

# Rexroth IndraMotion MTX micro MTX micro Trainer Turning - First Steps

R911341437  
Edition 01

## Application Description



<b>Title</b>	Rexroth IndraMotion MTX micro MTX micro Trainer Turning - First Steps						
<b>Type of Documentation</b>	Application Description						
<b>Document Typecode</b>	DOK-MTXMIC-TRAIN*TURN*-AP01-EN-P						
<b>Internal File Reference</b>	RS-7923826d9a01af5e0a6846a500a7976f-1-en-US-4						
<b>Purpose of Documentation</b>	This documentation describes the functions of the trainer for creating a turned part by using the Rexroth IndraMotion MTX micro.						
<b>Record of Revision</b>	<table border="1"><thead><tr><th>Edition</th><th>Release Date</th><th>Notes</th></tr></thead><tbody><tr><td>Edition 01</td><td>07.2013</td><td>First edition</td></tr></tbody></table>	Edition	Release Date	Notes	Edition 01	07.2013	First edition
Edition	Release Date	Notes					
Edition 01	07.2013	First edition					
<b>Copyright</b>	© Bosch Rexroth AG 2013 This document, as well as the data, specifications and other information set forth in it, are the exclusive property of Bosch Rexroth AG. It may not be reproduced or given to third parties without its consent.						
<b>Liability</b>	The specified data is intended for product description purposes only and shall not be deemed to be a guaranteed characteristic unless expressly stipulated in the contract. All rights are reserved with respect to the content of this documentation and the availability of the product.						
<b>Published by</b>	Bosch Rexroth AG Bgm.-Dr.-Nebel-Str. 2 ■ 97816 Lohr a. Main, Germany Phone +49 9352 18 0 ■ Fax +49 9352 18 8400 <a href="http://www.boschrexroth.com/">http://www.boschrexroth.com/</a> System Development Machine Tools, PeSc (FrWe/PiGe)						
<b>Note</b>	This document has been printed on chlorine-free bleached paper.						

# Table of Contents

	Page
<b>1 About this Documentation.....</b>	<b>3</b>
1.1 Validity of the Documentation.....	3
1.2 Required and Supplementing Documentation.....	3
1.2.1 Selecting.....	3
1.2.2 Configuring.....	4
1.2.3 Commissioning.....	4
1.2.4 Operating.....	5
1.2.5 Maintaining.....	5
1.2.6 Drive System: Commissioning and Project Planning.....	6
1.3 Information Representation.....	6
1.3.1 Safety Instructions.....	6
1.3.2 Symbols Used.....	7
1.3.3 Names and Abbreviations.....	7
<b>2 Introduction.....</b>	<b>9</b>
2.1 General Information.....	9
2.2 Operation.....	9
2.3 Programming Documents.....	12
2.3.1 Drawings.....	12
2.3.2 Unmachined Part.....	12
2.3.3 Machining Plan.....	13
2.3.4 Tools.....	13
2.4 Creating Tools.....	16
<b>3 NC Program External Machining.....</b>	<b>19</b>
3.1 General Information.....	19
3.2 Creating the NC Program "SHAFT_OP1".....	19
3.2.1 General Information.....	19
3.2.2 Creating the First NC Program "SHAFT_OP1.npg".....	19
3.2.3 OP1.1 - Face Turning with Allowance.....	21
3.2.4 OP1.2 - External Contour Roughing with Roughing Cycle G171.....	24
3.2.5 OP1.3 - External Contour Finishing.....	32
3.2.6 OP1.4 - Thread Cutting with Cycle G167.....	34
3.3 Complete Program "SHAFT_OP1".....	36
3.4 Simulating the NC Program "SHAFT_OP1".....	37
3.4.1 Entering the Zero Point Offset.....	37
3.4.2 Operating the Simulation.....	38
<b>4 NC Program Internal Machining.....</b>	<b>43</b>
4.1 Creating the NC Program "SHAFT_OP2".....	43
4.1.1 General Information.....	43
4.1.2 Creating the Second NC Program "SHAFT_OP2.npg".....	43
4.1.3 OP2.1 - Face Turning with Allowance.....	44

## Table of Contents

	Page
4.1.4 OP2.2 – Internal Boring D12.....	46
4.1.5 OP2.3 - Roughing Internal Contour with Allowance.....	48
4.1.6 OP2.4 - Finishing of Internal Contour.....	49
4.1.7 OP2.5 - Plane Surface and External Contour Finishing.....	51
4.1.8 OP2.6 - Groove External Turning.....	52
4.1.9 OP2.7 - Internal Thread M30x1.5 Edges.....	54
4.2 Complete Program "SHAFT_OP2".....	56
4.3 Simulating the NC Program "SHAFT_OP2".....	57
<b>5 Service and Support.....</b>	<b>59</b>
<b>Index.....</b>	<b>61</b>

# 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.

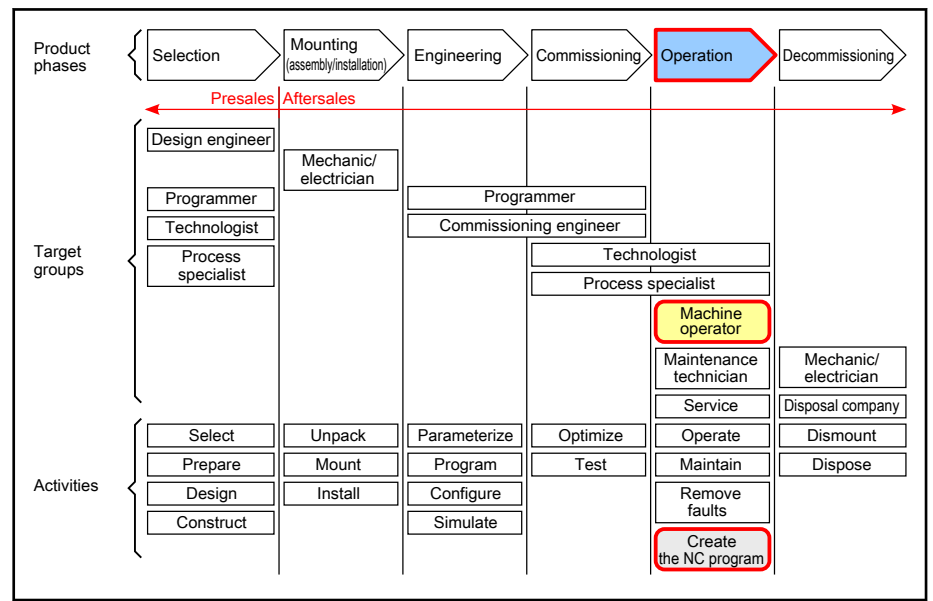


Fig. 1-1: Assigning this documentation to the target groups, product phases and target group activities

## 1.2 Required and Supplementing Documentation

### 1.2.1 Selecting

#### Documentation titles with type codes and parts numbers

<p><b>Rexroth IndraMotion MTX micro 12VRS System Description</b>                  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.</p>
<p><b>Rexroth IndraControl VDP 80.1 Machine Operator Panel Operator Display</b>                  DOK-SUPPL*-VDP*80.1***-PRxx-EN-P, R911329156                  This documentation contains a detailed description of the standard interface of the HMI operator panel.</p>
<p><b>Rexroth IndraDrive Drive Controllers HCQ, HCT</b>                  DOK-INDRV*-HCQ-T+HMQ-T-PRxx-EN-P, R911324185</p>

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

About this Documentation

## 1.2.2 Configuring

### Documentation titles with type codes and parts numbers

<p><b>Rexroth IndraMotion MTX micro 13VRS Functional Description Basics</b>  DOK-MTXMIC-NC*F*BA*V13-RExx-EN-P, R911336532</p> <p>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.</p>
<p><b>Rexroth IndraMotion MTX micro 13VRS Functional Description Extension</b>  DOK-MTXMIC-NC*F*EX*V13-RExx-EN-P, R911337298</p> <p>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.</p>
<p><b>Rexroth IndraMotion MTX micro 13VRS Machine Parameters</b>  DOK-MTXMIC-MA*PAR**V13-RExx-EN-P, R911336536</p> <p>This documentation describes the design and adjustment of the available parameters.</p>
<p><b>Rexroth IndraMotion MTX 13VRS PLC Interface</b>  DOK-MTX***-PLC*INT*V13-PRxx-EN-P, R911336344</p> <p>This documentation describes interface signals and program function blocks for the integrated PLC.</p>

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

## 1.2.3 Commissioning

### Documentation titles with type codes and parts numbers

<p><b>Rexroth IndraMotion MTX micro Easy Setup for Standard Turning and Milling Machines</b>  DOK-MTXMIC-EASY*****-COxx-EN-P, R911332281</p> <p>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.</p>
<p><b>Rexroth IndraWorks 13VRS Software Installation</b>  DOK-IWORKS-SOFTINS*V13-COxx-EN-P, R911336880</p> <p>This documentation describes the IndraWorks installation.</p>
<p><b>Rexroth IndraWorks 13VRS Engineering</b>  DOK-IWORKS-ENGINEE*V13-APxx-EN-P, R911336870</p> <p>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.</p>
<p><b>Rexroth IndraMotion MTX 13VRS Commissioning</b>  DOK-MTX***-STARTUP*V13-COxx-EN-P, R911336346</p> <p>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.</p>

**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. 1-4: *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. 1-5: *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. 1-6: *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

<p><b>Rexroth IndraDrive Drive Controllers HCQ, HCT</b>  DOK-INDRV*-HCQ-T+HMQ-T-PRxx-EN-P , R911324185</p> <p>It is used for the project planning of the drive systems Rexroth IndraDrive with the listed components</p> <ul style="list-style-type: none"> <li>• HCQ02</li> <li>• HCT02</li> </ul>
<p><b>Rexroth IndraDrive MPx-17 Functions</b>  DOK-INDRV*-MP*-17VRS**-APxx-EN-P, R911331236</p> <p>This documentation describes all the functional properties of the IndraDrive firmware in the variants MPB-17, MPM-17, MPC-17 and MPE-17.</p>
<p><b>Rexroth IndraDrive MPx-16 and MPx-17 Parameters</b>  DOK-INDRV*-GEN1-PARA**-RExx-EN-P, R911328651</p> <p>This documentation describes all parameters implemented in the firmware for drive controllers of the IndraDrive family. It supports the parameterization of the drive controllers.</p> <ul style="list-style-type: none"> <li>• FWA-INDRV*-MPx-16VRS</li> <li>• FWA-INDRV*-MPx-17VRS</li> </ul>
<p><b>Rexroth IndraDrive MPx-16 and MPx-17 Diagnostics</b>  DOK-INDRV*-GEN1-DIAG**-RExx-EN-P, R911326538</p> <p>This documentation describes the diagnostics implemented into the following firmwares:</p> <ul style="list-style-type: none"> <li>• FWA-INDRV*-MPx-16VRS</li> <li>• FWA-INDRV*-MPx-17VRS</li> </ul> <p>It supports the operating crew as well as the programmer at troubleshooting.</p>
<p><b>Rexroth IndraDrive MPx-17 Version Notes</b>  DOK-INDRV*-MP*-17VRS**-RNxx-EN-P, R911331588</p> <p>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.</p> <p>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.</p>
<p><b>Rexroth IndraDyn S Synchronous Motors QSK061, -075</b>  DOK-MOTOR*-QSK*****-PRxx-EN-P, R911330321</p> <p>This documentation...</p> <ul style="list-style-type: none"> <li>• explains the features of the product, possibilities for use, operating conditions and operational limits of QSK motors</li> <li>• contains technical data regarding available QSK motors</li> <li>• provides information on product selection, handling and operation</li> </ul>

xx Corresponding version or edition  
*Tab. 1-7: 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).

## About this Documentation

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. 1-8: Names and abbreviations used



## 2 Introduction

### 2.1 General Information

**About this documentation** This documentation is intended for users wanting to manufacture workpieces at turning 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 turned part, the entire process, starting from the drawing to the finished part, is explained using the **IndraMotion MTX micro Trainer**. Download the software under [www.boschrexroth.com/mtxtrainer](http://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 knowledge 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

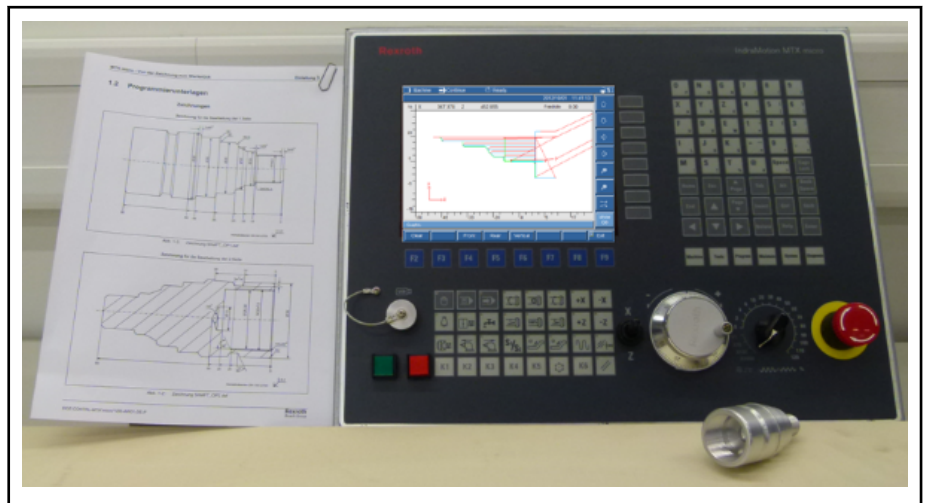


Fig.2-1: From drawing to workpiece

### 2.2 Operation

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

Introduction

MTX micro Trainer

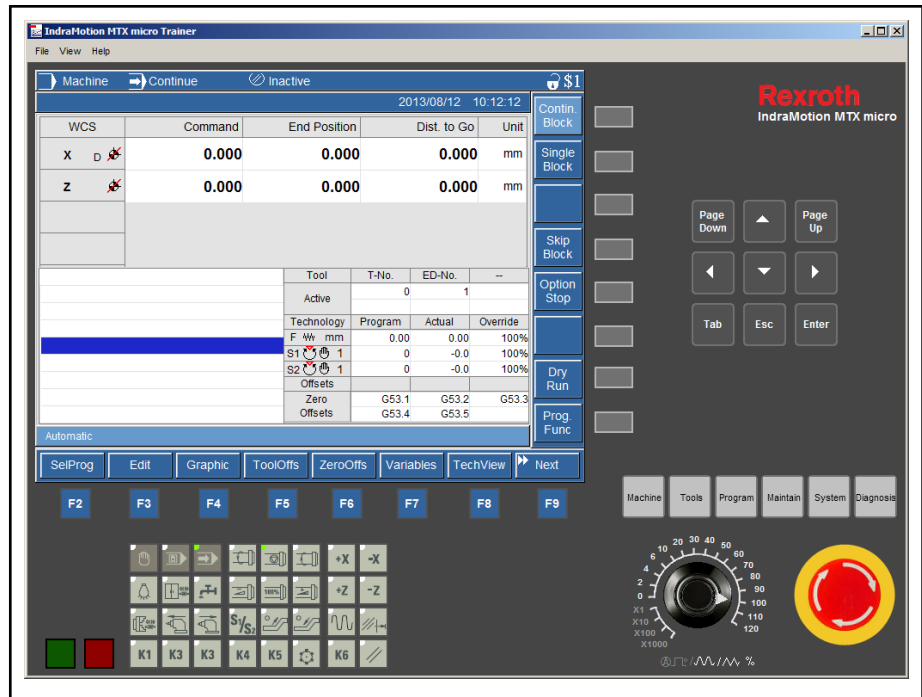


Fig.2-2:

Operation modes manual / MDI / automatic



Fig.2-3:

NC start / NC stop



Fig.2-4:

Operating areas

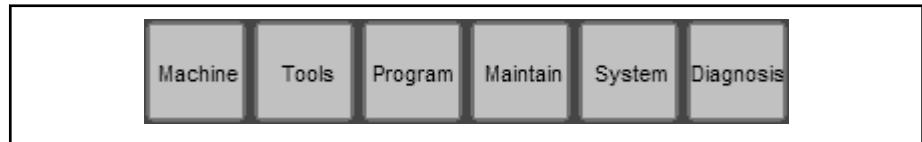


Fig.2-5:

M-Key 1 to 8

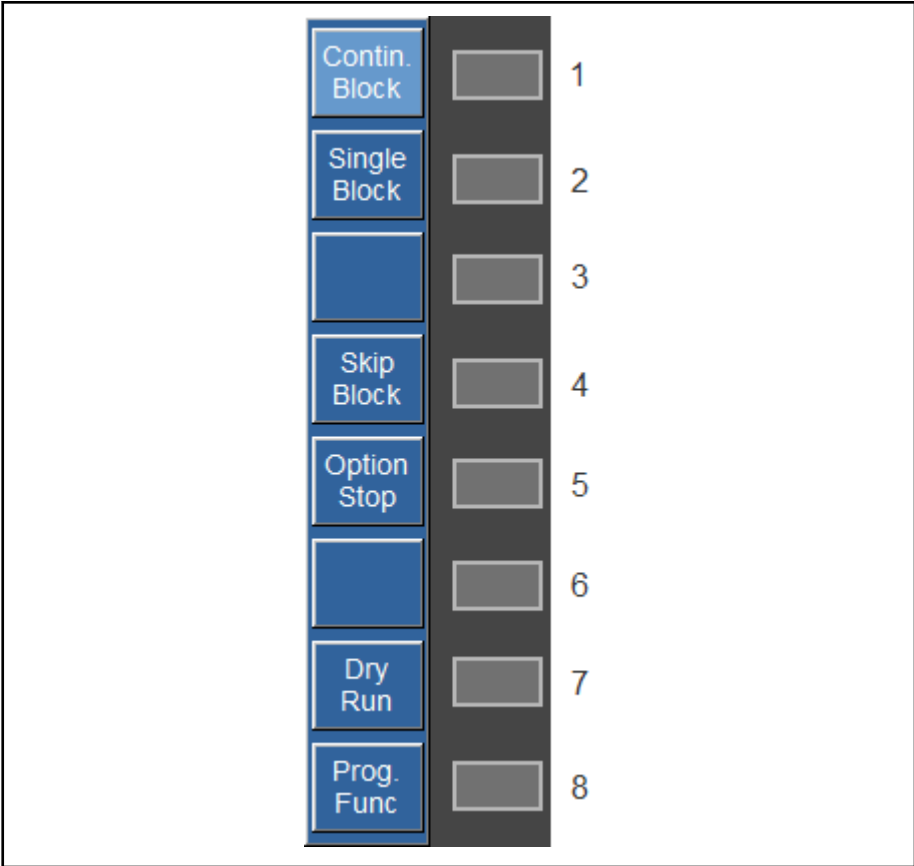


Fig.2-6:

Introduction

## 2.3 Programming Documents

### 2.3.1 Drawings

Drawing for machining of 1st side:

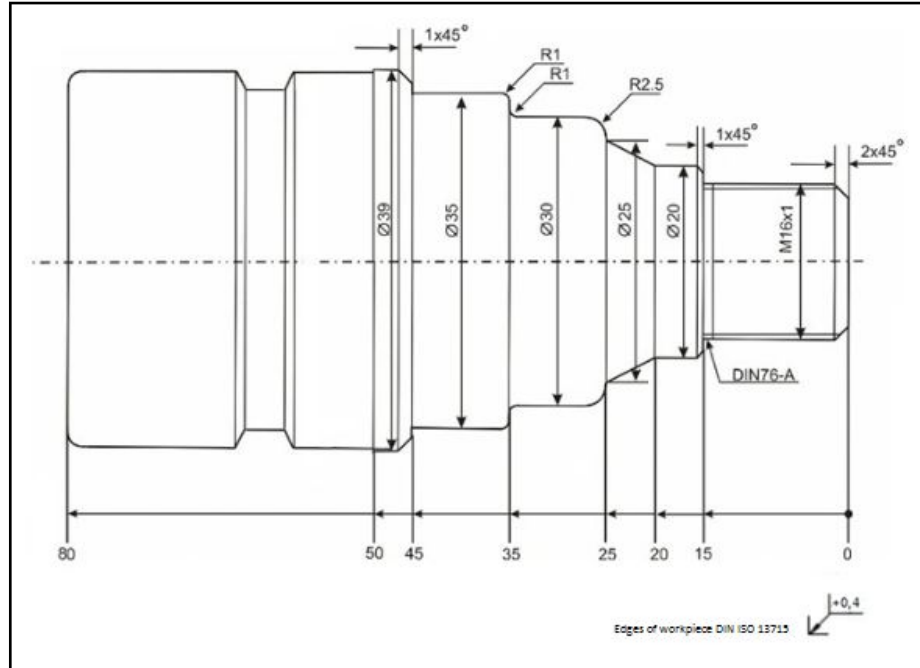


Fig.2-7: Drawing SHAFT\_OP1.dxf

Drawing for machining of 2nd side:

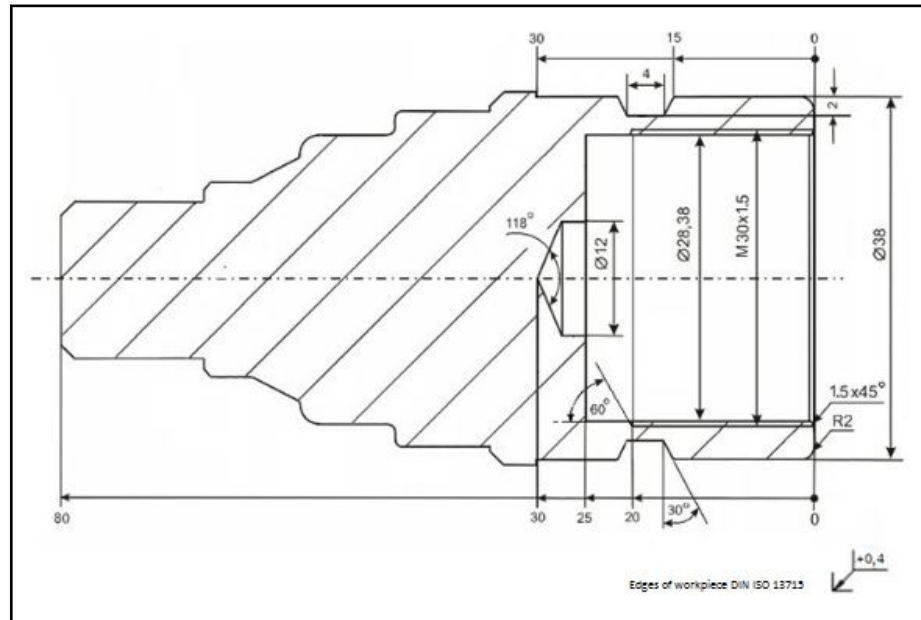


Fig.2-8: Drawing SHAFT\_OP2.dxf

### 2.3.2 Unmachined Part

Material: Aluminium

Diameter: D 40mm

Length: L 82mm

## 2.3.3 Machining Plan

Machining Plan OP\_No: Machining operation number

OP_No:	Machining operation	Remark	TNo.
<b>OP1 - 1st side</b>			
OP 1.1	Face turning with allowance	1mm for 2nd side	1
OP 1.2	External contour roughing with allowance	Radial 0.1mm Axial 0.1mm	1
OP 1.3	External contour finishing		2
OP 1.4	External thread M16x1 cutting		4
<b>OP2 - 2nd side</b>			
OP 2.1	Face turning with allowance	0.1mm	1
OP 2.2	Internal boring D12	Z-30	6
OP 2.3	Internal contour with allowance roughing	Radial 0.1mm Axial 0.1mm	7
OP 2.4	Internal contour finishing		
OP 2.5	Plane surface and external contour finishing		1
OP 2.6	Groove external turning		3
OP 2.7	Internal thread M30x1,5 cutting	60° run-out	8

Tab.2-9: Machining plan for 1st and 2nd setting

## 2.3.4 Tools

The edge position (orientation) has to be specified to machine the radius compensation G41/G42.

Introduction

Tool Edge Position Orientation

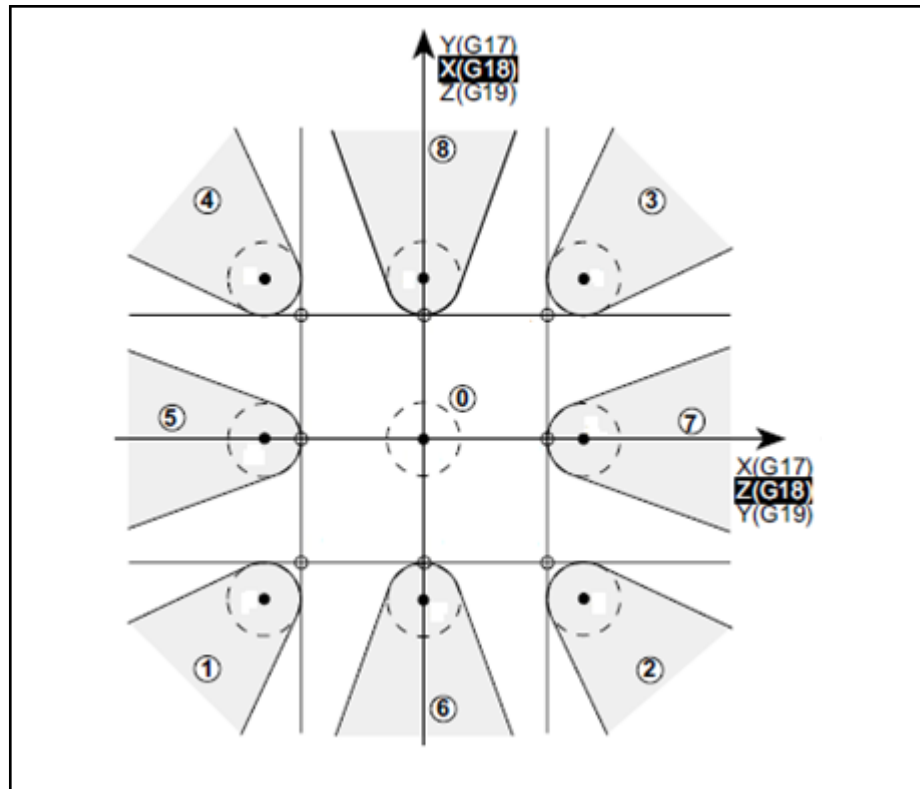
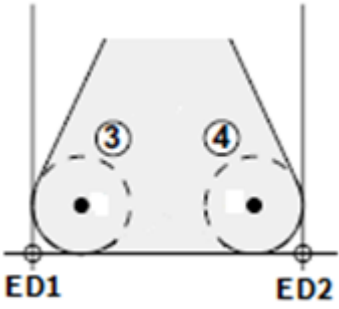
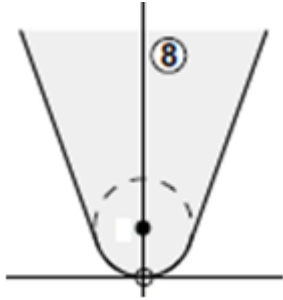
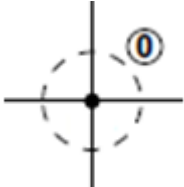
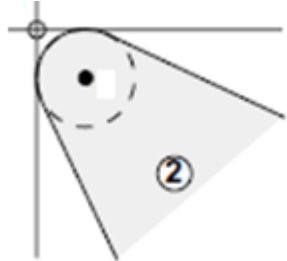
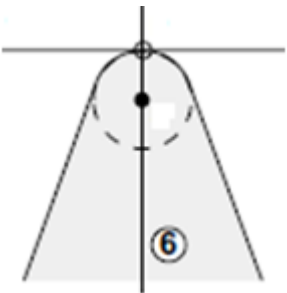


Fig.2-10:

Tool plan The following tools are required according to the machining plan:

Name	Turret pocket	Type	Radius	Edge position
T1	1	Roughing tool, external	0.4mm	
T2	2	Finishing tool, external	0.4mm	

Name	Turret pocket	Type	Radius	Edge position
T3	3	Grooving tool, external, width: 3mm. Specify ED1 and ED2	0.4mm	
T4	4	Threading tool, external	0	
T6	6	HSS drilling machine D12 118°	0	
T7	7	Boring bar	0.4mm	
T8	8	Thread boring bar M1.5		

Tab.2-11: Tool plan for 1st and 2nd setting

## Introduction

## 2.4 Creating Tools

Change to "Tools" machine operating area

Press "Tools".

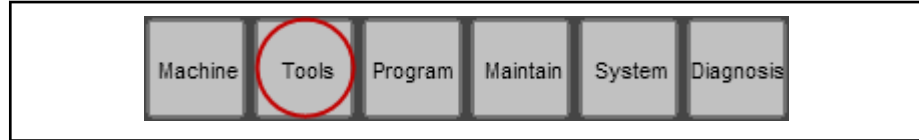


Fig.2-12:

### Tool list display

The turning machine is provided with a turret with 12 tool pocket locations. The is assigned the digit 1 in the "S" column of the tool list. The tool pocket locations of the turret are represented by the digits 1-12 in column "turretP".

The turret pocket can be locked using the <M-Key 1>, the tool can be locked using <M-Key 2>. This is visualized in the "PB" or "TL" column.

The line highlighted in orange refers to the active tool

The line highlighted in blue indicates the turret pocket selected to edit the tool data. Change this selection using <Cursor up> and <Cursor down>.

The lower image shows that the tool on turret pocket 2 is active and the empty turret pocket 3 has been selected for editing.

Pocket			T No.	TL	ED Curr./All	Geometry			
S	P	PB				Z	X	Radius	Ori
1	1	<input type="checkbox"/>	1	<input type="checkbox"/>	1/1	53.378	70.065	0.400	3
1	2	<input type="checkbox"/>	2	<input type="checkbox"/>	1/1	54.098	70.985	0.400	3
1	3	<input type="checkbox"/>		<input type="checkbox"/>					
1	4	<input type="checkbox"/>	4	<input type="checkbox"/>	1/1	47.145	66.311	0.000	8
1	5	<input type="checkbox"/>		<input type="checkbox"/>					
1	6	<input type="checkbox"/>	6	<input type="checkbox"/>	1/1	152.147	0.000	0.000	0
1	7	<input type="checkbox"/>	7	<input type="checkbox"/>	1/1	139.250	-5.811	0.400	2
1	8	<input type="checkbox"/>	8	<input type="checkbox"/>	1/1	140.738	-0.401	0.000	6
1	9	<input type="checkbox"/>		<input type="checkbox"/>					
1	10	<input type="checkbox"/>		<input type="checkbox"/>					
1	11	<input type="checkbox"/>		<input type="checkbox"/>					
1	12	<input type="checkbox"/>		<input type="checkbox"/>					

Fig.2-13:

### 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 3, select turret pocket 3 in the tool list by moving the blue bar.

To use more than one tool edge, e.g. a grooving tool, increase the number of edges in parameter "Offset Count (ED)".

### Entering tool data

Example: Press <F3>"Edit" and enter the following tool data:

- T number 3
- Offset Count (ED) 2

*Data for tool edge ED1:*

- Z Offset 54.559
- X Offset 71.580
- R Offset 0.400
- Ori 3

The screenshot shows the 'Tool Editor' window with the following data:

Tool Data		Actual Position	
	Value	Storage	Pocket
T Number	3	1	3
Offset Count (ED)	2	Origin Position	
		Storage	Pocket
		0	0

Tool Status		Tool Offset	
	Value	ED1	
Used Tool (tu)	<input type="checkbox"/>	Z Offset	54.559
Life Warning Limit (tw)	<input type="checkbox"/>	X Offset	71.580
Tool Locked (TL)	<input type="checkbox"/>	R Offset	0.400
Fixed Place Tool (TPC)	<input type="checkbox"/>	Ori	3
Tool Life (min)		Z Wear	0.000
Remaining	0.000	X Wear	0.000
Warning Limit	0.000	R Wear	0.000
Tool Life Time	0.000		

At the bottom, the 'Tool Editor' section contains buttons: Tool +, Tool -, Ed +, Ed -, OK, and Cancel.

*Fig.2-14:*

Subsequently, press <F4>"Ed+" to enter the data of the tool edge ED2.

*Data for tool edge ED2:*

- Z Offset 51.559
- X Offset 71.580
- R Offset 0.400
- Ori 4

Introduction

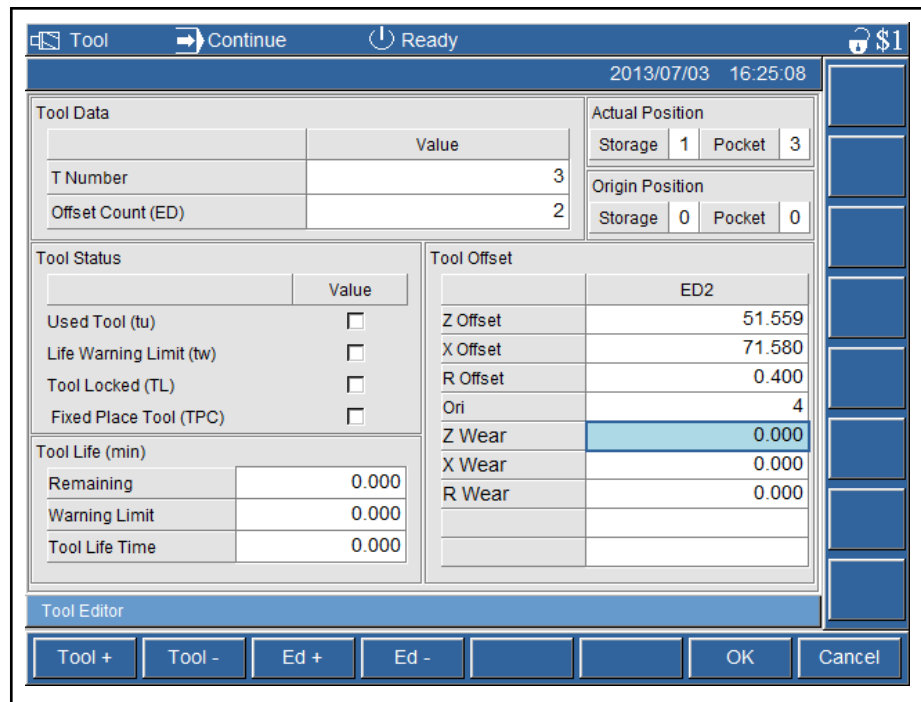


Fig.2-15:

Applying tool data Press <F8>"OK" to apply the data.

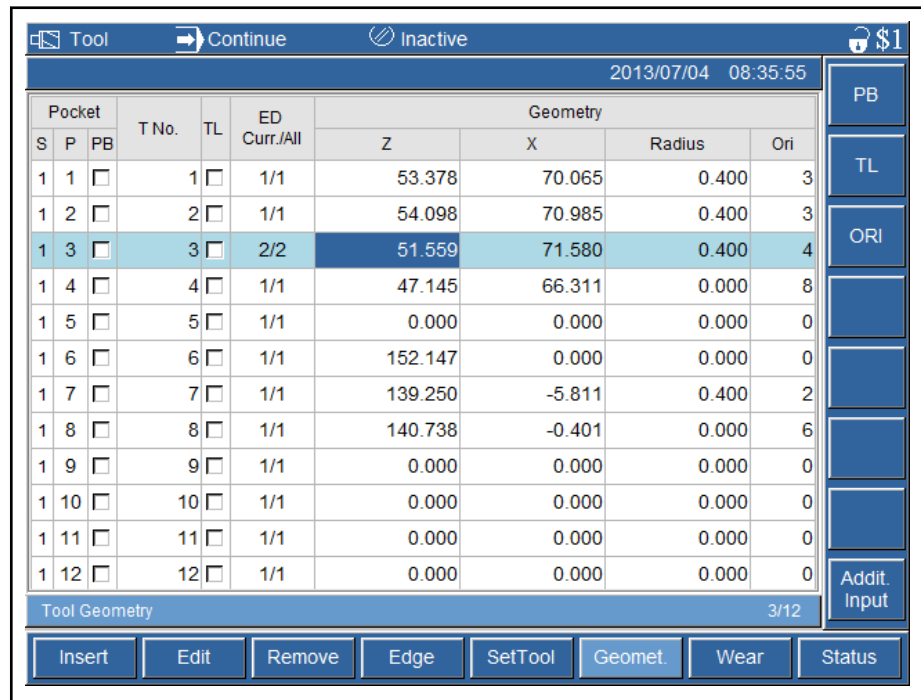


Fig.2-16:

The tool T3 has been inserted in turret pocket 3. 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 External Machining

### 3.1 General Information

Two machining programs are required for complete machining:

- SHAFT\_OP1 external machining (shaft, 1st setting)
- SHAFT\_OP2 internal machining (shaft, 2nd setting)

### 3.2 Creating the NC Program "SHAFT\_OP1"

#### 3.2.1 General Information

Program flow chart Dividing machining in programming steps

OP_No.	Step	Program 1.side	Note	To ol
	1.1	Generate program header		
OP1.1	1.2	Tool change, technology input		1
	1.3	Face turning		
OP1.2	1.4	Describing the external contour		
	1.5	External contour roughing with allowance	G171	
OP1.3	1.6	Tool change, technology input		2
	1.7	External contour finishing	G170	
OP1.4	1.8	Tool change, technology input		4
	1.9	External thread M16x1 cutting	G167	
	1.10	Approach position for unclamping		

Tab.3-1:

#### 3.2.2 Creating the First NC Program "SHAFT\_OP1.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.

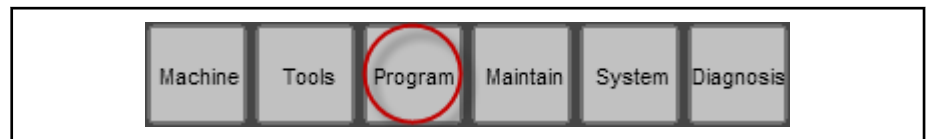


Fig.3-2:

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 <F7>"Save". Enter a program name, e.g. "SHAFT\_OP1.npg".



**Note:** The MTX micro file system is case sensitive. This refers to the names of directories and programs. For example, the program names "SHAFT\_OP1.npg", "Shaft\_Op1.npg" and "shaft\_op1.npg" would refer to three different programs.

## NC Program External Machining

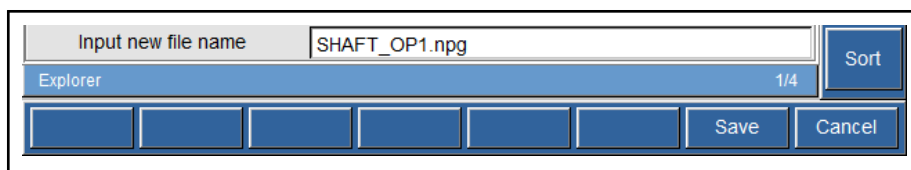


Fig.3-3:

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

The program that is still empty, is now saved under the name "SHAFT\_OP1.npg" in the "/prog" directory.

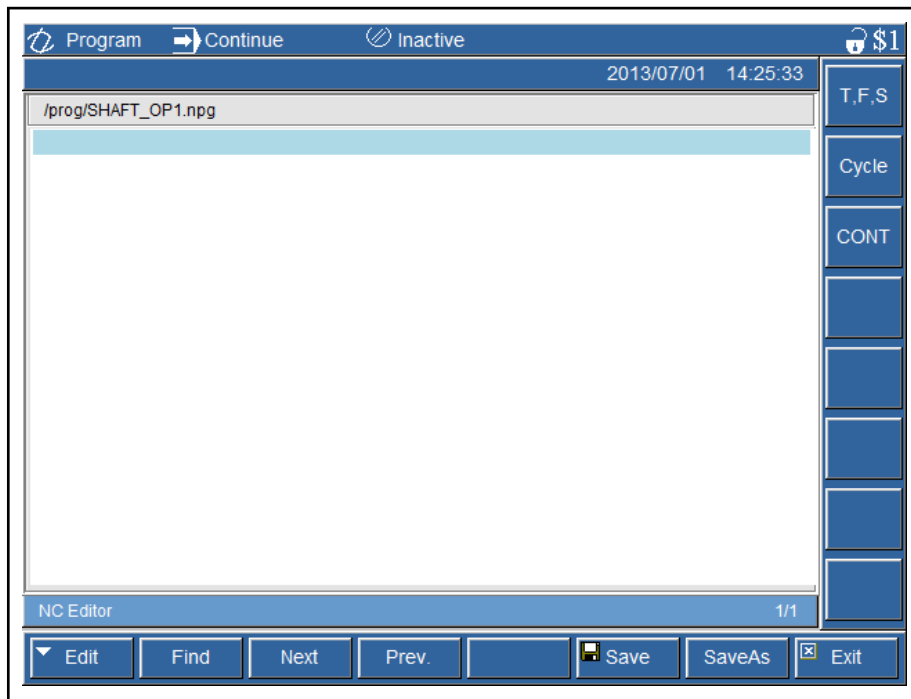


Fig.3-4:

## Step 1.1 - Creating the Program Header

**Comment** Enter comments at the beginning of the program to track changes.

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

*Program header*


---

```

;Millingmachine:  MTX_micro
;NC program name: SHAFT_OP1.npg  Drawing:          SHAFT_OP1.dxf
;Material:        Aluminium      Stock           Dimension: D40mm x 82mm
;Author:          Turner          Version: 1.0.0  Date: 24.02.2013

```

---

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 external contour
// begin
N1000
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
```

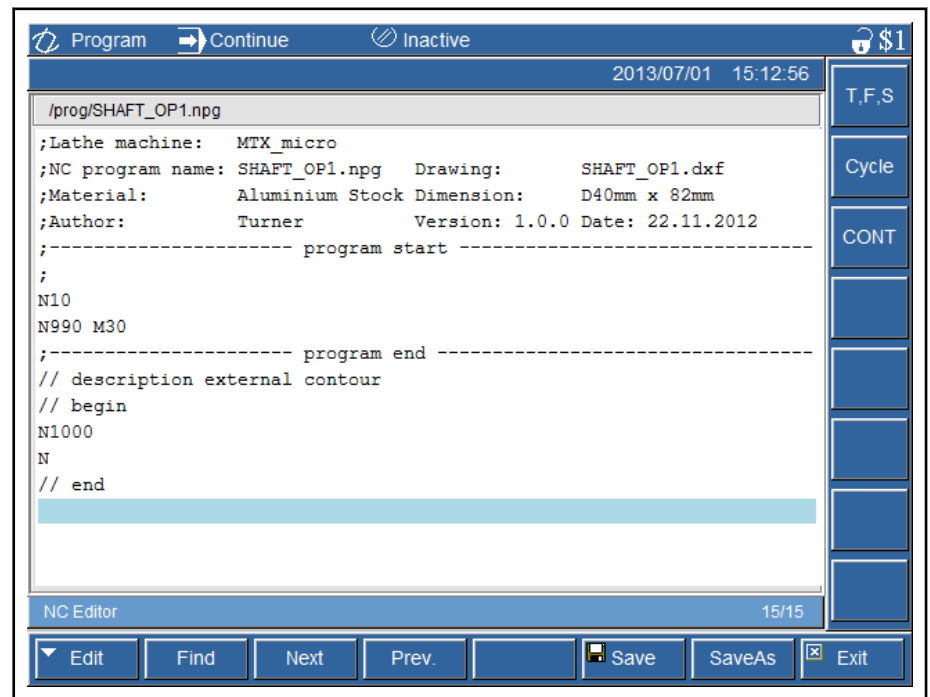


Fig.3-5:

### 3.2.3 OP1.1 - Face Turning with Allowance

#### General Information

During the first machining operation, the workpiece is face turned. The roughing tool 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.

```
;OP1.1 Face milling with allowance
```


Program a traversing motion to a safe, machine-based position (G76) before swivelling the turret.

Position the cursor after "N10" and enter the following NC block:

```
N10 G76 G0 X250 Z600
```

Press <Enter> to enter the next NC block.

NC Program External Machining

 The positions x250 z600 are used as an example. The positions at your machine may differ.

Step 1.2 - Tool Change T1

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

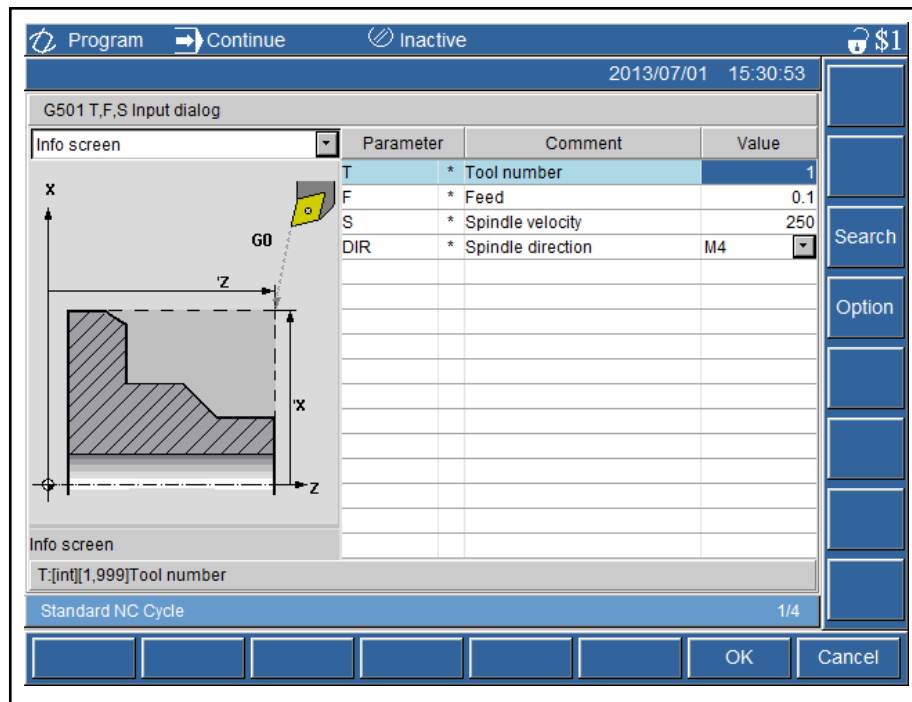



Fig.3-6:

Entering values in the cycle input masks

Select an input field in the cycle mask and enter the value. Complete the input 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 N20 and by pressing "T,F,S" or "Cycle".

Tool change T1 and technology

Specify all conditions for subsequent machining in the input mask:

T1	Tool, roughing tool, external
F0.1	Roughing feed 0.1 mm/rev
S250	Spindle velocity 250 m/min
M4	Direction of rotation, counterclockwise
SMX3000	Maximum spindle speed 1/min
ZG G54	Workpiece zero point G54

- ZB 1 Zero point bank 1
- TCH0 Tool change positioning 0=No

Tab.3-7:

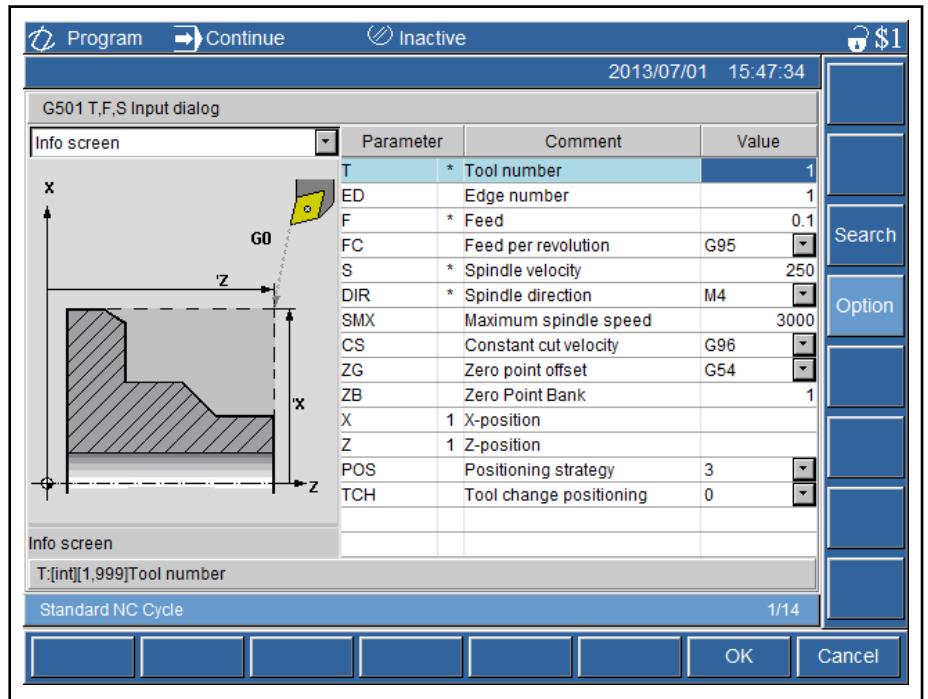


Fig.3-8:



**Note:** The input mask shows all parameters used in the cycle. Check the basic settings of the optional parameters. Comply with the maximum speed SMX.

**Applying the cycle into the program**

Complete the input by pressing <F8>"OK". The cycle G501 has been applied to the program. Insert a line break with <Enter> at the end of the line.

```

;----- program start -----
;OP1.1 Face milling with allowance
N10 G76 X250 Z600
N20 G501(T1,F0.1,S250,DIR M4,SMX3000,ZG G54,TCH 0)
N990 M30
;----- program end -----
    
```

Fig.3-9:

**Step 1.3 - 1st Side Face Turning**

**Face turning**



The specified allowance of 1mm serves to machine the machined part length of 80mm that is generated during face turning of the 2nd side. Thus, the length is 81mm after the face cut on the 1st side.

Insert the NC blocks N30 to N60:

```

N30 G0 X45 Z0
N40 G1 X-1
N50 G1 Z0.5
    
```

## NC Program External Machining

```
N60 G0 X41
```

```

;----- program start -----
;OP1.1 Face milling with allowance
N10 G76 X250 Z600
N20 G501(T1,F0.1,S250,DIR M4,SMX3000,ZG G54,TCH 0)
N30 G0 X45 Z0
N40 N40 G1 X-1
N50 G1 Z0.5
N60 G0 X41
N990 M30
;----- program end -----

```

Fig.3-10:

### 3.2.4 OP1.2 - External Contour Roughing with Roughing Cycle G171

#### General Information

For this machining operation, the contour is first described and subsequently machined.

Enter a comment for this machining operation:

```

;-----
;OP1.2 External contour roughing with roughing cycle G171

```

#### Step 1.4 - Describing the External Machined Part Contour

The external 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, chamfers and complex contour drafts. 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()

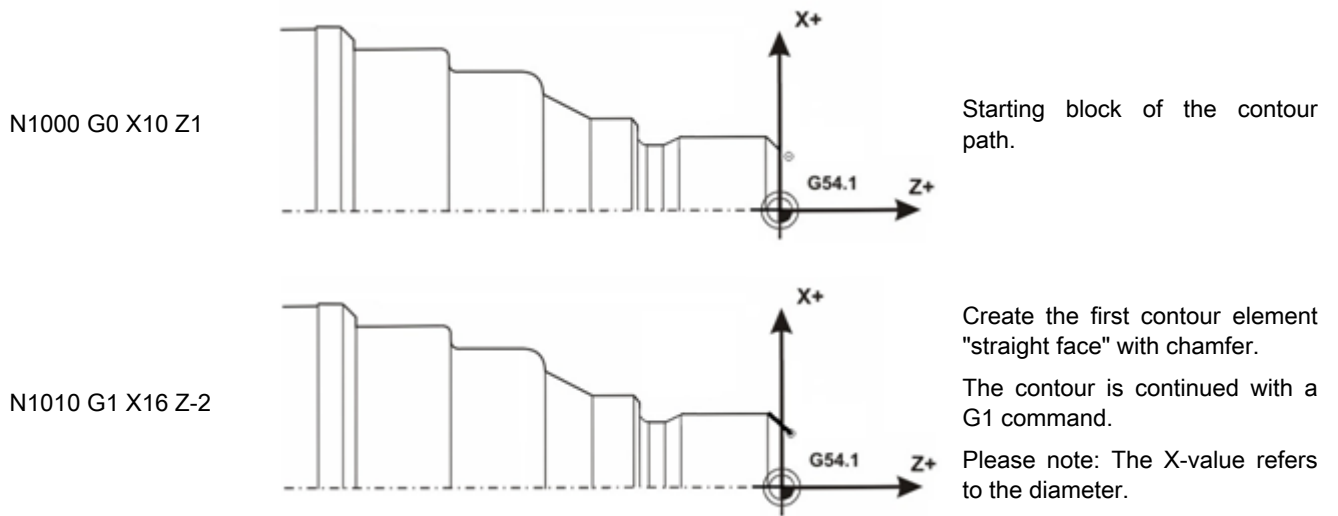
*Block-by-block active:*

- CHLL(<Chamfer length>)
- RNDL(<Rounding radius>)

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

##### Contour path definition

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



Tab.3-11:

### Including the Thread Undercut DIN 76 A in the Contour Description

**Selecting the cycle G196** First select the group "9" (turning cycles).



Use <M-Key 8>">>" to display the next 8 group. Use <M-Key 1>"<<" to see the previous 8 group.

Enter "9" in the input field and apply the group with <F8>"Select".

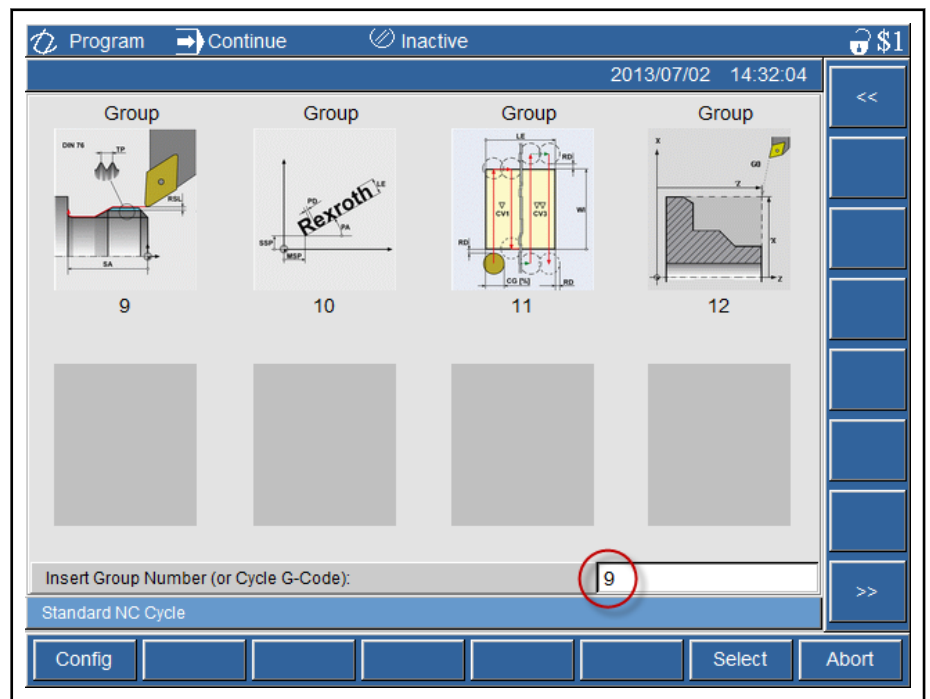


Fig.3-12:

Select the cycle 1 / G196. Enter "1" in the input field and apply the cycle with <F8>"Select".

## NC Program External Machining

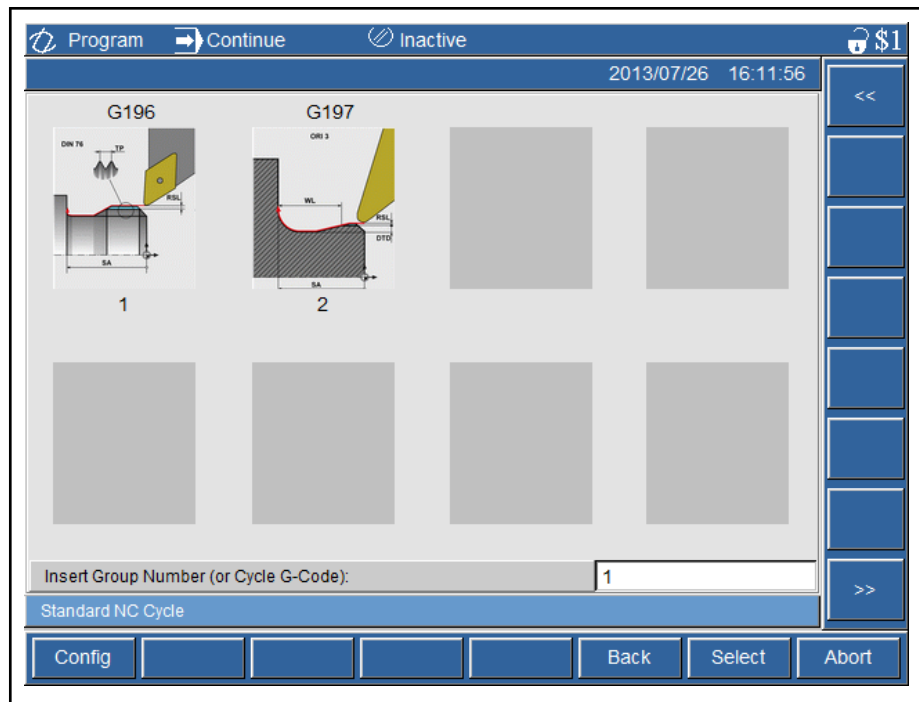


Fig.3-13:

## Releasing technology data for the undercut cycle



Only specify the diameter DN if the groove is to be machined as individual contour with approaching and leaving the contour. Within contours, DN is determined by the end point of the previous geometry element. In this case, specifying a value for DN would result in an error.

## Specify the parameters for the machining cycle G196 in the cycle input mask:

ORI	3	Position orientation
ART	1	Part of DIN 76
FORM	A	Contour acc. to DIN 76
SA	-15	Axial reference coordinate
TP	1	Pitch
ANG	30	Angle
RSL	0	Allowance

Tab.3-14:

NC Program External Machining

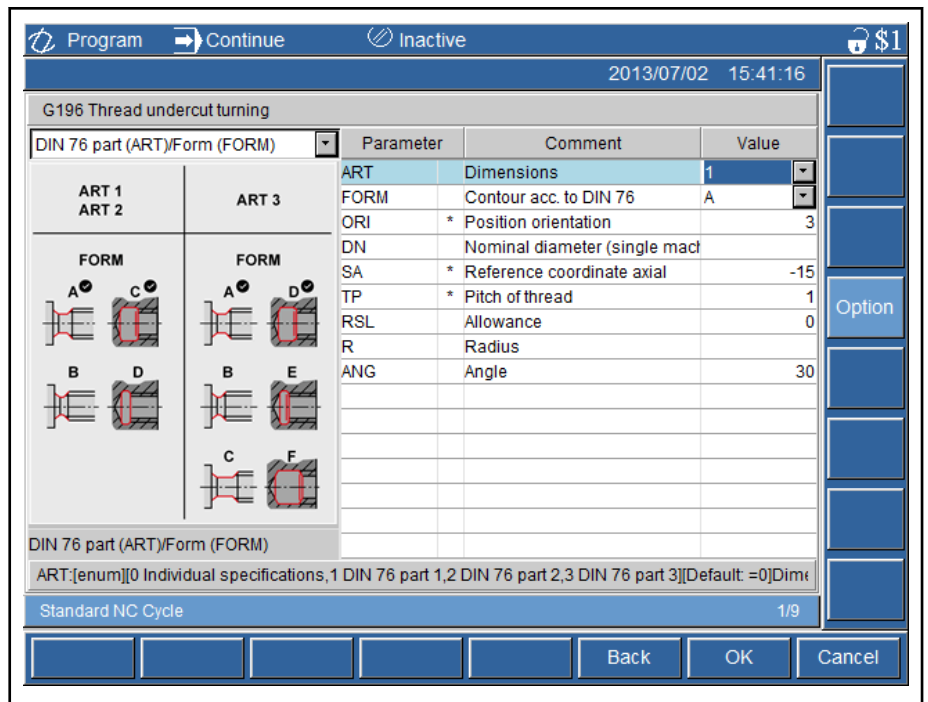
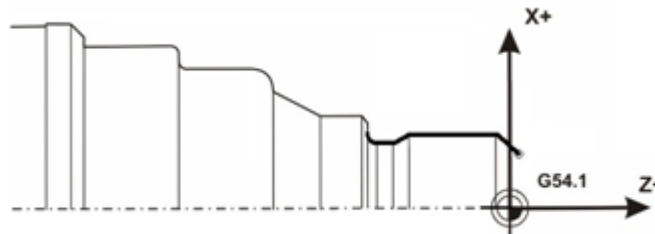


Fig.3-15:

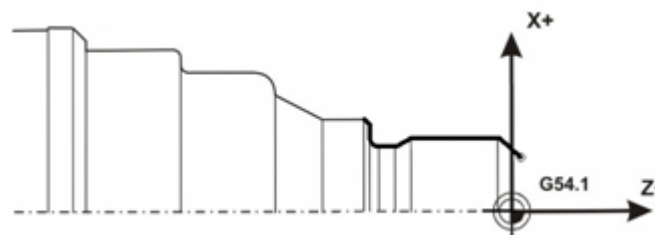
Complete the input by pressing <F8>"OK".

N1020  
G196(ORI3,ART  
1,FORM A,  
SA-15,TP1)



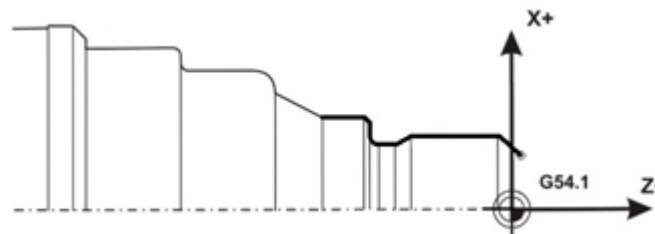
The thread undercut has been inserted by means of the input mask.

N1030 G1 X20  
CHLL(1)



Enter the contour element "straight line, face" with chamfer.

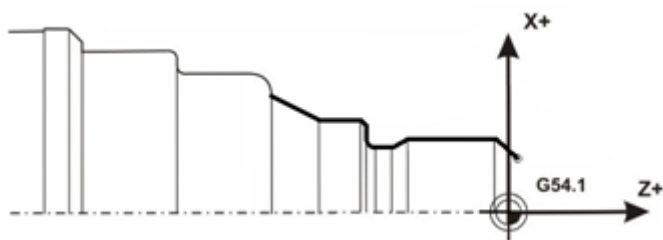
N1040 G1 Z-20



Enter the contour element "straight line, longitudinal".

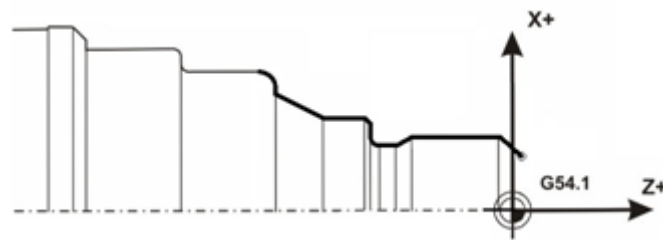
NC Program External Machining

N1050 G1 X25 Z-25



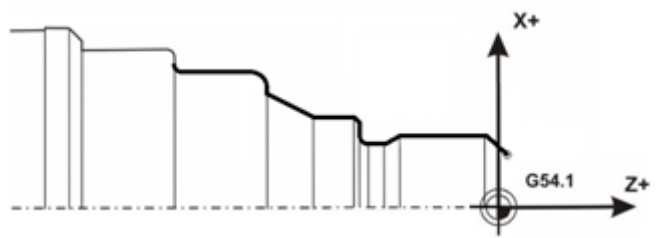
Enter the contour element "Inclined line".

N1060 G1 X30  
RNDL(2.5)



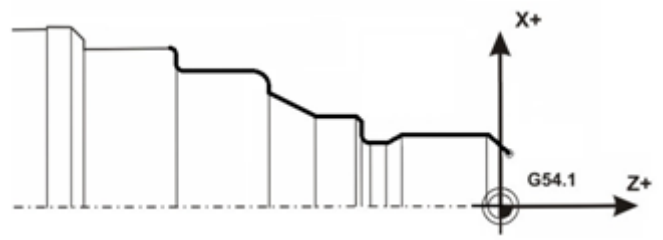
Enter the contour element "straight line, face with rounding".

N1070 G1 Z-35  
RNDL(1)



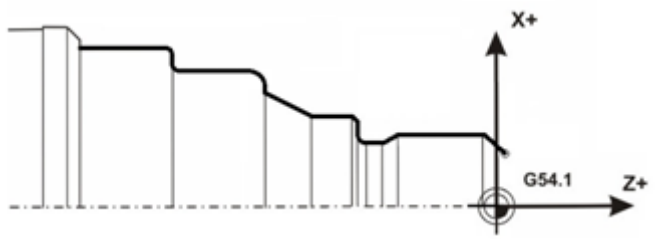
Enter the contour element "straight line, longitudinal with rounding".

N1080 G1 X35  
RNDL(1)



Enter the contour element "straight line, face with rounding".

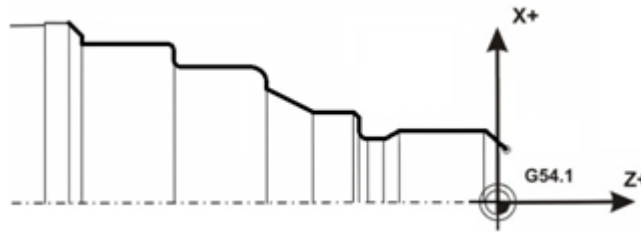
N1090 G1 Z-45



Enter the contour element "straight line, longitudinal".

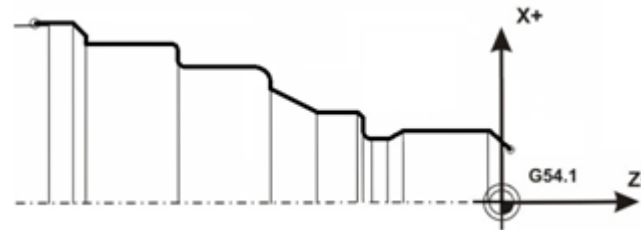
## NC Program External Machining

N1100 G1 X39  
CHLL(1)



Enter the contour element  
"straight line, face" with chamfer.

N1110 G1 Z-51



End block of contour path: Enter  
the contour element "straight  
line, longitudinal".

Tab.3-16:

External contour path

```
// description external contour
// begin
N1000 G0 X10 Z1
N1010 G1 X16 Z-2
N1020 G196(ORI3,ART 1,FORM A, SA-15,TP1)
N1030 G1 X20 CHLL(1)
N1040 G1 Z-20
N1050 G1 X25 Z-25
N1060 G1 X30 RNDL(2.5)
N1070 G1 Z-35 RNDL(1)
N1080 G1 X35 RNDL(1)
N1090 G1 Z-45
N1100 G1 X39 CHLL(1)
N1110 G1 Z-51
// end
```

Fig.3-17: External contour path description

## Step 1.5 - External Contour Roughing

Entering comment and NC block  
number

Navigate up and enter the block number "N70" after the comment for OP1.2.  
Press <M-Key 2>"Cycle". A list box with the cycle group is opened.

## NC Program External Machining

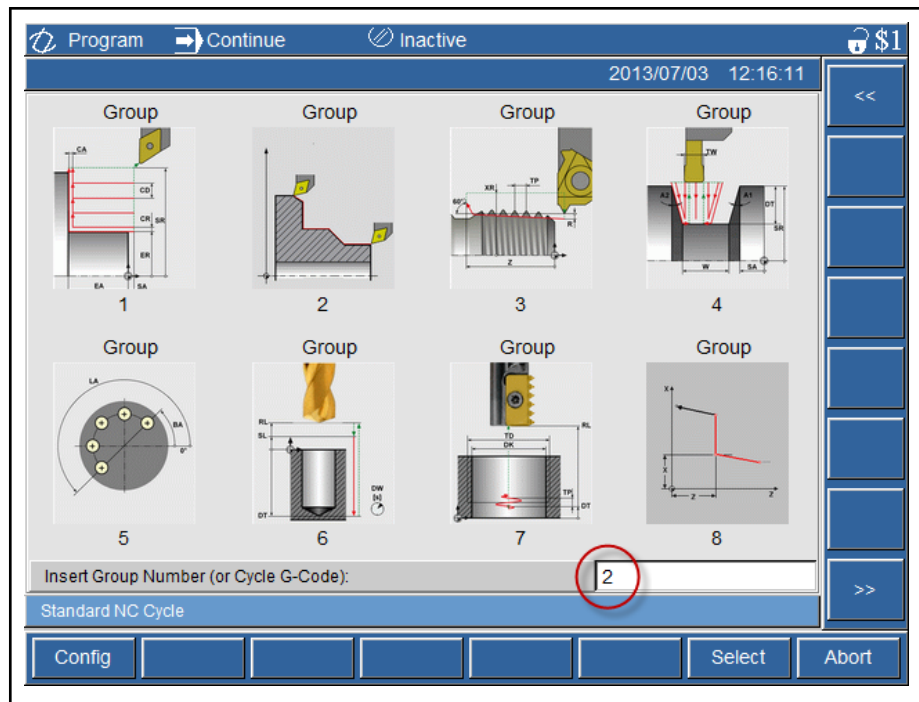


Fig.3-18:

## Selecting the group turning cycles

Select the group "2", to select the group turning cycles.

## Selecting the roughing cycle

The available cycles of the selected group are now displayed. Select the roughing cycle G171 by entering "2" in the input line.

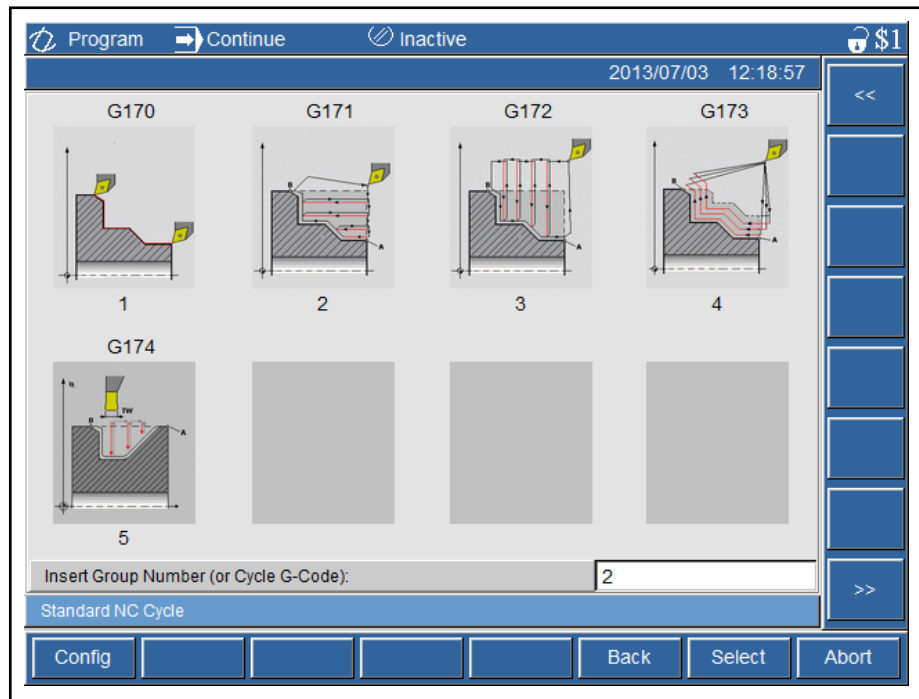


Fig.3-19:

Turning cycle G171 Specify the parameters for the roughing cycle G171 in the technology window:

S	1000	Starting block of the contour description
E	1110	End block of the contour description
CD	1	Maximum depth of cut
LD	1	Retract amount after cut
CR	0.1	Longitudinal allowance
CA	0.1	Plane allowance
UCV	3	Undercut variant

Tab.3-20:

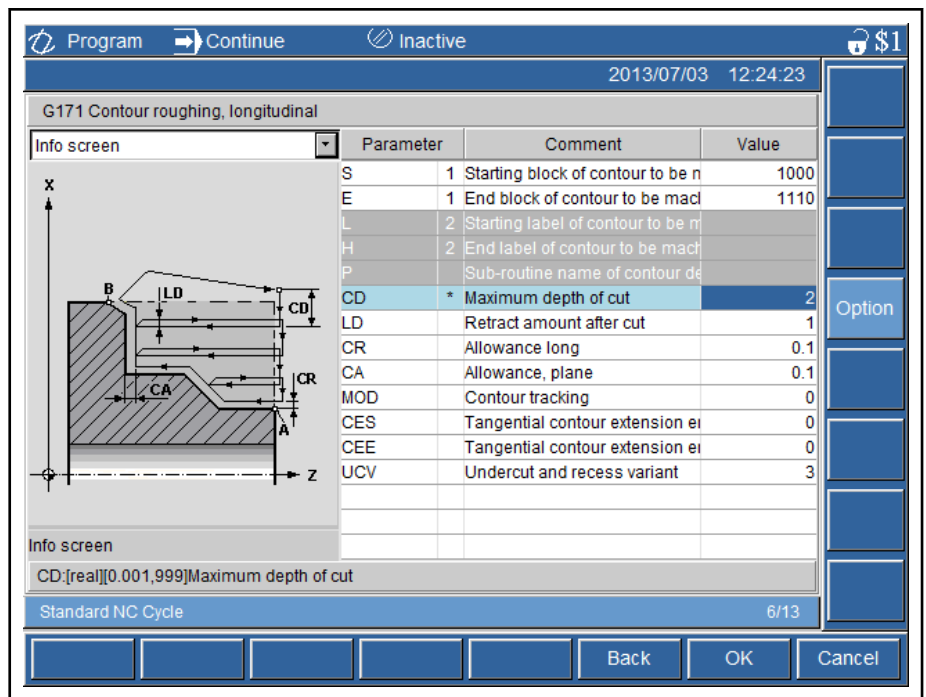


Fig.3-21:

Complete the input by pressing <F8>"OK". The cycle is inserted in the program.

```
N70 G171(S1000,E1110,CD1,LD1,CR0.1,CA0.1,UCV3)
```

Program a traversing motion to a safe, machine-based position (G76) before swivelling the turret:

```
N80 G76 G0 X250 Z600
```



The positions X250 Z600 are used as an example. The positions at your machine may differ.

## NC Program External Machining

```

;-----
;OP1.2 Roughing of external contour with roughing cycle G171
N70 G171 (S1000,E1110,CD1,LD1,CR0.1,CA0.1,UCV3)
N80 G76 G0 X250 Z600
;-----
N990 M30
;----- program end -----

```

Fig.3-22:

### 3.2.5 OP1.3 - External Contour Finishing

#### General Information

After roughing of the contour has been completed, the external contour is finished. Tool T2, a finishing tool, is swiveled. Enter a comment line for this machining operation:

```

;-----
;OP1.3 Finishing external countour

```

#### Step 1.6 - Tool Change T2

##### Tool change T2 and technology

The tool change is inserted by means of the "T,F,S" cycle.

**The contour is turned with the tool T2 using the following parameters:**

T	2	Tool, finishing tool, external
F	0.08	Finishing feed 0.08 rpm
S	250	Spindle velocity 250 m/min
M	4	Direction of rotation, counterclockwise
SMX	3000	Maximum spindle speed 1/min
ZG	G54	Workpiece zero point G54
ZB	1	Zero point bank 1
TCH	0	Tool change positioning 0=No

Tab.3-23:

```
N90 G501(T2, F0.08, S250, DIR M4, SMX3000, ZG G54, TCH0)
```

Subsequently, the edge radius compensation, right, is enabled in the N100 block with G42:

```
N100 G42
```

#### Step 1.7 - External Contour Finishing with Cycle G170

**Cycle G170** Select cycle "1" / G170 in the group "2" (turning cycles). Optionally, make entries directly in the input line "G170".

Specify the parameters for the machining cycle G170 in the cycle input mask:

S=1000 Starting block of the contour description

E=1110 End block of the contour description

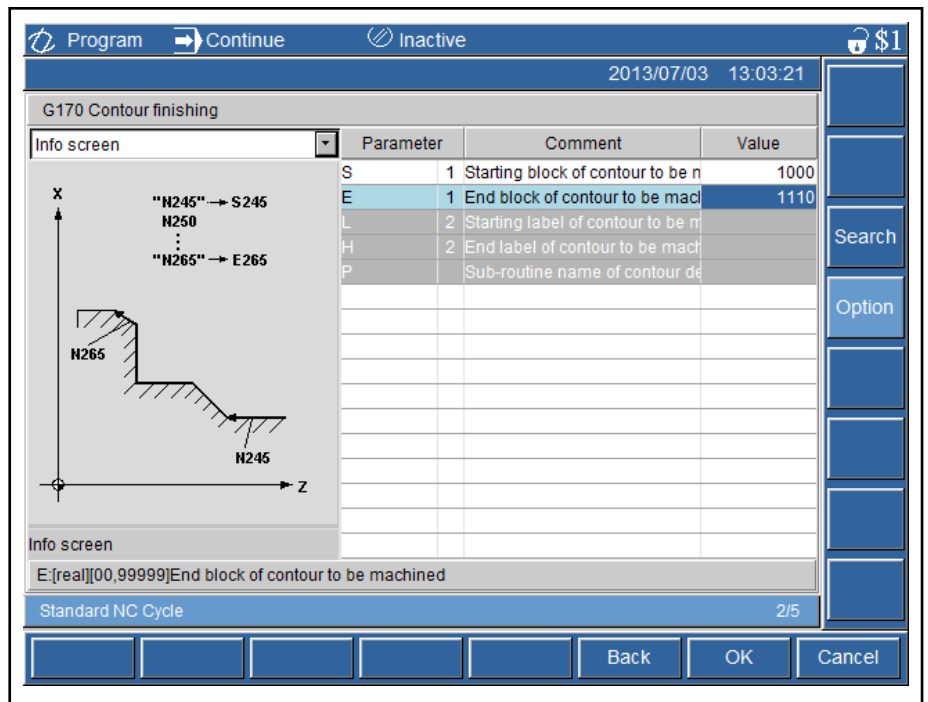


Fig. 3-24:

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

```
N110 G170 (S1000,E1110)
```

Disable the radius compensation and traverse a safe swivel position. Stop the spindle with M5 as the follow-on tool operates in the reverse direction of rotation.



**Note:** The contour cycle ends at the last point of the described contour draft. Ensure that collision-free approaching of the turret swiveling position is possible. In this case, add a corresponding intermediate position.

Insert the following NC blocks:

```
N120 G40 G0 X41 Z-50
N130 G76 G0 X250 Z600 M5
```



The positions X250 Z600 are used as an example. The positions at your machine may differ.

```

;-----
;OP1.3 Finishing external countour
N90 G501(T2,F0.08,S250,DIR M4,SMX3000,ZG G54,TCH 0)
N100 G42
N110 G170(S1000,E1110)
N120 G40 G0 X41 Z-50
N130 G76 G0 X250 Z600 M5
N990 M30
;----- program end -----
    
```

Fig. 3-25:

NC Program External Machining

### 3.2.6 OP1.4 - Thread Cutting with Cycle G167

#### General Information

During the next machining operation, the thread M16x1 is cut to the shaft end using the thread turning tool T4.

Enter a comment line for this machining operation:

```
;------  
;OP1.4 Cutting external thread M16x1
```

#### Step 1.8 - Tool Change T4

##### Tool change T4 and technology

Execute tool change using the T,F,S cycle.

The contour is turned with the tool T4 using the following parameters:

T	4	Tool, turning tool, external
F1	1	Finishing feed 1 rpm
S	1200	Spindle speed 1/min
M	3	Direction of rotation, clockwise
CS	G97	Direct speed programming
ZG	G54	Workpiece zero point G54
ZB	1	Zero point bank 1
TCH	0	Tool change positioning 0=No

Tab.3-26:

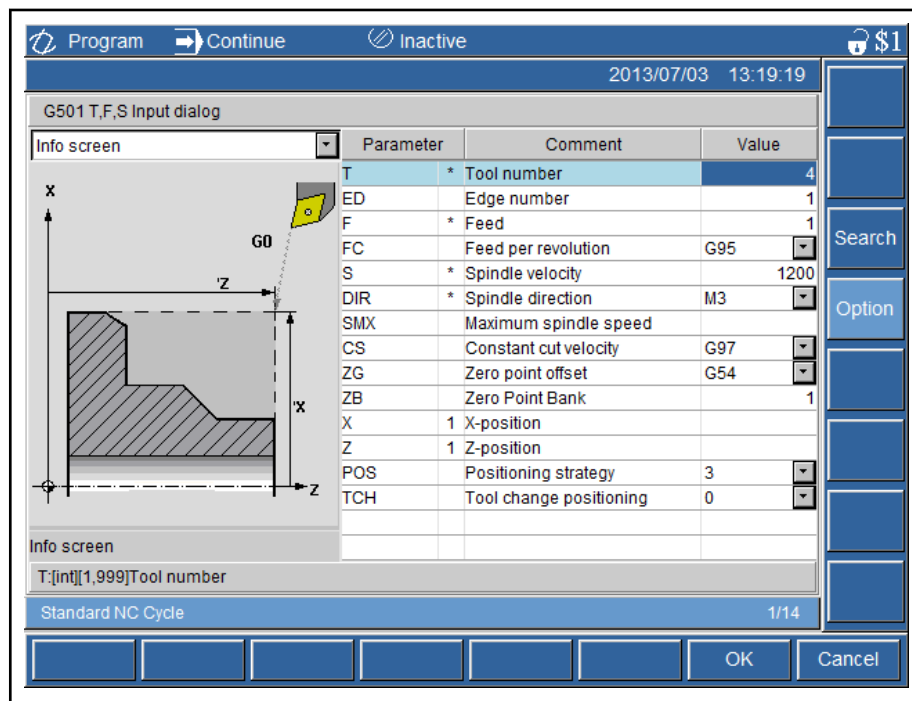


Fig.3-27:

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

```
N140 G501(T4, F1, S1200, DIR M3, CS G97, ZG G54, TCH0)
```

### Step 1.9 - Cutting the External Thread M16x1

Entering technology data for the thread turning cycle

The detailed allowance of the metric ISO fine thread has been taken from DIN 13, page 2.

Select cycle "3" / G167 in the cycle group "3".

Specify the parameters for the machining cycle G196 in the cycle input mask:

SR	16	Starting point, radial
SA	3	Starting point, axial
EA	-13.5	End point, axial
TP	1	Thread pitch
DT	-0.613	Thread depth
NI	6	Number of cuts
ROF	0	Retracting angle

Tab.3-28:

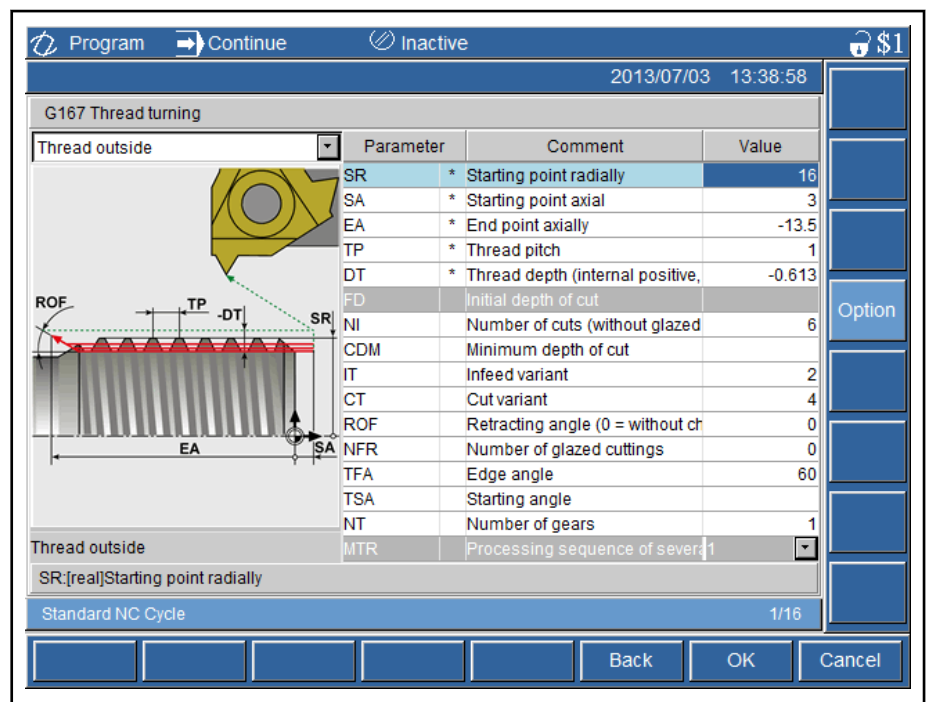


Fig.3-29:

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

```
N150 G167(SR16,SA5,EA-13.5,TP1,DT-0.613,NI6,ROF0)
```

### Step 1.10 - Traversing the Position for Unclamping

Traversing the position for unclamping

A fixed machine position is traversed and the spindle is stopped before the end of the program is reached. Enter a comment line and the N160 block.

```
N160 G76 X250 Z600 M5
```



The positions X250 Z600 are used as an example. The positions at your machine may differ.

## NC Program External Machining

```

;-----
;OP1.4 Cutting external thread M16x1
N140 G501(T4,F1,S1200,DIR M3,CS G97,ZG G54,TCH 0)
N150 G167(SR16,SA3,EA-13.5,TP1,DT-0.613,NI6,ROF0)
;-----
N160 G76 G0 X250 Z600 M5
N990 M30
;----- program end -----

```

Fig.3-30:

Saving the NC program Save the program with "Save" before exiting the editor.

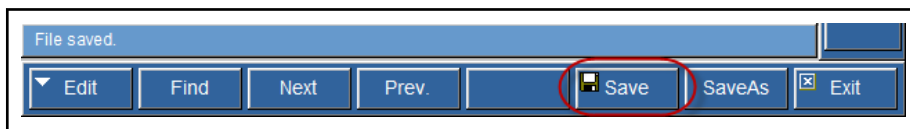


Fig.3-31:

### 3.3 Complete Program "SHAFT\_OP1"

#### Program listing SHAFT\_OP1

```

;Lathe machine:   MTX_micro
;NC program name: SHAFT_OP1.npg      Drawing:       SHAFT_OP1.dxf
;Material:        Aluminium Stock   Dimension:     D40mm x 82mm
;Author:          Turner              Version:      1.0.0
;Date:            26.06.2013
;----- program start -----
;OP1.1 Face milling with allowance
N10 G76 X250 Z600
N20 G501(T1,F0.1,S250,DIR M4,SMX3000,ZG G54,TCH 0)
N30 G0 X45 Z0
N40 N40 G1 X-1
N50 G1 Z0.5
N60 G0 X41
;-----
;OP1.2 Roughing of external contour with roughing cycle G171
N70 G171(S1000,E1110,CD1,LD1,CR0.1,CA0.1,UCV3)
N80 G76 G0 X250 Z600
;-----
;OP1.3 Finishing external contour
N90 G501(T2,F0.08,S250,DIR M4,SMX3000,ZG G54,TCH 0)
N100 G42
N110 G170(S1000,E1110)
N120 G40 G0 X41 Z-50
N130 G76 G0 X250 Z600 M5
;-----
;OP1.4 Cutting external thread M16x1
N140 G501(T4,F1,S1200,DIR M3,CS G97,ZG G54,TCH 0)
N150 G167(SR16,SA3,EA-13.5,TP1,DT-0.613,NI6,ROF0)
;-----
N160 G76 G0 X250 Z600 M5
N990 M30
;----- program end -----
// description external contour
// begin
N1000 G0 X10 Z1
N1010 G1 X16 Z-2
N1020 G196(ORI3,ART 1,FORM A, SA-15,TP1)
N1030 G1 X20 CHLL(1)
N1040 G1 Z-20
N1050 G1 X25 Z-25
N1060 G1 X30 RNDL(2.5)
N1070 G1 Z-35 RNDL(1)
N1080 G1 X35 RNDL(1)
N1090 G1 Z-45
N1100 G1 X39 CHLL(1)
N1110 G1 Z-51
// end

```

### 3.4 Simulating the NC Program "SHAFT\_OP1"

#### 3.4.1 Entering the Zero Point Offset

The machine zero point usually lies in the center of the workpiece spindle for turning machines. A zero point offset for the workpiece in the x-axis is not required. The zero point of the z-axis is usually positioned on the plane surface of the workpiece.

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 chuck into consideration:

1. Clamping: G54.1 Z = 214.5 mm

Clamping sketch, 1st side

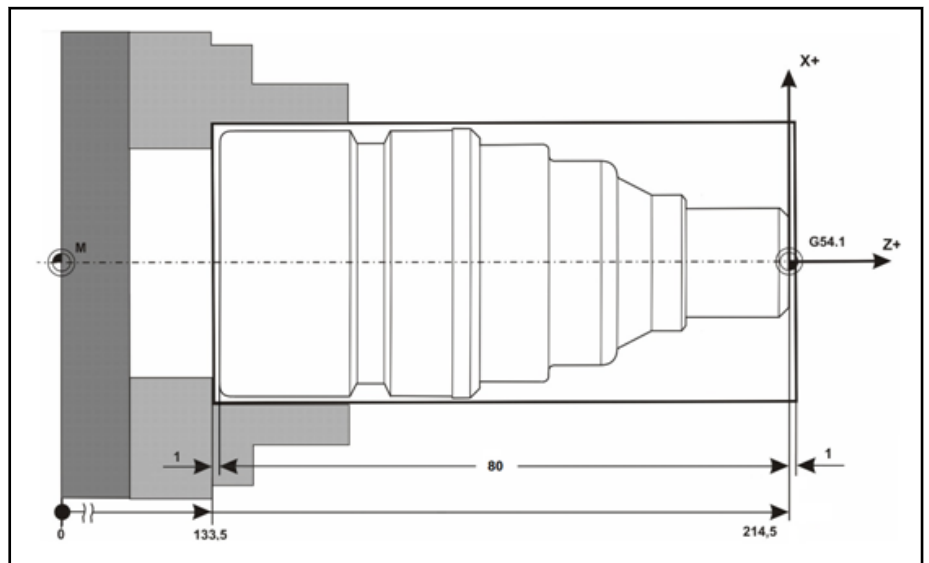


Fig.3-32: Clamping sketch OP1

2. Clamping: G55.1 Z = 189.4 mm

Clamping sketch, 2nd side

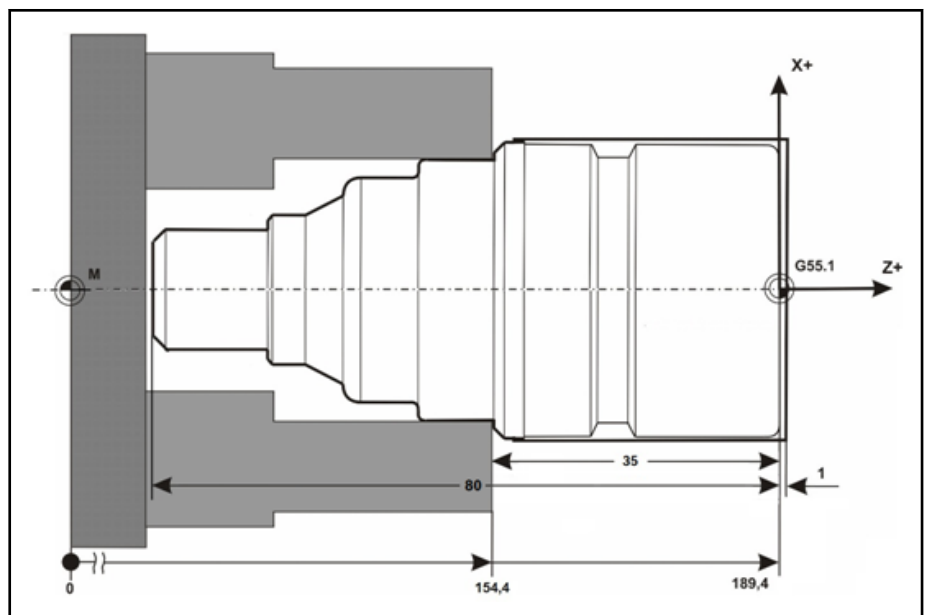


Fig.3-33: Clamping sketch OP2

Changing to "Machine" operating area

Press the key "Machine" at the control panel.

## NC Program External Machining

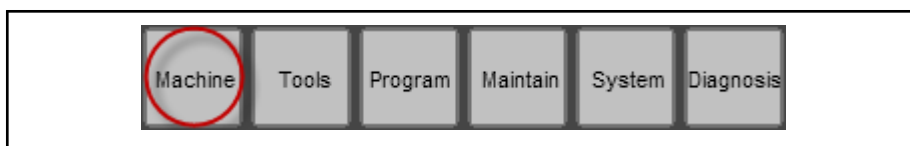


Fig. 3-34:

Selecting the zero point table and enter the offsets

Press <F6>"ZeroOffs".

Select the input field for G54.1 using the cursor keys. Enter the value 0 in the column X and the value 214.5 in the column Z.

Repeat the operation for G55.1 using the values X0 and Z189.4

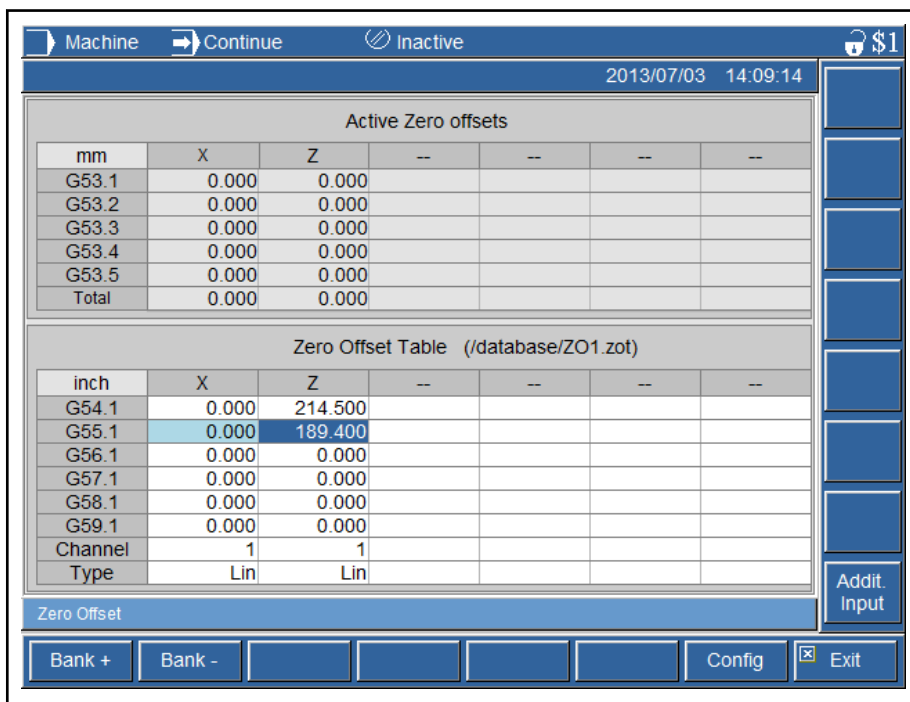


Fig. 3-35:

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 "SHAFT\_OP1" can be simulated to check for programming and input errors. The required operation of the IndraMotion MTX micro Trainer is described as follows:

Changing to the "Machine" operating area

Press the key "Machine" on the control panel:

NC Program External Machining

Changing to "Automatic" operation mode

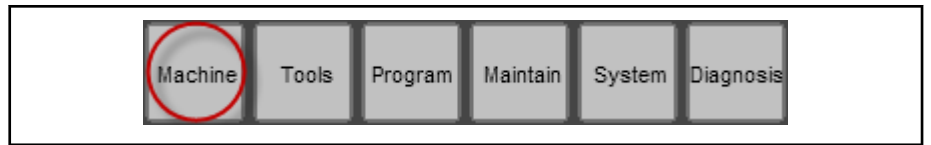


Fig. 3-36:

Press the key "Automatic" key on the control panel:



Fig. 3-37:

Selecting program

Press <F2>"SelProg":

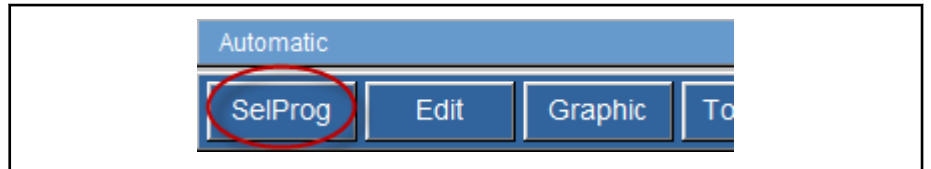


Fig. 3-38:

Select the program "SHAFT\_OP1" in the "prog" directory:



**Note:** Navigate using the <TAB> key as well as the cursor keys <up> and <down>.

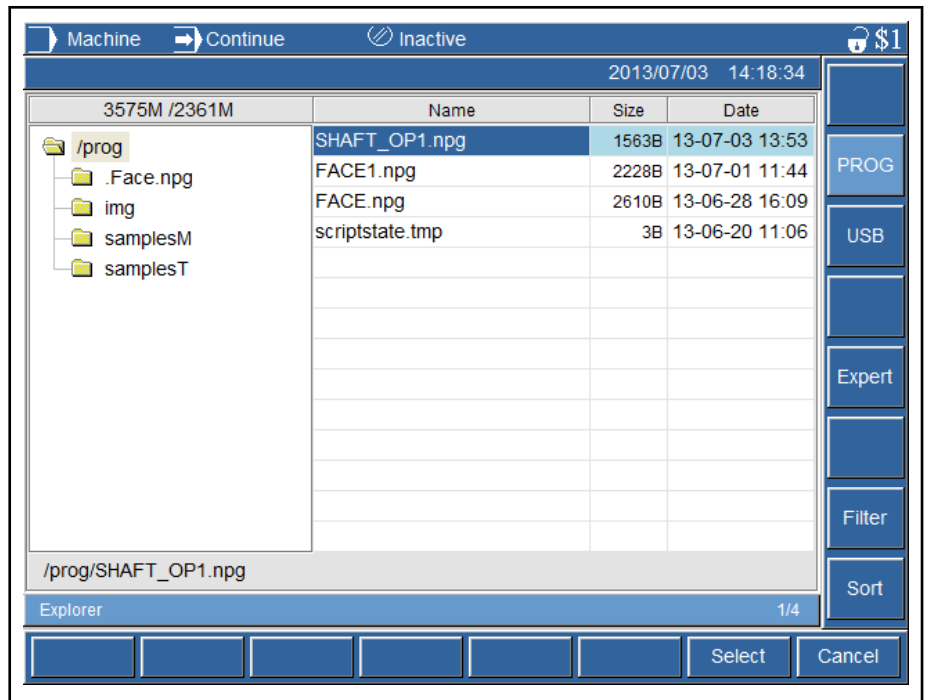


Fig. 3-39:

NC Program External Machining

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

**Start program**

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



Fig.3-40:

**Setting the graphic display**

Press <F4>"Graphic", the graphic display opens. Press <F5> "Rear" to swivel the coordinate system in the G18 plane - Machining behind the turning center.

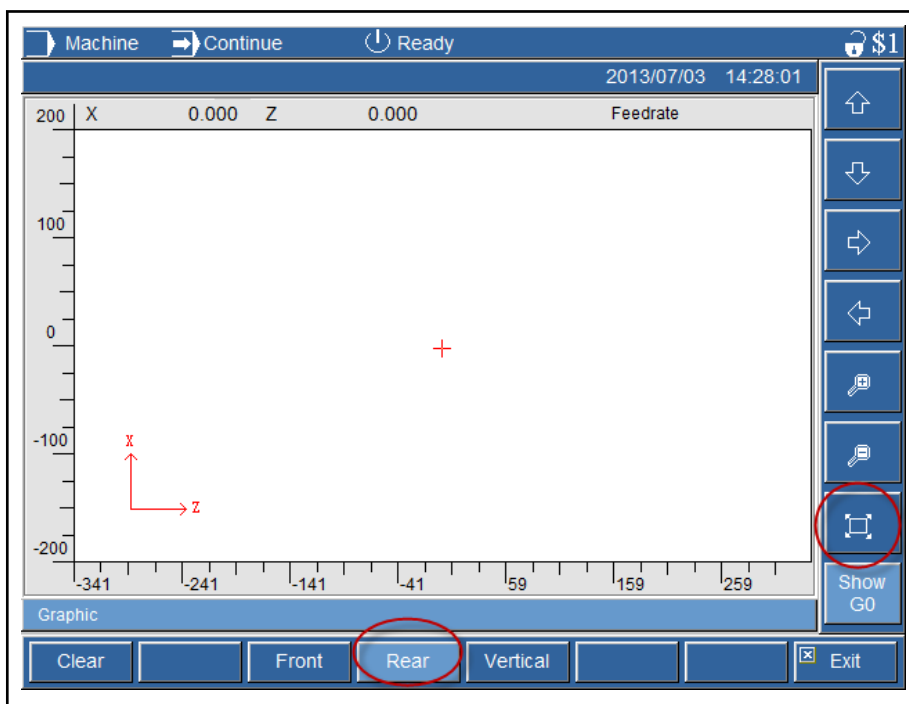


Fig.3-41:

**Start simulation**

Press the key "Start" at the control panel.

## NC Program External Machining

While the graphic is drawn, enable "Autozoom" with <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-Key 8> "Show G0", the rapid traverse motions (highlighted in red) can be shown or hidden.

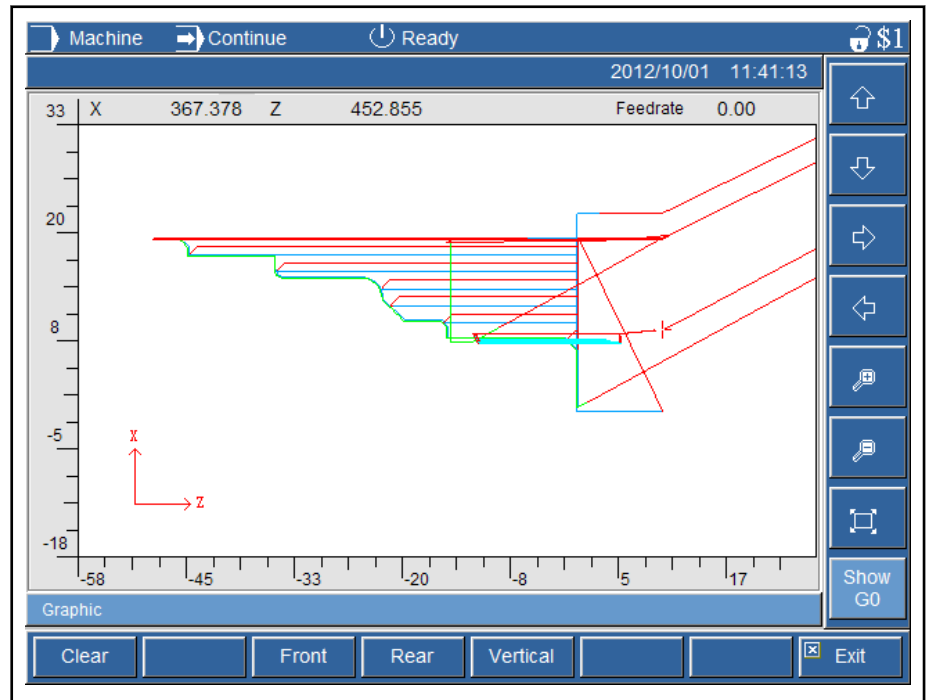


Fig.3-42:



## 4 NC Program Internal Machining

### 4.1 Creating the NC Program "SHAFT\_OP2"

#### 4.1.1 General Information

Program flow chart Dividing machining in programming steps

OP_No.	Step	Program 2nd side	Note	To ol
	2.1	Generate program header		
OP2.1	2.2	Tool change, technology input		1
	2.3	Face turning with allowance	0.1mm	
OP2.2	2.4	Tool change, technology input		6
	2.5	Rough-drilling D12	Z-30	
OP2.3	2.6	Tool change, technology input		
	2.7	Internal roughing	G171	
OP2.4	2.8	Internal contour finishing		
OP2.5	2.9	Tool change, technology input		1
	2.10	Finish turning the external contour		
OP2.6	2.11	Tool change, technology input		3
	2.12	Groove turning	G191	
OP2.7	2.13	Tool change, technology input		8
	2.14	Internal thread M30x1,5 cutting	G167	
	2.15	Approach position for unclamping		

Tab.4-1:

### 4.1.2 Creating the Second NC Program "SHAFT\_OP2.npg"

#### General Information

To machine the second side of the workpiece, reclamp the workpiece. For this side, a new NC program is created under the name of e.g. "SHAFT\_OP2.npg".

#### Creating the program

- Change to the "Program" machine operating area
- Press <F2>"New"
- Press <F3>"NCProg" to open the editor for a new NC program
- Press <F7>"Save"
- Enter the program name "SHAFT\_OP2.npg" in the input field

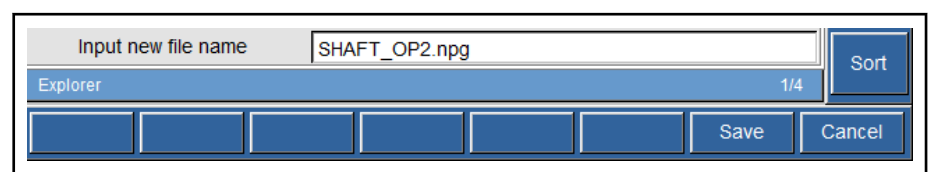


Fig.4-2:

- Complete the input with <ENTER> or <F8> "Save".

## NC Program Internal Machining

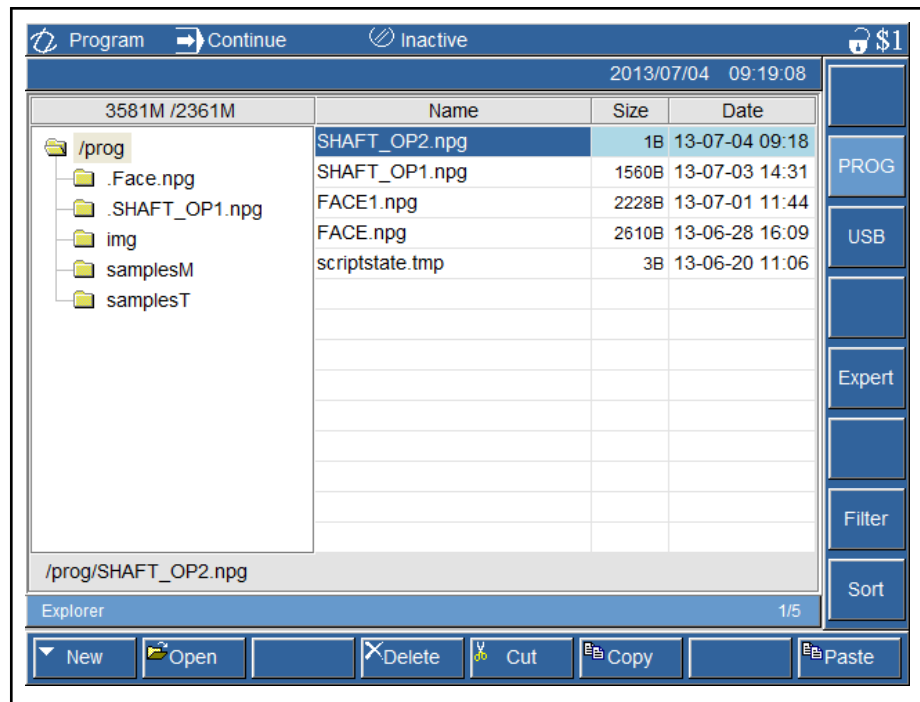


Fig.4-3:

The program that is still empty is now saved under the name "SHAFT\_OP2.npg" in the "/prog" directory.

## Step 2.1 - Creating the Program Header

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

*Program header*

```
;Millingmachine: MTX_micro
;NC program name: SHAFT_OP2.npg Drawing: SHAFT_OP2.dxf
;Material: Aluminium Stock Dimension: D40mm x 81mm
;Author: Turner Version: 1.0.0 Date: 24.02.2013
```

**Program start and end** Enter the following comment lines to structure the program:

*Program:*

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

**Contour description** *Program:*

```
// description inside contour
// begin
N1000
N
// end
;-----
```

## 4.1.3 OP2.1 - Face Turning with Allowance

## General Information

The end face is face milled with the roughing tool T1 during the first machining operation. Enter the comment for the first machining operation in the empty comment line after the "program start" comment.

```
;OP2.1 Face milling with allowance
```

NC Program Internal Machining

Program a traversing motion to a safe, machine-based position (G76) before swivelling the turret.

Position the cursor after "N10" and enter the following NC block:

```
N10 G76 G0 X250 Z600
```



The positions X250 Z600 are used as an example. The positions at your machine may differ.

Press <ENTER> to enter the next NC block.

### Step 2.2 - Tool Change T1

Tool change T1 and technology

Specify all conditions for subsequent machining in the input mask:

T	1	Tool, roughing tool, external
F	0.1	Roughing feed 0.1 mm/rev
S	250	Spindle velocity 250 m/min
M	4	Direction of rotation, counterclockwise
SMX	3000	Maximum spindle speed 1/min
ZG	G55	Workpiece zero point G55
ZB	1	Zero point bank 1
TCH	0	Tool change positioning 0=No

Tab.4-4:

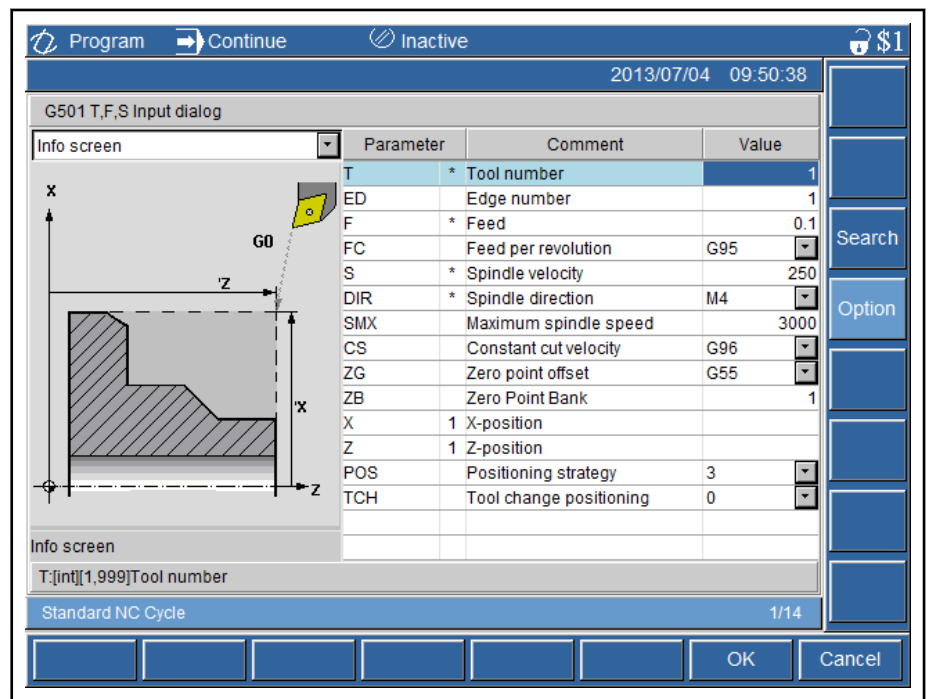


Fig.4-5:

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

```
N20 G501(T1,F0.1,S250,DIR M4,SMX3000,ZG G55, TCH0)
```

### Step 2.3 - Face Turning

Insert the following NC blocks:

## NC Program Internal Machining

**Pre-positioning** N30 G0 X45 Z0.1  
**Face turning** N40 G1 X-1  
**Retracting** N50 Z0.2

Program a traversing motion to a safe, machine-based position (G76) before swivelling the turret:

N60 G76 G0 X250 Z600 M5



The positions X250 Z600 are used as an example. The positions at your machine may differ.

```

;----- program start -----
;OP2.1 Face milling with allowance
N10 G76 G0 X250 Z600
N20 G501(T1,F0.1,S250,DIR M4,SMX3000,ZG G55,TCH 0)
N30 G0 X45 Z0.1
N40 G1 X-1
N50 Z0.2
N60 G76 G0 X250 Z600 M5
N990 M30
;----- program end -----

```

Fig.4-6:

## 4.1.4 OP2.2 – Internal Boring D12

### General Information

During the next machining operation, the diameter 12 mm is drilled to prepare for internal machining. Enter a comment for this machining operation:

```

;-----
;OP2.2 Internal boring D12

```

### Step 2.4 - Tool Change T5

#### Tool change T5 and technology

Enter the NC block number "N70". Press <M-Key 1> "T,F,S" to open the input mask for the tool change cycle:

**Specify all conditions for subsequent machining in the input mask:**

T	6	Tool, roughing tool, external
F	0.07	Boring feed 0.07mm/rev
S	1000	Spindle speed 1/min
M	3	Direction of rotation, clockwise
CS	G97	Constant spindle speed
ZG	G55	Workpiece zero point G55
ZB	1	Zero point bank 1
TCH	0	Tool change positioning 0=No

Tab.4-7:

```
N70 G501(T6, F0.07, S1000, DIR M3, CS G97, ZG G55, TCH0)
```

### Step 2.5 – Internal Boring D12

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

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

Parameter	Value	Description
IX	Z	Drill axis
SL	1	Safety height
DT	-30	Drilling depth
DW	0.5	Dwell time
RL	1	Retracting height

Tab.4-8:

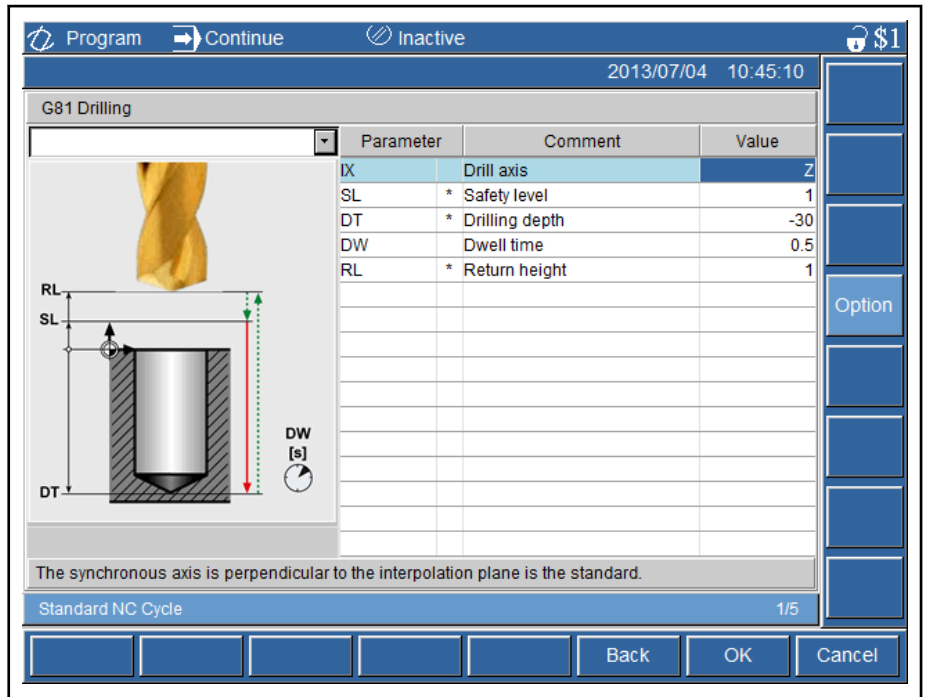


Fig.4-9:

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

```
N80 G81 ( IX Z , SL1 , DT-30 , DW0.5 , RL1 )
```

**Boring cycle, modal**

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

```

Drilling N90 G0 X0
Deleting modal mode of boring cycle N100 G80
    
```

Program a traversing motion to a safe, machine-based position (G76) before swivelling the turret:

```
N110 G0 G76 X250 Z600 M5
```



The positions X250 Z600 are used as an example. The positions at your machine may differ.

## NC Program Internal Machining

```

;-----
;OP2.2 Internal boring D12
N70 G501(T6,F0.07,S1000,DIR M3,CS G97,ZG G55,TCH 0)
N80 G81(IX Z,SL1,DT-30,DW0.5,RL1)
N90 G0 X0
N100 G80
N110 G0 G76 X250 Z600 M5
N990 M30
;----- program end -----

```

Fig.4-10:

## 4.1.5 OP2.3 - Roughing Internal Contour with Allowance

## General Information

During the third machining operation, the internal contour is roughened with cutting cycle G160.

Enter a comment for this machining operation:

```

;-----
;OP2.3 Roughing of internal contour with allowance

```

## Step 2.6 - Tool Change T7

## Tool change T7 and technology

Enter the NC block number 120. Press <M-Key 1>"T,F,S" to open the input mask for the tool change cycle.

**Specify all conditions for subsequent machining in the input mask:**

T	7	Tool, roughing tool, internal
F	0.06	Boring feed 0.06 mm/rev
S	250	Spindle velocity 250 m/min
M	4	Direction of rotation, counterclockwise
SMX	3000	Maximum spindle speed 1/min
ZG	G55	Workpiece zero point G55
ZB	1	Zero point bank
TCH	0	Tool change positioning 0=No

Tab.4-11:

```

N120 G501(T7,F0.06,S250,DIR M4,SMX3000,ZG G55, TCH 0)

```

## Step 2.7 - Roughing of Internal Contour with Allowance

## Specify referencing position

Insert the following NC blocks:

```

N130 G0 X10 Z1

```

Insert the description of the internal contour at the program end:

*Internal contour*

*Program:*

```

N1000 G41 G0 X32.917 Z0.5
N1010 G1 X28.917 Z-1.5
N1020 G1 Z-25
N1300 G1 X12
N1500 G40 (NOM)

```



Using the G40 (NOM) command, the radius compensation is disabled without a traversing motion.

```
// description inside contour
// begin
N1000 G41 G0 X32.917 Z0.5
N1010 G1 X28.917 Z-1.5
N1020 G1 Z-25
N1300 G1 X12
N1500 G40 (NOM)
// end
;-----
```

Fig.4-12:

Again navigate to a position after the NC block "N130" and enter the NC block "N140".

G171 Contour roughing

**Specify the parameters for the machining cycle G171 in the cycle input mask:**

S	1000	Starting point of the contour description
E	1500	End point of the contour description
CD	2	Maximum depth of cut
LD	1	Retract amount after cut
CR	0.1	Longitudinal allowance
CA	0.1	Plane allowance

Tab.4-13:

```
N140 G171(S1000,E1500,CD2,LD1,CR0.1,CA0.1)
```

```
;-----
;OP2.3 Roughing of internal contour with allowance
N120 G501(T7,F0.06,S250,DIR M4,SMX3000,ZG G55,TCH 0)
N130 G0 X10 Z1
N140 G171(S1000,E1500,CD2,LD1,CR0.1,CA0.1)
N990 M30
;----- program end -----
```

Fig.4-14:

## 4.1.6 OP2.4 - Finishing of Internal Contour

### General Information

Use the same tool to finish the internal contour with the radius compensation in this machining operation. Enter a comment for this machining operation:

```
;-----
;OP2.4 Internal contour finishing
```

### Step 2.8 - Internal Contour Finishing

**Pre-positioning** Prepositioning to enable the radius compensation

```
N150 G0 X30 Z1
```

## NC Program Internal Machining

Specify the parameters for the machining cycle G170 in the cycle input mask:

S	1000	Starting block of the contour description
E	1500	End block of the contour description

Tab.4-15:

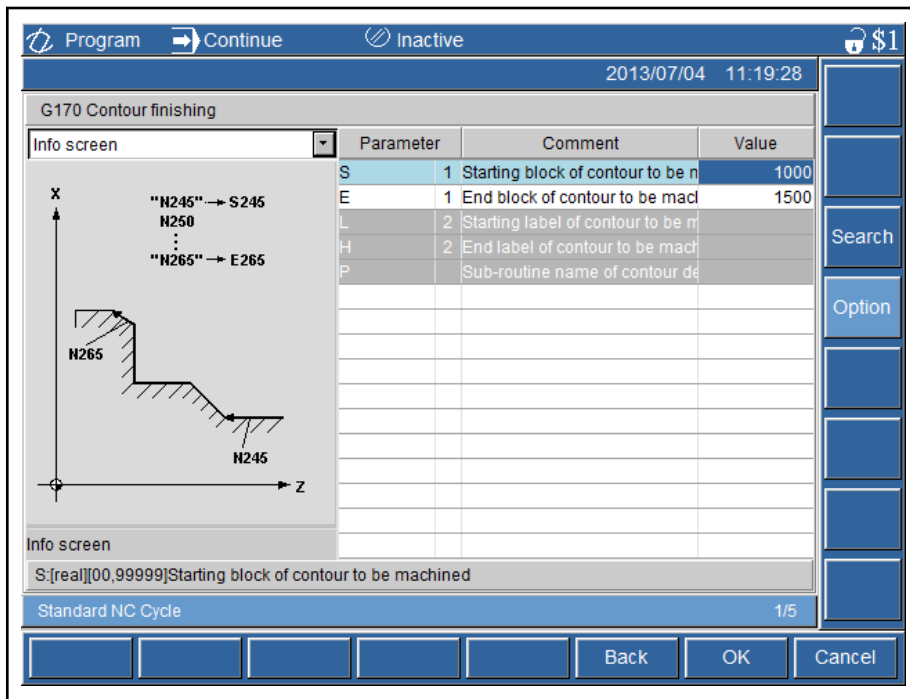


Fig.4-16:

N160 G170(S1000,E1500)

**Retracting** N170 G0 Z1

Program a traversing motion to a safe, machine-based position (G76) before swivelling the turret.

N180 G0 G76 X250 Z600



The positions X250 Z600 are used as an example. The positions at your machine may differ.

```

;-----
;OP2.4 Finishing of internal contour
N150 G0 X30 Z1
N160 G170(S1000,E1500)
N170 G0 Z1
N180 G0 G76 X250 Z600
N990 M30
;----- program end -----

```

Fig.4-17:

## 4.1.7 OP2.5 - Plane Surface and External Contour Finishing

### General Information

After roughing of the contour has been completed, the external contour is finished. Tool T1 is swiveled in. Enter a comment line for this machining operation:

```
;-----  
;OP2.5 Plane surface and external contour finishing
```

### Step 2.9 - Tool Change T1

Tool change T1 and technology

Specify all conditions for subsequent machining in the input mask:

T	1	Tool, roughing tool, external
F	0.1	Roughing feed 0.1 mm/rev
S	250	Spindle velocity 250 m/min
M	4	Direction of rotation, counterclockwise
SMX	3000	Maximum spindle speed 1/min
ZG	G55	Workpiece zero point G55
ZB	1	Zero point bank 1
TCH	0	Tool change positioning 0=No

*Tab.4-18:*

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".

```
N190 G501(T1,F0.1,S250,DIR M4,SMX3000,ZG G55,TCH 0)
```

### Step 2.10 - Plane Surface and External Contour Finishing

Contour with radius compensation

Insert the following NC blocks:

*Program:*

```
N200 G42 G0 X28 Z1  
N210 G1 Z0  
N220 G1 X38 RNDL(2)  
N230 G1 Z-30  
N240 G1 X40  
N250 G40 G0 G76 X250 Z600
```



The positions X250 Z600 are used as an example. The positions at your machine may differ.

## NC Program Internal Machining

```

;-----
;OP2.5 Finishing of plane surface and external contour
N190 G501(T1,F0.1,S250,DIR M4,SMX3000,ZG G55,TCH 0)
N200 G42 G0 X28 Z1
N210 G1 Z0
N220 G1 X38 RNDL(2)
N230 G1 Z-30
N240 G1 X40
N250 G40 G0 G76 X250 Z600
N990 M30
;----- program end -----

```

Fig.4-19:

## 4.1.8 OP2.6 - Groove External Turning

### General Information

The groove is turned with cycle G191 in this machining operation. Enter a comment for this machining operation:

```

;-----
;OP2.6 External recess turning

```

### Step 2.11 - Tool Change T3

Tool change T3 and technology

Execute tool change using the T,F,S cycle. The groove is turned with the tool T3 using the following parameters:

T	3	Tool, grooving tool
F	0.05	Feed 0.05 rpm
S	280	Spindle velocity 180 m/min
M	4	Direction of rotation, counterclockwise
SMX	3000	Maximum spindle speed 1/min
ZG	G55	Zero offset
ZB	1	Zero point bank
TCH	0	Tool change positioning 0=No

Tab.4-20:

```
N260 G501(T3, F0.05, S250, DIR M4, SMX3000, ZG G55, TCH 0)
```

### Step 2.12 - External Recess Turning

Specify referencing position

Insert the following NC blocks:

```
N270 G0 X40 Z-19
```

Open the cycle input mask group "4"Cycle "1" / G191 in the next NC block.

Entering technology data for the grooving cycle



This cycle uses the active edge. The tool width is contained in parameter TW!

Specify the parameters for the machining cycle G191 in the cycle input mask:

ORI	8	Groove position
SR	38	Starting point, radial

SA	-15	Starting point, axial
DT	2	Groove depth
W	4	Width of basic straight line
TW	3	Tool width
A1	30	Edge angle at starting point
A2	30	Edge angle at end point

Tab.4-21:

Fig.4-22:

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

```
N280 G191 (SR38 , SA-15 , DT2 , W4 , TW3 , A1=30 , A2=30)
```

Program a traversing motion to a safe, machine-based position (G76) before swivelling the turret:

```
N290 G76 G0 X250 Z600 M5
```



The positions X250 Z600 are used as an example. The positions at your machine may differ.

```

;-----
;OP2.6 Turning of external recess
N260 G501 (T3 , F0.05 , S180 , DIR M4 , SMX3000 , ZG G55 , TCH 0)
N270 G0 X40 Z-19
N280 G191 (SR38 , SA-15 , DT2 , W4 , TW3 , A1=30 , A2=30)
N290 G76 G0 X250 Z600 M5
N990 M30
;----- program end -----

```

Fig.4-23:

NC Program Internal Machining

## 4.1.9 OP2.7 - Internal Thread M30x1.5 Edges

### General Information

An internal thread M30x1.5 is cut using the thread turning tool T8 at the end of the shaft.

Enter a comment for this machining operation:

```
;-----  
;OP2.7 Internal thread M30x1.5 edges
```

### Step 2.13 - Tool Change T8

#### Tool change T8 and technology

Execute tool change with <M-Key 1>"T,F,S" cycle.

**Tool T8 and the following parameters are used for machining:**

T	8	Tool, turning tool, internal
F	1.5	Finishing feed 1 mm/rev
S	1200	Spindle speed 1/min
M	3	Direction of rotation, clockwise
CS	G97	Direct speed programming
ZG	G55	Zero point offset G55
ZB	1	Zero point bank 1
TCH	0	Tool change positioning 0=No

Tab.4-24:

```
N300 G501(T8,F1.5,S1200,DIR M3,CS G97,ZG G55,TCH 0)
```

### Step 2.14 - Internal Thread M30x1.5 Edges

The detailed allowance of the metric ISO fine thread has been taken from DIN 13, page 2.

The run-out 60° has to be taken into consideration when specifying the axial end point and has to be added to the usable thread length.

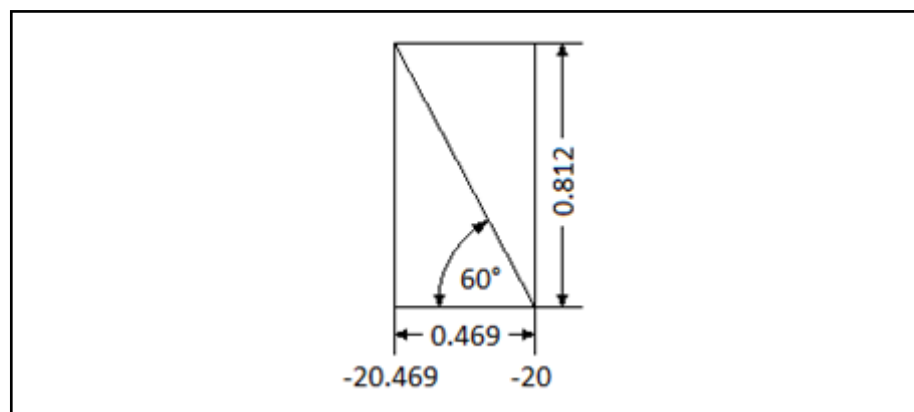


Fig.4-25: Taking the thread run-out into consideration

Cycle G167 With automatic run-out 60°

Specify the parameters for the machining cycle G167 thread cutting in the cycle input mask:

SR	28.38	Starting point, radial
SA	3	Starting point, axial
EA	-20.469	End point, axial
TP	1.5	Thread pitch
DT	0.812	Thread depth
FD	0.4	First cutting depth
IT	1	Infeed variant
CT	2	Cutting variant
NFR	1	Number of glazed cuts

Tab.4-26:

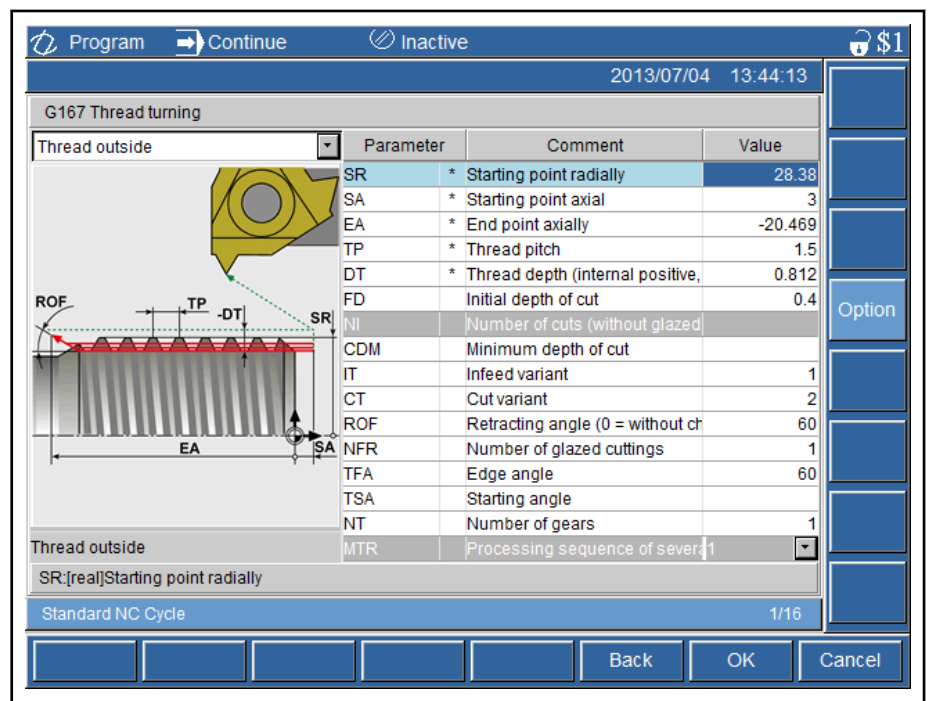


Fig.4-27:

N310 G167 (SR28.38, SA3, EA-20.469, TP1.5, DT0.812, FD0.4, IT1, CT2, NFR1)

```

;-----
;OP2.7 Internal thread M30x1.5 edges
N300 G501(T8,F1.5,S1200,DIR M3,ZG G55,TCH 0)
N310 G167(SR28.38,SA3,EA-20.469,TP1.5,DT0.812,FD0.4,IT1,CT2,NFR1)
N990 M30
;----- program end -----
    
```

Fig.4-28:

## NC Program Internal Machining

**Step 2.15 – Approaching Position for Unclamping****Traversing the position for unclamping**

A fixed machine position is approached and the the zero point offset is deselected before the end of the program is reached. Insert the block N320:

```
N320 G0 G76 X250 Z500 M5
```



The positions X250 Z600 are used as an example. The positions at your machine may differ.

```
N320 G0 G76 X250 Z500 M5
N990 M30
;----- program end -----
```

Fig.4-29:

**Saving the NC program**

Save the program with <F8>"Save" before exiting the editor.

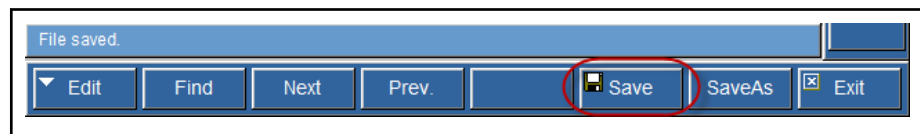


Fig.4-30:

Close the editor with <F9>"Exit"

**4.2 Complete Program "SHAFT\_OP2"***Program listing SHAFT\_OP2*

```
;Lathe machine: MTX_micro
;NC program name: SHAFT_OP2.npg Drawing: SHAFT_OP2.dxf
;Material: Aluminium Stock Dimension: D40mm x 81mm
;Author: Turner Version: 1.0.0 Date: 22.11.2012
;----- program start -----
;OP2.1 Face milling with allowance
N10 G76 G0 X250 Z600
N20 G501(T1,F0.1,S250,DIR M4,SMX3000,ZG G55,TCH 0)
N30 G0 X45 Z0.1
N40 G1 X-1
N50 Z0.2
N60 G76 G0 X250 Z600 M5
;-----
;OP2.2 Internal boring D12
N70 G501(T6,F0.07,S1000,DIR M3,CS G97,ZG G55,TCH 0)
N80 G81(IX Z,SL1,DT-30,DW0.5,RL1)
N90 G0 X0
N100 G80
N110 G0 G76 X250 Z600 M5
;-----
;OP2.3 Roughing of internal contour with allowance
N120 G501(T7,F0.06,S250,DIR M4,SMX3000,ZG G55,TCH 0)
N130 G0 X10 Z1
N140 G171(S1000,E1500,CD2,LD1,CR0.1,CA0.1)
;-----
;OP2.4 Finishing of internal contour
N150 G0 X30 Z1
N160 G170(S1000,E1500)
N170 G0 Z1
N180 G0 G76 X250 Z600
;-----
;OP2.5 Finishing of plane surface and external contour
N190 G501(T1,F0.1,S250,DIR M4,SMX3000,ZG G55,TCH 0)
N200 G42 G0 X28 Z1
N210 G1 Z0
N220 G1 X38 RNDL(2)
N230 G1 Z-30
N240 G1 X40
N250 G40 G0 G76 X250 Z600
;-----
;OP2.6 Turning of external recess
N260 G501(T3,F0.05,S180,DIR M4,SMX3000,ZG G55,TCH 0)
N270 G0 X40 Z-19
```

## NC Program Internal Machining

```

N280 G191(SR38,SA-15,DT2,W4,TW3,A1=30,A2=30)
N290 G76 G0 X250 Z600 M5
;-----
;OP2.7 Internal thread M30x1.5 edges
N300 G501(T8,F1.5,S1200,DIR M3,ZG G55,TCH 0)
N310 G167(SR28.38,SA3,EA-20.469,TP1.5,DT0.812,FD0.4,IT1,CT2,NFR1)
N320 G0 G76 X250 Z500 M5
N990 M30
;----- program end -----
// description inside contour
// begin
N1000 G41 G0 X32.917 Z0.5
N1010 G1 X28.917 Z-1.5
N1020 G1 Z-25
N1300 G1 X12
N1500 G40 (NOM)
// end
;-----

```

### 4.3 Simulating the NC Program "SHAFT\_OP2"



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 "SHAFT\_OP2" can be simulated to check for programming and input errors. The required operation of the **IndraMotion MTX micro Trainer** has already been explained.

Simulation with G0 traversing motion display

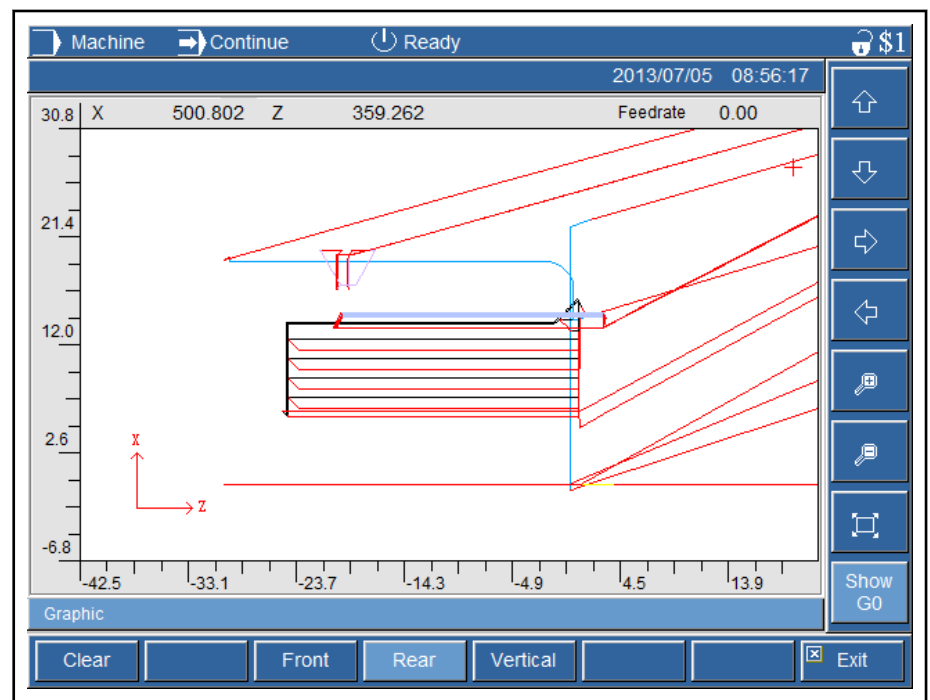


Fig.4-31:

NC Program Internal Machining

Simulation without G0 traversing  
motion display

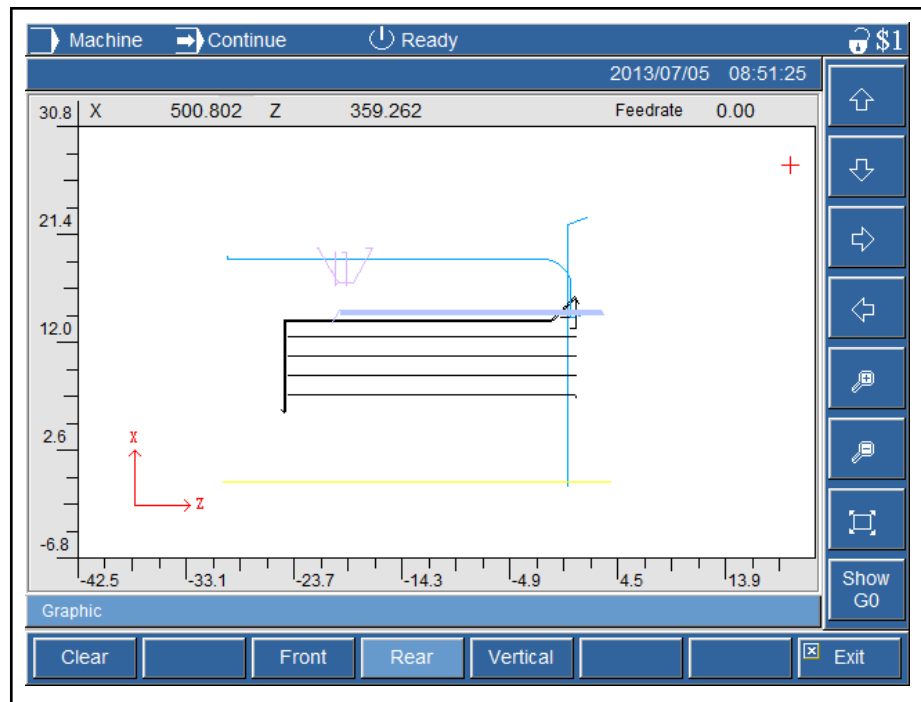


Fig.4-32:

## 5 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:	<b>+49 9352 40 5060</b>
Fax:	<b>+49 9352 18 4941</b>
E-mail:	<a href="mailto:service.svc@boschrexroth.de">service.svc@boschrexroth.de</a>
Internet:	<a href="http://www.boschrexroth.com">http://www.boschrexroth.com</a>

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

## A

Abbreviations.....	7
About this documentation.....	3
Information representation.....	6
Validity of the documentation.....	3

## C

Creating the first NC program SHAFT_OP1.npg	
General information.....	19
Step 1.1 - Creating the program header.....	20
Creating the NC program SHAFT_OP1	
Creating the first NC program	
SHAFT_OP1.npg.....	19
General information.....	19, 43
OP1.1 - Face turning with allowance.....	21
OP1.3 - External contour finishing.....	32
Step 1.7 - External contour finishing with	
cycle G170.....	32
Creating the NC Program SHAFT_OP1	
OP1.4 - Thread cutting with cycle G167.....	34
Creating the NC program SHAFT_OP2	
Creating the second NC program	
SHAFT_OP2.npg .....	43
OP2.1 - Face turning with allowance.....	44
OP2.2 - Internal boring D12.....	46
OP2.3 - Roughing internal contour with	
allowance.....	48
OP2.4 - Finishing of internal contour.....	49
OP2.5 - Plane surface and external con-	
tour finishing.....	51
OP2.6 - Groove external turning.....	52
Creating the NC program SHAFT_OP1	
OP1.2 - External contour roughing with	
roughing cycle G171.....	24
Step 1.6 - Tool change T2.....	32
Creating the NC program SHAFT_OP2	
OP2.7 - Internal thread M30x1.5 edges.....	54
Creating the second NC program SHAFT_OP2	
General information.....	43
Creating the second NC program SHAFT_OP2.npg	
Step 2.1 - Creating the program header.....	44

## I

Information representation	
Names and abbreviations.....	7
Safety instructions.....	6
Symbols used.....	7

## N

NC program external machining.....	19
Complete program SHAFT_OP1.....	36
Creating the NC program SHAFT_OP1.....	19
General information.....	19
Simulating the NC program SHAFT_OP1.....	37
NC program internal machining	

Complete program SHAFT_OP2.....	56
Creating the NC program SHAFT_OP2.....	43
Simulating the NC program SHAFT_OP2.....	57
NC Program internal machining.....	43

## O

OP1.1 - Face turning with allowance	
General information.....	21
Step 1.2 - Tool change T1.....	22
Step 1.3 - 1st side face turning.....	23
OP1.2 - External contour roughing with	
roughing cycle G171	
General information.....	24
Including the thread undercut DIN 76 A in	
the contour description.....	25
Step 1.4 - Describing the external ma-	
chined part contour.....	24
Step 1.5 - External contour roughing.....	29
OP1.4 - Thread cutting with cycle G167	
General information.....	34
Step 1.8 - Tool change T4.....	34
Step 1.9 - Cutting the External Thread M16x1	
Step 1.10 - Traversing the position for	
unclamping.....	35
OP2.1 - Face turning with allowance	
General information.....	44
Schritt 2.3 - Face turning.....	45
Step 2.2 - Tool change T1.....	45
OP2.2 - Internal boring D12	
General information.....	46
Step 2.4 - Tool change T5.....	46
Step 2.5 - Internal boring D12.....	47
OP2.3 - Roughing internal contour with allowance	
General information.....	48
OP2.3 - Roughing of internal contour with al-	
lowance	
Step 2.6 - Tool change T7.....	48
Step 2.7 - Roughing of internal contour	
with allowance.....	48
OP2.4 - Finishing of internal contour	
General information.....	49
OP2.4 - Internal contour finishing	
Step 2.8 - Internal contour finishing.....	49
OP2.5 - Plane surface and external contour	
finishing	
General information.....	51
Step 2.9 - Tool change T1.....	51
Step 2.10 - Plane surface and external	
contour finishing.....	51
OP2.6 - External recess turning	
Step 2.11 - Tool change T3.....	52
Step 2.12 - External recess turning.....	52
OP2.6 - Groove external turning	
General information.....	52
OP2.7 - Internal thread M30x1.5 edges	

## Index

General information.....	54
Step 2.14 - Internal thread M30x1.5 edges....	54
Step 2.13 - Tool change T8.....	54
Step 2.15 – Approaching position for un- clamping.....	56
<b>S</b>	
Simulating the NC program SHAFT_OP1	
Entering the zero point offset.....	37
Operating the simulation.....	38
Support	
See service hotline.....	59
<b>T</b>	
Target groups.....	3

# Notes

Bosch Rexroth AG  
Electric Drives and Controls  
P.O. Box 13 57  
97803 Lohr, Germany  
Bgm.-Dr.-Nebel-Str. 2  
97816 Lohr, Germany  
Tel. +49 9352 18 0  
Fax +49 9352 18 8400  
[www.boschrexroth.com/electrics](http://www.boschrexroth.com/electrics)



R911341437

DOK-MTXMIC-TRAIN\*TURN\*-AP01-EN-P