# SIEMENS

Introduction	1
Advantages of working with ShopTurn	2
Ensuring everything runs smoothly	3
Basics for beginners	4
Well equipped	5
Example 1: Taper shaft	6
Example 2: Input shaft	7
Example 3: Guide shaft	8
Example 4: Hollow shaft	9
Example 5: Plunge-turning	10
And now you can start manufacturing!	11
How accomplished are you in the use of ShopTurn?	12

1

# SINUMERIK Operate

# SinuTrain Turning made easy with ShopTurn

**Training Documents** 

#### Legal information

#### Warning notice system

This manual contains notices you have to observe in order to ensure your personal safety, as well as to prevent damage to property. The notices referring to your personal safety are highlighted in the manual by a safety alert symbol, notices referring only to property damage have no safety alert symbol. These notices shown below are graded according to the degree of danger.

#### 

indicates that death or severe personal injury will result if proper precautions are not taken.

#### WARNING

indicates that death or severe personal injury may result if proper precautions are not taken.

### 

with a safety alert symbol, indicates that minor personal injury can result if proper precautions are not taken.

#### CAUTION

without a safety alert symbol, indicates that property damage can result if proper precautions are not taken.

#### NOTICE

indicates that an unintended result or situation can occur if the relevant information is not taken into account.

If more than one degree of danger is present, the warning notice representing the highest degree of danger will be used. A notice warning of injury to persons with a safety alert symbol may also include a warning relating to property damage.

#### **Qualified Personnel**

The product/system described in this documentation may be operated only by **personnel qualified** for the specific task in accordance with the relevant documentation, in particular its warning notices and safety instructions. Qualified personnel are those who, based on their training and experience, are capable of identifying risks and avoiding potential hazards when working with these products/systems.

#### Proper use of Siemens products

Note the following:

#### 

Siemens products may only be used for the applications described in the catalog and in the relevant technical documentation. If products and components from other manufacturers are used, these must be recommended or approved by Siemens. Proper transport, storage, installation, assembly, commissioning, operation and maintenance are required to ensure that the products operate safely and without any problems. The permissible ambient conditions must be complied with. The information in the relevant documentation must be observed.

#### Trademarks

All names identified by <sup>®</sup> are registered trademarks of Siemens AG. The remaining trademarks in this publication may be trademarks whose use by third parties for their own purposes could violate the rights of the owner.

#### **Disclaimer of Liability**

We have reviewed the contents of this publication to ensure consistency with the hardware and software described. Since variance cannot be precluded entirely, we cannot guarantee full consistency. However, the information in this publication is reviewed regularly and any necessary corrections are included in subsequent editions.

Siemens AG Industry Sector Postfach 48 48 90026 NÜRNBERG GERMANY Order number: 6FC5095-0AB80-1BP1 (P) 09/2011 Copyright © Siemens AG 2011. Technical data subject to change

# Table of contents

1 Introduction						
2	Advant	tages of working with ShopTurn	9			
	2.1	You will save time for training	9			
	2.2	You will save time for programming	11			
	2.3	You will save time for manufacturing	14			
3	Ensurii	ng everything runs smoothly	17			
	3.1	Using ShopTurn	17			
	3.2 3.2.1 3.2.2 3.2.3	The contents of the main menu Machine Parameters Program				
	3.2.4	Program Manager	27			
	3.2.5	Diagnosis				
4	Basics for beginners					
	4.1 4.1.1 4.1.2 4.1.3 4.1.4 4.1.5	Geometrical basics Tool axes and work planes Points in the work space Absolute and incremental dimensioning Cartesian and polar dimensions Circular motions				
	4.2 4.2.1 4.2.2	Technological basics Cutting rate and speeds Feed				
5	Well ea	quipped				
	5.1 5.1.1 5.1.2 5.1.3	Tool management The tool list The tool wear list Magazine list	41 41 43 44			
	5.2	Tools used	44			
	5.3	Tools in the magazine	46			
	5.4	Gauging tools	47			
	5.5	Setting the workpiece zero	49			
6	Examp	ble 1: Taper shaft	51			
	6.1	Overview	51			
	6.2	Program management / creating programs	53			
	6.3	Calling a tool				

	6.4	Specifying the distance to be traversed	59
	6.5	Creating contours with the contour calculator and machining sequence	63
	6.6	Thread undercut	79
	6.7	Thread	82
	6.8	Grooves	
7	Examp	le 2: Input shaft	
	7.1	Overview	
	7.2	Face turning	
	7.3	Creating the contour, stock removal and residual stock removal	91
	7.4	Thread	110
Q	Fyomn	lo 2: Guido shoft	
0			
	8.1		
	8.2	Face turning	114
	8.3	Creating any blank contour	116
	8.4	Creating the machined part contour and stock removal	118
	8.5	Residual stock removal	128
	8.6	Groove	133
	8.7	Thread	136
	8.8	Drilling	138
	8.9	Milling the rectangular pocket	143
9	Examp	le 4: Hollow shaft	147
	9.1	Overview	147
	9.2	Creating the first workpiece side	148
	9.2.1	Face turning	149
	9.2.2	Drilling	150
	9.2.3	Blank contour	
	9.2.4	Machined part contour of the first side, external	
	9.2.5	Machined part contour of the first side internal	
	9.2.7	The work step editor.	
	9.2.8	Copying the contour	175
	9.3	Creating the second workpiece side	176
	9.3.1	Face turning	176
	9.3.2	Drilling	178
	9.3.3	Specifying a blank contour	
	9.3.4	Machined part contour of the second side, external	
	9.3.5 0.2 G	Creating an asymmetrical groove	
	9.3.0	ואומטוווויפט אמון כטוונטנו טו נוופ גפנטווט גוטפ, ווונפווומן	100

10	Examp	195			
	10.1	Overview	195		
	10.2	Plunge-turning			
	10.3	Creating the contour			
	10.4	Stock removal with the plunge-turning cycle			
11	And no	ow you can start manufacturing!			
	11.1	And now you can start manufacturing!			
12	How accomplished are you in the use of ShopTurn?				
	12.1	Exercise 1			
	12.2	Exercise 2			
	12.3	Exercise 3	211		
	12.4	Exercise 4	213		
	Index				

Table of contents

# Introduction

### Faster from the drawing to the workpiece - but how?

The technological development of machine tools is highly dynamic. Particularly with the creation of NC programs, the range has extended from pure CAM system programming to programming directly at the CNC machine. Special, productive programming methods are available for each area. With ShopTurn, SIEMENS therefore offers a programming method specially tailored to the shop floor which allows quick and practical programming of machining steps from the manufacture of single parts up to small batches. In conjunction with SINUMERIK Operate, the new operator interface for the control system, intuitive and effective working in the workshop is possible even for series production.

### The solution is: Creating a process plan instead of programming

The creation of a process plan with intuitive and operator-friendly handling sequences, allows the ShopTurn user to create the NC program directly from the drawing. Even changes and different variants of a workpiece can be quickly programmed due to the clear structure.

Even the most complicated contours and workpieces are simple to manufacture with ShopTurn thanks to the integrated, powerful tools for creating traversing paths. For this reason:

### Simpler and faster from the drawing to the workpiece - with ShopTurn.

Although ShopTurn is easy to learn, these ShopTurn Training Documents allow you to enter this world even faster. Before, however, it comes to actually working with ShopTurn, the first sections cover a few important basics:

- First, we will show you the advantages of working with ShopTurn.
- Then, we will explain the basics of the operation.
- And next, the geometrical and technological fundamentals will be introduced to the beginner.
- A short introduction to tool management will be given in a further section.

The theory is followed by practical exercises with ShopTurn:

- Five examples have been chosen to explain the machining possibilities with ShopTurn, whereby the degree of difficulty is increased continuously. At the beginning, all key actions are specified; later you will be prompted to proceed without help.
- Then you will learn how to cut in AUTOMATIC mode using ShopTurn.
- If you wish, at the end you may test your knowledge of ShopTurn.

Please note that the technology data used here are only examples due to the wide variety of situations in the workshop.

Just as ShopTurn itself was created with the help of skilled workers, these Training Documents were also drawn up by practical users. On that note, we wish you much pleasure and success in your work with ShopTurn.

# Advantages of working with ShopTurn

This section shows you the special advantages of working with ShopTurn.

### 2.1 You will save time for training....

• ShopTurn does not use any foreign-language terms you would otherwise have to learn. All required inputs are requested in plain text.



• When working with ShopTurn, you are offered clear assistance by colored help displays.



Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1 2.1 You will save time for training ....

- You can also integrate DIN/ISO commands into the **Graphical Process Plan** of ShopTurn. You may also program in DIN/ISO 66025 and use DIN cycles.
  - G 696 S320 LIMS=3000 M4 M8¶
  - G G18 G54 G90¶
  - G G0 X32 Z0¶
  - G G1 X-1.6 F0.1¶
  - G 60 Z2¶
  - G GØ G42 X22 Z2¶
  - G X30 Z-2¶
- You may switch between the individual work step and the workpiece graphic (broken-line graphics) at any time during the creation of a process plan.

							09/15/11 8:31 AM
NC/	<b>WKS/EXAMPLE1/TAPER</b>	_shaft				11	Select
Ρ	Program header		Work offset (	ì54		^	tool
Т	Turning T=ROUGHING_1	80_A S1=240	rev				
<b>→</b>	RAPID X80 Z0.3						Build
<b>→</b>	F0.3/rev X-1.6						group
<b>→</b>	Rapid Z1						
<b>→</b>	RAPID X82						Search
<b></b>	RAPID ZØ						
<b>→</b>	F0.25/rev X-1.6						
<b></b>	Rapid Z1						Mark
<b>→</b>	RAPID X120 Z200						
۲U	Contour		TAPER_SHAF	F_CONTOUR		=	
<b>)</b>	Stock removal	$\nabla$	T=ROUGHING	_T80_A F0.3/re	ev V=240m		Сору
Ì≁^	Stock removal	$\nabla \Delta \Delta$	T=FINISHING_	T35 A F0.15/re	v V=200m		
	Undercut thrd	$\nabla + \nabla \nabla \nabla$	T=FINISHING_	T35 A F0.15/re	v V=200m X0=	-30 20=-17	
M	Thread long.	$\triangle + \triangle \triangle \triangle$	T=THREAD_1	5 P1.5mm/rev	S=800rev Out	side X0=30	Paste
à.Ľ	Groove	$\nabla + \nabla \nabla \nabla$	T=PLUNGE_C	UTTER_3 A F0. <sup>-</sup>	1/rev V=150m	N2 X0=60	
END	End of program						Curt
							Cut
						>	
	7	🛛 🦊 Turn-	📕 Cont.		💶 Vari-	+ Simu-	NC Ex-
J		ing ing	📄 turn.	<b>Milling</b>	and a second	lation	🖶 ecute

Figure 2-1 Work step in a process plan



Figure 2-2 Broken-line graphics

2.2 You will save time for programming...

### 2.2 You will save time for programming...

• ShopTurn offers optimum assistance when it comes to entering technological values: You only have to enter the handbook values **feedrate** (or **feed**) and **cutting rate** – the speed is calculated by ShopTurn automatically.

Drilling centric			Drilling centric		
Т	DRILL_5	D 1	Т	DRILL_5	D 1
F	100.000	mm/min	F	0.040	mm/rev
V	40	m/min	S	2546.000	rpm
	Chip rem	oval		Chip rem	oval

• ShopTurn enables you to describe a complete machining sequence using only one work step, and the required positioning motions (in this case, from the tool change point to the workpiece and back) are created automatically.

NC/	WKS/TEST/TEST	
Р	Program header	Work offset G54
çez	Drilling centric	T=DRILL_5 F0.04/rev S2546rev X1=-→
END	End of program	

• All work steps are represented by ShopTurn in a compact and clear fashion in the **Graphical Process Plan**. This provides you a complete overview and thus better editing possibilities even if comprehensive manufacturing sequences are to be performed.

					09/15/11 8:34 AM
NC/	WKS/EXAMPLE4/HOL	LOW_SHAFT_SI	DE1	7	Select
Ρ	Program header		Work offset G54	^	tool
1	Stock removal	$\nabla$	T=ROUGHING_T80_A F0.2/rev V=240m Face X0=105		
fez -	Drilling	_+	T=DRILL_32 F0.1/rev V=240m Z1=-57		Build
N	001: Positions	□+	Z0=0 X0=0 Y0=0		group
ੁ\	Contour		Hollow_Shaft_Side_2_e		
Mi -	Stock removal	$\nabla$	T=ROUGHING_T80_A F0.3/rev V=260m		Search
<b>.</b> -	Stock removal	$\nabla \Delta \Delta$	T=FINISHING_T35 A F0.15/rev V=280m	<b>→</b>	
ð.H.	Groove	$\nabla + \nabla \nabla \nabla$	T=PLUNGE_CUTTER_3 A F0.08/rev V=180m X0=70		
<u>U</u> -	Contour		HOLLOW_SHAFT_SIDE_2_1		Mark
Mi-	Stock removal	$\nabla$	1=ROUGHING_180_A F0.25/rev V=280m		
national Contraction	Residual cutting	~	1=FINISHING_1351F0.12/rev V=240m	_	Comu
<i>?#//</i>	Stock removal	~~~	1=FINISHING_1351F0.12/rev v=280m		Cobà
ENU	End of program				
					Paste
					Cut
				~	
				>	
J	Edit 📑 Drillin	g 🚅 Turn- ing	Cont. T- Milling lations ous lations lations	iu- on	Ex-

 During stock removal, for example, several machining operations and contours can be linked together.

U 1 Contour		Hollow_Shaft_blank
U Contour		HOLLOW_SHAFT_SIDE1_E
Stock removal	$\nabla$	T=ROUGHING_T80 A F0.3/rev V260m
🔭 Residual cutting	$\nabla$	T=FINISHING_T35 A F0.2/rev V240m
🖌 Stock removal	$\nabla \nabla \nabla$	T=FINISHING_T35 A F0.15/rev V280m-

2.2 You will save time for programming...

• The integrated contour calculator can process all standard dimensions (Cartesian, polar); it is nevertheless very easy to handle and understand - thanks to colloquial input and graphic support.







2.2 You will save time for programming ...



 You may switch between the graphic view and parameter screenform with help display at any time.

Figure 2-5 Graphic view



• Process plan and manufacturing do not exclude each other. With ShopTurn, you can create a new process plan parallel to manufacturing.

2.3 You will save time for manufacturing ...

### 2.3 You will save time for manufacturing...

• You can optimize tool selection during the stock removal of contours:

Large volumes are removed using roughing tools, any residual material 1 is then detected and removed automatically using a pointed tool.



• The exact specification of the selected retraction plane avoids the use of unnecessary traversing paths and thereby saves valuable manufacturing time. This is possible using the settings **normal**, **extended** and **all**.

#### **Retraction plane: normal**



#### Retraction plane: extended



#### Retraction plane: all



2.3 You will save time for manufacturing...

• You can optimize your machining sequence with a minimum of work - thanks to the compact structure of the process plan (in this case, by saving of a tool change, for example).

_	ר 💥 🗋			09/15/11 8:44 AM
NC/	WKS/EXAMPLE1/TAPE	r_shaft		15 Select
Ρ	Program header		Work offset G54	▲ tool
Т	Turning T=ROUGHING_	T80_A S1=240	rev	
<b>→</b>	RAPID X80 Z0.3			Build
<b>→</b>	F0.3/rev X-1.6			group
<b>→</b>	Rapid Z1			
<b>→</b>	RAPID X82			Search
<b>→</b>	RAPID Z0			
<b>→</b>	F0.25/rev X-1.6			
<b>→</b>	RAPID 21			Mark
$\rightarrow$	RAPID X120 Z200			
<u>U</u> -	Contour		TAPER_SHAFT_CONTOUR	0
<i>Mi</i> -	Stock removal	▽	1=RUUGHING_180_H F0.3/rev V=240m	cobà
<b>***</b>	Stock removal	~~~	T=FINISHING_135 H F0.15/ rev V=200m	
	Inread long.	V+VVV	T=THKEHU_1.5 P1.5mm/rev 5=800rev Outside X0=3	Paste
3 6	Groove	V+VVV	T-PHISTING_133 H F0.13/160 V-20011 A0-30 20[	
SHA.	End of program		1-reorde_corren_3 + re.1/160 v-13011 H2 X0-00	,
	Life of program			Cut
_				
	Edit 📑 Drilling	Jurn-	Cont. Milling Sir Vari-	mu- NC Ex-

Figure 2-7 Original machining sequence

				09/15/11 8:52 AM
NC/	WKS/EXAMPLE1/TAPE	er_shaft		14 Select
Р	Program header		Work offset G54	▶ tool
Т	Turning T=ROUGHING	_T80_A S1=240	lrev	
<b>→</b>	RAPID X80 Z0.3			Build
<b>→</b>	F0.3/rev X-1.6			group
<b>→</b>	Rapid Z1			
<b></b>	Rapid X82			Search
<b>→</b>	RAPID Z0			
<b>→</b>	F0.25/rev X-1.6			
<b>→</b>	Rapid Z1			Mark
<b>→</b>	RAPID X120 Z200			
V-	Contour		TAPER_SHAFT_CONTOUR	
M.	Stock removal	$\nabla$	T=ROUGHING_T80_A F0.3/rev V=240m	Copy
<b>M</b>	Stock removal	$\nabla \Delta \Delta$	T=FINISHING_T35 A F0.15/rev V=200m	
	Undercut thrd	0 <b>+</b> 000	I=FINISHING_135 A F0.15/rev V=200m X0=30 20=-→	Bosto
	I hread long.	$\Delta + \Delta \Delta \Delta$	T=THREAD_1.5 P1.5mm/rev S=800rev Outside X0=30	Faste
di di la cara di la ca	Groove	$\Delta + \Delta \Delta \Delta$	I=PLUNGE_CUITER_3 A F0.1/rev V=150m N2 X0=60	
END	End of program			Cut
			>	
		" 📕 📕 Turn-	📕 Cont. 💶 Milling = Vari- 👍 Simu	- NC Ex-
Þ		y 💳 ing	turn. 💶 🖬 🔤 ous 💶 latio	n 🖼 ecute

Figure 2-8 Optimized machining sequence with cutting and pasting

 With ShopTurn, you can achieve extremely high feedrates with optimum repeat accuracy based on consistent digital technology (SINAMICS drives, ...., SINUMERIK control systems). Advantages of working with ShopTurn

2.3 You will save time for manufacturing...

# Ensuring everything runs smoothly

In this section you will learn the basics of ShopTurn operation with the help of examples.

### 3.1 Using ShopTurn

A powerful software is one thing, but it must also be easy to operate. Regardless of whether you work with the SINUMERIK 840D sl or SINUMERIK 828D shown here - you are always assisted by the clearly laid-out machine operator panel. The operator panel consists of 3 parts - the flat operator panel ①, the CNC full keyboard ② and the machine control panel (MCP) ③.



3.1 Using ShopTurn

The most important keys on the CNC full keyboard for navigation in ShopTurn are listed in the following:

Key	Function							
(i) HELP	<help> Calls the context-sensitive online help for the selected window.</help>							
SELECT	<select> Selects a listed value.</select>							
	Cursor keys The cursor is moved using the 4 cursor keys. Use the <cursor right="" the="" to=""> key to open a directory or program (e.g. a cycle) in the editor.</cursor>							
PAGE UP	<page up=""> Scrolling upwards in a menu screen.</page>							
PAGE DOWN	<page down=""> Scrolling downwards in a menu screen.</page>							
END	<end> Moves the cursor to the last input field in a menu screen or a table.</end>							
DEL	<b>&gt;DEL&gt;</b> <ul> <li>Edit mode:</li> <li>Deletes the first character to the right.</li> <li>Navigation mode:</li> <li>Deletes all characters.</li> </ul>							
HACKSPACE	<b>SACKSPACE&gt;</b> <ul> <li>Edit mode:</li> <li>Deletes a character selected to the left of the cursor.</li> <li>Navigation mode:</li> <li>Deletes all of the selected characters to the left of the cursor.</li> </ul>							
INSERT .	<insert> <ul> <li>Pressing the <insert> key opens the "Editing" mode, and when you press the key again, you will leave the "Editing" mode and you can call the "Navigation" mode.</insert></li> </ul></insert>							
INPUT	<ul> <li><input/></li> <li>Completes input of a value in the input field.</li> <li>Opens a directory or program.</li> </ul>							

The actual function selection in ShopTurn is performed using the keys located around the screen. Most of them are assigned directly to the individual menu items. Since the contents of the menus change depending on the situation, the term 'softkeys' is used.

All main functions can be called using the horizontal softkeys.

All subfunctions in ShopTurn can be called using the vertical softkeys.



The main menu can be called with this key at any time - irrespective of in which operating area you are at the moment.

### Main menu



### 3.2 The contents of the main menu

### 3.2.1 Machine

### Machine - Manual



Select the "Machine" softkey.



Press the "JOG" key.

3.2 The contents of the main menu

In this mode, the machine is set up; the tool is traversed in the MANUAL mode. It is also possible to gauge tools and to set workpiece zeros.

M									07/19/10 9:59 PM
1						SI	EMEN	S	Select tool
// Reset									
Machine	Posi	ition [mm]			T,F,S				Select
X1		0.000			Т				work offs.
74		0.000			•				
21		0.000			Fε	9.000			
					. 6	9.000	mm/min	120%	
					S1~6	)		Ø	
					Master 6	3		0.0%	
					<u>.</u>		50 .	100,	
T,S,M									
Т	FINISHIN	IG_T35 A	D 1						
Spindle	S1	200.000	rpm	Gear sta	ige				
Spindle 1	1 function	ى م							
Other M	function								
Work off	set								
Machinin	ieasure. In nlane								
	.g p.c0								~~
	_	_	_	_	_	_	_		Back
2		Mor		Meen	- Doni-			Stock	
<b>T,S,</b> ►	1 20 W	0 0 work	(p.	tool	tion		<b>-</b>	rem.	

Figure 3-1 Call of a tool and input of technological values

<b>M</b>		07/19/10 16:00 PM
		SIEMENS
🥢 Reset		
Machine	Position [mm]	T,F,S
X1	0.000	т
21	0 000	
	0.000	F 0.000
		0.000 mm/min 120%
		S1 🗸 🛛 🛛 🕅
		Master 0 0.0%
		Rapid
larget posit	tion	traverse traverse
		X 80.000
		Z 1.000 abs
		C abs
		SP2 abs
		Back
	A Sot A Moon	Maan Dania
📑 T,S,M	U Pa UO Po workp.	tool tion FUSI-

Figure 3-2 Input of a target position

Ensuring everything runs smoothly 3.2 The contents of the main menu

### Machine - AUTO



 Select the "Machine" softkey.

#### Press the AUTO key.

During manufacturing, the current work step is displayed. It is possible to switch to a running simulation using the relevant key ("Drawing"). When executing a process plan, you may insert work steps and/or create a new process plan.

				08/11/11 10:42 AM
NC/WKS/EXAMPLE3,	/Guide_shaft		SIEMENS	G
🔿 active				functions
Llorkniece I	Position [mm]	Dist_to_go	TES	
workpiece i				Auxiliary
∛ X	80.134	0.000	RUUGHING_180_H R0.800	functions
7	60.065	0 400	1 🗇 D1 X55.848	
- 2 -	-09.900	-0.403	E 0.220	Basic
			Г U.33U	DIOCKS
			0.300 mm/rev 110%	Time
			S1 516	lime /
			Master 1833 58%	counter
<b>⊞</b> G54			<u>0 50 100</u>	Program
NC/UKS/EXAMPLE3	/Guide_shaft			levels
℃   Fertigteil		GUIDE_SHAFT_CO	NTOUR	
🥁 - Abspanen	V	T=ROUGHING_T8	0_A F0.3/U V=260m	
🥼 - Restabspanen	$\nabla$	T=BUTTON_TOOL	_8 F0.25/U V=240m	
🖌 🛛 Abspanen	$\nabla \nabla \nabla$	T=FINISHING_T35	A F0.12/U V=280m	
📜 Einstich	$\nabla + \nabla \nabla \nabla$	T=PLUNGE_CUTT	ER_3_A F0.1/U V=150m X0=60	Act. values
🗯 Gewinde Längs	$\nabla$	T=THREAD_1.5 P	1.5mm/U S=800U außen X0=48	Machine
ן 🖾 Bohren	□+	T=DRILL_5 F0.06/	/U V=140m Z1=10ink	
🔊 🛛 001: Positionen	_+	20=0 X0=16 Y0=0	X1=0 Y1=-16 X2=-16 Y2=0 X3=0	
∧ . <b>V</b>	<u> </u>	TFOT		
			Black Simult	Prog 1
			search search	corr

Figure 3-3 Executing a process plan



Figure 3-4 Simultaneous recording of the execution

3.2 The contents of the main menu

### 3.2.2 Parameters

### Parameter lists



This key can be used to edit data for the tool management and for programs.

### Tool lists

No cutting without tools.

The tools can be managed in a tool list.

										08/11/11 12:55 PM		
<b>Tool lis</b>	st									Ma	gazine 1	Tool
Loc.	MT L0.	Туре	Tool name	ST	D	Н	Length X	Length Z	Radius			measure
1/1			ROUGHING_T80_A	1	1	0	55.848	39.124	0.800	+	93	
1/2		<b>6</b>	DRILL_32	1	1	0	80.000	185.124	32.000		180	
1/3		0	FINISHING_T35 A	1	1	0	123.976	57.370	0.400	←	93	
1/4			ROUGHING_T80_I	1	1	0	55.800	39.000	0.800	←	95	Edges
1/5		<u></u>	PLUNGE_CUTTER_3_A	1	1	0	85.124	44.124	0.200		3.00	Eugou
1/6		1	PLUNGE_CUTTER_3_I	1	1	0	85.952	41.300	0.200		3.00	
1/7		0	FINISHING_T35_I	1	1	0	-12.658	121.877	0.400	←	95	
1/8		Þ	Thread_1.5	1	1	0	66.326	33.333	0.100			
1/9		8	CUTTER_8	1	1	0	87.833	74.621	8.000	3		
1/10		<b>8</b>	DRILL_5	1	1	0	80.000	185.124	5.000		118	United
1/11			BUTTON_TOOL_8	1	1	0	88.112	38.123	2.000			uniuau
1/12			THREADCUTTER_M6	1	1	0	80.000	145.000	6.000		1.00	
1/13												Delete
1/14												tool
1/15												
1/16												Magazine
2/1												selection
2/2												
2/3											~	
						<		11			>	
				_							>	
	Tool list	ľ	Tool wear			laga- zine	Ua off:	set R.	User variable			SD Setting data

Figure 3-5 Tool list

### Magazine

ţ_O	¥ ا	₩ ŽG											08/11/11 12:47 PM
Magazi	ne										Magazin	ie 1	
Loc.	MT LO.	Туре	Tool name	ST	D	D	z	L				^	
1/1		i.	ROUGHING_T80_A	1	1								Unload
1/2		<b>6</b>	DRILL_32	1	1								all
1/3		0	FINISHING_T35 A	1	1								
1/4		•	ROUGHING_T80_I	1	1							=	
1/5		1	PLUNGE_CUTTER_3_A	1	1								
1/6		1	PLUNGE_CUTTER_3_I	1	1								
1/7		6	FINISHING_T35_I	1	1								
1/8		$\geq$	Thread_1.5	1	1								
1/9			CUTTER_8	1	1								
1/10		<b>6</b>	DRILL_5	1	1								Delegate
1/11			BUTTON_TOOL_8	1	1								nelocale
1/12		6333	THREADCUTTER_M6	1	1								
1/13		<b>6</b>	DRILL_10	1	1								Position
1/14													magazine
1/15													
1/16													
2/1													
2/2													
2/3													
2/4												~	
								_			>		
	Tool list		Tool wear <b>DEM</b> Tool		٢	1ag zin	a- e		Derk offset	R User			SD Setting data

Tools can be organized into a magazine.

Figure 3-6 Magazine

### Work offsets

Zero points are saved in a clearly laid-out zero-point table.

							08/12/11 5:32 PM
Work offset – Overview (mi	n]						
	\$° £ ∆∆	Х	Y	Z	A	C	
DRF		0.000	0.000	0.000	0.000	0.000	
Rotary table ref.		0.000	0.000	0.000	0.000	0.000	Active
Basic reference		0.000	0.000	0.000	0.000	0.000	
Total basic WO		0.000	0.000	0.000	0.000	0.000	
G54		9.992	15.008	0.000	0.000	0.000	Overview
Tool reference		0.000	0.000	0.000	0.000	0.000	
Workpiece ref.		0.000	0.000	0.000	0.000	0.000	
Programmed WO		0.000	0.000	0.000	0.000	0.000	Base
Cycle reference		0.000	0.000	0.000	0.000	0.000	
Total WO		9.992	15.008	0.000	0.000	0.000	
							G54
							G5/
							Details
7							
× .	1					>	
Tool Tool		- Morro		Hork	lleor		Cotting
list wear	OEM Tool	zine	•	offset	R variable		SD data

Figure 3-7 Work offsets

3.2 The contents of the main menu

### 3.2.3 Program

### **Editing programs**



This key can be used to edit programs.

If you have created a **ShopTurn program** in the Program Manager, you can now create the process plan with the complete machining sequence for the appropriate workpiece. An optimum sequence can only be achieved by a skilled worker with suitable experience.

							08/11, 10:56	/11 AM
NC/	<b>WKS/EXAMPLE2/DRIVE</b>	_shaft				3	Select	
Ρ	Program header		Work offset G	i54		^	tool	
	Stock removal	$\nabla \nabla \nabla$	T=ROUGHING	_T80_A F0.25/r	ev V=240m Fa	ace X0=80		
V٦	Contour		DRIVE_SHAFT	_CONTOUR		$\overline{}$	Build	
M	Stock removal	$\bigtriangledown$	T=ROUGHING	_T80_A F0.3/re	v V=240m		group	
)	Residual cutting	$\bigtriangledown$	T=FINISHING_	T35 A F0.2/rev	V=240m			
<u>کھر</u>	Stock removal	$\nabla \nabla \nabla$	T=FINISHING_	T35 A F0.12/re	v V=280m		Search	
W	Thread long.	$\nabla + \nabla \nabla \nabla$	T=THREAD_1.	5 P1.5mm/rev	S=800rev Out	side XØ=24		
END	End of program							
							Mark	
						_		
							Сору	
							Paste	
							Cut	
_						V		
		- T				>	-	
J	Edit Edit	ing	turn.	- Milling	Vari- ous	lation	ecuti	e

The contour to be machined is entered graphically as a machining step.





Geometry and technology constitute a unit in programming. The subsequent technological machining operations are applied to the contour.

Example for the dovetailing of geometry and technology:



This geometrical-technological interrelation is represented very clearly in the graphical display of the work steps by putting the appropriate symbols in brackets. The brackets mean linking of geometry and technology to one work step.

### Simulating programs

Before machining a workpiece on the machine, it is possible to display the program execution graphically on the screen.

- To this end, select the "Simulation" and "Start" softkeys.
- To stop simulation, select the "Stop" softkey.
- To cancel simulation, use the "Reset" softkey.

3.2 The contents of the main menu



The following views are available, among others, for simulation:

Figure 3-8 Side view (display tool path, activated)



Figure 3-9 3D view



Figure 3-10 2-window view

### 3.2.4 Program Manager

### Managing programs



With the Program Manager, you can create new programs at any time. You can similarly open existing programs to execute, modify, copy or rename them. Programs no longer required can be deleted.

							08/11/11 11:06 AM
	Name		Туре	Length	Date	Time	Directory
<ul> <li>Part programs</li> <li>Subprograms</li> <li>Workpieces</li> <li>EXAMPLE1</li> <li>EXAMPLE2</li> <li>EXAMPLE3</li> </ul>		New se	DIR DIR nin equential prog	ram	08/04/11 08/04/11 08/11/11 1 1	8:52:06 AM 8:52:06 AM 11:05:50 AM 10:08:52 AM 8:52:07 AM 11:05:50 AM	
- 🗂 TEMP	Tune		ShonTurn		1	11:03:04 AM	ShopTurn
	Name <mark>GU</mark>	ide_shaft	Shop furth				programGUIDE G code
							Any
							× Cancel
NC/Workpieces	_					Free: 2.4 MB	ŐK

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1 3.2 The contents of the main menu

₲ 2					07/26/10 13:57 PM
Name	Туре	Length	Date	Time	Execute
Part programs	DIR		07/21/10	11:47:28 AM	Exoouto
	DIR WPD		07/26/10 07/21/10	1:56:45 PM 11:47:57 AM	New 🕨
	WPD		07/26/10	1:56:44 PM	
■ Exhibites ■ Examples ■ Examples ■ Hollow Shaft Side1	WPD WPD MPF	3934	07/26/10 07/26/10 05/27/10	1:56:45 PM 2:39:10 PM	Open
HOLLOW_SHAFT_SIDE2	MPF	3780	05/27/10	2:41:38 PM	
• 🗅 EXAMPLE5	upd Upd		07/26/10 07/26/10	1:56:45 PM 1:54:32 PM	Mark
					Сору
					Paste
					Cut
NC/Workpieces/EXAMPLE4.WPD				Free: 2.4 MB	
NC 🔤 Local 🦿 USB					

Active programs are marked with a green symbol.

ψ USB

USB flash drives can be used for data exchange. For example, programs which were created on an external device can be copied and executed on the NC.

#### Creating a new workpiece

You can manage your programs and other files, such as tool data, zero points, magazine mapping, in a workpiece.

### Creating a new program

If you create a new program, you can specify the type of programming using the following softkeys:



ShopTurn program

G code

G code program

Ensuring everything runs smoothly 3.2 The contents of the main menu

### 3.2.5 Diagnosis

### Alarms and messages



Here you can see alarm lists, messages and alarm logs.

						07/19/10 10:42 PM
Alarm log						Display
Raised 🔻	Cleared	Number	Text		^	new
07/19/10 10:27:49.341 PM	07/19/10 10:27:55.810 PM	150202	Waiting for a connection t	to /PLC/PMC		
07/19/10 10:27:49.341 PM	07/19/10 10:27:55.808 PM	150202	Waiting for a connection t	to /PLC/DiagBuffer		
07/19/10 10:27:48.896 PM	07/19/10 10:27:53.653 PM	150202	Waiting for a connection t	to /NCK		
07/19/10 10:27:46.287 PM	07/19/10 10:27:46.287 PM	150204	Start alarm acquis	ition		
					_	
						Settings
						Save
						log
					~	
					>	
Alarm list	Sages Mes-	Alarm log	V NC/PLC variab.	Remote Ros diag.		V: Version

Figure 3-11 Alarm log

Ensuring everything runs smoothly

3.2 The contents of the main menu

# **Basics for beginners**

This section explains the general basics of the geometry and technology for turning. No inputs for ShopTurn are planned yet.

### 4.1 Geometrical basics

### 4.1.1 Tool axes and work planes

During turning, it is the workpiece, and not the tool, that rotates. The axis is the Z axis.

- G18 plane = machining with turning tools
- G17 plane = drilling and milling operations on the front face
- G19 plane = drilling and milling operations on the peripheral surface

Since the diameter of the turned workpieces is relatively easy to control, the dimensions of the transverse axis are based on the diameter. This means that you can directly compare the actual dimensions with the dimensions in the drawing.

### 4.1.2 Points in the work space

Various important reference points are provided for a CNC - such as the SINUMERIK 828D with ShopTurn - for orientation in the working space by means of the measuring system.



Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1

Basics for beginners
4.1 Geometrical basics



### Machine zero (M)

The machine zero (M) is specified by the manufacturer and cannot be changed. It lies in the origin of the machine coordinate system.



### Workpiece zero (W)

The workpiece zero (W) - also called program zero - is the origin of the workpiece coordinate system. It can be selected freely and should be located at a point from which the most dimensions start in the drawing.



### Reference point (R)

The reference point (R) is approached to set the measuring system to zero, as the machine zero cannot be approached in most cases. Thus, the control system finds the start of counting in the position measuring system.



### Toolholder reference point T

Toolholder reference point T is used to set up machines with tool revolvers with default tools. Its position and location hole permit setup with cutter holders for shank tools in accordance with DIN 69880 and VDI 3425.

### 4.1.3 Absolute and incremental dimensioning

### Absolute input

The entered values refer to the workpiece zero.

Straig	ht							
Х	50.000	abs						
Y		abs						
Z	-20.000	abs						
* G90 Absolute dimensions								



With absolute inputs, the **absolute** coordinate values of the **end point** must always be entered (the current position is not taken into account).

### Incremental input

SELECT

The entered values refer to the current position.

raig	ht		End point
{ {	20.000	inc abs	
Z	-5.000	inc	(+)20
G91 I	ncremental dime	nsions	
			_ <del>5</del> ← Current
			Positior

With incremental inputs, the difference values between the current position and the end point must always be entered, observing the direction.

Switching between absolute and incremental input is possible at any time using the SELECT key.

A few examples for the absolute/incremental combination can be found below:

Straig	ht	
Х	10.000	abs
Y		abs
Z	-35.000	inc

Straig	ht	
Х	25.000	inc
Y		abs
Z	-40.000	abs





Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1

### 4.1.4 Cartesian and polar dimensions

### Cartesian input

Input of the X and Z coordinates. The gray values in the example were calculated automatically.

ht ZX	
100.000	abs
40.000	inc
-40.000	abs
-30.000	inc
50.000	
126.870	0
320.906	0
	ht ZX 100.000 40.000 -40.000 -30.000 50.000 126.870 320.906



With absolute inputs, the **absolute** coordinate values of the **end point** must always be entered (the current position is not taken into account).

### Polar input

Input of the length and angle. The gray values in the example were calculated automatically.

Straig	ht ZX	
Х	100.000	abs
Х	40.000	inc
Z	-40.000	abs
Z	-30.000	inc
L	50.000	
α1	126.870	0
α2	320.906	0

 $\alpha$ **1** = Angle to the positive Z axis

 $\alpha 2$  = Angle to the preceding element



The angles can be entered...

... negative.



The Cartesian and polar inputs can also be combined. Here are two examples:

Straig	nt ZX	
Х	100.000	abs
Х	40.000	inc
Z	-40.000	abs
Z	-30.000	inc
L	50.000	
α1	126.870	0
α2	320.906	0

Input of the end point in X and the length
Straight ZX



Input of the end point in Z and an angle

Straig	ht ZX	
Х	100.000	abs
Х	40.000	inc
Z	-40.000	abs
Z	-30.000	inc
L	50.000	
α1	126.870	0
α2	320.906	0



### 4.1.5 Circular motions

In accordance with DIN, circular arcs are specified with the end point of the arc (X and Z coordinates in the G18 plane) and the center point (I and K in the G18 plane).

The ShopTurn contour calculator gives you the freedom to use any dimension from the drawing for circular arcs, without having to carry out conversions.

The following example shows two circular arcs, one of which is only partially determined.
4.1 Geometrical basics

#### Input of the R10 arc:

ion of rotation	Q
10.000	
50.000	abs
-35	abs
	abs
	abs
	•
	ion of rotation 10.000 50.000 -35

After the input:

Circle										
Direct	ion of rotation	2								
R	10.000									
х	50.000	abs								
Z	-35.000	abs								
1	50.000	abs								
K	-25.000	abs								
α1	180.000	0								
α2	Tangential									



Input of the R20 arc:



#### After the input:

Circle		
Directi	ion of rotation	Ş
R	20.000	
Х	30.000	abs
Х	15.000	inc
Z	-6.771	abs
Z	-6.771	inc
I I	0.000	abs
I I	0.000	inc
K	-20.000	abs
K	-20.000	inc
α1	90.000	•
β1	138.590	0
β2	48.590	0

All parameters

The following displays of all values appear when you have entered all known dimensions and pressed the **All parameters** softkey in the input window of the respective arc.

Circle		
Direct	ion of rotation	Q
R	10.000	
х	50.000	abs
х	10.000	inc
Z	-35.000	abs
Z	-10.000	inc
1	50.000	abs
1	10.000	inc
ĸ	-25.000	abs
ĸ	0.000	inc
α1	180.000	0
α2	Tangential	
β1	90.000	0
β2	90.000	0

In DIN format: G2 X50 Z-35 CR=10



Circle		
Directi	ion of rotation	Ş
R	20.000	
Х	30.000	abs
Х	15.000	inc
Z	-6.771	abs
Z	-6.771	inc
1	0.000	abs
1	0.000	inc
ĸ	-20.000	abs
ĸ	-20.000	inc
α1	90.000	•
β1	138.590	•
β2	48.590	•
In DIN	l format:	

G3 X30 Z-6.771 K-20

# 4.2 Technological basics

# 4.2.1 Cutting rate and speeds

The cutting rate is usually directly programmed for turning, in particular for roughing, finishing and plunge-cutting. The speed is programmed only for drilling and (usually) for thread cutting.

# Determination of the cutting rate

First, the optimum cutting rate is determined using either the manufacturer catalogs or a handbook.

Value:	vc = 180 m/min
Material of the <b>workpiece:</b>	Machining steel
Material of the <b>tool:</b>	Hard metal

# Constant cutting rate $v_c$ (G96) for rough cutting, finishing and plunge-cutting:



To ensure that the selected cutting rate remains constant for each workpiece diameter, the control uses the G96 command (= constant cutting rate) to adjust the appropriate speed. This is performed using direct-current or frequency-controlled threephase motors. As the diameter reduces, the speed will theoretically increase to infinity. To prevent danger caused by excessive centrifugal forces, a speed limitation of n = 3000 rpm, for example, must be programmed.

In DIN format, the block would have the following form:

**G96 S180 LIMS=3000** (LIMS = limit).

# Constant speed n (G97) for drilling and thread cutting:

Since a constant speed is used for drilling, the command G97 (= constant speed) must be used here.

The speed is dependent on the desired cutting rate (120 m/min selected here) and the tool diameter.

The inputs are then G97 S1900.



## 4.2.2 Feed

In the previous chapter, you learned how to determine the cutting rate and the speed. The tool can only perform machining if the cutting rate or the speed is assigned a tool feedrate.

#### Determination of the feed

Like the cutting rate, the value for the feed is also taken from the handbook, the documents of the tool manufacturer or from empirical knowledge.

Cutting material of the tool:	Hard metal
Material of the <b>workpiece:</b>	Machining steel
Determined value (handbook):	f = 0.2 - 0.4 mm
The mean value will be chosen:	f = 0.3 mm

4.2 Technological basics

# Relationship between feed and feedrate:

The constant feed **f** and the associated speed **n** produce the feedrate:



Since the speed is different, the feedrate is different for the various diameters, despite having the same feed.



# Well equipped

In this section you will learn how to create the tools required for the examples in the following sections. Furthermore, it is explained how to take into account the tool lengths and how to set the workpiece zero.

# 5.1 Tool management

ShopTurn offers three lists for tool management:

- the tool list
- the tool wear list
- the magazine list.

# 5.1.1 The tool list

The tool list displays all parameters and functions required to create and set up the tools.

<u>t</u> ⊡	) {	₩ 6										08/11/11 12:55 PM
Tool lis	t									Mag	jazine 1	Tool
Loc.	MT LO.	Туре	Tool name	ST	D	Н	Length X	Length Z	Radius			measure
1/1		i.	ROUGHING_T80_A	1	1	0	55.848	39.124	0.800	←	93	
1/2		<b>S</b>	DRILL_32	1	1	0	80.000	185.124	32.000		180	
1/3		Ø	FINISHING_T35 A	1	1	0	123.976	57.370	0.400	←	93 _	
1/4		•	ROUGHING_T80_I	1	1	0	55.800	39.000	0.800	←	95	Edges
1/5			PLUNGE_CUTTER_3_A	1	1	0	85.124	44.124	0.200		3.00	Luges
1/6		1	PLUNGE_CUTTER_3_I	1	1	0	85.952	41.300	0.200		3.00	
1/7		0	FINISHING_T35_I	1	1	0	-12.658	121.877	0.400	←	95	
1/8		$\geq$	Thread_1.5	1	1	0	66.326	33.333	0.100			
1/9			CUTTER_8	1	1	0	87.833	74.621	8.000	3		
1/10		<b>S</b>	DRILL_5	1	1	0	80.000	185.124	5.000		118	11-1
1/11			BUTTON_TOOL_8	1	1	0	88.112	38.123	2.000			Unioad
1/12			THREADCUTTER_M6	1	1	0	80.000	145.000	6.000		1.00	
1/13												Delete
1/14												tool
1/15												
1/16												Magazine
2/1												selection
2/2												
2/3											~	
						<					>	
				_							>	
	Tool list	Þ	Tool wear		ľ	laga- zine	Generation United States	set Ru	User ariable			SD Setting data

Figure 5-1 Example for tool lists

5.1 Tool management

Meanings	of the	most	important	parameters:
----------	--------	------	-----------	-------------

Location	Location number
Туре	Tool type
Tool name	The tool is identified by the name and the replacement tool number. You may enter the names as text or numbers.
ST	Replacement tool number (for replacement tool strategy)
D	Cutting edge number
Length X	Geometry data, length X
Length Z	Geometry data, length Z
Diameter	Tool diameter
Holder angle, point angle, board width	Holder angle (roughing tool and finishing tool), point angle (drill) and board width (plunge-cutter)
₽	Direction of spindle rotation
<del>ب</del>	Coolants 1 and 2 (e.g. internal and external cooling)

ShopTurn features various tool types (favorites, milling cutters, drills, turning tools and special tools). Tools can be created in the tool list by means of a predefined tool catalog. Various mounting positions and geometrical parameters exist (e.g. holder angle), depending on the tool type.

New too	I - favorites	
Туре	ldentifier	Tool position
500 ·	- Roughing tool	
510 ·	- Finishing tool	
520 ·	- Plunge cutter	
540 ·	- Threading tool	
550 ·	- Button tool	$\odot \odot \odot \odot$
560 ·	- Rotary drill	
580 -	- 3D turning probe	- <b>- - -</b>
730	- Stop	
120 -	- End mill	
140	- Facing tool	
150	- Side mill	
200	- Iwist drill	
240	- Tap	60000 <b>11</b> 3 <b>10000</b> 318

Figure 5-2 Example of Favorites list

# 5.1.2 The tool wear list

The wearing data for the appropriate tools are defined here.

ţ_O	) \ \	کم کړ									08/11/11 12:47 PM
Tool we	ear									Magazine 1	Sort
Loc.	MT L0.	Туре	Tool name	ST	D	∆Length X	∆Length Z	ΔRadius	T C	^	
1/1		•	ROUGHING_T80_A	1	1	0.000	0.000	0.000			Filter
1/2		<b>6</b>	DRILL_32	1	1	0.000	0.000	0.000			
1/3		0	FINISHING_T35 A	1	1	0.000	0.000	0.000			
1/4		•	ROUGHING_T80_I	1	1	0.000	0.000	0.000			Search
1/5		IJ	PLUNGE_CUTTER_3_A	1	1	0.000	0.000	0.000			Couron
1/6			PLUNGE_CUTTER_3_I	1	1	0.000	0.000	0.000			
1/7			FINISHING_T35_I	1	1	0.000	0.000	0.000			Details
1/8		$\triangleright$	Thread_1.5	1	1	0.000	0.000	0.000			Details
1/9		8	CUTTER_8	1	1	0.000	0.000	0.000			
1/10		<b>6</b>	DRILL_5	1	1	0.000	0.000	0.000			
1/11			BUTTON_TOOL_8	1	1	0.000	0.000	0.000			
1/12		6333	THREADCUTTER_M6	1	1	0.000	0.000	0.000			
1/13		<b>6</b>	DRILL_10	1	1	0.000	0.000	0.000			
1/14											
1/15											
1/16											
2/1											
2/2											
2/3										v	
						<		1			
										>	
	Tool list		Tool wear		٢	1aga- zine	Uork offset	R vari	ser iabl	e	SD Setting data

Figure 5-3 Tool wear list

The most important tool wearing parameters are:

Δ Length X	Length X wear				
Δ Length Z	Length Z wear				
Δ Radius	Radius wear				
ТС	Selection of tool monitoring				
	• by tool life (T)				
	• by count (C)				
	• by wear (W)				
Tool life or	Tool life				
workpiece count or	Number of workpieces				
wear *	Tool wear				
*Parameter depends on selection in TC					
Setpoint	Setpoint for tool life, workpiece count, or wear				
Prewarning limit	Specification of the tool life, workpiece count or wear at which a warning is displayed.				
G	The tool is disabled when the checkbox is selected.				

5.2 Tools used

# 5.1.3 Magazine list

All tools that are assigned to one or several tool magazines are contained in the magazine list. This list displays the condition of each tool. Individual magazine locations can be reserved or disabled for existing tools.

<u>t</u>	۲ ۲												08/11/11 12:47 PM
Magaz	ine										Magazin	e 1	
Loc.	MT LO.	Туре	Tool name	ST	D	D	z	L				^	
1/1			ROUGHING_T80_A	1	1								Unload
1/2		<b>S</b>	DRILL_32	1	1								all
1/3		0	FINISHING_T35 A	1	1								
1/4		•	ROUGHING_T80_I	1	1							=	
1/5		Ţ	PLUNGE_CUTTER_3_A	1	1								
1/6		1	PLUNGE_CUTTER_3_I	1	1								
1/7		9	FINISHING_T35_I	1	1								
1/8		Þ	Thread_1.5	1	1								
1/9		8	CUTTER_8	1	1								
1/10		20	DRILL_5	1	1								Pelocate
1/11		O	BUTTON_TOOL_8	1	1								neiocale
1/12		6888	THREADCUTTER_M6	1	1	닏	Ц	Ц					
1/13		- 55	DRILL_10	1	1	닏	Ш	Ш				-	Position
1/14						닏						-	magazine
1/15						님						- 1	
1/16						님						-	
2/1						님						-	
2/2						님						-	
2/3				_		님							
2/4										_		Ľ	
	<b>T</b> 1												0 W
	lool list		wear <b>OEM</b> Tool	955 2	ľ	Tâg zin	a- e	K	offset	R variable			SD data

Figure 5-4 Magazine list

Meanings of the most important parameters:

G	Disabling of the magazine location
Ü	Marking of a tool as oversized. The tool occupies two half locations left, two half locations right, one half location top and one half location bottom in a magazine.
Ρ	Fixed location coding
	The tool is permanently assigned to this magazine location.

# 5.2

# Tools used

In this section you will learn how to enter tools required for the later machining in the tool list. Select the "Parameters" area in the main menu.



Select the "Tool list" softkey.

<u>↓</u> O	) \ \	کی کی										08/11/11 12:49 PM
Tool lis	t									Mag	jazine 1	
Loc.	MT L0.	Туре	Tool name	ST	D	Н	Length X	Length Z	Radius		^	
1/1		•	ROUGHING_T80_A	1	1	0	55.848	39.124	0.800	←	93	New
1/2		<b>6</b>	DRILL_32	1	1	0	80.000	185.124	32.000		180	tool
1/3		Ø	FINISHING_T35 A	1	1	0	123.976	57.370	0.400	←	93	
1/4		•	ROUGHING_T80_I	1	1	0	55.800	39.000	0.800	←	95	
1/5		1	PLUNGE_CUTTER_3_A	1	1	0	85.124	44.124	0.200		3.00	
1/6		1	PLUNGE_CUTTER_3_I	1	1	0	85.952	41.300	0.200		3.00	
1/7		0	FINISHING_T35_I	1	1	0	-12.658	121.877	0.400	←	95	
1/8		$\geq$	Thread_1.5	1	1	0	66.326	33.333	0.100			
1/9		8	CUTTER_8	1	1	0	87.833	74.621	8.000	3		
1/10		Ø	DRILL_5	1	1	0	80.000	185.124	5.000		118	
1/11		.0	BUTTON_TOOL_8	1	1	0	88.112	38.123	2.000			
1/12		6333	THREADCUTTER_M6	1	1	0	80.000	145.000	6.000		1.00	
1/13		Ø	DRILL_10	1	1	0	80.000	120.000	10.000		118	
1/14												
1/15												
1/16												Magazine
2/1												selection
2/2												
2/3											~	
						<	_				>	
	_		4					. 4			>	
	Tool list	Þ	vear <b>CEM</b> Tool			laga- zine	Uo offs	set R	User ariable			SD Setting data

To create a new tool, call the tool list and search for a free location.

Figure 5-5 Tool list - free location

New tool Select the "New tool" softkey.

Select the desired tool type from the tool catalog displayed. This tool type is inserted in the tool list and you can enter the data of the tool.

#### Note

Milling cutter 8 (CUTTER\_8) must immerse, as it is used for milling a pocket.

#### Well equipped

5.3 Tools in the magazine

# 5.3 Tools in the magazine

In the following you will learn how to insert the tools into the magazine. In the tool list, select a tool without a location number.

ţ_C												08/11/1 12:56 PM
Tool lis	st									NC	memory	Tool
Loc.	MT LO.	Туре	Tool name	ST	D	Н	Length X	Length Z	ø		Tip 🗅 angli	measure
2/18												New
2/11												tool
2/12												
2/13												Edges
2/14												Lugos
2/15												
2/16												
			Milling_tool_8	1	1	0	0.000	50.000	2.000	4		
		=	Milling_tool_5	1	1	0	0.000	50.000	12.000	4		
		=	End_mill_5	1	1	0	0.000	50.000	12.000	4	_	heal
			End_mill_7	1	1	0	0.000	0.000	12.000	4		LUAU
		<b>2</b>	DRILL_10	1	1	0	80.000	120.000	10.000		118	
			Roughing_tool_C2	1	1	0	50.000	10.000	0.200	→	93	Delete 📐
		<b>P</b>	Plunge_cutter_1_C2	1	1	0	50.000	20.000	0.200		3.00	tool
		<b>1</b>	Plunge_cutter_2	1	1	0	50.000	20.000	0.200		3.00	
		T,	Plunge_cutter_4	1	1	0	50.000	10.000	0.200		3.00	Magazine
		I,	Plunge_cutter_4_C2	1	1	0	50.000	10.000	0.200		3.00	selection
		U,	Punge_cutter_3	1	1	0	50.000	10.000	0.200		3.00	
											×	<b>N</b> N
						<					>	
	_										>	
	Tool list		vear <b>DEM</b> Tool		ľ	laga- zine	l 💽 ₩a	set R	User variable			SD Setting data

Figure 5-6 Select the tool in the magazine

Press the "Load" key. The following dialog offers the first free magazine location for you to change or accept directly.

<u>j</u>												08/11/11 12:56 PM
Tool lis	st									HC I	memory	
Loc.	MT LO.	Туре	Tool name	ST	D	Н	Length X	Length Z	ø		Tip 🗅 angle	
2/10												
2/11												
2/12												
2/13				1.0	ad	on						
2/14	_			LU	au							
2/15												
2/16												
	_		mogozine		1		Locat		12	4		
			mayazine		•		LUCAL		13	4		
		8=	End mill 7	1	1	ß	0 000	0 000	12 000	4		Multitool
			DBILL 10	1	1	Å	80.000	120.000	10.000		118	
			Roughing tool C2	1	1	Ø	50.000	10.000	0.200	→	93	
			Plunge_cutter_1_C2	1	1	0	50.000	20.000	0.200		3.00	
			Plunge_cutter_2	1	1	0	50.000	20.000	0.200		3.00	
		T,	Plunge_cutter_4	1	1	0	50.000	10.000	0.200		3.00	X
			Plunge_cutter_4_C2	1	1	0	50.000	10.000	0.200		3.00	Cancel
		1	Punge_cutter_3	1	1	0	50.000	10.000	0.200		3.00	
											~	$\checkmark$
						<					>	OK
											_	

Figure 5-7 Enter and/or accept a magazine location

ţ_O	] \^ \	₩ DG											08/11/11 12:46 PM
Tool lis	t									Mag	gazine	1	Tool
Loc.	MT L0.	Туре	Tool name	ST	D	Н	Length X	Length Z	Radius			^	measure
1/1			ROUGHING_T80_A	1	1	0	55.848	39.124	0.800	←	93		
1/2		<b>8</b>	DRILL_32	1	1	0	80.000	185.124	32.000		180	L	
1/3		0	Finishing_t35 a	1	1	0	123.976	57.370	0.400	←	93		
1/4		•	ROUGHING_T80_I	1	1	0	55.800	39.000	0.800	←	95		Edges
1/5			PLUNGE_CUTTER_3_A	1	1	0	85.124	44.124	0.200		3.00	L	Luges
1/6		1	PLUNGE_CUTTER_3_I	1	1	0	85.952	41.300	0.200		3.00		
1/7		0	FINISHING_T35_I	1	1	0	-12.658	121.877	0.400	←	95		
1/8		$\geq$	THREAD_1.5	1	1	0	66.326	33.333	0.100			-1	
1/9			CUTTER_8	1	1	0	87.833	74.621	8.000	3			
1/10		<b>S</b>	DRILL_5	1	1	0	80.000	185.124	5.000		118		Unland
1/11		.0	BUTTON_TOOL_8	1	1	0	88.112	38.123	2.000				Univau
1/12			THREADCUTTER_M6	1	1	0	80.000	145.000	6.000		1.00		
1/13		<b>S</b>	DRILL_10	1	1	0	80.000	120.000	10.000		118	[	Delete 🔪
1/14													tool 🖊
1/15													
1/16												- [	Magazine
2/1													selection
2/2													
2/3												~	
				_							>		
	Tool list		Tool wear <b>DEM</b> Tool	65 Ø	۲	laga- zine	Uo off:	set Ru	User ariable				SD Setting data

After acceptance, the tool list may look like this.

Figure 5-8 Tool list following acceptance

# 5.4 Gauging tools

In the following you will learn how to calculate tools.

# Procedure



Insert a tool from the tool list into the spindle using the "T,S,M" softkey.

		09/15/11 6:03 AM
NC/MPF/CLOSURES	SIEMENS	Manual
1/ Reset		
Workpiece Position [mm]	T,F,S	
∞ X 33.920 Z −53.640	T FINISHING_T35 A R0,400 3 万 D1 257.000 F 0,000	Hutomatic
SP1 0.000° SP3 0.000°	0.000 mm/min 120%	
51.5	S1 9 X	
	Master 0 50%	
	0.50.100	
		Calibrate probe
		K Back
Real Set Real Maga	Mana Pasia	
T,S,M	tool tion Stock	

#### Well equipped

5.4 Gauging tools



Then switch to the "Gauge tool" menu.

Manual

Enter the probed or turned diameter.

	09/15/11 6:04 AM
NC/MPF/CLOSURES SIEMENS	Select
🖊 Reset	1001
Uorkpiece Position [mm] T.F.S	Course
	Save
∞ X 33.920 I I International 10.400	position
J EO CAO 3 🗗 D1 X124 88	
ζ -53.640	x
SP1 0.000 ° F 0.000	
SP3 0.000° 0.000 mm/min 120%	
	7
י צו	2
Master 0 50%	
Measure: length manual	
Tool data T FINISHING_T35 A D 1	
X 124.000 ST 1	
🚽 🗸 Z 57.000	
R 0.400 X0 80.000	
	Set
	length
	longui
	<b>**</b>
>	васк
Stock	
To I,S,M P20 U0 P workp. tool tion rem.	

Figure 5-9 Tool measurement - input of the X value



The current position of the tool is calculated taking into account the workpiece diameter.

NC/MPF/CLOSURES       Siemens       Select tool						09/15/11 6:04 AM
✓ Reset       Uorkpiece       Position [mm]       T.F.S         ○ X       80.000       T FINISHING_T35 A       R0.400         2       -53.640       3 □ D1       257.000         3 □ D1       257.000       80.000       X100.96         SP1       0.000°       0.000       mm/min       120%         SP3       0.000°       0.000       mm/min       120%         Ylaster       0       50       100         Ylaster       0       50       100         X0       Z       57.000       ST       1         X0       Z       57.000       X0       80.000       St         X0       Z       57.000       X0       80.000       St       1         X0       Z       57.000       X0       80.000       St       1         X0       Z       57.000       X0       80.000       Set       Iength         X0       Z       Stock       Y       Set       Set       Iength	NC/MPF/CLOSURES			SIE	<b>MENS</b>	Select
Uorkpiece       Position [mm]       T.F.S         ∞       X       80.000       Z       -53.640         SP1       0.000°       Image: Constraint of the state of t	// Reset					1001
∞ X       80.000         Z       -53.640         SP1       0.000°         B       0.000         B       0.000         C       0.000         C       0.000         SP3       0.000         C       0.000         SP3       0.000         C       0.000         SP3       0.000         SP3       0.000         C       0.000         Measure: length manual       Tool data         X       100.960         Z       57.000         R       0.400         X0       80.000         Set       Paint Measure: Posi-         Set       Posi-         Set       Posi-         Set       Posi-         Set       Posi-         Set       Posi-         Set       Posi-	Uorkpiece Position	[mm]	T.E.S	s		-
<ul> <li>x</li> <li>x</li></ul>		00		- FINICHING T35.0	D0 /00	Save
Z       -53.640         SP1       0.000°         SP3       0.000°         0.000°       0.000°         Image: Set of the set of	∞ X 80.0	100	I '		757 000	position
Z       -53.040       x         SP1       0.000°       F       0.000       x         SP3       0.000°       0.000       mm/min       120%         Measure: length manual       T       FINISHING_T35 A       D 1         X0       Z       57.000       ST       1         X0       Z       57.000       x0       80.000         Set       Meas.       F       Meas.       F       Stock         X1       Set       Meas.       F       Meas.       F       Stock	7 500	10	3	🗾 D1	X100 96	
SP1 0.000° SP3 0.000° Measure: length manual Tool data T FINISHING_T35 A D 1 X0 Z 57.000 R 0.400 X0 80.000 Set length Set weeks a track of the set weeks a track	2 -53.6	40	F	0.000	7100.00	х
SP3       0.000°       0.000 mm/min       120%         S1       0       Image: Set of the set of	SP1	0.000°	Г	0.000		
S1         0         Z           Master         0         50         50%           Master         0         50         100           Measure:         Length         Tool data         T         FINISHING_T35 A         D 1           X0         Z         57,000         ST         1         0         50           X0         Z         57,000         X0         80.000         Set         Iength           Iength         Stock         Image: Set         Image: Meas.	SP3	0.000°		0.000	mm/min 120%	
Master         0         50%           Measure:         length         50%         100           Master         0         50%         100           Measure:         length         Tool data         T         FINISHING_T35 A         D 1           X0         Z         57.000         ST         1         Set         Set         length           X0         R         0.400         X0         80.000         Set         length           Set         Set         Meas.         Posi-         Stock         stock			S	1 🗸 0	Ø	z
Measure: length manual         Tool data         T         FINISHING_T35 A         D 1           X0         Z         57,000         ST         1           X0         Z         57,000         80.000         Set           Image: Set			Mas	ster Ø	50%	
Measure:         Length manual           Tool data         T           X         100.960           Z         57.000           R         0.400           X0         80.000           Set         length           Image: Set         Meas.           Image: Set         Meas.           Image: Set         Meas.           Image: Set         Image: Set			0	. 5,0	. 100	
Tool data         T         FINISHING_T35 A         D 1           X0         2         57.000         ST         1           Z         57.000         X0         80.000         Set           Back         2         57.000         St         1           Junction         2         57.000         St         1           Junction         2         57.000         St         1           Junction         3         6.400         X0         80.000           Set         Junction         St         Set         Set           Junction         Junction         Junction         Set         Set	Measure: length manual					
X0 X 100.960 ST 1 Z 57.000 R 0.400 X0 80.000 Set length Set Back		Tool data		T FINISHING	T35 A D 1	
X0 Z 57,000 0.400 X0 80.000 Set length Set Back Set Longth Set Longth Set Longth Set Longth Set Longth Set Longth Set Longth Set Longth Set Longth Set Longth Set Longth Set Set Longth Set Set Set Set Set Set Set Set		Х	100.960	ST 1		
T.S.M Let Stock a usekn in the set of the se	Va	Z	57.000			
Set length		R	0.400	XØ	80.000	
T.S.M Le Set Meas. Posi- Stock Back						Set
T.S.M Les Looka Prosi- Stock training to be tool tool tool tool tool tool tool too						length
T.S.M Base Meas. Posi- a unkn Meas. Posi- ton tool						Tongar
Back						
T.S.M Set Meas. Meas. Stock						Rook
T,S,M Set Meas. Meas. Posi-					>	Dack
	T,S,M	Meas.	Meas.	Posi- tion	Jeff Stock	

Figure 5-10 Tool measurement - set length X

5.5 Setting the workpiece zero

Repeat for Z.

					09/15/11 6:05 AM
NC/MPF/CLOSURES			SIEME	NS	Select
🥢 Reset					1001
Workpiece Posit	ion [mm]	T,F,S			Saue
. X 80	9 000	T FIN	ISHING_T35 A	R0.400	position
× ∧ 00	0.000	3 🗖	D1	Z57.000	
Z -53	3.640	-		X100.96	v
SP1	0.000°	F	0.000		^
SP3	0.000°		0.000 mm/min	120%	
		S1	<mark>~ 0</mark>	Ø	Z
		Master	0	50%	
Measure: length manual		0	5,0 .	100	
r ieasuie. ienyui manuai	Teal date	т		D 1	
70	X	100.960 ST	1		
L0	Z Ż	57.000 Ref	point Workpiec	e edae	
	• R	0.400 <b>Z0</b>	0.	000	
-	_				Set
					length
					<b>~</b>
				>	Back
D TON Set	Meas.	Meas. Pos	- 1 1 .	Stock	
	workp.	tool 🖓 tior	ı 📔 📄	rem.	

Figure 5-11 Tool measurement - set length Z

# 5.5 Setting the workpiece zero

In the following you will learn how to set the workpiece zero.

# Procedure



To set the workpiece zero, switch to the **Machine - Manual** mode in the main menu. Move the workpiece zero if this does not lie on the end face of the workpiece.

	09/15/11 6:05 AM
NC/MPF/CLOSURES	SIEMENS Select
// Reset	Work ons.
Workpiece Position [mm]	T,F,S
∞ X 80.000	T FINISHING_T35 A R0.400 3 ₫ D1 257.000
∠ − <b>33.040</b> SP1 0.000° SP3 0.000°	F 0.000
515 0.000	
	Master 0 50%
Measure: front edge	
Z0 Z	UO Uork offset G54 8.888 Meas. only 0.888 Uork offset
Measur 20	red values Set U0
	Back
T,S,M Set Meas.	Meas. Posi-

Figure 5-12 Enter the work offset

5.5 Setting the workpiece zero



Confirm your input.

	09/15/11 6:06 AM
NC/MPF/CLOSURES	SIEMENS Select
🖊 Reset	Work otts.
Workpiece Position [mm]	T,F,S
<ul> <li>X</li> <li>80.000</li> <li>Z</li> <li>0.000</li> <li>SP1</li> <li>0.000°</li> <li>0.000°</li> </ul>	T       FINISHING_T35 A       R0.400         3       D1       257.000         X100.96       X100.96         0.000       mm/min       120%         S1<
<b>H</b> #ic54	Master 0 50%
Measure: front edge	3
20 Z0	0 Uork offset G54 -53,640 Meas. only 0.000 Uork offset
Measured 20	d values 0.000
T,S,M J= Set J= Meas.	Meas. V Posi- tool V tion rem.

Figure 5-13 Work offset set

# Example 1: Taper shaft

# 6.1 Overview

# Learning objectives

This section will explain the first steps to create a workpiece in detail. You will learn how to...

- create and manage programs;
- call tools;
- enter traversing paths;
- create any contours with the contour calculator;
- rough and finish contours;
- create a thread undercut,
- thread and
- grooves.

# Task



Figure 6-1 Workshop drawing - Example 1:

Example 1: Taper shaft

6.1 Overview



Figure 6-2 Workpiece - Example 1:

# Note

ShopTurn always saves the last setting selected with the toggle key. Therefore, make sure that all units, texts and symbols are specified as in the dialog boxes shown here in the relevant input fields and all toggle fields.

Whenever it is possible to switch, this is indicated in the help text (see screenshot below).



Figure 6-3 Example toggle field

# **Operating sequences**

After power-up of the control system, you are in the main menu.

M												05/20/1 12:36 Pt
								S	IEM	EN	S	G functions
// Reset			-1									
Plachine		Position [mi					,F,S F	_	_	-	_	Auxiliary
XI		0.00	0				1					Tunctions
Z1		0.00	0				-					
						1	-	0.000			05%	
							C 4	0.000	mm/	min	85%	
							51	0			100%	
						0			50		100 /0	
												Act. values
												Machine
	_	_	_						_		>	
на тен	4	Set	Meas.	۵t	Meas.	2.	Posi-				Stock	
<b>1,3,</b>	20	WO 🕨	ø workp.	Ľ↓	tool	50	tion	_			rem.	

Figure 6-4 Main screen



Open the main menu using the **MENU SELECT** key. You can call the various areas of ShopTurn from the main menu.

M	<b>∧</b> 20						05/20/10 12:36 Ph
// Denat					SIEM	ENS	→ AUTO
Machine	Position	[mm]		TES			
X1	0 G	100		T			MDA
21	0.0	200		•			
21	0.0	000		F	0.000		
					0.000 mm	n/min 85%	
				S1 ~	0	Ø	REPOS
				.0	8 50	100%	1
							×
.M.	IO		<b>F</b>	$\wedge$	2		
Machine	Parameter	Program	Program	Diag-	Start-up		
			manager	nostics			

Figure 6-5 Main menu



Select the Program Manager softkey. The Program Manager is opened.

In the Program Manager, you can manage process plans and contours (e.g. "New", "Open", "Copy", ...).

G	*** \$200							05/19/1 2:32 PI
		Name	)	Тур	be Leng	th Date	Time	Evecute
🖶 🗂 Par	t progra	ms		DI	R	05/19/10	2:19:47 PM	LAGGUIG
🖶 🗂 Sub	program	ns		DI	R	05/19/10	2:19:11 PM	
🖻 🗀 Workpieces		DI	R	05/19/10	2:30:40 PM			
							New	

Figure 6-6 Program Manager



The Program Manager displays a list of the existing directories. Use the cursor key to select the 'Workpieces' directory.



Open the "Workpieces" directory.



Enter the name 'EXAMPLE1' for the workpiece.

	New workpiece	
Туре	Workpiece WPD	~
Name EXAMP	LEI	

Figure 6-7 Creating a workpiece



Confirm your input. The following dialog box is opened:

	New sequential program	
Туре	ShopTurn	~
Name <mark>TAPE</mark>	r_shaft	

Figure 6-8 Creating a step sequence program



Select the input format using the **ShopTurn** and **ProgramGUIDE G code** softkeys. The program type can be specified via the **ShopTurn** softkey.

Specify the name of the process plan, in this case 'TAPER\_SHAFT'.



Confirm your input.

				06/27/1 11:38 Al
NC/WKS/EXAMPLE1/TAPER_SHAFT	Progra	am header		Work
p	Work	offset	G54	offset
END	descri	be	Yes	
	ZV	0.000		Graphic
	Blank	(	Cylinder	view
	XA	80.000		
	ZA	1.000		
	ZI	-100.000	abs	
	ZB	-92.000	abs	
	Retra	ct	Simple	
	XRH	5.000	inc	
	700	F 000		
	2КН	5.000	Inc	
	Tool	hongo noint	Harksiese	
	VT	nange point 100 000	workpiece	
	쓝	20.000		
	\$1	3500.000	rom	
	51	0.000	rpin	×
	SC	1.000	the set	Cancel
	Flachi	ned dir. of rota	υρ-сиτ	Gancer
				Occent
			>	несерг
Edit - Drilling - Turn- Cont.	Millin	o 📒 Vari-	te Simu-	NC Ex-
ing turn.		ous	lation	ecute

After confirming, the following interactive screenform is displayed to enter the workpiece data.

Figure 6-9 Program header - Help display

Enter the workpiece data and general program specifications in the program header. Enter the following values:

Field	Value	Selection via toggle key	Notes
Unit of measurement	mm	Х	
Work offset		Х	
Blank	Cylinder	Х	Select the blank shape (here cylinder) using the toggle button.
XA	80		
ZA	1		
ZI	-100 abs	Х	
ZB	-92 abs	Х	The value ZB indicates the distance from the chuck.
Retraction	normal	Х	See below Retraction
XRA	5 inc	Х	The dimensions of the
ZRA	5 inc	Х	retraction planes
Tool change point	WCS	Х	(absolute or incremental) and the tool change point
XT	120		are entered here.
ZT	200		
Safety clearance SC	1		
Speed limits S1	3500		
Machining direction	Synchronous	X	



Accept the entered values. After confirming, the program header is displayed.

_		
NC/	/WKS/EXAMPLE1/TAPER_SHAFT	
Р	Program header	$\overline{}$
END	End of program	

Figure 6-10 Program header, example 1 - Work step editor

Now the program has been created as the basis for further machining steps. It has a name (in the blue bar), a program header (pictogram "P") and a program end (pictogram "END"). The individual machining steps and contours are stored in the program one beneath the other. The later machining is performed from top to bottom.



You may call the program header again at any time to make changes or check the values.

#### Retraction

It is possible to switch the retraction plane between normal, extended and all. Depending on the retraction setting, the associated fields are enabled for the input of the distances.

# **normal** (for simple cylinders)

(IOI SIMPle Cylinders)



Retract	9	Simple			
XRA	5.000	inc			
ZRA	5.000	inc			

extended (for complex workpieces with internal machining)

Retract

xra Xri

ZRA

ZRA

Extended 5.000 inc

5.000 inc

5.000 inc

(for complex workpieces with internal machining and/or relief cuts)

ZRA



all

Retract	All	
XRA	5.000 inc	
XRI	5.000 inc	
ZRA	5.000 inc	
ZBI	0.000	

Example 1: Taper shaft 6.3 Calling a tool

# Softkeys

Graphic view Use this softkey to switch to the online graphic of the workpiece (see screenform below).

NC/UKS/EXAMPLE1/TAPER_SHAFT Program header Unit at the distant Q P X8 Uork offset GS4 offset as the describe No	) et
P XØ Uork offset G54 offs END describe No	eť
END describe No	
Grap	nic
Blank Cylinder view	J
XA 80.000	
199 70 1 999	
	_
ZR -92.888 abs	
Retract Simple	
XRA 5.000 inc	-
· · · · · · · · · · · · · · · · · · ·	
ZRA 5.000 inc	
	_
Tool change point Workpiece	
XI 120.000	
-100 21 200.000	-
51 5300.000 rpm	_
SU 1.000	ol
Plachined dir. of rota Up-cut	
-200 100 50 0	_
-188 -38 8 →Z	nt
Edit Torn Drilling I Turn- Cont. To Milling I Vari- Simu- Right e	Ex- cute

Figure 6-11 Program header - graphical view



Use this softkey to switch back to the help display.

# 6.3 Calling a tool

## **Operating sequences**

To call the required tool, proceed as follows:

Use this key to extend the horizontal softkey menu.



Select the Straight line Circle softkey.



Kreis

Select the Tool softkey.

6.3 Calling a tool

Select tool



To program

Use the cursor key to select the ROUGHING\_T80 A tool.

Accept the tool into your program. After accepting the tool, enter the following values in the interactive screenform (if necessary, change the unit using the toggle key):

Field	Value	Selection via toggle key	Notes
Spindle	V1	Х	Select main spindle V1.
Cutting rate	240 m/min	Х	
Plane selection	Turning	Х	

_			85/28/18 1:84 PM
NC/	WKS/EXAMPLE1/TAPER_SHAFT	Tool	Select
P	X	T ROUGHING_T80 A D 1	tool
END	1	V1 240 m/min	
	288	Ebenenanwahl Turning	

Figure 6-13 Tool - input



# 6.4 Specifying the distance to be traversed

# **Operating sequences**

Now enter the distances to be traversed:

Select the Straight line softkey.



Select the "Rapid traverse" softkey.

Enter the following starting point for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	82 abs	Х	
Z	0.3 abs	Х	



Enter the starting point for the traversing path



"Apply" the set values.

Straight

Select the Straight line softkey.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	-1.6 abs	Х	The tool has a 0.8 radius, so it must be traversed to diameter X -1.6.
F	0.3 mm/rev	Х	

6.4 Specifying the distance to be traversed



Figure 6-14 Specify the traversing path



"Apply" the set values.

Select the Straight line softkey.

Straight

Rapid traverse Select the "Rapid traverse" softkey. Move the tool away from the end face in rapid traverse. Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	1 abs	Х	



Figure 6-15 Enter the traversing path - moving away from the end face

#### Example 1: Taper shaft

6.4 Specifying the distance to be traversed



"Apply" the set values.

Straight

traverse

Rapid

Select the "Rapid traverse" softkey.

Select the Straight line softkey.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
x	82 abs	Х	This entry returns the tool to the starting point.



Figure 6-16 Enter the traversing path - returning to the starting point



"Apply" the set values.

Straight

Select the Straight line softkey.

6.4 Specifying the distance to be traversed

Create the four further traversing paths according to the following list of work steps.

			06/27/11 12:23 PM
NC/	uks/	/EXAMPLE1/TAPER_SHAFT 10	Tool
Ρ	HØ	Program header Work offset G54	1001
Т	N5	Turning T=ROUGHING_T80_A S1=240rev	
	N6	RAPID X82 Z0.3	Straight
	N7	F0.3/rev X-1.6	
<b>→</b>	N8	RAPID 21	Cirolo
<b>→</b>	N9	RAPID X82	center
	H10	RAPID 20	Conter
	H11	F0.25/rev X-1.6	Circle
	N12	RAPID 21	radius
<b></b>	N13	RAPID X120 Z200 →	
END		End of program	
			Polar
			Retract/
			Approach
		📕 Strght 🛛 🛛 🙀 Meas.	
		Circle Griefer Circle	

Figure 6-17 Enter the traversing path - four further traversing paths



Start the simulation.



Figure 6-18 Simulation – side view



You may end simulation by selecting either the **Simulation** softkey again or any other softkey.

# 6.5 Creating contours with the contour calculator and machining sequence

## Contour calculator

The integrated ShopTurn contour calculator allows you to enter even the most complicated contours easily.



With the graphical contour calculator, you can enter the contours faster and more easily than with conventional programming - and even without any mathematical knowledge.

## **Operating sequences**

Proceed as follows to enter the contour:



Select the Contour turning softkey.



Select the **New contour** softkey. Enter the name 'TAPER\_SHAFT\_CONTOUR' for the contour.

Each contour is assigned its own name. This provides for better legibility of the programs.



Figure 6-19 Creating the 'TAPER\_SHAFT\_CONTOUR' contour



Confirm your input.

You can accept the starting point for the contour line without making any changes (see illustration below).

#### Note

The contour line is in one sense the roughing limit and in another the finishing path.



Figure 6-20 Specifying the starting point

#### Note

If you deselect the Graphic view softkey, detailed help displays are shown.



Confirm your input.

‡

Enter the following values for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	30 abs	Х	
Transition to next element	Chamfer	Х	Attach the chamfer (FS)
FS	1.5		directly to the straight line as a transition element.



Figure 6-21 Specifying the vertical straight line contour



# Example 1: Taper shaft

6.5 Creating contours with the contour calculator and machining sequence



Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes	
Z	-17 abs	Х	A straight line appears up	
Transition to next element	Chamfer	Х	to Z-17.	
FS	0			
			The thread undercut is inserted later as an individual element.	



Figure 6-22 Specifying the horizontal straight line contour



<b>‡</b>

Enter the following value for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	40 abs	Х	Draw the vertical straight
Transition to next element	Radius	Х	line up to the
R	2.5		dimensioned intersection including the rounding to the next element.



Figure 6-23 Specifying the vertical straight line contour





In the interactive screenform, enter the following values for the end point of the inclined straight line:

Field	Value	Selection via toggle key	Notes
Х	50 abs	X	
Z	-30 abs	Х	
Transition to next element	Chamfer	Х	+X
FS	0		+Z



Figure 6-24 Specifying the contour end point of the inclined straight line



←•→
-----

Enter the following values for the horizontal straight line in the interactive screenform:

Value	Selection via toggle key	Notes
-44 abs	Х	
Radius	Х	
2.5		+X
	Value-44 absRadius2.5	ValueSelection via toggle key-44 absXRadiusX2.5



Figure 6-25 Specifying the horizontal straight line contour



## Example 1: Taper shaft

6.5 Creating contours with the contour calculator and machining sequence



Enter the following values for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
x	60 abs	Х	The paths (= main elements) do not meet <b>tangentially</b> .
			3 main elements



Figure 6-26 Specifying the vertical straight line contour



←•→	
-----	--

Enter the following values for the horizontal straight line in the interactive screenform:

	1		
Field	Value	Selection via toggle key	Notes
Z	-70 abs	Х	The grooves are entered later as individual elements in exactly the same way as the thread undercut.
Transition to next element	Radius	Х	
R	1		



Figure 6-27 Specifying the horizontal straight line contour



## Example 1: Taper shaft

6.5 Creating contours with the contour calculator and machining sequence



Enter the following value for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
X	66 abs	Х	
Transition to next element	Radius	Х	
R	1		
			+ Z



Figure 6-28 Specifying the vertical straight line contour


6.5 Creating contours with the contour calculator and machining sequence

|--|

Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	-75 abs	Х	
Transition to next element	Radius	Х	
R	1		+X
			+Z



Figure 6-29 Specifying the horizontal straight line contour



#### Example 1: Taper shaft

6.5 Creating contours with the contour calculator and machining sequence



Enter the following value for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	80 abs	Х	End point X80 with a
Transition to next element	Chamfer	Х	2x45° chamfer
FS	2		
			سر
			+ X + Z + Z



Figure 6-30 Specifying the vertical straight line contour



6.5 Creating contours with the contour calculator and machining sequence



Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	-90 abs	X	
Transition to next element	Chamfer	Х	
FS	0		+X +C
			The contour end point lies at X80 and Z-90 (2mm in front of the chuck).



Figure 6-31 Specifying the contour end point





Figure 6-32 Full contour

#### Example 1: Taper shaft

6.5 Creating contours with the contour calculator and machining sequence

Accept

Accept the contour into your process plan.



Figure 6-33 Contour in the process plan

To be able to machine the created contour, you must now create the following work steps. To this end, proceed as follows:

Stock removal Select the Stock removal softkey.

Select tool

Open the tool list and select ROUGHING\_T80 A.

To program Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes		
F	0.3				
V	240 m/min	Х			
Machining	longitudinal external roughing	X X X			
D	2.5				
UX	0.5				
UZ	0.2				
DI	0.0				
BL	Cylinder	Х			
XD	0.0 inc	Х			
ZD	0.0 inc	Х			
Relief cuts	No	Х			
Set machining area limits	No	Х			

6.5 Creating contours with the contour calculator and machining sequence



Figure 6-34 Roughing the contour



Accept the entered values.

Stock removal

> Select tool

Open the tool list and select FINISHING\_T35 A.



Accept the tool into your program.

Select the Stock removal softkey.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	Х	
Machining	Finishing	Х	

6.5 Creating contours with the contour calculator and machining sequence



Figure 6-35 Finishing the contour



Accept the entered values.

The two machining steps are linked in the work step editor.

							06/30/1 12:25 Ph
NC/	uks/example1/1	raper_shaft				13	New
Р	Program header		Work offset G	54		^	contour
т	Turning T=ROUGH	IING_T80_A S1=240	irev				Stock
	RAPID X80 20.3						removal
	FØ.3/rev X-1.6						Tomovar
<b>→</b>	RHPID 21						Cut resid
	RHPID X82						stock
	FR 2E / row V-1 6						
_	PODID 71						Groowing
_	ROPID 210 ROPID 2120 7200						arooving
10-	Contour		TOPER SHOE	CONTOUR			
Se.	Stock removal	$\nabla$	T=ROUGHING	T80 0 F0.3/re	u U=240m		Groove
1111. 	Stock removal	<u><u></u> <del> </del> <del> </del></u>	T=FINISHING	T35 A F0.15/re	v V=200m		resid.
END	End of program						Disease
						_	Plunge
						_	tarning
						_	Plunge
						_	turn.resid.
						_	
						>	
	à	Turn-	Cont		💻 llari-	simu-	INC Ex-
	Edit 📘 Dr	illing 📂 ing	turn.	Milling		lation	ecute

Figure 6-36 Linking of the work steps in the process plan



Select the Simulation softkey.



Select the Side view softkey.



The following simulation shows the manufacturing sequence for you to check before manufacturing the workpiece.

Figure 6-37 Simulation – side view

# 6.6 Thread undercut

# **Operating sequences**

Proceed as follows to create a thread undercut:



Figure 6-38 Thread undercut



Select the **Turning** softkey.

Select the Undercut softkey.

Select the Undercut thread softkey.



Open the tool list and select the finishing tool FINISHING\_T35 A .

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1

# *Example 1: Taper shaft 6.6 Thread undercut*

To program Accept the tool into your program.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	Х	
Machining Roughing/ finishing		x	
	Longitudinal	Х	
Position		Х	(see illustration above)
X0	(0 30		
ZO	-17		
X1	1.15 inc	Х	
Z1	4.5 inc	Х	
R1	0.8		
R2	0.8		
α 30			
VX	1 inc	Х	
D	0.8		
U	0.1	X (field)	



Figure 6-39 Thread undercut



Switch between the graphic view and the help display as required.

Figure 6-40 Thread undercut - graphic view

Accept the entered values.



Select the **Simulation** softkey. Check the thread undercut via the detailed view in the 3D view, for example.

Select the 3D view softkey.

Details

Select the **Details** softkey. You can manipulate the display using the Zoom +, Zoom -, Magnifying glass, etc. softkeys.



Figure 6-41 Detailed view of the simulation in the 3D display

6.7 Thread

# 6.7 Thread

# **Operating sequences**

Proceed as follows to create a thread:



Figure 6-42 Thread

Select the Thread softkey.

Thread

Open the tool list and select THREADING\_T1.5.

tool To program

Select

Accept the tool into your program.

Enter the following values for the thread in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Р	1.5 mm/rev		
G	0		
S	800 rpm	Х	
Machining	Roughing/finishi ng	x	
	Linear	Х	
	External thread	Х	
X0	30	Х	The following inputs
ZO	0		define the thread
Z1	Z1 -16 abs		geometrically.
LW	2		
LR	1		
H1	0.92		
αP	29	Х	
ND	8		
U	0.1		
NN	0		
VR	2		
Multiple threads	No	X	
α0	0		



Figure 6-43 Thread - graphic view

Switch to the help display when necessary.



Figure 6-44 Thread - help display



6.8 Grooves



Figure 6-45 Thread simulation

# 6.8 Grooves

### **Operating sequences**

Proceed as follows to create the two grooves:







Select the Groove softkey.



Select the Groove 2 softkey.



Open the tool list and select the PLUNGE\_CUTTER\_3 A plunge cutter.



Accept the tool into your program.

Field	Value	Selection via toggle key	Notes			
F 0.1						
V	150 m/min	Х				
Machining Roughing/finishi ng						
Position			(see illustration above)			
X0	60		The following inputs			
ZO	-65		define the grooves			
B1	6	X (field)	geometrically.			
T1	3 inc					
α1	1 0					
α2	α2 0					
FS1	0.5	X (field)				
R2	1	X (field)				
R3	1	X (field)				
FS4 0.5		X (field)				
D	3					
U	0.1	X (field)				
N	2					
DP	10					

Enter the following values for the grooves in the interactive screenform:



Figure 6-47 Grooves - graphic view

6.8 Grooves

_		~~~											07/01/10 1:14 AM	3
NC/U	JKS/EXA	MPLE	I/TAPER	_SHAF	Т			6	Groov	e 2			Select	
Р									Т	PLUNGE_CUTT	ER_3A [	) 1	tool	J
т									F	0.100	mm/rev			
									V	150	m/min		Graphic	
									Machi	ning	0+000		view	
									Pos.		1 1			
			DP						X0	60.000				
_		-							20	-65.000				
	_				_	_		_	B1	6.000				
	1		2	A .			N			3.000	inc		8.8	
1000									α 1 ~2	0.000	0			
							12		EC1	0.000			i i i i i i i i i i i i i i i i i i i	
51									R2	1 999			Br	
M									R3	1 899				
Mi.									FS4	0.500				
	_		-				-	-	D	3.000				
THE AUX									U	0.100			-	
END											Ristance of		Ű	
									H	2	grooves		×	
									DP	10.000			Cancel	
													1	ĺ
										_	_		Accept	
	_				-		0.1					-		ļ
	Edit		Drill.		ing		Cont.		Mill	Vari-		imu-	NC Ex-	
					mg		turii.			ous	i i i	auoli	ecute	1

Switch to the help display when necessary.

Figure 6-48 Grooves - help display



Accept the entered values.

							06/30/1 12:51 Ph
NC/	/UKS/EXAMPLE1/TAPE	R_SHAFT				16	Stock 📐
Ρ	Program header		Work offset G	54		^	removal
Т	Turning T=ROUGHING_	T80_A S1=240	lrev				
<b>→</b>	RAPID X80 20.3						Groove
<b>→</b>	F0.3/rev X-1.6						
<b>→</b>	Rapid 21						
<b>→</b>	Rapid X82						Undercut
	Rapid Z0						
<b>→</b>	F0.25/rev X-1.6						
	Rapid Z1						Thread
<b>→</b>	RAPID X120 Z200						
U,	Contour		TAPER_SHAFT	_CONTOUR			
nh.	Stock removal	$\nabla$	T=ROUGHING_	_T80_A F0.3/re	ev V=240m		Cutoff
Mr.	Stock removal	$\Delta \Delta \Delta$	T=FINISHING_	T35 A F0.15/re	v V=200m		
<u>ا</u>	Undercut thrd	44000	T=FINISHING_	T35 A F0.15/re	v V=200m X0	=30 20=-17	
	I hread long.	4400	T=THREADING	_11.5 P1.5mm	/rev S=800re	v Outside	
Soll.	Groove	44444	T=PLUNGE_C	JTTER_3 H FU.1	1/rev V=150m	N2 X0=ti→	
END	End of program					_	
						~	
						>	
	🛛 Edit 📑 🕇 Drilling	Turn- ing	Cont.	Milling	Vari- ous	Simu- lation	Ex- ecute

Figure 6-49 Process plan with grooves



Start the simulation in the side view or in the 2-window view, for example.



Figure 6-50 Simulation – side view

Select the Side view softkey.



Side view

Select the Further views softkey.



Select the 2-window softkey.



Figure 6-51 Simulation – 2-window view

Example 1: Taper shaft

6.8 Grooves

# 7.1 Overview

### Learning objectives

In this section you will learn the following new functions. You will learn how to...

- perform face turning;
- work with the contour calculator (advanced application);
- machine residual material.

### Task



Figure 7-1 Workshop drawing - Example 2:



Figure 7-2 Workpiece - Example 2:

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1 7.2 Face turning

# Preparation

Perform the following steps without help:

- 1. Create a new workpiece with the name 'EXAMPLE2'.
- 2. Create a new step sequence program with the name 'DRIVE\_SHAFT' .
- 3. Specify the blank dimensions (for the procedure, see example 1).



Figure 7-3 Creating a program header

Following creation of the program header, the process plan looks like this.

	05/20/1 3:29 P
NC/UKS/EXAMPLE2/DRIVE_SHAFT P Program header	1 Stock removal
END End of program	Groove

Figure 7-4 Work step program

# 7.2 Face turning

# **Operating sequences**

Proceed as follows to face turn the workpiece:



Select the **Turning** softkey.



Since face turning should be completed in one cut, switch to finishing during machining. Select the ROUGHING\_T80 A tool and enter the following values.



Figure 7-5 Face turning the workpiece

# 7.3 Creating the contour, stock removal and residual stock removal

#### **Operating sequences**

Proceed as follows to enter the contour:





Select the Contour turning softkey.

7.3 Creating the contour, stock removal and residual stock removal



Select the **New contour** softkey. Enter the name 'DRIVE\_SHAFT\_CONTOUR' for the contour.



Figure 7-6 Creating the contour



Confirm your input.

You can accept starting point X0/Z0 directly (see illustration below).

		₩2 9 9										05/2 3:33	0/10 3 PM
NC/	UKS	/EXAM	PLE2/DRIV	_shaft		St	tartin	g point					
P	⊕ END	1°				x	:	0.0	<mark>100</mark> a	bs			
END						2		0.0	300 a	bs		Graphic view	
						Tr	ransi	tion at con Radius	tour s	tart			
		0.1				R	1	0.0	000 ¥				
		0		-									
		-0.1											
		-0.2	-0.05	0	0.05	.,Z						$\sim$	
											>	нссерт	
	E	dit	Drill.	ing	turn.	•	Mill.		ari- ous		lation	ecu	te

Figure 7-7 Accepting the starting point



Confirm your input.



Enter the following values for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	16 abs	Х	
Transition to next element	Radius	Х	
R	2		+X +Z



Figure 7-8 Specifying the vertical straight line contour



7.3 Creating the contour, stock removal and residual stock removal



Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	-16 abs	Х	
Transition to next element	Chamfer	Х	
FS	0		+X +Z



Figure 7-9 Specifying the horizontal straight line contour





Enter the following value for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	24 abs	Х	
Transition to next element	Chamfer	Х	
FS	2		+X +Z



Figure 7-10 Specifying the vertical straight line contour



7.3 Creating the contour, stock removal and residual stock removal



Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	-38 abs	Х	
Transition to next element	Chamfer	Х	
FS	0		



Figure 7-11 Specifying the horizontal straight line contour





Enter the following values for the sloping straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	20 abs	Х	
α2	45	Х	
Transition to next element	Chamfer	Х	+X to +z
FS	0		The entered angle refers to the preceding element.



Figure 7-12 Specifying the sloping straight line contour



7.3 Creating the contour, stock removal and residual stock removal



Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	-53 abs	Х	
Transition to next element	Radius	Х	
R	1		+X +Z



Figure 7-13 Specifying the horizontal straight line contour





Enter the following values for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	36 abs	Х	
Transition to next element	Radius	Х	
R	0.4		+X +X +z
			Round the transition to the next element with R0.4.



Figure 7-14 Specifying the vertical straight line contour



7.3 Creating the contour, stock removal and residual stock removal



Enter the following values for the next section in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х		Х	
Z		Х	
α1	165.167°		+X to +z
Transition to next element	Radius	Х	Nothing more is known of
R	0.4		the path than the angle to the Z axis, 165.167°. In such cases, simply continue the construction with the next element.



Figure 7-15 Specifying the inclined straight line contour





Enter the following values for the next section in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Direction of rotation	Right	Х	
R	13		
Х			+X +C +Z
Z			The missing points of the
	60 abs	Х	previous contour element
К	-78 abs	Х	are calculated using the
Transition to next element	Chamfer	Х	arc.
R	0		Since there are several possibilities, it is important that the correct selection is made.







Select the recommended solution in accordance with the below illustration.

Figure 7-17 Confirming the contour selection

7.3 Creating the contour, stock removal and residual stock removal



Select the desired construction, then accept.

Since the end point of the arc is not known, simply continue with the construction. You could also use the **All parameters** softkey to enter the exit angle here.



Figure 7-18 Accepting the arc contour



Accept the contour section.

7.3 Creating the contour, stock removal and residual stock removal



A tangential path follows.

Select the Tangent to prec.elem. softkey.

Field	Value	Selection via toggle key	Notes
Х	80 abs	Х	
Transition to next element	Radius	Х	
R	0.4		+X +Z



Figure 7-19 Specifying the vertical straight line contour



7.3 Creating the contour, stock removal and residual stock removal



Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	-100 abs	Х	
Transition to next element	Chamfer	Х	
FS	0		+X +z
			The end point of the contour is at Z-100



Figure 7-20 Specifying the horizontal straight line contour



Accept the entered values.

Accept the contour into your process plan.



Figure 7-21 Accepting the contour

### Stock removal, residual stock removal and finishing

To be able to machine the created contour, you must now create the following work steps. To this end, proceed as follows:



Select the Stock removal softkey.



Open the tool list and select the ROUGHING\_T80 A tool.

To program Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.3		
S	240 rpm	Х	
Machining	contour-parallel external roughing	x x x	The machining of the contour, for example, is performed parallel to the contour here.
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Cylinder	Х	
XD	0.0 inc	Х	
ZD	0.0 inc	Х	
Relief cuts	No	X	
Set machining area limits	No	Х	



Figure 7-22 Roughing the contour

7.3 Creating the contour, stock removal and residual stock removal



view

Accept the entered values.

Select the Simulation softkey.





Figure 7-23 Roughing the contour - side view simulation



Select the Contour turning softkey.

Select the Residual stock removal softkey. Open the tool list and select the FINISHING\_T35 A tool.

То program

Accept the tool into your program.

Field	Value	Selection via toggle key	Notes
F	0.12		
V	240 m/min	Х	
Machining	longitudinal external roughing	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
Relief cuts	Yes	X	To be able to cut all residual material, you must switch the input field to <i>Yes</i> .
FR	0.2		
Set machining area limits	No	Х	

Enter the following values for residual stock removal in the interactive screenform:



Figure 7-24 Cutting contour residual material



Accept the entered values.

Select the Simulation softkey.





Extend the menu.

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1

Show tool path

7.3 Creating the contour, stock removal and residual stock removal

Resi	dual cuttin	g		_	Z2	700.00	0 Feedra	te	80%	00:03:14	
κ 4	11.054 Z	-69.9	948 Y	0.00	90 S1	Q	T FINIS	HING_T35	A	D1	
-100 -140	-120	-100	-80	-60	-40	-20	ç	20	40	60 7.5	Progra contro
-50											Detail
0											Fur the views
-50						_					3D view
100				1				R			Side view
NC/UKS,	/EXHMPLE	2/DRIVE	_SHAFT								Ø

Activate the display of traversing paths.

Figure 7-25 Cutting residual material - side view simulation



Select the Stock removal softkey.

Select the Contour turning softkey.



Open the tool list and select the FINISHING\_T35 A tool.

To program Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.12		
S	280 rpm	Х	
Machining	longitudinal external finishing	X X X	
Allowance	No	Х	
Relief cuts	Yes	Х	
Set machining area limits	No	Х	
#### Example 2: Input shaft

7.3 Creating the contour, stock removal and residual stock removal



Figure 7-26 Finishing the contour



Accept the entered values. After acceptance, the process plan looks like this.



Figure 7-27 Process plan

Start the simulation.



Select the **Details** softkey. Here you can expand or minimize the view.

### Example 2: Input shaft

7.4 Thread



Figure 7-28 3D view simulation - details

## 7.4 Thread

### **Operating sequences**

Proceed as follows to create a thread.



7.4 Thread

Field	Value	Selection via toggle key	Notes
Р	1.5 mm/rev	Х	
G	0		
S	800 rpm	Х	
Machining	Roughing + finishing	X	
	Linear	Х	
	External thread	Х	
X0	24		
Z0	-16		
Z1	-40 abs	Х	
LW	2		
LR	1		
H1	0.92		
αP	29	Х	
	Infeed with alternating flanks	Х	
ND	8		
U	0.1		
NN	0		
VR	2		
Multiple threads	No	Х	
α0	0		

Enter the following values for the thread in the interactive screenform:



Figure 7-29 Producing a thread



Accept the entered values.

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1 7.4 Thread





Figure 7-30 3D view simulation - details

# Example 3: Guide shaft

### 8.1 Overview

### Learning objectives

In this section you will learn the following new functions. You will learn how to...

- create any blank;
- perform stock removal of the difference material between the blank and the machined part;
- drill on the front face;
- mill on the front face;



Figure 8-1 Workshop drawing - Example 3:

### Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1

### Task

### 8.2 Face turning

### Preparation

Perform the following steps without help:

- 1. Create a new workpiece with the name 'EXAMPLE3'.
- 2. Create a new step sequence program with the name 'GUIDE\_SHAFT'.
- 3. Fill in the program header (see illustration below).

### Note

Although you can do this for any blank, select the *Cylinder* blank here. ShopTurn ignores this input and selects an arbitrary blank.



Figure 8-2 Creating a program header

## 8.2 Face turning

### **Operating sequences**



Proceed as follows to create a new program and face turn the blank to Z0: Select the **Turning** softkey.

Select the **Stock removal** softkey. Select the ROUGHING\_T80 A tool.

8.2 Face turning

Field	Value	Selection via toggle key	Notes
F	0.25		
V	240 m/min	Х	
Machining	Finishing	Х	
Position	(see illustration below)	X	
Machining direction	Chart	Х	
XO	60		Since the blank has a diameter of 60mm, you must also set dimension X0 to 60 in this work step.
Z0	2		
X1	-1.6 abs	Х	
Z1	0.0 abs	Х	
D	1.5		
UX	0.0		
UZ	0.2		

Enter the following values in the interactive screenform:



Figure 8-3 Face turning the workpiece



Accept the entered values.



Start the simulation to check the work step.

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1

### Example 3: Guide shaft

8.3 Creating any blank contour



# 8.3 Creating any blank contour

### **Operating sequences**

Enter the following blank contour without help:





Select the New contour softkey. Enter the name 'GUIDE\_SHAFT\_BLANK' for the contour.



Figure 8-5 Creating the contour

In the contour calculator, create the blank contour with the starting point at X0/Z0 (see illustration below).



Figure 8-6 Any blank contour



### Note

The contour must be closed.

8.4 Creating the machined part contour and stock removal

# 8.4 Creating the machined part contour and stock removal

### **Operating sequences**

Proceed as follows to enter the machined part contour:





Select the **Contour turning** softkey.



Select the **New contour** softkey. Enter the name 'GUIDE\_SHAFT\_CONTOUR' for the contour.

New contour
Please enter the new name
guide_shaft_contour

Figure 8-7 Creating the contour



Confirm your input.

Since the blank was assigned to Z0 in the first work step, you can accept the starting point X0/Z0 directly (see illustration below).

		05/20/1 4:29 Pt
NC/UKS/EXAMPLE3/GUIDE_SHAFT	Starting point	
	X 0.000 at Z 0.000 at	S Graphic view
END	Transition at contour st	art 🛛
0.1	FS 0.000 ¥	

Figure 8-8 Specifying the contour starting point



Confirm your input.

8.4 Creating the machined part contour and stock removal

# **‡**

Enter the following values for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	48 abs	Х	
Transition to next element	Chamfer	Х	
R	3		+X ₩⇒_
			÷z



Figure 8-9 Specifying the vertical straight line contour



Accept the entered values.

### Example 3: Guide shaft

8.4 Creating the machined part contour and stock removal



### Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z			$\neg$
Transition to next element	Radius	Х	
R	4		+X +Z
			The end point of the horizontal straight line is unknown. Enter only the transition to the next element with R4. The end point of the straight line is calculated automatically from the subsequent contour constructions.



Figure 8-10 Specifying the horizontal straight line contour



Accept the entered values.

8.4 Creating the machined part contour and stock removal



Enter the following values for the next section in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Direction of rotation	Right	Х	
R	23		
Х	60 abs	Х	+X
Z			∳⇒ +Z
I	80 abs	X	If several solutions are possible when entering the contour data (e.g. in the case of circular arcs here), you can select these via the <i>Selection</i> <i>dialog</i> softkey.



Figure 8-11 Specifying the arc contour



Select the recommended solution in accordance with the below illustration.

Figure 8-12 Selecting an arc contour

### Example 3: Guide shaft

8.4 Creating the machined part contour and stock removal

Dialog accept Select the desired construction, then accept.

Dialog select

Select the recommended solution in accordance with the below illustration.



Figure 8-13 Selecting an arc contour

Select the desired construction, then accept.

NC/UKS	s/example3/guide_shaft	Circle	
P 🕁	Xø	Direction of rotation	
1	60	R 23.000	
U1 +		X 60.000 abs	Graphic
	50	7 0.000 Inc	VIEW
END	50	-37 235 inc	Tongont
ENU		I 80.000 abs	trans
	-40	I 16.000 inc	trans.
		K abs	Change
	-30	K -16.523 inc	selection
		α1 225.921 °	
	28	α2 91 115 771 °	All para-
	20	B2 110 149 °	meters
		FB	
	10	Transition to next element	
		Cham	
	0	FS 0.000	
		FRC	×
	-10		Cancel
	10	CH 0.000	
	-15 -10 -5 0 5 10	15 <sub>2</sub>	
		>	нссерт

Figure 8-14 Accepting the selected arc contour

122

8.4 Creating the machined part contour and stock removal

To complete the arc, proceed as follows:

1. Enter center point K-35 (absolute dimension).



Figure 8-15 Specifying the arc contour center point

2. Enter the transition to the next element with R4.



You can use the existing contour data and the calculated selection options to construct the arc and the straight line (with unknown end point).



Accept the contour section.

### Example 3: Guide shaft

8.4 Creating the machined part contour and stock removal



Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	-75 abs	Х	$\neg$
Transition to next element	Radius	Х	
R	6		+X
			+Z



Figure 8-17 Specifying the horizontal straight line contour



Accept the entered values.

8.4 Creating the machined part contour and stock removal



Enter the following values for the inclined straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	90 abs	Х	
Z	-80 abs	Х	
Transition to next element	Radius	Х	+X
R	4		+Z



Figure 8-18 Specifying the inclined straight line contour



Accept the entered values.

### Example 3: Guide shaft

8.4 Creating the machined part contour and stock removal



Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	-90 abs	Х	
Transition to next element	Chamfer	Х	
FS	0		+X +2
			To prevent damage to the chuck, end the construction at Z-90.



Figure 8-19 Specifying the horizontal straight line contour



Accept the entered values.

Accept the contour into your process plan.



Figure 8-20 Accepting the contour

8.4 Creating the machined part contour and stock removal

### Stock removal

In the following work step you will perform contour stock removal.

To this end, proceed as follows: Select the **Stock removal** softkey.



Select

tool

Open the tool list and select the ROUGHING\_T80 A tool.

To program Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	260 m/min	Х	
Machining	longitudinal external roughing	X X X	
D	2.5		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Contour	X	The blank description must be switched to contour here.
Relief cuts	No	X	To ensure that the recess of radius 23 is not machined, you must switch to <i>No</i> .
Set machining area limits	No	Х	



Figure 8-21 Contour stock removal



Accept the entered values. Both contours and the work step are linked together following acceptance.



Select the Simulation softkey.



Figure 8-22 Contour stock removal simulation (with display of the traversing paths).

The traversing paths in the simulation clearly indicate how the previously constructed blank is taken into consideration.

# 8.5 Residual stock removal

### **Operating sequences**

Proceed as follows to cut the residual material:



The following illustration shows the process plan up to roughing machining:



Figure 8-23 Process plan including roughing machining

Open the tool list and select the BUTTON\_TOOL\_8 tool.



Select the **Contour turning** softkey.

Select the **Residual stock removal** softkey.

To	
program	

Accept the tool into your program.

Enter the following values for residual stock removal in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	240 m/min	Х	
Machining	longitudinal external roughing	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
Relief cuts	Yes	X	Machining with relief cuts must be switched to <i>Yes</i> here.
FR	0.2		
Set machining area limits	No	Х	

8.5 Residual stock removal



Figure 8-24 Cutting contour residual material



Accept the entered values. After acceptance, the list of work steps looks like this:

_							08/05/1 4:23 PM
NC/	WKS/EXAMPLE3/GUIDE	_shaft				6	New 📐
Ρ	Program header		Work offset G54			^	contour
3	Stock removal	$\nabla$	T=ROUGHING_T8	LA F0.25/rev V=2	240m Face X0:	-60	
νı	Blank		GUIDE_SHAFT_BL	ahk			Stock
J.	Fin. part		GUIDE_SHAFT_CO	NTOUR			removal
<b>)</b> -	Stock removal	$\nabla$	T=ROUGHING_T8	1_A F0.3/rev V=20	60m		0.1
de la companya de la comp	Residual cutting	V	T=BUTTON_TOOL	_8 F0.25/rev V=2	40m	$\Box$	stock
END	End of program						SLUCK
							Grooving
							Groove resid.
							Plunge turning
							Plunge turn.resid.
						~	
	Edit <b>E</b> dit Edit	Jurn- ing	Left Cont. Left turn.	Milling	Vari- ous	Simu- lation	NC Ex- ecute

Figure 8-25 Process plan with residual stock removal



Start the simulation.



Figure 8-26 Residual stock removal simulation

After roughing the contour, it must then be finished. Select the **Contour turning** softkey.



Stock removal Select the Stock removal softkey.



Open the tool list and select the FINISHING\_T35 A tool.

To program Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.12		
S	280 m/min	Х	
Machining	longitudinal external finishing	X X X	
Allowance	No	Х	
Relief cuts	Yes	Х	
Set machining area limits	No	Х	

8.5 Residual stock removal



Figure 8-27 Finishing the contour



Accept the entered values.



Start the simulation.



Figure 8-28 Finishing simulation - 3D view

## 8.6 Groove

### **Operating sequences**

Proceed as follows to create a groove:



After residual stock removal, the list of work steps looks like this:





Select the **Groove** softkey.



Select the second groove shape (groove 2).

Select tool Open the tool list and select PLUNGE\_CUTTER\_3 A.

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1 8.6 Groove

To program Accept the tool into your program.

Enter the following values for the groove in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.1 mm/rev		
V	150 m/min	Х	
Machining	Roughing + finishing	X	
Position	see illustration below	x	
X0	60		Here, you enter the
ZO	-67		position and allowance of
B1	4.2	X (field)	the groove.
T1	4 inc	Х	
α1	15		Here, you enter the flank
α2	15		angle and rounding at the
FS1	1	X (field)	corners.
R2	1	X (field)	
R3	1	X (field)	
FS4	1	X (field)	
D	4		
U	0.2	X (field)	
Ν	1		



Figure 8-30 Creating a groove



Accept the entered values. After acceptance, the list of work steps looks like this:



Figure 8-31 Process plan including groove



Start the simulation. You can check subareas of the workpiece using the **Magnifying glass** softkey.



Figure 8-32 Simulation - 3D view (magnifying glass)

8.7 Thread

#### 8.7 Thread

### **Operating sequences**

Proceed as follows to create a thread.





Select the Turning softkey.



Select the Thread softkey.



Open the tool list and select THREADING\_T1.5.



Accept the tool into your program.

Enter the following values for the thread in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Р	1.5 mm/rev	Х	
G	0		
S	800 rpm	Х	
Machining	Roughing Diminishing External thread	X X X	The thread is created with the <i>diminishing</i> setting. This setting causes the cutting division to be reduced for each cut, and so ensures that the cutting cross-section remains constant.
X0	48		
ZO	-3		
Z1	-23 abs	Х	
LW	4	X (field)	
LR	2		

8.7 Thread

Field	Value	Selection via toggle key	Notes
H1	0.92		
αP	29	X (field)	
	Infeed with alternating flanks	Х	
ND	8	X (field)	
U	0.1		
VR	2		
Multiple threads	No	X	
α0	0		



Figure 8-33 Producing a thread

Switch to the help display when necessary.



Figure 8-34 Help display - thread exit

8.8 Drilling



Accept the entered values.



Start the simulation. You can check subareas of the workpiece using the **Details** softkey.



Figure 8-35 3D view simulation - details

## 8.8 Drilling

### **Operating sequences**

Proceed as follows to create drill holes on the front face (C axis or complete machining sequence).



			05/20/10 4:52 PM
NC/WKS/EXAMPLE3/GUIDE_SH	IAFT	1	1 Drilling
P Program header			centric
Stock removal	$\nabla$	T=ROUGHING_T80 A F0.25/rev V240m	I
Contour		GUIDE_SHAFT_BLANK	Centering
U- Contour		GUIDE_SHAFT_CONTOUR	contorning
Stock removal	$\nabla$	T=ROUGHING_T80 A F0.3/rev V260m	
Residual cutting	$\nabla$	T=BUTTON_TOOL_8 F0.25/rev V240m	Drilling
Stock removal	$\nabla \nabla \nabla$	T=FINISHING_T35 A F0.12/rev V280m	Reaming
Groove	$\nabla + \nabla \nabla \nabla$	T=PLUNGE_CUTTER_3 A F0.1/rev V150m	Deer hele
🕅 Thread long.	V	T=THREADING_T1.5 P1.5mm/rev S800r	Deep hole
END End of program			urning
			Tapping
			Positions
	a Turn-	Font June Varia	Position repetit.
🚽 Edit 🗧 Drill. 📑	ing ing	turn.	ecute

After the thread has been created, the list of work steps looks like this:

Figure 8-36 Process plan following thread creation

	_			
	Field	Value	Selection via toggle key	Notes
program	Enter the following values	for the drill hole	in the interactive screenf	orm:
To	Accept the tool into your p	program.		
	• • • • • • •			
Select tool	Open the tool list and sele	ect DRILL_5.		
Drilling	Select the <b>Drilling</b> soffkey.			
	Coloct the Drilling cofficer			
Reaming	g	<b>g</b> ,		,,
Drilling	Select the <b>Drilling Reamin</b>	a softkey. The v	vorkpiece is drilled directly	/. i.e. without centering.
<b>Drill</b> .	Select the <b>Drilling</b> softkey.			
	O THE STOLE DUNING STOLE			

Field	Value	Selection via toggle key	Notes
F	0.06 mm/rev	Х	
V	140 m/min	Х	
	Face	Х	
	Shank	X	The depth reference is switched to <i>Shank</i> .
Z1	10 inc	X	The hole depth can be entered as 10mm incremental or -10mm absolute.
DT	0 s	Х	

8.8 Drilling



Figure 8-37 Drilling



Accept the entered values. After acceptance, the list of work steps looks like this:

			05/20/10 4:54 PM
NC/UKS/EXAMPLE3/GUIDE_SHA	AFT	12	Drilling
P Program header		<u>^</u>	centric
Stock removal	$\nabla$	T=ROUGHING_T80 A F0.25/rev V240m	
U 1 Contour		GUIDE_SHAFT_BLANK	Centering
Contour		GUIDE_SHAFT_CONTOUR	
Stock removal	$\nabla$	T=ROUGHING_T80 A F0.3/rev V260m	
Residual cutting		T=BUTTON TOOL 8 F0.25/rev V240m	Drilling
Stock removal	$\nabla \nabla \nabla$	T=FINISHING T35 A F0.12/rev V280m	Reaming
Groove	⊽+⊽⊽⊽	T=PLUNGE CUTTER 3 A F0.1/rev V150m	
M Thread long.	$\nabla$	T=THREADING_T1.5 P1.5mm/rev S800rev	Deep hole
🖉 - Drilling		□+ T=DRILL 5 F0.06/rev V140m Z1=10in	arilling
END End of program			
			Tapping
			Positions
			Position
		¥	renetit
		>	repetit
Edit Drill	Turn-	🧶 Cont. 💶 🧧 Mill 📒 Vari- 👍 Simu-	NC Ex-

Figure 8-38 Process plan following drilling

During the drilling work step, an open connector appears in the list of work steps. This is automatically linked to the drilling positions in the next step.

Positions

Select the **Positions** softkey.



For the purposes of the exercise, the four drill holes are entered as single positions. A simpler solution here would be to use the position circle.



Figure 8-39 Entering the positions



Accept the entered values. After acceptance, the list of work steps looks like this:

	08/05/11 4:50 PM
NC/WKS/EXAMPLE3/GUIDE_SHAFT	11 Drilling
P Program header Work offset G54	centric
🔚 Stock removal 🛛 🗸 T=ROUGHING_T80_A F0.25/rev V=240m Fac	ce X0=60
Un Blank GUIDE_SHAFT_BLANK	Centering
U   Fin. part GUIDE_SHAFT_CONTOUR	
Stock removal T=ROUGHING_T80_A F0.3/rev V=260m	D. 111
Residual cutting T=BUTTON_TOOL_8 F0.25/rev V=240m	Drilling
Stock removal VVV T=FINISHING_T35 A F0.12/rev V=280m	Reaming
Groove V+VVV T=PLUNGE_CUTTER_3 A F0.1/rev V=150m X	KØ=60
∭ Thread long. ⊽ T=THREADING_T1.5 P1.5mm/rev S=800rev	Outside drilling
S <sup>∞</sup> ] Drilling □• T=DRILL_5 F0.06/rev V=140m Z1=10inc	urning
√ <sup>1</sup> 001: Positions □• 20=0 X0=16 Y0=0 X1=0 Y1=-16 X2=-16 Y2=	<mark>-0 X3=0 →</mark> -
END End of program	Thread
	Donitions
	Posidolis
	Position
	> repetit.
Turn- Cont Vari-	+ Simu- NC Ex-
Edit Drilling ing turn. Milling ous	lation 🖶 ecute



The drill holes are now linked to the drilling positions.

8.8 Drilling



Start the simulation.



Figure 8-41 Simulation - 3D view



Figure 8-42 Simulation - front view

# 8.9 Milling the rectangular pocket

### **Operating sequences**

Proceed as follows to create a rectangular pocket on the front face (C axis or complete machining sequence).



Pocket Rectang. pocket Select the Milling softkey.

Select the **Pocket** softkey.

Select the **Rectangular pocket** softkey.

Select tool To program Open the tool list and select CUTTER\_8.

Accept the tool into your program.

Enter the following values for the rectangular pocket in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.03 mm/tooth	Х	
V	220 m/min	Х	
	Face	Х	
Machining	Roughing	Х	
	Single positions	Х	
X0	0	X (field)	
Y0	0	X (field)	
ZO	0		
W	23		
L	23		
R	4		

#### Example 3: Guide shaft

8.9 Milling the rectangular pocket

Field	Value	Selection via toggle key	Notes
α0	0		
Z1	3 inc	Х	
DXY	75%	Х	
DZ	1.5		
UXY	0		
UZ	0		
Insertion	Helical	Х	see insertion below
EP	1		
ER	7		



Figure 8-43 Producing a rectangular pocket



Accept the entered values. After acceptance, the list of work steps looks like this:

_											05/20/ 5:02	/11 Ph
NC/	uks/example	3/GUIDE_9	Shaft							14		
4	Program hea	der			T. DOUGUT	10 Too		- (	10.40	<b>^</b>		i.
1944	Stock remov	al	V		T=RUUGHIN	16_180 I	H FØ.2	5/rev	V240m			
U.	Contour				GUIDE_SH	IFI_BLH					Pocket	
2	Contour	-1	-		T-DOUCUT	1F1_CUN	100K	( II)	000-			
anti.	SLUCK IEMUV	al	v			10_100 I TOOL 0	E0 25	/rev v	2001		Multi-edg	e
nali.	Stock remou	al	000		T-FINICH	IUUL_0	0 60	12/reu	2408 11200m		spigot	2
3 8	SLUCK TEMOV	ai	01000		T-DI HNCE		205	$\frac{12}{100}$	V2000			
1111 1111	Thread long				T=THREAD	NG T1	5 P1 5	0.1/1C mm/reu	S800reu		Slot	
Arz.	Drilling			(T)+	T=DRTII	5 FØ Ø6	/reu U	140m 7	1=10inc		olot	1
1	001: Positi	ons		+	20=0 X0=1	6 Y0=0	X1=0	Y1=-16	X2=-16	2		
100	Rectang. noc	ket	⊽	0+	T=CUTTER	8 F0.0	3/t U2	20m X0		۱I	Thread	
END	End of prog	ram					-,			1	mining	i.
										1		
											Engraving	Þ
												E.
											Cont.	
										ŕ	mill.	r
	ð		I Turn-		Cont			Uori-	+ Simu		NC Ex-	
3	Edit	Drill.	ina ina	5	turn.	Mill.		ous	latio	n	ecute	e
				_				-		-		-

Figure 8-44 Process plan following rectangular pocket
8.9 Milling the rectangular pocket



Start the simulation.



Figure 8-45 Simulation - 3D view

## Insertion

Vertical insertion	Helical insertion	Oscillating insertion

Example 3: Guide shaft

8.9 Milling the rectangular pocket

# 9.1 Overview

## Learning objectives

In this section you will learn the following new functions. You will learn how to...

- perform internal machining on workpieces;
- work with the work step editor;
- create an undercut and
- an asymmetrical groove.

## Task



Figure 9-1 Workshop drawing - Example 4:



Figure 9-2 Blank contour

All non-dimensioned radii R10.

Note

On account of improved clamping, side 1 is manufactured first.

## 9.2 Creating the first workpiece side

#### Creating a process plan

Since the workpiece is to be machined from two sides (and produced without counterspindle), two process plans must be created.

Create the process plan for the left-hand side first ('HOLLOW\_SHAFT\_SIDE1')

#### Operating sequences

Create the program 'HOLLOW\_SHAFT\_SIDE1' without help.

	New sequential program	_
Туре	ShopTurn	~
Name HOLL	.ow_shaft_side1	

Figure 9-3 Creating a ShopTurn program

Enter the following data in the program header (see illustration).



Figure 9-4 Workpiece dimensions in the program header

9.2 Creating the first workpiece side

## 9.2.1 Face turning

#### **Operating sequences**

Proceed as follows to face turn the blank to Z0:

Furning Select the **Turning** softkey.



Select the Stock removal softkey.

Select the ROUGHING\_T80 A tool.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	Х	
Machining	Roughing	X	Since a large amount of material (5mm) remains on the front face, set the machining to roughing.
Position	(see illustration below)	X	
Machining direction	Chart	Х	
X0	105		
Z0	5		
X1	-1.6 abs	Х	
Z1	0 abs	Х	
D	2.5		
UX	0.0		
UZ	0.2		



Figure 9-5 Face turning the workpiece

9.2 Creating the first workpiece side

Accept	Accept the entered values. After acceptance, your work step program looks like this:
	NC/UKS/EXAMPLE4/HOLLOU_SHAFT_SIDE1 2 Stock
	P Program header
	🔚 Stock removal 🛛 🔻 T=ROUGHING_T80 A F0.2/rev V240m 🚍
	END End of program Groove
	Indexet
	Under cut
	Thread
	Cutoff
	cum
	📝 Edit 📑 Drill. 🗾 Turn- 🚽 Cont. T- Mill. 📑 Vari-

Figure 9-6 Process plan following face turning

# 9.2.2 Drilling

## **Operating sequences**

	Proceed as follows to drill the workpiece at the center.
<b>_</b> Drill.	Select the <b>Drilling</b> softkey.
Drilling centric	Select the <b>Drilling Centric</b> softkey.
Drilling centric	Select the <b>Drilling Centric</b> softkey.
Select tool	Open the tool list and select DRILL_32.
То	Accept the tool into your program.
program	Enter the following values for the drill hole in the interactive screenform:

9.2 Creating the first workpiece side

Field	Value	Selection via toggle key	Notes
F	0.1 mm/rev	Х	
S	2500 rpm	Х	
	Chip removal	Х	
Z0	0		
	Tip	Х	
Z1	-57 inc	Х	
D	57		





"Apply" the set values. After acceptance, the list of work steps looks like this:





## 9.2.3 Blank contour

#### **Operating sequences**

Enter the following blank contour without help. Since the workpiece is only machined on one side for each process plan, the blank contour only needs to be constructed to Z-65.





Select the New contour softkey. Enter the name 'HOLLOW\_SHAFT\_BLANK' for the contour.

New contour Please enter the new name HOLLOW\_SHAFT\_BLANK

Figure 9-9 Creating the contour

Create the blank contour in the contour calculator (see illustration below).



Figure 9-10 Creating a blank contour

## 9.2.4 Machined part contour of the first side, external

## **Operating sequences**

Proceed as follows to enter the machined part contour:



#### Note

The (red) contour of the machined part is intended not to correspond to the drawing. The machined part contour serves as the roughing machining limit, but more importantly it specifies the precise traversing path for finishing. Thus, the construction begins here at the drill hole diameter. This ensures that the end face is finished cleanly. The contour end is an extension of the chamfer extending outside the blank. The large diameter is produced only in the second clamping.



Select the Contour turning softkey.



Select the **New contour** softkey. Enter the name 'HOLLOW\_SHAFT\_SIDE1\_E' for the contour.



Figure 9-11 Creating the contour



Confirm your input.

Set the starting point to X32/Z0.



Figure 9-12 Specifying the contour starting point

Confirm your input.



Enter the following values for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	68 abs	Х	$\mathbf{Y}$
Transition to next element	Chamfer	Х	
F	1		



Figure 9-13 Specifying the vertical straight line contour



Accept the entered values.



Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	-5 abs	Х	)
Transition to next element	Chamfer	Х	
FS	0		



Figure 9-14 Specifying the horizontal straight line contour



Accept the entered values.

9.2 Creating the first workpiece side



Enter the following values for the next section in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Direction of rotation	Right	Х	\ \
R	20		
Х	68 abs	Х	
Z	-25 abs	Х	
Transition to next element	Chamfer	Х	
FS	0		



Figure 9-15 Specifying the arc contour

Select the desired construction.



Accept your selection.



Accept the contour section.

9.2 Creating the first workpiece side



Enter the following values for the horizontal straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Z	-55 abs	Х	>
Transition to next element	Chamfer	X	
FS	0		
			The undercut is inserted later as an individual element.



Figure 9-16 Specifying the horizontal straight line contour



Accept the entered values.

9.2 Creating the first workpiece side



Enter the following values for the vertical straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	98 abs	Х	
Transition to next element	Chamfer	Х	
FS	0		
			The inclined straight line remains as chamfer after the second side has been machined.



Figure 9-17 Specifying the vertical straight line contour



Accept the entered values.



Enter the following values for the inclined straight line in the interactive screenform:

Field	Value	Selection via toggle key	Notes
Х	106 abs	Х	<b>\</b>
α1	135	Х	
Transition to next element	Chamfer	Х	
FS	0		



Figure 9-18 Specifying the inclined straight line contour



Accept the entered values.

Accept the contour into your process plan.



Figure 9-19 Contour in the contour calculator

After acceptance, the process plan looks like this. Both contours are automatically linked together.

						00/40/4
						8:52 AM
NC/WKS/EXAMPLE4/H0LLO	u_shaft_s	IDE1			5	New 📐
P Program header		Work offset (	54		~	contour
Stock removal	$\nabla$	T=ROUGHING	_T80_A F0.2/re	v V=240m Fac	e X0=105	
Drilling centric		T=DRILL_32 F	0.1/rev S=250	0rev 21=-57in	c	Stock
U n Blank		HOLLOW_SH	ift_blank			removal
℃- Fin. part		HOLLOW_SH	FT_SIDE1_E		$\Box$	
END End of program						Stock
						Grooving
						Groove resid.
						Plunge
						turning
						Plunge
						turn.resid.
					~	
					>	
📝 Edit 📑 Drilling	Jurn- ing	Cont. turn.	Milling	Vari- ous	ter Simu- lation	Ex- ecute

Figure 9-20 Process plan following input of the contours

#### Stock removal, residual stock removal and finishing

In the following work step you will perform contour stock removal.

To this end, proceed as follows:



Select the Stock removal softkey.



Open the tool list and select the ROUGHING\_T80 A tool.

To program Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	260 m/min	Х	
Machining	longitudinal external roughing	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		

9.2 Creating the first workpiece side

Field	Value	Selection via toggle key	Notes
BL	Contour	x	In the blank descriptions you can choose between the following settings:
			<i>Cylinder:</i> Blank = cylinder
			<i>Contour:</i> Blank = constructed contour
			<i>Allowance:</i> Blank = constructed contour with defined allowance
Relief cuts	No	X	Insertion with the roughing tool is not recommended. Therefore, switch the Relief cuts field to <i>No</i> .
Set machining area limits	No	Х	



Figure 9-21 Roughing the contour



Accept the entered values.

Select the **Residual stock removal** softkey.



To program Accept the tool into your program. Before finishing, the residual material is cut in the concave fillet.

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1

Enter the following values for stock removal of residual material in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	X	
Machining	longitudinal external roughing	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
Relief cuts	Yes	X	To ensure that the concave fillet is taken into consideration, the Relief cuts field must be switched to <i>Yes</i> .
FR	0.2		
Set machining area limits	No	X	



Figure 9-22 Stock removal of residual material contour



Accept the entered values.

Select the Stock removal softkey.

Select tool Open the tool list and select the FINISHING\_T35 A tool.

9.2 Creating the first workpiece side



Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	280 m/min	Х	
Machining	longitudinal external finishing	X X X	
Allowance	No	Х	
Relief cuts	Yes	X	Switch Relief cuts to <i>Yes</i> here, too.
Set machining area limits	No	Х	



Figure 9-23 Finishing the contour

$\checkmark$
Accept

Accept the entered values. After acceptance, your work step program looks like this: The contours are automatically linked to the stock removal work steps.

							08	8/10/11 8:57 AM
NC,	/WKS/EXAMPLE4/HOLLO	)u_shaft_si	DE1				8 Ne	ωN
Ρ	Program header		Work offset G	54			< cont	our
1000	Stock removal	$\nabla$	T=ROUGHING	T80_A F0.2/re	v V=240m Fac	e X0=105		
Res	Drilling centric		T=DRILL_32 F	0.1/rev S=2500	Brev 21=-57in	c	Sto	ck 📐
J.	Blank		HOLLOW_SHP	ift_blank			remo	val
J.	Fin. part		HOLLOW_SHP	FT_SIDE1_E			Cuta	
de la composition de la compos	Stock removal	$\bigtriangledown$	T=ROUGHING	_T80_A F0.3/re	v V=260m		stor	2510
n.S.	Residual cutting	$\nabla$	T=FINISHING_	T35 A F0.2/rev	V=240m		310	
M.	Stock removal	$\nabla \Delta \Delta$	T=FINISHING_	T35 A F0.15/re	v V=280m			
END	End of program						Groo	ving
							Groo	id.
							Plun turni	ge ing
							Plun turn.re	ge esid.
_							- I	
	Edit 📑 Drilling	Jurn- ing	Cont. turn.	<b></b> Milling	Vari- ous	Simu- lation	e e	Ex- cute

Figure 9-24 Process plan following stock removal of the contour

## 9.2.5 Undercut

You can select from four different types of undercut:

Undercut form E	Undercut form F	Undercut DIN thread	Undercut thread
	ZI VX		21

## **Operating sequences**

Proceed as follows to create an undercut.



After residual stock removal, the list of work steps looks like this:



Figure 9-25 Process plan following stock removal



Enter the following values for the groove in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	Х	
Position	see illustration below	X	
	E 1.0 x 0.4	X	
X0	68		
Z0	-55		
X1	0 inc	Х	
VX	70 abs	Х	



Figure 9-26 Specifying the undercut



Accept the entered values. After acceptance, the list of work steps looks like this:



Figure 9-27 Process plan with undercut



Start the simulation.



Figure 9-28 Simulation - Cut active



Figure 9-29 Simulation - Side view with display of the traversing paths

## 9.2.6 Machined part contour of the first side, internal

#### **Operating sequences**

Proceed as follows to enter the machined part contour:





Select the Contour turning softkey.



Select the **New contour** softkey. Enter the name 'HOLLOW\_SHAFT\_SIDE1\_I' for the contour.



Figure 9-30 Creating the contour



Confirm your input.

Set the starting point to X50/Z0.



Figure 9-31 Specifying the contour starting point



Confirm your input.



Create the contour without help (see illustration below).

Figure 9-32 Machined part contour of the first side, internal

## Stock removal, residual stock removal and finishing

In the following work step you will perform contour stock removal. Enter the geometries into your process plan as follows.



Figure 9-33 Broken-line graphics



Select the Stock removal softkey.

Open the tool list and select the ROUGHING\_T80 I tool.

program

Accept the tool into your program.

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1

Enter the following values	for roughing ir	the interactive	screenform:
----------------------------	-----------------	-----------------	-------------

Field	Value	Selection via toggle key	Notes
F	0.25		
V	250 m/min	X	
Machining	longitudinal internal roughing	X X X	You must switch the machining to <i>Inside</i> .
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Cylinder	X	Since drilling has already taken place, you do not need to take a blank contour into consideration for internal machining. Switch to <i>Cylinder</i> .
XD	32 abs	Х	
ZD	0 inc	Х	
Relief cuts	No	Х	
Set machining area limits	No	X	



Figure 9-34 Roughing the contour



Accept the entered values.



Select the Stock removal softkey.

9.2 Creating the first workpiece side



Open the tool list and select the FINISHING\_T35 I tool.

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.12		
V	280 m/min	Х	
Machining	longitudinal internal finishing	X X X	
Allowance	No	Х	
Relief cuts	No	X	
Set machining area limits	No	X	



Figure 9-35 Finishing the contour



Accept the entered values.



Start the simulation to perform checks.



Figure 9-36 Simulation - Cut active

## Undercut

Proceed as follows to create an undercut:

Undercut

Select the **Undercut** softkey.



Select the **Undercut form E** softkey.

Create the undercut (see illustration below).



Figure 9-37 Creating an undercut

#### Note

Make sure that the undercut is in the correct position.



Start the simulation.



Figure 9-38 Undercut simulation (with display of the traversing paths)

The process plan for the first side of the workpiece looks like this.

				08/10/1 9:18 Al
NC/	WKS/EXAMPLE4/HOLL	ow_shaft_si	DE1 13	Select
Ρ	Program header		Work offset G54	tool
3	Stock removal	$\nabla$	T=ROUGHING_T80_A F0.2/rev V=240m Face X0=105	
Res	Drilling centric		T=DRILL_32 F0.1/rev S=2500rev 21=-57inc	Build
v.	Blank		Hollow_Shaft_Blank	group 🚩
J.	Fin. part		Hollow_Shaft_Side1_e	
×.	Stock removal	$\nabla$	T=ROUGHING_T80_A F0.3/rev V=260m	Search
A.S.	Residual cutting	$\nabla$	T=FINISHING_T35 A F0.2/rev V=240m	
×.	Stock removal	$\nabla \nabla \nabla$	T=FINISHING_T35 A F0.15/rev V=280m	
2.	Undercut E		T=FINISHING_T35 A F0.15/rev V=200m E1.0x0.4 X0=68	Mark
5	Contour		HOLLOW_SHAFT_SIDE_1_I	
×.	Stock removal	$\nabla$	T=ROUGHING_T80_I F0.25/rev V=250m	
No.	Stock removal	$\nabla \nabla \nabla$	T=FINISHING_T35_I F0.12/rev V=280m	Сору
14	Undercut E		T=FINISHING_T35_I F0.15/rev V=200m E1.0x0.4 X0 -	
END	End of program			
				Paste
				Cut
			<b>v</b>	
			>	
	Edit 📑 Drilling	Jan Turn-	Cont. Milling Vari- turn. Milling us lation	NC Ex-

Figure 9-39 Process plan with undercut

## 9.2.7 The work step editor

## Functions of the work step editor

The following information provides you with an overview of the functions of the work step editor.

Graphic view	Use this softkey to switch to the broken-line graphics.
Search	Use this softkey to search for texts in the program.
Mark	Use this softkey to select several work steps for further processing (e.g., "Copy" or "Cut").
Сору	Use this softkey to copy work steps to the clipboard.
Paste	Use this softkey to paste work steps from the clipboard to the process plan. The copied step is always inserted after the currently highlighted step.
Cut	Use this softkey to copy work steps to the clipboard; at the same time, it is deleted at its origin. This softkey can also be used for "pure" deletion.
	Use this softkey to switch to the extended menu.
Renumbering	Use this softkey to renumber the work steps.
Settings	Use this softkey to open the "Settings" dialog. Here you can specify, e.g. automatic numbering or whether you wish the end of the block to be represented as a symbol.
	Use this softkey to return to the previous menu.

You will need some of these functions in order to reuse the blank contour of the first side in the process plan for the second side of the workpiece. The blank contour will be copied to the clipboard and pasted into the process plan for the second side.



Figure 9-40 Blank contour

## 9.2.8 Copying the contour

#### **Operating sequence**

Proceed as follows to copy the blank contour to the clipboard: Navigate to the 'HOLLOW\_SHAFT\_BLANK' contour.

							08/10/1 9:20 Al
NC/	/WKS/EXAMPLE4/HOLLO	u_shaft_si	DE1			4	Select
Ρ	Program header		Work offset G	54		^	tool
3	Stock removal	$\nabla$	T=ROUGHING	T80_A F0.2/re	ev V=240m Fac	e X0=105	
Ces	Drilling centric		T=DRILL_32 F	0.1/rev S=250	0rev 21=-57in	c	Build
- ت	Blank		HOLLOW_SHA	FT_BLANK		$\overline{}$	group
· س	Fin. part		HOLLOW_SHA	FT_SIDE1_E			
1	Stock removal	$\nabla$	T=ROUGHING	T80_A F0.3/re	ev V=260m		Search
Je-	Residual cutting	$\nabla$	T=FINISHING	T35 A F0.2/rev	V=240m		
20	Stock removal	$\nabla \Delta \Delta$	T=FINISHING	T35 A F0.15/re	ev V=280m		
La	Undercut E		T=FINISHING	T35 A F0.15/re	ev V=200m E1.	0×0.4 X0=68	Mark
11.	Contour		HOLLOW SHA	FT SIDE 1 I			
Se-	Stock removal	$\nabla$	T=ROUGHING	T80   F0.25/r	ev V=250m	-	
Se-	Stock removal	$\nabla \Delta \Delta$	T=FINISHING	T35   F0.12/re	v V=280m		Сору
14	Undercut E		T=FINISHING	T35   F0.15/re	v V=200m E1.	0×0.4 X0=48	
END	End of program						
	Lina of program						Paste
						_	
						_	
						_	Cut
						_	
						~	
						>	
3	Edit 📑 Drilling	Jurn- ing	Cont.	- Milling	Vari- ous	Simu- lation	Ex-

Figure 9-41 Copying the contour to the clipboard.

Copy the blank contour to the clipboard. The contour remains on the clipboard until you copy another work step to the clipboard or shut down the control system.

Сору

9.3 Creating the second workpiece side

# 9.3 Creating the second workpiece side

### Creating a process plan

Proceed as follows to create a process plan for the second side of the workpiece.

#### **Operating sequences**

Create the program 'HOLLOW\_SHAFT\_SIDE2' without help.

	New sequential program	
Туре	ShopTurn	<b> </b> ~
Name <mark>HOLI</mark>	Low_shaft_side2	

Figure 9-42 Creating a ShopTurn program

Enter the following data in the program header (see illustration).



Figure 9-43 Workpiece dimensions in the program header

## 9.3.1 Face turning

#### **Operating sequences**



Proceed as follows to face turn the blank to X-1.6 and Z0: Select the **Turning** softkey.



Select the Stock removal softkey.

9.3 Creating the second workpiece side

Select the ROUGHING\_T80 A tool.

Enter the following values in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	Х	
Machining	Roughing	X	Since a large amount of material (5mm) remains on the front face, set the machining to roughing.
Position	(see illustration below)	x	
Machining direction	Chart	Х	
X0	105		
Z0	5		
X1	-1.6 abs	Х	
Z1	0 abs	Х	
D	2.5		
UX	0.0		
UZ	0.2		



Figure 9-44 Face turning the workpiece

9.3 Creating the second workpiece side

Accept	Accept the entered values. After acceptance, your work step program looks like this:
	P Program header
	Stock removal ▼ T=ROUGHING_T80 A F0.2/rev V240m → ENG End of program
	Undercut
	Thread
	Cutoff
	Edit Turn: Cont. C

Figure 9-45 Process plan following face turning

# 9.3.2 Drilling

## **Operating sequences**

	Proceed as follows to drill the workpiece at the center.
<b>Drill</b> .	Select the <b>Drilling</b> softkey.
Drilling centric	Select the Drilling Centric softkey.
Drilling centric	Select the Drilling Centric softkey.
Select tool	Open the tool list and select DRILL_32.
To program	Accept the tool into your program.

9.3 Creating the second workpiece side

Field	Value	Selection via toggle key	Notes
F	0.1 mm/rev	Х	
S	2500 rpm	Х	
	Chip removal	Х	
ZO	0		
	Тір		
Z1	-67 abs	Х	
D	67		
DT	0	Х	

Enter the following values for the drill hole in the interactive screenform:



Figure 9-46 Hole



"Apply" the set values. After acceptance, the list of work steps looks like this:

								08/10/1 9:29 AM
NC/	UKS/EXAMPLE4/I	HOLLOW_SH	IAFT_SIDE2				3	Drilling
Ρ	Program header		Work	offset G54			^	centric
3	Stock removal	V	T=R0	JGHING_T80_A	F0.2/rev V=24	40m Face 1	X0=105	
Res	Drilling centric		T=DRI	LL_32 F0.1/rev	S=2500rev Z	1=-67inc	$\Box$	Centering
END	End of program							
								D. III'
								Drilling
								ricaning
								Deen hole
								drilling
								Thread
								Positions
							_	Position
_							~	repetit.
							>	. spour
J	Edit - Dr	illing 🚅	Turn- ing	Cont. turn.	Milling	Vari- ous	ter Simu- lation	ecute

Figure 9-47 Process plan following input of the drilling position

9.3 Creating the second workpiece side

## 9.3.3 Specifying a blank contour

#### **Operating sequences**

Proceed as follows to paste the blank contour from the clipboard into your process plan:

Paste the blank contour from the clipboard. After pasting the contour, your process plan

First, navigate in the process plan to the most recently entered work step (see illustration).



Figure 9-48 Position for inserting the blank contour



Paste

Figure 9-49 Pasting the contour

should look like this.
## 9.3.4 Machined part contour of the second side, external

## **Operating sequences**

Proceed as follows to enter the machined part contour:



Note

The asymmetrical groove is produced later.



Select the Contour turning softkey.



Select the **New contour** softkey. Enter the name 'HOLLOW\_SHAFT\_SIDE2\_E' for the contour.



"Apply" your input.

Set the starting point to X57/Z0.



Figure 9-50 Specifying the contour starting point



"Apply" your input.



Create the contour up to the end point to Z-65 and X100 without help (see illustration below).

Figure 9-51 Contour in the contour calculator



Accept the contour into your process plan.

## Stock removal and finishing

In the following work step you will perform contour stock removal.

To this end, proceed as follows:



Select the Stock removal softkey.



Open the tool list and select the ROUGHING\_T80 A tool.



Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	260 m/min	Х	
Machining	longitudinal external roughing	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Contour	Х	
Relief cuts	No	Х	
Set machining area limits	No	X	



Figure 9-52 Roughing the contour



"Apply" the set values. After acceptance, your work step program looks like this:



Figure 9-53 Process plan following roughing



Select the Stock removal softkey.

Open the tool list and select the FINISHING\_T35 A tool.



Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	Х	
Machining	longitudinal external finishing	X X X	
Allowance	No	Х	
Relief cuts	No	X	
Set machining area limits	No	Х	



Figure 9-54 Finishing the contour

"Apply" the set values. After acceptance, your work step program looks like this:



Figure 9-55 Process plan following stock removal of the contour



Accept



Figure 9-56 Simulation - 3D view

## 9.3.5 Creating an asymmetrical groove

#### **Operating sequences**

Proceed as follows to create an asymmetrical groove.





Select the **Turning** softkey.

Select the **Groove** softkey.

Select the Groove 2 softkey.

Open the tool list and select the PLUNGE\_CUTTER\_3 A tool.



Select

tool

Accept the tool into your program.

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1

Enter the following v	values for the	groove in the	interactive	screenform:
-----------------------	----------------	---------------	-------------	-------------

Field	Value	Selection via toggle key	Notes
F	0.08		
V	180 m/min	Х	
Machining	Roughing + finishing	X	
Position	see illustration below	Х	
X0	70		
Z0	-55		
B1	10	X (field)	
T1	5.5 inc	Х	
α1	0		
α1	15		
R1	0	X (field)	
R2	2	X (field)	
R3	0	X (field)	
R4	0	X (field)	
D	3		
U	0.2	X (field)	
N	1		



Figure 9-57 Specifying a groove



Accept the entered values. After acceptance, the list of work steps looks like this:

_							
_							08/10/11 9:42 AM
NC/	uks/example4/hollo	w_shaft_si	DE2			8	Stock 📐
Ρ	Program header		Work offset (	i54		^	removal
100	Stock removal	$\nabla$	T=ROUGHING	_T80_A F0.2/re	v V=240m Fac	e X0=105	
Ges	Drilling centric		T=DRILL_32 F	0.1/rev S=250	0rev 21=-67in	c	Groove
Vι	Blank		HOLLOW_SHA	AFT_BLANK			
V	Fin. part		HOLLOW_SH	AFT_SIDE_2_E			
M	Stock removal	$\nabla$	T=ROUGHING	_T80_A F0.3/re	v V=260m		Undercut
لم الأ	Stock removal	$\nabla \nabla \nabla$	T=FINISHING_	T35 A F0.15/re	v V=200m		
L.	Groove	<b>∀+</b> ∀∀∀	T=PLUNGE_C	UTTER_3 A F0.0	08/rev V=180n	n X0=70 🕣	
END	End of program						Thread
						-	Cutoff
_						~	
						>	
J	Edit Trilling	Turn- ing	bont.	Milling	Vari- ous	Simu- lation	Ex- ecute

Figure 9-58 Process plan following groove



Start the simulation.



Figure 9-59 Simulation - 3D view (cut active)

## 9.3.6 Machined part contour of the second side, internal

### **Operating sequences**

Proceed as follows to enter the machined part contour:





Select the Contour turning softkey.



Select the **New contour** softkey. Enter the name 'HOLLOW\_SHAFT\_SIDE2\_I' for the contour.

"Apply" your input.

Set the starting point to X57/Z0.



Figure 9-60 Specifying the contour starting point



"Apply" your input.



Create the contour without help (see illustration below).

Figure 9-61 Machined part contour of the second side, internal

#### Note

When creating the contour, ensure that the arc elements merge tangentially.

Tangential merging applies only to main elements, i.e. the rounding is attached to the main element.

(See illustration below)



## Example 4: Hollow shaft

9.3 Creating the second workpiece side

			08/10/1 9:46 P
NC/UKS/EXAMPLE4/H	ollow_shaft_s	SIDE2	9 New
P Program header		Work offset G54	<ul> <li>contour</li> </ul>
Stock removal	$\nabla$	T=ROUGHING_T80_A F0.2/rev V=240m Face X0=	105
Drilling centric		T=DRILL_32 F0.1/rev S=2500rev Z1=-67inc	Stock
U 1 Blank		Hollow_Shaft_Blank	removal
े Fin. part		Hollow_shaft_side_2_e	
Stock removal	$\nabla$	T=ROUGHING_T80_A F0.3/rev V=260m	Cut resid
Stock removal	$\nabla \Delta \Delta$	T=FINISHING_T35 A F0.15/rev V=200m	Stuck
Groove	<b>44</b> 00	T=PLUNGE_CUTTER_3 A F0.08/rev V=180m X0=7	0
🕖 n Contour		HOLLOW_SHAFT_SIDE_2_I	Grooving Grooving
END End of program			
			Groove
			resid.
			Plunge
			turning
			_
			Plunge
			turn.resid.

am looks like this.

Figure 9-62 Process plan following input of the contour

#### Stock removal, residual stock removal and finishing

In the following work step you will perform contour stock removal.



Select the Stock removal softkey.



Open the tool list and select the ROUGHING\_T80 I tool.

To program Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	280 m/min	Х	
Machining	longitudinal internal roughing	X X X	You must switch the machining to <i>Inside</i> .
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Cylinder	X	Since drilling has already taken place, you do not need to take a blank contour into consideration for internal machining. Switch to <i>Cylinder</i> .

Example 4: Hollow shaft

9.3 Creating the second workpiece side

Field	Value	Selection via toggle key	Notes
XD	32 abs	Х	
ZD	0 abs	Х	
Relief cuts	No	Х	
Set machining area limits	No	Х	



Figure 9-63 Roughing the contour



"Apply" the set values.



Select tool Select the Residual stock removal softkey.

Open the tool list and select the FINISHING\_T35 I tool.



Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	Х	
Machining	longitudinal internal finishing	X X X	
Allowance	No	Х	
Relief cuts	Yes	Х	
FR	0.2		
Set machining area limits	No	Х	



Figure 9-64 Contour residual stock removal



"Apply" the set values.



Select the Stock removal softkey.

Select tool

То

program

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Open the tool list and select the FINISHING\_T35 I tool.

Field	Value	Selection via toggle key	Notes
F	0.12		
V	280 m/min	Х	
Machining	longitudinal internal finishing	X X X	
Allowance	No	Х	
Relief cuts	Yes	Х	
Set machining area limits	No	Х	



Figure 9-65 Finishing the contour



"Apply" the set values.



Start the simulation to perform checks.



Figure 9-66 Simulation - 3D view (cut active)

Example 4: Hollow shaft

9.3 Creating the second workpiece side

## Example 5: Plunge-turning

# 10

## 10.1 Overview

## Learning objectives

In this section you will learn how to use the plunge-turning functions.

Task



Figure 10-1 Workshop drawing - Example 5:

10.2 Plunge-turning

## Preparation

Perform the following steps without help:

- 1. Create a new workpiece with the name 'EXAMPLE5'.
- 2. Create a new step sequence program with the name 'PLUNGE\_TURNING'.
- 3. Fill in the program header (see illustration below).



Figure 10-2 Creating a program header

## 10.2 Plunge-turning

The achievable level of productivity during turning is limited, among other things, by the possible number of tools in the revolver and the frequent tool changes required for effective turning machining. With standard turning tools alone, not all possible contours can be produced, and residual material machining, therefore, is often performed by means of plunge-cutting. For the complete machining of a contour, therefore, it is always necessary to change between standard turning tools and plunge-cutting tools.

The aim of the plunge-turning cycle is to reduce the number of tool changes and to avoid empty cuts, such as those that occur during the backward movement of the turning tool, for example.

As a general rule, hardly any empty cuts exist during the plunge-turning cycle, as stock removal is performed during both forward and backward movement. This must be taken into account during program creation. ShopTurn offers optimum support for this. As you already know, you only need to describe the contour of the turning part and during the stock removal cycle you can select whether you wish to perform stock removal using a conventional procedure or by means of plunge-cutting or plunge-turning. ShopTurn automatically calculates the cuts and traversing movements of the tool based on the cycle. This means that empty cuts are eliminated to a great extent.

During the simulation, you can clearly analyze the calculated traversing movements of the tool. Even a combination of conventional turning machining and plunge-turning is possible, i.e. a standard tool is used for roughing, while plunge-turning is used for machining residual material, meaning that the contour can be thoroughly machined without the risk of damage.

Example 5: Plunge-turning 10.3 Creating the contour

## 10.3 Creating the contour

## **Operating sequences**

Create the contour without help.



Select the **Contour turning** softkey.



Select the **New contour** softkey. Enter the name 'CONTOUR\_E' for the contour. Confirm your input.

Set the starting point to X48/Z0.



Figure 10-3 Specifying the contour starting point

₩5 9 9 08/10/11 10:04 AM Delete element X α1 α2 Tra 50.000 abs 90.000 270.000 120 **←•**→ o next elemen m 0.000 ES ‡ 80 X 10171 I A Ť × Cancel END -46 Accept 40 > Lation Ex-📝 Edit 📑 Drilling 🛃 Turn-Lont. Milling Vari-

Create a contour (see illustration below).

Figure 10-4 Contour in the contour calculator

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1

## 10.4 Stock removal with the plunge-turning cycle

### **Operating sequences**

In the following work step you will perform contour stock removal. To this end, proceed as follows:



Select the **Plunge-turning** softkey.

Select the Contour turning softkey.



Part

Open the tool list and select the PLUNGE\_CUTTER\_3 A tool.



Accept the tool into your program.

Enter the following values for roughing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
FX	0.2		
FZ	0.25		
V	150 m/min	X	
Machining	longitudinal external roughing	X X X	
D	2.5		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Cylinder	X	
XD	50 abs	Х	
ZD	0 abs	Х	
Set machining area limits	No	Х	
Ν	1		



Roughing the contour



Accept the entered values.

Select the **Plunge-turning** softkey.

Open the tool list and select the PLUNGE\_CUTTER\_3 A tool.

To program

Accept the tool into your program.

Enter the following values for finishing in the interactive screenform:

Field	Value	Selection via toggle key	Notes
FX	0.15		
FZ	0.15		
V	200 m/min	Х	
Machining	longitudinal external finishing	X X X	
Allowance	No	Х	
Set machining area limits	No	Х	
Ν	1		



Finishing the contour



Accept the entered values. After acceptance, your work step program looks like this:



Figure 10-5 Work step program



Select the **Simulation** softkey.



Figure 10-6 Simulation - Side view (with display of the traversing paths)



Figure 10-7 Simulation - 3D view (with display of the traversing paths)

Example 5: Plunge-turning

10.4 Stock removal with the plunge-turning cycle

## And now you can start manufacturing!

After you have acquired a well-founded knowledge of the creation of process plans in ShopTurn by working with the examples, we will now machine workpieces.

To machine a workpiece, proceed as follows:

#### Approaching reference point

After turning on the control system and before traversing the axes according to the process plans or traversing manually, you will have to approach the reference point of the machine. In this way, ShopTurn will find the start of counting in the position measuring system of the machine.

Since approaching of the reference point is different depending on machine type and manufacturer, only a few hints can be given here for orientation:

- If necessary traverse the tool to a free point in the work space from which traversing is possible in all directions without collision. Make sure that the tool is then not beyond the reference point of the corresponding axis (since reference point approach is only performed in one direction for each axis; otherwise, this point cannot be reached).
- 2. Perform the reference point approach exactly according to the specifications of the machine manufacturer.

#### Clamping the workpiece

To guarantee machining in accordance with the specified dimensions and, naturally, also for your own safety, it is imperative to clamp the workpiece tightly. A three-block chuck is usually used for this.

#### Setting the workpiece zero

Since ShopTurn cannot guess where in the work space the workpiece is located, you must determine the workpiece zero in Z.

The workpiece zero is usually determined through sampling with a calculated tool in the Z axis.

#### Executing the process plan

Now the machine is prepared, the workpiece is set up, and the tools are gauged. At last we can start!

11.1 And now you can start manufacturing!

First select the program you want to execute in the Program Manager, e.g. HOLLOW\_SHAFT\_SIDE2.

					08/11/1 9:45 AI
Name	Туре	Length	Date	Time	Execute
C Part programs     Subprograms	DIR		08/04/11	8:52:06 AM	
Gubprograms     Gubprograms     Gubprograms	DIR		08/05/11	5:33:56 PM	
EXAMPLE1	WPD		08/04/11	8:52:07 AM	New
EXHMPLE2     EXAMPLE3	UPD UPD		08/04/11	2:52:38 PM	
e 🖻 Example4	WPD		08/10/11	10:11:09 AM	Open
HOLLOW_SHAFT_SIDE1	MPF	3645	08/10/11	9:17:20 AM	
e	WPD	5041	08/04/11	8:52:08 AM	Mark
	WPD		08/04/11	8:52:09 AM	THUR
	WPD		00/10/11	10:10:52 HFT	
					Сору
					Paste
					_
					Cut
NC/Workpieces/EXHMPLE4.WPD	_	_	_	Free: 2.4 MB	
			1 un		
drive VUSB		T ncu	Y US	B	

Figure 11-1 Selecting the program

Open the program.

											08/11/1 9:45 Al
NC/	uks/exampl	E4/HOLLOW	_SHAFT_SI	DE2						S	elect
Р	Program hea	der		Work offs	et G54						100
3	Stock remova	al	$\nabla$	T=ROUGH	ING_T80_	A F0.2/re	ev V=240m	Face X	0=105		
Nes.	Drilling centr	ic		T=DRILL_	32 F0.1/r	ev S=250	0rev 21=-	67inc		B	uild
J.	Blank			HOLLOW_	_SHAFT_B	lank				g	roup
J.	Fin. part			HOLLOW_	SHAFT_S	IDE_2_E					
M.	Stock remove	al	$\nabla$	T=ROUGH	ING_T80_	A F0.3/re	ev V=260m	1		Se	arch
Mi.	Stock remove	al	$\nabla \Delta \Delta$	T=FINISHI	NG_T35 A	F0.15/re	ev V=200m				
	Groove	,	7 <b>+</b> 000	T=PLUNG	E_CUTTEF	_3 A F0.	38/rev V=1	180m X0	=70		
J.	Contour			HOLLOW_	_SHAFT_S	IDE_2_I				۲	lark
Mi-	Stock remove	al	$\nabla$	T=ROUGH	ING_T80_	I F0.25/r	ev V=280n	n			
n.	Residual cut	ting	$\nabla$	T=FINISHI	NG_T35_I	F0.2/rev	V=240m				
Mi.	Stock remove	al	$\Delta \Delta \Delta$	T=FINISHI	NG_T35_I	F0.12/re	v V=280m			C	opy
END	End of progra	am									
									_	P	aste
									_		Cut
									~		
	Edit 🚦	Drilling	Turn- ing	Left Co	nt. rn.	Milling	Va 01	ri- JS	Simu- lation		Ex- ecute

Figure 11-2 Opening the process plan

11.1 And now you can start manufacturing!

NC	Ex-
	ecute

Select the **NC selection** softkey.



Figure 11-3 Executing



Due to the fact that the process plan has not yet been executed with control, turn the feedrate potentiometer to zero position to ensure that you keep everything under control from the beginning.



If you also want to see a simulation during machining, select the **Drawing** softkey before starting. Only then are all traversing paths and their effects are displayed.



Start machining and check the speed of the tool motions using the feedrate potentiometer.

And now you can start manufacturing!

11.1 And now you can start manufacturing!

## How accomplished are you in the use of ShopTurn? 12

## 12.1 Exercise 1

Will you manage this task using ShopTurn within 10 minutes?



Figure 12-1 Workshop drawing DIYS1

## Notes

In the process plan, the workpiece is milled to size in two work steps (see model below). For this reason, you can assign the the starting point of the CONTOUR\_1 contour to the start of the first chamfer.

## Model

HC/U P P	iks/shopturn/diys1			
P F	In succession in the second second			Select
aller S	rogram neader		Work offset G54	tool
1 - 0	Stock removal	$\nabla \Delta \Delta$	T=ROUGHING_T80_A F0.15/rev V=250m Face X0=60	
VIU	Contour		CONTOUR_1	Build
¥- 5	Stock removal	$\nabla$	T=ROUGHING_T80_A F0.3/rev V=260m	group
2.5	Stock removal	$\nabla \nabla \nabla$	T=FINISHING_T35 A F0.15/rev V=280m	_
1.6 6	Groove	<b>7+</b> 777	T=PLUNGE_CUTTER_3 A F0.15/rev V=280m N3 X0=50	Search
DT T	Thread long.	<b>7+</b> 777	T=GEWINDESTAHL_P1.5 P1.5mm/rev S=2000rev	
END E	End of program			
				Mark
				Сору
				Paste
				Cut
	Edit <b>F</b> Drilling	Jurn-	Cont. Milling Vari-	NC Ex-

Figure 12-2 Process plan

12.1 Exercise 1



Figure 12-3 Contour in the contour calculator



Figure 12-4 Workpiece simulation

12.2 Exercise 2

## 12.2 Exercise 2



Will you manage this task using ShopTurn within 10 minutes?

Figure 12-5 Workshop drawing DIYS2

## Notes

You can use the automatic cutting of residual material to great effect here.

## Model

						08/10/11 10:24 AM
NC/UKS/SHOPTURH/D	IYS2				1	Select
P Program header		Work offset	654			tool
Stock removal	$\nabla \Delta \Delta$	T=ROUGHING	_T80_A F0.15/1	rev V=250m Fa	ace X0=80	
ງ Contour		CONTOUR_2				Build
Stock removal	$\bigtriangledown$	T=ROUGHING	_T80_A F0.3/re	v V=260m		group
Residual cutting	$\bigtriangledown$	T=FINISHING_	T35 A F0.15/re	v V=240m		
Stock removal	$\nabla \Delta \Delta$	T=FINISHING_	T35 A F0.15/re	v V=280m		Search
END End of program						
						Mark
						Сору
						Paste
						Cut
📝 Edit [ 📲 Dri	lling 🛃 Turn	- 🤳 Cont. turn.	Milling	Vari- ous	Simu- lation	NC Ex- ecute

Figure 12-6 Process plan

12.2 Exercise 2



Figure 12-7 Contour in the contour calculator



Figure 12-8 Workpiece simulation

12.3 Exercise 3

## 12.3 Exercise 3



Will you manage this task using ShopTurn within 10 minutes?

Figure 12-9 Workshop drawing DIYS3

Notes

Construct radius 5 in two steps.

## Model

_												08/10/11 10:26 AM
NC/	UKS/SHOPTURN/DIYS3									6	S	elect
Ρ	Program header		Work	offset G	<b>3</b> 54					1	1	tool
3	Stock removal	$\nabla \Delta \Delta$	T=R0	UGHING	_T80_A F	FØ.15/r	ev V=2	250m Fa	ace XØ=	60		
5	Contour		CONT	OUR_3							B	Build
1	Stock removal	$\nabla$	T=R0	UGHING	_T80_A F	FØ.3/re	v V=28	60m			g	roup
3.8	Residual cutting	$\nabla$	T=BU	TTON_TO	00L_8 F	0.2/rev	V=24	Øm			_	
Mr	Stock removal	$\Delta \Delta \Delta$	T=FIN	ISHING_	T35 A F8	0.15/re	v V=28	80m		$\ominus$	Se	earch
END	End of program									1		
											1	1ark
											0	Сору
											Р	aste
												Cut
							<i>6</i>			>		
J	Edit <b>E</b> Drilling	Jan Tur	n- 📕	Cont. turn.	<b>[</b> r	1illing		Vari- ous	1	Simu- lation		Ex- ecute

Figure 12-10 Process plan

12.3 Exercise 3



Figure 12-11 Contour in the contour calculator



Figure 12-12 Workpiece simulation

12.4 Exercise 4

## 12.4 Exercise 4



Will you manage this task using ShopTurn within 15 minutes?

Figure 12-13 Workshop drawing DIYS4

#### Notes

In the process plan, the end face is roughed and finished first (see model below). Then the entire external area, including the undercut, is produced. Following this, the internal part of the contour is machined. The starting point of the internal contour is set to X70/Z0. You can copy the external and internal machining sequences using the work step editor by means of cut and paste.

12.4 Exercise 4

## Model

		1	18:37 A
NC/WKS/SHUPTURN/DITS	•	Hask affect CE4	tool
Program neader	000	T-DOUCHING TOR O E0 15/row II-250m Easo V0-120	
Stock removal	V V V	CONTOUR 40	Build
Stock remound	Ω.	T-DOILCHING T88 0 F8 3/reu II-258m	group
Stock removal	777	T-EINICHING T25 0 E9 15/rev U-200m	
Stock removal		T-EINICHING_133 H 10.13/160 V-20011	Court
Drilling centric		T-DDI 1 22 E9 1/reg S-2000reg 71-50	Search
Gen Drilling Centric		CONTOUR 4	
Contour		T-DOILCHING TOB   E0.2/row II-260m	Mork
Stock removal	2000	T-EINICHING T25   E0 15/reg ll-200m	TIdIK
Undercut E	***	T-FINISHING_135_178.15/189 V-20811	
		1-FINISHING_135_1F0.15/160 V-20011 E1.020.4 A0-50	Conu
Enu or program			oopy
			Paste
			Cut
		V	••
	J	Cont Vari- + Simu-	NC Ex-

Figure 12-14 Process plan



Figure 12-15 External contour in the contour calculator

How accomplished are you in the use of ShopTurn?

12.4 Exercise 4



Figure 12-16 Internal contour in the contour calculator



Figure 12-17 Workpiece simulation

## 12.4 Exercise 4



Figure 12-18 Workpiece simulation - Cut active
# Index

# Α

Absolute input, 32 Alarms, 29 Axes, 31

## В

Basics of operation, 17 Blank description, 127 Blank shape Cylinder, 55 Tube, 55 Blank shapes, 161 Broken-line graphics, 174

# С

C axis, 138 Calling dialogs, 56 Cartesian input, 34 Circular motions, 36 Clipboard, 175 Complete machining sequence, 138 Concatenation, 25 Constant speed, 39 Contour calculator, 12 Accept dialog, 122 Selection dialog, 121 Creating process plan, 90 Cut, 15 Cutting rate, 11, 38

### D

Directory, 54 Drilling positions, 141

# F

Face turning, 90 Feed, 39 Feed, 39 Feedrates, 39

Turning made easy with ShopTurn Training Documents, 09/2011, 6FC5095-0AB80-1BP1 Flank angle, 134

# G

Graphical process plan, 11 Grooves, 84

#### Η

Handbook, 38, 39

## I

Incremental input, 33 Internal machining, 170

## L

Load magazine, 46

#### Μ

Machine zero, 32 Magazine, 23 Magazine list, 44 Main menu, 19 Main screen, 53 Measure workpiece, 49 Messages, 29

#### Ρ

Paste, 15 Plunge-turning, 196 Points in the working space, 31 Polar input, 35 Production, 203 Program header, 55 Program management, 54 Program Manager, 27, 54

### R

Reference point, 32 Relief cuts, 107 Residual material, 14, 107 Residual stock removal, 107 Retraction all, 56 extended, 56 normal, 56 Retraction plane, 55

## S

Safety clearance, 55 Sequence editor Copy, 174 Cut, 174 Graphic view, 174 Highlight, 174 Menu backward, 174 Menu forward, 174 Paste, 174 Renumber, 174 Searching, 174 Settings, 174 Setting up the machine, 21 Simulation, 25 2-window view, 87 3D view, 62 Cut active, 167 Details, 81 Displaying tool paths, 108 Magnifying glass, 135 Side view, 79 Simultaneous recording, 205 Softkeys, 18 Speed, 38 Speed limitation, 38 Start key, 205

### Т

Thread, 82, 110 Diminishing, 136 Thread undercut, 79 Toggle field, 52 Tool list, 22, 42 Tool wear list, 43 Toolholder reference point, 32 Tools for the examples, 44

#### U

Undercut

DIN thread, 164 Form E, 164 Form F, 164 Thread, 164

#### W

Work offsets, 23 Workpiece zero, 32