

SCE Training Curriculums

Siemens Automation Cooperates with Education | 02/2016

CNC Technology Module 700-020 ShopTurn Basics



For unrestricted use in educational / R&D institutions. © Siemens AG 2016. All rights reserved.

Suitable SCE trainer packages to accompany these training curriculums

- SinuTrain for SINUMERIK Operate V4.7 Basic free download with no time limit <u>www.siemens.com/sinutrain-downloads</u>
- SinuTrain Classroom License for SINUMERIK Operate V4.5 6 single licenses + 40 student licenses;

Order no.: 6FC5870-1TC41-0YA0

- SinuTrain Student Licenses for SINUMERIK Operate V4.5 300 hours 20 student licenses; Order no.: 6FC5870-1SC41-0YA0
- SinuTrain Classroom License for SINUMERIK Operate V4.4 16 single licenses + 32 student licenses; Order no.: 6FC5870-1TC40-1YA0
- SinuTrain Student Licenses for SINUMERIK Operate V4.4 300 hours 32 student licenses; Order no.: 6FC5870-1SC40-1YA0

Note that these trainer packages will be replaced with successor packages when required. You can find an overview of the currently available SCE packages at: <u>siemens.com/sce/tp</u>

In-service training courses

Get in touch with your regional SCE contact for information on regional Siemens SCE in-service training courses.

siemens.com/sce/contact

Additional information on SinuTrain

In particular: Downloads, Getting Started, Videos, Tutorials, Manuals and Programming Guides. <u>siemens.com/sce/sinutrain</u>

Further information on SCE

siemens.com/sce

Notes on use

The SCE training curriculum for CNC Technology has been created for the "Siemens Automation Cooperates with Education (SCE)" program especially for educational purposes for public educational and R&D institutions. Siemens AG assumes no responsibility for the content.

This curriculum may be used only for initial education with respect to Siemens products/systems. That is, it may be copied in part or in whole and handed out to trainees for use within the framework of their education. Transmission and reproduction of this curriculum as well as communication of its content is permitted within public educational institutions for educational purposes.

Exceptions require written consent from the Siemens AG. Send all related requests to <u>scesupportfinder.i-</u> <u>ia@siemens.com</u>.

Offenders will be held liable. All rights including translation are reserved, particularly if a patent is granted or a utility model or design is registered.

Use for industry customers is expressly prohibited. Commercial use of the curriculum is not permitted.

We would like to thank Michael Dziallas Engineering, MOSER CNC Training and all those involved for their support in creating this curriculum.

Contents

0	Obje	ective		. 6
1	Intro	ducti	on	. 6
2	Adva	antag	es of working with ShopTurn	. 8
	2.1	You	save training time:	. 8
	2.2	You	save programming time:	10
	2.3	You	save machining time:	13
3	To e	ensure	e that everything function smoothly	15
	3.1	The	operation of ShopTurn	15
	3.2	The	contents of the main menu	17
	3.2.1	1	Machine	17
	3.2.2	2	Parameters	20
	3.2.3	3	Program	22
	3.2.4	1	Program Manager	26
	3.2.5	5	Diagnostics	27
4	Basi	cs fo	r beginners	28
	4.1	Basi	cs of geometry	28
	4.1.1	1	Tool axes and machining planes	28
	4.1.2	2	Points in the work area	28
	4.1.3	3	Absolute and incremental dimensioning	29
	4.1.4	1	Cartesian and polar dimensions	31
	4.1.5	5	Circular motions	34
	4.2	Basi	cs of technology	35
	4.2.1	1	Cutting rate and speeds	35
	4.2.2	2	Feed	37
5	Effe	ctive	setup	38
	5.1	Tool	management	38
	5.1.1	1	Tool list	38
	5.1.2	2	Tool wear list	40
	5.1.3	3	Magazine list	41
	5.2	Tool	s used	42
	5.3	Tool	s in the magazine	43
	5.4	Mea	suring tools	44
	5.5	Setti	ng the workpiece zero	47
6	Exar	mple	1: Taper shaft	48
	6.1	Ove	rview	48
	6.2	Prog	ram management and creating programs	50
	6.3	Ope	ning a tool	54
	6.4	Ente	ring the traversing path	56

6.5	Creating contours with the contour calculator and machining	60
6.6	Thread undercut	75
6.7	Thread	
6.8	Grooves	80
7 Exa	mple 2: Drive shaft	84
7.1	Overview	
7.2	Face turning	
7.3	Creating the contour, stock removal and residual stock removal	
7.4	Thread	
8 Exa	mple 3: Guide shaft	108
8.1	Overview	
8.2	Face turning	
8.3	Creating a random blank shape contour	
8.4	Creating the finished part contour and stock removal	113
8.5	Residual stock removal	125
8.6	Groove	
8.7	Thread	
8.8	Drilling	
8.9	Milling the rectangular pocket	
9 Exa	mple 4: Hollow shaft	
9.1	Overview	
9.2	Creating the first workpiece side	
9.2.	1 Face turning	
9.2.	2 Drilling	
9.2.	3 Blank contour	
9.2.	4 Machined part contour of the first side, external	150
9.2.	5 Undercut	163
9.2.	6 Finished part contour of the first side, internal	
9.2.	7 The work step editor	173
9.2.	8 Copying the contour	174
9.3	Creating the second workpiece side	175
9.3.	1 Face turning	175
9.3.	2 Drilling	177
9.3.	3 Specifying a blank contour	179
9.3.	4 Finished part contour of the second side, external	
9.3.	5 Creating the asymmetrical groove	
9.3.	5 Finished part contour of the second side, internal	
10 Exa	mple 5: Plunge-turning	
10.1	Overview	
10.2	Plunge-turning	
10.3	Creating the contour	
10.4	Stock removal with the plunge-turning cycle	

11	Machining the workpieces	203
----	--------------------------	-----

1. Objective

In this module, you will learn how you can easily go from a drawing to a finished workpiece with the help of the OPERATE machining plan sequential programming interface, ShopTurn.

2. 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 solution tailored to the workshop that allows quick programming of machining steps in line with real-world requirements, ranging from the machining of single parts up to small batches. In conjunction with SINUMERIK Operate, the new operator interface for the controller, intuitive and effective working in the workshop is made possible even for series production.

Creation of a machining plan instead of programming is the solution.

The creation of a machining plan with intuitive and operator-friendly handling sequences allows the ShopTurn user to create the NC program directly based on 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 machine with ShopTurn thanks to the integrated and powerful tools for creating traversing paths. For this reason:

Easier and faster from the drawing to the workpiece - with ShopTurn!

Although ShopTurn is easy to learn, this ShopTurn training curriculum allows you to get started in this world even faster. Before starting the actual work with ShopTurn, however, important basics are discussed in the first sections:

- First, we will show you the advantages of working with ShopTurn.
- We show then you the basics of the operation.
- The basics of geometry and technology for machining will be explained for beginners.
- 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 possibilities for machining with ShopTurn, whereby the degree of difficulty is increased continuously. At the beginning, all key strokes are specified. Later, you will be prompted to proceed on your own.
- You will then learn how to machine in Automatic mode using ShopTurn.

Note that the technology data used here only serves as an example due to the wide variety of situations in the workshop.

Just as ShopTurn was created with the help of skilled workers, this training curriculum was also elaborated by practitioners. In this sense, we hope you enjoy working with ShopTurn and wish you success.

3. Advantages of working with ShopTurn

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

3.1 You save training time:

• ShopTurn does not use any foreign-language terms you would otherwise have to learn, and all necessary inputs are prompted in plain text.



• When working with ShopTurn you are assisted optimally by colored help screens.



- You may also integrate DIN/ISO commands into the graphical machining plan of ShopTurn. You may also program in DIN/ISO 66025 and use DIN cycles.
 - G
 696 S320 LIMS=3000 M4 M8¶

 G
 618 654 690¶

 G
 60 X32 Z0¶

 G
 61 X-1.6 F0.1¶

 G
 60 Z2¶

 G
 60 642 X22 Z2¶

 G
 X30 Z-2¶
- You may switch between the individual machining step and the workpiece graphic (broken-line graphic) at any time when creating a machining plan.

	SIEME	NS									E 03/06/17 10:02 AM		200
NC,	/WKS/E	EXAMPLE1/EXAMPLE1									14		
Ρ	N10	Program header		G54 Cylind	er						^	Vi	ew 🕨
G	; Exa	mple by Turning m	ade easy ui	th ShopTu:	rn¶								_
G	; Exa	mple 1 : Taper sh	aft¶										
1	N20	Stock removal	$\nabla \nabla \nabla$	T=ROUGHIN	IG_T80 A	F=0.25/	ev V=240	m Face X	(0=80 Z0:	=1		Gra	phic
т	N30	Turning T=ROUGHING	_T80 A V1=240	m								vi	eω
→	N40	RAPID X82 Z0.3											
→	N50	F0.3/rev X-1.6											
	N60	RAPID 21											
	N70	RAPID X82										Renun	nbering
	N80	RAPID ZØ											
→	N90	F0.25/rev X-1.6											
→	N100	RAPID 21										Open	further
→	N110	RAPID X120 Z200										pro	gram
V	N120	Contour		TAPER_SH	AFT_CON	TOUR					\Box		
M.	N130	Stock removal	∇	T=ROUGHI	IG_T80 A	F=0.3/re	ev V=240m	n Longitu	dinal		-		
M.	N140	Stock removal	$\nabla \nabla \nabla$	T=FINISHIN	G_T35 A I	F=0.15/r	ev V=200n	n Longitı	udinal				
	N150	Undercut thrd	~+~~~	T=FINISHIN	G_T35 A	F=0.15/r	ev S=200r	ev X0=30	0 Z0=-17	X1=1.15	inc		
W	N160	Thread long.	$\nabla + \nabla \nabla \nabla$	T=THREAD	ING_1.5 P	1.5mm/ı	ev S=800	rev Outsi	ide X0=30) Z0=0			
<u>) - E</u>	N170	Groove	$\nabla + \nabla \nabla \nabla$	T=PLUNGE	_CUTTER_	_3 A F=0.	1/rev V=1	50m N2	X0=60 Z0	=-65 T1	=3inc		
G	N180	F_HOME¶										Cat	lingo
G	N190	T0 D01									_	Set	ungs
END		End of program											
												CL	000
													030
													4≣
											~		.=
											>		
	• E	dit 🔤 Drilling	Turi	n-	Cont.	<u>[</u> 3=4	Milling	NC	Vari-	Ŀ,	Simu-	NC	Ex-



3.2 You save programming time:

• ShopTurn assists you even when entering the technological values: You only have to enter the handbook values "feedrate" (or feed) and "cutting rate" – the speed is calculated by ShopTurn automatically.

Drillin	g centric		Drillin	g centric
Input		Complete	Input	
Т	DRILL_5	D 1	Т	DRILL_5
F	0.100	mm/rev	F	100.000
S	2000	rpm	V	40.000
	Chip remo	oval		Chip remo

 ShopTurn enables you to describe a complete machining operation with one machining step, and the required positioning motions (in this case, from the tool change point to the workpiece and reverse) are created automatically.



• All machining steps are represented by ShopTurn in a compact and clear fashion in the graphic machining plan. This provides you a complete overview and thus better editing possibilities even for comprehensive machining sequences.

	SIEME	NS										03/06/1 10:27 AP		
N	:/WKS/I	example4/example	4									g		Select
Ρ	N10	Program header		G	54 Cylinde	er						^		
G	; Exa	mple by Turning	made easy	uith	ShopTu	cn¶								_
G	; Exa	mple 4 : Hollou	shaft sid	le 1¶										
a	N20	Stock removal	∇	T	=ROUGHIN	IG_T80 A	A F=0.2/r	ev V=240	m Face X	0=105 20	=5			Build
8	N30	Drilling		⊡• T	=DRILL_3	2 F=0.1/	rev V=24	0m 21=-6	67inc					group
N	⁻ N40	001: Positions		⊡• Z	0=0 X0=0	Y0=0								
U	1 N50	Blank		H	ollow_s	HAFT_B	Lank							
U	- N60	Fin. part		Н	ollon_s	HAFT_S	DE1_E							Search
M	- N70	Stock removal	∇	T	=ROUGHIN	IG_T80 A	A F=0.3/r	ev V=260i	m Longit	udinal				
3,1	- N80	Residual cutting	∇	T	=FINISHIN	G_T35 A	F=0.2/re	ev V=240n	n Longitu	idinal				
)	J N90	Stock removal	$\nabla \Delta \Delta$	T	=FINISHIN	G_T35 A	F=0.15/1	ev V=280	m Longi	tudinal				
	N100	Undercut E		T	=FINISHIN	G_T35 A	F=0.15/1	ev V=200	m E1.0x	0.4 X0=68	20=-55			Mark
, U	N110	Contour		H	OLLOW_S	HAFT_S	DE1_I							- Mark
M	N120	Stock removal		1	-ROUGHIN	IG_1801	F=0.25/r	ev V=250	m Longit	udinal				
M	⁻ N130	Stock removal	$\nabla \Delta \Delta$	1	=HNISHIN	G_1351	F=0.12/r	ev V=280r	m Longiti	udinal			-	_
0	114.40	Settings		5	1: Retract				54.0.0	4.140 5.0	70 00			Сорц
1	N140	Undercut E		1	FINISHIN	G_1351	F=0.15/r	ev V=2001	m E1.0X0	.4 X0=50	20=-20			
0	LH CO	Settings		5	1: Retract							_		
G	N150	F_HUITE1												_
G	NIDO											_		Paste
EN		End of program										_		
														Cut
														-
												~		≣►
												>		
E	<u>ا</u>	dit 🚰 Drillin	g 🛃	Turn- ing		Cont. turn.	[84	Milling	NC	Vari- ous	Ŀ,	Simu- lation	NC.	Ex- ecute

• For stock removal, for example, several machining operations and contours can be linked.

<u>ں</u>	N50	Blank		Hollow_Shaft_Blank	
J.	N60	Fin. part		Hollow_Shaft_Side1_e	
M.	N70	Stock removal	∇	T=ROUGHING_T80 A F=0.3/rev V=260m Longitudinal	
3.8	N80	Residual cutting	∇	T=FINISHING_T35 A F=0.2/rev V=240m Longitudinal	
). Miri-	N90	Stock removal	$\nabla \nabla \nabla$	T=FINISHING_T35 A F=0.15/rev V=280m Longitudinal	

 The integrated contour calculator can process all standard dimensions (Cartesian and polar). It is nevertheless very easy to handle and understand – thanks to colloquial input and graphic support.





• You may switch between the graphic view and parameter screen form with help screen at any time.





• The creation and the machining of the machining plan are not mutually exclusive. With ShopTurn, you can create a new machining plan in parallel with machining.

3.3 You save machining time:

• You can optimize tool selection for the stock removal of contours: Large volumes are removed using roughing tools, and residual material ① is detected and removed automatically using a pointed tool.



• The ability to pinpoint the selected retraction plane means that unnecessary traversing motions can be avoided, thereby saving valuable machining time. This is possible using the settings "Simple", "Extended" and "All".

Retraction plane: Simple	Retraction plane: Extended	Retraction plane: All
ZRA XRA		

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

	SIEME	NS						SIN	UMERIK C	PERATI	03/06/1 10:45 AM		
NC,	/WKS/E	EXAMPLE1/TAPER_SHAF	T								17		
Ρ	N10	Program header		G54 Cyli	nder						^		
G	; Exa	mple by Turning ma	de easy u	vith Shop	urn¶								
G	; Exa	mple 1 : Taper sha	ft¶										
1	N20	Stock removal	$\nabla \nabla \nabla$	T=ROUG	IING_T80 A	F=0.25/	rev V=240n	n Face X	X0=80 Z0=	1			Build
т	N30	Turning T=ROUGHING_1	[80 A V1=24	40m								1	group
	N40	RAPID X82 Z0.3											
	N50	F0.3/rev X-1.6											
	N60	Rapid 21											aanah 📐
	N70	RAPID X82										3	earch
	N80	RAPID 20											
	N90	F0.25/rev X-1.6											
→	H100	Rapid 21											Mark
	N110	RAPID X120 Z200											Tark
J.	N120	Contour		TAPER_S	HAFT_CON	TOUR							
M.	N130	Stock removal	∇	T=ROUG	IING_T80 A	F=0.3/re	ev V=240m	Longitu	udinal		=		
M.	N140	Stock removal	$\nabla \nabla \nabla$	T=FINISH	ING_T35 A	F=0.15/r	ev V=200m	1 Longit	udinal				Conu
	N150	Undercut thrd	⊽≠⊽⊽⊽	T=FINISH	ING_T35 A	F=0.15/r	ev S=200re	ev X0=3	0 Z0=-17)	X1=1.15	inc 🖃		copy
W	N160	Thread long.	⊽ + ⊽⊽⊽	T=THREA	DING_1.5 P	1.5mm/	rev S=800r	ev Outs	ide X0=30	20=0			
3. E	N170	Groove	~ + 777	T=PLUN	E_CUTTER	_3 A F=0	1/rev V=15	50m N2	X0=60 Z0	=-65 T1	=3inc		
G	N180	F_HOME¶											Pooto
G	N190	TO DO											aste
END		End of program											
													Cut
													our
													≣⊾
											~		=
											>		
	e F	dit SSI Drilling	🛄 🤊 Tu	ırn- 🔜	Cont.	[[S=0	Milling	NC	Vari-		Simu-	NC	Ex-
5	· ·			ing 🛛 🖉	turn.	L	, many		ous		lation		ecute

	SIEME	NS											E 03/06/ 10:46 A	17 M		***
NC/	WKS/E	example1/tai	PER_SHAI	T									1	8	Sele	oou
Р	N10	Program hea	der		G	354 Culind	er							^		
G	; Exa	mple by Tur	ning ma	de easu	uith	ShopTu	rn¶									
G	; Exa	mple 1 : Ta	iper sha	ft¶										18		
1	N20	Stock remova	al	$\nabla \nabla \nabla$	T	=ROUGHI	NG_T80 F	F=0.25/	rev V=240	m Face	X0=80 Z0	=1			Build	
т	N30	Turning T=RC	DUGHING_	T80 A V1	=240m										grou	p 🕨
→	N40	RAPID X82 Z	0.3											le:		
	N50	F0.3/rev X-1.	.6													
→	N60	RAPID 21													6	
	N70	RAPID X82													Searc	n 🕨
→	N80	RAPID Z0												le:		
→	N90	F0.25/rev X-	1.6													
→	N100	RAPID 21													Mari	L
→	N110	RAPID X120	Z200												Fian	£
J.	N120	Contour			T	aper_sh	AFT_CON	TOUR								
M.	N130	Stock remova	al	∇	1	=ROUGHI	4G_T80 F	A F=0.3/r	ev V=240r	n Longitı	udinal					
M.	N140	Stock remova	al	$\nabla \nabla \nabla$	T	=FINISHIN	G_T35 A	F=0.15/r	ev V=200	m Longit	udinal				Com	
W	N160	Thread long.		∀+ ⊽⊽7	7 T	=THREAD	ING_1.5 I	P1.5mm/	rev S=800	rev Outs	ide X0=3	0 Z0=0			Cobi	P
	N150	Undercut thre	d	∆+ ∆∆∆	7 I	[=FINISHIN	G_T35 A	F=0.15/r	ev S=200	rev X0=3	0 Z0=-17	X1=1.15	inc 🔁			
3	N170	Groove		A+AA4	7 T	=Plunge	_CUTTER	_3 A F=0	.1/rev V=	150m N2	X0=60 Z	0=-65 T1	=3inc			
G	N180	F_HOME¶													Paet	
G	N190	T0 D0¶													rast	5
END		End of progra	am													
															Cut	
																≣►
_														~		-
													>			
	• E	dit 🔤 🔤	Drilling		Turn-		Cont.	S=0	Milling	NC	Vari-		Simu-	1	NC E	x-
-		-		-	ing		turn.	1			ous	-0	lation		⇒ ec	ute

• With ShopTurn, you can achieve extremely high feedrates with optimum repeat accuracy based on integrated digital technology (SINAMICS drives, etc., SINUMERIK controllers).

4. To ensure that everything function smoothly

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

4.1 The operation of ShopTurn

Powerful software is important, but it also has to be intuitive to operate. Regardless of whether you work with SINUMERIK 840D sl or SINUMERIK 828D as 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).



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

Key	Function
HELP	<help> Calls the context-sensitive online help for the selected window. </help>
SELECT	<select> (also called Toggle key) Selects a listed value. </select>
	 Cursor keys The cursor is moved using the four cursor keys. Use the <cursor right=""> key shown here to open a directory or program (e.g. a cycle) in the editor in edit mode.</cursor>
PAGE UP	<page up=""> Scroll upwards in a menu screen. </page>
PAGE DOWN	<page down=""> Scroll downwards in a menu screen. </page>
END	<end> Moves the cursor to the last text box in a menu screen or table. </end>
DEL	 Edit mode: Deletes the first character to the right. Navigation mode: Deletes all characters.
BACKSPACE	<backspace> Edit mode: Deletes a character selected to the left of the cursor. Navigation mode: Deletes all of the selected characters to the left of the cursor. </backspace>
INSERT	<insert> Press the key to enter Edit mode. Press the key again to exit Edit mode and go to Navigation mode. </insert>
INPUT	<input/> Complete input of a value in the text box. Open a directory or program.

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 commands. 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 opened with this key at any time – irrespective of the operating area you are in at the moment.

Main menu



4.2 The contents of the main menu

4.2.1 Machine

Machine - Manual



Select the "Machine" softkey.



Press the "JOG" key.

Here, the machine is set up and the tool is moved in manual operation. It is also possible to measure tools and to set workpiece zeros.

SI	EMENS												03/06/17 10:52 AM	Μ				
NC/M	PF/CLOSURI	ES												Sel	ect			
∥ Re	set				1	1RD								to	ol			
Work			Position [m	m]			T,F,9	6										
∘ X Z	X 0.000 Z 0.000					Т								Select work offs.				
SP1 SP3				100 °			F		0.0 0.0	900 900	mm,	/min	100%					
							Mas	ter	- 0 0		50		100%					
							<u>د</u>	53	• 0 0		50		100%					
T,S,M																		
	т	Too FINISHING_T	l name 35 A	D 1 ST 1														
	Spindle Spindle M fu	S1 Inction	Ω	200 rpm										-				
	Other M fun Work offset	ction																
														< Ba	K ck			
													>					
10	T,S,M	20 Set 20 WO	1	Meas. workp.	1	Meas. tool	2,1	Posi- tion				2	Stock rem.					

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



Figure 3-2 Specification of a target position

Machine - Auto



Select the "Machine" softkey.



Press the "AUTO" key.

During the machining, the current machining step is displayed. It is possible to switch to a simultaneously running simulation at the press of a key ("Simult. record."). During execution of a machining plan, you may add machining steps or start a new machining plan.

SIEMENS				RATE 03/06/17 11:03 AM	
NC/WKS/EXAMPLE3/GUIDE_	Shaft				G
interrupted		MRD A Stop: N	IC Stop active		functions
Work	Position [mm]	Dist-to-go	T,F,S		
• X	56 400	0 000	T ROUGHING T80 A	R 0.800	
° ^	30.400	0.000	1	7 39 888	Auxiliary
2	-13.322	-41.565	1 🗇 D1	¥ 55 888	Tunctions
- SP1	210.455 °	0.000		A 33.000	
SP3	0.000°	0.000	F 0.000		Basic
			0.300 mm/rev	100%	blocks
			S1 - 1467	2	
			Master 1467	100%	
				. 100	Time /
			53 *	8	counter
₩ 654			0 . 50	100%	
NC/UKS/EXAMPLE3/GUIDE_	SHAFT				Program
\/ N40 Fin. part	G	GUIDE_SHAFT_CONTOUR		<u>~</u>	levels
N50 Stock removal	⊽T	=ROUGHING_T80 A F=0.	3/rev V=260m Longitudinal		
N60 Stock removal	222 T	FINISHING_T35 A F=0.	12/rev V=280m Longitudinal		
N70 Groove	⊽+⊽⊽⊽ T	=PLUNGE_CUTTER_3 A	F=0.1/rev V=150m X0=60 20=-67 T	I=4inc	
N80 Thread long.		=IHREADING_1.5 P1.5n	11/10m 21=10inc	-3	
N100 001: Positions	 	POEN X0=16 Y0=0 X1=0 Y	-140m 21-10mc 11=-16 X2=-16 Y2=0 X3=0 Y3=16	_	
N110 Rectang.pocket	⊽ C+T	=CUTTER 8 F=0.03/t V=	220m X0=0 Y0=0 Z0=0 Z1=3inc		Act. values
END End of program		_			Machine
					≣≻
				>	
NC 00	er-	NC Prog.	Block	Simult.	= Prog.
📑 ste	ore	cntrl.	A search	- record.	Corr.

Figure 3-3 Executing the machining plan



Figure 3-4 Simultaneous recording of the execution

4.2.2 Parameters

Parameter lists



Here, data for the tool management and for programs can be edited.

Tool lists

Cutting is not possible without tools. These can be managed in a tool list.

SIE	MEN	IS									SINU	IME	RII	(0	PERATE 0	3/06/17 1:12 AM	ŗO	AUTO
Tool li	st														MAG	Gazin1	Т	ool
Loc.	Туре	Tool name	ST	D	Length X	Length Z	Radius		\bullet		PI. leng	Ψ,	ాన 1	∄ 2			mea	asure
1	Ū.	ROUGHING_T80 A	1	1	55.000	39.000	0.800	+	95.0	80	12.0	2	~					
2	.	DRILL_32	1	1	0.000	185.000	32.000					S	~					
3	Ø.	FINISHING_T35 A	1	1	124.000	57.000	0.400	←	93.0	35	12.0	ð	~					
4	•	ROUGHING_T80 I	1	1	-9.000	122.000	0.800	+	95.0	80	10.0	P	~				_	_
5	Ţ	PLUNGE_CUTTER_3 A	1	1	85.000	44.000	0.200		3.000		8.0	2	~					
6	1	PLUNGE_CUTTER_3 I	1	1	-12.000	135.000	0.100		3.000		4.0	P	~				_	
7	9	FINISHING_T35 I	1	1	-12.000	122.000	0.400	←	93.0	35	8.0	2	~				Ed	lges 🕨
8		THREADING_1.5	1	1	100.000	0.000	0.050				6.0	Q	~					- /
9	8	CUTTER_8	1	1	0.000	38.000	8.000	3				G	~					
10	62	DRILL_5	1	1	0.000	185.000	5.000		118.0			P	~				_	_
11	.0	BUTTON_TOOL_8	1	1	88.000	38.000	2.000					Q	~					
12		FINISHING_T35_R	1	1	124.000	23.000	0.400	→	93.0	35	10.0	2	~					
13	1	PLUNGE_CUTTER_3P	1	1	86.000	54.000	0.100		3.000		5.0	Q	~					
14		THREADING_3.5	1	1	100.000	0.000	0.050				6.0	P	~					
15	62	SOLIDDRILL_D16	1	1	0.000	50.000	16.000		118.0			P						
16																	Un	load
17																	_	_
18																		
19																	De	lata 1
20																	t	ool
																	Mag sele	azine
																		≣⊦
																>		
-	To lis	ol 📑 Tool tt 🥑 wear			4	Maga zine	a- 📀		Work offset		R	Us vari	er abl	в			SD	Setting data

Figure 3-3 Tool list

_ | _

Magazine

Tools can be organized in a magazine.

SIE	MEN	IS											DPERATE 11:13 A	ιo	AUTO
Magaz	ine												MAGAZIN	De	elete 🕟
Loc.	Туре	Tool name	ST	D	D	z									all
1		ROUGHING_T80 A	1	1											
2		DRILL_32	1	1										Ur	load N
3	Ø.	FINISHING_T35 A	1	1											all
4	•	ROUGHING_T80 I	1	1											_
5	Ţ	PLUNGE_CUTTER_3 A	1	1											
6	<u>.</u>	PLUNGE_CUTTER_31	1	1										1	⊿ heo
1		FINISHING_1351	1	1										-	all
8		THREHDING_1.5	1	1											
10	8	DDILL E	4	-											
11			- 1	-											
12	Ě	FINISHING T35 B	1	1											
13	-	PLUNGE CUTTER 3P	1	1											_
14		THREADING 3.5	1	1											
15	5	SOLIDDRILL D16	1	1											×
16														Rel	ocate 📄
17															
18															
19														De	altion
20														P0 mai	sition
														may	guzino
														Ma	
														sel	ection
														501	Cuon
															_
															≣►
							_	_	_	_					-
													>		-
	To	ol 🛛 Tool						Maga-		Work	R	User		SD	Setting
-	lis	t 🔽 wear						zine	-	orrset		variable			data

Figure 3-6 Magazine

Work offsets

SIEMENS						OPERATE 03/06/17 11:14 AM	
Work offset - Overview	[mm]						
	0.5 F		Х	Z	SP1	SP3	
Machine act value			125.000	250.000	19.983	138.864	
DRF			0.000	0.000	0.000	0.000	
Basic reference			0.000	0.000	0.000	0.000	
Total basic W0			0.000	0.000	0.000	0.000	Active
G54			0.000	0.000	0.000	0.000	
Transf. reference			0.000	0.000	0.000	0.000	
Programmed UO			0.000	0.000	0.000	0.000	
Cycle reference			0.000	0.000	0.000	0.000	Overview
Total UO			0.000	0.000	0.000	0.000	
Tool: CUTTER_8			0.000	38.000			
TOFF			0.000	0.000			
Work actual value			250.000	212.000	19.983	138.864	Base
							G57 Details
						>	
Tool Jist	Tool wear		Ma Zi	ga- 💽 Work ne 💽 offse	R User variable		SD Setting data
Figure 2.7		ark of	ffacto				

Zero points are saved in a clearly laid-out work offset table.

Figure 3-7 Work offsets

4.2.3 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 machining plan with the complete machining sequence for the appropriate workpiece. Your practical knowledge is required to create the optimal sequence.

SIEN	IENS												03/06/ 11:15 A	17 IM	⊃	AUTO
NC/UK	s/exampl	.E2/DR	IVE_SHAF	Г										3		elect
P N1	0 Program	n head	er		G54	Culinder								^		
L N2	0 Stock re	emoval		$\nabla \nabla \nabla$	T=R	DUGHING	_T80 A F	=0.25/re	ev V=240m	Face X	0=80 Z0=1				-	
ີ\ N3	0 Contour				DRIU	e_shaf	Т_СОНТО	UR					→			
M- N4	Ø Stock re	emoval		∇	T=R	DUGHING	_T80 A F	=0.3/reu	V=240m l	Face					F	Ruild 🔊
N5	0 Residua	l cutti	ng	∇	T=FI	NISHING	T35 A F	=0.12/re	v V=240m	Longitue	dinal				g	roup
M6	Ø Stock re	emoval		$\nabla \nabla \nabla$	T=FI	NISHING	_T35 A F	=0.12/re	v V=280m	Longitue	dinal			l.		
🗯 N7	0 Thread	long.		*+***	T=T	IREADIN	G_1.5 P1	.5mm/re	v S=800re	ev Outsid	le X0=24 Z	0=-16				
END	End of p	orogra	m													• b
															5	earch
														l.		
																lask
																lark
														=		
															(Conu
															,	νορφ
															P	aste
																uoto
															-	
																Cut
																≣►
														×		-
													>			
	Edit	<u> </u>	Drilling		Turn-		Cont.	S=0	Milling	NC	Vari-		Simu-		NC	Ex-
					ing		turn.	L			ous		lation			ecute

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 of the dovetailing of geometry and technology:



This relationship between geometry and technology is represented very clearly in the graphical display of the machining steps by a "bracketing" of the corresponding symbols. The "bracketing" signifies a linking of geometry and technology to form a machining step.

Simulating programs

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

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

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



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



For unrestricted use in educational / R&D institutions. © Siemens AG 2016. All rights reserved. SCE_EN_700_020_ShopTurn_4_7_17.docx



Figure 3-10 2-window view

4.2.4 Program Manager

Managing programs



You can create new programs at any time using the Program Manager. You can access existing programs to execute, modify, copy or rename them. Programs that are no longer needed may be deleted.

SIEMENS						03/06/17 11:28 AM	6	→ AUTO
Na	ame	Туре	Length	Date	Time			
Cart programs Cart programs Cart programs Cart programs Cart programs Cart programs		DIR DIR DIR		01/25/16 03/03/17 03/06/17	3:34:10 PM 8:04:34 AM 10:06:24 AM		Direct	ory
C EXAMPLE1 EXAMPLE1_SCE EXAMPLE2	N	ew sequential pr	ogram		0:43:44 AM :15:16 AM 1:14:58 AM			
 DRIVE_SHAFT EXAMPLE2 EXAMPLE3 	Tura	CharTur			1:19:16 AM :29:33 AM 1:27:49 AM			
EXAMPLE3	Name GUIDE SHAFT	ShopTur	n		1:07:10 AM	Ð	ShopT	urn
EXAMPLE5					:34:10 PM			
GHOPTURN GTEMP		WPD		03/06/17	0:06:51 AM 11:08:44 AM		program	GUIDE
							6 00	ae
							_	
							Anj	J
							× Carr	
							Gand	
							~	
NC/Workpieces/EXAMPLE3	3.WPD				Free	: 2.4 MB	OK	

Active programs are marked with a green symbol.

SIEMENS						PERATE 03/06/17 11:29 AM	ſ	→ AUTO
Part programs Subprograms Lorkpices EXAMPLE1 EXAMPLE3 EXAMPLE3 EXAMPLE3 EXAMPLE3 EXAMPLE4 EXAMPLE5 EXAMPLE5 SHOPTURN TEMP	Name IS SCE 54 54_2	Tupe DIR DIR UPD UPD UPD UPD UPD UPD UPD UPD UPD	Length 4030 4018	Date 01/25/16 03/03/17 03/06/17 03/06/17 03/06/17 03/06/17 03/06/17 03/06/17 03/06/17 03/06/17 03/06/17 03/06/17	Ti 3:34:10 8:49:34 8:49:34 8:15:16 11:44:56 3:34:10 11:29:05 3:34:10 10:27:34 10:26:51 11:08:44	ne 111 AM AM AM AM AM AM AM AM	Exe Nu Op Mu Co Co	inter inter
NC/Workpieces/E	Xample4.WPD					Free: 2.4 MB		-



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

Creating a new workpiece

You can manage your programs and other files, such as tool data, zero points and magazine loading, 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	"ShopTurn" program
programGUIDE	
G-Code	"G-Code program"

4.2.5 Diagnostics

Alarms and messages

<u>∕</u>»

Diagnostics



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

Figure 3-11 Alarm log

5. Basics for beginners

This section explains the general basics of the geometry and technology for turning. You do not have to enter anything in ShopTurn in this section.

5.1 Basics of geometry

5.1.1 Tool axes and machining planes

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

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

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

5.1.2 Points in the work area

A few important reference points are available so that a CNC – such as the SINUMERIK 828D with ShopTurn – can orient itself in the existing work area by way of the measuring system.



SCE Training Curriculum | CNC Technology Module 700-020, Edition 02/2016 | Digital Factory, DF FA



Machine zero (M):

The machine zero (M) is specified by the manufacturer and cannot be changed. It is located at 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 freely selected and should be located at the point from which the most dimensions start in the drawing.



Reference point (R):

The reference point (R) is approached for setting the measuring system to zero, as the machine zero cannot be approached in most cases. This is how the controller finds its count start in the position measuring system.



Tool carrier reference point (T):

The tool carrier reference point (T) is relevant for setting up in the case of machines with tool turrets and preset tools. Its position and locating hole enable setting up with tool holders for shank tools in accordance with DIN 69880 and VDI 3425.

5.1.3 Absolute and incremental dimensioning

Absolute input

The entered values are relative to the workpiece zero.

Straigh	t	
х	50.000	abs
Y		abs
Z	-20.000	abs

* G90 Absolute dimensions



With absolute specifications, the absolute coordinate values of the end point must always be entered (the current position is not considered).

Incremental input

SELECT

The entered values are relative to the current position.

Straight	1	
Х	20.000	inc
Y		abs
Z	-5.000	inc

* G91 Incremental dimensions



With incremental specifications, the difference values between the current position and the end point must always be entered while taking the direction into account.

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

A few examples combining absolute and incremental dimensions can be found below:

Straigh	t	
Х	10.000	abs
Y		abs
Z	-35.000	inc

Straigh	t	
Х	25.000	inc
Y		abs
Z	-40.000	abs



5.1.4 Cartesian and polar dimensions

Cartesian input

The coordinates X and Z are entered. The gray values in the example were calculated automatically.

Straight	t ZX		End point +X
Х	100.000	abs	
Х	40.000	inc	
Z	-40.000	abs	
Z [-30.000	inc	
L	50.000		Ø20
α1	126.870	0	
α2	320.906	0	+7
			-40 -10

With absolute specifications, the absolute coordinate values of the end point must always be entered (the current position is not considered).

Polar input

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

Straight	t 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

The angles can be entered as positive angles...



...and/or as negative angles.



You can also combine Cartesian and polar inputs. Two examples:

Specification of the end point in X and the length

Straight	t 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



Specification of the end point in Z and an angle

Straigh	t 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



5.1.5 Circular motions

According to DIN, for circular arcs the end point of the arc (X and Z coordinates in the G18 plane) is specified along with 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 – initially only partially defined – circular arcs.



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	

After the input:

Directio	n of rotation	S
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
K	-20.000	inc
α1	90.000	0
β1	138.590	0
ß2	48.590	0

```
All para-
meters
```

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.

ircle					Circle	
Direct	tion of rotation	2			Directio	n of rotation
R	10.000		4	м	R	20.000
Х	50.000	abs		E	(Ø50) X	30.000
Х	10.000	inc			X	15.000
Z	-35.000	abs	Ø60		Ø30 Z	-6.771
Z	-10.000	inc			Z	-6.771
1	50.000	abs		B20	Y_{+X} I	0.000
1	10.000	inc				0.000
K	-25.000	abs		M K=-20,	AT +Z K	-20.000
K	0.000	inc		-35	ι κ	-20.000
α1	180.000	0			α1	90.000
α2	Tangential					
ß1	90.000	0			61	138.590
ß2	90.000	0			B2	48.590

In DIN format: G2 X50 Z-35 CR=10 In DIN format: G3 X30 Z-6.771 K-20

5.2 Basics of technology

5.2.1 Cutting rate and speeds

Most of the time the cutting rate is directly programmed for turning, in particular for roughing, finishing and grooving. The speed is programmed only for drilling and (most of the time) for thread cutting.

Determination of the cutting rate

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

Material of the tool:

Material of the workpiece:

Hard metal Machining steel

Value:

Vc = 180 m/min

Constant cutting rate v_c (G96) for roughing, finishing and grooving:



To ensure that the selected cutting rate is the same at every workpiece diameter, the speed is adjusted by the controller with command G96 = Constant cutting rate. This is carried out by DC motors or frequency-controlled three-phase motors. As the diameter decreases, the speed increases theoretically to infinity. To avoid danger due to 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.

Therefore, the inputs are: G97 S1900.



$$n = \frac{v_c \cdot 1000}{d \cdot \pi}$$

$$n = \frac{120mm \cdot 1000}{20mm \cdot \pi \cdot min}$$
5.2.2 Feed

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

Determination of the feed

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

Cutting material of the tool:	Hard metal
Material of the workpiece:	Machining steel
Determined value (handbook):	f = 0.2 - 0.4 mm
The mean value is selected:	f = 0.3 mm

Relationship between feed and feedrate:



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



6. Effective setup

In this section, you will learn how to create the tools required for the examples in the following sections. Furthermore, the offset of the tool lengths and the setting of the workpiece zero is explained with examples.

6.1 Tool management

ShopTurn offers three lists for tool management:

- Tool list
- Tool wear list
- Magazine list

6.1.1 Tool list

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

SIE	MEN	IS														03/06/17 1:52 PM	ŗO	
Tool li	st														M	agazin1	То	ool
Loc.	Туре	Tool name	ST	D	Length X	Length Z	Radius				PI. leng	Ц.	る 1	ار 5			mea	sure
1		ROUGHING_T80 A	1	1	55.000	39.000	0.800	+	95.0	80	12.0	2	~					
2	.	DRILL_32	1	1	0.000	185.000	32.000					S	~					
3	Ø.	FINISHING_T35 A	1	1	124.000	57.000	0.400	←	93.0	35	12.0	P	~					
4	•	ROUGHING_T80 I	1	1	-9.000	122.000	0.800	←	95.0	80	10.0	2	~					_
5	Ţ	PLUNGE_CUTTER_3 A	1	1	85.000	44.000	0.200		3.000		8.0	ð	~					
6	1	PLUNGE_CUTTER_3 I	1	1	-12.000	135.000	0.100		3.000		4.0	P	~					
7		FINISHING_T35 I	1	1	-12.000	122.000	0.400	+	93.0	35	8.0	ð	~				Ed	aes 🕨
8	•	THREADING_1.5	1	1	100.000	0.000	0.050				6.0	ð	~					P
9	8=	CUTTER_8	1	1	0.000	38.000	8.000	3				G	~					
10	- 55	DRILL_5	1	1	0.000	185.000	5.000		118.0			S	~				_	
11	.0	BUTTON_TOOL_8	1	1	88.000	38.000	2.000					ð	~					
12		FINISHING_T35_R	1	1	124.000	23.000	0.400	→	93.0	35	10.0	2	~					
13	1	PLUNGE_CUTTER_3P	1	1	86.000	54.000	0.100		3.000		5.0	2	~					_
14	\mathbf{r}	THREADING_3.5	1	1	100.000	0.000	0.050				6.0	ð	~					
15	- 55	SOLIDDRILL_D16	1	1	0.000	50.000	16.000		118.0			S						
16																	Unl	oad
17																		_
18																		
19																		
20																	De	lete ool
																	Mag sele	azine ction
																		≣⊦
																>		
	To	ol 📑 Tool				Maga	-		Work offset		R	Us vari	er abl	e			SD SD	Setting data
		· would												-				

Figure 5-1 Example of tool list

Loc.	Location number
Туре	Tool type
Tool name	The tool is identified by the name and the sister tool number. You may enter the name as a text or number.
ST	Sister 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
Mounting bracket, tip angle, insert width	Mounting bracket (roughing tool and finishing tool), tip angle (drill) and insert width (grooving tool)
#	Direction of spindle rotation
μ	Coolants 1 and 2 (e.g. internal and external cooling)

Meanings of the most important parameters:

ShopTurn provides 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. There are various mounting positions and geometrical parameters (e.g. mounting bracket), depending on the tool type.

New tool – favorites	
Type Identifier	Tool position
500 – Roughing tool	
510 – Finishing tool	2 to 12 to 20</th
520 – Plunge cutter	┥ <u>ਗ਼</u> ੑਗ਼ੑੑੑੑਗ਼ੑੑੑੑੑ
540 - Threading tool	< <u>⊳ <</u> <u>⊳</u> <>
550 – Button tool	∢⊙ ⊙ ⊙)
560 – Rotary drill	
580 – 3D turning probe	← 🕹 ⇒ 📍
730 – Stop	┣╸┻╺╝╶┲
120 – End mill	
140 - Facing tool	シート 二〇 📍
150 – Side mill	
200 – Twist drill	<u>az (j zo (j</u>
240 – Tap	

Figure 5-2 Example of Favorites list

6.1.2 Tool wear list

SIE	MEN	IS								OPERATE 03/06/17 1:55 PM	ŗO	30G
Tool u	ear									NC memory	C -	
Loc.	Туре	Tool name	ST	D	ΔLength X	ΔLength Z	∆Radius	T C	D		50	n p
1		ROUGHING_T80 A	1	1	0.000	0.000	0.000					
2		DRILL_32	1	1	0.000	0.000	0.000					
3	0	FINISHING_T35 A	1	1	0.000	0.000	0.000				Filt	er 🖻
4	•	ROUGHING_T80 I	1	1	0.000	0.000	0.000					_
5	, T	PLUNGE_CUTTER_3 A	1	1	0.000	0.000	0.000					
6	1	PLUNGE_CUTTER_3 I	1	1	0.000	0.000	0.000					
7		FINISHING_T35 I	1	1	0.000	0.000	0.000				Sea	rch 🕨
8		THREADING_1.5	1	1	0.000	0.000	0.000					
9	8=	CUTTER_8	1	1	0.000	0.000	0.000					
10	2	DRILL_5	1	1	0.000	0.000	0.000					_
11		BUTTON_TOOL_8	1	1	0.000	0.000	0.000					
12		FINISHING_T35_R	1	1	0.000	0.000	0.000					
13	H	PLUNGE_CUTTER_3P	1	1	0.000	0.000	0.000					
14		THREADING_3.5	1	1	0.000	0.000	0.000					
15	2	SOLIDDRILL_D16	1	1	0.000	0.000	0.000				Catti	nac 📐
16											Jeili	liyə
1/												
18												
19												
20			-									
											Maga	zine
											selec	tion
										>		
	To	ol 📄 Tool				Ma	ga- 🛛	Work	R User		SD S	etting
	lis	t 🥄 wear				ziı	ne 🛛 🔼	 offset 	variable			data

The wear data for the respective tools is defined here.

Figure 5-3 Tool wear list

The most important tool wear parameters are:

Δ Length X	Length X wear
Δ Length Z	Length Z wear
Δ Radius	Radius wear
TC	 Selection of tool monitoring by tool life (T) by count (C) by wear (W)
Tool life or workpiece count or wear * *Parameter depends on selection in TC	Tool life Workpiece count Tool wear
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 if the check box is selected.

6.1.3 Magazine list

All tools that are assigned to one or more tool magazines are contained in the magazine list. This list displays the status of each tool. In addition, individual magazine locations can be reserved or locked for assigned tools.

SIE	MEN	IS														E 03/0		ĮΟ	JC JC	G
Magaz	zine														ł	IC mem	hory	Del	ete	
Loc.	Туре	Tool name	ST	D	D	z												a	11	2
1		ROUGHING_T80 A	1	1																
2	.	DRILL_32	1	1														Unl	nad	
3	Ð.	FINISHING_T35 A	1	1														a	II	
4		ROUGHING_T80 I	1	1															-	-
5	Ţ	PLUNGE_CUTTER_3 A	1	1																
6	1	PLUNGE_CUTTER_3 I	1	1															od	
7	1	FINISHING_T35 I	1	1														20	au II	
8		THREADING_1.5	1	1																
9	8	CUTTER_8	1	1																
10	22	DRILL_5	1	1																
12		EINICHING THE D	1	-																
12	-	DI UNCE CUTTED 2D	-	-															_	_
14		THREADING 3.5	1	1																
15	5	SOLIDDBILL_D16	1	1																
16		SOCIODINICE_DIV																		
17																				
18																				
19																			-	
20																				
																		Maga selec	azine ction	
																			-	
																			=	
																:	>			
	To	ol 📑 Tool t 🥑 wear					2	Maga- zine	۲	l	Jork ffset	R	User variab	r ole				SD S	ettir data	g

Figure 5-4 Magazine list

Meanings of the most important parameters:

D	Locking of the magazine location
Z	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.
L	Fixed location coding The tool is permanently assigned to this magazine location.

6.2 Tools used



In this section, the tools that are needed for working through the examples later are entered in the tool list.

Select the "Parameter" area in the main menu.

Select the "Tool list" softkey.

Tool
 list

To create a new tool, go to a free location in the tool list ("New tool" softkey must be displayed).

SI	EMEN	IS														03/06/17 1:58 PM	ŢŌ	
Tool I	ist														1	1agazin1		
Loc.	Туре	Tool name	ST	D	Length X	Length Z	Radius					Ψ,	ති 1	1 2				
1		ROUGHING_T80 A	1	1	55.000	39.000	0.800	←	95.0	80	12.0	P	~					
2	-	DRILL_32	1	1	0.000	185.000	32.000					P	~				Ne	ω
3	0	FINISHING_T35 A	1	1	124.000	57.000	0.400	+	93.0	35	12.0	Q	~				to	ol
4	-	ROUGHING_T80 I	1	1	-9.000	122.000	0.800	+	95.0	80	10.0	2	~				_	_
5	Ţ	PLUNGE_CUTTER_3 A	1	1	85.000	44.000	0.200		3.000		8.0	2	~					
6	1	PLUNGE_CUTTER_3 I	1	1	-12.000	135.000	0.100		3.000		4.0	Q	~					
7		FINISHING_T35 I	1	1	-12.000	122.000	0.400	←	93.0	35	8.0	2	~					
8		THREADING_1.5	1	1	100.000	0.000	0.050				6.0	Q	~					
9	8=	CUTTER_8	1	1	0.000	38.000	8.000	3				S	~					
10	62	DRILL_5	1	1	0.000	185.000	5.000		118.0			£	~					_
11	.0	BUTTON_TOOL_8	1	1	88.000	38.000	2.000					2	~					
12		FINISHING_T35_R	1	1	124.000	23.000	0.400	→	93.0	35	10.0	2	~					
13	1	PLUNGE_CUTTER_3P	1	1	86.000	54.000	0.100		3.000		5.0	Q	✓					
14		THREADING_3.5	1	1	100.000	0.000	0.050				6.0	Q	~					
15	62	SOLIDDRILL_D16	1	1	0.000	50.000	16.000		118.0			ጌ					Lo	ad 🕨
17																		· · ·
18																		
19																		_
20	-																	
20																		
																	Maga selec	zine tion
																		≣⊦
																>		
-	To lis	t 📑 Tool wear			4	Maga zine	-		Work offset		R	Us vari	ser able	e			SD S	etting data

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 is inserted in the tool list and you can enter the data of the tool.

Note:

The milling cutter with diameter 8 (CUTTER_8) must be insertable, as it will be used for milling a pocket.

6.3 Tools in the magazine

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

SIE	MEN	IS														03/06/17 2:19 PM	ŗO	
Tool li	st														NC	memory	Та	ol
Loc.	Туре	e Tool name	ST	D	Length X	Length Z	ø		Tip angle			Ϋ́	る 1	ተሻ 2			mea	sure
1		ROUGHING_T80 A	1	1	55.000	39.000	0.800	+	95.0	80	12.0	Q	~					
2	.	DRILL_32	1	1	0.000	185.000	32.000					ጌ	~				Ne	III N
3	Ø.	FINISHING_T35 A	1	1	124.000	57.000	0.400	←	93.0	35	12.0	P	~				to	ol 🕨
4	•	ROUGHING_T80 I	1	1	-9.000	122.000	0.800	+	95.0	80	10.0	2	~					
5	Ţ	PLUNGE_CUTTER_3 A	1	1	85.000	44.000	0.200		3.000		8.0	Q	~					
6	1	PLUNGE_CUTTER_3 I	1	1	-12.000	135.000	0.100		3.000		4.0	P	~				_	
7		FINISHING_T35 I	1	1	-12.000	122.000	0.400	+	93.0	35	8.0	P	~				Edg	jes 🕨
8		THREADING_1.5	1	1	100.000	0.000	0.050				6.0	Q	~					· .
9	8	CUTTER_8	1	1	0.000	38.000	8.000	3				G	~					
10	- 65	DRILL_5	1	1	0.000	185.000	5.000		118.0			S	~					
11	.0	BUTTON_TOOL_8	1	1	88.000	38.000	2.000					P	~					
12		FINISHING_T35_R	1	1	124.000	23.000	0.400	→	93.0	35	10.0	P	~					
13	° -1	PLUNGE_CUTTER_3P	1	1	86.000	54.000	0.100		3.000		5.0	ð	~					
14		THREADING_3.5	1	1	100.000	0.000	0.050				6.0	P	~					
15	- 55	SOLIDDRILL_D16	1	1	0.000	50.000	16.000		118.0			S						
16																	Lo	ad 📄
17																	_	
18																		
19																		
20																	Del	ete
		Milling_tool_8	1	1	0.000	50.000	2.000	4				S	~				10	01
	-8	Milling_tool_5	1	1	0.000	50.000	12.000	4				S	~					
	20	DRILL_10	1	1	0.000	80.000	120.000		118.0			S	~				_	
	1	Roughing_tool_C2	1	1	50.000	10.000	0.200	+	93.0	80	11.0	ð	~				Maga	azine
		Plunge_cutter_4	1	1	50.000	10.000	0.200		3.000		2.0	2	~				sele	ction
																		≣⊦
																		-
																>		
	То	ol 🗾 Tool			4	Maga	-		Work		R .	Us	er				SD S	etting
· ·	118	wear				- 200	-		Unset			/aft	au				_	uaid

Figure 5-6 Selecting the tool in the magazine

Press the "Load" key. The following dialog offers the first free magazine location, which you can change or accept as is.

SIE	MEN	IS														03/06/17 2:19 PM	ŢŌ	
Tool li	st														NC	memory		
Loc.	Туре	Tool name	ST	D	Length X	Length Z	ø		Pitch			ЦĻ	る 1	1 2				
1	-63	ROUGHING_T80 A	1	1	75.000	239.000	0.800	+	95.0	80	12.0	Ð						
2		DRILL_32	1	1	0.000	185.000	32.000					S						
3	ø	FINISHING_T35 A	1	1	124.000	57.000	0.400	+	93.0	35	12.0	Ð						
4		ROUGHING_TRAI	1	1	-9 000	122 000	0 800	4	95.0	80	10.0	ŝ	2					
5	Π	PLUNGE_CU				Load on												
6	1	PLUNGE_CU																
7	2	FINISHING_1																
8		THREADING																
9	8	CUTTER_8			loc.			1	6									
10	2	DRILL_5																
11		BUTTON_TOOL_8	1	1	88.000	38.000	2.000				10.0	5	M					
12		FINISHING_135_R	1	1	124.000	23.000	0.400	-	93.0	35	10.0	2						
13		PLUNGE_CUTTER_3P	1	1	86.000	54.000	0.100		3.000		5.0	5						
14		THREHDING_3.5	- 1	1	100.000	0.000	0.050		440.0		6.0	.12					_	
15	<u>a</u>	SULIDURILL_D16	1	1	0.000	50.000	16.000		118.0			<u>6</u> ,						
17	-																	
10																		
10																	_	
20																		
20	ш.	Milling tool 8	1	1	0 000	50 000	8 000	3				©.						
		Milling tool 5	1	1	0.000	50.000	12.000	3				ۍ ۲						
	55	DBILL 10	i	1	0.000	89,999	120.000		118.0			Ģ						
	(1111	TAP M6	1	1	0.000	30.000	6.000		1.000			ŝ					>	<
		Roughing tool C2	1	1	50.000	10.000	0.200	+	93.0	80	11.0	2					Car	cel
	1	Plunge cutter 4	1	1	50.000	10.000	0.200		3.000		2.0	2						
																		1
																	0	K
																	Ū	
										T								

Figure 5-7 Entering and/or accepting a magazine location

Load

SIE	MEN	IS									SINU	IME	RII	< 0	PERATE	03/06/17 2:20 PM	ŗO	
Tool li	st														M	AGAZIN1	То	ol
Loc.	Туре	Tool name	ST	D	Length X	Length Z	ø		Pitch			Ĥ	1	1/2			mea	sure
1	- 63	ROUGHING_T80 A	1	1	75.000	239.000	0.800	+	95.0	80	12.0	ð	~					
2		DRILL_32	1	1	0.000	185.000	32.000					ፍ	~					
3	0	FINISHING_T35 A	1	1	124.000	57.000	0.400	+	93.0	35	12.0	ð	~					
4	•	ROUGHING_T80 I	1	1	-9.000	122.000	0.800	+	95.0	80	10.0	Q	~					
5	T	PLUNGE_CUTTER_3 A	1	1	85.000	44.000	0.200		3.000		8.0	2	~					
6	1	PLUNGE_CUTTER_3 I	1	1	-12.000	135.000	0.100		3.000		4.0	ð	~					
7	2	FINISHING_T35 I	1	1	-12.000	122.000	0.400	+	93.0	35	8.0	2	~				Edd	ies 🕨
8		THREADING_1.5	1	1	100.000	0.000	0.050				6.0	ð	~					- P
9	-	CUTTER_8	1	1	0.000	38.000	8.000	3				S	~					
10	- 55	DRILL_5	1	1	0.000	185.000	5.000		118.0			S	~					
11		BUTTON_TOOL_8	1	1	88.000	38.000	2.000					2	~					
12		FINISHING_T35_R	1	1	124.000	23.000	0.400	→	93.0	35	10.0	2	~					
13	1	PLUNGE CUTTER 3P	1	1	86.000	54.000	0.100		3.000		5.0	2	~					
14		THREADING 3.5	1	1	100.000	0.000	0.050				6.0	2	~					
15	55	SOLIDDRILL D16	1	1	0.000	50.000	16.000		118.0			G						
16	60000	TAP_M6	1	1	0.000	30.000	6.000		1.000			S					Unk	bad
17		_										-						
18																		
19																		
20																	Del	ete 📐
	th,	Milling tool 8	1	1	0.000	50.000	8.000	3				ଜ	~				to	ol
	-8	Milling tool 5	1	1	0.000	50.000	12.000	3				G	~					
	5	DRILL 10	1	1	0.000	80.000	120.000	-	118.0			G	~					
	1	Roughing tool C2	1	1	50.000	10.000	0.200	+	93.0	80	11.0	3	~				Maga	azine
	T.	Plunge cutter 4	1	1	50.000	10.000	0.200		3.000		2.0	S S	~				selec	tion
																		-
																		≣►
_	_		_				_		_	_	_	_			_			
															_	>		
	To	ol 🔰 Tool			4	Maga	- 4		Work	1F	R	Us	er				SD S	etting
7	lis	t 🖓 wear			4	zine zine			offset	1-		vari	abl	e				data

Once applied, the tool list may look like this.

Figure 5-8 Tool list once applied

6.4 Measuring tools

In the following, you will learn how the tools are offset:

Procedure

T,S,M

Use the "T,S,M" softkey to insert a tool from the tool list into the spindle.

SIEMENS												03/06/17 2:44 PM	Μ	
NC/WKS/EXAMPL	e4/example4	_2		1	MRD								Sel to	ect ol
Work	P 1	osition (mm)				T,F	S	uiuo	2 799.0			D 0 900		
∞ X Z SP1	X 120.000 Z 200.000 sp1 appa [∞]			1	in C)1	а_тойн			Z 39.000 X 55.000	Sel work	ect offs.		
SP3		0.000	>			F	51 ·	0. 0. • 0	.000 .000	mm/	min	100%		
						Ma	ster	0		50		100%	-	-
							53	- 0 0		50		100%	-	
T,S,M	Tool	name	-											
т	1001	D	1 ST											
Spindle Spindle M fu	S1 Inction		rpm											
Other M fun Work offset	ction												•	•
													Ba	K ck
∎∳ T,S,M	P20 Set W0	P	eas. orkp.	<u>.</u>	Meas. tool	2,2	Posi- tion				. ?	> Stock rem.		



Then go to the "Meas. tool" menu.

Enter the probed or turned diameter.



Figure 5-9 Tool measurement – Specification of the X value

Set length The current position of the tool is offset taking into account the workpiece diameter.



Figure 5-10 Tool measurement – Setting length X



You must now repeat this operation for Z.

Figure 5-11 Tool measurement – Setting length Z

6.5 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. Shift the workpiece zero if this does not lie on the end face of the workpiece.



Set W0 Accept your input.

7. Example 1: Taper shaft

7.1 Overview

Learning objectives

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

- Manage and create programs
- Call tools
- Enter traversing paths
- Create any contours with the contour calculator
- Rough and finish contours
- Create a thread undercut
- Create a thread
- Create grooves



Figure 6-1 Workshop drawing - Example 1



Figure 6-2 Workpiece – Example 1

Note:

ShopTurn always saves the last setting you selected with the toggle key. You must therefore ensure that the settings of all units, texts and symbols in the text boxes as well as all toggle fields are the same as in the dialog boxes of the examples.

An available toggle option is always indicated in the help text (see figure below).

Х		
Y Z	-22.500	larget position 2 0
۲		an?
F	*Rapid tr.*	mm/min

Figure 6-3 Example of toggle field

7.2 Program management and creating programs

Operating sequences

SIEMENS					SINUMERIK	OPERATE	03/08/17 10:53 AM	Μ	20G
NC/WKS/EXAMP	le3/guide_shaft							G	
// Reset		MR	RD					funct	ions
Machine	Position [m	m]	T,F,9	S					
MX1 MZ1 MSP1	500.00 500.00	500.000 500.000		T FINISHING_T35 A R 0.400 3 ₪ D1 257.000 X 124.000					iary ions
MSP3	MSP3 0.000 °		F	: 1 -	0.000 0.000 mr	n/min	0.0%		
			Mas	ster	0 50		50%	-	-
			C	53 -	0		Ø		
			0		0 50		50%		
								Act. v Macl	alues hine
									≣⊦
						_	>		
T,S,M	Bo Set WO	Meas. workp.	Meas. tool	Posi- tion		2	Stock rem.		

When the controller starts up, you are in the main screen.



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

SIEMENS						PERATE 03/08/17 10:54 AM	М	
NC/WKS/EXAMPL	.e3/guide_shaft		MRD					То
Machine	Posi	tion [mm]		T,F,S				
MX1 500.000 M21 500.000		T FINISHING 3 👼 D1	R 0.400 2 57.000 X 124.000	m) A			
MSP3 0.000 °			F	0.000 0.000 mm,	/min 0.0%	۲¢ ۲¢	ት G	
				S1 Master	0 0 <u>50</u>	50%	0	L-
				S3 -	0	×	REF	os
				0	0 50	50%		
							→ REF.P	>— OINT
M	ŢŌ	\supset	G	\bigtriangleup	2			
Machine	Parameter	Program	Program manager	Diag- nostics	Setup			

Figure 6-5 Main menu

Figure 6-4 Main screen



Select the "Program Manager" softkey. The "Program Manager" is displayed. In the "Program Manager", you can manage machining plans and contours (e.g. "New", "Open", "Copy", etc.).

SIEMENS			SINU	JMERIK OPERATI	03/08/17 10:55 AM	Ū	
Name	Туре	Length	Date	Time			
👳 🗂 Part programs	DIR		01/25/16	3:38:29 PM			
🗉 🗂 Subprograms	DIR		01/25/16	3:38:30 PM		_	_
🖻 🗂 Workpieces	DIR		03/06/17	10:50:42 AM	\Box		
						Ne	ω 🕨

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	Ŧ
Name	EXAMPLE1		

Figure 6-7 Creating the workpiece



Confirm your entry. The following dialog box opens.

	New sequential program	
T	Ch Tu	_
iype	SnopTurn	· ·
Name	TAPER SHAFT	

Figure 6-8 Creating the sequential program



You can select the input format with the "ShopTurn" and "programGUIDE" G code softkeys.

You specify the program type via the "ShopTurn" softkey.

Enter the name of the machining plan, in this case "TAPER_SHAFT".



Accept your input.



Once applied, the following screen form is displayed for entering the workpiece data.

Figure 6-9 Program header – Help screen

Enter the workpiece data and general program specifications in the program header.

Enter the following details:

Field	Value	Selection via toggle key	Notes
Unit of measurement	mm	Х	
Work offset		Х	
Blank	Cylinder	X	Select the blank shape (here, cylinder) using the toggle key.
ХА	80		
ZA	1		
ZI	-100 abs	Х	
ZB	-92 abs	Х	The value ZB indicates the distance from the chuck.
Retract	Simple	Х	See below Retraction
XRA	5 inc	Х	The dimensions of the retraction planes
ZRA	5 inc	Х	(absolute or
Tool change point	Workpiece	Х	tool change
ХТ	120		here.
ZT	200		
Safety clearance SC	1		
Speed limits S1	3500		
Machined direction of rotation	Down-cut	x	



Accept the entered values. Once applied, the program header is displayed.

	SIEMENS		03/08/ 11:04 f
NC/	/WKS/EXAMPLE1/TAPER_SHAFT		
Р	Program header	G54 Cylinder	\rightarrow
END	End of program		

Figure 6-10 Program header, example 1 – Machining step editor

The program has now 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 program is subsequently executed from top to bottom.



You can open the program header again at any time to make changes or check the values.

Retraction

It is possible to switch the retraction plane between "Simple", "Extended" and "All". Depending on the retraction setting, the associated fields are enabled for the input of the distances.

"Simple" (for simple cylinders)	"Extended" (for complex workpieces with internal machining)	"All" (for complex workpieces with internal machining and/or relief cuts)		
ZRA XRA				
Retract Simple	Retract Extended	Retract All		
700 5.000 inc	XRI 5.000 inc ZR0 5.000 inc	XRI 5.000 inc ZR0 5.000 inc		
208 J.000 IIIC	200 3.000 1110	ZRI 0.000		

Softkeys



Use this softkey to change to the online graphic of the workpiece (see figure below).



Graphic view Figure 6-11 Program header – Graphic view Use the "Graphic view" softkey to change back to the help screen.

7.3 Opening a tool

Operating sequences

Follow the steps below to call the required tool:



Use this key to extend the horizontal softkey menu.



Select the "Straight Circle" softkey.



Select the "Tool" softkey.

Se	ect
to	lo

Open the tool list.

JKS	FXAM	IPI F1	TAPER SHAFT						Tool				Teel
X	,		, in ciconn i	_					T			D 1	liot
~									51		rnm		list
4									Plane s	election	ipin		
	Tool	alacti	08								M0C07IN1		
	TUUTS	GIECU		11/24	1000						TINUNZINT		
-1	Loc.	Туре	e Tool name	ST	D	Length X	Length Z	Radius			-		
	1	10	ROUGHING TRA A	1	1	55 000	39 000	0 800					
	2		DBILL 32	1	1	0.000	185.000	32,000					
	3	1	FINISHING T35 A	1	1	124 000	57 000	0 400					
-5	4	6	ROUGHING T80 I	1	1	-9.000	122,000	0.800					
Ĭ	5	-	PLUNGE CUTTER 3 A	1	1	85.000	44.000	0.200					
	6	'n	PLUNGE CUTTER 31	1	1	-12.000	135,000	0.100					
	7	0	FINISHING T351	1	1	-12,000	122,000	0.400			2		
	8		THREADING 1.5	1	1	100.000	0.000	0.050					
-0	9		CUTTER 8	1	1	0.000	38,000	8.000					
	10	20	DRILL 5	1	1	0.000	185,000	5.000					
	11		BUTTON TOOL 8	1	1	88.000	38,000	2.000					
	12		FINISHING T35 R	1	1	124.000	23.000	0.400					
	13	1	PLUNGE CUTTER 3P	1	1	86.000	54.000	0.100					
11	14												Magazi
	15												selectio
	16												
	17												
	18										~		X
			1	-	-								Cance
													ounce
	150												1
	150	-	50 0	5	0	10	9	150					OF
			1		-							_	UK
We	rt eing	egebe	en										





Use the cursor key to select the "ROUGHING_T80 A" tool.

Apply the tool to the program. After the tool has been applied, enter the following values in the screen form

(if necessary, change the unit using the toggle key):

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

s	IEMENS							03/08/17 12:18 PM
NC/	uks/example	1/TAPER_	Shaft		Tool			
Ρ	Х				T	ROUGHING_T80	A	D 1
END					V1	240.000	m/min	
4			Plane	selection				
						Turning		

Figure 6-13 Tool – Input



7.4 Entering the traversing path

Operating sequences

Now enter the traversing paths:



Select the "Straight" softkey.

Select the "Rapid traverse" softkey.

Enter the following starting point for roughing in the screen form:

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



Enter the starting point for the traversing path



Accept the entered values.



Select the "Straight" softkey.

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

Enter the following values in the screen form:



Figure 6-14 Entering the traversing path



Accept the entered values.

Select the "Straight" softkey.

Select the "Rapid traverse" softkey. Move the tool away from the end face in rapid traverse.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
Z	1 abs	Х	

SIEMENS		
NC/WKS/EXAMPLE1/TAPER_SHAFT	Straig	ht
P XØ T → -0.4	X Y Z	abs abs 1.000 abs
	SP3 F Radius	abs Graphic *Rap.trau.* mm/min s comp.
0.8		
1.2		
-1.6		
		Rapid traverse
2		
-2.4		
		Cancel
-2.8		
0.2 0.4 0.6	8 1 Z	Accept
		>
Straight Circle		

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



Accept the entered values.

Select the "Straight" softkey.

Rapid traverse

Select the "Rapid traverse" softkey.

Enter the following values in the screen form:

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

SIEMENS	SINUMERIK OPERATE 03/08/17 12:26 PM	
NC/WKS/EXAMPLE1/TAPER_SHAFT	Straight	
P XØ T 4 	X 82.000 abs Y abs Z abs SP1 abs SP3 abs F *Rap.traw.* mm/min	Graphic view
-80	Radius comp.	-
60	_	
40	_	Rapid traverse
-20	_	
-0		×
20	_	Cancel
-20 -10 0 10 20	z	Accept
	>	
Straight Circle		

Figure 6-16 Entering the traversing path – Returning to the starting point



Accept the entered values.

Select the "Straight" softkey.

Create the other four traversing paths according to the following list of machining steps.

	SIEMENS												03/08/11 12:31 PM		
NC,	/WKS/Examp	PLE1/TA	PER_SHAP	T									10		Select
Ρ	Program he	ader			G54 Cylin	der							^		
т	Turning T=F	OUGHI	NG_T80 A V	J1=240m	1										_
	RAPID X80	20.3													
→	F0.3/rev X-	1.6													Build
	RAPID Z1														group
→	RAPID X82														_
→	RAPID Z0														
	F0.25/rev X	-1.6													
	RAPID 21													5	earch
	RAPID X120	Z200											\rightarrow		
END	End of prog	ram													
															Mark
															TIAIK
													-		
															Сопц
															oopy
															_
															Paste
															Cut
															_
															= .
													~		=
													>		
	1 .10	200	Delline		Turn-		Cont.	1 18=0	Marin	NC	Vari-		Simu-	NC	Ex-
9	P Edit		Urilling	1	ing		turn.		Filling	-	ous	-0	lation	3	ecute

Figure 6-17 Entering the traversing path – Other four traversing paths



Start the simulation.



Simu

P.

You can end the simulation by selecting either the "Simulation" softkey again or any other horizontal softkey.

7.5 Creating contours with the contour calculator and machining

Contour calculator

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



With the graphic contour calculator, you can enter the contours faster and more easily than with conventional programming – and without any math.

Operating sequences



Follow the steps below to enter the contour: Select the "Cont. turn." softkey.

Select the "New contour" softkey. Enter the name "TAPER_SHAFT_CONTOUR" for the contour.

Each contour is assigned its own name. This makes programs easier to read.

	New contour
Please ente	er the new name
TAPER_SHA	AFT_CONTOUR

Figure 6-19 Creating the "TAPER_SHAFT_CONTOUR" contour



Accept your input.

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

Note:

The contour definition is the roughing limit on the one hand and the finishing distance on the other hand.



Figure 6-20 Entering the starting point

Note:

If you deselect the "Graphic view" softkey, detailed help screens are displayed.



Accept your input.

\$

Enter the following values for the vertical straight-line segment in the screen form:

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



Figure 6-21 Entering the vertical straight-line segment for the contour



←•→

Enter the following values for the horizontal straight line in the screen form:

Field	Value	Selection via toggle key	Notes
Z	-17 abs	Х	A straight line follows to Z-17.
Transition to next element	Cham	Х	
FS	0		+X +Z
			The thread undercut will be inserted later as an individual element.



Figure 6-22 Entering the horizontal straight-line segment for the contour

✓ Accept

‡

Enter the following values for the vertical straight-line segment in the screen form:

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



Figure 6-23 Entering the vertical straight-line segment for the contour



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

Enter the following values for the end point of the inclined straight line in the screen form:



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

Accept

←•→

Enter the following values for the horizontal straight line in the screen form:

Field	Value	Selection via toggle key	Notes
Z	-44 abs	Х	
Transition to next element	Radius	Х	+X +Z
R	2.5		



Figure 6-25 Entering the horizontal straight-line segment for the contour

V Übernehmen

ŧ

Enter the following values for the vertical straight line in the screen form:

Field	Value	Selection via toggle key	Notes
X	60 abs	Х	The line segments (= main elements) are not tangential. Rounding 3 main elements



Figure 6-26 Entering the vertical straight-line segment for the contour



←•→

Enter the following values for the horizontal straight line in the screen form:

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



Figure 6-27 Entering the horizontal straight-line segment for the contour



ŧ

Enter the following values for the vertical straight-line segment in the screen form:

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



Figure 6-28 Entering the vertical straight-line segment for the contour

←•→

Enter the following values for the horizontal straight line in the screen form:

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



Figure 6-29 Entering the horizontal straight-line segment for the contour

‡

Enter the following values for the vertical straight-line segment in the screen form:

Field	Value	Selection via toggle key	Notes
х	80 abs	Х	End point X80 with a
Transition to next element	Cham	Х	
FS	2		+X bo +Z



Figure 6-30 Entering the vertical straight-line segment for the contour



←•→

Enter the following values for the horizontal straight line in the screen form:

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



Figure 6-31 Specifying the contour end point





Figure 6-32 Complete contour
Accept

Accept the contour to apply it to your machining plan.

SIEM	IENS											03/08/17 1:02 PM		100 200	
NC/WKS	/EXAMPLE1/TA	PER_SHA	FT									11			
Р	Program he	ader		G	54 Cylind	er						^	יין	ieω	
Т	Turning T=R	OUGHING_	T80 A V1	=240m											
→	RAPID X80 Z	0.3													
→	F0.3/rev X-1	.6											Gr	aphic	
→	RAPID 21												U	iew	
→	RAPID X82														
→	RAPID 20														
→	F0.25/rev X-	-1.6											Dame		
→	RAPID 21												Renu	nberin	9
→	RAPID X120	Z200													
\r_1 N12	20 Contour			TA	PER_SH	AFT_COM	ITOUR					\ominus			
END	End of prog	am											Open	furthe	r
												_	pro	gram	
												_			
												=			
												_			
												_			
												_			
												_	Se	ttings	
												_		_	
												_			
												_	6	000	
												_	Ŭ	000	
												_		-	
												>			
-	201			Turn-	- 1	Cont	11 22-0		MC	Uari-		Simu-	NC	Ev-	
٤	Edit	Drilling		ing		turn.	1.0-0	Milling		ous	P 0	lation	3	ecute	
	-														

Figure 6-33 Contour in the machining plan

To be able to machine the created contour, you must now create the following machining steps. Proceed as follows:



Select the "Stock removal" softkey.

Open the tool list and select "ROUGHING_T80 A".

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	240 m/min	Х	
Machining	Roughing Longitudinal Outside	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	
Limit	No	Х	



Figure 6-34 Roughing the contour



OK

Accept the entered values.

Select the "Stock removal" softkey.

Open the tool list and select "FINISHING_T35 A".

Apply the tool to the program.

Enter the following values for finishing in the screen form:

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

The following simulation shows the machining sequence you need to check before machining the workpiece.



Figure 6-37 Simulation side view

7.6 Thread undercut

Operating sequences

Follow the steps below to create the 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".





Apply the tool to the program.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	X	
Machining	Roughing /	Х	
	Longitudinal	х	
Pos.		X	See figure above
X0	30		
Z0	-17		
X1	1.15 inc	X	
Z1	4.5 inc	X	
R1	0.8		
R2	0.8		
α	30		
VX	1 inc	X	
D	0.8		
UX	0.1	X	
UZ	0.1	Z	



Figure 6-39 Thread undercut



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

Figure 6-40 Thread undercut - Graphic view

Accept the entered values.

Accept

3D view

Details

P

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

Select the "3D view" softkey.

Select the "Details" softkey. You can manipulate the display as appropriate using the "Zoom +", "Zoom -", "Zoom", etc. softkeys.



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

7.7 Thread

Operating sequences

Follow the steps below to create the thread:



Figure 6-42 Thread



Select the "Thread" softkey.

Open the tool list and select the "THREADING_T1.5" drill.

Apply the tool to the program.

Enter the following values for the thread in the screen form:

Field	Value	Selection via toggle key	Notes
L	1.5 mm/rev	Х	
D	0		
S	800 rpm	Х	
Machining	Roughing/Finishing Linear External thread	X X X	
X0	30	Х	
Z0	0		
Z1	-16 abs	Х	
LW	2		You define the thread
LR	1		following inputs.
H1 0.92			5
αP	29	Х	
ND	8		

Field	Value	Selection via toggle key	Notes
U	0.1		
NN	0		
VR	2		
Multiple threads	No	Х	
α0	0		



Figure 6-43 Thread – Graphic view



Switch to the help screen when necessary.

Figure 6-44 Thread – Help screen







Figure 6-45 Simulation of thread

7.8 Grooves

Operating sequences

Follow the steps below to create the two grooves:



Figure 6-46 Grooves



Select the "Groove" softkey.

Select the "Groove 2" softkey.

Open the tool list and select the grooving tool "PLUNGE_CUTTER_3 A".

Apply the tool to the program.

Field	Value	Selection via toggle key	Notes
F	0.1		
V	150 m/min	Х	
Machining	Roughing/ Finishing		
Pos.			See figure above
X0	60		
Z0	-65		
B1	6	X (field)	
T1	3 inc	Х	
α1	0		You define the geometry
α2	0		of the grooves with the
FS1	0.5	X (field)	following inputs.
R2	1	X (field)	
R3	1	X (field)	
FS4	0.5	X (field)	
D	3		
U	0.1	X (field)	
Ν	2		
DP	10		

Enter the following values for the grooves in the screen form:



Figure 6-47 Grooves - Graphic view



Switch to the help screen when necessary.

Figure 6-48 Grooves - Help screen



Accept the entered values.

SIEME	NS						OPERATE 03/0 1.2	8/17 4 PM		
NC/UKS/E	EXAMPLE1/TAPER_S	HAFT						16	St	ock
P	Program header		G54 Cylinde	r				^	ren	noval
Т	Turning T=ROUGHIN	IG_T80 A V1=2	40m							
→	RAPID X80 20.3									
→	F0.3/rev X-1.6									N.
→	RAPID Z1								Gro	ove
→	RAPID X82									
	RAPID Z0									
→	F0.25/rev X-1.6									
→	RAPID 21								Und	ercut
→	RAPID X120 Z200									
U1 N120	Contour		TAPER_SHA	FT_CONTOUR	1					
×-	Stock removal		T=ROUGHIN	G_T80 A F=0.	3/rev V=240m	n Longitudinal				
¥-	Stock removal	$\nabla \nabla \nabla$	T=FINISHING	_T35 A F=0."	15/rev V=200n	n Longitudinal			Ih	read
100	Undercut thrd	⊽+⊽⊽⊽	T=FINISHING	_T35 A F=0.	15/rev V=200n	n X0=30 Z0=-17 >	(1=1.15inc			
W	Thread long.	4000	T=THREADI	G_1.5 P1.5n	nm/rev S=800	rev Outside X0=30	20=0	-		
1.6	Groove	0+000	T=PLUNGE_	CUTTER_3 A	F=0.1/rev V=1	50m N2 X0=60 Z0	=-65 T1=3in	•		
END	End of program				10				Cu	toff
									-	
									1	
								14		
				Cont		llori	Cimu		mal	Eve
E E	dit Drillin	1g 🚽 🚽	ing	turn.	Milling	ous	Lation	1	-	ecute

Figure 6-49 Machining plan with grooves



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



Select the "Side view" softkey.



Figure 6-50 Simulation – Side view



Select the "Further views" softkey.

2 windows

Select the "2 windows" softkey.



Figure 6-51 Simulation – 2-window view

8. Example 2: Drive shaft

8.1 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:

Preparations

Perform the following steps on your own:

- 1. Create a new workpiece with the name "EXAMPLE2".
- 2. Create a new sequential program with the name "DRIVE_SHAFT".
- 3. Enter the blank dimensions (for the procedure, see example 1).



Figure 7-3 Creating the program header

After creation of the program header, the machining plan looks like this:

	SIEMENS		03/08/ 1:34
NC	/WKS/EXAMPLE2/DRIVE_SHAFT		
Р	Program header	G54 Cylinder	\rightarrow
END	End of program		

Figure 7-4 Machining step program

8.2 Face turning

Operating sequences

Stock removal Follow the steps below to face the workpiece:

Select the "Turning" softkey.

Select the "Stock removal" softkey.

Since face turning is to be completed in one cut, switch the machining to finishing. Select the "ROUGHING_T80 A" tool and enter the following values.



Figure 7-5 Facing the workpiece

8.3 Creating the contour, stock removal and residual stock removal

Operating sequences

Follow the steps below to enter the contour:





Select the "Cont. turning" softkey.



Select the "New contour" softkey. Enter the name "DRIVE_SHAFT_CONTOUR" for the contour.



Figure 7-6 Creating the contour



Accept your input.



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

Figure 7-7 Applying the starting point

Accept your input.

 \checkmark

Accept

†

Enter the following values for the vertical straight-line segment in the screen form:

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



Figure 7-8 Entering the vertical straight-line segment for the contour

Accept the entered values.

←•→

Enter the following values for the horizontal straight-line segment in the screen form:

Field	Value	Selection via toggle key	Notes
Z	-16 abs	Х	
Transition to next element	Cham	х	,×
FS	0		7+



Figure 7-9 Entering the horizontal straight-line segment for the contour



‡

Enter the following values for the vertical straight-line segment in the screen form:

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



Figure 7-10 Entering the vertical straight-line segment for the contour



←•→

Enter the following values for the horizontal straight-line segment in the screen form:

Field	Value	Selection via toggle key	Notes
Z	-38 abs	Х	
Transition to next element	Cham	х	,×,
FS	0		Υ. Α .



Figure 7-11 Entering the horizontal straight-line segment for the contour





Enter the following values for the sloping straight-line segment in the screen form:

Field	Value	Selection via toggle key	Notes
Х	20 abs	Х	
α2	45	Х	
Transition to next element	Cham	Х	The entered angle is relative to the preceding element.
FS	0		



Figure 7-12 Entering the sloping straight-line segment for the contour

Accept

←•→

Enter the following values for the horizontal straight line in the screen form:

Field	Value	Selection via toggle key	Notes
Z	-53 abs	Х	
Transition to next element	Radius	х	·×
R	1		







‡

Enter the following values for the vertical straight line in the screen form:

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



Figure 7-14 Entering the vertical straight-line segment for the contour

Accept



Enter the following values for the next segment in the screen form:

Field	Value	Selection via toggle key	Notes
Х		Х	
Z		Х	
α1	165.167°		+X +Z
Transition to next element	Radius	Х	From the line segment, nothing further is known than the angle relative to the Z-axis of 165.167°. In such cases, continue the construction with the next element.
R	0.4		



Figure 7-15 Entering the inclined straight line for the contour





Enter the following values for the next segment in the screen form:

Field	Value	Selection via toggle key	Notes
Direction of rotation	right	Х	
R	13		
Х			+X +X +Z
Z			The missing points of
1	60 abs	Х	the previous contour element are calculated
К	-78 abs	Х	using the known dimensions of the arc. Since there are several possibilities, you must
Transition to next element	Cham	х	
R	0		select the correct one.



Figure 7-16 Entering the arc for the contour

Dialog select

> Dialog accept

Select the suggested solution corresponding to the figure below:



Figure 7-17 Confirming the contour selection

Once you have selected the desired construction, accept it.

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



Figure 7-18 Applying the arc for the contour



Accept the contour segment.

ŧ

A tangential line segment follows.

Tangent. trans.

Select the "Tangent trans." softkey.

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



Figure 7-19 Entering the vertical straight-line segment for the contour



←•→

Enter the following values for the horizontal straight line in the screen form:

Field	Value	Selection via toggle key	Notes
Z	-100 abs	Х	
Transition to next element	Cham	х	
FS	0		The end point of the contour is at Z-100.



Figure 7-20 Entering the horizontal straight-line segment for the contour



Accept the contour to apply it to your machining plan.



Figure 7-21 Applying the contour

Stock removal, residual stock removal and finishing



To be able to machine the created contour, you must now create the following machining steps. Proceed as follows:

Select the "Stock removal" softkey.



Open the tool list and select the "ROUGHING_T80 A" tool.

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.3		
S	240 rpm	Х	
Machining	Roughing Contour- parallel Outside	X X X	The machining of the contour is performed parallel to the contour here, for example.
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		

Field	Value	Selection via toggle key	Notes
BL	Cylinder	Х	
XD	0.0 inc	Х	
ZD	0.0 inc	Х	
Relief cuts	No	Х	
Limit	No	Х	



Figure 7-22 Roughing the contour



Accept the entered values.

Simulation

Select the "Simulation" softkey.

Side view

Select the "Side view" softkey.



Figure 7-23 Roughing the contour – Simulation side view



Select the "Cont. turning" softkey.



Select the "Cut resid stock" softkey.



Open the tool list and select the "FINISHING_T35 A" tool.

OK

Apply the tool to the program.

Enter the following values for residual stock removal in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.12		
V	240 m/min	Х	
Machining	Roughing Longitudinal Outside	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
Relief cuts	Yes	Х	To be able to cut all residual material, you must toggle the text box to "Yes".
FR	0.2		
Limit	No	Х	



Figure 7-24 Contour: Cutting the residual material



Accept the entered values.

Select the "Simulation" softkey.



Expand the menu.

Activate the display of traversing paths.



Figure 7-25 Cutting residual material - Simulation side view



Select the "Cont. turning" softkey.



Select the "Stock removal" softkey.

Tool OK

Open the tool list and select the "FINISHING_T35 A" tool.

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.12		
S	280 rpm	Х	
Machining	Finishing Longitudinal Outside	X X X	
Allowance	No	Х	
Relief cuts	Yes	Х	
Limit	No	Х	



Figure 7-26 Finishing the contour



Accept the entered values. Once applied, the machining plan looks like this:

SIEM	IENS								SIN	UMERIK	OPERATE	03/08/1 2:39 P	17 M		*	
NC/UKS	/EXAMPLE2/DR	ive_shaft	ſ										6	Ne	ω	
P N10	Program head	er		G54	Cylinder	T00.01	-0.05/		- F V(0_00_70_			^	cont	our	2
	Contour		~~~	DRI	JE SHAF	_ 100 H F	·=0.25/16	0 V=240n	1 Face X	0=00 20=	1		L			
M40	Stock removal		∇	T=R	OUGHING	A F	=0.3/rev	V=240m	Face					Sto	ck	
N50	Residual cuttir	Ig	∇	T=F	NISHING	T35 A F	=0.12/re	v V=240m	Longitu	dinal			U.	rem	oval	1
M.	Stock removal		$\nabla \nabla \nabla$	T=FI	NISHING	_T35 A F	=0.12/re	v S=280re	ev Longit	udinal			h			
END	End of program	n										_	Ŀ			
														Cut r	esid	b
													Ŀ	510	UK	1
														Groo	uina	5
														uroo	ving	2
														Plu	nae	Į,
													U.	res	id.	P
													h			
													Ŀ		-	
														Plur	inge	b
													Ŀ	turr	ing	1
													I.			
														Plu	nge	5
														turn.r	esid.	1
																1
												>				ł.
Ξ.	Edit Sal	Drilling		Turn-		Cont.	[[\$4]	Milling	NC	Vari-		Simu-	E	NC	Ex-	1
=>		Drining		ing		turn.		riming		ous		lation		3	ecute	

Figure 7-27 Machining plan



Select the "Details" softkey. Here, you can zoom into or out from the view.



Simulation

Ŀ,

The "Zoom +" softkey enlarges the view.



Figure 7-28 Simulation 3D view – Details

8.4 Thread

Operating sequences

Thread	
Select tool	
OK	

Select the "Thread" softkey.

Open the tool list and select the "THREADING_T1.5" drill.

Apply the tool to the program.

	ع ماء	مماريدهالم		1		ام م م م ما ۱	:	4	~ ~ ~ ~ ~ ~	1
Enter	me i	Ollowing	values	TOF	me	mean	In	me	screen	TOULD.
		onoming	varaoo	101		unouu			0010011	

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



Figure 7-29 Producing the thread



Simulation

Ŀ,



Figure 7-30 Simulation 3D view - Details

9. Example 3: Guide shaft

9.1 Overview

Learning objectives

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

- Create a random blank shape
- Remove the difference in material between the blank and the machined part
- Drill on the front face
- Mill on the front face

Task



Figure 8-1 Workshop drawing – Example 3
Preparations

Perform the following steps on your own:

- 1. Create a new workpiece with the name "EXAMPLE3".
- 2. Create a new sequential program with the name "GUIDE_SHAFT".
- 3. Fill in the program header (see figure below).

Note:

Despite the random blank shape, select the Cylinder blank shape here. ShopTurn ignores this input and orients itself to the random blank shape.



Figure 8-2 Creating the program header

9.2 Face turning

Operating sequences

Follow the steps below to create a new program and face the blank to Z0:



Field	Value	Selection via toggle key	Notes
F	0.25		
V	240 m/min	Х	
Machining	Finishing	Х	
Pos.	See figure below	Х	
Machining direction	Face	Х	
XO	60		Because the random blank shape has a diameter of 60 mm, you also need to set dimension X0 to 60 in this machining 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 screen form:





Accept



Figure 8-4 Face turning simulation

9.3 Creating a random blank shape contour

Operating sequences

Enter the following blank contour on your own:



New contour 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 figure below).



Figure 8-6 Random blank shape contour

Close contour

Note:

The contour must be closed.

9.4 Creating the finished part contour and stock removal

Operating sequences

Follow the steps below to enter the finished part contour:





Select the "Cont. turning" softkey.



Select the "New contour" softkey. Enter the name "GUIDE_SHAFT_CONTOUR" for the contour.



Figure 8-7 Creating the contour



Accept your input.

Since the blank was planned at Z0 in the first machining step, you can accept the starting point X0/Z0 directly (see figure below).

SIEMENS	SINUMERIK OPERATE 83/18/17
NC/WKS/EXAMPLE3/GUIDE_SHAFT	Starting point
P 🕁 Xø	GUIDE_SHAFT_CONTOUR
	X 0.000 abs
0.12	Z 0.000 abs Graphic
END	Transition at contour start view
	Cham
-0.08	FS 0.000 ¥
0.00	

Figure 8-8 Specifying the contour starting point



Accept your input.

ŧ

Enter the following values for the vertical straight-line segment in the screen form:

Field	Value	Selection via toggle key	Notes
Х	48 abs	Х	$\overline{}$
Transition to next element	Cham	Х	+×
R	3		t ⇒ +Z







←•→

Enter the following values for the horizontal straight-line segment in the screen form:

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







$\widehat{}$

Enter the following values for the next segment in the screen form:

Field	Value	Selection via toggle key	Notes
Direction of rotation	right	Х	
R	23		
Х	60 abs	Х	+X
Z			⇒+Z
1	80 abs	X	If several solutions are possible when contour data is entered (e.g. in the case of circular arcs here), you can select these via the "Dialog select" softkey.



Figure 8-11 Entering the arc for the contour

Select the suggested solution corresponding to the figure below:



Figure 8-12 Selecting an arc for the contour

Once you have selected the desired construction, accept it.

Dialog select

Dialog

accept

Select the suggested solution corresponding to the figure below:



Figure 8-13 Selecting an arc for the contour

Dialog select



Figure 8-14 Applying the selected arc for the contour

Follow the steps below to complete the arc:

Dialog accept



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

Figure 8-15 Entering the arc center point for the contour



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

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



Accept the contour segment.

←•→

Enter the following values for the horizontal straight line in the screen form:

Field	Value	Selection via toggle key	Notes
Z	-75 abs	Х	$\overline{}$
Transition to next element	Radius	Х	+×
R	6		÷≓Z



Figure 8-17 Entering the horizontal straight-line segment for the contour



 \mathbf{X}

Enter the following values for the inclined straight-line segment in the screen form:

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







←•→

Enter the following values for the horizontal straight line in the screen form:

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



Figure 8-19 Entering the horizontal straight-line segment for the contour

Accept



Accept the contour to apply it to your machining plan.

Figure 8-20 Applying the contour

Stock removal

You machine the contour in the next machining step. Proceed as follows:



Select the "Stock removal" softkey.



Open the tool list and select the "ROUGHING_T80 A" tool.

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	260 m/min	Х	
Machining	Roughing Longitudinal Outside	X X X	
D	2.5		
UX	0.2		
UZ	0.2		
DI	0.0		

Field	Value	Selection via toggle key	Notes
BL	Contour	X	Change the blank description to Contour.
Relief cuts	No	X	To ensure that the recess of radius 23 is not machined, you must toggle this setting to "No".
Limit	No	Х	



Figure 8-21 Removing stock from the contour



Accept the entered values. Once applied, the two contours and the machining step are linked.



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

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

9.5 Residual stock removal

Operating sequences

Simulation

Ŀ,

Follow the steps below to cut the residual material:





SIEMENS *** NC/WKS/EXAMPLE3/GUIDE_SHAFT New contou P N10 Program header N20 Stock removal G54 Culinder 434 Cylinder T=ROUGHING_T80 A F=0.25/rev V=240m Face X0=60 20=2 GUIDE_SHAFT_BLANK GUIDE_SHAFT_CONTOUR T=ROUGHING_T80 A F=0.3/rev V=260m Longitudinal Fin. para Stock removal End of program Stock remova Fin. part → Total time: Cut resid stock Grooving Plunge resid. Plunge turning Plunge turn.resid.? Vari-ous Ex-ecute Drilling Turn-SH Milling Þ, Simu-lation 1 Edit NC.

Figure 8-23 Machining plan including roughing machining

Open the tool list and select the "BUTTON_TOOL_8" tool.



Apply the tool to the program.

Select the "Cont. turning" softkey.

Select the "Cut resid stock" softkey.

Enter the following values for residual stock removal in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	240 m/min	Х	
Machining	Roughing Longitudinal Outside	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		

The following figure shows the machining plan up to and including the roughing machining:

Field	Value	Selection via toggle key	Notes
Relief cuts	Yes	X	Toggle the setting for machining with relief cuts to "Yes".
FR	0.2		
Limit	No	Х	



Figure 8-24 Cutting the residual material from the contour



Accept the entered values. Once applied, the list of machining steps looks like this:

S	IEMI	ENS												1:19 Pl	м .		10G
NC/	uks/	EXAMPLE	E3/GUI	de_shaf	T										6	Ne	ω
Ρ	N10	Program	heade	er		G54	Cylinder							1	^	cont	our
1113	N20	Stock re	moval		∇	T=R	oughing	_T80 A F	=0.25/re	v V=240m	Face X0	=60 20=2					
Մլ	N30	Blank				GUIE	DE_SHAFT	T_BLANK									
J.		Fin. part				GUIE	DE_SHAF	T_CONTO	UR							Sto	ick 📐
Mi.		Stock re	moval		V	T=R	OUGHING	_T80 A F	=0.3/rev	V=260m L	ongitudi	nal			E.	rem	oval
). <u></u>		Residual	cuttin	g	∇	T=B	UTTON_T	00L_8 F	=0.25/re	v V=240m	Longitud	linal		⊡			
END		End of p	rogran	1 I													
																Cut r sto	esid ck
																Groo	ving 🗼
																Plui res	id.
															Γ	Plur turr	nge ning
																Plur turn.r	nge resid.
														•	~		
														>			
	,	Edit	S 221	Drilling	-	Turn- ing	▶	Cont. turn.	S=4	Milling	NC	Vari- ous	Þ,	Simu- lation			Ex- ecute

Figure 8-25 Machining 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 "Cont. turning" softkey.



OK

Cont.

Select the "Stock removal" softkey.

Open the tool list and select the "FINISHING_T35 A" tool.

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.12		
S	280 m/min	Х	
Machining	Finishing Longitudinal Outside	X X X	
Allowance	No	Х	
Relief cuts	Yes	Х	
Limit	No	Х	



Figure 8-27 Finishing the contour



Accept the entered values.



Start the "Simulation".



Figure 8-28 Finishing simulation – 3D view

9.6 Groove

Operating sequences

Follow the steps below to create the groove:



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

- - -

SIE	MENS												1:24 F	m		1 1	ÅČ DG
NC/UK	(s/exai	1PLE3/GL	JIDE_SHAF											7	N	eω	
P N:	10 Prog	ram head	ler		G54	Cylinder								^	cor	tour	
L N/	20 Stoc	k remova	1	∇	T=R	OUGHING	6_T80 A	F=0.25/re	ev V=240n	1 Face X)=60 Z0=	2					
V ₁ ₩	30 Blan	k			GUI	DE_SHAF	T_BLAN	K									
<u>-</u> ک	Fin.	part			GUI	De_shaf	T_CONT	OUR							St	ock	
Mi -	Stoc	k remova	I.	∇	T=R	OUGHING	G_T80 A	F=0.3/rev	v V=260m	Longitud	inal				ren	noval	1
h.	Res	dual cutti	ng	∇	T=B	UTTON_1	18_1001	F=0.25/re	v V=240m	ı Longitu	dinal			h			
Mi.	Stoc	k remova	l i	$\nabla \nabla \nabla$	T=FI	NISHING	_T35 A F	=0.12/re	v V=280m	Longitud	dinal		\rightarrow				
END	End	of progra	m												Cut	resid	1
To	otal t	ime: 😐	2:18.73												st	ock	
															Gro	ouina	
															uro	oving	1
														۲.			
															Plu	nge	
														IL.	re	siu.	1
														U			
															DL		
															PIL	ninge	
														I.	tui	illing	
														U			
															Ph	nne	
															turn	resic	
														l.	-	-	
													>				
	F .40	Real	Detition		Turn-		Cont.	TS-I	Marin	NC	Vari-		Simu-	1	NC	Ex-	
_/	Edit		Urilling		ing		turn.	L	Milling		ous	-0	lation		=	ecut	e

Figure 8-29 Machining plan after stock removal

Turn- ing
Groove
M
Select tool

Select the "Turning" softkey.

Select the "Groove" softkey.

Select the second offered groove shape (Groove 2).

Open the tool list and select the grooving tool "PLUNGE_CUTTER_3 A".

Apply the tool to the program.

OK

Field	Value	Selection via toggle key	Notes
F	0.1 mm/rev		
V	150 m/min	Х	
Machining	Roughing / Finishing	x	
Pos.	See figure below	x	
X0	60		
Z0	-67		Here, you enter the
B1	4.2	X (field)	the groove.
T1	4 inc	Х	Ĵ
α1	15		
α2	15		Here you enter the
FS1	1	X (field)	flank angle and the
R2	1	X (field)	rounding at the
R3	1	X (field)	corners.
FS4	1	X (field)	
D	4		
U	0.2	X (field)	
Ν	1		

Enter the following values for the groove in the screen form:



Figure 8-30 Creating the groove



Accept the entered values. Once applied, the list of machining steps looks like this:

SIEM	IENS											03/10/ 1:33 P	17 111		
NC/UKS	/EXAMPLE3/GL	IDE_SHAF	т										8	Sto	nck I
L N20	Stock remova		∇	T=R	OUGHIN	G_T80 A I	F=0.25/r	ev V=240n	n Face X	0=60 Z0=	2		^	rem	oval
℃1 N30) Blank			GUI	DE_SHAF	T_BLAN	K								
<u>-</u> ک	Fin. part			GUI	DE_SHAF	T_CONT	OUR								
¥-	Stock remova	1	∇	T=R	OUGHIN	G_T80 A I	F=0.3/rev	V=260m	Longitue	linal				-	
3.1	Residual cutti	ng	∇	T=B	UTTON_	TOOL_8 F	F=0.25/re	v V=240m	n Longitu	Idinal				Gro	ove
¥.	Stock remova	1	$\nabla \nabla \nabla$	T=F	INISHING	_T35 A F	=0.12/re	v V=280m	n Longitu	dinal			le.		
L.	Groove		∀+ ∀∀∀	T=P	LUNGE_	CUTTER_	3 A F=0.1	/rev V=15	50m X0=	60 Z0=-6	7 T1=4ind	; 🖃			
END	End of progra	m												Unde	arcut
														onuc	arcut
														Thr	ead
													14		
													_		
														Cut	off
													L.	-	
													L.		
													l.		
													-		
												>			
	E-104	Deilling		Turn-		Cont.	1 1840	Million	NC	Vari-		Simu-	F	NC	Ex-
<u>=</u> >		Urining	2×	ing		turn.	L	riming		ous	-0	lation	1	-	ecute

Figure 8-31 Machining plan including groove



Start the "Simulation". You can check subareas of the workpiece using the "Zoom" softkey.



Figure 8-32 Simulation – 3D view (Zoom)

9.7 Thread

Operating sequences

Follow the steps below to create the thread:





Select the "Turning" softkey.

Select the "Thread" softkey.

Open the tool list and select the "THREADING_T1.5" thread cutting tool.

Apply the tool to the program.Enter the following values for the thread in the screen form:

Field	Value	Selection via toggle key	Notes
L	1.5 mm/rev	Х	
D	0		
S	800 rpm	Х	
Machining	Roughing Degressive External thread	X X X	The thread will be created with the Degressive setting. This setting causes the cut segmentation to be reduced for each cut, and so ensures that the cutting cross-section remains constant.
X0	48		
ZO	-3		
Z1	-23 abs	Х	

Field	Value	Selection via toggle key	Notes
LW	4	X (field)	
LR	2		
H1	0.92		
αP	29 Infeed with alternating flank	X (field) X	
ND	8	X (field)	
U	0.1		
VR	2		
Multiple threads	No	Х	
α0	0		



Figure 8-33 Producing the thread



Switch to the help screen when necessary.

Figure 8-34 Help screen - thread run-out

Accept the entered values.

Accept



Start the "Simulation". You can check subareas of the workpiece using the Details softkey.



Figure 8-35 Simulation 3D view - Details

9.8 Drilling

Operating sequences

Follow the steps below to create drill holes on the front face (C axis or complete machining sequence):



After the thread machining has been added, the list of machining steps looks like this:

S	IEM	ENS											E 03/10/ 1:40 P	17 M		222 JOG
NC/	JKS/	EXAMPLE3/GU	IDE_SHAF	T										9		
Р	N10	Program head	er		G54	4 Cylinde	r							^		
<u>.</u>	N20	Stock removal		∇	T=F	ROUGHIN	G_T80 A	F=0.25/r	ev V=240r	n Face X	0=60 Z0=	2			-	_
V1	N30	Blank			GU	ide_shaf	T_BLAN	K								
5		Fin. part			GUI	IDE_SHAP	T_CONT	OUR							Bui	ld 🕟
×-		Stock removal		∇	T=F	ROUGHIN	G_T80 A	F=0.3/rev	v V=260m	Longitud	linal				gro	up 🕨
Ne-		Residual cuttin	ng	∇	T=E	BUTTON_	TOOL_8	F=0.25/re	ev V=240n	n Longitu	dinal					
×.		Stock removal		$\nabla \nabla \nabla$	T=F	INISHING	_T35 A F	=0.12/re	v V=280m	n Longitu	dinal					
1.E		Groove		⊽+⊽⊽⊽	T=F	PLUNGE_	CUTTER_	3 A F=0.1	/rev V=1	50m X0=	60 20=-6	7 T1=4in	•			
W		Thread long.		⊽+⊽⊽⊽	T=1	FHREADIN	IG_1.5 P	1.5mm/re	ev S=800r	ev Outsid	de X0=48	Z0=-3	\rightarrow		Sear	rcn
END		End of program	m													
	Tota	al time: 😐 3	: 14.90											_		
															Ma	rk
															T fa	ĸ
														=		
															Cor	DU
															001	y
																_
															Pas	te
															_	_
															Cu	t
																_
																≣►
														×	_	
					T		0				11. 2		>		_	
		Edit State	Drilling		ing		turn		Milling	NC	vari-		Simu-	NC		EX-
-		_		1-	ing		culli.	-			045		adon			outo

Figure 8-36 Machining plan after thread machining



Select the "Drill." (horizontal) softkey.

Select the "Drilling Reaming" softkey. The workpiece is drilled directly, i.e. without centering.



Select the "Drilling" (vertical) softkey.



Open the tool list and select the "DRILL_5" twist drill.

ok

Apply the tool to the program.

Enter the following values for the drill hole in the screen form:

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



Figure 8-37 Drilling



Accept the entered values. Once applied, the list of machining steps looks like this:

5	IEM	ENS											03/10/1 1:43 P	17 M		
NC/	WKS,	/EXAMPLE3/G	uide_shaf										1	0	Dril	lina 🕠
Ρ	N10	Program hea	der		G54	1 Cylinde	r							^	cen	tric
1	N20	Stock remova	al	∇	T=F	ROUGHIN	G_T80 A I	F=0.25/re	ev V=240m	n Face X	0=60 20=	2		h		
<u>ل</u> -	N30	Blank			GUI	DE_SHAP	T_BLAN	< Contract of the second s						I.		
<u>ل</u> -		Fin. part			GUI	DE_SHAF	T_CONT	DUR							C	
÷.		Stock remova	al	∇	T=F	ROUGHIN	G_T80 A I	F=0.3/re	v V=260m	Longitu	dinal			U.	Cent	ering
he-		Residual cutt	ing	∇	T=E	BUTTON_	TOOL_8 F	=0.25/re	ev V=240m	I Longitu	ıdinal			h		
de la		Stock remova	al	$\nabla \Delta \Delta$	T=F	INISHING	i_T35 A F	=0.12/re	v V=280m	Longitu	Idinal			I.		
<u>LE</u>		Groove		~+~~~	T=F	PLUNGE_	CUTTER_	3 A F=0.1	/rev V=15	i0m X0=	60 Z0=-6	7 T1=4ind	;		Dril	ling
W		Thread long.		~+~~~	T=T	THREADIN	IG_1.5 P1	l.5mm/re	ev S=800re	ev Outsi	de X0=48	Z0=-3		U.	Rear	ning
/g== -		Drilling			⊡+ T=D	DRILL_5 I	F=0.06/re	iv V=140i	n Z1=10in	С			\Box	h		
END		End of progra	ım											I.		
															Deep	hole
														U	dril	ing 🖉
														h		
													-	١.		
															The	hee
														U.		au
														h		
														U.		
														I.		
														Ŀ		
															Posi	tions
														U.	1 001	
														II.		
															_	
															Pos	tion
														~	rep	etit. 🕖
													>			
		Edit	Drilling		Turn-		Cont.	[84	Milling	NC	Vari-		Simu-		NC	Ex-
=			orning		ing		turn.				ous		lation		-	ecute

Figure 8-38 Machining plan following drilling

For the drilling machining step, you may notice an open link in the list of machining steps. This will be automatically linked to the drilling positions in the next step.



Select the "Positions" softkey.



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



Figure 8-39 Entering the positions



Accept the entered values. Once applied, the list of machining steps looks like this:

SI	EMI	ENS										E 03/10/17 1:46 PM		
NC/L	JKS/	EXAMPLE3/GUIDE	_shaft									11		
P I	110	Program header		G54	1 Cylinder							^	•	
L. 1	120	Stock removal	∇	T=F	ROUGHING	_T80 A F	=0.25/re	ev V=240m	Face X)=60 Z0=2				_
VI	130	Blank		GUI	DE_SHAF	T_BLANK								
1		Fin. part		GUI	DE_SHAF	T_CONTO	UR							-• h
×		Stock removal	∇	T=F	ROUGHING	_T80 A F	=0.3/rev	V=260m l	Longitud	inal				
A.		Residual cutting	∇	T=B	BUTTON_1	00L_8 F	=0.25/re	v V=240m	Longitu	dinal				
×.		Stock removal	~~~	T=F	INISHING	_T35 A F	=0.12/re	v V=280m	Longitud	dinal				
LE.		Groove	0+ 000	T=F	PLUNGE_C	UTTER_3	3 A F=0.1	/rev V=15	0m X0=6	60 Z0=-67	T1=4ind	c	9	
W		Thread long.	* ****	T=T	HREADIN	G_1.5 P1	.5mm/re	v S=800re	ev Outsid	le X0=48 2	20=-3			
g∞ 1		Drilling		⊡+ T=D	DRILL_5 F	=0.06/re	v V=140r	n 21=10in	C					
1		001: Positions		⊡+ Z0=	•0 X0=16	Y0=0 X1=	0 Y1=-1	6 X2=-16	Y2=0 X3	=0 Y3=16		\Box		
END		End of program											9	
												_		
												-		
														₩ /
														₽
														«
												~	1	Back
		_		_	_	_				_		>		
		E.404		Turn-		Cont.	1 18=0	Million	NC	Vari-		Simu-	NC	Ex-
=/			rinng 🔤 🔤	ing		turn.		rining		ous	-0	lation	3	ecute

Figure 8-40 Machining plan after input of the position pattern

The drill holes are now linked to the drilling positions.



Start the "Simulation".



Figure 8-41 Simulation – 3D view



Figure 8-42 Simulation - Face view

9.9 Milling the rectangular pocket

Operating sequences

Follow the steps below to create the rectangular pocket on the front face (with the C axis and complete machining sequence).



OK

Apply the tool to the program.

Enter the following values for the rectangular pocket in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.03 mm/tooth	Х	
V	220 m/min	Х	
	Face	Х	
Machining	Roughing Single position	X X	
X0	0	X (field)	
Y0	0	X (field)	
ZO	0		
W	23		
L	23		
R	4		
α0	0		
Z1	3 inc	Х	
DXY	75%	Х	
DZ	1.5		
UXY	0		
UZ	0		
Insertion	Helical	Х	Insertion see below
EP	1		
ER	7		





Accept

Accept the entered values. Once applied, the list of machining steps looks like this:

-	SIEM	ENS										E 03/10/1 1:52 PM	7) X	¢ CG GG	
NC/	(UKS/	EXAMPLE3/GUI	DE_SHAFT									12	1			
Ρ	N10	Program head	er		G54 Cylinde	r						1				
1	N20	Stock removal	∇		T=ROUGHIN	G_T80 A	F=0.25/r	ev V=240r	n Face X	(0=60 Z0=	2					
5.	N30	Blank			GUIDE_SHA	FT_BLAN	ĸ									
5	-	Fin. part			GUIDE_SHA	FT_CONT	OUR									
×.	-	Stock removal	∇		T=ROUGHIN	G_T80 A	F=0.3/re	v V=260m	Longitu	dinal				Pocket		
Del.	-	Residual cuttin	g v		T=BUTTON_	T00L_8	F=0.25/re	ev V=240n	n Longiti	udinal						
de la		Stock removal	222		T=FINISHIN	G_T35 A F	F=0.12/re	v V=280n	n Longitu	ıdinal						
3.E		Groove	∇ + ∇7	∇	T=PLUNGE_	CUTTER_	3 A F=0.1	l/rev V=1	50m X0=	60 Z0=-6	7 T1=4in	с	1	Multi-edge		
W		Thread long.	∀+ ∀₹	∇	T=THREADI	NG_1.5 P	1.5mm/r	ev S=800r	ev Outsi	de X0=48	Z0=-3		L .	spigot	72	
Çez -	1	Drilling			T=DRILL_5	F=0.06/re	ev V=140	m Z1=10iı	10							
1		001: Positions			20=0 X0=16	Y0=0 X1	=0 Y1=-1	6 X2=-16	Y2=0 X	3=0 Y3=16						
<u>ل</u>		Rectang.pocke	t ⊽		T=CUTTER_	8 F=0.3/t	V=220m	X0=0 Y0=	=0 Z0=0 i	Z1=3inc		\ominus				
END		End of program	n											Slot		
												=		Thread milling Engraving	1	
	2	Edit 255	Drilling	Tu	m-	Cont.	5-1	Milling	NC	Vari-	E,	> Simu-	NC	Cont. mill. Ex-		

Figure 8-44 Machining plan after rectangular pocket



Start the "Simulation".



Figure 8-45 Simulation 3D view

Insertion

Perpendicular insertion	Helical insertion	Oscillating insertion			
		~~~			

# 10. Example 4: Hollow shaft

# 10.1 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 machining step editor
- Create an undercut
- Create an asymmetrical groove

Task



Figure 9-1 Workshop drawing - Example 4



Figure 9-2 Blank contour

All non-dimensioned radii are R10
Note:

Side 1 is machined first on account of the better clamping possibility.

# 10.2 Creating the first workpiece side

Creating the machining plan

Since the workpiece is to be machined from two sides (it will be machined without counterspindle), two machining plans must be created.

Create the machining plan for the left side first ("HOLLOW_SHAFT_SIDE1")

### Operating sequences

Create the program "HOLLOW_SHAFT_SIDE1" on your own.



Figure 9-3 Creating the ShopTurn program



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

Figure 9-4 Workpiece dimensions in the program header

# 10.2.1 Face turning

## **Operating sequences**



Follow the steps below to face the blank to Z0: Select the "Turning" softkey.



Select the "Stock removal" softkey.

Select the "ROUGHING_T80 A" tool.

Enter the following values in the screen form:

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



Figure 9-5 Facing the workpiece



Accept the entered values. Once applied, your machining step program looks like this:



10.2.2 Drilling

## **Operating sequences**



Follow the steps below to drill the workpiece in the center. Select the "Drill." softkey.



Select the "Drilling centric" softkey.



Open the tool list and select the "DRILL_32" solid drill.

ok

Apply the tool to the program.

Enter the following values for the drill hole in the screen form:

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



Figure 9-7 Drilling



Accept the entered values. Once applied, the list of machining steps looks like this:



### 10.2.3 Blank contour

Operating sequences

Enter the following blank contour on your own: Since the workpiece will be machined only from one side per machining plan, the blank contour is to be constructed only to Z-65.





Select the "New contour" softkey. Enter the name "HOLLOW_SHAFT_BLANK" for the contour.



Figure 9-9 Creating the contour



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

Figure 9-10 Creating the blank contour

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

### Operating sequences

Follow the steps below to enter the finished part contour:



### Note:

The (red) contour of the machined part intentionally does not correspond to the drawing. The machined part contour serves, on the one hand, as the boundary for the roughing machining limit, but more importantly it specifies the precise traversing path for finishing. The construction therefore 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 beyond the blank. The large diameter is machined only in the second clamping.



Select the "Cont. turning" softkey.



enSelect the "New contour" softkey. Enter the name "HOLLOW_SHAFT_SIDE1_E" for the contour.



Figure 9-11 Creating the contour



Accept your input and set the starting point to X32/Z0.



Figure 9-12 Entering the starting point for the contour



Accept your input.

Field	Value	Selection via toggle key	Notes
Х	68 abs	Х	
Transition to next element	Cham	Х	
F	1		

Enter the following values for the vertical straight-line segment in the screen form:







Accept the entered values.

**‡** 

←•→

Enter the following values for the horizontal straight-line segment in the screen form:

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







Enter the following values for the next segment in the screen form:

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	Cham	х	
FS	0		



Figure 9-15 Entering the arc for the contour



Accept your selection.

Accept the contour segment.

Select the desired construction.

←•→

Enter the following values for the horizontal straight line in the screen form:

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



Figure 9-16 Entering the horizontal straight-line segment for the contour



**‡** 

Enter the following values for the vertical straight-line segment in the screen form:

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



Figure 9-17 Entering the vertical straight-line segment for the contour

Accept



Enter the following values for the inclined straight-line segment in the screen form:

Field	Value	Selection via toggle key	Notes
Х	106 abs	Х	N
α1	135	Х	
Transition to next element	Cham	Х	
FS	0		



Figure 9-18 Entering the inclined straight-line segment for the contour

Accept



Accept Accept

Accept the contour to apply it to your machining plan.

Figure 9-19 Contour in the contour calculator

Once applied, the machining plan looks like this: The two contours are linked automatically.



Figure 9-20 Machining plan after input of the contours

Stock removal, residual stock removal and finishing



You machine the contour in the next machining step. Proceed as follows:

Select the "Stock removal" softkey.



Open the tool list and select the "ROUGHING_T80 A" tool.



Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key Notes	
F	0.3		
V	260 m/min	Х	
Machining	Roughing Longitudinal Outside	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Contour	X	You can choose between the following settings in the blank descriptions: Cylinder: Blank = cylinder Contour: Blank = constructed contour Allowance: Blank = constructed contour with defined allowance
Relief cuts	No	X	The roughing tool is not appropriate for insertion. Therefore, switch the Relief cuts field to" No".
Limit	No	Х	

SIEMENS	SINUMERIK OPERATE 9:09 AM	
SIEMENS NC/UKS/EXAMPLE4/HALLOU_SHAFT_SIDE1	SINUMERIK OPERATE         000000000000000000000000000000000000	Select tool
	DI 0.000 BL Contour Relief cuts No Limit No	
		Cancel
Edit 🚰 Drilling 🔐 Turn- 💕 Cont. 📭 Milling	Vari- Jui Ous Lation	Ex-

Figure 9-21 Roughing the contour



Accept the entered values.

Select the "Cut resid stock" softkey.

Open the tool list and select the "FINISHING_T35 A" tool.



Apply the tool to the program. Before the finishing, the residual material is cut in the hollow groove in this machining step.

Enter the following	n values foi	stock ro	moval of	rosidual	material in	the screen	form
Enter the following	y values io	SLOCK TE	movaror	residual	materiarii	The screen	IOIIII.

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	Х	
Machining	Roughing Longitudinal Outside	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
Relief cuts	Yes	x	So that the hollow groove is taken into consideration, the Relief cuts field must be switched to "Yes".
FR	0.2		
Limit	No	Х	



Figure 9-22 Stock removal of residual material from contour



Accept the entered values.

Select the "Stock removal" softkey.

Open the tool list and select the "FINISHING_T35 A" tool.

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	280 m/min	Х	
Machining	Finishing Longitudinal Outside	X X X	
Allowance	No	Х	
Relief cuts	Yes	Х	Switch Relief cuts to "Yes".
Limit	No	Х	



Figure 9-23 Finishing the contour



Accept the entered values. Once applied, your machining step program looks like this: The contours are automatically linked to the stock removal machining steps.

5	SIEM	ENS												E 03/13/ 9:14 A	17 IM	⊃	
NC/	'WKS/	/EXAMPLE4	/Hal	LOW_SH	IAFT_SID	E1									9		elect
Ρ	N10	Program h	eade	er		G54	1 Cylinde	r							^		
1	N20	Stock rem	oval		$\nabla$	T=F	ROUGHIN	G_T80 A	F=0.2/rev	v V=240m	Face X0	=105 20=	5				
des.	N30	Drilling cer	ntric			T=[	ORILL_32	F=0.1/re	ev S=250	Orev Z1=-	57inc						
<u>ل</u>	N40	Blank				HO	llow_sł	HAFT_BLI	ank								
<u>ل</u> -	N50	Fin. part				HO	LLOW_SH	HAFT_SID	E1_E								
No.	N60	Stock rem	oval		$\nabla$	T=F	ROUGHIN	G_T80 A	F=0.3/rev	v V=260m	Longitue	dinal			le.		
h.	N70	Residual c	uttin	g	$\nabla$	T=F	INISHING	i_T35 A F	=0.2/rev	V=240m	Longitud	inal					
×	N80	Stock rem	oval		$\nabla \nabla \nabla$	T=F	INISHING	i_T35 A F	=0.15/re	v V=280m	n Longitu	dinal					and b
END		End of pro	gram	1										$\Box$		56	earch
															l.		
																r	1ark
																	_
																0	ору
																_	
																P	aste
															l.		-
																	C+
																	out
																-	_
																	≣⊾
_															~		-
														>			
		Edit 🚽	33]	Drilling		lurn-		Cont.	S=0	Milling	NC	Vari-		Simu-		NC	Ex-
-		-			-	mg		turn.	-			ous		adon		-	could

Figure 9-24 Machining plan after machining the contour

# 10.2.5 Undercut

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

You can select from four different types of undercut:

## **Operating sequences**

Follow the steps below to create the undercut:



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

S	IEM	ENS											E 03/13/ 9:14 A	17 M	
NC/	uks/	/EXAMPLE4/	HALLOW_S	HAFT_SID	E1									9	Select
Ρ	N10	Program he	ader		G54	4 Cylinde	r							^	
3	N20	Stock remo	val	$\nabla$	T=F	ROUGHING	G_T80 A	F=0.2/re	v V=240m	Face X0	=105 20=	5			_
Ges	N30	Drilling cent	tric		T=[	DRILL_32	F=0.1/re	ev S=250	Orev Z1=-	57inc					
V	N40	Blank			HO	LLOW_SH	HAFT_BLI	ank							
<u>ل</u>	N50	Fin. part			HO	llow_sf	HAFT_SID	)E1_E							
A.	N60	Stock remo	val	$\nabla$	T=F	ROUGHIN	G_T80 A	F=0.3/re	v V=260m	Longitue	dinal				
de la	N70	Residual cu	tting	$\nabla$	T=F	INISHING	i_T35 A F	=0.2/rev	V=240m	Longitud	inal				
M.	N80	Stock remo	val	$\nabla \nabla \nabla$	T=F	INISHING	i_T35 A F	=0.15/re	v V=280m	n Longitu	dinal				Coursh
END		End of prog	ram										⊒		Search
															Mark
															_
															Сору
															Paste
														le.	_
															0.4
															Cut
															Ξ×
														~	= '
													>		
		Edit 🔜	a Drilling		Turn-		Cont.	<b>[</b> [S=0	Milling	NC	Vari-		Simu-	NC	Ex-
=			oning		ing		turn.		, ming	-	ous		lation		ecute

Figure 9-25 Machining plan after stock removal



Select the "Turning" softkey.

Select the "Undercut" softkey.

Select the "Undercut form E" softkey.



OK

Open the tool list and select the "FINISHING_T35 A" tool.

Apply the tool to the program.

Enter the following values for the groove in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	Х	
Pos.	See figure below E 1.0 x 0.4	x x	
X0	68		
ZO	-55		
X1	0 inc	Х	
VX	70 abs	Х	



Figure 9-26 Specifying the undercut



Accept the entered values. Once applied, the list of machining steps looks like this:

5	IEM	ENS										03/13/1 9:10 Al	7	
NC/	'UKS/	/EXAMPLE4/HA	ILLOW_SHAF	T_SIDE1								1		Select
Р	N10	Program head	ler		G54 Cyli	inder							3	
	N20	Stock removal		$\nabla$	T=ROUG	HING_T80	A F=0.2/re	v V=240m l	Face X0=	=105 Z0=5	i			_
in the second se	N30	Drilling centric	0		T=DRILL	32 F=0.1/	rev S=250	0rev 21=-5	57inc					
<u>ل</u> -	N40	Blank			HOLLOU	J_SHAFT_E	ILANK							Build
V-	N50	Fin. part			HOLLOU	J_SHAFT_S	IDE1_E							group
<b>.</b> -	N60	Stock remova	I	$\nabla$	T=ROUG	HING_T80	A F=0.3/re	v V=260m l	Longitud	inal				
ð	N70	Residual cutti	ng	$\nabla$	T=FINISI	HING_T35 A	A F=0.2/rev	V=240m L	.ongitudi	nal				
M.	N80	Stock remova		~~~	T=FINISI	HING_T35 A	A F=0.15/re	v V=280m	Longitud	linal				Cooreb
	N90	Undercut E			T=FINISI	HING_T35 A	A F=0.15/re	v V=200m	E1.0x0.4	X0=68 Z	0=-55		1	Search
END		End of progra	m											
														_
														Mark
														THUR
												2		
														Conu
														oopy
														_
														Paste
														Cut
														≣≻
_													1	
		_										>		
		Edit 🚽	Drilling	P T	urn-	Cont	· [S=0	Milling	NC	Vari-	Ŀ,	Simu- lation	NC	Ex-

Figure 9-27 Machining plan with undercut



### Start the "Simulation".



Figure 9-28 Simulation – Cut active



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

# 10.2.6 Finished part contour of the first side, internal

## **Operating sequences**

Follow the steps below to enter the finished part contour:





Select the "Cont. turning" softkey.



Select the "New contour" softkey. Enter the name "HOLLOW_SHAFT-SIDE1_I" for the contour.



Figure 9-30 Creating the contour



Accept your input.



Set the starting point to X50/Z0.

Figure 9-31 Entering the starting point for the contour

Accept your input.

 $\checkmark$ 

Accept

Create the contour on your own (see figure below).



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

## Stock removal, residual stock removal and finishing



You machine the contour in the next machining step. The geometries are present in your machining plan as follows:

Figure 9-33 Broken-line graphic



Select the "Stock removal" softkey.

Open the tool list and select the "ROUGHING_T80 I" tool.

**√** OK

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	250 m/min	Х	
Machining	Roughing Longitudinal Inside	X X X	You must switch the machining to "Inside".
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Cylinder	X	Because drilling has already been performed, you do not have to take a black contour into account for the internal machining. Switch to Cylinder.

Field	Value	Selection via toggle key	Notes
XD	32 abs	Х	
ZD	0 inc	Х	
Relief cuts	No	Х	
Limit	No	Х	



Figure 9-34 Roughing the contour



Accept the entered values.

Select the "Stock removal" softkey.

Open the tool list and select the "FINISHING_T35 I" tool.

Apply the tool to the program.

Field	Value	Selection via toggle key	Notes
F	0.12		
V	280 m/min	Х	
Machining	Finishing Longitudinal Inside	X X X	
Allowance	No	Х	
Relief cuts	No	Х	
Limit	No	Х	

Enter the following values for finishing in the screen form:



Figure 9-35 Finishing the contour





Start the simulation to perform checks.



Figure 9-36 Simulation - Cut active

## Undercut



Follow the steps below to create the undercut:

Select the "Undercut" softkey.

Select the "Undercut form E" softkey.

Create the undercut (see figure below).



Figure 9-37 Creating an undercut

## Note:

Make sure that the undercut is in the correct position!



Simulation

P.

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

The machining plan for the first side of the workpiece looks like this:

5	IEME	NS											03/13/ 10:00 A	17 M		
NC/	WKS/E	XAMPLE4/HA	LLOW_SH	IAFT_SIDE	1								1	3		
Р	N10	Program hea	der		G5	i4 Cylinde	er							^	Vi	ew 📄
1	N20	Stock remov	al	$\nabla$	T=	ROUGHIN	IG_T80 A	F=0.2/r	ev V=240n	n Face X	0=105 Z0:	=5				-
Čes	N30	Drilling centr	ic		T=	DRILL_3	2 F=0.1/r	rev S=25	00rev 21=	-57inc						
5-	N40	Blank			HC	DLLOW_S	HAFT_BL	ANK							Gra	phic
<u>ل</u> -ئ	N50	Fin. part			HC	DLLOW_S	HAFT_SI	DE1_E							vi	eω
1	N60	Stock remov	al	$\nabla$	T=	ROUGHIN	IG_T80 A	F=0.3/r	ev V=260n	n Longitu	udinal			L.		_
de la	N70	<b>Residual cut</b>	ting	$\nabla$	T=	FINISHIN	G_T35 A	F=0.2/re	v V=240m	l Longitu	dinal					
1	N80	Stock remov	al	$\nabla \nabla \nabla$	T=	FINISHIN	G_T35 A	F=0.15/r	ev V=280	m Longit	udinal					
ła	N90	Undercut E			T=	FINISHIN	G_T35 A	F=0.15/r	ev V=200	m E1.0x0	.4 X0=68	20=-55			Renur	nbering
J.	N100	Contour			HC	)LLOW_S	HAFT_SI	DE1_I						le.		
M.	N110	Stock remov	al	$\nabla$	T=	ROUGHIN	IG_T80 I	F=0.25/r	ev V=250i	m Longit	udinal					
M.	N120	Stock remov	al	$\nabla \nabla \nabla$	T=	FINISHIN	G_T35   F	F=0.12/re	ev V=280n	n Longitu	ıdinal				Open	further
1	N130	Undercut E			T=	FINISHIN	G_T35   F	F=0.15/re	ev V=200n	n E1.0x0.	4 X0=48 Z	20=-20	∍		pro	gram
END		End of progr	am													
														-		
															-	_
															Set	tinas
																ungo
															_	_
															CI	ose
															_	
																-
														-		
													>			
			Deilling		Turn-		Cont.	<b>1</b> 18=0	Million	NC	Vari-		Simu-		NC	Ex-
	/ E		Urilling		ing		turn.	L	rilling		ous		lation	1	•	ecute

Figure 9-39 Machining plan with undercut

# 10.2.7 The work step editor

Functions of the machining step editor

	Below is an overview of the functions of the machining step editor:
Graphic view	Use the "Graphic view" softkey to change to the broken-line graphic.
Search	You can use the "Search" softkey to search for texts in the program.
Mark	You can use the "Mark" softkey to select more than one machining step for further editing (e.g. copying or cutting).
Сору	You can use this softkey to copy work steps to the clipboard.
Paste	You can use the "Paste" softkey to paste machining steps from the clipboard to the machining plan. The machining step(s) are always inserted after the currently marked machining step.
Cut	You can use the "Cut" softkey to copy machining steps to the clipboard and simultaneously delete them from their point of origin. This softkey can also be used as a deletion key.
≣►	Use this softkey to go to the expanded menu.
Renumbering	Use the "Renumbering" softkey to renumber the machining steps.
Settings	Use this softkey to open the "Settings" dialog. Among other things, you specify whether numbering is to be automatic or whether the block end is to be displayed 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

You will need some of these functions in order to reuse the blank contour of the first side in the machining plan for the second side of the workpiece. You copy the blank contour to the clipboard and insert it in the machining plan for the second side.



# 10.2.8 Copying the contour

## **Operating sequence**

Proceed as follows to copy the blank contour to the clipboard: Navigate to the "HOLLOW_SHAFT_BLANK" contour.

5	IEME	NS							SIN	UMERIK	OPERATE	03/13/17 10:13 AM	$\Box$	
NC/	WKS/E	example4/Hall	ow_shaft_sid	E1								4		liou
P	N10	Program heade	r	G5	4 Cylinde	er					-	^		
100	N20	Stock removal	$\nabla$	T=	ROUGHIN	IG_T80 A	1 F=0.2/r	ev V=240n	n Face X	0=105 20	=5			
Sez.	N30	Drilling centric		=	DRILL_3	2 F=0.1/1	rev 5=25	90rev 21=	-5/inc					
	N40	Blank Fin port		HU									Gr	aphic
No.	NEO	Stock removal	7	T=	BUICHIN	THE TRA 0	DEI_E E=0.3/m	au 11=260m	n Longit	udinal				new
1111. 2 5 -	N70	Residual cutting	, v	T=	FINISHIN	G T35 0	F=0.2/re	u U=240m	n Longitu	dinal				
nati. de -	N80	Stock removal	, .	T=	FINISHIN	G T35 A	F=0.15/r	eu U=280	m Longita	udinal				_
Los	N90	Undercut F		T=	FINISHIN	G T35 A	F=0.15/r	eu U=200	m F1.0x6	.4 X0=68	20=-55		Renu	mbering
10-	N100	Contour		HO	LLOW S	HAFT SI	DE1 I							_
Ň	N110	Stock removal	$\nabla$	T=	ROUGHIN	IG T80 I	F=0.25/r	ev V=250i	m Lonait	udinal				
×-	N120	Stock removal	~~~	T=	FINISHIN	G_T35 I I	F=0.12/re	ev V=280n	n Longitu	udinal			Open	further
144	N130	Undercut E		T=	FINISHIN	G_T35 I I	F=0.15/re	ev V=200n	n E1.0x0	4 X0=48	Z0=-20		pro	gram
END		End of program												
												-	-	
														_
													Se	ttings
													C	lose
													_	_
														. =
												~		4≣
												>		
				Turn-		Cont.	<b>1</b> 18=0	Million	NC	Vari-		Simu-	NC	Ex-
E,	/ E			ing		turn.	L	rinning	111	ous	-0	lation	3	ecute

Figure 9-41 Copying the contour to the clipboard

#### Сору

Copy the blank contour to the clipboard. The contour remains on the clipboard until you copy another machining step to the clipboard or shut down the controller.

# 10.3 Creating the second workpiece side

Creating the machining plan

Proceed as follows to create a machining plan for the second side of the workpiece:

## Operating sequences

Create the program "HOLLOW_SHAFT_SIDE2" on your own.

	New sequential program	
-		
туре	ShopTurn	· ·
Name HOLL	ow_shaft_side2	

Figure 9-42 Creating the ShopTurn program

Enter the following data in the program header (see figure):



Figure 9-43 Workpiece dimensions in the program header

## 10.3.1 Face turning

## Operating sequences

Follow the steps below to face the blank to X-1.6 and Z0:



Select the "Turning" softkey.

Select the "Stock removal" softkey.

~ Accept

Select the "ROUGHING_T80 A" tool.

Enter the following values in the screen form:

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



Figure 9-44 Facing the workpiece



Accept the entered values. Once applied, your machining step program looks like this:



Figure 9-45 Machining plan after face turning

## 10.3.2 Drilling

## **Operating sequences**

	Follow the steps below to drill the workpiece in the center
Drilling	Select the "Drill." softkey.
Drilling centric	Select the "Drilling centric" softkey.
Select tool	Open the tool list and select the "DRILL_32" solid drill.
OK	Apply the tool to the program.

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 screen form:



Figure 9-46 Drilling



Figure 9-47 Machining plan after input of the drilling position

## 10.3.3 Specifying a blank contour

### Operating sequences

Accept

Follow the steps below to paste the blank contour from the clipboard into your machining plan:

First, navigate in the machining plan to the most recently entered machining step (see figure).



Figure 9-48 Position for inserting the blank contour

#### Paste

Paste the blank contour from the clipboard. After the contour is pasted, your machining plan should look like this:

s	IEMI	ENS								SINU	JMERIK C	PERATE	03/13/1 10:23 Af	7	
NC/ P	UKS/ N10	EXAMPLE4/HC	)LLOW_SH	AFT_SID	E2 G54	Culinder							4		Select
	N20 N30	Stock remova Drilling centric		⊽	T=R T=D	OUGHING	i_T80 A F F=0.1/rev	=0.2/reu ) S=2500	V=240m F Trev 21=-6	Face X0= 7inc	105 20=5				1001
Un END	N40	Contour End of progra	m		HOL	.low_sh	AFT_BLA	NK					Ð		Build group
														9	earch
															Mark
													-	-	
															Сору
															-
															Paste
															-
															Cut
															≣≻
E,		Edit Resa	Drilling	?	Turn-		Cont.		Milling	NC Sel	Vari-	Þ,	> Simu- lation	NC	Ex-
				-		-		-				_			

Figure 9-49 Pasting the contour

## 10.3.4 Finished part contour of the second side, external

#### **Operating sequences**

Follow the steps below to enter the finished part contour:



#### Note:

The asymmetrical groove is machined later.



Select the "Cont. turning" softkey.



Select the "New contour" softkey. Enter the name "HOLLOW_SHAFT_SIDE2_E" for the contour.



Accept your input.


Set the starting point to X57/Z0.

Figure 9-50 Entering the starting point for the contour

Accept your input.

(see figure below). SIEMENS NC/UKS/EXAMPLE4/H DILION SHAFT ht line 7 Delete element -65.000 abs 180.000 ° 90.000 ° 
 Z
 -65.000
 abs

 α1
 180.000
 °

 α2
 90.000
 °

 Transition to next element
 • 4200 Cham 0.000 ←•→ FS Ť -160 •--1 END 120  $\times$ -80 ( )-40 ≣► -0 × Cancel -40 -80 -20 -66 -40

Accept

Simu

Ex-

Create the contour up to the end point at Z-65 and X100 on your own

Figure 9-51 Contour in the contour calculator

Drilling

<u>(</u>23)

Accept

 $\checkmark$ 

Accept

Accept the contour to apply it to your machining plan.

Cont. Milling

NC

Vari-Ŀ,

### Stock removal and finishing

You machine the contour in the next machining step. Proceed as follows:



Select the "Stock removal" softkey.

OK

Open the tool list and select the "ROUGHING_T80 A" tool.

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.3		
V	260 m/min	Х	
Machining	Roughing Longitudinal Outside	X X X	
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Contour	Х	
Relief cuts	No	Х	
Limit	No	Х	



Figure 9-52 Roughing the contour



Accept the entered values. Once applied, your machining step program looks like this:

SIEMENS	SINUMERIK OPERATE 03/13/17 19:33 AT		30G
NC/WKS/EXAMPLE4/HOLLOW_SHAFT_SIDE2 P N10 Program header	6 G54 Cylinder	Vie	ew 🕨
N20 Stock removal N30 Drilling centric	T=ROUGHING_T80 A F=0.2/rev U=240m Face X0=105 20=5 T=DRILL_32 F=0.1/rev S=2500rev 21=-67inc		
Ur N40 Blank Ur N50 Fin. part	Hollow_Shaft_Blank Hollow_Shaft_Side2_e	Graj	phic ew
N60 Stock removal N0 End of program	T=ROUGHING_T80 A F=0.3/rev V=260m Longitudinal		
		Renum	nbering
		Open f prog	further Iram
	-		
		Sett	ings
		Clo	ose
			۹≣
Edit Calling	rn- Cont. Simu-	NC	Ex-

Figure 9-53 Machining plan after roughing

Select the "Stock removal" softkey.



Open the tool list and select the "FINISHING_T35 A" tool.

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.15		
V	200 m/min	Х	
Machining	Finishing Longitudinal Outside	X X X	
Allowance	No	Х	
Relief cuts	No	Х	
Limit	No	Х	



Figure 9-54 Finishing the contour



Accept the entered values. Once applied, your machining step program looks like this:

s	IEM	ENS												03/13/ 10:35 A	17 111	$\supset$	10 V	ĉ
NC/	UKS/	/EXAMP	LE4/H	)LLOW_SI	HAFT_SIC	)E2									7			
Ρ	N10	Progra	ım head	ler		G54	Cylinde	r							^		liew	P
1	N20	Stock	remova	1	$\nabla$	T=F	ROUGHIN	G_T80 A I	F=0.2/re	v V=240m	Face X0	=105 20=	5					
25	N30	Drilling	g centri	c		T=D	RILL_32	F=0.1/re	ev S=250	Brev Z1=-	67inc							
U1	H40	Blank				HOI	LOW_SF	iaft_bli	ank							Gr	aphic	
5	N50	Fin. pa	rt			HO	LOW_SH	IAFT_SID	E2_E								view	
Mi.	N60	Stock	remova	I	$\nabla$	T=F	ROUGHING	G_T80 A I	F=0.3/re	v V=260m	Longitue	dinal						
Ì.	N70	Stock	remova	1	$\nabla \Delta \Delta$	T=F	INISHING	_T35 A F	=0.15/re	v V=200m	n Longitu	dinal		→				
END		End of	progra	m												Donu	mhari	
																nenu	mbern	'Y
																Oper	ı furth	er
																pro	ogram	
															Ξ.			
															1			
																_	_	
																Se	ttings	
															1			
															10	_	_	
																0	lose	
																_	_	
															1			
																	•	1
														>				
_						Turn		Cont	Car			lloria		Cimu		and a	Ev-	
		Edit		Drilling		ing		turn.		Milling	NC 1	ous	L.	lation			ecute	
			_		-		-		1-			040			1		oodt	-

Figure 9-55 Machining plan after machining the contour



### 10.3.5 Creating the asymmetrical groove

### **Operating sequences**

Simulation

Ŀ,

Follow the steps below to create the asymmetrical groove:





OK

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.

Apply the tool to the program.

Field	Value	Selection via toggle key	Notes		
F	0.08				
V	180 m/min	Х			
Machining	Roughing/ Finishing	х			
Pos.	See figure below	х			
X0	70				
Z0	-55				
B1	10	X (field)			
T1	5.5 inc	Х			
α1	0				
α2	15				
R1	0	X (field)			
R2	2	X (field)			
R3	0	X (field)			
R4	0	X (field)			
D	3				
U	0.2	X (field)			
Ν	1				

Enter the following values for the groove in the screen form:



Figure 9-57 Specifying the groove



Accept the entered values. Once applied, the list of machining steps looks like this:

-	SIEM	ENS											03/13/1 10:41 Al	7	) [ {	
NC/	/WKS/	/EXAMPLE4/H	)LLOW_SH	AFT_SID	E2									3	Stock	
Ρ	N10	Program head	ler		G54	Cylinder	·						1	S 1	remova	d P
	N20	Stock remova	I.	$\nabla$	T=R(	DUGHING	6_T80 A F	=0.2/rev	V=240m	Face X0	=105 20=5					
n an	N30	Drilling centri	c		T=D	RILL_32	F=0.1/rev	) S=2500	)rev 21=-6	67inc						
J.	N40	Blank			HOL	low_sh	iaft_bla	NK							<b>.</b>	
J.	N50	Fin. part			HOL	LOW_SH	IAFT_SIDE	2_E							Groove	
de la comercia de la come	N60	Stock remova	I	$\nabla$	T=R(	DUGHING	6_T80 A F	=0.3/reu	v V=260m	Longitue	dinal					
de la compañía de la	N70	Stock remova	I	$\nabla \Delta \Delta$	T=FI	NISHING	_T35 A F=	=0.15/re	v V=200m	Longitu	Idinal					
<u>}</u> ∦		Groove		⊽+⊽⊽⊽	T=PL	_UNGE_(	CUTTER_3	A F=0.1	/rev V=15	i0m X0=	70 20=-55	T1=5.5i	nc 🖃		Inderci	
END		End of progra	m											· ·	nuercu	n //
															Thread	
															mout	
													1		_	
															Cutoff	
															_	
															_	
															_	
-															_	
					Turn		Cont	Pas			llori-		Cimu	and a	Ev	
	1	Edit	Drilling	2	ing		turn.		Milling	NC 141	ous	P.	lation		ecu	te

Figure 9-58 Machining plan after groove



### Start the "Simulation".

SIEMENS		18.45 ATI
NC/UKS/EXAMPLE4/HOLLOU_SHAFT_SIDE2		Cut active
		X+
		x -
		¥+
		¥ -
		Z+
		2-
XØ50.000 Z 156.000 Y S1 🔯 T PLUNG ENO End of program Rapid tra	E_CUTTER_3 A D NV 120% 00:	1 « 04:26 Back
Edit 2 Drilling 2 Turn- Scott. 1 Milling	NC Vari- Ous	Simu- Ex- lation ecute

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

### 10.3.6 Finished part contour of the second side, internal

### **Operating sequences**

Follow the steps below to enter the finished part contour:





Select the "Cont. turning" softkey.



Select the "New contour" softkey. Enter the name 'HOLLOW_SHAFT_SIDE2_I' for the contour.

Accept

Accept your input.

Set the starting point to X57/Z0.



Figure 9-60 Entering the starting point for the contour



Accept your input.



Create the contour on your own (see figure below).

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

### Note:

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

The tangential transition applies only to main elements, i.e. the rounding is attached to the main element (see figure below).





Accept the contour. Once applied of the contour, your work step program looks like this:

s	IEM	ENS		SINUMERIK OPERATE 83/13/17 10:49 M	$\supset$	
NC/	UKS/	EXAMPLE4/HOLLOL	J_SHAFT_SIDE2	9	U	iew 🕨
P	N10	Stock removal	⊽	G54 Cylinder T=ROUGHING T80 A F=0.2/rev U=240m Face X0=105 20=5		· · · ·
Čes.	N30	Drilling centric		T=DRILL_32 F=0.1/rev S=2500rev Z1=-67inc		
VI	N40	Blank		HOLLOW_SHAFT_BLANK	Gra	aphic
5	N50	Fin. part		HOLLOW_SHAFT_SIDE2_E	V	iew
Mr.	N60	Stock removal	$\nabla$	T=ROUGHING_T80 A F=0.3/rev V=260m Longitudinal		
	N/0	Stock removal	~~~	1=FINISHING_135 A F=0.15/ rev V=200m Longitudinal		_
anti.	N80	Groove	V+VVV	1=PLUNGE_CUTTER_3 H F=0.1/rev V=150m X0=/0 20=-55 T1=5.5inc	Renu	mbering
END	1190	End of program				-
		chu or program				
					Open	further
					pro	gram
						_
						_
					Co	Hinne
					58	tungs
						_
					C	lose
						_
						4
						1
				>		
E,		Edit 🔤 🔤 Drilli	ing 📑 T	urn- Cont. 📭 Milling 🔤 Vari- 📭 Simu-	NC	Ex-

Figure 9-62 Machining plan after input of the contour

### Stock removal, residual stock removal and finishing



You machine the contour in the next machining step:

Select the "Stock removal" softkey.



Open the tool list and select the "ROUGHING_T80 I" tool.



Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.25		
V	280 m/min	Х	
Machining	Roughing Longitudinal Inside	X X X	You must switch the machining to "Inside".
D	2.0		
UX	0.2		
UZ	0.2		
DI	0.0		

Field	Value	Selection via toggle key	Notes
BL	Cylinder	X	Because drilling has already been performed, you do not have to take a black contour into account for the internal machining. Switch to Cylinder.
XD	32 abs	Х	
ZD	0 abs	Х	
Relief cuts	No	Х	
Limit	No	Х	



Figure 9-63 Roughing the contour



Accept the entered values.



Select the "Cut resid stock" softkey.



OK

Open the tool list and select the "FINISHING_T35 I" tool.

Apply the tool to the program.

Field	Value	Selection via toggle key	Notes
F	0.2		
V	240 m/min	Х	
Machining	Finishing Longitudinal Inside	X X X	
Allowance	No	Х	
Relief cuts	Yes	Х	
FR	0.2		
Limit	No	Х	





Figure 9-64 Residual stock removal from contour



 $\checkmark$ 

OK

Accept the entered values.

Select the "Stock removal" softkey.

Open the tool list and select the "FINISHING_T35 I" tool.

Apply the tool to the program.

Field	Value	Selection via toggle key	Notes
F	0.12		
V	280 m/min	Х	
Machining	Finishing Longitudinal Inside	X X X	
Allowance	No	Х	
Relief cuts	Yes	Х	
Limit	No	Х	

Enter the following values for finishing in the screen form:



Figure 9-65 Finishing the contour

Accept

Accept the entered values.



Start the simulation to perform checks.



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

# 11. Example 5: Plunge-turning

## 11.1 Overview

Learning objectives

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

Task



Figure 10-1 Workshop drawing – Example 5

### Preparations

Perform the following steps on your own:

- 1. Create a new workpiece with the name "EXAMPLE5".
- 2. Create a new sequential program with the name "PLUNGE_TURNING".
- 3. Fill in the program header (see figure below).



Figure 10-2 Creating the program header

### 11.2 Plunge-turning

The level of productivity that can be achieved during turning is limited, among other things, by the possible number of tools in the turret and the frequent tool changes required for effective turning machining. Not all possible contours can be produced with standard turning tools alone. The residual material machining, therefore, is often performed using grooving. For the complete machining of a contour, therefore, it is always necessary to switch between standard turning tools and grooving 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 movements of the turning tool, for example.

As a general rule, there are hardly any empty cuts during the plunge-turning cycle, as stock is removed 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 turned part, and for the stock removal cycle you can select whether you want to machine with conventional methods or with grooving 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 effectively analyze the calculated traversing movements of the tool. Even a combination of conventional turning machining and plunge-turning is possible. For example, a standard tool can be used for roughing, while plunge-turning can be used for machining residual material. This means the contour can be machined completely and without damage.

# 11.3 Creating the contour

### **Operating sequences**

Create the contour on your own.



Accept

Select the "Cont. turning" softkey.

Select the "New contour" softkey. Enter the name 'CONTOUR_E' for the contour.

Accept your input.

Set the starting point to X48/Z0.



Figure 10-3 Entering the starting point for the contour



Create the contour (see figure below).

Figure 10-4 Contour in the contour calculator

## 11.4 Stock removal with the plunge-turning cycle

### **Operating sequences**

You machine the contour in the next machining step.

Proceed as follows:



OK

Select the "Cont. turning" softkey.

Select the "Plunge turning" softkey.





Enter the	following	values	for	roughing	in	the	screen	form:
			-					-

Field	Value	Selection via toggle key	Notes
FX	0.2		
FZ	0.25		
V	150 m/min	Х	
Machining	Roughing Longitudinal Outside	X X X	
D	2.5		
UX	0.2		
UZ	0.2		
DI	0.0		
BL	Cylinder	Х	
XD	50 abs	Х	
ZD	0 abs	Х	
Limit	No	Х	
Ν	1		



Figure 10-5 Roughing the contour



Accept the entered values.

Select the "Part" softkey.



Open the tool list and select the "PLUNGE_CUTTER_3 A" tool.

 $\checkmark$ OK

Apply the tool to the program.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
FX	0.15		
FZ	0.15		
V	200 m/min	Х	
Machining	Finishing Longitudinal Outside	X X X	
Allowance	No	Х	
Limit	No	Х	
Ν	1		



Figure 10-6 Finishing the contour



Simulation

Ŀ,

Accept the entered values. Once applied, your machining step program looks like this:

SIEM	ENS								SINU	JMERIK O	PERATE	03/13/17 11:09 AM			∧° G
NC/UKS/ P N10	/EXAMPLE5/PL Program head	UNGE_TUF Ier	Rhing	G54	Cylinder							4	c	New ontour	
M20	Contour Plunge turning	1	$\nabla$	CON T=P	tour_e Lunge c	UTTER	8 A FX0.2	/rev F20.2	5/rev V	150m Lond	itudinal				
	Plunge turning End of program	j m	$\nabla\nabla\nabla$	T=P	LUNGE_C	UTTER_	8 A F0.15	/rev V200i	m Longi	udinal		Ð	n	Stock emoval	)
													C	ut resid stock	)
													G	rooving	Þ
															l
													F	Plunge resid.	)
													F	Plunge urning	Þ
												_		<b>,</b>	i
													f	Plunge rn.resid	
												~			
												>			
<u></u>	Edit 🔤	Drilling	2	Turn- ing	▶	Cont. turn.	<b>.</b> [\$4	Milling	NC	Vari- ous	<b>P</b> , 1	Simu– lation	NC	Ex- ecute	

Figure 10-7 Machining step program



Figure 10-8 Simulation – Side view (with display of the traversing paths)



Figure 10-9 Simulation – 3D view (with display of the traversing paths)

# 12. Machining the workpieces

Now that you have worked through the examples and acquired a well-founded knowledge of the machining plan creation process in ShopTurn, the next step is to machine the workpieces.

The steps described below are required for the machining:

### **Reference point approach**

After turning on the controller and before executing the machining plans or traversing, you must approach the reference point of the machine manually. This is how ShopTurn finds the count start in the position measuring system of the machine.

Since the reference point approach differs depending on machine type and manufacturer, only basic information can be given here.

- If necessary, move the tool to a free spot in the work space from which it can traverse in all directions without collision. In doing so, ensure that the tool is then not already beyond the reference point of the corresponding axis (since the reference point approach is performed in only one direction for each axis, this point cannot be reached otherwise).
- 2. Perform the reference point approach exactly according to the specifications of the machine manufacturer.

### **Clamping the workpiece**

For dimensionally-correct machining and, naturally, also for your own safety, secure chucking that is appropriate for the workpiece is required. A three-jaw 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 in the Z axis through scratching with an offset tool.

### Executing the machining plan

The machine is now prepared, the workpiece is set up and the tools are measured. At last you can start the machining.

First, select the program you want to execute in the Program Manager, e.g. "HOLLOW_SHAFT_SIDE2".

SIEMENS			SINU	IMERIK OPERATE	03/13/17 11:14 AM	Ū	30G
Name	Туре	Length	Date	Time		Exer	ute
Cart programs Subprograms	DIR		01/25/16	3:34:10 PM 3:34:10 PM		EXO	, are
Gubpiograms Gubpiograms Gubpiograms	DIR		03/13/17	11:12:55 AM			
e 🗇 Example1	WPD		01/25/16	3:34:10 PM	i		
EXAMPLE2 EVOMPLE2	UPD		03/08/17	2:33:05 PM		Ne	ω 🕨
EXAMPLE4	WPD		03/13/17	11:13:12 AM	i		
- ■ HALLOW_SHAFT_SIDE1	MPF	3720	03/13/17	10:00:10 AM		_	
HOLLOW_SHAFT_SIDE2	MPF	3773	03/13/17	10:55:31 AM	⊢	Op	en
	LIPD		03/13/17	10:59:25 HIT			
E TEMP	WPD		03/13/17	8:02:08 AM			
						Ma	rk
						110	u K
						<b>C</b> -	
						0	py
						_	
							ste
						_	
						Ci	ıt
							≣►
NC/Workpieces/EXHMPLE4.WPD				Free:	2.4 MB	-	
	_						
NC NC							

Figure 11-1 Selecting the program



### Open the program.

	SIEME	NS											13.03.1 11:1	5	
NC	/WKS/I	example4/ho	ILLOW_SH	AFT_SI	DE2								1		naicht
Р	N10	Programmko	opf		G5-	4 Zylind	er							· ·	insicili
100	N20	Abspanen		$\nabla$	T=	ROUGHI	NG_T80 A	F=0.2/U	V=240m	plan X0=	105 20=5				
Gez	N30	Bohren Mitti	g		T=I	DRILL_3	2 F=0.1/l	J S=2500	U Z1=-67	'ink					
J	N40	Rohteil			HO	LLOW_9	Shaft_Bl	.ank						Gr	rafische
V	- N50	Fertigteil			HO	LLOW_9	Shaft_Si	DE2_E						F	Insicht
de la compañía	- N60	Abspanen		$\nabla$	T=	ROUGHI	NG_T80 A	F=0.3/U	V=260m	längs					
Ì.	N70	Abspanen		$\nabla \Delta \Delta$	T=	FINISHIN	IG_T35 A	F=0.15/l	J V=200m	längs					
<u>a p</u>	N80	Einstich		⊽+⊽⊽	⊽ <b>T</b> =	PLUNGE	_CUTTER	_3 A F=0	.1/U V=15	i0m X0=i	70 20=-55	T1=5.5i	ık		Neu
J	N90	Kontur			HO	LLOW_9	Shaft_Si	DE2_I						nu	mmerier.
Ì.	- N100	Abspanen		$\nabla$	T=	Roughii	NG_T80 I	F=0.25/l	J V=280m	längs					
3.1	N110	Restabspane	en	$\nabla$	T=	FINISHIN	IG_T35 I F	=0.2/U \	J=240m lä	ings					
3	N120	Abspanen		$\nabla \nabla \nabla$	T=	ROUGHII	NG_T80 I	F=0.12/l	J V=280m	längs				u	eiteres
END		Programmer	nde											Pro	g. öffnen
														Eine	t-11
														EINS	tellungen
														So	:hließen
_															₹
													>		
	۶ e	dit 🔤 🔤	Bohren		Drehen	2	Kontur drehen	S=0	Fräsen	NC	Diver- ses	Ŀ,	Simu- lation	NC	Anwahl

Figure 11-2 Opening the machining plan

```
Ex-
```

Select the "NC Execute" softkey.

-	SIEME	NS				SINUMERIK O	PERATE 03/13/17 11:16 AM	Μ	AUTO		
NC/	WKS/E	Xample4/Hollou	_shaft_side2						G		
Ø	interrup	oted		SB1 MRD 🕂 Stop:	Block ended ir	SBL mode		functions			
Wo	rk		Position [mm]	Dist-to-go	Dist-to-go T,F,S						
\$	X		10 000	0 0 0 0	<b>Α ΘΟΟ Τ</b> ROUGHING_T80 A R 0.800						
~	Ξ.		101 000	2 39.000 Z 39.000				functions			
	2		101.000	0.000	1 🗖 D1		X 55.000	Tarre			
-	SP1 SP3		174.396° 0.000°	0.000 0.000	F	0.000 0.200 mm	/rev 100%	Bablo	asic ocks		
					S1 - Master	3500 3500	100%	The			
					G2 -	0	X	cou	ne / inter		
					33	0	100%				
	654				٥	50	. 100				
HC/	'WKS/E	XAMPLE4/HOLLOU	J_SHAFT_SIDE2					Pro	gram		
Ρ	N10	Program header		G54 Cylinder			^	let	Jels		
1	N20	Stock removal	⊽	T=ROUGHING_T80 A F=0	.2/rev V=240n	n Face X0=105 Z0=	5				
Ges	N30	Drilling centric		T=DRILL_32 F=0.1/rev S	=2500rev Z1=	-67inc		_	_		
J.	N40	Blank		HOLLOW_SHAFT_BLANK	[						
J.	N50	Fin. part		HOLLOW_SHAFT_SIDE2	_E						
nh.	N60	Stock removal	$\nabla$	T=ROUGHING_T80 A F=0	.3/rev V=260n	n Longitudinal					
de la composition de la compos	N70	Stock removal	222	T=FINISHING_T35 A F=0.	15/rev V=200	m Longitudinal	-	0-1-1			
3.H.	N80	Groove	<b>A+</b> ΔΔΔ	T=PLUNGE_CUTTER_3 A	F=0.1/rev V=	150m X0=70 Z0=-5	5 T1=5.5inc	HCt. (	chine		
J.	N90	Contour		HOLLOW_SHAFT_SIDE2	J			That			
de la compañía	N100	Stock removal	▽	T=ROUGHING_T80 I F=0.	25/rev V=280i	m Longitudinal					
de la comercia de la come en en e	N110	Residual cutting	$\nabla$	T=FINISHING_T35   F=0.2/rev V=240m Longitudinal							
de la come	N120	Stock removal	$\nabla \nabla \nabla$	T=ROUGHING_T80   F=0.12/rev V=280m Longitudinal							
END		End of program						_	-		
							>				
		Sto	r- re	Prog. cntrl.	Block Search		Simult. record.	<u></u>	Prog. corr.		

Figure 11-3 Executing



Due to the fact that the machining 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 the machining, select the "Simult. record." softkey before starting. Only then are all traversing motions and their effects displayed.



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

To approach at reduced rapid traverse RG0, the percentage of the maximum rapid traverse of the machine, in the second level of the horizontal softkey bar, must be set under Settings, e.g. 50%.

SIEMENS			ATE 03/13/17 11:19 AM	M A
NC/WKS/EXAMPLE4/HOLLOW_S	iHaft_side2			
// Reset	MRD			
Work Po	osition [mm]	T,F,S		
: X 1	38 000	T ROUGHING_T80 I	R 0.800	
	70.000		Z 122.000	
2	78.000	4 🏰 D1	X-9.000	_
SP1	134.641 °	<b>F</b> 0.000		
553	0.000	F 0.000	1000	
		0.000 mm/min	100%	
		51 ~ 0	×	
		Master 0	100%	
		C2 - 0	X	
		55 0	100%	_
<b>⊞G54</b>		0 . 50	. 100	
Settings for automatic mode				Changeover
				Inch
Dry run feedrate DRY	5000.000	mm/min		
Deduced and it to make DOA	50.000	o.		
Reduced rapid traverse RGØ	50.000	%		
				_
Record machining time	block-bu-block			
				_
Save machining times	No			«
				Back
			>	
			Synch. action.	Settings

Figure 11-4 Setting RG0

Once the percentage is set, the reduced rapid traverse must also be selected under "Program control" with the toggle key.

SIEMENS				SINUMERIK OPERA	IE 03/13/17 11:19 AM	М	AUTO	
NC/WKS/EXAMPLE4/HOLL	ow_shaft_side2							
// Reset		RGØ						
Work	Position [mm]		TES					
* X Z	138.000 78.000	138.000 78.000		T     ROUGHING_T80 I     R 0.800       4     □     D1     Z 122.000       X-9.000     X-9.000     X-9.000				
SP3	SP3 0.000		F 0.000 0.000 mm/min		100%			
			S1 - Master	0 0 50 .	100%			
-			<b>S</b> 3 -	0	<b>X</b> 100%			
l∰•G54			0	50 .	100			
NC/WKS/EXAMPLE4/HOLL	OW_SHAFT_SIDE2		Program o	control				
N10     Program neade       N20     Stock removal       N30     Drilling centric       N40     Blank       N50     Fin, part       N60     Stock removal       N70     Stock removal       N80     Groove       N80     Contour       N100     Stock removal       N110     Stock removal       N110     Stock removal       N110     Stock removal       N110     Stock removal	⊽ ⊽⊽⊽⊽ ⊽+⊽⊽⊽ ⊽ ⊽ ⊽ ⊽	Los Quinder T=ROUGHING_T80 A T=DRILL_32 F=0.1/rev HOLLOU_SHAFT_BLAH HOLLOU_SHAFT_BLAH HOLLOU_SHAFT_SIDE: T=ROUGHING_T80 I T=RNUSHING_T80 I T=ROUGHING_T80 I	PRI DRY RG0 M01 DRF SKP MRD	No axis motion Dry run fedrate Reduced rapid trau, Programmed stop 1 Handuheel offset Skip block Display meas, result SB1: Single block rough		B	ack.	
END End of program					>	Di		
	over- store	Prog. No.	Block search	R.	Simult. record.	_	Prog. corr.	

Figure 11-5 Selecting RG0

### OPERATE keyboard shortcuts

Control key:	
CTRL + P	For screenshots – Storage location: Commis- sioning (password) – System data – HMI data – Logs – Screenshots
CTRL + L	Language switchover
CTRL + C	Сору
CTRL + X	Cut
CTRL + V	Paste
CTRL + Y	Redo (editor functionality)
CTRL + Z	Undo – max. five lines in the editor (editor functionality)
CTRL + A	Select all (editor functionality)
	Go to start of program
CTRL + END	Go to end of program
CTRL + ALT + S	Save complete archive – NCK/PLC/drives/HMI
CTRL + ALT + D	Backup log files on USB or CompactFlash card
CTRL + E	Control energy

Figure 11-6 Keyboard shortcuts

### Keyboard shortcuts, continued

CTRL + M	Maximum simulation speed					
CTRL + F	Search in all screen forms Wildcards "?" and ""*" can be used in search screen forms. "?" stands for any character. "*"					
	for any number of any characters.					
Miscellaneous:						
Shift + NSERT	Commenting out of cycles and direct editing of programGUIDE cycles					
Shift + END	Select up to end of block					
Shift +	Select up to start of line					
	Jump to start of line					
ALT + S	Enter Asian characters					
=	Calculator function					
HELP	Help function					
END	Jump to end of line					

Figure 11-7 Keyboard shortcuts

### Keyboard shortcuts, continued

Simulation and simultaneous recording:						
	Move					
Shift +	Rotate in 3D display					
	Move section					
	Override +/- (simulation)					
CTRL + S	Single block on/off (simulation)					
Insert key:						
RISERT	It brings you into the Edit mode for text boxes or into the Selection mode of combo boxes and toggle fields. You can exit this without making any changes by pressing Insert again.					
NSERT	Undo function, as long as no Input key is pressed or no data has already been trans-ferred to the fields.					
Toggle key:						
SELECT	You can directly switch between toggle fields using the Toggle key (Select) without having to open them. With Shift-Toggle you can switch through these in the reverse direction.					
Cursor key:						
	Open/close directory Open/close program Open/close cycle					

Figure 11-8 Keyboard shortcuts