

SCE Training Curriculums

Siemens Automation Cooperates with Education | 02/2016

CNC Technology Module 700-030 ShopMill 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
 www.siemens.com/sinutrain-downloads
- 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> ia@siemens.com.

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

1.	C	Obje	ective		. 5
2.	l	ntro	ducti	on	. 5
3.	A	\dva	antag	es of working with ShopMill	. 7
	3.1		You	save training time:	. 7
	3.2		You	save programming time:	10
	3.3		You	save machining time:	13
4.	Т	Го е	nsure	e that everything function smoothly	15
	4.1		The	operation of ShopMill	15
	4.2		The	contents of the main menu	17
	4	1.2.1		Machine	17
	4	1.2.2	2	Parameters	20
	4	1.2.3	3	Program	22
	4	1.2.4	1	Program Manager	25
	4	1.2.5	5	Diagnostics	26
5.	E	Basi	cs fo	r beginners	27
	5.1		Basi	cs of geometry	27
	5	5.1.1		Tool axes and machining planes	27
	5	5.1.2	2	Points in the work area	29
	5	5.1.3	3	Absolute and incremental dimensions	30
	5	5.1.4	1	Linear motions	31
	5	5.1.5	5	Circular motions	33
	5.2		Basi	cs of technology	35
	5	5.2.1		The tools in use	35
	5	5.2.2	2	Cutting rate and speeds	38
	5	5.2.3	3	Feed per tooth and feedrates	39
6.	E	Effec	ctive	setup	40
	6.1		Tool	management	40
	6	6.1.1		Tool list	40
	6	6.1.2	2	Tool wear list	42
	6	6.1.3	3	Magazine list	43
	6.2		Tool	s used	43
	6.3		Tool	s in the magazine	45
	6.4		Mea	suring tools	45
	6.5		Setti	ng the workpiece zero	47
7.	E	Exar	nple	1: Longitudinal guide	51
	7.1		Ove	rview	51
	7.2		Prog	ram management and creating programs	52

7.3	Opening a tool and setting the cutter radius compensation	
7.4	Entering the traversing path	
7.5	Creating drill holes and position repetitions	
8. Exa	mple 2: Injection mold	
8.1	Overview	72
8.2	Straight lines and circular paths using polar coordinates	74
8.3	Rectangular pocket	
8.4	Circular pockets on a position pattern	
9. Exa	mple 3: Mold plate	
9.1	Overview	
9.2	Path milling of open contours	
9.3	Solid machining, residual material and finishing of contour pockets	
9.4	Machining on several planes	
9.5	Taking obstacles into account	
10. Exa	mple 4: Lever	120
10.1	Overview	
10.2	Face milling	
10.3	Creating the border for the lever island	
10.4	Machining the lever	
10.5	Creating the border for the circular island	
10.6	Creating the 30 mm circular island	
10.7	Creating a 10 mm circular island	
10.8	Copying the 10 mm circular island	
10.9	Machining the circular island with the help of the editor	
10.10	Deep hole drilling	
10.11	Milling a helix	
10.12	Boring	
10.13	Thread milling	
10.14	Programming contours using polar coordinates	
11. Exa	mple 5: Flange	165
11.1	Overview	
11.2	Creating a subprogram	
11.3	Mirroring of machining steps	
11.4	Holes	
11.5	Rotation of pockets	
11.6	Chamfering contours	
11.7	Longitudinal and circumferential slots	
12. Mac	hining the workpieces	197

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 ShopMill.

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 and productive programming methods are available for each area. With ShopMill, 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 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 ShopMill user to create the NC program directly from the drawing. Even changes and different variants of a workpiece can be quickly programmed due to the clear structure.

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

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

Although ShopMill is easy to learn, this ShopMill Learn-/Training Documentallows you to get started in this world even faster. Before it comes to the actual work with ShopMill, however, important basics are discussed in the first sections:

- First, we will show you the advantages of ShopMill.
- Then we show you the basics of operation with SINUMERIK Operate.
- 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 ShopMill:

- Five examples have been chosen to explain the possibilities for machining with ShopMill, 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 ShopMill.
- Note that the technology data used here only serves as an example due to the wide variety of situations in the workshop.

Just as ShopMill was created with the help of skilled workers, this training curriculum was also elaborated by practitioners. In this sense, we wish you much pleasure and success in your work with ShopMill.

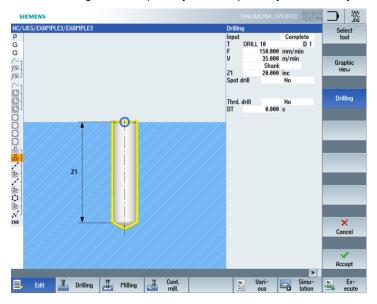
3. Advantages of working with ShopMill

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

3.1 You save training time:

• ShopMill 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 ShopMill, you are optimally assisted by colored help screens.

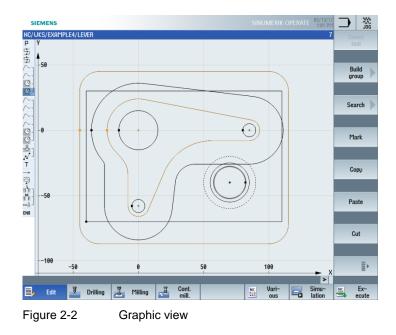
• You may also integrate DIN/ISO commands into the graphical machining plan of ShopMill. You may also program in DIN/ISO 66025 and use DIN cycles.

	i94
T N30 T=EM16	
G N35 GØ X85 Y22.5	
G N40 G0 Z2 S500 M3 M8	
G N45 G0 Z-10	
G N50 G1 X-85 F200	
G N55 G0 Y-22.5	
G N60 G1 X85	
G N65 G0 Z100 M5 M9	

• You may switch between the individual machining step and the workpiece graphic at any time when creating a machining plan.

5	IEME	NS							SIN	JMERIK (OPERATE	03/13/17 1:01 PM	\supset	ېنې اوو	
NC/	WKS/E	XAMPLE4/LEVER										7			
Ρ	N10	Program header		Block								^		iew 📄	
비바	N20	Face milling	∇	T=FACE	MILL 63	F=0.1/t V	=120m X0)=-40 Y0=	=-70 20	=5 21=0					i.
事	N30	Face milling	$\nabla\nabla\nabla\nabla$			F=0.08/t		(0=-40 Ye	0=-70 Z	0=5 Z1=0					
\sim -	N40	Contour				ular_Area	a					_	Gr	aphic	
~-	N50	Contour		LEVER_									U	iew	
0.	N60	Mill pocket	∇			=0.15/t V=						_			i.
9-	N70	Mill pocket	⊽⊽⊽₿	T=CUTT	ER 20 F	=0.08/t V=	=150m Z0=	=0 Z1=6in	IC			\Box			
\sim	N80	Contour		LEVER_	Lever_A	rea							Dame	mberina	
~-	N90	Contour		LEVER_	Circle_F	15						_	Renu	mbering	
~-	N100	Contour		LEVER_	Circle_F	15_A									ä
$\sim -$	N110	Contour		LEVER_	Circle_F	R5_B									
O-	N120	Mill pocket	∇	T=CUTT	ER 20 F	=0.15/t V=	=120m Z0=	=0 Z1=3in	IC				Open	further	
Q-			⊽⊽⊽₿	T=CUTT	ER 20 F	=0.08/t V=	=150m 20=	=0 Z1=3in	IC				pro	gram	
Bar-	N140	Drilling		T=PRED	RILL 30	F=0.1/rev	v V=120m	Z1=-21						_	ä
1	N150	001: Positions		Z0=-6 X	(0=70 Ye)=-40						-			
Т	N160	T=CUTTER 20 V=120m													
	N170	RAPID G40 X82 Y-40 Z-5										_			
ą		F=0.1/t I70 J-40 P3 Z-23													ä
a e -	N190	Boring		T=DRILL	Tool F	=0.08/rev	S=500re	v Z1=15in	C						
a e-	N200	Thread milling	∇	T=THRE	AD CUT	FER F=0.0	8/t V=150	m Z1=-2	3inc ø=	40 P2mm	n/rev H1=	1			
-B-	N210	Repeat position		001: Po:	sitionen								Se	ttings	
END		End of program													ä
														ose	
														lose	
															i.
														. =	
												~			
												>			i
		dit 7 Drilling	a		77	Cont.			NC	Vari-		Simu-	NC	Ex-	f
) E	dit 🛃 Drilling 🚪	I I	Milling	<u>_</u>	mill.			NC	ous	5	lation	-	ecute	
								_							f

Figure 2-1 Machining step in the machining plan



3.2 You save programming time:

 ShopMill assists you even when entering the technological values: You only have to enter the handbook values feed/tooth and cutting rate – speed and feedrate are calculated by ShopMill automatically.

Rectar	ngular pocket		Recta	Rectangular pocket				
Input		Complete	Input		Comple			
Г	CUTTER 10	D 1	Т	CUTTER 10				
F	0.150	mm/tooth	F	2292.000	mm/min			
V	120.000	m/min	S	3820	rpm			
Ref. p	oint	•	Ref. p	oint	•			
Machi	ning	∇	Machi	ning	∇			

 ShopMill 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.



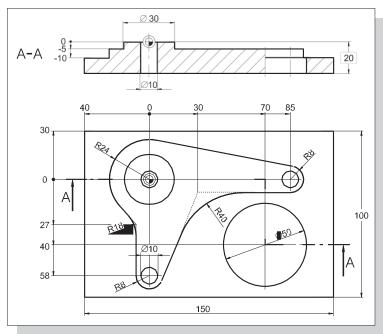
 All machining steps are represented by ShopMill in a compact and clear fashion in the graphic machining plan. This provides you a complete overview and thus better editing possibilities even for comprehensive machining sequences.

NC/UKS/EXAMPLE3/MOLD_PLATE 21	Select
P N10 Program header Block	tool
∼ N20 Contour MOLD_PLATE_OUTSIDE	
7% - N30 Path milling	
™ J N40 Path milling ∀∀∀ T=CUTTER 32 F=0.08/t V=150m Z0=0 Z1=10inc	Build
∼¬N50 Contour MOLD_PLATE_INSIDE	group
○ N60 Mill pocket ▼ T=CUTTER 20 F=0.15/t V=120m Z0=0 Z1=15inc	
N70 Mill pocket	
Image: Will pocket \not \not \not \not \not \no	Search
○ N90 Circular pocket	Search
N100 Circular pocket マママ T=CUTTER 20 F=0.1/t V=150m X0=0 Y0=0 Z0=0 Z1=-10	
○ N110 Circular pocket	
N110 Circular pocket ▼ T=CUTTER 20 F=0.15/t U=120m X0=0 Y0=0 Z0=-10 Z1=-20 N120 Circular pocket ▼ T=CUTTER 20 F=0.1/t U=150m X0=0 Y0=0 Z0=-10 Z1=-20 N130 Centering T=CUTTER Z0 F=0.1/t U=150m X0=0 Y0=0 Z0=-10 Z1=-20 N130 Centering T=CENTERDRILL 12 F=150/min S=500 eror Ø11 N140 Drilling T=DRILL 10 F=150/min V=35m Z1=20inc	Mark
N130 Centering T=CENTERDRILL 12 F=150/min S=500rev Ø11	Flark
- N150 001: Posit. row Z0=-10 X0=-42.5 Y0=-92.5 N=4 α0=90	
P N160 002: Obstacle Z=1	0
- N170 003: Posit. row Z0=-10 X0=42.5 Y0=-92.5 N=4 α0=90	Сору
PJ N180 007: Obstacle Z=1	
€ N190 004: Posit. circle Z0=-10 X0=0 Y0=0 R=22.5 N=6	
P1 N200 005: Obstacle Z=1	
N210 006: Positions 20=-10 X0=0 Y0=42.5	Paste
END End of program	
	Cut
	=
	≣►
×	
Edit Z Drilling Z Milling Cont. K Vari- Jusi Jation	NC Ex-
E Drilling 2 Philling mill.	=> ecute

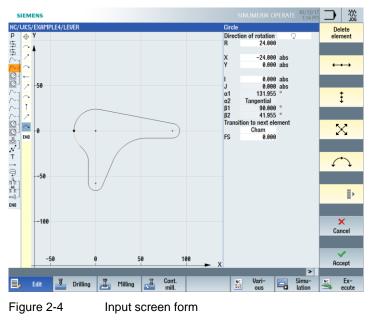
 In drilling, for example, several machining operations can be linked so that they need not be called repeatedly.

И	N1120	Centering	T=CENTERDRILL 12 F=150/min S=500rev Ø11
79-77	N140	Drilling	T=DRILL 10 F=150/min V=35m Z1=20inc
1-	N150	001: Posit. row	Z0=-10 X0=-42.5 Y0=-92.5 N=4 α0=90
)))	N160	002: Obstacle	Z=1
/-	N170	003: Posit. row	Z0=-10 X0=42.5 Y0=-92.5 N=4 α0=90
海-	N180	007: Obstacle	Z=1
O -	N190	004: Posit. circle	20=-10 X0=0 Y0=0 R=22.5 N=6
))- (県)	N200	005: Obstacle	Z=1
\mathcal{N}^{\perp}	N210	006: Positions	20=-10 X0=0 Y0=42.5
END		End of program	

 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.

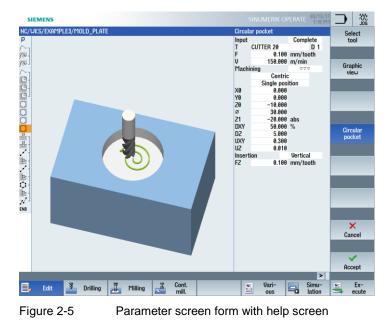






• You may switch between the graphic view and parameter screen form with help screen at any time.

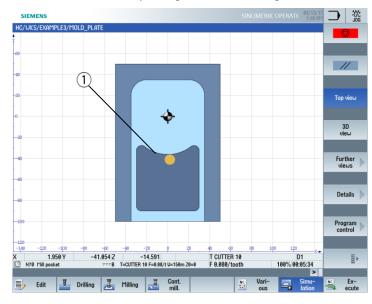




• Creating a machining plan and machining are not mutually exclusive. With ShopMill, you can create a new machining plan in parallel with machining.

3.3 You save machining time:

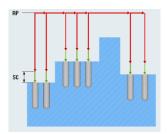
• You need not take into account the pocket radii when selecting the milling cutter for machining contour pockets: Any residual material ① is detected and removed automatically using a smaller milling cutter.



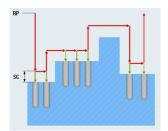
• There are no unnecessary infeed motions between the retraction plane and machining plane when positioning the tool. This is made possible by the settings "Retract to retraction plane "(RP) and "Optimized retraction".

The "Optimized retraction" setting is to be made by a skilled worker in the program header. The worker must take into account obstacles, such as clamping elements.

Retraction to retraction plane (RP)



Retraction to retraction planes = machining time saving



• 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 Xample3/mold_plate	_	SINUMERIK OPERATE 03/13/17 1:19 PT	
	Program header		21 Block	
	Contour		MOLD_PLATE_OUTSIDE	1001
	Path milling	⊽	T=CUTTER 32 F=0.15/t V=120m 20=0 21=10inc	
	Path milling	$\nabla \nabla \nabla$	T=CUTTER 32 F=0.08/t V=150m Z0=0 Z1=10inc	Build
n N50	Contour		MOLD PLATE INSIDE	group
- N60	Mill pocket	∇	T=CUTTER 20 F=0.15/t V=120m 20=0 Z1=15inc	graap
	Mill pocket	⊽⊽⊽₿	T=CUTTER 10 F=0.08/t V=150m 20=0 21=15inc	
N80	Mill pocket		T=CUTTER 10 F=0.08/t V=150m 20=0 21=15inc	
N90	Circular pocket		T=CUTTER 20 F=0.15/t V=120m X0=0 Y0=0 Z0=0 Z1=-10	Search
N100	Circular pocket	$\nabla \nabla \nabla$	T=CUTTER 20 F=0.1/t V=150m X0=0 Y0=0 20=0 21=-10	
N110	Circular pocket	∇	T=CUTTER 20 F=0.15/t V=120m X0=0 Y0=0 Z0=-10 Z1=-20	
N120	Centering		T=CENTERDRILL 12 F=150/min S=500rev Ø11	
	Drilling		T=DRILL 10 F=150/min V=35m Z1=20inc	Mark
- N140	001: Posit. row		20=-10 X0=-42.5 Y0=-92.5 N=4 α0=90	_
- N150	002: Obstacle		Z=1	
- N160	003: Posit. row		20=-10 X0=42.5 Y0=-92.5 N=4 α0=90	_
- N170	007: Obstacle		Z=1	Сору
- N180	004: Posit. circle		20=-10 X0=0 Y0=0 R=22.5 N=6	_
- N190	005: Obstacle		Z=1	
N200	006: Positions		20=-10 X0=0 Y0=42.5	
N210	Circular pocket	222	T=CUTTER 20 F=0.1/t V=150m X0=0 Y0=0 20=-10 21=-20	Paste
	End of program			_
				0.4
				Cut
				=.
			_	≣⊦
			>	
				NC Ex-
a E	dit 🛛 🖉 Drilling	J. I	Milling Cont. Vari- Simu-	Ex-

Figure 2-6 Original machining sequence

S	IEME	NS			E 03/13/17 1:20 PM	\supset	
		XAMPLE3/MOLD_PLATE			12		
		Program header		Block	^		
		Contour		MOLD_PLATE_OUTSIDE			
		Path milling	∇	T=CUTTER 32 F=0.15/t V=120m 20=0 21=10inc			
36 J		Path milling	$\Delta \Delta \Delta$	T=CUTTER 32 F=0.08/t V=150m 20=0 21=10inc			uild
21		Contour	_	MOLD_PLATE_INSIDE		gr	oup
20		Mill pocket		T=CUTTER 20 F=0.15/t V=120m Z0=0 Z1=15inc	_		
3		Mill pocket Mill pocket		T=CUTTER 10 F=0.08/t V=150m 20=0 21=15inc T=CUTTER 10 F=0.08/t V=150m 20=0 21=15inc	_	_	
1		Circular pocket		T=CUTTER 10 F=0.06/t V=150m 20=0 21=15inc T=CUTTER 20 F=0.15/t V=120m X0=0 Y0=0 20=0 21=-10		Sea	arch
1		Circular pocket	222	T=CUTTER 20 F=0.1/t V=150m X0=0 Y0=0 20=0 21=-10			
1		Circular pocket	~	T=CUTTER 20 F=0.15/t V=120m X0=0 Y0=0 20=-10 21=-20			
1		Circular pocket		T=CUTTER 20 F=0.1/t V=150m X0=0 Y0=0 20=-10 21=-20	Ð		_
1		Centering		T=CENTERDRILL 12 F=150/min S=500rev Ø11		M	ark
		Drilling		T=DRILL 10 F=150/min V=35m Z1=20inc			_
7		001: Posit, row		20=-10 X0=-42.5 Y0=-92.5 N=4 α0=90			
at -	N160	002: Obstacle		2=1			
	N170	003: Posit. row		20=-10 X0=42.5 Y0=-92.5 N=4 α0=90		Co	ppy
2j -	N180	007: Obstacle		2=1			
	N190	004: Posit. circle		20=-10 X0=0 Y0=0 R=22.5 N=6			
3j -		005: Obstacle		2=1		De	- 1 -
/	N210	006: Positions		20=-10 X0=0 Y0=42.5		Pa	ste
4D		End of program					
						6	ut
						Ŭ	
							≣⊦
					~		-
			-	P Cant	>		En
		dit 🖉 Drilling	I.	illing Cont. No Vari- mill.	Simu- lation	NC	Ex-

Figure 2-7 Optimized machining sequence with cutting and pasting the machining step

• With ShopMill, 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 ShopMill with the help of examples.

4.1 The operation of ShopMill

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 consist of three parts: the flat operator panel (1), the CNC full keyboard (2) and the machine control panel (MCP) (3).



The most important keys of the CNC full keyboard for navigation in ShopMill are listed in the following:

Кеу	Function
(i) 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 4 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 a 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 ShopMill 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 of ShopMill 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

M	ŢŌ	⊃	G	\bigtriangleup	عر
Machine	Parameter	Program	Program manager	Diag- nostics	Setup

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.

SIE	MENS												03/13/17 1:33 PM	Μ	200 J00
NC/UKS/EXAMPLE3/MOLD_PLATE											Select				
// Res	// Reset MRD													to	
Work			Pos	ition (m	m]			T,F	S						
X			2	3.00	A			Т						Sel	
Ŷ			_		-			· ·						Sei work	
				1.00											
Z			- 25	0.00				-							
SP	1			0.0	00°			F		0.000					
										0.000	mm	/min	100%		
								S	1	0			Ø		
								Ma	ster	0			100%	_	_
⊡•G54								0			50		100.		
T,S,M															
1		CUTTER	R 16		D 1 S1	r 1								_	
	Spindle Spindle M fu			Q	1200 rg	m									
	pinale 11 tu)ther M fun		_	.5	-										
	Jork offset	ction													
	1eas. un.														
1	1achining pl	lane													
														<	<
														Ba	
													>		
₽	T,S,M	9 20	Set	3	Meas.	Ī	Meas.	ĪŢ	Posi-			-	Face		
		2	WO		workp.		tool		tion				mill.		

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

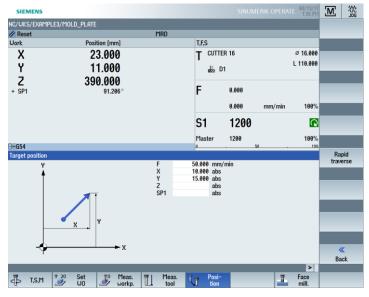


Figure 3-2 Specification of a target position

Machine - Auto

Machine

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.

SIEME	NS							TE 03/13/17 1:59 PM	М	→ AUTO
NC/WKS/E	XAMPLE3/MOLD_	Plate								G
// Reset				MRD					func	ctions
Work		Position (mm]		T,F,S					
Х		23.000			T CUIT	ER 16		Ø 16.000	_	
					1	2.11.10		L 110.000		iliary
Y		11.000				D1		L 110.000	func	tions
ż					1000					
		390.000			-					
+ SP1		91.20	6°		F	0.000				asic ocks
									DIC	JCKS
						0.000	mm/min	100%		
					C1	1000			Tim	ne /
					S1	1200		\mathbf{Q}		ne / inter
					Master	1200		100%	000	
+G54					D		50 .	100%		
Contract of the second s	XAMPLE3/MOLD	PL OTE			-		· ·		Pro	aram
P N10	Program header	FLAIL	Block					A		vels
	Contour		MOLD PLATE							
	Path milling	~			=120m Z0=0 Z1	=18inc				
	Path milling	~~~~			=150m 20=0 21					
,	Contour		MOLD PLATE		1301120-021	- Tome		-		
	Mill pocket	~			=120m 20=0 21	=15inc				
9999922	Mill pocket	⊽⊽⊽₿			=150m 20=0 21					
	Mill pocket				=150m 20=0 21					values
O N90	Circular pocket	~			=120m X0=0 Y0		10		Mad	chine
O N100	Circular pocket	~~~			150m X0=0 Y0=					
	Circular pocket	~			=120m X0=0 Y0					
O N120	Circular pocket	~~~	T=CUTTER 20	F=0.1/t U=1	150m X0=0 Y0=	0 20=-10 21=	-20			≣⊦
N130	Centering		T=CENTERDR	LL 12 F=15	0/min S=500re	w Ø11		~		= '
								>		
	NG OU		NC	Prog.	Block		2	Simult.	1	Prog.
		ore		cntrl	searc			record.		corr.

4.2.2 Parameters

Parameter lists



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

Tool lists

No cutting without tools.

These can be managed in a tool list.

ol li	st									_			MAGA	2111		
.oc.	Туре	Tool name	ST	D	Length	ø			Ψ,	≂ 1	2			^		
Щ		CUTTER 16	1	1	110.000	16.000			Q							
1		CUTTER 4	1	1	65.000	4.000			Q							
2		CUTTER 6	1	1	120.000	6.000			Q							
3		CUTTER 10	1	1	150.000	10.000		4	Q	\checkmark						
4																
5		CUTTER 20	1	1	100.000	20.000			മ							
6		CUTTER 32	1	1	110.000	32.000			Q							
7		CUTTER 60	1	1	110.000	60.000		6	Q	\checkmark						
8		FACEMILL 63	1	1	120.000	63.000			Q							
9		CENTERDRILL 12	1	1	120.000	12.000	90.0		Q					-		
10		DRILL 8.5	1	1	120.000	8.500	118.0		Q							
11		DRILL 10	1	1	120.000	10.000	118.0		Q	\checkmark						
12		PREDRILL 30	1	1	120.000	30.000	180.0		Q							
13		DRILL_Tool	1	1	110.000	25.000			ð	~						
14		THREAD CUTTER	1	1	110.000	20.000		1	P	\checkmark						
15	Ū	THREADCUTTER M10	1	1	130.000	10.000	1.500		ð	\checkmark						
16																
17																
18																_
19														-		
20																
21																
22																
23															Maga	
24															selec	ction
25																
26																
27																=
28																Ð
								_						>		

Figure 3-3 Tool list

Magazine

Tools can be organized in a magazine.

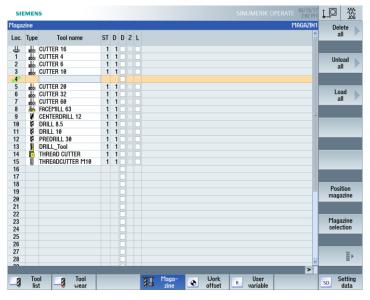


Figure 3-4 Magazine

Work offsets

SIEMENS				SINUMERIK O	PERATE 03/13/17 2:03 PM	lo ∛
Jork offset – Overview	[mm] 公子日 /私	X	Y	Z	SP1	
1achine act value	0 E 56	23.000	11.000	500.000	72.000	
DRF		0.000	0.000	0.000	0.000	
Basic reference		0.000	0.000	0.000	0.000	
Total basic UO		0.000	0.000	0.000	0.000	Active
G54		0.000	0.000	0.000	0.000	
Programmed U0		0.000	0.000	0.000	0.000	
Cucle reference		0.000	0.000	0.000	0.000	
otal UO		0.000	0.000	0.000	0.000	Overview
ool: CUTTER 16		0.000	0.000	110.000		
OFF		0.000	0.000	0.000		
Jork actual value		23.000	11.000	390.000	72.000	
						654 657
						Details
_	_	_	_	_	>	

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

Figure 3-5 Work offsets

4.2.3 Program

Editing programs

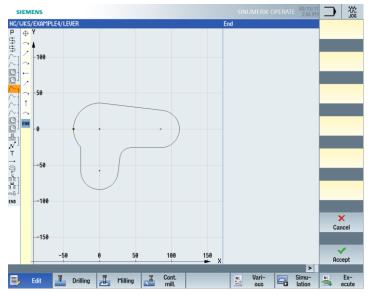


This key can be used to edit programs.

If you have created a ShopMill program in the Program Manager, you can now create the machining plan with the complete machining sequence for the appropriate workpiece. The practical knowledge of the skilled worker is required to create the optimal sequence.

	SIEME	NS		SINUMERIK OPERATE 83/13/1 2:84 Pt	1 =	
NC/	/WKS/E	Example4/lever		8		
Ρ	N10	Program header		Block		
5	N20	Face milling	V	T=FACEMILL 63 F=0.1/t V=120m X0=-40 Y0=-70 20=5 21=0	1	_
바매	N30	Face milling	$\nabla \nabla \nabla$	T=FACEMILL 63 F=0.08/t V=150m X0=-40 Y0=-70 Z0=5 Z1=0		
N.	N40	Contour		LEVER_Rectangular_Area		Build
\sim	N50	Contour		LEVER_Lever		group
O.	N60	Mill pocket	∇	T=CUTTER 20 F=0.15/t V=120m 20=0 21=6inc		
O.	N70	Mill pocket	⊽⊽⊽₿	T=CUTTER 20 F=0.08/t V=150m 20=0 21=6inc		
\sim	N80	Contour		LEVER_Lever_Area		N
\sim	N90	Contour		LEVER_Circle_R15		Search
\sim	N100	Contour		LEVER_Circle_R5_A		
~	N110	Contour		LEVER_Circle_R5_B		
O.	N120	Mill pocket	∇	T=CUTTER 20 F=0.15/t V=120m 20=0 21=3inc		
O.	N130	Mill pocket	VVVB	T=CUTTER 20 F=0.08/t U=150m 20=0 21=3inc		Mark
alle.	N140	Drilling		T=PREDRILL 30 F=0.1/rev V=120m Z1=-21		_
1	N150	001: Positions		20=-6 X0=70 Y0=-40		
т	N160	T=CUTTER 20 V=120m				
	N170	RAPID G40 X82 Y-40 Z-5	i			Сору
ą	N180	F=0.1/t I70 J-40 P3 Z-23	3			
ala	N190	Boring		T=DRILL_Tool F=0.08/rev S=500rev 21=15inc		
a E	N200	Thread milling	∇	T=THREAD CUTTER F=0.08/t V=150m Z1=-23inc Ø=40 P2mm/rev H1=1		
-R-	N210	Repeat position		001: Positions		Paste
END		End of program				
						Cut
						=.
						≣⊦
				>		
		7			NC	Ex-
	> E	dit 💆 Drilling	I.	Milling Cont. Vari- Simu-	NC	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:

Contour	\sim -
Path milling incl. approach and retraction strategies	/%
Circular pocket including technology and position	
Boring technology	
Position for boring	<u>N</u>
Centering technology	ך 🎢
Drilling technology	77.77.
Positions for centering and drilling	۲٦

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 for simulation:

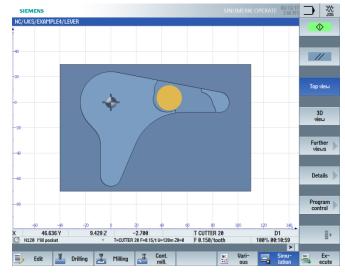


Figure 3-6 Top view

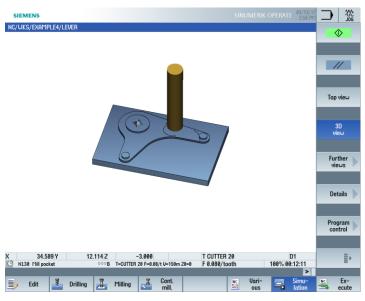


Figure 3-7 3D view

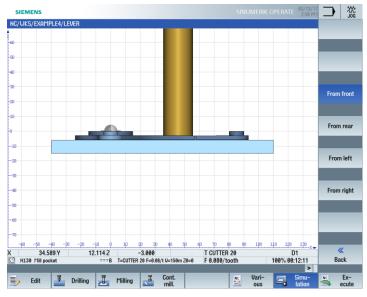


Figure 3-8 Side view

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				JMERIK OPERATE 03/13/13 3:02 PM		AUTO
Name +	Type DIR DIR	Length	Date 03/13/17 01/25/16	Time 1:08:15 PM 3:39:11 PM	Exec	ute
Workpieces EXAMPLE1 ELONGITUDINAL_GUIDE	Dir WPD MPF	1221	03/13/17 03/13/17 11/27/13	3:00:45 PM 3:00:52 PM 3:11:06 PM) Ne	u 🕨
	UPD UPD UPD UPD		01/25/16 03/13/17 03/13/17	3:39:11 PM 1:11:00 PM 12:59:50 PM		
• CRAMPLES • TAC • TEMP	WPD WPD WPD		03/13/17 03/13/17 03/13/17 03/13/17	3:00:40 PM 3:00:52 PM 12:54:04 PM	Ope	en
					Mai	rk
					Cop	y.
					Pas	te
					Cu	t
NC/Workpieces/EXAMPLE1.WPD				Free: 2.5 MB		∎►
NC Local drive USB						

Active programs are marked with a green symbol.



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:

ShopMill

"ShopMill Program"



"G-Code program"

4.2.5 Diagnostics

Alarms and messages



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

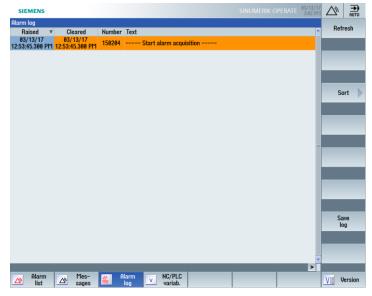


Figure 3-9 Alarm log

5. Basics for beginners

This section will explain the general basics of the geometry and technology for milling. You do not have to enter anything in ShopMill in this section.

5.1 Basics of geometry

5.1.1 Tool axes and machining planes

On universal milling machines, the tool can be mounted parallel to any of the three main axes. These perpendicular axes are aligned to the main guideway of the machine according to DIN 66217 or ISO 841.

The mounting position of the tool yields a corresponding machining plane. Z is the tool axis in most cases.

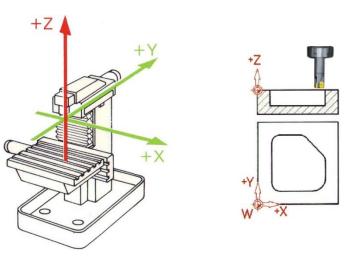


Figure 4-1 Vertical spindle

On modern machines, the tool mounting position is changed without the need for resetting measures and in a few seconds by way of a universal swivel head.

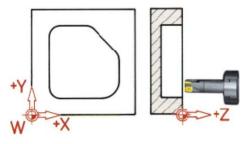


Figure 4-2 Horizontal spindle

If the coordinate system shown on the previous page is rotated accordingly, the axes and their directions in the respective machining plane (DIN 66217) will change.

With the "Various" and "Settings" softkeys, you can call a parameter screen form in which you can specify the machining planes in the program header.



Select the "Various" softkey.



Select the "Settings" softkey.

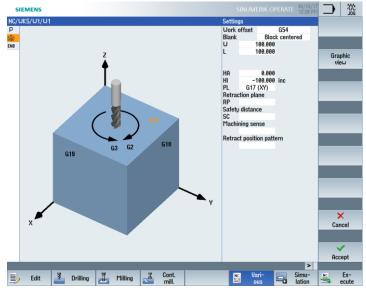
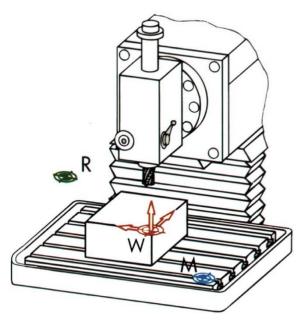


Figure 4-3 Machining planes parameter screen form

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 ShopMill – can orient itself in the existing work area by way of the measuring system.





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.

5.1.3 Absolute and incremental dimensions

Absolute input

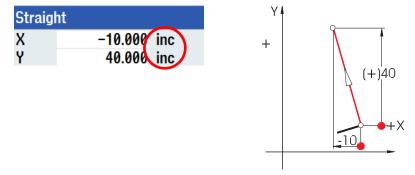
Straig	ht	+Y
X	20.000 abs	50
Y	50.000 abs	20 +X

The entered values are relative to the workpiece zero.

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

Incremental input

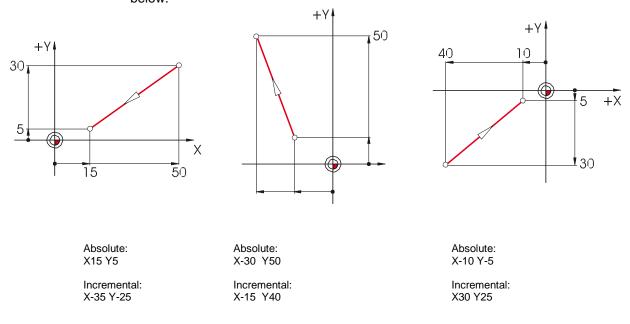
The entered values are relative to the starting point.



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

\bigcirc
SELECT

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



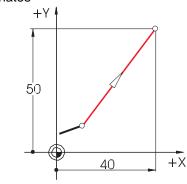
A few examples combining absolute and incremental dimensions can be found below:

5.1.4 Linear motions

Two specifications are required to define an end point unambiguously. These specifications could be:

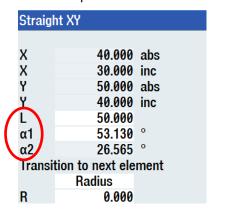
- Cartesian
 - Specification of the X and Y coordinates

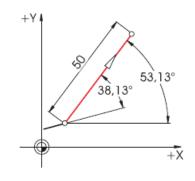
	Straig	ht XY					
1	X 🔪	40.000	abs				
	X	30.000	inc				
	Y /	50.000	abs				
	Y	40.000	inc				
	L	50.000					
	α1	53.130	0				
	α2	38.133	0				
	Transition to next element						
	Radius						



- Polar
 - Specification of the length and an angle

Angle 38.13° = Angle relative to the previous element or angle 53.13° = starting angle relative to the positive X axis

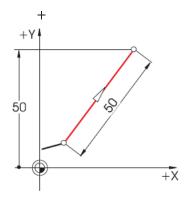




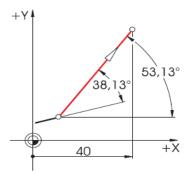
• Cartesian and polar

Cartesian and polar inputs can be combined, e.g.

- Specification of the end point in Y and the length



- Specification of the end point in X and an angle (either 38.13° or 53.13°)



5.1.5 Circular motions

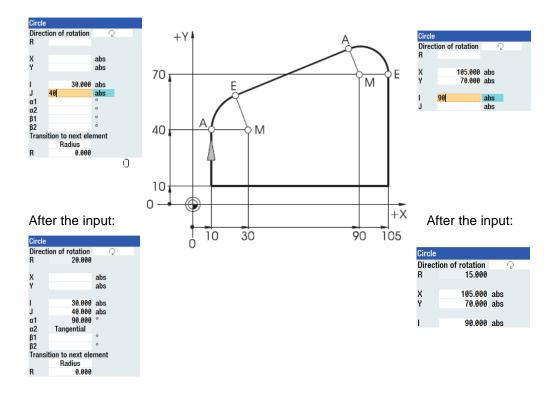
In the case of circular arcs, X and Y specify the end point; the circle center is specified with I and J. In ShopMill, these four values can be entered separately - either as absolute or incremental dimensions.

While X and Y are entered as absolute dimensions, the center point is specified with I and J as an incremental dimension in most controllers. Not only the difference from the starting point A to the center point M must be defined (often in combination with mathematic calculations), but also the direction and thus the sign.

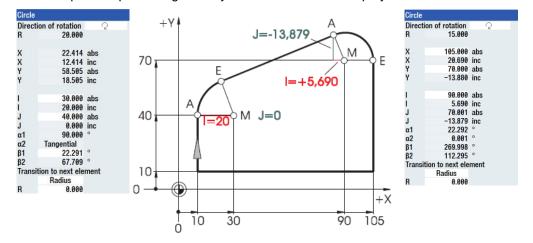
When working with ShopMill, however, you need not perform any calculations thanks to the possibility of entering the center point as an absolute dimension – even the most complicated contour can be defined easily using the graphical contour calculator.

Specification of the center point (absolute)

Values (here: radii) that result from data already entered are calculated by ShopMill automatically.



Display of all parameters



With ShopMill all possible geometry values can also be displayed:

A further advantage of absolute center-point dimensioning:

You need not recalculate the values for I and J when reversing the milling direction.

5.2 Basics of technology

Well-founded knowledge of the tools is a basic requirement for optimum machining, which means cutting materials of the tools, their possible applications and the optimum cutting data are meant. Although tools themselves account for only about 2 - 5 % of the total manufacturing costs of a workpiece, they influence more than 50% of production costs of a component through their performance.

5.2.1 The tools in use

Facing cutter



The facing cutter (also called facing head or milling head) is used to remove large amounts of material.

Shell end mill



The shell end mill is used to create rectangular contour sections with vertical shoulders.

Helical shank mill



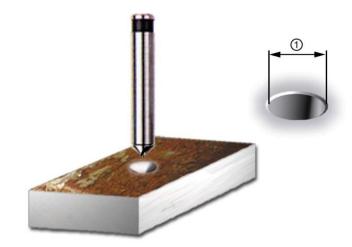
The helical shank mill is a multiple cutting-edge tool which provides especially smooth machining thanks to the spiral arrangement of the cutting edges.

Slotting end mill



The slotting end mill (also called drilling-groove cutter) cuts across the center and can therefore cut into the solid. Mostly, it possesses two or three cutting edges.

NC spotdrill



NC spotdrills are used to center and create a chamfer for the subsequent drilling. ShopMill calculates the depth automatically if you specify the outside diameter of the chamfer (1).

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

Twist drill



With ShopMill, you may select various drilling techniques (swarf milling, deep-hole drilling, etc.). The drill tip is offset automatically in ShopMill. provided that the tip angle of the drill was entered in the tool list.

Solid drill



Solid drills are fitted with indexable inserts and are only available for drill holes with larger diameter. The drilling process must always be performed without interruption.

5.2.2 Cutting rate and speeds

The appropriate optimum speed of a tool depends on the cutting material of the tool and on the material of the workpiece, as well as on the tool diameter. In practice, this speed is often entered immediately without a calculation, even if based on many years of experience. However, it is better to calculate the speed using the cutting rate taken from the relevant tables.

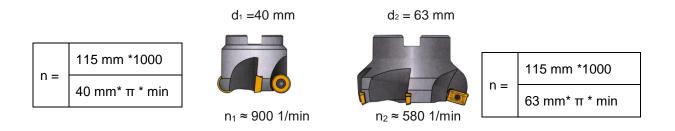
Example - Determination of the cutting rate

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

Material of the tool:	Hard metal
Material of the workpiece:	C45
Determined value:	$v_c = 80 - 150 \text{ m/min}$
The mean value is selected:	vc = 115 m/min

This cutting rate and the known tool diameter are used to calculate the speed n.

The speed for two tools is calculated in the following example:



In NC coding, the speed is specified with the letter S (from 'speed"). Therefore, the inputs are:

Path n	nilling	
Т	CUTTER40	D 1
F	0.150	mm/tooth
S	900	rpm

Note:

ShopMill calculates the spindle speed automatically based on the cutting rate and the tool diameter. This is useful for a cross-comparison, for example:

5.2.3 Feed per tooth and feedrates

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.

The basic value required to calculate the feedrate is the characteristic "feed per tooth". Like the cutting rate, the value for the feed per tooth is also obtained from the handbook, the documents of the tool manufacturer or practical knowledge.

Example - Determination of the feed per tooth

Cutting material of the tool:	Hard metal
Material of the workpiece:	C45
Determined value:	$f_z = 0.1 - 0.2 \text{ mm}$
The mean value is selected:	fz = 0.15 mm

The feedrate v_f is calculated using the feed per tooth, the number of teeth and the known speed.

 $V_f = fz * z * n$

The feedrate for two tools with different number of teeth is calculated in the following, example:

 $d_1 = 63 \text{ mm}, Z_1 = 4$ $d_2 = 63 \text{ mm}, Z_2 = 9$

V_{f1} ≈ 580 1/min * 0.15 mm * 4



V_{f1} ≈ 348 mm/min



V_{f2} ≈ 580 1/min * 0.15 mm * 9

V_{f2} ≈ 783 mm/min

D 1

In NC coding, the feedrate is specified with F (from 'feed'). Therefore, the inputs are:

Path r	milling	
Т	CUTTER63_Z4	D 1
F	340.000	mm/min
S	580	rpm

Note:

ShopMill calculates the feedrate automatically using the feed per tooth and the number of teeth. This is useful for a cross-comparison, for example:

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

ShopMill 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.

oc.	Tuno	Tool name	ST	D	Length	ø		ы	Ĥ	5	ا م	<u>^</u>	Too meas	
	type	Toor name	51	U	Lengui	~		n		1	2			
世 1	Ш	CUTTER 4	1	1	65.000	4.000		3	Q					
2		CUTTER 6	1	1	120.000	6.000			ð		H			
3		CUTTER 10	1	1	150.000	10.000			ົ		H			
4		CUTTER 16	1	1	110.000	16.000			ñ		H			
5		CUTTER 20	1	1	100.000	20.000			2		H			
6		CUTTER 32	1	1	110.000	32,000					n		Edg	
7		CUTTER 60	1	1	110.000	60,000			2		H		Lug	~
8		FACEMILL 63	1	1	120,000	63,000			ñ		n			
9		CENTERDRILL 12	1	1	120.000	12.000	90.0		2		H			
10		DRILL 8.5	1	1	120,000	8,500	118.0		ñ		H			
11		DRILL 10	1	1	120.000	10.000	118.0		2					
12		PREDRILL 30	1	1	120.000	30,000	180.0		2	~				
13		DRILL Tool	1	1	110.000	25.000				~				
14	F	THREAD CUTTER	1	1	110.000	20.000		1	2	~				
15	I	THREADCUTTER M10	1	1	130.000	10.000	1.500		2	~			Unlo	ad
16	J.	CUTTER40	1	1	120.000	40.000		4	2	~				
17	The second	CUTTER63	1	1	120.000	63.000		4	2	~				
18														
19													Dele	
20													100	•
21														
22														
23													Maga	
24													select	tion
25														
26														
														≣⊦

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	Tool length
Ø	Tool diameter
Tip angle or lead	Tip angle or lead
Ν	Number of teeth
#	Direction of spindle rotation
5	Coolants 1 and 2 (e.g. internal and external cooling)

Meaning of the most important parameters in the tool list:

ShopMill provides various tool types (favorites, milling cutters, drills, and special tools). Tools can be created in the tool list by means of a predefined tool catalog. The geometrical parameters (e.g. angle specifications for drills) are different for each tool type.

Tup	Bezeichner	Werkzeuglage
120	- Schaftfräser	
140	 Planfräser 	
200	- Spiralbohrer	8
220	- Zentrierer	Ŵ
240	- Gewindebohrer	Ú
	 3D–Messtaster 	- I I I I I I I I I I I I I I I I I I I
711	 Kantentaster 	÷
110	 Kugelkopf zylindr. 	Ú
111	 Kugelkopf kegelig 	U
121	 Schaftfräser Eckenverr. 	U
	 Kegelstumpffräser 	
	 Kegelstumpffräs. Eck. 	U
157	 Kegeliger Gesenkfräs. 	U

Figure 5-2 Example of Favorites list

6.1.2 Tool wear list

ool w							т		MAGAZIN1	Sor	t
	Туре	Tool name	ST	D	ΔLength	Δø	T C	D	-	_	
Ц,											
1		CUTTER 4	1	1	0.000	0.000					
2		CUTTER 6	1	1	0.000	0.000				Filte	r
3		CUTTER 10	1	1	0.000	0.000				_	_
4		CUTTER 16	1	1	0.000	0.000					
5		CUTTER 20	1	1	0.000	0.000				_	-
6	1	CUTTER 32	1	1	0.000	0.000				Sear	ch
7	-	CUTTER 60	1	1	0.000	0.000					
8		FACEMILL 63	1	1	0.000	0.000					
9	V	CENTERDRILL 12	1	1	0.000	0.000					
10	Ø	DRILL 8.5	1	1	0.000	0.000					
11		DRILL 10	1	1	0.000	0.000					
12	Ň	PREDRILL 30	1	1	0.000	0.000				_	
13	Ň	DRILL Tool	1	1	0.000	0.000					
14		THREAD CUTTER	1	1	0.000	0.000					
15		THREADCUTTER M10	1	1	0.000	0.000				Settin	ngs
16		CUTTER40	1	1	0.000	0.000					_
17		CUTTER63	1		0.000	0.000					
18										_	-
19											
20											
21											
22											
23										Magaz	zin
24										select	
25											
26											
27											
28											
20									×		

The wear data for the respective tools is defined here.

Figure 5-3 Tool wear list

The most important tool wear parameters are:

∆ Length	Length wear
∆ 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	s						SINUMERIK OPERATE 03/15/17 2:01 Pf1	ŢŌ	
1agaz	ine							MAGAZINI		lete
Loc.	Туре	Tool name	ST	D	D	z	L	<u>^</u>		all
Ц										
1		CUTTER 4	1	1					He	load
2		CUTTER 6	1	1						all
3		CUTTER 10	1	1						
4	100	CUTTER 16	1	1						
5		CUTTER 20	1	1						
6		CUTTER 32	1	1					L	oad all
7		CUTTER 60	1	1						all
8		FACEMILL 63	1	1						
9		CENTERDRILL 12	1	1						_
10		DRILL 8.5	1	1						
11		DRILL 10	1	1						
12		PREDRILL 30	1	1						
13		DRILL_Tool	1	1						
14		THREAD CUTTER	1	1						
15	Ū	THREADCUTTER M10	1							
16		CUTTER40	1							_
17		CUTTER63	1	1						
18									De	sition
19										azine
20									may	Jazine
21										
22										_
23									Mag	jazine
24									sel	ection
25										
26										
27										=.
28										∎⊧
20								>		
	То	ol 🙀 Tool								Setting
-8	lis	t 🥑 lool bear						Zine Work User variable	SD	data

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.

8	Tool
-0	list

Select the "Tool list" softkey.

	MEN	IS										SINUMERIK OPERATE 03/15/17 2:06 PM	-	ŝ
ool li:	st											MAGAZINI		
Loc.	Туре	Tool name	ST	D	Length	ø			ĥ	1 1	ا م 2	_		
Ц,														
1		CUTTER 4	1	1	65.000	4.000		3	P	~			Ne	
2													to	
3		CUTTER 10	1		150.000	10.000			Q					
4		CUTTER 16	1	1	110.000	16.000			P					
5		CUTTER 20	1	1	100.000	20.000			Q					
6		CUTTER 32	1	1	110.000	32.000			Q					
7		CUTTER 60	1	1	110.000	60.000			Q					
8		FACEMILL 63	1	1	120.000	63.000		6	Q					
9		CENTERDRILL 12	1	1	120.000	12.000	90.0		Q					
10	Ø	DRILL 8.5	1	1	120.000	8.500	118.0		Q					
11		DRILL 10	1	1	120.000	10.000	118.0		P					
12		PREDRILL 30	1	1	120.000	30.000	180.0		Q					
13		DRILL_Tool	1	1	110.000	25.000			Q					
14		THREAD CUTTER	1	1	110.000	20.000		1	Q					
15		THREADCUTTER M10	1		130.000	10.000	1.500		Q				Lo	ad
16		CUTTER40	1	1	120.000	40.000			P					
17		CUTTER63	1	1	120.000	63.000		- 4	Q	✓				
18														
19														
20														
21														
22														
23													Maga	Izin
24													selec	tio
25														
26														
27														E
28														1
20												>		
-	То	ol 🙀 Tool			7	TT Mag	a-	. 1		Jor	k	User	S	etti
8	lis				1	Mag zin		9		ffs				dat

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



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 cutters with diameter 6, 10, 20 and 32 (Cutter 6, 10, 20 and 32) must be insertable, as they will also be used for the milling of pockets in the following, examples.

6.3 Tools in the magazine

In the following, you will learn how to load the tools into the magazine: Select a tool without location number in the tool list and press the "Load" key.

Load

The following dialog offers the first free magazine location, which you can change or accept as is. For example, the magazine for the following exercises could look like this:

1aga:	zine							MAGAZ	IN1	ŢŌ	
-								T INUNC		L)elete
Loc.	Туре	Tool name	ST	D	D	z	L				an
Ц	the second secon	CUTTER 20	1	1							
1		CUTTER 4	1								nload
2		CUTTER 6	1	1							all
3		CENTERDRILL 12	1	1						_	
4		CUTTER 16	1	1							
√5 *											
6		CUTTER 32	1	1							Load
7		CUTTER 60	1	1							all
8		FACEMILL 63	1	1							
9		CUTTER 10	1	1					1		
10		DRILL 8.5	1	1							
11		DRILL 10	1	1							
12		PREDRILL 30	1	1							
13		DRILL_Tool	1	1							
14		THREAD CUTTER	1	1							
15		THREADCUTTER M10	1	1							
16		CUTTER63_24	1							_	
17		CUTTER63_Z9	1	1							
18		CUTTER63	1	1							osition
19		CUTTER40	1	1							agazine
20										IIIR	igazine
21											
22											
23										Ma	agazine
24										se	lection
25											
26										_	
27											=.
28											≣►
20								>			
	То	ol 🙀 Tool									Setting
8	lis	t lool wear						Maga- Work User zine offset R variable		SD	data

6.4 Measuring tools

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



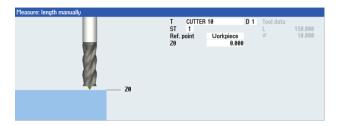
Use the "T,S,M" softkey to insert a tool from the tool list into the spindle.



Then go to the "Meas. tool" menu.



The "Length manual" function is used to measure the tool in the Z-direction.



SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

Diameter manually The "Diameter manual" function is used to measure the diameter of the tool.



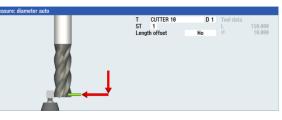
Length	i i
auto	

The "Length auto" function is used to measure the tool in the Z direction using a tool probe.



Diameter
auto

The "Diameter auto" function is used to measure the diameter of the tool using a tool probe.



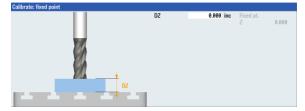


The "Calibrate probe" function is used to determine the position of the probe on the machine table relative to the machine zero.



Calibrate fixed pt.

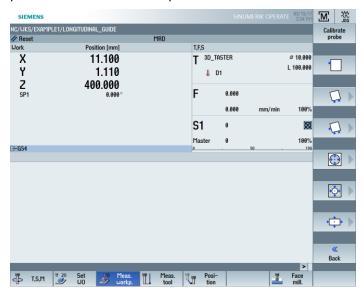
The "Calibrate fixed pt." function is used to determine the fixed point as the reference point for manual measurement of the tool length.



6.5 Setting the workpiece zero

To set the workpiece zero, switch to the Machine - Manual mode in the main menu.

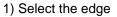
The submenu of the "Measure workpiece" option ("Meas. workp.") offers various possibilities to set the workpiece zero.





The zero point of a workpiece edge is set in the following, example using an edge probe.

SIEMENS				SINU	IMERIK OPERA	TE 03/15/17 2:35 PM	M	₩ J0G
NC/WKS/EXAMPLE1/LONG	itudinal_guide	MRD					Sel work	
Work	Position [mm]		T,F,S					
X Y Z	11.100 1.110 400.000		I -	aster D1		Ø 10.000 L 100.000	Meas	
SP1	400.000 0.000°		F	0.000		100%	Х	
				0.000	mm/min	100%		
			S1	0		Ø	Y	
			Master	0		100%		
Heasure: edge			٥	•	50 .	100.		
rieasure: edge		Work of	fset	G54	Values WO		Z	
		Meas.direct X0		0.000	X Y Z	0.000 0.000 0.000		_
+X					Measured va X0	lues		
								-
			_	_	_	>	Ba	CK
↓ T,S,M 20	Set 30 Meas. W0 Deckp.	Meas.	Posi-		3	Face mill.		





Define the measuring direction left (+) or (-). Parameter X0 can be used to specify an offset of the workpiece zero if this is not to lie on the workpiece edge.

2) Contact the workpiece edge with the probe.



3) The workpiece zero is set taking into account the edge probe diameter (5 mm).

This offset operation must now be repeated for Y with the edge probe and for Z (in most cases, with the milling cutter).

Since the workpieces to be machined are not always cube-shaped or cannot always be clamped at right angles, further offset possibilities are provided:



Example 1: Any corner

If the workpiece is positioned as shown here, the position/corner of the workpiece can be determined by approaching four points.

SIEMENS				SINU	JMERIK OPERA	TE 03/17/17 8:16 AM	100 M
NC/MPF/SWIVEL		MRD					Select work offs.
Work	Position [mm]		T,F,S			TC1 🗖	
X Y Z	1.230 4.560 333.000		T ^{3D_TA} ↓ D			Ø 10.836 L 100.000	Measure. report
A C SP1	0.000° 0.000° 0.000°		F	0.000 0.000	mm/min	100%	P1 saved
			S1	0		×	P2 saved
 G54			Master	0	50 .	100%	
Measure: any co	mer		b			100	P3
		Work o Corner	Any corner offset Outs.co Pos.		Values W0 X Y Z ⊗ີ Measured val		saved P4 saved
	Y PI B PI T a	X0 Y0		0.000 0.000	α β Χθ Υθ	0	Sateu ≪ Back
T,S,M	♥ 20 Set → Meas. → W0 → Workp.	Meas. tool	Posi- tion		- -	Face mill.	🗶 Swi vel

Electronic and mechanical 3D probes are available.

The signals of electronic probes can be processed directly by the controller.

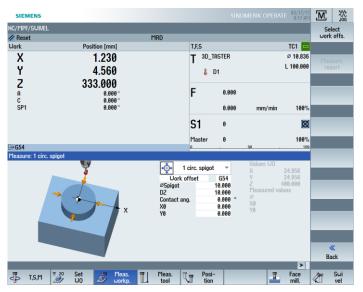




Example 2: Offset of 1 hole



Example 3: Offset of 1 circular spigot



For unrestricted use in educational / R&D institutions. © Siemens AG 2016 All rights reserved. SCE_EN_700_030_ShopMill_4_7_17.docx Calibrate probe

If an electronic 3D probe from the tool magazine is inserted into the spindle, clamping tolerances are involved. This would lead to incorrect results in further measurements. This can be avoided by calibration of the 3D probe at any reference surface or in any reference drill hole using the "Calibrate probe" cycle.



Figure 5-5 Calibrating the probe for the length

SIEMENS				SINU	MERIK OPERAT	E 03/17/17 8:20 AM	М	30G
NC/MPF/SWIVEL								
// Reset		MRD					_	_
Work	Position [mm]		T,F,S			TC1 🗖		
Х	1.230		T 3D_TAST	FER		Ø 10.836		
Y	4.560		- L D1			L 100.000		
ż			• • •					
A C	333.000 0.000 0.000		F	0.000			Len	gth
SP1	0.000 °			0.000	mm/min	100%		
			S1	0		Ø	Diam	eter
				0		100%		
⊞G54			ρ		50 .	100		
Calibrate: probe		ø	20.0	000	Meas, probe d Ø Trigger pts. -X +X -Y +Y ΔX ΔY	10.836	≪ Ba	
	20 Cat 10 Mars 1	Marca	Deci.			>		C- 4
🖞 Т,S,М	20 Set 20 Meas. 2 WO workp.	Meas. tool	Posi- tion			Face mill.	21	Swi vel

Figure 5-6 Calibrating the probe for the radius

7. Example 1: Longitudinal guide

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 and perform a cutter radius compensation
- Enter traversing paths
- Create drill holes and handle position repetitions



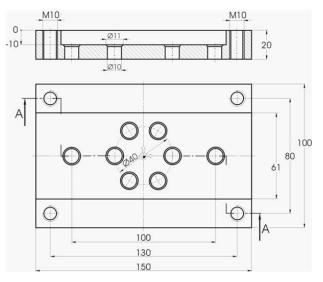


Figure 6-1 Workshop drawing - Example 1



Figure 6-2 Workpiece - Example 1

Note:

ShopMill 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		ahs
Y	-22.500	Target position Z 🔾
Z		abs
F	*Rapid tr.*	mm/min

7.2 Program management and creating programs

Operating sequences

M 38 SIEMEN MRD on (mn T,F,S TC1 X Y Z A C SP1 0.000 All G Т 0.000 0.000 F 0.000 0.00 **S1** ß 100% -G54 Zoom act. val. T,S,M 20 Set W Meas. Meas. Tool Vi tion 7

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

Figure 6-3 Main screen



Open the main menu using the MENU SELECT key. In the main menu, you can open various areas of ShopMill.

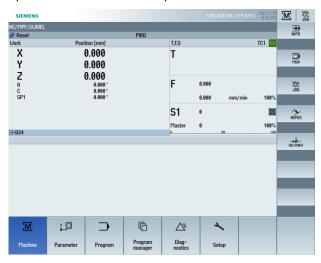


Figure 6-4 Main menu

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.).

				E 8:36 AM	Ē	
Туре	Length	Date	Time			
DIR		01/25/16	3:36:19 PM			
					_	_
DIR		03/15/17	2:40:09 PM	-		
					Ne	su 🌔
		DIR DIR	Type Length Date DIR 01/25/16 01/25/16	Type Length Date Time DIR 01/25/16 3:36:19 PM DIR 01/25/16 3:36:20 PM	DIR 01/25/16 3:36:19 PM DIR 01/25/16 3:36:20 PM	Type Length Date Time DIR 61/25/16 3:36:19 PH DIR 61/25/16 3:36:28 PH DIR 61/25/16 3:36:28 PH DIR 63/15/17 2:48:89 PH

Figure 6-5 Program Manager



Program manager

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



Open the "Workpieces" directory.



Enter the name 'EXAMPLE1' for the workpiece.

	New workpiec	8
-		22
Туре	Workpiece W	PD *
Name	EXAMPLE1	

Figure 6-6 Creating a workpiece

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

Accept	-	ir entry. The follo	owing dialog box opens.
	Type Name Longitudinal_guide	ShopMill	E
	Figure 6-7	Creating a seque	ential program

You can select the input format with the "ShopMill" and "programGuide G code" softkeys.

You specify the program type using the ShopMill softkey.

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



ShopMill

Accept your input.

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

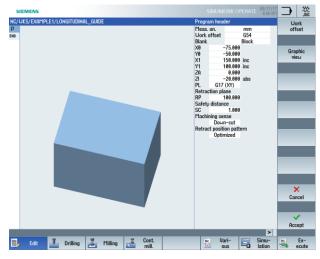


Figure 6-8 Program header – Help screen

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	G54	Х	
Blank	Block	Х	
X0	-75		
YO	-50		Since the workpiece zero is in the center of the workpiece surface, the coordinates of the left workpiece corner have negative values.
X1	150 inc	X (for selection of inc/abs)	
Y1	100 inc	X (for selection of inc/abs)	
ZA	0		
ZI	-20 abs	X (for selection of inc/abs)	
PL	G17 (XY)	Х	
Retraction plane	100		
Safety distance	1		
Machined direction of rotation	Down-cut	Х	
Retract position pattern	Optimized	х	See below Retraction position pattern

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



Figure 6-9 Program header, example 1 – Work 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.

Retract position pattern

The position pattern can be set to "Optimized" (= time-optimized traversing paths) or "To retraction plane".

SC

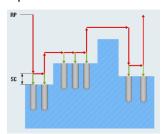
plane and

To retraction plane (standard)

The tool moves back to the retraction

is fed in to the new position.

Optimized retraction



The tool traverses over the workpiece at the safety distance dependent on the contour.

Softkeys



Use the "Graphic view" softkey to change to the online graphic of the workpiece. See figure below.

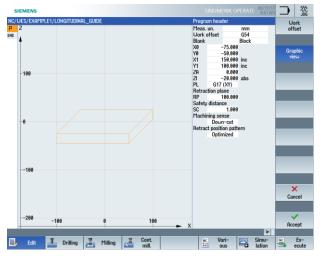


Figure 6-10 Program header – Graphic view

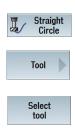


Use the "Graphic view" softkey to change back to the help screen.

7.3 Opening a tool and setting the cutter radius compensation

Operating sequences

Follow the steps below to call the required tool: Expand the horizontal softkey menu with the ETC key.



Select the "Straight Circle" softkey. Select the "Tool" softkey. Open the tool list.

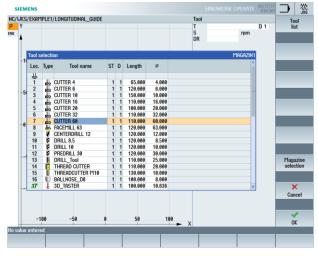


Figure 6-11 Tool list



Use the cursor key to select the "CUTTER60" tool.



Apply the tool to the program. After the tool has been applied, specify the cutting rate 80 m/min (if necessary, change the unit using the toggle key).

SIEMENS			PERATE 8:57 AM		JOG
NC/UKS/EXAMPLE1/LONGITUDINAL_GUIDE	Tool			Se	lect
PY	Т	CUTTER 60	D 1		lool
END	V	80.000	m/min		-
+	DR				
100					

Figure 6-12 Tool - Cutting rate



Accept the entered value.

7.4 Entering the traversing path

Operating sequences



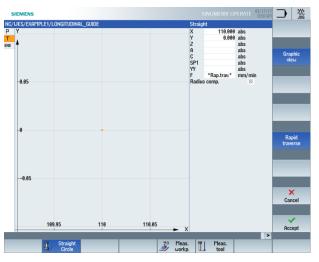
Now enter the traversing paths: Select the "Straight" softkey.



Select the "Rapid traverse" softkey.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
Х	110 abs	Х	
Υ	0 abs	Х	
Radius compensation	Off	х	See below Radius compensation







Accept the entered values.

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA



Select the "Straight" softkey.

Rapid traverse Select the "Rapid traverse" softkey.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
Z	-10 abs	Х	
Radius compensation	Empty field	Х	See below Radius compensation

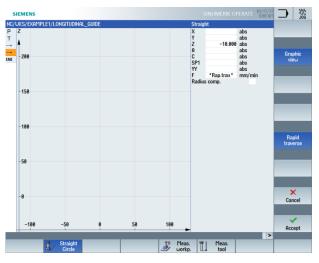


Figure 6-14 Entering the traversing path – Tool positioned in Z

✓ Accept

Accept the entered values.

Straight

Select the "Straight" softkey.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
Х	-110 abs	Х	
F	400 mm/min	Х	
Radius compensation	Empty field	х	See below Radius compensation

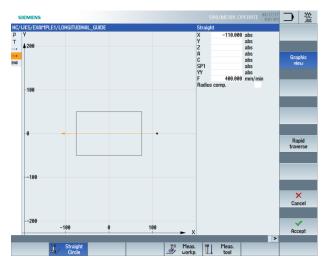
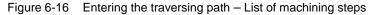


Figure 6-15 Entering the traversing path – First, machining path



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

Т	T=CUTTER 60 V=80m	
→	RAPID G40 X110 Y0	
→	RAPID Z-10	
_	F400/min X-110	—



Tool

Select the "Tool" softkey and perform the following machining steps without help.

Load the next tool "CUTTER16". After the tool has been applied, specify the cutting rate 100 m/min.

Create the traversing path according to the following list of machining steps:

Т	N60	T=CUTTER 16 V=180m
→	N70	RAPID X85 Y22.5
	N80	RAPID Z-10
→	N90	F200/min X-85
→	N100	RAPID Y-22.5
	N110	F200/min X85

Figure 6-17 Entering the traversing path – List of machining steps

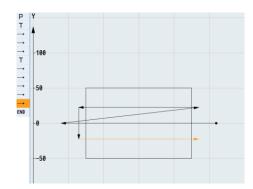


Figure 6-18 Entering the traversing path – Complete



Start the simulation.

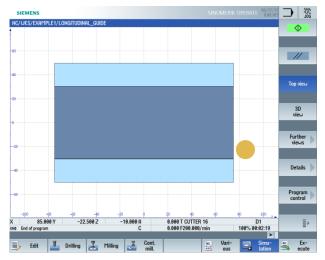


Figure 6-19 Simulation of traversing path

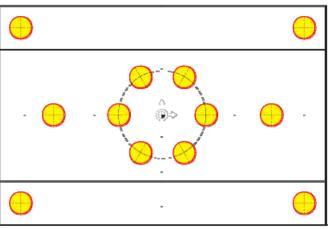
You can end the simulation by selecting either the "Simulation" softkey again or any other horizontal softkey.

Radius compensation

Selection	Result
×	Radius compensation is switched off. The milling cutter traverses with its center point along the created contour.
	The existing compensation setting is maintained.
č	The compensation is performed to the left of the contour in the milling direction.
<u>3</u> 85	The compensation is performed to the right of the contour in the milling direction.

7.5 Creating drill holes and position repetitions

Operating sequences



Now enter the values for the drill holes and position repetitions. In so doing so, you must center, through-drill and tap the 12 drill holes.

Figure 6-20 Drilling positions

Select the "Centering" softkey.

Select the "Drill." softkey.

Z Drilling Centering Select tool

Accept

Open the tool list. Use the cursor key to select the "CENTERDRILL12" tool. Apply the tool to the program. After the tool has been applied, enter the following values:

Field	Value	Selection via toggle key	Notes
F	150 mm/min	Х	
S	500 rpm	Х	
Diameter/tip	Diameter	X	Centering can be entered with reference either to the diameter or to the depth (tip). Since the drill holes have a 0.5 mm chamfer, you may specify a diameter of 11 mm here.

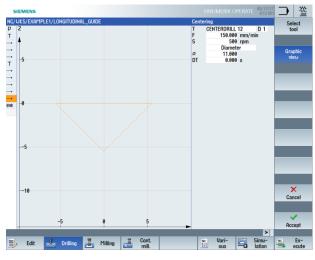


Figure 6-21 Centering



Accept the entered values.

The following steps are used to enter the drilling positions and link these positions with the cutting data.



Select the "Positions" softkey.

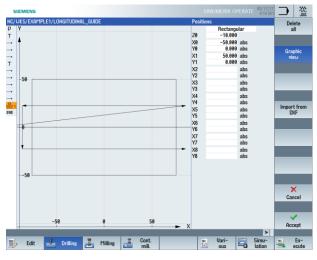


Figure 6-22 Positions – Individual drill holes

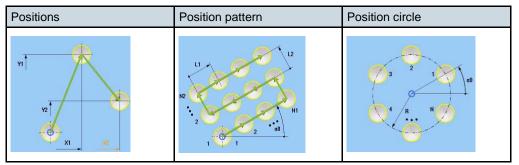
SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

Field	Value	Selection via toggle key	Notes
ZO	-10		The starting depth is -10 mm.
X0	-50		
Y0	0		
X1	50 abs	Х	
Y1	0 abs	Х	

Enter the following values for the two individual drill holes:

Note:

If you deselect the "Graphic view" softkey, detailed help screens are displayed (see table below).



Help screens - Positions



Accept the entered values.



Select the "Positions" softkey.



Select the "Position circle" softkey.



Figure 6-23 Position circle

Enter the following details:

Field	Value	Selection via toggle key	Notes
Pattern	Full circle	Х	
Z0	-10		
X0	0		
Y0	0		
α1	0		
R	20		
Ν	6		
Positioning	Straight	X	Use the "Positioning" field to define how the drill holes are approached within the drill hole pattern. If the drill holes lie in a circumferential slot, for example, do not use "Positioning - Straight" as this would cause a contour violation.

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA



Accept the entered values.



Select the "Positions" softkey.



Select the "Position pattern" softkey.

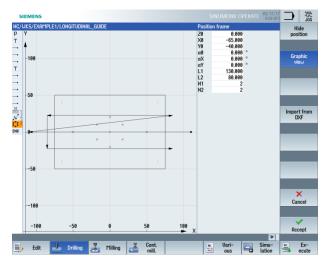


Figure 6-24 Positions - Grid

Enter the following details:

Field	Value	Selection via toggle key	Notes
Pattern	Grid	Х	
Z0	0		
X0	-65		
Y0	-40		
α0	0		
L1	130		
L2	80		
N1	2		
N2	2		



Accept the entered values.



Select the "Drilling Reaming" softkey.

Select tool Open the tool list. Use the cursor key to select the "DRILL8.5" tool.



Apply the tool to the program. After the tool has been applied, enter the following values:

Field	Value	Selection via toggle key	Notes
F	150 mm/min	Х	
V	35 m/min	Х	
Shank/tip	Shank	X	Specify the depth with reference to the shank as an incremental dimension. That is, the 1/3 D drill tip is taken into account automatically.
Z1	20 inc	Х	
DT	0 sec	X	Drilling will be carried out without dwell time.

Note:

The machining steps "Centering", "Drilling" and "Tapping" are linked with each other automatically.

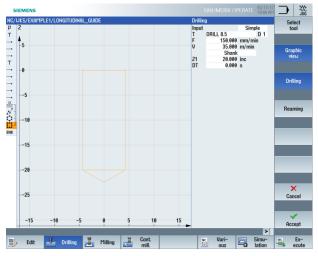


Figure 6-25 Drilling



Accept the entered values.

Thread

Select the "Thread" softkey.

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA



Select the "Tapping" softkey.

Accept

Apply the tool to the program. After the tool has been applied, enter the following values:

Open the tool list. Use the cursor key to select the "THREADCUTTER M10" tool.

Field	Value	Selection via toggle key	Notes
L	1.5 mm/rev	Х	
S	60 rpm	Х	
SR	60 rpm	Х	
Z1	22 inc	Х	The cutting depth must be entered as an incremental dimension.

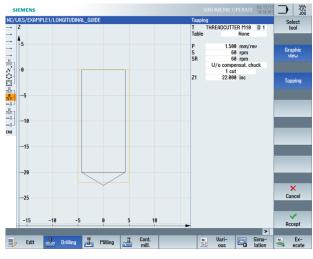


Figure 6-26 Thread



Accept the entered values.

Position repetit.

Select the "Position repetit." softkey.

The drilling positions are numbered consecutively during creation. The respective number stands directly after the block number of the respective position pattern. Enter hole matrix for position 3.

SIEMENS	SINUMERIK OPERATE 03/17/17 10:02 RM
NC/WKS/EXAMPLE1/LONGITUDINAL_GUIDE	Repeat position
PY	Position 3
Т	Position frame
→ IT	

Figure 6-27 Repeating a position



Accept the entered values. Once applied, the values, you can see the linking of the machining steps in the machining step editor.

2 2 2 1	Centering	
N	001: Positions	
0-	002: Posit. circle	
Π	003: Posit. frame	
7/7/7	Drilling	
18 73 12	Tapping	
-û	Repeat position	

Figure 6-28 Linking of machining steps



Select the "Drilling Reaming" softkey.



Open the tool list. Use the cursor key to select the "DRILL10" tool.

Apply the tool to the program. After the tool has been applied, enter the following values:

Field	Value	Selection via toggle key	Notes
F	150 mm/min	Х	
V	35 m/min	Х	
Shank/tip	Shank	Х	
Z1	20 inc	Х	
DT	0	Х	



Figure 6-29 10 mm drill holes



Accept the entered values.

Last, repeat the positions 001 and 002 for the 10 mm drill.

29 29/27 1	Drilling	T=DRILL 10 F=150/min V=35m Z1=20inc
7%77 	Repeat position	001: Positionen
−û j	Repeat position	002: Positionskreis 🖂

Figure 6-30 Repetition of positions 001 and 002 in the machining step editor.

Start the simulation for checking.

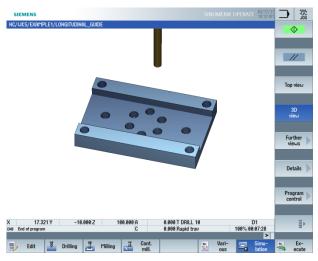


Figure 6-31 3D simulation

8. Example 2: Injection mold

8.1 Overview

Learning objectives

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

- Specify straight lines and circular paths using polar coordinates
- Create rectangular pockets
- Apply circular pockets to position patterns

Task

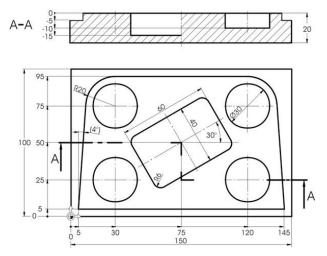


Figure 7-1 Workshop drawing – Example 2



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 "INJECTION_FORM".
- 3. Enter the blank dimensions (for the procedure, see example 1).

Note:

Pay attention to the new zero position!

- 4. Load the 20 mm milling cutter (V 80 m/min).
- 5. Position the tool to the point X-12/ X-12/ Z-5 at rapid traverse.
- 6. Define the starting point of the contour at X5 and Y5. The starting point is approached along a straight line (F 100 mm/min, cutter radius compensation left). After you have entered the traversing blocks, your machining plan should look like this:

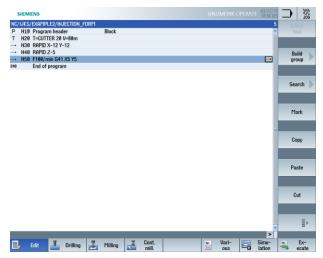


Figure 7-3 Machining step program

8.2 Straight lines and circular paths using polar coordinates

Operating sequences

Before you start entering the contour, observe the following note:

Note:

You can describe the end point of a traversing block not only by way of its X and Y coordinates, but if necessary also via a polar reference point.

X and Y are not known in our example. However, you can determine the point indirectly: It is located 20 mm from the center of the circular pocket, which marks the pole here. The polar angle 176° results from the calculation 180° - 4° (see workshop drawing).

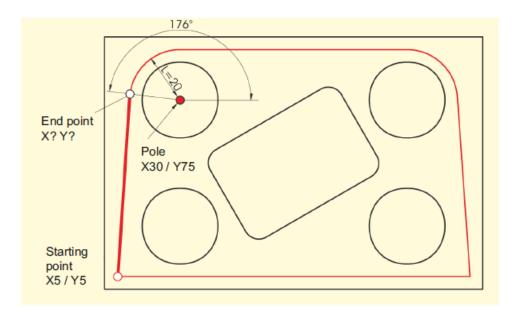


Figure 7-4 Determination of end point and polar angle

Follow the steps below to enter the contour:



Select the "Polar" softkey.

Select the "Pole" softkey.

Field	Value	Selection via toggle key	Notes
Х	30 abs	Х	
Υ	75 abs	Х	



Figure 7-5 Entering the pole



Select the "Straight polar" softkey.

Field	Value	Selection via toggle key	Notes
L	20		The length L specifies the distance of the end point of the straight line from the pole.
α	176		The polar angle specifies how far the length L must be rotated around the pole to reach the end point of the straight line. You may specify the polar angle in either the counterclockwise (176°) or clockwise direction (-184°).

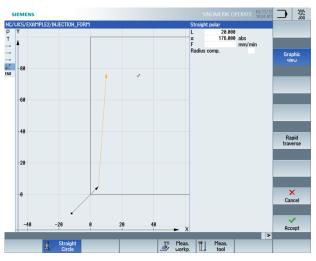


Figure 7-6 Entering the straight line using polar coordinates

Accept

Circle polar

Select the "Circle polar" softkey.

A circular path can also be specified using polar coordinates.

Field	Value	Selection via toggle key	Notes
α	90 abs		Since the pole applies both for the circular path and for the straight line, it need not be entered once more.
			The polar angle is 90° in this case. (See figure below)

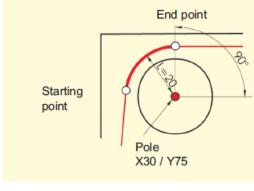


Figure 7-7 Pole starting/end points

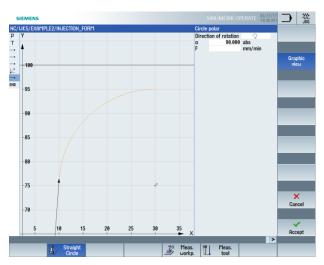


Figure 7-8 Entering the circular path





Select the "Back" softkey.



Select the "Straight" softkey.

Since the end point of the straight line is known unambiguously, you may use the "Straight" function here.

Field	Value	Selection via toggle key	Notes
Х	120	Х	

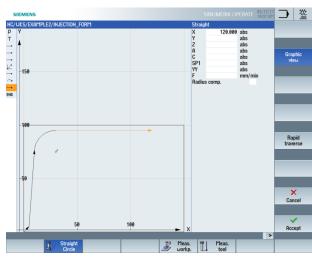


Figure 7-9 Entering the straight line



Select the "Polar" softkey.

Select the "Pole" softkey.

Since the end point of the next circular path is also not known, you must work with polar coordinates again here.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
Х	120 abs	Х	The pole of the circular path is known from the drawing.
Y	75 abs	Х	

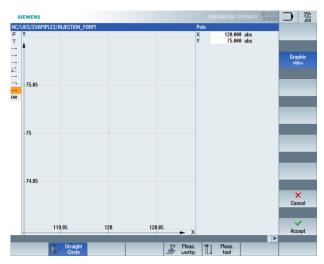


Figure 7-10 Entering the pole for circular path

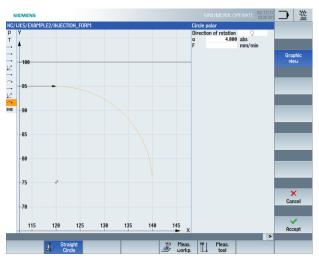


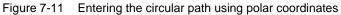
Accept the entered values.



Select the "Circle polar" softkey.

Field	Value	Selection via toggle key	Notes
α	4		The polar angle is also known because of the symmetry.





Select the "Back" softkey.

Select the "Straight" softkey.

The end point of the straight line is known so that you can enter it directly. Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
Х	145 abs		
Y	5 abs		

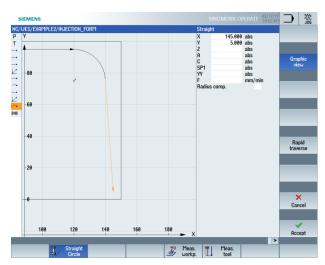


Figure 7-12 Entering the straight line





Select the "Straight" softkey.

The entire contour has been completely milled once with the last straight line.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
Х	-20 abs	Х	

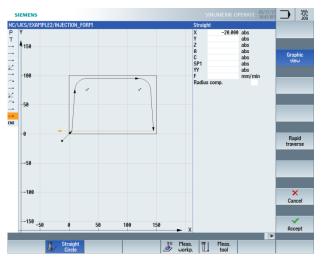


Figure 7-13 Entering the straight line



Accept the entered values.

Straight

Select the "Straight" softkey.

Field	Value	Selection via toggle key	Notes
Х	-12 abs	Х	
Y	-12 abs	Х	
Radius compensation	Off	Х	In the last traversing path, the tool traverses at the entered safety distance, and the radius compensation is switched off for this.

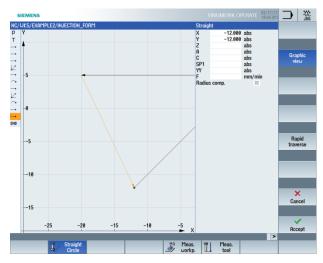


Figure 7-14 Entering the straight line - Safety distance





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

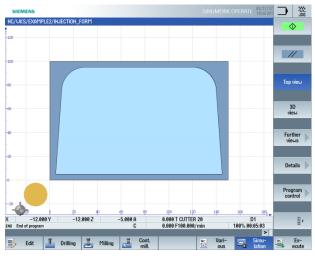


Figure 7-15 Simulation - Top view

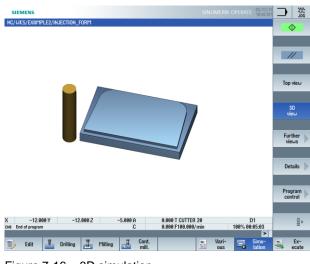


Figure 7-16 3D simulation

8.3 Rectangular pocket

Operating sequences

Follow the steps below to enter the rectangular pocket:

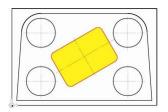


Figure 7-17 Rectangular pocket – Example 2

Milling	Select the "Mill." softkey.
Pocket	Select the "Pocket" softkey.
Rectangle pocket	Select the "Rectang. pocket" softkey.
Select tool	Open the tool list and select "CUTTER10".
Accept	Apply the tool to the program.

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	Х	
V	120 m/min	Х	
Reference point	Center	Х	
Machining	Roughing	X	Ensure that the toggle field is set to "Single position".
X0	75		Specify the
Y0	50		geometrical data for the rectangular
Z0	0		pocket in these fields:
W	40		Position, width,
L	60		length,
R	6		
α0	30		
Z1	-15 abs	Х	
DXY	80%	X	The max. infeed in the plane (DXY) specifies at which width the material is removed. This can be specified either as a percentage of the milling cutter diameter or directly in mm. The maximum infeed in the plane is specified in % here.
DZ	2.5		
UXY	0.3		
UZ	0.3		
Insertion	Helical	X	Select "helical insertion" if not already set.
EP	2 mm/rev	Х	
ER	2		

After the tool has been applied, enter the following values:

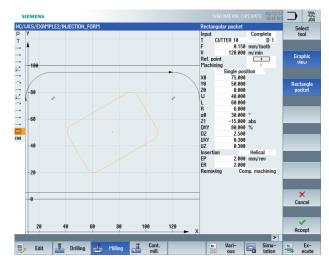


Figure 7-18 Roughing a rectangular pocket

Select the "Pocket" softkey.

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	Х	
V	150 m/min	Х	
Machining	Finishing	X	The wall and base are finished using these settings. Alternatively, you may also only finish the base or chamfer the pocket.



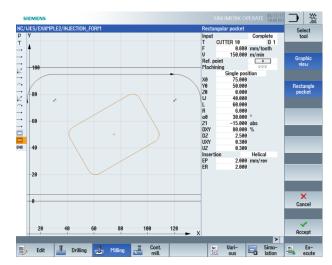


Figure 7-19 Finishing a rectangular pocket



Insertion

Helical insertion	Perpendicular insertion	Oscillating insertion
EP -ER-	F	F

8.4 Circular pockets on a position pattern

Operating sequences

Follow the steps below to enter the circular pockets:

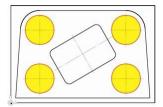


Figure 7-20 Circular pockets – Example 2



Select the "Mill." softkey.

Select the "Pocket" softkey.

Select the "Circular pocket" softkey.

Open the tool list and select "CUTTER10".

Apply the tool to the program.

After the tool has been applied, enter the following values:

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	Х	
V	120 m/min	Х	
Machining	Roughing	Х	
	Position pattern	Х	Similar to drilling, you can also create pockets on a position pattern.
Ø	30	Х	
Z1	-10 abs	Х	
DXY	80%	Х	Specify the maximum infeed in the plane in %.
DZ	5		
UXY	0.3		
UZ	0.3		

Field	Value	Selection via toggle key	Notes
Insertion	Helical	Х	
EP	2 mm/rev	Х	
ER	2		
Removing	Complete machining	Х	

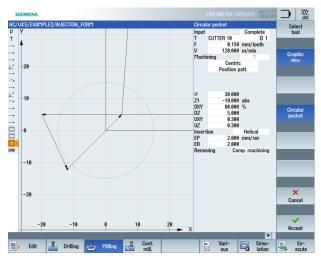


Figure 7-21 Roughing a circular pocket



Accept the entered values.

Select the "Pocket" softkey.



Select the "Circular pocket" softkey.

Enter the following details:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	Х	
V	150 m/min	Х	
Machining	Finishing	Х	

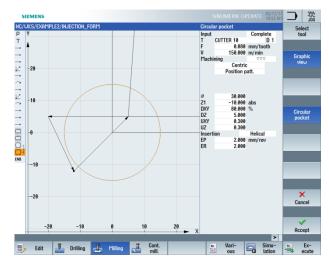


Figure 7-22 Finishing a circular pocket

Select the "Drill." softkey.

Select the "Positions" softkey.

Select the "Position grid" softkey.

Enter the following details:

Field	Value	Selection via toggle key	Notes
Pattern	Grid	X	Position patterns are described in the "Drilling" menu with the "Positions" submenu (independent of the machining method).
X0	30 abs		
Y0	25 abs		
α0	0		
L1	90		
L2	50		
N1	2		
N2	2		



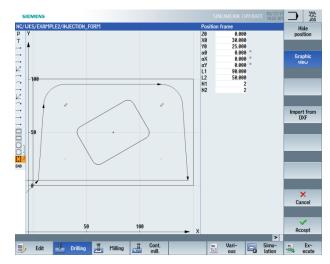


Figure 7-23 Positions of the circular pockets



Start the simulation.

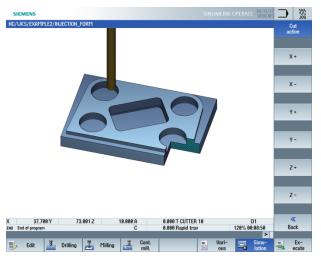


Figure 7-24 Simulation – Cut active

For unrestricted use in educational / R&D institutions. © Siemens AG 2016 All rights reserved. SCE_EN_700_030_ShopMill_4_7_17.docx

9. Example 3: Mold plate

9.1 Overview

Learning objectives

In this section, you will learn the following new functions, in particular the contour calculator. You will learn how to:

- Mill open contours
- Remove contour pockets from the solid, machine residual material and finish
- · Apply machining operations to several planes
- Take obstacles into account

Task

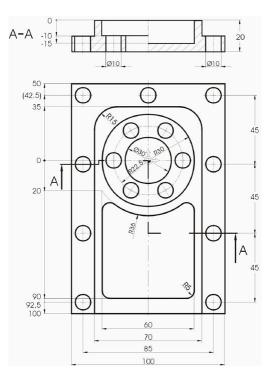


Figure 8-1 Workshop drawing - Example 3

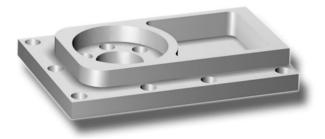


Figure 8-2 Workpiece – Example 3:

Preparations

Perform the following steps on your own:

- 1. Create a new workpiece with the name "Example3".
- 2. Create a new machining plan with the name "MOLD_PLATE".
- 3. Enter the blank dimensions (for the procedure, see example 1).

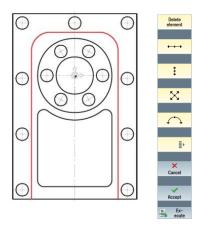
Note:

Pay attention to the new zero position!

9.2 Path milling of open contours

Contour calculator

The integrated ShopMill 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 "Contour milling" softkey.

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

Every contour is assigned its own name to make programs easier to read.



Figure 8-3 Creating the "MOLD_PLATE_Outside" contour



Accept your input.

Enter the following values for the starting point of the contour definition in the screen form:

Field	Value	Selection via toggle key	Notes
X Y	-35 -100		The starting point of the construction is also the starting point of the later machining of the contour.

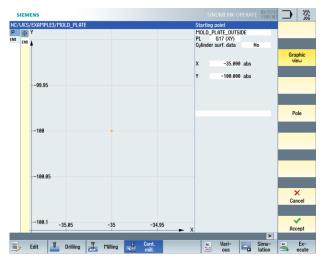


Figure 8-4 Entering the starting point

Note:

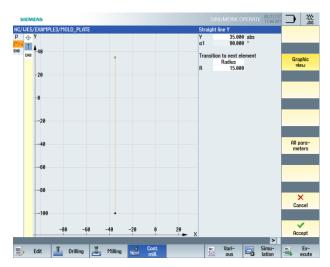
You only describe the workpiece contour here. The approach path and retraction path are not defined until later.

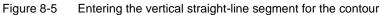


Accept the entered values.

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

Field	Value	Selection via toggle key	Notes
Y	35 abs	X	The first contour element is a vertical straight-line segment with the end point at Y=20. You can specify the subsequent circle contour very easily in this dialog – as a transition element to the next straight line. Therefore, the theoretical end point of the straight line lies at Y=35.
Transition to next element	Radius	Х	
R	15		







Accept the entered values.

←•→

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

Field	Value	Selection via toggle key	Notes
Х	35 abs	Х	
R	15		The radius is specified again as a rounding.

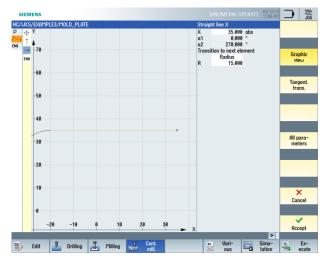


Figure 8-6 Entering the horizontal straight-line segment for the contour



Accept the entered values.

‡

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

Field	Value	Selection via toggle key	Notes
Y	-100 abs	Х	

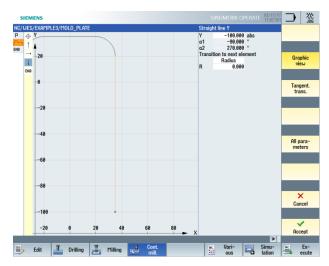


Figure 8-7 Entering the vertical straight-line segment for the contour



Accept the entered contour.

Accept the contour to apply it to your machining plan.

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



Select the "Path milling" softkey.



Open the tool list and select "CUTTER32".



Apply the tool to the program.

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	Х	
V	120 m/min	Х	
Machining	Roughing Forward	X X	With ShopMill V6.4 and higher, you may also mill backwards, opposite the construction direction.
Radius compensation	Left	X	The tool is to traverse to the left of the contour.
Z0	0		
Z1	10 inc	X	Switch the depth Z1 to "inc". The advantage of this is that in all cases only the actual depth of the pocket can be entered without a sign. This makes input easier for you, in particular with nested pockets.
DZ	5		
UZ	0.3		
UXY	0.3		
Approach	Straight	X	The approach can be in a quarter circle or semicircle, perpendicular or on a straight line. Here, it is appropriate to approach the contour tangentially on a straight line.
L1	5		For approach length L1, you do not have to take the cutter radius into account. It is offset automatically by ShopMill.
FZ	0.1 mm/tooth	Х	
Retract	Straight	Х	
L2	5		
Lift mode	To retraction plane	Х	

Enter the following values for roughing in the screen form:

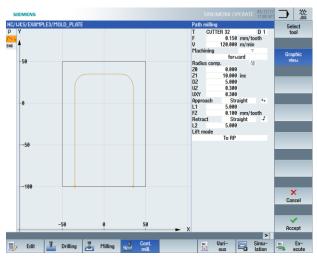


Figure 8-8 Roughing the contour



Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	Х	
V	150 m/min	Х	
Machining	Finishing		

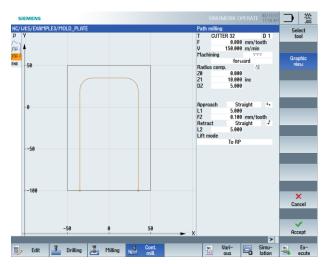


Figure 8-9 Finishing the contour



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

NC/	UKS/	EXAMPLE3/MOLD_PLATE			
Ρ	N10	Program header		Block	
		Contour		MOLD_PLATE_OUTSIDE	
<i>184</i> -	N30	Path milling	∇	T=CUTTER 32 F=0.15/t V=120m 20=0 21=10inc	
<i>78</i> 7	N40	Path milling	$\nabla\nabla\nabla$	T=CUTTER 32 F=0.08/t V=150m 20=0 21=10inc	\rightarrow
END		End of program			

Figure 8-10 Linking of the machining steps in the machining plan



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

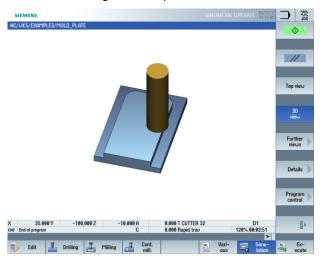


Figure 8-11 Simulation – Outside contour

9.3 Solid machining, residual material and finishing of contour pockets

Operating sequences

Follow the steps below to enter the pocket contour. The pocket is then removed from the solid by machining and finished.

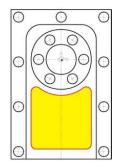


Figure 8-12 Pocket contour



Select the "Cont. mill." softkey.



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



Figure 8-13 Creating the "MOLD_PLATE_Inside" contour



Accept your input.

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

Field	Value	Selection via toggle key	Notes
Х	0 abs		
Y	-90 abs		

SIEMENS	SINUMERIK OPERATE 03/17/17 11:15 RM
NC/WKS/EXAMPLE3/MOLD_PLATE	Starting point
P 🕁 Y	MOLD_PLATE_INSIDE
	PL G17 (XY) Cylinder surf. data No
	X 0.000 abs
-80.05	Y -90.000 abs

Figure 8-14 Entering the starting point



←•→

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

Field	Value	Selection via toggle key	Notes
X	25 abs	Х	For practice, specify the first arc not as a rounding but rather as a separate element. Therefore, construct the straight line only up to X25.

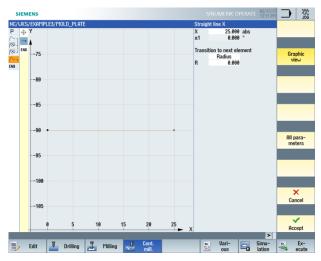


Figure 8-15 Entering the horizontal straight-line segment for the contour

 Accept the entered values.

~	
4	7

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

Field	Value	Selection via toggle key	Notes
Direction of rotation	Left	Х	
R	5		
Х	30 abs	Х	
Υ	-85 abs	Х	

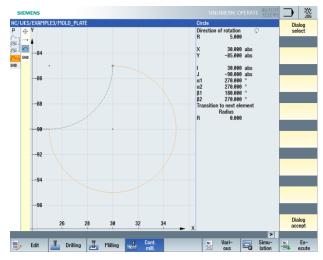


Figure 8-16 Arc for the contour (bottom right)

Dialog	
select	

Two construction solutions result after entering the Y end point. Select the desired solution using the "Dialog select" softkey. The selected solution then turns orange, and the alternative solution is displayed as a black dotted line.



Accept your selection. The geometry processor automatically detects that the programmed arc is connected tangentially to the straight line. The appearance of the "Tangent. trans." softkey changes (i.e. is pressed).

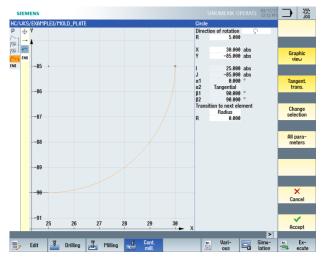


Figure 8-17 Arc for the contour – After selection

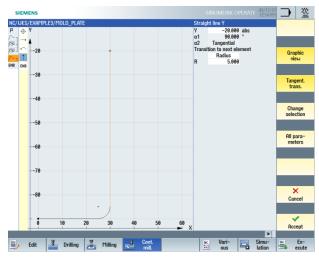


Accept the entered values.

т	
•	
Ť.	

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

Field	Value	Selection via toggle key	Notes
Y Transition to next element	-20 abs Radius 5	X X	The end point of the straight line is known. The transition to R36 is rounded with R5.



Accept the entered values.

Figure 8-18 Entering the vertical straight-line segment for the contour

✓ Accept



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

Field	Value	Selection via toggle key	Notes
Direction of rotation	Right	Х	
R	36		
Х	-30 abs	Х	
Υ	-20 abs	Х	
Transition to next element	Radius 5	Х	

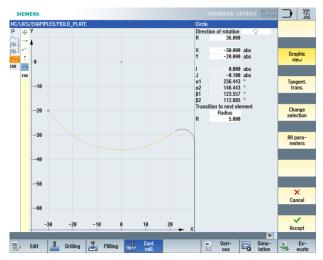


Figure 8-19 Entering the arc for the contour



Accept the entered values.

1

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

Field	Value	Selection via toggle key	Notes
Y	-90 abs	Х	
Transition to next element	Radius 5	Х	Specify the radius R5 as the rounding.

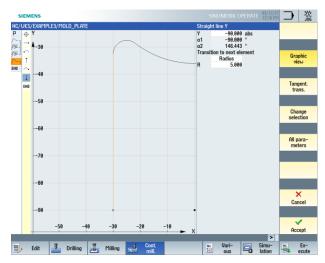


Figure 8-20 Entering the vertical straight-line segment for the contour



Accept the entered values.



Close the contour. The pocket contour is now fully described.

Figure 8-21 Closing the contour



Accept the contour to apply it to your machining plan.



Select the "Pocket" softkey.

Open the tool list and select "CUTTER20".

Apply the tool to the program.

Note:

The machining direction of the pocket has already been defined in the program header. The "Down-cut" setting was selected in this case.

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	Х	
V	120 m/min	Х	
Machining	Roughing	Х	
Z0	0		
Z1	15 inc	X	If you enter the machining depth as an incremental dimension, you must enter a positive value for the depth.
DXY	50%	Х	
DZ	5		
UXY	0.3		
UZ	0.3		
Starting point	automatically	X	If you select the "automatically" setting for the starting point (insertion position), the starting point is specified by ShopMill.
Insertion	Helical	Х	Set insertion to
EP	2 mm/rev	Х	Helical with 2 mm/revolution pitch
ER	2		and 2.00 mm radius.
Lift mode	To retraction plane	х	

Enter the following values for roughing in the screen form:

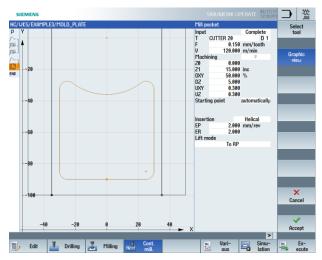


Figure 8-22 Roughing a pocket



Pocket res.mat.



Open the tool list and select "CUTTER10".

Apply the tool to the program.

Enter the following values i	in the	screen form:
------------------------------	--------	--------------

Field	Value	Selection via toggle key	Notes
F	0.1 mm/tooth	Х	
V	120 m/min	Х	
Machining	Roughing	Х	
DXY	50%		The maximum infeed in the plane must be 50%.
DZ	5		

Select the "Pocket res. mat." softkey. Because the 20 mm cutter cannot machine R5 radii, material will remain in the corners. Use the "Pocket reside. mat." function

to rough-machine the areas not yet machined with pinpoint accuracy.

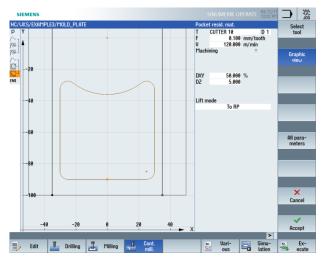


Figure 8-23 Machining residual material of the pocket



Select the "Pocket" softkey.

Open the tool list and select "CUTTER10".

Apply the tool to the program.

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	Х	
V	150 m/min	Х	
Machining	Base	Х	
UXY			The previously entered allowance for roughing must remain set for the values in the "Finishing allowance plane (UXY)" and "Finishing allowance depth (UZ)" fields. This value is important for automatic calculation of the traversing paths.

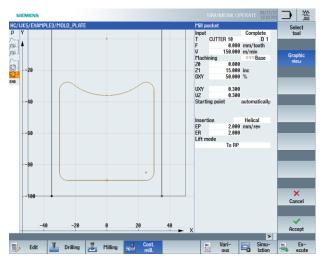


Figure 8-24 Finishing a pocket



Select the "Pocket" softkey.

Enter the following value for removing the residual material from the contour In the screen form:

Field	Value	Selection via toggle key	Notes
Machining	Wall	Х	

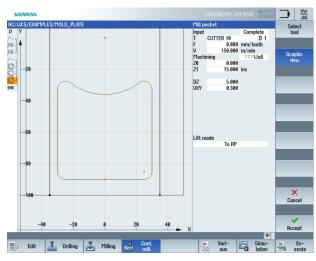


Figure 8-25 Finishing the wall



Accept the entered values.

9.4 Machining on several planes

Operating sequences

Mill the 60 mm circular pocket as in the "INJECTION_FORM" example in two machining steps.

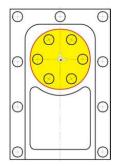
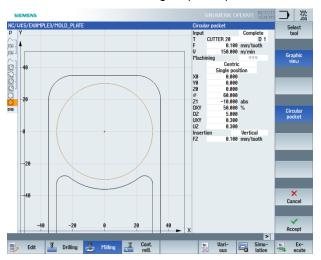


Figure 8-26 Circular pocket

1. In the first machining step, the pocket is machined by roughing with the 20 mm milling cutter to -9.7 mm.



Figure 8-27 Roughing the circular pocket



2. In the second machining step, the pocket is finished using the same tool.

Figure 8-28 Finishing the circular pocket

Use the following steps to specify the machining of the inside circular pocket: The circular pocket is then machined to a depth of -20 mm.

Note:

The starting depth is now no longer at 0 mm, but at -10 mm!

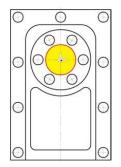


Figure 8-29 Inside circular pocket



Select the "Mill." softkey.

Pocket

Select the "Pocket" softkey.

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA



Enter the following values for machining of the circular pocket in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	Х	
V	120 m/min	Х	
Machining	Roughing	Х	
X0	0		
Y0	0		
Z0	-10		
Ø	30		
Z1	-20 abs	Х	
DXY	50%	Х	
DZ	5		
UXY	0.3		
UZ	0.3		
Insertion	Vertical	Х	
FZ	0.1 mm/tooth	Х	

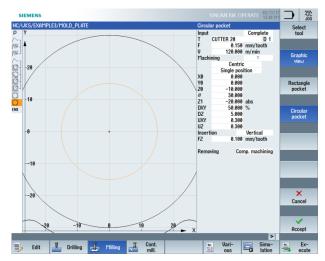


Figure 8-30 Roughing the inside circular pocket



Accept the entered values.



Select the "Mill." softkey.



Select the "Pocket" softkey.

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

Circular pocket Enter the following values for machining of the circular pocket in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	Х	
V	150 m/min	Х	

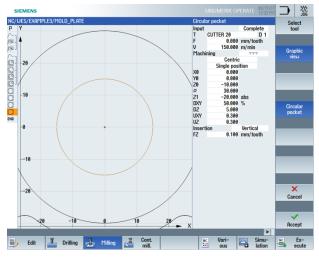


Figure 8-31 Finishing the inside circular pocket



Accept the entered values.



Start the simulation.

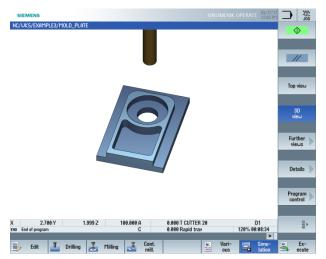


Figure 8-32 Simulation in 3D view

9.5 Taking obstacles into account

Operating sequences

The linking of various drill patterns, as you learned in example 1, can also be used for this workpiece as well. Here, however, it is necessary to bypass one or more obstacles – depending on the sequence of machining. Between the drill holes, the tool traverses to the safety distance or to the machining plane – depending on your setting.

First, create the centering and drilling machining steps as done in example 1.

1. Centering

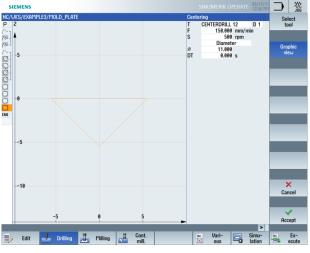


Figure 8-33 Centering machining step



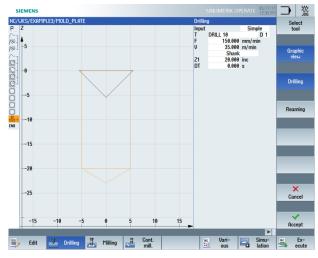


Figure 8-34 Drilling machining step

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

Follow the steps below to enter the associated drilling positions:



Select the "Positions" softkey.



First, create the left line of holes in the sequence from bottom to top.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
Pattern	Line	Х	
Z0	-10		
X0	-42.5		
Y0	-92.5		
α0	90		
LO	0		
L	45		
Ν	4		

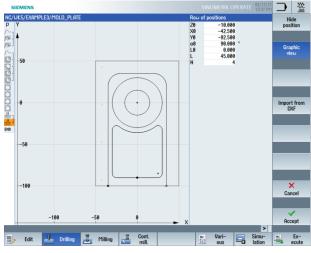


Figure 8-35 Entering the line of holes



Accept the entered values.



Select the "Positions" softkey.

Obstacle

Use the "Obstacle" function to specify a traversing path of 1 mm, since the right line of holes is to be drilled next from bottom to top next for practice purposes. You only have to enter the obstacle if you have switched the "Retraction position pattern" toggle field in the program header to "Optimized" beforehand.

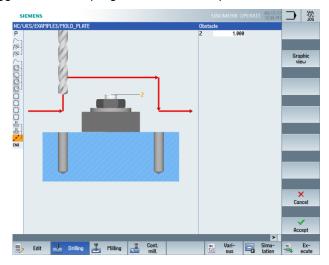


Figure 8-36 Entering the obstacle



Accept the entered values.

Select the "Positions" softkey.

Enter the following values for the second line of holes in the screen form:

Field	Value	Selection via toggle key	Notes
Pattern	Line	Х	
Z0	-10		
X0	42.5		
Y0	-92.5		
α0	90		
LO	0		
L	45		
Ν	4		

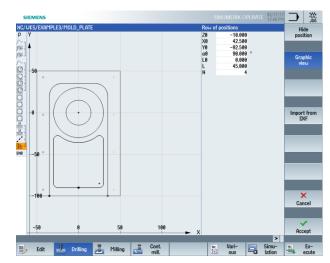


Figure 8-37 Entering the line of holes





Select the "Positions" softkey.



To get to the next drill pattern - the hole circle, another obstacle must be bypassed. Enter Z = 1.



Accept the entered value.

Select the "Positions" softkey.

Enter the following values for the six drill holes in the full circle in the screen form:

Field	Value	Selection via toggle key	Notes
Pattern	Full circle	Х	
Z0	-10		
X0	0		
Y0	0		
α0	0		
R	22.5		
Ν	6		
positioning	Straight	Х	

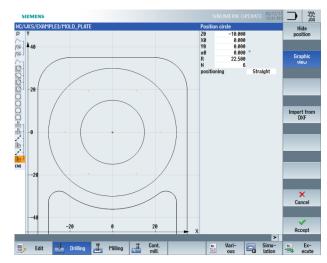


Figure 8-38 Entering the drill holes in the full circle

Select the "Positions" softkey.

To make the last drill hole, another obstacle must be by passed. Enter Z = 1.

Accept the entered value.

Select the "Positions" softkey.

Enter the following values for the last drilling positions in the screen form:

Note:

If necessary, delete any existing positions using the DEL key.

Field	Value	Selection via toggle key	Notes
Pattern	Rectangular	Х	
Z0	-10		
X0	0		
Y0	42.5		





SIEMENS				NUMERIK OF	PERATE 03/17/17 12:42 PM		30G
/WKS/EXAMPLE3/MOLD_PLATE			Position	ns Rectangu	llar		elete
			20	-10.000			- Can
]			X0 Y0	0.000 42.500			
42.55			X1	42.000	abs	Gr	aphic view
			Y1		abs	``	icω
			X2 Y2		abs		
42.55			¥2 X3		abs abs		
			Y3		abs		
			X4 Y4		abs		
			14 X5		abs abs	Impo	ort from
			Y5		abs		DXF
42.5			X6		abs		
42.0			Y6 X7		abs abs		
			¥7		abs		
			X8		abs		
			Y8		abs		
-42.45							
							x
							ancel
							<u></u>
-0.05	0	0.05	► X				ccept
			- 1		>		
🤊 Edit 🎿 Drilling 🍃	🖌 Milling 💡	Cont.	NC	Vari-	Simu-		Ex-

Figure 8-39 Entering the drilling positions



Note:

This programming example is intended to familiarize you with the "Obstacle" function. There are naturally more elegant methods of programming drilling positions, including with only one obstacle.

Try out different strategies and decide which is the best for you.



Start the simulation.



Figure 8-40 Simulation - Top view

10. Example 4: Lever

10.1 Overview

Learning objectives

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

- Perform face milling
- · Create borders (auxiliary pockets) for solid machining around islands
- Create and copy circular islands
- · Work with the machining step editor to machine islands
- Perform deep-hole drilling, helix milling, boring and thread milling
- Program contours using polar coordinates (version 6.4 and higher)

Task

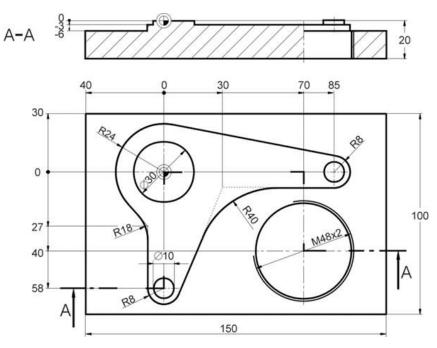


Figure 9-1 Workshop drawing – Example 4



Figure 9-2 Workpiece – Example 4

Preparations

Perform the following steps on your own:

- 1. Create a new workpiece with the name "Example4".
- 2. Create a new machining plan with the name "LEVER".
- 3. Enter the blank dimensions (for the procedure, see example 1).

Note:

Note that the blank is to be 25 mm thick and you must therefore set ZA to 5 mm. After entering the data, the program header should look like this:

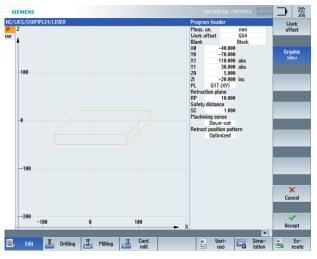


Figure 9-3 Workpiece dimensions in the program header

10.2 Face milling

Operating sequences



Select the "Mill." softkey.

Select the "Face milling" softkey.

Open the tool list and select "FACEMILL63".

Apply the tool to the program.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.1 mm/tooth	Х	
V	120 m/min	Х	
Machining	Roughing	Х	
Direction	Changing	Х	
X0	-40		
Y0	-70		
ZO	5		
X1	110 abs	Х	
Y1	30 abs	Х	
Z1	0 abs	Х	
DXY	30%	Х	
DZ	5		
UZ	1		

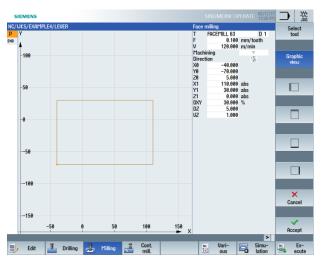


Figure 9-4 Roughing the area



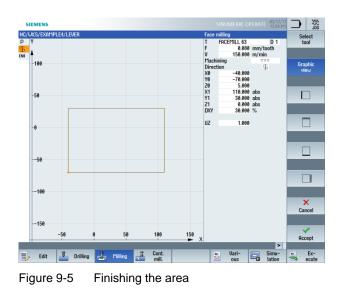
Select the "Face milling" softkey.

Enter the following values for finishing in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	Х	
V	150 m/min	Х	
Machining	Finishing	Х	

Note:

The finishing allowance must have the same value for both roughing and finishing, because it refers to the subsequent finishing machining for the roughing step, and it refers to the thickness of the material still to be removed for the finishing step.





10.3 Creating the border for the lever island

Operating sequences

Note:

Just like pockets, islands are described as a contour in the graphic contour calculator. They only become islands through linking in the machining plan. The first contour in the machining plan always describes the pocket. One or more subsequent contours are interpreted as islands.

Since there is no pocket in the case of the 'LEVER' example, you must put an imagined auxiliary pocket around the external contour. This serves as a necessary outside boundary for the traversing paths and thus forms the frame in which the tool motions take place.



Select the "Cont. mill." softkey.



Create a new contour with the name "LEVER_Rectangular_Area".

	New contour
PI	ease enter the new name
LE	VER_Rectangular_Area

Figure 9-6 Creating the contour

Create the following contour on your own.

Round the corners with R15. Be sure to select the values in such a way that the workpiece corners are covered by the pocket.

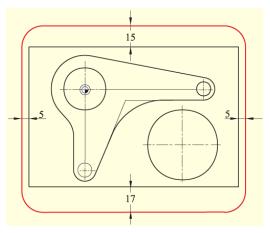
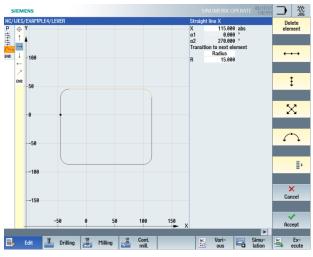


Figure 9-7 Border for the lever island



Compare your contour with the figure below.

Figure 9-8 Completely constructed contour

10.4 Machining the lever

Operating sequences

Follow the steps below to enter the contour:

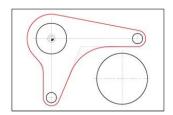


Figure 9-9 Lever contour



Select the "Cont. mill." softkey.



Create a new contour with the name 'LEVER_Lever'.

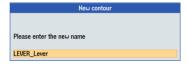


Figure 9-10 Creating the contour

Once applied, enter the following values for the starting point of the contour line in the screen form.

Field	Value	Selection via toggle key	Notes
Х	-24 abs		
Y	0 abs		

	/Example4/lever	Starting point	
END	Y ▲	LEVER_LEVER PL G17 (XY) Cylinder surf. data No	
0	-0.06	X -24.000 abs	Graphic view
	-0.04	Y 0.000 abs	
	-6.82		Pole
	-8	_	
	0.02	_	
	0.04	_	×
	0.06	_	Cancel
	-24.04 -24.02 -24 -23.98 -23.96	- x	Accept

Figure 9-11 Creating the starting point





Enter the following values for the first arc in the screen form:

Field	Value	Selection via toggle key	Notes
Direction of rotation	Clockwise	х	
R	24		Radius and center
1	0		point are known.

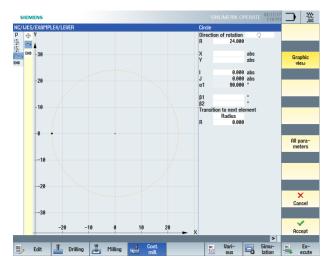


Figure 9-12 Arc for the contour



Accept the entered values.



Create the inclined straight line tangential to the preceding element.



Select the "Tangent trans." softkey.

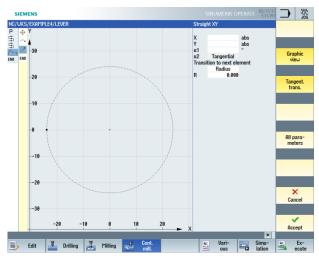


Figure 9-13 Inclined straight line for the contour



Accept your input.



Enter the tangential circular arc.

Tangent. trans.

Select the "Tangent trans." softkey.

Field	Value	Selection via toggle key	Notes
Direction of rotation	right	Х	
R	8		Radius, center point and end point are known.
Х	85 abs	Х	
Y	-8 abs	Х	
1	85 abs	Х	

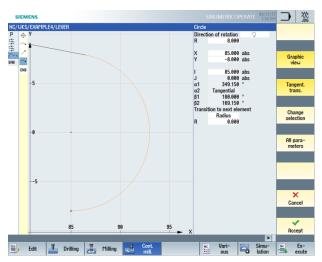


Figure 9-14 Arc for the contour



Accept the contour suggestion.



Accept the entered values.

←•→

Enter the following values for the horizontal straight-line segment up to end point X30 in the screen form:

Field	Value	Selection via toggle key	Notes
Х	30 abs	Х	
R	40		Enter 40 mm for the radius to the subsequent element.

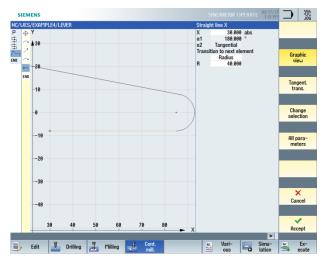


Figure 9-15 Horizontal straight-line segment for the contour



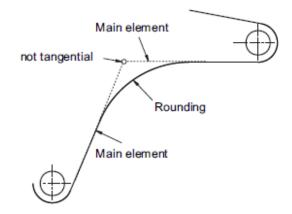
Accept the entered values.



Read the note below for the subsequent inclined straight-line segment:

Note:

The tangential transition is always referenced to the main element only. This means that the straight line does not connect tangentially in this case (see figure below).



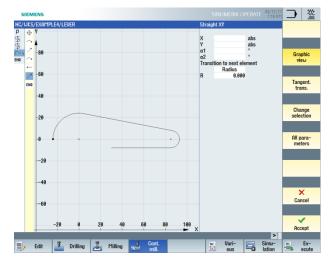
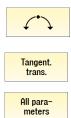


Figure 9-16 Inclined straight line for the contour



Accept your input.

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA



Enter the tangential circular arc.

Select the "Tangent trans." softkey.

Select the "All parameters" softkey.

The "All parameters" function gives you detailed information on the arc.

This can be used, for example, to check the entered values (e.g.: Does the arc end vertically?).

Enter the following values for the circular arc in the screen form:

Field	Value	Selection via toggle key	Notes
Direction of rotation	right	Х	
R	8		
Y	-58 abs		
1	0 abs		
J	-58 abs		

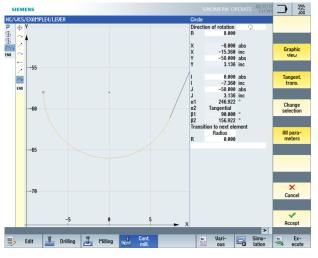


Figure 9-17 Arc for the contour



Select the desired contour suggestion.



Accept the contour suggestion.



Accept your input.



Specify the vertical straight-line segment (automatically tangential) up to the end point Y-27.

Tangent. trans.

Select the "Tangent trans." softkey.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
Y	-27 abs	Х	
R	18	Х	Round the transition to the subsequent straight line using R18.

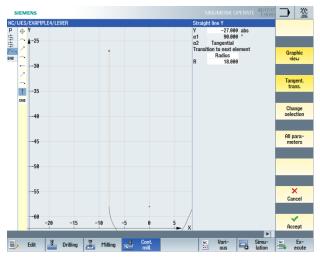


Figure 9-18 Vertical straight-line segment for the contour



Accept the entered values.



Specify the inclined straight line.

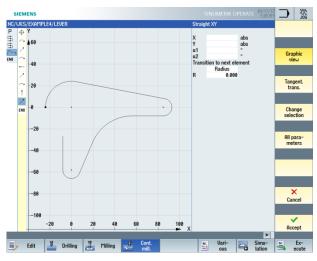


Figure 9-19 Inclined straight line for the contour



Accept your input.

 \leftarrow

Close the contour to the starting point with an arc.

```
Tangent.
trans.
```

Select the "Tangent trans." softkey.

Enter the following values for the starting point of the contour definition in the screen form:

Field	Value	Selection via toggle key	Notes
R	24		
Х	-24	Х	
Υ	0	Х	
1	0	Х	

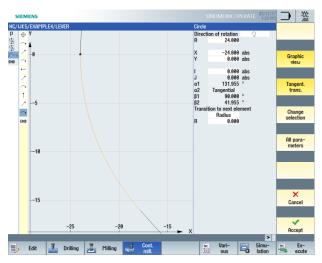


Figure 9-20 Arc for the contour





Accept the contour.

Follow the steps below to rough and finish the pocket taking into account the lever contour:

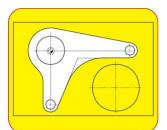


Figure 9-21 Roughing and finishing around the lever



Select the "Pocket" softkey.



Open the tool list and select the "CUTTER20" milling cutter.



Apply the tool to the program.

Field	Value	Selection via toggle key	Notes
F	0.15 mm/tooth	Х	
V	120 m/min	Х	
Machining	Roughing	Х	
Z0	0		
Z1	6 inc	Х	
DXY	50%	Х	Specify the maximum infeed in the plane in %.
DZ	6		
UXY	0		
UZ	0.3		
Starting point	automatically	Х	
Insertion	Vertical	Х	
FZ	0.15 mm/tooth	Х	
Lift mode	To RP	Х	

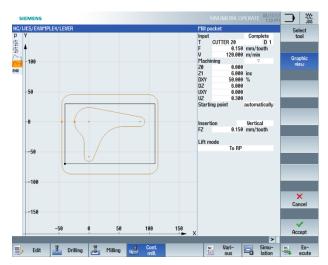


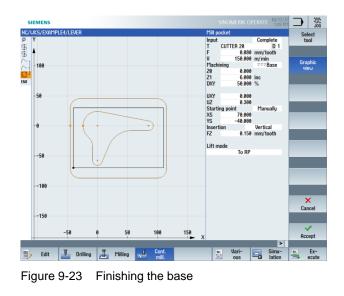
Figure 9-22 Roughing the contour



Pocket

Select the "Pocket" softkey.

Field	Value	Selection via toggle key	Notes
F	0.08 mm/tooth	Х	
V	150 m/min	Х	
Machining	Finishing Base	х	
Z0	0		
Z1	6 inc	Х	
DXY	50%	X	Specify the maximum infeed in the plane in %.
UXY	0		
UZ	0.3		
Starting point	Manual	Х	
XS	70		
YS	-40		
Insertion	Vertical	Х	
Lift mode	To RP	Х	





10.5 Creating the border for the circular island

Operating sequences

Create the border as the traversing path boundary for milling on your own Mill to a depth of -3.

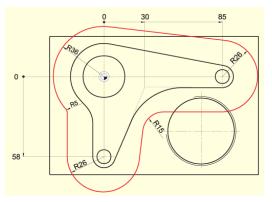


Figure 9-24 Contour of border for the circular islands

Note:

The values R36 and R26 result from the corresponding island radius + cutter diameter (here, 20 mm + 1 mm allowance)

The radii R5 and R15 are freely selected.



Select the "Cont. mill." softkey.

Create a new contour with the name "LEVER_Lever_Area".

New contour	
Please enter the new name	
LEVER_Lever_Area	

Figure 9-25 Creating the contour

Construct the boundary of the traversing paths as described above such that the 20 mm milling cutter fits through everywhere between the boundary and the islands. Enter this boundary contour in the same way as the lever contour.

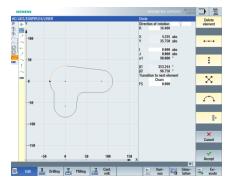


Figure 9-26 Arc contour section, left

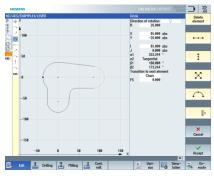


Figure 9-27 Arc contour section, right

10.6 Creating the 30 mm circular island

Operating sequences

Follow the steps below to create the 30 mm circular island shown in the figure.

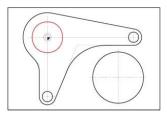


Figure 9-28 30 mm circular island

Select the "Cont. mill." softkey.



Create a new contour with the name "LEVER_Circle_R15".



Figure 9-29 Creating the contour

Create the circular contour on your own (see figure below). The starting point of the circle construction is at X-15 and Y0.

Note:

Make sure that some values are specified with incremental dimensions!

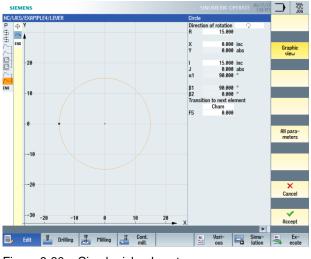


Figure 9-30 Circular island contour

10.7 Creating a 10 mm circular island

Operating sequences

Follow the steps below to create the 10 mm circular island shown in the figure:

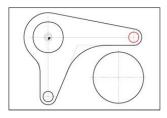


Figure 9-31 10 mm circular island



Select the "Cont. mill." softkey.



Create a new contour with the name "LEVER_Circle_R5_A".



Figure 9-32 Creating the contour

Create the circular contour on your own (see figure below). The starting point of the circle construction is at X80 and Y0.

Note:

Since this circular island will be copied in the next step, you must specify the contour with incremental dimensions so that you only need to change the starting point when copying.

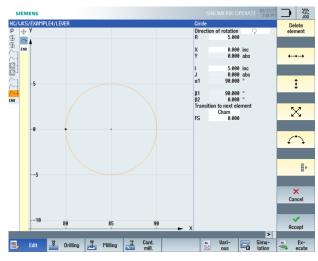
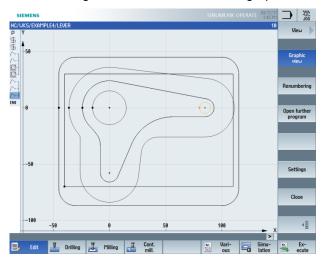


Figure 9-33 10 mm circular island contour



After entering the circle, the broken-line graphic looks like this.

Figure 9-34 Broken-line graphic

10.8 Copying the 10 mm circular island

Operating sequences

Follow the steps below to copy the circular island created in the previous step:

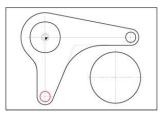


Figure 9-35 10 mm circular island

Сору

Navigate to the "LEVER_Circle_R5_A" contour and copy it.

SIEM	ENS					SINU	JMERIK	OPERATI	03/17/17 1:42 PM		2005 J00
NC/LIKE	/EXAMPLE4/LEVER								10	_	JOG
P N10			G54 Block						10	U	ieu 🕨
5 N20		~		F=0.1/t U	=120m X0=-40 Y6	8=-78 28	=5 21=8				
다 N20 다 N30		~~~			J=150m X0=-40)			
~- N40			LEVER Rectan							Gre	phic
~- N50	Contour		LEVER LEVER								ew
3- N60	Mill pocket	∇	T=CUTTER 20 F	=0.15/t V=	120m 20=0 21=6	inc					
O N70	Mill pocket	AAA B	T=CUTTER 20 F	=0.08/t V=	150m 20=0 21=6	inc					
~1 N80	Contour		LEVER_Lever_	Area							
∧- N90			LEVER_Circle_							Renur	nbering
/~ N10	0 Contour		LEVER_Circle_	R5_A							
END	End of program									_	
											further
										pro	gram
										_	
										Set	tings
											_
										CI	ose
											∢≣
									~		1
									>		
Ξ.	Edit Drilling	, <u>I</u> , I	Milling 📕	Cont.		NC	Vari-		Simu-	×	Ex-
-				mill.			ous	-0	lation	-	ecute

Figure 9-36 Copying the contour

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

SIEME	NS					03/17/17 1:44 PM	
C/UKS/E	EXAMPLE4/LEVER					10	
	Program header		G54 Block			^	
	Face milling	∇	T=FACEMILL 63 F=0.1/t	V=120m X0=-40 Y0=-70 Z	0=5 21=0		_
5 N30	Face milling	222	T=FACEMILL 63 F=0.08/1	V=150m X0=-40 Y0=-70	20=5 21=0		
~1 N40	Contour		LEVER_Rectangular_Are	a			
~- N50	Contour		LEVER_LEVER				
2- N60			New contou				
2 N70			Hew Contou				
~1 N80							Create
~- N90							new
	Contour	Please ente	r the new name			Ð	
10	End of program	LEVER_Circ					Import from
					-		DXF
						-	_
							_
							_
							_
							_
							-
						=	
						=	_
						=	×
							× Cancel
						22	
						22	Cancel
						22	Cancel
							Cancel
						=	Cancel
	_						Cancel

Figure 9-37 Entering the name for the copied contour



Accept your input.

Once applied, your machining plan should look like this:

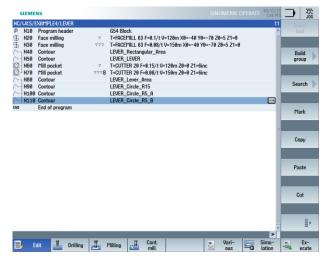


Figure 9-38 Pasted contour in the machining step editor

Now you only have to change the starting point, as you have specified the contour using incremental dimensions.

Paste

Paste the copied contour and name it "LEVER_Circle_R5_B".



Open the contour. You can now use this key in the open contour to open the selected geometry element for changing.

Enter the following values for the starting point of the contour definition in the screen form:

Field	Value	Selection via toggle key	Notes
Х	-5		
Y	-58		



Figure 9-39 Changing the starting point



10.9 Machining the circular island with the help of the editor

Operating sequences

Follow the steps below to machine the three circular islands: In doing so, you will learn additional functions of the machining step editor that will help you to reuse and manage parts of the machining plan (see Functions of the machining step editor).

The following contour serves as the traversing path boundary for machining the islands.

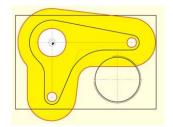


Figure 9-40 Traversing path boundary

Your machining plan will look as follows:

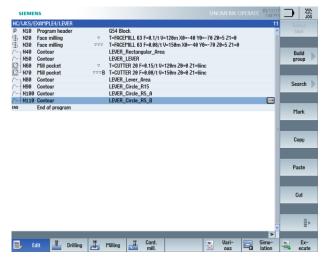


Figure 9-41 Machining plan

Mark

Mark the two machining steps for roughing and finishing the pocket.

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

Сору

Copy the marked machining steps.

										. 03/17/17		~~~
	IEME									1:50 PF		
NC/	uks/e	Xample4/Lever									S	
	N10	Program header		G54 Block						^		
		Face milling	∇	T=FACEMILL 63								
\$		Face milling	~~~	T=FACEMILL 63) Y0=-70 Z	0=5 21=0				
		Contour		LEVER_Rectan	gular_Area						E	uild 📐
		Contour		LEVER_LEVER							g	roup
		Mill pocket	⊽	T=CUTTER 20 F								
61	N70	Mill pocket	277 B	T=CUTTER 20 F	=0.08/t V=	150m 20=0 21=	=6inc			Ð		
\sim_1	N80	Contour		LEVER_Lever_	Area							arch
		Contour		LEVER_Circle_							56	arcn
		Contour		LEVER_Circle_								
		Contour		LEVER_Circle_								
Q1	N120	Mill pocket	∇	T=CUTTER 20 F	=0.15/t V=	120m 20=0 21=	6inc					lark
END		End of program										lark
										_		
												opu
												opy
											P	aste
												asto
												Cut
												out
												≣⊦
										~		= '
										>		
I.		dit 📱 Drill	ing 📕	Milling 🛃	Cont.		NC	Vari-	5	Simu-	<u></u>	Ex-
Ξy					mill.			ous	-0	lation	3	ecute

Figure 9-42 Marked machining steps

Paste

Paste the machining steps below the contours. This links the solid machining technologies with the contours.

SIEME	NS		SINUMER	IK OPERATE 03/17/17 1:51 PM	
	Xample4/Lever			13	
P N10	Program header		54 Block	<u>^</u>	
5 N20 5 N30	Face milling		=FACEMILL 63 F=0.1/t V=120m X0=-40 Y0=-70 20=5 21=		
	Face milling	~~~	'=FACEMILL 63 F=0.08/t V=150m X0=-40 Y0=-70 20=5 21	1=0	
~_ N40	Contour		EVER_Rectangular_Area		Build
V- N50	Contour		EVER_LEVER		group
3 - N60	Mill pocket		=CUTTER 20 F=0.15/t V=120m 20=0 21=6inc		
3 N70	Mill pocket	222 B	=CUTTER 20 F=0.08/t V=150m 20=0 21=6inc		
∼ 1 N80	Contour		EVER_Lever_Area		
V- N90	Contour		EVER_Circle_R15		Search
- N100	Contour		EVER Circle R5 A		
- N110	Contour		EVER Circle R5 B		
3- N120	Mill pocket	~	=CUTTER 20 F=0.15/t V=120m 20=0 21=6inc		
	Mill pocket	⊽⊽⊽₿	=CUTTER 20 F=0.08/t V=150m 20=0 21=6inc		Mark
NO	End of program				
					Сору
					Paste
					Cut
				v	I⊧
				>	
🥠 Б	dit 📕 Drilli	ng 📕 I	Iling Cont. Vari-		Ex-

Figure 9-43 Pasted machining steps

The two solid machining technologies – roughing and finishing – still have to be adapted to the new machining depth:



Open the machining step for roughing.

Enter the following values for roughing in the screen form:

Field	Value	Selection via toggle key	Notes
Z1	3 inc	Х	
Starting point	Manual	Х	
XS	70		
YS	-10		

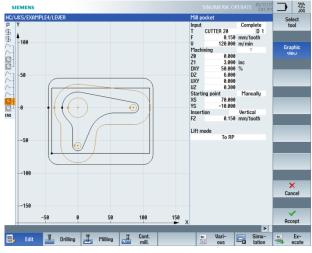


Figure 9-44 Adaptations for roughing



Accept the entered values.



Open the machining step for finishing. Change the values similar to roughing.



Figure 9-45 Adaptations for finishing



Accept the entered values.

Graphic view

This will show which geometries belong to the finishing technology (machining plan graphic).

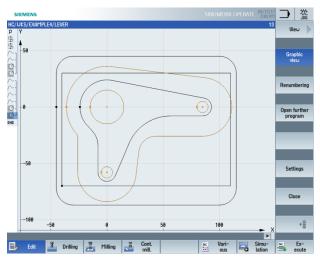


Figure 9-46 Broken-line graphic



Check your intermediate result by way of simulation.

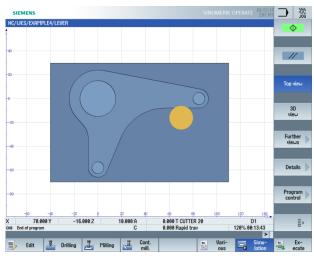


Figure 9-47 Simulation - Top view

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 the "Copy" softkey to copy machining 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.
Ē	Use this softkey to go to the expanded menu.
Renumbering	Use the "Renumbering" softkey to renumber the machining steps.
Settings	Use the "Settings" softkey to open the Settings dialog. Here, you set whether numbering is to be automatic or whether the block end, among other things is to be displayed as a symbol.
₹	Use this softkey to return to the previous menu.

10.10 Deep hole drilling

Operating sequences

Follow the steps below to predrill:

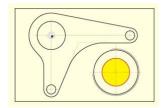


Figure 9-48 Deep hole drilling



Select the "Drill." softkey.



Select the "Drilling Reaming" softkey.



Open the tool list and select "PREDRILL30".



Apply the tool to the program.

Enter the following values for deep hole drilling in the screen form:

Field	Value	Selection via toggle key	Notes
F	0.1 mm/rev	Х	
V	120 m/min	Х	
Depth reference	Tip	Х	
Z1	-21 abs	Х	
DT	0 s	Х	

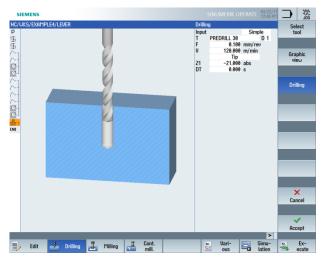


Figure 9-49 Entering the drill hole



Accept the entered values.



Enter the following values for the drilling position in the screen form:

Field	Value	Selection via toggle key	Notes
Positions	Rectangular	Х	
Z0	-6		
X0	70		
Y0	-40		

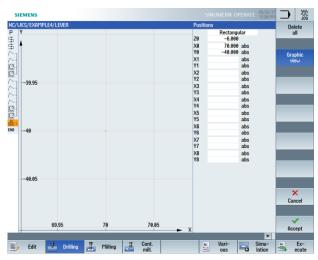


Figure 9-50 Entering the position



10.11 Milling a helix

Operating sequences

Follow the steps below to remove the residual material of the circular ring remaining after the drilling by way of a helical motion:

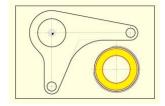


Figure 9-51 Milling a helix



Select the "Strght Circle" softkey.



Open the tool list and select "CUTTER20".



Apply the tool to the program.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
V	120 m/min	Х	

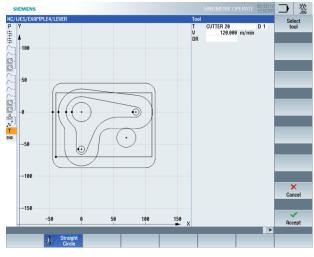


Figure 9-52 Milling a helix



Accept your input.

Straight

Select the "Straight" softkey.

Rapid traverse

Select the "Rapid traverse" softkey.

Enter the following values for the starting point of the contour definition in the screen form:

Note:

Since milling is performed here without cutter radius compensation, you must position the milling cutter with its circumference to the tap hole diameter (here: 45.84 mm) minus the finishing allowance.

Field	Value	Selection via toggle key	Notes
Х	82	Х	
Y	-40	Х	
Z	-5	Х	
Radius compensation	Off	Х	

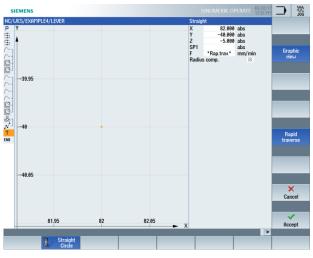


Figure 9-53 Positioning



Helix

Select the "Helix" softkey. Enter the following values for the helix in the screen form:

Field	Value	Selection via toggle key	Notes
1	70	Х	
J	-40	Х	
L	3 mm/rev		The pitch of the helix is 3.
Z	-23 abs	Х	
F	0.1 mm/tooth	Х	

Note:

Since the tool traverses along an inclined path, 6 revolutions are created here to prevent any residual material from remaining (even though the end depth is already reached after 5 revolutions).

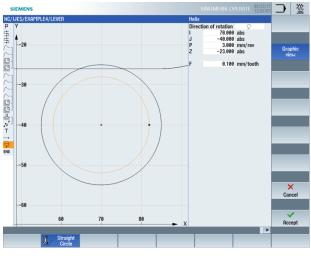


Figure 9-54 Entering the helix



10.12 Boring

Operating sequences

Follow the steps below to machine the circular pocket to the required dimensions using a boring tool:

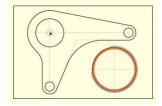


Figure 9-55 Boring a circular pocket

Select the "Drilling" softkey.



Select the "Boring" softkey.

Open the tool list and select "DRILL_tool".

Apply the tool to the program.

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

Field	Value	Selection via toggle key	Notes
F	0.08 mm/rev	Х	
S	500 rpm	Х	
Z1	15 inc	Х	
DT	0 s	Х	
SPOS	45		
Lift mode	Lift	X	The "Lift" option withdraws the tool from the contour before it retracts from the drill hole. This option may only be used with one-edged tools.
D	0.5		

Note:

The angular position of the tool during lifting is specified by the machine manufacturer.

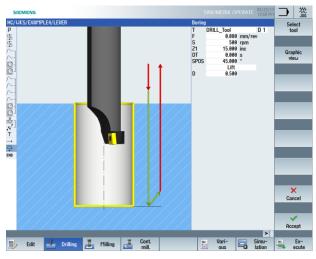


Figure 9-56 Boring



Accept the entered values.

Positions

Position the tool to the drill hole center. The dimension 45.84 mm is specified by the set tool diameter.

Instead of entering the position, you can also use the Repeat position function here.

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

Field	Value	Selection via toggle key	Notes
Z0	-6		
X0	70		
Y0	-40		

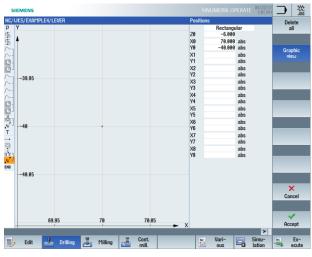


Figure 9-57 Positioning



10.13 Thread milling

Operating sequences

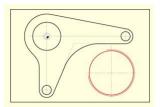


Figure 9-58 Thread milling



Select the "Milling" softkey.

Select the "Thread milling" softkey.



Open the tool list and select "THREADCUTTER".



Apply the tool to the program.

Mill the thread from top to bottom. The "THREADCUTTER" is used for this (F 0.08 mm/tooth, v 150 m/min and pitch 2 mm). A right thread is to be milled to Z-23 (absolute dimension). Due to the overtravel of 3 mm, the thread is always milled cleanly down to the workpiece lower edge even if the lowest tooth is slightly worn.

The help screens are very useful for the input. Compare your entries with the figure below:

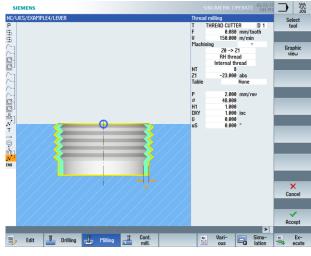


Figure 9-59 Thread milling



Positions

Specify the position for the thread.

Enter the following values in the screen form:

Field	Value	Selection via toggle key	Notes
ZO	-6		
X0	70		
YO	-40		

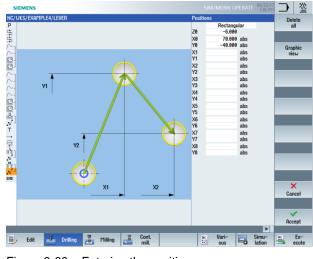


Figure 9-60 Entering the position



10.14 Programming contours using polar coordinates

Programming with polar coordinates

Contour elements in workpiece drawings often reference a pole point. In this case, you do not know the Cartesian coordinates (X/Y), but instead the polar coordinates, i.e. the distance and the angle to this pole.

Now we will slightly modify the lever as a further exercise: The lower "lever arm" is no longer perpendicular to the zero point at X0, but is rotated clockwise by 10°.

In this example, you will learn how to program this position graphically without using the pocket calculator or any auxiliary constructions.

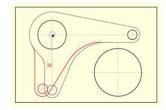
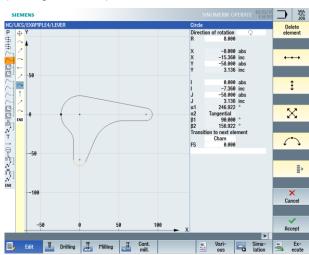


Figure 9-61 Programming the lever using polar coordinates

Operating sequences



First, move the cursor to the arc, whose center point has to be re-dimensioned (see figure below).

Figure 9-62 Cursor on arc

∎ **•**

Expand the menu.

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

Pole

Accept

Position the cursor on the element in front of the arc and paste the pole at this point. Place the pole at the zero point.

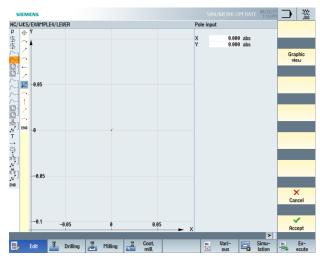


Figure 9-63 Entering the pole

Accept your input.

Next, adjust the values of the arc:

1. In the dialog window of the arc, delete the values that are no longer valid: Y-58, I0 and J-58.

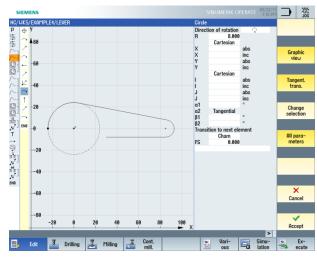


Figure 9-64 Deleting the values

2. Toggle the coordinates for input of the center point from "Cartesian" to "Polar". Enter the distance to the pole and the polar angle (see figure below).

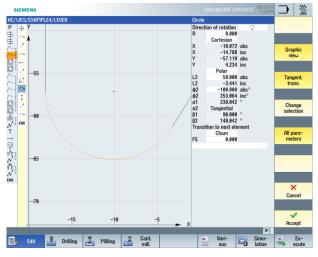


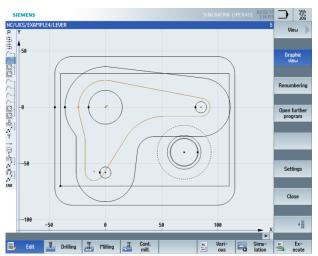
Figure 9-65 Entering the distance to the pole and polar angle



Accept

Accept your input.

Accept the change.



The broken-line graphic shows that the auxiliary pocket "LEVER_Lever_Area" and the circular island "LEVER_Circle_R5_B" must also be adapted in the same way.

Figure 9-66 Broken-line graphic after shifting

Change these two contours on your own. In doing so, note the following information.

Notes:

For the auxiliary pocket, you may proceed a bit "rougher" and approximate the polar dimensions of the center point of arc R26 with Cartesian dimensioning (X-10/Y-57). Then, the contour can be closed directly with a vertical line.

The starting point for the circular island is already dimensioned with polar coordinates. Only the center point of the full-circle arc must be changed.

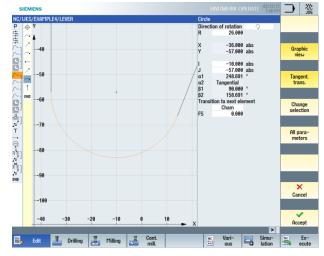


Figure 9-67 Adapting the border

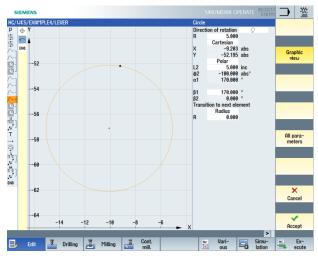
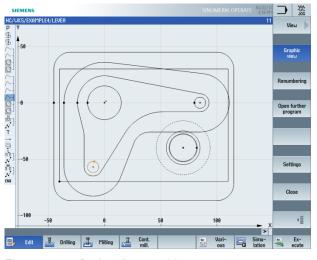


Figure 9-68 Adapting the circular island



After successful adaptation, your broken-line graphic looks like this:

Figure 9-69 Broken-line graphic

11. Example 5: Flange

11.1 Overview

Learning objectives

In this section, you will learn how to

- Create a subprogram
- Mirror machining steps
- Chamfer any contours
- Create longitudinal and circular slots

Task

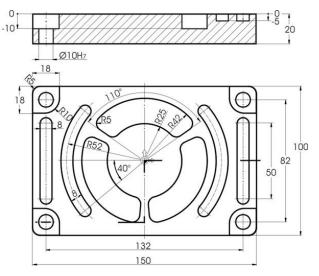
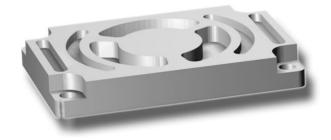


Figure 10-1 Workshop drawing - Example 5



Note:

All machining steps have been explained and nearly all softkeys / keys to be selected / pressed have been shown in the previous examples. In this example, the whole sequence of inputs will no longer be specified, but instead only essential information and the most important softkeys and keys to be pressed.

11.2 Creating a subprogram

Operating sequences

The creation and functioning of subprograms is explained taking the example of the workpiece "CORNER_MACHINING".

Following the steps below you can machine the four corners with the help of a subprogram and the Mirror function.

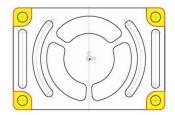


Figure 10-3 Contour of the four corners

New

Create a new sequential program with the name "CORNER_MACHINING". You will later link this program as a subprogram.

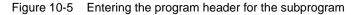
	New sequential program	
Туре	ShopMill	Ŧ
	CORNER_MACHINING	-

Figure 10-4 Creating a subprogram

Enter the following data for the program header. The blank dimensions will be

SELMENS
SINUAL RUC OPERATE SAME
Nor-UCK/SMMPLES/CORNER_MACINANG
Program header
Uork offset
Heads un, mm
Uork offset
Blank
None
PL 617 (XY)
Retraction plan
PD 18 899
Safety distance
SC 2899
Machining sense
Dour-out
Retract position pattern
Optimized

specified later centrally in the main program.





Accept the entered values.



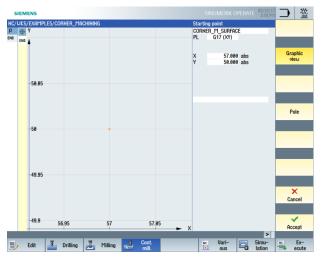
Select the "Cont. mill." softkey.



Create a new contour with the name "CORNER_M_SURFACE".



Figure 10-6 Creating the contour



Specify the starting point. The top right corner, for example, will be constructed.

Figure 10-7 Entering the starting point



Accept the entered values.

Create the contour. After entering the two contour elements, your screen should look like this: Accept the contour to apply it to your machining plan.

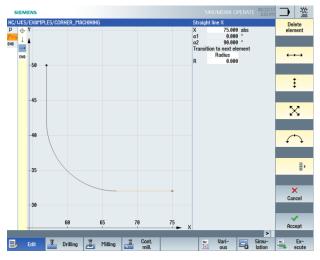


Figure 10-8 Subprogram for top right corner contour



The contour is to be roughed with the 20 mm milling cutter (F = 0.15 mm/tooth and V = 120 m/min).

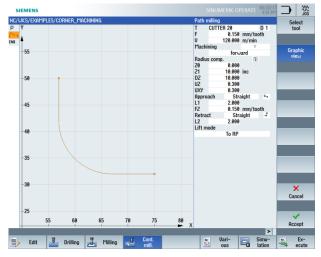


Figure 10-9 Roughing the contour

The approach and retraction distances are traversed in a straight line. The length values are the distances between the cutter edge and the workpiece.

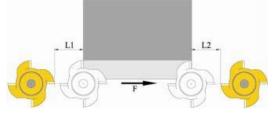


Figure 10-10 Approach and retraction distances in a straight line





The contour is to be finished with the same milling cutter (F = 0.08 mm/tooth and V = 150 m/min).

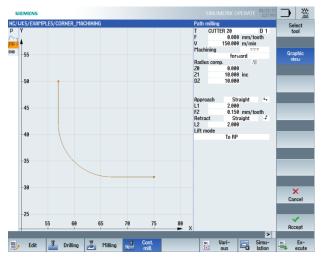


Figure 10-11 Finishing the contour



Accept the entered values.

Cont. mill.

Select the "Cont. mill." softkey.



Create a new contour with the name "CORNER_M_ARC".

In the next few steps, the corner of the blank block is to be rounded using R5:



Figure 10-12 Creating the contour

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

SIEMENS	SINUMERIK OPERATE 83/22/17 328 PT
NC/UKS/EXAMPLE5/CORNER_MACHINING	Starting point CORNER_M_GRC PL G17 (XY)
-50.05	X 70.000 abs Graphic Y 50.000 abs
-50	Pole
-49.95	
-49.95	×
	Cancel
-49.9 69.95 70	78.85 X
📄 Edit 🗾 Drilling 🛃 Milling 💽	Cont. Vari- mill. Simu- Ex-

Specify the starting point.

Figure 10-13 Entering the starting point



Accept the entered values.

Specify the contour and the associated machining steps:

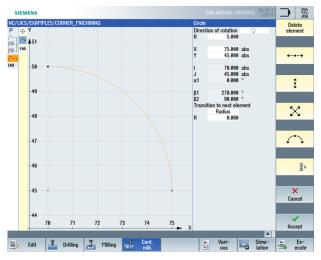


Figure 10-14 Entering the geometry

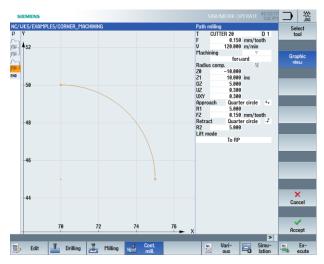


Figure 10-15 Roughing the contour

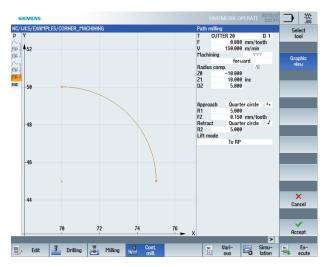


Figure 10-16 Finishing the contour

Program header		G54 Blank: None	
∼ ₁ Contour		CORNER_M_SURFACE	
9 - Path milling	∇	T=CUTTER 20 F=0.15/t V=120m 20=0 21=10inc	
Path milling	$\nabla \nabla \nabla$	T=CUTTER 20 F=0.08/t V=150m 20=0 21=10inc	
Contour		CORNER_M_ARC	
🧏 - Path milling	∇	T=CUTTER 20 F=0.15/t V=120m 20=-10 21=10inc	
Path milling	$\nabla \nabla \nabla$	T=CUTTER 20 F=0.08/t V=150m 20=-10 21=10inc	F

Figure 10-17 Complete subprogram in the machining step editor

11.3 Mirroring of machining steps

Task

Now that you have finished the subprogram, create the main program. By using the "Mirroring" function in the "Transformations" menu, you can use the subprogram for all four workpiece corners.

Mirroring can be performed in two different ways:

• new:

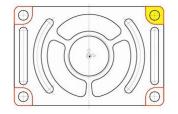
Mirroring is performed from the location at which the 1st machining took place.

• additive:

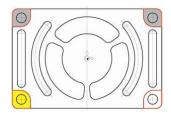
Mirroring is performed from the last machined location

The sequence of machining is then shown schematically with setting "New":

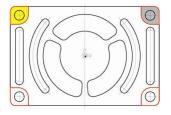
1st machining (see subprogram) (the X values are mirrored here)



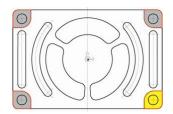
3rd machining: Mirroring of the X and Y axes (the X and Y values are mirrored here)



2nd machining: Mirroring of the X axis



4th machining: Mirroring of the Y axis (the Y values are mirrored here)



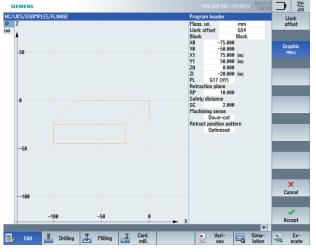
Operating sequences

Create the main program with the name "FLANGE".

	New sequential program	
Type Name <mark>FLANG</mark>	ShopMill	-
Name FLANG	Ē	

Figure 10-18 Creating the main program

Enter the program header.





Entering the program header of the main program



Accept the entered values.



Select the "Various" softkey.



Insert the subprogram into the main program.

Note:

If you have created the subprogram in the same directory as the main program, the "Path/Workpiece" text box may remain empty.

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

SIEMENS NC/UKS/EXAMPL P N10 Program	n header	Block	_		SINU	JMERIK OPER	ATE 03/22/17 3:45 PM 1	
		s	elect subprogram					New
Local driv Local drive Loc	rces PLE1.UPD PLE2.UPD PLE3.UPD PLE3.UPD PLE4.UPD				03/13/17 01/25/16 01/25/16 03/13/17 01/25/16 03/24/17	1:24:58 PM 3:39:11 PM 3:39:11 PM 1:15:38 PM 3:39:11 PM 1:54:56 PM		Search
COL E COL E FL E TEMP Part pro- Subprog - USB	RNER_MACHINING. IMPLE5.MPF INGE.MPF WPD Igrams			1612 4135 115	03/15/17 03/27/17	2:28:19 PM 7:39:57 AM 2:28:57 PM 1:24:58 PM 3:39:11 PM 3:39:11 PM	2.5 MB	
NC/Workpiece	s/EXAMPLE5.WPD					Free:	2.5 MB	X Cancel
		_					v	OK

Figure 10-20 Inserting the subprogram



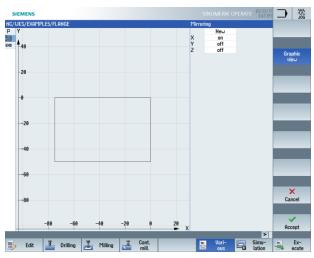
Accept your input. Once applied, your machining step program looks like this:

NC/	'WKS/EXAMPLE5/FLANGE		
Ρ	Program header	G54 Block	
	Execute	"Corner_machining.mpf"	\rightarrow
END	End of program		

Figure 10-21 Subprogram inserted in the main program



The axes can be shifted, rotated, etc., by selecting the "Transformations" softkey.



Preparation for the 2nd machining: Mirroring of the X values.

Figure 10-22 Mirroring



Accept your input.

To mirror the remaining machining operations, proceed as follows:

Copy the subprogram after the "Mirroring" machining step. The 2nd machining occurs.

You must then repeat the "Mirroring" and "Subprogram call" operations for the two other corners.

	IEMENS		SINUM	ERIK OPERATE	03/22/17 3:51 PM	\supset	20G
	WKS/EXAMPLE5/FLANGE				4	Uo off:	rk
P	Program header	G54 Block			^	off	set 🗾
調	Execute	"CORNER_MACHINING.MPF"					
4+⊾	Mirroring	x			_	_	
調	Execute	"CORNER_MACHINING.MPF"			Ð	Off	set 🕨
END	End of program					011	
						_	_
						Rota	tion 🕨
							- P
						Sca	ling 🗼
						Firre	oring 🕨
						_	
						_	
							_
						<	·
						Ba	
					×	Da	UK
		T Milling I Cont.		ari- 📼	Simu-	-	Ex-
	> Edit 💆 Drilling	Hilling Cont. mill.		ari- Jus - 1	lation		ecute

Figure 10-23 Copying the subprogram

The help screen illustrating this procedure will help you. After you have entered all four machining operations, you must disable the mirroring in all three axes.

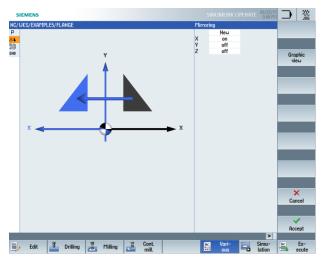


Figure 10-24 Mirroring help screen

s	IEMENS	SINUMERIK OPERATE 03/22/17	
NC/	WKS/EXAMPLE5/FL	NGE 9	Work N
Ρ	Program header	G54 Block	offset
騆	Execute	"CORNER_MACHINING.MPF"	
4+1	Mirroring	X	
調	Execute	"CORNER_MACHINING.MPF"	Offset
4	Mirroring	XY	Unset
韻	Execute	"CORNER_MACHINING.MPF"	
	Mirroring	Y	
韻	Execute	"CORNER_MACHINING.MPF"	Rotation >
1+1	Mirroring		Rotation
END	End of program		
			Scaling 🕨
			Mirroring 🔪
			« Back
		>	
	🕨 Edit	Drilling 🛃 Milling 🛃 Cont. 🛅 Vari- 🤤 Simu-	Ex- ecute

Your machining step program will look like this:



Check your work by now using the simulation.

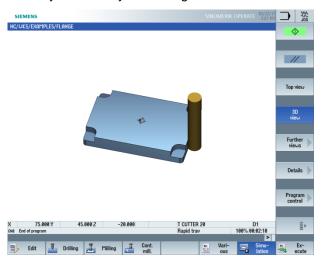


Figure 10-26 Simulation in 3D view

11.4 Holes

Operating sequences

With the next machining steps, you will create the four drill holes in the corners. Since an obstacle lies between the individual drill holes, it must be specified between the positions.

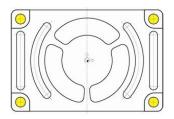


Figure 10-27

Holes

Figure 10-28 Centering

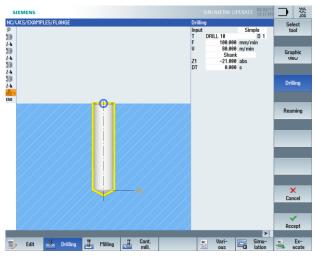


Figure 10-29 Drilling

SIEME		SINUMERIK OPERATE 03/24/17 12:23 PM	
	EXAMPLE5/FLANGE	18	- 22
N10	Program header	Block	••
) N20	Execute	"CORNER_MACHINING"	
N30	Mirroring	X	_
N40	Execute	"CORNER_MACHINING"	مر
N50	Mirroring	XY	••
N60	Execute	"CORNER_MACHINING"	
N70	Mirroring	Y	
B N80	Execute	"CORNER_MACHINING"	H
N90	Mirroring		
	Centering	T=CENTERDRILL 12 F=150/min S=500rev Ø11	
- N110	Drilling	T=DRILL 10 F=100/min V=80m Z1=-21	
	001: Positions	20=-10 X0=-66 Y0=-41	
	002: Obstacle	Z=1	
	003: Positions	20=-10 X0=66 Y0=-41	
	004: Obstacle	Z=1	
	005: Positions	20=-10 X0=66 Y0=41	÷
	006: Obstacle	Z=1	
/ N180	007: Positions	20=-10 X0=-66 Y0=41 🖃	
0	End of program		
			.
			Obstacle
			«
		~	Back
		>	
<u></u> р Е	dit 🛃 Drilling	Tilling Cont. Vari- ous lation	Ex-
	dit 🎿 Drilling	Milling Milling Milling Milling	= ecute

Figure 10-30 Entering the positions of the obstacles

11.5 Rotation of pockets

Operating sequences

Follow the steps below to program the contour and the machining for the pocket highlighted in yellow.

By rotating the coordinate system, the other two pockets are then created.



Select the "Cont. mill." softkey.



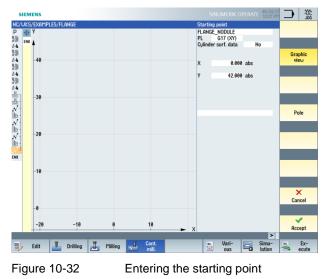
Create a new contour with the name "FLANGE_NODULE".



Figure 10-31

Creating a new contour

Specify the starting point.







Select the "Arc" softkey.

All parameters

Select the "All parameters" softkey.

The arc R42 is described unambiguously, e.g. via the radius, the center point in X and the runout angle. Construct in the counterclockwise direction to ensure that the pocket can also be finished by synchronized milling.

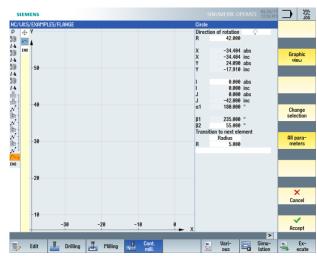


Figure 10-33 Entering the arc



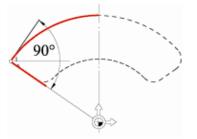


Select the "Diagonal" softkey.

All parameters

Select the "All parameters" softkey.

Create the diagonal straight-line segment.



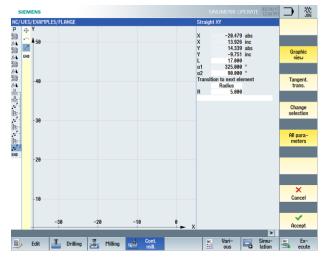


Figure 10-34 Entering the diagonal





Select the "Arc" softkey.

Create the second arc.

All parameters

Select the "All parameters" softkey.

SIEMENS NC/UKS/EX6 P ⊕ ♥ 部部 公本 288 ~ 44 ~ 588 ~ 44 ~ 588 ~ 488 ~ 58 rotation 25.000 20.479 abs 14.339 abs Grap 0.000 abs Ι J α1 α2 β1 β2 Tanger trans Change selectio All para-meters 10 × Cancel ~ -30 -20 -10 0 10 20 Accept Edit 🛃 Drilling 📕 Milling Į, NC 142 Vari-Ex-Simu-

Figure 10-35 Entering the arc



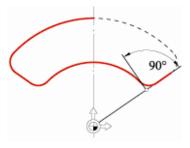


Select the "Diagonal" softkey.

All parameters

Select the "All parameters" softkey.

Create the second diagonal straight-line segment.



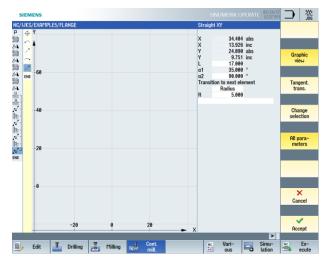


Figure 10-36 Entering the diagonal



()

Select the "Arc" softkey.





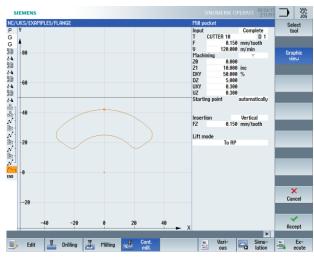
Figure 10-37 Entering the final arc



Accept the entered values.



Accept the contour pocket to apply it to your machining plan.



Create the following machining steps on your own:

Figure 10-38 Roughing a pocket

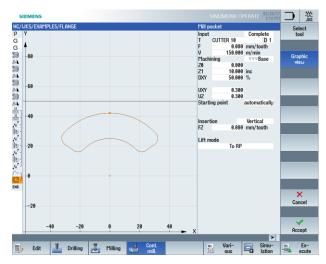


Figure 10-39 Finishing the pocket base

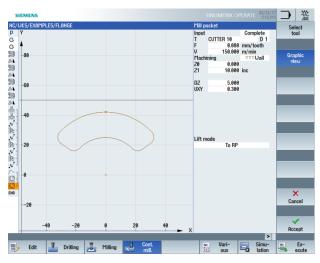


Figure 10-40 Finishing the pocket wall

Follow the steps below to copy the created machining step sequence for machining the three pockets:

Mark

Mark the complete machining step sequence describing the pocket machining in the machining step editor.

Сору

Copy the machining step sequence to the clipboard.

	IEME								SIN	UMERIK	OPERATI	2:20 PT1		۲. JOG
	WKS/E	Example5/Flange												
Ρ	N10	Program header		Block								^		
鮰	N20	Execute		"CORN	ER_MAC	HINING"								
1+1	N30	Mirroring		х										
酿	N40	Execute		"CORN	ER_MAC	HINING"							F	Build 🕟
1+1	N50	Mirroring		XY										roup
酿	N60	Execute		"CORN	ER_MAC	HINING"								
1+1	N70	Mirroring		Y										
韻	N80	Execute		"CORN	ER_MAC	HINING"								
1+1	N90	Mirroring											S	earch 🕨
U., .	N100	Centering		T=CEN1	FERDRIL	L 12 F=15	0/min S=50	00rev Ø11	1					
Ц.	N110	Drilling		T=DRIL	L 10 F=1	00/min U	=80m Z1=-	-21						
1	N120	001: Positions		20=-10	X0=-66	Y0=-41								
h.	N130	002: Obstacle		Z=1										1ark
1	N140	003: Positions		20=-10	X0=66 1	/0=-41							_	
pj -	N150	004: Obstacle		Z=1										
1	N160	005: Positions		20=-10	X0=66 1	/0=41								
þ.	N170	006: Obstacle		Z=1									(Copy
1	N180	007: Positions		20=-10	X0=-66	Y0=41							_	
~.	N190	Contour		FLANG	E_NODU	E								
0	N200	Mill pocket	∇	T=CUTT	TER 10 F	=0.15/t V	=120m Z0=	0 Z1=10in	C					
0	N210	Mill pocket		T=CUT1	FER 10 F	=0.08/t V	=150m 20=	0 Z1=10in	C				P	aste
0	N220	Mill pocket	⊽⊽⊽U	T=CUTT	TER 10 F	=0.08/t V	=150m Z0=	0 Z1=10in	IC			•	_	
END		End of program										_		
														Cut
												_	_	
												_		
												_		=.
												~		≣⊦
												>		
	_	7	ज .		T	Cont.			NC	Vari-		Simu-	NC	Ex-
Ę	р Е	dit 📕 Drilling	1	Milling	1	mill.				ous	-	lation	-	ecute

Figure 10-41 Copying the machining steps



Select the "Various" softkey.

Transformations

Select the "Transformations" softkey.



The coordinate system is rotated around the Z axis by 120°.

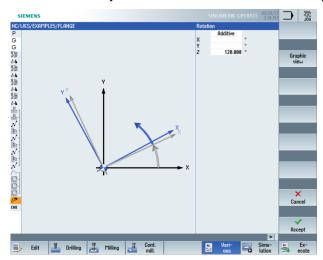


Figure 10-42 Rotation around the Z axis

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

Ex-ecute

Vari-ous Simu-lation

Accept Paste		: your inpu	ut. I machining steps		
	SIEMENS			INUMERIK OPERATE 03/24/17 2:30 PM	
	NC/WKS/EXAMPL	E5/ELONGE		27	
	P N10 Progra		Block		View
	新日 N20 Execut		"CORNER_MACHINING"		· · · ·
	AN N30 Mirror		X		
	調 N40 Execut		"CORNER_MACHINING"		Graphic
	∆+k N50 Mirror		XY		view
	語 N60 Execut		"CORNER_MACHINING"		
	∆+⊾ N70 Mirror		Y		
	18 N80 Execut		"CORNER_MACHINING"		Renumbering
	∆+k N90 Mirror				Renumbering
	N100 Center	ring	T=CENTERDRILL 12 F=150/min S=500rev Ø11		
	N110 Drilling	1	T=DRILL 10 F=100/min V=80m Z1=-21		
	N N120 001: P	ositions	20=-10 X0=-66 Y0=-41		Open further
	p: N130 002: 0	bstacle	Z=1		program
	N- N140 003: P	ositions	20=-10 X0=66 Y0=-41		
	p1 - N150 004: 0		Z=1		
	N - N160 005: P		20=-10 X0=66 Y0=41		
	p1 - N170 006: 0		2=1		
	N180 007: P		20=-10 X0=-66 Y0=41		
	∧ N190 Contou		FLANGE NODULE		
	C N200 Mill po		T=CUTTER 10 F=0.15/t V=120m 20=0 21=10inc		
	C N210 Mill po		T=CUTTER 10 F=0.08/t U=150m 20=0 21=10inc		Settings
	N220 Mill po		T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc		-
	A™ N230 Rotatio		add Z=120		
	N240 Contou		FLANGE_NODULE		Close
	C - N250 Mill po		T=CUTTER 10 F=0.15/t V=120m 20=0 21=10inc		01000
	C - N260 Mill po		T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc		
	S N270 Mill po		T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc		
	END End of	program			

Figure 10-43 Pasting the copied machining steps

Transfor-mations

Select the "Transformations" softkey.

Rotation

Enter another rotation by 120°.

Edit Z Drilling A Milling Cont.

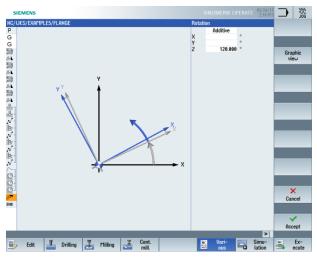


Figure 10-44 Rotation around the Z axis



Accept your input.

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA

Paste

Paste the copied machining steps.

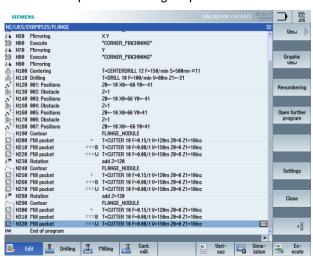


Figure 10-45 Pasting the copied machining steps

Rotation

Select New and specify the value 0° to undo the rotation.

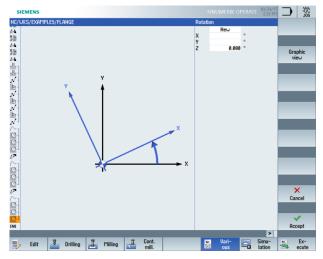


Figure 10-46 Undoing the rotation



Accept your input.

11.6 Chamfering contours

Operating sequences

Chamfer the circular pocket last machined without help.

For chamfering, you need a tool type which allows a tip angle to be entered, in the example "CENTERDRILL12".

Loc.	Туре	Tool name	ST	D	Length	ø	Tip angle		Ĥ	ಸ 1	년 2	
ЦЦ.												
1		CUTTER 4	1	1	65.000	4.000		3	2	~		
2		CUTTER 6	1	1	120.000	6.000		3	2	~		
3	V	CENTERDRILL 12	1	1	120.000	12.000	90.0		2	\checkmark		

Figure 10-47	Centering drill

Select "Chamfer" for the machining. The machining of the chamfer is programmed using the chamfer width (FS) and the insertion depth of the tool tip (ZFS).

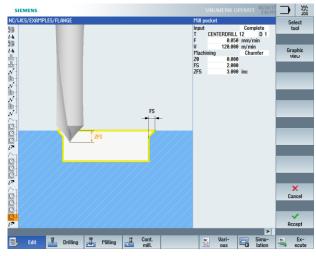


Figure 10-48 Chamfering

_	uks/e	XAMPLE5/FLANGE		SINUMERIK OPERATE 43/24/ 2:46	33	_	JOG
<u> </u>		Mirroring			^	Vie	ew
	N80	Execute		, "Corner Machining"			
		Mirroring					
51		Centering		T=CENTERDRILL 12 F=150/min S=500rev @11		Gra	phic
	N110	Drilling		T=DRILL 10 F=100/min V=80m Z1=-21		vie	èω
2	N120	001: Positions		20=-10 X0=-66 Y0=-41			
, .	N130	002: Obstacle		2=1	1.		
1	N140	003: Positions		20=-10 X0=66 Y0=-41		Renum	
, .	N150	004: Obstacle		2=1		nenum	inetini
1	N160	005: Positions		20=-10 X0=66 Y0=41			
, .	N170	006: Obstacle		2=1	1.		
•]	N180	007: Positions		20=-10 X0=-66 Y0=41		Open f	
۲٦	N190	Contour		FLANGE_NODULE		prog	jram
2-	N200	Mill pocket	∇	T=CUTTER 10 F=0.15/t U=120m 20=0 21=10inc			
1	N210	Mill pocket	∀∀∀ B	T=CUTTER 10 F=0.08/t U=150m 20=0 21=10inc			
		Mill pocket	⊽⊽⊽U	T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc			
•	N230	Rotation		add Z=120			
۲٦	N240	Contour		FLANGE_NODULE			
		Mill pocket	∇	T=CUTTER 10 F=0.15/t V=120m 20=0 21=10inc	15		
		Mill pocket	⊽⊽⊽B	T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc		Sett	tings
		Mill pocket	va∆n	T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc		5011	anya
		Rotation		add Z=120			
		Contour		FLANGE_NODULE			
		Mill pocket	∇	T=CUTTER 10 F=0.15/t V=120m 20=0 21=10inc		Clo	nse
2-	N310	Mill pocket	⊽⊽⊽₿	T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc		one	
		Mill pocket		T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc			
		Mill pocket	Cham.	T=CENTERDRILL 12 F=0.05/min V=120m 2FS=3 20=0 🖂			
•	N340	Rotation		2=0			∎
0		End of program			~		.=
				>			
		dit 📕 Drilling		illing 📕 Cont. 🔛 Vari- ous 🖬 lation	- 6	NC	Ex-

Figure 10-49 Chamfering machining step in the machining step editor

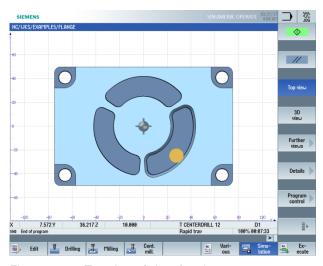


Figure 10-50 Top view of chamfered contour

11.7 Longitudinal and circumferential slots

Operating sequences

Finally, program the slots. They will be positioned correctly using Position pattern and Positioning to full circle.

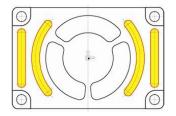


Figure 10-51

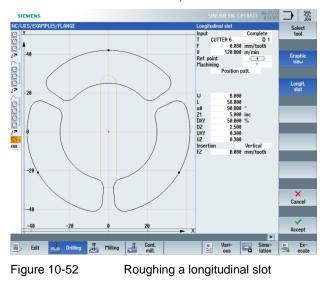
Longitudinal and circular slots



Select the "Milling" softkey.

Select the "Slot" softkey.

Select tool For roughing the longitudinal slots, use the "CUTTER6" tool (F = 0.08 mm/tooth and v = 120 m/min).





SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA



Use the same tool for finishing (F = 0.05 mm/tooth and V = 150 m/min).

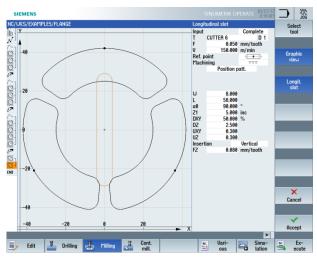


Figure 10-53 Finishing a longitudinal slot



Accept the entered values.

Select the "Drilling" softkey.

Specify the positions of the longitudinal slots below. The reference point lies in the center of the slot.

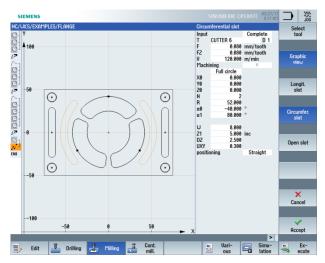


Figure 10-54 Entering the positions for the longitudinal slot





Select the "Milling" softkey.

Select the "Slot" softkey.

Rough the circumferential slots with the "CUTTER6" tool (F = 0.08 mm/tooth and FZ= 0.08 mm/tooth and V = 120 m/min).

The Full circle option positions the circumferential slots automatically with equal spacing. The reference point in X/Y/Z refers to the center point of the circumferential slots.

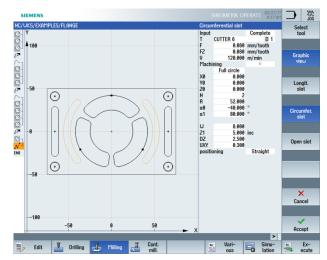


Figure 10-55 Roughing a circumferential slot



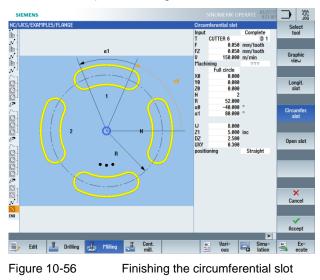
Accept the entered values.

Select the "Slot" softkey.

SCE Training Curriculum | CNC Technology Module 700-030, Edition 02/2016 | Digital Industry, DI FA



Use the same tool (F = 0.05 mm/tooth, FZ = 0.05 mm/tooth and V = 150 m/min) for finishing.





Accept the entered values.

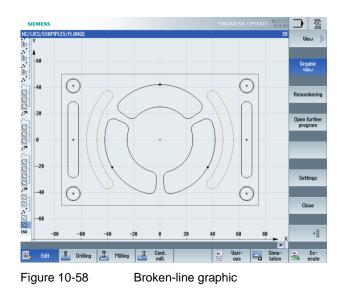
Machining plan

C/WKS/E	XAMPLE5/FLANGE		39	View
- N130	002: Obstacle		Z=1	View
- N140	003: Positions		20=-10 X0=66 Y0=-41	
- N150	004: Obstacle		Z=1	
N160	005: Positions		20=-10 X0=66 Y0=41	Graphic
- N170	006: Obstacle		Z=1	view
' N180	007: Positions		20=-10 X0=-66 Y0=41	
	Contour		FLANGE_NODULE	
	Mill pocket	V	T=CUTTER 10 F=0.15/t V=120m 20=0 21=10inc	Renumbering
	Mill pocket		T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc	
	Mill pocket	A∆∆∆	T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc	
	Rotation		add 2=120	
	Contour		FLANGE_NODULE	Open further
	Mill pocket	∇	T=CUTTER 10 F=0.15/t V=120m 20=0 21=10inc	program
	Mill pocket		T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc	
	Mill pocket	vv⊽U	T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc	
	Rotation		add Z=120	
	Contour		FLANGE_NODULE	
	Mill pocket	∇	T=CUTTER 10 F=0.15/t V=120m 20=0 21=10inc	1
	Mill pocket		T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc	
- N320	Mill pocket		T=CUTTER 10 F=0.08/t V=150m 20=0 21=10inc	Settings
	Mill pocket	Cham.	T=CENTERDRILL 12 F=100/min V=120m ZFS=3 20=0	
	Rotation		Z=0	
	Longitudinal slot	~	T=CUTTER 6 F=0.08/t V=120m Z1=5inc U=8 L=58	
	Longitudinal slot	222	T=CUTTER 6 F=0.05/t V=150m 21=5inc U=8 L=58	Close
	008: Positions		20=0 X0=66 Y0=0 X1=-66 Y1=0	
	Circumfer. slot	V	T=CUTTER 6 F=0.08/t V=120m X0=0 Y0=0 20=0 21=5inc	
	Circumfer. slot	444	T=CUTTER 6 F=0.05/t V=150m X0=0 Y0=0 20=0 21=5inc 🖃	-
	End of program			
			>	
	_		tilling T Cont. NC Vari- mill. NC Vari- ous Cont.	NC Ex-

Figure 10-57

Extract from machining plan

Broken-line graphic



Simulation in 3D view

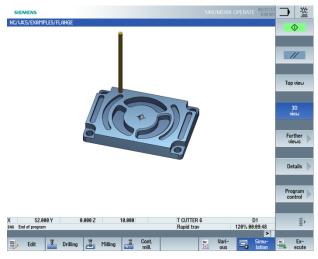


Figure 10-59 3D view

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 ShopMill, the next step is to machine the workpieces.

The following steps 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 ShopMill 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. Machine jaw vices or clamps are normally used for this.

Setting the workpiece zero

Since ShopMill cannot guess where in the work area the workpiece is located, you must determine the workpiece zero.

In the plane, the workpiece zero is set in most cases using one of the following:

- Contacting with a 3D probe
- Contacting with an edge probe

In the tool axis, the workpiece zero is set in most cases using one of the following:

- Contacting with the 3D probe
- Scratching with a tool

Note:

Observe the manufacturer's specifications for use of measuring tools and measuring cycles.

Executing the machining plan

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

First, select the program you want to use for the machining in the Program Manager, e.g. "INJECTION_FORM".

SIEMENS					03/27/17 8:28 AM	G	200 J00
Name Part programs Subprograms	Type DIR DIR	Length	Date 01/25/16 01/25/16	Time 3:39:11 PM 3:39:11 PM		Exec	
Good Status Good Stat	DIR		01/25/16 01/25/16	3:39:11 PM 3:39:11 PM			
EXAMPLE2 INJECTION_FORM	UPD MPF	1172	03/27/17 11/27/13	8:27:26 AM 3:11:06 PM	Ð	Ne	ω 🕨
EXAMPLE3 EXAMPLE4	UPD UPD		01/25/16 01/25/16	3:39:11 PM 3:39:11 PM			
EXAMPLE5 EXAMPLE5	UPD		03/27/17	7:52:34 AM		Op	en
						-	
						Ma	rk
						-	
						Co	py .
						Pas	ste
						G	a
							~
							∎⊧
NC/Workpieces/EXAMPLE2.WPD				Free	2.5 MB	-	-
NC Local VUSB							

Figure 11-1 Selecting the program



Open the program.

_		NS Xample2/Injection_F0	DM					S	INUMERIK	OPERATI	03/27/17 8:29 Af		
	N10	Program header	JNIT	Block									
	N20	T=CUTTER 20 V=80m		DIOCK								_	1001
	N30	RAPID X-12 Y-12											
	N40	RAPID Z-5											Build
	N50	F100/min G41 X5 Y5											roup
2		X=30 Y=75										-	
	N70	L20 α176											
×	N80	G2 a90											
•	N90	X120										S	earch
	N100	X=120 Y=75											
	N110	G2 α4											
•	N120	X145 Y5											
•	N130	X-20											1ark
•	N140	G40 X-12 Y-12											
j.	N150	Rectang.pocket	∇	T=CUTT	'ER 10 F	=0.15/t V	=120m X0=	75 Y0=50 Z0	=0 21=-15		-		
ĵ.		Rectang.pocket	~~~	T=CUTT	'ER 10 F	=0.08/t V	=150m X0=	75 Y0=50 Z0	=0 21=-15				
		Circular pocket	∇				=120m Z1=						Сору
3-		Circular pocket	222				=150m Z1=	-10 Ø30					_
IJ	N190	001: Position grid		20=0 X	9=30 YO	=25 N1=2	N2=2						
0		End of program											aste
													aste
													Cut
													≣⊦
											>		
		dit 📕 Drilling	I.		-	Cont.		NC	Vari-		Simu-	4	Ex-

Figure 11-2 Opening the machining plan



Select the "NC Execute" softkey.

SIEMENS								E 03/27/17 8:31 AM	Μ	
NC/WKS/EXAMPLE2	/INJECTION_FORM								G	
// Reset			М	RD					funct	ions
Work	Position	[mm]			T,F,S					
X Y	0.0	000 000			т				Auxil funct	lary ions
Z SP1	1000.0	0.000 °			F	0.000			Bas bloc	
						0.000	mm/min	100%		
					S1	0		Ø	Tim	
					Master	0		100%		
⊞ 654		_			٩		50 .	100.	Deer	
NC/UKS/EXAMPLE2									Prog	ram els
P N10 Program T N20 T=CUTTE	header R 20 V=80m	Block						^		
→ N30 RAPID X-										
→ N40 RAPID Z										
	G41 X5 Y5									
V N60 X=30 Y=										
→ N70 L20 α17	5									
→ N80 G2 α90									Act. w Maci	alues
→ N90 X120									Maci	aine
12 N100 X=120 Y	75									and the second value of th
· N110 G2 α4	15									1.00
→ N110 G2 α4 → N120 X145 Y5	-/5									≣⊦
· N110 G2 α4	=/5			_		_	_	>		₽

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.

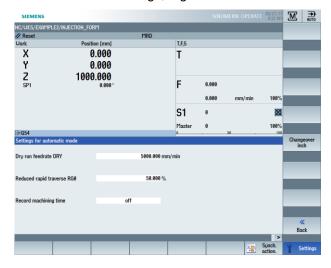
Simult. record.

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 move in 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%.



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

SIEME	NS			SINUMERIK OPERAT	E 03/27/17 8:33 AM	М	→ AUTO
NC/WKS/E	XAMPLE2/INJECTION_FORM						
// Reset		MRD RG0					
Work	Position [mm]		T,F,S				
Х	0.000		Т				_
Y	0.000						
Z	1000.000					_	
SP1	0.000 °		F	0.000			
				0.000 mm/min	100%		
			S1	0	Ø		
H-G54			Master	0 50	100%		
	XAMPLE2/INJECTION_FORM		Program		100		
P N10	Program header Block			No axis motion			
T N20	T=CUTTER 20 V=80m		DRY	Dru run feedrate			
	RAPID X-12 Y-12		✓ RG0	Reduced rapid trav.			_
→ N40	RAPID Z-5		M01	Programmed stop 1			
	F100/min G41 X5 Y5		DRF	Handwheel offset			
	X=30 Y=75		SKP MRD	Skip block Display meas, result			
	L20 α176		✓TINU	SB1: Single block rough			
	G2 a90		-	Soft Single block rough			
	X120 X=120 Y=75						
∠ N100 → N110							
	X145 Y5					<	C
→ N130			~			Ba	ck
					>		
	Over- store	Prog. cntrl.	Block search	3 1 1	Simult. record.		Prog. corr.

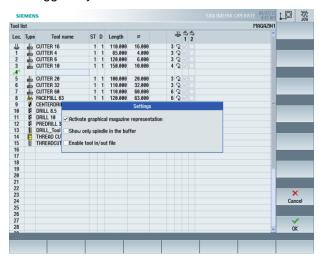
SI	EMEN	IS										SINUMERIK OPERATE 8:34 AP	17	ĮΩ	200 J00
lool I	ist											Magazin	1		
Loc.	Туре	Tool name	ST	D	Length	ø			Ψ,	ನ 1	≂⊃ 2		^	_	_
Щ	*	CUTTER 16	1	1	110.000	16.000		3	ð	~			u		
1		CUTTER 4	1	1	65.000	4.000		3	ð	~			IE		
2	#	CUTTER 6	1	1	120.000	6.000			ð						
3		CUTTER 10	1	1	150.000	10.000		- 4	ð	~			L		
4													u		
5	the second	CUTTER 20	1	1	100.000	20.000		3	ð	~			h	_	_
6		CUTTER 32	1	1	110.000	32.000		3	2	~					
7		CUTTER 60	1	1	110.000	60.000		6	ð	~			U.		
8	1	FACEMILL 63	1	1	120.000	63.000		6	2	~					
9	V	CENTERDRILL 12	1	1	120.000	12.000	90.0		ð			-	١.		
10	Ň	DRILL 8.5	1	1	120.000	8.500	118.0		ñ	~					
11	Ň	DRILL 10	1	1	120.000	10.000	118.0		2	~			U.		
12	Ň	PREDRILL 30	1	1	120.000	30.000	180.0		2	~			h	_	_
13	Ň.	DRILL_Tool	1	1	110.000	25.000			ð	~			u		
14	E.	THREAD CUTTER	1	1	110.000	20.000		1	2	~			IF		
15	T.	THREADCUTTER M10	1	1	130.000	10.000	1.500		ð	~					
16													IL.		
17													u		
18													L.	_	_
19													4		
20													Ш		
21													1		
22													I.		
23													1	Mag	azine
24													I	sele	ction
25													ľ		
26															
27															=.
28															≣►
20							-					>	1		
8	To	ol Tool			2	Mag zin	a-	•	l	Jor	k	User	ſ	co 5	Setting
- 3	lis	st 🥑 wear			9	2 zin		Ø	0	ffs	et	R variable		SD 1	data

Set the graphic tool display in the tool list.

Go to the next level in the vertical softkey bar, and select the "Settings" softkey.

ool I	st											MAGAZINI	1		ŝ		
Loc.	Туре	Tool name	ST	D	Length	ø			Щ.	る 1	≂ 2	<u></u>		So	rt		
Щ.	#	CUTTER 16	1		110.000	16.000			2								
1		CUTTER 4		1	65.000	4.000			2								
2		CUTTER 6	1		120.000	6.000			2					Filt	er		
3		CUTTER 10	1	1	150.000	10.000		- 4	2	~			L	_	_		
4	_																
5	the second se	CUTTER 20	1	1	100.000	20.000		3	2	~				_	-		
6		CUTTER 32	1	1	110.000	32.000		3	2	~				Sea	rch	b	
7		CUTTER 60	1	1	110.000	60.000		6	2	~						r .	
8	1	FACEMILL 63	1	1	120.000	63.000		6	2	~							
9	N	CENTERDRILL 12	1	1	120.000	12.000	90.0		a.				١				
10	Ň	DRILL 8.5	1	1	120.000	8.500	118.0		Q I	~							
11		DRILL 10	1	1	120.000	10.000	118.0		ົ								
12		PREDRILL 30	1	1	120.000	30.000	180.0		a l				le:	_	-		
13	Ň	DRILL Tool	1	1	110.000	25,000			ñ.								
14		THREAD CUTTER	1		110.000	20.000			à								
15		THREADCUTTER M10		1	130.000	10.000	1.500		2					Setti	ngs		
16	e			-												· ·	
17																	
18															-		
19																	
20													11				
21																	
22																	
23																	
24																	
25															-		
26																	
27																	
28																	
20												>	1			•	

Activate the "Activate graphical magazine representation" in the dialog window with the toggle key.



	st		- 1	5			Ī	Ţ	Ţ	10	, 1	SINUMERIK OP	ERATE 83/27/17 ESS 81 MAGAZINI	Sort
Loc.	Туре	Tool name	ST	D	Length	ø			Щ	う 1	≂ 2		^	
Ц	the second se	CUTTER 16	1	1	110.000	16.000		3	2	~				
1		CUTTER 4	1	1	65,000	4,000			2					Settings
2		CUTTER 6	1	1	120,000	6,000			ð					Setungs
3		CUTTER 10	1	1	150,000	10.000		4	2	V				
-4°									-					
5	#	CUTTER 20	1	1	100.000	20.000		3	2	7				
6		CUTTER 32	1	1	110.000	32,000			2					
7		CUTTER 60	i	1	110.000	60.000			2					
8		FACEMILL 63	1	1	120.000	63.000			2					
9	Ű.	CENTERDRILL 12	1	1	120.000	12.000	90.0			~				
10	ù.	DRILL 8.5	1	1	120.000	8.500	118.0			V				
11	ù.	DBILL 10	1	1	120.000	10.000	118.0			V				
12	ù	PREDRILL 30	1	1	120.000	30.000	180.0			V				
13	Ň	DRILL Tool	1	1	110.000	25.000				Ż				
14	÷.	THREAD CUTTER	i	1	110.000	20.000		1	ñ					.=
15	1	THREADCUTTER M10	1	1	130.000	10.000	1.500			~			~	×≣
8	To lis	ol 🙀 Tool				Mag zin	a-	۲		Wor		R User variable	>	SD Setting data

Result of the setting: The tools are represented above the tool list.

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				

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 +	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
(i) HED	Help function
END	Jump to end of line

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:					
NSERT	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.				
RISERT	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				