# Training Document for Integrated Automation Solutions Totally Integrated Automation (TIA)

Module S01

# Fundamentals of CNC Programming with SinuTrain

This document was prepared by Siemens AG for training purposes for the project 'Siemens Automation Cooperates with Education' (SCE).

Siemens AG does not guarantee the content of this document.

Passing on this document, copying it and using and sharing its content is allowed within public training facilities and continued education facilities. Exceptions require written approval by Siemens AG (Michael Knust michael.knust@siemens.com).

Offenders are held liable. All rights reserved, including translation rights, particularly in the event a patent is granted or a utility model or design is registered.

We thank the Michael Dziallas Engineering corporation, the teachers of vocational schools others for their support during the preparation of this document.

#### Table of Contents

1		Preface	5	
2		Introduction	7	
	2.1	Development Phases of CNC Technology	7	
	2.2	Controller Requirements for the New Millennium	7	
	2.3	Advantages of CNC Programming with SinuTrain SHOPMILL, SHOPTURN	7	
3		Programming a Turned Part in G-Code	8	
4		Programming a Turned Part according to the Machining Step Procedure	9	
	4.1	Starting SINUTRAIN	9	
	4.2	Selecting the Machine	10	
	4.3	SINUTRAIN Basic Display	10	
	4.4	Generating the Turning Program	11	
	4.5	Setting Up a New Directory/Program Generation	11	
	4.6	Setting Up a New Workpiece Program	12	
	4.7	Program Header	13	
	4.8	Program Editor	14	
	4.9	Tool Selection	14	
	4.10	Face Turning	15	
	4.11	Workpiece Contour	17	
	4.12	Stock Removal Cycle Roughing	22	
	4.13	Finishing the Outside Contour	23	
	4.14	Thread Undercut	23	
	4.15	Thread Cutting	24	
	4.16	Making Grooves	25	
	4.17	Simulation	26	
5		Programming a Milled Part in G-Code	28	
6		Programming a Milled Part according to the Machining Step Process	31	
6	6.1	Programming a Milled Part according to the Machining Step Process	<b>31</b> 32	
6	6.1 6.2	Programming a Milled Part according to the Machining Step Process Start SINUTRAIN Selecting the Machine	<b>31</b> 32 32	
6	6.1 6.2 6.3	Programming a Milled Part according to the Machining Step Process Start SINUTRAIN Selecting the Machine SINUTRAIN Basic Display	<b>31</b> 32 32 32	
6	6.1 6.2 6.3 6.4	Programming a Milled Part according to the Machining Step Process Start SINUTRAIN Selecting the Machine SINUTRAIN Basic Display Setting Up a New Directory/Program Generation	31 32 32 32 33	
6	6.1 6.2 6.3 6.4 6.5	Programming a Milled Part according to the Machining Step Process Start SINUTRAIN Selecting the Machine SINUTRAIN Basic Display Setting Up a New Directory/Program Generation Setting Up a New Workpiece Program	32 32 32 32 33 33	
6	6.1 6.2 6.3 6.4 6.5 6.6	Programming a Milled Part according to the Machining Step Process Start SINUTRAIN Selecting the Machine SINUTRAIN Basic Display Setting Up a New Directory/Program Generation Setting Up a New Workpiece Program Program Header	32 32 32 33 34 35	
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7	Programming a Milled Part according to the Machining Step Process Start SINUTRAIN Selecting the Machine SINUTRAIN Basic Display Setting Up a New Directory/Program Generation Setting Up a New Workpiece Program Program Header Outside Contour	31 32 32 32 33 34 35 36	
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8	Programming a Milled Part according to the Machining Step Process Start SINUTRAIN	31 32 32 33 34 35 36 37	
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9	Programming a Milled Part according to the Machining Step Process Start SINUTRAIN Selecting the Machine SINUTRAIN Basic Display Setting Up a New Directory/Program Generation Setting Up a New Workpiece Program Program Header Outside Contour Roughing the Outside Contour Selecting the Tool	31 32 32 33 34 35 36 37 37	
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10	Programming a Milled Part according to the Machining Step Process Start SINUTRAIN Selecting the Machine SINUTRAIN Basic Display Setting Up a New Directory/Program Generation Setting Up a New Workpiece Program Program Header Outside Contour Roughing the Outside Contour Selecting the Tool Finishing the Outside Contour	31 32 32 33 34 35 36 37 37 38	
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10 6.11	Programming a Milled Part according to the Machining Step Process Start SINUTRAIN	31 32 32 33 34 35 36 37 37 38 pur	39
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10 6.11 6.12	Programming a Milled Part according to the Machining Step Process Start SINUTRAIN	31 32 32 33 34 35 36 37 37 38 pur 41	39
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10 6.11 6.12 6.13	Programming a Milled Part according to the Machining Step Process	32 32 32 33 34 35 36 37 37 38 our 41 42	39
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10 6.11 6.12 6.13 6.14 6.15	Programming a Milled Part according to the Machining Step Process	31 32 32 33 34 35 36 37 37 37 38 pur 41 42 43	39
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10 6.11 6.12 6.13 6.14 6.15 6.12	Programming a Milled Part according to the Machining Step Process	31 32 32 33 34 35 36 37 37 38 our 41 42 43 44	39
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10 6.11 6.12 6.13 6.14 6.15 6.16 6.17	Programming a Milled Part according to the Machining Step Process	31 32 32 33 34 35 36 37 37 37 38 our 41 42 43 43 44 45	39
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10 6.11 6.12 6.13 6.14 6.15 6.16 6.17 6.12	Programming a Milled Part according to the Machining Step Process	31 32 32 33 34 35 36 37 37 37 38 our 41 42 43 43 45 45	39
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10 6.11 6.12 6.13 6.14 6.15 6.16 6.17 6.18 6.19	Programming a Milled Part according to the Machining Step Process	31 32 32 33 34 35 36 37 37 38 our 41 42 43 44 45 45 46	39
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10 6.11 6.12 6.13 6.14 6.15 6.16 6.17 6.18 6.19 6.20	Programming a Milled Part according to the Machining Step Process         Start SINUTRAIN         Selecting the Machine         SINUTRAIN Basic Display         Setting Up a New Directory/Program Generation         Setting Up a New Workpiece Program         Program Header         Outside Contour         Roughing the Outside Contour         Selecting the Tool         Finishing the Outside Contour         Stock Removal, Residual Material and Finishing of Contour Pockets and Inside Contou         Stock Removal of Contour Pocket Roughing         Removing Residual Material from the Corners         Finishing the Base         Finishing the Edge         Roughing the Circular Pocket Diameter 60         Roughing the Circular Pocket Diameter 30         Finishing the Circular Pocket Diameter 30         Contoring and Drilling toking Obstructions into Account	31 32 32 33 34 35 36 37 37 37 38 our 41 42 43 44 45 45 46 46	39
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10 6.11 6.12 6.13 6.14 6.15 6.16 6.17 6.18 6.19 6.20 6.21	Programming a Milled Part according to the Machining Step Process         Start SINUTRAIN         Selecting the Machine         SINUTRAIN Basic Display         Setting Up a New Directory/Program Generation         Setting Up a New Workpiece Program         Program Header         Outside Contour         Roughing the Outside Contour         Selecting the Tool         Finishing the Outside Contour         Stock Removal, Residual Material and Finishing of Contour Pockets and Inside Contou         Stock Removal of Contour Pocket Roughing         Removing Residual Material from the Corners         Finishing the Base         Finishing the Edge         Roughing the Circular Pocket Diameter 60.         Finishing the Circular Pocket Diameter 30.         Finishing the Circular Pocket Diameter 30.         Centering and Drilling taking Obstructions into Account.	31 32 32 33 34 35 36 37 37 38 our 41 42 43 44 45 46 46 46	39
6	6.1 6.2 6.3 6.4 6.5 6.6 6.7 6.8 6.9 6.10 6.11 6.12 6.13 6.14 6.15 6.16 6.15 6.16 6.17 6.18 6.19 6.20 6.21 6.22	Programming a Milled Part according to the Machining Step Process         Start SINUTRAIN         Selecting the Machine         SINUTRAIN Basic Display         Setting Up a New Directory/Program Generation         Setting Up a New Workpiece Program         Program Header         Outside Contour         Roughing the Outside Contour         Selecting the Tool         Finishing the Outside Contour         Stock Removal, Residual Material and Finishing of Contour Pockets and Inside Contou         Stock Removal of Contour Pocket Roughing         Removing Residual Material from the Corners         Finishing the Edge         Roughing the Circular Pocket Diameter 60.         Finishing the Circular Pocket Diameter 30.         Finishing the Circular Pocket Diameter 30.         Finishing the Circular Pocket Diameter 30.         Centering and Drilling taking Obstructions into Account.         Machining Step Centering.	31 32 32 33 34 35 36 37 37 38 34 37 38 37 38 37 38 37 38 34 37 38 37 38 37 38 37 38 37 38 37 38 34 37 38 37 38 34 37 38 34 37 38	39
6	6.1         6.2         6.3         6.4         6.5         6.6         6.7         6.8         6.9         6.10         6.11         6.12         6.13         6.14         6.15         6.16         6.17         6.18         6.20         6.21         6.22         6.23	Programming a Milled Part according to the Machining Step Process         Start SINUTRAIN         Selecting the Machine         SINUTRAIN Basic Display         Setting Up a New Directory/Program Generation         Setting Up a New Workpiece Program         Program Header         Outside Contour         Roughing the Outside Contour         Selecting the Tool         Finishing the Outside Contour         Stock Removal, Residual Material and Finishing of Contour Pockets and Inside Contou         Stock Removal, Residual Material and Finishing of Contour Pockets and Inside Contou         Stock Removal of Contour Pocket Roughing         Removing Residual Material from the Corners         Finishing the Base         Finishing the Edge         Roughing the Circular Pocket Diameter 60         Finishing the Edge         Roughing the Circular Pocket Diameter 60         Finishing the Edge         Roughing the Circular Pocket Diameter 30         Finishing the Circular Pocket Diameter 30         Centering and Drilling taking Obstructions into Account         Machining Step Drilling         Machining Step Drilling	31 32 32 33 34 35 36 37 37 37 37 37 37 37 37 37 37 37 37 37 37 37 37 37 34 35 37 34 35 36 37 37 36 37 37 37 36 37 37 37 38 37 37 37 38 34 37 34 41 42 45 45 46 45 46 46 46 46 46	39
6	6.1         6.2         6.3         6.4         6.5         6.6         6.7         6.8         6.9         6.10         6.11         6.12         6.13         6.14         6.15         6.16         6.17         6.18         6.19         6.20         6.21         6.22         6.23         6.24	Programming a Milled Part according to the Machining Step Process         Start SINUTRAIN         Selecting the Machine         SINUTRAIN Basic Display         Setting Up a New Directory/Program Generation         Setting Up a New Workpiece Program         Program Header         Outside Contour         Roughing the Outside Contour         Selecting the Tool         Finishing the Outside Contour         Stock Removal, Residual Material and Finishing of Contour Pockets and Inside Contor         Stock Removal, Residual Material and Finishing of Contour Pockets and Inside Contor         Stock Removal of Contour Pocket Roughing         Removing Residual Material from the Corners         Finishing the Base         Finishing the Edge         Roughing the Circular Pocket Diameter 60         Finishing the Edge         Roughing the Circular Pocket Diameter 60         Finishing the Circular Pocket Diameter 30         Finishing the Circular Pocket Diameter 30         Finishing the Circular Pocket Diameter 30         Centering and Drilling taking Obstructions into Account         Machining Step Drilling         Inputting the Line of Holes to the Left         Eirer Obstruction	31 32 32 33 34 35 36 37 37 38 our 41 42 43 44 45 46 46 46 46 46 47 47	39

6.25	Inputting the Line of Holes to the Right	48
6.26	Second Obstruction	48
6.27	Drilling the Full Circle	.48
6.28	Third Obstruction	48
6.29	Drilling a Single Hole	48
6.30	Simulation	49
7	Additional Tasks for Turning	50
7.1	Step Shaft	50
7.2	Shaft02	51
8	Additional Tasks Milling	52
8.1	Flange	52
8.2	Exercise11	53

#### 1 Preface

This training document **'Fundamentals of CNC Programming'** is the prerequisite for working with the additional modules with SINUTRAIN.



Today, CNC controllers are considered the most essential part of any automation. Depending on the problem definition, the most varied tasks in the areas of turning, milling, lasering, and grinding as well as in many other areas can be carried out economically with the controllers ShopMill and ShopTurn.

#### Training objective:

The CNC S01module is structured in a way that you have before you a completely programmed workpiece in G-code programming (DIN 66025).

The same workpiece is generated step by step with this document according to the machining step programming. In this way, you will learn the simple operation of this software, and control and optimize it with the simulation.

#### **Required Hardware and Software**

- 1 PC, operating system Windows XP Professional starting with SP1 with 500 MHz and 256 MB RAM, free disk storage approx. 400 MB, of that 50 MB on the system drive, 1GB for installing all products, MS Internet Explorer starting with 6.0
- 2 Software SINUTRAIN 802D/ 810D/840D/840Di/Programming & Training, SinuTrain/JopShop



#### 2 Introduction

#### 2.1 Development Phases of CNC Technology

- At the beginning of the eighties, first CNC machines with simple controllers

- In the middle of the eighties, more powerful controllers with cycles because of faster processors as well as machine tools with greater processing speed
- At the end of the eighties, machine tools with 5 and more axes and special software tools for external programming by using CAD/CAM systems

- At the beginning of the nineties, flexible manufacturing systems with extensive supplementary functions such as palette systems and multiple clamping with multiple spindle drives

- In the middle of the nineties, continued development of tool systems and the use of special tools for processing complex workpiece contours with only one tool
- Find of the pinetice, control programming systems for program

- End of the nineties: central programming systems for programming several different controllers at different machine tools

#### 2.2 Controller Requirements for the New Millennium

- Openness: It is to be possible for the machine manufacturer or the user to configure and expand controllers according to their own requirements

- Independence: Programming by means of a uniform controller interface for the most varied CNC processing

- Equality: All machine data is to be available also at the external programming units.

Programming at the external programming units is the same as on the machine tool.

- Saving programming time: With graphic machining plans and help displays, it is to be possible to generate complex workpiece contours very easily and quickly

- Editing capability: Extensive editing functions provide for fast and simple program changes/program expansion

#### 2.3 Advantages of CNC Programming with SinuTrain SHOPMILL, SHOPTURN

The controller is continuously optimized and can be adapted any time to the individual requirements of the machine manufacturers. Moreover, cycles and functions can be integrated later.

Regardless of whether turning, milling, or any other type of processing is performed, always the same program interface and the same menus or functions are used.

Retrofit: This means: Siemens can retrofit also older CNC machines to ShopMill and ShopTurn.

Advantage: Operating the software and the menu structure has to be learned only once.

By transferring the machine data to the programming system of SINUTRAIN, programming at the external programming unit is the same as on the machine tool.

By using contour calculators and CAD readers, simple programming is possible without technical terms. By directly entering technological values, no external calculations have to be made beforehand. The integrated contour calculator is able to process all conceivable dimensions, yet is very simple to handle. Through work step programming and many online help functions, extensive programming tasks can be solved very quickly. Convenient programming is possible with functions such as Copy, Cut, and Insert. Since the program is generated in the editor as a graphic machining plan by means of individual work steps, all editing steps are provided in a straightforward arrangement.

#### 3 Programming a Turned Part in G-Code

SIEMENS



WERKSTOECK_G_COD	30	KUNTUR	KONTOR_SCHL
T="SCHRUPPER_80"¶ G96 S250 LIMS=5000 M4 M8¶ G18 G54 G90¶ G0 X84 Z.2¶ G1 X-1.6¶ G0 X80 Z2¶ CYCLE95("Kontur",2.5,0.1,0.5,0.1,0.3,0.15,,1,0,0,1)¶ G0 X200 Z5 ¶ T="SCHLICHTER_35"¶ G96 S320 M4 M8¶ G0 X-1.8 Z1¶ G42¶ G1 X0 Z0¶ Kontur_Sch1¶		G18 G90 DIAMON¶ G1 X24 Z1¶ G1 X30 Z-2¶ G1 Z-20¶ G1 X40 RND=2.5¶ G1 X50 Z-30¶ G1 Z-44 RND=2.5¶ G1 X60 CHR=1¶ G1 Z-70 RND=1¶ G1 X66 RND=1¶ G1 Z-75 RND=1¶ G1 X76¶ G1 X82 Z-78¶ M17¶	G18 G90 DIAMON¶ G1 X0 Z0¶ G1 X26¶ G1 X26¶ G1 X20 Z-2¶ G1 Z-16¶ G1 Z-16¶ G1 Z-20 RND=.8¶ G1 Z-20 RND=.8¶ G1 X40 RND=2.5¶ G1 X50 Z-30¶ G1 Z-44 RND=2.5¶ G1 Z-44 RND=2.5¶ G1 Z-70 RND=1¶ G1 Z-70 RND=1¶ G1 Z-75 RND=1¶
G40¶ G0 X200 Z100¶ T="GEWINDESTAHL"¶			G1 X76¶ G1 X82 Z-78¶ M17¶
G95 M3 M8¶ G95 M3 M8¶ G0 X30 Z8¶ CYCLE97(1.5,,0,-17,30,30,3,2,0.92,0.03,30,0,8,2,1,1,0)¶ G0 X200 Z100 ¶ T="STECHER"¶ G96 S120 F.07 M4 M8¶ G0 X62 Z-56¶ CYCLE93(60,-56,4,3,0,0,0,-1,-1,0.1,0.1,0.1,0.1,3,0.2,11,0)¶ G0 X200 Z100¶ M30¶ ¶		Schrupper = roughing Schlichter = finishing t Gewindestahl = threac Stecher = grooving too	tool ool ling tool )l
==eof==			

#### 4 Programming a Turned Part according to the Machining Step Procedure

Using this drawing, we will generate a program in the machining step procedure.



#### 4.1 Starting SINUTRAIN



To start SinuTrain, double click on <sup>Start SinuTrain</sup>, or start SinuTrain using the start bar.

Dinternet Explorer N	/erknüpfung mit hopMill_6.4_pdf	ShopMill Dokumentation     ShopTurn Dokumentation     SinuTrain Dokumentation     Hilfe zur Maschinenkonfiguration
<ul> <li>Programmzugriff und -stand</li> <li>Windows Update</li> <li>Windows-Katalog</li> <li>SIMATIC</li> </ul>	dards yg mit , 4_pdf	Hino Edi Faschinomonioningi datori     Hinoweise zu SinuTrain     ReadMe Chinese     Start SinuTrain     Uninstall SinuTrain
🖬 Programme	🔸 🛗 SinuTrain Version 06.03	Edition 3 🔸 😜
🗋 Dokumente	🕨 🛅 Zubehör	F
📴 Einstellungen	🕨 🛅 DOConCD	8
🔎 Suchen	Adobe Acrobat 6.0 Prof	essional
(i) Hilfe und Support	Microsoft Office	E.
Ausführen	¥	
<ul> <li>"Heinz Nahlik" abmelden</li> <li>Computer ausschalten</li> </ul>	jutrainse 7	

#### 4.2 Selecting the Machine

After booting up the computer, a menu is displayed with a selection of machines with the respective version number.

- 1. Select turning machine with >> ShopTurn Open V 06.04 <<
- 2. Click the Start button



#### 4.3 SINUTRAIN Basic Display

After selecting the machine, the basic display of SINUTRAIN appears. From here, all SINUTRAIN actions are started.



#### Generating the Turning Program 4.4

#### 4.5 Setting Up a New Directory/Program Generation

Various actions can be performed in the Program Manager (for example: New, Rename, Copy, etc.)

	NC Pro-
ECT	🖳 gramm

the available directories are opened in the

By pressing the softkeys operating area **DIRECTORY** <<**VERZEICHNIS**>> under ShopTurn.

VER	ZEICHNIS						
	Name	Тур	Geladen	Größe	Datum/Zeit		
	SHOPTURN_UEBUNGEN	WPD	×	NCK-Dir.	27.05.2007	15:28	
	SIEMENS_SHOPTURN	WPD	×	NCK-Dir.	03.04.2007	15:47	
	TEMP	WPD	×	NCK-Dir.	03.04.2007	15:47	Neu
							Um- benennen
							Markieren
							Kopieren
							Einfügen
							Aus- schneiden
Fre	ier Speicher	Festp	latte:	3.5 GBytes	NC: 15	79432	Weiteres
	NC 🏭 Disk A	USB 🖏	tte		ĺ		

Neu , we are now setting up a new directory with the name "Work pieces" With the softkey <<Werkstuecke>>.

VER	ZEICHNIS						
	Name	Тур	Geladen	Größe	Datum/Zeit		Ver-
	SHOPTURN_UEBUNGEN	WPD	×	NCK-Dir.	27.05.2007	15:28	zeichnis
	SIEMENS_SHOPTURN	WPD	×	NCK-Dir.	03.04.2007	15:47	-
	TEMP	WPD	×	NCK-Dir.	03.04.2007	15:47	
	Neues Verzeichnis Bitte geben Sie d	len nei	uen Namer	n ein:			
	WERKSTUECI	(E <mark>)</mark>					
							Abbruch
Fre	ier Speicher	Fest	olatte:	3.5 GBytes	NC: 15	79432	ок

Confirm the input with **OK**.

The directory with the name "Work pieces" is set up.

VER	ZEICHNIS							
	Name	Тур	Geladen	Größe	Datum/Zeit			
	SHOPTURN_UEBUNGEN	WPD	×	NCK-Dir.	27.05.2007	15:28		
	SIEMENS_SHOPTURN	WPD	х	NCK-Dir.	03.04.2007	15:47		
	TEMP	WPD	×	NCK-Dir.	03.04.2007	15:47	Neu	
	WERKSTUECKE	WPD	×	NCK-Dir.	23.07.2007	09:46		
							Um- benennen	
							Markieren	
							P	
							Kopieren	
							Einfügen	
							wittebw	
							Aus-	
		-				70.000	END	
Fre	ler Speicher	rest	olatte:	3.5 uBytes	NC: 15	79432	Weiteres	

By pressing the arrow key on the CNC keyboard or with the cursor key toward the right, the selected directory opens.

#### Note

The cursor keys can be used for navigating in SinuTrain.

#### 4.6 Setting Up a New Workpiece Program

After the directory is opened, existing programs are displayed, or new programs can be set up.

By pressing the softkey and the selection Shopturn or G-Code (vertical softkeys), a new program name "Name of the machining plan" is entered, in our case "Shaft" <<"Welle">>.

VERZ	ΈIC	HNIS							
	Nam	e		Тур	Geladen	Größe	Datum/Z	eit	
t)	WER	KSTUECKE.	WPD\						
									ShopTurn Programm
									G-Code Programm
	Ne	ues ShopT	urn Program	m		<u>-</u>			
		Bitte	geben Sie d Welle <mark>e</mark>	en neu	ien Namen	ein:			
									Abbruch
Fre:	ier	Speicher		Festp	latte:	3.5 GByte	s NC:	1579432	ок

Confirm input with **OK**.

#### 4.7 Program Header

The program header of the new program opens automatically. In the program header, the following is entered: the workpiece data, zero shift, raw part pattern, distance to the chuck, retraction planes, tool change point, safety clearance incremental or absolute, and the spindle speed limitation.

With the softkey you can now call the associated help display.



Make all inputs as indicated in the display above

()

and accept Ubernahme the input into the machining plan.

#### Note

With the softkey **Alternat.** you can alternate, for example, between different zero points, units of mass, mm, inch, raw part shapes, cylinder, pipe, rectangle, corner, etc. The blue line above always shows you the current input type.

#### 4.8 **Program Editor**

Entered/edited in the program editor in the form of machining steps.

PRC	)gramm		
WEL	NS WELLE		Werkzeug
END	Programmende	N=1	Gerade

#### 4.9 Tool Selection

First, you need a tool for face turning and for roughing the outside contour.

Click on		
PROGRAMM		
WELLE	Werkzeugn	ame
P	Werkzeug	
	т	D1
END	S4 U/min	Werkzeuge
	Ebenenanwahl:	

Then, by clicking on "Tools" again, call the tool list. Here, the tools that have already been set up are provided. But you can also create new ones.

Accept the roughing tool. The finishing tool and the threading tool are added later from the tool list. The grooving tool has to be set up, however.

Click	or	Werkzeuge		F	ROUGH	IING T	OOL		ins Progr	s amm
Werkz	eug	liste								U
P1.	Тур	Werkzeugname	DP :	L. Schne Länge XI	eide Länge Z	Radius	0	1	ੀat. 12 late	Alternat.
1		SCHRUPPER	1	0.000	0.000	0.800	<b>€</b> 93.0	55	11.0Q	Ins Programm
2	Ø	SCHLICHTER	1	0.000	0.000	0.000	<b>← 93.</b> 0	55	11.02	Herkzeug
3									121-1-	löschen
4	622	BOHRER	1	0.000	0.000	10.000	118.0		5	
5										Entladen
7	۵	GEWINDESTAHL	1	0.000	0.000	0.000			2	
8										
9										
10										Schneiden
11										
12										Sortieren
13										
14										
	erkz .ist	e Werkz. versch		Ma z	ga- in 💽	Nullp. versch	R R-Pa	ra er		

Now, enter the data for the constant cutting speed and the plane.

PROGRAMM		
WELLEHEINZ	Mantel/Stirn/Drehen	0
P	Werkzeug	Alternat.
т	T SCHRUPPER D1	
	V1 200 m/min	Werkzeuge
END	Ebenenanwahl: Drehen	

Accept the values with the input key.

If needed, change the unit or selec	ction with	O Alternat.
and finally confirm your input with	Übernał	nme

The roughing tool is included in the program, with all the data.

#### 4.10 Face Turning

Positioning for face turning in rapid feed. By using line/circle, we can program a travel movement.



Next, face the front of the workpiece to X-1.6 (2 x plate radius size R=0.8) so no residual material remains in the center.





Now, position on X90 Z5 at rapid feed.

After accepting, you are back in the program.

PRC	PROGRAMM						
WEL	LE						
Р	N5	WELLE	Nullpktv. 1 G54				
Т	N10	Drehen	T=SCHRUPPER V1=200m				
	N15	EILG. X81 Z0.1					
$\rightarrow$	N20	F0.2/U X-1.6					
	N25	EILG. X90 Z5					

#### 4.11 Workpiece Contour

The contour is generated with the contour calculator.

📕 Kontur



Click on the button below and then in the upper right on Assign a name to the new contour; then the contour calculator opens.

Neue Kontur	
Bitte geben Sie den neuen Namen ein	r:
Welle_Kontur <mark>_</mark>	

The first input specifies the starting point.



Here, the first diameter of the finished part contour is described.

A chamfer 2 x 45 degrees as transition element is appended to the end of the element.







Click on the button and enter X82 and FS3.

The raw part is diameter 80 in the program and is run to 82, in order to gain distance from the tool. For that reason, the chamfer size has to be programmed 1mm larger.





The finished part contour has now been described, and is accepted into the program with



After that, you are back in the program editor.

PRC	OGRAMM		
WEL	.LB		
Р	N5 WELLE	Nullpktv. 1 G54	
Т	N10 Drehen	T=SCHRUPPER V1=200m	
$\rightarrow$	N15 EILG. X81 Z0.1	L	
$\rightarrow$	N20 F0.2/U X-1.6		
$\rightarrow$	N25 EILG. X90 Z5		
٦.	N30 WELLE_KONTUR		

#### 4.12 Stock Removal Cycle Roughing

Click on the button

The stock removal of a contour is performed in the area Contouring and Stock Removal. Using the "Help Key" brings up the help display.



Abspanen

Select the tool and enter the cutting data. Accept all other data in the picture.



Here, you should test the program with a simulation display.



PROGRAMM Seite sich  $\bigcirc$ -50 0 50 Details -50.000 Z WELLE X N5 100.000 Eilgang T=SCHRUPPER\_80 2:42:05 D1 Ende Simulation Gerade Kreis 🛃 Drehe Kontur drehen Diver Abar-beiter 8 - Bohren 📻 Fräsen

Clicking on the button "End" takes you back to the program editor.

#### 4.13 Finishing the Outside Contour

The outside contour is executed with the finishing tool contour parallel.



Accept all other data from the figure.



#### 4.14 Thread Undercut

We incorporate the undercut DIN 76



We can see here that the contour during roughing and finishing is connected with stock removal. The thread undercut was generated with its own cycle.

<u>ان</u>	N30	WELLE_KONTUR		
1	N35	Abspanen	$\nabla$	T=SCHRUPPER F0.3/U V250m
۲	N40	Abspanen 🗸 🗸	$\overline{\mathcal{M}}$	T=SCHLICHTER F0.1/U V250m
	N45	Freistich Gew. 🖓+	+777	T=SCHLICHTER F0.1/U V120m X0=30
END		Programmende		N=1

Here, you can test your program with the simulation display.



#### 4.15 Thread Cutting



A thread of M30 X 1.5 is to be manufactured.

#### 4.16 Making Grooves

Finally, we are making two grooves.





Exit menu with



We are done with our machining step program.

1				Werkzeug
N5	WELLE		Nullpktv. 1 G54	, in the second s
N10	Drehen		T=SCHRUPPER V1=200m	
N15	EILG. X81 Z	9.1		Gerade
N20	F0.2/U X-1.0	3		
N25	EILG. X90 Z	5		Kreis Mittelp
7 N30	WELLE_KONTUR			
- N35	Abspanen	$\nabla$	T=SCHRUPPER F0.3/U V250m	Kreis
J N40	Abspanen	$\overline{\mathbf{AAA}}$	T=SCHLICHTER F0.1/U V250m	Radius
N45	Freistich Gew.	<b>∀+</b> ₩	T=SCHLICHTER F0.1/U V120m X0=30	
N50	Gewinde Längs	∀+∀∀∀	T=GEWINDESTAHL P1.5mm S1000U Außen	Polar
N55	Einstich	∇+∇∇∇	T=STECHER F0.1/U V120m N0 X0=60	1997 19
	Programmende		N=1	Anfahren Abfahrer

#### 4.17 Simulation

Finally, test your program with a simulation display. First in the side view.



Next in the 3 window view



and in the volume model.

PROGRAMM		
Volumenmodell		Details
X -50.000 Z 100.000	Eilgang 3:01:44	B
N5 WELLE	T=STECHER_3 A D:	L Ende
Gerade	Fräsen Diver- Simu:	la 🛛 Abar-
Kreis Bohren 🛃 Drehen 💕 Kontur	ses tior	beiten

#### Using **EXIT**, we exit the ShopTurn software.

Gerade Kreis <b>E</b> - Bohrer	Drehen 💕 Kontur	<b>r</b> - Fräsen	Diver-	Simula tion	Abar- beiten
MACHINE				ME	
EXIT	RESET BLOCK				(i) HELP

#### 5 Programming a Milled Part in G-Code



FORMPLATTE G CODE 3 N1 G0 G90 G40 G17 N2 T="PLANFRAESER63" N3 M61 N4 G94 S400 F300 M3 M81 N5 G54 G0 X-20 Y-135 Z3 ¶ N8 CYCLE71(3,2,2,0,-50,-100,100,150,0,1.8,55,2,0.2,300,22,) N10 G0 Z1501 N11 GØ X200 Y2001 N12 T="FRAESER32" N13 M6¶ N14 G94 S1000 M3 M8 F2001 N15 GØ X-51 Y-112 Z1¶ N16 GØ Z-101 N18 G17 G90 DIAMOF ;\*GP\*1 N19 G1 X-35 Y-100 G41 ;\*GP\*¶ N20 Y35 RND=15 ;\*GP\*1 N21 X35 RND=15 ;\*GP\*¶ N22 Y-120 ;\*GP\*1 N23 G40 1 N32 G0 Z100 N33 X200 Y150 N34 T="FRAESER10" N35 M6 M81 N36 G94 S2000 M3 M8 F2001 N37 GØ XØ Y-84 Z1¶ N38 G1 Z-15 ¶ N40 G17 G90 G41 DIAMOF ;\*GP\*¶ N41 G1 X0 Y-90 :\*GP\*¶ N42 X30 RND=5 ;\*GP\*1 N43 Y-19.9 RND=5 ;\*GP\*¶ N44 G2 X-30 I=AC(0) J=AC(0) RND=5 ;\*GP\*1 N45 G1 Y-90 RND=5 ;\*GP\*¶ N46 X0 ;\*GP\*1 N47 G401 N48 GØ Y-841 N49 GØ Z1¶ N64 \_ZSD[2]=1 ;\*R0\*¶ N65 POCKET3(,0,1,-15,60,54,5,-30,-90,0,5,0.2,0.2,200,100,0,31,6,,,, 12,)¶ N70 ZSD[2]=1 ;\*R0\*[ N71 POCKET3(2,0,1,-15,60,54,5,-30,-90,0,5,0.2,0.2,200,100,0,32,6,,, ,12,)¶ N73 GØ Z1001 N74 GØ X200 Y150¶ N75 T="FRAESER20"1 N76 G94 S1000 M3 M8 F2001 N79 POCKET4(2,0,1,-10,30,0,0,5,0.2,0.2,200,100,0,21,10,,,12,5) N83 POCKET4(2,0,1,-10,30,0,0,5,0.2,0.2,200,100,0,22,10,,,12,5) N87 POCKET4(2,-10,1,-21,15,0,0,5,0.2,0,200,100,0,21,10,.,12,5) N91 POCKET4(2,-10,1,-21,15,0,0,5,0.2,0,200,100,0,22,10,,,12,5) N93 GØ Z1001 N94 GØ X200 Y150 N95 T="ZENTRIERER12" N96 G94 S2000 M3 M8 F2001 N97 GØ Z21 N100 MCALL CYCLE82(2,0,2,-15.3,,0.1)

N104 HOLES1(-42.5,-92.5,90,0,45,4) N106 GØ XØ Y42.5¶ N109 HOLES1(42.5,42.5,-90,0,45,4)¶ N111 MCALL¶ N114 GØ Z1¶ N118 MCALL CYCLE82(2,0,2,-15.3,,0.1)¶ N122 HOLES2(0,0,22.5,0,60,6)¶ N124 MCALL N125 GØ Z100¶ N126 GØ X200 Y150¶ N127 T="BOHRER10" N128 G94 S2000 M3 M8 F2001 N131 MCALL CYCLE83(2,0,1,-24,,-15,,5,,3,1,1,3,10,,,1) N135 HOLES1(-42.5,-92.5,90,0,45,4) N137 GØ XØ Y42.5¶ N140 HOLES1(42.5,42.5,-90,0,45,4)¶ N142 MCALL N143 GØ Z21 N146 MCALL CYCLE83(2,0,1,-24,,-15,,5,,3,1,1,3,10,,,1) N150 HOLES2(0,0,22.5,0,60,6)¶ N152 MCALL¶ N153 GØ Z100¶ N154 GØ X200 Y150 N155 M301

PLANFRAESER = facing tool FRAESER = milling tool ZENTRIERER = centering tool

#### 6 Programming a Milled Part according to the Machining Step Process



#### 6.1 Start SINUTRAIN



To start SinuTrain, double click on Start SinuTrain or start SinuTrain using the start bar.

#### 6.2 Selecting the Machine

After you booted the computer, a menu is displayed with the selection of machines with the respective version number.

- 1. Select milling machine >> ShopMill Open V 06.04 <<.
- 2. Click on the Start button.

on: Ub.U3 Edition 3 [15.12.2005 ] aschinenauswahl	JUL
1.Sprache: deutsch - 2.Sprache: english Fräsmaschine mit>> ShopMill Open V 06.04 <<	
Fräsmaschine mit>> ShopMill Open V 06:02 <<	~
Fräsmaschine mit>> ShopMill Open V 06.03 <<	
Fräsmaschine mit>> ShopMill Open V 06.04 <<	
Drehmaschine mit>> ShopTurn Open V 06.03 <<	
Drehmaschine mit ->> ShopTurn Open V 06.04 <<	
Drehmaschine mit ->> ManualTurn V06.02 <<	
Fräsmaschine SINUMERIK 802D	
Drehmaschine SINUMERIK 802D	
st0604_GY	
st0604_Y	*
🦳 Maschine im Vollbildmodus anzeigen	
Start	Abbruck

#### 6.3 SINUTRAIN Basic Display

After you selected the machine, the basic display of SINUTRAIN appears. All SINUTRAIN actions are started from here.



#### 6.4 Setting Up a New Directory/Program Generation

Different actions can be performed in the program manager.



, the available directories are opened under

By clicking on the softkeys SELECT ShopMill, in the operating area DIRECTORY

Name	Тур	Geladen	Größe	Datum/Zeit		
BEISPIELPROGRAMME	WPD	×	NCK-Dir.	02.03.2007	19:36	
CAD_PROGRAM	WPD		NCK-Dir.	11.02.2007	15:14	19 17
SHOPMILL	WPD	×	NCK-Dir.	28.03.2007	15:08	Neu
SIEMENS_SINUTRAIN	WPD	x	NCK-Dir.	27.05.2007	16:04	
ТЕМР	WPD	×	NCK-Dir.	28.03.2007	15:09	Um- benenne
						Markier Kopiere Einfüge
						Aus-
reier Speicher 또 NC 한다 Disk A 한다 ।	Festp JSB	olatte:	3.3 GBytes	NC: 12	49176	Aus- schneid Weitere
reier Speicher NC 행용 Disk 4 행용 U Vith the softkey Pattern plate".	Festp JSB EU	we are	3.3 GBytes	NC: 12	:49176 w direc	Aus- schneid Weitere
reier Speicher ■ NC 원을 Disk A 원을 U Vith the softkey Pattern plate". ERZEICHNIS	Festp JSB EU	we are	3.3 GBytes	NC: 12	249176 w direc	Aus- schneid Weitere
reier Speicher NC الله Disk A الله الله Vith the softkey Pattern plate". ERZEICHNIS	Festp JSB EU	We are	3.3 GBytes e now setti Größe	NC: 12 ng up a new Datum/Zeit	49176 w dired	Aus- schneid Weitere

	SHOPMILL SIEMENS_SINUTRAIN TEMP	WPD WPD	x x	NCK-Dir.	28.03.2007	15:08 16:04	
	SHOPMILL SIEMENS_SINUTRAIN TEMP	WPD	× ×	NCK-Dir.	28.03.2007	15:08 16:04	
	SIEMENS_SINUTRAIN	WPD	x	NCK-Dir.	27.05.2007	16:04	
]	ТЕМР	LIDD					Transmission and the second
		WPD	×	NCK-Dir.	28.03.2007	15:09	
	Bitte geben Sie Formpla	tte <b>l</b>	n Name	n ein:			Abbru

Confirm the input with **OK**.

The selected directory opens by pressing the arrow key **I** on the CNC keyboard, or with the cursor key toward the right.

VER	ZEICHNIS						
	Name	Тур	Geladen	Größe	Datum/Zeit		
	BEISPIELPROGRAMME	WPD	×	NCK-Dir.	02.03.2007	19:36	
	CAD_PROGRAM	WPD		NCK-Dir.	11.02.2007	15:14	-
	FORMPLATTE	WPD	×	NCK-Dir.	02.09.2007	16:55	Neu
	SHOPMILL	WPD	×	NCK-Dir.	28.03.2007	15:08	
	SIEMENS_SINUTRAIN	WPD	×	NCK-Dir.	27.05.2007	16:04	Um- benennen
	ТЕМР	WPD	×	NCK-Dir.	28.03.2007	15:09	
							Markieren Kopieren
							Einfügen
							Aus- schneiden
Fre	ier Speicher	Festp	latte:	3.3 GBytes	NC: 12	25624	Weiteres

#### 6.5 Setting Up a New Workpiece Program

After opening the directory, existing programs are displayed, or new programs can be set up.

By pressing the softkey and selecting ShopMill or G-Code (vertical softkeys), a new program name "Name of the work plan" is entered; in our case "Pattern plate".

VERZ	EICHNIS	;							
	Name			Тур	Geladen	Größe	Datum/2	Zeit	
2	FORMPLA	TTE.WPD	N						
									ShopMill Programm
									G-Code Programm
	Neues	ShopMi.	1 Program	m					4
	E	litte ge	ben Sie d	en neu	len Namer	n ein:			
			Formplatte						
									<b>X</b> Abbruch
Frei	ler Spej	cher		Festp	latte:	3.3 GByte	s NC:	1249176	OK

Confirm the input with **OK**.

#### 6.6 Program Header

With the softkey

The program header of the new program opens automatically. The following is entered in the program header: the workpiece data, zero shift, raw part pattern, distance to the chuck, retraction planes, tool change point, safety clearance incremental or absolute, and the spindle speed limitation.



you can now call the associated help display.



Enter all data as in the figure above

and accept the inputs in the machining plan with

 $\odot$ 



#### Note

With the softkey **Alternat.** you can alternate, for example, between different zero points and measurement units, mm, inch. The blue line above always shows the current input type.

#### 6.7 Outside Contour

The contour is generated with the contour calculator.

Click on the button	on Neue Kontu
Assign a name to the new contour; after that, the contour calculator ope	ens.
Neue Kontur	
Bitte geben Sie den neuen Namen ein: FORMPLATTE_Aussen	

The first input specifies the starting point.



Complete programming the outside contour with the contour calculator.



The outside contour is now described, and is accepted into the program with

#### 6.8 Roughing the Outside Contour

To perform the machining step roughing on the outside, with contour compensation on the left, do the following:



#### 6.9 Selecting the Tool

Here the tools that have already been set up are provided. But you can also generate new ones. Here, take the MillingTool32.



Click on Programm to accept the selected tool.

FORMF	LATTE					Robe for	Anfahrmod	us Ebene	O
P						Bann+r	asen		Alternat.
$\sim$						TFR	AESER32	D1	
	-					F	0.150	mm/Zahn	Werkzeuge
- 1						V.	120	m/min	
ND	50-					Radius	korrektur	: Öl	
		1	<del></del>	<u></u>		VO	rvärts		
						Bearbe	itung:	V	
	<u>и</u> -					ZØ	0.000	abs	1
						Z1	10.000	ink	
						DZ	5.000		A
	-50-						0.300		
						UNT	0.3001	171171	
						Anfahr	en:Gerade		
	-100-			4		L1	5.000	502.1	
	-					FZ	0.100	mm/Zahn	
	-169-					Abfahr	en:Gerade	t	
	Y -10	a -50	ø	50	100	L2	5.000		×
	Ъx					1 11 11			Abbruch
						Abhebe	modus:		
						auf Rü	ickzugsebei	ne	$\checkmark$
	86	2004	10		-		101	i	Übernahme
🚽 G	erade 🕴 👝			Kontu	C	iver-		Simu-	NC Abar-

Accept all data from the graphic and exit the menu with



#### 6.10 Finishing the Outside Contour

To perform the machining step Finishing Outside, with contour compensation to the left, do the following:

Click on Bahnfräsen and er

and enter the values.



Accept all values from the figure, and exit the menu with



Here, you can test your program with the simulation display.

# 6.11 Stock Removal, Residual Material and Finishing of Contour Pockets and Inside Contour

The contour is generated with the contour calculator.

FORMPLATTE\_Innen



The first input specifies the starting point.



With the contour calculator, complete programming the inside contour.



The inside contour is now described and is accepted into the program with

#### 6.12 Stock Removal of Contour Pocket Roughing



```
n and enter the values.
```

Select the tool and enter the cutting data. Accept all additional data from the figure.



#### 6.13 Removing Residual Material from the Corners



Tasche Restmat.

and enter the values.

Select the tool and enter the cutting data.

Accept all additional data from the figure.



#### 6.14 Finishing the Base



Tasche fräsen

and enter the values.

Select the tool and enter the cutting data.

Accept all additional data from the picture.



#### 6.15 Finishing the Edge





and enter the values.

Select the tool and enter the cutting data.

Accept all additional data from the picture.



Click on

and enter the values.

#### 6.16 Roughing the Circular Pocket Diameter 60



Select the tool and enter the cutting data.

Accept all additional data from the picture.



#### 6.17 Finishing the Circular Pocket Diameter 60



Select the tool and enter the cutting data. Accept all additional data from the picture.



#### 6.18 Roughing the Circular Pocket Diameter 30

Rough the circular pocket yourself. Important: The starting depth is no longer at "0" but at "-10mm"

#### 6.19 Finishing the Circular Pocket Diameter 30

Rough the circular pocket yourself. Important: The starting depth is no longer at "0" but at "-10mm"

#### 6.20 Centering and Drilling taking Obstructions into Account

For centering and drilling, first the corresponding machining steps are programmed, and then the individual positions are specified. The obstructions have to be taken into account when positioning.

#### 6.21 Machining Step Centering





and enter the values.

Select the tool and enter the cutting data. Accept all additional data from the picture.





#### 6.22 Machining Step Drilling



#### 6.23 Inputting the Line of Holes to the Left



#### 6.24 First Obstruction

Click on And enter the Z value.



#### 6.25 Inputting the Line of Holes to the Right



#### 6.27 Drilling the Full Circle



#### 6.28 Third Obstruction





#### 6.30 Simulation

Finally, test your program by using the different simulation displays.



#### 7 Additional Tasks for Turning

#### 7.1 Step Shaft



### 7.2 Shaft02

SIEMENS



#### 8 Additional Tasks Milling

#### 8.1 Flange



#### 8.2 Exercise11

SIEMENS

