

SIEMENS

SINUMERIK 840D/810D ShopMill

Operation/Programming

11.98 Edition

User Documentation

SIEMENS

**SINUMERIK 840D/810D
ShopMill**

Operation/Programming

User Documentation

Valid for

<i>Control</i>	<i>Software Version</i>
SINUMERIK 810D/DE	2
SINUMERIK 840D/DE	4

11.98 Edition

Introduction	1
Operation	2
Programming	3
Simulation	4
Alarms and Messages	5
Appendix	A

Contents

Introduction

1.1 The ShopMill product	1-17
1.2 Handling instructions	1-17
1.3 Operating components	1-18
1.3.1 OP031 slimline operator panel	1-18
1.3.2 OP032S slimline operator panel	1-19
1.3.3 Important function keys	1-20
1.4 Elements on machine control panel	1-21
1.4.1 Emergency STOP	1-21
1.4.2 Operating modes	1-22
1.4.3 Machine functions	1-22
1.4.4 Feed control	1-23
1.4.5 Spindle control	1-23
1.4.6 Keyswitch	1-24
1.4.7 Program control	1-25
1.5 Graphics interface	1-26
1.6 Fundamentals	1-27
1.6.1 Rectangular coordinate system	1-27
1.6.2 Plane designations	1-28
1.6.3 Polar coordinates	1-29
1.6.4 Absolute dimension	1-29
1.6.5 Incremental dimension	1-30
1.6.6 Important softkeys for operation and programming	1-30
1.6.7 Pocket calculator function	1-31
1.6.8 Inch/metric switchover	1-31

Operation

2.1 Power ON and reference point approach	2-35
2.2 Manual mode and settings for manual mode	2-38
2.2.1 Traversing the machine axes	2-38
2.2.2 Load tool from list into spindle	2-39
2.2.3 Entering a new tool in the list and load to spindle	2-40
2.2.4 Entering a new tool in the list and load to magazine	2-41
2.2.5 Start, stop and position spindle manually	2-41
2.2.6 Machine-specific functions	2-43
2.2.7 Switch over machining plane / tool axis	2-43
2.2.8 Switch over to mm or inches	2-44
2.2.9 Switch over between machine and workpiece coordinate systems	2-44
2.3 Set new position value (basic offset)	2-45
2.4 Set up workpiece	2-46
2.4.1 Accepting a zero offset from the zero offset list	2-46

2.4.2 Manual setup with edge probe.....	2-47
2.4.3 Manual setup with 3D probe.....	2-48
2.4.4 Calibrate workpiece probe.....	2-52
2.5 Measuring tools in Manual area	2-54
2.5.1 Accept tool from the tool list.....	2-54
2.5.2 Measure tools using "scratching" method	2-54
2.5.3 Measure tools with a tool probe.....	2-55
2.5.4 Calibrate tool probe.....	2-56
2.6 Machining in Manual mode	2-57
2.6.1 Change settings.....	2-57
2.6.2 Positioning	2-57
2.6.3 Face milling	2-58
2.7 MDA mode	2-59
2.8 Automatic mode.....	2-60
2.8.1 Switchover between "T, F, S", "G functions" and "auxiliary functions" displays.....	2-60
2.8.2 Select a program for execution	2-61
2.8.3 Start / stop / abort program	2-61
2.8.4 Interrupt program.....	2-62
2.8.5 Enter a program at a random point.....	2-63
2.8.6 Program control.....	2-64
2.8.7 Simulation before machining.....	2-65
2.8.8 Simulation during machining.....	2-66
2.9 Execute a trial program run.....	2-67
2.9.1 Single block	2-67
2.9.2 Correct program.....	2-68
2.10 File management.....	2-69
2.10.1 Select a file	2-69
2.10.2 Copy/rename a file.....	2-70
2.10.3 Delete files.....	2-70
2.10.4 Read out files.....	2-70
2.10.5 Read in files.....	2-71
2.10.6 Error log.....	2-72
2.11 Tools and tool offsets.....	2-73
2.11.1 Select tool list	2-75
2.11.2 Create a new tool.....	2-76
2.11.3 Create tool offset blocks for tool edge1/ 2	2-77
2.11.4 Change tool name.....	2-77
2.11.5 Create a replacement tool	2-78
2.11.6 Manual tools	2-78
2.11.7 Tool offsets	2-79
2.11.8 Special tool functions.....	2-82
2.11.9 Create tool wear data.....	2-83
2.11.10 Tool monitoring	2-84
2.11.11 Magazine list.....	2-85

2.11.12 Delete tool.....	2-86
2.11.13 Change tool type	2-86
2.11.14 Load a tool	2-86
2.11.15 Unload a tool.....	2-87
2.11.16 Sorting tools in the tool list	2-88
2.12 Zero offsets.....	2-89
2.12.1 General	2-89
2.12.2 Display zero offset	2-90
2.12.3 Select/deselect settable zero offset in Manual area	2-92
2.13 Switch over to CNC operation	2-93

Programming

3.1 Fundamental programming principles	3-97
3.2 Program structure	3-100
3.2.1 Screen window in program	3-101
3.2.2 Toggle between screen windows	3-102
3.3 Create a program.....	3-103
3.3.1 Create a new program; define a blank.....	3-103
3.3.2 Program new blocks.....	3-106
3.3.3 Edit (change, delete, move, copy, insert) program blocks.....	3-108
3.4 Program tool, offset value and spindle speed.....	3-110
3.5 Mill a contour	3-111
3.5.1 Programming freely defined contours.....	3-112
3.5.2 Description of softkeys for free contour programming function	3-114
3.5.3 Description of parameters for line/circle contour elements.....	3-116
3.5.4 Programming examples for freely defined contours.....	3-117
3.5.5 Path milling of open and closed contours	3-119
3.5.6 Rough drilling in contour pockets.....	3-122
3.5.7 Machine (rough cut) pocket with islands	3-124
3.5.8 Remove residual material	3-126
3.5.9 Finish cut pocket with islands	3-127
3.6 Programming simple path motions.....	3-129
3.6.1 Line.....	3-129
3.6.2 Circle with known center point.....	3-130
3.6.3 Circle with known radius.....	3-131
3.6.4 Helix	3-132
3.6.5 Polar coordinates	3-133
3.6.6 Straight line polar.....	3-134
3.6.7 Circle polar	3-135
3.6.8 Programming examples for polar coordinates	3-136
3.7 Drilling	3-137
3.7.1 Centering	3-138
3.7.2 Drilling and reaming.....	3-139
3.7.3 Deep hole drilling	3-140

3.7.4 Boring	3-141
3.7.5 Tapping	3-142
3.7.6 Thread cutting.....	3-143
3.7.7 Machining positions, freely programmable or positioning patterns	3-146
3.7.8 Freely programmable positions	3-147
3.7.9 Positioning pattern Line.....	3-148
3.7.10 Positioning pattern Matrix.....	3-149
3.7.11 Positioning pattern Full Circle	3-150
3.7.12 Positioning pattern Pitch Circle	3-152
3.7.13 Repeat positions	3-153
3.7.14 Obstacle	3-154
3.7.15 Programming examples for drilling.....	3-155
3.8 Milling	3-157
3.8.1 Face milling	3-157
3.8.2 Mill a rectangular pocket	3-159
3.8.3 Mill a circular pocket	3-162
3.8.4 Milling rectangular spigots.....	3-164
3.8.5 Milling circular spigots.....	3-166
3.8.6 Milling longitudinal slots	3-168
3.8.7 Milling circumferential slots in full or partial circle.....	3-170
3.8.8 Use of positioning patterns for milling	3-171
3.9 Measurements	3-174
3.9.1 Workpiece measurements	3-174
3.9.2 Tool measurements	3-177
3.10 Miscellaneous functions	3-178
3.10.1 Subroutines and program section repeats	3-178
3.10.2 Altering the safety clearance or return plane	3-179
3.10.3 Zero offsets.....	3-180
3.11 G code programming	3-186
Simulation	
4.1 General.....	4-191
4.2 Start/abort program.....	4-191
4.3 Representation as plan view	4-192
4.4 Representation in the 3-plane view	4-193
4.5 Zoom a finished part viewport.....	4-194
4.6 Three-dimensional representation of finished part.....	4-195
4.6.1 Change position of finished part.....	4-195
4.6.2 Cut open finished part.....	4-196
4.6.3 Refresh finished part display	4-196
Alarms and Messages	
5.1 Cycle alarms and messages	5-199

5.1.1 Error treatment in machining cycles	5-199
5.1.2 Cycle alarm overview.....	5-199
5.1.3 Messages in cycles	5-202
5.2 Alarms in ShopMill.....	5-203
5.2.1 Overview of alarms.....	5-203
5.2.2 Select alarm/message overview.....	5-203
5.2.3 Description of alarms	5-204

Appendix

A Abbreviations	A-217
B Terms.....	A-219
C References.....	A-227