

SIEMENS

SINUMERIK 810D
Software Version 2
ShopMill

Operation/Programming

10.97 Edition

User Documentation

**SINUMERIK 810D
Software Version 2
ShopMill**

Operation/Programming

User Documentation

Applies to

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

10.97 Edition

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

Contents

Introduction

1.1 The ShopMill product	1-15
1.2 Handling instructions	1-15
1.3 Operating components	1-16
1.3.1 OP031 slimline operator panel	1-16
1.3.2 OP032S slimline operator panel	1-17
1.3.3 Important function keys	1-18
1.4 Elements on machine control panel	1-19
1.4.1 Emergency STOP	1-19
1.4.2 Operating modes	1-20
1.4.3 Machine functions	1-20
1.4.4 Feed control	1-21
1.4.5 Spindle control	1-21
1.4.6 Keyswitch	1-22
1.4.7 Program control	1-23
1.5 Graphics interface	1-24
1.6 Fundamentals	1-25
1.6.1 Rectangular coordinate system	1-25
1.6.2 Plane designations	1-26
1.6.3 Polar coordinates	1-27
1.6.4 Absolute dimension	1-27
1.6.5 Incremental dimension	1-28
1.6.6 Important softkeys for operation and programming	1-28
1.6.7 Pocket calculator function	1-29

Operation

2.1 Power ON and reference point approach	2-33
2.2 Manual mode and settings for manual mode	2-35
2.2.1 Traversing the machine axes	2-35
2.2.2 Load tool from list into spindle	2-37
2.2.3 Load a new tool into the spindle and enter it in the list	2-38
2.2.4 Load a new tool into the magazine and enter it in the list	2-38
2.2.5 Start, stop and position spindle manually	2-39
2.2.6 Machine-specific functions	2-40
2.2.7 Machining plane/switch over tool axis	2-41
2.2.8 Switch over to mm or inches	2-41
2.2.9 Switch over between machine and workpiece coordinate systems	2-41
2.3 Set new position value (basic offset)	2-42
2.4 Set up workpiece	2-43
2.4.1 Manual setup with edge probe	2-43
2.4.2 Manual setup with 3D probe	2-45
2.4.3 Calibrate workpiece probe	2-48

2.5 Measuring tools in Manual area	2-49
2.5.1 Measure tools using "scratching" method	2-49
2.5.2 Measure tools with a tool probe	2-50
2.5.3 Calibrate tool probe	2-51
2.6 Machining in Manual mode	2-52
2.6.1 Change settings	2-52
2.6.2 Positioning	2-53
2.6.3 Face milling	2-53
2.7 Automatic mode	2-54
2.7.1 Select a program for execution	2-55
2.7.2 Start / stop / abort program	2-55
2.7.3 Interrupt program	2-56
2.7.4 Enter a program at a random point	2-57
2.7.5 Program control	2-57
2.7.6 Program test/simultaneous recording (option)	2-59
2.8 Execute a trial program run	2-60
2.8.1 Single block	2-60
2.8.2 Correct program	2-61
2.9 File management	2-62
2.9.1 Select a file	2-62
2.9.2 Copy/rename a file	2-63
2.9.3 Delete files	2-63
2.9.4 Read out files	2-63
2.9.5 Read in files	2-64
2.9.6 Error log	2-65
2.10 Tools and tool offsets	2-65
2.10.1 Select tool list	2-66
2.10.2 Create a new tool	2-67
2.10.3 Create tool offset blocks for tool edge 1/ 2	2-68
2.10.4 Change tool name	2-68
2.10.5 Create a replacement tool	2-69
2.10.6 Tool offsets	2-69
2.10.7*Special functions for tool type	2-72
2.10.8 Create tool wear data	2-73
2.10.9 Delete a tool	2-75
2.10.10 Load a tool	2-75
2.10.11 Unload a tool	2-76
2.10.12 Sort magazine locations	2-76
2.10.13 Tool monitoring	2-77
2.11 Zero offset	2-78
2.11.1 General	2-78
2.11.2 Display zero offset	2-80
2.11.3 Select/deselect settable zero offset in Manual area	2-81
2.12 Switch over to CNC operation	2-82

Programming

3.1 Fundamental programming principles.....	3-85
3.2 Program structure	3-88
3.2.1 Screen window in program	3-89
3.2.2 Toggle between screen windows	3-90
3.3 Create a program.....	3-91
3.3.1 Create a new program; define a blank.....	3-91
3.3.2 Program new blocks	3-94
3.3.3 Edit (delete, move, copy, paste) program blocks.....	3-96
3.4 Program tool, offset value and spindle speed.....	3-98
3.5 Mill a contour.....	3-99
3.5.1 Programming freely defined contours.....	3-100
3.5.2 Description of softkeys for free contour programming function	3-102
3.5.3 Description of parameters for line/circle contour elements.....	3-104
3.5.4 Programming examples for freely defined contours	3-105
3.5.5 Path milling of open and closed contours	3-108
3.5.6 Machine (rough cut) pocket with islands.....	3-110
3.5.7 Remove residual material	3-112
3.5.8 Finish cut pocket with islands	3-113
3.6 Programming of simple positioning motions.....	3-115
3.6.1 Line	3-115
3.6.2 Circle with known centre point	3-116
3.6.3 Circle with known radius	3-117
3.6.4 Helix	3-118
3.6.5 Polar coordinates	3-119
3.6.6 Polar line	3-120
3.6.7 Polar circle	3-121
3.6.8 Programming examples for polar coordinates.....	3-122
3.7 Drilling	3-123
3.7.1 Centering	3-124
3.7.2 Drilling and reaming.....	3-125
3.7.3 Deep hole drilling	3-126
3.7.4 Boring.....	3-127
3.7.5 Tapping.....	3-128
3.7.6 Thread cutting.....	3-129
3.7.7 Machining positions, freely programmable or positioning patterns.....	3-132
3.7.8 Freely programmable positions.....	3-133
3.7.9 Position pattern Line	3-134
3.7.10 Position pattern Matrix	3-135
3.7.11 Position pattern Full Circle	3-136
3.7.12 Position pattern Pitch Circle.....	3-138
3.7.13 Repeat positions	3-139
3.7.14 Obstacle.....	3-140
3.7.15 Programming examples for drilling	3-141

3.8 Milling	3-143
3.8.1 Face milling	3-143
3.8.2 Mill a rectangular pocket	3-145
3.8.3 Mill a circular pocket	3-148
3.8.4 Mill a rectangular pivot	3-150
3.8.5 Mill a circular pivot	3-152
3.8.6 Mill a longitudinal groove	3-154
3.8.7 Mill circumferential grooves on full circle or pitch	3-156
3.8.8 Use of positioning patterns for milling	3-157
3.9 Measurements	3-160
3.9.1 Workpiece measurements	3-160
3.9.2 Tool measurements	3-162
3.10 Miscellaneous functions	3-163
3.10.1 Subroutines and program section repeats	3-163
3.10.2 Insertion of ISO blocks	3-164
3.10.3 Altering the safety clearance or return plane	3-166
3.10.4 Settable and programmable offsets	3-166

Simulation

4.1 General	4-175
4.2 Start/abort program	4-175
4.3 Representation as plan view	4-176
4.4 Representation in 3-plane view	4-176
4.5 Zoom a finished part viewport	4-177
4.6 Three-dimensional representation of finished part	4-178
4.6.1 Change position of finished part	4-179
4.6.2 Cut open finished part	4-179
4.6.3 Refresh finished part display	4-179

Alarms and Messages

5.1 Cycle alarms and messages	5-183
5.1.1 Error treatment in machining cycles	5-183
5.1.2 Cycle alarm overview	5-183
5.1.3 Messages in cycles	5-186
5.2 Alarms in ShopMill	5-187
5.2.1 Overview of alarms	5-187
5.2.2 Select alarm/message overview	5-187
5.2.3 Description of alarms	5-188

Appendix