

TRAK[®] Vertical Machining Centers Siemens SINUMERIK ONE CNC Safety & Operating Manual

Document:	34105
Version:	042624

Covers Current Models:

- TRAK VMC7si
- TRAK VMC10si
- TRAK VMC12si
- TRAK VMC14si

TRAK MACHINE
TOOLS



SOUTHWESTERN INDUSTRIES, INC.

2615 Homestead Place
Rancho Dominguez, CA 90220-5610 USA
T | 310.608.4422 | F | 310.764.2668
Service Department: 800.367.3165
e-mail: sales@trakmt.com | service@trakmt.com | web: www.trakmt.com

Copyright 2021, Southwestern Industries, Inc. All rights are reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted, in any form or by any means, mechanical, photocopying, recording or otherwise, without the prior written permission of Southwestern Industries, Inc.

While every effort has been made to include all the information required for the purposes of this guide, Southwestern Industries, Inc. assumes no responsibility for inaccuracies or omission and accepts no liability for damages resulting from the use of the information contained in this guide.

All brand names and products are trademarks or registered trademarks of their respective holders.

TRAK Machine Tools

Southwestern Industries, Inc.

2615 Homestead Place

Rancho Dominguez, CA 90220-5610

Phone 310.608.4422 ▪ Fax 310.764.2668

Service Department

Phone 800.367.3165 ▪ Fax 310.886.8029

Web: www.trakmt.com

Table of Contents

1.0	Introduction	7
1.1	Using This Manual	8
2.0	Safety	9
2.1	Safety Publications.....	9
2.2	Danger, Warning, Caution, and Note Labels and Notices as Used in This Manual	9
2.3	Safety Precautions	14
3.0	Description	15
3.1	Siemens SINUMERIK ONE Control Specifications.....	15
3.2	Siemens SINUMERIK ONE Options	18
3.2.1	ONE Dynamics – Job Shop	18
3.2.2	ONE Dynamics – Surfacing.....	18
3.2.3	ShopMill Option	18
3.2.4	DXF Reader	18
3.2.5	Residual Material.....	18
3.2.6	3D Simulation	19
3.2.7	Simultaneous Recording – 2D or 3D Simultaneous During Run	19
3.2.8	TRACYL – Cylinder Surface Transformation	19
3.2.9	Advanced Surface.....	19
3.2.10	Top Surface	19
3.2.11	Measuring Cycle for Auto Mode	19
3.2.12	CNC User Memory – 4.6 GB	19
3.2.13	Memory - Execution from External Memory	20
3.2.14	Handwheel Run Option	20
3.2.15	Access MyMachine /OPC UA	20
3.2.16	Run MyVirtual Machine – Operate.....	20
3.2.17	Run MyVirtual Machine – 3D	20
3.2.18	RTAC – Real Time Adaptive Feedrate Control	20
3.2.19	APO – Auto power Off	20
3.3	Machine Details	21
3.3.1	Physical Layout and Dimensions VMCsi 7/10	21
3.3.2	Physical Layout and Dimensions VMCsi 12	23
3.3.3	Physical Layout and Dimensions VMCsi 14	25
3.3.4	Machine Specifications.....	27
3.4	Optional Equipment	29
3.4.1	Machine Options	29
3.4.2	SINUMERIK ONE Control Options	31
3.5	Standard Equipment	33
3.5.1	Touch Screen SINUMERIK ONE Control	33
3.5.2	Multiple USB Ports.....	33
3.5.3	Remote Handwheel	33
3.5.4	Side Mount Arm Style 24 Position Tool Magazine.....	33
3.5.5	12K Spindle W/CTS	33
3.5.6	Opening Side Doors.....	33
3.5.7	Oil Cooler for Spindle (VMCsi 7/10).....	34
3.5.8	Oil Chiller for Spindle (VMCsi 12/14).....	34
3.5.9	Chip Conveyor.....	34

3.5.10	Separate Coolant, Washdown and CTS Pumps	34
3.5.11	Two Coolant Nozzles	34
3.5.12	Two Air Nozzles.....	34
3.5.13	Air & Wash Hose	34
3.5.14	Work Lamps.....	34
4.0	Basic Operation	35
4.1	Powering Up / Powering Down the TRAK VMCSi.....	35
4.1.1	Powering Up	35
4.1.2	Powering Down	35
4.2	Machine Control Panel Overview	35
4.2.1	Operator Controls.....	35
4.2.2	Program Control.....	36
4.2.3	Operating Modes, Machine Functions.....	36
4.2.4	Traversing Axes	36
4.2.5	Spindle Control.....	37
4.2.1	Feed Control	37
4.3	Gestures	38
4.4	Display Pendant Overview	38
4.4.1	Function Key Block	39
4.4.2	Expandable Menu	39
4.4.3	Side Screen Widgets.....	40
4.5	Changing Modes	40
4.6	Remote Handwheel.....	41
4.7	4th Axis Option.....	41
4.7.1	Start and Stop 4th Axis Operation	41
4.7.2	4th Axis Specifications	42
4.7.3	Mounting the 4th Axis.....	43
4.8	Automation Ready Option	46
4.8.1	Automation Ready Feature Descriptions.....	46
4.8.2	Enable / Disable Automation Ready	47
4.8.3	Interface Specifications.....	48
4.9	VMCSi Auto Doors	52
4.9.1	Features Description.....	52
4.9.2	Enable/Disable Auto Doors.....	53
4.9.3	Interface Specifications.....	54
4.9.4	Trouble shooting	56
5.0	Machine Mode	58
5.1	Automatic (AUTO) sub-mode.....	59
5.1.1	Overstore	60
5.1.2	Prog. Cntrl.	60
5.1.3	Block Search	61
5.1.4	Simult. Record	62
5.1.5	Prog. Corr.	63
5.1.6	Settings	63
5.2	Machine Data Automatic (MDA/MDI) sub-mode	64
5.2.1	Loading an MDI-program.....	65
5.2.2	Saving an MDI-Program.....	65
5.2.3	Editing/executing a MDI program	65

5.2.4	Deleting an MDA/MDI program	65
5.3	JOG sub-mode.....	66
5.3.1	T, S, M (Tool, Spindle and Machine Commands).....	66
5.3.2	Set WO.....	68
5.3.3	Meas. Workp.....	69
5.3.4	Meas. Tool.....	70
5.3.5	Position	72
5.3.6	Face mill	72
5.3.7	Retract	72
5.3.8	Settings	73
5.4	Reposition (REPOS) sub-mode.....	73
5.4.1	Repositioning the axes.....	74
5.5	Reference Point (REF POINT) sub-mode	74
5.6	TEACH-IN sub-mode.....	75
5.6.1	Overstore	75
5.6.2	Teach Prog.	75
5.6.3	Prog. Cntrl	77
5.6.4	Simult. Record	77
5.6.5	Prog. Corr.....	77
6.0	Parameter Mode.....	78
6.1	Tool List.....	79
6.1.1	Create a new tool.....	80
6.1.2	Load tool to spindle / via spindle	80
6.1.3	Unloading tool from spindle / via spindle	80
6.1.4	Delete tool.....	81
6.1.5	Selecting Multiple Tools	81
6.1.6	Managing several cutting edges	81
6.1.7	Tools with additional geometry data	81
6.1.8	Changing a tool type	81
6.1.9	Sorting tool list, tool wear and magazine	82
6.1.10	Filtering tool list, tool wear and magazine	82
6.1.11	Specific search in tool list, tool wear and magazine	82
6.2	Tool Wear	83
6.2.1	Entering tool wear.....	84
6.2.2	Disabling tool.....	84
6.2.3	Reactivating tool	84
6.3	Magazine	85
6.3.1	Relocate a tool.....	86
6.3.2	Magazine selection	86
6.3.3	Deleting / unloading / loading / relocating all tools	86
6.4	Work offset	87
6.4.1	Zero offset basics	87
6.4.2	Work offset - Active.....	88
6.4.3	Work offset - Overview	88
6.4.4	Work offset - Base	89
6.4.5	Work offset - G54...G599	89
6.5	Setting Data	90
6.5.1	Working area limit.....	90
6.5.2	Spindle data.....	91

6.5.3	Data Lists	91
7.0	Program Mode.....	92
7.1	Edit.....	92
7.1.1	Building a program group	92
7.1.2	Searching in programs.....	93
7.1.3	Replacing program text	93
7.1.4	Copying/pasting/deleting program blocks.....	94
7.1.5	Renumbering a program	94
7.1.6	Editor Settings	95
7.2	Simulation	95
7.2.1	Simulation prior to machining.....	95
7.2.2	Setting the model quality	95
7.2.3	Different views of the workpiece	96
7.2.4	Program control during simulation	97
7.3	Execute.....	98
8.0	Program Manager Mode.....	99
8.1	Memory Locations.....	101
8.1.1	NC.....	101
8.1.2	NC Extend	101
8.1.3	External Memory	101
8.2	Execute.....	102
8.2.1	Executing a program	102
8.3	New.....	102
8.3.1	Starting a new NC-program	102
8.3.2	Creating a new directory.....	102
8.3.3	Creating a new workpiece.....	103
8.3.4	Storing any new file.....	103
8.4	Open	103
8.4.1	Opening a program	103
8.5	Mark.....	103
8.5.1	Selecting a Program	103
8.5.2	Selecting several directories/programs.....	104
8.5.3	Closing a program	104
8.6	Copy / Paste / Cut	104
8.6.1	Copying and pasting a directory/program	104
8.7	Archive	105
8.7.1	Generating an archive via system data	105
8.7.2	Read in archive from system data	106
8.8	Preview window	106
8.8.1	Displaying the program in the Preview.....	106
8.9	Search	106
8.9.1	Searching directories and files.....	106
8.10	Multiple Clamping	107
8.11	Properties	107
8.12	Delete.....	107
8.12.1	Deleting a directory/program	107
9.0	Diagnostics Mode.....	108

9.1	Alarm List.....	108
9.1.1	Cancel alarms, messages and logs.....	109
9.1.2	Sorting alarms, messages and logs.....	110
9.2	Messages.....	112
9.3	Alarm log.....	113
9.3.1	Save Log.....	113
9.3.2	Sort.....	113
9.4	NC/PLC variab.	114
9.4.1	Changing and deleting values.....	115
9.4.2	Editing a variable list.....	117
9.4.3	Changing operands.....	117
9.4.4	Saving and loading screen forms.....	118
10.0	Setup Mode	121
10.1	HMI.....	121
10.1.1	Assigning a Computer File Location.....	121
10.2	System Data.....	122
10.2.1	Screenshots.....	122
10.2.2	Generating an archive via the system data.....	122
10.2.3	Read in archive from system data.....	123
10.3	TRAK MT Options.....	124
10.4	Licenses.....	124
11.0	Context help.....	126
12.0	Network & Operator Panels.....	128
12.1	Configuring a Network Drive.....	128
12.2	Computer – Shared Folder.....	131
12.2.1	Creating a New Folder.....	133
12.2.2	Sharing the New Folder.....	134
12.3	Logical Drive Set-Up.....	139
12.3.1	Mapping the Shared Folder.....	139
13.0	G and M Codes.....	142
13.1	List of recognized ProtoTRAK G-codes for Siemens.....	142
13.2	List of Siemens G-codes.....	143
13.2.1	G Group 1: Modality valid motion commands.....	143
13.2.2	G Group 2: Non-modality valid motion commands.....	143
13.2.3	G Group 3: Programmable frame, working area limitation and pole programming.....	144
13.2.4	G Group 4: FIFO.....	144
13.2.5	G Group 6: Plane Selection.....	144
13.2.6	G Group 7: Tool radius compensation.....	145
13.2.7	G Group 8: Settable work offset.....	145
13.2.8	G Group 9: Frame and tool offset suppression.....	145
13.2.9	G Group 10: Exact stop – continuous path mode.....	145
13.2.10	G Group 11: Exact stop, non-modal.....	146
13.2.11	G Group 12: Block change criteria at exact stop (G60/G9).....	146
13.2.12	G Group 13: Workpiece measuring inch/metric.....	146
13.2.13	G Group 14: Workpiece measuring absolute/incremental.....	146
13.2.14	G Group 15: Feed type.....	146
13.2.15	G Group 16: Feedrate override at inside and outside curvature.....	147

13.2.16	G Group 17: Approach and retraction response, tool offset.....	147
13.2.17	G Group 18: Corner behavior, tool offset.....	147
13.2.18	G Group 19: Curve transition at beginning of spline.....	147
13.2.19	G Group 20: Curve transition at end of spline.....	147
13.2.20	G Group 21: Acceleration profile.....	148
13.2.21	G Group 22: Tool offset type.....	148
13.2.22	G Group 23: Collision monitoring at inside hours.....	148
13.2.23	G Group 24: Precontrol.....	149
13.2.24	G Group 25: Tool orientation reference.....	149
13.2.25	G Group 26: Repositioning mode for REPOS (modal).....	149
13.2.26	G Group 27: Tool offset for change in orientation at outside corners.....	149
13.2.27	G Group 28: Working area limitation.....	149
13.2.28	G Group 29: Radius/diameter programming.....	149
13.2.29	G Group 30: NC block compression.....	150
13.2.30	G Group 33: Settable fine tool offset.....	150
13.2.31	G Group 34: Tool orientation smoothing.....	150
13.2.32	G Group 35: Punching and nibbling.....	150
13.2.33	G Group 36: Punching with delay.....	150
13.2.34	G Group 37: Punching with delay.....	150
13.2.35	G Group 38: Assignment of fast inputs/outputs for punching/nibbling.....	151
13.2.36	G Group 39: Programmable contour accuracy.....	151
13.2.37	G Group 40: Tool radius compensation constant.....	151
13.2.38	G Group 41: Interruptible threat cutting.....	151
13.2.39	G Group 42: Tool carrier.....	151
13.2.40	G Group 43: SAR approach direction.....	151
13.2.41	G Group 44: SAR path segmentation.....	152
13.2.42	G Group 45: Path reference for FGROUP axes.....	152
13.2.43	G Group 46: Plane selection for fast retraction.....	152
13.2.44	G Group 47: Mode switchover for external NC code.....	152
13.2.45	G Group 48: Approach and retraction response with tool radius and compensation.....	152
13.2.46	G Group 49: Point to point motion.....	152
13.2.47	G Group 50: Orientation programming.....	153
13.2.48	G Group 51: Interpolation type for orientation programming.....	153
13.2.49	G Group 52: Interpolation type for orientation programming.....	153
13.2.50	G Group 53: Frame rotation in relation to tool.....	153
13.2.51	G Group 54: Vector rotation for polynomial programming.....	154
13.2.52	G Group 55: Rapid traverse with/without linear interpolation.....	154
13.2.53	G Group 56: Taking into account tool wear.....	154
13.2.54	G Group 57: Corner deceleration.....	154
13.2.55	G Group 59: Dynamic response mode for path interpolation.....	155
13.2.56	G Group 60: Working area limitation.....	155
13.2.57	G Group 61: Tool orientation smoothing.....	155
13.2.58	G Group 62: Repositioning mode for REPOS (non-modal).....	155
13.2.59	G Group 64: Grinding frames.....	155
13.3	List of recognized ProtoTRAK M-codes for Siemens.....	157

TRAK Warranty Policy.....158

1.0 Introduction

Congratulations! Your TRAK Vertical Machining Center with the Siemens SINUMERIK ONE is a brand-new versatile machine that delivers powerful capability with an easy-to-use interface. The strengths of Siemens and TRAK Machine Tools come together as one in the TRAK VMCSi.

Full integration of the Digital Twin unlocks the future of manufacturing by enabling the IoT and Automation applications that work for your shop. The open architecture of the SINUMERIK ONE ensures you will be able to make use of the latest applications developed as you progress through your journey to higher and higher productivity.

Integrate different programming elements into the same program and the SINUMERIK ONE runs them as a contiguous whole. The diverse formats shown below are processed by the SINUMERIK ONE for you, eliminating the need for you to convert everything into the same format.

- Sub programs
- Workpiece definition
- Probing cycles
- Logic statements
- Conversational events
- Standard G-Code
- Macros

State of the art software analyses your program and automatically converts points to splines. Most CNCs run the “point cloud” in small straight lines, the SINUMERIK ONE reconstructs the part from the point cloud and even fixes problems in the CAM software’s interpretation of the solid.

Intuitive operation embedded throughout including: touchscreen interface, animated elements while programming, tool tips that describe fields, context-sensitive help and more. These painstakingly designed features enable you to quickly learn and use the high-end features of the SINUMERIK ONE.

Conversational Programming enables you to write full or partial programs right on the shop floor, using the shop language already familiar to you.

Easy navigation throughout, whether you are searching through programs in memory or inspecting an open program, your work is made easier through carefully designed screens and the touchscreen interface of the SINUMERIK ONE.

Advanced simulations in program creation, program verification and program execution. The Digital Twin capabilities of the SINUMERIK ONE take simulations into the realm of Industry 4.0, giving you the ability to use your real machine to its maximum value by running simulations for programming, integrating automation, quoting and more.

Same SINUMERIK ONE for Milling, Turning, Grinding, Robotic, Horizontal and Additive technologies. The same user interface enables you to build on what you learn in 3-axis milling to adopt advanced 4- and 5-axis capability.

TIA Portal – The Totally Integrated Automation Portal is part of the Siemens Digital Enterprise Software Suite. The SINUMERIK ONE can be fully integrated into the TIA Portal, enabling it to be a part of the digitalized network running the shop floor, or even the entire facility.

1.1 Using This Manual

This manual provides enough information for most users in most situations. If you ever need more information or clarification, we want to help.

- Call us at 800-421-6875 and ask for Applications
- Visit our website <http://www.trakmt.com>
- Talk to your local TRAK representative

Section 2 of this manual provides important safety information. It is highly recommended that all operators of this product review this safety information.

Section 3 provides a description of the TRAK VMcsi 7, 10, 12 and 14, and the Siemens SINUMERIK ONE. This section includes specifications, control options, display pendant information, machine information, machine options and description of standard equipment.

Section 4 describes the basic operation of the vertical machining center and some basic operations of the Siemens SINUMERIK ONE.

Section 5 describes the Machine menu. The Machine menu displays information such as the axis positions, distances to go, feed rates, spindle speeds, active tool, etc.

Section 6 describes the Parameter menu. The Parameter menu displays information such as tool/magazine data, work offsets, user variables etc. It is possible to carry out functions such as creating tools, editing existing tool/magazine data, editing work offsets/variables/setting data.

Section 7 describes the Program menu. The Program menu is used to display a part program, only if a program is already open will its contents be displayed. Editing of the part program is possible here. If a part program is not already open, the Program Manager will be open.

Section 8 describes Program Manager menu. The Program Manager menu displays the "Part program/subprogram/workpiece" directories. Programs/subprograms/workpieces can be created or opened for editing. It is also possible to save to/from controller using memory devices.

Section 9 describes the Diagnostics menu. The Diagnostics menu is used for diagnosing machine problems, information about current/past alarms can be viewed. The status of drives, axes, networks, NC/PLC variables, utilization etc., can be viewed. It is also possible to check the software/hardware versions of the system. Machine information and service reports are also entered via the diagnostics menu.

Section 10 describes the Setup menu. The Setup menu is used for a wide variety of functions, these include passwords, machine data modifications, archives, alarm files, time/date settings, licensing/option selection, system data management, servo optimization, etc.

Section 11 describes the Context help screen function, which provides context-sensitive help for every input window.

Section 12 describes the common G-Codes, formatting, start and end blocks and M-Code functions.

2.0 Safety

The safe operation of the TRAK VMCSi depends on its proper use and the precautions taken by each operator.

- Read and study this manual. Be certain every operator understands the operation and safety requirements of this machine before its use.
- Never run the machine with enclosure doors open.
- Always wear safety glasses and safety shoes.
- Always stop the spindle and check to ensure the CNC control is in the stop mode before changing or adjusting the tool or workpiece.
- Never wear gloves, rings, watches, long sleeves, neckties, jewelry, or other loose items when operating or around the machine.
- Use adequate point of operation safeguarding. It is the responsibility of the employer to provide and ensure point of operation safeguarding per OSHA 1910.212 - Milling Machine.

2.1 Safety Publications

Refer to and study the following publications for assistance in enhancing the safe use of this machine.

Safety Requirements for Machining Centers and Automatic, Numerically Controlled Milling, Drilling and Boring Machines (ANSI B11.23-2002) (R2007). Available from The American National Standards Institute, 1819 L Street N.W., Washington D.C. 20036

Concepts and Techniques of Machine Safeguarding (OSHA Publication Number 3067). Available from The Publication Office - O.S.H.A., U.S. Department of Labor, 200 Constitution Avenue, NW, Washington, DC 20210.

2.2 Danger, Warning, Caution, and Note Labels and Notices as Used in This Manual

DANGER - Immediate hazards that will result in severe personal injury or death. Danger labels on the machine are red in color.

WARNING - Hazards or unsafe practices that could result in severe personal injury and/or damage to the equipment. Warning labels on the machine are orange in color.

CAUTION - Hazards or unsafe practices that could result in minor personal injury or equipment/product damage. Caution labels on the machine are yellow in color.

NOTE - Call attention to specific issues requiring special attention or understanding.

Safety & Information Labels Used on TRAK VMCsi Machines

It is forbidden by OSHA regulations and by law to deface, destroy or remove any of these labels.

208 VOLTS

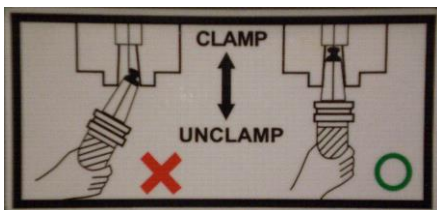
480 VOLTS



Safety & Information Labels Used on TRAK VMCsi Machines



Air pressure setting	
Max.pressure setting	115psi / 8kg/cm ²
Min.pressure setting	70psi / 5kg/cm ²
Pressure sensor setting	60psi / 4 kg/cm ²



SAFETY INSTRUCTIONS

1. Read and understand Operator's Manual and all warnings on the sign before operating. Failure to follow these instructions and warnings can result in serious injury or death.
2. This machine starts and moves automatically. Never place any part of your body near or on moving parts of this machine.
3. Always stop the spindle completely before touching the work piece, tool or spindle.
4. Do not operate this machine unless all guards, interlocks and other safety devices are in place and functioning.
5. Always clamp work piece and cutting tool securely. avoid excessive feeds and spindle speeds.
6. Remove rings, watches, jewelry and loose fitting clothing. Keep your hair away from moving parts of the machine.
7. Always wear safety glasses, safety shoes and hearing protection when operating this machine.
8. Service or installation of this machine must be performed by qualified personnel only, following procedures described in the Maintenance Manual. Turn off and lock out power at main electrical panel before servicing.

It is the responsibility of the user to be sure that this machine is in safe operating condition at all times and that the operator follows the safe operating procedures described in the Operator and Maintenance Manuals and all signs attached to this machine. If you have any questions concerning the safe operation of this machine, contact your supervisor or local Distributor.

Please do not remove or disfigure this sign.



Safety & Information Labels Used on TRAK VMCsi Machines

MAINTENANCE SCHEDULE

DAILY

- Remove majority of chips from around the axis slide ways, work table, and ballscrew covers.
- Drain the water from the air regulator assembly by either turning off the air to the machine or depressing the valve at the bottom of the water collector tank.
- Check to make sure there are no obvious oil leaks to the lubrication system.
- Visually check coolant level and add if low.

WEEKLY

- Visually check lubrication pump oil level and make sure it is always above the minimum line. Fill with ISO VG32/SAE10W.
- Check the oil level in the pneumatic system lubricator and add if low.
- Remove and clean air filters on the door and the top of the electrical cabinet and replace if needed.

MONTHLY OR AS SPECIFIED

- Visually check the air regulator filter and replace if needed.
- Check the oil level of the Tool Change Air Cylinder Oil Cup, and add if low.
- Perform all maintenance on the ATC as listed in the Machine Service Manual.
- Every 3 months remove and drain the coolant tank, clean all debris and fill with new coolant.
- Every 6 months check the level and tram of the machine and adjust as necessary.
- Every 6 months check the spindle motor belt tension and adjust as necessary.

YEARLY

- Check backlash on each axis and adjust as necessary. Refer to the machine manual for service codes.
- Remove all covers, vacuum chips and debris that may have built up, and wipe down machine from top to bottom.
- Inspect the tool change air cylinder.
- Inspect machine for any unusual wear and play.
- Check cables, coolant hoses and pneumatic lines for any excess abrasion or cuts.

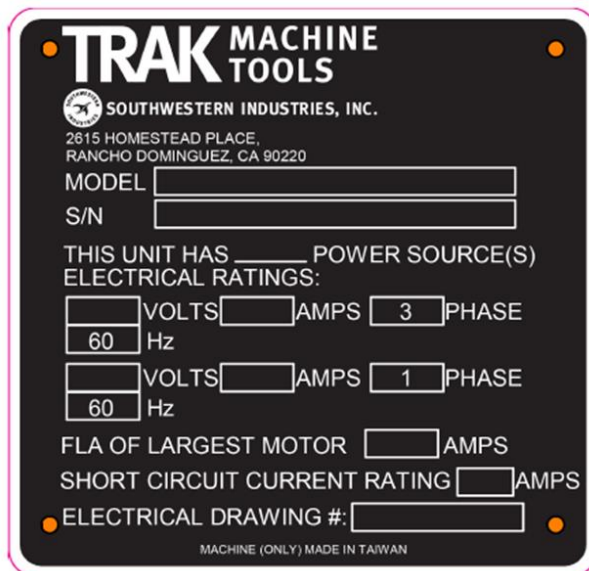
Please contact SWI Service Dept. for help with any machine maintenance procedures.

Service Dept. contact number is 1-800-367-3165.

Safety & Information Labels Used on TRAK VMCsi Machines



TRAK Serial Number Plate will have the specific information engraved applicable to the mill.



2.3 Safety Precautions

1. Do not operate this machine before the TRAK VMCsi Service and Operation manuals have been studied and understood.
2. Do not run this machine without knowing the function of every control key, button, knob, or handle. Ask your supervisor or a qualified instructor for help when needed.
3. Protect your eyes. Always wear approved safety glasses (with side shields). Don't use compressed air to remove the chips or clean the machine.
4. Before operating this machine remove all jewelry including watches and rings, neckties, and any loose-fitting clothing. Keep your hair away from moving parts. Wear adequate safety headgear.
5. Protect your feet. Wear safety shoes with oil-resistant, anti-skid soles, and steel toes.
6. Take off gloves before you start the machine. Gloves are easily caught in moving parts.
7. Remove all tools (wrenches, check keys, etc.) from the machine before you start. Loose items can become dangerous flying projectiles.
8. Never operate a milling machine after consuming alcoholic beverages, taking strong medication, or while using non-prescription drugs.
9. Protect your hands. Stop the machine spindle and ensure that the CNC control is in the stop mode before:
 - a. changing tools, changing parts, clearing away the chips, oil or coolant
 - b. making an adjustment to the part, fixture, coolant nozzle or take measurements.
 - c. opening safeguards (protective shields, etc.). Never reach for the part, tool, or fixture around a safeguard.
10. Disconnect power to the machine before you change belts, pulley, or gears.
11. Keep work areas well lit. Ask for additional light if needed.
12. Do not lean on the machine while it is running.
13. Prevent slippage. Keep the work area dry and clean. Remove the chips, oil, coolant, and obstacles of any kind around the machine.
14. After an emergency stop, release the E-stop and press the RESET button.
15. Avoid getting pinched in places where the table, saddle, tool changer, or spindle head create "pinch points" while in motion.
16. Securely clamp, and properly locate the workpiece in the vise, on the table, or in the fixture. Use stop blocks to prevent objects from flying loose. Use proper holding clamping attachments and position them clear of the tool path.
17. Use correct cutting parameters (speed, feed, depth, and width of cut) to prevent tool breakage.
18. Use proper cutting tools for the job. Pay attention to the rotation of the spindle: Left hand tool for counterclockwise rotation of spindle, and right-hand tool for clockwise rotation of spindle.
19. Prevent damage to the workpiece or the cutting tool. Never start the machine (including the rotation of the spindle) if the tool is in contact with the part.
20. Check the direction (+ or -) of movement of the table, saddle, and spindle when using jog.
21. Don't use dull or damaged cutting tools. They break easily and become airborne. Inspect the sharpness of the edges, and the integrity of cutting tools and their holders. Use proper length tool for the job.
22. Large overhang on cutting tools when not required result in accidents and damaged parts.
23. Inspect the retention knobs for damage or excessive wear before each use.
24. Prevent fires. When machining certain materials (magnesium, etc.) the chips and dust are highly flammable. Obtain special instruction from your supervisor before machining these materials. Keep flammable materials and fluids away from the machine and hot, flying chips.
25. Always be certain the door is closed during program run when the machine is in operation.
26. Make sure coolant lines and nozzles are facing away from the operator.
27. Use only the retention knobs that each machine is designed to use. The use of the incorrect retention knobs or incorrect usage of the retention knobs may result in injury or property damage.

3.0 Description

3.1 Siemens SINUMERIK ONE Control Specifications

Control Structure & Configuration

- 19" LED
- Multi-Touch Screen
- Maintenance Free Design, no battery, no fan

Spindle Functions

- Thread cutting with constant or variable pitch
- Tapping with compensating chuck and rigid tapping

Measuring

- Measuring cycles for drilling/milling – O
 - Calibrating workpiece probes – O
 - Workpiece measurement – O
 - Tool measurement – O
- Logging of measurement results

Interpolations

- Typical block change times (block processing time) 0.7 ms
- Floating point accuracy 80 bits
- Max number linear interpolating axes: 12
- Circle via center point and end point
- Circle via interpolation point
- Helical interpolation (2D+6)
- Universal interpolator NURBS (non-uniform rational B splines)
- Continuous-path mode with programmable rounding clearance
- Continue machining at the contour (retrace support) – O
- Spline interpolation (A, B, C splines) – O
- Polynomial interpolation – O
- Involute interpolation – O
- Multi-axis interpolation – O

CNC Programming Language

- Programming language DIN 66025 and high-level language expansion
- Main program call from main program and subprogram
- Subroutine levels, maximum: 16
- Interrupt routines, maximum: 2
- Number of subprogram passes: 9999
- Number of levels for skip blocks: 0...8
- Polar coordinates
- 1/2/3-point contours
- Dimensions metric/inch, changeover via operator action or program
- Inverse-time feedrate
- User variables, configurable
- Predefined user variables (arithmetic parameters)
- Read/write system variables
- Program jumps and branches
- Comparison operations and logic combinations
- Arithmetic and trigonometric functions
- Macro techniques
- Control structures: IF-ELSE-ENDIF
- Control structures: WHILE, FOR, REPEAT, LOOP
- STRING functions
- Program Functions:
 - Preprocessing memory, dynamic FIFO
 - Look Ahead, recorded part program blocks (MDynamics, Top Surface or COMPSURF active): 3000
 - Look Ahead, IPO blocks, buffered: 1000
 - Frame concept
 - Inclined-surface machining with frames
 - Axis/spindle interchange
 - Program preprocessing

Simulation

- Quickview for mold-making programs
- 2D simulation (finished part)
- 3D simulation 1 (finished part) – O
- Real-time simulation of current machining operation – O

Program Support

- Program editor in SINUMERIK ONE:
 - Text editor: selecting, copying, deleting
 - Dual editor
 - Multi-editor, maximum: 4
 - Write protection for lines
 - Suppression of lines in the display
- Technology cycles in SINUMERIK ONE:
 - Drilling
 - Milling
 - Pocket milling with free contour definition and islands
- programGUIDE in SINUMERIK ONE:
 - Programming support for cycles
 - Dynamic programming graphics
 - Animated elements
- DXF Reader – O
- Accepting contours – O
- Accepting point patterns – O
- Residual material detection and machining for contour pockets and stock removal – O
- Access protection for cycles – O

Communication / Data Management

- CNC user memory for programs and OEM cycles, buffered internally on NCU/PPU: 10 MB
- Manage additional drives via:
 - Ethernet: 4
 - USB
- Data backup:
 - SD card of the NCU/PPU (backup/restore) on flash drive or on network
 - With Ghost (backup/restore) on network
- IT security:
 - Secure and unique identification by means of device certificate
 - Software integrity thanks to signed software and Secure Boot
 - Secure protection of access data thanks to hardware-supported Security Controller
 - Program block and cycle encryption
 - Encrypted communication with OPC UA

Program / Workpiece Management

- Part programs, maximum: 1000
- Program/workpiece management: 250
- Templates for programs and workpieces
- Job lists
- Program/workpiece management on additional HMI user memory
- Program/workpiece management on USB storage
- Program/workpiece management on the network drive
- Basic frames, maximum number: 16
- Settable offsets, maximum number: 100
- Work offsets, programmable (frames)
- Scratching, determining work offset

Compensations

- Backlash compensation
- Leadscrew error compensation
- Measuring system error compensation
- Circularity test
- Quadrant error compensation
- Feedforward control:
 - Velocity-dependent
 - Acceleration-dependent
- Weight counterbalance, electronic
- Temperature compensation

Operating Modes

- AUTO CNC operating mode:
 - Execute directly:
 - from real CNC user memory: 10 MB
 - from CNC user memory, expanded – O
 - from CNC user memory on SD card of the NCU – O
 - Process from external source (EXTCALL):
 - Data storage medium on USB interface

- from a network drive
 - Execution from External Storage (EES): – O
 - One part program memory for several NCUs – O
 - Backward jumps, far jumps, long program loops (GOTOF/GOTOB) – O
 - DRF offset
 - Program control
 - Program editing
 - Block search with/without calculation
 - Overstore
 - Configured Stop – O
- JOG CNC operating mode:
 - Handwheel selection
 - Inch/metric changeover
 - Manual measurement of work offset
 - Manual measurement of tool offset
 - Automatic tool/workpiece measurement
 - Reference point approach automatic/via CNC program
 - Repositioning on the contour via operator action, semiautomatically and via program
- MDI CNC (Manual Data Automatic/Input) operating mode:
 - Input in text editor
 - Save MDI program
 - Teach positions in MDI buffer
 - Teach-in function Handling

Tools

-
- Tool Types:
 - Drilling/milling
 - Groove sawing
 - Turning tools
 - Probing tools
 - Tool radius compensations in plane with:
 - Approach and retract strategies
 - Transition circle or transition ellipse at outside corners
 - Configurable intermediate blocks with tool radius compensation active
 - Tool carrier with orientation capability
 - Look-ahead detection of contour violations
 - Wheel peripheral speed, programmable
 - Tool length compensation, online
 - Identify tools with meaningful tool names
 - Operation with tool management:
 - Operation with tool management, up to 4 magazines
 - Operation with tool management, with more than 4 magazines – O
 - Monitoring for maximum tool speed/acceleration – O
 - System displays in standard software
 - User-friendly commissioning via system displays
 - Tool list
 - Configurable tool list
 - Operation without tool management:
 - Editing of tool data
 - Tool offset selection via T and D numbers
 - Tools in the tool list: 600
 - Cutting edges in the tool list: 1500
 - Quantity:
 - Tools in the tool list: 600
 - Cutting edges in the tool list: 1500
 - Magazine list
 - Configurable magazine list
 - Empty location search and place positioning
 - Easy empty location search using softkeys
 - Loading and unloading of tools
 - More than one loading and unloading point per magazine
 - Tool life monitoring and workpiece count
 - Multi-tool with tools, maximum: 64
 - Adapter data
 - Location-dependent offsets

3.2 Siemens SINUMERIK ONE Options

3.2.1 ONE Dynamics – Job Shop

The One Dynamics - Job Shop is the perfect technology package to get you started programming right away! This software bundle includes:

- ShopMill
- DXF Reader
- Residual Material
- 3D Simulation
- Simultaneous Recording – 2D or 3D Simultaneous During Run
- TRACYL – Cylinder Surface Transformation
- Memory - Execution from External Memory

3.2.2 ONE Dynamics – Surfacing

This bundle includes everything in the One Dynamics – Job Shop, as well as additional options for enhanced surface finishes, in-process probing, and expanded user programming memory. This software bundle includes:

- ShopMill
- DXF Reader
- Residual Material
- 3D Simulation
- Simultaneous Recording – 2D or 3D Simultaneous During Run
- Advanced Surface
- Top Surface
- TRACYL – Cylinder Surface Transformation
- Measuring Cycle for Auto Mode
- User Programming Memory
- Memory - Execution from External Memory

3.2.3 ShopMill Option

There is no G-Code required with ShopMill. It is a conversational programming language that enables the simple operation of machine tools, supporting all operator actions with graphic help displays.

NOTE – These ShopMill cycles, can be used in conjunction with G-Code programs.

3.2.4 DXF Reader

The DXF Reader allows DXF files to be opened directly on the CNC within SINUMERIK ONE so that contours and points can be extracted from the files. If you work with CAD drawings, we highly recommend the DXF file converter.

3.2.5 Residual Material

Contour areas that cannot be machined with large tools are automatically detected/recognized by the cycle for contour pockets or the stock removal cycle. The operator can rework these regions using a smaller tool.

Some benefits of this option include:

- Shorter machining times using a large tool for the substantial part of the stock removal and a smaller tool for the remaining residual material
- Avoidance of non-cutting movements while achieving extremely simple programming

3.2.6 3D Simulation

This option provides you with a 3D simulation of the machining process in programming mode. This extension also applies to the simultaneous recording function explained below.

3.2.7 Simultaneous Recording – 2D or 3D Simultaneous During Run

During machining, the tool paths can be simultaneously recorded on the CNC screen in 3-side view or 3D view (requires 3D Simulation P25). Graphic simulation corresponds directly to the current state/position of the machine.

3.2.8 TRACYL – Cylinder Surface Transformation

With the 4th Axis, you have the ability to purchase the TRACYL option which allows you to draw a 2D shape, and then wrap it about a cylinder.

3.2.9 Advanced Surface

Advanced Surface and **Top Surface** are both advanced algorithms that analyze look ahead and velocity. The software analyses the program and converts points to splines to create a superior toolpath that is processed smoothly through the SINUMERIK ONE run engine.

You are able to choose the between maximizing Machining Velocity, Surface Quality and Dimensional Accuracy. For example, turning on the "Smoothing" function (in Top Surface) will greatly increase part surface quality but may possibly slow the machining. This advanced software puts the trade-offs into your hands – where it belongs.

The Advanced Surface function analyzes your part file in order to optimize the program's motion and speed, and provide more accurate contours, and even more perfect surfaces. Best for high-speed roughing applications, but also great for achieving optimal results for general milling. Highly recommended, and part of the One Dynamics Options Package for Job Shops.

3.2.10 Top Surface

This option only works if the Advanced Surface option was also purchased. It guarantees even more enhanced workpiece surfaces, and is considerably more proficient for demanding CAD/CAM programs. This option provides additional benefits to the programmer and machine operator, in order to ensure that workpieces are top quality.

The newer of the algorithms, this advanced software will actually fix problems with the imported CAM data, detecting gaps in the points distribution that will result in jerking or hesitations as the program runs. This option is best for high-speed finishing in mold making where the very best surface quality is required. Part of the One Dynamics Options Package for Surfacing.

3.2.11 Measuring Cycle for Auto Mode

This option allows you to use the native measuring (probing) cycles inside a program (auto mode).

3.2.12 CNC User Memory – 4.6 GB

This option provides the user with 4.6 GB of internal memory. This allows all programs and data, such as part programs, subprograms, comments, tool offsets, and work offsets/frames, as well as channel and program user data, can be stored in the shared CNC user memory.

3.2.13 Memory - Execution from External Memory

Enables the use of an external memory device for running and editing programs. Ideal for optimizers of large programs, it allows your memory device to function like an expansion of your internal SINUMERIK ONE memory buffer without any cache restrictions on programs, subprograms, commands or even transferring to another machine.

3.2.14 Handwheel Run Option

This provides a more user-friendly operation of the machine in auto mode by allowing you to run through a program using the remote handwheel pendant, in the forward direction with multiple speed resolutions.

3.2.15 Access MyMachine /OPC UA

The Access MyMachine /OPC UA function enables the communication channel for the OPC UA server. OPC Unified Architecture (OPC UA) is a standardized, industrial communication protocol for the access to control data, e.g. by control systems. The OPC UA Data Access method permits the reading and writing of CNC and PLC variables.

3.2.16 Run MyVirtual Machine – Operate

This option provides you with a digital-twin of the control, right on your desktop/laptop, that you can use for remote programming or training.

3.2.17 Run MyVirtual Machine – 3D

This option turns the Run My Virtual Machine software into a true digital twin of the TRAK VMCsi. This is an extension to the option above, and has additional features such as virtual job preparation, simulation of drilling, milling and turning operations, simulation of material removal, display log of detected collision, delete log entries, and much more.

3.2.18 RTAC – Real Time Adaptive Feedrate Control

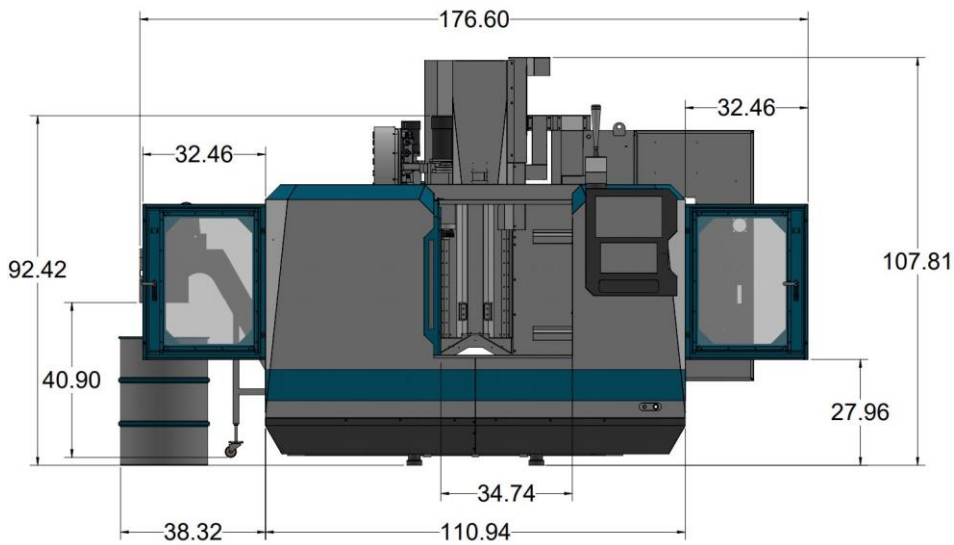
Optimize MyMachining/ Real Time Adaptive Feedrate Control (RTAC) is a real-time adaptive control application. RTAC monitors the spindle load of the machine and automatically adjusts the feedrate to its optimal level during each operation. This allows you to manufacture more parts with shorter machining time and achieve higher process stability with tool breakage prevention.

3.2.19 APO – Auto power Off

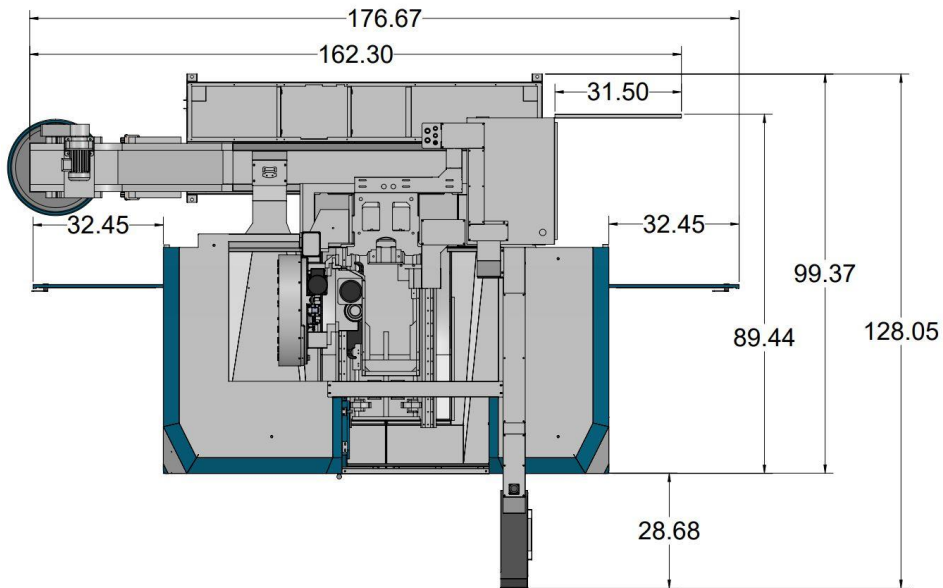
Auto Power Off will command the SINUMERIK ONE to shut down power at the end of a program run. This option is great for untended machining, so the machine will shut down after the last part is run. Simply press the APO button on the control panel after the run is initiated.

3.3 Machine Details

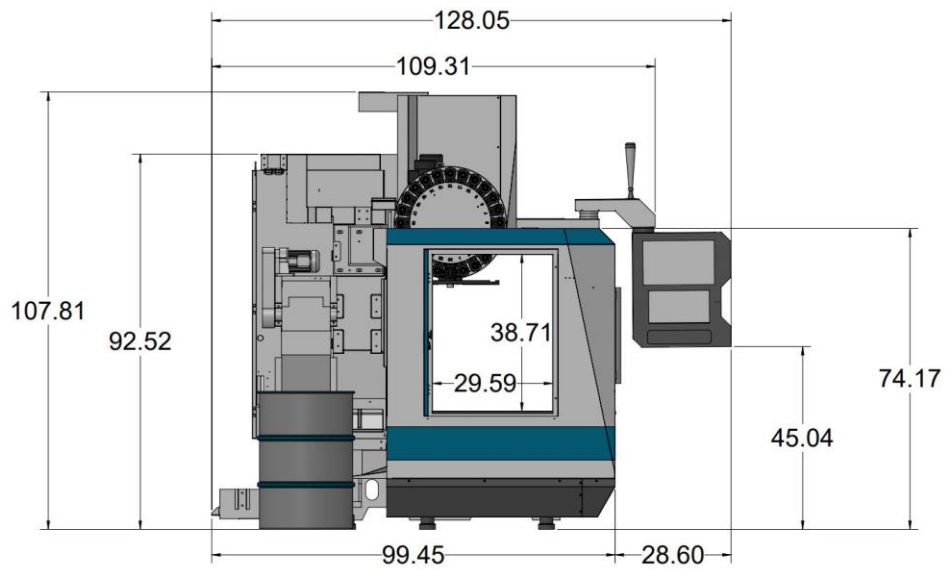
3.3.1 Physical Layout and Dimensions VMCsi 7/10



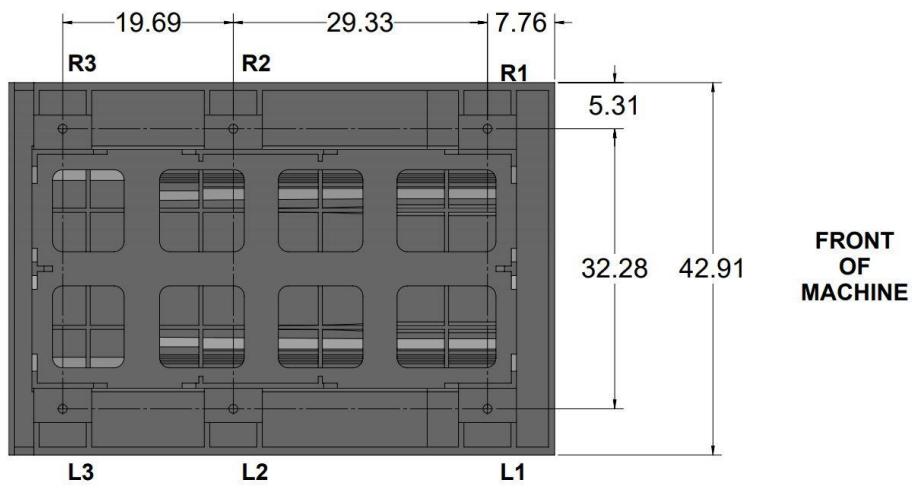
Front View VMCsi 7/10 (inches)



Top View VMCsi 7/10 (inches)



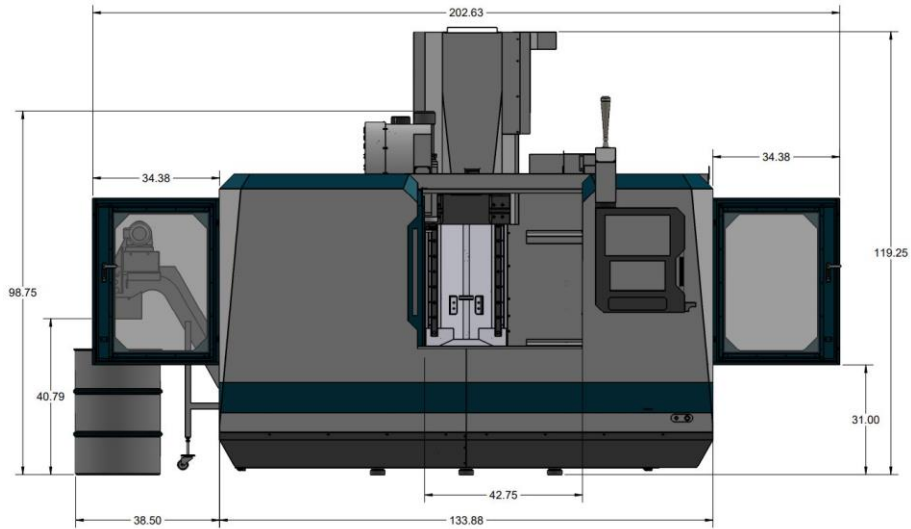
Left Side VMCsi 7/10 (inches)



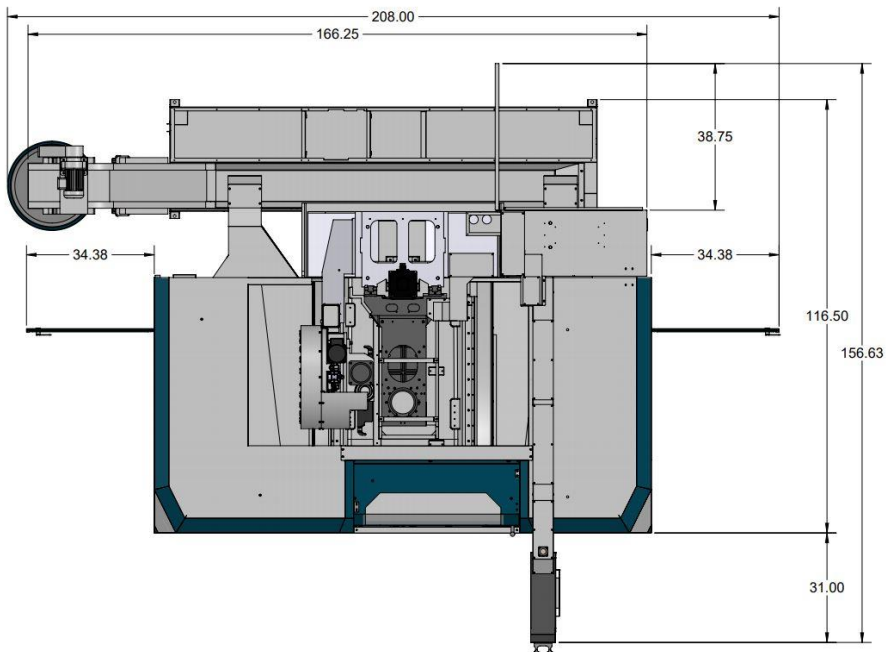
**FRONT
OF
MACHINE**

Bottom Side VMCsi 7/10 (inches)

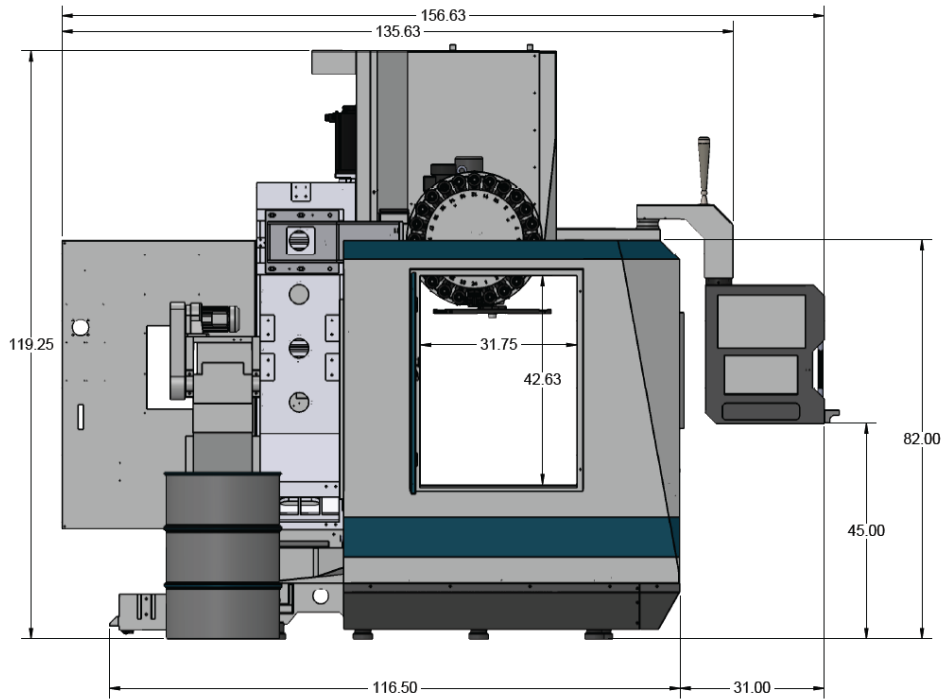
3.3.2 Physical Layout and Dimensions VMCsi 12



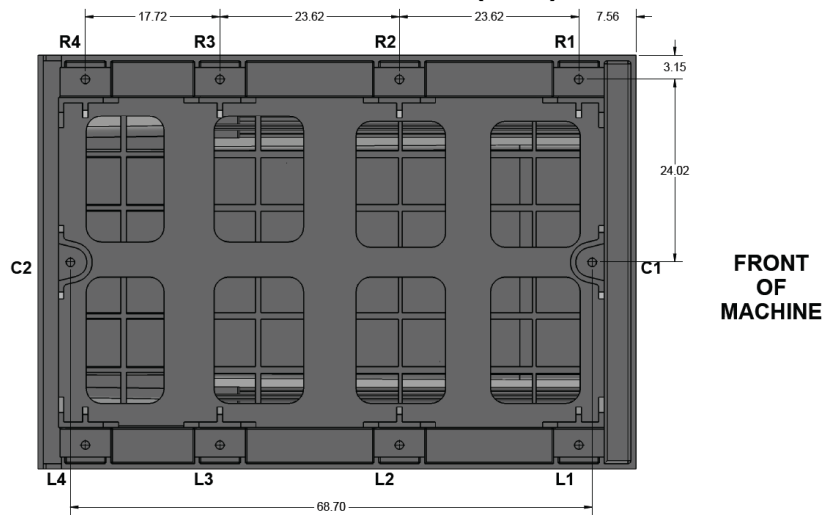
Front View VMC12si (inches)



Top View VMC12si (inches)

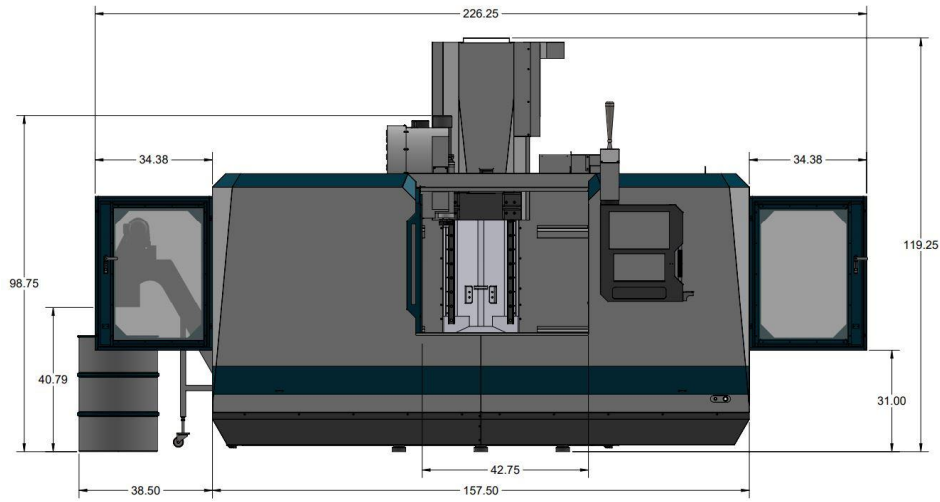


Left Side VMC12si (inches)

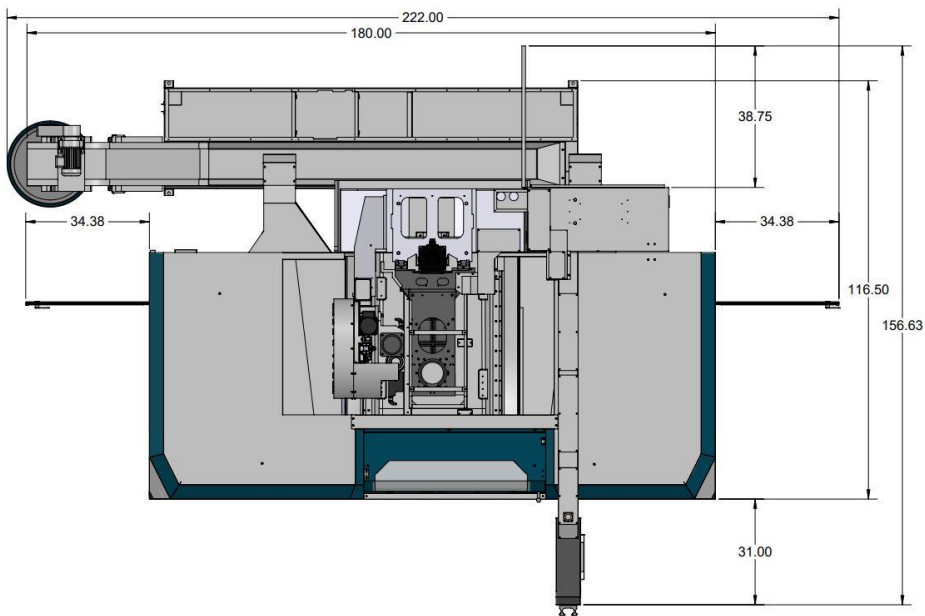


Bottom Side VMCsi 14 (inches)

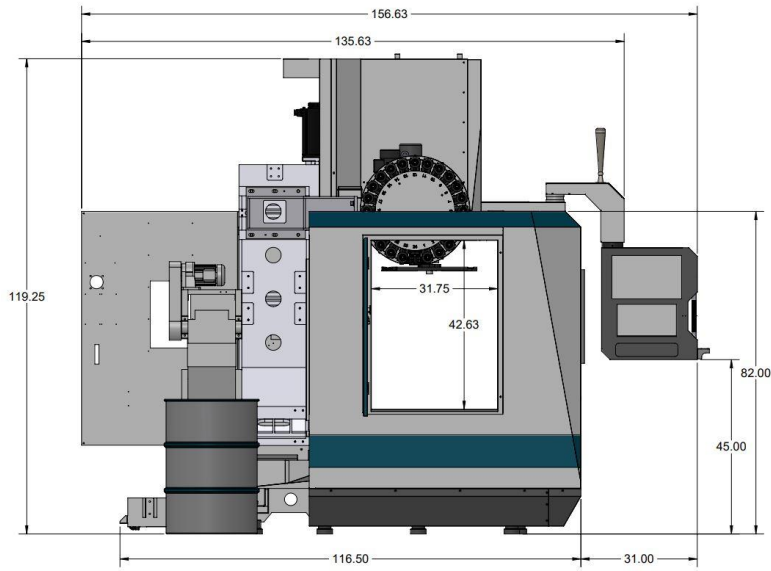
3.3.3 Physical Layout and Dimensions VMCsi 14



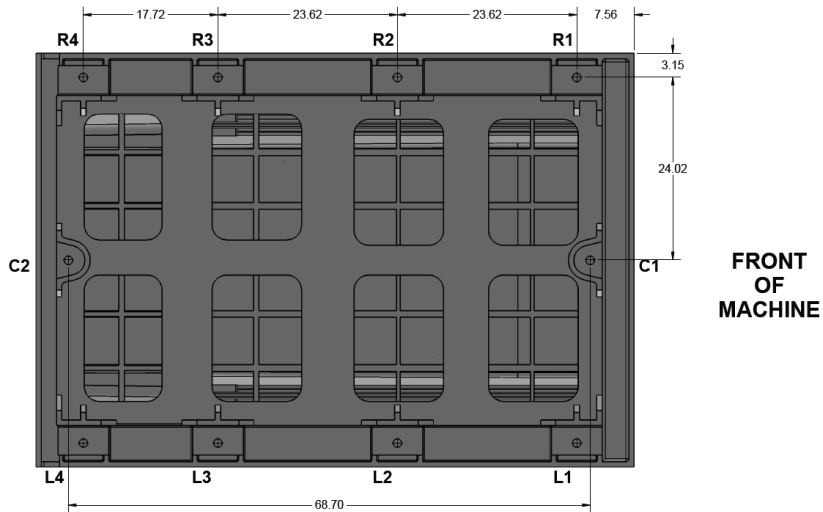
Front View VMC14si (inches)



Top View VMC14si (inches)



Left Side VMC14si (inches)



Bottom Side VMCsi 14 (inches)

3.3.4 Machine Specifications

Machine Specs	VMC7si	VMC10si	VMC12si	VMC14si
Table size	35.43 x 19.69"	44.09 x 19.69"	51.18 x 23.62"	62.99 x 23.62"
Travel (X, Y, Z axis)	30 x 20 x 20"	40 x 20 x 20"	50 x 27.5 x 25"	60 x 27.5 x 25"
Rapid traverse X, Y, Z	1,000 IPM			
Cutting max. speed X, Y, Z	1,000 IPM			
T-slots (number x width x pitch)	5 x .709 x 3.937		5 x .709 x 4.921	
Maximum weight of workpiece	1,230 lbs	1,760 lbs	2,200 lbs	2,640 lbs
Height of table from bottom of floor	38"		42"	
Min. spindle nose to table distance	3.5"		3"	
Max. spindle nose to table distance	23.50"		28"	
Distance of ATC arm to table	20"		25.75"	
Min. machine height	93.5"		106"	
Max. machine height (head all the way up)	108"		119.25"	
Overall width of machine (side doors open + chip conveyor)	177"		202.63"	
Overall length of machine (electrical cabinet fully open)	150.25"		156.63"	
Footprint of machine (chip conveyor & coolant tank)	144 x 112"		168.25 x 126.50"	180 x 126.50"
Weight net / shipping (lbs)	8,360 / 8,710	9,900 / 10,250	15,400 / 15,900	17,600 / 18,100
Spindle Specs	VMC7si	VMC10si	VMC12si	VMC14si
Spindle taper	40 Taper			
Spindle speed range	10 - 12,000 RPM			
Spindle nose diameter	3"		3.75"	
Diameter of spindle	60 mm		70 mm	
Tool clamping force (90 psi)	1,500 lbs		2,200 lbs	
Tool holder type	CAT 40 standard			
ATC tool capacity	24			
Maximum tool weight (incl. holder)	17.6 lbs			
Maximum tool diameter	3.07			
ATC arm tool change time	2.5 seconds			
Spindle motor HP - peak	41.5 HP		67 HP	
Spindle motor HP - continuous	20.1 HP		26.8 HP	
Spindle Motor Torque - continuous	72 N-m or 53 ft-lbs. at 2000 RPM 96 N-m or 70.8 ft-lbs. at 500 RPM		98 N-m or 72 ft-lbs. at 1750 RPM 125 N-m or 92 ft-lbs. at 500 RPM	
Spindle Motor Torque - peak	148 N-m or 109 ft-lbs. up to 2000 RPM		240 N-m or 177 ft-lbs. up to 1750 RPM	

Spindle center to spindle head	18.25"	26.75"		
Spindle cooler oil capacity	~1 gallon	~4.25 gallon		
Power Requirements	VMC7si	VMC10si	VMC12si	VMC14si
Power requirements - standard	480V (456-506V is okay), 3P, 41A		480V (456-506V is okay), 3P, 69A	
Power requirements - optional	208V with transformer option (198-218V is acceptable), 3P, 94A		n/a	
Coolant System	VMC7si	VMC10si	VMC12si	VMC14si
Tank capacity	60 gallons		78 gallons	
Pressure and volume	8.7 gpm @ 42 psi			
Pump and wash power	750 watts			
Coolant wash pump	750 watts		1,290 watts	
CTS (Coolant Through Spindle) pump	3 KW / 4 HP			
CTS pressure	319 psi			
Air Requirements	VMC7si	VMC10si	VMC12si	VMC14si
Air - CFM, SCFM (at 90 PSI)	2.5 CFM, 18 SCFM			
Air quality	Air dried/water separator upstream of VMC			
Lubrication Requirements	VMC7si	VMC10si	VMC12si	VMC14si
Lubrication pump capacity	2 liters			
Lube Oil Type	ISO 32 or 10 W			

Drive Train Specs	VMCsi 7/10	VMCsi 12/14
# of X, Y and Z axis linear bearing blocks	2	
X, Y and Z axis linear guide size	35 mm	45 mm
X axis linear guide way span	300 mm	380 mm
Y axis linear guide way span	530 mm	900 mm
Z axis linear guide way span	386 mm	440 mm
X, Y and Z axis ballscrew diameter	32 mm	40 mm
Pitch of ballscrew (X, Y and Z axis)	10 mm	
Ballscrew accuracy class	C3	
Rapid traverse X, Y, Z	1000 ipm	
Cutting max speed X, Y, Z	1000 ipm	

ATC Info	VMCsi 7/10	VMCsi 12/14
ATC Tool Capacity	24	
Maximum tool weight including holder	8 kg /17.6 lbs.	
Maximum tool diameter	78 mm /3.07"	
Maximum tool length	300 mm /11.81"	
Arm ATC tool change time	2.5 seconds	

Specifications subject to change.

3.4 Optional Equipment

3.4.1 Machine Options

3.4.1.1 Retention Knobs – CAT 40 Tooling

The VMCsi uses CAT40 retention knobs, as shown below. They can also be purchased through TRAK Machine Tools under part number: RETN KNOB KIT-CTS-VMCSI

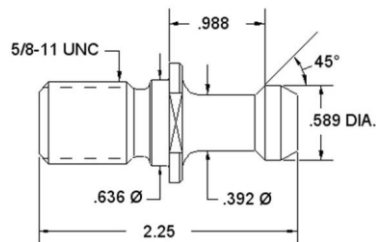


Figure 3.a – VMC Retention Knobs

Tightening to the proper torque value is important for all retention knobs. Please see the retention knob manufacturer for the proper torque. Retention knobs purchased through TRAK Machine Tools should be torqued to 30 ft-lbs.

WARNING!

Retention knobs come in a wide variety of designs; however, they often look similar and appear to be interchangeable, but they are not. Use only the knob that the machine is designed to use. The use of the incorrect knob, or the incorrect use of a knob, may result in injury and/or damage to the mechanism.

3.4.1.2 BT40 Option

If a user would prefer to run BT40 tooling, the BT40 option is required. This option replaces the arm on the ATC as well as the tool pots installed in the ATC magazine. BT40 pull studs are available from TRAK Machine Tools.

3.4.1.3 Memory Option - TRAK USB Device

We highly recommend that you use the high-quality Delkin® brand thumb drive we provide with the USB Memory Option. This thumb drive is industrial grade, built to handle demanding read/write speeds as well as the environmental hazards of a manufacturing environment.

3.4.1.4 Transformer Option (VMCsi 7/10): 208V to 480V

Both the TRAK VMC7si, and VMC10si, run on 480-volt, 3 phase-power. Shops with 200 to 240-volt incoming power, will require this optional step-up transformer for the VMC7/10si. The transformer output is set to 480-volts; to convert your incoming power from 208 volts, to 480 volts.

3.4.1.5 Skimmer Option

The oil skimmer option directly bolts to the VMCsi coolant tank. This option extends the life of your coolant and reduces coolant maintenance.

3.4.1.6 Vise Kit Option

The vise kit offered includes a 6" Kurt DX6 vise (9" opening) as well as the mounting hardware for the VMCsi T-slots.

3.4.1.7 Tool Measurement Cart

This option includes a Huot ToolScoat cart, granite plate, preset base tool, and scale. It is used to measure tool lengths outside of the machine when a tool probe is not available.

3.4.1.8 Fixture Cart

Heavy duty cart for the set up and transportation of fixtures. Designed at the correct table height with an overhang to get the fixture as close to the table as possible.

3.4.1.9 Probe Options

TRAK Machine Tools offers three probing options to choose from: Part Probe, Tool Probe, as well as a package combining the Part and Tool probes.

The Part Probe loads into the spindle to set work coordinates using the SINUMERIK native probing cycles and to inspect dimensions during a program run. Adjustments can be made to the work coordinate or tool table based on the inspection.

The Tool Probe mounts inside the machine space and SINUMERIK native cycles are available to measure the tool length as well as the diameter. These same cycles can be used during program runs to check and adjust for tool wear or breakage.

NOTE - If a probing system is purchased from a vendor other than TRAK Machine Tools, please contact the probe manufacturer for assistance with installation.

3.4.1.10 4th Axis Option

The TRAK 4th axis system for vertical machining centers provides full control of 4th axis machining through the Siemens SINUMERIK ONE CNC. The system can also be used as an indexer. Optional tailstock sold separately.

3.4.1.11 4th Axis Ready Option

The electronics cabinet is pre-wired for the optional 4th axis rotary unit. You would want this to have each machine in your shop be able to run the 4th axis and then swap the actual rotary unit between them. It also simplifies the installation of the option on the machine later without a lot of downtime or expense.

3.4.1.12 Tailstock Option - 4th Axis

The tailstock is optional for the 4th axis. It is useful when machining large parts with the 4th axis, to maintain part accuracy on long, complex, or off-balance parts.

3.4.1.13 ATC Option- 30 or 40 Tool ATC

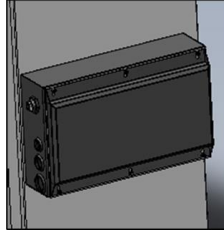
This option increases ATC capacity to 30 or 40 tools.

3.4.1.14 Automation Ready

Automation Ready option will allow users to interface with an automation system which includes robots or industrial robots via digital Input/Output.

The Automation Ready key features include:

- Remote Cycle Start – allows robot to start the program in memory
- Remote Cycle Stop – allows robot to stop a program
- Cycle End Output – signals to the robot that the program has finished
- 4 Automation Request Outputs and Finish Input commands. The M-codes allow CNC to control when the robot interacts with the CNC.
- E-stop input and output
- Front Door open/close outputs
- Side Door open/close outputs
- Front and Side Door open/close requests



3.4.2 SINUMERIK ONE Control Options

3.4.2.1 One Dynamics Job Shop (S41) Package

Options included in this package (descriptions of individual options can be found below):

- ShopMill (P17)
- DXF Reader (P56)
- Residual Material Detection (P13)
- 3D Simulation 1 (P25)
- Simultaneous 3D Run (P22)
- TRACYL/Cylinder Surface Transformation (M27)
- Execution from External Storage (P75)
- Advanced Surface Motion Control (S07)

3.4.2.2 One Dynamics Surfacing (S42) Package

Options included in this package (descriptions of individual options can be found below):

- Job Shop (S41)
- Top Surface (S17)
- Measuring Cycles for Drilling/Milling (P28)
- User program memory 4.6 GB (P12)

3.4.2.3 Auto Power Off (APO)

Auto Power Off will command the SINUMERIK ONE to shut down power at the end of a program run. This option is great for untended machining so the machine will shut down after the last part is run. Simply press the APO button on the control panel after the run is initiated.

3.4.2.4 3D Simulation (P25)

3D Solid digital twin of your part, faithful in every programmed detail. You can check your program by close inspection of each detail using the touchscreen operation of the 19" screen of the SINUMERIK ONE on the VMCSi.

3.4.2.5 Advanced Surface (S07)

Advanced algorithms that analyze look ahead and velocity. The software analyses the program and converts points to splines to create a superior toolpath that is processed smoothly through the SINUMERIK ONE run engine.

Best for high-speed roughing applications, but also great for achieving optimal results for general milling. Highly recommended, and part of the One Dynamics Options Package for Job Shops.

3.4.2.6 Top Surface – Cycle 832 (S17)

Advanced algorithms that analyze look ahead and velocity. The software analyses the program and converts points to splines to create a superior toolpath that is processed smoothly through the SINUMERIK ONE run engine.

The newer of the algorithms, this advanced software will actually fix problems with the imported CAM data, detecting gaps in the points distribution that will result in jerking or hesitations as the program runs. This option is best for high-speed finishing in mold making where the very best surface quality is required. Part of the One Dynamics Options Package for Surfacing.

3.4.2.7 CNC Memory, 4.6 Gigabytes (P12 & P77)

A combination of Siemens options that, together, adds internal memory to the SINUMERIK ONE to bring in and integrate large files (such as CAD and CAM). Extends the internal CNC memory by 4.6 GB.

3.4.2.8 DXF Reader (P56)

Open DXF files on the SINUMERIK ONE to easily transfer data into your program.

3.4.2.9 Measuring Cycle for Auto Mode (P28)

Enables you to use a probe within the Auto Mode operation. Insert a probing canned cycle to measure tool wear for the SINUMERIK ONE to apply compensation.

3.4.2.10 Execute from External Memory (P75)

Enables the use of an external memory device for running and editing programs. Ideal for optimizers of large programs, it allows your memory device to function like an expansion of your internal SINUMERIK ONE memory buffer without any cache restrictions on programs, subprograms, commands or even transferring to another machine.

3.4.2.11 Residual Material (P13)

The SINUMERIK ONE automatically recognizes Rest Machining situations, simply call up the Residual Machining canned cycle, enter a few simple inputs and the control does the rest!

Match the tool to the operation for fastest machining: large tool for hogging, smaller tool for regions the large tool won't reach. Multiple Residual Machining cycles in the same feature gives you the flexibility to minimize cycle times.

3.4.2.12 Shopmill (P17)

True conversational programming you can do right on the shop floor. Write complete part programs or insert a feature within another program. With guidance from Animated Elements, Tool Tips, Dynamic Graphics, Canned Cycles and Context-Sensitive Help to guide you can create programs with ease. You won't have to go back to the person who programmed the part to fix or add to your program.

3.4.2.13 Simulation Run During Machining (P22)

3D solid model graphics real time while running a program. It is a great way to check the progress of your job, especially when you are running a lot of coolant.

3.4.2.14 TRACYL (M27)

Wraps a 2D shape around a cylinder, enabling you to program features onto a cylinder using the 4th Axis option of your VMCSi machine.

3.4.2.15 RTAC – Real Time Adaptive Feedrate Control

Optimize MyMachining / RTAC (Real Time Adaptive Feedrate Control) is a real-time adaptive control application. RTAC monitors the spindle load of the machine and automatically adjusts the feedrate to its optimal level during each operation. This allows you to manufacture more parts with shorter machining time and achieve higher process stability with tool breakage prevention.

3.4.2.16 Handwheel Run (M08)

Run the programmed moves by turning the handwheel. You control the feedrate by moving the Manual Pulse Generator on the control panel and the actual programmed path is moved. Adjust the relationship between the handwheel turns and the feedrate with 4 different resolution settings. Seamlessly go between Handwheel Run and fully automatic CNC run.

3.5 Standard Equipment

3.5.1 Touch Screen SINUMERIK ONE Control

Touchscreen for data entry and mode selection w/ intuitive gestures for navigation and manipulation of 3D simulation.

3.5.2 Multiple USB Ports

The VMCSi comes with multiple USB ports, located on the right side of the pendant, which can be utilized for external memory, keyboard or mouse.

3.5.3 Remote Handwheel

This is used to move the axes around in the manual mode. You select which axis and resolution you want to move. It is also used with the Handwheel Run Option, which allows the operator to manually jog thru the program continuously in the forward direction, as well as backwards, to the start of the block.

3.5.4 Side Mount Arm Style 24 Position Tool Magazine

This is a side mount arm style automatic tool changer that has a capacity of 24 tools. Automatic pre-staging of next tool in ShopMill mode (option), and follows the shortest path to maximum tool change efficiency.

3.5.5 12K Spindle W/CTS

The VMCSi comes standard with a 12,000 RPM spindle. Compared to flood coolant, the CTS option can increase tool life, allow for higher spindle speeds and a corresponding reduction of cycle times, and allow for better surface finishes.

3.5.6 Opening Side Doors

The large opening side doors allow you to comfortably reach through the side of the machine and place/remove your workpiece, or change tools in the tool magazine, and aids in cleaning and maintenance of the machine.

3.5.7 Oil Cooler for Spindle (VMCsi 7/10)

The VMCsi 7 & 10 come equipped with a cooler, to help cool the oil running through the spindle. This helps keep the spindle operating within temperature range and helps keep tighter tolerances.

3.5.8 Oil Chiller for Spindle (VMCsi 12/14)

The VMCsi 12 & 14 come equipped with a chiller, to help chill the oil running through the spindle. This helps keep the spindle operating within temperature range and helps keep tighter tolerances.

3.5.9 Chip Conveyor

The conveyor removes chips from the machine by carrying them and unloading them into a container. The conveyor can be controlled directly through M-Code functionality inside a program, as well as, with a button located on the machine control panel (MCP). The conveyor can also run-in reverse to help free up tangled chips in the conveyor shoot.

3.5.10 Separate Coolant, Washdown and CTS Pumps

In order to have enough pressure for each function, more than one pump is required. You can find these located at the back of the machine, behind the chip conveyor.

3.5.11 Two Coolant Nozzles

On the head, there are two coolant nozzles to help clear chips, reduce heat buildup, and add lubrication.

3.5.12 Two Air Nozzles

Next to the coolant nozzles, on the head, are two air nozzles to help clear chips from the workpiece.

3.5.13 Air & Wash Hose

There is a handheld air and coolant wash hose located on the right side of the pendant.

3.5.14 Work Lamps

The VMCsi comes standard with two work lamps; one located above each side door, which come on automatically when the power is turned on, and can be shut off manually from the control panel.

4.0 Basic Operation

4.1 Powering Up / Powering Down the TRAK VMCsi

Before powering up / powering down the TRAK VMCsi, the E-stop should be pressed.

4.1.1 Powering Up

The power switch for the TRAK VMCsi is located on the outside of the electrical box, on the right side of the machine.

NOTE - Make sure your Spindle and Feed are enabled, by pressing "Spindle Start" and "Feed Start". If they are not enabled, they will not run.

4.1.2 Powering Down

To shut power down the TRAK VMCsi, flip the switch OFF at the electrical box.

4.2 Machine Control Panel Overview

You use the machine control panel to initiate actions on the machine tool such as traversing an axis or starting the machining of a workpiece.

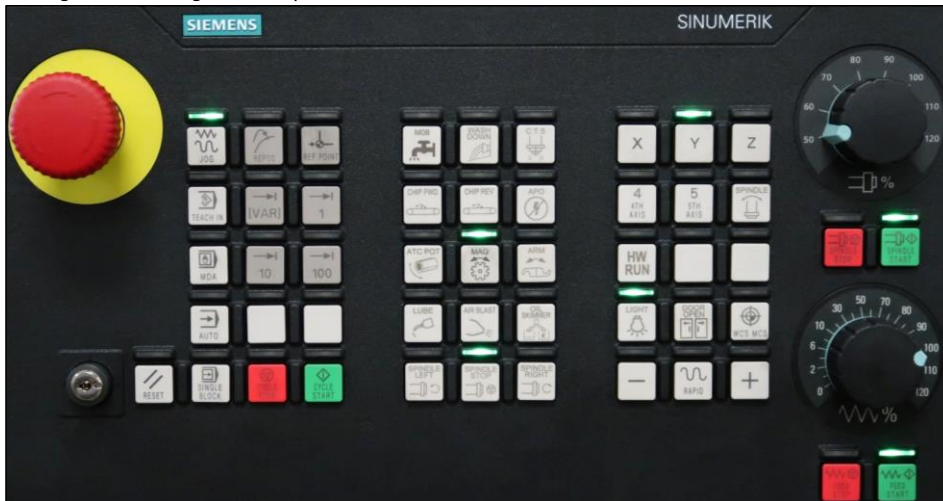


Figure 4.a - Machine Control Panel

4.2.1 Operator Controls



E-STOP: Press the button in situations where life is at risk, or there is the danger of a machine or workpiece being damaged. All drives will be stopped with the greatest possible braking torque.



RESET:

- Stop processing the current programs.
- Cancel alarm.

4.2.2 Program Control



SINGLE BLOCK:
Single block mode on/off.



CYCLE START:
The key is also referred to as NC Start.
Execution of a program or command is started.



CYCLE STOP:
The key is also referred to as NC Stop.
Execution of a program is stopped.

4.2.3 Operating Modes, Machine Functions



JOG:
Go to Jog Sub-Mode



REF POINT:
Approach reference point.



TEACH-IN:
Go to TEACH-IN Mode



Inc VAR Incremental Feed Variable
Incremental mode with variable increment size.



MDA/MDI:
Go to MDA/MDI (MDI) Sub-Mode



Inc 1 (Incremental feed)
Incremental mode with predefined increment size of 1.



AUTO:
Go to AUTO Sub-Mode



Inc 10 (Incremental feed)
Incremental mode with predefined increment size of 10.



REPOS:
Go to REPOS Sub-Mode

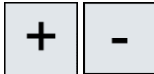


Inc 100 (Incremental feed)
Incremental mode with predefined increment size of 100.

4.2.4 Traversing Axes



Axis Keys
Select an axis.



Direction Keys
Select the traversing direction.

- When the door is closed, you can jog the axis in the + or - direction at 20"/min.
- When the door is opened, you are limited to 20% of the feed speed.



RAPID
Traverse axis in rapid traverse while pressing the direction key.

- When the door is closed, pressing the + or - key along with the rapid key will jog the selected axis at 400"/min
- When the door is opened, you are limited to 20% of the RAPID feed speed.



WCS MCS
Switches between the workpiece coordinate system (WCS) and machine coordinate system (MCS).

4.2.5 Spindle Control



SPINDLE STOP
Stop spindle (disable)



SPINDLE START
Start spindle (enable)



Override Switch (%)

Reverse/Stop/Forward

The spindle can then be manually jogged through the spindle left/stop/right keys located on the machine control panel. You must enter an initial command (ex: in TSM mode), in order for the spindle to know what RPM to run at, then use the following keys:



Figure 4.b – Spindle Left/Stop/Right (#1 in Figure 4.a)

- **Spindle Left** – Turn the spindle counter clockwise.
- **Spindle Stop** – Stop the spindle.
- **Spindle Right** – Turn the spindle clockwise

NOTE - For your safety, you will not be allowed to unlock the door, when the spindle is running higher than 50 RPM. If the door is open, the spindle will not run faster than 50 RPM. Once it is closed, it may run at the speed you programmed it to.

4.2.5.1 Spindle Override

The spindle override is the knob located on the top, right side of the machine control panel.

1. Select the override percentage.
2. Press "Spindle Start" to begin the override.
3. Press "Spindle Stop", to end the override.

Percentage override is on the knob itself, but there is also a visual on the top right of the display screen, labeled "S".

4.2.1 Feed Control



FEED STOP
Stops execution of the running program and shuts down axis drives.



FEED START
Enable for program execution in the current block and enable for ramp-up to the feedrate value specified by the program.



Override Switch (%)

4.2.1.1 Feed Override

The feed override is the knob located on the bottom, right side of the machine control panel.

1. Select the override percentage.
2. Press "Feed Start" to begin the override.
3. Press "Feed Stop", to end the override.

Percentage override is on the knob itself, but there is also a visual on the top right of the display screen, labeled "F".

4.3 Gestures

- **Swipe horizontally with one finger:** Scrolls lists with many columns.
- **Swipe vertically with one finger:** Scroll through list with many rows.
- **Swipe vertically with two fingers:** Scroll one page up/down
- **Swipe vertically with three fingers:** Scroll to top/bottom
- **Spread / Pinch:** These gestures are used to zoom in or out of a drawing. Use your thumb and finger.
- **Pan with one finger:** Pan is used to move graphic contents around the screen, and move list contents.
- **Pan with two fingers:** Rotate graphic contents.
- **Tap and hold:** Open input fields to change, and activate/deactivate edit mode.
- **Tap and hold using two fingers:** open cycles line by line (without input screen form).

4.4 Display Pendant Overview

The display pendant consists of about six different sections, as displayed in Figure 3.a. below.

NOTE - Different modes will yield different screen displays, and options (in some cases).

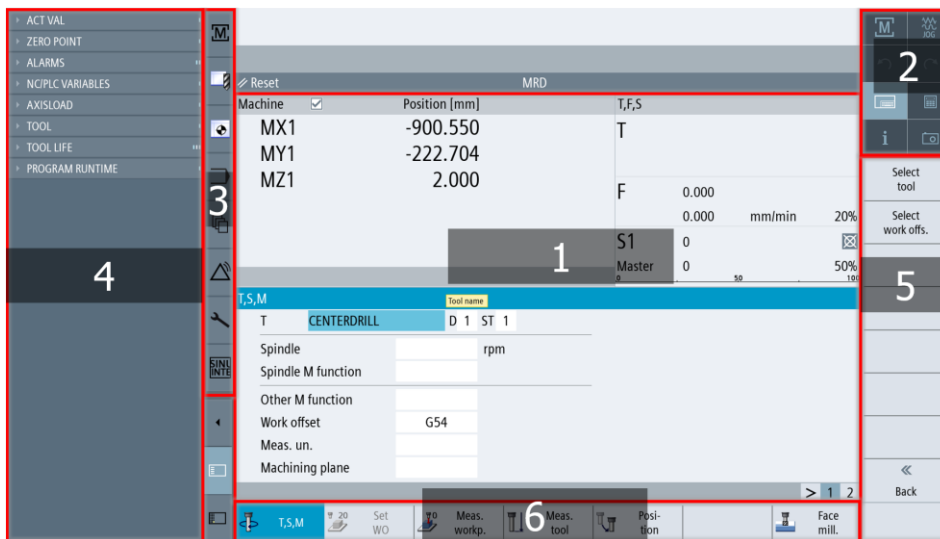


Figure 4.c – Screenshot of control in Machine Mode, and Jog Sub-Mode.

Screen Overview

1. **Main Display** - Screen contents will change, depending on the Mode, Sub-mode, or task you are in, or performing.
2. **Function Key Block** – Contains some frequently used functions and shortcuts.
3. **Expandable Menu** – Contains shortcuts to the modes, and side screen widgets.
4. **Side Screen Widgets** – Part of the Expandable Menu.
5. **Vertical Softkeys** - These will change depending on the mode and sub-mode you are in. They are selectable options based on your current mode/sub-mode as well as current access level. Use the < > or BACK keys to expand or collapse these panels.

- 6. **Horizontal Softkeys** - These will change depending on the mode and sub-mode you are in, as well as current access level. They are selectable options based on your current mode/sub-mode. Use the < > or BACK keys to expand or collapse these panels.

4.4.1 Function Key Block

- **Menu Select** – Opens the Mode menus as HSKs and Sub-modes as VSKs
- **Sub-mode** – Displays the sub-mode you are in.
- **Undo** – Undo the last step. Multiple changes are undone one by one. As soon as a change has been completed in an input field, this function is no longer available.
- **Restore** – Restore the last step. Multiple changes are restored one by one. As soon as a change has been completed in an input field, this function is no longer available.
- **Virtual keyboard** is displayed. Allows you to enter data into fields.
- **Virtual calculator** is displayed.
- **Context help screen** is displayed.
- **Camera** – Generates a screenshot of the current screen.

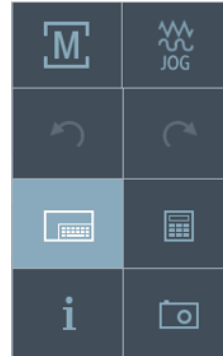


Figure 4.d - Right Shortcut Menu

4.4.2 Expandable Menu

- **Machine Mode** – This is the mode you want to be in when you want to physically interact the machine, and/or generate machine movement in any way. In this mode, you can jog the machine manually, setup your workpiece, use the auto probe cycles, setup or probe your tools, and allows you quickly program. This is the mode you will be taken to when running your program.
- **Tools List / Parameter Mode** – In this mode you can enter information about your tooling, including tool wear, magazine assignments, assigning sister tools, make fine adjustments, and edit any of the work offsets.
- **Workpiece Offset / Parameter Mode** - Displays the Workpiece Offset area menu.
- **Program Mode** – In this mode, you can write and edit programs using G-Code or ShopMill (*option*).
- **Program Manager Mode** – In this mode, you can view all the part programs, subprograms, and work pieces. You can also navigate through the internal/external storage options and access network drive files.
- **Alarm List / Diagnostics Mode** – In this mode, you can see the alarms, alarm lists and message lists, and diagnostics.
- **Setup Mode** – In this mode, you can see machine data, system data, licenses, and network setup.
- **Expand/Collapse** the Side Screen.

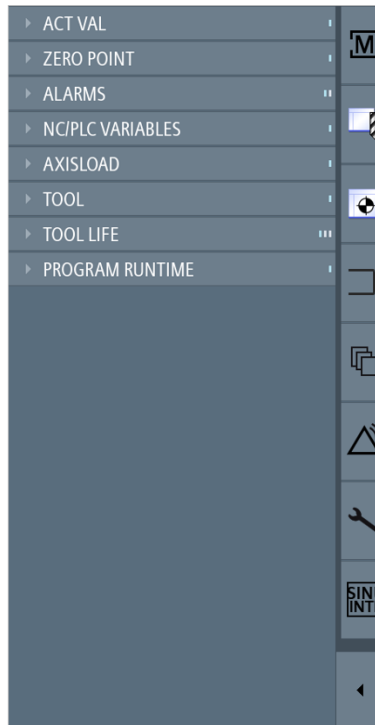


Figure 4.e - Left Shortcut Menu and Side Screen

4.4.3 Side Screen Widgets

Selecting widget item will expand the window.

- **ACT VAL** – The widget contains the position of the axes in the displayed coordinate system. The distance-to-go for the current NC block is displayed while a program is running.
 - **ZERO POINT** – The widget includes values of the active work offset for all configured axes. The approximate and detailed offset, as well as rotation, scaling and mirroring are displayed for each axis.
 - **ALARMS** – The widget contains all the messages and alarms in the alarm list. The alarm number and description are displayed for every alarm. An acknowledgment symbol indicates how the alarm is acknowledged or canceled. Vertical scrolling is possible if multiple alarms are pending. Swipe horizontally to switch between alarms and messages.
 - **NC/PLC VARIABLES** – The widget displays the NC and PLC variables. The variable name, data type and value are shown for each variable. Only those variables that are currently displayed in the "NC/PLC variables" screen in the "Diagnostics" operating area are shown. To update the list in the "NC/PLC variables" widget following a change in the "NC/PLC variables" screen in the "Diagnostics" operating area, collapse and expand the widget again. Vertical scrolling is possible.
 - **AXISLOAD** – The widget shows the load on all axes in a bar chart. Vertical scrolling is possible if multiple axes are present.
 - **TOOL** – The widget contains the geometry and wear data for the active tool. The following information is additionally displayed depending on the machine configuration:
 - EC: Active location-dependent offset - setting up offset
 - SC: Active location-dependent offset - additive offset
 - TOFF: Programmed tool length offset in WCS coordinates, and programmed tool radius offset
 - Override: Value of the overridden movements that were made in the individual tool directions
 - **TOOL LIFE** – The widget displays the tool monitoring in relation to the following values:
 - Operating time of tool (standard time monitoring)
 - Finished workpieces (quantity monitoring)
 - Tool wear (wear monitoring)
- NOTE** - Multiple cutting edges: If a tool has multiple cutting edges, the values of the edge with the lowest residual service life, quantity and wear is displayed. It's possible to alternate between views by scrolling horizontally.
- **PROGRAM RUNTIME** – The widget contains the following data:
 - Total runtime of the program
 - Time remaining to end of program
- This data is estimated for the first program run. Additionally, progress of the program is visualized in a bar chart as a percentage.

4.5 Changing Modes

To change the mode you are in, there are two ways:

1. Using the Left Shortcut Menu, click on the mode you would like to go to.
2. On the Right Shortcut Panel, click on the "Menu Select" button.
 - The HSK will display the different modes available: Machine, Parameter, Program, Program manager, Diagnostics, and Setup.
 - The VSK will display the sub-modes available within the selected mode.

4.6 Remote Handwheel

In order to move the axes around in the manual mode:

1. Use the **left black knob**, to select the axis you wish to move (X, Y, Z, 4th).
2. Use the **right black knob** to select the speed resolution (x100 = 0.0100", x10 = 0.0010", x1 = 0.0001").
3. Press down on the **two dead man switches** on each side of the remote, to activate the remote handwheel (a green light above the wheel in the center will light up green, meaning the remote is now active)
4. When the remote is active, use the **wheel in the center** to jog the axis in the positive (clockwise) direction, or negative (counter-clockwise) direction.
5. An **E-Stop** switch is located on the top left corner of the remote.



4.7 4th Axis Option

TRAK Machine Tools manufactures a 4th axis option that can be retro-fitted onto a TRAK VMCSi at any time after its initial purchase. See figure below.

For additional information about the 4th Axis option, aside from what is contained in this manual, see the *TRAK VMCSi Mills with Siemens Sinumerik CNC Safety, Installation, Maintenance, Service & Parts List Manual* (P/N 34104), Section 7.0.

The 4th axis model's key features include:

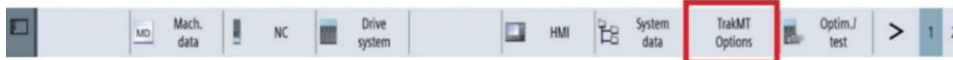
- The unit doesn't require air.
- The unit is compact.
- The unit is sealed and lubricated for 20,000 hours of operation therefore no maintenance is required.



Figure 4.f - 4th Axis Option

4.7.1 Start and Stop 4th Axis Operation

TRAK has a customized screen available for enabling and disabling the 4th axis operation. To access the screen, at the upper right corner of the display, press [M], press "SETUP" from the left VSK menu bar then press "TrakMT Options" from the HSK as shown below:



The current status will be displayed on the top line of the screen. To change 4th axis status, press the highlighted second row and select the new option from pop up menu. If you want to activate the change, press "Accept" in the lower right VSK. If you press "Cancel", the changes will not be effective. See figure below.

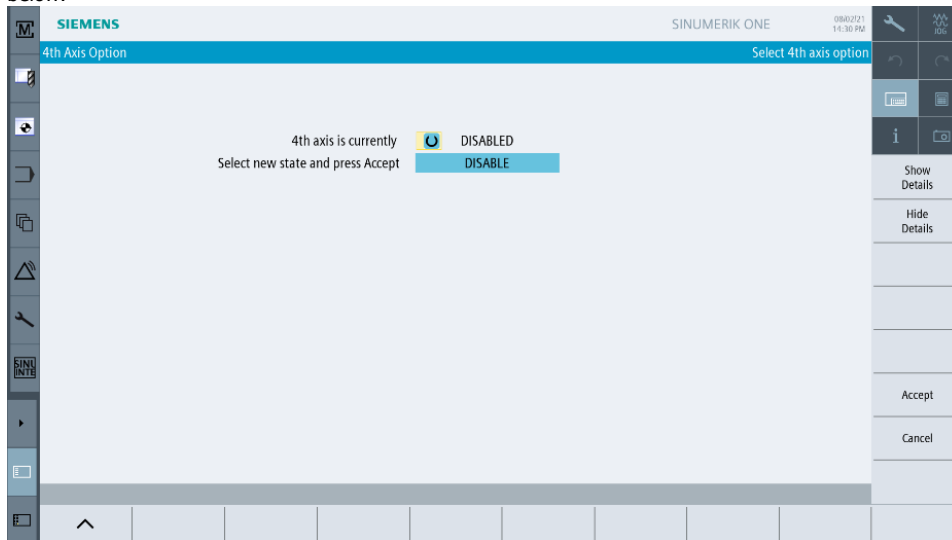


Figure 4.g – 4th Axis Option screen

4.7.2 4th Axis Specifications

Feature	Specification
Spindle Diameter	8.07" (205mm)
Chuck Diameter	7.874" (200mm)
Overall Height of 4 th Axis	13.82" (351mm)
Centerline Height of Spindle	6.30" (160 mm)
Minimum Resolution of System	0.001°
Maximum RPM	40
Repeatability	+/- 2 arc seconds
Indexing Accuracy	25 arc seconds
4 th Axis Weight	190 lbs (86 Kg)
4 th Axis Keyway Size	18 mm
Max distance between 4th axis chuck face & tailstock with center*	VMC7si – 14.14" (360mm) VMC10si – 22.70" (580mm) VMC12si – 29.86" (758mm) VMC14si – 41.67" (1058mm)

* Chuck and adapter plate can be removed to add up to 4.65" (113mm) of additional distance. Chuck and adapter plate feed-through hole is 2.28" (58mm).

4.7.3 Mounting the 4th Axis

The images below demonstrate how the 4th axis unit and the optional tailstock mount to the VMCSi table, figure 6.3a and figure 6.3b and 6.3d with the tailstock option. The 4th axis unit has an eyebolt attached to the top of the unit for lifting purposes. As shown in the image below, the 4th axis and tailstock have a key or clamping block to drop in and slide to any T slot, see figure 6.3c.

The 4th axis unit requires (2) cables to be connected and disconnected each time you add or remove the unit from the machine. See figure 6.3e. The cables are the electrical hookup to the servomotor and encoder. Make sure the cables are securely fastened and locked in place.



Figure 4.h –4th Axis and Tailstock Option Mounted on VMCSi Table

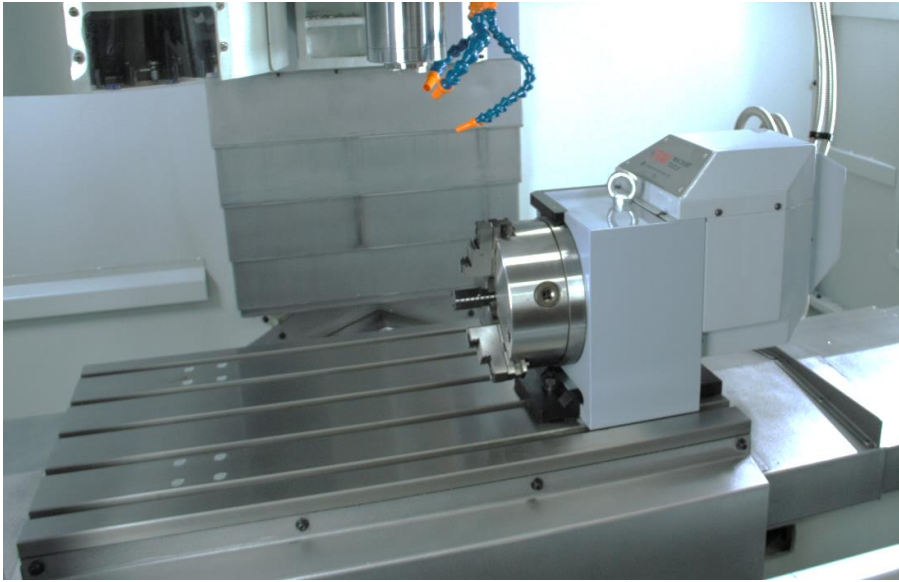


Figure 4.i – 4th Axis Mounted on VMCSi Table without the Tailstock Option

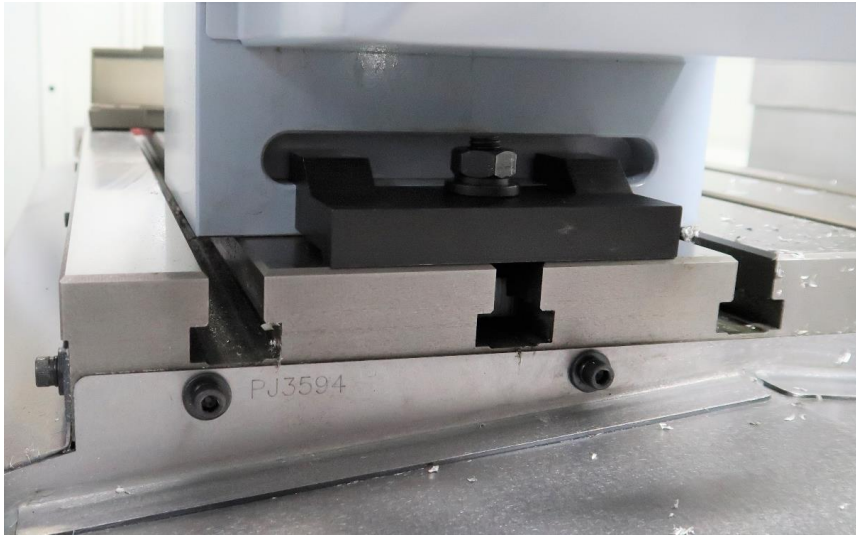


Figure 4.j – Clamping Block



Figure 4.k – Tailstock Option

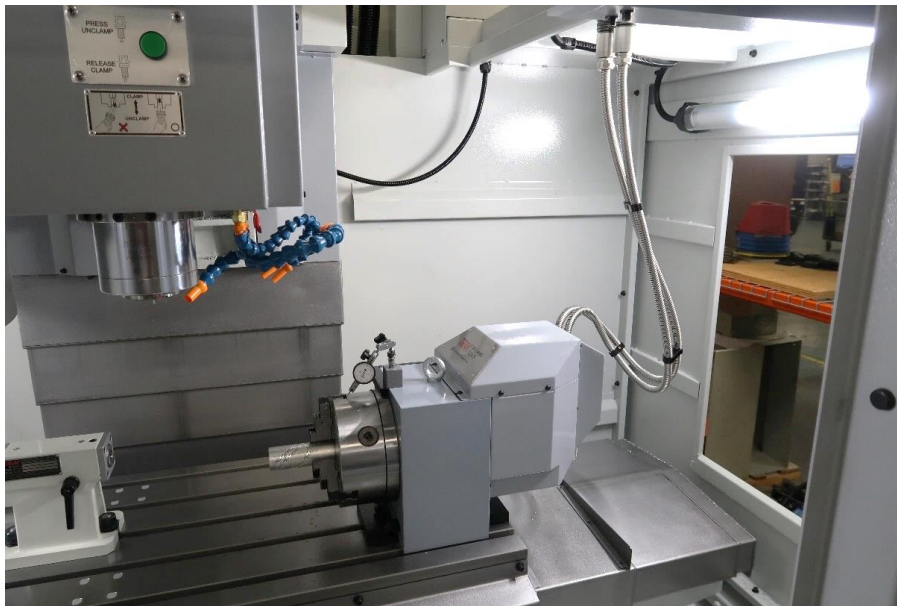


Figure 4.l – 4th Axis displaying the (2) cables that are to be connected and disconnected each time the unit is added or removed from the machine.

4.8 Automation Ready Option

4.8.1 Automation Ready Feature Descriptions

Feature	Description
Remote Cycle Start Input:	Will allow the machine to run any program that is ready to be executed. Example: When the operator presses the Cycle Start button.
Remote Cycle Stop Input:	Will allow the machine to stop at any point in the program.
4 x Automation Request Digital Outputs	M Code (M22-M29) controlled outputs that may be turned OFF or ON. These outputs can also be enabled to cause the machine to stop motion and wait for the Automation Finish Input, via the option screen. See section 13.3 for more details regarding M Codes.
Automation Request Finish Input	Used with all the Automation Request Outputs to indicate to the CNC that the Robot has completed the task associated with the Automation Request Output and will cause the machine to continue running the program.
Vise(s) Clamped Input	Signals the CNC system that the vise or vises are in an unclamped state, triggering an alarm and preventing program execution on the machine. Manual movement of the machine remains available. This input can be disabled through the option screen.
Cycle End Output	Will turn on when an M30 end of program has been activated.
E-stop Input	A pair of dual contacts which would trigger an E-stop condition on the CNC. Requires Normally Open contacts and should be closed when the CNC is running.
E-stop Output	A pair of relay contacts that may be used to connect the CNC E-stop to the automation connected with the machine.
Front Door Close Output	A pair of relay contacts that may be used to indicate to the automation system that the operator front door is closed.
Front Door Open Output	With the Front Door Option installed, the Front Door Open switch provides a normally open contact. This functions as an indicator, signaling the fully open position of the machine's Front Door.
Side Door Open Output	With the Side Door Option installed, the Front Door Open switch provides a normally open contact. This functions as an indicator, signaling the fully open position of the machine's Side Door.
Side Door Close Output	A pair of relay contacts that may be used to indicate to the automation system that the operator side door is closed.
Request Front Door Open Input	With the Front Door Option installed, the input will trigger the Front Door to open if there is No E-stop condition and the spindle is not turning.
Request Front Door Close Input	With the Front Door Option installed, the input will trigger the Front Door to Close if there is No E-stop condition.
Request Side Door Open Input	With the Side Door Option installed the input will trigger the Side Door to open if there is No E-stop condition and the spindle is not turning.
Request Side Door Close Input	With the Side Door Option installed, the input will trigger the Side Door to Close if there is No E-stop condition.
Network port	Used for programming and debugging by the manufacturer.

4.8.2 Enable / Disable Automation Ready

TRAK has a customized screen available for enabling and disabling the Automation Ready Options. To access the screen, at the upper right corner of the display, press [M], press "SETUP" from the left VSK menu bar then press *TrakMT Options* in figure 4.8.2a seen below from the HSK:



Figure 4.8.2a

The user will need to scroll down to the Automation Ready section.

Automation Ready option can be Enabled or Disabled in the select drop down menu (see figure 4.8.2b). The machine does not need to be restarted. When Enabled all features listed on the specification page will be available.

As shown in figure 4.8.2b the *Wait for Finish Input on Automation Request* can be individually enabled to wait for the Automation Finish Input. When the selection is disabled the corresponding M Code will turn the output ON and OFF.

As shown in figure 4.8.2b when *Select Vise(s) Unclamp Alarm Input* is Enabled the machine will expect an input from the workholding fixture or vise that it is clamped before running the machine. When disabled the CNC will ignore all input and run the program regardless of the state of the input.

The inputs and outputs can also be monitored in the TRAK Diagnostic Window (see figure 4.8.2c for an example).

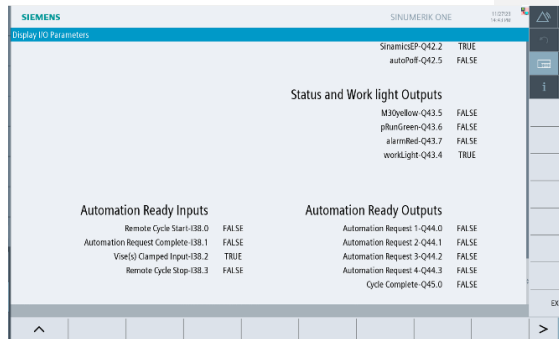
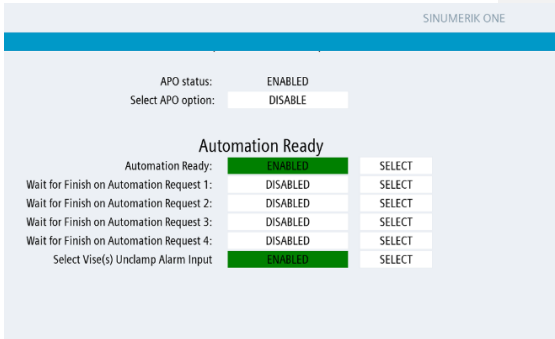


Figure 4.8.2c

4.8.3 Interface Specifications

P 17		X333		P17		P18		P19		P20		P21		P22		P23		P24		P25		P26		P27		P28		P29		P30		P31		P32		P33		P34		P35		P36		P37		P38		P39		P40		P41		P42		P43		P44		P45		P46		P47		P48		P49		P50		P51		P52		P53		P54		P55		P56		P57		P58		P59		P60		P61		P62		P63		P64		P65		P66		P67		P68		P69		P70		P71		P72		P73		P74		P75		P76		P77		P78		P79		P80		P81		P82		P83		P84		P85		P86		P87		P88		P89		P90		P91		P92		P93		P94		P95		P96		P97		P98		P99		P100		P101		P102		P103		P104		P105		P106		P107		P108		P109		P110		P111		P112		P113		P114		P115		P116		P117		P118		P119		P120		P121		P122		P123		P124		P125		P126		P127		P128		P129		P130		P131		P132		P133		P134		P135		P136		P137		P138		P139		P140		P141		P142		P143		P144		P145		P146		P147		P148		P149		P150		P151		P152		P153		P154		P155		P156		P157		P158		P159		P160		P161		P162		P163		P164		P165		P166		P167		P168		P169		P170		P171		P172		P173		P174		P175		P176		P177		P178		P179		P180		P181		P182		P183		P184		P185		P186		P187		P188		P189		P190		P191		P192		P193		P194		P195		P196		P197		P198		P199		P200	
------	--	------	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	-----	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--	------	--

All the connections between the CNC and the Automation (Robot) are to be done in the Automation Ready Box.

All input and output terminals are push-in terminals. Recommend use 22-16awg with ferrule.

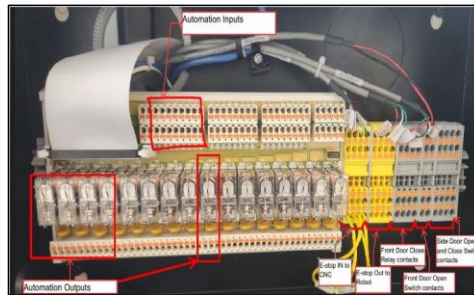


Figure 4.8.3a

4.8.3.1 E-Stop Input to CNC

Dual channel E-stop input to CNC. When connecting Automation to the E-stop input terminals, remove jumpers, and add contacts to corresponding terminals.

- Channel 1 (24-T12B) between TB901 and TB902
- Channel 2 (0-T22B) between TB903 and TB904.
- Internal Circuit Power 24V, 100mA

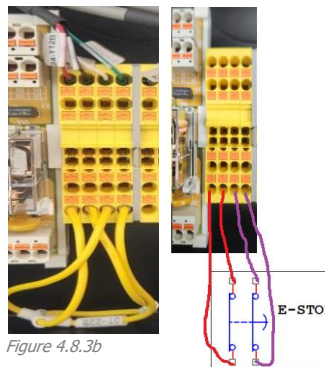


Figure 4.8.3b

4.8.3.2 E-Stop Output to Automation (Robot)

Dual channel E-stop Relay Output contacts. These contacts will open when any of the E-stops is pressed on the CNC or power is turned off.

- Channel 1 (EO1) is between TB905 and TB906
- Channel 2 (EO2) between TB907 and TB908.
- E-stop output circuits are rated up to 250V, 3amps.

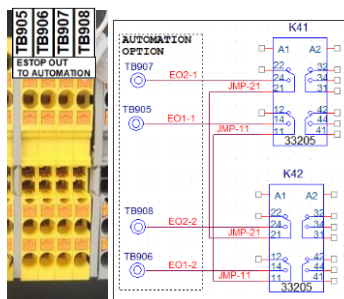


Figure 4.8.3d

4.8.3.3 Front Door Close Output

This is a dual channel Relay Output contacts. These contacts will open when the Front Door is not fully closed.

- Channel 1 (FDCR11 & FDCR12) is between TB909 and TB910.
- Channel 2 (FDCR21 & FDCR22) between TB911 and TB912.
- Door Close circuits are rated up to 250V, 3amps.

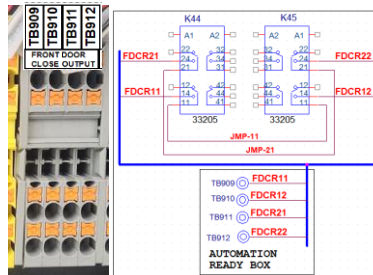


Figure 4.8.3e

4.8.3.4 Automation Ready Outputs:

Automation Request and Cycle End:
Circuit and relay contact output SPDT, 250V, 3A.

- Automation Request 1: Relay-Y0, Internal Signal: Q44.0
- Automation Request 2: Relay-Y1, Internal Signal: Q44.1
- Automation Request 3: Relay-Y2, Internal Signal: Q44.2
- Automation Request 4: Relay-Y3, Internal Signal: Q44.3
- Cycle End (M30): Relay-Y8 Internal Signal: Q45.0

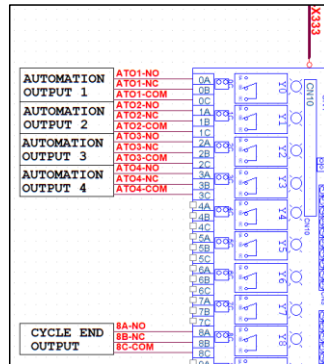


Figure 4.8.3f

4.8.3.5 Automation Ready Inputs

Inputs require a contact closure between P0 and associated input to activate.

- Input nominal rating: 24V 10mA, Sourcing Digital input.
- P0 provided internal 24V for inputs.
- (Remote Cycle Start), Signal: I38.0
Connection: P0 & A2 terminals on x333 I/O board.
- (Remote Cycle Stop), Signal: I38.3
Connection: P0 & B3 terminals on x333 I/O board.
- (Automation Finish), Signal: I38.1
Connection: P0 & B2 terminals on x333 I/O board.
- (Vise Clamp), Signal: I38.2
Connection: P0 & A3 terminals on x333 I/O board.

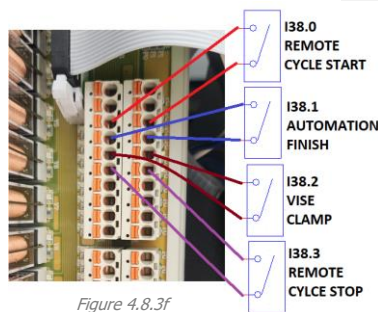


Figure 4.8.3f

4.8.3.6 Front Door Open Output

This feature is available with the Auto Front Door Option. This is a Normally Open single switch contact that is triggered when the Front Door is fully open.

- Circuits Rating Max: 250V, 3amps.
- (FDO23 & FDO24) between TB913 and TB914.

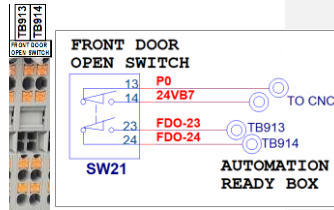


Figure 4.8.3g

4.8.3.7 Request Front Door Open/Close Inputs

This feature is available with the Auto Front Door Option.

- Input nominal rating: 24V 10mA, Sourcing Digital input.
- (Request Front Door Open) Signal: I38.6
Connection: P0 & A5 terminals on x333 I/O board.
- (Request Front Door Close) Signal: I38.7
Connection: P0 & B5 terminals on x333 I/O board.

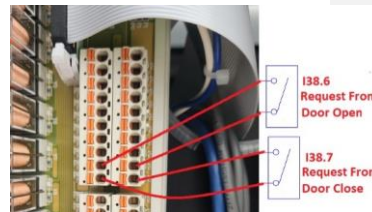


Figure 4.8.3h

4.8.3.8 Side Door Open and Close Output

This feature is available with the Auto Side Door Option. These are Normally Open single switch contacts that are triggered when the Side Door is fully open or fully closed.

- Circuits Rating Max: 250V, 3amps.
- (SDO23 & SDO24) between TB915 and TB916.
- (SDC23 & SDC24) between TB917 and TB918.

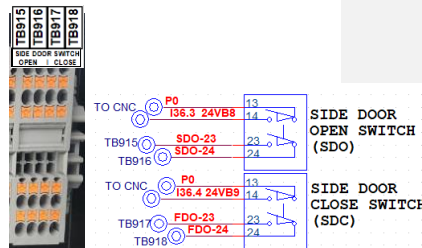


Figure 4.8.3i

4.8.3.9 Request Side Door Open/Close Inputs

This feature is available with the Auto Side Door Option.

- Input nominal rating: 24V 10mA, Sourcing Digital input.
- (Request Side Door Open) Signal: I38.4
Connection: P0 & A4 terminals on x333 I/O board.
- (Request Side Door Close) Signal: I38.5
Connection: P0 & B4 terminals on x333 I/O board.

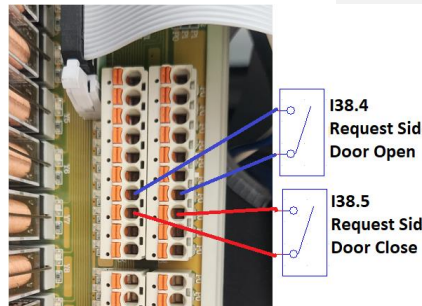


Figure 4.8.3j

4.8.3.10 Automation Box Cable Entry Port

The Automation Box has three sets of 3 different standard hole sizes for attaching conduits or cable glands for wiring all the external connection going to a Robot Arm or Automation system. Each hole is sealed with a panel plug, that will need to be removed when adding a cable conduit fitting or cable gland.

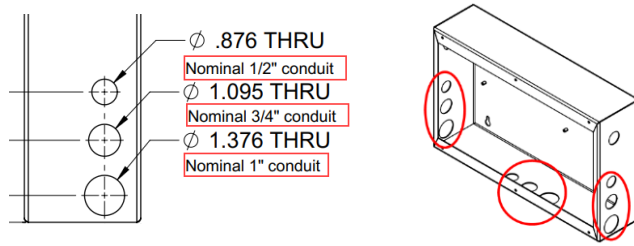


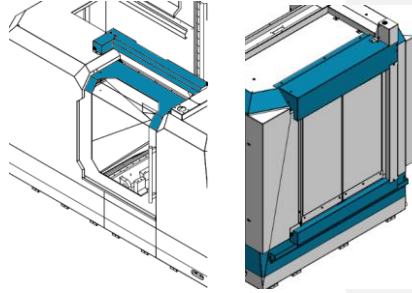
Figure 4.8.3k

4.9 VMCsi Auto Doors

TRAK manufactures Auto Front and Side Door options that will allow an end user to have the machine open and close the Front or a Side door automatically.

The Auto Door key features include:

- Position and forced controlled servo motor.
- End of program auto open.
- Machine cycle stop with Auto Door open.
- Digital I/O controlled.
- Front Door open and close outputs contacts.
- Side Door open and close outputs contacts.
- Front and Side Door remote open/close request.



4.9.1 Features Description

Feature	Description
MCP Auto Door Open/Close	When enabled the Auto Door will toggle open and close with every press of the MCP Auto Door button.
M0 Auto Door Open	When enabled the Auto Door will open automatically when an M0 is processed.
M30 Auto Door Open	When enabled the Auto Door will open automatically when a M30 is processed.
Auto Door Close with Remote Cycle Start	When enabled the Auto Door will close whenever a Remote Cycle Start input is triggered.
Front Door Close Output	These are a pair of relay contacts that can be used to indicate to the automation system that the operator Front Door is closed.
Front Door Open Output	If Front Door Option is installed. This is a normally open contact from the Front Door open switch that can be used to indicate when the machines front door is fully open.
Side Door Open Output	If Side Door Option is installed. This is a normally open contact from the Side Door open switch that can be used to indicate when the machines Side Door is fully open.
Side Door Close Output	If Side Door Option is installed. This is a normally open contact from the Side Door close switch that can be used to indicate when the machines Side Door is fully closed.
Request Front Door Open Input	If Front Door Option is installed. This input will trigger the Front Door to open if there is No E-stop condition and the spindle is not turning.
Request Front Door Close Input	If Front Door Option is installed. This input will trigger the Front Door to close if there is No E-stop condition.

Feature	Description
Request Side Door Open Input	If Side Door Option is installed. This input will trigger the Side Door to open if there is No E-stop condition and the spindle is not turning.
Request Side Door Close Input	If Side Door Option is installed. This input will trigger the Side Door to close if there is No E-stop condition.
M-code driven Auto Door Open	When triggered it will cause the Auto Door to open. M33 – Open Front Door M35 – Open Side Door

4.9.2 Enable/Disable Auto Doors

TRAK has a customized screen available for enabling and disabling the Automation Ready Options. To access the screen, at the upper right corner of the display, press [M], press "SETUP" from the left VSK menu bar then press "TrakMT Options" from the HSK as shown below:



Figure 4.9.2a

4.9.2.1 Auto Front/Side Door Option

can be Enabled or Disabled in the select drop down menu. The selection will take effect immediately, no need to restart the machine. When "Enabled" all the features described in the specification page will be available.

4.9.2.2 Open Auto Front/Side Door with M0:

When Enabled it will cause the Auto Door to open everytime an M0 is processed.

4.9.2.3 Open Auto Front/Side Door with M30:

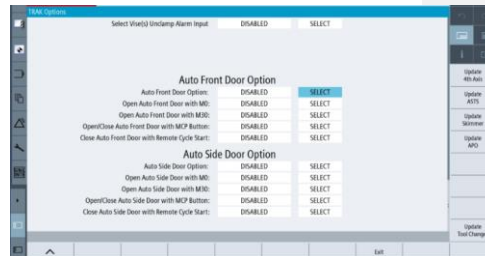
When Enabled it will cause the Auto Door to open everytime an M30 is process.

4.9.2.4 Open/Close Auto Front/Side Door with MCP Button:

When Enabled it will cause the Auto Door to toggle between open and close every time the MCP Auto Door button is pressed. Note if both Front and Side doors are enabled it will toggle both at the same time.

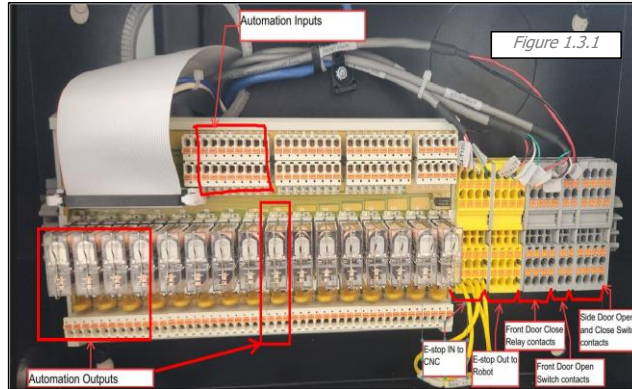
4.9.2.5 Close Auto Front/Side Door with Remote Cycle Start:

When Enabled it will cause the Front/Side Door to close whenever the Remote Cycle Start input is set high(24V).



4.9.3 Interface Specifications

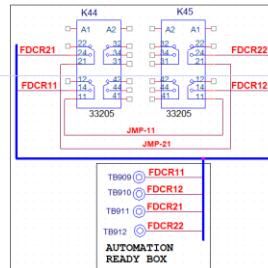
All the I/O connections for the Auto Front/Side Door are available in the Automation Ready Box.



4.9.3.1 Front Door Close Output

This is a dual channel Relay Output contacts. These contacts will open when the Front Door is not fully closed.

- Channel 1 (FDCR11 & FDCR12) is between TB909 and TB910.
- Channel 2 (FDCR21 & FDCR22) between TB911 and TB912.
- Door Close circuits are rated up to 250V, 3amps.



Commented [LG1]: [@Roman Ortu](#) Please review these 2 sentences. You use "is" but then use contacts in a plural form. Which is correct?

Commented [RO2R1]: [@Leo Gammatori](#) the contacts are plural but the door is single thus "is" in this case is the door in this case.

4.9.3.2 Front Door Open Output

This feature is available with the Auto Front Door Option. This is a Normally Open single switch contact that is triggered when the Front Door is fully open.

- Circuits Rating Max: 250V, 3amps.
- (FDO23 & FDO24) between TB913 and TB914.

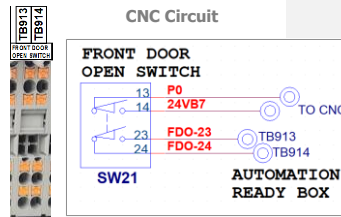


Figure 1.3.2

4.9.3.3 Request Front Door Open/Close Inputs

This feature is available with the Auto Front Door Option.

- Input nominal rating: 24V 10mA, Sourcing Digital input.
- (Request Front Door Open) Signal: I38.6
Connection: P0 & A5 terminals on x333 I/O board.
- (Request Front Door Close) Signal: I38.7
Connection: P0 & B5 terminals on x333 I/O board.

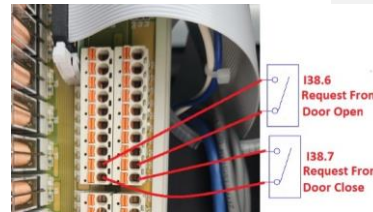


Figure 1.3.3

4.9.3.4 Side Door Open and Close Output

This feature is available with the Auto Side Door Option. These are Normally Open single switch contacts that are triggered when the Side Door is fully open or fully closed.

- Circuits Rating Max: 250V, 3amps.
- (SDO23 & SDO24) between TB915 and TB916.
- (SDC23 & SDC24) between TB917 and TB918.

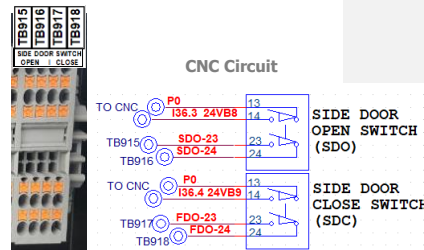


Figure 1.3.4

4.9.3.5 Request Side Door Open/Close Inputs

This feature is available with the Auto Side Door Option.

- Input nominal rating: 24V 10mA, Sourcing Digital input.
- (Request Side Door Open) Signal: I38.4
Connection: P0 & A4 terminals on x333 I/O board.
- (Request Side Door Close) Signal: I38.5
Connection: P0 & B4 terminals on x333 I/O board.

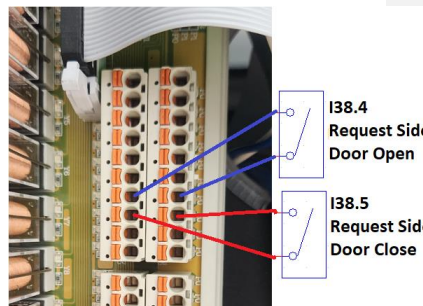


Figure 1.3.5

4.9.3.6 Automation Ready Inputs (Reference)

Inputs require a contact closure between P0 and associated input to activate.

- Input nominal rating: 24V 10mA, Sourcing Digital input.
- P0 provided internal 24V for inputs.
- (Remote Cycle Start), Signal: I38.0
Connection: P0 & A2 terminals on x333 I/O board.
- (Remote Cycle Stop), Signal: I38.3
Connection: P0 & B3 terminals on x333 I/O board.
- (Automation Finish), Signal: I38.1
Connection: P0 & B2 terminals on x333 I/O board.
- (Vise Clamp), Signal: I38.2
Connection: P0 & A3 terminals on x333 I/O board.

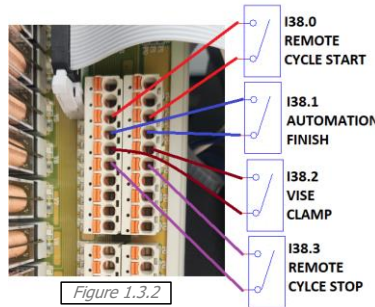
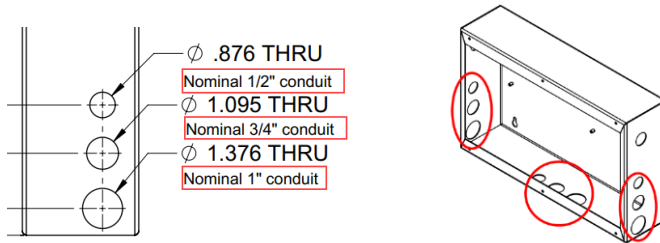


Figure 1.3.2

4.9.3.7 Automation Box Cable Entry Port (Reference)

The Automation Box has three sets of 3 different standard hole sizes for attaching conduits or cable glands for wiring all the external connections going to a Robot Arm or Automation system. Each hole is sealed with a panel plug, that will need to be removed when adding a cable conduit fitting or cable gland.



4.9.4 Troubleshooting

Message		Messages are information displayed on machine that will not cause the machine current activity to stop.
#	Description	Definition
701247	Auto Front Door Opening	<ul style="list-style-type: none"> • This message will be displayed when the door is in the process of opening due to a command from the MCP, M-code or Digital Input Request • This message will be cleared automatically when the door reaches the open position or the option has been disabled.
701248	Auto Front Door Fully Open	<ul style="list-style-type: none"> • This message will be displayed when the door has triggered the Door Open Switch. • This message will be cleared automatically when the door is off the Door Open Switch or the option has been disabled.
701249	Auto Front Door Partially Open	<ul style="list-style-type: none"> • This message will be displayed when the door is not on the Open or Close switch. • This message will be cleared automatically when the door is on the open or close switch or the option has been disabled.
701250	Auto Front Door Closing	<ul style="list-style-type: none"> • This message will be displayed when the door is in the process of closing due to a command from the MCP, M-code or Digital Input Request • This message will be cleared automatically when the door reaches the closed position or the option has been disabled.
701252	Auto Side Door Opening	<ul style="list-style-type: none"> • This message will be displayed when the door is in the process of opening due to a command from the MCP, M-code or Digital Input Request • This message will be cleared automatically when the door reaches the open position or the option has been disabled.

Commented [LG3]: [@Roman Ortiz](#) Sentence needs rewording.

Message		Messages are information displayed on machine that will not cause the machine current activity to stop.
#	Description	Definition
701253	Auto Side Door Fully Open	<ul style="list-style-type: none"> This message will be displayed when the door has triggered the Door Open Switch This message will be cleared automatically when the door is off the Open Switch or the option has been disabled.
701254	Auto Side Door Partially Open	<ul style="list-style-type: none"> This message will be displayed when the door is not on the Open or Close switch. This message will be cleared automatically when the door is on the open or closed switch or the option has been disabled.
701255	Auto Side Door Closing	<ul style="list-style-type: none"> This message will be displayed when the door is in the process of closing due to a command from the MCP, M-code or Digital Input Request This message will be cleared automatically when the door reaches the closed position or the option has been disabled.

Commented [LG3]: @Roman Orti Sentence needs rewording.

5.0 Machine Mode

The Machine menu displays information such as the axes positions, distances to go, feed rates, spindle speeds, active tool, etc. Functionality such as manual tool selection, tool/work-piece measurement, handwheel selection can be performed by selecting the relevant soft-key. The functionality depends upon the selected operating mode.

Vertical Softkeys

Below are some vertical softkeys that will be repeated while in Machine Mode. Within the individual sub-modes, we will list any VSKs that are unique to that specific sub-mode.

- **G Functions** - The most important G-functions are displayed in a sub-window
 1. From Machine Mode, go to JOG, MDA/MDI or AUTO sub-mode.
 2. Press the "G functions" softkey. The "G Functions" window is opened.
 3. Press the "G functions" softkey again to hide the window.
- **Auxiliary Functions** - Available auxiliary functions are displayed in a sub at the time of the output.
 1. From Machine Mode, go to JOG, MDA/MDI or AUTO sub-mode.
 2. Press the "Auxiliary Functions " softkey. The "Auxiliary Functions" window is opened.
 3. Press the "Auxiliary Functions" softkey again to hide the window.
- **Act. values Machine** - The coordinate system will be toggled between the machine coordinate system (MCS) and the workpiece coordinate system (WCS).
- **All G functions** - All G-functions will be shown.
 1. From Machine Mode, go to JOG, MDA/MDI or AUTO sub-mode.
 2. Press the "All G functions" softkey. The "G Functions" window is opened.
 3. Press the "All G functions" softkey again to hide the window.
- **Zoom act. val.** - All actual axes positions in the selected coordinate system as well as the currently active feed rate and feed override of each individual axis are displayed full screen.

5.1 Automatic (AUTO) sub-mode

In this section, we will go over the HSKs within the sub-mode, and describe some basic operation procedures. Below is a diagram of all the VSK and HSK options found within the AUTO sub-mode for your reference.

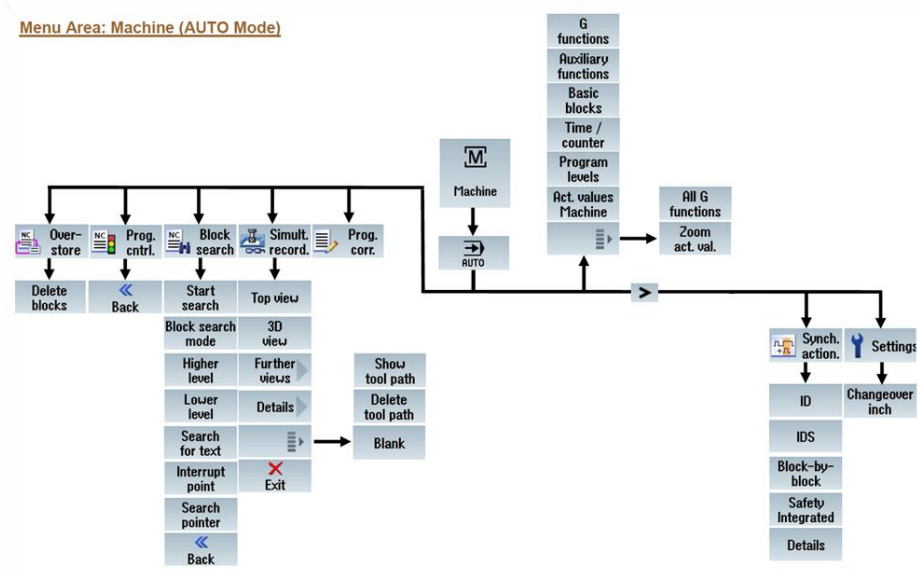


Figure 5.a - Machine Mode > Auto Sub-Mode: All menu options displayed

Vertical Softkeys

- **Basic Blocks** - all G-code commands that trigger a function on the machine will be displayed. The display updates both in the test operation and in the actual machining of the workpiece at the machine.
- **Time/Counter** - the program run time, the rest of the program run time and the number of machined workpieces will be displayed.
- **Program Levels** - you can display the current program level during the execution of a large program with several subprograms.

Horizontal Softkeys

Below is a brief description of the unique HSKs found in this sub-mode. More detailed descriptions and operations may be found in the sub-sections that follow.

- **Overstore** - allows you to overstore technological parameters (for example, auxiliary functions, axis feed, spindle speed, programmable instructions, etc.) for a program run in the main memory of the NCK.
- **Prog. Cntrl.** - The working window for controlling the program run will be opened.
- **Block Search** - Execute program from any point selected or from the last program run interrupt point.
- **Simult. Record** - You will see a full machine simulation of the current status of the machine during machining of the workpiece, to monitor the result of the programming.

- **Prog. Corr.** - (program correction) The program editor opens.
- **Synch action** - the screen which shows the current synchronized actions is displayed.
- **Settings** - a window opens up where you can adjust the settings for manual operation on the SINUMERIK ONE.

5.1.1 Overstore

In the "Overstore" editor view you can overstore technological parameters (for example, auxiliary functions, axis feed, spindle speed, programmable instructions, etc.) for a program run in the main memory of the NCK.

NOTE - The program to be corrected has to be in the STOP or RESET mode. The programs in the part program memory are not changed while using the function "Overstore". You cannot change the operating mode while you are in overstore mode.

Procedure

1. Press the "Overstore" HSK. The "Overstore" window will open.
2. Enter the required data and NC block.
3. Press the "CYCLE START" key on the MCP. The blocks you have entered are stored. You can observe execution in the "Overstore" window.
4. After the entered blocks have been executed, you can append blocks again.
5. Press the "Back" VSK. The "Overstore" window closes.
6. Press the "CYCLE START" key on the MCP again. The program selected before overstoreing continues to run.

NOTE - Block-by-block execution - The <SINGLE BLOCK> key is also active in the overstore mode. If several blocks are entered in the overstore buffer, then these are executed block-by-block after each NC start.

Deleting Blocks

1. Press the "Delete blocks" VSK to delete program blocks you have entered.

5.1.2 Prog. Cntrl.

You can control the program sequence however you wish by selecting and clearing the relevant checkboxes. This HSK is also available in MDA/MDI Mode and TEACH-IN Mode.

NOTE - The "Program Control - Skip Blocks" window is only available when more than one skip level is set up.

Program Control Option Descriptions

- **PRT** (no axis motion) - The program is started and executed with auxiliary function outputs and dwell times. In this mode, the axes are not traversed. The programmed axis positions and the auxiliary function outputs are controlled this way.
NOTE - Program processing without axis motion can also be activated with the function "Dry run feedrate".
- **DRY** (Dry run feedrate) - The traversing velocities programmed in conjunction with G1, G2, G3, CIP and CT are replaced by a defined dry run feedrate. The dry run feedrate also applies instead of the programmed revolution feedrate.

CAUTION!

Workpieces must not be machined when "Dry run feedrate" is active because the altered feed rates might cause the permissible tool cutting rates to be exceeded and the workpiece or machine tool could be damaged.

- **RG0** (Reduced rapid trav.) - In the rapid traverse mode, the traversing speed of the axes is reduced to the percentage value entered in RG0.
- **M01** (Programmed stop 1) - The processing of the program stops at every block in which supplementary function "M01" is programmed. In this way you can check the already obtained result during the processing of a workpiece.
- **DRF** (Handwheel offset) – Enables an additional incremental zero offset while processing in automatic operation mode with an electronic handwheel. This function can be used to compensate for tool wear within a programmed block.
- **SB** (Individual blocks)
 - **SB1** – Single block, coarse: The program stops only after blocks which perform a machine function.
 - **SB2** – Data block: The program stops after each block.
 - **SB3** – Single block, fine: The program stops also in cycles after blocks, which perform a machine function.
- **SKP** – Skipped blocks are skipped during machining.
- **MRD** - In the program, the measurement results screen display is active while machining.

5.1.2.1 Controlling the program run

1. Press the "Prog. cntrl." HSK. The "Program control" window opens and shows a list of program control options.
2. Select the desired program controls. *See descriptions above.*
3. Press the "Back" VSK to go back to the main screen in the sub-mode "AUTO".

5.1.3 Block Search

Block Search is used to start a program from any point using a search for previous modal commands (i.e., M-Code activation, work coordinate, etc.).

5.1.3.1 Start a Block Search

1. A desired program is selected and the machining was discontinued with pressing "RESET" or "CYCLE STOP" or the control unit is generally in RESET state.
2. Press the "Block search" HSK.

5.1.3.2 Simple search target definition

1. Start a block search. *Shown above.*
2. Select a particular program block on the screen.

5.1.3.3 Higher Level

1. Use this VSK to change the program level you want to search.

5.1.3.4 Lower Level

1. Use this VSK to change the program level you want to search.

5.1.3.5 Search for text

1. Start a Block search.
2. Press the "Search for text" VSK.
3. Select the search direction.

4. Enter the search text and confirm.
5. Press the "OK" VSK.
6. Press the "Start search" VSK. The search starts. Your specified search mode will be taken into account (indicated in the upper blue title bar of the search target window). The current block will be displayed and marked in the "Program" window as soon as the target is found.
7. If the located target (for example, when searching via text) does not correspond to the program block, press the "Start search" VSK again until you find your target.
8. Press the "CYCLE START" key on the MCP twice. Processing is continued from the defined position.

5.1.3.6 Interrupt point

1. Start a Block Search.
2. Press the "Interrupt point" VSK. The interruption point is loaded.
3. If the "Higher level" VSK and the "Lower level" VSK are available, use these to change the program level.
4. Press the "Start search" VSK. The search starts. The specified search mode will be taken into account (indicated in the upper blue title bar of the search target window).
5. The search screen closes. The current block will be displayed and marked in the "Program" window as soon as the target is found.
6. Press the "CYCLE START" key on the machine control panel (MCP) twice. The execution will continue from the interruption point.

5.1.3.7 Search pointer

1. Start a block search.
2. Press the "Search pointer" VSK. The "Search pointer" window opens.
3. Enter the full path of the program as well as the subprograms, if required, in the input fields.
4. Press the "Start search" VSK. The search starts. The specified search mode will be taken into account (indicated in the upper blue title bar of the search target window).
5. The search screen closes. The current block will be displayed and marked in the "Program" window as soon as the target is found.
6. Press the "CYCLE START" key on the machine control panel (MCP) twice. Processing is continued from the defined location.

5.1.4 Simult. Record

Before machining the workpiece on the machine, you can graphically display the execution of the program on the screen to monitor the result of the programming. You can replace the programmed feedrate with a dry run feedrate to influence the speed of execution. Simultaneous record can also be turned on, if machining is already running. You can also use simultaneous recording during machining of a workpiece. This helps if the view towards the inside of the cabin is obstructed by coolant. In each different view of the "Simultaneous recording" window you can adjust the view by using the blue cursor keys and zoom in or out by using the plus ("+") and minus ("-") keys on the keyboard. The traversing paths of the tool in the "Simultaneous recording" window are displayed in different colors: red for rapid traverse and green for feed motion.

5.1.4.1 Before machining of the workpiece

1. Load a program in AUTO mode.
2. Press the "Prog. cntrl." HSK and activate the checkboxes "PRT" and "DRY".
The program is executed without axis movement. The programmed feedrate is replaced by a dry run feedrate.

-OR-

Leave the "DR" box unchecked. Simultaneous recording is performed with the programmed feedrate.

3. Press the "Simult. Record" HSK. The "Simultaneous recording" window opens.
4. Press the "CYCLE START" key on the MCP. The execution of the program on the machine is started and displayed graphically on the screen.
5. Press "CYCLE STOP" to stop machining and the "Simult. record." HSK again to close the "Simultaneous recording" window.

5.1.4.2 During machining of the workpiece

1. Press the "Simult. Record" HSK. The "Simultaneous recording" window opens.
2. Press the "CYCLE START" key on the MCP. The machining of the workpiece is started and graphically displayed on the screen.
3. Press the "CYCLE STOP" key and the "Simult. record." HSK again to stop the recording and to close the "Simultaneous recording" window.

5.1.5 Prog. Corr.

In this HSK, the program editor window opens for any corrections of the program. It is essentially a shortcut key that takes you back to "Program Mode".

- STOP state: Only program lines that have not yet been executed can be edited.
- RESET state: All program lines can be edited

5.1.6 Settings

The following HSK will re-direct you to the settings for automatic mode.

Vertical Softkeys

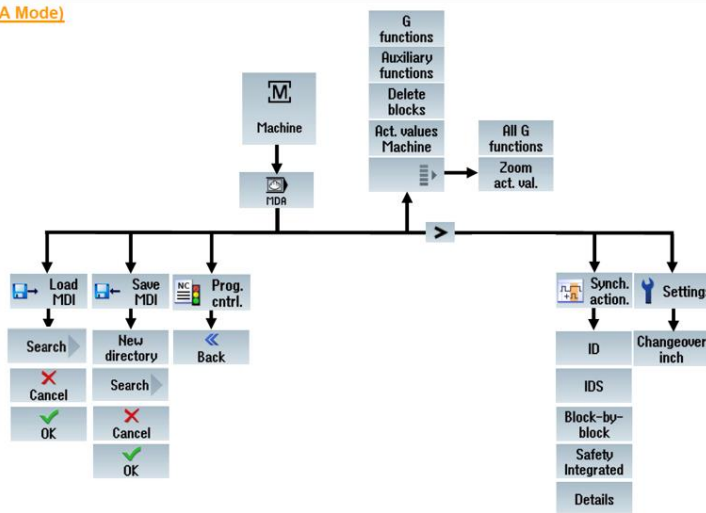
- **Changeover inch** – the measuring units are converted from the metric to the imperial (inch) dimension system. New values have to be entered in inches. By pressing this key the key function switches to "Changeover metric".
- **Changeover metric** - the measuring units are converted from the imperial (inch) to the metric dimension system. New values have to be metric. By pressing this key the key function switches to "Changeover inch".

5.2 Machine Data Automatic (MDA/MDI) sub-mode

In "MDA/MDI" mode (Manual Data Automatic), you can enter G-code commands block-by-block and immediately execute them for setting up the machine. You can load an MDA/MDI program straight from the Program Manager into the MDA/MDI buffer. You may also store programs which were rendered or changed in the MDA/MDI operating window into any directory of the program manager.

In this section, we will go over the HSKs within the sub-mode, and describe some basic operation procedures. Below is a diagram of all the VSK and HSK options found within the MDA/MDI sub-mode for your reference.

Menu Area: Machine (MDA Mode)



Vertical Softkeys

- **Delete Blocks** – The entered program blocks can be deleted.

Horizontal Softkeys

- **Load MDI** - The program manager window opens so you can select the program you want to run.
- **Save MDI** – Manual Data Input – If you need to write a brief G-Code program, you can run it and save it.
- **Prog. Cntrl.** - You can control the program sequence however you wish by selecting and clearing the relevant checkboxes. Please see section [5.1.2](#) for information on this HSK.
- **Settings** - Please see section [5.1.6](#) for information on this HSK.

5.2.1 Loading an MDI-program

1. Press "Load MDI" – The "Load into MDI" Program manager window opens.
2. Mark the program you want to run. The selected program will be highlighted orange.
3. Press "OK" VSK.
4. The window closes and the program is ready for machining.

5.2.2 Saving an MDI-Program

1. Create the MDI program.
2. Press "Save MDI".
3. The "Load into MDI" Program manager window opens. It shows you a view of the program manager.
4. Select the drive to which you want to save the MDI program you have created.
5. Press "OK" VSK.
6. Enter a name for the program.
7. Press "OK" VSK.
8. The program will be saved under the specified name in the selected directory.

5.2.3 Editing/executing a MDI program

1. When you go to MDA/MDI sub-mode, the MDI editor opens.
2. Enter the desired G-code commands using the operator's keyboard.
-OR-
Enter a standard cycle, e.g., CYCLE62 ().

Editing G-code commands/program blocks

3. Edit G-code commands directly in the "MDI" window.
-OR-
Select the required program block (e.g., CYCLE62) and press the "OK" VSK.
When editing a cycle, either the help screen or the graphic view can be displayed.
4. Press the <CYCLE START> key on the MCP.
5. The control executes the input blocks.

When executing G-code commands and standard cycles, you have the option of controlling the sequence as follows:

- Executing the program block-by-block
- Testing the program - Settings under program control
- Setting the test-run feedrate - Settings under program control

5.2.4 Deleting an MDA/MDI program

Before you proceed, make sure the MDI editor contains a program that you created in the MDI window or loaded from the program manager.

Procedure

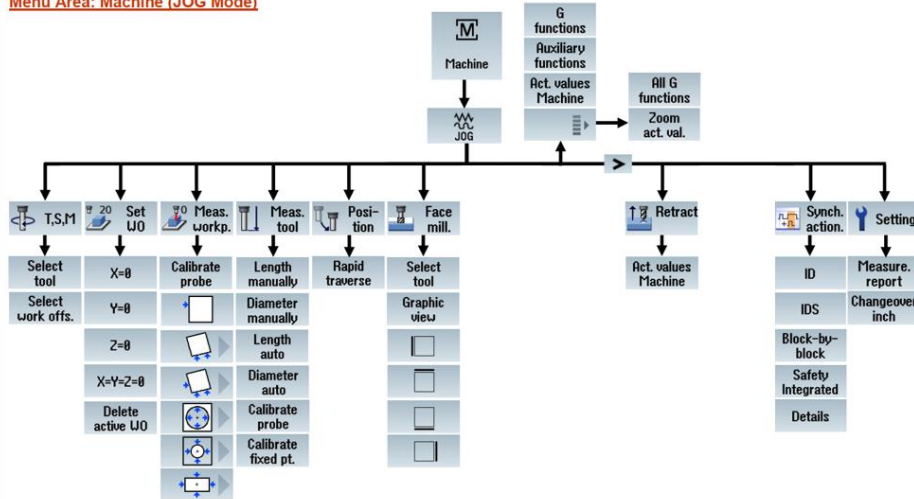
1. Press the "Delete blocks" softkey. The program blocks displayed in the program window are deleted.

5.3 JOG sub-mode

Operation mode "JOG" is used for setting up the machine for program runoff, or if you simply want to traverse the axes on the machine.

In this section, we will go over the HSKs within the sub-mode, and describe some basic operation procedures. Below is a diagram of all the VSK and HSK options found within the JOG sub-mode for your reference.

Menu Area: Machine (JOG Mode)



Horizontal Softkeys

- **T,S,M** – Tool, Spindle and Machine Commands
- **Set WO** - The input screen for setting the work offset will be activated.
- **Meas. Workp.** – The input mask for measuring a workpiece will be activated.
- **Meas. Tool** - The function "Measure tool" will be activated and the extended options "Length/Radius manual", "Length/Radius auto", "Calibrate Probe", and "Calibrate fixed pt." will be available in the VSK.
- **Position** - The input screen "Target position" will be activated.
- **Face mill.** - The input screen "Face milling" will be activated. This is not covered in this manual.

5.3.1 T, S, M (Tool, Spindle and Machine Commands)

When you select the T,S,M HSK, the bottom portion of the display window will be activated (blue title bar indicates the window is active). Here, the user may enter the tool, spindle and machine commands in the fields provided below.

Field Descriptions

- **T** – Tool Name
- **D** – Cutting edge number of the tool.
- **ST** – Sister Tool
- **Spindle** - Enter the spindle speed (RPM)
- **Spindle M Function** - Activate M functions
– There is a drop-down menu with a handful of options to select M-Functions from.
- **Other M Function** – Any other M-Function, not listed in the drop-down menu. You need to manually enter the code.
- **Work offset** - Activates work offsets
- **Meas. Un** - Change measurement units between mm and inches.
- **Machining plane** - Changes machining plane to XY, ZX or YZ

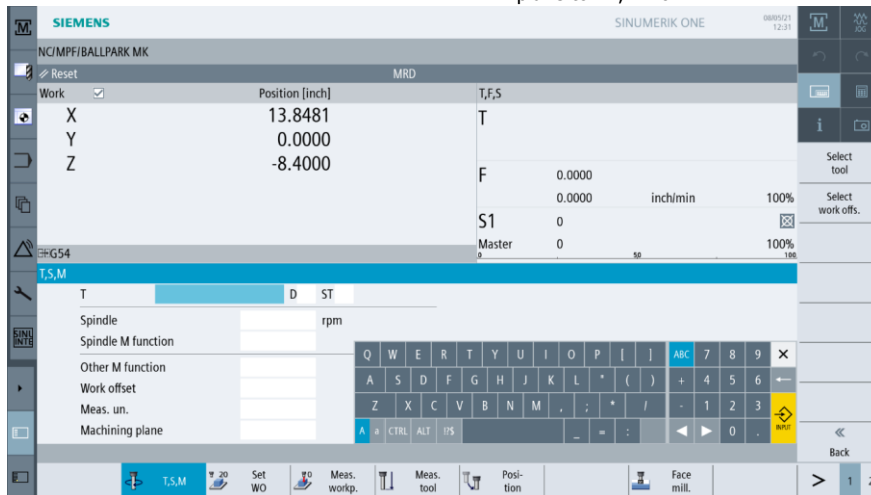


Figure 5.b - Inputting Tool, Spindle and Machine Commands

To change the tool

1. Select the field to the right of T.
2. Press "Select tool" VSK. The tool list will open.
3. Select a tool.
4. Click the "Ok" VSK.
5. Press "Cycle start"

To turn on the spindle

1. Enter the RPM value using the on-screen keyboard.
2. Select Direction under "Spindle M function"
3. Press "Cycle start"

To activate M-Functions

1. Select one of the drop-down options.
2. Press "Cycle start"

To activate any other M-Functions

1. Enter the M-Functions using the on-screen keyboard
2. Press "Cycle start"

To activate work offsets

1. Select the work offset.
2. Press "Cycle start"

To change measurement units

1. Select between "mm" or "inches".
2. Press "Cycle start"

To change machining plane

1. Select between "XY", "ZX", or "YZ".
2. Press "Cycle start"

5.3.2 Set WO

Setting the work offset is only available when you have a work coordinate active (press cycle start to activate).

Field Descriptions:

- X | Y | Z – The axis position / The selected axis you want to modify.
- A – Actual
- C – Current

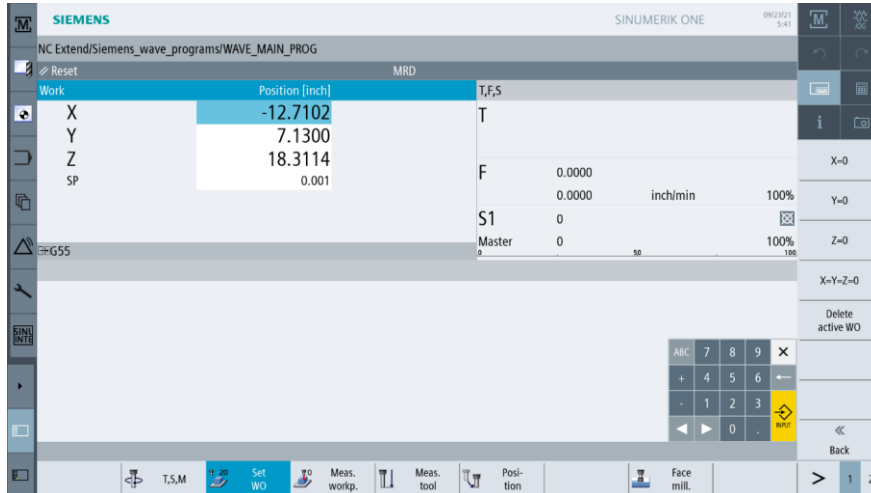


Figure 3 - Machine Mode > JOG Sub-mode > Set WO

5.3.2.1 Set work offsets

1. Press Set WO. The top left window will be editable.
2. Select the axis you would like to modify (it will be highlighted orange when selected)
3. Manually type in: The entered values for a zero-point offset of the axis will be accepted and displayed in the workpiece coordinate system (WCS). The difference between the original position to the new entered value will be written to the active zero-point offset.
- OR -
To reset any or all of the axis work offsets to zero, use the VSK's.
4. After setting a position or deleting a zero-point offset you switch back automatically to the main screen.

5.3.3 Meas. Workp.

The workpiece zero can be measured manually, or automatically with a part probe. The reference point for programming a workpiece is always the workpiece zero. You can determine the workpiece zero on the following workpiece elements; Edge, Corner, Pocket and hole, and Spigot.

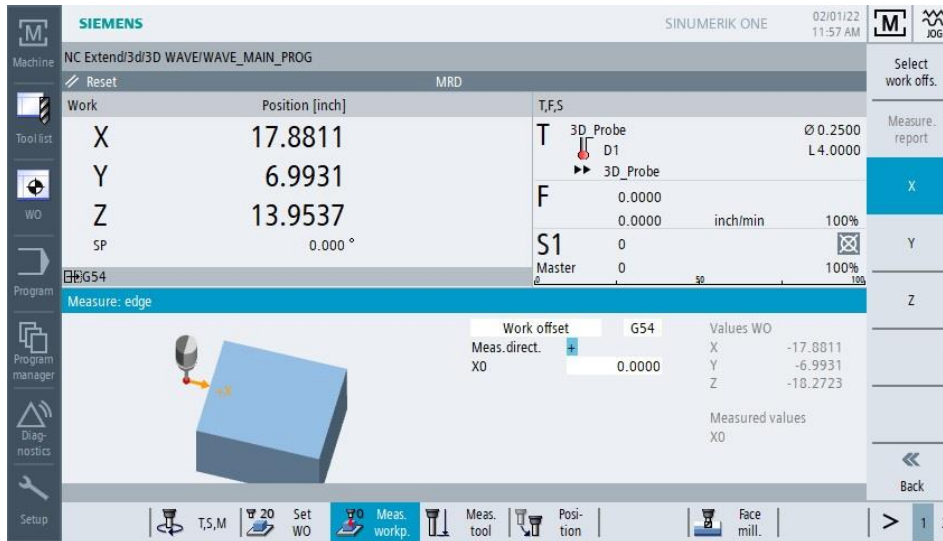


Figure 5.d - Machine Mode > Meas. workp.

5.3.3.1 Calibrate Work Probe

Please see *TRAK VMCSi Mills with Siemens Sinumerik CNC Safety, Installation, Maintenance, Service & Parts List Manual (P/N 34104)*, Section 2.15, for step-by-step instructions on how to calibrate the work probe.

5.3.3.2 Measure work piece automatically

For automatic measurements always use electronic 3-D or mono workpiece probes.

1. You must calibrate the electronic workpiece probes beforehand.
2. Press "Meas. Workp." VSK.
3. Press "Probing Geometry" VSK.
4. Follow the on-screen positioning guide, this will show you where to place the probe.
5. As soon as you start the process with the "CYCLE START" key, the workpiece probe automatically approaches the workpiece at measuring feedrate and then returns to the starting position at rapid traverse.

5.3.3.3 Measure work piece manually

To measure the zero point manually, you need to traverse your tool manually up to the workpiece. You can use edge probes, sensing probes, or dial gauges with known radii and lengths. You can also use any other tool of which you know the radius and length. The tools used for measuring must not be electronic probes.

5.3.4 Meas. Tool

The geometries of the machining tool must be taken into consideration when executing a part program. These are stored as tool offset data in the tool list. Each time the tool is called, the control considers the tool offset data. When programming the part program, you only need to enter the workpiece dimensions from the production drawing. After this, the controller independently calculates the individual tool path.

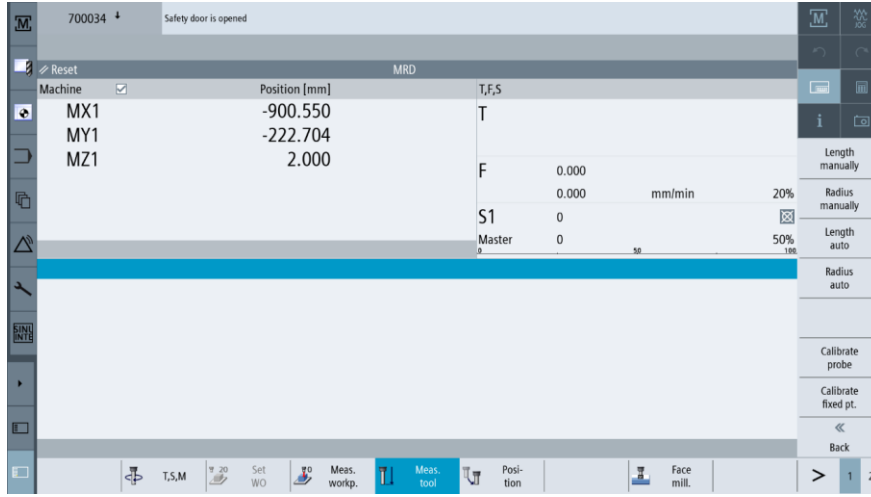


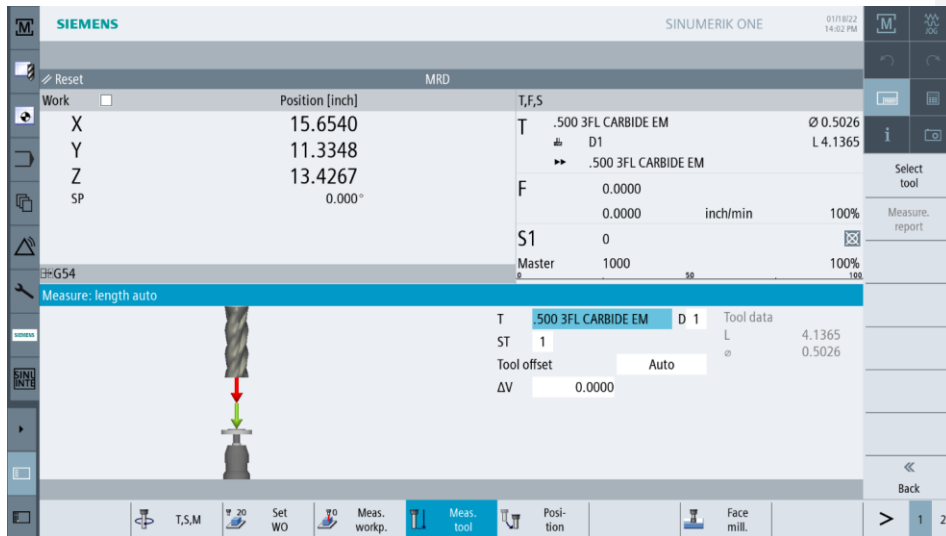
Figure 5.e - Machine Mode > Meas. tool

5.3.4.1 Calibrate tool probe

Please see *TRAK VMCSi Mills with Siemens Sinumerik CNC Safety, Installation, Maintenance, Service & Parts List Manual* (P/N 34104), Section 2.15, for step-by-step instructions on how to calibrate the tool probe.

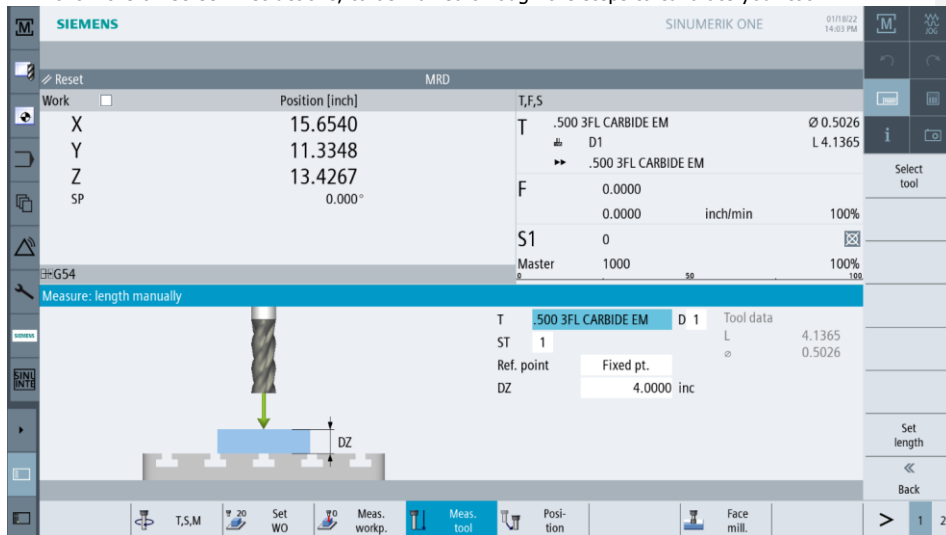
5.3.4.2 Measure tools automatically

1. Make sure the tool probe has been calibrated.
2. Press the "Length auto" VSK.
3. If the tool currently loaded is NOT the tool you are going to probe, then press "Select tool" VSK.
4. Once the correct tool is selected, press "CYCLE START" on the MCP.



5.3.4.3 Measure tools manually

1. Press "Calibrate fixed pt." VSK.
2. Touch the spindle off on a fixed, known location.
3. Load the tool you want to measure.
4. Touch the tool off the exact same point you touched off the spindle.
5. Now your fixed point has been calibrated.
6. Press "Length manually" VSK.
7. Follow the on-screen instructions, to be walked through the steps to calibrate your tool.



5.3.5 Position

In manual mode, you can traverse individual or several axes to certain positions in order to implement simple machining sequences. The feedrate / rapid traverse override is active during traversing.

Field Descriptions

- **F** – Feed rate
- **X | Y | Z** - Target position of selected axes
- **SP** – Degree of target angle

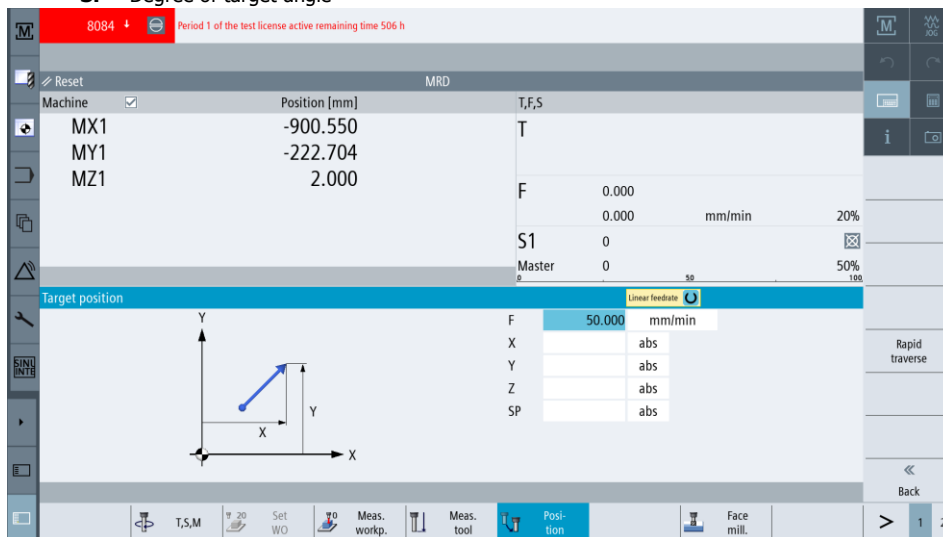


Figure 5.f - Position HSK

5.3.5.1 Command a position move (X, Y, Z, Spindle) at a set feed rate or rapid

1. Specify a value for "F" (feed rate),
-OR-
press "Rapid" VSK (this will display "Rapid tr." In the "F" field)
2. Enter the target position or target angle for the axis or axes to be traversed.
3. Press "CYCLE START" on the MCP
4. The axis/axes is/are traversed to the specified target position.

5.3.6 Face mill

This HSK opens up the face mill canned cycle, that requires programming. Please see the Siemens programming manual for more details.

5.3.7 Retract

The "Retract" function allows tools to be retracted in a safe direction after a program interruption. This is available after interrupting machining operations or a RESET at the machine control panel.

Retraction is useful when the coordinate system is swiveled, i.e., the infeed axis is not in the vertical position.

NOTE – In the case of tapping, the form fit between the tap and the workpiece is considered and the spindle moved according to the thread. The control will coordinate Z movement in relation to the spindle being jogged at the pitch of the threading operation that was interrupted.

5.3.7.1 Retract tool from part

1. The power feed to the machine is interrupted.
-OR-
<RESET> interrupts an active part program.
2. After a power supply interruption, switch on the controller.
3. Select the JOG mode.
4. Press the Menu forward key.
5. Press the "Retract" HSK. The "Retract Tool" window opens.
6. The softkey is available only when an active tool and retraction data are present.
7. Select the "WCS" coordinate system on the machine control panel.

5.3.8 Settings

Please see section [5.1.6](#) for information on inch to metric / metric to inch conversion procedures.

5.3.8.1 Measurement report settings

The settings for the measuring report can be configured.

1. Press the ">>" and "Settings" HSKs.
2. Press "Measure. Report" VSK. The "Settings for measurement screen" window opens.
3. Select/Enter a value for the following fields:
 - o Report Format
 - o Log Data
 - o Report directory
 - o Name of report file

5.4 Reposition (REPOS) sub-mode

The "REPOS" function is used for repositioning to a defined position. After a program interrupt (e.g., to correct tool wear values), move the tool away from the contour in "JOG" mode. The path differences traversed in "JOG" mode are displayed in the actual value window as the "REPOS" offset. "REPOS" offsets can be displayed in the machine coordinate system (MCS) or workpiece coordinate system (WCS).

Repositioning axes

After a program interruption in automatic mode (e.g. after a tool breaks), you can move the tool away from the contour in manual mode. The coordinates of the interrupt position will be saved. The distances traversed in manual mode are displayed in the actual value window. This path difference is called "REPOS offset".

Resuming program execution

You use the "REPOS" function to return the tool to the contour of the workpiece to continue executing the program. The interrupt position is not passed as it is blocked by the control system. The feedrate/rapid traverse override is in effect.

CAUTION!

RISK OF COLLISION! When repositioning the axes, move with the programmed feedrate and linear interpolation, i.e. in a straight line from the current position to the interrupt point. Therefore, you must first move the axes to a safe position in order to avoid collisions.

If you do not use the "REPOS" function after a program interrupt, traverse the axes in manual mode, then on changing to automatic mode and starting the machining process, the control automatically traverses the axes in straight lines back to where they were at point of interruption.

The following prerequisites must be met when repositioning the axes:

- The program execution was interrupted using <CYCLE STOP>.
- The axes were moved from the interrupt point to another position in manual mode.

5.4.1 Repositioning the axes

1. Press the <REPOS> key.
2. Select the axes to be traversed one after the other.
3. Press the <+> or <-> key for the relevant direction.
4. The axes are moved to the interrupt position.

5.5 Reference Point (REF POINT) sub-mode

The "REF POINT" function is used to synchronize the control and the machine. For this purpose, you approach the reference point in "JOG" mode.

Referencing Axes

Your machine tool can be equipped with an absolute or incremental path measuring system. An axis with incremental path measuring system must be referenced after the controller has been switched on – however, an absolute path measuring system does not have to be referenced.

For the incremental path measuring system, all the machine axes must therefore first approach a reference point, the coordinates of which are known to be relative to the machine zero-point.

Prior to the approach, the axes must be in a position from where they can approach the reference point without a collision.

CAUTION!

Risk of collision: If the axes are not in a collision-free position, you must first traverse them to safe positions in "JOG" or "MDI" mode. You must follow the axis motions directly on the machine! Ignore the actual value display until the axes have been referenced! The software limit switches are not active!

Procedure

1. Press the <JOG> key.
 2. Press the <REF. POINT>. key
 3. Select the axis to be traversed.
 4. Press the <-> or <+> key. The selected axis moves to the reference point. If you have pressed the wrong direction key, the action is not accepted and the axes do not move.
- A symbol is shown next to the axis if it has been referenced.

The axis is referenced as soon as the reference point is reached. The actual value display is set to the reference point value. From now on, path limits, such as software limit switches, are active.

End the function by going to AUTO or JOG sub-mode.

5.6 TEACH-IN sub-mode

The "Teach in" function can be used to edit programs in the "AUTO" and "MDA/MDI" modes. You can create and modify simple traversing blocks. You traverse the axes manually to specific positions in order to implement simple machining sequences and make them reproducible. The positions you approach are applied.

- In the "AUTO" teach-in mode, the selected program is "taught".
- In the "MDA/MDI" teach-in mode, you teach to the MDA/MDI buffer.

External programs, which may have been generated offline, can therefore be adapted and modified according to your specific requirements.

NOTE - When an "execute from external memory" program has been selected, it is not possible to teach-in programs.

General sequence

1. Activate teach-in mode.
2. Insert a block. To do this, position the cursor at the desired point in the program and insert an empty line.
3. Press the relevant VSK "Teach position", "Rapid traverse G01", "Straight line G1", or "Circle interpolation position CIP" and "Circle end position CIP".
-OR-
Change an existing block. To do this, mark the desired program block, and press the corresponding VSK "Teach in position", "Rapid traverse G01", "Straight line G1", or "Circle interpolation point CIP" and "Circle end point CIP".
4. You can only overwrite a block with a block of the same type.
5. Traverse the axes.
6. Press the "Accept" VSK to teach-in the modified or newly created program block.

NOTE - All defined axes are "taught in" in the first teach-in block. In all additional teach-in blocks, only axes modified by axis traversing or manual input are "taught in". If you exit teach-in mode, this sequence begins again.

NOTE - You can select the axes to be included in the teach-in block in the "Settings" window. You also specify here whether motion and transition parameters are offered for teach-in.

Operating mode or operating area switchover

If you switch to another mode or sub-mode while in teach-in mode, the position changes will be canceled and teach-in mode will be cleared.

5.6.1 Overstore

Please see section [5.1.1](#) for information on this HSK.

5.6.2 Teach Prog.

Vertical Softkeys

- **Teach position** - You traverse the axes and write the current actual values directly into a positioning block.
- **rap. Tra. G0** - You traverse the axes and teach-in a rapid traverse block with the approached positions.

- **Straight G1** - You traverse the axes and teach-in a machining block (G1) with the approached positions.
- **Inter. Circl pt.CIP, Circ. End pos. CIP** - Enter the intermediate and end positions for the circle interpolation CIP. You teach-in each of these separately in a separate block. The order in which you program these two points is not specified.

NOTE - Make sure that the cursor position does not change during teach-in of the two positions.

- You teach-in the intermediate position in the "Circle intermediate position CIP" window.
 - You teach-in the end position in the "Circle end position CIP" window.
 - The intermediate or interpolation point is only taught-in with geometry axes. For this reason, at least 2 geometry axes must be set up for the transfer.
- **ASPLINE** – Command to activate A spline interpolation.

5.6.2.1 Inserting a block

Before proceeding, make sure an empty line is selected. The windows for pasting program blocks contain input and output fields for the actual values in the WCS. Depending on the default setting, selection fields with parameters for motion behavior and motion transition, are available.

When first selected, the input fields are empty, unless axes had already been traversed before the window was selected.

All data from the input/output fields are transferred to the program with the "Accept" VSK.

1. While in Teach-in mode, select the desired point in the program. If an empty row is not available, insert one.
 2. Press the softkeys "Rap. tra. G0", "Straight line G1", or "Circ. interm. pos. CIP" and "Circ. end pos. CIP". The relevant windows with the input fields are displayed.
 3. Traverse the axes to the relevant position.
 4. Press the "Accept" VSK. A new program block will be inserted at the cursor position.
- OR-
- Press the "Cancel" softkey to cancel your input.

5.6.2.2 Editing a block

You can only overwrite a program block with a teach-in block of the same type. The axis values displayed in the relevant window are actual values, not the values to be overwritten in the block.

NOTE - If you wish to change any variable in a block in the program block window other than the position and its parameters, then we recommend alphanumerical input.

1. While in Teach-In mode, select the program block to be edited.
 2. Press the relevant softkey "Teach position, "Rap. tra. G0", "Straight line G1", or "Circ. interm. pos." CIP", and "Circ. end pos. CIP". The relevant windows with the input fields are displayed.
 3. Traverse the axes to the desired position and press the "Accept" VSK.
 4. The program block is taught with the modified values.
- OR-
- Press the "Cancel" softkey to cancel the changes.

5.6.2.3 Selecting a block

You have the option of setting the interrupt pointer to the current cursor position. The next time the program is started, processing will resume from this point. With teach-in, you can also change program

areas that have already been executed. This automatically disables program processing. You must press reset or select a block to resume the program.

1. When in Teach-in mode, select the desired program block.
2. Press the "Block selection" VSK.

5.6.2.4 Deleting a block

In teach-in mode, you can delete both a teach-in block and a program block entirely.

1. When in Teach-in mode, select the program block to be deleted.
2. Press the ">>" and "Delete block" VSKs. The selected program block is deleted.

5.6.3 Prog. Cntrl

Please see section [5.1.2](#) for information on this HSK.

5.6.4 Simult. Record

Please see section [5.1.4](#) for information on this HSK.

5.6.5 Prog. Corr.

A program must be selected for execution in "AUTO" mode. As soon as a syntax error in the part program is detected by the controller, program execution is interrupted and the syntax error is displayed in the alarm line.

Depending on the state of the control system, you have various options of correcting the program.

- Stop state - Only change lines that have not been executed
- Reset status - Change all lines

NOTE - The "program correction" function is also available for execute from external; however, when making program changes, the NC channel must be brought into the reset state.

5.6.5.1 Correcting a program

1. The program to be corrected is in the Stop or Reset mode.
2. Press the "Prog. corr." HSK. The program is opened in the editor.
3. The program preprocessing, and the current block are displayed. The current block is also updated in the running program, but not the displayed program section, i.e. the current block moves out of the displayed program section.
4. If a subprogram is executed, it is not opened automatically. Make the necessary corrections.
5. Press the "NC Execute" VSK. The system switches back to machine mode and selects "AUTO" mode.
6. Press the "CYCLE START" on the MCP to resume program execution.

6.0 Parameter Mode

In Parameter Mode, you have a choice of selecting between various lists (e.g. Tool list, tool wear, magazine list, offsets, user variables and setting data).

The lists differ from each other by the displayed parameters and the VSK functions. The Parameter Mode menu displays information such as tool/magazine data, work offsets, user variables etc. It is possible to carry out functions such as creating tools, editing existing tool/magazine data, editing work offsets/variables/setting data.

- **Tool list:** All parameters and functions required to create and set up tools are displayed.
- **Tool wear:** All parameters and functions that are required during operation, e.g. wear and monitoring functions, are listed here.
- **Magazine:** Magazine and magazine location-related parameters and functions for the tools and magazine locations are listed here.

In this section, we will go over the HSKs, and describe some basic operation procedures. Below is a diagram of all the VSK and HSK options found within Parameter Mode for your reference.

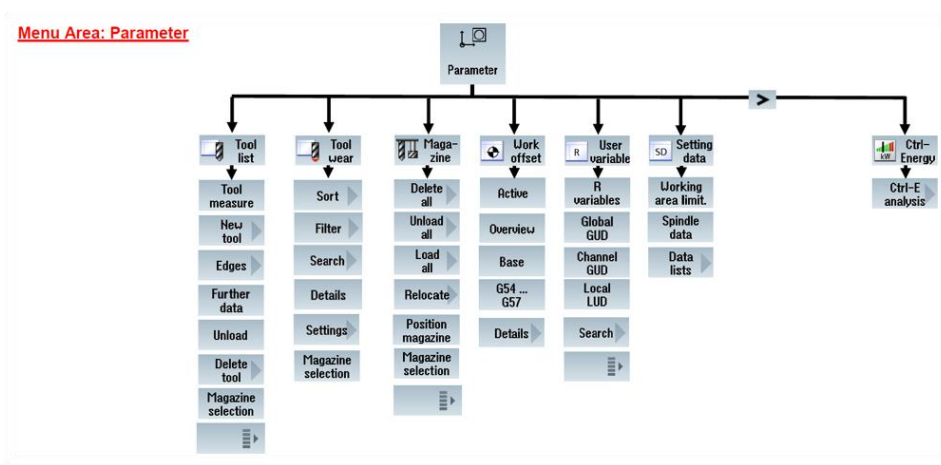


Figure 6.a - Parameter Mode

6.1 Tool List

In the tool list, all parameters and functions that are required to create and set up the tools are displayed, regardless whether the tools are assigned or not, to a magazine location.

Each tool is uniquely identified by the location number, the tool name and the replacement tool number. Geometrical and technological tool data can be assigned to each tool type. Depending on the tool type different correction data are necessary.

Loc.	Type	Tool name	ST	D	H	Length	ø	N	1	2	CTS
		.500 3FL CARBIDE EM	1	1	0	4.1365	0.5026		3	☑	☐
1		.50 3FL Carbide EM	2	1	0	5.5536	0.5000		5	☑	☐
2		FACING TOOL	1	1	0	3.2615	3.0000		8	☑	☐
3		.75 3FL Carbide EM	1	1	0	5.1026	0.7504		3	☑	☐
4		CHAMFER_MILL	1	1	0	6.1596	0.3750	90.0	☑	☑	☐
6		.5 EM	1	1	0	4.7480	0.4837		0	☑	☐
7		.257 DRILL	1	1	0	8.0281	0.2570	118.0	☑	☑	☑
8		.375 Corner Rounding EM	1	1	0	4.2155	0.3650		4	☑	☐
9		.375 Spot/Chamfer 90 Deg	1	1	0	3.3328	0.3750	90.0	☑	☑	☐
10		3/8 BALL ENDMILL	1	1	0	3.8140	0.3750		2	☑	☐
11		1/4 BALL ENDMILL	1	1	0	4.3999	0.2359		2	☑	☐
12		375_ENDMILL	1	1	0	3.9157	0.3750		3	☑	☐
13		.50 3FL Course Rougher	1	1	0	4.4269	0.5000		0	☑	☐
14		4.0 Staggered Tooth Side M	1	1	0	5.8220	4.0000	0.5000	8	☑	☑
15		KM_KSSM_5004	1	1	0	3.8150	3.0000		8	☑	☐

Figure 6.b - Parameter Mode > Tool List HSK > Selecting an empty row

Table Column Headers

- **Loc.** – Magazine / location number
 - Spindle Location
 - Gripper 1
 - Gripper 2
 - **1** Magazine Number
- **Type** – milling, cutting, drill, special tools, etc.
- **Tool Name** – to identify a tool you can enter a tool name as text or a T-number. If a new tool is created, tool names are pre-assigned as default.
- **ST** – (Sister Tool) If a new tool with the same name as an already existing tool is being created, then the new tool gets the ST "2". This way it is possible to define a replacement tool.
- **D** – (Cutting edge number) For tools with multiple cutting edges, each tool receives its own correction data field. Up to 9 edges per tool can be managed.
- **H** –When running an ISO G-code program: The H code tells the control which length offset value to use, when length compensation is active (as selected by G43 or G44). Generally, the H code is the same as the tool number.
- **Length** - (Tool length) Geometry length of the tool.

- **Ø** – (Tool radius/diameter) For every tool, information about the tool radius or diameter can be entered here. The changeover from diameter to radius or vice versa can be set via a machine datum.

Vertical Softkeys

- **Tool measure** – the “measuring tool” window opens.
- **New Tool** – a new tool can be created. The VSK only appears when an empty row has been selected.
- **Edges** – assign new cutting edges, and delete existing cutting edges. If a tool has several cutting edges, each edge has its own set of correction data.
- **Further Data** – more info about the tool is displayed. Only available for tools with more additional information.
- **Load / Unload** – the selected tool will be loaded to the magazine, or unloaded to the bottom of the magazine list.
- **Delete tool** – the selected tool will be deleted from the tool list.
- **Magazine Selection** – by pressing the key multiple times, this allows you to jump between buffer location (spindle and gripper), magazine and nc-memory (unloaded tools) and back to the buffer location.
- **Sort** – you can sort the tools list by magazine, name, type or T-Number.
- **Filter** – you can filter the tool list.
- **Search** – you can search by tool, magazine location, or empty location.
- **Details** - you can see tool data, cutting edge data or monitoring data.

6.1.1 Create a new tool

1. Highlight the location in which the new tool is to be created.
2. Press the “New tool” VSK.
3. From the menu, select the Favorites List, or a tool type (Cutters, Drill, Turn tools, Spec tool).
4. Press the “OK” VSK, to accept the selected tool/type and to jump back to the “Tool list” window.
5. From here you can edit the tool fields such as the tool name, sister tool, cutting edge number, tool length, tool radius/diameter, tip angle. *See 5.3.4 to set tool length offsets.* .

6.1.2 Load tool to spindle / via spindle

This allows you to load a tool via the spindle, or to the spindle.

1. Create a new tool, or select a tool from the library/tool list.
2. Press the “LOAD” VSK, and the control will suggest an empty station, but instead, choose to put that tool into the spindle.
3. Go to Machine Mode > Jog sub-mode > T,S,M HSK.
4. Press the “T” field on the screen. Select the tool in the spindle.
5. Press “Cycle Start” on the MCP. The ATC arm will load the tool in the spindle.
 - a. If you have not loaded the tool yet, load the tool into the spindle now.
6. Go back to the T,S,M screen.
7. In the “T” field, enter “0”, and then “CYCLE START”. This will place the tool back into the ATC in any available empty pot.
8. Go back to Parameter > Tool list, or Magazine, and you will see the tool loaded, in the appropriate pot.

6.1.3 Unloading tool from spindle / via spindle

This allows you to unload a tool via the spindle, or from the spindle. When it is unloaded, it is removed from the magazine and stored in the NC memory.

1. From the Machine Mode > Jog sub-mode > T,S,M HSK, press the "T" field.
2. Select the tool that you want to unload from the list. This will place it in the row for the spindle.
3. Press "CYCLE START" on the MCP. The machine will get the tool from the ATC, and put it in the spindle.
4. Go to Parameter mode > Tool list HSK. Select the tool in the spindle (should be the same tool you want to unload).
5. Manually remove the tool from the spindle.
6. Press "Unload" VSK. This will move the tool from the ATC to the library (bottom of the tool list).
7. Press "Reset" hard key.

6.1.4 Delete tool

This allows you to permanently remove a tool from the tool list.

1. Always unload the tool you want to delete, first.
2. Select the tool.
3. Press "Delete tool". A window will pop up, asking you to confirm you want to delete this tool.
4. Press "OK" VSK. Tool will be deleted along with all its data.

6.1.5 Selecting Multiple Tools

1. Highlight the first tool you want to select. The tool will be highlighted blue.
2. Press the "MARK" VSK. The tool highlight will change from blue to yellow.
3. Now, you may press on all the rows/tools you wish to select.
4. From here you can load, unload or delete multiple tools.

6.1.6 Managing several cutting edges

In the case of tools with more than one cutting edge, a separate set of offset data is assigned to each cutting edge. The number of possible cutting edges depends on the controller configuration. Tool cutting edges that are not required can be deleted.

6.1.6.1 Adding Cutting Edges

1. Select the tool for which you would like to store more cutting edges.
2. Press the "Edges" VSK in the "Tool list".
3. Press the "New cutting edge" VSK. A new data set is stored in the list.
The cutting-edge number is incremented by one and the offset data is assigned the values of the cutting edge that is selected.
4. Enter the offset data for the 2nd cutting edge.
5. Repeat this process if you wish to create more tool edge offset data.

6.1.6.2 Delete Cutting Edges

1. Select the cutting edge that you want to delete and press the "Delete cutting edge" VSK.
2. The data set is deleted from the list. The first tool cutting edge cannot be deleted.

6.1.7 Tools with additional geometry data

1. While the tool list is opened, select an appropriate tool, e.g., an angle head cutter.
2. Press the "Additional data" VSK. The "Additional Data - ..." window opens. *The "Additional data" VSK is only available when the tool type supports this function.*

6.1.8 Changing a tool type

1. The tool list, the wear list, or the magazine must be opened.

2. Select the column "Type" of the tool that you wish to change. The "Tool types - Favorites" window opens.
3. Select the desired tool type in the list of favorites or select using the VSKs "Cutters 100-199", "Drill 200-299", or "Spec. tool 700-900".
4. Press the "OK" VSK. The new tool type is accepted and the corresponding icon is displayed in the "Type" column.

6.1.9 Sorting tool list, tool wear and magazine

1. From the "Tool list", "Tool wear" or "Magazine" HSK, press the ">>" and "Sort" VSKs.
2. The lists are displayed sorted numerically according to magazine location. Tool types are used to sort tools with the same magazine location.
 - OR-
 - Press the "By name" VSK to display the tool names in alphabetical order. The replacement tool numbers are used to sort tools with the same names.
 - OR-
 - Press the "By type" VSK to display the tools arranged by tool type. Identical types (e.g., milling cutters) are sorted according to their radius value.
 - OR-
 - Press the "By T number" VSK to display the tool names sorted numerically.
3. The list is sorted according to the specified criteria.

6.1.10 Filtering tool list, tool wear and magazine

The filter function allows you to filter-out tools with specific properties in the tool management lists.

1. Press the "Tool list", "Tool wear" or "Magazine" HSK.
2. Press the ">>" and "Filter" VSKs. The "Filter" window opens.
3. Activate the required filter criterion and press the "OK" VSK. The tools that correspond to the selection criteria are displayed in the list. The active filter is displayed in the window header.

Filter criteria:

- Only display the first cutting edge
- Only tools that are ready to use
- Only tools with active code
- Only locked tools
- Only tools that have reached the prewarning limit
- Only tools with remaining quantity of _ to _
- Only tools with residual tool life of _ to _
- Only tools with unloading marking
- Only tools with loading marking

6.1.11 Specific search in tool list, tool wear and magazine

There is a search function in all tool management lists, where you can search for the following objects:

Tools

- You enter a tool name. You can narrow down your search by entering a replacement tool number.
- You have the option of only entering a part of the name as search term.
- Enter the D number and, if necessary, activate the "Active D number" checkbox.

Magazine locations or magazines

- The search is made for the magazine location.

Empty locations

- If the lists with the location type are used, then the empty location search is made using the location type and location size.

1. From the Parameter Mode, press the "Tool list", "Tool wear" or "Magazine" HSK.

2. Press the ">>" and "Search" VSKs.
3. Press the "Tool" VSK if you wish to search for a specific tool.
 - OR-
 - Press the "Magazine location" VSK if you wish to search for a specific magazine location.
 - OR-
 - Press the "Empty location" VSK if you wish to search for a specific empty location.

6.2 Tool Wear

The Tool Wear tab allows you to monitor tool life based on time in cut, diameter modifiers, length modifiers, or quantity of parts. The control will monitor these values, notify the operator when approaching tool end-of-life, automatically disable the tool when tool end-of-life is reached, and will also automatically switch to a sister tool if one has been defined.

Loc.	Type	Tool name	ST	D	ΔLength	Δ∅	T	C
		.500 3FL CARBIDE EM	1	1	0.0000	0.0000		
1		.50 3FL Carbide EM	2	1	0.0000	0.0000		
2		FACING TOOL	1	1	0.0000	0.0000		
3		.75 3FL Carbide EM	1	1	0.0000	0.0000		
4		CHAMFER_MILL	1	1	0.0000	0.0000		
6		.5 EM	1	1	0.0000	0.0000		
7		.257 DRILL	1	1	0.0000	0.0000		
8		.375 Corner Rounding EM	1	1	0.0000	0.0000		
9		.375 Spot/Chamfer 90 Deg	1	1	0.0000	0.0000		
10		3/8 BALL ENDMILL	1	1	0.0000	0.0000		
11		1/4 BALL ENDMILL	1	1	0.0000	0.0000		
12		375_ENDMILL	1	1	0.0000	0.0000		
13		.50 3FL Course Rougher	1	1	0.0000	0.0000		
14		4.0 Staggered Tooth Side N	1	1	0.0000	0.0000		
15		KM_KSSM_5004	1	1	0.0000	0.0000		

Figure 6.c - Parameter Mode > Tool wear HSK

Table Column Headers

See section 6.1.1 for descriptions on Loc., Type, and Tool name.

- **ΔLength** (Length wear) – In this field, changes for the tool length are entered.
- **Δ∅** (Radius wear) – The SINUMERIK ONE checks the entered values, and whether they exceed an absolute or incremental threshold, or not.
 - The **incremental threshold** is the maximum difference between present wear and new wear.
 - The **absolute threshold** is the maximum total wear value that can be entered.
- **T** (Tool monitoring by tool life) –With the tool life T (Time), the service life for tool with machining federate is monitored in minutes.
- **C** (Tool monitoring by count) – With the count C, the number of workpieces machined by the tool is counted.
- **W*** (Tool monitoring by wear) - With wear W, the greatest value in the wear parameters ΔLength X, ΔLength Z, ΔRadius or Δ∅ in the wear list is monitored.

*The wear monitoring is configured via a machine data item.

- **D** - Single tools can also be disabled by hand, if these tools are not in use anymore, or if the tools life ran off. (The tool is disabled if the checkbox is activated).

Vertical Softkeys

- **Sort** – you can sort the tools list by magazine, name, type or T-Number.
- **Filter** – you can filter the tool list.
- **Search** – you can search by tool, magazine location, or empty location.
- **Details** - you can see tool data, cutting edge data or monitoring data.
- **Reactivate** - locked tools, and tools that have reached their pre-warning limit can be made operational again.

6.2.1 Entering tool wear

1. Select the tool from the list, that you want to enter wear data for.
2. Enter values for length, radius, setpoint, prewarning and tool life.

6.2.2 Disabling tool

1. Select the tool from the list, that you want to disable.
2. Activate the "D"-parameter checkbox for disabling the tool manually.

6.2.3 Reactivating tool

1. Select the disabled tool from the list, that you want to reactivate.
2. Press the "Reactivate" VSK.
3. The value entered as the setpoint is entered as the new tool life workpiece count.
4. The disabled tool is active again.

Reactivating and positioning

When the "Reactivate with positioning" function is configured, the selected tool's magazine location will also be positioned at a loading point. You can exchange the tool.

Reactivation of all monitoring types

When the "Reactivation of all monitoring types" function is configured, all the monitoring types set in the NC for a tool are reset during reactivation.

6.3 Magazine

Tools are displayed with their magazine-related data in the magazine list. Here, you can take specific actions relating to the magazines and the magazine locations. Individual magazine locations can be location-coded or disabled for existing tools.

In the magazine list, all magazine locations are shown. It is indicated whether a magazine location is available, disabled or occupied by a tool.

- If a magazine location is defective, this location can be disabled.
- If an oversized tool is inserted, which uses more than a half of the neighboring magazine locations, then the neighboring magazine locations will be disabled.

Loc.	Type	Tool name	ST	D	D	Z	L
1	DRILL	DRILL_201 1/4 TAP DRILL	2	1			
2	1/4 BALL ENDMILL	1/4 BALL ENDMILL	1	1			
3	1/4 BALL ENDMILL	1/4 BALL ENDMILL	2	1			
4	1_2_IN_EM	1_2_IN_EM	1	1			
5							
6							
7							
8	CALABRATION TOOL	CALABRATION TOOL	1	1			
9							
10							
11							
12							
13	CUTTER	CUTTER	1	1			
14							
15							

Figure 6.d - Parameter Mode > Magazine HSK

Column Descriptions

See section 6.1.1 for descriptions on Loc., Type, Tool name, ST, and D (edge number).

- **D** – Magazine location disabled
- **Z** (Oversized tool) - Marking 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. Only tools that are not loaded yet can be marked as oversized.
- **L** – Fixed location coding. The tool is fixed to the magazine location.

Vertical Softkeys

- **Unload all** - all tools can be transferred from the magazine location to the tool library, "Unload all" must be confirmed with the "OK" VSK.
- **Relocate** - a tool can be transferred from one magazine location to another or, with pressing the "Spindle" VSK, it can be transferred to the spindle. The selected target location must be confirmed with the "OK" VSK or can be aborted with the "Cancel" VSK.
- **Position magazine** - you can position magazine locations directly on the loading point.

6.3.1 Relocate a tool

1. Select the tool you wish to relocate. Press the "Relocate" VSK.
2. Select the magazine location you wish to relocate it to, or press the "Spindle" VSK, to transfer it to the spindle.
3. Press the "OK" VSK to confirm the location. The tool will now be transferred to the designated location.

6.3.2 Magazine selection

You can directly select the buffer memory, the magazine, or the NC memory.

1. Press the "Magazine selection" HSK.
2. You will move from one area to the next (i.e., from the buffer memory to the magazine, from the magazine to the NC memory, and from the NC memory back to the buffer memory) each time you press the softkey.

6.3.3 Deleting / unloading / loading / relocating all tools

You have the option of deleting or unloading all tools in the magazine list, loading them into the magazine list or relocating them in the magazine list. With one task, the tools are deleted or unloaded from the list or loaded, relocated in the list. The magazine management must be set up, and there can't be a tool in the buffer/spindle, so that the "Delete all", "Unload all", "Load all" or "Relocate all" VSK is displayed and available.

Procedure

1. When the magazine list is open,
2. Press the "Delete all", "Unload all", "Load all", or "Relocate all" VSK.
3. A prompt is displayed as to whether you really want to delete, unload, load or relocate all tools.
4. Press the "OK" VS to continue with deleting, unloading, loading or relocation of the tools.
The tools are deleted, unloaded, loaded or relocated in the magazine in ascending magazine location number order.

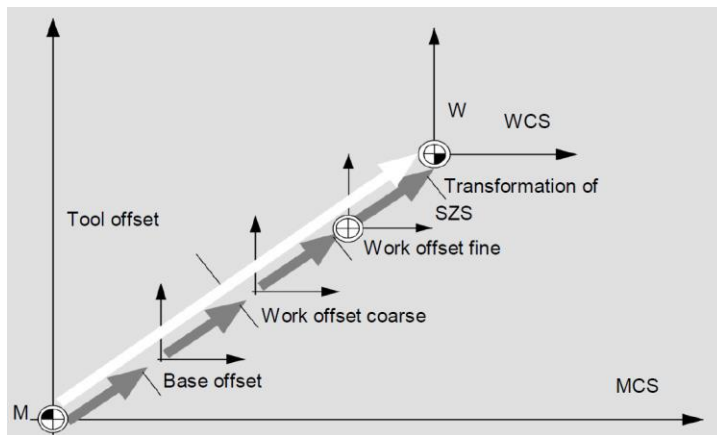
6.4 Work offset

6.4.1 Zero offset basics

Following reference point approach, the actual value display for the axis coordinates is based on the machine zero (M) of the machine coordinate system (MCS). The program for machining the workpiece, however, is based on the workpiece zero (W) of the workpiece coordinate system (WCS).

The machine zero and workpiece zero are not necessarily identical. The distance between the machine zero and workpiece vary in accordance with the type of tool and the way it is clamped. These zero offsets are taken into account during execution of the program and can be a combination of different offsets.

On the SINUMERIK ONE, the position actual value display refers to the SZS-coordinate system (settable zero system). The position of the active tool relative to the workpiece zero is displayed. The offsets are added as follows:



Base offset

The base offset is a zero offset that is always active. If you have not defined a base offset, its value will be zero. You determine the base offset via "Measure workpiece zero" in JOG sub-mode.

Zero offsets

Every zero offset (G54 to G57, G505 to G599) consists of a coarse offset and a fine offset. You can call the work offsets from any sequence program (coarse and fine offsets are added together). You can save the workpiece zero, for example, in the coarse offset, and then store the offset that occurs when a new workpiece is clamped between the old and the new workpiece zero in the fine offset.

6.4.2 Work offset - Active

The following work offsets are displayed in the "Work offset - active" window:

- Work offsets, for which offsets are included, or for which values are entered
- Adjustable work offsets
- Total work offset

	X	Y	Z	SP
Act.value Machine	-59.8100	-16.7270	-1.2008	0.000
G500	0.0000	0.0000	0.0000	0.000
Total WO	0.0000	0.0000	0.0000	0.000
Tool: CUTTER_3/8_ENDMILL	0.0000	0.0000	2.0504	0.000
Work actual value	-59.8100	-16.7270	-3.2512	0.000

Figure 6.e - Parameter Mode > Work offset HSK > Active VSK

This window is generally used only for monitoring. The availability of the offsets depends on the access level.

6.4.3 Work offset - Overview

In the "Work offset - Overview" window, all active offsets and system offsets are displayed for all set-up axes. In addition to the offset, the rotation, scaling and mirroring defined using this are also displayed.

	X	Y	Z	SP
Act.value Machine	-59.8100	-16.7270	-1.2008	0.000
DRF	0.0000	0.0000	0.0000	0.000
Rotary table ref.	0.0000	0.0000	0.0000	0.000
Basic reference	0.0000	0.0000	0.0000	0.000
Total basic WO	0.0000	0.0000	0.0000	0.000
G500	0.0000	0.0000	0.0000	0.000
Tool reference	0.0000	0.0000	0.0000	0.000
Workpiece ref.	0.0000	0.0000	0.0000	0.000
Transf. reference	0.0000	0.0000	0.0000	0.000
Programmed WO	0.0000	0.0000	0.0000	0.000
Cycle reference	0.0000	0.0000	0.0000	0.000
Total WO	0.0000	0.0000	0.0000	0.000
Tool: CUTTER_3/8_ENDMILL	0.0000	0.0000	2.0504	0.000
Work actual value	-59.8100	-16.7270	-3.2512	0.000

Figure 6.f - Parameter Mode > Work offset HSK > Overview VSK

This window is generally used for monitoring.

6.4.4 Work offset – Base

	X	Y	Z	SP
1. Channel basic WO	0.0000	0.0000	0.0000	0.0000
Fine	0.0000	0.0000	0.0000	0.0000

Figure 6.g - Parameter Mode > Work Offset HSK > Base VSK

The defined channel-specific and global base offsets, divided into coarse and fine offsets, are displayed for all set-up axes in the "Work offset - Base" window.

Procedure

1. In Parameter Mode, press the "Work offset" HSK.
2. Press the "Base" VSK. The "Work Offset - Base" window is opened. You can edit the values directly in the table.

NOTE - The offsets specified here are immediately active.

6.4.5 Work offset - G54...G599

All settable offsets, divided into coarse and fine offsets, are displayed in the "Work offset - G54...G599" window. Rotation, scaling and mirroring are displayed.

	X	Y	Z	SP
G54	0.0000	-19.1575	-22.5475	0.0000
Fine	0.0000	0.0000	0.0000	0.0000
G55	-29.0710	-19.1585	-23.7274	0.0000
Fine	0.0000	0.0000	0.0000	0.0000
G56	0.0000	0.0000	0.0000	0.0000
Fine	0.0000	0.0000	0.0000	0.0000
G57	0.0000	0.0000	0.0000	0.0000
Fine	0.0000	0.0000	0.0000	0.0000
G505	0.0000	0.0000	0.0000	0.0000
Fine	0.0000	0.0000	0.0000	0.0000
G506	0.0000	0.0000	0.0000	0.0000
Fine	0.0000	0.0000	0.0000	0.0000
G507	0.0000	0.0000	0.0000	0.0000
Fine	0.0000	0.0000	0.0000	0.0000
G508	0.0000	0.0000	0.0000	0.0000
Fine	0.0000	0.0000	0.0000	0.0000
G509	0.0000	0.0000	0.0000	0.0000
Fine	0.0000	0.0000	0.0000	0.0000
G510	0.0000	0.0000	0.0000	0.0000

Figure 6.h - Parameter Mode > Work offset HSK > Overview VSK

6.4.5.1 Displaying and editing settable zero offset:

1. From Parameter Mode, press the "Work offset" HSK.
2. Press the "G54...G599" VSK. The "Work offset - G54...G599" window opens.
3. Values can be edited directly in the table.

NOTE - The settable work offsets must first be selected in the program before they have an impact.

6.4.5.2 Deleting a zero offset

1. Press the "Overview", "Basis" or "G54...G599" VSK.
2. Press the "Details" VSK.
3. Select the work offset you would like to delete.
4. Press the "Clear offset" VSK. A confirmation prompt is displayed as to whether you really want to delete the work offset.
5. Press the "OK" VSK to confirm that you wish to delete the work offset.

6.5 Setting Data

6.5.1 Working area limit

Using the "Working area limitation" function you can limit the range within which a tool should traverse in all channel axes. This function allows you to set up protection zones in the working area that are inhibited for tool motion. In this way, you are able to restrict the traversing range of the axes in addition to the limit switches.

NOTE - You can only make changes in "AUTO" mode when in the RESET condition. These changes are then immediate.

NOTE - You can make changes in "JOG" mode at any time. These changes, however, only become active at the start of a new motion.

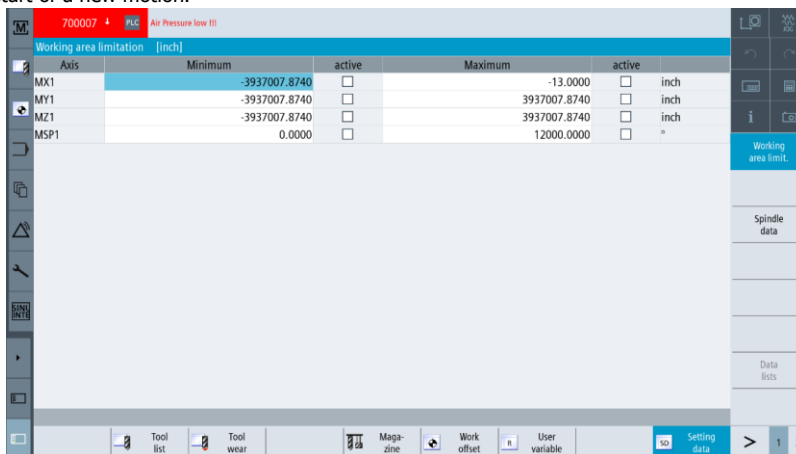


Figure 6.i - Parameter Mode > Setting data HSK > Working area limit VSK

Procedure

1. Press the "Setting data" HSK. The "Working Area Limitation" window appears.
2. Select the required field and enter the new values. The upper or lower limit of the protection zone changes according to your inputs.
3. Click the "active" checkbox to activate the protection zone.

6.5.2 Spindle data

The speed limits set for the spindles that must not be under- or overshoot are displayed in the "Spindles" window. You can limit the spindle speeds in fields "Minimum" and "Maximum" within the limit values defined in the relevant machine data.

Spindle speed limitation at constant cutting rate

In field "Spindle speed limitation at G96", the programmed spindle speed limitation at constant cutting speed is displayed together with the permanently active limitations.

This speed limitation, for example, prevents the spindle from accelerating to the max. spindle speed of the current gear stage (G96) when performing tapping operations or machining very small diameters.

NOTE - The "Spindle data" softkey only appears if a spindle is configured.

Procedure

1. Press the "Setting data" HSK, and "Spindle data" VSK. The "Spindles" window opens.
2. If you want to change the spindle speed, place the cursor on the "Maximum", "Minimum", or "Spindle speed limitation at G96" and enter a new value.

6.5.3 Data Lists

You can display lists with configured setting data.

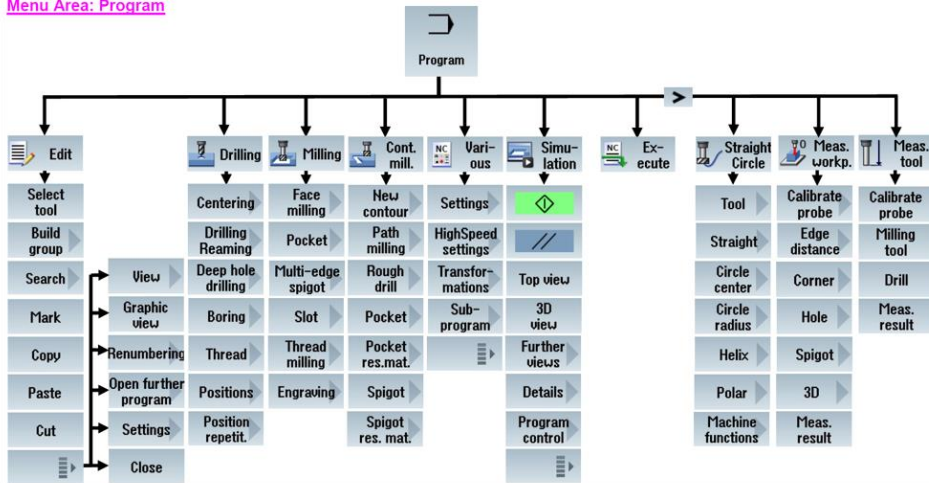
1. Press the "Setting data" HSK.
2. Press the "Data lists" VSK. The "Setting Data Lists" window opens.
3. Press the "Select data list" VSK and in the "View" list, select the required list with setting data.

7.0 Program Mode

The Program menu is used to display a part program, only if a program is already open will its contents be displayed. Editing of the part program is possible here. If a part program is not already open, the Program Manager will open.

In this section, we will go over the HSKs, and describe some basic operation procedures. Below is a diagram of all the VSK and HSK options found within Program Mode for your reference.

Menu Area: Program



7.1 Edit

Calling the editor

- The editor is started via the "Program correction" HSK in Machine Mode.
- The editor is called via the "Open" VSK in Program Manager Mode.
- The editor opens in Program Mode with the last executed part program, if this was not explicitly exited via the "Close" VSK.

NOTE - Please note that the changes to programs saved in the NC memory take immediate effect.

- If you are editing on a local drive or external drives, you can also exit the editor without saving, depending on the setting. Programs in the NC memory are always automatically saved.
- Exit the program correction mode using the "Close" VSK to return to Program Manager Mode.

7.1.1 Building a program group

In order to structure programs to achieve a higher degree of organization, you have the option of combining several G-code blocks to form program groups. Program groups can be created in two stages. This means that you can form additional groups within a group (nesting). You then have the option of opening and closing these groups depending on your requirement.

Procedure

1. Go to Program Manager Mode.
2. Select the storage location and create a program or open a program. The program editor opens.
3. Select the required program blocks that you wish to combine to form a group.
4. Press the "Build group" VSK. The "Build Group" window opens.
5. Enter a designation for the group.

Opening and closing groups

6. Press the ">>" and "View" VSKs.
7. Press the "Open groups" VSK if you wish to display the program with all groups.
8. Press the "Close groups" VSK if you wish to display the program again in a structured form.

Remove group

9. Open the group.
10. Select the end of the group.
11. Press the "Remove group" VSK.

NOTE - DEF statements in program groups or group generation in the DEF part of a part program / cycle are not permitted.

7.1.2 Searching in programs

1. Press the "Search" VSK. A new vertical softkey bar appears. The "Search" window opens at the same time.
2. Enter the desired search term in the "Text" field.
3. Select "Whole words" if you want to search for whole words only.
-OR-
Activate the "Exact expression" checkbox if, for example, you want to search for place holders ("*", "?") in program lines.
4. Select the "Direction" field and choose the search direction (forward, backward)
5. Press the "OK" VSK to start the search.
6. If the text you are searching for is found, the corresponding line is highlighted.
7. Press the "Continue search" VSK if the text located during the search does not correspond to the point you are looking for.
-OR-
Press the "Cancel" VSK when you want to cancel the search.

7.1.3 Replacing program text

You can find and replace text in one step. The desired program must be opened in the editor.

1. Press the "Search" VSK. A new vertical softkey bar appears.
2. Press the "Find and replace" VSK. The "Find and Replace" window appears.
3. In the "Text" field, enter the term you are looking for and in the "Replace with" field, enter the text you would like to insert automatically during the search.
4. Select the "Direction" field. Select the search direction (forward, backward).
5. Press the "OK" VSK to start the search. If the text you are searching for is found, the corresponding line is highlighted.
6. Press the "Replace" VSK to replace the text.
-OR-
Press the "Replace all" VSK to replace all text in the file that corresponds to the search term.
-OR-

Press the "Continue search" VSK if the text located during the search should not be replaced.

-OR-

Press the "Cancel" VSK when you want to cancel the search.

7.1.4 Copying/pasting/deleting program blocks

In the editor, you can edit both basic G-Code, as well as program steps such as cycles, blocks, and subprogram calls.

Inserting program blocks

The editor responds depending on what type of program block you insert.

- If you insert a G-Code, then the program block is directly inserted where the write mark is located.
- If you insert a program step, then the program block is always inserted at the next block, independent of the position of the write mark within the actual line. This is necessary as a cycle call always requires its own line.
- This behavior is in all applications, irrespective of whether the program step is inserted with a screen form using "Accept" or "Insert" is used as editor function.

NOTE - If you cut out a program step at a specific location and you then directly reinsert it again, the sequence changes. Press the shortcut (key combination) <CTRL> + <Z> to undo what you have cut out.

Procedure

1. Press the "Mark" VSK, and select the desired program blocks.
 2. Press the "Copy" VSK in order to copy the selection to the buffer memory.
 3. Select the desired insertion point in the program and press the "Paste" VSK. The content of the buffer memory is pasted.
- OR -
Press the "Cut" VSK to delete the selected program blocks and to copy them into the buffer memory.

NOTE - When editing a program, you cannot copy or cut more than 1024 lines. While a program that is not on the NC is opened (progress display less than 100%), you cannot copy or cut more than 10 lines or insert more than 1024 characters.

7.1.5 Renumbering a program

You can modify the block numbering of programs opened in the editor at a later point in time, but the program must be opened in the editor.

Procedure

1. Press the ">>" VSK. A new vertical softkey bar appears.
2. Press the "Renumber" VSK. The "Renumbering" window appears.
3. Enter the values for the first block number and the increment to be used for numbering.
4. Press the "OK" VSK. The program is renumbered.

NOTE

- If you only want to renumber a section, before the function call, select the program blocks whose block numbering you want to edit.
- When you enter a value of "0" for the increment size, then all of the existing block numbers are deleted from the program and/or from the selected range.

7.1.6 Editor Settings

Enter the default settings in the "Settings" window that are to take effect automatically, when the editor is opened.

NOTE - All entries that you make here are effective immediately.

Procedure

1. Press the ">>" and "Settings" VSK. The "Settings" window opens.
2. Make the required changes.
3. Press the "Del. machining times" VSK if you wish to delete the machining times.
4. The machining times that have been determined are deleted from the editor as well as from the actual block display. If the machining times are saved to an .ini file, then this file is also deleted.
5. Press the "OK" VSK to confirm the settings.

7.2 Simulation

7.2.1 Simulation prior to machining

Before machining the workpiece on the machine, you have the option of performing a quick run-through in order to graphically display how the program will be executed. This provides a simple way of checking the result of the programming.

Feedrate override

The rotary switch (override) on the control panel only influences the functions of Machine Mode. Press the "Prog. Cntrl" HSK to change the simulation feedrate. You can select the simulation feedrate in the range of 0 - 120%.

Procedure

1. Go to Program Manager mode.
2. Select the storage location of the program to be simulated.
3. Select desired program and press "Open" VSK. The selected program is opened in Program Mode.
4. Press the "Simulation" HSK. The program execution is displayed graphically on the screen. The machine axes do not move.
5. Press the "Stop" VSK if you wish to stop the simulation.
-OR-
Press the "Reset" VSK to cancel the simulation.
6. Press the "Start" VSK to restart or continue the simulation.

NOTE - The simulation is exited if you switch into another mode. If you restart the simulation, then this starts again at the beginning of the program.

7.2.2 Setting the model quality

In order for the model quality to be set, the simulation needs to be in progress.

Procedure

1. Press the ">>" and "Model quality" VSKs. The "Model quality" windows open and displays the available values.
2. Select the desired model quality.
3. Confirm your selection with the "OK" VSK.

NOTE - For complex workpieces, switching the model quality can take some time. In this case, a progress bar will be displayed.

7.2.3 Different views of the workpiece

In the graphical display, you can choose between different views so that you constantly have the best view of the current workpiece machining, or in order to display details or the overall view of the finished workpiece.

7.2.3.1 Top view

The workpiece is shown from above in the top view. You can increase or decrease the size of the simulation graphic and move it, as well as change the segment.

Procedure

1. With the simultaneous recording or the simulation is started, press the "Top view" softkey.

7.2.3.2 3D view

The workpiece is shown in 3D view. You can increase or decrease the size of the graphic, move it, turn it, or change the segment. You can display/move cutting planes X, Y, and Z.

NOTE - You require the "3D simulation" software option for the simulation.

Procedure

1. With the simultaneous recording or the simulation is started, press the "Other views" and "3D view" VSK.

7.2.3.3 Side view

The workpiece is shown displaying further side views.

Procedure

1. With the simultaneous recording or the simulation started, press the "Other views" VSK.
2. Press the "From front" VSK if you wish to view the workpiece from the front.
- OR -
Press the "From rear" VSK if you wish to view the workpiece from the rear.

7.2.3.4 Cut View

Procedure

1. With the simultaneous recording or the simulation is started, press the "Details" and "Cut active" VSKs.

7.2.3.5 Mold making view

For large mold making programs such as those provided by CAD/CAM systems, you have the option to display the machining paths by using a fast view. This provides you with a fast overview of the program, and you have the possibility of correcting it.

You can check the following with Mold making view:

- Does the programmed workpiece have the correct shape?
- Are there large traversing errors?
- Which program block hasn't been correctly programmed?
- How is the approach and retraction realized?

NOTE - The following NC blocks are not supported for the mold making view:

- Helix programming
- Rational polynomials
- Other G codes or language commands

All NC blocks that cannot be interpreted are simply overread.

Starting the mold making view

Procedure

1. From Program manager mode, select the program that you would like to display in the mold making view.
2. Press the "Open" VSK. The program is opened in the editor.
3. Press the ">>" and "Mold making view" VSKs. The editor splits up into two areas. The G code blocks are displayed in the left half of the editor. The workpiece is displayed in the mold making view on the right-hand side of the editor. All of the points and paths programmed in the part program are represented.

Adapting the mold making view

You can adapt the graphic in various ways to better assess the workpiece in the mold making view. Make sure the program is opened in mold making view already, and the "Graphic" VSK is active.

1. Press the softkey "Hide G1/G2/G3" if you want to conceal the machining paths.
-OR-
2. Press the softkey "Hide G0" if you want to deactivate the approach and retraction paths.
-OR-
Press softkey "Hide points" to conceal all the points in the graphic.
NOTE - You have the option of simultaneously hiding G1/G2/G3 and G0 lines. In this case softkey "Hide points" is deactivated.
-OR-
Press the softkeys ">>" and "Vectors" to display all orientation vectors.
NOTE - This softkey can only be operated if vectors are programmed.
-OR-
Press the softkeys ">>" and "Surface" to calculate the surface area of the workpiece.
-OR-
Press the softkeys ">>" and "Curvature". The "Curvature" input window opens. Enter the desired minimum and maximum value and press "OK" to confirm the entry and to highlight the curvature changes in color.

7.2.4 Program control during simulation

7.2.4.1 Changing the feedrate

You have the capability of changing the feedrate at any time during the simulation. You can track the changes in the status bar.

NOTE - If you are working with the "Simultaneous recording" function, use the rotary switch (override) on the control panel.

Procedure

1. With the simulation started, press the "Program control" softkey.

2. Press the "Override +" or "Override -" VSK to increase/decrease the feedrate by 5%, respectively.
-OR-
Press the "100% override" VSK to set the feedrate to 100%.
-OR-
Press the "<<" VSK to return to the main screen and perform the simulation with changed feedrate.

7.2.4.2 Simulating the program block by block

You have the capability of controlling the program execution during the simulation, i.e., to execute a program, e.g., program block by program block.

Procedure

1. With the simulation started, press the "Program control" and "Single block" softkeys.
2. Press the "Back" and "Start SBL" VSK. The pending program block is simulated and then stops.
3. Press "Start SBL" as many times as you want to simulate a single program block.
4. Press the "Program control" and the "Single block" softkeys to exit the single block mode.

7.3 Execute

When you select a program for execution, the control switches automatically to the "Machine" operating area.

Procedure

1. Select the "Program manager" operating area.
2. Select the desired storage location, and position the cursor on the workpiece/program that you would like to execute.
3. Press the "Select" softkey. The control switches automatically into the "Machine" operating area.
-OR-
If the selected program is already opened in the "Program" operating area, press the "Execute NC" softkey.
4. Press the <CYCLE START> key.
5. Machining of the workpiece is started.

8.0 Program Manager Mode

All NC-Programs, which are created with the SINUMERIK ONE, are stored in the NC-work memory. These programs can be accessed via the program manager for execution, alteration, copying, renaming, and/or deletion.

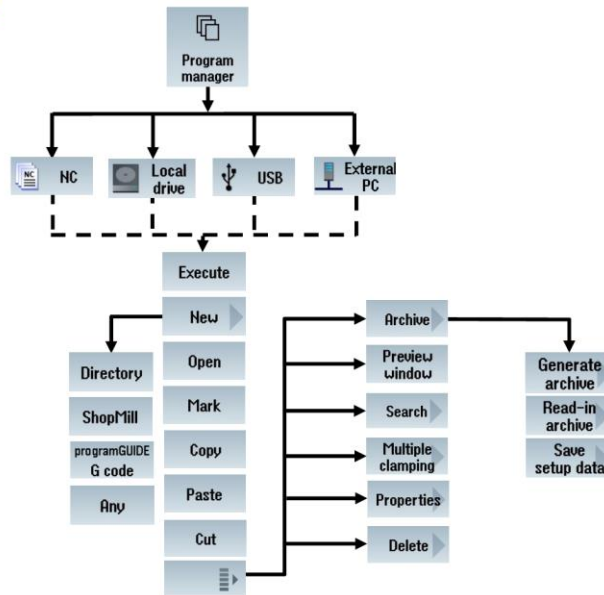
The SINUMERIK ONE provides the following means of data transmission of NC-programs to other storage media:

- NC memory
- Local drive memory
- Its own hard disk (PCU 50.x)
- Network connection
- USB-storage (stick or drive)

The Program Manager menu displays the “Part program/subprogram/workpiece” directories. Programs/subprograms/workpieces can be created or opened for editing. It is also possible to save to/from controller using memory devices (CF/USB).

In this section, we will go over the HSKs, and describe some basic operation procedures. Below is a diagram of all the VSK and HSK options found within Program Manager Mode for your reference.

Menu Area: Program Manager



Horizontal softkey bar

- **NC** - By pressing the HSK 1 "NC" all directories, folders and files of the NC/Hard disc will be displayed in a directory tree view in the program manager window.
- **Local Drive** - By pressing the HSK 2 "Local drive" all programs and directories on an allocated network drive or on an allocated user memory on a CF-Card at the NCU will be displayed. Prerequisite for this function is that the "Additional 256 MB HMI user memory on CF card of NCU" option is activated. For larger CF-cards also more than 256 MB memory can be enabled.
- **USB** - By pressing the HSK 3 "USB" all programs and directories on an USB drive will be displayed in a directory tree view in the program manager window. Programs created on an external PC can be copied to an USB drive and transferred to the NC via the USB interface where they can be processed further. Direct processing from the USB flash drive is not recommended. The text on the Softkey e.g "USB" can also be replaced by a drive letter e.g. "G".

Vertical Softkeys

Below are some vertical softkeys that will be repeated while in Program Manager Mode. Within the individual sub sections, we will list any VSKs that are unique to that specific sub-mode.

- **Execute** – you can select a program and change over to Machine Mode in order to start machining the selected program.
- **New** – you can create a new directory. In the selected directory you can create a new ShopMill or a ProgramGUIDE program.
- **Open** – the selected program (marked with an orange cursor) will be opened.
- **Mark** – several programs or directories can be marked for copying or cutting.
- **Copy** – one or several programs or directories can be copied
- **Paste** – the copied program(s) or directorie(s) are inserted into the selected place in the directory tree of the NC, of a local drive, or a USB drive.
- **Cut** – one to several programs or directories can be cut out and inserted somewhere else on a location in the directory tree of the NC, a local drive or on an external USB drive.
- **Archive** – a new vertical softkey bar is opened.
- **Preview Window** – a sub window opens below the file browser window, with a preview of the program code of the selected program.
- **Search** – you can search for "File name", using the Vertical softkeys.
- **Multiple clamping** - can optimize tool changes over several workpiece clamping.
- **Properties** – the "Properties of ..." input mask opens where you can:
 - View program path and modify the program name.
 - View the time and date of creation.
 - View the time and date of last changing of the program or folder.
 - User rights for execution, writing, listing and reading of files and folders.
- **Delete** - the program or folder marked with the cursor will be deleted.

Types of Programs

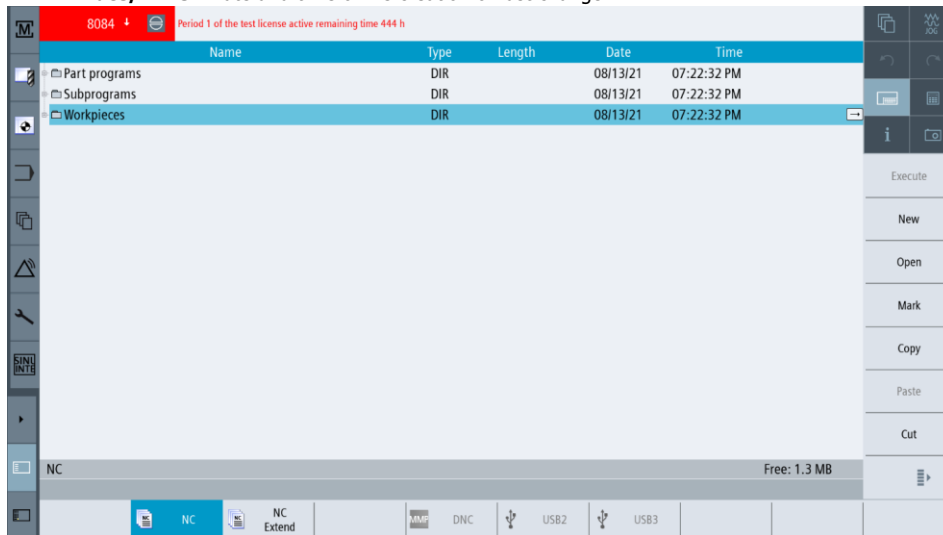
- **Part programs** - Programs written in g-code (DIN or ISO) format or programmed conversationally with ShopMill to execute machining functions.
- **Subprograms** - Generally a sub-program is identical to a part program but is only a part of the machining process and is usually called up during a part program. Sub-programs can be written in g-code (DIN or ISO) format or conversationally in ShopMill.
- **Workpieces** - This folder allows for non-program files to be stored on the hard drive of the control. eg: pdf, ini, jpg, etc.

8.1 Memory Locations

8.1.1 NC

The complete NC-memory is displayed along with all workpieces, the main programs and subroutines. The directories and programs are listed with the following information:

- **Name** - The name can contain up to 28 characters (24 characters for the name + dot + 3-character extension, e.g., MPF). Permissible characters include all upper-case letters (without accents), numbers, and underscores. Do not use spaces.
- **Type**
 - Directory/ Programs *.WPD Directory (Workpiece Directory)
 - *.MPF Program (Main program File)
 - *.SPF Subprogram (Subprogram File)
- **Size** - The size of files of the selected directory is displayed in bytes.
- **Date/Time** - Date and time of file creation or last change



8.1.2 NC Extend

Extends user memory on board the control open to all file types.

NOTE – This option requires the “CNC User Memory 4.6 GB” software option.

8.1.3 External Memory

8.1.3.1 USB Drives

USB drives enable you to exchange data. For example, you can copy to the NC and execute programs that were created externally.

NOTE - Direct execution and simulation from the USB flash drive is not recommended, because machining can be undesirably interrupted, therefore resulting in workpiece damage.

8.1.3.1 Network drives

Network drives enable you to exchange data. For example, you can copy to the NC and execute programs that were created externally.

NOTE – More detailed information can be found in section 13.

8.2 Execute

8.2.1 Executing a program

1. Select the desired storage location, and then select the workpiece/program that you would like to execute.
2. Press the "Select" VSK. The control switches automatically to Machine Mode.
-OR-
If the selected program is already opened in program Mode, press the "Execute NC" VSK.
3. Press the <CYCLE START> key on the MCP. Machining of the workpiece is started.

8.3 New

8.3.1 Starting a new NC-program

You can create programs directly at the control.

Procedure

1. Select the desired storage location and then select the folder, in which you would like to create the program.
2. Press the "New" VSK. The "New Sequential Program" window appears.
3. Select the type of program: "ShopMill" (option) or "G-Code".
4. If necessary, select a template if any are available. Select the file type (MPF or SPF).
5. If you are in the NC memory and have selected the file "Subprograms" or "Part programs", you can create only one subprogram (SPF) or main program (MPF) respectively.
6. Enter the desired program name and press the "OK" VSK.
7. The program type is appropriately specified. From here, you may begin to generate the new program.

8.3.2 Creating a new directory

Directory structures help you to manage your programs and data. At all storage locations, you can create subdirectories for this purpose.

Procedure

1. Select the desired storage medium, i.e. a local or USB drive.
2. If you want to create a new directory in the local network, select the topmost folder and press the "New" and "Directory" VSKs. The "New Directory" window opens.
3. Enter the desired directory name and press the "OK" VSK.
 - o Directory names must end in .DIR or .WPD.
 - o The maximum name length is 28 characters including the extension.
 - o The maximum path length for nested workpieces, including all supplementary characters, is 100 characters.
 - o These names are automatically converted to upper-case letters.
 - o This limitation does not apply for work on USB/network drives.

8.3.3 Creating a new workpiece

You can set up various types of files such as main programs, initialization files, tool offsets, etc., in a workpiece. You also have the option of nesting tool directories.

Workpiece directories

Note that the length of the call line is restricted. You will be informed if the maximum number of characters is reached when entering the workpiece name.

Procedure

1. Select the desired storage location and then select the folder in which you would like to create a workpiece.
2. Press the "New" VSK. The "New Workpiece" window appears.
3. If necessary, select a template if any are available.
4. Enter the desired workpiece name and press the "OK" VSK. The directory type (WPD) is set by default.
5. A new folder with the workpiece name will be created. The "New G-Code Program" window will open.
6. Press the "OK" VSK again if you want to create the program. The program will open in the editor.

8.3.4 Storing any new file

In each directory or subdirectory, you can create a file in any format that you specify.

NOTE - In the NC memory, the extension must have 3 characters, and DIR or WPD are not permitted. In the NC memory, you can create the following file types under a workpiece using the "Any" VSK.

Procedure

1. In Program Manager mode, select the desired storage location and select the folder in which you would like to create the file.
2. Press the "New" and "Any" VSKs. The "Any New Program" window opens.
3. Select a file type from the "Type" selection field (for example, "Definitions GUD") and enter the name of the file to be created when you have selected a workpiece directory in the NC memory.
4. The file automatically has the selected file format.
- OR -
Enter a name and file format for the file to be created (e.g., My_Text.txt).
5. Press the "OK" VSK.

8.4 Open

8.4.1 Opening a program

1. Select the desired storage location and select the program that you would like to edit.
2. Press the "Open" VSK. The selected program is opened in the "Editor" operating area.
3. Now make the necessary program changes.
4. Press the "NC Select" softkey to switch to Machine Mode and begin execution. When the program is running, the softkey is deactivated.

8.5 Mark

8.5.1 Selecting a Program

1. From the Program Manager Mode, the directory overview is opened.
2. Select the location where the program is archived (e.g., "NC")

3. Select the directory containing the program that you want to select. The directory contents are displayed.
4. Select the desired program.
5. When the program has been successfully selected, an automatic changeover to the "Machine" operating area occurs.

8.5.2 Selecting several directories/programs

You can select several files and directories for further processing. When you select a directory, all directories and files located beneath it are also selected.

NOTE - If you have selected individual files in a directory, then this selection is canceled when the directory is closed. If the complete directory with all of the files included in it are selected, then this selection is kept when closing the directory.

Procedure

1. Choose the desired storage location and select the file or directory from which you would like your selection to start.
2. Press the "Mark" VSK. The softkey is active.
3. Select the required directories/programs.

Canceling a selection

4. By reselecting an element, the existing selection is canceled.

8.5.3 Closing a program

1. Press the ">>" and "Exit" VSKs to close the program and editor again.
2. To reopen a program, you have exited with "Close", press the "Program" key.

8.6 Copy / Paste / Cut

8.6.1 Copying and pasting a directory/program

To create a new directory or program that is similar to an existing program, you can save time by copying the old directory or program and only changing selected programs or program blocks. The capability of copying and pasting directories and programs can also be used to exchange data with other systems via USB/network drives (e.g., USB Flash Drive). Copied files or directories can be pasted at a different location.

NOTE - You can only paste directories on local drives and on USB or network drives. If the current directory is write-protected for the user, then the function is not listed.

NOTE - When you copy directories, any missing endings are added automatically. All letters (except accented characters), numbers, and underscores are permitted when assigning names. The names are automatically converted to upper-case letters, and extra dots are converted to underscores.

Procedure

1. From Program Manager mode, choose the desired storage location, then select the file or directory which you would like to copy/cut.
2. Press the "Copy"/"Cut" VSK.
3. Select the directory in which you want to copy/cut your copied/cut directory/program.
4. Press the "Paste" VSK.

8.7 Archive

8.7.1 Generating an archive via system data

If you only want to backup specific data, then you can select the desired files directly from the data tree and generate an archive.

Archive formats

You can save your archive in the binary format.

You can display the content of the selected files (XML, ini, hsp, syf files, programs) using a preview.

You can display information about the file, such as path, name, date of creation and change, in a properties window.

Storage locations

- SD Card under
/archives/user
- All configured logical drives (USB, network drives)

NOTE - In order to save archives on the SD Card in the user area, you require the "CNC User Memory – 4.6 GB" software option is required.

NOTE - Possible data loss when using USB flash drives. USB-Flash Drives are not suitable as persistent memory media.

Procedure

1. Go to Setup Mode, press the "System data" HSK. The data tree opens.
2. In the data tree, select the required files from which you want to generate an archive.
-OR -
If you want to back up several files or directories, press the "Mark" VSK, and select the files.
3. If you press the ">>" VSK, further softkeys are displayed on the vertical bar.
4. Press the "Preview window" VSK.
The contents of the selected file are displayed in a small window.
Press the "Preview window" VSK again to close the window.
5. Press the "Properties" VSK.
Information about the selected file is displayed in a small window.
Press the "OK" VSK to close the window.
6. Press the "Search" VSK.
Enter the required search term in the search dialog and press the "OK" VSK if you wish to search for a specific directory or subdirectory.
NOTE - The place holders "*" (for any character string) and "?" (for any character) make it easier for you to perform a search.
7. Press the "Archive" and "Generate archive" VSK.
The "Generate Archive: Select Storage Location" window opens.
The "Archive" folder with the subfolders "User" and "Manufacturer" as well as the storage media (e.g., USB) are displayed.
8. Select the required location for archiving and press the "New directory" VSK to create a suitable subdirectory.
The "New Directory" window opens.
9. Enter the required name and press the "OK" softkey.
The directory is created below the selected folder.
10. Press the "OK" softkey.

The "Generate Archive: Name" window opens.

11. Select the format, enter the required name and press the "OK" softkey to archive the file/files. A message informs you if archiving was successful.
12. Press the "OK" softkey to confirm the message and end the archiving operation. An archive file is created in the selected directory.

8.7.2 Read in archive from system data

If you want to read in a specific archive, you can select this directly from the data tree.

Procedure

1. Go to Setup mode, and press the "System data" HSK.
2. In the data tree below the "Archive" directory, in the "User" folder, select the file that you wish to read in.
3. Press the "Read in" VSK.
4. Press the "OK" or "Overwrite all" VSK to overwrite existing files.
-OR-
Press the "Do not overwrite" softkey if you do not want to overwrite already existing files.
-OR-
Press the "Skip" softkey if the read-in operation is to be continued with the next file.
The "Read in Archive" window opens and a progress message box appears for the read-in process.
You will then obtain a "Read error log for archive" in which the skipped or overwritten files are listed.
Press the "Cancel" softkey to cancel the read-in process.

8.8 Preview window

8.8.1 Displaying the program in the Preview

You can show the content on a program in a preview before you start editing.

Procedure

1. From the Program Manager mode, select a storage location and select the relevant program.
2. Press the ">>" and "Preview window" VSKs. The "Preview: ..." window opens.
3. Press the "Preview window" VSK again to close the window.

8.9 Search

8.9.1 Searching directories and files

You have the possibility of searching in the Program Manager for certain directories and files.

NOTE - Search with place holders - The following place holders simplify the search:

- "*": Replaces any character string
- "?": Replaces any character

If you use place holders, only directories and files are found that correspond exactly to the search pattern. Without place holders, directories and files are found that contain the search pattern at an arbitrary position.

Search strategy

The search is made in all of the selected directories and their subdirectories. If a file is already selected, then a search is made from the higher-level directory.

Searching in opened directories

Open the closed directories for a successful search.

Procedure

1. From the Program Manager, select the storage location in which you wish to perform the search and then press the ">>" and "Search" VSKs. The "Find File" window opens.
2. Enter the desired search term in the "Text" field.
NOTE - When searching for a file with place holders, enter the complete name with extension (e.g., DRILLING.MPF).
3. When required, activate the "Observe upper and lower case" checkbox.
4. Press the "OK" VSK to start the search. If a corresponding directory or file is found, then it is marked.

8.10 Multiple Clamping

Allows you to run multiple ShopMill programs (same or different) across various WCS. The control will automatically run these programs tool-by-tool to optimize cycle time.

8.11 Properties

Information on directories and files can be displayed in the "Properties for ..." window. Information on the creation date is displayed near the file's path and name. You can change names.

Procedure

1. From the Program Manager, choose the desired storage location and select the file or directory whose properties you want to display or change.
2. Press the ">>" and "Properties" VSKs. The "Properties from ..." window appears.
3. Enter any necessary changes.
NOTE - You can save changes via the user interface in the NC memory.
4. Press the "OK" VSK to save the changes.

8.12 Delete

8.12.1 Deleting a directory/program

Delete programs or directories from time to time that you are no longer using to maintain a clearer overview of your data management. Back up the data beforehand, if necessary, on an external data medium (e.g., USB Flash Drive) or on a network drive.

Please note that when you delete a directory, all programs, tool data and zero-point data and subdirectories that this directory contains are deleted.

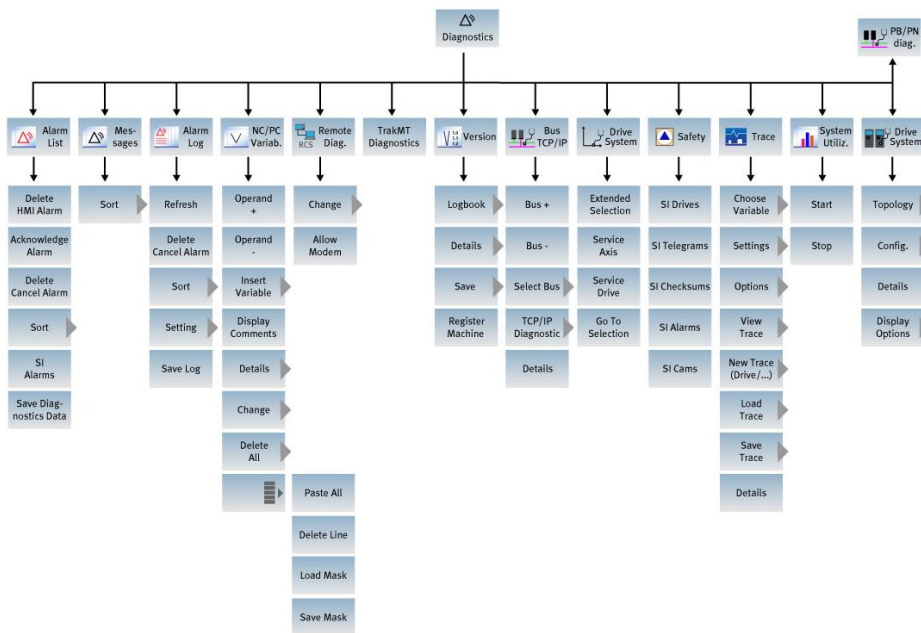
Procedure

1. From the Program Manager, choose the desired storage location and select the file or directory that you would like to delete.
2. Press the ">>" and "Delete" VSK. A prompt appears as to whether you really want to delete the file or directory.
3. Press the "OK" VSK to delete the program/directory.
-OR -
Press the "Cancel" VSK to cancel the process.

9.0 Diagnostics Mode

The Diagnostics menu is used for diagnosing machine problems, information about current/past alarms can be viewed. The status of drives, axes, networks, NC/PLC variables, utilization, etc., can be viewed. It is also possible to check the software/hardware versions of the system. Machine information and service reports are also entered via the diagnostics menu.

In this section, we will go over the HSKs, and describe some basic operation procedures. Below is a diagram of all the VSK and HSK options found within Diagnostics Mode for your reference.



9.1 Alarm List

If the machine develops a fault in operation, an alarm is generated and machining is possibly interrupted. The error text that is displayed together with the alarm number gives you more detailed information on the error cause.

CAUTION!

Carefully check the system, based on the description of the active alarm(s). Resolve the cause of the alarms. Then acknowledge the alarms in the specified way. Failure to observe this warning will place your machine, workpiece, saved settings and possibly even your own safety at risk.

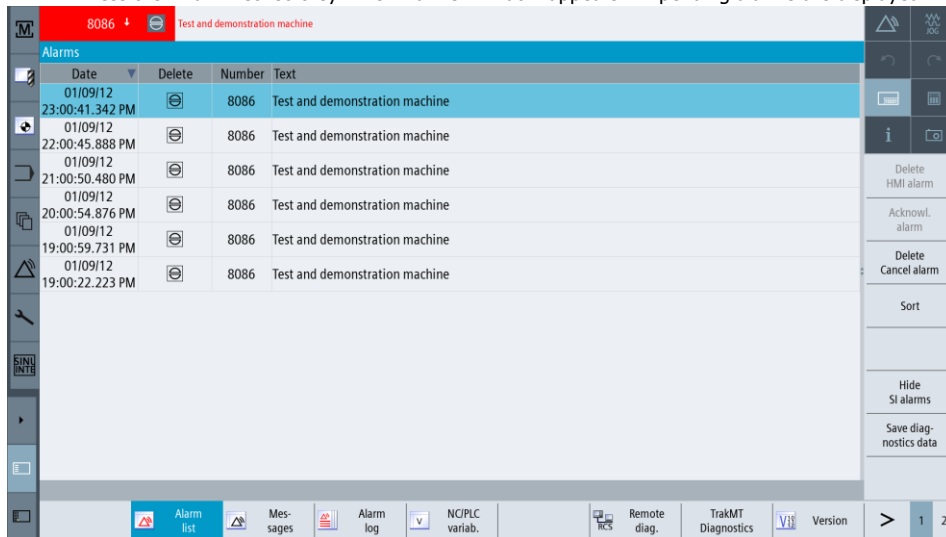
Alarm overview

You can display all upcoming alarms and acknowledge them. The alarm overview contains the following information:

- Date and time
 - Cancel criterion
 - Alarm number
 - Alarm text
- The delete criterion specifies the key or softkey that can be used to acknowledge the alarm.

Procedure

1. Select the "Diagnostics" operating area.
2. Press the "Alarm list" softkey. The "Alarms" window appears. All pending alarms are displayed.












3. The "Hide SI alarms" softkey is displayed if safety alarms are pending.
4. Press the "Hide SI alarms" softkey if you do not wish to display SI alarms.
5. Press the "Save diag. data" softkey if the cause of the alarm is unknown.
6. The function collects all available LOG files of the operating software and saves them to the following directory: \user\sinumerik\didac\out_<Date-Time>.7z

9.1.1 Cancel alarms, messages and logs

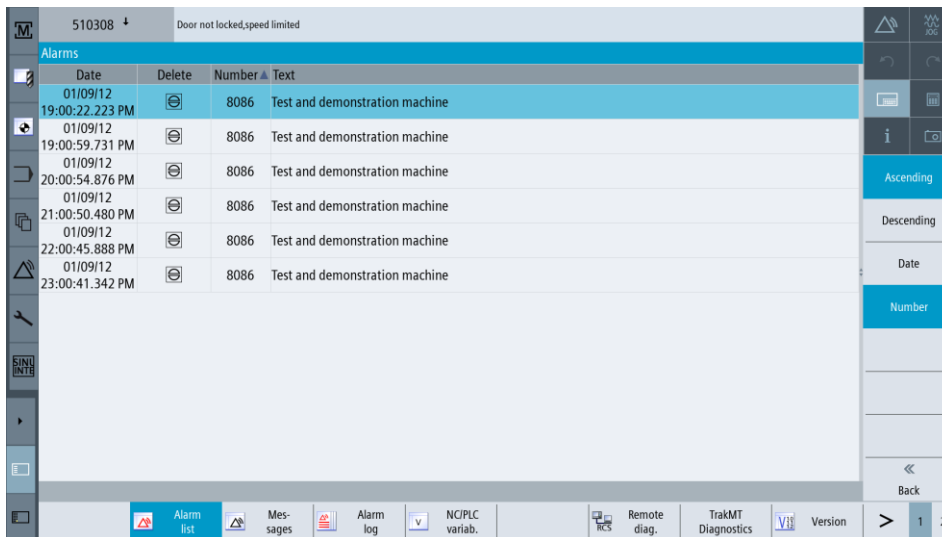
1. In the "Cancel" column it is symbolized how you delete the pending alarms from the alarm list.
2. Position the cursor on an alarm.
3. If an **NCK-POWER ON alarm** is displayed, turn the unit off and back on (main switch), or press NCK-POWER ON.
 - OR - If an **NC-Start alarm** is displayed, press the <NC-Start> key.
 - OR - If a **RESET alarm** is displayed, press the <RESET> key.
 - OR - If a **Cancel alarm** is displayed, press the <ALARM CANCEL> key or press the "Cancel Alarm delete" softkey.
 - OR - If an **HMI alarm** is displayed, press the "Delete HMI alarm" softkey.
 - OR - If a **dialog alarm** of the HMI is displayed, press the <RECALL> key.

- OR - If a **PLC alarm** is displayed, press the key provided by the machine manufacturer.
 - OR - If a **PLC alarm of the type SQ** is displayed, press the "Acknowl. alarm" softkey.
4. The softkeys are activated when the cursor is on the corresponding alarm.

Acknowledge symbols

Symbol	Meaning	Symbol	Meaning	Symbol	Meaning
	NCK POWER ON		Cancel alarm		PLC alarm
	NC start		HMI alarm		PLC alarm of the SQ type (alarm number from 800000)
	RESET alarm		Dialog alarms of the HMI		Safety alarms

9.1.2 Sorting alarms, messages and logs



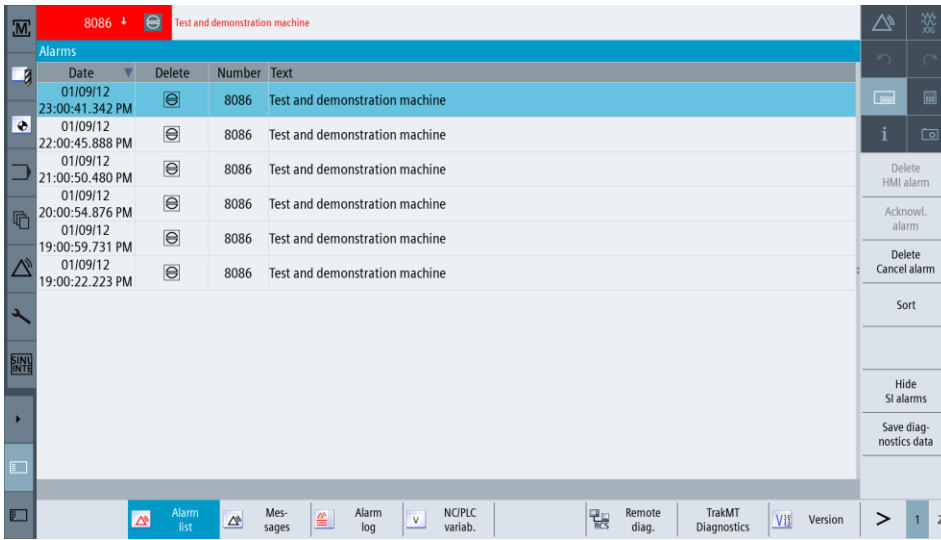
If a large number of alarms, messages or alarm logs are displayed, you have the option of sorting these in an ascending or descending order according to the following criteria:

- Date (alarm list, messages, alarm log)
- Number (alarm list, messages)

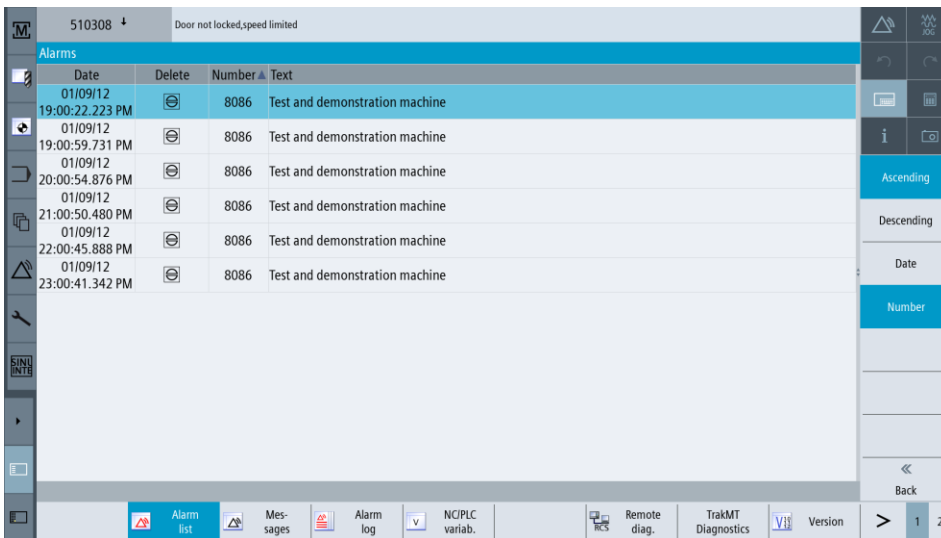
As a consequence, for every extensive list, you can obtain the required information faster.

Procedure

1. Press the "Alarm list", "Messages" or "Alarm log" softkey to display the requested messages and interrupts.



2. Press the "Sort" softkey. The list of entries is sorted in descending order according to date, i.e. the most recent information is at the beginning of the list.



3. Press the softkey "Ascending" to sort the list in the ascending order. The most recent event is shown at the end of the list.
4. Press the "Number" softkey if you wish to sort the alarm list or the list with messages according to numbers.

Date	Delete	Number	Text
01/09/12 23:00:41.342 PM	⊗	8086	Test and demonstration machine
01/09/12 22:00:45.888 PM	⊗	8086	Test and demonstration machine
01/09/12 21:00:50.480 PM	⊗	8086	Test and demonstration machine
01/09/12 20:00:54.876 PM	⊗	8086	Test and demonstration machine
01/09/12 19:00:59.731 PM	⊗	8086	Test and demonstration machine
01/09/12 19:00:22.223 PM	⊗	8086	Test and demonstration machine

5. Press the "Descending" softkey to display the list in decreasing/descending order again.

9.2 Messages

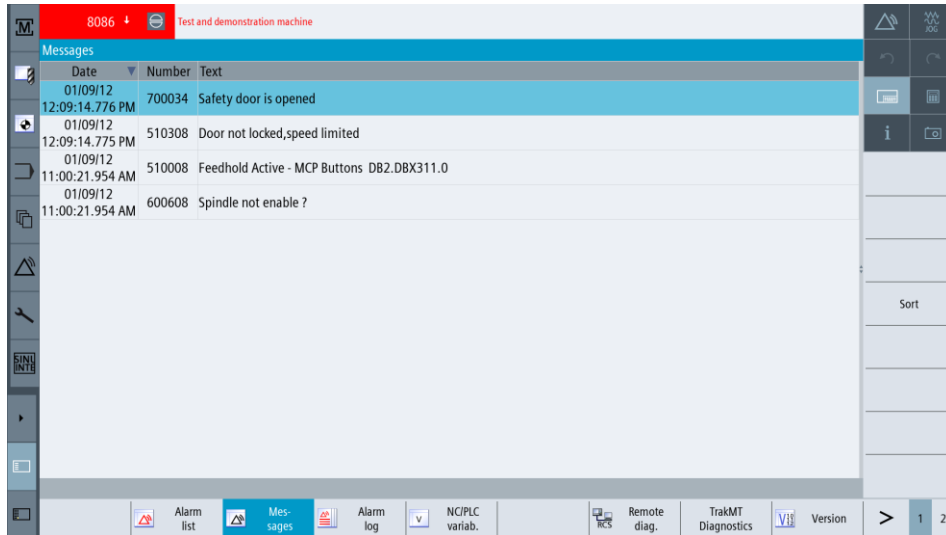
PLC and part program messages may be issued during machining. These messages will not interrupt the program execution. Messages provide information with regard to a certain behavior of the cycles and with regard to the progress of machining and are usually kept beyond a machining step or until the end of the cycle.

Overview of messages

You can display all issued messages. The message overview contains the following information:

- Date
- Message number
is only displayed for PLC messages
- Message text

Procedure



1. Select the "Diagnostics" operating area.
2. Press the "Messages" softkey. The "Messages" window appears.

9.3 Alarm log

A list of all the alarms and messages that have occurred so far are listed in the "Alarm Log" window. Up to 500 administered, incoming and outgoing events are displayed in chronological order.

9.3.1 Save Log

1. Select the "Diagnostics" operating area.
2. Press the "Alarm log" softkey. The "Alarm Log" window opens.
3. All of the coming and going events - that have occurred since the HMI was started - are listed.
4. Press the "Display new" softkey to update the list of displayed alarms/messages.
5. Press the "Save Log" softkey. The log that is currently displayed is stored as text file alarmlog.txt in the system data in directory /HMI data/Logs/Alarm logs.

9.3.2 Sort

See section 9.1.2

9.4 NC/PLC variab.

Changes can only be made to the NC/PLC variables with the appropriate password.

WARNING!

Changes in the states of NC/PLC variables have a considerable influence on the machine. Incorrect configuration of the parameters can endanger life and cause damage to the machine.

Variable	Comment	F	Value
DB95.DBD48		F	-0.318543314933777
db21.dbb7		B	00000001
\$A_IN[8]		D	0
db2.dbx563.4		B	0
db2.dbx493.0		B	0
db20.dbx115.2		B	0
DB2.DBX554.1		B	0
\$AA_ALARM_STAT[MSP1]	Display of active alarms	D	0
/Nck/AlarmEvent/alarmNo[1]	Alarm number	D	0
/Nck/AlarmEvent/textIndex[1]	Alarm number (actual alarm)	D	0
/Nck/MachineAxis/aaaAlarmStat[6]	Display of active alarms	D	0
/Channel/State/chanAlarm[u1, 1]	NCK alarm pending	D	0
DB21.DBB4		B	00001100
QD40		B	00000001000000000000000011100010000
QD44		B	000000000000000000000000000000000000
QD48		B	000000000000000000000000000000000000
QB40		B	00000001
QB41		B	00000000
QB42		B	00000111
QB43		B	00010000
QB44		B	00000000
QR45		R	00000000

In the "NC/PLC Variables" window, enter the NC system variables and PLC variables that you want to monitor or change in the list:

- Variable
Address for NC/PLC variable.
Incorrect variables have a red background and are displayed with a # character in the value column.
- Comment
Any comment on the variable.
The columns can be displayed and hidden.
- Format
Specify the format in which the variable is to be displayed.
The format can be specified (e.g., floating point)
- Value
Displays the actual value of the NC/PLC variables

Notation examples

Permissible notation for variables:

- PLC variables:
 - Addresses: EB2, A1.2, DB2.DBW2
 - Names: LBP_NC.E_NCKready, LBP_HMI.A_MCPChan
- NC variables:

- NC system variables: Notation \$AA_IM[1]
- User variables / GUD: Notation GUD/MyVariable[1,3]
- OPI notation: /CHANNEL/PARAMETER/R[u1,2]

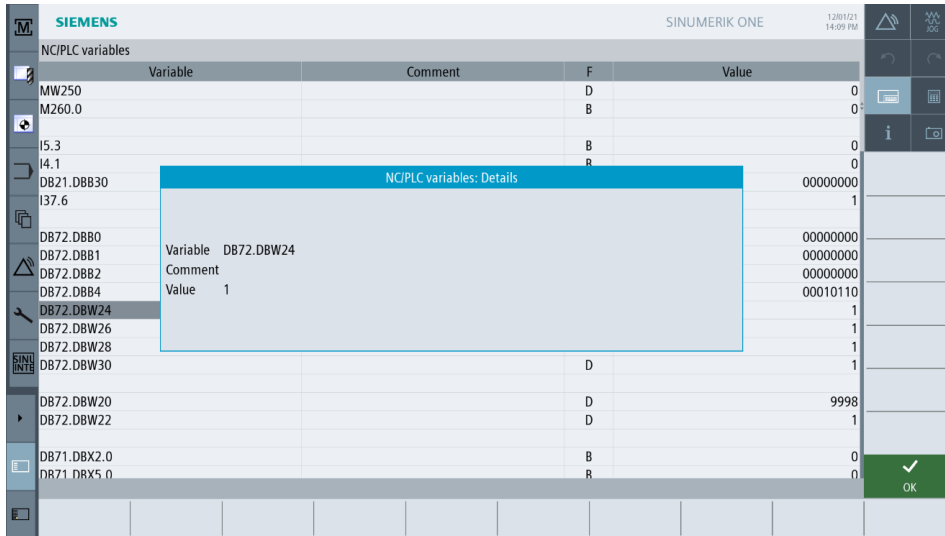
NOTE - If the PLC user program writes a string in an NC/PLC variable, the string will only be displayed correctly if the variable is parameterized as a field variable of the type "A" (ASCII) on the NC side.

9.4.1 Changing and deleting values

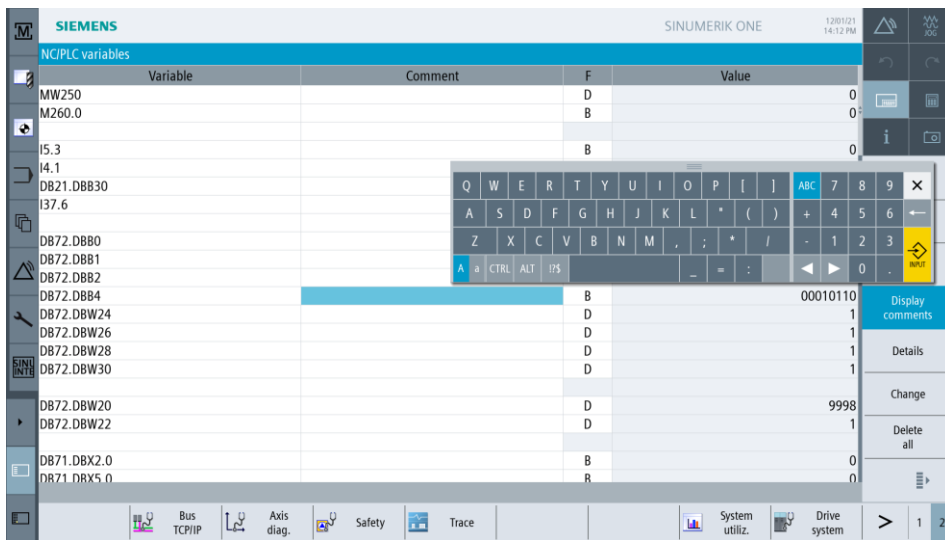
1. From Diagnostics Mode, press the "NC/PLC variables" HSK. The "NC/PLC Variables" window opens.

Variable	Comment	F	Value
DB95.DB48		F	-0.318543314933777
db21.dbb7		B	00000001
\$A_IN[8]		D	0
db2.dbx563.4		B	0
db2.dbx493.0		B	0
db20.dbx115.2		B	0
DB2.DBX554.1		B	0
\$AA_ALARM_STAT[MSP1]	Display of active alarms	D	0
/Nck/AlarmEvent/alarmNo[1]	Alarm number	D	0
/Nck/AlarmEvent/textIndex[1]	Alarm number (actual alarm)	D	0
/Nck/MachineAxis/aaAlarmStat[6]	Display of active alarms	D	0
/ChannelState/chanAlarm[u1, 1]	NCK alarm pending	D	0
DB21.DBB4		B	00001100
QD40		B	00000001000000000000011110010000
QD44		B	00000000000000000000000000000000
QD48		B	00000000000000000000000000000000
QB40		B	00000001
QB41		B	00000000
QB42		B	00000111
QB43		B	10010000
QB44		B	00000000
QB45		B	00000000

2. Select the "Variable" column and enter the required variable. The operand is displayed with the value.
3. Press the "Details" VSK.
The "NC/PLC Variables: Details" window opens. The information for "Variable", "Comment" and "Value" is displayed in full length.



4. Select the "Format" field and select the required format.
5. Press the "Display comments" VSK.
The "Comments" column is displayed. You have the option of creating comments or editing existing comments.
Press the "Display comments" VSK once again to hide the column.



6. Press the "Change" VSK if you would like to edit the value.

- The "Value" column can be edited.
7. Press the "Insert variable" VSK if you wish to select a variable from a list of all existing variables and insert this.
The "Select variable" window opens.
 8. Press the "Filter/search" VSK to restrict the display of variables (e.g. to mode group variables) using the "Filter" selection box and/or select the desired variable using the "Search" input box.
 9. Press the "Delete all" VSK if you would like to delete all the entries for the operands.
 10. Press the "OK" VSK to confirm the changes or the deletion.
-OR -
Press the "Cancel" VSK to cancel the changes.

Variable	Comment	F	Value
DB95.DBD48		F	-0.318543314933777
db21.dbb7		B	00000001
\$A_IN[8]		D	0
db2.dbx563.4		B	0
db2.dbx493.0		B	0
db20.dbx115.2		B	0
DB2.DBX554.1		B	0
\$AA_ALARM_STAT[MSP1]	Display of active alarms	D	0
/Nck/AlarmEvent/alarmNo[1]	Alarm number	D	0
/Nck/AlarmEvent/textIndex[1]	Alarm number (actual alarm)	D	0
/Nck/MachineAxis/aaAlarmStat[6]	Display of active alarms	D	0
/Channel/State/chanAlarm[u1, 1]	NCK alarm pending	D	0
DB21.DBB4		B	00001100
QD40		B	0000000100000000000000001100010000
QD44		B	00000000000000000000000000000000
QD48		B	00000000000000000000000000000000
QB40		B	00000001
QB41		B	00000000
QB42		B	00000111
QB43		B	00010000
QB44		B	00000000
QB45		B	00000000

9.4.2 Editing a variable list

You can edit the variable list using the "Insert line" and "Delete line" VSK.

- If you press the VSK, a new line inserted before the line marked by the cursor.
- You can only use the "Insert line" VSK if there is at least one empty line at the end of the variable list.
- The VSK is deactivated if there is no empty line.
- If you press the "Delete line" VSK, the line marked by the cursor is deleted.
- An empty line will be added at the bottom of the variable list.

9.4.3 Changing operands

Depending on the type of operand, you can increment or decrement the address by 1 place at a time using the "Operand +" and "Operand -" softkeys.

NOTE - For axis names, the "Operand +" and "Operand -" softkeys do not act as index, e.g., for \$AA_IM[X1].

Examples of Operand+

Starting: DB97.DBX2.5
Operand+
Result: DB97.DBX2.6

Starting: \$AA_IM[1]
Operand +
Result: \$AA_IM[2]

Examples of Operand-

Starting: MB201
Operand-
Result: MB200

Starting: /Channel/Parameter/R[u1,3]
Operand-
Result: /Channel/Parameter/R[u1,2]

9.4.4 Saving and loading screen forms

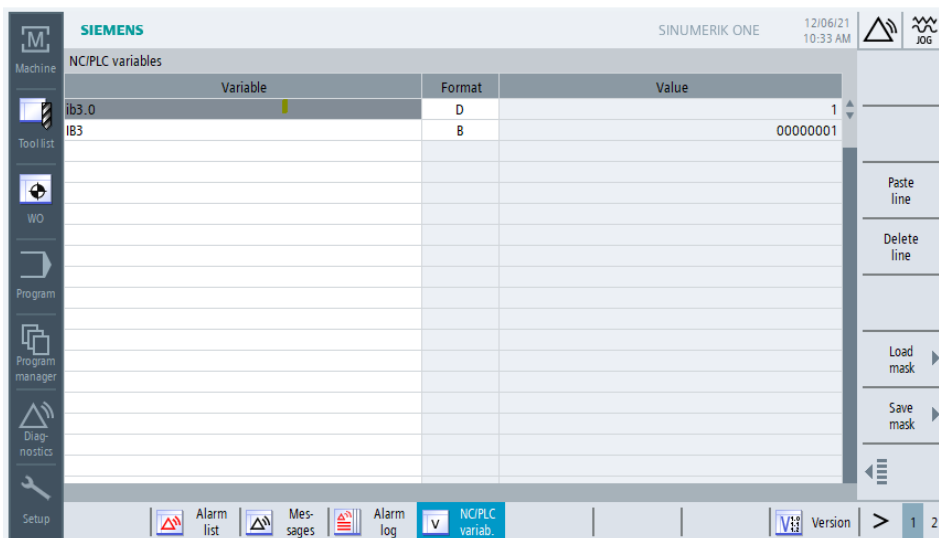
You have the option of saving the configurations of the variables made in the "NC/PLC variables" window in a screen form that you reload again when required.

Editing screen forms

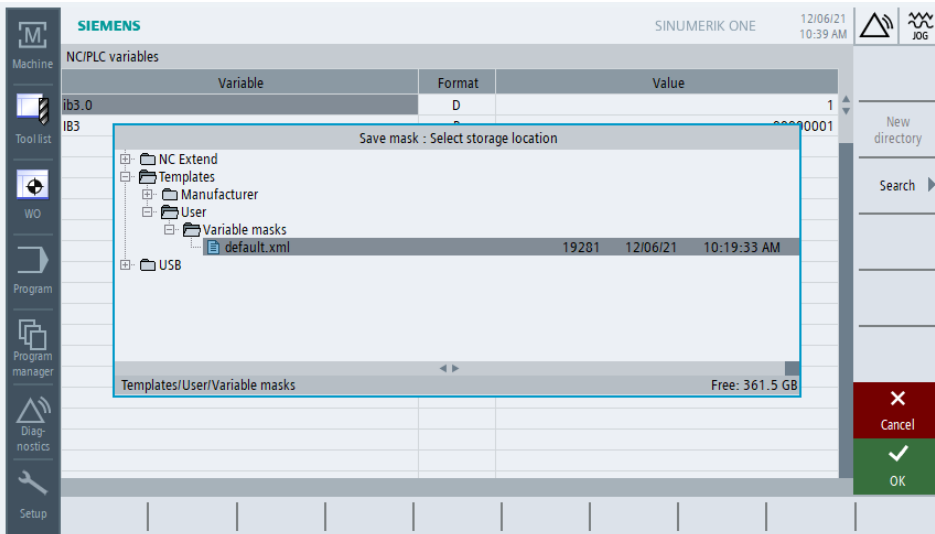
If you change a screen form that has been loaded, then this is marked using with * after the screen form name. The name of a screen form is kept in the display after switching-off.

Procedure

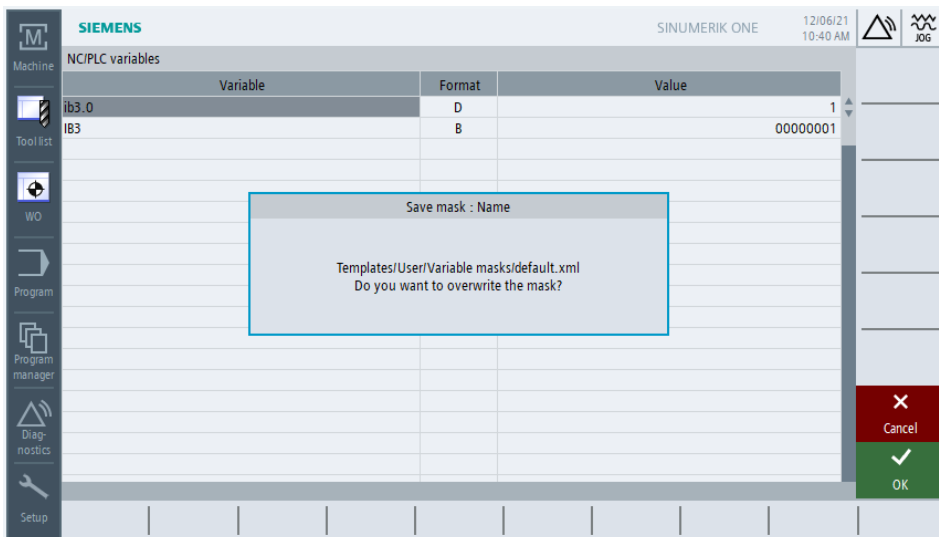
1. Once you have entered values for the desired variables in the "NC/PLC variables" window, press the ">>" VSK.



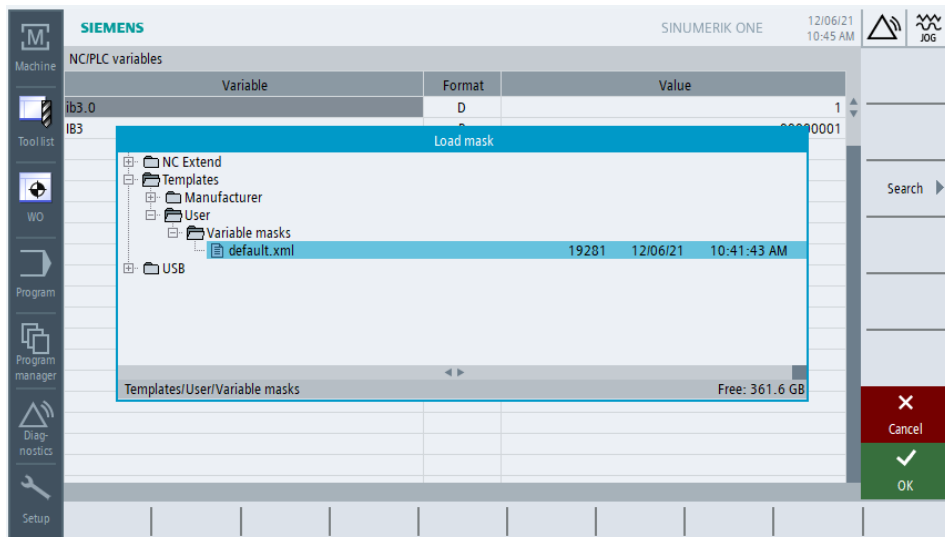
2. Press the "Save screen" VSK. The "Save screen: Select archiving" window opens.



3. Select the template folder for variable screen forms in which your actual screen form should be saved and press the "OK" VSK. The "Save screen: Name" window opens.
4. Enter the name for the file and press the "OK" VSK.
A message in the status line informs you that the screen form was saved in the specified folder. If a file with the same name already exists, then you will receive a prompt.



5. Press the "Load screen" VSK. The "Load screen" window opens and displays the sample folder for the variable screen forms.

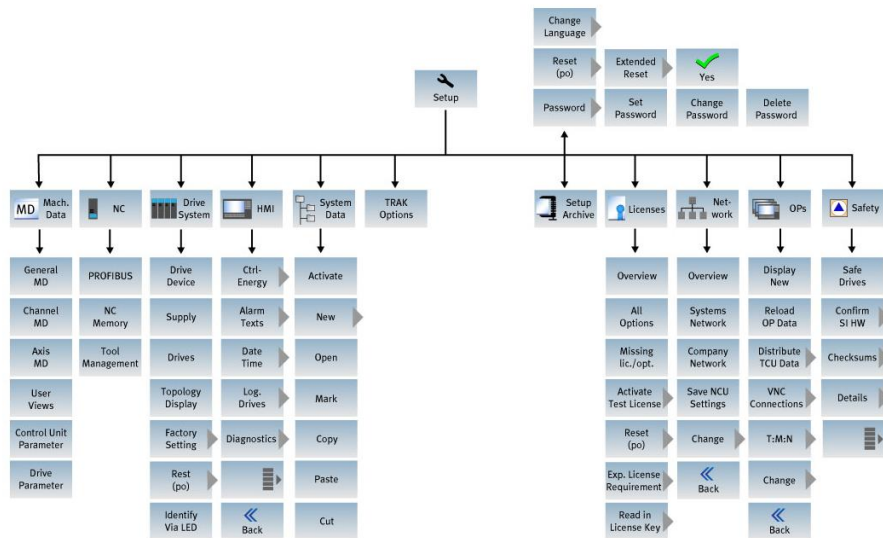


6. Select the desired file and press the "OK" vsk. You return to the variable view. The list of all of the predefined NC and PLC variables is displayed.

10.0 Setup Mode

The Setup menu is used for a wide variety of functions, these include password entry, machine data modifications, creating archives, creating alarm files, time/date settings, network set-up, licensing/option selection, system data management, servo optimization etc.

In this section, we will go over the HSKs, and describe some basic operation procedures. Below is a diagram of all the VSK and HSK options found within Setup Mode for your reference.



10.1 HMI

10.1.1 Assigning a Computer File Location

1. From Setup Mode, press the "HMI" HSK.
2. Press the "Log Drives" VSK (only available at Customer access level or higher).
3. Select the desired slot (note that you can select a slot marked 'reserved')
4. Type: select "windows drive (PCU)"
5. Path: enter in the path to your files – i.e.: D:\Siemens\Programs
6. Symbolic: give it a name – no spaces – i.e.: Name_2
7. Access level: Select appropriate access level ie: Key switch 1, 2, 3, Customer, Service, Manufacturer
8. Select "OK".

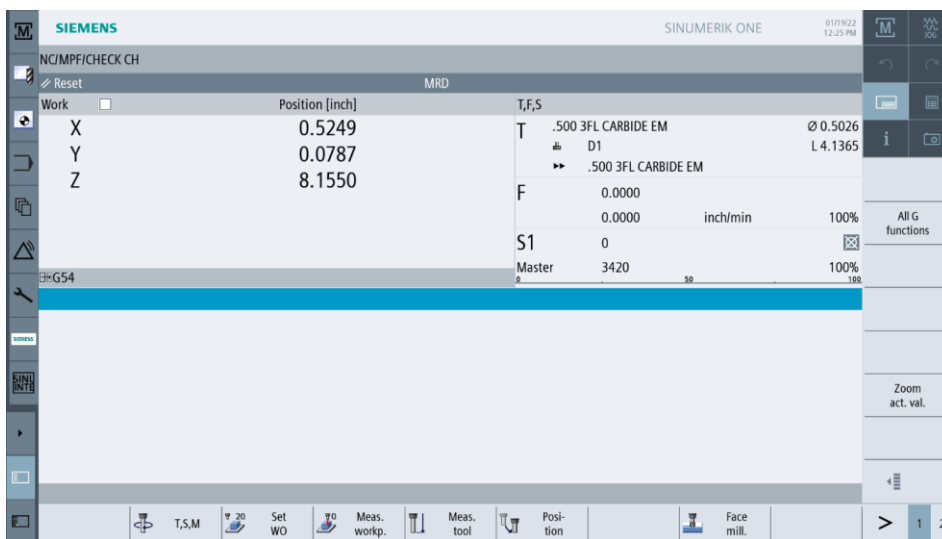
Now you can have your own local computer folder to access and load files.

10.2 System Data

10.2.1 Screenshots

You can create screenshots of the current user interface. Each screenshot is saved as a file and stored in the following folder: HMI data/Logs/Screenshots

A screenshot of the current user interface is created in .png format. The file names assigned by the system are in ascending order from "SCR_SAVE_0001.png" to "SCR_SAVE_9999.png". You can create up to 9,999 screenshots.



Procedure

1. Select the "Setup" operating area.
2. Press the "System data" softkey.
3. Open the folder specified above, and select the required screenshots.
4. Press the "Copy" softkey.
-OR -
Press the "Cut" softkey.
5. Open the required archive directory, e.g. on a USB flash drive and press the "Paste" softkey.

10.2.2 Generating an archive via the system data

If you only want to backup specific data, then you can select the desired files directly from the data tree and generate an archive.

Archive formats

You can save your archive in the binary format. You can display the content of the selected files (XML, ini, hsp, syf files, programs) using a preview. You can display information about the file, such as path, name, date of creation and change, in a properties window.

Requirement

The access rights depend on the relevant areas and range from protection level 7 (key switch position 0) to protection level 2 (password: Service).

Storage locations

- SD Card under
/user/sinumerik/data/archive or
/oem/sinumerik/data/archive
- All configured logical drives (USB, network drives)

Software option

In order to save archives on the SD Card in the user area, you require the "Additional HMI user memory on NCU memory card" option.

NOTE - Possible data loss when using USB flash drives. USB-Flash Drives are not suitable as persistent memory media.

Procedure

1. From Setup Mode, press the "System data" HSK. The data tree opens.
2. In the data tree, select the required files from which you want to generate an archive.
-OR -
If you want to back up several files or directories, press the "Mark" VSK. Select the files.
If you press the ">>" VSK, further softkeys are displayed on the vertical bar.
3. Press the "Preview window" VSK. The contents of the selected file are displayed in a small window. Press the "Preview window" VSK again to close the window.
4. Press the "Properties" VSK. Information about the selected file is displayed in a small window. Press the "OK" VSK to close the window.
5. Press the "Search" VSK. Enter the required search term in the search dialog and press the "OK" VSK if you wish to search for a specific directory or subdirectory.
Note: The place holders "*" (for any character string) and "?" (for any character) make it easier for you to perform a search.
6. Press the "Archive" and "Generate archive" VSK. The "Generate Archive: Select Storage Location" window opens. The "Archive" folder with the subfolders "User" and "Manufacturer" as well as the storage media (e.g. USB) are displayed.
7. Select the required location for archiving and press the "New directory" VSK to create a suitable subdirectory. The "New Directory" window opens.
8. Enter the required name and press the "OK" VSK. The directory is created below the selected folder.
9. Press the "OK" VSK. The "Generate Archive: Name" window opens.
10. Select the format, enter the required name and press the "OK" VSK to archive the file/files. A message informs you if archiving was successful.
11. Press the "OK" VSK to confirm the message and end the archiving operation. An archive file is created in the selected directory.

10.2.3 Read in archive from system data

If you want to read in a specific archive, you can select this directly from the data tree.

Procedure

1. From the Setup mode, press the "System data" HSK.

2. In the data tree below the "Archive" directory, in the "User" folder, select the file that you wish to read in.
3. Press the "Read in" VSK.
4. Press the "OK" or "Overwrite all" VSK to overwrite existing files.
-OR -
Press the "Do not overwrite" VSK if you do not want to overwrite already existing files.
-OR -
Press the "Skip" VSK if the read-in operation is to be continued with the next file.
The "Read In Archive" window opens and a progress message box appears for the read-in process.
You will then obtain a "Read error log for archive" in which the skipped or overwritten files are listed.
5. Press the "Cancel" VSK to cancel the read-in process.

10.3 TRAK MT Options

This menu is to enable/disable the 4th Axis Option. See Section 4.7 for more information.

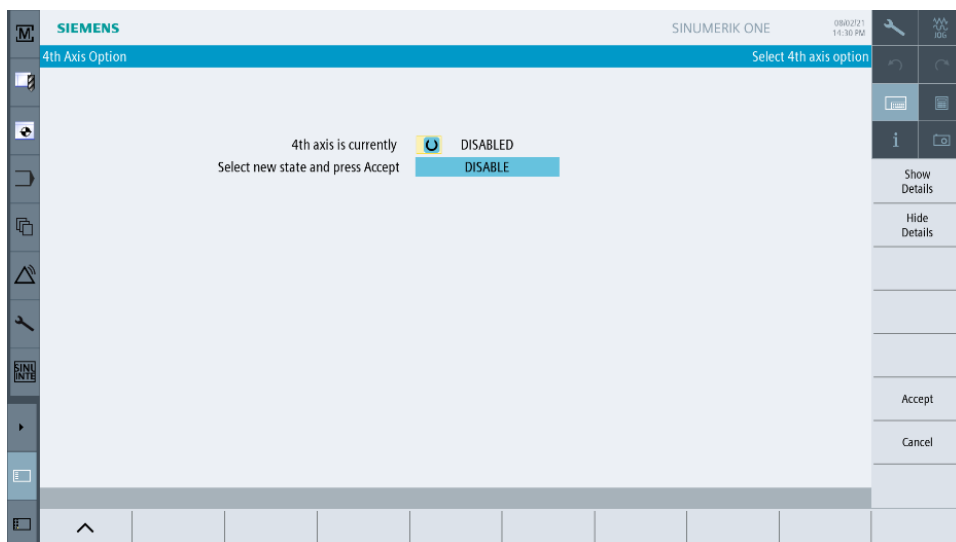


Figure 10.a – 4th Axis Option screen

10.4 Licenses

For additional / renewal licenses, please contact our Customer Service department. They will supply you with a link to download your license files.

Procedure

1. Download your files from the link provided by Customer Service and save them onto a USB drive.
2. Go to the controller and insert your USB onto the right side of the pendant.
3. From Setup mode, press the ">" and "Licenses" HSKs. The Vertical Soft Keys (VSK) show the options available to access your license.

4. Press the "Read in license key" VSK. You will see a window open with files, select your USB, and find your new license file (.zip file).
5. Press the "OK" VSK. The files window will close.
6. Press E-stop, then press the "Reset (po)" VSK to restart your controller, without powering off the machine.
7. Once restarted, the new license should now be active.

11.0 Context help

The Context help screen can assist you in better understanding the window you are in, and its field requirements. The Context help screen is often supported with guiding graphics.

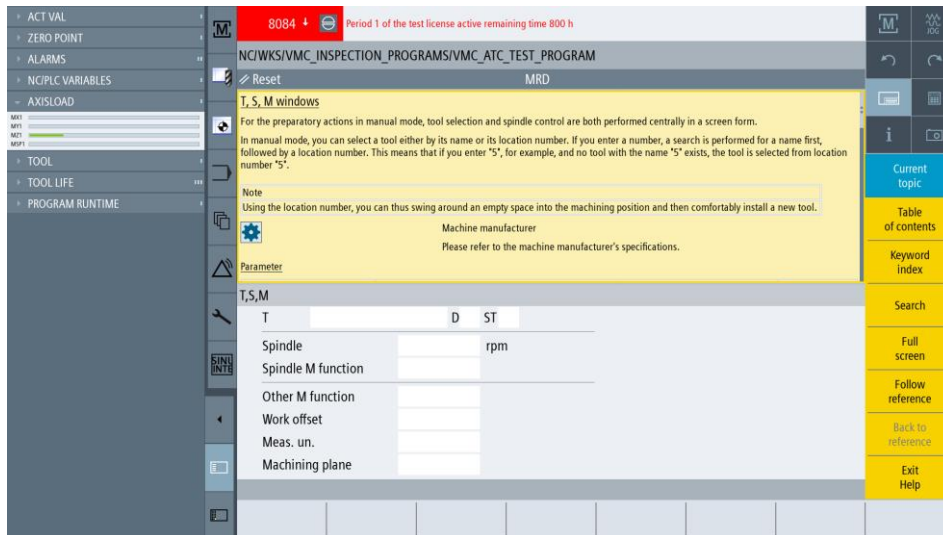


Figure 11.a - T,S,M Help Screen

T, S, M windows

For the preparatory actions in manual mode, tool selection and spindle control are both performed centrally in a screen form.

In manual mode, you can select a tool either by its name or its location number. If you enter a number, a search is performed for a name first, followed by a location number. This means that if you enter "5", for example, and no tool with the name "5" exists, the tool is selected from location number "5".

Note
Using the location number, you can thus swing around an empty space into the machining position and then comfortably install a new tool.


 Machine manufacturer
Please refer to the machine manufacturer's specifications.

Figure 11.b - T,S,M Context help screen - Information about the window you are in

Parameter

Parameter	Meaning	Unit								
T	Input of the tool (name or location number) You can select a tool from the tool list using the "Select tool" softkey.									
D	Cutting edge number of the tool (1 - 9)									
ST	Sister tool number (1 - 99 for sister tool strategy)									
Spindle	Spindle selection, identification with spindle number									
Spindle M function	<table border="0"> <tr> <td>⊗</td> <td>Spindle off: Spindle is stopped</td> </tr> <tr> <td>⌚</td> <td>CW rotation: Spindle rotates clockwise</td> </tr> <tr> <td>⌚</td> <td>CCW rotation: Spindle rotates counterclockwise</td> </tr> <tr> <td>⌚</td> <td>Spindle positioning: Spindle is moved to the desired position.</td> </tr> </table>	⊗	Spindle off: Spindle is stopped	⌚	CW rotation: Spindle rotates clockwise	⌚	CCW rotation: Spindle rotates counterclockwise	⌚	Spindle positioning: Spindle is moved to the desired position.	
⊗	Spindle off: Spindle is stopped									
⌚	CW rotation: Spindle rotates clockwise									
⌚	CCW rotation: Spindle rotates counterclockwise									
⌚	Spindle positioning: Spindle is moved to the desired position.									

Figure 11.c - T,S,M Context help screen > Parameter

See also:
[Selecting a tool](#)
[Starting and stopping a spindle manually](#)
[Position spindle](#)

Figure 11.d - T,S,M Context help screen - Additional Links

To access the Context help screen:

- you may do so at any time, by just pressing the “i” button, located in the Function Key Block (top right side of screen).

Vertical Softkeys

- **Current topic** - Refers to the content-sensitive context help button. In this case it would give you help based on the function you were in at the time of pressing the help button.
- **Table of contents** - Allows you to browse through a table of contents to access all the manuals included under help.
- **Keyword index** - You can search for specific terms, functions, commands, etc. by giving you a list.
- **Search** - You will type in your search criteria.
- **Full screen** - Full screen allows you to expand the Context help screen, to full screen.
- **Follow reference** - Opens whatever is highlighted by the cursor. Same as simply selecting something with the touch screen.
- **Back to reference** - Goes back to the first (current topic)/previous page.
- **Exit help** - Closes the context help window.

12.0 Network & Operator Panels

12.1 Configuring a Network Drive

The transfer of programs can be achieved by mapping a soft-key to a networked Computer. The soft-key will appear in the Program Manager area of the controller.

The computer connected to the network must have been logged in with a username and password. This information will be required for inputting on the controller.

A shared folder is required on the PC for the storage of files.

For reasons of security it is possible to restrict network access to the shared folder by setting "Permissions".

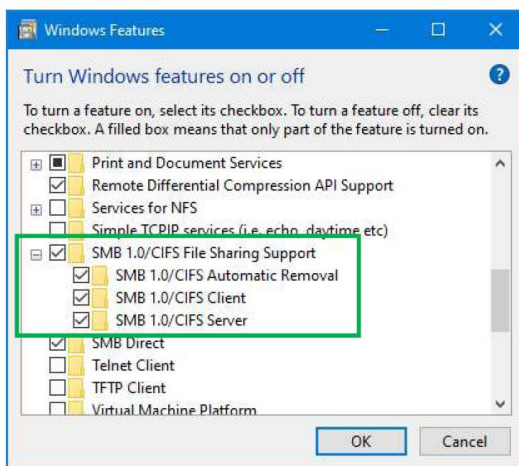
The following example will show how to create a folder on the D drive and how to set up sharing and permissions.

The option "Use Sharing Wizard" should be deselected via the "Tools" menu in Windows@ Explorer:

Procedure

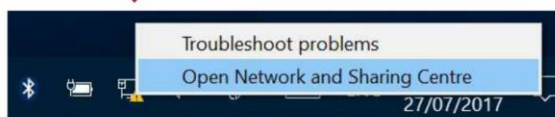
1. Switch on a Windows feature.
2. Go to: "**Apps and Features**", then "**Program Features**"
3. Select "**Turn Windows features on or off**"
4. Select "**SMB I.O/CIFS Files Sharing Support**"
Including
 - "**SMB 1 .O/CIFS Automatic Removal**"
 - "**SMB I.O/CIFS Client**"
 - "**SMB I.O/CIFS Server**"

This feature turns on file sharing between Windows and Linux.

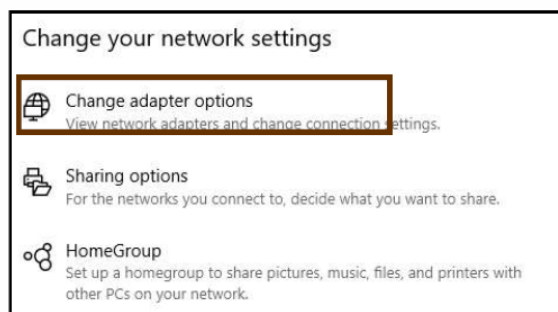


5. Restart the PC

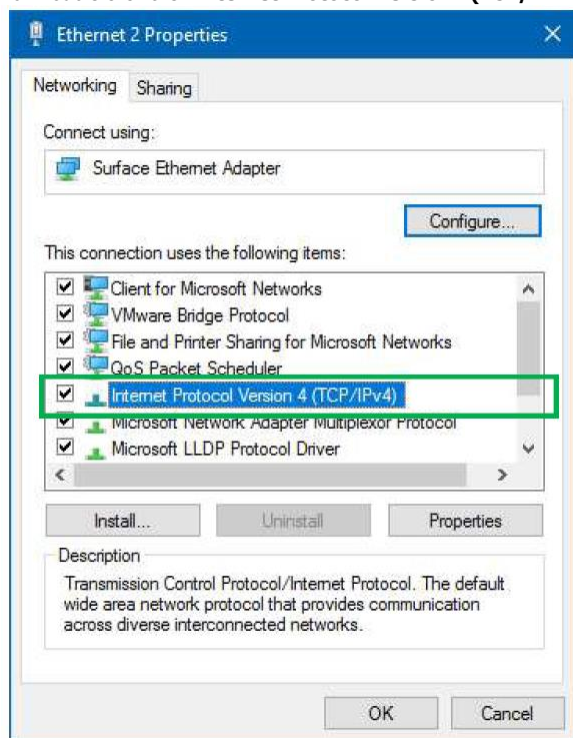
6. The computer must now be given an IP address that is in the same subnet mask.
 - The IP address for this example is: **192.168.100.2** with a sub-net mask of **255.255.255.0**.
7. Locate and open the "Network and Sharing Center" of the computer to be connected to the controller.
8. Right click the network icon.



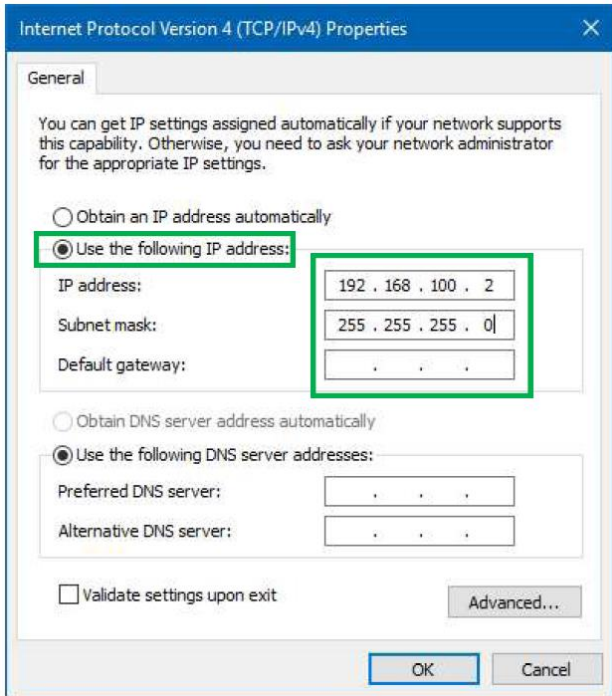
9. Select "Open Network and Sharing Center" as shown above, then follow the steps shown in the images below.



10. Double click the "Internet Protocol Version 4(TCP/IPv4)" option.



11. The IP address and the subnet mask can now be entered:



12. Press "OK".

12.2 Computer – Shared Folder

A shared folder allows it to be accessed via the network. The folder is required for the storage of programs which will be accessible from the controller.

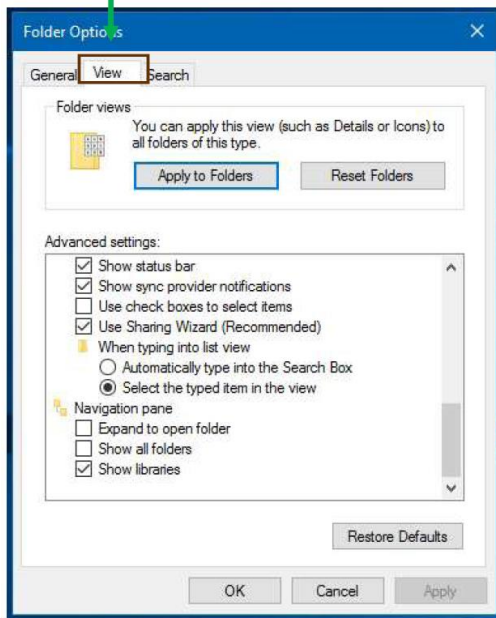
For reasons of security it is possible to restrict access to the shared folder by setting permissions.

The following example will show how to create a folder on the D drive and how to set up sharing and permissions.

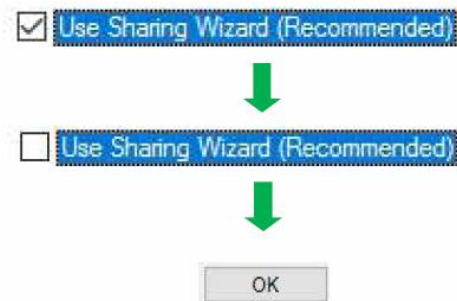
The option "Use sharing Wizard" can be deselected, via the Tools menu in Windows Explorer, to ease setup:

Procedure

1. From Windows@ Explorer, select the **"Tools"** menu, followed by Folder **"Options"**:
2. Select **"View"**



3. Deselect the **"Use Sharing Wizard"** option:



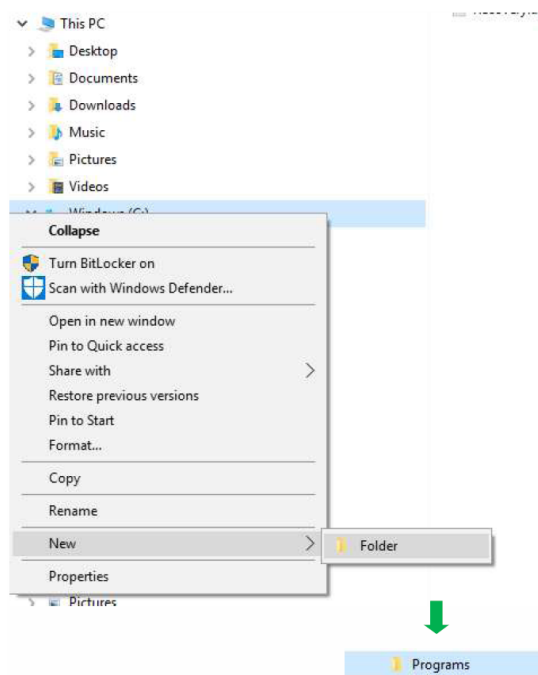
4. Press **"OK"**

12.2.1 Creating a New Folder

Procedure (Example)

In Windows® Explorer:

1. Select the "D" drive.
2. With the mouse pointer in the right hand window, right click and **create a new folder**.

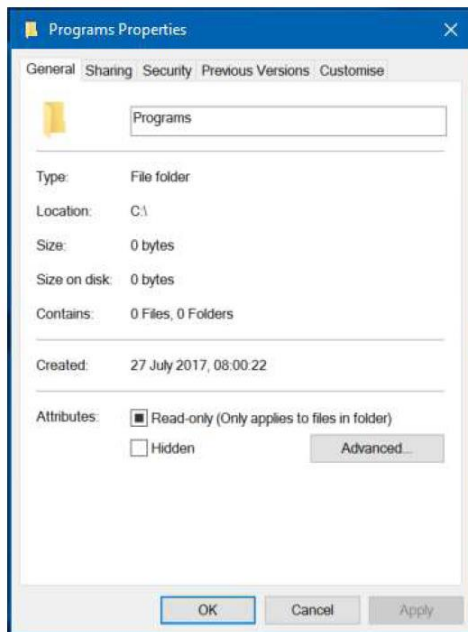


3. Give a suitable name to the folder.
Example: **Programs**.

12.2.2 Sharing the New Folder

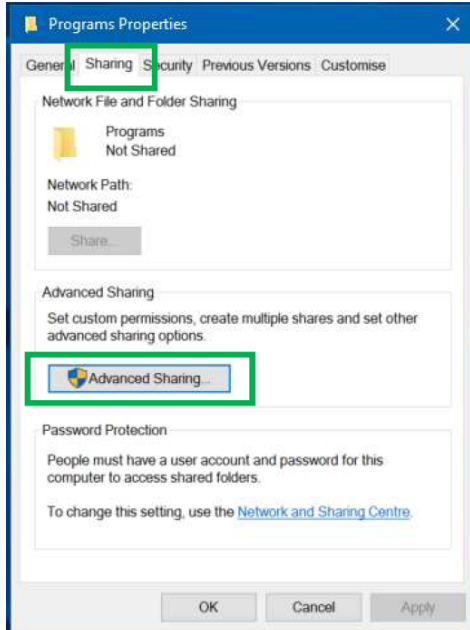
Procedure

1. Right click with the mouse pointer on the “**Programs**” folder.
2. Select the “**Properties**” option from the list.

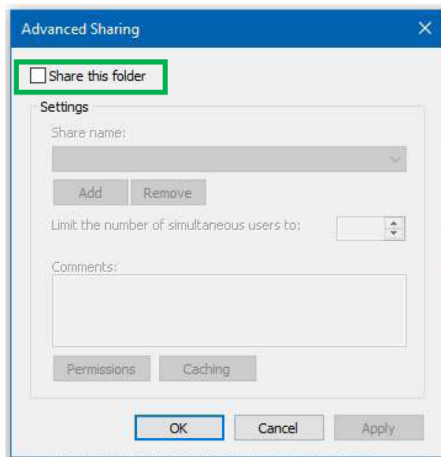


3. Select the “**Sharing**” tab.

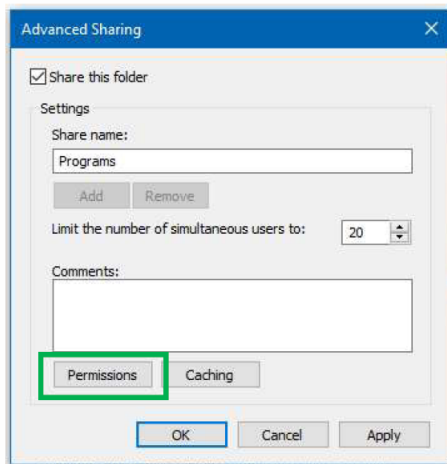
4. Select the “**Advanced Sharing**” tab.



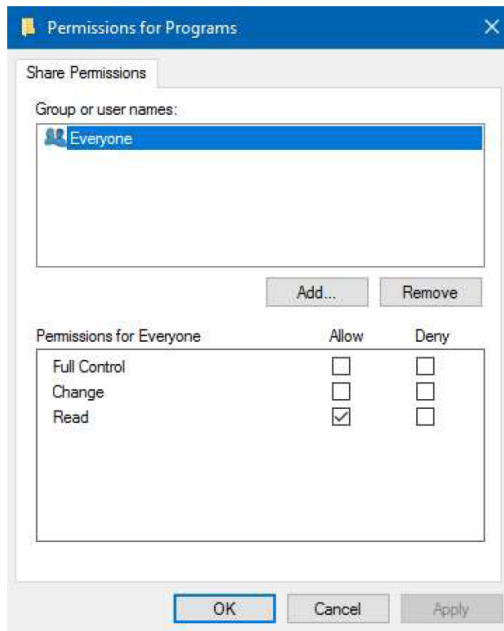
5. Select “**Share this folder**”



6. Select "Permissions"

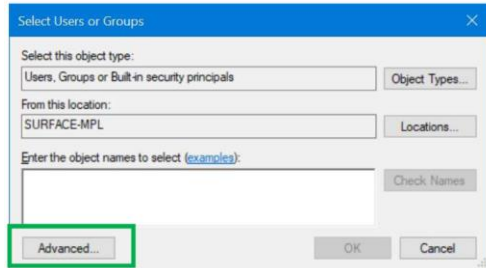


The default setting is for everyone to have read-only access to the folder. It is possible to restrict access to the shared folder by other users of the PC.



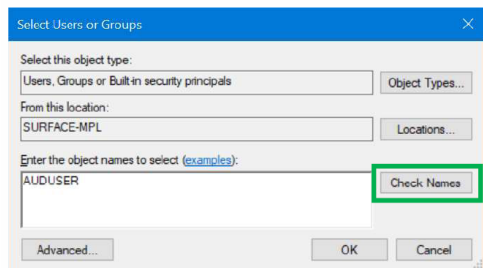
7. If restrictions are required, the Everyone user can be removed or the permissions deselected. Users can then be added via the "Add" button.

8. The required users of the computer can be searched for in the "Advanced..." field.

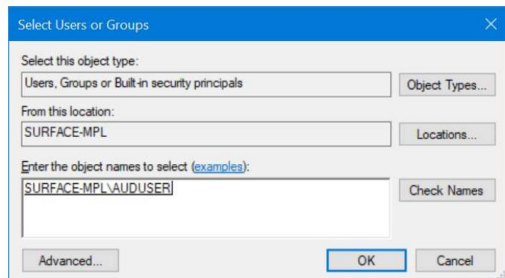


In the following example, the user **AUDUSER** is going to be given permission to access the shared folder.

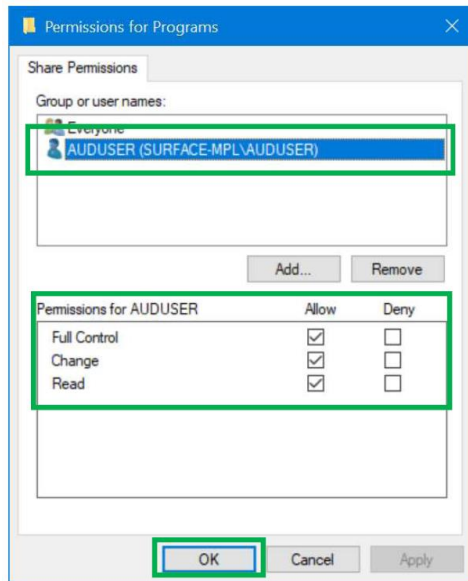
9. Enter **AUDUSER** in the "Check Names" field, then select the "Check Names" button.



10. If the user exists, it will be displayed.



Example: *AUDUSER* has been granted full access to the shared "**Programs**" folder. The Share permission for *Everyone* has been removed.



12.3 Logical Drive Set-Up

12.3.1 Mapping the Shared Folder

The folder previously set up for sharing now needs to be mapped from the controller to allow programs to be transferred between the controller and PC.

The following procedure will create a soft-key in the Program manager area which, when selected will display the content of the shared folder and allow file transfer using copy and paste functions.

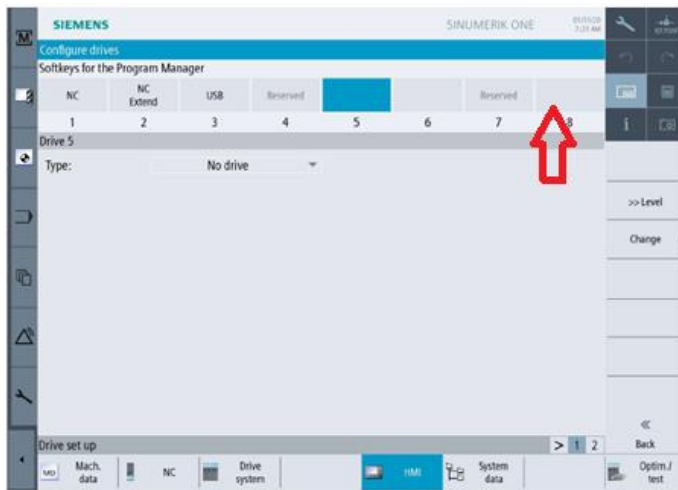
Procedure

1. Use the following key selection to reach the Logical Drives configuration screen:

Enter the mfg password to get to LOG DRIVES vertical soft key.

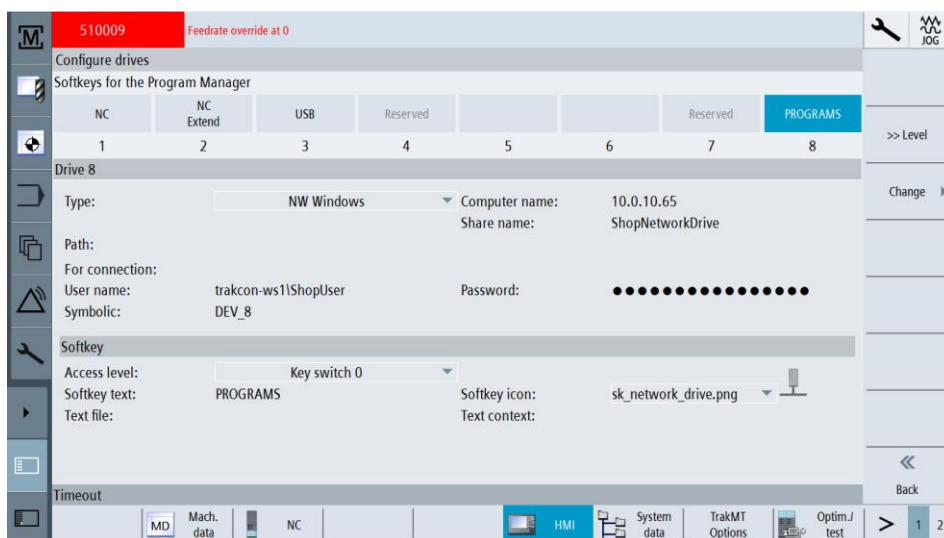


Press the button shown with the red arrow. The shared folders on the network computer will be set up as a drive on the control using this button.

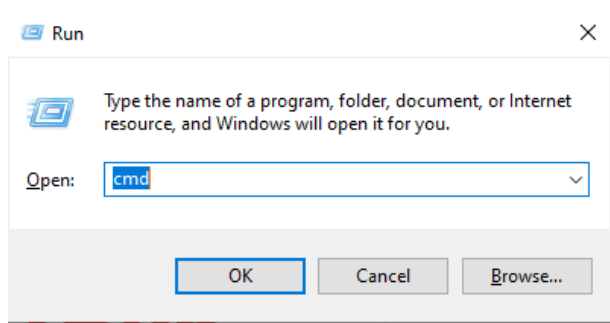


2. Press Change button.
3. Select **Type** and from drop down, change to NW Windows.

4. This screen below will now show up. Fill in the networked computer name, share name of folder you are sharing, user name and password on networked computer and then softkey text. The softkey text is the name you want on the new button. In our case we used **PROGRAMS**.

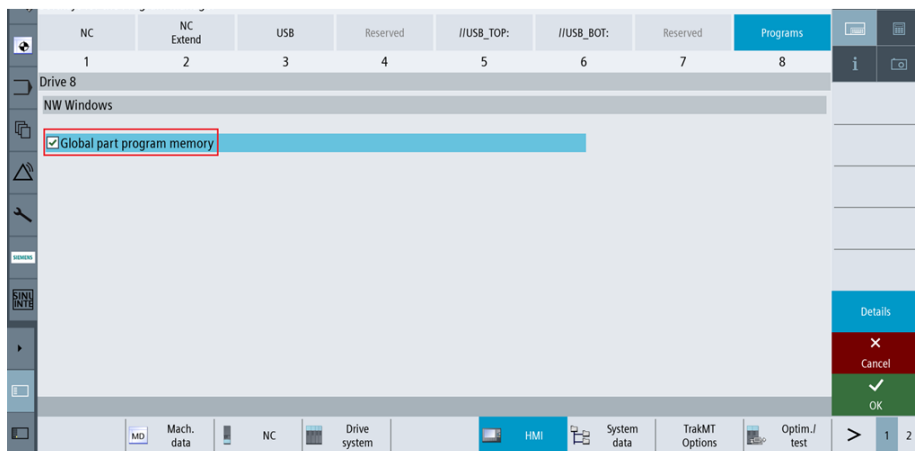


5. To find the computers User Name.
 - a. Press windows (keyboard button) + r

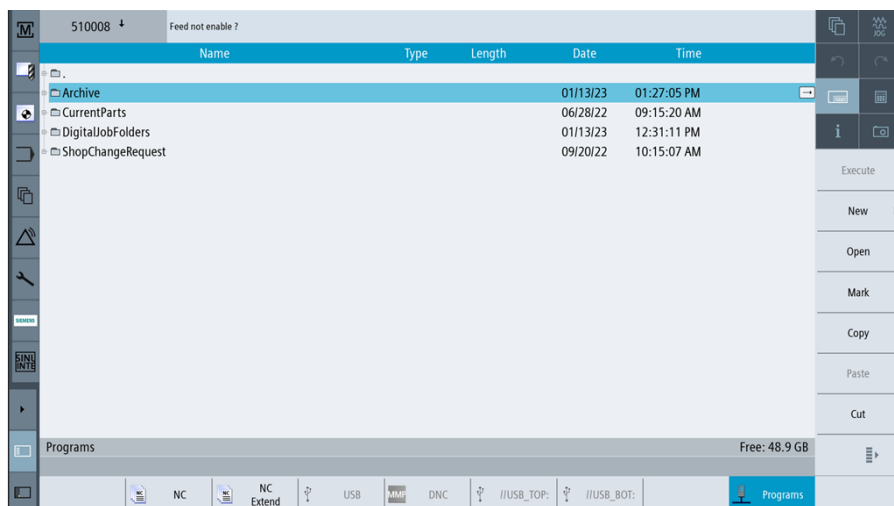


- b. In the Run screen, type in cmd.
- c. In the command prompt type in whoami.
- d. The format that it expects Domain\UserName if you are on a domain, or ComputerName\UserName if you are not.

6. Change **Access Level** to key switch 0.
7. Once all info is entered, press Details button and make sure the Global part program memory box is checked. Note – requires the P75 software option license. This allows users to run programs from external memory.



8. Press **OK** once complete and the control will now try to connect to the network computer. A please wait message will be on the screen.
9. Once this stops, go to the PROGRAM MANAGER folder and make sure you see the new softkey in bold as shown below named Programs and folders you are sharing. In our example, we are sharing the following 4 folders.



13.0 G and M Codes

13.1 List of recognized ProtoTRAK G-codes for Siemens

G Code	Description	Acceptable Format	Notes
G00	Position at maximum feedrate	G00 X0.0 Y0.0 Z0.0	
G01	Linear Interpolation	G01 X0.0 Y0.0 Z-1. F15.	F = Feedrate in Inches per minute
G02	Circular Interpolation (Clockwise)	G02 X1. Y2. Z0. I0. J2. K0.	I, J, & K define the X, Y, & Z center points respectively, incremental from the starting point of the arc.
G03	Circular Interpolation (Counter-clockwise)	G03 X2. Y0. Z0. I2. J0. K0.	
G17	Select XY Plane	G17	G17 - G19 only apply to G2 and G3.
G18	Select XZ Plane	G18	
G19	Select YZ Plane	G19	
G20	Input in Inch	G20	
G21	Input in Metric	G21	
G40	Cutter compensation cancel	G40	
G41	Cutter compensation left	G41	
G42	Cutter compensation right	G42	
G54	Fixture offset 1 (Base Offset)	G54	
G55	Fixture offset 2	G55	The offset difference from each other is defined conversationally on our control.
G56	Fixture offset 3	G56	
G57	Fixture offset 4	G57	
G58	Fixture offset 5	G58	
G59	Fixture offset 6	G59	
G80	Canned Cycle Cancel	G80	
G81	Drilling Cycle	G81 X0. Y0. Z-1. R.1 F15.	Z = final depth R = rapid position
G82	Drilling Cycle, Dwell at Bottom	G82 X0. Y0. Z-1. R.1 P500 F15.	P = dwell time in milliseconds
G83	Peck Drilling Cycle – Variable Peck	G83 X0. Y0. Z-1. R.1 I.25 J.1	I = Initial peck amount J = Minimum peck
G84	Tapping cycle	G84 X0. Y0. Z-2. R.1 S300	S defines RPM Q defines Pitch Feed = S x Q
G85	Boring Cycle	G85 X0. Y0. Z-1 R.1 F15.	
G90	Absolute Programming	G90	Default
G91	Incremental Programming	G91	
G94	Feed per Minute	G94	Default
G95	Feed per Rev	G95	

13.2 List of Siemens G-codes

The G commands are divided into G groups. In part programs or synchronized actions, in a block, only a G command of a G group can be written. A G command can be modal or non-modal.

Modal: up to programming another G command of the same G group.

13.2.1 G Group 1: Modality valid motion commands

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G0	Rapid Traverse	+	m		
G1	Linear interpolation (linear interpolation)	+	m	x	
G2	Circular interpolation clockwise	+	m		
G3	Circular interpolation counter-clockwise	+	m		
CIP	Circular interpolation through intermediate point	+	m		
ASPLINE	Akima spline	+	m		
BSPLINE	B spline	+	m		
CSPLINE	Cubic spline	+	m		
POLY	Polynomial interpolation	+	m		
G33	Thread cutting with constant lead	+	m		
G331	Tapping	+	m		
G332	Retraction (tapping)	+	m		
OEMIPO1	Reserved	+	m		
OEMIPO2	Reserved	+	m		
CT	Circle with tangential transition	+	m		
G34	Thread cutting with linear increasing lead	+	m		
G35	Thread cutting with linear decreasing lead	+	m		
INVCW	Involute interpolation clockwise	+	m		
INVCCW	Counter-clockwise involute interpolation	+	m		
G335	Turning a convex thread in clockwise direction	+	m		
G336	Turning a convex thread in counter-clockwise direction	+	m		

13.2.2 G Group 2: Non-modality valid motion commands

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G4	Dwell time, preset	-	s		
G63	Tapping without synchronization	-	s		
G74	Reference point approach with synchronization	-	s		
G75	Fixed-point approach	-	s		
REPOSL	Linear repositioning	-	s		
REPOSQ	Repositioning in a quadrant	-	s		
REPOSH	Repositioning in semicircle	-	s		
REPOSA	Linear repositioning with all axes	-	s		
REPOSQA	Linear repositioning with all axes, geometry axes in quadrant	-	s		
REPOSHA	Repositioning with all axes; geometry axes in semicircle	-	s		
G147	Approach contour with straight line	-	s		
G247	Approach contour with quadrant	-	s		
G347	Approach contour with semicircle	-	s		

G148	Leave contour with straight line	-	S		
G248	Leave contour with quadrant	-	S		
G348	Leave contour with semicircle	-	S		
G5	Oblique plunge-cut grinding	-	S		
G7	Compensatory motion during oblique plunge-cut grinding	-	S		

13.2.3 G Group 3: Programmable frame, working area limitation and pole programming

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
TRANS	TRANSLATION: Programmable offset	-	S		
ROT	ROTATION: Programmable rotation	-	S		
SCALE	SCALE: Programmable scaling	-	S		
MIRROR	MIRROR: Programmable mirroring	-	S		
ATRANS	Additive TRANSLATION: Additive programmable translation	-	S		
AROT	Additive ROTATION: Programmable rotation	-	S		
ASCALE	Additive SCALE: Programmable scaling	-	S		
AMIRROR	Additive MIRROR: Programmable mirroring	-	S		
G25	Minimum working area limitation/spindle speed limitation	-	S		
G26	Maximum working area limitation/spindle speed limitation	-	S		
G110	Pole programming relative to the last programmed setpoint position	-	S		
G111	Polar programming relative to origin of current workpiece coordinate system	-	S		
G112	Pole programming relative to the last valid pole	-	S		
G58	Absolute programmable work offset	-	S		
G59	Additive programmable work offset	-	S		
ROTS	Rotation with solid angle	-	S		
AROTS	Additive rotation with solid angle	-	S		

13.2.4 G Group 4: FIFO

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
STARTFIFO	Start FIFO Execute and simultaneously fill preprocessing memory	+	m	x	
STOPFIFO	STOP FIFO Stop machining; fill preprocessing memory until STARTFIFO is detected, FIFO is full or end of program	+	m		
FIFOCTRL	Activation of automatic preprocessing memory control	+	m		

13.2.5 G Group 6: Plane Selection

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G17	Plane selection 1. – 2. Geometry axis	+	m	x	

G18	Plane selection 3. – 1. Geometry axis	+	m		
G19	Plane selection 2. – 3. Geometry axis +	+	m		

13.2.6 G Group 7: Tool radius compensation

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G40	No tool radius compensation	+	m	x	
G41	Tool radius compensation left of the contour	-	m		
G42	Tool radius compensation right of the contour	-	m		

13.2.7 G Group 8: Settable work offset

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G500	Deactivation of settable work offset (G54 to G57, G505 to G599)	+	m	x	
G54	1 st settable work offset	+	m		
G55	2 nd settable work offset	+	m		
G56	3 rd settable work offset	+	m		
G57	4 th settable work offset	+	m		
G505	5 th settable work offset	+	m		
...	...	+	m		
G599	99 th settable work offset	+	m		

Each of the G commands in this G group is used to activate an adjustable user frame \$P_UIFR[]. G54 corresponds to frame \$P_UIFR[1], G505 corresponds to frame \$P_UIFR[5]. The number of adjustable user frames and therefore the number of G commands in this G group can be set using machine data MD28080 \$MC_MM_NUM_USER_FRAMES.

13.2.8 G Group 9: Frame and tool offset suppression

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G53	Suppression of current frames: Programmable frame including system frame for TOROT and TOFRAME and active adjustable frame (G54 to G57, G505 to G599)	-	s		
SUPA	As for G153 including suppression of system frames for actual value setting, scratching, external work offset, PAROT including handwheel offsets (DRF), [external work offset], overlaid movement	-	s		
G153	As for G53 including suppression of all channel-specific and/or NCU-global basic frames	-	s		
SUPD	Suppression of the active tool offsets	-	s		

13.2.9 G Group 10: Exact stop – continuous path mode

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G60	Exact stop	+	m	x	
G64	Continuous-path mode	+	m		
G641	Continuous-path mode with smoothing according to distance criterion (= programmable rounding clearance)	+	m		

G642	Continuous-path mode with smoothing within the defined tolerances	+	m		
G643	Continuous-path mode with smoothing within the defined tolerances (block-internal)	+	m		
G644	Continuous-path mode with smoothing with maximum possible dynamic response	+	m		
G645	Continuous-path mode with smoothing and tangential block transitions within defined tolerances	+	m		

13.2.10 G Group 11: Exact stop, non-modal

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G60	Exact stop	+	m		

13.2.11 G Group 12: Block change criteria at exact stop (G60/G9)

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G601	Block change at exact stop fine	+	m	x	
G602	Block change at exact stop coarse	+	m		
G603	Block change at IPO block end	+	m		

13.2.12 G Group 13: Workpiece measuring inch/metric

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G70	Input system inches (length)	+	m		
G71	Input system metric mm (lengths)	+	m	x	
G700	Input system inch, inch/min (lengths + velocity + system variable)	+	m		
G710	Input system metric mm, mm/min (lengths + velocity + system variable)	+	m		

13.2.13 G Group 14: Workpiece measuring absolute/incremental

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G90	Absolute dimension	+	m	x	
G91	Incremental dimensions	+	m		

13.2.14 G Group 15: Feed type

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G93	Inverse-time feedrate rpm	+	m		
G94	Linear feedrate in mm/min, inch/min	+	m	x	
G95	Revolutional feedrate in mm/rev, inch/rev				
G96	Revolutional feedrate (as for G95) and constant cutting rate				
G97	Revolutional feedrate and constant spindle speed (constant cutting rate OFF)				
G931	Feedrate specified by means of traversing time, deactivate constant path velocity				

G961	Linear feedrate (as for G94) and constant cutting rate				
G971	Linear feedrate and constant spindle speed (constant cutting rate OFF)				
G942	Linear feedrate and constant cutting rate or constant spindle speed				
G952	Revolutional feedrate and constant cutting rate or constant spindle speed				
G962	Linear feedrate or revolutional feedrate and constant cutting rate				
G972	Linear feedrate or revolutional feedrate and constant spindle speed (constant cutting rate OFF)				
G973	Revolutional feedrate without spindle speed limitation and constant spindle speed (G97 without LIMS for ISO mode)				

13.2.15 G Group 16: Feedrate override at inside and outside curvature

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
CFC	Constant feedrate at contour effective for internal and external radius	+	m	x	
CFTCP	Constant feedrate in tool center point (center point path)	+	m		
CFIN	Constant feedrate for internal radius only, acceleration for external radius	+	m		

13.2.16 G Group 17: Approach and retraction response, tool offset

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
NORM	Normal position at starting and end points	+	m	x	
KONT	Travel around contour at starting and end points	+	m		
KONTT	Approach/retraction with constant tangent	+	m		
KONTC	Approach/retraction with constant curvature	+	m		

13.2.17 G Group 18: Corner behavior, tool offset

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G450	Transition circle (tool travels around workpiece corners on a circular path)	+	m	x	
G451	Intersection of equidistant paths (tool backs off from the workpiece corner)	+	m		

13.2.18 G Group 19: Curve transition at beginning of spline

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
BNAT	Natural transition to first spline block	+	m	x	
BTAN	Tangential transition to first spline block	+	m		
BAUTO	Definition of the first spline section by means of the next 3 points	+	m		

13.2.19 G Group 20: Curve transition at end of spline

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
ENAT	Natural transition to next traversing block	+	m	x	
ETAN	Tangential transition to next traversing block	+	m		
EAUTO	Definition of the last spline section by means of the last 3 points	+	m		

13.2.20 G Group 21: Acceleration profile

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
BRISK	Fast non-smoothed path acceleration	+	m	x	
SOFT	Soft smoothed path acceleration	+	m		
DRIVE	Velocity-dependent path acceleration	+	m		

13.2.21 G Group 22: Tool offset type

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
CUT2D	2½D TRC	+	m	x	
CUT2DF	2½D TRC relative to the current frame (inclined plane)	+	m		
CUT3DC	3D TRC for circumferential milling	+	m		
CUT3DF	3D TRC for face milling with change in orientation	+	m		
CUT3DFS	3D TRC for face milling with constant orientation. The tool orientation is defined by G17 - G19 and is not influenced by frames.	+	m		
CUT3DFF	3D TRC for face milling with constant orientation. The tool orientation is the direction defined by G17 - G19 and, in some cases, rotated by a frame.	+	m		
CUT3DCC	3D TRC for circumferential milling taking into account a limitation surface with 3D radius compensation: Contour on the machining surface	+	m		
CUT3DCCD	3D TRC for circumferential milling taking into account a limitation surface with a differential tool on the tool center-point path: Infeed to the limitation surface	+	m		
CUT2DD	2½ D TRC in relation to the differential tool	+	m		
CUT2DFD	2½D TRC in relation to a differential tool relative to the current frame (inclined plane)	+	m		
CUT3DCD	3D TRC in relation to a differential tool for circumferential milling	+	m		
CUT3DFD	3D TRC in relation to a differential tool for face milling with change in orientation	+	m		

13.2.22 G Group 23: Collision monitoring at inside hours

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
CDOF	Collision detection OFF	+	m	x	
CDON	Collision detection ON	+	m		
CDOF2	Collision detection OFF for 3D circumferential milling	+	m		

13.2.23 G Group 24: Precontrol

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
FFWOF	Feedforward control OFF	+	m	x	
FFWON	Feedforward control ON				

13.2.24 G Group 25: Tool orientation reference

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
ORIWKS	Tool orientation in workpiece coordinate system (WCS)	+	m	x	
ORIMKS	Tool orientation in machine coordinate system (MCS)	+	m		

13.2.25 G Group 26: Repositioning mode for REPOS (modal)

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
RMB	Repositioning to start of block	+	m	x	
RMI	Repositioning to interrupt point	+	m		
RME	Repositioning to end of block	+	m		
RMN	Repositioning to the nearest path point	+	m		

13.2.26 G Group 27: Tool offset for change in orientation at outside corners

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
ORIC	Orientation changes at outside corners are superimposed on the circle block to be inserted	+	m	x	
ORID	Orientation changes are performed before the circle block	+	m		

13.2.27 G Group 28: Working area limitation

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
WALIMON	Working area limitation ON	+	m	x	
WALIMOF	Working area limitation OFF	+	m		

13.2.28 G Group 29: Radius/diameter programming

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
DIAMOF	Modal channel-specific diameter programming OFF Deactivation activates channel-specific radius programming.	+	m	x	
DIAMON	Modal independent channel-specific diameter programming ON The effect is independent of the programmed dimensions mode (G90/G91).	+	m		
DIAM90	Modal dependent channel-specific diameter programming ON	+	m		

	The effect is dependent on the programmed dimensions mode (G90/G91).				
DIAMCYCOF	Modal channel-specific diameter programming during cycle processing OFF	+	m		

13.2.29 G Group 30: NC block compression

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
COMPOF	NC block compression OFF	+/-	m	x	
COMPCAD	Compressor function COMPCAD ON	+/-	m		
COMPSURF	COMPSURF EIN compressor function	+/-	m		

13.2.30 G Group 33: Settable fine tool offset

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
FTOCOF	Online fine tool offset OFF	+	m	x	
FTOCON	Online fine tool offset ON	-	m		

13.2.31 G Group 34: Tool orientation smoothing

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
OSOF	Tool orientation smoothing OFF	+	m	x	
OSC	Continuous tool orientation smoothing	+	m		
OSS	Tool orientation smoothing at end of block	+	m		
OSSE	Tool orientation smoothing at start and end of block	+	m		
OSD	Block-internal smoothing with specification of path length	+	m		
OST	Block-internal smoothing with specification of angular tolerance	+	m		

13.2.32 G Group 35: Punching and nibbling

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
SPOF	Stroke US, punching and nibbling OFF	+	m	x	
SON	Nibbling ON	+	m		
PON	Punching ON	+	m		
SONS	Nibbling ON in interpolation cycle	-	m		
PONS	Punching ON in interpolation cycle	-	m		

13.2.33 G Group 36: Punching with delay

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
PDELAYON	Punching with delay ON	+	m	x	
PDELAYOF	Punching with delay OFF	+	m		

13.2.34 G Group 37: Punching with delay

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
FNORM	Feedrate normal to DIN 66025	+	m	x	

FLIN	Feed linear variable	+	m		
FCUB	Feedrate variable according to cubic spline	+	m		

13.2.35 G Group 38: Assignment of fast inputs/outputs for punching/nibbling

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
SPIF1	Fast NC inputs/outputs for punching/nibbling byte 1	+	m	x	
SPIF2	Fast NC inputs/outputs for punching/nibbling byte 2	+	m		

13.2.36 G Group 39: Programmable contour accuracy

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
CPRECOF	Programmable contour accuracy OFF	+	m	x	
CPRECON	Programmable contour accuracy ON	+	m		

13.2.37 G Group 40: Tool radius compensation constant

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
CUTCONOF	Constant tool radius compensation OFF	+	m	x	
CUTCONON	Constant tool radius compensation ON	+	m		

13.2.38 G Group 41: Interruptible thread cutting

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
LFOF	Interruptible thread cutting OFF	+	m	x	
LFON	Interruptible thread cutting ON	+	m		

13.2.39 G Group 42: Tool carrier

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
TCOABS	Determine tool length components from the current tool orientation	+	m	x	
TCOFR	Determine tool length components from the orientation of the active frame	+	m		
TCOFRZ	Determine tool orientation of an active frame on selection of tool, tool points in Z direction	+			
TCOFRY	Determine tool orientation of an active frame on selection of tool, tool points in Y direction	+			
TCOFRX	Determine tool orientation of an active frame on selection of tool, tool points in X direction				

13.2.40 G Group 43: SAR approach direction

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G140	SAR approach direction defined by G41/G42	+	m	x	
G141	SAR approach direction to left of contour	+	m		
G142	SAR approach direction to right of contour	+	m		

G143	SAR approach direction tangent-dependent	+	m		
------	------------------------------------------	---	---	--	--

13.2.41 G Group 44: SAR path segmentation

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G340	Spatial approach block; in other words, infeed depth and approach in plane in one block	+	m	x	
G341	Start with infeed on perpendicular axis (Z), then approach in plane	+	m		

13.2.42 G Group 45: Path reference for FGROU P axes

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
SPATH	Path reference for FGROU P axes is arc length	+	m	x	
UPATH	Path reference for FGROU P axes is curve parameter	+	m		

13.2.43 G Group 46: Plane selection for fast retraction

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
LFTXT	The plane is determined from the path tangent and the current tool orientation	+	m	x	
LFWP	The plane is determined by the current working plane (G17/G18/G19)	+	m		
LFPOS	Axial retraction to a position				

13.2.44 G Group 47: Mode switchover for external NC code

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G290	Activate SINUMERIK language mode	+	m	x	
G291	Activate ISO language mode	+	m		

13.2.45 G Group 48: Approach and retraction response with tool radius and compensation

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
G460	Collision detection for approach and retraction block ON	+	m	x	
G461	Extend border block with arc if no intersection in TRC block	+	m		
G462	Extend border block with straight line if no intersection in TRC block	+	m		

13.2.46 G Group 49: Point to point motion

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
CP	Path motion	+	m	x	
PTP	Point-to-point motion (synchronized axis motion)	+	m		
PTPG0	Point-to-point motion only with G0, otherwise path motion CP	+	m		

PTPWOC	Point-to-point motion without compensatory motion, which is caused by orientation changes	+	m		
--------	-------------------------------------------------------------------------------------------	---	---	--	--

13.2.47 G Group 50: Orientation programming

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
ORIEULER	Orientation angle via Euler angle	+	m	x	
ORIRPY	Orientation angle via RPY angle (rotation sequence XYZ)	+	m		
ORIVIRT1	Orientation angle via virtual orientation axes (definition 1)	+	m		
ORIVIRT2	Orientation angle via virtual orientation axes (definition 2)	+	m		
ORIAPOS	Orientation angle via virtual orientation axes with rotary axis positions	+	m		
ORIRPY2	Orientation angle via RPY angle (rotation sequence ZYX)	+	m		

13.2.48 G Group 51: Interpolation type for orientation programming

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
ORIVECT	Large-circle interpolation (identical to ORIPLANE)	+	m	x	
ORIAxes	Linear interpolation of machine axes or orientation axes	+	m		
ORIPATH	Tool orientation trajectory referred to path	+	m		
ORIPLANE	Interpolation in plane (identical to ORIVECT)	+	m		
ORICONCW	Interpolation on the peripheral surface of a taper in the clockwise direction	+	m		
ORICONCCW	Interpolation on the peripheral surface of a taper in the counter-clockwise direction	+	m		
ORICONIO	Interpolation on a conical peripheral surface with intermediate orientation setting	+	m		
ORICONTO	Interpolation on a peripheral surface of the cone with tangential transition	+	m		
ORICURVE	Interpolation with additional space curve for orientation	+	m		
ORIPATHS	Tool orientation in relation to the path, kinks in the orientation characteristic are smoothed	+	m		

13.2.49 G Group 52: Interpolation type for orientation programming

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
PAROTOF	Frame rotation in relation to workpiece OFF	+	m	x	
PAROT	Frame rotation in relation to workpiece ON The workpiece coordinate system is aligned on the workpiece.	+	m		

13.2.50 G Group 53: Frame rotation in relation to tool

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
TOROTOF	Frame rotation in relation to tool OFF	+	m	x	

TOROT	Align Z axis of the WCS by rotating the frame parallel to the tool orientation	+	m		
TOROTZ	As TOROT	+	m		
TOROTY	Align Y axis of the WCS by rotating the frame parallel to the tool orientation	+	m		
TOROTX	Align X axis of the WCS by rotating the frame parallel to the tool orientation	+	m		
TOFRAME	Align Z axis of the WCS by rotating the frame parallel to the tool orientation	+	m		
TOFRAMEZ	As TOFRAME	+	m		
TOFRAMEY	Align Y axis of the WCS by rotating the frame parallel to the tool orientation	+	m		
TOFRAMEX	Align X axis of the WCS by rotating the frame parallel to the tool orientation	+	m		

13.2.51 G Group 54: Vector rotation for polynomial programming

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
ORIROTA	Vector rotation absolute	+	m	x	
ORIROTR	Vector rotation relative	+	m		
ORIROTT	Vector rotation tangential	+	m		
ORIROTC	Tangential rotational vector in relation to path tangent	+	m		

13.2.52 G Group 55: Rapid traverse with/without linear interpolation

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
RTLION	Rapid traverse motion with linear interpolation ON	+	m	x	
RTLIOF	Rapid traverse motion with linear interpolation OFF Rapid traverse motion is achieved with single-axis interpolation.	+	m		

13.2.53 G Group 56: Taking into account tool wear

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
TOWSTD	Initial setting value for offsets in tool length	+	m	x	
TOWMCS	Wear values in the machine coordinate system (MCS)	+	m		
TOWWCS	Wear values in the workpiece coordinate system (WCS)	+	m		
TOWBCS	Wear values in the basic coordinate system (BCS)	+	m		
TOWTCS	Wear values in the tool coordinate system (toolholder ref. point T at the toolholder)	+	m		
TOWKCS	Wear values in the coordinate system of the tool head for kinetic transformation (differs from machine coordinate system through tool rotation)	+	m		

13.2.54 G Group 57: Corner deceleration

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH

FENDNORM	Corner deceleration OFF	+	m	x	
G62	Corner deceleration at inside corners when tool radius compensation is active (G41/G42)	+	m		
G621	Corner deceleration at all corners	+	m		

13.2.55 G Group 59: Dynamic response mode for path interpolation

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
DYNNORM	Standard dynamic response	+	m	x	
DYNPOS	Positioning mode, tapping	+	m		
DYNROUGH	Roughing	+	m		
DYNSEMIFIN	Rough finishing	+	m		
DYNFINISH	Finishing	+	m		
DYNPREC	Smooth finishing	+	m		

13.2.56 G Group 60: Working area limitation

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
WALCS0	Workpiece coordinate system working area limitation OFF	+	m	x	
WALCS1	WCS working area limitation group 1 active	+	m		
WALCS2	WCS working area limitation group 2 active	+	m		
WALCS3	WCS working area limitation group 3 active	+	m		
WALCS4	WCS working area limitation group 4 active	+	m		
WALCS5	WCS working area limitation group 5 active	+	m		
WALCS6	WCS working area limitation group 6 active	+	m		
WALCS7	WCS working area limitation group 7 active	+	m		
WALCS8	WCS working area limitation group 8 active	+	m		
WALCS9	WCS working area limitation group 9 active	+	m		
WALCS10	WCS working area limitation group 10 active	+	m		

13.2.57 G Group 61: Tool orientation smoothing

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
ORISOF	Tool orientation smoothing OFF	+	m	x	
ORISON	Tool orientation smoothing ON	+	m		

13.2.58 G Group 62: Repositioning mode for REPOS (non-modal)

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
RMBBL	Repositioning to start of block	-	S		
RMIBL	Repositioning to interrupt point	-	S	x	
RMEBL	Repositioning to end of block	-	S		
RMNBL	Repositioning to the nearest path point	-	S		

13.2.59 G Group 64: Grinding frames

G Command	Meaning	MD20150 ²⁾	W ³⁾	STD ⁴⁾	
				SAG	MH
GFRAME[0]	Grinding frame of the data management \$P_GFR[0] (null frame)	+	m	x	

GFRAME[1]	Grinding frame of the data management \$P_GFR[1]	+	m		
GFRAME[2]	Grinding frame of the data management \$P_GFR[2]	+	m		
GFRAME[3]	Grinding frame of the data management \$P_GFR[3]	+	m		
...	...	+	m		
GFRAME[100]	Grinding frame of the data management \$P_GFR[100]	+	m		

²⁾ Configurability of the G command as a reset setting for the G group on power up, reset or end of part program (with MD20150 \$MC_GCODE_RESET_VALUES):

- + Configurable
- Not configurable

³⁾ Effectiveness of the G Command

- m modal
- s non-modal

⁴⁾ Reset setting, see the following machine data:

- MD20149GCODE_RESET_S_VALUES (reset position of G groups (fix))
 - MD20150 \$MC_GCODE_RESET_VALUES (reset position of the G groups)
 - MD20151GCODE_RESET_S_MODE (reset behavior of G groups (fix))
 - MD20152 \$MC_GCODE_RESET_MODE (reset behavior of G groups)
 - MD20154 \$MC_EXTERN_GCODE_RESET_VALUES (reset position of the G groups in ISO mode)
 - MD20156 \$MC_EXTERN_GCODE_RESET_MODE (reset behavior of external G groups)
- SAG Default setting Siemens AG
MM Default setting Machine Manufacturer

13.3 List of recognized ProtoTRAK M-codes for Siemens

M Code	Description
M00	Program Stop
M01	Option Stop
M02	End of program
M03	Spindle CW start
M04	Spindle CCW start
M05	Spindle Stop
M06	Tool Change
M07	Air Coolant ON
M08	Flood Coolant ON
M09	Coolant OFF
M17	Fed of subroutine
M19	Spindle oriented
M22	Automation Request 1 ON
M23	Automation Request 1 OFF
M24	Automation Request 2 ON
M25	Automation Request 2 OFF
M26	Automation Request 3 ON
M27	Automation Request 3 OFF
M28	Automation Request 4 ON
M29	Automation Request 4 OFF
M30	End of Program
M48	Coolant through spindle ON
M49	Coolant through spindle OFF
M50	Coolant through spindle ON from tool table
M55	Wash Down Coolant ON
M56	Wash Down Coolant OFF
M57	Chip Conveyor Start
M58	Chip Conveyor Stop
M59	Skimmer ON
M60	Skimmer OFF
M64	Activate probing cycle work area limits
M65	Activate the default work area limits
M66	Reset condition at the end of the program (Set the Z to machine zero and put the first
M67	Activate probe automatically when it is in the spindle
M68	Deactivate probe
M69	Door Open
M71	Work Light ON
M72	Work Light OFF

TRAK Machine Tools
Southwestern Industries, Inc

TRAK Warranty Policy

Warranty

TRAK products are warranted to the original purchaser to be free from defects in workmanship and materials for the following periods:

Product	Warranty Period
New TRAK/ProtoTRAK	1 Year
New Siemens CNC	1 Year

The warranty period starts on the date of the invoice to the original purchaser from Southwestern Industries, Inc. (SWI) or their authorized distributor.

If a product, subsystem or component proves to be defective in workmanship and fails within the warranty period, it will be repaired or exchanged at our option for a properly functioning unit in similar or better condition. Such repairs or exchanges will be made FOB Factory/Los Angeles or the location of our nearest factory representative or authorized distributor.

Warranty Disclaimers

- This warranty is expressly in lieu of any other warranties, express or implied, including any implied warranty of merchantability or fitness for a particular purpose, and of any other obligations or liability on the part of TMT/SWI (or any producing entity, if different).
- Warranty repairs/exchanges do not cover incidental costs such as installation, labor, freight, etc.
- TMT/SWI is not responsible for consequential damages from use or misuse of any of its products.
- TRAK products are precision mechanical/electromechanical/electronic systems and must be given the reasonable care that these types of products require. Evidence that the product does not receive adequate Preventative Maintenance may invalidate the warranty. Excessive chips built up around ballscrews and way surfaces is an example of this evidence.
- Accidental damage, beyond the control of TMT/SWI, is not covered by the warranty. Thus, the warranty does not apply if a product has been abused, dropped, hit, or disassembled.
- Improper installation by or at the direction of the customer in such a way that the product consequently fails, is considered to be beyond the control of the manufacturer and outside the scope of the warranty.
- Warranty does not cover wear items that are consumed under normal use of the product. These items include, but are not limited to: windows, bellows, wipers, filters, drawbars and belts.

F10302-2 rev 012822