



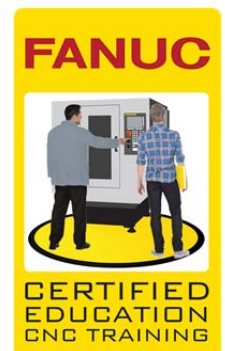
White Paper

Document No. MWA-017-EN_05_1312

December 2013

NCGuide Academic packages

Authentic FANUC CNC software on a PC
for the most effective learning environment



1 Introduction	5
1.1 NCGuide Academic packages	6
1.2 FANUC CNCs supported	6
2 Academic Applications	7
2.1 Instructor demonstration	7
2.2 Classroom exercises	7
2.3 Students homework	8
2.4 Flipping the classroom	8
2.5 Distance learning	8
2.6 CAD/CAM integration	8
2.7 Operation training	8
2.8 Part programming training	9
2.9 CNC Certification Cart	9
3 Supporting the CNC workflow	10
3.1 Creating part programs	10
3.2 Loading part programs	10
3.3 Tool setup	11
3.4 Graphical part program verification	11
3.5 Editing part programs	12
3.6 Work offsets	12
3.7 Run program	12
3.8 Tool wear offsets	12
4 CNCs, displays, MDI panels and options	13
4.1 FANUC CNC models supported	13
4.2 FANUC CNC displays supported	13
4.2.1 Changing the CNC display	14
4.2.2 Changing the display mode	14
4.3 FANUC MDI key panels supported	14
4.3.1 Changing the MDI key panel	14
4.4 Examples of CNC display and MDI panel combinations	15
4.4.1 Series 0i-D lathe, 8.4" display, picture mode	15
4.4.2 Series 0i-D mill, 10.4" display, picture mode	16
4.4.3 Series 31i MODEL B mill, 10.4" display, picture mode	17
4.4.4 Series 0i-D mill, 10.4" display, window mode	18
4.5 List of options	19
4.6 Lathe G-codes	20
4.7 Mill G-codes	25

5 Operation	30
5.1 Starting NCGuide	30
5.2 Components	31
5.2.1 0i-D components	31
5.2.2 30i, 31i and 35i components	32
5.3 CNC Mode selection	33
5.3.1 EDIT - Program editing mode	33
5.3.2 MEM – Memory operation	33
5.3.3 MDI operation mode	33
5.4 MDI key operation	34
5.4.1 MDI keys	35
5.5 Soft key operation	36
5.6 Mini operator panel operation	39
5.7 Screens display	41
5.7.1 Position screen	41
5.7.2 Program screen	42
5.7.3 Setting screen	43
5.7.4 Parameter screen	44
5.7.5 Message screen	45
5.7.6 Graphic screen	46
5.8 Editing part programs	47
5.8.1 Creating part programs	47
5.8.2 Altering a word	47
5.8.3 Inserting a word	48
5.8.4 Deleting a word	48
5.9 Selecting part programs	48
5.9.1 Selection using program call operation	48
5.9.2 Selection using the part program list	48
5.10 Deleting part programs	49
5.10.1 Deletion using program call operation	49
5.10.2 Deletion using the part program list	49
5.11 Part program execution	49
5.11.1 Part program execution	49
5.11.2 Program graphic function check	50
5.11.3 Single block execution	50
5.12 Multi-language display	50
5.12.1 Language switch procedure	51
5.13 Exiting NCGuide	51

6 MANUAL GUIDE <i>i</i>	52
6.1 What is MANUAL GUIDE <i>i</i>	52
6.2 Academic applications of MANUAL GUIDE <i>i</i>	52
6.3 Main Features of MANUAL GUIDE <i>i</i>	53
6.4 Navigation to MANUAL GUIDE <i>i</i>	54
6.4.1 Starting MANUAL GUIDE <i>i</i>	54
6.4.2 MANUAL GUIDE <i>i</i> screen components	55
6.5 Overview of creating a MANUAL GUIDE <i>i</i> part program	56
6.6 Notes on creating part programs	57
6.6.1 Commands required before machining cycle input	57
6.6.2 Machining cycles	58
7 Machine compositions	59
7.1 Machine Composition setting tool	59
7.2 Option Setting	60
7.3 Parameters	60
8 NCGuide Academic packages	61
8.1 System requirements	61
8.2 Differences from the CNC in operation	62
9 Summary	62

1 Introduction

NCGuide Academic packages are FANUC CNC software running on a PC, providing a 100% authentic operation and part programming environment at a fraction of the cost of using a hardware simulator or a production machine tool. Comprehension and retention is enhanced as students perform repetitive hands-on exercises in an ergonomically friendly environment.

Students can practice common procedures and experiment safely with minimum supervision and without risks to people, tooling or machines.

NCGuide Academic packages provide access to the latest CNC technology and support multiple control configurations for both turning and milling.

NCGuide Academic packages improves the effectiveness of CNC instruction through:

- Instructor demonstrations in the classroom and online for distance learning
- Classroom exercises that provide a superior hands on experience with minimum supervision
- Homework that can be completed anytime and anywhere
- Distance learning that is supported through both online instructor demonstrations or to complement online video instruction, and independent student exercises
- Enhanced CAD/CAM generated part program testing
- Comprehensive operation and part programming support, including Custom Macro programming

NCGuide Academic packages support the complete CNC programming, setup operation and part program execution workflow:

- Create part programs on the CNC with conventional G-code editors or using the MANUAL GUIDE *i* conversational programming
- Upload part programs created externally with text editors or CAD/CAM systems
- Complete tool setup for tool geometry
- Verify the part program using 3D solid model animation or tool path plotting visualizations and using the part program check screen
- Part programs can be modified using the G-code editor
- Work offsets can be established
- Part programs can be executed at the programmed feed rate, or in accelerated simulation rates
- Tool offsets can be adjusted

1.1 NCGuide Academic packages

NCGuide Academic package for Classroom provides network licensing. Up to 16 concurrent students can use the simulation software at the same time, from a larger pool of potential users. Each individual student can use any of the controls and configurations supported. As long as students can access the network within the campus or via the Internet they can run the simulator on any PC that has the software installed. This provides flexibility for students and staff for practical exercises, homework and distance learning.

NCGuide Academic package for Homework is value priced for an individual student, or for an instructor to test drive the software before committing to the Classroom package. The software is identical to the classroom package, except the licensing is provided with a plug-in USB key that is only valid for one year. The price of the package makes it suitable for including in a year-long certificate program.

1.2 FANUC CNCs supported

The current versions of NCGuide Academic packages support:

- FANUC Series 0i-D (0i-TD and 0i-MD)
- FANUC Series 31i-B
- FANUC Series 30i-B
- FANUC Series 31i-A
- FANUC Series 31i-LB (laser cutting CNC)
- FANUC Series 31i-PB (punch press CNC)
- FANUC Series 35i-B

2 Academic Applications

2.1 Instructor demonstration

NCGuide Academic packages can be projected to a large classroom display just like any other PC-based program, providing an effective visual aid for demonstrating concepts and examples during lectures.

The instructor can share his screen to remote students using suitable software for instructor-led distance learning applications. Student may also be able to control the presenting computer for interactive exercises.

All aspects of CNC operation and part programming can be demonstrated including:

- Milling and turning applications
- Creating a part programming on the CNC
- Loading a program into the control from an external source
- Tool geometry setup and adjustment
- Graphical part program verification for syntax and sequence of operation errors
- Part program editing
- Work offset setting and adjustment
- Part program execution
- Adjusting tool wear offsets



2.2 Classroom exercises

Students can perform operation and programming exercises in the classroom. This is especially useful if, as is typical, there are insufficient machines for all the students to use at the same time. The instructor can focus on monitoring students using machine tools to ensure safe operation, while the remaining students can use NCGuide to pre-test part programs or to learn additional concepts.

The instructor can provide a specific machine composition for the exercise, with the CNC type, options, parameter settings, existing part programs and subprograms, tool and work offsets all preselected.



2.3 Students homework

Students that can access the schools network remotely can also run NCGuide. The software is loaded on the student's computer and only the licensing is controlled via the network. This means the student gets the performance benefits of a local application and the network is not loaded with large data transfers.

An instructor can provide a customized machine composition for exercises, with the CNC type, options, parameter settings, existing part programs and subprograms, tool and work offsets all pre-configured. Problems can be designed for the student to solve.



2.4 Flipping the classroom

NCGuide Academic packages are ideal when lectures and standard exercises are performed with minimal supervision, reserving the primary instructor interactions for in-class homework assignments.

2.5 Distance learning

NCGuide is an effective distance learning tool that allows the user to perform comprehensive exercises in CNC operation and programming, complimenting existing distant learning systems. Licensing is controlled via the network so the student gets the performance benefits of a local application and the network is not taxed with large data transfers. An instructor can also demonstrate the application across the web for instructor-led distance learning applications.

2.6 CAD/CAM integration

NCGuide a valuable tool in CAD/CAM class. The first execution of the part program after completing the CAM post processing can be performed on NCGuide. Conventional tool path plotting graphics or the 3D solid model animation can be used to check syntax and tool paths in a safe, low-cost environment.

2.7 Operation training

NCGuide is ideal for operational training. All standard CNC operational screens can be selected and all standard procedures can be practiced. You can:

- create and edit part programs
- search for words and safe start blocks
- upload and download part programs
- test for syntax and tool path geometry errors
- edit and visualize workpiece coordinate offsets
- edit and visualize tool geometry and tool wear offsets

Users can expand their knowledge by learning the features available with newer controls - even if the control is not available in the workshop. Academic students can get hands on experience with the latest controls to enhance their value to hiring companies, and workers already in industry can build experience prior to new machine purchases.

For the most realistic and effective learning environment, users can setup machine compositions to emulate a particular machine's CNC.

2.8 Part programming training

NCGuide supports both conventional G-code part programming and the easy-to-use, yet powerful MANUAL GUIDE *i* conversational part programming. You can:

- create and edit machining center, lathe and compound machining part programs
- visualize conventional part programs with tool path simulation
- visualize part programs with 3D solid model animation or tool path plotting
- generate cycle time estimates
- create and test Custom Macro subroutines
- visualize the effect of workpiece and tool offsets
- visualize the details of canned cycles and advanced interpolation modes
- test CAD/CAM programs prior to sending to a machine

Manual Guide *i* conversational part programs can be developed on the simulator and then converted to conventional G-code to run on any FANUC CNC.

2.9 CNC Certification Cart

NCGuide can be used with the Levil CNC Certification Cart to bring complete programming and setup instruction into the classroom. Students can create, test and debug part programs using NCGuide, save the program to a USB memory port or network drive directory and then load the programs into the CNC Certification Cart. All setup including tool geometry and workpiece coordinate system offsets can be set and a part produced. By allowing the complete programming and setup process to be experienced in the classroom means that students are more efficient and effective when using workshop machines.

Schools without workshop resources can use NCGuide and the CNC Certification Cart to provide an effective introduction to CNC machining, preparing them for future courses at college.



3 Supporting the CNC workflow

3.1 Creating part programs

Part programs can be created on the CNCs using the standard G-code editors or using MANUAL GUIDE *i* conversational programming.

The FANUC traditional G-code word editor and the more contemporary character editor are supported.

MANUAL GUIDE *i* conversational programming simplifies the generation of a part programming and focuses the student on the sequence of operations required rather than on the detailed G-code.

MANUAL GUIDE *i* also provides fixed sentence programming to generate multiple lines of G-code with just a few keystrokes. Fixed sentences can be established for sequence of operations such as program starting blocks, tool change blocks, material feeds and speeds and part program end blocks.

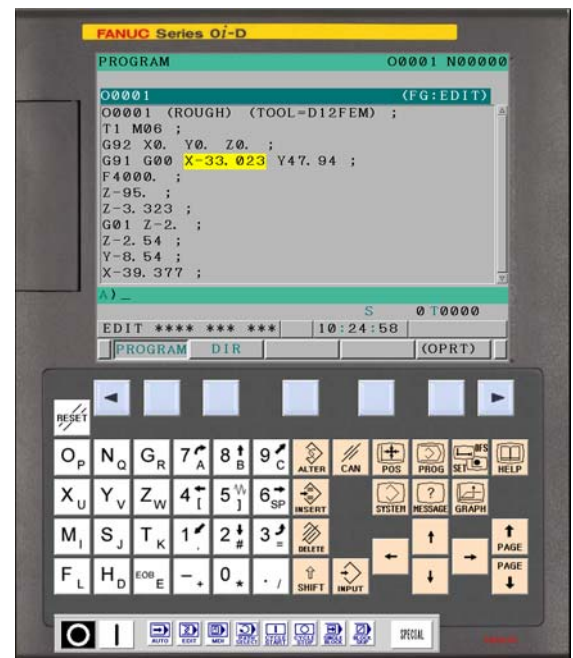
NCGuide Academic packages support a wide range of G-codes from the basic through to advanced concepts including Custom Macros.

3.2 Loading part programs

Part programs can be created externally using a PC-based editor or by a CAD/CAM post-processor and input into the control using standard CNC screens.

Part programs are loaded using the memory card interface. The virtual memory card may be located on a USB memory stick for student convenience, or on any directory on the PC or a network drive.

Instructors can provide sample part programs to be uploaded into the CNC. The part programs can include syntax or sequence of operations errors, for the student to test and debug.



3.3 Tool setup

Setting the tool geometry data accurately is a critical step in machine setup. The correct tools must be loaded into the correct tools stations to correspond with the part program assignments (or the part program must be edited).

For milling machines the correct tool length and diameter must be entered into the tool offset tables. NCGuide supports Tool Offset Memory C, displaying tool data and tool wear offsets for length and diameter in separate column, eliminating the need for error prone math calculations. Legacy tool offset systems are also supported.

For turning machines, the correct tool geometry data for X and Z must be entered along with the appropriate tool nose radius and tip offset.

When using MANUAL GUIDE *i*, additional data is required to describe the type and set of the tool to support the solid model machining simulation.

3.4 Graphical part program verification

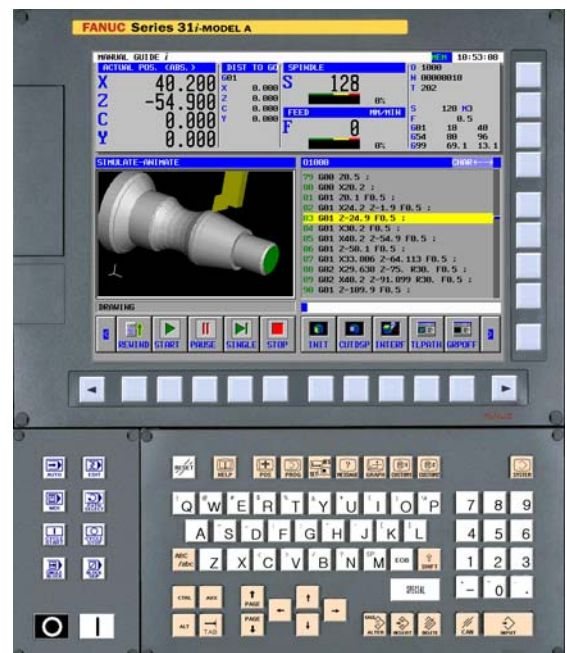
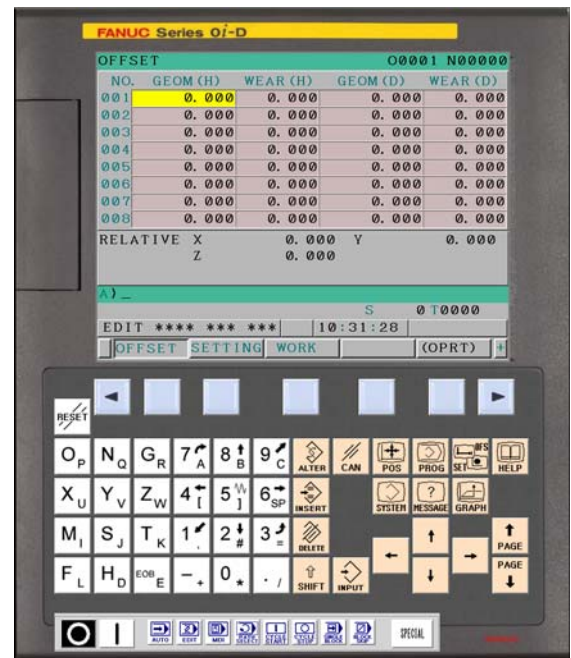
Before executing a part program with the material in place, it is usual to check it for syntax problems and ensure the correct order of operations.

Verification is best achieved using the graphics display. Both the standard GRAPH display and the MANUAL GUIDE *i* solid model animation are supported in NCGuide Academic packages.

Solid model animation is superior for several reasons:

- Easier to visualize tooling and machining operations
- Simulation is performed without moving axes
- 3D tool path plotting is also selectable

Conventional part programs can be simulated using the 3D solid model animation. A block must be added at the beginning of the part program to describe the material blank, the tool types must be defined in the tool data table, and a D-code must be specified at each tool change.



Advanced screens available with the FANUC Series 31i-MODEL B control simplify the testing and debugging of Custom Macro programs by displaying both the macro statements and the result values simultaneously.

3.5 Editing part programs

Part programs can be edited on the CNC using the standard part program editors or in the MANUAL GUIDE *i* environment. Students can practice all editing operations from the simple alter or delete commands through to advanced search and replace commands.

3.6 Work offsets

Work offsets can be set to shift the working coordinate system to the part program zero point.

For machining centers, G54 through G59 and G54 P1 through G54 P48 are available.

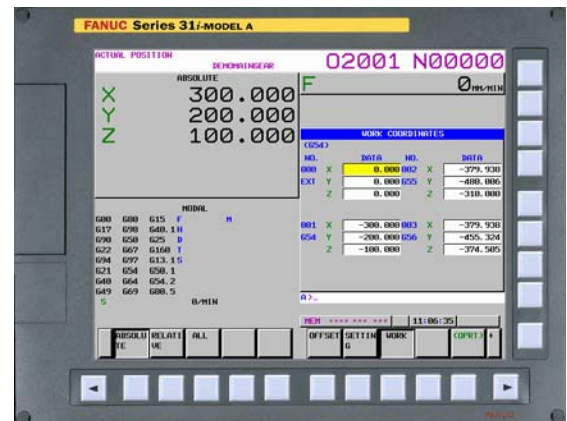
For turning, G54 through G59 and the external work shift is available.

3.7 Run program

The part program can be executed at the programmed feed rate. The program check screen displays the part program and the active positions and model G-codes simultaneously.

3.8 Tool wear offsets

Though NCGuide cannot simulate actual tool wear, the tool wear offset tables can be updated and their effect on axis positions can be demonstrated.



4 CNCs, displays, MDI panels and options

NCGuide Academic packages support a wide range of FANUC CNC models, displays and MDI panels.

4.1 FANUC CNC models supported

NCGuide Academic packages support the following FANUC CNC models:

- FANUC Series 0i MODEL D (lathe and mill)
- FANUC Series 31i MODEL A (lathe and mill)
- FANUC Series 31i MODEL B (lathe, mill, laser and punching)
- FANUC Series 30i MODEL B (lathe and mill up to 10 axes)
- FANUC Series 35i MODEL B

Multiple machine compositions can be created for each model with the following maximum machine configuration:

Maximum controlled axes	4 axes
Maximum controlled paths	1 path
Maximum part program storage size	32 KB
Maximum number of registered part programs	64

4.2 FANUC CNC displays supported

NCGuide Academic packages support the following FANUC displays:

Display	Models	Optimum PC Screen Resolution
8.4"	All	1280 x 1200
10.4"	All	1280 x 1200
15"	Series 30i/31i MODEL A/B	2048 x 1536

If NCGuide does not fit the screen in picture mode due to a low display resolution, scroll bars are displayed. Alternatively, NCGuide can use Window mode to display the CNC screen, MDI panel and mini operator's panels in separate windows. The PC keyboard can be used for most MDI inputs. Window mode and PC keyboard input may be preferred for laptop computers.

4.2.1 Changing the CNC display

The display size can be changed quickly by selecting from the choices listed on the Display Size menu or by using the Setting Management tool. In Picture Mode, the Display Size menu is displayed by right-clicking on the CNC control. In Window Mode, the Display Size menu is accessible from the View menu. After changing the display size, you must exit and restart NCGuide to activate the new screen size.

4.2.2 Changing the display mode

The display mode can be changed quickly by selecting Window Mode or Picture Mode on the Display Mode menu or by using the Setting Management tool. In Picture mode, the Display Mode menu is displayed by right-clicking on the CNC control. In Window mode, the Display Mode menu is accessible from the View menu. After changing the display mode, you must exit and restart NCGuide to activate the new screen mode.

4.3 FANUC MDI key panels supported

NCGuide Academic packages support the following FANUC MDI key panels:

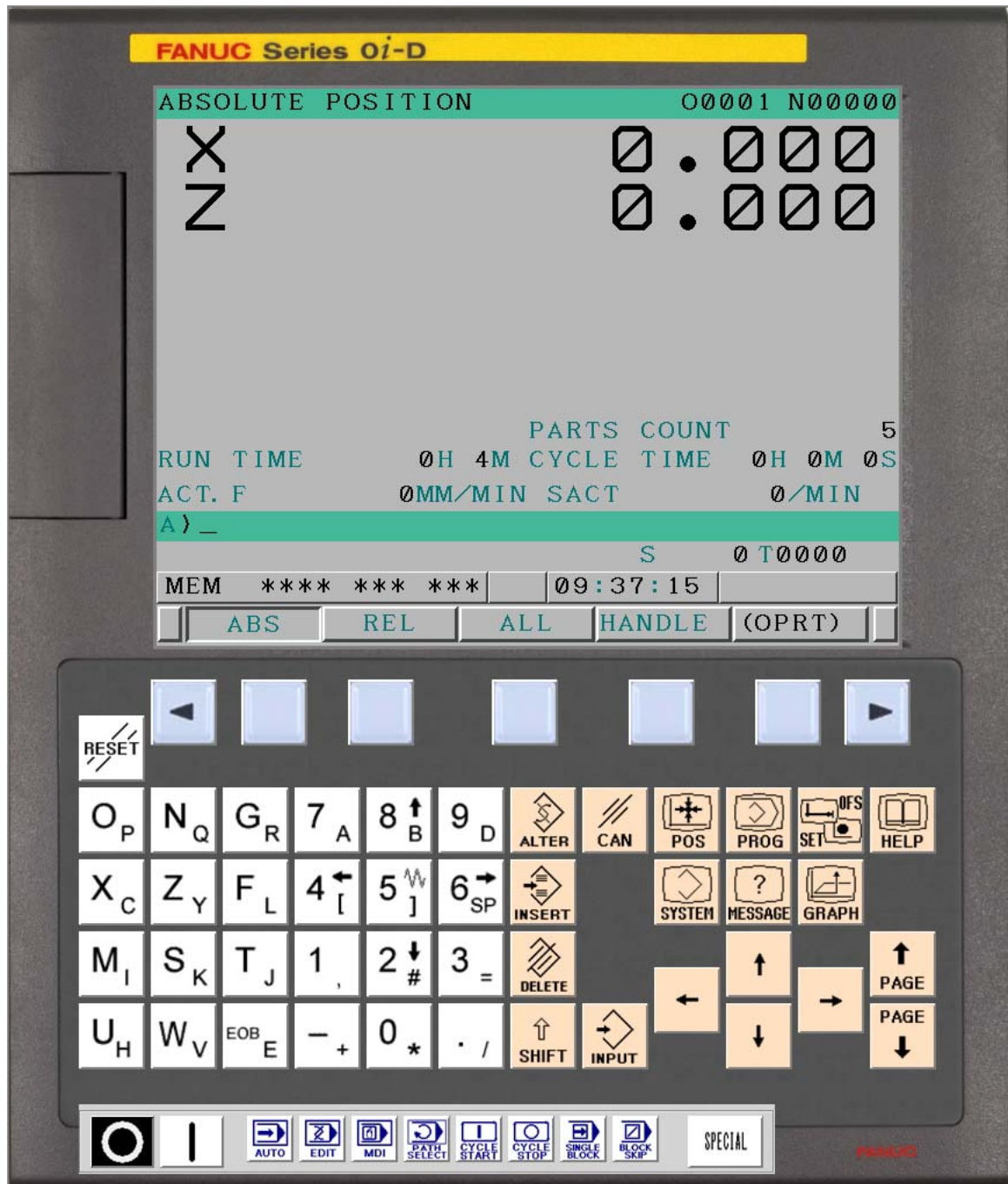
MDI Panel	Models
ONG 'T'	All
ONG 'M'	All
QWERTY	Series 30i/31i MODEL A/B

4.3.1 Changing the MDI key panel

The MDI key panel can be changed quickly by selecting from the choices listed on the Display Size menu or by using the Setting Management tool. In Picture Mode, the MDI Key menu is displayed by right-clicking on the CNC control. In Window Mode, the MDI menu is accessible from the View menu. Note that in Window mode you can chose not to display an MDI key panel if you intend to provide all input though the PC keyboard. After changing the MDI key, the new MDI key panel is displayed.

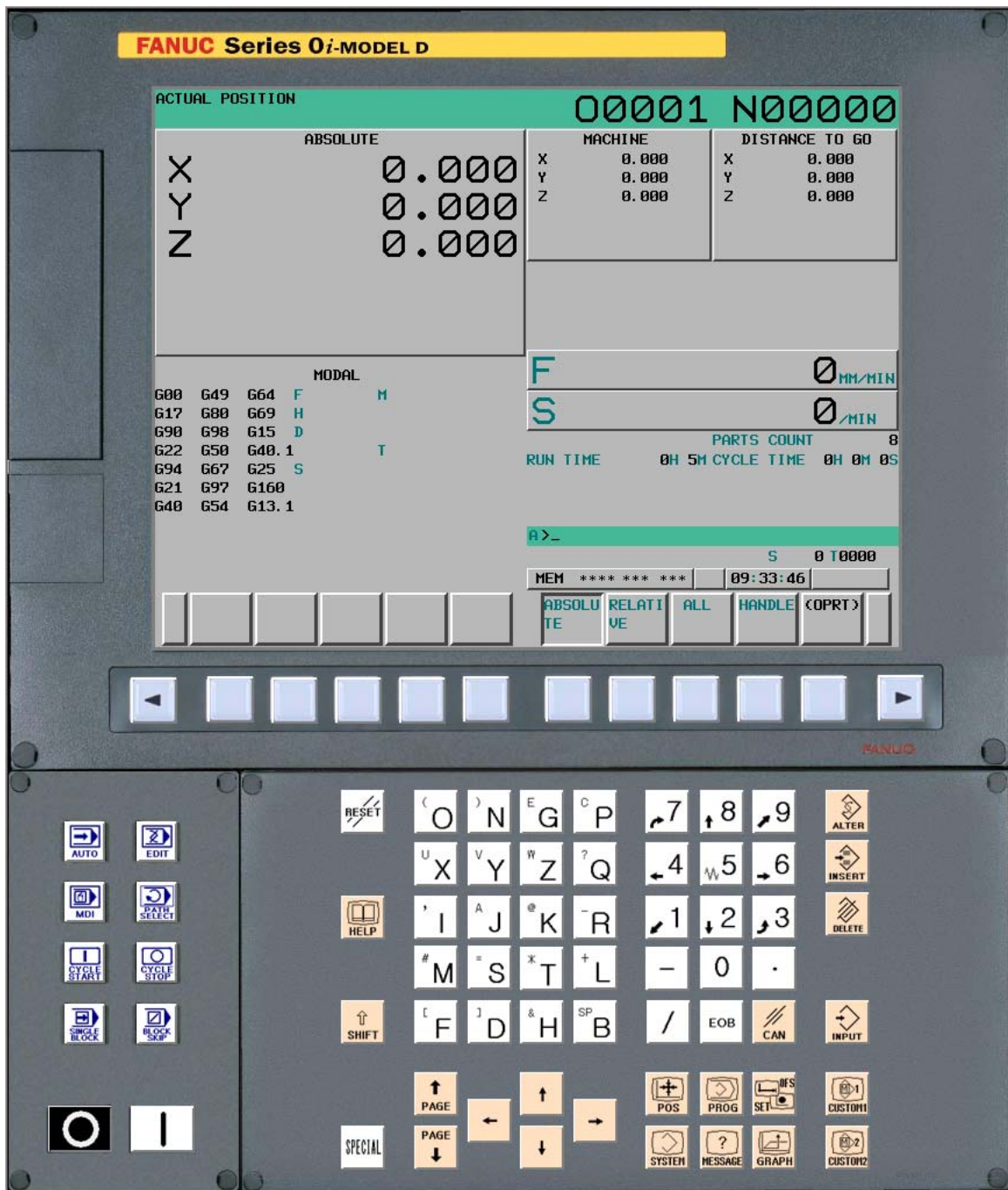
4.4 Examples of CNC display and MDI panel combinations

4.4.1 Series 0i-D lathe, 8.4" display, picture mode

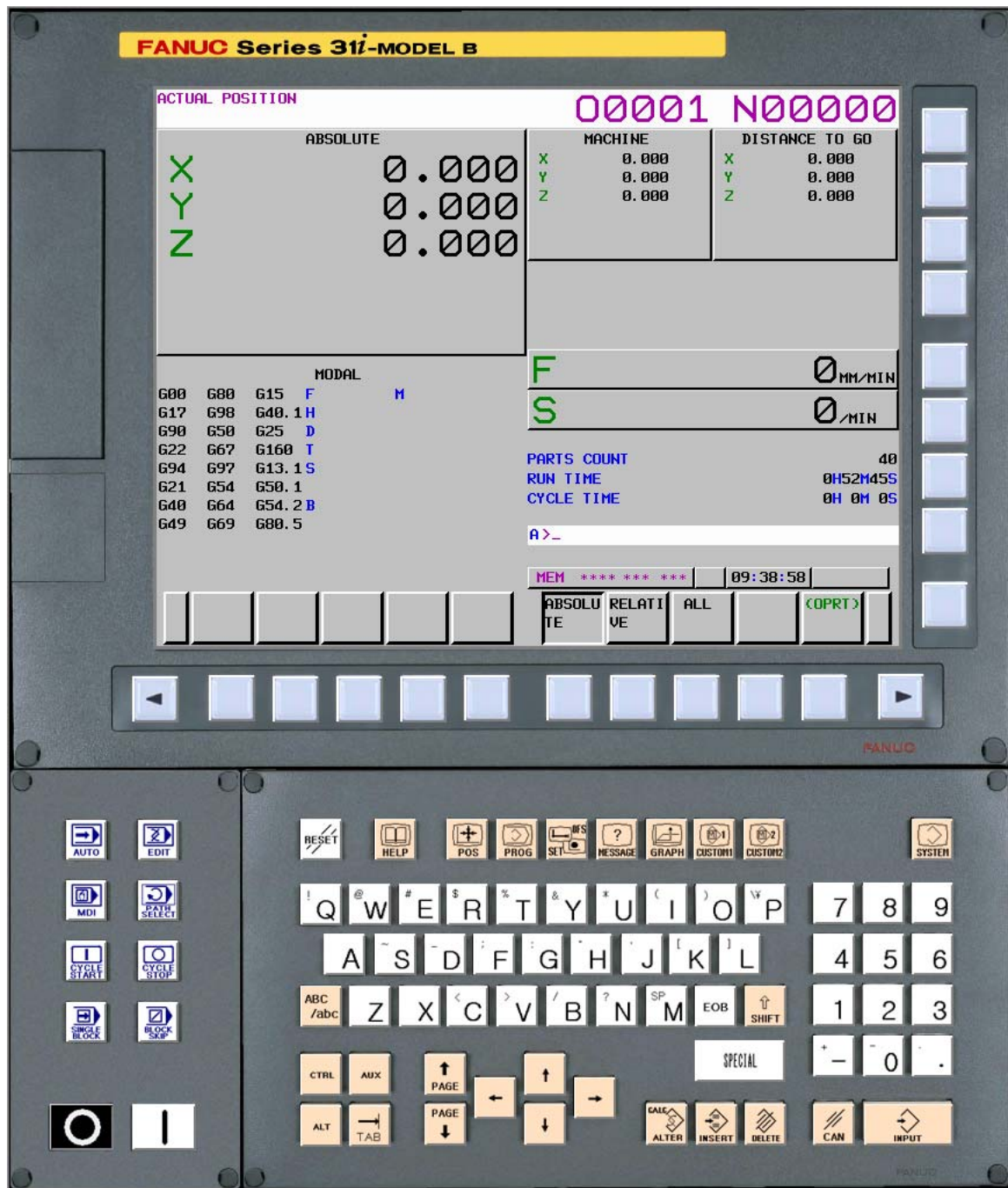


0i-D CNC, 8.4" display, ONG 'T' MDI panel in lathe mode

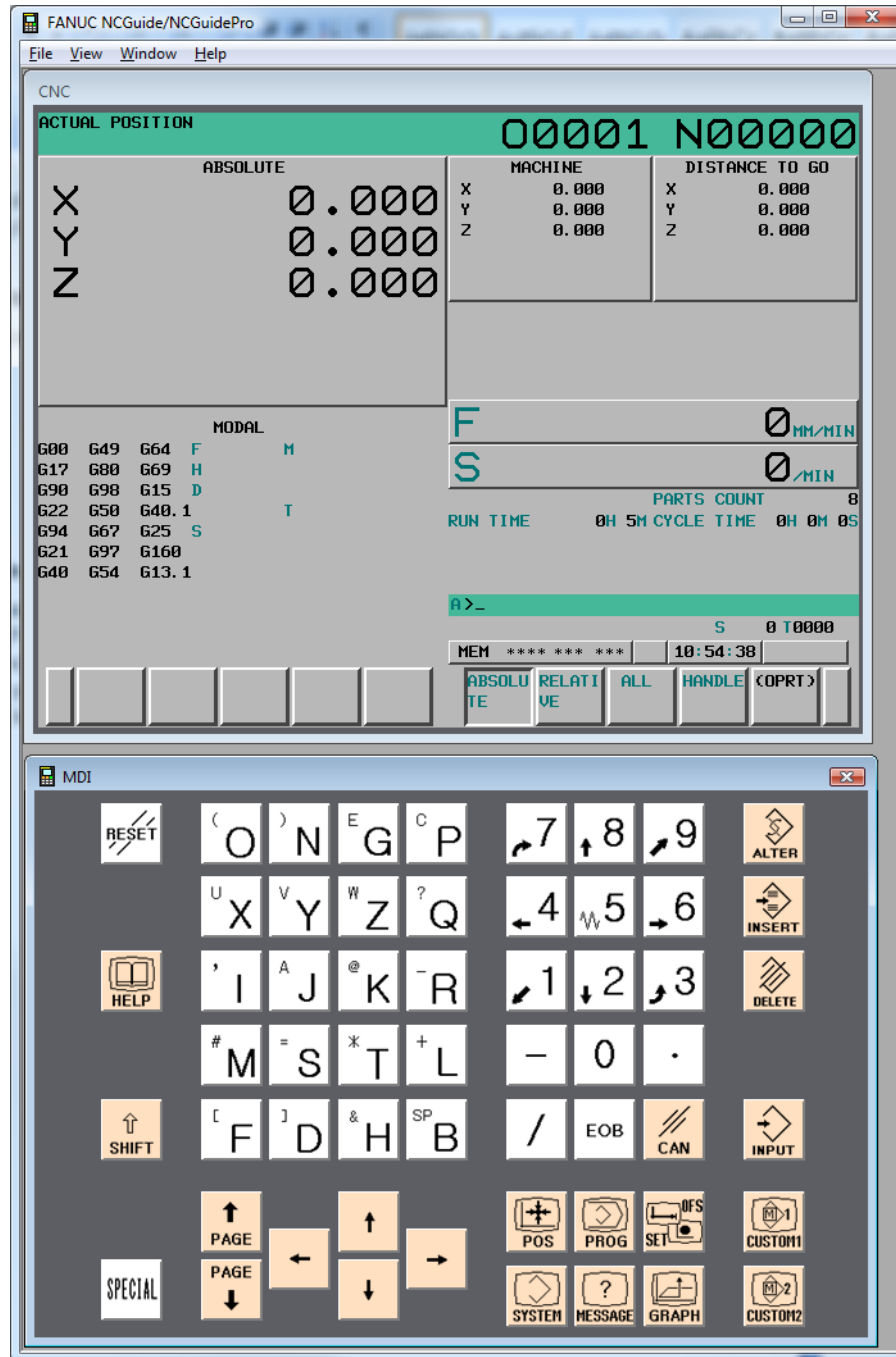
4.4.2 Series 0i-D mill, 10.4" display, picture mode



0i-D CNC, 10.4" display, ONG 'M' MDI panel in machining mode



4.4.4 Series 0i-D mill, 10.4" display, window mode



0i MODEL D CNC, window mode

4.5 List of options

Feature Description	0i-MODEL D		31i-MODEL A/B	
	Machining	Lathe	Machining	Lathe
INCH/METRIC SELECTION	●	●	●	●
SEQUENCE NUMBER COMPARISON & STOP	●	●	●	●
INCREMENTAL SYSTEM C	●	●	●	●
STORED STROKE LIMIT 2ND/3RD	●	●	●	●
PRE-TRAVEL STROKE LIMIT CHECK	●	●	●	●
CHUCK/TAILOSTOCK BARRIER		●		●
PROGRAM RESTART	●	●	●	●
EXPONENTIAL FUNCTION INTERPOLATION			●	●
POLAR COORDINATE INTERPOLATION		●	●	●
CYLINDRICAL INTERPOLATION	●	●	●	●
HELICAL INTERPOLATION	●		●	●
INVOLUTE INTERPOLATION			●	●
CONICAL/SPIRAL INTERPOLATION			●	●
F1-DIGIT FEED	●		●	
INVERSE TIME FEED	●		●	
NORMAL DIRECTION CONTROL	●		●	
POLAR COORDINATE COMMAND	●		●	
SINGLE DIRECTION POSITIONING	●		●	
3RD/4TH REFERENCE RETURN	●	●	●	●
WORKPIECE COORDINATE SYSTEM	●	●	●	●
WORKPIECE COORDINATE SYSTEM PRESET	●	●	●	●
ADDITION OF WORKPIECE COORDINATE SYSTEM 48-PAIRS	●		●	●
DIRECT DRAWING DIMENSION		●		●
G CODE SYSTEM B/C		●		●
CHAMFERING/CORNER R		●		●
OPTIONAL ANGLE CHAMFERING/CORNER R	●		●	
CUSTOM MACRO	●	●	●	●
ADDITIONAL COMMON MACRO VARIABLES	●	●	●	●
PATTERN DATA INPUT	●	●	●	●
MULTIPLE REPETITIVE CYCLES		●		●
MULTIPLE REPETITIVE CYCLES II		●		●
CANNED CYCLES FOR DRILLING	●	●	●	●
SCALING	●		●	●
AUTOMATIC CORNER OVERRIDE	●		●	
COORDINATE SYSTEM ROTATION	●		●	●
PROGRAMMABLE MIRROR IMAGE	●		●	●
FIGURE COPY			●	
2ND AUXILIARY FUNCTION	●	●	●	●
TOOL OFFSET MEMORY 400 PAIRS	●		●	
TOOL OFFSET MEMORY C	●		●	
Y AXIS OFFSET		●		●
TOOL OFFSET	●		●	
TOOL RADIUS/TOOL NOSE RADIUS COMPENSATION	●	●	●	●
TOOL GEOMETRY/WEAR OFFSET		●		●
TOOL LIFE MANAGEMENT	●	●	●	●
BACKGROUND EDITING	●	●	●	●
MACHINE TIME STAMP			●	●
RUN HOUR AND PARTS COUNT	●	●	●	●
GRAPHIC DISPLAY	●	●	●	●
BACKGROUND EDITING	●	●	●	●
MANUAL GUIDE I	●	●	●	●

4.6 Lathe G-codes

G code system			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW or helical interpolation CW
G03	G03	G03		Circular interpolation CCW or helical interpolation CCW
G02.2	G02.2	G02.2		Involute interpolation CW
G02.3	G02.3	G02.3		Exponential interpolation CW
G02.4	G02.4	G02.4		3-dimensional coordinate system conversion CW
G03.2	G03.2	G03.2		Involute interpolation CCW
G03.3	G03.3	G03.3		Exponential interpolation CCW
G03.4	G03.4	G03.4		3-dimensional coordinate system conversion CCW
G04	G04	G04	00	Dwell
G05	G05	G05		AI contour control (command compatible with high precision contour control), High-speed cycle machining, High-speed binary program operation
G05.1	G05.1	G05.1		AI contour control / Nano smoothing / Smooth interpolation
G05.4	G05.4	G05.4		HRV3, 4 on/off
G06.2	G06.2	G06.2	01	NURBS interpolation
G07	G07	G07	00	Hypothetical axis interpolation
G07.1 (G107)	G07.1 (G107)	G07.1 (G107)		Cylindrical interpolation
G08	G08	G08		Advanced preview control
G09	G09	G09		Exact stop
G10	G10	G10		Programmable data input
G10.6	G10.6	G10.6		Tool retract and recover
G10.9	G10.9	G10.9		Programmable switching of diameter/radius specification
G11	G11	G11		Programmable data input mode cancel
G12.1 (G112)	G12.1 (G112)	G12.1 (G112)	21	Polar coordinate interpolation mode
G13.1 (G113)	G13.1 (G113)	G13.1 (G113)		Polar coordinate interpolation cancel mode
G17	G17	G17	16	XpYp plane selection
G18	G18	G18		ZpXp plane selection
G19	G19	G19		YpZp plane selection
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G22	G22	G22	09	Stored stroke check function on
G23	G23	G23		Stored stroke check function off
G25	G25	G25	08	Spindle speed fluctuation detection off
G26	G26	G26		Spindle speed fluctuation detection on

G code system			Group	Function
A	B	C		
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G28.2	G28.2	G28.2		In-position check disable reference position return
G29	G29	G29		Movement from reference position
G30	G30	G30		2nd, 3rd and 4th reference position return
G30.2	G30.2	G30.2		In-position check disable 2nd, 3rd, or 4th reference position return
G31	G31	G31		Skip function
G31.8	G31.8	G31.		EGB-axis skip
G32	G33	G33	01	Threading
G34	G34	G34		Variable lead threading
G35	G35	G35		Circular threading CW
G36	G36	G36		Circular threading CCW (When bit 3 (G36) of parameter No. 3405 is set to 1) or Automatic tool offset (X axis) (When bit 3 (G36) of parameter No. 3405 is set to 0)
G37	G37	G37		Automatic tool offset (Z axis) (When bit 3 (G36) of parameter No. 3405 is set to 0)
G37.1	G37.1	G37.1		Automatic tool offset (X axis) (When bit 3 (G36) of parameter No. 3405 is set to 1)
G37.2	G37.2	G37.2		Automatic tool offset (Z axis) (When bit 3 (G36) of parameter No. 3405 is set to 1)
G38	G38	G38		Tool radius/tool nose radius compensation: with vector held
G39	G39	G39		Tool radius/tool nose radius compensation: corner rounding interpolation
G40	G40	G40	07	Tool radius/tool nose radius compensation : cancel
G41	G41	G41		Tool radius/tool nose radius compensation : left
G42	G42	G42		Tool radius/tool nose radius compensation : right
G41.2	G41.2	G41.2		3-dimensional cutter compensation : left (type 1)
G41.3	G41.3	G41.3		3-dimensional cutter compensation : (leading edge offset)
G41.4	G41.4	G41.4		3-dimensional cutter compensation : left (type 1) (FS16i-compatible command)
G41.5	G41.5	G41.5		3-dimensional cutter compensation : left (type 1) (FS16i-compatible command)
G41.6	G41.6	G41.6		3-dimensional cutter compensation : left (type 2)
G42.2	G42.2	G42.2		3-dimensional cutter compensation : right (type 1)
G42.4	G42.4	G42.4		3-dimensional cutter compensation : right (type 1) (FS16i-compatible command)
G42.5	G42.5	G42.5		3-dimensional cutter compensation : right (type 1) (FS16i-compatible command)
G42.6	G42.6	G42.6		3-dimensional cutter compensation : right (type 2)
G40.1	G40.1	G40.1	19	Normal direction control cancel mode
G41.1	G41.1	G41.1		Normal direction control left on
G42.1	G42.1	G42.1		Normal direction control right on

G code system			Group	Function
A	B	C		
G43	G43	G43	23	Tool length compensation + (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G44	G44	G44		Tool length compensation - (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G43.1	G43.1	G43.1		Tool length compensation in tool axis direction (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G43.4	G43.4	G43.4		Tool center point control (type 1) (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G43.5	G43.5	G43.5		Tool center point control (type 2) (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G43.7 (G44.7)	G43.7 (G44.7)	G43.7 (G44.7)		Tool offset (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G49 (G49.1)	G49 (G49.1)	G49 (G49.1)		Tool length compensation cancel (Bit 3 (TCT) of parameter No. 5040 must be "1".)
G50	G92	G92	00	Coordinate system setting or max spindle speed clamp
G50.3	G92.1	G92.1		Workpiece coordinate system preset
-	G50	G50	18	Scaling cancel
-	G51	G51		Scaling
G50.1	G50.1	G50.1	22	Programmable mirror image cancel
G51.1	G51.1	G51.1		Programmable mirror image
G50.2 (G250)	G50.2 (G250)	G50.2 (G250)	20	Polygon turning cancel
G51.2 (G251)	G51.2 (G251)	G51.2 (G251)		Polygon turning
G50.4	G50.4	G50.4	00	Cancel synchronous control
G50.5	G50.5	G50.5		Cancel composite control
G50.6	G50.6	G50.6		Cancel superimposed control
G51.4	G51.4	G51.4		Start synchronous control
G51.5	G51.5	G51.5		Start composite control
G51.6	G51.6	G51.6		Start superimposed control
G52	G52	G52		Local coordinate system setting
G53	G53	G53		Machine coordinate system setting
G53.1	G53.1	G53.1		Tool axis direction control
G53.6	G53.6	G53.6		Tool center point retention type tool axis direction control
G54 (G54.1)	G54 (G54.1)	G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55	G55	G55		Workpiece coordinate system 2 selection
G56	G56	G56		Workpiece coordinate system 3 selection
G57	G57	G57		Workpiece coordinate system 4 selection
G58	G58	G58		Workpiece coordinate system 5 selection
G59	G59	G59		Workpiece coordinate system 6 selection
G54.4	G54.4	G54.4	26	Workpiece setting error compensation
G60	G60	G60	00	Single direction positioning
G61	G61	G61	15	Exact stop mode
G62	G62	G62		Automatic corner override mode
G63	G63	G63		Tapping mode
G64	G64	G64		Cutting mode
G65	G65	G65	00	Macro call

G code system			Group	Function
A	B	C		
G66	G66	G66	12	Macro modal call A
G66.1	G66.1	G66.1		Macro modal call B
G67	G67	G67		Macro modal call A/B cancel
G68	G68	G68	04	Mirror image on for double turret or balance cutting mode
G68.1	G68.1	G68.1	17	Coordinate system rotation start or 3-dimensional coordinate system conversion mode on
G68.2	G68.2	G68.2		Tilted working plane command
G68.3	G68.3	G68.3		Tilted working plane command by tool axis direction
G68.4	G68.4	G68.4		Tilted working plane command (incremental multi-command)
G69	G69	G69	04	Mirror image off for double turret or balance cutting mode Cancel
G69.1	G69.1	G69.1	17	Coordinate system rotation cancel or 3-dimensional coordinate system conversion mode off
G70	G70	G72	00	Finishing cycle
G71	G71	G73		Stock removal in turning
G72	G72	G74		Stock removal in facing
G73	G73	G75		Pattern repeating cycle
G74	G74	G76		End face peck drilling cycle
G75	G75	G77		Outer diameter/internal diameter drilling cycle
G76	G76	G78		Multiple-thread cutting cycle
G71	G71	G72	01	Traverse grinding cycle
G72	G72	G73		Traverse direct sizing/grinding cycle
G73	G73	G74		Oscillation grinding cycle
G74	G74	G75		Oscillation direct sizing/grinding cycle
G80	G80	G80	10	Canned cycle cancel for drilling Electronic gear box : synchronization cancellation
G80.4	G80.4	G80.4	28	Electronic gear box: synchronization cancellation
G81.4	G81.4	G81.4		Electronic gear box: synchronization start
G80.5	G80.5	G80.5	27	Electronic gear box 2 pair: synchronization cancellation
G81.5	G81.5	G81.5		Electronic gear box 2 pair: synchronization start
G81	G81	G81	10	Spot drilling (FS15-T format) Electronic gear box : synchronization start
G82	G82	G82		Counter boring (FS15-T format)
G83	G83	G83		Cycle for face drilling
G83.1	G83.1	G83.1		High-speed peck drilling cycle (FS15-T format)
G83.5	G83.5	G83.5		High-speed peck drilling cycle
G83.6	G83.6	G83.6		Peck drilling cycle
G84	G84	G84		Cycle for face tapping
G84.2	G84.2	G84.2		Rigid tapping cycle (FS15-T format)
G85	G85	G85		Cycle for face boring
G87	G87	G87		Cycle for side drilling
G87.5	G87.5	G87.5		High-speed peck drilling cycle
G87.6	G87.6	G87.6		Peck drilling cycle
G88	G88	G88		Cycle for side tapping
G89	G89	G89		Cycle for side boring
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Threading cycle
G94	G79	G24		End face turning cycle
G91.1	G91.1	G91.1	00	Maximum specified incremental amount check

G code system			Group	Function
A	B	C		
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G96.1	G96.1	G96.1	00	Spindle indexing execution (waiting for completion)
G96.2	G96.2	G96.2		Spindle indexing execution (not waiting for completion)
G96.3	G96.3	G96.3		Spindle indexing completion check
G96.4	G96.4	G96.4		SV speed control mode ON
G93	G93	G93	05	Inverse time feed
G98	G94	G94		Feed per minute
G99	G95	G95		Feed per revolution
-	G90	G90	03	Absolute programming
-	G91	G91		Incremental programming
-	G89	G98	11	Canned cycle : return to initial level
-	G99	G99		Canned cycle : return to R point level

4.7 Mill G-codes

G code	Group	Function
G00	01	Positioning (rapid traverse)
G01		Linear interpolation (cutting feed)
G02		Circular interpolation CW or helical interpolation CW
G03		Circular interpolation CCW or helical interpolation CCW
G02.1, G03.1		Circular thread cutting B CW/CCW
G02.2, G03.2		Involute interpolation CW/CCW
G02.3, G03.3		Exponential interpolation CW/CCW
G02.4, G03.4		3-dimensional coordinate system conversion CW/CCW
G04		Dwell
G05	00	AI contour control (high-precision contour control compatible command), High-speed cycle machining, High-speed binary program operation
G05.1		AI contour control / Nano smoothing / Smooth interpolation
G05.4		HRV3, 4 on/off
G06.2		NURBS interpolation
G07	00	Hypothetical axis interpolation
G07.1 (G107)		Cylindrical interpolation
G08		AI contour control (advanced preview control compatible command)
G09		Exact stop
G10		Programmable data input
G10.6		Tool retract and recover
G10.9		Programmable switching of diameter/radius specification
G11		Programmable data input mode cancel
G12.1	21	Polar coordinate interpolation mode
G13.1		Polar coordinate interpolation cancel mode
G12.4	00	Groove cutting by continuous circle motion (CW)
G13.4		Groove cutting by continuous circle motion (CCW)
G15	17	Polar coordinates command cancel
G16		Polar coordinates command
G17	02	XpYp plane selection
G18		ZpXp plane selection
G19		YpZp plane selection
G20 (G70)	06	Input in inch
G21 (G71)		Input in mm
G22	04	Stored stroke check function on
G23		Stored stroke check function off
G25	19	Spindle speed fluctuation detection off
G26		Spindle speed fluctuation detection on
G27	00	Reference position return check
G28		Automatic return to reference position
G28.2		In-position check disable reference position return
G29		Movement from reference position
G30		2nd, 3rd and 4th reference position return
G30.1		Floating reference position return
G30.2		In-position check disable 2nd, 3rd, or 4th reference position return
G31		Skip function
G31.8		EGB-axis skip

G code	Group	Function
G33	01	Threading
G34		Variable lead threading
G35		Circular threading CW
G36		Circular threading CCW
G37	00	Automatic tool length measurement
G38		Tool radius/tool nose radius compensation : preserve vector
G39		Tool radius/tool nose radius compensation : corner circular interpolation
G40	07	Tool radius/tool nose radius compensation : cancel 3-dimensional cutter compensation : cancel
G41		Tool radius/tool nose radius compensation : left 3-dimensional cutter compensation : left
G42		Tool radius/tool nose radius compensation : right 3-dimensional cutter compensation : right
G41.2		3-dimensional cutter compensation : left (type 1)
G41.3		3-dimensional cutter compensation : leading edge offset
G41.4		3-dimensional cutter compensation : left (type 1) (FS16i-compatible command)
G41.5		3-dimensional cutter compensation : left (type 1) (FS16i-compatible command)
G41.6		3-dimensional cutter compensation : left (type 2)
G42.2		3-dimensional cutter compensation : right (type 1)
G42.4		3-dimensional cutter compensation : right (type 1) (FS16i-compatible command)
G42.5		3-dimensional cutter compensation : right (type 1) (FS16i-compatible command)
G42.6		3-dimensional cutter compensation : right (type 2)
G40.1	18	Normal direction control cancel mode
G41.1		Normal direction control on : left
G42.1		Normal direction control on : right
G43	08	Tool length compensation +
G44		Tool length compensation -
G43.1		Tool length compensation in tool axis direction
G43.3		Nutating rotary head tool length compensation
G43.4		Tool center point control (type 1)
G43.5		Tool center point control (type 2)
G45	00	Tool offset : increase
G46		Tool offset : decrease
G47		Tool offset : double increase
G48		Tool offset : double decrease
G49 (G49.1)	08	Tool length compensation cancel
G44.9	27	Spindle unit compensation
G49.9		Spindle unit compensation cancel
G50	11	Scaling cancel
G51		Scaling

G code	Group	Function
G50.1	22	Programmable mirror image cancel
G51.1		Programmable mirror image
G50.2	31	Polygon turning cancel
G51.2		Polygon turning
G50.4	00	Cancel synchronous control
G50.5		Cancel composite control
G50.6		Cancel superimposed control
G51.4		Start synchronous control
G51.5	00	Start composite control
G51.6		Start superimposed control
G52		Local coordinate system setting
G53		Machine coordinate system setting
G53.1		Tool axis direction control
G53.6		Tool center point retention type tool axis direction control
G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G54.2	23	Rotary table dynamic fixture offset
G54.4	33	Workpiece setting error compensation
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	00	Macro call
G66	12	Macro modal call A
G66.1		Macro modal call B
G67		Macro modal call A/B cancel
G68	16	Coordinate system rotation start or 3-dimensional coordinate conversion mode on
G69		Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off
G68.2		Tilted working plane command
G68.3		Tilted working plane command by tool axis direction
G68.4		Tilted working plane command (incremental multi-command)

G code	Group	Function
G70.7	00	Finishing cycle
G71.7		Outer surface rough machining cycle
G72.7		End rough machining cycle
G73.7		Closed loop cutting cycle
G74.7		End cutting off cycle
G75.7		Outer or inner cutting off cycle
G76.7		Multiple threading cycle
G72.1		Figure copying (rotary copy)
G72.2		Figure copying (linear copy)
G73	09	Peck drilling cycle
G74		Left-handed tapping cycle
G75	01	Plunge grinding cycle
G76	09	Fine boring cycle
G77	01	Plunge direct sizing/grinding cycle
G78		Continuous-feed surface grinding cycle
G79		Intermittent-feed surface grinding cycle
G80	09	Canned cycle cancel Electronic gear box : synchronization cancellation
G80.4	34	Electronic gear box: synchronization cancellation
G81.4		Electronic gear box: synchronization start
G80.5	24	Electronic gear box 2 pair: synchronization cancellation
G81.5		Electronic gear box 2 pair: synchronization start
G81	09	Drilling cycle or spot boring cycle Electronic gear box : synchronization start
G81.1	00	Chopping
G82	09	Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G84.2		Rigid tapping cycle (FS15 format)
G84.3		Left-handed rigid tapping cycle (FS15 format)
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90	03	Absolute programming
G91		Incremental programming
G91.1	00	Checking the maximum incremental amount specified
G92		Setting for workpiece coordinate system or clamp at maximum spindle speed
G92.1		Workpiece coordinate system preset
G93	05	Inverse time feed
G94		Feed per minute
G95		Feed per revolution
G96	13	Constant surface speed control
G97		Constant surface speed control cancel

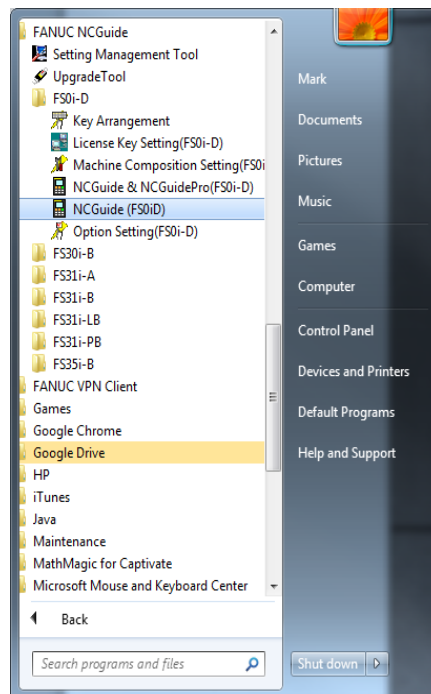
G code	Group	Function
G96.1	00	Spindle indexing execution (waiting for completion)
G96.2		Spindle indexing execution (not waiting for completion)
G96.3		Spindle indexing completion check
G96.4		SV speed control mode ON
G98	10	Canned cycle : return to initial level
G99		Canned cycle : return to R point level
G107	00	Cylindrical interpolation
G112	21	Polar coordinate interpolation mode
G113		Polar coordinate interpolation mode cancel
G160	20	In-feed control cancel
G161		In-feed control

5 Operation

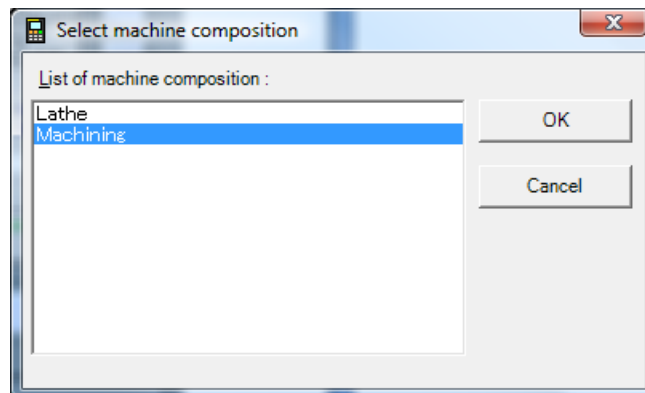
5.1 Starting NCGuide

Before starting NCGuide, the hardware key supplied with the product must be installed in a USB port on the PC, or the License Key Authorize Setting utility must be used to provide access a network license.

From the Windows Start menu, select the NCGuide program for the CNC model you wish to use. There is a program structure under FANUC NCGuide for each of the CNC models installed. The display bellow is typical of a Windows 7 display.



When the select machine composition dialog box is displayed, select either Lathe or Machining. Note that addition machine compositions may be created to configure specific CNC configurations.

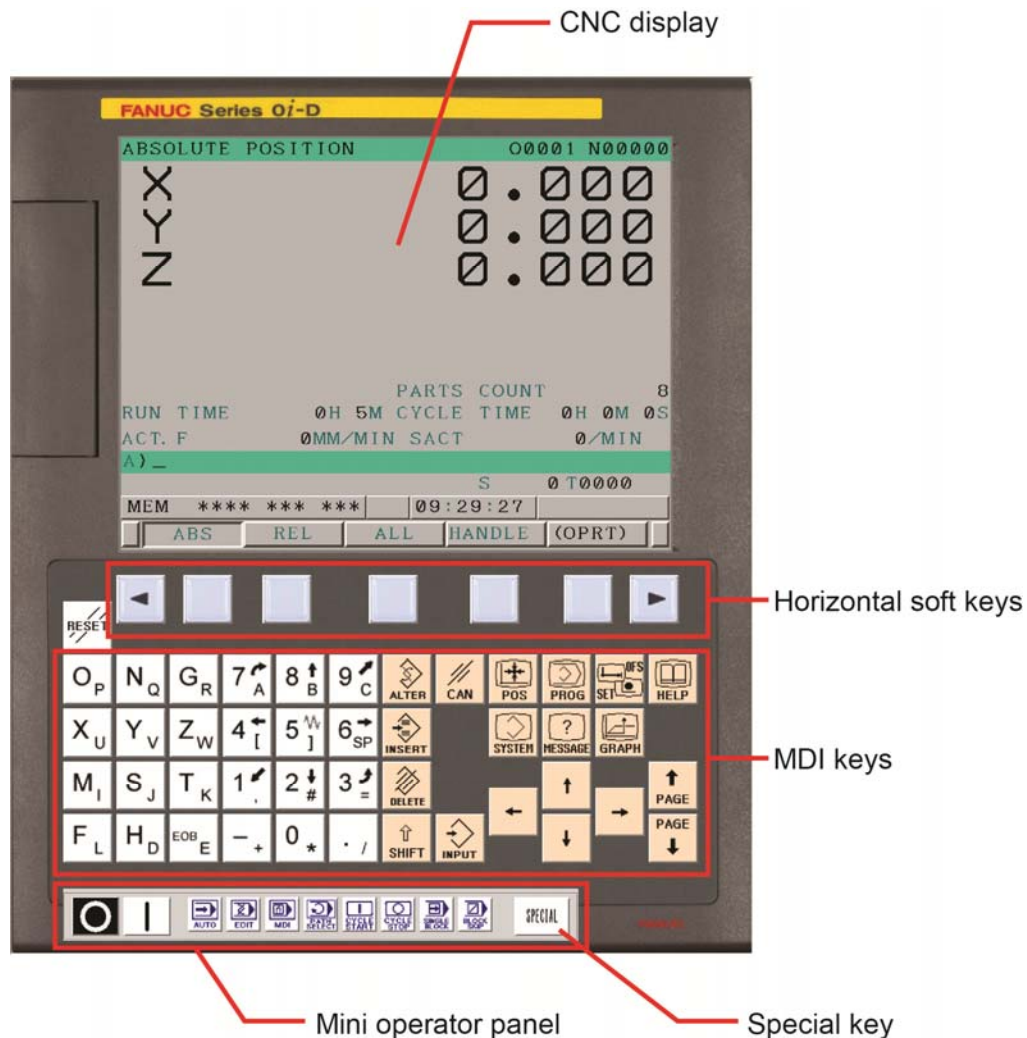


NCGuide will now start.

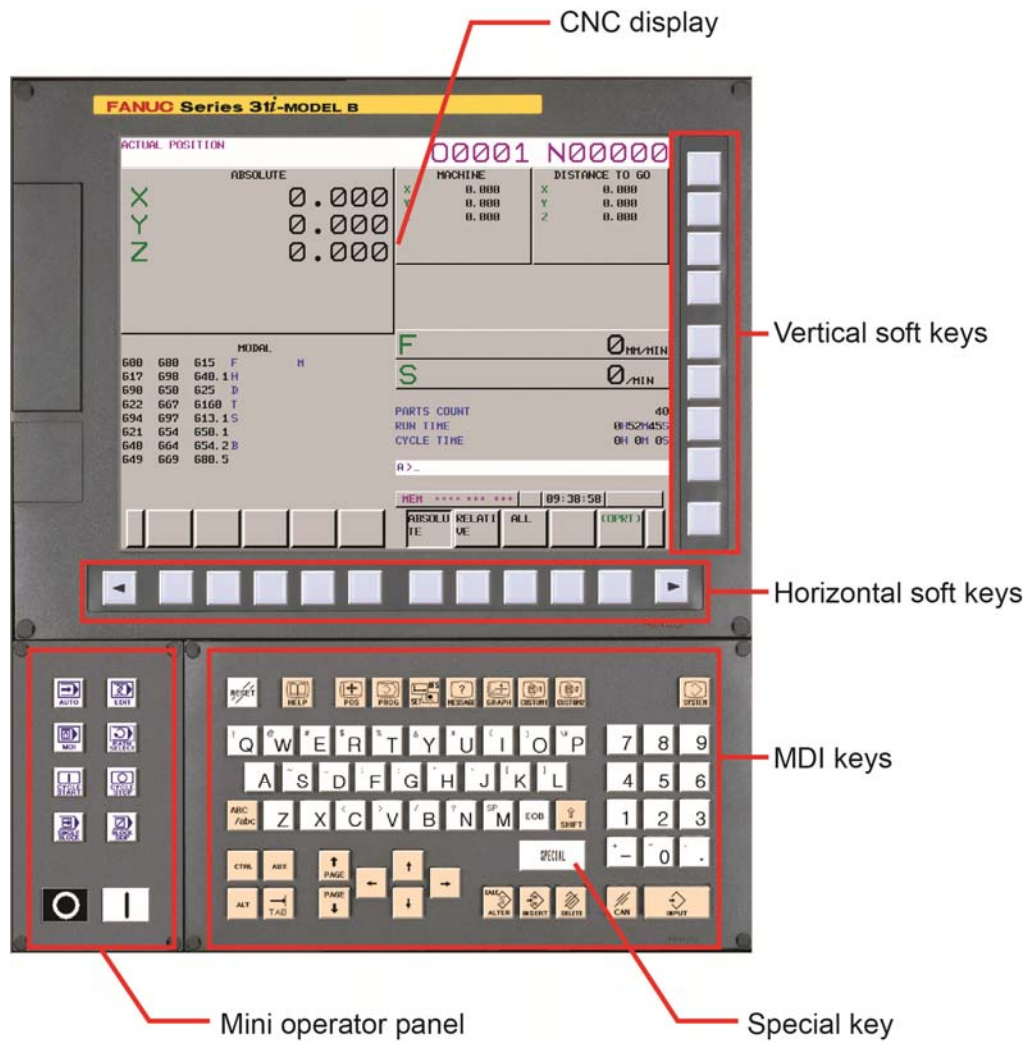
5.2 Components

Component	Function
CNC display	Show the CNC screens
Horizontal/vertical soft keys	Clicking them allows you to perform their corresponding soft key operations. The vertical soft keys are not available on the 0i MODEL D controls.
MDI keys	Clicking them allows you to perform corresponding MDI operations
Mini operator panel	Click the buttons allows you to perform the functions associated to individual keys
Special key	Combined-key operations are executed using this special key

5.2.1 $0i$ -D components



5.2.2 30i, 31i and 35i components



5.3 CNC Mode selection

CNC operation is determined by selecting the CNC mode. The modes supported by NCGuide are EDIT, AUTOMATIC (MEM) and MDI. The mode can be selected with mini operator's panel displayed by NCGuide.

The active mode is displayed in the lower right of the CNC screen.



5.3.1 EDIT - Program editing mode

The following program editing operations are possible.

1. Creating new part programs
2. Editing part programs
3. Deleting part programs

5.3.2 MEM – Memory operation

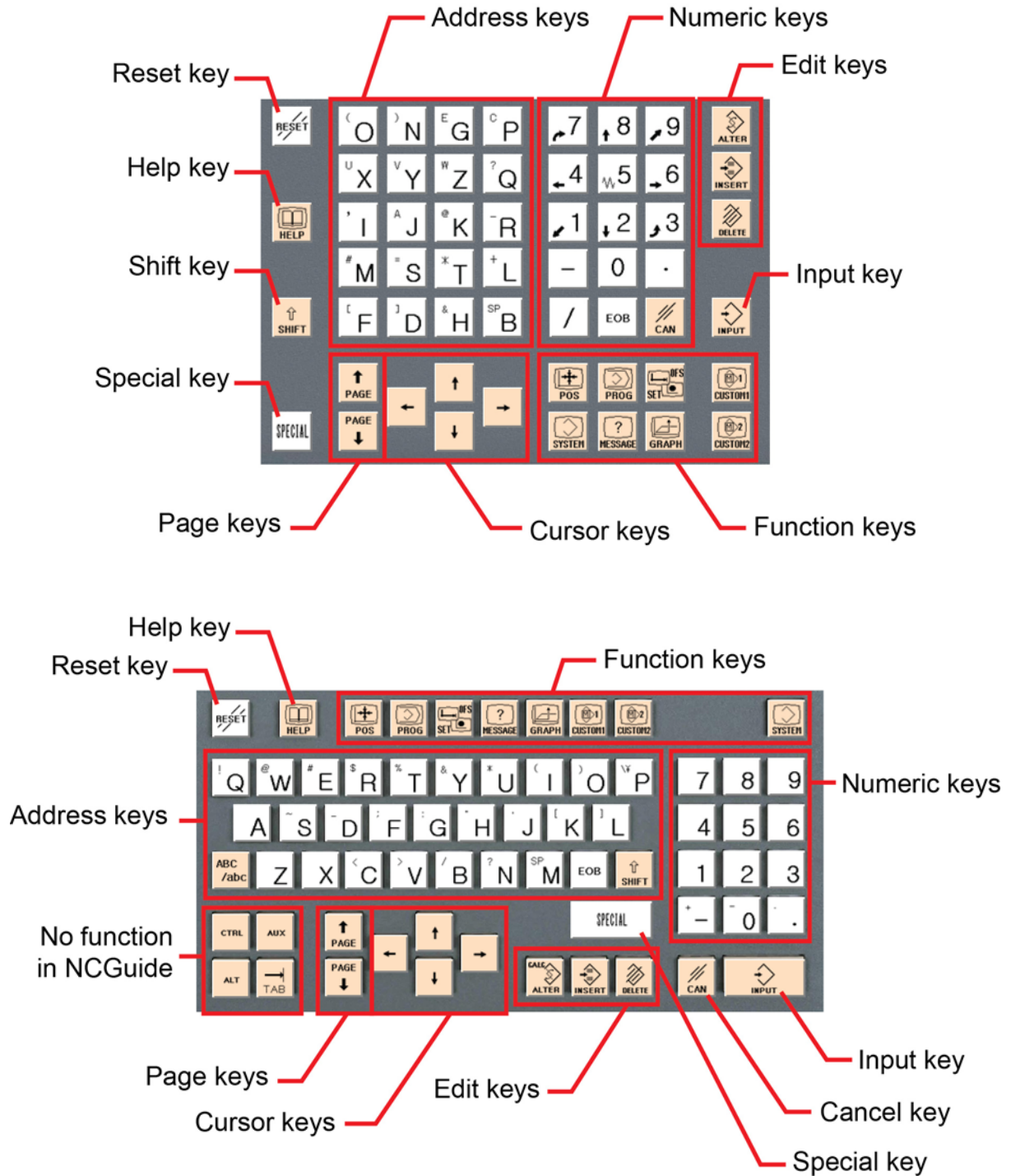
The part programs registered in the CNC memory can be executed.

5.3.3 MDI operation mode

Part program blocks can be entered and executed using the MDI keys.

5.4 MDI key operation

The MDI keys may be used in the same way as those on an actual CNC, either by using a mouse or a touch screen. Though the MDI key panel layout may change depending on the CNC model or when the display size is changed, the components are readily identifiable.



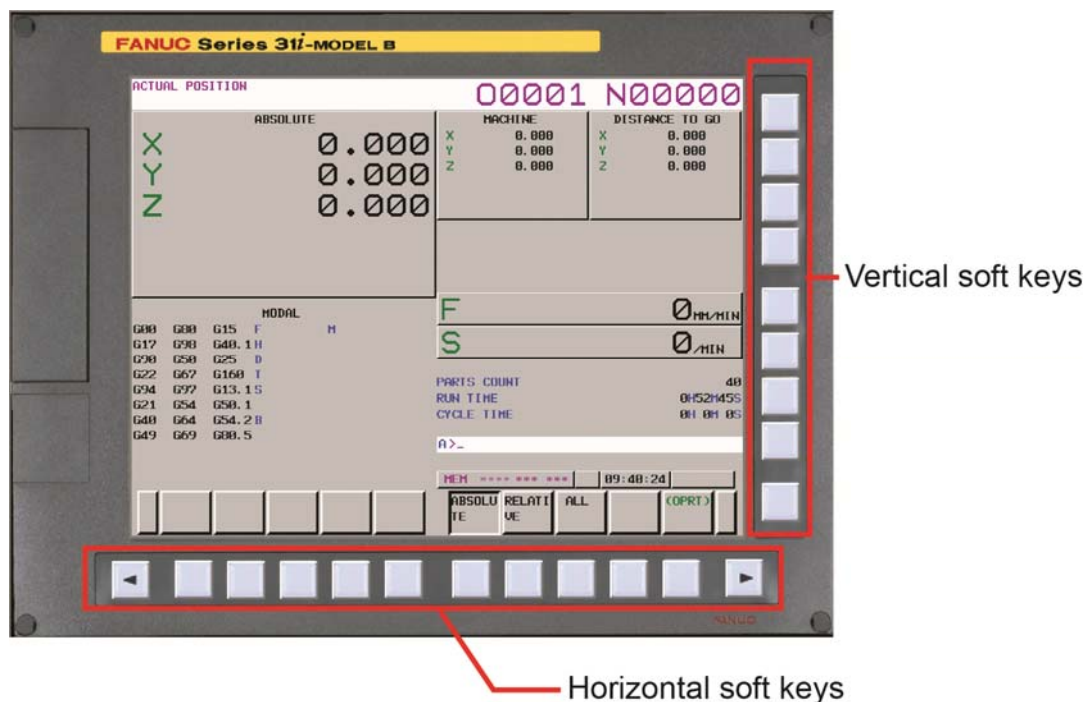
5.4.1 MDI keys

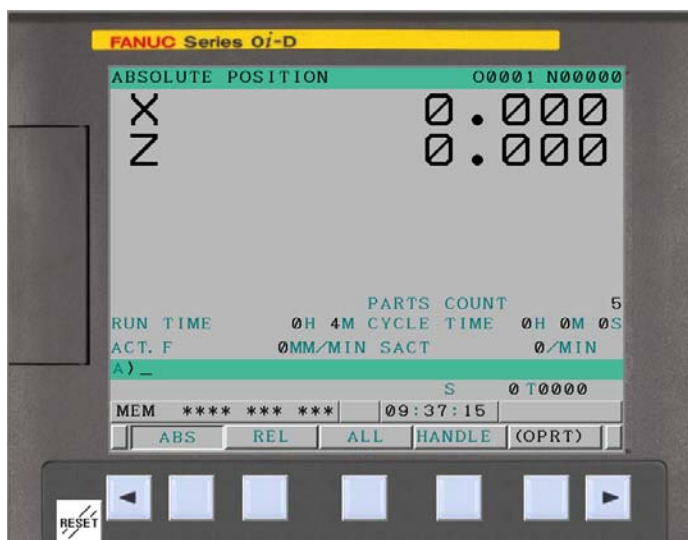
Key(s)	Function	PC keyboard
Reset key	Reset the CNC	Esc
Help key	Detailed information on how to operate the CNC is displayed	Home
Function keys	Selects the CNC screens	
POS	CNC position screen is displayed	Ctrl-r
PROG	Part program screen is displayed	Ctrl-p
OFS/SET	Offset or setting screens are displayed	Ctrl-o
MESSAGE	Alarm screen is displayed	Ctrl-m
GRAPH	Graphic screen is displayed	Ctrl-g
CUSTOM1	Not used by NCGuide	n/a
CUSTOM2	Not used by NCGuide	n/a
SYSTEM	Parameter or diagnostic screens are displayed	Ctrl-s
Numeric keys	Allow numerical values to be entered	Numeric keys 0..9
Input key	Data in edit key buffer is entered in CNC	Enter
Cancel key	Data in edit key buffer is cancelled	Backspace
Address keys	Allows letter address characters to be entered	Letter keys A..Z
Cursor keys	Move cursor up, down, left, right on CNC	Cursor keys
Page keys	Pages screen up and down on CNC screen	Page up / page down
Edit keys	Allow the part program to be modified	
Alter key	The character or word at the cursor position is modified with the keys pressed	n/a
Insert key	The data entered is inserted at the cursor position	Insert
Delete key	The character or word at the cursor position is deleted	Del
Special key	Used for an operation that normally requires the simultaneous pressing of two keys on actual CNC. This key is not available on an actual CNC. To simulate the simultaneous pressing of two keys, first click the special key, then click the two keys in succession in any order. As soon as the second key is clicked, the corresponding function will be executed.	n/a

Note that the PC keyboard assignments can be customized using the Key Arrangement tool selectable in the NCGuide program start menu structure. Combinations of a key together with the Shift, Ctrl and Alt keys are possible.

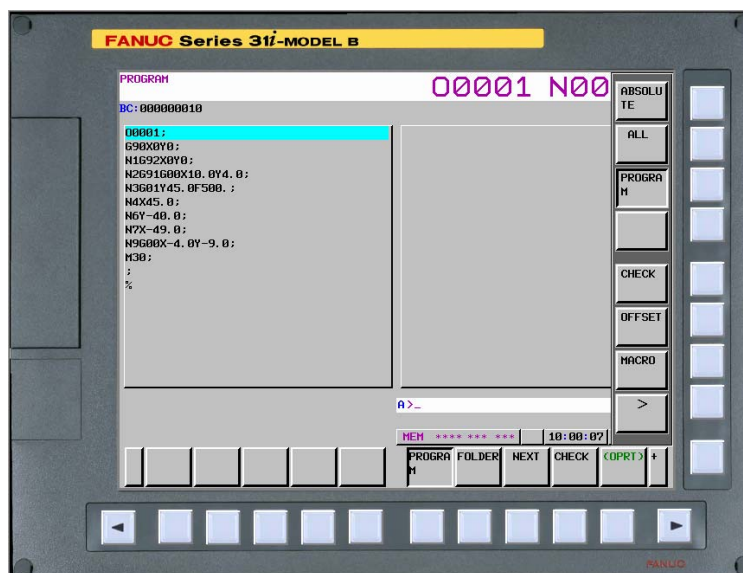
5.5 Soft key operation

The horizontal and vertical soft keys may be used in the same way as those on an actual CNC, either by using a mouse or a touch screen. The horizontal soft keys are used to perform operations describe by the text above each key. The vertical soft keys are used as chapter selection soft keys, an alternative to the tradition function key selection. By pressing one of the vertical keys, the screen (chapter) belonging to each function can be selected. With the 10.4" screen, the text associate with each vertical key is displayed by pressing the lowermost vertical key. Note that the Oi-D control does not support the vertical soft keys.

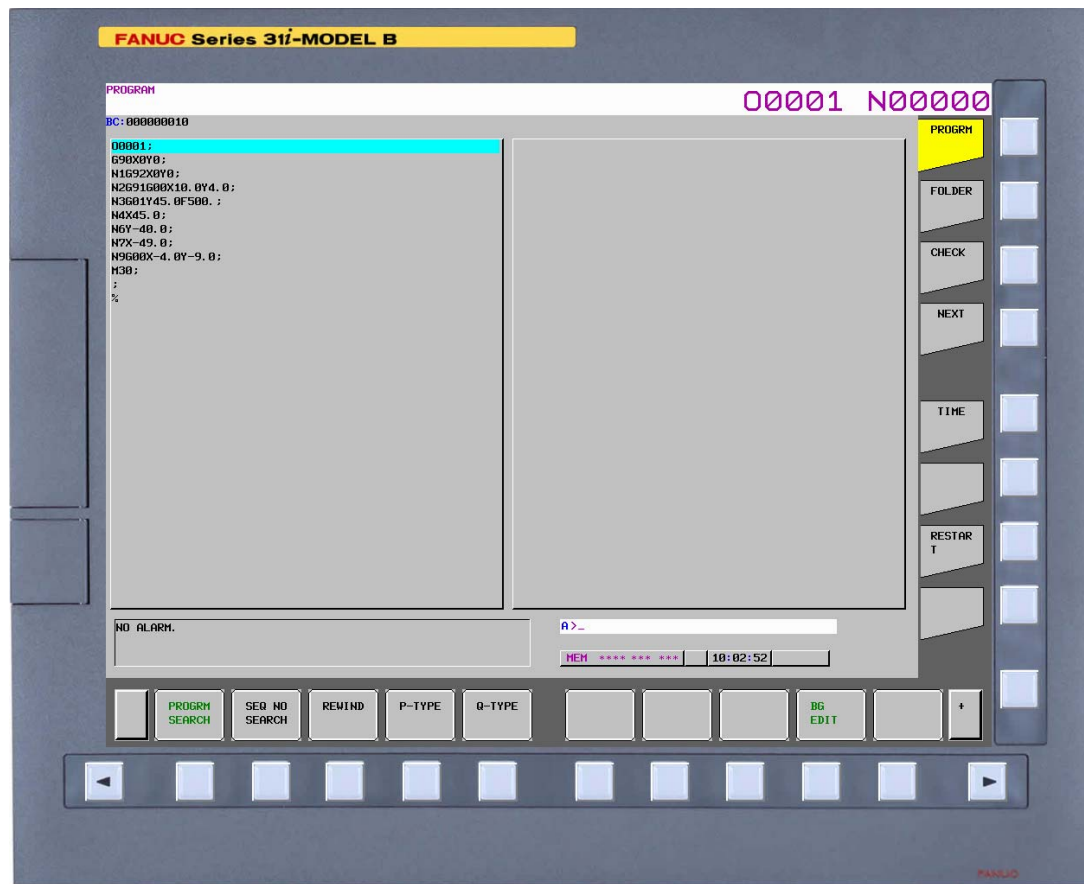




0i-D 8.4" display horizontal soft keys (no vertical available)



31i-B 10.4" display soft keys (with vertical text displayed)



31i-B 15" display soft keys

The PC keyboard keys to control the horizontal soft keys are F1 through F12, assigned from left to right. The PC keyboard keys to control the vertical soft keys are Ctrl-F1 through Ctrl-F9, assigned from top to bottom.

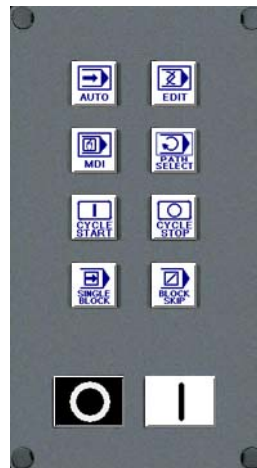
Note that the PC keyboard assignments can be customized using the Key Arrangement tool selectable in the NCGuide program start menu structure. Combinations of a key together with the Shift, Ctrl and Alt keys are possible.

5.6 Mini operator panel operation

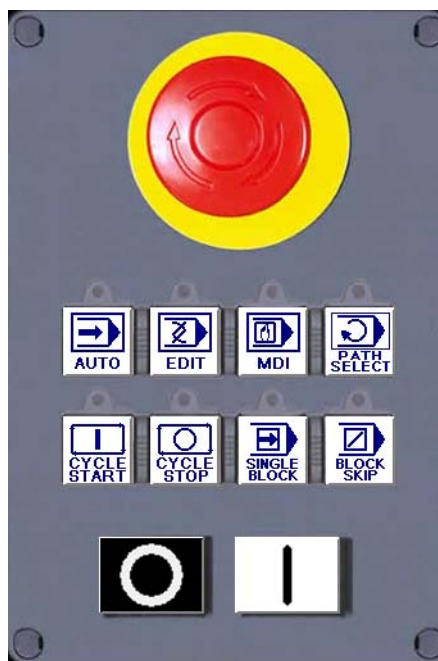
The mini operator panel is provided with the basic operations necessary to run part programs in CNC guide. Clicking the operation keys allows you to perform CNC mode switching, cycle start and stop, single block execution and block skip. Though the visual placement of the keys may change with each display, the functions remains the same.













0i-D mini operator panel



31i 10.4" mini operator panel



31i 15" mini operator panel

Key(s)	Function	PC keyboard
	Exits NCGuide	
	No function, just provided for display purposes	
	Switches the CNC to auto mode	Ctrl-1
	Switches the CNC to edit mode	Ctrl-3
	Switches the CNC to MDI mode	Ctrl-2
	No function in NCGuide Academic packages because they only support single path controls	
	Cycle start the execution of a part program or MDI block	Ctrl-[
	Stop (pause) part program execution	Ctrl-]
	Execute the part program one block at a time	On Ctrl-\, off Ctrl-^
	Skip blocks in the part program mark with a slash '/'	

Note that the PC keyboard assignments can be customized using the Key Arrangement tool selectable in the NCGuide program start menu structure. Combinations of a key together with the Shift, Ctrl and Alt keys are possible.

5.7 Screens display

The following are examples of the common screens displayed in NCGuide.

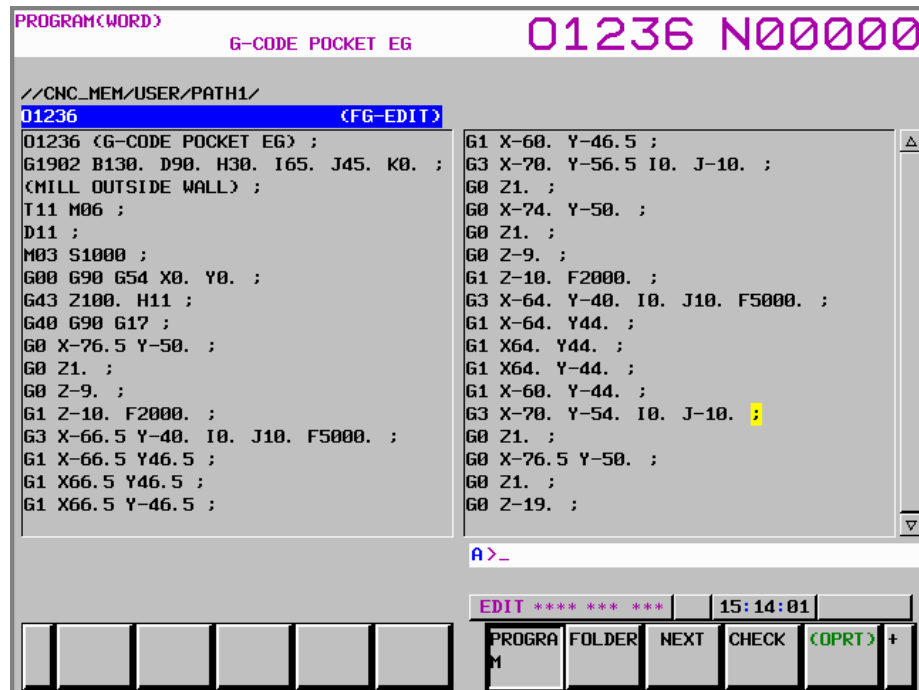
5.7.1 Position screen

ACTUAL POSITION				TOOL OFFSET		00003 N00000	
ABSOLUTE				MACHINE		DISTANCE TO GO	
X		0.000	X	0.000	X	0.000	
Y		0.000	Y	0.000	Y	0.000	
Z		0.000	Z	0.000	Z	0.000	
MODAL				F		0 MM/MIN	
G00	G00	G15	F		S		0 /MIN
G17	G98	G40.1	H				
G90	G50	G25	D				
G22	G67	G160	T				
G94	G97	G13.1	S				
G21	G54	G50.1					
G40	G64	G54.2	B				
G49	G69	G80.5					
				PARTS COUNT		59	
				RUN TIME		1H 9M 0S	
				CYCLE TIME		0H 0M 0S	
				A>_			
				MEM ****		15:07:45	
				ABSOLUTE		RELATIVE	
				ALL		OPRT	

Position screen - Series 31i 10.4" display

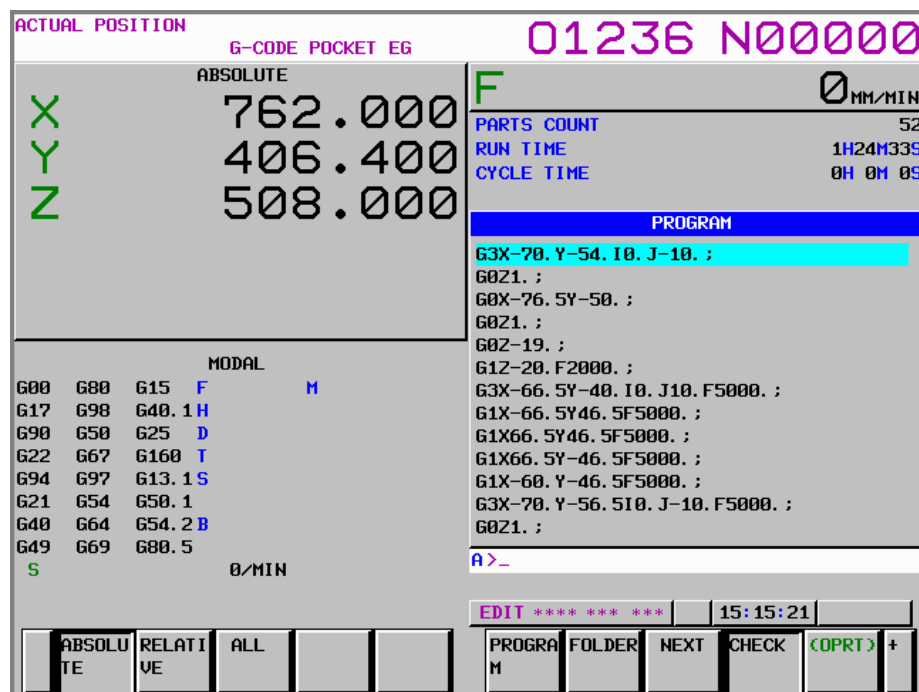
The current positions of each axis is displayed in the workpiece coordinate system, relative coordinate system, machine coordinate system and distance to go.

5.7.2 Program screen



Program screen - Series 31i 10.4" display

The part program may be entered, displayed, edited and executed.



Program check screen - Series 31i 10.4" display

The program check screen provides useful information to execute the part program, displaying the part program, position data and modal information all on a single screen.

5.7.3 Setting screen

Tool offsets and work coordinate offsets can be display and modified.

OFFSET 01236 N00000

G-CODE POCKET EG

NO.	(LENGTH)		(RADIUS)		RELATIVE
	GEOM	WEAR	GEOM	WEAR	
001	0.000	0.000	152.400	0.000	X 0.000
002	0.000	0.000	254.000	0.000	Y 0.000
003	0.000	0.000	0.000	0.000	Z 0.000
004	0.000	0.000	152.400	0.000	
005	0.000	0.000	304.800	0.000	
006	0.000	0.000	76.200	0.000	
007	0.000	0.000	0.000	0.000	
008	0.000	0.000	0.000	0.000	
009	0.000	0.000	0.000	0.000	
010	0.000	0.000	0.000	0.000	
011	0.000	0.000	4.000	0.000	
012	0.000	0.000	3.000	0.000	
013	0.000	0.000	2.000	0.000	
014	0.000	0.000	0.000	0.000	
015	0.000	0.000	0.000	0.000	
016	0.000	0.000	0.000	0.000	

RELATIVE
X 0.000
Y 0.000
Z 0.000

ABSOLUTE
X 762.000
Y 406.400
Z 508.000

MACHINE
X 762.000
Y 406.400
Z 508.000

A>_

EDIT ***** 15:19:04

OFFSET SETTING WORK (OPRT) +

Tool offset screen - Series 31i 10.4" display

ACTUAL POSITION 01236 N00000

G-CODE POCKET EG

ABSOLUTE
X 762.000
Y 406.400
Z 508.000

MODAL
G00 G80 G15 F M
G17 G98 G40.1 H
G90 G50 G25 D
G22 G67 G160 T
G94 G97 G13.1 S
G21 G54 G50.1
G40 G64 G54.2 B
G49 G69 G80.5 S
0/MIN

WORK COORDINATES
(G54)
NO. DATA NO. DATA
000 X 0.000 002 X 0.000
EXT Y 0.000 G55 Y 0.000
Z 0.000 Z 0.000
001 X 0.000 003 X 0.000
G54 Y 0.000 G56 Y 0.000
Z 0.000 Z 0.000

A>_

EDIT ***** 15:20:08

ABSOLUTE RELATIVE ALL (OPRT) +

Work coordinate offset screen - Series 31i 10.4" display

Several handy settings are available to change input and output settings. Primarily the input/output will be set to the MEMCARD in NCGuide.

ACTUAL POSITION
G-CODE POCKET EG 01236 N00000

ABSOLUTE
X 762.000
Y 406.400
Z 508.000

MODAL
G00 G80 G15 F M
G17 G98 G40.1 H
G90 G50 G25 D
G22 G67 G160 T
G94 G97 G13.1 S
G21 G54 G50.1
G40 G64 G54.2 B
G49 G69 G80.5
S 0/MIN

SETTING (HANDY)
PARAMETER WRITE= 1 (0:DISABLE 1:ENABLE)
TV CHECK = 0 (0:OFF 1:ON)
PUNCH CODE = 0 (0:EIA 1:ISO)
INPUT UNIT = 0 (0:MM 1:INCH)
I/O CHANNEL = 4 (0-35:CHANNEL NO.)
SEQUENCE NO. = 0 (0:OFF 1:ON)
PROGRAM FORMAT = 0 (0:NO CNV 1:F15)
SEQUENCE STOP = 0 (PROGRAM NO.)
SEQUENCE STOP = 0 (SEQUENCE NO.)
CONTRAST (+ = [ON:1] - = [OFF:0])

EDIT ** * 15:23:04**

ABSOLUTE RELATIVE ALL OFFSET SETTING WORK (OPRT) +

Setting (handy) screen - Series 31i 10.4" display

5.7.4 Parameter screen

Parameters can be set that modifies functions of the CNC.

PARAMETER
G-CODE POCKET EG 01236 N00000

00000 SEQ INI ISO TVC 00022 INP CHANNEL/B. G.
00001 FCV 00023 OUT CHANNEL/B. G.
00002 SJZ 00024
00010 PEC PRM PZS 00100 ENS IOP NCR CRF CTU
00012 RMV MIR 00101 NFD ASI SB2
00020 I/O CHANNEL 00102 IO SELECT CH0
00021 OUT CHANNEL/F. G. 00103 BAUDRATE CH0
00110 I04
00111 NFD ASI SB2

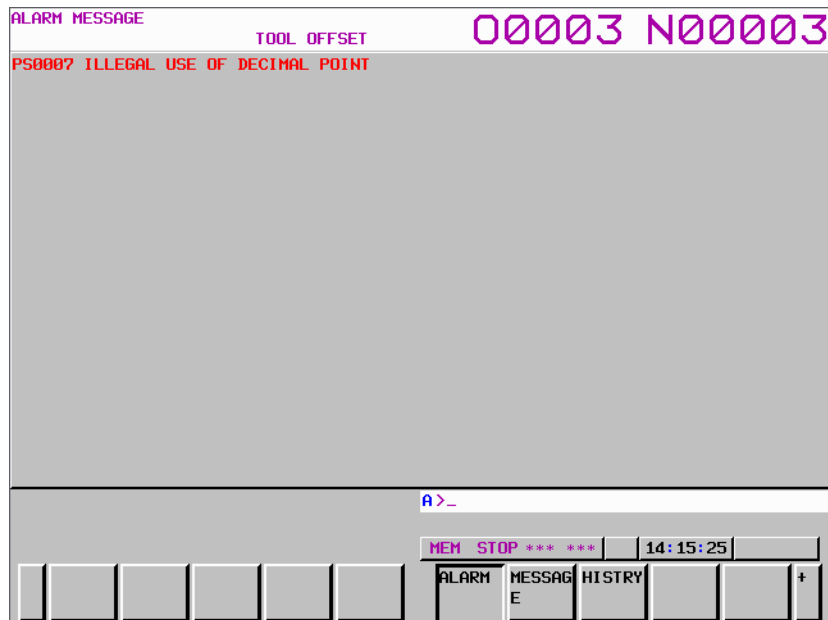
EDIT ** * 15:24:25**

PARAMETER DIAGNOSIS SERVO GUIDE SYSTEM (OPRT) +

Parameter screen - Series 31i 10.4" display

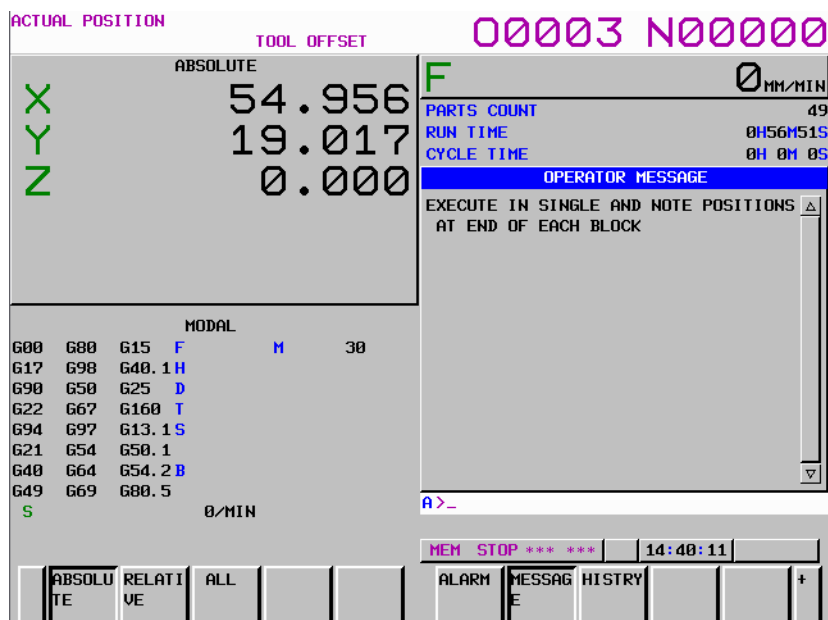
5.7.5 Message screen

All usual alarm and operator messages are displayed.



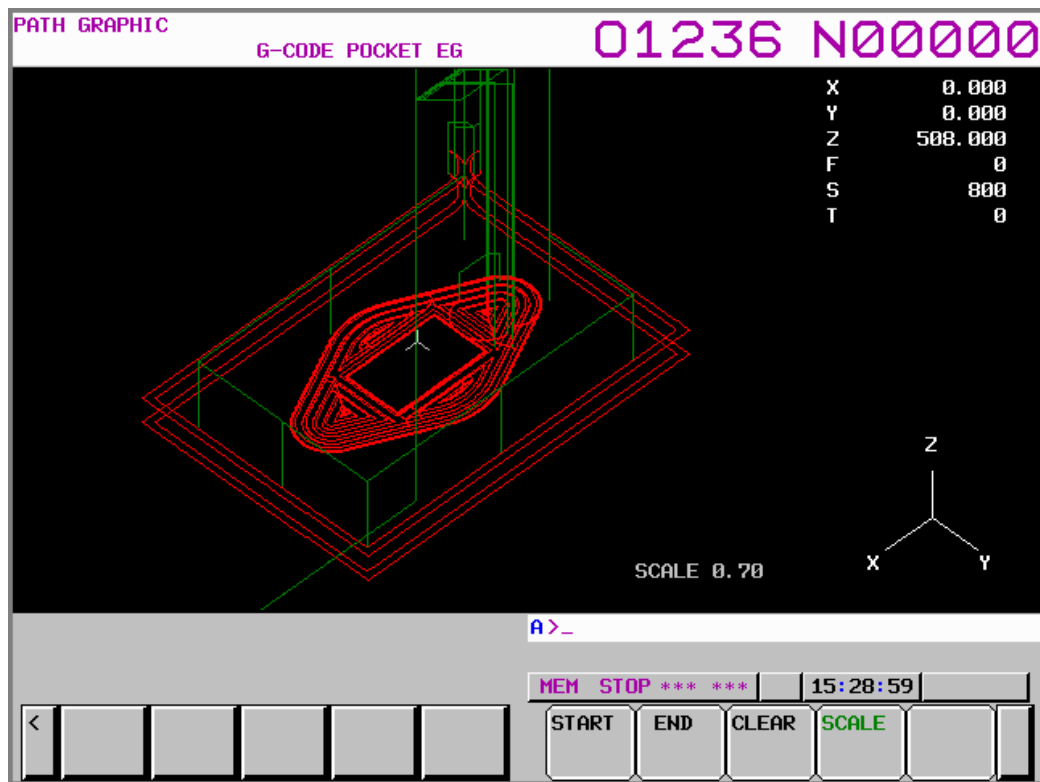
Alarm screen - Series 31i 10.4" display

Alarms and messages can be generated in the part program or subprograms using Custom Macro variables (#3000 Alarm, #3006 Message) to provide feedback or instructions to the student.



Operator message generated with #3006 Custom Macro - Series 31i 10.4" display

5.7.6 Graphic screen



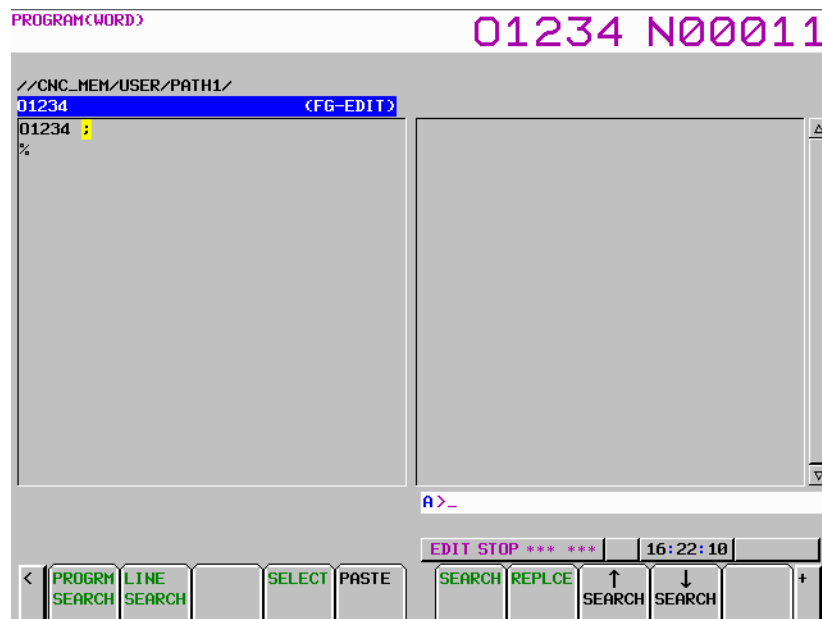
Graph screen - Series 31i 10.4" display

5.8 Editing part programs

5.8.1 Creating part programs

Before you can enter a part program through the MDI key panel, it must be registered. For example, the procedure to register the part program number O1234 is as follows:

1. Select EDIT mode.
2. Press the PROG function key.
3. Press the 'O' letter address key.
4. Press the '1', '2', '3' and '4' numeric keys in order.
5. Press the INSERT key.
6. Press the EOB (end of block) key.
7. Press the INSERT key.
8. "O1234" is displayed on the screen. Note that the "%" end of program marker is generated automatically.
9. Now you can add blocks to the registered part program.



Registering part program 'O1234' screen - Series 31i 10.4" display

5.8.2 Altering a word

The procedure for altering a word is as follows:

1. Select the EDIT mode.
2. Press the PROG function key.
3. Move the cursor to the word to be altered using the page and cursor keys.
4. Use the address and numeric keys to enter the new text in the input buffer.
5. Press the ALTER key to replace the selected word with the text in the input buffer.

5.8.3 Inserting a word

1. Select the EDIT mode.
2. Press the PROG function key.
3. Move the cursor to the word before the position you want to insert new text using the page and cursor keys. Note to add a new block, position the cursor on the EOB symbol ';' at the end of the block prior to the insertion point.
4. Use the address and numeric keys to enter the new text in the input buffer. If adding a complete new block, do not forget to include a new EOB symbol ';'.
5. Press the INSERT key to insert the selected word/text you typed in the input buffer.

5.8.4 Deleting a word

1. Select the EDIT mode.
2. Press the PROG function key.
3. Move the cursor to the word you want to delete using the page and cursor keys.
4. Press the DELETE key to delete the selected word.
5. Repeat the process to delete multiple words.

5.9 Selecting part programs

Once part programs have been registered in the CNC, they can be selected by the following two methods:

1. Selection using the program call operation.
2. Selection from the program list. (not available in 0i-D)

5.9.1 Selection using program call operation

The procedure for selecting part programs using the program call operations is as follows:

1. Select EDIT or AUTO (MEM) mode.
2. Press the PROG function key.
3. Enter the program number you want to select using the address and numeric keys.
4. Press the CURSOR DOWN key.
5. The active part program is changed. If the part program is not registered, the warning "SPECIFIED PROGRAM NOT FOUND" is displayed below the input buffer on the CNC screen.

5.9.2 Selection using the part program list

The procedure for selecting part programs using the part program list is as follows:

1. Select EDIT or AUTO (MEM) mode.
2. Press the PROG function key.
3. Press the soft key [FOLDER]
4. Move the cursor to the part program to select using the cursor keys.
5. Press the soft key [(OPRT)]

6. Press the soft key [MAIN PROGRAM].

5.10 Deleting part programs

Once part programs have been registered in the CNC, they can be deleted by the following two methods:

1. Deletion using the program call operation.
2. Deletion from the program list. (not available in 0i-D)

5.10.1 Deletion using program call operation

The procedure for deleting part programs using the program call operation is as follows:

1. Select EDIT or AUTO (MEM) mode.
2. Press the PROG function key.
3. Enter the program number you want to delete using the address and numeric keys. For example to delete "O0001", enter 'O', '0', '0', '0' and then '1' in order, or just 'O' and then '1' (lead zeros are not required).
4. Press the DELETE key.
The following message is displayed "DELETE PROGAM (O0001) ?"
5. Press the soft key [EXEC] if you want to confirm the program deletion, otherwise click the soft key [CAN] to cancel the deletion operation.

5.10.2 Deletion using the part program list

The procedure for deleting part programs using the part program list is as follows:

1. Select EDIT or AUTO (MEM) mode.
2. Press the PROG function key.
3. Press the soft key [FOLDER]
4. Move the cursor to the part program to delete using the cursor keys.
5. Press the soft key [(OPRT)]
6. Press the soft key [DELETE].
The following message is displayed "DELETE PROGAM ?"
7. Press the soft key [EXEC] if you want to confirm the program deletion, otherwise click the soft key [CAN] to cancel the deletion operation.

5.11 Part program execution

5.11.1 Part program execution

Part programs can be executed in MEM (auto) mode and checked with the graphic function.

1. Select the AUTO (MEM) mode.
2. Click the PROG key.
3. Check that the cursor position is at the top of the part program screen. If not, click the soft key [(OPRT)] and then click the soft key [REWIND].
4. Click CYCLE START.

5.11.2 Program graphic function check

1. Select the AUTO (MEM) mode.
2. Click the PROG key.
3. Check that the cursor position is at the top of the part program screen. If not, click the soft key [(OPRT)] and then click the soft key [REWIND].
4. Click the GRAPH key.
5. Press the soft key [(OPRT)] (this step is not necessary in the 0i-D version)
6. Press the soft key [START] (this step is not necessary in the 0i-D version)
7. Click CYCLE START.

5.11.3 Single block execution

When SINGLE BLOCK is active, program blocks are executed one block at a time. CYCLE START must be clicked for each block. To activate SINGLE BLOCK, click the SINGLE button on the mini operator panel. To deactivate SINGLE BLOCK, click the SINGLE button on the mini operator panel again.

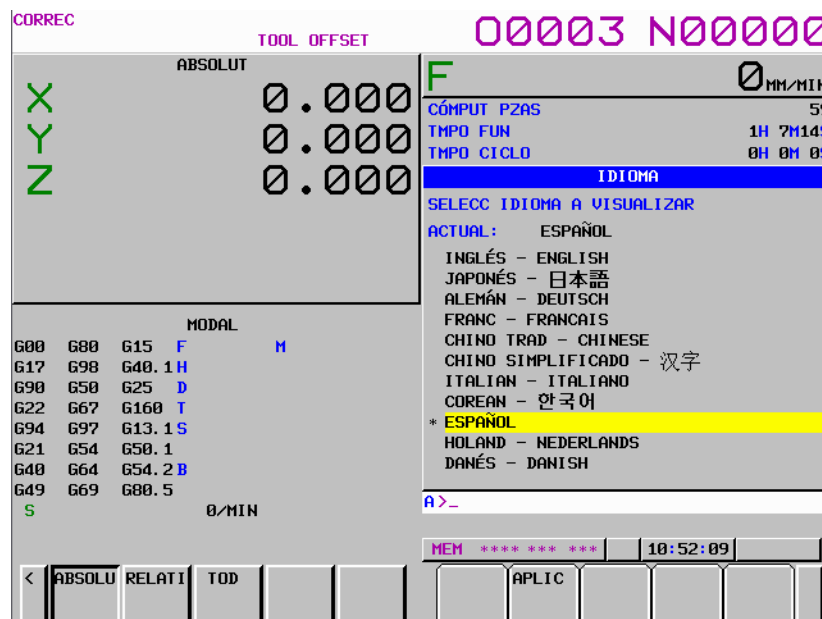
5.12 Multi-language display

The following languages are supported by the CNCs and NCGuide.

- Chinese (simplified)
- Chinese (traditional)
- Czech
- Danish
- Dutch
- English
- French
- German
- Italian
- Hungarian
- Japanese
- Korean
- Portuguese
- Russian
- Spanish
- Swedish
- Turkish

5.12.1 Language switch procedure

1. Click the OFS-SET function key.
2. Click the continuous menu key several times until the [LANGUAGE] soft key text is displayed.
3. Click the soft key [LANGUAGE] to display the language screen.
4. Click the page and cursor keys to move the cursor to the desired display language.
5. Click the soft key [(OPRT)].
6. Click the soft key [APPLY]. The display is switched to the selected language, which remains active even if the CNC is turned off and on.



Setting the display language screen - Series 31i 10.4" display

5.13 Exiting NCGuide

There are several methods to exit NCGuide

In picture mode:

- Click the "Off" button on the mini operator panel
- Right-click anywhere on the CNC and select Exit from the popup menu.

In window mode:

- Select Exit from the File menu.

6 MANUAL GUIDE *i*

6.1 What is MANUAL GUIDE *i*

MANUAL GUIDE *i* is an easy-to-use yet powerful conversational part programming and operation environment. It guides operator's on how to program and operate CNC controls installed on machine tools such as lathes and machining centers. It features a single screen that can be used to create machining part programs, check them using animation, set up tooling and workpieces, and perform the actual machining.

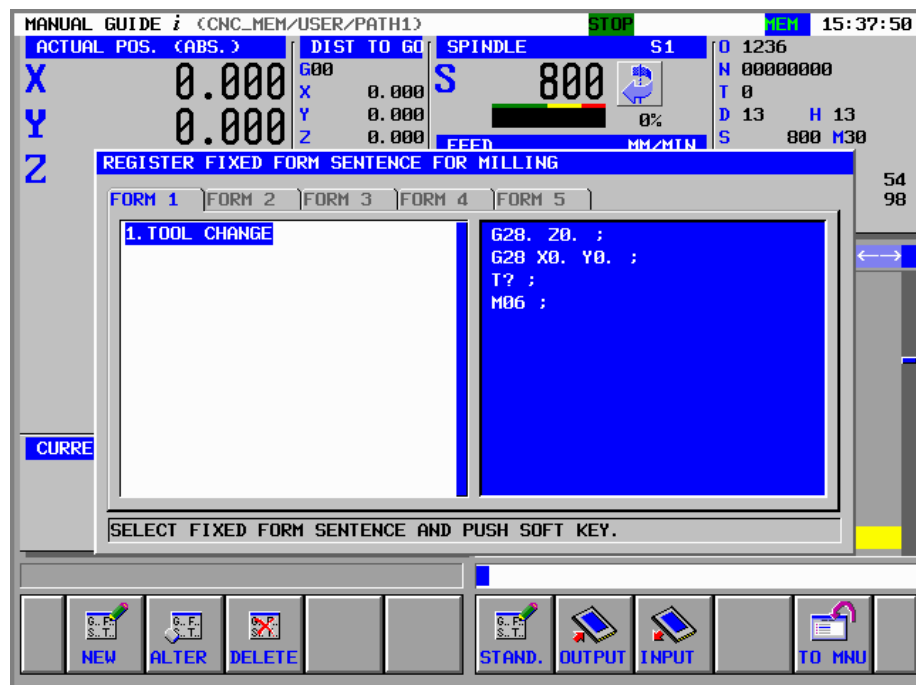
MANUAL GUIDE *i* programs are high-level G-code programs using advanced machining cycles. They can be quickly converted to lower-level G-code programs that will run on a wide range of FANUC CNC models.

6.2 Academic applications of MANUAL GUIDE *i*

NCGuide provides a powerful environment to learn about conversational part programming. Conversational program is rapidly replacing manual machining in many toolrooms and some job shops because it provides more flexibility, accuracy and speed yet still uses journeyman knowledge and skills to create parts directly from the workpiece drawing.

Conversational programming allows the student to learn processes and sequence of operations rather than focus on the specific G-codes used.

MANUAL GUIDE *i* also provides fixed sentence programming to generate multiple lines of G-code with just a few keystrokes. Fixed sentences can be established for sequence of operations such as program starting blocks, tool change blocks, material feeds and speeds and part program end blocks.



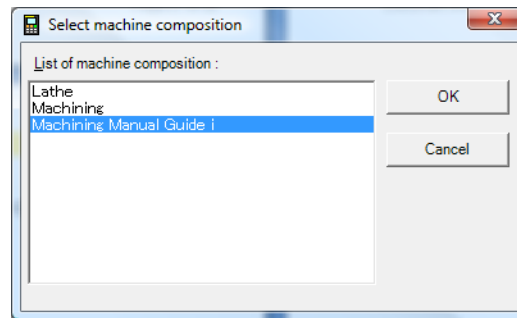
6.3 Main Features of MANUAL GUIDE *i*

- **Integrated operation screen that facilitate most routine machining operations**
A single integrated operation screen provides for routine machining operations including machining part program input/editing, animated simulation based machining program checks, production machining and MDI operations. Manual operations with JOG and HANDLE are also supported (not in NCGuide).
- **Simple part program generation**
Simple menu-driven conversation programming screens guide the operator through a series of common machining operations. These high-level operations eliminate the trouble of repeatedly generating the same multiple blocks of G-code.
- **Realistic animated simulation**
Machining programs can be checked easily using the 3D solid model animated simulation for all operations for both milling and turning. It realistically shows the surface being removed with a specific type of tool tip as if a real workpiece is being machined.
- **Advanced machining using machining cycles**
Advanced milling machining cycles are available to perform complex machining, but the conversation environment makes creating and running these programs easy.
- **Machining programs in G-code format**
Common G-code is also supported by MANUAL GUIDE *i* part programs, which enables the operator to specify simple operations such as straight lines and arc with simple G-code commands, and the more complicated machining operations using the advanced machining cycles.
- **Affinity with CAD/CAM**
Machining programs created using CAD/CAM can be still be used without modification. Adding advanced machining cycles to these machining programs makes them perfect machining programs. They can be checked easily, using animated simulation.
- **Advanced machining program editing**
Using advanced editing functions, such as substring search and cut/paste via the clipboard, enables easy editing of machining programs.
- **M-code menu**
It is possible to input M codes easily by referencing explanations displayed in an M code menu. Machine tool builders can create the explanations easily.
- **Advanced set-up guidance (option)**
It is possible to set up machining operations to ensure the precision machining easily. Using an advanced set-up guidance functions all measurements can be easily taken, from tool offset measurement through to the measurement of the workpiece. (not supported by NCGuide)
- **Wide support for various machining centers**
The 3+2 axis machining center with a tilting head as well as vertical and horizontal machining centers are supported.

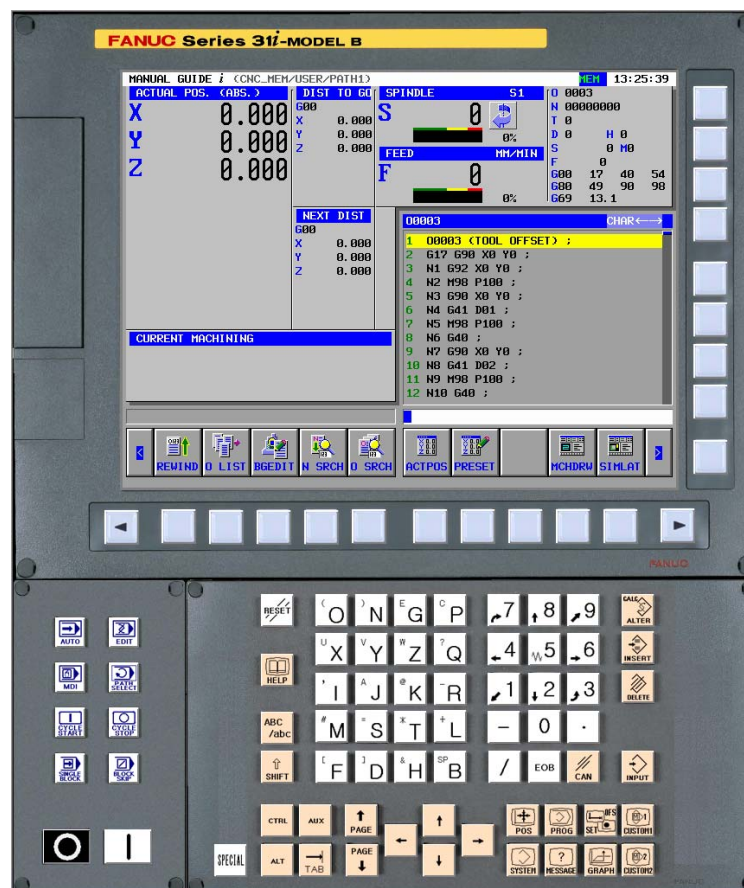
6.4 Navigation to MANUAL GUIDE *i*

6.4.1 Starting MANUAL GUIDE *i*

1. Select NCGuide from the Windows menu.
2. Select a machine composition that includes MANUAL GUIDE *i* then click the OK button.

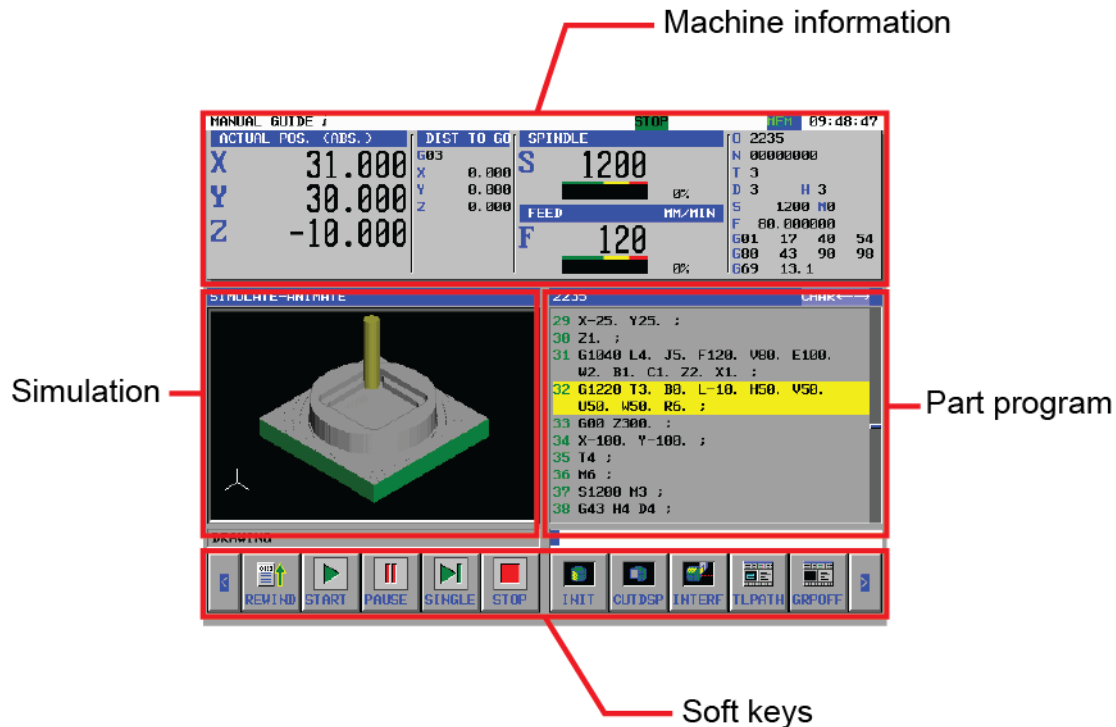


3. Click the GRAPH function key on the CNC when start up is complete.



6.4.2 MANUAL GUIDE *i* screen components

All part programming and operation is achieved from a single primary screen. When additional inputs are required from the operator/part programmer, popup screens are displayed.



Component	Function
Machine information	<ul style="list-style-type: none"> • Axis positions • Distance to go • Spindle speed and direction • Spindle load • Feed rate • Modal G-codes
Simulation	Simulation of part program execution <ul style="list-style-type: none"> • 2D or 3D • Solid model or tool path wire frame
Part program	Part program being executed or edited
Soft keys	All navigation from these keys

6.5 Overview of creating a MANUAL GUIDE *i* part program

A MANUAL GUIDE *i* program is created in the following steps. This follows planning session to define the operations and sequence required to make the part and deciding which tools to use.

1. Analyze the workpiece drawing and determine blank size, geometry and material type, determine a sequence of operations, select tooling to be used and calculate feeds and speeds.
2. Define part blank size (required for part program machining simulation)
3. Select the program start conditions (modal G-codes)
4. Move to the tool-change position, select tool, start spindle, coolant, set tool length offset move to the part approach position
5. Select a machining cycle (e.g. pocketing, drilling, slot milling) and define cutting conditions and feature geometry.
6. Repeat steps 4 and 5, to add additional machining processes for part features
7. Select program end conditions

Example

(define the part blank)

G1902 B100. D100. H30. I0. J0. K1.;

(select a new tool)

G0 Z300.;	(retract Z)
X100. Y100.;	(move X/Y to the tool change position)
T1;	(select the next tool)
M6;	(change tool)
S1200 M3;	(start the spindle)
M08;	(start the coolant)
G43 H1 D1;	(activate tool length and radius offsets)
X-25. Y-25.;	(approach the workpiece for the cycle)
G0 Z2.;	(move Z to the working plane)

(part feature operation)

(cycle cutting conditions)

G1020 T1. L38. F200. E150. W2. P2. V0. C1. M1. A1. B4. Z2. I100.;

(cycle part feature geometry)

G1220 T1. B0. H50. V50. U100. W100.;

(end the part program)

M09;	(turn off coolant)
G0 Z300.;	(retract the Z axis)
X-100. Y-100.;	(move X/Y to a safe position)
T0;	(cancel tool offsets)
M30;	(end and rewind the part program)

Note that many of these steps can be repetitive. For example, step 11 may define many “boiler plate” G-codes to end a part program, turning off coolant, moving the axis to a safe position so the operator can get access to the part, turning off offsets, and rewinding the part program (M30).

Similarly, steps 2 through 5 define those required to make a safe tool change:

```
G0 Z300.;           (retract Z)
X100. Y100.;        (and move X/Y to the tool change position)
T1;                 (select the next tool)
M6;                 (change tool)
S1200 M3;           (start the spindle)
M08;                (start the coolant)
G43 H1 D1           (Activate tool length and radius offsets)
X-25. Y-25.;        (approach the workpiece for the cycle)
G0 Z2.;
```

These common repetitive code sections may be encapsulated and inserted into the part program in a single step using the Fixed Form Sentence feature in MANUAL Guide *i*, simplifying part program creation.

Note that the only true unique parts of a part program are typically those that define the cutting conditions and the geometric properties of a part feature.

6.6 Notes on creating part programs

This section provides notes on creating part programs. Read the notes before creating a part program.

6.6.1 Commands required before machining cycle input

1. For milling controls, a cutter radius offset using the D letter address, even if cutter radius compensation is not used. This selects the tool geometry for 3D solid model animation.
2. Be sure to enter a spindle rotation command, and specify whether to enable or disable constant surface speed control.
3. No feed rate command needs to be specified in startup blocks, because feed rate commands are included in the machining cycles. However, specify a command for feed per revolution or feed per minute as required. For example, operator know-how on executing milling in pocketing by feed per revolution can be used.
4. Enter an M-code for switching between the spindle and C-axis, and a command for C-axis reference position return as required.
5. Enter M-codes such as for coolant ON/OFF as required.
6. Upon completion of machining cycle, the tool always returns to the position it was before the start of the machining cycle. By setting bit 7 (ESC) of parameter No. 27002 to 1, the tool can be prevented from returning to the position before the start of the machining cycle.

6.6.2 Machining cycles

1. Specify an I point coordinate not as a distance from point R but as a coordinate value.
2. Figure data should be entered as a subprogram for utilization in roughing, finishing, and chamfering. This eliminates the need to enter figure data each time.
3. In machining of a projected portion (island figure), efficient machining can be achieved by using pocketing with an island instead of using contouring. In this case, specify a blank as the outer wall of a pocket.
4. In contouring (side facing), enter a start point figure and end point figure so that both figures contact each other. This means that by starting not from a corner of a figure but from an intermediate point on a straight line, undercutting due to approaching the figure and retraction can be eliminated.

7 Machine compositions

A machine composition is a collection of settings that define a particular CNC system to be simulated. It defines the machine type to be milling or turning. It also defines the machine axis configuration.

You can create multiple machine compositions and switch between them as necessary. The instructor may predefine a machine composition, select options, set parameters and load part programs and distribute that machine composition to students as a platform for exercises or homework assignments.

Machine compositions may be established to closely match CNC on actual machines, CNCs not actually available to expand the student's experience, or to present problem solving situations.

The files that represent the machine composition can be archived and distributed freely. The Machine Composition Setting tool creates these five files when a new or copy machine composition action is taken.

Each CNC system to be simulated can be customized using:

- The Machine Composition Setting tool
- The Option Settings tool
- CNC parameters

7.1 Machine Composition setting tool

The Machine Composition Setting tool allows you to specify the following parameters depending on the CNC model:

CNC	System Type	Axes	MGi
31i-MODEL A	Only at creation	Only at edit	Creation and edit
30i/31i-MODEL B	Only at creation	Only at edit	Creation and edit
0i-MODEL D	Only at creation	n/a	Creation and edit
35i-MODEL B	n/a	Creation and edit	n/a

The machine composition tool can create, copy, edit and delete machine compositions. NCGuide does not have to be running to use the machine composition tool. The USB or network license key is necessary to use the Machine Composition Setting tool.

The Machine Composition Setting tool also makes it possible to select one machine composition as the default – it will be used whenever the NCGuide simulator is started for that CNC model. Alternatively, the user can be presented a list of machine compositions to choose from when the NCGuide simulator is started for a CNC model.

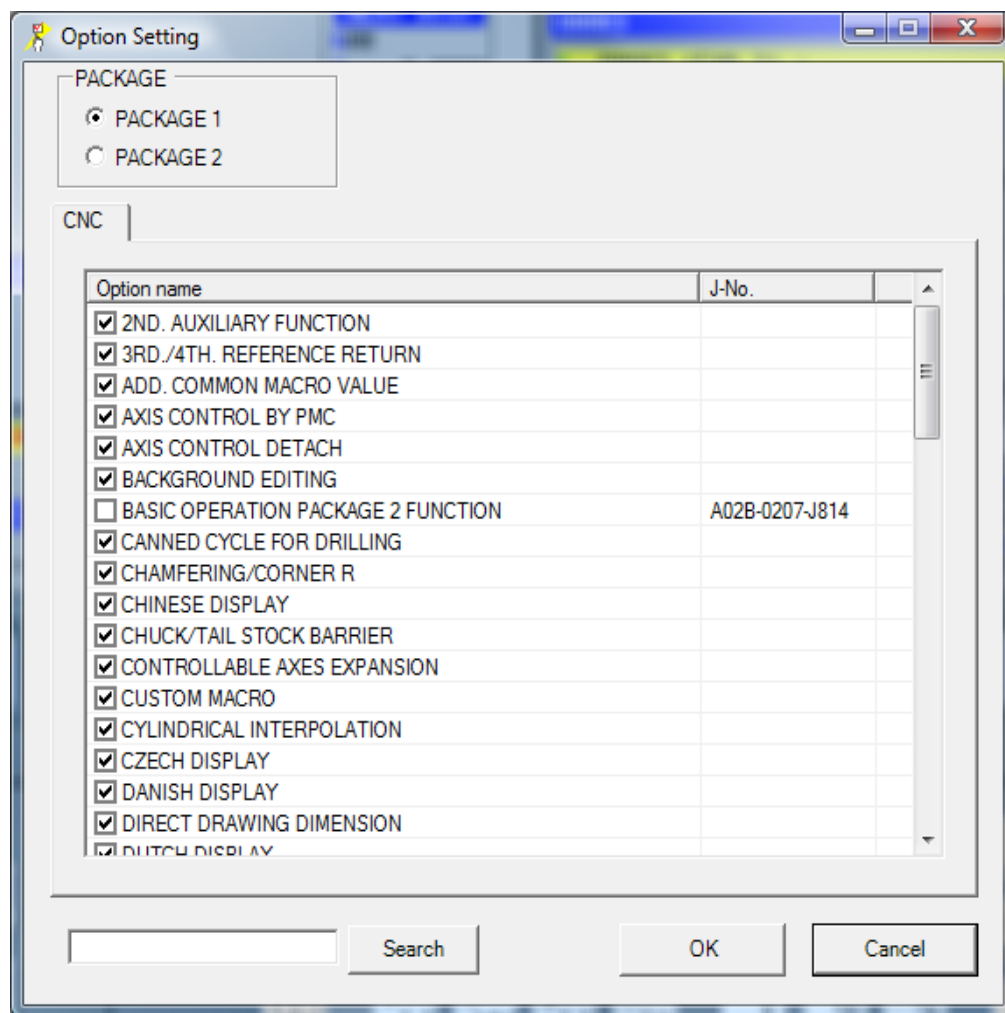
7.2 Option Setting

In the NCGuide Academic package, a basic option set is pre-initialized, therefore the following operations will typically not be necessary. However, sometime an options may have to be activated, for example, when MANUAL GUIDE i is activated using the Machine Composition Setting tool.

The options available are dependent on the machine model.

NCGuide must be running to change the option settings for a particular machine composition. In the procedures below, make sure you select the Option Setting tool for the CNC model that is running.

In NCGuide options are activated and deactivated using the Option Setting tool.



7.3 Parameters

Setting parameters is exactly the same in NCGuide as on the real CNC. Refer to the relevant parameter manual for the specific control type. The parameters set in NCGuide as shipped should be fine for most situations.

8 NCGuide Academic packages

8.1 System requirements

Operating systems

- Windows 2000 (Service Pack 4)
- Windows XP Home/Professional Edition
- Windows Vista Business (Service Pack 2)
- Windows 7 Professional

The following packages are also required

- .NET Framework 1.1 (Service Pack 1) *
- .NET Framework 2.0 (Service Pack 1)

Hardware system

- CPU - Pentium4 1.3GHz or higher (Pentium4 2GHz or higher recommended) or Intel Core Duo 1.83GHz or higher
- Memory - 256 MB or greater
- Free hard disk space - 800MB or greater (about 200MB per CNC model)
- Display resolution - 1280 x 1024 (SXGA) or higher
- DVD drive
- USB port

Windows NT, Windows 2000, and Windows XP, Windows Vista are trademarks of Microsoft Corporation in the United States.

Microsoft, Windows, and .NET Framework are registered trademarks of Microsoft Corporation in the United States.

Pentium is a registered trademark of Intel Corporation in the United States.

8.2 Differences from the CNC in operation

The functions of NCGuide Academic packages differ from the actual CNC as follows:

- Only EDIT, MEM, and MDI modes are selectable.
- The following functions are not supported.
 - Functions requiring the hardware of the CNC or related to it
 - Data server
 - Serial communication
 - Ethernet communication
 - Servo/spindle control
 - Option board functions such as customers' and C language boards
 - Operation History
 - Maintenance screens cannot be displayed or manipulated.
 - FANUC PICTURE
 - C-executor
 - Macro Executor
 - PMC simulation
 - Machine Signal Simulation
- Interpolation commands other than G00, G01, G02 and G03 approximate to a G01 command for execution.
- The cycle time of part program operation differs from that of the actual CNC.

9 Summary

Training students on the latest FANUC CNCs is challenging. Class size, safety concerns, CNC and machine tool costs typically limit the amount of hands experience available to the individual student. NCGuide and NCGuide Academic packages provide a realistic operation and part programming environment at a fraction of the cost of using a real machine tool. Comprehension and retention is enhanced as students perform repetitive hands-on exercises in an ergonomically friendly environment - away from the noise of the workshop. Operator, G-code programming and CAM programming students can practice common procedures and test part programs without risks to people, tooling or machines.

FANUC America Corporation
1800 Lakewood Boulevard
Hoffman Estates, IL 60192
888-FANUC-US (888-326-8287)
Find more information at www.fanucamerica.com

Technical data is subject to change without prior notice. No part of this document may be reproduced in any form. All rights reserved.
© 2013 FANUC FA America Corporation