

CNC TURRET PUNCH PRESS
VIPROS-255 ^{US}

OPERATOR'S MANUAL

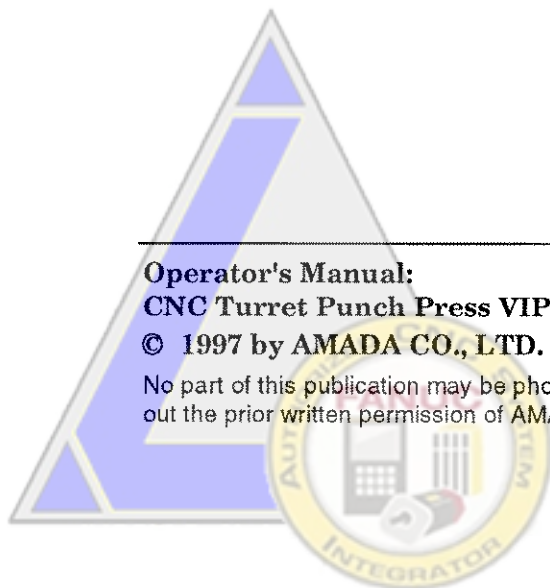
LAPIS SERVICES
(219) 464-9131



Amada Support,
AMADA And
Fanuc Control Upgrades.

LAPISINC.COM VIP255 OPE US-E01-199703

PREFACE Read this manual carefully to obtain a thorough knowledge of the machine and its installation, operation, and maintenance. Correctly operate the machine as described in the manual to prevent personal injuries and machine damage. Do not operate the machine by guesswork. Keep the manual at hand and refer to it whenever you are not sure of how to perform any of the procedures.



LAPIS SERVICES
(219) 464-9131

Operator's Manual:
CNC Turret Punch Press VIPROS-255
© 1997 by AMADA CO., LTD.

No part of this publication may be photocopied or otherwise reproduced without the prior written permission of AMADA CO., LTD.

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

Printed in Japan

CONTENTS

Safety Rules & Safety Functions

Safety rules	vi
Safety functions	x

General Information

Identification of machine parts	xii
Specifications	xiii

Part I

Installation

Location & foundation	I-2
Lifting & placing	I-2
Cleaning	I-3
Leveling	I-3
Air connection	I-4
Electrical connection	I-5
Hydraulic & cooling water connections	I-6
Confirming electrical connection	I-7
Floor plan drawing	I-8

Part II

Controls

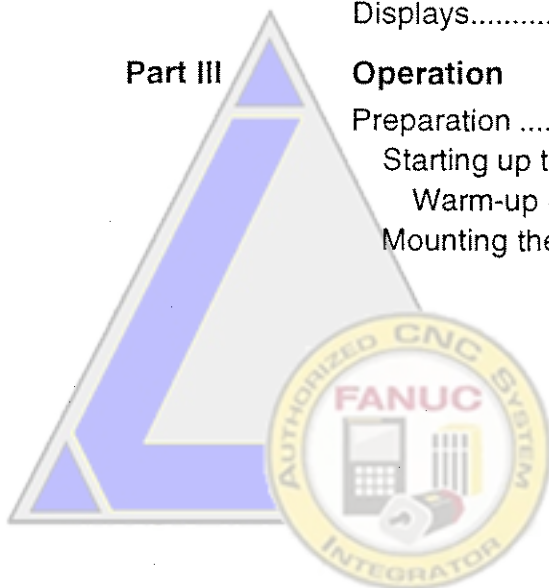
Electrical control cabinet	II-2
Main control panel	II-4
Subcontrol panel "A"	II-12
Subcontrol panel "B"	II-13
Auxiliary control stand	II-14
Foot switch & interface	II-16
Displays	II-17

Part III

Operation

Preparation	III-2
Starting up the machine	III-2
Warm-up operation	III-3
Mounting the tool	III-3

(Continued on next page.)



LAPIS SERVICES

(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

Operation.....	III-4
MEMORY mode operation.....	III-4
MDI mode operation.....	III-7
REMOTE mode operation.....	III-10
MANUAL mode operation.....	III-12
Punching in the MANUAL mode.....	III-12
PHNC switch settings.....	III-13
Operation interruption.....	III-15
Shutting down the machine.....	III-18

Part IV

Program Management

Program management.....	IV-2
Registration.....	IV-2
Memory capacity.....	IV-2
Setup for program management.....	IV-3
Reading from an input device.....	IV-3
Outputting to an output device.....	IV-4
Direct programming & registration.....	IV-5
Search.....	IV-6
Check.....	IV-6
Deletion.....	IV-7
Program editing.....	IV-8
Editing.....	IV-8
Modifying a word.....	IV-8
Inserting a word or block.....	IV-8
Deleting a word or block.....	IV-8
Background editing.....	IV-9

Part V

PHNC Parameters

PHNC parameters.....	V-2
"FORMING" parameters.....	V-3
"MARKING" parameters.....	V-7
"KNOCKOUT" parameters.....	V-11
"SLITTING" parameters.....	V-15
"PUNCH-NIBBLE" parameters.....	V-19
"TIME DATA" parameters.....	V-22
"TOOLING DATA" parameters.....	V-24

Part VI

Alarms

CNC alarms.....	VI-2
PHNC alarms.....	VI-34
Machine alarms.....	VI-36
Self-diagnosis (CNC).....	VI-41
Self-diagnosis (PHNC).....	VI-41

Part VII

Maintenance

Daily maintenance	VII-2
Periodical maintenance	VII-4
Every week	VII-4
Every month	VII-5
Every 3 months	VII-5
Every 6 months	VII-8
Every 12 months	VII-9
Lubrication	VII-11
Major operation faults	VII-13
Fuses & motor breaker	VII-14
Inputting/Outputting custom macro programs	VII-15
Inputting/Outputting parameters	VII-17

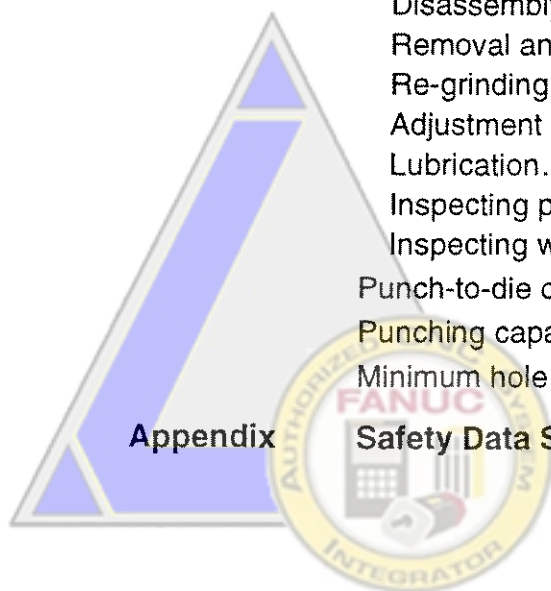
Part VIII

Tooling

Station arrangement	VIII-2
31-station turret	VIII-2
Tool types	VIII-3
Type A (1/2")	VIII-3
Type B (1-1/4")	VIII-4
Type C (2")	VIII-5
Type D (3-1/2")	VIII-6
Mounting and dismounting punches and dies	VIII-7
1/2" inner-track turret stations	VIII-7
1/2" outer-track, 1-1/4" , 2" and 3-1/2" turret stations	VIII-8
Auto-index turret stations	VIII-9
Keys and keyways in tools and turret stations	VIII-10
Keys and keyways in tools	VIII-10
Keys and keyways in turret stations	VIII-11
Maintenance of punches and dies	VIII-12
Disassembly and assembly of punches (Types A and B) ...	VIII-12
Removal and installation of punch tips (Types C and D)	VIII-13
Re-grinding punches and dies	VIII-14
Adjustment of punch and die height	VIII-14
Lubrication	VIII-16
Inspecting punches and dies	VIII-16
Inspecting worksheets	VIII-16
Punch-to-die clearance	VIII-17
Punching capacity	VIII-17
Minimum hole diameter	VIII-19

Appendix

Safety Data Sheet



Safety Rules & Safety Functions

SAFETY RULES

Observe these safety rules to prevent injuries and accidents.

- a)** Place a fence around the machine to remind you and others of dangers inside the fence. *The area enclosed by the fence is referred to as the hazardous area in this manual.*

The worksheet will thrust outward during operation — in the front of the machine during zero-returning after automatic repositioning or both in the front and the back if the worksheet is oversized.

We recommend additional installations of optional safety devices — pressure detection mats and optical sensors. They will stop the machine instantly upon detection of careless entry in the hazardous area.

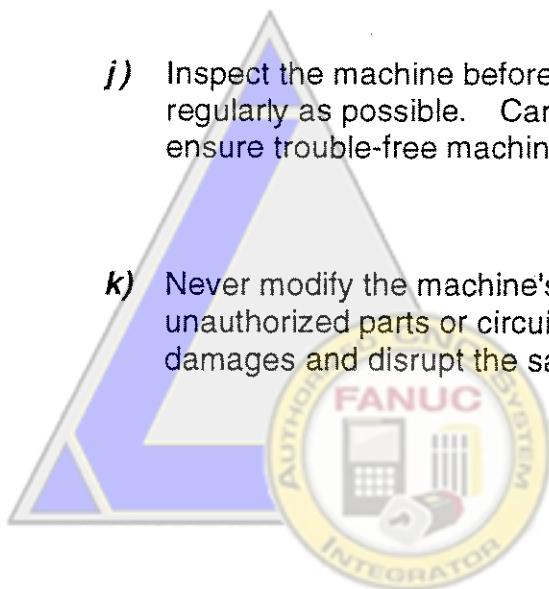
- b)** Have the machine operated by a single, trained person who has read this manual and acquired a thorough knowledge of the machine and its operation. If more than one authorized operator must be involved, coordinate their work to ensure utmost safety.

- c)** Don't wear loose clothing, a necktie, or a muffler when operating the machine. They can be extremely dangerous if any part of them is caught in the machine.

- d)** Observe the following punching precautions. If you are not sure of whether or not a particular worksheet can be punched, consult the AMADA engineer.

- Do not punch worksheets that exceed the nominal press capacity of the machine.
- Do not punch worksheets that exceed the maximum allowable thickness.

- Do not punch worksheets of glass, stone, or any other material that may scatter when punched.
 - Do not punch worksheets of wood, plaster, and any other material that generate powder or crumble when punched.
 - Do not punch worksheets that exceed the tool usage conditions and punching conditions.
- e)* Press the STOP button and turn the TOOL CHANGE keyswitch to ON when working inside the hazardous area around the machine.
- f)* Clear the hazardous area around the machine of people and obstacles before starting the machine — don't overlook the area behind the machine.
- g)* Don't touch the hydraulic unit and piping during and immediately after operation. The unit and piping become as hot as 70°C (158°F).
- h)* Don't leave the keys in the machine's keyswitches. Keep the keys in the custody of the supervisor when they are not in use.
- i)* Turn off the power and shut down the compressed air supply when carrying out a maintenance operation. Then padlock the machine circuit breaker switch lever. If the power is needed, prevent the machine from being started by mistake —press the STOP button and turn the TOOL CHANGE keyswitch to ON.
- j)* Inspect the machine before starting the day's work and as frequently and regularly as possible. Carry out the periodical maintenance operations to ensure trouble-free machine operations.
- k)* Never modify the machine's parts or electric circuits or change them with unauthorized parts or circuits. Doing so will cause machine disorders and damages and disrupt the safety of the machine and the operator.



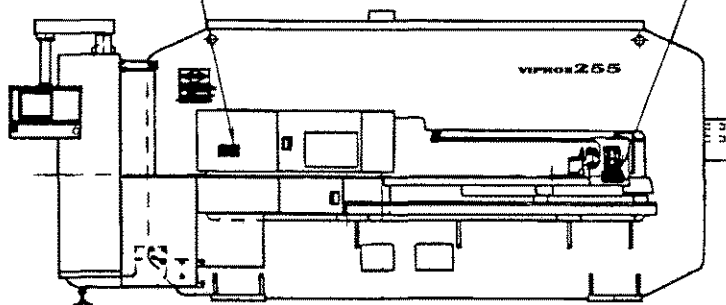
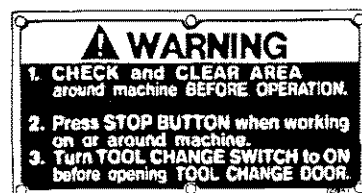
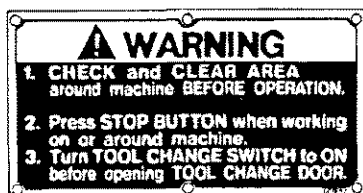
(219) 464-9131

Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

DANGER and WARNING plates

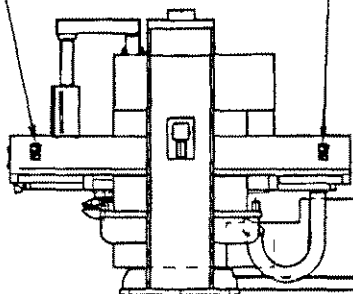
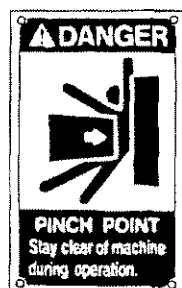
Keep the DANGER and WARNING plates well noticeable and never remove them.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM



Hazard seriousness level



DANGER

Indicates an imminently hazardous situation which, if not avoided, will result in death or serious injury.



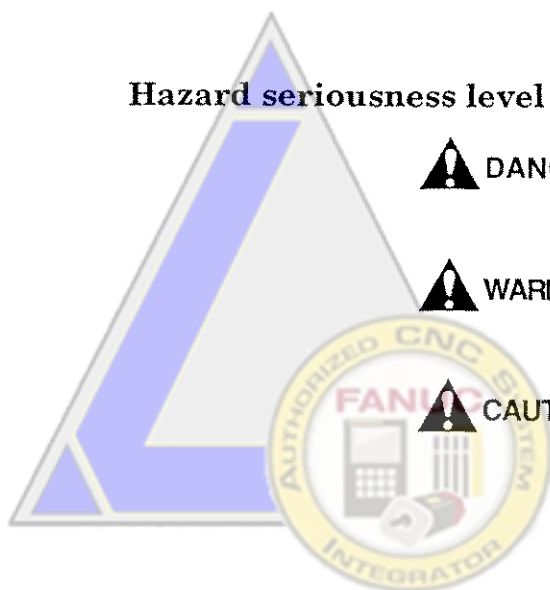
WARNING

Indicates a potentially hazardous situation which, if not avoided, could result in death or serious injury.



CAUTION

Indicates a potentially hazardous situation which, if not avoided, may result in minor or moderate injury.



LAPIS SERVICES

(212) 944-9131

Amada Support,

Reborn CNC

Fanuc Control Upgrades.

LAPISINC.COM

SAFETY FUNCTIONS

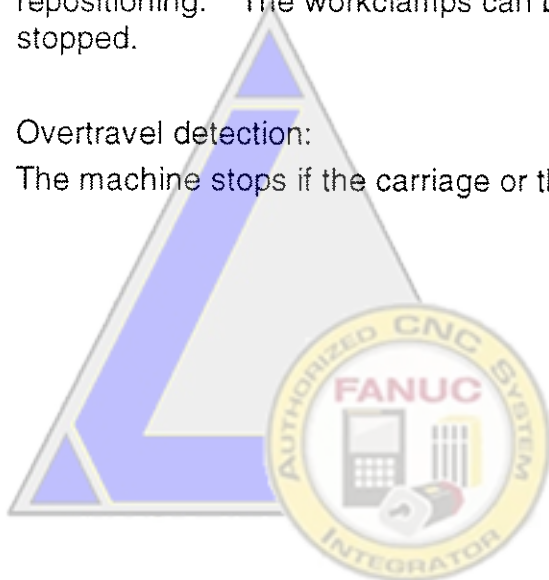
The following functions ensure the safe operations of the machine:

Operator safety

- Tool-change protection:
A switch is installed to lock the carriage and table during a tool change.
- Tool change door interlock:
The machine does not start if the tool change doors are not closed. The machine stops if any of the tool change doors is opened during operation.

Machine safety

- Servo-system protection:
The machine stops if an irregularity is detected in the servo-system, such as an overloaded servomotor.
- Stripping failure detection:
The machine stops if the worksheet is not stripped off the punch or if stripping is delayed.
- Workclamp/Workholder protection:
The machine stops temporarily if a workclamp enters an area in which it may interfere with the punch. The machine also stops temporarily if there is a danger of workclamp-and-workholder collision during a worksheet repositioning. The workclamps can be opened only when the machine is stopped.
- Overtravel detection:
The machine stops if the carriage or the table overtravels.



LAPIS SERVICES

(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

- Overtravel precheck:
An alarm is caused if any data which will make the carriage or table overtravel are detected. The detection is made automatically by the CNC in the MEMORY mode prior to program execution.
- Turret index pin & striker position detection:
Punching will not be performed if a turret index pin is not inserted or if the striker is not positioned over the proper turret track.
- X-gauge block protection:
The machine does not start if the X-gauge block is not lowered. The machine stops if the block is raised erroneously during operation.
- Air pressure detection:
The machine stops if the operating air pressure has been reduced below the required pressure.
- Hydraulic system protection:
The machine stops if the hydraulic pressure has been reduced below the required pressure or the oil temperature has risen extremely high.



LAPIS SERVICES
(219) 464-9131

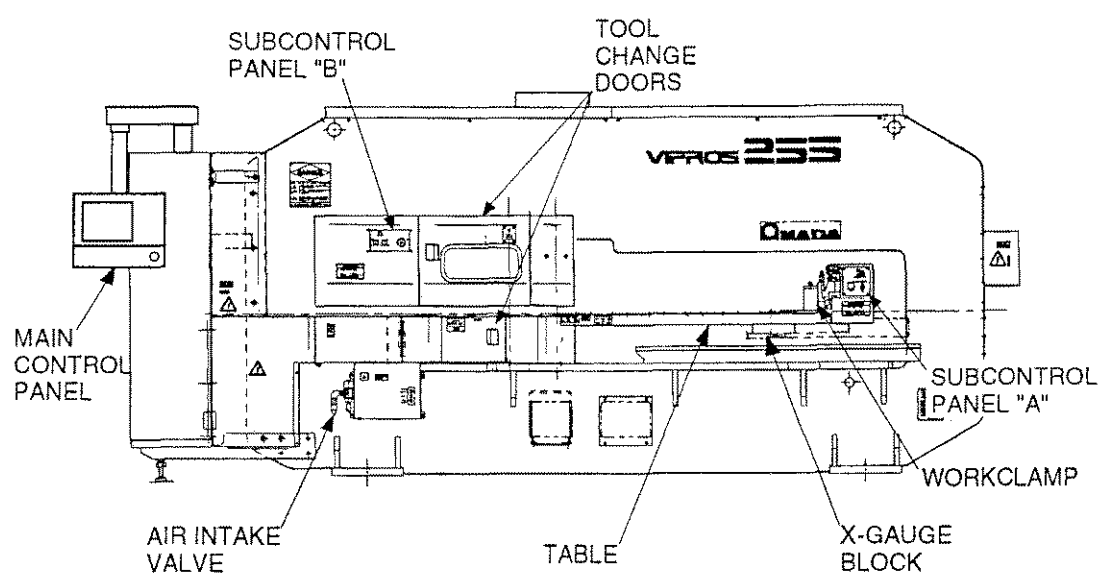
**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

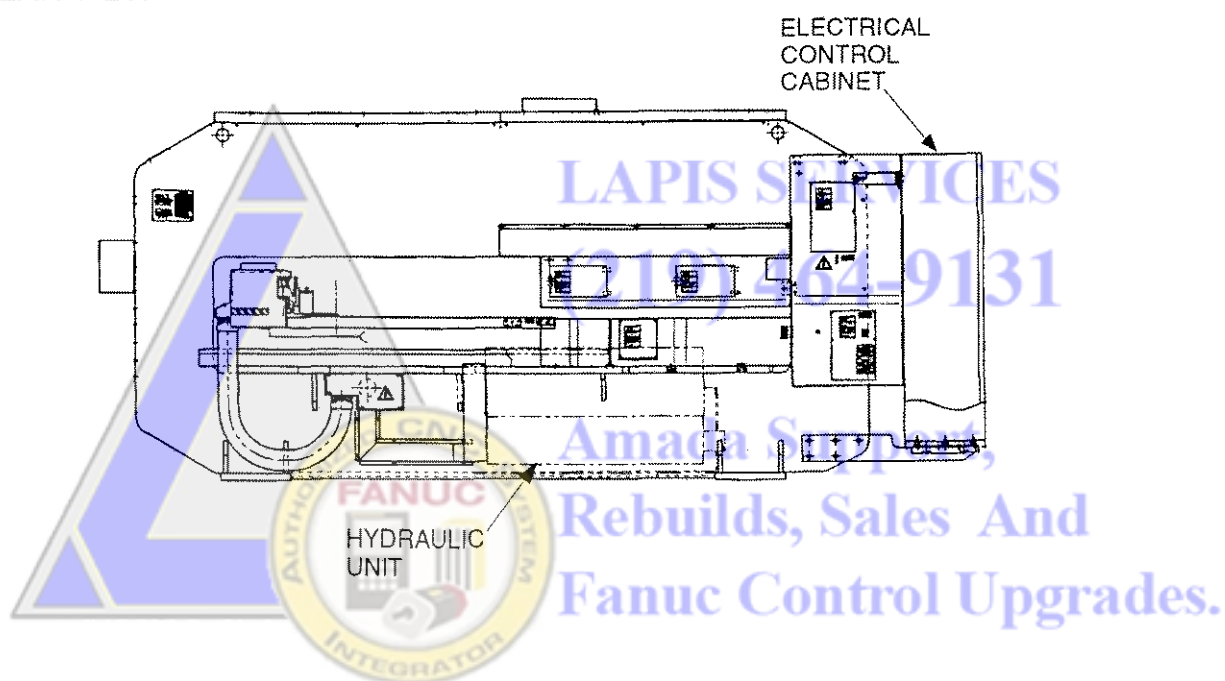
General Information

IDENTIFICATION OF MACHINE PARTS

FRONT VIEW



REAR VIEW



SPECIFICATIONS

Machine specifications

Press model	VIPROS-255
Press capacity	196 kN (20 metric tons or 22 U.S. tons)
Max. sheet size	1270 x 2540 mm (50 x 100")
Max. sheet thickness	6.2 mm (0.24")
Max. sheet weight	105 kg (231 lb) at axial feed-speed F4 62 kg (136 lb) at axial feed-speed F1
Y-axis travel	1270 mm (50")
X-axis travel	1270 mm (50")
Punching accuracy	±0.07 mm (0.003")* *±0.05 mm (0.002") in high-accuracy mode operation
Strokes per minute X	8 mm pitch: 410 strokes, 25.4 mm pitch: 340 strokes
Strokes per minute Y	8 mm pitch: 380 strokes, 25.4 mm pitch: 280 strokes
Max. ram stroke length	40 mm (1.57")
Max. feed speed X	65 m/min (213 ft/min)
Max. feed speed Y	50 m/min (164 ft/min)
Turret speed	35 rpm
Power supply	22 kVA
Air supply	250 liters/min (8.8 ft ³ /min)
Operating air pressure	0.49 MPa (5.0 kg/cm ² or 72 psi)
Width	5030 mm (198")
Depth	2630 mm (104")
Height	2160 mm (85")
Weight	9.2 metric tons (10.1 U.S. tons)

NC specifications

NC model	FANUC 18-PC
Controlled axes	4 axes simultaneously controlled
Program code	ISO or EIA
Program format	Address and data
Min. dimension	0.01 mm (0.001"), 0.01°
Memory capacity	160 m (520 ft) in tape length

Operating environment

Ambient temperature	0 to 40°C (32 to 104°F)
Max. relative humidity	75 % without dew condensation
Illumination	Over 300 lux.

Never use the machine in an explosive environment.



LAPIS SERVICES

(219) 464-9131

Amada Support,
Rebuild, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Sound level

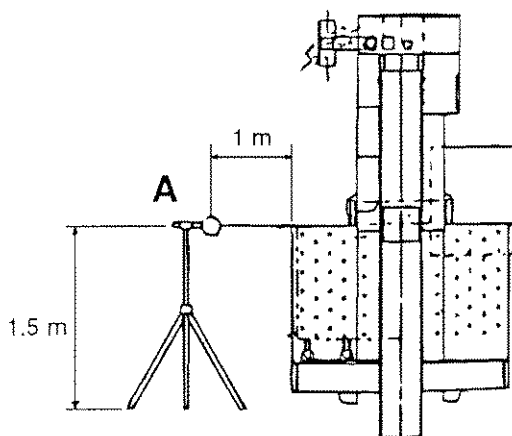
Sound level (measured at position A shown below):

$L_A \text{ eq} = 78.3 \text{ dB (A)}$	No load (Carriage and table move according to program without punching.)
$L_A \text{ eq} = 90.9 \text{ dB (A)}^*$	Mild steel sheet SPCC (JIS), measuring 2.3 mm in thickness and $914 \times 914 \text{ mm}$ in size, is punched by tool without shear angle
$L_A \text{ eq} = 86.5 \text{ dB (A)}$	Mild steel sheet SPCC (JIS), measuring 0.8 mm in thickness and $914 \times 914 \text{ mm}$ in size, is punched by tool without shear angle
$L_P \text{ peak} < 130 \text{ dB}$	Mild steel sheet SPCC (JIS), measuring 6.0 mm in thickness and $914 \times 914 \text{ mm}$ in size, is punched by tool without shear angle

Measuring height: 1.5 m from floor

Measurements made: according to ISO 8500

*90.9 dB (A) > 85 dB (A)



Machine acoustic power level:

LWA = 117.0 dB(A) (501 mW) in full load.

Determination in accordance with ISO 9614-1.

Advice for decreasing sound level

- Keep the operator away from the machine.
- Use a lower press ram speed and lower carriage and table feed speed.
- Use tools with shear angle.
- Use tools of smaller size.
- Enclose the machine with soundproof walls or the like.

It is also recommended to wear ear protectors.

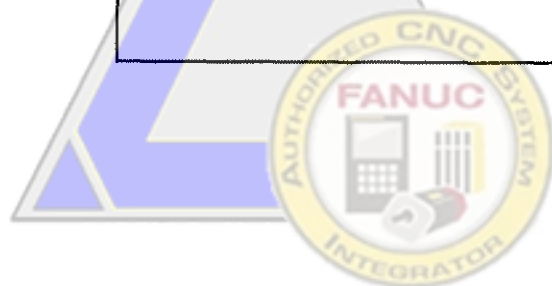
LAPIS SERVICES
(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

LIST OF THE REGULATIONS AND STANDARDS TAKEN IN REFERENCE IN THE CONCEPTION AND CONSTRUCTION OF THE PUNCHING MACHINE

Subject	French standard	European standard standardized to the project
Machine directive/Regulations and procedures	Act 92 765/67	89/392 CE
Graphic symbols used in the doc/diagrams		CEI 03201 to 03211
Graphic symbols of signals		CEI 417
Classification of the protection index on envelops	C 20010	CE 529/76
Colors of lights and buttons	C 20070	CEI 73/84
Conventional rotation of the movements	C 20090	CEI 447/74
Separation transformer		EN 60742
Electric equipment of machines	C 79130	EN 60204.1
Graphic, hydraulic and pneumatic symbols	E 04056	
Emergency stop		EN 418/92
Technical prevention of accidents and mechanical risks	E 09001	
Protection of upper organs moving away	E 09010	EN 294/92
Elements to take into account in the T.M.conception	E 09020	
Electrical control footswitches	E 09031/32	
Two-hand control	E 09034/35	
Locking and interlocking device	E 09051	
Machines stopping times	E 09052	
Safety in T.M.conception		EN 292.1/2
Control organs rotation of the T.M.	E 60031	
Specifications symbolization on T.M.	E 60032	
Measurement of the acoustic pressure at working station	S 31048	
Measurement of the acoustic power with intensimetry		ISO 9614.1
List of graphic symbols for industrial	X 05005	ISO 7000
Colors and safety signals	X 08003	
Ergonomic principles	X 35001	
Signalization applicable to working station	X 35101	ISO 3864
Behaviours and dimensions for the human being at work	X 35104	
Operation organs	X 35105	
Effort limits	X 35106	
Access dimensions to machines	X 35107	
Technical documentation redaction	X 60200	
Electro-magnetic compatibility directive	Act 92/31	89/336/EEC
Body approach speed		pr EN 999
Minimum distance		EN 349
Risk analysis		pr EN 1050/94
Electro-magnetic compatibility		EN 50081.2
Measurements of radio-electric interferences		EN 55011/91 EN 50.082-2-95 EN 61-4-2/4/8



Amada Support
Rebuilds, Sales And
Fanuc Control Upgrades.



LAPIS SERVICES
(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

Part I

Installation

Location & foundation	I-2
Lifting & placing	I-2
Cleaning	I-3
Leveling	I-3
Air connection	I-4
Electrical connection	I-5
Hydraulic & cooling water connections	I-6
Confirming electrical connection	I-7
Floor plan drawing	I-8

The machine should be installed according to the instructions described in this Part as well as by following the instructions of the AMADA engineer.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

LOCATION & FOUNDATION

Select a location where ample space can be provided around the machine, air compressor and tool storage after their installation, as well as for work handling and maintenance operation. The air compressor should be installed within a distance of 5 m (15') from the machine.

The location must be isolated from any nearby equipment that produces dust or vibration and at least 3 m (10') away from any equipment that produces electrical interference, such as welders, drilling machines, sanders, and grinders.

Provide a firm, level concrete floor for the installation according to the floor plan provided by AMADA. (See page I-8.)

LIFTING & PLACING

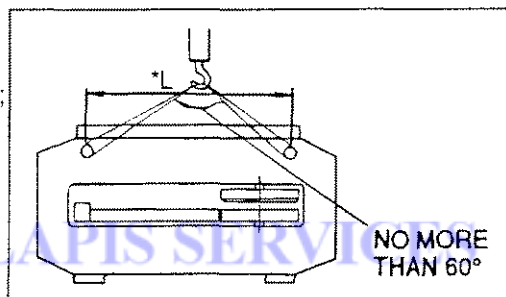


WARNING

- Have qualified personnel perform the lifting work. And use a crane and wire ropes that have ample capacities. And don't spread the wire ropes exceeding 60 degrees when slinging and lifting.

When lifting and carrying the machine, use a crane and wire ropes that have ample capacities. The machine weighs 9.2 metric tons (10.1 U.S. tons), including the hydraulic unit (approx. 650 kg or 1433 lb).

To lift the machine, attach wire ropes to the four ribs located below the top covers; balance, lift, and place the machine carefully while protecting it from impacts.



*L (distance between ribs) = 3175 mm (125")



Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

CLEANING

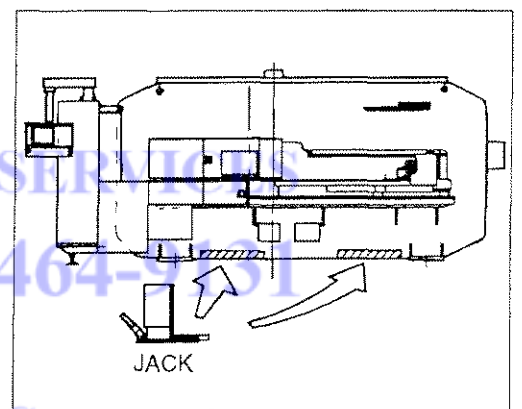
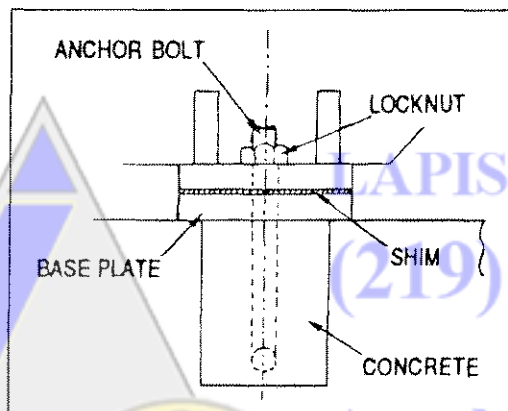
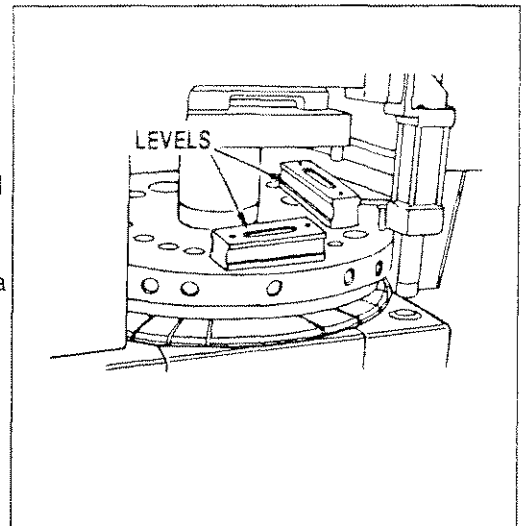
After the machine has been installed, remove the rust preventive from its surfaces by using cleaning oil and then apply machine oil to the parts that may rust.

NOTICE

- Never use a scraper or solvents which may damage the coated surfaces of the machine.

LEVELING

Place spirit levels on the top surface of the upper turret disk in the X- and Y-axis directions. Insert the accessory shims between the machine frame base and the base plate to level the machine. Lift the machine (see lower right figure) with a hydraulic jack to insert the shims. Do not lift the bottom of the electrical control cabinet. The machine must be level to within ± 0.05 mm/m in both the X- and Y-axis directions.



When the leveling of the machine is finished, insert the anchor bolt through the holes in the base plate and the machine frame base from below, fit the locknut on the anchor bolt, and fix the anchor bolt with concrete. When the concrete is completely dried, securely tighten the locknut to fix the machine.

AIR CONNECTION

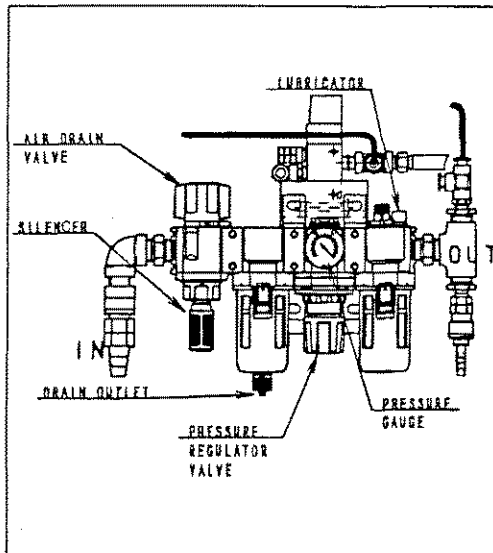
The machine requires clean, dry air of 250 liters/min (8.8 ft³/min) with an operating pressure of 0.49 MPa (5.0 kg/cm² or 72 psi) and, therefore, it is necessary to use an air compressor with a capacity of over 2.2 kW (3 HP). Install the compressor within a distance of 5 m (15') from the machine in order to minimize the loss of pressure.

Use a pipe or hose with an inside diameter of 1/2" to connect the air compressor with the machine and connect the pipe or hose to the 1/2" air inlet of the air pressure control unit.

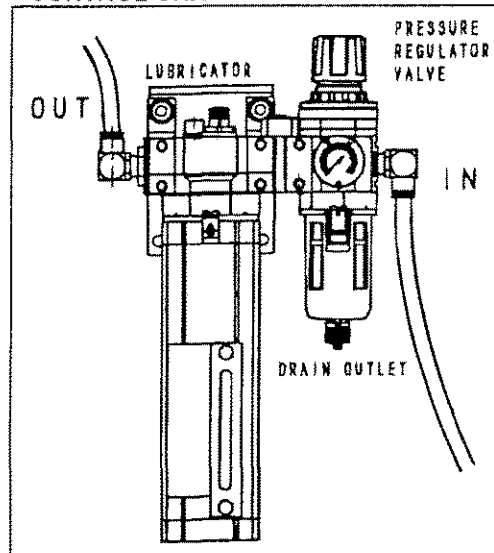
NOTE

- If the compressor must be installed more than 5 m (15') away from the machine, use a pipe or hose with an inside diameter of more than 3/4".

MAIN AIR PRESSURE CONTROL UNIT

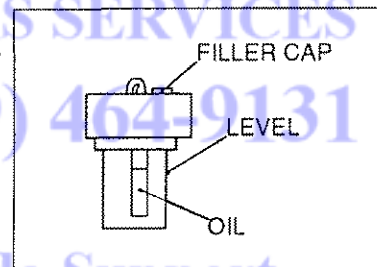


STRIKER AIR BLOW PRESSURE CONTROL UNIT



Set the operating pressure at 0.49 MPa (5.0 kg/cm² or 72 psi) for both the main air system and the striker air blow system by pulling out the knob of the regulator valve and turning the knob clockwise to increase the pressure or counterclockwise to decrease it — push in the knob when the pressure is set.

Keep the level of oil in the lubricator above the line marked on the lubricator of each air pressure control unit. The recommended oil is ESSO Teresso 32, SHELL Tellus Oil C32, or MOBIL DTE Oil Light. Refer to Part VII, Maintenance, for adjustment of the amount of oil drops.



Drain the water accumulated in the air filter reservoir of each air pressure control unit by pushing the drain valve located at the bottom of the air filter.

ELECTRICAL CONNECTION



WARNING

- Have a qualified electrician carry out all electrical work in order to prevent injuries or accidents.

Use a power supply that is not shared by a welder, drilling machine, sander, or grinder, which can cause electrical interference. This machine requires a power supply of 22 kVA and the allowable voltage fluctuation is within $\pm 10\%$. Use a power cable with conductors each having a proper cross section. Connect the power cable conductors to the input terminals in the electrical control cabinet. Connect the ground conductor to the ground terminal. The lead of the ground conductor should be longer than those of the power cable conductors. The connection must be checked for correctness later after completing hydraulic and cooling water connections.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

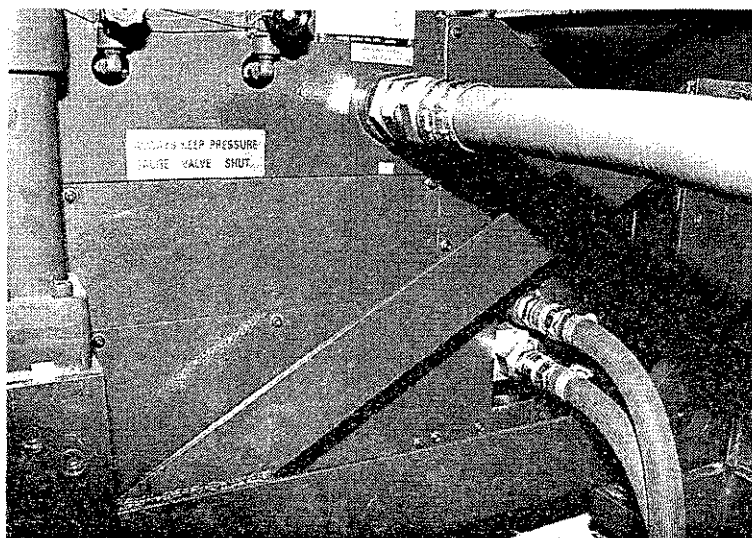
LAPISINC.COM

HYDRAULIC & COOLING WATER CONNECTIONS

Connect the hydraulic hoses to the hydraulic unit as described below.

NOTICE

- Don't place the hoses where they come in contact with piping or wiring.



A cooling water unit with a cooling capacity of over 11.6 kW (10000 kcal/h) is required to cool the hydraulic oil. Connect the cooling water pipes to the hydraulic unit as shown above.

NOTE

- Use pipes with an inside diameter of over 27.6 mm and bushings to minimize resistance even though the size of the water inlet and outlet on the hydraulic unit is PT 3/4". Use hoses with an equivalent size if they are used instead of pipes.

Detach the side cover from the hydraulic unit and fill the hydraulic oil tank with a recommended oil. The required amount of hydraulic oil is 150 liters (40 U.S. gal).

Recommended oils:

AMADA Hydraulic
Press TV-110X or
MOBIL NZ46



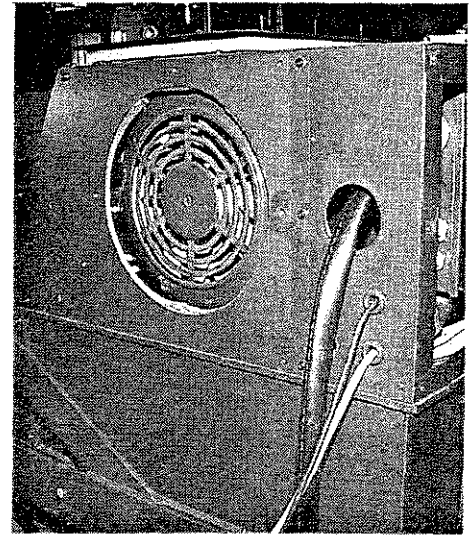
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

CONFIRMING ELECTRICAL CONNECTION

Confirm that the electrical connection has been properly made by inspecting the rotating direction of the hydraulic pump cooling fan in the following manner after air, hydraulic, and cooling water connections have been completed:

- 1 Turn on the shop power, the machine circuit breaker switch, and then the CNC.
- 2 Check to see if the cooling fan is rotating in the direction marked by an arrow.
- 3 If the cooling fan is rotating in the opposite direction, immediately turn off the CNC, the machine circuit breaker switch, and then the shop power.



- 4 Interchange any two of the three wire connections (power cable) on the input terminals in the electrical control cabinet.
- 5 Confirm that the cooling fan is now rotating in the correct direction.

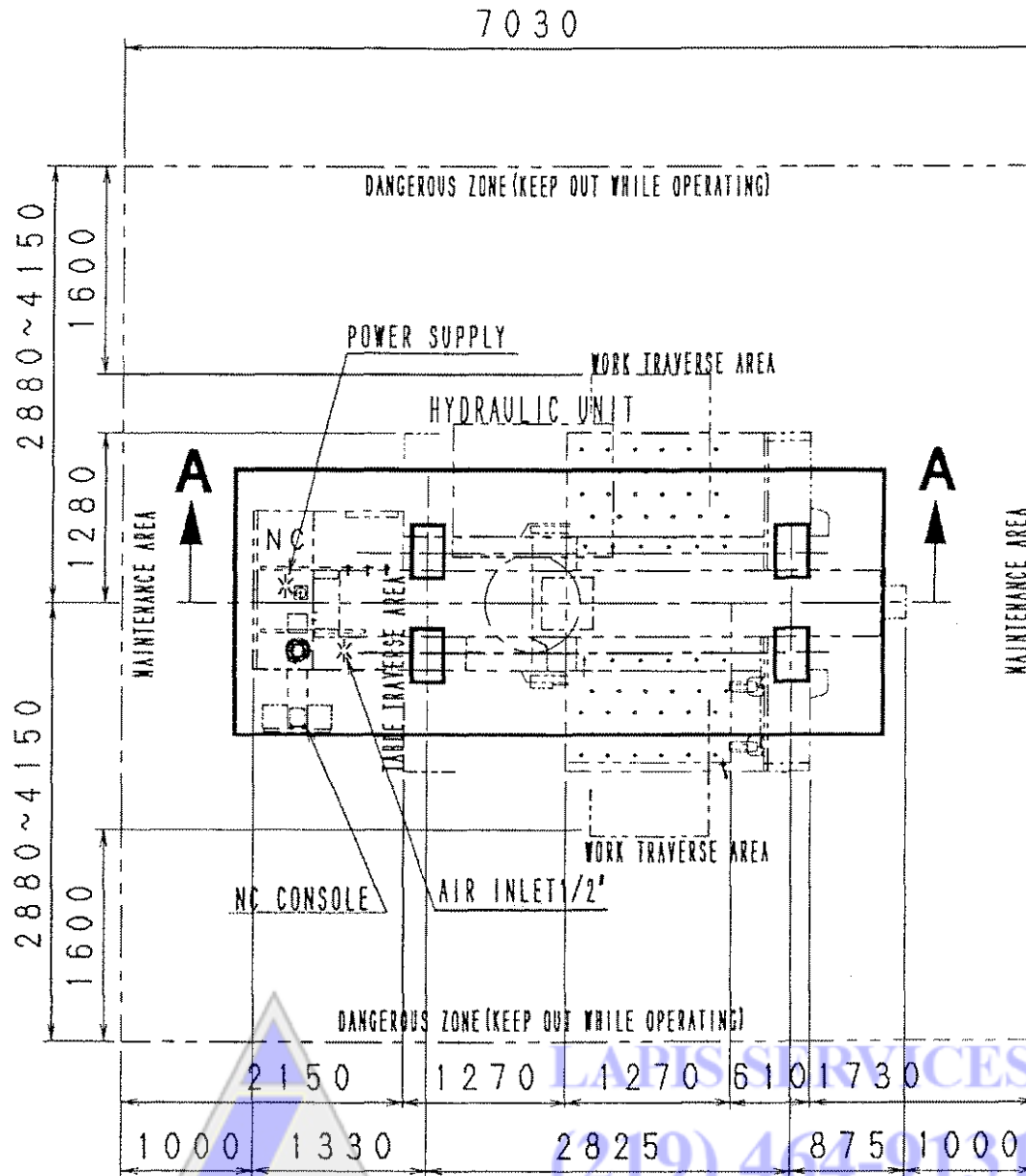


LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

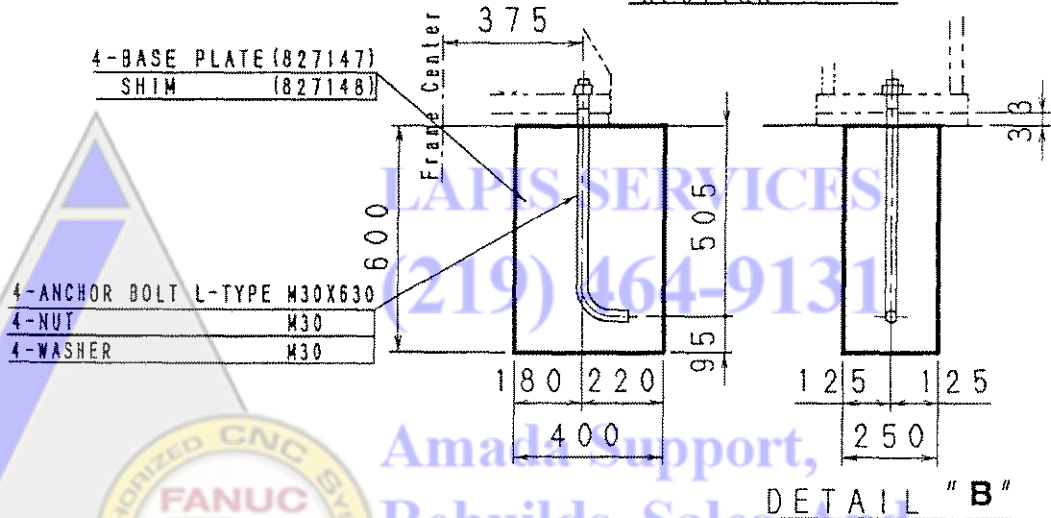
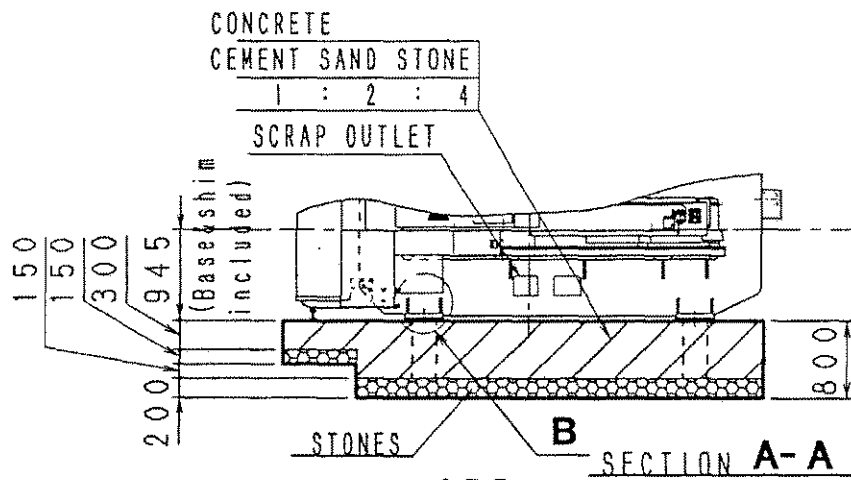
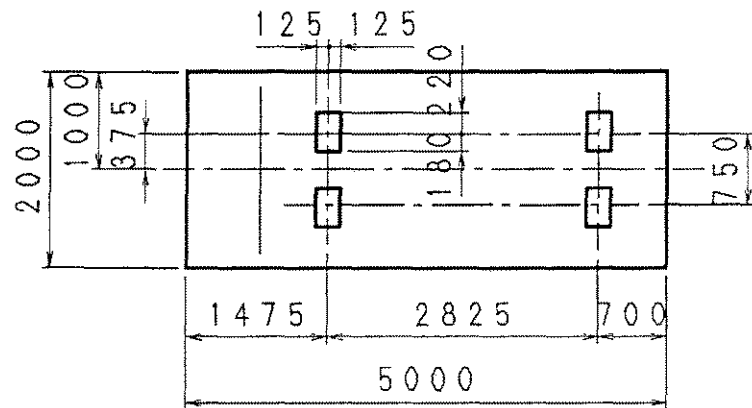
FLOOR PLAN DRAWING



AUX. CONTROL STAND

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.





- 4-ANCHOR BOLT L-TYPE M30X630
- 4-NUT M30
- 4-WASHER M30

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.



LAPIS SERVICES
(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM 

Part II

Controls

Electrical control cabinet.....	II-2
Main control panel.....	II-4
Subcontrol panel "A"	II-12
Subcontrol panel "B"	II-13
Auxiliary control stand.....	II-14
Foot switch & interface.....	II-16
Displays.....	II-17

The switches and others which are used to operate and control the machine are described in this Part.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

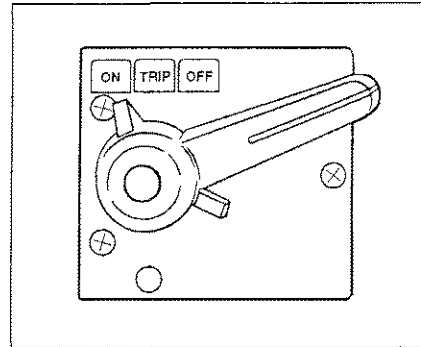
LAPISINC.COM

ELECTRICAL CONTROL CABINET

Machine circuit breaker switch

This switch turns the power on and off for the machine and the CNC unit. It will be turned off automatically if an overcurrent is supplied.

The switch lever can be locked with the padlock when the switch is turned off.



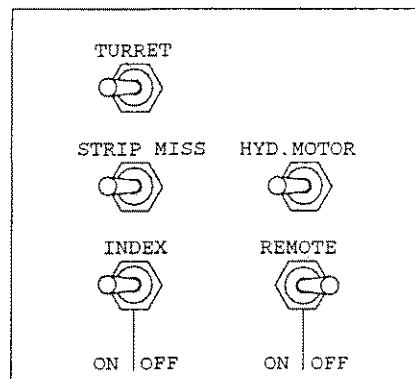
Switches inside the electrical control cabinet

TURRET switch

The turret will be rotated as commanded when this switch is turned to ON. Keep the switch turned to ON normally.

STRIP MISS switch

The stripping failure detection function will be effective when this switch is turned to ON. Keep the switch turned to ON normally.



HYD. MOTOR switch

The hydraulic pump motor will be started as soon as the NC READY light on the main control panel is lighted when this switch is turned to ON. Keep the switch turned to ON normally.

INDEX switch

The auto-index device will be rotated as commanded when this switch is turned to ON. Keep this switch turned to ON at all times.

REMOTE switch

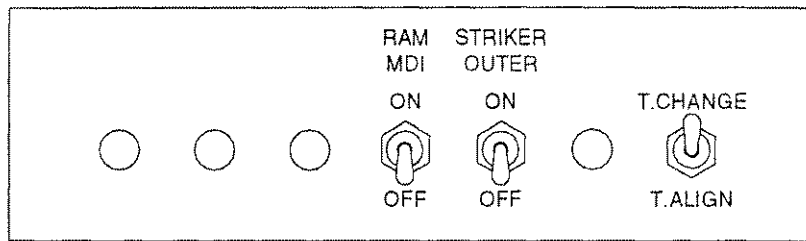
The remote operation can be performed when this switch is turned to ON.

LAPIS SERVICES

(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM



RAM MDI switch

The ram can be stopped at random when this switch is turned to ON. Keep the switch turned to OFF normally.

STRIKER OUTER switch

This switch is used for maintenance purposes. Keep the switch turned to OFF.

T.CHANGE / T.ALIGN switch

The hydraulic pump motor will be stopped when the TOOL CHANGE switch on the main control panel is turned to ON, and when this switch is turned to T.ALIGN. Keep the switch turned to T.CHANGE normally. The T.ALIGN position is only to be used for maintenance purposes.

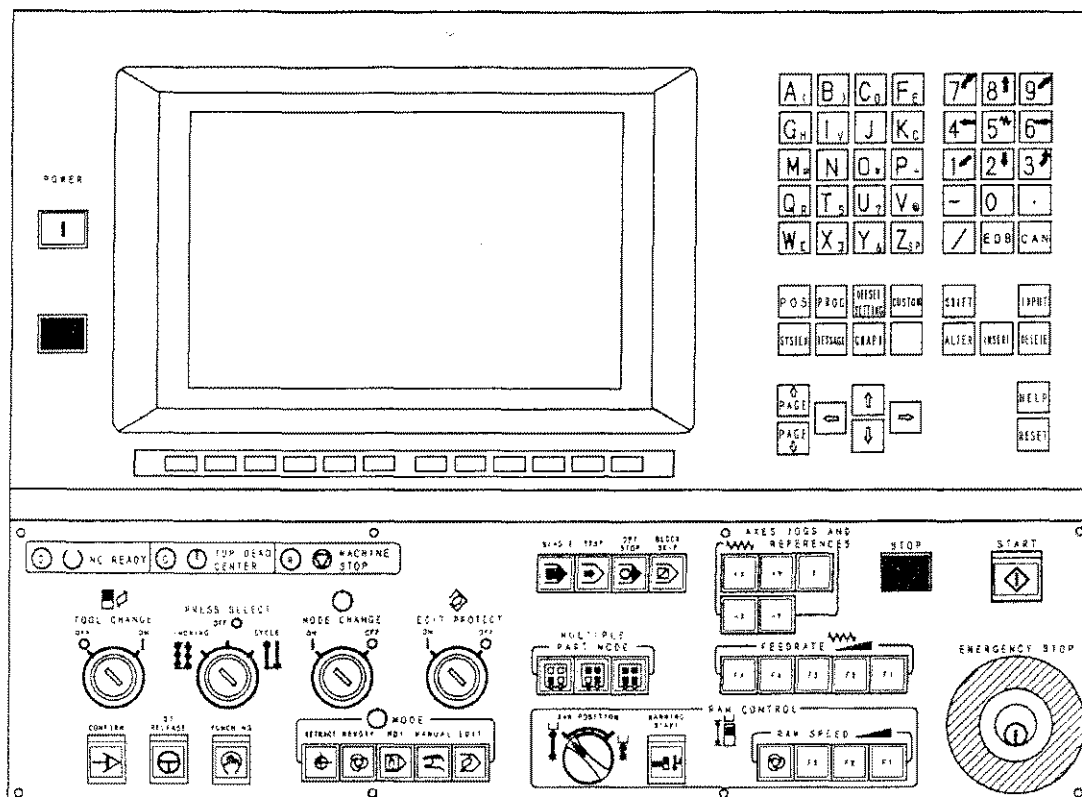


LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

MAIN CONTROL PANEL



POWER ON & OFF buttons

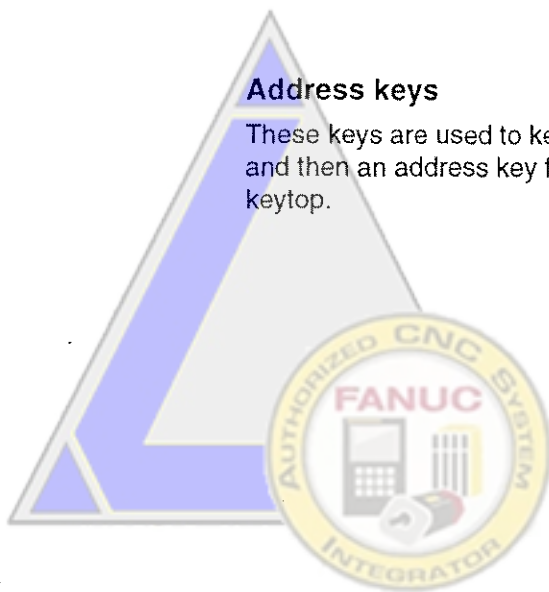
These buttons are used to turn the CNC unit on and off.

Softkeys

There are ten unlabeled keys below the screen. Their current functions will be defined and displayed at the bottom of the screen during operation. If a key is not defined, that key has no function.

Address keys

These keys are used to key-in address characters. Press the Shift key and then an address key for the character shown on the lower right keytop.



key-in address characters. Press the Shift key for the character shown on the lower right

Amada Support, Rebuilds, Sales And Fanuc Control Upgrades.

LAPISINC.COM

Numeric keys

The numeric and decimal point keys are used to key-in numeric data. The symbols shown on the left of the keytops are not available.

Slash key

This key is used to key-in a slash character.

EOB key

This key is used to key-in an end-of-block code(;).

Cancel(CAN)key

This key is used to erase a keyed-in character in the display's buffer area.

Shift key

Press this key in combination with an address key to key-in the character shown on the lower right keytop of the address key.

Input key

This key is used to enter the keyed-in data into memory.

Alter key

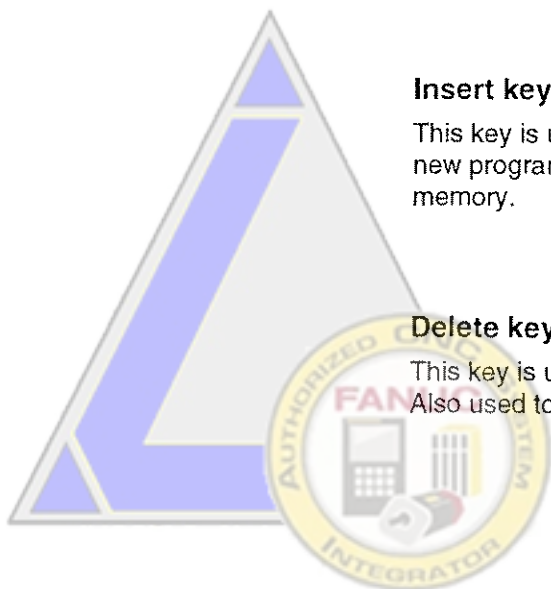
This key is used to change a word with a new word in the program in memory.

Insert key

This key is used to enter a program block into memory when creating a new program. Also used to insert a word or words in the program in memory.

Delete key

This key is used to delete a word or words in the program in memory. Also used to delete an entire program.



LAPIS SERVICES

(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

Display selection keys

These keys are used to select displays.

Position(POS)key

Shows the Position display.

Program(PROG)key

Shows the Program display.

Offset/Setting key

Shows the Offset/Setting display.

Custom key

This key is not used.

System key

Shows the System display.

Message key

Shows the Message display.

Graphic key

This key is not used.

Page()keys

These keys are used to change the page displayed on the screen.

Cursor()keys

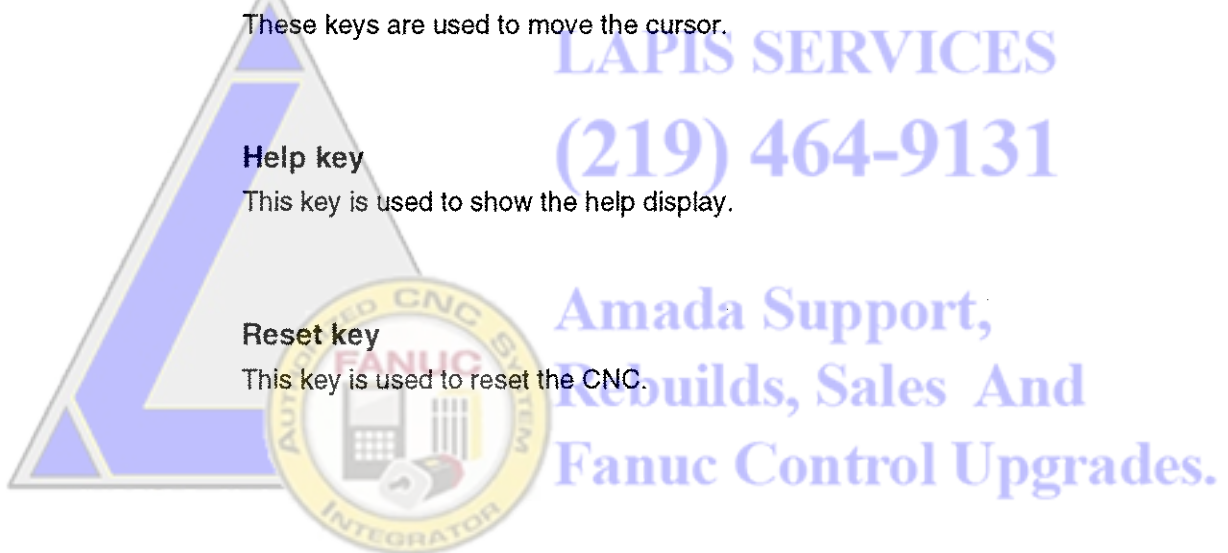
These keys are used to move the cursor.

Help key

This key is used to show the help display.

Reset key

This key is used to reset the CNC.



Indicator lights

NC READY light (green)

Lights when the CNC is ready for operation.

TOP DEAD CENTER light (green)

Lights when the ram is at the top dead center.

MACHINE STOP light (red)

Lights when the machine is stopped due to a program stop (M00) or optional stop (M01) command.

TOOL CHANGE keyswitch

The carriage and table will be locked when this switch is turned to ON in the MANUAL mode — the turret can then be rotated safely in the same mode. The switch key can be removed in the ON position.

NOTE

- Never turn this switch to ON during operation in a mode other than MANUAL.

PRESS SELECT keyswitch

Selects the manner of punching as follows:

Switch position	"INCHING"	"OFF"	"CYCLE"
MANUAL mode	Inch-punching	No punching	No punching
MDI mode	Inch-punching	No punching	Single cycle
MEMORY mode	No punching	No punching	Continuous

Inch-punching and single-cycle punching are started by pressing the PUNCHING button in the MANUAL or MDI mode.

Continuous punching in the MEMORY mode is performed according to the program.

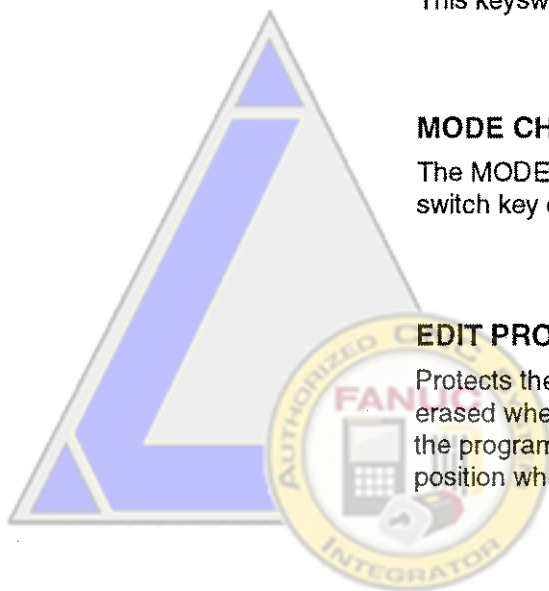
This keyswitch will be locked in position when the key is removed.

MODE CHANGE keyswitch

The MODE buttons are enabled when this switch is turned to ON. The switch key can be removed in the OFF position.

EDIT PROTECT keyswitch

Protects the program in the CNC memory from being overwritten or erased when turned to ON. Keep the switch in the ON position unless the program must be edited. The switch will be locked in the ON position when the key is removed.



LAPIS SERVICES

(918) 466-2431

Canada Support,
Fanuc Control Upgrades.

LAPISINC.COM

CONFIRM button

Restarts the machine to resume the operation which has been interrupted to confirm the safety of the workclamps from interfering with the punch.

NOTICE

- Do not press this button until visually confirming that the operation can safely be continued.

OT RELEASE button

Used to release an overtravel of the carriage or table in combination with the corresponding AXES JOGS AND REFERENCES button — the two buttons must be pressed together.

PUNCHING button

Used to perform punching according to the setting on the PRESS SELECT keyswitch in the MDI or MANUAL mode — when the INDEX PIN IN light on the subcontrol panel "B" is lighted.

MODE buttons

These buttons are used to select CNC modes. They are enabled only when the MODE CHANGE keyswitch is turned to ON. The button light comes on when the mode is selected.

RETRACT button

Selects the RETRACT mode to permit manual zero-return of an axis or axes.

MEMORY button

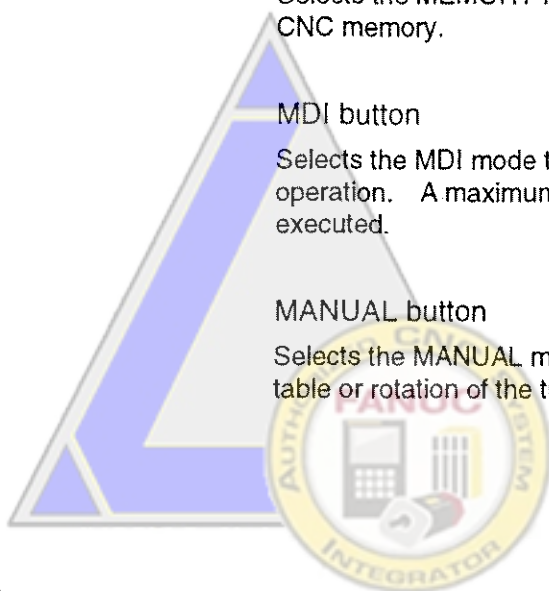
Selects the MEMORY mode to permit execution of a program in the CNC memory.

MDI button

Selects the MDI mode to permit a warming-up operation and test operation. A maximum of 10 blocks can be programmed and executed.

MANUAL button

Selects the MANUAL mode to permit manual feed of the carriage or table or rotation of the turret.



LAPIS SERVICES

(219) 464-9131

Amade Support

Rebuilds, Sales And

Fanuc Control Upgrades.

LAPISINC.COM

EDIT button

Selects the EDIT mode to permit program editing and registration, direct programming, etc.

Program execution selection buttons

These buttons are used to select the mode of program execution. The button light comes on when the mode is selected.

SINGLE button

Permits the single-block operation.

TEST button

Permits a program check without operating the machine.

OPT STOP button

Permits the optional-stop operation.

BLOCK SKIP button

Permits the block-skip operation.

MULTIPLE PART MODE buttons

These buttons are used to select the method of punching for a multi-part punching program. The button light comes on when the method is selected.

First part()button

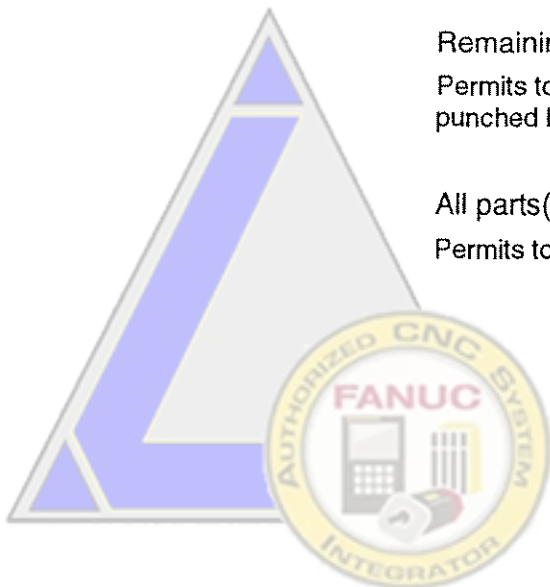
Permits to punch only the first part which is used as basis to punch the remaining parts.

Remaining parts()button

Permits to punch the remaining parts after the first part has been punched by using the First part button.

All parts()button

Permits to punch all the parts.



**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

AXES JOGS AND REFERENCES buttons

These buttons have different functions in the RETRACT mode and the MANUAL mode.

+X, +Y, T buttons

These three buttons are used to return the carriage (+X), table (+Y), and turret and auto-index device (T) to their origins in the RETRACT mode. The button light comes on when the corresponding axis is returned to its origin.

+X, -X, +Y, -Y buttons

These four buttons are used to feed the carriage (+X, -X) and table (+Y, -Y) in the indicated direction in the MANUAL mode.

FEEDRATE buttons

These buttons set feedrates for the carriage and table as well as indexing speeds for the turret and auto-index device. The button light comes on when the speeds are selected.

Button	Carriage	Table	Turret	AI device
F1	65 m/min (2560 ipm)	50 m/min (1968 ipm)	30 rpm	60 rpm
F2	48.75 m/min (1919 ipm)	37.5 m/min (1476 ipm)	30 rpm	60 rpm
F3	32.5 m/min (1280 ipm)	25.0 m/min (984 ipm)	15 rpm	30 rpm
F4	16.25 m/min (640 ipm)	12.5 m/min (492 ipm)	15 rpm	30 rpm
FA	High-accuracy mode operation			

The carriage and table feedrates and the turret and auto-index device indexing speeds will be 50% of the setting by the F-code command in the program when the FA button is pressed.

The carriage and table feedrates in the MANUAL mode will be 5.2 m/min (204 ipm) regardless of the selection.

The turret indexing speed in the MANUAL mode will be 3 rpm regardless of the selection.

START button

Starts operation when pressed in the MEMORY or MDI mode.

STOP button

Stops operation. The button light comes on when the button is pressed and for the duration of the stopped condition.

LAPIS SERVICES

(219) 464-9131

Amada Support,

Reliable Sales And

Fanuc Control Upgrades.

LAPISINC.COM

EMERGENCY STOP button

Stops the machine operation instantly when pressed in any operation mode — the button will then be locked. Unlock the button by using the key, normalize the condition, and return all axes to their origins before starting another operation. The interrupted operation cannot be resumed.

RAM POSITION switch

Selects the ram stopping position at the top dead center or the top stroke end in a MEMORY mode operation. Keep the switch turned to the right position (top stroke end) normally and turn it to the left position (top dead center) if the worksheet has been warped by punching during the operation.

WARMING START button

Starts a warming up operation for the machine when pressed and lighted in the MDI mode — the button remains lighted during the operation.

RAM SPEED buttons

Selects a punching speed for operations in the MEMORY or MDI mode. F1 to F3 buttons select preset speeds — the speed is highest at F1 and lowest at F3. Punching speed will be selected automatically according to the worksheet material and thickness and the tool used when the Auto () button is pressed. (Refer to “PHNC switch settings” in Part III, Operation.)

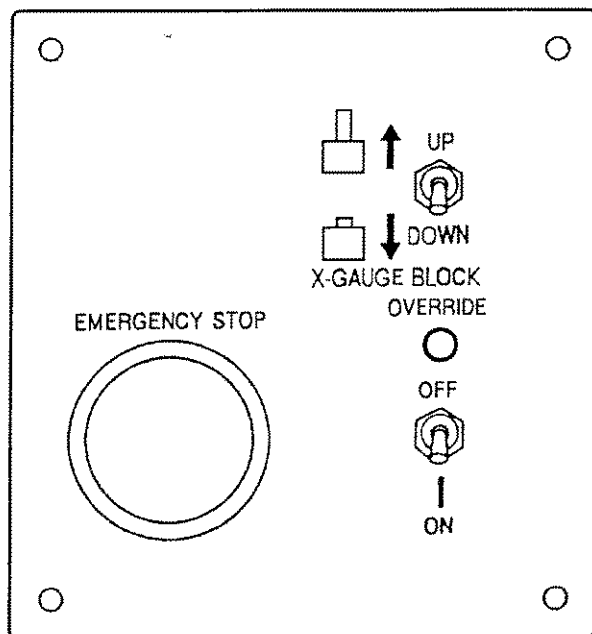


LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

SUBCONTROL PANEL “A”



OVERVERRIDE ON/OFF switch

Turns on or off the function of detecting the workclamp position in relation to the punching position and the workholder positions to prevent interference. The function is enabled regardless of the setting of this switch when the REMOTE switch in the electrical control cabinet is turned to ON.

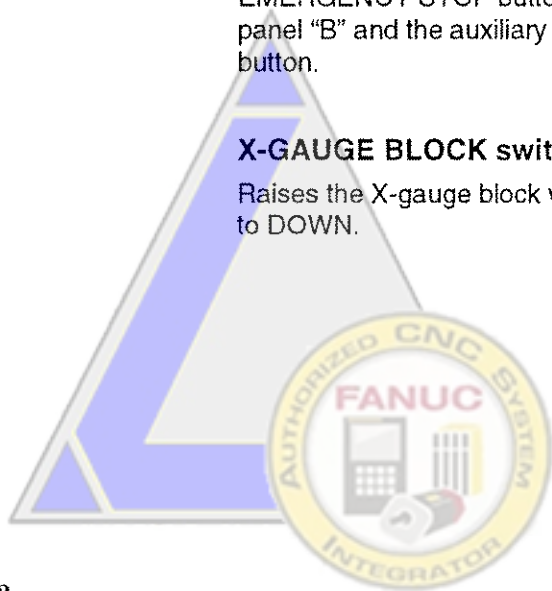
EMERGENCY STOP button

Stops the machine operation instantly when pressed in any operation mode — the button will then be locked. Unlock the button by turning it clockwise, normalize the condition, and return all axes to their origins before starting another operation. The interrupted operation cannot be resumed.

EMERGENCY STOP buttons on the main control panel, subcontrol panel “B” and the auxiliary control stand have the same function as this button.

X-GAUGE BLOCK switch

Raises the X-gauge block when turned to UP and lowers it when turned to DOWN.



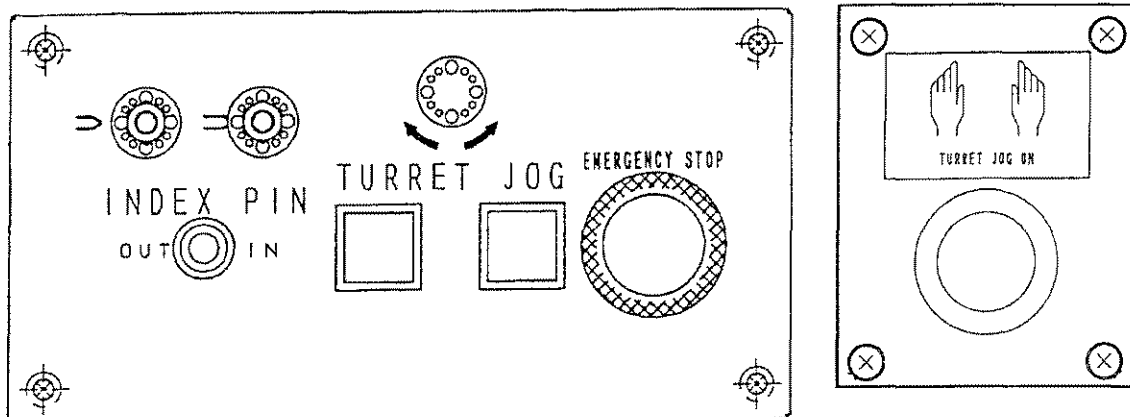
LAPIS SERVICES

(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

SUBCONTROL PANEL "B"



INDEX PIN switch, IN & OUT lights

The switch is used to operate the turret index pins in the MANUAL mode — it is disabled in other modes. The pins are inserted in turret disks by turning the switch to IN, lighting the IN light. The pins are withdrawn when the switch is turned to OUT, lighting the OUT light — keep the OUT light on normally. The switch is spring-returned to the neutral position when it is released.

TURRET JOG buttons

These buttons are used to rotate the turret in the MANUAL mode — when the TOOL CHANGE keyswitch on the main control panel is turned to ON. Use the left button to rotate the turret clockwise and the right button to rotate it counterclockwise — the turret will rotate at 3 rpm. The turret will stop to index the nearest turret station in the turret's rotating direction when the button is released.

The turret index pins must be out of turret disks when using these buttons. And each button must be pressed within 0.5 second after pressing and holding the TURRET JOG ON button (see below).

The TURRET JOG buttons are disabled in modes other than MANUAL.

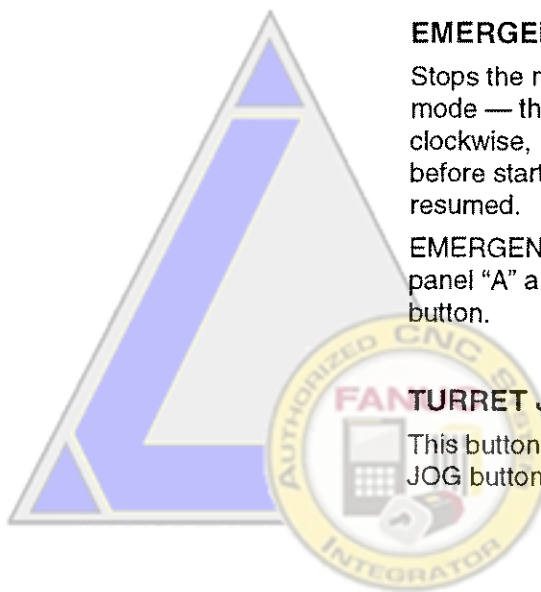
EMERGENCY STOP button

Stops the machine operation instantly when pressed in any operation mode — the button will then be locked. Unlock the button by turning it clockwise, normalize the condition, and return all axes to their origins before starting another operation. The interrupted operation cannot be resumed.

EMERGENCY STOP buttons on the main control panel, subcontrol panel "A" and the auxiliary control stand have the same function as this button.

TURRET JOG ON button

This button is used to rotate the turret in combination with the TURRET JOG buttons (see above) in the MANUAL mode.

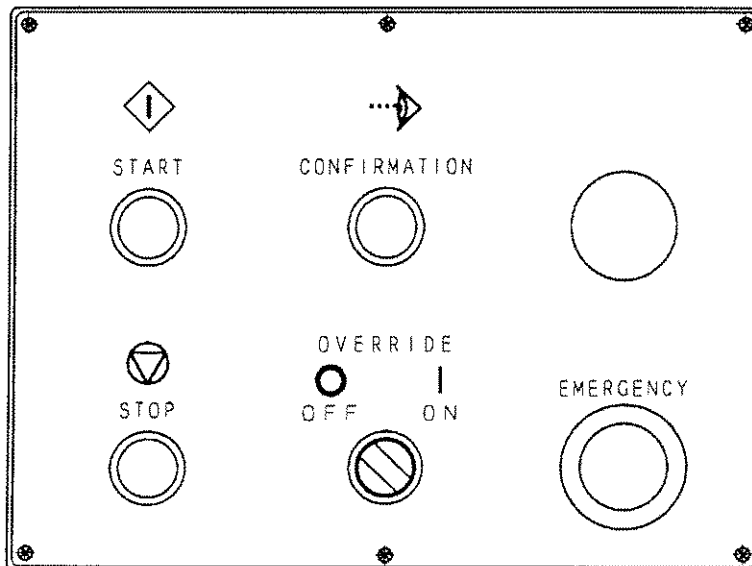


Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

AUXILIARY CONTROL STAND

This control stand is used to operate and control the machine from the outside of the hazardous area.



START button

Starts operation when pressed in the MEMORY or MDI mode.

STOP button

Stops operation when pressed in any mode. The STOP button on the main control panel will be lighted when this button is pressed.

CONFIRMATION button

Restarts the machine to resume the operation which has been interrupted to confirm the safety of the workclamps from interfering with the punch.

NOTICE

- Do not press this button until visually confirming that the operation can safely be continued.

LAPIS SERVICES
(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

OVERRIDE ON/OFF switch

Turns on or off the function of detecting the workclamp position in relation to the punching position and the workholder positions to prevent interference. The function is enabled regardless of the setting of this switch when the REMOTE switch in the electrical control cabinet is turned to ON.

EMERGENCY STOP button

Stops the machine operation instantly when pressed in any operation mode — the button will then be locked. Unlock the button by turning it clockwise, normalize the condition, and return all axes to their origins before starting another operation. The interrupted operation cannot be resumed.

EMERGENCY STOP buttons on the main control panel and subcontrol panels "A" and "B" have the same function as this button.



LAPIS SERVICES
(219) 464-9131

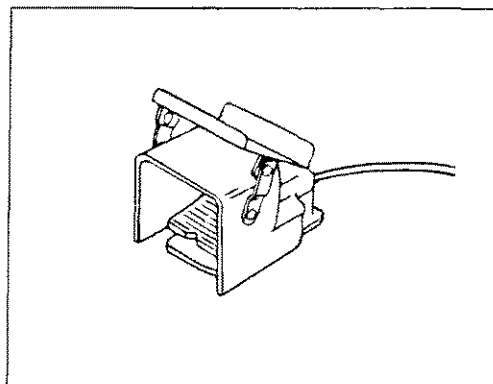
**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

FOOT SWITCH & INTERFACE

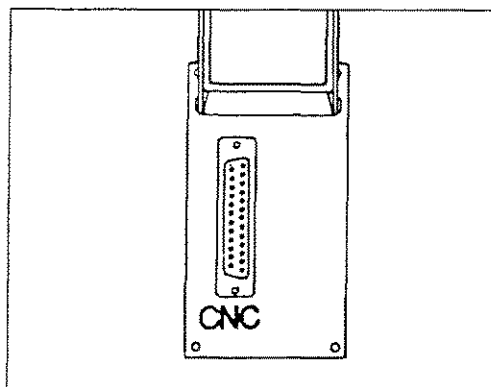
Foot switch

The foot switch is used to open and close the workclamps. During an automatic operation, it is enabled only when the operation is stopped by the program stop command.



Interface for input/output device

An input/output device (tape puncher/reader, floppy disk drive, etc.) with the EIA-RS232C interface can be connected to the interface on the electrical control cabinet. Use the interface for the input/output of CNC programs. (The connector configuration is shown below.)



CONNECTOR CONFIGURATION

1	2	3	4	5	6	7	8	9	10	11	12	13
FG	SD	RD	RS	CS	DR	SG	CD					
14	15	16	17	18	19	20	21	22	23	24	25	
						ER					+24N	

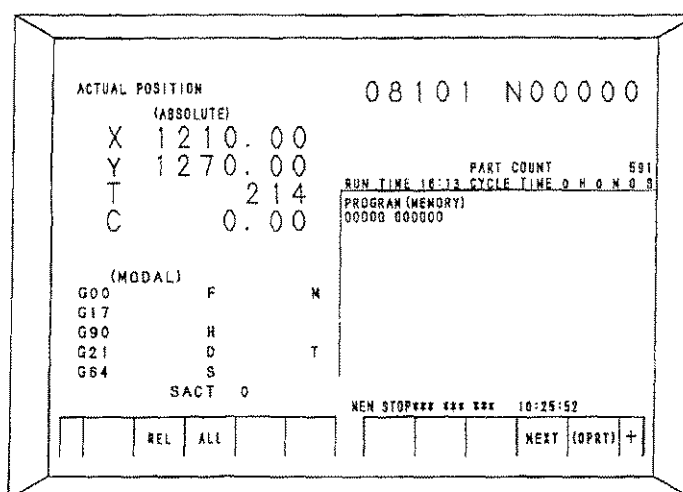
**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

DISPLAYS

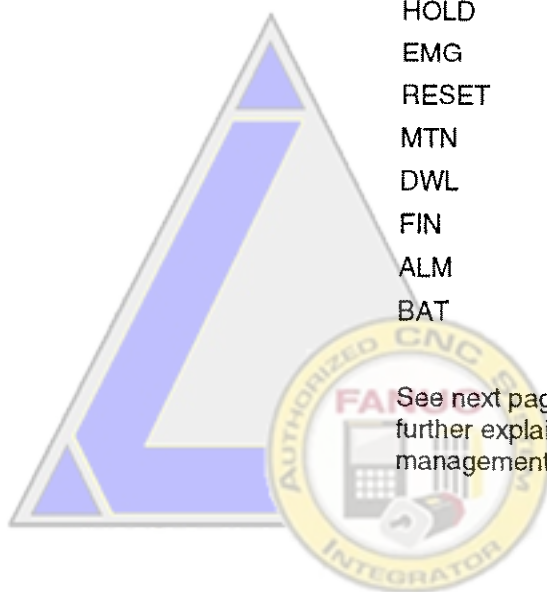
A number of displays can be shown selectively on the screen on the main control panel by pressing a corresponding display selection key. Some of these displays have a series of additional displays — they can be shown by using the assigned softkeys identified on the screen.

In the lower right section of each display, the current CNC status is indicated by the code words described below:



EDIT	EDIT mode
MEM	MEMORY mode
RMT	Remote operation mode
MDI	MDI mode
INC	Incremental feed mode
HND	Handle feed mode
JOG	MANUAL mode
REF	RETRACT mode
STRT	Start state
STOP	Stop state
HOLD	Hold state
EMG	Emergency stop state
RESET	Reset state
MTN	Axis traveling
DWL	Dwelling
FIN	Waiting for auxiliary function finish
ALM	Alarm state
BAT	Low battery

See next page for the description of displays. Their applications are further explained in Part III, Operation, and Part IV, Program management.



LAPIS SERVICES
(219) 464-9131

Amada Support,
Fanuc Control Upgrades.

LAPISINC.COM

Setting displays

The Setting displays are used to set the setup data, etc. One of the Setting displays is shown by pressing the Offset/Setting key and then the SETING softkey. It can be changed to another Setting display by pressing the Page key.

SETTING HANDY display

This display shows the following items.

PARAMETER WRITE

Enables or disables alteration of parameters.

0 : Disable

1 : Enable

TV CHECK

Enables or disables the TV check function.

0 : Disable

1 : Enable

PUNCH CODE

Selects the code system for the output of data.

0 : EIA

1 : ISO

INPUT UNIT

Selects the unit for the input of data.

0 : Millimeter

1 : Inch

I/O CHANNEL

Selects the channel of the I/O interface.

0 : Channel 0

1 : Channel 1

2 : Channel 2

3 : Channel 3

SEQUENCE NO

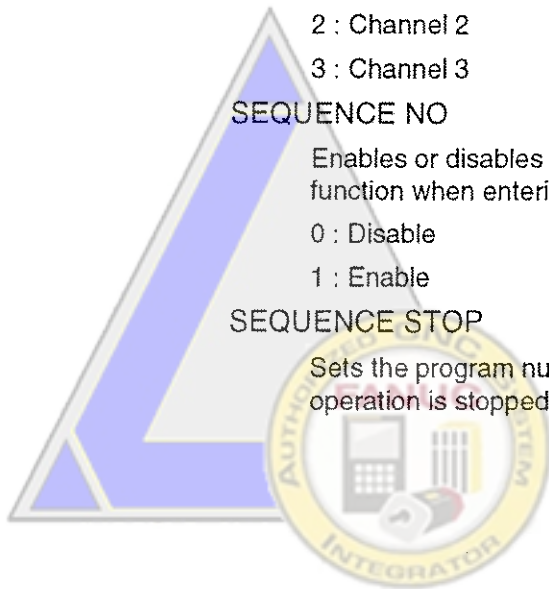
Enables or disables the automatic sequence number insertion function when entering a program in the EDIT mode.

0 : Disable

1 : Enable

SEQUENCE STOP

Sets the program number and the sequence number at which the operation is stopped.



LAPIS SERVICES

(219) 461-9151

Amada Support,

Rebuilds, Sales And

Fanuc Control Upgrades.

LAPISINC.COM

SETTING (TIMER) display

This display shows the following items.

PARTS TOTAL

Indicates a total number of punched worksheets.

PARTS REQUIRED

Sets a number of worksheets to be punched.

PARTS COUNT

Indicates a number of worksheets punched. The count will be reset to zero when it reaches the number set at PARTS REQUIRED.

POWER ON

Indicates an accumulated period of time in which the CNC has been turned on.

OPERATING TIME

Indicates an accumulated period of time in which the machine has been operated automatically.

CUTTING TIME

Not used.

FREE PURPOSE

Not used.

CYCLE TIME

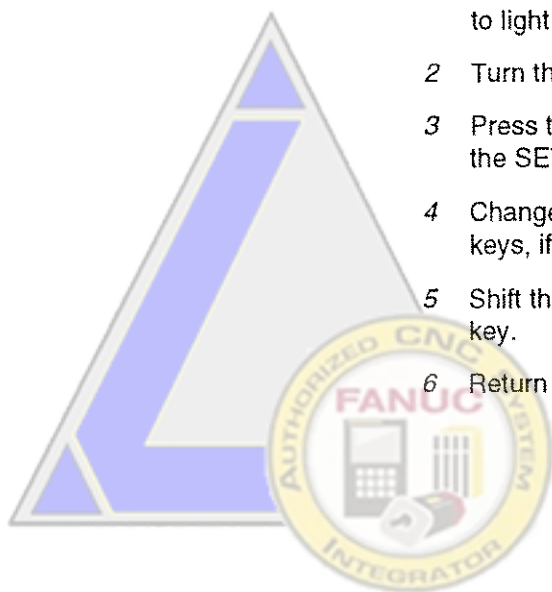
Indicates a period of time in which the machine has been operated automatically.

DATE and TIME

Indicates the date and time. The date and time can be set here.

Changing the setup data on the SETTING display

- 1 Turn the MODE CHANGE keyswitch to ON, press the MDI button to light its light, and then return the keyswitch to OFF.
- 2 Turn the EDIT PROTECT keyswitch to OFF.
- 3 Press the Offset/Setting key and then the SETING softkey to show the SETTING display.
- 4 Change the display to another SETTING display by using the Page keys, if necessary.
- 5 Shift the cursor to the required item, enter data and press the Input key.
- 6 Return the EDIT PROTECT keyswitch to ON.



LAPIS SERVICES
(219) 464-9131
Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

Program displays

The PROGRAM display shows the program which is registered in the CNC memory and the current axial positions. It is displayed by pressing the Program key. It can be changed to the PROGRAM DIRECTORY display, which shows a list of program numbers and comments for the programs stored in the CNC memory as well as the remaining memory capacity. This display is shown by pressing the softkey which is labeled DIR on the screen.

Position display

The ACTUAL POSITION display shows the current positions of the axes (of program and machine coordinate systems), the remaining travel distances, a number of punched worksheets, and accumulated periods of time in which the CNC and the machine have been turned on. It is displayed by pressing the Position key.

System displays

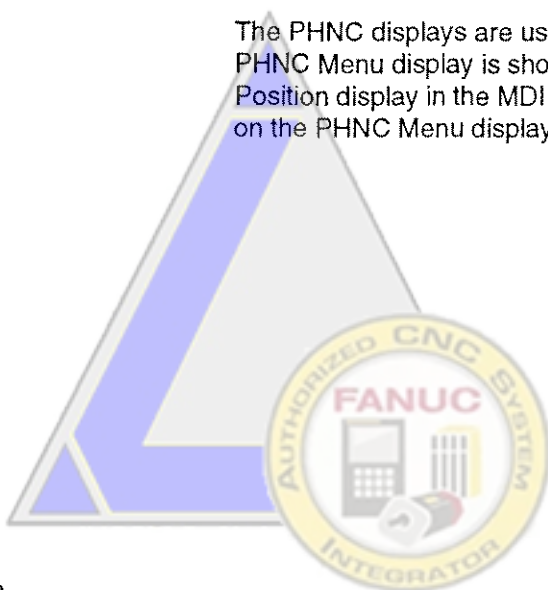
The System displays are used for maintenance purposes. One of the System displays is shown by pressing the System key. It can be changed to another System display by pressing a softkey.

Message displays

The ALARM display is used to identify the cause of an alarm. It is shown by pressing the Message key. It can be changed to the ALARM HISTORY and MESSAGE displays by pressing the HSTRY and MSG softkeys, respectively.

PHNC displays

The PHNC displays are used to set the PHNC parameters, etc. The PHNC Menu display is shown by pressing the PHNC softkey on the Position display in the MDI mode. The PHNC displays can be selected on the PHNC Menu display.



(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Part III

Operation

Preparation.....	III-2
Starting up the machine	III-2
Warm-up operation	III-3
Mounting the tool.....	III-3
Operation.....	III-4
MEMORY mode operation.....	III-4
MDI mode operation.....	III-7
REMOTE mode operation.....	III-10
MANUAL mode operation	III-12
Punching in the MANUAL mode	III-12
PHNC switch settings.....	III-13
Operation interruption	III-15
Shutting down the machine	III-18



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

PREPARATION

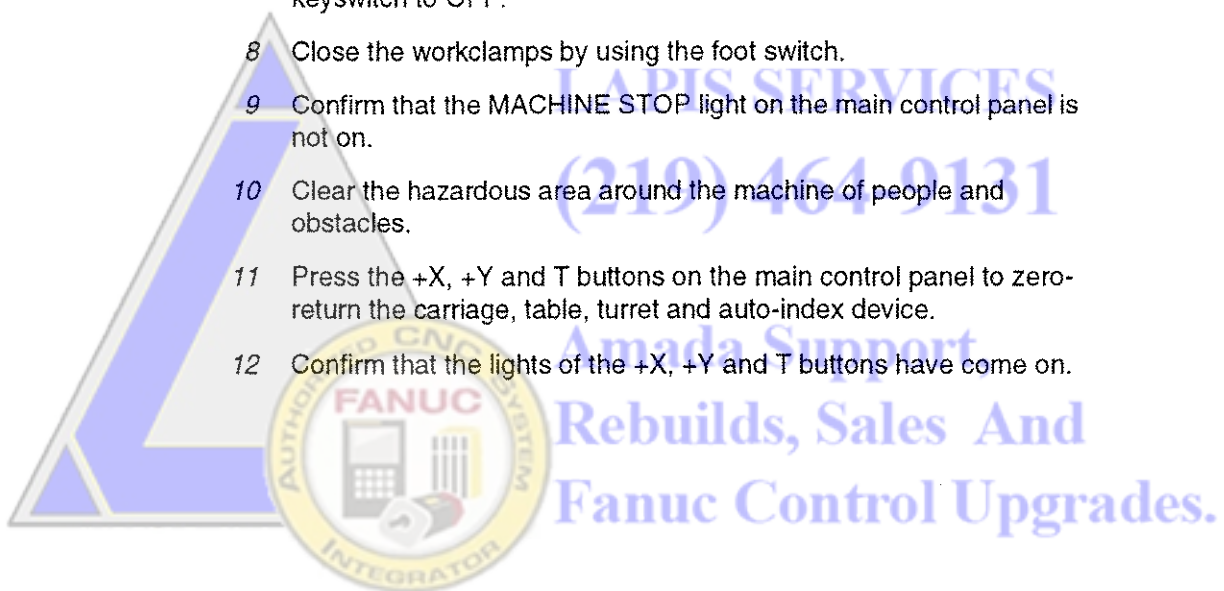
The preparation procedure for a punching operation is described in this section.

Be sure to inspect the machine before starting the day's work (refer to Part VII, Maintenance) and then prepare the machine for the operation.

Starting up the machine

	WARNING	● Be sure that the hazardous area around the machine is cleared of people and obstacles before starting the machine.
---	----------------	--

- 1 Turn on the shop power.
- 2 Turn on the air compressor and open the air intake valve.
- 3 Turn on the cooling water unit.
- 4 Turn the machine circuit breaker switch on the electrical control cabinet to ON.
- 5 Press the POWER ON button on the main control panel.
- 6 Confirm that the NC READY and TOP DEAD CENTER lights on the main control panel are on.
- 7 Confirm that the RETRACT button on the main control panel is lighted. If not, turn the MODE CHANGE keyswitch on the same panel to ON, press the RETRACT button, and then return the keyswitch to OFF.
- 8 Close the workclamps by using the foot switch.
- 9 Confirm that the MACHINE STOP light on the main control panel is not on.
- 10 Clear the hazardous area around the machine of people and obstacles.
- 11 Press the +X, +Y and T buttons on the main control panel to zero-return the carriage, table, turret and auto-index device.
- 12 Confirm that the lights of the +X, +Y and T buttons have come on.



WARM-UP OPERATION

It is necessary to run the machine to warm it up as well as the hydraulic oil before operating the machine for punching if the oil temperature is below 10°C (50°F). Warm up the machine in the following manner:

- 1 Turn the MODE CHANGE keyswitch on the main control panel to ON, press the MDI button on the same panel to light its light, and then return the keyswitch to OFF.
- 2 Press the WARMING START button on the main control panel—the button will light and a warm-up run will start. The ram will be operated at a stroke of 5 mm (0.2") for three minutes and then the machine will stop automatically.

NOTE

- To stop the warm-up run before it is stopped automatically, press one of the STOP buttons — the machine will then stop and the WARMING START button will be unlighted. The stopped warm-up run can be restarted by pressing the WARMING START button again.

Mounting the tool

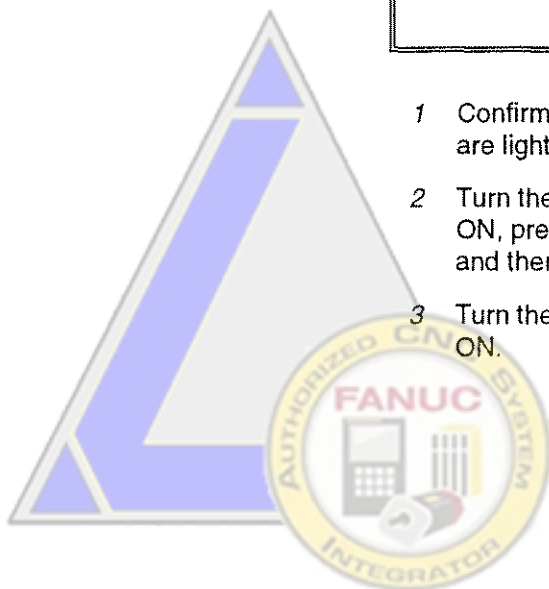
Mount punching tools on the turret. The turret can be rotated in the MANUAL mode to bring the required station to the tool change position. Refer to Part VIII, Tooling, for the mounting/dismounting procedure.



WARNING

- Turn the TOOL CHANGE keyswitch to ON when mounting or dismounting tools.
- Before rotating the turret, check that there are no persons near the turret.

- 1 Confirm that the +X, +Y and T buttons on the main control panel are lighted.
- 2 Turn the MODE CHANGE keyswitch on the main control panel to ON, press the MANUAL button on the same panel to light its light, and then return the keyswitch to OFF.
- 3 Turn the TOOL CHANGE keyswitch on the main control panel to ON.



LAPIS SERVICES
(210) 464-9131
**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

- 4 Bring the required turret station to the tool change position by using the TURRET JOG and TURRET JOG ON buttons on the subcontrol panel "B" — the turret will rotate as long as the TURRET JOG ON-button is pressed together with one of the TURRET JOG buttons.
- 5 Release the buttons to stop the turret when the required station is in the tool change position.
- 6 Open the tool change doors.
- 7 Mount, dismount, or change tools (see Part VIII, Tooling).
- 8 Close the tool change doors.
- 9 Return the TOOL CHANGE keyswitch on the main control panel to OFF.

OPERATION

A punching operation can be performed by using a program registered in the CNC in the MEMORY mode, by sending a program from an external computer in the REMOTE mode, or by creating and executing a new program (up to 10 blocks) in the MDI mode.

MEMORY mode operation

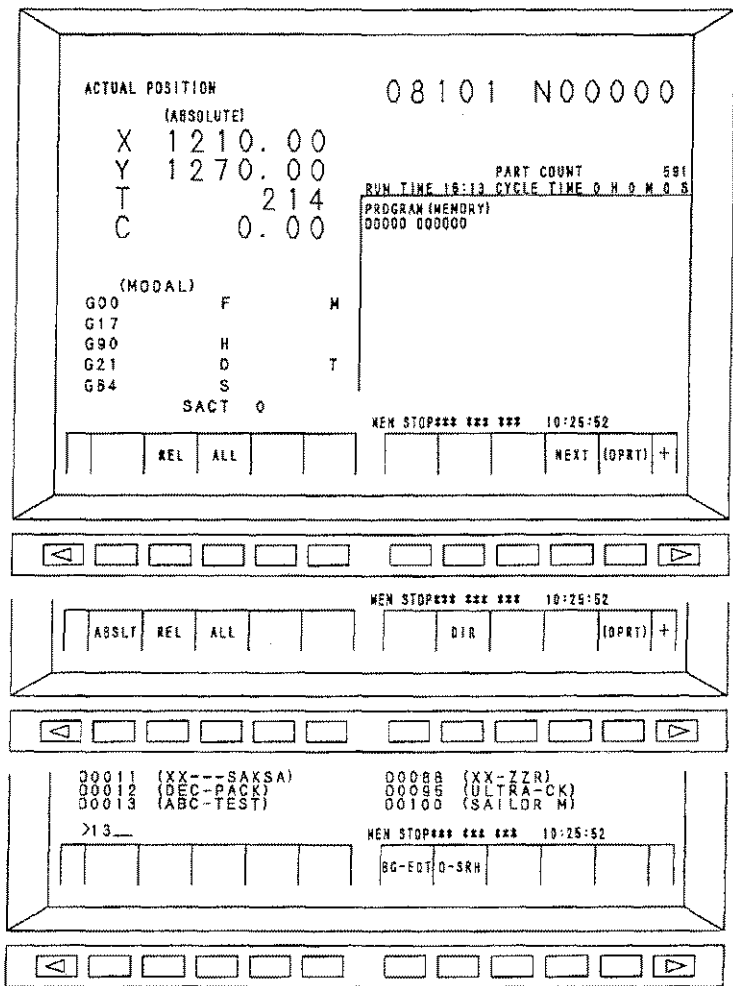
The MEMORY mode operation can be performed in the following manner when the machine has been prepared and the program is registered and available for the operation:



WARNING

- Turn the TOOL CHANGE keyswitch to ON when working inside the hazardous area.
- Do not press the START button until the hazardous area around the machine is completely cleared of people and obstacles.

- 1 Turn the MODE CHANGE keyswitch on the main control panel to ON, press the MEMORY button on the same panel to light its light, and then return the keyswitch to OFF.
- 2 Press the Program key on the main control panel.



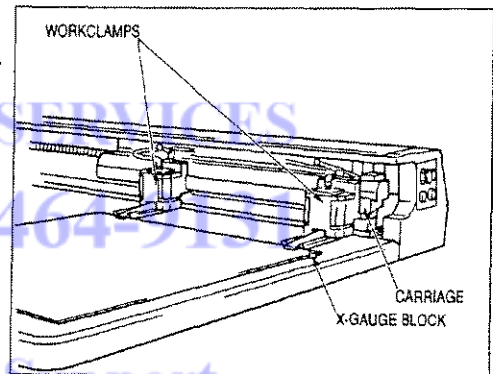
- 3 Press the right arrowhead softkey and the DIR softkey.
- 4 Show the program to be used on the display — key-in the number for the program and then press the O-SRH softkey. Then shift the cursor to the address character "O" of the shown program number. (If the cursor is somewhere else in the program, the program will be executed from where the cursor is.)

- 5 Open the workclamps by using the foot switch.

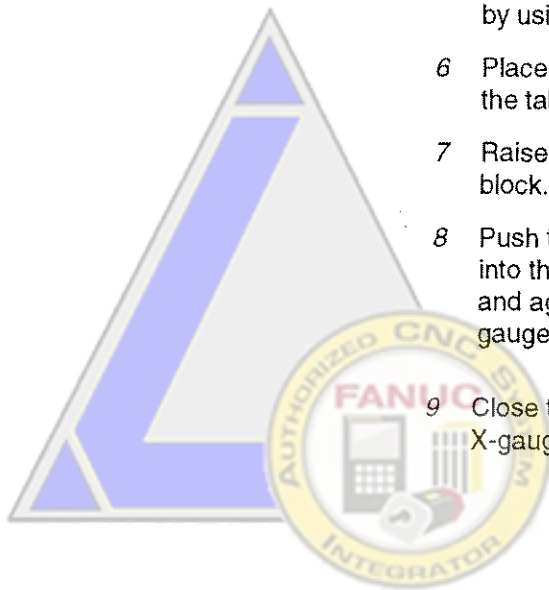
- 6 Place the worksheet on the table.

- 7 Raise the X-gauge block.

- 8 Push the worksheet into the workclamps and against the X-gauge block.



- 9 Close the workclamps by using the foot switch and then lower the X-gauge block.



Amada Support,
 Tools, Sales And
 Fanuc Control Upgrades.

- 10 Confirm that no alarm messages appear on the screen. Also check to ensure that switches* and PHNC parameters are properly set for the operation. Take necessary steps as required.
- * • BLOCK SKIP button
 - EDIT PROTECT keyswitch (turned to ON)
 - FEEDRATE button
 - MODE CHANGE keyswitch (turned to OFF)
 - MULTIPLE PART MODE button
 - OPT STOP button
 - OVERRIDE ON/OFF switch
 - PRESS SELECT keyswitch (turned to CYCLE or OFF)
 - SINGLE button
 - TEST button
 - TOOL CHANGE keyswitch (turned to OFF)
 - RAM POSITION switch
 - RAM SPEED button
- 11 Clear the hazardous area around the machine of people and obstacles.
- 12 Press the START button to execute the program. The cursor on the display will shift to the program block which is being executed as the operation progresses.
- 13 Upon completion of the program, the carriage and table will then be returned to their origins and the lights of the +X and +Y buttons on the main control panel will come on. The cursor will be returned to the start of the program.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

MDI mode operation

The MDI mode operation can be performed in the following manner when the machine has been prepared for the operation:

**WARNING**

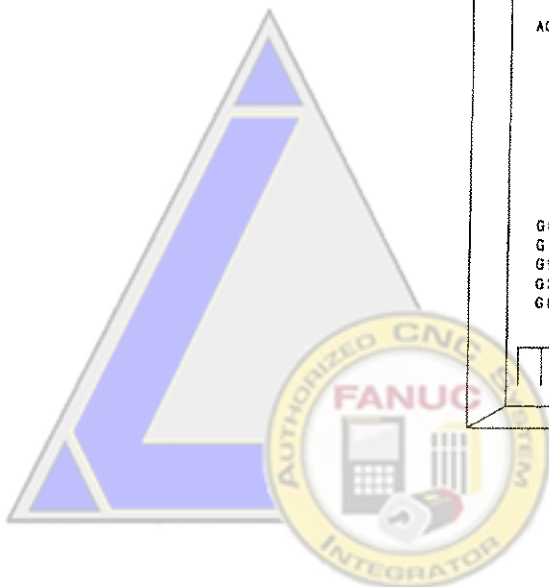
- Turn the TOOL CHANGE keyswitch to ON when working inside the hazardous area.
- Do not press the START button until the hazardous area around the machine is completely cleared of people and obstacles.

- 1 Turn the MODE CHANGE keyswitch on the main control panel to ON, press the MDI button on the same panel to light its light, and then return the keyswitch to OFF.
- 2 Press the MDI softkey.
- 3 Open the workclamps by using the foot switch.
- 4 Place the worksheet on the table.
- 5 Raise the X-gauge block.
- 6 Push the worksheet into the workclamps and against the X-gauge block.
- 7 Close the workclamps by using the foot switch and then lower the X-gauge block.
- 8 Create the new program (up to 10 blocks) on the screen as follows:

NOTE

- If a punching pattern is used for the entry of data or two or more blocks are programmed, specify the single block function by pressing the SINGLE button. Otherwise only positioning will be executed and the machine will then be stopped.

ACTUAL POSITION		08101 N00000	
(ABSOLUTE)			
X	1210.00		
Y	1270.00		
T	214		
C	0.00		
(MODAL)			
G00	F	M	
G17			
G90	H	T	
G21	D		
G84	S		
SACT 0			
		MDI RUN STOP 10:25:52	
		REL ALL NEXT (OPR1) +	



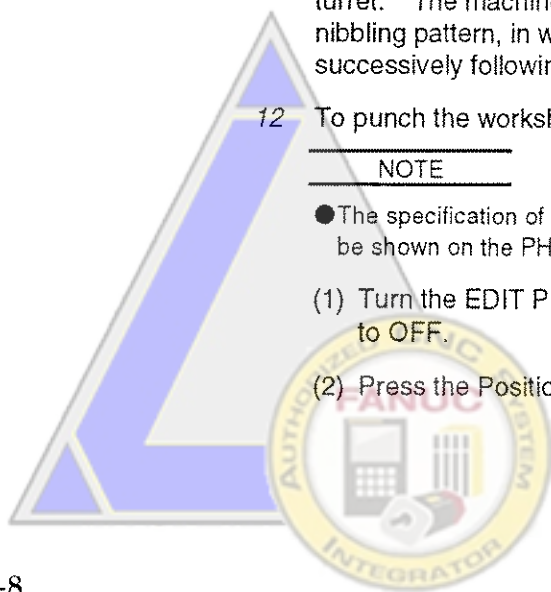
Amada Support
Rebuilds, Sales And
Fanuc Control Upgrades.

- (1) Turn the EDIT PROTECT keyswitch on the main control panel to OFF.
 - (2) Press the ABSLT softkey to specify the entry of absolute data. The code word "ABS" will appear in the lower right section of the screen.
 - (3) Key-in data for the first block of the program by using address and numeric keys on the main control panel. Then press the EOB key on the same panel to enter the end-of-block code (;).
 - (4) Then press the Insert key on the same panel.
 - (5) Enter data for the successive blocks in the same manner.
 - (6) After the entry of data has been completed, press the REWIND softkey to shift the cursor to the start of the created program.
 - (7) Return the EDIT PROTECT keyswitch on the main control panel to ON.
- 9 Confirm that no alarm messages appear on the screen. Also check to ensure that switches* and PHNC parameters are properly set for the operation. Take necessary steps as required.
- * • BLOCK SKIP button
 • EDIT PROTECT keyswitch (turned to ON)
 • FEEDRATE button
 • MODE CHANGE keyswitch (turned to OFF)
 • OPT STOP button
 • PRESS SELECT keyswitch
 • SINGLE button
 • TOOL CHANGE keyswitch (turned to OFF)
 • RAM POSITION switch
 • RAM SPEED button
- 10 Clear the hazardous area around the machine of people and obstacles.
- 11 Press the START button to position the worksheet and index the turret. The machine will then stop temporarily — except for a nibbling pattern, in which case punching can be performed successively following the positioning and the indexing.
- 12 To punch the worksheet, set up the PHNC as follows:

NOTE

- The specification of the worksheet used in the previous operation can be shown on the PHNC Monitor display.

- (1) Turn the EDIT PROTECT keyswitch on the main control panel to OFF.
- (2) Press the Position key on the main control panel.



LAPIS SERVICES
(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

(3) Press the PHNC softkey to show the PHNC Menu display on the screen.

(4) Shift the cursor to item Manual Press and then press the Input key on the main control panel. The Manual Press display will appear on the screen.

PHNC Menu	
1	Press Pattern Edit
2	Tooling Data Edit
3	PHNC Monitor
4	TIME Data
5	Diagnose
6	Manual Press
7	PHNC Maintenance

(5) Key-in data for the M-code command to be used and press the Input key on the main control panel.

MANUAL Press	
Press Pattern	(M)
Thickness	(0.0)
Material (0:OCR 1:SUS 2:AL)	()
0:Bottom pos 1:Bottom pos end	(M)

NOTE

● Two ram-lowering positions Bottom Pos (preset position at the stroke end) and Bottom Pos End are available in the slitting press mode.

(6) After the entry of data has been completed, press the REG. DATA softkey to store them in the memory.

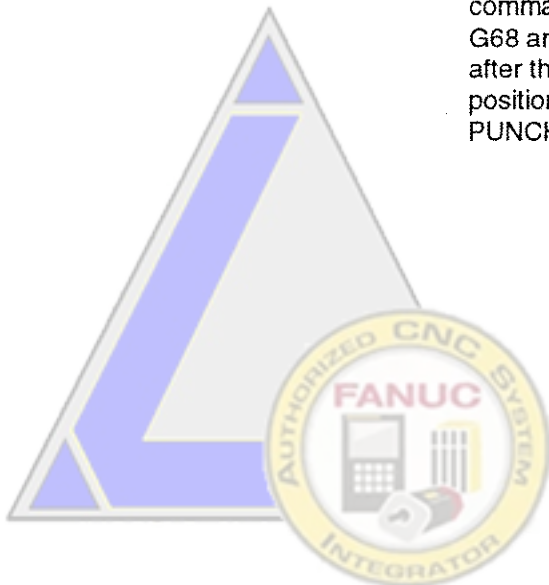
(7) Return the EDIT PROTECT keyswitch on the main control panel to ON.

13 Press to light a RAM SPEED button on the main control panel as required.

14 Set the PRESS SELECT keyswitch on the main control panel to CYCLE or INCHING.

15 Press the PUNCHING button on the main control panel to punch the worksheet. Single nibbling will be executed completely if it is commanded by G68 or G69. When executing a pattern other than G68 and G69, or two or more blocks, the machine will be stopped after the first punching — to proceed, press the START button to position the worksheet for the next punching and then press the PUNCHING button each time the machine stops.

(219) 464-9131



**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

REMOTE mode operation

The REMOTE mode operation can be performed in the following manner when the machine has been prepared and the program is available for the operation:


WARNING

- Turn the TOOL CHANGE keyswitch to ON when working inside the hazardous area.
- Do not press the START button until the hazardous area around the machine is completely cleared of people and obstacles.

- 1 Open the electrical control cabinet, turn the REMOTE switch in the cabinet to ON, and then close the cabinet.
- 2 Turn the MODE CHANGE keyswitch on the main control panel to ON and press the MDI button on the same panel to light its light.
- 3 Turn the EDIT PROTECT keyswitch on the main control panel to OFF.
- 4 Press the Offset/Setting key on the main control panel.
- 5 Press the SETING softkey to show the SETTING HANDY display on the screen.

SETTING HANDY

PARAMETER WRITE=0 (0:DISABLE, 1:ENABLE)

TV CHECK =0 (0:OFF, 1:ON)

PUNCH CODE =1 (0:EIA, 1:ISO)

INPUT UNIT =0 (0:MM, 1:INCH)

I/O CHANNEL =0 (0-3:CHANNEL NO)

SEQUENCE NO =0 (0:OFF, 1:ON)

TAPE FORMAT =0 (0:NO CNV, 1:F15)

SEQUENCE STOP =0 (PROGRAM NO)

SEQUENCE STOP =0 (SEQUENCE NO)

- 6 Shift the cursor to item I/O CHANNEL and key-in "1" to specify the channel number. Then press the Input key and the Offset/Setting key on the main control panel.
- 7 Return the EDIT PROTECT keyswitch on the main control panel to ON.
- 8 Press the MEMORY button on the main control panel to light its light and then return the MODE CHANGE keyswitch on the same panel to OFF.
- 9 Press the Program key on the main control panel.

SETTING FOR INPUT DEVICE

PROTOCOL : B	SUBPROGRAM : NON
ISO PARITY : ADD	NC PROGRAM FORM : LF/CR
BAUDRATE : 4800bps	BEGIN OF DATA : LF/CR
STOP BIT : 2bit	END OF DATA : LF/CR
PARITY BIT : NON	COMPATIBILITY : FLOPPYCASSETTE
RECEIVE CODE : ISO/EIA	EOB CONVERSION : NO CONVERSION
SEND CODE : ISO	
CHANNEL : RS-232-C	

- 10 Prepare the external computer for the operation.
- 11 Open the workclamps by using the foot switch.
- 12 Place the worksheet on the table.
- 13 Raise the X-gauge block.
- 14 Push the worksheet into the workclamps and against the X-gauge block.
- 15 Close the workclamps by using the foot switch and then lower the X-gauge block.
- 16 Confirm that no alarm messages appear on the screen. Also check to ensure that switches* and PHNC parameters are properly set for the operation. Take necessary steps as required.
 - * • BLOCK SKIP button
 - EDIT PROTECT keyswitch (turned to ON)
 - FEEDRATE button
 - MODE CHANGE keyswitch (turned to OFF)
 - MULTIPLE PART MODE button
 - OPT STOP button
 - PRESS SELECT keyswitch (turned to CYCLE or OFF)
 - SINGLE button
 - TEST button
 - TOOL CHANGE keyswitch (turned to OFF)
 - RAM POSITION switch
 - RAM SPEED button
- 17 Clear the hazardous area around the machine of people and obstacles.
- 18 Press the START button to execute the program.
- 19 Upon completion of the program, the carriage and table will then be returned to their origins and the lights of the +X and +Y buttons on the main control panel will come on.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

MANUAL mode operation

Punching, carriage and table travel, turret rotation, turret index pin operation, striker position changing, and workclamp operation can be performed in the MANUAL mode.

PUNCHING IN THE MANUAL MODE

- 1 Turn the MODE CHANGE keyswitch on the main control panel to ON, press the MANUAL button on the same panel to light its light, and then return the keyswitch to OFF.
- 2 Turn the TOOL CHANGE keyswitch on the main control panel to ON.
- 3 Set the turret station in the punching position by using the switches on the subcontrol panel "B" and in the electrical control cabinet.
- 4 Turn the INDEX PIN switch on the subcontrol panel "B" to IN.
- 5 Return the TOOL CHANGE keyswitch on the main control panel to OFF.
- 6 Position the worksheet on the table by using the jog buttons (+X, -X, +Y and -Y) on the main control panel.
- 7 To inch-operate the punch, change the CNC mode to MDI and set up the PHNC. (See Step 12 in "MDI mode operation" earlier in this Part.)
- 8 Turn the PRESS SELECT keyswitch on the main control panel to INCHING.
- 9 Press the PUNCHING button on the main control panel to perform punching.

NOTE

- The ram will be lowered to the Lower Limit End position in the slitting mode.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

PHNC switch settings

The RAM SPEED buttons (F1, F2, F3, and Auto) on the main control panel are used to select a punching speed in the MEMORY or MDI mode.

Manual speed selection

Punching speed can be preset by pressing and lighting a RAM SPEED button (F1, F2 or F3) other than Auto. Punching will be performed at the highest speed at F1 to permit high productivity and the lowest speed at F3 to permit a lower noise level. (This function is applicable only to the punching press mode and not applicable to forming, marking, knockout, slitting, and nibbling press mode operations.)

Automatic speed selection

Punching speed can be selected automatically according to the worksheet material and thickness and the tool by pressing and lighting the Auto button. If the program is for punching a multiple number of worksheets of identical specifications, the punching speed for the second and successive worksheets will be regulated automatically according to the result of the first worksheet punching. (This function is applicable only to the punching press mode and not applicable to forming, marking, knockout, slitting, and nibbling press mode operations.)

When pressing and lighting the Auto button, punching speed can be specified by using M-code commands in the program and the time-controlled punching speed function can be selected by using the three timers (Time Data parameters). M-code commands and the time-controlled punching speed function can be used in combinations as described below.

- Time-controlled operation at automatically regulated punching speeds (Time Data parameters set, M-code commands not used):

In addition to the ordinary automatic punching speed control, punching speeds will be changed according to the time ranges of the day preset by the parameters.



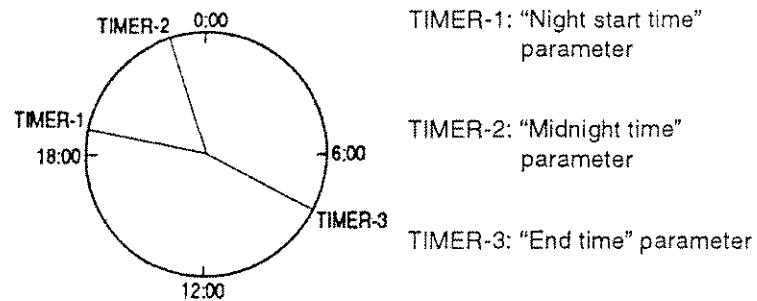
LAPIS SERVICES
(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

- Time-controlled operation at commanded punching speeds (Time Data parameters set, M-code commands used):

Punching speeds will be changed according to the time ranges of the day preset by the parameters and as specified by M-code commands. The lowest punching speed (F3 button function) is preset for the time range covered by Timer-2. M681 to M683 commands are used and correspond to the functions of the F1 to F3 buttons.



- Operation at automatically regulated punching speeds (Time Data parameters not set, M-code commands not used):

Punching will be performed under the ordinary automatic punching speed control throughout the operation.

- Operation at commanded punching speeds (Time Data parameters not set, M-code commands used):

Punching will be performed at punching speeds specified by M-code commands throughout the operation.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

Operation interruption

The machine will stop and the execution of the program will be interrupted if one of the conditions listed below arises. The cause of the condition can be identified by the corresponding indicator light or lights on the main control panel, which have come on or gone out, or by the corresponding alarm number or message shown on the screen.

Some interrupted operations can be restarted by resuming the execution of the program from the point where it has been interrupted — after confirming or normalizing the condition. However, some other interrupted operations cannot be resumed and must be restarted from the beginning of the program — after resetting the CNC or PHNC and returning the machine axes to their origins.

	WARNING	● Turn the TOOL CHANGE keyswitch to ON when working inside the hazardous area. ● Clear the hazardous area of people and obstacles before resuming or restarting the operation.
---	----------------	--

STOP button has been pressed:

The operation can be resumed after the purpose of the stop has been served or the cause of the stop has been corrected. Press the START button to resume the operation.

EMERGENCY STOP button has been pressed:

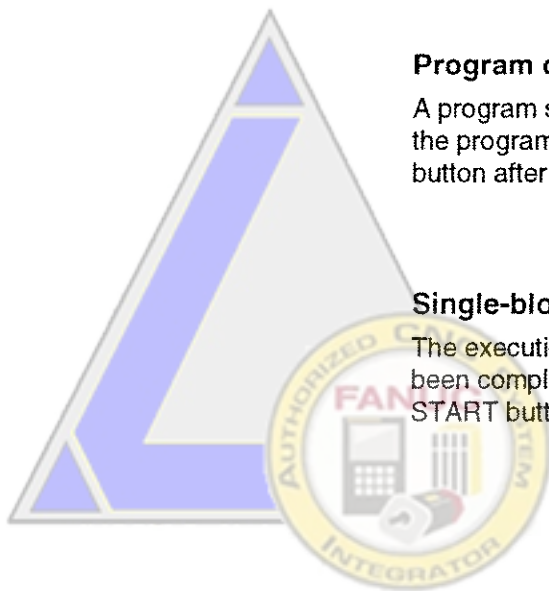
The operation must be restarted from the beginning after the purpose of the stop has been served or the cause of the stop has been corrected. Unlock the pressed button and then follow the normal procedure to start the operation.

Program or optional stop has been commanded:

A program stop (M00) or optional stop (M01) has been commanded by the program. The operation can be resumed by pressing the START button after serving the purpose of the command.

Single-block execution has been completed:

The execution of one block programmed by the single block function has been completed. The operation can then be resumed by pressing the START button.



LAPIS INC.
(219) 464-9131

Authorized Service
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Tool change door has been opened erroneously:

Close the tool change door. The operation can then be resumed by pressing the START button.

Workclamps have been opened erroneously:

Close the workclamps. The operation can then be resumed by pressing the START button.

Stripping failure has been caused:

Confirm that the punch is not stuck in the worksheet. The operation can then be resumed by pressing the START button. If the punch is stuck in the worksheet, dismount the punch and die and check them.

Workclamp-punch interference is suspected:

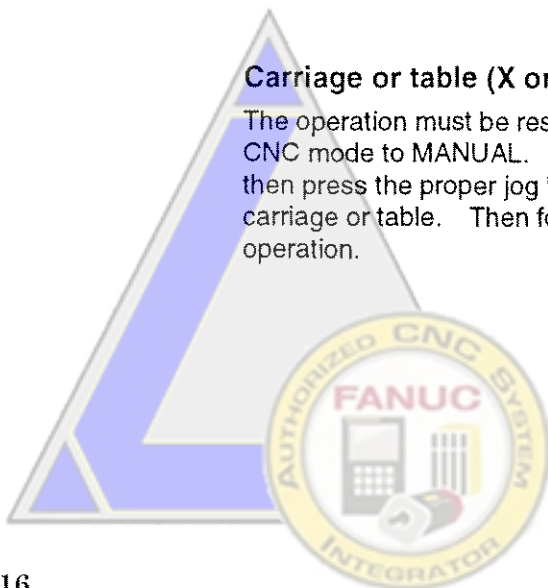
The workclamps have come too close to the punch — detection is made when the workclamp override detection function is enabled. If the workclamps are in no danger of being punched, the operation can be resumed by pressing the CONFIRM (CONFIRMATION) button. If there is the danger, the operation must be restarted from the beginning in the normal procedure — it is necessary to change the workclamp positions or modify the program.

Workclamp-workholder interference is suspected:

The workclamps have come too close to the workholders — detection is made when the worksheet repositioning detection function is enabled. If there is no danger of interference, the operation can be resumed by pressing the START button. If there is the danger, the operation must be restarted from the beginning in the normal procedure after modifying the program.

Carriage or table (X or Y axis) has overtraveled:

The operation must be restarted from the beginning. First change the CNC mode to MANUAL. Press and hold the OT RELEASE button and then press the proper jog button (+X, -X, +Y or -Y) to retract the carriage or table. Then follow the normal procedure to start the operation.



**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

Operating air pressure has been reduced:

Confirm that the air compressor is turned on and the air intake valve is open, and press the Reset key. The message "AIR PRESSURE" disappears from the screen. The operation must then be restarted from the beginning in the normal procedure.

Operating oil pressure has been reduced:

Normalize the condition. The operation must then be restarted from the beginning in the normal procedure.

Oil temperature has risen extremely high:

Normalize the condition. The operation must then be restarted from the beginning in the normal procedure.

To zero-return all the machine axes by canceling the ongoing operation:

- 1 Press any STOP button.
- 2 Press the Reset key.
- 3 Change the CNC mode to RETRACT.
- 4 Press the +X, +Y and T buttons to zero-return all the machine axes.

To cancel the ongoing slitting operation:

- 1 Press any STOP button.
- 2 Press the Reset key.
- 3 Change the CNC mode to MANUAL.
- 4 Cut the chip off from the worksheet (refer to "Punching in the MANUAL mode").
- 5 Zero-return all the machine axes.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

SHUTTING DOWN THE MACHINE

- 1 Press the POWER OFF button on the main control panel.
- 2 Turn the machine circuit breaker switch on the electrical control cabinet to OFF.
- 3 Close the air intake valve and turn off the air compressor.
- 4 Turn off the cooling water unit.
- 5 Turn off the shop power.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

Part IV

Program Management

Program management	IV-2
Registration	IV-2
Memory capacity	IV-2
Setup for program management	IV-3
Reading from an input device	IV-3
Outputting to an output device	IV-4
Direct programming & registration	IV-5
Search.....	IV-6
Check.....	IV-6
Deletion.....	IV-7
Program editing.....	IV-8
Editing	IV-8
Modifying a word.....	IV-8
Inserting a word or block	IV-8
Deleting a word or block.....	IV-8
Background editing	IV-9



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

PROGRAM MANAGEMENT

The registration of CNC programs in the CNC memory and other program management procedures are described in this section.

Registration

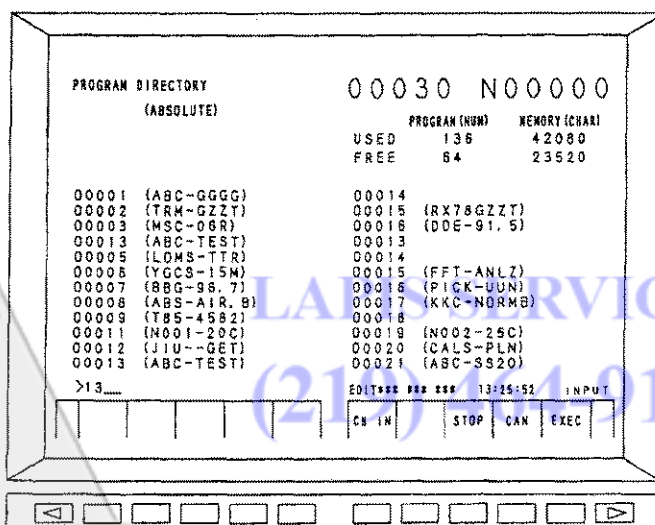
The CNC program to perform a punching operation must first be registered in the CNC memory. The program may be a CNC program read from an input device, or a CNC program programmed directly on the CNC (see "Direct programming & registration" below).

MEMORY CAPACITY

The currently available memory capacity — a number of programs which can be registered and a total number of characters which can be saved — can be confirmed on the PROGRAM DIRECTORY display. Confirm the capacity before registering a program — the information appears in the upper right section of the screen.

The PROGRAM DIRECTORY display can be shown on the screen in the following manner:

- 1 Change the CNC mode to MEMORY.
- 2 Press the Program key.
- 3 Press the right arrowhead softkey.
- 4 Press the DIR softkey.



SETUP FOR PROGRAM MANAGEMENT

The CNC must be set up in order to perform a program management operation — confirm or change the setup as required on the SETTING HANDY display.

For the setup procedures and more information, refer to “Displays” in Part II, Controls.

READING FROM AN INPUT DEVICE

Read the program from an input device and register it in the CNC memory in the following manner:

- 1 Change the CNC mode to EDIT.
- 2 Turn the EDIT PROTECT keyswitch to OFF.
- 3 Press the Program key to show the PROGRAM display.
- 4 Press the DIR softkey to show the PROGRAM DIRECTORY display.
- 5 Press the (OPRT) softkey and then the right arrowhead softkey.
- 6 Connect the input device to the interface on the electrical control cabinet and prepare it for reading.
- 7 Key-in address O and the number for the program to be read.
To read all programs, key-in “O-9999.”
- 8 Press the READ softkey.
- 9 Press the EXEC softkey to start reading the program.
- 10 When the reading of the program is completed, press the STOP softkey.
- 11 Return the EDIT PROTECT keyswitch to ON.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

OUTPUTTING TO AN OUTPUT DEVICE

Output the program to an output device in the following manner:

- 1 Change the CNC mode to EDIT.
- 2 Press the Program key to show the PROGRAM display.
- 3 Press the DIR softkey to show the PROGRAM DIRECTORY display.
- 4 Press the (OPRT) softkey and then the right arrowhead softkey.
- 5 Connect the output device to the interface on the electrical control cabinet and prepare it for outputting.
- 6 Key-in address O and the number for the program to be output. To output all programs, key-in "O-9999."
- 7 Press the PUNCH softkey.
- 8 Press the EXEC softkey to start outputting the program.
- 9 When the outputting of the program is completed, press the STOP softkey.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

DIRECT PROGRAMMING & REGISTRATION

A CNC program can be composed and registered directly in the CNC memory in the following manner:

- 1 Change the CNC mode to EDIT.
- 2 Turn the EDIT PROTECT keyswitch to OFF.
- 3 Press the Program key to show the PROGRAM display.
- 4 Press the DIR softkey to show the PROGRAM DIRECTORY display.
- 5 Key-in "O" and a number (0001 to 7999) for the program to be entered.
- 6 Press the Insert key.
- 7 Key-in data for the first block of the program. Up to 32 characters can be entered in one line — change the line if the block has more characters (do not divide a word between the two lines at this time). Any erroneously keyed-in character can be erased by using the Cancel key.
- 8 Press the Insert key to input the block if the keyed-in data are correct.
- 9 Key-in data for the next block and press the Insert key. Enter data for the remaining blocks in the same manner.
- 10 Press the REWIND softkey or the Reset key to return the cursor to the start of the program to end the session.
The completed program is now registered in the CNC memory and its status is the same as any other registered programs.
- 11 Return the EDIT PROTECT keyswitch to ON.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

Search

A registered program can be searched for in the following manner:

Show the PROGRAM or PROGRAM DIRECTORY display in the MEMORY or EDIT mode.

On the PROGRAM display, key-in address O and the program number and then press the Cursor down (↓) key or the O-SRH softkey.

The shown program can be changed to the successive program by keying-in "O" and pressing the Cursor down (↓) key or the O-SRH softkey.

All registered programs can be shown and changed successively in this manner.

On the PROGRAM DIRECTORY display, key-in just the program number and then press the O-SRH softkey.

Check

The registered program can be checked for format errors and axial position data prior to its execution — without operating the machine — in the following manner:

- 1 Confirm that the lights of the +X, +Y and T buttons on the main control panel are on. Take necessary steps if they are not on.
- 2 Change the CNC mode to MEMORY.
- 3 Press the Program key.
- 4 Show the contents of the registered program on the display and place the cursor at the start of the program.
- 5 Press the TEST button, lighting its light.
- 6 To check the program block by block, press the SINGLE button, lighting its light. To show the current position display during the checking process, change the display to the ACTUAL POSITION display.
- 7 Press the START button to start the check.
- 8 The check will be interrupted if an error has been found, which will be identified by the corresponding alarm number on the screen. (Refer to Part VI, Alarms.)
Correct the error as described below.
 - (1) Press the Reset key on the main control panel.
 - (2) Press the TEST button, unlighting its light.

- (3) Change the CNC mode to EDIT, turn the EDIT PROTECT keyswitch to OFF, and then modify the program (see "Program editing" in this Part for detail.)

The canceled check process cannot be resumed and check must be repeated from the beginning.

- 9 When the check has been completed, end the check as described below.
 - (1) Press the TEST button on the main control panel, unlighting its light.
 - (2) Change the CNC mode to MANUAL.
 - (3) Move the carriage and the table about 200mm (8") away from their origins by using the -X and -Y buttons on the main control panel.
 - (4) Change the CNC mode to RETRACT.
 - (5) Zero-return the carriage and table.

Deletion

Registered programs can be erased and deleted from the CNC memory in the following manner:

With the EDIT PROTECT keyswitch turned to OFF in the EDIT mode, show the PROGRAM or PROGRAM DIRECTORY display.

Key-in address O (not necessary on the PROGRAM DIRECTORY display) and the program number, and then press the Delete key. To delete all registered programs, key-in -9999 for the program number.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

PROGRAM EDITING

Procedures for program editing and background program editing are described in this section.

Editing

To edit a registered program, change to the EDIT mode, turn the EDIT PROTECT keyswitch to OFF, show the PROGRAM display, and show the contents of the program on the display.

MODIFYING A WORD

Place the cursor under the address character of the word which must be changed. Or search for the word by keying-in its address character, or the whole word, and pressing the Cursor down (↓) key. The search will be made toward the end of the program from the current cursor position and then the cursor will be placed under the address character which has been found first, or the address character of the specified word. (The cursor can be returned to the start of the program by pressing the REWIND softkey or the Reset key.)

With the cursor placed under the address character of the word to be changed, key-in the new address character and data and then press the Alter key. Modification can be made for one word at a time.

INSERTING A WORD OR BLOCK

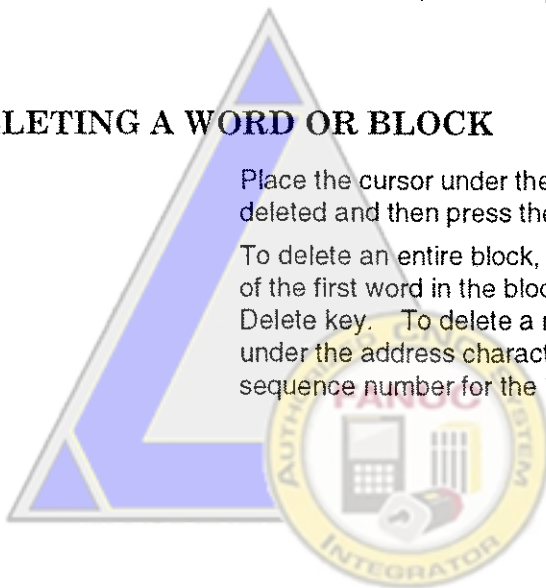
Place the cursor under the address character of the word after which a new word or words must be inserted, key-in the new word or words, and then press the Insert key.

To insert a new block, place the cursor under the EOB code (;) of the block after which a new block or blocks must be inserted, key-in the new block or blocks, and then press the Insert key.

DELETING A WORD OR BLOCK

Place the cursor under the address character of the word which must be deleted and then press the Delete key.

To delete an entire block, place the cursor under the address character of the first word in the block, press the EOB key, and then press the Delete key. To delete a number of successive blocks, place the cursor under the address character of the first word in the first block, key-in the sequence number for the last block, and then press the Delete key.



LAPIS SERVICES

(219) 464-9131

Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Background editing

Program editing, search, deletion, and registration can be performed during the operation in any CNC mode by using the background editing function. The program currently in use in the foreground cannot be edited by this function.

ENABLING THE FUNCTION

The background editing function can be enabled in the following manner:

- 1 Press the Program key.
- 2 Press the (OPRT) softkey.
- 3 Press the BG-EDT softkey.

The word "BG-EDT" appears in the lower center of the screen.

EDITING & OTHER PROCEDURES

Editing, search, deletion, and registration procedures are the same as those described previously except that it is not necessary to change the mode.

TERMINATING THE FUNCTION

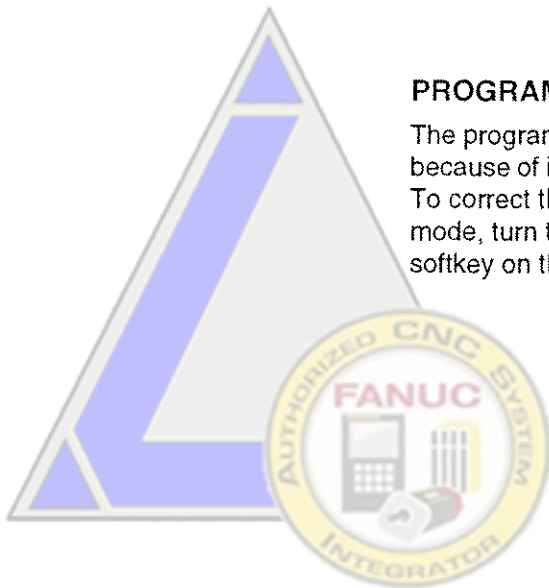
To end background editing, press the (OPRT) softkey and then the BG-END softkey.

NOTE

- Do not turn off the CNC without terminating the background editing function. Doing so will erase any program which has been newly registered during background editing.

PROGRAM REGISTRATION

The program which has been background-edited may not be registered because of its size, which has been expanded during background editing. To correct the condition, terminate the function, change to the EDIT mode, turn the EDIT PROTECT keyswitch to ON, and press the DIR softkey on the PROGRAM display.



LAPIS SERVICES
Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

PROGRAM READING OR OUTPUTTING

Press the EOB key to interrupt program reading or outputting during background editing.

NOTE

- Do not press the Reset key to interrupt program reading or outputting. Doing so will stop the on-going operation.

ALARMS

An alarm will be caused if background editing is attempted on the program currently in use in the foreground.

The message "BP/S ALARM" appears in the lower left section of the screen when an alarm is caused during background editing. The alarm can be reset, erasing the message, by pressing any address or numeric key.

NOTE

- Do not press the Reset key to reset the alarm caused during background editing. Doing so will stop the on-going operation.

Alarms caused in the background will not affect the operation in the foreground — unless the Reset key is used to reset the alarm. Alarms caused in the foreground will not affect the operation in the background either.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

Part V

PHNC Parameters

PHNC parameters.....	V-2
"FORMING" parameters.....	V-3
"MARKING" parameters.....	V-7
"KNOCKOUT" parameters	V-11
"SLITTING" parameters.....	V-15
"PUNCH-NIBBLE" parameters	V-19
"TIME DATA" parameters	V-22
"TOOLING DATA" parameters.....	V-24



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

PHNC PARAMETERS

The PHNC parameters control the operation of the machine's press function. The parameters are grouped into the following three categories:

PRESS PATTERN

Selection can be made from six press modes: forming, marking, knockout, slitting, punch and nibble. A number of M-code commands are allocated to each mode and parameters can be set for each command to control the press operation.

TIME DATA

The current time and three timers can be set to control the operating time periods for the machine in a 24-hour period.

TOOLING DATA

Parameters can be set to register tooling information for each turret station so that programming and punch height adjustment can be simplified and the ram speed in the punching press mode can be optimized.

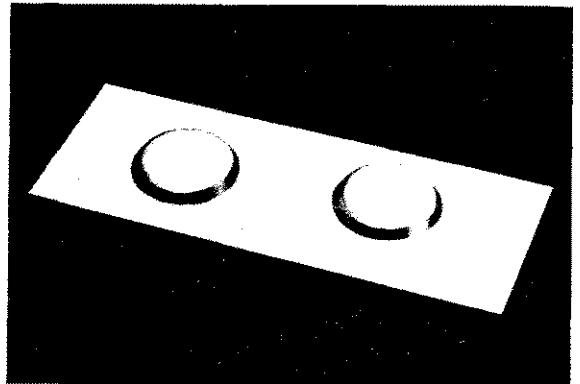
These PHNC parameters must be set according to the press requirements before operating the machine. See below for the procedure.

PHNC parameter categories

PHNC MENU					
1. PRESS PATTERN	2. TIME DATA	3. TOOLING DATA			
	1. CURRENT TIME	1. T101			
	2. TIMER-1	2. T102			
	3. TIMER-2	:			
	4. TIMER-3	31. T207			
1. FORMING	2. MARKING	3. KNOCKOUT	4. SLITTING	5. PUNCH	6. NIBBLE
1. M510	1. M560	1. M502	1. M506	1. M500	1. M12
2. M511	2. M561	2. M503	2. M507	2. M501	
3. M512	3. M562	3. M504	3. M508		
4. M513	4. M563	4. M505	4. M509		
5. M514	5. M564	5. M570			
6. M515	6. M565	6. M571			
7. M516	7. M566	7. M572			
8. M517	8. M567	8. M573			
:	9. M568	9. M574			
50. M559	10. M569	10. M575			
51. M800					
52. M801					
:					
250. M999					

“FORMING” parameters

“Forming” is a press process in which a worksheet is formed into a shape (see photo). The process can be registered and executed by using M-code commands — each command is defined by eight parameters (see below) and up to 250 commands (M510 to M559 and M800 to M999) can be used at a time.



TOP POS parameter

Distance between the top of the worksheet and the tip of the punch in its standby position when the same tool is used continually.

SLOW POSITION parameter

Distance above the top of the worksheet at which the punch decelerates after it has been rapidly lowered.

BOTTOM POS parameter

Descending distance for the punch below the top of the worksheet.

DIE HEIGHT parameter

Stroke of the ram from the top dead center required until the punch tip touches the die top.

MAX TONNAGE parameter

Maximum tonnage which can be applied to the worksheet.

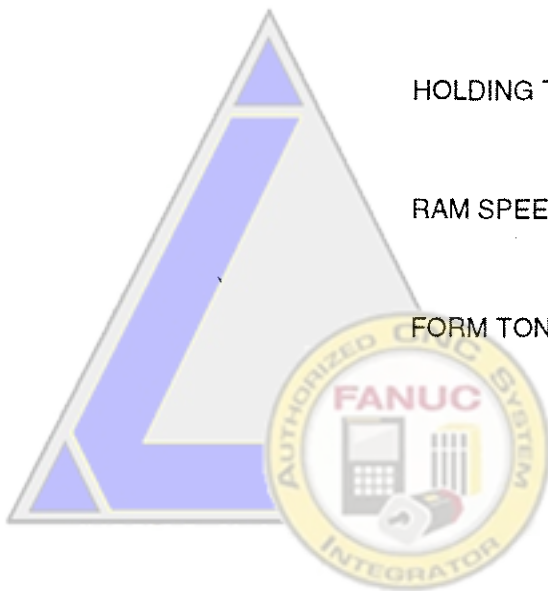
HOLDING TIME parameter

Period of time during which the punch is held at its bottom position.

RAM SPEED parameter

Descending speed for the punch after deceleration.

FORM TONNAGE parameter Tonnage required for forming.



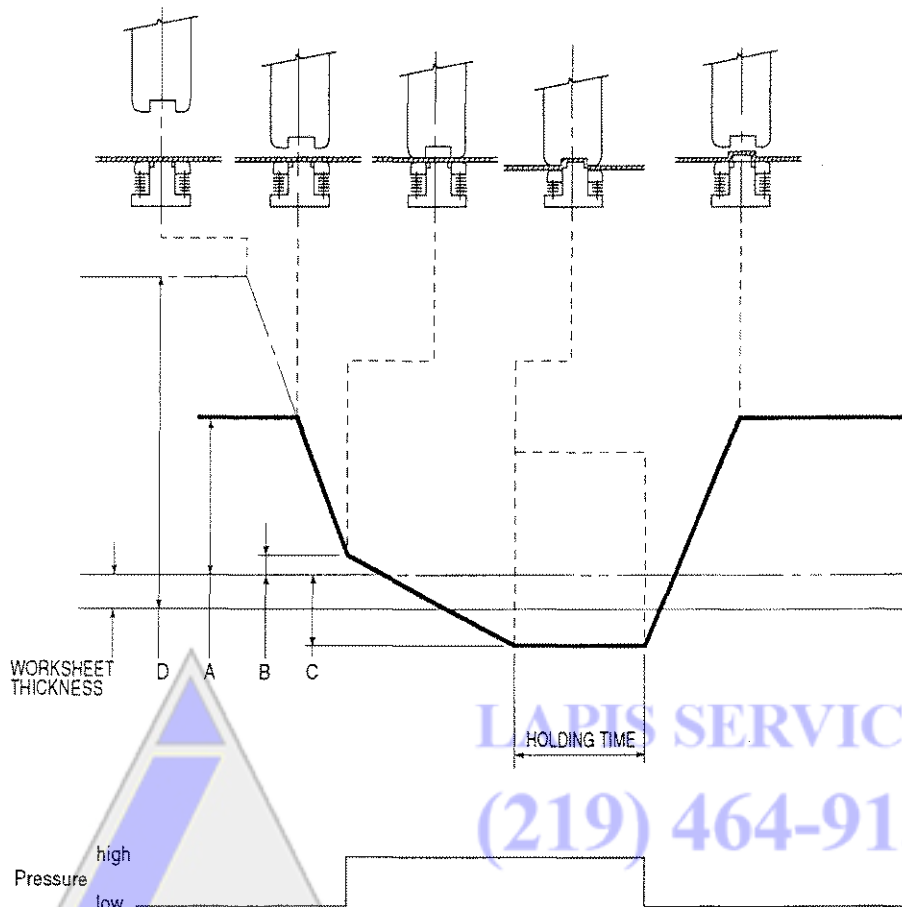
LAPIS SERVICES
(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

PARAMETER	RANGE OF SETTING	RECOMMENDED SETTING
TOP POS	0 to 35 mm (0 to 1.378")	Formed height plus 5 to 8 mm (0.197 to 0.315")
SLOW POSITION	0 to 35 mm (0 to 1.378")	0
BOTTOM POS	0 to 30 mm (0 to 1.181")	0 (for trial)
DIE HEIGHT	5 to 35 mm (0.197 to 1.378")	267 mm (10.512") minus punch & die heights
MAX TONNAGE	0 to 382.2 kN (0 to 39 metric tons or 0 to 42.9 U.S. tons)	196 kN (20 metric tons or 22 U.S. tons)
HOLDING TIME	0 to 9999 ms	100 ms
RAM SPEED	High or low	High (for trial)
FORM TONNAGE	0 to 382.2 kN (0 to 39 metric tons or 0 to 42.9 U.S. tons)	176.4 kN (18 metric tons or 19.8 U.S. tons)

Press pattern



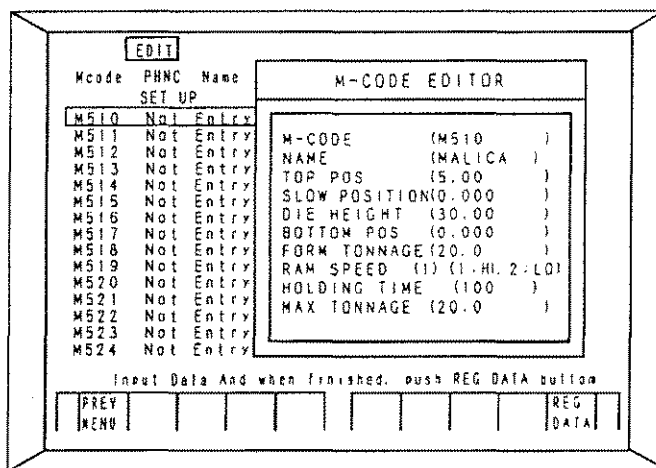
LAPIS SERVICES
(219) 464-9131

Amada Support
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Set the "forming" parameters in the following manner:

- 1 Turn the MODE CHANGE keyswitch to ON, change the CNC mode to MDI, and then return the keyswitch to OFF.
- 2 Turn the EDIT PROTECT keyswitch to OFF.
- 3 Press the Position key.
- 4 Press the PHNC softkey to show the PHNC Menu display on the screen.
- 5 Shift the cursor to item Press Pattern Edit and then press the Input key to show the Press Pattern Menu display.
- 6 Shift the cursor to item Forming Pattern and then press the Input key.
- 7 Use the PAGE UP and DOWN softkeys to show the page containing the required M-code commands.
- 8 Shift the cursor to an M-code command with the "Not Entry" prompt, for which parameters must be set, and then press the Input key. The M-CODE EDITOR display will appear.



NOTE

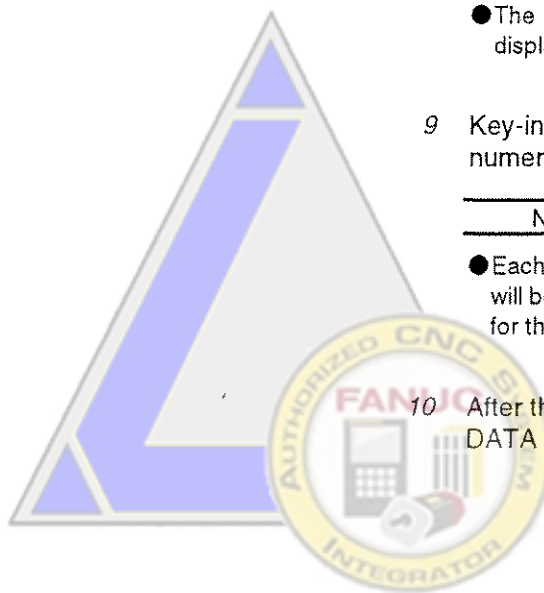
- The PREV.MENU softkey returns the current display to the previous display.

- 9 Key-in parameters for the M-code command by using address and numeric keys and press the Input key.

NOTE

- Each parameter must be set within a predetermined range — an error will be caused if an entry of out-of-range data is attempted. See above for the range of parameter setting.

- 10 After the entry of parameters has been completed, press the REG. DATA softkey to store them in the memory.



LAPIS SERVICES
(219) 464-9131

Amada Support,
Fanuc Control Upgrades.

LAPISINC.COM

Returning to default settings

M-code commands, for which parameters have been set previously, are shown with an "Entry" prompt in Step 6 above. Return the parameters set for an M-code command to the default settings as follows:

- 1 Shift the cursor to an M-code command with the "Entry" prompt and then press the Delete key.
- 2 Press the YES softkey to return to the default settings. (To cancel this operation, press the NO softkey.)

The parameter settings can be determined or verified in the following manner:

- 1 Prepare the following program and register it in the memory to perform trial forming:

G06 A__B__;

(A: Worksheet thickness, 0.1 to 6.4 mm (0.002 to 0.253"), B: Worksheet material, 0: Mild steel, 1: Stainless steel, 2: Aluminum)

G92 X__Y__;

M__; (M-code for which parameters have been set)

G90 X__Y__T__;

M13;

G50;

- 2 Perform trial forming in the MEMORY mode. (Refer to "MEMORY mode operation" in Part III, Operation.)
- 3 Check the formed worksheet. Revise the BOTTOM POS setting so that the required forming height is obtained. Revise the TOP POS setting if the punch interferes with the formed worksheet.
- 4 Repeat the procedure until ideal parameter settings are achieved.
- 5 Upon completion of the procedure, return the EDIT PROTECT keyswitch to ON.



LAPIS SERVICES
(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

“MARKING” parameters

“Marking” is a press process in which a shape is embossed on a worksheet (see photo). The process can be registered and executed by using M-code commands — each command is defined by four parameters (see below) and ten M-code commands (M560 to M569) can be used at a time.



TOP POS parameter

Distance between the top of the worksheet and the tip of the punch in its standby position when the same tool is used continually.

SLOW POSITION parameter

Distance above the top of the worksheet at which the punch decelerates after it has been rapidly lowered.

BOTTOM POS parameter

Descending distance for the punch below the top of the worksheet.

DIE HEIGHT parameter

Stroke of the ram from the top dead center required until the punch tip touches the die top.



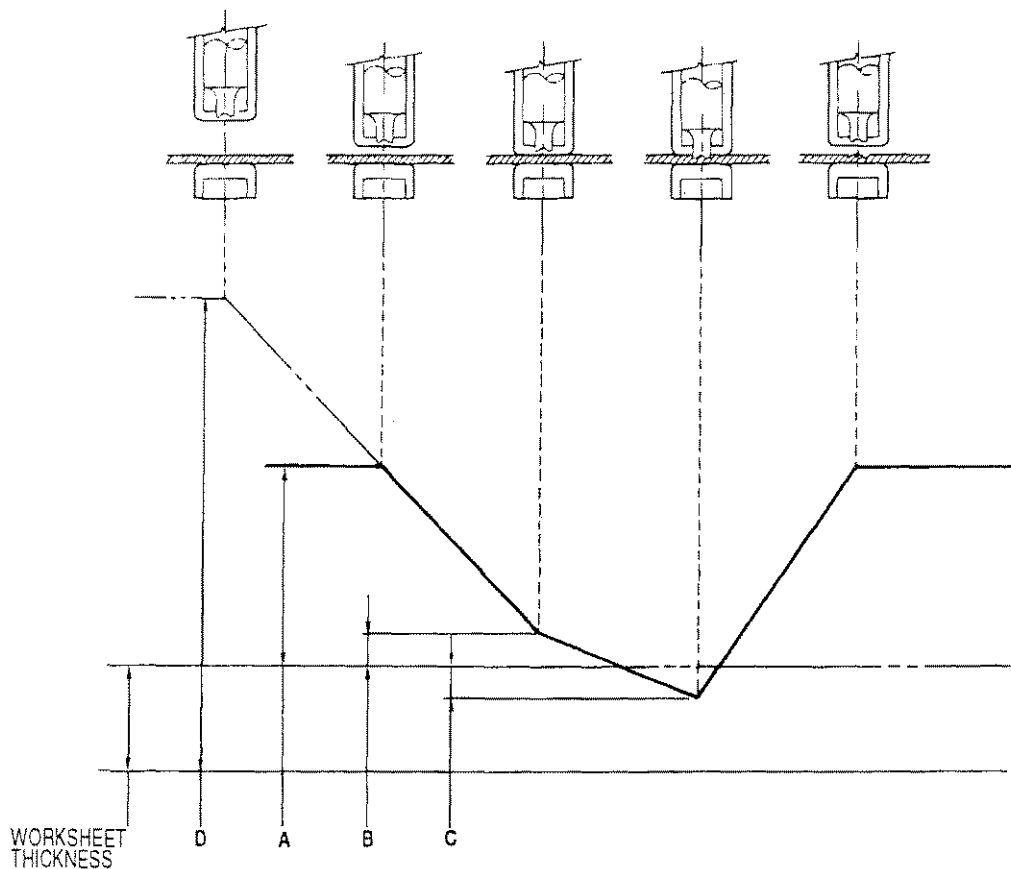
LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

<u>PARAMETER</u>	<u>RANGE OF SETTING</u>	<u>RECOMMENDED SETTING</u>
TOP POS	2 to 35 mm (0.079 to 1.378")	5 to 8 mm (0.197 to 0.315")
SLOW POSITION	0 to 35 mm (0 to 1.378")	0
BOTTOM POS	0 to 30 mm (0 to 1.181")	0 (for trial)
DIE HEIGHT	5 to 35 mm (0.197 to 1.378")	267 mm (10.512") minus punch & die heights

Press pattern



Pressure high
low

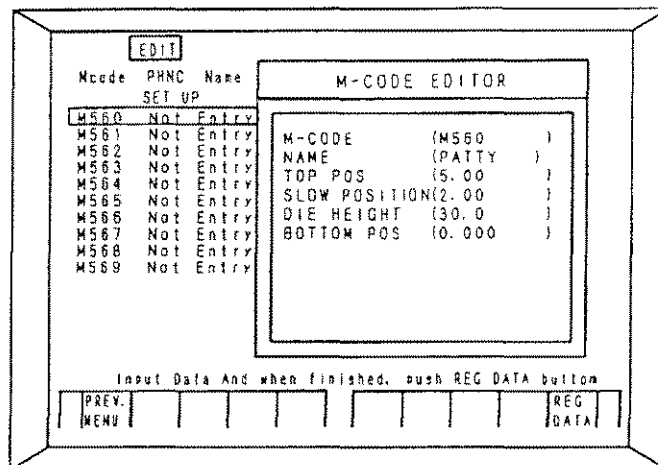
LAPIS SERVICES
(219) 464-9131

A: Top pos
B: Slow position
C: Bottom pos
D: Die height

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

Set the "marking" parameters in the following manner:

- 1 Turn the MODE CHANGE keyswitch to ON, change the CNC mode to MDI, and then return the keyswitch to OFF.
- 2 Turn the EDIT PROTECT keyswitch to OFF.
- 3 Press the Position key.
- 4 Press the PHNC softkey to show the PHNC Menu display on the screen.
- 5 Shift the cursor to item Press Pattern Edit and then press the Input key to show the Press Pattern Menu display.
- 6 Shift the cursor to item Marking Pattern and then press the Input key.
- 7 Shift the cursor to an M-code command with the "Not Entry" prompt, for which parameters must be set, and then press the Input key. The M-CODE EDITOR display will appear.



NOTE

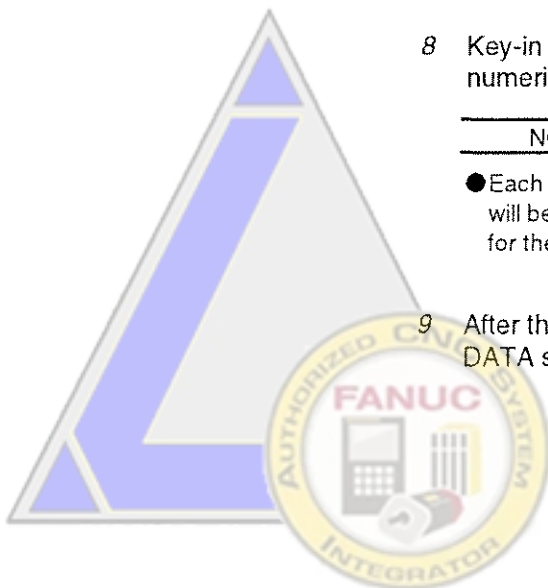
- The PREV.MENU softkey returns the current display to the previous display.

- 8 Key-in parameters for the M-code command by using address and numeric keys and press the Input key.

NOTE

- Each parameter must be set within a predetermined range — an error will be caused if an entry of out-of-range data is attempted. See above for the range of parameter setting.

- 9 After the entry of parameters has been completed, press the REG. DATA softkey to store them in the memory.



Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Returning to default settings

M-code commands, for which parameters have been set previously, are shown with an "Entry" prompt in Step 6 above. Return the parameters set for an M-code command to the default settings as follows:

- 1 Shift the cursor to an M-code command with the "Entry" prompt and then press the Delete key.
- 2 Press the YES softkey to return to the default settings. (To cancel this operation, press the NO softkey.)

The parameter settings can be determined or verified in the following manner:

- 1 Prepare the following program and register it in the memory to perform trial marking:

G06 A___B___;

(A: Worksheet thickness, 0.1 to 6.4 mm (0.002 to 0.253"), B: Worksheet material, 0: Mild steel, 1: Stainless steel, 2: Aluminum)

G92 X___Y___;

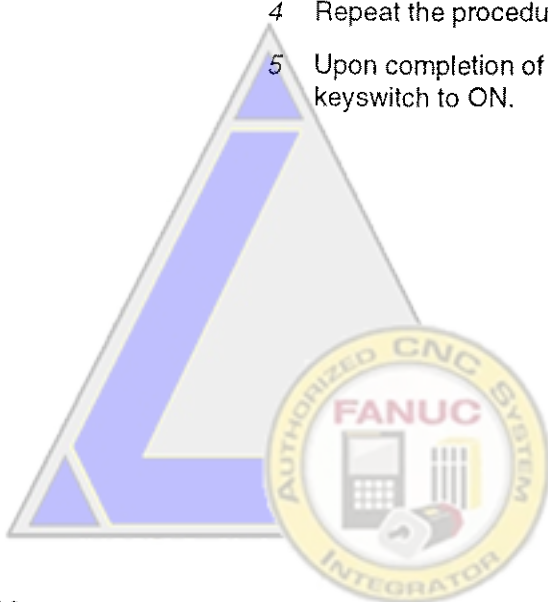
M___; (M-code for which parameters have been set)

G90 X___Y___T___;

M13;

G50;

- 2 Perform trial marking in the MEMORY mode. (Refer to "MEMORY mode operation" in Part III, Operation.)
- 3 Check the embossed worksheet. Revise the BOTTOM POS setting by increasing or decreasing 0.1 mm (0.004") at a time so that the required depth is obtained.
- 4 Repeat the procedure until ideal parameter settings are achieved.
- 5 Upon completion of the procedure, return the EDIT PROTECT keyswitch to ON.



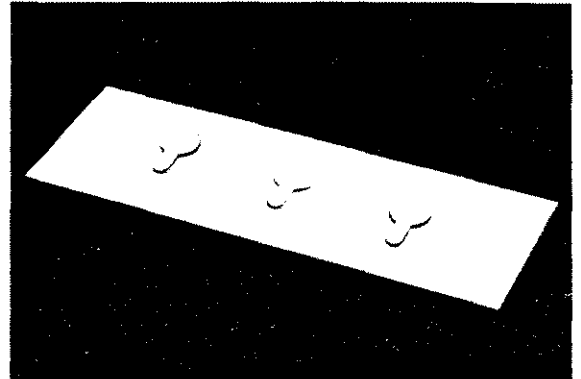
LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

“KNOCKOUT” parameters

“Knockout” is a press process in which a hole is punched in a worksheet but the punched-out part partially remains attached to the worksheet (see photo). The process can be registered and executed by using M-code commands — each command is defined by four parameters (see below) and ten M-code commands (M502 to M505 and M570 to M575) can be used at a time.



TOP POS parameter

Distance between the top of the worksheet and the tip of the punch in its standby position when the same tool is used continually.

SLOW POSITION parameter

Distance above the top of the worksheet at which the punch decelerates after it has been rapidly lowered.

BOTTOM POS parameter

Descending distance for the punch below the top of the die.

DIE HEIGHT parameter

Stroke of the ram from the top dead center required until the punch tip touches the die top.



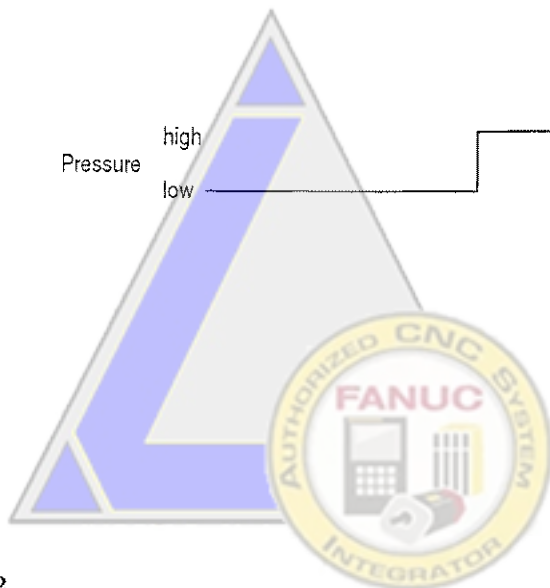
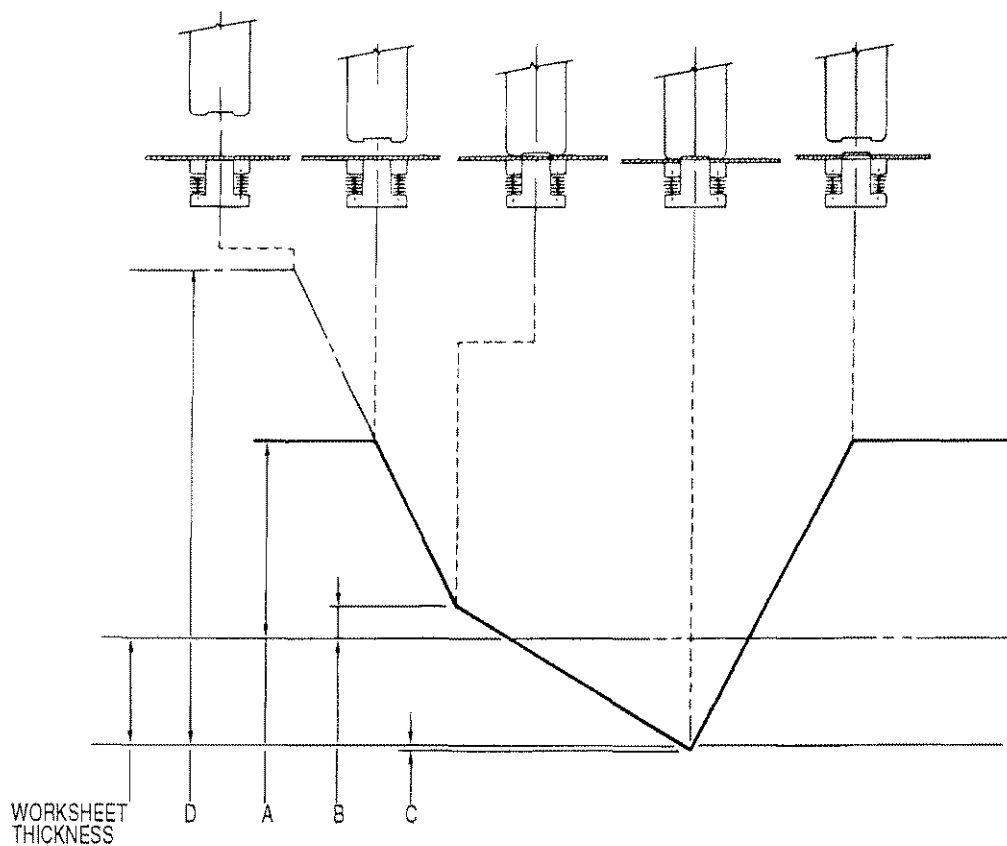
LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

PARAMETER	RANGE OF SETTING	RECOMMENDED SETTING
TOP POS	0 to 35 mm (0 to 1.378")	Worksheet thickness plus 5 to 8 mm (0.197 to 0.315")
SLOW POSITION	0 to 35 mm (0 to 1.378")	0
BOTTOM POS	0 to 30 mm (0 to 1.181")	0 (for trial)
DIE HEIGHT	5 to 35 mm (0.197 to 1.378")	267 mm (10.512") minus punch & die heights

Press pattern



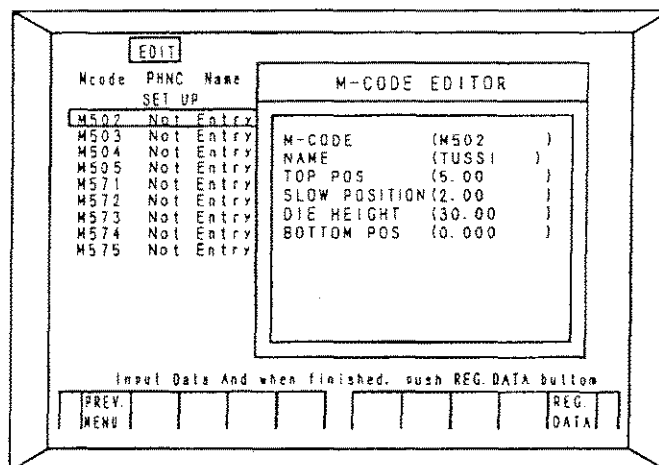
LAPIS SERVICES
(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Set the "knockout" parameters in the following manner:

- 1 Turn the MODE CHANGE keyswitch to ON, change the CNC mode to MDI, and then return the keyswitch to OFF.
- 2 Turn the EDIT PROTECT keyswitch to OFF.
- 3 Press the Position key.
- 4 Press the PHNC softkey to show the PHNC Menu display on the screen.
- 5 Shift the cursor to item Press Pattern Edit and then press the Input key to show the Press Pattern Menu display.
- 6 Shift the cursor to item Knockout Pattern and then press the Input key.
- 7 Shift the cursor to an M-code command with the "Not Entry" prompt, for which parameters must be set, and then press the Input key. The M-CODE EDITOR display will appear.



NOTE

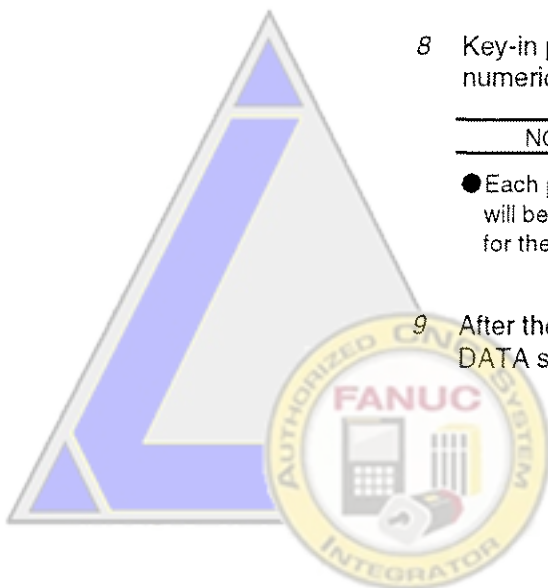
- The PREV.MENU softkey returns the current display to the previous display.

- 8 Key-in parameters for the M-code command by using address and numeric keys and press the Input key.

NOTE

- Each parameter must be set within a predetermined range — an error will be caused if an entry of out-of-range data is attempted. See above for the range of parameter setting.

- 9 After the entry of parameters has been completed, press the REG. DATA softkey to store them in the memory.



Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Returning to default settings

M-code commands, for which parameters have been set previously, are shown with an "Entry" prompt in Step 6 above. Return the parameters set for an M-code command to the default settings as follows:

- 1 Shift the cursor to an M-code command with the "Entry" prompt and then press the Delete key.
- 2 Press the YES softkey to return to the default settings. (To cancel this operation, press the NO softkey.)

The parameter settings can be determined or verified in the following manner:

- 1 Prepare the following program and register it in the memory to perform trial knockout:

G06 A__B__;

(A: Worksheet thickness, 0.1 to 6.4 mm (0.002 to 0.253"), B: Worksheet material, 0: Mild steel, 1: Stainless steel, 2: Aluminum)

G92 X__Y__;

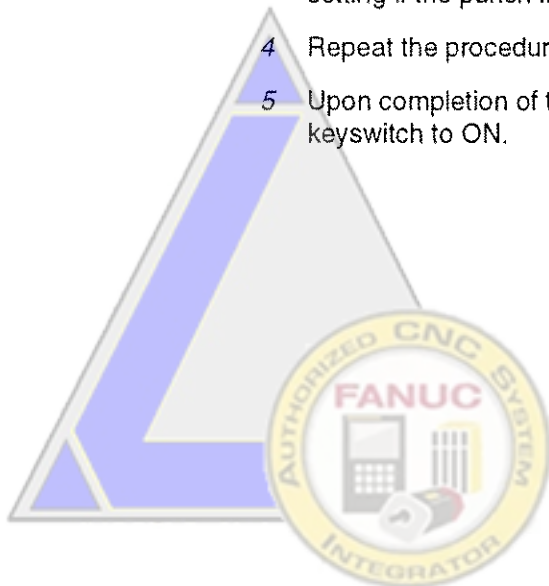
M__; (M-code for which parameters have been set)

G90 X__Y__T__;

M13;

G50;

- 2 Perform trial knockout in the MEMORY mode. (Refer to "MEMORY mode operation" in Part III, Operation.)
- 3 Check the punched worksheet. Revise the BOTTOM POS setting gradually at a time so that the punched-out part remains attached to the worksheet in an appropriate amount. Revise the TOP POS setting if the punch interferes with the punched-out part.
- 4 Repeat the procedure until ideal parameter settings are achieved.
- 5 Upon completion of the procedure, return the EDIT PROTECT keyswitch to ON.



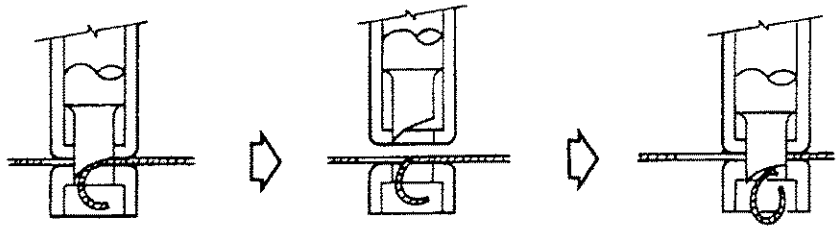
LAPIS SERVICES
(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

“SLITTING” parameters

“Slitting” is a press process in which holes are punched continuously without leaving seams between holes (see figure). The process can be registered and executed by using M-code commands — each command is defined by four parameters (see below) and four M-code commands (M506 to M509) can be used at a time.



TOP POS parameter

Distance between the top of the worksheet and the tip of the punch in its standby position when the same tool is used continually.

BOTTOM POS parameter

Descending distance for the punch below the top of the die during slitting.

DIE HEIGHT parameter

Stroke of the ram from the top dead center required until the punch tip touches the die top.

BOTTOM POS END parameter

Descending distance for the punch below the top of the die to where cutoff is made.



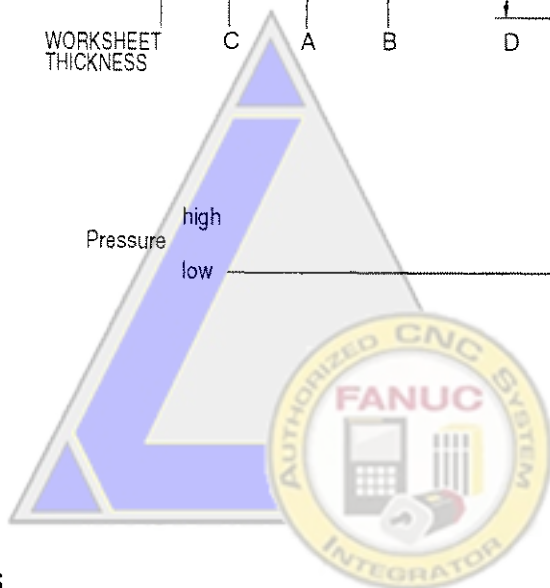
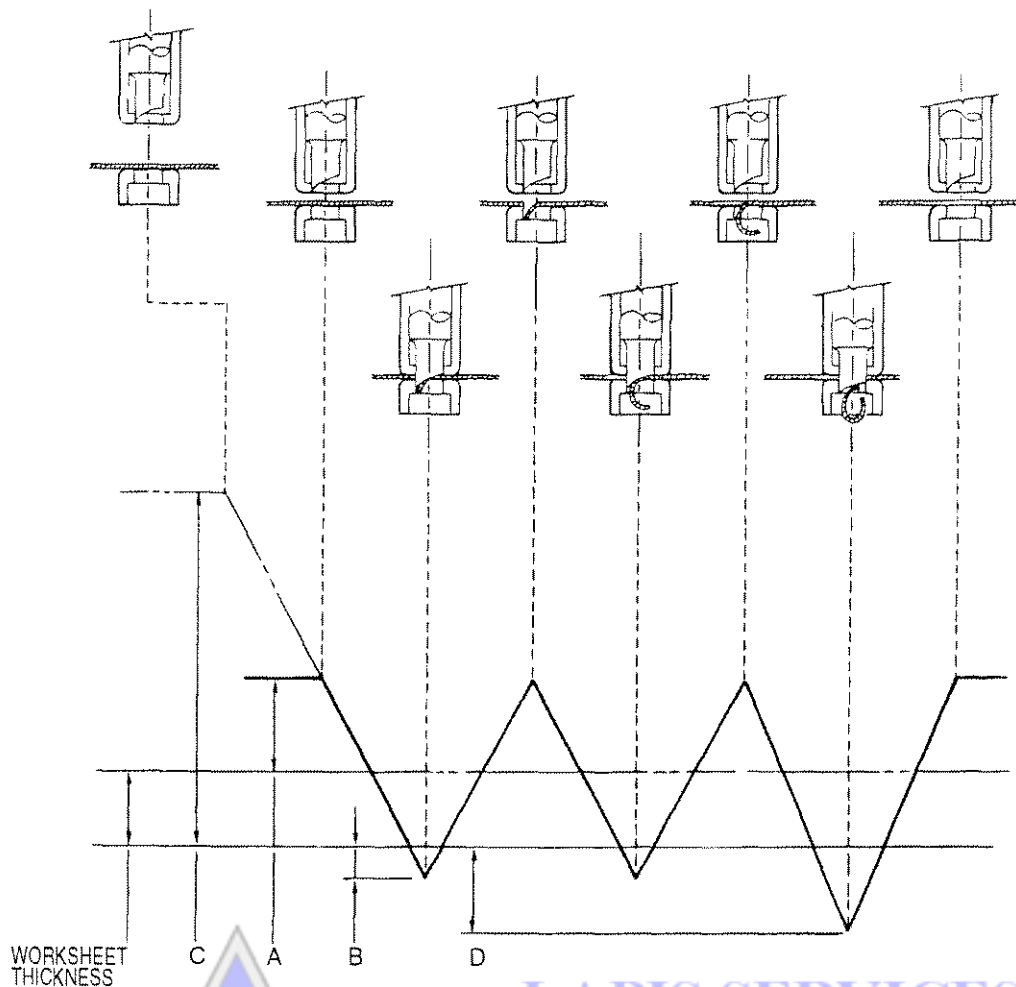
LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

PARAMETER	RANGE OF SETTING	RECOMMENDED SETTING
TOP POS	0 to 35 mm (0 to 1.378")	5 to 8 mm (0.197 to 0.315")
BOTTOM POS	0 to 15 mm (0 to 0.590")	4 to 5 mm (0.157 to 0.197")
DIE HEIGHT	5 to 35 mm (0.197 to 1.378")	267 mm (10.512") minus punch & die heights
BOTTOM POS END	0 to 15 mm (0 to 0.590")	39 mm (1.535") minus die height

Press pattern



LAPIS SERVICES
(219) 464-9131

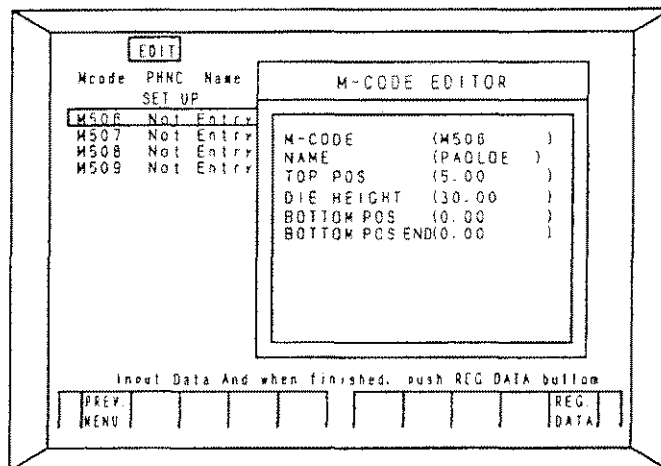
Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

A: Top pos
B: Bottom pos
C: Die height
D: Bottom pos end

LAPISINC.COM

Set the "slitting" parameters in the following manner:

- 1 Turn the MODE CHANGE keyswitch to ON, change the CNC mode to MDI, and then return the keyswitch to OFF.
- 2 Turn the EDIT PROTECT keyswitch to OFF.
- 3 Press the Position key.
- 4 Press the PHNC softkey to show the PHNC Menu display on the screen.
- 5 Shift the cursor to item Press Pattern Edit and then press the Input key to show the Press Pattern Menu display.
- 6 Shift the cursor to item Slotting Pattern and then press the Input key.
- 7 Shift the cursor to an M-code command with the "Not Entry" prompt, for which parameters must be set, and then press the Input key. The M-CODE EDITOR display will appear.



NOTE

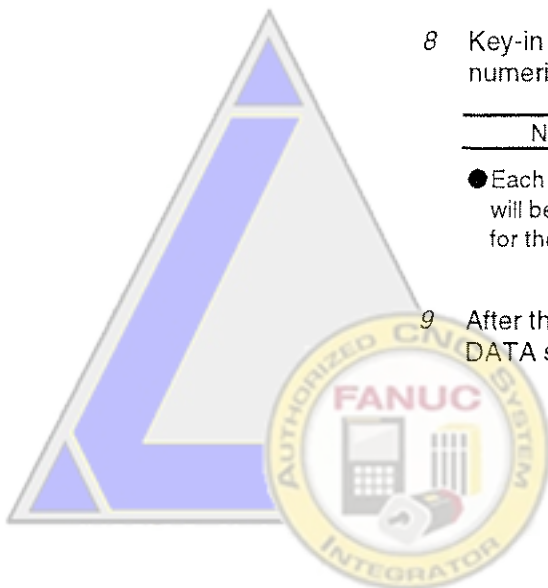
- The PREV.MENU softkey returns the current display to the previous display.

- 8 Key-in parameters for the M-code command by using address and numeric keys and press the Input key.

NOTE

- Each parameter must be set within a predetermined range — an error will be caused if an entry of out-of-range data is attempted. See above for the range of parameter setting.

- 9 After the entry of parameters has been completed, press the REG. DATA softkey to store them in the memory.



Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Returning to default settings

M-code commands, for which parameters have been set previously, are shown with an "Entry" prompt in Step 6 above. Return the parameters set for an M-code command to the default settings as follows:

- 1 Shift the cursor to an M-code command with the "Entry" prompt and then press the Delete key.
- 2 Press the YES softkey to return to the default settings. (To cancel this operation, press the NO softkey.)

The parameter settings can be determined or verified in the following manner:

- 1 Prepare the following program and register it in the memory to perform trial slitting:

G06 A__ B__;

(A: Worksheet thickness, 0.1 to 6.4 mm (0.002 to 0.253"), B: Worksheet material, 0: Mild steel, 1: Stainless steel, 2: Aluminum)

G92 X__ Y__;

M690; (M-code for air-blower operation)

M__; (M-code for which parameters have been set)

G90 X__ Y__ T__;

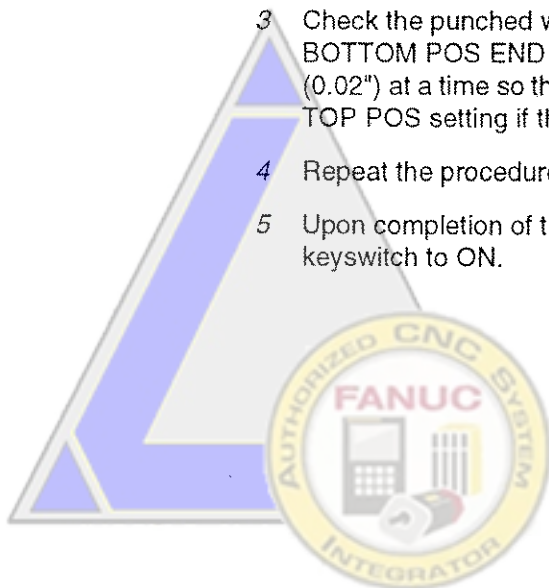
G28 I5. J0 K10 ;

M13;

G28 I5. J0 K1 ;

G50;

- 2 Perform trial slitting in the MEMORY mode. (Refer to "MEMORY mode operation" in Part III, Operation.)
- 3 Check the punched worksheet. Revise the BOTTOM POS and BOTTOM POS END settings by increasing or decreasing 0.5 mm (0.02") at a time so that the required result is obtained. Revise the TOP POS setting if the punch interferes with the worksheet.
- 4 Repeat the procedure until ideal parameter settings are achieved.
- 5 Upon completion of the procedure, return the EDIT PROTECT keyswitch to ON.



**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

“PUNCH-NIBBLE” parameters

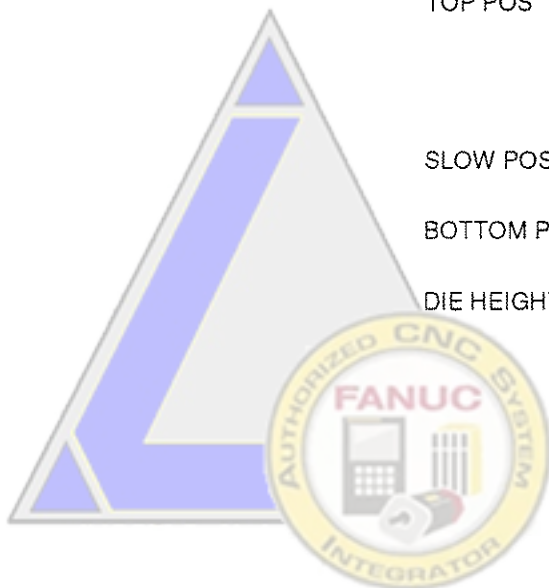
Parameters are preset for normal punching or nibbling mode press operations and it is not necessary to set them even when the “punch-nibble” mode is selected. The process can be registered and executed by using three M-code commands (M500, M501, M12) — M500 and M501 are defined by four preset parameters and M12 is defined by three preset parameters (see below). M500 is for punching by a punch without a shear angle, M501 is for punching by a punch with a shear angle, and M12 is for nibbling.

NOTE

- PUNCH-NIBBLE parameters can be changed in the same manner for other parameters.

TOP POS parameter	Distance between the top of the worksheet and the tip of the punch in its standby position when the same tool is used continually.
SLOW POSITION parameter	(For punching only) Distance above the top of the worksheet at which the punch decelerates after it has been rapidly lowered.
BOTTOM POS parameter	Descending distance for the punch below the top of the die.
DIE HEIGHT parameter	Stroke of the ram from the top dead center required until the punch tip touches the die top.

PARAMETER	RANGE OF SETTING	DEFAULT SETTING
TOP POS	2 to 35 mm (0.079 to 1.378") for M500 and M501, 3 to 35 mm (0.118 to 1.378") for M12	5 mm (0.197") for M500 and M12, 7.5 mm (0.295") for M501
SLOW POSITION	0 to 35 mm (0 to 1.378")	2 mm (0.079") for M500, 4.5 mm (0.177") for M501
BOTTOM POS	0 to 10 mm (0 to 0.394")	2.5 mm (0.098") for M500, M501 and M12
DIE HEIGHT	5 to 35 mm (0.197 to 1.378")	30 mm (1.181") for M500 and M12, 32.5 mm (1.280") for M501

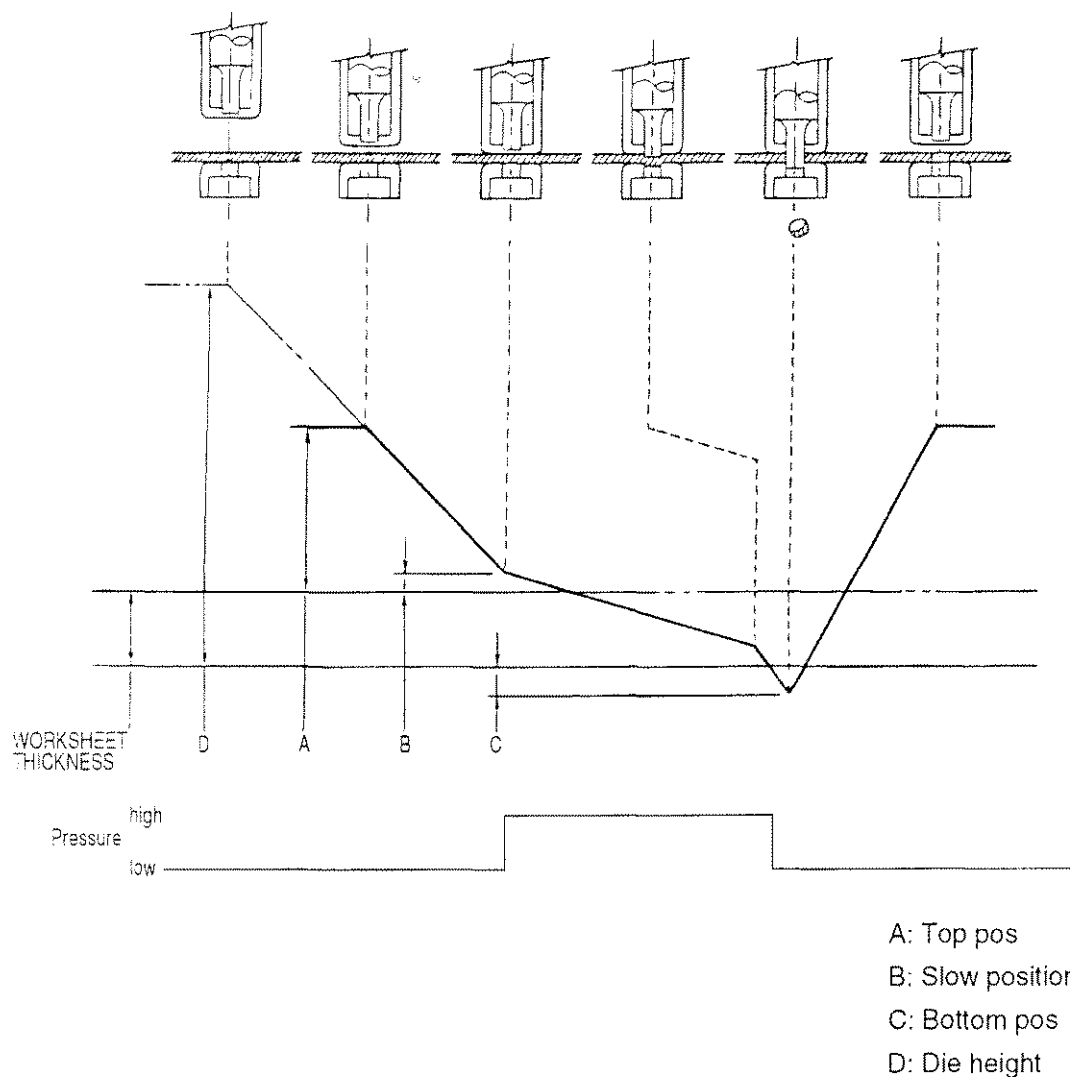


LAPIS SERVICES
(219) 464-9131

Amada Support
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

PUNCHING (M500, M501)



NOTE

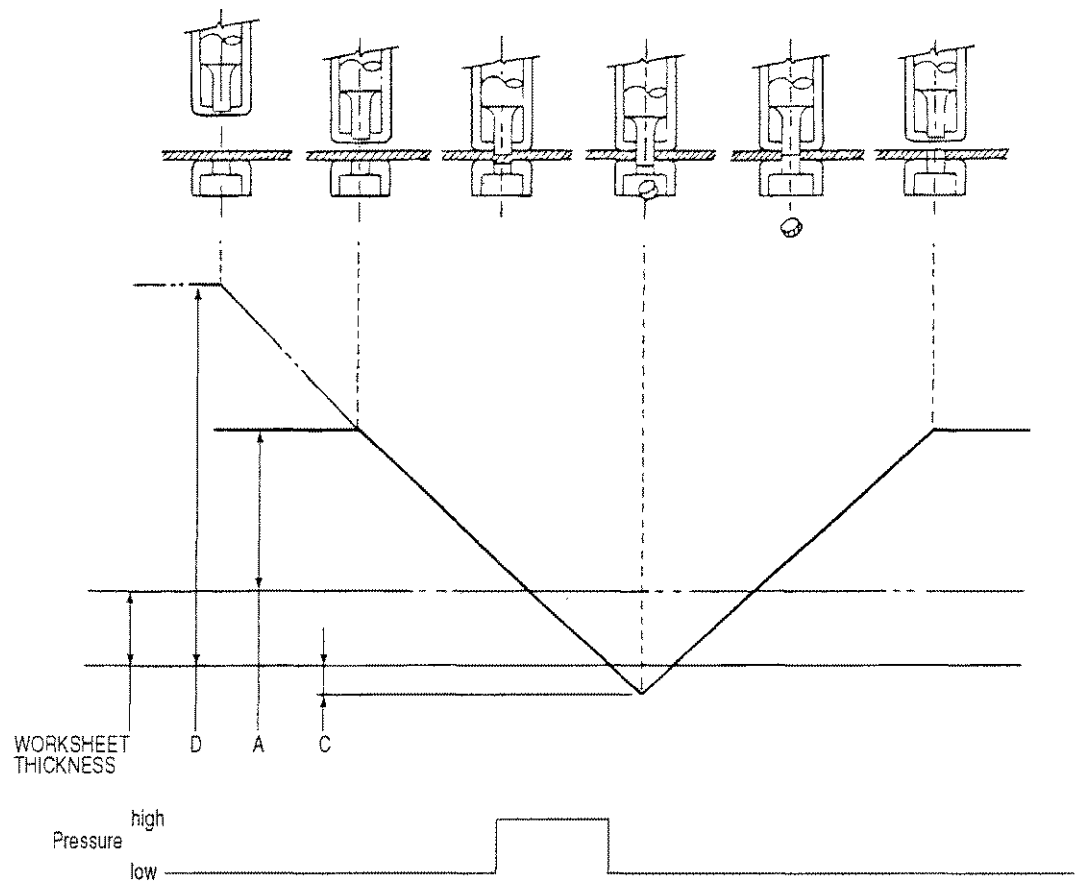
- Hydraulic pressure does not change when the punching tonnage is less than approx. 73 kN (7.5 metric tons or 8.2 U.S. tons).
- When pressing and lighting the RAM SPEED "Auto" button, punching speed can be selected automatically according to the worksheet material and thickness (programmed G06 command) and the tool (Tooling Data parameter settings). If the program is for punching a multiple number of worksheets of identical specifications, the punching speed for the second and successive worksheets will be regulated automatically according to the result of the first worksheet punching.

(219) 464-9131

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

NIBBLING (M12)



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

“TIME DATA” parameters

These parameters are used to set time periods for operations in the MEMORY mode for the reduction of operation noise — when M500 or M501 is programmed and the RAM SPEED “Auto” button is lighted.

The Time parameter sets the current time; the Night start time parameter (Timer-1) sets the night operation start time, or the end of the daytime operation; the Midnight time parameter (Timer-2) sets the midnight operation start time, or the end of the night operation; and the End time parameter (Timer-3) sets the daytime operation start time, or the end of the midnight operation.

The use of M681 to M683 punching-speed commands in the program also affects punching speeds in addition to the settings on the screen:

Operation:	DAYTIME	NIGHT	MIDNIGHT
Starting timer:	Timer 3	Timer 1	Timer 2
M681-683:	As programmed	As programmed	Lowest speed*
No M681-683:	High speed**	Med. speed**	Low speed**

* Same as RAM SPEED “F3” button function.

** Automatically controlled according to worksheet material and thickness and tool size.

Set the “Time Data” parameters in the following manner:

- 1 Turn the MODE CHANGE keyswitch to ON, change the CNC mode to MDI, and then return the keyswitch to OFF.
- 2 Turn the EDIT PROTECT keyswitch to OFF.
- 3 Press the Position key.
- 4 Press the PHNC softkey to show the PHNC Menu display on the screen.
- 5 Shift the cursor to item Time Data and then press the Input key to show the TIME INFO display.

TIME INFO

Time	(14:15)
Night start time	(00:00)
Midnight time	(00:00)
End time	(00:00)

AUTO TIMER ON/OFF

PREV. MENU TIMER ON TIMER OFF

NOTE

- The PREV.MENU softkey returns the current display to the previous display.

- 6 Shift the cursor to an item for which time must be set.
- 7 key-in parameters (00 to 23) for hours by using numeric keys and press the Input key.
- 8 key-in parameters (00 to 59) for minutes by using numeric keys and press the Input key.
- 9 Press the TIMER ON softkey to enable the timers. (To disable them, press the TIMER OFF softkey.)
- 10 After the entry of parameters has been completed, press the REG.DATA softkey to store them in the memory.
- 11 Return the EDIT PROTECT keyswitch to ON.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

“TOOLING DATA” parameters

Tooling parameters can be set for each turret station in the manner described below. The TOOL RANGE and TOOL TYPE parameters set sizes and types of the tools mounted on the turret; the PUNCH HEIGHT and TOP POS ADJ parameters simplify the punch height adjustment; the CUT LINE parameter controls the ram speed in a punching press mode operation; and the SHEAR parameter simplifies the programming procedure.

TOOL RANGE parameter

The TOOL RANGE parameter is used to identify the size of the mounted tool at each turret station:

A: 1/2", B: 1-1/4", C: 2", D: 3-1/2"

TOOL TYPE parameter

The TOOL TYPE parameter specifies the type of the mounted tool at each turret station. A PHNC alarm will be caused stopping the machine if the parameter is not collated with the programmed press-mode command (M12, M500 to M575, and M800 to M999). The message “Tools not in agreement” identifies the alarm on the screen.

Enter the number (0 to 2) assigned to the type of the mounted tool as follows:

- 0: Punching/nibbling tool (corresponds to M12, M500, M501) — alarm is not caused if “1” is entered instead of “0”.
- 1: Slitting tool (corresponds to M506 to M509) — slitting tools can be used only when the optional air blower is equipped.
- 2: Knockout, forming, or marking tool (corresponds to M502 to M505, M510 to M575, M800 to M999) — alarm is not caused if “0” or “1” is entered instead of “2”.

The default settings are “1” for all turret stations.

PUNCH HEIGHT parameter

It is necessary to set this parameter when the punch has been reground or a reground punch is to be used (see “TOP POS ADJ parameter” and note below). Otherwise the parameter is set at the following default values:

Turret station size:	1/2"	1-1/4"	2"	3-1/2"
Default setting:	207.5 mm (8.17")	207.5 mm (8.17")	208.0 mm (8.19")	209.0 mm (8.23")

CUT LINE parameter

The CUT LINE parameter is preset for the peripheral length of the punch to be used in the "punching" press mode. Never change the default settings.

Turret station size:	1/2"	1-1/4"	2"	3-1/2"
Default setting:	39.0 mm (1.57")	99.5 mm (3.92")	159.5 mm (6.28")	279.1 mm (10.99")

TOP POS ADJ parameter

It is necessary to set this parameter when the punch has been reground or a reground punch is to be used (see "PUNCH HEIGHT parameter" above and note below). The default setting is 0.0.

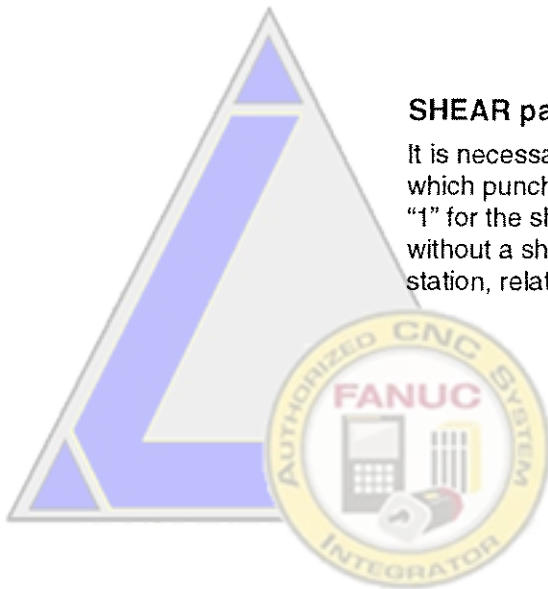
NOTE

● The PUNCH HEIGHT and TOP POS ADJ parameters should be set for a turret station in the following manner when the punch in the station has been reground or when using a reground punch:

- a) When the punch height has been adjusted by the head or shims:
 - Do not change the PUNCH HEIGHT parameter, keep it at the default setting.
 - For 1/2" and 1-1/4" turret stations, set the TOP POS ADJ parameter at a height lost by regrinding.
 - For 2" and 3-1/2" turret stations, do not change the TOP POS ADJ parameter, keep it at the default setting of 0.0.
- b) When the punch height is not adjusted (applicable to all turret stations):
 - Set the PUNCH HEIGHT parameter at an actually measured height.
 - Set the TOP POS ADJ parameter at a height lost by regrinding.
ex.) Set the TOP POS ADJ parameter at 0.5 mm when the punch has been reground by 0.5 mm.

SHEAR parameter

It is necessary to set this parameter for each of the turret stations in which punches other than those without a shear angle are used. Enter "1" for the shear-angled punch — the default setting is for the punch without a shear angle (0). When this parameter is set for every turret station, related M-code commands can be omitted in a program.



LAPIS SERVICES

Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Part VI

Alarms

CNC alarms.....	VI-2
PHNC alarms	VI-34
Machine alarms.....	VI-36
Self-diagnosis (CNC)	VI-41
Self-diagnosis (PHNC).....	VI-41



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

CNC ALARMS

A CNC alarm can be identified by a number and brief message shown on the screen. To proceed, remove the cause of the alarm, reset the CNC, correct the program if necessary, return all the axes to their origins, and then restart operation from the beginning.

NOTE

- Press any address or numeric key on the main control panel to reset the CNC alarm caused during the background editing. Pressing the Reset key will stop the on-going automatic operation.

If any alarm that is not described in this section has been caused, contact AMADA.



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

Program errors / Alarms on program and operation (P/S alarm)

Number	Message	Contents
000	PLEASE TURN OFF POWER	A parameter which requires the power off was input, turn off power.
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
003	TOO MANY DIGITS	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)
004	ADDRESS NOT FOUND	A numeral or the sign " - " was input without an address at the beginning of a block. Modify the program .
005	NO DATA AFTER ADDRESS	The address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program.
006	ILLEGAL USE OF NEGATIVE SIGN	Sign " - " input error (Sign " - " was input after an address with which it cannot be used. Or two or more " - " signs were input.) Modify the program.
007	ILLEGAL USE OF DECIMAL POINT	Decimal point " . " input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.) Modify the program.
009	ILLEGAL ADDRESS INPUT	Unusable character was input in significant area. Modify the program.
010	IMPROPER G-CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.
014	ILLEGAL LEAD COMMAND (T series)	In variable lead threading, the lead incremental and decremental outputted by address K exceed the maximum command value or a command such that the lead becomes a negative value is given. Modify the program.
	CAN NOT COMMAND G95 (M series)	A synchronous feed is specified without the option for threading / synchronous feed.
015	TOO MANY AXES COMMANDED	An attempt was made to move the machine along the axes, but the number of the axes exceeded the specified number of axes controlled simultaneously. Alternatively, in a block where the skip function activated by the torque-limit reached signal (G31 P99/P98) was specified, either moving the machine along an axis was not specified, or moving the machine along multiple axes was specified. Specify movement only along one axis.
	TOO MANY AXES COMMANDED (T series)	An attempt has been made to move the tool along more than the maximum number of simultaneously controlled axes. Alternatively, no axis movement command or an axis movement command for two or more axes has been specified in the block containing the command for skip using the torque limit signal (G31 P99/98). The command must be accompanied with an axis movement command for a single axis, in the same block.
020	OVER TOLERANCE OF RADIUS	In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 3410.

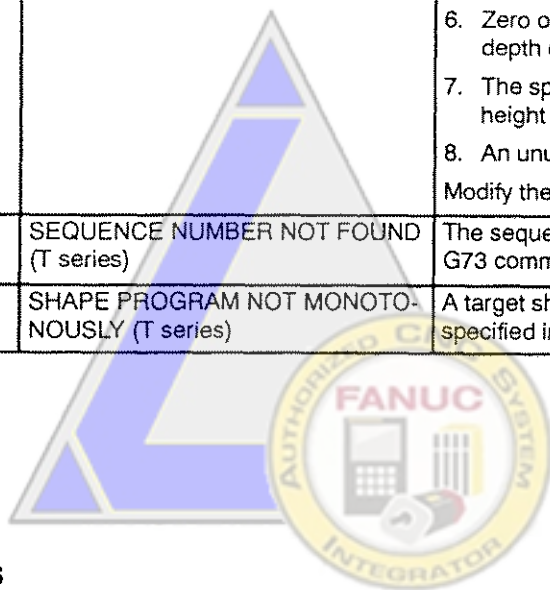
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

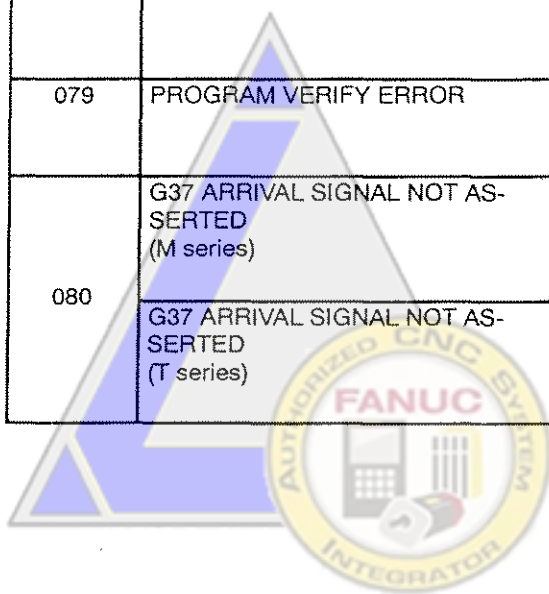
Number	Message	Contents
021	ILLEGAL PLANE AXIS COMMANDED	An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program.
022	NO CIRCLE RADIUS	The command for circular interpolation lacks arc radius R or coordinate I, J, or K of the distance between the start point to the center of the arc.
023	ILLEGAL RADIUS COMMAND (T series)	In circular interpolation by radius designation, negative value was commanded for address R. Modify the program.
025	CANNOT COMMAND F0 IN G02/G03 (M series)	F0 (fast feed) was instructed by F1 –digit column feed in circular interpolation. Modify the program.
027	NO AXES COMMANDED IN G43/G44 (M series)	No axis is specified in G43 and G44 blocks for the tool length offset type C. Offset is not canceled but another axis is offset for the tool length offset type C. Modify the program.
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.
029	ILLEGAL OFFSET VALUE (M series)	The offset values specified by H code is too large. Modify the program.
	ILLEGAL OFFSET VALUE (T series)	The offset values specified by T code is too large. Modify the program.
030	ILLEGAL OFFSET NUMBER (M series)	The offset number specified by D/H code for tool length offset or cutter compensation is too large. Modify the program.
	ILLEGAL OFFSET NUMBER (T series)	The offset number in T function specified for tool offset is too large. Modify the program.
031	ILLEGAL P COMMAND IN G10	In setting an offset amount by G10, the offset number following address P was excessive or it was not specified. Modify the program.
032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
033	NO SOLUTION AT CRC (M series)	A point of intersection cannot be determined for cutter compensation. Modify the program.
	NO SOLUTION AT CRC (T series)	A point of intersection cannot be determined for tool nose radius compensation. Modify the program.
034	NO CIRC ALLOWED IN ST-UP / EXT BLK (M series)	The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation C. Modify the program.
	NO CIRC ALLOWED IN ST-UP / EXT BLK (T series)	The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation. Modify the program.
035	CAN NOT COMMANDED G39 (M series)	G39 is commanded in cutter compensation B cancel mode or on the plane other than offset plane. Modify the program.
	CAN NOT COMMANDED G31 (T series)	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
036	CAN NOT COMMANDED G31 (M series)	Skip cutting (G31) was specified in cutter compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN CRC (M series)	G40 is commanded on the plane other than offset plane in cutter compensation B. The plane selected by using G17, G18 or G19 is changed in cutter compensation C mode. Modify the program.
	CAN NOT CHANGE PLANE IN NRC (T series)	The offset plane is switched in tool nose radius compensation. Modify the program.

Number	Message	Contents
038	INTERFERENCE IN CIRCULAR BLOCK (M series)	Overcutting will occur in cutter compensation C because the arc start point or end point coincides with the arc center. Modify the program.
	INTERFERENCE IN CIRCULAR BLOCK (T series)	Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.
039	CHF/CNR NOT ALLOWED IN NRC (T series)	Chamfering or corner R was specified with a start-up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R. Modify the program.
040	INTERFERENCE IN G90/G94 BLOCK (T series)	Overcutting will occur in tool nose radius compensation in canned cycle G90 or G94. Modify the program.
041	INTERFERENCE IN CRC (M series)	Overcutting will occur in cutter compensation C. Two or more blocks are consecutively specified in which functions such as the auxiliary function and dwell functions are performed without movement in the cutter compensation mode. Modify the program.
	INTERFERENCE IN NRC (T series)	Overcutting will occur in tool nose radius compensation. Modify the program.
042	G45/G48 NOT ALLOWED IN CRC (M series)	Tool offset (G45 to G48) is commanded in cutter compensation. Modify the program.
043	ILLEGAL T-CODE COMMAND (M series)	In a system using the DRILL-MATE with an ATC, a T code was not specified together with the M06 code in a block. Alternatively, the Tcode was out of range.
044	G27-G30 NOT ALLOWED IN FIXED CYC (M series)	One of G27 to G30 is commanded in canned cycle mode. Modify the program.
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command.
047	ILLEGAL AXIS SELECT (M series)	Two or more parallel axes (in parallel with a basic axis) have been specified upon start-up of three-dimensional tool compensation or three-dimensional coordinate conversion.
048	BASIC 3 AXIS NOT FOUND (M series)	Start-up of three-dimensional tool compensation or three-dimensional coordinate conversion has been attempted, but the three basic axes used when Xp, Yp, or Zp is omitted are not set in parameter No. 1022.
050	CHF/CNR NOT ALLOWED IN THRD BLK (M series)	Optional chamfering or corner R is commanded in the thread cutting block. Modify the program.
	CHF/CNR NOT ALLOWED IN THRD BLK (T series)	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
051	MISSING MOVE AFTER CHF/CNR (M series)	Improper movement or the move distance was specified in the block next to the optional chamfering or corner R block. Modify the program.
	MISSING MOVE AFTER CHF/CNR (T series)	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
052	CODE IS NOT G01 AFTER CHF/CNR (M series)	The block next to the chamfering or corner R block is not G01, G02 or G03. Modify the program.
	CODE IS NOT G01 AFTER CHF/CNR (T series)	The block next to the chamfering or corner R block is not G01. Modify the program.

Number	Message	Contents
053	TOO MANY ADDRESS COMMANDS (M series)	For systems without the arbitrary angle chamfering or corner R cutting, a comma was specified. For systems with this feature, a comma was followed by something other than R or C. Correct the program.
	TOO MANY ADDRESS COMMANDS (T series)	In the chamfering and corner R commands, two or more of I, K and R are specified. Otherwise, the character after a comma(",") is not C or R in direct drawing dimensions programming. Modify the program.
054	NO TAPER ALLOWED AFTER CHF/CNR	A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.
055	MISSING MOVE VALUE IN CHF/CNR (M series)	In the arbitrary angle chamfering or corner R block, the move distance is less than chamfer or corner R amount.
	MISSING MOVE VALUE IN CHF/CNR (T series)	In chamfering or corner R block, the move distance is less than chamfer or corner R amount.
056	NO END POINT & ANGLE IN CHF/CNR (T series)	Neither the end point nor angle is specified in the command for the block next to that for which only the angle is specified (A). In the chamfering command, I(K) is commanded for the X(Z) axis.
057	NO SOLUTION OF BLOCK END (T series)	Block end point is not calculated correctly in direct dimension drawing programming.
058	END POINT NOT FOUND (M series)	In an arbitrary angle chamfering or corner R cutting block, a specified axis is not in the selected plane. Correct the program.
	END POINT NOT FOUND (T series)	Block end point is not found in direct dimension drawing programming.
059	PROGRAM NUMBER NOT FOUND	In an external program number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background editing.
060	SEQUENCE NUMBER NOT FOUND	Commanded sequence number was not found in the sequence number search. Check the sequence number.
061	ADDRESS P/Q NOT FOUND IN G70-G73 (T series)	Address P or Q is not specified in G70, G71, G72, or G73 command. Modify the program.
062	ILLEGAL COMMAND IN G71-G76 (T series)	1. The depth of cut in G71 or G72 is zero or negative value. 2. The repetitive count in G73 is zero or negative value. 3. the negative value is specified to Δi or Δk is zero in G74 or G75. 4. A value other than zero is specified to address U or W though Δi or Δk is zero in G74 or G75. 5. A negative value is specified to Δd, though the relief direction in G74 or G75 is determined. 6. Zero or a negative value is specified to the height of thread or depth of cut of first time in G76. 7. The specified minimum depth of cut in G76 is greater than the height of thread. 8. An unusable angle of tool tip is specified in G76. Modify the program.
063	SEQUENCE NUMBER NOT FOUND (T series)	The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. Modify the program.
064	SHAPE PROGRAM NOT MONOTONOUSLY (T series)	A target shape which cannot be made by monotonic machining was specified in a repetitive canned cycle (G71 or G72).



Number	Message	Contents
065	ILLEGAL COMMAND IN G71–G73 (T series)	1. G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command. 2. Address Z(W) or X(U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively. Modify the program.
066	IMPROPER G-CODE IN G71–G73 (T series)	An unallowable G code was commanded between two blocks specified by address P in G71, G72, or G73. Modify the program.
067	CAN NOT ERROR IN MDI MODE (T series)	G70, G71, G72, or G73 command with address P and Q. Modify the program.
069	FORMAT ERROR IN G70–G73 (T series)	The final move command in the blocks specified by P and Q of G70, G71, G72, and G73 ended with chamfering or corner R. Modify the program.
070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
071	DATA NOT FOUND	The address to be searched was not found. Or the program with specified program number was not found in program number search. Check the data.
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125 (option), 200 (option), 400 (option) or 1000 (option). Delete unnecessary programs and execute program registration again.
073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registration again.
074	ILLEGAL PROGRAM NUMBER	The program number is other than 1 to 9999. Modify the program number.
075	PROTECT	An attempt was made to register a program whose number was protected.
076	ADDRESS P NOT DEFINED	Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.
077	SUB PROGRAM NESTING ERROR	The subprogram was called in five folds. Modify the program.
078	NUMBER NOT FOUND	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. Otherwise, a called program is being edited in background processing. Correct the program, or discontinue the background editing.
079	PROGRAM VERIFY ERROR	In memory or program collation, a program in memory does not agree with that read from an external I/O device. Check both the programs in memory and those from the external device.
080	G37 ARRIVAL SIGNAL NOT AS- SERTED (M series)	In the automatic tool length measurement function (G37), the measurement position reach signal (XAE, YAE, or ZAE) is not turned on within an area specified in parameter 6254 6255 (value ε). This is due to a setting or operator error.
	G37 ARRIVAL SIGNAL NOT AS- SERTED (T series)	In the automatic tool compensation function (G36, G37), the measurement position reach signal (XAE or ZAE) is not turned on within an area specified in parameter 6254 (value ε). This is due to a setting or operator error.



Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Number	Message	Contents
081	OFFSET NUMBER NOT FOUND IN G37 (M series)	Tool length automatic measurement (G37) was specified without a H code. (Automatic tool length measurement function) Modify the program.
	OFFSET NUMBER NOT FOUND IN G37 (T series)	Automatic tool compensation (G36, G37) was specified without a T code. (Automatic tool compensation function) Modify the program.
082	H-CODE NOT ALLOWED IN G37 (M series)	H code and automatic tool compensation (G37) were specified in the same block. (Automatic tool length measurement function) Modify the program.
	T-CODE NOT ALLOWED IN G37 (T series)	T code and automatic tool compensation (G36, G37) were specified in the same block. (Automatic tool compensation function) Modify the program.
083	ILLEGAL AXIS COMMAND IN G37 (M series)	In automatic tool length measurement, an invalid axis was specified or the command is incremental. Modify the program.
	ILLEGAL AXIS COMMAND IN G37 (T series)	In automatic tool compensation (G36, G37), an invalid axis was specified or the command is incremental. Modify the program.
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.
088	LAN FILE TRANS ERROR (CHANNEL-1)	File data transfer via OSI-ETHERNET has been stopped due to a transfer error.
089	LAN FILE TRANS ERROR (CHANNEL-2)	File data transfer via OSI-ETHERNET has been stopped due to a transfer error.
090	REFERENCE RETURN INCOMPLETE	The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return. Check the program contents.
091	REFERENCE RETURN INCOMPLETE	Manual reference position return cannot be performed when automatic operation is halted.
092	AXES NOT ON THE REFERENCE POINT	The commanded axis by G27 (Reference position return check) did not return to the reference position.
094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to the operator's manual.
095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.) Perform the correct operation according to the operator's manual.
096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.) Perform the correct operation according to the operator's manual.

Number	Message	Contents
097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or P / S 94 to 97 reset, no automatic operation is performed.) Perform automatic operation.
098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI. Move axis before a move command or don't interrupt MDI operation.
100	PARAMETER WRITE ENABLE	On the PARAMETER(SETTING) screen, PWE(parameter writing enabled) is set to 1. Set it to 0, then reset the system.
101	PLEASE CLEAR MEMORY	The power turned off while rewriting the memory by program edit operation. If this alarm has occurred, press <RESET> while pressing <PROG>, and only the program being edited will be deleted. Register the deleted program.
109	P/S ALARM	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
110	DATA OVERFLOW	The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.
111	CALCULATED DATA OVERFLOW	The result of calculation turns out to be invalid, an alarm No.111 is issued. -10 ⁴⁷ to -10 ⁻²⁹ , 0, 10 ⁻²⁹ to 10 ⁴⁷ Modify the program.
112	DIVIDED BY ZERO	Division by zero was specified. (including tan 90°) Modify the program.
113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
114	FORMAT ERROR IN MACRO	There is an error in other formats than <Formula>. Modify the program.
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro or in high-speed cycle machining. The header contents are improper. This alarm is given in the following cases: High speed cycle machining 1. The header corresponding to the specified machining cycle number called is not found. 2. The cycle connection data value is out of the allowable range (0 - 999). 3. The number of data in the header is out of the allowable range (0 - 32767). 4. The start data variable number of executable format data is out of the allowable range (#20000 - #85535). 5. The last storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format data is overlapped with the variable number used in the header. Modify the program.
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.
118	PARENTHESIS NESTING ERROR	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.

Number	Message	Contents
119	ILLEGAL ARGUMENT	The SQRT argument is negative. Or BCD argument is negative, and other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	FOUR FOLD MACRO MODAL-CALL	The macro modal call is specified four fold. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
124	MISSING END STATEMENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	<Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DOn, $1 \leq n \leq 3$ is not established. Modify the program.
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coexist. Modify the program.
128	ILLEGAL MACRO SEQUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argument Designation> is used. Modify the program.
130	ILLEGAL AXIS OPERATION	An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program.
131	TOO MANY EXTERNAL ALARM MESSAGES	Five or more alarms have generated in external alarm message. Consult the PMC ladder diagram to find the cause.
132	ALARM NUMBER NOT FOUND	No alarm No. concerned exists in external alarm message clear. Check the PMC ladder diagram.
133	ILLEGAL DATA IN EXT. ALARM MSG	Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.
135	ILLEGAL ANGLE COMMAND (M series)	The index table indexing positioning angle was instructed in other than an integral multiple of the value of the minimum angle. Modify the program.
	SPINDLE ORIENTATION PLEASE (T series)	Without any spindle orientation, an attempt was made for spindle indexing. Perform spindle orientation.
136	ILLEGAL AXIS COMMAND (M series)	In index table indexing. Another control axis was instructed together with the B axis. Modify the program.
	C/H-CODE & MOVE CMD IN SAME BLK. (T series)	A move command of other axes was specified to the same block as spindle indexing addresses C, H. Modify the program.
137	M-CODE & MOVE CMD IN SAME BLK.	A move command of other axes was specified to the same block as M-code related to spindle indexing. Modify the program.
138	SUPERIMPOSED DATA OVERFLOW	The total distribution amount of the CNC and PMC is too large during superimposed control of the extended functions for PMC axis control.
139	CAN NOT CHANGE PMC CONTROL AXIS	An axis is selected in commanding by PMC axis control. Modify the program.
141	CAN NOT COMMAND G51 IN CRC (M series)	G51 (Scaling ON) is commanded in the tool offset mode. Modify the program.
142	ILLEGAL SCALE RATE (M series)	Scaling magnification is commanded in other than 1 – 999999. Correct the scaling magnification setting (G51 P _p or parameter 5411 or 5421).
143	SCALED MOTION DATA OVERFLOW (M series)	The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. Correct the program or scaling magnification.
144	ILLEGAL PLANE SELECTED (M series)	The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program.

Number	Message	Contents
145	ILLEGAL CONDITIONS IN POLAR COORDINATE INTERPOLATION	The conditions are incorrect when the polar coordinate interpolation starts or it is canceled. 1) In modes other than G40, G12.1/G13.1 was specified. 2) An error is found in the plane selection. Parameters No. 5460 and No. 5461 are incorrectly specified. Modify the value of program or parameter.
146	IMPROPER G CODE	G codes which cannot be specified in the polar coordinate interpolation mode was specified. Modify the program.
148	ILLEGAL SETTING DATA (M series)	Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters (No.1710 to No.1714)
149	FORMAT ERROR IN G10L3 (M series)	A code other than Q1,Q2,P1 or P2 was specified as the life count type in the extended tool life management.
150	ILLEGAL TOOL GROUP NUMBER	Tool Group No. exceeds the maximum allowable value. Modify the program.
151	TOOL GROUP NUMBER NOT FOUND	The tool group commanded in the machining program is not set. Modify the value of program or parameter.
152	NO SPACE FOR TOOL ENTRY	The number of tools within one group exceeds the maximum value registerable. Modify the number of tools.
153	T-CODE NOT FOUND	In tool life data registration, a T code was not specified where one should be. Correct the program.
154	NOT USING TOOL IN LIFE GROUP (M series)	When the group is not commanded, H99 or D99 was commanded. Correct the program.
155	ILLEGAL T-CODE IN M06 (M series)	In the machining program, M06 and T code in the same block do not correspond to the group in use. Correct the program.
	ILLEGAL T-CODE IN M06 (T series)	Group No.ΔΔ which is specified with TΔΔ 88 of the machining program do not included in the tool group in use. Correct the program.
156	P/L COMMAND NOT FOUND	P and L commands are missing at the head of program in which the tool group is set. Correct the program.
157	TOO MANY TOOL GROUPS	The number of tool groups to be set exceeds the maximum allowable value. (See parameter No. 6800 bit 0 and 1) Modify the program.
158	ILLEGAL TOOL LIFE DATA	The tool life to be set is too excessive. Modify the setting value.
159	TOOL DATA SETTING INCOMPLETE	During executing a life data setting program, power was turned off. Set again.
160	MISMATCH WAITING M-CODE T series (At two-path)	Different M code is commanded in heads 1 and 2 as waiting M code. Modify the program.
	G72.1 NESTING ERROR (M series)	A subprogram which performs rotational copy with G72.1 contains another G72.1 command.
161	G72.1 NESTING ERROR (M series)	A subprogram which performs parallel copy with G72.2 contains another G72.2 command.
163	COMMAND G68/G69 INDEPENDENTLY T series (At two-path)	G68 and G69 are not independently commanded in balance cut. Modify the program.
169	ILLEGAL TOOL GEOMETRY DATA T series (At two-path)	Incorrect tool figure data in interference check. Set correct data, or select correct tool figure data.
175	ILLEGAL G107 COMMAND	Conditions when performing circular interpolation start or cancel not correct. To change the mode to the cylindrical interpolation mode, specify the command in a format of "G07.1 rotation-axis name radius of cylinder."

Fanuc Control Upgrades.

LAPISINC.COM

Number	Message	Contents
176	IMPROPER G-CODE IN G107 (M series)	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning: G28,, G73, G74, G76, G81 – G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G52, G92, 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.
	IMPROPER G-CODE IN G107 (T series)	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning: G28, G76, G81 – G89, including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G50, G52 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.
177	CHECK SUM ERROR (G05 MODE)	Check sum error Modify the program.
178	G05 COMMANDED IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAM. (NO. 7510) SETTING ERROR	The number of controlled axes set by the parameter 7510 exceeds the maximum number. Modify the parameter setting value.
180	COMMUNICATION ERROR (REMOTE BUF)	Remote buffer connection alarm has generated. Confirm the number of cables, parameters and I/O device.
181	FORMAT ERROR IN G81 BLOCK (M series)	G81 block format error (hobbing machine) 1) T (number of teeth) has not been instructed. 2) Data outside the command range was instructed by either T, L, Q or P. Modify the program.
182	G81 NOT COMMANDED (M series)	G83 (C axis servo lag quantity offset) was instructed though synchronization by G81 has not been instructed. Correct the program. (hobbing machine)
183	DUPLICATE G83 (COMMANDS) (M series)	G83 was instructed before canceled by G82 after compensating for the C axis servo lag quantity by G83. (hobbing machine)
184	ILLEGAL COMMAND IN G81 (M series)	A command not to be instructed during synchronization by G81 was instructed. (hobbing machine) 1) A C axis command by G00, G27, G28, G29, G30, etc. was instructed. 2) Inch/Metric switching by G20, G21 was instructed.
185	RETURN TO REFERENCE POINT (M series)	G81 was instructed without performing reference position return after power on or emergency stop. (hobbing machine) Perform reference position return.
186	PARAMETER SETTING ERROR (M series)	Parameter error regarding G81 (hobbing machine) 1) The C axis has not been set to be a rotary axis. 2) A hob axis and position coder gear ratio setting error Modify the parameter.
190	ILLEGAL AXIS SELECT (M series)	In the constant surface speed control, the axis specification is wrong. (See parameter No. 3770.) The specified axis command (P) contains an illegal value. Correct the program.
194	SPINDLE COMMAND IN SYNCHRO-MODE	A contour control mode, spindle positioning (Cs-axis control) mode, or rigid tapping mode was specified during the serial spindle synchronous control mode. Correct the program so that the serial spindle synchronous control mode is released in advance.

Number	Message	Contents
195	MODE CHANGE ERROR	Switching command to contouring mode, Cs axis control or rigid tap mode or switching to spindle command mode is not correctly completed. (This occurs when the response to switch to the spindle control unit side with regard to the switching command from the NC is incorrect. This alarm is not for the purposes of warning against mistakes in operation, but because continuing operation in this condition can be dangerous it is a P/S alarm.)
197	C-AXIS COMMANDED IN SPINDLE MODE	The program specified a movement along the Cs-axis when the signal CON(DGN=G027#7) was off. Correct the program, or consult the PMC ladder diagram to find the reason the signal is not turned on.
199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value is out of the range or is not specified. Modify the program.
201	FEEDRATE NOT FOUND IN RIGID TAP	In the rigid tap, no F value is specified. Correct the program.
202	POSITION LSI OVERFLOW	In the rigid tap, spindle distribution value is too large. (System error)
203	PROGRAM MISS AT RIGID TAPPING	In the rigid tap, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
204	ILLEGAL AXIS OPERATION	In the rigid tap, an axis movement is specified between the rigid M code (M29) block and G84 or G74 for M series (G84 or G88 for T series) block. Modify the program.
205	RIGID MODE DI SIGNAL OFF	Rigid mode DI signal is not ON when G84 or G74 for M series (G84 or G88 for T series) is executed though the rigid M code (M29) is specified. Consult the PMC ladder diagram to find the reason the DI signal (DGNG061.1) is not turned on.
206	CAN NOT CHANGE PLANE (RIGID TAP) (M series)	Plane changeover was instructed in the rigid mode. Correct the program.
210	CAN NOT COMAND M198/M199	M198 and M199 are executed in the schedule operation. M198 is executed in the DNC operation. Modify the program. 1) The execution of an M198 or M99 command was attempted during scheduled operation. Alternatively, the execution of an M198 command was attempted during DNC operation. Correct the program. 2) The execution of an M99 command was attempted by an interrupt macro during pocket machining in a multiple repetitive canned cycle.
211	G31 (HIGH) NOT ALLOWED IN G99	G31 is commanded in the per revolution command when the high-speed skip option is provided. Modify the program.
212	ILLEGAL PLANE SELECT (M series)	The arbitrary angle chamfering or a corner R is commanded or the plane including an additional axis. Correct the program.
	ILLEGAL PLANE SELECT (T series)	The direct drawing dimensions programming is commanded for the plane other than the Z-X plane. Correct the program.



Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Number	Message	Contents
213	ILLEGAL COMMAND IN SYNCHRO-MODE	Movement is commanded for the axis to be synchronously controlled. Any of the following alarms occurred in the operation with the simple synchronization control. 1) The program issued the move command to the slave axis. 2) The program issued the manual continuous feed/manual handle feed/incremental feed command to the slave axis. 3) The program issued the automatic reference position return command without specifying the manual reference position return after the power was turned on. 4) The difference between the position error amount of the master and slave axes exceeded the value specified in parameter NO.8313.
	ILLEGAL COMMAND IN SYNCHRO-MODE (T series)	A move command has been specified for an axis subject to synchronous control.
214	ILLEGAL COMMAND IN SYNCHRO-MODE	Coordinate system is set or tool compensation of the shift type is executed in the synchronous control. Correct the program.
217	DUPLICATE G51.2 (COMMANDS) (T series)	G51.2/G251 is further commanded in the G51.2/G251 mode. Modify the program.
218	NOT FOUND P/Q COMMAND IN G251 (T series)	P or Q is not commanded in the G251 block, or the command value is out of the range. Modify the program.
219	COMMAND G250/G251 INDEPENDENTLY (T series)	G251 and G250 are not independent blocks.
220	ILLEGAL COMMAND IN SYNCHRO-MODE (T series)	In the synchronous operation, movement is commanded by the NC program or PMC axis control interface for the synchronous axis.
221	ILLEGAL COMMAND IN SYNCHRO-MODE (T series)	Polygon machining synchronous operation and axis control or balance cutting are executed at a time. Modify the program.
222	DNC OP. NOT ALLOWED IN BG.-EDIT (M series)	Input and output are executed at a time in the background edition. Execute a correct operation.
224	RETURN TO REFERENCE POINT (M series)	Reference position return has not been performed before the automatic operation starts. Perform reference position return only when bit 0 of parameter 1005 is 0.
	TURN TO REFERENCE POINT (T series)	Reference position return is necessary before cycle start.
225	SYNCHRONOUS/MIXED CONTROL ERROR (T series (At two-path))	This alarm is generated in the following circumstances. (Searched for during synchronous and mixed control command. 1 When there is a mistake in axis number parameter setting. 2 When there is a mistake in control commanded. Modify the program or the parameter.
226	ILLEGAL COMMAND IN SYNCHRO-MODE T series (At two-path)	A travel command has been sent to the axis being synchronized in synchronous mode. Modify the program or the parameter.
229	CAN NOT KEEP SYNCHRO-STATE (T series)	This alarm is generated in the following circumstances. 1 When the synchro/mixed state could not be kept due to system overload. 2 The above condition occurred in CMC devices (hardware) and synchro-state could not be kept. (This alarm is not generated in normal use conditions.)
230	R CODE NOT FOUND (GS series)	The infeed quantity R has not been instructed for the G161 block. Or the R command value is negative. Correct the program.

Number	Message	Contents
231	ILLEGAL FORMAT IN G10 OR L50	Any of the following errors occurred in the specified format at the programmable-parameter input. 1 Address N or R was not entered. 2 A number not specified for a parameter was entered. 3 The axis number was too large. 4 An axis number was not specified in the axis-type parameter. 5 An axis number was specified in the parameter which is not an axis type. Correct the program. 6 An attempt was made to reset bit 4 of parameter 3202 (NE9) or change parameter 3210 (PSSWD) when they are protected by a password. Correct the program.
232	TOO MANY HELICAL AXIS COMMANDS (M series)	Three or more axes (in the normal direction control mode two or more axes) were specified as helical axes in the helical interpolation mode.
233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS-232-C interface, other users were using it.
239	BP/S ALARM	While punching was being performed with the function for controlling external I/O units, background editing was performed.
240	BP/S ALARM	Background editing was performed during MDI operation.
241	ILLEGAL FORMAT IN G02.2/G03.2 (M series)	The end point, I, J, K, or R is missing from a command for involute interpolation.
242	ILLEGAL COMMAND IN G02.2/G03.2 (M series)	An invalid value has been specified for involute interpolation. • The start or end point is within the basic circle. • I, J, K, or R is set to 0. • The number of rotations between the start of the involute curve and the start or end point exceeds 100.
243	OVER TOLERANCE OF END POINT (M series)	The end point is not on the involute curve which includes the start point and thus falls outside the range specified with parameter No. 5610.
244	P/S ALARM (T series)	In the skip function activated by the torque limit signal, the number of accumulated erroneous pulses exceed 32767 before the signal was input. Therefore, the pulses cannot be corrected with one distribution. Change the conditions, such as feed rates along axes and torque limit, and try again.
245	T-CODE NOT ALLOWED IN THIS BLOCK (T series)	One of the G codes, G50, G10, and G04, which cannot be specified in the same block as a T code, was specified with a T code.
250	Z AXIS WRONG COMMAND (ATC) (M series)	A value for the Z-axis has been specified in a block for the tool exchange command (M06T_) on a system with DRILL-MATE ARC installed.
251	ATC ERROR (M series)	This alarm is issued in the following cases (DRILL-MATE): • An M06T_ command contains an unusable T code. • An M06 command has been specified when the Z machine coordinate is positive. • The parameter for the current tool number (No. 7810) is set to 0. • An M06 command has been specified in canned cycle mode. • A reference position return command (G27 to G44) and M06 command have been specified in the same block. • An M06 command has been specified in tool compensation mode (G41 to G44). • An M06 command has been specified without performing reference position return after power-on or the release of emergency stop. • The machine lock signal or Z-axis ignore signal has been turned on during tool exchange. • A pry alarm has been detected during tool exchange. Refer to diagnosis No. 530 to determine the cause.

Number	Message	Contents
252	ATC SPINDLE ALARM (M series)	An excessive error arose during spindle positioning for ATC. For details, refer to diagnosis No. 531. (Only for DRILL-MATE)
253	G05 IS NOT AVAILABLE (M series)	Alarm details Binary input operation using high-speed remote buffer (G05) or high-speed cycle machining (G05) has been specified in advance control mode (G08P1). Execute G08P0; to cancel advance control mode, before executing these G05 commands.
5000	ILLEGAL COMMAND CODE (M series)	The specified code was incorrect in the high-precision contour control (HPCC) mode.
5003	ILLEGAL PARAMETER (HPCC) (M series)	There is an invalid parameter.
5004	HPCC NOT READY (M series)	High-precision contour control is not ready.
5006	TOO MANY WORD IN ONE BLOCK (M series)	The number of words specified in a block exceeded 26 in the HPCC mode.
5007	TOO LARGE DISTANCE (M series)	In the HPCC mode, the machine moved beyond the limit.
5009	PARAMETER ZERO (DRY RUN) (M series)	The maximum feedrate (parameter No. 1422) or the feedrate in dry run (parameter No. 1410) is 0 in the HPCC model.
5010	END OF RECORD	The end of record (%) was specified. I/O is incorrect. modify the program.
5011	PARAMETER ZERO(CUT MAX) (M series)	The maximum cutting feedrate (parameter No. 1422) is 0 in the HPCC mode.
5012	G05 P10000 ILLEGAL START UP (HPCC) (M series)	Function category: High-precision contour control Alarm details: G05 P10000 has been specified in a mode from which the system cannot enter HPCC mode.
5013	HPCC: CRC OFS REMAIN AT CANCEL (M series)	G05P0 has been specified in G41/G42 mode or with offset remaining.
5014	TRACE DATA NOT FOUND (M series)	Transfer cannot be performed because no trace data exists.
5015	(M series)	The specified rotation axis does not exist for tool axis direction handle feed.
5016	ILLEGAL COMBINATION OF M CODE	M codes which belonged to the same group were specified in a block. Alternatively, an M code which must be specified without other M codes in the block was specified in a block with other M codes.
5018	POLYGON SPINDLE SPEED ERROR (T series)	Function category: Polygon turning Alarm details: In G51.2 mode, the speed of the spindle or polygon synchronous axis either exceeds the clamp value or is too small. The specified rotation speed ratio thus cannot be maintained.
5020	PARAMETER OF RESTART ERROR	An erroneous parameter was specified for restarting a program. A parameter for program restart is invalid.
5030	ILLEGAL COMMAND (G100) (T series)	The end command (G110) was specified before the registration start command (G101, G102, or G103) was specified for the B-axis.
5031	ILLEGAL COMMAND (G100, G102, G103) (T series)	While a registration start command (G101, G102, or G103) was being executed, another registration start command was specified for the B-axis.
5032	NEW PRG REGISTERED IN B-AXIS MOVE (T series)	While the machine was moving about the B-axis, an attempt was made to register another move command.
5033	NO PROG SPACE IN MEMORY B-AXIS (T series)	Commands for movement about the B-axis were not registered because of insufficient program memory.

Number	Message	Contents
5034	PLURAL COMMAND IN G110 (T series)	Multiple movements were specified with the G110 code for the B-axis.
5035	NO FEEDRATE COMMANDED B-AXIS (T series)	A feedrate was not specified for cutting feed about the B-axis.
5036	ADDRESS R NOT DEFINED IN G81-G86 (T series)	Point R was not specified for the canned cycle for the B-axis.
5037	ADDRESS Q NOT DEFINED IN G83 (T series)	Depth of cut Q was not specified for the G83 code (peck drilling cycle). Alternatively, 0 was specified in Q for the B-axis.
5038	TOO MANY START M-CODE COMMAND (T series)	More than six M codes for starting movement about the B-axis were specified.
5039	START UNREGISTERED B-AXIS PROG (T series)	An attempt was made to execute a program for the B-axis which had not been registered.
5040	CAN NOT COMMANDED B-AXIS MOVE (T series)	The machine could not move about the B-axis because parameter No.8250 was incorrectly specified, or because the PMC axis system could not be used.
5041	CAN NOT COMMANDED G110 BLOCK (T series)	Blocks containing the G110 codes were successively specified in tool-tip radius compensation for the B-axis.
5043	TOO MANY G68 NESTING (M series)	Three-dimensional coordinate conversion G68 has been specified three or more times.
5044	G68 FORMAT ERROR (M series)	A G68 command block contains a format error. This alarm is issued in the following cases: 1. I, J, or K is missing from a G68 command block (missing coordinate rotation option). 2. I, J, and K are 0 in a G68 command block. 3. R is missing from a G68 command block.
5046	ILLEGAL PARAMETER (ST.COMP)	The parameter settings for straightness compensation contain an error. Possible causes are as follows: 1. A parameter for a movement axis or compensation axis contains an axis number which is not used. 2. More than 128 pitch error compensation points exist between the negative and positive end points. 3. Compensation point numbers for straightness compensation are not assigned in the correct order. 4. No straightness compensation point exists between the pitch error compensation points at the negative and positive ends. 5. The compensation value for each compensation point is too large or too small.
5050	ILL-COMMAND IN CHOPPING MODE (M series)	A command for switching the major axis has been specified for circular threading. Alternatively, a command for setting the length of the major axis to 0 has been specified for circular threading.
5051	M-NET CODE ERROR	Abnormal character received (other than code used for transmission)
5052	M-NET ETX ERROR	Abnormal ETX code
5053	M-NET CONNECT ERROR	Connection time monitoring error (parameter No. 175)
5054	M-NET RECEIVE ERROR	Polling time monitoring error (parameter No. 176)
5055	M-NET PRT/FRT ERROR	Vertical parity or framing error
5057	M-NET BOARD SYSTEM DOWN	Transmission timeout error (parameter No. 177) ROM parity error CPU interrupt other than the above
5058	G35/G36 FORMAT ERROR (T series)	A command for switching the major axis has been specified for circular threading. Alternatively, a command for setting the length of the major axis to 0 has been specified for circular threading.
5059	RADIUS IS OUT OF RANGE (T series)	A radius exceeding nine digits has been specified for circular interpolation with the center of the arc specified with I, J, and K.

Number	Message	Contents
5063	IS NOT PRESET AFTER REF. (M series)	Function category: Workpiece thickness measurement Alarm details The position counter was not preset before the start of workpiece thickness measurement. This alarm is issued in the following cases: (1) An attempt has been made to start measurement without first establishing the origin. (2) An attempt has been made to start measurement without first presetting the position counter after manual return to the origin.
5064	DIFFERENT AXIS UNIT (IS-B, IS-C) (M series)	Circular interpolation has been specified on a plane consisting of axes having different increment systems.
5065	DIFFERENT AXIS UNIT (PMC AXIS) (M series)	Axes having different increment systems have been specified in the same DI/DO group for PMC axis control. Modify the setting of parameter No. 8010.
5066	RESTART ILLEGAL SEQUENCE NUMBER (M series)	Sequence number 7xxx has been read during search for the next sequence number at program restart for the return/restart function.
5068	G31 P90 FORMAT ERROR (M series)	No movement axis or more than one movement axis has been specified.
5073	NO DECIMAL POINT	No decimal point has been specified for an address requiring a decimal point.
5074	ADDRESS DUPLICATION ERROR	The same address has been specified two or more times in a single block. Alternatively, two or more G codes in the same group have been specified in a single block.
5082	DATA SERVER ERROR	This alarm is detailed on the data server message screen.

Note

HPCC : High precision contour control



LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

Background edit alarm

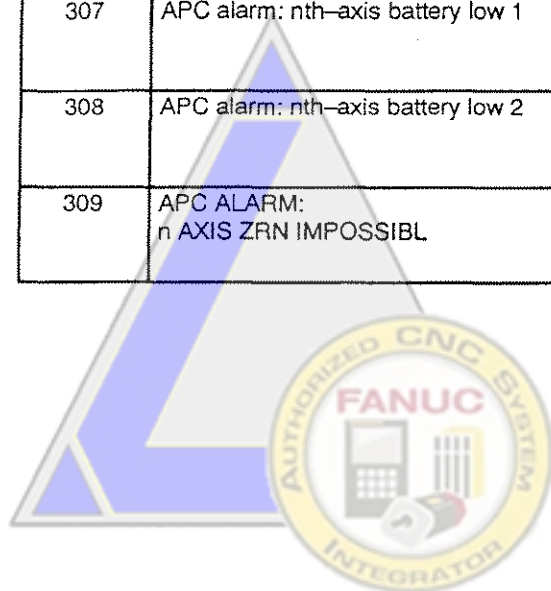
Number	Message	Contents
???	BP/S alarm	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit. (070, 071, 072, 073, 074 085,086,087 etc.)
140	BP/S alarm	It was attempted to select or delete in the background a program being selected in the foreground. (Note) Use background editing correctly.

Note

Alarm in background edit is displayed in the key input line of the background edit screen instead of the ordinary alarm screen and is resettable by any of the MDI key operation.

Absolute pulse coder (APC) alarm

Number	Message	Contents
300	nth-axis origin return	Manual reference position return is required for the nth-axis (n=1 – 8).
301	APC alarm: nth-axis communication	nth-axis (n=1 – 8) APC communication error. Failure in data transmission Possible causes include a faulty APC, cable, or servo interface module.
302	APC alarm: nth-axis over time	nth-axis (n=1 – 8) APC overtime error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
303	APC alarm: nth-axis framing	nth-axis (n=1 – 8) APC framing error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
304	APC alarm: nth-axis parity	nth-axis (n=1 – 8) APC parity error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
305	APC alarm: nth-axis pulse error	nth-axis (n=1 – 8) APC pulse error alarm. APC alarm. APC or cable may be faulty.
306	APC alarm: nth-axis battery voltage 0	nth-axis (n=1 – 8) APC battery voltage has decreased to a low level so that the data cannot be held. APC alarm. Battery or cable may be faulty.
307	APC alarm: nth-axis battery low 1	nth-axis (n=1 – 8) axis APC battery voltage reaches a level where the battery must be renewed. APC alarm. Replace the battery.
308	APC alarm: nth-axis battery low 2	nth-axis (n=1 – 8) APC battery voltage has reached a level where the battery must be renewed (including when power is OFF). APC alarm. Replace battery.
309	APC ALARM: n AXIS ZRN IMPOSSIBL	Return to the origin has been attempted without first rotating the motor one or more times. Before returning to the origin, rotate the motor one or more times then turn off the power.



Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Serial pulse coder (SPC) alarms

When either of the following alarms is issued, a possible cause is a faulty serial pulse coder or cable.

Number	Message	Contents
350	SPC ALARM: n AXIS PULSE CODER	The n axis (axis 1–8) pulse coder has a fault. Refer to diagnosis display No. 202 and No. 204 for details.
351	SPC ALARM: n AXIS COMMUNICATION	n axis (axis 1–8) serial pulse coder communication error (data transmission fault) Refer to diagnosis display No. 203 for details.

- The details of serial pulse coder alarm No.350

	#7	#6	#5	#4	#3	#2	#1	#0
202		CSA	BLA	PHA	PCA	BZA	CKA	SPH

- #6 (CSA) : Check sum alarm has occurred.
- #5 (BLA) : Battery low alarm has occurred.
- #4 (PHA) : Phase data trouble alarm has occurred.
- #3 (PCA) : Speed count trouble alarm has occurred.
- #2 (BZA) : Battery zero alarm has occurred.
- #1 (CKA) : Clock alarm has occurred.
- #0 (SPH) : Soft phase data trouble alarm has occurred.

- The details of serial pulse coder alarm No.351

	#7	#6	#5	#4	#3	#2	#1	#0
203	DTE	CRC	STB					

- #7 (DTE) : Data error has occurred.
- #6 (CRC) : CRC error has occurred.
- #5 (STB) : Stop bit error has occurred.



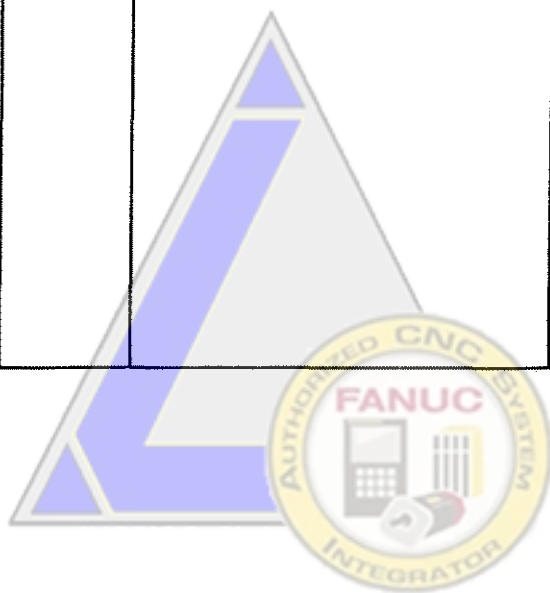
LAPIS SERVICES
(219) 464-9131

**Amada Support,
Rebuilds, Sales And
Fanuc Control Upgrades.**

LAPISINC.COM

Servo alarms

Number	Message	Contents
400	SERVO ALARM: n-TH AXIS OVER-LOAD	The n-th axis (axis 1-8) overload signal is on. Refer to diagnosis display No. 201 for details.
401	SERVO ALARM: n-TH AXIS VRDY OFF	The n-th axis (axis 1-8) servo amplifier READY signal (DRDY) went off.
404	SERVO ALARM: n-TH AXIS VRDY ON	Even though the n-th axis (axis 1-8) READY signal (MCON) went off, the servo amplifier READY signal (DRDY) is still on. Or, when the power was turned on, DRDY went on even though MCON was off. Check that the servo interface module and servo amp are connected.
405	SERVO ALARM: (ZERO POINT RETURN FAULT)	Position control system fault. Due to an NC or servo system fault in the reference position return, there is the possibility that reference position return could not be executed correctly. Try again from the manual reference position return.
407	SERVO ALARM: EXCESS ERROR	The difference in synchronous axis position deviation exceeded the set value.
409	SERVO ALARM: n AXIS TORQUE ALM	Abnormal servo motor load has been detected. Alternatively, abnormal spindle motor load has been detected in Cs mode.
410	SERVO ALARM: n-TH AXIS - EXCESS ERROR	The position deviation value when the n-th axis (axis 1-8) stops is larger than the set value.
411	SERVO ALARM: n-TH AXIS - EXCESS ERROR	The position deviation value when the n-th axis (axis 1-8) moves is larger than the set value.
413	SERVO ALARM: n-th AXIS - LSI OVERFLOW	The contents of the error register for the n-th axis (axis 1-8) exceeded $\pm 2^{31}$ power. This error usually occurs as the result of an improperly set parameters.
414	SERVO ALARM: n-TH AXIS - DETECTION RELATED ERROR	N-th axis (axis 1-8) digital servo system fault. Refer to diagnosis display No. 200 and No.204 for details.
415	SERVO ALARM: n-TH AXIS - EXCESS SHIFT	A speed higher than 511875 units/s was attempted to be set in the n-th axis (axis 1-8). This error occurs as the result of improperly set CMR.
416	SERVO ALARM: n-TH AXIS - DIS-CONNECTION	Position detection system fault in the n-th axis (axis 1-8) pulse coder (disconnection alarm). Refer to diagnosis display No. 201 for details.
417	SERVO ALARM: n-TH AXIS - PARAMETER INCORRECT	This alarm occurs when the n-th axis (axis 1-8) is in one of the conditions listed below. (Digital servo system alarm) 1) The value set in Parameter No. 2020 (motor form) is out of the specified limit. 2) A proper value (111 or -111) is not set in parameter No.2022 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 2023 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 2024 (number of position feedback pulses per motor revolution). 5) Parameters No. 2084 and No. 2085 (flexible field gear rate) have not been set. 6) A value outside the limit of {1 to the number of control axes} or a non-continuous value. Parameter 1023 (servo axis number) contains a value out of the range from 1 to the number of axes, or an isolated value (for example, 4 not preceded by 3) was set in parameter No. 1023 (servo axis number).



Rebuilds, Sales And
Fanuc Control Upgrades.

LAPISINC.COM

Number	Message	Contents
420	SERVO ALARM: n AXIS SYNC TORQUE (M series)	During simple synchronous control, the difference between the torque commands for the master and slave axes exceeded the value set in parameter No. 2031.
421	SERVO ALARM: n AXIS EXCESS ER (D)	The difference between the errors in the semi-closed loop and closed loop has become excessive during dual position feedback. Check the values of the dual position conversion coefficients in parameters No. 2078 and 2079.

● **Details of servo alarm No.414**

The details of servo alarm No. 414 are displayed in the diagnosis display (No. 200 and No.204) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
200	OVL	LV	OVC	HCA	HVA	DCA	FBA	OFA

#7 (OVL) : An overload alarm is being generated.

#6 (LV) : A low voltage alarm is being generated in servo amp.

#5 (OVC) : A overcurrent alarm is being generated inside of digital servo.

#4 (HCA) : An abnormal current alarm is being generated in servo amp.

#3 (HVA) : An overvoltage alarm is being generated in servo amp.

#2 (DCA) : A regenerative discharge circuit alarm is being generated in servo amp.

#1 (FBA) : A disconnection alarm is being generated.

#0 (OFA) : An overflow alarm is being generated inside of digital servo.

	#7	#6	#5	#4	#3	#2	#1	#0
204		OFS	MCC	LDA	PMS			

#6 (OFS) : A current conversion error has occurred in the digital servo.

#5 (MCC) : A magnetic contactor contact in the servo amplifier has welded.

#4 (LDA) : The LED indicates that serial pulse coder C is defective

#3 (PMS) : A feedback pulse error has occurred because the feedback cable is defective.

● **Details of servo alarms No. 400 and No.416**

The details of servo alarms No. 400 and No. 416 are displayed in the diagnosis display (No. 201) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
201	ALD			EXP				

When OVL equal 1 in diagnostic data No.200 (servo alarm No. 400 is being generated):

#7 (ALD) 0 : Motor overheating
1 : Amplifier overheating

When FBAL equal 1 in diagnostic data No.200 (servo alarm No. 416 is being generated):

ALD	EXP	Alarm details
1	0	Built-in pulse coder disconnection (hardware)
1	1	Separately installed pulse coder disconnection (hardware)
0	0	Pulse coder is not connected due to software.