



**DMNC/DMTRT SERIES**  
**OPERATIONS MANUAL**  
**VERSION 1.00**



**YUASA INTERNATIONAL**  
**1320 LANDMIERE ROAD**  
**ELK GROVE VILLAGE, ILLINOIS**  
**60188**  
**1-800-323-7427 OFFICE**  
**1-847-981-0737 OUTSIDE U.S.**  
**1-847-981-0757 FAX**

# **!!MANDATORY READ THIS!!**

## **INTRODUCTION:**


Thank you for choosing the DMNC or DMTRT Programmable Rotary Table. These precision tables have been designed and manufactured for the highest possible durability, accuracy, and flexibility that is required for today's diverse machining needs.

To insure that you get the most out of your new unit, please **COMPLETELY** read all the aspects of programming and operating these units before actual setup and operation.

## **SPECIAL CAUTIONS:**

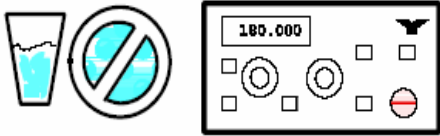
1. Power source should be 110-220 VAC single phase
2. Never apply voltage to lines 2 and 3(Step Start) or 8 and 13(E-STOP) on our remote cable. ( We require an isolated relay to provide a contact closer only)
3. Position the controller in a place free from oils, coolants, chips, and excessive heat. Do not allow filters to become clogged or blocked. Air must be able to circulate freely to insure no overheating of the internal circuitry.
4. Always use a Filtered/Regulated/Lubricated/ air supply for these units. (90-120 P.S.I.) Failure to do so will cause moisture to accumulate in the braking mechanisms. This can cause severe damage to the servos and mechanics of the rotary table.
5. Periodically check motor cabling and conduit for splits, cracks, and puncture that can allow coolant and other contaminates into the motor housing. If damage is found fix or replace immediately to avoid costly NON-WARRANTY repairs.
6. Avoid tool and work pieces from crashing into these units.
7. When mounting jigs, truniuns, and other fixtures, be sure the brake is in the clamped position. Failure to do so will result in damage to the drive gear and costly NON-WARRANTY repairs.
8. When using truniuns and long work pieces use tailstocks with LIVE centers or BEARING style end supports.

## UDNC-M1 & M2 WARNINGS




**ELECTRICAL  
WARNING**

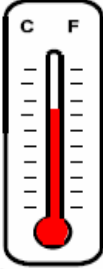
KEEP AWAY FROM COOLANT, WATER, OILS



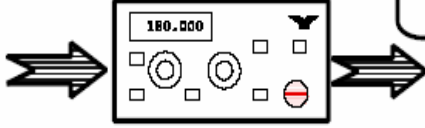
AND OTHER CONDUCTIVE LIQUIDS!



**ELECTRICAL  
WARNING**



PROVIDE PLENTY OF AIR FLOW!  
EXCESSIVE HEAT WILL DAMAGE  
CIRCUITRY AND COMPONENTS.



**DO NOT BLOCK FANS OR FLOW WAYS!**  
**DO NOT RUN WITHOUT SCREEN FILTERS!**  
CHECK & CLEAN FILTERS REGULARLY.

**NOTE:** It absolutely necessary that the 110-220 VAC uses a true Earth Ground. If a cabinet or chassis ground is used, the control box will be susceptible to voltage spikes and floating ground noise which will create undesirable effects to the system and your production.

## SECTION 1: GETTING STARTED

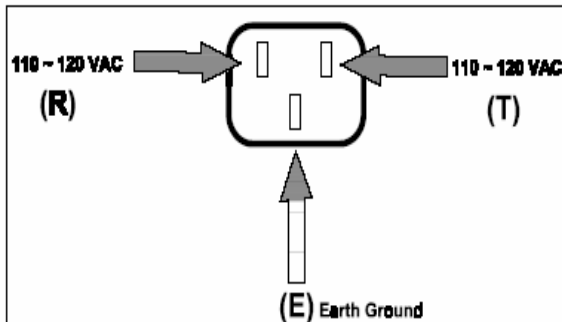
## Un-crating:

This is a fairly minor task yet an important one. Please make sure you maintain the warranty card, test report, and manual. You will notice the test report provides you with the center height specifications and serial number for your unit. Save this data as it provides us specific information about your unit. As a precaution we also recommend that you maintain the original shipping crate for at least 30 days.

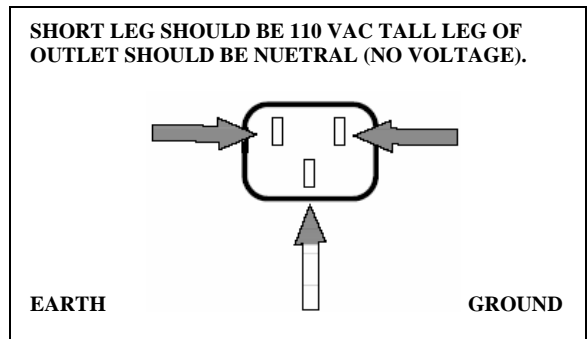
## Preliminary Set Up:

First connect all motor cables from rotary table to the control box. Be sure to thread Amphenol connector ALL THE WAY to insure proper connection. Then plug in power cord to 110-220 VAC power source.

220 single phase plug example



110 vac plug example



**{Do not hook up to 3 phase A/C or you will blow the safety fuses and possibly damage the controllers circuitry and PLC's.}**

## Tech Tip:

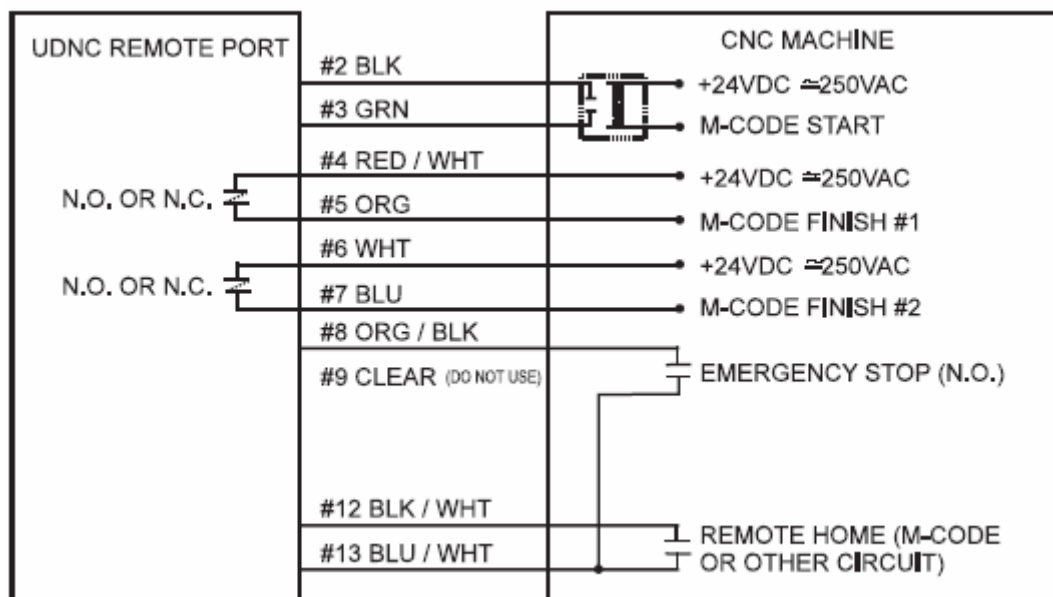
If you are hard wiring to the single phase transformer on your CNC, connect to the connector as shown in the picture above. Most CNC machines have these lines marked for easy identification. You must always confirm voltages with a multi-meter prior to connecting any voltage lines.

## SIMPLE HOOK UP TO CNC

**WARNING:** If you are not an electrician or have not interfaced an M-function product before, we strongly recommend, having a qualified electrician do this interface. Like anything else it's easy if you know how and very dangerous if you do not.

Supplied with your new rotary table is what we classify as a remote cable. On one end is a cannon plug and the other has thirteen terminated wires. These wires have several functions, as shown in the electrical schematic, below, but for this quick reference we will only concern ourselves with four wires. (#2 black and #3 green and #4 red/white and #5 orange)

**Warning:** When installing the remote cable pay very close attention to the wire colors as some are very similar. Connecting these wires incorrectly can create very expensive NON-WARRANTY repairs of both your CNC machine and rotary table.



**Tech Tip:** Do not leave unused bare wires exposed or twisted together. If you know you will only be using the step start and finish (#2, 3, 4, & 5) then be sure to cut back all remaining wire. Do not have remote cable connected when cutting extra wires.

## Remote Cycle Start (M-Code)

1. Determine first, where your M-code is outputting. In some machines the M-code may have been wired out to a terminal strip. On other machines may also have included a relay for the M-code out put.
2. If the relay is already installed on your machine, simply connect wires #2 black and #3 green of our remote cable, to the normally open contacts on the relay or terminal strip, if there are wires already running from the relay.
3. If you machine does not have this indexer M-code relay, you may need to install one. On most machines the M-code output acts as switch for completing a circuit, others output voltage either 24v DC or 110v AC.
4. When adding the relay you must first determine whether you need 24v DC or 110v AC, check your machines wiring schematic to determine this and verify with a meter. Then make sure you have the correct type of relay as the machine may have a receptacle in place. Most machines utilize an “ice cube” relay.
5. If your machines M-code outputs voltage, either 24v or 110v, then run a wire from the M-code output to the one side of the coil on the relay. Hook the other side of the relay to ground.
6. If you M-code is switch type, like most machines, then run a wire from the M-code to one side of the coil and voltage to the other side typically 24v DC.
7. Then always connect wires #2 black and #3 green of the remote cable to the normally open contacts of the relay.

## **Remote M-Code Finish**

The way most machines work once the M-code is initiated the rotary table controller will execute the next step in the program. Once the rotary has reached the programmed position the encoder verifies position and either a contact closer happens, if using normally open contact, or a contact open will occur, if using normally closed contacts. Most CNC machines utilize normally open contacts and operate with a momentary close, either 0.5 seconds or 1.0 second. Again as always consult with your machines electrical drawings for confirmation. If your machine is looking for open contacts on the return signal you would connect wire #4 red/white of our remote cable to 24v DC and #5 orange to the M-Finish of your CNC machine.

## **WARNING READ THIS:**

**Note: When the control box is in the SAVING MODE treat it like you would your Computer.** If for any reason the box is powered off during the saving

process, you will lose all the information contained in the UDNC. To correct this problem, the UDNC will have to be sent back to Yuasa International for reprogramming. Please wait 20 to 30 seconds after the SAVE command is finished before recycling the power.

**WARNING:** Do not access the Program or Terminal modes from the Key pad with the RS-232 cable connected to the UDNC-M1/M2.

## **Section 2: UDNC M1 & M2**

### **Panel Switches**

1. **On & off** Be sure to check incoming voltage, prior to powering up. The off switch totally shuts off the entire system.

2. **Program select** is used to access any of the programs. These programs are recognized when the program select key is pushed & the program # is set to the desired program #. Then press the Enter key to enter program #. Next press the Program Reset key to activate the desired program. The program that is being run will be displayed on the lower left side of the display.

**NOTE:** The UDNC must be in the Manual mode when using the front panel switches.

3. **Jog cw, Jog ccw, & Jog return:** these keys will make the rotary table rotate CW or CCW respectively, & the return key will send the rotary table back to the position where it was at prior to being jogged.

**NOTE:** Rotary table will perform incremental Jog if the feed rate is at 10,20,30,40, or 50%.

10%=0.0001 increment change. If key is held down 3 seconds table will go into continuous jog.

20%=0.0010 increment change. If key is held down 3 seconds table will go into continuous jog.

30%=0.0100 increment change. If key is held down 3 seconds table will go into continuous jog.

40%= 0.1000 increment change. If key is held down 3 seconds table will go into continuous jog.

50%=1.0000 increment change. If key is held down 3 seconds table will go into continuous jog.

60~200% initiates a continuous jog.

4. **Feed rate % increase/decrease** controls both the jog feed & the program feed. This function operates in 10% settings. You will note that the Feed rate will reach a maximum of 200%. Yes you can run over 100%, but torque is greatly sacrificed. Use extreme caution when exceeding 100%.

5. **Manual/Remote:** Manual is used for operating the front panel keys & for use of the pulse generator. In this mode, the brake will be disabled & no remote I/O will

be accepted. In addition, the auto stepping function (BLK# key) will be enabled. In Remote the following front panel keys (W-Z set, W-Z return, jog cw, jog ccw, & cycle start) will be disabled.

6. **BLK#:** **Auto-stepping feature:** (Must be in Manual Mode), press the BLK# key, press the block # in your program you wish to auto-step to, then press the enter key to set the desired block #. Enter the desired BLK# of your program that you wish to auto-step to on the next cycle start. If it is greater than the current block #, on the next cycle start the program will auto-step to this block #. The BLK# Key will terminate the entry without setting the block #. Switching to remote mode will terminate the auto-stepping.

**NOTE:** If the BLK# is larger than the # of blocks in the program, auto-stepping will continue until terminated by selecting the remote mode. This allows you to continuously run a program, very useful when demonstrating at shows or when doing a dry run.

7. **Shift key:** This key activates any key that is hi-lighted in Yellow. The shift key must be held down when used with any yellow hi-lighted key.

8. **Cycle Start/Machine Zero:** The cycle start key initiates a step start. Each press of the cycle start key will step thru 1 line of the program. To initiate a Machine zero you must press & hold the Shift key then press the Machine zero key. This will begin a search of the rotary table spindle, When the M/Z switch is activated this will begin the search for the "C" channel of the encoder, which is classified as machine zero. This offset is set by changing the value of parameters i126= (axis1), i226= (axis 2).

9. **Work Zero Set:** This key sets the current position of the rotary table as work zero. Once you have pressed this key, any previous W-Z setting will be lost, & the controller will recognize this new W-Z position.

**NOTE:** Please refer to setting work zero in the following section.

10. **Work Zero Return:** One press of this key will return you to the current W-Z position. This will also reset the program to the zero block #.

11. **E-Stop:** Emergency stop is used to totally stop all movement. To restart the controls simply press the Program Reset key.

12. **Axis Select:** with each press of this key the display will cycle the axis selected between 1, 2, or both. The selected axis directs the actions of the front panel keys for Jog (CW, CCW, & Return), W-Z Rtn. W-Z set.

## **WARNING READ THIS:**

**Note:** When the control box is in the **SAVING MODE** treat it like you would your **Computer**. If for any reason the box is powered off during the saving process, you will lose all the information contained in the UDNC. To correct this problem, the UDNC will have to be sent back to Yuasa International for reprogramming. Please wait 20 to 30 seconds after the **SAVE** command is finished before recycling the power.

**WARNING:** Do not access the Program or Terminal



modes from the Key pad with the RS-232 cable connected to the UDNC-M1/M2.

### **Special Program Editing Control Keys**

1. **Yellow Shift & Edit:** Displays Terminal <- or Program Edit->. Press the <- (left arrow key) for terminal mode (Edit parameters, Set home offsets) or press the-> (right arrow key) for program edit. Pressing the Yellow Shift & Edit keys terminates the program edit or terminal modes.
2. **Backspace:** If in the program edit mode, pressing this key backspaces the last entered character. This is active only in program mode.
3. **Up arrow:** If in the program edit mode, pressing this key scrolls the displayed program Up 1 line. This is active only in program mode.
4. **Yellow Shift & Up arrow:** If in the program edit mode, it scrolls the displayed program Up 1 page. This is active only in program mode.
5. **Down arrow:** If in the program edit mode, it scrolls the displayed program Down 1 line. This is active only in program mode.
6. **Yellow Shift & down arrow:** If in the program edit mode, it scrolls the displayed program Down 1 page. This is active only in program mode.
7. **Right arrow:** If in the program edit mode. It moves the cursor to the right stopping at the end of the line. This is active only in program mode. To answer questions YES press this key.
8. **Yellow Shift & Right arrow:** If in the program edit mode. It moves the cursor to the right to the end of the line. This is active only in program mode.
9. **Left arrow:** If in the program edit mode. It moves the cursor to the left stopping at the beginning of the line. Active only in program mode. To answer questions NO press this key.
10. **Yellow Shift & Left arrow:** If in the program edit mode. it moves the cursor to the left to the beginning of the line. Active only in program mode.
11. **DEL:** If in the program edit mode deletes the character to the right.
12. **Yellow Shift & DEL Line:** If in the program edit mode it deletes that line leaving the cursor at the beginning of the blank line.

13. **Enter**: Completes the line entry of data or is an answer YES for questions like **SAVE** or **Download**. To answer questions YES press this key.

14. **INS**: This key toggles the Edit mode between insert & overlay.

15. **Yellow Shift & INS line**: Inserts a blank line leaving the cursor at the beginning of the line.

## **Cable connections**

**Remote Port**: Is to connect the provided Remote cable for M-Code Start/Finish signals, E-Stop, & Remote Home.

**Motor 1 or 2**: Connect Motor cables here. X-axis Rotary=Motor#1. Y-axis Tilt=Motor#2.

**Hand wheel Port**: Connect the MPG-300 pulse generator here. (Optional Accessories).

## **WARNING READ THIS:**

**Note**: When the control box is in the **SAVING MODE** treat it like you would your **Computer**. If for any reason the box is powered off during the saving process, you will lose all the information contained in the UDNC. To correct this problem, the UDNC will have to be sent back to Yuasa International for reprogramming. Please wait 20 to 30 seconds after the **SAVE** command is finished before recycling the power.

**WARNING**: Do not access the Program or Terminal modes from the Key pad with the RS-232 cable connected to the UDNC-M1/M2.

## **Setting Work Zero,**

1. Power on the UDNC.
2. Jog the spindle 15 degrees or more from vernier "0" using the Jog CW key to make sure that the Home Switch is not on the Home position Dog.
3. Press & hold the Yellow Shift key & the Machine zero keys together. This will initiate a Machine Zero return.
4. Press the Jog CW or the Jog CCW keys until you reach the desired work position zero, or use the optional MPG-300.

5. Push the Work Zero set key once you have reached the desired position. The display will indicate SAVING then read 0.0000.
6. Press the WZ Return key when you need to return your work piece to home setting.

To recover a previous work zero setting after power off:  
Repeat steps 1~3 then press WZ Return.

NOTE: If you do not perform steps 1~3 after power down then work zero setting will not be correct.

## **WARNING READ THIS:**

**Note:** When the control box is in the SAVING MODE treat it like you would your Computer. If for any reason the box is powered off during the saving process, you will loose all the information contained in the UDNC. To correct this problem, the UDNC will have to be sent back to Yuasa International for reprogramming. Please wait 20 to 30 seconds after the SAVE command is finished before recycling the power.

**WARNING:** Do not access the Program or Terminal modes from the Key pad with the RS-232 cable connected to the UDNC-M1/M2.

## **Section 3: Programming Codes, Context, & Examples**

### M & G Codes

The programming of this controller utilizes standard M & G codes, found in common CNC controls. Below is listing of these codes, & there meaning. A more detailed description follows.

#### G Codes

G04	Dwell.
G26	Machine Zero axis #1
G27	Machine Zero axis #2
G28	Machine Zero axis #1 & #2
G70	Start Loop
G71	End Loop
G72	Auto Divide
G90	Absolute Positioning
G91	Incremental Positioning

#### M Codes

M18	Disables step finish relay #1
-----	-------------------------------

M19	Enable step finish relay #1
M20	Disables step finish relay #2
M21	Enable step finish relay #2
M22	Disables Brake on
M23	Enables Brake on
M30	End of Program
M40	Enables Continuous Motion
M41	Disables Continuous Motion

### G Code Description

#### G04, Dwell (7) digits, 1 to 8388607 milliseconds:

A G04X1000 would cause the axis to remain in position 1 second before being ready to execute the next block.

Example:

```
F1000 G90
G04X5000A90
M30
```

This program will cause the table to dwell 5 seconds then move to the 90 degree position.

#### G26 Machine Zero Axis #1:

A G26 in the program will cause the #1 axis to do a machine zero return.

Example:

```
F1000
G26
M30
```

This program will cause the table to execute a machine zero return of axis #1 (Rotary Axis)

#### G27 Machine Zero Axis #2:

A G27 in the program will cause the #2 axis to do a machine zero return.

Example:

```
F1000
G27
M30
```

This program will cause the table to execute a machine zero return of axis #2 (Tilt Axis)

#### G28 Machine Zero Return Axis #1 & #2:

A G28 in the program will cause the #1 & #2 Axis' to do a machine zero return.

Example:

```
F1000
G28
M30
```

This program will cause the table to execute a machine zero return of Axis #1 & #2 (rotary & tilt)

**G70, G71 Start & End Loop:**

**G70X##### (5 digits) 0~65535. 0= infinite.**

All Blocks between G70 & G71 commands are executed as many times as specified in the G70 command.

**Example (A):**

**F1000G91  
G70X15A180  
G71  
M30**

Table will move incrementally 180 degrees 15 times.

**Example (B):**

**F1000G91  
G70X0A90  
G71  
M30**

Table will move incrementally 90 degrees with every start signal.

Note: A looping command May not be used inside a loop. The first loop will reject the statement of the secondary command.

**G72 Auto-Divide:**

**G72X##### (5 digits) 2~65535.**

This command will instruct the unit to divide the next movement commanded into the number of equal divisions you require. The division takes care of residual encoder counts & insures that no cumulative error occurs during incrimination, & that the last movement, will execute correctly.

**Example (A):**

**F1000G91  
G72X10A360  
M30**

This program will evenly divide 360 degrees 10 times.

**Example (B):**

**F1000G91  
G72X10A180  
M30**

This program will evenly divide 180 degrees 10 times.

**G90 Selects Absolute Positioning:**

This command is seen as default, if G91 (incremental) is not initiated. An absolute movement that is relative to Zero.

**Example:**

**F1000G90**

A180  
A270  
M30

This program will move the rotary table, (with each start signal) first to the 180 degree position, then to the 270 degree position, all referenced from zero.

**G91 Selects Incremental Positioning:**

An incremental move is one that is relative to the current position, not the zero location.

**Example:**

F1000G91

A90

A180

M30

This program will move the rotary table, first 90 degrees then an additional 180 degrees, from the 90 degree position, a total of 270 degrees.

**Note: G90 & G91 are modal commands & will remain in effect until changed.**

**WARNING READ THIS:**

**Note:** When the control box is in the SAVING MODE treat it like you would your Computer. If for any reason the box is powered off during the saving process, you will lose all the information contained in the UDNC. To correct this problem, the UDNC will have to be sent back to Yuasa International for reprogramming. Please wait 20 to 30 seconds after the SAVE command is finished before recycling the power.

**WARNING:** Do not access the Program or Terminal modes from the Key pad with the RS-232 cable connected to the UDNC-M1/M2.

**M Code Description**

**M18 Step Finish #1 Disable:**

Writing this code into a program block will disable the step finish signal on relay #1.

**M 19 Step Finish #1 Enable:**

Writing this code into a program block will enable the finish signal relay #1 if M18 has been used.

**Note: It is not necessary to put this code in a program; the relay is turned on by factory default.**

**M20 Step Finish #2 Disable:**

Writing this code into a program block will disable the step finish signal on relay #2.

**M21 Step Finish #2 Enable:**

Writing this code into a program block will enable the finish signal relay #2 if M20 has been used

**M22 Brake Function Disable:**

This command will disable the “Brake on” function in remote mode.  
{Default=Brake On in remote mode}.

**M23 Brake Function Enabled:**

This command will enable the “Brake on” function in remote mode.  
{Default=Brake On in remote mode}

**M30 End of Program:**

All programs must be terminated by an M30 command.

**M40/M41 Enable/Disable Continuous Motion:**

This code will execute all movement commands that follow it without stopping at the end of each move to flawlessly blend them into 1 fluid motion without the need of repeated start signals. Ideal for Helical work. This M-code can be enabled or disabled anywhere in the program.

Another use is for generating unusual motion profiles, such as hyperbolic, parabolic or arbitrary curves. This command provides a continuous motion string, blending different programmed moves & federates, with 1 step start command. This is extremely useful if you are attempting to develop a helical move, synchronizing an “X” axis move with a rotational move. Many times the lead or pitch of a particular part cut will change, necessitating the need to alter the rotary feedrate & movement without stopping for another step start command. The movements & feedrates entered between M40 & M41 will execute in a continuous fashion, permitting this type of move.

**Example:**

```
F1000G90  
M40A90F25A45F50A30  
M30
```

This program will move to the 90 degree position at 100% feedrate, then without stopping will move to the 45 degree position at 25% feedrate, then without stopping will move to the 30 degree position at 50% feed rate.

**Note: The M41 is not always required, program will finish at M30 & reset without M41.**

## **Special, & motion Commands**

**(F)Feed rate Settings:**

The (F) setting in the program, acts as a percentage of feed, based on the maximum RPM of the unit(s) you are programming.

Example: A DMNC-130 has a maximum RPM of 90 of 100% Feed rate. If you have a fixture that you plan to run at 45 RPM, you would have a feed rate of: F50 or 50% of max speed.

**Note:** The “F” command is modal, & will remain in effect until changed.

**(TM) Time Mode Programming:**

Where: ##### is 1 to 8388607 in milliseconds.

This command may be used whenever a specified move time is desired rather than a feed rate. Its primary use in the controller is if a timed programming is used on multiple axis, regardless of the move sizes on each axis, all moves will terminate simultaneously, thereby, providing an 8 axis linear interpolation capability.

Feed rate “F” programming, which is modal, can be used interchangeably with time programming “TM” in any program. Care must be taken to re-specify the desired feed rate by placing an “F” command in the program immediately after the last “TM” move.

The time round off error for very small and slow moves is traded in favor of velocity round off error. This permits multiple controller systems to remain synchronized through out the full dynamic range of the controlled operation.

The velocity round-off is small, approximately equivalent to 200 encoder counts, for a 2 hour move: i.e. if you tell the controller to move 2,000,000 counts in two hours, it will terminate the move exactly on time, but may make a 100 to 150 encoder count move at the very end of the time period, instead of equally spreading the move, thus creating a small velocity jerk. Time and position remain accurate. This error rapidly diminishes as the move times become shorter. The same move made in 20 seconds, would have an undetectable velocity error of ½ of an encoder count.

**Time Constant:** 1 second is equal to a “TM” value of 1000.

If you wanted to move 360 degrees in 1 minutes, you would first multiply 60 (seconds) and 1000 which equals: 60000. Your “TM” value would appear as: TM60000A360.

**WARNING READ THIS:**

**Note:** When the control box is in the **SAVING MODE** treat it like you would your **Computer**. If for any reason the box is powered off during the saving process, you will loose all the information contained in the UDNC. To correct this problem, the UDNC will have to be sent back to Yuasa International for reprogramming. Please wait 20 to 30 seconds after the **SAVE** command is finished before recycling the power.

**WARNING:** Do not access the Program or Terminal modes from the Key pad with the RS-232 cable connected to the UDNC-M1/M2.

**Program Examples**

**Accessing, & Writing New Programs:**

1. Press the program select key.



2. Select the program # you wish to access. Note: Motor must be connected.
3. Press the Enter key.
4. Press the program reset key.
5. Press & hold the shift key & the edit keys together.
6. You will be asked to select Terminal <-, or Program edit? ->, Select Program Edit by pressing the Right arrow key -> or ENTER.
7. Edit Program as necessary or writing new program.
8. Press & hold the Shift key & Edit keys together.
9. You will be asked to download Program? NO<- or YES ->. To answer YES press the Right arrow -> key or press the Enter key. To answer NO press the Left Arrow <- key.
10. You will be asked to save? To answer YES press the Right arrow -> key or press the Enter key. To answer NO press the Left Arrow <- key.
11. The Box will now say SAVING PROGRAM.

## **WARNING READ THIS:**

**Note:** When the control box is in the **SAVING MODE** treat it like you would your **Computer**. If for any reason the box is powered off during the saving process, you will loose all the information contained in the UDNC. To correct this problem, the UDNC will have to be sent back to Yuasa International for reprogramming. Please wait 20 to 30 seconds after the **SAVE** command is finished before recycling the power.

**WARNING:** Do not access the Program or Terminal modes from the Key pad with the RS-232 cable connected to the UDNC-M1/M2.

In this first example, we have first set the AXIS as A, enable the block counter, and set the feed rate to 100%. We will then move incrementally 90 deg., from zero, then incr. minus 180 degrees. Then we switched to absolute (G90) and moved to the 170 deg. position, relative to zero. Then back to incr. (G91) minus 95.3025 deg. then home. You will note that we used a G90A0 for work zero; you must be in ABS (G90) mode for a machine zero return.

```
F100G91
A90
A-180
G90A170
G91A-95.3025
G90A0
M30
```

In this second example, we wanted to rotate 360° CW then 360° CCW (without stopping) at 50% feed rate.

```
F50G91
M40A360A-360
M30
```

In this third example, we are positioning in 15 degree increments.

```
F100G91  
G72X24A360  
M30
```

In this example, we will be commanding both axes simultaneously. (UDNC-M2 will operate 2 axes whereas the UDNC-M1 is for single axis.) In this program we will command the "B" axis to rotate 57 degrees. When "B" has stopped, "A" will be indexed 3.6 degrees 100times. "B" will then move 33 degrees at which time "A" will index 6 degrees 3 times. Then both axes will be returned to absolute zero.

```
F100G90  
B57  
G72X100A360  
B33  
G72X3A18  
G90A0B0  
M30
```

This last example will have Axis "A" perform a Timed Move using the example from Time Move description, while "B" moves at 3% feed rate 30 degrees. After which both axes will return to absolute 0 degrees.

```
TM247940A45G91F3B30  
F100G90A0B0  
M30
```

Other program styles are available; these are merely examples of what can be written in a simple format. If you have any questions and/or need programming advise, please call our technical staff at 800-323-7427 for assistance.

## **WARNING READ THIS:**

**Note:** When the control box is in the **SAVING MODE** treat it like you would your **Computer**. If for any reason the box is powered off during the saving process, you will loose all the information contained in the UDNC. To correct this problem, the UDNC will have to be sent back to Yuasa International for reprogramming. Please wait 20 to 30 seconds after the **SAVE** command is finished before recycling the power.

**WARNING:** Do not access the Program or Terminal modes from the Key pad with the RS-232 cable connected to the UDNC-M1/M2.

#### Section 4: P & I Variables (parameters)

**Warning:** Changing parameter settings may cause controller faults. Check with the Yuasa factory before altering these settings.

**NOTE:** Whenever an I or P variable has been changed & saved the UDNC must be powered off then on to save the changes.

#### Accessing & Editing P & I Variables:

##### To View Variables

1. Press & hold the shift key & the edit keys together.
2. You will be asked to select Terminal <- or Program Edit? ->, Press the Left Arrow <- key for terminal mode.
3. Using the Shift key to access the I & P keys, input the variable # you wish to access.

Example: P32 press Enter the display will look like this.

P32-> (Parameter #)  
1-> (Parameter Value)

##### To Change a variable

Perform steps 1~3

4. After selecting the variable you wish to change input an equal sign (=) then input the value you wish to change this variable to.

Example: P32=2

5. Press the Enter key.
6. Press & hold the Shift key & Edit keys together.
7. You will be asked to Save? To answer YES press the Right arrow -> key or press the Enter key. To answer NO press the Left Arrow <- key.

8. The Box will now say SAVING PROGRAM.

#### **WARNING READ THIS:**

**Note:** When the control box is in the SAVING MODE treat it like you would your Computer. If for any reason the box is powered off during the saving process, you

will lose all the information contained in the UDNC. To correct this problem, the UDNC will have to be sent back to Yuasa International for reprogramming.

**WARNING:** Do not access the Program or Terminal modes from the Key pad with the RS-232 cable connected to the UDNC-M1/M2.

**NOTE:** Whenever an I or P variable has been changed & saved the UDNC must be powered off then on to save the changes.

#### P Variables

P11= Axis #1 gear ratio  
P21= Axis #2 gear ratio  
Example: P11=90 (90:1 gear ratio)

P21=48 (48:1 gear ratio)  
P15= Axis #1 Jog speed & direction factor.

P25= Axis #2 Jog speed & direction factor.  
P16= Axis #1 Machine Zero speed & direction factor.  
P26= Axis #2 Machine Zero speed & direction factor.

**P29 (Axis#1) & P39 (Axis#2): Brake off before Step**  
This variable determines 2 things. A) If a brake is being used and (B) how long it takes before the brake disengages.  
Units without pneumatic brakes will have a value of 0, signifying no braking time allowed. This variable is set in units of 10 milliseconds times the value entered.

If this value is lowered below factory setting, the table will try to index before the brake has completely release which can cause the UDNC to report "Brake Error", "Pos Follow Error", or the unit may fail to move at all. In the last case, a step finish signal maybe fired to the CNC making it begin its next cycle, thus scrapping your part. Although 500ms may work fine for most purposes, it is recommended that you check your work with a dry run and confirm that this value is substantial.

**P30: Brake on after Step**  
This variable determines how long after the axis has reached position before the

brake engages. If this value is too low it will try to engage before the axis has reached its designated position. As with P29, its value is multiplied by 10ms.

#### **P31: Step Finish Delay**

This variable sets the length of the Step Finish signal that will be fed back across the remote lines when a non hand shaking signal is used (i.e.: P32 = 1 or 3.) It is important the length be longer than P30 or signaling and braking will conflict.

#### **P32: Step Start type**

Type: 0 - No Step Finish.

1 - (Momentary contacts) No handshake with a delay set by "P31"

2 - (Full handshaking) Handshake by remote step input, going low. {Default}

This means the start signal stays on until the CNC receives a Finish.

3 - Handshake by second remote step input, going high. The remote step should stay set until it sees the remote step finish change state, or remain set for at least the time set in "P29"

4 - Step Finish is synchronous with motion start and resets the step finish when the Remote CYCLE START resets.

#### **P33: Step Finish Relay Polarity**

Polarity refers to either normally Open (N.O.) or Normally Closed (N.C.) relays. If your CNC is looking for the relay to be closed during indexing and to open upon completion and then closes, this is N.C. If the CNC is looking for an Open during indexing then a closure and reopen at the end of the move, this in N.O.

The UDNC is set up with 2 step finish relays #1 & #2. These relays can be set up to work together or independently of each other.

P33=0 relay #1=N.C. Relay #2=N.C.

P33=1 relay #1=N.O. Relay #2=N.C.

P33=2 relay #1=N.C. Relay #2=N.O.

P33=3 relay #1=N.O. Relay #2=N.O.

#### **P42: Enable Front Panel Cycle Start in Remote**

P42=0 Cycle start key Disabled in remote mode.

P42=1 Cycle start key Enabled in remote mode.

#### **P44: Program Input Remote/NC**

P44=0 Will run program that is stored in the program buffer.

P44=1 Will run Remote NC RS-232 program input

NOTE: All program lines must be executed via M-Code or with the Cycle Start key.

#### **P46: Auto step delay**

P46=Sets the auto step delay in units of 1/2000 seconds.

### **I Variables**

Ix13overtravel (OT) position Software Limit: (x = Motor 1 or Motor 2) Axis travel in the Positive or Clockwise direction. This limit is set based on the Gear Ratio to Encoder value multiplied by the degrees to limit. I113 is Axis #1 I213 is Axis #2.

[See Chart Below for GRE Values].

<b>GRE VALUE CHART</b>		
<b>MODEL</b>	<b>ROTARY VALUE</b>	<b>TILT VALUE</b>
DMNC-5C/130	990	NA
DMNC-170	1320	
DMNC-220	1980	
DMNC-280/320	2640	
DMNC-400/500	3960	NA
DMTRT-170	1584	7920
DMTRT-220	1980	7920
UTRT-280/320	2640	7920

**GRE Formula: 22 times gear ratio = GRE Value=1 degree**

**Special Note:** If parameter P11 for A axis, P21 for B axis, is a positive value then i113 is your OT positive value and i114 is the OT negative value. If parameter P11 or P21 is a negative value then i113 is the OT negative value and i114 is the OT positive value.

Regardless i113 must be entered as a positive value and i114 must be entered as a negative value. Example i113=9000 and i114=-9000.

**Example:** You have a DMTRT-170 and would like to limit the amount of positive travel on the tilting axis to 75 degrees to prevent the table accidentally crashing due to the size of your work fixture. You would take the value of 2640 (The tilt axis on the DMTRT-170 is 120:1) and multiply by 75. [2640 x 75 = 198000] You will then input this value in the i213 variable, I213=198000.

**Ix14 Over travel position Software Limit:** (x = Motor 1 or Motor 2) Axis travel in the negative or Counter Clockwise direction. This limit is set based on the Gear Ratio to Encoder value multiplied by the degrees to limit. {See Chart Above for GRE Values} As with the example for ix13, you would follow the same procedure, only in this case, the value you store in Ix14 will be a negative value, i.e., 2640 x -5 = -13200 or -5 degrees in CCW.

I114 is Axis #1 I214 is Axis #2.

**Ix26 Machine Zero Offset.** This is the setting relative to the home trigger signal. If you send the unit to true Machine Zero and the unit is at 358.0000, then by increasing this parameter by increments of 16, you can adjust the axis to true zero. Again, positive values will move Clock Wise and negative values Counter Clock Wise. Range: -8,388,608 ~ 8,388,607 {to figure how many encoder counts = 1 degree follow this formula: 22 X gear ratio= Z. Multiply Z x 16 = 1 degree.}.

Below is a simple gear ratio chart.

<b>GEAR RATIO CHART PER 1 DEGREE</b>		
<b>TABLE</b>	<b>GEAR RATIO</b>	<b>1 DEGREE VALUE</b>
DMNC-5C/5CA	45:1	16000

DMNC-5CA- 2/3/4		
DMNC-130		
DMNC-170	60:1	21333
DMNC-220	90:1	32000
DMNC-280/320	120:1	42666
DMNC-400	180:1	64000
OTHER GEAR RATIOS		
GEAR RATIO	1 DEGREE VALUE	
57:1	20266	
72:1	25600	
360:1	128000	

**I54 Baud Rate Settings.** By default, this value is 12 or 38400 baud. Even values change the baud rate higher or lower.

Value	Baud Rate
2	1200
4	2400
6	4800
8	9600
10	19200
12	38400
14	57600
16	115200

**I65 Parity & Bits Default =0, 8 bit/no parity.**

**1= 7 bit/odd parity.**

**3= 7 bit/even parity.**

## **WARNING READ THIS:**

**Note:** When the control box is in the **SAVING MODE** treat it like you would your **Computer**. If for any reason the box is powered off during the saving process, you will lose all the information contained in the UDNC. To correct this problem, the UDNC will have to be sent back to Yuasa International for reprogramming. Please wait 20 to 30 seconds after the **SAVE** command is finished before recycling the power.

**WARNING:** Do not access the Program or Terminal modes from the Key pad with the RS-232 cable connected to the UDNC-M1/M2.

**NOTE:** Whenever an I or P variable has been changed & saved the UDNC must be powered off then on to save the changes.

## Remote RS-232 program input.

NC requirements:

Macro B or equivalent

Note: Your machine program look ahead parameter must be deactivated.

NC parameters must be set for LF/CR (Line feed/Carriage return after DPRNT statement).

Baud rate & Stop bit/parity must match (See parameters I54 & I65).

UDNC-M control default is I54=12 & I65=0 (Please refer to the parameter section of this manual.

Please refer to the Macro programming section of your NC programming manual.

Getting the UDNC-M1/M2 control ready

Set parameter P44 to =1, Save it, recycle power to change parameter.

Switch the UDNC to REMOTE MODE, the UDNC is now open to RS-232 input from the NC machine.

All program lines must be executed via M-Code or with the Cycle Start key.

Program examples:

NC program should look like this.

Program #

POPEN;

DPRNT [G91F100A120M2];

PCLOS;

M21 (Example of CNC machine M-code to fire step start)

M30;

%

Once you execute this program the rotary table should move incrementally 120 degrees. Essentially what is happening here is that the RS232 port on the CNC control is being opened for communication "POPEN", then the CNC control is being told to send the characters inside the square brackets [ ] out thru the port "DPRNT", in ASCII format. Once the UDNC-M1/M2 controller receives this data. The CNC machine M-Code will automatically execute a move.

NOTE: If indexer fails to move. The block counter must be reset before another attempt is made. To do this press the manual remote switch. Then press the program reset button on the control panel. Return to remote mode and test again.



### Direct Programming As A Variable

The following is a method of placing the Macro “DPRNT” command line, in a variable. This allows the programmer to use a simple program line to execute any rotary movement. NOTE: If you do not have experience with Macro programming, it is recommended that you contact your CNC dealer. Yuasa does not guarantee this method of programming will work in all cases.

#### CNC Machine MACRO program (single axis sample only)

```
%  
:9010;  
POPEN;  
DPRNT [G90F100A#1[33] M2];  
PCLOS;  
M21; (example of Machine M-code)  
M99;  
%
```

#### (Dual axis sample only)

```
%  
: 9010;  
POPEN;  
DPRNT [G90F100A#1[33] B#2[33] M2];  
PCLOS;  
M21; (example of Machine M-code)  
M99;  
%
```

### Explanation

Programs beginning with a “9” are considered Macro program numbers. Some of these programs are called with a G-CODE (example, G15), and others are called with M-codes, (please refer to your control manual). In the DPRNT line note the A#1&B#2, this requests the program to look at the A or B “index movement” in the main program line. The [33] states that the index degrees will be 3 digits (maximum), for example “180” degrees, and to look 3 digits left of the decimal. If you need to use fractional degrees like 180.5025 degrees, then you may wish to input [73] 7 total digits, with 3 digits left of the decimal. Also note the “M21” after the “PCLOS”. This is a sample of the M-code need to execute the DPRNT command line. This M-code will change depending on the brand of machine. This M-code will be a spare M-code accessed via the remote cable supplied with the UDNC-M1/M2.

**NOTE:** In no way does the RS232 report back a position from the UDNC to the CNC control. The UDNC control only listens to commands.

**Main CNC Program Line Example**

Example G15 calls the “9010 Macro”

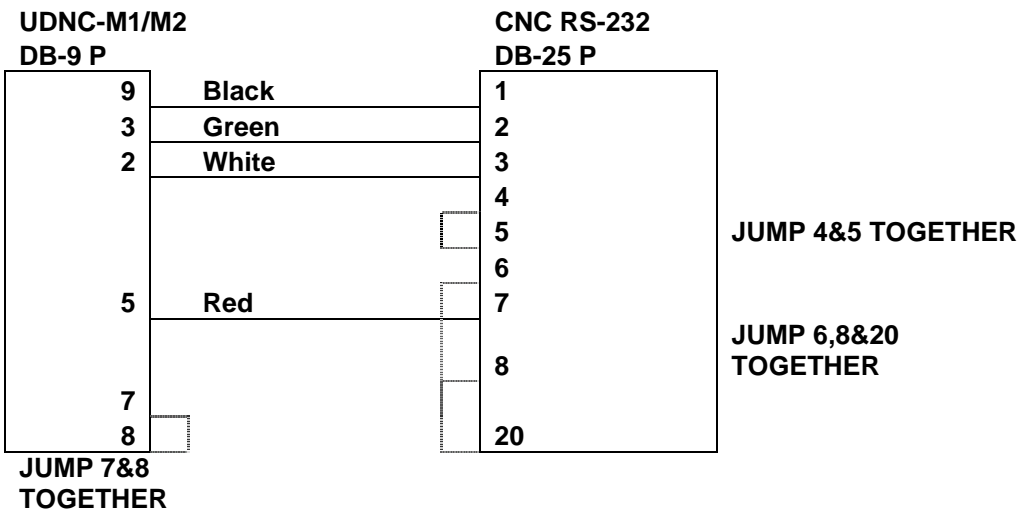
G15A180 (This line in the main program would provide an absolute move of 180 degrees).

G15A120B90 (This line in the main program would provide absolute moves for the A axis to move 120 degrees & the B axis to move 90 degrees).

NOTE: you may use another Macro program changing only the “G90” to “G91”.

**CNC Machine RS-232 Wiring for UDNC-M1/M2**

When using the “DPRNT” function with Macro “B” on CNC Controllers, the CNC needs to have a Null Modem Confirmation: However, the UDNC-M1/M2 needs a NULL connection. To complete this operation, you will need to go from 25 pin to 9 pin RS232 connector cable to be wired as below:



By making the jumped circuit shown, the CNC Null modem requirement is completed.

NOTE: depending upon the Mach. Control, some times you may find that you need to run line 5 on the 25 pin to line 7 on the 9 pin & line 4 on the 25 pin to line8 on the 9 pin side. However, please try the connection above 1<sup>st</sup>.

## **ERROR CODES**

**POSS\_ERR:** This is the most common error code. This simply means the indexer did not reach the programmed position. A possible cause for this is the braking mechanism is rusted or stuck. Another possibility is the indexer is bound and or the motor is not working. If your unit is still under warranty period DO NOT disassemble anything with out the consent of the Yuasa Service Dept. (1-800-323-7427).

**IPKA\_ERR:** This is error means the over current sensors have detected an excess current through the motor leads. This can be related to excessive drag on the unit, such as improperly matched tailstocks or end supports, which creates excessive drag. If you do not have this type of set up contact the service department.

**IPKB\_ERR:** This is identical to IPKA\_ERR only it refers to the B axis.

**PWMA\_ERR:** The system has detected a short circuit in the output. Check motor wires to each other and ground. If the error continues even when disconnected from the motor cables, there is an internal problem with the drives. Contact the service dept.

**PWMB\_ERR:** Same a PWMA only refers to axis B.

**IGBTA\_ERR:** This is an excess heat warning. Be sure the airflow for cooling fans is not obstructed. Clean or replace filters. Make sure ambient temperature does not exceed 45c.

**IGBTB\_ERR:** Same as IGBTA only this refers to axis B.

**BUSHI\_ERR:** The bus voltage has exceeded a factory pre-set limits. The incoming voltage is too high. If this is not the case contact the service dept.

**BUSLO\_ERR:** The bus voltage has dropped below the factory pre-set limit. The incoming voltage is too low. If this error tends to come and go check your 110/220 volt power to see if it properly connected. If you are using an extension cord this can be a common result. If you are not using a dedicated circuit your voltage may be fluctuating and dropping below the required 16-30Vdc.

**Credit policy**

1. All returned goods **must have prior authorization.** The MRA (Merchandise Return Authorization) number must appear prominently on the shipping label. **Please refer to your original invoice number when requesting MRA's. Merchandise sent in without an MRA # will be refused by our shipping department. All returns must be sent back pre-paid.**
2. Defective goods will be replaced, repaired, or credited to you after the item has been inspected by our Service Department; unless prior authorization has been made.
3. Merchandise will not receive if: **It is no longer under warranty, has been misused, or has been altered or damaged by end user.**
4. If a unit is deemed to be defective & not repairable, credit will be issued. **Should part(s) be missing from the returned unit(s), the part(s) price will be deducted from the customer's credit.**
5. Any warranty merchandise that is repairable, will be repaired & sent back to the customer, therefore, no credit will be issued.
6. There will be a \$150.00 restocking fee on any & all items not returned in their original packaging &/or not covered under warranty.
7. No other policy, other than the foregoing. Will be authorized or accepted by Yuasa International.

## **Service policy**

1. Warranty cards must be on file within 30 days of invoice date for a warranty repair. If the warranty card is not on file, proof of purchase must be submitted when requesting service. If neither is supplied, **all repairs will be charged to the customer.**
2. There will be a minimum charge of \$150.00 for handling & inspection fee on all items sent in for inspection. If no defect is found {warranty} or if customer declines service on unit(s) {non-warranty} all such shipments will be sent back "**Freight Collect.**"
3. Units must not be previously altered, repaired, or serviced by anyone other than an Authorized Yuasa service Center. **Any & all misuse of an item will void warranty.**
4. Before any item will be repaired, the customer will receive a written estimate which must be returned within 15 days, signed: Approve or Disapprove repair. **If after 30 days we have not received authorization either way, there will be a \$150.00 fee charged to the customer & merchandise will be sent back "freight Collect".**
5. All repairs performed at the Yuasa International service Center: Illinois carry a 30 day warranty; warranty limited to only the exact items repaired or replaced.
6. **All merchandise must be sent in prepaid.** All repairs covered as warranty, will have freight charges credited back to their account.