CNC Programming Manual

Bosch CC 100 M

.

www.DaltonsWadkin.com

INTRODUCTION TO CNC PROGRAMMING

X and Y Axes



WHAT IS A POINT?



- 2 -

www.DaltonsWadkin.com

On the graph below plot the point and draw lines between the points to describe the shape:-



Now write down the intersection points for the shape below, decide which direction to plot the shape and follow round in a logical order from X0 Y0.



This is the basis of the CNC PROGRAMMING CO-ORDINATED SYSTEM. The table of the router is measured just like a graph but with a much finer spacing (resolution).

The resolution of the WADKIN ROUTER is 0.01 mm or 1/1000 inch depending which units you are using.

NOTE 1

Whether a program for a component is written clockwise or anti-clockwise around the shape all depends on the type of material to be cut, the type of cutter being used and the finish required.

- 3 -

www.DaltonsWadkin.com

STRAIGHT LINES

All straight line moves need to be preceded by a Gl code

To describe this component shape (with the graph taken off) with all straight line moves could be as follows:-



When working in INCHES the feed is in units of 1/10 inch/minute. (Maximum programmable is 4000 1/10 inch/minute (400 inches/minute)).

PROBLEM 1





07/92

ł

ຫ

1

www.DaltonsWadkin.com

CURVES

All curves need to be preceded by a code to tell the direction of rotation.

G2 - CLOCKWISE CIRCULAR MOVEMENT.

G3 - ANTI-CLOCKWISE CIRCULAR MOVEMENT.

The G2 or G3 is determined by the direction of rotation from the start point of a curve to the end point of a curve.



07/92

<u>Rule 2</u>

If the angular movement of the curve is LESS than 180 degrees then the radius is programmed as a negative 'R' value.

eg.







X550 Y600: End point of arc R200: Curve of 180 degrees, so positive radius value

PROBLEM 2



. •

1

- 9 -



CUTTER COMPENSATION

Consider this example:-



To enable us to still program the component as previous, but still get a true sized component we use CUTTER COMPENSATION.

This compensates for the size of the cutter, ie. moves away from the programmed path a specified amount usually 1/2 the diameter of the cutter (the radius).

This radius value is stored in the control when the component is ready to be cut or when the radius of the cutter is known, and it is stored in the TOOL STORE in memory under the R= column, which is accessed as follows:-

- 1. MEMORY
- 2. TOOLS
- 3. ACCESS ON
- 4. EDIT
- 5. Select required tool store number.

There are 48 tool stores available, which is used is left up to the programmer. Each tool store consists of four columns:-

Eq. T1 R= DR= L= S=

R= Radius
DR= Wear factor (not applicable)
L= Tool length (for Z axis)
S= Spindle speed (not applicable)

To call a tool store up into a program it must be with two digits ie. T01, T09, T15.

RULE FOR ACTIVATING CUTTER COMPENSATION

w.Do



Cutting CLOCKWISE round the OUTSIDE of the component use G41. (To the left hand side of the workpiece)

Cutting ANTI-CLOCKWISE round the OUTSIDE of the component use G42. (To the right hand side of the workpiece).

of the component use G42. (To the right hand side of the workpiece).

Cutting ANTI-CLOCKWISE round the INSIDE of the component use G41. (To the left hand side of the workpiece).

G41 - CUTTER COMPENSATION TO THE LEFT HAND SIDE OF THE WORKPIECE.
 G42 - CUTTER COMPENSATION TO THE RIGHT HAND SIDE OF THE WORKPIECE.
 G40 - CANCEL CUTTER COMPENSATION.

G41/G42 and G40 must be activated by a STRAIGHT LINE MOVE.



ie. Tl DR=0 L=0 S=0 R=7

The distance of the straight line move which activates and cancels cutter compensation (point 1 to 3 and point 4 to 6) must be more than the distance of the radius of the cutter being used

NOTE 3

It is good machining practice to activate and cancel cutter compensation away from the workpiece and also to feed on to and off the component in a straight a line as possible.

Example 1



T7 R= 11 DR=0 L=0 S=0

Example 2



Example Tool Store No. 3

T3 R=7 DR=0 L=0 S0





COMPONENT DATUM POINTS (MOVING ZERO POINT)

We have used the bottom left hand corner of the workpiece as the component datum (ie. all dimensions are relative to this point, X0 Y0). Which in turn would be relative to the bottom left hand corner of the machine table.

In practice the component would be situated somewhere on the machine table, thus:-



The component datum has a distance from the bottom left hand corner of the machine table (Machine reference point). In this case it is 50 along the X axis and 30 up in the Y axis.

If there wasn't the 'moving zero point' facility then the program would have to have 50 added on to every X dimension and 30 on to every Y dimension.

eg. G1 X50 ¥30	<pre>(Previously X0 Y0)</pre>
G1 X350 ¥30	(Previously X300 Y0)
G2 X550 ¥30 R100	(Previously X500 Y0)
G1 X750 ¥30	(Previously X700 Y0)
G1 X750 ¥330	(Previously X700 Y300)
G1 X550 ¥530	(Previously X500 Y500)
G1 X50 ¥530	(Previously X0 Y500)
G1 X50 ¥30	(Previously X0 Y0)

What we need to do is set the bottom left hand corner of the workpiece (or desired co-ordinates) to equal X0 Y0 when on the machine table.

To find the co-ordinates for a component zero point :-

- Firstly the component must be securely clamped to the machine (1)table in the position ready for cutting, and a stylus inserted into the router head (the left hand head on multi-head machines).
- (2) Go into MANUAL MODE, and MDI and key in G53 T00 and press CYCLE START. This will cancel any zero points still active so giving a true machine reading.
- (3) If a multi-head machine lower the left hand head off its ram, this is usually done via the code M8. It can be activated via MDI (key the code in and press cycle start) or via the MTBbuttons, if programmed.
- Now via the manual panel lower the 'Z' axis using the jog (4)buttons or the handwheel to a position above the workpiece.
- Again via the manual panel, drive the router head over the (5) position where the component zero point is required. Note the X and Y readings displayed on the CRT display, these are the values to be inserted into the program to register the position as component X0 Y0.

The main way setting a component zero point is via the programming of Variable Offsets.

USE OF VARIABLE OFFSETS



G90 - Absolute programming, where all dimensional values are relative a fixed (absolute) zero position, ie. X0 Y0

- VX= VY= Two variable stores, which are allocated with the values of the X and Y component zero positions for further reference.
- Is a cycle set be Wadkin, which is stored in the control, G8 calls up a cycle and '20' is the cycle number. A cycle is G820 used for a sequence of codes or moves which are to be used repeatedly.

Cycle 20 automatically loads the values of VX and VY into the Zero Shift store 54, under the X and Y columns, and so sets the values to equal X0 Y0 each time.

Part of a typical Cycle 20 would be as follows:-(This is for information only)

Ν	1	G53	Cancels any previous offsets.
N	2	TRF = G54 X = VX Y = VY	Transfers the values of VX and VY into Zero
			shirt 54 into the x and Y columns
			respectively.
Ν	3	G54	Call up Zero shift 54 and so sets the
			values in X and Y to equal X0 Y0
N	4	M2	End of cycle

PROBLEM 8





.

.

PROBLEM 11



23 1

www.DaltonsWadkin.com

PROBLEM 13



07/92

.

- 24 -

www.DaltonsWadkin.com

TOOL LENGTH COMPENSATION



When a program has been written it will contain at least one tool number (eg. T01), which is used to activate the tool length compensation and/or cutter compensation. Before the program can be executed a datum must be set for the tool/tools being used in the appropriate tool store number.

The following procedure must be followed for the tool length to be activated accurately.

- 1. Secure the desired tool(s) in the router/drill head(s).
- 2. Make sure the particular workpiece is vacuumed/clamped down.
- 3. If necessary lower the head/drill down on its pneumatic slide (this is done by the use of a 'M' code, see machine list)
- 4. Wind the head down, using the handwheel on the manual panel, until the tool is just touching the top of the workpiece.
- 5. On the screen will be a value for the tool length displayed as a Z axis reading.
- To ensure that this is the correct figure, select M.D.I. and key in the following commands pressing cycle start after each:-

G53 T00 G92

The correct figure will now be displayed as all offsets have been cancelled.

- 7. Activate the following keys to store the tool length value:
 - a) Page out once, (green round key)
 - b) Memory mode
 - c) Tool store, ((far left softkey)
 - d) Access on, (far left softkey)
 - e) Edit, (middle softkey)
- 8. This brings up the tool store page with a list of tools from 1 to 48. There are four columns for each tool store.

ie. T1 R=0 DR=0 L=0 S=0

R = Tool radius DR = Wear factor (seldom used) L = Tool length S = Spindle speed (seldom used)

To insert the tool length value, select the appropriate tool store number, as used in the program, using the arrow keys.

- 9. Move the cursor across to the 'L' column and key in the value from the Z axis reading from machine mode, and then press enter. Make sure it is a negative (-) value. (If a positive value is entered then the program will error as the Z axis will try to go upwards instead of downwards when the program is executed).
- 10. Now page out and go back to Machine mode.
- 11. If the head/drill has been lowered via an 'M' code use the opposite code to retract the head/drill (see machine list)
- 12. To move the Z axis back to its top position, select M.D.I and key in G0 Z0 T00 and press cycle start.

The whole procedure should be repeated for all the tools/heads being used.

<u>Note</u>

When setting a tool length for a drill, key in M61 in MDI (Machine mode) and press cycle start, this will stop the drill from rotating when it is in its down position. This is cancelled by M62.

GO - RAPID TRAVERSE

GO - is used for mid-air moves to reduce cycle time. It is an alternative to G1, but it defaults to a set feedspeed rather than a defined feedspeed, as in G1 F5000.

Rapid traverse speeds, X & Y - 12 m/min Z - 6 m/min.

Rapid traverse is overridden by the feedspeed override key.

UNCONDITIONAL JUMP

If a machine is fitted with AC drives (inverter) a signal to the heads is cut out when M30 is read and this then stops the router head (s). To eliminate this stop an unconditional jump using G24 has to be inserted into the program as follows:-

.			
Programs	s snould	start	\$99
			MO
			(Press cycle start to continue)
			(11000 C)OLO DUALE DO CONCLINAC,
			G53
			GO ZO TOO
			VX = VY =
			G820
			elov
mba line		to M20 should	
The True	prior	co M30 should	
now read	l		G24 P99
			M30
		0 line is mood	the immediately immed to the

When the G24 P99 line is read it immediately jumps to the corresponding \$ sign (\$99) at the start of the program, the program continues to be read, but M0 is program stop and will hold until cycle start is pressed to continue. Thus giving exactly the same function as M30.

Leave M30 in the program, although it is not used for programming it does signify the end of the program.

EXAMPLE FOR A UX2F/C3

.

\$99 MO G90 G53 G 0 Z0 T00	<pre>\$ sign for return to start of program. Program stop, press cycle start to continue Absolute programming Cancel all offsets Cancel tool store and retract Z to Machine Z0</pre>
VX=50 VY=50	
G820	Set Component datum point
M8	Lower left hand head
GO X-15 YO 25 TO9	Rapid traverse to start position, bringing cutter to 5 mm above workpiece using tool length offset in tool 9.
G1 F2500	
G42 X-5 Y0	Call up cutter compensation, activate by moving to X0 Y0
G1 Z-10 F2000	Plunge cutter into workpiece 10 mm.
G1 X50 Y0 F5000 G1 X50 Y50 G1 X0 Y50 G1 X0 Y-5	com
G1 Z5	Retract cutter to 5 mm above workpiece
G40 X0 Y-15	Cancel cutter compensation
M9	Retract left hand head
G53	Cancel offsets
GO ¥600 ZO TOO	Off load position to Y600 and cancel tool store retracting Z to Machine Z0.
G24 P99	Unconditional jump to \$99
M30	End of program

NB:- With other machine head configurations the 'M' codes may change, please refer to the individual M code list for each machine.

.

EXAMPLE FOR A SINGLE HEAD ROUTER

\$99 M0 G90 G53 G0 Z0 T00 VX=50 VY=50	<pre>\$ sign to return to start of program Program stop, press cycle start to continue Absolute programming Cancel all offsets Cancel tool store and retract 2 to Machine 20</pre>
G820 G0 X-15 Y0 <mark>Z5 T09</mark>	Set Component datum point Rapid traverse to start position, bringing cutter to 5 mm above workpiece using tool length offset in tool 9.
G1 F2500	Set straight line move and feedspeed for G42
G42 X0 Y0	Set cutter compensation to datum point
G1 Z-10 F2000	Plunge cutter into workpiece 10 mm.
G1 X50 Y0 F5000	
G1 X50 Y50	
G1 X0 Y50	
G1 X0 Y-5	
G1 Z50	Retract cutter to 5 mm above workpiece
G40 X0 Y-15	Cancel cutter compensation
G53	Cancel OIISets
GU 1600 20 100	off foad position to food and cancel tool
C24 D00	Unconditional jump to S99
G24 F33	End of program
M20	
	130.
PROBLEM 14	
	XO.
C	
NN	
400	200
150	150
75	
300	300
50 ¦	

.

SUB-PROGRAMS

A sub-program is used when information needs to be repeated within one program. It may be used to cut a number of components in one run of a program, or where a certain sequence needs to be repeated at different positions, or just to clarify different parts of a program.

Sub-programs are entered directly under the main program (after M30) and can only be used within the program they are written for.

- G22 Calls up a sub-program
- P? Is the number of the sub-program, and can be anything from 1 to 99.
 - (eg. G22 P1 will automatically call up and activate subprogram 1 once.)
- L? Is the repetition factor (after the initial run of the subprogram). A sub-program can have a L' factor of upto 99.
- \$? Can be the start of a sub-program, and must correspond with the P value from the call up of the sub-program (1 to 99).
- G99 Is the end of a sub-program.

Mary!

VARIABLES

VA to VZ and V1 to V99

Are allocation codes to which a value can be assigned for use within a program, they can be used to do mathematical calculations on values.

www.DaltonsWadkin.com

ABSOLUTE SUB-PROGRAMS



\$99	\$ sign for return to start of program
MO	Program stop, press cycle start to continue
G90	Absolute programming
G53	Cancel offsets
GO ZO TOO	Cancel tool store, retract Z to machine ZO
VX=50 VY=50	Component zero point for first component
G820	
MB	Lower left hand head
G22 P1 L2	Call up sub-program 1 and repeat 2 times.
М9	Raise left hand head
G53	Cancel offsets
GO ¥600 ZO TOO	Off load position, retract Z to machine ZO.
G24 P99	Unconditional jump to \$99
M30	End of main program
\$1	Start of sub-program 1.
GO X-15 YO Z5 T16	Rapid traverse to start of component, 5 mm above the workpiece using offset in tool 16.
G1 F3000	Set feedspeed for cutter compensation move
G42 X-5 YO	Cutter compensation
G1 Z-10 F2000	Plunge cutter into workpiece 10 mm.
G1 X250 Y0 F5000	
G1 X250 Y250	
G1 X0 Y250	
G1 X0 Y-5	
G1 Z5	Retract cutter 5 mm above workpiece
G40 X0 Y-15	Cancel cutter compensation
VI=270	Distance between component zero points
VX=VX+VI ·VY=VY	Add the value of V50 to the current VX value, VY remains as the last stated VY
G820	Will zero the new VX and VY position
G99	End of Sub-program

- 31 -

PROBLEM 15

• --



PROGRAM FOR MULTI-ROWS USING SUB-PROGRAMS

\$99 MO G90 G53 G0 Z0 T00	<pre>\$ sign for return to start of program Program stop, press return to continue Absolute programming Cancel all offsets Cancel all tool stores, retract to machine Z0</pre>
VX=50 VY=50	Component zero position for first component first row.
G820 M8 G22 P1 L2	Set component zero point Lower left hand router head Call up sub-program 1 and repeat 3 times.
VX=50 VY=350	Component zero position for first component second row.
G820 G22 P1 L2	Set component zero point Call up sub-program 1 and repeat 3 times.
VX=50 VY=650	Component zero position for first component third row.
G820	Set component zero point
G22 P1 L2	Call up sub-program 1 and repeat 3 times.
VX=50 VY=950	Component zero position for first component fourth row.
G820	Set component zero point
G22 P1 L2	Call up sub-program 1 and repeat 3 times
M9	Retract left nand nead
G53	Cancel all offsets
GO YI300 ZU TUU	Unconditional jump to \$99
G24 P95 M30	End of main program
\$1	
GO X-15 YO Z5 T10	5
G1 F3000	$\bigcap_{i=1}^{n}$
G42 X-5 Y0	\mathbf{V}
G1 Z-10 F2000	
G1 X250 Y0 F5000	*
G1 X0 Y250	
G1 X0 Y-5	
G1 Z5	
G40 X0 Y-15	
VSU=270 VX=VX+V50 VY=VY	
G820 G99	

.

•

MULTI-ROW USING SUB-PROGRAMS



•

www.DaltonsWadkin.com

.



,

INCREMENTAL PROGRAMMING - G91

G91 - All dimensional values relate to the respective previous positions. It is advisable to use G91 for contours which need to be machined repeatedly in different places.

It is advisable to keep any Z axis moves in absolute (G90).



www.DaltonsWadkin.com







CYCLES

There are 69 cycles available. A cycle is activated via the code G8 plus the cycle number, eg. G820.

Cycle store has exactly the same format as program store, the difference is that you can call any cycle up into any program via the above code.

Cycles are used where parts of programs, or sequences of codes are repeated either in different programs or different parts of the control, ie. in machine mode.

Certain cycles are reserved, these are generally cycles 20, 21, 22, 23, 78 and 79. Do not delete or alter any of these cycles without prior instructions from Wadkin.

Cycles 20, 21, 22, 23 are used for the offsets for the router heads and drills (which will be in individual machines depends on the specification of the machine, please check).

Cycle 79 is known as the 'Reference cycle' and is activated indirectly. In Machine mode the second option is 'Ref. cycle' (it is activated after the machine and drives are switched on to reference the machine), when the key is pressed the control activates cycle 79 in memory automatically.

Cycle 78 is known as the MTB Buttons cycle. It relates to the 10 function keys on the manual panel (F1 to F10). To check the function of each of these keys go into Memory mode and cycles and call up cycle 78 and enter. The cycle displays ten lines of programming whatever is written on each line is what is activated via the equivalent function key.

ie. If N 1 reads M8, then function key F1 will lower the left hand router head.

If N 10 reads M10, then function key F10 will stop the current running router head.

These lines can be edited to read whatever the function key is required to do.

All other cycles can be used by the customer.

EXAMPLE CYCLE FOR CIRCLE



PROGRAM USING A SINGLE ROUTER HEAD MACHINE

Single router head is used to cut panel and circle.

\$99 \$ sign for return to start of program MO Program stop, press cycle start to continue G90 Absolute programming Cancel all offsets G53 GO ZO TOO Cancel active tool store, retract to m/c Z0 VX=50 VY=50 Component zero position G820 Set component zero point GO X-15 YO Z5 T07 Start position, move to 5 mm above workpiece using tool store 7 G1 Z-11 F2000 Plunge cutter into workpiece 11 mm G42 X-5 YO Set cutter comp. to left of workpiece G1 X1000 Y0 F5000 G1 X1000 Y1000 G1 Y1000 XO Y-5Gl X0 Retract cutter 5 mm above workpiece G1 $\mathbf{Z5}$ G40 X0 Y-15 Move to centre position of circle. Call up cycle 11 GO X500 ¥500 G811 Cancel all offsets G53 Off load position, cancel tool length GO Y1300 ZO TOO Unconditional jump to \$99 G24 P99 End of main program M30 To produce further circles on the same component, all that would be needed to be inserted would be :-GO X500 Y750 (Centre of next circle) G811

This would be after the first G811.

- 40 -

PROGRAM USING A TWIN ROUTER HEAD MACHINE

Left hand router head is used to cut panel and right hand router head is used to cut the circle.

\$99 \$ sign for return to start of program MO Program stop, press cycle start to continue G90 Absolute programming Cancel all offsets G53 GO ZO TOO Cancel active tool store, retract to m/c ZO Component zero position VX=50 VY=50 Set component zero point G820 Lower left hand head MÅ Start position, move to 5 mm above GO X-15 YO Z5 T07 workpiece using tool store 7 Plunge cutter into workpiece 11 mm G1 Z-11 F2000 Set cutter comp. to left of workpiece G42 X-5 Y0 G1 X1000 YO F5000 G1 X1000 Y1000 G1 X0 Y1000 G1 X0 Y-5 Retract cutter 5 mm above workpiece Cancel cutter compensation G1 $\mathbf{Z5}$ G40 X0 Y-15 Retract left hand head Switch active zero point to right hand head M9 G823 GO X500 Y500 Z5 T08 Move to centre position of circle, moving 5 mm above workpiece using tool store 8. Lower right hand router head M68 Call up cycle 11 G811 M69 Retract right hand router head Cancel all offsets G53 Off load position, cancel tool length G0 Y1300 Z0 T00 Unconditional jump to \$99 G24 P99 End of main program M3 0 To produce further circles on the same component, all that would be needed to be inserted would be :-

G0 X500 Y750 (Centre of next circle) G811

This would be after the first G811.



DRILLING

The drills are set up for a program in the same way as for the router heads, that is the tool length as to be set for each of the drill heads to be used in a program. Each should be allocated a tool number and the tool length offset is stored in the 'L' column of the appropriate tool store. (The radius column will be left blank).

To set the tool length for a drill head, drill rotation has to be switch off, this is done via the code 'M61' programmed via Machine mode, MDI. To switch drill rotation back on 'M62' should be programmed.

.

Example program for drilling

\$99 MO G90 G53 G0 Z VX=5	0 TOO 0 VY=	50	<pre>\$ sign for return to start of program Program stop, press cycle start to continue Absolute programming Cancel all offsets Cancel active tool store, retract to m/c Z0 Component zero point relative to left hand router head</pre>
G820 M8 G0	X-15	YO Z5 T04	Set component zero point Lower left hand router head Start position, activate tool length
G1 F G42 G1	'3000 X-5 Z-11	Y0 F2000	compensation using tool store 4. Set feedspeed for cutter comp. move Activate cutter comp. to left of workpiece Plunge cutter in to workpiece 11 mm.
G1 G1 G1	X1000 X1000 X0	Y0 F5000 Y1000 Y1000	<i>toll</i>
G1 G40 M9	Z10 X0	Y-15	Retract cutter 10 mm above workpiece Cancel cutter compensation Retract left hand router head
G821	-	- Ar	Switch active zero point relative to left hand drill (G822 if right drill)
G0	X200	¥200 25 T05	Rapid to first hole position activating tool length offset using tool store 5
M20 G1	F2000		Lower left hand drill (M66 for right drill) Set G1 and feedspeed for Z axis movement
G81 G61	V1=5	V2=-11	In position logic on
GO	X200 X800 X800	¥200 ¥200 ¥800	when drilling is required.
G62 G80 M38 G53	X200	1800	In position logic off Cancels drilling cycle. Retract lowered drill head Cancel all offsets
GO	¥1300	Z0 T00	Off-load position, cancel tool length retract to machine Z0
G24 M30	P99		Unconditional jump to \$99 End of program

4

G81 - Calls up fixed drilling cycle.

V1 - Retract out in rapid to height above workpiece, ie. 5 mm V2 - Depth to drill down to at state feedspeed, ie. -11 mm Do not alter the variable value, they must read V1 and V2.



TEACH IN

Teach In is used to produce co-ordinates for a drawing/component which has limited or no dimensions avaialable to program from.

- 1. Securely clamp the job to the machine table (if a drawing tape to a board and then clamp).
- 2. Secure the pointer/stylus into the left hand router head.
- 3. Enter into Machine mode.
- 4. If a multi-head machine is being used lower the left hand router head using either the function buttons (if appropriate) or go into MDI and key in M8 and press cycle start.
- 5. In M.D.I. key in G53 T00 and press CYCLE START and then key in G92 and press CYCLE START. This will clear any offsets still active.
- 6. Lower the Z axis until the pointer is just above the workpiece.
- 7. If a component zero point is required (all co-ordinates taught in would be in relation to the point, as in VX and VY in a program) move the pointer to the position to be set as zero and make a note of the co-ordinates on the screen.
- 8. In MDI key in G92 X0 Y0 and press CYCLE START the control should now read X0 Y0 on the screen.
- 9. Page out.
- 10. Enter into TEACH IN.
- 11. If there are lines of programming between the Z axis display and the edit line, this means that a program called TEACH IN already exists and if you were to continue with the new Teach In it would be added to the end of the existing one. To clear the Teach In option the existing Teach In should be renamed.
 - a) Page out
 - b) Memory
 - c) Programs
 - d) Key in TEACH IN (or find the correct program number)
 - e) Enter
 - f) Command
 - g) Rename
 - h) Type in the program name for the existing teach in.
 - i) Enter
 - j) Page out, and return to Machine Mode and the TEACH IN option.
- 12. Key in a start line for the Teach In ie. GO XO YO. (In classroom G1 XO YO F6000)
- 13. You are now going to manually move the pointer around the component/drawing.

- 45 -

- 14. If a straight line is required, move directly to the end of the straight line and press RECORD. This will record on the screen the position of the end point.
- 15. If a curve is required, without moving from the end point of the last position press CIRCLE COMPUTE and then:-

press RECORD POINT 1 - this will record the start of the curve.

now move to a mid-point on the curve and press RECORD POINT 2

finally move to the end point of the curve and press RECORD POINT 3

- 16. Now the codes for the programming of the curve will be displayed on the edit line, <u>PRESS RECORD AGAIN</u> to record the line into memory.
- 17. If a straight line is required after a curve, before moving from the end of the curve, key in G1 and press record. If further straight line moves are then recorded there is no need to program another G1.
- 18. Continue with the above procedures for the whole of the component/drawing.
- 19. When the shape is complete, page out of Teach In, the control will automatically record a 'M2' at the end of the program.
- 20. If the shape needs testing, go into Automatic Mode and call up the program called TEACH IN jeither via name or program number)
- 21. Run the program round in either Normal or Rapid, the pointer should remain just above the workpiece and the shape can be followed easily using the STEP function (press STEP, it will flash '1', press enter) this will step the program one block at at time, press cycle start to read the next block.
- 22. When the test is complete return to Machine Mode and if a component zero point was set, this must be cancelled.
- 23. Go into MDI and key in G92 and press CYCLE START this will re-reference the position of the pointer relative to machine zero.
- 24. To retract the router head key in GO ZO TOO this will retract the head to machine ZO.
- 25. If a twin head machine and the left hand router head was lowered to retract the head use either the appropriate function key or key in MDI M9 and press cycle start.
- 26. Now go into Memory Mode and you can edit the program (which will be called TEACH IN) so it will run normally on the machine.
- 27. If it is the end of the Teach In, the program can also be renamed to allow a further Teach In to be carried out.

NOTE 3

TEACH IN can also be used in a similar way to MDI, that is any codes can be keyed in and activated manually and not recorded. The code is keyed into the edit line and activated via cycle start and then is cleared using the CLEAR BLOCK softkey. This does not interrupt the on-going Teach IN.

Circle Compute is pressed and then the table moved via the eg. manual panal before RECORD POINT 1 is pressed. The Teach In will be incorrect if point one is not recorded at the correct position.

Point one is the same position as the last recorded co-ordinates displayed on the screen, so key in GO X? Y? (the last displayed line above the edit line) and press CYCLE START. The pointer should now move back over point one.

Press the far right softkey CLEAR BLOCK the command should disappear from the edit line and now RECORD POINT 1 can be pressed and the Teach In continued with as normal.

with with with WMM. Dations Wadkin

SCALING

To activate scaling the code is G36 X or G36 Y. Although only one axis is defined the opposite axis will automatically be scaled at the same rate.

The scaling range is 0.00001 - 99.00 and must be defined in decimal format.

G36 X.5 (will scale by half) eq. G36 X.2 (will scale by a fifth)

To de-activate scaling use G36 X1, this puts scaling back on a factor of 1:1

Scaling works around the active zero point.

Scaling does affect variables, ie. VX and VY, if activated prior to the offset the zero point will be scaled.

Scaling does not affect the Z axis or the tool stores.

Scaling must be activated prior to cutter compensation (G41 and G42) and must be de-activated after cutter compensation is cancelled (G40).

> Original Component

Scaled

Component



07/92

ł

100



The code to activate mirroring is G38 plus the axis you wish to mirror in.

- eg. G38 X reverses all the X axis signs.
 - G38 XY reverses all the X and Y axes signs.
 - G38 Y reverses all the Y axis signs.

Each of the above codes cancels the other out, but to cancel mirroring completely the code is G39.

The control automatically converts G41 to G42 and G2 to G3 and vise versa.

Mirroring does affect the variables within a program.

G820, G821, G822, G823 and any other offset cycles can not be programmed with mirroring active, G39 must be programmed prior to the code and then mirroring can be re-activated after the code.

EXAMPLE MIRRORING PROGRAM



\$99	\$ sign for return to start of program
MO	Program stop, press return to continue
G90	Absolute programming
G53	Cancel all offsets
GO ZO TOO	Cancel active tool store, retract to m/c 20
VX=500 VY=500	Component zero position
G820	Set component zero point
M8	Lower left hand router head
G22 P1	Call up sub-program 1, activate once
VX=460 VY=500	
G820	
G38 X	Mirror everything in the X axis
G22 P1	Call up sub-program 1, activate once
G39	Cancel mirroring
M9	Retract left hand router head
G53	Cancel all offsets
GO Y1000 ZO TOO	Off load position, cancel tool length
G24 P99	Unconditional jump to \$99
M30	End of main program
\$1	Start of sub-program 1
G0 X-15 Y0 Z5 T11	Move to start position, set tool length
G1 F3000	
G42 X-5 Y0	Set cutter compensation
G1 Z-10 F2000	Plunge cutter 10 mm into workpiece
G1 X300 Y0 F6000	
G3 X320 Y20 R-20	
G1 X320 ¥320	
G1 X0 Y320	
G1 X0 Y-5	
G1 Z10	Retract cutter 10 mm above workpiece
G40 X0 Y-15	Cancel cutter compensation
G99	End of sub-program

.

IN-POSITION LOGIC & CONTOUR TRANSITIONS

- G61 In-position logic ON
- G62 In-position logic OFF (active on switch on)
- G61 -With G1,2 and 3 the control waits for the cutter to be in position before readin the next block. Once the position has been reached the control stops for a short tiem, this is determined by a machine parameter.
- When the In-position function is switched off the control G62 reads the next block whilst the cutter is actioning the previous block. This results in a 'cutting of corners', but saves time.
- G61 is programmed prior to movements required to be in position and is cancelled with G62. 130Kin.com
- G61 eg. X500 Y200 G1
- or G61 X500

G62 to cancel.

- Arc (active on switch on) G68 G69 - Intersection

If cutter compensation is active the control must create transitions for outside corners. These transitions can either be intersections or automatically generated arcs.



G69 is used in conjunction with G61 and is cancelled by G68.

G1 X0 Y0 eq. G61 G69 X300 Y0 X300 Y50 **Y50** X0 G62 G68 X0 Y-5

- 52 -

MISCELLANEOUS

MO - PROGRAM STOP (feedhold condition)

Is used when a stop is required in a program for an uncertain length of time, operator controlled. To continue the program cycle start must be activated. It can be used to change clamping during the execution of a program, ie. turning over the component.

() - Anything entered in brackets in a program is classed as information only, the control will ignore these lines.

Information could be entered at the beginning of the program to ease the job of the operator, i.e. tooling information - (T1 = RIGHT HEAD T2 = LEFT DRILL)

or when used in conjunction with M0 the line below the M0 will be displayed during the feedhold time. ie. M0

(TURN COMPONENT, PRESS CYCLE START TO CONTINUE)

G4 F - Dwell

The machine will dwell for 'X' number of seconds, the range is from 0.01 to 9 999 999 seconds. The number of seconds must be in decimal format.

ie. G4 F10 will dwell for 10 seconds G4 F.5 will dwell for 1/2 second

SHORTENING PROGRAM

All codes in a program will remain modal until cancelled by an equivalent code. So there is no need to repeat information which remains the same on a following line.

ie.	Gl	X0	Y0	F5000	G1 X0	¥0	F5000
	G1	X100	YO		X100		
	Gl	X100	¥50		¥50		
	Gl	XO	¥50		X0		
	G1	XO	YO		Ϋ́O		

Example shortened program:-

GO X-10 YO Z5 T01 Gl F5000 G42 X0 YO G1 Z-10 F2000 X500 Y0 F3000 G1 X500 Y300 G1 X500 Y500 R100 G2 G1 X500 ¥550 XO G1 ¥550 G1 X0 Y-5 GO Z10 G40 X0 Y-15

G0 X-10 Y0 Z5 T01 G1 F5000 G42 X0 Z-10 F2000 X500 F3000 Y300 G2 Y500 R100 G1 Y550 X0 Y-5 G0 Z10 G40 Y-15

www.Dationswadkin.com

LIST OF 'G' CODES

- G0 - RAPID TRAVERSE TO POSITION G1 - MOVE TO POSITION IN A STRAIGHT LINE G2 - CLOCKWISE CIRCULAR MOVE - ANTI-CLOCKWISE CIRCULAR MOVE G3 - PROGRAMMED DWELL (WITH 'F' IN SECONDS) G4 G17 - PLANE SELECTION X.Y G18 - PLANE SELECTION X,Z G1.9 - PLANE SELECTION Y,Z G20 - SETTING POLE FOR POLAR CO-ORDINATES G22 - UNCONDITIONAL CALL UP OF SUB-PROGRAM G24 - UNCONDITIONAL JUMP TO JUMP TARGET (\$?) - ACTIVATE SCALING G36 G38 - SWITCH ON PROGRAMMED MIRROR IMAGE G39 - SWITCH OFF MIRRORING G40 - CANCEL CUTTER COMPENSATION G41 - CUTTER COMPENSATION TO THE LEFT OF THE WORKPIECE G42 - CUTTER COMPENSATION TO THE RIGHT OF THE WORKPIECE - CANCEL ZERO SHIFT OFFSETS G53 G54 - G59 ZERO SHIFT OFFSETS - SWITCH ON 'IN-POSITION' LOGIC G61 - SWITCH OFF 'IN-POSITION' LOGIC G62 - DISABLE SPINDLE & FEEDRATE OVERIDE POTENTIOMETER G63 - ENABLE SPINDLE & FEEDRATE OVERIDE POTENTIOMETER G66 - AUXILLARY ARC ON OUTSIDE CORNERS G68 - INTERSECTION ON OUTSIDE CORNERS G69 - PROGRAMMED TRAVERSE TO REFERENCE G74 - CANCEL FIXED CYCLES G80 G81 - DRILLING CYCLE G90 - ABSOLUTE PROGRAMMING G91 - INCREMENTAL PROGRAMMING G92 - SET COMPONENT ZERO POINT
- G99 END OF SUB-PROGRAM

STANDARD 'M' CODE LIST

Please check individual machine specification for correct codes.

M0 M2 M30	PROGRAM STOP END OF MAIN PROGRAM END OF MAIN PROGRAM WITH RETURN TO START
M3 M5	SPINDLE ROTATION CLOCKWISE SPINDLE STOP
MS	LOWER HEAD 1
M9	RAISE HEAD 1
M68	LOWER HEAD 2
M69	RAISE HEAD 2
M73	LOWER HEAD 3
M74	RAISE HEAD 3
M90	LOWER HEAD 4
M91	RAISE HEAD 4
M75	LOWER HEADS 1 + 3
M76	RAISE HEADS 1 + 3
M84	LOWER HEADS 2 + 4
M85	RAISE HEADS 2 + 4
M78	LOWER ALL HEADS
M/9	RAISE ALL HEADS
M17	HEAD 1 ON
M11	HEAD 1 OFF
M18	HEAD 2 ON
M12	HEAD 2 OFF
M70	HEAD 3 ON
M15	HEAD 3 OFF
M83	HEAD 4 ON
M16	HEAD 4 OFF
M46	HEADS 1 + 3 ON
M4,7	HEADS 1 + 3 OFF
M48	HEADS 2 + 4 ON (UX M/C, GUARD 2 RAISE CC M/C)
M49	HEADS $2 + 4$ OFF (UX M/C, GUARD 2 LOWER CC M/C)
M65	ALL HEADS ON
M10	ALL HEADS OFF
M61	DRILL ROTATION OFF
M62	DRILL ROTATION ON
M20	LOWER DRILL 1
M66	LOWER DRILL 2
M28	LOWER DRILL 3 (ON UX M/C, GUARD 1 RAISE ON CC M/C)
M29	LOWER DRILL 4 (ON UX M/C, GUARD 1 LOWER ON CC M/C)
M80	LOWER DRILLS 1 + 3
M81	RAISE DRILLS 1 + 3
M31	LOWER DRILLS 2 + 4
M39	RAISE DRILLS 2 + 4
M38	RAISE ALL DRILLS

.

.

1

.

M50 M51 M52 M53 M54 M55 M56 M57 M58 M63 M71 M72	VACUUM PUMP 1 ON, RIGHT HAND SIDE VACUUM PUMP 1 OFF, RIGHT HAND SIDE VACUUM PUMP 2 ON, LEFT HAND SIDE VACUUM PUMP 2 OFF, LEFT HAND SIDE VACUUM TABLE CLAMP (ON), RIGHT HAND SIDE) VACUUM TABLE CLAMP (OFF), RIGHT HAND SIDE) VACUUM TABLE CLAMP (ON), LEFT HAND SIDE) VACUUM TABLE CLAMP (OFF), LEFT HAND SIDE)
M82	GUARDS 1 + 2 RAISE CC $M/C'S$ ONLY
M86	GUARDS 1 + 2 LOWER CC M/C'S ONLY
М9 З	M CODE TO SOTP VACUUM UNCLAMP DURING AUTOMATIC
M94	M CODE TO CANCEL M93
M67	CUSTOMER DEFINED USE, M67 ON
M2 3	CUSTOMER DEFINED USE, M23 ON
M24	CUSTOMER DEFINED USE, M24 ON
M25	CUSTOMER DEFINED USE, M25 ON
M26	CUSTOMER DEFINED USE, M26 ON
M27	CUSTOMER DEFINED USE, M27 ON
M32	CUSTOMER DEFINED USE, M67 OFF
M33	CUSTOMER DEFINED USE, M33 OFF
M34 M35	CUSTOMER DEFINED USE, M34 OFF
MJE	CUSTOMER DEFINED USE, M35 OFF
M37	CUSTOMER DEFINED USE, M37 OFF
110 /	
	www.Daitor

ERRORS

Errors are described with a three digit code, the first digit being the error group and the second and third digits being the error number.

In the edit option in memory the error will automatically displayed on error input.

In MDI in Machine mode the error will again be automatically displayed on incorrect input.

In Automatic mode when an error appears on a run, the error description can be found in Information mode, under MACHINE STATUS and MESSAGE LIST.

Whilst in Automatic mode if Information is pressed twice the error message list will automatically be displayed.

. /

These are the most common errors:-

ERROR MESSAGE GROUP 0:

DATA MEANING

1	syntax	error
2	syntax	error

- computer spelling error - computer spelling error

10	repetition (L) without subroutine call
11	cutter comp. programmed without tool
12	this G code must be alone in block
13	this G/M code is not allowed with TEACH IN or MDI
15	R or I, J, K not allowed
21	enter Dwell time (F)
22	unadmissible G number
23	enter S without sign
24	value too large
25	with D, F, or R, zero not allowed
26'	repetition of address not allowed
31	enter jump target (P)
32	jump target (P) allowed with G21/22/23/24
33	G code required with P or L
34	axis without value not allowed
35	only integer value
36	this M code must be alone in block
38	unadmissible tool number
39	E axis must be programmed alone
44	axis value not allowed
45	max. 2 digits with \$, P, L or M
46	only 2 or 4 digits with T
47	too many digits
48	max. 1 axis with value allowed
49	max. 4 digits with S
50	sign not allowed
52	enter value
68	E not allowed

í

- 58 -

82 unadmissible G number

.

-59 -

DATA MEANING

1	99 programs exist
2	memory full
3	memory too small for jump target table
4	checksum error
5	undefined jump target
6	parity memory
7	duplication of jump target
8	69 cycles exist
9	memory too small to copy
10	file protected
12	unadmissible file
13	file already exists
14	device not ready
15	parity error
16	incorrect data format
17	incorrect baud rate
23	reference cycle does not exist
24	interruption, abort with clear block
25	inch/metric selection incorrect
26	no corresponding cycle
27	undefined key
28	reference not allowed with length comp.
32	too many characters for one block
35	no machine reference, end axes to reference
39	circle calculation not possible
41 50	emergency stop
50	axis error
	G
	\sim°
	N.
	7

07/92

- 60 -

TO INSERT A PROGRAM INTO MEMORY

- 2. PROGRAMS
- 3. KEY IN NAME OF PROGRAM (UPTO 15 CHARACTERS)
- 4. ENTER
- 5. EDIT
- 6. KEY IN EACH LINE OF THE PROGRAM (NO SPACES ARE REQUIRED) AND PRESS ENTER AT THE END OF EACH LINE

TO RUN A PROGRAM (FOR USE IN THE CLASSROOM)

- 1. PAGE OUT OF MEMORY MODE
- 2. AUTOMATIC
- 3. KEY IN NAME OR NUMBER OF PROGRAM (IF NOT ALREADY THERE)
- 4. ENTER
- 5. PRESS THE FAR LEFT SOFTKEY TO ACTIVATE 'DRYRUN'
- 6. PRESS CYCLE START

IF NO ERROR OCCURS PROCEED TO NO.

IF 'ERROR' APPEARS IN THE TOP RIGHT OF THE SCREEN GO TO 'IF ERROR OCCURS ON DRYRUN' BELOW.

7. PRESS FAR LEFT SOFTKEY TWICE, TO ACTIVATE 'NORMAL', SET PLOTTER AND PRESS CYCLE START TO PLOT.

IF ERROR OCCURS ON DRYRUN

- 1. PRESS INFORMATION TWICE (MESSAGE LIST WILL APPEAR)
- 2. NOTE THE ERROR AND THE LINE NUMBER IT IS ON FROM THE SCREEN
- 3. REPRESS AUTOMATIC (THIS WILL STOP 'AUTOMATIC' FLASHING)
- 4. PAGE OUT
- 5. MEMORY
- 6. PROGRAMS
- 7. KEY IN NAME OR NUMBER OF PROGRAM TO EDIT (IF NOT ALREADY THERE)
- 8. ENTER
- 9. EDIT
- 10. CORRECT THE PROGRAM ERROR AND REPEAT 'TO RUN A PROGRAM'.

TO INSERT A NEW LINE INTO AN EXISTING PROGRAM

- 1. MEMORY
- 2. PROGRAMS
- 3. KEY IN NAME OR NUMBER OF PROGRAM (IF NOT ALREADY THERE)
- 4. ENTER
- 5. EDIT
- 6. SCROLL (HOLD SCROLL DOWN AND USE THE UP AND DOWN ARROW KEYS) TO THE LINE ABOVE WHERE THE NEW LINE IS REQUIRED.

IE. N 2 G1 X200 Y0 N 3 G1 X400 Y100 N 4 G1 X400 Y500

G1 X400 Y100

- 7. SELECT THE INSERT MODE, PRESS THE FAR LEFT SOFTKEY TO HIGHLIGT 'INSERT'.
- 8. KEY IN THE MISSING LINE IE. G2 X400 Y400 R150
- 9. ENTER
- 10. THE SCREEN WILL NOW APPEAR

N 2 G1 X400 Y0 N 3 G1 X400 Y100 N+3 G2 X400 Y400 R150 N 4 G1 X400 Y500

(Any number of lines can be inserted into an exisiting program, but they would all be a + number of they line they are inserted below.)

- 11. TO RESQUENCE THE THE LINE NUMBERS:
 - a) PAGE OUT
 - b) PRESS 'COMMAND' TWICE
 - c) RESEQUENCE This will put the block numbers in consectutive order.

12. PAGE OUT TWICE.

SEARCH

TO USE THIS OPTION YOU MUST BE IN:-

- 1. MEMORY
- 2. PROGRAMS
- 3. KEY IN NAME OR NUMBER OF PROGRAM (IF NOT ALREADY THERE)
- 4. ENTER
- 5. EDIT

BLOCK NUMBERS OR CHARACTERS CAN NOW BE SEARCHED FOR QUICKLY AND EASILY.

- 6. PRESS THE SEARCH & GRAPHIC SOFTKEY (FAR LEFT SOFTKEY)
- 7. THE SCREEN WILL PROMPT FOR A 'STRING'

A STRING CAN EITHER BE A BLOCK NUMBER OR A STRING OF CHARACTERS.

IE. N 10, N 99 (A BLOCK NUMBER MUST BE KEYED IN AS ON THE SCREEN, A SPACE BETWEEN THE N AND NUMBER)

CHARACTERS CAN BE ANYTHING, M30 X100, R, F2000.

BLOCK NUMBERS CAN BE SEARCHED FOR FROM ANY POSITION WITHIN THE PROGRAM

CHARACTERS CAN ONLY BE SEARCHED FOR BELOW THE LINE THE CURSOR IS ON.

- 8. KEY IN THE STRING YOU WISH TO SEARCH FOR.
- 9. ENTER

THE CONTROL WILL JUMP TO THE BLOCK WHICH HOLDS THE FIRST CORRESPONDING STRING.

10. IF THE NEXT STRING TO BE SEARCHED FOR IS THE SAME, JUST PRESS SEARCH AND ENTER (AS THE LAST STATED STRING WILL BE ACTIVE).

EXTRA OPTIONS FOR PROGRAMS

- 1. MEMORY
- 2. PROGRAMS
- 3. KEY IN NAME OR NUMBR OF PROGRAM (IF NOT ALREADY THERE)
- 4. ENTER

5. COMMAND



TO ESCAPE FROM ANY OF THE ABOVE OPTIONS USE THE ROUND GREEN PAGE OUT KEY.

TO DELETE

1. <u>A CHARACTER.</u>

ie. COLLEEGE

CURSOR

To delete the second 'E', move the cursor, via the two softkeys \checkmark \diamond \rightarrow over the 'G' and press the red delete button on the right hand side of the alpha/numeric pad.

NB:- Always take the cursor to the right of the character you want to delete.

2. PART OF A LINE.

ie. G1 X0 Y0 F9000 - To delete the feedspeed in one go, take the cursor over the first character of the part you want to delete, 'F' and press

SHIFT, DELETE

3. <u>A LINE.</u>

Place the cursor at the start of the line and press SHIFT, DELETE & ENTER

4. INDIVIDUAL PROGRAMS.

- a) MEMORY
- b) PROGRAMS
- c) KEY IN NAME OR NUMBER OF PROGRAM TO BE DELETED
- d) ENTER
- e) COMMAND
- f) DELETE

This will delete the selected program unless 'File Protection' is on. NO PROMPT is given.

5. THE WHOLE OF THE PROGRAM MEMORY STORE.

<u>(</u>

- a) INFORMATION
- b) RESET/DELETE
- c) DELETE PROGRAMS
- d) ENTER (to confirm)

This will delete all programs in memory, even the programs with 'file protection' on.

To escape having pressed DELETE PROGRAMS press the round page out key.