PROGRAMMING MANUAL

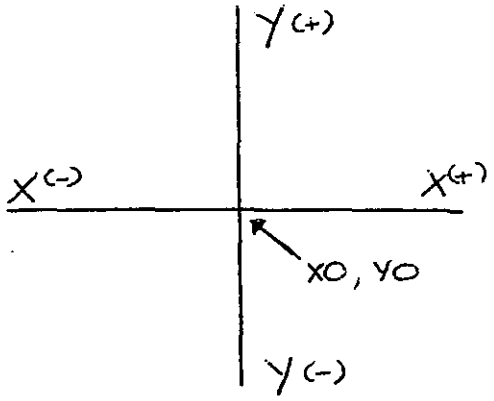
FOR BOSCH ALPHA 3 CONTROL

WADKIN COLNE, DIVISION OF WADKIN PLC
LODGE HOLME,TRAWDEN,COLNE,LANCS BB8 8RB

A MEMBER OF THE THOMAS ROBINSON GROUP PLC.
TEL:0282 866717 TELEX:635032 FAX:0282 862541

CNC PROGRAMMING FOR THE BOSCH ALPHA CONTROL

X and Y axis



Notes:-

(1) G90 - Absolute programming, the co-ordinates of a data point are defined in relation to an absolute zero.

(2) G91 - Incremental programming, the co-ordinates of a data point are defined in relation to the preceding point.

Which ever type of programming is used needs to be stated at the beginning of the program. G90 and G91 cancel each other out.

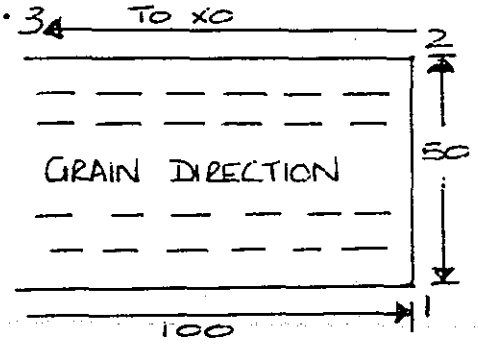
(2) The control generates its own block numbers, ie. when you input a line of program it will be issued with the respective block number, ie. N1, N2 ... N13 etc.

(3) The first line of a program using the Alpha control must be **G78** (start of main program), whatever block this is on needs to be noted, for later on in the program.
All programs are entered consecutively, so it is one continuous run, it is therefore important to note what line each program starts on, for running the correct one.

(3) The last line of a program will read **M30** (end of main program, with jump address), **ADR=A** (address of first line of program, G78's block number), this will jump the program back to the start each time it is run.

(4) When actually cutting you need to tell the control how fast to travel. This is done by entering a feedspeed on the <u>first</u> G1, G2 or G3 line, once it is in it will remain active for the rest of the program, unless otherwise altered. Feedspeeds can be altered on any line of the program, if you want to tell the control to slow down, say if you were cutting across the grain, or cutting a curve the new feedspeed would be programmed on the same line as the information to cut.

eg.




```
(1) G1   X100   Y0    F4000
(2) G1   X100   Y50   F500
(3) G1   X0     Y50   F4000
```

When working in METRIC the feed is in units of 1 milimetre/minute (maximum programmable is 9999 mm/min, or virtually 10 metres/minute)

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

(5) When actually running program, if the cutter is in 'mid-air' (ie. not cutting) then the machine needs to go as fast as possible to minimise cycle time, (for instance moving to component datum to set zero). The code for this rapid traverse is **G0**, so the move to component datum could be G0 X700 Y300, as opposed to G1 X700 Y300.

(6) Entering a line of program:-

G - 1, 2, 3 etc.
X - co-ordinate.
Y - co-ordinate.
Z - for radius value on curve,
for Z axis movement.
F - feedspeed
S - spindle speed
M - 'M' modes, and end of program with jump address, ADR=A.

B - repetition factor (for use with G22, G23)
C - sub-program number (G98)
H - dwell in 0.1 seconds (G4)
T - tool and radius compensation (G41, 42 and 43)

(7) When machine is switched on the control displays two values:-

eg. 6132 8192

amount of memory still available — total amount of memory available

(8) On switching on the memory is protected.

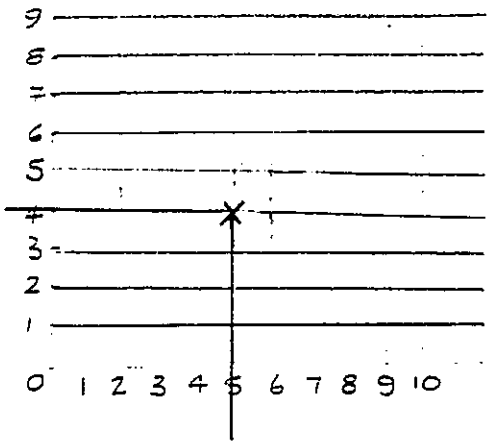
Mode + 9 (Manual Data Input)

then G37 - Memory enable

G38 - Memory protect

G39 - Memory clear, ? enter for yes.

WHAT IS A POINT?

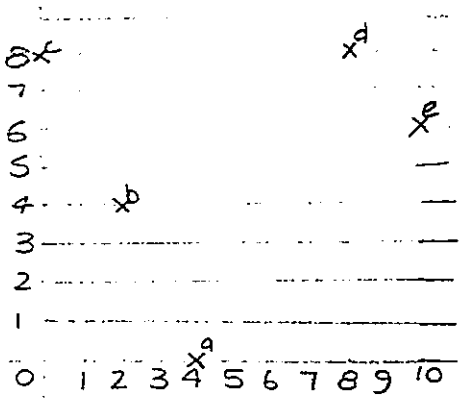


A line which is 5 units along the 'X' axis is know as X5, similarly a line which is 4 up in the 'Y' axis is known as Y4.

Where these two lines intersect is known as a point or co-ordinate.

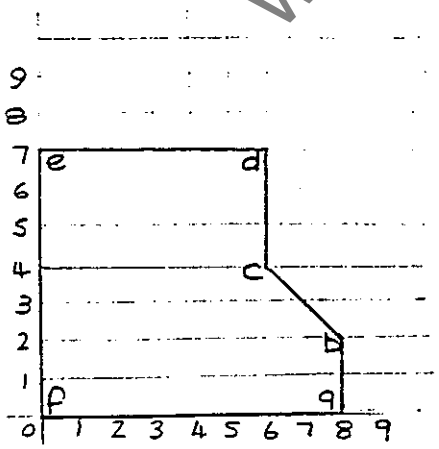
So the point described is known as X5 Y4.

Describe the points below:-



(a)
(b)
(c)
(d)
(e)

POINTS FOR A SHAPE

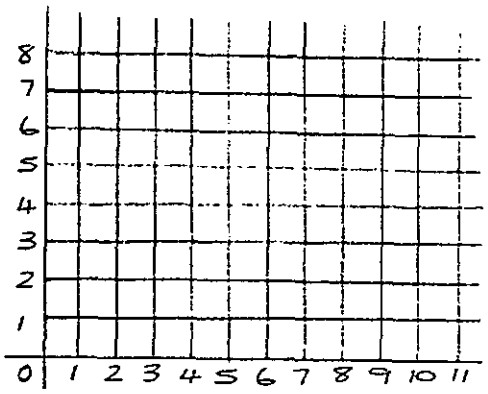


If we draw a shape or component on the graph, all the intersectio points of it will lie on points of the graph, thus

	X	Y
(a)	X8	Y0
(b)	X8	Y2
(c)	X6	Y4
(d)	X6	Y7
(e)	X0	Y7
(f)	X0	Y0

On the graph below plot the points and draw lines between the points to describe the shape.

Start from X0 Y0
X6 Y0
X8 Y3
X8 Y7
X2 Y7
X0 Y3
X0 Y0



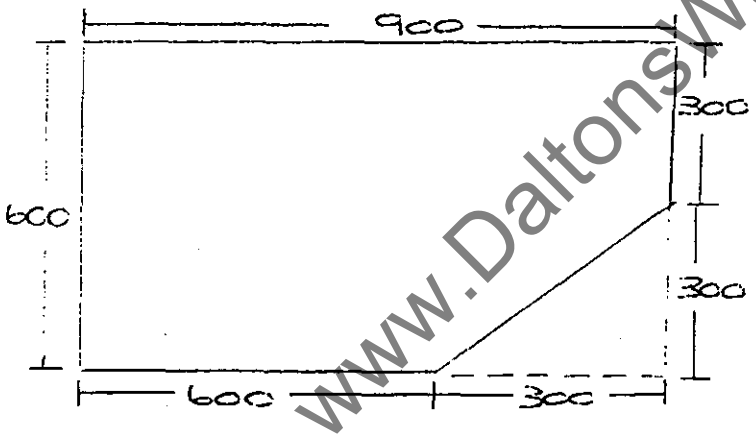
This is the basis of the CNC PROGRAMMING CO-ORDINATE SYSTEM. The table of the router is measured as a grid just like the graph but with a much finer spacing (resolution).

The resolution of the Wadkin Router is .01 mm or 1/1000 inch depending which units you are using.

STRAIGHT LINES AND CURVES

i) Straight Lines

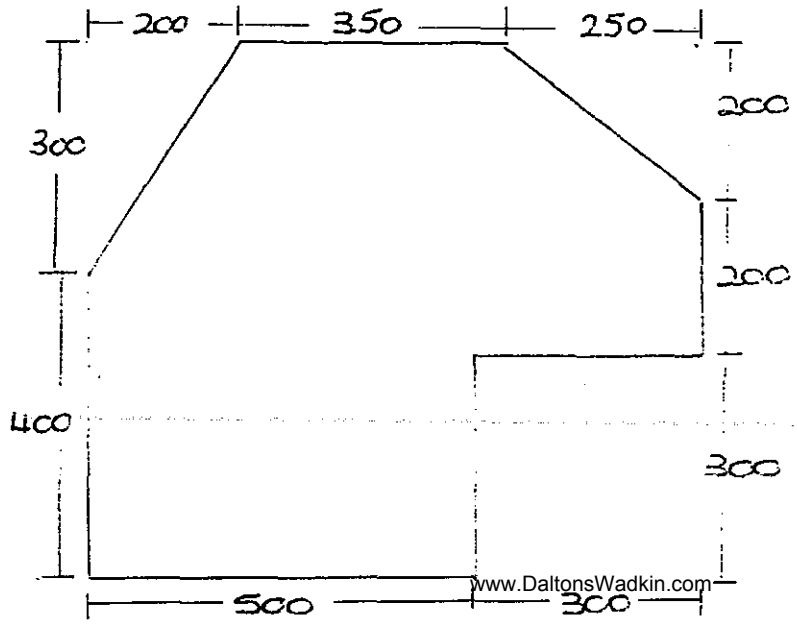
All straight line moves need to be preceded by a **G1** code.



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

G1 X0 Y0
G1 X600 Y0
G1 X900 Y300
G1 X900 Y600
G1 X0 Y600
G1 X0 Y0

Write down the points to describe the shape below:-



ii) Curves

All curves need to be preceded by a G2 or G3 code depending whether the circular movement is clockwise or anti-clockwise.

G2 = Clockwise circular movement

G3 = Anti-clockwise circular movement

The radius is also required, this is programmed as a + or - Z value.

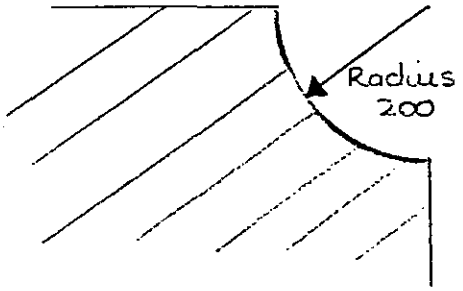
The control automatically assumes that the value is positive unless otherwise stated, ie only the negative (-) sign is required.

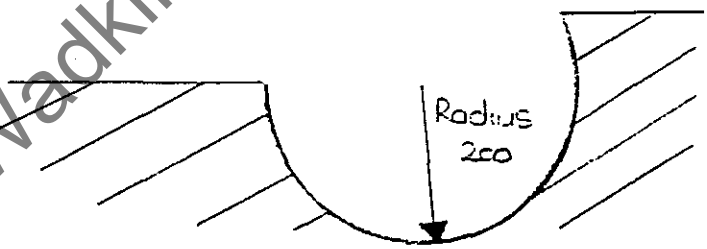
Rule for radius value:-

If the angular movement of the curve is 180 degrees or more then the radius is programmed as a positive value, eg. Z200.

If the angular movement of the curve is less than 180 degrees then the radius is programmed as a negative value, eg. Z-150.

EG. (1) Less than 180^{o} Z-200 (2) Greater than 180^{o} Z200

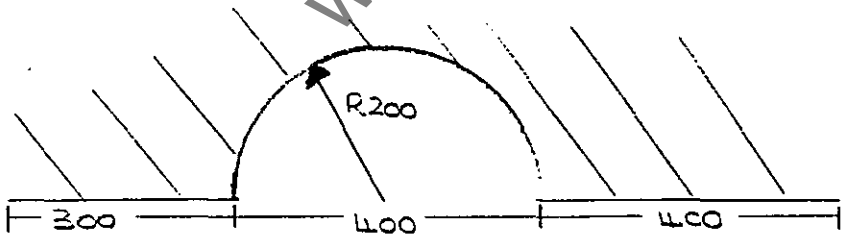




The X and Y co-ordinates are the end point of the arc.

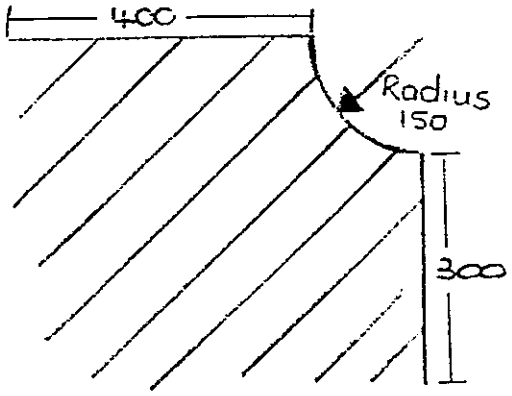
Example 1

```
G1 X300  Y0
G2 X700  Y0 Z200
G1 X1100 Y0
```



G2: Clockwise movement
X700 Y0: End point of the arc
Z200: Radius of 180 degrees, so positive value

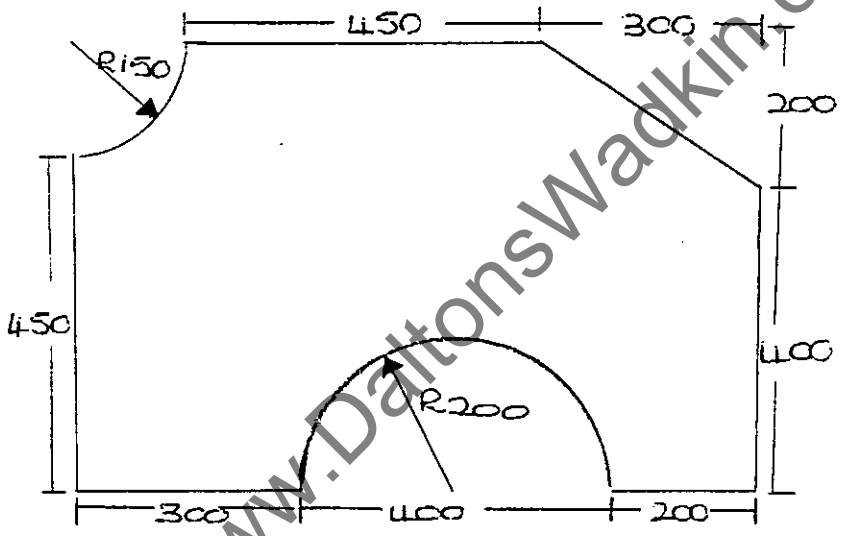
Example 2




```
G1 X400 Y0
G3 X550 Y-150 Z-150
G1 X550 Y-450
```

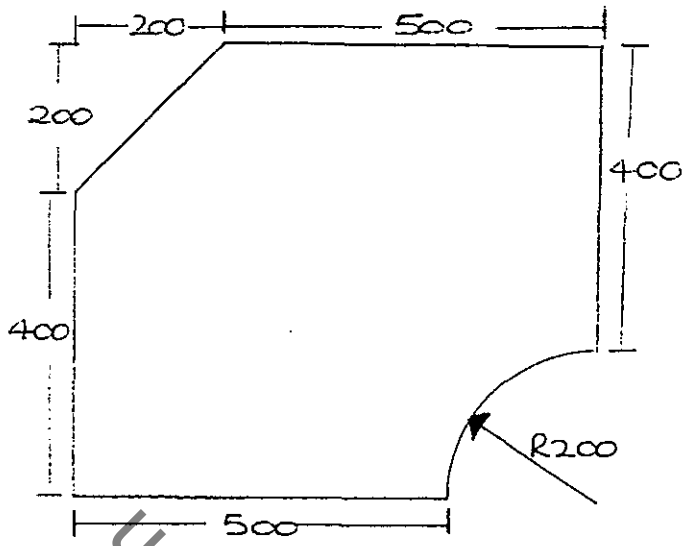
G3: Anti-clockwise movement
X550 Y-150: End point of arc
Z-150: Radius of less than 180°, so a negative value

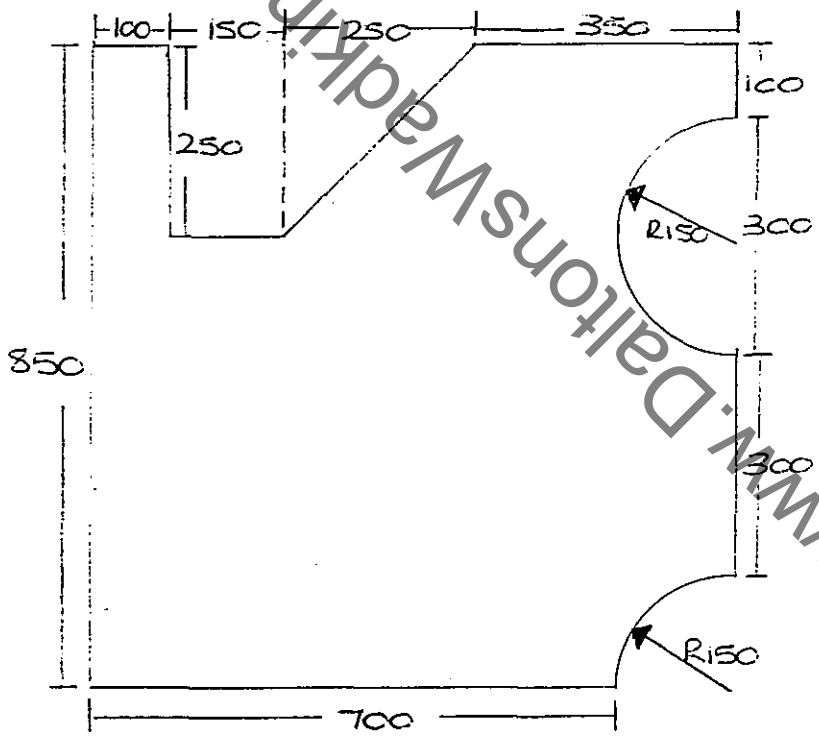
Putting these 'G' codes into a complete example:-

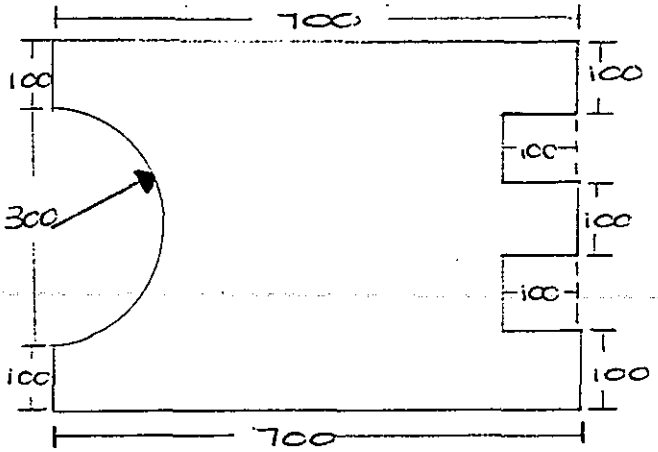



```
 1.  G78
 2.  G90
 3.  G1 X0    Y0
 4.  G1 X300  Y0
 5.  G2 X700  Y0 Z200
 6.  G1 X900  Y0
 7.  G1 X900  Y400
 8.  G1 X600  Y600
 9.  G1 X150  Y600
10.  G2 X0    Y450 Z-150
11.  G1 X0    Y0
12.  M30 A1
```

Try the following examples:-

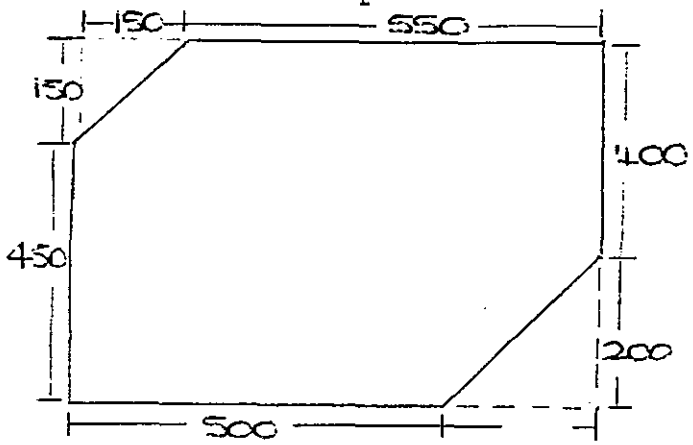






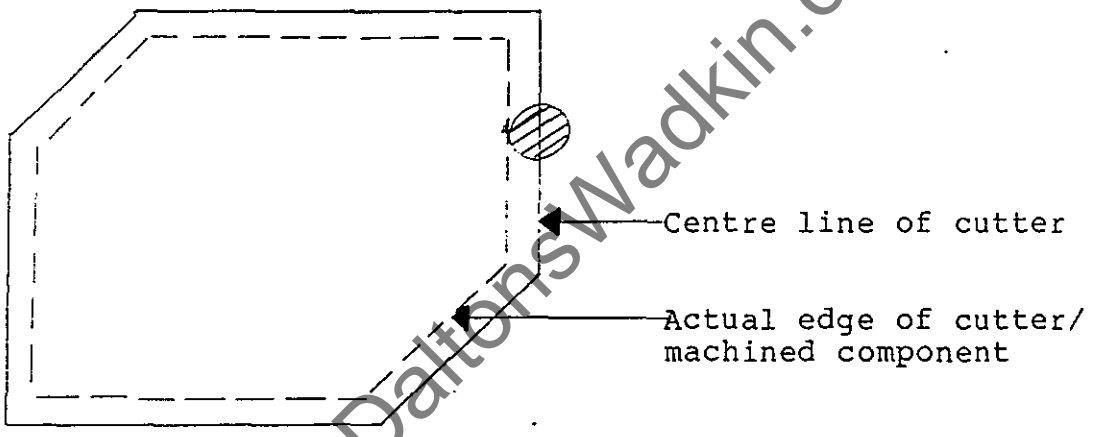
CUTTER COMPENSATION

Consider this example:-




```
G1 X0   Y0
G1 X500 Y0
G1 X700 Y200
G1 X700 Y600
G1 X150 Y600
G1 X0   Y450
G1 X0   Y0
```

If the above component was cut with a cutter of 10 mm diameter then the component would be cut undersize by 5 mm (the radius of the cutter) because we programmed the **CENTRE LINE** of the cutter.



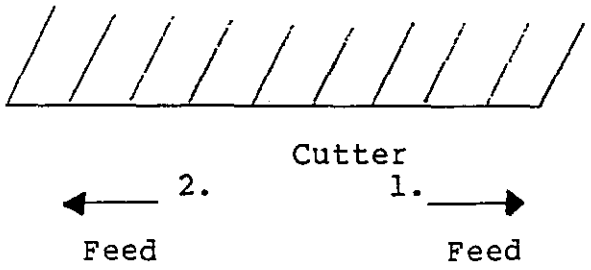
To enable us to still program the component as previous, but still get a true sized component we use CUTTER COMPENSATION.

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

This amount is stored in Mode +

There are 32 tool stores and to call up a tool store within a program you enter the tool store number under T.

RULE (1)

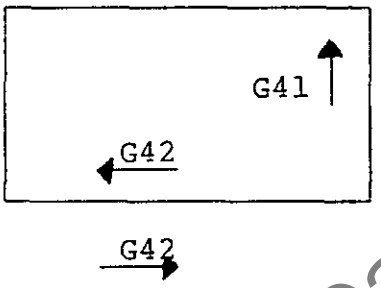


If you can imagine yourself stood on top of the cutter, facing the direction of the feed:-

1. You are on the RIGHT-HAND side of the workpiece you use **G42**

2. You are on the LEFT-HAND side of the workpiece you use **G41**

RULE (2)



Cutting **anti-clockwise** round the **outside** of the component use **G42.**

Cutting **clockwise** round the **outside** of the component use **G41.**

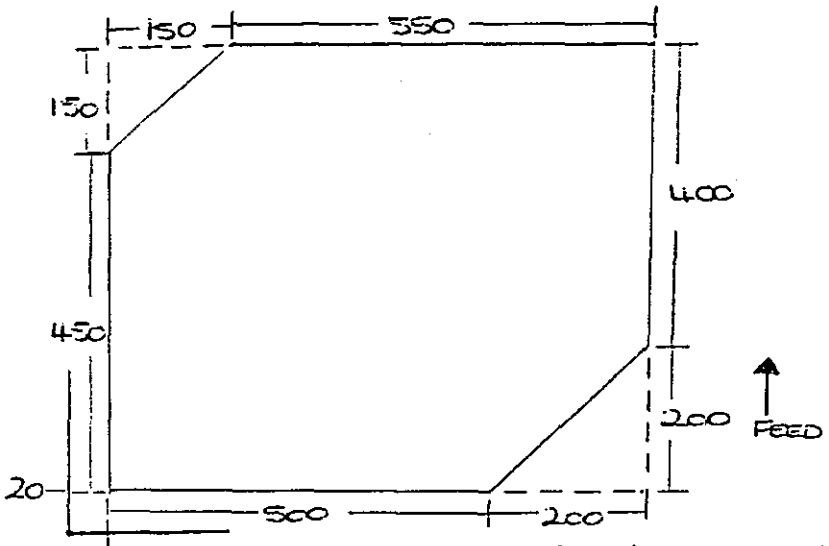
Cutting **anti-clockwise** round the **inside** of the the component use **G41.**

Cutting **clockwise** round the **inside** of the component use **G42.**

To **CANCEL COMPENSATION** we use **G40.**

Cutter compensation must be put on and cancelled with a **straight line move** (of a distance which is more than the radius of the cutter being used) before and after cutting respectively.

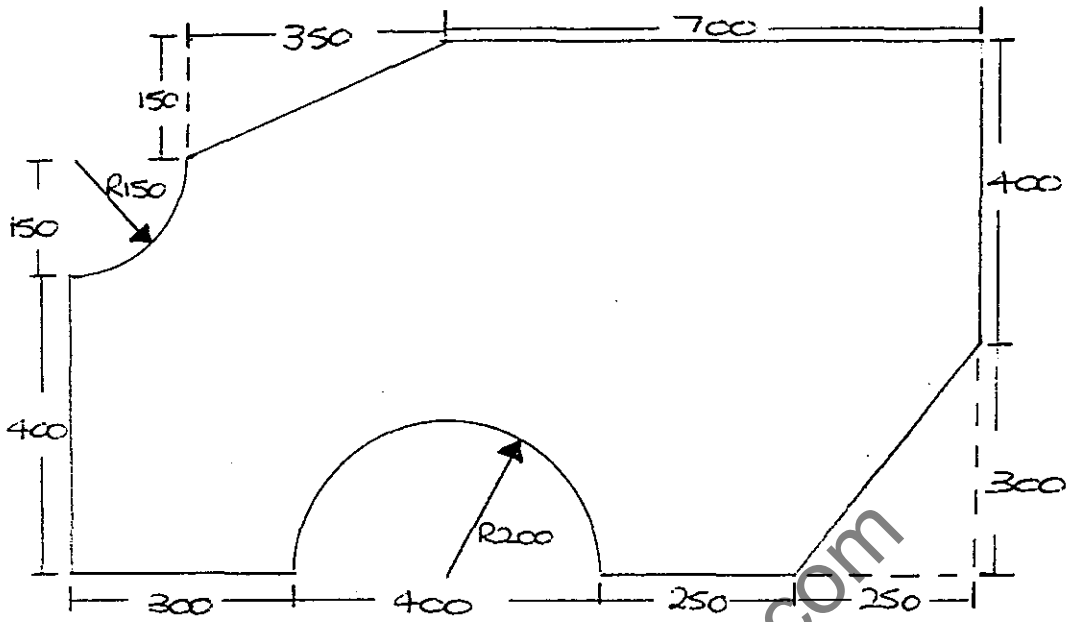
Consider the previous example:-

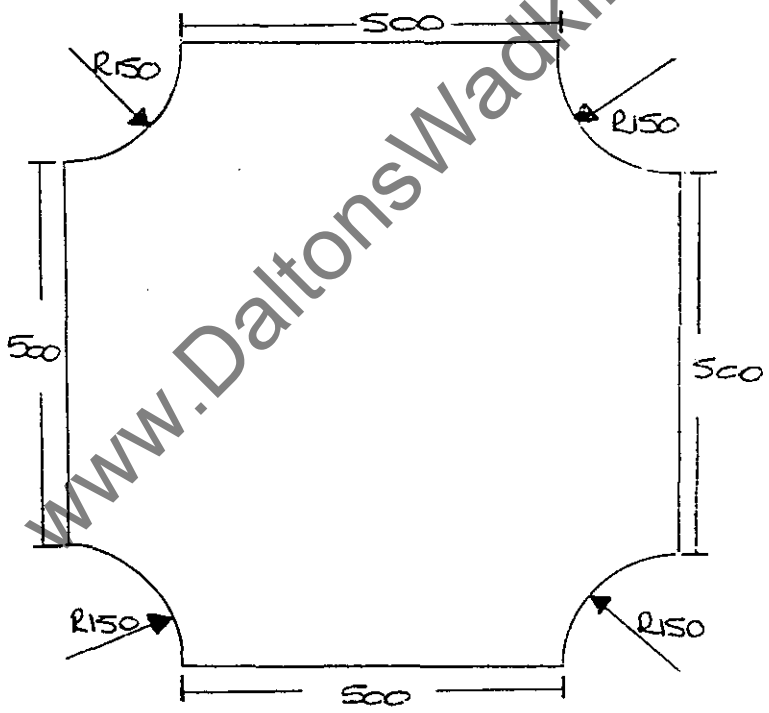



```
G78                    (Start of main program)
G90                    (Absolute programming)
G1  X-20  Y0           (A move away from the component so that
                        compensation can be set, this must be
                        at least the distance of the radius of
                        the cutter being used.)
G42 T1 - 32            (G42 - cutter compensation to the right
                        of the workpiece (anti-clockwise round
                        the outside) using the chosen tool store
                        number.  It is not necessary to know the
                        exact size of the cutter at this moment,
                        it can be entered later.
G1  X0    Y0           (Straight line move to activate cutter
                        compensation)
G1  X500  Y0
G1  X700  Y200
G1  X700  Y600
G1  X150  Y600
G1  X0    Y450
G1  X0    Y0
G40                    (Code for cancelling compensation)
G1  X0    Y-20         (Straight line move to activate cancellation)
M30 A1                 (End of main program)
```


NB:- It is good machining practice to feed on to and off the component in a straight a line as possible for compensation, this saves the cutter having to do any unnecessary moves.

Try these two examples using the previous example as a guide line:-

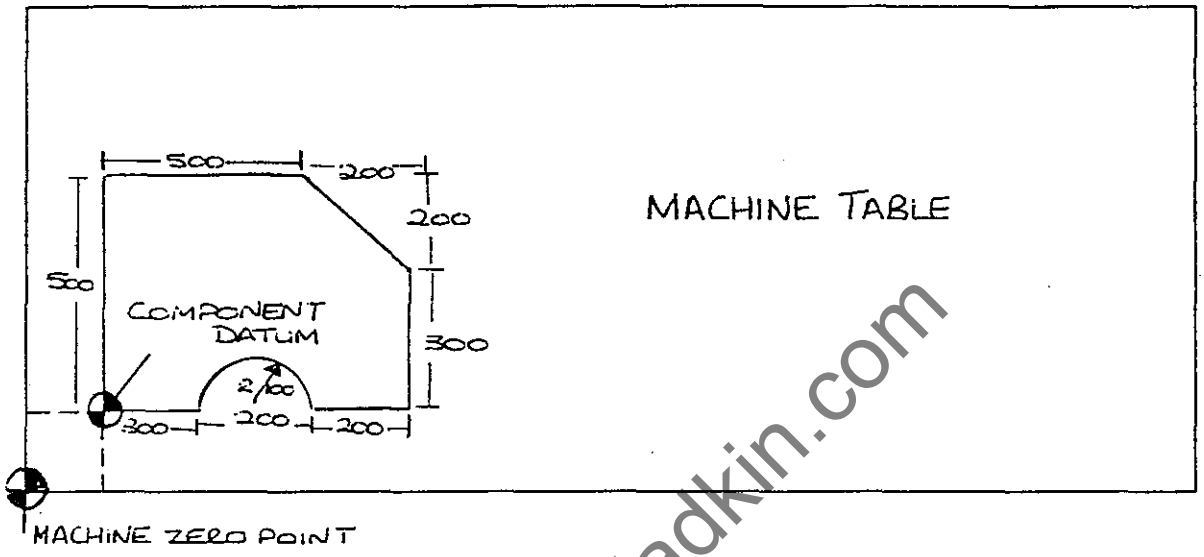




COMPONENT DATUM POINTS (MOVING ZERO POINTS)

We have used the bottom left hand corner of the workpiece as the component datum (ie. all dimensions are relative to this point).

In practice the component would be situated somewhere on the machine table.



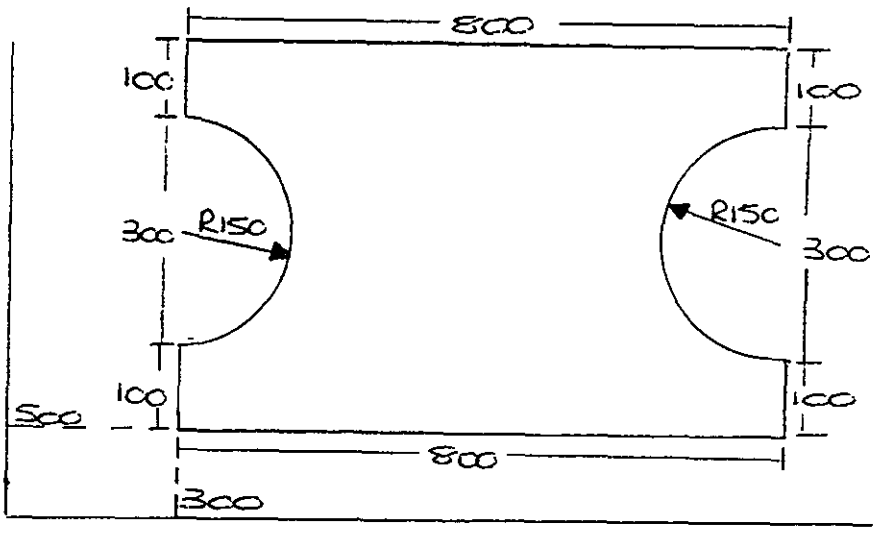
The component datum has a distance from the bottom left hand corner of the machine table (Machine Zero Point). In this case its 50 along the 'X' axis and 30 up in the 'Y' axis.

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

```
eg:  G1 X50  Y30          (Previously X0   Y0  )
     G1 X350 Y30          (Previously X300 Y0  )
     G2 X550 Y30  R100    (Previously X500 Y0  )
     G1 X750 Y30          (Previously X700 Y0  )
     G1 X750 Y330         (Previously X700 Y300)
     G1 X550 Y530         (Previously X500 Y500)
     G1 X50  Y530         (Previously X0   Y500)
     G1 X50  Y30          (Previously X0   Y0  )
```

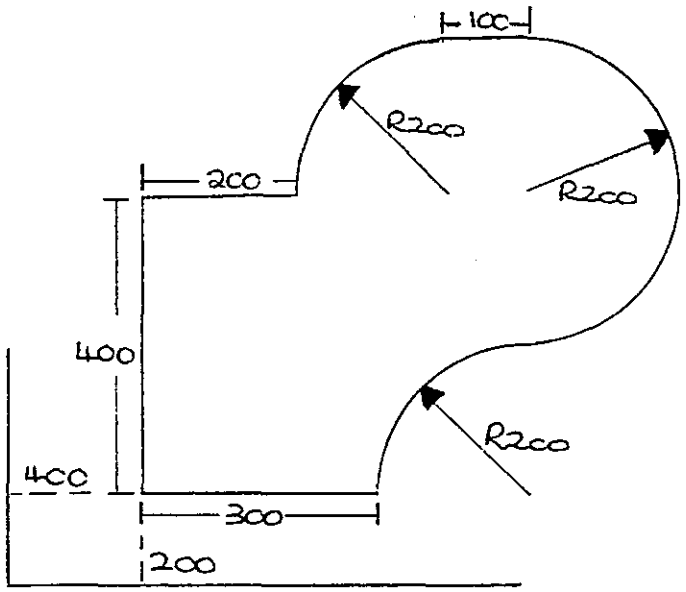
What we need to do is set the bottom left hand corner of the workpiece (or the desired co-ordinates) to equal X0 Y0 when on the machine table.

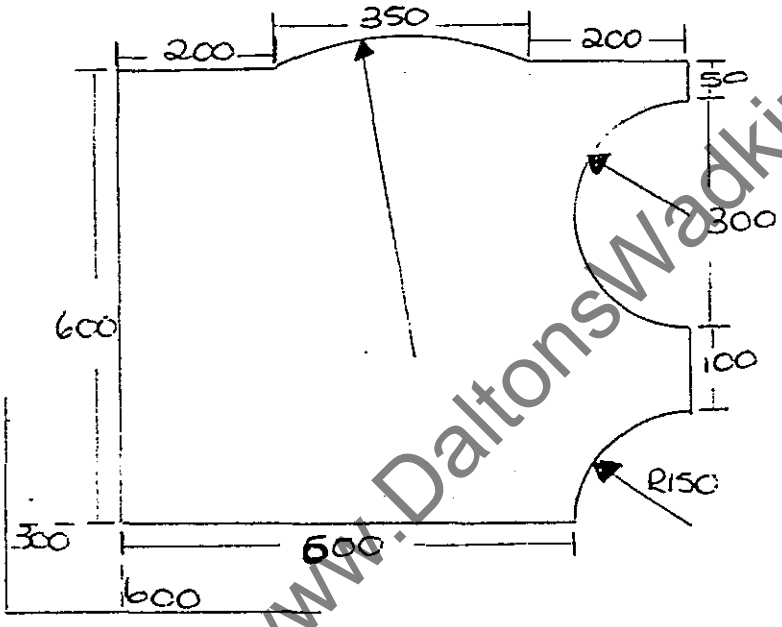
USE OF G92

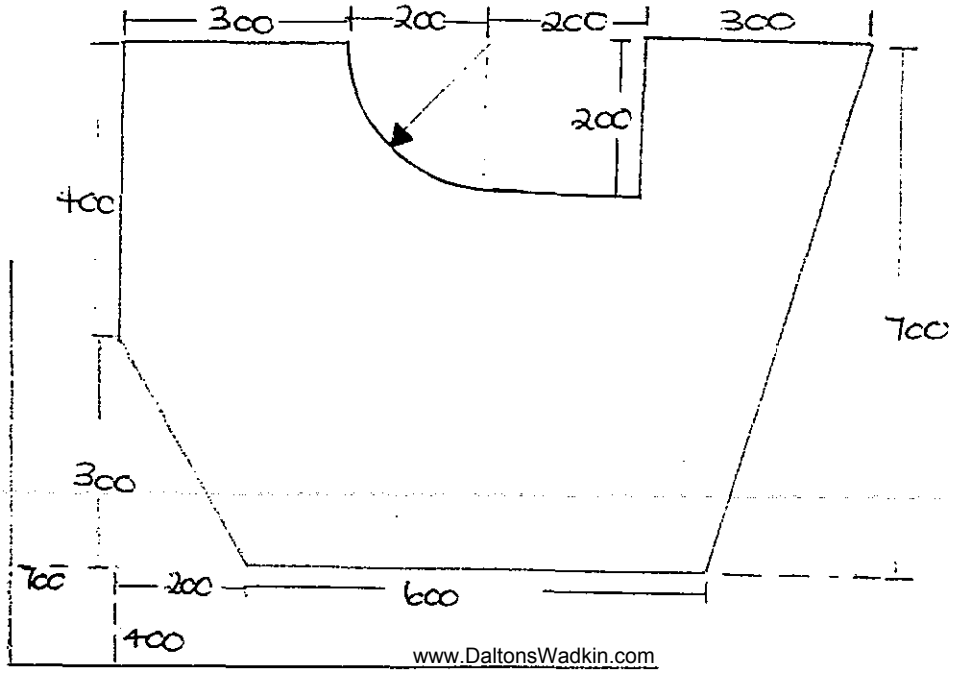



```
G78                         (Start of main program)
G90                         (Absolute programming)
G0  X500  Y300              (This move takes us to the component datum point
                             relative to the machine zero point).
G92 X0    Y0                (The G92 will set the X an Y co-ordinates to
                             whatever you want them to be, in this and most
                             cases it is X0 Y0, so now the component can be
                             programmed as before)
G1  X-20  Y0                (Move away from the component to enable us to
                             set cutter compensation)
G42 T2                      (Cutter compensation to the right of the workpiece)
G1  X0    Y0                (Straight line move to activate compensation)
G1  X800  Y0
G1  X800  Y100
G2  X800  Y400 Z150
G1  X800  Y500
G1  X0    Y500
G1  X0    Y400
G2  X0    Y100 Z150
G1  X0    Y0
G40                         (Cancel compensation)
G1  X0    Y-20              (Straight line move to activate cancellation)
G1  X0    Y0                (Back to the component datum point so that we
                             can reset its true position relative to the
                             machine zero point)
G92 X500  Y300              (We are giving the X and Y co-ordinates their
                             true position, notice that they must
                             correspond exactly with the values set at the
                             beginning)
M30 A1                     (End of main program)
```

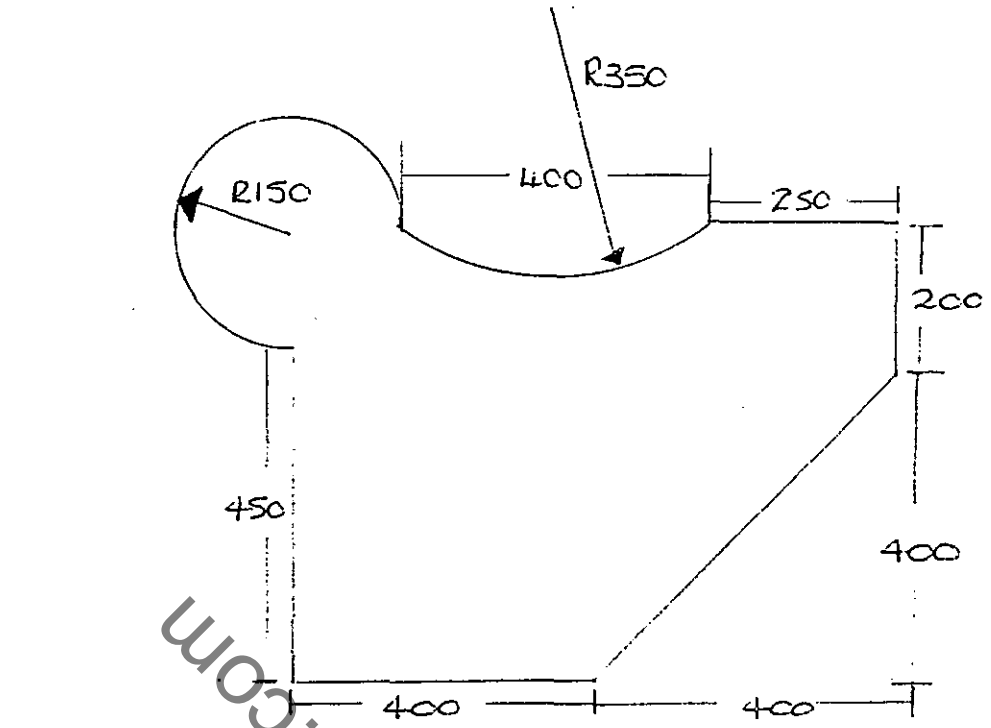
Examples - with Component Datum Points

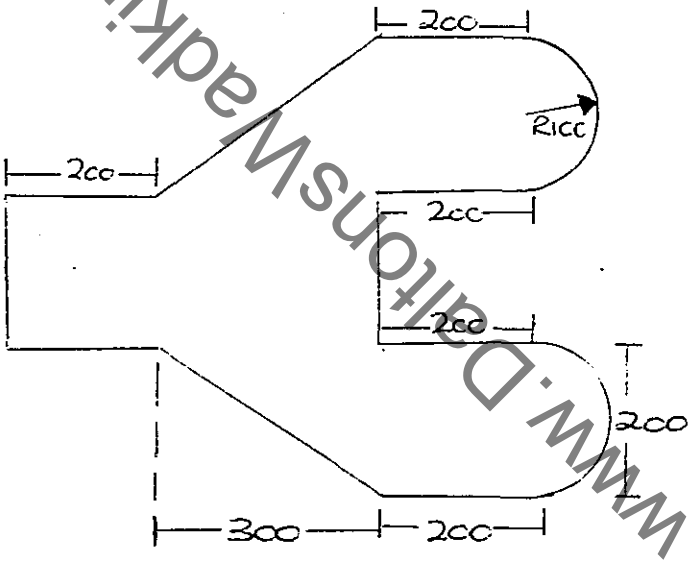


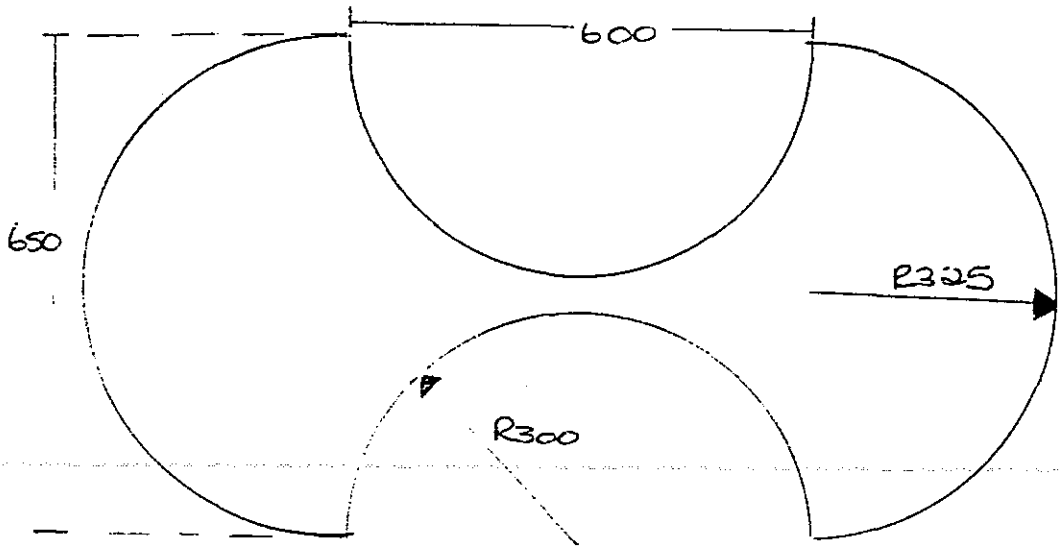


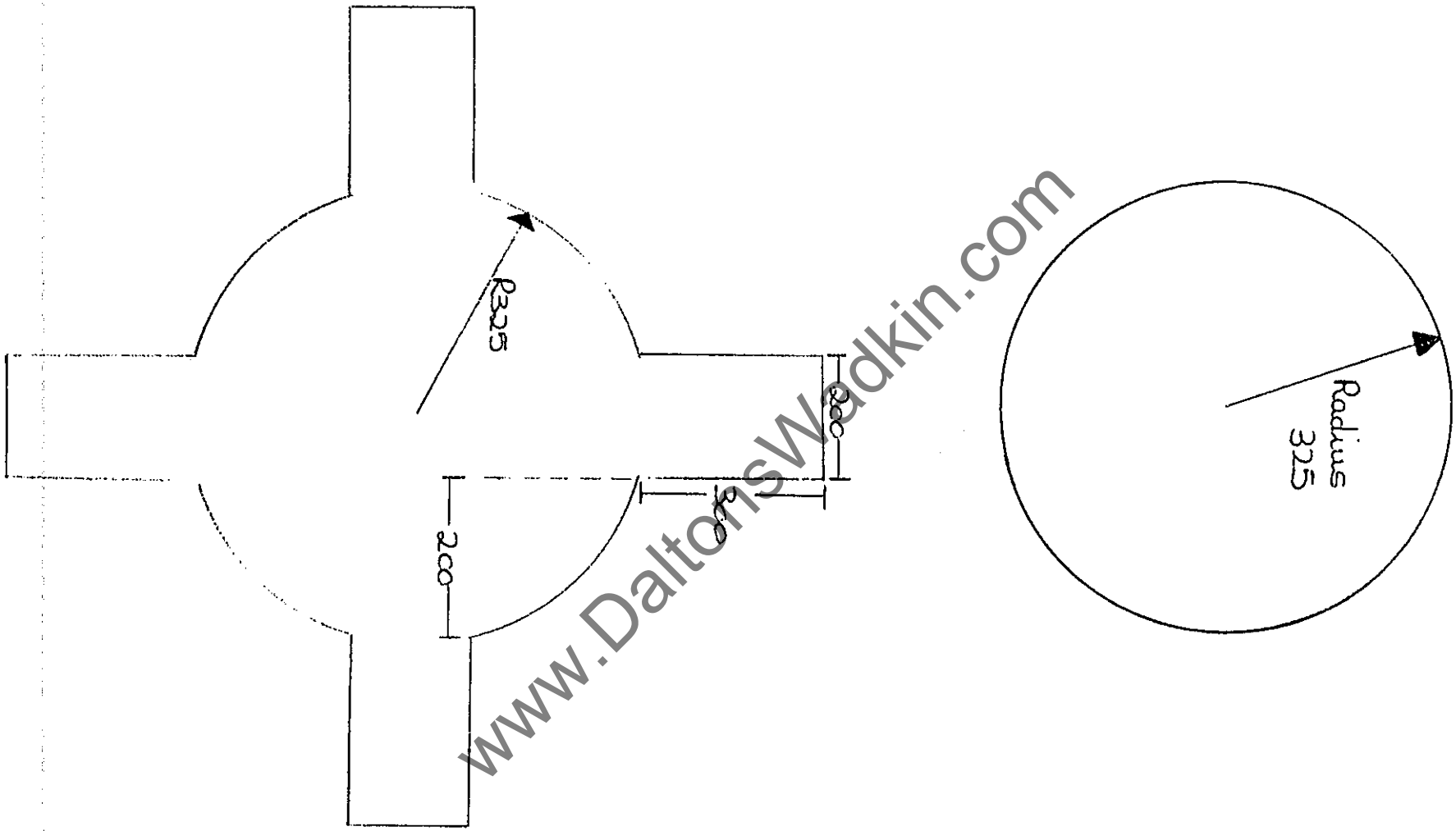


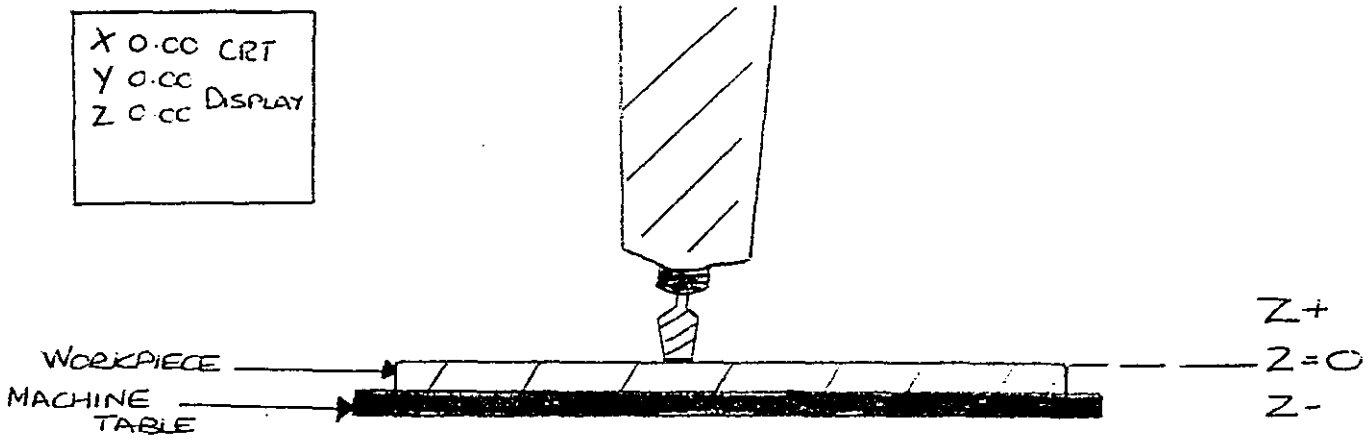
EXAMPLES.











How to set tool length compensation.

Bring cutter down so that it is just touching the top of the workpiece using the manual panel and handwheel (vacuum clamping must be on if vacuum pumps are switched on).

Note the reading of the 'Z' axis on the CRT display.

Go into Mode + , enter the number of the tool store you require and then under the 'L' column enter the Z reading as a negative value.

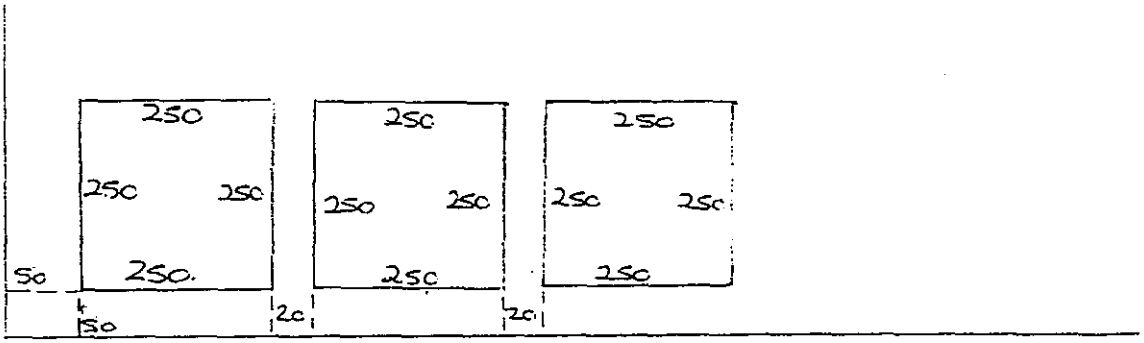
When the tool store is called up into the program it will set the top of the workpiece to equal zero.

eg.

```
G78                        (Start of main program)
G90                        (Absolute programming)
G0  X200  Y200             (Rapid traverse to component datum)
G92 X0    Y0               (Set component datum to equal zero)
G0  X-20  Y0               (Move away to start feed in position)
G42 T1                     (Call up of cutter compensation using Tool store 1.
G1  X-10  Y0   F5000       (Straight line move to activate compensation)
M8                         (Select left hand head down (on a two headed machin
G43 Z5                     (Call up cutter length offset to Z=5 using tool 1.)
G1  Z-10 F500              (Feed down to cutter depth)
G1  X0    Y0
G1  X100  Y0
G1  X100  Y100
G1  X0    Y100
G1  X0    Y-10
G0  Z5                     (Retract cutter to above the workpiece)
G40 X0    Y-20             (Cancel cutter compensation with straight line move
M9                         (Retract left hand head)
G44 Z0                     (Cancel tool length offset to machine zero)
G0  X0    Y0               (Re-position over component zero)
G92 X200  Y200             (Re reference control with respect to machine absol
G0  X0    Y800             (Off load position)
M30 A?                     (End of main program with jump address)
```

SUB PROGRAMS

ABSOLUTE SUB-PROGRAMS




```
G78                                  Start of main program
G90                                  Absolute programming
G0                                   Rapid traverse
G43 X50   Y50   Z5  T1               Tool length offset of 5 mm above using T1.
M8                                   Select left hand head down
G92 X0    Y0                         Set component absolute zero
G22 C1    B2                         See below
M9                                   Retract left hand head
G92 X860  Y50                        Select left head down
*
G44 X0    Y800 Z0 M30 A?             Cancel tool length offset to machine Z0
G98 C1                               Start of sub-program number 1
G0  X-20  Y0                         Move away to feed in position
G1  Z-5         F200                 Feed down to cutter depth
G42 X0     Y0   F5000                Cutter compensation to the right.
G1  X250   Y0
G1  X250   Y250
G1  X0     Y250
G1  X0     Y-10
G40 X0     Y-20                      Cancel cutter compensation
G0  Z5                               Retract cutter to 5mm above workpiece
G0  X270   Y0                       Go to next start position
G92 X0     Y0                        Set new component datum
G99                                  (End of sub-program)
```

NB:- A sub-program is global ie. can be called up from any main program.

G22 Call up of a sub-program.
C? Name (number) of sub-program.
B? Is the repetition factor, the sub-program will automatically run once with the call up of it, so the repetition is after it has run once.

* If you want to repete the pattern above, insert the following:-

```
G0  X50   Y350
G92 X0    Y0
G22 C1    B2
G92 X860  Y350
```

To repete further just keep increasing the Y value each time.