PART 2 OF MACHINE AND CONTROLLER

9. Polar coordinates r and a KEYS

10. Zero Coordinates, Zero At KEYS.

11. > REF COODS and X Y > REF 0 KEYS.

12. Tool compensation and qualifiers inside (i) and outside (o).

13. Qualifiers Fast (f) and Comeback (c).

14. The move instructions GO and GR keys.

15. Polar coordinates with inside (i) and outside (o).

16. Examples of the geometry of the tool path.
    1. RECTANGULAR FRAME WITH CORNERS
    2. TRIANGULAR FRAME

17. The CS and CYCLE KEYS.

18. DWELL KEY.

19. DISPLAY KEY.

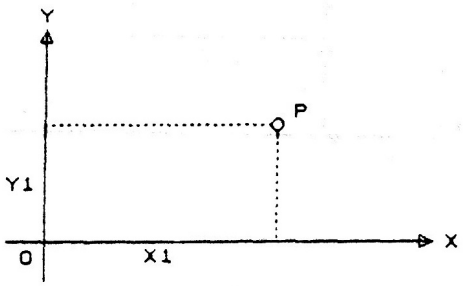
20. SPINDLE ON/OFF KEY.

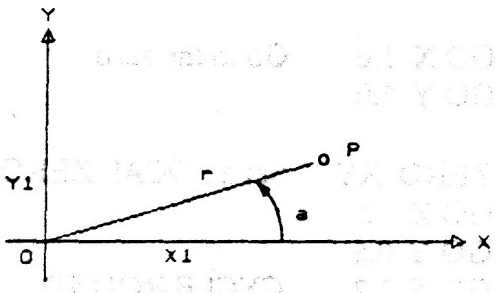
21. HALT KEY - PROGRAM RUN EXIT AND RE-START.

22. TOOL CALIBRATION AND THE TOOL CHANGE KEY.

9. POLAR COORDINATES r and a KEYS

In the X Y plane we can represent a point P either by an X1, Y1 coordinate or in polar coordinates r and a.





For doing arcs in the X Y plane we need polar coordinates.
The keys r (the radius) and a (the angle) are below the X Y Z keys on the keyboard, a is - for clockwise direction, + for counter clockwise. The entry is in degrees ± nnn.nnn the same as U, the rotary table. A zero must be established around which we can swing the arc and to do this we use a LOCAL ZERO. To set one we use the following instructions.

10. ZERO COODS AND ZERO AT KEYS

For ease of entering tool path coordinates and for centering of arcs we need the ability to create a LOCAL ZERO. All subsequent moves are referenced from this zero. Other LOCAL ZEROS can be set from the first local zero. It can be set in two ways.

ZERO COODS

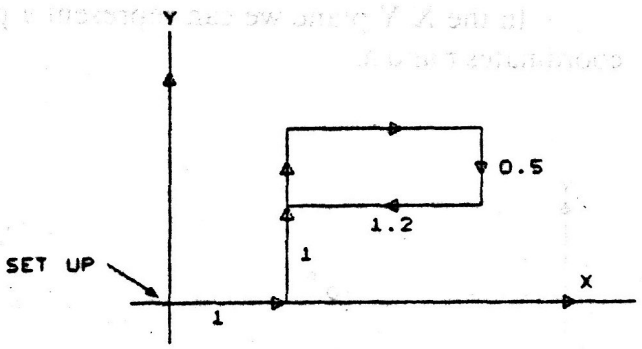
This zeros the selected axis or axes AT THE CURRENT TOOL POSITION.
Press ZERO COODS KEY. The display will ask
ZERO AXIS ?
Enter any combination of X Y Z U you require zeroed.
Since it requires moving the tool tip to the requisite point this may not be convenient so instead use : -

ZERO AT

This zeros at the following coordinates location.
Press ZERO AT KEY. Display will show ZERO AT.
Press NEXT key. Press TWICE any one of X Y Z U r a and enter the value. Press the NEXT key again, then press TWICE the next coordinate, enter the value and so on.

EXAMPLE 1

Suppose you wanted to cut a rectangle 1.2 inches by 0.5 inches at a location 1,1 as shown. Move the tool to the lower left side

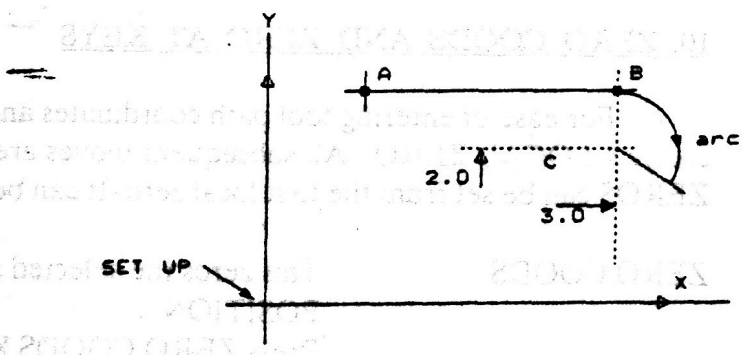


GO X 1.0 Go to the zero
GO Y 1.0

ZERO XY Set a LOCAL ZERO HERE
GO Z -.2
GO Y 0.5
GO X 1.2 CYCLE ROUND
GO Y 0.0
GO X 0.0
Z > Z MAX

EXAMPLE 2

You are at A, want to go to B and swing an arc round C of radius .75 . Do this



ZERO AT
x = 3.0 Zero at C
y = 2.0

GO r 0.75 Moves tool A to B
a 90.00

GO a -45 Cut arc.

11. > REF COODS AND X Y > REF 0 KEYS

These are one line instructions.

> REF COODS KEY. When you create a local zero to do tool moves around, you also need the ability to "UN ZERO" and to reference all subsequent moves to the SET UP (the original reference coordinates). This instruction will do it. There is no tool movement involved.

X Y > REF 0 KEY. This one line instruction MOVES the tool in X and Y to the SET UP point via the home positions in X and Y. It is used to confirm the X Y location particularly after very long or repetitive arc moves. It is a version of END NEWPART in x and y. Take care as these two instructions are easily confused.

12. Tool compensation and qualifiers. Inside (i) & Outside (o)

Sometimes it is necessary to make the tool path independent of the part geometry and dependent only on the tool diameter. Thus if the tool diameter is changed it is not necessary to recalculate the tool path instructions in the program. This is called tool compensation. The qualifiers to do this are called Inside (i) and Outside (o). They are added to the move instruction in single axis moves X,Y,Z and in various combinations in polar moves with r and a. Essentially in linear moves "inside" subtracts off half the tool diameter or tool radius from the move while "outside" adds on half the tool diameter or tool radius to the move. In angular moves the tool is placed on either side of the angle or radius or both.

13. Qualifiers Fast (f) and Comeback (c)

These can be added to the move instruction.

f FAST makes the move at maximum speed

c COMEBACK make the move and comes back.

14. THE MOVE INSTRUCTION GO ABS AND GO REL KEYS

The GO ABS or GO ABSOLUTE key means GOTO this coordinate. The instruction display as GO thus.

GO X 2.2	means goto the X coordinate 2.2
GO X 2.2 GO Y 1.3	means goto the XY coordinates (2.2,-1.3)
GO X 2.2 GO Y 1.3 GO Z 0.5	means goto the XYZ coordinate (2.2,1.3,0.5)
GO r 2.3 a 23.5	means goto the polar coordinates (2.3,23.5) (in the XY plane)

These are example of a single axis, two axes, three axes and two axes moves. The GO REL or GO RELATIVE key means FROM WHERE YOU ARE go THIS DISTANCE. It is displayed as GR thus.

GR Y 1.2	means move from here 1.2 positively in Y.
GR Y 2.3 Z -3.2	means move from here 2.3 in Y and -3.2 in Z simultaneously
GR X -0.1 Y +0.2 Z -0.3	means move from here -0.1 in X +0.2 in Y and -0.3 in Z simultaneously

GR r 2.0 means move from here 2.0 along r (angle a remains constant)

GR a 23.0 means move the tool in an arc 23 degrees clockwise from where you are. (radius r remains constant)

Again you have a single axis, two axes, three axes and two, two axes moves. In the examples above each axis selected if two or more, must be pressed TWICE to enter it into the controller.
THE QUALIFIERS inside i, outside o, comeback c, fast f only apply to single axis moves in X Y Z. Thus : -

GO c X 2.2 Means go and comeback to coordinate 2.2 in X

GO f X 3.2 Means go 3.2 fast from where you are in X.

GO i Y 1.2 Means go to coordinate (1.2 - tool radius) in Y.

GO o Z 3.2 Means go (3.2 + tool radius) in Z from where you are.

For the U axis only c and f apply.

15. QUALIFIERS WITH POLAR MOVES

The qualifiers Inside i, Outside o can be used in polar moves. With r and angle a to position the tool correctly. There are nine possibilities :-

Thus point 1 is given by

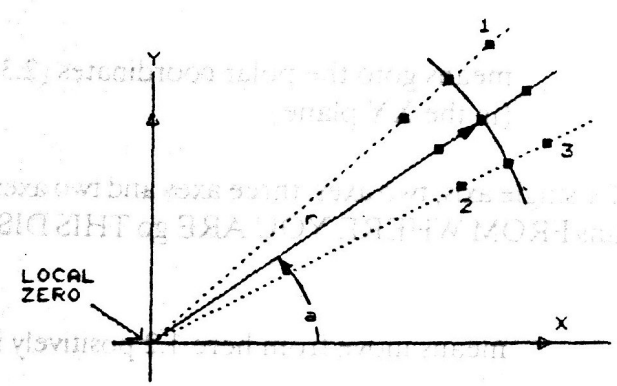
GO o r
o a

point 2 is given by

GO i r
i a

point 3 is given by

GO o r
i a



The controller calculates the "tool radius angle" and adds or subtracts it if required.

16. GEOMETRY OF TOOL PATH

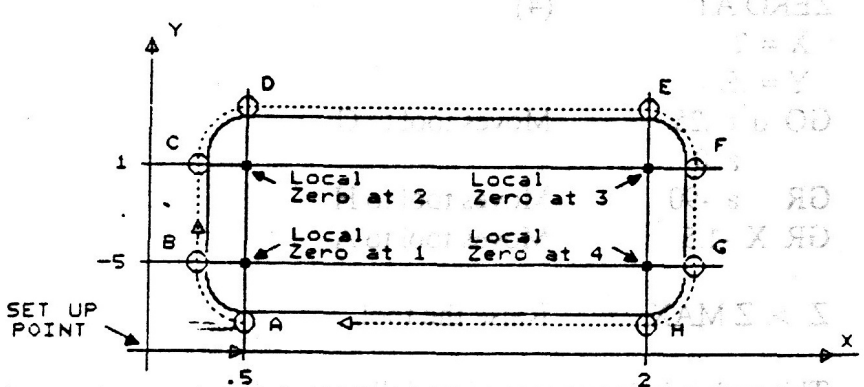
Most of the geometry in the X Y plane consists of connecting lines to lines, lines to arcs and arcs to arcs. The tool moves are generally outside the contour or inside and provided there is sufficient information on the drawing it is fairly easy to write down the tool path. However we have seen triangles in which the sum of the interior angles do not equal 180; closed contours in which the total rotation of the tool swing does not equal 360 and very insufficient information on blend radii centers.

Care is needed in selecting cartesian or polar coordinates in making tool moves as the following examples will show.

For more complex geometries involving three axes moves the calculations can only be done with a CAM program and since we use three lines per point you can rapidly generate 25000 lines on some shapes.

EXAMPLES 1 RECTANGLE FRAME WITH CORNERS

Cut out a rectangle with corners of radius .25
Rectangle is 1 by 2. Depth is .05
The required tool path is shown in dotted lines.
First locate the local zero around which we swing the arcs.
The first arc's center is at (.5,.5) so after SET UP we do this.



ZERO AT (1)
X = .5
Y = .5

GO o r .25 Move tool from SET UP to A.
a 270

Drop the tool GO Z -.05 in at A
Swing the arc GR a -90 to B
Switch back to old coordinates

> REF COODS

ZERO AT (2)
X = .5
Y = 1.0
GO o r .25 Moves tool from B to C
a 180

Swing the arc GR a -90 to D

```
> REF COODS

ZERO AT          (3)
  X = 2
  Y = 1
GO o r .25       Moves tool to E
      a 90
GR    a - 90     Moves tool to F

> REF COODS

ZERO AT          (4)
  X = 2
  Y = .5
GO o r .25       Moves tool to G
      a 0
GR    a -90      Moves tool to H
GR X -1.5        Moves tool to A

Z > Z MAX        Raise the tool.
```

This path is independent of tool diameter. It ~~is~~ really an arc to arc set of moves.

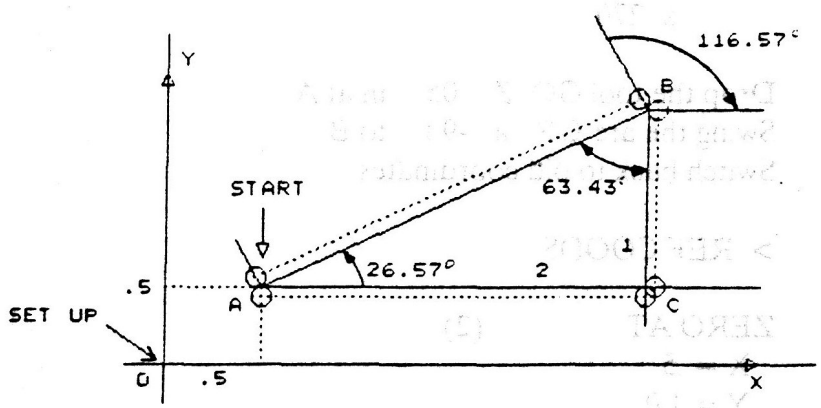
EXAMPLE 2 TRIANGULAR FRAME

Cut out a triangle of base 2 altitude 1.
After SET UP zero at A

```
ZERO AT
  X = 0.5
  Y = 0.5
```

Position tool at the bottom of the hill

```
GO o r 0.0
       a 116.57
```



Drop the tool

```
GO    Z -0.5
```

RE ZERO WITH RESPECT TO THIS ZERO AT B

ZERO AT
r = 2.24
a = 26.57

Move the tool up the hill to B

GO o r 0.0
a 116.57

Swing the tool round on the B point so it is set to go B to C

GR a -116.57

RE ZERO ON C WITH RESPECT TO ZERO AT B.

ZERO AT
r = 1
a = 270

Move the tool from B to C

GO o r 0.0
a 0.0

Swing the tool round on C 90 degrees

GR a -90.0
GR x -2.0 Moves tool from C to A.

Z > Z max

Again the path is independent of the tool diameter. It is not necessary of course to put in the qualifiers if the tool diameter is fixed.
In many cases it much simpler to think in terms of polar moves instead of cartesian coordinate moves.

17. THE CS KEY and the CYCLE KEY instruction

Many parts exhibit symmetry which may be exploited by these two instructions. You have to zero in the center of the part.

CS KEY

Simply means change sign of either X or Y or Z and go there.
If the tool tip is at x = 2, the instruction

C - SIGN X

Will move the tool to x = -2.
To cut a rectangle, zero at the center, drop the tool at one corner then use

C - SIGN X
C - SIGN Y
C - SIGN X
C - SIGN Y

to cut each side.

CYCLE KEY

Displayed as CYCLE XY, this instruction combines the four instructions above to generate a complete "rectangular" move. Depending on which corner the tool is dropped the rectangular move is either clockwise or counter clockwise. Sliding in along a diagonal and repeating this instruction will generate a pocket rectangle.

18. DWELL KEY

Display will show DWELL nn. Enter 01, 02, etc for seconds of required dwell time.

19. DISPLAY KEY

This instruction DISPLAY (X) (Y) (Z) (r) (a) can be put in anywhere and as many times as required. It will display the current value of the selected axis or axes. It is useful for debugging programs.

20. SPINDLE OFF / ON KEY

When pressed the display will show SPINDLE OFF.
To change to SPINDLE ON press + / - key. These instructions should be put in after the program is working (after SET UP - SPINDLE ON, before END NEWPART - SPINDLE OFF and where necessary around a TOOL CHANGE instruction). Before running a program with these instructions, switch the spindle PROGRAM / LOCAL to PROGRAM, turn the spindle ON and then set the rpm adjust.

21. HALT KEY PROGRAM RUN EXIT & RE-START

At any stage you can put in a HALT instruction. This halts the controller in program run. To continue, press the NEXT key. If the HALT KEY is pressed during program run, the controller will halt after it has finished the current instruction, or function.

In both cases in halt, you can exit the program run mode to any other mode (e.g. change an instruction in program enter mode) go back a few lines and press program run again. Functions (canned routines) should always be restarted at their beginning. You can also go forward to END or END NEWPART and program run from here. This will recycle to SET UP which can also be exited by pressing HALT to allow program modification. Restart at SET UP with program run. Care should be exercised in this. Exiting from local zeroes and repeat loops can lead to problems in re-starting.

22. TOOL CALIBRATION AND THE TOOL CHANGE KEY

For multiple tools in one program you need to calibrate them first before running the program. Calibration means measuring the Z differences (or offsets) in height - each one from TOOL 1. TOOL 1 is used in the SET UP and so the difference in height is automatically factored in when TOOL 2, TOOL3 etc. is called up in the program to a maximum of 8 TOOLS. The offsets are stored in the controller until changed.

To load them in, put in TOOL 1 and place the probe on a flat surface under the tool. Go to MANUAL mode, answer YES to TOOL CALIB?, NO to PROBE. TOOL 1 will be displayed. Press NEXT and TOOL 1 will descend to touch the probe then the display will show TOOL Z. Press NEXT again and the Z axis will go up to display TOOL 2. Put in TOOL 2, press NEXT and the tool will descend to touch the probe. The display will change to TOOL Z. Press NEXT and it will go up again and display TOOL 3. To exit from this, press HALT if less than 8 TOOLS.

If any tool breaks during a program ALL the tools have to be re-calibrated from TOOL 1 again - when the new tool replaces the broken one.

To enter a tool change instruction press the TOOL CHANGE KEY. Answer YES to the display TOOL n? then enter the desired number. E.G. TOOL 3.

It should be preceded by a spindle OFF instruction, followed by a spindle ON instruction, followed by a TOOL DIAMETER instruction.

During program run, the program stops, the spindle is raised and you insert the correct TOOL number. Press NEXT to continue running the program.