Drilling operations

Q.1 → 200 X 100 X 15 mm

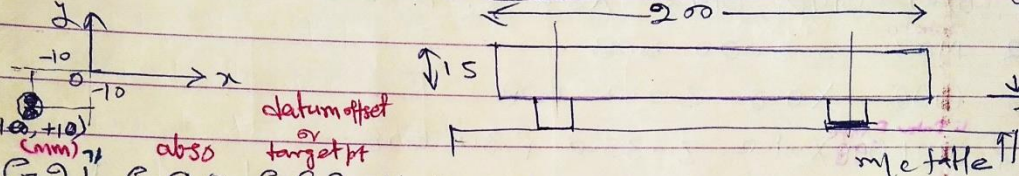
feed 200 mm/rev

speed 800 rev/min.

target x = 10 mm

y = −10 mm

z = 10 mm



N01 G21 G90 G92 X-10 Y-10 Z10 EOB

N02 G00 X20.0 Y10.0 EOB (rapid traverse)

N03 G01 G95 Z-18.0 F200.00 S800.00 M03 EOB (linear interp, feed mm/rev, spindle start) (hol

N04 G01 Z10.0 EOB taking out from hole-1

N05 G00 X180.0 Y10.0 EOB

N06 G01 G95 Z-18.0 F200.00 EOB (hol

N07 G01 Z10.0 EOB taking out drill bit from the hole.

N08 G00 X180.00 Y90.0 EOB

N09 G01 G95 Z-18.0 F200.0 EOB (hole-

N10 G01 Z10.0 EOB taking out drill bit from the hole

N11 G00 X20.0 Y90.0 EOB

N12 G01 G95 Z-18.0 F200.0 EOB (hole

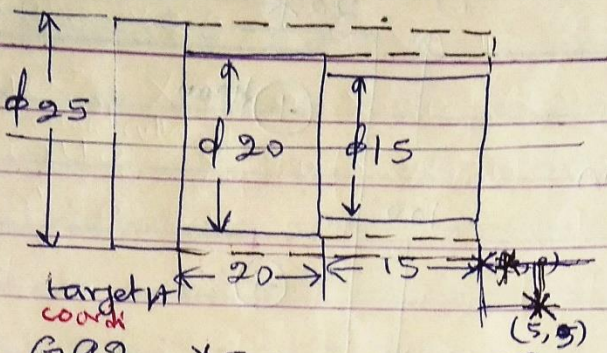
N13 G01 Z10.0 EOB

N14 G00 X-10.0 Y-10 EOB rapid traverse to target pt

N15 M05 M30 EOB. (spindle stop, end of program, m/c stop)

Turning opns take tra as a co-ord

S = 3000 rpm
f = 30 mm/min.



N01 G21 G90 G92 X5.0 Z5.0 EOB
N02 M03 S3000 EOB
N03 G00 X0.0 Z2.0 EOB
N04 G01 G98 X-1.0 Z-35.0 F30 EOB
N05 G00 X0.0 Z2.0 EOB tool retrieves alo
N06 G01 X-2.0 Z-35.0 EOB
(N07) G00 X0.0 Z2.0 tool retrieves
N08 G01 X-2.5 Z-35.0 EOB
N09 G00 X0.0 Z2.0 EOB
N10 G01 X-3.5 Z-15.0 EOB Depth of cut = 1.0 turning
(N11) G00 X0.0 Z2.0 EOB
N12 G01 X-4.5 Z-15 EOB
N13 G00 X0.0 Z2.0 EOB
(N14) G01 X-5.0 Z-15.0 EOB final cut
N15 G00 X5.0 Z5.0 rapid traverse to target pt.
N16 M05 M30 EOB

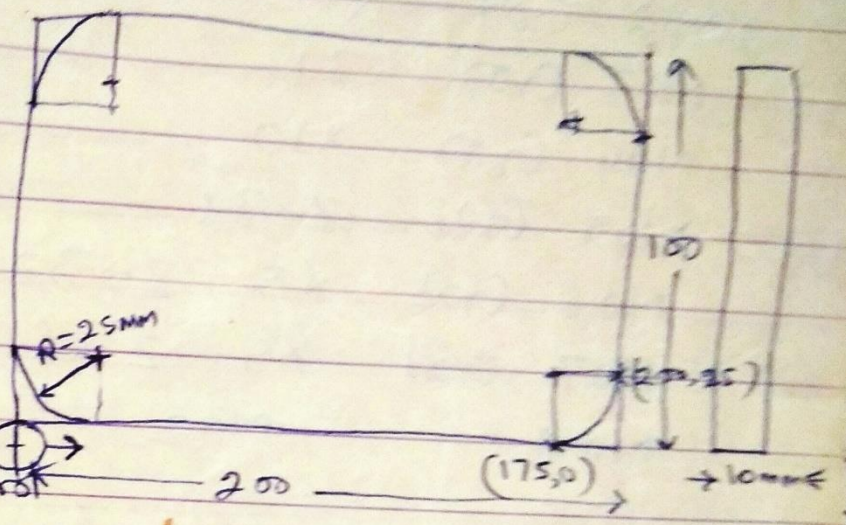
end mill cutter = 25 mm dia

f rate = 300 mm/min

S = 800 r/pm

target (−30 mm, −30 mm, +10 mm)

cutter radius compensation = difference of programmed dia & cutter. (afterwards the control sys will generate a new cutter path.



mm abs target pt

N05 G21 G90 G92 X−30 Y−30 Z10.0 EOB

N10 G00 X0.0 Y0.0 Z−20.0 S800 M03 EOB

Lin cutter offset(R) feed. number

N15 G01 G42 G94 X175.0 Y0.0 D04 F30 EOB

cutter dia data (−12.5 mm) saved in register

xy plane circ

N20 G17 G03 X200.0 Y25.0 R25.0 EOB

Lin

N25 G01 Y~~75~~.0 EOB

N30 G17 G03 X175 Y100.0 R25.0 EOB

N35 G01 X25.0 Y100.0 EOB

N40 G17 G03 X0.0 Y~~7~~5.0 R 25.0 EOB

N45 G01 X0.0 Y25.0 EOB

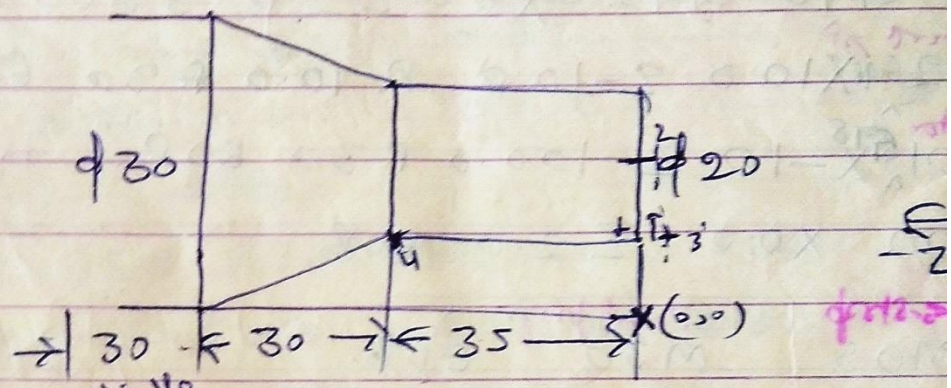
N50 G17 G03 X25.0 Y0.0 R25.0 EOB

N55 G00 G40 (canned cutter off) X-30.0 Y-30.00 EOB

N60 M05 M30 EOB

(5) S = 800

f = 200 mm/min



frame mon ↑spindle cw

N01 G91 G71 M03 S800 EOB

N02 G01 X-15 F200 EOB — facing opⁿ

N03 G00 Z1 EOB — clearance of

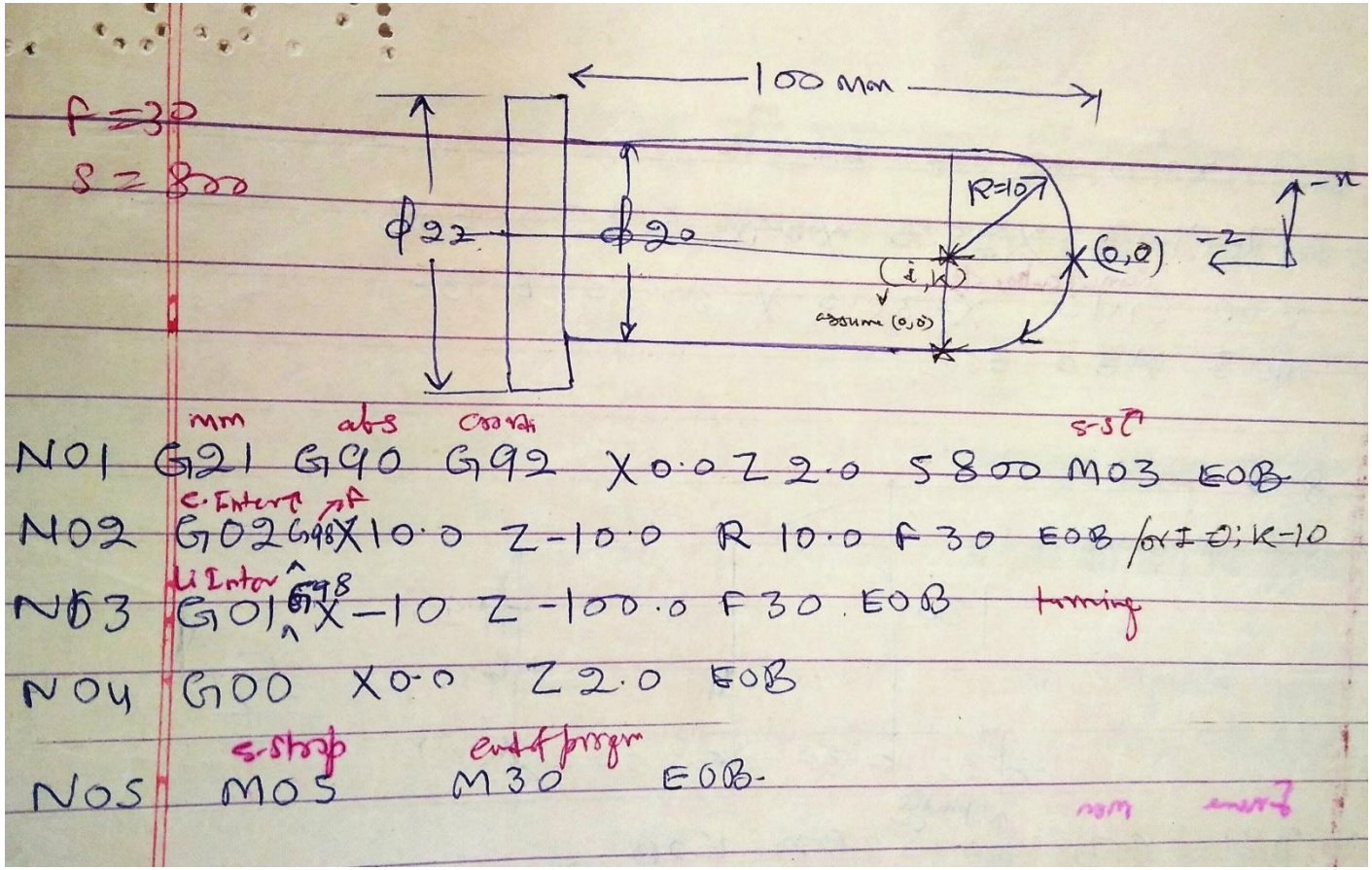
N04 G00 X10 EOB - tool move to

N05 G01 Z-36 EOB - turning opⁿ

N06 G01 X5 Z-30 EOB — taper tu

N07 G01 X5 Z66. EOB — tool back

N08 M02 EOB — end of



mm abs coord s-st

N01 G21 G90 G92 X0.0 Z2.0 S800 M03 EOB

c. Entert

N02 G02 G98 X10.0 Z-10.0 R 10.0 F30 EOB for I=0; K=-10

Li Entor

N03 G01 G98 X-10 Z-100.0 F30 EOB turning

N04 G00 X0.0 Z2.0 EOB

s-stop end of prog

N05 M05 M30 EOB.

Example

02 (All dimensions are in mm).

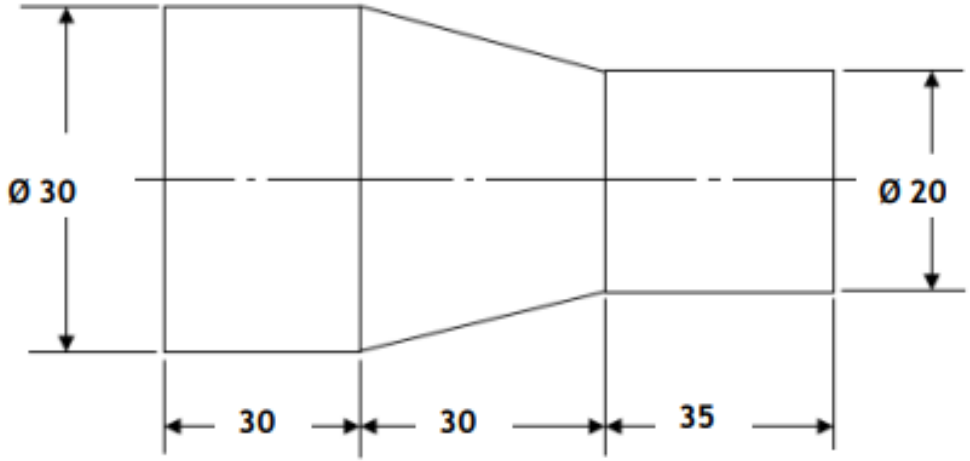


Figure 4.12 : Taper Turning

```
% 2000;                          (Main programme)
N01 G54 G91 G71 G94 M03 S800;    (Parameters Setting)
N05 G01 X-15 Z0 F2;              (Facing the job)
N10 G00 Z1;                      (Tool clearance)
N15 G00 X10;                     (Tool clearance from the centre)
N20 G01 Z-36;                    (Turning operation)
N25 G01 X5 –Z30;                 (Taper turning operation)
N30 G00 X1 Z66;                  (Final position of tool)
N35 M02;                         (End of programme)
```

Advanced Part Programming

Several advance technique are used such that a sequence can be programmed just once and given an identity so that it can be called back into the main programme as and when required. These sequences are referred to in a number of ways like cycle, subroutines and loops, etc

A fixed cycle is a combination of machine moves resulting in a particular machining function such as drilling, milling, boring and tapping. By programming one cycle code number, as many as distinct movements may occur. These movements would take blocks of programme made without using Fixed or Canned cycles.

The advantages of writing a part programme with these structures are :

(a) Reduced lengths of part programme.
(b) Less time required developing the programme.
(c) Easy to locate the fault in the part programme.
(d) No need to write the same instructions again and again in the programme.
(e) Less memory required in the control unit.

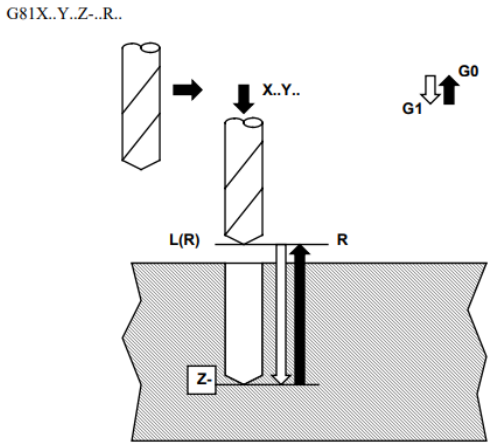
Canned Cycles /Fixed Cycle

The sequence of some machining operations is may be the same for any part and for any machine. For example, drilling a hole

Some Commonly Used Canned Cycle				
Code	***Function***	***Down feed***	***At bottom***	***Retraction***
G81	Drilling	Continuous feed	No action	Rapid
G82	Spot face, counterbore	Continuous feed	Dwell	Rapid
G83	Deep hole drilling	Peck	No action	Rapid
G84	Tapping	Continuous feed	Reverse spindle	Feed rate
G85	Through boring(in & out)	Continuous feed	No action	Feed rate
G86	Through boring(in only)	Continuous feed	Stop spindle	Rapid

G81-DRILL CYCLE-

N-G81X-Y-Z-R-



abs presetting

N01 G92 X-25 Y-25 Z2 EOB

N02 G90 G71 (abs programming, metric programming)

spindle CW

N03 T01 S592 M03

N031 G00 X0 Y0 Z2 EOB

N04 G81 X25 Y25 Z-14 R2 F100

125

cannedcycle start at 'O'

N05 Y100 Drilly at 'A'

N06 X50 Y 62-5 Drilly at 'C'

'B'

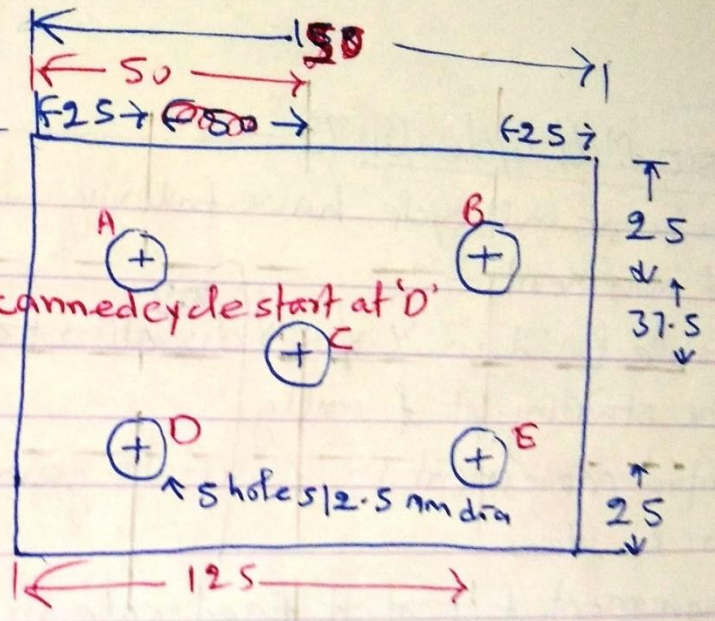
N07 X125 Y100

N08 Y25 Drilly at 'E'

N09 G80 M05 canned cycle off & spindle off

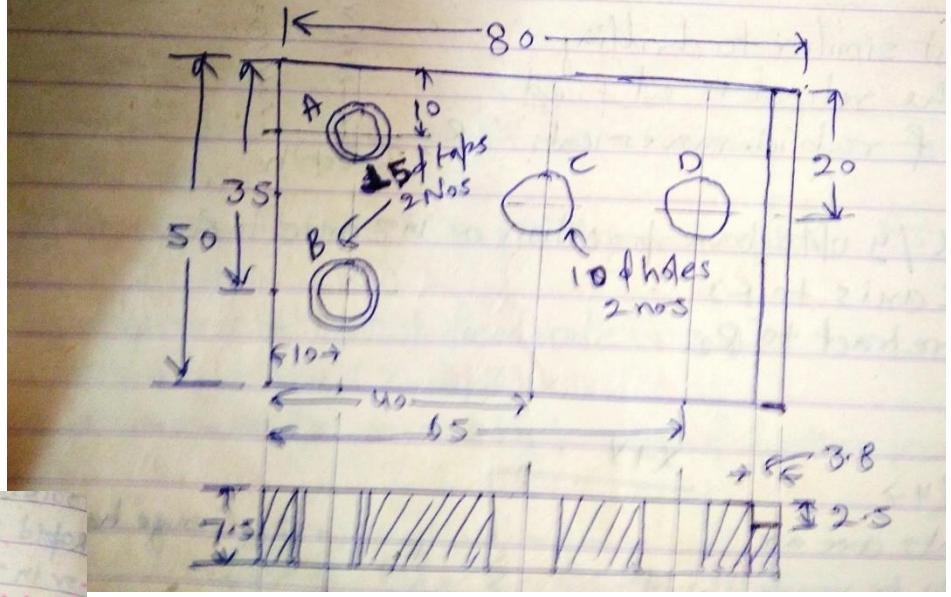
N10 G00 X-25 Y-25 tool goes rapidly to clearance pt

N11 M30 end of program & tape rewind.



BASIC MILL CYCLE G79–

Eg:- Write the Part program for given drawing-




```
N01 G71 G90 G92 X0.0 Y0.0 Z2.0 EOB
N02 G81 X65 Y30 Z-10 R2.0 F48 M03 S800 T01 EOB
N03     X40 EOB
N04     X10 Y15 EOB
N05     Y40 M06 EOB
N06 G81 X10 Y40 Z-10 R2.0 F48 M03 S800 T02 EOB
N07     Y15 M06 EOB
N08 G81 X40 Y30 Z-10 R2.0 F48 M03 S800 T03 EOB
N09     X65 M06 EOB
N10 G84 X10 Y40 Z-10 R2.0 F45 M03 S800 T04 EOB
N11     Y15 M06 EOB
N12 G80 X79.95 Y3.75 Z2.0
N13 G99 Z-2.5 F400 M03 S400 T05 EOB
N14     Y46.25
N15 G80 M02 EOB
```

DO-LOOPS –

In a few jobs some portion of the programme needs to be repeated, which do not fit into standardized category. Some of the non-standardized cycles are Do-loops and Subroutines. Do-loop is a number of operations repeated over a number of equal steps for a previously fixed number of times.

Do-loops always are implemented on incremental mode because each previous position becomes reference for next iteration. Do-loop is actually jumping back to an already written initial portion of the program for the number of times a loop count.

N... G25 Pppp Qqqq Ll
Nppp X/Y/Z
N... X/Y/Z
N... X/Y/Z
Nqqq X/Y/Z

Where: G25—Signals the start of a loop.
P—Specifies the beginning block number of the loop.
Q—Specifies the ending block number of the loop.
L—Specifies the number of times to perform the loop.

SUBROUTINE-

A subroutine is a portion of a programme, complete in itself, which is stored in computer after programming once. It is called with required data when required again in a programme.

CALLING A SUBPROGRAM-

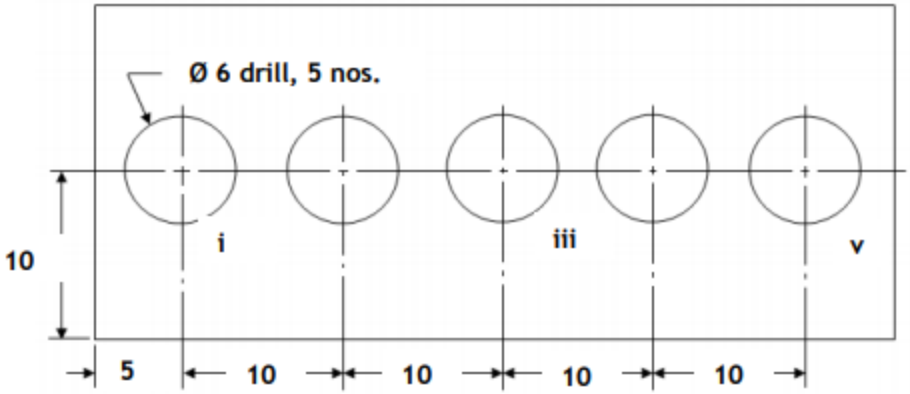
On FANUC and FANUC-style controllers, a subprogram is a program in its own right. It has its own “O” number to identify it, and is sequence numbered independently of the parent program.

The format for calling a subprogram is:

MAIN PROGRAM-
O0001
N001 X/Y/Z
N002 .
N003 .
N004 M98P2000L1
N005 .
N006 .
N007 .
N008 M30

SUBPROGRAM-
O2000
N001 X/Y/Z
N002 .
N003 .
N004 M99

Where M98 — Instructs the MCU to jump to a subprogram.
P2000 — Tells the MCU that O2000 is the subprogram ID.
L1 — Instructs the MCU to execute the subprogram one time




```
% 500;
N2 G71 G90 G94;
N4 G92 X0 Y0 Z0 ;
N6 T1 M06;
N8 G81 G99 X5 Y10 Z-8 R.2 F100 S500 M03 M08;     (Canned Drill cycle)
N10 G51 P4;                                      (Start loop 4 times)
N12 G91 X 10 ;
N14 G50 ;
N16 G80 G90 M09;                                 (Cancel cycle)
N18 T2 M06 ;
N20 G81 G99 X5 Y10 Z-8 R.2 F100 S500 M03 M08;     (Canned Drill cycle)
N22 G51 P4 ;                                     (Start loop 4 times)
N24 G91 X 10 ;
N26 G50;
N28 G80 G90 M09;                                 (Cancel cycle)
N30 M30;
```

%1001;

N2 G17 G71 G94 G90 G54;

N4 T1 L90;

N6 G00 D5 Z5 M3 S500 X9 Y16;

N8 G01 Z0 F500;

N10 L100 P1; (Call the subroutine)

N12 G00 X34 Y16;

N14 G01 Z0 F500;

N16 L100 P1; (Call the subroutine)

N18 G00 X9 Y41;

N20 G01 Z0 F500;

N22 L100 P1; (Call the subroutine)

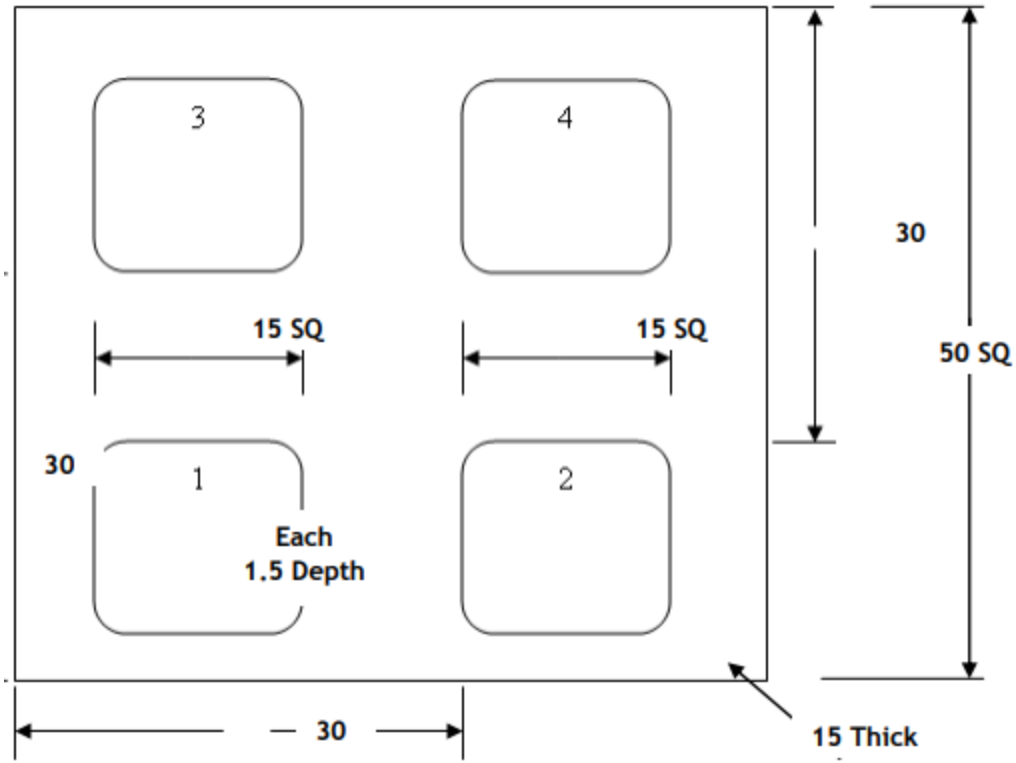
N24 G00 X34 Y41;

N26 G01 Z0 F500;

N28 L100 P1; (Call the subroutine)

N30 G00 Z100;

N32 M02;



Subroutine Programme is below :

L100;

N2 G01 G91 Z-1.5 F100 M7;

N4 X7;

N6 Y-7;

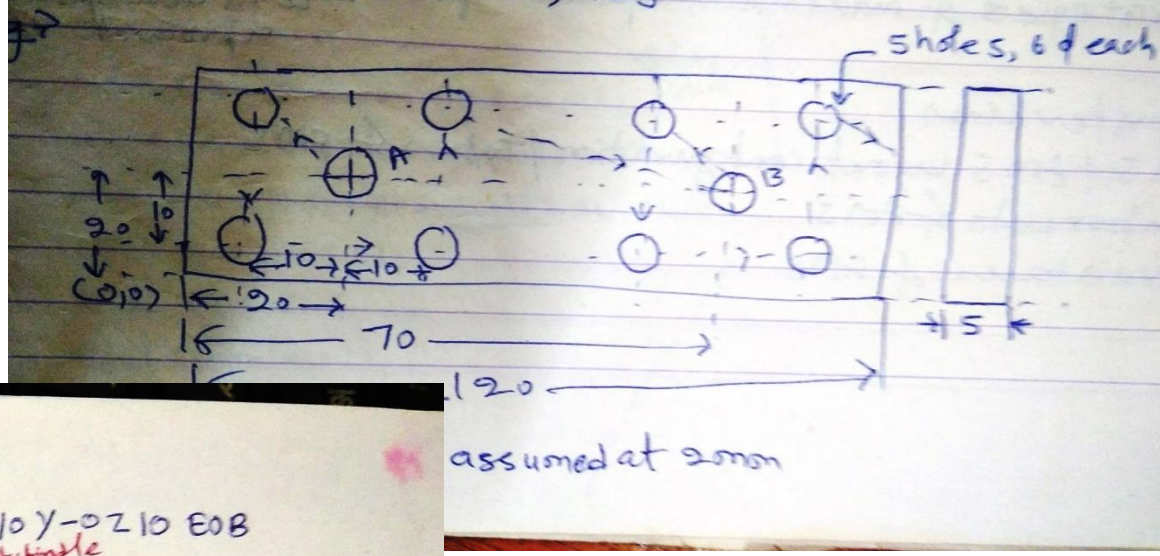
N8 X-7;

N10 Y7;

N12 G00 G90 Z5 M9;

N14 M17;

This has been called as a subroutine in the main programme as above.



assumed at 2mm

main programme

datum offset

metric cancel any previous cycle

set absolute

N001 G99 G71 G80 G90 EOB X-10 Y-0 Z10 EOB

set speed

feed

tool change

N002 S3500 F105 M06 T1 M03 EOB start spindle

rapid

N003 G00 X20 Y20 M08 coolant on - mist

at location P009

N004 P009 M98 call the subroutine of drilling 5 holes

N005 L009 02 call the subroutine '009' two times for location at B

cancel cycle

N006 G80 Z9 M09 coolant off

N007 G00 X-10 Y0 Z2 M05 spindle stop

N008 M30 end of programme

subroutine

N009 O09: start of subroutine

N001 G91 set incremental mode

drill cycle

N002 G81 Z-7 R02 drill hole at that very location

N003 X-10 Y10 drill hole at location '1'

N004 Y-20 at location '2'

N005 X+20 '3'

N006 Y+20 '4'

N007 G80 cancel cyle

N008 G00 X+50 Y-10 move to next relative position 'B'

N009 M99 Return back to main programme