CNC PART PROGRAMMING AND COST ANALYSIS ON VERTICAL MACHINING CENTRE (VTC)

Dr. V.S.S. Murthy[1] P. Sreenivas[2]
[1](Professor and Principal of K.S.R.M *College of Engg, Dept of Mechanical Engineering, Kadapa, Andhra Pradesh, India)*
[2]*(Assistant Professor, Dept of Mechanical Engineering,*
K.S.R.M College of Engg, Kadapa, Andhra Pradesh, India)

ABSTRACT: *In the present study in view of the latest development and revolutionary changes taking place in CNC field through the world, Mechanical elements have to be designed and manufactured to precision, which is perfectly and easily possible through these modern CNC machines. This work is based on the capacity and capability of vertical machining Centre (VTC) with auto tool changer. The top slide which was part programmed can be machined using VTC. And Machining Time is compared in between carbide and hardened tools. The "Top slide" of lathe's called for powerful NC programming technique were used absolute position type data input system using G codes, M codes, polar coordinate programs, circular and linear interpolation, canned cycles etc. The above mentioned component – top slide being manufactured by using various Conventional machine tools like horizontal milling, vertical milling, surface grinding, boring machine and slotting machines. This involved a considerable lead time and usually delayed the assembly schedule. it has been modified and adopted for regular production on this machine, in two setups there by boosting their productivity and ensuring quality in each and every piece. Finally, we can establish for regular production.*

KEYWORDS: *CNC Programming, Machining Time, carbide & hardened Tools.*

I. INTRODUCTION

I.1.NUMERICAL CONTROL: Numerical control (NC) can be defined as a form of programmable automation in which the process is controlled by numbers, letters and symbols. In NC, the numbers form a program of instructions of designed for a particular work part or job.

The definition of NC given by electronic industries association (EIA) is "A system in which actions are controlled by direct insertion of numerical data at some point. The system must automatically interpret at least some portion of this data. "

A Numerical control (NC) system is used when

- The number of components per component is large
- Size of batches is medium
- Labour cost for the component is high
- The component requires special tooling
- Ratio of cutting time to non-cutting time is high
- Design changes are frequent

I.2. BASIC ELEMENTS OF A NC SYSTEM:

An operational numerical control system consists of three basic components

- Controller unit also known as machine control unit (MCU)
- Machine tool or other machining centre
- The program of instructions serves as the input to the controller unit, which in turn commands the machine tool or other process to be controlled.

I.2.1 PROGRAM OF INSTRUCTIONS: The program of instructions is the detailed step-by-step set of directions which instructs the machine tool what to do. It is coded in numerical or symbolic form on some type of input medium that can be interpreted by the controller unit. The input media used can be punched cards/ magnetic disk or tape/punched tape. There are two methods of inputs in the NC system.

- By manual entry of instructional data to the controller unit and this method is called manual data input (MDI) and is appropriate only for relatively simple jobs where the order will not be protected.
- By means of a direct link with a computer. This is called direct numerical control (DNC).

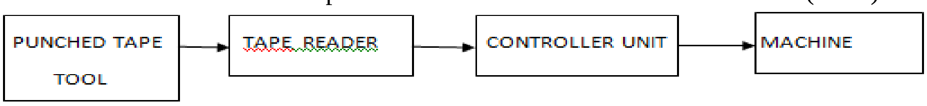


Fig.1 Basic Components of NC system

- Part programmer prepares the program of instructions. The programmer's job is to provide a set of detailed instructions by which the sequences of processing steps are to be performed.
- The processing steps for a machining operation are the relative movement between the cutting tool and the work piece

I.2.2 CONTROLLER UNIT: The controller unit consists of the electronics and hardware that reveals and interprets the program of instructions and converts it into mechanical actions of the machine tool. The controller unit elements are tape reader, a data buffer, signal output to the machine tool Feedback, channel from the machine tool and data decoding control area.

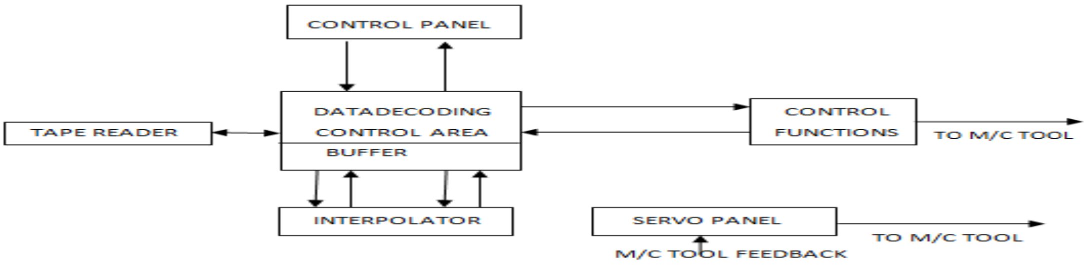


Fig 2. Machine controller unit

I.3. NUMERICAL CONTROL (NC) PROCEDURE:

The basic steps in NC procedure to utilize NC in manufacturing are

- Process planning
- Part programming
- Part program entry/ tape preparation
- Proving the part programs/ tape verification
- Production

I.3.1. PROCESS PLANNING: Process planning is the procedure of deciding what operations to be carried on the component, in what order and with what tooling and work holding facility. Both process planning and part programming for manufacturing occur after the detail drawing a component has been prepared.

1.3.2. PART PROGRAMMING: In part programming, sequence of steps to be performed by NC planned and documented. There are two methods by which a part program is accomplished manual part programming and computer assisted part programming.

In manual part programming the relative cutter/work piece positions which must be following to machine the part are listed in a format known as part program manuscript. For complex work piece geometries and jobs with many machining steps, computer-assisted part programming is used.

I.4 TOOLINGS FOR CNC (VERTICAL MACHINING CENTRE): The modern machine tools are designed to operate at higher speeds and feeds. They possess improved accuracy, higher rigidity and reduced noise levels. The cost of raw material input is very high-of the order of 40% for general purpose machine tools. This calls for optimizing the design of machine elements, selecting the right type of materials, judiciously imparting effective fabrication and treatment method.

I.5 GENERAL PRINCIPLES IN THE SELECTION OF MATERIALS FOR MACHINE TOOLS:

I.5.1 FUNCTIONAL REQUIREMENTS: The functional requirements must be met in terms of various properties. For example, in the selecting material for the main spindle of a machine tool, the modulus of elasticity and the surface hardness required for the spindle nose, bore and the locations of the bearings are important properties which need to be considered. Generally, low nickel-chromium alloy case carburized steel such as 15CrNi6 (as per DIN 17210) is selected, which meets the functional requirements.

1.5.2 EASE OF FABRICATION: The process of fabrication should be such that the part or component should be easy to make. If it is required in batch quantity, casting process is adopted. For example, machine tool elements or parts such as bed, headstock, etc. required in batch quantity are made out of casting process in the foundry. If the requirement is one or two numbers, a welding process is used to fabricate the part.

1.5.3 MACHINABILITY: This is another important parameter to be considered for selecting the raw material of machine tool components as extensive machining is involved. Construction steels such as medium carbon steel (C45 as per DIN 17200) and low alloy steel (15CrNi6 as per DIN 17210, 36CrNiMo4 as per DIN 17200 and 34CrAIMo5 as per DIN 17211) are chosen for many of the parts which have good machinability. In case of castings, grey cast iron is selected.

1.5.4. COST: Since the raw material cost plays a significant role in the overall cost of the machine tool, it becomes as important factor to be considered.

1.5.5. AVAILABILITY: The chosen material must be easily available so that the cost and delivery time are kept low. In fact, all the raw materials required for machine tools are easily available in India.

1.5.6. MATERIALS FOR CUTTING TOOLS: One of the main qualities that a cutting tool must possess is that it retains it's hardness at high temperatures generated during the cutting process. The most common cutting tool materials used in CNC application or HSS sintered carbides.

II. CANNED CYCLES

A canned cycle (fixed cycle) defines a series of machining sequences for drilling, boring, tapping.

The canned cycles G81 to G89 are stored as subroutines L81 to L89.

The user may deviate from a standard fixed cycle and redefine it to suit his specific machine or tooling requirements. The parameters R00 to R11 are used by subroutines to define the variable values necessary to correctly execute a fixed cycle prior to a subroutine call; all necessary parameters must be defined in the main program.

A fixed cycle call is initiated with G80 to G89. G81 to G89 are fixed cycles that are cancelled with G80. A boring cycle can be called with L81 to L89, however, L81 to L89 are not model. L81-L89 is performed only once in the block in which it is (notable tungsten carbides), ceramic and polycrystalline diamond. High speed steel is tougher than cemented carbide but not so hard and therefore, cannot be used at such high rate of metal removal, not suitable from higher cutting speeds.

The hardness of the cemented carbide is almost equal to that of diamond. It deserves this hardness from its main constituent, tungsten carbide. In its pure form tungsten carbide is too brittle to be used as a cutting tool. So it is pulverized and mixed with cobalt. The mixture of tungsten carbide and cobalt powder is pressed into the required shape and then sintered. The cobalt metal binds the tungsten carbide gains in to a dense, non-porous structure.

In addition to tungsten carbide, the other metals as titanium and titanium carbides (TIC) are used and by providing tungsten carbide tool with a thin layer of titanium carbide tool, resistance of to wear and useful life are increased up to 5 times. Programmed. At the end of a fixed cycle the tool is re-positioned at the starting point.

II.1 USING CANNED CYCLES IN PROGRAMS

CALL-UP G81 (DRILLING, BORING, CENTERING, BORING AXIS Z)

```
N8101 G90 S48 F460 LF                   - Spindle ON
N8102 G00 D01 Z500 LF                   - Active tool offset
N8103 X100 Y150 LF                      - First drill position
N8104 G81 R02 360 R03 250 R11 3 LF      - Call cycle
N8105 X250 Y300 LF                      - Second drill position and automatic G81 call
N8110 G80 Z500 LF                       - Cancelling G81 and returning to  Starting plane
```

CALL-UP WITH L81:

```
N8101 G90 S48 F460              LF
N8102 G00 D01 Z500              LF
N8103 X100 Y150                 LF
N8104 L81 R02 360 R03 250 R11 3 LF – Call up drilling cycle first hole
N8105 X250 Y300                 LF
N8106 L81 R02. . .              LF – Call- up drilling cycle, second hole
N8107 Z500                      LF
```

As opposed to the call-up with G81, here the drilling cycle must be called up a new at every new drill position

CALL – UP G82 (DRILLING, COUNTER, SINKING)

```
N8201 …   M03 F460                     LF
N8202 G00 D01 Z500                     L F
N8203 X100 Y150                        LF
N8204 G82 R02 360 R03 250 R04 1 R11 3  LF
N8205 X250 Y300                        LF
N8206 G80 Z500                         LF
```

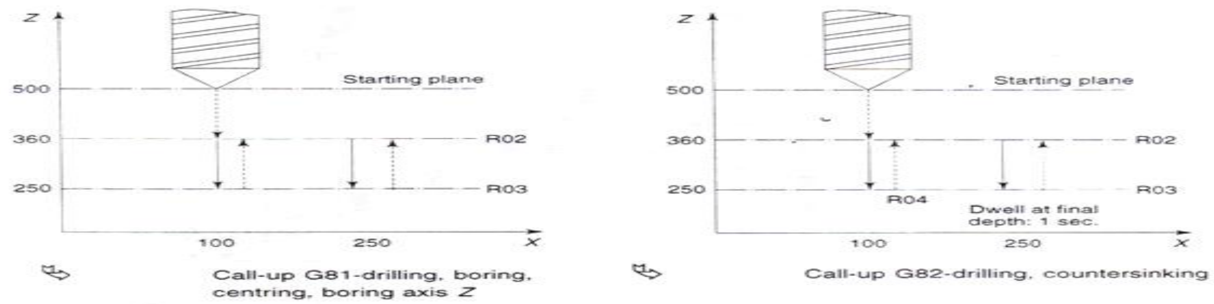


Fig 3

CALL – UP G83 (DEEP HOLE DRILLING)

First drilling depth	50 mm	R01 50
Reference plane = retract plane	146 mm	R02 146
Final drilling depth	5 mm	R03 5
Dwell at starting point	5 s	R00 5
Dwell at final depth	1 s	R04 1
Degression value	20 mm	R05 20
Drilling axis (z)	3	R11 3

```
N8301    …  S48 M03 F460                   LF
N8302 G00 D01 Z500                         LF
N8303 X100 Y150                            LF
N8304 G83 RO1 50 R02 146 R03 5 R00 5
           RO4 1   R05   20   R11   3      LF
N8305  X250  Y300                          LF
N8306  G80  Z500                           LF
```

At the rapid traverse advance with respect to the new drilling depth, a safety distance of 1mm is kept (on account of the chips still remaining in the hole). With the inch system (G70) the safety distance must be changed accordingly.

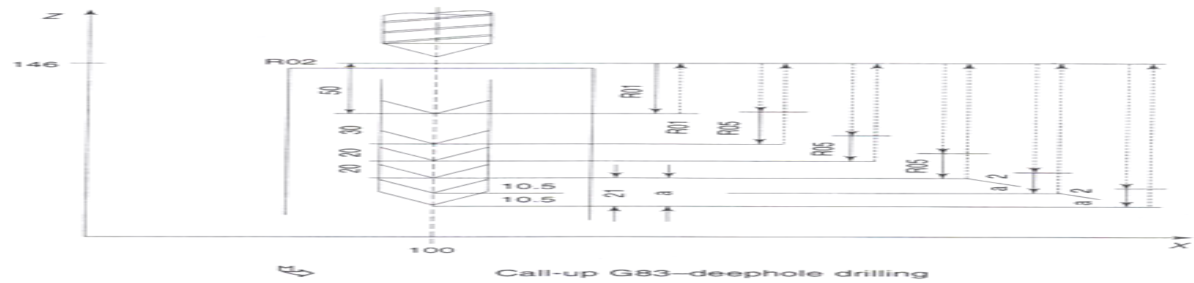


Fig 4

CALL – UP G84 (tapping cycle):-
N8401 . . . S48 M03 F460 LF
N8402 G00 D01 Z500 LF
N8403 X100 Y150 LF
N8404 G84 R02 360 R03 340 R06 04
R07 03 R11 3 LF
N8405 X250 Y300 LF
N8406 G80 Z500 LF

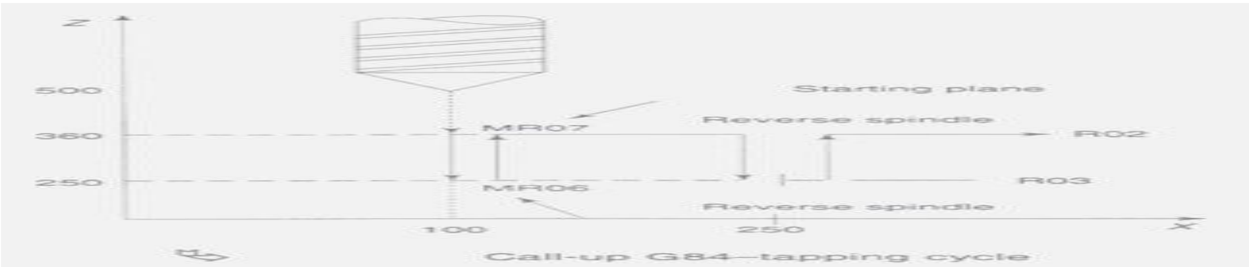


Fig 5

CALL – UP G85 (BORING 1) :
N8501 . . . S48 M03 F460 LF
N8502 D00 D01 Z500 LF
N8503 X100 Y150 LF
N8504 G85 R02 360 R03 250 R10 380 R11 3 LF
N8505 X250 Y300 LF
N8506 G80 Z500 LF
CALL – UP G86 (BORING 2):-
N8601 . . . S48 M03 F460 LF
N8602 G00 D01 Z500 LF
N8603 X100 Y150 LF
N8604 G86 R02 360 R03 250 R07 03
R10 380 R11 3 LF
N8605 X250 Y300 LF
N606 G80 Z500 LF

CALL – UP G87 (BORING 3):
N8701 . . . S48 M03 F460 LF
N8702 G00 D01 Z500 LF
N8703 X100 Y150 LF
N8704 G87 R02 360 R03 250 R07 03 R11 3 LF
N8705 X250 YY300 LF
N8706 G80 Z500 LF

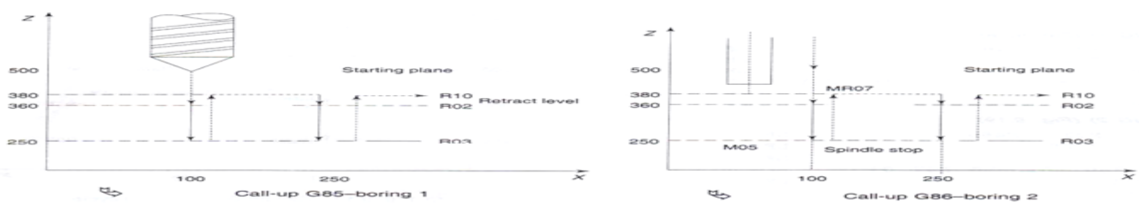


Fig 6

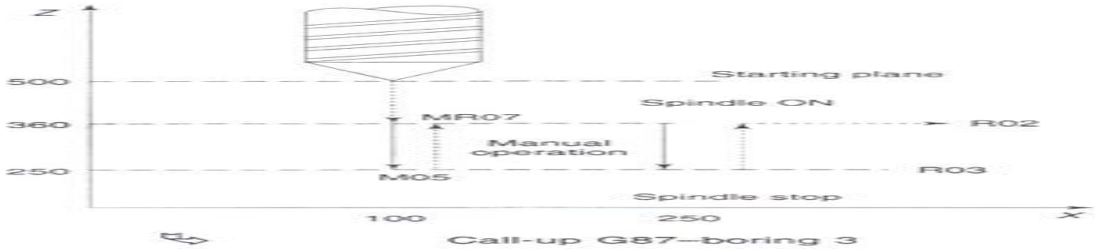


Fig 7

CALL – UP G88 (BORING 4):
N8801 . . . . . S48 MO3 F460 LF
N8802 G00 D01 Z500 LF
N8803 X100 Y150 LF
N8804 G88 R02 360 R03 250 R04 1 LF
R07 03 R11 3
N8805 X250 Y300 LF
N8806 G80 Z500 LF
CALL-UP G89 (BORING 5):-
N8901 . . . . S48 M03 F460 LF
N8902 G00 D01 Z500 LF
N8903 X100 Y150 LF
N8904 G89 R02 360 R03 250 R04 LF
1 R1103
N8905 X250 Y300 LF
N8906 G80 Z500 LF

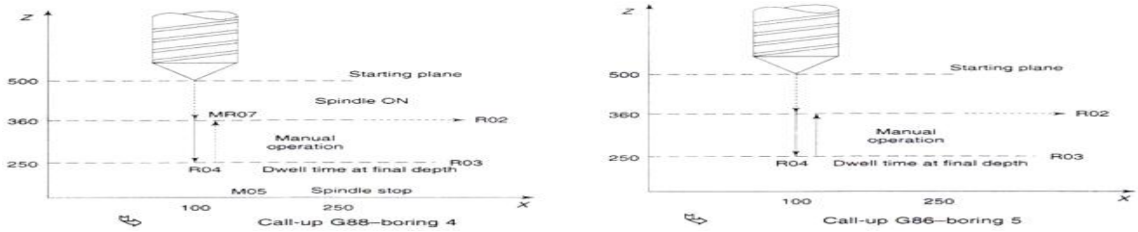


Fig 8

II.2 CALLING BORING CYCLES IN A SUBROUTINES:

If boring cycles are called in a subroutine, the following procedure is necessary:
%1R02 360 R03 250 R00 81 R11 3 LF- supply boring cycle parameters
L0101 LF-boring positions
M30* LF
L0101 (Boring positions)
GR00 X1 Y1 LF-First boring location
X2 Y2 LF-Second boring location
X5 Y5 LF-Third boring location
X10 LF-Fourth boring location
G80 M17 LF-Deselect boring cycle and end of subroutine
R02 360 R03 250 R00 81 R11 3 LF- supply boring cycle parameters
L0101 LF-boring positions
M30* LF
L0101 (Boring positions)
GR00 X1 Y1 LF-First boring location
X2 Y2 LF-Second boring location
X5 Y5 LF-Third boring location
X10 LF-Fourth boring location
G80 M17 LF-Deselect boring cycle and end of subroutine

II.3 POLAR COORDINATES G10/G11/G12/G13: Drawing dimensioned with an angle and radius can be entered directly in the program with the aid of the polar coordinates.
The following preparatory functions are available for Programming with coordinates:-
G10 Linear interpolation, rapid traverse
G11 Linear interpolation, federate (F)
G12 Circular interpolation, clockwise

G13 Circular interpolation, counter clockwise.

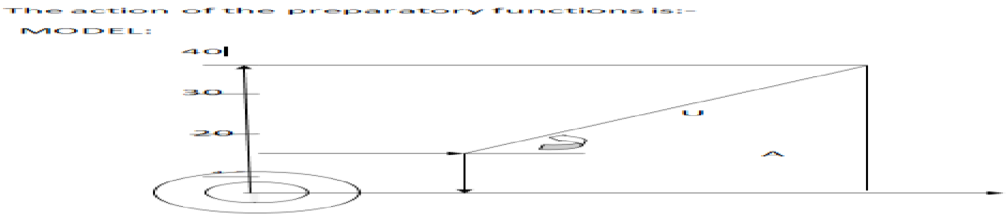


Fig 9

W - Work piece zero
M - Centre point of polar coordinate system
A - Angle
U - Radius

In order to determine the traverse path , the control requires the centre point, the radius and the angle .the centre point is entered with perpendicular coordinates (X,Y,Z) and on initial programming using absolute position data . A subsequent incremental position data input (with G91) always refers to the last centre point programmed.

2.3.1. Polar coordinates g110/g111: The functions G110 and G111 are used to adopt a new centre point or zero point when programming polar coordinates.
Using the new centre point, the angles are again taken from the horizontal and the radius is calculated from the new centre point. G110 and G111 have the following meanings.
G110 Adopt the set point reached as the new centre point
G111 Centre point programming with angle and radius without axis movement
(Example: setting the arc centre of a hole circle)
The following traversing movement must be programmed using G110
Ex: polar coordinates G110
Z385
(G110 polar coordinates) Lf
N05 G90 G 10 X0 Y0 U0 F1000 Lf
N10 G11 U30 A45 Lf
N15 G11 U20 A30 Lf
N20 M30 Lf

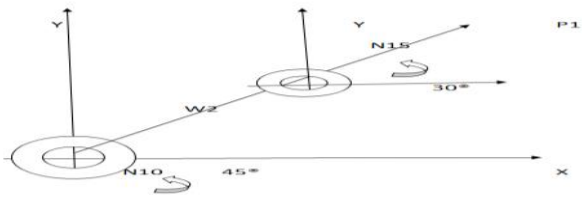


Fig 10

Feed rate F, G94/G95/G98
The federate F is programmed in mm/min or mm/rev
G94 F federate in mm/min
G95 F federate in mm/rev (is assigned to the leading spindle)
G98 F federate in rev/min (for rotary axes only)

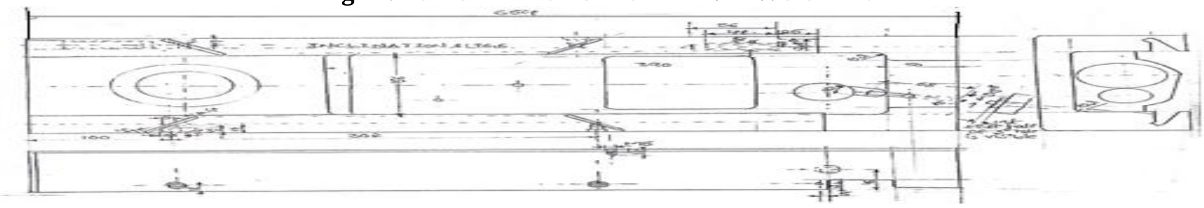
II.3.2.Thread cutting G33/G34/G35: Threads can be cut both on drilling or boring and milling machines with a boring tool or a facing tool. These are various types of thread which can be cut as follows:

i. Threads with a constant lead
ii. Threads with a variable lead
iii. Single or multiple threads
iv. External or internal threads

The following preparatory functions are available for machining threads;

G33 threads cutting with constant lead
G34 thread cutting with linear lead increases
G35 thread cutting linear lead decreases

Fig 11.TOP SLIDE SECTIONAL VIEWS SET UP -1



SCALE 1:3

FIG 12. Cross section of top slide for setup-1

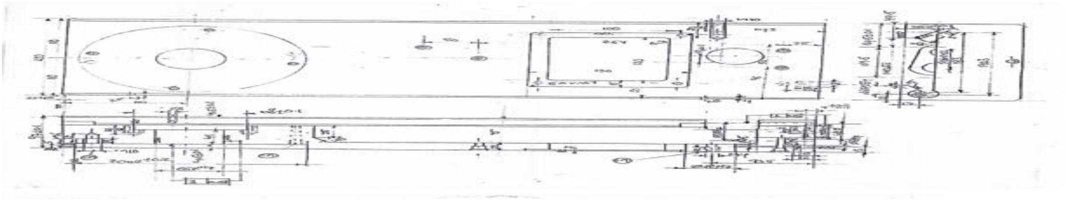


Fig 13. Cross section of top slide for setup-II

TOOL	DIAMETER
BW drill	60
Boring bar	64.5
BW drill	38
Boring bar	21.8
Inserted type end mill	25
Three lip end mill	12
Boring bar(Finishing)	65
Boring bar(finishing)	22
Shoulder milling cutter	63

III.1. MACHINE SPECIFICATIONS

VMC	1200
SOFTWARE	SIEMENS
AXIS MOVEMENT	3
TOOLPOST CARRYING CAPACITY	24
X-AXIS	1200 MM
Y-AXIS	600 MM
Z-AXIS	600 MM
GUIDEWAYS	T-SLOT BED
SPEED MAXIMUM	600 R.P.M
SPEED MINIMUM	50 R.P.M
FEED RATE MM/MIN	10 TO 250

III.2. PART PROGRAM:

III.2.1 SETUP – I:

```
N5    L90; ……. [DIA60 CORE DRILL]
N10   T1
N15   M06
N20   G54   S200   M3   F30   D1
N25   G0   X0   Y0
N40   CYCLE 82   (150, 0, 2, -90, 1 )
N50   MCALL
N55   M0
N60   L90: ….. [DIA64.5   SIF   B. BAR]
N65   T2
N70   M06
N75   G54   S300   F30   D1
N80   G0   X0   Y0
N95   CYCLE 86 [DIA 38   BW   DRL]
N105   MCALL
N110   M0
N115   N90; …. [DIA38 BW DRL]
N120   T3
N125   M06
N130   G54   S350   M03   F35   D1
N135   G0   X453.5   Y18.5
N150   CYCLE82 (150, 0, 2, -28, , 1)
N160   MCALL
N170   L90; …. [DIA   41.5   SIF   B.BAR]
N175   T4
N180   M06
N185   G54   S400   M03   F30   D1
N190   G0   X453.5   Y1895
N205   CYCLE 86 (200, 0, 2, -28, , 2, 3, , , )
N205   MCALL
```

```
N215  M0
N220  M0
N225  L90; …. [DIA 63  S/MILL]
N230  T5
N235  M06
N240  G54  S350  M03  F150  D1
N245  G0  X0  Y0
N250  R20 = 27  R21 = 3  R22 = 14.7
N255  MS1: G0  Z = -R20
N260  G01  X-50
N265  G02  X-50  Y0  I50  J0
N270  G01  X-67.5
N275  G02  X-67.5  Y0  I67.5  J0
N280  G01  X0  Y0
N285  IF  R20 == R22  GOTOF  MS2
N290  R20 = R20+R21
N295  G0T0B  MS1
N300  MS2  GO  Z200
N305  M0
N310  L90; ….[DIA25 END MILL]
N315  T6
N320  M06
N325  G54  S800  M03  F100  D1
N330  TRANS  X348  Y0
N335  GO  X-25  Y0
N340  Z-3.5
N345  G01  X-52.5
N350  Y67.5
N355  X52.5
N360  Y-67.5
N365  X-52.5
N370  Y0
N375  X-25
N380  TRANS  X0  Y0
N385  G0  Z200
N390  M0
N395  L90; ….[DIA12 END MILL]
N400  T7
N405  M06
N410  G54  S1000  M03  F100  D1
N415  GO  X-40  Y0
N425  Z-3.5
N430  G01  X-59
N435  Y74
N440  X59
N445  Y-74
N450  X-59
N455  Y0
N460  X-40
N465  TRANS  X0  Y0
N470  G0  Z200
N475  M0
N480  L90; ……[SPOT]
N485  T8
N490  M06
N495  G54  S1000  M03  F100  D1
N500  G0  X509  Y-19
N515  MCALL  CYCLE  82 (10, 0, 2, - 3.5, , 1)
N525  X509  Y-19
N530  Y31
N535  X242  Y0
N540  X192  Y31
N545  MCALL
N550  G0  Z200
N555  M0
N560  L90; …..[DIA200  SIF  BORE]
N565  T9
N570  M06
N575  G54  S100  M03  F10  D1
N580  G0  X0  Y0
N595  CCYCLE86 (200, 0, 2, -15, , 2, 3, , , , )
N605  MCALL
N610  M0
N615  L90; …… [DIA 42 H7]
N620  T10
N625  M06
N630  G54  S400  M03  F30  D1
N635  G0  X0  Y0
N650  CYCLE86 (150, -12, 2, -85, , 2, 3, , ,)
N660  MCALL
N665  M0
N670  L90; …… [DIA 42H7]
N675  T11
N680  M06
N685  G54  S600  M03  F30  D1
N690  G0  X453.5  Y18.95
N705  CYCLE86 (150, 0, 2, -28, , 2, 3, , , ,)
N715  MCALL
N720  M0
N725  L90; …. [DIA 100  SIM]
N730  T12
N735  M06
N740  G54  S300  M03  F200  D1
N745  G0  X-175  Y70
N750  Z0
N755  G01  X610
N760  G0  Y0
N765  G01  X-175
N770  G0  Y-70
N775  G01  X610
N780  G0  Z200
N785  M30
= = eof = =
```

III.2.2.SETUP - II

```
N5  L90; …..[DIA 63  S/M]
T5
M06
N10  G54  S400  M03  F200  D1
G0  X-55  Y202
Z-1
G01  X570
Y28
X-10
Y-40
N15  G0  X700  Y130
Z0
G01  X560
G0  Z250
M0
N16  L90; …..[SPOT]
T8
M06
N20  G54  S1000  M03  F50  D1
G0  X100  Y17
```

```
MCALL   CYCLE   82 (10, 0, 2, -3.5, , 0)
X100   Y17
X402
Y213
X100
MCALL
G0   Z200
M0
N30   L90; …..[OIL  GROVING]
T14
M06
N40   G54   S1000   M03   F100   D1
G111   X100   Y17
RP = 17.5   AP = 225   F = 100
G0   Z0
G01   Z-17
G111   X100   Y17
AP = 45   RP = 29
G0   Z2
G111   X402   Y17
RP = 17.55   AP = 315
G01   Z-1.7
G111   X402   Y17
RP = 29   AP = 13.5
G0   Z2
G111   X402   Y213
RP = 17.5   AP = 45
G01   Z-1.7
G111   X402   Y213
AP = 225   RP = 22.5
G0   Z2
G111   X100   Y213
RP = 17.5   AP = 135
G01   Z-1.7
G111   X100   Y213
AP = 315   RP = 29
G0   Z300
M0
N50   L90; ……..[DIA 63   S/MILL]
T5
M06
N60   S400   M03   F200   D1
G0   X-35   Y155.22
R20 = 4.4   R21 = 3   R22 = 25.4
MS1: G0   Z = -R20
G01   X0
X558   Y162.52
X563
G0   Y100
G01   Y74
X-35
G0   Y155.22
IF   R 20 == R22   GOTOF   MS2
R20 = R20 + R21
GOTOB   MS1
MS2: GO   Z100
N70   GO   X-40   Y115
Z-20
G01   X260
GO   Z-25.4
G01   X-45
G0   Z-25
N70   TRANS   X110   Y115
G0   X0   Y0
R20 = =29.4   R21 = 4   R22 = 41.4
PR1: G0   Z = -R20
G01   X17.5
G02   X17.5   Y0   I-17.5   J0
G01   X0   Y0
IF   R20 ==R22   G0T0F   PR2
R20 == R20 + R21
GOTOB   PR1
PR2: TRANS   X0   Y0
G0   Z200
M0
N70   L90; …..[50  DOVETAIL]
T15
M06
N80   G54   S80   M03   F50   D1
R20 = 74   R21 = 2   R22 = 66
XY:   X-45   Y = R20
Z-25.5
G01   X562
Y90
G0   X550
Z80
IF R20 == R22   G0TOF   XY2
R20 = R20-R21
GOTOB   XY1
XY2: M0
N90   X65.5
Z-25.5
G01   X562
Y90
G0   X550
Z80
N100   R20 = 5   R21 = 2   R22 = 63
X21: TRANS   Y = R20
G0   X-45   Y144.72
Z-25.5
G01   X0
X558   Y152.005
X562
TRANS   Y0
Y150
G0   X550
Z80
IF   R20 ==R22   GOTOF   XZ2
R20 = R20+R21
GOTOB   XZ1
MZ 2:  M0
N110   TRANS   X0   Y0
G0   X-55   Y115
Z-25.5
G01   X260
G0   Z300
M30
N90   L90; …[DIA 100   S/F]
T16
M06
N90   G54   S200   M03   F30   D1
G0   X110   Y115
CYCLE 86 (150, -24, 2, -41.5,  , 2, 3,  ,  ,  ,  ,  )
MCALL
M0
N100   L90
```

```
T17
M06
N110   S600   M03   F150   D1
G0   X574   Y-20
R20 = 8   R21 = 8   R22 = 32
JK 1: G0   Z= -R20
G01   Y70
N800   G01   Y701
N805   G0   Z10
N810   Y-20
N815   IF  R20 == R22   G0T0F   JK2
N820   R20 = R20 + R21
N825   G0T0B   JK1
N830   JK2:  G0   Z10
N835   G0   X574   Y12
N840   Z32
N845   G01   X674
N850   G0   Y40
N855   G01   X574
N860   G0   Z300
N865   Y400
N870   M0
N875   G0   X574   Y250
N880   R20 = 8   R21 = 8   R22 = 32
N885   JH1:  Z = -R20
N890   G01   Y175
N895   G0   Z10
N900   Y250
N905   IF  R20 == R22  G0T0F   JH2
N910   R20 = R20 + R21
N915   GOTOB   JH1
N920   JH2: G0   Z10
N925   G0   X574   Y250
N930   Z-32
N935   G01   Y218
N940   X674
N945   G0   Y188
N950   G01   X580
N955   G0   X200
N960   M30
= =eof = =
```

IV. MACHINIG TIME

IV.1 INTRODUCTION

Machining process converts raw material into useful finished product, surface finishing is needed to the foundry castings certain amount of material is added as a machining allowance for this purpose the size of the casting should be slightly over size than the dimensions shown on the finished drawings the machining operations generally performed on vertical machining center are:

- Drilling
- Boring
- Shaping
- Grinding
- Reaming
- Milling etc.

IV.2 PURPOSE OF ESTIMATING MACHINING TIME:

Estimation of machining time for different processes is required for the following processes:

To estimate the manufacturing time
To fix the delivery dates
To determine the cost of labour charges
To find out the cost of manufacturing different parts

IV.3 MACHINING TIME

Estimation of machining time means calculation of time required to finish the given component according to the drawings supplied after giving number of allowances in addition to the actual time taken for machining operations certain amount of extra time is given to the workers. They are:

- Setup time
- Handling inspection of jobs
- Team down time
- Fatigue allowance
- Tool changing allowance
- Measurement checking allowance
- Other allowances for cleaning
- Getting stock etc.

There for total machining time is the actual time for machining and all the time allowances as given above.

To calculate actual machining time the basic general formula used is

Machining time = length of cut / (feed × rpm)

IV.4 CUTTING SPEED:

The cutting speed of a cutting tool may be defined as the speed at which the cutting edge passes over the material. Cutting speed is generally expressed in m/min.

An estimator should consider the following while selecting a suitable cutting speed.

Low cutting speeds are required for hand materials.

- High speed steel cutting tools content high speeds and can bide tipped tools cut still higher speeds.
- If the depth of cut and feed is more/less cutting speed may be taken and vice-versa

- Cutting speeds can be increased by using good cutting fluids and coolants.
- The amount of stock removed in inversely proportional to the cutting speeds when stock is removed at high speeds.
- If the cutting speed increases the heat generated also increases and tool life decreases.

Cutting speed is given by the formula:
S = (πDN) / 1000 meters/min

IV.5. MACHINING TIME FOR CARBIDE TOOLS:

4.5.1. SETUP – I:

For DIA60mm core drill,
Cutting speed (s) =π DN/1000= (1000 × 45)/(π ×60)= 200 rpm
Feed rate (F) = feed/tooth × no. of teeth × Rpm = 0.1 × 2 × 200 F = 30 mm/min
M/C time (or) operating time = length/feed= 92/30
Time = 4 min

For DIA 64.5 S/F B. BAR
Operating time = length/feed = 92/30 T = 4 min
For DIA 38 between DRL
C/s =πdn/1000
N = (45 × 1000) / (π ×38) N = 350 rpm (spindle speed)
Feed = 0.1× 1×350 = 35 mm/min
Operating time = 30/35 (length/feed) T = 1 min

For DIA 41.5 S/F B. BAR
C/s = πdn/1000
N = (1000 × 45) / (π × 41.5) = 300 rpm
F = 0.1 × 1 × 300= 30 mm/min
Operating time = length / feed= 30/30 = 1 min

For DIA 63 S/MILL S/s = 350 rpm [R = 67.5min]
Feed = 150 mm/min [no. of passes = 5]
Operating time = 2πR × no. of passes / feed= 2π×67.5×5/150= 15 min

For DIA 25 END MILL s/s = 800 rpm
Feed = 100 mm/min
Operating time = length/feed= 480/100 T = 5 min

For DIA 12 END MILL S/s = 1000 rpm
Feed = 100 mm/min
Operating time = length / feed= 532 /100 T = 6 min

For DIA 200 S/F BORE s/s = 100 rpm
Feed = 10 mm/min
Operating time = length /feed = 17 /10= 2 min

For DIA 42 H_7 S/S = 400 rpm
Feed = 30 mm/min
Operating time = length / feed= (87+30)/30 T = 4 min

For DIA 100 S/M S/s = 300 rpm
Feed = 200 mm/min
Operating time = length/feed= 1850/200 T = 10 min

TOTAL OPERATING TIME FOR SETUP – 1
= 4+4+1+1+15+15+6+2+4+10 T = 52 min

8.5.2.SETUP-II

For DIA 63 S/M S/s = 400 rpm
Feed = 200 mm/min
Operating time = length / feed= 1218 /200 T = 7 min
For spot
T = 5.5/50 = 1 min
For oil grooving
Feed = 100 mm/min
Time = length / feed= [(29 + 17.5)/100] × 4= 2 min
For 63 s/m
F = 200 mm/min
Operating time = length/feed= (length × no. of passes) / feed
= [(558×2 + 162×2)/200] × 8 T = 58 min
Total length = 2πR × no. of passes= 2π × 17.5 × 4
Time = (2π × 17.5 × 4)/200= 3 min

For 50 DOVETAIL S/s = 80 rpm
Feed = 50 mm/min
Operating time = (total length × no. of passes)/feed= [(607+90-74)×5]/50 T = 62 min
Operating time = [603+ (152.005-144.72)×4]/50 T = 49 min
Operating time = (260+55+155)/50 T = 8 min

For DIA 100 S/F S/s = 200 rpm
Feed = 30 mm/min
T = 43.5/30=2 min
No. of passes = 4
Total time = (574+70+20)*4/150= 17 min
IF R_{20} == R_{22}, S/s= 600 rpm
F= 150 mm/min
Total time = (total length × no. of passes)/feed= [((674+250)×2)/150]×4= 49 min

TOTAL OPERATING TIME FOR SETUP-II
= 7+1+2+58+3+3+62+49+8+2+17+9+49 T = 270 min

Total operating time for both setup-I & II = 270+52 = 322 min

IV.6. MACHINING TIME FOR HARDEND TOOLS

If the cutting tool is H.S.S/HARDEND cutting speed = 15 m/sec

4.6.1.SETUP-I

For DIA 60 core drill
C/s = πDN/1000
N = (1000×15) / (π×60)
N = 50 rpm
Feed = (feed/tooth)× no. of teeth's× rpm=0.1×2×50
F = 10 mm/min
Operating time = length / feed= 92/10 = 10 min

For DIA 64.5 S/F B. BAR s/s = 50 rpm
F = 10 mm/min
Operating time = length/feed = 92/10= 10 min

For DIA 38 BW DRL
C/s = πDN/1000 N = (1000×15)/ (π×38)= 120 rpm
Feed = 0.1×1×120= 12 mm/min
Operating time = 30/12= 3 min

For DIA 41.5 S/F BAR
C/S = πDN/1000N = (1000×15)/ (π×41.5)= 100 rpm
F = 0.1×1×100= 10 mm/min
Operating time = length/feed= 30/10 =3 min

For DIA 63 S/MILL
Spindle speed = 120 rpm
Feed = 50 mm/min
Operating time = (2πR× no. of passes)/feed= (2π×67.5×5)/50= 42 min

For DIA 25 END MILL
Spindle speed = 300 rpm
Feed = 30 mm/min
Operating time = length/feed= 480/30= 16 min

For DIA 12 END MILL
Spindle speed = 350 rpm
Feed = 35 mm/min

Operating time = 532/35 = 16 min

For DIA 200 S/F BORE

Spindle speed = 30 rpm

Feed = 5 mm/min

Operating time = length/feed= 17/5 = 4 min

For DIA 42 H_7.

Spindle speed = 150 rpm

Feed = 15 mm/min

Operating time = (87+30)/15 = 8 min

For DIA 100 S/M

Spindle speed = 100 rpm

Feed = 50 mm/min

Operating time = length/feed= 1850/50 = 37 min

Total operating time for setup –I

= 10+10+3+3+42+16+5+4+8+37= 154 min

IV.6.2 SET UP-II:

For DIA 63 s/m

Spindle speed = 150 rpm

Feed = 50 mm/min

Operating time = length/feed= 1218/50= 25 min

For spot

T= 5.5/20= 1 min

For oil grooving

Feed = 30 mm/min

Time = length/feed= [(29+17.5)/30]×4= 8 min

For 63 S/M F= 50 mm/min

Operating time = (length×no. of passes)/feed= (558×2+162×2)8/60= 230 min

Total length = 2πR ×no. of passes= 2π×17.5×4

Time = (2π×17.5×4)/80= 7 min

No. of passes = 4

Total time = (570+70+20)×4/40=67 min

Spindle speed = 200 rpm

Feed = 50 mm/min

Total time = [(674+250)×2 ×4]50= 147 min

Total time for setup-II= 25+1+8+230+7+160+145+22+5+67+147= 817 min

Total time for both setups I & II = 148+817= 965 min

Total machining time for carbide tools = 322 min

Total machining time for hardened tools = 965 min

By observing the carbide & hardened tools

The machining time for hardened tools is 3 times greater than the carbide tools

IV.7. COST ANALYSIS:

Given

M/c hour rate = Rs.800

For carbide tools total manufacturing cost for 4 pieces = (322×800×40)/60= Rs.17, 173

For Hardened tools total manufacturing cost for 4 pieces = (965×800×4)/60= Rs.51, 466

By using carbide tools, we saved Rs.34, 293 for 4 pieces.

IV. CONCLUSION

The ‘TOP SLIDE’ of horizontal lathe was machined successfully in two set ups on vertical set ups on vertical machining centre (VTC). It has been realised that CNC programming technique is so powerfull,that components of intricate profile and tight dimensional tolerance are machined with the sophisticated machine tool like VTC Machining of such components give way to a compact and revolutionary changes in industrial product. Concept of interchagibility is being felt verymuch .thus,maintance becomes very simple and economical.

It is also seen that numerically controlled machines because of their high initial cost and high machining hour rate are used mainly for highly intricate components requiring high degree of accuracy .It has been observed that the machining time for carbide tools is less as compared to hardened tools.

REFERENCES

[1]. NC MACHINES- By S.J. MARTIN,peace publishers Moscow 1965
[2]. NC MACHINES- By YOREM KOREM,industrial press new delhi,1993
[3]. MECHATRONICS- By TATA McGraw HILL pvt limited,new delhi
[4]. CAD/CAM- By ISHRAT M. MIRZANA radient publishers,2008