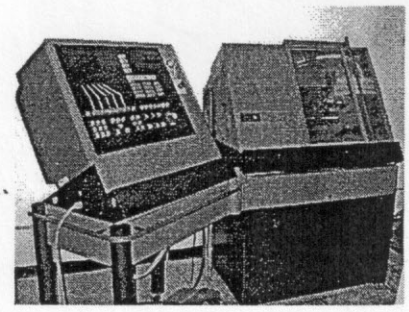
CNC Programming

Lecture 14, 15

Advanced Manufacturing



Overview

A numerical control, or "NC", system controls many machine functions and movements which were traditionally performed by skilled machinists.

Numerical control developed out of the need to meet the requirements of high production rates, uniformity and consistent part quality.

Programmed instructions are converted into output signals which in turn control machine operations such as spindle speeds, tool selection, tool movement, and cutting fluid flow.

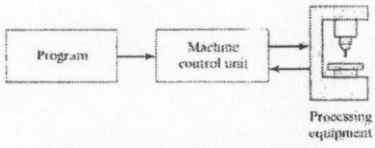


Figure 6.1 Basic components of an NC system.

Overview

By integrating a computer processor, computer numerical control, or "CNC" as it is now known, allows part machining programs to be edited and stored in the computer memory as well as permitting diagnostics and quality control functions during the actual machining.

All CNC machining begins with a part program, which is a sequential instructions or coded commands that direct the specific machine functions. The part program may be manually generated or, more commonly, generated by computer aided part programming systems.

Figure 1-8. Slant bed CNC lathe with hydraulic chucking and an electronically indexed 12 station turret. (Cincinnati Industrial, Inc.)

Basic CNC Principles

All computer controlled machines are able to accurately and repeatedly control motion in various directions. Each of these directions of motion is called an axis. Depending on the machine type there are commonly two to five axes.

Additionally, a CNC axis may be either a linear axis in which movement is in a straight line, or a rotary axis with motion following a circular path.

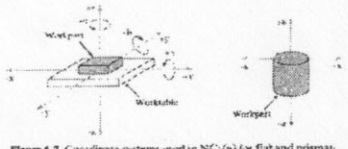


Figure 6.2 Coordinate systems used in NC: (a) for flat and prismatic work and (b) for rotational work. (On most turning machines, the z-axis is horizontal rather than vertical as we have shown it.)

Each axis consists of a mechanical component, such as a slide that moves, a servo drive motor that powers the mechanical movement, and a ball screw to transfer the power from the servo drive motor to the mechanical component. These components, along with the computer controls that govern them, are referred to as an axis drive system.

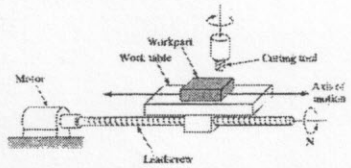
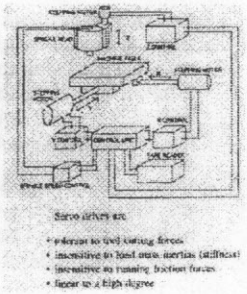


Figure 6.26 Motor and leadscrew arrangement in an NC positioning system.

Basic CNC Principles

Using a vertical mill machining center as an example, there are typically three linear axes of motion. Each is given an alphabetic designation or address. The machine table motion side to side is called the "X" axis. Table movement in and out is the "Y" axis, while head movement up and down the column is the "Z" axis.



Basic CNC Principles

If a rotary table is added to the machine table, then the fourth axis is designated the "b" axis.

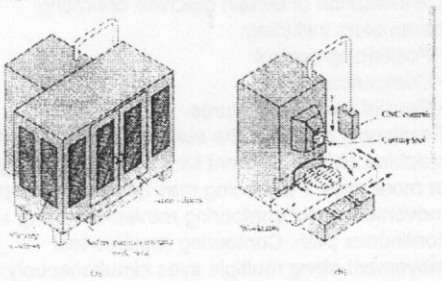


Figure 6.11 (a) Horizontal CNC machine with safety panels installed and (b) with side panels removed to show typical axis configuration for the horizontal spindle.

Work Positioning

The method of accurate work positioning in relation to the cutting tool is called the "rectangular coordinate system." On the vertical mill, the horizontal base line is designated the "X" axis, while the vertical base line is designated the "Y" axis. The "Z" axis is at a right angle, perpendicular to both the "X" and "Y" axes.

Increments for all base lines are specified in linear measurements, for most machines the smallest increment is one ten-thousandth of an inch (.0001). If the machine is graduated in metric the smallest increment is usually one thousandth of a millimeter (.001mm).

The rectangular coordinate system allows the mathematical plotting of points in space. These points or locations are called "coordinates." The coordinates in turn relate to the tool center and dictate the "tool path" through the work.

Basic CNC Principles

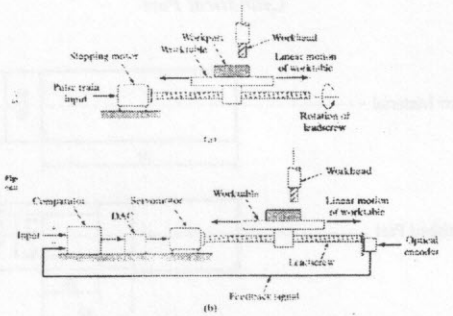


Figure 6.27 Two types of motion control in NC: (a) open loop and (b) closed loop.

CNC Programming Basics

CNC instructions are called part program commands.

When running, a part program is interpreted one command line at a time until all lines are completed.

Commands, which are also referred to as blocks, are made up of words which each begin with a letter address and end with a numerical value.

CNC Programming Basics

- Each letter address relates to a specific machine function. "G" and "M" letter addresses are two of the most common. A "G" letter specifies certain machine preparations such as inch or metric modes, or absolutes versus incremental modes.
- A "M" letter specifies miscellaneous machine functions and work like on/off switches for coolant flow, tool changing, or spindle rotation. Other letter addresses are used to direct a wide variety of other machine commands.

Program Command Parameters

Optimum machine programming requires consideration of certain machine operating parameters including:

- Positioning control
- Compensations
- Special machine features

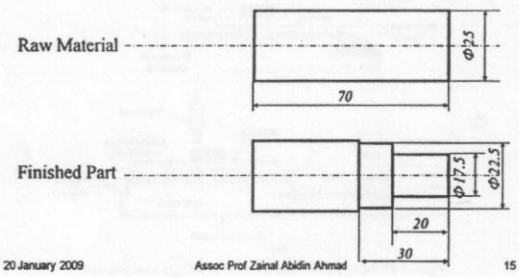
Positioning control is the ability to program tool and machine slide movement simultaneously along two or more axes. Positioning may be for point-to-point movement or for contouring movement along a continuous path. Contouring requires tool movement along multiple axes simultaneously.

Program Command Parameters

- Contouring requires tool movement along multiple axes simultaneously. This movement is referred to as "Interpolation" which is the process of calculating intermediate values between specific points along a programmed path and outputting those values as a precise motion.
- Interpolation may be linear having just a start and end point along a straight line, or circular which requires an end point, a center and a direction around the arc.

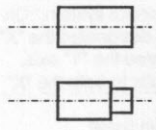
Programming Example

Cylindrical Part



Programming Example

Cylindrical Part



```
O0013
N0005 G53
N0010 T0303
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X22.50 Z2.0 S500
N0080 G01 Z-30.0 F100
N0090 G00 X23.0 Z2.0 S500
N0100 G84 X17.5 Z-20.0 D0=200 D2=200 D3=650
N0110 G00 Z2.0
N0120 X50.0 Z50.0
N0130 M30
```


Program Interpretation

O0013
Program identification number

Program Interpretation

O0013

N0005 G53
To cancel any previous working zero point

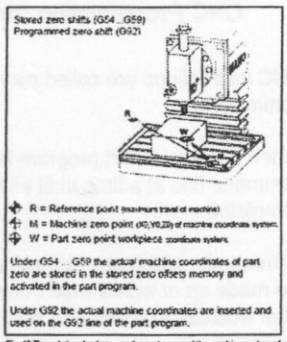


Fig. 13 The relationship between the part zero and the machine system of coordinates. (Deckel Maho, Inc.)

Program Interpretation

O0013
N0005 G53
N0010 T0303

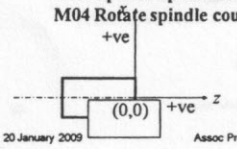
N0010 Sequence number
T0303 Select tool number 303

Program Interpretation

O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.0 Z0.0 S500 M04

G57 To set the working zero point as saved
G00 Rapid movement (no cutting)
X26.0 *X* location (as a diameter; 13 form zero)
Z0.0 *Z* location
S500 Spindle speed is 500 rpm
M04 Rotate spindle counterclockwise



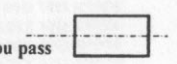
Program Interpretation

O0013

N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100

G01 Linear interpolation (cutting)
X-0.20 Move only in x direction until you pass the center by 0.1 mm (facing)
F100 Set feed rate to 100 mm/min.



Program Interpretation

O0013
N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0

G00 Move rapidly away from workpiece (no cutting)
Z2.0 the movement is 2 mm away from the face.

Program Interpretation

O0013

N0005 G53

N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0

Go to a safe location away from the workpiece [x = 50 (25 from zero), z = 50] to change the tool.

Program Interpretation

O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404

T0404 Select tool number 404

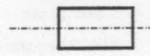
Program Interpretation

```
O0013
N0005 G53
N0010  T0404
N0020  G57 G00 X26.00 Z0.0 S500 M04
N0030  G01 X-0.20 F100
N0040  G00 Z2.0
N0050  X50.0 Z50.0
N0060  T0404
N0070  G57 G00 X22.50 Z2.0 S500
```

G57 PS0
G00 Rapid movement (no cutting)
X22.50 *X* location (as a diameter; 11.25 form zero)
Z2.0 *Z* location
S500 Spindle speed is 500 rpm

Program Interpretation

```
O0013
N0005 G53
N0010  T0404
N0020  G57 G00 X26.00 Z0.0 S500 M04
N0030  G01 X-0.20 F100
N0040  G00 Z2.0
N0050  X50.0 Z50.0
N0060  T0404
N0070  G57 G00 X25.00 Z2.0 S500 M04
N0080  G01 Z-30.0 F100
```



G01 Linear interpolation (cutting)
Z-30 Move only in z direction (external turning)
F100 Set feed rate to 100 mm/min.

Program Interpretation

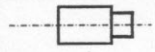
```
O0013
N0005 G53
N0010  T0404
N0020  G57 G00 X26.00 Z0.0 S500 M04
N0030  G01 X-0.20 F100
N0040  G00 Z2.0
N0050  X50.0 Z50.0
N0060  T0404
N0070  G57 G00 X25.00 Z2.0 S500 M04
N0080  G01 X22.5 Z-70.0 F100
N0090  G00 X23.0 Z2.0 S500
```

G00 Move rapidly away from workpiece (no cutting) to location x= 23.0 (11.50 from zero) and z = 2.0.

Program Interpretation

```
O0013
N0005 G53
N0010  T0404
N0020  G57 G00 X26.00 Z0.0 S500 M04
N0030  G01 X-0.20 F100
N0040  G00 Z2.0
N0050  X50.0 Z50.0
N0060  T0404
N0070  G57 G00 X25.00 Z2.0 S500 M04
N0080  G01 X22.5 Z-70.0 F100
N0090  G00 X26.0 Z2.0 S500
N0100  G84 X17.5 Z-20.0 D0=200 D2=200 D3=650
```

G84 Turning cycle for machining the step
X17.5 final diameter
Z-20 length of step is 20 mm
D0=200 Finish allowance in X direction (0.2 mm)
D2=200 Finish allowance in Z direction (0.2 mm)
D3=650 Depth of cut in each pass (0.65 mm)



Program Interpretation

```
O0013
N0005 G53
N0010  T0404
N0020  G57 G00 X26.00 Z0.0 S500 M04
N0030  G01 X-0.20 F100
N0040  G00 Z2.0
N0050  X50.0 Z50.0
N0060  T0404
N0070  G57 G00 X25.00 Z2.0 S500 M04
N0080  G01 X22.5 Z-70.0 F100
N0090  G00 X26.0 Z2.0 S500
N0100  G84 X17.5 Z-20.0 D0=200 D2=200 D3=650
N0110  G00 Z2.0
```

G00 Move rapidly away from workpiece (no cutting)
Z2.0 the movement is 2 mm away from the face.

Program Interpretation

```
O0013
N0005 G53
N0010  T0404
N0020  G57 G00 X26.00 Z0.0 S500 M04
N0030  G01 X-0.20 F100
N0040  G00 Z2.0
N0050  X50.0 Z50.0
N0060  T0404
N0070  G57 G00 X25.00 Z2.0 S500 M04
N0080  G01 X22.5 Z-70.0 F100
N0090  G00 X26.0 Z2.0 S500
N0100  G84 X17.5 Z-20.0 D0=200 D2=200 D3=650
N0110  G00 Z2.0
N0120  X50.0 Z50.0
```

X50.0 Z50.0 Move to the tool changing location

Program Interpretation

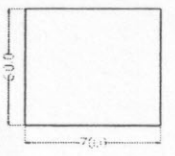
O0013

N0005 G53
N0010 T0404
N0020 G57 G00 X26.00 Z0.0 S500 M04
N0030 G01 X-0.20 F100
N0040 G00 Z2.0
N0050 X50.0 Z50.0
N0060 T0404
N0070 G57 G00 X25.00 Z2.0 S500 M04
N0080 G01 X22.5 Z-70.0 F100
N0090 G00 X26.0 Z2.0 S500
N0100 G84 X17.5 Z-20.0 D_0=200 D_2=200 D_3=650
N0110 G00 Z2.0
N0120 X50.0 Z50.0 T00
N0130 M30

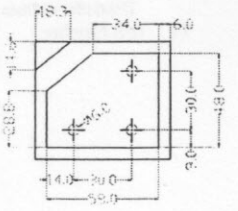
M30 Program End

Programming Example

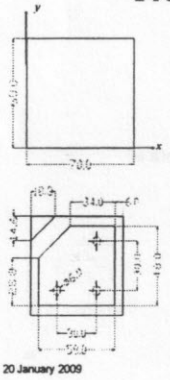
Raw Material



Finished Part

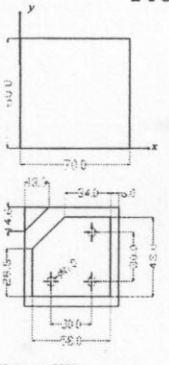


Programming Example



G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
N011 M05
N012 M02

Programming Example

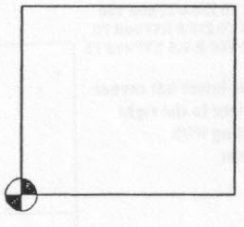


Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05
N014 M02

Program Interpretation

G55 X200 Y80

Setting the datum to the lower left corner of the work piece



Program Interpretation

G55 X200 Y80
Program 1

Program Identification Number

Program Interpretation

```
G55 X200 Y80
Program 1
N001 M06 T1
```

N001 Sequence Number
M06 Tool Change (End Mill with Diameter=12mm
T1 Tool Number

Program Interpretation

```
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
```

Start rotating the spindle clockwise with 400 rpm

Program Interpretation

```
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
```

Go to Safe Position with feed 150mm/min

Program Interpretation

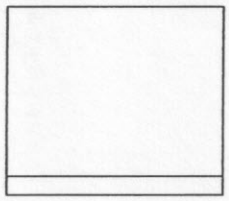
```
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
```

Lower the end mill to determine the depth of cut

Program Interpretation

```
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
```

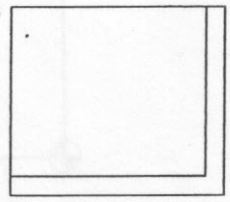
Move from the lower left corner of the work piece to the right lower one cutting with feed=75mm/min



Program Interpretation

```
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
```

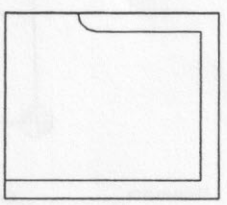
Move from the lower left corner of the work piece to the right lower one cutting with feed=75mm/min



Program Interpretation

```
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
```

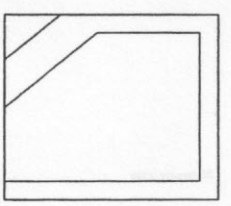
Cutting the horizontally up to X=30



Program Interpretation

```
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
```

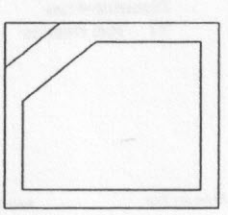
Cutting to X=0 & Y=40



Program Interpretation

```
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
```

Complete the countering



Program Interpretation

```
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
```

Repeat 7 times blocks from N003 to N009 with incremental offset of Z=-0.5

Program Interpretation

```
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
N011 M05
```

Spindle Off

Program Interpretation

```
G55 X200 Y80
Program 1
N001 M06 T1
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X-8 Y0 Z-0.5 ZFeed 150
N005 G01 X70 Y0 Z-0.5 XYFeed 75
N006 G01 X70 Y60 Z-0.5 XYFeed 75
N007 G01 X30 Y60 Z-0.5 XYFeed 75
N008 G01 X0 Y40 Z-0.5 XYFeed 75
N009 G01 X0 Y0 Z-0.5 XYFeed 75
N010 G81 R3 E9 N7 Z-0.5
N011 M05
N012 M02
```

End Program

Program Interpretation

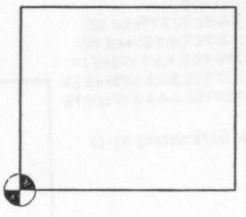
Tool Change

Changing the tool

Program Interpretation

Tool Change
G55 X200 Y80

Setting the datum to the lower left corner of the work piece



Program Interpretation

Tool Change
G55 X200 Y80
Program 2

Program Identification Number

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2

N001 Sequence Number
M06 Tool Change (Drill with Diameter=6mm
T2 Tool Number

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400

Start rotating the spindle clockwise with 400 rpm

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150

Go to Safe Position with feed 150mm/min

Program Interpretation

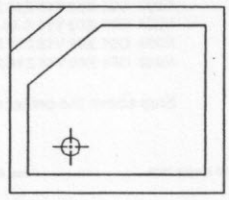
Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150

Stop above the center of the first hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75

Start Drill the first hole



Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150

Retract to a position above the hole

Program Interpretation

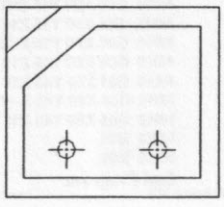
Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150

Stop above the center of the second hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75

Drill the second hole



Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150

Retract to a position above the second hole

Program Interpretation

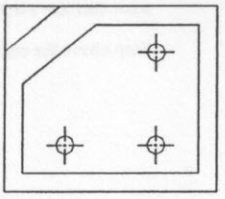
Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150

Stop above the center of the third hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75

Drill the third hole



Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150

Retract to a position above the third hole

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05

Spindle off

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05
N014 M02

End Program

Program Interpretation

Tool Change
G55 X200 Y80
Program 2
N001 M06 T2
N002 M03 rpm 400
N003 G01 X-8 Y0 Z0 XYFeed 150
N004 G01 X20 Y15 Z10 XYFeed 150 ZFeed 150
N005 G01 X20 Y15 Z-10 ZFeed 75
N006 G01 X20 Y15 Z10 ZFeed 150
N007 G01 X50 Y15 Z10 ZFeed 150
N008 G01 X50 Y15 Z-10 ZFeed 75
N009 G01 X50 Y15 Z10 ZFeed 150
N010 G01 X50 Y45 Z10 ZFeed 150
N011 G01 X50 Y45 Z-10 ZFeed 75
N012 G01 X50 Y45 Z10 ZFeed 150
N013 M05
N014 M02

End Program