CNC Programming

By

Dr. M Vishnu Vardhan

Objectives

- To be able to know when and what type of NC should be used
- To be able to read and interpret an NC part program
- To be able to create NC part programs for milled parts
- To understand the difference between world, machine and part coordinates
- To understand how to set machine offsets
- To execute an NC part program

Introduction

CNC stands for **Computer Numerical Control.**

It is the technology of controlling a machining operation using a computer program, which is called Numerical Control (NC) Program.

In other words, a computer rather than a person will directly control the machine tool.

Introduction

The most important computerized machine tools that are used extensively in the industry are:

CNC Turning (Lathe) machine

CNC Milling machine

Turning Machine axes

CNC Turning machine has at least 2 controllable feed axes, marked as X and Z; Fig. 1.8

Turning Machine axes

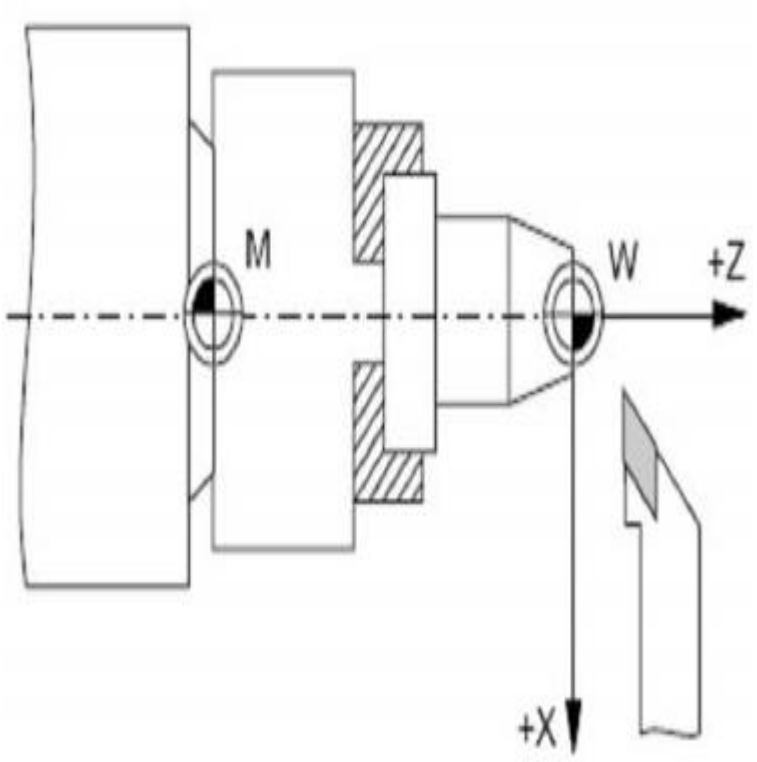
 When the cutting tool moves toward and backward the machine spindle, this is called movement along Z axis.

 When the cutting tool moves in cross direction to the longitudinal axis of the workpiece, this is called movement along X axis.

Turning Machine axes

- Positive Z direction is when the tool moves away from the workpiece in Z axis.

- Positive X direction is when the tool moves away from the work part in X axis.



Dimensioning

There are two types of Dimensioning

Absolute Dimensioning

Incremental Dimensioning

Motivation and uses

To manufacture complex curved geometries in 2D or 3D was extremely expensive by mechanical means (which usually would require complex jigs to control the cutter motions)

Machining components with repeatable accuracy

Unmanned machining operations

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.

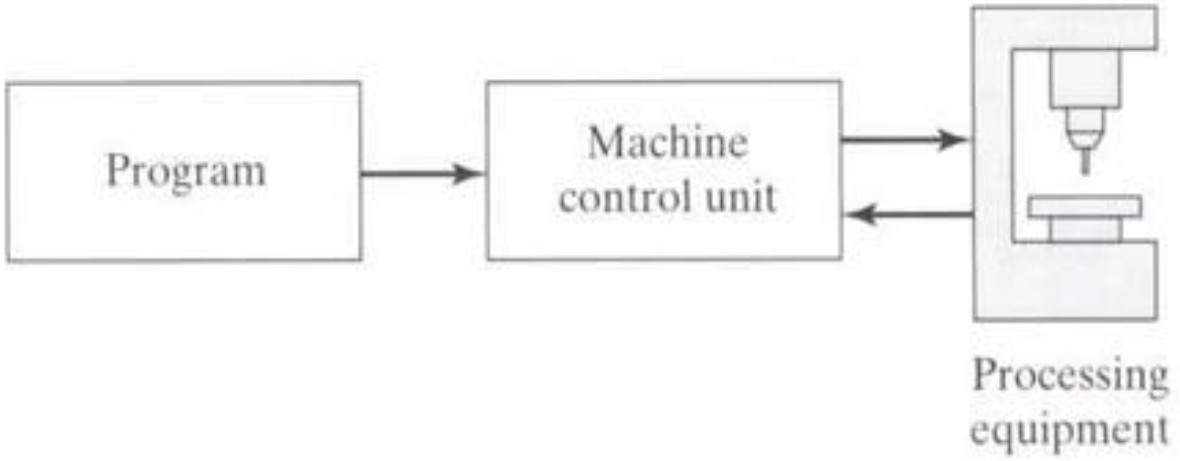
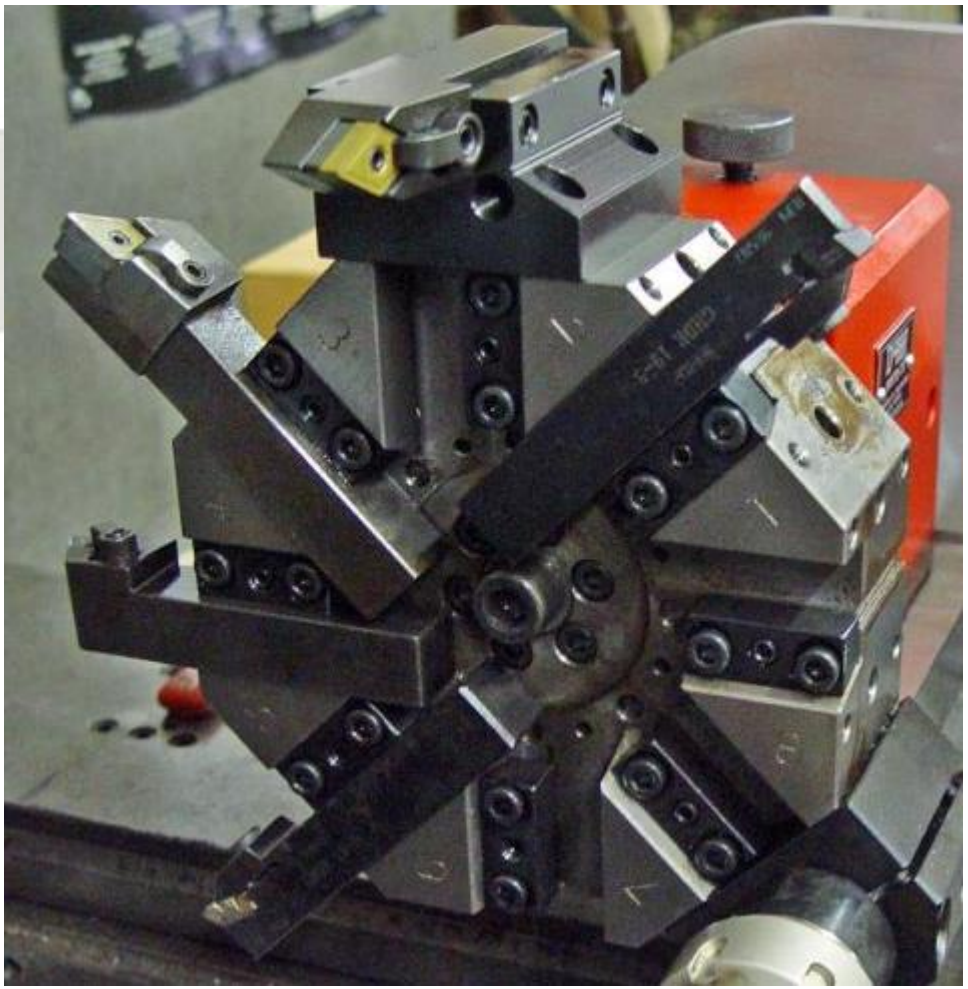


Fig. Basic components of NC system

CNC Tooling



Advantages of CNC

- Easier to program;
- Easy storage of existing programs;
- Easy to change a program
- Avoids human errors
- safer to operate
- Complex geometry is produced as cheaply as simple ones
- Usually generates closer tolerances than manual machines

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.

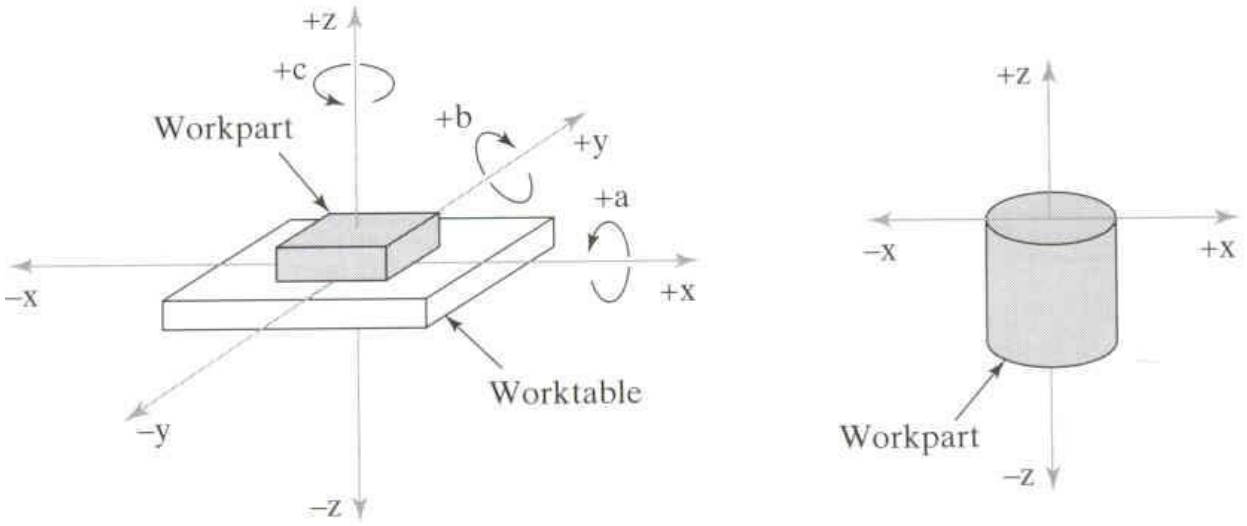


Figure 6.2 Coordinate systems used in NC: (a) for flat and prismatic work and (b) for rotational work. (On most turning machines, the *z*-axis is horizontal rather than vertical as we have shown it.)

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.

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.

CNC programming

Important things to know:

- Coordinate System
- Units, incremental or absolute positioning
- Coordinates: X,Y,Z
- Feed rate and spindle speed
- Coolant Control: On/Off
- Tool Control: Tool and tool parameters

CNC programming

- Programming consists of a series of instructions in form of letter codes
- Preparatory Codes: G codes- Initial machining setup and establishing operating conditions
- Miscellaneous codes – M codes For coolant control and other activities
- N codes- specify program line number to executed by the MCU
- Axis Codes: X,Y,Z - Used to specify motion of the slide along X, Y, Z direction
- Feed and Speed Codes: F and S- Specify feed and spindle speed
- Tool codes: T – specify tool number

Programming Key Letters

- O - Program number (Used for program identification)
- N - Sequence number (Used for line identification)
- G - Preparatory function
- X - X axis designation
- Y - Y axis designation
- Z - Z axis designation
- R - Radius designation
- F – Feed rate designation
- S - Spindle speed designation
- H - Tool length offset designation
- D - Tool radius offset designation
- T - Tool Designation
- M - Miscellaneous function

G codes

- G00 Rapid Transverse
- G01 Linear Interpolation
- G02 Circular Interpolation, CW
- G03 Circular Interpolation, CCW
- G17 XY Plane,G18 XZ Plane,G19 YZ Plane
- G20/G70 Inch units
- G21/G71 Metric Units
- G40 Cutter compensation cancel
- G41 Cutter compensation left
- G42 Cutter compensation right
- G43 Tool length compensation (plus)
- G43 Tool length compensation (plus)
- G44 Tool length compensation (minus)
- G49 Tool length compensation cancel
- G80 Cancel canned cycles
- G81 Drilling cycle
- G82 Counter boring cycle
- G83 Deep hole drilling cycle
- G90 Absolute positioning
- G91 Incremental positioning

M codes

- M00 Program stop
- M01 Optional program stop
- M02 Program end
- M03 Spindle on clockwise
- M04 Spindle on counterclockwise
- M05 Spindle stop
- M06 Tool change
- M08 Coolant on
- M09 Coolant off
- M10 Clamps on
- M11 Clamps off
- M30 Program stop, reset to start

CNC Programming format

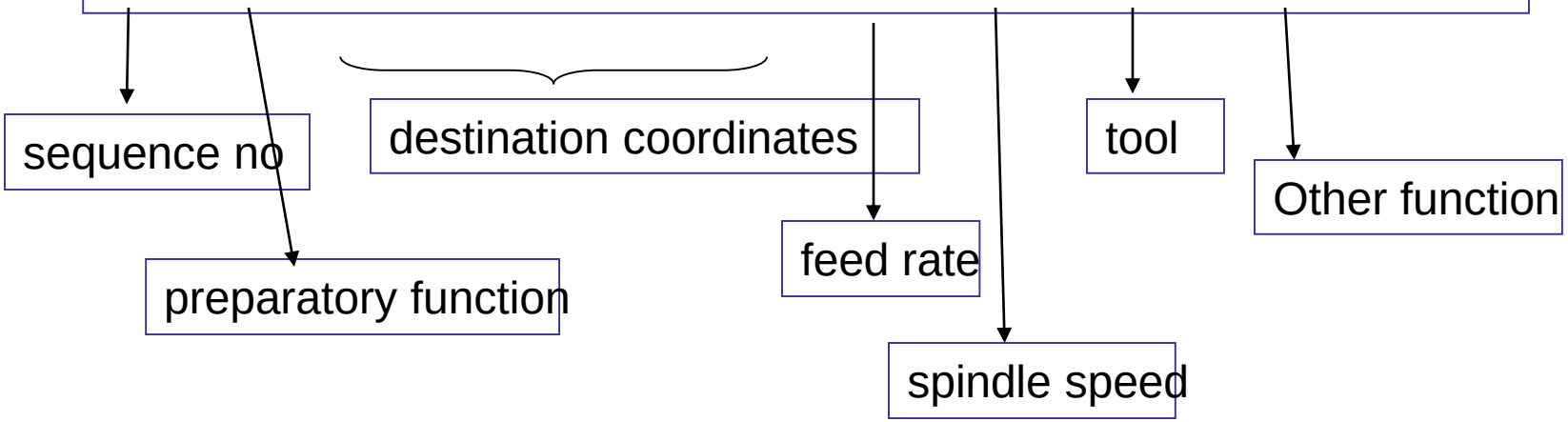
The programming format is called as a **word address format**

Each line of program == 1 **block**

Each block is composed of several instructions, or (words)

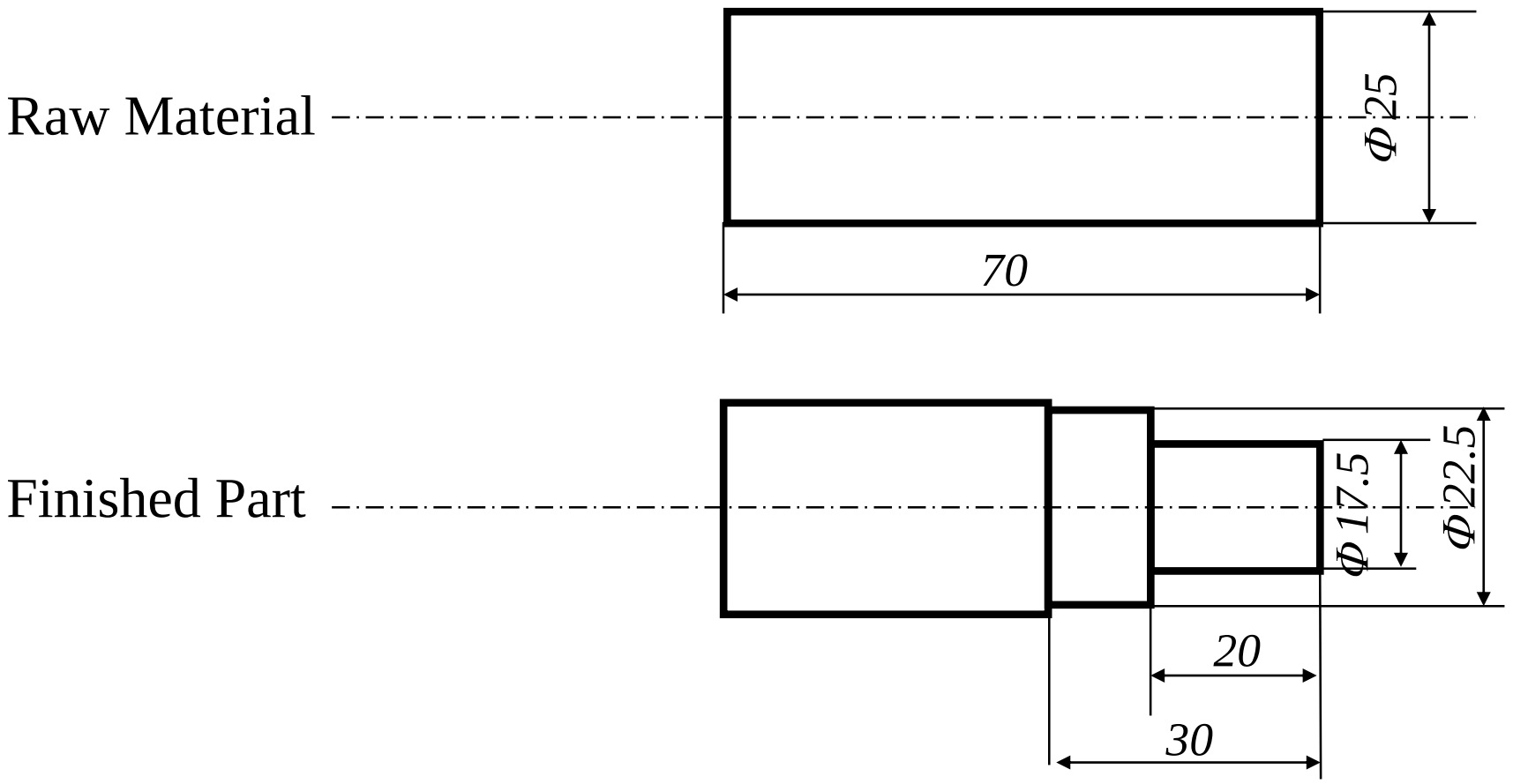
Sequence and format of words:

N3 G02 X+1.4 Y+1.4 Z+1.4 F3.2 S4 T0404 M2



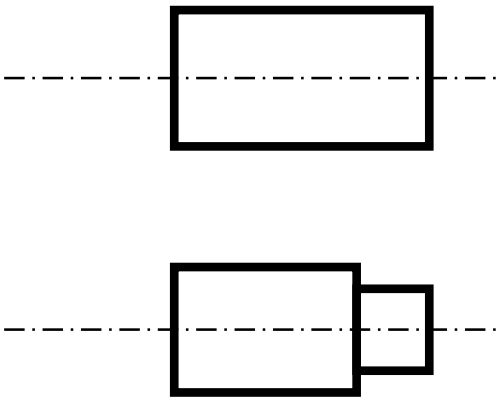
APT Programming Example

Cylindrical Part



NC 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
```

APT Program Interpretation

O0013

Program identification number

APT Program Interpretation

O0013

N0005 G53

To cancel any previous working zero point

APT Program Interpretation

O0013
N0005 G53
N0010 T0303

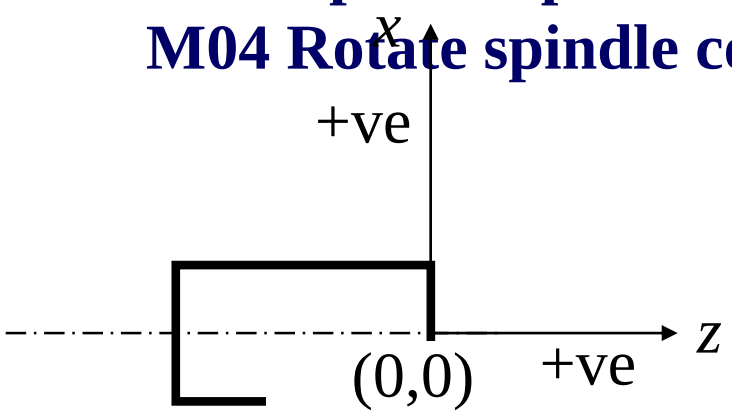
N0010 Sequence number
T0303 Select tool number 303

APT 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

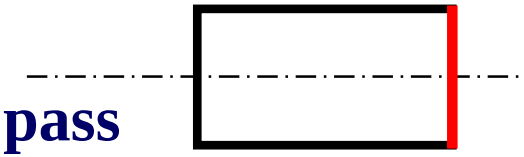


APT Program Interpretation

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.

APT 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.

APT 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.

APT 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

APT 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

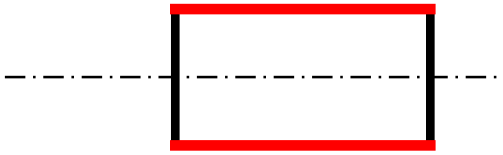
G57 PS0
G00 Rapid movement (no cutting)
X22.50 *X* location (as a diameter; 11.25 form zero)
Z2.0 *Z* location
S500 Spindle speed is 500 rpm

APT 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.

APT 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

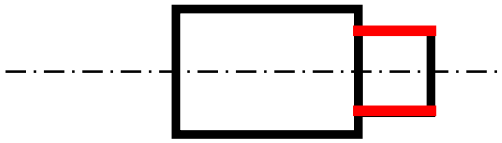
G00 Move rapidly away from workpiece (no cutting) to location x= 23.0 (11.50 from zero) and z = 2.0.

APT 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)

APT 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

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

APT 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

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

APT 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