



# SNS COLLEGE OF TECHNOLOGY



Coimbatore-35  
An Autonomous Institution

Accredited by NBA – AICTE and Accredited by NAAC – UGC with 'A+' Grade  
Approved by AICTE, New Delhi & Affiliated to Anna University, Chennai

## DEPARTMENT OF MECHANICAL ENGINEERING

### 19MEB301 – CAD CAM and AUTOMATION

#### III YEAR V SEM

### UNIT 3 – CNC CONTROL SYSTEMS & PART PROGRAMMING

#### TOPIC – NC dimensioning –reference points



# Prepared By

# Janagarathinam P

Assistant Professor

Dept. of mechanical Engineering

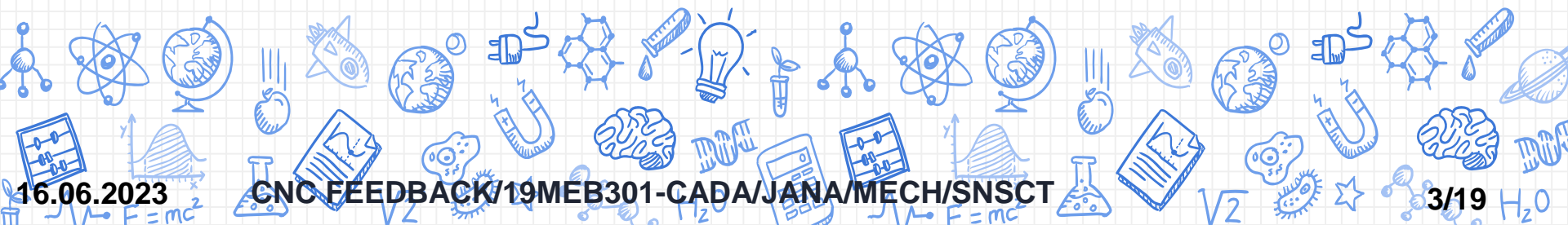
SNS College of technology

Email: [janagan12345@gmail.com](mailto:janagan12345@gmail.com)



# WHAT IS NC dimensioning – reference points ?

Let's start with the first question



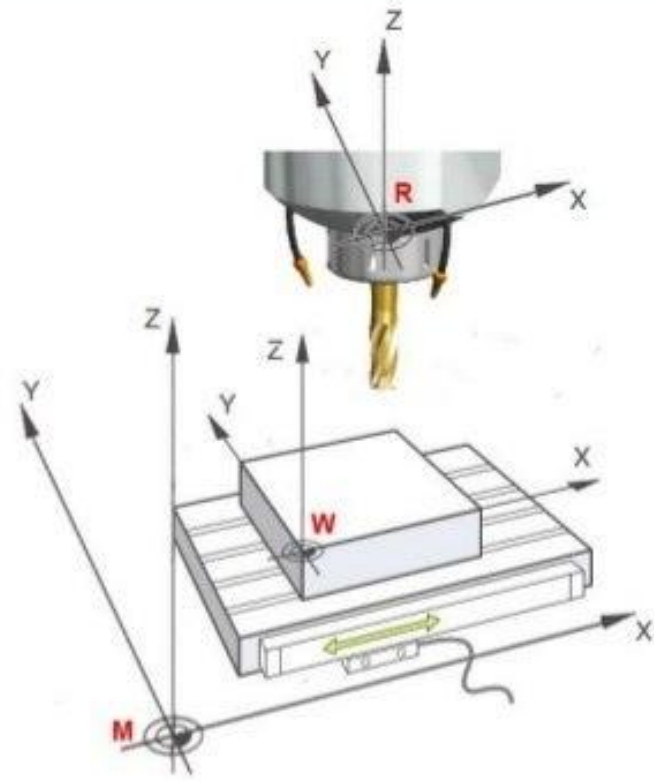
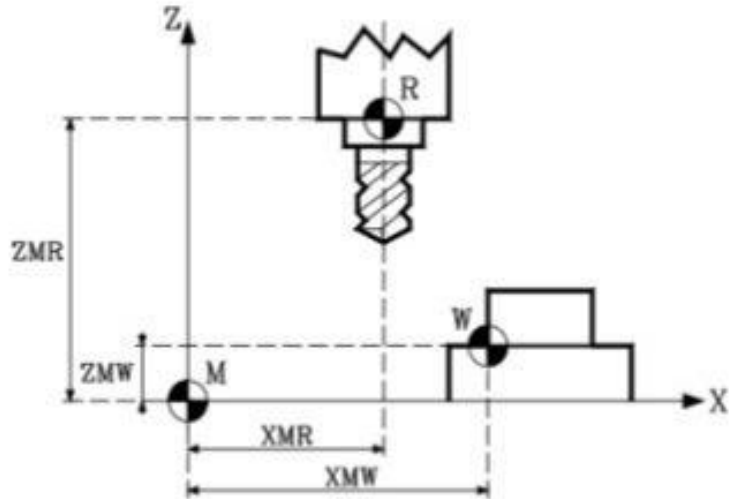
# What is CNC Machine Reference Point?

During every part setup, two major locations, called reference points, have to be carefully managed.

Those are points that establish the relationship between the known reference point of the machine (machine zero) and the unknown reference point of the part (part zero).





# CNC MACHINE REFERENCE POINT

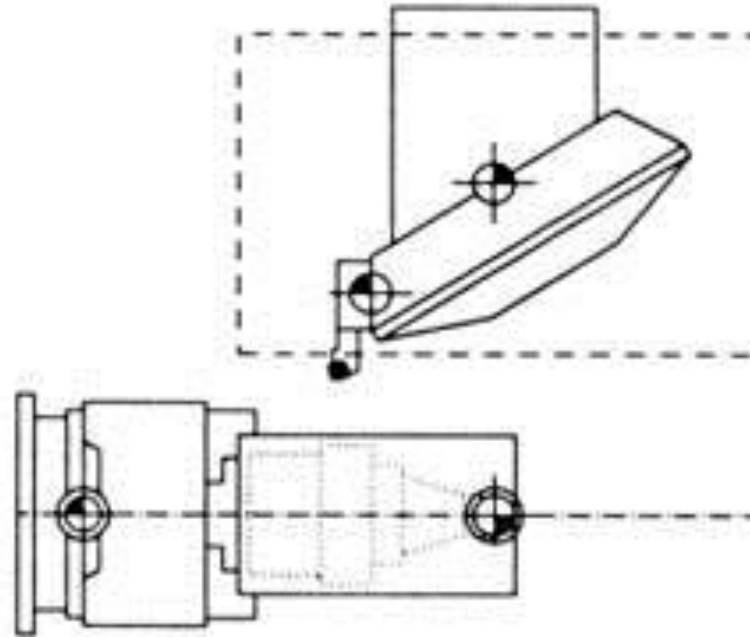
## Reference systems



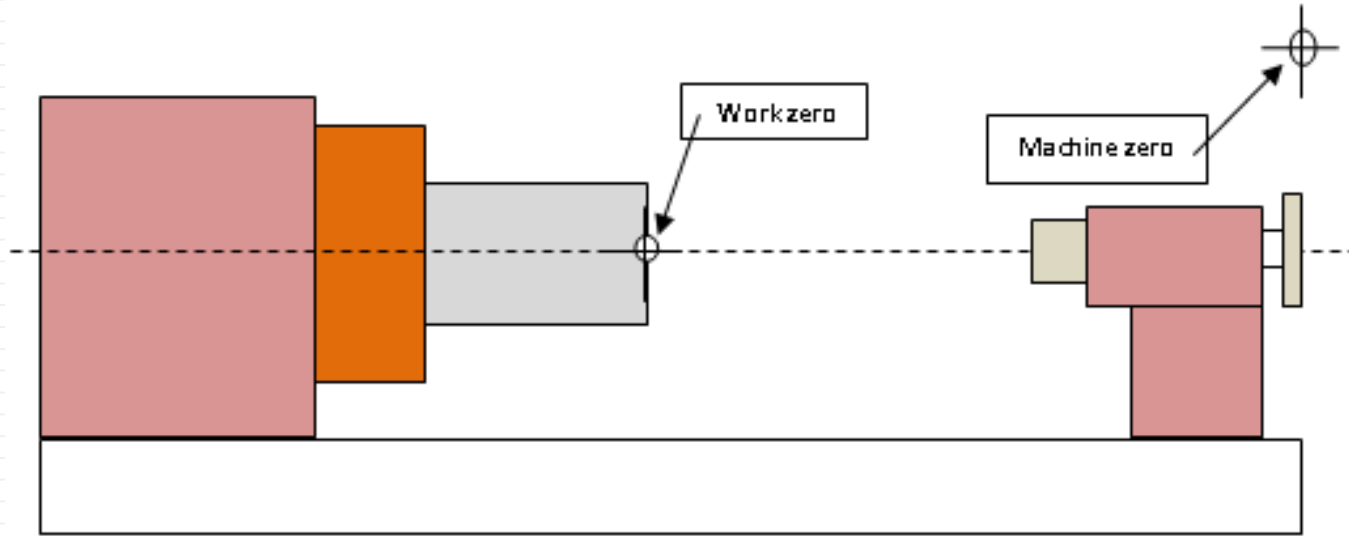
- M Machine zero
- W Part zero
- R Machine reference point
- XMW, YMW, ZMW... Coordinates of part zero
- XMR, YMR, ZMR... Coordinates of machine reference point ("REFVALUE")

# CNC MACHINE REFERENCE POINT

-  Machine zero point
-  Turret reference point
-  Tool reference point
-  Workpiece zero point



# CNC MACHINE REFERENCE POINT



- X Machine zero** - This point typically is located at the farthest positive direction along the X-, Y-, and Z-axes, and it cannot be changed by anyone after it leaves the original manufacturer
- X Work Zero** - normally set at the front face and at center of the job



# Zero and reference points on CNC



**M** machine zero point



**W** work part zero point



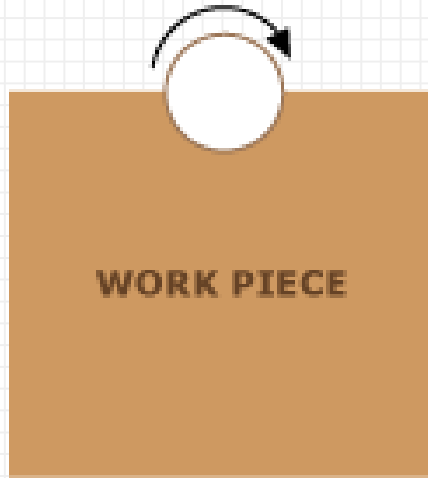
**R** reference point



**E** tool reference point

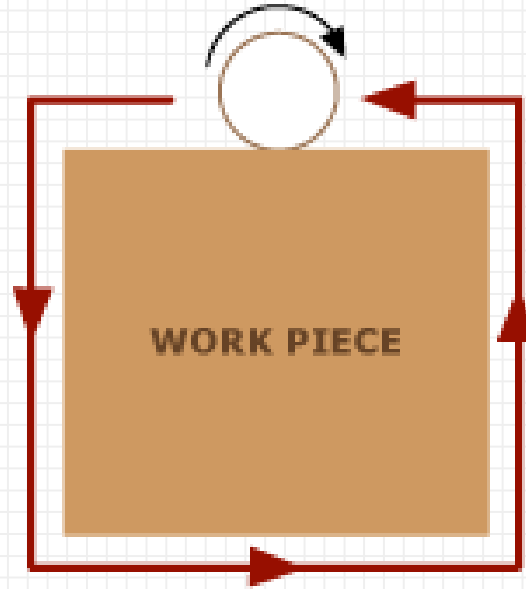


# Tool Offsets



## No offset

Cutter is on the centre of the programmed line.



## Offset

Cutter is moved off the programmed line.

# Tool Offsets

- The word 'offset' refers to the allowance made by the CNC machine for the diameter and length of the tool to cut the job
- Programming on a CNC machine is always done according to the centre point of the cutter
- The cutting tool runs along the programmed line. If the offset value of a tool is not set, the tool will move according to this centre point of the cutter rather than according to the tool being used
- This means that the tool will be cutting in the wrong part of the work piece

# What Is Tool Offset?

There are three essential measuring components we use to carry out CNC cutting. These are:

X G41: This particular component helps us to execute cutter radius compensation to the left.

X G42: This component allows us to execute cutter radius compensation to the right.

X G40: This component helps us to cancel cutter radius compensation when needed.

# What Is Work Offset?

X The average distance between the machine zero and the work zero in the X-Z plane is called the Work Offset. The measurement is either diameter wise or lengthwise according to the need. There are some specific components in the work offsets. These are:

- X G52: Takes input in any of the axis.
- X G54: Takes input in all the axis.
- X G55: Takes negative input on any axis.
- X G56: Takes only positive input.
- X G57: It can take double input.

# CNC Dimensioning

- **Absolute Dimensioning**
- **Incremental Dimensioning**

IMAGE CREDIT: [motioncontroltips.com](http://motioncontroltips.com)

# CNC Dimensioning

## What Is Absolute?

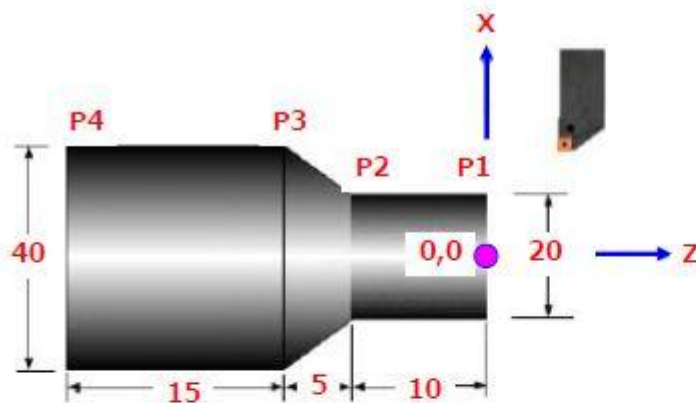
When programming in absolute, all of your coordinates and movement values will come from the origin (0,0) point. If you want to be in Absolute, the G-code that defines this is G90, which is a modal code.

## What Is Incremental?

How is Incremental different from Absolute? Well, instead of all of your coordinates/numbers coming from one location (0,0 offset), each move is the distance from your current location.

IMAGE CREDIT: sciencedirect.com

# CNC Dimensioning



P1 → P2 → P3

Absolute:

X20.0 Z-10.0

X40.0 Z-15.0

Incremental:

U0.0 W-10.0

U20.0 W-5.0

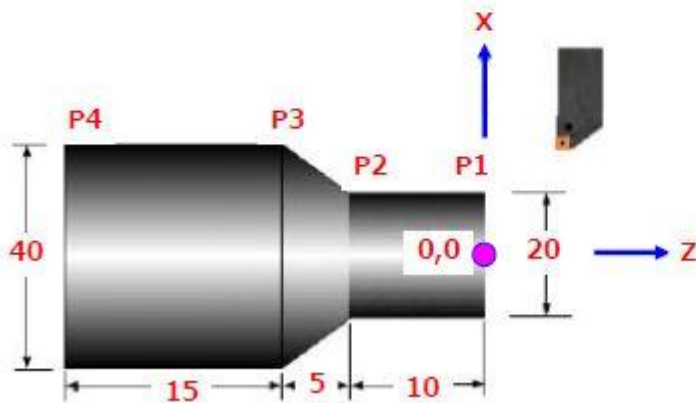
G00 Rapid Traverse:

Format:

G00 X\_ Z\_

Example:

G00 X20.0 Z0.0





# Recap

- **CNC Machine Reference Point**
- **Types of Machine References**
- **Symbols for Machine reference**
- **Tool offset**
- **Work offset**
- **CNC Dimensioning**
- **Absolute and Incremental Dimensioning**

# Assessment

## X Rapid Fire:

Differentiate between different reference points.

Can you recall the codes used for tool offset?

Which is better? Absolute or Incremental dimensioning?



# THANKS!