

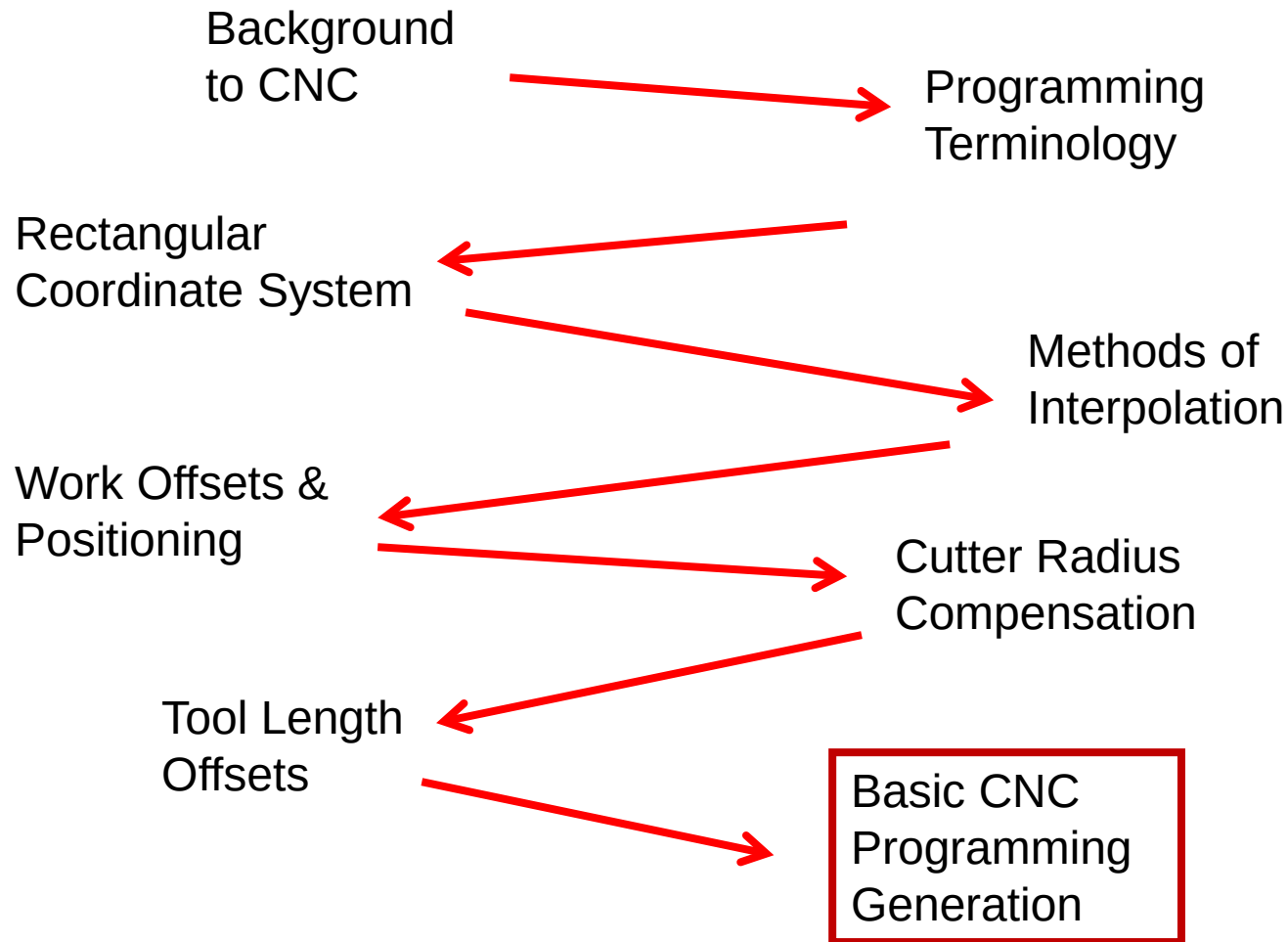
Computer Numerical Control (CNC) Concepts & Theory



Computer Numerical Control (CNC) Concepts & Theory

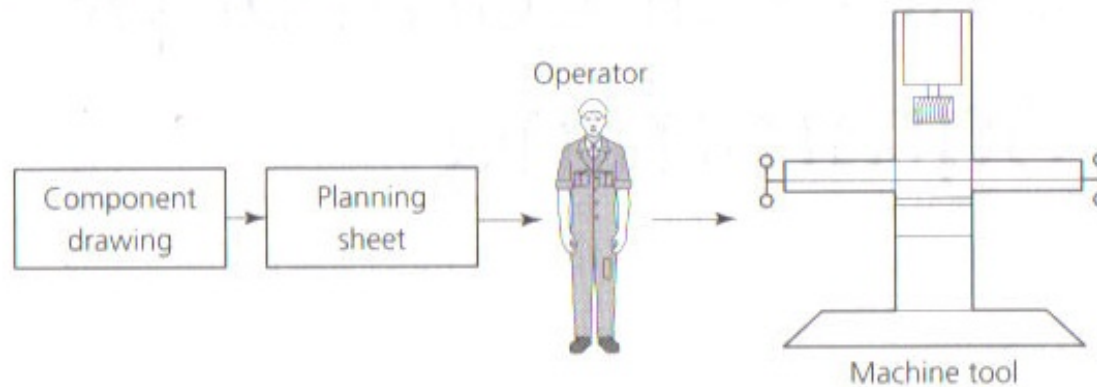


CNC Concepts & Theory



Background

Manually Controlled Machine Tools



Starting/Stopping
Relative Movements
Cutting Fluid (on/off)
Altering Spindle Speed
Altering Feed Rate
Changing Tools

Quality
Manufacturing Time

Judgement
Experience
Machine Operator
Skill
Concentration

Background

Manually Controlled Machine Tools



Batch Production



Repetitive Actions

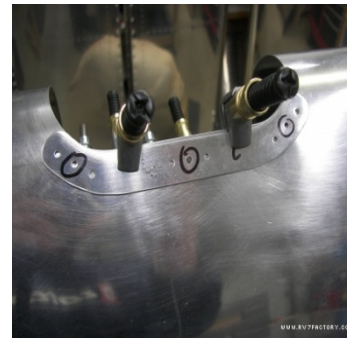


Jigs

Fixtures

Templates

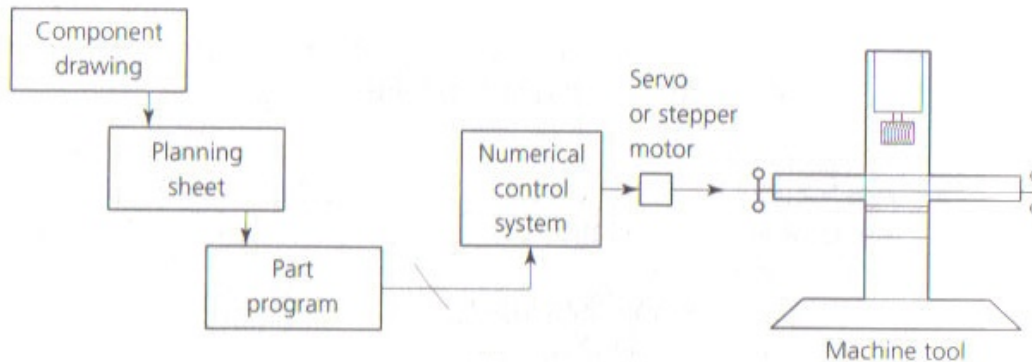
Automatic
Machine Tools



Background

Automatically Controlled Machine Tools

Computer Numerical Controlled (CNC) Machine Tools

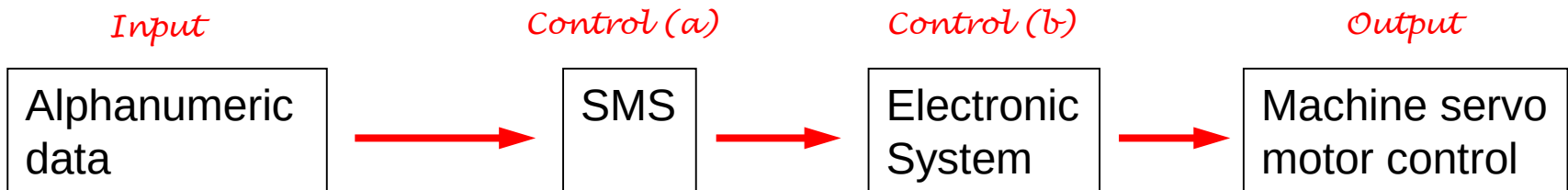


CNC now used exclusively

Dedicated, build-in computers

Larger memories

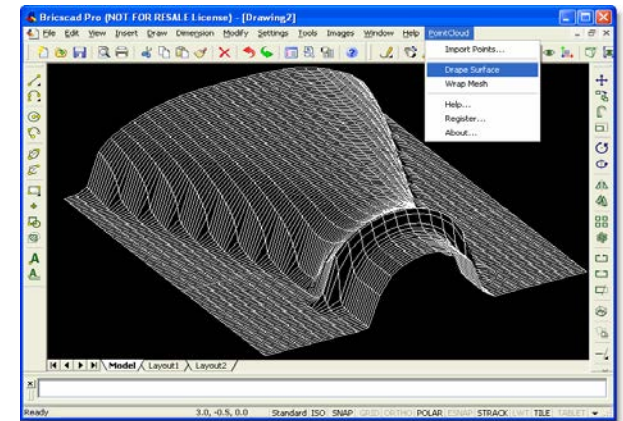
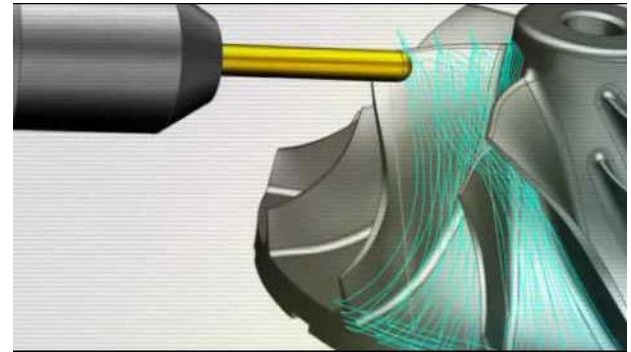
- Sophisticated system management software (SMS)
- Canned cycles
- Part Programs



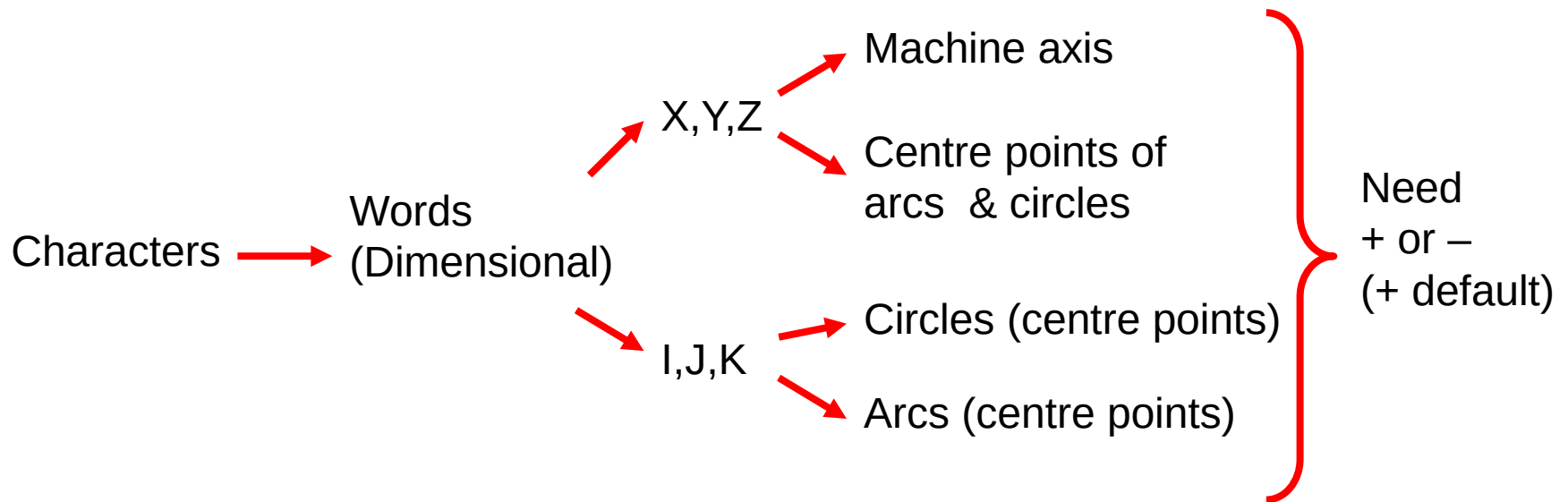
CNC Programming

CNC Control

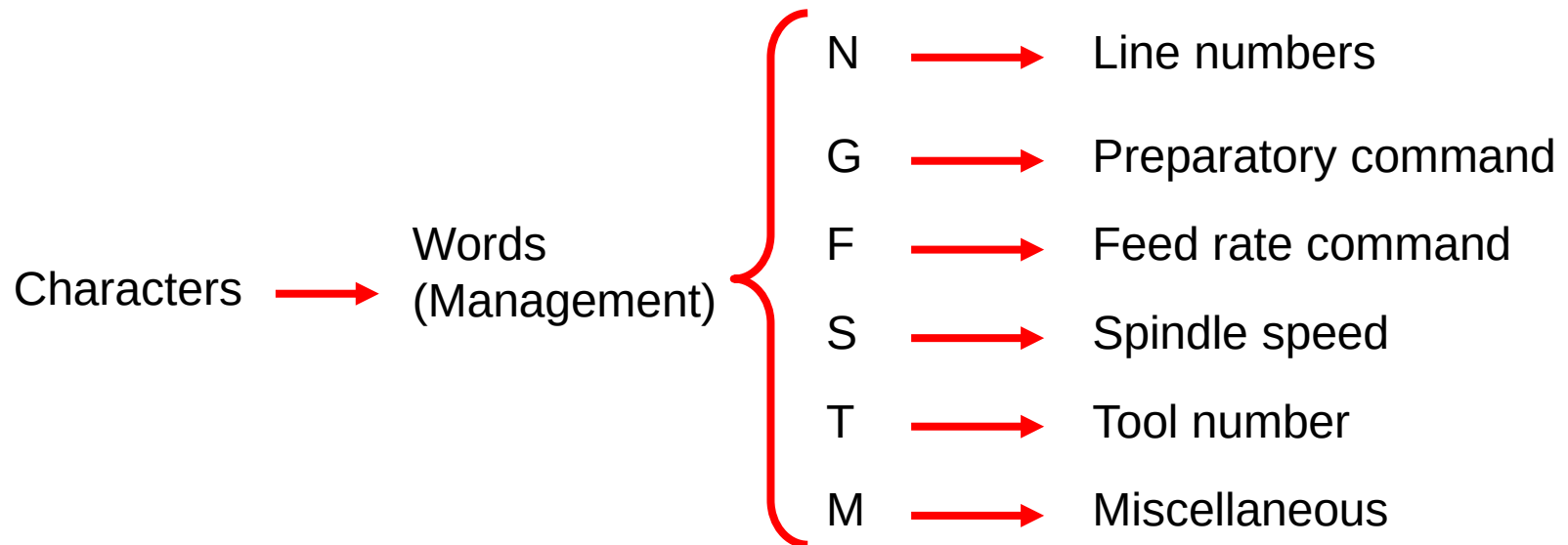
- Manual control
- Numerical control (NC)
 - Manual programming
 - CAM generated programs
 - R E technology



Program Terminology

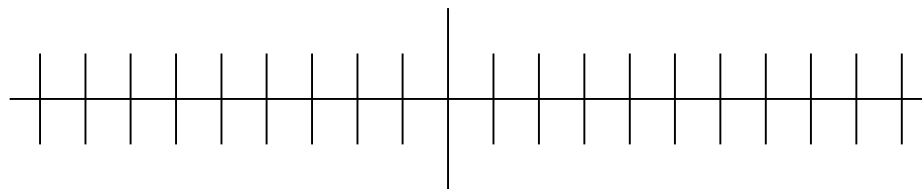


Program Terminology

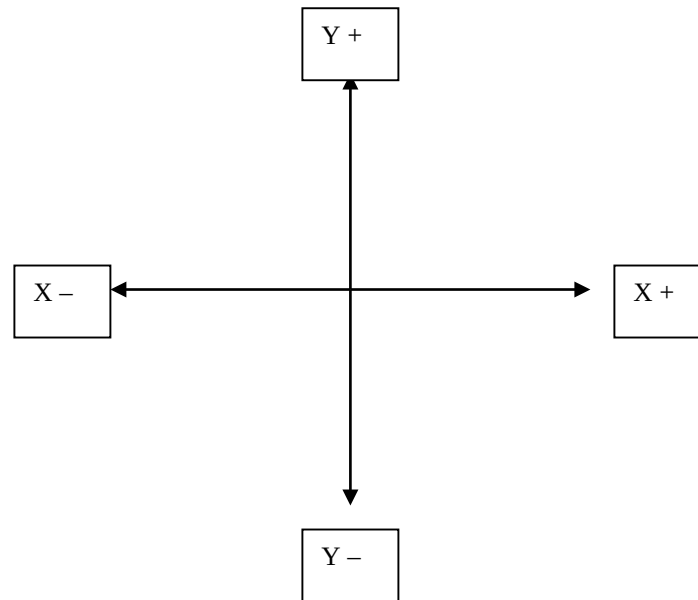


Rectangular Coordinate Systems

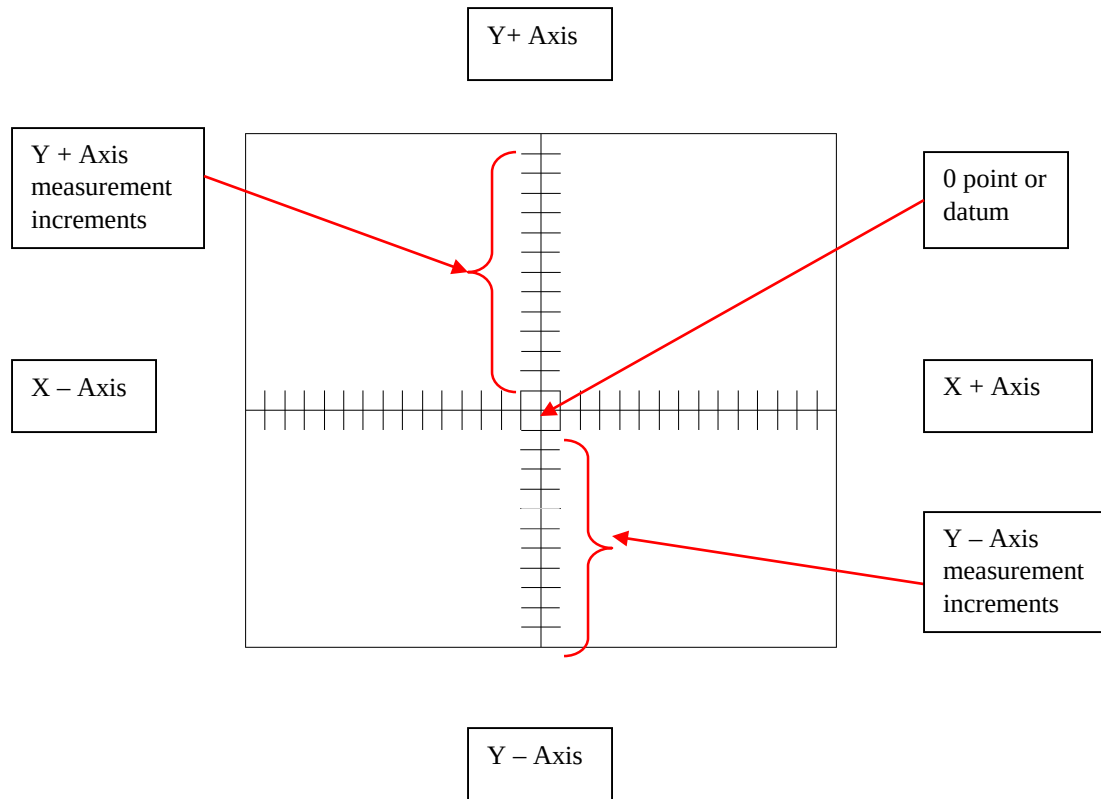
The Number Scale



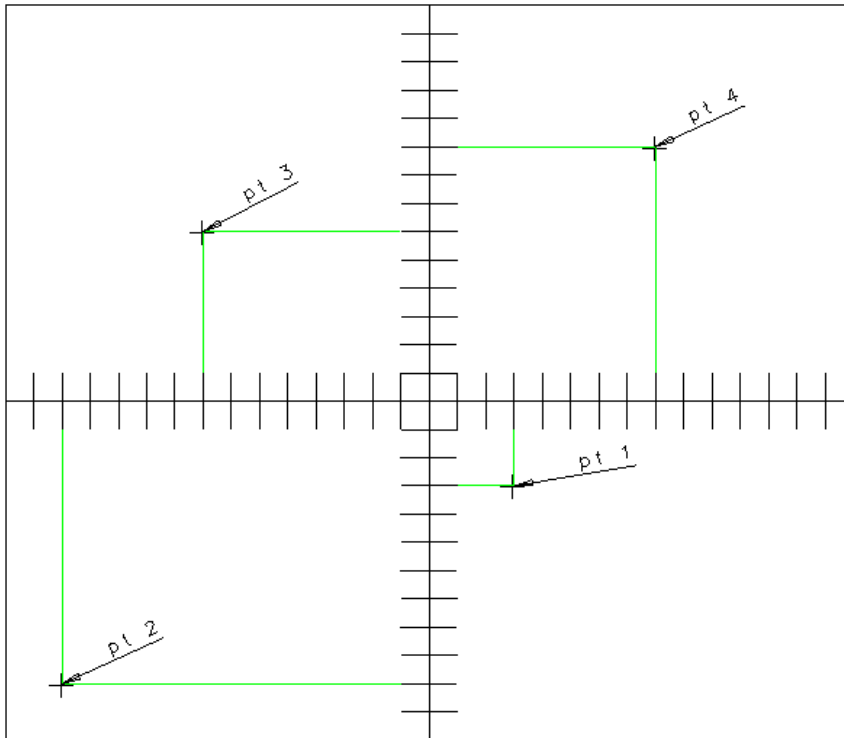
Axes



Axes & Planes



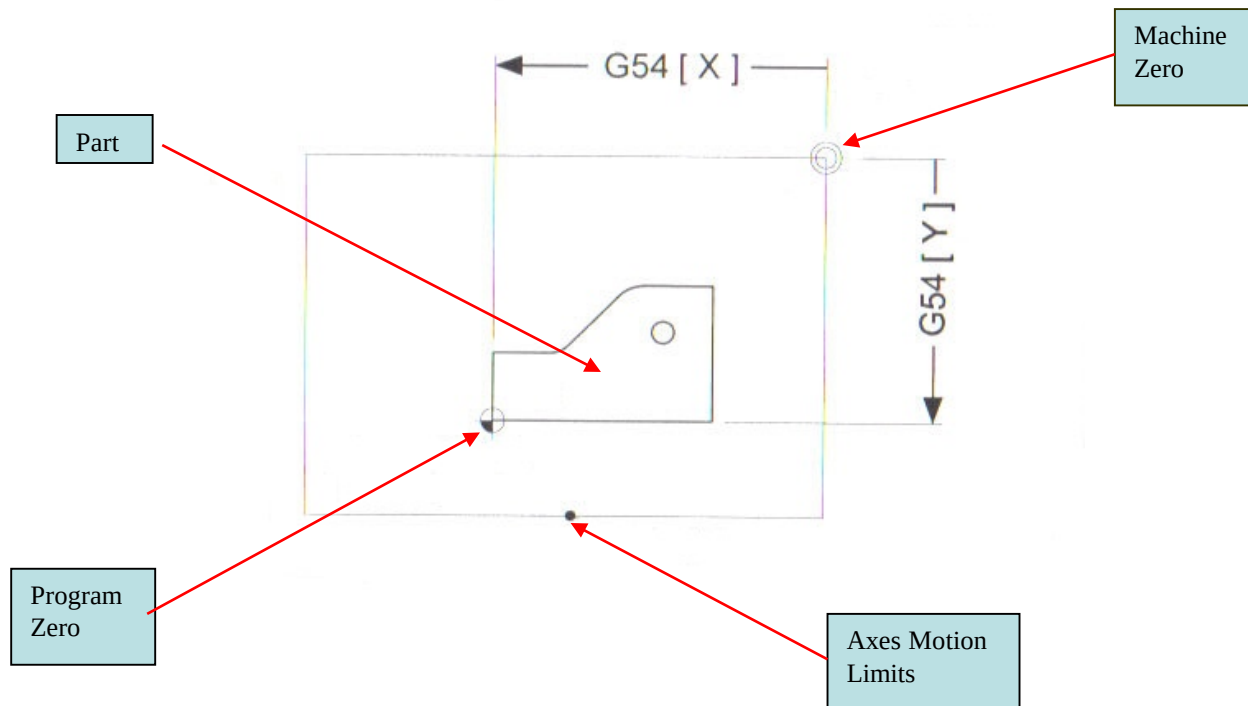
Points Defined by the Rectangular Coordinate System



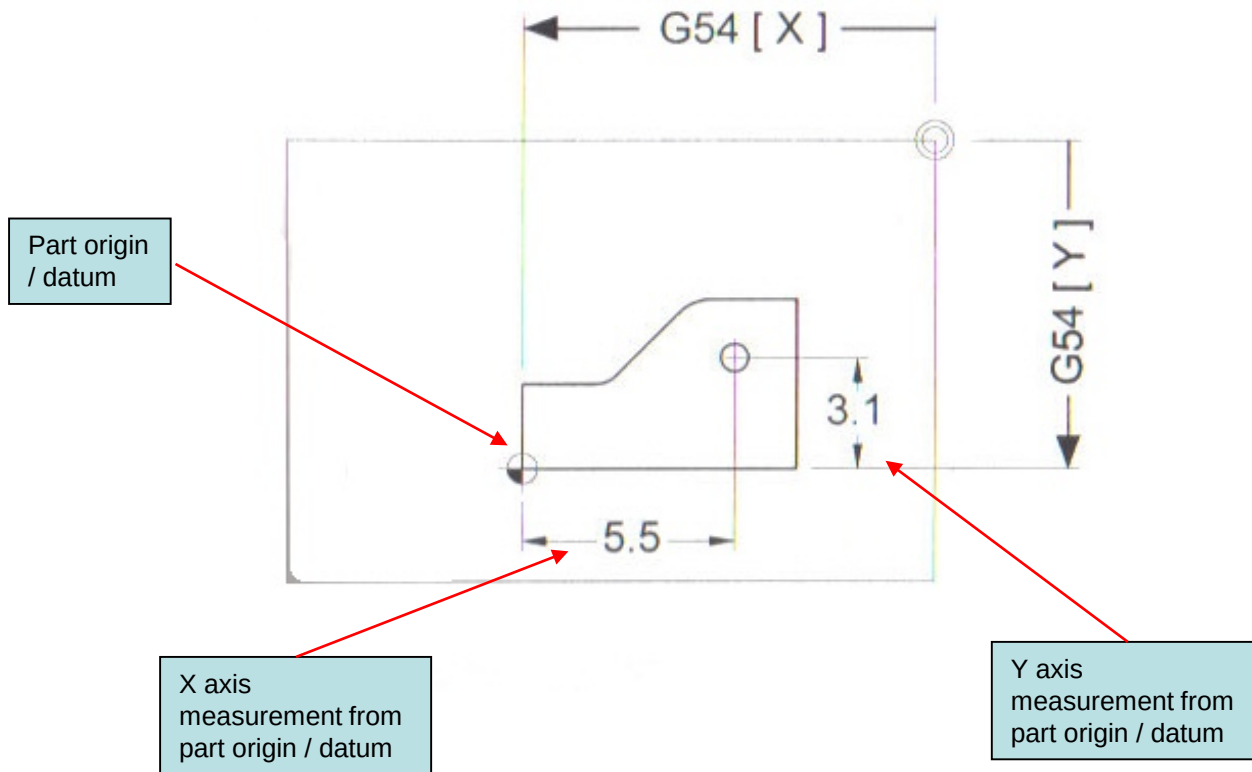
Point Number	X Axis Value	Y Axis Value
P1	3	-3
P2	-13	-10
P3	-8	6
P4	8	9

Work Offsets

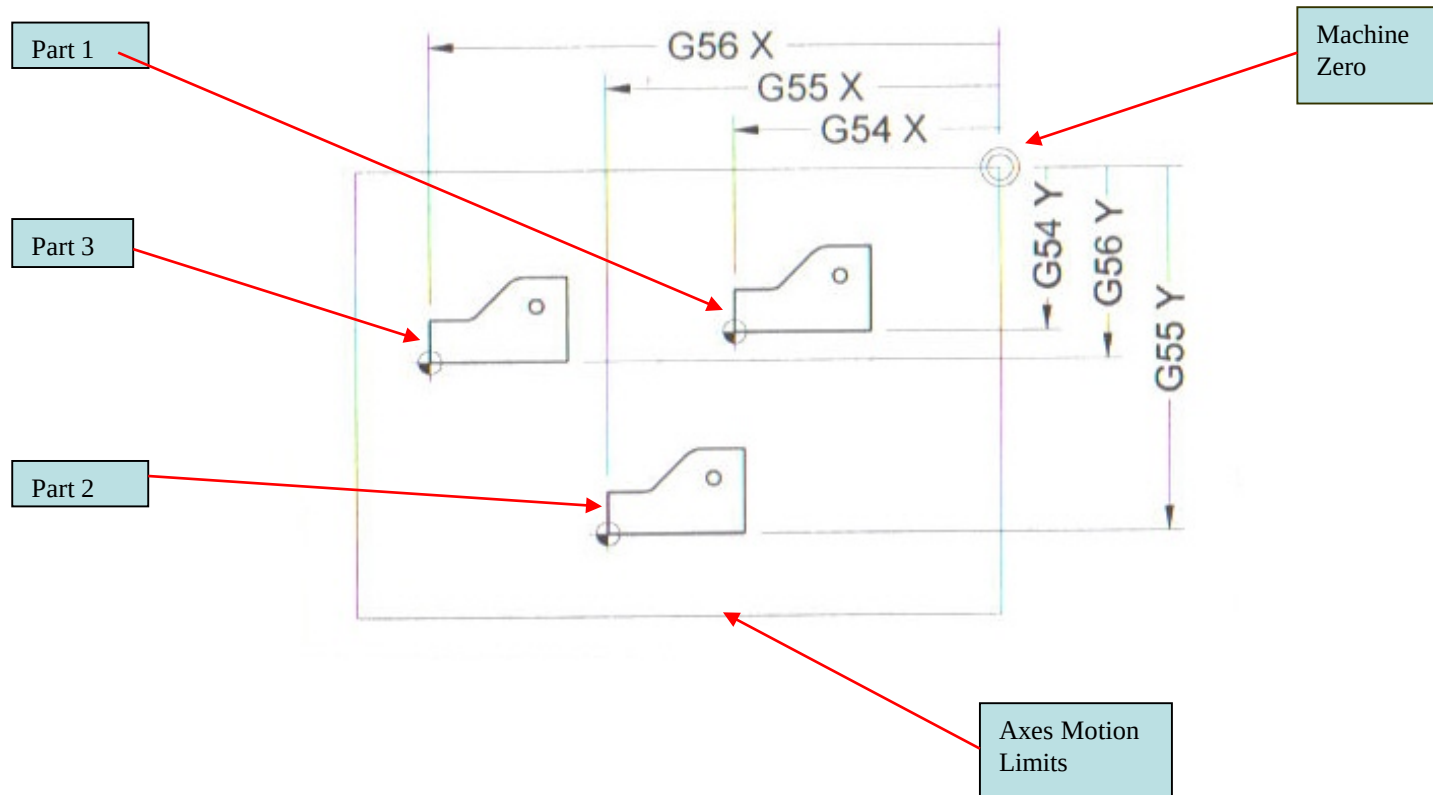
Work Offsets



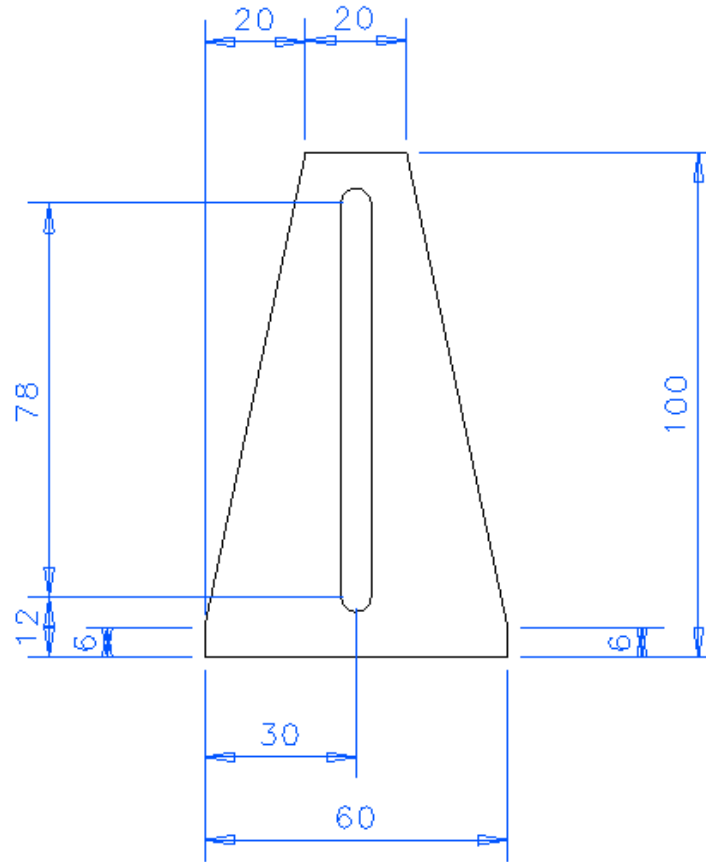
Work Offset



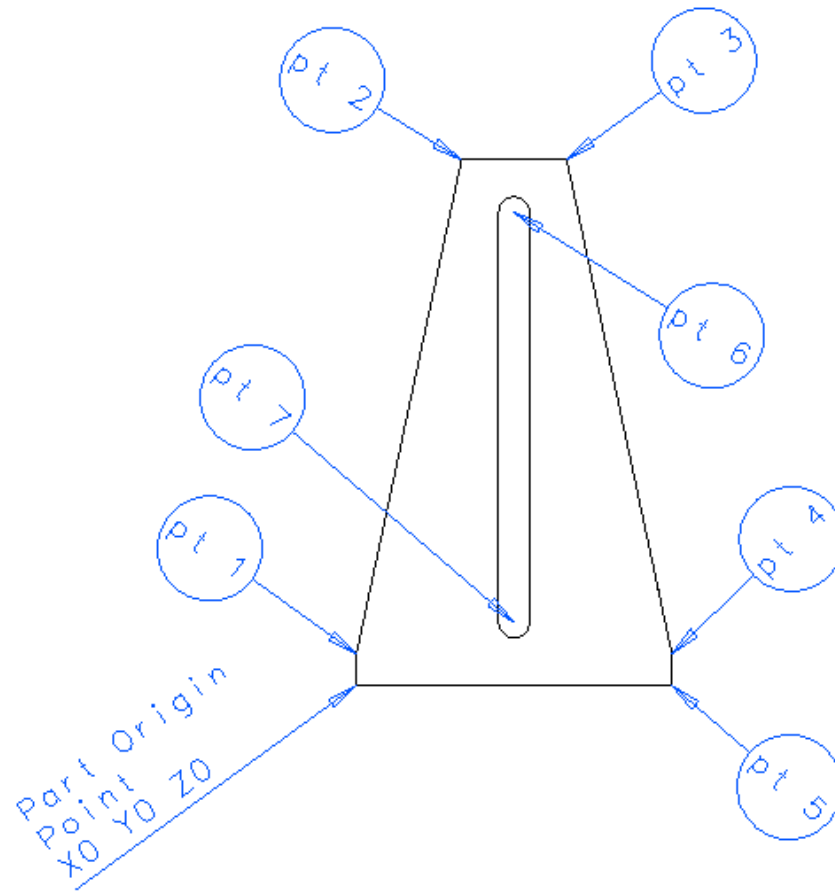
Multiple Work Offsets

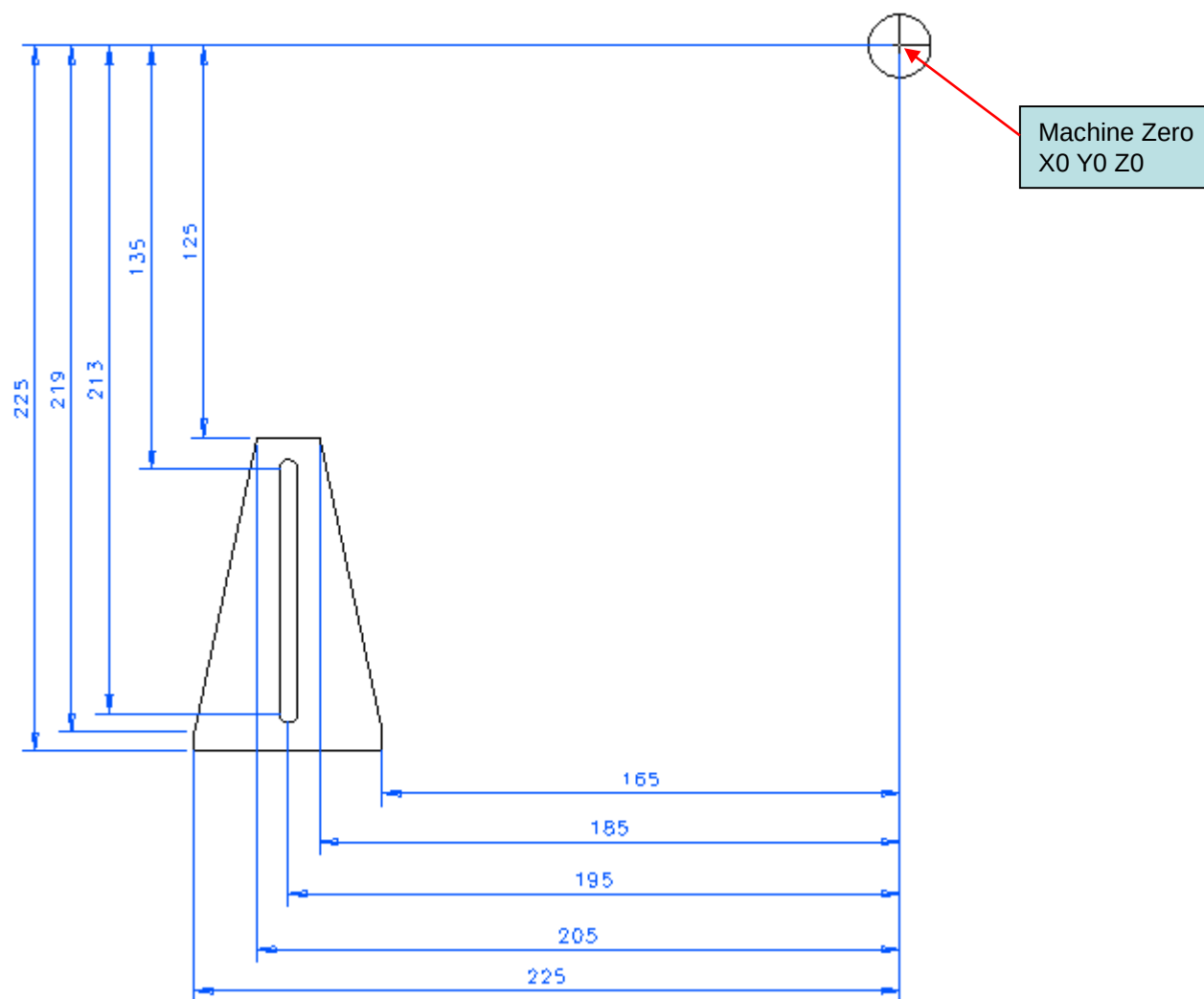


Part Dimensions



Part Target Points

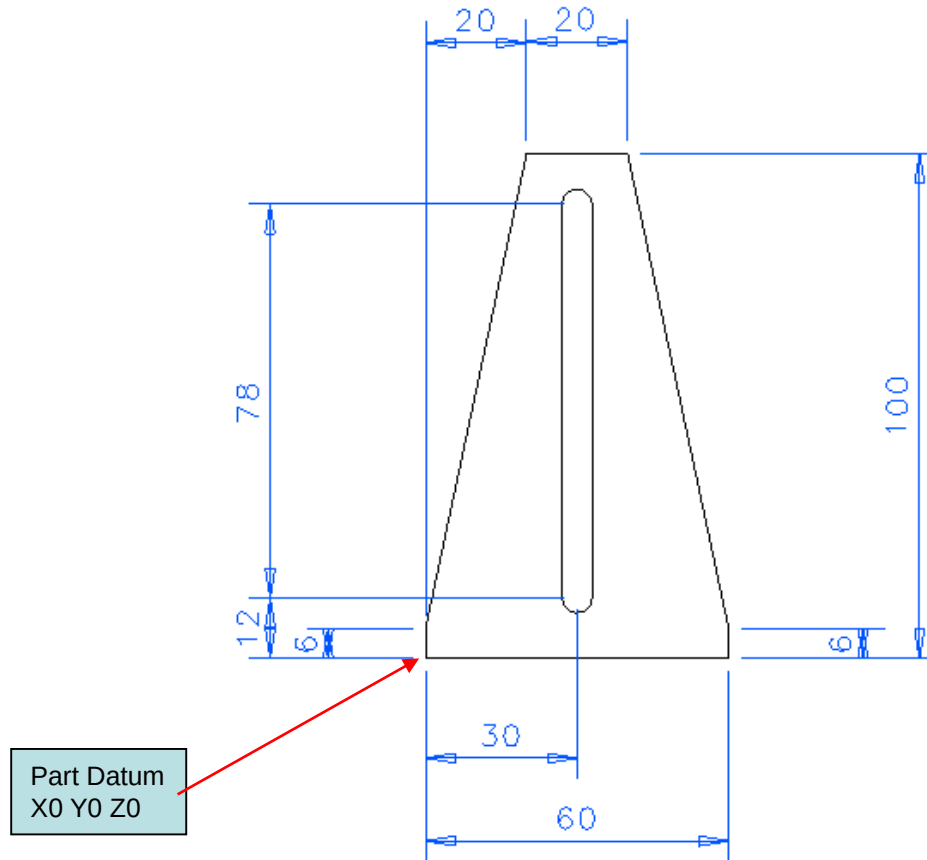




Angle Bracket Coordinates (Machine Zero)

Point #	X Coordinate	Y Coordinate
0	-225	-225
1	-225	-219
2	-205	-125
3	-185	-125
4	-165	-219
5	-165	0
6	-195	-135
7	-195	-213

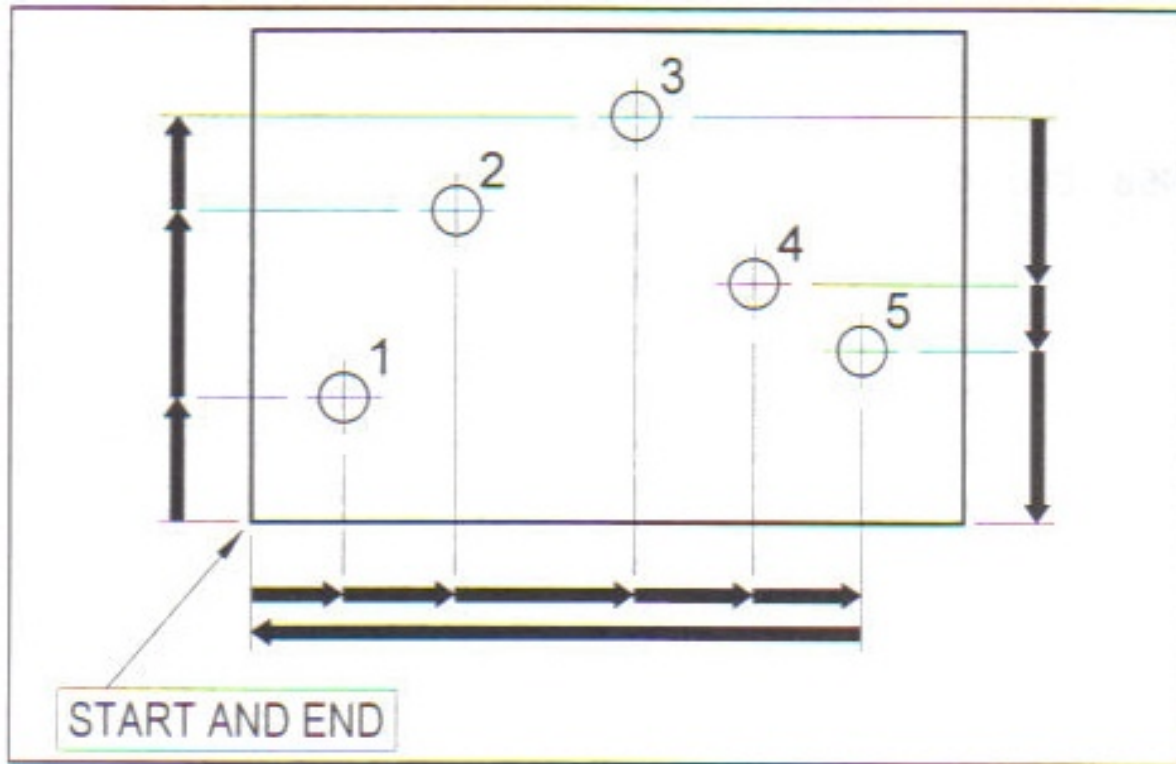
Part Datum Position



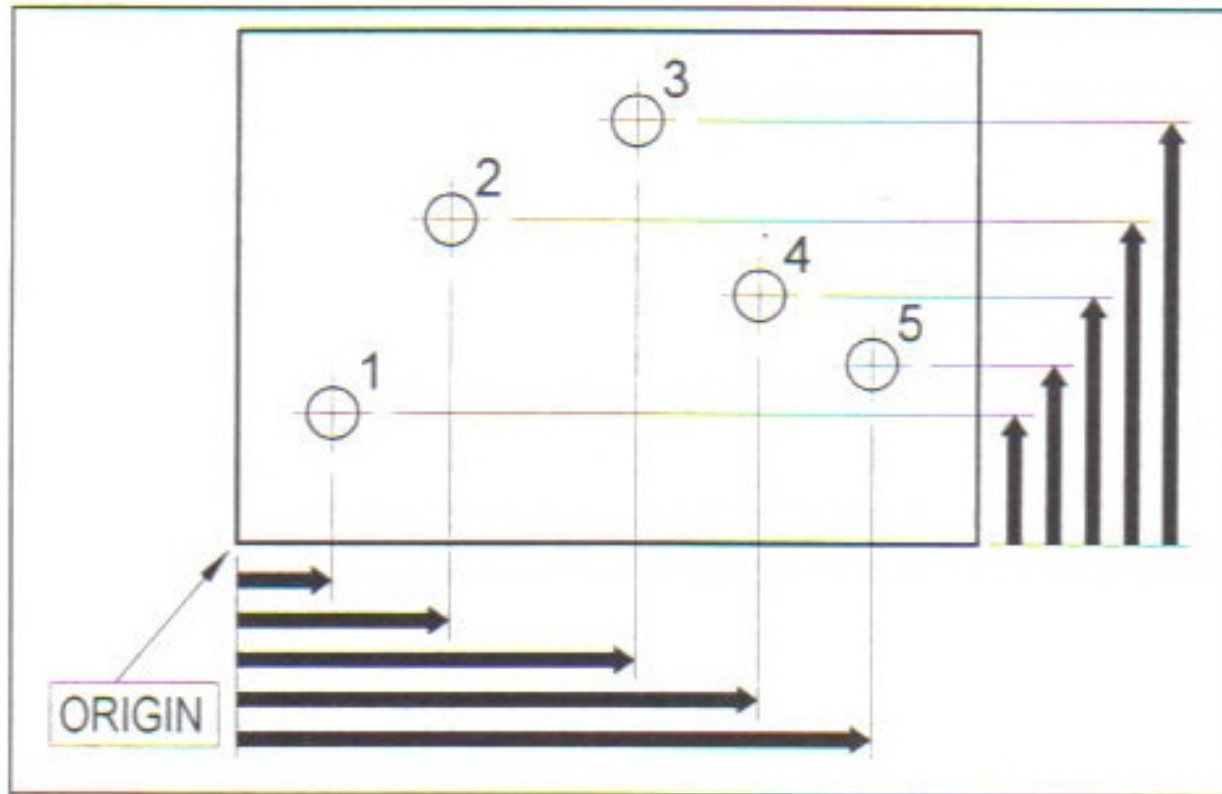
Angle Bracket Coordinates (part datum)

Point #	X Coordinate	Y Coordinate
Datum	0	0
1	0	6
2	20	100
3	40	100
4	60	6
5	60	0
6	30	90
7	30	12

Incremental Positioning



Absolute Positioning



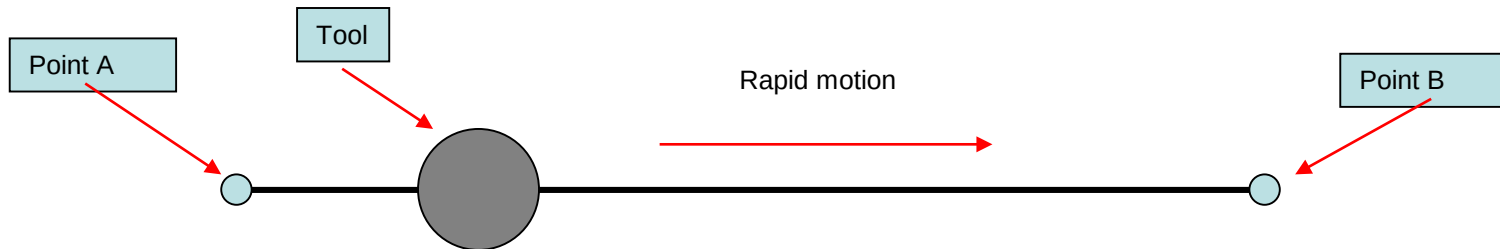
Linear & Circular Interpolation

Target Position Movements

- *There are 4 types of basic point to point movement within CNC programs*
 - *G00 Rapid positioning mode*
 - *G01 Linear interpolation*
 - *G02 Circular interpolation (clockwise)*
 - *G03 Circular interpolation (anti-clockwise / counter-clockwise)*

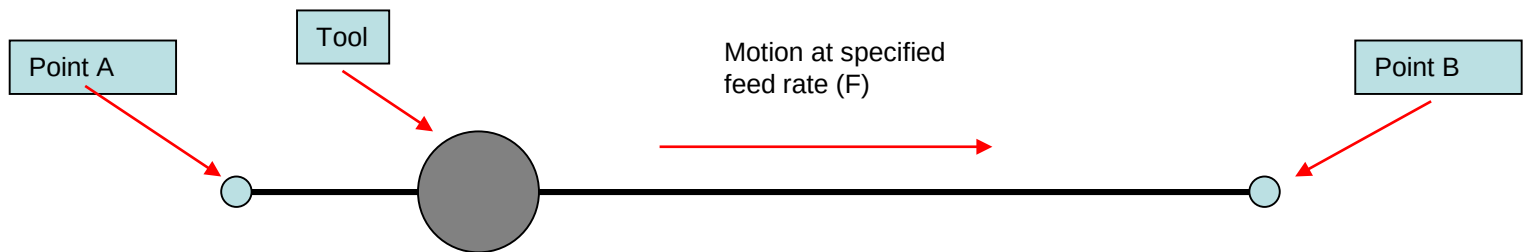
Rapid positioning mode

- *G00 Rapid positioning mode*
- *Used to get from point to point as quickly as possible i.e. when tool is not in contact with the work piece*
- *Example:- G00 Z20.0*



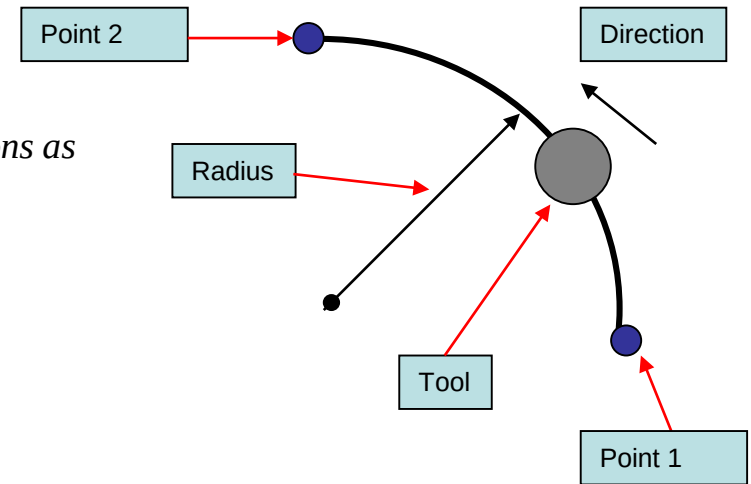
Linear Interpolation

- *Linear Interpolation is used in part programming to make a straight cutting motion from the start position of the cut to its end position.*
- *It will always use the shortest distance the cutting tool path can take*
- *The motion of a linear interpolation will always be in a straight line (point to point)*
- *G01 Linear interpolation*
- *Used to get from point to point at a specified velocity or feed rate*
- *Must be accompanied by an F value on the same line of the program.*
- *The F value determines the velocity of the movement in mm/min, deg/min or in/min depending on the mode selected*
- *Example:- G01 X90.0 Y78.0 F20*



Circular Interpolation

- *G02 Circular interpolation (clockwise)*
- *G03 Circular interpolation (anti-clockwise)*
- *Used to program arcs or complete circles in such applications as*
 - *external radii*
 - *internal radii*
 - *circular pockets*
 - *spherical or conical shapes*
 - *radial recesses*
 - *grooves*
 - *large counter bore*
- *Must be accompanied by an F value to specify the velocity of the cutting tool relative to the work piece*
- *When specifying a circular movement we must input 4 pieces of information*
 - *1 Start target point*
 - *2 Direction*
 - *3 Radius*
 - *4 Finish target point*
- *Example:- G01 X32.0 Y10.00 F20.0 (start point coordinates & feed rate)*
- *G02 X27.0 Y15.00 R5 F20.0 (end point coordinates, radius value & feed rate)*

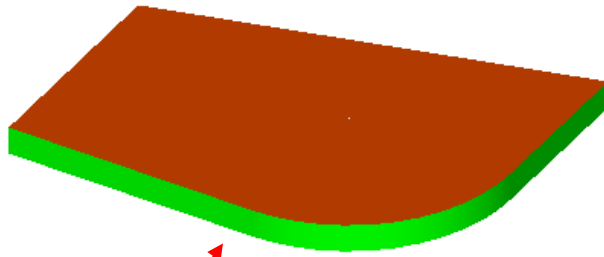


Cutter Radius Compensation

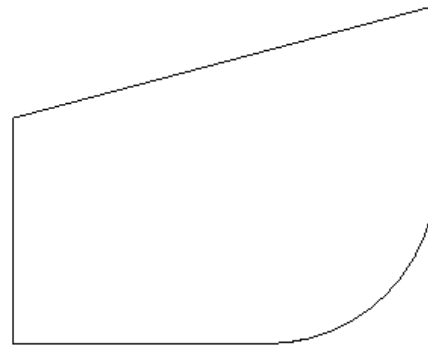
What is CRC?

- *Cutter Radius Compensation (CRC) is a function of modern CNC machines which uses data stored in the machines memory to correct the coordinates within a program for a given tool diameter.*
- *Data of this type (tool lengths, tool diameters, part datum positions etc) is stored in the machines offset register.*

Defining the Part

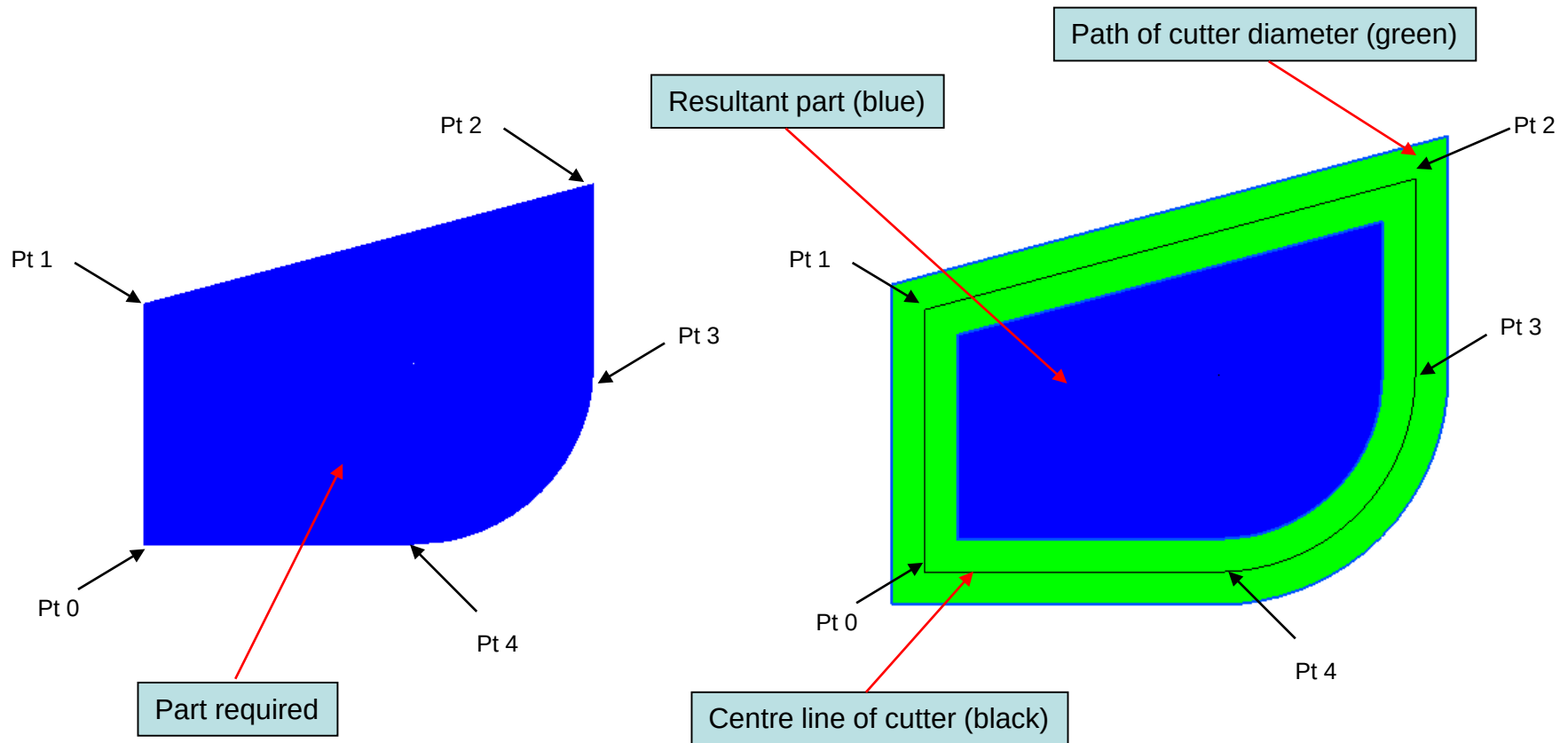


Isometric View

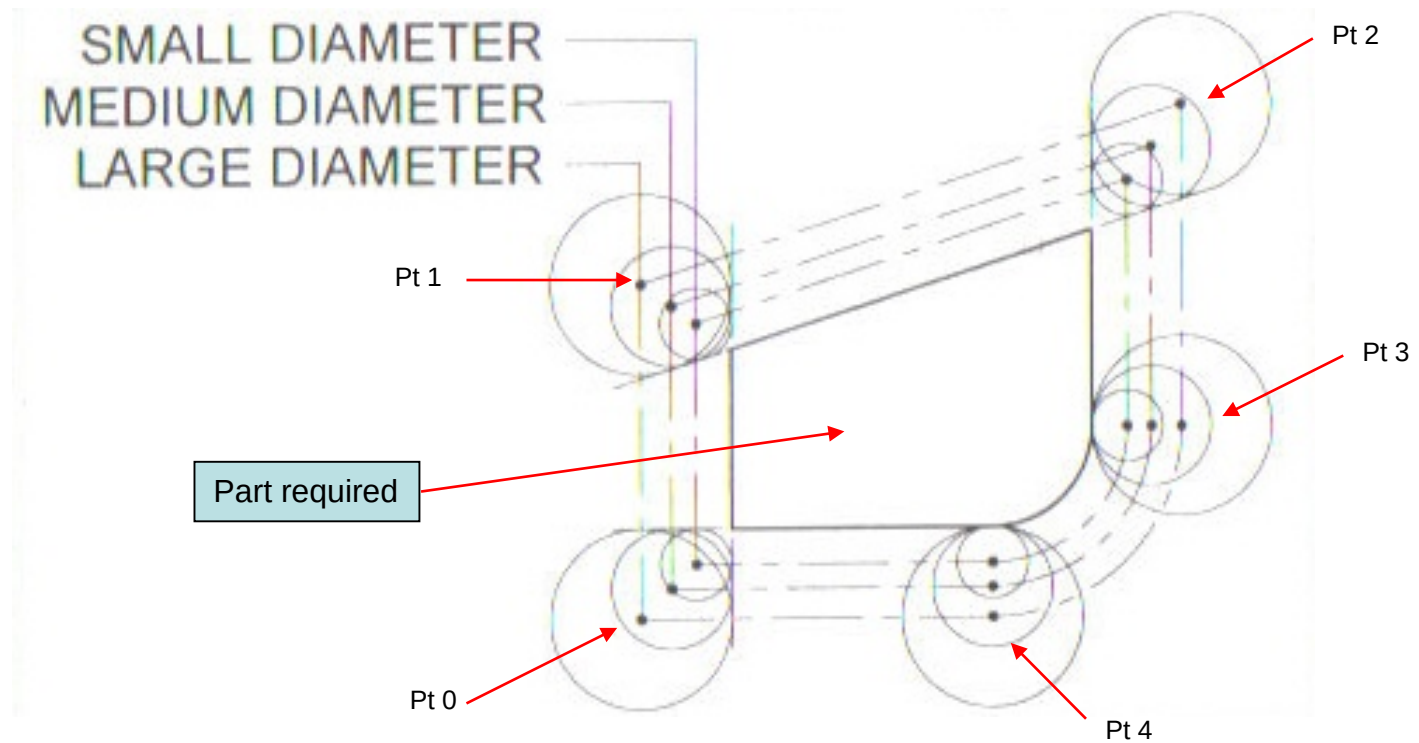


Plan View

Using no offset commands



Applying a manually calculated offset



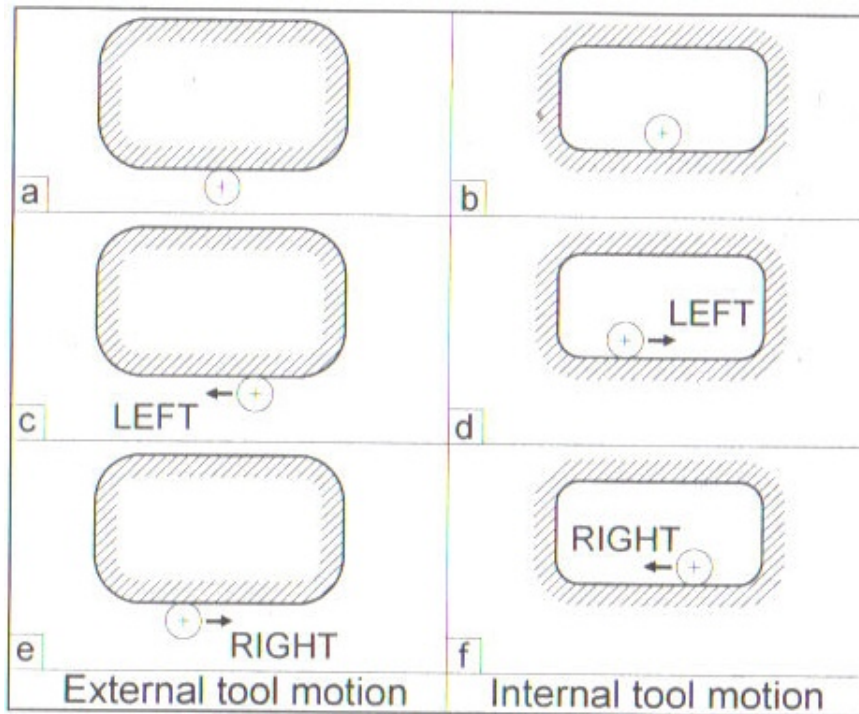
Representing CRC in G&M code

- *The codes **G41** & **G42** are used within a CNC program to indicate that the coordinates which follow should be compensated for cutter width.*
- ***G40** is the code which is used to cancel CRC*
- *The program line which contains the CRC command must also specify which of the offset registers contains the correct tool diameter information*
- ***D0(offset register number)** e.g. **D01**, **D02** or **D03** is the code used to detail which offset register contains the correct tool diameter data*

Direction of Compensation

- *Cutter compensation is classified by **left** or **right** & not by clockwise or anticlockwise*
- *G41 is the code used to identify cutter compensation left (CCL)*
- *G42 is the code used to identify cutter compensation right (CCR)*

Direction of Compensation

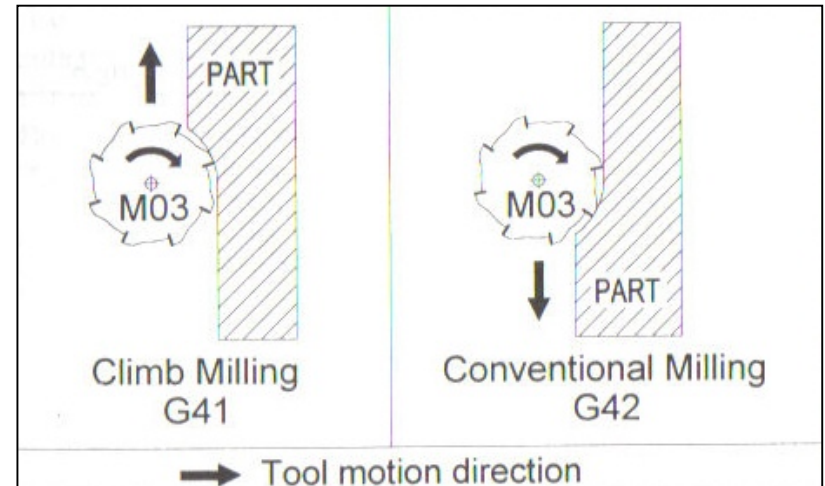


The diagram shows applications of G41 (CCL) & G42 (CCR)

- (a-b) No motion direction shown – left & right is unknown*
- (c-d) Cutter positioned to the left - G41*
- (e-f) Cutter positioned to the right - G42*

Climb & Conventional Milling

- *Cutter Compensation determines Climb or Conventional milling action*
- *If a standard (clockwise) rotation of the tool is used (M03)*
 - *G41 will produce a climb milling condition*
 - *G42 will produce a conventional milling condition*



Using CRC within a CNC program

N	G	M	X	Y	XY Feed	Z	Z Feed
12	G01					Z-7.0	F20
13	G01			Y90.0	F20		
14				Y12.0			
15	G00					Z20.0	
16	G41		D01				
17	G00	X0.0	Y0.0				

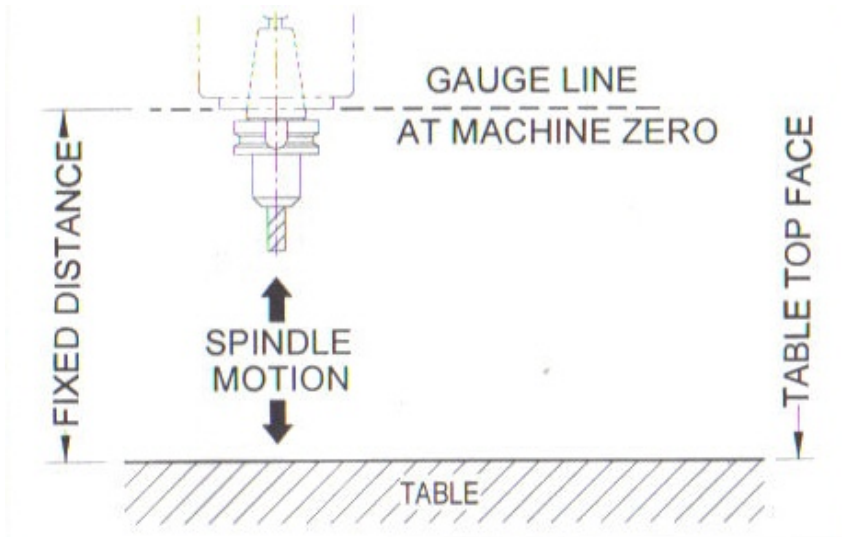
Tool Length Offsets (TLO)

General Principles

- *The length of each cutting tool used must be accounted for in every CNC program*
- *The reason – Each tool has a different length*
- *TLO's can be divided into two groups:*
 - *Actual tool length is known*
 - *Actual tool length is unknown*

- *Standardised tooling systems*
 - *HSK63, HSK100, BT40 and CAT50*
- *Tool length must be associated with the tool holder and in relation to the machine design (gauge line).*

The Gauge Line



- Tool holders use a precision taper pulled in by a pull-bar to accurately locate the tool in the machine spindle
- Precision manufacturing allows the consistent location of the tool holders
- This position is commonly called the gauge line
- The gauge line is an imaginary line used for gauging or measuring along the Z axis

Table Top Face

- *Every CNC machining centre has a built in machine table onto which parts & fixtures can be mounted*
- *The table surface is precision ground to ensure flatness & squareness to the machine axis*
- *The table is located a certain fixed distance from the gauge line. Its position can not be altered (even pallet tables)*

Tool Length Offset Commands

- *The three commands relating to TLO are all preparatory G codes*
 - **G43** *Positive tool length offset*
 - **G44** *Negative tool length offset*
 - **G49** *Tool length offset cancel*
- *Unlike work offset commands G54-G59, G43 & G44 cannot be used without further specification.*
- *They can only be used with an offset number designated by the address H. The H must be followed by up to three digits.*
- *A typical program entry would be the G43 command followed by a Z axis target position and the H offset number e.g.*

N66 G43 Z10. H04
- *This is also a convenient block to add the coolant function for the current cutting tool*

N66 G43 Z10. H04 M08

Procedure for Setting H values on Haas VF-0 Machining Centre

- *Use Automatic Tool changer to select the correct tool in the spindle*
- *Use Jog handle to position the tool on the Z0. position*
- *Select settings/offset and scroll down to the correct tool number in the offset register*
- *Press the Tool Offset Measure key*
- *Select Reset and select the next tool*