

TRC3801 - Mechatronics and Manufacturing Summary

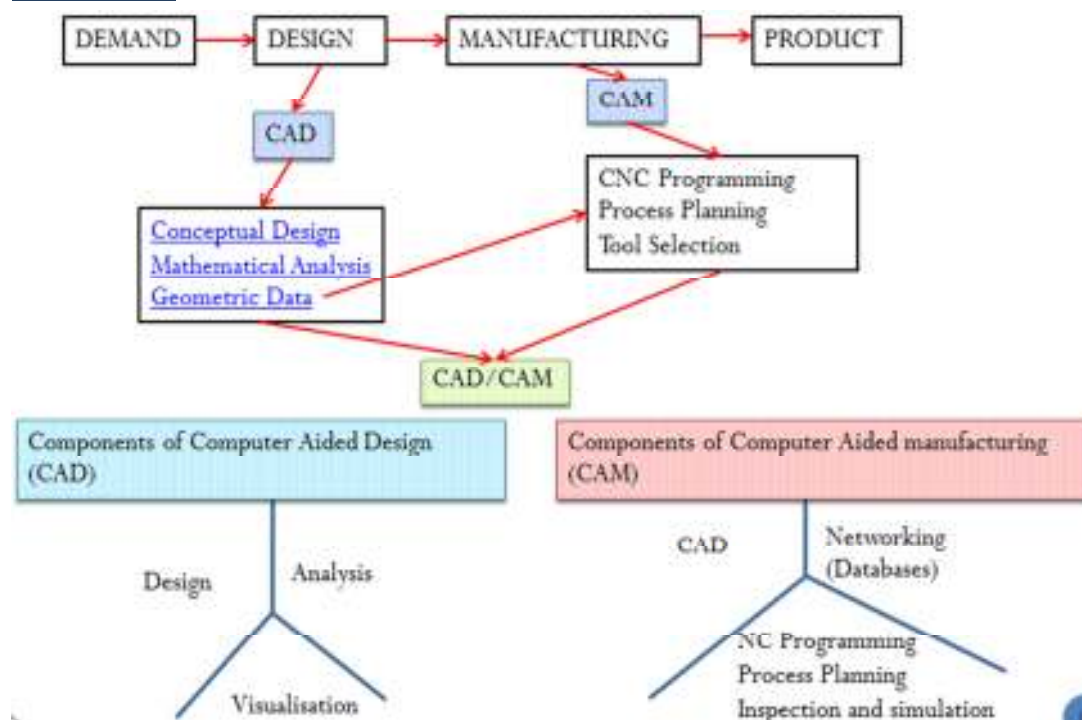
Notes

Contents

Overview.....	3
CNC Manufacturing	3
Basic Operations.....	4
Programming Codes.....	4
Positioning Systems	5
Motion Control.....	5
Cutter Compensation.....	5
Code List	6
Drilling	7
Spot Drilling.....	7
Peck Drilling.....	8
Drilling Codes	8
Polar Coordinates.....	8
Other Canned Cycles	10
Circular Interpolation	10
Manufacturing Systems	11
Factory Layout.....	11
Fixed Position Layout	11
Product Based Layout	11
Process Based Layout.....	12
Cell/Group Layout.....	12
Production Relationships	13
Manufacturing Philosophies	13
Concurrent Engineering	13
Lean Manufacturing.....	14
Agile Manufacturing	14
Virtual Manufacturing.....	14
Push-Pull Production System	14

Manufacturing Cells	15
Analysis	15
Process Planning.....	16
Group Technology	17
GT Coding Systems.....	17
Optiz System	18
Vuoso-Praha Coding System	19
Flexible Manufacturing System	19
Production Flow Analysis	19
Rank Order Cluster Algorithm.....	20
Line Balancing Algorithms	21
Automated Feature Recognition.....	23

Overview

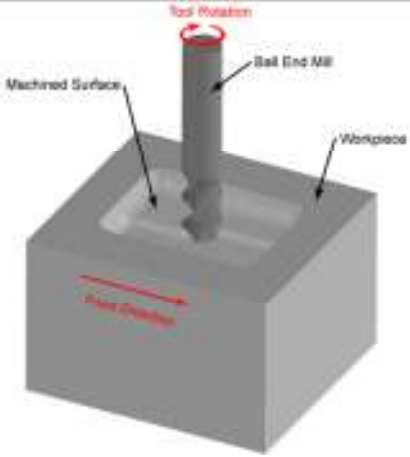
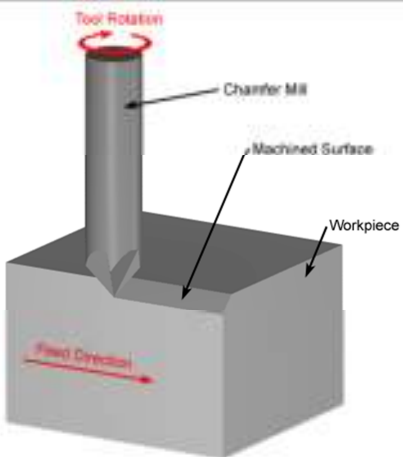


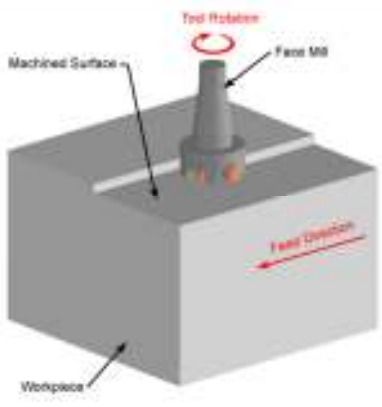
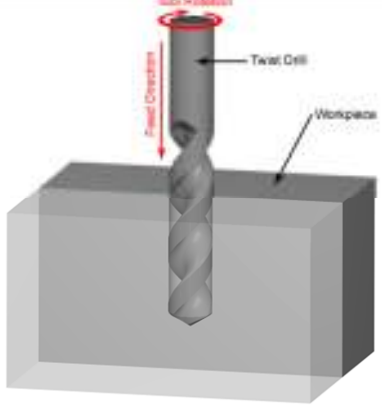
CNC – Computer Numerical Control.

- A specified instruction set is used to execute commands on a CNC machine in order to manufacture a part

CNC Manufacturing

Milling operations:

End Milling	Chamfer Milling
Makes peripheral or slot cuts to machine a feature such as profile or slot.	Makes peripheral cut along edge of workpiece to create angled surface.
	

Face Milling	Drilling
Machines a flat surface to provide a smooth finish.	Cuts a blind or through hole.
	

Others:

- **Boring** – commonly after drilling a hole in order to enlarge diameter or obtain precise dimension
- **Countersinking** – Enlarges top portion of existing hole to a cone-shape
- **Reaming** – Enlarges existing hole, obtain more accurate diameter and smooth finish
- **Tapping** – Cuts internal threads into an existing hole

Basic Operations

Programming Codes

■ 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 - Action code or Machine code or Miscellaneous function
--	--

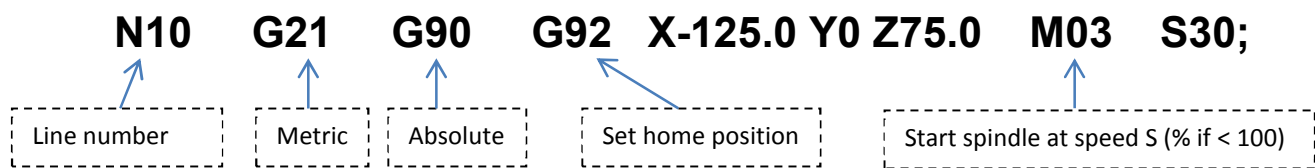
For example:

O0001;

N010 G02 X1500 Y2000 R20 F100 S400;

Omitted terms are assumed to be 0 or as previously defined.

General initialisation:



Positioning Systems

Absolute – position for movement is specified with respect to axis origin (G90)

Incremental/Relative – Specifies movement relative to the current tool location (G91)

Motion Control

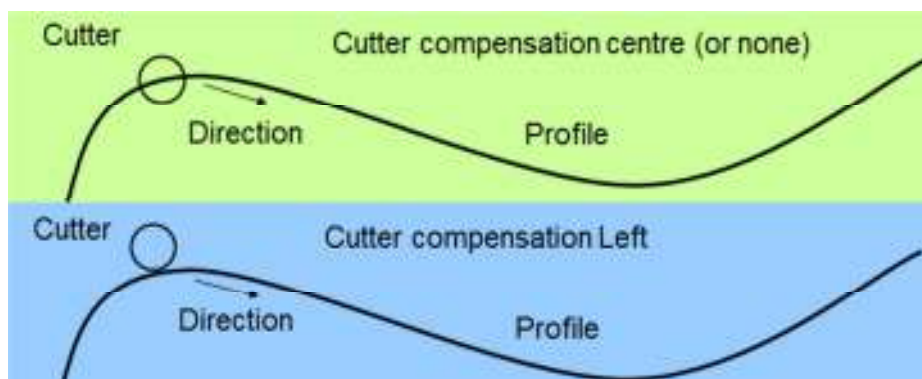
Point-to-Point – movement at maximum rate to a specified point, to position tool for other operations (G00)

Continuous path – Slower, controlled movement with specified feed rate for cutting (G01 (linear), G02, G03 (circular))

Cutter Compensation

Can be used to automatically account for tool radius so that exact part profile can be specified for cutting motion (G41 – left, G42 – right).

- Activating cutter compensation in the block **before** movement (separate blocks) causes compensation to be effective from the start



Must previously store radius of tool using G10, and access it when G41 is called:

```
O0001;  
-----  
N 0010G10P01R1.5;  
N0020M08;  
N0030 G17 G41 G01 X-10.0 Y-10.0 D01 F80.0;  
N0040 Y10.0;
```

'L' word can be used at end of line to repeat the execution of the line a specified number of times (usually used in incremental distance mode G91).