Design for manufacturing
MED 421

By
Dr.Gomaa Ahmed Abdelrazek

2025

Contents

No	Title	Page
	CHAPTRR.1:	
1.1	Introduction to computer aided integrated design and manufacturing,	4
1.2	Hardware and software ,	17
1.3	Two and three - dimensional simulations, planned analysis,	23
	CHAPTRR.2	
2.1	pre –processing and post – processing,	25
2.2	product, Cycle, CAD /CAM	35
2.3	Automation programming of lathe, drilling and milling machines, computer assisted part programming,	43
	CHAPTRR.3	
3.1	DNC, CNC, Adaptive control, industrial robotics,	75
3.2	Manufacturing CLM,FMS, Manufacturing cells,	86

MED 421 Design for manufacturing

3 Cr . hrs =[2Lect. + 2Tut + 1Lab] Prerequisite: MED 321

Introduction to computer aided integrated design and manufacturing, Hardware and software ,Two and three -dimensional simulations, planned analysis, pre –processing and post – processing, product, Cycle, CAD /CAM and Automation programming of lathe, drilling and milling machines, computer assisted part programming, DNC, CNC, Adaptive control, industrial robotics, Manufacturing CLM,FMS, Manufacturing cells, practical training.

مكص ٤٢١ التصميم للتصنيع

مقدمة فى تكامل التصميم للتصنيع، البرامج والمعدات الخاصة،المحاكاة ثنائية وثلاثية والابعاد، التحويلات التخطيطية والتجهيزات المسبقة ، والتحضيرات التابعة،اتمته والفريزة والمكشطة) برمجة ماكينات التشغيل ذات التحكم العددى ، الروبوتات الصناعية وبرمجة الروبوت، مجموعة التكنولوجيا ، نظم التصنيع المرنة والخلايا الصناعية، تدريب عملى

Laboratory:

1. Practice on CAD /CAM computer packages with case studies.
2. Programming of Lathe, Drilling, and Milling Machines with producing standard pieces.
3. Programming of aRobot to follow aspecified path in space for loading and unloading CNC Machines.

References:

Pahl and W Beitz, Engineering Design: A systematic approach, springer - verlag London 3rdEd., 2007.

- Xun xu, lntegrating Advanced computer –Aided Design, Manufacturing, and Numerical control: principles and implantations,(premier Reference) lnformation science Reference, 1stEd., 2008.

1.1.Introduction to Computer Integrated Design and Manufacturing (CAD/CAM/CIM)

Introduction:

1-Technologies concerning computer-aided design, process planning, manufacturing and numerical control, have matured to a point that commercialized software solutions and industrial systems can be acquired readily.

2-The term “islands of automation” has been used to describe these disconnected groups of systems with no obvious integration points other than the end user.

3-In some cases, where integration does not exist among these computer-aided solutions, promising product technologies may come to a sudden halt against these barriers.

4- CAD and CAM systems are unified by the same data format so that data incompatibilities between them are eliminated.

Introduction to CAD/CAM
(Computer Aided Design and Computer Aided Manufacturing)

- CAD/CAM = Computer Aided Design and Computer Aided Manufacturing. It is the technology concerned with the use of computers to perform design and manufacturing functions.

Computer-aided design CAD can be defined as:

- 1- the use of computer systems to perform certain functions in the design process.
- 2- the use of computer systems to assist in the creation, modification, analysis, or optimization of a design.
- 3- the creation and manipulation of pictures (design prototypes) on a computer to assist the engineer in the design process

Computer-aided manufacturing (CAM) can be defined as :

- 1- the use of computer systems to plan, manage and control the operations of manufacturing plant through either direct or indirect computer interface with the plant's production resources.

- Need for CAD/CAM:
  - To increase productivity of the designer
  - To improve quality of the design
  - To improve communications
  - To create a manufacturing database
  - To create and test tool paths and optimize them
  - To help in production scheduling and models
  - To have effective shop floor control

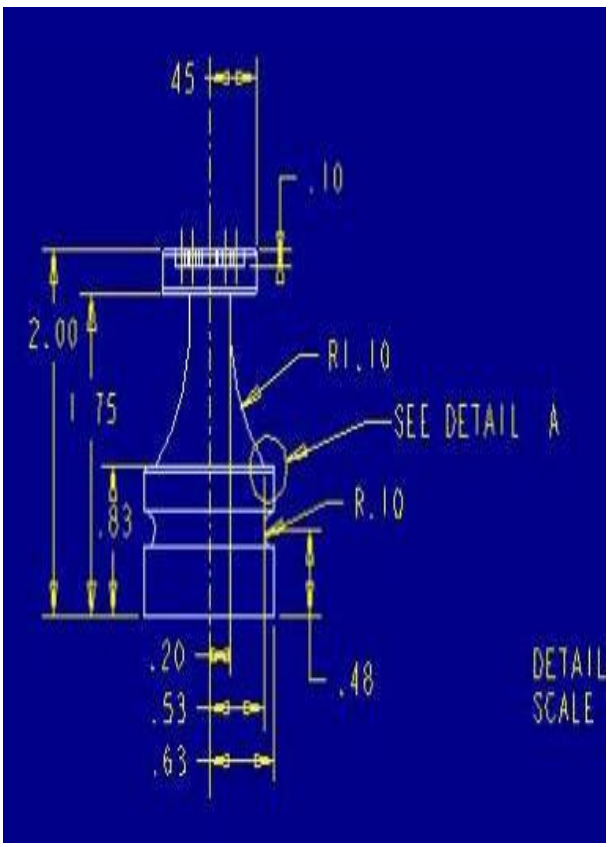
How do CAD/CAM systems work?

Developing NC(numerical control) code requires an understanding of:

1. Part geometry
2. Tooling
3. Process plans
4. Tolerances
5. Fixturing

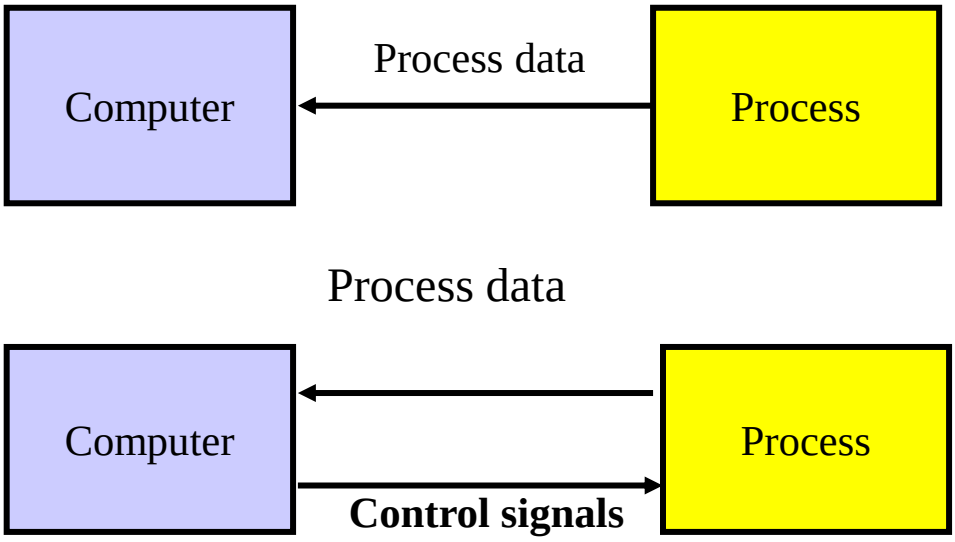
- Most CAD/CAM systems provide access to:

1. Part geometry
2. Tooling

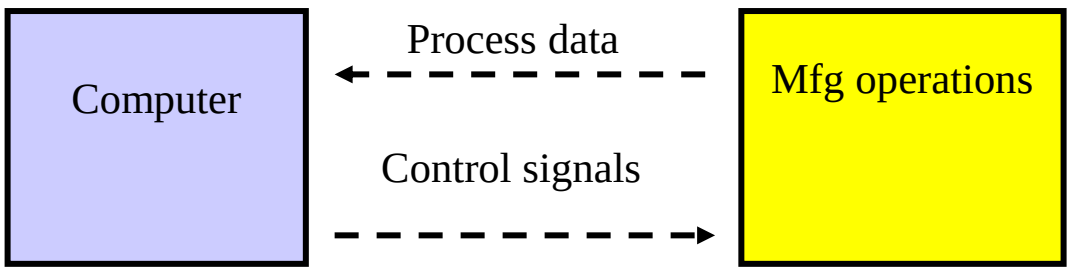


From CAM definition, the application of CAM falls into two broad categories:

1. Computer monitoring and control .



2. Manufacturing support application .



Advantages of CAD/CAM systems

- Greater flexibility.
- Reduced lead times.
- Reduced inventories.
- Increased Productivity.
- Improved customer service.
- Improved quality.
- Improved communications with suppliers.

- Better product design.
- Greater manufacturing control.
- Supported integration.
- Reduced costs.
- Increased utilization.
- Reduction of machine tools.
- Less floor space.

CAD Packages

- AutoCAD
- ProEngineer
- Ideas
- SolidWorks
- Unigraphics
- Others

Computer Integrated Manufacturing (CIM)

- Definition:
- the extensive use of computers in design, analysis, drafting and manufacturing
- Enables integration of all functions to minimize costs by optimizing communication in a flexible environment

- Objective:
- Automate and integrate the manufacturing system at the following integrate levels:
  - Obtain product specification
  - Design a method for producing the product (including design, part list, processes, manufacturing plan)
  - Schedule for production
  - Plan to purchase raw materials in accordance with schedule
  - Production in the factory
  - Monitor results for quality, cost, etc.
  - Shipping, distribution, marketing

CIM Impact

- Design of product, the manufacturing process and the redesign or modification of the process
- Continuous communication between designers and manufacturing engineers (geometry of part = geometry of cutting tool path)
- One common database or archive for product , everyone looking at latest version
- Provides infrastructure for concurrent engineering , design for manufacturability taking into account th design specs, esthetics, cost, materials, etc.

Characteristics of CIM:

- Complete automation of a manufacturing plant
- Processes functioning under computer control
- Digital information tied them together

CIM technology flexibility:

Manufacturing engineers are required to achieve the following objectives to be competitive in a global context.

- Reduction in inventory
- Lower the cost of the product
- Reduce waste
- Improve quality
- Increase flexibility in manufacturing to achieve immediate and rapid response to:
    - Product changes
    - Production changes
    - Process change
    - Equipment change
    - Change of personnel

Typical Elements of a CIM System



- manufacturing planning & control
  - shop floor control, inventory control, ...
- manufacturing engineering
  - CAD/CAM, CAPP, coding & classification, ...
- manufacturing processes
  - NC/CNC/DNC, FMS, robots, material handling systems, ...
- indirect elements:
  - sales order processing (& marketing)
  - finance & accounting

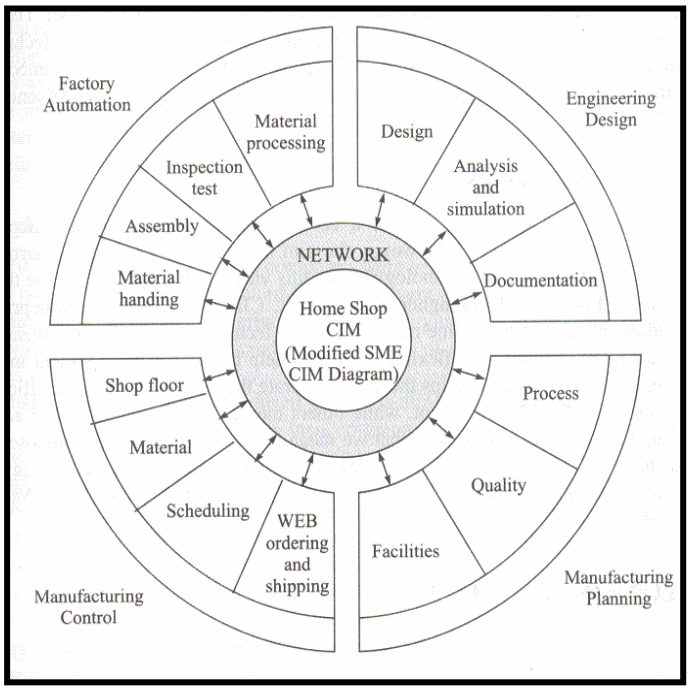
CIM is recognized as Islands of Automation.
They are:

1. CAD/CAM/CAE/GT
2. Manufacturing Planning and Control.
3. Factory Automation
4. General Business Management

Key Challenges

- Integration of components from different suppliers
- Data integrity
- Process control

(The computer and automated systems association of the society of
Manufacturing Engineers) (CASA/SME)



CASA/SME's CIM Wheel

Concepts Terminology

- CIM - Computer - Integrated Manufacturing
- CHIM – Computer - Human - Integrated Manufacturing
- IMS - Integrated Manufacturing Systems
- CAD - Computer - Aided Design
- CAM - Computer - Aided Manufacturing
- CAPP - Computer - Aided Process Planing
- CAD/CAM - Computer - Aided Design and Manufacturing
- CADMAT - Computed - Dided Design, Manufacturing and Testing
- PLM/PDM - Process Lifecycle Management / Process Data Management
- FMS - Flexible Manufacturing Systems
- RE – Reverse Engineering
- CE - Concurrent Engineering
- RP – Rapid Prototyping

1.2.Hardware and Software

CIM HARDWARE AND CIM SOFTWARE:

CIM Hardware:

It most comprises the following:

I. Manufacturing equipment such as CNC machines or computerized work centers,robotic work cells, DNC/FMS systems, work handling and tool handling devices, storage devices, sensors, shop floor data collection devices, inspection machines etc.

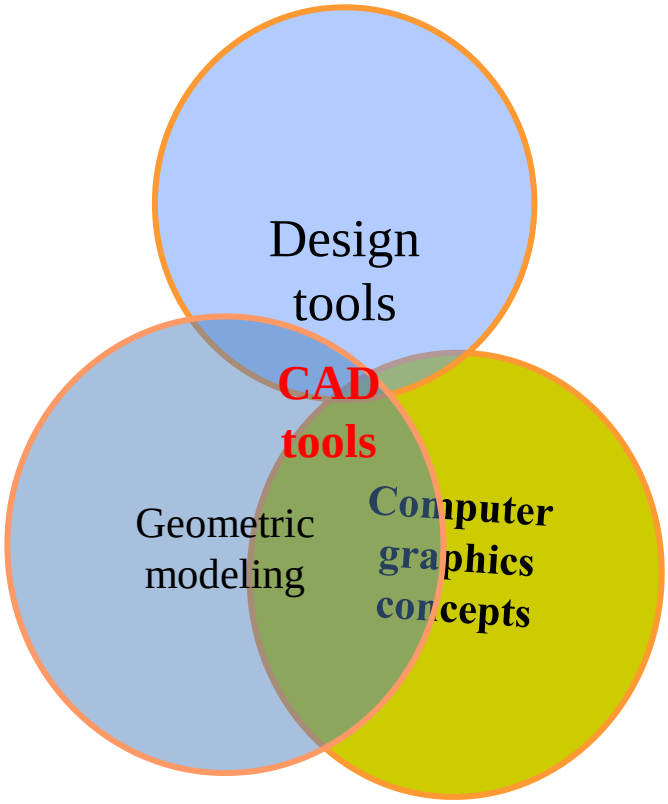
II. Computers, controllers, CAD/CAM systems, workstations / terminals, data entry terminals, bar code readers, printers, plotters and other peripheral devices , modems, cables, connectors etc.,

CIM software :

It most comprises computer programs to carry out the following functions:

- Management Information System
- Job Tracking
- Sales
- Inventory Control
- Marketing
- Shop Floor Data Collection
- Finance
- Order Entry
- Database Management
- Materials Handling
- Modeling and Design
- Device Drivers
- Analysis
- Process Planning
- Simulation
- Manufacturing Facilities Planning
- Communications
- Work Flow Automation

Definitions of CAD Tools Based on Their Constituents



Definition of CAD Tools Based on Their Implementation in a Design Environment

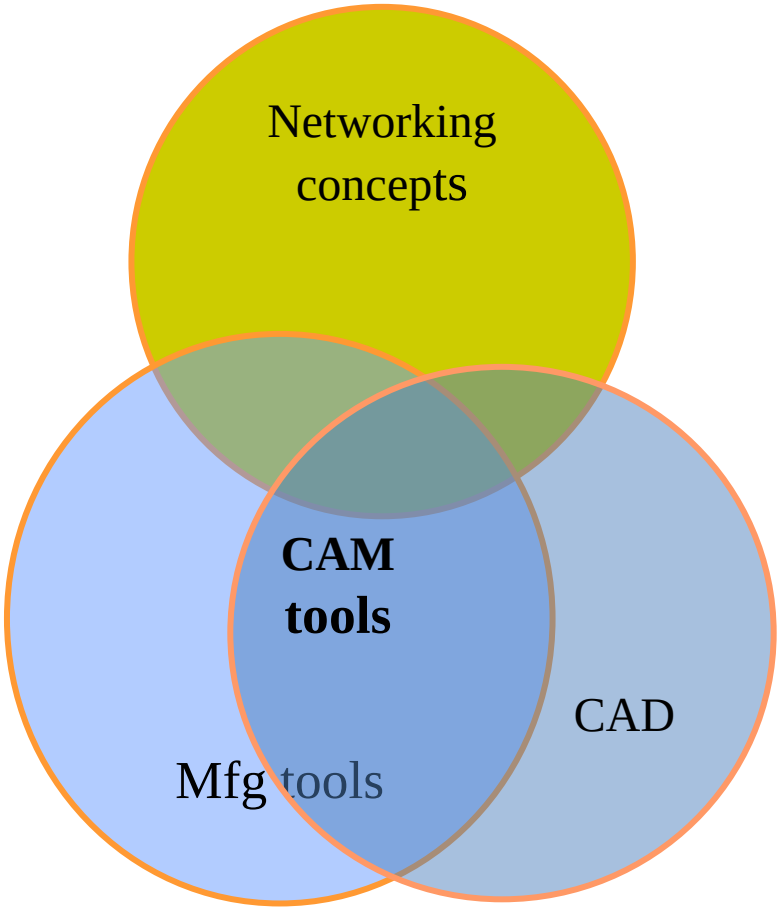
Hardware
(control unit; display terminals; I/O devices)

Design tools + Computer

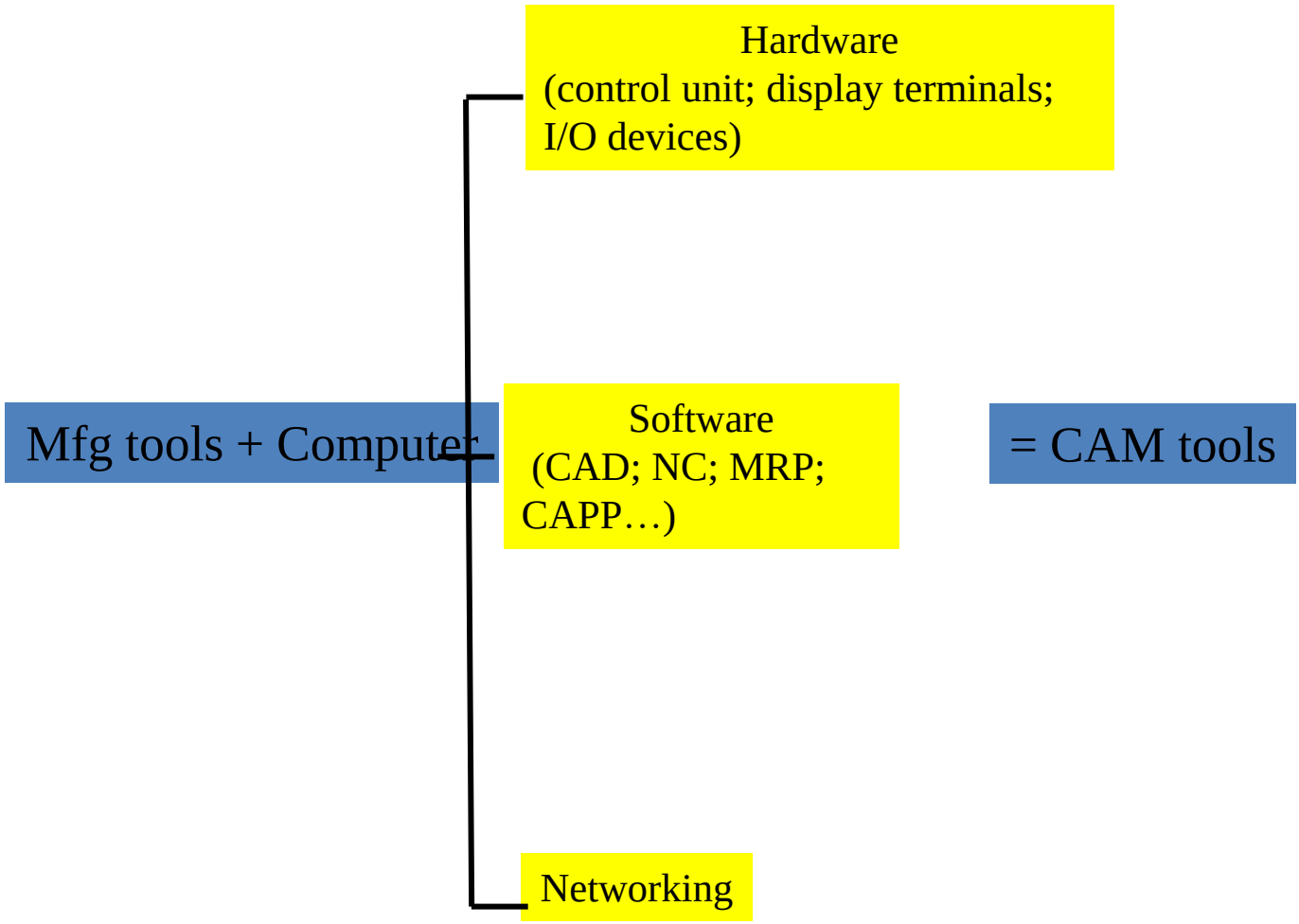
= CAD tools

Software
(graphics; modeling; applications, programs)

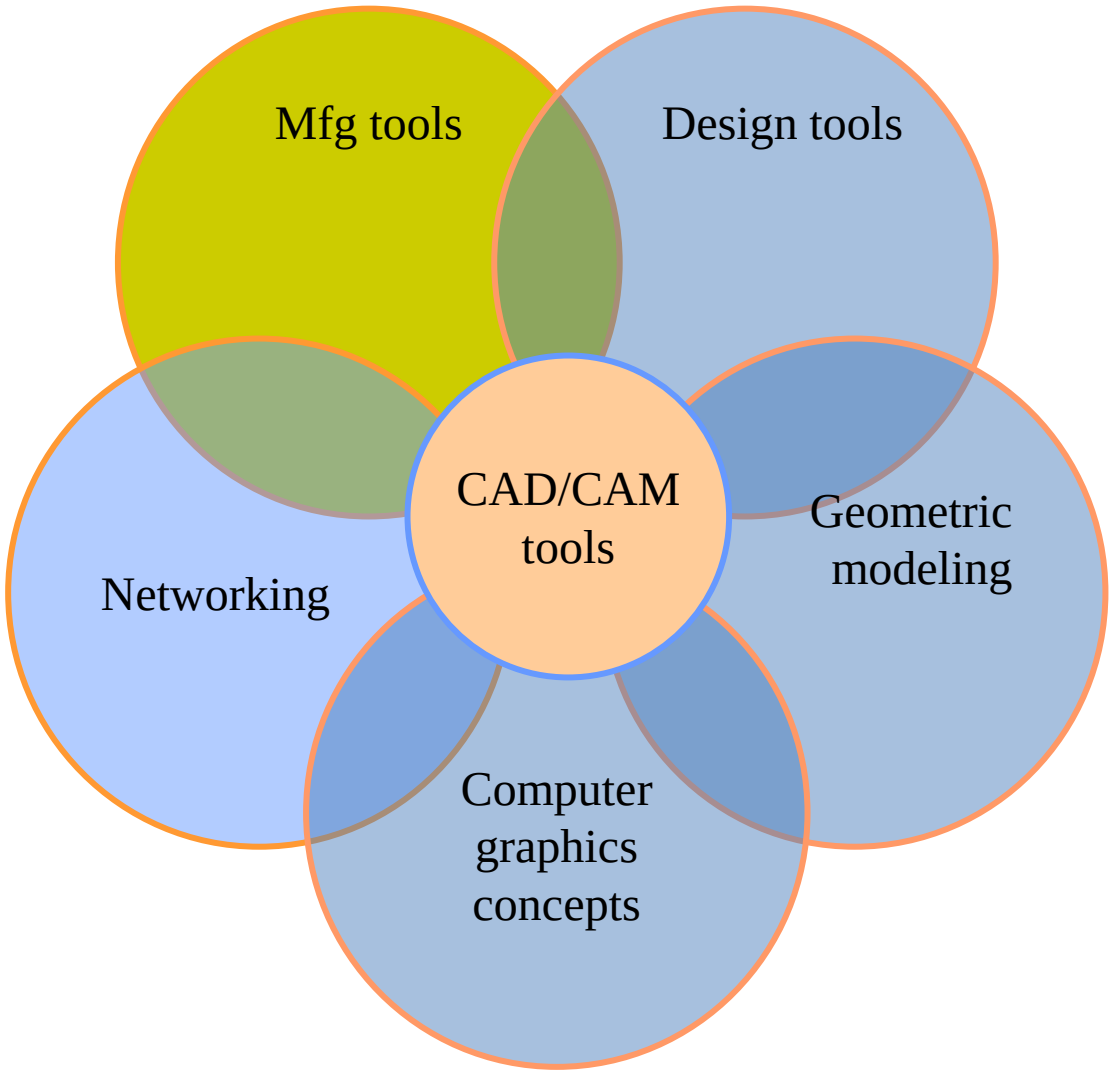
Definitions of CAM Tools Based on Their Constituents



Definition of CAM Tools Based on Their Implementation in a Manufacturing Environment



Definitions of CAD/CAM Tools Based on Their Constituents



Definition of CAD/CAM Tools Based on Their Implementation
in an Engineering Environment

Hardware

Design and Mfg tools + Computer + Software = CAD/CAM tools

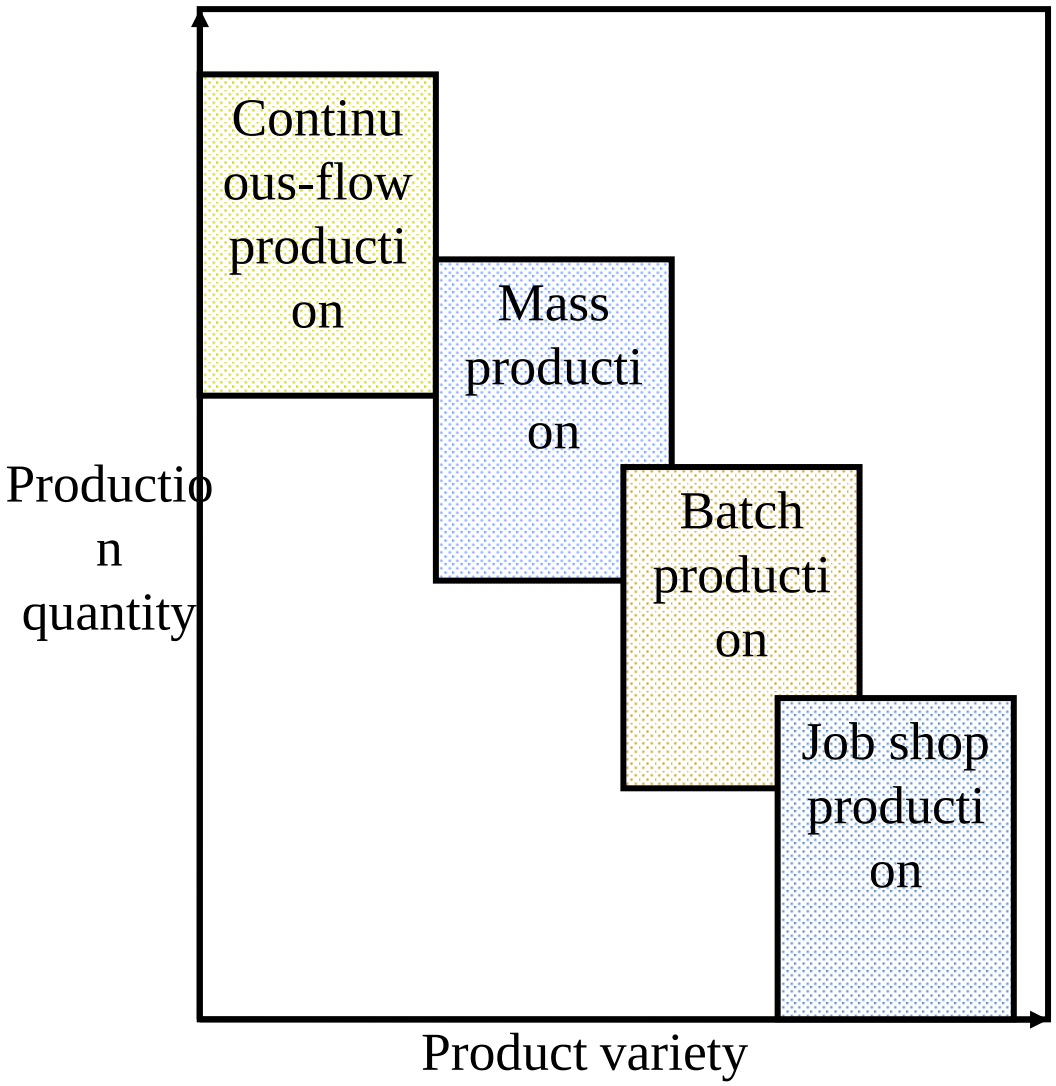
Networking

Automation and CAD/CAM

Automation can be defined as the technology concerned with the application of complex mechanical, electronic, and computer-based systems in the operation and control of manufacturing systems.

Most of the automated production systems implemented today make use of computers. CAD/CAM in addition to its particular emphasis on the use of computer technology, is also distinguished by the fact that it includes not only the manufacturing operations but also the design and planning functions that precede manufacturing.

manufacturing systems.



Category	Automation achievements
Continuous-flow process	•Flow process from beginning to end •Sensors technology available to measure important process variables •Use of sophisticated control and optimization strategies •Fully computer automated lines
Mass production of discrete products	•Automated transfer machines •Dial indexing machines •Partially and fully automated assembly lines •Industrial robots for spot welding, part handling, machine loading, spray painting, etc. •Automated material handling systems •Computer production monitoring
Batch production	•Numerical control (NC), direct numerical control (DNC), computer numerical control (CNC). •Adaptive control machining •Robots for arc welding, parts handling, etc. •CIM systems.
Job shop production	•Numerical control, computer numerical control

2-D&3-D Simulation

INTRODUCTION

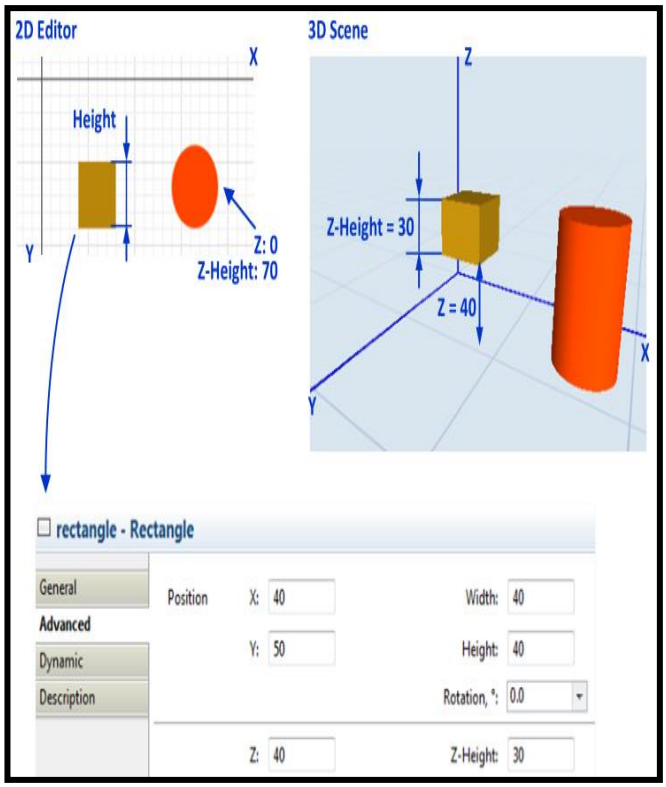
2-D&3-D Simulation

1- 2D and 3D space in simulation models, enable us to create high-quality interactive technical-looking 2D and 3D animations.

2-The graphical editor is two-dimensional, and the natural scenario of building a 3D animation starts with a 2D (X,Y) plane that is the "XY projection" .

3- The models originally designed as 2D can be converted into 3D easily by defining the Z-properties of the 2D shapes.

4- In 3D, the rectangle shape will become parallelepiped (a solid whose faces are all parallelograms) and the oval will become a cylinder (see the Figure).



Definitions:

1-A model is a physical, mathematical, or logical representation of a system entity, phenomenon, or process.

2-A simulation is the electronic implementation of a mode or process .

3-A simulation brings a model to life and shows how a particular object or phenomenon will behave.

4-It is useful for testing, analysis or training where real-world systems or concepts can be represented by a model.

CLASSES OF SIMULATIONS:

The three classes of models and simulations are virtual, constructive, and live:

• Virtual simulations:

represent systems both physically and electronically. Examples are air-craft trainers, the Navy's Battle…..

• Constructive simulations:

Represent a system and its employment and include:

computer models, analytic tools, mockups, Flow Diagrams, and Computer-Aided Design/ Manufacturing (CAD/CAM).

• Live simulations:

Include simulated operations with real operators and real equipment. Examples are fire drills, operational tests,..

Constructive Systems Simulations

1-The purpose of systems simulation is to develop descriptions of system solutions.

2-Accordingly, constructive simulations are important products in all key system engineering tasks and activities.

3-Computer- aided tools can allow more in-depth and complete analysis of system requirements early in design.

Computer-Aided Design (CAD) simulation tools :

1-CAD simulation tools are used to describe the product electronically to facilitate and support design decisions.

2-It can model diverse aspects of the system such as how diagnostics will be performed.

3-It is used to lay out systems or components for sizing, positioning, and space allocating using two- or three-dimensional displays.

4-It uses three-dimensional "solid" models to ensure that assemblies, surfaces, intersections, interfaces, etc.

5-Most CAD tools automatically generate isometric and exploded views of detailed dimensional and assembly drawings, and determine component surface areas, volumes, weights, moments of inertia,centers of gravity, etc.

6-Additionally, many CAD tools can develop three-dimensional models of facilities, operator consoles, maintenance workstations, etc.

Computer-Aided Engineering (CAE).

1-CAE provides automation of requirements and performance analyses

2- It normally would automate technical analyses such as stress ,thermodynamic, acoustic, vibration, or heat transfer analysis.

3-Additionally, it can provide automated processes for functional analyses such as fault isolation and testing, failure mode, and safety analyses.

4-CAE can also provide automation of life-cycle-oriented analysis necessary to support the
design.

5-Maintainability, producibility, human factor, logistics support, and value/cost analyses are
available with CAE tools.

Simulation of CAD/CAM system in CNC Machining:

1-After a NC programmer finishes programming, and before an operator begins to manufacture the geometry part, it is very helpful for them to watch a graphic representation of the path superimposed on the part geometry.

2-A CAD/CAM system can generate automatically the optimal cutter path, to move from one machining feature to the next. **This is simulation.**

3-CAD/CAM system makes CNC manufacturing process simulation more powerful and clear than any other simulations

Toolpath Simulation and Machine Tool Control:

1-Simulation provided a basic toolpath simulation in a 2D simulations .

2- Machine works for 3D simulation and performs solid toolpath verification .

3-The simulation software provides graphical optimal tool path simulation .

Simulate machining:

1- Simulation occurs before posting and before the G-codes are computed in the post-processor.

2- Parts can be rotated during verification.

3-The entire machine tool including fixtures and the part are simulated in 5-axis operations. In 2- and 3-axis operations the entire machine tool is not simulated.

Benefits of selecting Simulation software:

1-Simulation software prevents the NC programs errors that can cause serious problems during the communication and machining.

2-For instance, if the dimension of the drill operation is too deep, the cutter may drill into a

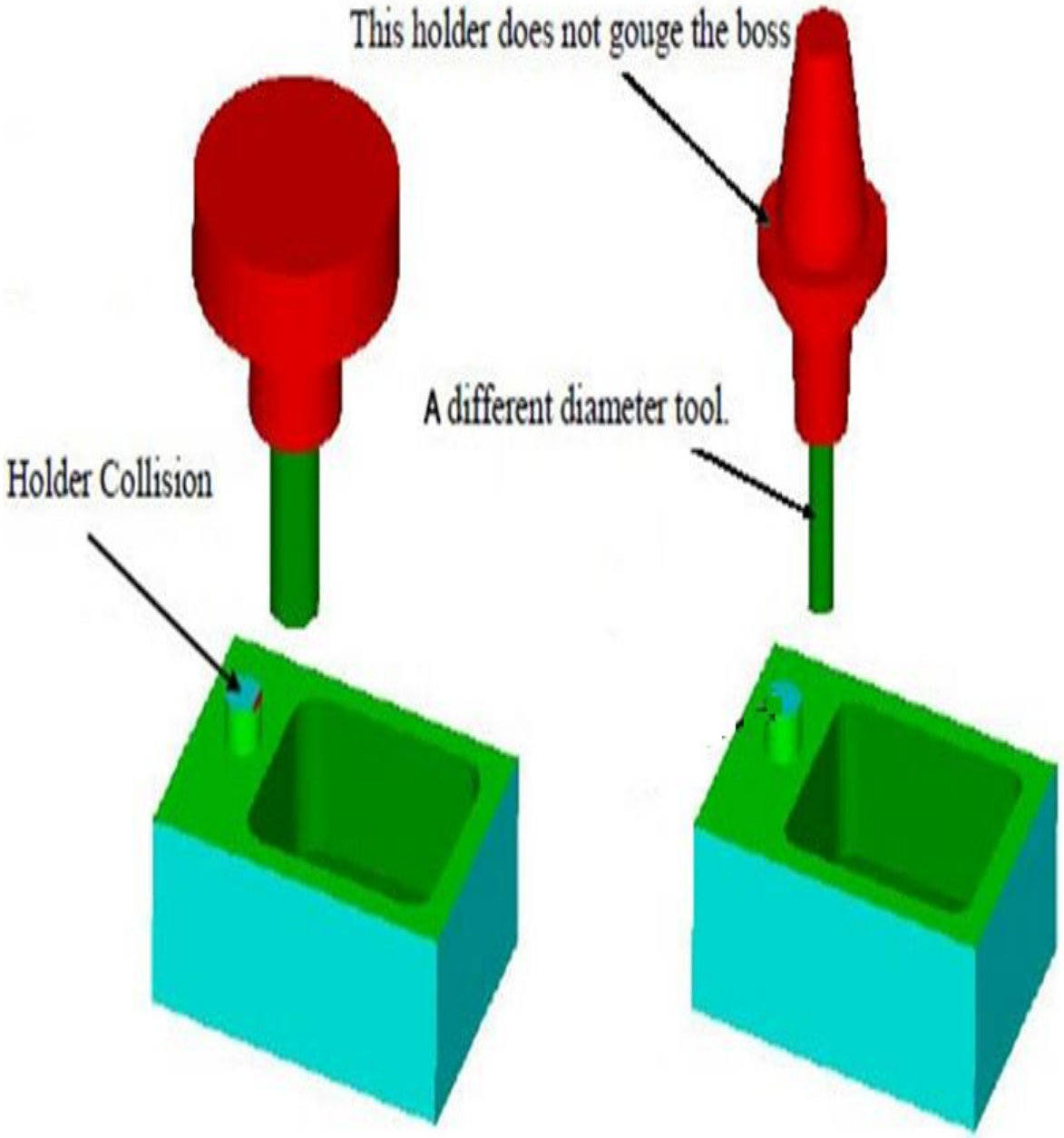
clamp or a fixture.

3-That prevents breakage of the tool and damage to the clamp or fixture.

4-Some programs may give hints to the solution of the error or hints to the cause of error and display an alarm message.

5- After simulating in some complicated cases manufacturing start machining by wooden ,wax , or foam as prototypes, for verifying the toolpath and examining it.

6- It can be used to fulfill the education requirements and saves unnecessary costs.



Solving the errors that appear by simulation

Simulation results:

1- Simulation strategy can make convenient design and manufacturing issues, and give easier way in complicated manufacturing problems.

2 -It reduces the time to teach and implement a new CNC machines.

3-Simulation prevents the danger when dealing with the CAM steps and CNC machine crashes and increases safety .

4- Simulation provides train programmers and machinists without using production time or risking a crash.

5-Simulation strategy improve process efficiency and reduce costs.

CHAPTRR.2

3.1.Planned analysis , pre-processing and post processing

Planned analyses

Planned analyses: are planned and specified study protocols before starting data collection. These achieved by inspecting the data.

Multiple statistical comparisons result in misleading findings .

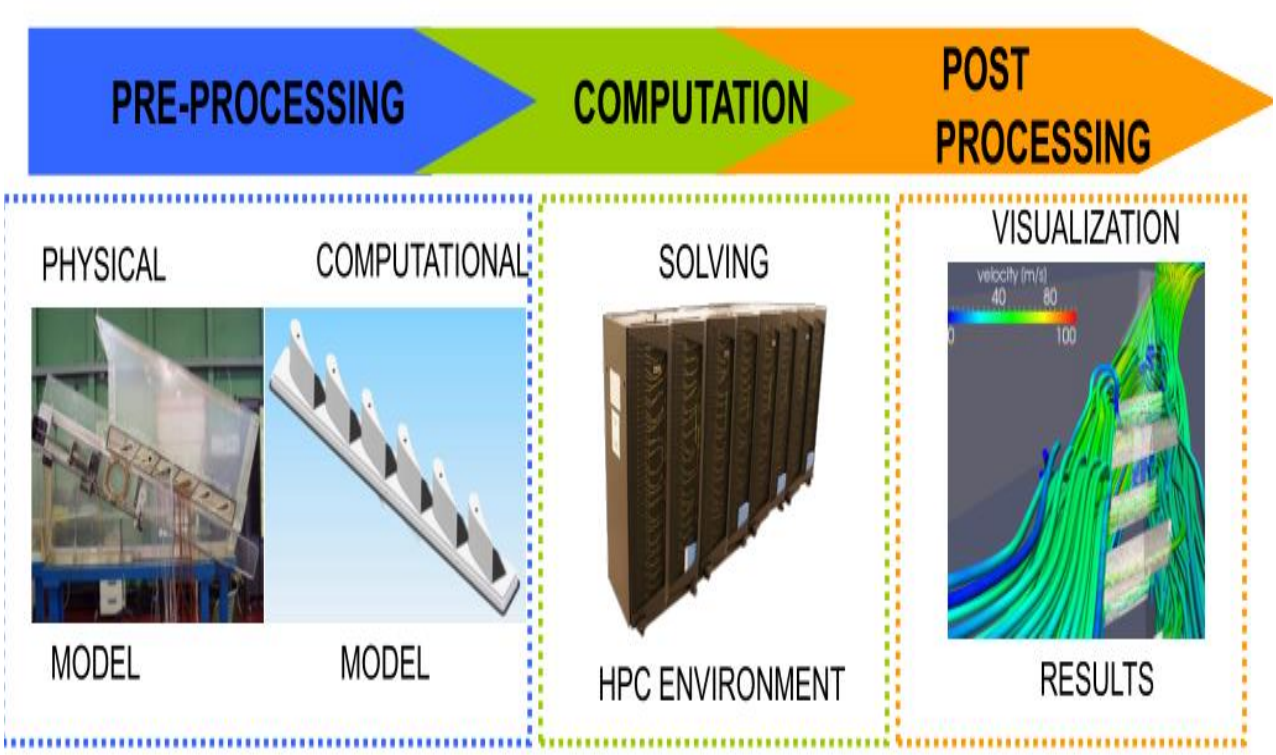
Computer Aided Engineering

1-CAE provides automation of requirements and performance analyses , it can provide automated processes for functional analyses such as fault isolation and testing, failure mode, and safety analyses.

2-Software tools that have been developed to support these activities are considered CAE tools.

3-CAE tools are being used, for example, to analyze the robustness and performance of components and assemblies.

4-The term encompasses simulation, validation, and optimization of products and manufacturing tools.In the future, CAE systems will be major providers of information to help support design teams in decision making



Computer Aided Engineering

“Pre- and Post- Processing” methodology where a cyclic process of experiments with Machine Learning(given a certain task, and a data set that constitutes the

task) tool, its data, data description language and parameters attempts to optimize learning performance.

In general, there are three phases in any computer-aided engineering task:

• Pre-processing .

• Computing .

• Post-processing of results .

This cycle is iterated, often many times, either manually or with the use of automation techniques or using optimization software.

Pre-processing

1-Definition:–

It is defined as the geometry model, the physical model and the boundary conditions

Usage of Pre-processing:

-Is used to enhance the reliability of the collected data.

-It includes data cleaning, for example, handling missing attributes…….

2. Pre-Processing Techniques

Pre-processing technique includes :

Data cleaning

Fill in missing values, identify or remove outliers, and resolve inconsistencies

Data integration

Integration of multiple databases, data cubes, or files

Data transformation

Normalization and aggregation

Data reduction

Obtains reduced representation in volume but produces the same or similar analytical results

Post-processing

• Post-processing of results (using scientific visualization tools &techniques)

post-processing components can be categorized into the following groups:

Post Processing Techniques

post processing includes following techniques:

a. Knowledge Filtering

In post processing phase, the main objective is to extract meaningful results to perform certain decision making step

b. Evaluation

It is essential to evaluate the performance of the system.

Classification accuracy, error,

computational complexity etc. are used to evaluate the classifier model

c. Information Visualizatione

. It is important to visualize the extracted knowledge in such form so that user can gain insight into data for better decision making

d. Knowledge Integration

New decision making system combines the results produced from different models which appear as an efficient approach

Example:

To transform the part ‘X’(raw material) into new part ‘Y’(product).

-This process consists of three following major steps(the fig.):

1-Pre-Processing of the part ‘X’: this step includes:

a-Cleaning and inspection operations of selected part ‘X’.

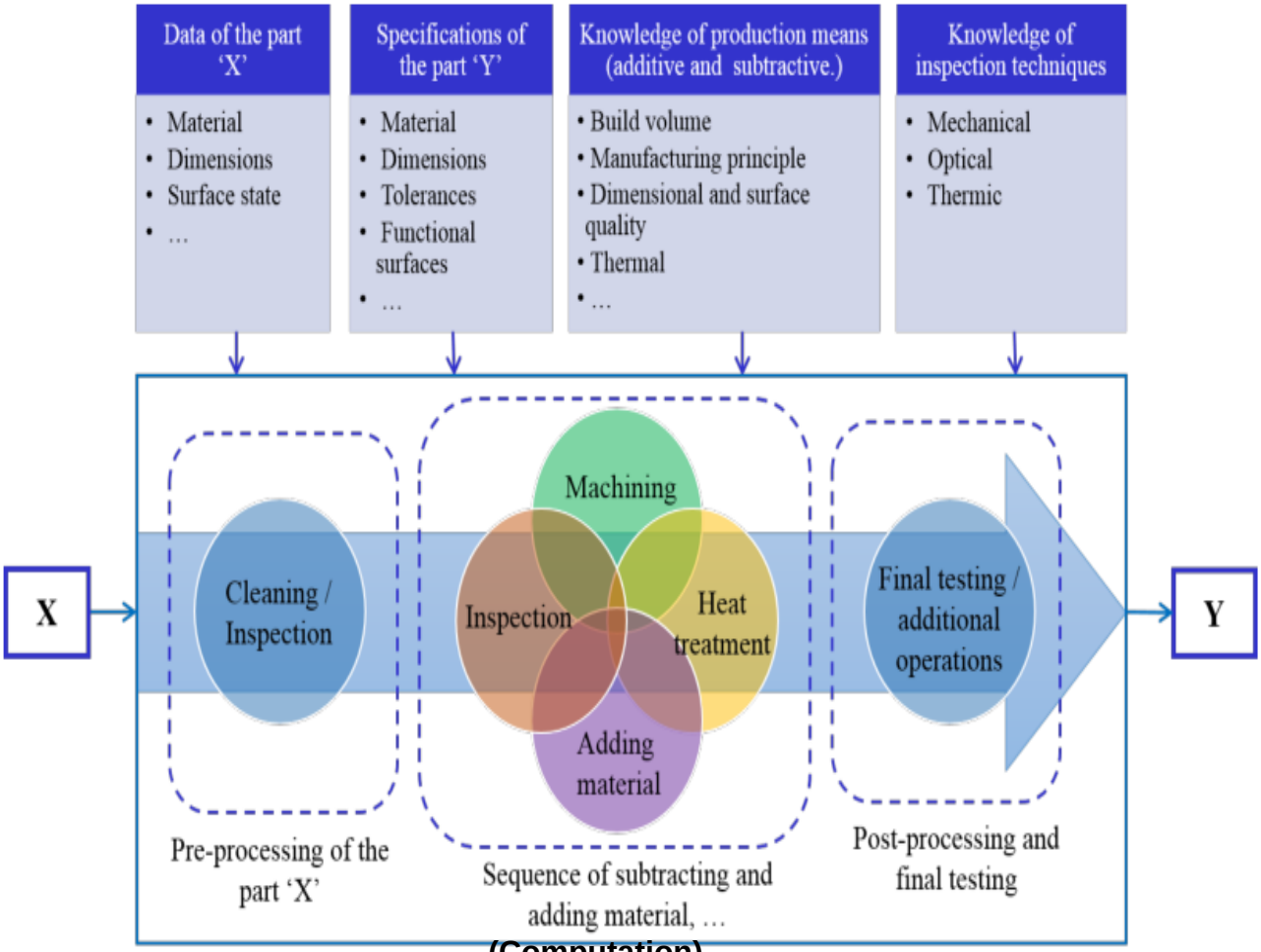
b-The raw material geometry will be identified in this steps by initial inspections.

2-Computation:

a- CAD model of the part ‘X’ can be generated.

b-This information is one of needed inputs to design the process (part “y”).

3- Post-processing and final testing: The final testing of the part and additional operations, such as labeling , shipping ,…, are performed during this step.



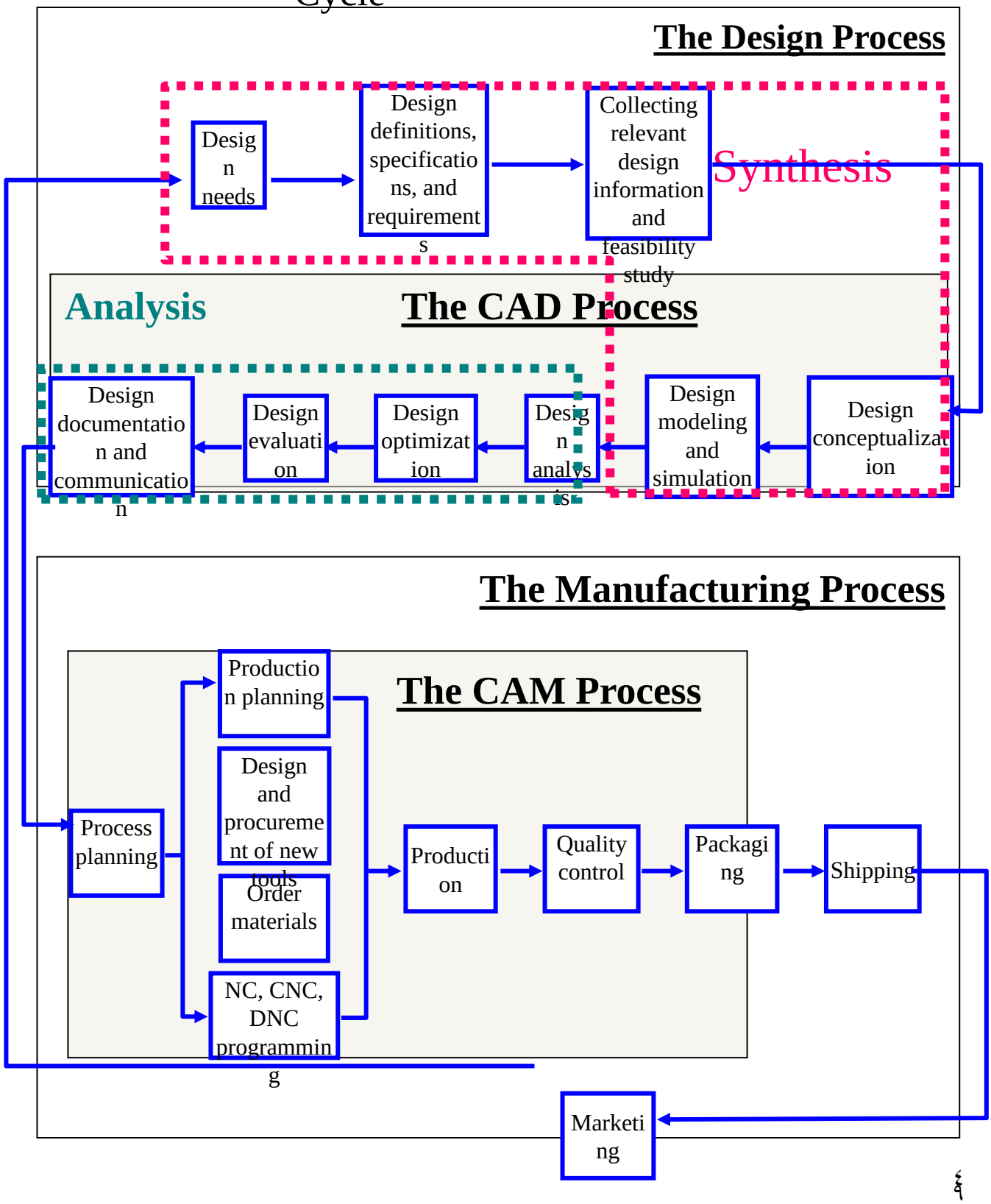
Generic process of the proposed strategy

2.2.The Product Cycle

The Product Cycle and CAD/CAM

In order to establish the scope and definition of CAD/CAM in an engineering environment and identify existing and future related tools, a study of a typical product cycle is necessary. The following Figure shows a flowchart of such a cycle.

Typical Product Life Cycle



The product begins with a need which is identified based on customers' and markets' demands.

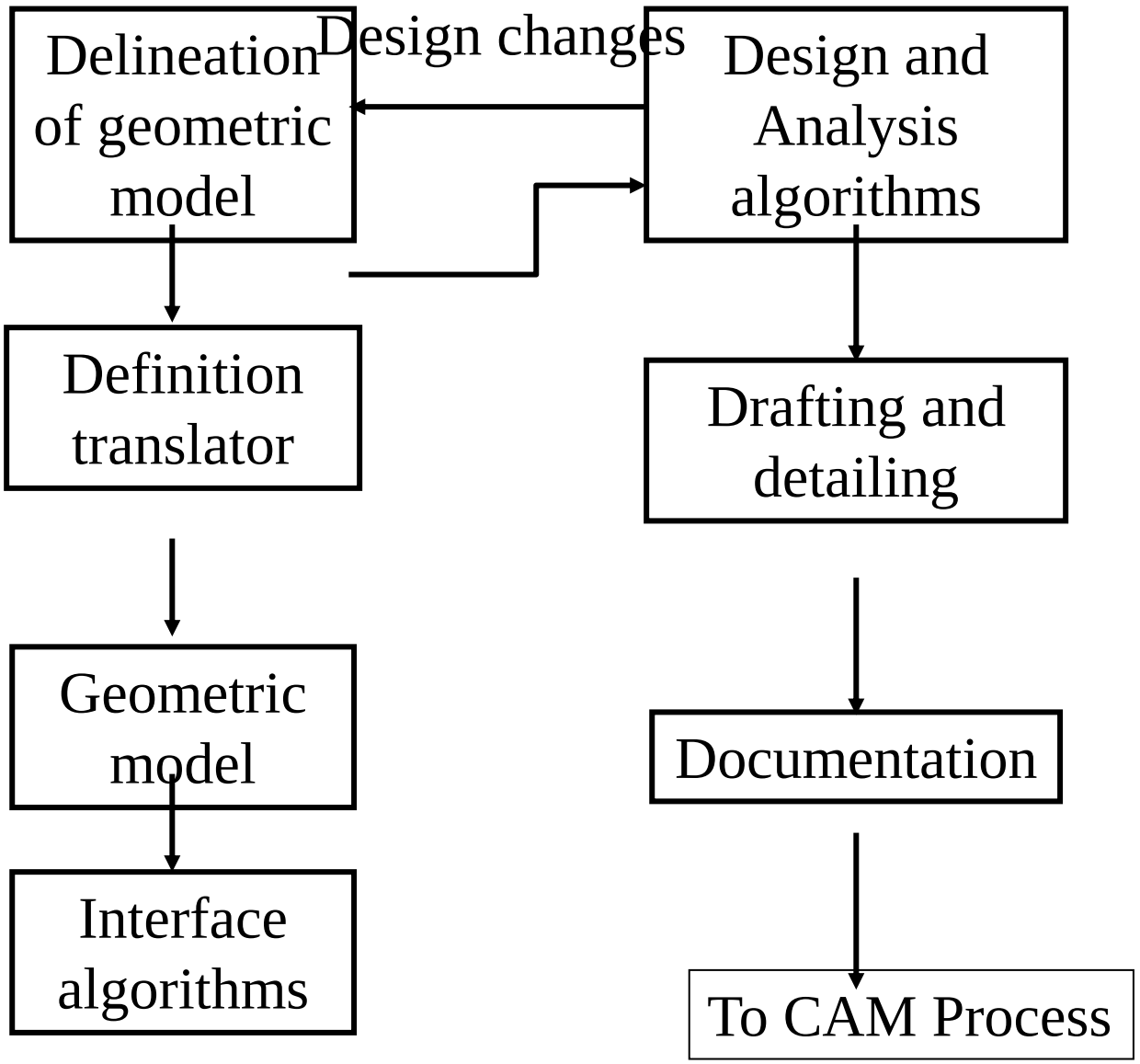
The product goes through two main processes from the idea conceptualization to the finished product:

1. **The design process.**
2. **The manufacturing process.**

The main sub-processes that constitute the design process are:

1. Synthesis.
2. Analysis.

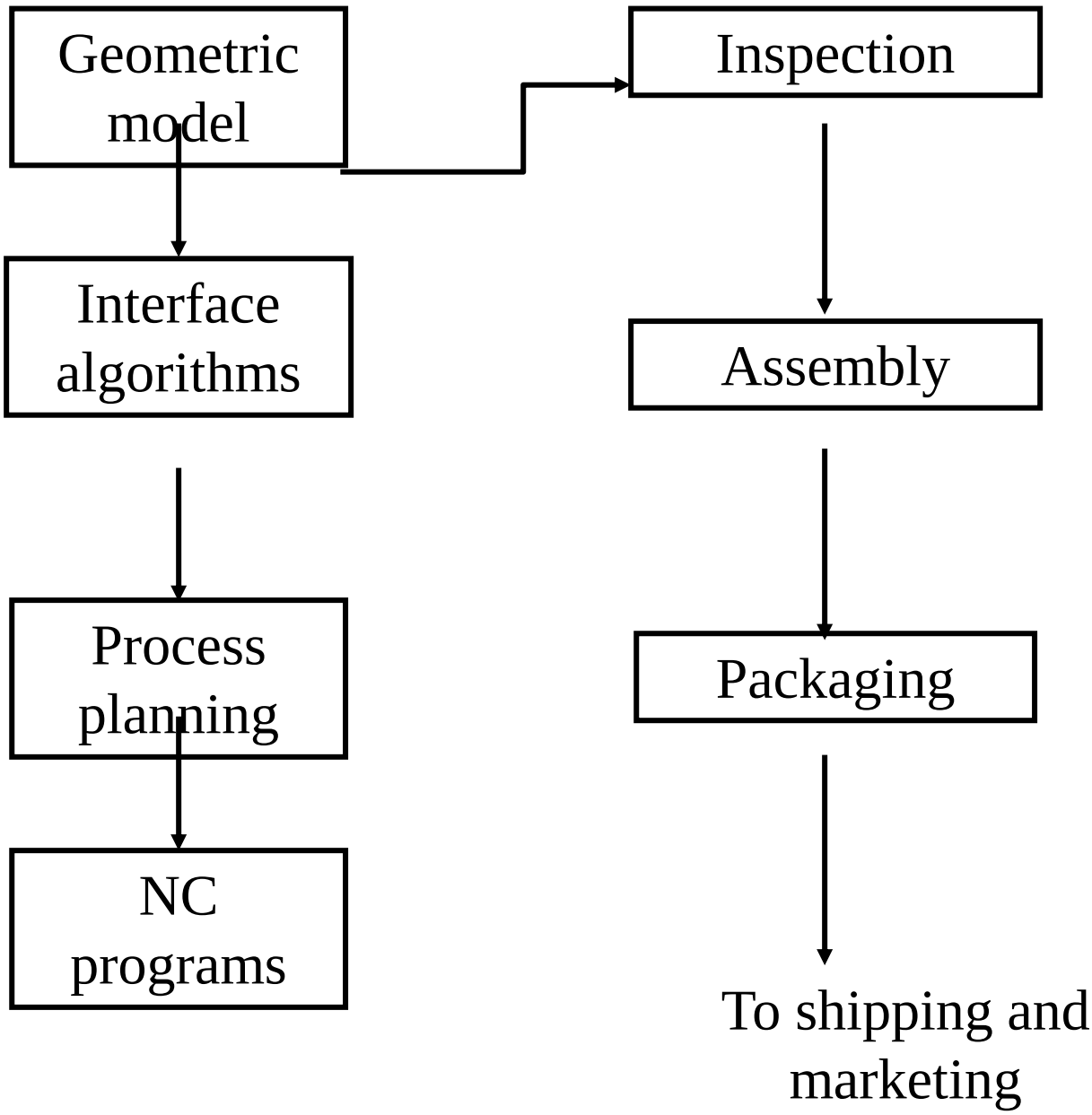
Implementation of a Typical CAD Process on a CAD/CAM system



CAD Tools Required to Support the Design Process

Design phase	Required CAD tools
Design conceptualization	**Geometric modeling techniques; Graphics aids; manipulations; and visualization**
Design modeling and simulation	**Same as above; animation; assemblies; special modeling packages.**
Design analysis	**Analysis packages; customized programs and packages.**
Design optimization	**Customized applications; structural optimization.**
Design evaluation	**Dimensioning; tolerances; BOM; NC.**
Design communication and documentation	**Drafting and detailing…**

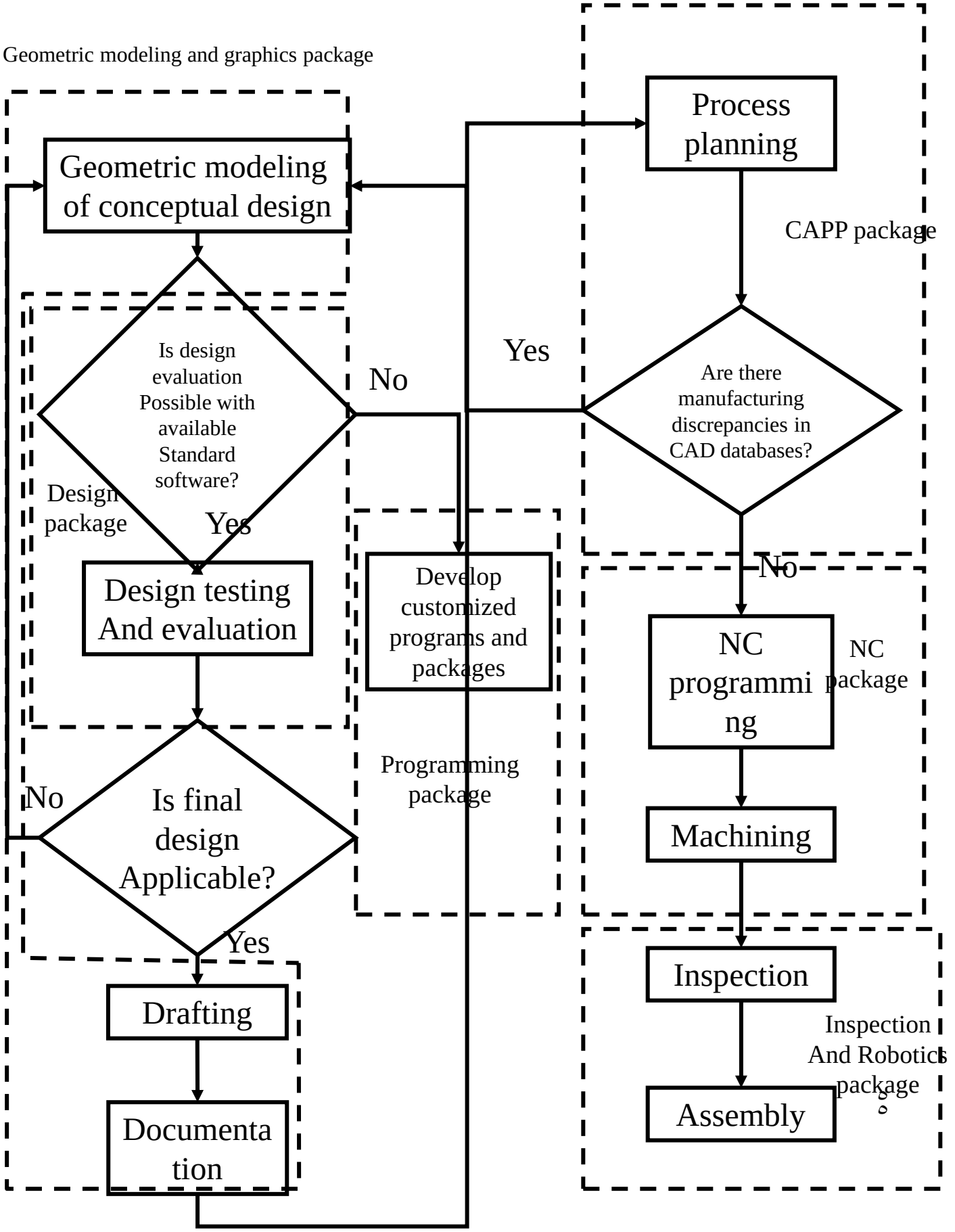
Implementation of a Typical CAM Process on a CAD/CAM system



CAM Tools Required to Support the **Manufacturing** Process

Manufacturing phase	**Required CAM tools**
Process planning	CAPP techniques; cost analysis; material and tooling specification.
Part programming	NC programming
Inspection	CAQ; and Inspection software
Assembly	Robotics simulation and programming

Typical Utilization of CAD/CAM Systems in an Industrial Environment



Automation and CAD/CAM

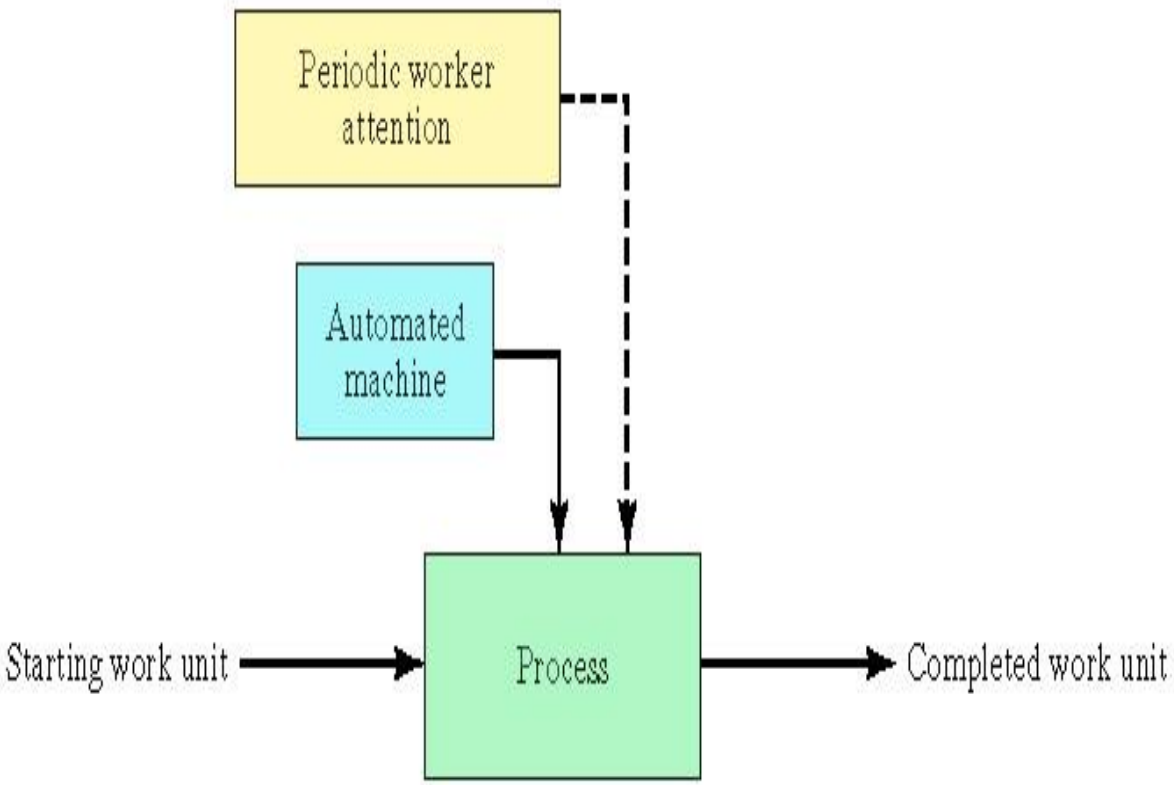
Automation can be defined as the technology concerned with the application of complex mechanical, electronic, and computer-based systems in the operation and control of manufacturing systems.

Types of Manufacturing Systems

1. **Continuous-flow processes**. Continuous dedicated production of large amount of bulk product. Continuous manufacturing is represented by chemicals, plastics, petroleum, and food industries.
2. **Mass production of discrete products**. Dedicated production of large quantities of one product (with perhaps limited model variations). Examples include automobiles, appliances and engine blocks.
3. **Batch production.** Production of medium lot sizes of the same product. The lot may be produced once or repeated periodically. Examples: books, clothing and certain industrial machinery.
4. **Job-shop production**. Production of low quantities, often one of a kind, of specialized products. The products are often customized and technologically complex. Examples: prototypes, aircraft, machine tools and other equipment.

Automated System

Fig. 1.2. (c)



Automation in Production Systems

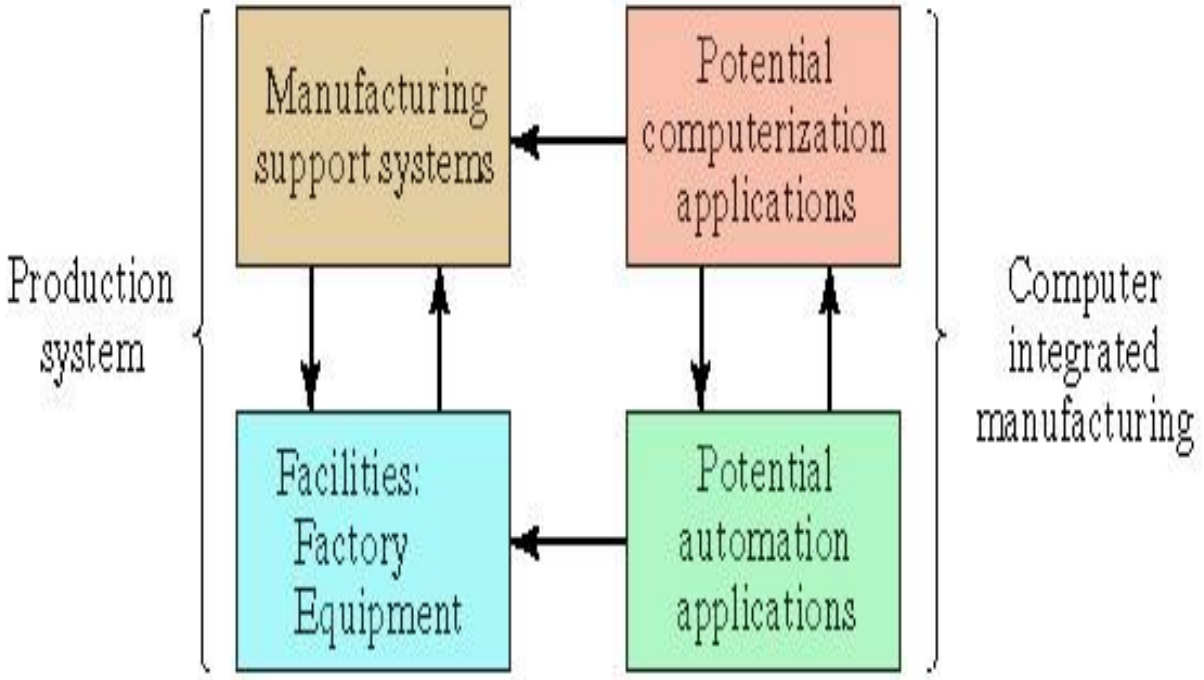
Two categories of automation in the production system:

1. Automation of manufacturing systems in the factory
2. Computerization of the manufacturing support systems

The two categories overlap because manufacturing support systems are connected to the factory manufacturing systems

Computer-Integrated Manufacturing (CIM)

Computer Integrated Manufacturing



Automated Manufacturing Systems

Examples:

Automated machine tools

Transfer lines

Automated assembly systems

Industrial robots that perform processing or assembly operations

Automated material handling and storage systems to integrate manufacturing operations

Automatic inspection systems for quality control

Three types:

1. Fixed automation
2. Programmable automation
3. Flexible automation

Fixed Automation

A manufacturing system in which the sequence of processing (or assembly) operations is fixed by the equipment configuration

Typical features:

Suited to high production quantities

High initial investment for custom-engineered equipment

High production rates

Relatively inflexible in accommodating product variety

Programmable Automation

A manufacturing system designed with the capability to change the sequence of operations to accommodate different product configurations

Typical features:

High investment in general purpose equipment

Lower production rates than fixed automation

Flexibility to deal with variations and changes in product configuration

Most suitable for batch production

Physical setup and part program must be changed between jobs (batches)

Flexible Automation

An extension of programmable automation in which the system is capable of changing over from one job to the next with no lost time between jobs

Typical features:

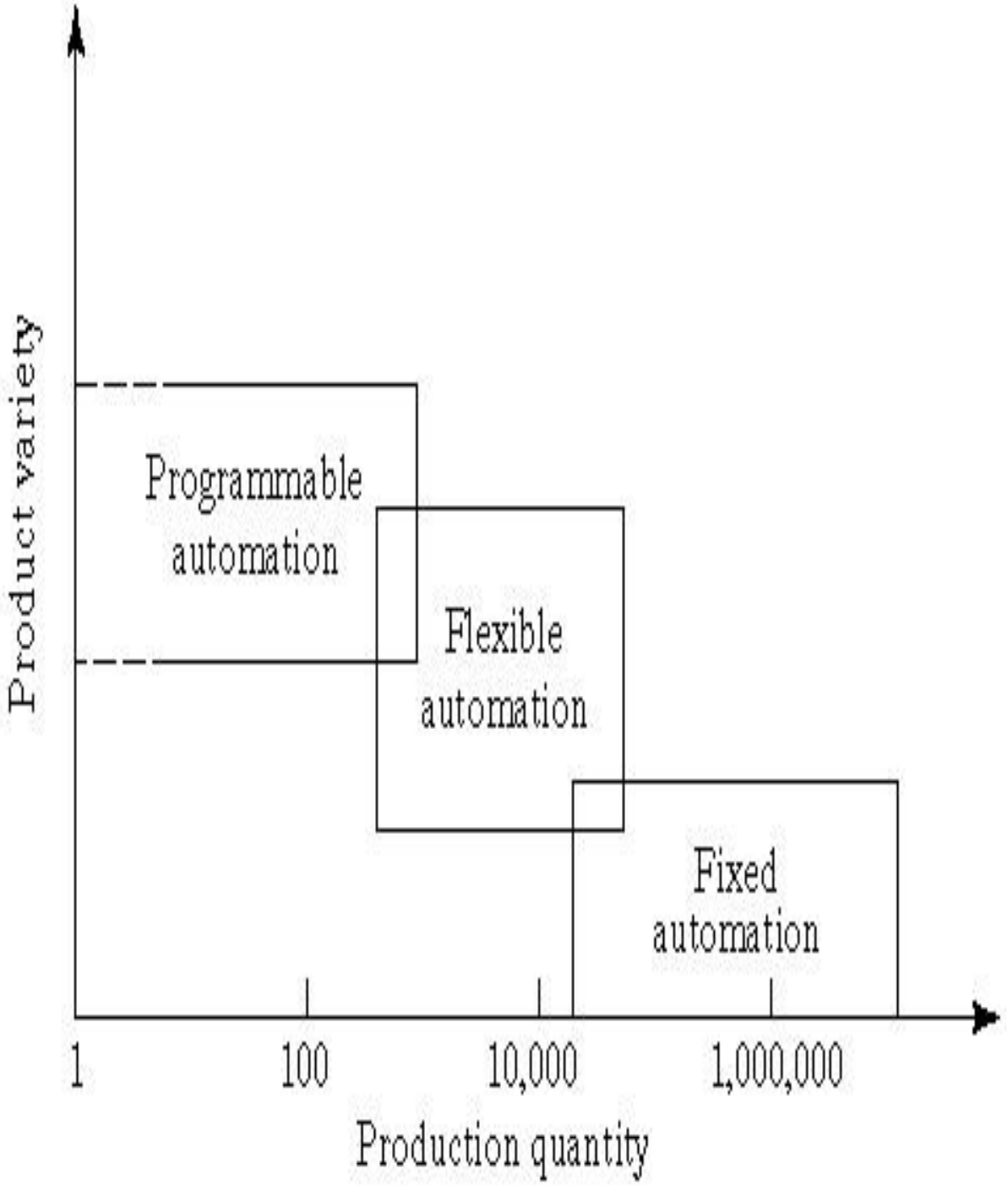
High investment for custom-engineered system

Continuous production of variable mixes of products

Medium production rates

Flexibility to deal with soft product variety

Product Variety and Production Quantity for Three Automation Types



Computerized Manufacturing Support Systems

Objectives of automating the manufacturing support systems:

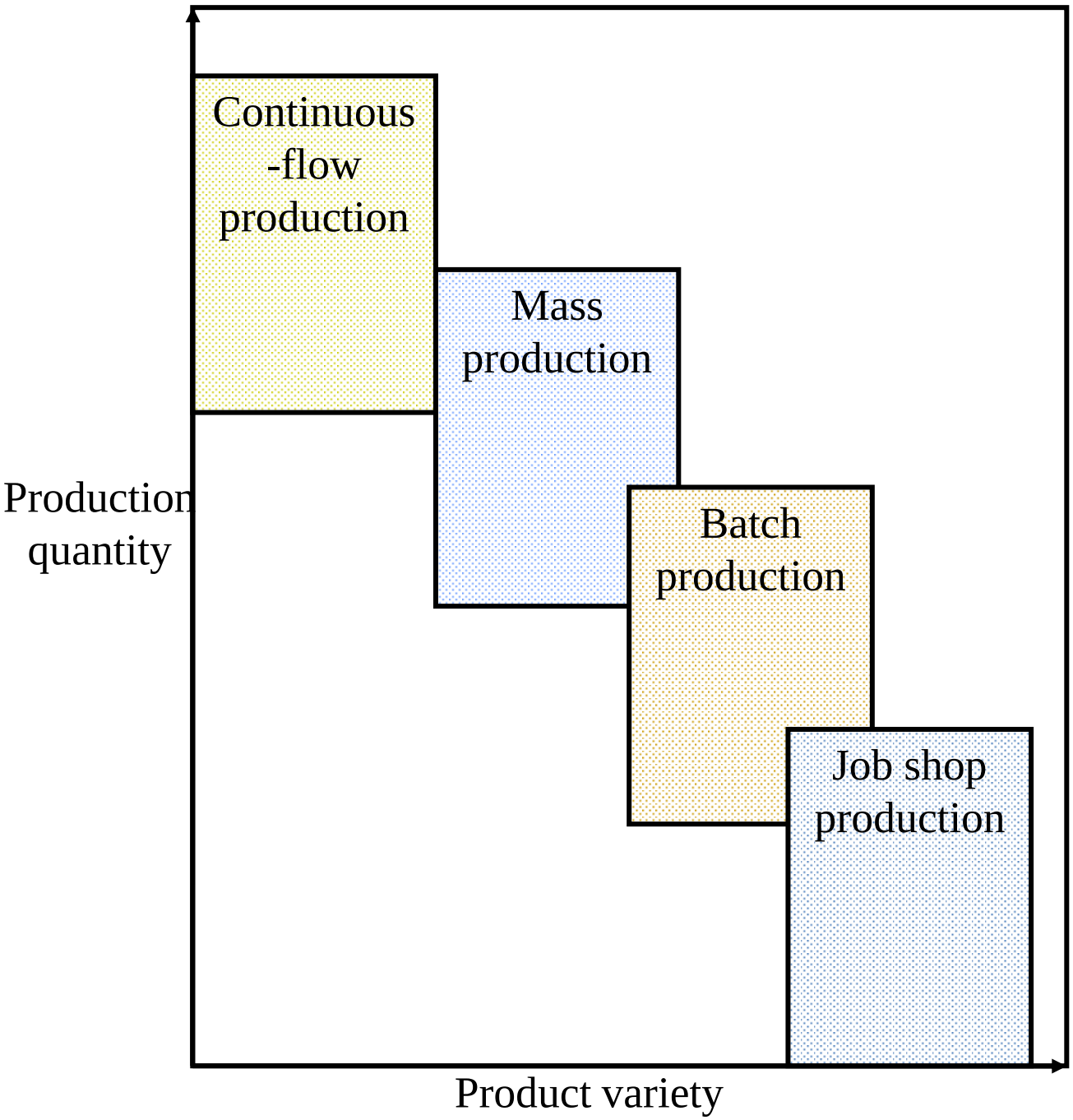
To reduce the amount of manual and clerical effort in product design, manufacturing planning and control, and the business functions

Integrates computer-aided design (CAD) and computer-aided manufacturing (CAM) in CAD/CAM

CIM includes CAD/CAM and the business functions of the firm

Reasons for Automating:

1. To increase labor productivity
1. To reduce labor cost
2. To mitigate the effects of labor shortages
3. To reduce or remove routine manual and clerical tasks
4. To improve worker safety
5. To improve product quality
6. To reduce manufacturing lead time
7. To accomplish what cannot be done manually
8. To avoid the high cost of not automating


Continuous -flow production
Mass production
Batch production
Job shop production
Production quantity
Product variety

Category	Automation achievements
Continuous -flow process	•Flow process from beginning to end •Sensors technology available to measure important process variables •Use of sophisticated control and optimization strategies •Fully computer automated lines
Mass production of discrete products	•Automated transfer machines •Dial indexing machines •Partially and fully automated assembly lines •Industrial robots for spot welding, part handling, machine loading, spray painting, etc. •Automated material handling systems •Computer production monitoring
Batch production	•Numerical control (NC), direct numerical control (DNC), computer numerical control (CNC). •Adaptive control machining •Robots for arc welding, parts handling, etc. •CIM systems.
Job shop production	•Numerical control, computer numerical control

Computer Technology in Automation

Most of the automated production systems implemented today make use of computers. CAD/CAM in addition to its particular emphasis on the use of computer technology, is also distinguished by the fact that it includes not only the manufacturing operations but also the design and planning functions that precede manufacturing.

To emphasize the differences in scope between automation and CAD/CAM, consider the following mathematical model

$$TT_{lc} = BQT_1 + BT_2 + T_3$$

TT_{cl} = total time during the product life cycle

B = The number of batches produced throughout the product life cycle.

Q = The number of units produced in each batch.

T_1 = The time required to produce one unit of product.

T_2 = The time associated with planning and setting up for each batch of production.

T_3 = The time required for designing the product and for all the other activities that are accomplished once for each different product.

T_{lc} = The average time spent on each unit of product during its life cycle $= T_1 + \frac{T_2}{Q} + \frac{T_3}{BQ}$

T_1 The most important term in mass production and batch production

T_2 & T_3 become very important in job shop manufacturing.

Automation technology is concerned with reducing

T_1 & T_2 the with emphasis on the unit production time

T_1

CAD/CAM concerned with reducing all three terms, but is perhaps focused on terms. The emphasis in T_2 & T_3 CAD/CAM includes the design and planning function of the product life cycle.

Advantages of CAD/CAM systems

Greater flexibility.

Reduced lead times.

Reduced inventories.

Increased Productivity.

Improved customer service.

Improved quality.

Improved communications with suppliers.

- Better product design.
- Greater manufacturing control.
- Supported integration.
- Reduced costs.
- Increased utilization.
- Reduction of machine tools.
- Less floor space.

CHAPTRER 3..
NC, CNC, Adaptive control, industrial robotics,

Industrial Robot Definition

A general-purpose, programmable machine possessing certain anthropomorphic characteristics

Area of Industrial Robot uses

Hazardous work environments

Repetitive work cycle

Consistency and accuracy

Difficult handling task for humans

Multishift operations

Reprogrammable, flexible

Interfaced to other computer systems

Robot Anatomy

Manipulator(it is often used to describe the robot itself, minus the power supply and controller) consists of joints and links

Joints provide relative motion

Links are rigid members between joints

Various joint types: linear and rotary

Each joint provides a “degree-of-freedom”

Most robots possess five or six degrees-of-freedom



Robot manipulator consists of two main sections:

Body-and-arm – for positioning of objects

in the robot's work volume

Wrist assembly – for orientation of objects

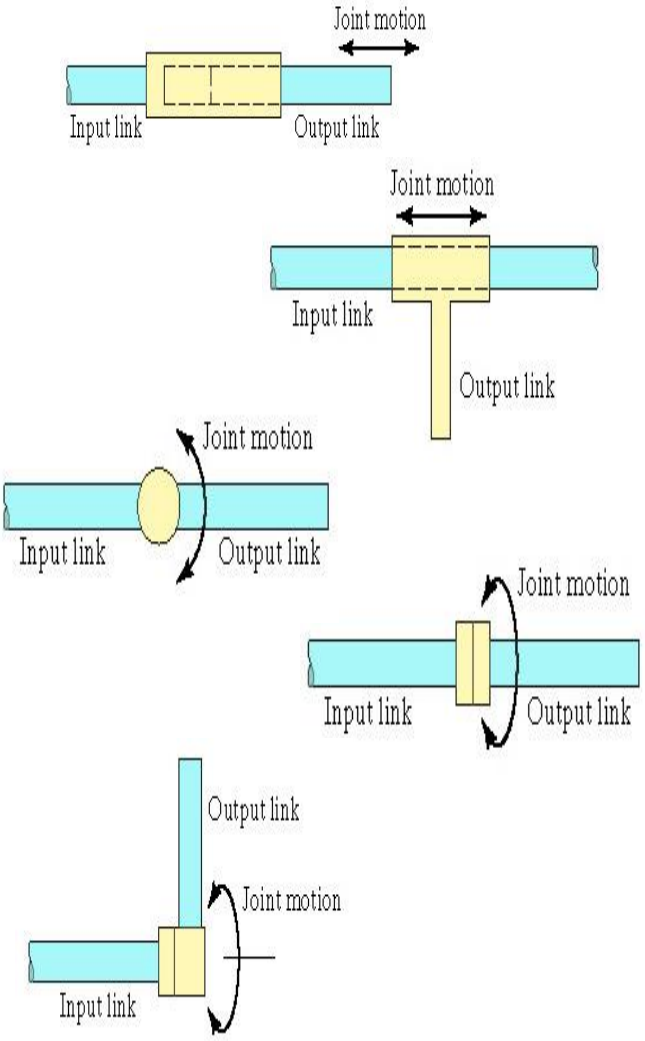
Manipulator Joints

Translational motion

- Linear joint (type L)
- Orthogonal joint (type O)

Rotary motion

- Rotational joint (type R)
- Twisting joint (type T)
- Revolving joint (type V)



Joint Notation Scheme

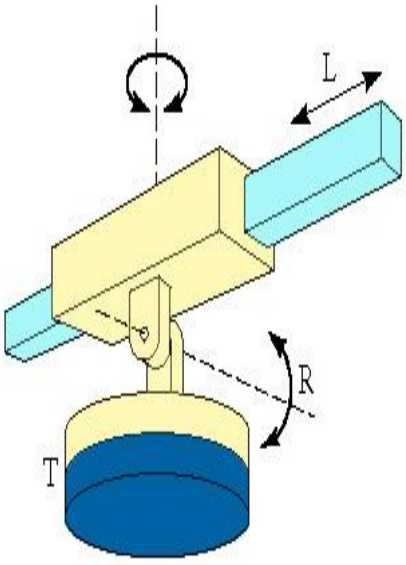
- Uses the joint symbols (L, O, R, T, V) to designate joint types used to construct robot manipulator
- Separates body-and-arm assembly from wrist assembly using a colon (:)
- Example: TLR : TR

Polar Coordinate Body-and-Arm Assembly

Notation TRL:

Consists of a sliding arm (L joint) actuated relative to the body, which can rotate about both a vertical axis (T joint) and horizontal axis (R joint)





Cylindrical Body-and-Arm Assembly

- Notation TLO:
- Consists of a vertical column, relative to

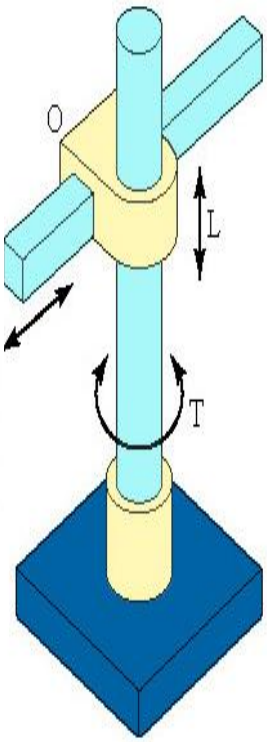
which an arm assembly is moved up or

down

- The arm can be moved in or out relative

to the column



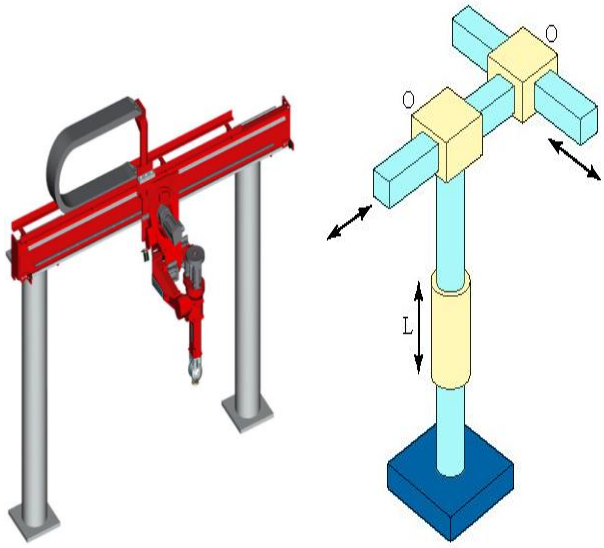


Cartesian Coordinate Body-and-Arm Assembly

Notation LOO:

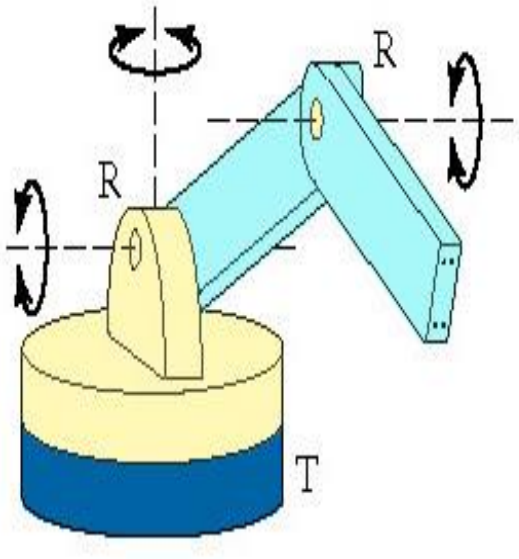
Consists of three sliding joints, two of which are orthogonal

Other names include rectilinear robot and x-y-z robot



Jointed-Arm Robot

- Notation TRR:



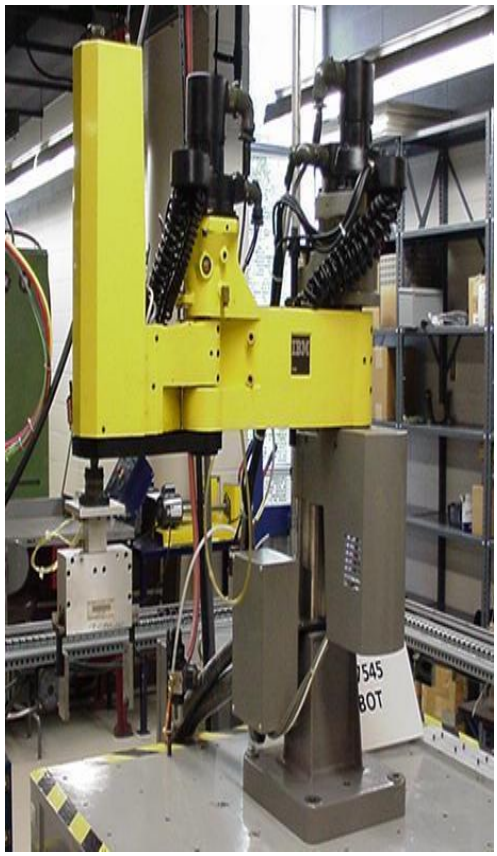


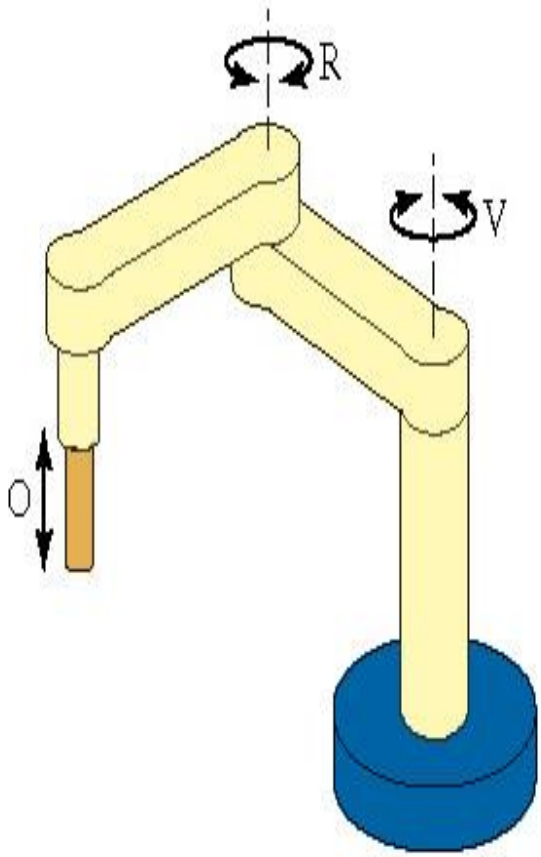
SCARA
Robot

Notation VRO

SCARA stands for Selectively Compliant Assembly Robot Arm

Similar to jointed-arm robot except that vertical axes are used for shoulder and elbow joints to be compliant in horizontal direction for vertical insertion tasks





Wrist Configurations

Wrist assembly is attached to end-of-arm

End effector is attached to wrist assembly

Function of wrist assembly is to orient end effector

Body-and-arm determines global position of end effector

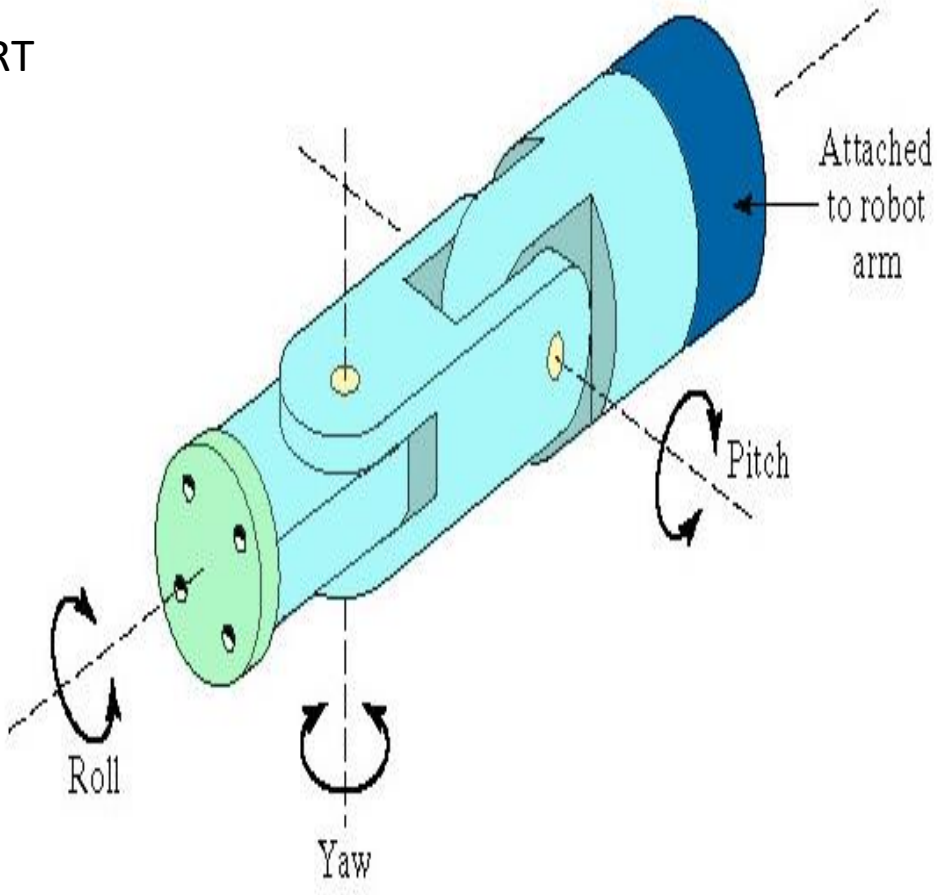
Two or three degrees of freedom:

Roll

Pitch

Yaw

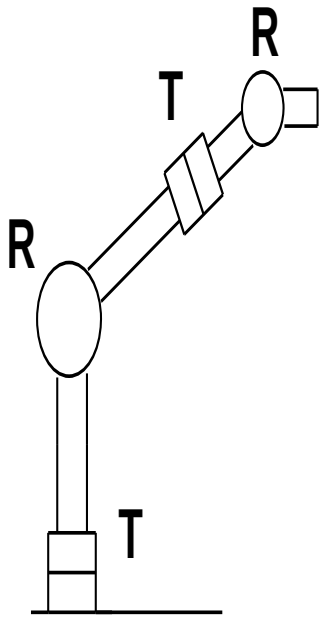
Notation :RRT



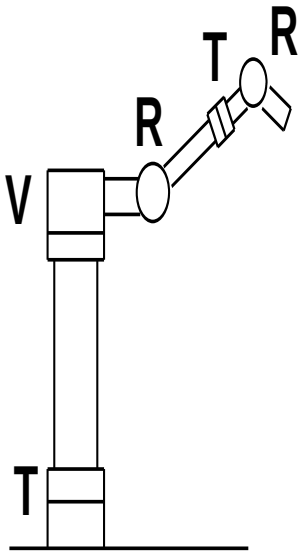
Example

Sketch following manipulator configurations
(a) TRT:R, (b) TVR:TR, (c) RR:T.

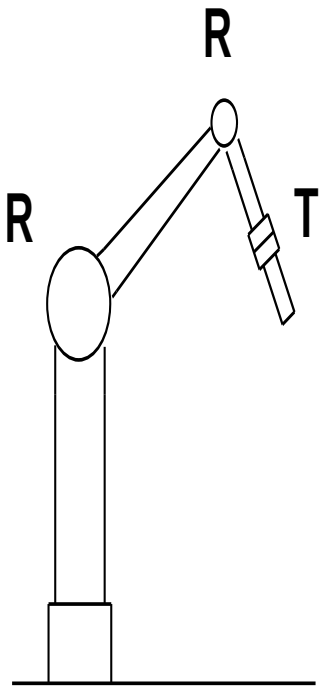
Solution:



(a) TRT:R



(b) TVR:TR



(c) RR:T

Joint Drive Systems



Electric

- Uses electric motors to actuate individual joints
- Preferred drive system in today's robots

Hydraulic



- Uses hydraulic pistons and rotary vane actuators
- Noted for their high power and lift capacity

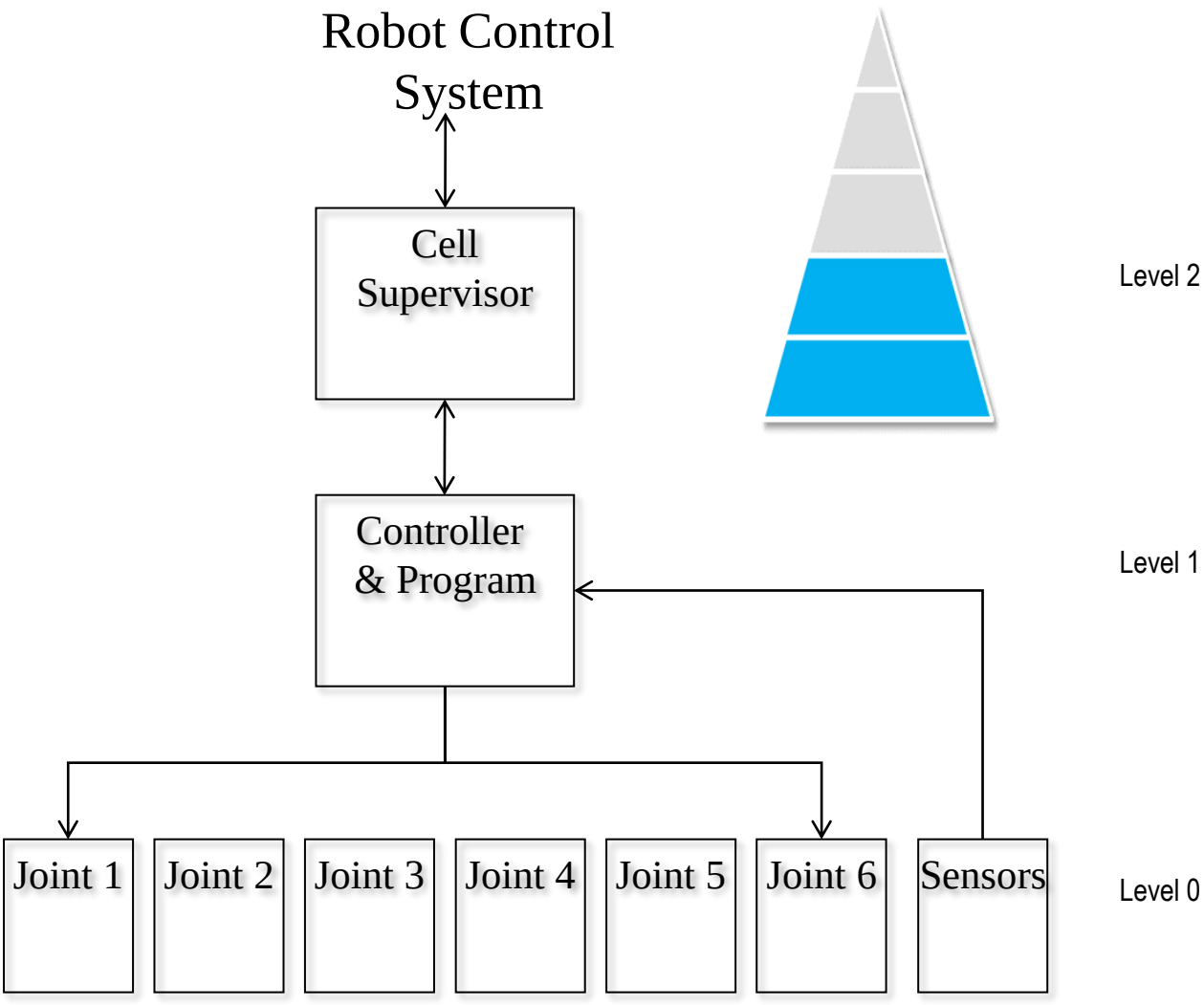
Pneumatic

- Typically limited to smaller robots and simple material transfer applications



Robot Control Systems

- **Limited sequence control** – pick-and-place operations using mechanical stops to set positions
- **Playback with point-to-point control** – records work cycle as a sequence of points, then plays back the sequence during program execution
- **Playback with continuous path control** – greater memory capacity and/or interpolation capability to execute paths (in addition to points)
- **Intelligent control** – exhibits behavior that makes it seem intelligent, e.g., responds to sensor inputs, makes decisions, communicates with humans



End Effectors



The special tooling for a robot that enables it to perform a specific task

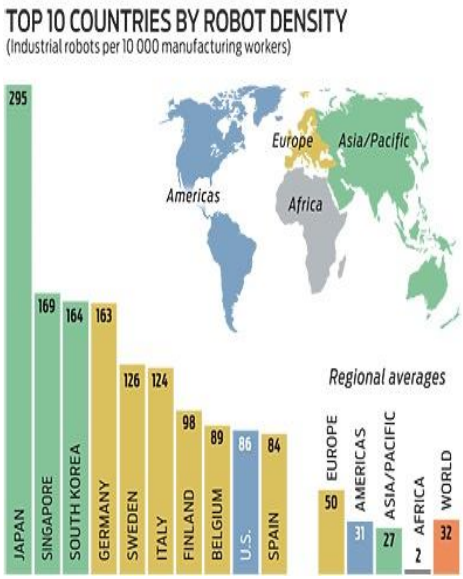
Two types:

Grippers – to grasp and manipulate objects (e.g., parts) during work cycle

Tools – to perform a process, e.g., spot welding, spray painting

Industrial Robot Applications

1. Material handling applications
   - Material transfer – pick-and-place, palletizing
   - Machine loading and/or unloading
2. Processing operations
   - Welding
   - Spray coating
   - Cutting and grinding
3. Assembly and insp ection



Robot Programming

Leadthrough programming

Work cycle is taught to robot by moving the manipulator through the required motion cycle and simultaneously entering the program into controller memory for later playback

Robot programming languages

Textual programming language to enter commands into robot controller

Simulation and off-line programming

Program is prepared at a remote computer terminal and downloaded to robot controller for execution without need for leadthrough methods

Leadthrough Programming

1. Powered leadthrough
   - Common for point-to-point robots
   - Uses teach pendant
2. Manual leadthrough
   - Convenient for continuous path control robots
   - Human programmer physical moves manipulator

Leadthrough Programming
Advantages

Advantages:

- Easily learned by shop personnel
- Logical way to teach a robot
- No computer programming

Disadvantages:

- Downtime during programming
- Limited programming logic capability
- Not compatible with supervisory control



Group technology

Group technology is a manufacturing technique and philosophy to increase production efficiency by exploiting the “underlying sameness” of component :shape, dimensions, process route, etc.

Group technology benefits

- Average lot size decreasing
- Part variety increasing
- Increased variety of materials with diverse properties
- Requirements for closer tolerances

•Part family :is the group of similar parts and

• Machine cell: is the group of machineries used to process an individual part family.



It is not necessary for each part of a part family to be processed by every machine of corresponding machine cell.

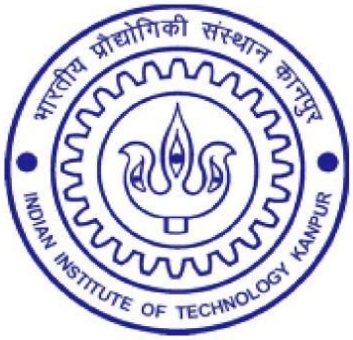
3.2.Cellular manufacturing :

This type of manufacturing in which a part family is produced by a machine cell.

The manufacturing efficiencies are generally increased by employing GT because the required operations may be confined to only a small cell and thus avoiding the need for transportation of in-process parts.

LABORATORY MANUAL

MANUFACTURING PROCESSES – 1



TA 202 LAB

Department of Mechanical Engineering INDIAN INSTITUTE OF TECHNOLOGY KANPUR

GENERAL INSTRUCTIONS

1. **Every student should obtain a set of instruction sheets entitled manufacturing processes Laboratory.**
2. **For reasons of safety, every student must come to the laboratory in shoes. It is unsafe for the students to come to the laboratory wearing garments with parts that that hang about loosely. Students should preferably use half-sleeve shirts. The Students should also ensure that floor around the machine is clear and dry (not oily) to avoid slipping.**
3. **An apron will be issued to each student. Students not wearing an apron will not be permitted to the work in the laboratory.**
4. **Instruments and tools will be issued from the tool room. Every student must produce his identity card for the purpose. Tools, etc. must be retuned to the tool room on the same day.**
5. **The student should take the permission of the Lab Staff / Tutor before handling any machine.**
6. **The student should not lean on the machine when it is working.**
7. **Power to the machines will be put off 10 minutes before the end of laboratory session to allow the students to return the tools.**
8. **Students are required to clear off the chips from the machine and lubricate the guides etc. at the end of the session.**
9. **Laboratory reports should be submitted on A4 size sheets.**
10. **Reports will not be returned to the students. Students may see the graded reports in the laboratory**

GENERAL INFORMATION

This laboratory is aimed at providing an introduction to the Know-how of common processes used in industries for manufacturing parts by removal of material in a controlled manner. Auxiliary methods for machining to desired accuracy and quality will also be covered. The emphasis throughout the laboratory course will be on understanding the basic features of the processes rather than details of constructions of machine, or common practices in manufacturing or acquiring skill in the operation of machines. Evidently, acquaintance with the machine is desirable and the laboratory sessions will provide adequate opportunity for this.

LABORATORY EXERCISE

I	Turning (T) & NC Demonstration	6 hr.
II	Milling	3 hr.
III	Drilling and Fitting (D&F)	3 hr.
IV	CNC Exercise	3 hr.

PROJECT

The Project consists of conceptualization a device designing, planning of machining and other operations, fabrication of the components and assembly of the device. Before the final evaluation of the completed project a report has to be submitted. This report should contain general description of the completed project, design details, detailed drawings, and fabrication of the components, assembly I and testing, suggestions for improvements.

I. First turn	VI Project groups should be formed and names to be given to tutor.
II. Second turn	Project idea discussion (minimum 3 ideas) and freezing
III Third	Isometric view submission (as per engineering standard)
turn IV	Part drawing submission (as per engineering standard) Lab
Fourth	Examination + CNC Exercise
V Fifth	

 # Number of components in the project should not be more than 25 (Excluding standard parts)

LABORATORY EXERCISES SCHEDULE.

EXERCISE POWER TRANSMITION Video Clip (10 MIN,)			
CLASS	LATHE	MILLING	DRILLING & FITTING
I	A+B	C	D
2	CNC DEMO. (60MIN.) A+B	D	C
3	C+D	A	B
4	CNC DEMO. (60 MIN.) C+D	B	A
5	CNC EXERCISE & Lab Exam		

NOTES: Each Section will be divided in to four groups (A, B, C & D).

6th to 12th Projects.

MACHINING PROCESSES

Machining is one of the processes of manufacturing in which the specified shape to the work piece is imparted by removing surplus material. Conventionally this surplus material from the workpiece is removed in the form of chips by interacting the workpiece with an appropriate tool. This mechanical generation of chips can be carried out by single point or multi point tools or by abrasive operations these are summarized below:

Machining Processes

Single point tool operations	Multi-point tool operations	Abrasive operations
1. Turning	1. Milling	1. Grinding
2. Boring	2. Drilling	2. Lapping
3. Shaping	3. Tapping	3. Honing
4. Planing	4. Reaming	4. Super-finishing
	5. Hobbing	
	6. Broaching	
	7. Sawing	

The process of chip formation in metal cutting is affected by relative motion between the tool and the workpiece achieved with the aid of a device called machine tool. This relative motion can be obtained by a combination of rotary and translatory movements of either the tool or the workpiece or both. The kind of surface that is produced by the operation depends on the shape of the tool and the path it traverses through the materials. When the workpiece is rotated about an axis and the tool is traversed in a definite path relative to the axis, a surface of revolution is generated. When the tool path is parallel to the axis, the surface generated is a cylinder as in straight turning (Fig.1.) or boring (Fig.2.) operations. Similarly, planes may be generated by a series of straight cuts without rotating the workpiece as in shaping and planning operations (Fig.3). In shaping the tool is reciprocating and the work piece is moved crosswise at the end of each stroke. Planning is done by reciprocating the workpiece and crosswise movement is provided to the tool.

Surface may be machined by the tools having a number of cutting edges that can cut successively through the workpiece materials. In plane milling, the cutter revolves and moves over the work piece as shown (Fig.4). The axis of the cutter is parallel to the surface generated. Similarly in drilling, the drill may turn and be fed into the workpiece of the workpiece may revolve while the drill is fed into it (Fig.5).

The machine tools, in general, provide two kinds of relative motions. The primary motion is responsible for the cutting action and absorbs most of the power required to perform the machining action. The secondary motion of the feed motion may proceed in steps or continuously and absorbs only a fraction of the total power required for machining. When the secondary motion is added to the primary motion, machine surfaces of desired geometric characteristics are produced. .

Consider a situation where both the cutting motions as well as the feed motion (provided at the end of each stroke) are rectilinear but perpendicular to each other. Here the machined surface produced is a plane. The line generated by the primary motion (cutting motion) is called the generatrix, while the line representing the secondary motion (feed motion) is called the directrix (Fig. 6a).

Depending upon the shapes of the generatrix and the directrix and their relative orientation various geometries can be produced on the workpiece. Consider another case when the generatrix is a circle and the directrix is a line perpendicular to the plane of the generatrix. It is clear that in this situation the surface produced will be a cylinder (Fig. 6b). A tapered surface can be produced by merely changing the angle that the directrix makes with the plane of the generatrix. When the directrix is in the plane of the circular generatrix (Fig. 6c), lines are generated which results in a plain surface when a number of generatrices and directrices are placed side by side in the direction perpendicular to the plane of the generatrix. In actual practice, the cutting is performed by cutting edge and not a point. Thus, a series of generatrix directrix combinations are involved and the relative motion produces a surface rather than a line.

Basically there are two methods of producing new surfaces, the tracing method and the generation method. In the tracing method the surface is obtained by direct tracing of the generatrices and when the surface produced is the envelope of the generatrics the process is known as generation. Figs. 6(a) & 6{b), the plane and the cylindrical surfaces *are* obtained by direct tracing, while in Fig. 6(c) the final surface geometry is the envelope of the generatrices.

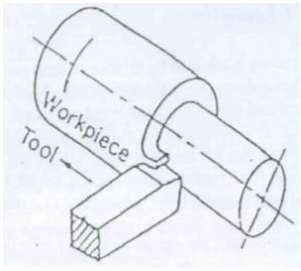


Fig. 1 Straight turning



Fig. 2 Straight boring

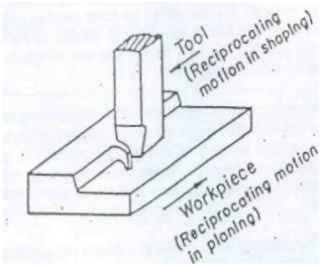


Fig. 3 Shaping and planning

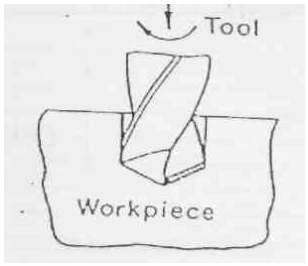


Fig. 5 Drilling

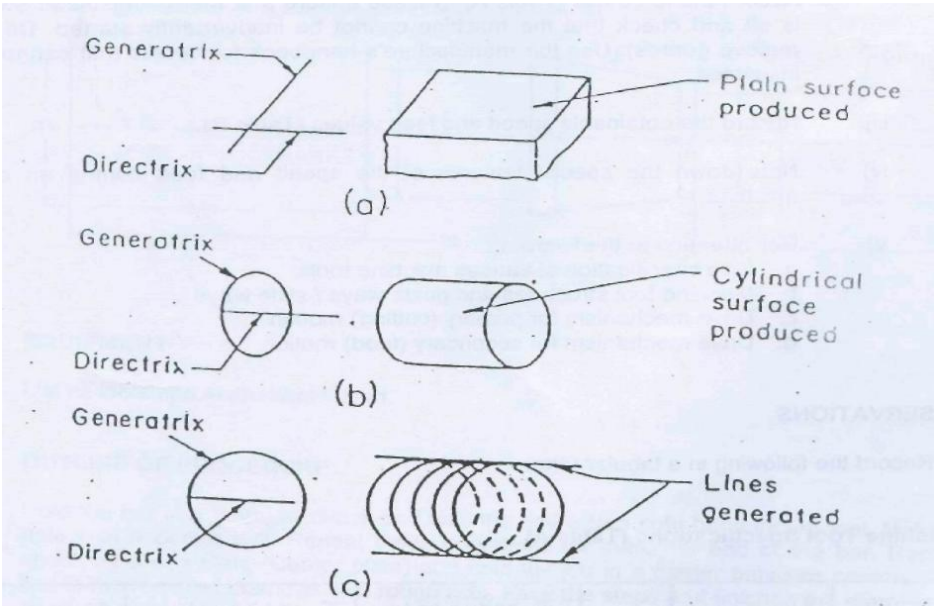


Fig.6 Concept of generatrix and directrix. (a) Rectilinear generatrix and directrix.
(b) Directrix perpendicular to the plane of generatrix.
(c) Directrix in the plane of generatrix.

INTRODUCTION TO LATHE MACHINE AND EXERCISE ON TURNING

PART (A)

OBJECTIVE : To study the characteristic features of lathe.

OUTLINE OF PROCEDURE

i) Run the machine at low speed and observe the motions, which control the shapes of the surfaces produced. Note particularly the features, which control the geometrical form of the surface.

ii) Learn the names of the major units and the components of each machine. Record these details (Table A). (Please ensure that the main isolator switch is off and check that the machine. cannot be inadvertently started. Do not remove guards). Use the manufacture's handbook for details that cannot be inspected.

Record the obtainable speed and feed values (Table B).

iii) Note down the special features of the speed and feed control on each machine. Pay attention to the

iv) following:

v) a.Size specification of various machine tools.
b.Machine tool structures and guide ways / slide ways.
c.Drive mechanism for primary (cutting) motion.
d.Drive mechanism for secondary (feed) motion.

OBSERVATIONS: (a) Record the following in a tabular form:

Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	
lathe							

Speed and Feed Data (Table B)

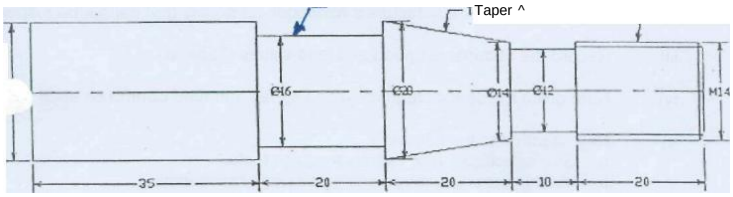
No.	Lathe	
	Speed	Feed
1.		
2.		
3.		
4.		
5.		
6.		
7.		
8.		

(b) Plot the lathe speeds against No. from Table B on a semi-log graph paper and show that the speed steps are in G.P.

PART(B)

OBJECTIVE: To make the part shown in the sketch from a mild steel rod on a Lathe.

22



(All dimensions are in mm)

EQUIPMENT: List all tools and instruments used.

OUTLINE OF PROCEDURE

Hold the bar in a three jaw chuck and face the end with a right hand facing tool. Make central hole with a center drill and support the work piece with revolving center. Turn the bar to the required diameter with rough cuts. Face the steps and finishes the diameters to the required sizes. Machine the groove with form tools. Machine the taper with the help of the cross-slide swiveling arrangement required surface. Cut the threads.

OBSERVATIONS

(a) Measure all dimensions (up to second decimal place) on the specimen turned by your group. Make a neat sketch and indicate all measured dimensions.
(b) Discuss briefly how tapered portion was turned.
(c) Show the calculation of the required gear ratio for thread cutting.
(d) Sketch the main drive unit of 'the- lathe and show how the speed steps are obtained.

Question Bank for laboratory quiz (ME Lab.) All answers must be brief and to the point. Sketches must be neat and self-explanatory.

1. What are the units of cutting speed and feed on machine tools?
2. What is the use of back gear arrangement in a lathe headstock?
3. How is the rotation imparted to a part, which is to be turned between centers?
4. What are the different ways of mounting the work on a lathe?
5. What is the use of Center drilling?
6. What is the function of chasing dial?
7. What is the difference between the lead and the pitch of a multistsrt thread?
8. Calculate the gear ratio between the spindle and the lead screw for cutting a screw with X threads per mm. The lead screw pitch isY mm.
9. Why is the main spindle of a lathe hollow?
10. List the type of surfaces produced by turning.
11. Sketch the plan/top view of different types of cutting tools you have used during the lathe exercise and indicate their respective names.
12. What are the instruments for measuring the diameters of turned shafts?
13. What is the instrument normally used for-measuring lengths of various parts?
14. Explain how you can determine the taper angle of a taper pin.
15. Calculate the time required for single pass straight turning of a cylindrical bar (diameter D length L) at a spindle speed of N rpm and feed F in appropriate units.
16. Main scale of a Venire has 10 divisions/cm and the least count of the instrument is 0.01mm. What should be the length of each division on the Vernier scale?
17. How are the spindle speeds changed?
18. What is the relation between spindle speed and cutting speed?
19. What is the relation between feed rate (mm/rev.) and spindle speed?
20. Are the back gears used to get lower or higher spindle speeds?
21. Draw figures of left hand and right hand turning tools.
22. How is the size of lathe specified?
23. Why are follow-rest and steady-test used?
24. What is Live Center and Dead center of the lathe?

MILLING: INTRODUCTION AND PRACTICE

PART (A)

OBJECTIVE: To study the characteristic features of Milling machine.

OUTLINE OF PROCEDURE

i) Run the machine at low speed and observe the motions, which control the shapes of the surfaces produced. Note particularly the features, which control the geometrical form of the surface.

ji) Learn the names of the major units and the components of each machine. Record these details (Table A). (Please ensure that the main isolator switch is off and check that the machine cannot be inadvertently started. Do not remove guards). Use the manufacture's handbook for details that cannot be inspected.

jii) Record the obtainable speed and feed values (Table B).

iv) Note down the special features of the speed and feed control on each machine.

v) Pay attention to the following:
a. Size specification of various machine tools.
b. Machine tool structures and guide ways / slide ways.
c. Drive mechanism for primary (cutting) motion.
d. Drive mechanism for secondary (feed) motion.

OBSERVATION : Record the following in a tabular form:

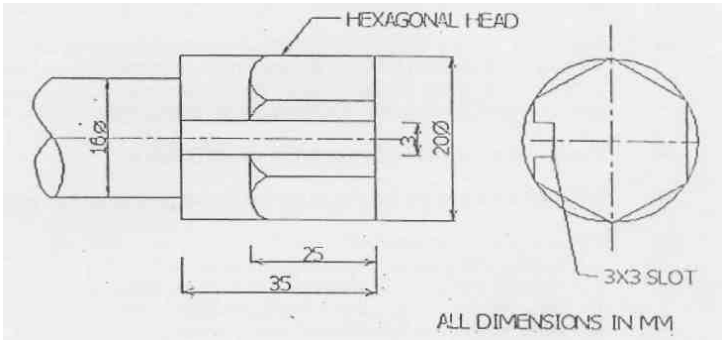
Machine Tool Specifications (Table A)

Machine Type & Make	Size	Speed given to		Feed given to Surface		Type of Produced	
		Tool	Work	Tool	Work		
Milling m/c							

Speed and Feed Data (Table B)

No.	Milling m/c.	
	Speed	Feed
1.		
2.		
3.		
4.		
5.		

PART (B) OBJECTIVE: To machine the hexagonal head and the slot shown in the sketch on the specimen,



EQUIPMENT: List all tools / cutters and instruments used.

OUTLINE OF PROCEDURE

Fit the helical cutter on the arbor and the specimen between the centers of the dividing head and the tail center. Carefully adjust the work piece so that the cutter just touches the top surface of the specimen. Calculate the necessary depth of cut and then mill the six *faces* of the hexagonal head in succession. Change the cutter and mill the rectangular slot.

OBSERVATIONS

(a)Measure all dimensions (up to second decimal place) on the specimen milled by your group. Make a neat sketch and indicate all measured dimensions.
(b)Explain in brief how the required indexing was obtained with the dividing head.
(c)Explain up-milling and down-milling operations. Which one did you use *for* slot milling and why?
(d)Explain the advantages of using a helical milling cutter.

Question Bank for laboratory quiz (ME Lab.) All answers must be brief and to the point. Sketches must be neat and self-expanatory.

1.How is a milling cutter mounted?
2.What is the main difference between a horizontal and a vertical milling machine?
3.Explain what is meant by a universal milling machine.
4.Why are helical tooth milling cutters usually preferred over straight tooth cutters for slab milling?
5.Why is down milling generally avoided?
6.What are the advantages of up milling?
7.What special - attachment is needed in the milling machine to perform down milling?
8.In what respect does a slitting saw differ from a narrow milling cutter?
9.How are milling cutters generally classified?
10.What is the difference between a plain milling cutter and a side-milling cutter? 11.How does a universal dividing head differ from a plain dividing head? 12.When does a universal indexing become essential?

SHAPING: INTRODUCTION AND PRACTICE

PART (A)

OBJECTIVE: To study the characteristic features of Shaper.

OUTLINE OF PROCEDURE

i)Run the machine at low speed and observe the motions, which control the shapes of the surfaces produced. Note particularly the features, which control the geometrical form of the surface.

ii)Learn the names of the major units and the components of each machine. Record these details (Table A). (Please ensure that the main isolator switch is off and check that the machine cannot be inadvertently started. Do not remove guards). Use the manufacture's handbook for details that cannot be inspected.

iii} Record the obtainable speed and feed values (Table B).

iv)Note down the special features of the speed and feed control on each machine.

v)Pay attention to the following:

a. Size specification of various machine tools.
b. Machine tool structures and guide ways I slide ways.
c. Drive mechanism for primary (cutting) motion.
d. Drive mechanism for secondary *(feed)* motion.

OBSERVATION Record the following in a tabular form:

Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	
Shaper M/c.							

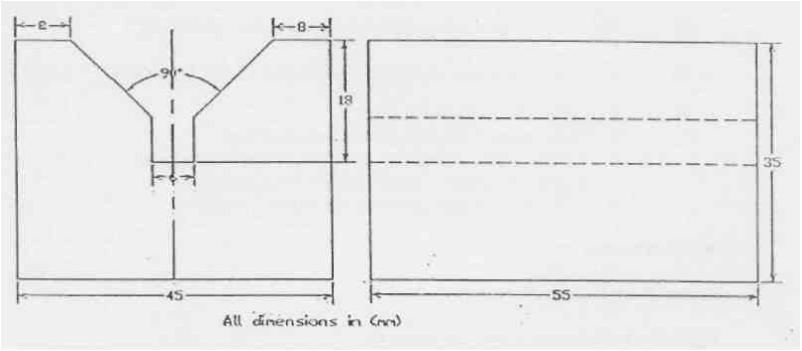
Speed and Feed Data (Table 2)

No.	Shaper M/c.	
	Speed	Feed
1.		
2.		
3.		
4.		
5.		

PART (B)

OBJECTIVE:

To machine a V-block as shown in the sketch out of the workpiece provided.



EQUIPMENT

List all tools and instruments used.

OUTLINE OF PROCEDURE

Hold the work piece in a vice and machine the bottom surface shown in the sketch. Invert the casting in the vice and machine the top surface till the desired height is obtained. Machine the inclined faces using right and left hand tools. Finally machine the groove.

OBSERVATIONS

(a)Measure all dimensions (up to second decimal place) on he specimen machined by your group. Make a neat sketch and indicate all measured dimensions.
(b)Explain -the quick return mechanism.
(c)Explain the use of clapper box on the machine.

Question Bank for laboratory quiz (ME Lab.) All answers must be brief and to the point. Sketches must be neat and self-explanatory.

1. What is the driving mechanism on the shaping machine? Why is quick return
2. effect important?
3. What happens to the quick return ratio when the stroke length is reduced?
4. How is feeding done on a shaping machine? Why is clapper box provided on
5. a shaper?

DRILLING AND FITTING

PART (A)

OBJECTIVE: To study the characteristic features of Drilling machine.

OUTLINE OF PROCEDURE

i) Run the machine at low speed and observe the motions, which control the shapes of the surfaces produced. Note particularly the features, which control the geometrical form of the surface.

ji) Learn the names of the major units and the components of each machine. Record these details (Table A). (Please ensure that the main isolator switch is off and check that the machine cannot be inadvertently started. Do not remove guards). Use the manufacture's handbook for details that cannot be inspected.

jii) Record the obtainable speed and feed values (Table B).

iv)Note down the special features *of* the speed and feed control on each machine,

v)Pay attention to the following:
a, Size specification *of* various machine tools,
b, Machine tool structures and guide ways I slide ways.
c. Drive mechanism for primary (cutting) motion,
d. Drive mechanism for secondary (feed) motion. Vi) Drill geometry - check for angles

OBSERVATION

Record the following in a tabular form:

Machine Tool Specifications (Table A)

Machine	Type & Make	Size	Speed given to		Feed given to		Type of Surface Produced
			Tool	Work	Tool	Work	
Drilling m/c							

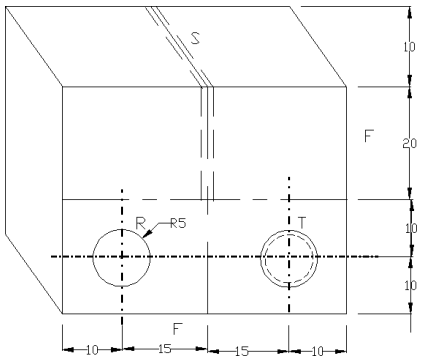
Speed and Feed Data (Table B)

No.	Drilling M/c.	
	Speed	Feed
1		
2		
3		
4		

PART (B)

OBJECTIVE:

To drill, file, as shown in the sketch, ream and tap holes on the mild steel plate.



R - Reamed hole 10mm T - Tapped hole M 10 S - Saw cut
F - Filled edge

EQUIPMENT : List all tools and instruments used.

OUTLINE OF PROCEDURE

File only two surfaces which make 90° each other mild steel work piece ensuring with a trisquare that the angle is 90°. Mark two centers of the holes. Punch at the center and also punch full length of the hexa cut from top to bottom. Drill And ream the holes as required. Tap the hole a set of three taps

OBSERVATIONS

(a) Measure all dimensions (up to second decimal place) on the specimen made by your group. Make a neat sketch and indicate all measured dimensions.
(b) Explain how power is transmitted *from* drill spindle to drill shank.' Sketch a reamer and show
(c) its main features.
(d) Explain why a set of three taps was used.

Question Bank for laboratory quiz (ME Lab.) All answers must be brief and to the point. Sketches must be neat and self-explanatory.

1. To which elements (tool and work) the speeds and feeds are provided on
2. Lathe (ii) Milling machine (m) Shaper and (iv) Drilling machine.
3. What type of speed variation mechanism is provided in the drilling machines you have studies?
4. What material is generally selected for the machine tool structure?
5. What types of guides are used for the main sideways of the basic machine tools?
6. How are the sizes of various basic machine tools specified?
7. Why are square threads used on driving screws of machine tools?
9. Which of the following process are intermittent? (a) Milling (b) Drilling (c) Shaping (d) Turning
10. What makes the simultaneous rotation of the spindle and the feed motion possible on drilling machines?
11. What are the functions of flutes on a twist drill?
12. Explain how power is transmitted from the drill spindle the drill shank?
13. Why are drilled holes generally slightly larger than drill diameter?
14. What is the primary purpose of reaming?
15. Sketch a twist drill and name its principle pats.
16. Name the principal kinds of reamer.
17. How is the diameter of a drilled hole measured?
18. What is the approximate order of magnitude by which hole diameter increases after reaming?
19. What will happen when the drilling is done with dull drill?
20. What are the types tapes in a hand operated tap set?
21. Why is a course file used for soft work materials?
22. Describe very briefly the important features of files?
23. Why the number of flutes on a reamer always even?
24. How is the drill held in spindle?
25. Why is the cutting fluid generally not used during drilling cast iron?
26. How are the files classified?
27. Distinguish between second cut and double cut files.
28. What is a scraper?
29. Does a saw blade cut on a return stroke?
30. What operation other than hole drilling can be performed on drilling machine?

Overview of Numerical Control (NC)

Various components present in NC machine tool are:

*Machine tool

*Machine Control Unit (MCU)

*Communication interface and accessories

The machine tool may be any type *of* machine tool used in the manufacturing industry. Machine control unit (MCV) is the control unit that reads and interprets the numerical data/part programme from the tap or any other media and passes on this information in the form of electrical signals to various activators / drive mechanisms *of* the machine to operate the machine tool in the desired way.

Numerical control Machines are classified based on the type of motion control, the presence *of* feed-back loops, the power drives the positioning system used and the number *of* axes *of* motion which can be controlled.

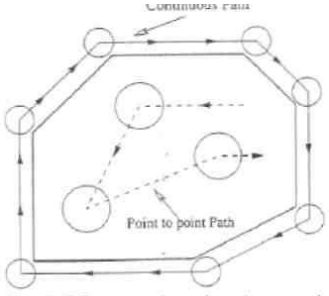


Fig. 15.15 Point-to-point and continuous path.

Two types *of* motion controls are used on NC machines. They are point-to-point or continuous-path controlled. As the name implies, the point-to-point (PTP) controlled machines move in a series *of* steps from one point to the other point. A machine with PTP control can perform very limited number *of* machining opera ions. In almost all the cases where PTP control is used, its function is to move the machine table or the spindle to a desired position to perform the machining operation at that position/point. In general, no machining is performed when the movement from one point to other is taking place, hence movement is made as rapidly as possible. When there is no need *of* machining between two points, path to be followed between them is not specified. Continuous path machines move uniformly and evenly by controlling the motion of two are more axes simultaneously. Here, the paths to be followed along with the destinations have to be mentioned as the machining is invariably done along that path. Figure 15.15 shows the cutting paths followed by PTP and continuous path machines.

Depending on the presence of the feedback, they are classified as open-loop system or closed-loop system (Fig.

15.16 and 15.17). In closed-loop systems, a transducer sends the current table position to the control unit and the driving unit (typically motor) stops running as the table reaches its desired position, hence accurate positioning can be achieved with closed-loop control systems. Closed-loop NC systems use servomotors where as open-loop systems use stepper motors.

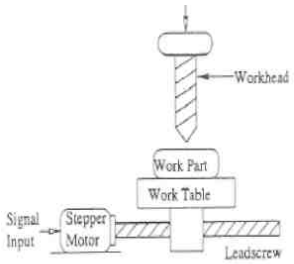


Fig. 15.16 Open-loop system.

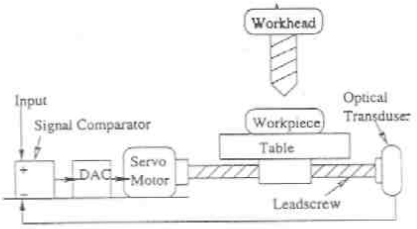


Fig. 15.17 Closed-loop system.

NC machines are classified into three types: hydraulic, pneumatic or electric based on the type of power used to drive the axes and spindle. Most of the modern NC machines use electric drives (AC or DC servo motors) to drive its axes and spindle because of its compact size, ease of control and low cost.

Depending on the positioning system used on the machine, they are classified as incremental or absolute positioning machines. Modern NC machine tools allow the programmer to choose either of the above mentioned positioning systems or both of them together through part programme. In absolute positioning system, the end point of the tool for a particular move has to be mentioned in the programme with respect to the origin. But in incremental system, the end point of the tool for a particular move has to be mentioned with respect to the current tool position.

Origin is a reference point used within the NC part program as the basis of defining tool location and other geometric entities. While developing an NC programme the programmer assumes that the tool is located at some specific point relative to the workpiece to be machined, and this point is designated as origin. In practice, during set-up and before starting the programme the operator has to move tile tool to a position which is designated as origin in that particular programme and then presses a button on control panel to select that as the origin. It means that the origin could be any where within the machine work space, hence it is called a floating zero. In the older generation NC machine tools, this facility was not available. The origin was a fixed point and the programmer had to write the programme with respect to that particular point.

The purpose of any production system is to produce a part in the most economic way without compromising on the desired qualities. Many large and small industries have undertaken the conversion from manual machines and processes to NC to get benefits. Some of these are listed below:

* Flexibiiity is high
* Scheduling is easier
* Set-up lead and processing times arc less
* Greater uniformity and accuracy in cutting
* Lover overall tooling costs
* Low inventory
* Inspection cost is less
* Higher degree of interchangeability of parts and tools

In spite of the above mentioned benefits, there are some negative points as well. The most prominent ones are:

* Large initial investments
* Programming costs
* Training and retraining costs for the existing work force

Numerical-Control Programming

1. NC PART PROGRAMMING

1. Coordinate Systems

In an NC system, each axis of motion is equipped with a separate driving source that replaces the hand wheel of the conventional machine. The driving source can be a DC motor, a stepping motor, or a hydraulic actuator. The source selected is determined mainly based on the precision requirements of the machine, as described in Chapter 9.

The relative movement between tools and workpieces is achieved by the motion of the machine tool slides. The three main axes of motion are referred to as the *X, Y.* and Z axes. The Z axis is perpendicular to both the *X* and Y axes in order to create a right-hand coordinate system, as shown in Figure 10.1.A positive Motion in the Z direction moves the cutting tool away from the workpiece. This is detailed as follows:

Z AXIS

1.On a workpiece -rotating machine, such as a lathe, the Z axis is parallel In the spindle, and the positive Motion moves the tool away from the workpiece (figure 10.2).

2.On a tool-rotating machine, such as a milling or boring machine, the Z axis is parallel to the tool axis, and the positive motion moves the tool away from the workpiece (Figures 10.3 and 10.4).

3. On other machines, such as a press, a planing machine, or shearing machine, the Z axis is perpendicular to the tool set, and the positive motion increases the distance between the tool and the workpiece.

X AXIS

1. On a lathe, the X axis is the direction of tool movement, and the positive motion moves the tool away from the workpiece
2. On a horizontal milling machine, the *X* axis is parallel to the table.
3. On a vertical milling machine, the positive *X* axis points to the right when the programmer is facing the machine.

The *Y* axis is the axis left in a standard Cartesian coordinate system.

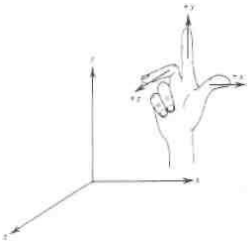


Figure 10.1 A right-hand coordinate system. (Y. Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.)

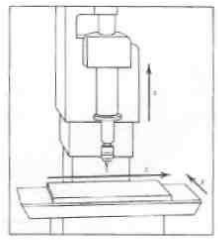


Figure 10.3 Coordinate system for a drill. (Y. Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.)

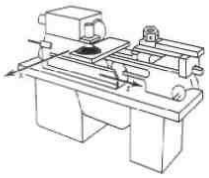


Figure 10.2 Coordinate system for a lathe. (Y. Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.)



Figure 10.4 Coordinate system for a mill. (Y. Koren, *Computer Control of Manufacturing Systems*, McGraw-Hill, 1983.)

10.1.2 NC Program Storage Media

Modern CNC controllers provide several ways of transferring data. Perhaps the most typical data-communication methods used to transfer part program files is an RS-232C interface (see Chapter 8). An NC part program is stored in a file on a computer or a CNC controller. The file download (or upload) can be initiated by setting up a transfer mode on the CNC controller. On the other side of the communication cable is a computer that sends or receives data byte by byte. The operator must start and end the data-transfer process on both the CNC controller and the computer. Some machines use higher-level protocols to ensure an error-free data transfer. Two of the higher-level protocols used are Kermit and *X*modem. Kermit and *X*modem arc widely accepted in the computer-to-computer telecommunication file-transfer process. These protocols allow the file transfer to be controlled by either the computer or the controller. The computer can send and retrieve data directly. Some machines also provide local-area network (LAN) instead of serial communication. Ethernet and MAP are two technologies used. Some CNC controllers allow the entire controller function to be initiated from a remote computer through the data-communication network.

10.1.5.1 Basic requirement *of* NC machine control.

To control a machine, it is necessary to begin by defining the coordinates *of* the tool motion. It is necessary to specify whether the motion is a positioning motion (rapid traverse) *or* a feed motion (cutting). 'The feed motion includes linear motion and circular motion. Linear motion requires the destination coordinates. When circular interpolation is used, the center of the circle must be given in addition to the destination. Before a cutting motion is called out, the spindle must be turned to the desired rpm and the feed speed must be specified. The spindle can rotate either clockwise *or* counterclockwise.

Sometimes coolant is required in machining, and the coolant may be applied in flood *or* mist form. If an automatic tool changer is present, the next tool number has to be known to the controller before a tool can be changed to the machine spindle. The sequence to change the tool also needs to be specified. It is often desirable to aggregate a fixed sequence *of* operations such as drilling holes into a cycle. Using cycle codes can drastically reduce programming effort. Additional information is needed for specific cycle operations. Finally, there are other programming functions, such as units-inch *or* metric-positioning system-absolute *or* incremental, and so on. All *of* these activities can (and in some cases must) be controlled through the NC controller and related part program. These control functions and data requirements are summarized in what follows:

(a)Preparatory functions: the words specify which units, which interpolator, Absolute *or* incremental Programming, which circular interpolation plane, cutter compensation, and so on.
(b)Coordinates: define three translational (and three rotational) axes.
(c)Machining parameters: specify feed and speed.
(d)Tool control: specifics tool diameter, next tool number, tool change, and so on.
(e)Cycle functions: specify drill cycle, ream cycle, bore cycle, mill cycle, and clearance plane.
(d) Coolant control: specify the coolant condition, that is, coolant on/off, flood. And mist.
(g) Miscellaneous control: specifies all other control specifics, that is, spindle *on/of,* tape rewind, spindle rotation direction, pallet change, clamps control, and so on. (h) Interpolators:
linear, circular interpolation, circle center, and so on.

These control functions are programmed through program words (codes).

1 0.1.5.2 NC words A specific NC function may be programmed using an *NC* word *or* a combination of NC words. All functions can be programmed in one block *of* a program. Many CNC controllers allow several of the same “word" be present in the same block. Thus, several functions can be included in one block, this is normally done by using a word-address format, which is the most popular format used in modern CNC controllers. The sequence *of* the words within one block is usually not important, except for the sequence number that must be the first word in the block. In order to make a program more readable, it is a good practice to follow a fixed sequence. Each word consists *of* a symbol and a numeral. The symbol is either N, G, X, V, and so on. Numerals follow as data in a prespecified format. For example, the format for an *X* word might he ".3.4:' which means three, digits before the decimal point and four digits after the decimal are used. The function of each NC word (code) and their applications are discussed in what follows: **N-code**. A part program block usually begins with an “N” word. The N word specifies the sequence number. It is used to identify the block within the program. It is especially useful for program editing. For example when the format is “4” a proper sequence number would be

N00I0

It is a good practice to program N values in increments of 10 or greater. This allows additional blocks to be inserted between two existing blocks.
G-code. The G-code is also called preparatory code or word. It is used to prepare the MCU for control functions. It indicates that a given control function is requested or that a certain unit or default be taken. There are modal functions and non modal functions. Modal functions are those that do not change after they have been specified once, such as unit selection. Non modal functions are active in the block where they are specified. For example, circular interpolation is a non modal function. Some commonly used G-codes arc listed in the Table 10.1. Some of these functions are explained in what follows.
G00 is the rapid traverse code that makes the machine move at maximum speed. It is used for positioning motion. When G01. G02, or G03 are specified, the machine moves at the feed speed. G01 is linear interpolation; G02 and G03 are for circular interpolation. For circular interpolation, the tool destination and the circle center are

Programmed in one block (explained later). G04 (dwell) is used to stop the motion for a time specified in the block. G08 and G09 codes specify acceleration and deceleration, respectively. They are used to increase (decrease) the speed of motion (feed speed) exponentially to the desired speed. Before an abrupt turn, decelerate the tool. Rapid acceleration in the new direction may cause a tool to break. The best accuracy can be obtained with acceleration and deceleration codes on and set to lower values. Most NC controllers interpolate circles on only

TABLE 10.1 G CODES

GO Rapid traverse	**G40 Cutter compensation cancel**
G01 Linear interpolation	**G41 Cutter compensation left**
G02 Circular interpolation CW	**G42 Cutter compensation right**
G03 Circular interpolation CCW	**G7 Inch format**
G04 Dwell	**0 Metric format**
	G7 1
G08 Acceleration	**G74 Full-circle programming off**
G09 Deceleration	**G75 Full-circle programming on**
G18 Z-X Plane	**G81-89 Fixed cycles**
G17 X-Y plane	**G80 Fixed-cycle cancel**
G19 Y-Z Plane	**G90 Absolute dimension program**
	G91 Incremental dimension

XY, YZ, and XZ planes. The interpolation plane can be selected using G 17, G18 or G 19. When a machine is equipped with thread-cutting capability, (G33-G35), the part program must specify the proper way to cut the thread. Codes G4O-G43 deal with cutter compensation. They simplify the cutter-center offset calculation. More details of cutter compensation are discussed later in Section 10.2.2. Most canned cycles are manufacturer-defined. They include drilling, peck drilling, spot drilling, milling, and profile turning cycles. The machine-tool manufacturer may assign them to one of the nine G codes reserved for machine manufacturers (GS)-G89). A user also can program the machine using either absolute (G90) or incremental (G91) coordinates. In the same program, the coordinate system can be changed. In order to simplify the presentation, most of the examples given in this chapter use absolute coordinate. Many controllers also allow the user to use either inch units (G70) or metric units (G71). Because hardwired NC circular interpolators work only in one quadrant and many CNC systems allow full-circle interpolation, a (G74) code emulates NC circular interpolation for CNC controllers. G75 returns the CNC back to the full-circle circular interpolation mode. *X,Y,Z ,A, B and C-Codes.* These words provide the coordinate positions of the tool. *X,* Y and Z define the three translational (Cartesian) axes of a machine. A. B. C are used for the three rotational axes about the *X, Y,* and Z axes. For a three axis there can be only three translational axes. Most applications only require *X.* Y. and Z codes in part programs. However, for four, five, or six-axis machine tools. A, B, and C are also used. The coordinates may be specified in decimal number (decimal programming) or integer number (BLU programming). For a controller with a data format of "3.4", to move the cutter to (1.12,2.275, 1.0), the codes are :

X1.1200 Y2.2750 Z1.OOO

In BLU programming, the programmer also may need to specify leading zero(s), or trailing-zero formats. A leading-zero format means that zeros must be entered in the space proceeding the numeric value. In this format, the controller locates the decimal point by counting the digits from the beginning of a number. In trailing-zero format, it is reversed. The number specified is in the BLU unit. The data format "3.4" implies that a BLU equals 0.0001 in. (fourth decimal place). By using the data from preceding example, the leading-zero program would be

X.0112 Y002275 Z00l

In the trailing-zero format, the program looks like

X.11200 Y22750 Z10000

For circular motion, more information is needed. A circular are is defined by the start and end points, the center, and the direction. Because the start point is always the current tool position, only the end point, the circle center. And the direction needs to be specified. I, J. and K words are used to specify the center. Usually, circular interpolation works only on either *X- Y, Y-Z.* or *X-Z* planes. When interpolating a circular are on the X- *Y* plane,

the I word provides the X. coordinate value of the circle center and the J word provides the Y coordinate value. X and Y words specify the end point. Clockwise or counterclockwise motions me specified by the G-code (G02 versus G03). There are many variations in circular interpolation programming. Each NC controller vendor has its own form and formal. Also they can depend on the combination of absolute or incremental, full-circle on or off modes. The following example is based on absolute programming with full circle on for a hypothetical controller.

F-Code. The F-code specifics the feed speed of the tool motion. It is the relative speed between the cutting tool and the work piece. It is typically specified in in./min (ipm). From a machinability data handbook, feed is given in in./rev (ipr). A conversion has to be done either by hand or on-hoard the controller. Some Controllers offer a G-code that specifies the ipr programming mode. When the ipr programming mode is used, the tool diameter and the number of teeth must he specified by the operator. The F-code must be given before G01, G02 or G03 can be used. Feed speed can be changed frequently in a program, as needed. When an F-code is present in a block, it takes effect immediately. To specify a 6.00-ipm feed speed for the cutting motion in Figure 10.7, one would program .

N0100 G02 X7.000 Y2.000 I5.000 J2.000

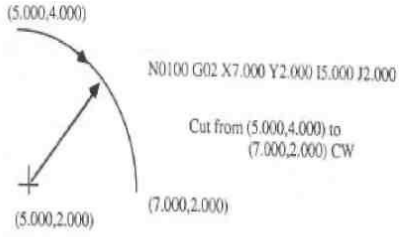


Figure 10.7 Cutting a circular arc.

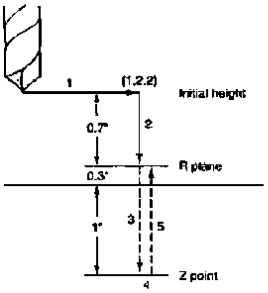


Figure 10.8 Drill cycle.

S-Code. The S-code is the cutting-speed code. Cutting speed is the specification of the relative surface speed of the cutting edge with respect to the work piece. It is the result of the tool (or workpiece in turning) rotation. Therefore, it is programmed in rpm. The Machinability Data Handbook (Machinability Data Center, 1980*)* gives these values in surface feet per minute (sfpm), and conversion is required before programming is done. When a controller is equipped with a sfpm programming option, the operator must specify the tool diameter. The S-code is specified before the spindle is turned on The S-code does not turn on the spindle. The spindle is turned on by an M-code. To specify a 1000-rpm spindle speed, the program block is

N0010 S1000

I-Code. The I-code is used to specify the tool number. It is used only when an automatic tool changer is present. It specifies the stot number on the tool magazine in which the next tool is located. Actual tool change does not occur until a tool-change M-code is specified.

R –Code. The R -code is used for cycle parameter. When a drill cycle is specified, one must give a clearance height (R plane) (see-Figure 10.8). The R-code is used to specify this clearance height. In Figure 10.8, the drill cycle consists of five operations:

1.Rapid to location (1.2.2).

2.Rapid down to the R plane.

3.Feed to the x point, the bottom of the hole.

4.Operation at the bottom of the hole, for example, dwelling.

5.Rapid or feed to either the R plane or the initial height. The cycle may be programmed in one block, such as (cycle programming is vendor-specific)

N0010. G81 X1.000 Y2.000 Z0.000 R 1.300

M-Code. The M-code is called the miscellaneous word and is used 10 control miscellaneous functions of the machine. Such functions include turn the spindle on/off, start/stop the machine, turn on/off the coolant, change the tool, and rewind the program (tape) (Table 10.2). M00 and M01 both stops the machine in the middle of a program. M01 is effective only when the optional stop button on the control panel is depressed. The program can be resumed through the control panel. M02 marks the end of the program. M03 turns on the spindle (clockwise). The spindle rpm must be specified in the same line or in a previous line. M04 is Similar to M03, except it turns the spindle on counterclockwise. M05 turns off the spindle. M06 signals the tool-change operation. On a machine equipped with an automatic tool changer, it stops the spindle, retracts the spindle to the tool-change position, and then changes the tool to the one specified in the T -code. M07 and M08 turn on different modes of coolant. MO9 turns off the coolant. M30 marks the end of the tape. It stops the spindle and rewinds the program (tape). On some controllers, more than one M-code is allowed in the same block.

TABLE 10.2 M-CODES

M00	**Program stop**	**M06**	**Tool change**
M01	**Optional stop**	**M07**	**Flood coolant on**
M02	**End of program**	**M08**	**Mist coolant on**
M03	**Spindle CW**	**M09**	**Coolant off**
M04	**Spindle CCW**	**M30**	**End of tape**

2. MANUAL PART PROGRAMMING

1. Part Programs

In manual part programming, the machining instructions are recorded on a document, called a part-program manuscript (see Figure 10.9) by the part programmer. The manuscript is essentially on ordered list of program blocks. The manuscript is other entered as a computer file or punched on a paper tape. Each symbol on the numeric or special characters, corresponds to a perforation(s) on the tape (or magnetic bit pattern on a disk) and is referred to as a character. Each line:

of the manuscript is equivalent to a block on the: punched tap and is followed by an EOB (end-of-block) character. When it is stored in a computer file, a tape-image format is used.

Because a part program records a sequence of tool motion and operations to produce the final part geometry one must prepare a process plan with setups and fixtures before writing the program (see Chapter 13).The work piece location and orientation, features (holes. slots, pockets) to be machined, tools and cutting parameters used need to be determined. We will use an example to illustrate how a part is programmed.

Example 10.1

The part drawing shown in Figure 10.10 is to he machined from a 4-in. x 4-in.x 2-in, work piece. The work piece material is low-carbon steel. We will use a hypothetical 3-axis CNC machining center for the process. The process plan for the part is as follows:

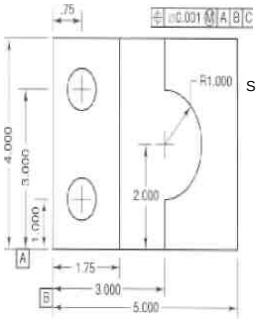
1. Set the lower left bottom corner of the part as the machine zero point (floating-zero programming).
2. Clamp the work piece in a vice.
3. Mill the slot with a ¾-in four flute flat ,end mill made of carbide. Form the machinability data hand book, the recommended feed in 0.005 in./tooth/rev and the recommended cutting speed is 620 fpm.
4. Drill two holes with a 0.75-in –diameter twist drill. Use 0.18-ipr feed and 100-rpm speed.

Figure 10.11 shows the setup, fixturing and cutter path. Write a part program for the part.

Solution The cutting parameters need be converted into rpm and ipm. Milling:

$$rpm = \frac{12V}{\pi D} = \frac{12 \times 620 \text{ fpm}}{\pi \times 0.75 \text{ in}} = 3157 \text{ rpm}$$

$$Vf = n\prime rpm = 4 \text{ tpr} \times 0.005 \text{ iprpt} \times 3157 \text{ rpm} = 63 \text{ ipm}$$



S

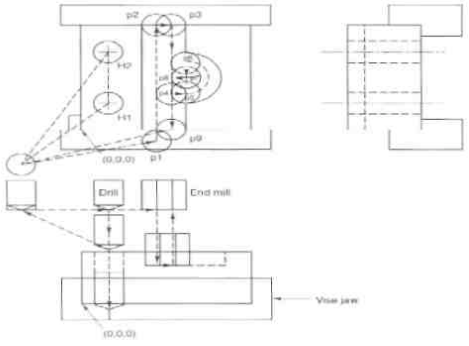


Figure 10.11 Setup and cutter path for the example part.

The coordinates of each point (cutter location) are

p1': (1.75 + 0.375, −0.1 − 0.375,400) = (2.125,−0.475,4.000)
p1: (2.125, −0.475,2.000 − 0.5000) = (2.125,−0.475,1.500)
p2: (2.125,4.000 + 0.100,1.500) = (2.125,4.100,1.500)
p3: (3.000 − 0.375,4.100,1.500) = (2.625,4.100,1.500)
p4: (2.625,2.000 − 1.000 + 0.375,1.500) = (2.625,1.375,1.500)
p5: (3.000,2.000 − 1.000 + 0.375,1.500) = (3.000,1.375,1.500)
p6: (3.000,2.625,1.500)
p7: (3.000,2.000,1.500)
p8: (2.625,2.000,1.500)
p9: (2.625, −0.100,1.500)
p9': (2.625, −0.100,4.000)

Drilling: All dimensions in inches, All tolerance A * 0.001 in.

$$\text{Rpm} = \frac{12V}{\pi D} = \frac{12 \times 100}{\pi \times 0.75\ \text{in}} = 509\ \text{rpm}$$

$Vf = f\,\text{rpm} = 0.018\ \text{ipr} \times 509\ \text{rpm} = 9.16\ \text{ipm}$

For the milling operation, the cutter is smaller than the slot, and two passes are required. The cutter first moves to p1 (the prime denotes the upper point).There must be clearance between the cutter and the workpiece, so that the cutter will not touch the workpiece during rapid positioning. We will use 0.1in as the clearance. The cutter then plunges down to p1, which is on the slot bottom level. Both p2 and p3 are outside the workpiece to ensure the slot edges are completely cut. The center of the cutter overhangs the edge by 0.1 in. The cutter moves to p4 from p3 to clear the slot. p5 is the beginning of a circular interpolation and p6 is the end. From p6, the cutter moves to p7 to clear the center of the circular area and then moves to p8. After the milling operation, a drill is installed in the spindle through an automatic tool change. The two holes are drilled using a drilling cycle. Combining the information from the process plan and the cutter-location data, a part program can be written. The program for the example part is shown in Figure 10.12. A step-by-step explanation is presented on the right-hand-side of the figure. The part program is verified using a program called Mac CNCS, and the results are in Figure 10.13 the result in figure 10.13 is the same as what is shown in Figure 10.11: Thus the program is correct. A three-dimensional view can also be found in Figure 10.14.

Part program	Explanation
N0010 G70 G90 T08 M06	Set the machine to the inch format and absolute dimension programming.
	Rapid to p1
N0020 G00 X2.I25 V-0.475 Z4.000 53157	Down feed to pl. spindle CW
N0030 G01 Z1500 F63 M03	Feed to p2
N0040 G01 Y4.100	To p3
N0050 G01 X2.625	To p4.
N0060 G01 Y.1375	To p5.
N0070 G01 X3.000	Circular interpolation to p6.
N0080 G03 Y2.625 I 3.000 J2.000	To p7.
N0090 G01 Y2.000	To p8
N0100 G01 X2.625	To p9.
N0110 G01 Y-0.100	To p9 spindle off, tool sign.2.
N0120 G00 Z4.000 T02 M05	Tool change, set new feed and speed.
N0130 F9.16 S509 M06	Drill hole 1.
N0140 G81 X0.750 Y1.000 Z-0.1 R2.100 M03	Drill hole 2.
N0150 G81 X0.750 Y3.000 Z-0.1 R2.100	Move to home position, stop the machine.
N0160 G00 X. 1.000 Y-1.000 M30	

Figure 10.12 Part program for the part in Figure 10.10

FOR LATHE

Survey of **G commands** for command Definition C

G00. . ..... Positioning (rapid traverse) G01 ..... Linear interpolation (feed)
G02 ..... Circular interpolation clockwise
G03 ..... Circular interpolation counterclockwise G04+ ..... Dwell
G7.1 ..... Cylindrical Interpolation G10 ..... Data setting
G11 ..... Data setting Off
G12.1 ..... Polar Coordinate Interpolation ON G13.1 ..... Polar Coordinate Interpolation OFF G17 ..... Plane selection)0(
G18 ..... Plane selection ZX G19 ..... Plane selection YZ
G20 ..... Longitudinal turning cycle G21 ..... Thread cutting cycle
G24 ..... Face turning cycle
G28+ ..... Return to reference point G33 ..... Thread cutting
G40 ..... Cancel cutter radius compensation
G41 ..... Cutter radius compensation left
G42 ..... Cutter radius compensation right
G70 ..... Inch data Input
G71 ..... Metric data Input G72+ ..... Finishing cycle

G73+ Stock removal in turning Stock removal
G74+ in facing Pattern repeating
G75+ Deep hole drilling, cut-in cycle in Z Cut-
G76+ in cycle in X
G77+ Multiple threading cycle
G78+
G80 ..... Cancel cycles (G83 up to G85) G83 ..... Drilling cycle'
G84 ..... Tapping cycle
G85 ..... Reaming cycle
G90. . ..... Absolute programming
G91 Incremental programming
G92+ ..... Coordinate system setting, spindle speed l it G94 ..... Feed per minute
G95. . ..... Feed per revolution
G96 Constant cutting speed
G97. . ..... Direct spindle speed programming G98. . ..... Return to initial plane
G99 ..... Return to withdrawal plane

1. . ..... Initial status
+ ..... Block wise effective

Survey of **M commands**

M00 Programmed stop
M01 Programmed stop, conditional M02 Program End

M03 Main spindle on CW M05* Main spindle off M08 Coolant ON
M10 Lock dividing head M19 Oriented spindle Stop
M25 Release Clamping Device
M30 Program End
M71 Puff blowing ON M98 Sub program Call

M04 Main spindle on CCW
M06 Tool change M09* Coolant OFF M11 Unlock dividing head

M26 Close Clamping Device

M72* Puff blowing OFF
M99 Sub program End

* Initial status

FOR MILLING

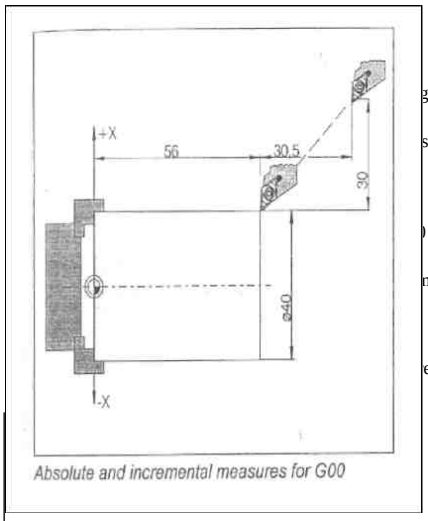
G commands

G00* Positioning (Rapid Traverse) G01 Linear Interpolation
G02 Circular Interpolation Clockwise
G03 Circular Interpolation Counterclockwise G04# Dwell
G09# Exact Stop G10 Data Setting
G11 Data Setting Off
G15* End Polar Coordinate Interpolation
G16 Begin Polar Coordinate Interpolation G17* Plane Selection XY
G18 Plane Selection ZX G19 Plane Selection YZ G20 Measuring in Inches G21 Measuring in Millimeter
G28# Approach Reference Point
G40* Cancel Cutter Radius Compensation
G41 Cutter Radius Compensation left G42 Cutter Radius Compensation right G43 Tool Length Compensation positive G44 Tool Length Compensation negative G49* Cancel Tool Length Compensation G50* Cancel Scale Factor
G51 Scale Factor
G52# Local Coordinate System G53# Machine Coordinate System G54* Zero Offset 1
G55 Zero Offset 2 G56 Zero Offset 3 G57 Zero Offset 4 G58 Zero Offset 5 G59 Zero Offset 6 G61 Exact Stop Mode
G62 Automatic Corner Override G63 Thread Cutting Mode On G64* Cutting mode
G68 Coordinate System Rotation ON
G69 Coordinate System Rotation OFF G73 Chip Break Drilling Cycle
G74 Left Tapping Cycle G76 Fine Drilling Cycle
G80* Cancel Drilling Cycles (G83 bis G85)
G81 Drilling Cycle
G82 Drilling Cycle with Dwell G83 Withdrawal Drilling Cycle G84 Tapping Cycle
G85 Reaming Cycle
G86 Drilling Cycle with Spindle Stop
G87 Back Pocket Drilling Cycle
G88 Drilling Cycle with Program Stop
G89 Reaming Cycle with Dwell
G90* Absolute Programming G91 Incremental Programming
G92# Coordinate System Setting G94* Feed per Minute
G95 Feed per Revolution G97* Revolutions per Minute
G98* Retraction to Starting Plane (Drilling Cycles) G99 Retraction to Withdrawal Plane

* Einschaltzustand
# Nur satzweise wirksam

LATHE Description of G Commands

G00 Positioning (Rapid Traverse)



Absolute and incremental measures for G00

Format

N…. G00 X(U)….Z(W)….

The slides are traversed at maximum speed to the grammed target point

Incremental and absolute commands can be used at same time.

Note

A programmed feed F will be suppressed while

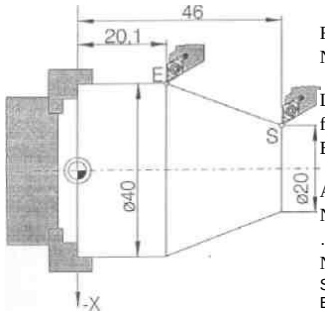
The maximum feed is defined by the producer of machine.

The feed override switch is limited to 100%

Example Absolute G90 N50 G00 X40 Z56 emental G91 N50 G00 U-30 W-30.5

G01 Linear Interpolation (Feed)

i+X



Format

N…. G01 X(U)….Z(W)….F…

Linear slide movements (face, longitudinal, taper turning) at the programmed federate.

Example

Absolute G90

N50

…

N20 G01 X40 Z20.1 F0.1

S ..... Start point

E .....End point

Incremental G91 N..

G95 F0.1

Absolute and incremental measures for GOO

N20 G01 X20 W-25.9

CNC Questions.

1.What are the different components of NC machine tool?

2.What is MCU in NC and write down its function?

3.Clearly explain with figure point-to-point motion control and continuous path mo ion control used in NC machines? Which is control system is generally used to perform machining on the work pieces?

4.What is closed loop control system and open loop control system used in NC machine? What are the differences between them? What are the motors used for each type?

5.What is adaptive control system?

11. Write down the differences between incremental positioning and absolute positioning system used in NC machine?
12. Which system of co-ordinate system is used in NC machine? Write down the directions of positive X,Y,Z axis in case of lathe.
13. Briefly explain about the different types of tape format

    a) Fixed sequential format
    b) Block-address format
    c) Tab-sequential format,
    d) Word address format

14. What is ATC in CNC machine?
15. Write down the functions of few G codes like G01, 002 003,004 etc. 16. N0100 G02 X9.000 Y8.000 I5.000 J2.000 F6.000

Explain each .term for this block of part program.

17.What is BLU programming used in NC part programming?
18.Explain the terms “tool length compensation" and "cutter diameter compensation".
19.What is liner interpolation and what is circular interpolation?
20.What is canned cycle used in NC part program?
21.What is fixed zero system and floating zero system?
22.What is rapid traverse, fine feed and creep feed?
23.What are the different types of tool used in NC machine?

Part A 1st Lab

Objective--- To Study the characteristics of CNC Lathe and CNC Milling Machine Outlines of

Procedure.

1.Specification ,Size, Model No. of various M/C
2.Note down the names of the measure parts and component of the machine.
3.Note down the particularly feature, Control Systems and geometric form of surfaces

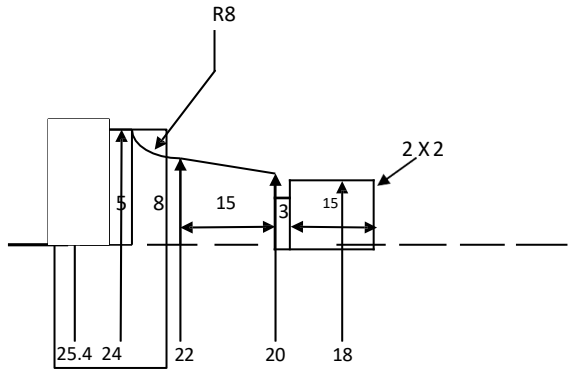
Observation

Machine	Type & Make	Size	Type of Bed	Speed given to Tool Work		Feed given to Work Tool		Types of profile or Contour genrated
CNC Lathe								
CNC Milling								

Part B

To write a Program of given Drawing for CNC Lathe

01



Format write a program

O_ _ _ _ (Program No.)

N5 T0101; (Tool selection)

N10 G90 G71; (G90-Absolute programming, G71-Metric data input)

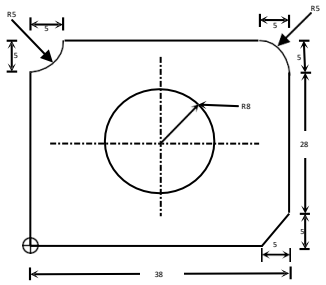
N15 G95 S… F… M04; (G95-Feed MM per revolutions-Spindle speed (rpm), F-Feed, M04-Main spindle on CCW) N20

.

.

N.. M30; (M30-Program End)

To write a Program of given Drawing for CNC Mill



O_ _ _ _ (Programm No.)

N5 G54; (G54-Zero Offset)

N10 T01 M06 G43 H1; (T01-Tool No.,M06- Tool Change,G43- Tool Length

Compensation Positive, H1-Offset Value for tool No. 1)

N15 G90 G21; (G90-Absolute programming,G21-Metric data input)

N20 G94 S..F..M03; (G94-Feed MM Per Minute, S-Spindle speed (rpm),F-Feed,M03-Main spindle on CW

.

.

.

N..M30; (M30-Program End)