

MACHINING

Level - III

Learning Guide 2

Unit of Competence: Perform Basic CAD/CAM Applications

Module Title: Performing Basic CAD/CAM Applications

LG Code: IND MAC3 02 0217

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Instruction Sheet	Learning Guide #1
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This learning guide is developed to provide you the necessary information regarding the following **content coverage** and topics:

- Determine job requirements
- Design detailed part drawing
- Translate CAD in CNC machine program
- Perform appropriate CAM / CNC operations
- Comply with quality assurance

This guide will also assist you to attain the learning outcome stated in the cover page. Specifically, **upon completion of this Learning Guide, you will be able to:**

- checked and interpreted drawing from work order
- limits and fits, surface texture, datum references and geometric tolerances to ensure functional operation and suitability according to standard
- Two-dimensional Computer Aided/Automated Design (CAD) blueprint of the part is reviewed for CAM application in compliance with standards
- Program is written into a standardized Computer Aided/Automated Manufacturing (CAM) code / process
- CAM/CNC operations are performed to produce part according to CAD drawing specifications

Learning Instructions:

1. Read the specific objectives of this Learning Guide.
2. Follow the instructions described below 3 to 35.
3. Read the information written in the information “Sheet.
4. Accomplish the “Self-check test.
5. Do the “LAP test”.

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Determine job requirements

Drawing set

Three main sets of production drawings include the following:

- Detail of each non-standard part on a drawing sheet, usually one part per sheet
- Assembly drawing showing all parts on one sheet
- A Bill of materials (BOM), essentially of each part

Elements of production drawings

The basic elements of production drawings include:

- ❖ Size and shape of component
- ❖ Format of drawing sheet
- ❖ Process sheet
- ❖ Projection method
- ❖ Limits, fits, and tolerances of size, form, and position
- ❖ Production method
- ❖ Indication of surface roughness and other heat treatments
- ❖ Material specification and Shape such as Castings, Forgings, Plates, Rounds, etc.
- ❖ Conventions used to represent certain machine components
- ❖ Inspection and Testing Methods

Specification of Standard Components

Basic principles of dimensioning in production drawings

The basic principles of dimensioning in production drawings include the following:

- The drawing module should dimension each feature only once.
- The drawing should show no more dimensions than necessary.
- Place dimensions outside the drawing view as far as possible.
- Represent dimensions by visible outlines rather than by hidden lines.
- Avoid dimensioning the center line, except when it passes through the center hole.
- Avoid intersecting projection or dimension lines.
- If the space for dimensioning is insufficient, you may reverse arrow heads and replace adjacent arrow heads with dots.

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Dimensioning technique

Any engineering drawing requires specifications in terms of dimensions.

Dimensions are classified as the following:

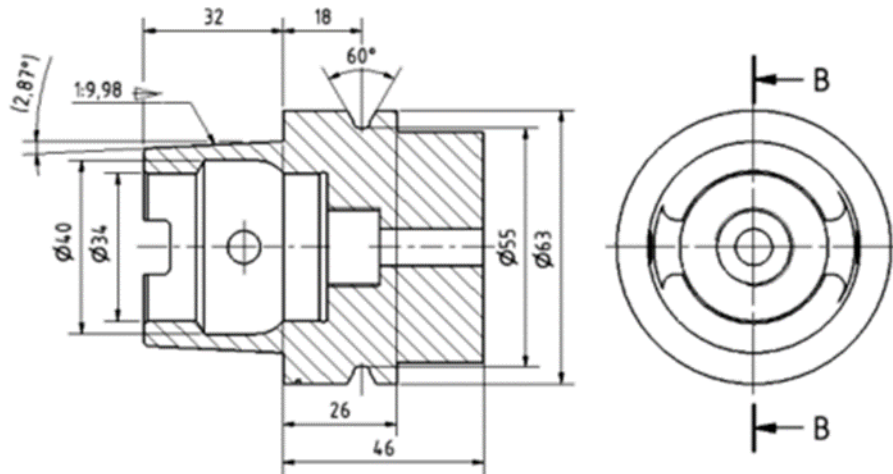
- Functional dimensions
- Non-functional dimensions
- Auxiliary dimensions

Non-functional dimensions are required for manufacturing.

Auxiliary dimensions do not govern manufacturing or inspection of parts. They are arranged in the following ways:

- ✓ Chain dimensioning: This method can be used only where the accumulation of tolerances does not affect the functional requirements.
- ✓ Parallel dimensioning: In this type of dimensioning, a number of single dimension lines are drawn parallel to one another, spaced so as to accommodate the dimensional values.
- ✓ Running dimensioning: This type of dimensioning is similar to parallel dimensioning; the only difference is that the dimensions are superimposed in one line. In this case, the origin point should be marked.
- ✓ Coordinate dimensioning: The location of each hole and its size is given by specifying X and Y coordinates from the defined origin and tabulating them.

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Production drawing in limits, fits and tolerance

Limit system

There are three terms used in the limit system:

- ❖ **Tolerance:** Deviation from a basic value is defined as Tolerance. It can be obtained by taking the difference between the maximum and minimum permissible limits.
- ❖ **Limits:** Two extreme permissible sizes between which the actual size is contained are defined as limits.
- ❖ **Deviation:** The algebraic difference between a size and its corresponding basic size. There are two types of deviations: 1) Upper deviation 2) Lower deviation

The fundamental deviation is either the upper or lower deviation, depending on which is closer to the basic size.

Tolerances

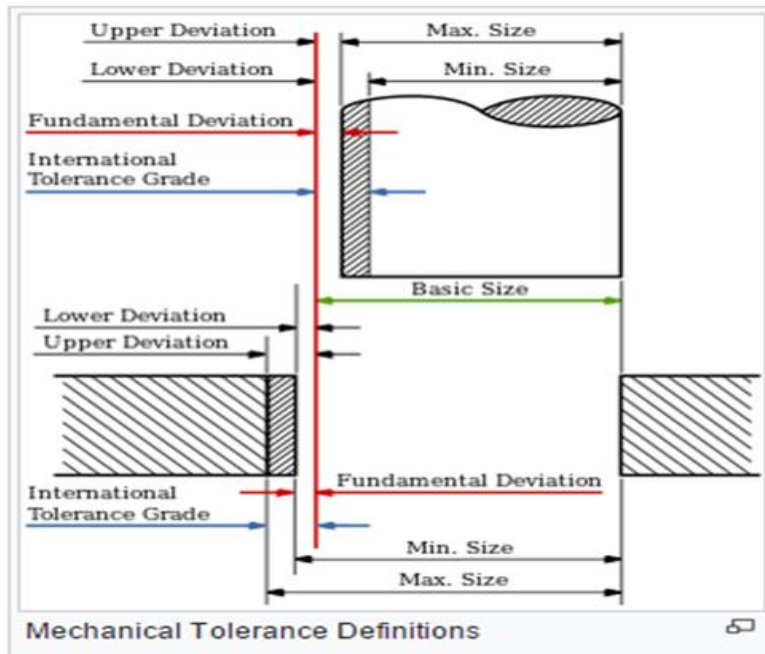
Mechanical Tolerance Definitions

Due to human errors, machine settings, etc., it is nearly impossible to manufacture an absolute dimension as specified by the designer. Deviation in dimensions from

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the basic value always arises. This deviation of dimensions from the basic value is known as Tolerance.

The figure shows mechanical tolerances which occur during operations.



Fits

The relation between tightness and looseness between two mating parts is called fit. Depending upon the actual limits of the hole or shaft sizes, fits may be classified as clearance fit, transition fit and interference fit.

Clearance fit

Clearance fit is defined as a clearance between mating parts. In clearance fit, there is always a positive clearance between the hole and shaft.

Transition fit

Transition fit may result in either an interference or clearance, depending upon the actual values of the tolerance of individual parts.

Interference fit

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Interference fit is obtained if the difference between the hole and shaft sizes is negative before assembly. Interference fit generally ranges from minimum to maximum interference.

The two extreme cases of interference are as follows:

❖ ***Minimum interference***

The magnitude of the difference (negative) between the maximum size of the hole and the minimum size of the shaft in an interference fit before assembly.

❖ ***Maximum interference***

The magnitude of the difference between the minimum size of the hole and the maximum size of the shaft in an interference or a transition fit before assembly.

Hole Basis and shaft basis system: In identifying limit dimensions for the three classes of fit, two systems are in use:

- ✚ Hole basis system: The size of the shaft is obtained by subtracting the allowance from the basic size of the hole. Tolerances are then applied to each part separately. In this system, the lower deviation of the hole is zero. The letter symbol indication for this is 'H'.
- ✚ Shaft basis system: The upper deviation of the shaft is zero, and the size of the hole is obtained by adding the allowance to the basic size of the shaft. The letter symbol indication is 'h'.

Production drawing in surface roughness

The properties and performance of machine components are affected by the degree of roughness of various surfaces; the higher the smoothness of the surface, the greater the fatigue strength and corrosion resistance will be.[9] Friction between mating parts is also reduced by a smoother surface finish. The geometrical characteristics of the surface, in relation to roughness, are as follows:

- Macro-deviations
- Surface waviness
- Micro irregularities

Surface roughness can be evaluated with the height and mean roughness index of micro-irregularities. Surface roughness is defined by the following terms:

- Actual profile
- Reference profile
- Datum profile

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- Main profile
- Mean roughness index
- Surface roughness number, etc.

Surface roughness number: The surface roughness number [R(a)] represents the average departure of the surface from the projections over the sampling length, which is expressed in micrometers. It is given by $R(a) = \{h_1 + h_2 + h_3 + \dots + h_n\} / n$.

Surface roughness can be measured using some of the following terms:

- Surface gauge
- Straight edge
- Profilograph
- Profilometer
- Optical flat, etc.

Introduction to CAD/CAM

Throughout the history of our industrial society, many inventions have been patented and whole new technologies have evolved. Perhaps the single development that has impacted manufacturing more quickly and significantly than any previous technology is the digital computer.

Computers are being used increasingly for both design and detailing of engineering components in the drawing office.

Computer-aided design (CAD) is defined as the application of computers and graphics software to aid or enhance the product design from conceptualization to documentation. CAD is most commonly associated with the use of an interactive computer graphics system, referred to as a CAD system. Computer-aided design systems are powerful tools and in the mechanical design and geometric modeling of products and components.

There are several good reasons for using a CAD system to support the engineering design function:

- To increase the productivity
- To improve the quality of the design
- To uniform design standards
- To create a manufacturing data base
- To eliminate inaccuracies caused by hand-copying of drawings and inconsistency between drawings

Computer-aided manufacturing (CAM) is defined as the effective use computer technology in manufacturing planning and control. CAM is most closely associated with functions in manufacturing engineering, such as process and production planning, machining, scheduling, management, quality control, and numerical control (NC) part programming.

Computer-aided design and computer-aided manufacturing are often combined CAD/CAM systems. This combination allows the transfer of information from the design into the stage of planning for the manufacturing of a product, without the need to reenter the data on part geometry manually. The database developed during CAD is stored; then it is processed further, by CAM, into the necessary data and instructions for operating and controlling production machinery, material-handling equipment, and automated testing and inspection for product quality.

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Rationale for CAD/CAM

The rationale for CAD/CAM is similar to that used to justify any technology-based improvement in manufacturing. It grows out of a need to continually improve productivity, quality and competitiveness. There are also other reasons why a company might make a conversion from manual processes to CAD/CAM:

- Increased productivity
- Better quality
- Common database with manufacturing
- Reduced prototype construction costs
- Faster response to customers

CAD/CAM Hardware

The hardware part of a CAD/CAM system consists of the following components

- one or more design workstations,
- digital computer,
- plotters and other output devices, and
- storage devices.

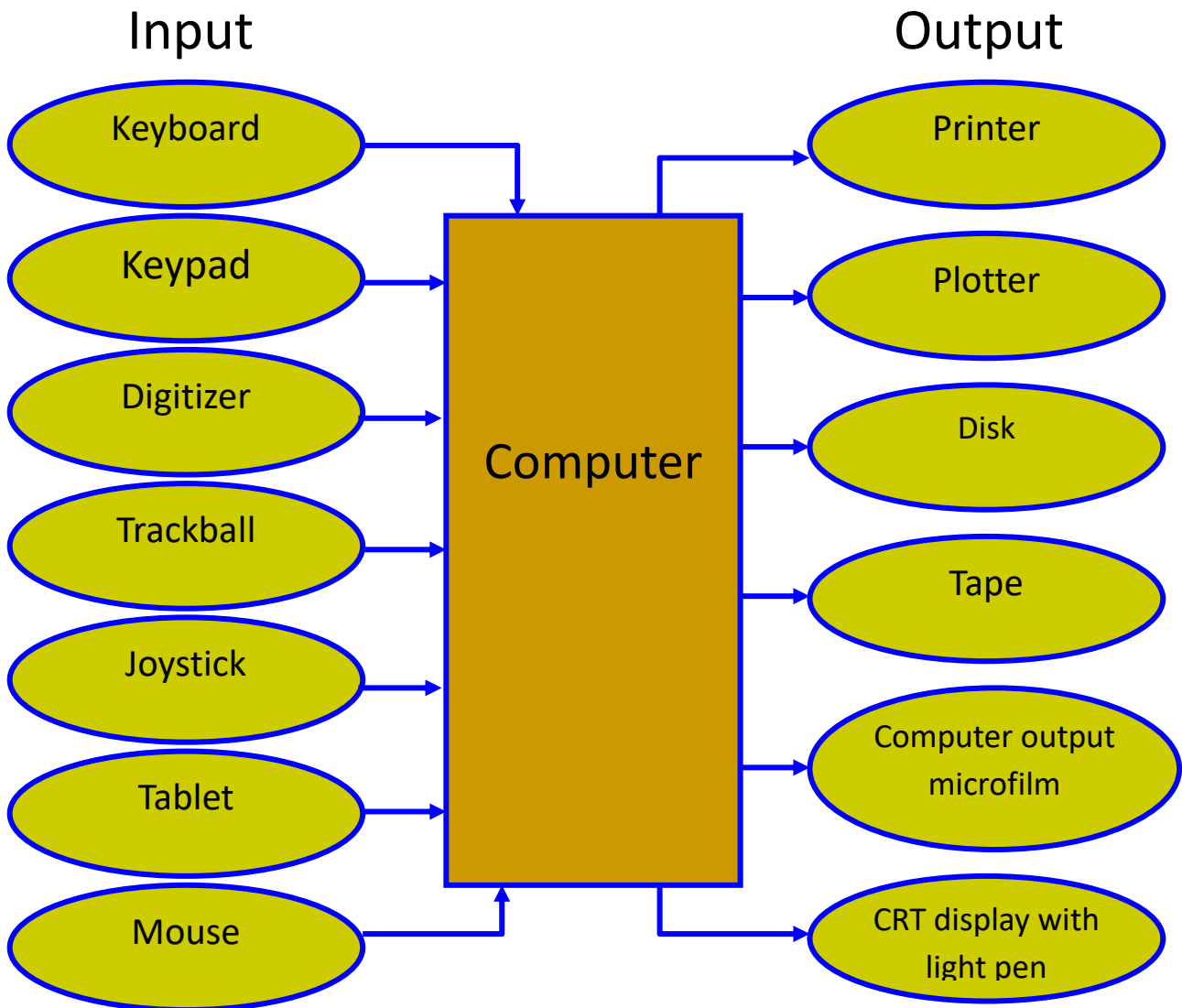
In addition, the CAD/CAM system would have a communication interface to permit transmission of data to and from other computer systems, thus enabling some of the benefits of computer integration.

The workstation is the interface between computer and user in the CAD system. The design of the CAD workstation and its available features have an important influence on the convenience, productivity, and quality of the user's output. The workstation must include a digital computer with a high-speed control processing unit (CPU). It contains a logic/arithmetic section for the system. The most widely used secondary storage medium in CAD/CAM is the hard disk, floppy diskette, or a combination of both.

Input devices are generally used to transfer information from a human or storage medium to a computer where "CAD functions" are carried out. There are two

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basic approaches to input an existing drawing: model the object on a drawing or digitize the drawing.



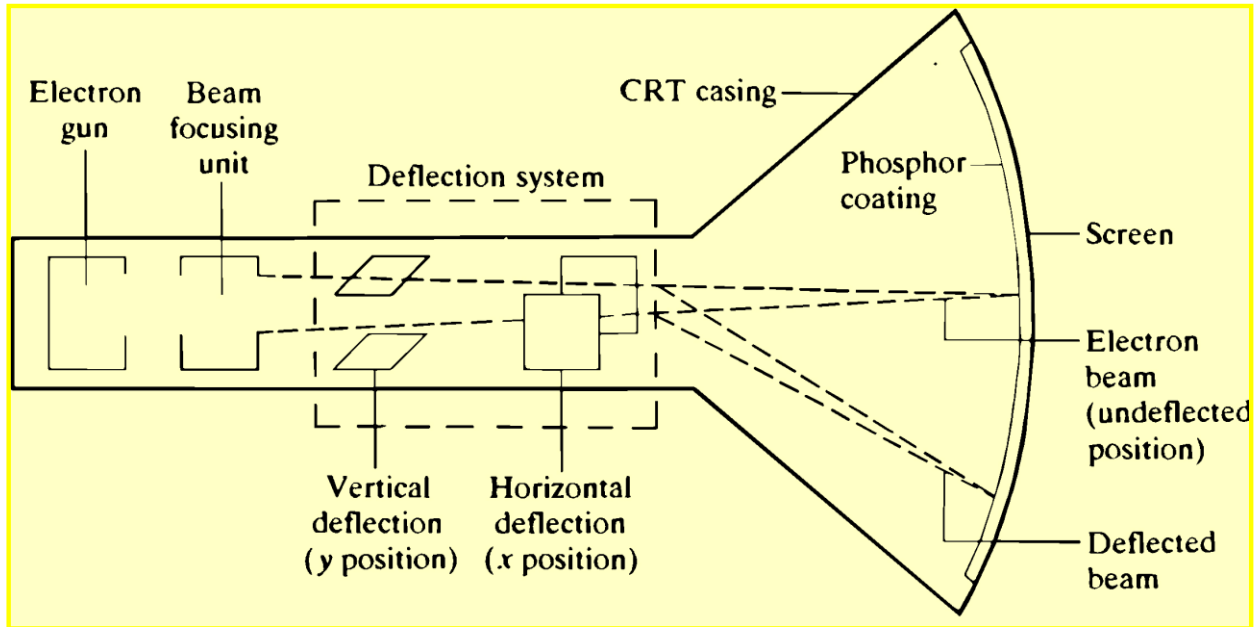
The standard output device for CAD/CAM is a CRT display.

There are two major of CRT displays: random-scan-line-drawing displays and raster-scan displays. In addition to CRT, there are also plasma panel displays and liquid-crystal displays.

CRT = Cathode ray tube

The standard output device for CAD is a monitor display. Among the available technologies, the CRT is the most dominating and has produced a wide range of extremely effective graphics display.

Schematic diagram of a CRT



CAD/CAM Software

Software allows the human user to turn a hardware configuration into a powerful design and manufacturing system. CAD/CAM software falls into two broad categories, 2-D and 3-D, based on the number of dimensions. 2-D representations of 3-D objects is inherently confusing. Equally problem has been the inability of manufacturing personnel to properly read and interpret complicated 2-D representations of objects. 3-D software permits the parts to be viewed with the 3-D planes-height, width, and depth-visible. The trend in CAD/CAM is toward 3-D representation of graphic images. Such representation approximate the actual shape and appearance of the object to be produced; therefore, they are easier to read and understand.

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Applications of CAD/CAM

The emergence of CAD/CAM has had a major impact on manufacturing, by standardizing product development and by reducing design effort, tryout, and prototype work; it has made possible significantly reduced costs and improved productivity.

Some typical applications of CAD/CAM are as follows:

- Programming for NC, CNC, and industrial robots;
- Design of dies and molds for casting, in which, for example, shrinkage allowances are preprogrammed;
- Design of tools and fixtures and EDM electrodes;
- Quality control and inspection----for instance, coordinate-measuring machines programmed on a CAD/CAM workstation;
- Process planning and scheduling.

AutoCAD is a computer-aided drafting and design system implemented on a personal computer. It supports a large number of devices. Device drivers come with the system and include most of the digitizers, printer/plotters, video display boards, and plotters available on the market.

AutoCAD supports 2-D drafting and 3-D wire-frame models. The system is designed as a single-user CAD package. The drawing elements are lines, poly-lines of any width, arcs, circles, faces, and solids. There are many ways to define a drawing element. For example, a circle can be defined by center and its radius, three points, and two end points of its diameter. The system always prompts the user for all options.

Of course, the prompt can be turned off by advanced users. Annotation and dimensioning are also supported. Text and dimension symbols can be placed on anywhere on the drawing, at any angle, and at any size. A variety of fonts and styles are also available.

Computer-Aided Manufacturing

The effective use of computer technology in manufacturing planning and control

- Most closely associated with functions in manufacturing engineering, such as process planning and NC part programming

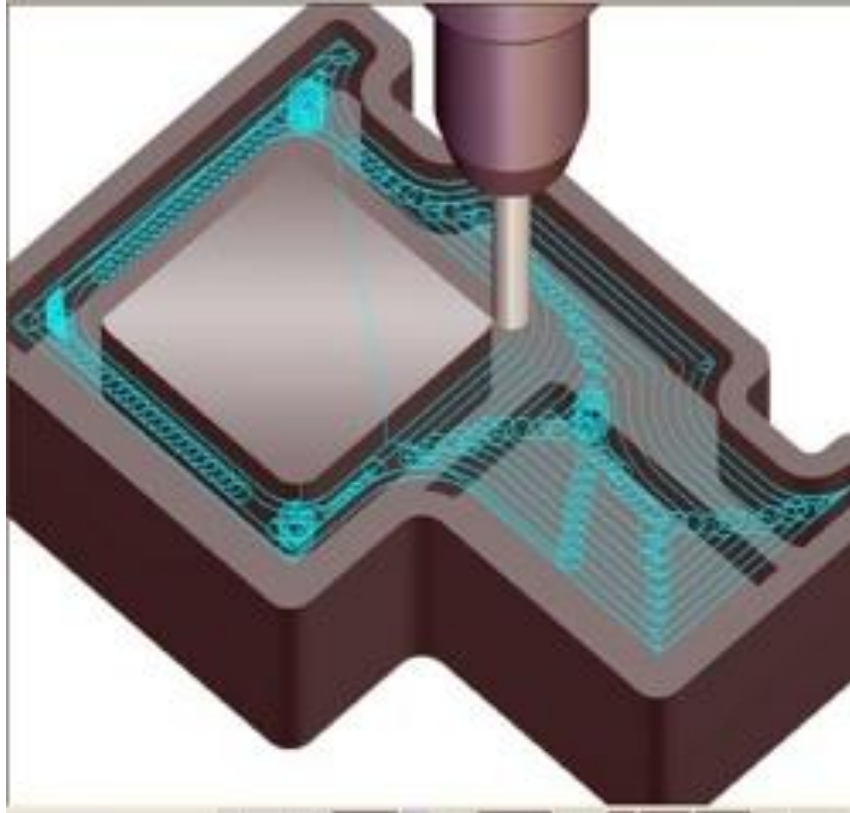
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- CAM applications can be divided into two broad categories:
 1. Manufacturing planning
 2. Manufacturing control

Includes the use of CAD files to:

1. Define the machine tool that will process the part
2. Define the stock or material
3. Define the features to machine
4. Generate operations
5. Select the origin
6. Generate tool paths
7. Simulate tool paths
8. Generate NC code
9. Download NC programs
10. Operate the CNC machine that will cut the part

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Data Communication Processes in CAD/CAM

In CAD/CAM:

- Data is created
 - Data is simulated
 - Data is translated
 - Data is stored
 - Data is manipulated
 - Data is analyzed
 - Data is interpreted
- Data is generated
 - Data is transformed
 - Data is cleaned
 - Data is communicated
 - Data is managed
 - Data is retrieved

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CAM Applications in Manufacturing Planning

- Computer-aided process planning (CAPP)
- Computer-assisted NC part programming
- CAD/CAM assisted NC part programming
- Computerized machinability data systems
- Computerized work standards
- Cost estimating
- Production and inventory planning
- Computer-aided assembly line balancing

CAM Applications in Manufacturing control

- Process monitoring and control
- Quality control
- Shop floor control
- Inventory control
- Just-in-time production systems

What Is CAM?

The use of a computer to aid in manufacturing is called computer-aided manufacturing, or simply CAM. This is the stage during which you define the steps for turning your virtual part into a reality.

The first stage in the CAM process is fixture design and tool selection, as shown in Figure 1. During this stage, the part designer determines what is needed to make the particular part. This includes determining the tools that are required to

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manufacture the part and how the part will be secured for machining. Figure 2 shows various tools that can be loaded on the machine.

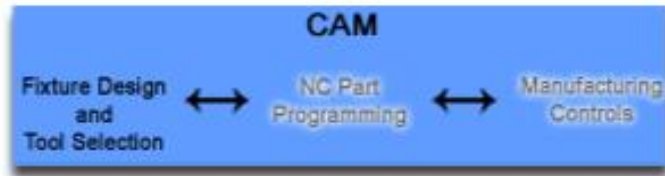


Figure 1. Fixture design and tool selection is the first stage of CAM.



Figure 2. The designer specifies which cutting tools the operator must use to make the part.

CAM uses CAD information to create toolpaths for various cutting operations such as milling, drilling, and turning. Figure 3 shows a sample toolpath. A computer generates and verifies these instructions, though they may be modified by a programmer for optimization. Likewise, an engineer may inspect the toolpath to make sure there is no interference between tools and clamps, fixtures, and other machine components.

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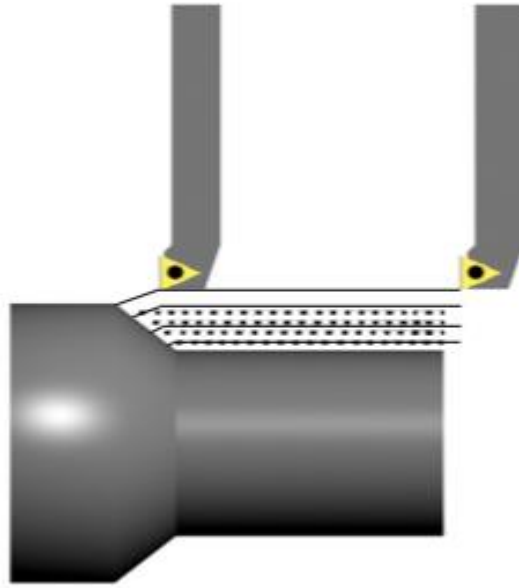


Figure 3. Toolpaths can be generated, inspected, and verified.

Manual and Computer-Assisted Part Programming

Traditional part programs are manually created by painstakingly translating a drawing of a part into instructions for the machine tool and the machine control unit. This process is tedious, time intensive, and prone to error.

Manual programming is still common and has been made easier with machine control features such as canned cycles and subprograms. Canned cycles are predetermined tool movements that act as programming shortcuts. They execute common, repetitive machining operations such as drilling, tapping, and threading. Instead of writing out a long list of tool movements, a programmer can plug a canned cycle into any part program with a single line of programming code. Figure 1 shows a common canned cycle.

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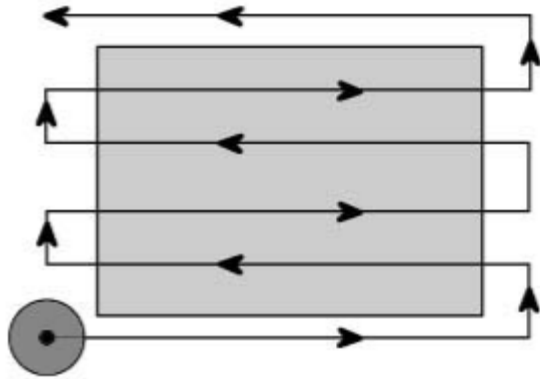


Figure 1. Canned cycles eliminate the tedious step-by-step programming of common machine cuts, like the one above.

Subprograms, also known as subroutines, are sequences of commands in a part program that may be called on repeatedly, whenever they are needed. Subprograms are used to simplify manual part programming by creating common sequences once. They are called on in the main program when they are needed. Figure 2 illustrates how a subprogram is called upon when needed.

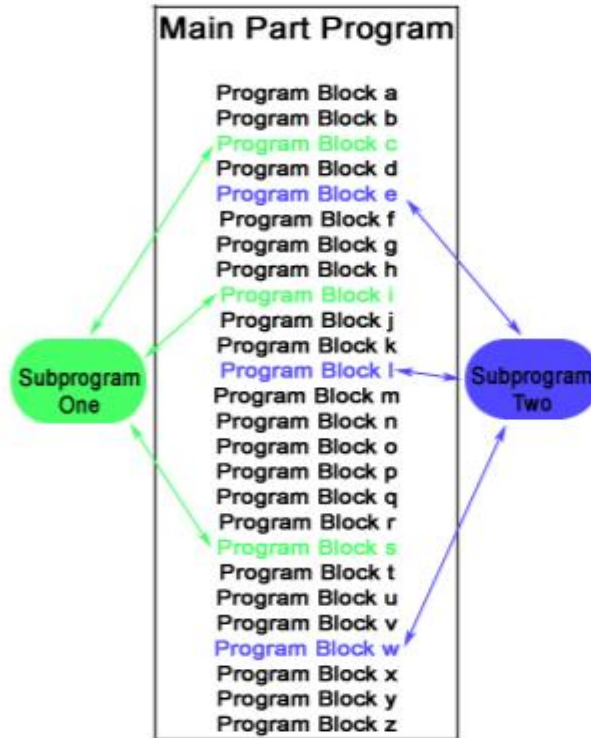


Figure 2. Subprograms allow programmers to call on repeated operations whenever they are needed.

CAD/CAM Part Programming

CAD/CAM part programming, shown in Figure 1, involves interpreting the part’s geometry and preparing the toolpaths directly from the data prepared in CAD. In this method, the programmer does not have to manually encode the tool motion and part geometry, which are both prone to error. Figure 2 shows a portion of a part program.

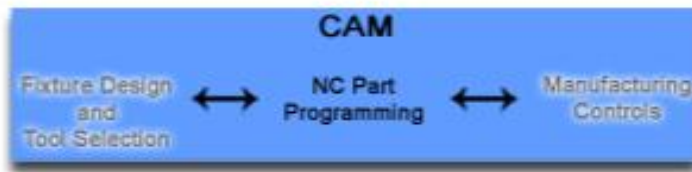


Figure 1. CAD/CAM part programming prepares the toolpath directly from CAD data.

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```

N200(Drill)
G97S750M03
G99T0202M08
G00X0.0Z.25
G83Q.15R.25Z-.85F.003
G00X8.0Z8.0
M01

```

Drill

```

N300(Bore)
G50S3500
G96S600M03
G99T0404M08
G00X.48Z.1
G71P310Q320U0.0W0.0D.05F.006
N310G00X1.0Z6
G01X.75Z-.038
Z-.25
X.681
X.625Z-.278
Z-.375
X.556
N320X.48Z-.413
G00X8.0Z8.0
M01

```

Bore

```

N400(Fin Turn)
G50S3500
G96S700M03
G99T0303M08
G00X1.75Z.1
G70P110Q120F.008
G00X8.0Z8.0
M01

```

**Finish
Turn**

```

N30
Z

```

Figure 2. Writing part programs without the aid of a CAD/CAM system is a process that is prone to error.

CAD/CAM programming also allows the user to visually verify the programmed motion of the tools along the workpiece. In other words, the CAD/CAM software displays a preview of how the machining sequences will appear on the actual machine. This interactive graphic display capability is a major advantage. The user may also edit the toolpath as required by adding new tool motions if needed.

One disadvantage of CAD/CAM part programming is that the programs produced tend to be very large. This may present some storage problems, especially if a machine control has limited storage capabilities.

Neutral Instructions and Post Processing

Post processing, which is shown in Figure 1, is an important link in the CAD/CAM chain that connects the CAM system and a CNC machine. The CAM system creates instructions for the CNC machine in a neutral language file called a cutter

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location file, which is then post processed. You can think of the post processor as translation software that converts neutral instructions from the CAM system into machine control data (MCD) for a specific machine.

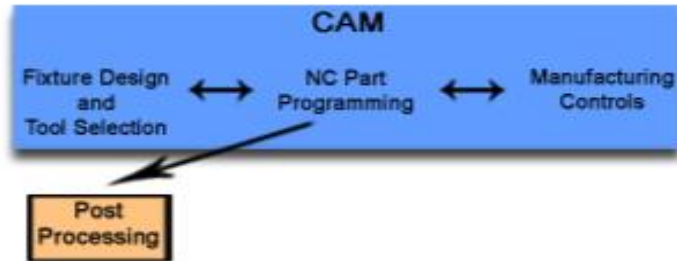


Figure 1. Post processing is a vital link in CAD/CAM.

An important link in the CAM process, this code file is responsible for generating specific machine control information, as shown in Figure 2. Typically, each CAD/CAM software package has its own post processor that converts the cutter location file into G codes and M codes for a specific machine/control combination. There is currently a movement in place that is trying to standardize the format of the cutter location file. This would enable the interchange of data between CAD/CAM systems and also make it possible for a cutter location file to be loaded directly into a machine control with a built-in post processor.

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Figure 2. Post processing connects CAM and the CNC machine.

Initial Graphics Exchange Specifications

As illustrated in Figure 1, one of the foremost concerns regarding CAD/CAM systems is the ability to communicate design and manufacturing information freely within an organization and between different organizations. Unfortunately, most of the early CAD/CAM systems were incompatible with each other. To overcome these difficulties, the industry has developed standards for information exchange. Although there are several standards for graphics and communications, this class will focus on data exchange standards.

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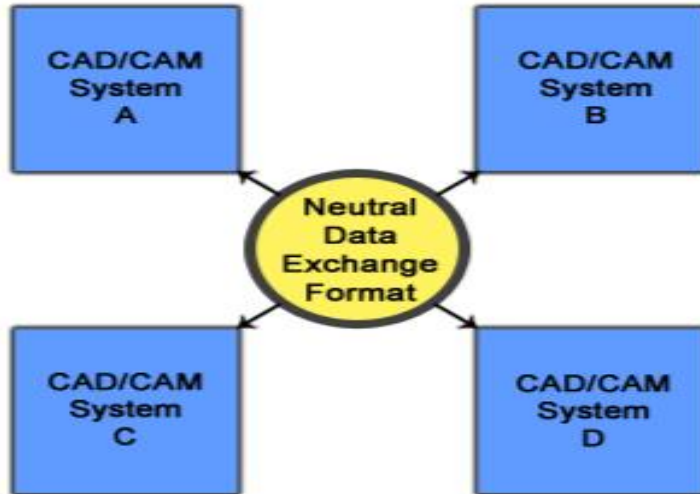


Figure 1. Neutral data exchange formats break down barriers between different CAD/CAM systems.

The first significant data exchange standard was developed in 1979 by major U.S. CAD vendors. This exchange standard, the Initial Graphics Exchange Specifications (IGES), is a neutral communications system that helps overcome differences in CAD systems. Using IGES, different CAD vendors translate data in two directions, in and out of their systems. Different CAD systems translate information into the neutral format and also translate information from neutral into their own system.

Drawing Exchange Format

The Drawing Exchange Format (DXF), illustrated in Figure 1, is a standard translation format for the interchange of CAD data between different software packages. It was developed by a company called Autodesk and was initially used on personal computers. DXF is a readable format that uses ASCII, the American Standard Code for Information Interchange. Most personal computer-based systems support DXF, which has become a standard for the interchange of data.

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Figure 1. DXF allows the interchange of CAD data between different software packages.

CAD/CAM companies support DXF so that the data created by other software companies can be compatible with their own systems. The major advantage with this arrangement is that data from one CAD/CAM system may be read on another CAD/CAM system by using DXF. Also, the standard allows other software companies to create programs that complement any CAD system that uses DXF.

Important Manufacturing Considerations

Figure 1 shows the last stage in the CAM process, which is the addition of manufacturing controls. Some of the newest CAD/CAM systems have additional capabilities that aid in manufacturing beyond the creation of its toolpaths. Process planning capabilities help create route sheets for the manufacturing process. For example, CAD/CAM software may help group parts with similar features into families of parts. The software determines the essential features of a particular part family. This is called variant process planning. By grouping similar parts together, the software helps a company find the most efficient way to manufacture these parts.

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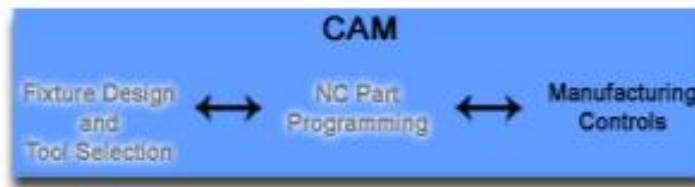


Figure 1. Some CAD/CAM systems can aid manufacturing beyond the part creation stage.

Some CAD/CAM systems may also gauge several other manufacturing variables, including forecasting demand for the part, planning needed capacity or material needs, balancing production lines, and controlling quality. CAM allows part planners to simulate production in order to determine its feasibility and effectiveness. All of these software tools move beyond the creation of the part program and help a company to consider a wide range of factors that improve production.

Summary

Computers have greatly improved the efficiency and productivity of part design and manufacturing. CAD/CAM systems decrease error and time commitments and improve the ways manufacturers store and manage information crucial to making parts.

The CAD/CAM part creation process begins with the design of a part with CAD. Geometric modeling is the process used to describe the shape of a part. The necessary machining details, such as the toolpath, are defined using CAM.

After part geometry, toolpaths, and general machine instructions have been determined, a cutter location file is processed. A post processor extracts machine control data and communicates with the CNC machine. Finally, the part is made by the CNC machine.

There are several standards such as IGES, DXF, and STEP that enable the transfer of information between manufacturing entities. IGES was the first data exchange standard. DXF is a personal computer-based standard for data exchange. Finally, STEP is the most developed standard with an international reach.

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CAD/CAM allows valuable data to inform part creation at any stage in the process. CAD/CAM has proven that computers can and will continue to make design and manufacturing more productive and cost-effective into the future.

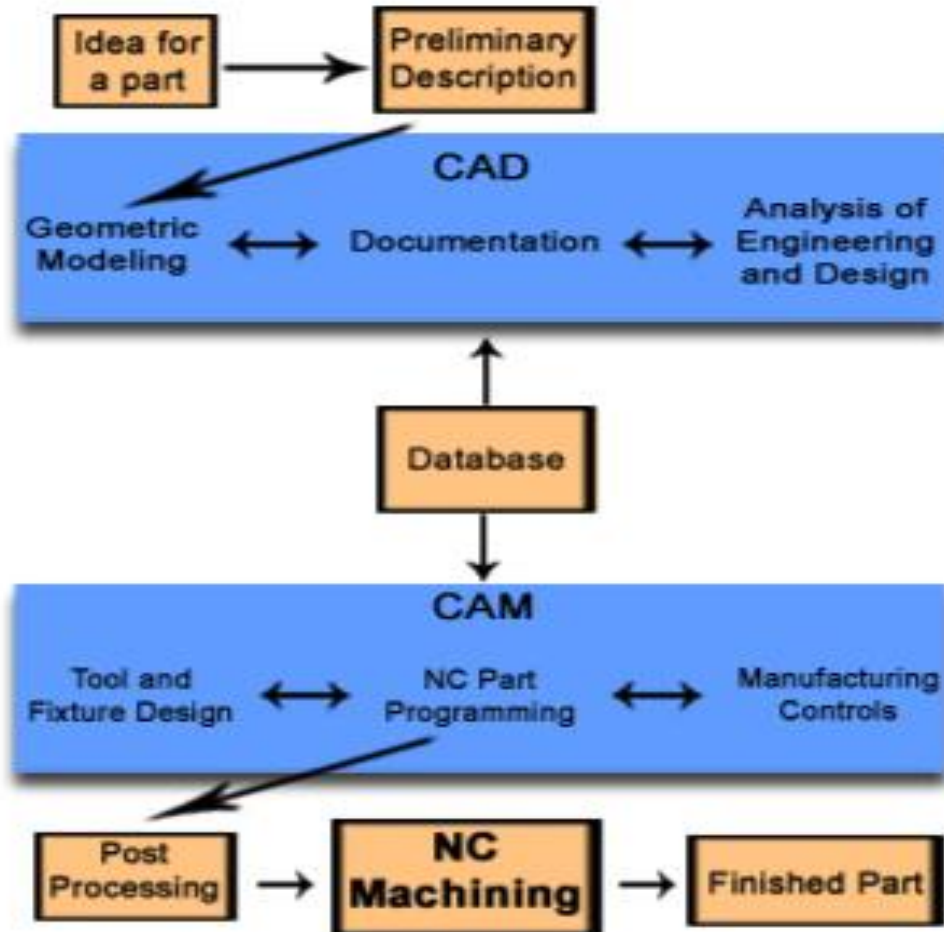


Figure 1. The complete CAD/CAM part creation process.

Mastercam

Mastercam is a three-dimensional geometry creation engine along with features to aid in tool path generation and verification.

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MasterCAM allows tool path planning and NC code generation for a given part. This part can either be drawn in MasterCAM or imported from other CAD packages.

MasterCAM drawing

Geometrical part drawing

- In-built CAD package

Two-dimensional parts

Three-dimensional parts

- Translators (include)

IGES (international Graphics Exchange Standard)

DXF (AutoCad)

CADL (CADKey)

Tool Path Generation using MasterCAM

- Tool path generation

Extensive Tool library

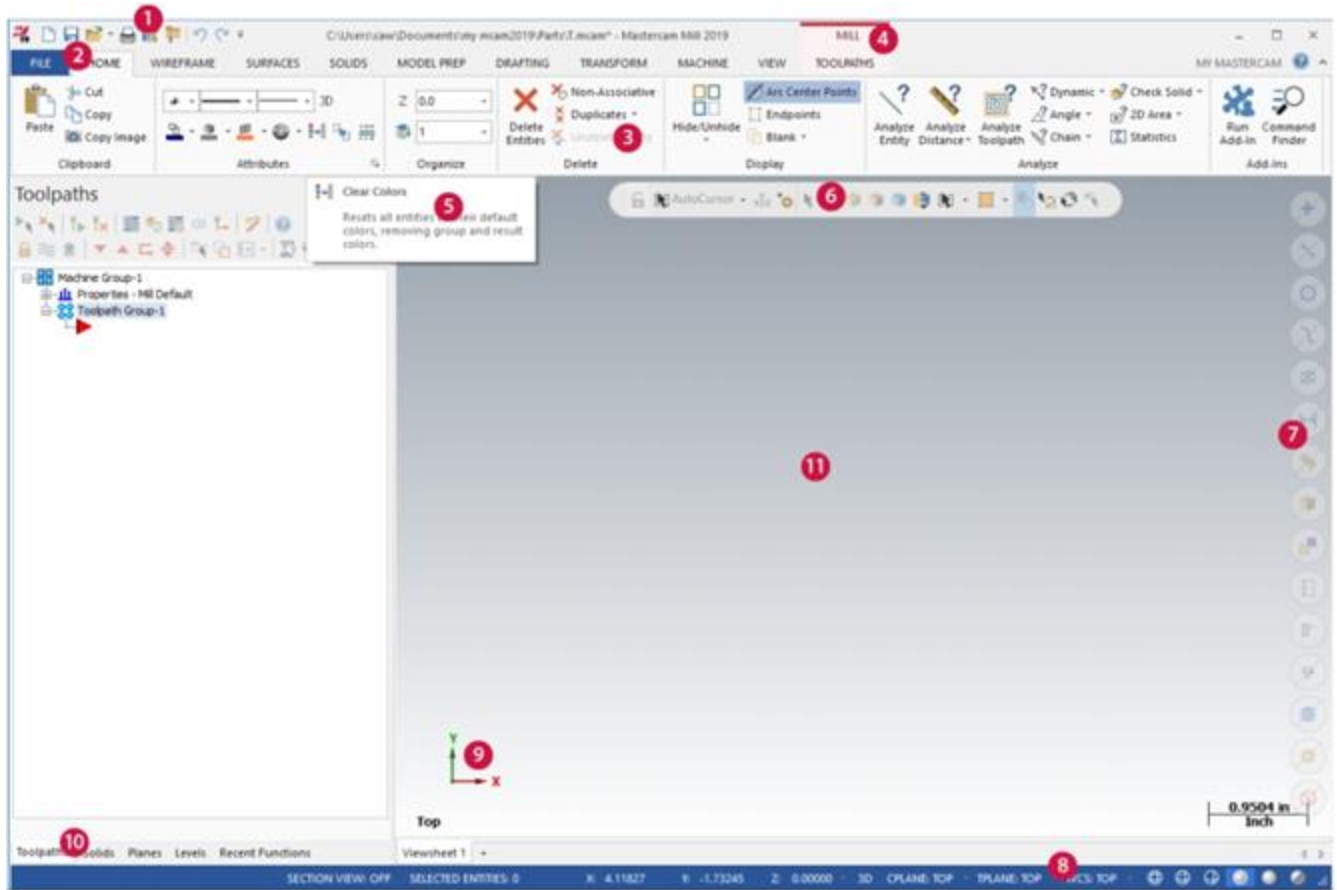
Machining parameter selection

NC program generator

Animation to visualize machining operations

Mastercam interface overview

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1. **Quick Access Toolbar (QAT):** A customizable set of commonly used functions that are always available in the interface. The QAT can be located above or below the ribbon.
2. **Tab:** Groupings of related controls. Tabs are organized in a simple-to-complex workflow from left to right.
3. **Tab Group:** A region of the tab that contains a set of related controls.
4. **Contextual Tab:** A tab that displays when you make a specific selection in Mastercam. A contextual tab presents relevant controls and commands to your current activity.
5. **Tooltip:** A small window with descriptive text that displays when you hover over a command or control.
6. **Selection Bar:** A toolbar that combines Auto Cursor controls and general selection tools used to select entities in the graphics window. There are two

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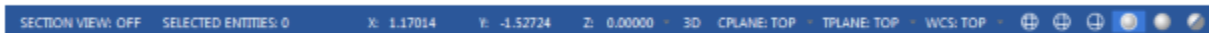
selection modes, Standard Selection and Solid Selection, which are activated based on the function you are using. Auto Cursor controls allow you to detect and snap to locations as you move the cursor over geometry in the graphics window. Auto Cursor becomes active whenever Mastercam prompts you to select a position in the graphics window.



7. **Quick Masks:** A group of controls that helps you to select all entities of a certain type, or to select only entities of certain type. Most Quick Mask controls are divided in half. Clicking the left or right side of the control toggles between selection modes. When a Quick Mask control is selected it highlights to indicate the selection mask is on. You can toggle multiple Quick Masks at a time.

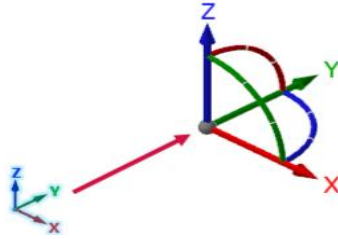


8. **Status Bar:** A bar across the bottom of the workspace that shows the coordinate position of the cursor and provides quick access to modify the Cplane, Tplane, WCS, and Z Depth of geometry and toolpaths in the graphics window. The right side of the Status Bar has wireframe, shading, and trans- lucency controls that change the appearance of your part.



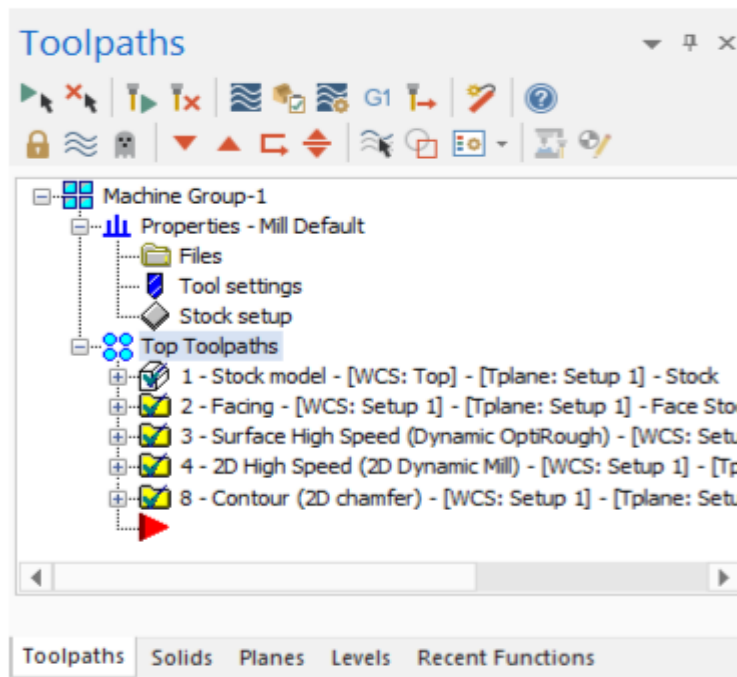
9. **Dynamic Gnomon:** The on-screen gnomon allows you to manipulate geometry and planes inter- actively. The gnomon comprises three axes connected at the origin, with selection points that let you choose different types of transformations.

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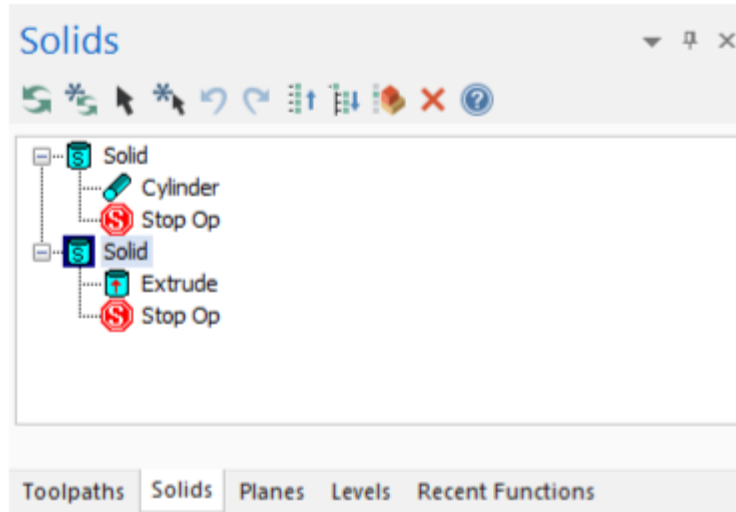
10. **Managers:** Managers include controls for toolpaths, solids, planes, levels, recent functions, and Art. The Man-agers can be stacked, docked, floating, or hidden.

- ❖ **Toolpaths Manager:** Lists the toolpath groups and machine types for the current file. Use the Toolpaths Manager to generate, sort, edit, regenerate, and post operations.

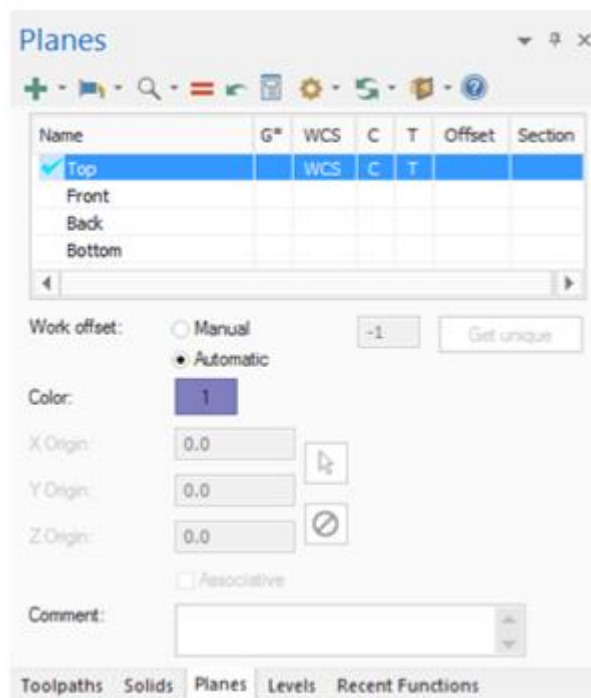


- ❖ **Solids Manager:** Lists each solid in the current file along with its operation history and associated toolpaths. Use the Solids Manager to edit solids and their operations.

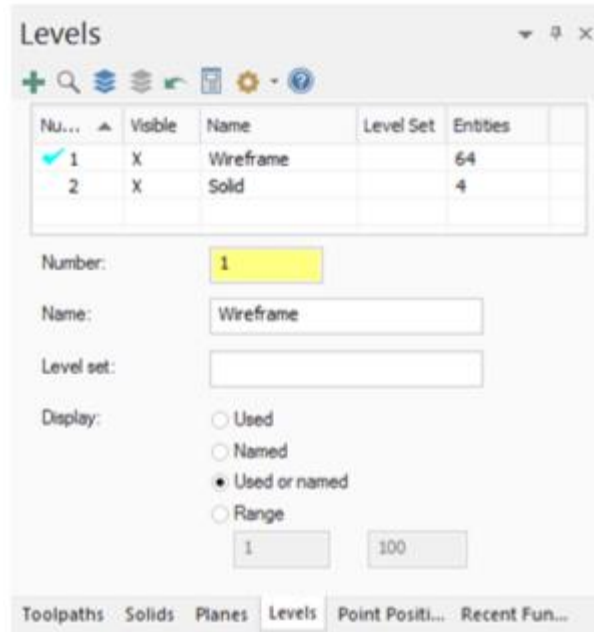
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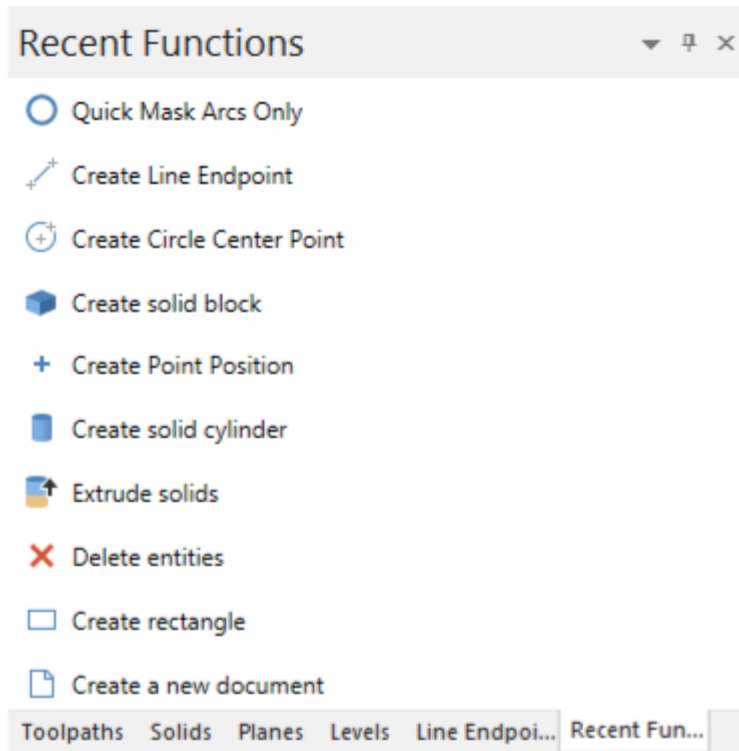
- ❖ **Planes Manager:** Shows the planes in the current file. Use the Planes Manager to display, create, edit, select, and manage planes and work offsets.



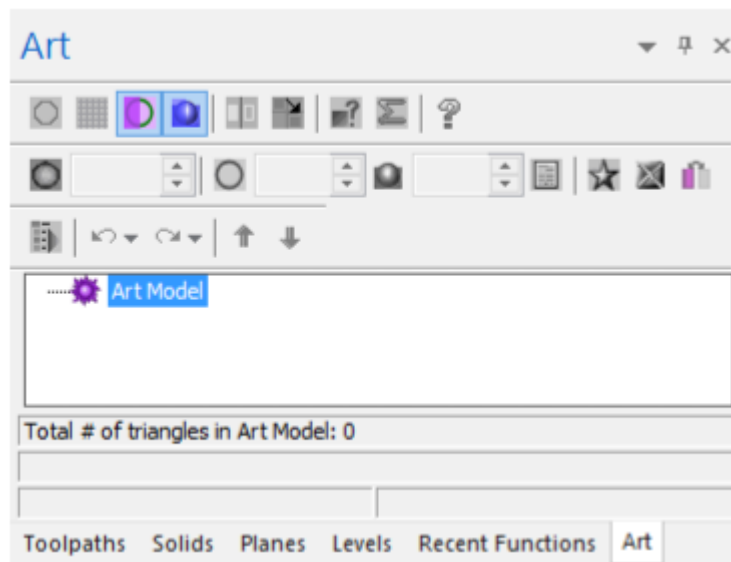
- ❖ **Levels Manager:** Shows the levels in the current file. Use the Levels Manager to create, select, hide or show levels, and set the active level.



- ❖ **The Recent Functions Manager:** Provides quick access to recently used functions.



- ❖ **Art Manager:** Manages the elements of the current Art model. (Only available if Mastercam Art is installed.)




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11. **Graphics window:** The workspace in which you view, create, and modify your parts. The graphics window also displays information about the current measurement system (inches or millimeters), and the coordinate axes for the current view or plane.

Common Interface Controls

The following table shows common controls used in the Mastercam interface. The icons may vary slightly depending on their location in Mastercam.

	OK and Create New		Cancel
	OK/Save		Help
	Apply		About Mastercam

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