

RESEARCH ARTICLE



ISSN: 2321-7758

MODELLING, ANALYSIS AND MANUFACTURING OF AIR FOIL SECTION

K.P.PREETHI MANOHARI SAI¹, Dr.P. SREENIVAS RAO², M.GANESH³

¹B.Tech Student, Aeronautical Department, MLR Institute of Technology, Dundigal, Hyderabad.

²(Development Officer) Tech At NSIC (National Small Industries Corporation Ltd.), Hyderabad.

³Associate Professor, Aeronautical Department, MLR Institute of Technology, Dundigal, Hyderabad



ABSTRACT

The present work aims Modeling, Analysis and Manufacturing of Airfoil Section (Aero plane wing) model by Using CAD/CAM software, Analysis by ANSYS Software and Manufacturing by CNC vertical machining Centre. CNC& CAD/CAM technologies are being used in almost all the industries in manufacturing sector. In Order to improve the productivity at the same time with high rate accuracy of machining. The modeling of model is to be done by using the part programme with G -Codes, M-Codes and the part programme generates by the cam module. The machining of model is carried out in CNC vertical machining Centre. A part from the live model, we also performs simple model analysis of a wing of the airplane and finding the natural frequencies and mode shapes. The wing will have a uniform cross section along its length. The wing fixed to the body of the aero plane at one end and hangs freely at the other end. The advantages of this project, comparing with conventional machines, found that, it is more flexible, high accuracy, good finish and also Complex shape. We can also read the results of analysis like deformed shapes, plotting the mode shapes, stresses, etc.

Keywords: Airfoil, NACA 4412, CAD/CAM Software, CNC Machine etc.

©KY PUBLICATIONS

1. INTRODUCTION

Production and productivity indicates the development of nation. Quantities of production of general-purpose items, automobiles, machines tools, etc., are increasing. The quality of these items is also need to improve. The requirements of high production and also quality of production is coupled with repeatability of accuracy and flexibility in machining operations are the prime factors responsible for improving productivity for the continuous development in the field of machining.

The special purpose machine tools are developed for components required in mass production. Numerically controlled machines tools are used for small batch productions in view of the flexibility built into the system.

With the development of computers and microprocessors the logic control function are achieved through computers .such systems are called "Computerized numerical control system (CNC)".some of the advanced of CNC machines are: High repeatability and

accuracy of work, reduction in rejection and also we can machine complicated jobs not normally possible on conventional machines, product changeover is also easy, elimination of skilled operators etc., is possible. But high knowledge persons with CNC programming are required.

CAD/CAM software packages originally, referred to those tasks dealing only with the initial phase of design and begun to mean computer based tools which are used to do everything a design engineer does. That includes the conceptual phase through detailed analysis and simulation. It even does into the interface with in manufacturing and documentation.

CAD/CAM technologies are very much important in the present now days. Almost all industries have to adopt these technologies and employment for young engineers in the field is very much progressive. Hence we selected our project in modelling and CNC programming, Machining.

2. LITERATURE SURVEY

About CAD/CAM

The term CAD/CAM is a shortening of computer-Aided design (CAD) and Computer-Aided manufacturing (CAM). It is the technology concerned with the use of digital computer to perform certain functions in design and production. This technology is moving in the direction of greater integration of design and manufacturing. CAD/CAM will provide the technology base for the computer-integrated factory of the future. Computer-Aided design (CAD) can be defined as the use of computer systems to assist in the creation, modification, analysis or optimization of a design. The CAD software consists of the computer programs to implement computer graphics on the system plus applications programs to facilitate the engineering functions of the user company. CAD/CAM software uses CAD drawing tools to describe geometries used by the CAM portion of the program to define a tool path that will direct the motion of a machine tool to machine the exact shape that was drawn.

CAM computer-aided manufacturing can be defined as the use of computer systems to plan, manage, and control the operations of a manufacturing plan through either direct to indirect computer interface with the plant's production resources. As indicated by the definition, the applications of a computer-aided manufacturing fall into two broad categories.

1. **Computer monitoring and control.** These are the direct applications in which the computer is connected directly to the manufacturing process for the purpose of monitoring or controlling the process.
2. **Manufacturing support applications** these are the indirect applications in which the computer is used in support of the production operations in the plant, but there is no direct interface between the computer and the manufacturing process.

3. HISTORY OF CATIA

- CATIA (Computer Aided Three-Dimensional Interactive Application) started as an in-house development in 1977 by French aircraft manufacturer Avions Marcel Dassault, at that time customer of the CAD/CAM CAD software to develop Dassault's Mirage fighter jet, then was adopted in the aerospace, automotive, shipbuilding, and other industries.
- Initially named CATI (*Conception Assistée Tridimensionnelle Interactive* — French for *Interactive Aided Three-dimensional Design*), it was renamed CATIA in 1981 when Dassault created a subsidiary to develop and sell the software and signed a non-exclusive distribution agreement with IBM. In 1984, the Boeing

Company had chosen CATIA V3 as its main 3D CAD tool, becoming its largest customer.

- In 1988, CATIA V3 was ported from mainframe computers to UNIX.
- In 1990, General Dynamics Electric Boat Corp chose CATIA as its main 3D CAD tool to design the U.S. Navy's Virginia class submarine. Also, Boeing had been selling its CADAM CAD system worldwide through the channel of IBM since 1978.
- In 1992, CADAM had been purchased from IBM, and the next year CATIA CADAM V4 was published.
- In 1996, it was ported from one to four Unix operating systems, including IBM AIX, Silicon Graphics IRIX, Sun Microsystems SunOS, and Hewlett-Packard HP-UX.
- In 1998, V5 was released and was an entirely rewritten version of CATIA with support for UNIX, Windows NT and Windows XP since 2001.
- In 2008, Dassault announced and released CATIA V6. While the server can run on Microsoft Windows, Linux or AIX, client support for any operating system other than Microsoft Windows was dropped.
- In November 2010, Dassault launched Catia V6R2011x, the latest release of its PLM2.0 platform, while still continuing to support and improve its Catia V5 software.
- In June 2011, Dassault launched V6 R2012.

4. SCOPE OF APPLICATION

Commonly referred to as a 3D Product Lifecycle Management software suite, CATIA supports multiple stages of product development (CAx), including conceptualization, design (CAD), manufacturing (CAM), and engineering (CAE). CATIA facilitates collaborative engineering across disciplines, including surfacing & shape design, mechanical engineering, and equipment and systems engineering.

CATIA provides a suite of surfacing, reverse engineering, and visualization solutions to create, modify, and validate complex innovative shapes, from subdivision, styling, and Class A surfaces to mechanical functional surfaces.

CATIA enables the creation of 3D parts, from 3D sketches, sheet metal, composites, and moulded, forged or tooling parts up to the definition of mechanical assemblies. It provides tools to complete product definition, including functional tolerances as well as kinematics definition.

CATIA facilitates the design of electronic, electrical, and distributed systems such as fluid

and HVAC systems, all the way to the production of documentation for manufacturing.

Systems engineering

CATIA offers a solution to model complex and intelligent products through the systems engineering approach. It covers the requirements definition, the systems architecture, the behaviour modelling and the virtual product or embedded software generation. CATIA can be customized via application programming interfaces (API). CATIA V5 & V6 can be adapted using Visual Basic and C++ programming languages via CAA (Component Application Architecture), a component object model (COM)-like interface.

HIGHLIGHTS AND BENEFITS OF CATIA

CATIA is the leading product development solution for all manufacturing organizations, from OEMs, through their supply chains, to small independent producers. The range of CATIA capabilities allows it to be applied in a wide variety of industries, such as aerospace, automotive, industrial machinery, electrical, electronics, shipbuilding, plant design, and consumer goods, including design for such diverse products as jewellery and clothing.

CATIA is the only solution capable of addressing the complete product development process, from product concept specification through product-in-service, in a fully integrated and associative manner. Based on an open, scalable architecture, it facilitates true collaborative engineering across the multidisciplinary extended enterprise, including style and form design, mechanical design and equipment and systems engineering, managing digital mock-ups, machining, analysis, and simulation. By enabling enterprises to reuse product design knowledge and accelerate development cycles, CATIA helps companies to speed-up their responses to market needs.

In conjunction with ENOVIA and ENOVIA Smart Team for lifecycle management and decision support and DELMIA for manufacturing engineering, CATIA is a key component of collaborative PLM.

Much beyond pure CAD software packages, which provide geometry modelling features for design-centric companies, CATIA delivers the keys to PLM for process-centric companies:

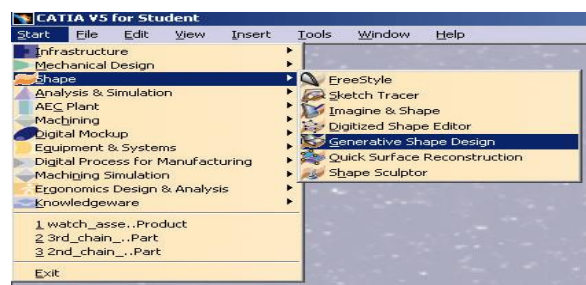
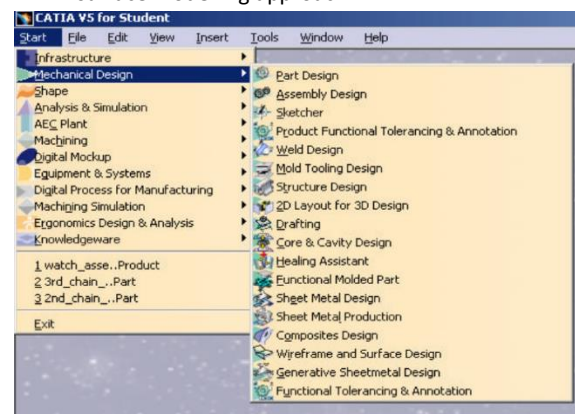
- **Product to market.** CATIA is about product creation. From the earliest product concept to production tooling, its concurrent engineering and design-in-context capabilities create value by enabling companies to create products and bring them to the market.

- **Time to market.** The unequalled process coverage of CATIA, combined with the native associativity among all of its applications, gives CATIA customers the means to shorten the time to market.
- **Right to market.** CATIA's integrated analysis, simulation, synthesis, and optimization applications provide product engineering validation at each design step to ensure product quality and market acceptance.
- **Lead the market.** CATIA's advanced capabilities for collaborative engineering, knowledge capture, and re-use boost innovation and help to lead the market.

5. CATIA CONCEPTS

The Workbench Concept: Each workbench contains a set of tools that is dedicated to perform a specific task. The following workbenches are the commonly used:

- **Part Design:** Design parts using a solid modelling approach
- **Sketcher:** Create 2D profiles with associated constraints, which is then used to create other 3D geometry.
- **Assembly Design:** Assemble parts together with constraints
- **Drafting:** Create drawings from parts or assemblies
- **Generative Shape Design:** Design parts using a surface modelling approach



User Interface:

Below is the layout of the elements of the standard CATIA application.

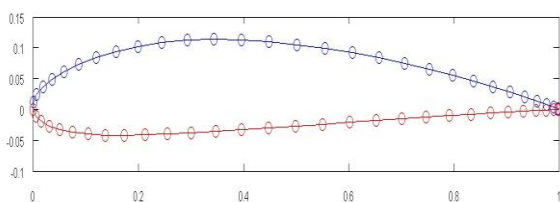
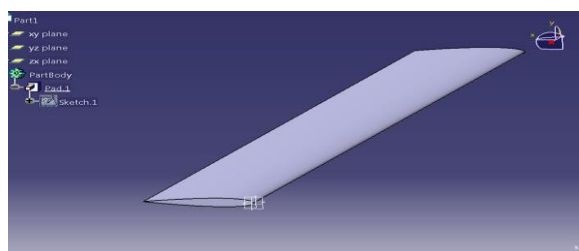
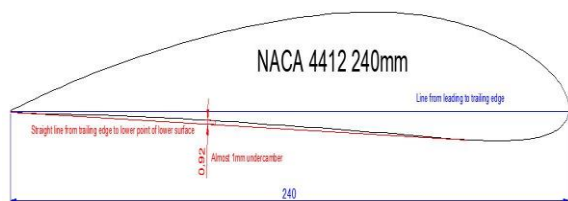
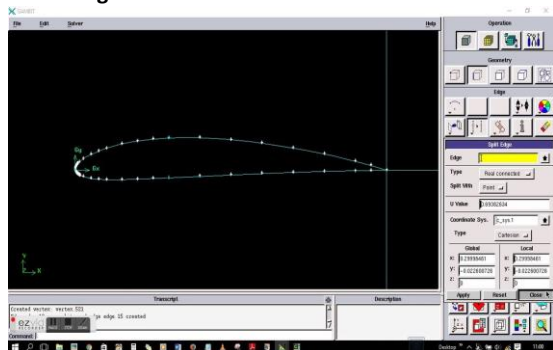
- A. Menu Commands
- B. Specification Tree
- C. Filename and extension of current document
- D. Icon of the active workbench
- E. Toolbars specific to the active workbench
- F. Standard Toolbar
- G. Compass
- H. Geometry area

Type of Documents:

The common documents are:

- A. A part document (.CATPart)
- B. An assembly document (.CATProduct)
- C. A drawing document (.CATDrawing)

Modelling of airfoil



Air foil coordinates

Start Loft	StartCurve		
0	1	0.0013	
0	0.95	0.0147	
0	0.9	0.0271	
0	0.8	0.0489	
0	0.7	0.0669	
0	0.6	0.0814	
0	0.5	0.0919	
0	0.4	0.098	
0	0.3	0.0976	
0	0.25	0.0941	
0	0.2	0.088	
0	0.15	0.0789	
0	0.1	0.0659	
0	0.075	0.0576	
0	0.05	0.0473	
0	0.025	0.0339	
0	0.0125	0.0244	
0	0	0	
0	0.0125	-0.0143	
0	0.025	-0.0195	
0	0.05	-0.0249	
0	0.075	-0.0274	
0	0.1	-0.0286	
0	0.15	-0.0288	
0	0.2	-0.0274	
0	0.25	-0.025	
0	0.3	-0.0226	
0	0.4	-0.018	
0	0.5	-0.014	
0	0.6	-0.01	
0	0.7	-0.0065	
0	0.8	-0.0039	
0	0.9	-0.0022	
0	0.95	-0.0016	
0	1	0.0013	

End Curve

End Loft

End

6. AIRFOIL

An airfoil-shaped body moved through a fluid produces an aerodynamic force. The component of this force perpendicular to the direction of motion is called lift.

The component parallel to the direction of motion is called drag Subsonic Flight airfoils have a characteristics shape with a rounded leading edge, followed by a sharp trailing edge, often with a symmetric curvature of upper and lower surfaces.

Foils of similar function designed with water as the working fluid are called hydrofoils.

The lift on an airfoil is primarily the result of its angle of attack and shape. When oriented at a suitable angle, the airfoil deflects the oncoming air, resulting in a force on the airfoil in the direction opposite to the deflection. This force is known as aerodynamic force and can be resolved into two components: lift and drag.

Most foil shapes require a positive angle of attack to generate lift, but cambered airfoils can generate lift at zero angle of attack. This "turning" of the air in the vicinity of the airfoil creates curved streamlines which results in lower pressure on one side and higher pressure on the other.

This pressure difference is accompanied by a velocity difference, via Bernoulli's principle, so the resulting flow field about the airfoil has a higher average velocity on the upper surface than on the lower surface. The lift force can be related directly to the average top/bottom velocity difference without computing the pressure by using the concept of circulation and the Kutta-Joukowski theorem.

A fixed-wing aircraft's wings, horizontal, and vertical stabilizers are built with airfoil-shaped cross sections, as are helicopter rotor blades. Airfoils are also found in propellers, fans, compressors and turbines. Sails are also airfoils, and the underwater surfaces of sailboats, such as the centre board and keel, are similar in cross-section and operate on the same principles as airfoils. Swimming and flying creatures and even many plants and sessile organisms employ airfoil/hydrofoils: common examples being bird wings, the bodies of fish, and the shape dollars. An airfoil-shaped wing can create down force on an automobile or other motor vehicle, improving traction.

Any object with an angle of attack in a moving fluid such as a flat plate, a building, or the deck of bridge, will generate an aerodynamic force (called lift) perpendicular to the flow. Airfoils are more efficient lifting shapes, able to generate more lift (up to a point). And to generate lift with less drag.

A lift and drag curve obtained in wind tunnel testing is shown on the right. The curve represents an airfoil with a positive camber so some

lift is produced at zero angle of attack. With increased angle of attack, lift increases in a roughly linear relation, called the slope of the lift curve. At about 18 degrees this airfoil stalls, and lift falls off quickly beyond that. The drop in lift can be explained by the action of the upper-surface boundary layer, which separates and greatly thickens over the upper surface at and past the stall angle. The thickened boundary layer's displacement thickness changes the airfoils effective shape, in particular it reduces its effective camber, which modifies the overall flow field so as to reduce the circulation and the lift. The thicker boundary layer also causes a large increase in pressure drag, so that the overall drag increases sharply near and past the stall point.

Airfoil design is a major facet of aerodynamics. Various airfoils serve different flight regimes. Asymmetric airfoils can generate lift at zero angle of attack, while a symmetric airfoil may better suit frequent inverted flight as in an aerobatic airplane. In the region of the ailerons and near a wingtip a symmetric airfoil can be used to increase the range of angles of attack to avoid spin-stall. Thus a large of angles can be used without boundary layer separation. Subsonic airfoils have a round leading edge. This is naturally insensitive to the angle of attack. The cross section is not strictly circular, however; the radius of curvature is increased before the wing achieves maximum thickness to minimize the chance of boundary layer separation. This elongates the wing and moves the point of maximum thickness back from the leading edge.

Supersonic airfoils are much more angular in shape and can have a very sharp leading edge, which is very sensitive to angle of attack. A supercritical airfoil has its maximum thickness close to the leading edge to have a lot of length to slowly shock the supersonic flow back to subsonic speeds. Generally such transonic airfoils and also the supersonic airfoils have a low camber to reduce drag divergence. Modern aircraft wings may have different airfoil sections along the wing span, each one optimized for the conditions in each section of the wing.

Movable high-lift devices, flaps and sometimes slats, are fitted to airfoils on almost

every aircraft. A trailing edge flap acts similarly to an aileron; however, it, as opposed to an aileron, can be retracted partially into the wing if not used.

A laminar flow wing has a maximum thickness in the middle camber line. Analyzing the Navier-stokes equations in the linear regime shows that a negative pressure gradient along the flow has the same effect as reducing the speed. So with the maximum camber in the middle, maintaining a laminar flow over a larger percentage of the wing at a higher cruising speed is possible. However, with rain or insects on the wing, or for jetliner speeds, this does not work. Since such a wing stalls more easily, this airfoil is not used on wingtips (spin-stall again)

Schemes have been devised to define airfoils- an example is the NACA system. Various airfoil generation systems are also used. An example of a general purpose airfoil that finds wide application, and predates the NACA system, is the Clark-Y. Today, airfoils can be designed for specific functions using inverse design programs such as PROFOIL, XFOIL and Aerofoil. XFOIL is an online program created by Mark Drela that will design and analyze subsonic isolated airfoils.

6.1 AIRFOIL TERMINOLOGY:

The various terms related to airfoils are defined below

- The suction surface (a.k.a. upper surface) is generally associated with higher velocity and low static pressure.
- The pressure surface (a.k.a lower surface) has a comparatively higher static pressure than the suction surface. The pressure gradient between these two surfaces contributes to the lift force generated for a given airfoil.

The geometry of the airfoil is described with a variety of terms:

- The leading edge is the point at the front of the airfoil that has maximum curvature (minimum radius).
- The trailing edge is defined similarly as the point of maximum curvature at the rear of the airfoil.
- The chord line is the straight line connecting leading and trailing edges. The chord length or simply chord is the length of the chord

line. That is the reference dimension of the airfoil section.

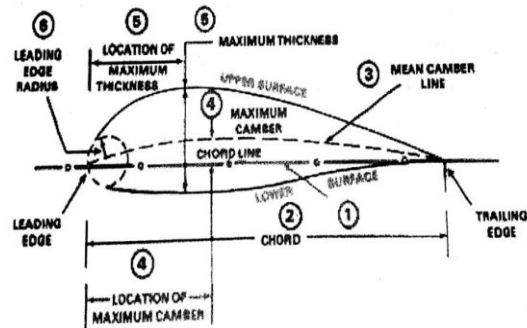


FIGURE 2-10. AIRFOIL TERMINOLOGY.

SHAPE OF THE AIRFOIL:

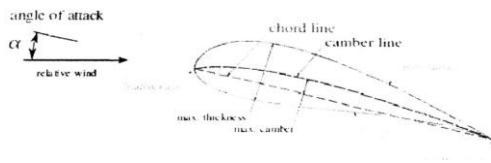
The shape of the airfoil is defined using the following geometrical parameters:

- The mean camber line or mean line is the locus of point's midway between the upper and lower surfaces. Its shape depends on the thickness distribution along the chord;
- The thickness of an airfoil varies along the chord. It may be measured in either of two ways.
- Thickness measured perpendicular to the camber line. This is sometimes described as the "American convention".
- Thickness measured perpendicular to the chord line. This is sometimes described as the "British convention".

Some important parameters to describe an airfoil's shape are its camber and its thickness. For example, an airfoil of the NACA 4-digit series such as the NACA 2415 describes an airfoil with a camber of 0.02 chords located at 0.40, with 0.15 chord of maximum thickness.

Finally, important concepts used to describe the airfoil's behaviour when moving through a fluid are:

- The aerodynamic centre, which is the chord-wise length about which the pitching moment, is independent of the lift coefficient and the angle of attack.
- The centre of pressure, which is the chord-wise location which the pitching moment is zero.

**AIRFOIL MOULD:****PART PROGRAMMING OF AN AIRFOIL:**

(MAIN PROGRAM):

O0001;

N1;

G00 G40 G80 G49;

G28 G91 Z0.0;

M06 T01;

G90 G54 X5.0 Y2.5;

G54 Z50.0;

M03 S2500;

G00 Z5.0;

M08;

G01 Z0.0 F50;

M98 P030 O0002;

G90 G00 Z50.0;

M05;

M09;

G28 G91 Z0.0;

G28 G91 Y0.0;

M30;

(SUB PROGRAM)

O0002;

G91 G01 Z-0.5 F10;

G90 G41 G01 X66.04 D10 F500;

G03 X75.569 Y5.9 R1 1.5;

G03 X76.51 Y7.266 R1.0;

G03 X62.9758 Y14.827 R18.0;

G03 X62.5120 Y14.80 R1.3;

G01 X33.2 Y5.0 Y2.5;

M99;

7. ANSYS

INTRODUCTION TO FEA: The finite Element Analysis is a computing technique that is used to obtain approximate solution to the boundary value problem in engineering. It uses a numerical technique called the finite element method to solve the boundary value problems. FEA involves a computer model of a design that is loaded and analysed for specific results. The finite element

analysis was first developed by 'RICHARD COURANT' in 1943. He used the Ritz method of numerical analysis and minimization of variation calculus for getting approximate solutions to vibration systems. Later, the academic and industrial researchers created the finite element method for structural analysis.

The concept of FEA can be explained with a small explain of measuring the perimeter of a circle, to measure the perimeter of a circle without using the conventional formula, divide the circle into equal segments. Next, join the start point and end point of each of the segments by straight line. Now, you can very easily measure the length of straight line, and thus, the perimeter of the circle.

If you divide the circle into four segments only, you will not get accurate results. For accuracy divide the circle into more number of segments. However, with more segments the effort required will be more. The same concept applies to FEA also, and therefore, there is always a compromise between accuracy speeds while using this method. This makes it an approximate method.

The FEA was first developed to be used in the aerospace and nuclear industries, where the safety of structures is critical. Nowadays, the simplest of the products rely on the FEA for design evaluation.

The FEA simulates the loading conditions of a design and determines the design response in those conditions. The design is modelled using the discrete building blocks called elements. Each element has some equations that describe how it responds to certain loads. The some of the response of all the elements in the model gives the total response of the design.

GENERAL WORKING OF FEA: Better knowledge of FEA will help you build more accurate models. It will also help you understand the backend working of ANSYS. The simplest model is discussed here to give you a brief over view of working of FEA.

1. Stiffness matrix
2. Degrees of Freedom
3. Boundary Conditions

1. STIFFNESS MATRIX: In the previous equation, the following part represents the stiffness matrix (k):

$$\begin{bmatrix} k_1 + k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix}$$

This matrix is relatively simple because it comprises only one pair of springs, but it turns complex when the number of springs increases,

2. DEGREE OF FREEDOM: Degree of freedom is defined as the ability of a node to translate or transmit the load. In the previous example, you are only concerned with the displacement and forces. By making one endpoint fixed, one degrees of freedom for displacement is removed from the three possible degree of freedom. So, now the model has two degrees of freedom. The number of degrees of freedom in a model determines the number of equations required to solve the mathematical model

3. BOUNDARY CONDITIONS: The boundary conditions are used to eliminate the unknown in the systems. A set of equations that is solvable is meaningless without the input. In the previous example, the boundary condition was $X_0=0$, and the input forces were F_1 & F_2 . In the either ways, the displacements could have been specified in place of forces as boundary conditions and the mathematics model could have been solved for the forces. In the other words, the boundary conditions help you reduce or eliminate unknown in the systems.

4. NODES, ELEMENTS AND ELEMENT SHAPES:
NODES: An independent entity in space is called a node. Node are similar to the points in geometry and represent the corner points of an element. The element shape can be changed by moving the nodes in space.

ELEMENTS: Element is an entity into which the system under study is divided. An element shapes is specified by nodes. The shape (area, length and volume) of an element depends on the nodes with which it is made.

ELEMENT SHAPES: The following are the basic shapes of the elements:

POINT ELEMENT: A point element is in the form of a point and therefore has only one node.

LINE ELEMENT: A line element has the shape of a line or curves, therefore a minimum of two nodes are required to define it. There can be higher order elements that have additional nodes. The element

that does not have a node at the middle of the edge of the element is called a linear element. The element with node at the mid of the edges is called quadratic or second order elements.

AREA ELEMENT: An area element has the shape of a quadrilateral or a triangle therefore it requires a minimum of three or four nodes to define it.

VOLUME ELEMENT: A volume element has the shape of a Hexahedron, wedge, tetrahedron or a pyramid.

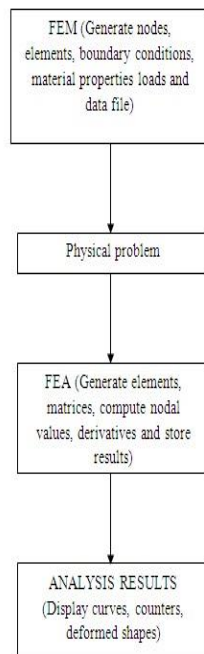
AREAS FOR APPLICATION OF FEA: FEA is a very important tool for designing. It is used in the following areas.

1. Structural strength design
2. Structural interaction with fluid flows
3. Analysis of shock
4. Acoustics
5. Thermal analysis
6. Vibrations
7. Crash simulations
8. Fluid flows
9. Electrical analysis
10. Mass diffusion
11. Buckling problems
12. Dynamic analysis
13. Electromagnetic analysis
14. Coupled analysis

General Procedure Of Conducting Finite Element Analysis:

1. Set the type of analysis to be used
2. Create the model
3. Define the element type
4. Divide the given problem into nodes and elements(mesh the model)
5. Apply material properties and boundary conditions
6. Derive the element matrices and equations
7. Assemble the element equations
8. Solve the unknown quantities at nodes
9. Interpret the results

FEA THROUGH ANSYS: In ANSYS, the general process of finite element analysis is divided into three main phases, pre-processor, solution and post processor



PREPROCESSOR: The pre-processor that process the input data to produce the output that is used as input to the subsequent phase (solution). Following are the input data that needs to be given to the pre-processor.

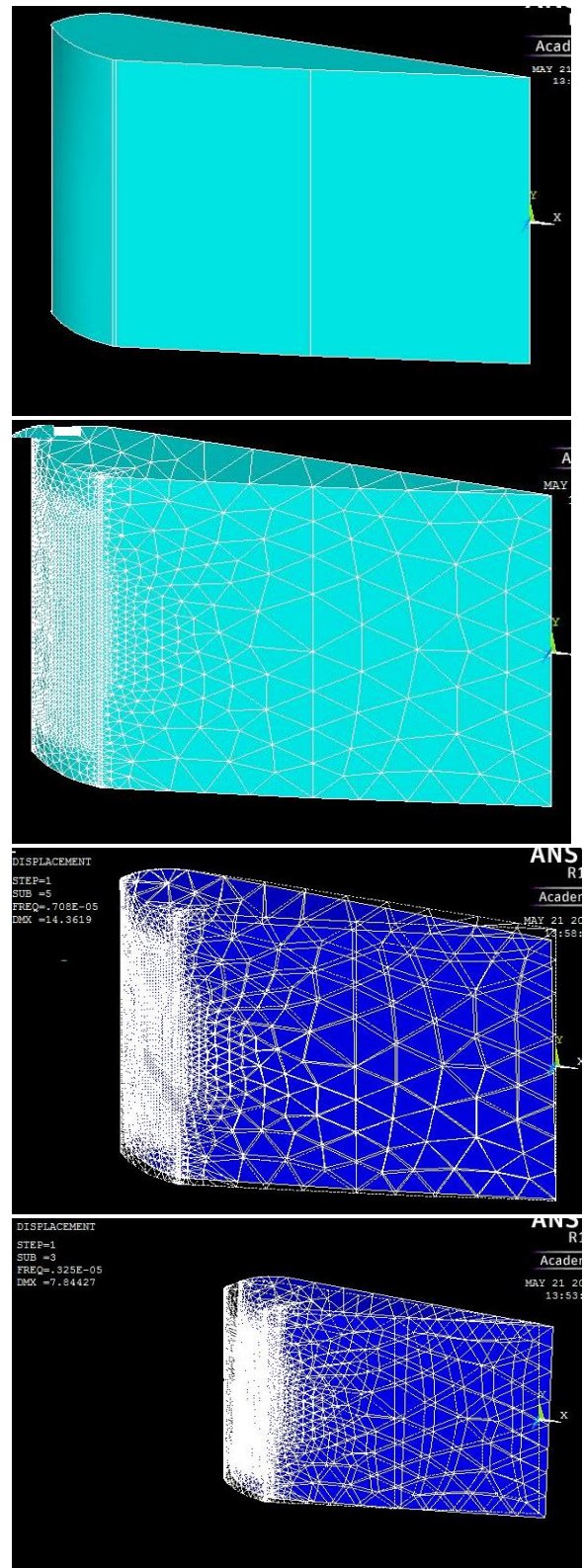
1. Type of analysis (structural or thermal, static or dynamic and linear or non-linear)
2. Element type
3. Real constants
4. Material properties
5. Geometric model
6. Meshed model
7. Loading and boundary conditions

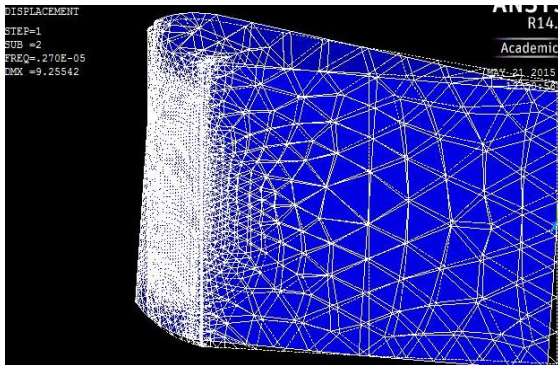
SOLUTIONS: Solution phase is completely automatic. The FEA software generates the element matrices, computes nodal values and derivatives and stores the results data in files. This files are further used by subsequent phase (postprocessor) to review and analyze the results through the graphic display and tabular listings.

POSTPROCESSOR: The output from the solution phase (result data files is in the numerical form and consists of nodal values of the field variable and its derivatives. For example, in structural analysis, the output is nodal displacement and stress in the elements. The postprocessor process the result data and displace them in the graphical form to check of analyze the result. The graphical output gives the detailed information about the required data. The

postprocessor phase is automatic and generates the graphical output in the form specified by the user.

ANALYSIS OF AN AIRFOIL USING ANSYS





8. NUMERICAL CONTROL MACHINES (NC)

Numerical control is a form of programmable automation in which the mechanical actions of a machine tool or other equipment are controlled by a programme containing coded alphanumeric data. The alphanumeric data represent relative positions between a work head part as well as other instructions needed to operate the machine.

Numerical Control can be applied to a wide variety of processes. The applications divide into two categories.

- 1) Machine tool applications, such as drilling, milling, turning and other metal working.
- 2) Machine tool applications, such as assembly, drafting and inspection. The common operating feature of NC in all these applications is control of the work head movement relative to the work part.

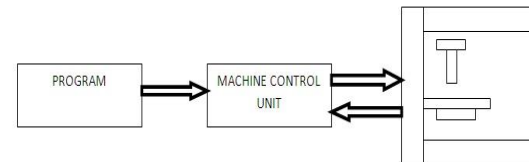
Basic Component of NC system: An NC system consists of three basic components.

- 1) A program of instructions.
- 2) A machine control unit and
- 3) Processing equipment. The general relationship among the three components is illustrated in figure.

The program if instructions are the detailed step-by step commands that direct the actions of the processing equipment. In machine tool applications, the program is called a part programmer.

In modern NC technology, the machine control units (MCU) consists of a microcomputer and related control hardware that stores the program of instructions and executions it by converting each command into mechanical actions of the processing equipment.

The third basic component of an NC system is the processing equipment that performs useful work. Its operation is directed by the MCU, which in turn is driven by instructions contained in the part program.



PROCESSING EQUIPMENT

Basic component of an NC system

FEATURES OF CNC: Computer Numerical Control systems include additional features beyond what is feasible with conventional; hard-wire NC. Many of features are standard on most of the CNC machines and some are optional. Features of the CNC systems are:

- Storage of more than one part program.
- Various forms of program input
- Program editing at the machine tool
- Fixed cycles and programming sub routines
- Interpolation
- Positioning features for setup
- Cutter length and size compensation
- Acceleration and deceleration calculations
- Communication interface
- Diagnostics.

CLASSIFICATION OF CNC

The CNC machines can be classified on many basis, some of them are:-

Open loop system & closed loop system: The basis of this classification is one the feedback operation. In open loop system there is not feedback mechanism, i.e., the information or data cannot be transmitted connect in the circuit. This feedback device transfers the data processed by the sensors (speed sensors, displacement sensors, etc) back to the machine control unit. The MCU then compares the feedback values with the original values and gives the appropriate orders to that particular device, so that required value is obtained.

On the basis of tool movement: Point to point CNC (PTP): PTP is also sometimes called positioning system. In PTP the objective of the machine tool

control system is to move the cutting tool to a predetermined location. The speed or path by which this movement is accomplished is not important in point to point system. Once the tool reaches the desired location, the machining operation is performed at that location.

a) **Straight cut CNC:** Straight cut control systems are capable of moving the cutting parallel to one of the major axis at a controlled rate suitable for machining. It is therefore appropriate for performing milling operation to fabricate rectangular work pieces. With this type of NC system is not possible to combine movements in more than a single axis direction. Therefore angular cut on work piece would not be possible a CNC machine capable of straight cut movement is also capable of PTP movements.

b) **Contouring CNC:** contouring is the most complex and most flexible and the most expensive type of machine tool control. It is capable of performing both PTP and straight cut operation. In addition, the distant feature of contouring CNC system is their capacity to control simultaneously more than one axis movement of the machine tool. The path of the cutter is continuously controlled to generate desired geometry of the work piece. For this reason, contouring, contouring systems are also called continuous path CNC system.

On the basis of number of axis: The CNC machines classified under number of axes are:

Two axes (2) CNC- in two axis CNC machine any two of the three axis (X, Y and Z) are present. i.e. XY or YZ or ZX are only present. That means the tool or machine base can move only in two directions.

Two and half axes (2 ½) CNC – in 2 ½ axis CNC machine though there are three axis (X Y Z) the machine tool or the machine base can be moved only in any two directions simultaneously. For example - if it has to move in all three directions the tool is moved in X and Y direction first simultaneously and then in Z direction.

Three axes (3) CNC – In three axis CNC contrary to 2 ½ CNC the machine tool or the machine base can be moved in the entire three axes simultaneously.

- Three and half CNC (3 ½) (X Y Z a)
- Four axis CNC (4) (X Y Z a)
- Five axis CNC (5) (X Y Z a b)

- Six axis CNC (6) (X Y Z a b c)
- *a, b, c are the angular rotation about x, y and z axes respectively.

COORDINATE SYSTEM IN CNC MACHINES

In order for the part programmer to plan the sequence of positions and movements of the cutting tool relative to the work piece, it is necessary to establish a standard axis system by which the relative position can be specified.

Two axis X and Y are defined in the plane of the table. The Z axis is perpendicular to this plane and movement in Z direction is controlled by the vertical motion of the spindle.

FIXED ZERO AND FLOATING ZERO: The programmer must determine the position of the tool relative to the origin of the coordinate system. CNC machines have either of two methods for specifying the zero point. The first possibility is for the machine to have a fixed zero. In this case, the origin is always located at same position on the machine table. Usually, that position is in the south west corner of the table and all tool location will be defined positive X and Y Coordinates.

The second and more common feature on modern CNC machines allows the machine operator to set the zero point at any position on the machine table. This feature is called floating zero. The part programmer is the one who decides where the zero point should be located. The decision is based on part programming convenience.

9. **CNC SOFTWARE:** The computer in CNC machine operated by means of software. There are three types of software programmes used in CNC systems.

- 1) Operating system software.
- 2) Machine interface software and
- 3) Application software

The principle functions of the operating system software are to interpret the NC part program and generate the corresponding control signals to drive the machine tool axis. It is installed by the controller manufacturer and is stored in ROM in the MCU. The operating system software consists of the following:

- 1) An editor which permits the machine operator to input and edit NC part programs

and perform other file management functions.

- 2) A control program, which decodes the part program instructions, performs interpolation and acceleration/deceleration calculations and accomplishes other related functions to produce the co-ordinate control signals for each axis.
- 3) An executive program, which manages the execution of the CNC software as well as I/O operations of MCU.

The machine interface software is used to operate the communication link between the CPU and the machine tool to accomplish the CNBC auxiliary functions.

The application software consists of the NC part programs that are written for machining (or other) application in the user's plant.

CNC PART PROGRAMMING LANGUAGES

A part programming languages consists of software packages plus the special rules conventions and vocabulary words for using that software. Its purpose is to make it convenient for a part programmer to communicate the necessary part geometry and tool motion information to the computer so that the desired part program can be prepared. The vocabulary words are typically mnemonic and English like, to make the language easy to use.

They have probably been over 100 part programming languages developed. Most of the languages were developed to meet particular needs and have not survived the test of time. Here are some of the part programming languages. Ex-APT, ADAPT.

AUTOMATICALLY PROGRAMMED TOOLS (APT)

The APT language was the product of the MIT developmental work on CNC programming system. Its development began in June, 1956 and it was first used in production around 1959. Today it is most widely used language in United States. Although first intended as contouring languages, modern versions of APT can be used for positioning and continuous path programming in up to five axis. Version of APT for particular process include APTURN (for Lathe operation, APTMIL (for milling and drilling operations).

ADAPTATION OF APT (ADAPT)

Several part programming languages are based directly on the APT program. One of these is ADAPT, which was developed by IBM under AIR FORCE contract. It was intended to provide many of the features of APT but to utilize a smaller computer. ADAPT is not a powerful of APT, but it can be used to program for both positioning and contouring jobs.

CNC STANDARD CONTROLLERS

There is large number of CNC controller available today. A few of them are listed below:-

- 1) Aeramatic
- 2) AEG
- 3) Bendix
- 4) Bosch
- 5) Fanuc OM, OT series
- 6) GE 550, CNC 2000
- 7) HEIDENHAIN
- 8) MAZAK
- 9) MELDAS
- 10) SINUMERIC
- 11) PHILIPS
- 12) TOSHIBA

CNC SYSTEM AVAILABLE IN INDIA

FANUC 'O'	-	FANUC	INDIA,
BANGALORE HINUMERIC	-HMT,	BANGALORE	
ANILAM CRUSADER	-	KIRLOSKAR,	MYSORE
ALLEN-BRADLEY	-LASHMI	ELECTIC	
CONTROLS, COIMBATORE			
ELPRO	-ELPRO,	PUNE	
ELECTRAMERIC	-ELECTRONICA,	PUNE	

DIRECT NUMERICAL CONTROL (DNC)

Direct numerical control (DNC) link is high speed industrial local area network. A personal computer is used as the DNC computer. The NC program from the CAD/CAM workstation is transferred to the PC. All CAM software have a DNC link which serves as a means of transferring machining program directly from the PC to machine tool controller. This enables the user to handles large programs associated with 3D machining of complex shapes. Other benefits include time savings, increased reliability and less maintenance. DNC can take on a variety of physical configurations, depending on the number of machine tools included, job complexity security requirements and

equipment availability and preferences. There are several ways to configure a DNC system.

There are two types of DNC network: (1) switching networks and (2) LAN

- 1) **Switching Network:** the switching network is the simplest DNC system to configure. It uses a data switching box to make a connection from the central computer to a given CNC Machine for down loading part programs or uploading data.
- 2) **LAN :** in this network, the computer system is organized as a hierarchy, with the central (host) computer co-ordinating several satellite computers that re each responsible for a number of CNC machines

MACHINING CENTRE

The CNC machining centre will have Automatic Tool Changer (ATC) and Automatic Pallet Changer (APC).

AUTOMATIC TOOL CHANGER (ATC)

During the operation of a machine tool, considerable amount of time is spent in idle movement of tool such as tool engagement and disengagement, tool change and tool setup. To improve the machine utilization, it is necessary to minimize these idle motions. To that extent automatic tools changes plays a very important role. These particularly in machining applications where a number of tools are to be used for finishing the job.

AUTOMATIC ROOL CHANGER (APC)

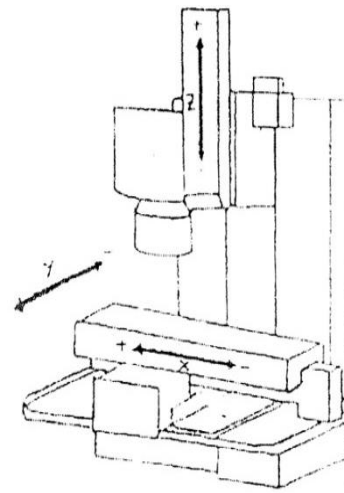
Generally the components machined on horizontal machining centre (HMC) or vertical Machining Centre (VMC) requires large cycle times. Further complex job used to these machines requires relatively complex setup times. Thus the actual settings up of the work pieces need to be done away from the machine tool such that the machine utilization is improved. For this purpose pallets are most commonly used with the machining centre pallet may be considered as a small table having standardized dimensions. These are normally available in standard sizes such as 500x500 mm, 630x630 mm etc. They come in generally two forms. One with precisions drilled holes, the other with T-slots.

CNC PART PROGRAMMING

The part program provides the instructions to the CNC control to execute the machine operation.

Coordinate System of the Machine

The information "move the longitudinal slide in the spindle head direction" is a very long one. Besides, in each language it would be different. That's why the traverse path movements in machine tools are described within the coordinate system



X – Axis: movement of table.

Y – Axis: movement of saddle.

Z – Axis; movement of spindle

+X movement: movement of the table to the left of the operator.

-X movement: movement of the table to the right of the operator.

-Y movement: movement of the table towards the operator.

-Y movement: movement of the table away from the operator.

-Z movement: movement of the spindle head up (away from the job)

-Z movement: movement of the spindle head down (towards the job)

PROGRAMMING METHOD

There are two methods to describe the path of motion in a CNC Machine

- Absolute Programming Method.
- Incremental Programming Method.

In Absolute Programming method one point on work piece is taken as reference, or zero point and all other dimensions are represented with respect to that point.

In incremental Programming Method the next point dimension in each block is referred with respect to PROGRAMMING METHODS COMPARISON

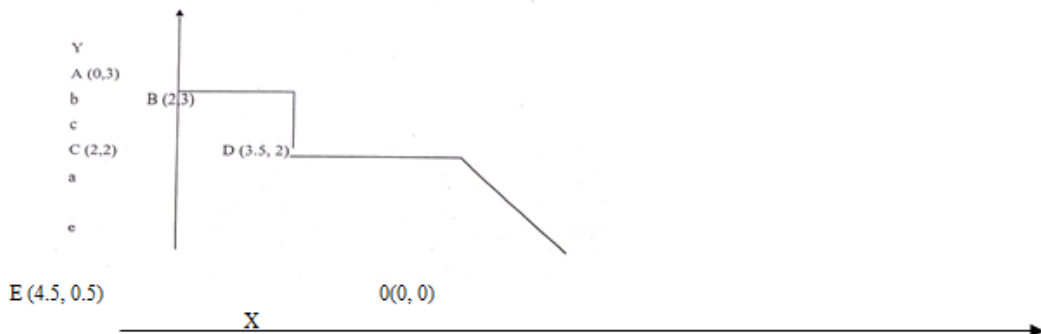
the final point on the previous block.

Absolute System

The path information given is always Calculated from a definite starting point.

Incremental System

the zero reference point for each Information is the current position Slides.



Path	X	Y
a	0	3
b	2	0
c	0	2

Point O is the reference point.

Path	X	Y
a	0	3
b	2	0
c	0	-1

Point O, A, B, C & D are new reference Points.

Program Configuration

MAJOR FUNCTIONS AND ADDRESS

Function Meanings	Address	Range
Programme number	O	1-9999
Sequence number	N	1-9999
Preparatory function	G	0-255
Specifies a motion mode (linear & arc)		
Dimension work	X, Y, Z	Coordinate
Axis of the arc centre		
Feed function	F	1-24000 mm/min
Rate of feed per minute		0.01-500 mm/rev
Spindle feed function	S	0-20000 rpm
Speed		
Tool function	T	0-20 Tool
Number		
Auxiliary/Miscellaneous Function	M	On/Off Control On
The machine tool		
Offset number	D, H	
Offset number		
Dwell	P, X	0-9999.999s
Dwell time		
Number of repetitions	P	1-999
Number of sub-programme Repetitions		

DEFINITIONS:

O: "O" followed by digits (O1702) indicates the program number or identity

N: N indicates the sequence number of operations.

G: preparatory commands "G" codes are used to initiate control commands, positioning of contouring modes of operation and machining cycles.

G codes are divided into two types:

- Modal G codes – These remain in the program until cancelled by a contradictory code.

- Non modal G codes – These are active only in the block in which they are programmed.

X Command for X-Axis movement

Y Command for Y-Axis movement

Z Command for Z- axis movement

The dimensions of these words specify the direction and the distance which is to be traversed.

Multiple values for the same axis in a block are not accepted.

I : Incremental vector @ X-Axis

J : Incremental vector @Y-Axis

K : Incremental vector @ Z- Axis

These are the paraxial coordinates of the distance vector from the starting position to the centre point of the circle/arc.

The values of the I, J & k must always be entered as incremental position data irrespective of whether the values of X , Y & Z are entered in absolute or incremental coordinates. The sign depends on the direction of the coordinates from the starting position to the centre point of the circle.

F: The feed function controls the feed rate of the tool. The feed rate is programmed in mm/min, inch/min, mm/rev, as determined by the G code active in that block.

S: The spindle speed can be controlled by specifying a value following the address S

T: By specifying two digit numerical value following the address T, tools can be selected on the machine.

M: Miscellaneous function contains primarily technological specifications. Usually only one M code can be specified in one block. In some cases however, up to three M codes can be specified.

D: Selection of tool CRC number

H: Selection of tool length offset number

P, X: By specification a dwell, the execution of the next block is delayed by the specified time in seconds

When X is used, the dwell is: second(s), and when P is used, the dwell is: mill seconds (0.001)

P: number of repetition of subprogram.

COMMONLY USED G-CODES

*G00 - Rapid positioning.
*G01 - Linear interpolation
*G02 - Circular interpolation clockwise
*G03 - Circular interpolation counter

clockwise

*G04 - Dwell

*G17 - Selection of XY plane

*G18 - Selection of ZX plane

*G19 - Selection of YZ plane

*G20 - Inch mode

*G21 - Metric mode

*G28 - Return to reference position

*G40 - Cancellation of cutter radius

compensation

*G41 - Cutter radius compensation left

*G42 - Cutter radius compensation right

*G43 - Tool length compensation

*G49 - Cancellation of tool length

compensation

*G54-G59 - Work piece coordinate system

(zero offset)

*G73 - High speed peck drilling cycle

*G74 - Left hand tapping cycle

*G80 - Cancellation of any canned cycle

*G81 - Drilling cycle

*G82 - Drilling cycle with dwell / Counter

boring cycle

*G83 - Peck drilling cycle

*G84 - Tapping cycle

*G85 - Boring cycle

*G86 - Boring cycle

*G87 - back boring cycle

*G90 - absolute command

*G91 - incremental command

*G92 - Work piece coordinate system

(Position Preset)

*G94 - Feed per minute

*G95 - Feed per Revolution

Commonly used M-Codes

M00 - Program stop

M01	-	Optional stop
M02	-	Program end
M03	-	Spindle rotation Clockwise
M04	-	Spindle Rotation Counter Clockwise
M05	-	Spindle Rotation Stop
M06	-	Tool Change
M08	-	Coolant on
M09	-	Coolant off
M30	-	Program end with rewind
M98	-	Sub Program call
M99	-	End of sub program

Explanation of G-Codes

G00: Rapid Positioning

The Code specifies a linear movement at the rapid feed rate. X, Y, Z can be moved individually or simultaneously. The feed rate need not be programmed.

G01- Linear interpolation

The tool moves along the straight line to the specified position at the programmed feed rate.

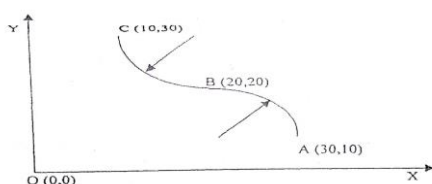
G02- Circular interpolation clockwise

G03- Circular interpolation counter clockwise

Circular interpolation in which the curvature of the path of the tool with respect to the work piece is clockwise/counter clockwise when the plane of motion is viewed in the negative direction of the axis perpendicular to it.

The starting point of the circle/ arc is determined by the previous block. The end point is fixed by the respective axes, values programmed in the block. The circle centre is determined by I, J & K vectors, which specify the incremental (signed) distance of the circle centre point from the circle starting point. I is the incremental (signed) distance parallel to the Z axis and J is the incremental (signed) distance parallel to Z axis, these will remain unchanged irrespective of whether the program is in absolute or incremental mode. G02 and G03 are effective until cancelled by G00 or G01.

Circular interpolation of any two axes at a time is possible. Example



The machine the profile starting from A to C: To machine from C to A

```
G90;
G00 XI 0.0Y 30.0;
G03 X20.0 Y20.0 I1 0.0 JO;
G02 X30.0 Y10.0 I1 10 J-10.0
```

To machine from C to A

```
G90;
G00 X30.0 Y10.0;
G03 X20.0 Y20.0 I-1-10.0 JO; ( A to B)
G02 XI0.0 Y30.0 I1 10 J10.0; (B to C)
```

G04: Dwell

The dwell time is programmed under the address X or P, Dwell is programmed as an unsigned number. The dwell time is effective only in the block in which it is programmed. Dwell is not accepted in MDI mode.

Example: Dwell time of 5 seconds is programmed as

G04 X5 or 04P500

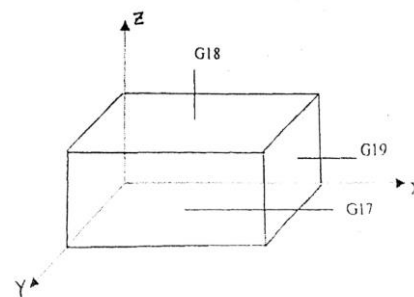
G17: selection of XY plane

G18: selection of ZX plane

G19: Selection of YZ plane

By programming G17/G18/G19 the plane is defined in which cutter radius compensation is available. If a plane has not been selected at the start of the program, the default plane will be defined by G17

When movement of X & Y is intended G17 code is to be programmed. It is a modal code and will be effective until cancelled by G18/G19 same way for Z & Y and Y & Z axis.



G20: Inch Mode

The use of this code will enable the control to accept part program dimension in "Inches", once programmed G20 remains in effect until cancelled by G21

G21: Metric Mode

The basic mode of the system is metric. So it is not necessary to program G21 except when

cancelling G20. G21 enables the control to accept part program dimensions in millimetres. The system remains in metric mode until G20 is program dimensions in millimetres. The system remains in metric mode G20 is programmed.

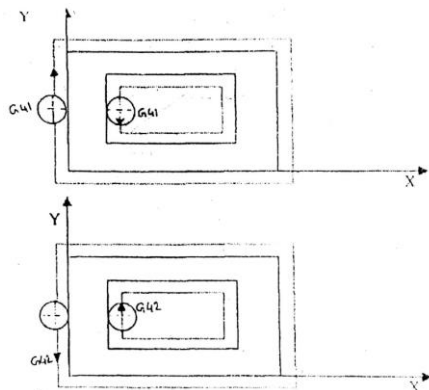
G28: Return to Reference point

The reference is a fixed on a machine tool to which the tool can easily be moved by the reference position function. For example, the reference position is used as a position at which tools are automatically changed.

G40: Cancellation of Cutter Radius Compensation

G41: Cutter Radius Compensation to the left of the Tool path

G42: Cutter Radius Compensation to the Right of the Tool Path



When CRC is selected the control reads in two further blocks in advance during processing of the current block and calculates the intersection point of the compensated paths.

The compensation mode is selected with G41/G42 and offset number D. The D value has to be programmed in the same block as G41/G42.

The compensation mode is selected in the program block with G00 or G01 or in the block previous to a linear interpolation (G00/G01) block.

Circular interpolation is not permitted as the first movement block on selection of CRC (G41/G42).

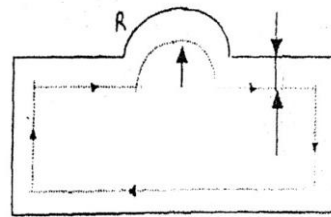
Offset number DO is assigned the compensation value O. The CRC is active in the selected plane. The radius compensation value has to be programmed in geometric offset page.

Corresponding D value has to be programmed in the program either in the same block where G41/G42 is programmed or earlier to it. Different plane selection or position preselect are not allowed after G41/G42 is active. G90 or G91, G20

or G21 etc should be programmed before G41/G42 is active. G41/G42 activation should not follow a circular interpolation block.

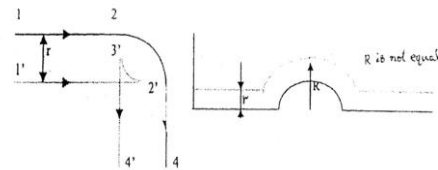
A linear interpolation block has to be programmed along the diameter of the circle.

If the inside profile contain a line segment and a circle segment cutter CRC value should be less than the radius of the circle



The CRC value should not be equal to the radius of any arc programmed in the path while machining an outside profile.

CRC value larger than the smallest linear program motion or larger than the smallest radius of the circle programme is not allowed.



G43: Tool Length Offset

This function compensates for the difference in tool length along the z-axis. When G43 is specified, the tool length offset value (stored in offset memory) specified with the H code is added to the coordinates of the end position specified by a command in the program.

G49: Cancellation of Tool length Offset

After G49 is specified the system immediately cancels the offset mode.

G54-G59: Work piece coordinate system (zero Offsets)

A coordinates system used for machining a work piece is referred to as a work piece coordinate system. It is the difference between the machine zero point and the work piece zero point (to which the part program dimensions are related). By specifying G54 to G59 six work piece coordinate systems can be selected.

G73: High Speed Peck Drilling Cycle

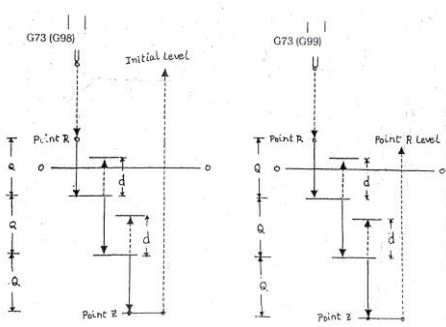
It performs intermittent cutting feed to the bottom of a hole while removing chips from the hole format: G73 G98/99 Z R Q F K

Z=Depth of hole

R=Distance from job surface to point R level

Q=Depth of cut for each cutting feed (Pecking depth)

F=Feed



K=Number of repeats

The High speed peck Drilling Cycle performs intermittent feeding along the Z axis. When this cycle is used, chips can be removed from the hole easily.

G76: Fine Boring Cycle

This cycle bores a hole precisely. When the bottom of the hole has been reached, the spindle stops and the tool is moved away from the machined surface of the work piece and retracted.

Format: G76 G98/99 Z R Q P F K

Z = Depth of hole

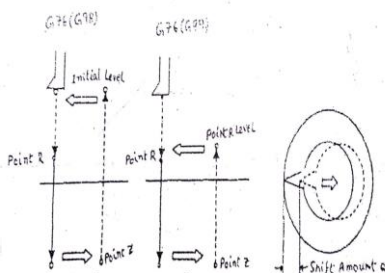
R = Distance from surface to pointer level

Q = Shift amount at the bottom of the hole

P = Dwell time at the bottom of hole

F = Feed

K = Number of repeats.



When the bottom of the hole has been reached, the spindle is stopped at the fixed rotation position, and the tool is moved in the direction opposite to the tool tip and retracted. This ensures that the machined surface is not damaged and enables precise and efficient boring to be performed

G80: Cancellation of any canned cycle.

All canned cycle are to be cancelled to perform normal operation.

G81: Drilling cycle

This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. The tool is then retracted from the bottom of the hole in rapid traverse.

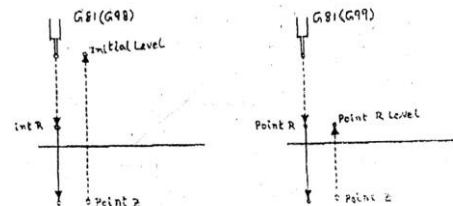
Format: G81 G98/99ZRFK

Z = Depth of hole

R = Distance from surface to point R level

F = Feed

K = Number of repeats.

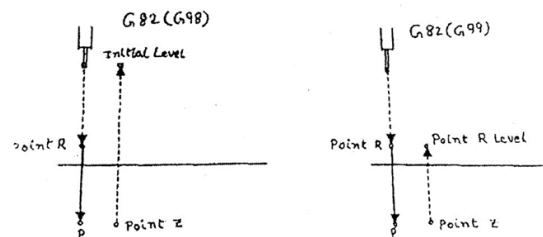


From initial position, rapid traverse is performed to point R. Drilling is performed from point R to point Z. The tool is then retracted in rapid traverse.

G82: Drilling cycle with Dwell/Counter Boring Cycle

This cycle is used for normal drilling. Cutting feed performed to the bottom of the hole. At the bottom, a dwell is performed, and then the tool is retracted in rapid traverse.

Format: G82 G98/99 Z R P FK



From initial position, rapid traverse is performed to point R. Drilling is then performed from

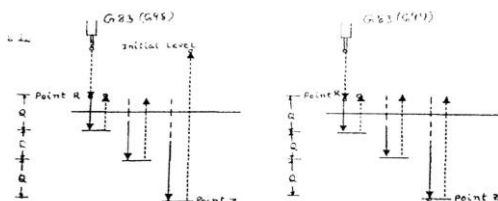
point R to point Z, when the bottom of the hole has been reached a dwell is performed. The tool is then retracted in rapid traverse.

G83: Peck Drilling Cycle

It performs intermittent cutting feed to the bottom of a hole while removing chips from the hole.

Format: G83 G98/99 Z R Q F K

- Z = Depth of hole
- R = Distance from surface to point R level
- Q = Depth of cut for each cutting feed (pecking depth)
- F = Feed
- K = Number of repeats.



Q represents the depth of cut for each cutting feed. It must always be specified as an incremental value.

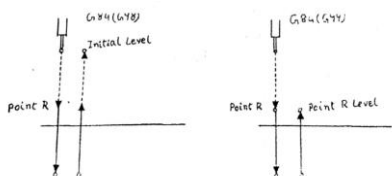
In the second and subsequent cutting feeds, rapid traverse is performed up to a point just before where the last drilling ended and cutting feed is performed again.

G84: Tapping Cycle

In the cycle, when the bottom of the hole has been reached, the spindle is rotated in the reverse direction.

Format: G84 G98/99 Z R P F K

- Z = Depth of hole
- R = Distance from surface to point R level
- P = Dwell time
- F = Feed
- K = Number of repeats.



Tapping is performed by rotating the spindle clockwise. When the bottom of the hole has been reached, the spindle is rotated counter clockwise for retraction.

Feed rate overrides are ignored during the cycle. Feed hold does not stop the machine until the return operation is completed.

G85: Boring Cycle

Format: G85 G98/99 Z R F K

- Z = Depth of hole
- R = Distance from surface to point R level
- F = Feed
- K = Number of repeats.

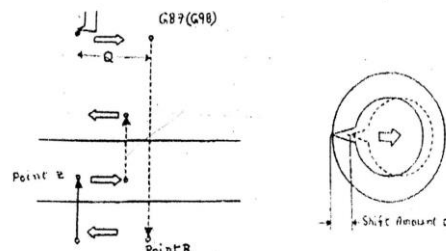
From initial position, rapid traverse is performed to point R. Boring is performed from point R to point Z. When point Z has been reached, cutting feed is performed to return to point R.

G87: Back Boring Cycle

This cycle performs accurate boring

Format: G87 G98 Z R Q P F K

- Z = Distance from the surface to point Z (bottom of the hole)
- R = Distance from surface to point R
- Q = Shift amount of the tool
- P = Dwell time
- F = Feed
- K = Number of repeats.



After positioning, the spindle is stopped at the fixed rotation position. The tool is moved in the direction opposite to the tip. Positioning (rapid traverse) is performed to the bottom of the hole (point R). The tool is shifted in the direction of the tool to and the spindle is rotated clockwise. Boring is performed in the positive direction along Z-axis until point Z is reached as point Z, the spindle is stopped at the fixed rotation position again; the tool is shifted in the direction to the tool tip, and then the tool is returned to the initial level. The tool is then shifted in the direction of the tool tip.

G90: Absolute mode

This code is used to select the absolute mode for X, Y & Z axes dimensions and defines the

tool path as the tool distance from a fixed zero position. This mode will remain in effect until changed by programming a G91 code.

G91: Incremental Mode

If this code is programmed X, Y & Z words are taken to be incremental path departures from the present position. The state of the control will remain in the incremental mode until G90 is programmed.

G92: Work coordinate system (position preset)

This code sets the position registers to the values specified by X, Y & Z codes programmed in the same block. Position preset occurs only for the axes programmed and will not cause axis motion. No other G codes are accepted with this code.

Example: G92 X100.0 Y 50.0;

After executing the above block, X position will be preset to 100 mm and Y position will be preset to 50mm.

G94: Feed Rate per Minute

The code allows the feed rate to be programmed in mm per revolution or inch per revolution. The value programmed in the F word represents the feed rate directly in mm/min or inch/min depending on whether G20 or G21 is active. G94 will remain active until G95 is programmed.

G95: Feed Rate per Revolution

If a program consist of a fixed sequence of frequently repeated pattern, such a sequence of pattern can stored as a subprogram in memory to simplify the program. These are used where repetition of a part of a program is required. They are stored at the end of the main program and can be called any time and any number of times for execution by simple instruction words in the main program. For calling a subprogram, word M98 is used along with P (number of the subprogram to be called) in the main program of the subroutine needs to be repeated, say twice, and then the word P is followed by L2. The subroutine ends with word M99 subroutine can be called J subroutine and thereby nesting can be done use of subroutines helps in short lining the main program and also prevent repetition of statements where the same features are required to be machined more than once.

A subprogram can be called from main program. Three different subprogram can be called from a subprogram. Any number of subprograms can be defined in a main program subprogram cannot be defined in a other subprogram.

Format:

O ____ (subprogram number);

M99 ;(program end)

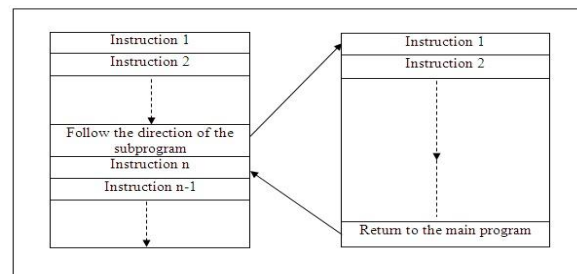
Subprogram call

M98 P-----O-----;

P (number of times subprogram is repeated)

O (subprogram number)

When number P is specified, the subprogram is called just once.



Manufacturing process by using CAD/CAM packages,

The steps involved are

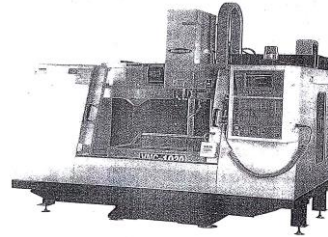
1. Enter Part Geometry
2. Identify process plan
3. Identify cut Boundary
4. Identify cutting Tools
5. Identify cutting process parameters
6. Identify pocket clearance method

- ↓
7. Generate tool path/Cutter location data(with prep processor)
- ↓
8. Program Generation (with Post processor)
- ↓
9. Transmitting the generated program from computer to machine with DNC

MAIN PROGRAMME

```
O0001;
G00 G40 G80 G49;
G28 G91 Z0.0;
M06 T01;
G90 G54 X-20.0 Y-20.0;
G54 Z50.0;
M03 S2500;
G00 Z5.0;
M08;
G01 Z0.0 F100;
M98 P198 0002;
G90 G00 Z50.0;
M05;
M09;
G28 G91 Z0.0;
G28 G91 X0.0 Y0.0;
M30;
SUB - PROGRAMME (1: 80)
G91 G01 Z-0.5 F100;
G90 G42 G01 Y0.0 D10 F900;
X64.5 Y1.25;
G03 X76.06 Y2.12 R73.9;
G03 X78.40 Y2.82 R14.0;
G03 X79.44 Y3.53 R4.4;
G03 X79.88 Y4.60 R1.3;
G03 X79.13 Y5.90 R2.2;
G03 X74.82 Y8.20 R13.6;
G03 X54.67 Y10.78 R61.0;
G03 X0.0 Y0.0 R195.0;
G40 G00 X-20.0;
Y-20.0;
M99;
```

CNC VERTICAL MACHINING CENTRE



CNC VERTICAL MACHINING CENTRE

CONCLUSIONS

The variation of the aerodynamic characteristics of the related air foils with the geometric characteristics investigated may be summarized as follows:

Variation with thickness ratio:

1. The slope of the lift curve in the normal working range decrease with increased thickness, varying from 95 to 81 percent, approximately, of the theoretical slope for thin airfoils.
2. The angle of zero lift moves toward zero with increased thickness.
3. The highest values of the maximum lift are obtained with sections of normal thickness ratios (9 to15 percent).
4. The greatest instability of the air flow at maximum lift is encountered with the moderately thick, low –cambered sections.
5. The magnitude of the moment at zero lift decreases with increased thickness, varying from 87 to 64 percent, approximately (for normally shaped airfoils), of the values obtained by thin-airfoil theory.
6. The axis of constant moment usually passes slightly forward of the quarter-chord point, the displacement increasing with increased thickness.
7. The optimum lift coefficient (The lift coefficient corresponding to the minimum profile-drag coefficient) approaches zero as the thickness is increased.
8. The ratio of the maximum lift to the minimum profile drag is highest for airfoils of medium thickness ratios (9 to 12 Percent).

Variation with thickness ratio:

1. The slope of the lift curve in the normal working range is little effected by the camber; a slight decrease in the slope is

- indicated as the position of the camber moves back.
 2. The angle of zero lift is between 100 and 75 percent, approximately, of the value given by thin-air foil theory, the smaller departures being for air foils with the normal camber positions.
 3. The maximum lift increases with increased camber, the increase being more rapid as the camber moves forward or back from a point near the 0.3c position.
 4. Greater stability of the airflow at maximum lift is obtained with increased camber if the camber is in the normal position (0.3c to 0.5c).
 5. The moment at zero lift is nearly proportional to the camber. For any given thickness, the difference between the experimental value of the constant of proportionality and the value predicted by thin-air foil theory is not appreciably effected by the position of the camber except for the sections having the maximum camber well back, where the difference becomes slightly greater.
 6. The axis of constant moment moves forward as the camber moves back.
 7. The maximum profiles drag increase with increased camber, and also with a rear ward movement of the camber.
 8. The optimum lift coefficient increase with the camber and for the highly cambered sections a definite increase accompanies a forward movement of the camber.
- [5]. Eppers, R and Somers,D.: A computer program for the Design and Analysis of Low-Speed Airfoils, NASA TM-80210,1980
 - [6]. <http://www.techniatranscat.com/solutions/dassault-systemes-software/catia>
 - [7]. http://www-903.ibm.com/kr/solutions/plm/download/PLM_Infrastructure.pdf
 - [8]. <http://samuelstoicescu.com/ctutorials.html>

Bibliography

- [1]. Hepperle,Martin: Neue Profile Fur NurFlugel modelle, FMT-Kolleg 8, Verlag fur technik und Handwerk, baden-Baden, Germany, 1988.
- [2]. Unverfehrt, Hans-Jurgen: Faszination Nurflugel, Verlag fur Technik und Hnadwerk,Baden-Baden, Germany, 1990
- [3]. Hansen, H.: Windkanalmessungen in Reynoldzahlbereich von Modellflugzeugen, FMT-Kolleg 6, Verlag fur Technik und Handwerk, Baden-Baden, Germany, 1989.
- [4]. Meuller, T.et al.: Low Reynolds Number Wind Tunnel Measurements: The Importance of being Earnest, Conference on Aerodynamics at Low Reynolds Numbers, London, 1986.