RAJALAKSHMI ENGINEERING COLLEGE

THANDALAM - 602 105

DEPARTMENT OF MECHANICAL ENGINEERING

ME 1356 – CAD/CAM LAB MANUAL

COMPLIED BY:

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CAD INTRODUCTION:

Computer-aided design is essentially based on a versatile and powerful technique called computer graphics, which basically means the criterion and manipulation of pictures on a display device with the aid of a computer. Computer graphics originated at the Massachusetts institute of technology (MIT) in 1950when the first computer-driven display, linked to a Whirlwind 1 computer, and was used to generate some pictures. The first important step forward in computer graphics came in 1963 when a system called SKETCHPAD was demonstrated at the Lincoln Laboratory of MIT. This system consists of a cathode ray tube (CRT) driven by TX2 computer. The CRT had a keyboard and a light pen. Pictures could be drawn on the screen and then manipulated interactively by the user via the light pen.

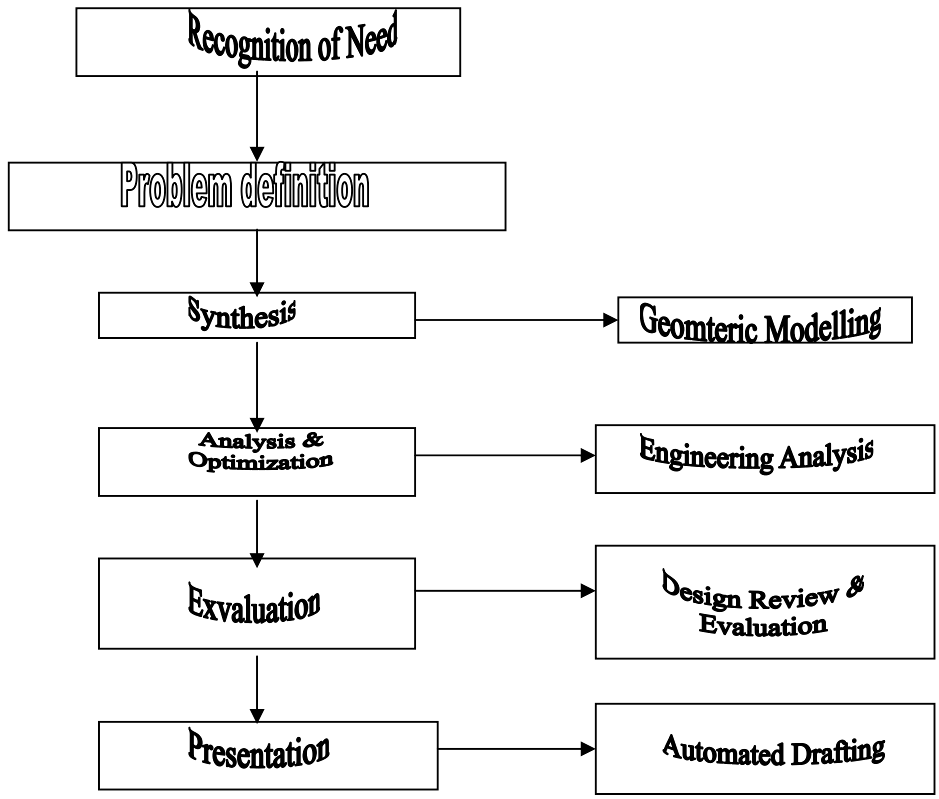
This demonstration clearly showed that the CRT could potentially be used as a designer's electronic drawing board with common graphic operations such as scaling, translation, rotation, animation and simulation automatically performed at the 'push of a button'. At that time, these systems were very expensive; therefore they were adopted only in such major industries as the aircraft and automotive industries where their use in design justified the high capital costs. Another crucial factor preventing computer graphics from being generally applied to engineering industries was that there was a lock of appropriate graphics and application software to run on these systems. However, a computer-based design system was clearly emerging. Since these pioneering developments in computer graphics, which had captured the imagination of the engineering industry all over the world, new and improved hardware, which is faster in processing speed, larger in memory, cheaper in cost and smaller in size, have become widely available.

Sophisticated software techniques and packages have also been gradually developed. Consequently, the application of CAD in industry has been growing rapidly. Initially CAD systems primarily were automated righting stations in which computer controlled plotters produced engineering drawings. The system were later linked to graphic display terminals where geometric model describing part dimensions were created, and the resulting database in the computer was then used to produce drawings. Nowadays, CAD systems can do much more than mere righting. Some systems have analytical capabilities that allow parts to be evaluated with techniques such as the finite element method. There are also kinematics analysis programs that enable the motion of mechanism to be studied. In addition, CAD system includes testing techniques to perform model analysis on structures, and to evaluate their response to pinpoint any possible defects.

Computer Aided Design is the process of developing and using computer assisted design tools in the design process. The advent of computers has contributed to significant advance in calculation, data handling and utilization applications. The ability to use the computers in these application areas enhances the capability of the design team significantly. Drafting and geometric modeling play significant roles in CAD. The module therefore concentrates on the general design process with specific consideration to drafting and geometric modeling. Three different CAD systems are referred to in the module. The syllabus includes: historical development, the design process, traditional drawing practice and the development of the CAD industry, system hardware, computers micros to mainframes, output devices, storage, workstations, networked systems, examples of CAD systems; simple entity descriptions: points, lines, arcs, made-edge lists, free-form curves, free-form surfaces; transformations: pan, rotate and scale, 3D transformations, observer angles, perspective, depth cueing; geometric modeling: wire frame modelers, surface modelers, solid modelers (CSG and B- rep), hidden line removal and mass properties; user interface: input devices, menus, graphics interface language, parametric. LEARNING OBJECTIVES: To understand and handle design problems in a systematic manner. To be able to use the capabilities provided by computers for calculations, data handling and visualization applications. To gain practical experience in handling 2D drafting and 3D modeling software systems. To be able to apply CAD in real life applications.

ROLE OF COMPUTERS IN DESIGN:

As manual design process has several risk factors including human fatigue and the vealution of design based on his previous experience. With the advent of computer and the development in the field of computer graphics, various design & manufacturing process takes place new faster rate with minimum or potimum error. The below figure shown the implementation of computer in design :



Computers in Design .

Implementation of computer in the design stage becomes the subset of design process. Once the conceptual design materializes in the designer mind the geometric model starts by the appropriate CAD software. The choice of geometric model to CAD is analogous to the choice. The various design related tasks which are performed by a modern computer-aided design system can be grouped into four functional areas :

1. *Geometric Modelling*
2. *Engineering Anlaysis*
3. *Design review and evaluation*
4. *Automated drafting.*

Geometric Modelling :

It is concerned with the computer compatible mathematical description of the geometry of an object. The mathematical description allows the image of the object to be displayed &

manipulated on a graphics terminal through signals from the CPU of the CAD system. The software that provides geometric modeling capabilities must be designed for efficient use both by the computer & the human designer.

During the geomtric modelling computer converts the command into a mathematical model, stores it in the computer data files, and display it as an image on the CRT screen. Object can be represneted by geometric model by wireframe, surface model or solid model.Another feature of CAD system is color graphics capability. By means of color, it is possible to diplay more information on the graphics screen.

Engineering Analysis :

The analysis may involve stress-strain calculation, heat transfer computaion etc., of the system being displayed. The computer can be used to aid in this analysis work. It is often necessary that specific programs be developed internnaly vy the enginnering analysis group to solve particular design problem. In other situtaion , commercially available general purpose programs can be used to perform the engineering analysis. Analysis may be :

a. *Mass property analysis.*
b. *Finite element analysis.*

The analysis of mass properties is the anlysis feature of CADsystem which providesproperties of solid objectbeing analysed, suchh as the surface area, weight, volume, centre of gravity and momnet of inertia.

In FEA the object is divided into large number of finite elements which form an interconnecting network network concentared nodes. By using a computer with significant computational Capabilities, the entire object can be anlysed for stress- strain, heat transfer coefficient at nodes.By determining the intterrelating behaviours of all nodes in the system, the behaviour of the entire object can be assesed.

Design review &Evaluation ;

Checking the accuracy of the design can be accomplished conviently on the graphical terminal. Semiautomatic dimensioning and tolerancing routines which assign size specification to surface indicated by the user help you to reduce the posibility of dimensioning errors. The designer can zoom inon part design details and magnify the imageon the graphics screen for close scrutiny

One of the most important evaluation feature available on some compueter aided design systems is KINEMATICS. The available kinematics packages provide the capability to animate the motion of the simple designed mechanisms such as hinged component & linkages.Commercial kinematicas software avaiolable is ADAMS(Autamatic DynamicAnalysi of Mechanical Systems).

Automated Drafting :

It involves the creation of hard-copy engineering drawings directly from the CAD data base. Most of the CAD systems are capable of generating as many as six views of the parts. Engineering drawings can be made into company drafting standard by programming the standards into the CAD system.

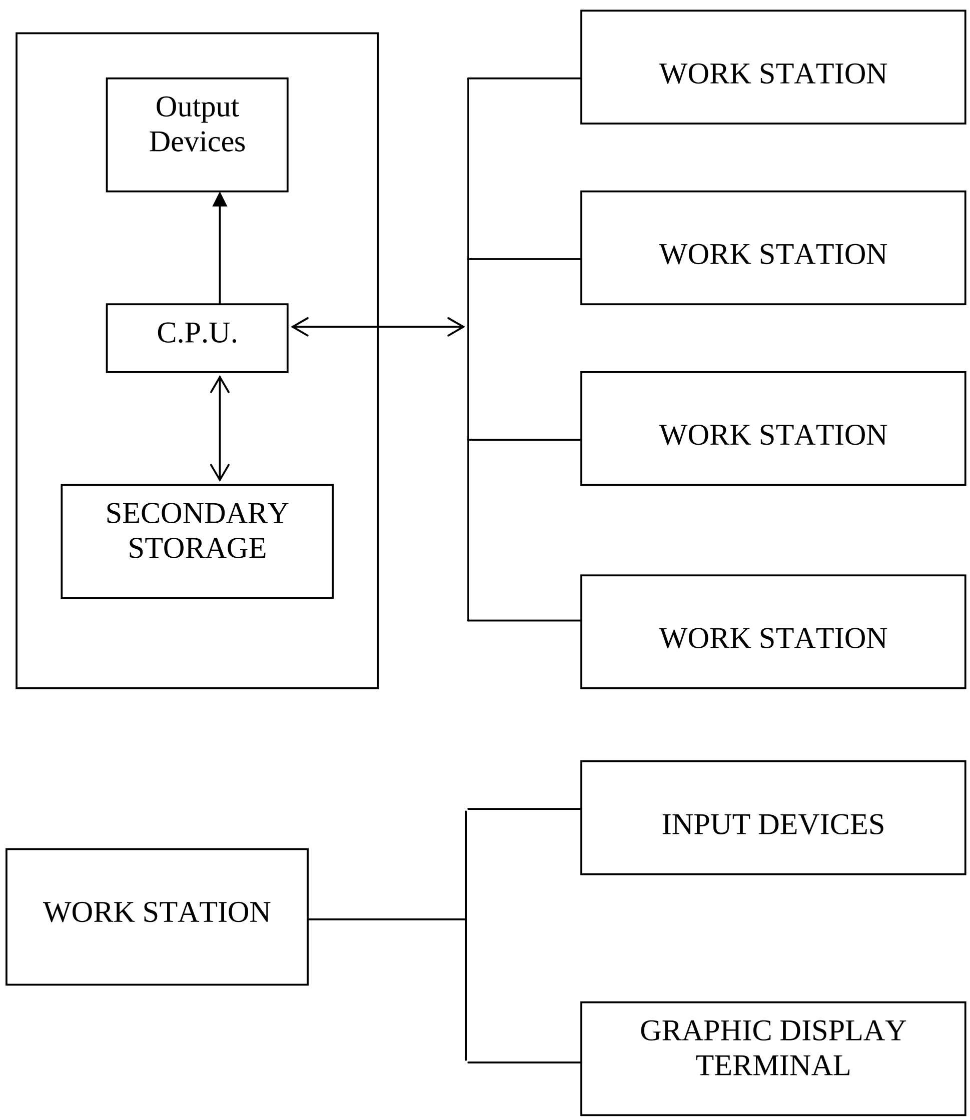
Implementation of computer in the design stage becomes the subset of design process. Once the conceptual design materialize in the designer mind the geometric model starts by the appropriate CAD software. The choice of geometric model to CAD is analogous to the choice of a mathematical model to engineering analysis. A valid geometric model is created by definition translator, which converts the designer input into the proper database format. In order to apply engineering analysis in geometric model, interface algorithms are provided by the system to extract the required data from the model database to perform the analysis. In case of FEA, these algorithms from the finite element modeling package of the system. Design testing & evaluation may require changing the geometric model before finalizing it.

When the final design is achieved the drafting & detailing of the model starts, followed by documentation & production of final drawings

COMPONENTS OF CAD SYSTEM

The components of a typical CAD system are illustrated in the following figure. The central processing unit (CPU) is the brain of the entire system. It contains of integrated circuits of (IC) of three parts – ALU, controller and main memory unit. The arithmetic logic unit (ALU) consists of electronic circuits, which perform logic and mathematical operations. Controller circuits are used to regulate various operations carried out in the computer. Main memory circuits store processed data, such as results of calculations and program instructions inside the computer. Hundreds of electronic circuits are reduced and etched on chip as small as a pinhead. The CPU is, therefore, one of the miracles of modern electronic technology.

Elements of CAD system



In the CAD system, the functions o the CPU is as follows:

i) To receive information from the work station and display the output on a CRT screen;

ii) To red the data stored in a secondary memory storage unit;

iii) To give instructions to output devices such as plotters to create permanent drawings; and

iv) To transmit data to and from magnetic tapes.

In addition to main memory circuits in CPU, secondary storage capacity is provided to reduce the cost of the main computer.

The functions of the secondary storage unit are as follows:

i) To store files related to the engineering drawings;

ii) To store CAD software; and

iii) To store programs required to give instructions to output devices like plotters.

The secondary storage unit consists of magnetic tapes and disks. Magnetic tape is similar to the tape used in a tape recorder. It consists of Mylar tape coated with magnetic material. The data are stored in the form pf magnetized spots. The data can be erased and reused. The data are stored sequentially, i.e. to find a certain piece of data on the tape; one must wind the tape till the data are reached. This is called the sequential access method. Magnetic tapes are cheap but the access time for data retrieval is more due to sequential access. They are mainly used for archiving drawings.

There are two types of magnetic disks – flexible and hard. The appearance of flexible disk is similar to that of a phonographic record. It is, however, thin and flexible compared with records, hence name floppy disk. The flexible disk is made of plastic like material – Mylar – with a thin coating of magnetic material such as ferric oxide. The data can be stored on the one side of the disk (single) or on both surfaces (dual). The standard diameters of floppy disks are 131mm and 200mm the disk is always kept in a square vinyl jacket for protection against dust particles and scratching. There is a small cut section in the jacket, called window. Reading and writing is accomplished through this window by means of a drive-head. The speed of rotation of the disk is usually 300r.p.m. The construction of the hard disk is similar to that of a flexible disk. It is, however, made from thin aluminium plate coated with ferric oxide. The disk is usually sealed in an airtight container and rotates at a much faster speed of 3600rpm. This increases speed of storage and retrieval of information. A hard disk is more durable than a floppy disk. Cost is the main limitation of this disk. There are two methods to store data on flexible as well as hard disks

– sequential and random access methods. In the sequential search method, data are stored in a sequence and the drive head has to search for a piece of information, starting from the beginning of the track. This increases the search depending upon the location of the information. The random access method is also called the direct access method. In this method, data stored on the disk are divided into two or more sections. When the section number is specified, the drive head directly moves to the relevant section and starts searching the data. Random access method is a faster method of data retrieval.

The computer systems used for CAD are of three types – mainframe, mini and micro. The mainframe system consists of a large capacity computer kept in a remote air-conditioned room. Strict environmental controls are needed for this system. The workstations are located at some distance from this central system. The mainframe system executes a number of functions, CAD being one of them. This system is more powerful than mini or micro systems, with fast computing speeds. Due to large memory capacity it can process the most difficult programs. Compared with the mainframe system, the microcomputer is small and inexpensive. It, however, operates at a slightly lower speed and is not able to process some of the difficult programs, which can be run on the mainframe system. Minicomputers are usually housed in an air-conditioned room. The microcomputer is the smallest type of CAD system. It does not require strict environmental controls. A graphic display station and keyboard is normally combined in to a micro unit. These units are called desktop computers. A microcomputer system is called a dedicated system, because it operates for the sole purpose of one user at a time. This system is cheap and easily available, but has limited capacity and speed.

The workstation is a visible part of the CAD system, which provides interaction between the operator and the system. There are two elements of a basic workstation – a CRT display and an alphanumeric keyboard. Other input devices, such as cursor control devices, digitizers and graphic tablets, are provided on elaborate workstations. Graphic display terminals and input devices are discussed in the forthcoming sections. The output devices used with the CAD systems are pen plotters, hardcopy units and electrostatic plotters.

GEOMETRIC MODELING

A geometric modeling is defined as the complete representation of an object that includes in both graphical and non-graphical information.

In computer-aided design, geometric modeling is concerned with the computer compatible mathematical description of the geometry of an object. The mathematical description of the geometry of an object to be displayed and manipulated on a graphics terminal through signal from CPU of the CAD system. The software that provides geometric modeling capabilities must be designed for efficient use of both by the computer and the human designer.

To use geometric modeling, the designer construct the graphical image of the object on the CRT screen of the IGS system by inputting three types of commands to the computer. The first type of command generates basic geometric elements such as points, lines, and circles. The second command types is used to accomplish scaling, rotation or other transformations of these elements. The third type of command causes the various elements to be joined into desired shape of the object being created on the ICG system.

During this geometric modeling process the computer converts the commands into mathematical model, stores it in the computer data files and displays it as an image on the screen. The model can be subsequently being called from the data files for review, analysis or alteration. The most advanced method of geometric modeling is solid modeling in three dimensions. This method uses solid geometry shapes called primitives to construct the object.

Basically there are three types of modeling, they are

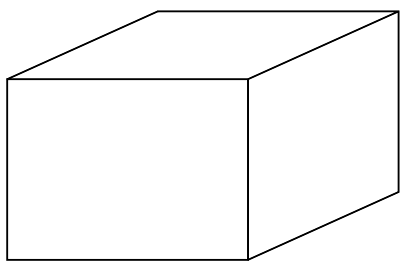
a. **Wire Frame Modeling**

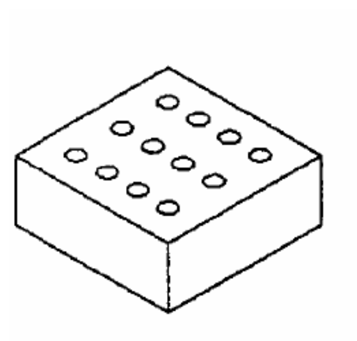
b. **Surface Modeling**

c. **Solid Modeling**

WIRE FRAME MODEL-INTRODUCTION:

This is the basic form of modeling; here the objects drawn will be simple but more verbose, geometric model that can be used to represent it mathematically in the computer. It is sometimes referred as a stick figure or an edge representation of the object. Typical CAD/CAM system provides users with possibly three modes to input coordinates: Cartesian, Cylindrical or Spherical. Each mode has explicit or implicit inputs. Explicit input could be absolute or incremental coordinates. Implicit input involves user digitizes..A wire frame model consists of points, lines, arcs, circles &curves. Early wire frame modeling techniques developed in 1960's were 2-dimensional. They are not centralized &associative. Later in 1970's the centralized, associative database concepts enabled modeling of 3D objects as wire frame models that can be subject to 3-dimensional transformations.





WIRE FRAME ENTITIES

Wire frame Entities are divided into 2 types are:

a. Synthetic Entities----------→ Splines & Curves
b. Analytic Entities----------→ Points, lines, Circles, arcs, conics, fillet, chamfer

Applications:

1. Two-dimensional drafting.
2. Numerical control tool path generation.

Advantages:

1.It is simple to construct model.

2. Less computer memory to store the object.

3.CPU time to retrieve, edit or update a wireframe model is less.

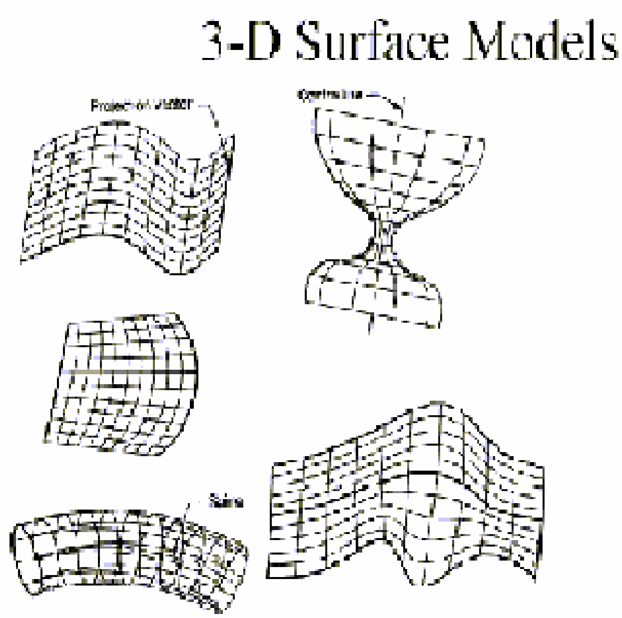
4.Doesnot require extensive training.

Disadvantages:

1 .It is ambiguous representation of real object.
2 It lack in visual coherence and information to determine the object.
3. User or terminal time needed to prepare & or input data increases with complexity of object.
4. Inability to detect interference between components.
5. No facility for automatic shading.
6. Difficult in calculating Physical properties like Mass, surface area, centre of gravity etc.,

Surface Modeling:

A surface model of an object is more complete and less ambiguous representation than it wire frame model. It is also richer in associated geometric contents, which make it more suitable for engineering and design applications. Surface model takes one step beyond wire frame models by providing information on surfaces connecting the object edges. Creating a surface have some quantitative data such as point & tangents & some qualitative data like desired shape & smoothness. Choice of surface form depends on type of application.



Surface Entities:

Similar to wire frame entities, existing CAD / CAM systems provide designers with both analytic and synthetic surface entities. Analytic entities include plane surface, ruled surface, surface of revolution, and tabulated cylinder. Synthetic entities include the bicubic Hermite spline surface, B – spline surface, rectangular and triangular bezier patches, rectangular and triangular Coons patches, and Gordon surface. The mathematical properties of some of these entities are covered in this chapter for two purposes. First, it enables users to correctly choose the proper surface entity for the proper application. For example, a ruled surface is a linear surface and does not permit any twist while a B – spline surface is a general surface. Second users will be in a position to better understand CAD/CAM documentation and the related modifiers to each surface entity command available on a system. The following are descriptions of major surface entities provided by CAD/CAM systems

Application:

1. Calculating mass properties.
2. Checking for interference between mating parts.
3. Generating cross-sectioned views.
4. Generating finite element mesh.

Advantages:.

1. They are less ambiguous than wireframe model.

2. Surface model provides hidden line and surface algorithms to add realism to the displayed geometry.

2. Surface model can be utilized in volume and mass property calculations, finite element modeling, NC path generation, and cross section &interference detections.

3. Change in finite element mesh size produce more accurate results in FEA

Disadvantages:

1. Surface models are generally more complex and thus require more terminal and CPU time and computer storage to create than wireframe models.

2. Surface models are sometimes awkward to create and may require unnecessary manipulations of wireframe entities.

3. It requires more training to create.

4. It does not provide any topological information.

Solid Modeling:

A solid model of an object is more complete representation than its surface model. It is unique from the surface model in topological information it stores which potentially permits functional automation and integration. Defining an object with the solid model is the easiest of the available three modeling techniques. Solid model can be quickly created without having to define individual locations as with wire frames. The completeness and unambiguity of solid models are attributed to the information that is related database of these models stores **(Topology--→ It determine the relational information between objects.)**

To model an object completely we need both geometry & topological information. Geometry is visible, whereas topological information are stored in solid model database are not visible to user. Two or more primitives can be combined to form the desire solid. Primitives are combined by Boolean Operations.

Different Boolean operations are:

1. ***Union (U)***
2. ***Intersection (n)***
3. ***Difference (-)***

SOLID ENTITIES

There are a wide variety of primitives available commercially to users. However, the four most commonly used are the *block, cylinder, cone, and sphere*. These are based on the four natural quadrics: planes cylinders, cones, and spheres.

INTRODUCTION TO SOLID WORKS:

Solid Works is a powerful 3D modeling program. The models it produces can be used in a number of ways to simulate the behavior of a real part or assembly as well as checking the basic geometry. This tutorial guides you through construction of the model steam engine shown here. First you'll learn the basics of creating solid features needed to build the major functional parts and assemble them. In later sessions you'll generate engineering drawings and experiment with animation and 'photo-realistic' rendered views too. This should give you the knowledge needed to create more complex designs as you explore the enormous functionality of Solid Works.

STARTING A NEW SESSION OF Solid Works 2006:

To start a new session of Solid Works 2006, choose
Start > Programs > Solid Works 2006SP0.0 > Solid Works 2006 SP0.0 from the **Start** menu or double-click on the **SolidWorks2006 SP0.0** icon placed on the desktop of your computer. The Solid Works 2006 window will be displayed. If you are starting Solid Works application for the first time after installing it, the **Welcome to Solid Works** dialog box will also be displayed, as shown in Figure 1.. This dialog box welcomes you to Solid Works and helps you customize Solid Works installation. The options available in this dialog box are discussed next.

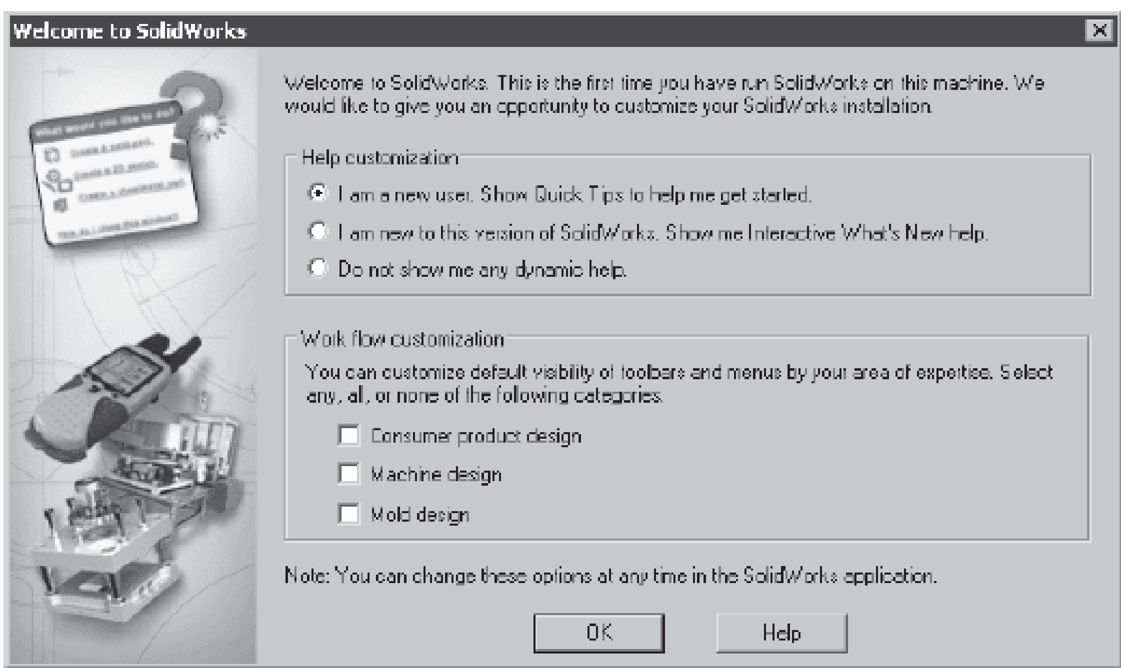


Fig 1: Solid works Welcome Window

DRAWING SKETCHES FOR SOLID MODELS:

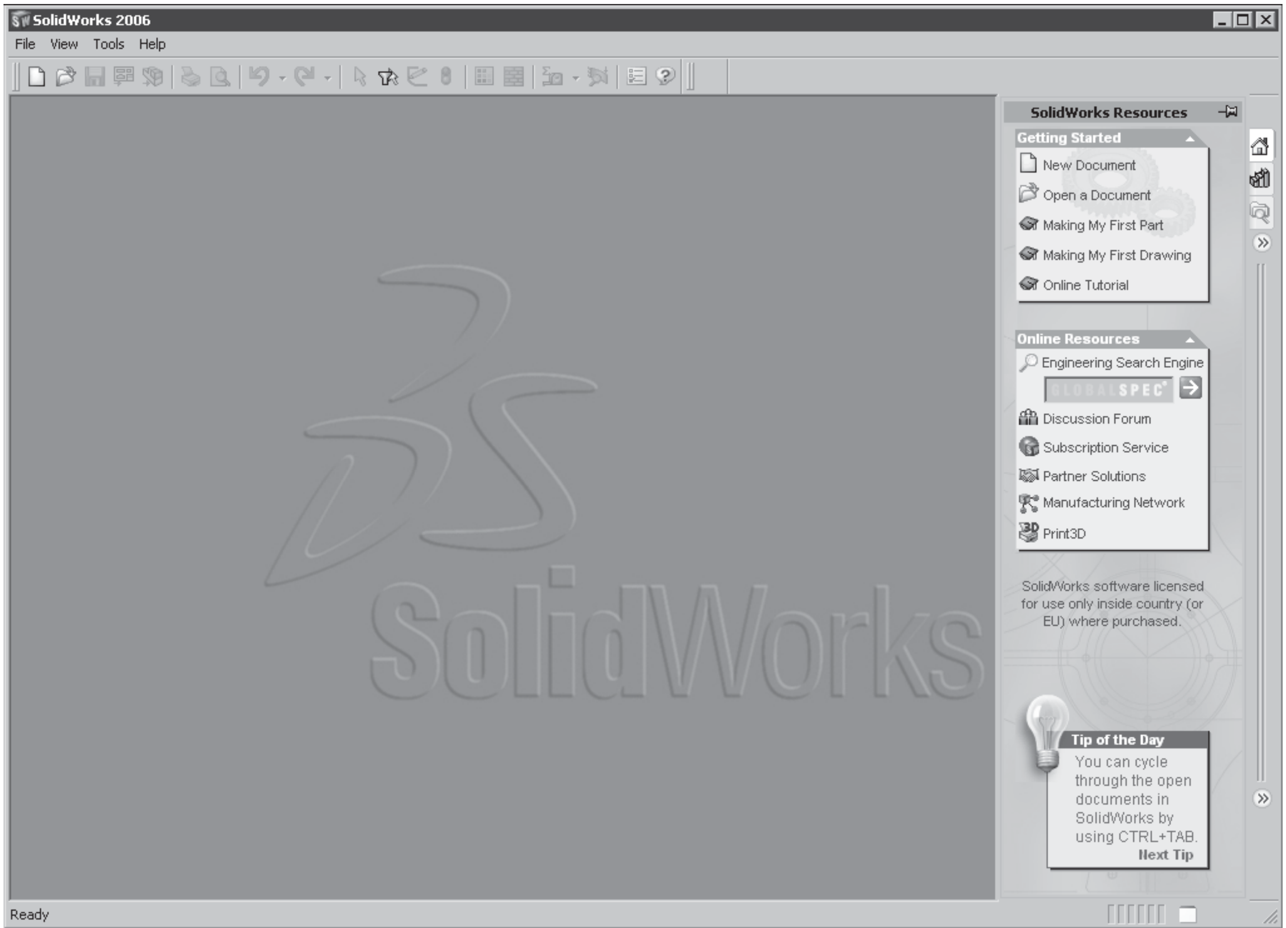


Fig 2: Solid works 2006 Windows

STARTING A NEW DOCUMENT IN Solid Works 2006:

To start a new document in Solid Works 2006, choose the **New Document** option from the **Getting Started** group of the **Solid Works Resources Task Pane**;

The **New Solid Works Document** dialog box will be displayed, as show in Figure 3.

You can also invoke this dialog box by choosing the **New** button from the **Standard** toolbar. The options provided in this dialog box are discussed next.

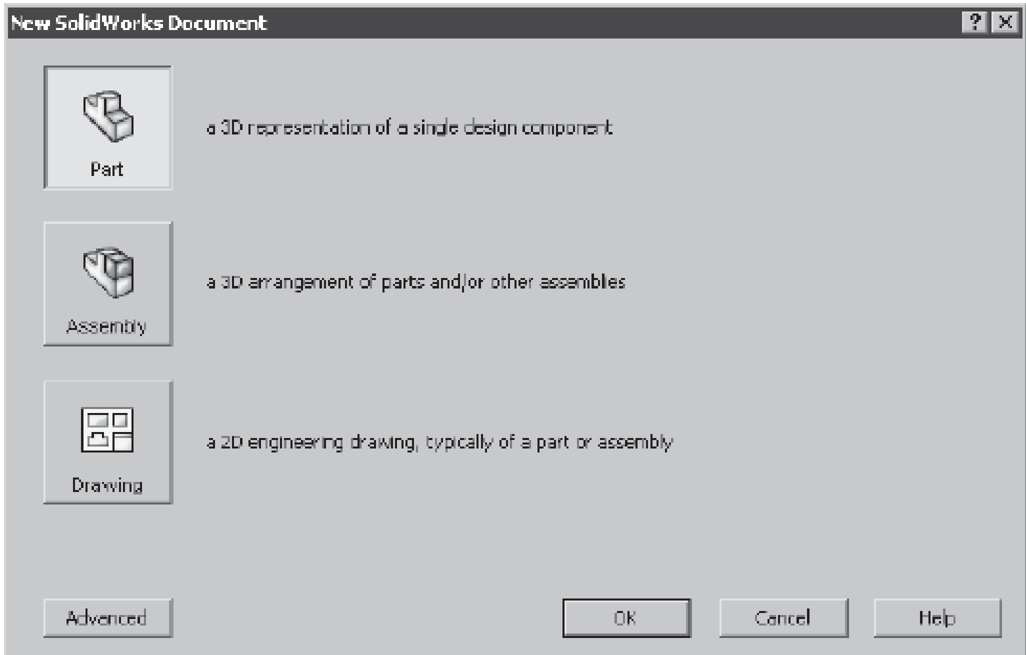


Fig 3: Solid works Startup Dialog Box

Part

The **Part** button is chosen by default in the **New Solid Works Document** dialog box. Choose the **OK** button to start a new part document to create solid models or sheet metal components. When you start a new part document, you will enter the **Part** mode

Assembly

Choose the **Assembly** button and then the **OK** button from the **New Solid Works Document** dialog box to start a new assembly document. In an assembly document, you can assemble the components created in the part documents. You can also create components in the assembly document.

Drawing

Choose the **Drawing** button and then the **OK** button from the **New Solid Works Document** dialog box to start a new drawing document. In a drawing document, you can generate or create the drawing views of the parts created in the part documents or the assemblies created in the assembly documents.

THE SKETCHING ENVIRONMENT:

Whenever you start a new part document, by default you are in the part modeling environment. But, you need to start the design by first creating the sketch of the base feature in the sketching environment. You can invoke the sketching environment using the **Sketch** tool available in the **Standard** toolbar.

You can also choose the **Sketch** button from the **Command Manager** (Figure 4) to invoke the **Sketch Command Manager**.

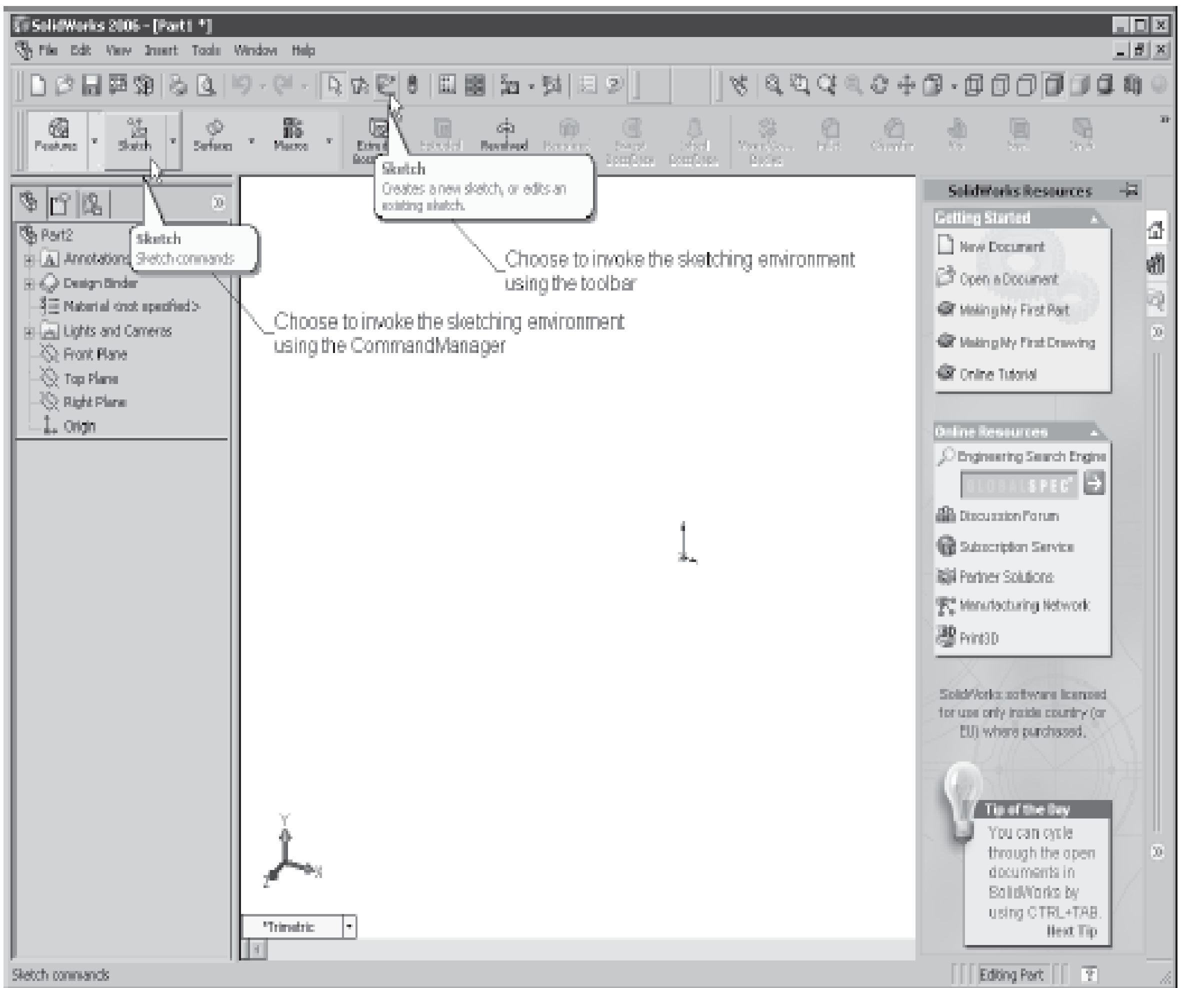


Fig: 4 Sketching Environment in Solid works

When you choose the **Sketch** button from the **Standard** toolbar or

Choose any tool from the **Sketch Command Manager**; the **Edit Sketch Property Manager** is displayed and you are prompted to select the plane on which the sketch will be created. Also, the three default planes available in Solid Works 2006 (**Front Plane**, **Right Plane**, and **Top Plane**) are temporarily displayed on the screen, as shown in Figure 5.

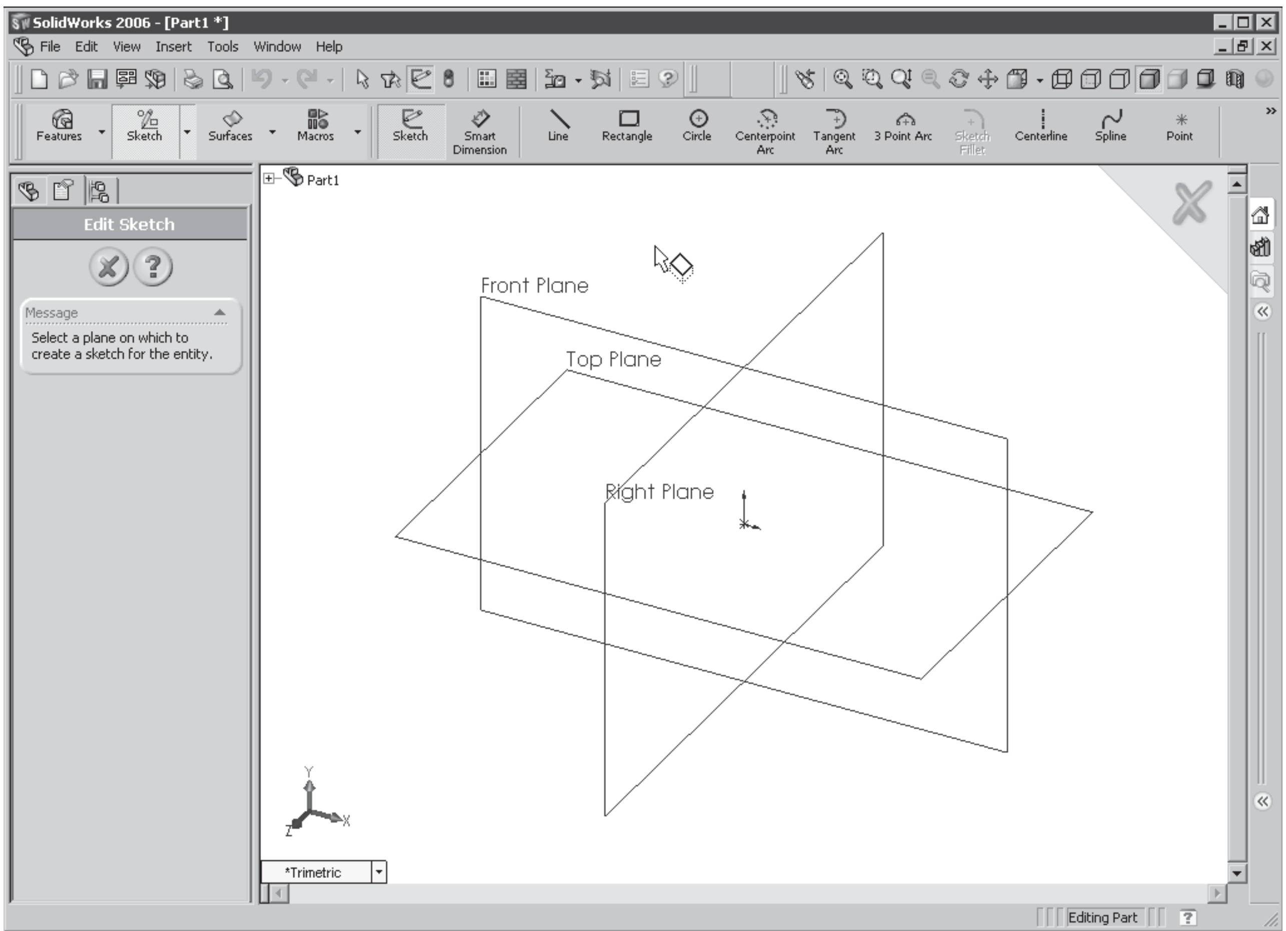


Fig 5: Planes displayed in Solid Works

Ex: 1 Exercise on Extrusion

AIM:

To model the given object using the Extrusion feature as per the dimensions given.

Description of Extrusion Feature:

Base Feature:

- **The first feature that is created.**
- **The foundation of the part.**
- **The base feature geometry for the box is an extrusion.**
- **The extrusion is named** *Extrude1***.**

To Create an Extruded Base Feature:

1. Select a sketch plane.

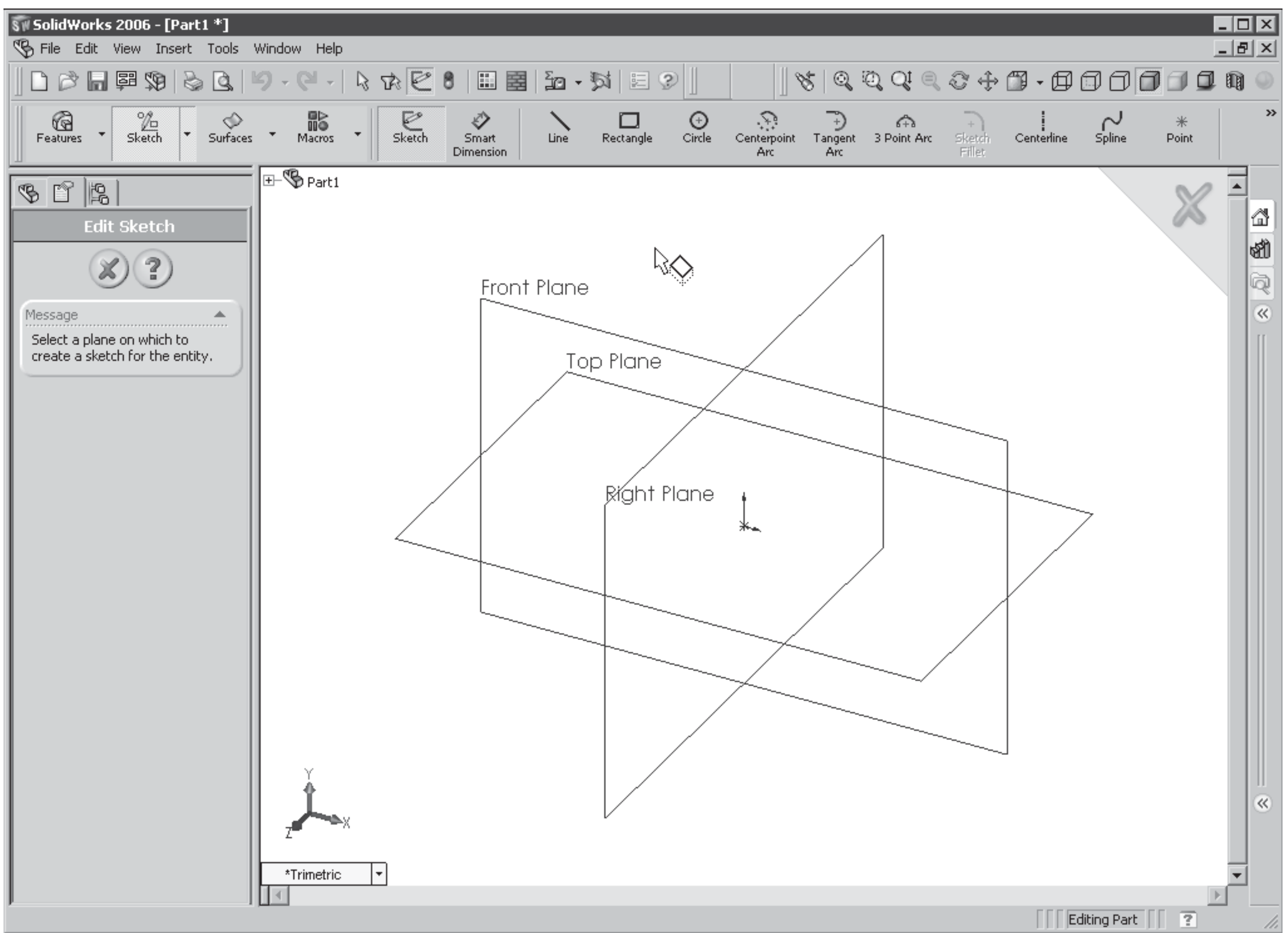
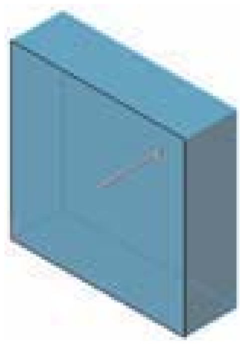
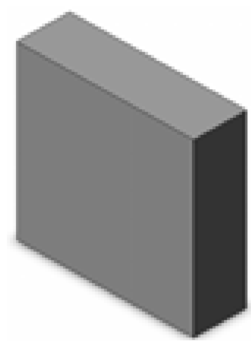


Fig:6 Solid works Plane display

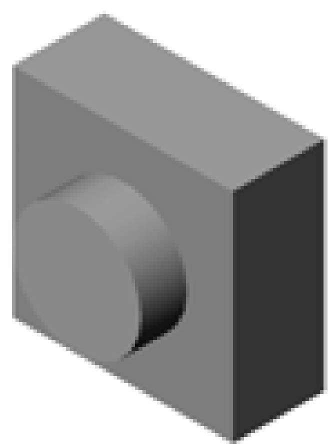
1. **Sketch a 2D profile of the model**
2. **Extrude the sketch perpendicular to sketch plane.**





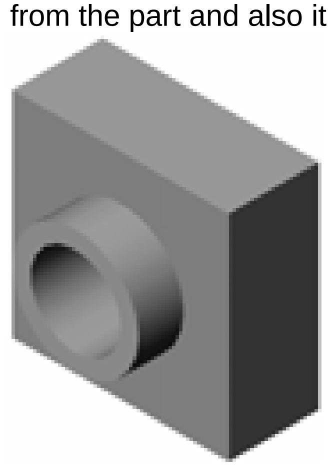
Extruded Boss Feature:

- It Adds material to the part and requires a sketch.



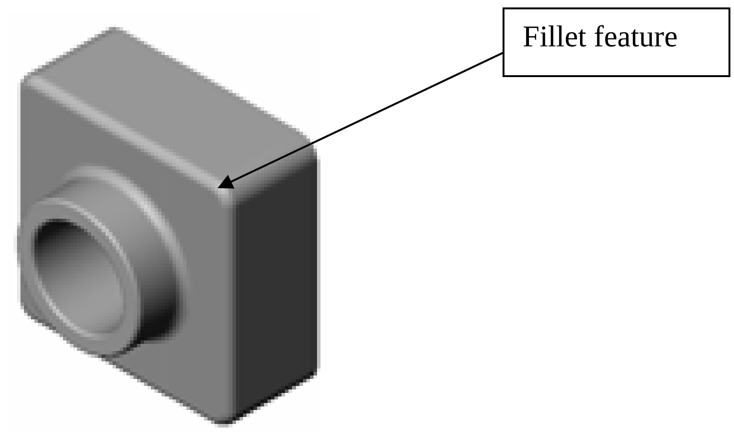
Extruded Cut Feature:

- It Removes material from the part and also it requires a sketch.



Fillet Feature:

- ❖ Rounds the edges or faces of a part to a specified radius.



Procedure:

1. Select a sketch plane.(Front, top or Side)
2. Sketch a 2D profile of the model.
3. Dimension the model using Smart Dimension icon.
4. Check the sketch is fully defined.
5. Extrude the sketch perpendicular to sketch plane.
6. Use extruded cut feature to cut the solid as given in the drawing.

Result:

Thus the given model is extruded.

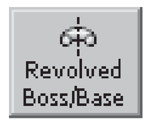
Ex: 2 **Exercise on Revolve**

AIM:

To model the given object using the Revolve feature as per the dimensions given.

Description of Revolve Feature:

Command Manager: Features > Revolved Boss/Base
Menu: Insert > Boss/Base > Revolve

Toolbar: Features > Revolved Boss/Base. 

Using this tool, the sketch is revolved about the revolution axis. The revolution axis could be an axis, an entity of the sketch, or an edge of another feature to create the revolved feature. Note that whether you use a centerline or an edge to revolve the sketch, the sketch should be drawn on one side of the centerline or the edge.

After drawing the sketch, as you choose this tool, you will notice that the sketching environment is closed and the part modeling environment is invoked. Similar to extruding the sketches, the resulting feature can be a solid feature or a thin feature, depending on the sketch and the options selected to be revolved. If the sketch is closed, it can be converted into a solid feature or a thin feature. However, if the sketch is open, it can be converted only into a thin feature.

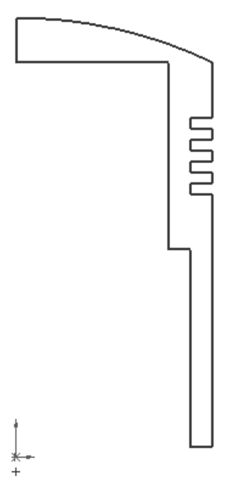


Fig:7 Sketch of piston to be revolve

After you have completed drawing and dimensioning the closed sketch and converted it into fully defined sketch, choose the **Revolved Boss/Base** button from the **Features** toolbar. You will notice that the view is automatically changed to a 3D view, and the **Revolve Property Manager** is displayed,

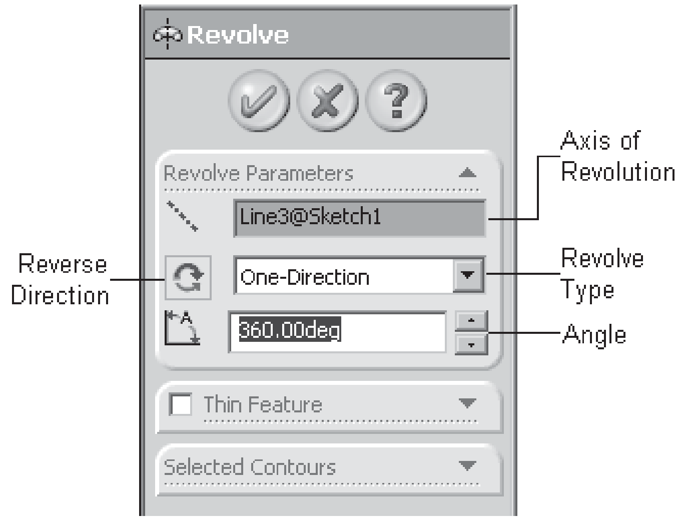


Fig: 8 Revolve Property Manager

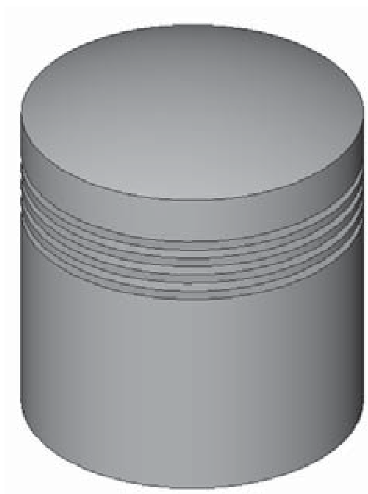


Fig:9 Feature Created after revolving to 360 ^{0}

Procedure:

1. Select a sketch plane.(Front, top or Side)
2. Sketch a 2D profile of the model.
3. Dimension the model using Smart Dimension icon.
4. Check the sketch is fully defined.
5. Revolve the sketch.

Result:

Thus the given model is drawn using revolve feature.

Ex: 3 **Exercise on RIB**

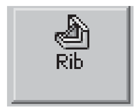
AIM:

To model the given object and construct rib portion in it.

Description of RIB Feature:

Command Manager: Features > Rib
Menu: Insert > Features > Rib

Toolbar: Features > Rib



Ribs are defined as the thin walled structures that are used to increase the strength of the entire structure of the component, so that it does not fail under an increased load. In Solid Works, the ribs are created using an open sketch as well as a closed sketch. To create a rib feature, invoke the **Rib Property Manager** and select the plane on which you need to draw the sketch for creating the rib feature. Draw the sketch and exit the sketching environment. Specify the rib parameters in the **Rib Property Manager** and view the detailed preview using the **Detailed Preview** button. The **Rib** tool is invoked by choosing the **Rib** button from the **Features**

Command Manager or by choosing **Insert > Features > Rib** from the menu bar.

After invoking the **Rib** tool, draw the sketch and exit the sketching environment.

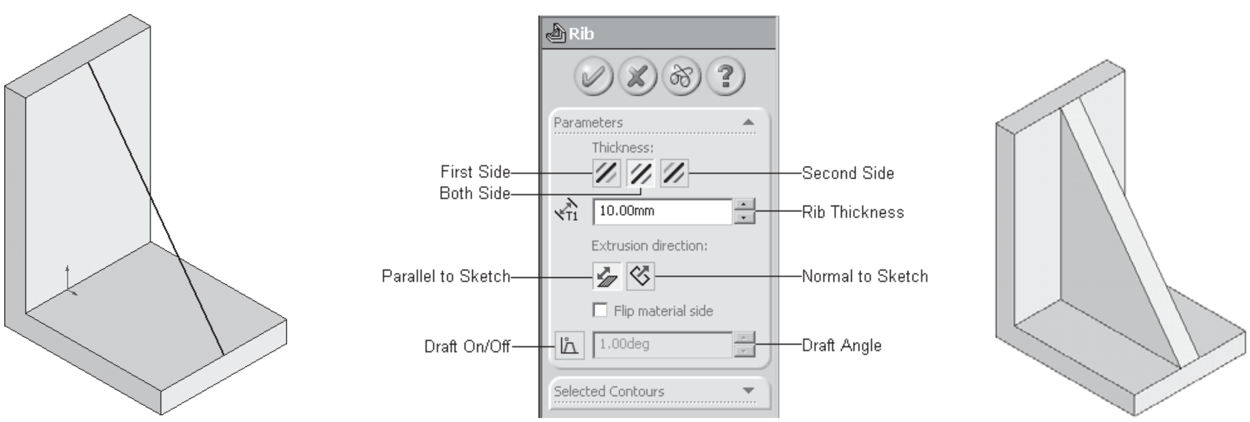


Fig:10 Rib construction procedure

Procedure:

1. Select a sketch plane.(Front, top or Side)
2. Sketch a 2D profile of the model.
3. Dimension the model using Smart Dimension icon.
4. Check the sketch is fully defined.
5. Extrude the sketch.
6. Using Rib Feature complete the model.

Result:

Thus the given model is drawn and completed using rib feature.

Ex: 4 **Exercise on Shell**

AIM:

To model the given object and remove the material using shell option.

Description of SHELL Feature:

- Removes material from the selected face.
- Creates a hollow block from a solid block.
- Very useful for thin-walled, plastic parts.
- You are required to specify a wall thickness when using the shell feature.

Fig: 11 Shell feature

Procedure:

1. Select a sketch plane.(Front, top or Side)
2. Sketch a 2D profile of the model.
3. Dimension the model using Smart Dimension icon.
4. Check the sketch is fully defined.
5. Extrude the sketch.
6. Select the face in which you are going to draw the cut profile.
7. Make that plane to normal to you.
8. Sketch the cut profile & dimension it.
9. Use Extruded cut feature remove the portion.
10. Select the Shell feature.
11. Select the face in which material to be removed using shell.
12. Specify the shell thickness.

Result:

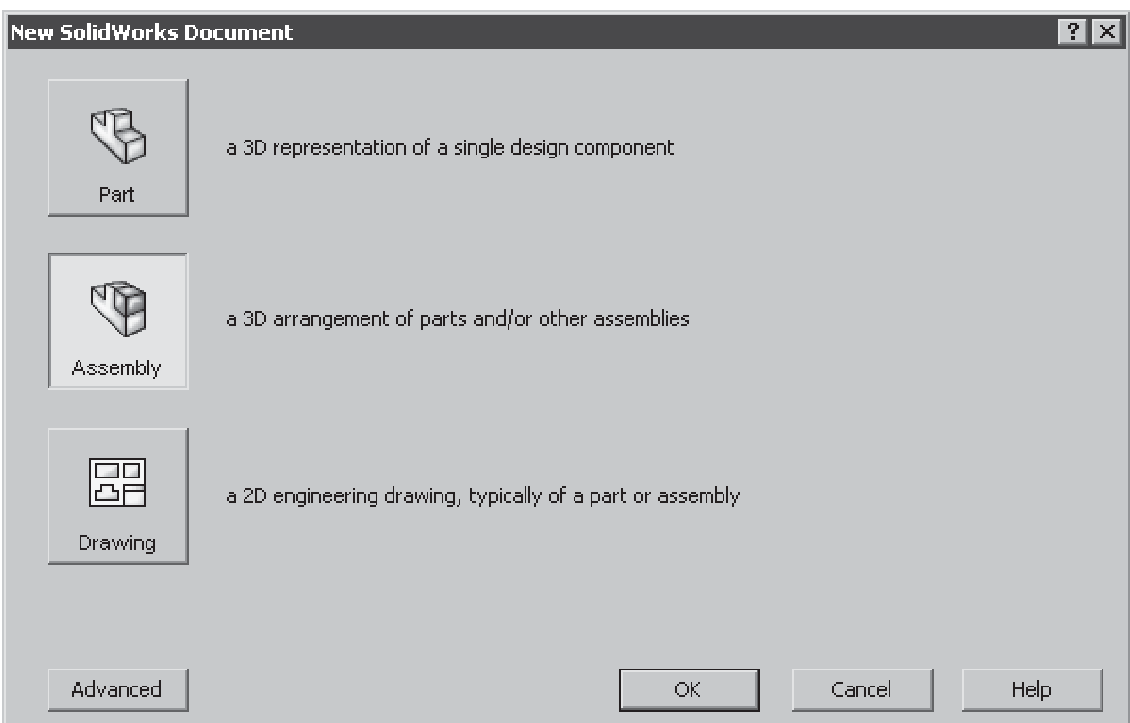
Thus the given model is drawn and completed using shell feature.

ASSEMBLY IN SOLID WORKS:

An assembly design consists of two or more components assembled together at their respective work positions using the parametric relations. In SolidWorks, these relations are called mates. These mates allow you to constrain the degrees of freedom of the components at their respective work positions.

To proceed to the **Assembly** mode of SolidWorks, invoke the **New SolidWorks**

Document dialog box and choose the **Assembly** button as shown in Figure below. Choose the **OK** button to create a new assembly document; a new SolidWorks document will be started in the **Assembly** mode



PLACING COMPONENTS IN THE ASSEMBLY DOCUMENT:

Command Manager: Assemblies > Insert Components

Menu: Insert > Component > Existing Part/Assembly

Toolbar: Assembly > Insert Components

When you start a new SolidWorks document in the **Assembly** mode, the **Insert Component Property Manager** will be displayed as below which enable us to insert the component into assembly file.

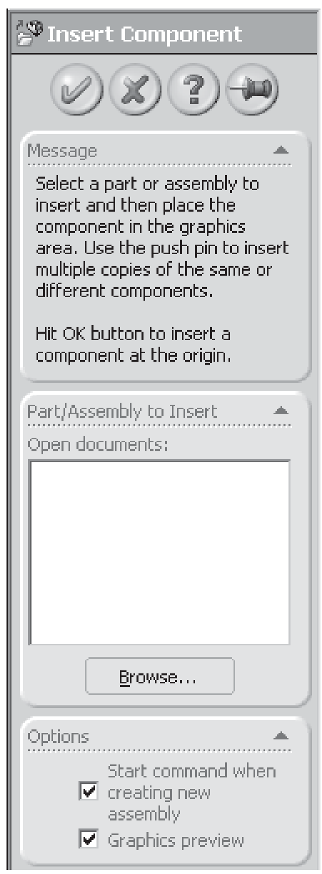
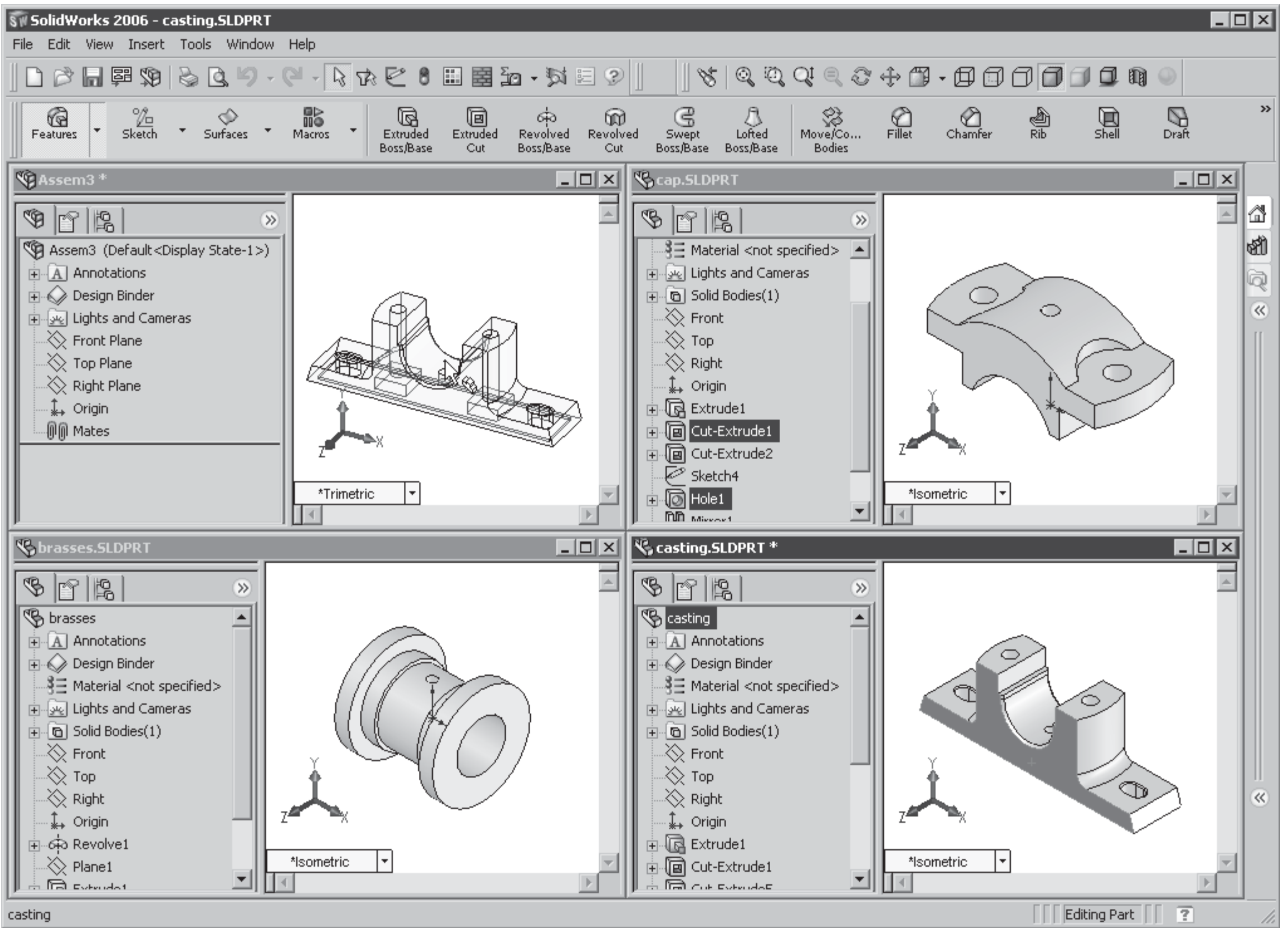


Fig:12 Insert component Manager

Place the component one by one in the proper fashion in the new assembly file.



ASSEMBLING COMPONENTS:

After placing the components in the assembly document, you need to assemble them. By assembling the components, you will constrain their degrees of freedom. As mentioned earlier, the components are assembled using mates. Mates help you precisely place and position the component with respect to the other components and the surroundings in the assembly. You can also define the linear and rotatory movement of the component with respect to the other components. In addition, you can create a dynamic mechanism and check the stability of the mechanism by precisely defining the combination of mates. There are two methods of adding mates to the assembly. The first method is using the **Mate Property Manager** and the second and the most widely used method of adding mates to the assembly is using the **Smart Mates.**

Ex: 5 **Exercises on Assembly of Flange Coupling**

AIM:

To model and assemble the flange coupling as per the dimensions given and also convert the 3D model into different vies with Bill of materials.

Description about Flange Coupling:

A flange coupling is the simplest type of rigid coupling most extensively used in the general power transmission application. It consist of two C.I or steel bosses projected flange plates at on of their ends. The flange plates are drilled with a number of equidistant bolt holes on their flat faces with their centers lying on a imaginary circle called “pitch Circle”. Each of the flange bosses is securely keyed to the end of each shaft using a tapered key driven from inside. While assembling generally two flanges are set such that the keys fitted in them are out of alignment by 90 ^{0} to each other, then the flanges are bolted together by a number of bolts and nuts. Power is transmitted from one shaft to other through the bolts. These bolts are in close running fit in the holes which are drilled and placed in the flanges in order that the load is taken smoothly with out any impact which would take place if the bolts are fitted loose in the holes.. Correct alignment of the two shafts is assured irrespective of the bolts, by allowing the end of the shaft to another a small distances in bosses bore of the other flange.

Procedure:

1. Model different parts of a flange coupling using Extrude, Revolve etc., features.
2. Select the assembly in solid works main menu.
3. Using Insert component icon of property manager, insert base component & next components to be assemble.
4. Assemble using MATE Feature.
5. Continue the inserting the component & mating until the entire component are assembled.
6. Save the assembly.
7. From the main menu of solid works select the drawing option.

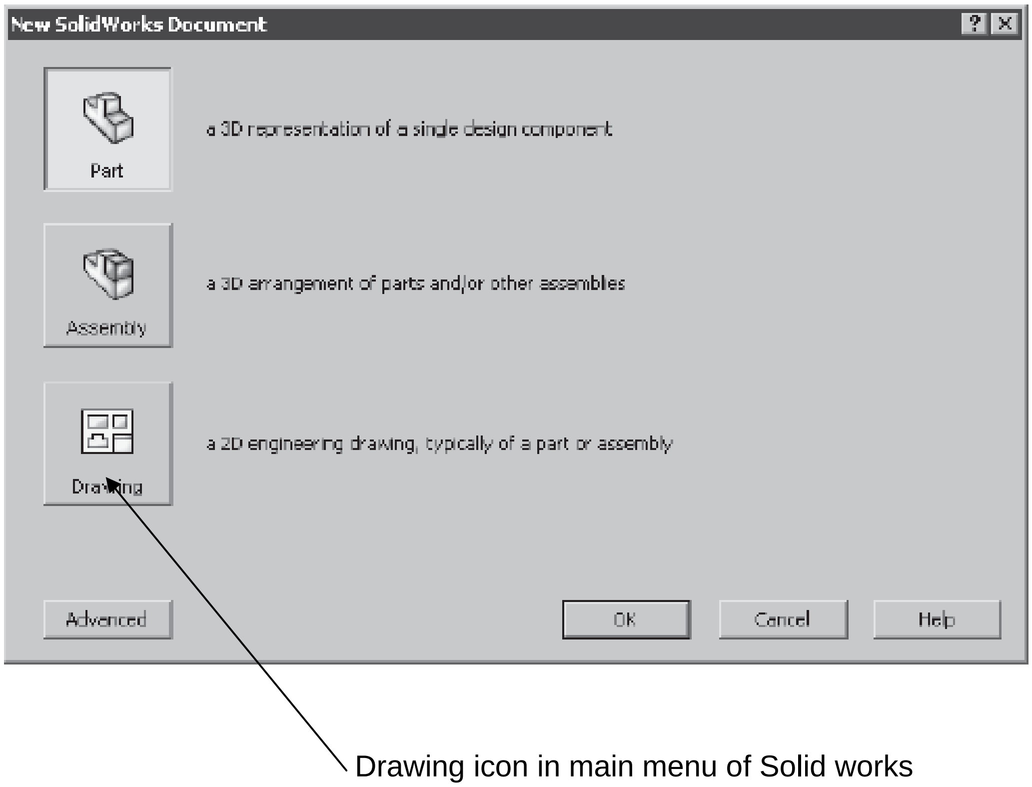
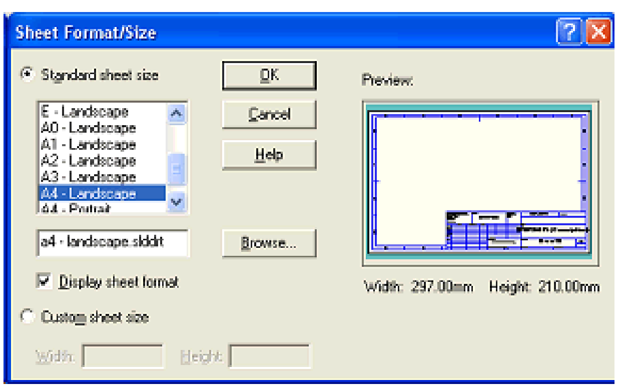
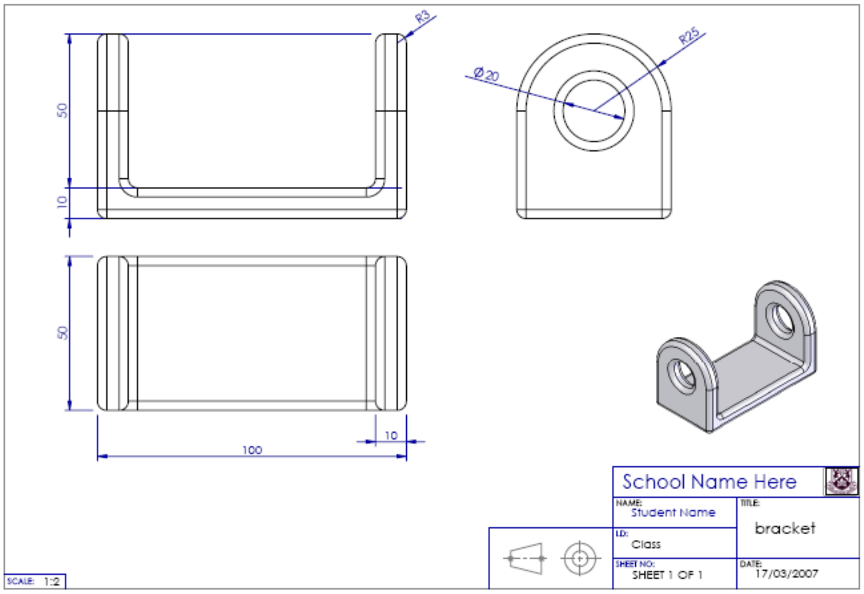


Fig:13 Solid works Main menu

8. Select the drawing sheet format size as – A4 Landscape.



9. Using the model view manager browse the document to be open.



10. Click the view orientation from the model view manager & place the drawing view in the proper place in the sheet as shown above.
11. Using the placed view as parent view project the other or needed views
12. Move cursor to any one view and right click the mouse button.
13. Select the Table – BOM.
14. Place the BOM in the proper place in the drawing sheet.
15. Save the drawing sheet.

Result:

Thus the given flange coupling is modeled, assembled & different views are taken.

Ex:6 Exercises on Assembly of Screw Jack

AIM:

To model and assemble the Screw jack as per the dimensions given and also convert the 3D model into different vies with Bill of materials.

Description about Screw jack:

A Screw Jack, manually operated is a contrivance to lift heavy object over a small height with a distinct Mechanical Advantages. It also serves as a supporting aid in the raised position. A screw Jack is actuated by a square threaded screw worked by applying a moderate effort at the end of a Tommy bar inserted into the hole of the head of the screw.

The body of the screw jack has an enlarged circular base which provides a large bearing area. A gun metal nut is tight fitted into the body at the top. A screw spindle is screwed through the nut. A load bearing cup is mounted at the top of the screw spindle and secured to it by a washer and a CSK screw. When the screw spindle is rotated, the load bearing cup moves only up or down along with the screw spindle but will not rotate with it. The Tommy bar is inserted into the hole in the head of the screw spindle only during working and will be detached when not in use.

Procedure:

1. Model different parts of a Screw Jack using Extrude, Revolve etc., features.
2. Select the assembly in solid works main menu.
3. Using Insert component icon of property manager, insert base component & next components to be assemble.
4. Assemble using MATE Feature.
5. Continue the inserting the component & mating until the entire component are assembled.
6. Save the assembly.
7. From the main menu of solid works select the drawing option.
8. Drawing icon in main menu of Solid works
9. Select the drawing sheet format size as – A4 Landscape.

10. Using the model view manager browse the document to be open.
11. Click the view orientation from the model view manager & place the drawing view in the proper place in the sheet.
12. Using the placed view as parent view project the other or needed views
13. Move cursor to any one view and right click the mouse button.
14. Select the Table – BOM.
15. Place the BOM in the proper place in the drawing sheet.
16. Save the drawing sheet.

Result:

Thus the given Screw Jack is modeled, assembled & different views are taken

Ex: 6 **Exercises on Assembly of Strap joint with GIB and Cotter**

AIM:

To model and assemble the strap joint of Gib & cotter as per the dimensions given and also convert the 3D model into different vies with Bill of materials.

Description about Gib & Cotter Joint:

When the rods of square or rectangular cross sections subjected to axial forces have to be connected temporarily, a strap joint is used. In this type of cotter joint, the end of one of the rods is formed into a fork into which the end of the other rod fits. The forked end of the rod is called STRAP. Since the strap is open on one side, if only a cotter is used to connect the two rods as explained earlier and when t he rods are subjected to axial forces, the end of the strap opens out. To prevent the opening out of the ends of the strap, a gib is used in conjunction with the cotter. The gib is a wedge shaped piece of steel of rectangular in cross section with one side tapered and the other straight and has two projections, called gib-heads. These gib heads act like hooks prevent and prevent the opening out of the ends of the straps. The use of gib along with the cotter facilitates the cutting of the slots with straight faces.

Procedure:

1. Model different parts of a gib & cotter joint using Extrude, Revolve etc., features.
2. Select the assembly in solid works main menu.
3. Using Insert component icon of property manager, insert base component & next components to be assemble.
4. Assemble using MATE Feature.
5. Continue the inserting the component & mating until the entire component are assembled.
6. Save the assembly.
7. From the main menu of solid works select the drawing option.
8. Drawing icon in main menu of Solid works
9. Select the drawing sheet format size as – A4 Landscape.
10. Using the model view manager browse the document to be open.
11. Click the view orientation from the model view manager & place the drawing view in the proper place in the sheet.

12. Using the placed view as parent view project the other or needed views
13. Move cursor to any one view and right click the mouse button.
14. Select the Table – BOM.
15. Place the BOM in the proper place in the drawing sheet.
16. Save the drawing sheet.

Result:

Thus the given strap joint of Gib & cotter is modeled, assembled & different views are taken.

CAM LAB EXERCISES

CNC – Lathe

List of G – codes

G00 – Rapid Traverse

G01 – Linear interpolation

G02 – Circular interpolation – clockwise

G03 – Circular interpolation – counter clockwise

G21 – Dimensions are in mm

G28 – Home position

G40 – Compensation Cancel

G50 – Spindle speed clamp

G70 – Finishing cycle

G71 – Multiple turning cycle

G75 – Multiple grooving cycle

G76 – Multiple threading cycle

G90 – Box turning cycle

G98 – Feed in mm/min

List of M-codes

M03 – Spindle ON in clockwise direction

M05 – Spindle stop

M06 – Tool change

M10 – Chuck open

M11 – Chuck close

M30 – Program stop and rewind

M38 – Door open

M39 – Door close

BOX TURNING CYCLE

Exercise : 1
Date :

Aim:

To write the manual part program to the given dimensions and execute in CNC Lathe. (G90).

Material required:

Material : Aluminium
Size : Diameter 25mm and Length 50mm

Program:

```
[BILLET X25 Z50;
G21 G98 G40;
G28 U0 W0;
G50 S2000;
M06 T01;
M03 S1200;
G00 X26 Z1;
G90 X24 Z-30 F45;
     X23;
     X22;
     X21;
     X20;
G00 X21 Z1;
G90 X19 Z-10 F45;
     X18;
     X17;
     X16;
     X15;
     X14;
     X13;
     X12;
     X11;
     X10;
M05;
G28 U0 W0;
M30;
```

Result:

Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

MULTIPLE TURNING CYCLE

Exercise : 2
Date :

Aim:

To write the manual part program to the given dimensions and execute in CNC Lathe. (G71).

Material required:

Material : Aluminium
Size : Diameter 25mm and Length 50mm

Program:

```
[BILLET X25 Z50;
G21 G98 G40;
G28 U0 W0;
G50 S2000;
M06 T01;
M03 S1200;
G00 X26 Z1;
G71 U0.5 R1;
G71 P100 Q200 U0.1 W0.1 F45;
N100 G01 X0;
Z0;
G03 X10 Z-5 R5;
G01 X10 Z-15;
G02 X20 Z-25 R10;
G01 X20 Z-30;
N200 G01 X25 Z-40;
M03 1500;
G70 P100 Q200 F25;
M05;
G28 U0 W0;
M30;
```

Note:

G71 U0.5 R1
Where,
U0.5 – depth of cut in mm
R1 - relief in mm

G71 P100 Q200 U0.1 W0.1 F45;
Where,
P100 – first line number
Q200 – last line number
U0.1 – finishing allowance in x-axis
W0.1 – finishing allowance in z-axis

G70 – Finishing cycle between first and last line number.

Conditions:

- In the first line number only G01 and X codes must only be written.
- Z code for the first coordinate must be written in the next line.
- G71 will not work for left downward taper.
- Between G71 cycle only G01, G02 and G03 must be written.

Result:

Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

TAPER TURNING CYCLE

Exercise : 3
Date :

Aim:

To write the manual part program to the given dimensions and execute in CNC Lathe. (G90).

Material required:

Material : Aluminium
Size : Diameter 25mm and Length 50mm

Program:

```
[BILLET X25 Z50;
G21 G98 G40;
G28 U0 W0;
G50 S2000;
M06 T01;
M03 S1200;
G00 X26 Z1;
G90 X24 Z-39 F45;
X23;
G00 X24 Z1;
G90 X22 Z-34 F45;
    X21;
    X20;
G00 X21 Z1;
G90 X19 Z-10 F45;
    X18;
    X17;
    X16;
    X15;
    X14;
    X13;
    X12;
    X11;
    X10;
G00 X11 Z1;
G90 X10 Z-10 R-0.5 F40;
    X10 R-1;
    X10 R-1.5;
    X10 R-2;
    X10 R-2.5;
    X10 R-3;
    X10 R-3.5;
    X10 R-4;
    X10 R-4.5;
    X10 R-5;
G00 X21 Z-10;
```

Note:

$$R = \frac{\text{Initial dia.} - \text{Final dia.}}{2}$$

```
G00 X20 Z-15;
G90 X19 Z-22 R0.5 F40;
    X18 R1;
    X17 R1.5;
    X16 R2;
    X15 R2.5;
G00 X21 Z-21;
G90 X20 Z-29 R-0.5 F40;
    X20 R-1;
    X20 R-1.5;
    X20 R-2;
    X20 R-2.5;
G00 X25 Z1;
M05;
G28 U0 W0;
M30;
```

Result:

Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

MULTIPLE GROOVING CYCLE

Exercise : 4
Date :

Aim:

To write the manual part program to the given dimensions and execute in CNC Lathe. (G75).

Material required:

Material : Aluminium
Size : Diameter 40mm and Length 55mm

Program:

```
[BILLET X40 Z55;
G21 G98 G40;
G28 U0 W0;
G50 S2000;
M06 T01;
M03 S1200;
G00 X41 Z1;
G71 U0.5 R1;
G71 P100 Q200 U0.1 W0.1 F45;
N100 G01 X16;
Z0;
G01 X30 Z-2;
G01 X30 Z-35;
N200 G01 X40 Z-45;
G28 U0 W0;
M06 T02;
M03 S700;
G00 X31 Z-17;
G75 R1;
G75 X24 Z-30 P1000 Q1750 F10;
G01 X33;
M05;
G28 U0 W0;
M30;
```

Note:
G75 R1
Where,
R1 – relief in mm

G75 X24 Z-30 P1000 Q1750 F10;
Where,
X24 – minor dia. of groove
Z-30 – final point in length
P1000 – increment in X- axis in microns
Q1750 – increment in Z- axis in microns

Result:

Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

MULTIPLE THREADING CYCLE

Exercise : 5
Date :

Aim:

To write the manual part program to the given dimensions and execute in CNC Lathe. (G76).

Material required:

Material : Aluminium
Size : Diameter 40mm and Length 55mm

Program:

```
[BILLET X40 Z55;
G21 G98 G40;
G28 U0 W0;
G50 S2000;
M06 T01;
M03 S1200;
G00 X41 Z1;
G71 U0.5 R1;
G71 P100 Q200 U0.1 W0.1 F45;
N100 G01 X16;
Z0;
G01 X30 Z-2;
G01 X30 Z-35;
N200 G01 X40 Z-45;
G28 U0 W0;
M06 T02;
M03 S700;
G00 X31 Z-17;
G75 R1;
G75 X24 Z-30 P1000 Q1750 F10;
G01 X33;
G00 Z1;
G28 U0 W0;
M06 T03;
M03 S350;
G00 X31 Z1;
G76 P031560 Q50 R0.1;
G76 X27.546 Z-16 P1227 Q60 F2;
M05;
G28 U0 W0;
M30;
```

Note:
G76 P031560 Q50 R0.1
Where,
P031560 -
03 – no. of finishing passes
15 – pull out angle
60 – angle of thread
Q50 – depth of cut in microns
R0.1 – finishing allowance

G76 X27.546 Z-16 P1227 Q60 F2;
Where,
X27.546 – core diameter for M30x2 fine series
Z-16 – length of thread
P1227 – depth of thread in microns
Q60 – first depth of cut
F2 – pitch of the thread

Result:

Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

CNC – Milling

List of G – codes

G00 – Rapid Traverse
G01 – Linear interpolation
G02 – Circular interpolation – clockwise
G03 – Circular interpolation – counter clockwise
G21 – Dimensions are in mm
G28 – Home position
G40 – Compensation Cancel
G50 – Spindle speed clamp
G83 – Peck drilling cycle
G90 – Absolute coordinate system
G91 – Incremental coordinate system
G94 – Feed in mm/min
G170, G171 – Circular Pocketing
G172, G173 – Rectangular Pocketing

List of M-codes

M03 – Spindle ON in clockwise direction
M05 – Spindle stop
M06 – Tool change
M10 – Chuck open
M11 – Chuck close
M30 – Program stop and rewind
M38 – Door open
M39 – Door close
M70 – Mirroring ON in X-axis
M71 - Mirroring ON in Y-axis
M80 – Mirroring OFF in X-axis
M81 – Mirroring OFF in Y-axis
M98 – Sub program call statement
M99 – Sub program terminate

LINEAR AND CIRCULAR INTERPOLATION

Exercise : 6
Date :

Aim:

To write the manual part program to the given dimensions and execute in CNC Milling.

Material required:

Material : Acrylic sheet
Size : Length 100mm, Width 100mm and Thickness 5mm

Program:

```
[BILLET X100 Y100 Z5;
[EDGEMOVE X0 Y0;
[TOOLDEF T1 D5;
G21 G94 G40;
G91 G28 Z0;
G28 X0 Y0;
G28
M06 T01;
M03 S1500;
G90 G00 X0 Y0 Z5;
G00 X25 Y10;
G01 Z-2 F40;
G03 X10 Y25 R15;
G01 X10 Y75;
G02 X25 Y90 R15;
G01 X75 Y90;
G03 X90 Y75 R15;
G01 X90 Y25;
G02 X75 Y10 R15;
G01 X25 Y10;
G01 Z5;
M05;
G91 G28 X0 Y0 Z0;
M30;
```

Result:

Thus the manual part program was written to the given dimensions and executed in CNC Milling.

CIRCULAR POCKETTING

Exercise : 7
Date :

Aim:

To write the manual part program to the given dimensions and execute in CNC Milling.

Material required:

Material : Acrylic sheet
Size : Length 100mm, Width 100mm and Thickness 5mm

Program:

```
[BILLET X100 Y100 Z5;
[EDGEMOVE X-50 Y-50;
[TOOLDEF T1 D5;
G21 G94 G40;
G91 G28 Z0;
G28 X0 Y0;
M06 T01;
M03 S1500;
G90 G00 X0 Y0 Z5;
G01 Z0 F300;
G170 R0 P0 Q1 X0 Y0 Z-3 I0.5 J0.1 K-25;
G171 P75 S2500 R75 F250 B3500 J200;
G00 Z5;
M05;
G91 G28 X0 Y0 Z0;
M30;
```

Note:
G170 R0 P0 Q1 X0 Y0 Z-3 I0.5 J0.1 K-25;
Where,
R0 – reference point
P0 – roughing; P1 – finishing
Q1 – depth of each cut
X,Y – center coordinate of circle measured from datum point
Z-3 – total depth
I0.5 – finishing allowance at side
J0.5 – finishing allowance at bottom
K-25 – radius of pocket

G171 P75 S2500 R75 F250 B3500 J200;
Where,
P75 – percentage of cut
S2500 – speed
R75 – feed in Z-axis
F250 – feed in X and Y axis
B3500 – finishing speed
J200 – finishing feed

Result:

Thus the manual part program was written to the given dimensions and executed in CNC Milling.

RECTANGULAR POCKETTING

Exercise : 8
Date :

Aim:

To write the manual part program to the given dimensions and execute in CNC Milling.

Material required:

Material : Acrylic sheet
Size : Length 100mm, Width 100mm and Thickness 5mm

Program:

```
[BILLET X100 Y100 Z5;
[EDGEMOVE X-50 Y-50;
[TOOLDEF T1 D5;
G21 G94 G40;
G91 G28 Z0;
G28 X0 Y0;
M06 T01;
M03 S1500;
G90 G00 X0 Y0 Z5;
G01 Z0 F300;
G172 I-60 J-50 K0 P0 Q1 R0 X-30 Y-25 Z-3;
G173 I0.5 K0.1 P75 T1 S2500 R75 F250 B3000 J200 Z5;
G00 Z25;
M05;
G91 G28 X0 Y0 Z0;
M30;
```

Note:
G172 I-60 J-50 K0 P0 Q1 R0 X-30 Y-25 Z-3;
Where,
I – length of the rectangle in X-axis
J - width of the rectangle in Y-axis
K – corner radius
P0 – roughing: P1 – finishing
Q – depth of each cut
R – reference point
X,Y – left downward coordinate
Z – total depth of cut

Note:
G173 I0.5 K0.1 P75 T1 S2500 R75 F250 B3000 J200 Z5;
Where,
I&K – finishing allowance at side & base
P – percentage of cut
T – tool
S- speed
R – feed in Z-axis
F – feed in X and Y axis
B&J – finishing speed&feed
Z – safe height

Result:

Thus the manual part program was written to the given dimensions and executed in CNC Milling.

PECK DRILLING

Exercise : 9
Date :

Aim:

To write the manual part program to the given dimensions and execute in CNC Milling.

Material required:

Material : Acrylic sheet
Size : Length 100mm, Width 100mm and Thickness 5mm

Program:

```
[BILLET X100 Y100 Z5;
 [EDGEMOVE X0 Y0;
[TOOLDEF T1 D5;
G21 G94 G40;
G91 G28 Z0;
G28 X0 Y0;
M06 T01;
M03 S1500;
G90 G00 X25 Y25 Z5;
G83 G99 X25 Y25 Z-3 Q1 R2 F200;
X75 Y25;
X50 Y50;
X25 Y75;
G98 X75 Y75;
G80;
G00 Z25;
M05;
G91 G28 X0 Y0 Z0;
M30;
```

Note:
G83 G99 X25 Y25 Z-3 Q1 R2 F200;
Where,
G83 – peck drilling cycle
G99 – return to R in canned cycle
X&Y – first drill coordinate
Z-3 – total depth of cut
Q1 – depth of each cut
R2 – starting point of drilling cycle in Z-axis
F – feed in Z-axis

G98 – return to initial point in canned cycle
G80 – canned cycle cancel

Result:

Thus the manual part program was written to the given dimensions and executed in CNC Milling.

MIRRORING

Exercise : 10
Date :

Aim:

To write the manual part program to the given dimensions and execute in CNC Milling.

Material required:

Material : Acrylic sheet
Size : Length 100mm, Width 100mm and Thickness 5mm

Program:

```
[BILLET X100 Y100 Z5;
[EDGEMOVE X-50 Y-50;
[TOOLDEF T1 D5;
G21 G94 G40;
G91 G28 Z0;
G28 X0 Y0;
G50 S3000;
M06 T1;
M03 S2000;
G01 Z0 F300;
G90 G00 X10 Y10 Z5;
M98 P0011000;
M70;
M98 P0011000;
M80;
M71;
M98 P0011000;
M70;
M98 P0011000;
M80;
M05;
G90 G28 X0 Y0 Z0;
M30;
```

Note:
M98 P0011000;
Where,
M98 – sub program call
P0011000 –
P001 means Number of times to repeat
1000 means Sub program file name

M70 – Mirroring ON in X-axis
M71 - Mirroring ON in Y-axis
M80 – Mirroring OFF in X-axis
M81 – Mirroring OFF in Y-axis
M99 – Sub program terminate

Sub program (file name : 1000)

```
G90 X10 Y10 Z5;
G01 Z-3;
X40 Y10;
X25 Y40;
X10 Y10;
G00 X0 Y0 Z5;
M99;
```

Result:

Thus the manual part program was written to the given dimensions and executed in CNC Milling.