A manual on

FreeCAD

(An open source software)

Academy of Technology

Hooghly, 712121

Part -2

(For 2nd Sem)

Course objective:

- To make the students understand and interpret drawings of machine components
- To gain practical experience in handling 2D drafting and 3D modeling software systems.
- To prepare the foundation for CAD/CAM and additive manufacturing.

The topics to be covered in

Engineering Graphics & Design (ES-ME291) and Workshop/Manufacturing Practices (ES-ME292) Laboratories:

1. Part modelling and drafting of a support using FreeCAD software.
2. Part modelling and drafting of a Bracket using FreeCAD software.
3. Part modelling and drafting of a Bracket using FreeCAD software.
4. Part modelling and drafting of a Bearing support using FreeCAD software.
5. Part modelling and drafting of a Bearing support using FreeCAD software.
6. Part modelling and drafting of a Bearing support using FreeCAD software.

Course Outcomes:

Upon the completion of this course the students will be able to

CO1 model different basic components used to provide support.

CO2 develop basic concepts in part modelling and constructing basic geometries.

CO3 create the drawing of parts using software

1.1.Introduction

FreeCAD is a free, open-source parametric 3D modeling application. Parametric is a term used to describe a dimension's ability to change the shape of model geometry as soon as the dimension value is modified. For e.g. the parametric modeling describes a certain type of modeling, where the shape of the 3D objects you design are controlled by parameters. For example, the shape of a brick might be controlled by three parameters, such as height, width and length. Feature-based is a term used to describe the various components of a model. For example, a part can consists of various types of features such as holes, grooves, fillets, and chamfers. A 'feature' is the basic unit of a parametric solid model. It is made primarily to model real-world objects, ranging from the small electronic components up to buildings and civil engineering projects, with a strong focus on 3D-printable objects. The data you produce with FreeCAD is fully yours, and can be recovered without FreeCAD. FreeCAD is also fundamentally a social project, as it is developed and maintained by a community of developers and users united by their passion for FreeCAD. FreeCAD is also multiplatform (it runs exactly the same way on Windows, Mac OS and Linux platforms).

The official website of FreeCAD is at http://www.freecadweb.org

Part, Sketcher, Part Design and Drawing and dimensioning workbenches of FreeCAD will be used in FreeCAD, Part 2 for students of second semester. The workbenches to be used in 2nd semester, have been discussed below.

1.1.1. Part

The Part Workbench provides basic tools (Table 1) for working with solid parts: primitives, such as cube and sphere, and simple geometric operations and boolean operations. Being the main anchor point with OpenCasCade, the Part workbench provides the foundation of FreeCAD's geometry system, and almost all other workbenches produce Part-based geometry.

Table 1: Tools in Part Workbench

Tool	Description	Tool	Description
Box	Draws a box	Cone	Draws a cone
Cylinder	Draws a cylinder	Sphere	Draws a sphere
Torus	Draws a torus (ring)	Create Primitives	Creates various other parametric geometric primitives
Shape Builder	Create more complex shapes from primitives	Fuse	Fuses (unions) two objects
Common	Extracts the common (intersection) part of two objects	Cut	Cuts (subtracts) one object from another
Join Connect	Connects interiors of walled objects	Join Embed	Embeds a walled object into another walled object
Join Cutout	Creates a cutout in a wall of an object for another walled object	Extrude	Extrudes planar faces of an object
Fillet	Fillets (rounds) edges of an object	Revolve	Creates a solid by revolving another object (not solid) around an axis
Section	Creates a section by intersecting an object with a section plane	Section Cross	Creates multiple cross sections along an object
Chamfer	Chamfers edges of an object	Mirror	Mirrors the selected object on a given mirror plane
Ruled Surface	Create a ruled surface between selected curves	Sweep	Sweeps one or more profiles along a path
Loft	Lofts from one profile to another	Offset	Creates a scaled copy of the original object
Thickness	Assign a thickness to the faces of a shape		

1.1.2. Sketcher

The Sketcher Workbench contains tools (Table 2) to build and edit complex 2D objects, called sketches. The geometry inside these sketches can be precisely positioned by the use of constraints. They are meant primarily to be the building blocks of Part Design geometry, but are useful everywhere in FreeCAD.

Table 2: Tools in Sketcher Workbench

Tool	Description	Tool	Description
Point	Draws a point	Line by 2 points	Draws a line segment from 2 points
Arc	Draws an arc segment from center, radius, start angle and end angle	Arc by 3 points	Draws an arc segment from two endpoints and another point on the circumference
Circle	Draws a circle from center and radius	Circle by 3 points	Draws a circle from three points on the circumference
	Draws an ellipse by center point, major radius point and	Ellipse	Draws an ellipse by major diameter (2

center	minor radius point	by 3 points	points) and minor radius point
Arc of ellipse	Draws an arc of ellipse by center point, major radius point, starting point and ending point	Polyline	Draws a line made of multiple line segments. Several drawing modes available
Rectangle	Draws a rectangle from 2 opposite points	Triangle	Draws a regular triangle inscribed in a construction geometry circle
Square	Draws a regular square inscribed in a construction geometry circle	Pentagon	Draws a regular pentagon inscribed in a construction geometry circle
Hexagon	Draws a regular hexagon inscribed in a construction geometry circle	Heptagon	Draws a regular heptagon inscribed in a construction geometry circle
Octagon	Draws a regular octagon inscribed in a construction geometry circle	Slot	Draws an oval by selecting the center of one semicircle and an endpoint of the other semicircle
Fillet	Makes a fillet between two lines joined at one point	Trim	Trims a line, circle or arc with respect to a clicked point

External Geometry	Creates an edge linked to external geometry	Construction Mode	Toggles an element to/from construction mode. A construction object will not be used in a 3D geometry operation and is only visible while editing the Sketch that contains it
Coincident constraint	Affixes a point onto (coincident with) one or more other points.	Point On Object constraint	Affixes a point onto another object such as a line, arc, or axis.
Vertical constraint	Constrains the selected lines or polyline elements to a true vertical orientation. More than one object can be selected before applying this constraint.	Horizontal constraint	Constrains the selected lines or polyline elements to a true horizontal orientation. More than one object can be selected before applying this constraint.

			constraint.
Parallel constraint	Constrains two or more lines parallel to one another.	Perpendicular constraint	Constrains two lines perpendicular to one another, or constrains a line perpendicular to an arc endpoint.
Tangent constraint	Creates a tangent constraint between two selected entities, or a co-linear constraint between two line segments.	Equal Length constraint	Constrains two selected entities equal to one another. If used on circles or arcs their radii will be set equal.
Symmetric constraint	Constrains two points symmetrically about a line, or constrains the first two selected points symmetrically about a third selected point.	Lock constraint	Constrains the selected item by setting vertical and horizontal distances relative to the origin, thereby locking the location of that item

Horizontal Distance constraint	Fixes the horizontal distance between two points or line endpoints. If only one item is selected, the distance is set to the origin.	Vertical Distance constraint	Fixes the vertical distance between 2 points or line endpoints. If only one item is selected, the distance is set to the origin.
Length constraint	Defines the distance of a selected line by constraining its length, or defines the distance between two points by constraining the distance between them.	Radius constraint	Defines the radius of a selected arc or circle by constraining the radius.
Internal Angle constraint	Defines the internal angle between two selected lines.	Snell's Law constraint	Constrains two lines to obey a refraction law to simulate the light going through an interface
Internal Alignment constraint	Aligns selected elements to selected shape (e.g. a line to become major axis of an ellipse)	Map sketch to face	Maps a sketch to the previously selected face of a solid
Merge	Merge two or more sketches	Mirror	Mirrors selected elements of a sketch

1.1.3. Part Design

The Part Design Workbench contains advanced tools to build solid parts. It also contains all the tools from the sketcher. Since it can only produces solid shapes, it is the main workbench to use when designing parts to be manufactured or 3D-printed, as you will always obtain a printable object. The tools of Part Design workbench have been shown in Table 3.

Table 3: Tools in Part Design Workbench

Tool	Description	Tool	Description
Pad	Extrudes a solid object from a selected sketch	Pocket	Creates a pocket from a selected sketch. The sketch must be mapped to an existing solid object's face
Revolution	Creates a solid by revolving a sketch around an axis	Groove	Creates a groove by revolving a sketch around an axis
Fillet	Fillets (rounds) edges of an object	Chamfer	Chamfers edges of an object
Draft	Applies angular draft to faces of an object	Mirrored	Mirrors features on a plane or face
Linear Pattern	Creates a linear pattern of features	Polar Pattern	Creates a polar pattern of features
Scaled	Scales features to a different size	MultiTransform	Allows creating a pattern with any combination of the other transformations
Shaft wizard	Generates a shaft from a table of values and allows to analyze forces and moments	Involute Gear wizard	Allows you to create several types of gears

1.1.4. Drawing

The Drawing Workbench handles the creation and manipulation of 2D drawing sheets, used for displaying views of your 3D work in 2D. These sheets can then be exported to 2D applications in SVG or DXF formats, to a PDF file. The tools have been shown in Table 3.

Table 3: Tools in Drawing Workbench

Tool	Description	Tool	Description
New sheet	Creates a new drawing sheet	Insert view	Inserts a view of the selected object in the active drawing sheet
Annotation	Adds an annotation to the current drawing sheet	Clip	Adds a clip group to the current drawing sheet
Browser preview	Opens a preview of the current sheet in the browser	Ortho Views	Automatically creates orthographic views of an object on the current drawing sheet
Symbol	Adds the contents of a SVG file as a symbol on the current drawing sheet	Draft View	Inserts a special Draft view of the selected object in the current drawing sheet
Export	Saves the current sheet as a SVG file		

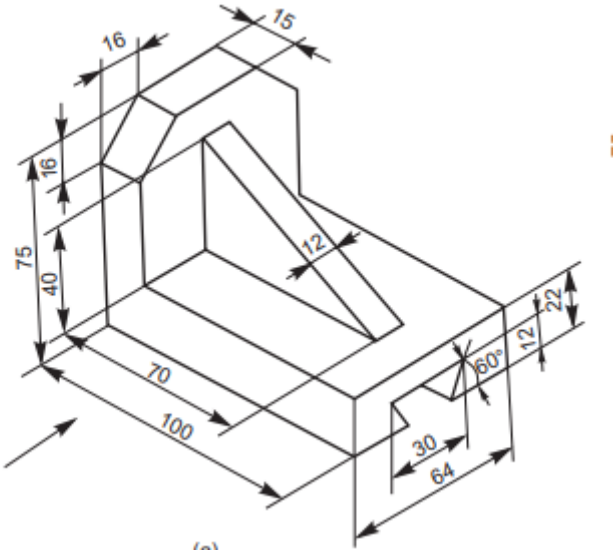
1.2.External workbenches

A number of other very useful workbenches produced by FreeCAD community members also exist. Although they are not included in a standard FreeCAD installation, they are easy to install as plug-ins. They are all referenced in the FreeCAD-addons repository. Among the most developed are:

- The Drawing Dimensioning Workbench: This offers many new tools to work directly on Drawing Sheets and allow you to add dimensions, annotations and other technical symbols with great control over their aspect.

Assignment 1

1. Do the following Part modelling and drafting using FreeCAD.

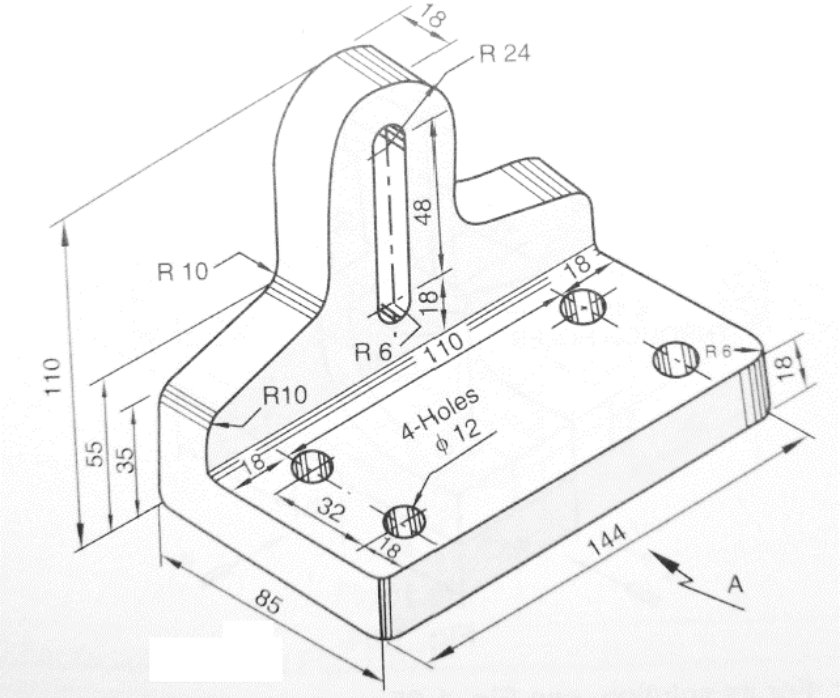


All dimensions are in mm.
Fig.A-02-01

Drawing Name: Support **No: FCAD/2/A01**

Assignment 2

2. Do the following Part modelling and drafting using FreeCAD.

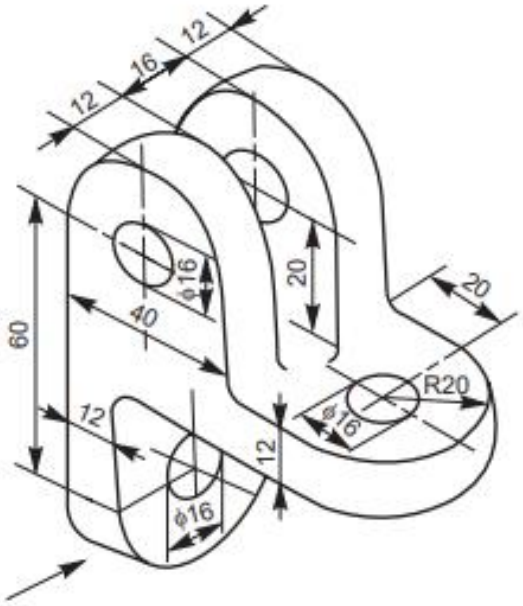


All dimensions are in mm.
Fig.A-02-02

Drawing Name: Bracket **No: FCAD/2/A02**

Assignment 3

3. Do the following Part modelling and drafting using FreeCAD.



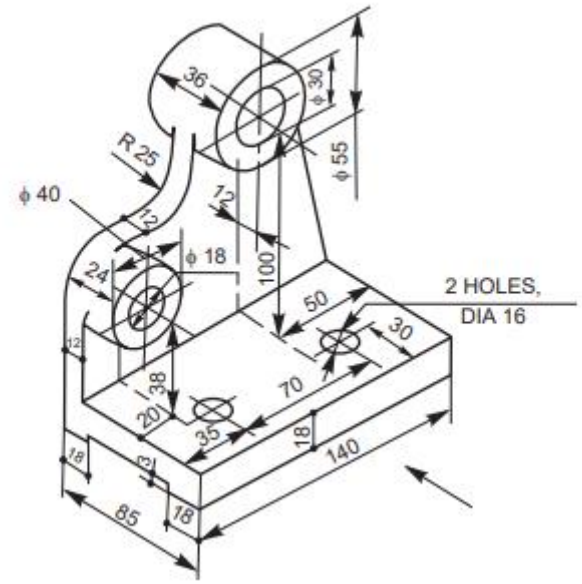
All dimensions are in mm.

Fig.A-02-03

Drawing Name: Bracket **No: FCAD/2/A03**

Assignment 4

4. Do the following Part modelling and drafting using FreeCAD.



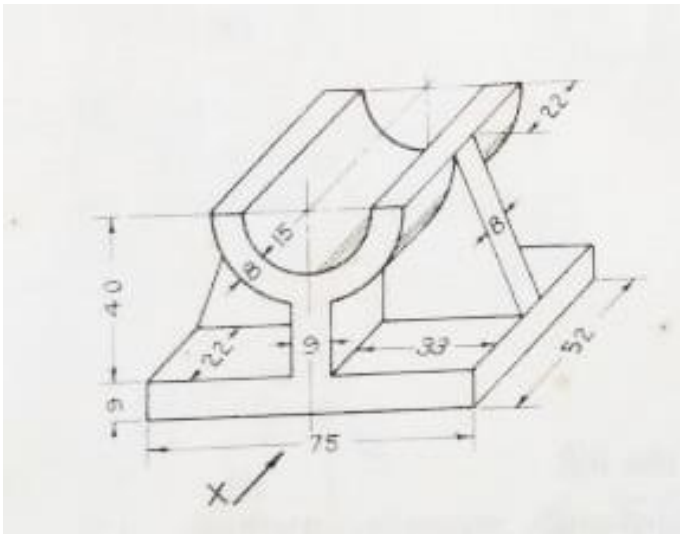
All dimensions are in mm.

Fig.A-02-04

Drawing Name: Bearing support **No: FCAD/2/A04**

Assignment 5

5. Do the following Part modelling and drafting using FreeCAD.



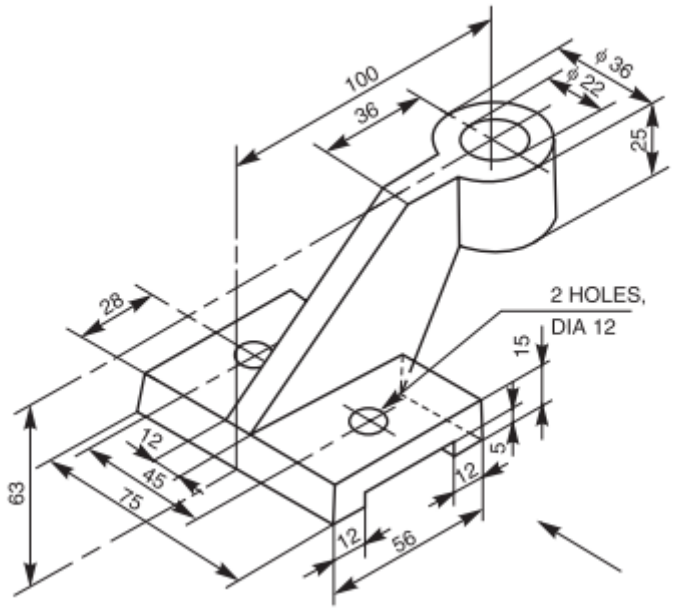
All dimensions are in mm.

Fig.A-02-05

Drawing Name: Bearing support **No: FCAD/2/A05**

Assignment 6

6. Do the following Part modelling and drafting using FreeCAD.



All dimensions are in mm.
Fig.A-02-06

Drawing Name: Bearing support **No: FCAD/2/A06**