Engineering Graphics Using CAD
A Practical Manual

First Edition

(For Undergraduate Engineering Students)

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Preface

Engineering Graphics is a fundamental subject for all engineering students, as it helps in developing the ability to visualize, understand, and represent objects accurately. Traditionally, engineering drawing was done manually using drawing instruments. However, with the advancement of technology, computer-aided design (CAD) tools have become an essential part of engineering practice.

This book, Engineering Graphics Using CAD: A Practical Manual, is designed to provide students with a clear understanding of engineering graphics concepts through hands-on practice using CAD software. The focus of this manual is to bridge the gap between theoretical knowledge and practical application.

The contents of this book are organized in a simple and systematic manner. It begins with basic 2D drafting exercises such as plane figures and progresses to 3D modelling of solids including prisms, pyramids, cylinders, cones, and combinations of solids. Further, it includes exercises on sectioning of solids, helping students understand truncated shapes and their practical significance.

Each experiment in this manual is presented in an NBA (Outcome-Based Education) format, including Aim, Outcomes, Theory, Procedure, and Result. The procedures are written in simple English so that students can easily follow and perform the exercises using CAD software such as FreeCAD.

This manual is intended for undergraduate engineering students. It will also be useful for beginners who want to learn CAD-based engineering drawing in a structured manner.

The author hopes that this book will help students develop strong visualization skills, improve their CAD proficiency, and gain confidence in engineering drawing.

Dr. MANICKAVASAHAM G

Acknowledgement

I would like to express my sincere gratitude to all those who have supported and encouraged me in the preparation of this book, Engineering Graphics Using CAD: A Practical Manual.

I would like to acknowledge the valuable knowledge gained from standard textbooks and reference materials in the field of Engineering Graphics, which have greatly helped in shaping the content and structure of this manual.

I also extend my appreciation to my students, whose curiosity and interest in learning CAD-based engineering graphics inspired me to design this book in a simple and practical manner. Their feedback played an important role in shaping this manual.

I would like to acknowledge the developers and contributors of FreeCAD for providing an open-source CAD platform that made it possible to create and demonstrate the exercises included in this book.

I am grateful to my family for their constant support, encouragement, and understanding during the preparation of this work.

Finally, I thank everyone who directly or indirectly contributed to the completion of this book.

Dr. MANICKAVASAHAM G

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Abbreviations, Symbols and Notations

Abbreviations

mm – Millimetre

V.P./VP – Vertical plane

H.P./ HP – Horizontal plane

XY – Reference line

F.V./FV – Front view

T.V./TV – Top view

R.S.V./RSV – Right side view

L.S.V./LSV – Left side view

CAD – Computer-Aided Design

2D – Two-dimensional

3D – Three-dimensional

Symbols

R – Radius of circle/arc

ϕ - Diameter of circle

θ – True inclination with H.P.

Notations

A, B, C – Names of the points

a, b, c – Top views of A, B, C

a', b', c' – Front views of A, B, C

a'', b'', c'' – Side views of A, B, C

O – Centre point

1. Introduction to Engineering Graphics and CAD

Engineering Graphics is the language used by engineers to represent ideas, designs, and technical information in the form of drawings. It provides a clear and accurate way to communicate the shape, size, and specifications of engineering components and structures. Engineering drawings follow standardized conventions such as line types, dimensioning methods, symbols, and projection techniques so that engineers, designers, and manufacturers can easily understand and interpret the drawings.

Traditionally, engineering drawings were prepared manually using drawing boards and instruments. With the advancement of computer technology, Computer-Aided Design (CAD) has become the modern method for creating engineering drawings.

1.1 Importance of engineering graphics in engineering communication

The importance of engineering graphics in engineering communication is multifaceted, serving as a foundational element in the education and practice of engineering. It facilitates the clear expression of ideas, enhances collaboration among engineers, and supports the design process through various graphical techniques. The following sections outline the key aspects of this significance.

1.1.1. Historical Context and Evolution

Engineering graphics has evolved from manual drafting to advanced computer-aided design (CAD) and 3D modeling, reflecting technological advancements. Historical achievements in engineering often relied on graphical communication, underscoring its longstanding relevance (Barr, 2004).

1.1.2. Educational Relevance

Engineering graphics is integral to the engineering curriculum, fostering essential visual communication and technical drawing skills (Mamasalievna, 2023). Recent surveys indicate a consensus on the necessity of graphical communication skills as part of ABET accreditation requirements (Barr, 2004).

1.1.3. Practical Applications

Proficiency in engineering graphics enhances productivity and efficiency, making engineers with these skills highly valuable in the workforce (Martinez, 1999). The integration of sketching and CAD in teaching methods allows for a comprehensive understanding of design principles (Kopp, 1999). The advancements in technology may render traditional engineering graphics skills less critical, as automated tools increasingly handle design tasks. However, the ability to interpret and create graphics remains essential for effective communication and problem-solving in engineering contexts (Martinez, 1999).

1.2 Manual drafting vs Computer-Aided Design

The comparison between manual drafting and Computer-Aided Design (CAD) reveals significant differences in efficiency, accuracy, and overall productivity. CAD systems have transformed various industries, including garment production and landscape architecture, by streamlining processes and enhancing design capabilities. The following sections outline the key advantages of CAD over manual drafting.

1.2.1. Efficiency and Speed

CAD significantly reduces the time required for drafting. For instance, CAD construction can be five times quicker than manual methods in garment pattern-making (Jankoska & Stojanovska, 2024). In roadway design, productivity ratios indicate that CAD can reduce manual drafting time by approximately threefold (Johnson, 1985).

1.2.2. Accuracy and Quality

CAD offers higher precision in designs, producing more accurate patterns and drawings compared to manual techniques (Jankoska & Stojanovska, 2024 and Sibbald, 1985). The ability to easily modify designs and generate detailed outputs enhances the quality of work, as seen in landscape architecture where CAD allows for intricate 3D modeling (Lallawmzuali & Pal, 2023).

1.2.3. Educational Impact

Studies show that both manual and CAD methods are effective for teaching technical drawing, but CAD may provide a more engaging learning experience for students (Smith & Glenn, 1997).

1.3 Applications of CAD in engineering industries

Computer-Aided Design (CAD) has become a cornerstone in various engineering industries, enhancing design processes through advanced modeling, simulation, and analysis capabilities. The integration of CAD with other technologies like CAM and CAE has significantly improved efficiency and productivity in manufacturing and design workflows. Below are key applications of CAD across different engineering sectors.

1.3.1. Enhanced Product Development

CAD allows for the rapid creation of virtual models, enabling engineers to visualize and modify designs before physical production (Bi, 2021). It automates complex design tasks, reducing time and increasing precision in product development ("Computer-aided Design", 2019).

1.3.2. Interdisciplinary Applications

CAD is utilized in diverse fields such as mechanical, electrical, and architectural engineering, facilitating the design of intricate systems and structures. The integration of CAD with intelligent systems (ICAD) enhances creativity by automating routine tasks, allowing designers to focus on innovative aspects (Tomiyama, 1990).

1.3.3. Improved Manufacturing Processes

The combination of CAD with CAM and CAE systems leads to streamlined manufacturing processes, resulting in higher reliability and productivity. Case studies indicate that industries adopting these technologies experience significant reductions in production time without compromising quality (Țîțu & Pop, 2024).

1.4 Introduction to CAD software

Computer-Aided Design (CAD) is a pivotal technology in modern engineering and manufacturing, enabling the creation, analysis, and simulation of products and systems with enhanced precision and efficiency. Initially developed for geometric representation, CAD has evolved to incorporate advanced modeling techniques, including parametric modeling and knowledge-based engineering. This evolution has significantly transformed the design process, allowing for rapid prototyping and complex simulations.

1.4.1. Key Features of CAD Software

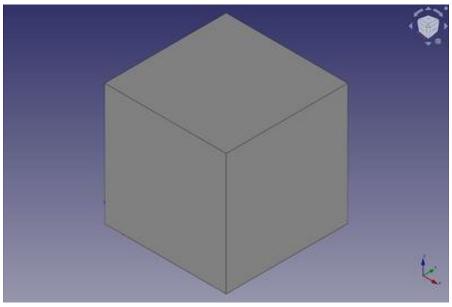
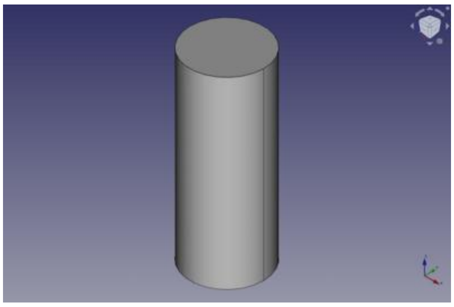
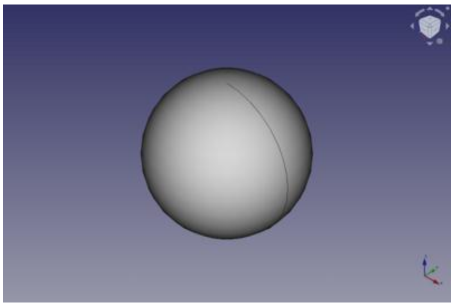
- ***Modeling Techniques:*** CAD software employs various modeling techniques, such as solid modeling and parametric modeling, to create detailed virtual representations of products (Bi, 2021).
- ***Automation of Design Tasks:*** CAD automates repetitive design tasks, which accelerates the product development process and reduces human error ("Computer-aided Design", 2019 and Kennedy et al., n.d.).
- ***User Interface and Tools:*** Software like AutoCAD provides a user-friendly interface and tools for creating intricate designs, making it accessible for engineers and designers ("Application System AutoCAD", 2022).

2. FreeCAD workbenches

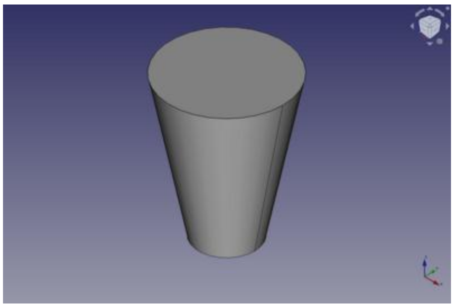
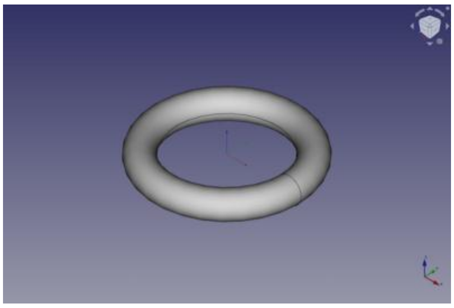
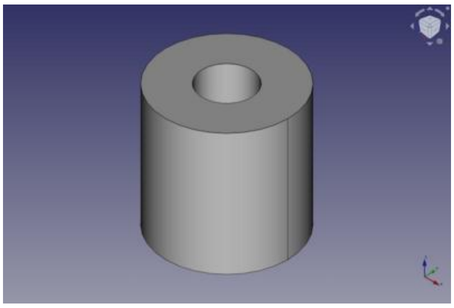
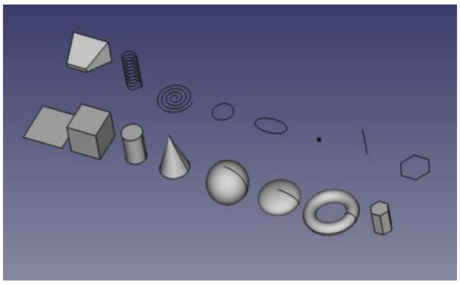
FreeCAD offers various workbenches, each dedicated to different applications. Although the multitude of options might seem overwhelming at first, each workbench is designed to cater to specific tasks, making the overall workflow more efficient and tailored to various project requirements. For instance, the Part Design workbench is ideal for creating and modifying solid 3D models, while the Draft workbench is perfect for 2D drafting and drawing. This modular approach allows users to customize their interface and toolset according to their specific needs and preferences.

2.1 Part

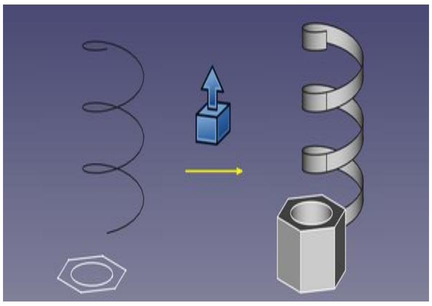
The Part Workbench offers fundamental tools for working with solid parts, including primitives like cubes and spheres, as well as basic geometric and boolean operations.

Tool	Description	Illustration
Box	The Part Box command creates a parametric box solid, a rectangular cuboid	
Cylinder	The Part Cylinder command creates a parametric cylinder solid. It is the result of extruding a circular arc along a straight path.	
Sphere	The Part Sphere command creates a parametric sphere solid. It is the result of revolving a circular arc profile around an axis.	

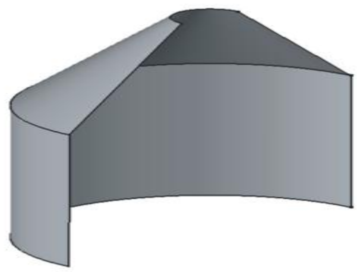
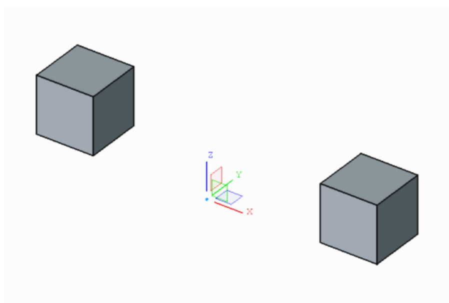
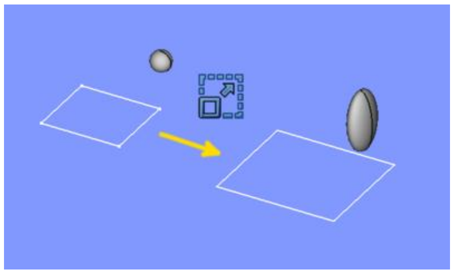
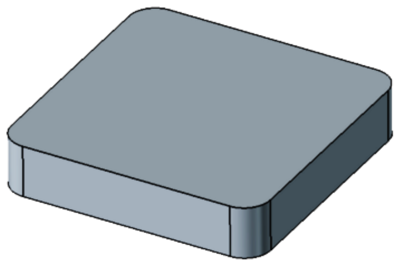
Source: FreeCAD.org

Tool	Description	Illustration
Cone	The Part Cone command creates a parametric cone solid. In the coordinate system defined by its Data Placement property, the bottom face of the cone lies on the XY-plane with its center at the origin.	
Torus	The Part Torus command creates a parametric torus solid, a doughnut shape. It is the result of sweeping a circular profile around a circular path.	
Tube	The Part Tube command creates a parametric tube solid. In the coordinate system defined by its Data Placement property, the bottom face of the tube lies on the XY-plane with its center at the origin.	
Part Primitives	The Part Primitives command opens a dialog to create one or more parametric primitives. 16 primitive types are available. Creates a plane, box, cylinder, cone, sphere, ellipsoid, torus, prism, wedge, helix, spiral, circular arc, elliptical arc, point, line, and regular polygon.	
Shape Builder	The Part Builder command creates more complex shapes from various geometric primitives.	

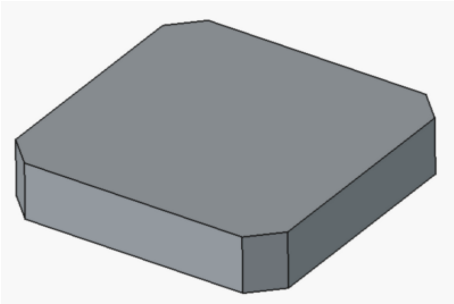
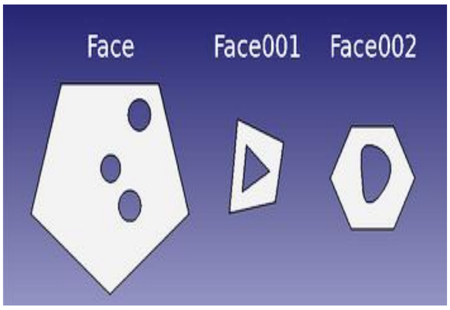
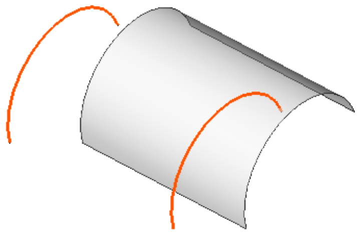
Source: FreeCAD.org

Tool	Description	Illustration
Extrude	The Part Extrude command extends a shape by a specified distance, in a specified direction. The output shape type will vary depending on the input shape type and the options selected. In most common scenarios, the following lists the expected output shape type from a given input shape type, • Extruding a Vertex (point) will produce a straight Edge (Line). • Extruding an open edge (e.g. line, arc) will produce an open face (e.g. plane). • Extruding a closed edge (e.g. circle) will optionally produce a closed face (e.g. an open-ended cylinder) or if the parameter "solid" is "true" will produce a solid (e.g. a closed solid cylinder). • Extruding an open Wire (e.g. a Draft Wire) will produce an open shell (several joined faces). • Extruding a closed Wire (e.g. a Draft Wire) will optionally produce a shell (several joined faces) or if the parameter "solid" is "true" will produce a solid. • Extruding a face (e.g. plane) will produce a solid (e.g. Cuboid). • Extruding a Draft Shape String will produce a compound of solids (the string is a compound of the letters which are each a solid). • Extruding a shell of faces will produce a Compsolid.	

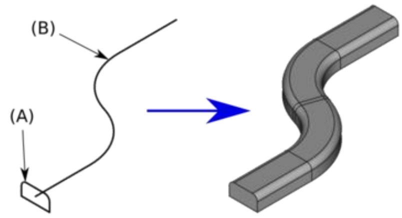
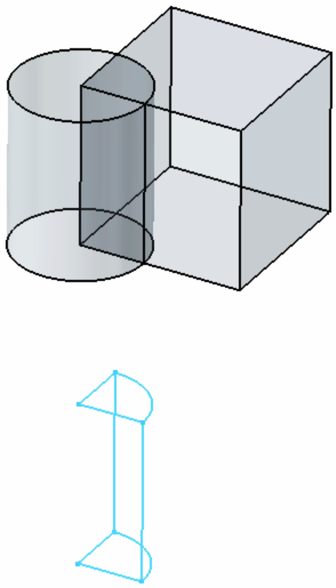
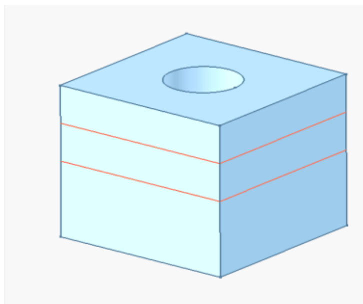
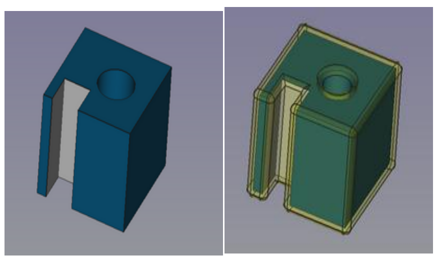
Source: FreeCAD.org

Tool	Description	Illustration
Revolve	The Part Revolve command revolves the selected object around a given axis. The following shape types are allowed, and lead to the listed output shapes: **Input shape** — **Output shape** Vertex — Edge Edge — Face Wire — Shell Face — Solid Shell — Compound solid (Compsolid)	
Mirror	The Part Mirror command creates a new object (image) which is a reflection of the original object (source). The image object is created behind a mirror plane. The mirror plane may be standard plane (XY, YZ, or XZ), any plane parallel to a standard plane.	
Scale	The Part Scale command scales shapes by a specified factor in all directions or by distinct factors in each cardinal direction. In the case of distinct factors, the shapes may be distorted.	
Fillet	The Part Fillet command creates fillets (rounds) on the selected edges of a shape. A dialog allows choosing which shape and which edges to work on.	

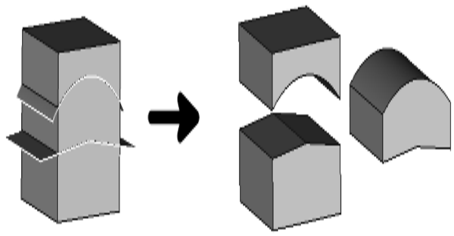
Source: FreeCAD.org

Tool	Description	Illustration
Chamfer	The Part Chamfer command chamfers the selected edge(s) of an object. A dialog allows you to choose which edges to work on as well as modify various chamfer parameters.	
MakeFace	The Part Make Face command creates a planar face from one or more coplanar closed wires (contours). They can be any valid wire, i.e. created with the Part Workbench, the Draft Workbench or the Sketcher Workbench. The contours should not self-intersect, or intersect each other. They can be nested to create voids.	
Ruled Surface	The Part Ruled Surface command creates a ruled surface spanning between two selected edges/wires.	
Loft	The Part Loft command creates a face, a shell, or a solid shape from two or more profiles (cross-sections).	 Loft from three profiles which are two Part Circles and one Part Ellipse. Properties are Solid "True" and Ruled "True".

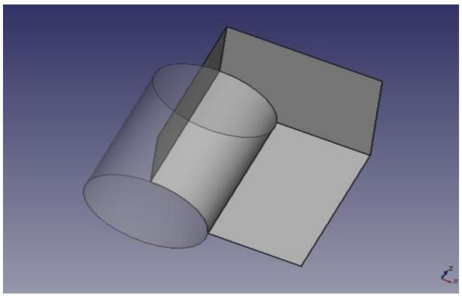
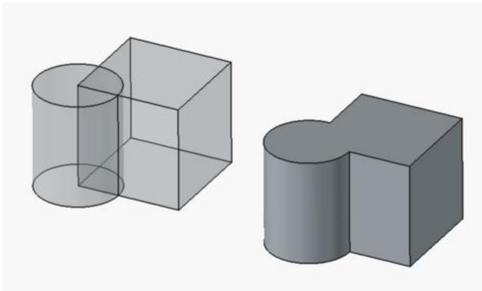
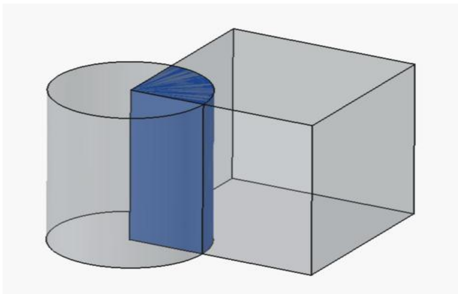
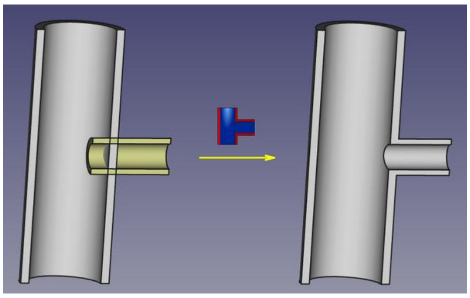
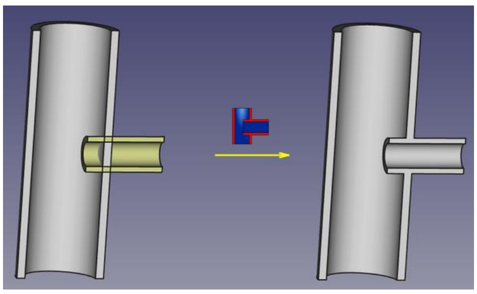
Source: FreeCAD.org

Tool	Description	Illustration
Sweep	The Part Sweep command creates a face, a shell, or a solid shape from one or more profiles (cross-sections) distributed along a spine. The Part Sweep command is similar to Part Loft with the addition of a spine.	
Section	The Part Section command creates wire geometry at the intersections of two objects. The result is fully parametric. • An intersection of two solids/faces results in one or more section lines. • An intersection of two lines, or a line and a solid/face, results in one or more points.	
Cross Sections	The Part Cross Sections command creates one or more cross-sections through the selected shape, parallel to one of the default global planes (XY, XZ or YZ).	
Offset	The Part Offset command creates parallel copies of a selected shape at a certain distance from the base shape, giving a new object.	

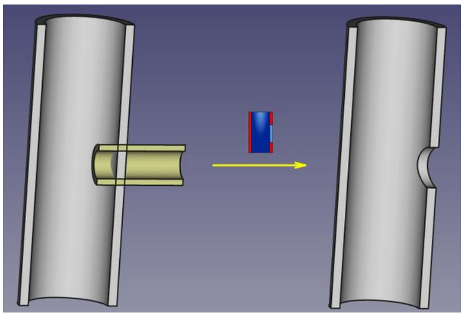
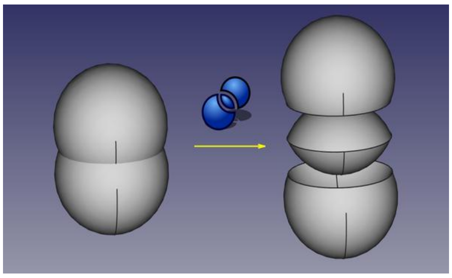
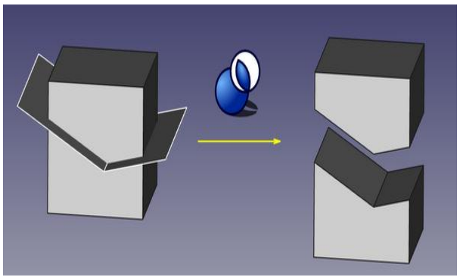
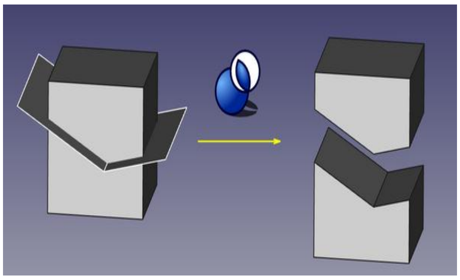
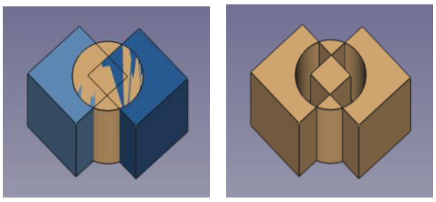
Source: FreeCAD.org

Tool	Description	Illustration
Thickness	The Part Thickness command works on a solid shape and transforms it into a hollow object, giving to each of its faces a defined and constant thickness. On some solids it allows you to significantly speed up the work, and avoids making extrusions and pockets.	
Compound	The Part Compound command creates a compound of objects with a topological shape such as solid objects and other objects with faces and/or edges. It cannot handle meshes as they do not have a topological shape.	
Explode Compound	The Part Explode Compound command splits a compound of shapes, to make each contained shape (child) available as a separate object.	
Compound Filter	The Part Compound Filter command can be used to extract the individual pieces of the result of e.g. a Part Slice operation, with which you have split an object.	
Boolean	The Part Boolean command can perform four boolean operations. A task panel is used to specify the operation and the objects. For quicker access to the boolean operations, use Part Cut, Part Fuse, Part Common and Part Section.	

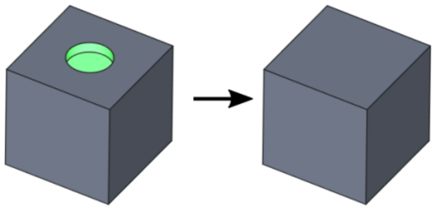
Source: FreeCAD.org

Tool	Description	Illustration
Cut	The Part Cut command cuts (subtracts) selected Part objects, the last one being subtracted from the first one. This operation is fully parametric and the components can be modified and the result recomputed.	
Fuse	The Part Fuse command fuses (unites) selected Part objects into one. This operation is fully parametric and the components can be modified and the result recomputed. This command is an automated form of the Boolean Operation.	
Common	The Part Common command extracts the common part (intersection) between selected Part objects. This operation is fully parametric and the components can be modified and the result recomputed.	
Join Connect	The Part Join Connect tool connects the interiors of two walled objects (e.g. pipes). It can also join shells and wires.	
Join Embed	The Part Join Embed tool embeds a walled object (e.g. a pipe) into another walled object.	

Source: FreeCAD.org

Tool	Description	Illustration
Join Cutout	The Part Join Cutout tool creates a cutout in a walled object (e.g. a pipe) to fit another walled object.	
Boolean Fragments	The Part Boolean Fragments command computes all fragments that can result from applying Boolean operations between input shapes. For example, for two intersecting spheres, three non-overlapping but touching solids are generated.	
Slice Apart	The Part Slice Apart command splits shapes by intersection with other shapes. For example, for a box and a plane, two solids are created.	
Slice to compound	The Part Slice command splits shapes by intersection with other shapes. For example, for a box and a plane, a compound of two solids is created.	
Boolean XOR	The Part XOR command removes geometry shared by an even number of objects and leaves a void space between the involved objects. For two objects it represents a symmetric version of Part Cut.	

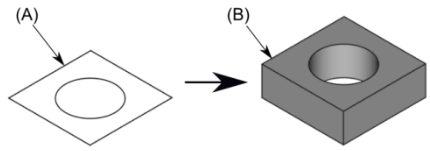
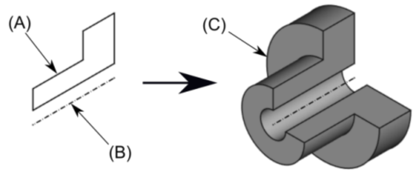
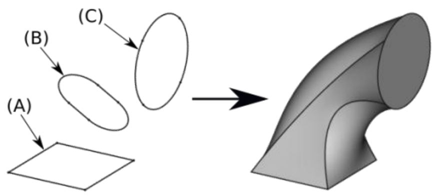
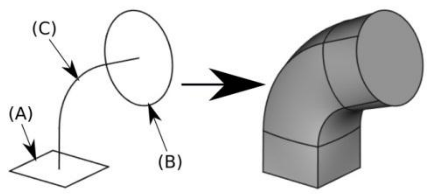
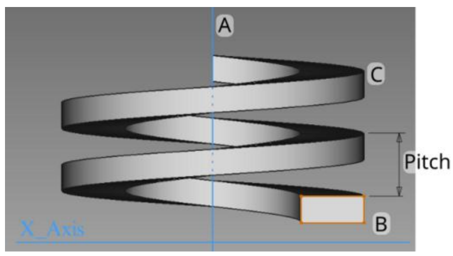
Source: FreeCAD.org

Tool	Description	Illustration
Check Geometry	The Part Check Geometry command runs a verification and reports if geometry is a valid solid. The command checks if the Boundary representation (BRep or B-rep) of the model is valid.	
Defeaturing	The Part Defeaturing command can remove selected form features, such as holes, protrusions, gaps, chamfers, fillets etc. from a model. The defeaturing command can be very useful in different contexts: • To edit an imported solid where no history of operations is available. • Fixing defects in the model, e.g. filling gaps, holes etc. • Model simplification for numeric analysis, display on mobile devices, etc. The removed form features are filled by the extension of the adjacent faces, thus no unexpected parts should appear in the result. Please note that the result is a new shape that is not linked to the original; thus, it is non-parametric.	

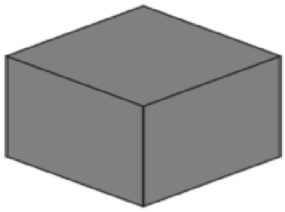
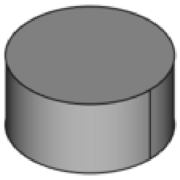
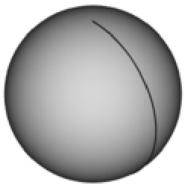
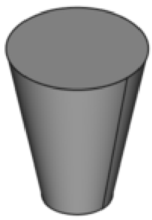
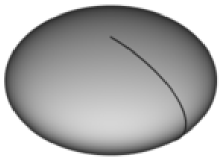
Source: FreeCAD.org

2.2 Part Design

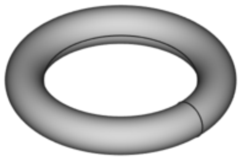
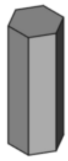
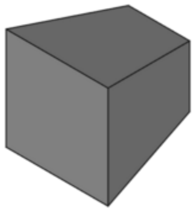
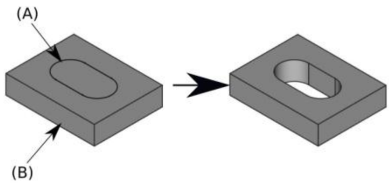
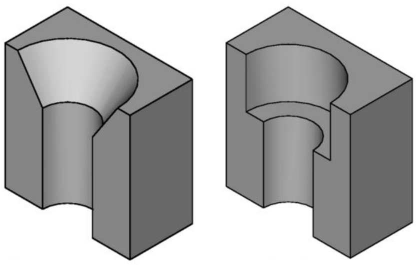
The Part Design Workbench is an essential tool for creating and modifying solid 3D models. It allows users to design complex parts by sketching 2D profiles and then applying various operations such as extrusions, lofts, and revolutions to generate 3D geometry. This workbench also supports the creation of features like pockets, holes, fillets, and chamfers, providing a comprehensive set of tools for detailed part design. Additionally, the Part Design Workbench integrates seamlessly with the Sketcher Workbench, enabling users to define and constrain sketches that serve as the foundation for 3D models. This integration facilitates a parametric design approach, allowing for easy adjustments and updates to the model throughout the design process.

Tool	**Description**	**Image**
Pad	The Pad tool extrudes a sketch or a face along a straight path.	
Revolution	The Revolution tool creates a solid by revolving a selected sketch or 2D object about a given axis.	
Additive Loft	Additive Loft creates a solid in the active Body by making a transition between two or more sketches (also referred to as cross-sections). If the Body already contains features, the additive loft will be merged to them.	
Additive Pipe	Additive Pipe creates a solid in the active Body by sweeping one or more sketches (also referred to as cross-sections) along an open or closed path. If the Body already contains features, the additive pipe will be merged to them.	
Additive Helix	The Additive Helix tool creates a solid by sweeping a selected sketch or 2D object along a helix path.	

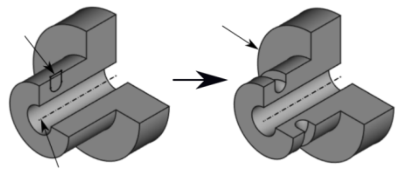
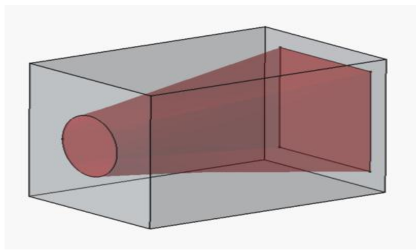
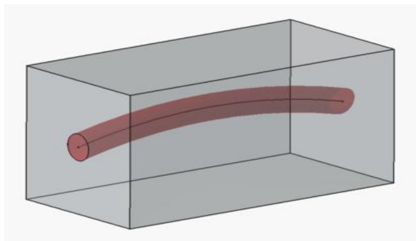
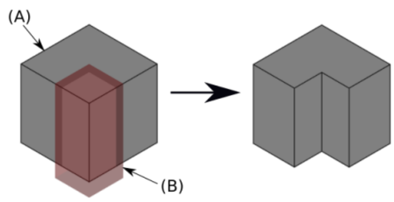
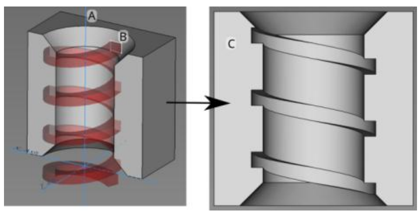
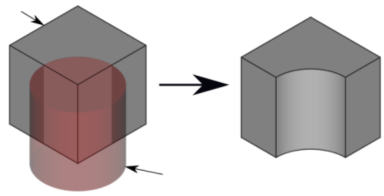
Source: FreeCAD.org

Tool	Description	Image
Additive Box	Inserts a primitive box in the active Body as the first feature, or fuses it to the existing feature(s).	
Additive Cylinder	Inserts a primitive cylinder in the active Body as the first feature, or fuses it to the existing feature(s).	
Additive Sphere	Inserts a primitive sphere in the active Body as the first feature, or fuses it to the existing feature(s).	
Additive Cone	Inserts a primitive cone in the active Body as the first feature, or fuses it to the existing feature(s).	
Additive Ellipsoid	Inserts a primitive ellipsoid in the active Body as the first feature, or fuses it to the existing feature(s).	

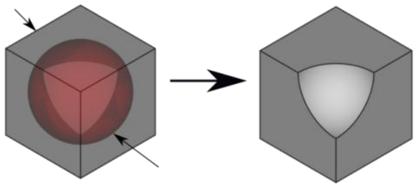
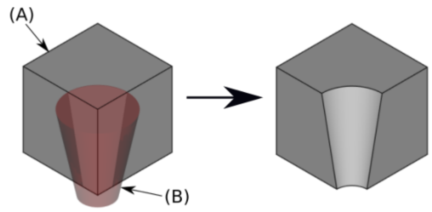
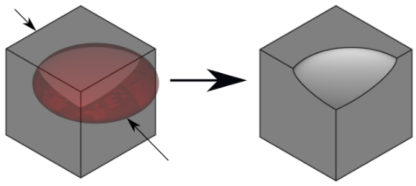
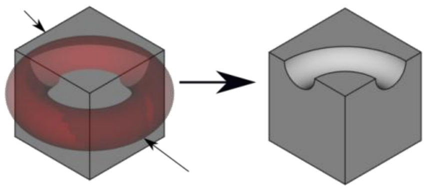
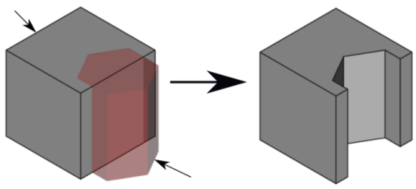
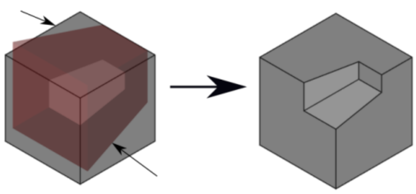
Source: FreeCAD.org

Tool	Description	Image
Additive Torus	Inserts a primitive torus in the active Body as the first feature, or fuses it to the existing feature(s).	
Additive Prism	Inserts a primitive prism in the active Body as the first feature, or fuses it to the existing feature(s).	
Additive Wedge	Inserts a primitive wedge in the active Body as the first feature, or fuses it to the existing feature(s).	
Pocket	The Pocket tool cuts solids by extruding a sketch or a face of a solid along a straight path.	
Hole	The Hole feature creates one or more holes from a selected sketch's circles and arcs. If arcs are present they must be part of closed contours. All non arc/circle entities are ignored but they still must form closed contours. Many parameters can be set such as threading and size, fit, hole type (countersink, counter bore, straight) and more. The centers of the circles and arcs are used to position the holes, but please note that their radii are not taken into account. The generated holes will be identical even if the radii vary.	

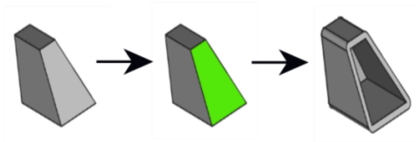
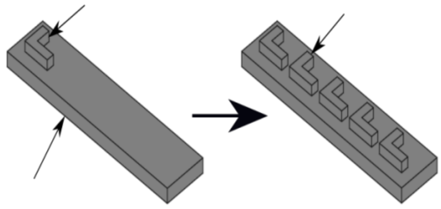
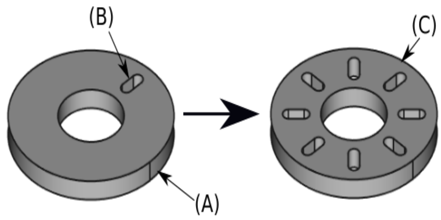
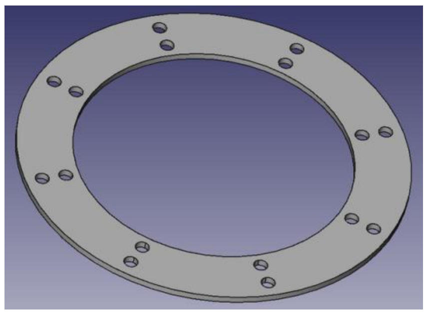
Source: FreeCAD.org

Tool	Description	Image
Groove	The Groove tool revolves a selected sketch or profile about a given axis, cutting out material from the support.	
Subtractive Loft	Subtractive Loft creates a subtractive solid in the active Body by making a transition between two or more sketches (also referred to as cross-sections). Its shape is then subtracted from the existing solid.	
Subtractive Pipe	Subtractive Pipe creates a subtractive solid in the active Body by sweeping one or more sketches (also referred to as cross-sections) along an open or closed path. Its shape is then subtracted from the existing solid. Subtractive Pipe is often used in connection with Part Helix and Part Design Shape Binder to create a thread; see the Thread for Screw Tutorial for details.	
Subtractive Box	Inserts a subtractive box in the active Body. Its shape is subtracted from the existing solid.	
Subtractive Helix	The Subtractive Helix tool modifies a solid by sweeping a selected sketch or 2D object along a helix path cutting away the material.	
Subtractive Cylinder	Inserts a subtractive cylinder in the active Body. Its shape is subtracted from the existing solid.	

Source: FreeCAD.org

Tool	Description	Image
Subtractive Sphere	Inserts a subtractive sphere in the active Body. Its shape is subtracted from the existing solid.	
Subtractive Cone	Inserts a subtractive cone in the active Body. Its shape is subtracted from the existing solid.	
Subtractive Ellipsoid	Inserts a subtractive ellipsoid in the active Body. Its shape is subtracted from the existing solid.	
Subtractive Torus	Inserts a subtractive torus in the active Body. Its shape is subtracted from the existing solid.	
Subtractive Prism	Inserts a subtractive prism in the active Body. Its shape is subtracted from the existing solid.	
Subtractive Wedge	Inserts a subtractive wedge in the active Body. Its shape is subtracted from the existing solid.	

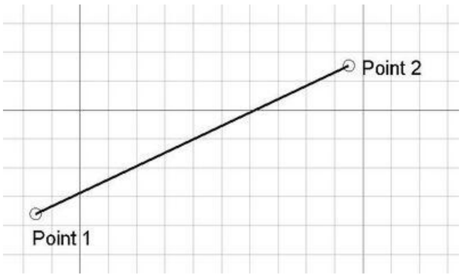
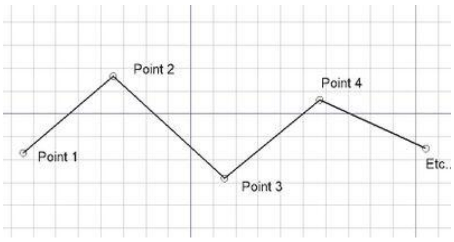
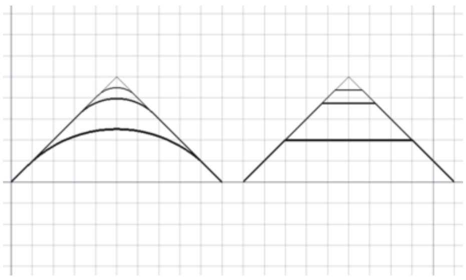
Source: FreeCAD.org

Tool	Description	Image
Thickness	The Part Design Thickness tool transforms a solid body into a hollow object with at least one open face, giving to each of its remaining faces a uniform thickness. It adds a Thickness object to the document with its corresponding representation in the Tree View.	
Linear Pattern	The Part Design Linear Pattern tool creates a linear pattern of one or more features.	
Polar Pattern	The Part Design Polar Pattern tool creates a polar pattern of one or more features.	
Multi Transform	The Part Design Multi Transform tool creates a pattern of one or more features. The pattern can include multiple transformations where each subsequent transformation is applied to the result of the previous transformation. The available transformations are: Mirror, Linear Pattern, Polar Pattern and Scale. The first three are also available as separate tools.	

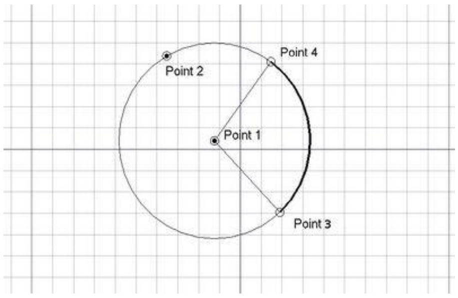
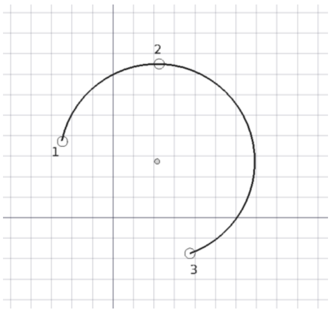
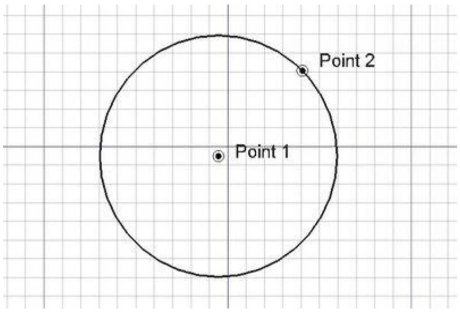
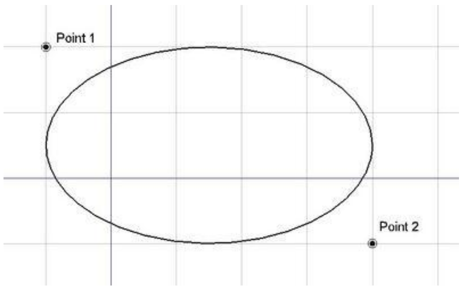
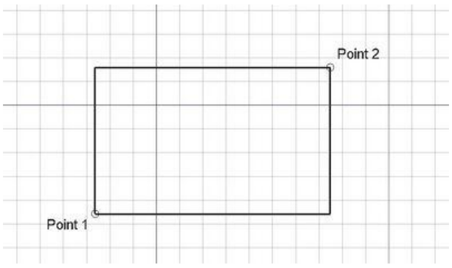
Source: FreeCAD.org

2.3 Draft

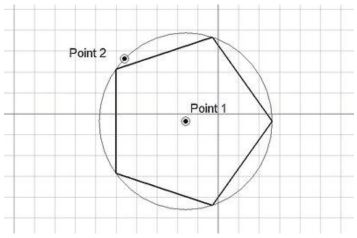
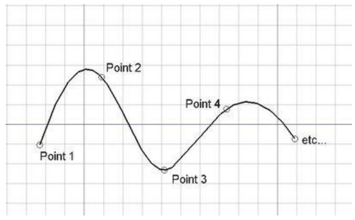
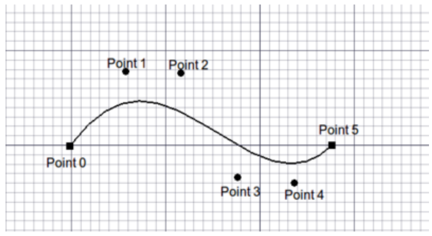
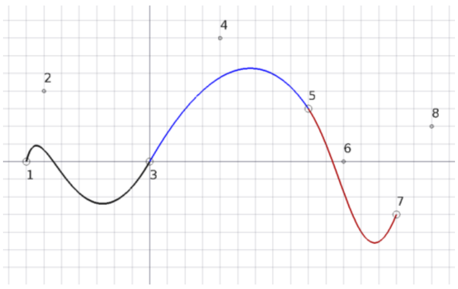
The Draft Workbench is designed for creating and editing 2D drawings. It offers a range of tools for drafting, including lines, arcs, circles, and text. Users can also perform operations like trimming, extending, and offsetting. The Draft Workbench is particularly useful for architectural drawings and can be used to create complex 2D geometries that can be later transformed into 3D models. It integrates seamlessly with other workbenches, providing a versatile foundation for both 2D and 3D design projects.

Tool	Description	Image
Line	The Draft Line command creates a straight line.	
Polyline	The Draft Wire command creates a polyline, a sequence of several connected line segments. The command can also be used to join Draft Lines and Draft Wires. The corners of a Draft Wire can be filleted (rounded) or chamfered by changing its Data Fillet Radius or Data Chamfer Size respectively. It is also possible to subdivide the edges of a Draft Wire by changing its Data Subdivisions property.	
Fillet	The Draft Fillet command creates a fillet, a rounded corner, or a chamfer, a straight edge, between two selected edges.	

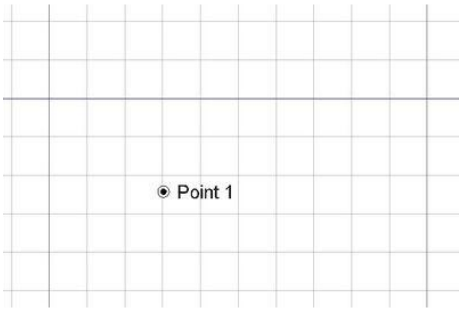
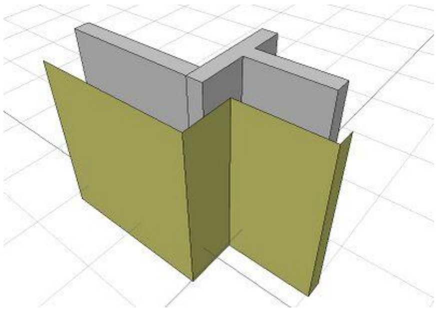
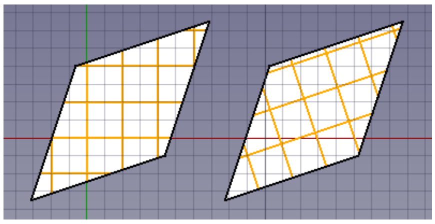
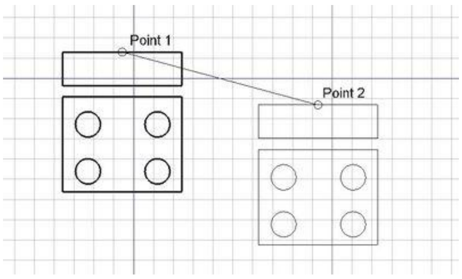
Source: FreeCAD.org

<table>
<tr><th>Tool</th><th>Description</th><th>Image</th></tr>
<tr><td>Arc</td><td>The Draft Arc command creates a circular arc on the current working plane from a center, a radius, a start angle and an aperture angle. The radius and the angles can be defined by picking points.</td><td>
</td></tr>
<tr><td>Arc 3 Points</td><td>The Draft Arc 3Points command creates a circular arc on the current working plane from three points that define its circumference. The center and radius are calculated from these points.</td><td>
</td></tr>
<tr><td>Circle</td><td>The Draft Circle command creates a circle on the current working plane from a center and a radius. The radius can be defined by picking a point.</td><td>
</td></tr>
<tr><td>Ellipse</td><td>The Draft Ellipse command creates an ellipse on the current working plane from two points defining a rectangle in which the ellipse will fit.</td><td>
</td></tr>
<tr><td>Rectangle</td><td>The Draft Rectangle command creates a rectangle on the current working plane from two points.
The corners of a Draft Rectangle can be filleted (rounded) or chamfered by changing its Data Fillet Radius or Data Chamfer Size respectively. It is also possible to subdivide a Draft Rectangle by changing its Data Columns and/or Data Rows property.</td><td>
</td></tr>
</table>

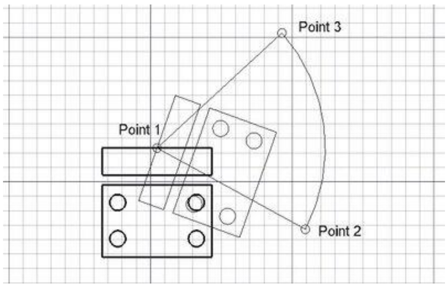
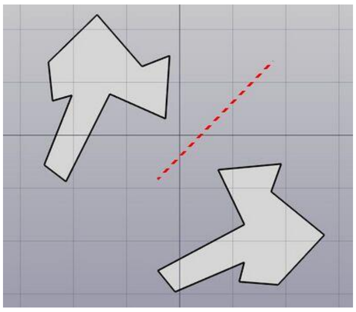
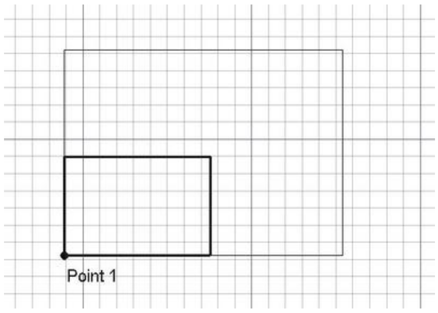
Source: FreeCAD.org

Tool	Description	Image
Polygon	The Draft Polygon command creates a regular polygon on the current working plane from a center and a radius. The radius can be defined by picking a point.	
B-Spline	The Draft B-Spline command creates a B-spline curve from several points. The Draft B-Spline command specifies the exact points through which the curve will pass.	
Bezier Curve	The Draft Bezier Curve command creates a Bezier curve from several points. The command creates a single Bezier curve with a Degree that is number of points. It can be transformed into a piecewise Bezier curve by reducing this property. The Draft Bezier Curve and the Draft Cubic Bezier Curve commands use control points to define the position and curvature of the spline.	
Cubic Bezier Curve	The Bezier Curve is one of the most commonly used curves in computer graphics. This command allows you to create a continuous spline made up of several 3rd-degree Bezier segments, in a way that is similar to the Bezier tool in Inscape. A general Bezier curve of any degree can be created with the Draft Bez Curve command. The Draft Bez Curve and the Draft Cubic Bez Curve commands use control points to define the position and curvature of the spline. The Draft B-Spline command, on the other hand, specifies the exact points through which the curve will pass.	

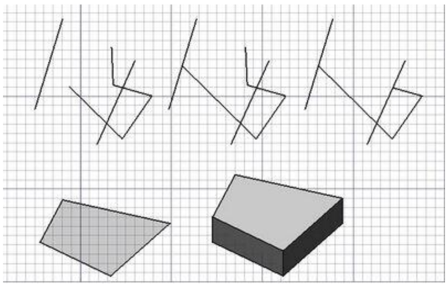
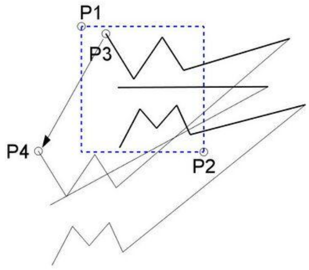
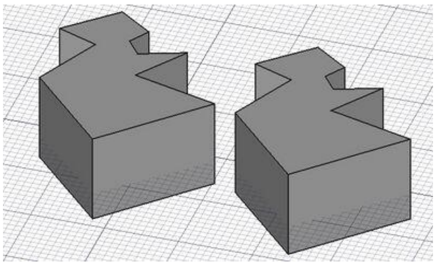
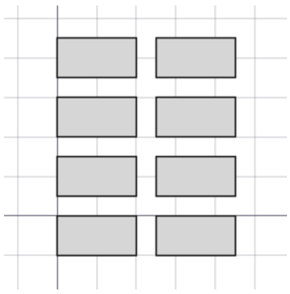
Source: FreeCAD.org

Tool	**Description**	**Image**
Point	The Draft Point command creates a simple point. Draft points can be useful as a reference for placing lines, wires or other objects.	
Face binder	The Draft Face binder command creates a surface object from selected faces. A Draft Face binder is parametric, it will update if you modify its source object(s). It can be used to create an extrusion from a collection of faces. This extrusion can for example represent a wall finish in architectural design.	
Hatch	The Draft Hatch command creates hatches on the planar faces of a selected object.	
Move	The Draft Move command moves or copies selected objects from one point to another. In subelement mode the command moves selected points and edges, or copies selected edges, of Draft Lines and Draft Wires. The command can be used on 2D objects created with the Draft Workbench or Sketcher Workbench, but also on many 3D objects such as those created with the Part Workbench, Part Design Workbench or BIM Workbench.	

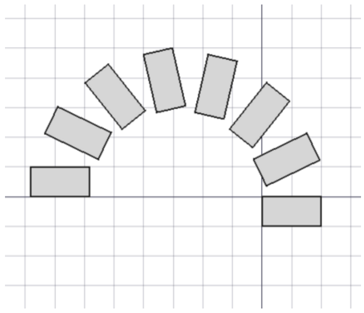
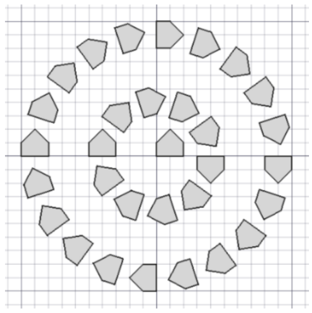
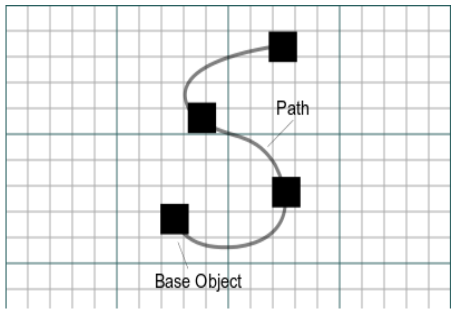
Source: FreeCAD.org

Tool	Description	Image
Rotate	The Draft Rotate command rotates or copies selected objects around a center point by a given angle. The axis of rotation is perpendicular to the current working plane and the rotation angle is relative to that plane. In subelement mode the command rotates selected points and edges, or copies selected edges, of Draft Lines and Draft Wires. The command can be used on 2D objects created with the Draft Workbench or Sketcher Workbench, but also on many 3D objects such as those created with the Part Workbench, Part Design Workbench or BIM Workbench.	
Mirror	The Draft Mirror command creates mirrored copies, Part Mirror objects, from selected objects. A Part Mirror object is parametric, it will update if its source object changes. The command can be used on 2D objects created with the Draft Workbench or Sketcher Workbench, but also on many 3D objects such as those created with the Part Workbench, Part Design Workbench or BIM Workbench.	
Scale	The Draft Scale command scales or copies selected objects around a base point. In subelement mode the command scales selected points and edges of Draft Lines and Draft Wires. The command can be used on 2D objects created with the Draft Workbench or Sketcher Workbench, but also on many 3D objects such as those created with the Part Workbench, Part Design Workbench or BIM Workbench.	

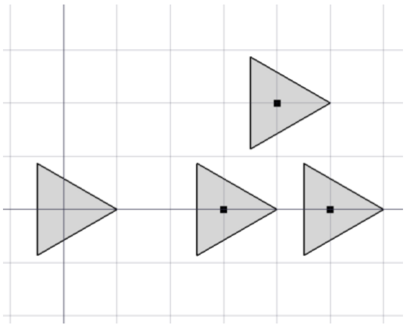
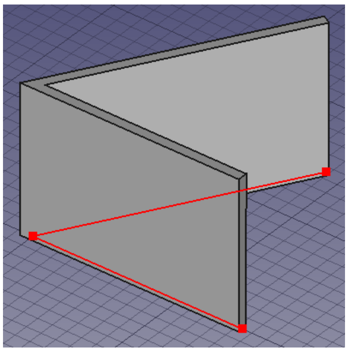
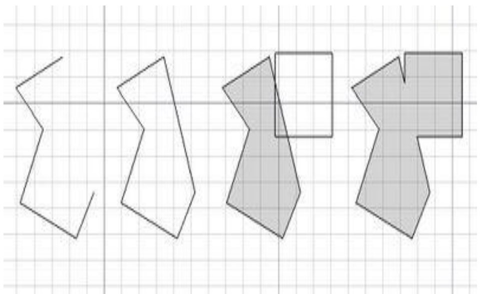
Source: FreeCAD.org

Tool	Description	Image
Offset	The Draft Offset command offsets each segment of a selected object over a given distance, or creates an offset copy of the selected object.	
Trimex	The Draft Trimex command trims or extends a selected object. Intersections with the edge of another object can be used to determine new endpoints. The command can also be used to extrude a face, in which case it creates a Part Extrude object.	
Stretch	The Draft Stretch command stretches objects by moving selected points.	
Clone	The Draft Clone command creates linked copies, clones, of selected objects. The shape of a clone is parametric, it will update if its source object changes. But a clone does have its own position, rotation, and scale, and its own View properties. For BIM objects the command creates a special type of clone: an Arch Clone.	
Ortho Array	The Draft Ortho Array command creates an orthogonal (3-axes) array from a selected object. The command can optionally create a Link array, which is more efficient than a regular array.	

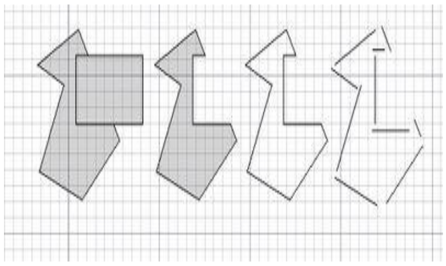
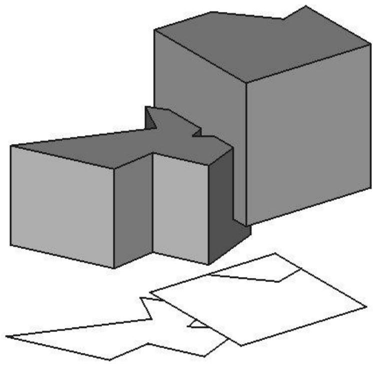
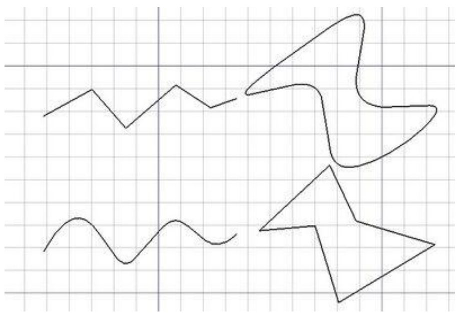
Source: FreeCAD.org

Tool	Description	Image
Polar Array	The Draft Polar Array command creates an array from a selected object by placing copies along a circle. The command can optionally create a Link array, which is more efficient than a regular array.	
Circular Array	The Draft Circular Array command creates an array from a selected object by placing copies along concentric circles. The command can optionally create a Link array, which is more efficient than a regular array.	
Path Array	The Draft Path Array command creates a regular array from a selected object by placing copies along a path. Use the Draft Path Link Array command to create a more efficient Link array instead. Except for the type of array that is created, Link array or regular array, the Draft Path Link Array command is identical to this command.	
Path Link Array	The Draft Path Link Array command creates a Link array from a selected object by placing copies along a path. Use the Draft Path Array command to create a less efficient regular array instead. Except for the type of array that is created, Link array or regular array, this command is identical to the Draft Path Array command.	

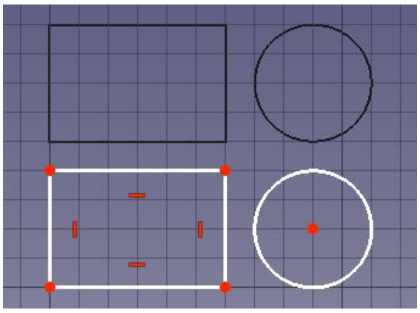
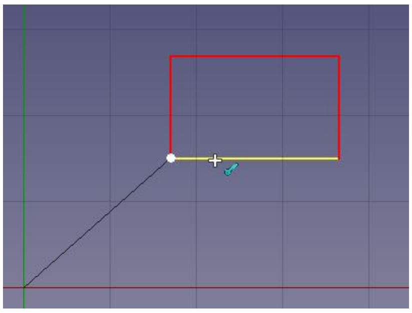
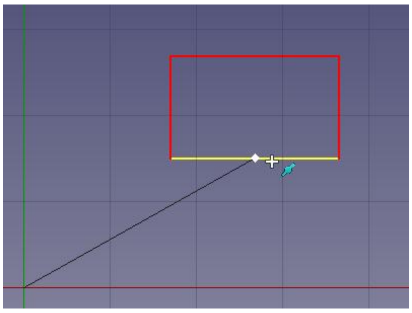
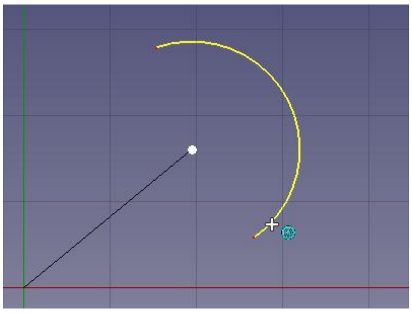
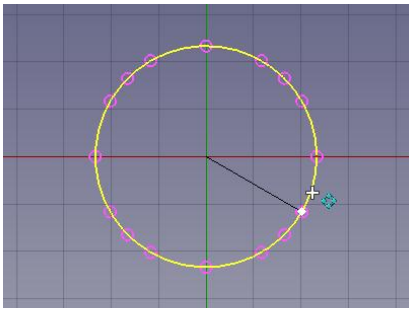
Source: FreeCAD.org

Tool	Description	Image
Point Array	The Draft Point Array command creates a regular array from a selected object by placing copies at the points from a point object. Use the Draft Point Link Array command to create a more efficient Link array instead. Except for the type of array that is created, Link array or regular array, the Draft Point Link Array command is identical to this command.	
Subelement Highlight	The Draft Subelement Highlight command temporarily highlights selected objects, or the base objects of selected objects. It is intended to be used in conjunction with the subelement mode of the Draft Move command, the Draft Rotate command or the Draft Scale command.	
Upgrade	The Draft Upgrade command upgrades selected objects. The result depends on the number of selected objects and their type. The command can for example fuse elements and create faces. It is worth trying to upgrade a selection several times to see if a better result can be obtained. See the example in the image. Note that not all objects can be upgraded. This command is the counterpart of the Draft Downgrade command.	

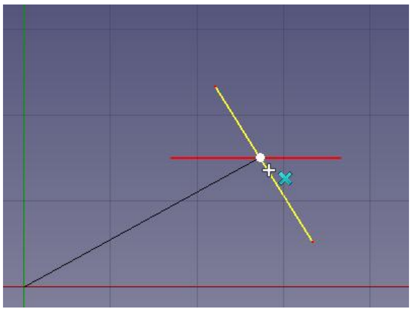
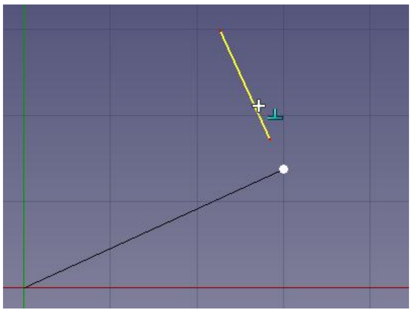
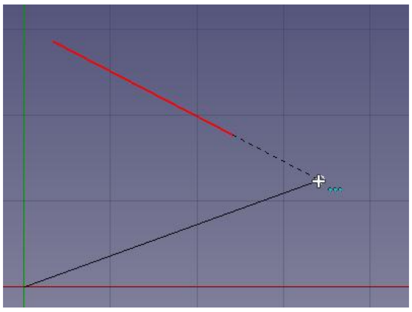
Source: FreeCAD.org

Tool	Description	Image
Downgrade	The Draft Downgrade command downgrades selected objects. The result depends on the number of selected objects and their type. The command can for example deconstruct a 3D solid into separate faces and a wire into separate edges. If two faces are selected a Part Cut object is created from them. Note that not all objects can be downgraded. This command is the counterpart of the Draft Upgrade command.	
Shape 2D View	The Draft Shape 2D View command creates 2D projections from selected objects, usually 3D solids or Arch Section Planes. The projections are placed in the 3D View. Draft Shape 2D View projections can be displayed on a Tech Draw Workbench page using the Tech Draw Draft View command. Alternatively the Tech Draw Workbench offer its own projection commands. But these create projections that are only displayed on the drawing page and not in the 3D View.	
Wire To B-Spline	The Draft Wire To B-Spline command converts Draft Wires to Draft B-Splines and vice versa.	

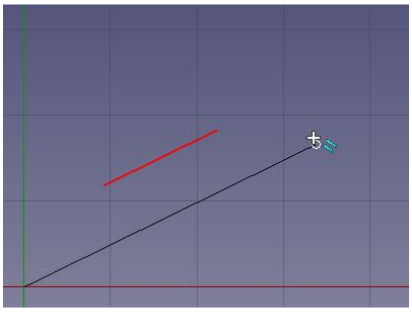
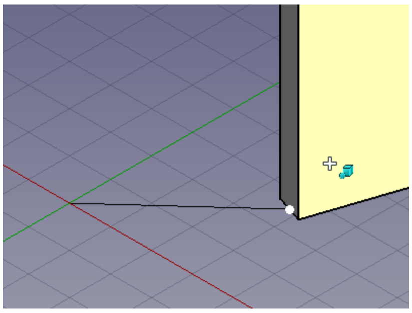
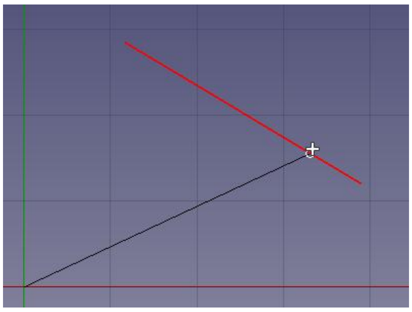
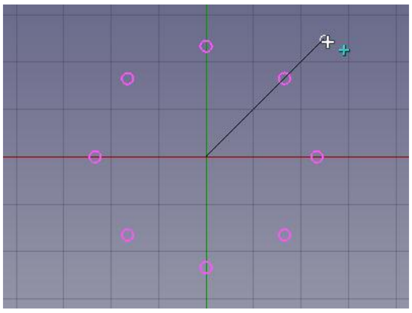
Source: FreeCAD.org

Tool	Description	Image
Draft to Sketch	The Draft, Draft to Sketch command converts Draft objects to Sketcher Sketches and vice versa.	
Snap Endpoint	The Draft Snap Endpoint option snaps to the endpoints of edges. The edges can belong to Draft or BIM objects but also to objects created with other workbenches.	
Snap Midpoint	The Draft Snap Midpoint option snaps to the midpoint of edges. The edges can belong to Draft or BIM objects but also to objects created with other workbenches.	
Snap Center	The Draft Snap Center option snaps to the center point of faces and circular edges, and to the Data Placement point of Draft Working Plane Proxies and Arch Building Parts. The faces and edges can belong to Draft or BIM objects but also to objects created with other workbenches.	
Snap Angle	The Draft Snap Angle option snaps to the special cardinal points on circular edges, at multiples of 30° and 45°. The edges can belong to Draft or BIM objects but also to objects created with other workbenches.	

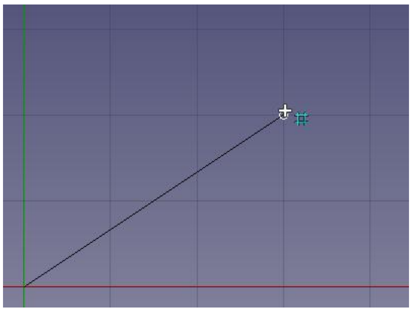
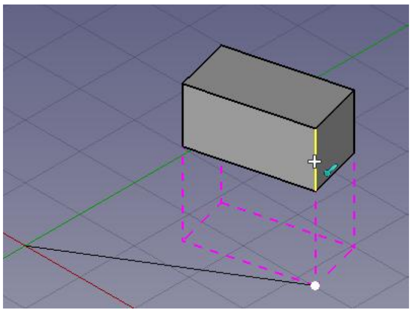
Source: FreeCAD.org

Tool	Description	Image
Snap Intersection	The Draft Snap Intersection option snaps to the intersection of two edges, and the intersection of a face and an edge. The faces and edges can belong to Draft or BIM objects but also to objects created with other workbenches. This snap option will also find apparent intersections of (extended) straight edges if Draft Snap Working Plane is active as well.	
Snap Perpendicular	The Draft Snap Perpendicular option snaps to the perpendicular projections of a previous point on faces and edges. The faces and edges can belong to Draft or BIM objects but also to objects created with other workbenches. This snap option will also find points on extended faces and edges.	
Snap Extension	The Draft Snap Extension option snaps to an imaginary line that extends beyond the endpoints of straight edges. The edges can belong to Draft or BIM objects but also to objects created with other workbenches. Up to 8 edges can be referenced by this snap option and Draft Snap Parallel, making it possible to snap to virtual intersections. Both snap options can also be combined with other snap options.	

Source: FreeCAD.org

Tool	Description	Image
Snap Parallel	The Draft Snap Parallel option snaps to an imaginary line parallel to straight edges. The edges can belong to Draft or BIM objects but also to objects created with other workbenches. Up to 8 edges can be referenced by this snap option and Draft Snap Extension, making it possible to snap to virtual intersections. Both snap options can also be combined with other snap options.	
Snap Special	The Draft Snap Special option snaps to special points defined by the object. The supported objects are Arch Walls, Arch Structures and Arch Equipment.	
Snap Near	The Draft Snap Near option snaps to the nearest point on faces and edges. The faces and edges can belong to Draft or BIM objects but also to objects created with other workbenches.	
Snap Ortho	The Draft Snap Ortho option snaps to imaginary lines that cross the previous point at multiples of 45°. The lines and angles are relative to the XY-plane of the working plane coordinate system.	

Source: FreeCAD.org

Tool	Description	Image
Snap Grid	The Draft Snap Grid option snaps to the intersections of grid lines.	
Snap Working Plane	The Draft Snap Working Plane option projects snap points onto the current working plane. It can only be used in combination with another snap option.	

Source: FreeCAD.org

3. CAD Modelling of 2D Objects

In engineering graphics, CAD modelling of 2D objects involves creating accurate two-dimensional representations of geometric shapes and plane surfaces using computer-aided design software. Instead of drawing manually with drafting instruments, CAD software such as AutoCAD or FreeCAD enables designers and students to construct precise drawings using digital tools and commands. These 2D models form the foundation for engineering drawings and are often used as the basis for developing more complex 3D models.

In engineering graphics, planes (also referred to as laminae, sheet surfaces, or flat solids) are thin, flat objects having negligible thickness and well-defined boundaries. The boundary of a plane may be polygonal, such as a triangle, square, rectangle, pentagon, or hexagon, or curvilinear, such as a circular plane. These plane surfaces represent two-dimensional faces of engineering components and are frequently used to understand projection, dimensioning, and geometric construction.

Using CAD tools, these plane shapes can be created by applying basic drawing commands such as line, circle, polygon, and arc, along with modification commands like trim, extend, and offset. CAD modelling of 2D objects allows users to control dimensions precisely, edit drawings easily, and maintain standard drawing conventions used in engineering practice.

3.1 Types of Planes Considered

In this exercise, the following planes (laminae or sheet surfaces) are used for CAD modelling and projection practice:

- ❖ Rectangular Plane (Rectangular Lamina)
- ❖ Square Plane (Square Lamina)
- ❖ Pentagonal Plane (Pentagonal Lamina)
- ❖ Hexagonal Plane (Hexagonal Lamina)
- ❖ Triangular Plane (Triangular Lamina)
- ❖ Circular Plane (Circular Sheet or Disk)

Each plane is assumed to be uniform and thin, representing a two-dimensional surface of a solid body. These planes help students understand how flat surfaces are represented and manipulated in CAD drawings. Figure 1 illustrates the basic terminologies and elements associated with plane surfaces, including edges, boundaries, vertices, and reference directions used in engineering graphics.

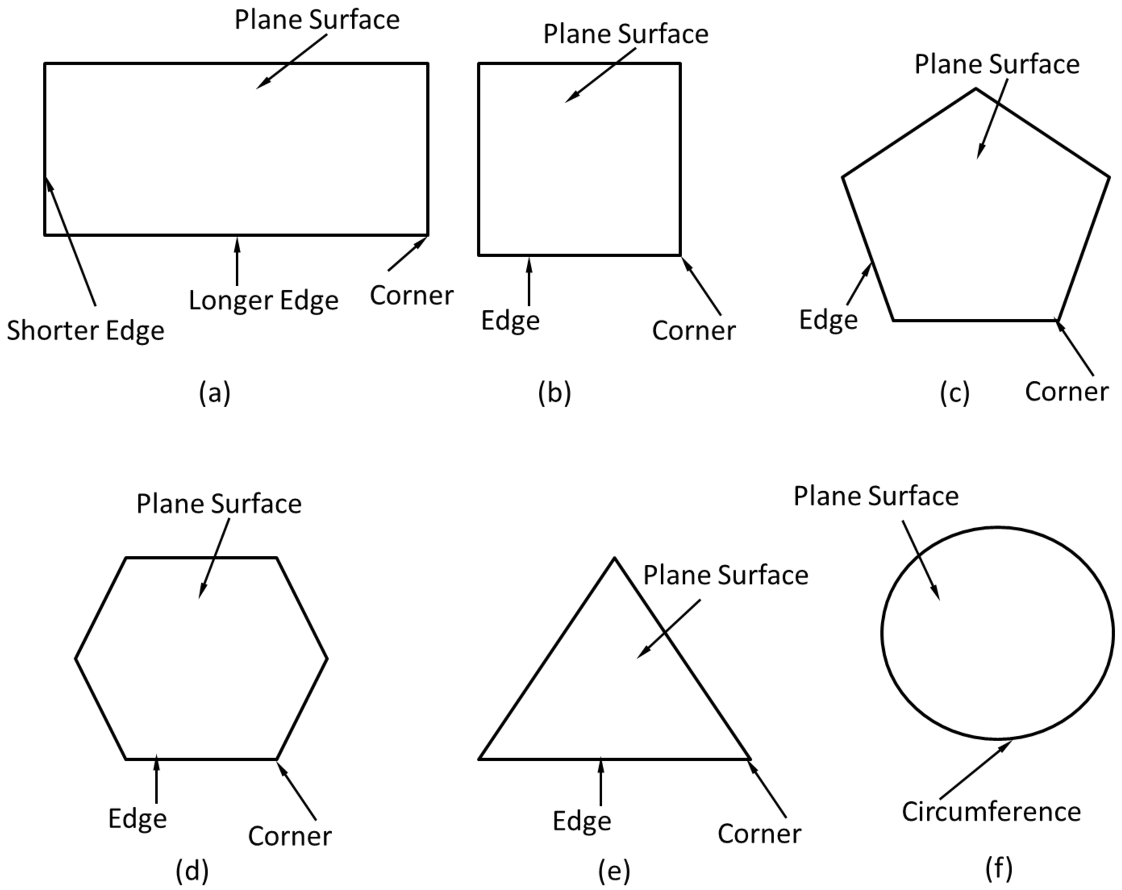


Figure 1 a) Rectangular plane, b) Square plane, c) Pentagonal plane, d) Hexagonal plane, e) Triangular plane, and f) Circular plane.

4. Virtual Demonstration of Sectioning of Truncated Solids

In engineering graphics, solids such as prisms, pyramids, cylinders, and cones are commonly used to represent real-world objects. Sometimes, these solids are not complete and are cut by an imaginary plane. This process is called sectioning of solids, and the remaining part is known as a truncated solid.

A truncated solid is formed when a solid is cut by a section plane and a portion of it is removed. The cut surface created by this process is called the section. Understanding how solids are sectioned is very important in engineering because many machine parts and structures are not in simple shapes; they are often cut, drilled, or modified.

Traditionally, sectioning of solids is taught using manual drawing methods. However, with the advancement of technology, virtual demonstration using CAD software has become an effective way to understand these concepts. Using software like AutoCAD or FreeCAD, students can visualize how a solid is cut and how the internal structure appears after sectioning.

In a virtual environment, the student can:

- Clearly see the cutting plane and its position
- Understand how the sectional view is formed
- Observe the true shape of the section
- Rotate and view the object from different angles

This makes learning more interactive and easier compared to traditional methods.

The section plane can cut a solid in different ways. In this exercise, the section plane is considered to be inclined to the base of the solid. This means the cutting plane is neither parallel nor perpendicular to the base, but at a certain angle. When a solid is cut by an inclined plane, the shape of the cut surface becomes more complex and is called the true shape of the section. Understanding this type of section is important because many engineering components are cut at angles in real applications. By using CAD software such as AutoCAD or FreeCAD, students can clearly visualize how the inclined section plane cuts the solid and how the truncated shape is formed. Each type of section produces a different shape, and understanding these variations is important for solving engineering problems.

Virtual demonstration helps students to develop spatial visualization skills, which are essential for engineers. It also improves accuracy and reduces errors in drawing. By observing the sectioning process step-by-step in CAD, students can better understand how to draw the sectional views in their engineering graphics exercises.

In this topic, students will learn how different solids are truncated using a section plane and how the resulting shapes are represented. The use of CAD tools will help in clearly visualizing the sectioning process and understanding the relationship between the original solid and the truncated part.

Overall, the virtual demonstration of sectioning of truncated solids provides a clear, practical, and modern approach to learning engineering graphics concepts, making it easier for students to understand and apply them in real engineering applications.

5. CAD Modelling of 3D Objects

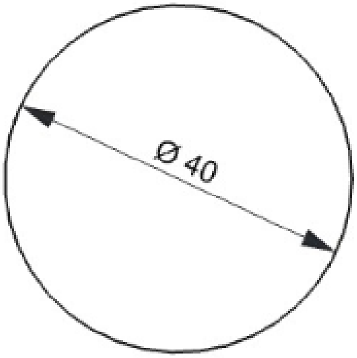
CAD modelling of 3D objects is the process of creating three-dimensional shapes using computer software. In engineering graphics, 3D modelling helps students and engineers to understand the actual shape, size, and structure of an object more clearly than 2D drawings.

Using CAD software such as AutoCAD or FreeCAD, 3D objects can be created by using basic commands like extrude, revolve, union, and subtract. These tools allow users to build solid models from simple shapes and modify them easily.

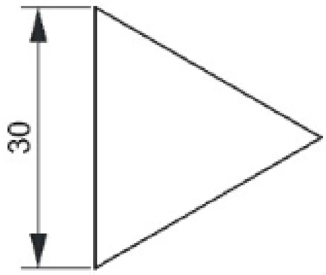
In this topic, 3D models are developed by interpreting given drawings or by creating shapes directly using CAD tools. Students can rotate, view, and analyze the object from different angles, which improves their visualization skills.

CAD modelling of 3D objects is very important in engineering because it is widely used in design, manufacturing, and product development. It helps in creating accurate models, reducing errors, and saving time.

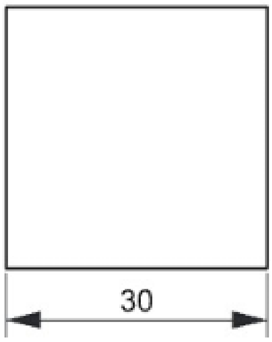
Overall, learning 3D CAD modelling provides a strong foundation for understanding real engineering components and preparing students for practical applications in industry.



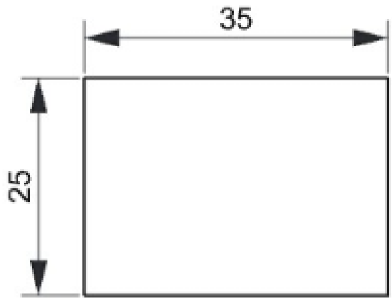
Circular Plane



Equilateral Triangular Plane

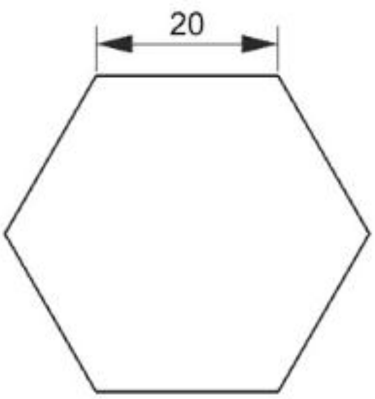


Square Plane

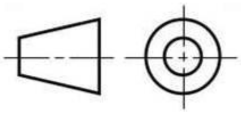


Rectangular Plane

Pentagonal Plane



Hexagonal Plane



SCALE 1:1

ALL DIMENSIONS ARE IN mm

Expt. No.: 1

Date:

CAD Modelling of Basic Plane Surfaces (2D Objects)

Aim

To create 2D CAD models of basic plane surfaces such as rectangular, square, triangular, circular, pentagonal, and hexagonal planes using CAD software.

Outcomes

After completing this experiment, the student will be able to:

- Understand the concept of plane surfaces (laminae).
- Create different 2D geometric shapes using CAD tools.
- Construct regular polygons such as pentagon and hexagon.
- Apply accurate dimensions and drawing techniques.
- Develop skills in CAD-based drafting.

Software/Hardware Required

- AutoCAD / FreeCAD
- Computer system

Theory

In engineering graphics, plane surfaces (also called laminae or sheets) are flat objects having negligible thickness and defined boundaries.

The boundary of a plane may be:

- Polygonal → Triangle, Square, Rectangle, Pentagon, Hexagon
- Curvilinear → Circle

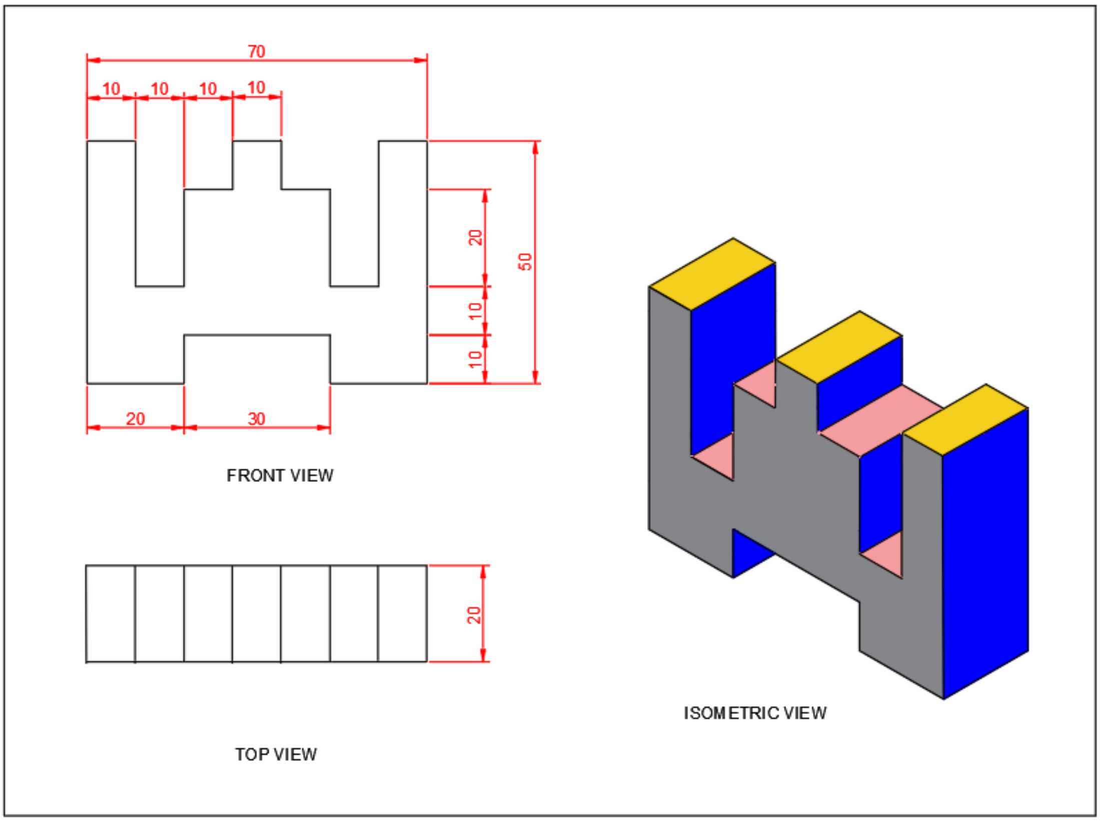
These plane surfaces are the basic building blocks for engineering drawings and 3D modelling. Using CAD software, these shapes can be created accurately using simple commands like line, circle, and polygon.

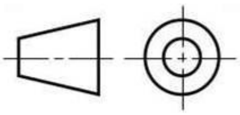
Procedure

1. Open the CAD software.
2. Set units to millimetres (mm).
3. Set drawing limits.
4. Draw Rectangular Plane (Use Rectangle tool)
5. Enter given length and width (35x25).
6. Draw Square Plane (Use Rectangle tool)
7. Enter equal length and width (30x30).
8. Draw Triangular Plane (Use Line tool or Polygon (3 sides, side 30))
9. Connect all sides properly.
10. Draw Circular Plane (Use Circle command)
11. Enter radius or diameter (40).
12. Draw Pentagonal Plane (Use Polygon command)
13. Enter number of sides = 5.
14. Specify size 20 (inscribed or circumscribed as per drawing).
15. Draw Hexagonal Plane (Use Polygon command)
16. Enter number of sides = 6.
17. Specify size 20.
18. Dimensioning
19. Apply dimensions to all shapes.
20. Arrange drawings neatly.
21. Save the drawing.

Result

The 2D CAD models of plane surfaces including rectangular, square, triangular, circular, pentagonal, and hexagonal planes were successfully created using CAD software.





SCALE 1:1

ALL DIMENSIONS ARE IN mm

Expt. No.: 2

Date:

CAD Modelling of Stepped Block with Slots from Orthographic Views

Aim

To create a 3D CAD model directly from the given orthographic views and generate its isometric view using CAD software.

Outcomes

After completing this exercise, the student will be able to:

- Interpret orthographic views (front and top view).
- Visualize the 3D shape from 2D drawings.
- Create 3D models directly without redrawing 2D views.
- Use basic 3D commands in AutoCAD or FreeCAD.
- Generate and understand isometric views of objects.

Software/Hardware Required

- AutoCAD / FreeCAD
- Computer system

Theory

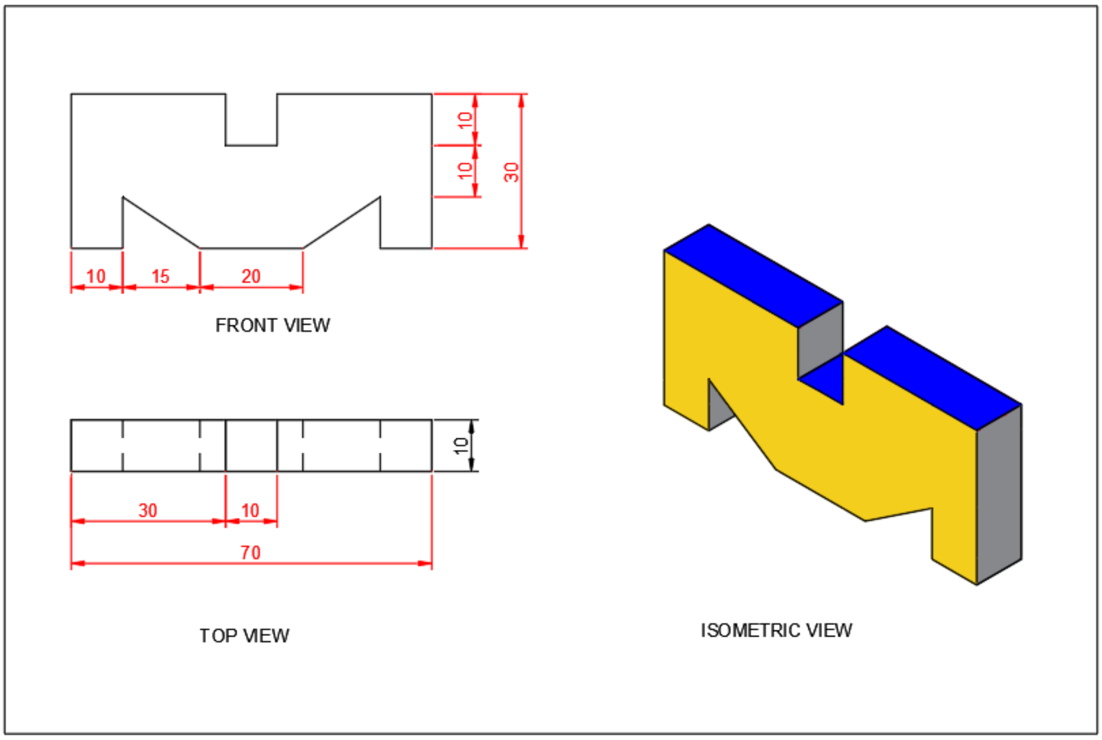
In engineering graphics, orthographic views provide all the necessary dimensions and shape information of a 3D object. By carefully analyzing these views, a 3D model can be constructed without recreating the 2D drawings. This method improves spatial visualization skills, which is essential in engineering design and modelling.

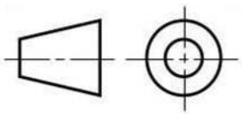
Procedure

1. Observe the front view and top view carefully.
2. Identify:
   - Overall length, width, and height
   - Step features and cut portions
   - Thickness of each section
3. Start CAD software and switch to 3D modelling workspace.
4. Create the base block using the overall length, width, and height.
5. Use commands like BOX or EXTRUDE.
6. Based on the front view, create raised portions.
7. Use EXTRUDE to build different levels.
8. Combine parts using UNION.
9. Identify the cut portions from the views.
10. Create cutting shapes.
11. Use SUBTRACT command to remove unwanted material.
12. Use 3D view tools to display the object in isometric form.
13. Apply shading or visual styles if required.
14. Verify dimensions with the given views.
15. Save the model.

Result

The 3D model of the stepped block with slots was successfully created from the given orthographic views, and its isometric view was generated.





SCALE 1:1

ALL DIMENSIONS ARE IN mm

Expt. No.: 3

Date:

CAD Modelling of Notched Stepped Block from Orthographic Views

Aim

To create a 3D CAD model of a Notched Stepped Block directly from the given orthographic views (front and top view) and generate its isometric view.

Outcomes

After completing this exercise, the student will be able to:

- Understand and interpret orthographic views.
- Visualize the 3D shape from 2D drawings.
- Create a 3D model directly without drawing 2D views.
- Use basic 3D modelling commands in AutoCAD or FreeCAD.
- Generate and display the isometric view of a 3D object.

Software/Hardware Required

- AutoCAD / FreeCAD
- Computer system

Theory

In engineering graphics, orthographic views such as front view and top view provide complete information about the shape and size of a 3D object. By carefully studying these views, the object can be visualized and modelled in three dimensions.

In this exercise, the given object is a Notched Stepped Block, which consists of:

- Step features
- Inclined surfaces
- Notches (cut portions)

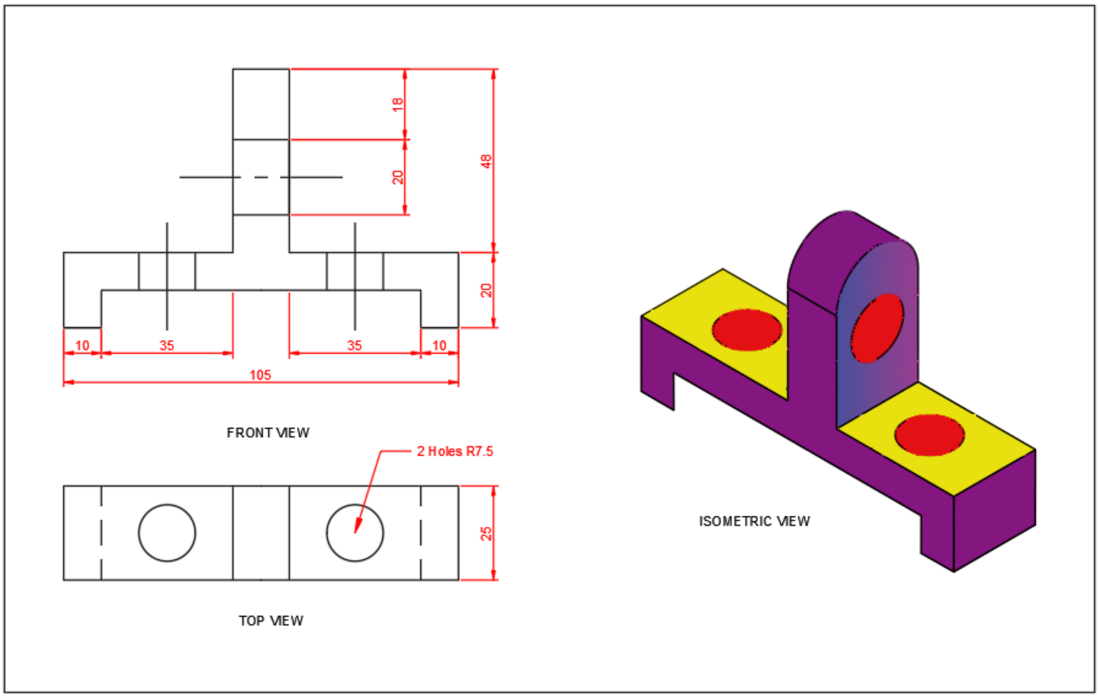
Students must interpret the given views and create the 3D model directly using CAD tools.

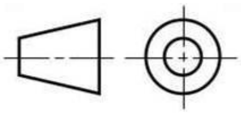
Procedure

1. Carefully observe the front view and top view.
2. Identify:
    - Overall length, width, and height
    - Step levels
    - Inclined faces
    - Notches and cut sections
3. Open the CAD software and switch to 3D modelling workspace.
4. Set the appropriate units and view.
5. Create a base block using overall dimensions from the top and front views.
6. Use BOX or EXTRUDE command.
7. Build the stepped portions by adding material.
8. Use EXTRUDE to raise required sections.
9. Combine solids using UNION.
10. Identify the inclined face from the front view.
11. Create the inclined cut using appropriate sketch and SUBTRACT command.
12. Draw the notch profiles based on given dimensions.
13. Remove material using SUBTRACT.
14. Use 3D view tools (Orbit/ViewCube) to obtain the isometric view.
15. Apply shading or visual style if required.
16. Check the model with given dimensions.
17. Save the file.

Result

The 3D CAD model of the Notched Stepped Block was successfully created from the given orthographic views, and the isometric view was obtained.





SCALE 1:1 ALL DIMENSIONS ARE IN mm

Expt. No.: 4

Date:

CAD Modelling of Bracket with Vertical Support and Holes from Orthographic Views

Aim

To create a 3D CAD model of a Bracket with Vertical Support and Holes directly from the given orthographic views (front and top view) and generate its isometric view.

Outcomes

After completing this exercise, the student will be able to:

- Interpret orthographic views of an object.
- Visualize and understand the 3D structure from 2D drawings.
- Create a 3D model directly without drawing 2D views.
- Use basic 3D modelling commands in AutoCAD or FreeCAD.
- Develop skills in creating holes and vertical features in 3D objects.
- Generate the isometric view of the object.

Software/Hardware Required

- AutoCAD / FreeCAD
- Computer system

Theory

In engineering graphics, orthographic views provide complete information about an object's geometry. By analyzing these views, a 3D object can be visualized and modelled.

The given object is a Bracket with Vertical Support and Holes, which includes:

- A base plate
- A vertical curved support
- Through holes in the base and support

This type of component is commonly used in mechanical supports and mounting applications.

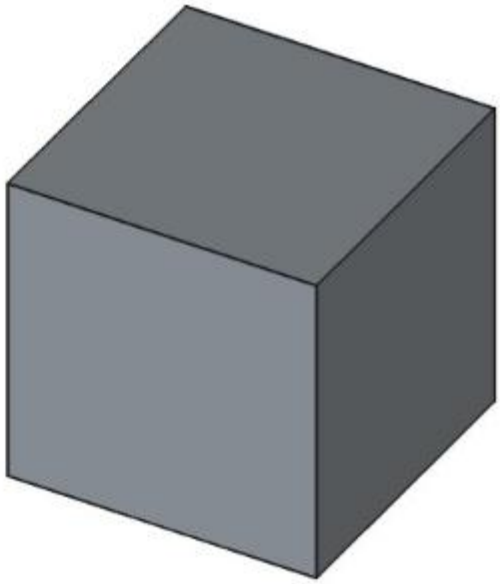
Procedure

1. Observe the front view and top view carefully.
2. Identify:
    - Overall length, width, and height
    - Thickness of base plate
    - Position of vertical support
    - Diameter and position of holes
3. Open the CAD software and switch to 3D modelling workspace.
4. Set units and view.
5. Create a rectangular base using BOX or EXTRUDE.
6. Use the given dimensions for length, width, and thickness.
7. Draw the profile of the vertical support (including curved top).
8. Use EXTRUDE to give thickness.
9. Place it correctly on the base.
10. Draw circles on the top surface at specified positions.
11. Use EXTRUDE (cut) or SUBTRACT to create holes in the base.
12. Similarly, create the hole in the vertical support.
13. Use UNION to combine solids if required.
14. Check alignment and dimensions.
15. Use 3D Orbit/View tools to obtain the isometric view.
16. Apply shading or visual styles if needed.

17. Verify all dimensions with the given views.
18. Save the model.

Result

The 3D CAD model of the Bracket with Vertical Support and Holes was successfully created from the given orthographic views, and the isometric view was generated.



Expt. No.: 5

Date:

CAD Modelling of Cube

Aim

To create a 3D CAD model of a cube of side 30 mm and generate its isometric view using CAD software.

Outcomes

After completing this experiment, the student will be able to:

- Understand the geometry of a cube.
- Create basic 3D solid models using CAD tools.
- Develop models directly from given drawings/problem statements.
- Generate and visualize the isometric view of a 3D object.
- Improve spatial visualization skills.

Software/Hardware Required

- AutoCAD / FreeCAD
- Computer system

Theory

A cube is a three-dimensional solid having six equal square faces, twelve edges, and eight vertices. All edges of a cube are equal in length.

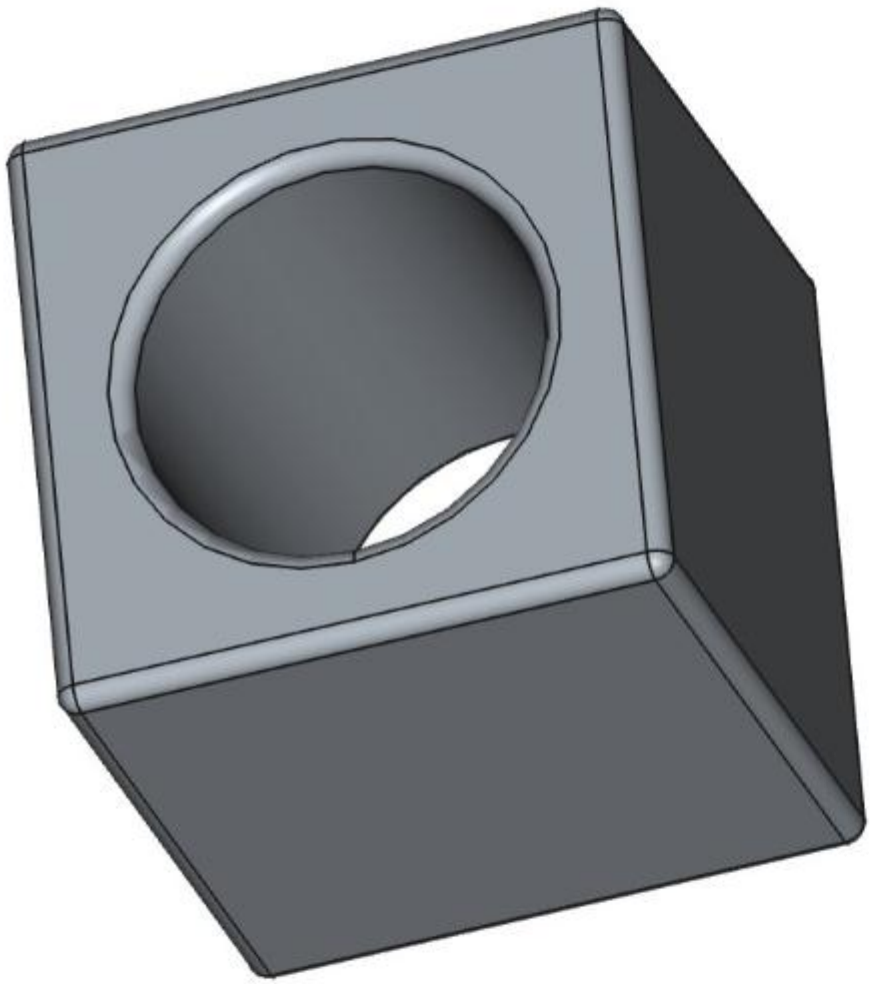
In CAD modelling, a cube can be easily created using commands like BOX or by extruding a square. Once created, the object can be viewed from different directions, including the isometric view, which shows the 3D shape clearly.

Procedure

1. Open FreeCAD.
2. Switch to Part Design (Parametric Part) workbench.
3. Create a New Body
4. Click on Create Body (Parametric Part).
5. Select the body to start modelling.
6. Step 3: Create Sketch
7. Click on Create Sketch.
8. Select XY Plane as the base plane.
9. Draw a Square (Use Sketcher Geometries tools)
10. Apply constraints and set dimensions: Length = 30 mm, Width = 30 mm.
11. Create 3D Solid (Click Pad (Extrude))
12. Enter height = 30 mm.
13. Click OK to create the cube.
14. Apply shading for better visualization.
15. Verify dimensions.
16. Save the file.

Result

The 3D CAD model of a cube of side 30 mm was successfully created, and its isometric view was generated.



Expt. No.: 6

Date:

CAD Modelling of Cube with Through Hole and Fillet

Aim

To create a 3D CAD model of a cube with a through hole and filleted edges using CAD software and generate its isometric view.

Outcomes

After completing this experiment, the student will be able to:

- Create a basic 3D solid (cube).
- Apply sketch-based modelling techniques.
- Create a through hole using sketch and cut operation.
- Apply fillet to edges and circular features.
- Develop skills in parametric modelling using CAD tools.

Software/Hardware Required

- FreeCAD
- Computer system

Theory

In CAD modelling, complex objects are created by combining basic operations such as sketching, extrusion, cutting, and filleting.

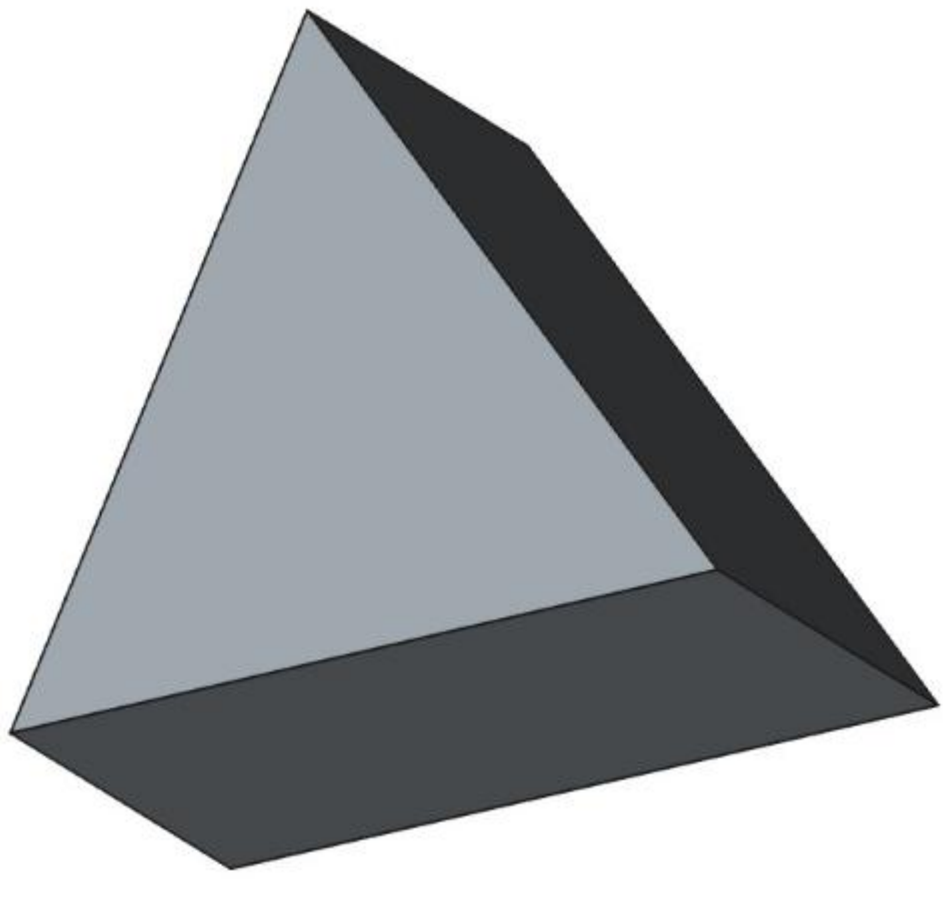
Procedure (Using FreeCAD – Part Design Workbench)

1. Open FreeCAD.
2. Switch to Part Design (Parametric Part) workbench.
3. Create Base Cube
4. Click Create Body. Click Create Sketch. Select XY Plane as base plane.

5. Use Sketcher Geometries to draw a square (Apply dimensions: 30 mm × 30 mm).
6. Close the sketch.
7. Click Pad and enter height = 30 mm.
8. Select the top face of the cube.
9. Click Create Sketch.
10. Draw a circle at the center (Set diameter = 20 mm).
11. Close the sketch.
12. Use Pocket (cut) and set depth = 30 mm (through all).
13. Apply Fillet
14. Select all outer edges of the cube.
15. Apply Fillet with suitable radius (Radius 1 mm).
16. Select circular edge of the hole.
17. Apply Fillet to smooth the edge.
18. Generate Isometric View
19. Use View → Isometric.
20. Apply shading for better visualization.
21. Verify dimensions and features.
22. Save the model.

Result

The 3D CAD model of a cube with a through hole and filleted edges was successfully created using FreeCAD, and the isometric view was generated.



Expt. No.: 7

Date:

CAD Modelling of Equilateral Triangular Prism

Aim

To create a 3D CAD model of an equilateral triangular prism and generate its isometric view using CAD software.

Outcomes

After completing this experiment, the student will be able to:

- Understand the geometry of a triangular prism.
- Create 2D sketches and convert them into 3D solids.
- Use parametric modelling tools in CAD software.
- Generate and visualize the isometric view of a 3D object.
- Improve modelling accuracy and visualization skills.

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A triangular prism is a three-dimensional solid having two triangular bases and three rectangular faces. In an equilateral triangular prism, the base triangle has all sides equal.

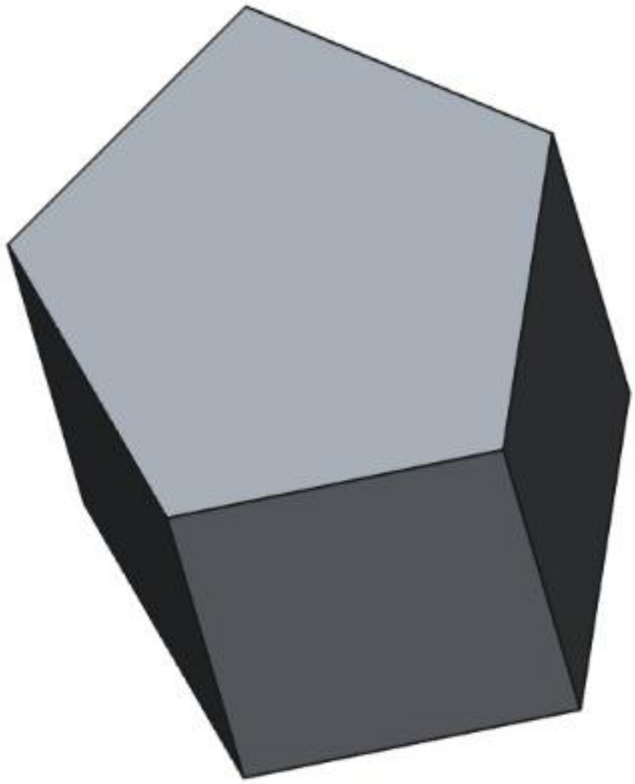
In CAD modelling, such solids are created by drawing the base shape and then extending it along a direction using the Pad (extrude) operation. This helps in converting a 2D sketch into a 3D solid.

Procedure

1. Open FreeCAD.
2. Switch to Part Design (Parametric Part) workbench.
3. Click Create Body. Click Create Sketch. Select XY Plane as the base plane.
4. Draw a triangle (Use Sketcher Geometries tools)
5. Apply constraints to make it equilateral. Set each side = 30 mm.
6. Close the sketch.
7. Click Pad.
8. Enter height = 50 mm.
9. Click OK to create the prism.
10. Use View → Isometric view.
11. Apply shading for better visualization.
12. Verify dimensions.
13. Save the model.

Result

The 3D CAD model of an equilateral triangular prism was successfully created using FreeCAD, and the isometric view was generated.



Expt. No.: 8

Date:

CAD Modelling of Pentagonal Prism

Aim

To create a pentagonal prism of side 30 mm and height 70 mm using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model using Pad
- Understand parametric modelling concepts

Software/Hardware Required

- FreeCAD
- Computer system

Theory

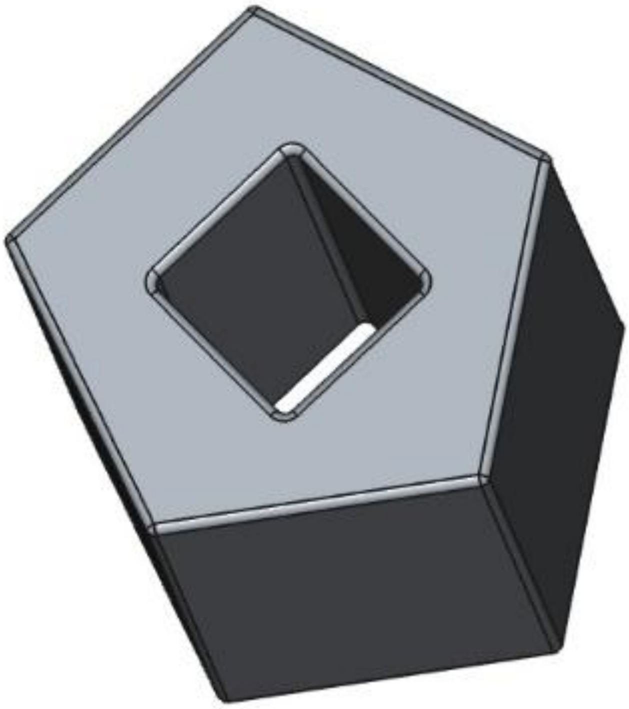
A pentagonal prism is a three-dimensional geometric solid that consists of a pentagon as its base and five rectangular faces as its lateral surfaces, with a uniform height. In CAD modelling, such solids are created using a parametric approach where a two-dimensional sketch is first developed and then converted into a three-dimensional model. The sketch defines the base geometry with proper dimensions and constraints, ensuring accuracy and flexibility in design modification. The Pad operation in FreeCAD is used to extrude the 2D sketch along a specified direction to obtain the required 3D shape. This method allows easy editing and updating of the model by changing parameters like side length and height.

Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body. Click Create Sketch and select XY Plane.
3. Draw a pentagon using Sketcher tools. Apply constraints so that each side is 30 mm
4. Close the sketch
5. Select the sketch and click Pad
6. Enter height as 70 mm and click OK to create the prism
7. Use View → Isometric view
8. Apply shading for better visualization
9. Verify dimensions
10. Save the model

Result

The CAD model of a pentagonal prism with side 30 mm and height 70 mm is successfully created using FreeCAD.



Expt. No.: 9

Date:

CAD Modelling of Pentagonal Prism with Square Hole and Fillet

Aim

To create a pentagonal prism of side 30 mm and height 70 mm with a central square hole of side 20 mm and depth 70 mm, and apply fillet of radius 1 mm to all edges using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model using Pad
- Create internal features like holes using sketch and cut operations
- Apply fillet to edges
- Understand parametric modelling concepts

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A pentagonal prism with internal features is a three-dimensional solid that consists of a pentagonal base and rectangular lateral faces, along with a square hole passing through the entire height. In CAD modelling, such components are created using a parametric approach where the base sketch is first developed and then converted into a three-dimensional model using the Pad operation. Additional features like holes are created by sketching on the surface and removing material using pocket or cut operations.

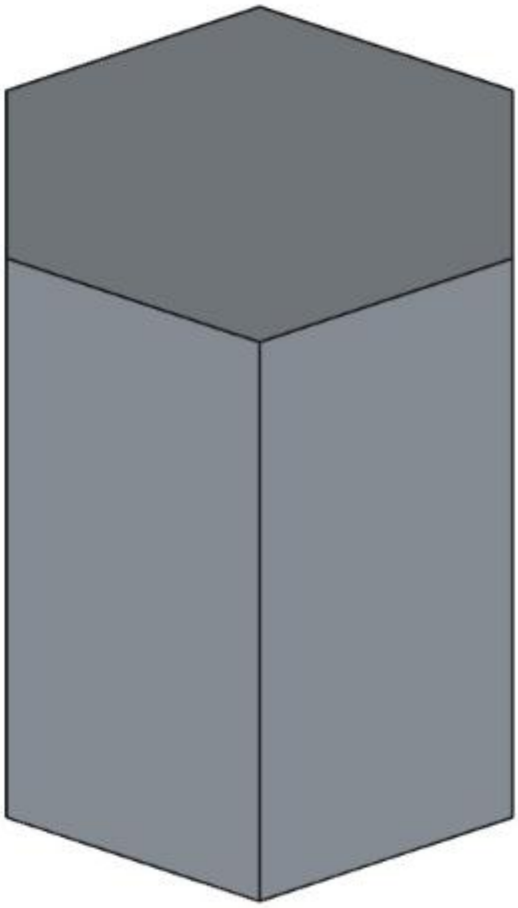
Fillet is applied to edges to smooth sharp corners and improve the design. This approach ensures accuracy, flexibility, and easy modification of dimensions such as side length, hole size, and fillet radius.

Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body. Click Create Sketch and select XY Plane.
3. Draw a pentagon using Sketcher tools. Apply constraints so that each side is 30 mm.
4. Close the sketch.
5. Select the sketch and click Pad.
6. Enter height as 70 mm and click OK to create the prism.
7. Select the top face of the prism and click Create Sketch.
8. Draw a square at the center using Sketcher tools. Apply constraints so that each side is 20 mm.
9. Close the sketch.
10. Select the sketch and use Pocket (cut) operation.
11. Set depth as 70 mm to create a through square hole.
12. Select all outer edges and inner square edges.
13. Apply Fillet with radius 1 mm.
14. Use View → Isometric view.
15. Apply shading for better visualization.
16. Verify dimensions.
17. Save the model.

Result

The CAD model of a pentagonal prism with a central square hole and fillet is successfully created using FreeCAD.



Expt. No.: 10

Date:

CAD Modelling of Hexagonal Prism

Aim

To create a hexagonal prism of side 30 mm and height 70 mm using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model using Pad
- Understand parametric modelling concepts

Software/Hardware Required

- FreeCAD
- Computer system

Theory

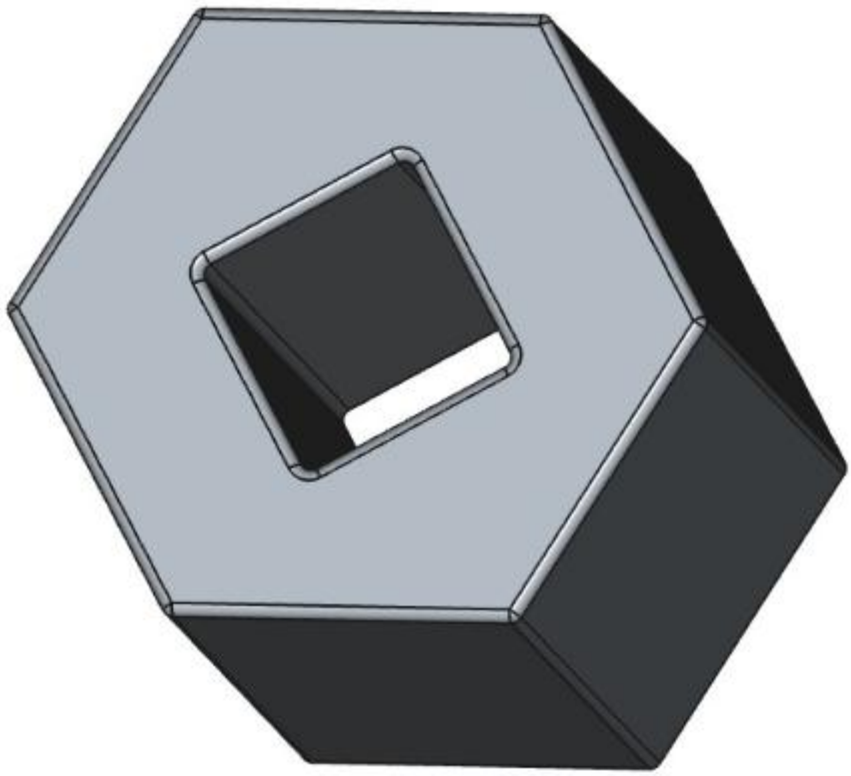
A hexagonal prism is a three-dimensional geometric solid that consists of a hexagon as its base and six rectangular faces as its lateral surfaces, with a uniform height. In CAD modelling, such solids are created using a parametric approach where a two-dimensional sketch is first developed and then converted into a three-dimensional model. The sketch defines the base geometry with proper dimensions and constraints, ensuring accuracy and flexibility in design modification. The Pad operation in FreeCAD is used to extrude the 2D sketch along a specified direction to obtain the required 3D shape. This method allows easy editing and updating of the model by changing parameters like side length and height.

Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body. Click Create Sketch and select XY Plane.
3. Draw a hexagon using Sketcher tools. Apply constraints so that each side is 30 mm.
4. Close the sketch.
5. Select the sketch and click Pad.
6. Enter height as 70 mm and click OK to create the prism.
7. Use View → Isometric view.
8. Apply shading for better visualization.
9. Verify dimensions.
10. Save the model.

Result

The CAD model of a hexagonal prism with side 30 mm and height 70 mm is successfully created using FreeCAD.



Expt. No.: 11

Date:

CAD Modelling of Hexagonal Prism with Square Hole and Fillet

Aim

To create a hexagonal prism of side 30 mm and height 70 mm with a central square hole of side 20 mm and depth 70 mm, and apply fillet of radius 1 mm to all edges using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model using Pad
- Create internal features like holes using sketch and cut operations
- Apply fillet to edges
- Understand parametric modelling concepts

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A hexagonal prism with internal features is a three-dimensional geometric solid that consists of a hexagon as its base and six rectangular lateral faces, along with a square hole passing through the entire height. In CAD modelling, such components are created using a parametric approach where the base sketch is first developed and then converted into a three-dimensional model using the Pad operation.

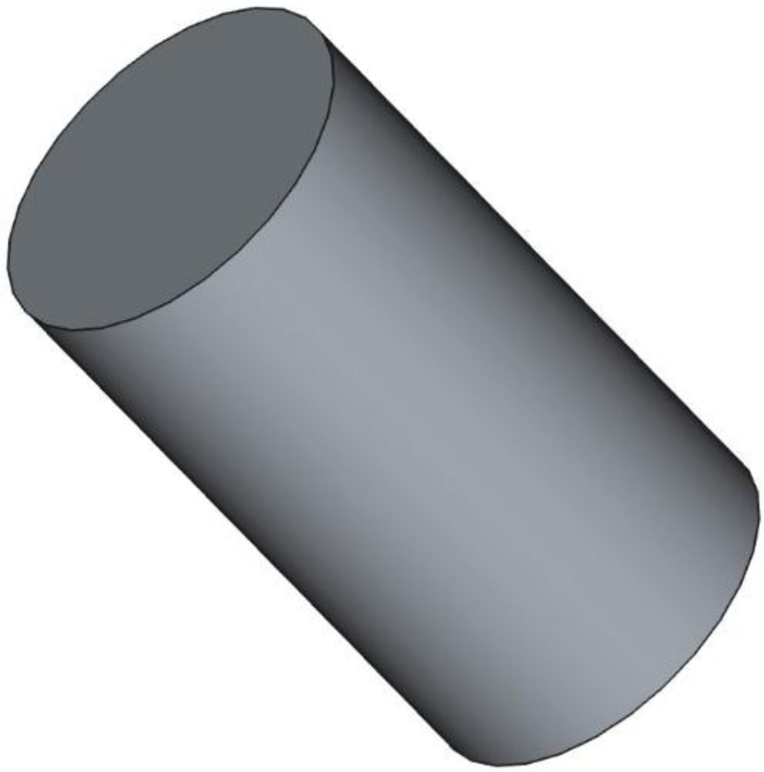
Additional features like holes are created by sketching on the surface and removing material using pocket or cut operations. Fillet is applied to edges to smooth sharp corners and improve the design. This approach ensures accuracy, flexibility, and easy modification of dimensions such as side length, hole size, and fillet radius.

Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body. Click Create Sketch and select XY Plane.
3. Draw a hexagon using Sketcher tools. Apply constraints so that each side is 30 mm.
4. Close the sketch.
5. Select the sketch and click Pad.
6. Enter height as 70 mm and click OK to create the prism.
7. Select the top face of the prism and click Create Sketch.
8. Draw a square at the center using Sketcher tools. Apply constraints so that each side is 20 mm.
9. Close the sketch.
10. Select the sketch and use Pocket (cut) operation.
11. Set depth as 70 mm to create a through square hole.
12. Select all outer edges and inner square edges.
13. Apply Fillet with radius 1 mm.
14. Use View → Isometric view.
15. Apply shading for better visualization.
16. Verify dimensions.
17. Save the model.

Result

The CAD model of a hexagonal prism with a central square hole and fillet of radius 1 mm is successfully created using FreeCAD.



Expt. No.: 12

Date:

CAD Modelling of Cylinder

Aim

To create a cylinder of diameter 40 mm and height 70 mm using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model using Pad
- Understand parametric modelling concepts

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A cylinder is a three-dimensional geometric solid that consists of a circular base and a curved lateral surface with a uniform height. In CAD modelling, cylinders are created using a parametric approach where a two-dimensional circle is first drawn and then converted into a three-dimensional model. The sketch defines the base geometry with proper dimensions and constraints to ensure accuracy. The Pad operation in FreeCAD is used to extrude the 2D circular sketch along a specified direction to obtain the required 3D cylindrical shape. This method allows easy modification of dimensions such as diameter and height.

Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body. Click Create Sketch and select XY Plane.
3. Draw a circle using Sketcher tools. Apply constraint so that diameter is 40 mm.
4. Close the sketch.
5. Select the sketch and click Pad.
6. Enter height as 70 mm and click OK to create the cylinder.
7. Use View → Isometric view.
8. Apply shading for better visualization.
9. Verify dimensions.
10. Save the model.

Result

The CAD model of a cylinder with diameter 40 mm and height 70 mm is successfully created using FreeCAD.

Expt. No.: 13

Date:

CAD Modelling of Cylinder with Through Hole and Fillet

Aim

To create a 3D CAD model of a cylinder with a through hole and fillet applied along the circumference using CAD software and generate its isometric view.

Outcomes

After completing this experiment, the student will be able to:

- Create a basic cylindrical solid.
- Develop models using sketch-based parametric design.
- Create a through hole using sketch and cut operation.
- Apply fillet to edges and circular features.
- Improve 3D modelling and visualization skills.

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A cylinder is a three-dimensional solid having circular top and bottom faces. The boundary of these circular faces is called the circumference.

In this model:

- A cylinder of diameter 40 mm and height 70 mm is created
- A through hole of diameter 20 mm is made along the axis
- Fillet is applied along the circumference of the outer surface and the hole to smooth the model.

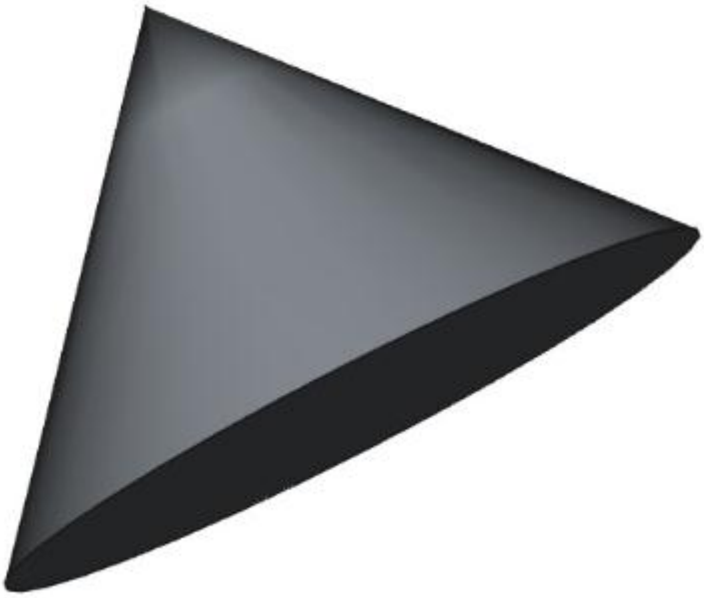
This improves both the appearance and functional performance of the component.

Procedure

1. Open FreeCAD and switch to Part Design workbench.
2. Click Create Body.
3. Click Create Sketch and select XY Plane.
4. Use Sketcher Geometries to draw a circle and set diameter = 40 mm.
5. Close the sketch and click Pad.
6. Enter height = 70 mm to create the cylinder.
7. Select the top face and click Create Sketch.
8. Draw a circle at the center and set diameter = 20 mm.
9. Close the sketch and use Pocket (cut) with depth = 70 mm (through all).
10. Select the circumference of the top face, bottom face, and hole.
11. Apply Fillet with suitable radius (radius 1 mm).
12. Set the view to Isometric and apply shading.
13. Save the model.

Result

The 3D CAD model of a cylinder with a through hole and fillet applied along the circumference was successfully created using FreeCAD, and the isometric view was obtained.



Expt. No.: 14

Date:

CAD Modelling of Cone

Aim

To create a 3D CAD model of a cone and generate its isometric view using CAD software.

Outcomes

After completing this experiment, the student will be able to:

- Create a conical solid using CAD tools.
- Develop models using a parametric sketch-based approach.
- Understand the geometry of a cone (base and apex).
- Convert a 2D sketch into a 3D object.
- Improve 3D modelling and visualization skills.

Software / Hardware Required

- FreeCAD
- Computer system

Theory

A cone is a three-dimensional solid having a circular base and a single point called the apex. The surface smoothly tapers from the base to the apex.

In this model:

- Base diameter = 60 mm
- Height = 70 mm

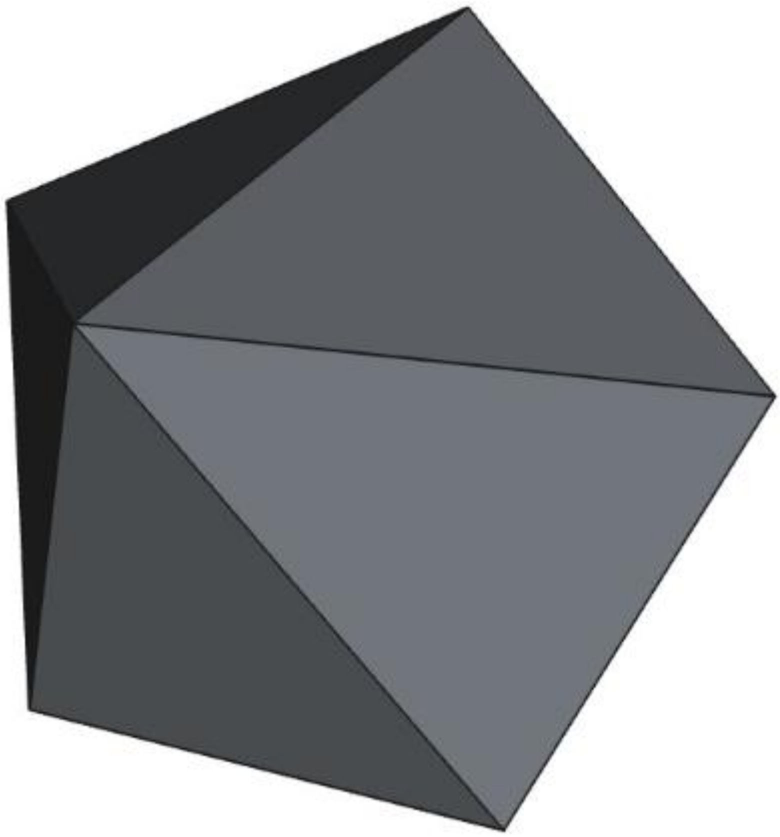
In CAD, a cone can be created using sketch-based modelling and appropriate solid features. It is widely used in engineering applications such as funnels, nozzles, and machine parts.

Procedure

1. Open FreeCAD and switch to Part Design workbench.
2. Click Create Body.
3. Click Create Sketch and select XY Plane as the base plane.
4. Use Sketcher Geometries to draw a circle.
5. Set diameter = 60 mm.
6. Close the sketch.
7. Create the cone using appropriate 3D operation (such as Additive Loft or Part tool for cone creation).
8. Set height = 70 mm and ensure the top converges to a point (apex).
9. Set the view to Isometric.
10. Apply shading for better visualization.
11. Save the model.

Result

The 3D CAD model of a cone (diameter 60 mm and height 70 mm) was successfully created using FreeCAD, and the isometric view was obtained.



Expt. No.: 15

Date:

CAD Modelling of Pentagonal Pyramid

Aim

To create a 3D CAD model of a pentagonal pyramid and generate its isometric view using CAD software.

Outcomes

After completing this experiment, the student will be able to:

- Create a pentagonal base using CAD tools.
- Develop models using a parametric sketch-based approach.
- Understand the geometry of a pyramid (base and apex).
- Convert a 2D sketch into a 3D object.
- Improve 3D modelling and visualization skills.

Software / Hardware Required

- FreeCAD
- Computer system

Theory

A pentagonal pyramid is a three-dimensional solid having a pentagonal base and five triangular faces that meet at a common point called the apex.

In this model:

- Base: Regular pentagon of side 30 mm
- Height: 70 mm

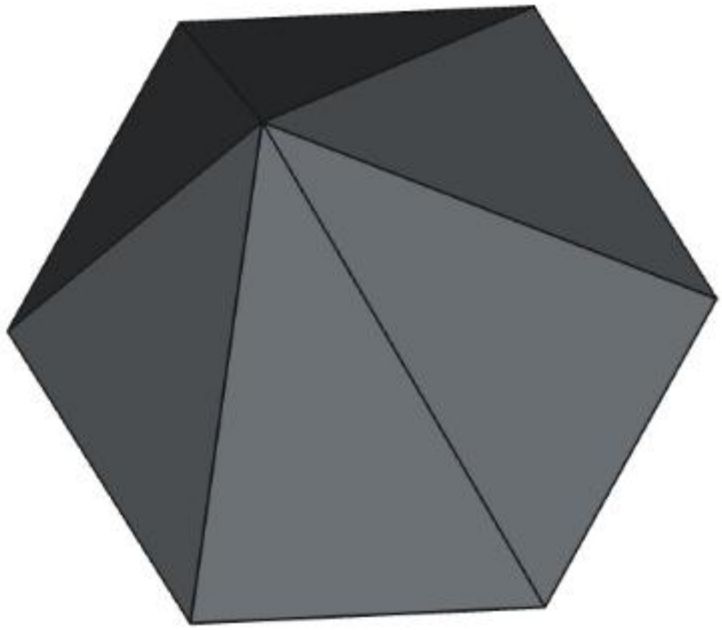
In CAD modelling, pyramids are created by forming a base and then joining it to an apex point using suitable operations.

Procedure

1. Open FreeCAD and switch to Part Design workbench.
2. Click Create Body.
3. Click Create Sketch and select XY Plane as the base plane.
4. Use Sketcher Geometries to draw a regular pentagon.
5. Set side length = 30 mm.
6. Close the sketch.
7. Create the pyramid using suitable 3D operation (such as loft or draft method to form apex).
8. Set the height of the pyramid = 70 mm.
9. Set the view to Isometric.
10. Apply shading for better visualization.
11. Save the model.

Result

The 3D CAD model of a pentagonal pyramid (base side 30 mm and height 70 mm) was successfully created using FreeCAD, and the isometric view was obtained.



Expt. No.: 16

Date:

CAD Modelling of Hexagonal Pyramid

Aim

To create a 3D CAD model of a hexagonal pyramid and generate its isometric view using CAD software.

Outcomes

After completing this experiment, the student will be able to:

- Create a hexagonal base using CAD tools.
- Develop models using a parametric sketch-based approach.
- Understand the geometry of a pyramid (base and apex).
- Convert a 2D sketch into a 3D object.
- Improve 3D modelling and visualization skills.

Software / Hardware Required

- FreeCAD
- Computer system

Theory

A hexagonal pyramid is a three-dimensional solid having a hexagonal base and six triangular faces that meet at a common point called the apex.

In this model:

- Base: Regular hexagon of side 30 mm
- Height: 70 mm

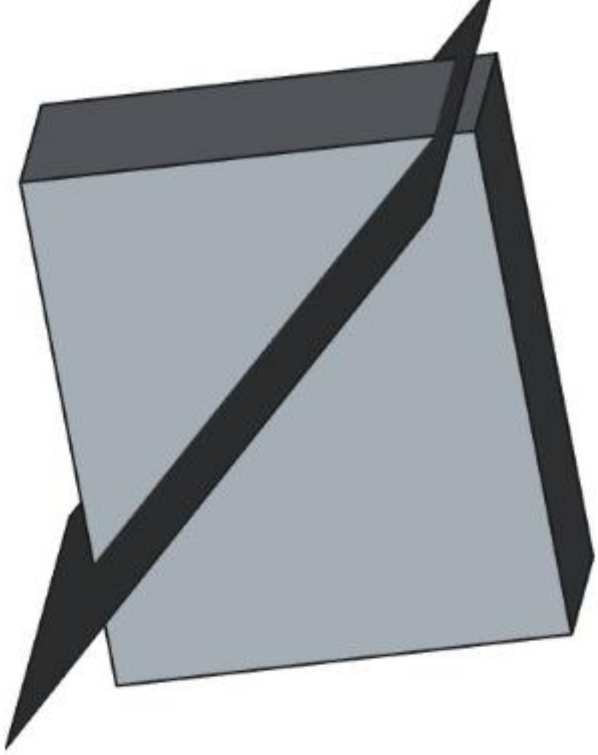
In CAD modelling, such solids are created by first drawing the base and then forming the sloping faces meeting at the apex using suitable operations.

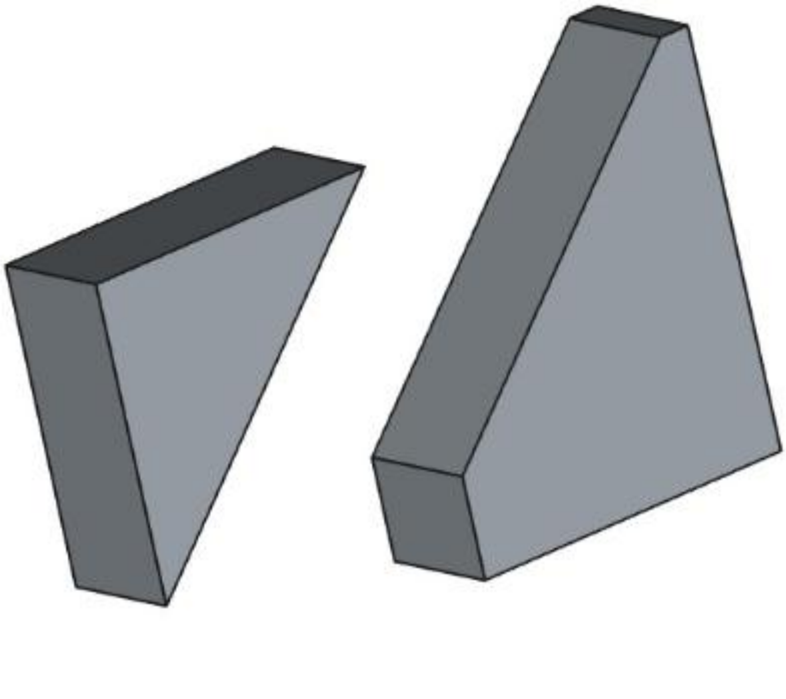
Procedure

1. Open FreeCAD and switch to Part Design workbench.
2. Click Create Body.
3. Click Create Sketch and select XY Plane as the base plane.
4. Use Sketcher Geometries to draw a regular hexagon.
5. Set side length = 30 mm.
6. Close the sketch.
7. Create the pyramid using a suitable 3D operation (such as loft or draft method to form the apex).
8. Set the height of the pyramid = 70 mm.
9. Set the view to Isometric.
10. Apply shading for better visualization.
11. Save the model.

Result

The 3D CAD model of a hexagonal pyramid (base side 30 mm and height 70 mm) was successfully created using FreeCAD, and the isometric view was obtained.





Expt. No.: 17

Date:

CAD Modelling of Truncated Rectangular Prism

Aim

To create a 3D CAD model of a truncated rectangular prism using a cutting plane and generate its isometric view.

Outcomes

After completing this experiment, the student will be able to:

- Create a rectangular prism using sketch-based modelling.
- Understand the concept of truncation of solids.
- Apply cutting plane operations in CAD.
- Perform splitting of solids using a plane.
- Visualize sectioned and truncated solids.

Software / Hardware Required

- FreeCAD
- Computer system

Theory

A truncated solid is obtained when a solid is cut by a plane and a portion is removed.

In this exercise:

- A rectangular prism of base 20 mm × 50 mm and height 70 mm is created

A cutting plane is applied which is:

- Inclined at 45° to the Horizontal Plane (H.P)
- Perpendicular to the Vertical Plane (V.P)

- The plane cuts the solid at a point 10 mm from the left side of the right top corner

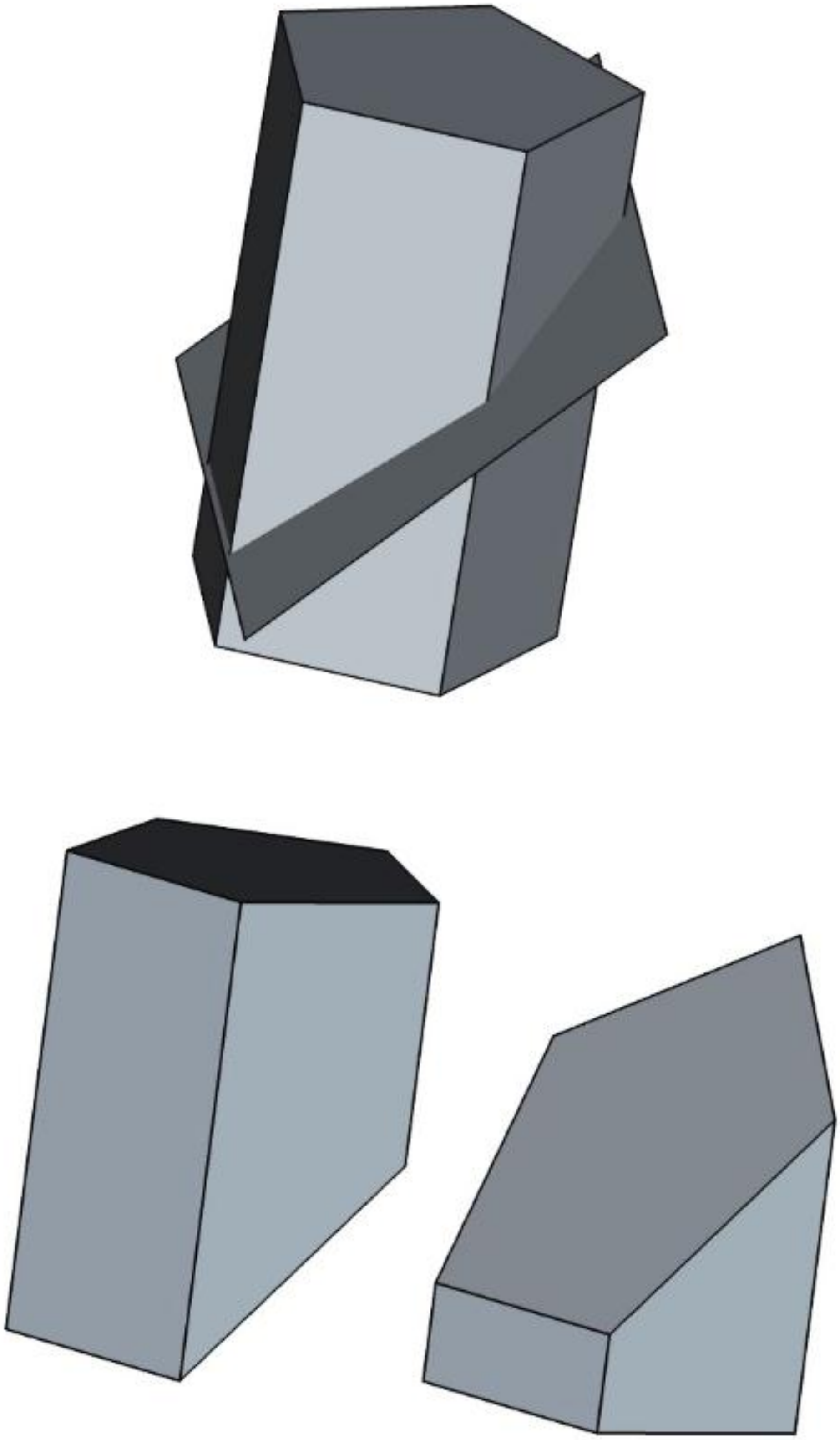
After cutting, the upper portion is removed to obtain a truncated rectangular prism.

Procedure

1. Open FreeCAD.
2. Switch to Part Design workbench.
3. Click Create Body.
4. Click Create Sketch and select XY Plane.
5. Use Sketcher Geometries to draw a rectangle.
6. Set dimensions: Length = 50 mm, Width = 20 mm, Close the sketch.
7. Click Pad and enter height = 70 mm to create the rectangular prism.
8. Switch to Part workbench.
9. Click Create Primitives → Plane.
10. Position the plane such that: It is inclined at 45° to H.P and perpendicular to V.P, it passes through a point 10 mm from the left side of the right top corner.
11. Select both the solid and the plane.
12. Use Split → Split Apart to cut the solid.
13. Delete or remove the unwanted portion.
14. Keep the required truncated part.
15. Set the model to Isometric view.
16. Apply shading for better visualization.
17. Save the model.

Result

The 3D CAD model of a truncated rectangular prism was successfully created by applying a cutting plane inclined at 45° to H.P and perpendicular to V.P, using FreeCAD.



Expt. No.: 18

Date:

CAD Modelling of Truncated Pentagonal Prism

Aim

To create a pentagonal prism of base side 30 mm and height 90 mm, cut by a plane inclined at 45° to HP and perpendicular to VP, bisecting the axis of the solid using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model using Pad
- Create cutting planes and perform sectioning operations
- Understand parametric modelling concepts

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A truncated pentagonal prism is a solid obtained when a pentagonal prism is cut by an inclined plane. The cutting plane in this case is inclined at 45° to the horizontal plane (HP) and is perpendicular to the vertical plane (VP), passing through the midpoint of the prism axis. In CAD modelling, the base solid is first created using a parametric approach by sketching a pentagon and extruding it to the required height. Then, a reference plane is created at the required angle and position.

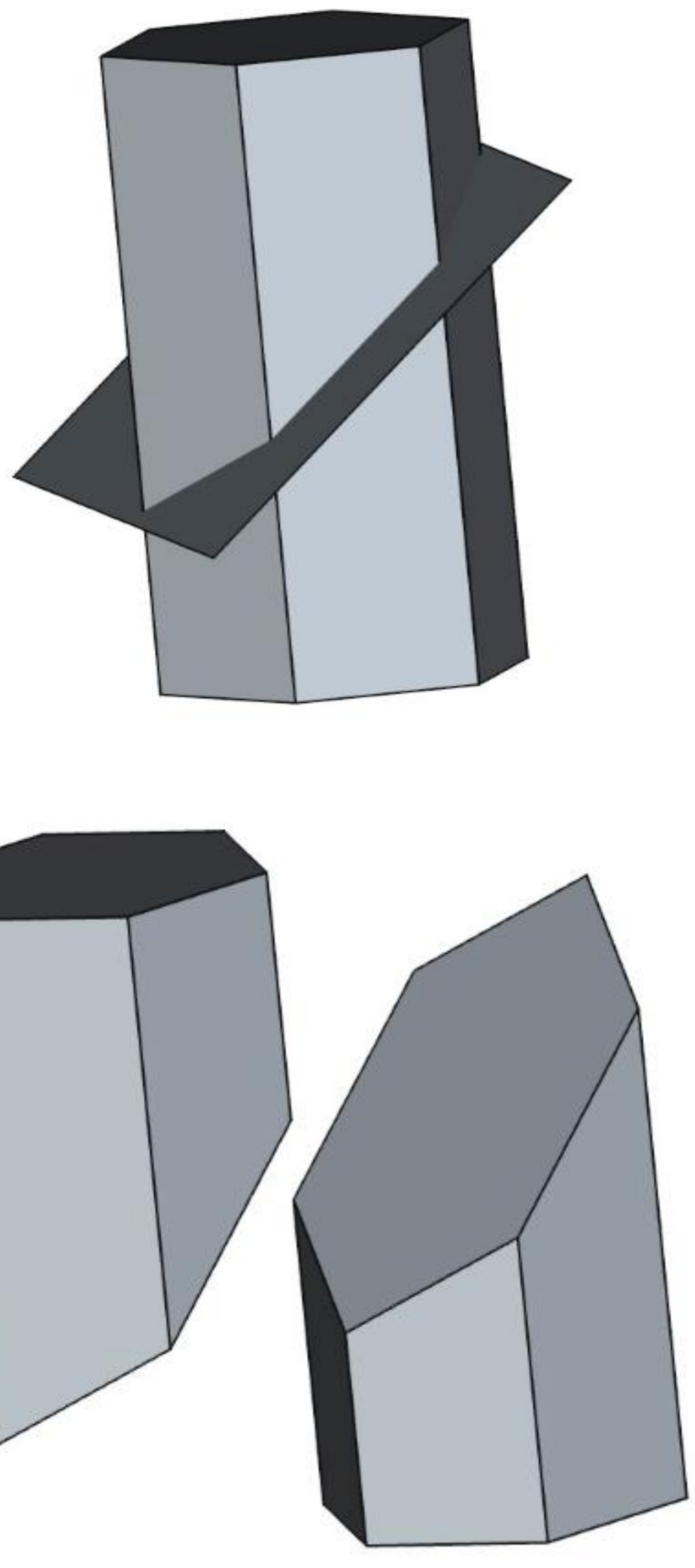
The solid is cut using this plane to obtain the truncated shape. This method helps in understanding sectioning of solids and allows easy modification of cutting angles and dimensions.

Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body. Click Create Sketch and select XY Plane.
3. Draw a pentagon using Sketcher tools. Apply constraints so that each side is 30 mm.
4. Close the sketch.
5. Select the sketch and click Pad.
6. Enter height as 90 mm and click OK to create the prism.
7. Create a reference plane using Plane option.
8. Use Plane Transform to incline the plane at 45° to HP and position it to bisect the axis of the prism.
9. Ensure the plane is perpendicular to VP.
10. Use Part workbench tools to perform splitting operation.
11. Apply Split (Split Apart) to cut the solid using the plane.
12. Convert the required portion into solid using Convert to Solid option.
13. Remove the unwanted portion.
14. Use View → Isometric view.
15. Apply shading for better visualization.
16. Verify dimensions.
17. Save the model.

Result

The CAD model of a pentagonal prism truncated by a plane inclined at 45° to HP and perpendicular to VP is successfully created using FreeCAD.



Expt. No.: 19

Date:

CAD Modelling of Truncated Hexagonal Prism

Aim

To create a hexagonal prism of base side 30 mm and height 120 mm, cut by a plane inclined at 30° to HP and perpendicular to VP at a height of 70 mm using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model using Pad
- Create cutting planes and perform sectioning operations
- Understand parametric modelling concepts

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A truncated hexagonal prism is formed when a hexagonal prism is cut by an inclined plane. In this case, the cutting plane is inclined at 30° to the horizontal plane (HP) and is perpendicular to the vertical plane (VP), intersecting the solid at a height of 70 mm from the base. In CAD modelling, the base solid is first created using a parametric approach by sketching a hexagon and extruding it to the required height. Then, a reference plane is created and oriented at the required angle and position.

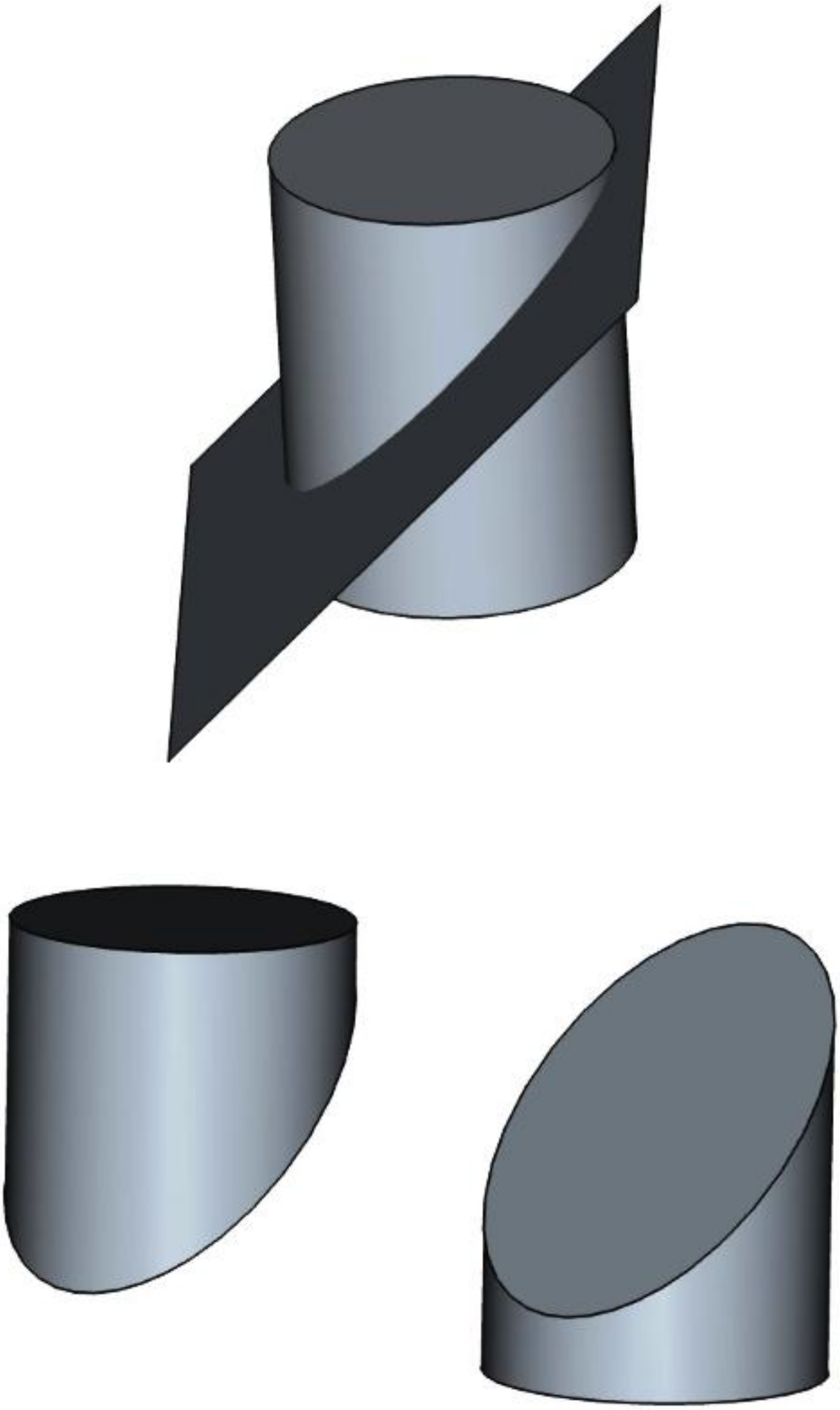
The solid is then cut using this plane to obtain the truncated shape. This method helps in understanding sectional views of solids and allows easy modification of dimensions and cutting angles.

Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body. Click Create Sketch and select XY Plane.
3. Draw a hexagon using Sketcher tools. Apply constraints so that each side is 30 mm.
4. Close the sketch.
5. Select the sketch and click Pad.
6. Enter height as 120 mm and click OK to create the prism.
7. Create a reference plane using Plane option.
8. Use Plane Transform to incline the plane at 30° to HP.
9. Position the plane at a height of 70 mm from the base and ensure it is perpendicular to VP.
10. Switch to Part tools for splitting operation.
11. Apply Split (Split Apart) to cut the solid using the plane.
12. Convert the required portion into solid using Convert to Solid option.
13. Remove the unwanted portion.
14. Use View → Isometric view.
15. Apply shading for better visualization.
16. Verify dimensions.
17. Save the model.

Result

The CAD model of a hexagonal prism truncated by a plane inclined at 30° to HP and perpendicular to VP at a height of 70 mm is successfully created using FreeCAD.



Expt. No.: 20

Date:

CAD Modelling of Truncated Cylinder

Aim

To create a cylinder of diameter 40 mm and height 50 mm, cut by a plane inclined at 30° to HP and perpendicular to VP, bisecting the axis of the solid using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model using Pad
- Create cutting planes and perform sectioning operations
- Understand parametric modelling concepts

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A truncated cylinder is obtained when a cylinder is cut by an inclined plane. In this case, the cutting plane is inclined at 30° to the horizontal plane (HP) and is perpendicular to the vertical plane (VP), passing through the midpoint of the cylinder axis. In CAD modelling, the base solid is first created by sketching a circle and extruding it to the required height using the Pad operation. Then, a reference plane is created and oriented at the required angle and position.

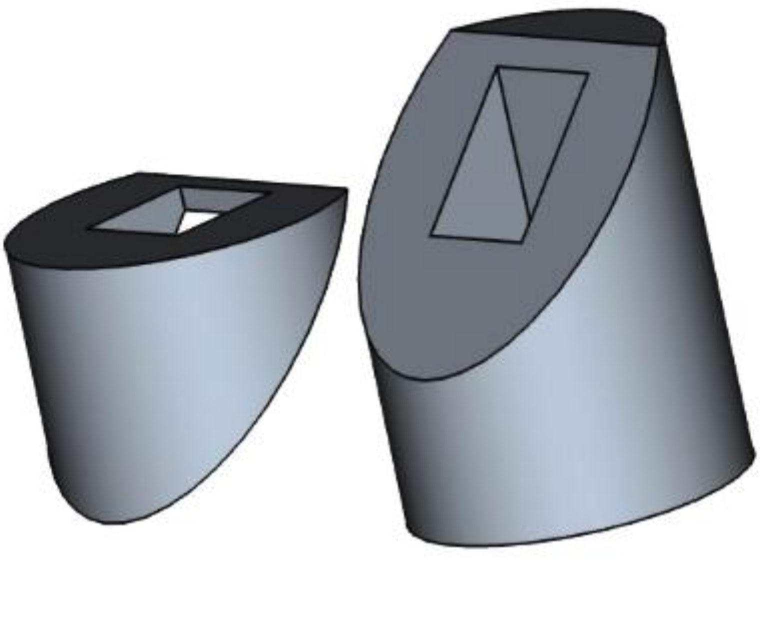
The solid is cut using this plane to obtain the truncated shape. This approach helps in understanding sectioning of solids and allows easy modification of dimensions and cutting angle.

Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body. Click Create Sketch and select XY Plane.
3. Draw a circle using Sketcher tools. Apply constraint so that diameter is 40 mm.
4. Close the sketch.
5. Select the sketch and click Pad.
6. Enter height as 50 mm and click OK to create the cylinder.
7. Create a reference plane using Plane option.
8. Use Plane Transform to incline the plane at 30° to HP.
9. Position the plane such that it bisects the axis of the cylinder and is perpendicular to VP.
10. Switch to Part tools for splitting operation.
11. Apply Split (Split Apart) to cut the solid using the plane.
12. Convert the required portion into solid using Convert to Solid option.
13. Remove the unwanted portion.
14. Use View → Isometric view.
15. Apply shading for better visualization.
16. Verify dimensions.
17. Save the model.

Result

The CAD model of a cylinder truncated by a plane inclined at 30° to HP and perpendicular to VP, bisecting the axis is successfully created using FreeCAD.



Expt. No.: 21

Date:

CAD Modelling of Truncated Cylinder with Square through Hole

Aim

To create a cylinder of diameter 40 mm and height 50 mm, with a central square through hole of size 15 mm × 15 mm, and cut by a plane inclined at 45° to HP and perpendicular to VP, meeting the axis at a point 15 mm from the top using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model using Pad
- Create internal features like holes using sketch and cut operations
- Create cutting planes and perform sectioning operations
- Understand parametric modelling concepts

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A truncated cylinder with internal features is formed when a cylinder is modified by both a through hole and an inclined cutting plane. The base cylinder is created by sketching a circle and extruding it to the required height. A square through hole is introduced at the center of the cylinder axis by sketching and removing material.

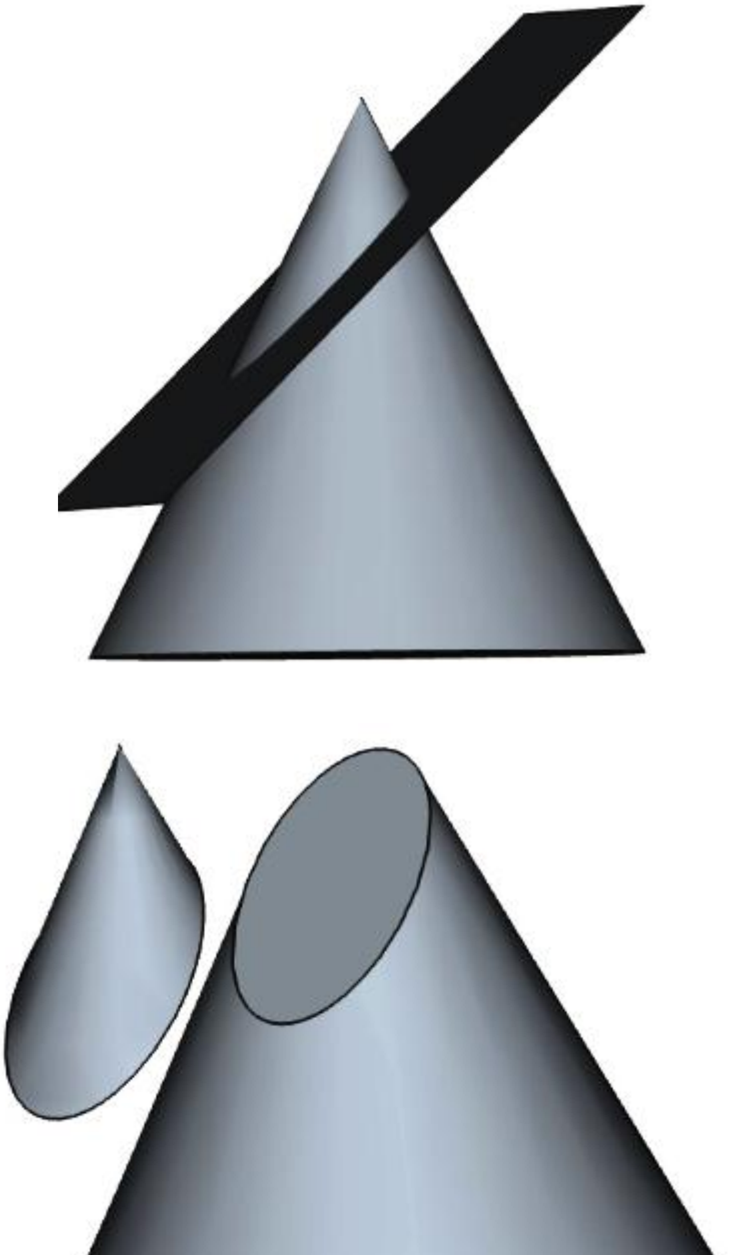
The solid is then cut by a plane inclined at 45° to the horizontal plane (HP) and perpendicular to the vertical plane (VP), intersecting the axis at a point 15 mm from the top. In CAD modelling, this is achieved using a parametric approach where the geometry and cutting plane can be easily controlled and modified, ensuring accuracy and flexibility in design.

Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body. Click Create Sketch and select XY Plane.
3. Draw a circle using Sketcher tools. Apply constraint so that diameter is 40 mm.
4. Close the sketch.
5. Select the sketch and click Pad.
6. Enter height as 50 mm and click OK to create the cylinder.
7. Select the top face of the cylinder and click Create Sketch.
8. Draw a square at the center using Sketcher tools. Apply constraints so that size is 15 mm × 15 mm.
9. Close the sketch.
10. Select the sketch and use Pocket (cut) operation.
11. Set depth as 50 mm to create a through square hole.
12. Create a reference plane using Plane option.
13. Use Plane Transform to incline the plane at 45° to HP.
14. Position the plane such that it intersects the axis at a point 15 mm from the top and is perpendicular to VP.
15. Switch to Part tools for splitting operation.
16. Apply Split (Split Apart) to cut the solid using the plane.
17. Convert the required portion into solid using Convert to Solid option.
18. Remove the unwanted portion.
19. Use View → Isometric view.
20. Apply shading for better visualization.
21. Verify dimensions.
22. Save the model.

Result

The CAD model of a cylinder with a central square through hole, truncated by a plane inclined at 45° to HP and perpendicular to VP, meeting the axis at 15 mm from the top is successfully created using FreeCAD.



Expt. No.: 22

Date:

CAD Modelling of Truncated Cone

Aim

To create a cone of base diameter 60 mm and height 150 mm, cut by a plane inclined at 40° to HP and perpendicular to VP, meeting the axis at 30 mm from the apex using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model using Pad or primitive features
- Create cutting planes and perform sectioning operations
- Understand parametric modelling concepts

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A truncated cone is formed when a cone is cut by an inclined plane. In this case, the cutting plane is inclined at 40° to the horizontal plane (HP) and is perpendicular to the vertical plane (VP), intersecting the axis at a point 30 mm from the apex. In CAD modelling, the cone is first created either by sketching and revolving or by using primitive features. A reference plane is then created and oriented at the required angle and position. The solid is cut using this plane to obtain the truncated shape. This method helps in understanding sectional views of solids and allows easy modification of dimensions and cutting angles.

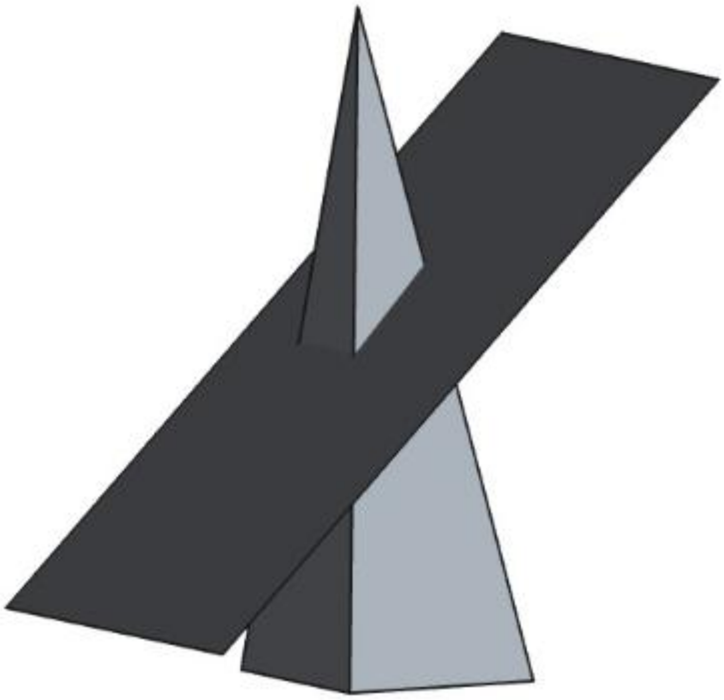
Procedure

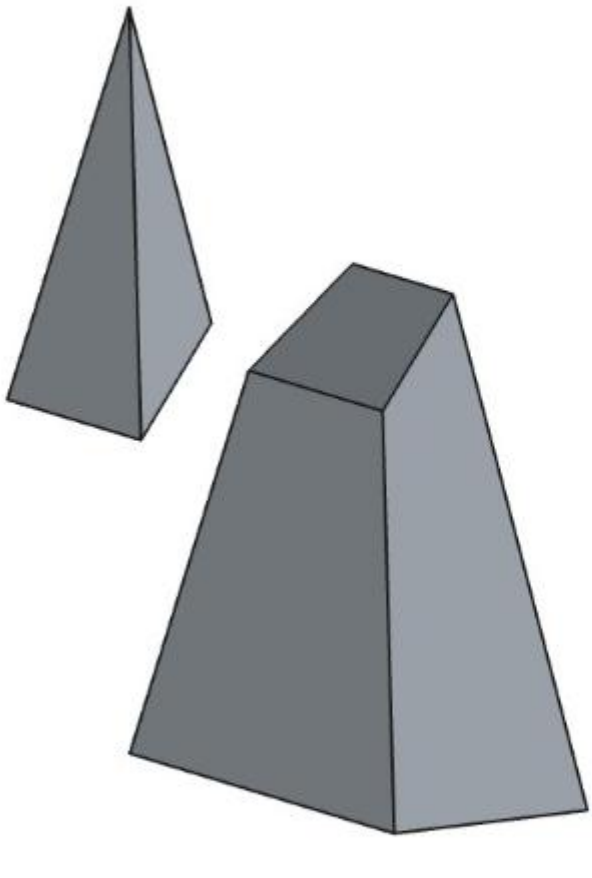
1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body.
3. Create the cone using Create Primitives option and set base diameter as 60 mm and height as 150 mm.

4. Position the cone properly on the XY Plane.
5. Create a reference plane using Plane option.
6. Use Plane Transform to incline the plane at 40° to HP.
7. Position the plane such that it intersects the axis at a point 30 mm from the apex and is perpendicular to VP.
8. Switch to Part tools for splitting operation.
9. Apply Split (Split Apart) to cut the solid using the plane.
10. Convert the required portion into solid using Convert to Solid option.
11. Remove the unwanted portion.
12. Use View → Isometric view.
13. Apply shading for better visualization.
14. Verify dimensions.
15. Save the model.

Result

The CAD model of a cone truncated by a plane inclined at 40° to HP and perpendicular to VP, meeting the axis at 30 mm from the apex is successfully created using FreeCAD.





Expt. No.: 23

Date:

CAD Modelling of Truncated Rectangular Pyramid

Aim

To create a rectangular pyramid of base 20 mm × 40 mm and height 70 mm, cut by a plane inclined at 45° to the axis and intersecting the axis at a point 40 mm from the base using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model
- Create cutting planes and perform sectioning operations
- Understand truncation of pyramidal solids

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A truncated rectangular pyramid is formed when a rectangular pyramid is cut by an inclined plane and a portion of the solid is removed.

In this case:

- The base of the pyramid is 20 mm × 40 mm
- The height of the pyramid is 70 mm
- The cutting plane is inclined at 45° to the axis
- The plane intersects the axis at a point 40 mm from the base

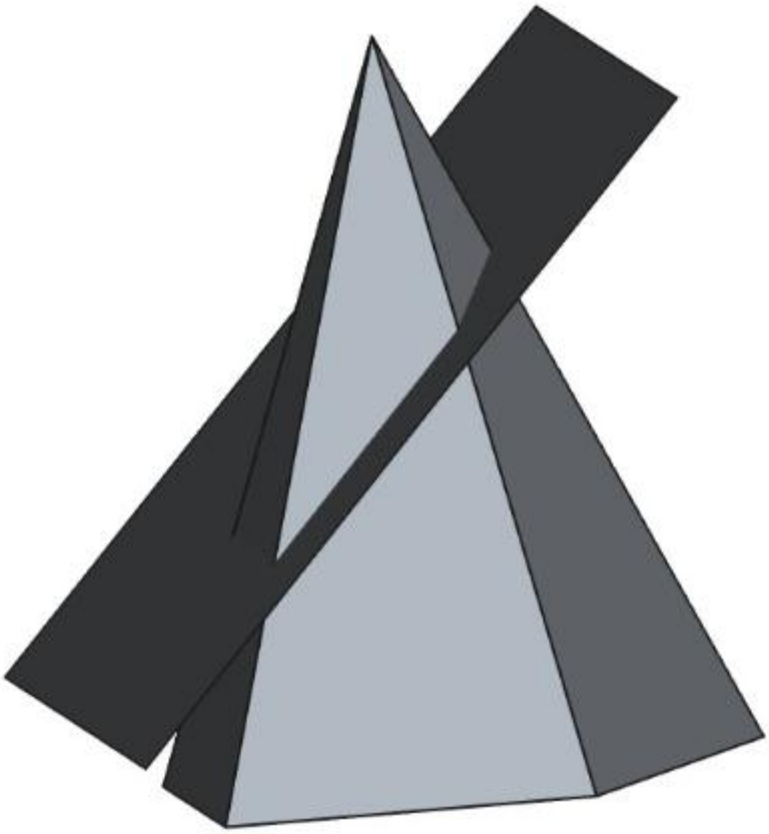
In CAD modelling, the pyramid is first created using sketch-based or primitive methods. A reference plane is then created and oriented at the required angle. The solid is cut using this plane to obtain the truncated shape. This helps in understanding sectioning of solids and 3D visualization.

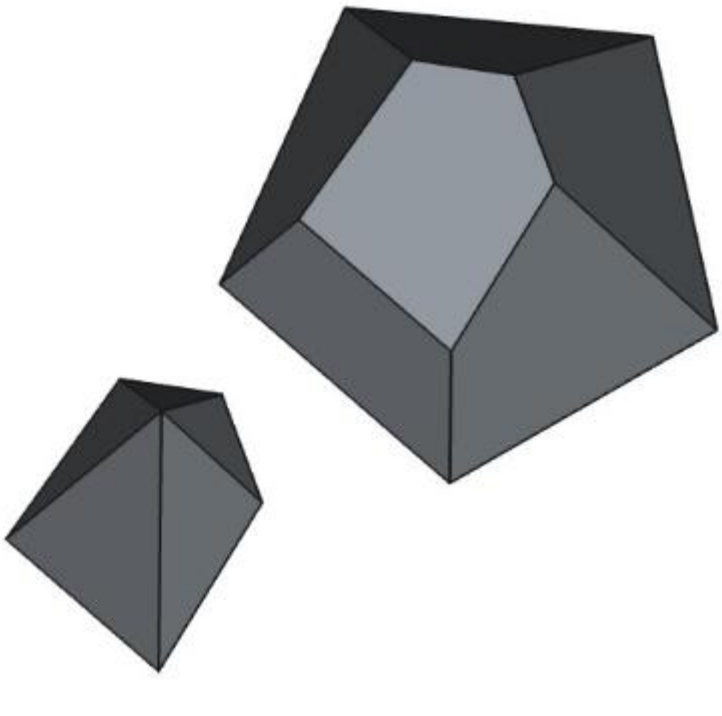
Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body.
3. Use Create Sketch on XY Plane.
4. Draw a rectangle of size 20 mm × 40 mm.
5. Close the sketch.
6. Create the pyramid using suitable feature (loft/draft/primitive).
7. Set height = 70 mm.
8. Position the pyramid properly on the XY Plane.
9. Create a reference plane using Plane option (Part workbench).
10. Use Plane Transform to incline the plane at 45° to the axis.
11. Position the plane such that it intersects the axis at a point 40 mm from the base.
12. Switch to Part tools for splitting operation.
13. Apply Split (Split Apart) to cut the solid using the plane.
14. Convert the required portion into solid using Convert to Solid option.
15. Remove the unwanted portion.
16. Use View → Isometric view.
17. Apply shading for better visualization.
18. Verify dimensions.
19. Save the model.

Result

The CAD model of a rectangular pyramid truncated by a plane inclined at 45° to the axis and intersecting it at 40 mm from the base is successfully created using FreeCAD.





Expt. No.: 24

Date:

CAD Modelling of Truncated Pentagonal Pyramid

Aim

To create a pentagonal pyramid with base side 30 mm and height 70 mm, cut by a plane inclined at 45° to the Horizontal Plane (HP) and meeting the axis at a point 30 mm from the top, using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model
- Create cutting planes and perform sectioning operations
- Understand truncation of pyramidal solids

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A truncated pentagonal pyramid is formed when a pentagonal pyramid is cut by an inclined plane and a portion of the solid is removed.

In this case:

- The base is a regular pentagon of side 30 mm
- The height of the pyramid is 70 mm
- The cutting plane is inclined at 45° to HP
- The plane meets the axis at a point 30 mm from the top (apex)

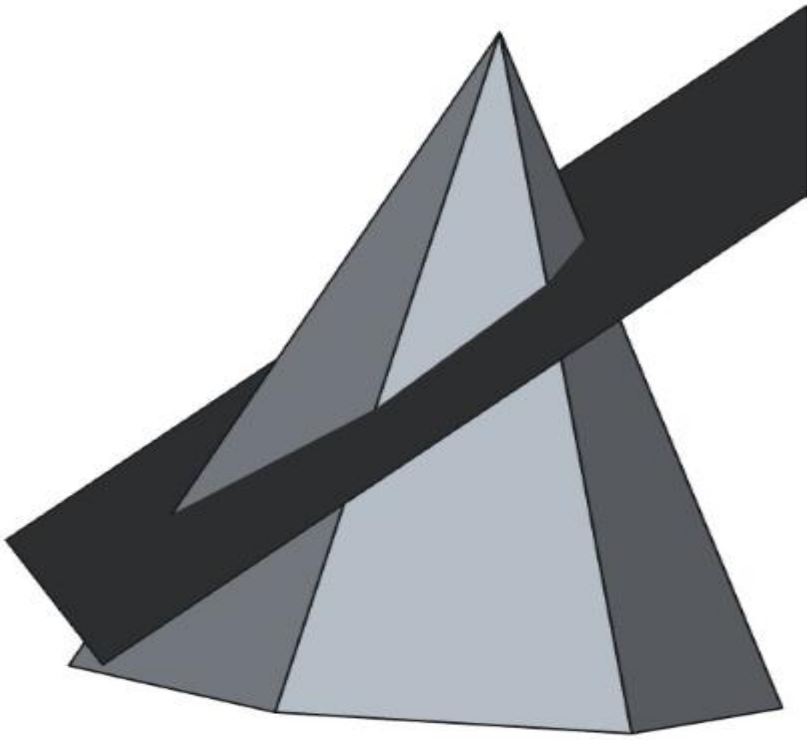
In CAD modelling, the pyramid is first created using sketch-based or primitive methods. Then a reference plane is created and inclined at the required angle. The solid is cut using this plane to obtain the truncated shape. This helps students understand sectioning of solids and 3D visualization.

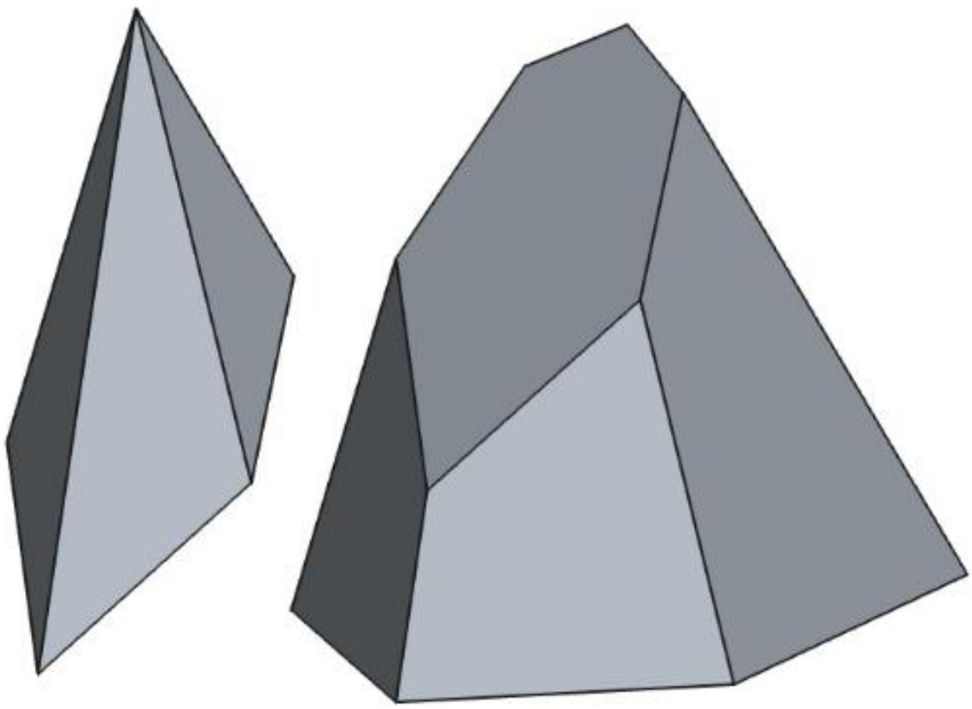
Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body.
3. Click Create Sketch on XY Plane
4. Draw a regular pentagon using Sketcher Geometries
5. Set side length = 30 mm
6. Close the sketch
7. Create the pyramid using suitable feature (loft/draft/primitive)
8. Set height = 70 mm
9. Position the pyramid properly on the XY Plane.
10. Create a reference plane using Plane option (Part workbench).
11. Use Plane Transform to incline the plane at 45° to HP.
12. Position the plane such that it meets the axis at a point 30 mm from the top (apex).
13. Switch to Part tools for splitting operation.
14. Apply Split (Split Apart) to cut the solid using the plane.
15. Convert the required portion into solid using Convert to Solid option.
16. Remove the unwanted portion.
17. Use View → Isometric view.
18. Apply shading for better visualization.
19. Verify dimensions.
20. Save the model.

Result

The CAD model of a pentagonal pyramid truncated by a plane inclined at 45° to HP and meeting the axis at 30 mm from the top is successfully created using FreeCAD.





Expt. No.: 25

Date:

CAD Modelling of Truncated Hexagonal Pyramid

Aim

To create a hexagonal pyramid of base side 30 mm and height 70 mm, cut by a plane perpendicular to the Vertical Plane (VP) and inclined at 30° to the Horizontal Plane (HP), meeting the axis at a point 40 mm from the base, using FreeCAD in the Part Design workbench.

Outcomes

After completing this experiment, the student will be able to:

- Create basic 2D sketches using CAD tools
- Apply constraints to define geometry
- Convert 2D sketch into 3D model
- Create cutting planes and perform sectioning operations
- Understand truncation of pyramidal solids

Software/Hardware Required

- FreeCAD
- Computer system

Theory

A truncated hexagonal pyramid is formed when a hexagonal pyramid is cut by an inclined plane and a portion of the solid is removed.

In this case:

- The base is a regular hexagon of side 30 mm
- The height of the pyramid is 70 mm
- The cutting plane is perpendicular to VP and inclined at 30° to HP
- The plane meets the axis at a point 40 mm from the base

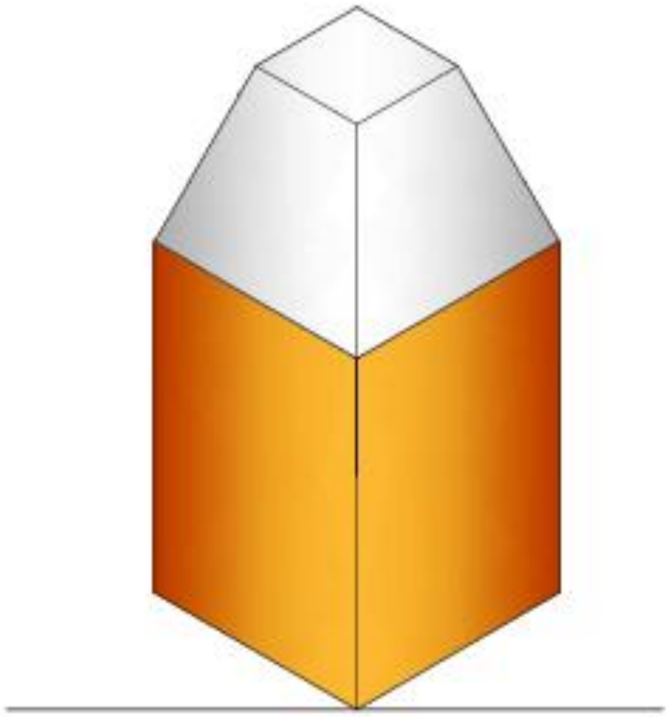
In CAD modelling, the pyramid is first created using sketch-based or primitive methods. Then a reference plane is created and oriented at the required angle and position. The solid is cut using this plane to obtain the truncated shape. This helps in understanding sectioning of solids and 3D visualization.

Procedure

1. Open FreeCAD and select Part Design Workbench.
2. Click Create Body.
3. Click Create Sketch on XY Plane
4. Draw a regular hexagon using Sketcher Geometries
5. Set side length = 30 mm
6. Close the sketch
7. Create the pyramid using suitable feature (loft/draft/primitive)
8. Set height = 70 mm
9. Position the pyramid properly on the XY Plane.
10. Create a reference plane using Plane option (Part workbench).
11. Use Plane Transform to incline the plane at 30° to HP.
12. Ensure the plane is perpendicular to VP.
13. Position the plane such that it meets the axis at a point 40 mm from the base.
14. Switch to Part tools for splitting operation.
15. Apply Split (Split Apart) to cut the solid using the plane.
16. Convert the required portion into solid using Convert to Solid option.
17. Remove the unwanted portion.
18. Use View → Isometric view.
19. Apply shading for better visualization.
20. Verify dimensions.
21. Save the model.

Result

The CAD model of a hexagonal pyramid truncated by a plane perpendicular to VP and inclined at 30° to HP, meeting the axis at 40 mm from the base is successfully created using FreeCAD.



Expt. No.: 26

Date:

CAD Modelling of Combination of Solids

(Square Prism with Frustum of Square Pyramid)

Aim

To create the isometric view of a 3D solid consisting of a square prism and a frustum of a square pyramid using CAD software.

Outcomes

After completing this exercise, the student will be able to:

- Understand the concept of combination of solids.
- Develop 3D models directly from given problem statements.
- Apply CAD commands to create prismatic and tapered solids.
- Generate the isometric projection of a 3D object.
- Improve visualization and modelling skills.

Software Required

- AutoCAD / FreeCAD
- Computer system

Theory

A combination of solids is formed by joining two or more basic solids. In this exercise, the object consists of:

- A square prism as the base
- A frustum of a square pyramid placed on top

The frustum is obtained by cutting the top portion of a pyramid, resulting in a smaller square surface at the top.

Using CAD software, such solids can be created using commands like extrude, loft, union, and subtract, and then viewed in isometric form.

Given Problem

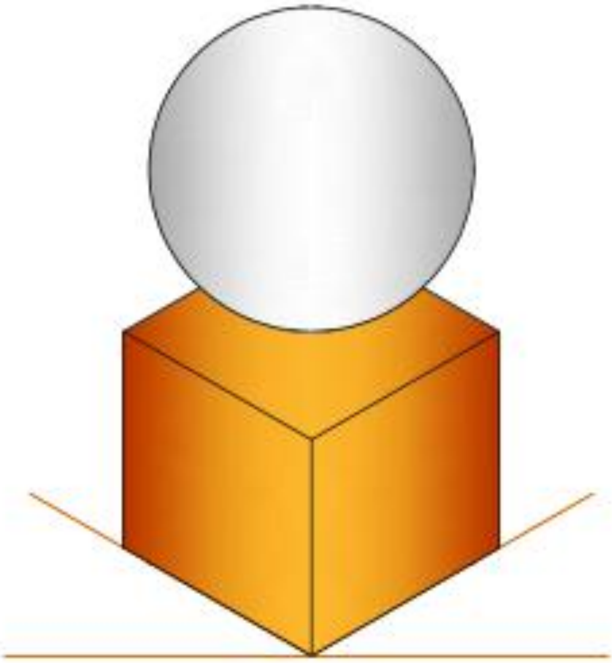
A solid is in the form of a square prism of side 40 mm and height 60 mm. Above this, it tapers into a frustum of a square pyramid whose top surface is a square of 20 mm side. The total height of the solid is 90 mm. Draw the isometric view of the solid.

Procedure

1. Open the CAD software and switch to 3D modelling workspace.
2. Set units in millimetres (mm).
3. Draw a square base of 40 mm × 40 mm.
4. Use EXTRUDE to create a prism of 60 mm height.
5. Draw a square of 40 mm × 40 mm on top of the prism.
6. Draw another square of 20 mm × 20 mm at a height of 90 mm total (i.e., 30 mm above prism).
7. Use LOFT (or similar command) to create the frustum shape.
8. Join the prism and frustum using UNION command.
9. Use 3D view tools to display the object in isometric view.
10. Apply shading or visual style if required.
11. Check all dimensions.
12. Save the model.

Result

The isometric view of the combination of solids (square prism with frustum of square pyramid) was successfully created using CAD software.



Expt. No.: 27

Date:

CAD Modelling of Combination of Solids

(Cube with Sphere)

Aim

To create the isometric view of a 3D object consisting of a cube surmounted by a sphere using CAD software.

Outcomes

After completing this exercise, the student will be able to:

- Understand the concept of combination of solids.
- Interpret given engineering drawings and problem statement.
- Create 3D models directly without drawing orthographic views.
- Use basic 3D modelling commands in AutoCAD or FreeCAD.
- Generate isometric views of combined solids.

Software Required

- AutoCAD / FreeCAD
- Computer system

Theory

A combination of solids is formed by placing or joining two or more simple solids. In this exercise, the object consists of:

- A cube resting on one of its faces
- A sphere placed centrally on the top face of the cube

The sphere touches the cube at the center of the top surface. This type of model helps students understand positioning and alignment of different solids in 3D space.

Given Data

A cube of size 40 mm is resting on the ground on one of its faces, surmounted centrally by a sphere of radius 30 mm. Draw the isometric view of the solid.

Procedure

1. Open the CAD software and switch to 3D modelling workspace.
2. Set units in millimetres (mm).
3. Draw a square of 40 mm × 40 mm.
4. Use EXTRUDE or BOX command to create a cube of 40 mm height.
5. Identify the center point of the top face of the cube.
6. Use SPHERE command.
7. Enter the radius as 30 mm.
8. Position the sphere so that it is centrally placed on the cube.
9. Ensure the bottom of the sphere just touches the top surface of the cube.
10. Adjust position if required.
11. Use 3D view tools to display the object in isometric view.
12. Apply shading or visual styles if required.
13. Verify dimensions and alignment.
14. Save the model.

Result

The isometric view of the combination of solids (cube with sphere) was successfully created using CAD software.

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