

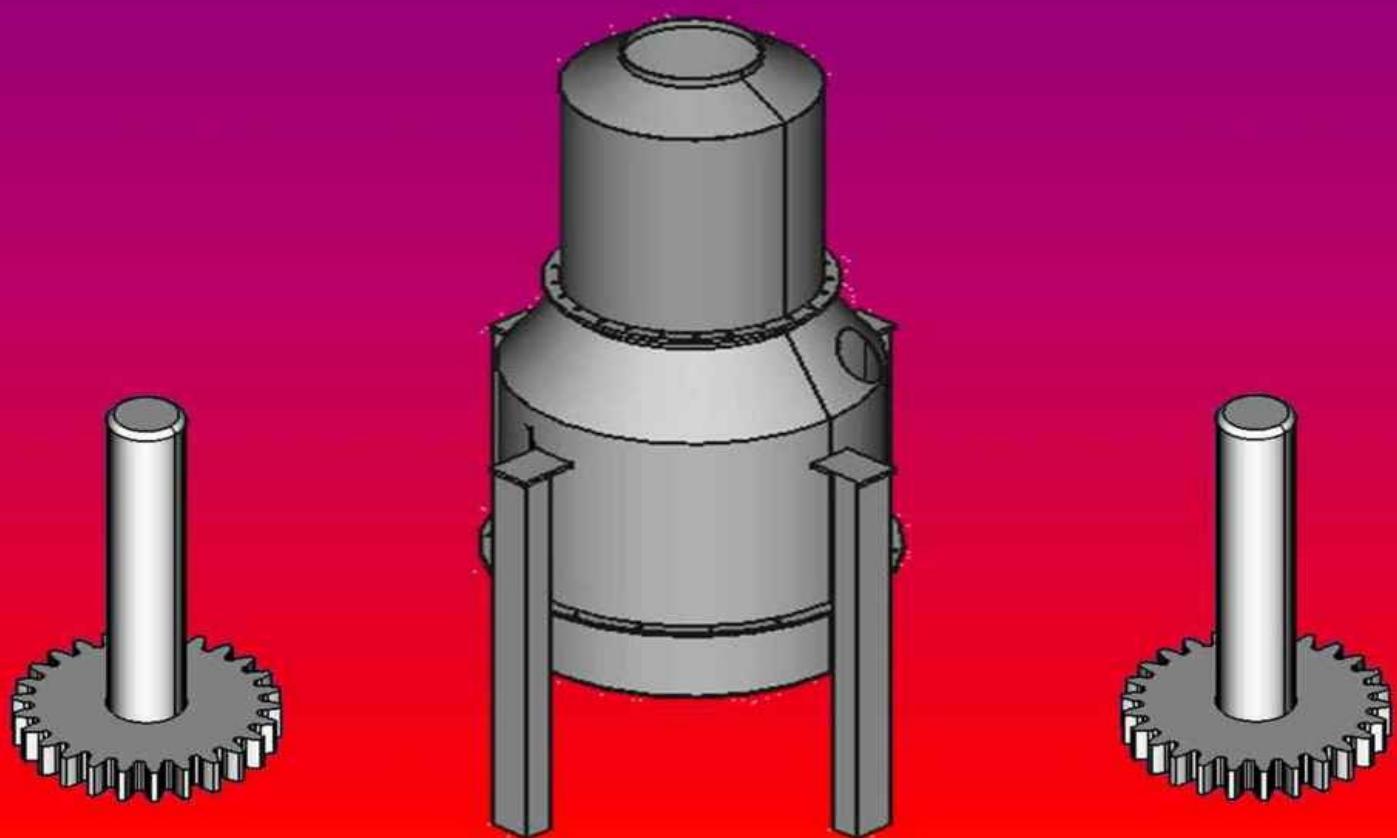
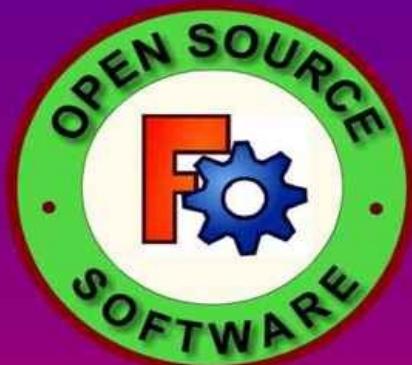


V. K. CHAUDHARY



FreeCAD

[Learn Easily & Quickly]



A 3D Solid Modeling Software Based On Latest Version
Of FreeCAD

FreeCAD

[Learn Easily & Quickly]

BY

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DEDICATION

Dedicated to Our Beloved parents
Smt. Shyama Devi
Late Sri Ram Pujan Chaudhary

Foreword

The book “**FreeCAD: [Learn Easily & Quickly]**” is the latest book in the FreeCAD world. This book has been written on the basis of latest version of FreeCAD. This book include **Video Tutorial Link** at chapter number **5, 9, 11 & 14** for easy and better understanding. The main advantages of this book is simple in language and clear screenshot.

A great thankful to two German engineers **Jürgen Riegel** and **Werner Mayer**, who was started the first FreeCAD project in 2002.

We shall be grateful to readers of this book to point out any error which can be rectified in future editions.

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1. INTRODUCTION



:- FreeCAD is open source and completely 3D modular for CAD, MCAD, CAX, CAE and PLM modular. It is 100% free software (GPL & LGPL License). FreeCAD is aimed directly at mechanical engineering and Product design but also fits in wide range of uses around engineering, Such as architecture or other engineering specialties. It is based on the OpenCascade. FreeCAD runs on Windows, Mac OSX and Linux platforms.

1.1 CHARACTERISTICS OF FreeCAD

- FreeCAD is a multi-platform .
- FreeCAD run as a command line application .
- FreeCAD can be imported as python module.
- Workbench concept.
- FreeCAD is a full Graphical User Interface (GUI) application.
- Graphical modification operation.
- Graphical creation of simple planer geometry.
- Boolean operation .
- Full macro editing and recording.
- Undo / Redo framework.
- Built in python console.
- Testing and repairing tools for meshes.
- Full customizable /scriptable Graphical User Interface.
- Modelling with straight or Revolution Extrusions, Sections and Fillets.
- Compound (ZIP based) document save format.
- Parametric primitive creation (box, cylinder, sphere). Modification

1.2 WHOM IS FreeCAD FOR?

- The home users/hobbyist.
- The experienced CAD users.
- The programmer.
- The educator.

1.3 FEATURES OF FreeCAD

1) SUPPORTED LANGUAGE

- Chinese
- Czech
- Dutch
- English
- Finnish
- French
- German
- Greek
- Hungarian
- Italian

2) WRITABLE FILES

- DXF
- IFC
- IGES
- NASTRAN
- OBJ
- PDF
- STEP
- STL
- VRML

3) READABLE FILES

- Bitmaps
- DXF
- IFC
- IGES
- NASTRAN
- OBJ
- STEP
- STL
- VRML

4) OPERATING SYSTEMS

- Windows 2000
- Windows 7
- Windows 8
- Windows Vista
- Windows XP
- Mac OS X
- Novell SUSE Linux

5) MAIN FUNCTIONALITY

- Parts & Assembly Modeling
- Simulation & Analysis
- Animations & Rendering
- Documentation

1.4 INSTALLATION OF FreeCAD ON WINDOWS

- Search the FreeCAD (<http://www.freecadweb.org>) on Google and download the latest version of FreeCAD. The latest version of FreeCAD is **FreeCAD 0.15.4671_x86_setup. Exe** and it is applicable for both windows 32-bit and 64-bit. For downloading the latest version of FreeCAD you can follow the link-
<http://sourceforge.net/projects/freecad/files/FreeCAD%20Windows/>

(See the below fig.)

The screenshot shows the 'FreeCAD' page on SourceForge. At the top, there's a navigation bar with links to Home, Browse, Graphics, 3D Modeling, FreeCAD, and Files. Below the navigation is the FreeCAD logo and the text 'a parametric 3D CAD modeler'. It also mentions 'Brought to you by: jnigiel, wmayer, yorikvanhavre'. A red arrow points to a button labeled 'External Link ▾' in the top menu. Below this, a red box highlights a download link: 'Looking for the latest version? Download FreeCAD-0.15.4671_x86_setup.exe (175.3 MB)'. The main content area shows a table of files:

Name	Modified	Size	Downloads / Week
Parent folder			
FreeCAD 0.16 development	2015-05-21	1,275	
FreeCAD 0.15	2015-04-08	13,702	

- After downloading the FreeCAD, click on the FreeCAD setup and run it. Follow the instructions and install the FreeCAD.
- After installation, the FreeCAD software is ready for use.
- For feature use, you can also download the latest version of Python (<https://www.python.org>). The latest version of python is **python —2.7.10.msi** and follow the download link -
<https://www.python.org/downloads/>
- After downloading the python, click on the python setup and run it. Follow the instructions and install the python.

END

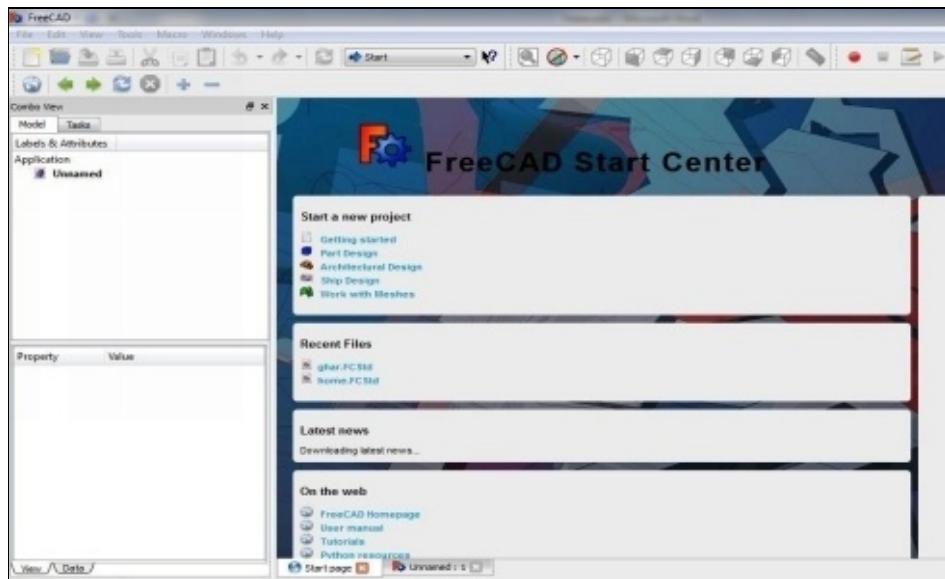
2. WELCOME TO FreeCAD



: - Really, this software is very useful and easy to use according to my experienced. In this chapter you will know about the look of FreeCAD, customization of FreeCAD, about Tools, navigation of 3D models and etc.

2.1 HOW TO OPEN FreeCAD

- After installation of FreeCAD, go to the Start menus in windows and find the FreeCAD.
- Click on it.
- Then explore the FreeCAD.
- When you start FreeCAD for the first time, it displayed with the Start Center.

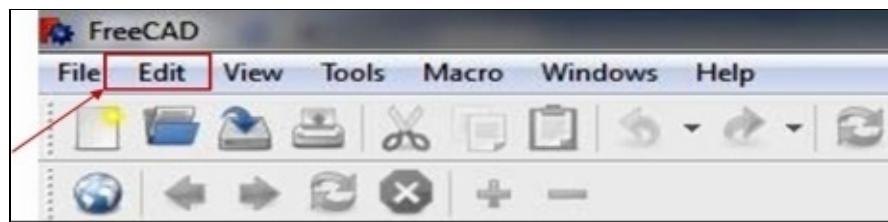


The Start Center allows you to quickly jump to one of the most common workbenches, open one of the recent files, or see the latest news from the FreeCAD world. You also find the FreeCAD tutorial, python resources, user manual etc, from this start Center. You can change the default workbench (Start)

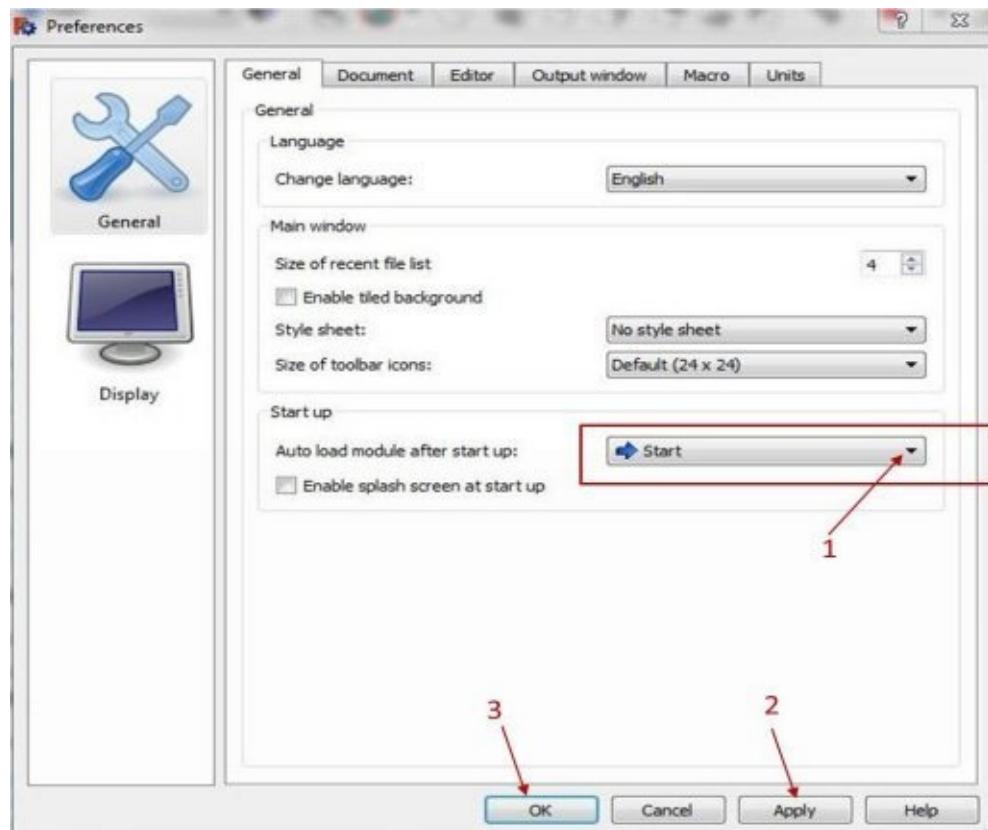
in the **preference**.

2.2 HOW TO SET USER PREFERENCES

- Open the FreeCAD.
- Go to the Edit menu and switch the Preferences.



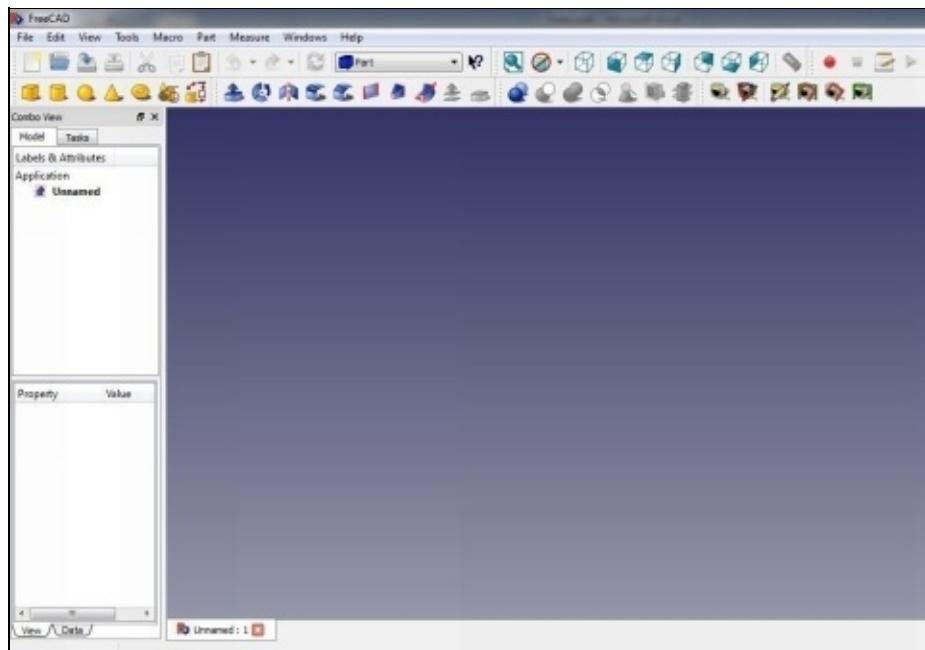
- A Preferences dialog box will open



- In this dialog box, go to the Start up and select the drop down key (see

above Fig.)

- Then select the listed workbenches which you want.
- I recommend to select the **Part Workbench**.
- Then click on Apply button and then Ok (see the above Fig.)
- You can also set the all others properties in the Preferences dialog box like Display.
- Then close the FreeCAD and again restart the FreeCAD. FreeCAD will appear like this,



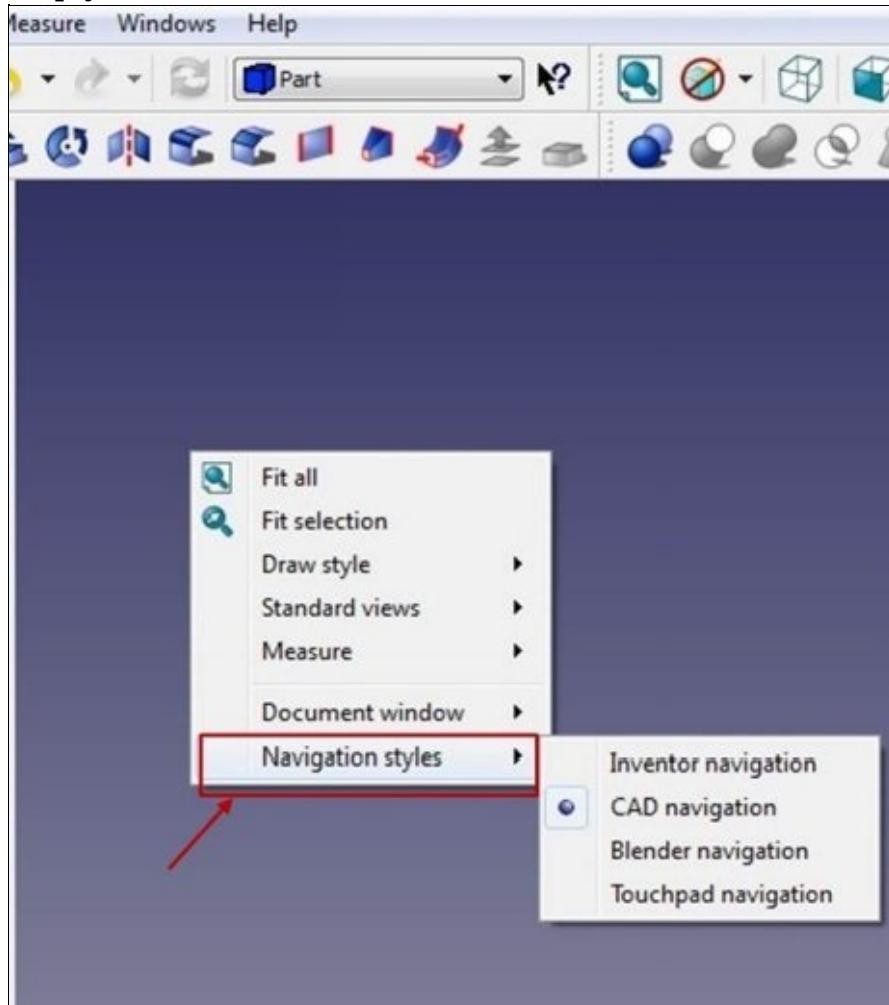
2.3 HOW TO NAVIGATE MODELS IN THE 3D SPACE

: - FreeCAD has four different **Navigation Styles**, available that change the way you use your mouse to interact with the objects in the 3D view and the view itself. These navigation styles are:-

1. Inventor navigation

2. Cad navigation
3. Blander navigation and
4. Touchpad navigation.

You can quickly change the current navigation mode by right-clicking on an empty area of the 3D view. See the Fig. below,

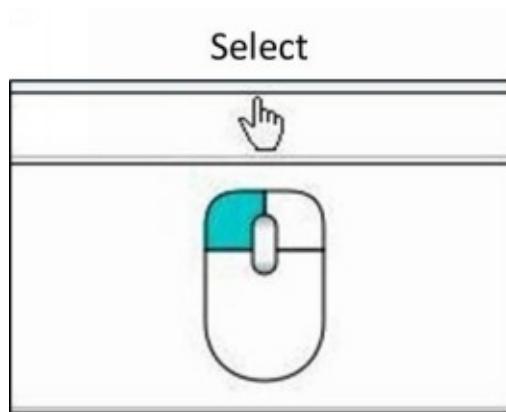


The default navigation style is referred to as "**CAD navigation**," and is very simple and practical and i also recommend to use CAD navigation, but FreeCAD also provides alternative navigation styles that you can choose according to your preferences.

PROCEDURE TO NAVIGATE THE MODELS IN 3D

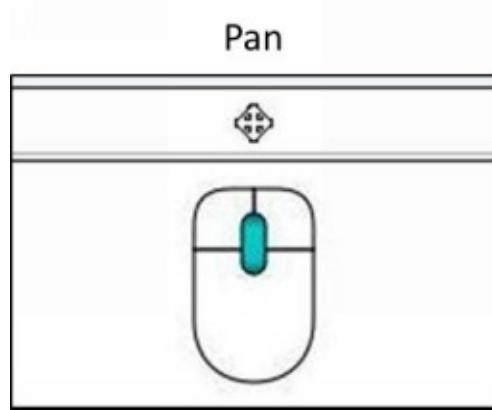
SPACE BY THE USE OF MOUSE WITH CAD NAVIGATION STYLE

a) For select the object or its surface:-



Press the left mouse button over an object you want to select. Holding down ctrl allows the selection of multiple objects.

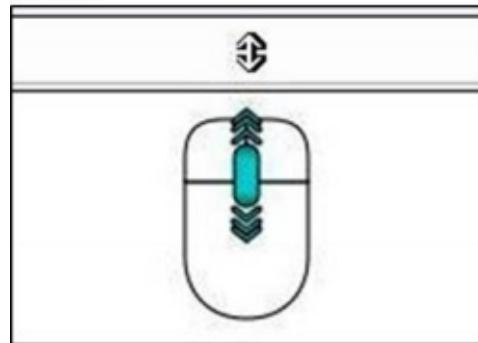
b) For Pan:-



Click and hold the middle mouse button and move the object around to pan.

c) For Zoom in and out:-

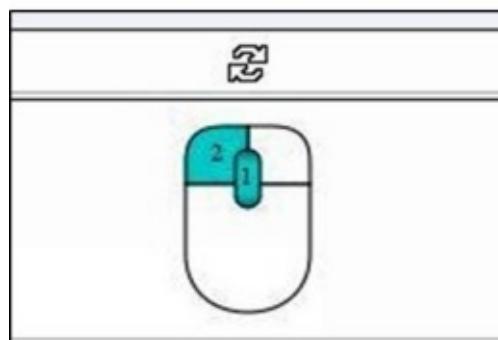
Zoom in and out



Rotate the mouse wheel up and down to zoom in and out.

d) For Rotate view:-

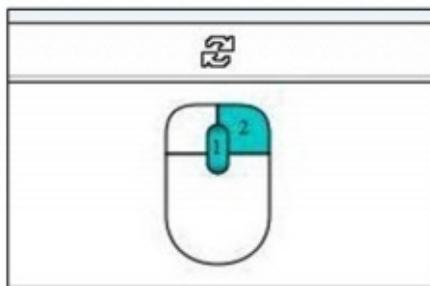
Rotate view



Click first with the middle mouse or wheel button and hold it, and then click the left mouse button and drag the mouse in the desired direction. The cursor location at the middle mouse button click determines the center of rotation. Rotation works like spinning a ball which rotates around its center. If the buttons are released before you stop the mouse motion, the object continues spinning if this is enabled. For again reset the position of object, click on the Isometric view.

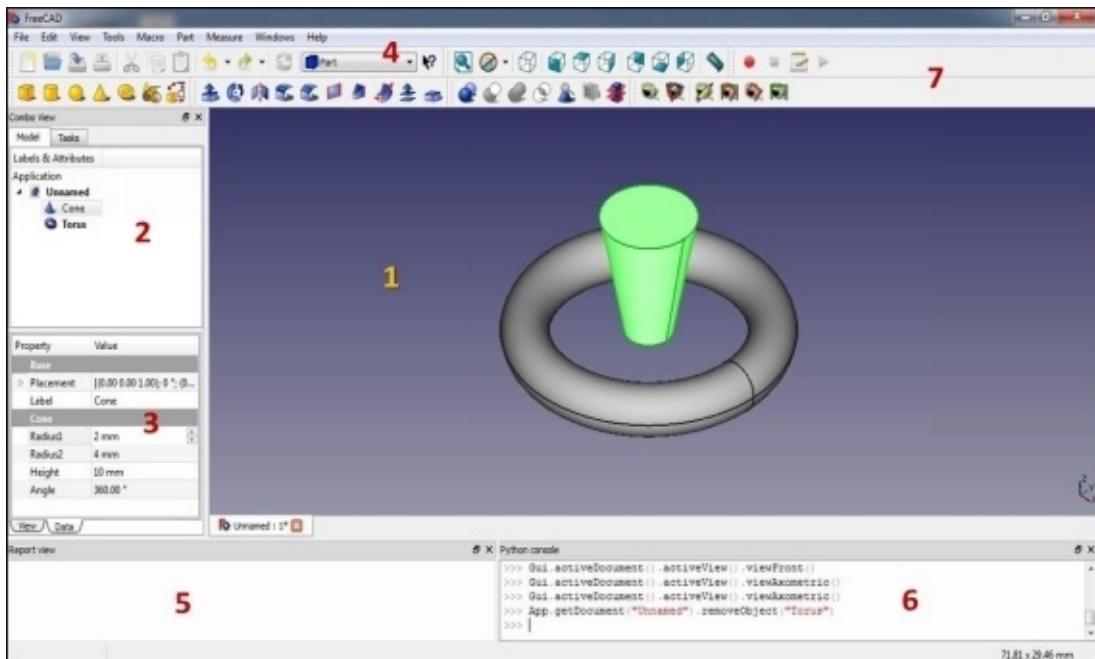
e) For Rotate view (alternate method):-

Rotate view (alternate method)



Click first with the middle mouse or wheel button and hold it, and then click the right mouse button and drag the mouse in the desired direction. This method works just like the previously described rotate view that uses middle mouse button + left mouse button, except that the middle mouse button may be released after the right mouse button is pressed. Users who use the mouse with their right hand may find this Rotate View method easier than the previous method.

2.4 EXPLORING FreeCAD



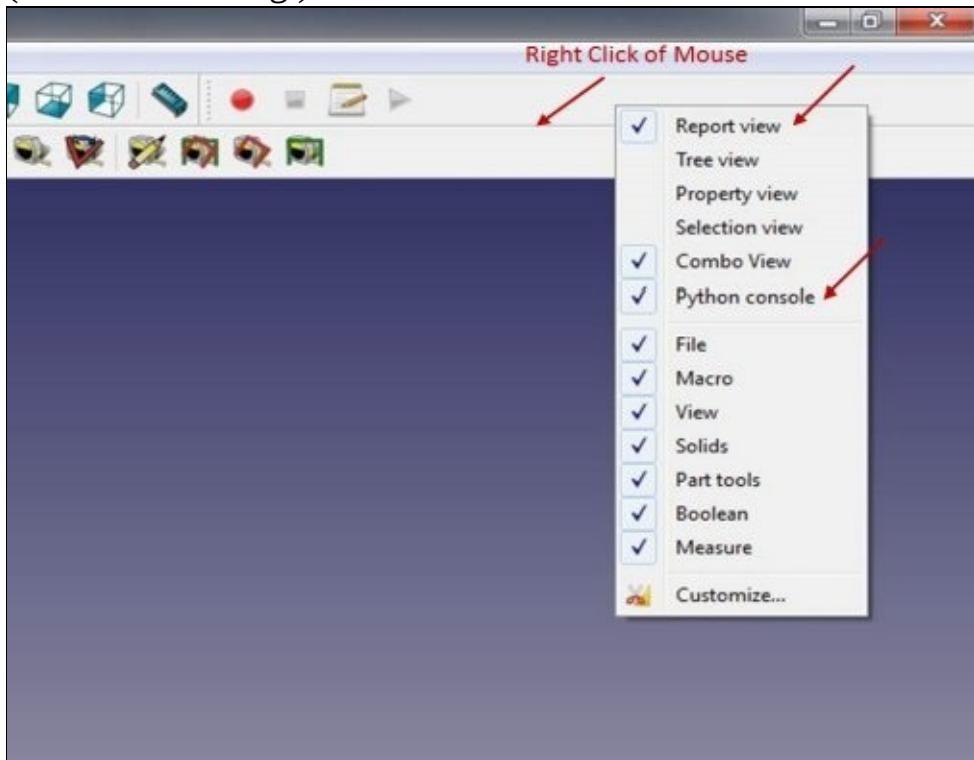
When you open or explore the FreeCAD, it will look like the above screenshot. The number (1, 2, 3, 4, 5, 6, and 7) which I mark in the screenshot is the part of FreeCAD and their explanations are:-

1. **The 3D view** → This view showing the contents or object(s) of your document.
2. **The Combo view** → which shows the hierarchy and construction history of all the objects in your document. You can also select the object(s) from this view for the purpose of modification and deleting.
3. **The Property view** → which allows you to view and modify properties of the selected object(s).
4. **Switch between Workbenches** → where you select the active workbench.
5. **The Report view** → which is where FreeCAD prints messages, warnings and errors.
6. **The python console** → where all the commands executed by FreeCAD are printed, and where you can enter python code.
7. **Toolbars Blank Space** → where you can arrange the tools and also select the different types of views and properties by the use of Right Click of Mouse.

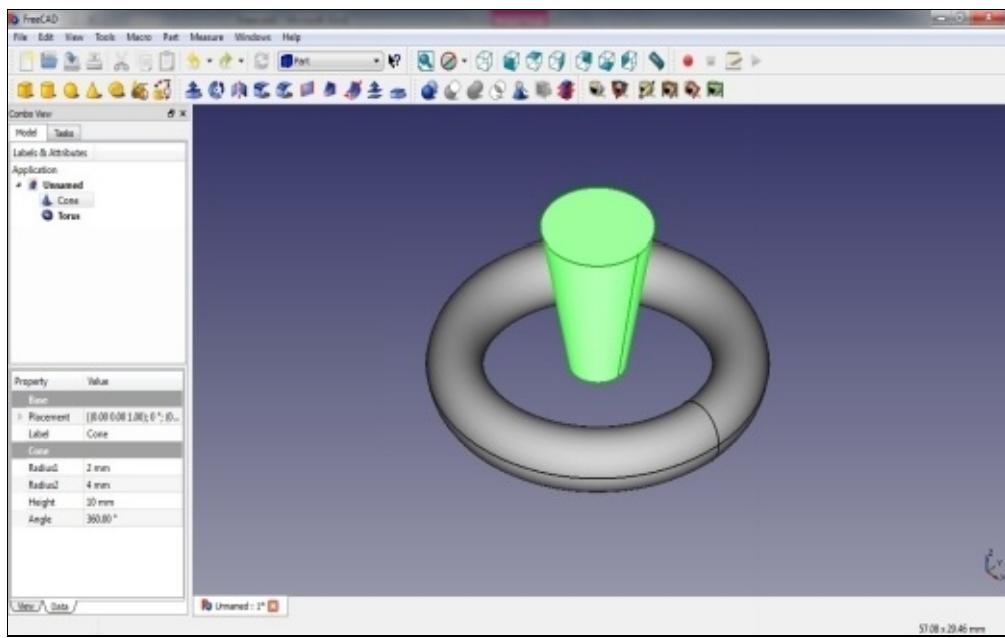
NOTE

: - If you want to remove the Report view and Python Console view from the Taskbar of the FreeCAD, follow the below instruction:-

- 1) Open or explore the FreeCAD, 2) Right click of the mouse on the Toolbars blank space, 3) Then uncheck the Report view and Python console.
- 4) You can also insert the all other views by check or uncheck on the views (see the below Fig.)



- 5) Now your FreeCAD looks like this,



6) You can also close the Report view and python console directly. See the below fig.

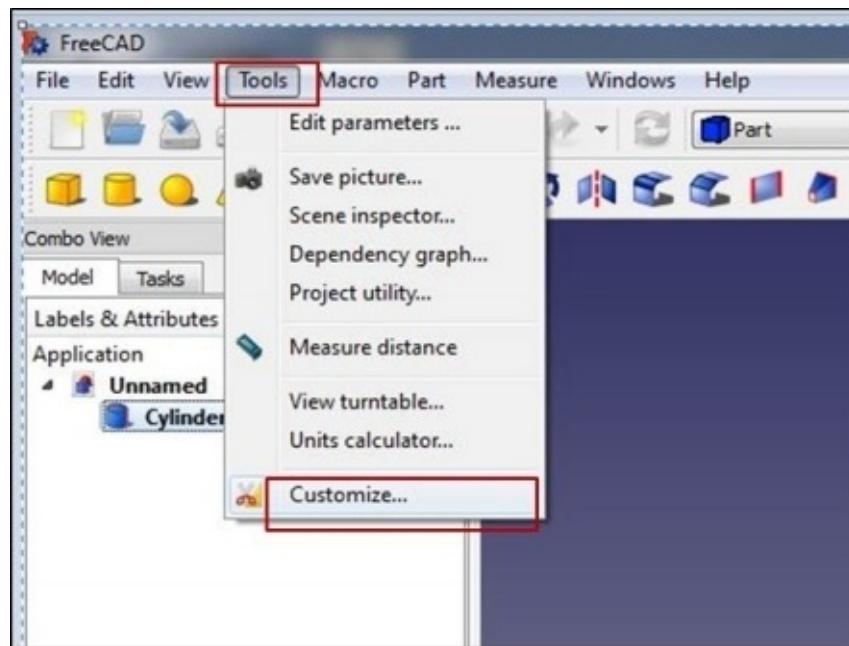


2.5 HOW TO CUSTOMIZE THE INTERFACE

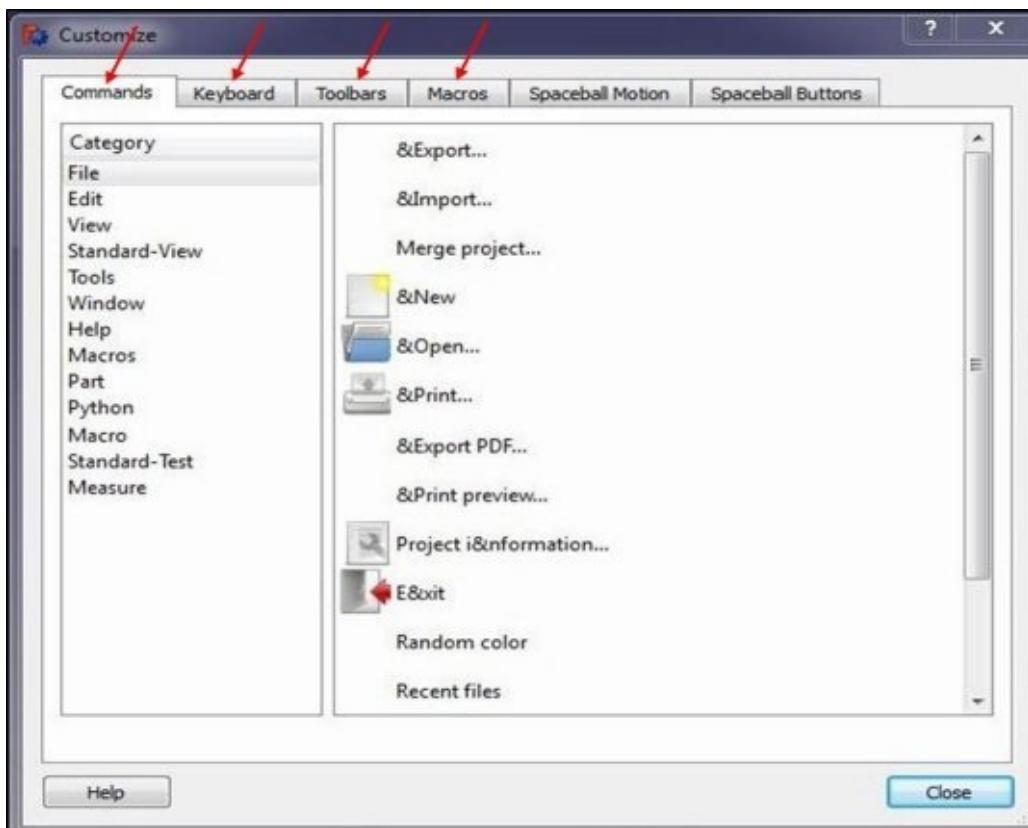
: - Since FreeCAD interface is based on the modern **Qt** toolkit, it has a state-of-the-art organization. Widgets, menus, toolbars and other tools can be modified, moved, shared between workbenches, keyboard shortcuts can be set, modified,

and macros can be recorded and played. The processes of the Customization of window are:-

a) Go the **Tools** and click on the **Customize** (see the below Fig).



b) A Customize dialog box will open (see the below Fig).



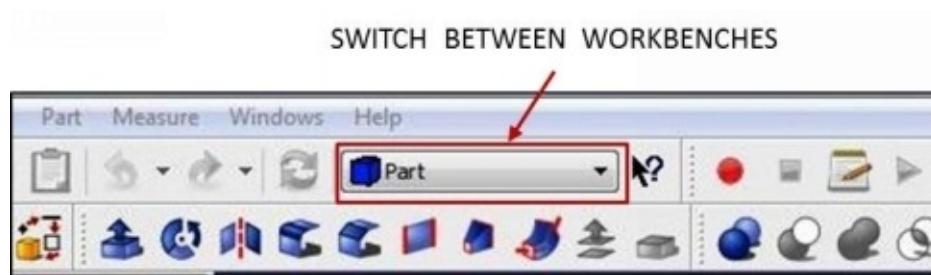
In this Customize dialog box, the six tabs are located at top position, the descriptions are:-

1. **Commands**: - The Commands tab lets you browse all available FreeCAD commands, organized by their category.
2. **Keyboard**: - In Keyboard, you can see the keyboard shortcuts associated with every FreeCAD command, and if you want, modify or assign new shortcut to any command. This is where to come if you use a particular workbench often, and would like to speed up its use by using the keyboard.
3. **Toolbars**: - The Toolbars and Toolbox bars tabs let you modify existing toolbars, or create your own custom toolbars.
4. **Macros**: - The Macros tab lets you manage your saved Macros.
5. **Spaceball Motion** and **Spaceball Buttons** are in development phase.

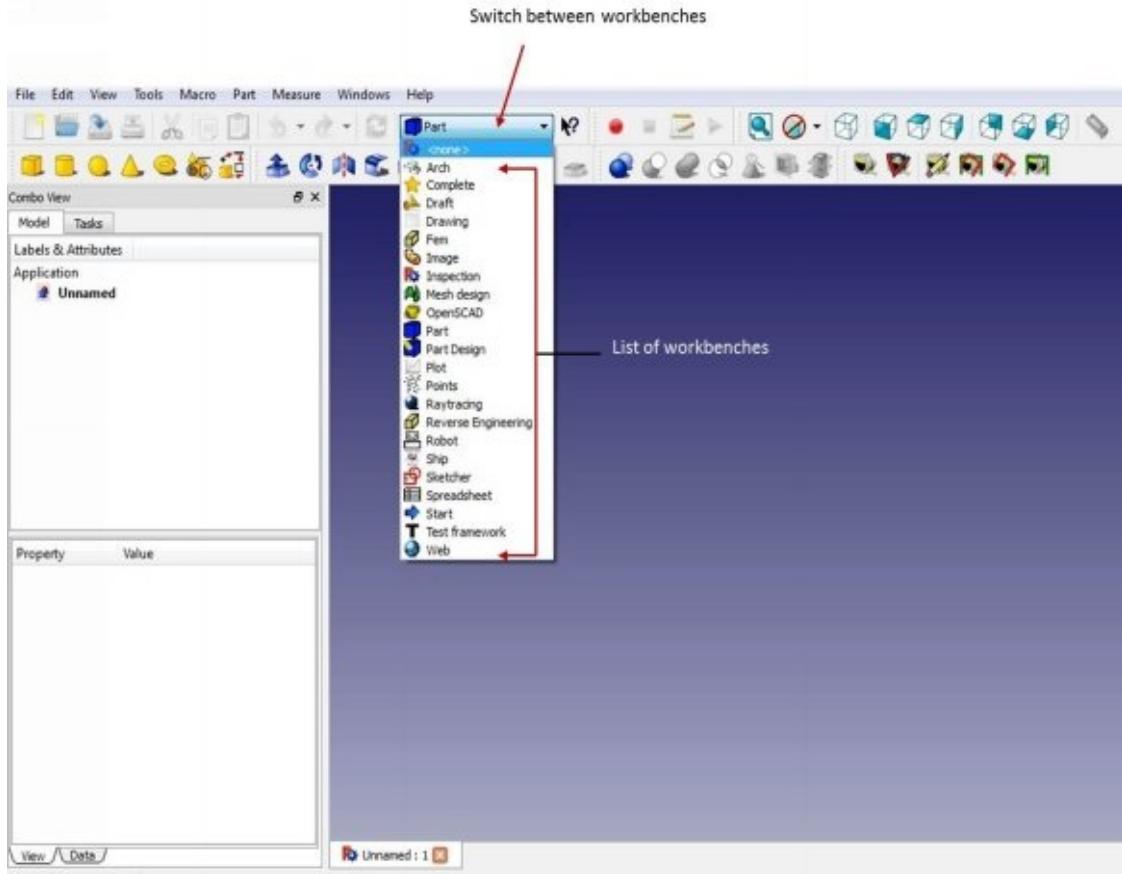
END

3. FreeCAD WORKBENCH

Workbench : - A **workbench** can be considered as a set of tools specially grouped for certain task. It is very helpful for crating and modifying the objects at time in one workplace. In FreeCAD, tools are grouped into workbenches according to the tasks.



3.1 LIST OF WORKBENCHES



In FreeCAD, the following important workbenches are available:

<u>S.N</u>	<u>SYMBOL</u>	<u>WORKBENCHES</u>	<u>DESCRIPTION</u>
1.		ARCH	For creating an Architectural design.
2.		COMPLETE	It indicates the all tools those are listed in other workbenches.
3.		DRAFT	For creating 2D object.
4.		DRAWING	For displaying 3D design into 2D on drawing sheet.
5.		FEM	For Pre-and Post-processing FEM

Note:-
In
FreeCAD,
total active

			studies
6.		IMAGE	For working with bitmap images.
7.		INSPECTION	This workbench is in development phase.
8.		MESH DESIGN	For working with triangulated meshes.
9.		OPENS CAD	For interoperability with OpenSCAD and repairing CSG model history.
10.		PART	For creating and working with standard parts.
11.		PART DESIGN	For modifying and creating a shape on parts.
12.		PLOT	The Plot module allows to edit and save output plots created from other modules and tools.
13.		POINTS	The Points module is made to give you specific tools for working with point clouds. It is still in development.
14.		RAYTRACING	For working with Raytracing (Rendering)
15.		REVERSE ENGINEERING	This workbench is still in development.
16.		ROBOT	For studying robot movement.
17.		SHIP	FreeCAD-ship work over ship entities.

18.		SKETCHER	For working with geometry-constrained sketches.
19.		SPREADSHEET	For creating and manipulating spread sheet data.
20.		START	The Start workbench allows you to quickly jump to one of the most common workbenches.
21.		TEST FRAMEWORK	The test framework (testing) is based on a set of Python scripts, which are located in the test module.
22.		WEB	This workbench allows you to connect directly with FreeCAD web documentations and help.

workbenches are 22, but in this, some workbenches are in development phase. In FreeCAD the **most important** and **useful** Workbenches are:

1. DRAFT WORKBENCH
2. SKETCHER WORKBENCH
3. PART WORKBENCH
4. PART DESIGN WORKBENCH
5. DRAWING WORKBENCH
6. ARC WORKBENCHES
7. ROBOT WORKBENCH
8. SHIP WORKBENCH
9. IMAGE WORKBENCH
10. RAYTRACING WORKBENCH

END

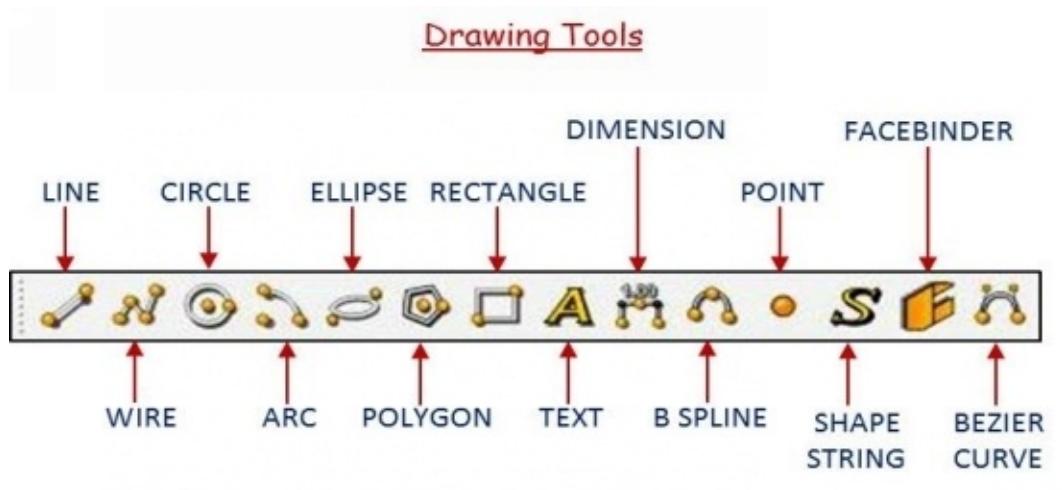
4. DRAFT WORKBENCH



- The Draft workbench allows to quickly drawing simple 2D objects in the current document, and offers several tools to modify them afterwards. Some of these tools also work on all other FreeCAD objects, not only those created with the Draft workbench. It also provides a complete snapping system, and several utilities to manage objects and settings.

6.1 DRAWING TOOLS

In draft workbench, these are tools for creating the objects with the help of drawing object.



The descriptions of **Drawing Tools** are listed below:-

1.	
LINE	Draws a line segment between 2 points.



2.

WIRE

Draws a line made of multiple line segments (polyline).



3.

CIRCLE

Draws a circle from center and radius.



4.

ARC

Draws an arc segment from center, radius, start angle and end angle.



5.

ELLIPSE

Draws an ellipse from two corner points.



6.

POLYGON

Draws a regular polygon from a center and a radius.



7.

RECTANGLE

Draws a rectangle from 2 opposite points.



8.

TEXT

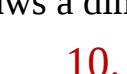
Draws a multi-line text annotation.



9.

DIMENSION

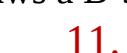
Draws a dimension annotation.



10.

B SPLINE

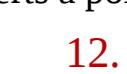
Draws a B-Spline from a series of points.



11.

POINT

Inserts a point object.



12.

SHAPE STRING

The Shape String tool inserts a compound shape representing a text string at a given point in the current document.



13.

FACEBINDER

Creates a new object from selected faces on existing objects.



14.

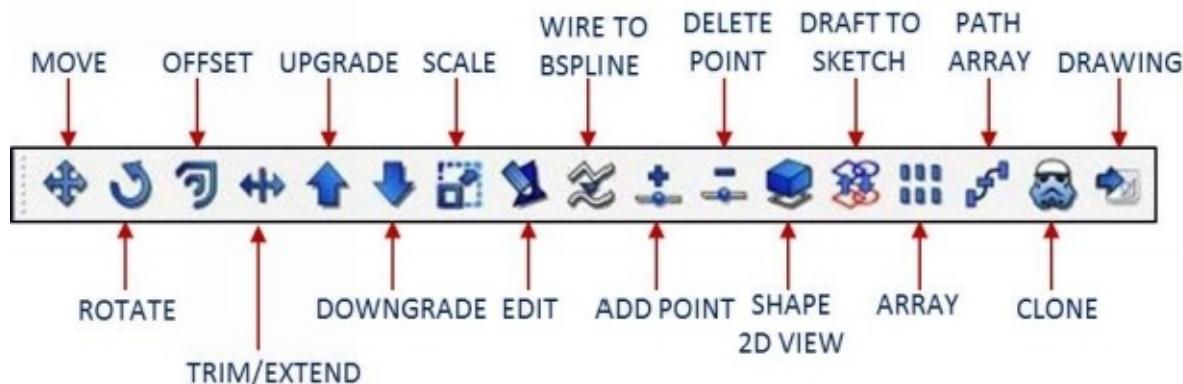
BEZIER CURVE

Draws a Bezier curve from a series of points.

6.2 MODIFYING TOOLS

These are tools for modifying existing objects. They work on selected objects, but if no object is selected, you will be invited to select one .

Modifying Tools



The descriptions of **Modifying Tools** are listed below:-



1.

MOVE

Moves object(s) from one location to another.



2.

ROTATE

Rotates object(s) from a start angle to an end angle.

3. **OFFSET**

Moves segments of an object about a certain distance.

4. **TRIM/EXTEND**

Trims or extends an object.

5. **UPGRADE**

Joins objects into a higher-level object.

6. **DOWNGRADE**

Explodes objects into lower-level objects.

7. **SCALE**

Scales selected object(s) around a base point.

8. **EDIT**

Edits a selected object.

9. **WIRE TO BSPLINE**

Converts a wire to a B Spline and vice-versa.

10. **ADD POINT**

Adds a point to a wire or B Spline.

11. **DELETE POINT**

Deletes a point from a wire or B Spline.

12. **SHAPE 2D VIEW**

Creates a 2D object which is a flattened 2D view of another 3D object.

13. 

DRAFT TO SKETCH

Converts a Draft object to Sketch and vice-versa.



15.

ARRAY

Converts a Draft object to Sketch and vice-versa.



16.

PATH ARRAY

Creates an array of objects by placing the copies along a path.



17.

CLONE

Clones the selected objects.



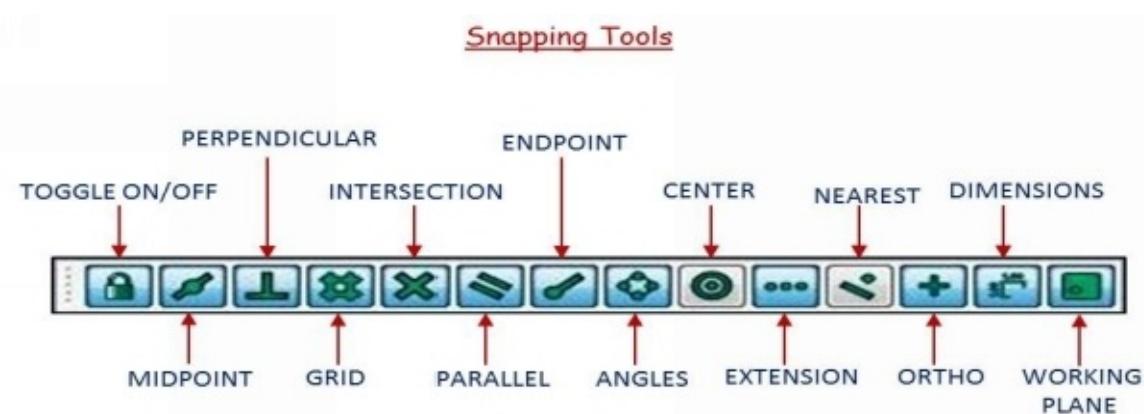
18.

DRAWING

Writes selected objects to a Drawing sheet

6.3 SNAPPING TOOLS

Snapping tools is a tool which is used for “Gluing” your next 3D point onto existing point. Snapping is available with most Draft and Arch tools.



Snapping tools can be enabled and disabled globally with the (Snap on/off) command. Each snap tool can also be enabled or disabled individually by clicking the corresponding button on the snap toolbar.

The descriptions of Snapping Tools are listed below:-

<p>1. </p> <p>TOGGLE ON/OFF</p> <p>Turns snapping on/off globally.</p> <p>2. </p> <p>MIDPOINT</p> <p>The middle point of line and arc segments.</p> <p>3. </p> <p>PERPENDICULAR</p> <p>On line and arc segments, perpendicularly to the latest point.</p> <p>4. </p> <p>GRID</p> <p>The nodes of the Draft grid, if visible.</p> <p>5. </p> <p>INTERSECTION</p> <p>The intersection of 2 line or arc segments. However the mouse over the two desired objects to activate their intersection snaps.</p> <p>6. </p> <p>PARALLEL</p> <p>On an imaginary line parallel to a line segment. However the mouse over the desired object to activate its parallel snap.</p> <p>7. </p> <p>ENDPOINT</p> <p>The endpoints of line, arc and spline segments.</p> <p>8. </p> <p>ANGLES</p> <p>The special cardinal points of circles and arcs, at 45° and 90°.</p> <p>9. </p> <p>CENTER</p>	6.4
--	-----

The center point of arcs and circles.

10. 

EXTENSION

On an imaginary line that extends beyond the endpoints of line segments. However the mouse over the desired object to activate its extension snaps.

11. 

NEAREST

The closest point on the nearest object.

12. 

ORTHO

On imaginary lines that cross the last point, and extend at 0° , 45° and 90° .

13. 

DIMENSIONS

To make the dimensions on drafting.

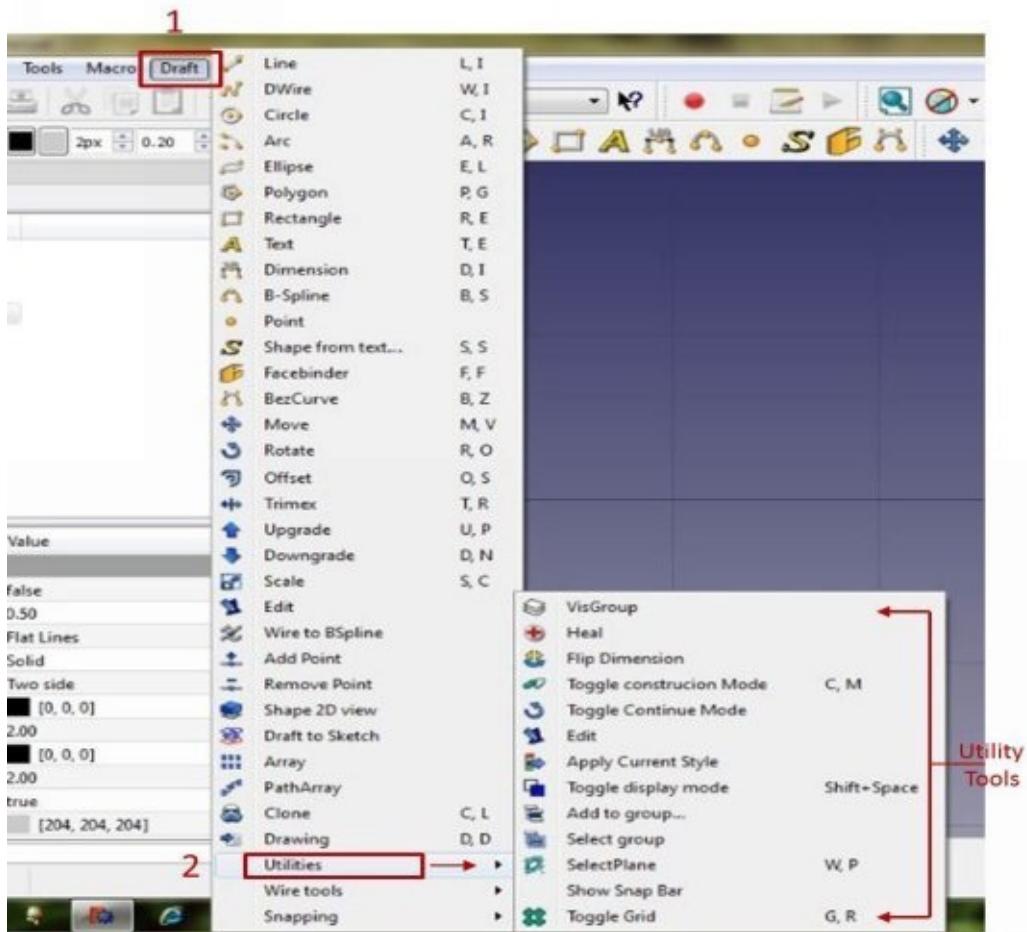
14. 

WORKING PLANE

Always places the snapped point on the current working plane even if you snap to a point outside that working plane.

UTILITY TOOLS

Utility tools are used for make a final touch in your 2d drafting. It is an Additional tool available via right-click context menu, depending on the selected objects. See the Fig. below.



For find the utility tools, first click on the **Draft** menu **utilities** then select object which you want.

The descriptions of Utility Tools are listed below:-



1.

VISGROUP

Creates a VisGroup in the current document.



2.

HEAL

Heals problematic Draft objects found in very old files.



3.

FLIP DIMENSION

Flips the orientation of the text of a dimension.



4.

TOGGLE CONSTRUCTION MODE
Toggles the Draft construction mode on/off.

5.



TOGGLE CONTINUE MODE
Toggles the Draft continue mode on/off.

6.



EDIT
Edits a selected object.

7.



APPLY CURRENT STYLE
Applies the current style and colour to selected objects.

8.



TOGGLE DISPLAY MODE
Switches the display mode of selected objects between "flat lines" and "wireframe".

9.



ADD TO GROUP
Quickly adds selected objects to an existing group.

1.



SELECT GROUP
Selects the contents of a selected group.

2.



SELECT PLANE
Sets a working plane from a standard view or a selected face.

3.



SHOW SNAP BAR
Shows/hides the **snapping** toolbar.

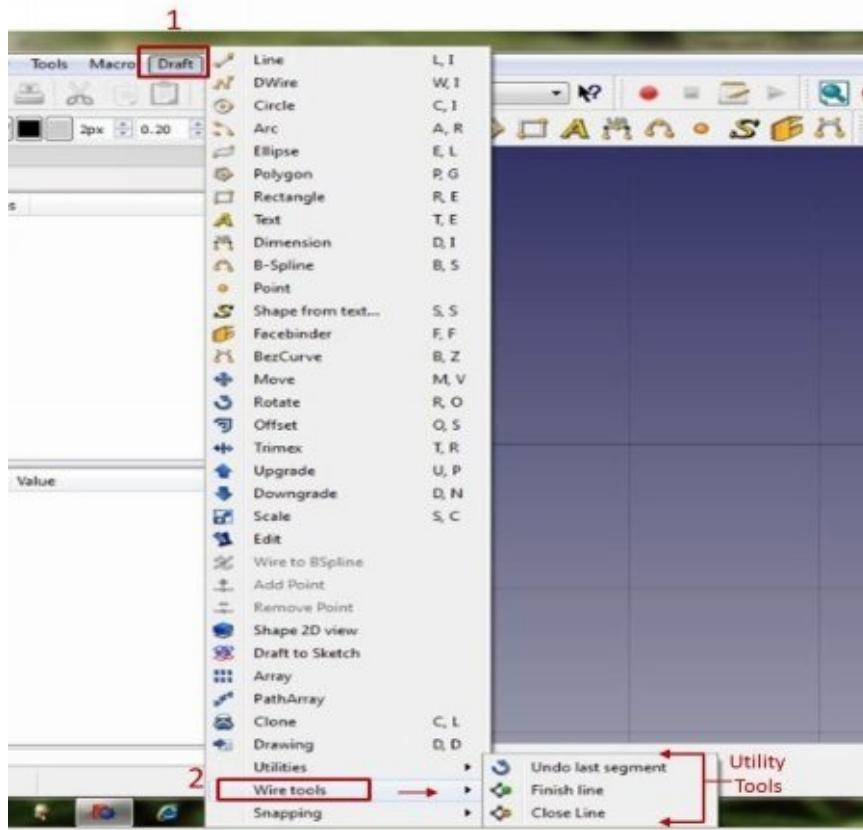
4.



TOOGLE GRID
Toggles the grid on/off.

6.1 WIRE TOOLS

Wire tools are basically used for making the polyline or wire with the help of  wire tool (Polyline). See the Fig. below,



For find the Wire tools, first click on the **Draft** menu → **Wire tools** → then select object which you want.

The descriptions of **Wire Tools are listed below:-**



1. UNDO LINE

Undoes the last segment of a line.



2. FINISH LINE

Ends the drawing of the current wire or Bspline, without closing it.



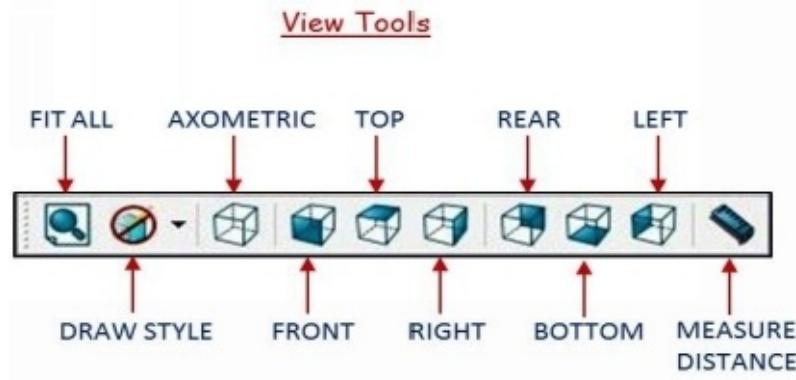
3.

CLOSE LINE

Ends the drawing of the current wire or Bspline, and closes it.

6.2 VIEW TOOLS

This tools are very usefull for showing the object in different location. This tools are always available in all workbenches.



The descriptions of View Tools are listed below:-



FIT ALL

Fit the whole content on the screen without the zooming mode.



DRAW STYLE



AS IS

It is draw style and keeps the object as it is.



FLAT LINES

This tool shows the object in flat mode.



SHADED

This tool shows the object in shaded mode.

2.4 

WIREFRAME

This tool shows the object like wireframe.

2.5 

POINTS

This tool shows the object like points

3. 

AXOMETRIC VIEW

Set the object to Isometric view.

4. 

FRONT VIEW

Set the object to Front view.

5. 

TOP VIEW

Set the object to Top view.

6. 

RIGHT VIEW

Set the object to Right view.

7. 

REAR VIEW

Set the object to Rear view.

8. 

BOTTOM VIEW

Set the object to Bottom view.

9. 

LEFT VIEW

Set the object to Left view.

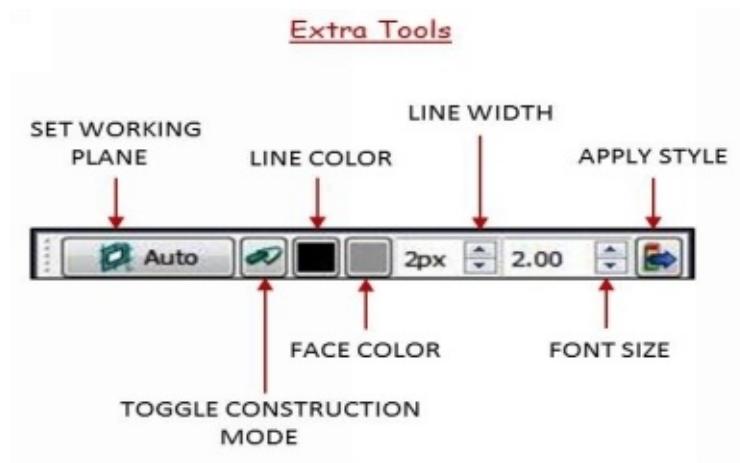
10. 

MEASURE DISTANCE

To use for measure the distance point to point.

6.3 EXTRA TOOLS

It is generally located at left side corner of the **Draft Command Bar**. On the Draft command bar, you will see three buttons: a line width setting, a line colour button, and an "apply" button. If objects are selected when you change those values, they will receive automatically the new values. If no object is selected, the changes you make will apply to objects you will create later. At any moment, you can hit the "apply" button to apply current settings to selected objects.



The descriptions of Extra Tools are listed below:-



SET WORKING PLANE

Sets a working plane from a standard view or a selected face.



TOGGLE CONSTRUCTION MODE

Toggles the Draft construction mode on/off.



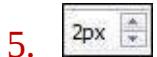
LINE COLOR

Set the line in different colour.



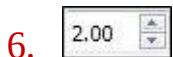
FACE COLOR

Set the face in different colour.



LINE WIDTH

Set the width of line.



FONT SIZE

Set the size of font.

7. 

APPLY STYLE

Applies the current style and colour to selected objects.

END

5. OPERATION WITH – DRAFT



: - We have already discussed about the draft or draft workbench. The Draft workbench allows to quickly drawing simple 2D objects in the current document, and offers several tools to modify them afterwards. Now we will discuss that, how to make drawing or module with draft workbench. During operation with draft, the **drawing tools** and **modifying tools** play very important roles for drawing the 2D objects. So first, we will learn, how to make simple 2D drawing like Line, Polyline, Circle etc and its modification.

WORKING WITH DRAWING TOOLS

5.1 HOW TO MAKE LINE



LINE:- The Draft Line tool creates, straight line or two points line in the current work plane. The Line tool behaves exactly like the [Draft Wire](#) tool, except that it stops after two points. we can also change the colour and width of line by using Extra tools.



- OPERATION

1. Run or open the FreeCAD software.

2. Switch the Workbench and select the Draft.
3. Then, select the  **Line** or press **L** then **I** from keyboard.
4. Click a first point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
5. Click a second point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).

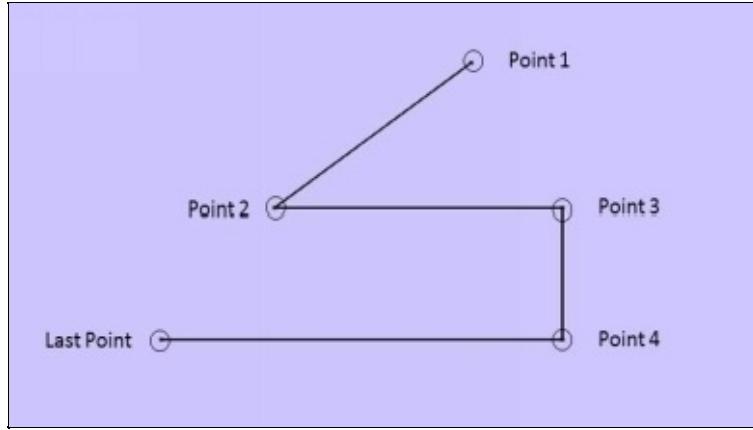
- **PROPERTIES**

:- The properties value tab are used for modification of line. Select the Line from the Combo view → Model and then put the value what ever you want, in Property Value.

- 1. For placement to the line: - **Property** → **Data** → Placement.
 2. For Line color, Line width, Point color of the line: - **Property** → **View**.
 3. For starts the point of the line: - **Property** → **Data** → Start.
 4. For end the point of the line: - **Property** → **Data** → Start.

5.2 HOW TO MAKE WIRE

 **WIRE:** - The Draft Wire tool creates, a polyline or wire in the current work plane. This Wire tool behaves like the Draft Line tool, except that it doesn't stop after two points. We can also change the colour and width of wire by using Extra tools.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Draft.
3. Then, select the  **Wire** or press **W** then **I** from keyboard.
4. Click a first point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
5. Click an additional point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
6. Select  **Finish** button for finish the wire from Combo View → Task.
7. Select  **Close** button for close the wire from Combo View → Task.
8. Or you can also press **F** or **C** form keyboard, or double-click the last point, or click on the first point to finish or close the wire.

- PROPERTIES

:- The properties value tab are used for modification of wire.

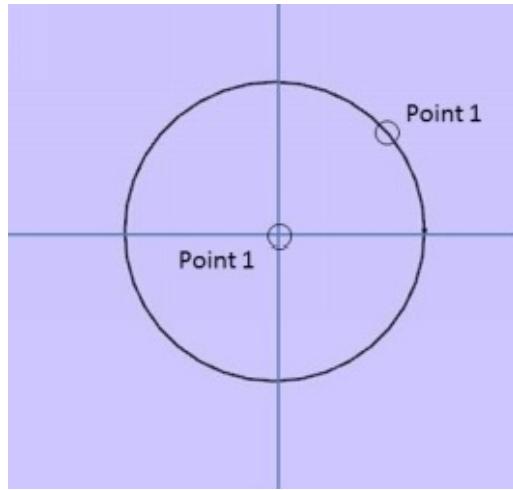
Select the Wire from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement to the wire: - **Property** → **Data** → Placement.
2. For chamfer size: - **Property** → **Data** → Chamfer Size.
3. For closed the wire: - **Property** → **Data** → Closed (false \ true).
4. For fillet radius: - **Property** → **Data** → Fillet Radius.
5. For wire color, wire width, Point color of the wire: - **Property** → **View**.
6. For pattern style of the closed wire: - **Property** → **View** → Pattern.
7. For pattern size of the closed wire: - **Property** → **View** → Pattern Size.

5.3 HOW TO MAKE CIRCLE



CIRCLE: - The Draft Circle tool creates a circle in the current work plane by entering two points, the centre and the radius. we can also change the colour and width of circle by using these tools.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Draft.
3. Then, select the  **Circle** or press **C** then **I** from keyboard.
4. Click a first point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
5. Click a second point on 3D view or enter a radius value (Combo View → Task → Radius).

- PROPERTIES

:- The properties value tab are used for modification of circle.
Select the Circle from the Combo view → Model and then put the value whatever you want, in Property Value.

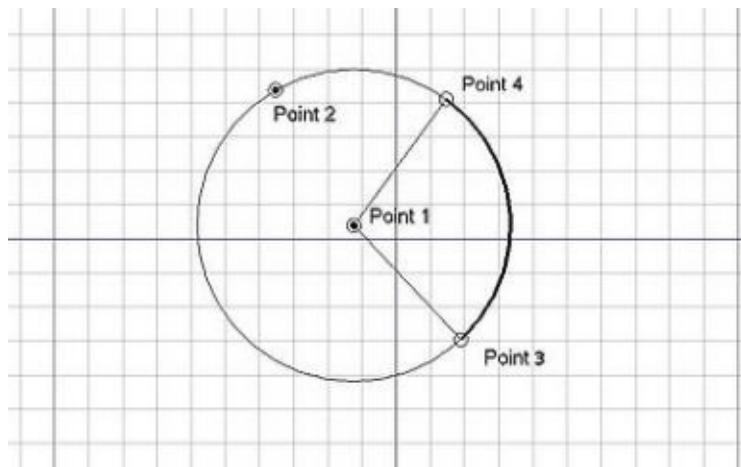
1. For placement to the Circle: - **Property** → **Data** → **Placement**.
2. For first angle: - **Property** → **Data** → **First Angle**.

3. For last angle: - **Property** → **Data** → Last Angle.
4. For make a face: - **Property** → **Data** → Make Face (false/true).
5. For radius: - **Property** → **Data** → Radius value.
6. For circle color, circle width, Point color of the circle: - **Property** → **View**

5.4 HOW TO MAKE ARC



ARC: - The Arc tool creates an arc in the current work plane by the use of four points, the centre, the radius, the first point and the last point. The arc is also creates my drawing tangents or my any combination of those. we can also change the colour and width of circle by using these tools.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Draft.
3. Then, select the  **Arc** or press **A** then **R** from keyboard.
4. Click a first point on 3D view or type a coordinate (Combo View →

Task and set X, Y, Z coordinate).

5. Click a second point on 3D view or entre a radius value (Combo View → Task → Radius).
6. Click a third point on 3D view or enter a start angle (Combo View → Task and put start angle value).
7. Click a last point on 3D view or enter an Aperture value (Combo View → Task and put Aperture value).

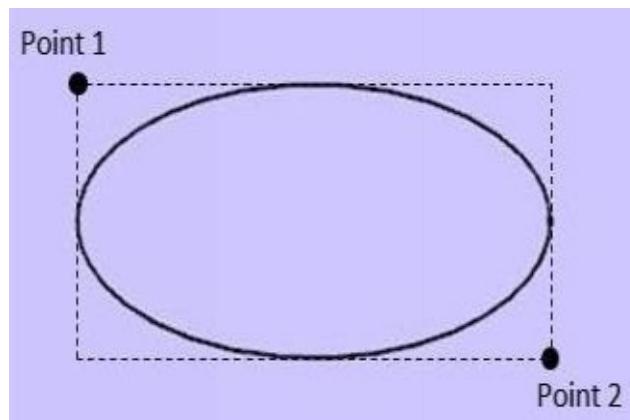
- PROPERTIES

:- The properties value tab are used for modification of arc. Select the Arc from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement to the Arc: - **Property** → **Data** → **Placement**.
2. For first angle: - **Property** → **Data** → **First Angle**.
3. For last angle or Aperture: - **Property** → **Data** → **Last Angle**.
4. For make a face: - **Property** → **Data** → **Make Face (false/true)**. For closed Arc.
5. For radius: - **Property** → **Data** → **Radius** value.
6. For arc color, arc width, Point color of the arc: - **Property** → **View**.

5.5 HOW TO MAKE ELLIPSE

 **ELLIPSE** : - The ellipse tool creates an ellipse in the current work plane by entering two points, defining the corner of a rectangular box in which the ellipse will fit.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Draft.
3. Then, select the  Ellipse or press E then L from keyboard.
4. Click a first point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
5. Click a second point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).

- PROPERTIES

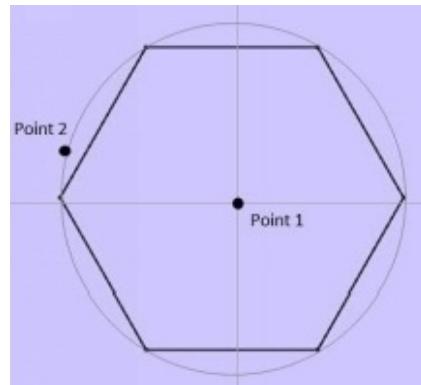
:- The properties value tab are used for modification of ellipse.
Select the Ellipse from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement to the Ellipse: - **Property** → **Data** → **Placement**.
2. For first angle: - **Property** → **Data** → **First Angle**.
3. For last angle or Aperture: - **Property** → **Data** → **Last Angle**.
4. For make a major radius: - **Property** → **Data** → **Major Radius** value.
5. For make a face: - **Property** → **Data** → **Make Face (false/true)**.
6. For make a minor radius: - **Property** → **Data** → **Minor Radius** value.
7. For Line color, Line width, Point color of the line: - **Property** → **View**.

5.6 HOW TO MAKE POLYGON



POLYGON : - The polygon tool creates a regular polygon in the current work plane by picking two points, the center and a second point defining a radius.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Draft.
3. Then, select the **Polygon** or press **P** then **G** from keyboard.
4. Click a first point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
5. Set the desired number of sides of polygon (Combo View → Task and put the Sides value).
6. Click a second point on 3D view or type a radius value (Combo View → Task and set Radius value).

- PROPERTIES

:- The properties value tab are used for modification of Polygon. Select the Polygon from the Combo view → Model and then put the value what

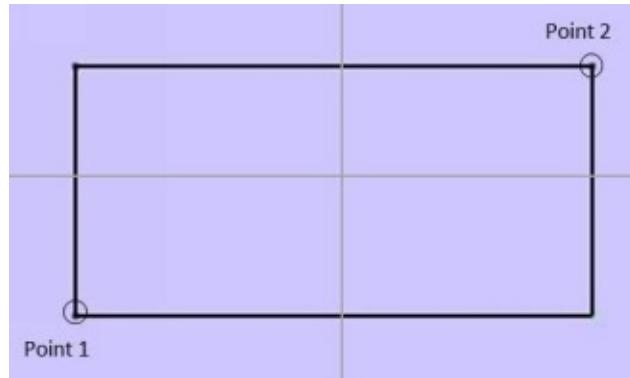
ever you want, in Property Value.

1. For placement to the Polygon: - **Property** → **Data** → **Placement**.
2. For chamfer size: - **Property** → **Data** → **Chamfer Size**.
3. For draw mode: - **Property** → **Data** → **Draw Mode**
(inscribed/circumscribed).
4. For face number or number of sides: - **Property** → **Data** → **Face Num....**
5. For make a face: - **Property** → **Data** → **Make Face (false/true)**.
6. For make a curvature radius: - **Property** → **Data** → **Radius** value.
7. For ellipse color, ellipse width, Point color of the ellipse: - **Property** → **View**.

5.7 HOW TO MAKE RECTANGLE



RECTANGLE : - The draft rectangle tool creates a Rectangle in the current work plane by taking two opposite points.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Draft.
3. Then, select the  **Rectangle** or press **R** then **E** from keyboard.
4. Click a first corner point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
5. Click a second opposite corner point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).

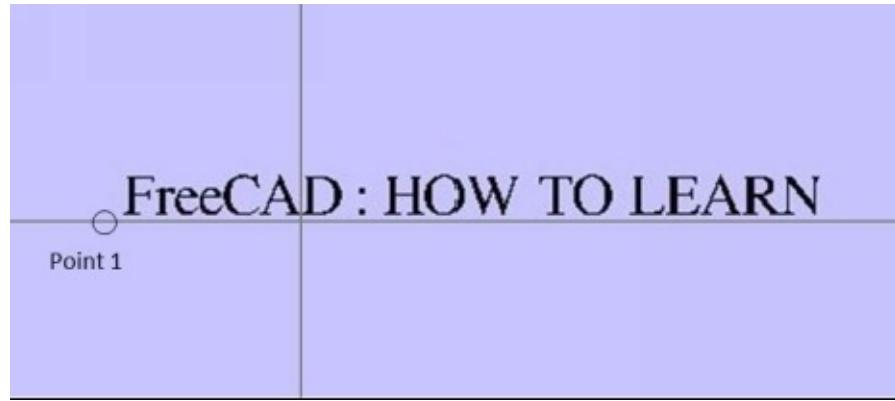
- **PROPERTIES**

:- The properties value tab are used for modification of rectangle. Select the Rectangle from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement to the rectangle: - **Property** → **Data** → **Placement**.
2. For chamfer size: - **Property** → **Data** → **Chamfer Size**.
3. For make a fillet radius: - **Property** → **Data** → **Fillet Radius** value.
4. For make a height: - **Property** → **Data** → **Height** value.
5. For make a length: - **Property** → **Data** → **Length** value.
6. For make a face: - **Property** → **Data** → **Make Face (false/true)**.
7. For rectangle color, rectangle width, Point color of the rectangle: - **Property** → **View**.

5.8 HOW TO MAKE TEXT ANNOTATION

A TEXT : - The text tool is use for draw a multi-line annotation text at a given point in the current work plane.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Draft.
3. Then, select the **A** Text or press **T** then **E** from keyboard.
4. Click a first corner point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
5. Enter the desired text, pressing **ENTER** between each line (Combo View → Task and type the desired Text).
6. Press **ENTER** twice to finished the operation.

- PROPERTIES

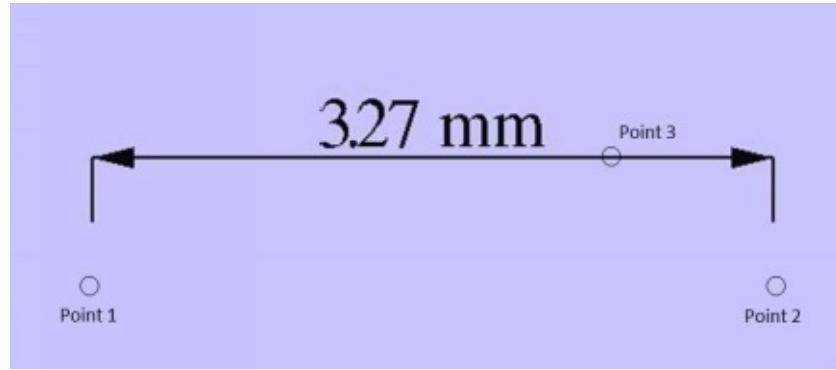
:- The properties value tab are used for modification of text. Select the text from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For label Text: - **Property** → **Data** → **Label Text**.
2. For position of text: - **Property** → **Data** → **Position**.

3. For make display mode: - **Property** → **View** → **Display Mode** (Screen/World).
4. For make a font size: - **Property** → **View** → **Font Size**.
5. For make an aligned of text: - **Property** → **View** → **Justification** (Left/Right/Centre).
6. For line spacing: - **Property** → **View** → **Line Spacing**.
7. For rotation axis: - **Property** → **View** → **Rotation Axis**.
8. For text color: - **Property** → **View** → **Text Color**.
9. For visibility of text: - **Property** → **View** → **Visibility** (false/true).

5.9 HOW TO MAKE DIMENSION

 **DIMENSION**: - The dimension tool use to create a dimension in the current work plane with two points defining the distance to measure, and a third point specifying where the dimension line passes.



- OPERATION

1. Run or open the FreeCAD software.

2. Switch the Workbench and select the Draft.
3. Then, select the  **Dimension** or press **D** then **I** from keyboard.
4. Click a first point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
5. Click a second point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
6. Click a third point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).

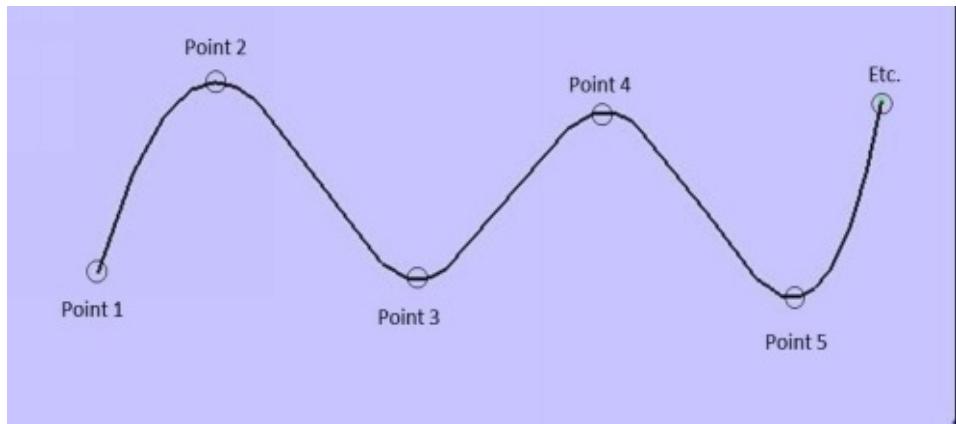
- **PROPERTIES**

:- The properties value tab are used for modification of dimension. Select the dimension from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For arrow size of dimension: - **Property** → **View** → **Arrow Size**.
2. For arrow type of dimension: - **Property** → **View** → **Arrow Type**.
3. For dimension color, width, Point color of the dimension: - **Property** → **View**.
4. For start point of dimension: - **Property** → **View** → **Start Point**.
5. For end point of dimension: - **Property** → **View** → **End Point**.

5.10 HOW TO MAKE B-SPLINE

 **B-SPLINE** : - The B-spline tool is use to create a B-spline curve with the help of several points in the current work plane. This tool all most behaves like the Draft tool.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Draft.
3. Then, select the  **B-spline** or press **B** then **S** from keyboard.
4. Click a first point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
5. Click an additional point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
6. Then press **F** or **C** or double click on the last point.
7. You can also click on the first point for finish or close the spline.

- PROPERTIES

:- The properties value tab are used for modification of B-spline. Select the B-spline from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For closed the spline: - **Property** → **Data** → **Closed** (false/true).
2. For make a face: - **Property** → **Data** → **Make face** (false/true – only for closed spline).
3. For spline color, width, Point color of the B-spline: - **Property** → **View**.

5.11 HOW TO MAKE POINT

● **POINT**: - The point tool is used to create a simple point in current work plane.



- **OPERATION**

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Draft.
3. Then, select the ● **Point** or press **P** then **T** from keyboard.
4. Click a point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).

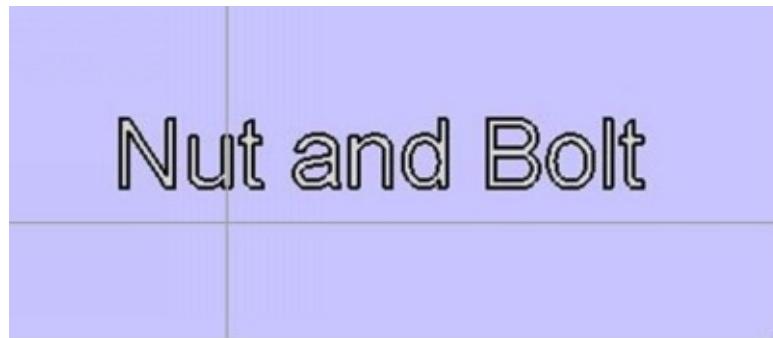
- **PROPERTIES**

:- The properties value tab are used for modification of Point. Select the point from the Combo view → Model and then put the value whatever you want, in Property Value.

1. For Placement: - **Property** → **Data** → X or Y or Z value.
2. For Point color, point size: - **Property** → **View**.

5.12 **HOW TO MAKE SHAPE STRING**

S **SHAPE STRING**: - The shape String tool is used to inserts a compound shape representing a text string at a given point in the current work plane.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Draft.
3. Then, select the **S** Shape string or press S then S from keyboard.
4. Click a point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
5. Put the desired text (Ex - Nut and Bolt) and press the **Enter**.
6. Put the text height and press **Enter**.
7. Put the desired tracking and press **Enter**.
8. Press the **Enter** and accept the displayed font file or set the font.

NOTE: -

You can set the defulet Font design form following steps,

1. Select the **Edit** menu.
2. Then select the **Preferences** and press the **Draft**.
3. Select the **Visual setting** and set the desired font design from Default Shape String Font File (see the figure).



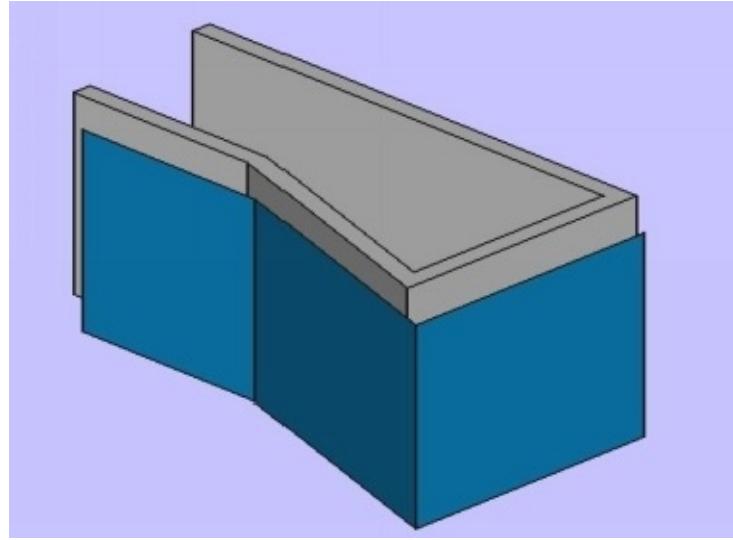
- PROPERTIES

:- The properties value tab are used for modification of Shape String. Select the Shape String from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For Placement: - **Property** → **Data** → **Placement (X or Y or Z value).**
2. For font file: - **Property** → **Data** → **Font File.**
3. For size of string: - **Property** → **Data** → **Size.**
4. For string test: - **Property** → **Data** → **String.**
5. For tracking: - **Property** → **Data** → **Tracking.**
6. You can also set the color of string form **Property** → **View.**

5.13 HOW TO MAKE FACEBINDER

 **FACEBINDER** : - The facebinder is a very simple object constructed from selected faces of other object. It can also use for example for making an extraction out of a collection of faces from other objects.



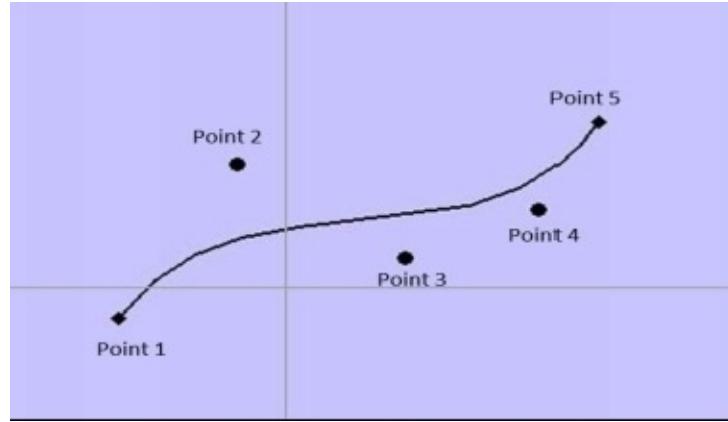
- OPERATION

1. Run or open the FreeCAD software.
2. Follow the **video tutorial link** for further procedure and for better understanding.

[**https://www.youtube.com/watch?v=QfhbCkWt2iU**](https://www.youtube.com/watch?v=QfhbCkWt2iU)

(How to use Facebinder tool in FreeCAD) 5.14 HOW TO MAKE BEZIER CURVE

 **BEZIER CURVE** : - The Bezcurve or Bezier Curve tool is used to create a Bezier curve or a piecewise Bezier curve in current work plane with the help of several points.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Draft.
3. Then, select the  **Bezcurve** or press **B** then **Z** from keyboard.
4. Click a point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
5. Click another or additional point on 3D view or type a coordinate (Combo View → Task and set X, Y, Z coordinate).
6. Double-click the last point, or click on the first point to finish and close the curve or Press **F** or **C**.

- PROPERTIES

:- The properties value tab are used for modification of Bezier curve. Select the Bezier curve from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For Placement: - **Property** → **Data** → **Placement (X or Y or Z)**

value).

2. For close the Bezcurve: - **Property** → **Data** → **Closed** (false/true).
3. For degree of the Bezcurve: - **Property** → **Data** → **Degree**.
4. For make a face of the Bezcurve: - **Property** → **Data** → **Make face** (only for closed curve).

WORKING WITH MODIFYING TOOLS

: - In Draft workbench, the modifying tools play a great roll to modifying the objects. They work on selected objects. If you will use the modifying tools without selecting the object, it gives error so make it correct.

1. HOW TO USE MOVE, ROTATE, OFFSET & TRIMEX TOOLS

 **MOVE**: - The Move tool moves or copies the selected objects from one point to another on the current work plane.

 **ROTATE** : - This tool rotates or copies the selected objects by a given angle around a point on the current work plane .

 **OFFSET** : - The Offset tool offsets the selected object by a given distance on the current work plane .

 **TRIMEX** : - This tool is use to trims/cuts and extends lines and polylines, and extrudes faces on current work plane.

• OPERATION

Follow the **video tutorial link** for how to use these tools.

<https://youtu.be/-mzOmATi0Ws>

[HOW TO USE MODIFYING TOOLS IN FreeCAD (PART -1)]

2. HOW TO USE UPGRADE, DOWNGRADE, SCALE & EDIT TOOLS



UPGRADE : - This tool is use to upgrades selected objects in different ways. If no object is selected, you will be invited to select one.



DOWNGRADE: - This too is use to downgrades selected objects in different ways. If no object is selected, you will be invited to select one.



SCALE : - This tool is use to scales selected object(s) around a base point. If no object is selected, you will be invited to select one. It can also be used to mirror objects.



EDIT: - The edit tool allows you to edit graphically certain properties of the selected object, such as the vertices of a Wire, or the length and width of a Rectangle, or the radius of a Circle.

• OPERATION

Follow the **video tutorial link** for how to use these tools.

<https://youtu.be/N1bk2XEKxBo>

[HOW TO USE MODIFYING TOOLS IN FreeCAD (PART -2)]

3. HOW TO USE WIRE TO BSPLINE, ADD POINT, DELETE POINT & SHAPE 2D VIEW



WIRE TO BSPLINE : - The wire to Bspline tool is use to converts Wires to Bsplines, and vice-versa.



ADD POINT : - This tool allows you to add additional points to wires and Bsplines.



DELETE POINT : - This tool allows you to delete or remove additional points to wires and Bsplines.



SHAPE 2D VIEW : - This tool places in the document a 2D object which is a flattened view of a selected Shape - based object.

- **OPERATION**

Follow the **video tutorial link** for how to use these tools.

<https://youtu.be/a01TTbkL-o8>

[HOW TO USE MODIFYING TOOLS IN FreeCAD (PART -3)]

4. **HOW TO USE DRAFT TO SKETCH, ARRAY, PATH ARRAY, CLONE & DRAWING**



DRAFT TO SKETCH : - This tool is use to convert Draft objects to Sketcher objects and vice-versa.



ARRAY : - The Array tool creates an orthogonal (3-axes) or polar array from a selected object. If no object is selected, you will be invited to select one.



PATH ARRAY : - The Path Array tool places copies of a selected shape along a selected path. The path can be a Wire or one or more Edges. The shapes can optionally be aligned with the tangent of the path. If required, a translation Vector can be specified to shift the shapes so the centroid is on the path. If no objects are selected, you will be invited to select them.



CLONE : - This tool produces a clone (a copy that is parametrically bound to the original object) of a selected object. If the original object changes, the clone changes too, but keeps its position, rotation and scale .



DRAWING : - This tool is used to allow you to put selected objects on a svg Drawing sheet If no sheet exists in the document, a default one will be created.

- **OPERATION**

Follow the **video tutorial link** for how to use these tools.

[**https://youtu.be/C38zrtad-M**](https://youtu.be/C38zrtad-M)

[HOW TO USE MODIFYING TOOLS IN FreeCAD (PART -3)]

END

6. SKETCHER WORKBENCH



: - The **Sketcher Workbench** is used to create 2D geometries intended for use in the Part Design Workbench and other workbenches. This workbench itself features constraints - allowing 2D shapes to be constrained to precise geometrical definitions and a constraint solver which calculates the constrained-extent of 2D geometry and allows interactive exploration of sketch degrees-of-freedom.

WHAT ARE CONSTRAINTS ?

Constraints are used to limit the degrees of freedom of an object. For example, a line without constraints has 4 degrees of freedom: it can be moved horizontally or vertically, it can be stretched, and it can be rotated.

Applying a horizontal or vertical constraint, or an angle constraint (relative to another line or to one of the axes), will limit its capacity to rotate, thus leaving it with 3 degrees of freedom. Locking one of its points in relation to the origin will remove another 2 degrees of freedom. And applying a dimension constraint will remove the last degree of freedom. The line is then considered fully-constrained.

Multiple objects can be constrained between one another. Two lines can be joined through one of their points with the coincident point constraint. An angle can be set between them, or they can be set perpendicular. A line can be tangent to an arc or a circle, and so on.

DIFFERENCE BETWEEN TRADITIONAL DRAFTING & CONSTRAINT SKETCHING

Traditional Drafting: -

The traditional way of CAD drafting inherits from the old drawing board. Orthogonal (2D) views are drawn manually and intended for producing technical drawings (also known as blueprints). Objects are drawn precisely to the intended size or dimension. If you want to draw an horizontal line 100mm in length starting at (0,0), you activate the line tool, either click on the screen or input the (0,0) coordinates for the first point, then make a second click or input

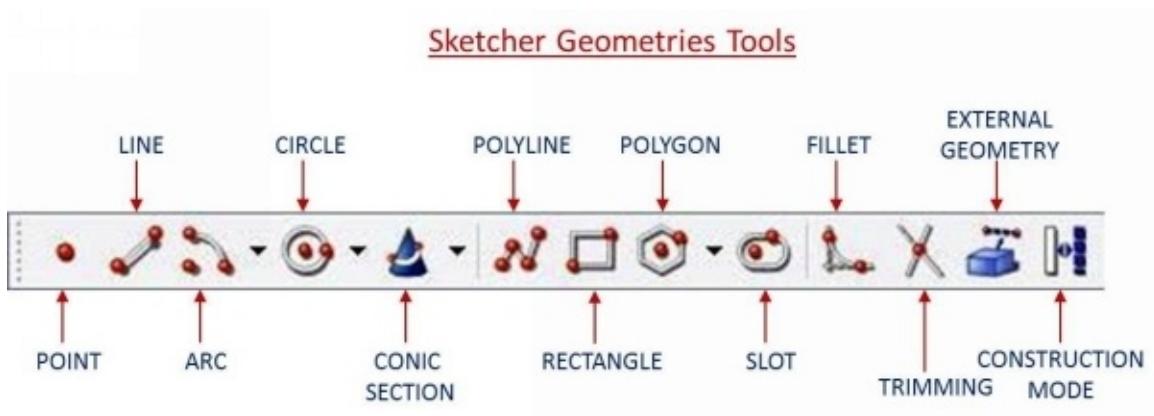
the second point coordinates at (100,0). Or you will draw your line without regard to its position, and move it afterward. When you've finished drawing your geometries, you add dimensions to them.

Constraint Sketching: -

The Sketcher moves away from this logic. Objects do not need to be drawn exactly as you intend to, because they will be defined later on by constraints. Objects can be drawn loosely, and as long as they are unconstrained, can be modified. They are in effect "floating" and can be moved, stretched, rotated, scaled, and so on. This gives great flexibility in the design process.

6.1 SKETCHER GEOMETRIES TOOLS

The Sketcher Geometries tools are used to create the 2D object in sketcher workbench.



The descriptions of Sketcher Geometries Tools are listed below:-

a)



This tool is use to create a point in the current sheet of Sketcher.

b)



The Line tool is use to create a two point line in the current sheet.

c)



- This tool is used to draw an arc in 3D view.

-



Arc By Centre and End Points: This tool draws an arc by picking three points: the center, the start angle along the radius, and the end angle.

-



Arc By End Points and Rim Point:

This tool draws an arc segment from two endpoints and another point on the circumference .

d)



CIRCLE: - This tool is used to draw a

circle in current sheet of Sketcher.

-



Circle By Centre and Rim Point:

This tool draws a circle by picking two points: the center, and a point along the radius.

-



Circle By Three Rim Points: This

tool is used to draw a circle from three points on the circumference.

e)



CONIC SECTION: - This tool is used to

create a conic section on sketcher workbench environment.

-



Ellipse By Centre: This tool is used to

draw an ellipse by center point, major radius point and minor radius point.

-



Ellipse By 3 Points: This tool is used

to create an ellipse by major diameter (2 points) and minor radius point.

-  **Arc Of Ellipse**: This tool is used to draw an arc of ellipse by center point, major radius point, starting point and ending point.

f)  **POLYLINE**: - This tool works like the Sketcher Line tool, but creates continuous line segments connected by their vertices.

g)  **RECTANGLE**: - This tool is used to create a rectangle from two opposite points.

h)  **POLYGON**: - This tool is used to create a regular polygon in the current sheet of sketcher.

-  **Triangle**: This tool is used to draw an equilateral triangle inscribed in a construction geometry circle .

-  **Square**: This tool is used to create a regular square inscribed in a construction geometry circle.

-  **Pentagon**: This tool is used to create a regular pentagon inscribed in a construction geometry circle.

-  **Hexagon**: This tool is used to create a regular hexagon inscribed in a construction geometry circle.

-  **Heptagon**: This tool is used to create a regular heptagon inscribed in a construction geometry circle.

-  **Octagon**: This tool is used to create a

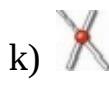
regular octagon inscribed in a construction geometry circle.



- i) **SLOT**: - This tool is used to draw a slot by selecting the center of one semicircle and an endpoint of the other semicircle .



- j) **FILLET**: - This tool is used to make a fillet between two lines joined at one point. Activate the tool, then select both lines or click on the corner point.



- k) **TRIMMING** : - This tool trims a line or circle to the nearest overlapping line.



- l) **EXTERNAL GEOMETRY**: - This tool is used to constrain elements of a sketch with reference to an element of an external solid, to which the sketch has been mapped.

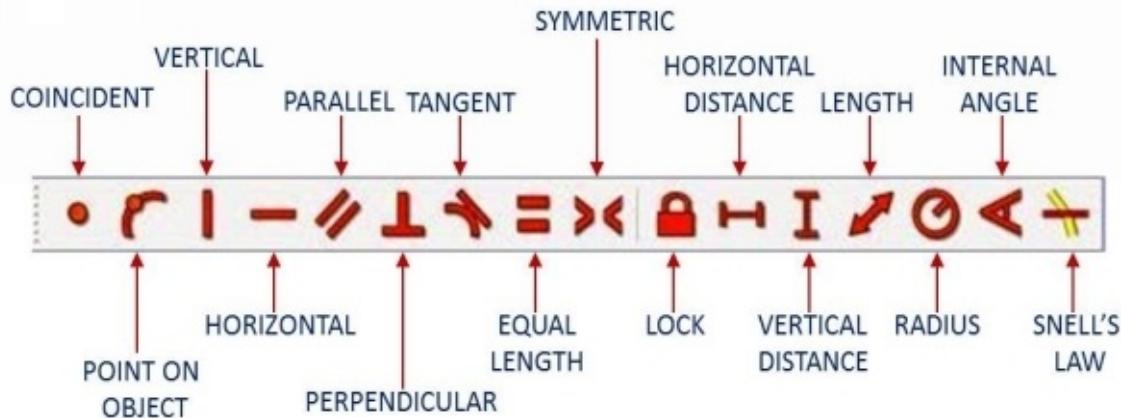


- m) **CONSTRUCTION MODE** : - This tool toggles sketch geometry from/to construction mode. It can be used on any type of geometry: line, arc or circle. Construction geometry is an important tool of the sketcher.

6.2 SKETCHER CONSTRAINTS TOOLS

Sketcher Constraints tools are used to define lengths, set rules between sketch elements, and to lock the sketch along the vertical and horizontal axes.

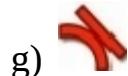
Sketcher Constraints Tools



The descriptions of Sketcher Constraints Tools are listed below:-

- a) **COINCIDENT** : - This constraint tool takes two points as its argument and serves to make the two points coincident. (Meaning to make them as one-point).
- b) **POINT ON OBJECT** : - This tool is used to affixes a point onto another object such as a line, arc, or axis.
- c) **VERTICAL** : - This tool is used to creates a vertical constraint to the selected lines or polylines elements. In this more than one object can be selected before applying this constraint.
- d) **HORIZONTAL** : - This tool is used to creates a horizontal constraint to the selected lines or polylines elements. In this more than one object can be selected before applying this constraint.
- e) **PARALLEL** : - This tool is used to make constraints two or more lines parallel to one another.
- f) **PERPENDICULAR** : - Perpendicular Constraint makes two lines to be perpendicular to each other, or two curves to be perpendicular at their

intersection. In this lines are treated as infinite, and arcs are treated as full circles/ellipses.



g) **TANGENT** :- Tangent Constraint makes two curves to touch each other (be tangent). Lines are treated as infinite, and arcs are treated as full circles/ellipses. This constraint is also capable of connecting two curves, forcing them tangent at the joint, thus making the joint smooth.



h) **EQUAL LENGTH** : - The Constrain Equal constraint forces two or more line segments in a line , polyline or rectangle to have equal length. If applied to arcs or circles the radii are constrained to be equal. It cannot be applied to geometry primitives which are not of the same type (e.g. line segments and arcs).



i) **SYMMETRIC** : - This tool is use to constrains two selected points to be symmetrical around a given line, i.e., both selected points are constrained to lie on a normal to the line through both points and are constrained to be equidistant from the line. Alternatively it can constrain two points to be symmetric with respect to a third one.



j) **LOCK** : - This tool is use to constraint any item fully. (Note:- it is advised that this tool is exclusively used on points for the time being.) k) 
HORIZONTAL DISTANCE : - This tool is use to fixes the horizontal distance between 2 points or line ends. If only one item is selected, the distance is set to the origin.



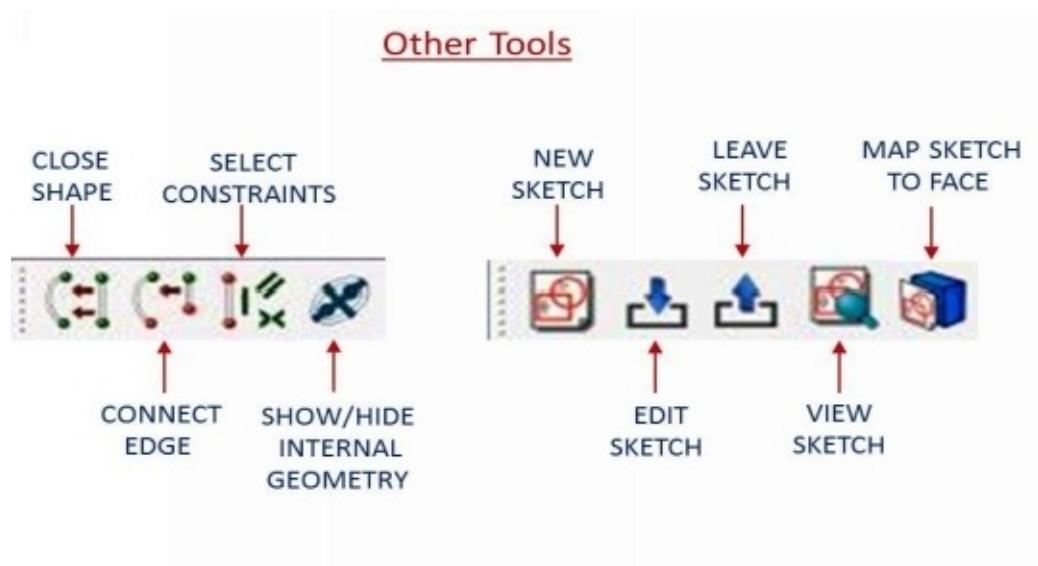
l) **VERTICAL DISTANCE** : - This tool is use to fixes the vertical distance between 2 points or line ends. If only one item is selected, the distance is set to the origin.



m) **LENGTH** : - Constraint Length constrains the length of a line, the perpendicular distance between a point and a line or the distance between two points to have a specified value.

- n)  RADIUS: - This constraint, constrains the value of the radius of a circle or arc to have a specific value. Only one arc or circle can be constrained at a time.
- o)  INTERNAL ANGLE: - This tool is a datum constraint intended to fix angles in sketch .
- p)  SNELLS LAW: - This tool is use to constrains two lines to follow the law of refraction of light as it penetrates through an interface, where two materials of different refraction indices meet.

6.3 OTHER TOOLS



The descriptions of Other Tools are listed below:-

- a)  CLOSE SHAPE: - This tool is use to produce shape by link end point of element with next element's.
- b)  CONNECT EDGE: - This tool is use to connect edge by link end point of element with next element's.



c) SELECT CONSTRAINTS: - This tool is used to select the constraints associated to the selected elements.



d) SHOW/HIDE INTERNAL GEOMETRY: - This tool is used to delete unused elements aligned to internal geometry, or recreates the missing ones.



e) NEW SKETCH: - This tool is used to create a new sketch on a selected face or plane.



f) EDIT SKETCH: - This command is used to edit the selected Sketch.



g) LEAVE SKETCH: - This command is used to leave the Sketch editing mode.



h) VIEW SKETCH: - This tool sets the model view perpendicular to the sketch plane. It is useful when the user has changed the model view orientation to examine another aspect of the model and wants to return to a view normal to the sketch.

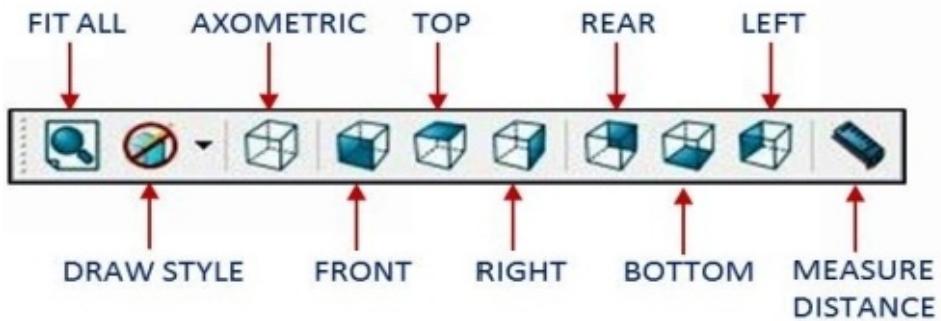


i) MAP SKETCH TO FACE: - This tool maps an existing sketch on the face of a shape.

6.4 VIEW TOOLS

These tools are very useful for showing the object in different locations. These tools are always available in all workbenches.

View Tools



NOTE: -

The description of this command is already discussed in **chapter Number – 4.**

END

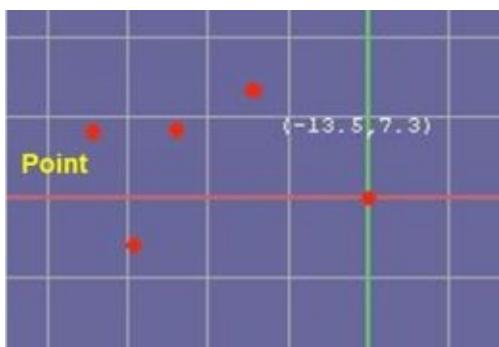
7. OPERATION WITH – SKETCHER



: - The **Sketcher Workbench** is used to create 2D geometries intended for use in the Part Design Workbench and other workbenches. This workbench itself features constraints - allowing 2D shapes to be constrained to precise geometrical definitions and a constraint solver which calculates the constrained-extent of 2D geometry and allows interactive exploration of sketch degrees-of-freedom.

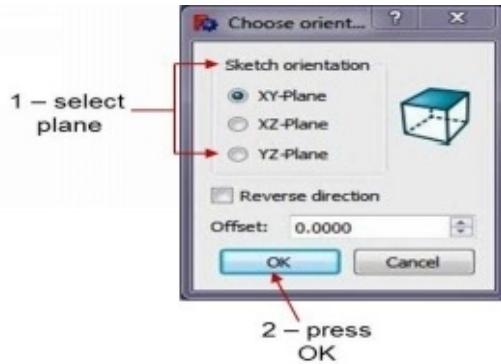
WORKING WITH SKETCHER GEOMETRIES TOOLS

7.1 HOW TO DRAW POINT



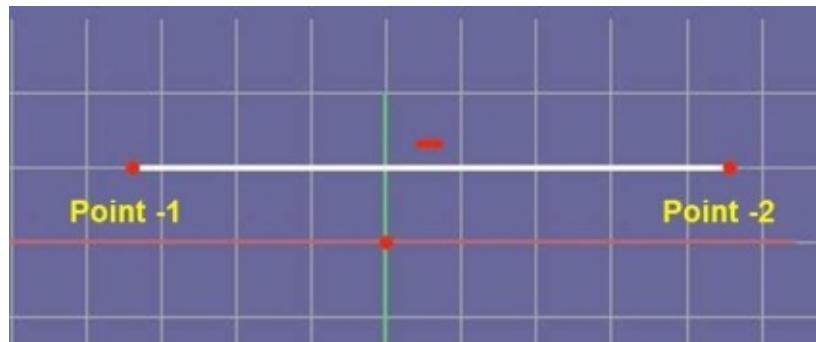
- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.



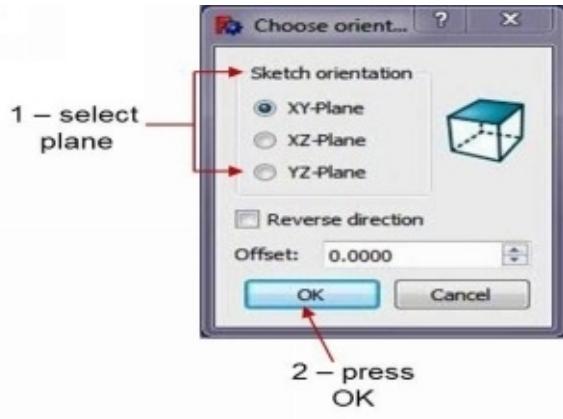
5. Select the  **Point tool**.
6. Click a point on 3D view, where you want to draw a point.
7. Then press the **Esc** button or click on **Close** tap from left side of the combo view.

7.2 HOW TO DRAW 2 POINT LINE



- **OPERATION**

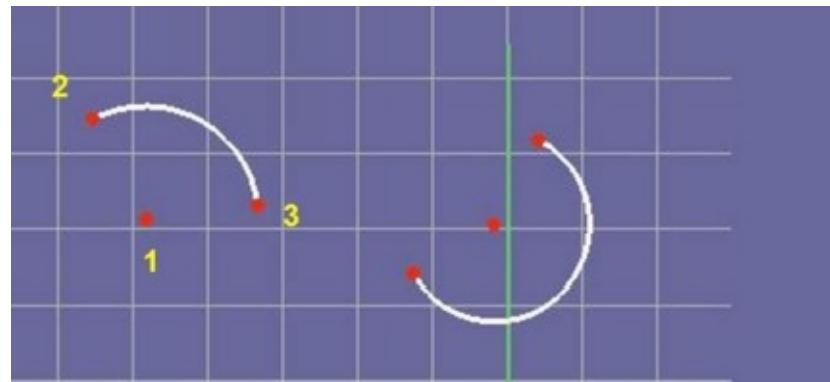
1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on  new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.



5. Select the  **Line tool**.
6. Click a first point on 3D view, where you want to make a line.
7. Click a another or second point on 3D view.
8. Then press the **Esc** button or click on **Close** tap from left side of the combo view.

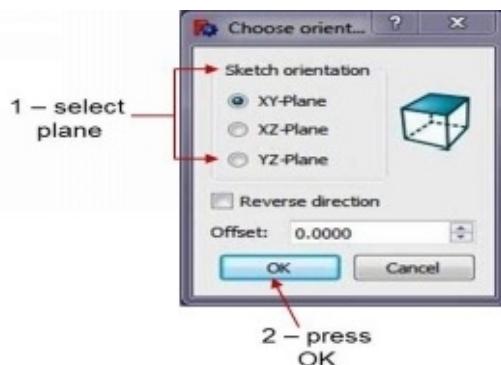
7.3 HOW TO DRAW ARC

The Arc tool is used to create two types of arc. First one is arc by center & end points and second one is arc by end point & rim point.



• OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on  new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.

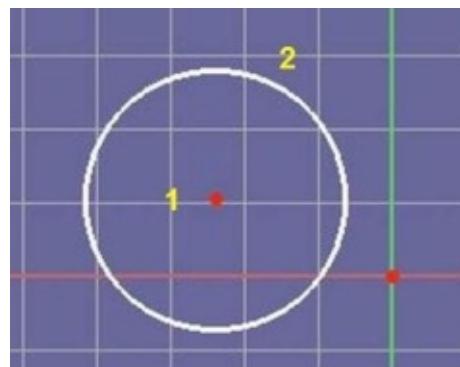


5. Select the  **Arc tool** by centre & end point.

6. Or select the  **Arc tool** by end & rim point.
7. Click a first point or centre point on 3D view.
8. Click a second point or set radius on 3D view.
9. Click a third point at same radius on 3D view.
10. Then press the **Esc** button or click on **Close** tap from left side of the combo view.

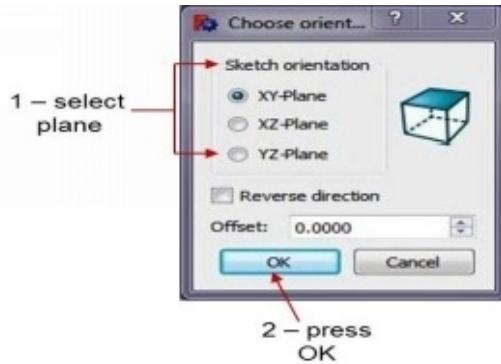
7.4 HOW TO DRAW CIRCLE

The Circle tool is used to create two types of circle. First one is circle by center & rim points and second one is circle by 3 rim points.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on  new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.

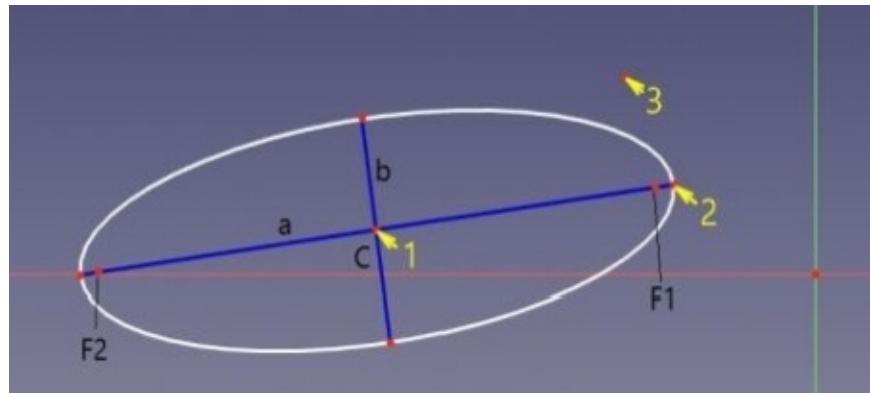


5. Select the  **Circle tool** by centre & rim point.
6. Or select the  **Circle tool** by 3 rim points.
7. Click a first point or centre point on 3D view.
8. Click a second point or set radius on 3D view.
9. Then press the **Esc** button or click on **Close** tap from left side of the combo view.

7.5 HOW TO DRAW CONIC SECTIONS (Ellipse)

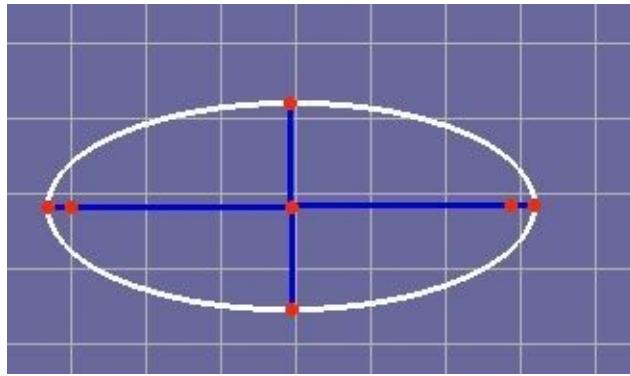
This tool is use to create a three types of conic section on sketcher workbench environment. First one is Ellipse by centre, second one is Ellipse by 3 points and third one is arc of Ellipse.

1. **ELLIPSE BY CENTRE, MAJOR RADIUS & MINOR RADIUS POINTS :-**



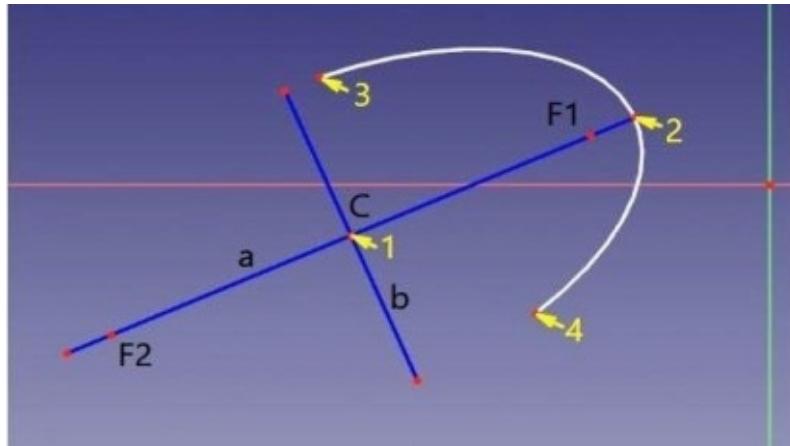
- OPERATION

1. Run or open the FreeCAD software.
 2. Switch the Workbench and select the **Sketcher**.
 3. Click on new sketch tool.
 4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.
 5. Select the **Ellipse tool**.
 6. Click a first point or centre point on 3D view.
 7. Click a second point or major radius point on 3D view.
 8. Click a third point or minor radius point on 3D view.
 9. Then press the **Esc** button or click on **Close** tap from left side of the combo view.
-
2. **ELLIPSE BY PERIAPSIS, APOAPSIS & MINOR RADIUS POINTS:** -



- OPERATION

1. Run or open the FreeCAD software.
 2. Switch the Workbench and select the **Sketcher**.
 3. Click on  new sketch tool.
 4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.
 5. Select the  **Ellipse tool**.
 6. Click a first point or periapsis point on 3D view.
 7. Click a second point or apoapsis point on 3D view.
 8. Click a third point or minor radius point on 3D view.
 9. Then press the **Esc** button or click on **Close** tap from left side of the combo view.
3. **ELLIPSE BY CENTRE, MAJOR RADIUS STARTING & END POINTS:** -



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on  new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.
5. Select the  **Ellipse tool**.
6. Click a first point or ellipse centre point on 3D view.
7. Click a second point or major radius point on 3D view. This makes orientation of the ellipse.
8. Click a third point or start of the arc point on 3D view.
9. Click a fourth point oe end of the arc point on 3D view.
10. Then press the **Esc** button or click on **Close** tap from left side of the combo view.

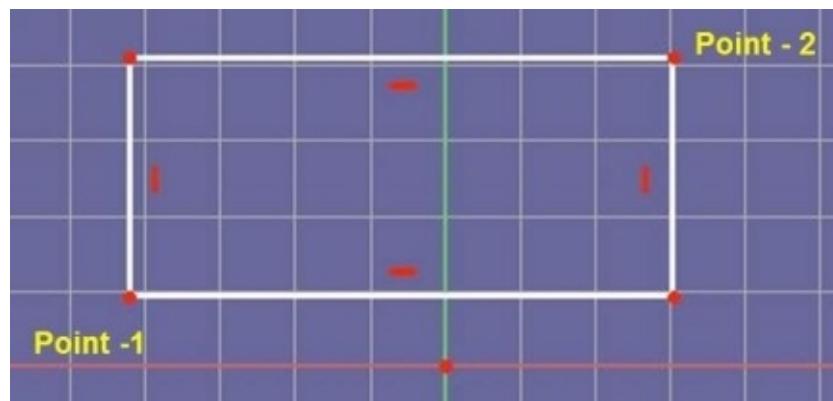
7.6 HOW TO DRAW POLYLINE



- OPERATION

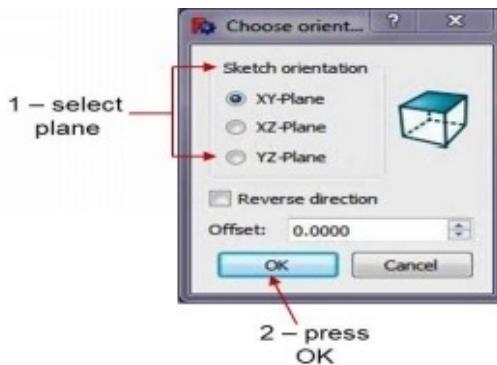
1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on  new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.
5. Select the  **Polyline tool**.
6. Click a first point, second point , third point, fourth and so, on sketcher environment.
7. Then press the **Esc** button or click on **Close** tap from left side of the combo view.

7.7 HOW TO DRAW RECTANGLE



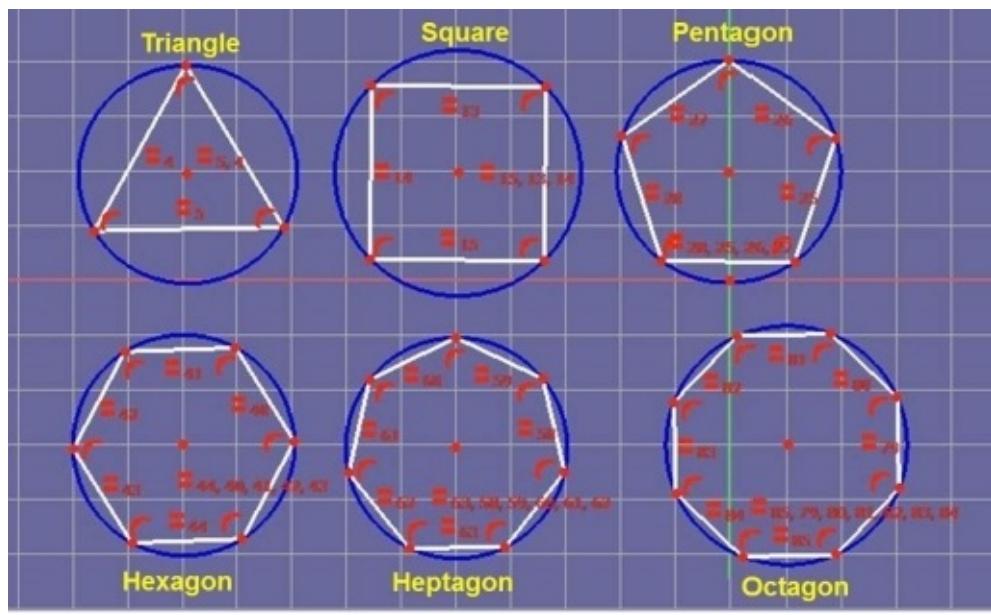
- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on  new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.



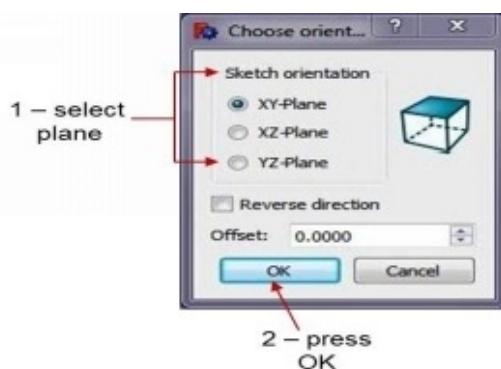
5. Select the  **Rectangle tool**.
6. Click a first point on sketcher environment.
7. Then move the mouse and click a second point on 3D view.
8. Then press the **Esc** button or click on **Close** tap from left side of the combo view.

7.8 HOW TO DRAW REGULAR POLYGON



• OPERATION

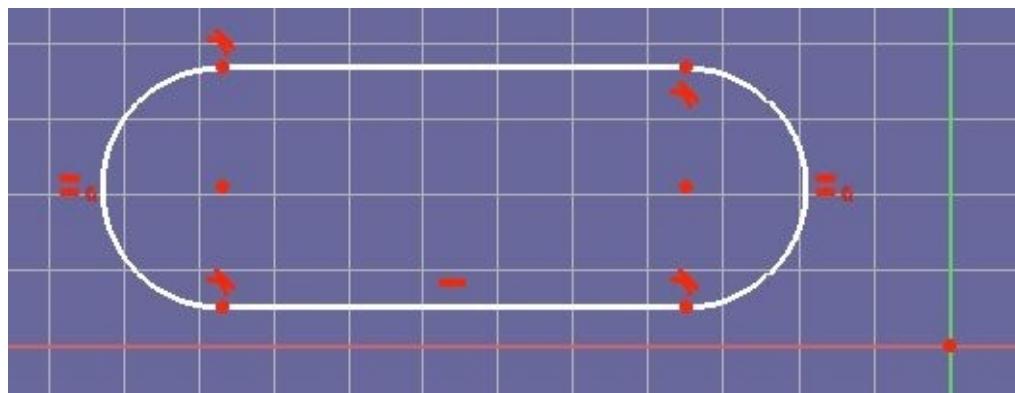
1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.



5. Select the **Polygon tool**.
6. Click a first point on sketcher environment.
7. Then move the mouse and click a second point on 3D view.

8. Then press the **Esc** button or click on **Close** tap from left side of the combo view.

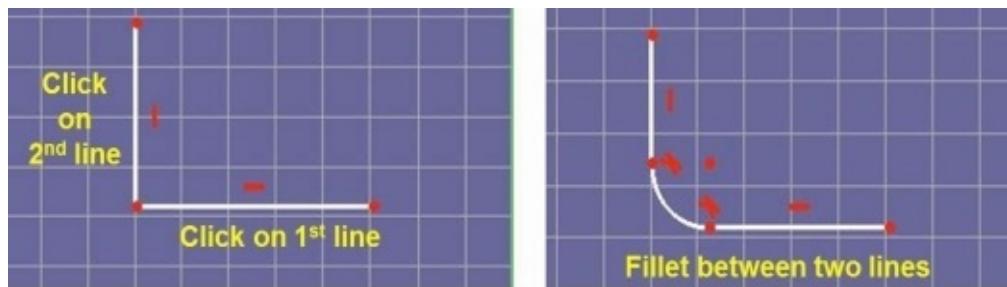
7.9 HOW TO DRAW SLOT



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on  new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.
5. Select the  **Slot tool**.
6. Click a first point on sketcher environment.
7. Then move the mouse and click a second point on 3D view.
8. Then press the **Esc** button or click on **Close** tap from left side of the combo view.

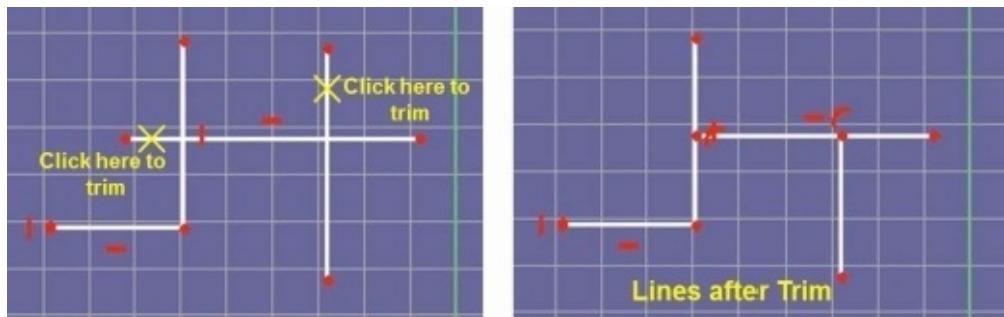
7.10 HOW TO MAKES FILLET



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on  new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.
5. Draw a two lines as you want or follow the above figure.
6. Select the  **Fillet tool**.
7. Click at first line on sketcher environment.
8. Click at second line on sketcher environment to make fillet.
9. Then press the **Esc** button or click on **Close** tap from left side of the combo view.

7.11 HOW TO MAKES TRIMMING

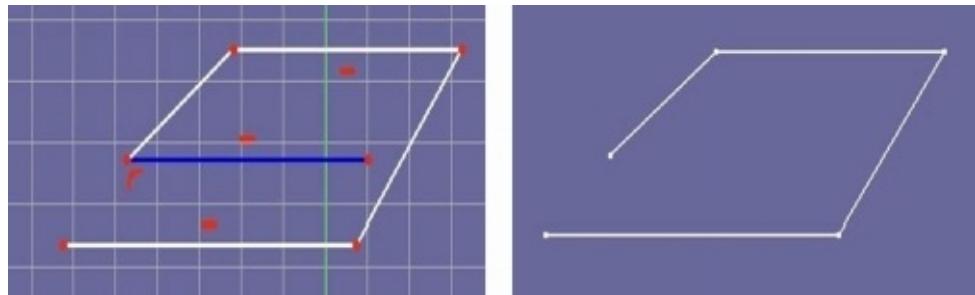


- OPERATION

1. Run or open the FreeCAD software.
 2. Switch the Workbench and select the **Sketcher**.
 3. Click on  new sketch tool.
 4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.
 5. Draw a lines as you want or follow the above figure.
- 
6. Select the **Trimming tool**.
 7. Then click on the line that you want to make a trim on sketcher environment.
 8. Then press the **Esc** button or click on **Close** tap from left side of the combo view.

7.12 HOW TO MAKE EXTERNAL GEOMETRY

This tool is basically used in Part design workbench. The operation of this tool is explained in next chapter(see the chapter No. – 11) 7.13 HOW TO USE CONSTRUCTION MODE TOOL



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.
5. Draw a lines as you want or follow the above figure.
6. Then click on the line that you want to make a construction mode on sketcher environment.
7. Select the **Construction mode tool**.
8. Then press the **Esc** button or click on **Close** tap from left side of the combo view.
9. once your finished sketch, sketch and leaving the forms used in

construction mode, have become invisible to the screen.

WORKING WITH SKETCHER CONSTRAINTS

1. HOW TO MAKE COINCIDENT CONSTRAINT

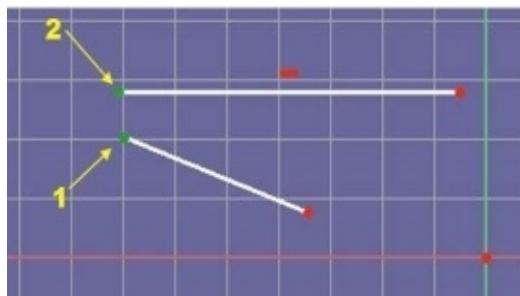


Fig. - 1

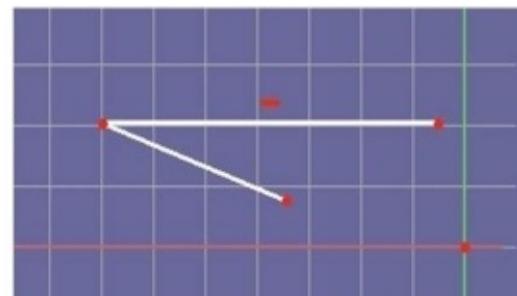


Fig. - 2

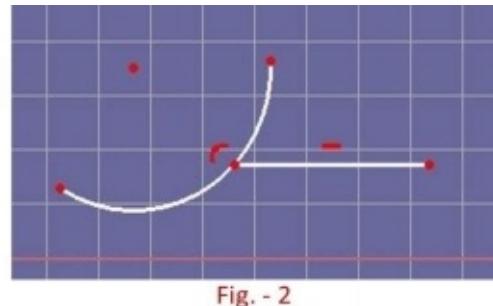
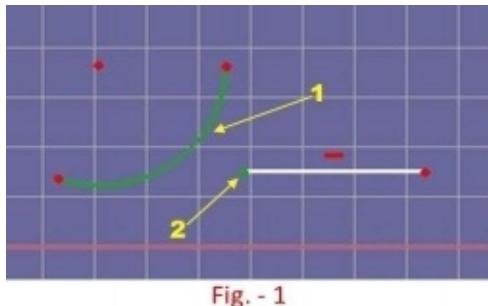
• OPERATION

1. First of all, select the one point of line to make the highlight of point (see Fig. – 1).
2. Again select the second point of line to make the highlight of point (see Fig. – 1).
3. A highlighted item will change colour to green.
4. Subsequent items can be highlighted by repeating the above

procedure(s) NOTE: There is no-need to hold-down any special key like Ctrl to achieve multiple item selection in a drawing.

5. Then click on the  **Coincident constraint tool**.
6. The two points to become coincident and be replaced by a single point(see Fig. – 2).
7. Then press the **Esc** button or click on **Close** tap from left side of the combo view.
8. If you again want to edit in sketch, make a **Double Click** on Sketch form Combo view.

2. HOW TO MAKE POINT ON OBJECT CONSTRAINT

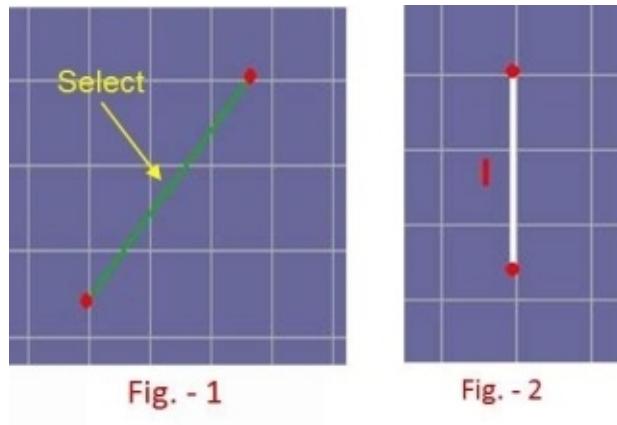


• OPERATION

1. First of all, select the one object (arc or etc) to make the highlight (see Fig. – 1).
2. Again select the point of line or point of object to make the highlight of point.

3. A highlighted item will change colour to green.
4. Then click on the  **Point on object constraint tool**.
5. The two objects to become point on object (see Fig. – 2).
6. Then press the **Esc** button or **click** on Close tap from left side of the combo view.
7. If you again want to edit in sketch, make a **Double Click** on Sketch form Combo view.

3. **HOW TO MAKE VERTICAL CONSTRAINT**

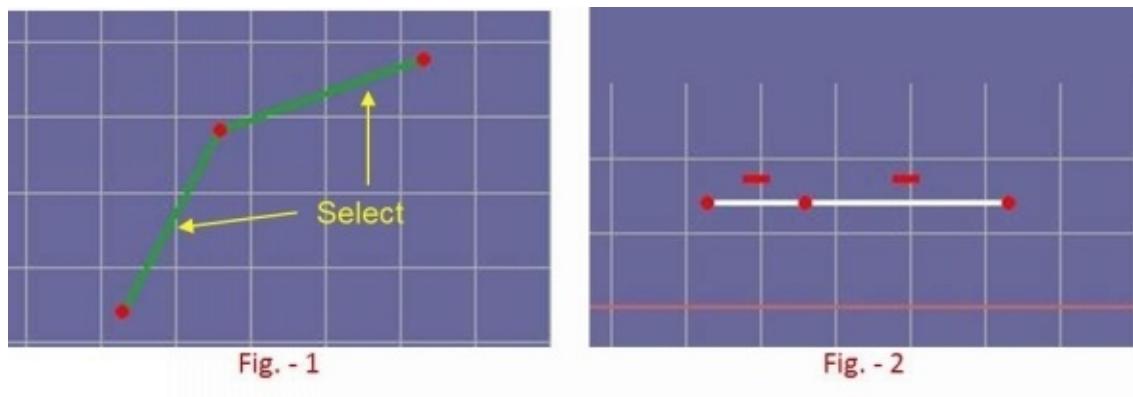


• **OPERATION**

1. First of all, select the line or polyline to make the highlight (see Fig. – 1).
2. In this, more than one object can be selected.
3. A highlighted item will change colour to green.

4. Then click on the  **Vertical constraint tool**.
5. A line to become vertical line (see Fig. – 2).
6. Then press the Esc button or click on Close tap from left side of the combo view.
7. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

4. HOW TO MAKE HORIZONTAL CONSTRAINT

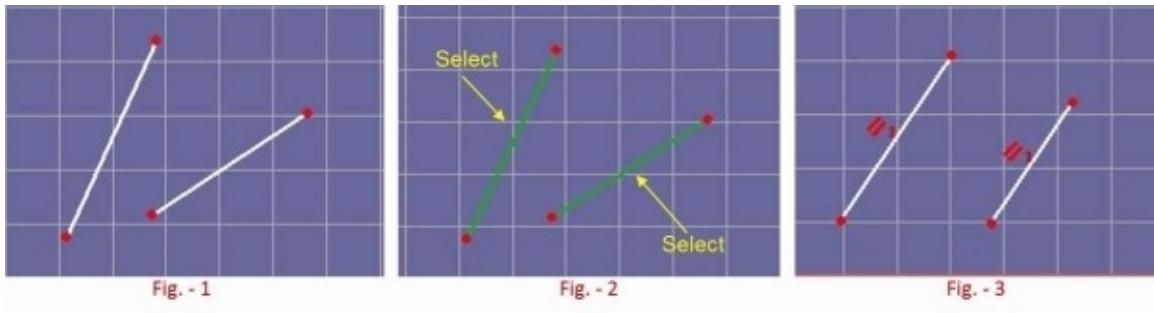


• OPERATION

1. First of all, select the line or polyline to make the highlight (see Fig. – 1).
2. In this, more than one object can be selected.
3. A highlighted item will change colour to green.
4. Then click on the  **Horizontal constraint tool**.

5. A lines to become horizontal line (see Fig. – 2).
6. Then press the Esc button or click on Close tap from left side of the combo view.
7. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

5. HOW TO MAKE PARALLEL CONSTRAINT



• OPERATION

1. First of all, draw the two lines (see Fig. – 1).
2. select the lines to make the highlight (see Fig. – 2).
3. A highlighted item will change colour to green.
4. Then click on the  **Parallel constraint tool**.
5. A lines to become parallel line (see Fig. – 3).
6. Then press the Esc button or click on Close tap from left side of the combo view.
7. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

6. HOW TO MAKE PERPENDICULAR CONSTRAINT

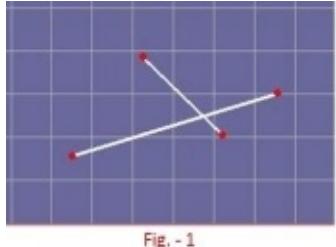


Fig. - 1

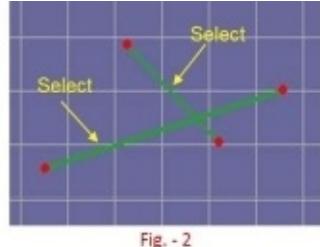


Fig. - 2

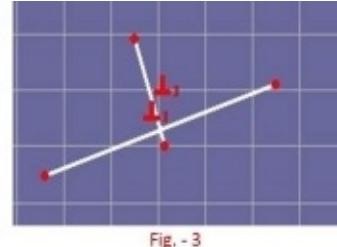


Fig. - 3

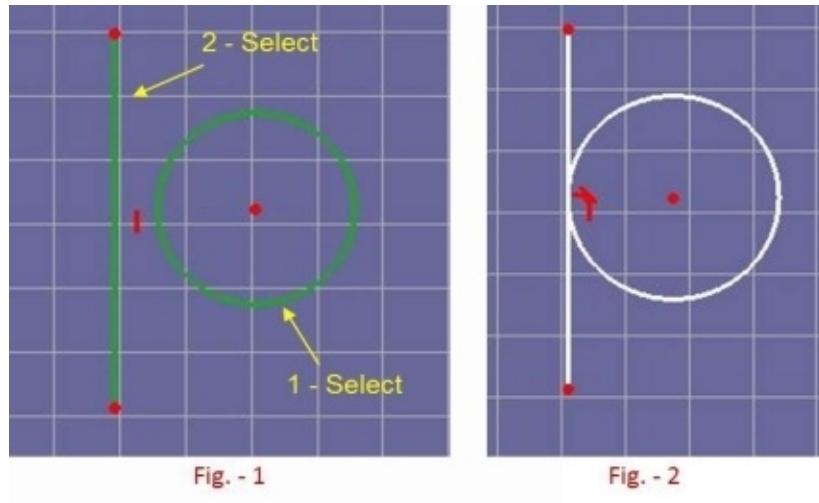
- OPERATION

1. First of all, draw the two lines (see Fig. – 1).
2. select the lines to make the highlight (see Fig. – 2).
3. A highlighted item will change colour to green.
4. Then click on the  **Perpendicular constraint tool**.
5. A lines to become parallel line (see Fig. – 3).
6. Then press the Esc button or click on Close tap from left side of the combo view.
7. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

Note: There are four different ways the perpendicular constraint can be applied: a) between two curves (available not for all curves).

- b) between two endpoints of a curve.
- c) between a curve and an endpoint of another curve.
- d) between two curves at user-defined point.

7. HOW TO MAKE TANGENT CONSTRAINT



- OPERATION

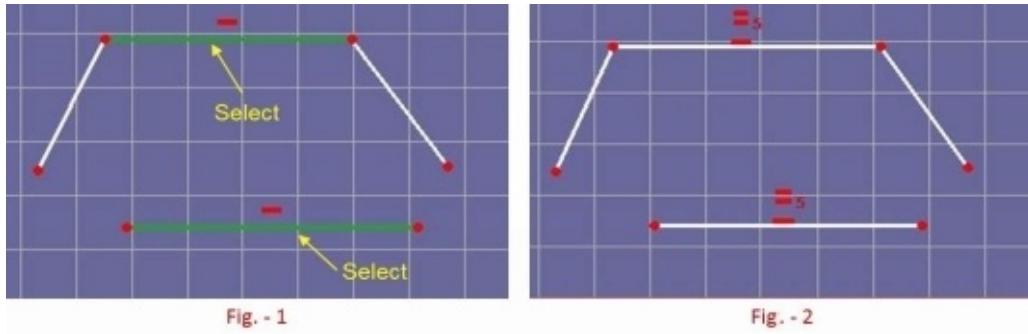
1. First of all, draw the two objects (see Fig. – 1).
2. select the objects to make the highlight.
3. A highlighted item will change colour to green.
4. Then click on the  **Tangent constraint tool**.
5. The line become tangent with circle (see Fig. – 2).
6. Then press the Esc button or click on Close tap from left side of the combo view.
7. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

Note: There are four different ways the perpendicular constraint can be applied: a) between two curves (available not for all curves).

- b) between two endpoints of a curve.
- c) between a curve and an endpoint of another curve.

d) between two curves at user-defined point.

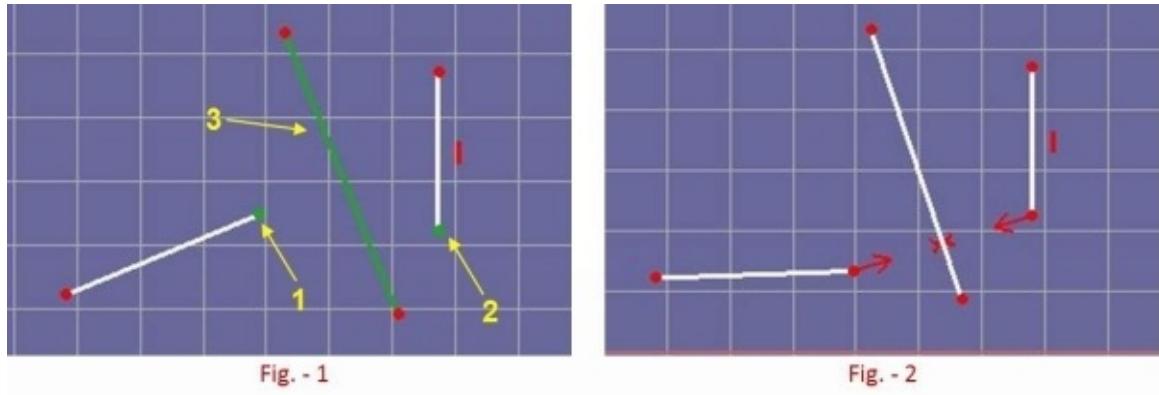
8. HOW TO MAKE EQUAL LENGTH CONSTRAINT



• OPERATION

1. First of all, draw the two objects (line, poly-line, rectangle, arc and circle. see Fig. – 1).
2. select the objects to make the highlight.
3. A highlighted item will change colour to green.
4. Then click on the  **Equal length constraint tool**.
5. The object become equal in length (see Fig. – 2).
6. Then press the Esc button or click on Close tap from left side of the combo view.

9. HOW TO MAKE SYMMETRIC CONSTRAINT



- OPERATION

1. Select the two points(vertices) in the sketch to make the highlight (see Fig. – 1).
2. Then select a line to make the highlight (see Fig. – 1).
3. A highlighted item will change colour to green.
4. Then click on the  **Symmetric constraint tool**.
5. Then it became symmetric (see Fig. – 2).
6. Then press the Esc button or click on Close tap from left side of the combo view.
7. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.
10. **HOW TO MAKE SYMMETRIC CONSTRAINT**

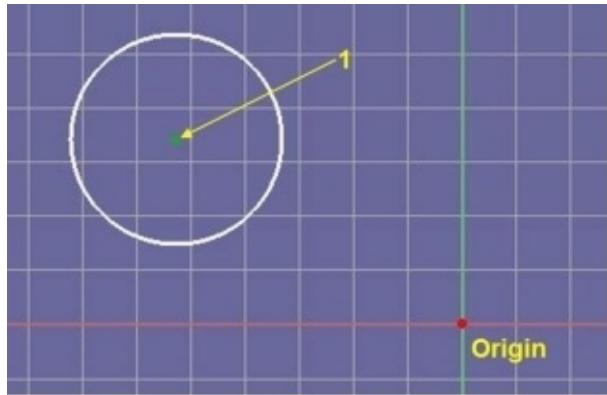


Fig. - 1

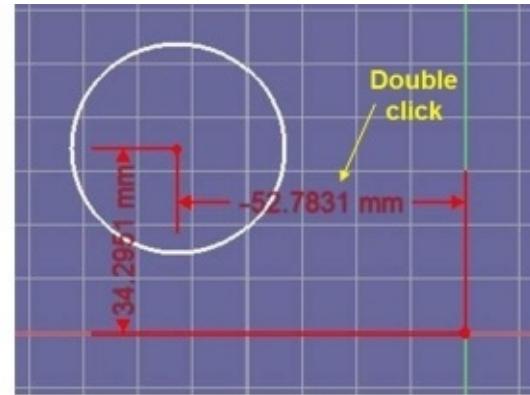


Fig. - 2

- OPERATION

1. Select the point of the object in the sketch to make the highlight (see Fig. – 1).
2. A highlighted item will change colour to green.
3. Then click on the **Lock constraint tool**.
4. Then usually it manifests as two constraints: a horizontal distance constraint from the drawing axis origin, and a vertical constraint from the drawing axis origin (see Fig. – 2).
5. The vertical and horizontal constraints forming the lock can be edited by double clicking on the appropriate constraint (see Fig. – 2) to be edited either in the drawing itself or in the Constraint tab of the Combo View pane.
6. Then press the Esc button or click on Close tap from left side of the combo view.
7. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

11. HOW TO SET HORIZONTAL DISTANCE

CONSTRAINT

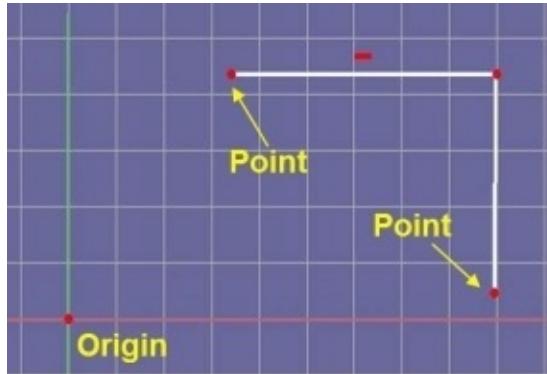


Fig. - 1

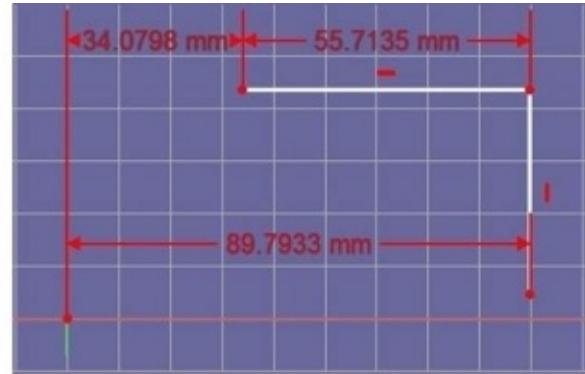


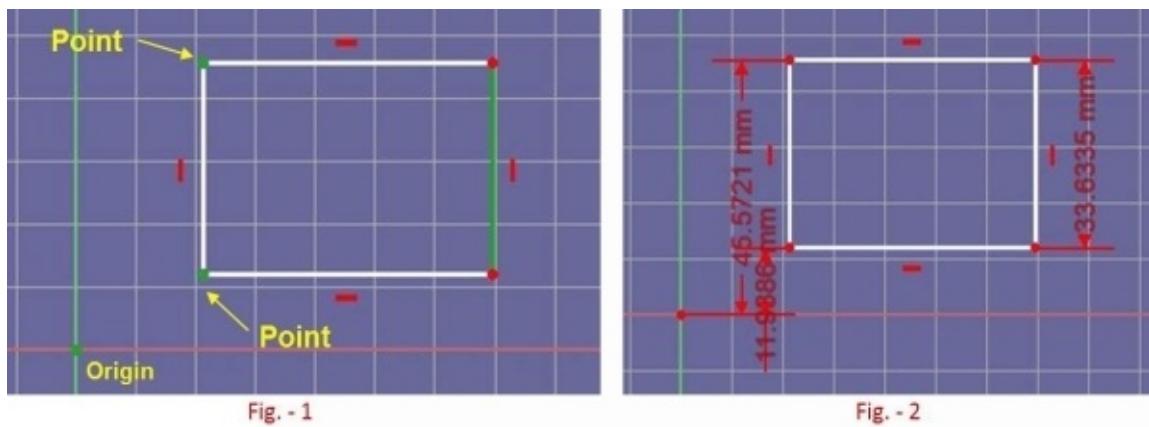
Fig. - 2

• OPERATION

1. Select the two points(vertices) you wish to set a horizontal distance (see Fig. – 1).
2. You can also select the line.
3. A selected items becomes highlighted and it will change colour to green.
4. Then click on the  **Horizontal distance constraint tool**.
5. Then horizontal distance constraint will appear (see Fig. – 2).
6. you can also edit the distance value by double clicking on the appropriate constraint.
7. Then press the Esc button or click on Close tap from left side of the combo view.

- If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

12. HOW TO SET VERTICAL DISTANCE CONSTRAINT



- OPERATION

- Select the two points(vertices) you wish to set a horizontal distance (see Fig. – 1).
- You can also select the line.
- A selected items becomes highlighted and it will change colour to green.
- Then click on the  **Vertical distance constraint tool**.
- Then vertical distance constraint will appear (see Fig. – 2).
- you can also edit the distance value by double clicking on the

appropriate constraint.

7. Then press the Esc button or click on Close tap from left side of the combo view.
8. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

13. HOW TO SET LENGTH CONSTRAINT

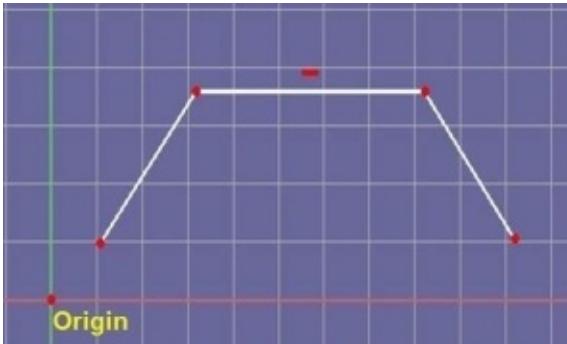


Fig. - 1

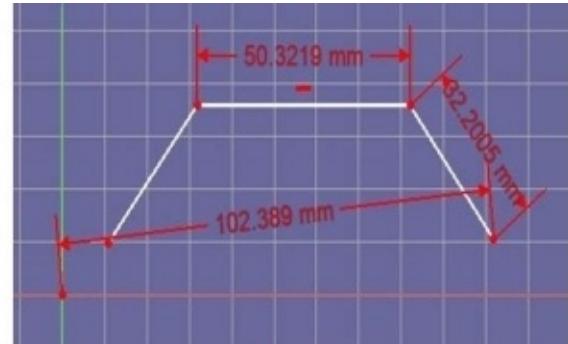
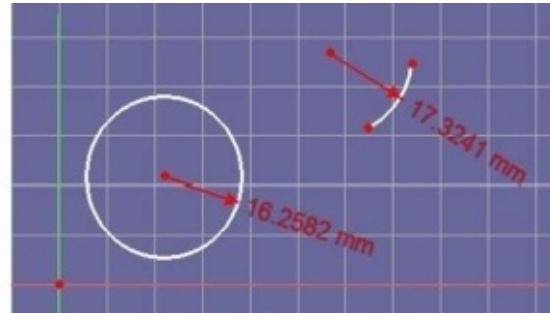
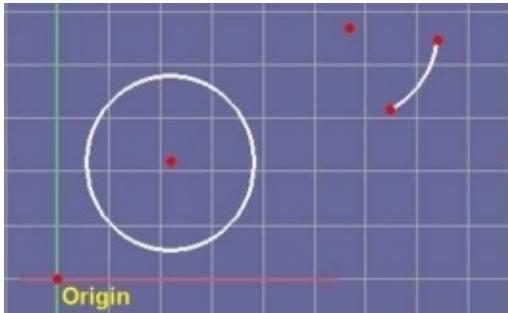


Fig. - 2

• OPERATION

1. Select the two points (vertices) you wish to set a length (see Fig. – 1).
2. You can also select the line.
3. A selected items becomes highlighted and it will change colour to green.
4. Then click on the  **Length constraint tool**.
5. Then length constraint will appear (see Fig. – 2).
6. you can also edit the length value by double clicking on the appropriate constraint.
7. Then press the Esc button or click on Close tap from left side of the combo view.
8. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

14. HOW TO MAKE RADIUS CONSTRAINT



• OPERATION

1. Select the Circle or Arc you wish to make radius constraint (see Fig. – 1).
2. A selected items becomes highlighted and it will change colour to green.
3. Then click on the
- Radius constraint tool.**
4. Then radius constraint will appear (see Fig. – 2).
5. you can also edit the radius value by double clicking on the appropriate constraint.
6. Then press the Esc button or click on Close tap from left side of the combo view.
7. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

15. HOW TO MAKE INTERNAL ANGLE

CONSTRAINT

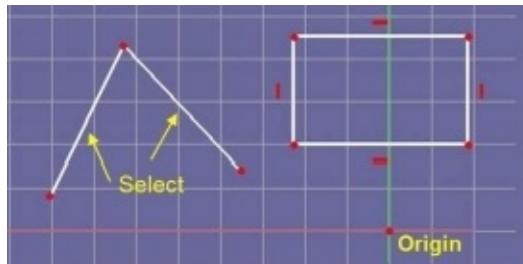


Fig. - 1

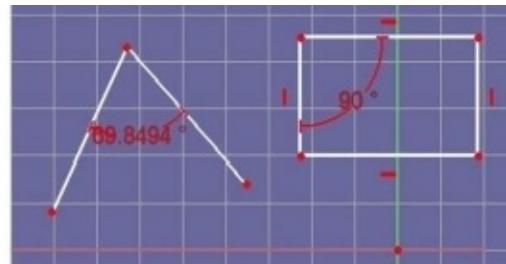


Fig. - 2

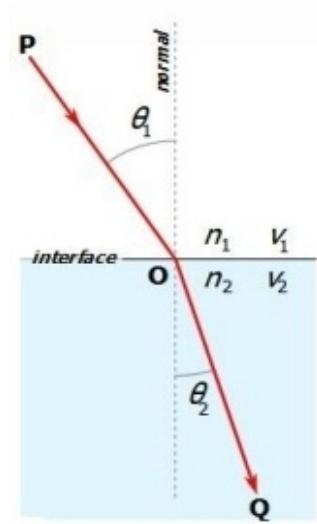
There are four different ways the constraint can be applied:
a) to individual lines
b) between lines c) to intersections of curves d) to arcs of circles

- OPERATION

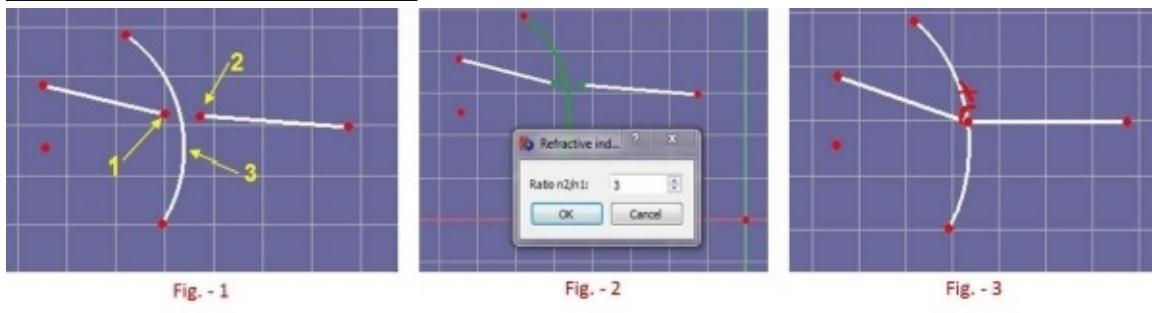
1. Select one, two or three entities in the sketch wish to make radius constraint
(see Fig. – 1).
2. A selected items becomes highlighted and it will change colour to green.
3. Then click on the  **Internal angle constraint tool**.
4. Then internal angle constraint will appear (see Fig. – 2).
5. you can also edit the angle value by double clicking on the appropriate constraint.
6. Then press the Esc button or click on Close tap from left side of the combo view.
7. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

16. HOW TO MAKE SNELL'S LAW CONSTRAINT

Constrains two lines to follow the law of refraction of light as it penetrates through an interface, where two materials of different refraction indices meet.



USE OF SNELL'S LAW :



• OPERATION

1. Select endpoint of one line, an endpoint of second line and a curve to act as a interface,
(see Fig. – 1).

2. A selected items becomes highlighted and it will change colour to green.
3. Then click on the  **Snell's law constraint tool.**
4. A dialog will appear asking for a ratio of indices of refraction n_2/n_1 (see Fig. – 2).
5. In this, n_2 corresponds to the medium where the second selected endpoint's line resides, n_1 is for the first line.

6. Then Snell's law constraint will appear (see Fig. – 3).
7. Then press the Esc button or click on Close tap from left side of the combo view.
8. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

WORKING WITH OTHER SKETCHER TOOLS

1. [HOW TO MAKE CLOSE SHAPE](#)

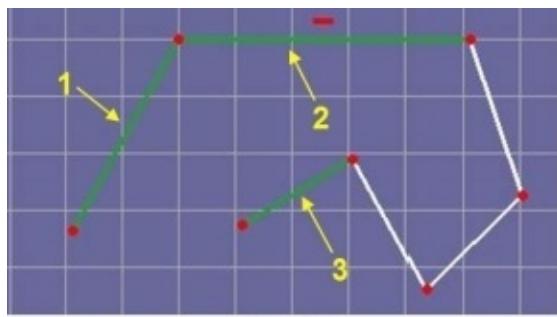


Fig. - 1

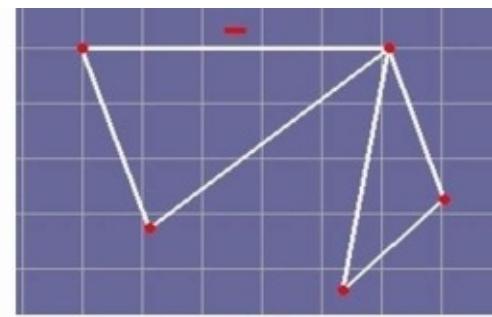


Fig. - 2

- **OPERATION**

1. Select more than two edges of an element you wish to make a close shape
(see Fig. – 1).
2. A selected items becomes highlighted and it will change colour to green.
3. Then click on the  **Close shape constraint tool**.
4. Then close shape constraint will appear (see Fig. – 2).
5. Then press the Esc button or click on Close tap from left side of the combo view.
6. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

2. **HOW TO MAKE CONNECT EDGES**

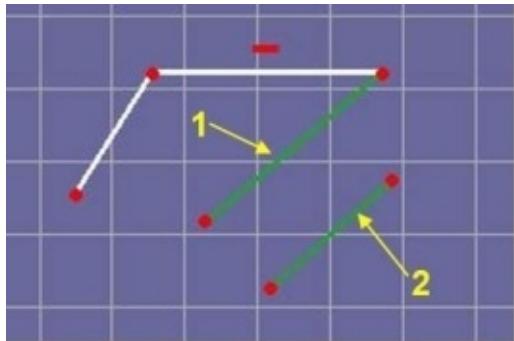


Fig. - 1

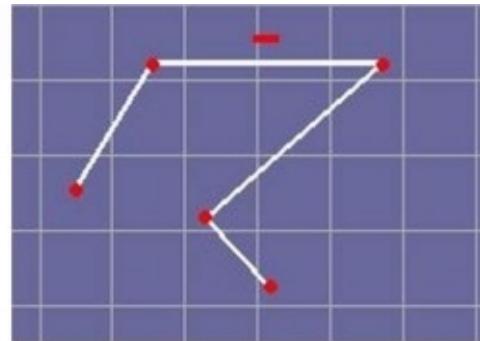


Fig. - 2

- OPERATION

1. Select two edges of an element you wish to make a connect edges (see Fig. – 1).
 2. A selected items becomes highlighted and it will change colour to green.
 3. Then click on the  **Connect edges constraint tool**.
 4. Then connect edges constraint will appear (see Fig. – 2).
 5. Then press the Esc button or click on Close tap from left side of the combo view.
 6. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.
3. **HOW TO MAKE SHOW/HIDE GEOMETRY**

This too is currently working with only Ellipse or with Arc of ellipse.

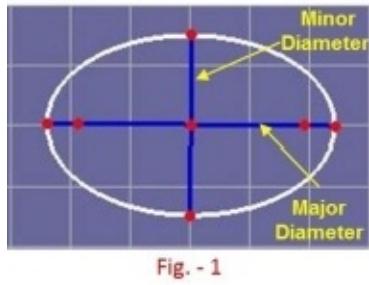


Fig. - 1

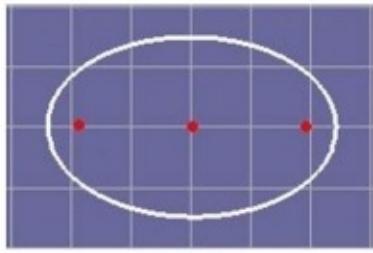


Fig. - 2

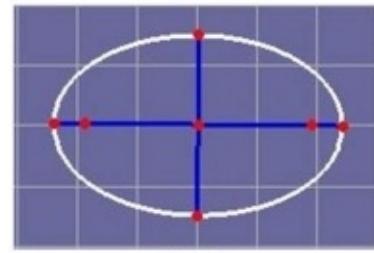


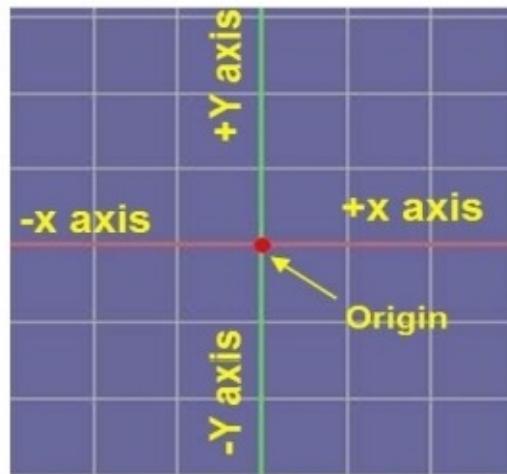
Fig. -3

- OPERATION

1. Create a new ellipse.
2. Select minor or major diameter or both and hit the Delete button (see Fig. – 1).
3. A selected items becomes highlighted and it will change colour to green.
4. Then the diameter is gone, but the ellipse remains (see Fig. – 2).
5. Click on the  **Show/Hide geometry constraint tool**.
6. The diameter is restored (see Fig. – 3).
7. Then press the Esc button or click on Close tap from left side of the combo view.
8. If you again want to edit in sketch, make a Double Click on Sketch form Combo view.

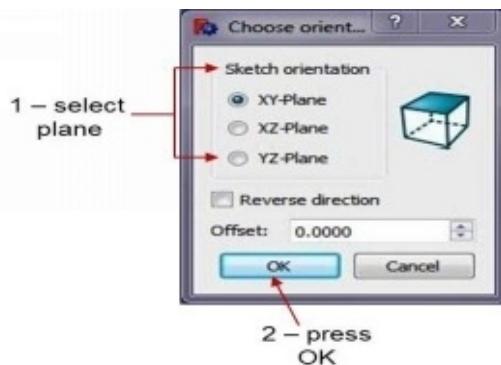
4. HOW TO SET NEW SKETCH

This will create a new sketch in the working environment .



- OPERATION

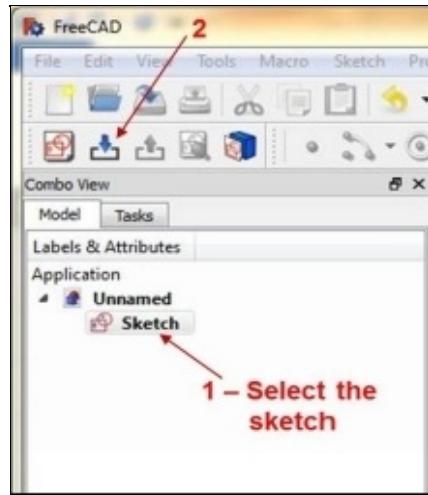
- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the **Sketcher**.
- c) Click on new sketch tool.
- d) Then select the Plane(XY, XZ, YZ) from Choose orientation window.



- e) You can change an offset to any of the three planes and the side of the offset.

5. HOW TO USE EDIT SKETCH

This command is used to edit the selected Sketch.



- How to Use

1. Run or open the FreeCAD software.
 2. Switch the Workbench and select the **Sketcher**.
 3. Click on  new sketch tool.
 4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.
 5. Make a some sketch (line, polyline, arc etc.).
 6. Then press the **Esc** button or click on **Close** tap from left side of the combo view.
 7. Now select the **Sketch** from combo view (see the above figure).
 8. Then click on  **Edit sketch command**.
 9. You will again come back on the sketcher environment or edit mode environment .
6. **HOW TO USE LEAVE SKETCH**

This command is used to leave the Sketch editing mode.



- How to Use

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the **Sketcher**.
3. Click on new sketch tool.
4. Then select the Plane(XY, XZ, YZ) from Choose orientation window.
5. Make a some sketch (line, polyline, arc etc.).
6. Then press the Esc button or click on **Close** tap from left side of the combo view.
7. Now select the **Sketch** from combo view (see the above figure).
8. Then click on **Edit sketch command**.
9. You will again come back on the sketcher environment or edit mode

environment .

10. Now click on  **Leave sketch command.**

11. You will again come back on the non sketcher environment.

7. HOW TO USE VIEW SKETCH

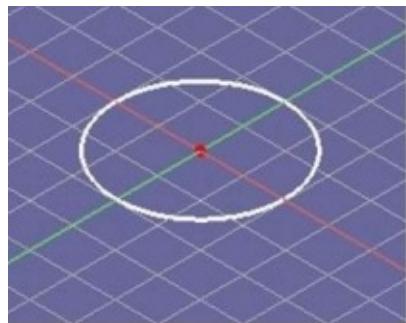


Fig. - 1

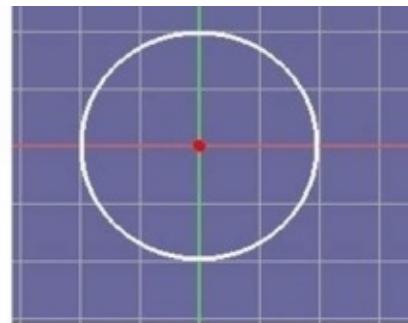


Fig. - 2

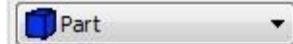
This command is useful when the user has changed the model view orientation to examine another aspect of the model and wants to return to a view normal to the sketch.

8. HOW TO USE MAP SKETCH TO FACE COMMAND

The use of this command will discuss in chapter number - 11 (11. WORKING WITH - PART DESIGN).

END

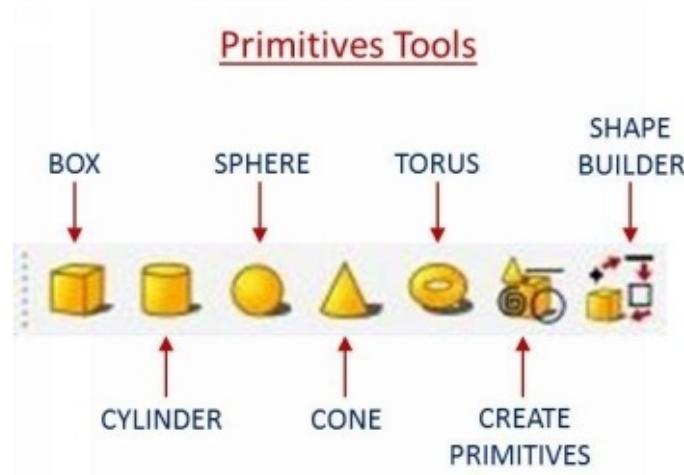
8. PART WORKBENCH



The part workbench allows to quickly drawing simple 3D objects in the current document. It includes basic tools for creating and modifying the objects.

8.1 PRIMITIVES TOOLS

Primitives tool are used for creating primitive objects.



The descriptions of Primitives Tools are listed below:-

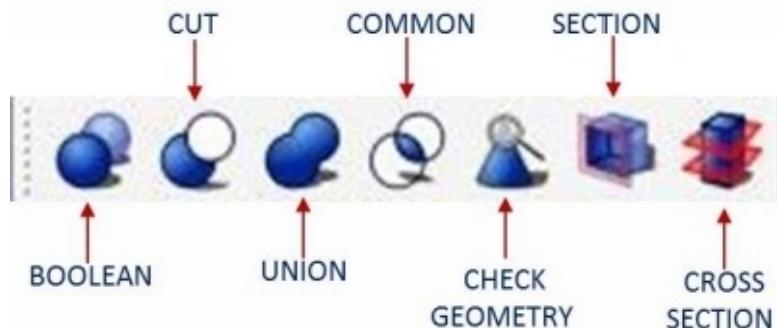
- a) BOX : - Draws a box by specifying its dimensions.
- b) CYLINDER : - Draws a cylinder by specifying its dimensions.
- c) SPHERE : - Draws a sphere by specifying its dimensions.
- d) CONE : - Draws a cone by specifying its dimensions.
- e) TORUS : - Draws a torus (ring) by specifying its dimensions.

- f)  [CREATE PRIMITIVES](#): - A tool to create various parametric geometric primitives.
- g)  [SHAPE BUILDER](#): - A tool to create more complex shapes from various parametric

8.2 [MODIFYING OBJECT TOOLS - 1](#)

These are tools for modifying existing objects. They will allow you to choose which object to modify.

[Modifying Object Tools -1](#)



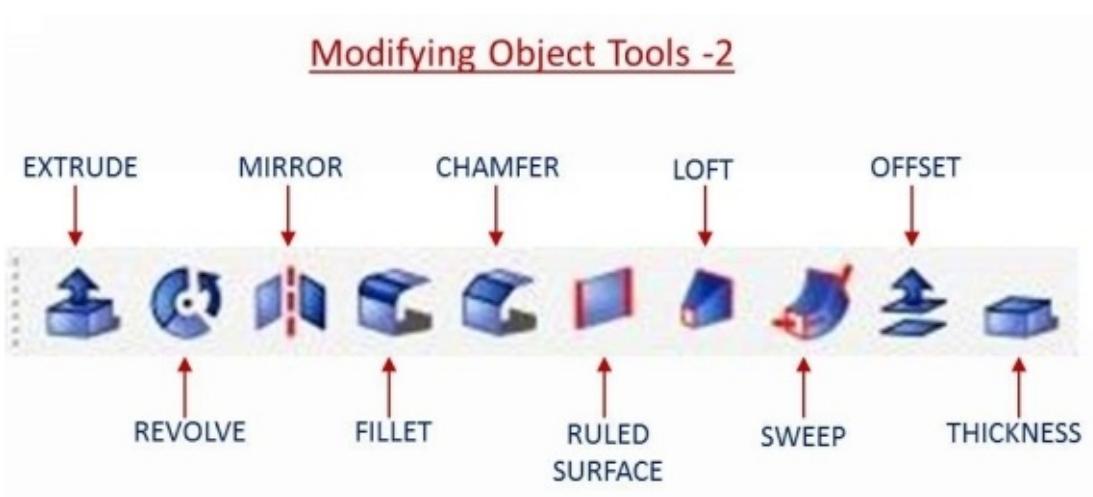
[The descriptions of Modifying Object Tools - 1](#) are listed below:-

-  [BOOLEAN](#): - Performs Boolean operations on objects.
-  [CUT](#): - Cuts (subtracts) one object from another.
-  [UNION](#): - Fuses (unions) two objects.
-  [COMMON](#): - Extracts the common (intersection) part of two objects
-  [CHECK GEOMETRY](#): - The check geometry tool allows you to verify if you have a valid solid.

- f)  **SECTION**: - Creates a section by intersecting an object with a section plane g)  **CROSS SECTION**: - Creates a cross section by intersecting an object with a section plane.

8.3 MODIFYING OBJECT TOOLS - 2

These are also tools for modifying existing objects. They will allow you to choose which object to modify.



The descriptions of **Modifying Object Tools - 1** are listed below:-

- a)  EXTRUDE : - Extrude planar faces of an object.
- b)  REVOLVE : - Creates a solid by revolving another object (not solid) around an axis.
- c)  Mirror : - Mirrors the selected object on a given mirror plane.
- d)  FILLET : - Fillets (rounds) edges of an object.
- e)  CHAMFER : - Chamfers edges of an object
- f)  RULED SURFACE : - Ruled the surface of an object.
- g)  LOFT : - Lofts from one profile to another.

- h)  [**SWEET**](#) : - Sweeps one or more profiles along a path.
- i)  [**OFFSET**](#) : - Creates a scaled copy of the original object.
- j)  [**THICKNESS**](#) : - Assign a thickness to the faces of a shape.

8.4 OTHER TOOLS

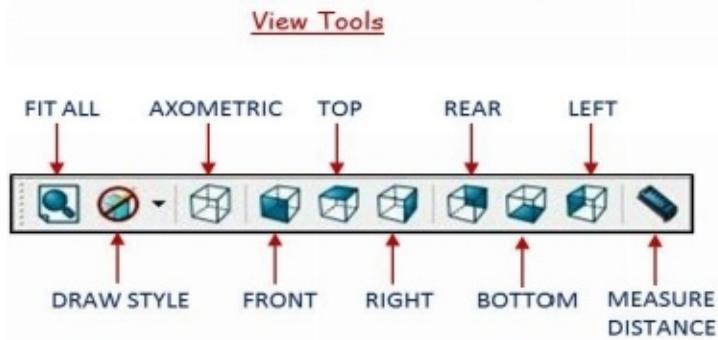


The descriptions of **Other Tools** are listed below:-

- a)  [**MEASURE LINER**](#) : - Allows you to make linear measurements.
- b)  [**MEASURE ANGULAR**](#) : - Allows you to make angular measurements.
- c)  [**CLEAR ALL**](#) : - Deletes all measures.
- d)  [**TOGGLE ALL**](#) : - Shows or hides all measures .
- e)  [**TOGGLE 3D**](#) : - Shows or hides 3D measurements.
- f)  [**TOGGLE DELTA**](#) : - Shows or hides Delta measurements.

6.5 VIEW TOOLS

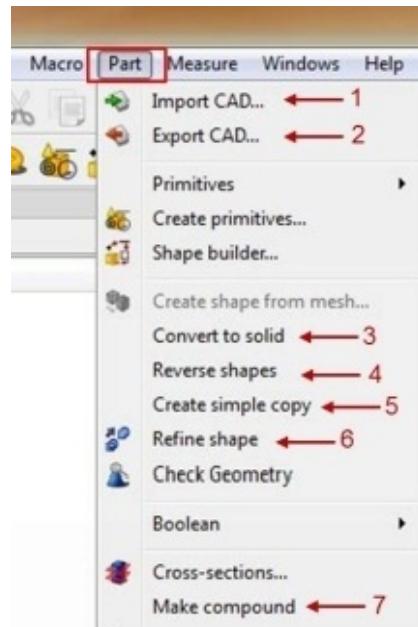
This tools are very usefull for showing the object in different location. This tools are always avilable in all workbenches.



NOTE: -

The description of this command is already discussed in **chapter Number – 4**.

6.6 SOME EXTRA TOOLS



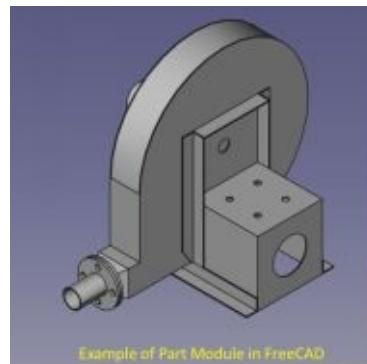
These tools are in development phase except tool number 3& 4 (see above fig.). Above listed tools are not much use in part workbench.

END

9. OPERATION WITH – PART



The part workbench allows to quickly drawing simple 3D objects in the current document. It includes basic tools for creating and modifying the objects. The Part module allows FreeCAD to access and use the Open CasCade objects and functions. OpenCascade is a professional-level CAD kernel, that features advanced 3D geometry manipulation and objects.

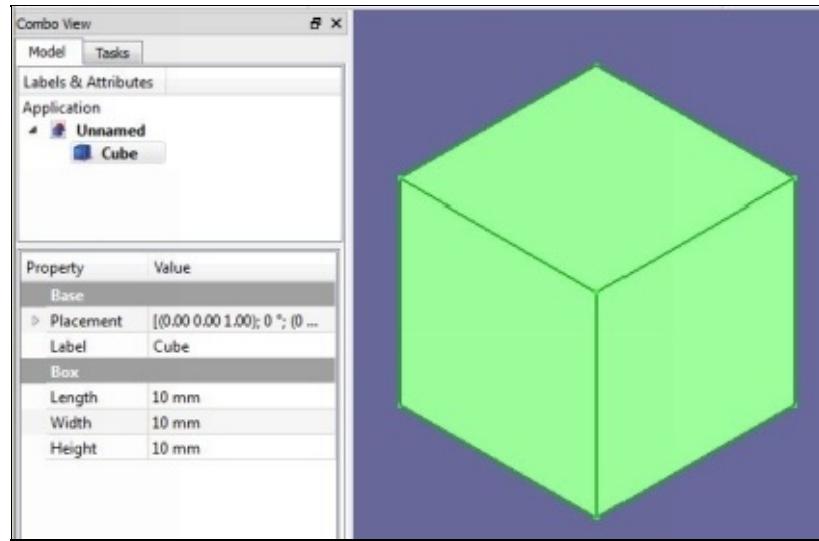


Example of Part Module in FreeCAD

WORKING WITH PART PRIMITIVES TOOLS

9.1 HOW TO MAKE SOLID CUBE

This tool is use to create a default box or solid cube in 3D environment.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Part.
3. Then, select the **Cube tool**.
4. A box or cube with standard dimension and position will be created.

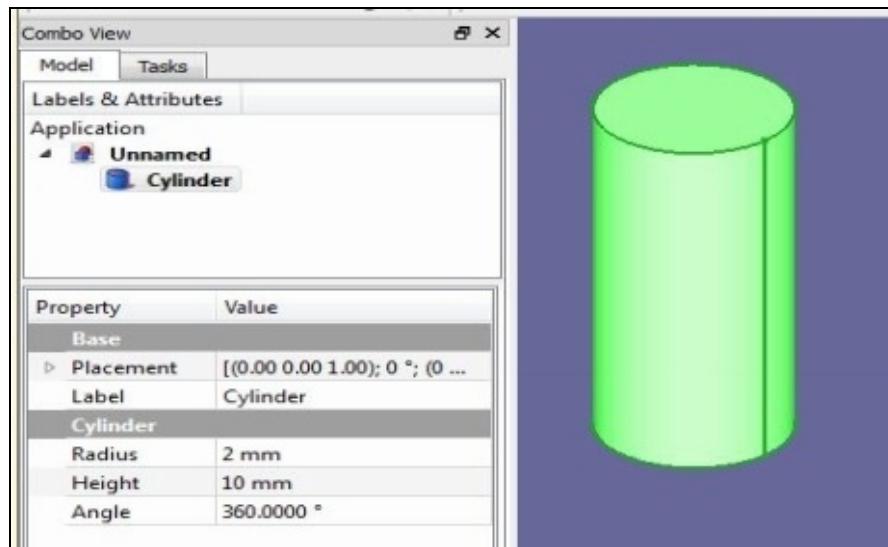
- PROPERTIES

:- The properties value tab are used for modification of Cube. Select the Cube from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement of cube: - **Property** → **Data** → **Placement**.
2. For length: - **Property** → **Data** → **Length** value.
3. For width: - **Property** → **Data** → **Width** value.
4. For height: - **Property** → **Data** → **Height** value.
5. For box Shape color, Draw style, Transparency etc: - **Property** → **View**.

9.2 HOW TO MAKE CYLINDER

This tool is used to create a simple cylinder with position, radius height and angle in 3D environment.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Part.
3. Then, select the  **Cylinder tool**.
4. A cylinder with standard dimension and position will be created.

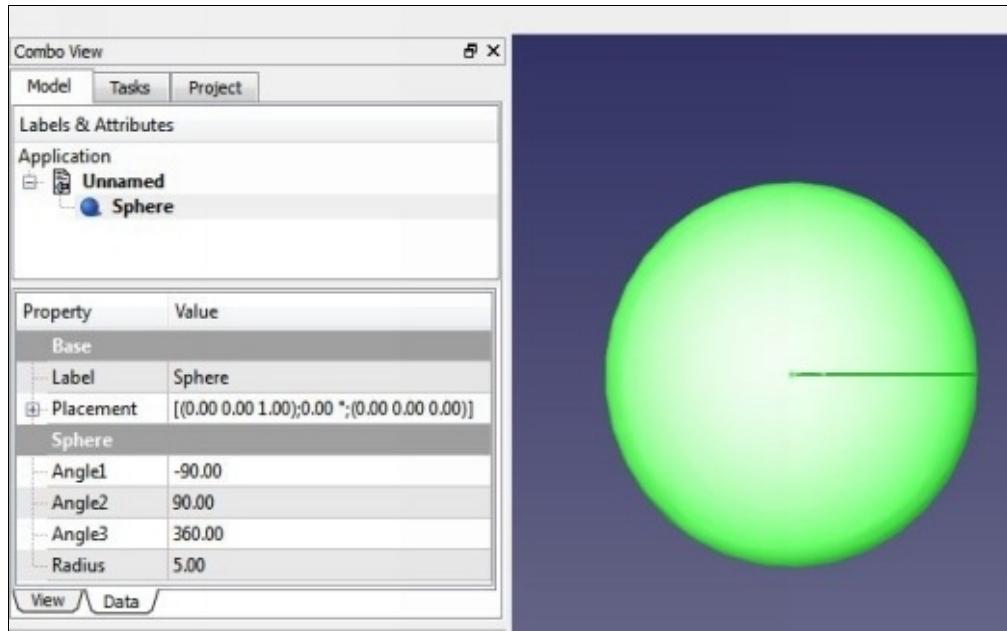
- PROPERTIES

:- The properties value tab are used for modification of Cylinder. Select the Cylinder from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement of cylinder: - **Property** → **Data** → **Placement**.
2. For radius: - **Property** → **Data** → **Radius** value.
3. For height: - **Property** → **Data** → **Height** value.
4. For angle: - **Property** → **Data** → **Angle**.
5. For cylinder Shape color, Draw style, Transparency etc: - **Property** → **View**.

9.3 HOW TO MAKE SOLID SPHERE

This tool is use to create a solid sphere with position, radius, angle1, angle2 & angle3 in 3D environment.



- **OPERATION**

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Part.
3. Then, select the  **Sphere tool**.

4. A solid sphere with standard dimension and position will be created.

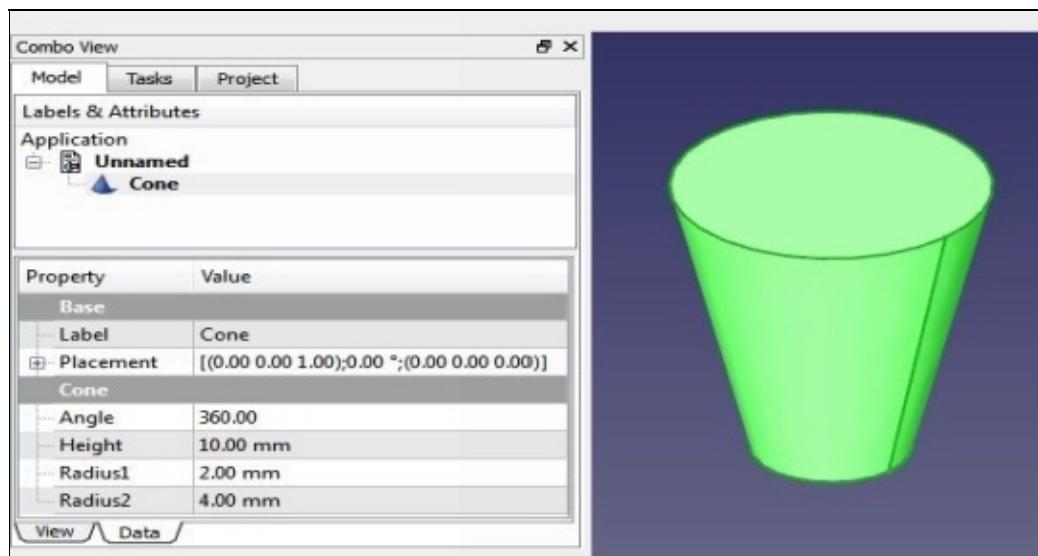
- **PROPERTIES**

:- The properties value tab are used for modification of Sphere. Select the Sphere from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement of sphere: - **Property** → **Data** → **Placement**.
2. For radius: - **Property** → **Data** → **Radius** value.
3. For angle1: - **Property** → **Data** → **Angle1** value.
4. For angle2: - **Property** → **Data** → **Angle2** value.
5. For angle3: - **Property** → **Data** → **Angle3** value.
6. For sphere Shape color, Draw style, Transparency etc: - **Property** → **View**.

9.4 HOW TO MAKE SOLID CONE

This tool is use to create a truncated parametric solid cone with position, radius1, radius2, height & angle in 3D environment.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Part.
3. Then, select the  **Cone tool**.
4. A solid cone with standard dimension and position will be created.

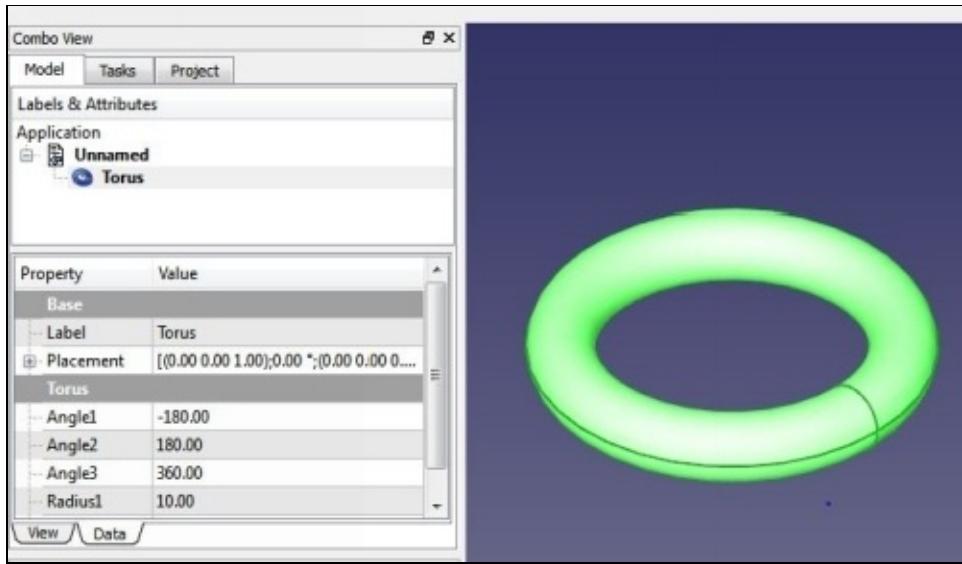
- PROPERTIES

:– The properties value tab are used for modification of cone. Select the cone from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement of cone: - **Property** → **Data** → **Placement**.
2. For radius1: - **Property** → **Data** → **Radius1** value.
3. For radius2: - **Property** → **Data** → **Radius2** value.
4. For height: - **Property** → **Data** → **Height** value.
5. For angle: - **Property** → **Data** → **Angle** value.
6. For cone Shape color, Draw style, Transparency etc: - **Property** → **View**.

9.5 HOW TO MAKE SOLID TORUS

This tool is use to create a parametric solid torus with position, radius1, radius2, angle1, angle2 & angle3 in 3D environment.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Part.
3. Then, select the  **Torus tool**.
4. A solid torus with standard dimension and position will be created.

- PROPERTIES

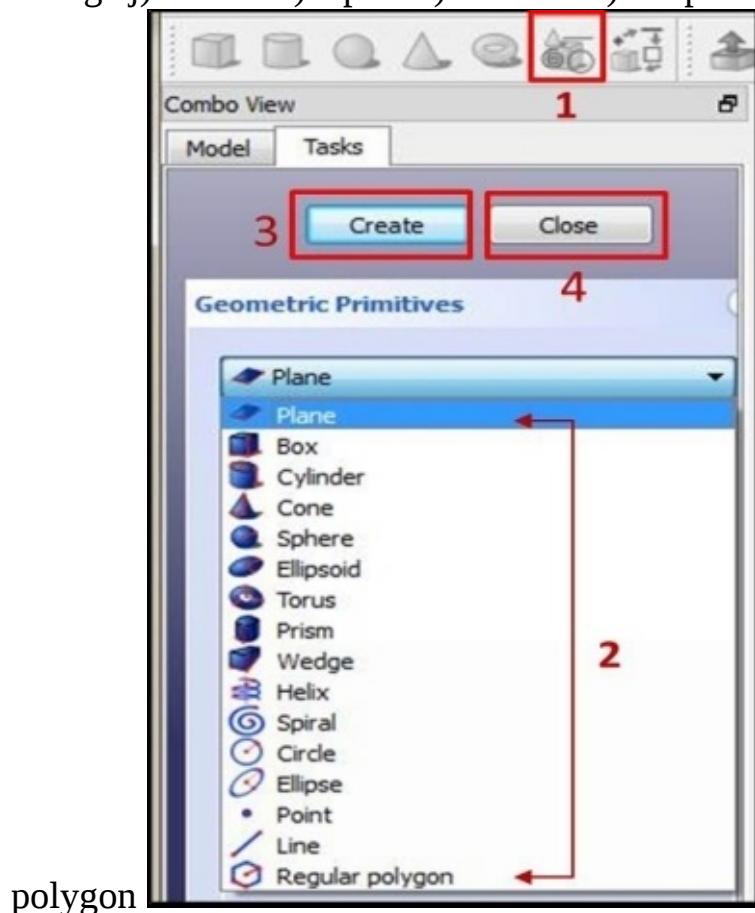
:- The properties value tab are used for modification of torus. Select the torus from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement of torus: - **Property** → **Data** → **Placement**.
2. For radius1: - **Property** → **Data** → **Radius1** value.
3. For radius2: - **Property** → **Data** → **Radius2** value.
4. For angle1: - **Property** → **Data** → **Angle1** value.
5. For angle2: - **Property** → **Data** → **Angle2** value.
6. For angle3: - **Property** → **Data** → **Angle3** value.

7. For torus Shape color, Draw style, Transparency etc: - **Property** → **View**.

- **HOW TO CREATE VARIOUS PRIMITIVES**

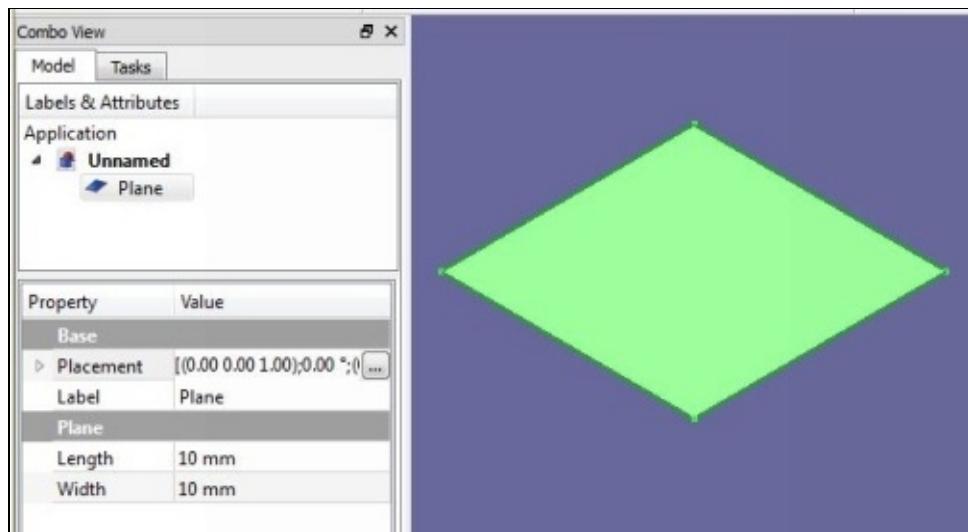
This tool is used to create various parametric geometric primitives in current working plane. Currently this tools can create a following parametric: a) Plane b) Box c) Cylinder d) Cone e) Sphere f) Ellipsoid g) Torus h) Prism i) Wedge j) Helix k) Spiral l) Circle m) Ellipse n) Point o) Line p) Regular polygon



In this primitives, i have already discussed about how to create Box, Cylinder, Cone, Sphere and Torus. The operation with reaming geometry primitives are discussed below: -

9.6 HOW TO CREATE PLANE

This tool is use to create a parametric plane (10 x 10)mm with position, length & width in 3D environment.



• OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Part.
3. Then, click on the  **Create primitives tool**.
4. A Geometric Primitives tab will open in Combo view.
5. Select the  **Plane** tool form dropdown menu.
6. You can edit the parameter value in parametric input.
7. Then click on **Create**.
8. A plane with standard dimension and position will be created.
9. Once you have created the plane you have the possibility to edit its parameters from properties value.
10. And last click on **Close** to exit the operation.

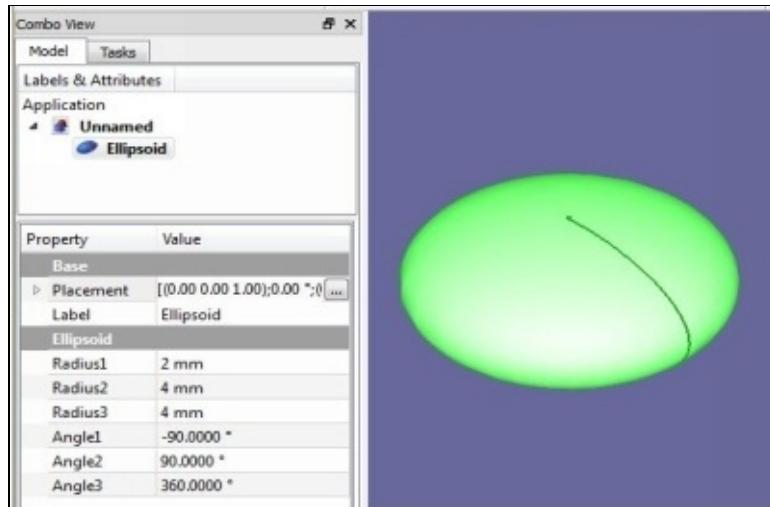
- PROPERTIES

:- The properties value tab are used for modification of plane. Select the plane from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement of plane: - **Property** → **Data** → **Placement**.
2. For length: - **Property** → **Data** → **Length** value.
3. For width: - **Property** → **Data** → **Width** value.
4. For plane Shape color, Draw style, Transparency etc: - **Property** → **View**.

9.7 HOW TO CREATE ELLIPSOID

This tool is use to create a parametric solid Ellipsoid with position, radius1, radius2, radius3, angle1, angle2, & angle3 in 3D environment.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Part.

3. Then, click on the  **Create primitives tool**.
4. A Geometric Primitives tab will open in Combo view.
5. Select the  **Ellipsoid** tool from dropdown menu.
6. You can edit the parameter value in parametric input.
7. Then click on **Create**.
8. A Ellipsoid with standard dimension and position will be created.
9. Once you have created the Ellipsoid you have the possibility to edit its parameters from properties value.
10. And last click on **Close** to exit the operation.

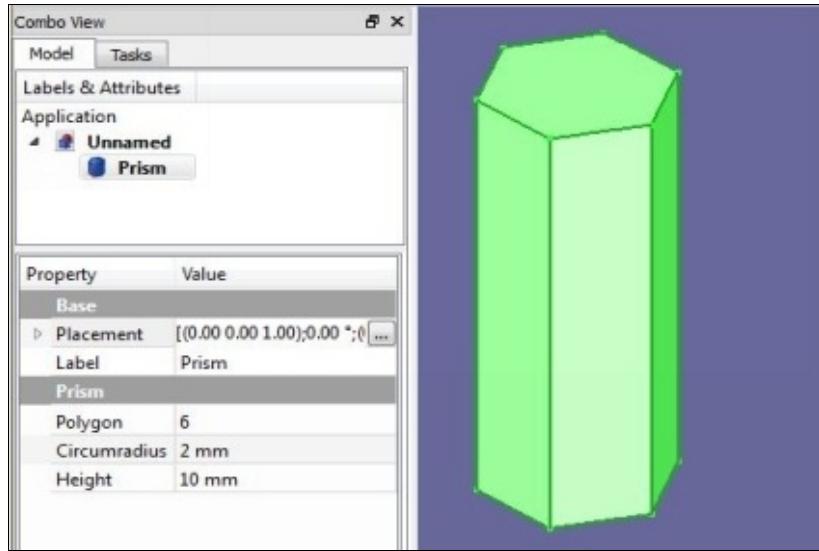
- **PROPERTIES**

:- The properties value tab are used for modification of Ellipsoid. Select the Ellipsoid from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement of Ellipsoid: - **Property** → **Data** → **Placement**.
2. For radius1: - **Property** → **Data** → **Radius1** value.
3. For radius2: - **Property** → **Data** → **Radius2** value.
4. For radius3: - **Property** → **Data** → **Radius3** value.
5. For angle1: - **Property** → **Data** → **Angle1** value.
6. For angle2: - **Property** → **Data** → **Angle2** value.
7. For angle3: - **Property** → **Data** → **Angle3** value.
8. For Ellipsoid Shape color, Draw style, Transparency etc: - **Property** → **View**.

9.8 HOW TO CREATE SOLID PRISM

This tool is use to create a parametric solid prism with position, polygon, circumradius & height in 3D environment.



- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Part.
3. Then, click on the  **Create primitives tool**.
4. A Geometric Primitives tab will open in Combo view.
5. Select the  **Prism** tool from dropdown menu.
6. You can edit the parameter value in parametric input.
7. Then click on **Create**.
8. A solid prism with standard dimension and position will be created.
9. Once you have created the prism you have the possibility to edit its parameters from properties value.
10. And last click on **Close** to exit the operation.

- PROPERTIES

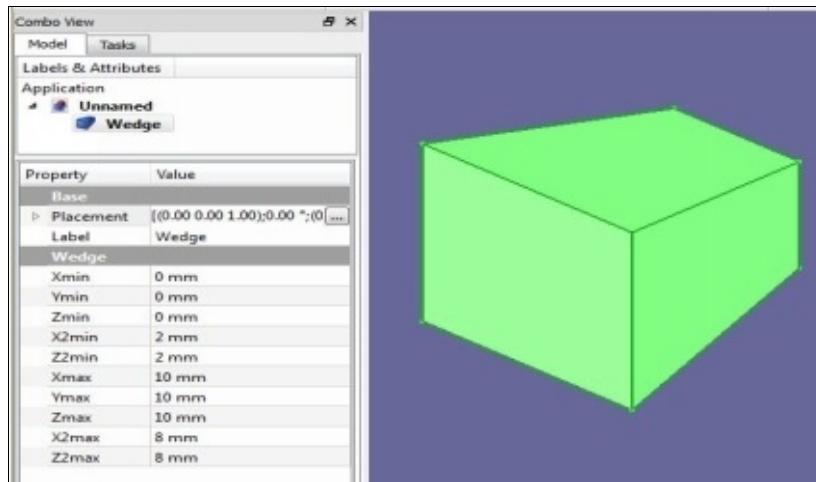
:- The properties value tab are used for modification of prism. Select the prism from the Combo view → Model and then put the value what ever you want, in

Property Value.

1. For placement of prism: - **Property** → **Data** → **Placement**.
2. For number of polygon: - **Property** → **Data** → **Polygon** value.
3. For circumradius: - **Property** → **Data** → **Circumradius** value.
4. For height: - **Property** → **Data** → **Height** value.
5. For Prism Shape color, Draw style, Transparency etc: - **Property** → **View**.

9.9 HOW TO CREATE WEDGE

This tool is used to create a parametric wedge in 3D environment. This Wedge defaults to a larger square base and a smaller square top.

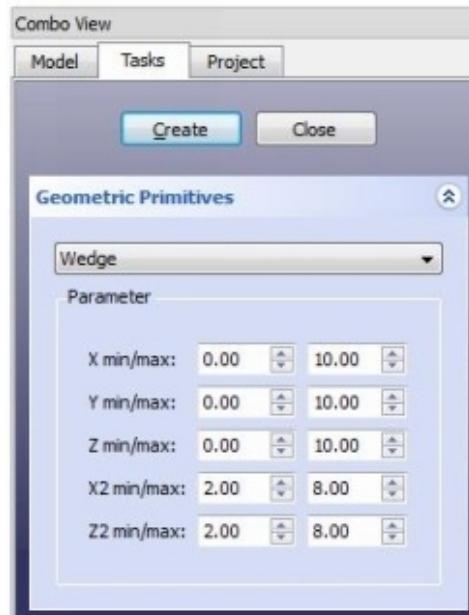


• OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Part.
3. Then, click on the  **Create primitives tool**.
4. A Geometric Primitives tab will open in Combo view.



5. Select the **Wedge** tool from dropdown menu.
6. You can edit the parameter value in parametric input.



The meaning of parameter input are: -

- **X min/max** : Base face X axis span.
 - **Y min/max**: Wedge height span.
 - **Z min/max** : Base face Z axis span.
 - **X2 min/max** : Top face X axis span.
 - **Z2 min/max** : Top face Z axis span.
7. Then click on **Create**.
 8. A wedge with standard dimension and position will be created.
 9. Once you have created the wedge you have the possibility to edit its parameters from properties value.
 10. And last click on **Close** to exit the operation.

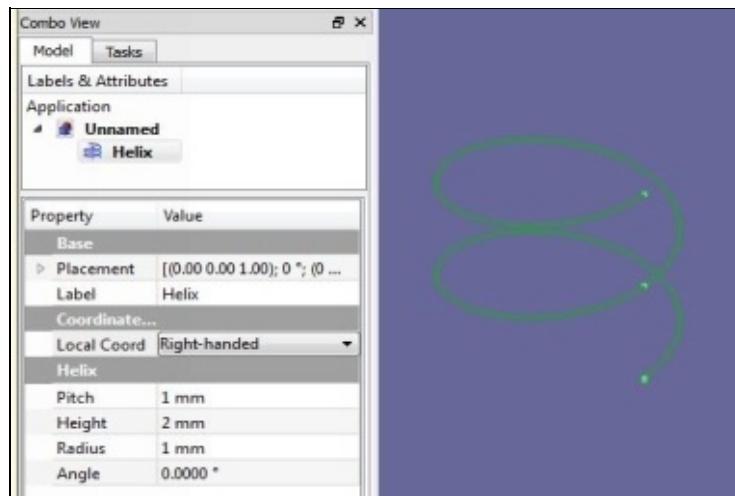
- **PROPERTIES**

:- The properties value tab are used for modification of wedge. Select the wedge from the Combo view → Model and then put the value what ever you want, in Property Value.

1. For placement of wedge: - **Property** → **Data** → **Placement**.
2. For required parameters: - **Property** → **Data** → **Parameters** value.
3. For wedge Shape color, Draw style, Transparency etc: - **Property** → **View**.

9.10 HOW TO CREATE HELIX

This tool is used to create a wire helix with position, pitch, height, radius, angle & coordinate systems in 3D environment.



Description Of Helix Parameter: a) **Pitch:** - The pitch corresponds to the space between two consecutive "turns" of the helix measured along the main axis of the helix.

b) **Height:** - The height corresponds to the overall height of the helix measured along the main axis of the helix.

c) **Radius:** - The radius corresponds to the radius of the circle built by the helix by viewing the helix from the top / bottom.

d) **Angle:** - This angle corresponds to the angle of the conus. The value must be comprised between -90 deg. and +90 deg.

e) **Coordinate systems:** - This parameter specifies the right-handed or left-handed of the helix.

- OPERATION

1. Run or open the FreeCAD software.
2. Switch the Workbench and select the Part.
3. Then, click on the  **Create primitives tool**.
4. A Geometric Primitives tab will open in Combo view.
5. Select the  **Helix** tool from dropdown menu.
6. You can edit the parameter value in parametric input.
7. Then click on **Create**.
8. A helix with standard dimension and position will be created.
9. Once you have created the helix you have the possibility to edit its parameters from properties value.
10. And last click on **Close** to exit the operation.

- PROPERTIES

:- The properties value tab are used for modification of helix. Select the helix from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For placement: - **Property** → **Data** → **Placement**.
- b) For coordinate: - **Property** → **Data** → **Local coord** value.
- c) For pitch: - **Property** → **Data** → **Pitch** value.
- d) For height: - **Property** → **Data** → **Height** value.
- e) For radius: - **Property** → **Data** → **Radius** value.
- f) For angle: - **Property** → **Data** → **Angle** value.
- g) For helix Shape color, Draw style, Transparency etc: - **Property** →

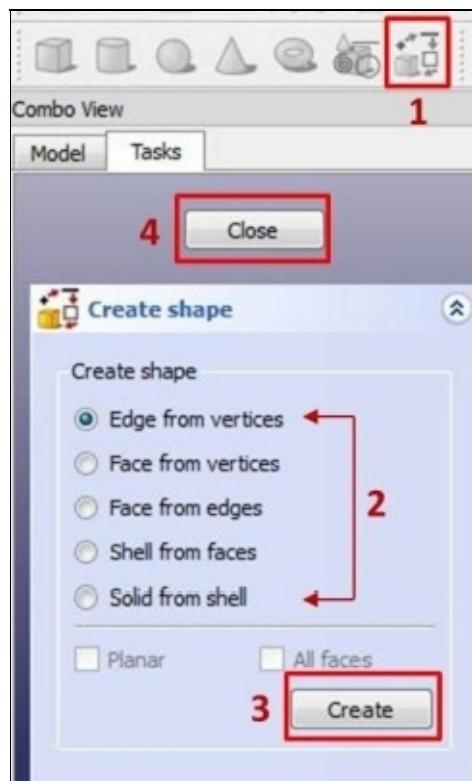
View.

Note:

The rest of the geometric primitives like **Spiral**, **Circle**, **Ellipse**, **Point**, **Line** and **Regular polygon** are creates by the same procedure those described above.

9.11 HOW TO SHAPE BUILDER

This tool is use to create more complex shapes from various parametric geometric primitives.



In this, there are five types of shapes can be create. The description are as follows: 1) [CREATE EDGE FROM VERTICES](#):

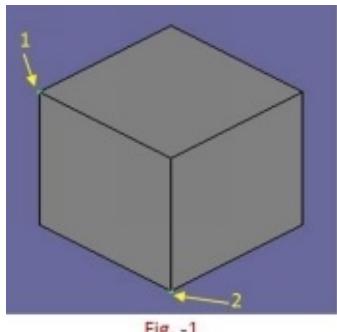


Fig. -1

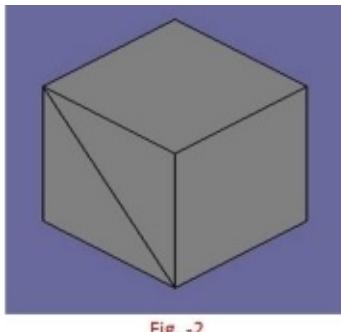


Fig. -2

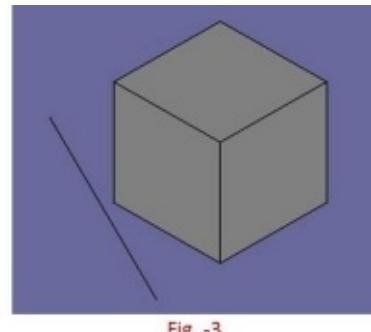


Fig. -3

- OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the Part.
- c) Make a one part module(cube, cylinder etc.).



- d) Then, click on the **Shape builder tool**.
- e) A **create shape** tab will open in Combo view.
- f) Select the **Edge from vertices**.
- g) Then select the two vertices with the help of **ctrl** in part module (see fig. -1).
- h) A selected items becomes highlighted and it will change colour to green.
- i) Then click on **Create**.
- j) A edge will be created (see fig. -2).
- k) Then click on **Close** to exit the operation.

- l) You can select the edge from the combo view and give the placement to the edge from properties value (see fig. -3).

2) CREATE FACE FROM VERTICES:

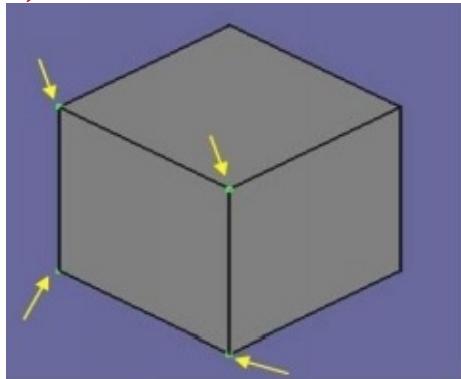


Fig. -1

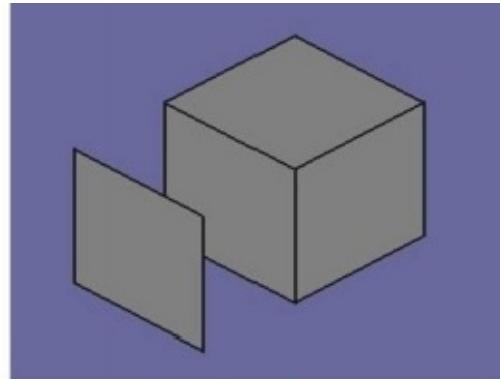


Fig. -2

• OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the Part.
- c) Make a one part module(cube, cylinder etc.).
- d) Then, click on the  **Shape builder tool**.
- e) A **create shape** tab will open in Combo view.
- f) Select the  **Face from vertices**.
- g) Then select the three or more vertices with the help of **ctrl** in part module (see fig. -1).
- h) A selected items becomes highlighted and it will change colour to green.

- i) You can make a check in **Planer** also.
- j) Then click on **Create**.
- k) A face will be created.
- l) Then click on **Close** to exit the operation.
- m) You can select the face from the combo view and give the placement to the face from properties value (see fig. -2).

3) CREATE FACE FROM EDGES:

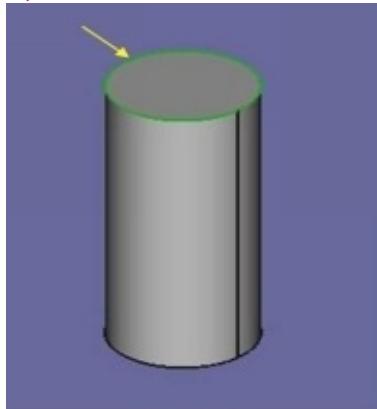


Fig. -1

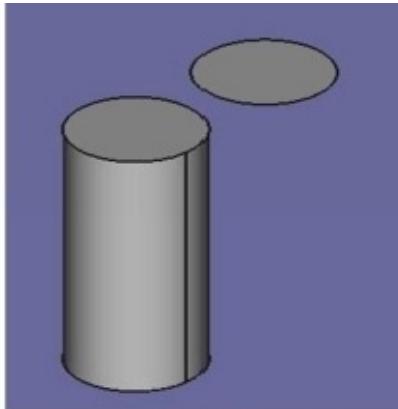


Fig. -2

- **OPERATION**

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the Part.
- c) Make a one part module(cube, cylinder etc.).
- d) Then, click on the  **Shape builder tool**.
- e) A **create shape** tab will open in Combo view.

- f) Select the  **Face from edges**.
- g) Then select the one or more vertices with the help of **ctrl** in part module (see fig. -1).
- h) A selected items becomes highlighted and it will change colour to green.
- i) You can make a check in **Planer** (do not check for make a plane face).

- j) Then click on **Create**.
- k) A face will be created.
- l) Then click on **Close** to exit the operation.
- m) You can select the face from the combo view and give the placement to the face from properties value (see fig. -2).

4) CREATE SHELL FROM FACES:

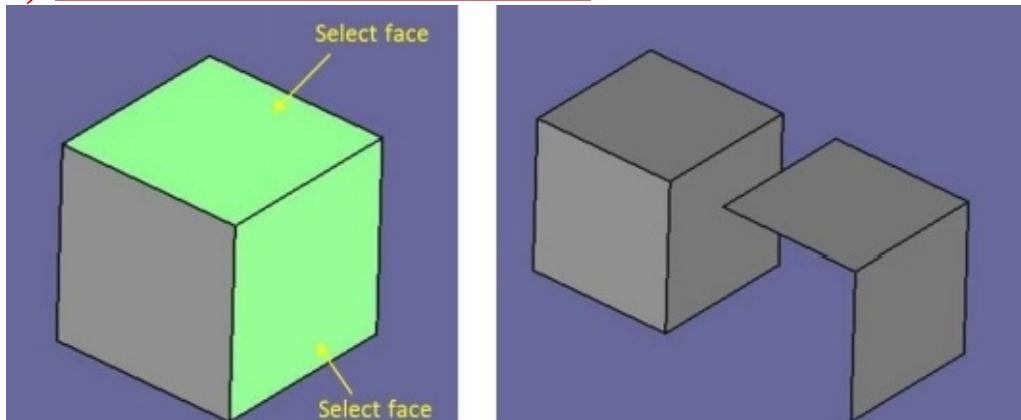


Fig. -1

Fig. -2

- OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the Part.
- c) Make a one part module(cube, cylinder etc.).



- d) Then, click on the **Shape builder tool**.
- e) A **create shape** tab will open in Combo view.
- f) Select the **Shell from faces**.
- g) Then select the two or more faces with the help of **ctrl** in part module (see fig. -1).
- h) A selected items becomes highlighted and it will change colour to green.
- i) You can make a check in **All faces** (for make a complete hollow shell) .
- j) Then click on **Create**.
- k) A shell will be created.
- l) Then click on **Close** to exit the operation.
- m) You can select the Shell from the combo view and give the placement to the Shell from properties value (see fig. -2).

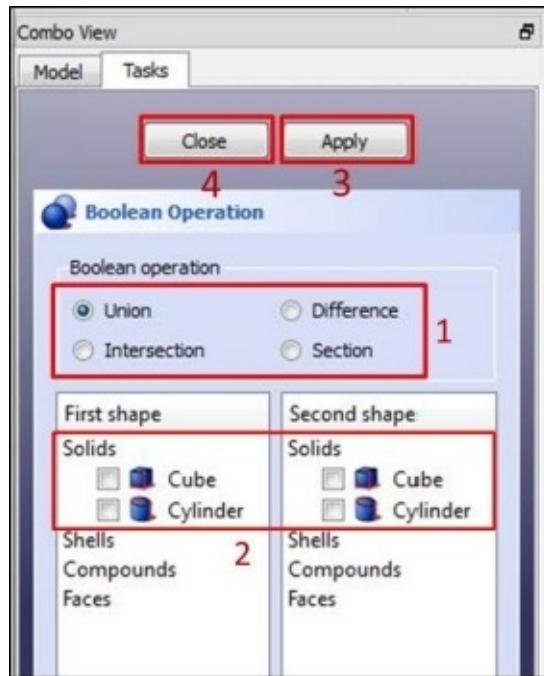
5) CREATE SOLID FROM SHELL:

- OPERATION

- a) Run or open the FreeCAD software.
 - b) Switch the Workbench and select the Part.
 - c) Make a one part module(cube, cylinder etc.).
- d) Then, click on the  **Shape builder tool**.
- e) A **create shape** tab will open in Combo view.
- f) Then make a Shell object (follow the above described procedure).
- g) Select the Shell from the combo view and give the placement to the Shell from properties value.
- h) Then, again click on the  **Shape builder tool**.
- i) Select the  **Solid from shell**.
- j) Then select the only one part of shell object.
- k) A selected items becomes highlighted and it will change colour to green.
- l) Then click on **Create**.
- m) A solid shell will be created.
- n) Then click on **Close** to exit the operation.
- o) You can select the Solid from the combo view and give the placement to the Solid from properties value **WORKING WITH
MODIFYING OBJECT TOOLS -1**

1) HOW TO PERFORM BOOLEAN OPERATION

The Boolean command is an all-in-one command. This command is used to perform **Union**, **Difference(Cut)**, **intersection** and **section** operation through one dialog.



• OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the Part.
- c) Draw two or more shapes (cube, cylinder etc.) in 3D environment.
- d) Then, click on the  **Boolean operation tool**.
- e) A Boolean dialog box will open in Combo view.
- f) Select the Boolean operation like **Union**, **Difference(Cut)**, **intersection** and **section**.
- g) Then check the part object from the **First shape** and form the **Second shape**.

- h) Then click on the **Apply** button.
- i) A new object will be created.
- j) Then click on **Close** to exit the operation.

- **PROPERTIES**

:— The properties value tab are used for modification of object. Select the object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For placement: - **Property** → **Data** → **Placement**.
- b) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

2) **HOW TO MAKE CUT OF TWO SHAPES**

This tool is use to make a cuts (subtracts) of two shapes or objects. In this, the last one object is being subtracted from the first one object. This operation is fully parametric and the components can be modified and the result recomputed. This operation can also be performed by the Boolean operation.

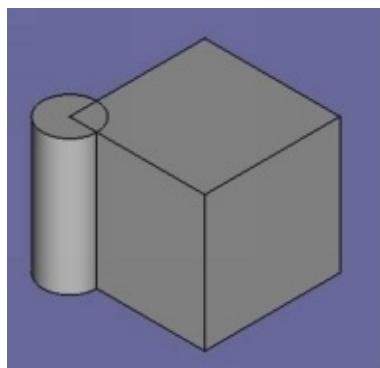


Fig. -1

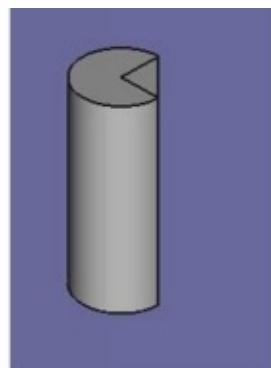


Fig. -2

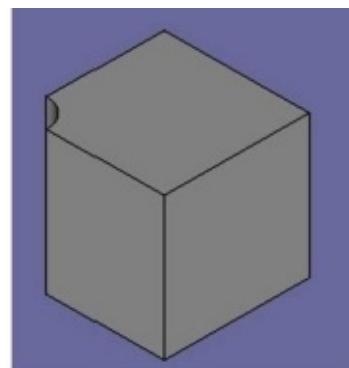


Fig. -3

- **OPERATION**

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the Part.

- c) Draw two or more shapes (cube, cylinder etc.) in 3D environment (see fig. -1).
- d) Select the Cylinder first then press ctrl and select the Cube.
- e) A selected items becomes highlighted and it will change colour to green.

- f) Click on the  **Cut tool**.
- g) A new cut object will be created (see fig. -2).
- h) Or select the Cube first then press ctrl and select the cylinder.
 **Cut tool**.
- i) Click on the  **Cut tool**.
- j) A new cut object will be created (see fig. -3).

- **PROPERTIES**

:- The properties value tab are used for modification of object. Select the cut object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For placement: - **Property** → **Data** → **Placement**.
- b) For edit in cylinder parameters: - **Property** → **Data** → **Base** → **Cylinder**.
- c) For edit in cube parameters: - **Property** → **Data** → **Tool** → **Cube**.
- d) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

3) HOW TO MAKE UNION OF SEVERAL SHAPES

This tool is use to make union (fuses) of several shapes into one. This operation is fully parametric and the components can be modified and the result recomputed. This operation can also be performed by the Boolean operation.

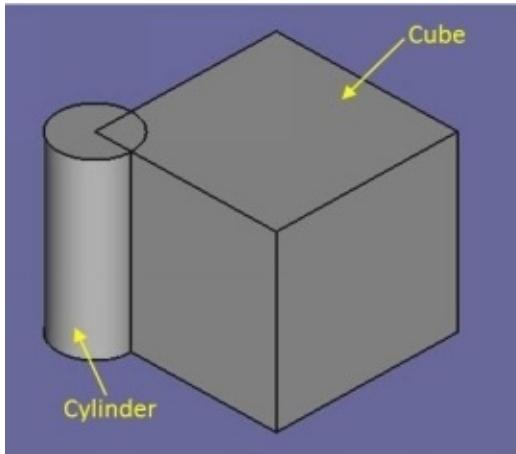


Fig. -1

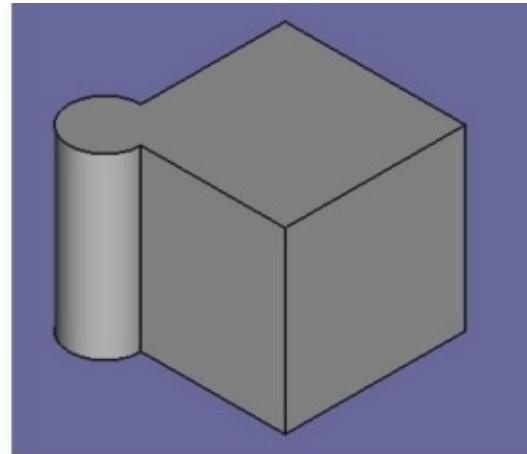


Fig. -2

• OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the Part.
- c) Draw two or more shapes (cube, cylinder etc.) in 3D environment (see fig. -1).
- d) Select two or more shapes form combo view.
- e) A selected items becomes highlighted and it will change colour to green.
- f) Click on the  **Union (Fuse) tool**.
- g) A new Fusion object will be created (see fig. -2).

• PROPERTIES

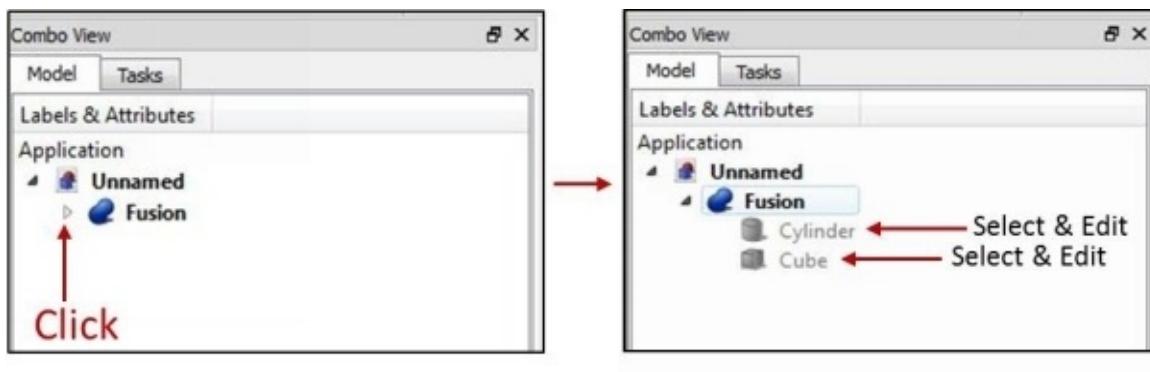
:- The properties value tab are used for modification of fusion object. Select the

fusion object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For placement: - **Property** → **Data** → **Placement**.
- b) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

NOTE: -

If you want make a change in object parameters like length, radius, height etc. click on arrow button behind fusion object and select the shapes (cube, cylinder etc.) and put the required value in properties value dialog (see fig. below).



4) HOW TO MAKE INTERSECTION OF TWO SHAPES

This tool is use to make intersection (common) of two shapes. This operation is fully parametric and the components can be modified and the result recomputed. This operation can also be performed by the Boolean operation.

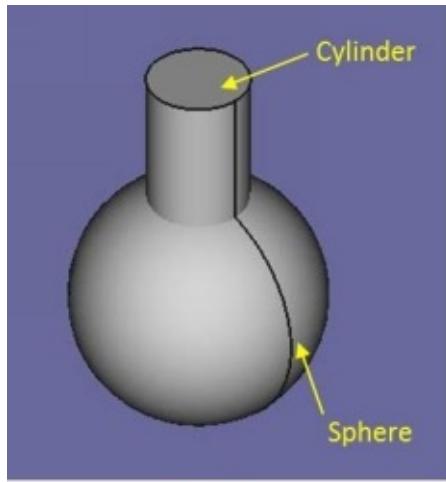


Fig. -1

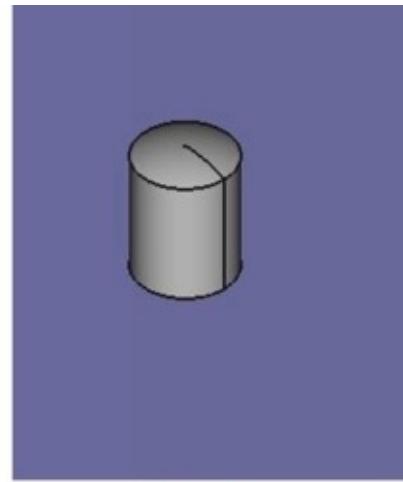


Fig. -2

- OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the Part.
- c) Draw two or more shapes (cube, cylinder etc.) in 3D environment (see fig. -1).
- d) Select two shapes from combo view.
- e) A selected items becomes highlighted and it will change colour to green.

- f) Click on the  **Intersection (Common) tool**.
- g) A new Common object will be created (see fig. -2).

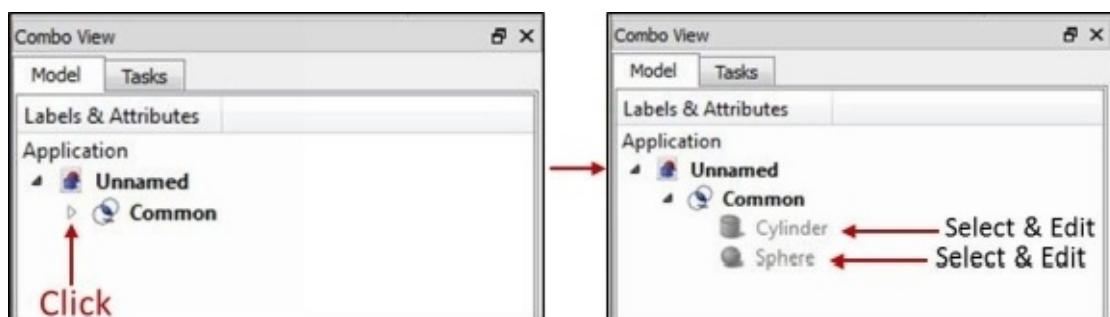
- PROPERTIES

:- The properties value tab are used for modification of Common object. Select the common object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For placement: - **Property** → **Data** → **Placement**.
- b) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

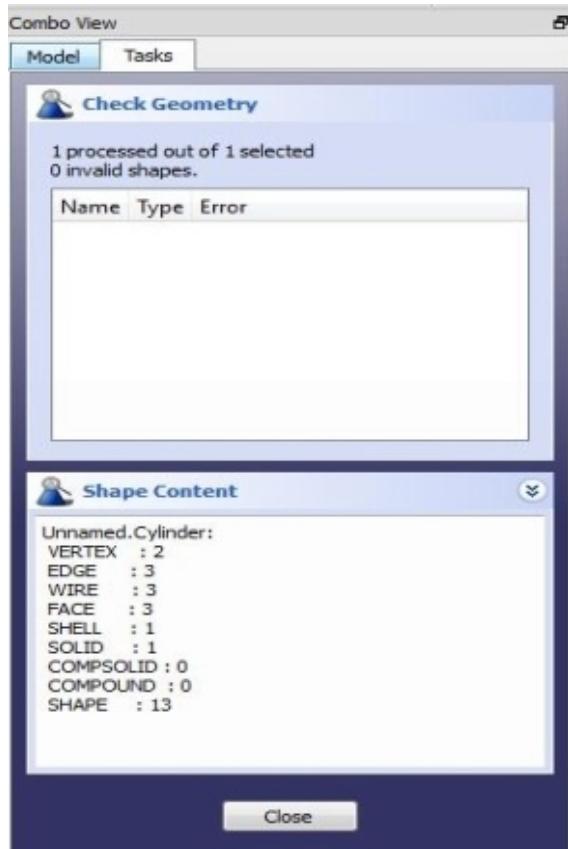
NOTE: -

If you want make a change in object parameters like length, radius, height etc. click on arrow button behind Common object and select the shapes (cube, cylinder etc.) and put the required value in properties value dialog (see fig. below).



5) HOW TO USE CHECK GEOMETRY TOOL

This tool is use to check the geometry errors in a part module.



- OPERATION

- Select one or more shapes (object) form combo view.
- A selected items becomes highlighted and it will change colour to green.
- Click on the  **Check error tool**.
- A Check Geometry dialog will be open in combo view (see above fig.).
- In this, upper part of dialog box shows the errors information and lower part of dialog box shows the Shape content information (see above fig.).
- Then click on **Close** to exit the operation.

6) HOW TO MAKE A SECTION OF TWO SHAPES

This tool is used to make a section of two shapes. In this, the second one being used as a section plane. This operation is fully parametric and the components can be modified and the result recomputed. This operation can also be performed by the Boolean operation.

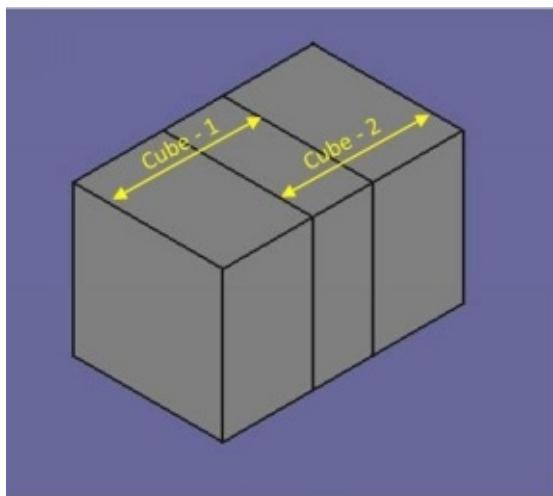


Fig. -1

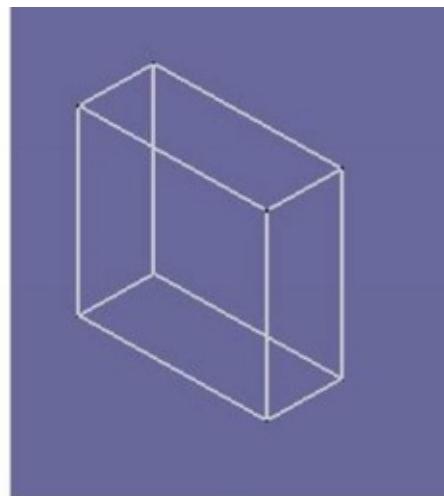


Fig. -2

• OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the Part.
- c) Draw two shapes (cube -1, cube -2) in 3D environment (see fig. -1).
- d) Give the placement of cube – 2 in positive Y direction (see fig. -1)
- e) Select two shapes (cube - 1 & cube – 2) from combo view.
- f) A selected items becomes highlighted and it will change colour to green.



- g) Click on the **Section tool**.
- h) A new Section object will be created (see fig. -2).

• PROPERTIES

:- The properties value tab are used for modification of Section object. Select the Section object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For placement: - **Property** → **Data** → **Placement**.
- b) For edit in cube - 1 parameters: - **Property** → **Data** → **Base** → **Cube**.
- c) For edit in cube - 2 parameters: - **Property** → **Data** → **Tool** → **Cube001**.
- d) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

7) HOW TO MAKE A CROSS-SECTION OF SHAPE

This tool is use to make a section object. This operation is fully parametric and the components can be modified and the result recomputed.

- **OPERATION**

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the Part.
- c) Make a cut object or whatever you want to design in 3D environment (see fig. -1).

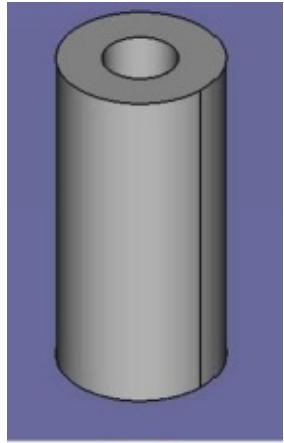


Fig. -1

- d) Then select the cut object form combo view.
- e) A selected items becomes highlighted and it will change colour to green.
- f) Click on the  **Cross-section tool**.
- g) A cross-section dialog will open in combo view (see fig. -2).

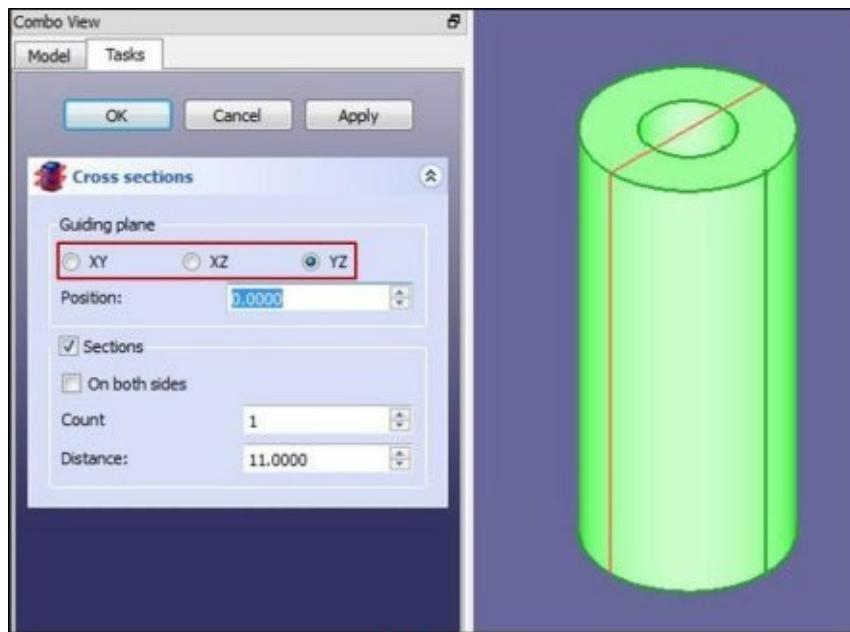


Fig. -2

-
- h) You can change the Guiding plane and position from dialog box (see fig. -2).
 - i) you can also check the Sections and On both side form dialog box.
 - j) you can also change the number of count and distance from dialog box.
 - k) Then click on **Apply** button and select the **OK** to exit the operation.
 - l) A cross-section object will created.
 - m) Then right click on cut object and select the Hide selection to see the cross-section object (see fig. – 3).

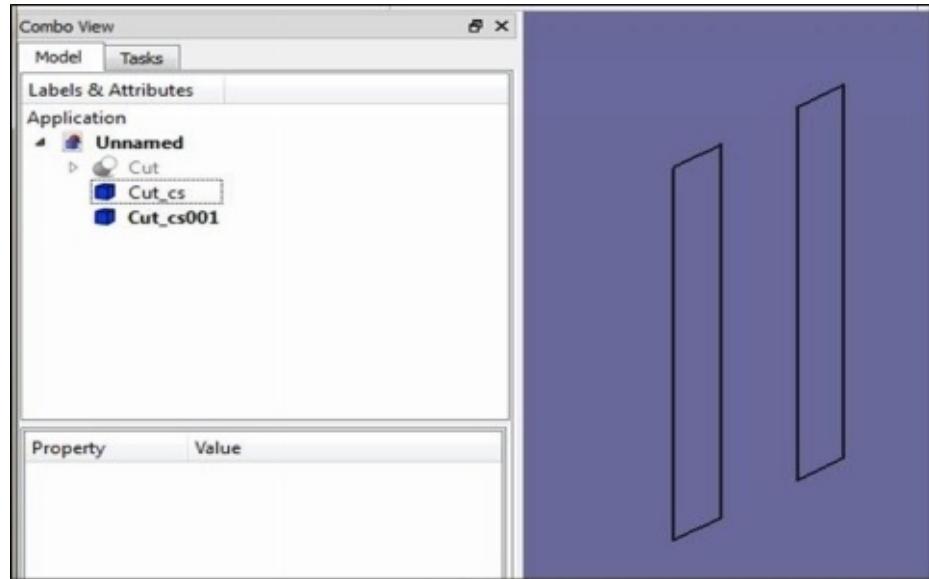


Fig. -3

- **PROPERTIES**

a) For placement: - **Property** → **Data** → **Placement**.

WORKING WITH MODIFYING OBJECT TOOLS -2

1) HOW TO USE EXTRUDE TOOL

This tool is used to extend a shape by a specified distance and in a specified direction. During operation, the output shape of object varies and depending on the input shape and the options selected.

The extrude tool is commonly used in following operation:

- Extrude a Vertex (point), will produce a linear Edge (Line).
- Extrude an open edge (e.g. line, arc), will produce an open face (e.g. plane).
- Extrude 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).
- Extrude an open Wire (e.g. a Draft Wire), will produce an open shell (several joined faces).

- Extrude 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.
- Extrude a face (e.g. plane), will produce a solid (e.g. Cuboid).
- Extrude a **Draft Shape String**, will produce a compound of solids (the string is a compound of the letters which are each a solid).

- OPERATION

- a) Select the shape(s) in the 3D view or from the combo view.
- b) A selected items becomes highlighted and it will change colour to green.
- c) Click on the  **Extrude tool**.
- d) A Extrude dialog will open in combo view (see fig. -1).

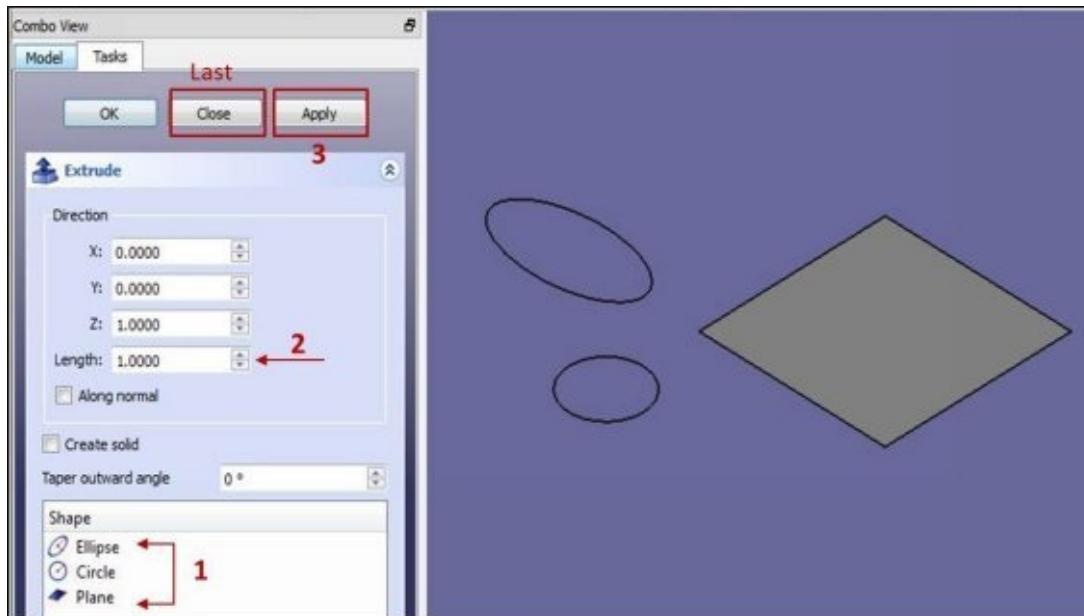


Fig. -1

- e) You can also select the shape(s) form Extrude dialog (see fig. -1).

- f) Set the direction and distance value in Extrude dialog (see fig. -1).
- g) You can also set the other options like Taper angle, Create solid, Along normal etc.
- h) Click on Apply.
- i) A extrude object will created in 3D environment (see fig. – 2).

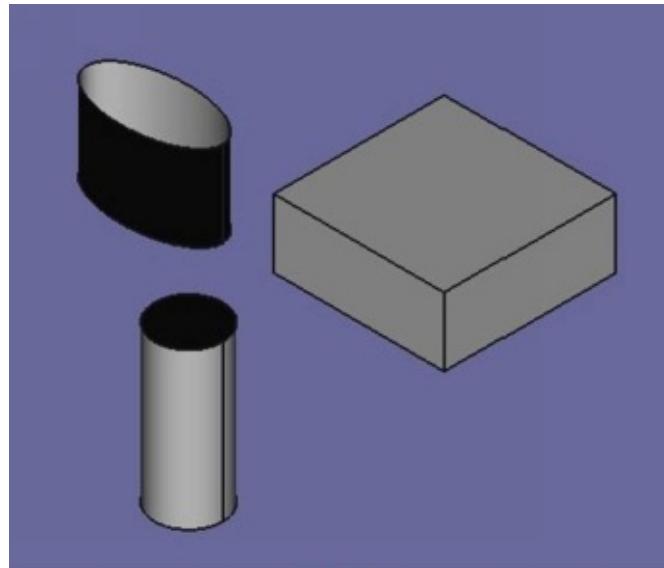


Fig. -2

- j) Click on Close button to exit the operation.

• PROPERTIES

:- The properties value tab are used for modification of Extrude object. Select the extrude object from the Combo view → Model and then put the value what ever you want, in Property Value.

- e) For edit in shape parameters: - **Property** → **Data** → **Base**.
- f) For direction of shape: - **Property** → **Data** → **Dir.**
- g) For make a solid of shape: - **Property** → **Data** → **Solid**(false/true).

- h) For taper angle: - **Property** → **Data** → **Taper Angle**.
- i) For placement of the shape: - **Property** → **Data** → **Placement**.
- j) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

2) HOW TO USE REVOLVE TOOL

This tool is use to make a revolves of the selected object around a given axis. In this, the following shape types are allowed to make a revolve shapes or object:

- Vertex
- Edge
- Wire
- Face
- Shell

Note:-

Solids or compound solids are not allowed as input shapes. Normal compounds are currently not allowed, too.

- **OPERATION**

- a) Select the object (circle etc.) in the 3D view or from the combo view.
- b) A selected items becomes highlighted and it will change colour to green.

- c) Click on the  **Revolve tool**.
- d) A Revolve dialog will open in combo view (see fig. -1).

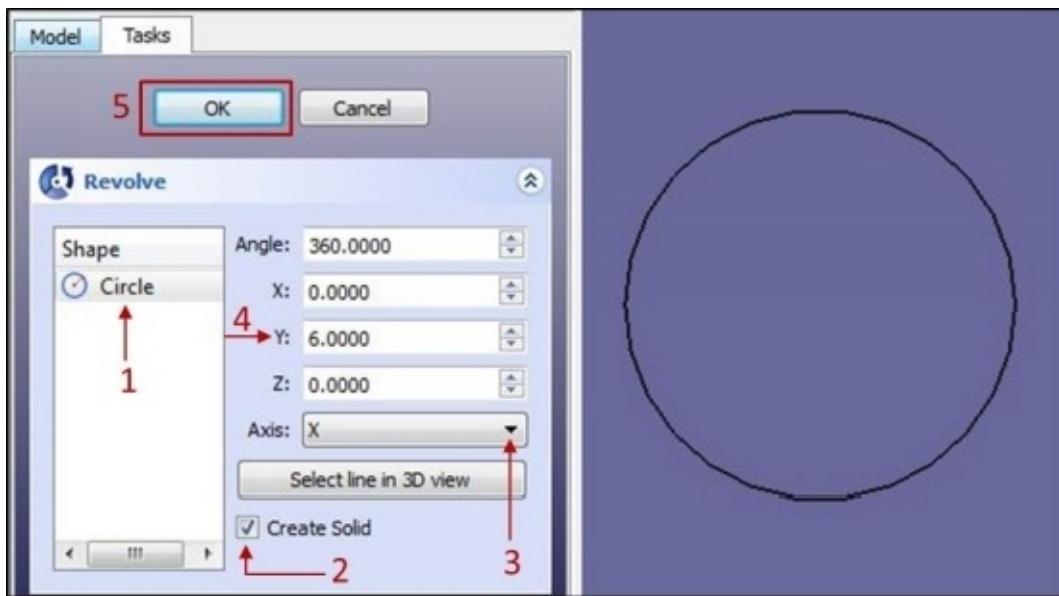


Fig. -1

- e) You can also select the object from dialog box (see fig. -1).
- f) Set the value you want or see the above fig.
- g) Then click on **OK** button.
- h) A Revolve object will created in 3D view (see fig. – 2).

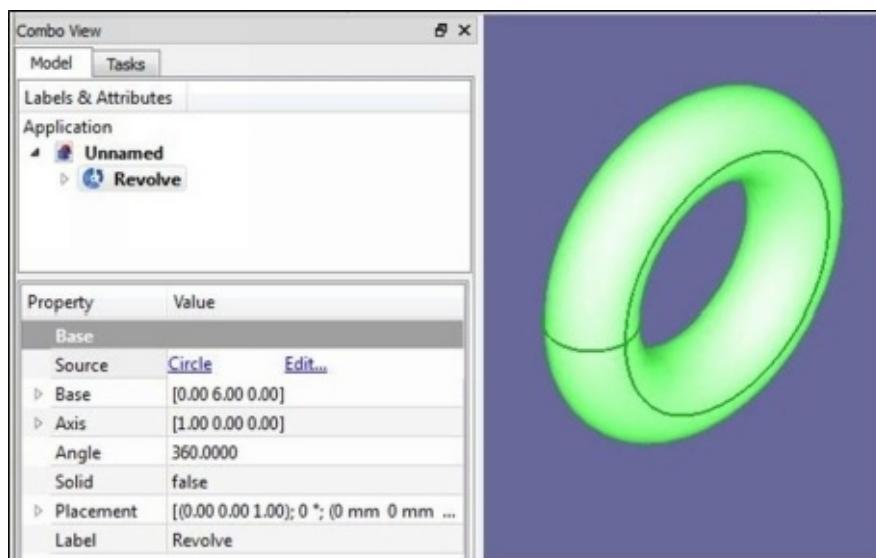


Fig. -2

- PROPERTIES

:- The properties value tab are used for modification of Revolve object. Select the Revolve object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For edit in shape parameters: - **Property** → **Data** → **Source**.
- b) For angle: - **Property** → **Data** → **Angle**.
- c) For make a solid of shape: - **Property** → **Data** → **Solid**(false/true).
- d) For placement of the shape: - **Property** → **Data** → **Placement**.
- e) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

3) HOW TO USE MIRROR TOOL

This tool is use to creates a new object (image) which is a reflection of the original object. The image object is created behind a mirror plane. The mirror plane may be standard plane (**XY**, **YZ**, or **XZ**), or any plane parallel to a standard plane.

- **OPERATION**

- a) Select the object (cylinder, circle etc.) in the 3D view or from the combo view.
- b) A selected items becomes highlighted and it will change colour to green.
- c) Click on the  **Revolve tool**.
- d) A Mirroring dialog will open in combo view (see fig. -1).

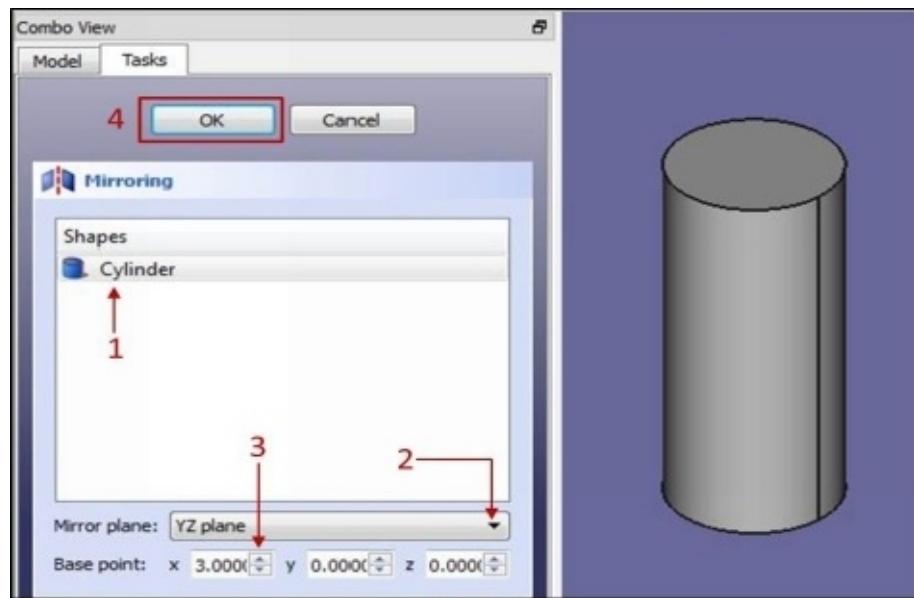


Fig. -1

- e) You can also select the object from dialog box (see fig. -1).
- f) Set the value you want or see the above fig.
- g) Then click on **OK** button.
- h) A Mirror object will created in 3D view (see fig. – 2).

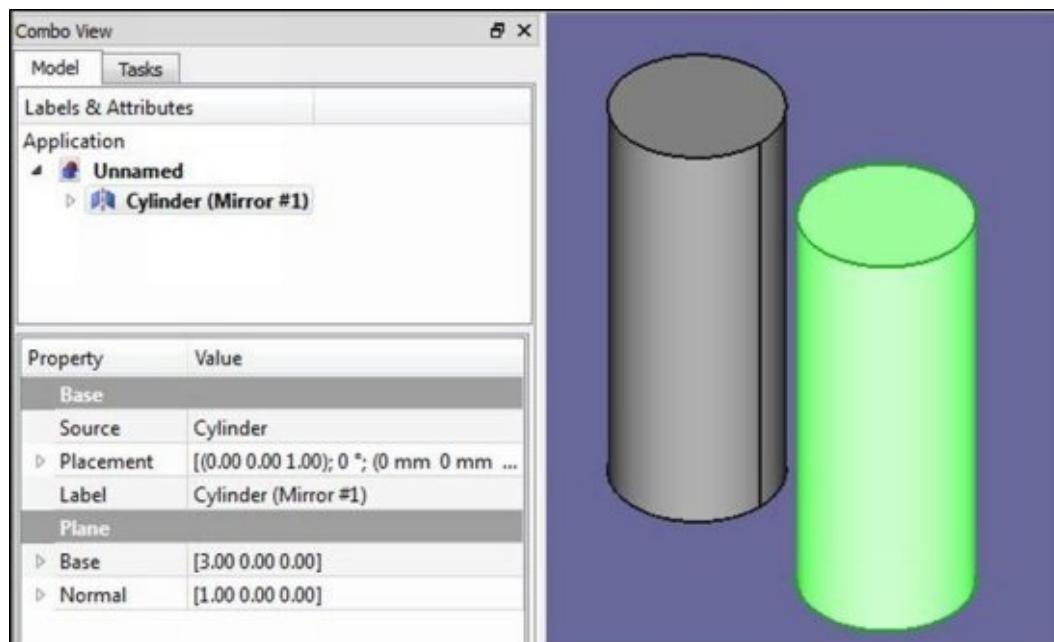


Fig. -2

- PROPERTIES

:- The properties value tab are used for modification of Mirror object. Select the Mirror object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For edit in shape parameters: - **Property** → **Data** → **Source**.
- b) For placement of the shape: - **Property** → **Data** → **Placement**.
- c) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

4) HOW TO USE FILLET TOOL

This tool is use to creates a fillet (round) on the selected edges of an object. A dialog allows you to choose which objects and which edges to work on .

- OPERATION

- a) Draw an object like Cylinder, Cube *etc.* in current working plane.
- b) Click on the  **Fillet tool**.
- c) A Fillet dialog will open in combo view (see fig. -1).

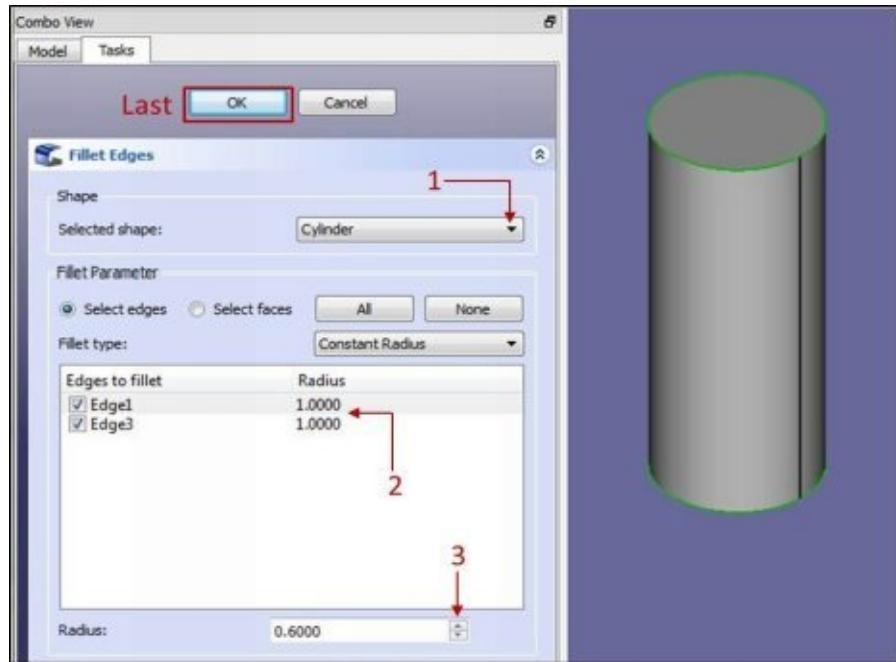


Fig. -1

- d) Select the object (cylinder) from drop down list in dialog box (see fig. -1).
- e) Then check edge(s) of object you wish to make fillet (see fig. -1).
- f) Or also select the edges form 3D view.
- g) A selected edges becomes highlighted and it will change colour to green.
- h) Set the value you want like fillet radius or see the above fig.
- i) Then click on **OK** button.
- j) A Fillet object will created in 3D view (see fig. – 2).

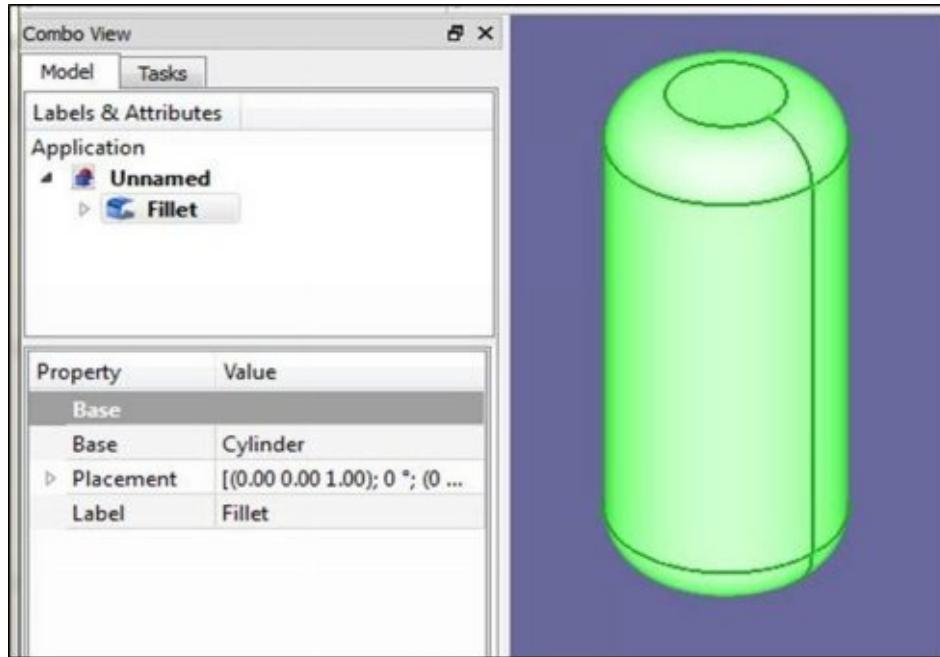


Fig. -2

- PROPERTIES

:= The properties value tab are used for modification of Fillet object. Select the Fillet object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For edit in object parameters: - **Property** → **Data** → **Base**.
- b) For placement of the object: - **Property** → **Data** → **Placement**.
- c) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

5) HOW TO USE CHAMFER TOOL

This tool is use to creates a Chamfers the selected edges of an object. A dialog allows you to choose which objects and which edges to work on.

- OPERATION

- a) Draw an object like Cylinder, Cube *etc.* in current working plane.

- b) Click on the  **Chamfer tool**.
- c) A Chamfer dialog will open in combo view (see fig. -1).

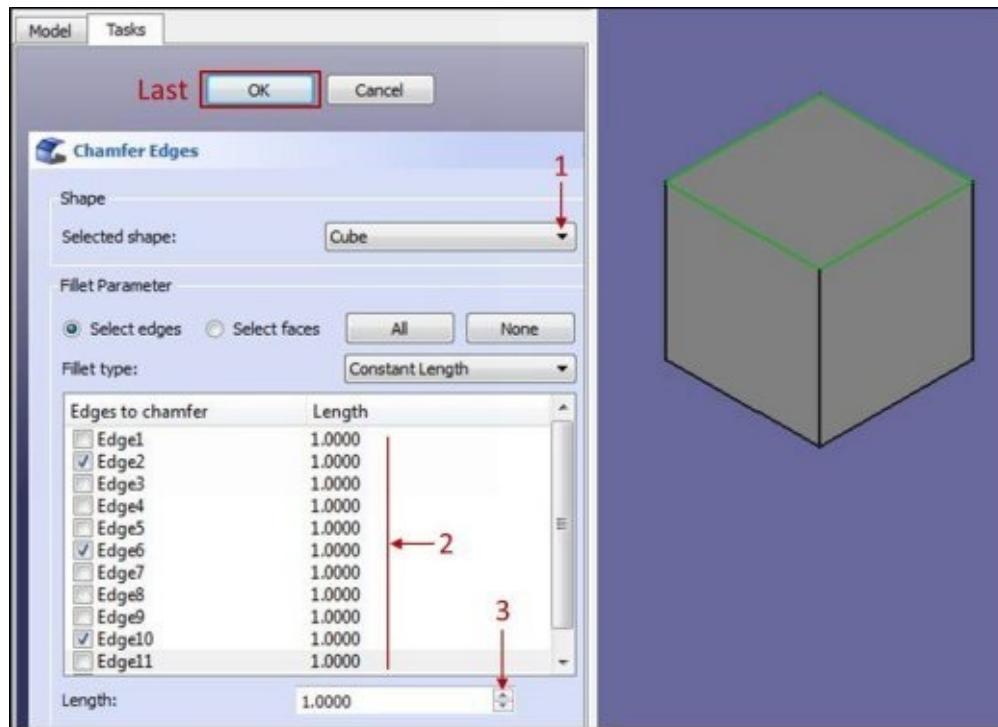


Fig. -1

- d) Select the object(cube) from drop down list in dialog box (see fig. -1).
- e) Then check edge(s) of object you wish to make chamfer (see fig. -1).
- f) Or also select the edges form 3D view.
- g) A selected edges becomes highlighted and it will change colour to green.
- h) Set the value you want or see the above fig.
- i) Then click on **OK** button.
- j) A Chamfer object will created in 3D view (see fig. – 2).

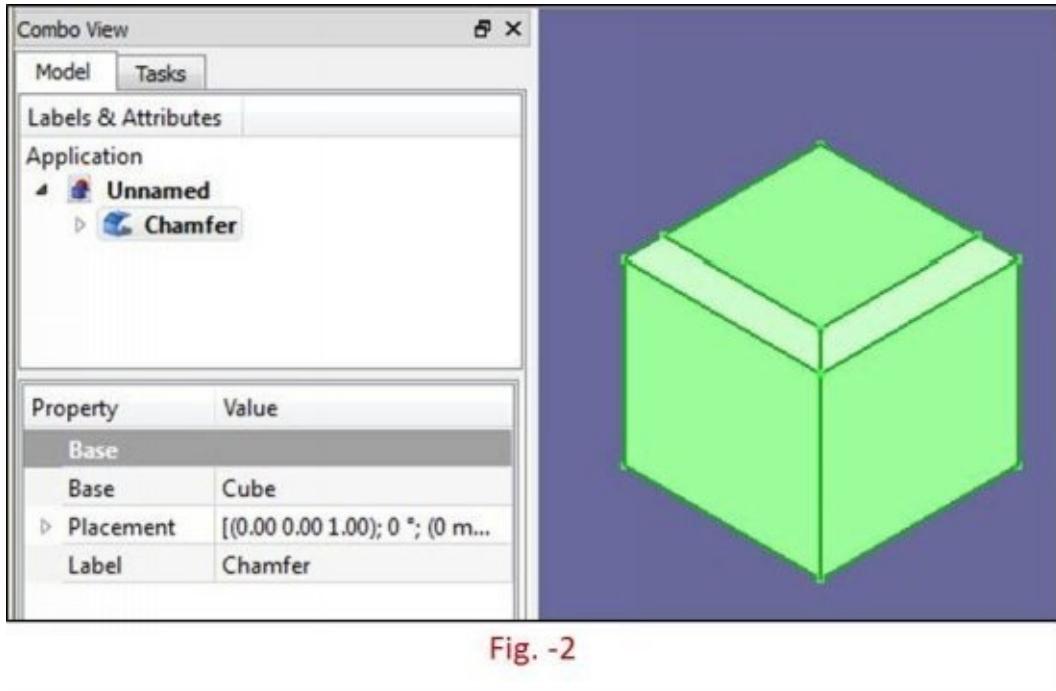


Fig. -2

- PROPERTIES

:= The properties value tab are used for modification of Chamfer object. Select the Chamfer object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For edit in object parameters: - **Property** → **Data** → **Base**.
- b) For placement of the object: - **Property** → **Data** → **Placement**.
- c) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

6) HOW TO USE RULED SURFACE TOOL

This tool is use to create a ruled surface form either two edges or two wires. By the use of this tool you can easily to make a curved plate of shapes.

- OPERATION

- a) Draw two half circle in current work plane (see fig. -1).

b) Select the first circle and press the ctrl then select the second circle (see fig. -1).

c) Now click on the  **Ruled surface tool**.

d) A Ruled surface object will created in 3D view (see fig. - 2).

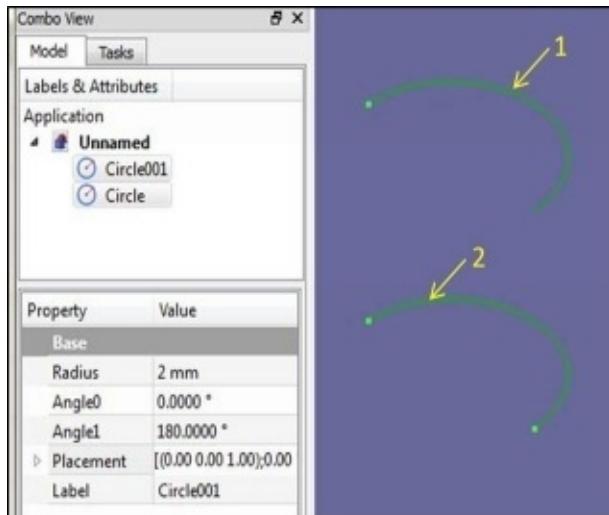


Fig. -1

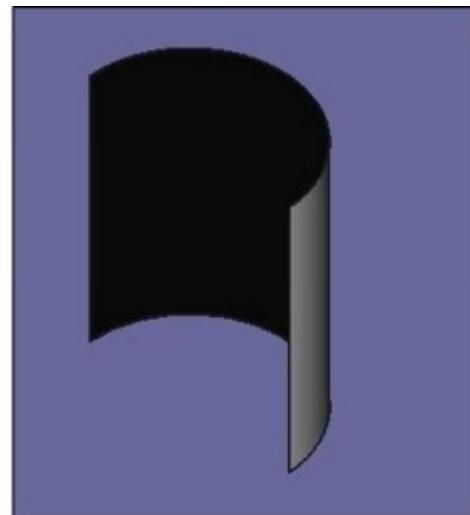


Fig. -2

• PROPERTIES

:- The properties value tab are used for modification of Chamfer object. Select the Chamfer object from the Combo view → Model and then put the value what ever you want, in Property Value.

- For placement of the object: - **Property** → **Data** → **Placement**.
- For orientation: - **Property** → **Data** → **Orientation**.
- For Shape color, Draw style, Transparency etc: - **Property** → **View**.

7) HOW TO USE LOFT TOOL



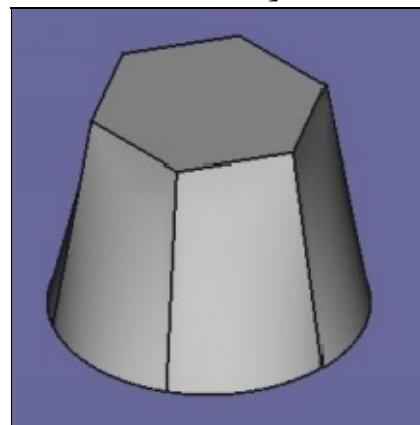
The Loft tool is use to create a Face, Shell or a solid shape from two or more profiles. In this, the profiles can be vertex, line or edge, wire or face. Edges and wires may be either closed or open.

- OPERATION

Follow the **video tutorial link** for how to use Loft tool.

<https://youtu.be/GzrxU8OmD9c>

[HOW TO USE LOFT TOOL IN FreeCAD]



8) HOW TO USE SWEEP TOOL



The Sweep tool is used to create a face, shell or a solid shape from one or more profiles, projected along a path. The Part Sweep tool is similar to **Part Loft** with the addition of a path to define the projection between profiles.



- OPERATION

Follow the **video tutorial link** for how to use Sweep tool.

<https://youtu.be/tmYy54CtOtk>

[HOW TO MAKE SOLID SPRING BY THE USE OF SWEEP TOOL IN FreeCAD]

9) HOW TO USE PART OFFSET TOOL

The Part Offset tool is used to create copies of a selected shape at a certain distance from the base shape.

- OPERATION

- a) Select the part object (cylinder) from the combo view or from the 3D view (see fig. -1).
- b) A selected object becomes highlighted and it will change colour to green.

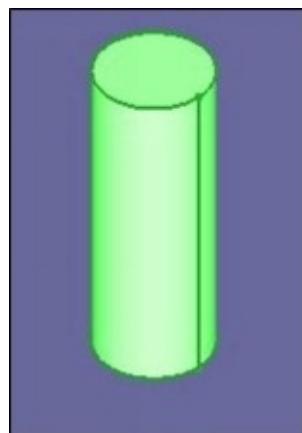
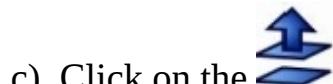


Fig. -1



- c) Click on the **Part Offset tool**.
- d) A Offset dialog will created in combo view (see fig. – 2).

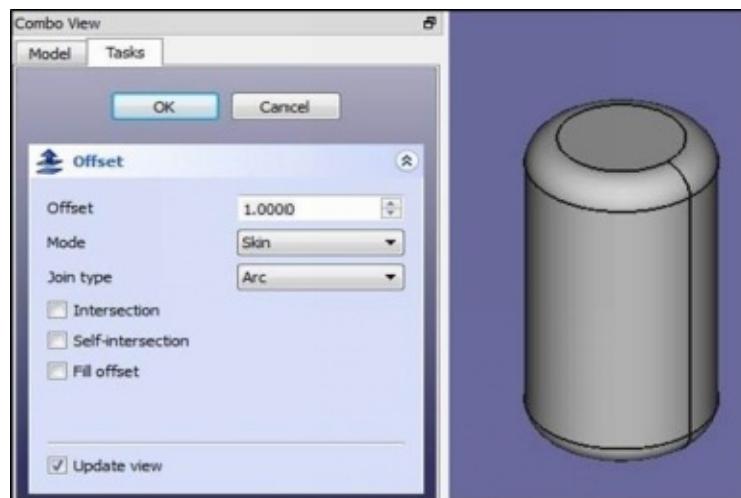


Fig. -2

-
- e) A part offset object will also created in 3D view (see fig. – 2).
 - f) Set the value you want or see the above fig.
 - g) Then click on **OK** button to exit the operation.
 - h) Give the some placement to the offset object in Y direction to see the Part Offset (see fig. – 3).

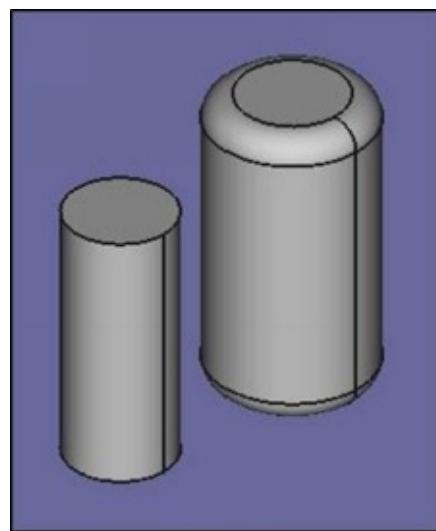


Fig. -3

- PROPERTIES

:- The properties value tab are used for modification Part offset. Select the Part offset from the Combo view → Model and then put the value what ever you want, in Property Value.

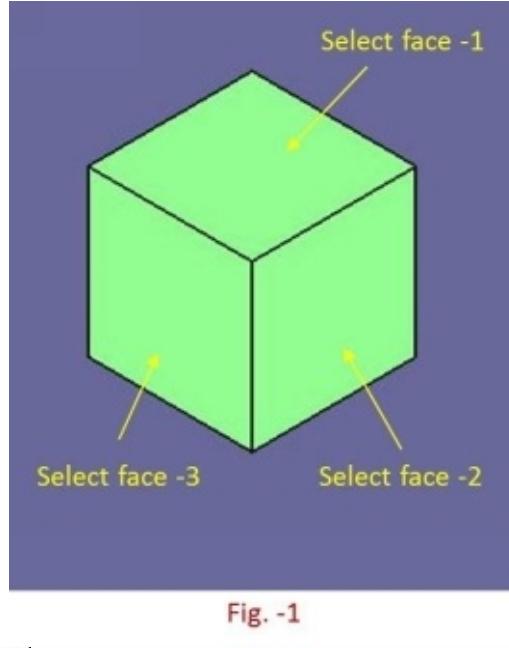
- d) For placement of the object: - **Property** → **Data** → **Placement**.
- e) For edit in object parameters: - **Property** → **Data** → **Source**.
- f) For offset value: - **Property** → **Data** → **Value**.
- g) For offset mode: - **Property** → **Data** → **Mode**.
- h) For join type: - **Property** → **Data** → **Join**.
- i) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

10) HOW TO USE THICKNESS TOOL

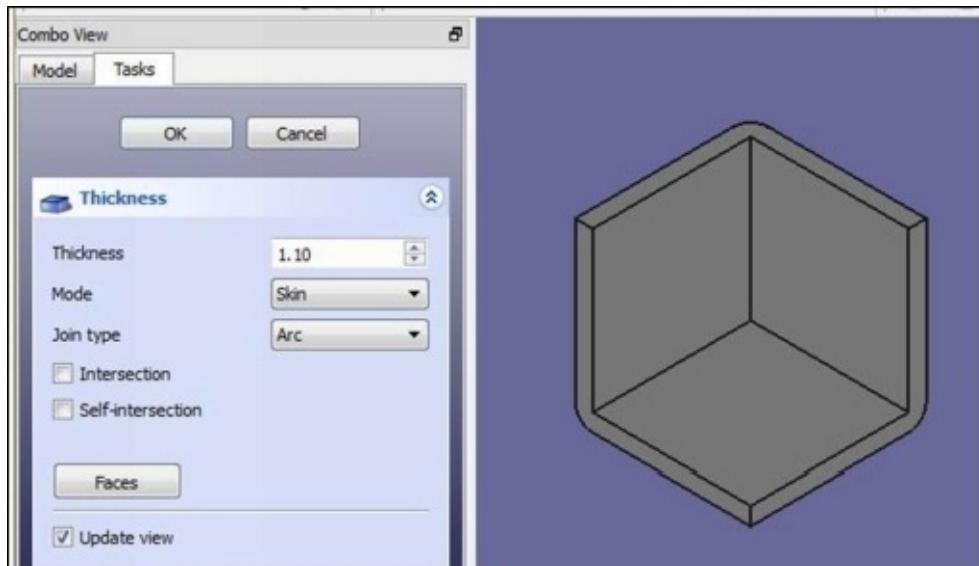
The **Thickness** tool works on a solid shape and transforms it into a hollow object, giving to each of its faces a defined thickness. On some solids it allows you to significantly speed up the work, and avoids making extrusions and pockets.

- OPERATION

- a) Create a solid object like Cube.
- b) Select one or more faces of object from the 3D view (see fig. -1).
- c) A selected object becomes highlighted and it will change colour to green.



- d) Click on the  Thickness tool.
- e) A Thickness dialog will created in combo view (see fig. – 2).



- f) A Thickness object will also crated in 3D view (see fig. – 2).
- g) Set the value you want or see the above fig.
- h) Then click on **OK** button to exit the operation.

- **PROPERTIES**

:- The properties value tab are used for modification Part offset. Select the Part offset from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For placement of the Thickness: - **Property** → **Data** → **Placement**.
- b) For thickness value: - **Property** → **Data** → **Thickness**.
- c) For thickness mode: - **Property** → **Data** → **Mode**.
- d) For join type: - **Property** → **Data** → **Join**.
- e) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

END

10. PART DESIGN WORKBENCH



: - The **Part Design Workbench** is used to create a Single, Connected solid and Multiple solids. This workbench provides tools for modelling complex solid parts in 3D environment. This workbench is most useful and helpful to make a modification in solid part module. This workbench is also used to draw 2D sketch in 3D work plane and on selected face. The Part Design workbench is based on a **Feature editing methodology** to produce a single contiguous solid.

- Tools In Part Design

The Part Design tools are all located in the **Part Design** menu that appears when you load the Part Design module. In part design workbench, the tools are divided into two categories.

1. SKETCHER TOOLS

- a) Sketcher Geometries Tools
- b) Sketcher Constraints Tools
- c) Other Tools

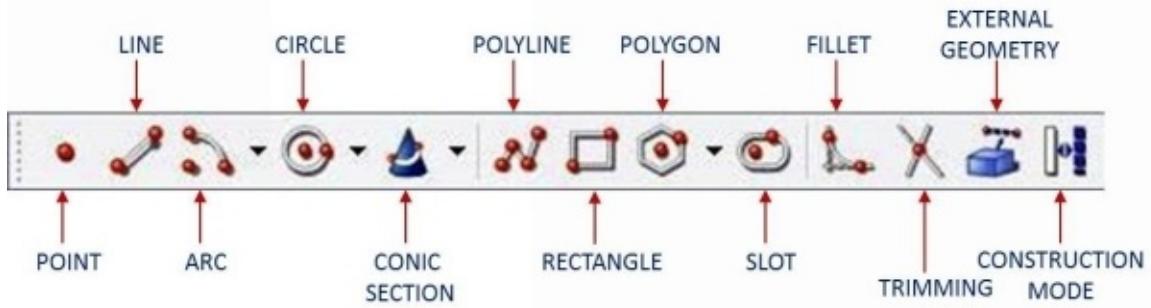
2. PART DESIGN TOOLS

- a) Construction Tools
- b) Modification Tools
- c) Transformation Tools
- d) Extra Tools

1. SKETCHER TOOLS

a) Sketcher Geometries Tools The Sketcher geometries tools are used to create the 2D object in 3D work plane.

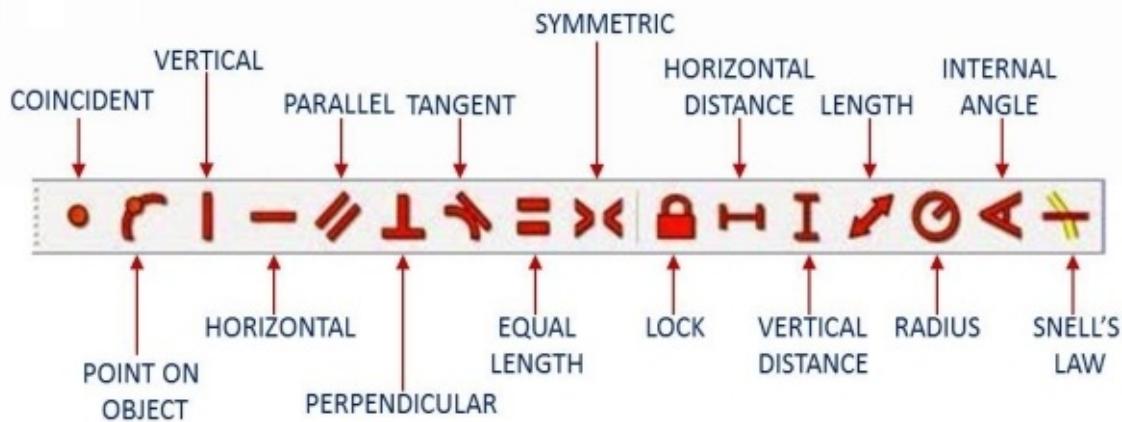
Sketcher Geometries Tools



NOTE: -The descriptions of **Sketcher Geometries Tools** are already discussed in chapter number – 6 (SKETCHER WORKBENCH).

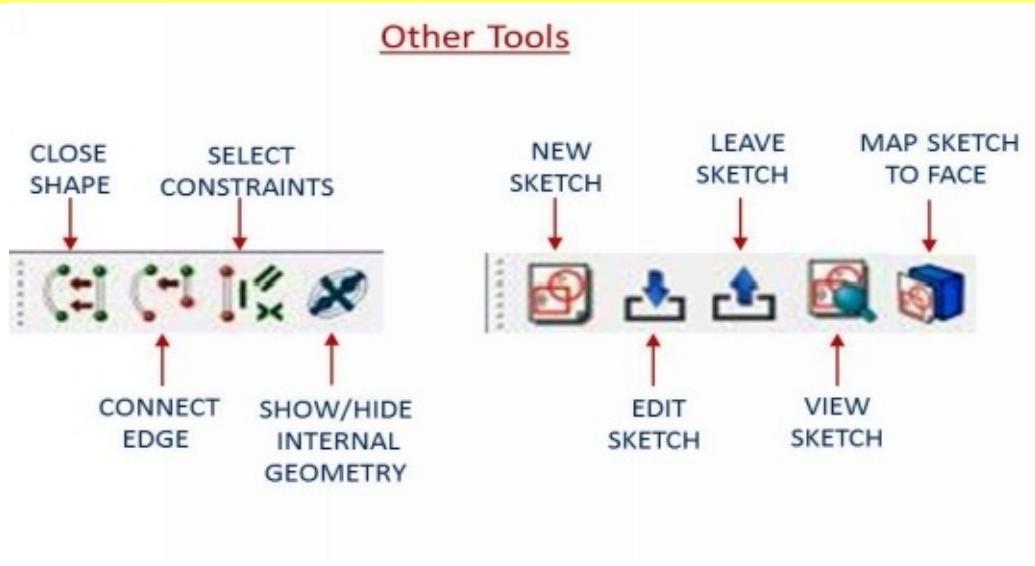
b) **Sketcher Constraints Tools** The Sketcher Constraints tools are used to define lengths, set rules between sketch elements, and to lock the sketch along the vertical and horizontal axes.

Sketcher Constraints Tools



NOTE: -The descriptions of **Sketcher Constraints Tools** are already discussed in chapter number – 6 (SKETCHER WORKBENCH).

c) Other Tools The other tools are used for some extra operation during sketch.

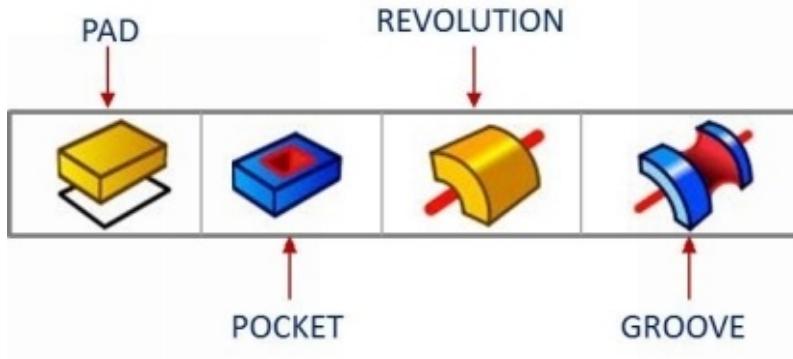


NOTE: -The descriptions of **Sketcher Constraints Tools** are already discussed in chapter number – 6 (SKETCHER WORKBENCH).

2. PART DESIGN TOOLS

a) Construction Tools Construction tools are used for creating solid objects or removing material from an existing solid object .

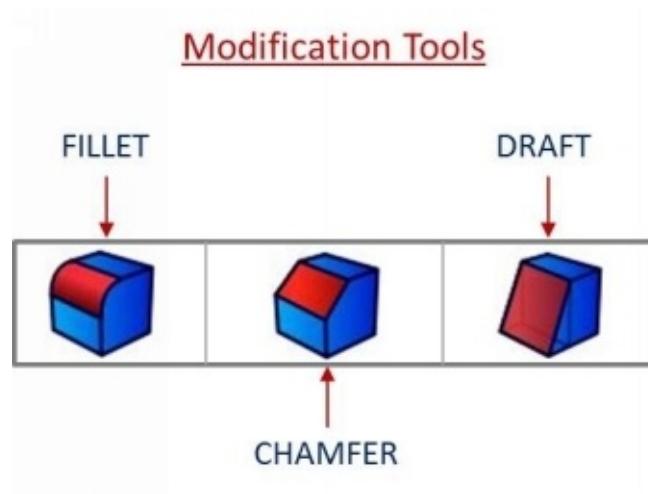
Construction Tools



The descriptions of Construction Tools are listed below:-

- **PAD** : - This tool is used to extrude a solid object from a selected sketch.
- **POCKET** : - This tool is used to create a pocket from a selected sketch. The sketch must be mapped to an existing solid object's face.
- **REVOLUTION** : - This tool is used to create a solid by revolving a sketch around an axis. The sketch must be a closed profile to get a solid object.
- **GROOVE** : - This tool is used to create a groove by revolving a sketch around an axis. The sketch must be mapped to an existing solid object's face.

b) Modification Tools Modification tools are used for modifying existing objects. They will allow you to choose which object to modify.

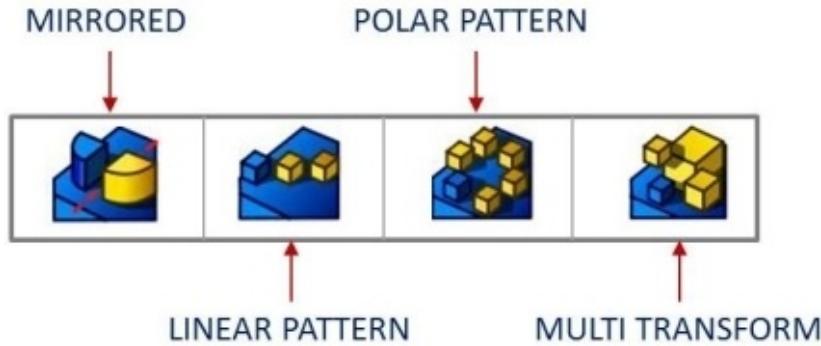


The descriptions of **Construction Tools** are listed below:-

-  **FILLET:** - This tool is used to make a fillet (round) edge(s) of an object.
-  **CHAMFER:** - This tool is used to create a chamfer edge(s) of an object.
-  **DRAFT:** - This tool is used to make an angular draft to face of an object.

c) **Transformation Tools** The transformation tool are used for transforming existing features. They will allow you to choose which features to transform.

Transformation Tools



The descriptions of Construction Tools are listed below:-

- **MIRRORED:** - This tool is used to Mirror features on a plane or face.
- **LINEAR PATTERN:** - This tool is used to create a linear pattern of features.
- **POLAR PATTERN:** - This tool is used to create a polar pattern of features.
- **MULTI TRANSFORM:** - This tool is used to create a pattern with any combination of the other transformations.

d) Extra Tools In Part Design Workbench Some optional functionality has been also created by the Extra tools.

In this workbench the extra tools are as follows:

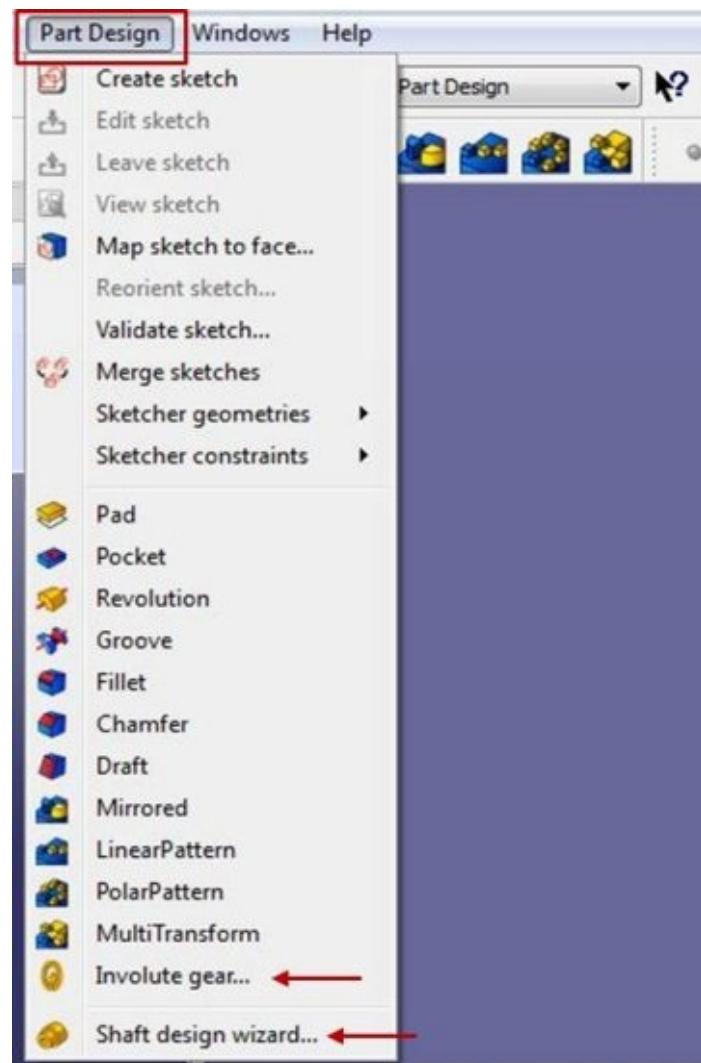
-  **INVOLUTE GEAR** : - This tool allows you to create gear.
-  **SHAFT DESIGN WIZARD**: - This tool is used to create a shaft from a table of values and allows to analyse forces and moments.

Extra Tools



NOTE :-

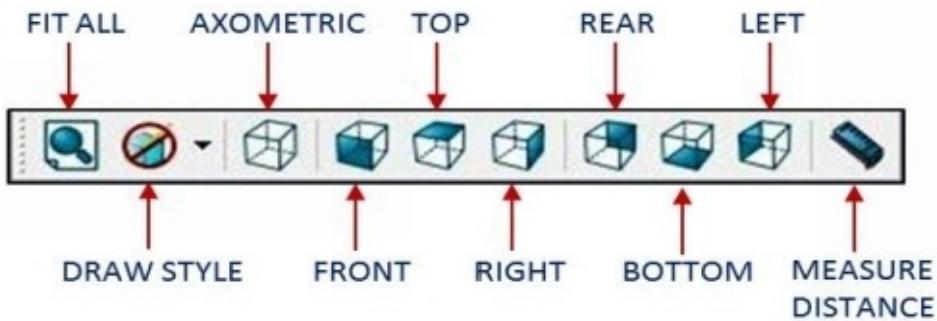
In this workbench, the Extra tools is located in Part Design tab. Click on **Part Design** tab and select the Extra tools wish you want.



VIEW TOOLS

This tools are very usefull for showing the object in different location. This tools are always available in all workbenches.

View Tools



NOTE: -

The description of this command is already discussed in **chapter Number – 4.**

END

11. OPERATION WITH - PART DESIGN



The **Part Design Workbench** is one of the most important workbench in FreeCAD. In the Part Design workbench, a very important concept is that, it has **sketch support**. Sketches can be created on standard planes (**XY**, **XZ**, **YZ** and planes parallel to them) or on the face of an existing solid. For this last case, the existing solid becomes the support of the sketch. Several tools will only work with sketches that have a support, for example, **Pocket** - without a support there would be nothing to remove material from.

In this chapter, we will discussed that how to use the different tools with the help of examples.

NOTE: :-

- How to use Sketcher tools in part design workbench is already discussed in Chapter – 6 (OPERATION WITH – SKETCHER). You know that the Part Design Workbench is the combination of **Sketcher Workbench** and **Part Workbench** so there is no need to explain once more in this chapter also.

WORKING WITH PART DESIGN TOOLS

1. Working With Construction Tools:

1.1 HOW TO USE PAD TOOL



The Pad tool is use to extrudes a solid object form 2D sketch. In this the **Pad** tool takes a selected sketch as its input and from it produces a "pad" feature. A pad is essentially an extrusion of a sketch into a solid.

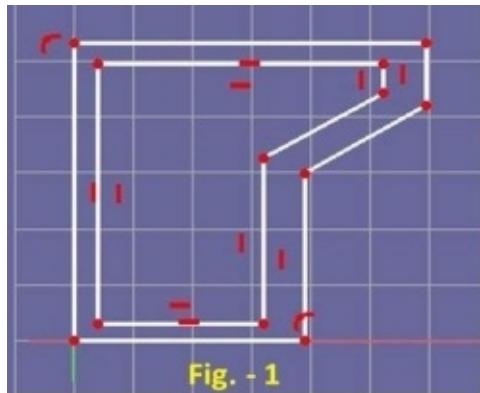


Fig. - 1

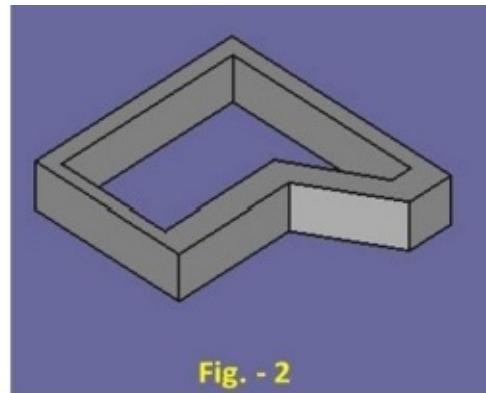
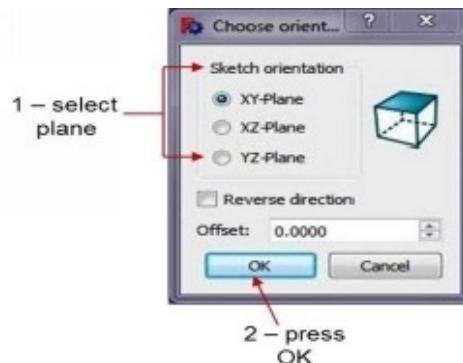


Fig. - 2

- OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the **Part Design**.
- c) Click on new sketch tool.
- d) Then select the Plane(XY, XZ, YZ) from Choose orientation window.



- e) You can change an offset to any of the three planes and the side of the offset.
- f) Make an one sketch in 3D work plane and hit the Close button to exit the sketches function (see Fig. – 1).
- g) Then select the sketch from combo view.
- h) Now press the **Pad** tool.
- i) A Pad parameters dialog box will open in Combo view and a Pad object

will created in 3D work plane (see Fig. – 2).

j) Set the Pad parameters as per requirement.

k) click on OK button to exit the function.

- OPTIONS

The description of **Pad parameters** are as follows: a) **Type**: - Type offers five different ways of specifying the length to which the pad will be extruded.

- **Dimension**: - Enter a numeric value for the length of the pad.
- **Two dimensions**: - This allows to enter a second length in which the pad should extend in the opposite direction (into the support).
- **To last** : - The pad will extrude up to the last face of the support in the extrusion direction. If there is no support, an error message will appear
- **To first**: - The pad will extrude up to the first face of the support in the extrusion direction. If there is no support, an error message will appear.
- **Up to face** : - The pad will extrude up to a face in the support that can be chosen by clicking on it. If there is no support, no selections will be accepted.



- b) **Length** : - Defines the length of the pad. Multiple units can be used independently of the user's units preferences (m, cm, mm, nm, ft or ', in or ").
- c) **Symmetric to plane**: - Tick the checkbox to extend half of the given length to either side of the sketch plane.
- d) **Reversed**: - Reverses the direction of the pad.

- **PROPERTIES**

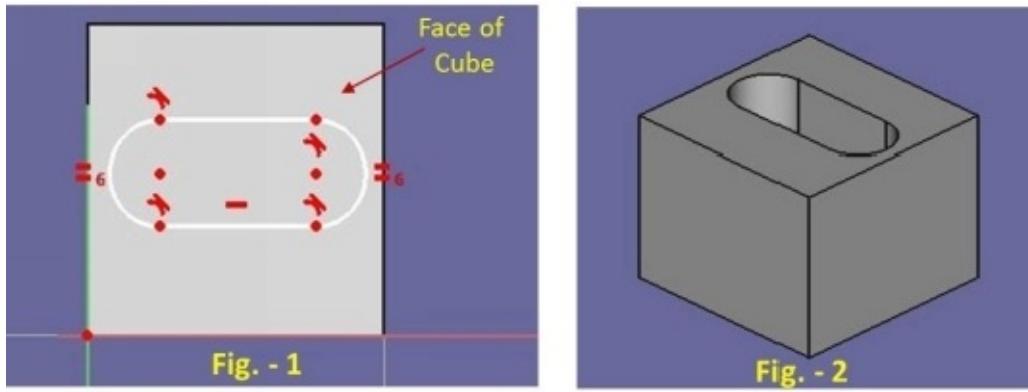
:- The properties value tab are used for modification of Pad object. Select the Pad object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For placement of the object: - **Property** → **Data** → **Placement**.
- b) For modification in sketch parameters: - **Property** → **Data** → **Sketch**.
- c) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

1.2 HOW TO USE POCKET TOOL



This tool is use to 'Create a pocket with the selected sketch'. This tool takes a selected sketch as its input, and produces with it a pocket. A pocket being essentially an extrusion of a sketch that subtracts from the geometry it protrudes into.



- OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the **Part Design**.
- c) Draw a part object (cube, Cylinder etc.) form Combo View → Task or form Part workbench.
- d) Then select the face of part object where you want to make sketch.
- e) A selected Face becomes highlighted and it will change colour to green.
- f) Click on new sketch tool.
- g) Make a sketch on the face of part object and hit the Close button to exit the sketches function (see Fig. – 1).
- h) Then select the sketch from combo view.
- i) Now press the **Pocket** tool.
- j) A **Pocket parameters** dialog box will open in Combo view and a Pocket object will created in 3D work plane (see Fig. – 2).
- k) Set the Pocket parameters as per requirement.
- l) click on OK button to exit the function.

- OPTIONS

The description of **Pocket parameters** are as follows:

- a) **Type:** - Type offers four different ways of specifying the length (depth) to which the pocket will be extruded.

- **Dimension:** - Enter a numeric value for the depth of the pocket.
The default direction for extrusion is into the support.
- **Through all:** - The pocket will cut through all material in the extrusion direction. With the option Symmetric to plane the pad will cut through all material in both directions.
- **To first:** - The pocket will extrude up to the first face of the support in the extrusion direction. In other words, it will cut through all material until it reaches an empty space.
- **Up to face :** - The pocket will extrude up to a face in the support that can be chosen by clicking on it.



- b) **Length :** - Defines the depth of the Pocket. Multiple units can be used independently of the user's units preferences.
- c) **Symmetric to plane:** - Tick the checkbox to extend half of the given depth to either side of the sketch plane.
- d) **Reversed:** - Reverses the direction of the pocket.

- PROPERTIES

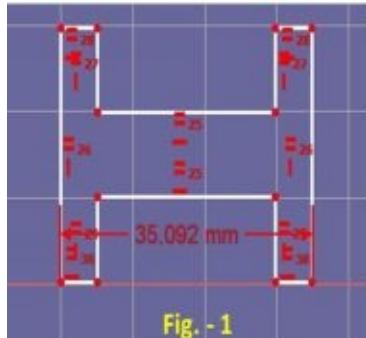
:- The properties value tab are used for modification of Pocket object. Select the Pocket object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For placement of the object: - **Property** → **Data** → **Placement**.
- b) For modification in sketch parameters: - **Property** → **Data** → **Sketch**.
- c) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

1.3 HOW TO USE REVOLUTION TOOL

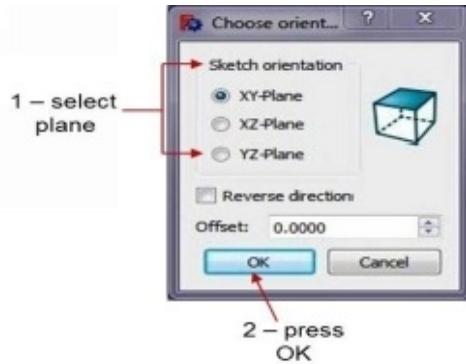


The revolution tool is use to revolves a selected sketch or 2D object about a given axis.



- OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the **Part Design**.
- c) Click on new sketch tool.
- d) Then select the Plane(XY, XZ, YZ) from Choose orientation window.

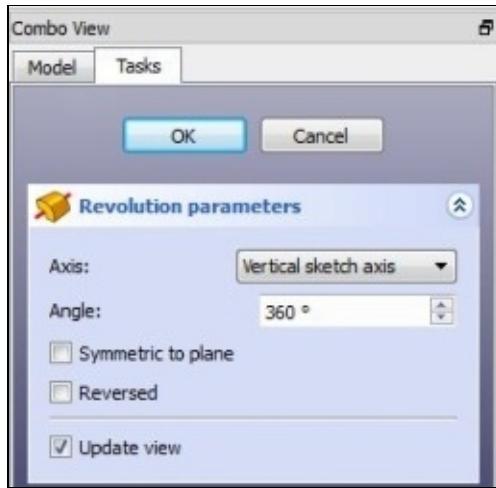


- e) You can change an offset to any of the three planes and the side of the offset.
- f) Make an one sketch in 3D work plane and hit the Close button to exit the sketches function (see Fig. – 1).
- g) Then select the sketch from combo view.
- h) Now press the  **Revolution** tool.
- i) A Revolution parameters dialog box will open in Combo view and a Revolve object will created in 3D work plane (see Fig. – 2).
- j) Set the Revolution parameters as par requirement.
- k) click on OK button to exit the function.

- **OPTIONS**

The description of **Revolution parameters** are as follows:

- a) **Axis:** - This option specifies the axis about which the sketch is to be revolved. Currently, by default only the horizontal or vertical sketch axis can be selected here.



- b) **Angle:** - This controls the angle through which the revolution is to be formed, e.g. 360° would be a full, contiguous revolution.
- c) **Symmetric to plane:** - The revolution will extend half of the specified angle in both directions from the sketch plane.
- d) **Reversed:** - The direction of revolution will be reversed.

- **PROPERTIES**

:- The properties value tab are used for modification of Revolve object. Select the Revolve object from the Combo view → Model and then put the value what ever you want, in Property Value.

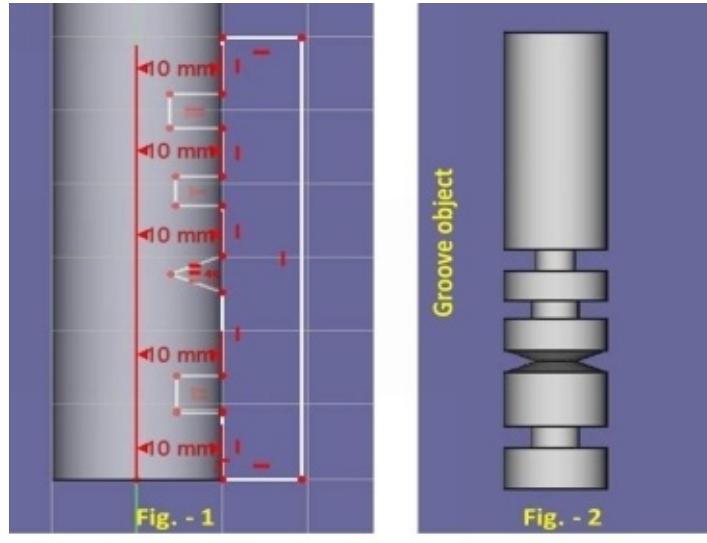
- a) For placement of the object: - **Property → Data → Placement.**
- b) For modification in sketch parameters: - **Property → Data → Sketch.**
- c) For Shape color, Draw style, Transparency etc: - **Property → View.**

1.4 HOW TO USE GROOVE TOOL



This tool is use to revolves a selected sketch or 2D object about a given axis and cutting out material from the support. During operation when creating a

groove, the 'groove parameters' dialogue offers several parameters specifying how the sketch should be revolved. They have exactly the same meaning as for the **revolution** feature.



- OPERATION

Follow the **video tutorial link** for how to use Groove tool.

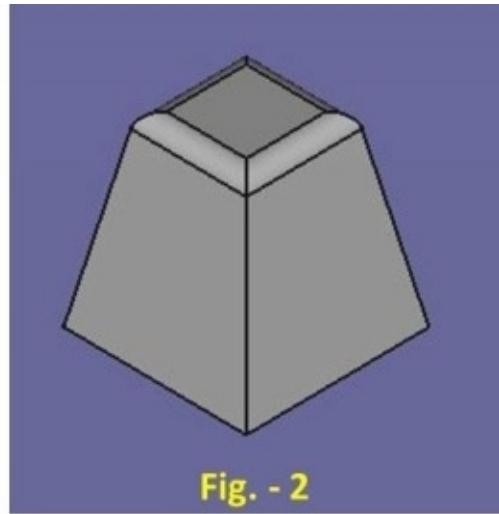
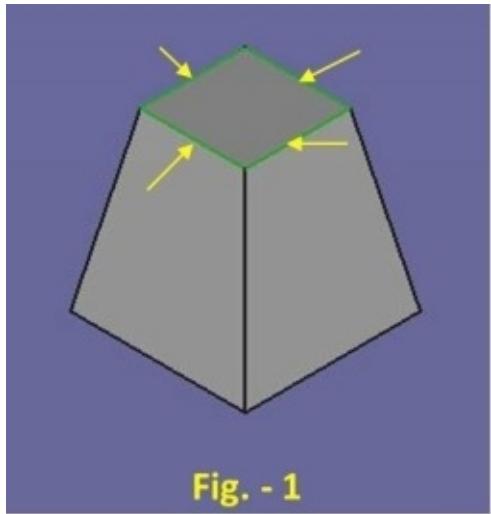
<https://youtu.be/cHuXl7evbOs>

[HOW TO USE GROOVE TOOL IN PART DESIGN WORKBENCH]

2. Working With Modification Tools:

2.1 HOW TO USE FILLET TOOL

 The **Fillet** tool is used to create fillets (rounds) on the selected edges of an object. A new separate Fillet entry (followed by a sequential number if there are already existing fillets in the document) is created in the Project tree.



- OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the **Part Design**.
- c) Draw a part object (cube, Cylinder etc.) from Combo View → Task or from Part workbench.
- d) Then select a single or multiple edges on an object.
- e) A selected edge(s) becomes highlighted and it will change colour to green (see Fig. – 1).

- f) Now press the  **Fillet** tool.
- g) A **Fillet parameters** dialog box will open in Combo view and a Fillet object will be created in 3D work plane (see Fig. – 2).
- h) Then set the fillet radius either by entering the value, or by clicking on the up/down arrows.
- i) click on OK button to exit the function.

- PROPERTIES

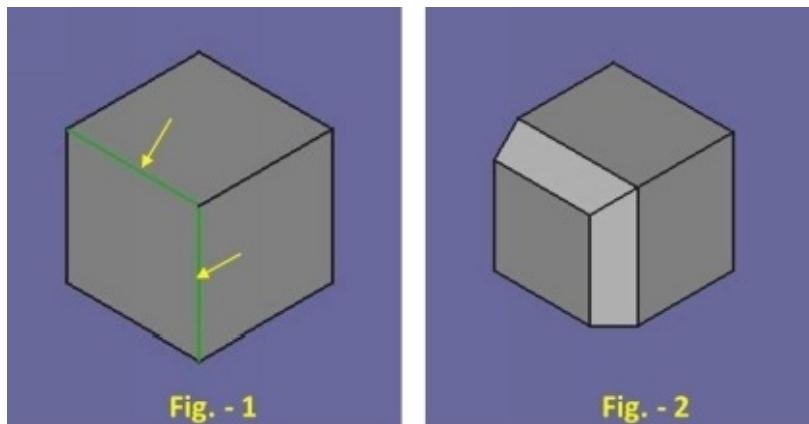
:- The properties value tab are used for modification of Fillet object. Select the Fillet object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For Fillet radius: - **Property** → **Data** → **Radius**.
- b) For placement of the object: - **Property** → **Data** → **Placement**.
- c) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

2.2 HOW TO USE CHAMFER TOOL



This tool is use to make chamfers edge(s) of an object.



- **OPERATION**

- a)** Run or open the FreeCAD software.
- b)** Switch the Workbench and select the **Part Design**.
- c)** Draw a part object (cube, Cylinder etc.) form Combo View → Task or form Part workbench.
- d)** Then select a single or multiple edges on an object.
- e)** A selected edge(s) becomes highlighted and it will change colour to green

(see Fig. – 1).

- f) Now press the  **Chamfer** tool.
- g) A **Chamfer parameters** dialog box will open in Combo view and a Fillet object will created in 3D work plane (see Fig. – 2).
- h) Then set the Chamfer size either by entering the value, or by clicking on the up/down arrows.
- i) click on OK button to exit the function.

- **PROPERTIES**

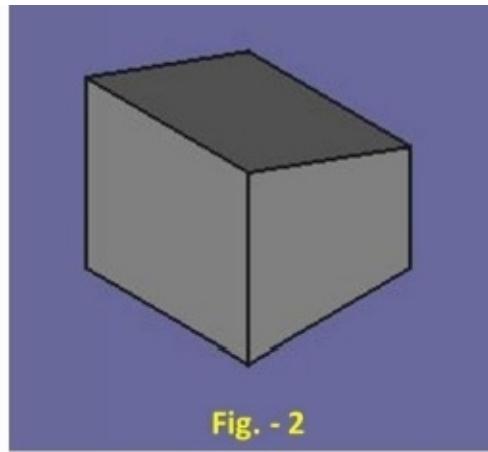
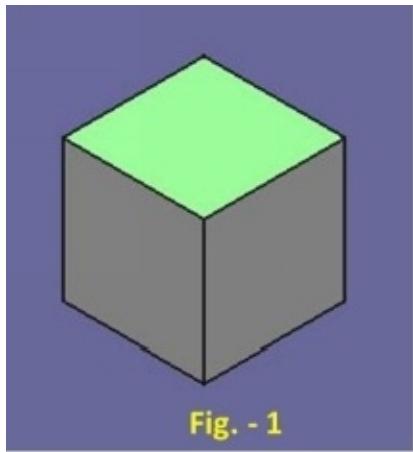
:– The properties value tab are used for modification of Chamfer object. Select the Chamfer object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For Chamfer size: - **Property** → **Data** → **Size**.
- b) For placement of the object: - **Property** → **Data** → **Placement**.
- c) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

2.3 HOW TO USE DRAFT TOOL



This tool is use to creates an angular draft on the selected faces of an object. A new separate Draft entry (followed by a sequential number if there are already existing drafts in the document) is created in the Project tree.



- OPERATION

- Run or open the FreeCAD software.
- Switch the Workbench and select the **Part Design**.
- Draw a part object (cube, Cylinder etc.) from Combo View → Task or from Part workbench.
- Select one or more faces on an object.
- A selected edge(s) becomes highlighted and it will change colour to green (see Fig. – 1).

- Now press the  **Draft** tool.
- A Draft parameters dialog box will open in Combo view.
- Set the Draft parameters as per requirement.
- click on OK button to exit the function.

- OPTIONS

The description of **Drft parameters** are as follows:

- Add face/Remove face:** - Click Add Face or Remove Face, then select a single face to update the list of active faces. Repeat as needed.



- b) **Draft angle:** - Set the Draft Angle by entering a value or by clicking on the up/down arrows. The applied draft angle is shown in real time.
- c) **Neutral plane:** - Click Neutral Plane, then select the plane that must not change dimensionally. The change is made in real time.
- d) **Pull direction:** - Click Pull Direction, then select an edge. Pull Direction is only effective if the Neutral Plane has been set. Results can be unpredictable.
- e) **Reverse pull direction:** - Checking Reverse Pull Direction will toggle the draft between positive and negative angles.

- PROPERTIES

:- The properties value tab are used for modification of Draft object. Select the draft object from the Combo view → Model and then put the value what ever you want, in Property Value.

- a) For Draft angle: - **Property → Data → Angle.**

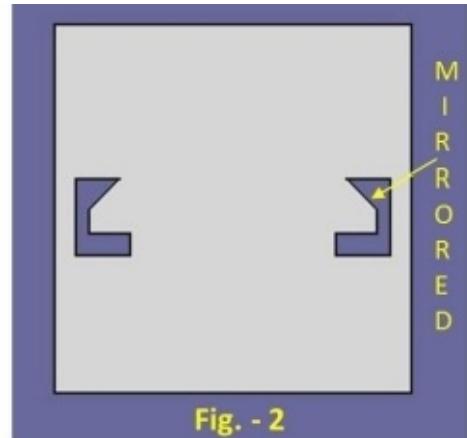
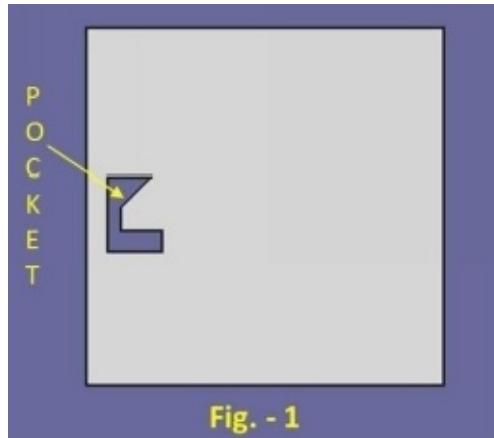
- b) For reversed direction: - **Property** → **Data** → **Reversed** (false/true).
- c) For placement of the object: - **Property** → **Data** → **Placement**.
- d) For Shape color, Draw style, Transparency etc: - **Property** → **View**.

3. Working With Transformation Tools:

3.1 HOW TO USE MIRRORED TOOL

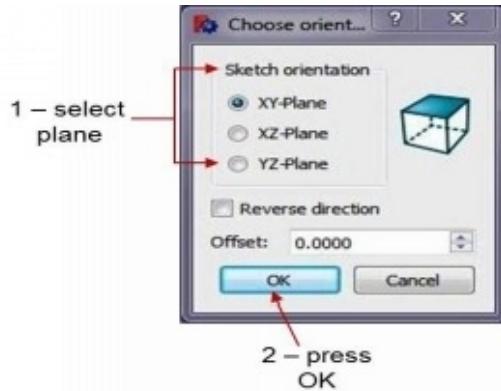


This tool is used to create a set of one selected features as its input (the 'original'), and produces with it a second set of features mirrored on a plane.



- OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the **Part Design**.
- c) Click on new sketch tool.
- d) Then select the Plane(XY, XZ, YZ) from Choose orientation window.

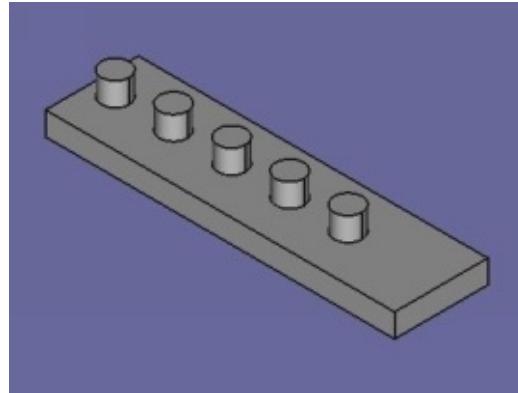


- e) Make an one sketch (rectangle) in 3D work plane and hit the Close button to exit the sketches function (see Fig. – 1).
- f) Then select the sketch from combo view.
- g) Now press the  **Pad tool** and make a pad of object.
- h) Then select the face of Pad object and click on new sketch tool.
- i) Make a sketch on that face whatever you want.
- j) Then select the sketch001 form combo view and click on  **Pocket tool**.
- k) A pocket will created on the face of pad object (see Fig. -1).
- l) Now select the pocket object and click on  **Mirrored tool**.
- m) A Mirrored parameters will open in combo view.
- n) Set the plane of mirrored for drop down list and hit OK button to exit the function.
- o) A mirrored object will created on 3D environment (see Fig.-2).

3.2 HOW TO USE LINEAR PATTERN TOOL



This tool is use to creates a set of one or more selected features as its input (the 'originals'), and produces with it a second set of features translated in a given direction.



- OPERATION

Follow the **video tutorial link** for how to use Linear pattern tool.

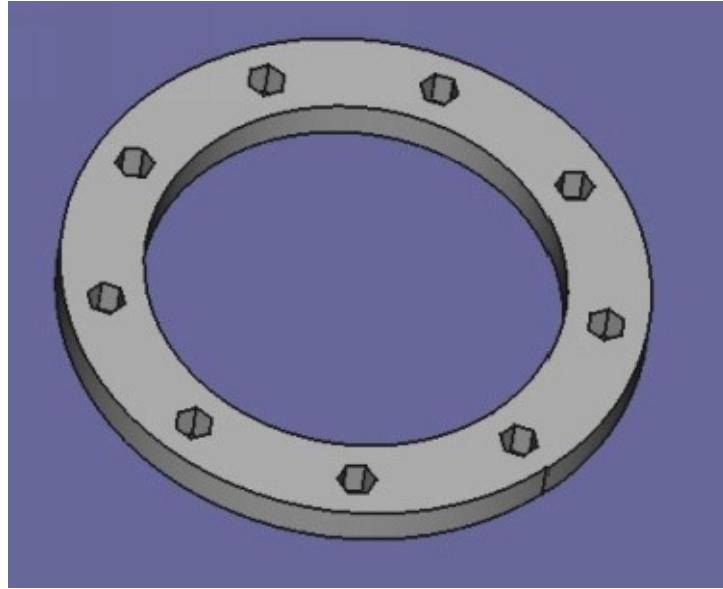
<https://youtu.be/NJsUTG4TSzk>

[HOW TO WORK WITH TRANSFORMATION TOOL IN PART DESIGN WORKBENCH]

3.3 HOW TO USE POLAR PATTERN TOOL



This tool is used to create a set of one or more selected features as its input (the 'originals'), and produces with it a second set of features rotated around a given axis.



- OPERATION

Follow the **video tutorial link** for how to use Polar pattern tool.

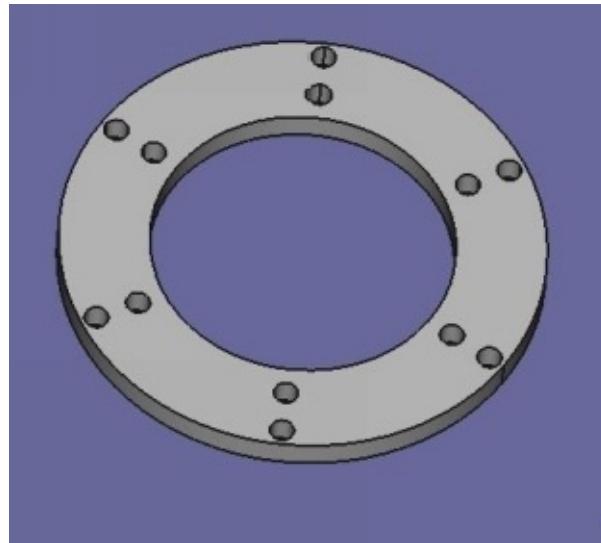
<https://youtu.be/NJsUTG4TSzk>

[HOW TO WORK WITH TRANSFORMATION TOOL IN PART DESIGN WORKBENCH]

3.4 HOW TO USE MULTI-TRANSFORM TOOL



This tool takes a set of one or more selected features as its input (the 'originals'), and allows to apply several transformations in sequence to them. For example, to produce a flange with a double row of holes, the hole (the 'original') is first patterned in a linear pattern in the X direction, and then patterned eight times in a polar pattern around the Y axis.



- OPERATION

Follow the **video tutorial link** for how to use Multi-transform tool.

<https://youtu.be/NJsUTG4TSzk>

[HOW TO WORK WITH TRANSFORMATION TOOL IN PART DESIGN WORKBENCH]

4. Working With Extra Tools:

4.1 HOW TO USE INVOLUTE GEAR TOOL

 This tool is used to create a 2D profile of an involute gear. This 2D profile is fully parametric, and can be padded with the Part Design Pad feature.

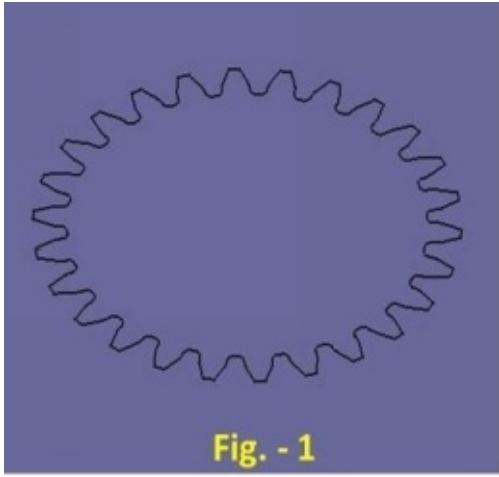


Fig. - 1

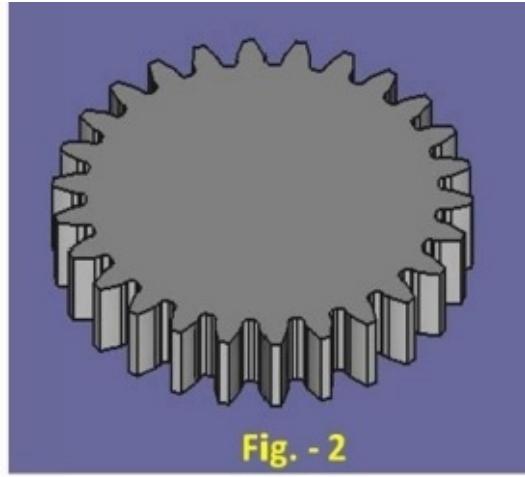


Fig. - 2

- OPERATION

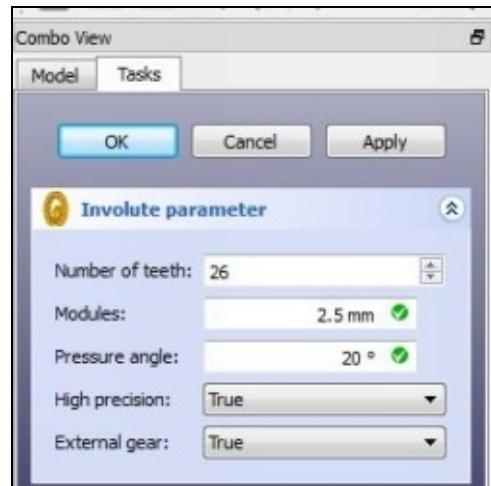
- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the **Part Design**.
- c) Then go to the Part Design menu and select the  **Involute Gear tool**.
- d) A Involute parameters dialog box will open in Combo view.
- e) Set the Involute parameters as per requirement and .
- f) Click on OK button to exit the function.
- g) Now select the Involute gear form combo view and click on **Pad tool**.
- h) Set the Pad parameters and a 3D gear will created in work plane (see Fig. – 2).
- i) Click on OK button to exit the function.

- OPTIONS

The description of **Involute Parameters** are as follows:

- a) **Number of teeth:** - Sets the number of teeth of gear.

- b) **Modules:** - Modules is the pitch diameter divided by the number of teeth.
- c) **Pressure angle:** - Pressure angle is the acute angle between the line of action and a normal to the line connecting the gear centres. Default is 20 degrees.

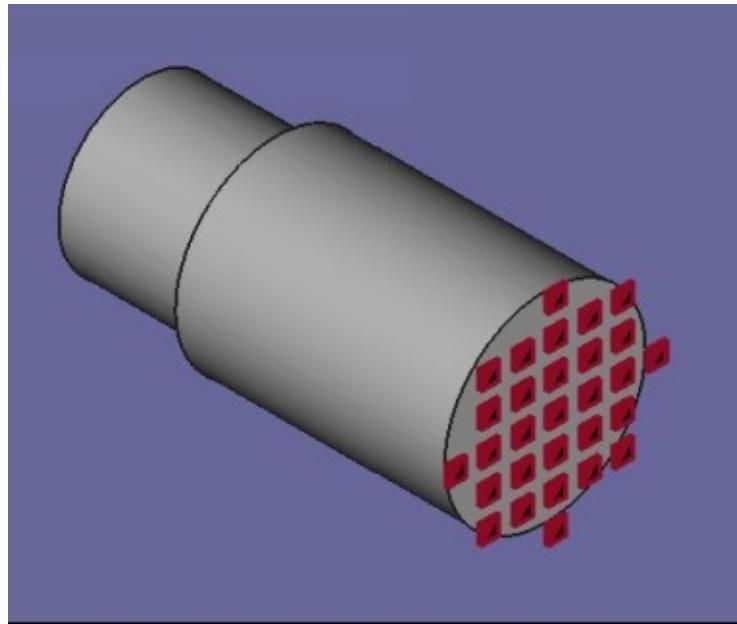


- d) **High precision:** - Make True or false.
- e) **External gear:** - Make True or false.

4.2 HOW TO USE SHAFT DESIGN WIZARD



This tool is used to create a shaft from a table of values, and to analyse forces and moments. You can start the wizard from the Part Design menu or by typing.

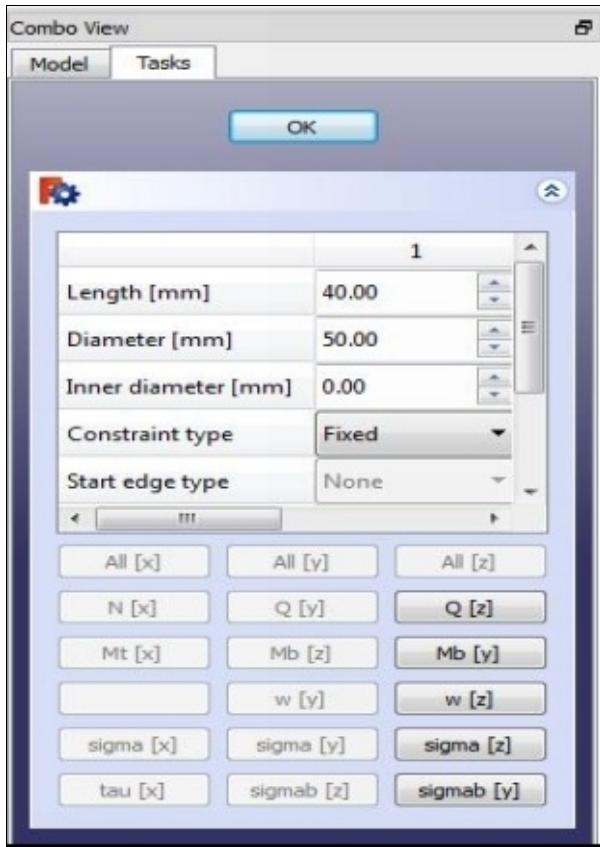


- OPERATION

- a) Run or open the FreeCAD software.
- b) Switch the Workbench and select the **Part Design**.
- c) Then go to the Part Design menu and select the  **Shaft design wizard**.
- d) A Shaft design parameters dialog box will open in Combo view.
- e) Set the parameters as per requirement and .
- f) Click on OK button to exit the function.

- OPTIONS

The description of **Shaft Design Wizard Parameters** are as follows:



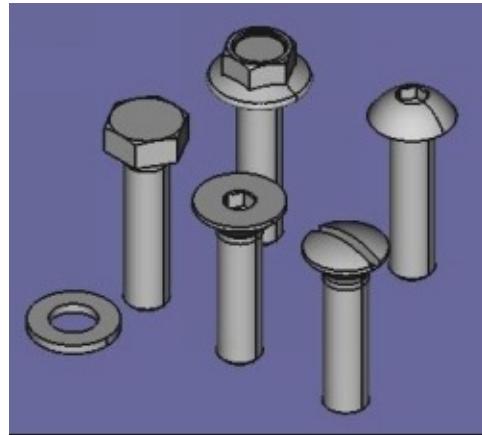
- Length of the segment
- Diameter of the segment
- Load type. Note that you have to click on the desired entry in the menu after scrolling to it, otherwise it will not be selected!
 - None: No load
 - Fixed: The end of the shaft is fixed (e.g. welded to another part). This load type can only be defined for the first or last segment.
 - Static: There is a static load on this shaft segment
- Load on the shaft segment
- Location where the load is applied to the segment. The location is counted from the left-hand edge of the segment

(Other rows and load types exist but no functionality has been implemented yet)

HOW TO MAKE SCREW WITH SCREW MAKER



This tool is used to create a screw with or without thread, according to ISO standards. The function of this tool is based on macro.



- OPERATION

- First of all you need to download the latest version of Python.
- The latest version of python is **python—2.7.10.msi** and follow the download link -
<https://www.python.org/downloads/>
- After downloading the python, click on the python setup and run it. Follow the instructions and install the python.
- Then, download the latest version of Screw Maker. The latest version of Screw Maker is screw maker 1.7.
- Follow the Link - <http://forum.freecadweb.org/viewtopic.php?f=22&t=6558#p52887> for download the zip file of screw maker.

ATTACHMENTS

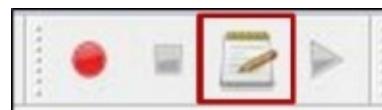
0 [screw_maker1_7.py.zip](#)
(18.83 KiB) Downloaded 4275 times

Last edited by ulrich1a on Sat Aug 08, 2015 10:13 pm, edited 1 time in total.

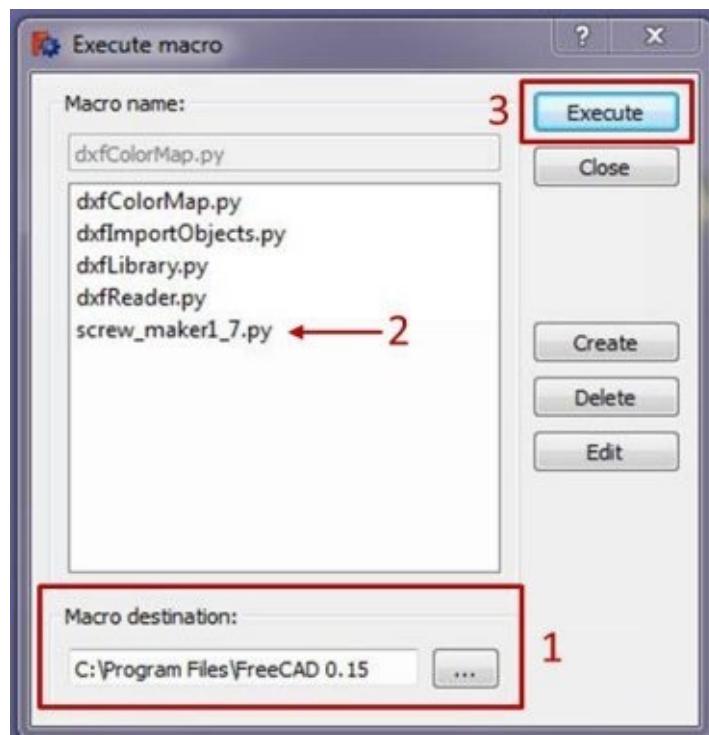
- f) Click on screw maker and download the Zip file (see above Fig.).
- g) Then Extract the zip file and get the Python file of screw maker.
- h) Copy the screw maker file and paste in Program files of FreeCAD where FreeCAD is installed (see below Fig.).



- i) Now run the FreeCAD and switch the Part design or part workbench.
- j) Then click on  **execute a recorded macro** (see the below Fig.).



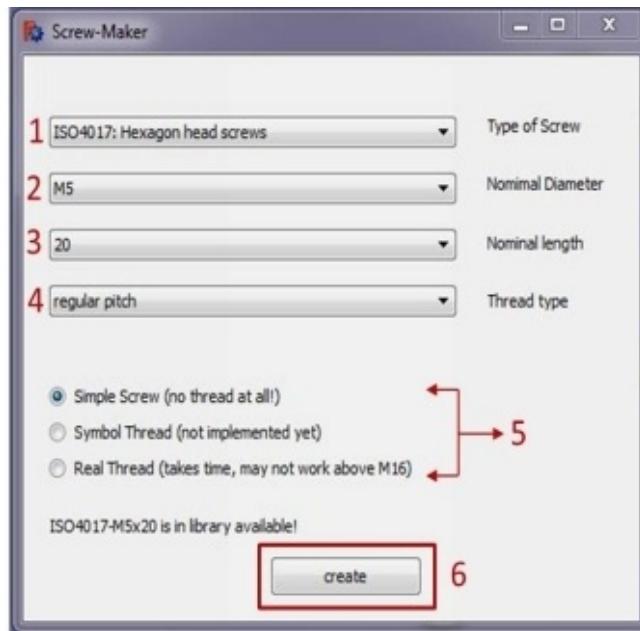
- k) A Execute macro dialogue will open (see below Fig.).



- l) First select the Macro destination where you paste the screw maker file.
- m) And then select the screw maker form Macro name and click on **Execute**

button.

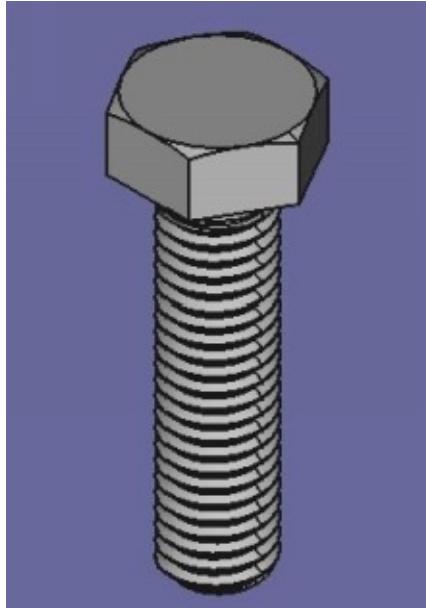
n) A Screw-Maker dialogue will open (see below Fig.).



o) Now select the characteristics of the screw form drop down list and click on the create button.

p) Then a screw or bolt or wisher of different shape and size will created in 3D view Note:-

The creation of the thread takes a long time. Be patient and have a look at the CPU-usage.



SOME ONLINE TUTORIALS

- **Basic Part Design Tutorial**

[http://www.freecadweb.org/wiki/index.php?
title=Basic_Part_Design_Tutorial&oldid=116110](http://www.freecadweb.org/wiki/index.php?title=Basic_Part_Design_Tutorial&oldid=116110)

- **Part Design Bearing holder Tutorial – I**

[http://www.freecadweb.org/wiki/index.php?
title=PartDesign_Bearingholder_Tutorial_I&oldid=108035](http://www.freecadweb.org/wiki/index.php?title=PartDesign_Bearingholder_Tutorial_I&oldid=108035)

- **Part Design Bearing holder Tutorial – II**

[http://www.freecadweb.org/wiki/index.php?
title=PartDesign_Bearingholder_Tutorial_II&oldid=108036](http://www.freecadweb.org/wiki/index.php?title=PartDesign_Bearingholder_Tutorial_II&oldid=108036)

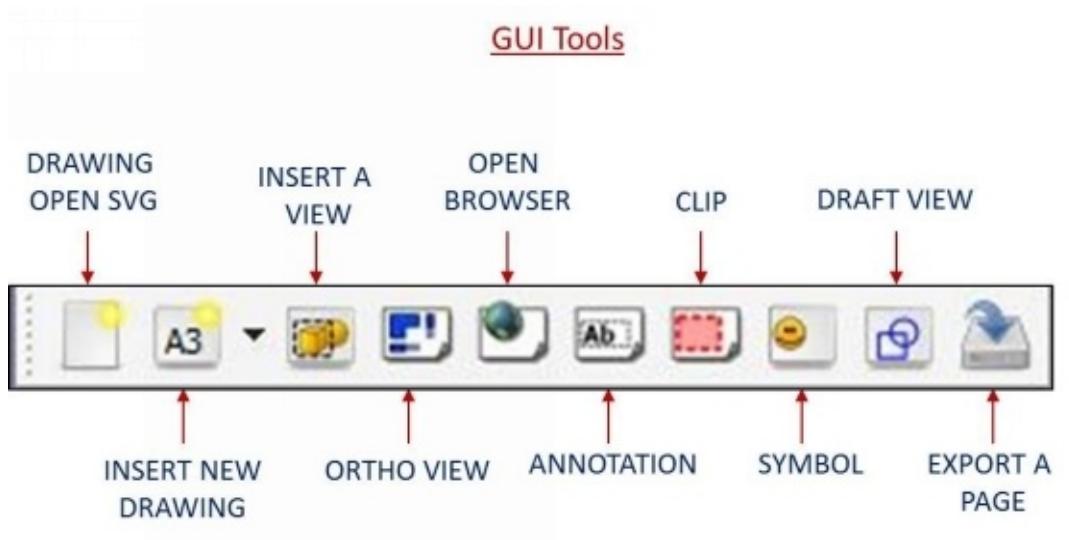
END

12. DRAWING WORKBENCH



The Drawing module allows you to put your 3D work on paper. That is, to put views of your models in a 2D window and to insert that window in a drawing, for example a sheet with a border, a title and your logo and finally print that sheet. The Drawing module is currently under construction and more or less a technology preview.

GUI Tools : - In Drawing workbench, the GUI tools are used for creating, configuring and exporting 2D drawing sheets.



The description and working of GUI tool are as follows,

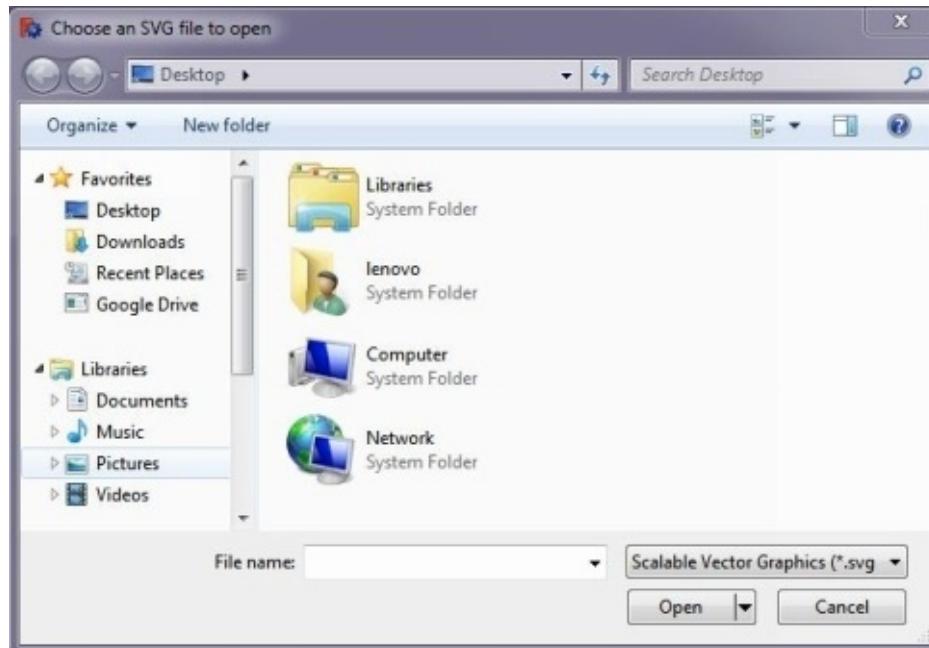
1. **USE OF DRAWING OPEN SVG**



This tool is used to open a drawing sheet previously saved as an SVG (scalable vector graphics) file. It can also be used to display any SVG.

• OPERATION

- a) Click on  Drawing open SVG tool.
- b) A Choose an SVG file to open dialogue will open.



- c) Then, select the file where you kept and click on Open button.
- d) This will open the drawing sheet in SVG format.

2. USE OF INSERT NEW DRAWING TEMPLATES



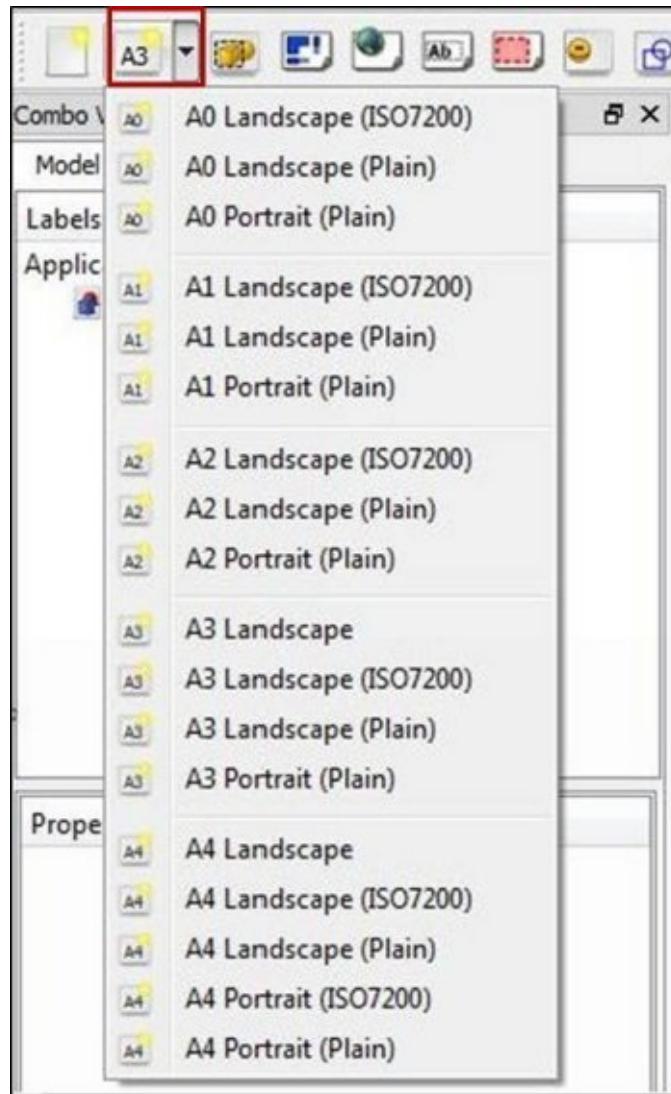
This tool is used to create a new drawing sheet from already installed templates. Currently, in drawing workbench, form A0 (Landscape/Portrait) to A4 (Landscape/Portrait) template is available.

To open the Drawing viewer to display the page, simply double-click on the **Page** object, or right-click → Show drawing. The page will be opened in a new tab. You can close the tab and open it again at any time the same way.

If the page does not display, click on the refresh icon in the main toolbar, or go to Edit → Refresh menu.

- OPERATION

- a) Click on **A3** **Insert new drawing templates** tool at drop down Menu.
b) Then select the required template (A0, A1, A4 etc.).



- c) A selected template will be opened in a new tab and also a Page will create in combo view.

- **PROPERTIES**

:- You can also change the template and other things form properties value tab. Select the Page from the Combo view → Model and then put the value what

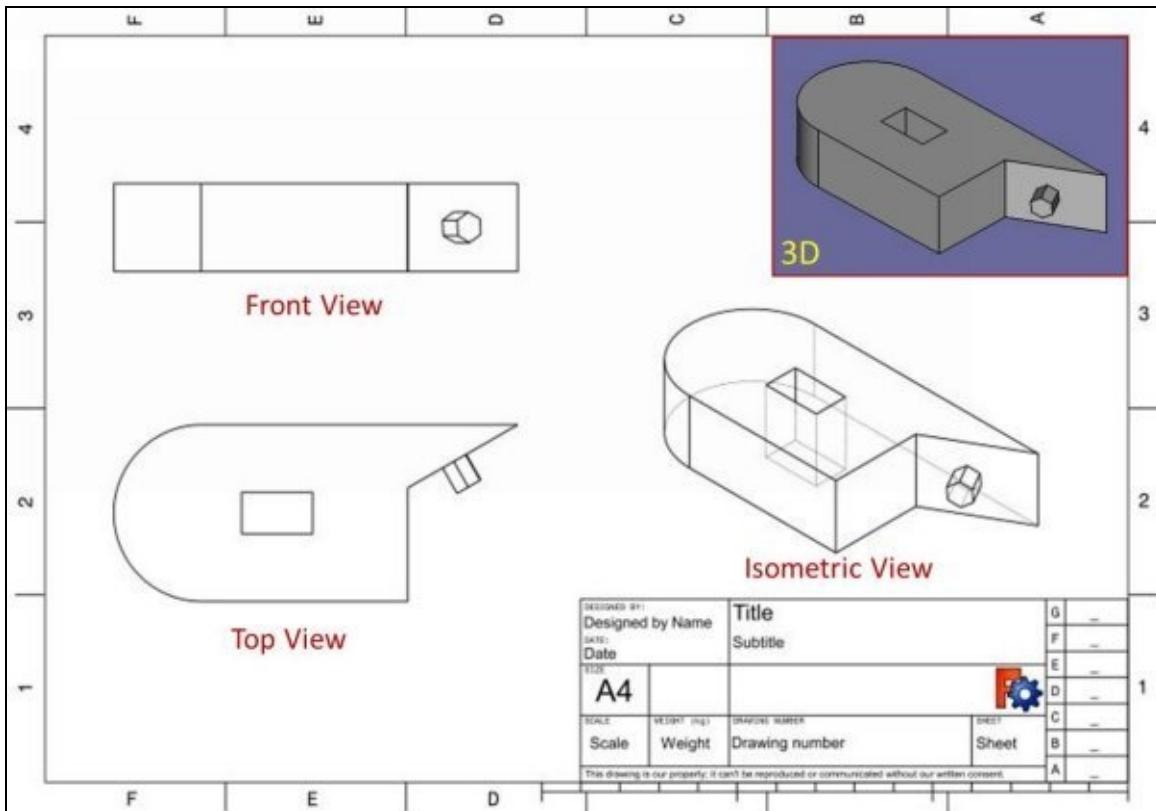
ever you want, in Property Value.

- For page result: - **Property** → **Data** → **Page Result**.
- For template of page: - **Property** → **Data** → **Template**.
- For editable texts of drawing sheet: - **Property** → **Data** → **Editable Texts**.

3. USE OF INSERT A VIEW



This tool is used to create a new view of the selected object in the active drawing sheet.



• OPERATION

- Switch to Drawing workbench.

b) Select an object (Pad, Pocket, Cube etc.) either in 3D view or from the Combo view.



- c)** Then click on **Insert view drawing** tool.
- d)** A drawing page will open in new tab.
- e)** By default, a top view scaled at 1:1 (real scale) will be placed at the top left of the page. It may not be visible if it's too small or too big for the page.
- f)** If you want to add more view in Drawing sheet, select the object from combo view or from project Tree and click the **insert view drawing** tool.
- g)** Then you need to modify the position, rotation *etc.* of object of drawing view from properties value.

- **OPTIONS**

The description of **property Value** are as follows:

- **Label:** changes the view's label in the combo view or Project tree. You can also click on the View in the tree and right-click → Rename.
- **Rotation:** rotates the view. For example, an isometric view will require a 60 degree rotation (see also Direction parameter below)
- **Scale:** sets the view scale.
- **X:** sets the view's horizontal position on the page in millimeters.
- **Y:** sets the view's vertical position on the page in millimeters. Please note that coordinate (0,0) is located at the top left of the page, so the higher the number, the lower in the page the view will be.
- **Direction:** changes the view direction. It is set by xyz values that define a vector normal to the page. Top view will be (0,0,1), and isometric will be (1,1,1). Values can be negative.
- **Show Hidden Lines:** toggles the hidden lines visibility on or off by selecting true/false
- **Show Smooth Lines:** toggles the smooth lines visibility on or off by selecting True or False. Smooth lines are also called tangency edges. These edges indicate surface changes between tangent surfaces.

1. How to set Top View in drawing sheet,

Select the **View** form combo view and the select the **Property → Data** and make the change as follows:

- Change Lable → View into **Top**.
- Change X → 10 into **50**.
- Change Y → 10 into **154**.
- Change Scale → 1 into **1.5**.
- Set Direction value → **(0,0,1)**.

2. How to set Front View in drawing sheet,

Select the **View001** form combo view and the select the **Property → Data** and make the change as follows:

- Change Lable → View001 into **Front**.
- Change X → 10 into **50**.
- Change Y → 10 into **70**.
- Change Scale → 1 into **1.5**.
- Change Rotation → 0° into **90°**
- Set Direction value → **(0,-1,0)**.

3. How to set Isometric View in drawing sheet,

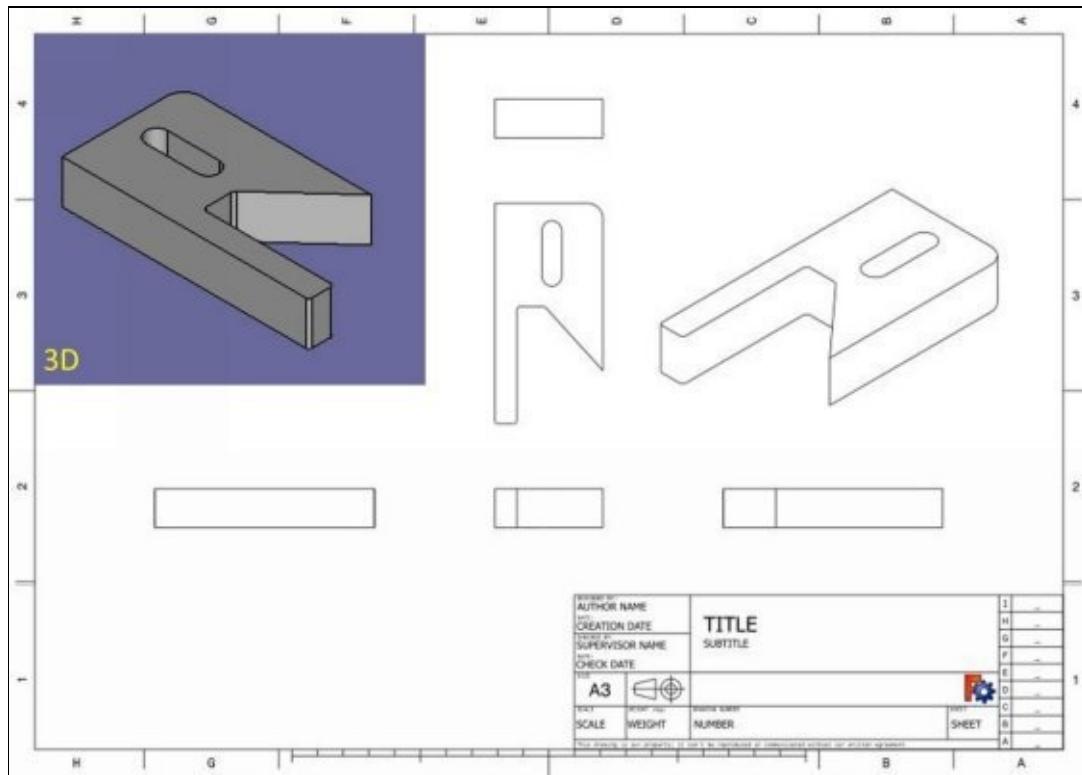
Select the **View002** form combo view and the select the **Property → Data** and make the change as follows:

- Change Lable → View002 into **Iso**.
- Change X → 10 into **175**.
- Change Y → 10 into **120**.
- Change Scale → 1 into **1.5**.
- Change Rotation → 0° into **60°**
- Set Direction value → **(1,-1,1)**.

4. USE OF ORTHO VIEW



This tool is use to insert an orthographic projection of the selected object in the active drawing sheet.



- **OPERATION**

- First of all switch to Drawing workbench.
- Click on **Insert new drawing templates** tool at drop down Menu.
- A drawing page will open in new tab.
- Then, select an object (Pad, Pocket, Cube etc.) either in 3D view or from the Combo view.
- Now, click on **Ortho view** tool.
- By default, a front view of object will be placed at the centre of the drawing sheet and **orthographic projection** dialogue will also open in combo view (see the below Fig.).



g) You can set the required data whatever you want. Check the **Secondary view** and get more View in drawing sheet (see the above Fig.).

5. USE OF OPEN BROWSER TOOL



This tool is used to allow you to display a selected **Drawing page** using FreeCAD's internal web browser. The normal Drawing page viewer of FreeCAD is based on **Qt's built-in SVG rendering module**, which only supports a tiny subset of the full SVG specification. As a result, some more advanced SVG features, such as pattern fills or multiline texts are not supported by this viewer.

- OPERATION

- a) First of all switch to Drawing workbench.
- b) Click on  **Insert new drawing templates** tool at drop down Menu.
- c) A drawing page will open in new tap.
- d) Then, select an object (Pad, Pocket, Cube etc.) either in 3D view or from the Combo view.
- e) Now, click on  **Ortho view** tool.
- f) By default, a front view of object will be placed at the centre of the drawing sheet and **orthographic projection** dialogue will also open in combo view (see the below Fig.).
- g) Add some more view or other content into your page.
- h) Then press the  **Open browser tool**.
- i) A Web browser page will open in new tab.

6. USE OF ANNOTATION TOOL



This tool is use to place a block of text on a **Drawing sheet**.

- OPERATION

- a) First of all switch to Drawing workbench.
- b) Click on  **Insert new drawing templates** tool at drop down Menu.
- c) A drawing page will open in new tap.
- d) Then, select an object (Pad, Pocket, Cube etc.) either in 3D view or from the Combo view.
- e) Now, click on  **Ortho view** tool.

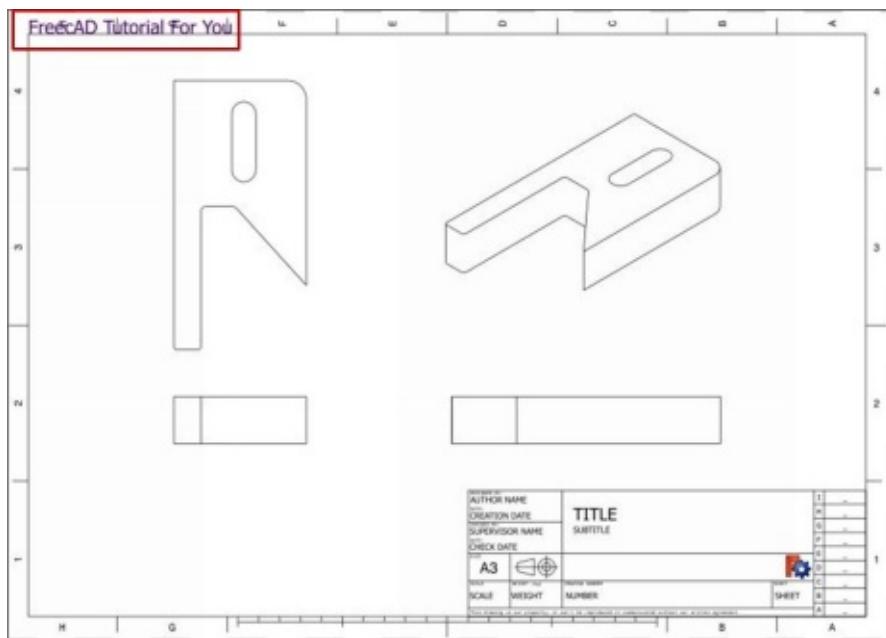
f) By default, a front view of object will be placed at the centre of the drawing sheet and **orthographic projection** dialogue will also open in combo view (see the below Fig.) g) Add some more view or other content into your page.

h) Then press the  **Annotation tool**.

i) A Annotation view will create inside the Page.

j) Click on annotation view form combo view and adjust the desired properties, such as text contents, font, size and position form **Properties value**.

k) Then text will create at top left of the drawing sheet (see the below Fig.).



7. USE OF DRAWIGN CLIP TOOL



This tool is use to place a clipping rectangle on a **Drawing sheet**. Drawing View objects can then be added to that clipping rectangle, and their display will be truncated by the borders of the rectangle.

- OPERATION

- a) First of all switch to Drawing workbench.
- b) Click on  **Insert new drawing templates** tool at drop down Menu.
- c) A drawing page will open in new tab.
- d) Then, select an object (Pad, Pocket, Cube etc.) either in 3D view or from the Combo view.
- e) Now, click on  **Ortho view** tool.
- f) By default, a front view of object will be placed at the centre of the drawing sheet and **orthographic projection** dialogue will also open in combo view (see the below Fig.)
- g) Add some more view or other content into your page.
- h) Then press the  **Drawing Clip** tool.
- i) Now adjust the desired properties, such as size and position etc. form Properties value.
- j) And last Drag and Drop **Drawing View** objects on the Clip object in the Combo or Tree View.

Note:-

Clipping objects are not displayed properly by the internal Qt-based Svg viewer, but the **Open Browser** command shows them correctly.

8. USE OF DRAWIGN SYMBOL TOOL

 This tool is use to add the contents of a SVG image on a selected **Drawing Sheet**. These contents can then be moved and rescaled on the page. The contents of the SVG image are copied into the FreeCAD document, so it is independent from the original file, and will display the same way on another computer that doesn't have the original SVG file.

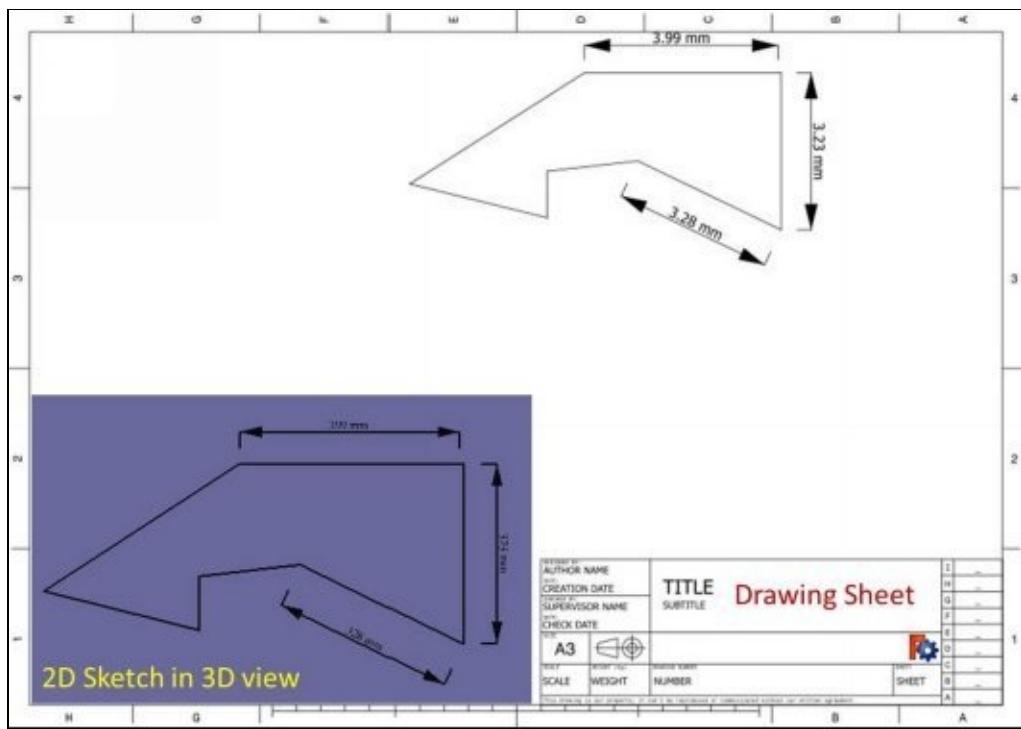
- OPERATION

- a) First of all switch to Drawing workbench.
- b) Click on  **Insert new drawing templates** tool at drop down Menu.
- c) A drawing page will open in new tab.
- d) Then, select an object (Pad, Pocket, Cube etc.) either in 3D view or from the Combo view.
- e) Now, click on  **Ortho view** tool.
- f) By default, a front view of object will be placed at the centre of the drawing sheet and **orthographic projection** dialogue will also open in combo view (see the below Fig.)
- g) Add some more view or other content into your page.
- h) Then press the  **Drawing Symbol** tool.
- i) Select the SVG file.
- j) Now adjust the desired properties, such as size and position etc. form Properties value.

9. USE OF DRAFT VIEW TOOL



This tool is use to put a selected objects on a svg **Drawing sheet**. During operation, If no sheet exists in the document, a default one will be created. This tool works similarly to the Drawing view tool, but is optimized for Draft object, and can render flat 2D object with a face filling. It can also handle a couple of specific Draft objects, such as dimensions and text, that the drawing view tool can't handle.



- **OPERATION**

- First of all switch to Draft workbench.
- Set the Top view.
- Then draw a Closed wire object (see the above Fig.).
- Make a dimension(s) of an object or wire.
- Now switch to Drawing workbench.
- Then select the 2D object with dimension with the help of **ctrl + A**.
- Press the  **Draft view tool**.
- A Page will create in Combo or Tree View.
- Then make double click on page and a Drawing sheet will open in new tab.
- Now you can adjust the desired properties, such as size and position etc. from Properties value.

Note:-

This tool will work best with flat 2D objects from the **Draft** or **Sketcher** modules.

10. USE OF EXPORT A PAGE TOOL



This tool is used to Export or saves the current Drawing sheet as an **SVG** (scalable vector graphics) file. Such a file can then be edited in a scalable vector graphics program such as **Inkscape**.

SVG files are common and can be viewed in most modern browsers and image viewers. It can be a useful way to share a design with people who don't have FreeCAD installed on their PC.

END

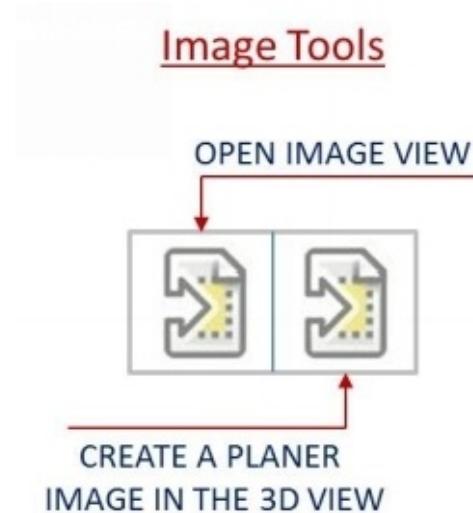
13. IMAGE WORKBENCH



The image workbench manages different types of **bitmap images**, and lets you open them in FreeCAD. Currently, the modules lets you open .bmp, .jpg, .png, .xpm etc. file formats in a separate viewer window.

The major use is tracing over the image, in order to generate a new part at using the image as template.

Image Tools : - In Image workbench, the image tools are used for importing the bitmap images and use as template for creating the different object .



The description and working of Image tool are as follows,

1. USE OF OPEN IMAGE VIEW TOOL



This tool is use to open an image in 3D work plane.

- OPERATION

- a) First of all switch to **Image workbench**.

-  b) Then press the **Open image view tool**.
- c) A **choose an image file to open** dialogue will open.
- d) Then select the file where you kept an image file and click on open button.



- e) A image will open in new tab.

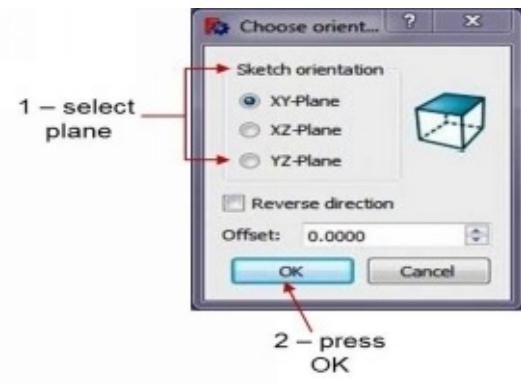
2. USE OF OPEN IMAGE VIEW TOOL



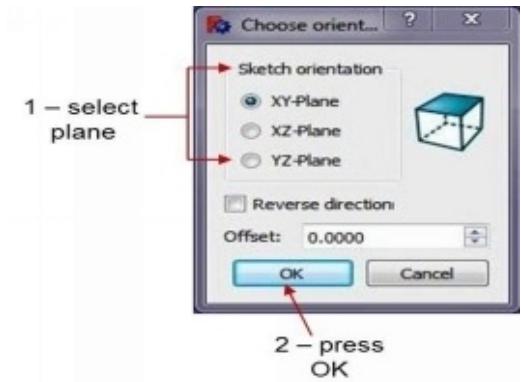
This tool is use to allow to open an image on a plane in the 3D-space of FreeCAD.

- OPERATION

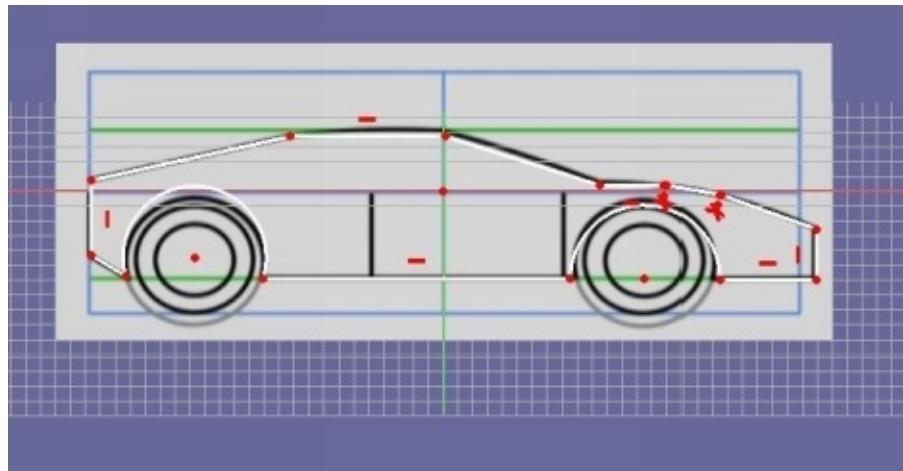
- a) First of all **switch to Image workbench**.
-  b) Then press the **Image in the 3D view tool**.
- c) A **choose an image file to open** dialogue will open.
- d) Now, select the file where you kept an image file and click on open button.
- e) Then a Choose orientation window will open.



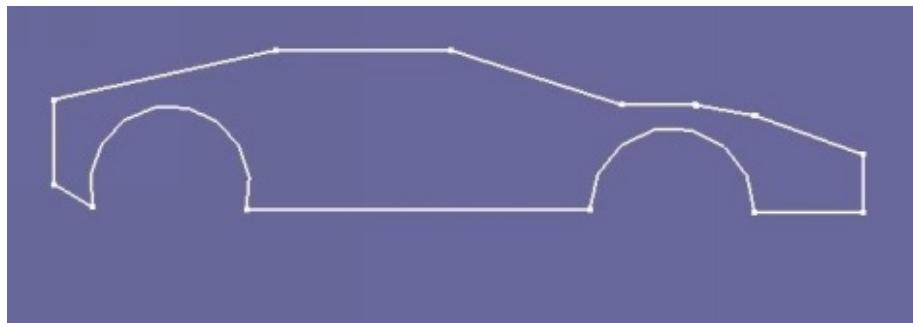
- f) Select the Plane(XY) from Choose orientation window and click on OK button..
- g) A image will open in 3D view.
- h) Now switch to Part design workbench.
- i) Click on  **New sketch** tool.
- j) Then select the Plane(XY) from Choose orientation window.



- k) Make an one sketch according to figure in 3D work plane and hit the Close button to exit the sketches function (see below Fig.).

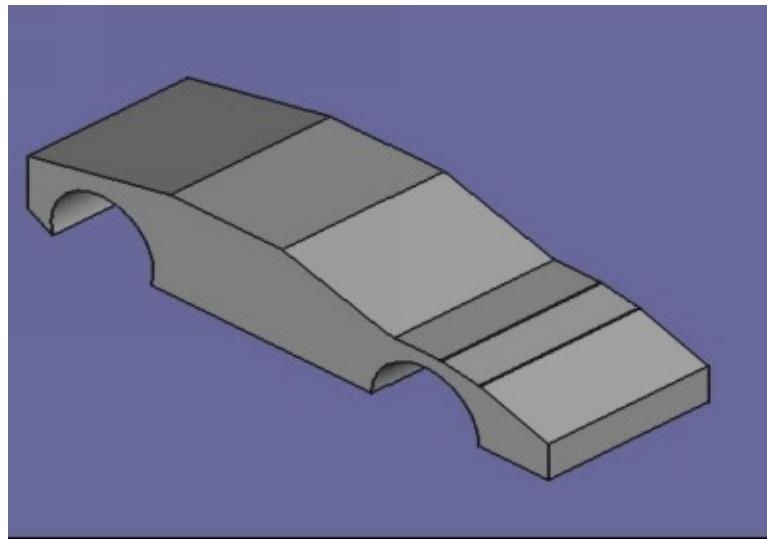


- l) Then select the ImagePlane from combo view and make hide object
(Right click on ImagePlane and select Hide selection).



m) Then it looks like above figure.

- n) Then select the Sketch form combo view o) Now press the  Pad tool and make a pad of object.



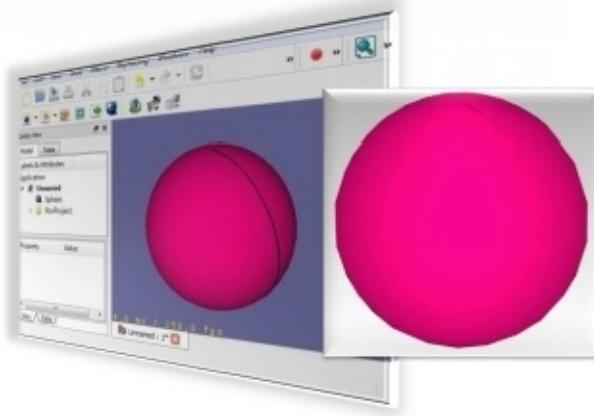
- p) Then it looks like above figure.
- q) Hit the OK button to exist the function.

END

14. RAYTRACING WORKBENCH



The Raytracing workbench is used to generate photorealistic images of your models by rendering them with an external renderer. The Raytracing workbench works with **templates**, the same way as the **Drawing workbench**, by allowing you to create a Raytracing project in which you add views of your objects. The project can then be exported to a ready-to-render file, or be rendered directly.



Currently, in Raytracing workbench two renderers are supported: **Povray** and **luxrender**. To be able to render directly from FreeCAD, at least one of those renderers must be installed on your system, and its path must be configured in the FreeCAD Raytracing preferences. Without any renderer installed, though, you are still able to export a scene file that can be used in any of those renderers later, or on another machine.

NOTE:-

I recommended to use **Povray** because it is open source software and very easy to use.

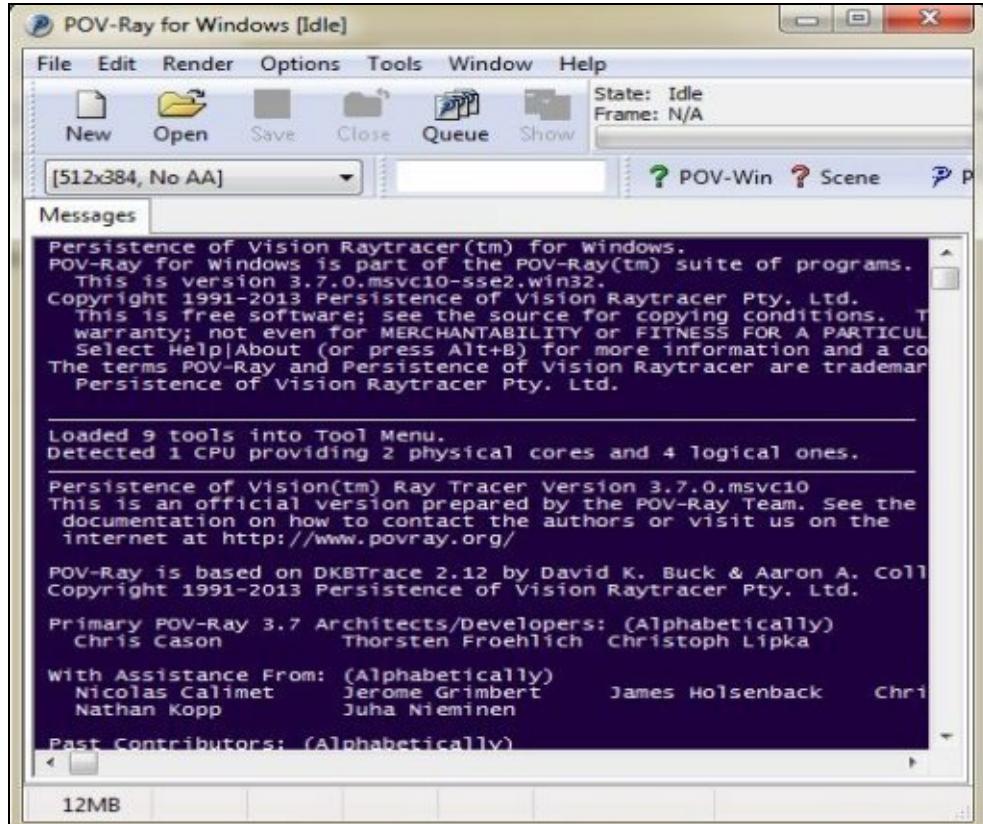
HOW TO INSTALLED POVRAY

- Search the Povray (<http://www.povray.org/>) on Google and download

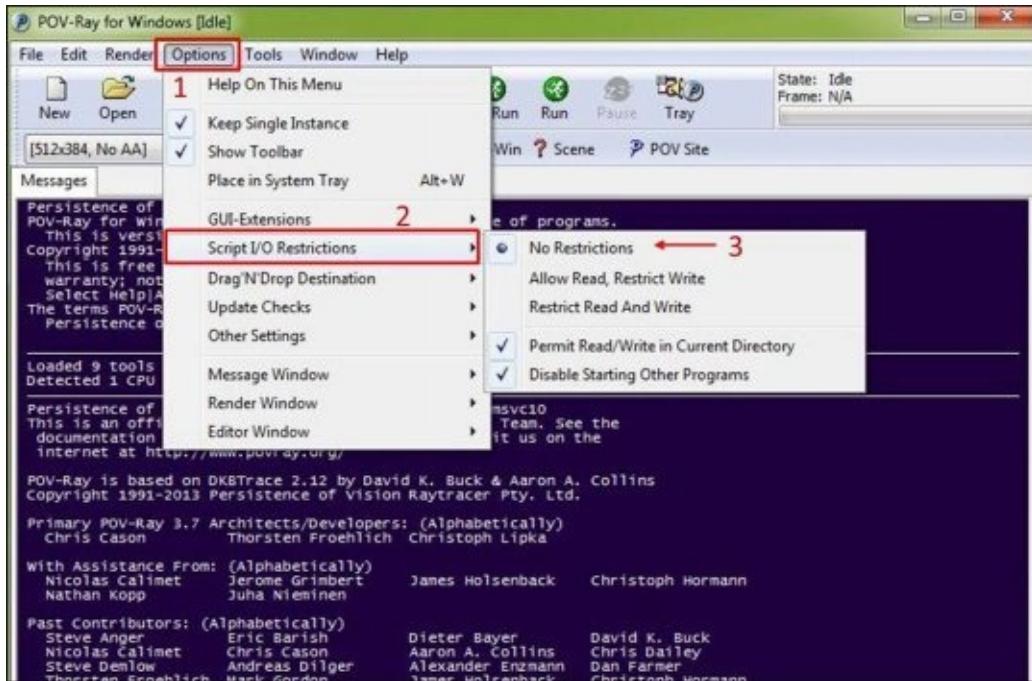
the latest version of Povray. For downloading the latest version of Povray you can also follow the link-

(<http://www.povray.org/ftp/pub/povray/Official/povwin-3.7-agpl3-setup.exe>)

- After downloading the Povray, click on the Povray setup and run it. Follow the instructions and install the Povray.
- After installation, open the Povray.



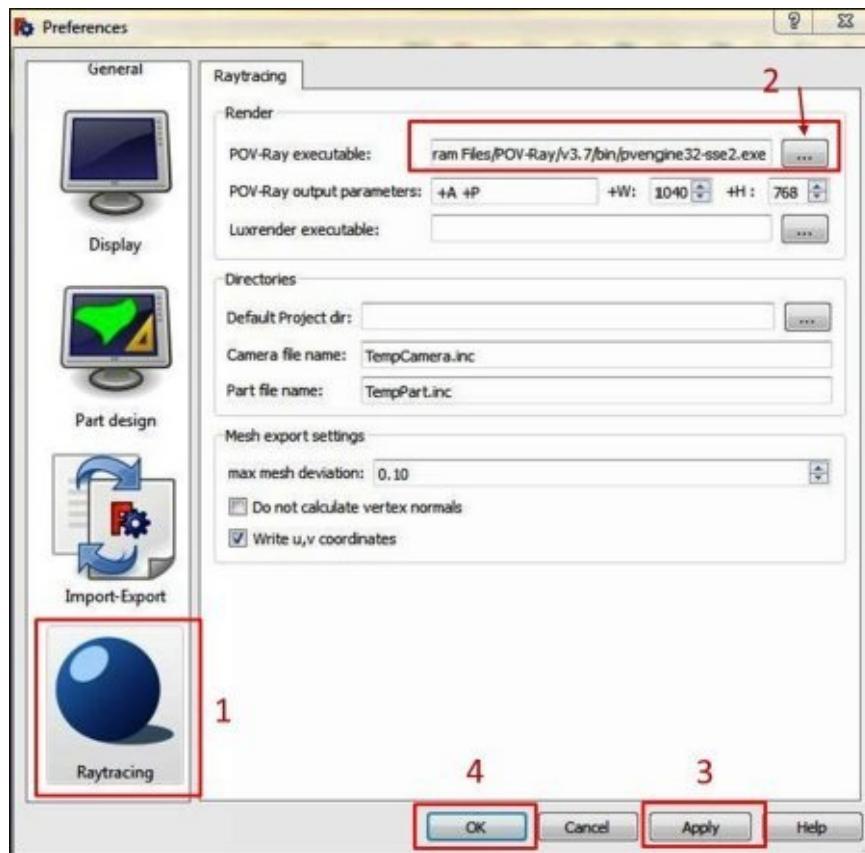
- Then click on **Option** button and check **No Restriction** in **Script I/O Restriction** (see the below Fig.).



- Now, your Povray are ready to use with FreeCAD.

HOW TO SET RAYTRACING PATH IN FreeCAD

- In FreeCAD, the Raytracing workbench is work only, when its path must be configured in the FreeCAD Raytracing preferences.
- Open the FreeCAD and switch to Raytracing workbench.
- Then, select the **Edit** and click on **Preferences** in FreeCAD.
- A preference tab will open in 3D view.
- Select the **Raytracing** and set the POV-Ray executable path (see the below Fig.).



- Generally, the POV-Ray executable path is: -
[C:/Program Files/POV-Ray/v3.7/bin/pvengine32-sse2.exe]
- Then hit the **Apply** button and click on **Ok** to exit the Preferences function.
- Now, FreeCAD is ready to use for Raytracing project with **Povray** render.

Tools in Raytracing Workbench

1. RAYTRACING PROJECT TOOLS

Raytracing project tools are the main tools for modifying your 3D work with external renderers.

Raytracing Project Tools



The description Raytracing project tools are as follows,

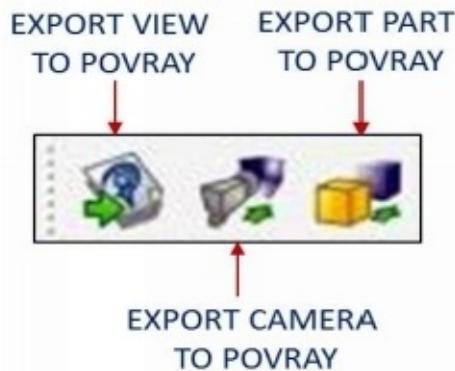
- **NEW POVRAY PROJECT** : - This tool is used to insert a new POV-Ray project in the current work. This tool consists of two parts, first one is **ProjectStd** and second one is **RadiosityNormal**. You can choose any one, whatever you want.
- **NEW LUXRENDER PROJECT** : - This tool is used to insert a new LuxRender project in the current work. This tool also consists of two parts, first one is **LuxClassic** and second one is **LuxOutdoor**. You can choose any one, whatever you want.
- **INSERT PART** : - This tool is used to insert a view of a part in a current Raytracing project.
- **RESET CAMERA** : - This tool is used to reset the camera position of a Raytracing project to the current work.

-  **RESET CAMERA** : - This tool is used to export a Raytracing project to a scene file for rendering in an external render(PovRay/LuxRender).
-  **RENDER** : - This tool is used to Render directly with an external render (PovRay/LuxRender).

2. UTILITIES TOOLS

Utilities tools are helper tools to perform specific tasks manually.

Utilities Tools



The description utilities tools are as follows,

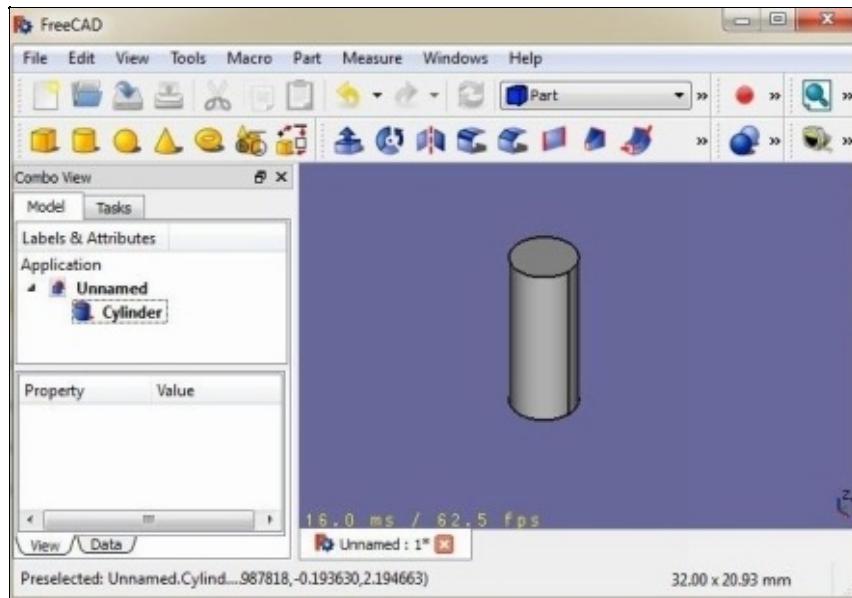
-  **EXPORT VIEW TO POVRAY** : - This tool is used to write the active 3D view with camera and all content to a PovRay file.
-  **EXPORT CAMERA TO POVRAY** : - This tool is used to export camera position of the active 3D view in PovRay format to a file.
-  **EXPORT PART TO POVRAY** : - This tool is used to write the selected object as a PovRay file.

OPERATION WITH RAYTRACING WORKBENCH

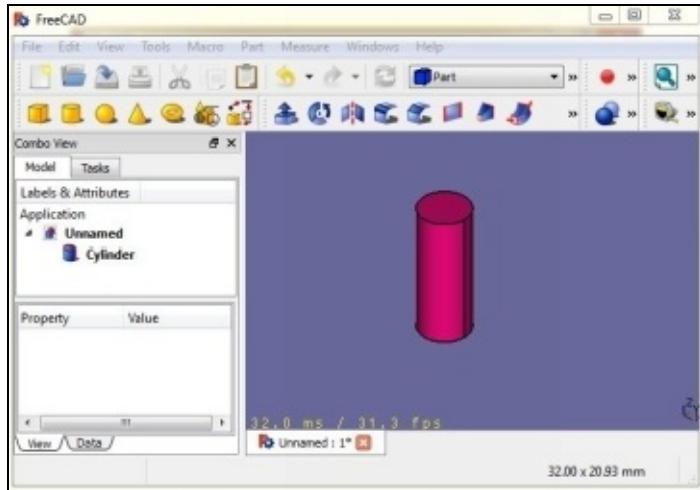
In FreeCAD, the Raytracing workbench play great roll to make the photorealistic images of part or object. In this workbench, all the tools have same value to make the rendered image. The procedure for how to make photorealistic images are as follows:-

PROCEDURE

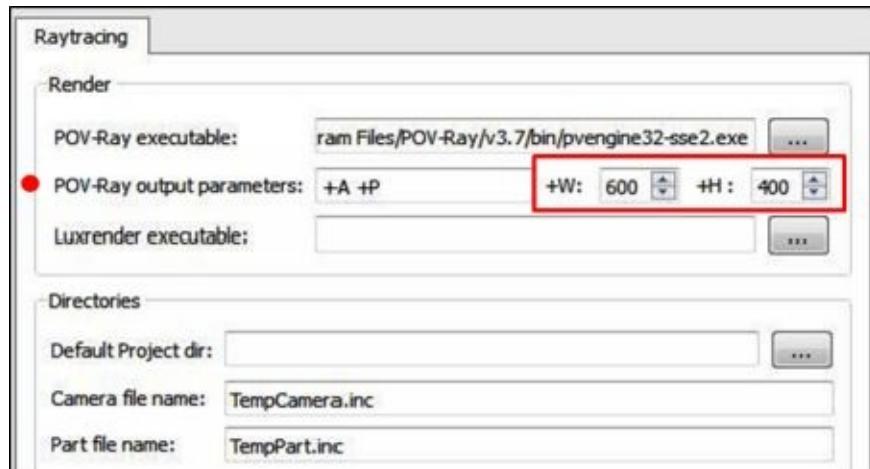
- Open the FreeCAD and create a new document.
- Switch to Part workbench.
- Then create a new part(Cylinder, Cube, Sphere etc.).



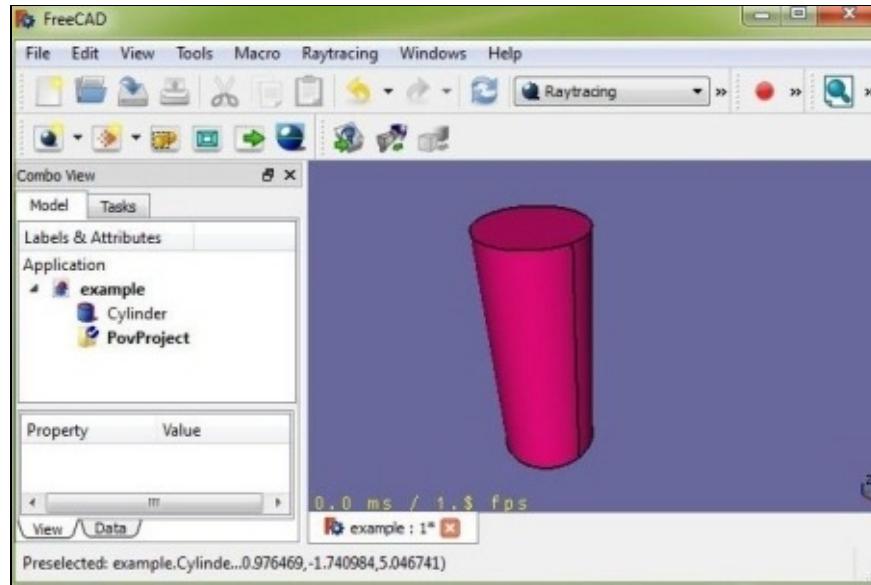
- You can also change the Shape Color of Cylinder for better look (see below Fig.).



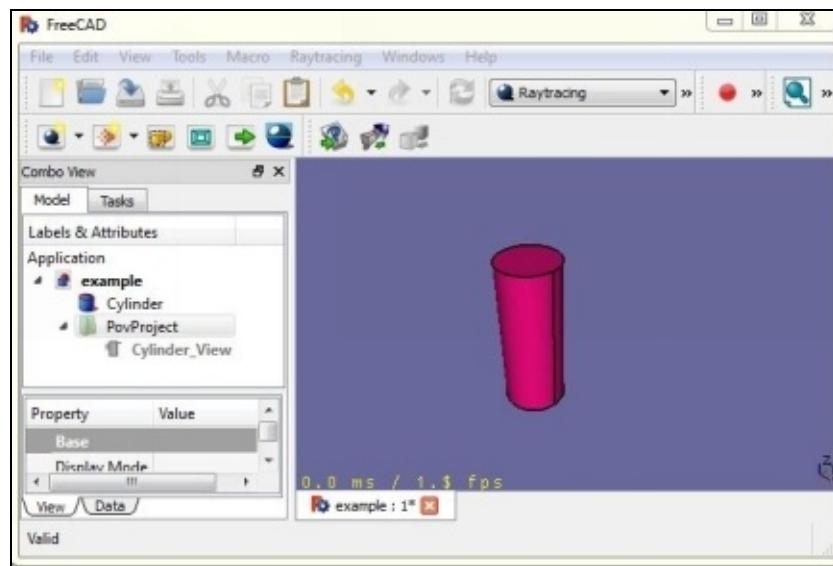
- Now switch to Raytracing Workbench.
- Then go to **View menu** and make the View to **Perspective view**. you can also set the view form keyboard. Letter (**O**) for Orthography view and letter (**P**) for Perspective view.
- You can also set the rendered image size. Go to the **Edit menu** and select the **Preferences**. Then select the **Raytracing** and set the **POV-Ray output parameters** like (**W+ & H+**).



- Now click on **New PovRay project** and select the project (ProjectStd or RadiosityNormal) from dropdown menu. I am selecting here **RadiosityNormal**.

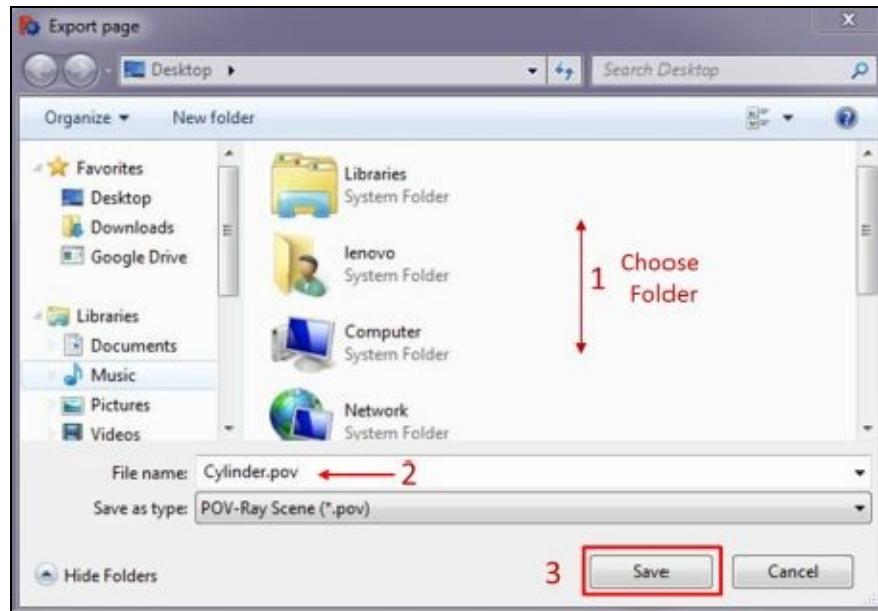


- Then select the **Cylinder** form **Combo view** and click on the  **Insert part** tool.
- Now select the **PovProject** form the **Combo view** and click on the  **Reset Camera** tool.

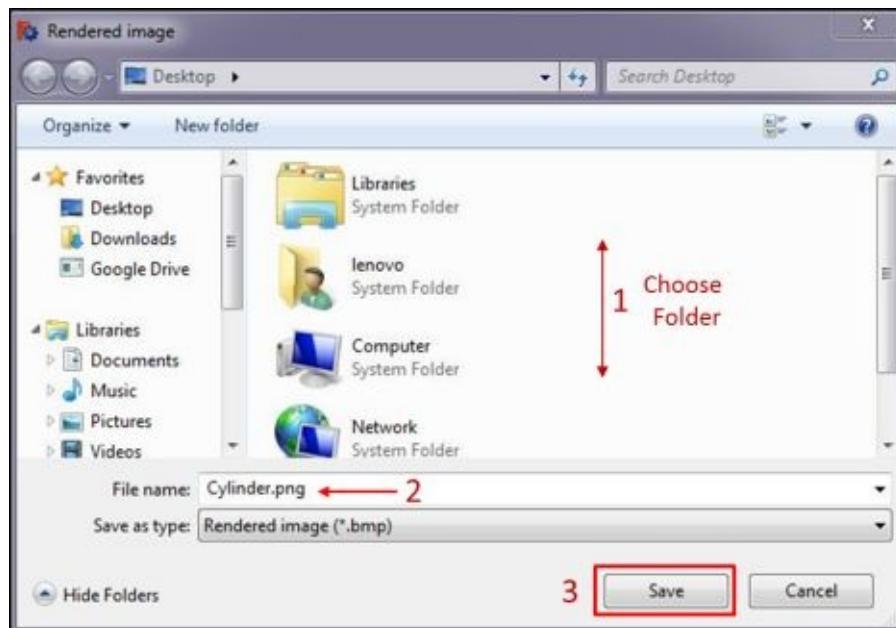


- Then again select **PovProject** form the **Combo view** and click on the  **Export Project** tool.
 - A **Export page** window will open in

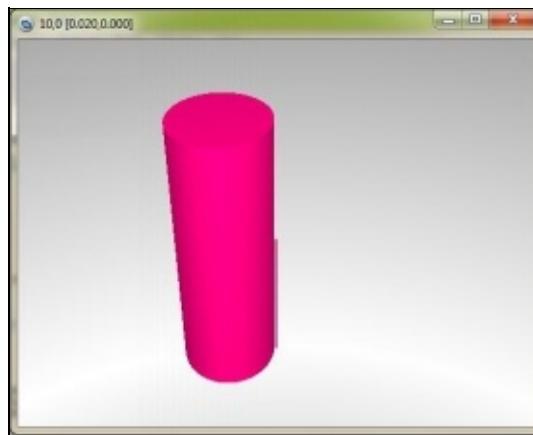
current document (see below Fig.).



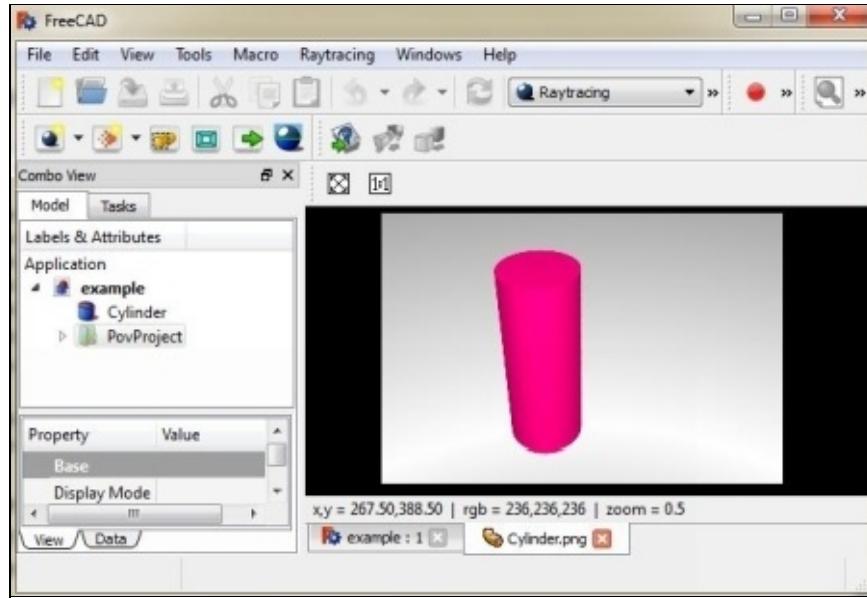
- Then choose the folder where you want to export the rendered project and write the **file name** (cylinder.pov) and hit the **Save** button.
- Then again select **PovProject** from the **Combo view** and click on the  **Render** tool. This tool make the rendered image directly with the help of external render software like POV-Ray.
 - A **Rendered image** window will open in current document (see below Fig.).



- Then choose the folder where you want to export or save the rendered image and write the **file name** (cylinder.png) and hit the **Save** button.
- Then wait for rendering to finish. This may take a while.



- After finished the rendering, the FreeCAD will immediately open the rendered image in new window (see below Fig.).



- Now operation has finished.
- You can also follow the same procedure for another project of POV-Ray as **ProjectStd**.

NOTE:-

For better understanding, Follow the **video tutorial link**:

<https://youtu.be/XOqMLWh2JzM>

[HOW TO USE RAYTRACING WORKBENCH IN FreeCAD]

END

15. IMPORTANT NOTE

NOTE - 1

- In FreeCAD, the offline FreeCAD manual and tutorial is in built. This tool is looks like: -  and the name is “**What is this**”.

- This tool is located in right side  of the workbench.
- Triple click on this tool and find or search the all things which are related to FreeCAD.
- In this, the FreeCAD documentation contents are very important. Click on **FreeCAD documentation** and find the related question and notes.
- For find the **FreeCAD tutorial**, type the Tutorial in search box and select the all listed notes.

NOTE - 2

FreeCAD is Very easy to use software no dot, but is totally depending on that how much time you are expanding with FreeCAD. It just like a, how a new person handle the smartphone easily without guidance. I know mistake will happen during handle the FreeCAD, but I also know you will correct it shortly.

Finally, it is universal truth “Practice Makes Perfect”. So do not wait, let’s start.....

Bibliography

- <http://www.freecadweb.org/>
- <https://en.wikipedia.org/wiki/FreeCAD>
-

<http://www.freecadweb.org/wiki/index.php?title=Category:Tutorials>

- [And website.](#)

[Thank You For Reading]

END