

Using FreeCAD

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FreeCAD is a 3D graphics application design to support computer aided design(CAD). It allows parametric object creation and easy shaping of parts. However, the online document is lacking - to say the least. This document is intended to capture useful tips, basic processes, etc.

Views








Most work in FreeCAD is done from one of six views, though an axometric view is also supported. Views can be selected from view icons at the top of the application.



Getting a feel for these views will help when you start building models. Each of these views can be accessed from the numerical keypad when the app is in view mode, selected by the tab at the bottom of the Property pane:



The icons are as follows:

View	Icon	Keypad
Axometric		0
Front		1
Right		2
Top		3
Back		4
Left		5
Bottom		6

There is also an icon to fit the whole content into the view pane:



Workshops

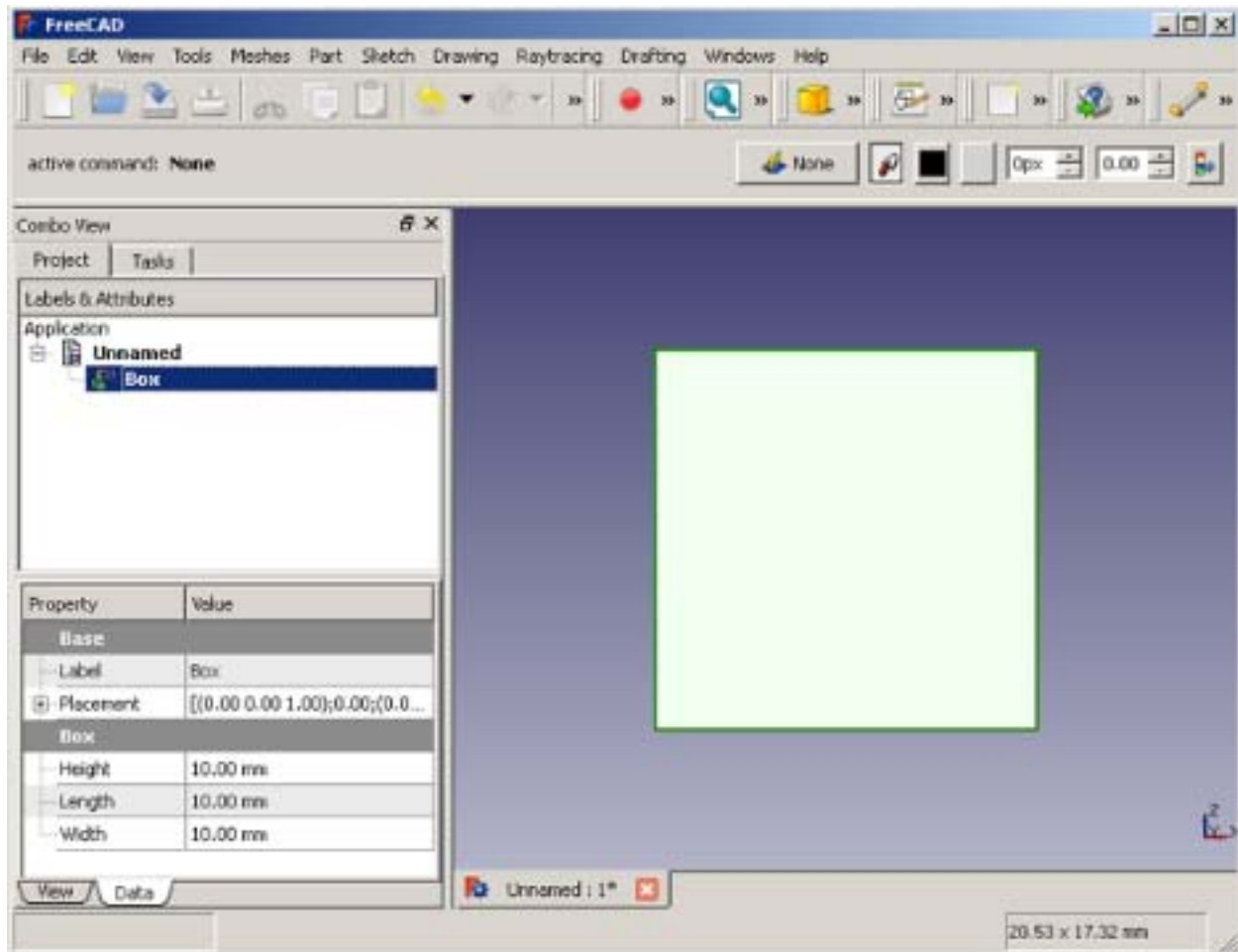
Different modeling techniques are called “workshops” in FreeCAD. Of particular importance to OSE is the Parts Workshop that allow complex shapes to be built up out of primitives (cube, cylinder, sphere, cone, and torus). These are selectable from a set of icons at the top of the application:



Clicking on one of these icons will cause that primitive part to be added to your document. It has a default name like “box001”, which can easily be renamed by right clicking on the item in the Labels & Attributes pane and selecting “Rename”. It can also be changed by left clicking on the item, and editing the Label property in the Properties pane.

Sizing a Primitive Part

Create a new document and add a box by clicking on the cube icon. It shows as a light green square:



Click on the data tab. In the Properties pane, the Height, Length, and Width properties can be modified by selecting the value and modifying it with the keypad. Units default to millimeters (mm).

How is the default units for a document specified?

Change the values to:

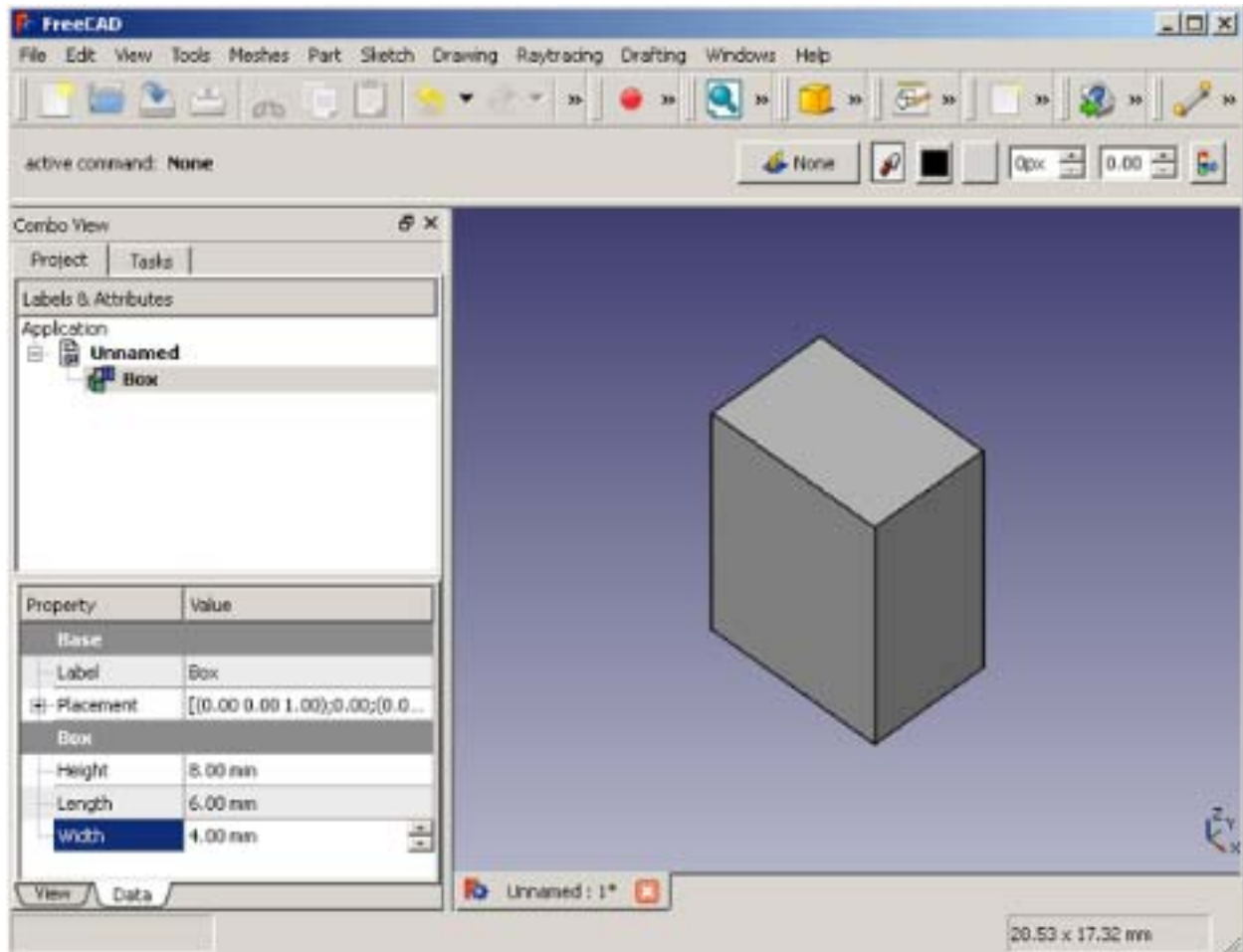
Height = 8.0

Length=6.0

Width=4.0

Height is the Z axis, Length is the X axis, and Width is the Y axis.

Click on the view tab and experiment with different views. In axonometric view, the sized cube looks like this:



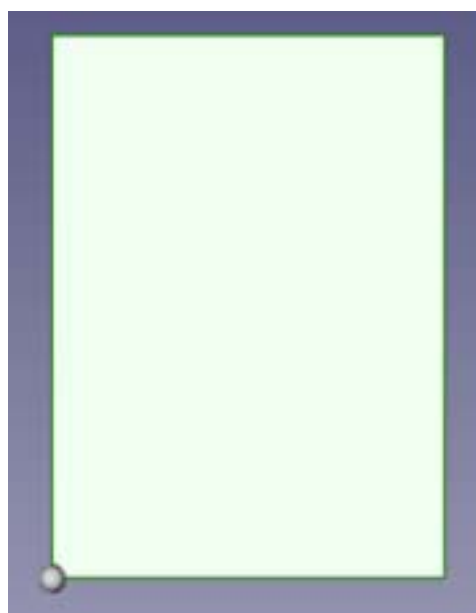
Positioning a Part

Part primitives can be moved around in several ways. Precise positioning is done in the Properties pane. Click on the “+” sign next to “Placement”, then click on the plus sign next to “Position”.

The properties for the box will look like this (by default):

Property	Value
Base	
Label	Box
<input type="checkbox"/> Placement	[(0.00 0.00 1.00);0.00;(0.0...
Angle	0.00 °
<input checked="" type="checkbox"/> Axis	[0.00 0.00 1.00]
<input type="checkbox"/> Position	[0.00 0.00 0.00]
x	0.00
y	0.00
z	0.00
Box	
Height	8.00 mm
Length	6.00 mm
Width	4.00 mm

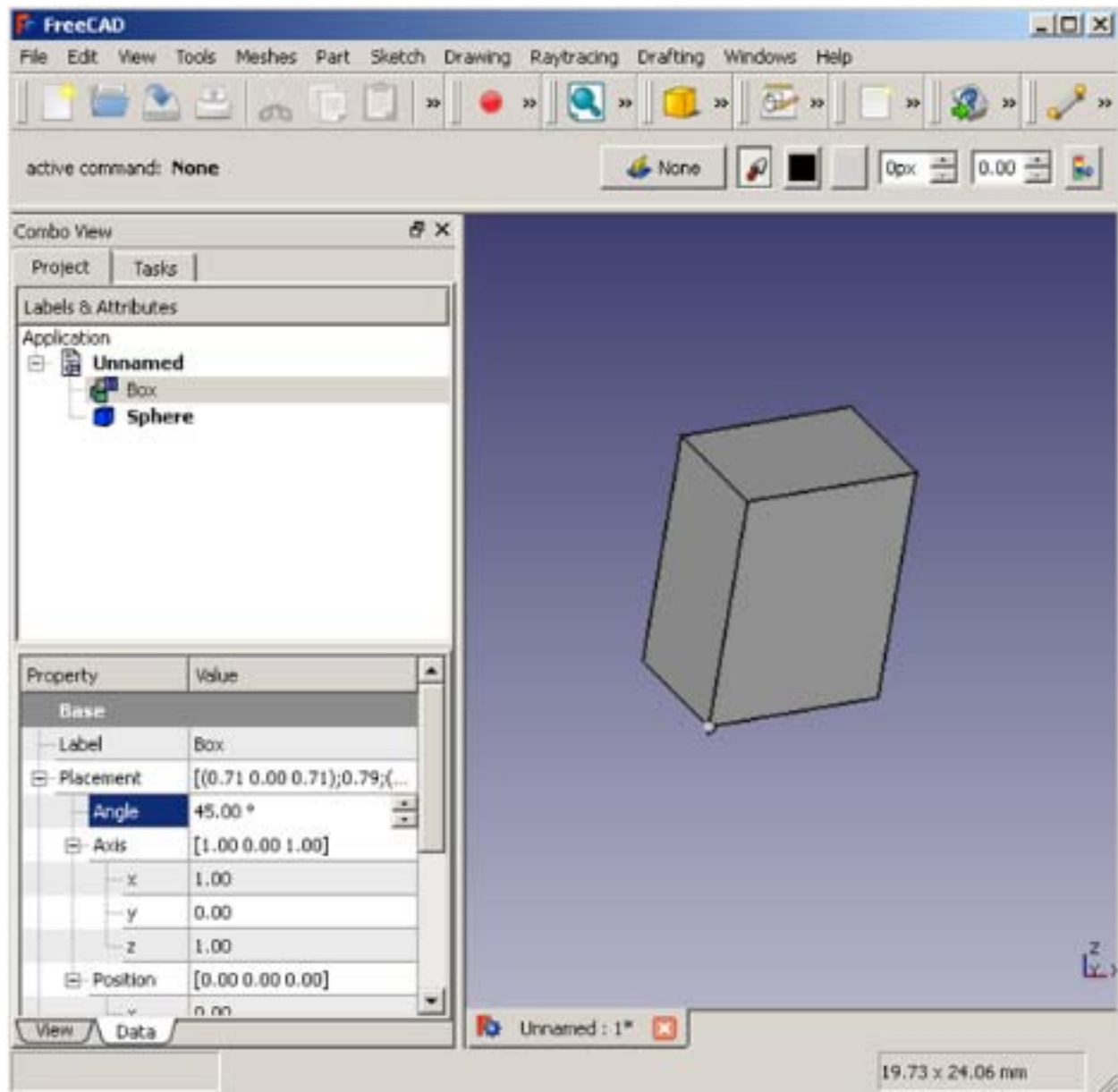
The placement of the box is at position [0.00, 0.00, 0.00], which is the origin of the placement coordinates system. On useful trick is to create a sphere and give it a small radius (say 0.2). By default, it should appear at the origin. If not, set the postition to [0.00, 0.00, 0.00]. This gives you a visual representation of where the origin is, which can help when positioning other parts.



Parts can be rotated in the coordinate space as well. Click on the plus sign next to “Axis”. You should see:

Property	Value
Base	
Label	Box
<input type="checkbox"/> Placement	[(0.00 0.00 1.00);0.00;(0.0...
Angle	0.00 °
<input type="checkbox"/> Axis	[0.00 0.00 1.00]
x	0.00
y	0.00
z	1.00
<input type="checkbox"/> Position	[0.00 0.00 0.00]
x	0.00
y	0.00
z	0.00
Box	
Height	8.00 mm
Length	6.00 mm
Width	4.00 mm

The axis properties define an “axis” that you can rotate the select part around. In the above illustration, the angle of rotation will be about the z-axis. To make this a more interesting example, set x to 1.00 and then set Angle to 45 degrees. This will cause the box to be rotated about a line extending from 0,0,0 to 1,0,1. The front view now looks like this:



Making Cuts

Cutting away material is an essential way to form more complex parts. Supposed we wanted to drill a hole through our box.

Click on the cylinder icon to add a cylinder primitive to the document.

Select it. In the data tab, change the radius to 1.00.

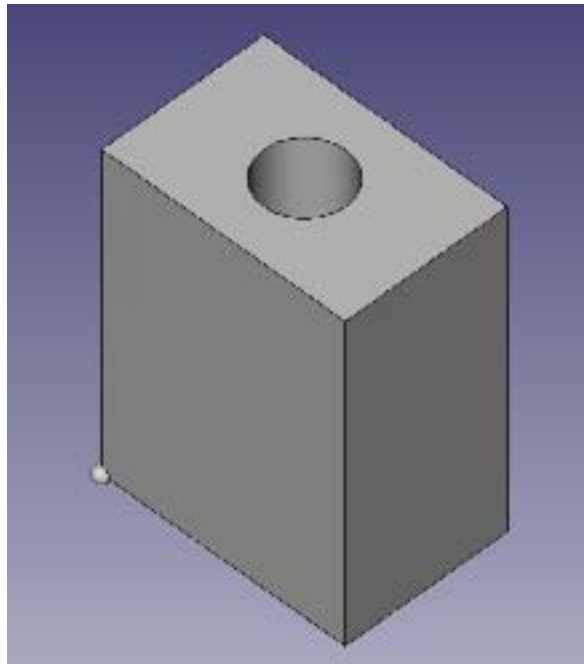
Set the position to [3.00, 2.00, -1.00]. Change the view to top to see the positioned cylinder.¹²

To make the hole, click on “Box” in the Labels & Attributes pane, and then control click on “Cylinder”. The order is important because we are going to subtract the second

from the first. Now click on the Cut icon at the top of the application window:



If you shift to axonomic view, you should see this:



FreeCAD creates a new name for the combined object, “Cut” in this case. If you click on the plus sign next to this part in the Labels & Attributes pane, you’ll see the two primitives that went into making this new part.

Besides drilling holes, the cut operation can be used to make angle, bevels, counter-sunk holes, etc.

Combining Parts

Rotating a Part