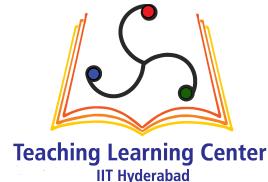




# Conic Sections using FreeCAD



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**Abstract**—This manual deals with Sketching of Conic Sections and designing 3D objects like Sphere, Ellipsoid, Hyperboloid, Paraboloid, using FreeCAD. This manual explains combination of various constraints and tools which is useful for designing complicated parts.

## 1 SPHERE

### 1.1 Solid Sphere

To design a sphere(CAD),

- Open FreeCAD by typing the following command in the command terminal.

```
sudo freecad
```

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Fig. 1: Sketching tools



Fig. 2: constraints

- In FreeCAD go to **View menu** click on **Workbench** and select **Sketcher Workbench**.
- Go to **Sketch menu** and select **create sketch** in xy-plane.
- Now create a circle using **sketcher geometries** in **sketch menu**.
- Select the circle and apply **Fix radius** constraint to 25mm using **sketcher constraints** in **Sketch menu**
- Apply **coincident** constraint to center of circle and origin.
- Create a line and using **Fix to shape** constraint fix the end points of the line to the circle, apply **horizontal** constraint, and using **Fix horizontal distance** constraint fix distance to 50mm.
- Using **Trim edge** in **Sketcher geometries** remove lower half of the circle, resulting in a semi-circle. Leave sketch in sketch menu.
- Go to **Part Workbench** in **View menu** and open **revolve** feature in **Part menu**, click on **select line in 3D** and choose the horizontal line, make **create solid** true and press **Ok** to create a sphere.
- Choose the feature(**revolve**) and in **File Menu**

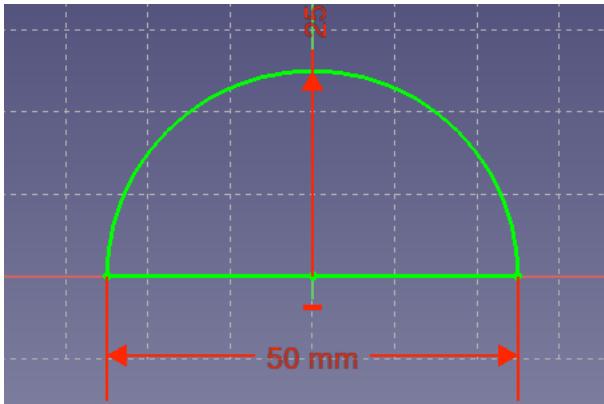


Fig. 3: Semi-circle

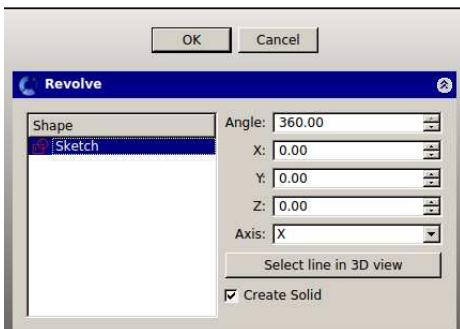


Fig. 4: revolve feature

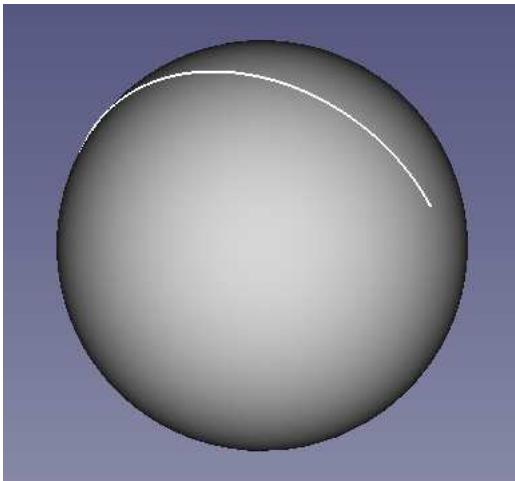


Fig. 5: Solid Sphere

export it as **Sphere.stl** in separate folder.

To 3D print the sphere(CAM),

- Open **Cura** Software and open **Sphere.stl** in it, then set **Layer height** to 0.2mm, **infill** to 50, and **infill pattern** to **Tri-Hexagon**.
- Make **Generate support** and **Build Adhesion**

true.

- **Slice** and generate the G-code and save it as **Sphere.gcode** file, which can be used to print.

The sliced model can be examined in Layer View. Layer view shows detailed path of the extruder of the printer.



Fig. 6: 3D-Print of Sphere

## 1.2 Hollow Sphere

To design Hollow-sphere(CAD),

- In FreeCAD go to **Sketcher Workbench**, and in xy-plane create two concentric circles of radius 25mm and 20mm and fix their center to origins.
- Create a vertical line of length 50mm, and fix its end points to the outer circle.
- Remove the edges towards left of the circle using **trim edges**, and also delete the line joining the circles, and close the sketch.
- Draw two vertical lines joining the ends of the two semi-circles.
- In **Part Workbench** click **create primitives** in **Part menu** and select a circle, choose the radius to be 12.5mm, in location (0,0,0,), direction Y-axis, press create and close.
- Open **Utility sweep** in **Part menu** load the sketch, select sweep path as the circle, make **create solid** and **Frenet** true, and press ok.

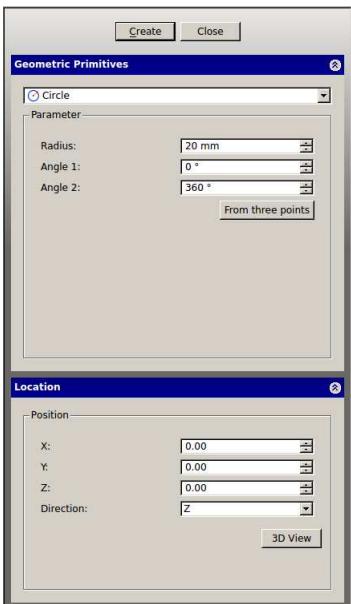


Fig. 7: Geometric primitives

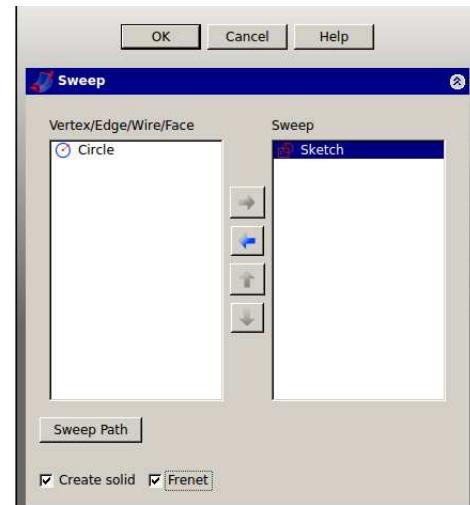


Fig. 9: Utility Sweep Feature

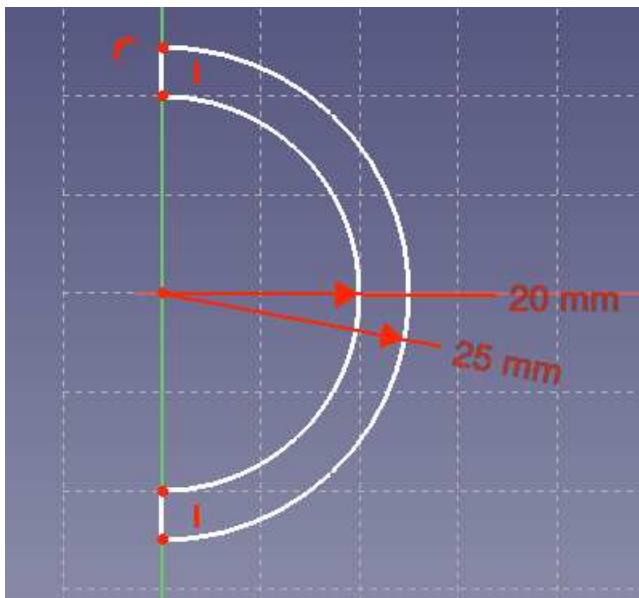


Fig. 8: Sketch for hollow sphere

- Select the **sweep** object and export it into a new folder as **Hollow\_Sphere.stl** file.

To 3D print the Hollow-sphere(CAM),

- Open **cura** in it open the **Hollow\_Sphere.stl** file.
- In **custom** settings set the **infill** to 50, **infill pattern** as **concentric**, and make **Generate Support** and **Build Adhesion** true.
- Slice the file and save it as **Hol-**

## **low\_Sphere.gcode.**

### 2 ELLIPSOID

#### 2.1 Solid Ellipsoid

To design Ellipsoid(CAD),

- In FreeCAD go to **Sketcher Workbench**, Create an ellipse in xy-plane using **conic section** tool. Select **Ellipse by center, major radius, point** from **Sketcher geometries** and create an Ellipse.
  - Select major axis of the ellipse(blue line) and apply horizontal constraint, make center of the ellipse coincident with origin.
  - Select major axis and set length to 70mm, and minor axis to 50mm.
  - Select the major axis and use **toggle** tool and make it a normal line(white color).
  - Using trim tool remove the lower part of the ellipse, resulting in a semi-ellipse, and close sketch.
  - In **Part Workbench** revolve the ellipse about the X-axis, resulting in an ellipsoid. Select the ellipsoid and export it in a new folder as **Ellipsoid.stl**.
- To 3D print the Ellipsoid(CAM),
- Open the **Ellipse.stl** in **cura**.
  - In custom settings set **Layer hight** to 0.2, **infill** to 40, and **infill pattern** to **Tri-Hexagons**, and

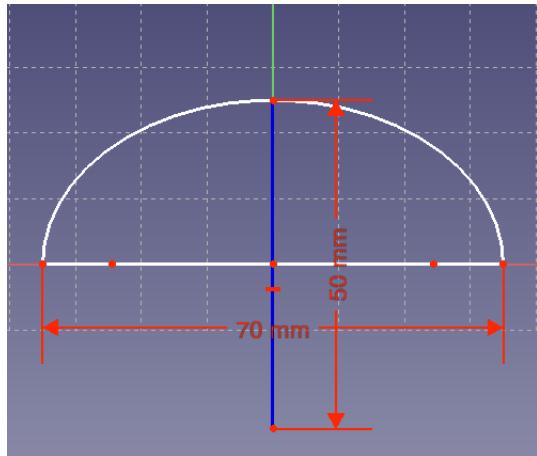


Fig. 10: Semi-Ellipse



Fig. 12: 3D-Printed model

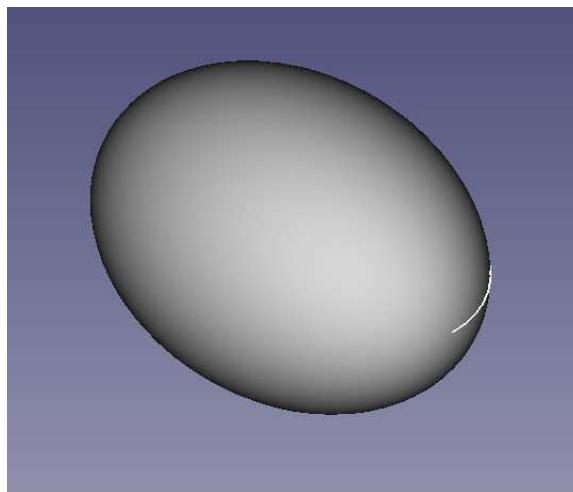


Fig. 11: Ellipsoid

## 2.2 Hollow Ellipsoid

**Problem 2.1:** Design a hollow Ellipsoid, such that it has two holes on opposite sides.

**Hint:** Refer Steps used in Hollow Sphere[1.2], create cylinder and perform **cut** operation from **Boolean** present in **Part menu** between hollow ellipse and cylinder to result in the shape.

**Solution:**

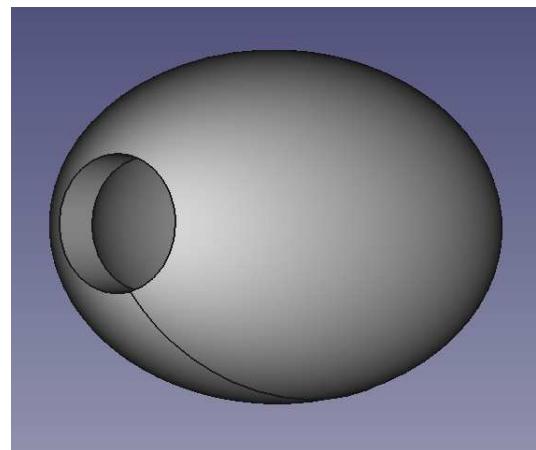


Fig. 13: Hollow ellipsoid with holes on side

make **Generate Support** and **Build Adhesion**, slice it, and save it as Ellipse.gcode.

## 3 HYPERBOLOID

The idea of designing a hyperboloid is to design an reverse ellipse resulting a Hyperboloid, as FreeCAD doesn't provide any specific tool for hyperbola.

### 3.1 Solid hyperboloid

To Design a Solid Hyperboloid(CAD),

- In FreeCAD go to **Sketcher Workbench**, on xy-plane create a sketch in Conic sections by selecting **Arc of Ellipse by center, major radius, end points**, and select the center at (30,0), minor radius as 15mm, major radius as 35mm, and place random end points.
- Use the **coincident** constraint and fix the end points to the major axis, make the major axis vertical.
- Fix the Length of minor axis as 30mm and major axis as 70mm, fix the center of the arc to th x-axis, Fix the distance between origin and center of the ellipse as 30mm.
- Draw two horizontal lines such that their end points lie at the vertices of the arc, and y-axis.
- draw vertical line conneting the end points of the two lines on the y-axis, and close the sketch.
- In **Part Workbench** use **revolve** tool, revolve the sketch about Y-axis, and make **create solid true**.

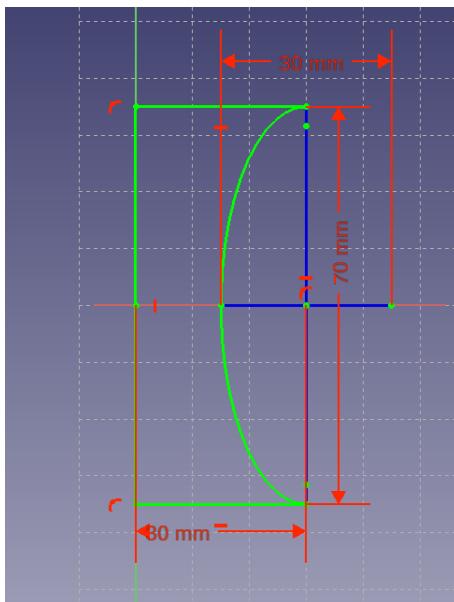


Fig. 14: Sketch of Solid Hyperboloid

- Select the solid Hyperboloid and export it as **Hyperboloid.stl**

To 3D print an Solid Hyperboloid,

- Open **Hyperboloid.stl** in **cura**.
- In custom settings set **infill** to 40 and **infill pattern** to Tri-hexagon, make **Generate support** and **Build Adhesion** true.

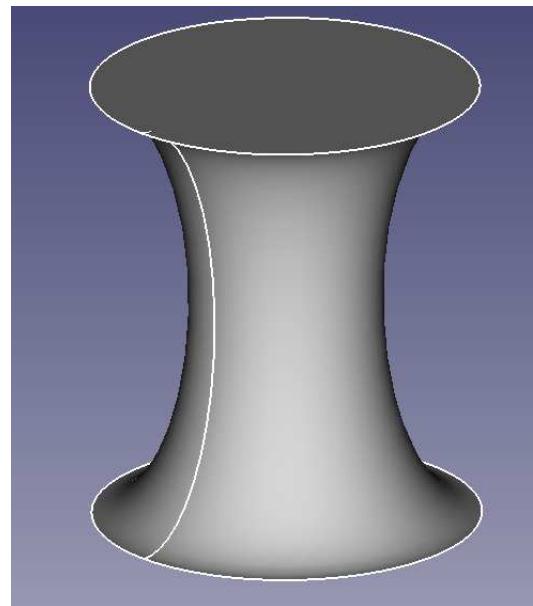


Fig. 15: Solid Hyperboloid

- Slice the file and save it as **Hyperboloid.gcode**.

### 3.2 Hollow Hyperboloid

**Problem 3.1:** Design an Hollow Hyperboloid.

**Solution:**

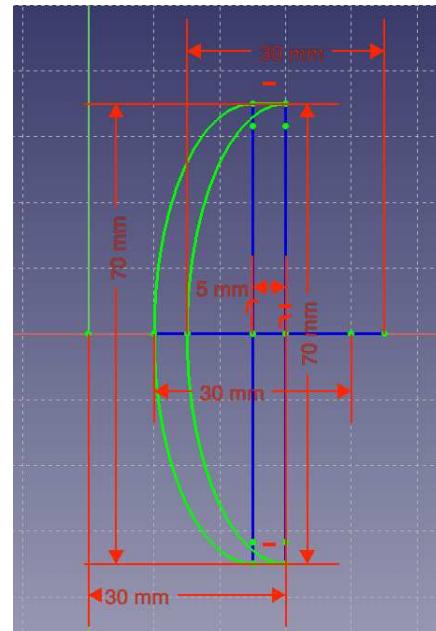


Fig. 16: Sketch of Hollow Hyperboloid

## 4 PARABOLOID

The idea of designing a Parabola is to design a elliptical curve and use it to make a soild paraboloid.

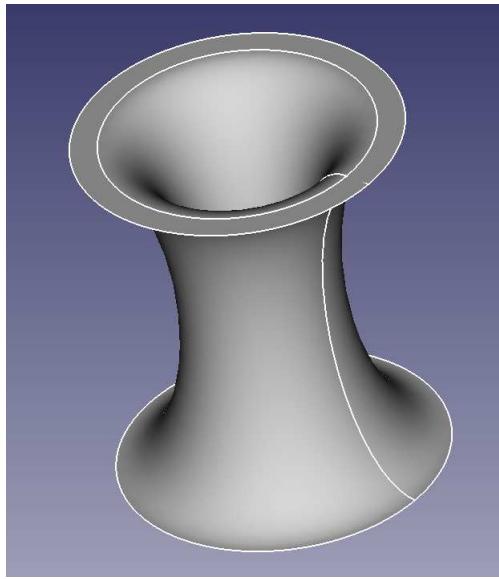


Fig. 17: Hollow Hyperboloid

#### 4.1 Solid Paraboloid

To Design a Paraboloid,

- In FreeCAD go to **Sketcher Workbench** and using **Arc of ellipse by center, major radius, minor radius, end points** draw an elliptical arc with center at origin, major radius 70mm, and minor radius 25mm.
- Fix one of the end points to the minor axis, and other to the major axis. The resulting shape is a semi-parabola, make the major axis vertical.
- Create two lines joining the end points of the arc to center using **Sketcher geometries**, and close the sketch.

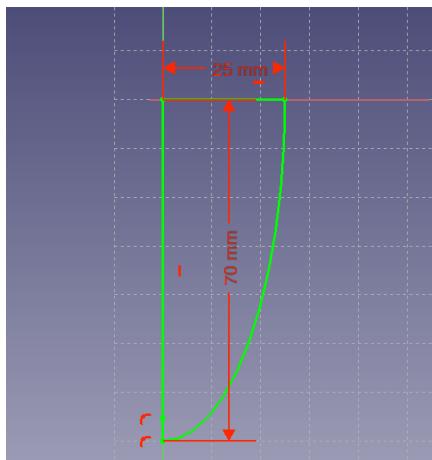


Fig. 18: Sketch of semi-parabola

- Using the **revolve** feature revolve the shape about Y-axis(make **create solid** true) to result

in solid parabola.

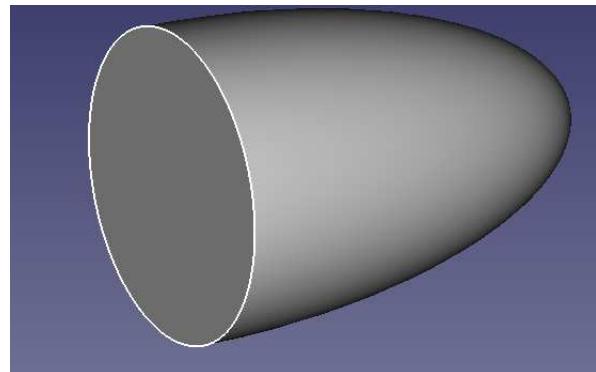


Fig. 19: Solid Paraboloid

#### 4.2 Hollow Paraboloid

**Problem 4.1:** Design a Hollow Paraboloid.

**Hint:** Refer the steps used for Hollow Sphere[1.2] and Hollow Sphere[1.2].

**Solution:**



Fig. 20: 3D-Print of Hollow Paraboloid

#### 5 CONCLUSION

In Computer Aided Designing the same design can completed in many different ways but which way is faster, or more efficient can be learnt through experience and practice. Each design shown in manual can be designed in some or the other way, so try combinations of various tools to create your own new design and keep exploring.

#### 6 REFERENCES

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