

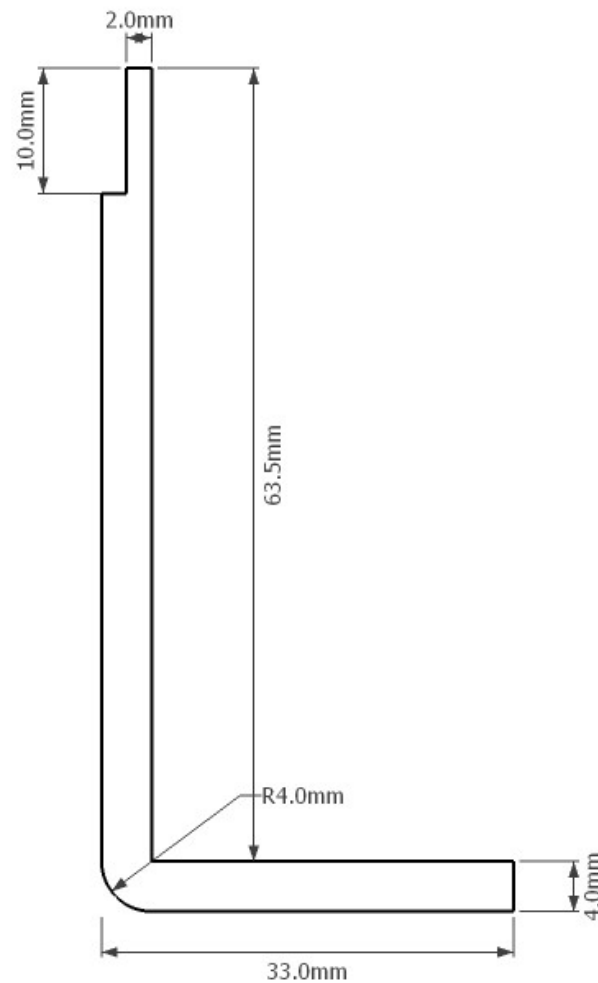
Modelling a can with SketchUp

Start with a front view

Camera → Standard Views → Front

Create a Profile

Dimensions:



Create a Profile in One Go

Draw a line beginning at the origin.

Pull the pointer right along the red axis, but do not click.

Type “33”.

Pull the pointer up parallel to the blue axis.

Type “4”.

Pull the pointer left, parallel to the red axis.

Type “29”.

Create a Profile in One Go

Pull the pointer up parallel to the blue axis.

Type “63.5”.

Pull the pointer left along the red axis.

Type “2”.

Pull the pointer down parallel to the blue axis.

Type “10”.

Pull the pointer left along the red axis.

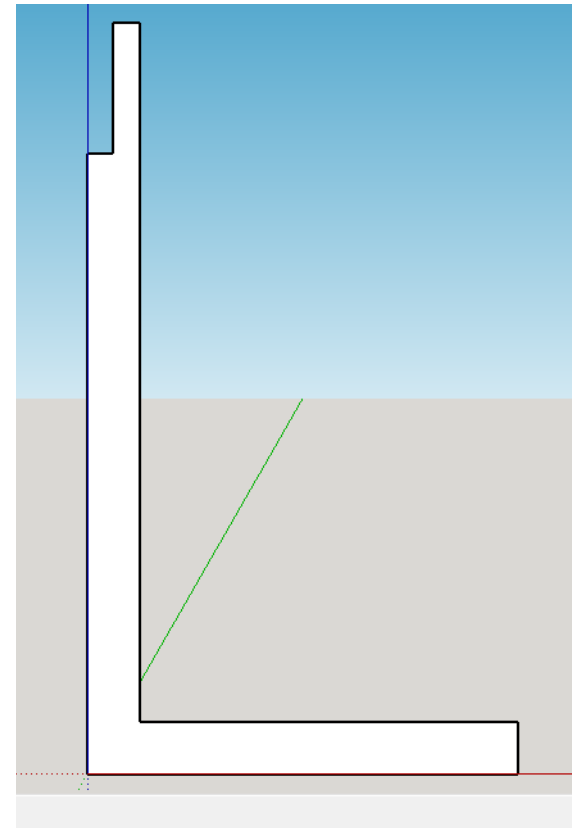
Type “2”.

Create a Profile in One Go

Pull the pointer down along the blue axis.

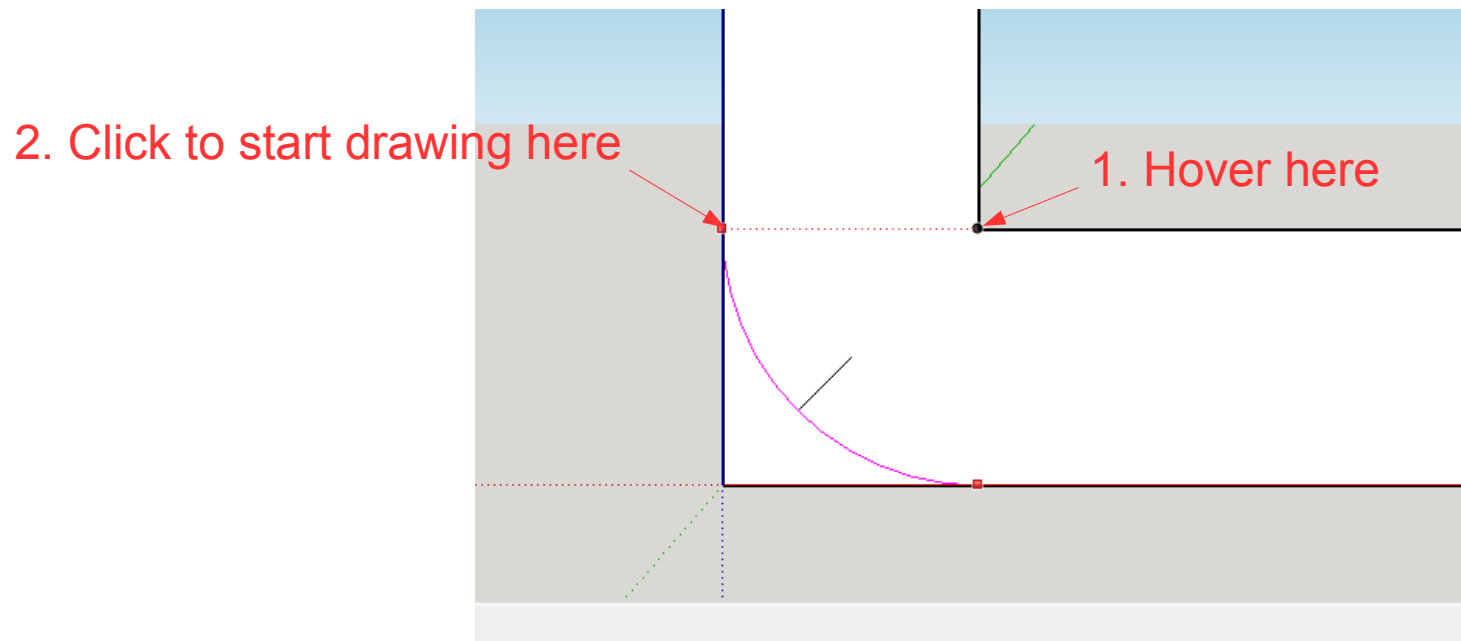
Click once on the endpoint: the origin, where you started.

You should have a shape like the one on the right.



Round the Profile

Draw an arc on the bottom right of the shape:
Hover the pointer over the inside corner, then
move the pointer over to the opposite edge and
click once to start the arc.

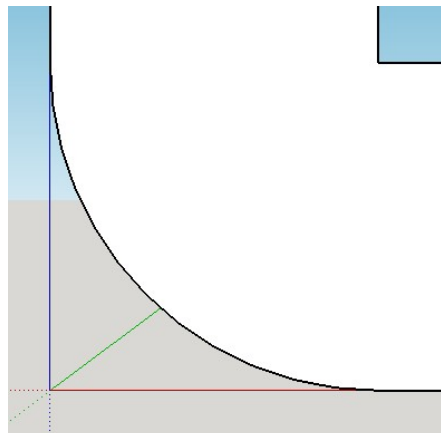


Round the Profile

Pull the pointer down towards the bottom edge, opposite the hover corner.

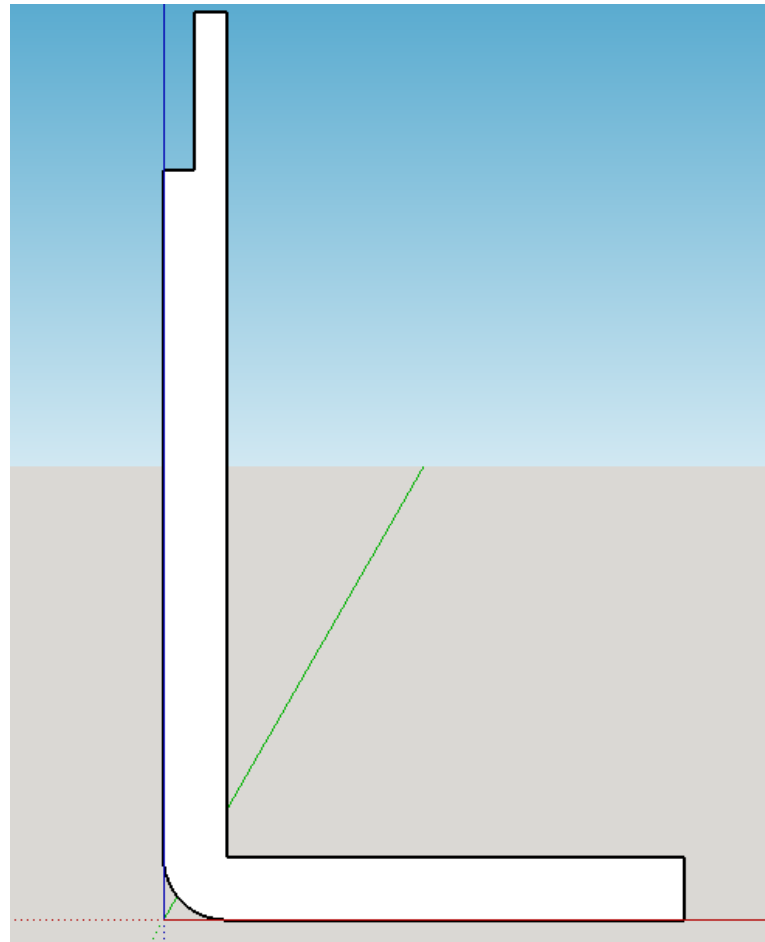
Watch the arc colour. When it changes from cyan to magenta, the arc is 90° .

Double-click to complete the arc.



Create a Profile

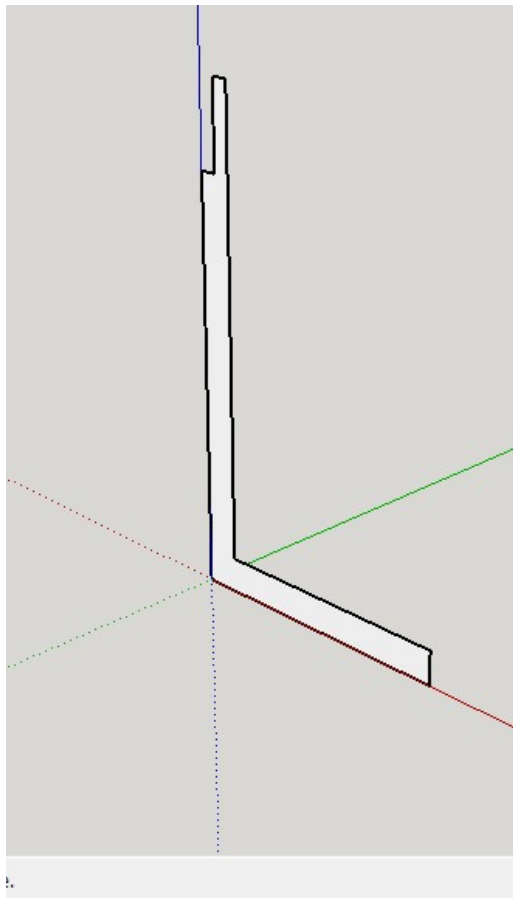
Your profile should now look like this:



Create an Extrusion Path

Switch the camera view to Isometric:

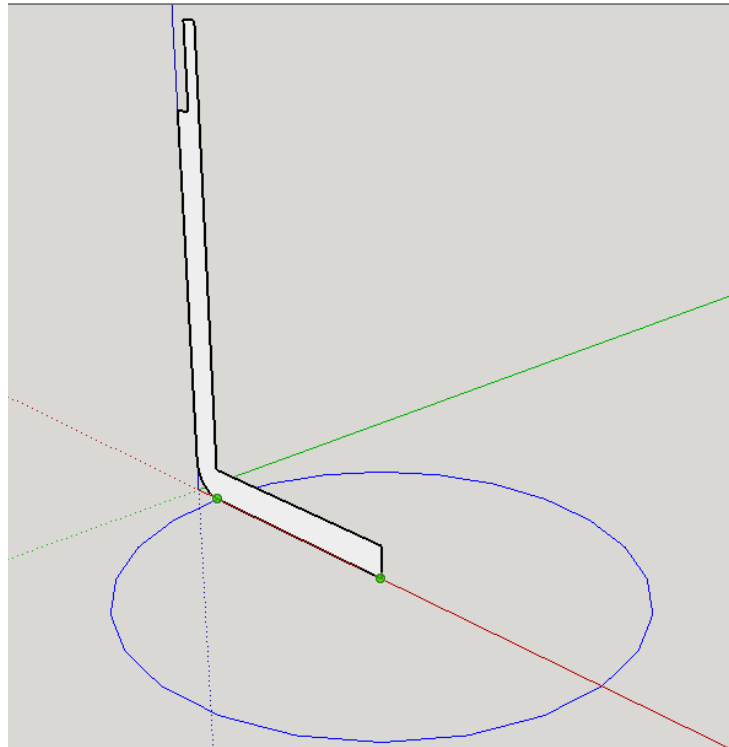
Camera → Standard Views → Iso



Create an Extrusion Path

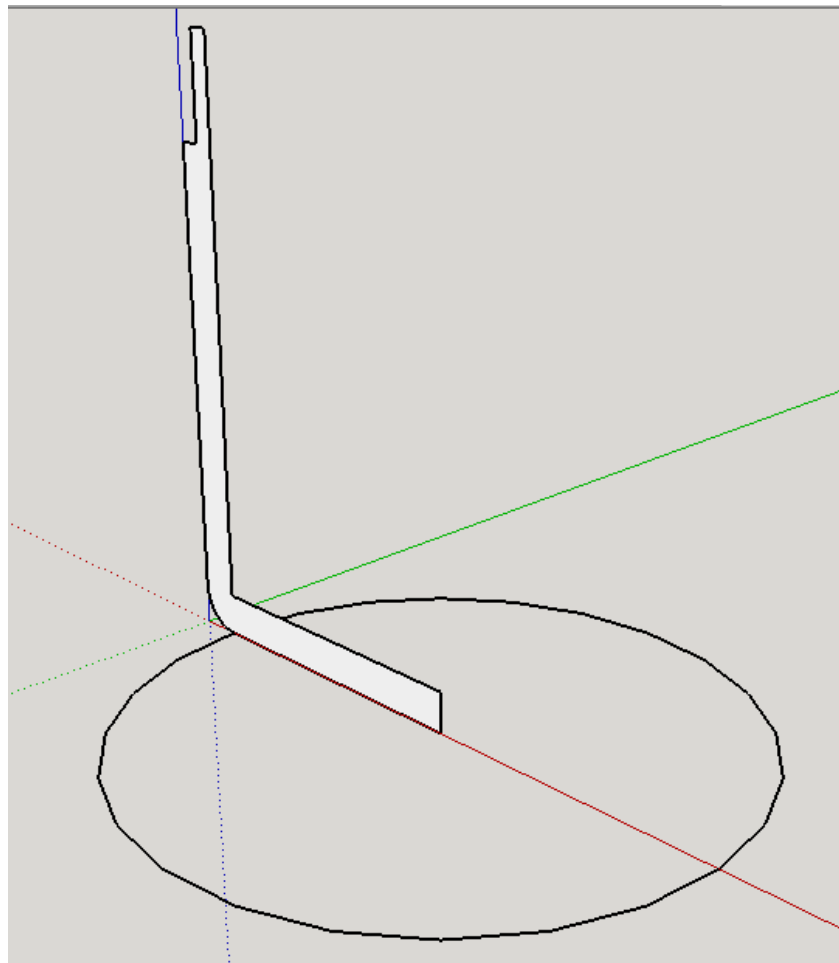
Draw a circle in the horizontal plane.

Use the bottom right corner of the profile as the centre. Use the end of the arc as the radius.



Create an Extrusion Path

Delete the circle face, leaving just the edge.

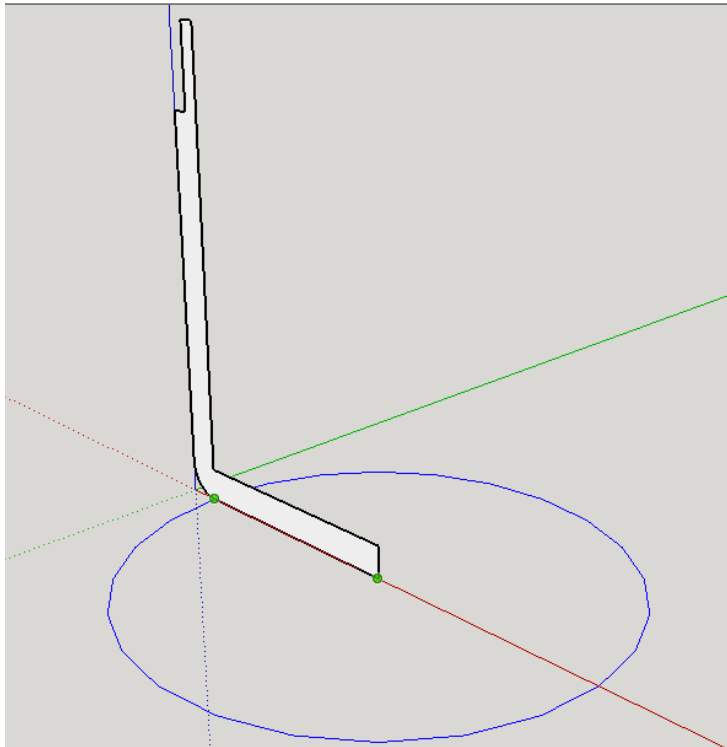


Follow the Circle

Select the circle edge on its own.

Click on the Follow Me tool.

Click on the face of the profile.



Final Result

