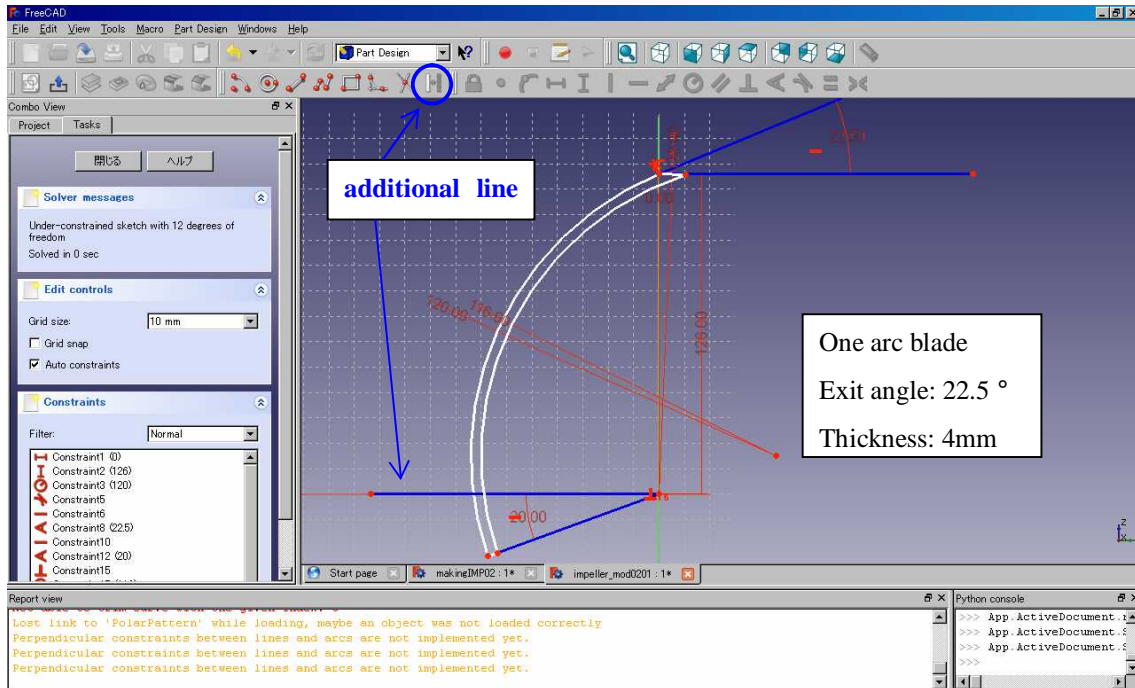


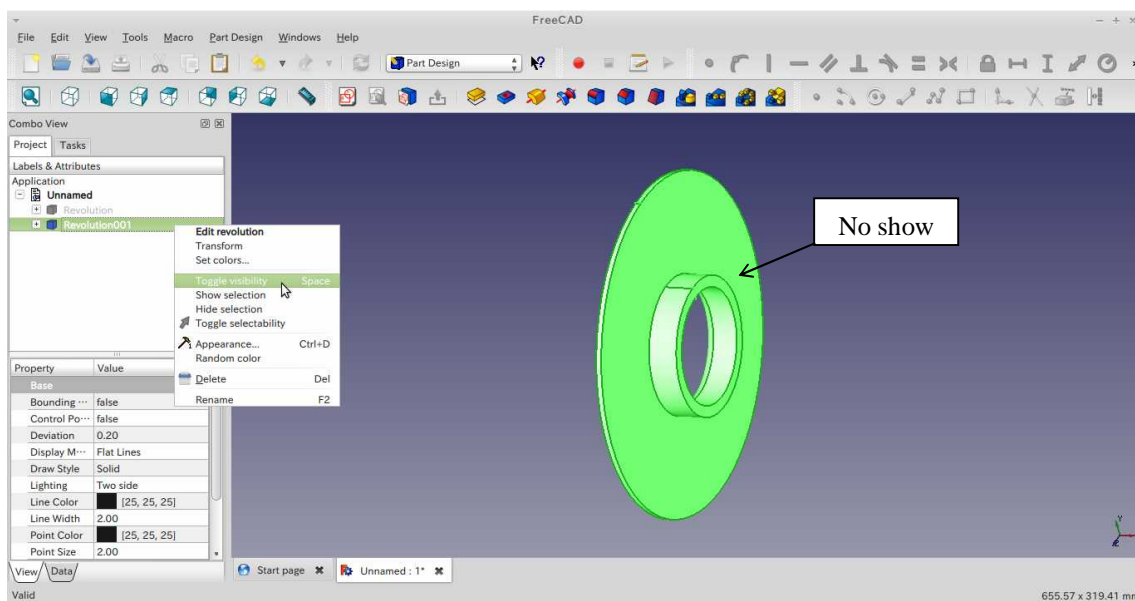
### Create blades

The complete drawing of the blade is shown below.

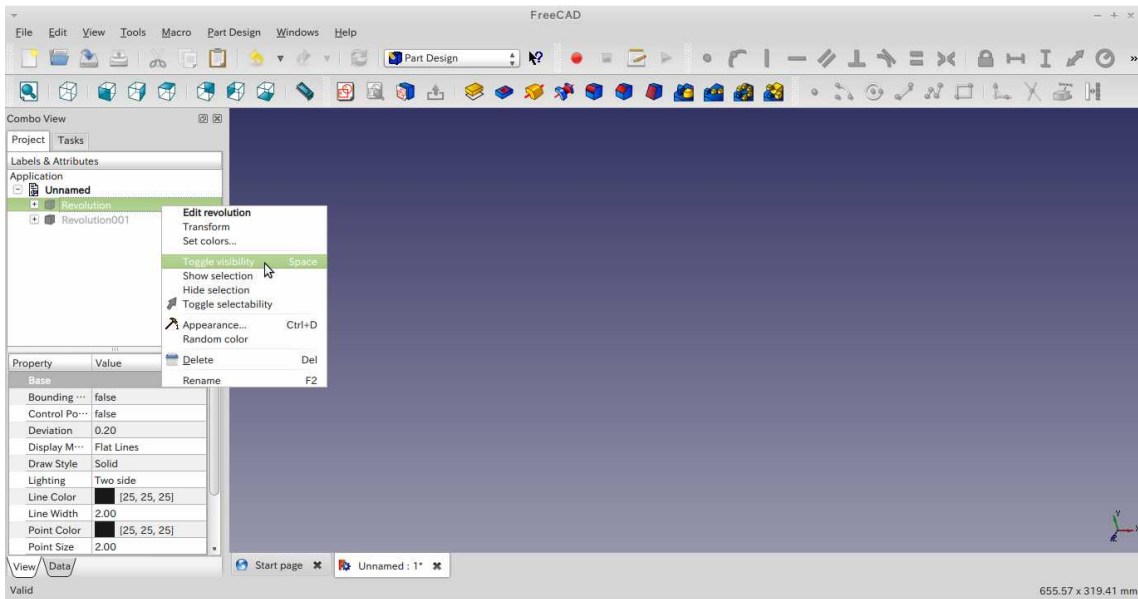
The making procedure is shown as follows.



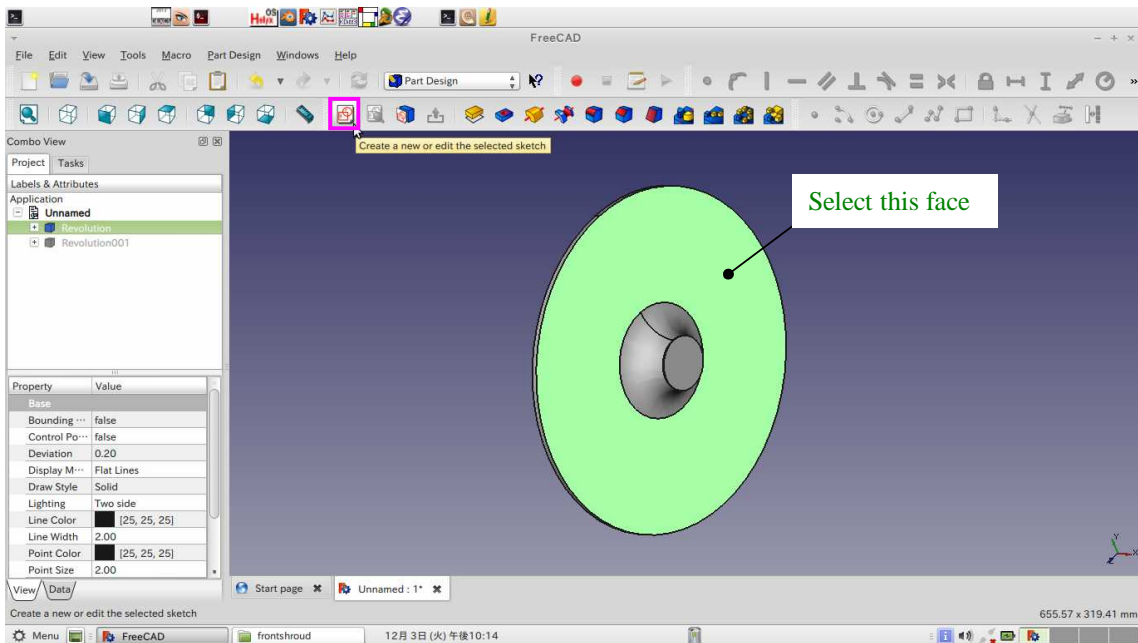
Click "Revolution001"(front-shroud) and right-click. Then, select "Toggle visibility".



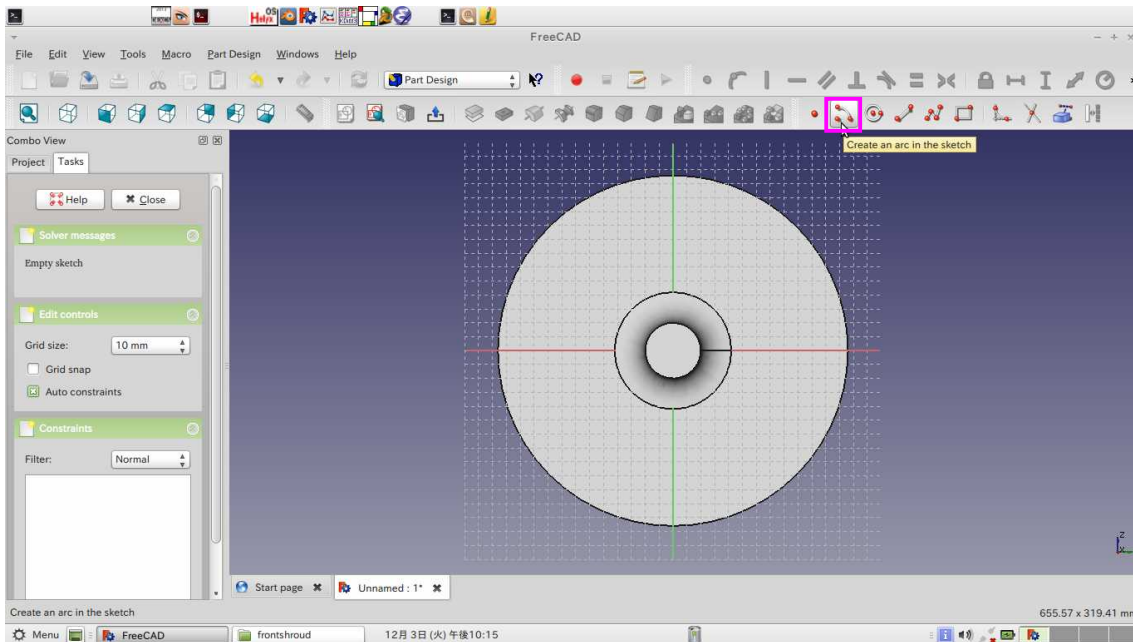
Click “Revolution” (back-shroud) and right-click. Then, select “Toggle visibility”.



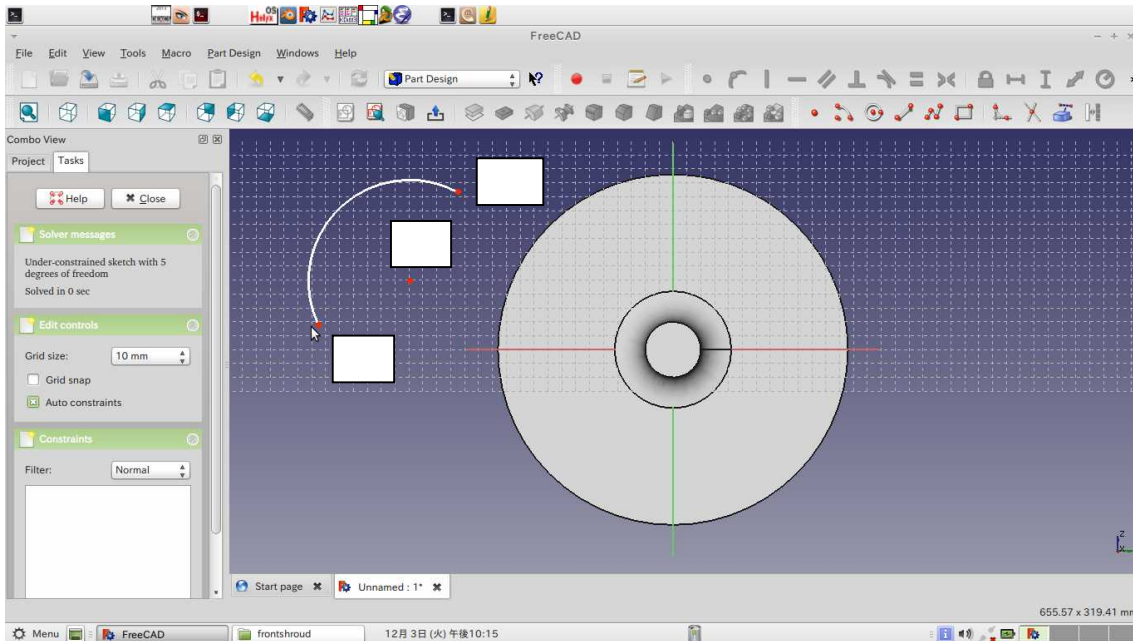
Click the face shown in the figure and click “Create a new or edit the selected sketch” icon.



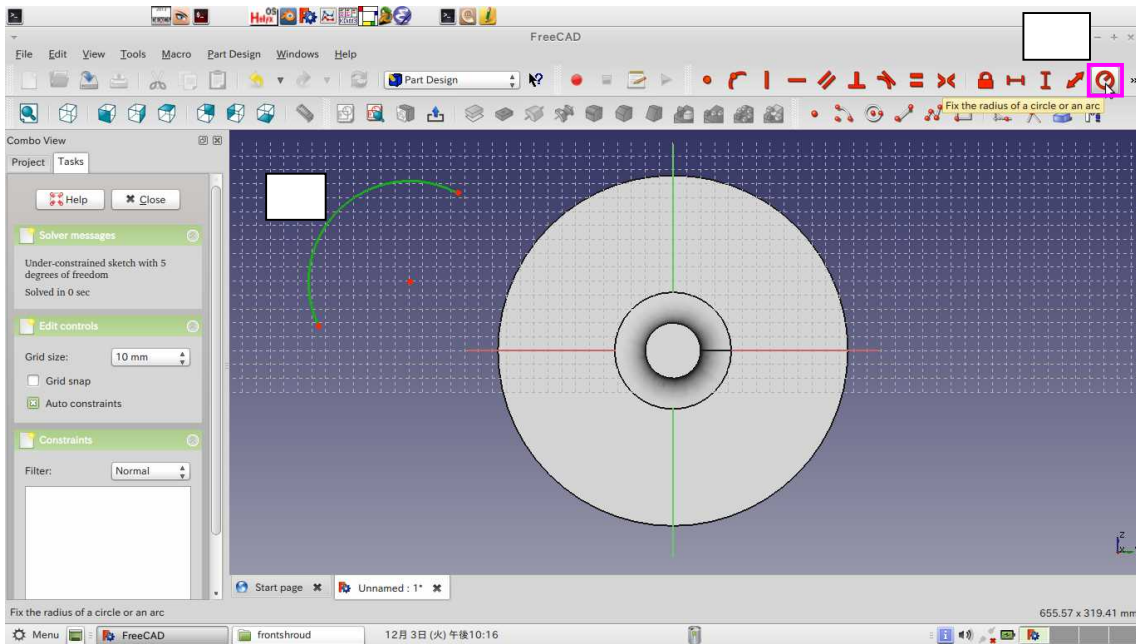
Click “Create an arc in the sketch” icon.



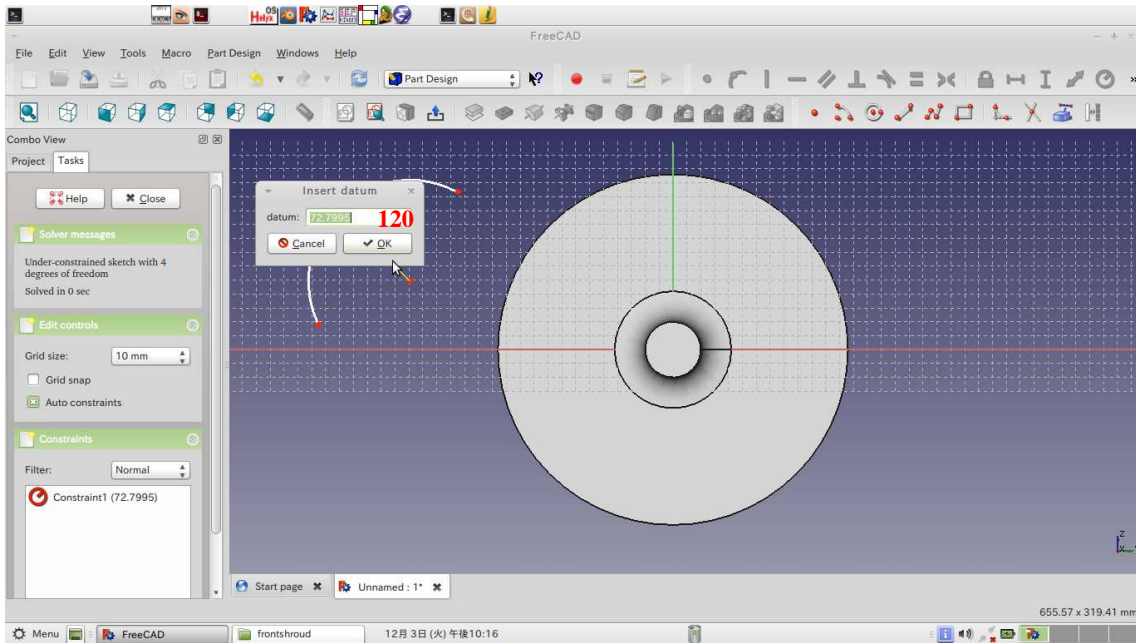
Click three times in the sketch window. An arc is created.



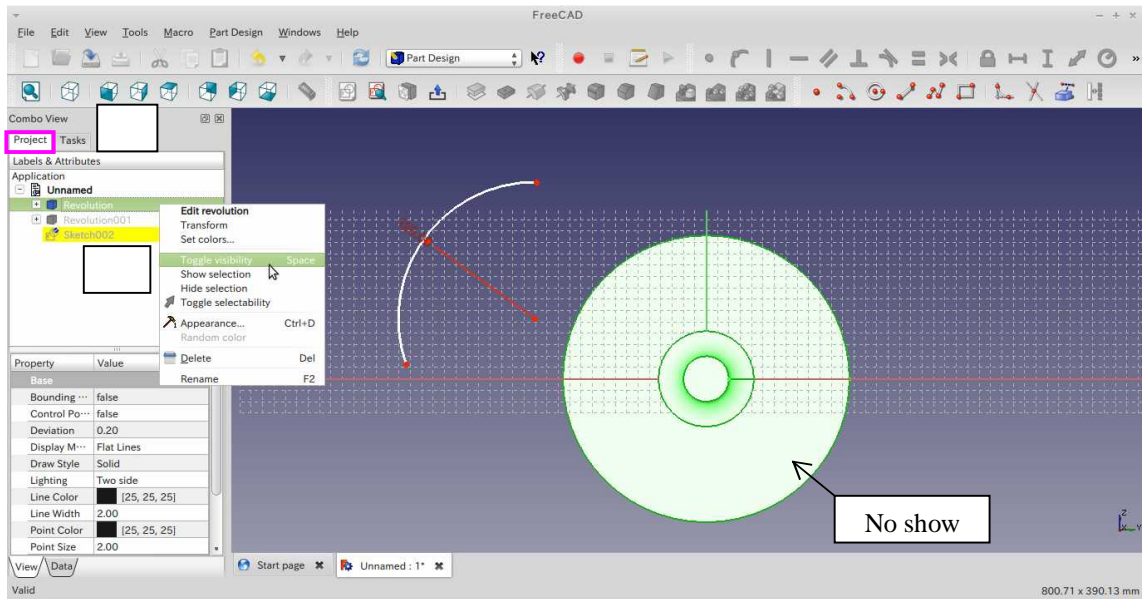
Select the arc and click “Fix the radius of a circle or an arc” icon .



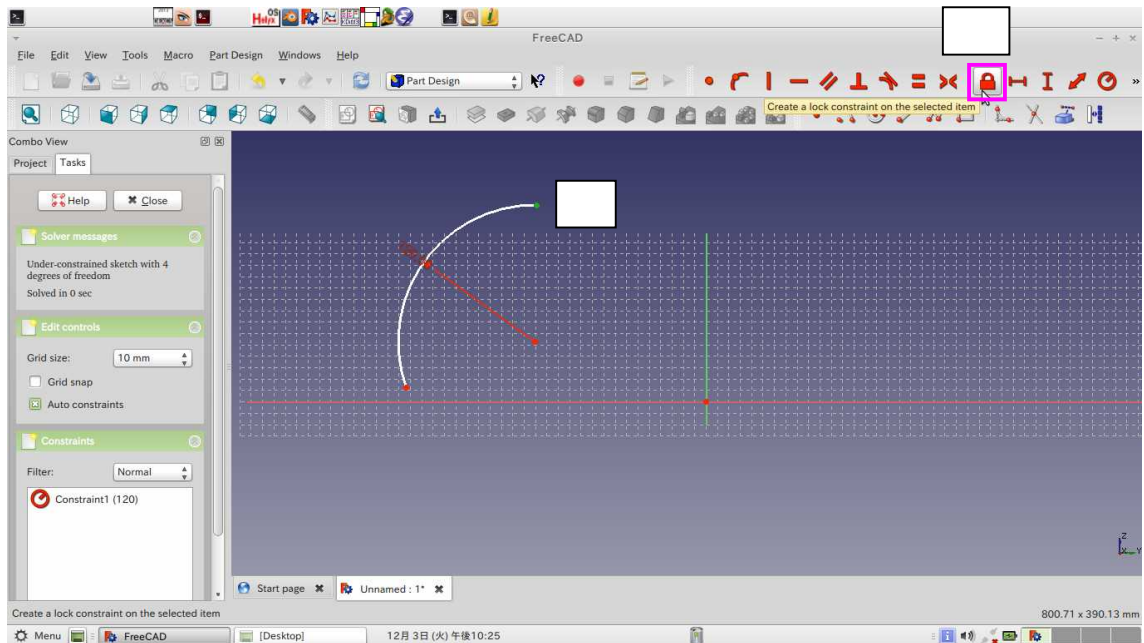
Double-click the dimension and replace the datum: 120



Click the “Project” tab . Select “Revolution” and right-click and select “Toggle visibility”.

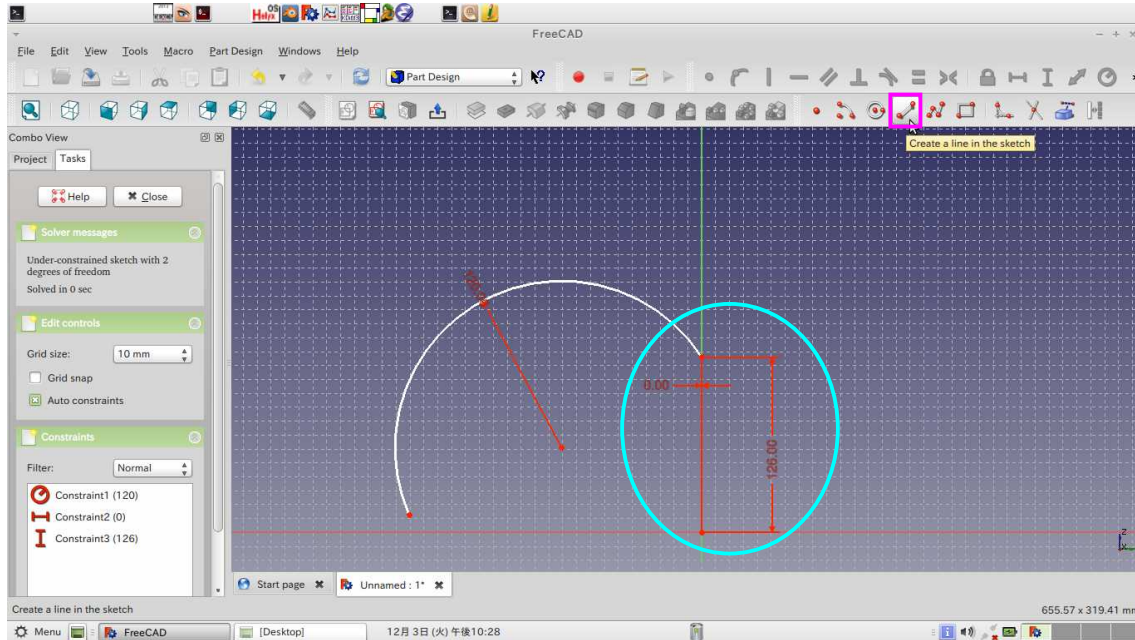


Select the point shown in the figure and click “Create a lock constraint on the selected item” icon .

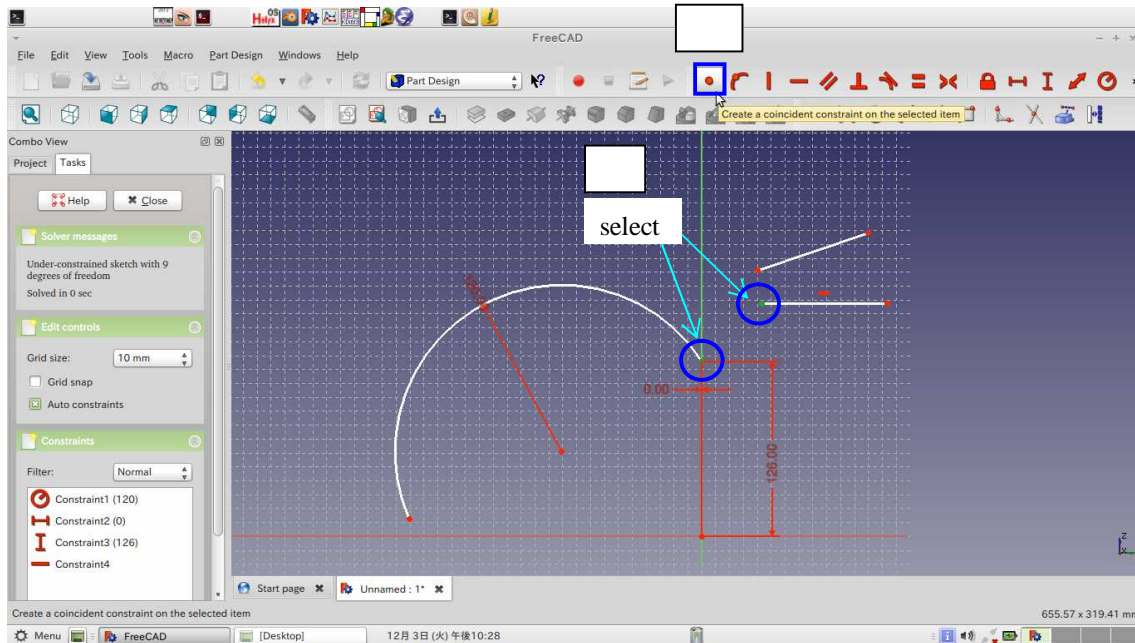


Double-click the dimension and set  $(Y, Z) = 0, 126$  as shown in the figure.

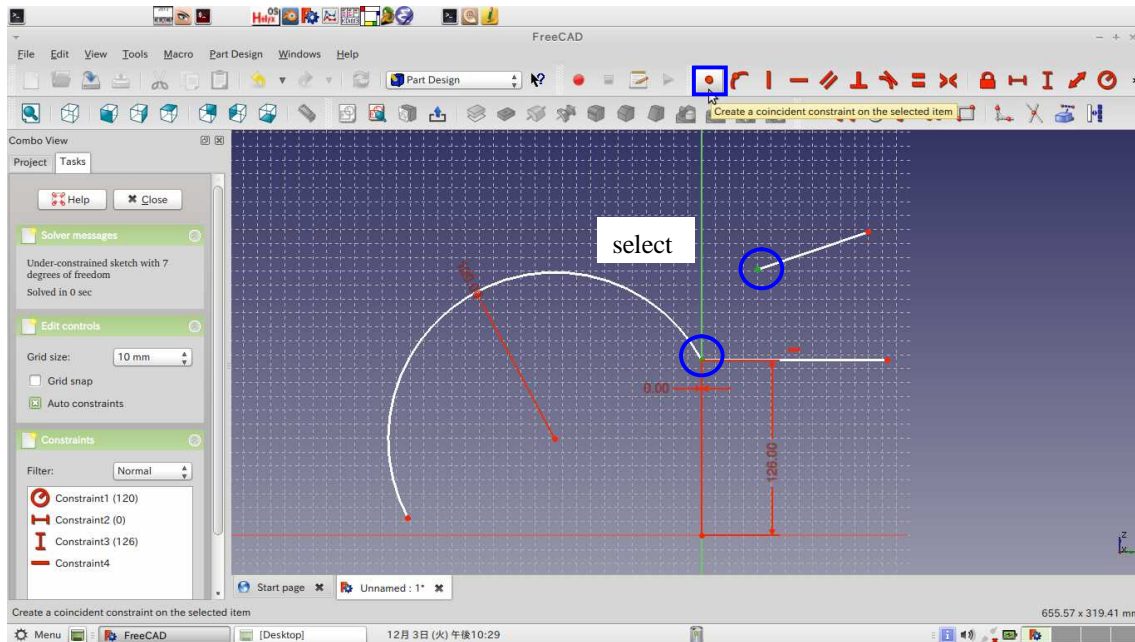
Click “Create a line in the sketch” icon.



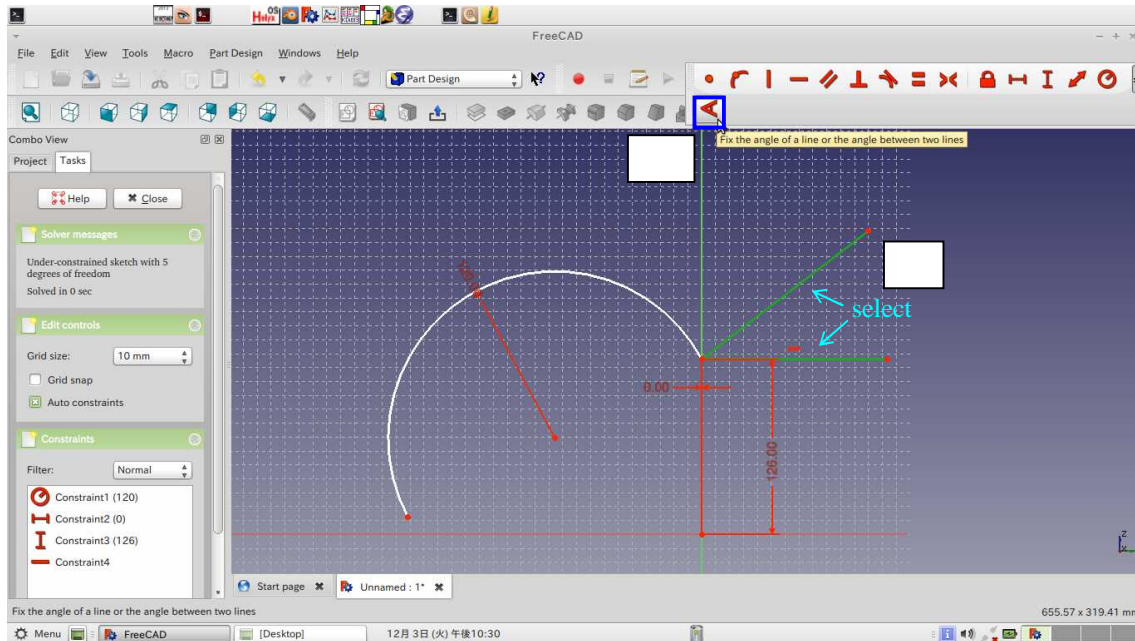
Add two lines. Select two points and click “Create a coincident constraint on the selected item” icon.



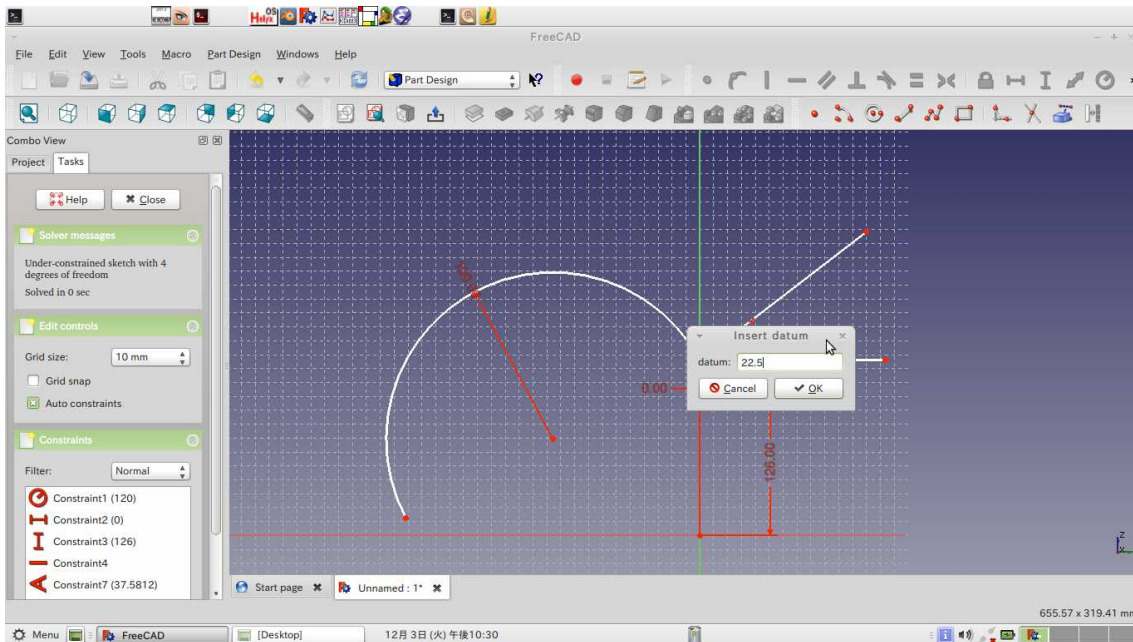
Implement same operation for another line.



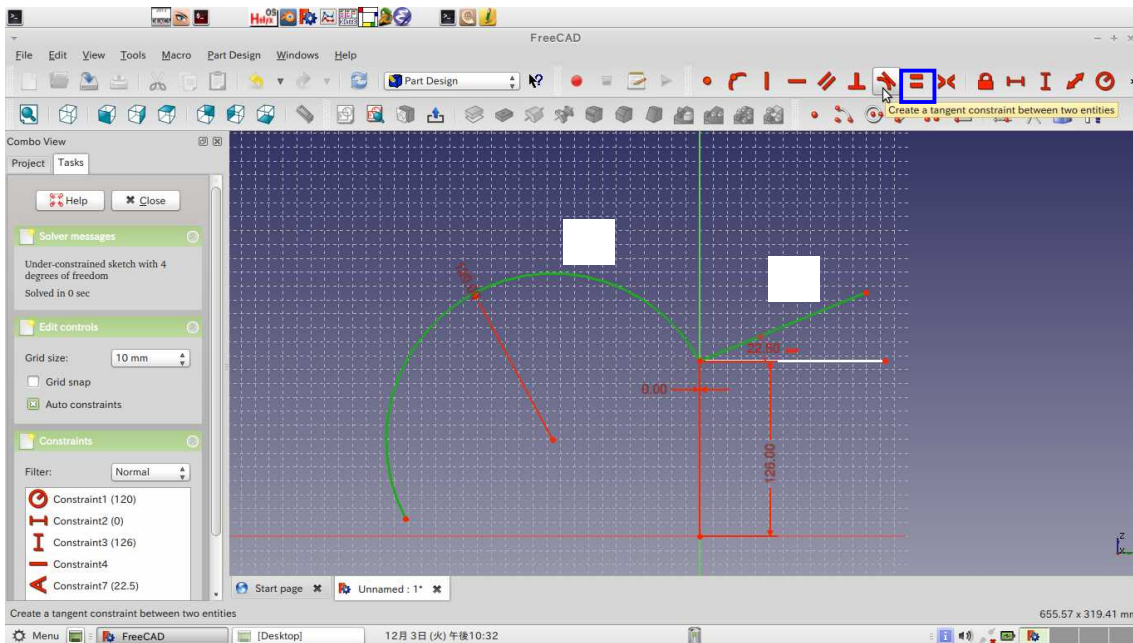
Select two lines and click “Fit the angle of a line or the angle between two lines” icon.



Double-click the dimension and replace the datum: 22.5 (recommended by A., J., Stepanoff)

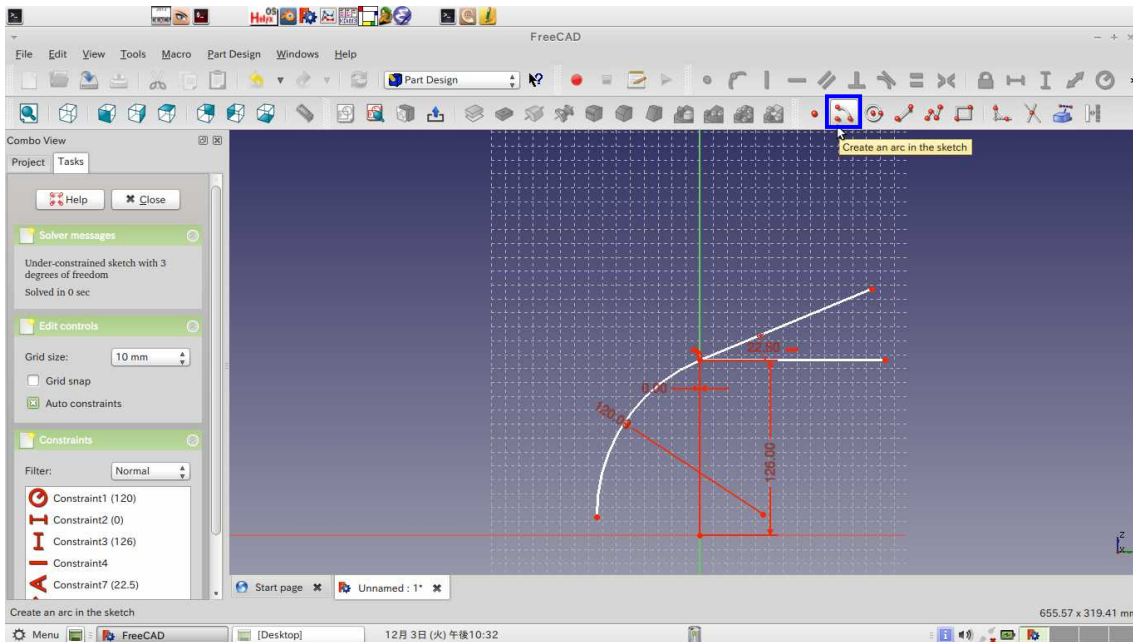


Select the arc and the line. Then, click the “Create a tangent constraint between two entities” icon.

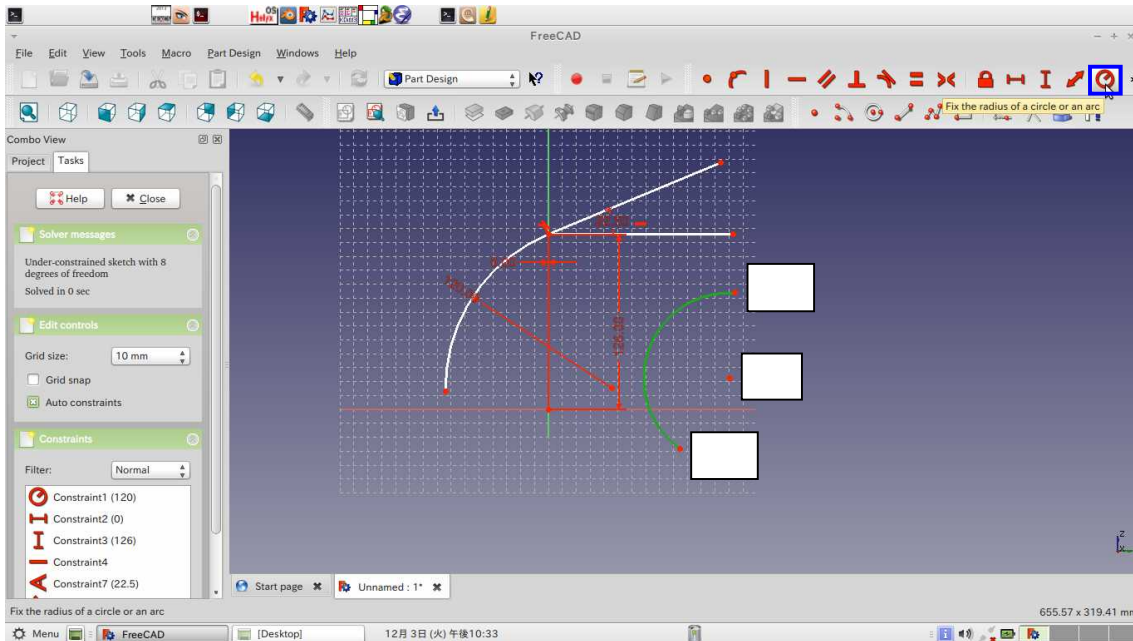




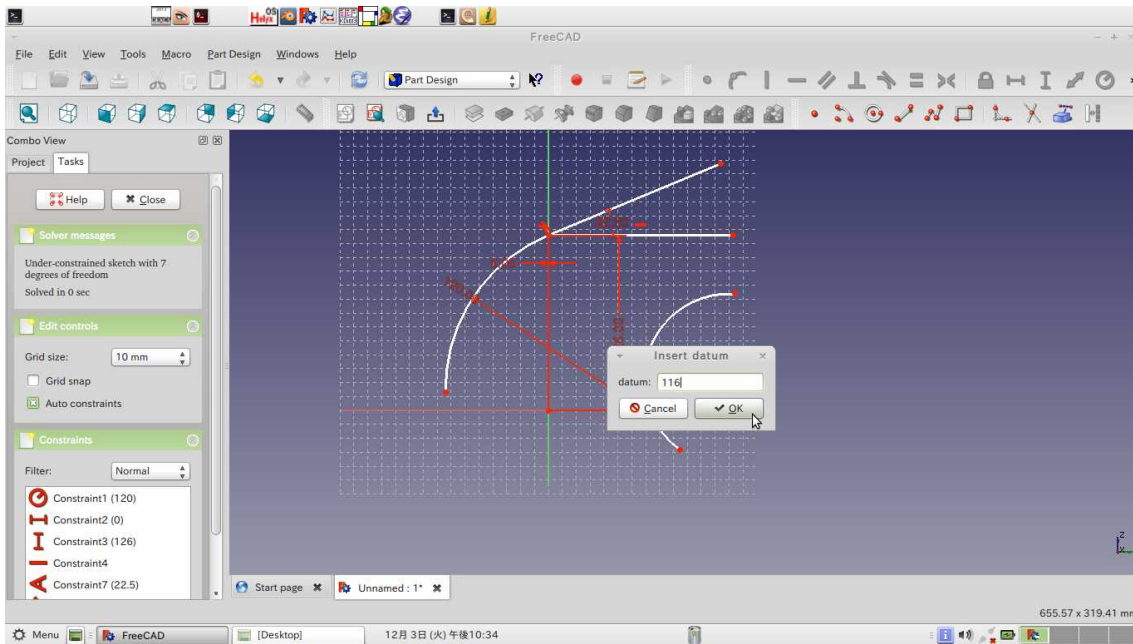
Click “Create an arc in the sketch” icon.



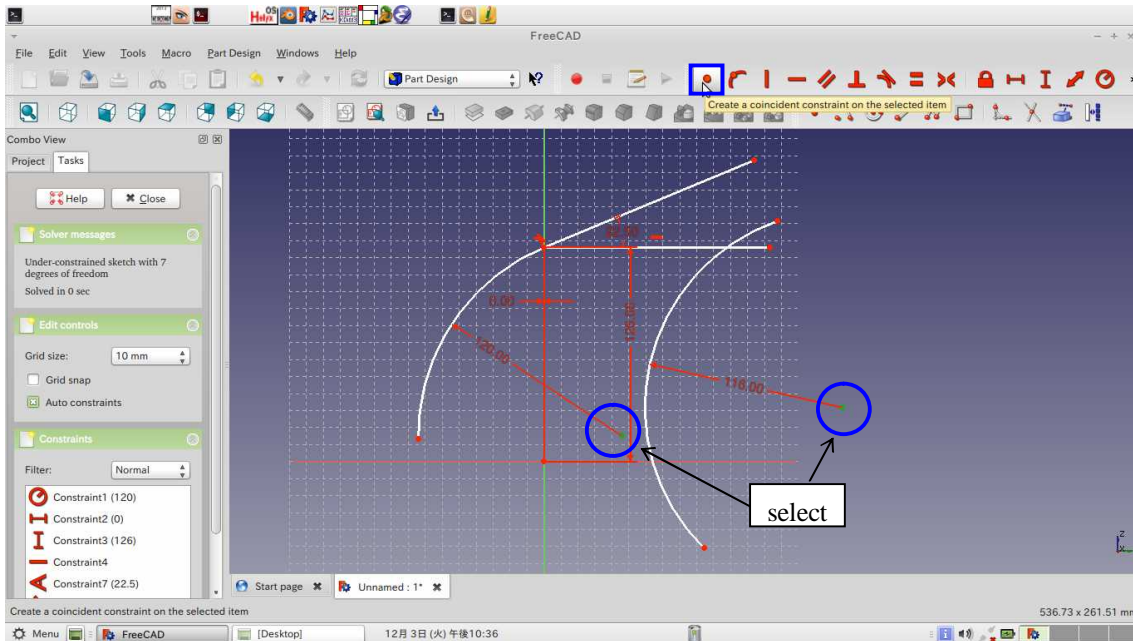
Click three times in the sketch window and click “Fix the radius of a circle of an arc” icon.



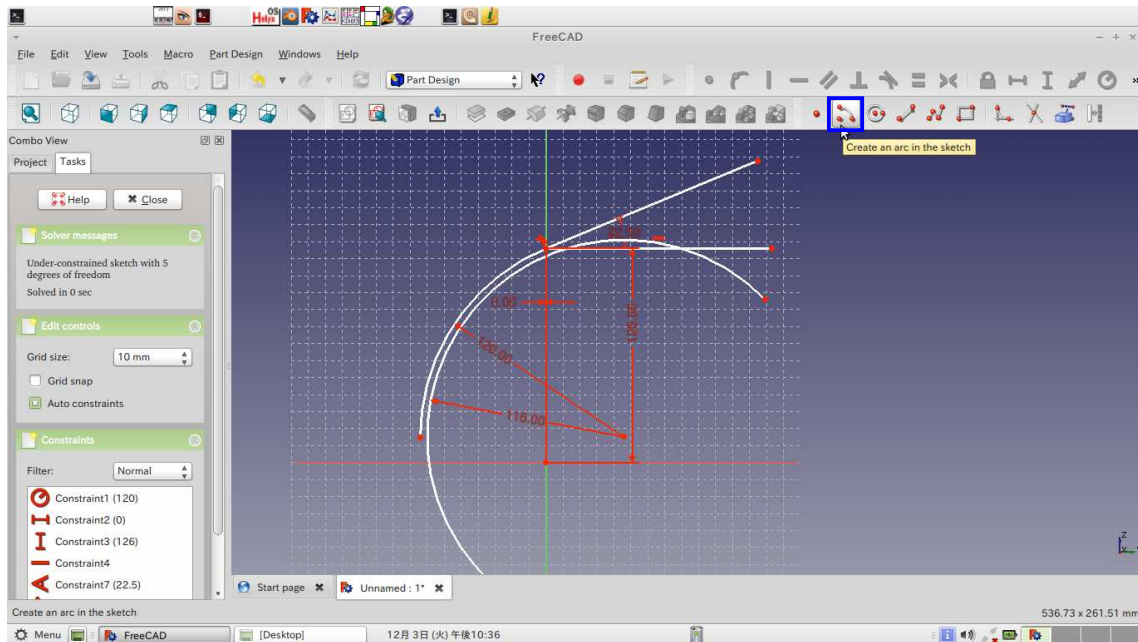
Double-click the dimension and replace the datum: 116



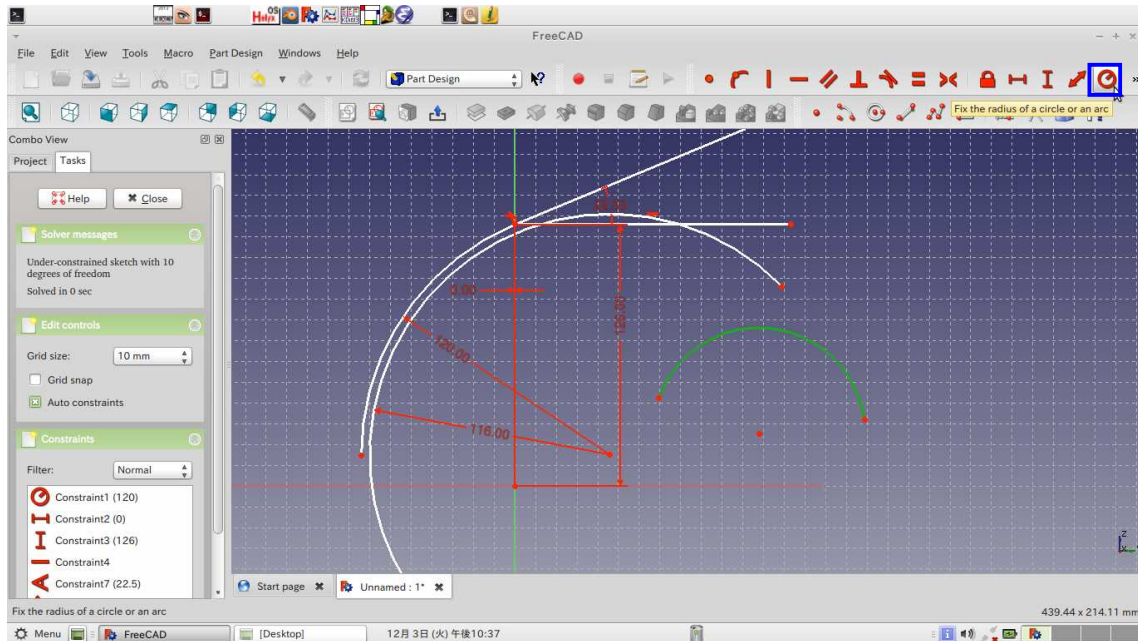
Select two points (centre of the arc) and click “Create a coincident constraint on the selected item” icon.



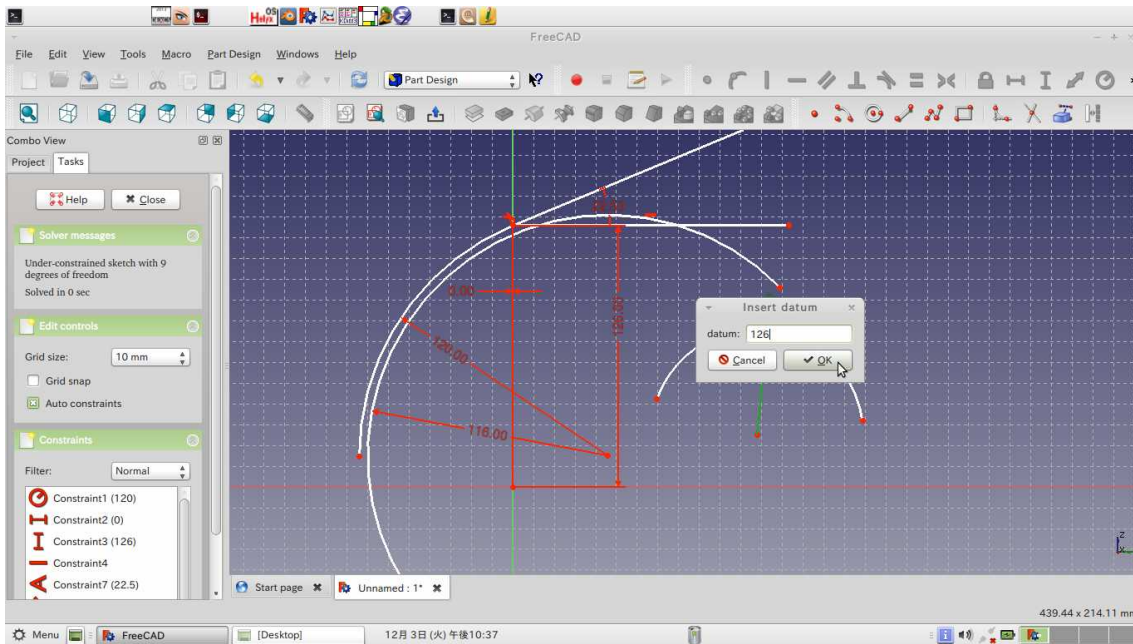
Draw a trailing Edge. Click “Create an arc in the sketch” icon.



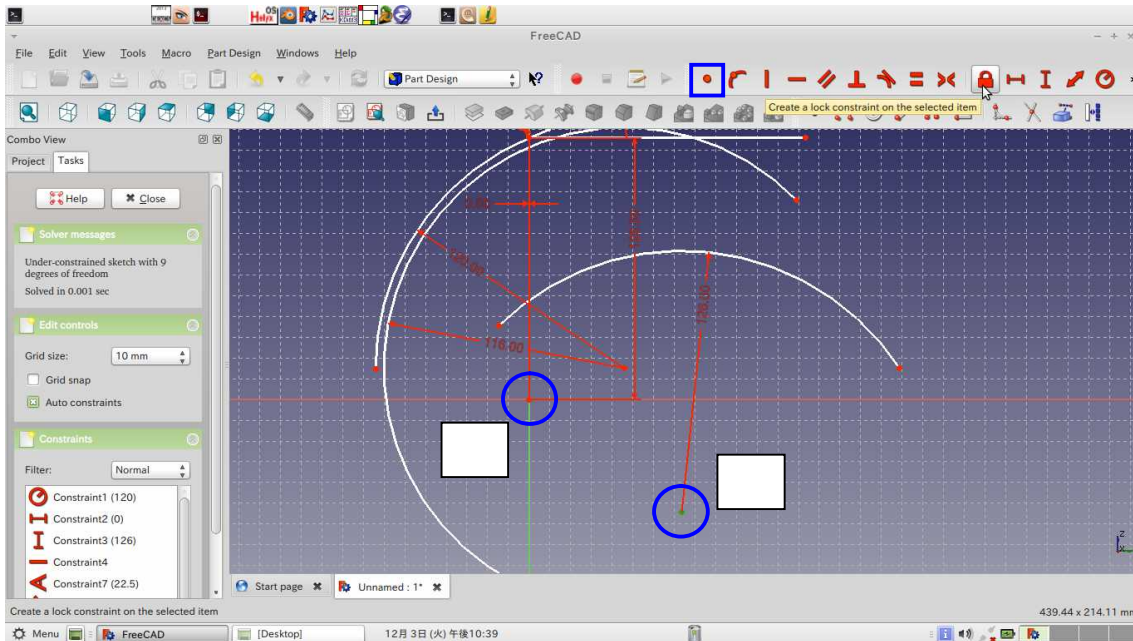
Click three times in the sketch window. Then, an arc is created.  
Select the arc and click “Fix the radius of a circle or an arc” icon.



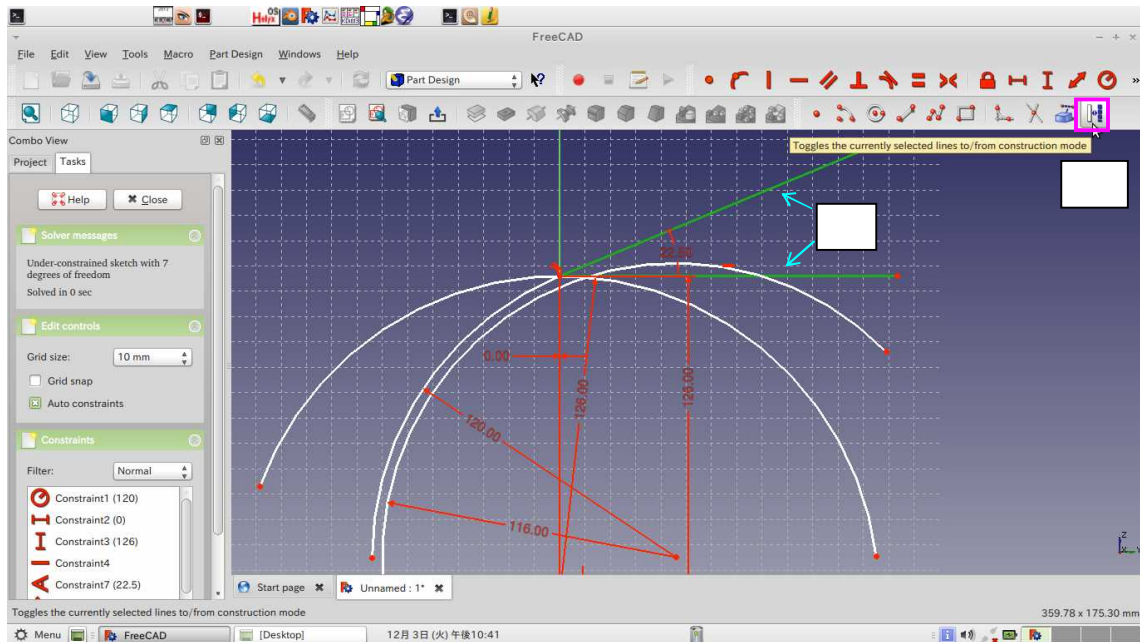
Double-click the dimension and replace the datum: 126



Select two points and click “Create a coincident on the selected item” icon.

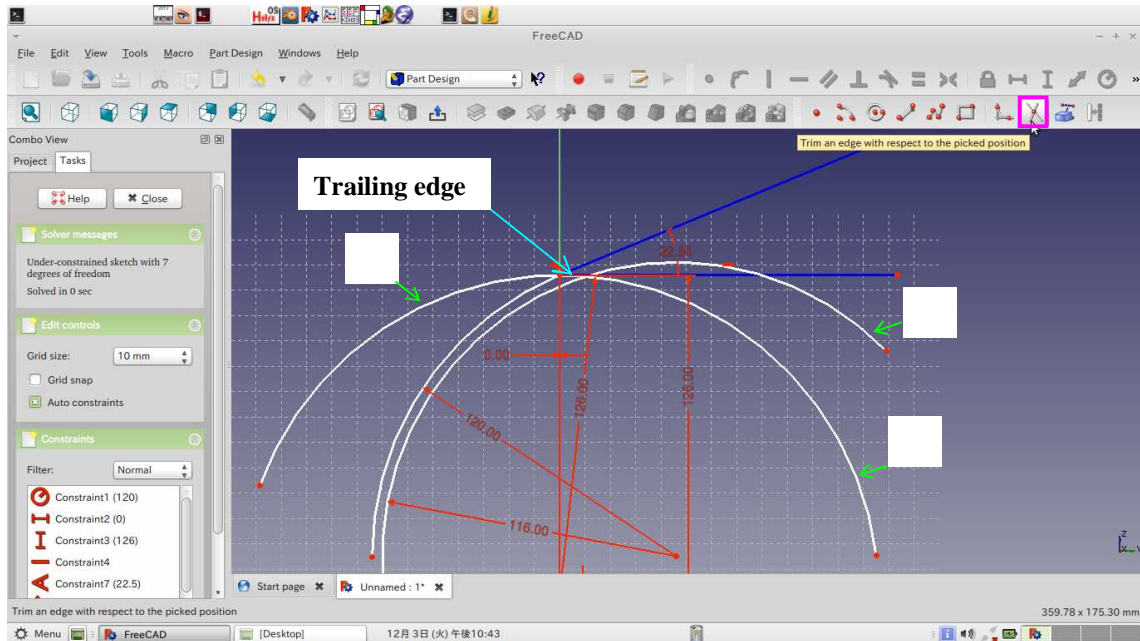


Select two lines and click “Toggles the currently selected lines to/from construction mode” icon.

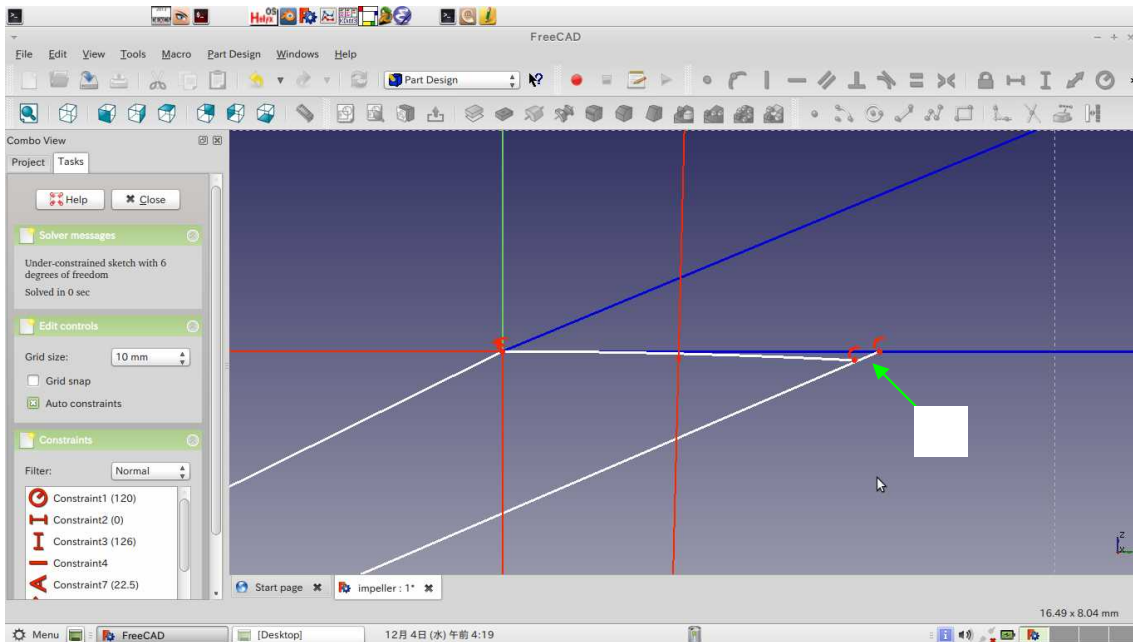


Click “Trim an edge with respect to the picked position” icon.

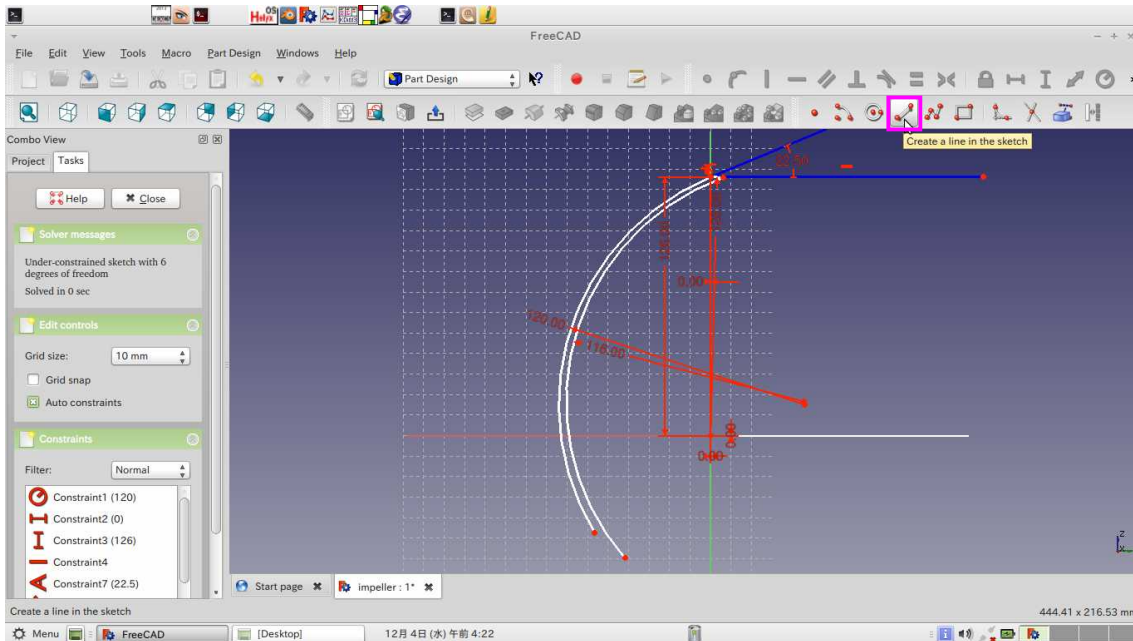
Click the arc from to .



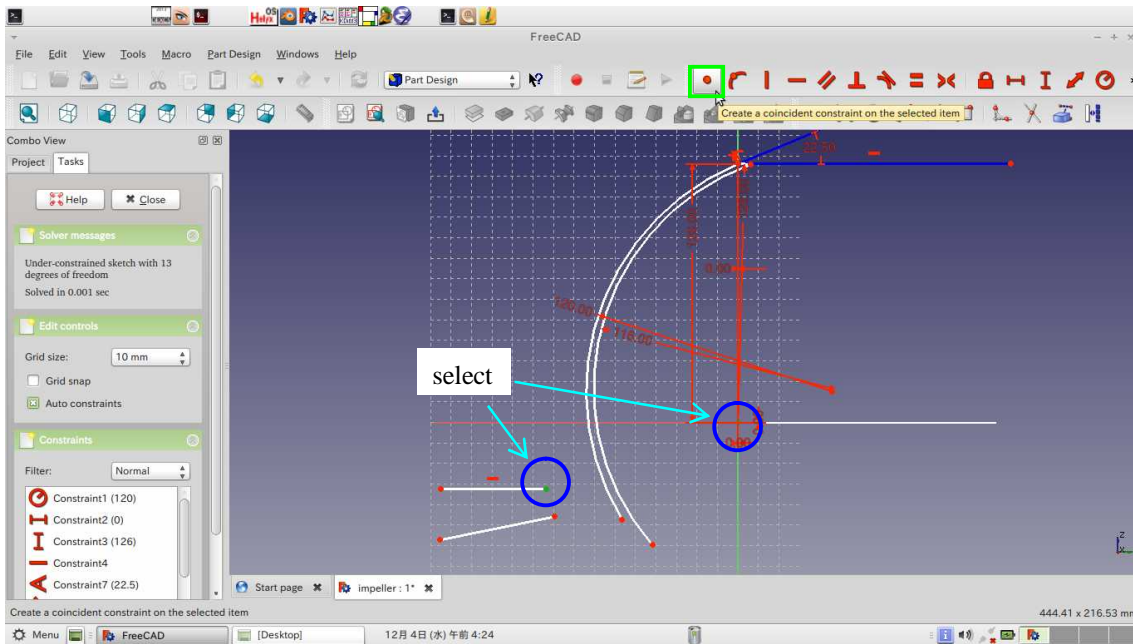
Finally, click (trim small arc). Then, trailing Edge is created.



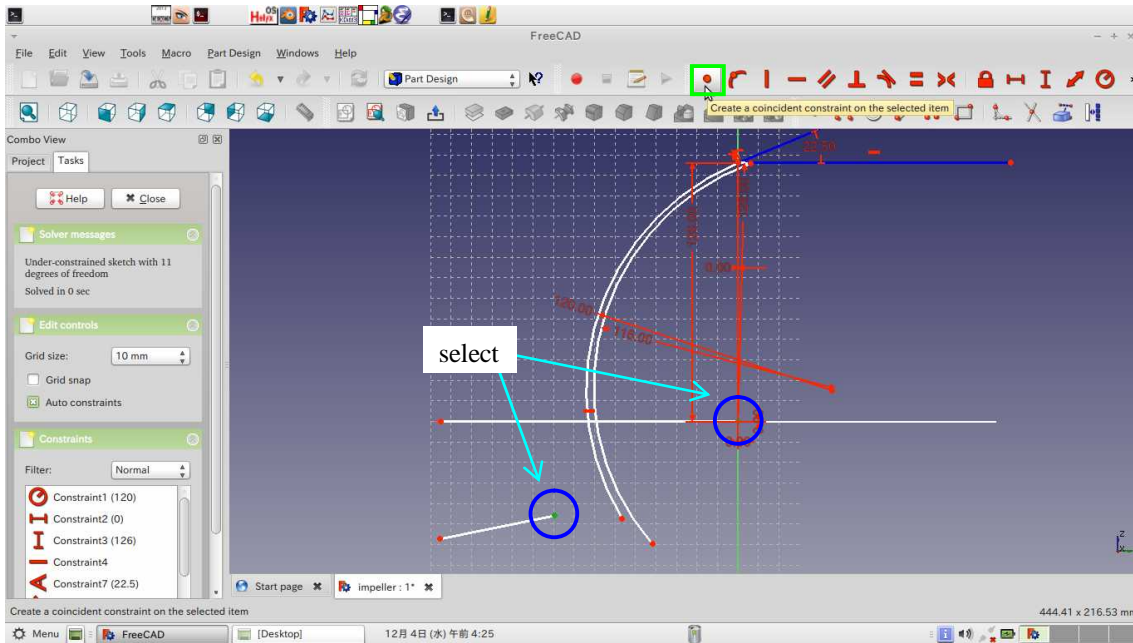
Draw a leading edge. Click “Create a line in the sketch” icon and add two lines.



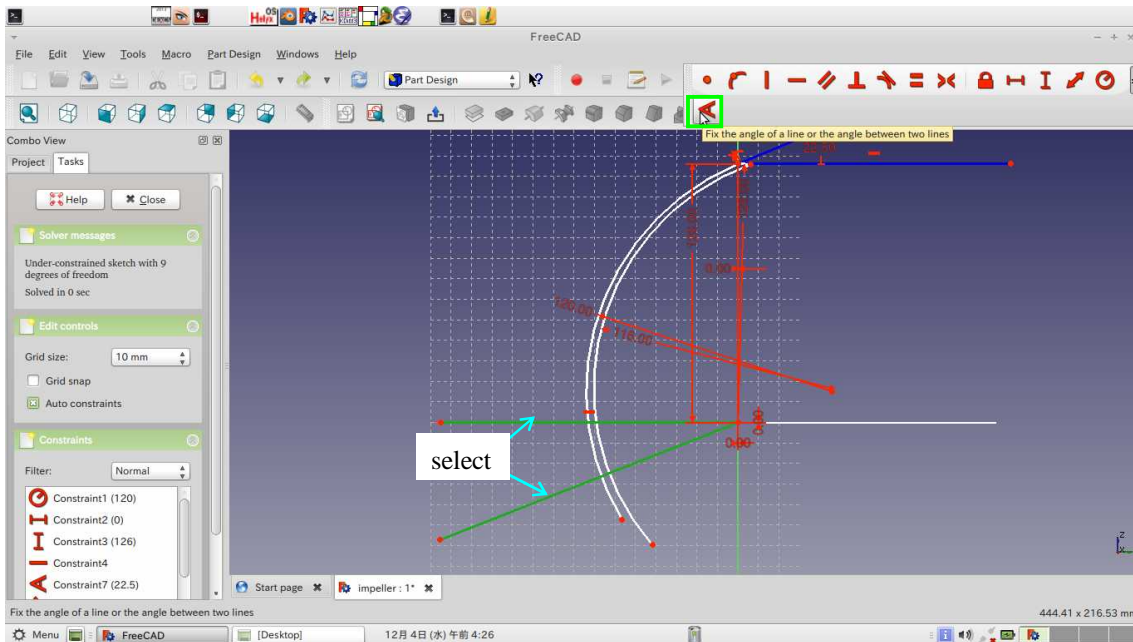
Select two points and click “Create a coincident constraint on the selected item” icon.



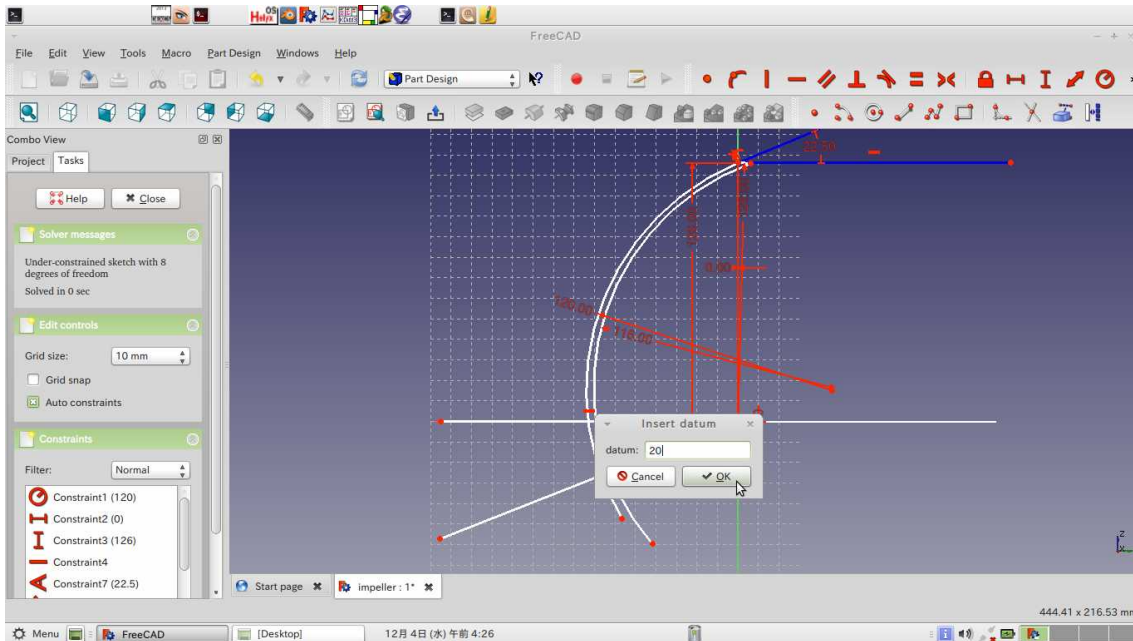
Select two points and click “Create a coincident constraint on the selected item” icon.



Select two lines and click “Fix the angle of a line or the angle between two lines” icon.

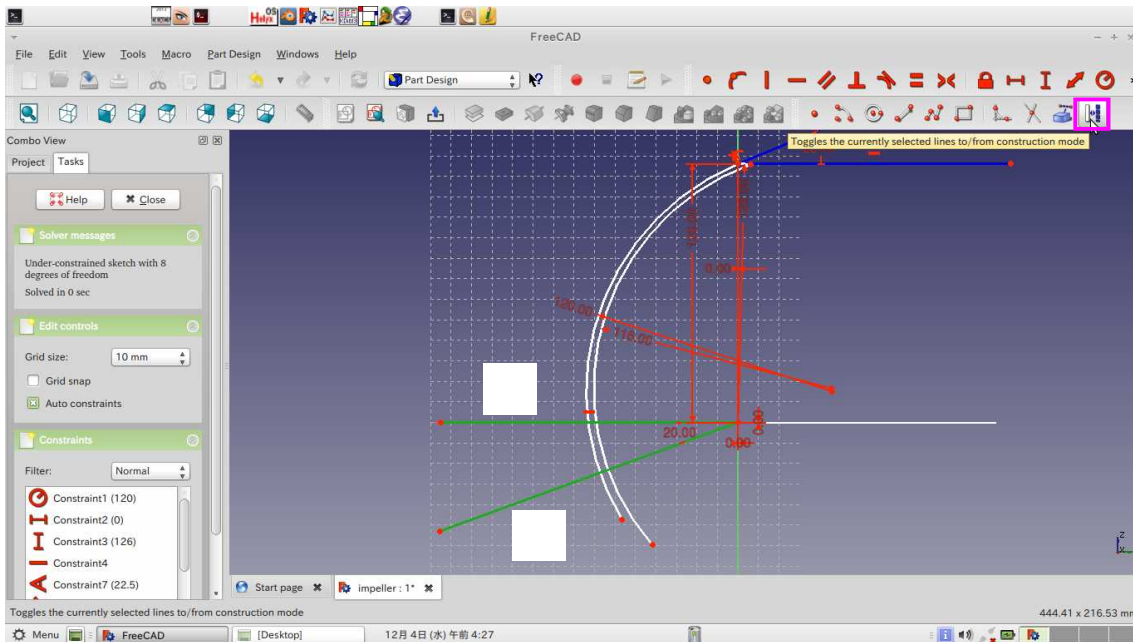


Double-click the dimension and replace the datum: 20

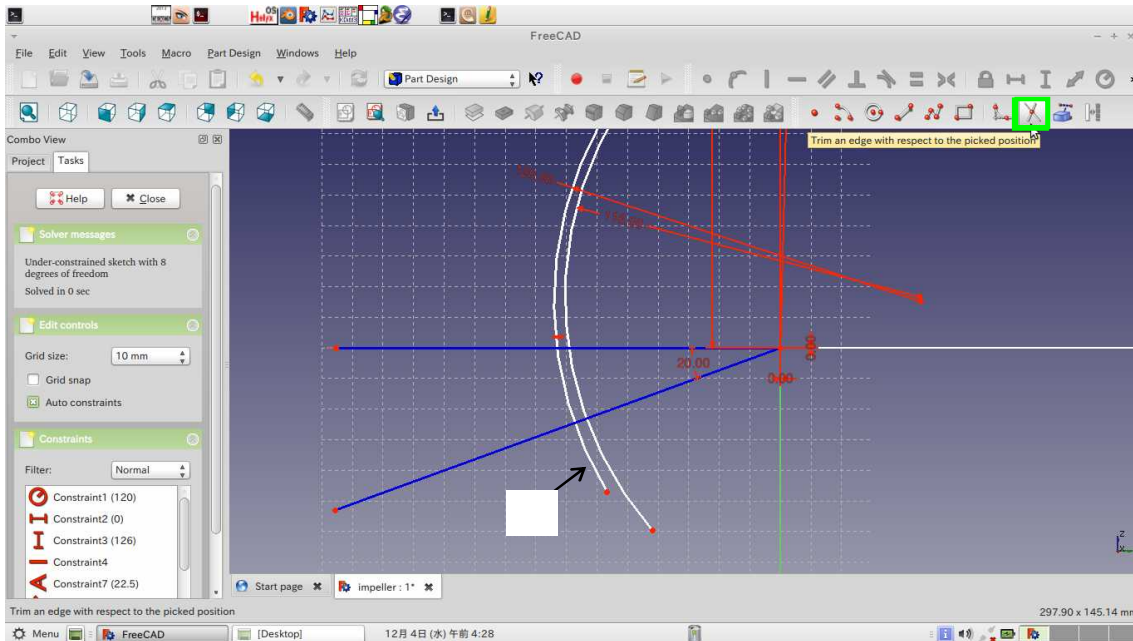




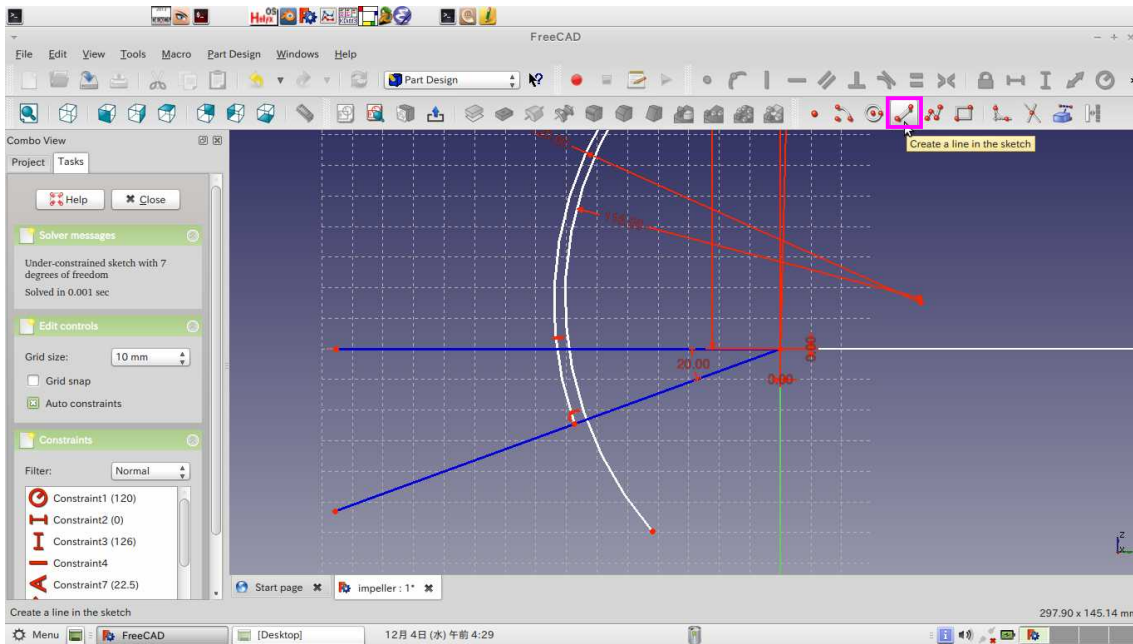
Select two lines and click “Toggle the currently selected lines to/from construction mode” icon.



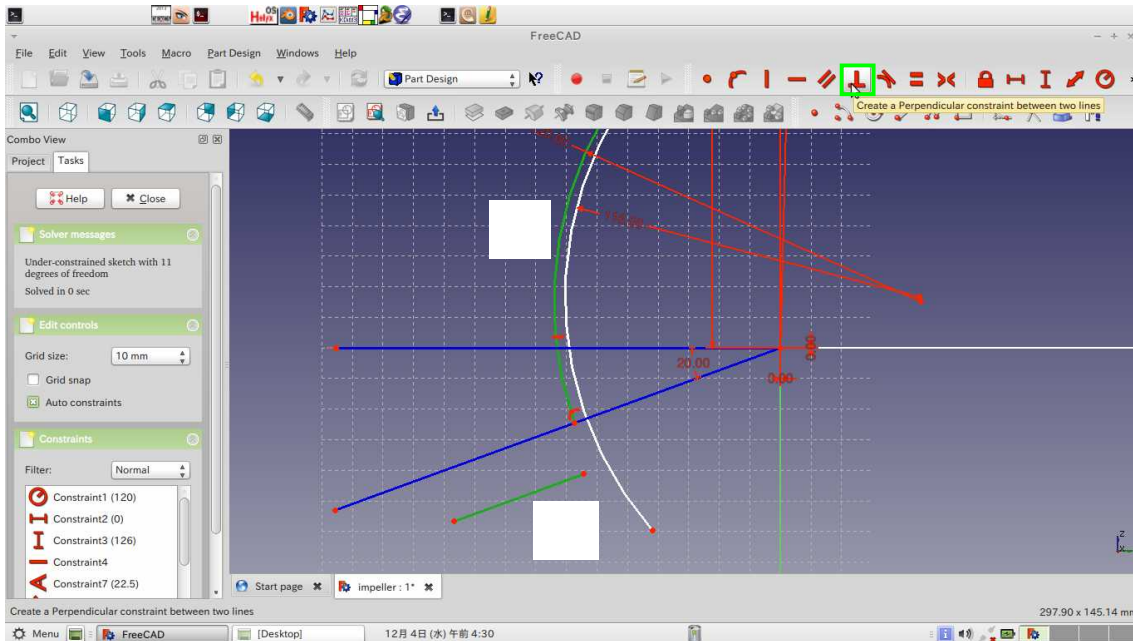
Click “Trim an edge with respect to the picked position” icon and select .



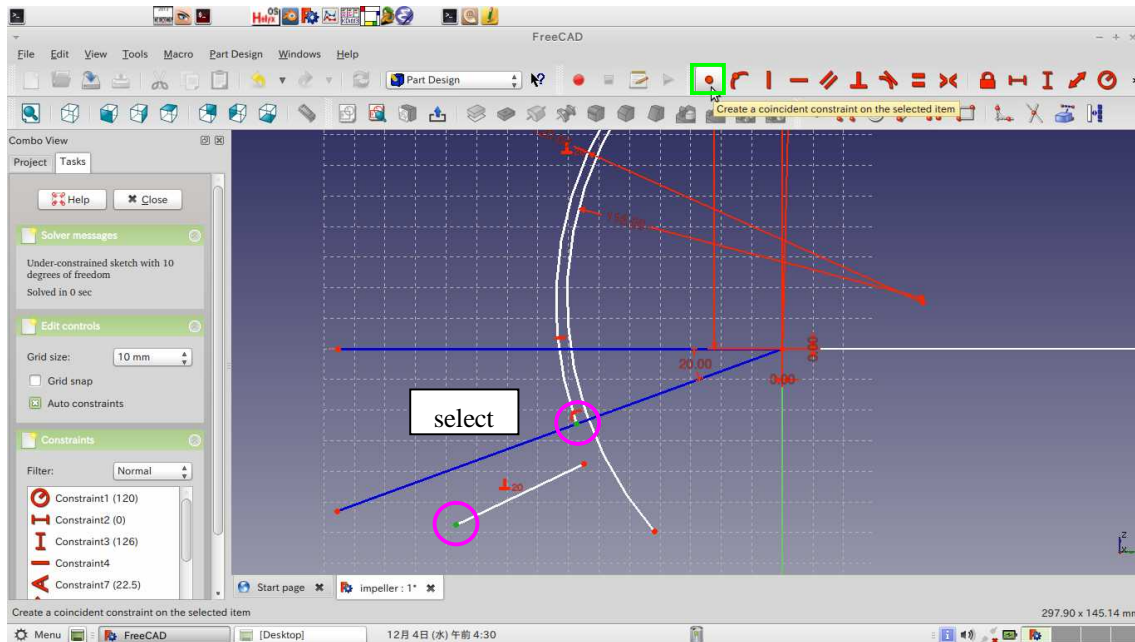
Click “Create a line in the sketch” icon and add a line.



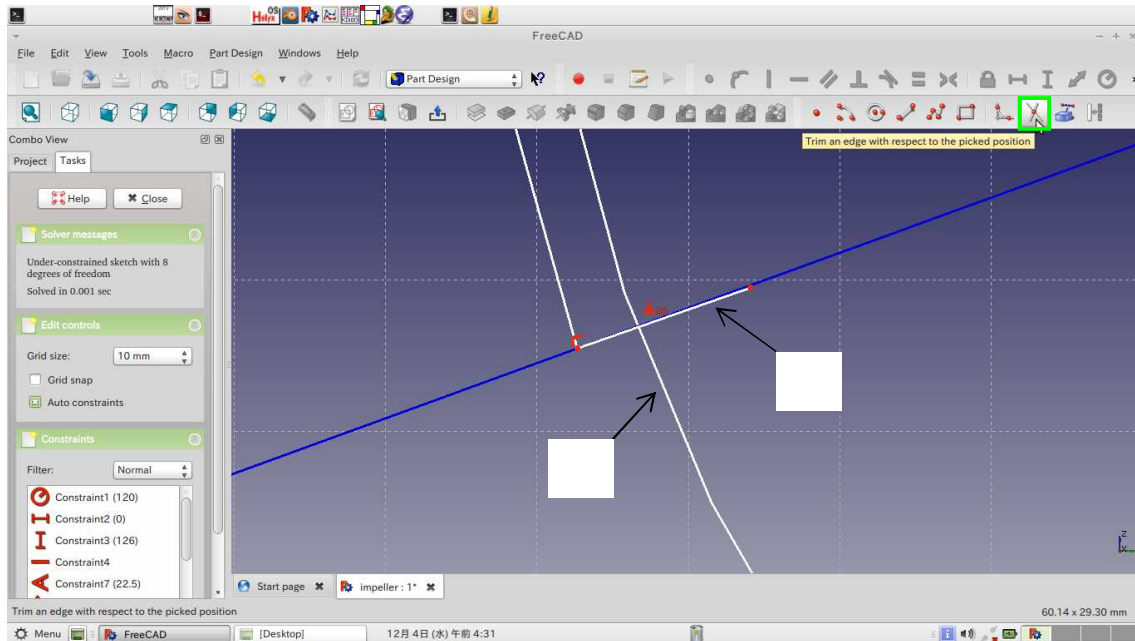
Select the arc and the line. Click “Create a Perpendicular constraint between two lines” icon.



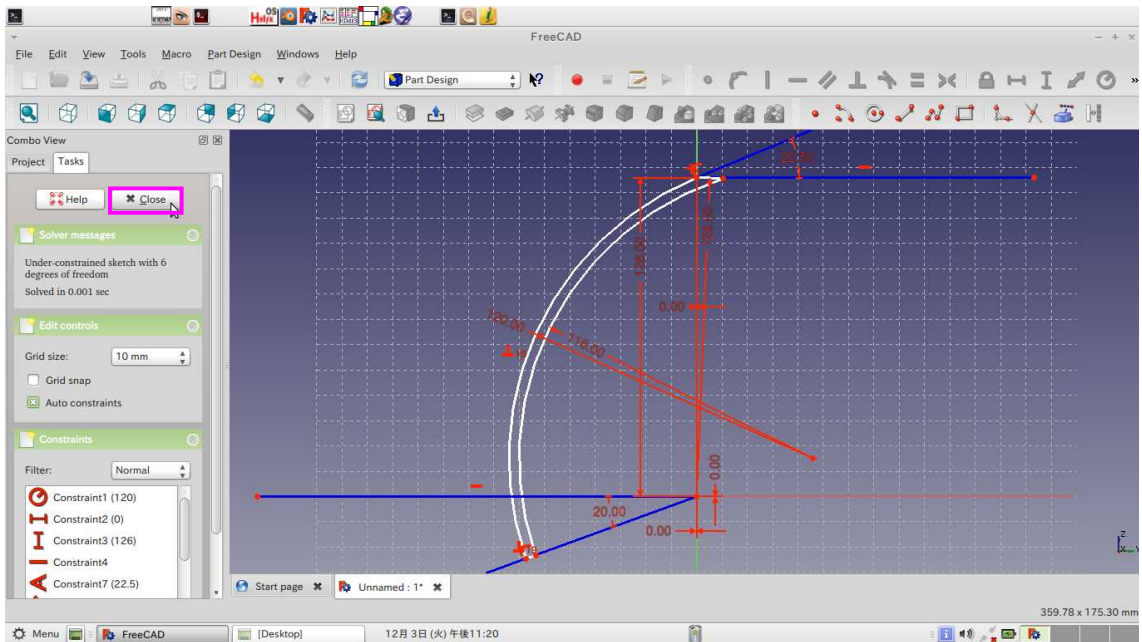
Select two points and click “Create a coincident constraint on the selected item” icon.



Click “” icon. Select the arc and the line .

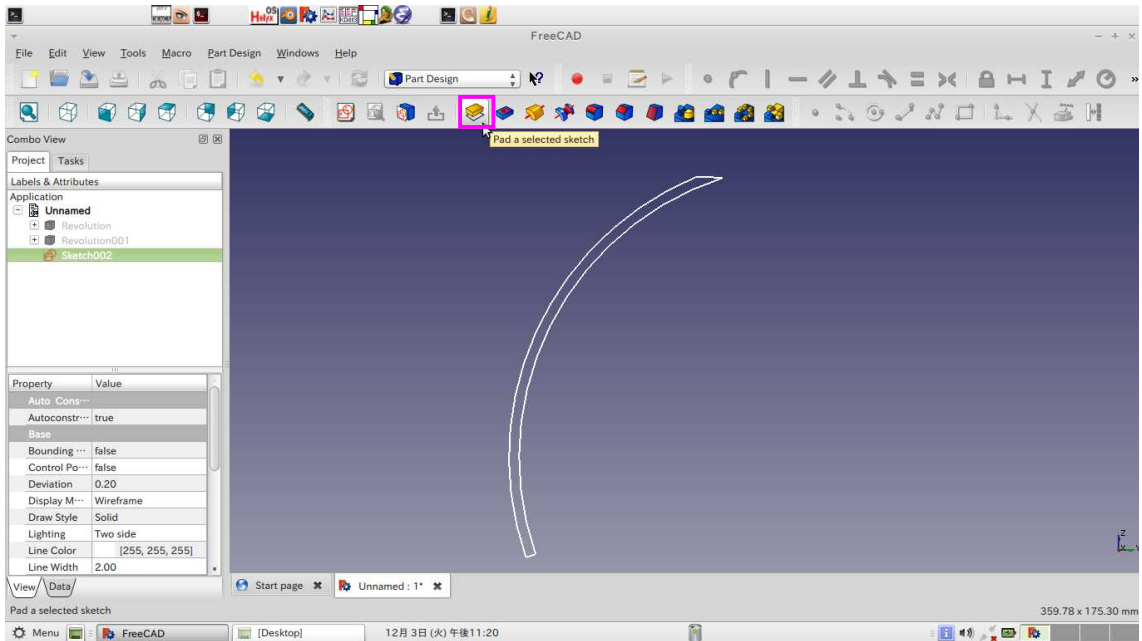


The drawing is finished. Click “Close”.

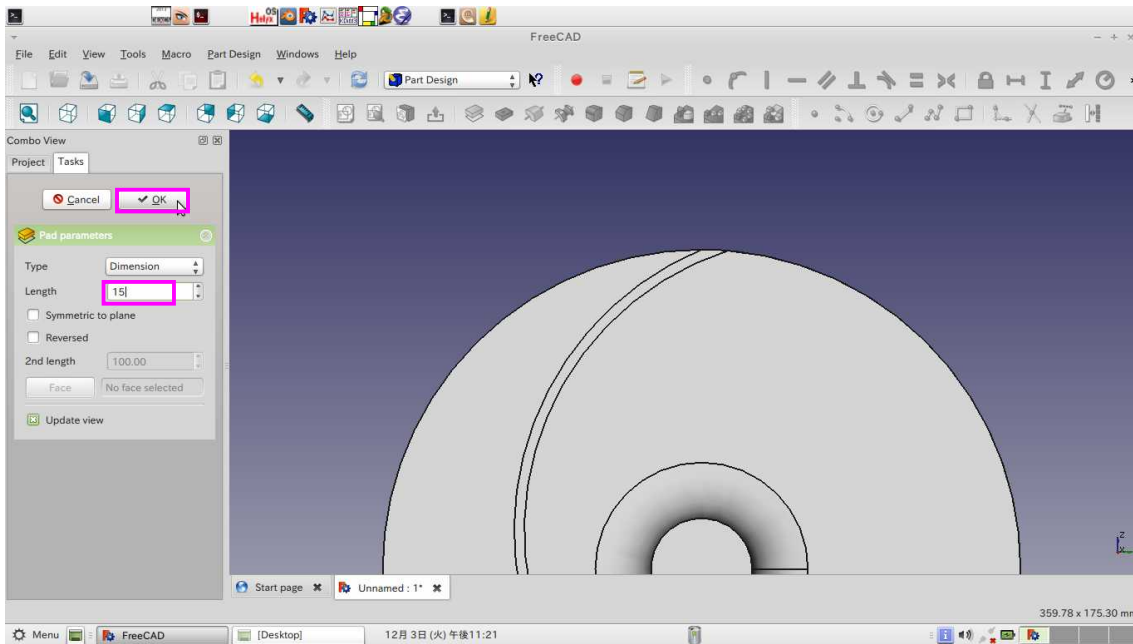


Create the 3D blade

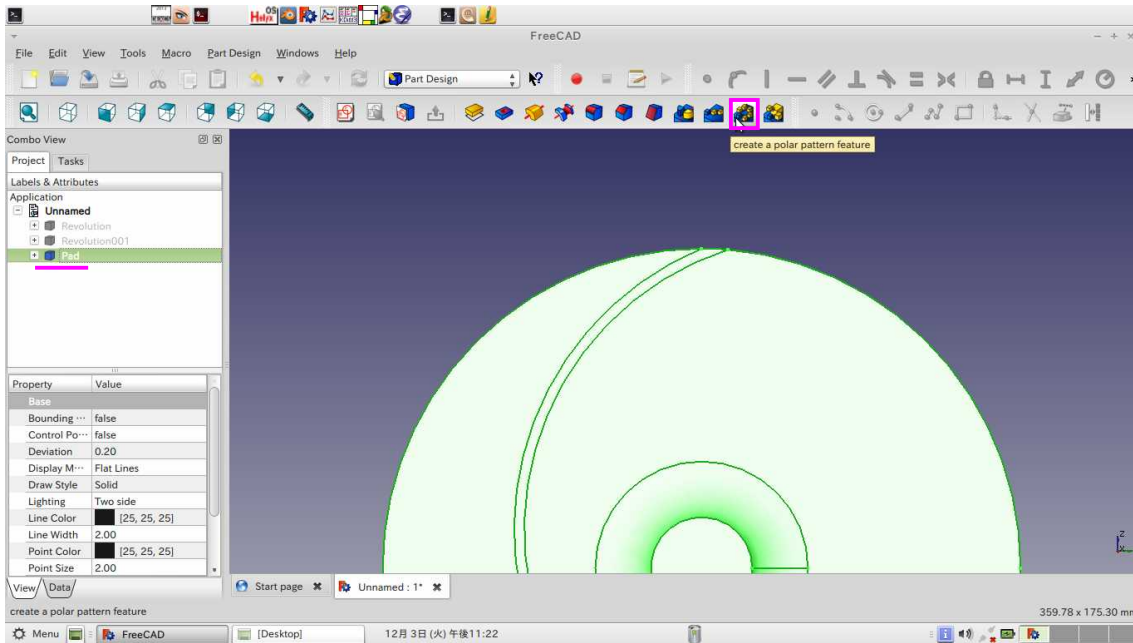
Click “Pad a selected sketch” icon.



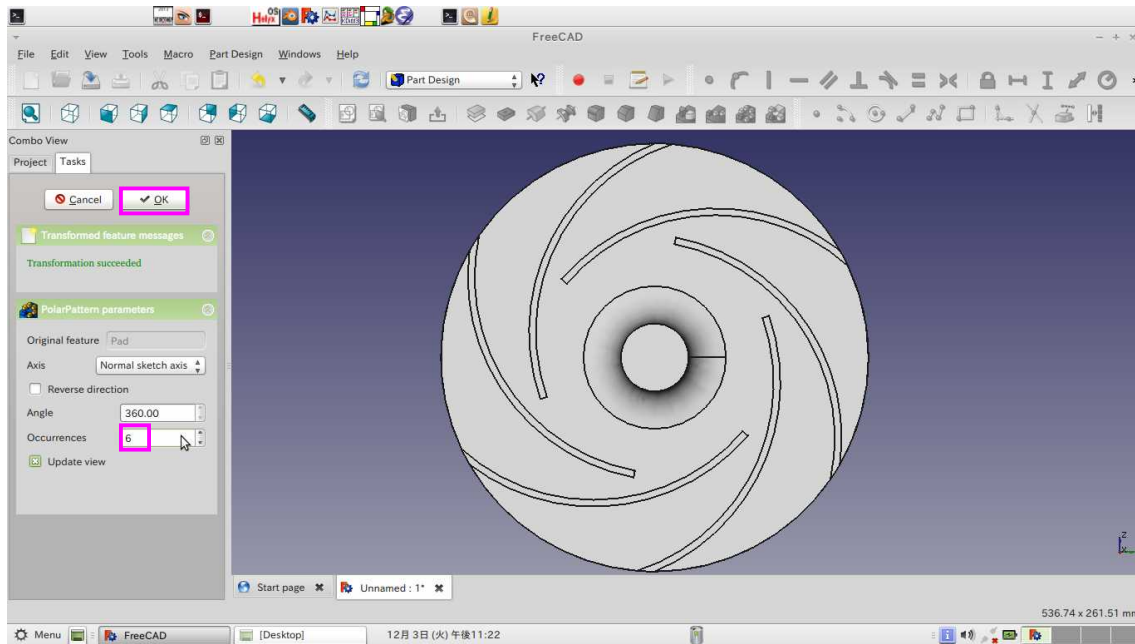
Replace the Length: 15. Click “OK”.



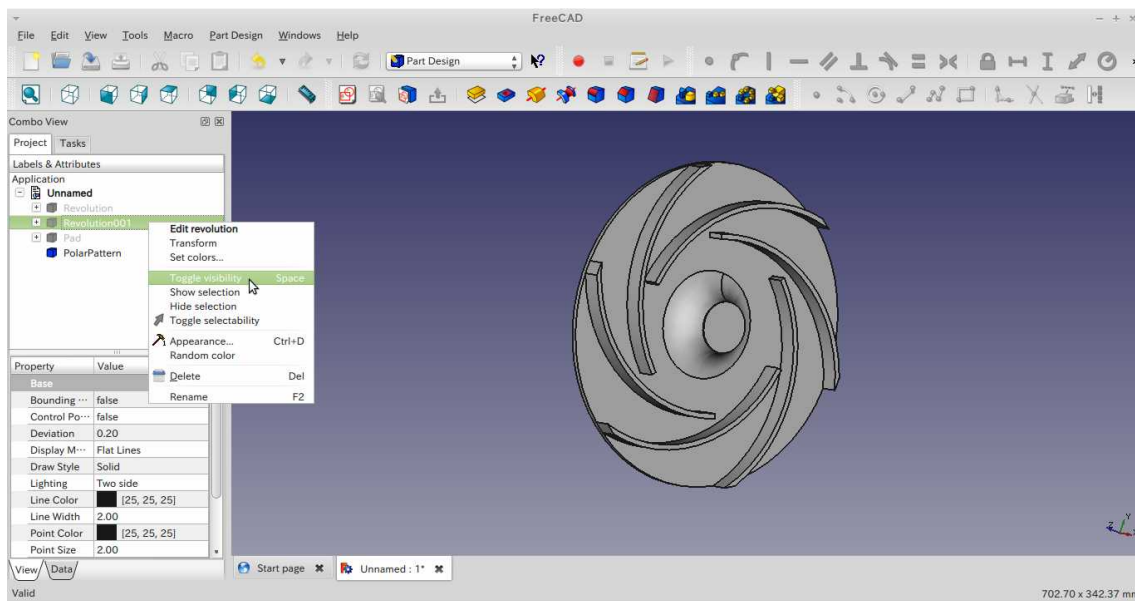
Select “Pad” and click “create a polar pattern feature”



Replace the Occurrences: 6 and click “OK”.

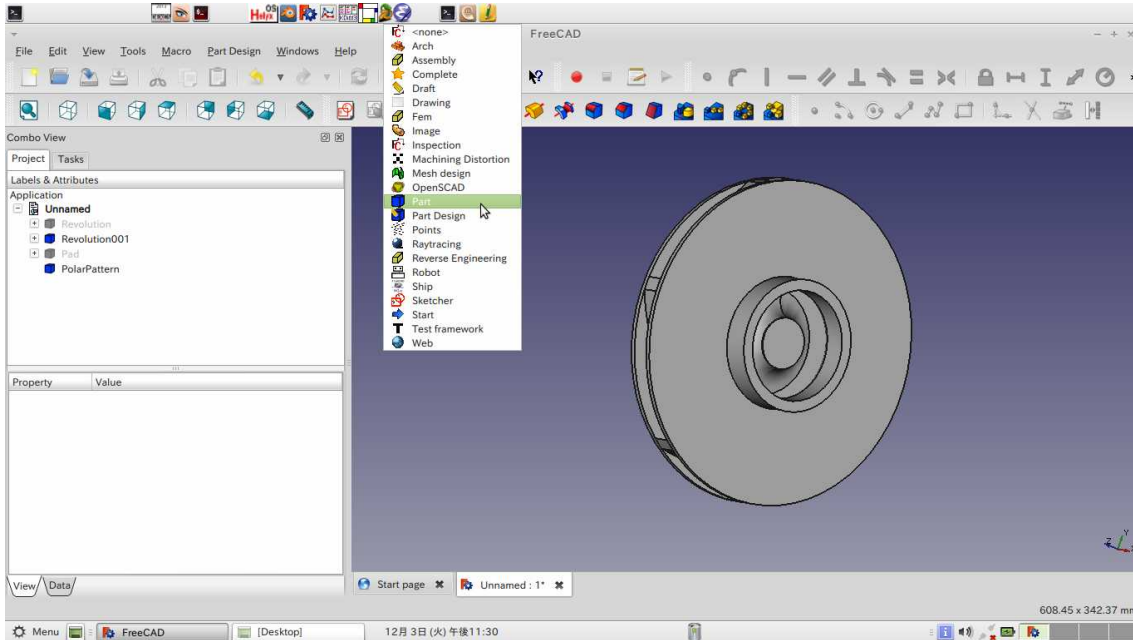


Select “Revolution001” and right-click. Then, select “Toggle visibility”.

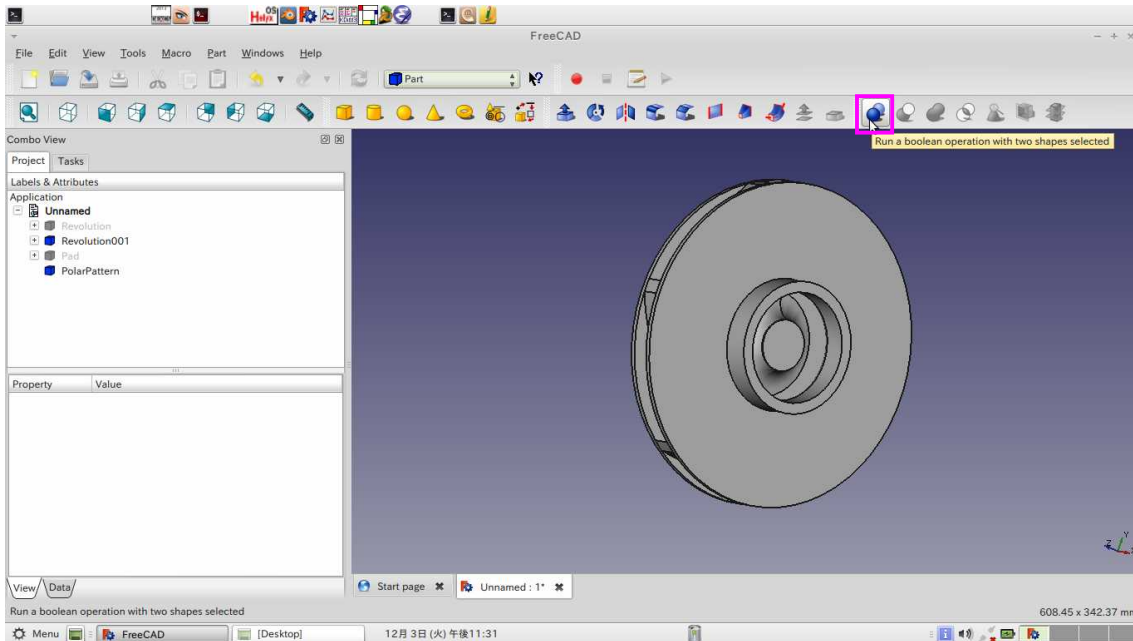


### Boolean operation

Select "Part".



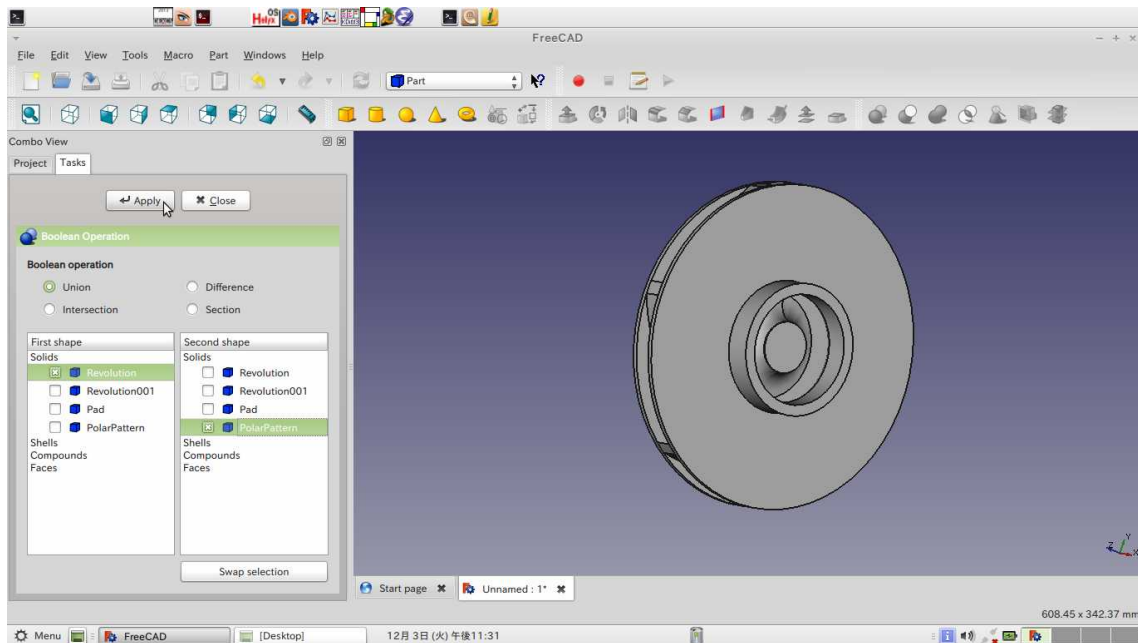
Click "Run a boolean operation with two shapes selected" icon.



Boolean Operation : Union

First shape : Revolution (back-shroud), Second shape : Polar Pattern

Click “Apply”.

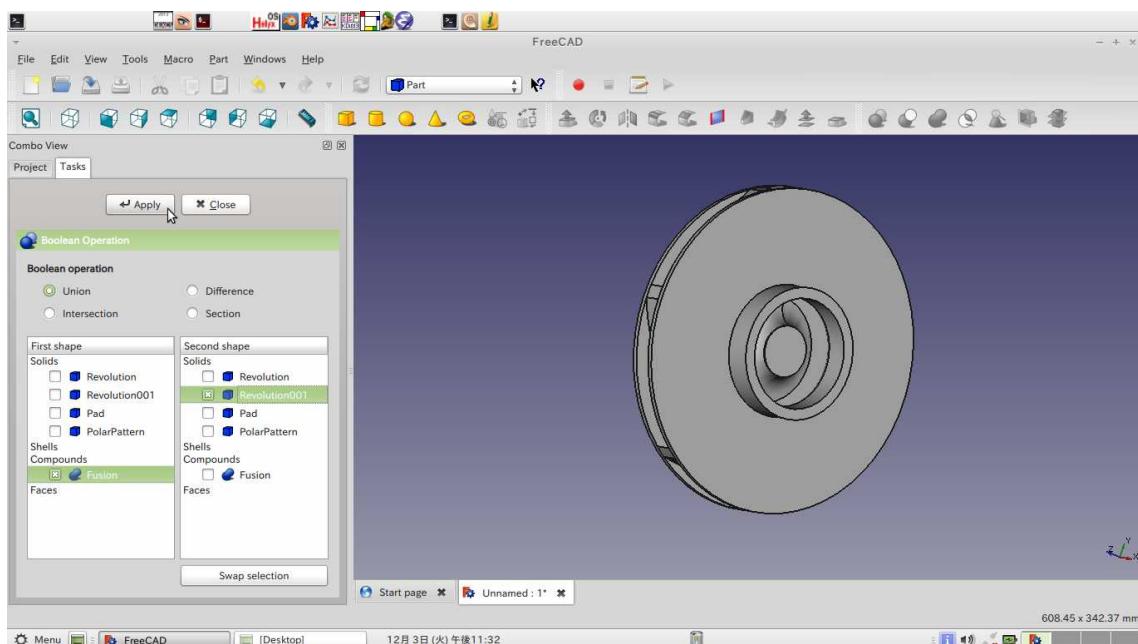


Boolean Operation : Union

First shape : Fusion, Second shape : Revolution001 (front-shroud)

Click “Apply” and “Close”.

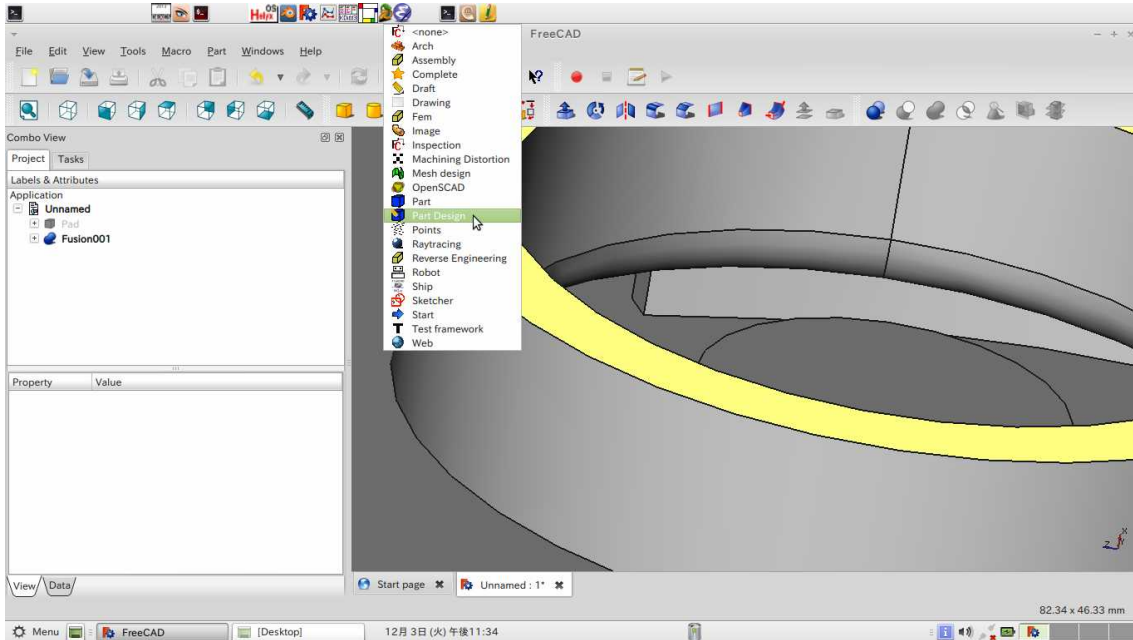
The front-shroud, the back-shroud and blades are united.





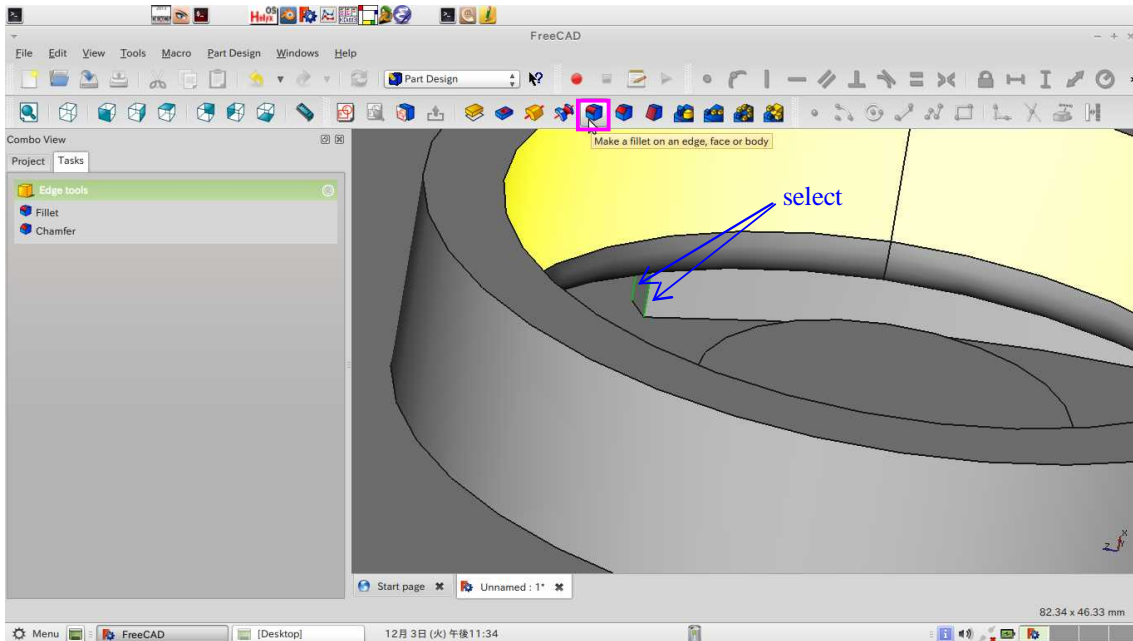
Create the edge fillet

Select “Part Design”

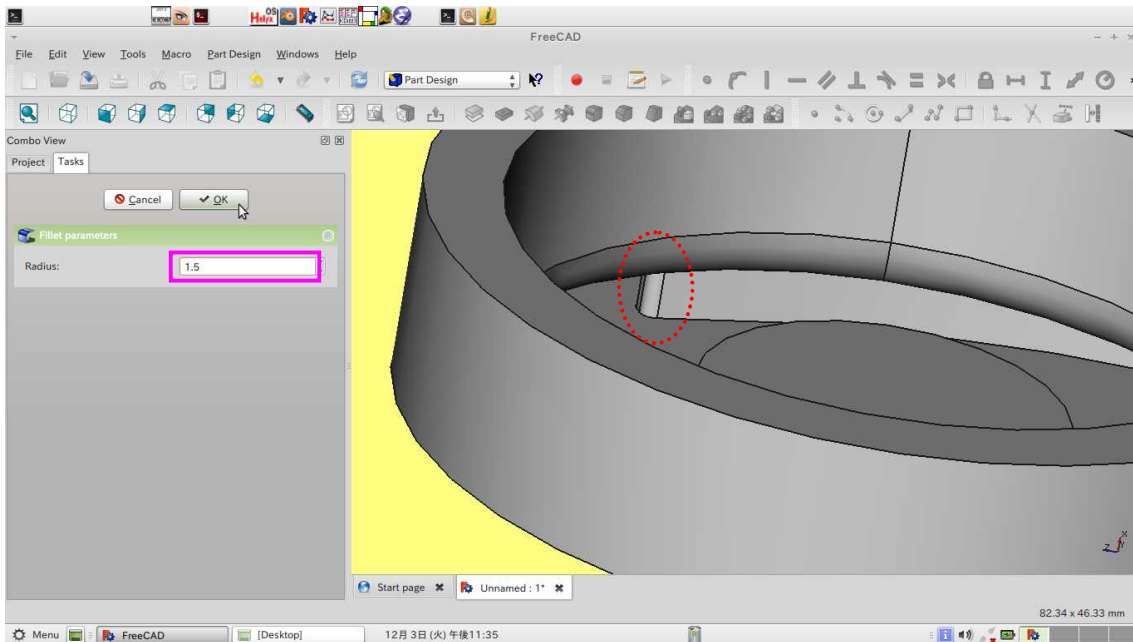


Select two edges. ( **Ctrl** + left-click)

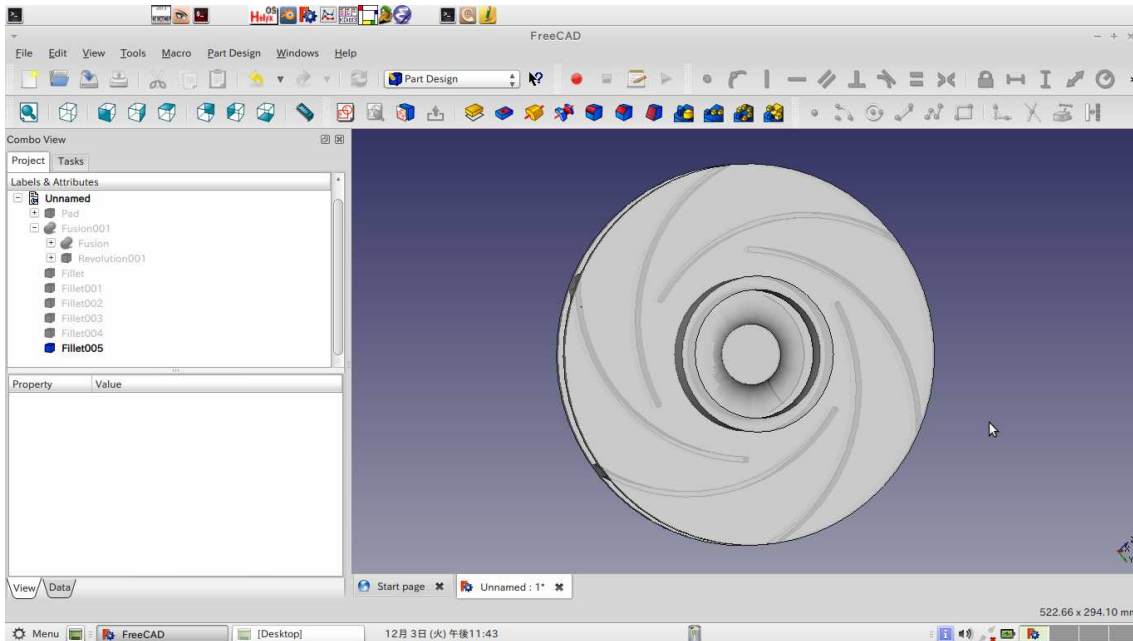
Click “Make a fillet on an edge, face or body” icon.



Replace the Radius: 1.5 and click “OK”.

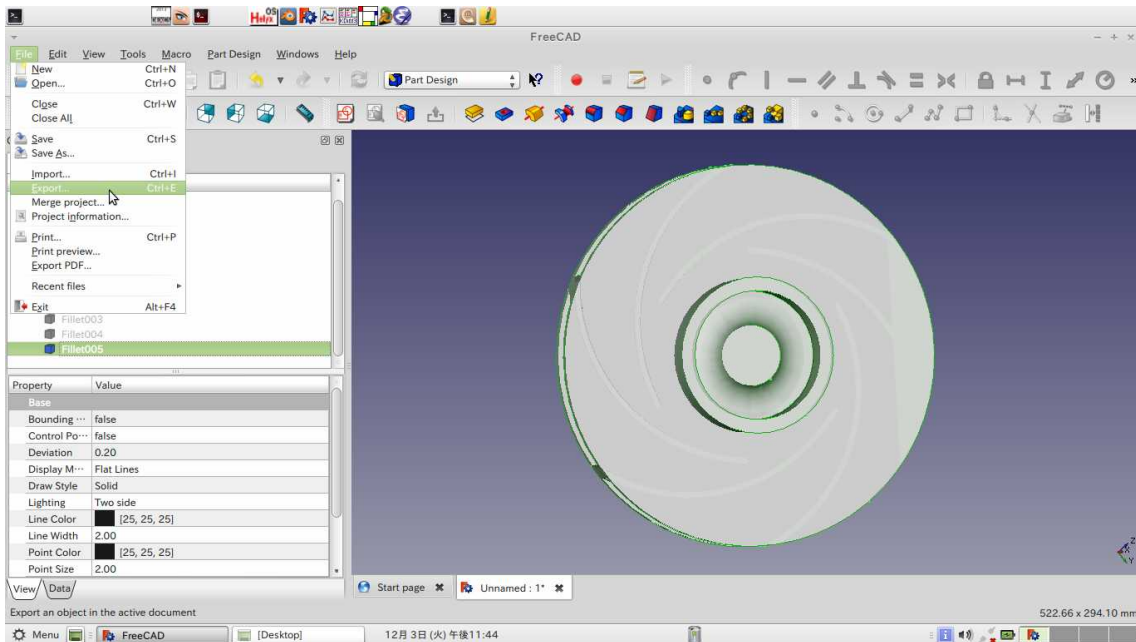


Make edge fillet for the other blades.

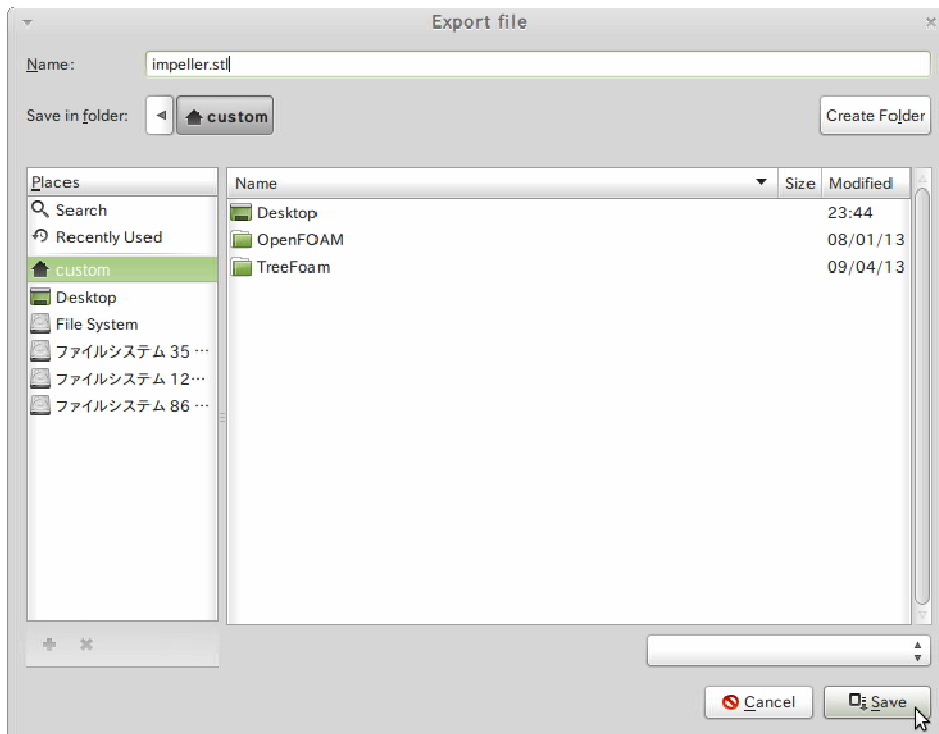


The impeller was finally created.

Select “File - Export”.



Input the Name: impeller.stl and click “Save”.



Select “File – Save As”. Input the Name: impeller.FCStd

Thank you.