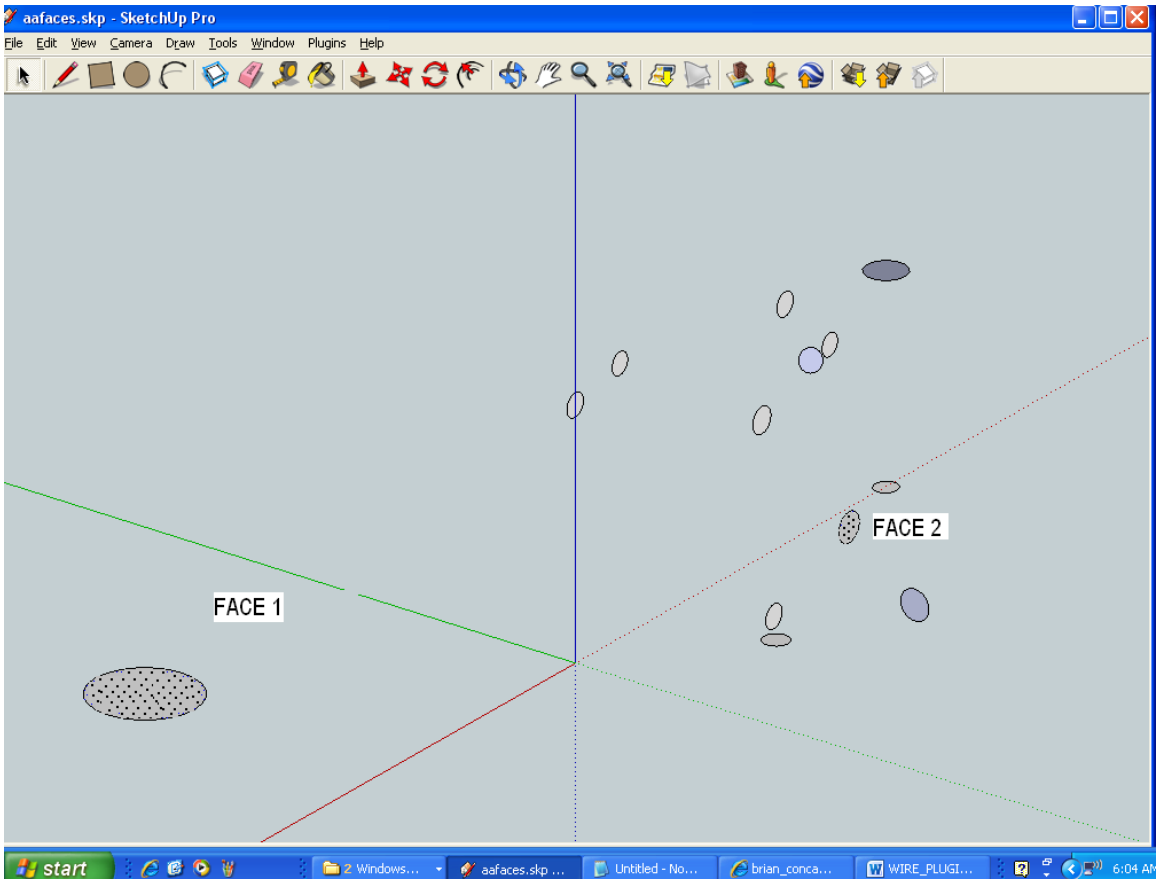
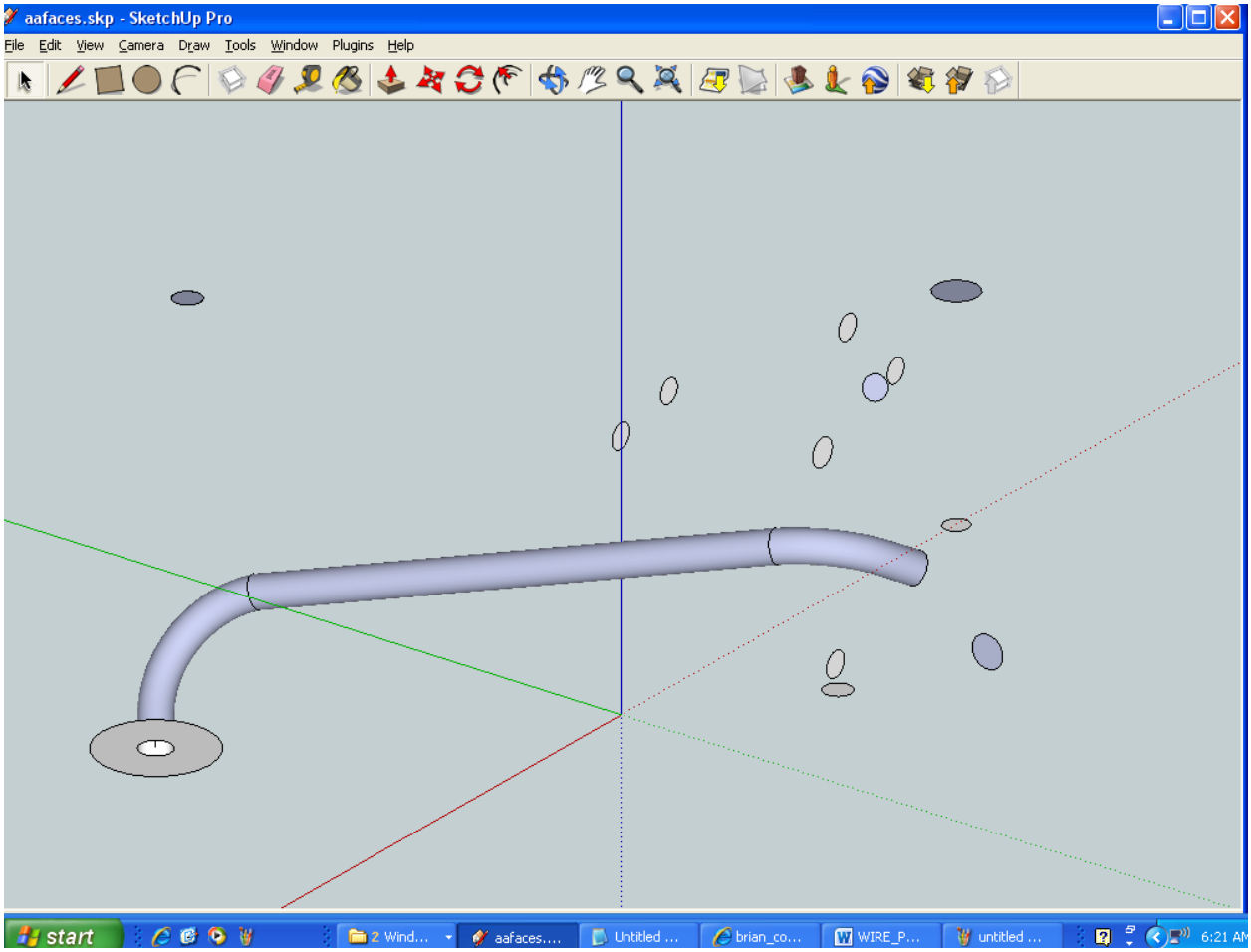
Face2Face Sketchup Plugin V1.04

This is a Sketchup plugin that starts from two arbitrary faces. It generates the smooth continuous central path for connecting the faces, and adds the outside surfaces between the selected faces.

- Select any two faces at any orientations.



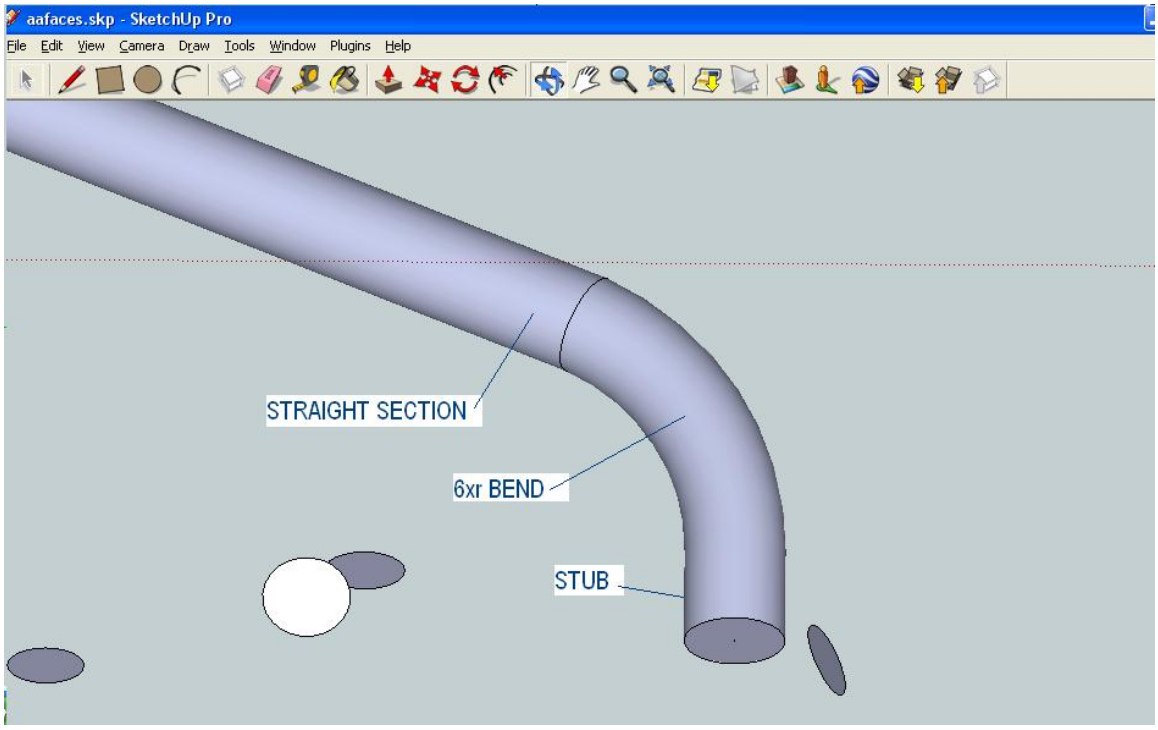
- Use menu selections Plugins, then Face2Face extrusion.
- The plugin then generates the surface that fits between the faces.



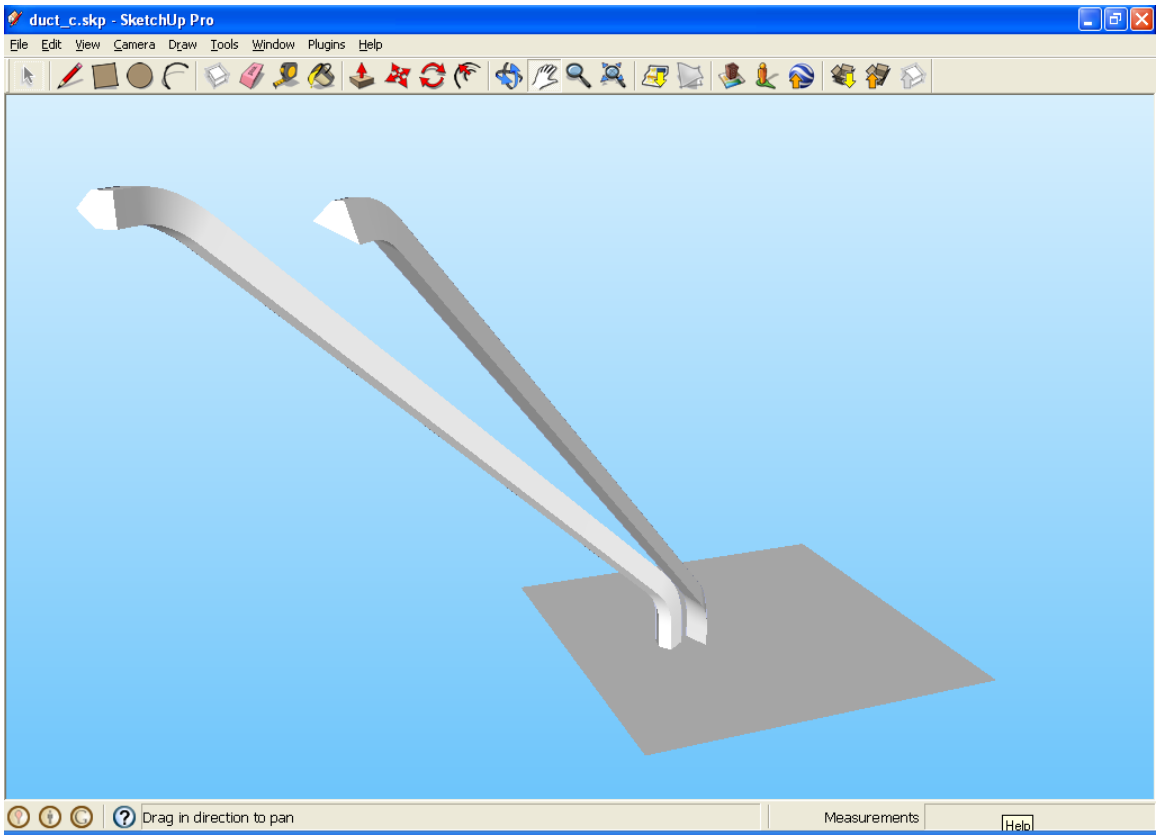
The result is an extrusion in the shape of the smaller face selected. . There is a small straight section at each face called a stub that assures a straight fit of the extrusion to the face. There is a circular bend attached to each stub that is turned so that it points toward the bend at the other face. There is a straight section that connects the two bends.

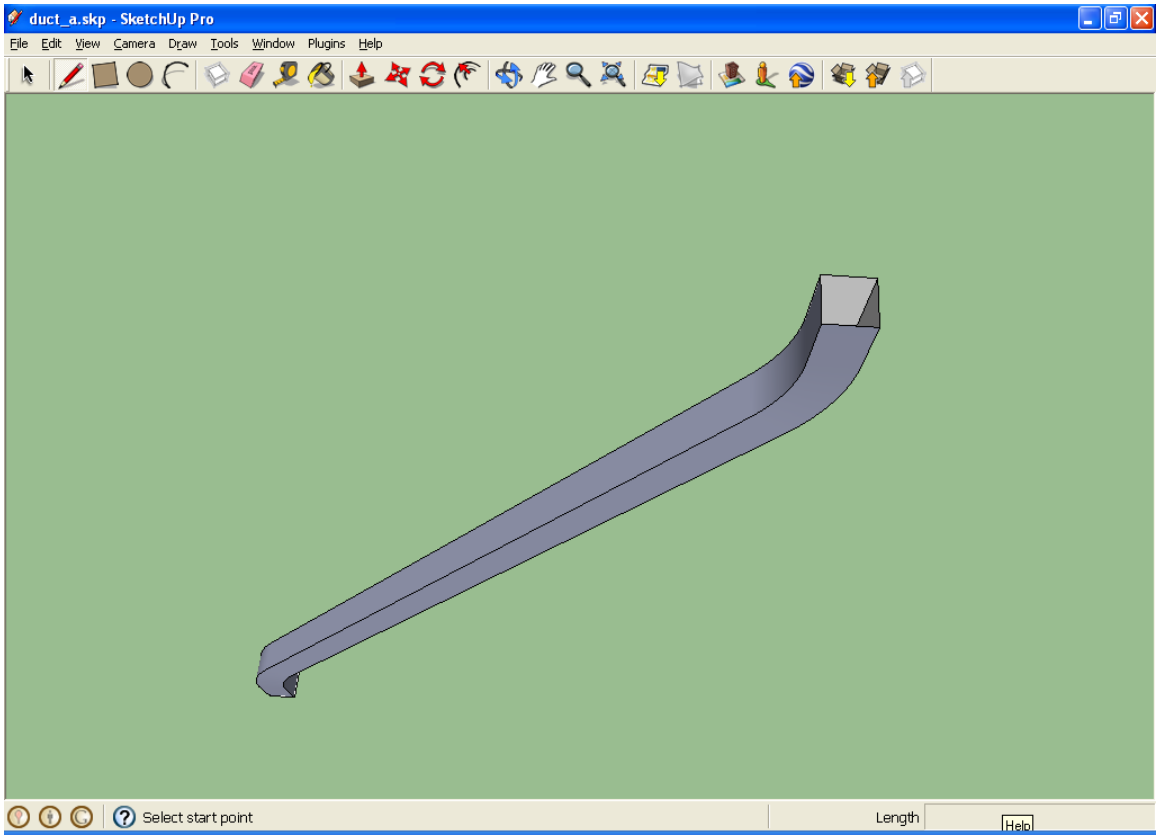
If the faces are too close together to allow for the stubs and the bends, then the plugin will tell you so and will not generate the extrusion.

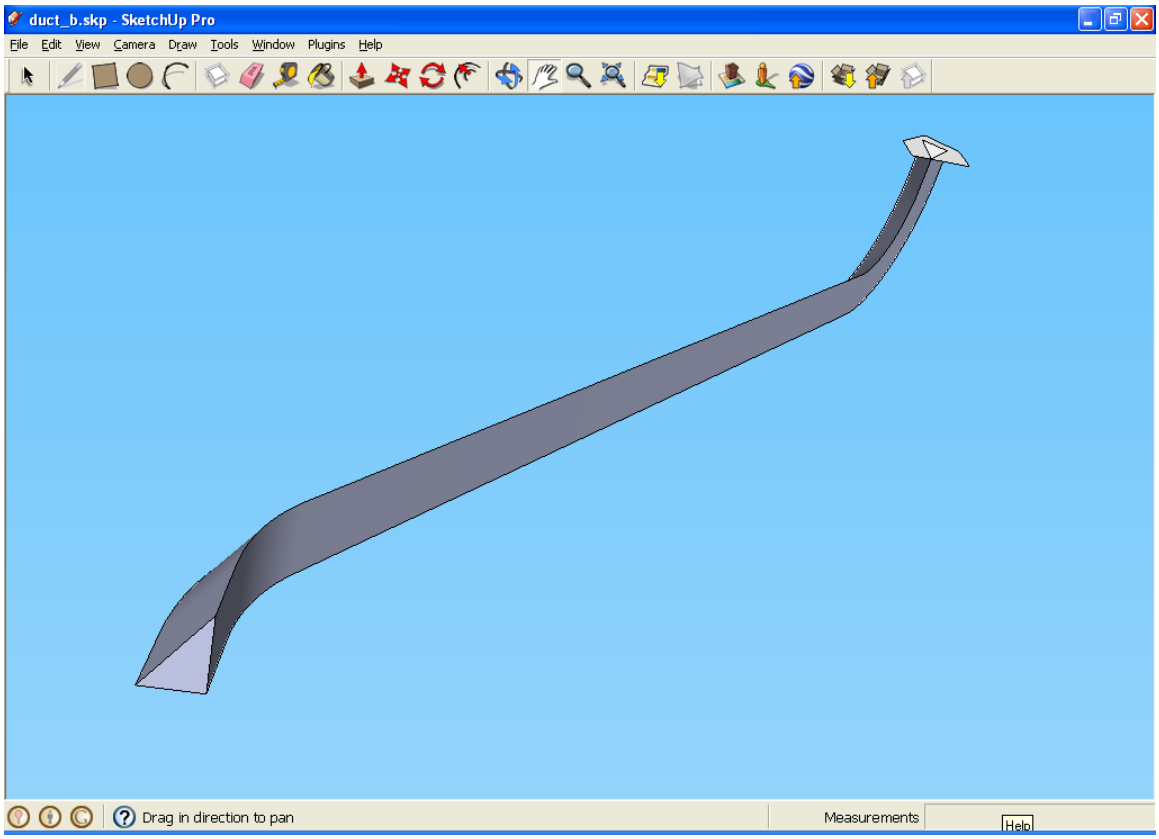
If one face is much larger than the other, as in this example, an annular ring will be left on the larger face, which can easily be deleted if desired. If the selected faces have the same radius, the ends will look solid. Here is a rotated view of the smaller end.



Here are some example results for noncircular faces:

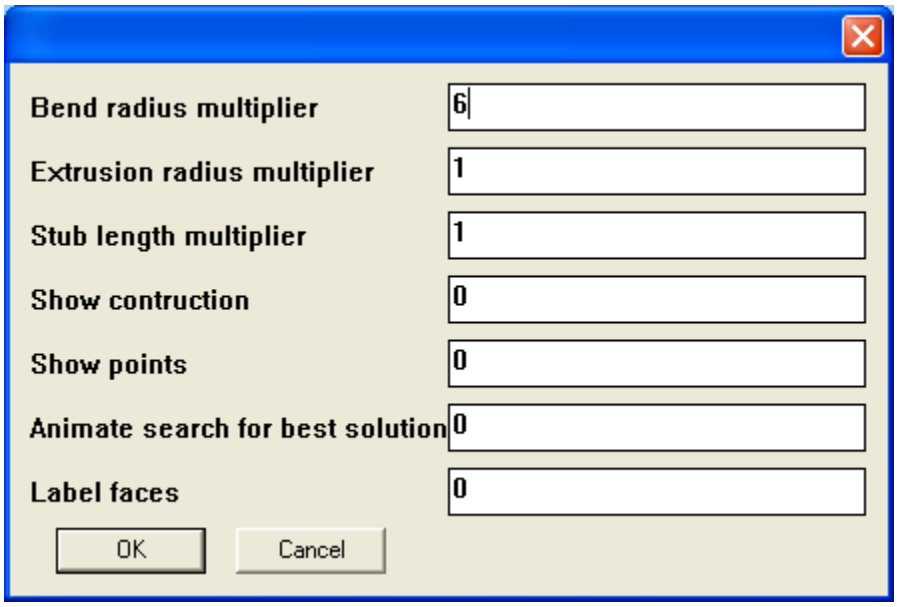



duct_a.skp - SketchUp Pro
File Edit View Camera Draw Tools Window Plugins Help
Select start point
Length
Help


duct_b.skp - SketchUp Pro
File Edit View Camera Draw Tools Window Plugins Help
Drag in direction to pan
Measurements
Help

The extrusion can be undone using control-z. This undoes the surfaces of the extrusions, leaving the central path. Using control-z again undoes the central path as well.

Adjustments can be made to the extrusions by the menu selections Plugins then Face2Face parameters. The following input box appears to allow the user some additional freedoms:



The bend radius, the extrusion size or radius, and the stub lengths can all be varies by supplying values between 0.01 and 100. The last four parameters allow the user to view the progress of the construction of the central path of the extrusion, and to label the faces ‘face1’ and ‘face2’. The last four values must be 0 or 1 for false or true, respectively.

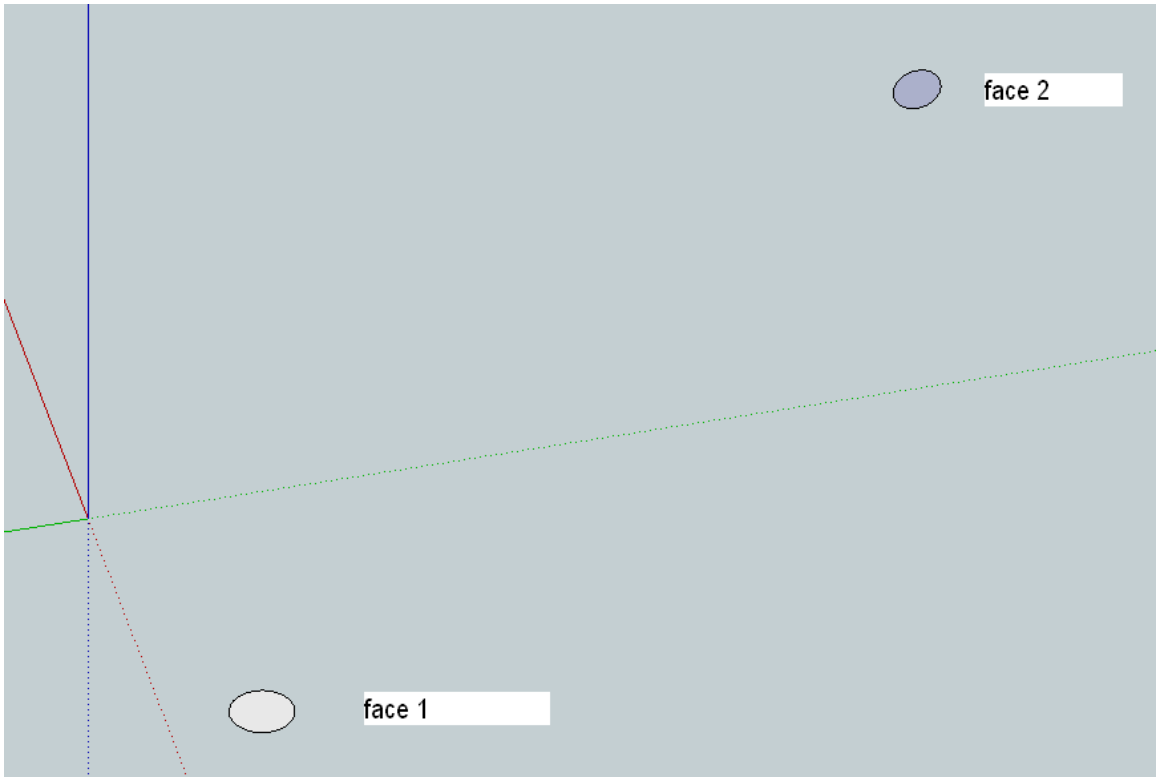
The face to be extruded is the smaller face by default, but this can be changed by selecting a face edge at the larger face, then invoking the plugin again. (The user will probably use control-z’s to erase the previously created extrusion). Then invoking the plugin again will extrude the larger face along the same central path, onto the plane of the smaller face.

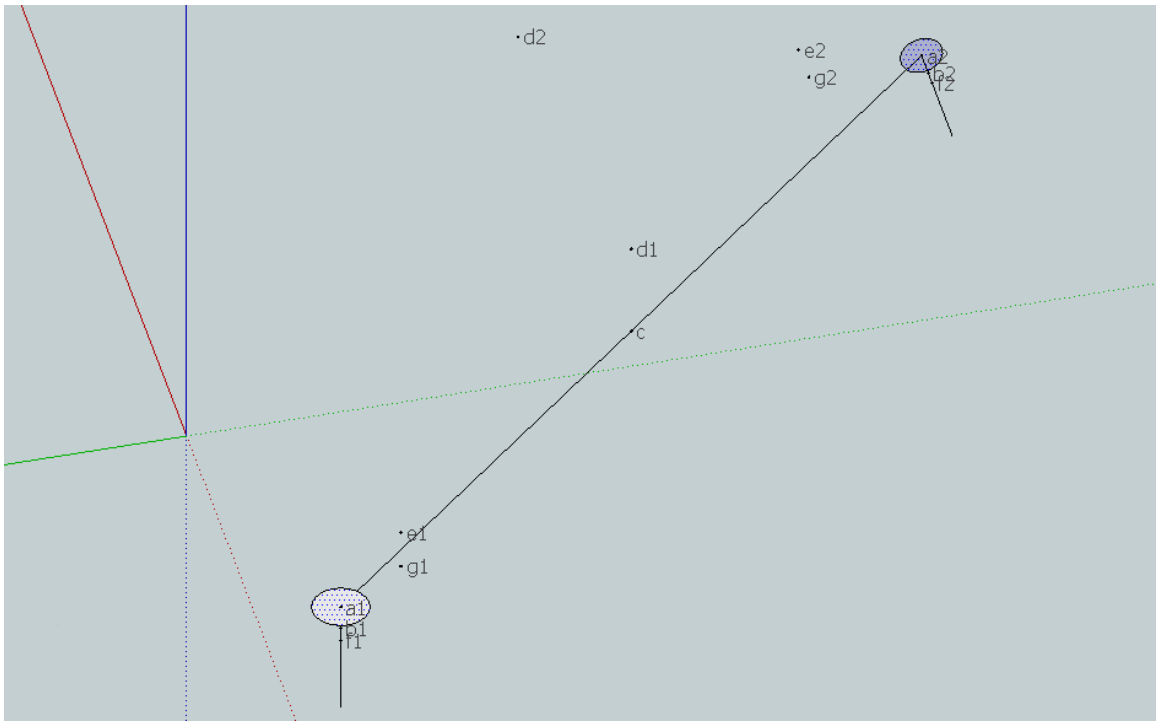
Sometimes the user may want to route the extrusion down instead of up or vice versa. This can be done by selecting a nearby entity along with the faces, where the routing is to be reversed.

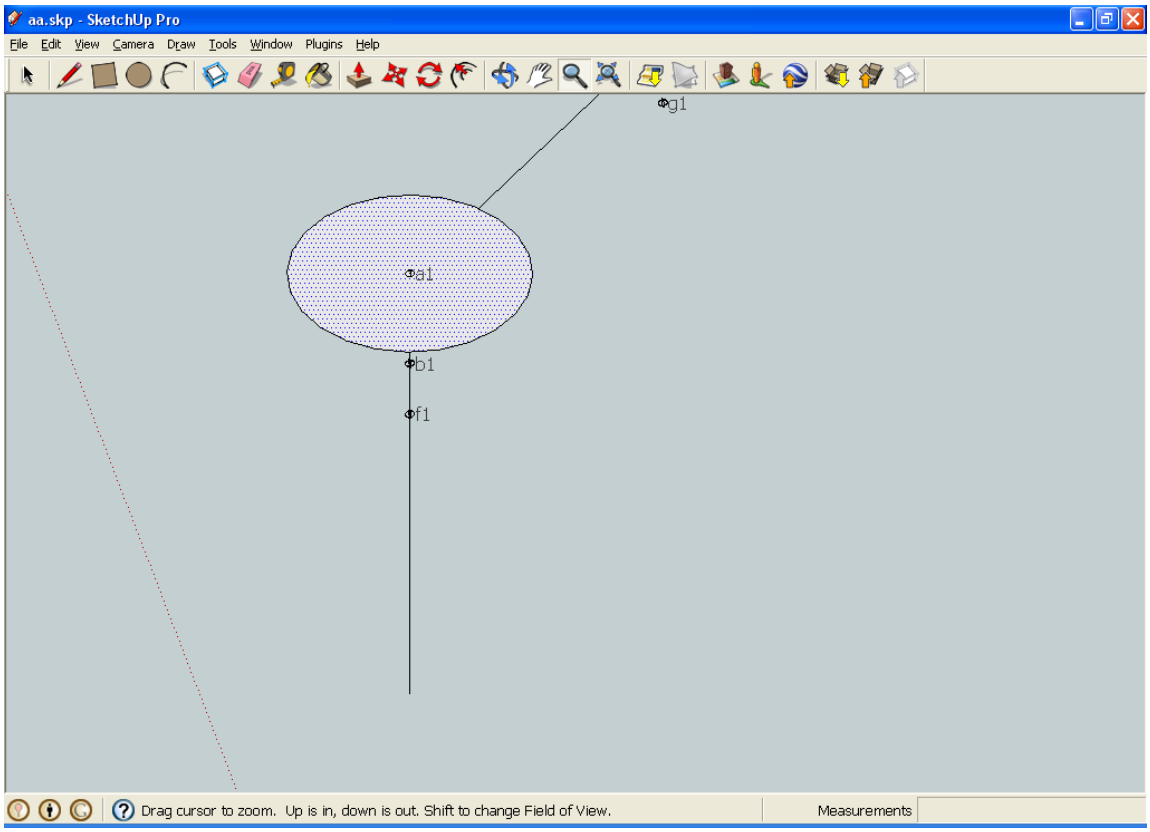
Algorithm

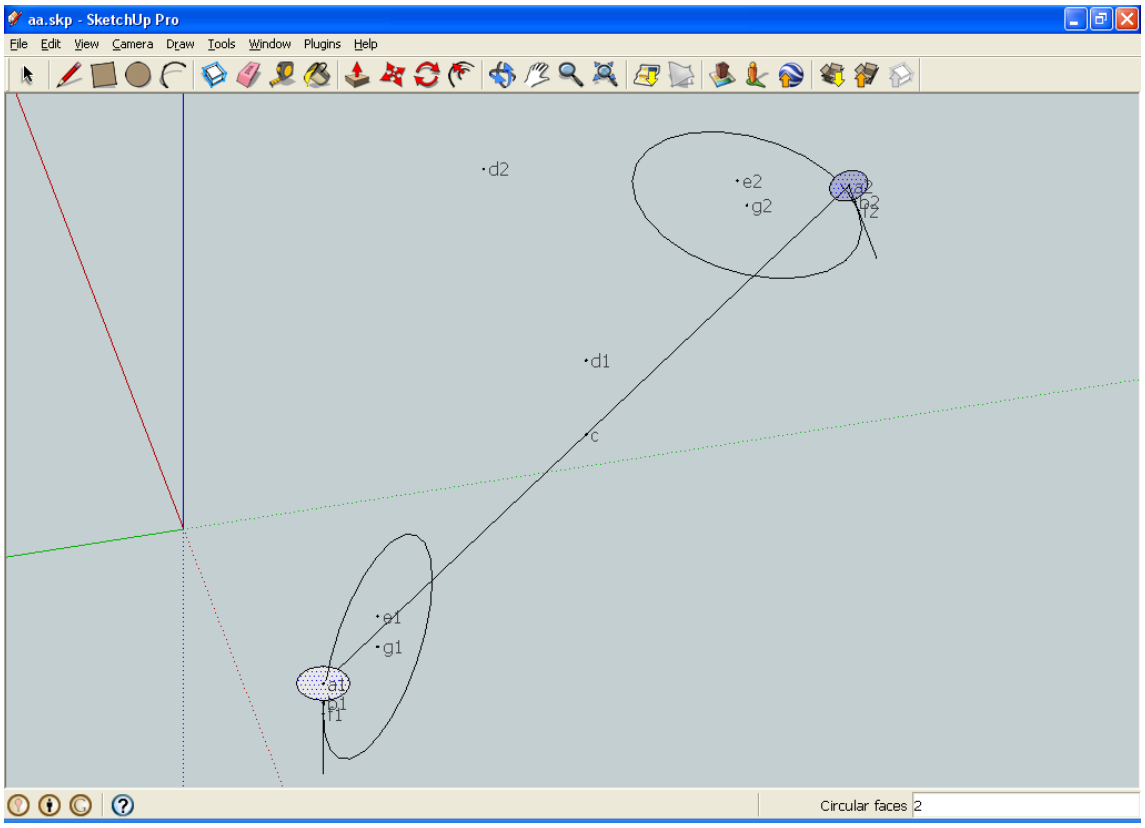
- Determine and use the size (call it the radius if circular) of the smaller face
- Call the face centerpoints a1 and a2.
- Construct outward unit normals to face1 and face2 at a1 and a2.
- The ends of the normals will be called b1 and b2. The distance between b1 and b2 may not be as desired because the normal directions, out of the faces, are generated in unpredictable directions because there are two possible directions for a normal to any face. Try all four combinations of the possible normal directions, and select the combination that gives the minimum b1-b2 distance.
- Check that sufficient space is available between b1 and b2 to accommodate bends.
- Create a straight line between a1 and a2.
- Create point c = midpoint of this line
- Project a point from the line onto the planes of face1 and face2, call them d1 and d2
- Find the intersections of the face edges with the lines from a to d, 6 radii out from a. Call these points e1 and e2.
- Project face1 and face2 in their normal directions by distance=2*radius. These faces face1a and face2a will be at the start of the wire bends and prevent the bent wire ends from bleeding through the original two faces. The points on faces 1a and 2a corresponding to a and e are h and g.
- The right circular cylinders formed by face 1-face1a and face2-face2a are the "stubs".
- At faces 1a and 2a, create a circle perpendicular to the face, of radius 6*r, at centerpoints g. Call these circle1 and circle2
- Create a line parallel to line a1-a2 that is "somewhat tangent" to circles 1 and 2.
- Swing the planes of circles 1 and 2 by iteration until "somewhat tangent" becomes
- "as close as possible to tangent".
- Create one connected path by connecting the central paths from face1, along stub1, along wire bend 1, along the straight section, along wire bend 2, along stub 2 to face2.
- Use the Followme feature to create a surface along the continuous path.
- Done!

Diagrams for preceding description:






aa.skp - SketchUp Pro
File Edit View Camera Draw Tools Window Plugins Help
g1
a1
b1
f1
Drag cursor to zoom. Up is in, down is out. Shift to change Field of View.
Measurements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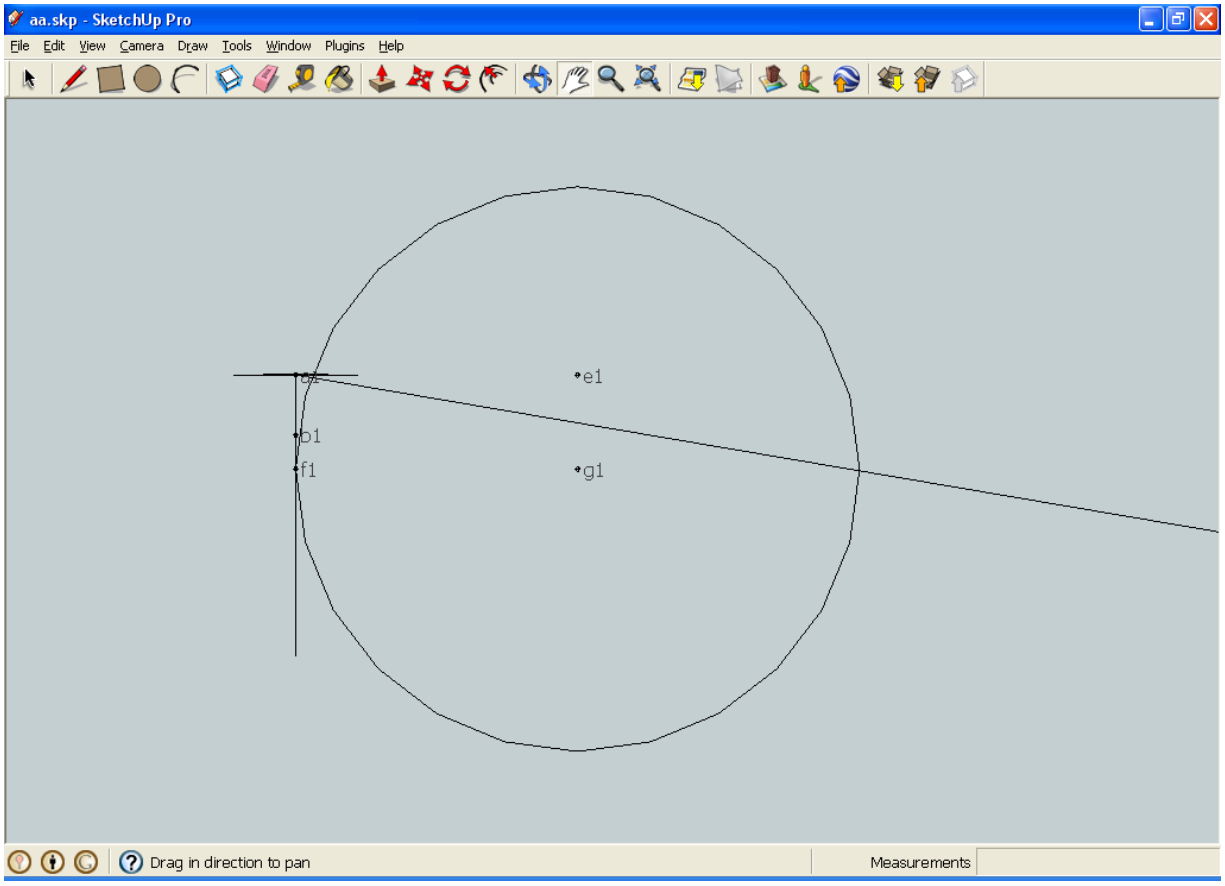
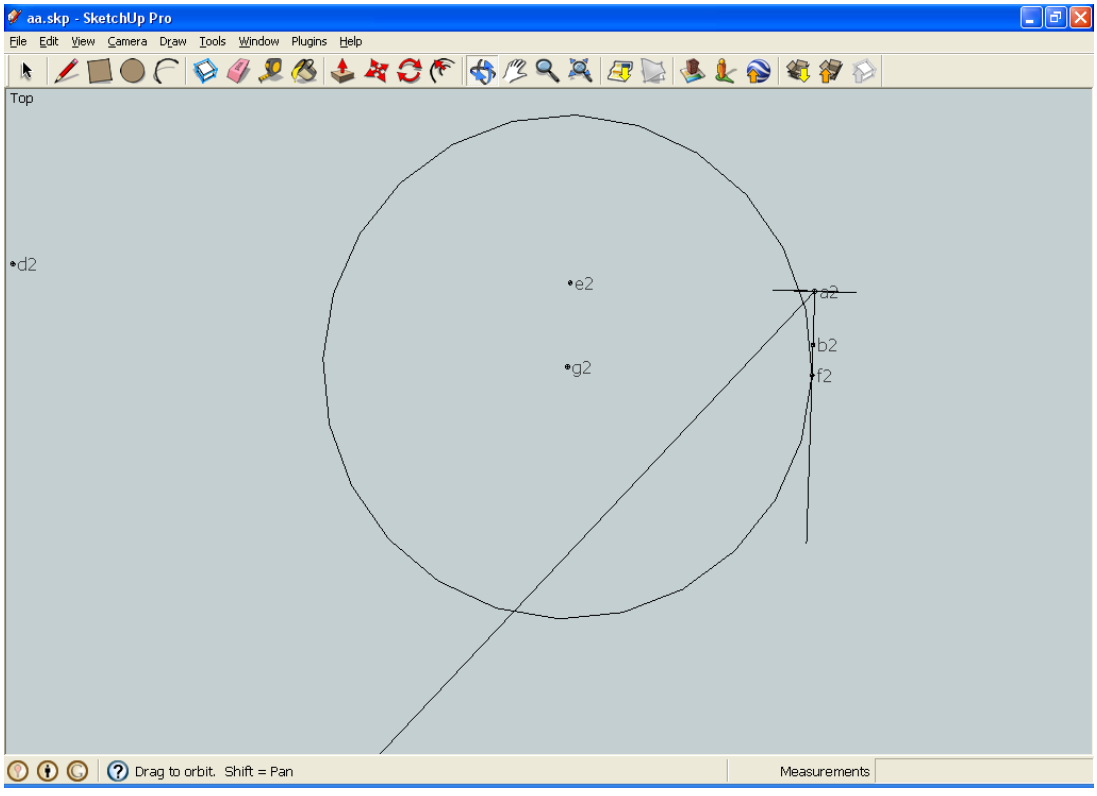
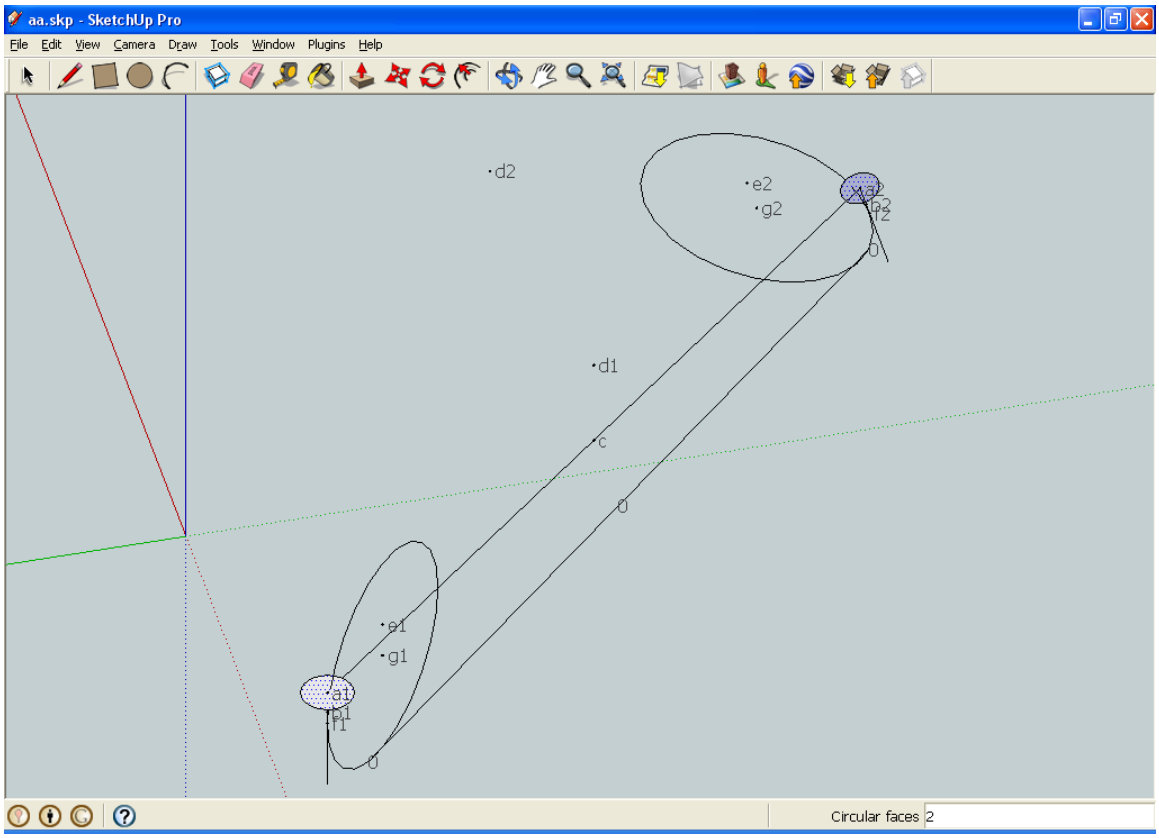
File Edit View Camera Draw Tools Window Plugins Help
d2
e2
g2
d1
c
e1
g1
a1
b1
f1
Circular faces 2

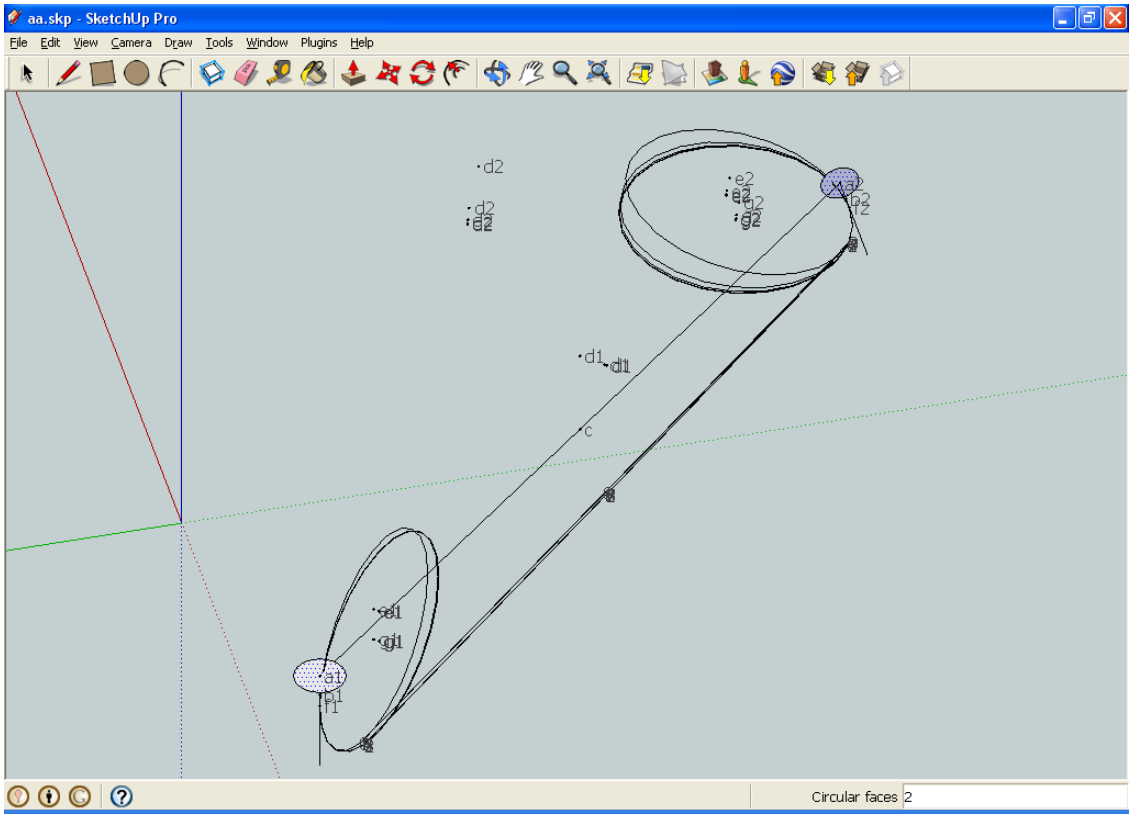

aa.skp - SketchUp Pro
File Edit View Camera Draw Tools Window Plugins Help
a1
e1
b1
f1
g1
Drag in direction to pan
Measurements


aa.skp - SketchUp Pro
File Edit View Camera Draw Tools Window Plugins Help
Top
d2
e2
a2
b2
g2
f2
Drag to orbit. Shift = Pan
Measurements

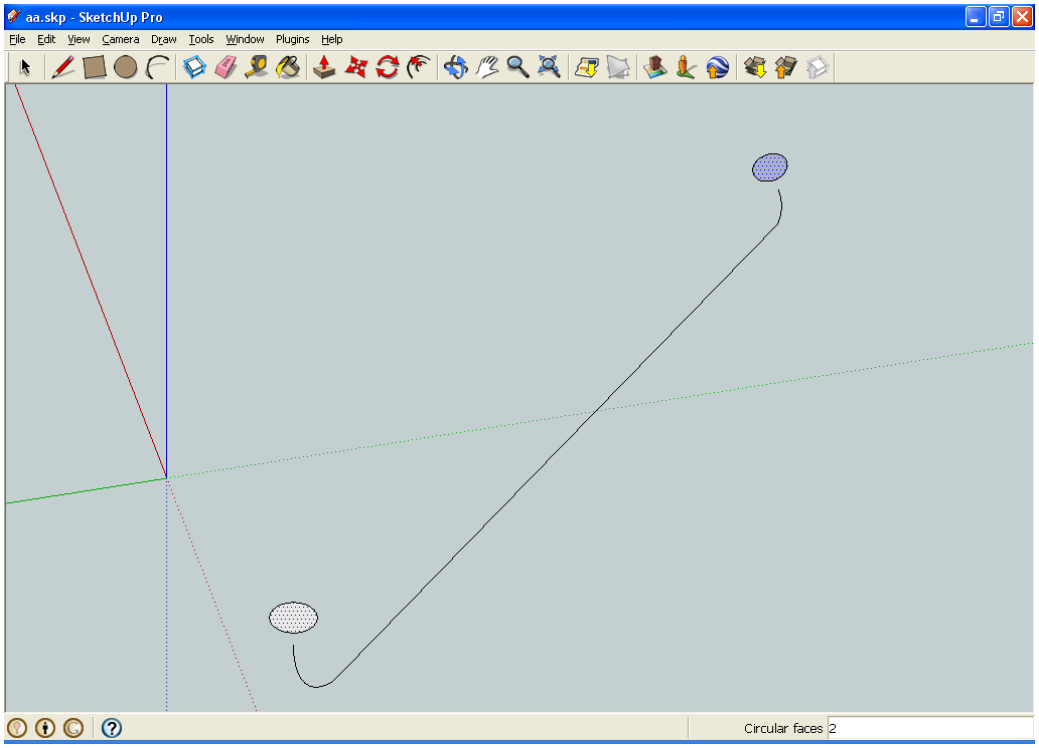
The plugin then finds the ‘best’ tangent line (minimizes angle errors of line with circle segments) to give:



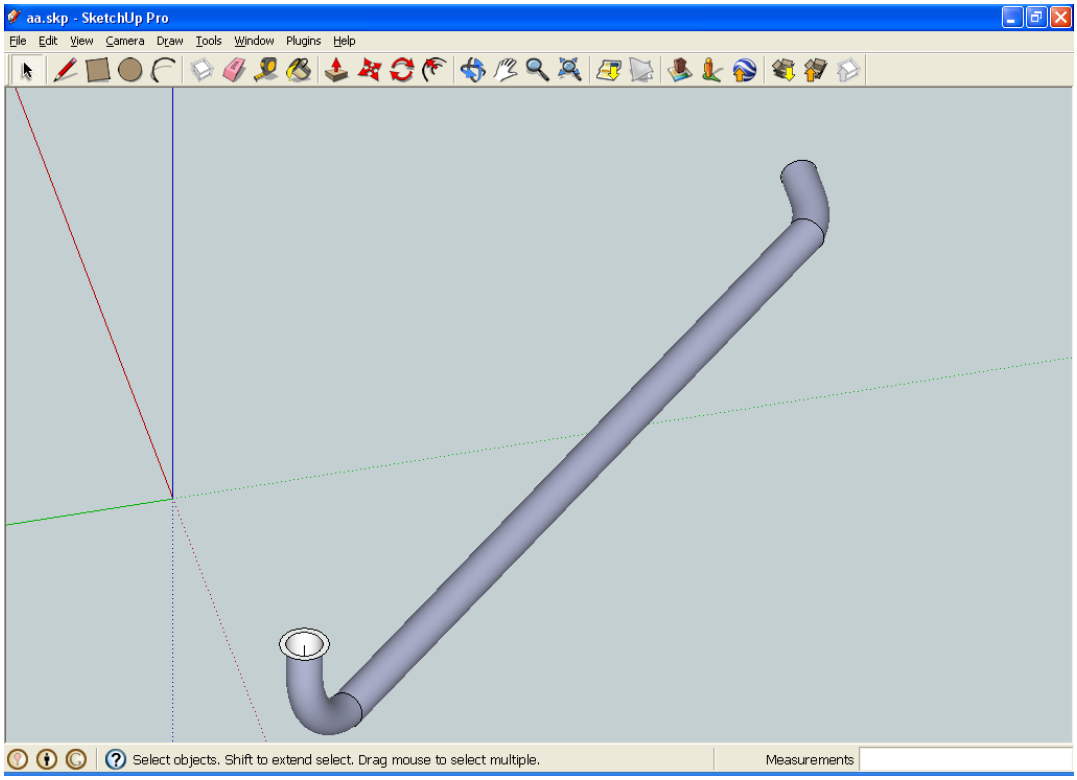
This is good but not good enough. The plugin repeatedly swings the major circles to minimize the angle errors between tangent lines and major circle segments:



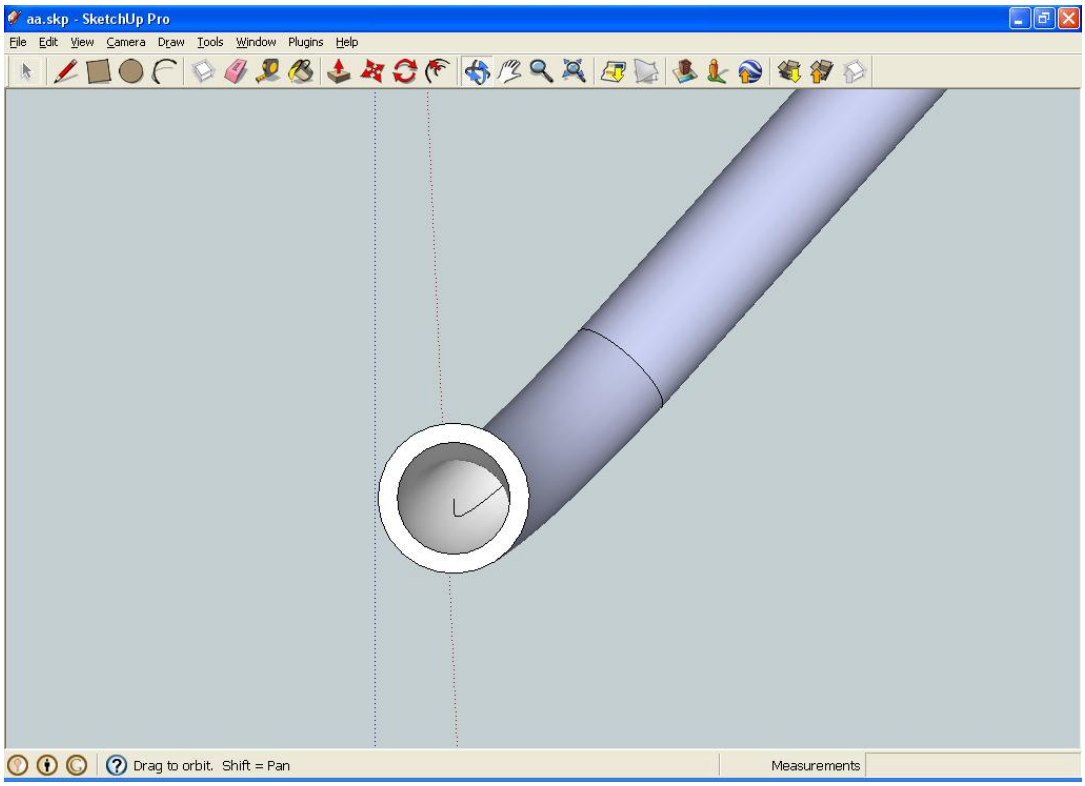
Then the best one is automatically selected.

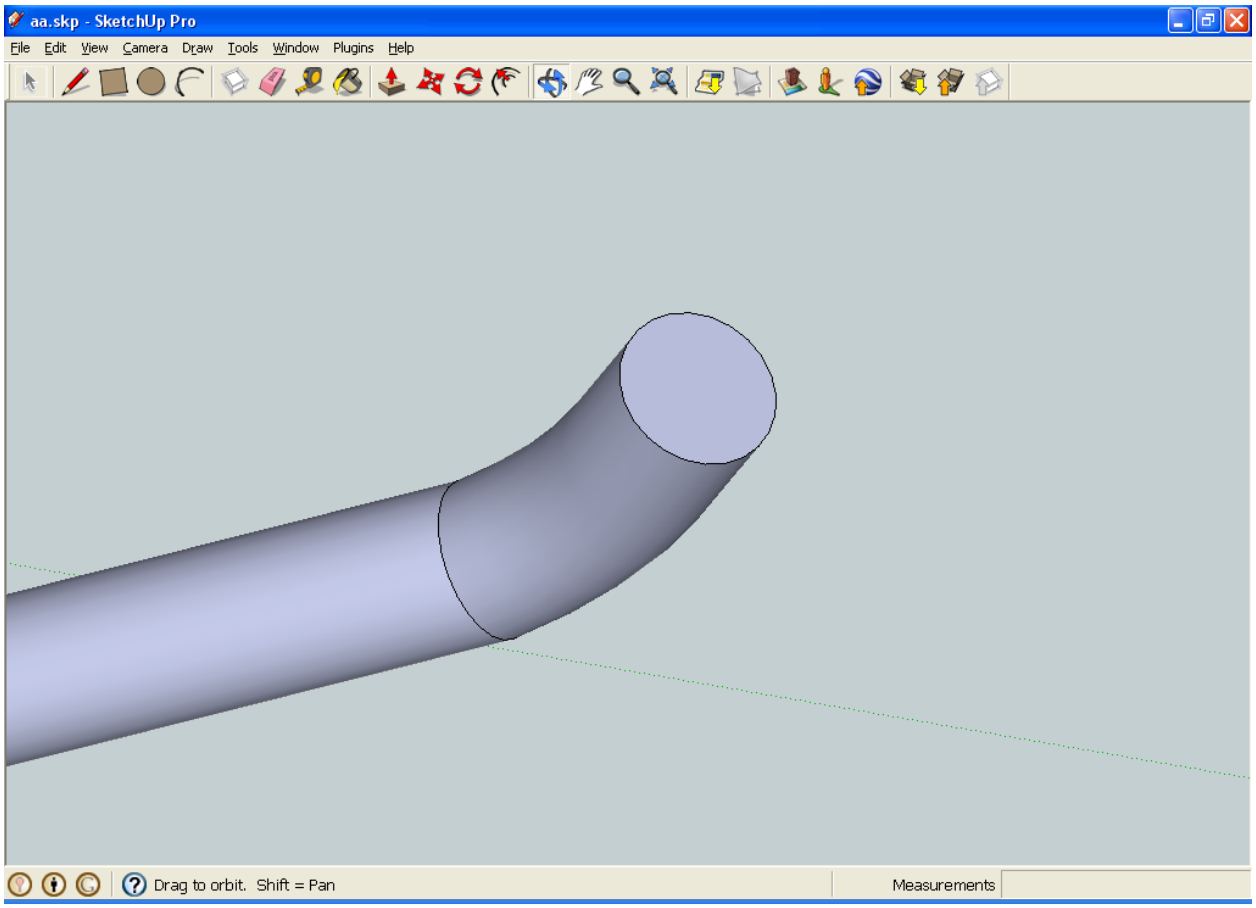


The Followme tool is automatically called to give this:



Which is the end result.




aa.skp - SketchUp Pro
File Edit View Camera Draw Tools Window Plugins Help
Drag to orbit. Shift = Pan
Measurements