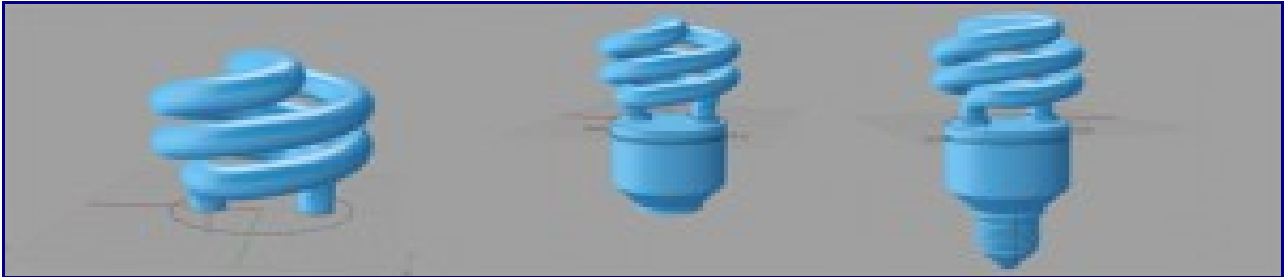


In this modeling tutorial you will be shown how to model a light bulb. This is english version of my tutorial, if you run on the video of my croatian version, its cool.

## Introduction

So like I said, in this tutorial you will learn how to model a light bulb in Rhino. The first “glass” part will be just a bit tricky because we need to control the curves little bit more than usual. After that we will make bulb screw threads, where the method is similar if not the same as the one explained on Rhino tips website. If only I known for it when I made this tutorial heh. Anyway, this one is pretty easy, but needs some special attention in some parts.

If you like, you can download source 3dm files step by step just to keep on the track.



source 3dm files step by step

This is what you will have in the end:



final model

## STEP 1

First we will start from top to bottom. So we're making the lighting spiral. Start with creating [Circle](#) from top view of radius 5cm. And create two small circles of radius 1cm. You can use Snap to make them really easy:

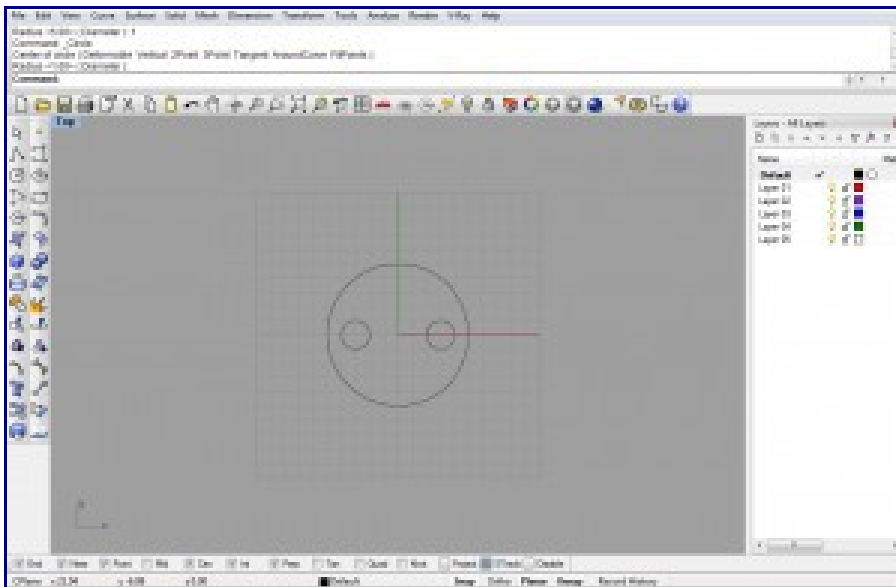


image 1

Next, we need a [Spiral](#). Using [Spiral](#) command make one starting from origin. Once you start the [Spiral](#) command just input 0 (zero) and press enter, now from Front viewport make it high 5cm. And for the radius use your already made circle that is 5cm in radius. [Move](#) that spiral 3cm up from front viewport and you should get something like on the image 2 and image 3.

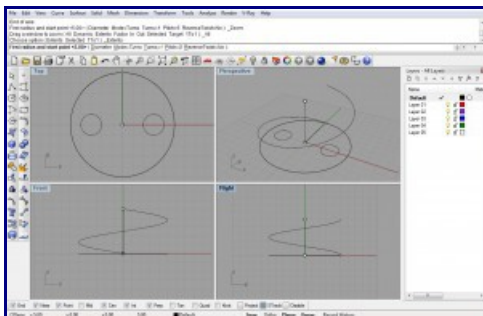


image 2

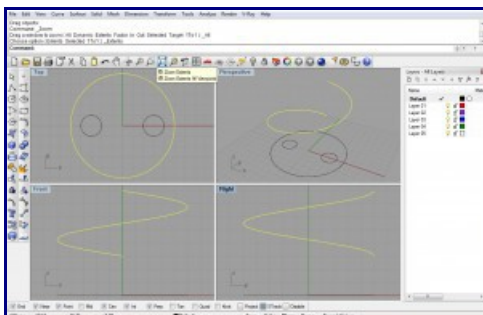


image 3

Now we need a little help line. You can make one starting from center of the right smaller circle. It should be straight vertical line, so you can finish it either from front or right viewport:

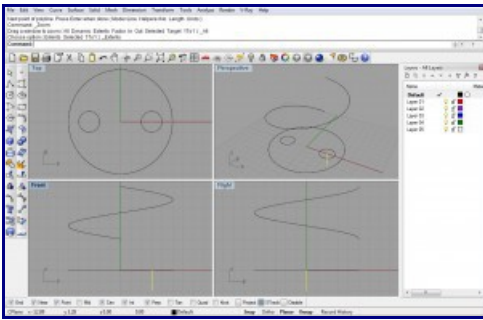


image 4

Now, using [BlendCrv](#) blend the lower end of spiral to the little help line with G1 continuity on both sides:

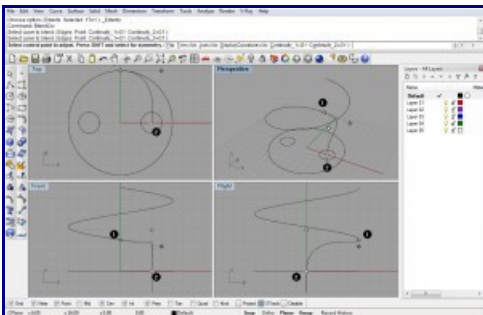


image 5

Using [Copy](#) command, copy the spiral and blended curve. While in the [Copy](#) command in the command line click on InPlace option. Or simply use [Copy](#) & Paste commands (ctrl+c, ctrl+v). While having selected two copied curves, [Rotate](#) them from top viewport by 180 degrees. Check if you have something like on the image 6:

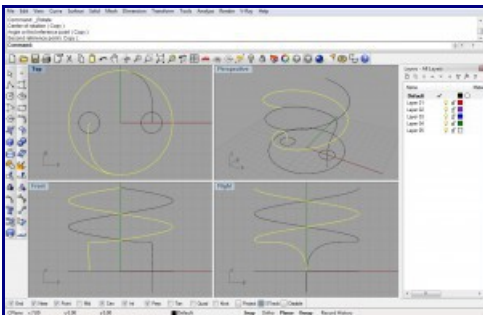


image 6

Now, with create a [Point](#) from right viewport (or front would work fine too) exactly 9cm from origin:

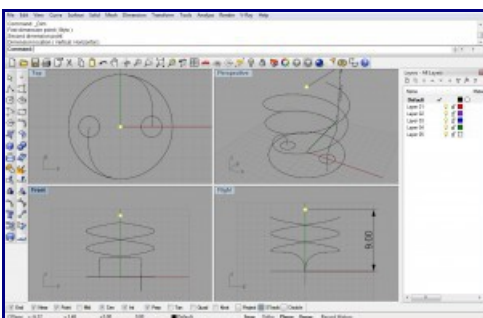


image 7

Now, start [Extend](#) command, and press enter for dynamic extend. Enable OSnap and check Point

option, and click on each upper end of spiral and extend it to the point.

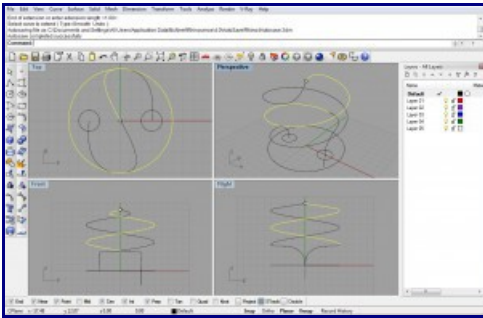


image 8

Now the endings of two spirals are not tangent. So with [Match](#) command we will match those two curves. Make sure both ends are set to Tangency and Average Curves is checked:

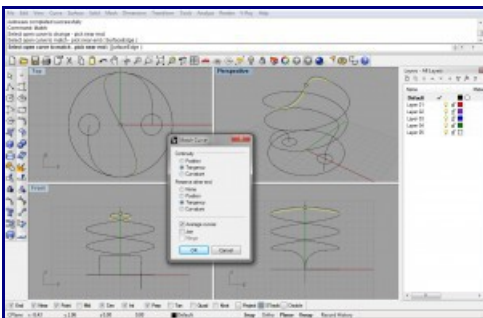


image 9

Now select all 4 parts of 2 spirals and join them together. Now, using [Pipe](#) command we will make a surface out of this curve. For the starting and ending radius you can use two small circles as reference. Near OSnap option is a good way for this. Ofcourse, you could use sweep1 or even sweep2 commands.

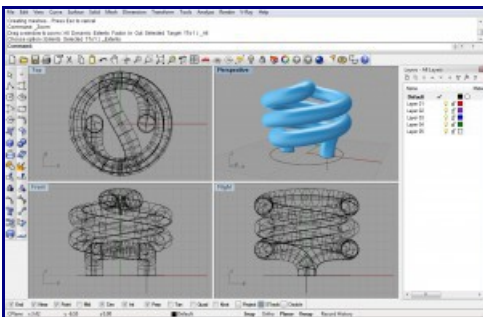


image 10

Now the big circle will be used to create the body of our bulb. Select it and run [ExtrudeCrv](#) command. Make sure the Cap is set to Yes and BothSides to No. We will extrude it down by 8cm, so you can either use snap and count for yourself, or simply input in the command line -8 and press enter.

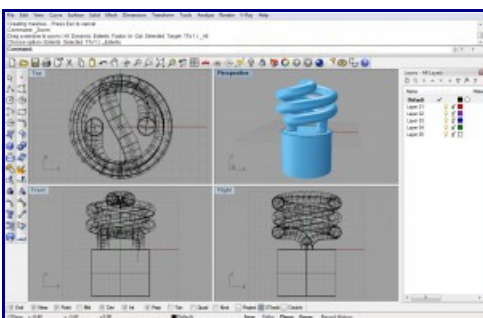


image 11

Using [FilletEdge](#) command fillet the upper edge of the cilinder by 0.5cm.

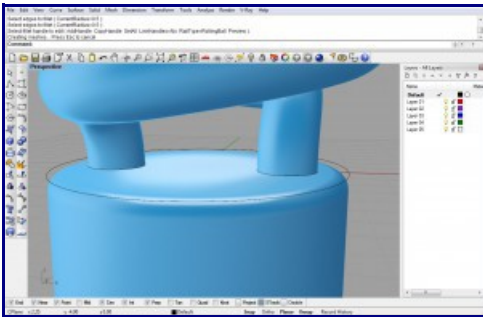


image 12

Using one of two smaller circles offset one by 0.3cm. And move it up vertically by 0.6cm.

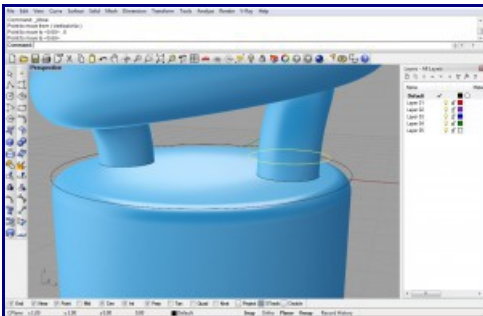


image 13

Using [Rib](#) command and the body of the bulb create a surface like on the image 14. [Rib](#) distance is 0.2:

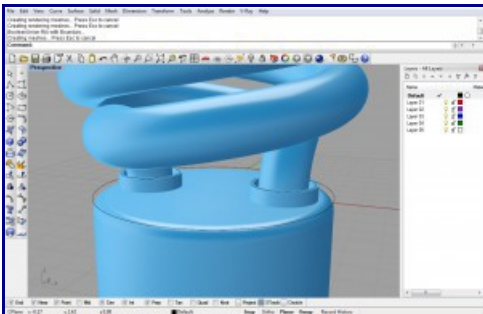


image 14 - mirrored curve for rib

Now for the ribs, we can make them look even prettier by filleting edges. So with [FilletEdge](#) command fillet those three edges for each rib with 0.05cm as radius:

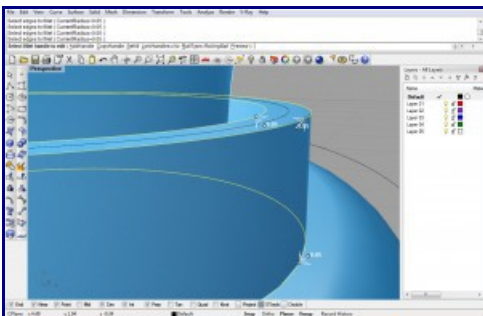


image 15 - filleting

Now lets get back to the lower part of the bulb body. Like we filleted upper edge, we will chamfer

the lower one with [ChamferEdge](#) command using 2cm as CurrentChamferDistance option.

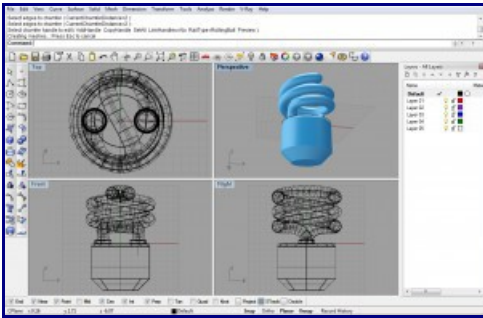


image 16 - chamferedge

Now, we will FilletEdge the middle edge with 0.5cm as fillet radius.

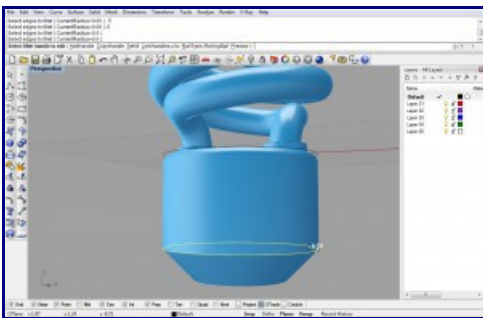


image 17 - filletededge radius 0.5cm

Using [ExtractSrf](#) command we will extract the bottom cap surface:

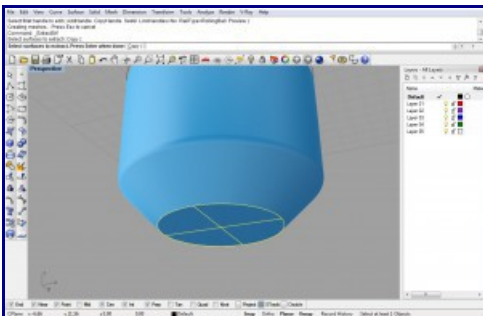


image 18 - omg, we're using ExtractSrf for the first time :D

When you extract this surface, you can delete it. And using the edge where this surface was extrude it by 1cm down. So just input in command line when extruding -0.5. Make sure the Cap option is set to No.

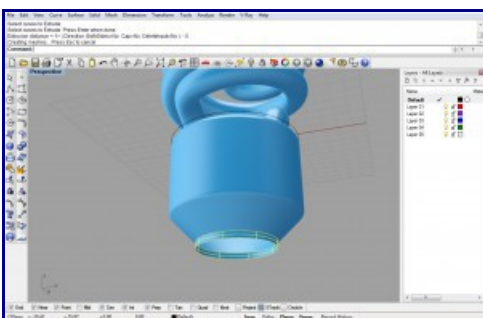


image 19

Now, you can create [PlanarSrf](#) to “cap” it on one side: (you could also join all the upper surfaces of this bulb body and simply use Cap command to close it)



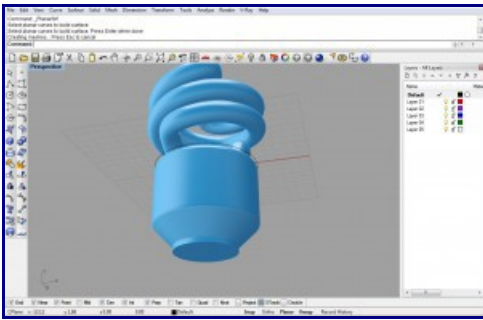


image 20

Lets do some details on the body. From front viewport create one line and offset it down by 0.3cm... Like on the image 21

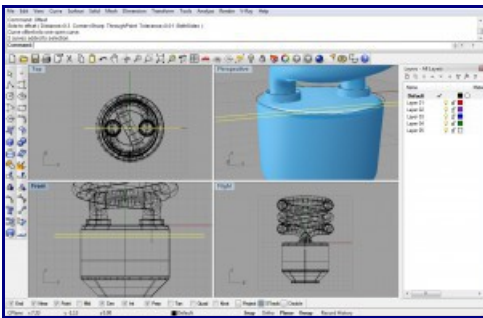


image 21

With [Trim](#) command trim the body surface between two lines:

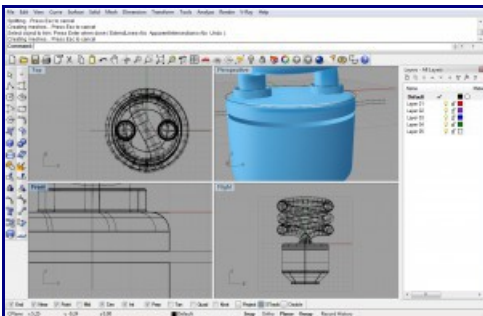


image 22

Using [Offset](#) command, offset the two edges inwards by 0.2cm. The easiest way is to first have everything deselected and run the command offset, then click on one edge from perspective view, and then from top viewport simply move your mouse pointer inside the cilinder and in command line type in 0.2 and press enter. Do the same for the other edge and you are set to go.

Then, using [Loft](#) command click on the lower edge first, then two offset circles inside, and then the upper edge, and with style set to Straight sections create loft:

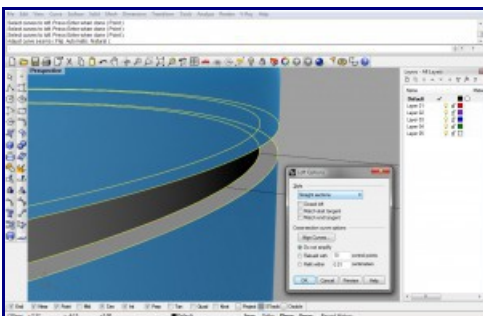


image 23

Now, join those surface, and make it prettier with FilletEdge using 0.05cm as radius.

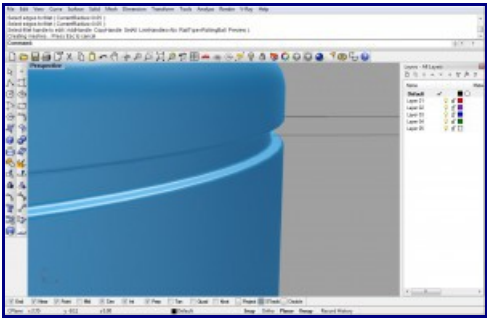


image 24 - smooth edges with FilletEdge