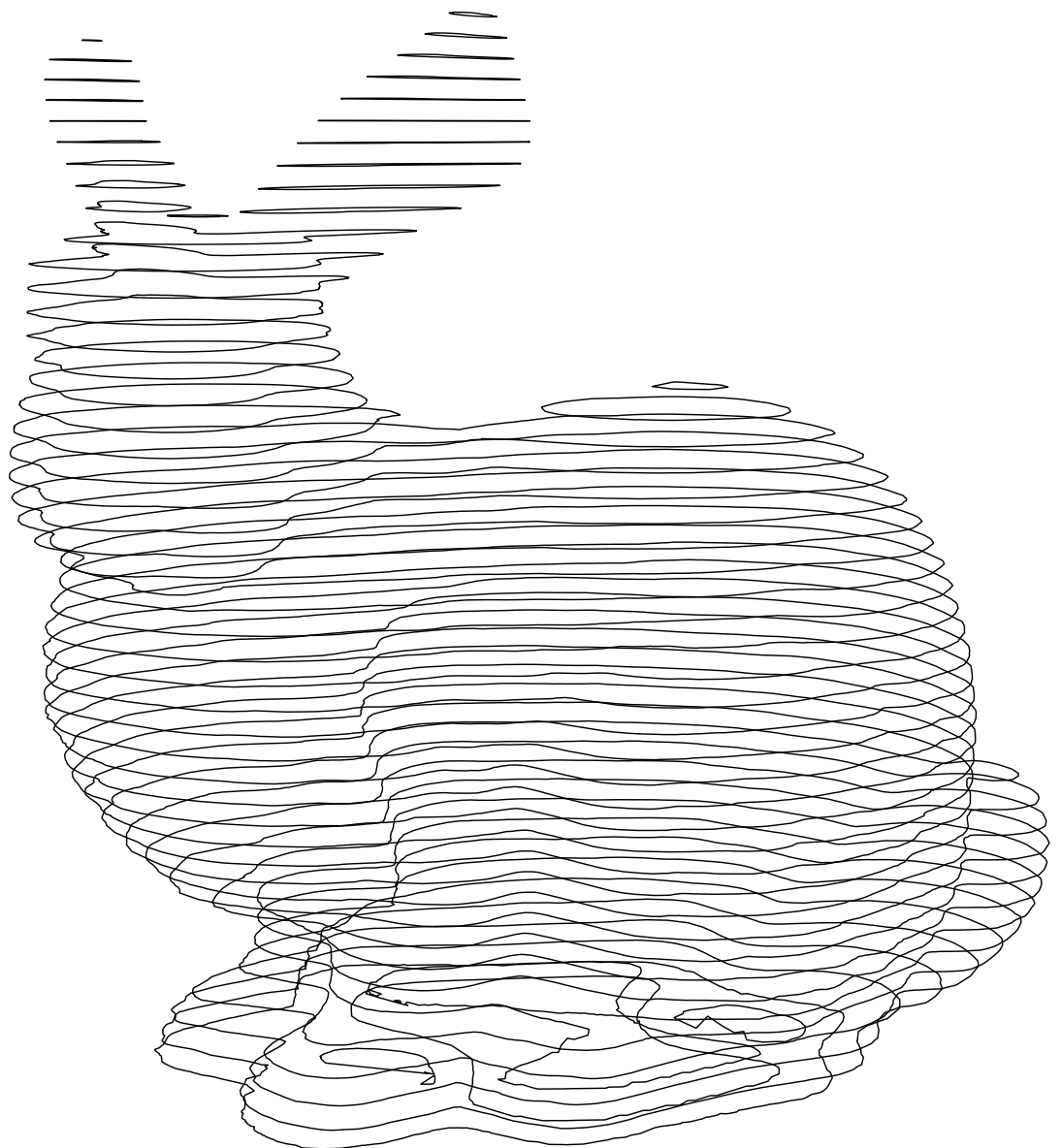


# Rhino 3D - Digital Crafting Tutorial

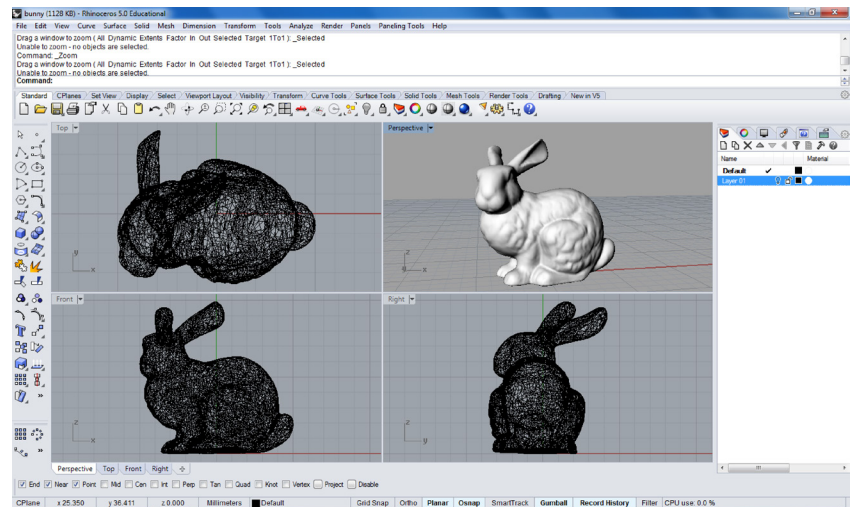
instructions for simple shape fabrication

## Contouring

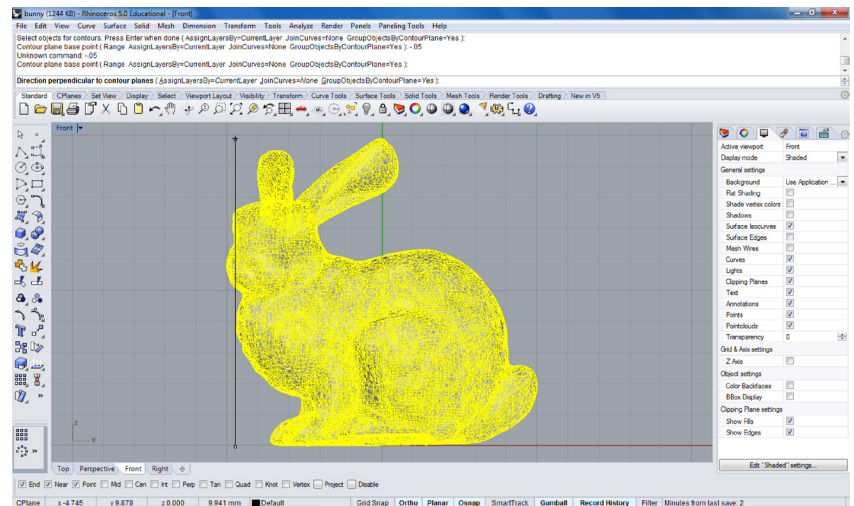


# Rhino 3D Digital Crafting Tutorial: Contouring

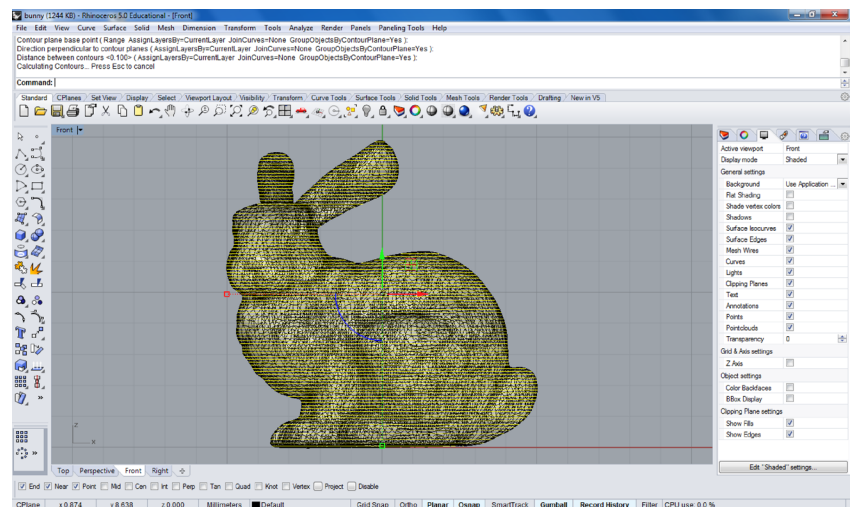
1. Draw or import your geometry into the Rhino workspace.



2. Switch to Front View. Use the command \_Contour and select your object. Draw a vertical line starting just below the geometry and press Enter.



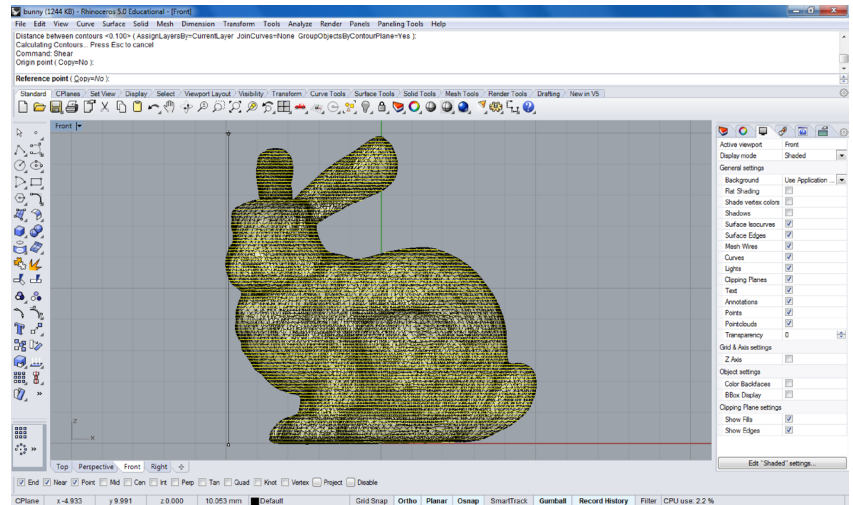
3. Now define the distance in between contour lines, which should be equal to the material thickness you are going to use.



# Rhino 3D Digital Crafting Tutorial: Contouring

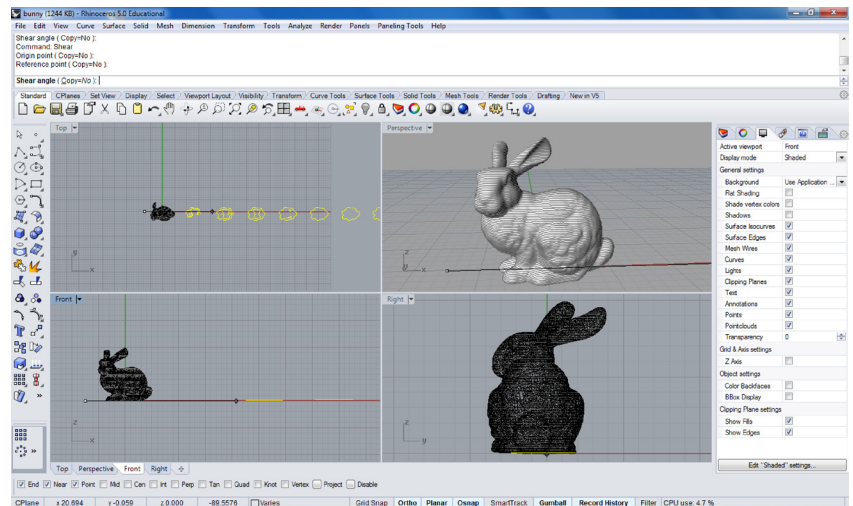
4.

Remaining in Front View and while having contour lines still selected type the command `_Shear`. Again draw a vertical line next to your geometry.



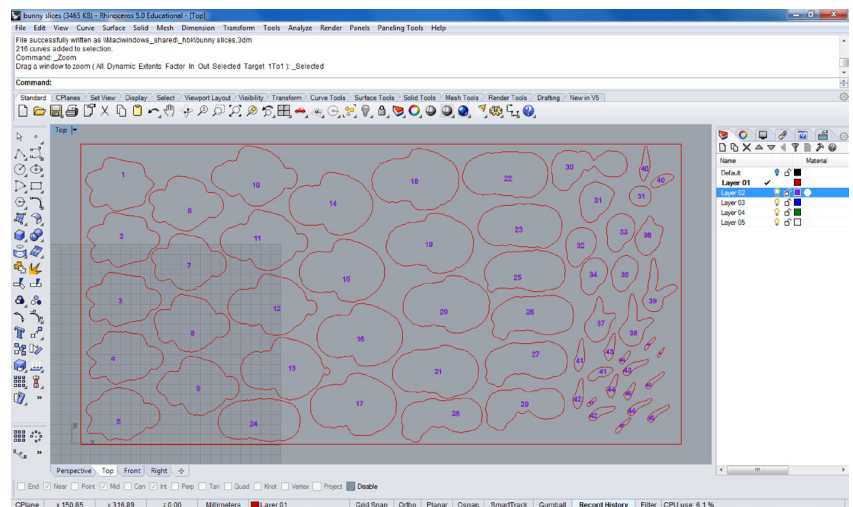
5.

While the `_Shear` command is still active switch back to all four windows and zoom out of Front and Top View. In the Front View window move your mouse to the right until in Top View the layers don't overlap any more. Turn off Osnap if necessary.



6.

Switch to Top View and use the command `_ProjectToCPlane`. Choose 'Delete input objects? Yes'. Arrange all lines so they can be sent to a laser cutter. In case certain lines need to be deleted the shapes might have to be ungrouped (`_Ungroup`) first. Labels can be added by using the command `_TextObject` (Curves, Allow Single Stroke Fonts). Single stroke fonts allow for faster manufacture. Here are some fonts that work well for engraving: <http://wiki.mcneel.com/rhino/engravingfonts>







finished object