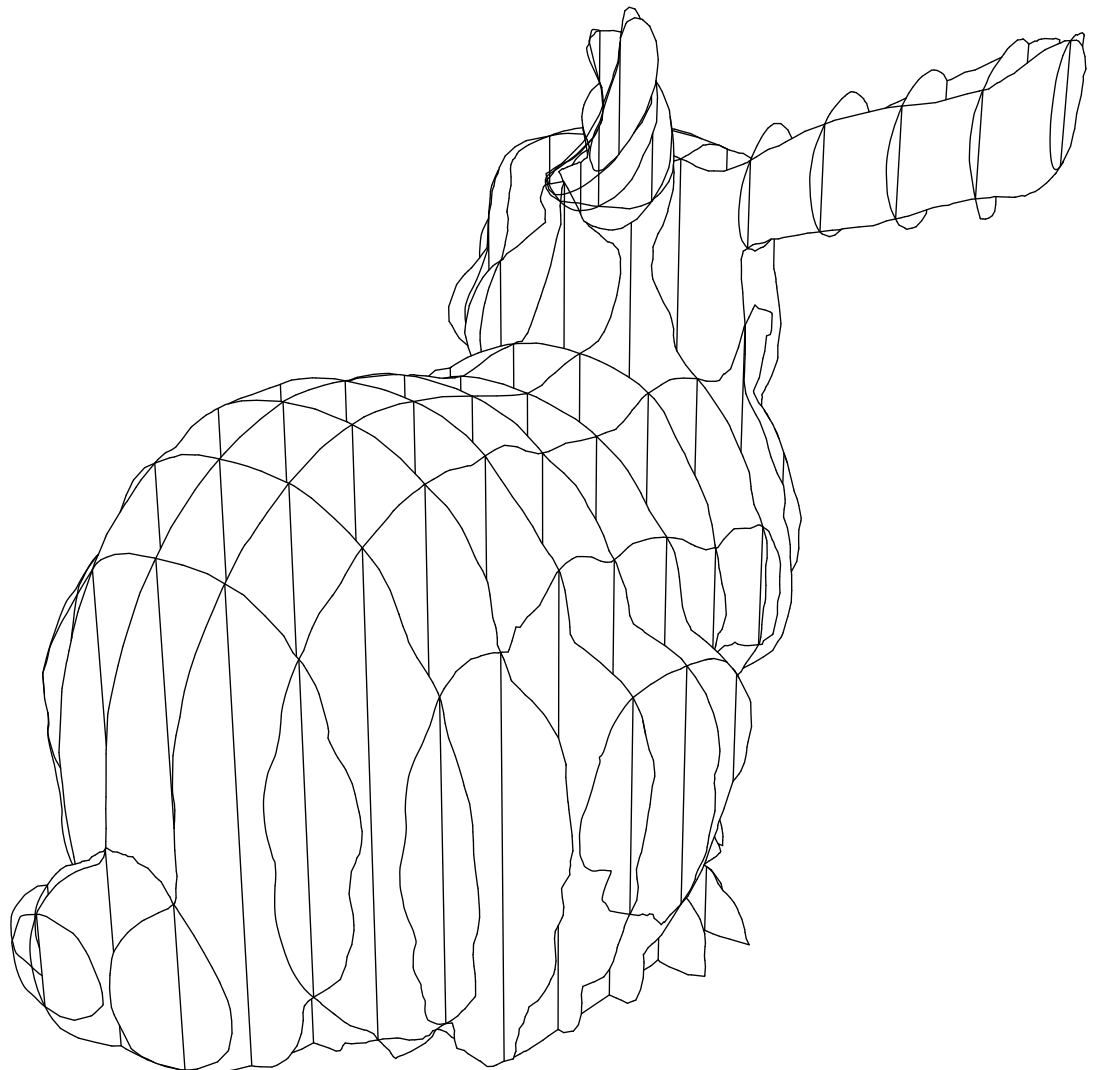


Rhino 3D - Digital Crafting Tutorial

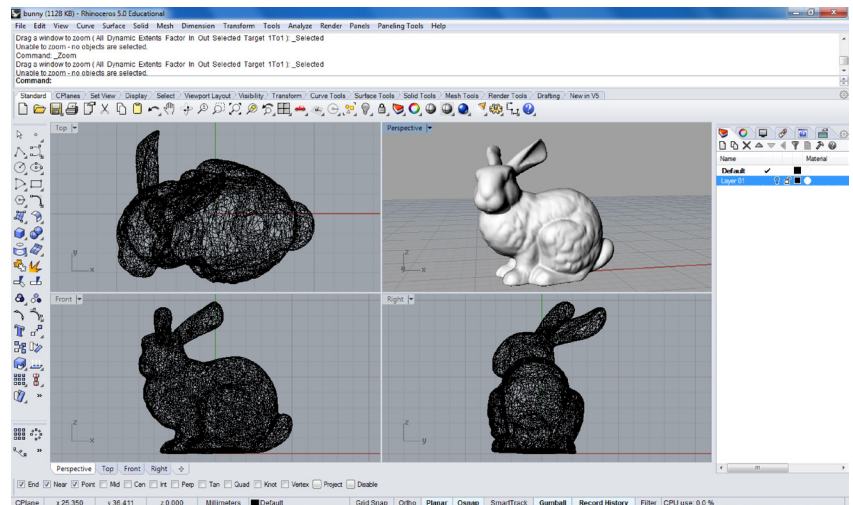
instructions for simple shape fabrication

Waffle

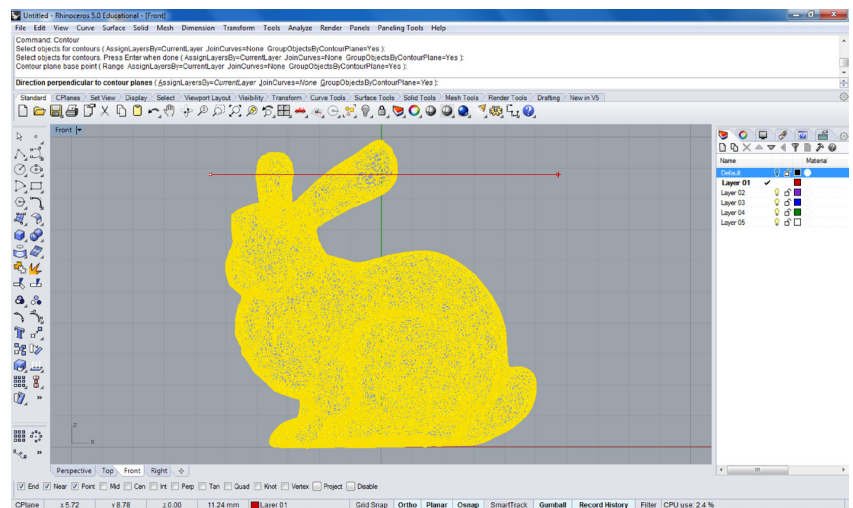


Rhino 3D Digital Crafting Tutorial: Waffle

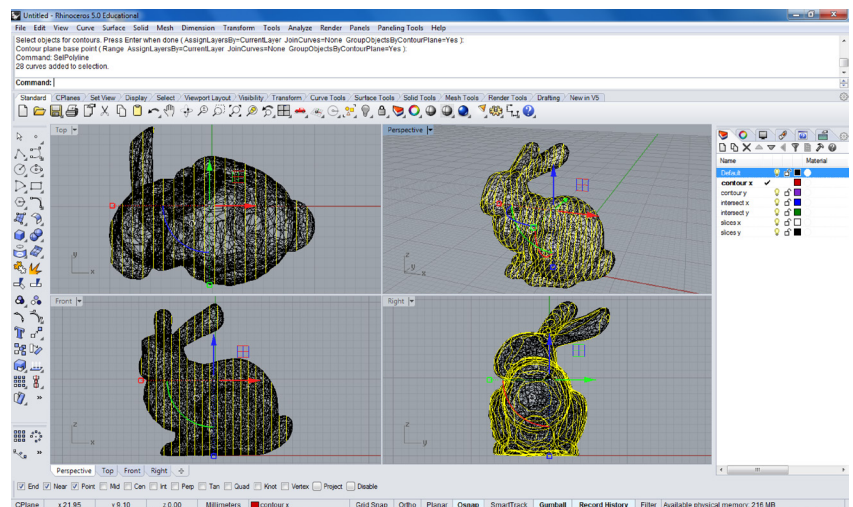
1. Draw or import your geometry into the Rhino workspace.



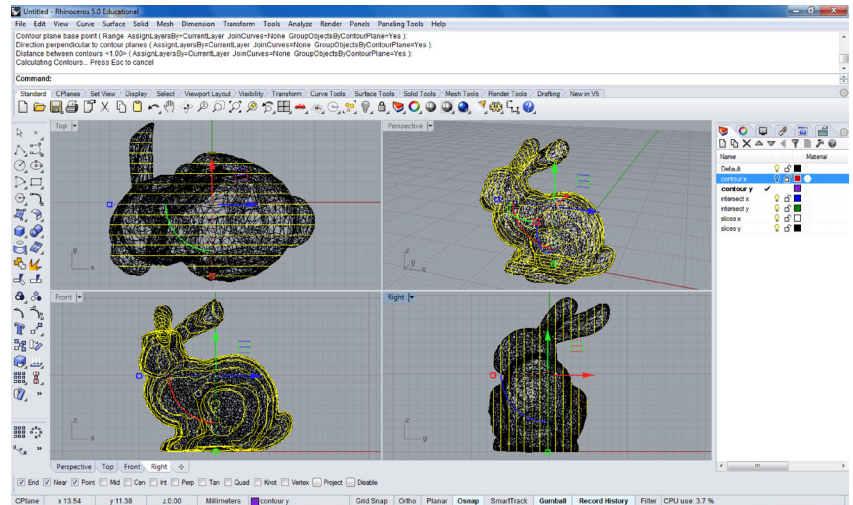
2. Create and select a new layer or rename an existing one to 'Contour X'. Switch to Front View. Use the command `_Contour` and select your object. Draw a horizontal line starting just before the geometry and press Enter.



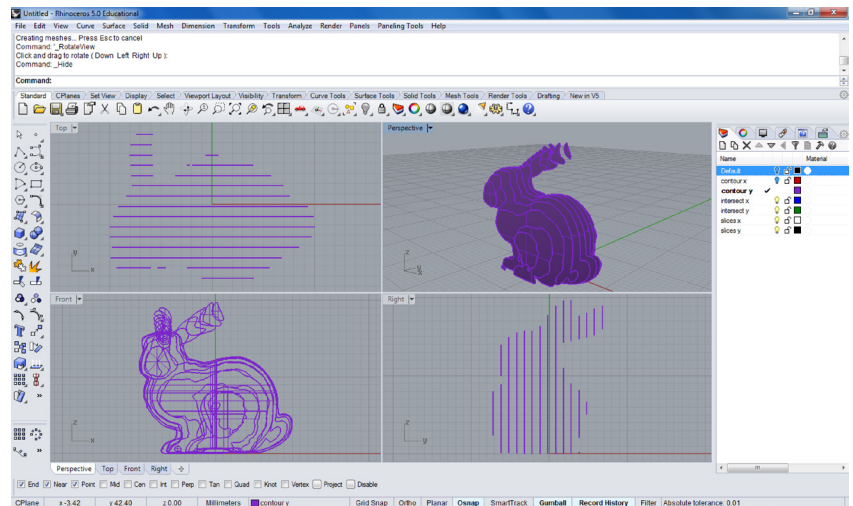
3. Now set the distance in between contour lines, which will define the spacing of the waffle grid.



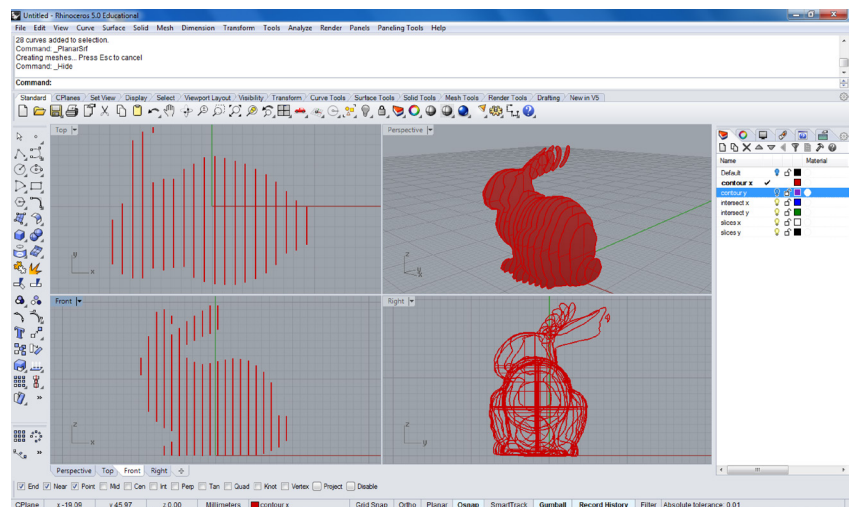
4.
Create another layer or rename an existing one to 'Contour Y'. Switch to Right View and again use the command `_Contour`. The spacing can vary from the previous, depending on the grid you want to create.



5.
Turn off layer 'Contour X', select all lines on layer 'Contour Y' (`_SelPolyline`) and type `_PlanarSrf`, which creates planar surfaces. Do not hide or delete the contour lines since they will still be needed later.

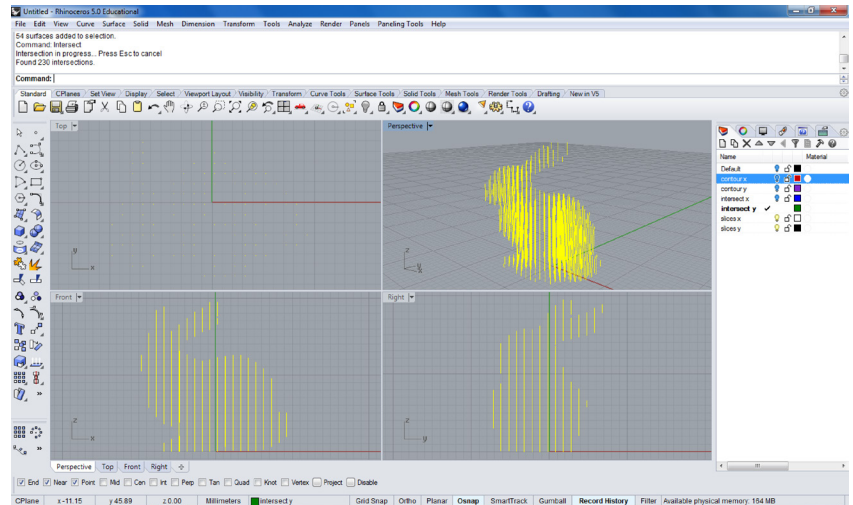


6.
Repeat the procedure for layer 'Contour X' while turning layer 'Contour Y' off. If an error concerning 'self-intersecting lines' occurs, rebuilding (`_Rebuild`) the contour lines with a point count similar to the original might solve the problem.



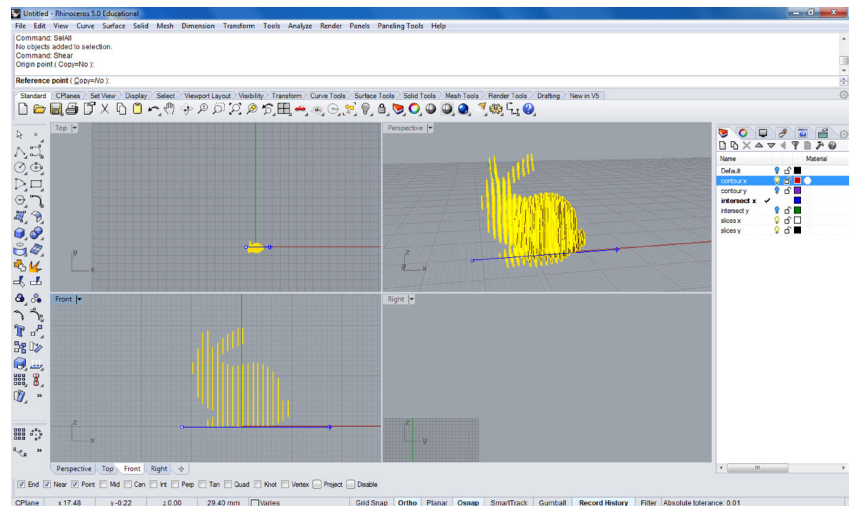
7.

Turn on layers 'Contour X' and 'Contour Y' and switch to a new layer 'Intersect X'. Select all surfaces (_SelSrf) and type _Intersect. Depending on the size of your geometry this process will take some time. Copy the lines, while they are still selected, to the layer 'Intersect Y' by right-clicking on the layer and choosing 'copy objects to layer'.



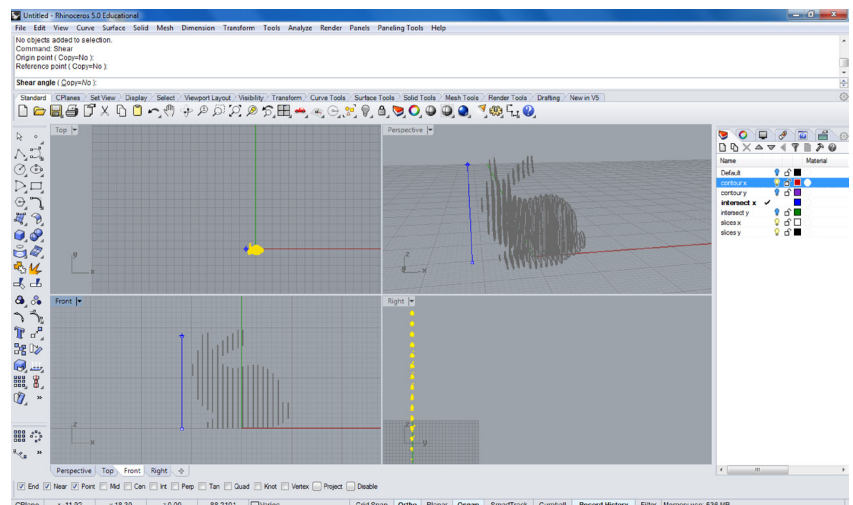
8.

Turn on layers 'Contour X' and 'Intersect X' and select all visible objects (_SelAll). Switch to Front View and type _Shear. Now draw a horizontal line starting just left before your object towards the right.



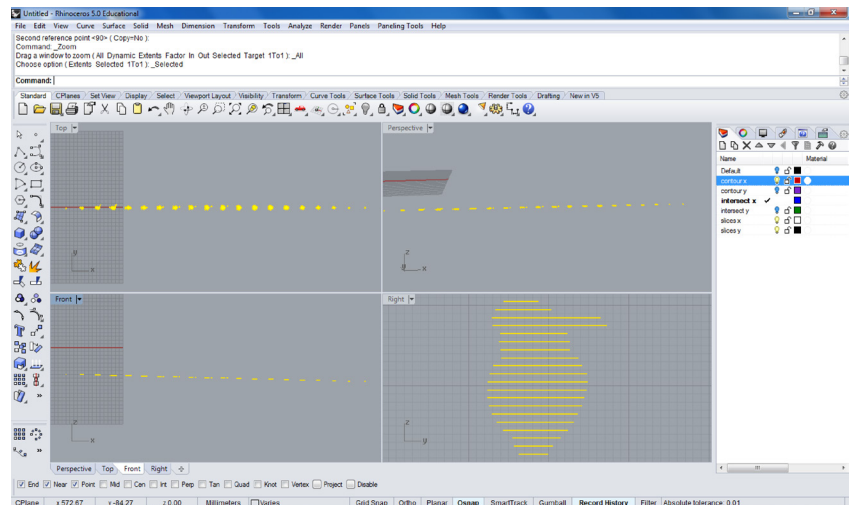
9.

While the _Shear command is still active zoom out of Front and Right View. In the Front View window move your mouse to the top until in Right View the surfaces no longer overlap. Turn off Osnap if necessary.



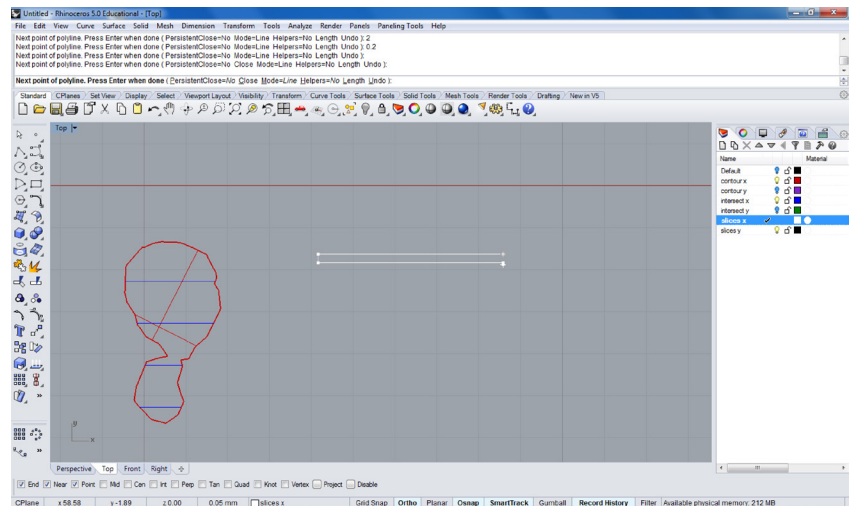
10.

Select all objects (**_SelAll**) and in Front View **_Rotate** them for 90 degrees. In Top View use **_ProjectToCPlane** ('Delete input objects? Yes') to map them onto the X-Plane.



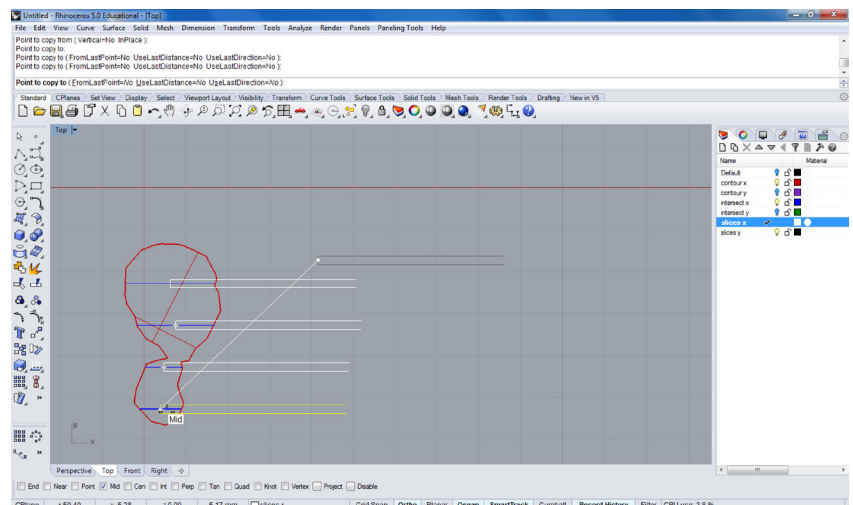
11.

Enlarge Top View and switch to the layer 'Slices X'. Zoom into your first slice and now draw a profile notch (**_Polyline**), with the thickness of your material.

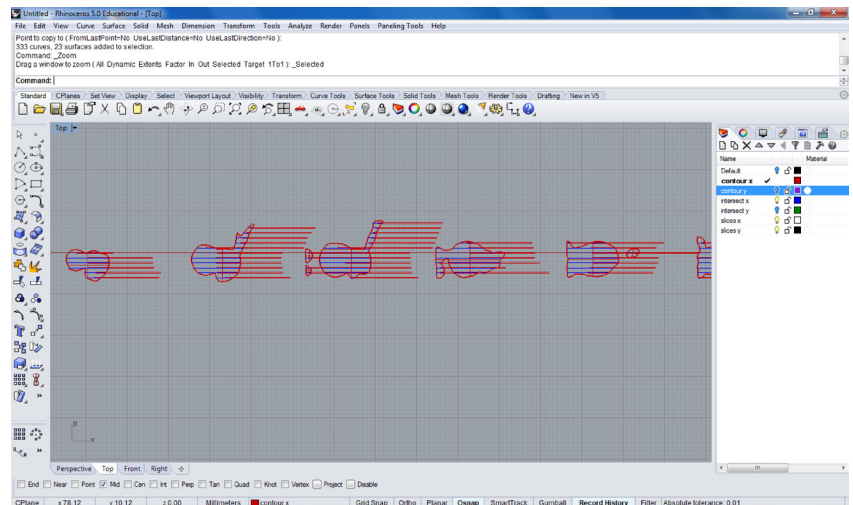


12.

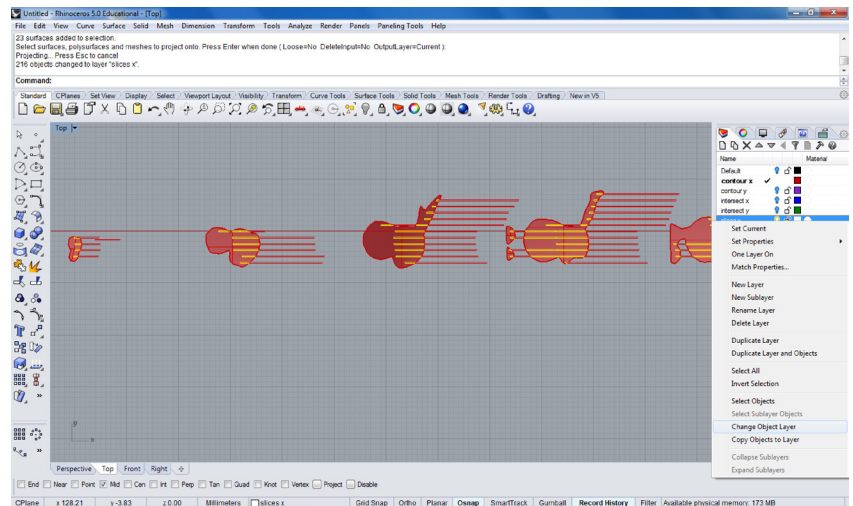
In your Osnap deselect everything except for 'Midpoint'. Now copy the profile onto every single slice-line on your shapes.



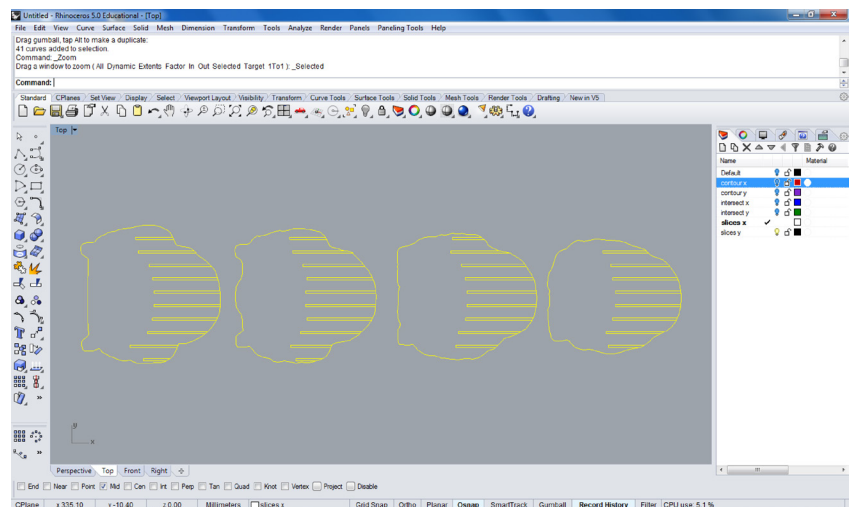
13.
Repeat the process until all shapes are filled.



14.
Turn off the layer 'Intersect X' and select all lines (_SelPolyline). Use _Project and when asked to 'select surfaces to project onto' type SelSrf. Move the projected lines onto the layer 'Slices X'. Use _SelClosedCrv to select the original contour curves and also move them to the layer 'Slices X'.

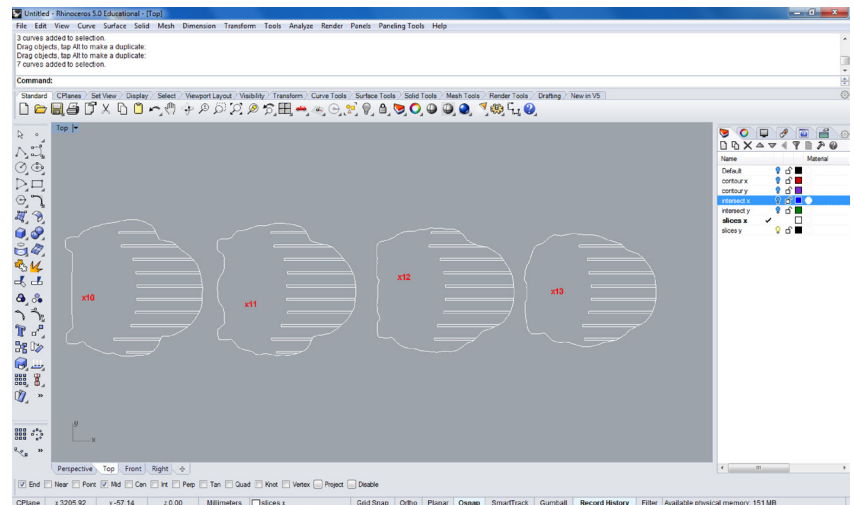


15.
Double check your created shapes for overlapping or intersecting lines.



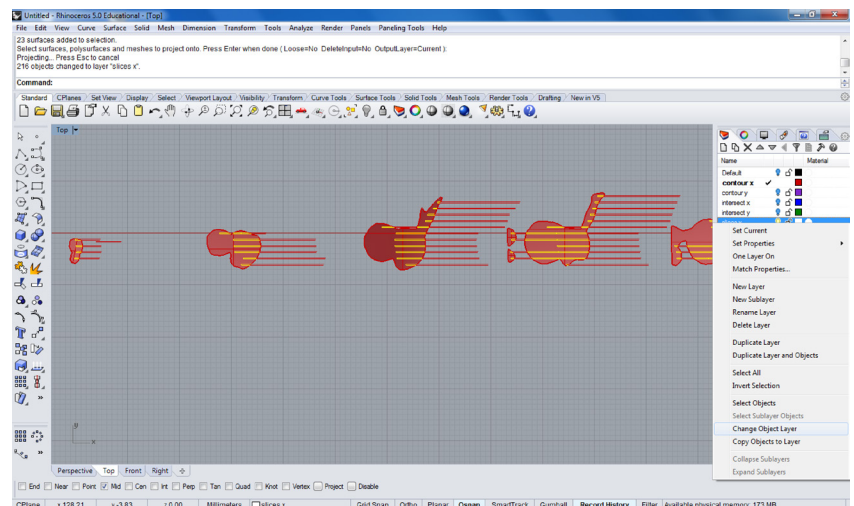
16.

Turn off all layers except for 'Slices X' select the first slice. 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>. Type `_Group` to group all lines and labels to the respective slice and repeat the process for each slice.



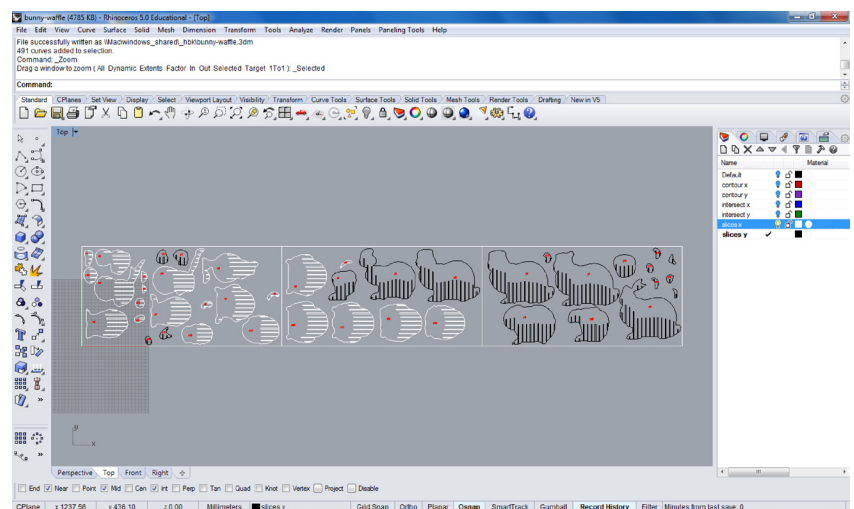
17.

Repeat steps 8 to 16 for 'Slices Y'.



18.

When finished with all slices draw a rectangle (`_Rectangle`) the size of your laser-cutter bed and arrange the pieces neatly.





finished object