

Burcu Nimet Dumlu

Digital Architectural Design & Modelling

Column Capital Modelling and Fabricating with Different Tools

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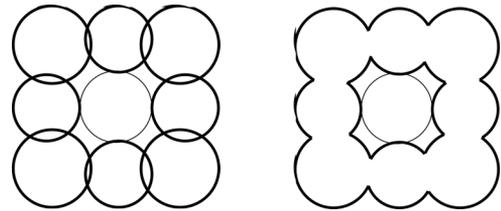
This project is about Parametric column capital design with grasshopper modelling. After modelling with grasshopper we create it in rapid prototype technique. Start point of this column capital is twelve surface which drawn in rhinoceros with curve tool and loft between bottom and top. These curves' base point is the circle. Creating one circle and multiplying it twelve times. Corner circles' radiuses are bigger than the middle circles. These circles are crossing. Then they are trimmed with trim tool and joined them together. It became two curves one inside one outside. At this point output of loft tool cannot be predicted, so they both keep to loft. If inside curve is not necessary for this model, it could be deleted after preview of loft tool output. These curves are on the X-Y plane. There must be curves on Z plane to creating loft surfaces. So Z elevation has been divided two parts; first one is the middle point where column capital is shrink, second one is end point.

Then making loft between base and z plane curves. All surfaces are ready for grasshopper phase.

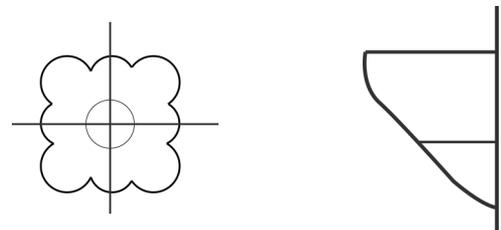
PROTOTYPE

After this process grasshopper phase starts. "Surface" element is created in panel and these twelve surface is introduced with "set multiple surfaces". Then my process is continued with "populate geometry" element. this element creates random points on my column surfaces. my intend of creating this points is assigning geometric elements inside these points and these geometries will define my parametric column capital. It will be parametric because every change in the populate geometry counts or geometric dimensions affects the last product of column capital. My prototype will be composed with SPheres which i defined R.60 u.Count.60 v.count.30 parameters. I baked the model with grasshopper baking. The parameters are like the following diagram. pop.geo. count60 -Seed. 100. and msphere. r.60 u.count.60. v.count.30.

Then this model is created in 3d with rapid prototype machine. I named my model - column of clouds.

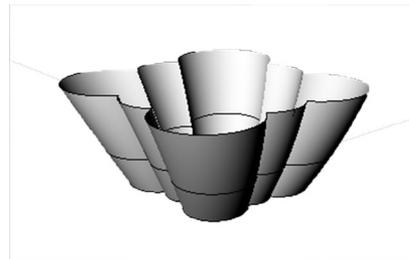


1 Base of Column Capital with Circles and Curves

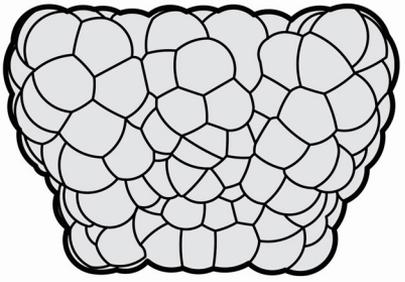


2 Outside curve of the column base

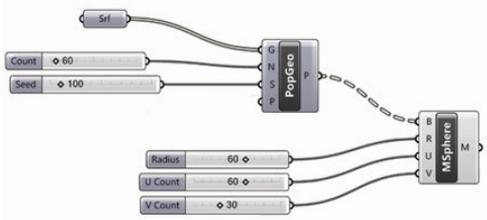
3 Z Plane Curves of Column. (Mid and Top)



4 Column Capital With Loft in Rhinoceros



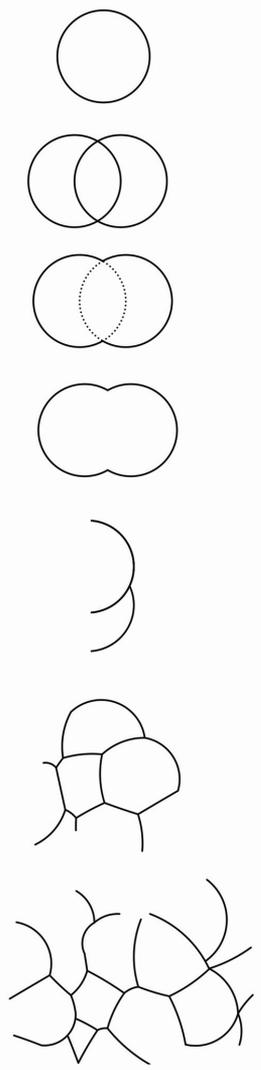
6 Frontal Diagram of Sphere Shapes



7 Grasshopper Codes of Column Model

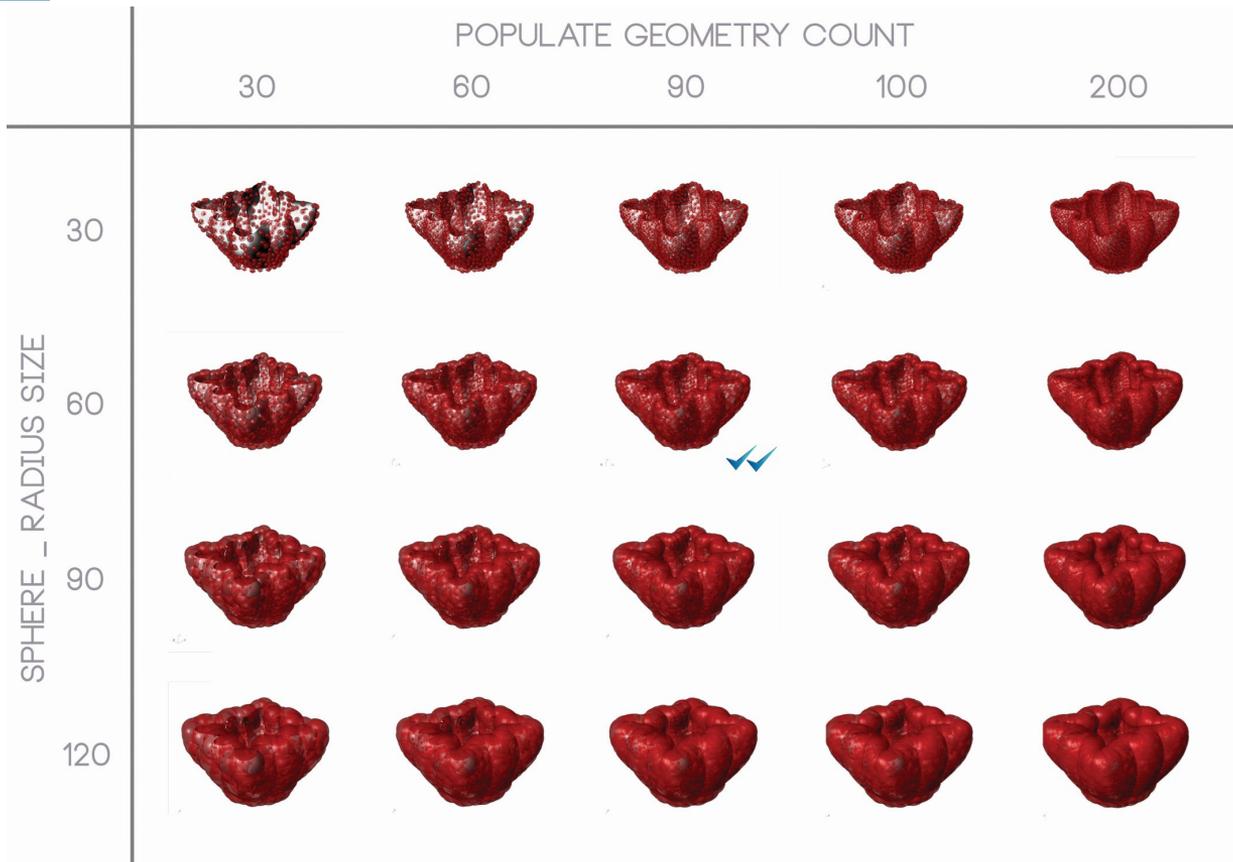
Geometric Shape Diagram

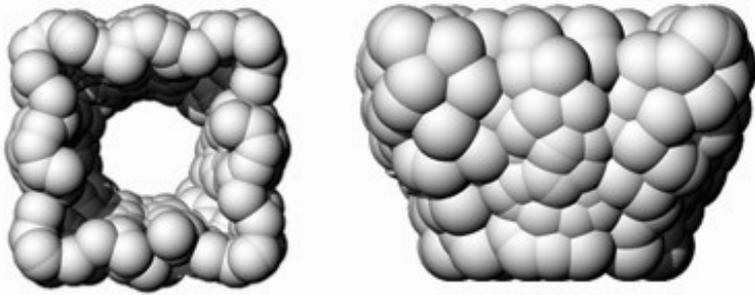
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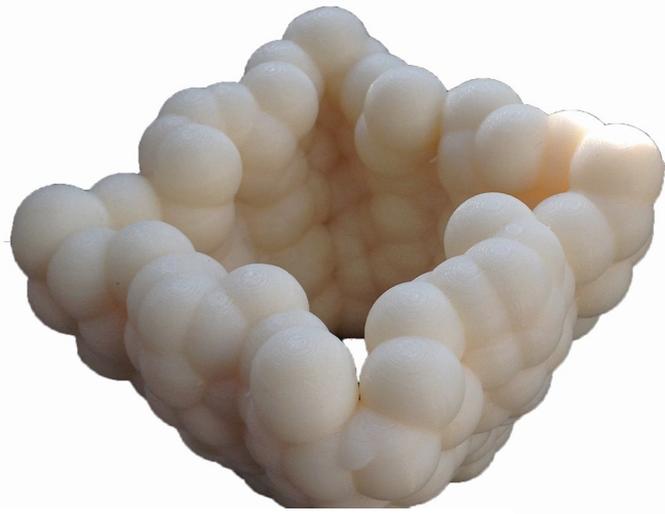
Probabilities of the Spheres on Surface. The answer of the What happens when Geometry counts are changed and Radius sizes are changed. This matrix reveals it.

4





9 Codes are baked then this model reveals in Rhino



10 Fabricating with Rapid Prototype Machine

PARAMETERIZE

6

This project is about parametric column capital design with grasshopper modelling. After modelling with grasshopper we fabricate it with laser cutting technique. Start point of this column capital is circular geometry. Firstly one big circle has been drawn in rhinoceros then multiply it to the corner points. Secondly the other circle which is smaller than the first one has been created. Trimmed them to make a closed curve. Loft it with a z plane drawn curve. Now surface came along.

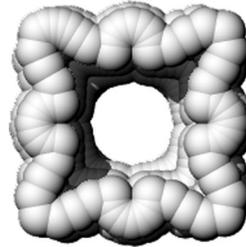
After this process grasshopper phase starts. "Surface" element is created in panel and this surface is introduced with "set one surface". Then my process is continued with "divide surface" element. This element coordinated points on my column surfaces orderly. That step becomes different from my rapid prototype modelling. If i create it with the same technique, it can not be fabricated with laser cutter. This coordinates are important for the system.

"Sdivide" has two parameter connections which are "u count" and "v count". "U count" represents vertical points. "v count" represents horizontal points. Now I can control points with sliders. My models valid counts for "u" and "v" coordinates are the same. The number is "5".

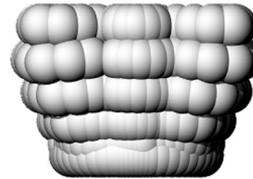
After that "Sphere" tool has been used with the "10" radius size. I assigned them on "S divide" element. Then it becomes orderly placed spheres on the surface.

When every steps are created, I baked the model. The spheres are individual elements on rhino. I used "BooleanzObject" in Rhino then Boolean them together to make the model solid on X-Y and Z plane. Now a solid model reveals.

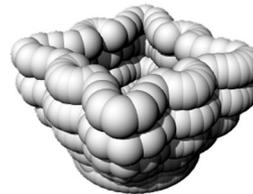
Then I used 123d make by autodesk to make a waffle structural model. Dimensions are 30x45x45 cm. I have fifteen horizontal planes and thirteen vertical planes. So I cut that planes with laser cutter machine.



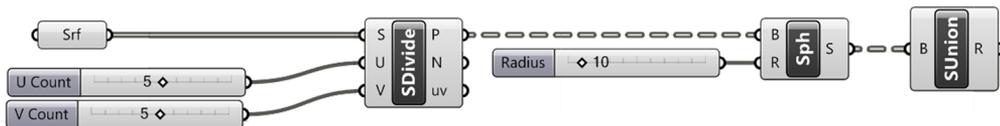
11 Top View of the Model



12 Top View of the Model

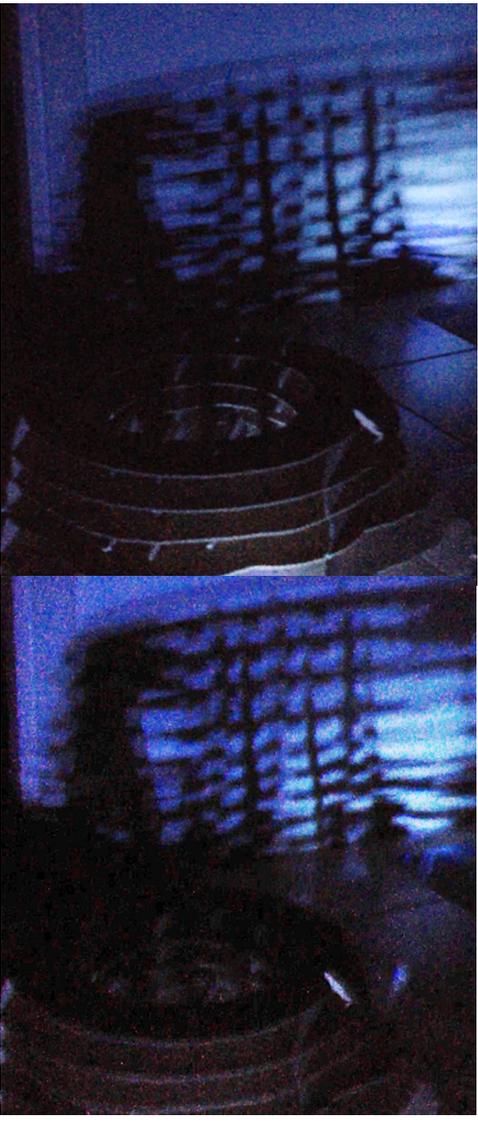


13 Top View of the Model



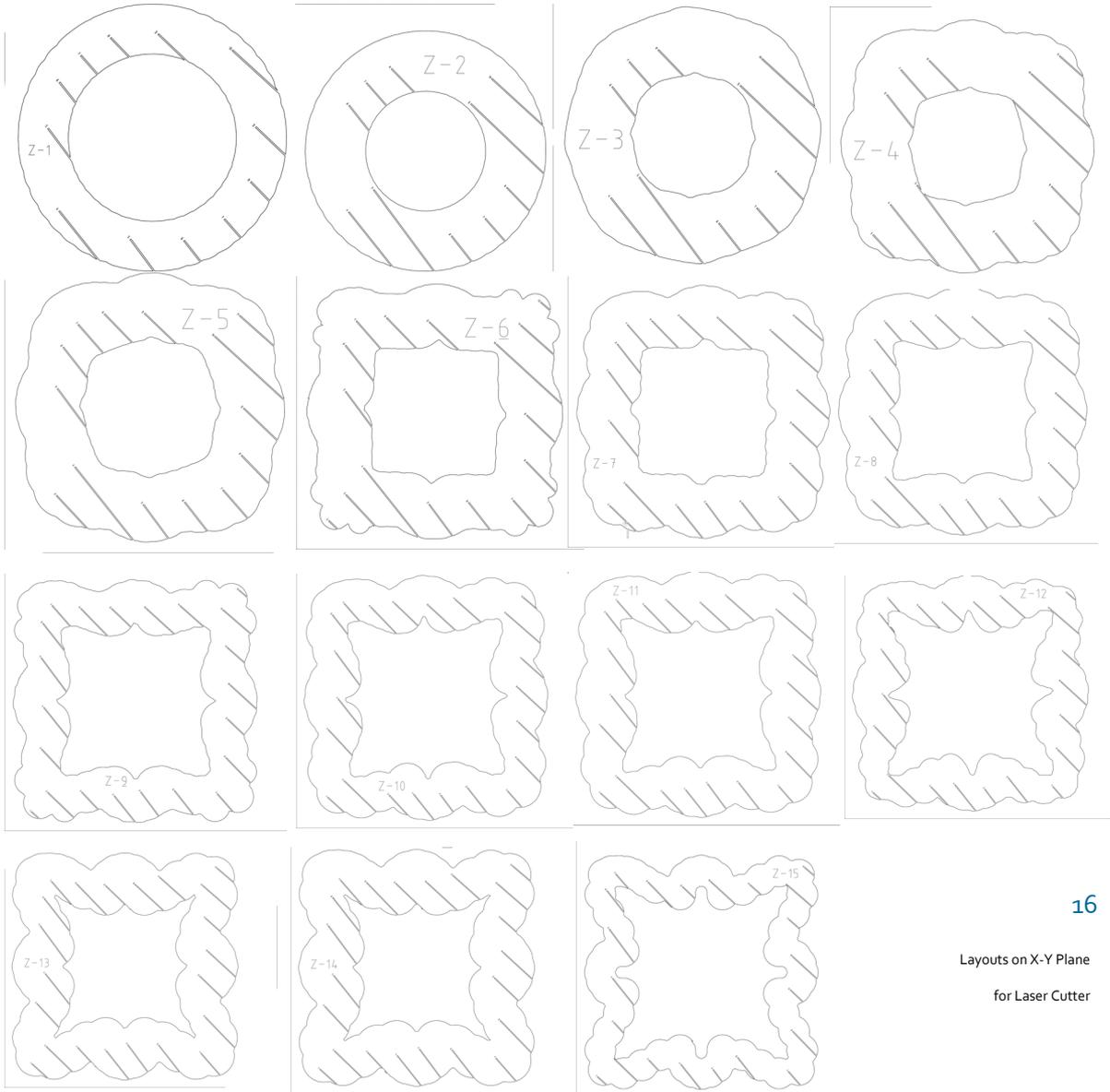
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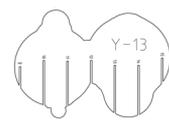
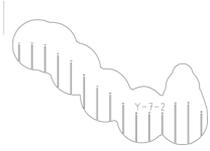
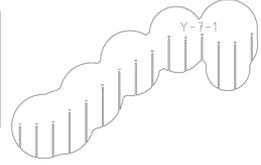
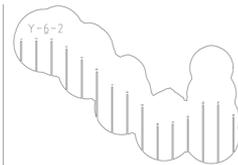
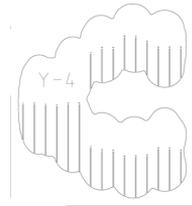
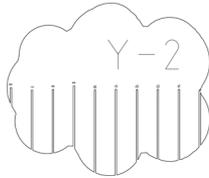
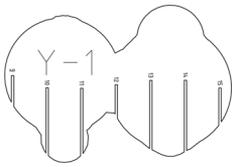
Grasshopper Codes of Column Model



15 Light Projection on Waffle Structural Fabricated Model



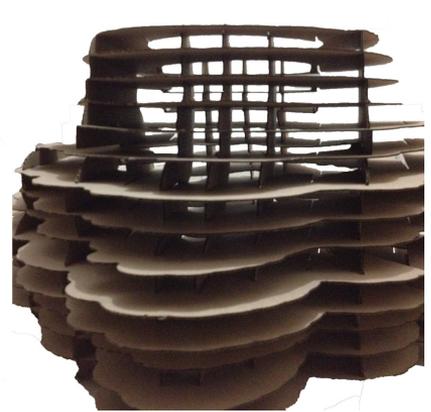




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Layouts on Z Plane for Laser Cutter

Fabricated Model with Laser Cutter 18



DIFFERENTIATE

This project is about creating a surface which is designed with parametric tools of grasshopper. First Step is drawing curves in rhino making a surface which is not linear Plane. This surface has points on axis x , y and z . For this reason when this surface is fabricated with laser cutter, it cannot stand up. So i project the surface on a linear plane surface then i create side surfaces to connect them. After these steps, i have a shape which is like a rectangular prism. Then I process same steps for all four surfaces. "divide Domain" tool for defining "subsurface" then i use "area" tool for center point. Points are created on rectangular subsurfaces. i offset the subsurfaces to guideline to my shape which will assign on subsurfaces. next step i assign "sphere" element on these center points to boolean them from my surfaces. spheres are my main structure from prototype to differentiate models. i tried to make spheres' radiuses random counts. but my subsurfaces HAVE equal dimensions, so spheres be formed as same radius on surfaces. one of my surfaces' shape is look like triangle. this surface's subsurfaces are not equal, so sphere has not random radius counts. they are equal too.

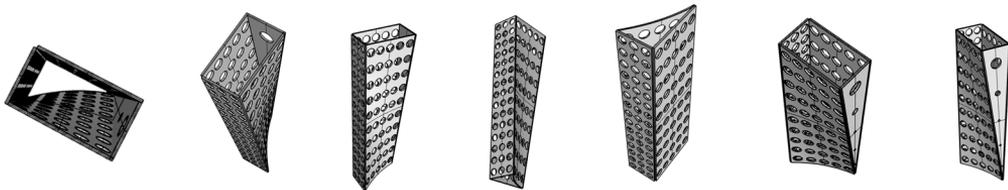
Now i made a research on grasshopper3d.com and i find solution to that problem. First of all not linear surface caused some problems like when we divide it into points they are not on the surface they stucked on air so that non-linear surface must transformed to a plane. there are some tools to create these moves.

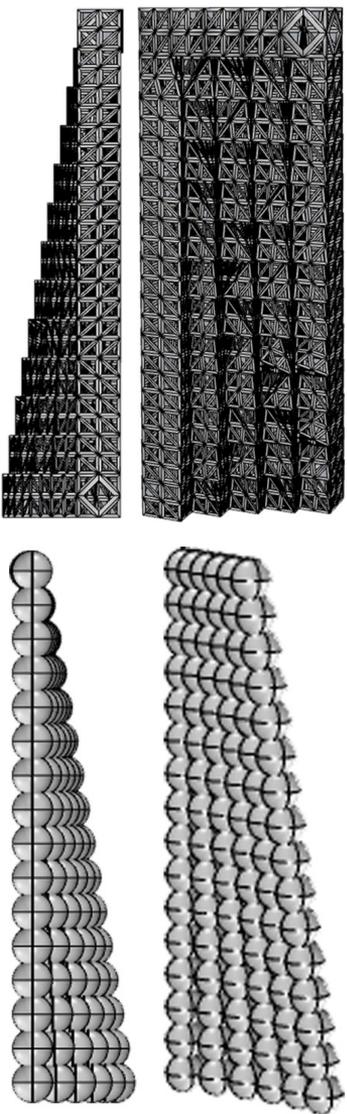
For Informing coming back to the grasshopper steps, i set the not-linear surface to "surface" tool. then "divide domain" tool with "u count" 20 and "v count" 5 has been used to dividing points. after that "sub-surface" tool has been used to make partition on this divided points. "area" tool is created to classify the center points on these subsurfaces. After all these, "surface closest point" tool is established to move divided points on non-linear surface. then "evaluate surface" has been created to complete "surface closest point tool". Although next step must be "plane" to define non-linear surface as a plane. after these patchings now I can assign

the spheres on this non linear surface. these steps created a solution to "surface as a plane" but i still need random radiuses to spheres. next step is creating "sphere" tool and connecting it with "plane" tool. for radius count connection, i created a "random" tool and "multiplication" tool which will solve the problem of random radius assign. "multiplication" has to be connected with the counts same as the surface "divide domain" u and v counts. this will defined the random radius count, so it is important. Now i have two things. first one is the surface, second one is the sphere groups with random radiuses.

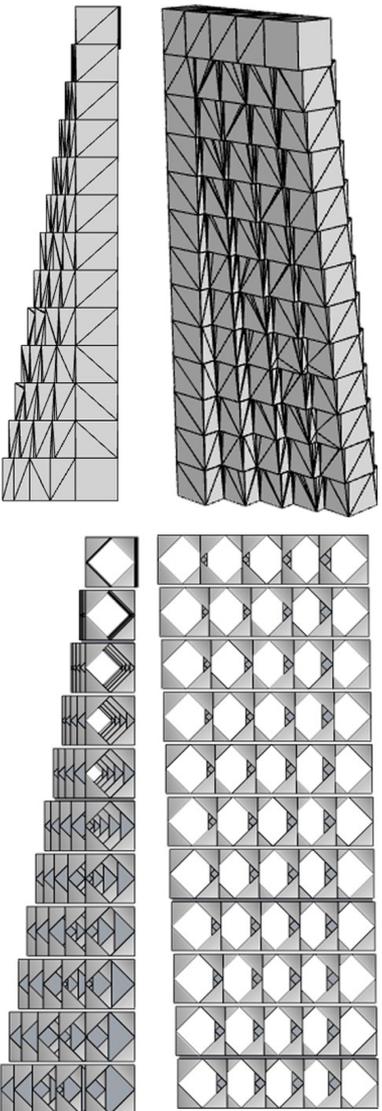
From now on i worked for "solid difference" tool. solid difference is work with two solid objects. therefore I have to "extrude" the non linear surface and then use "solid union" to make the surface solid object. spheres are the solid objects. now i created the "solid difference" tool and connect it with surface and sphere tools. then i get a surface with circular holes which are not the same radiuses. Then i use project tool to project this surface to a back surface for making my wall then i have one linear surface with the same holes of first surface.

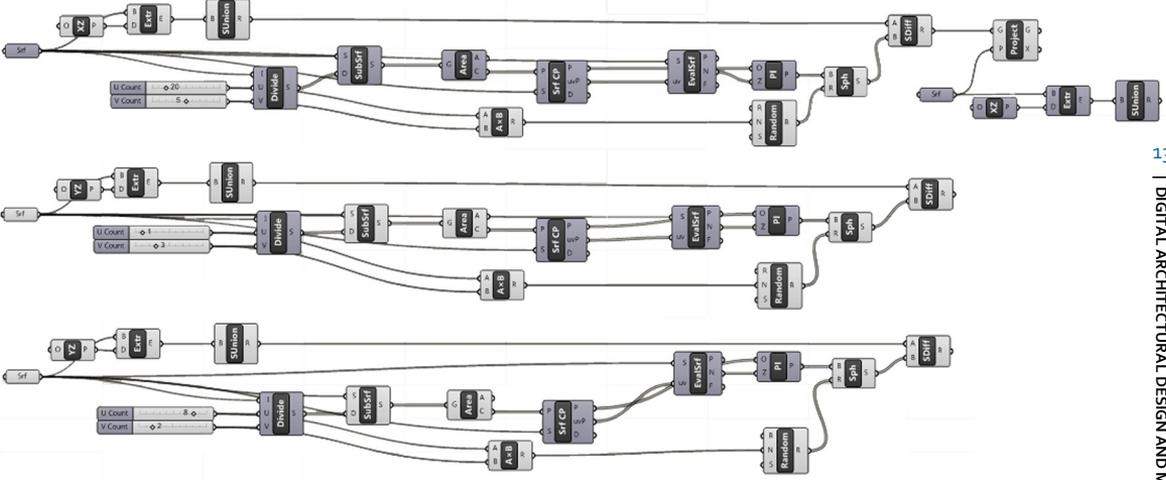
There are two surface left. I made same steps with different u and v counts on "divide domain" tool. then i baked all four surfaces. after that i make some steps on rhinoceros. i explode all surfaces and loft some circles to make support cylinders between surfaces.





20 Geometric Shape Diagram

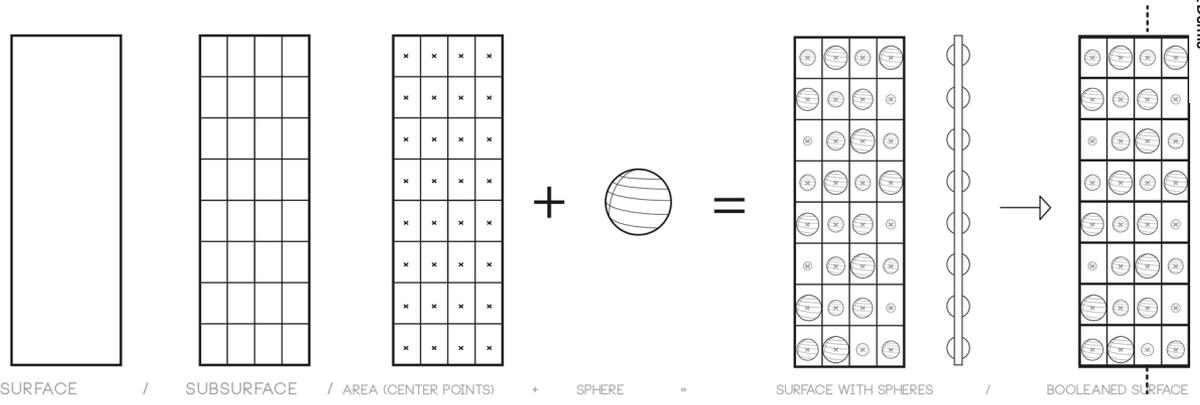


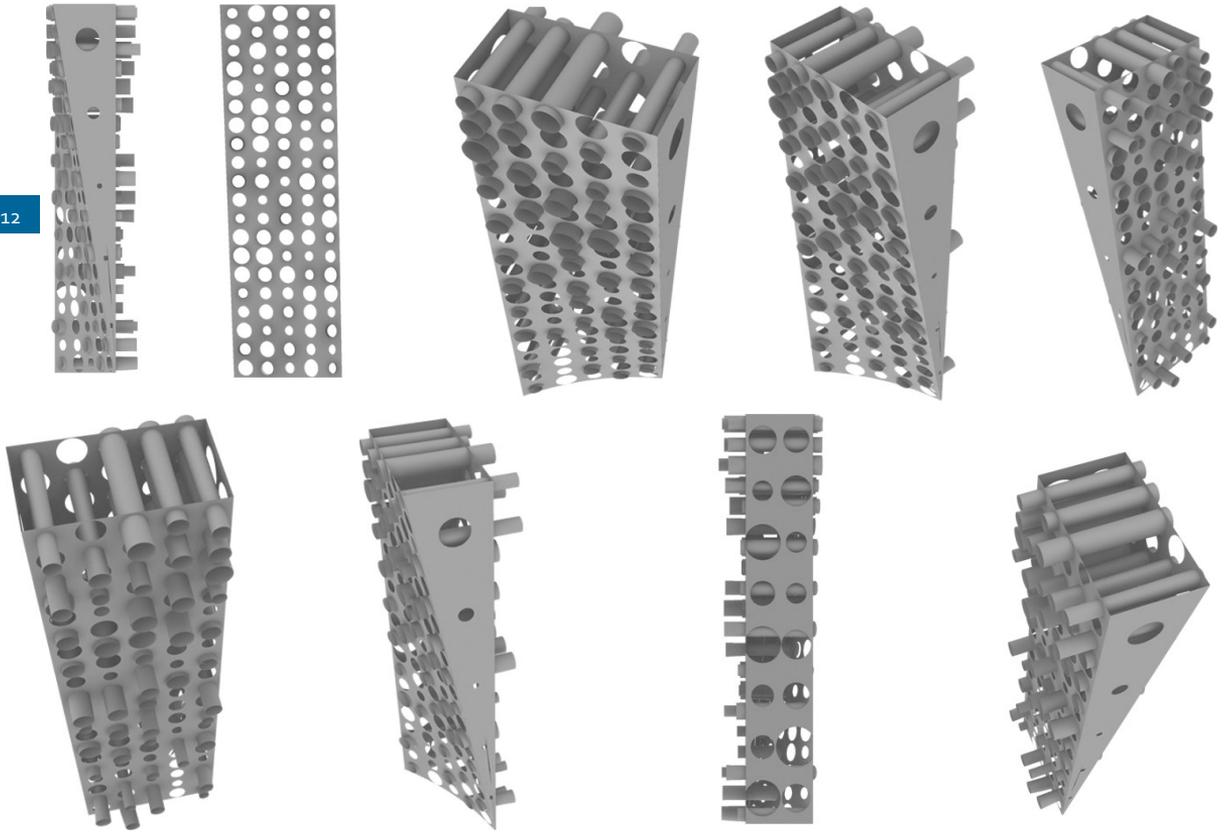


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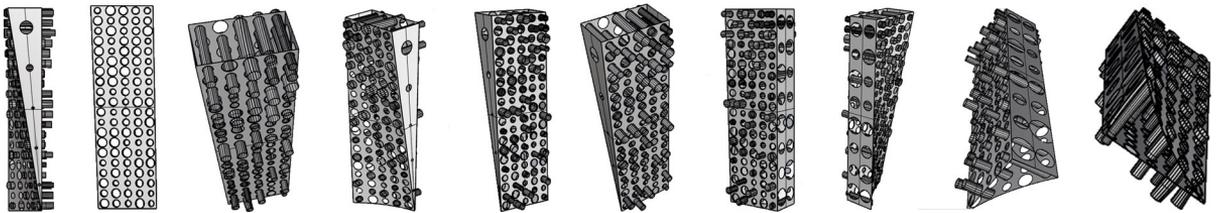
Grasshopper Codes for Fabricated Wall Model

22 .Diagram of Fabricated Wall Model. Random Sphere Radius sizes.





23 .Rendered Perspective Views of the Wall Modelling

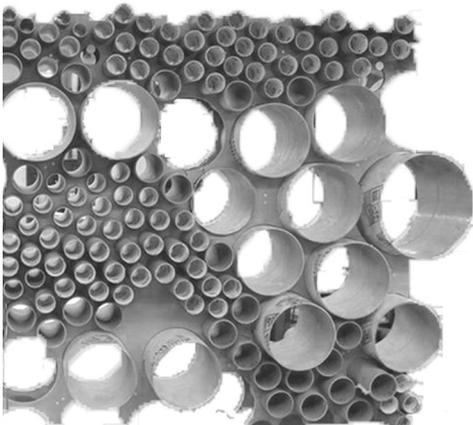


24 Shaded Perspective Views of the Wall Modelling



25 Fabricated Model Views

Recycling In Practice: Perkins + Will Finds New Life for Cardboard Tubes



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[Görseller ve Fotoğraflar]

<http://www.archdaily.com/> (Recycling In Practice: Perkins + Will Finds New Life for Cardboard Tubes) (26)

[Kaynaklar]

Url-1: www.grasshopper3d.com