CS211

Week 6 - Making a Lego-compatible model

27 March 2018

In order to have a complete AR game, we offer to 3D print a model that will serve as a tangible cylinder in the game.

To prepare your model for 3D printing, you need to fulfill the following main requirements:

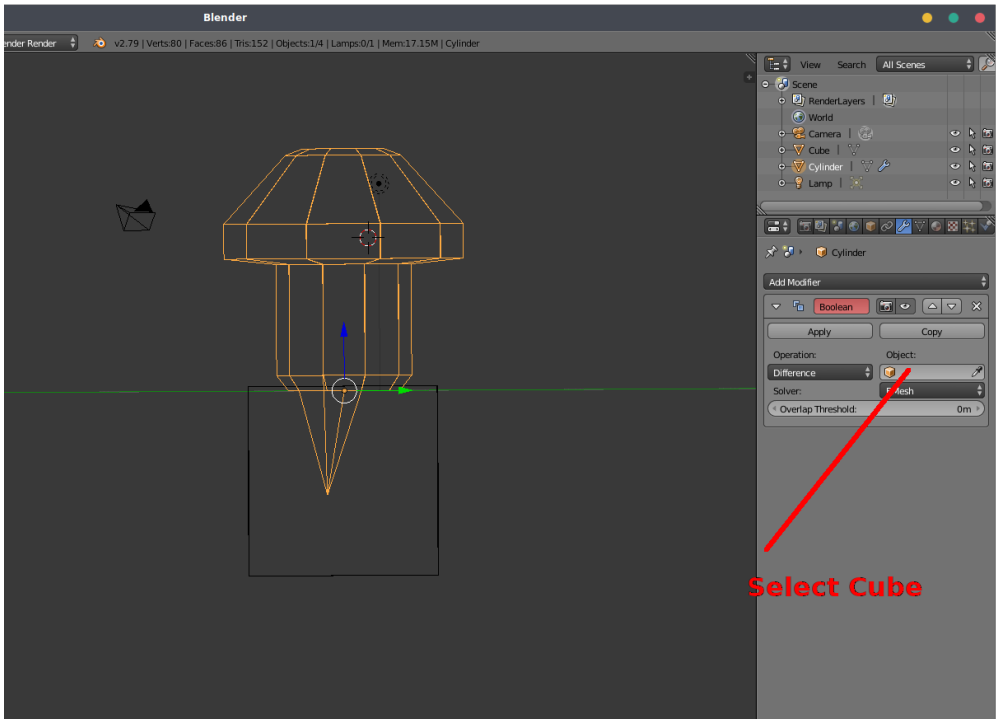
- The volume of your object should not exceed 4x4x6 cm.
- You should prepare an object that will fit on 2x2, 2x3 or 3x3 Lego knobs.
- Your object should respect the 3D printing constraints and minimize the need for manual retouch of the printed object.

To be able to fit the object on the Lego board, we need to ensure the bottom of the object is flat, and then we must **substract** the shape of the Lego knobs out of it.

Step 1 – Flatten the bottom part of the object with the Boolean modifier

Create a cube larger than your object, and place it below the object.

Select your object and add a `Boolean Modifier` to it (see the screenshot below). Set the operation to `Difference`, and the `Object` field to the cube you just created.

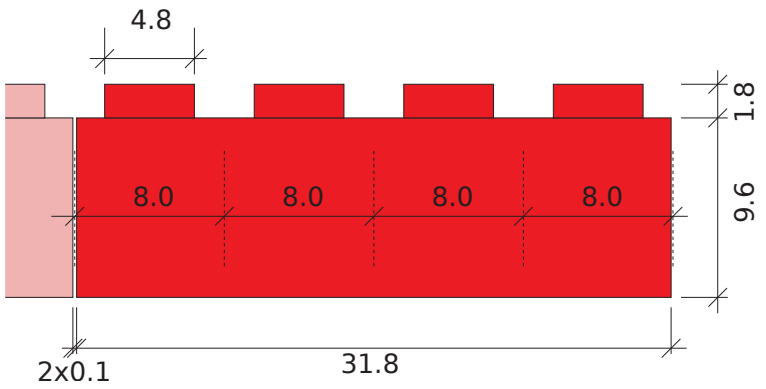


Switch to Wireframe rendering **Z**, and move the cube in order to place it in position.

Click on Apply in the modifier panel to apply the boolean operation. You can then delete the cube (select it and press X).

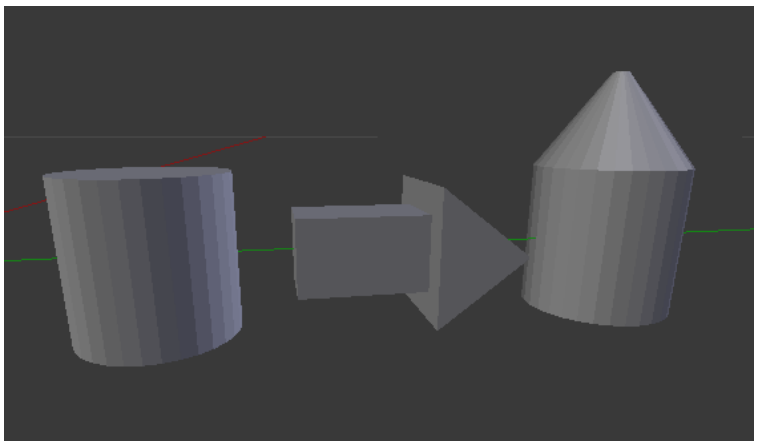
Step 2 – Model the Lego brick

> **Note**
>
> Remember to setup the units and set the real size of your model to millimeters



The diagram above provides you with the dimensions of a regular Lego brick. You do not need to model the full brick: Simply model *one* knob by scaling appropriately a cylinder (switch to Object Mode if necessary, and ⇧+ A to add a new cylinder primitive).

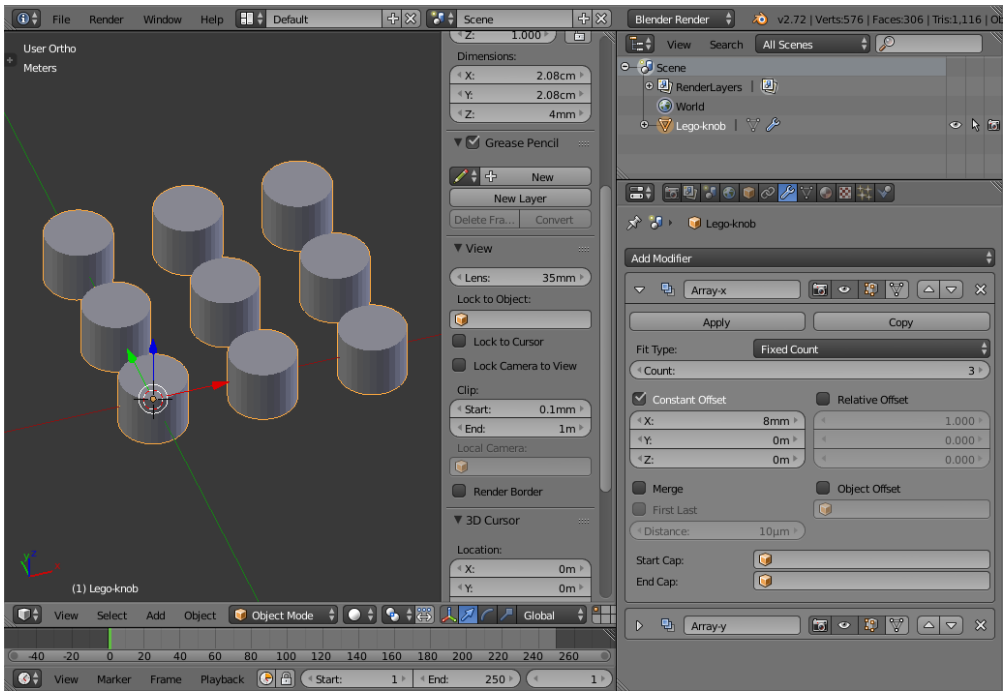
Extrude and then scale the top face to turn the knob cap into a cone:



Note

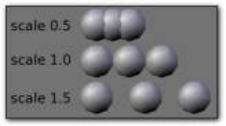
Add a margin of 0.4 millimeters to the **diameter** of the knob to ensure a nice fit on the Lego board.

Then, use 2 Array Modifiers to copy the knob over the X and Y axis and quickly recreate a full Lego board:



Note

Constant Offset adds a constant translation component having origin from the object's center.
Relative Offset adds a translation equal to the object's bounding box size along each axis, multiplied by a scaling factor, to the offset.



Step 3 – Finalize the Lego object

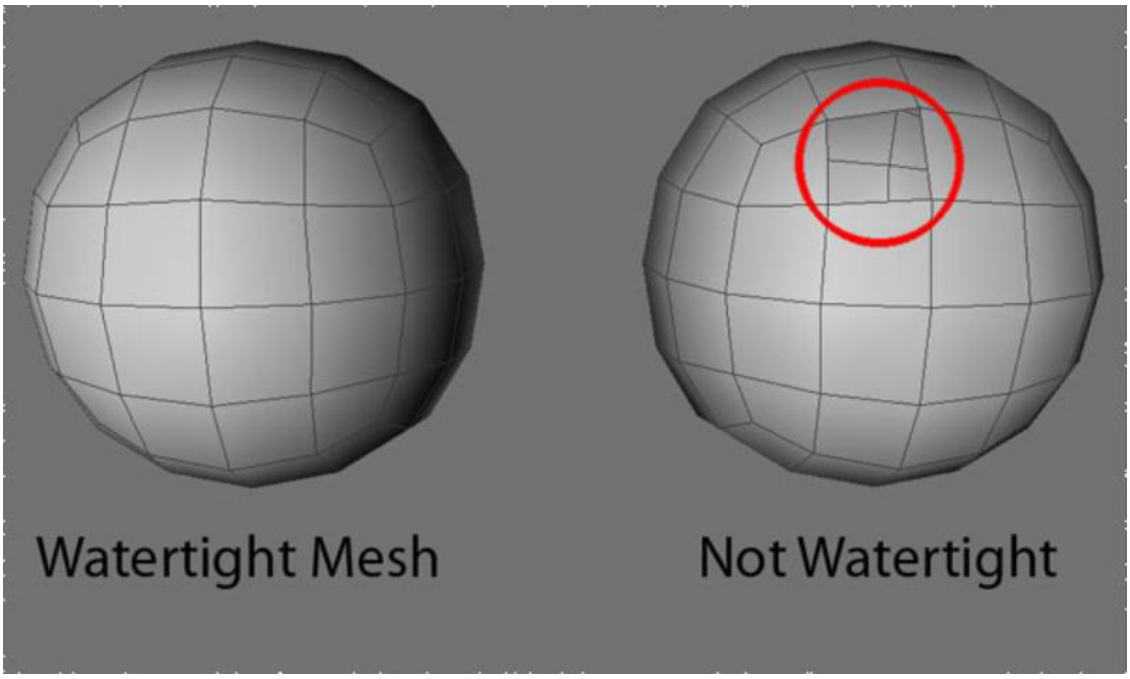
Use a boolean modifier to substract your array of Lego knobs from your object.

Designing for 3D printing

When you're designing a 3D model for a render or animation there's little need to pay any attention to the reality of the physical world. 3D Modeling and Digital Scultping for 3D Printing can be quite different. 3D Printing is a type of additive manufacturing in which a part is constructed layer by layer. They use an extrusion process called Fused Deposition Modeling (FDM) to create parts. PLA plastic in wire form is pulled into the printer head where it is heated to 230°C and extruded from a nozzle. This molten plastic is positioned on the build surface based on instructions the printer receives from a program that slices models into layers.

Making your Mesh Water-Tight

A water-tight mesh is achieved by having closed edges creating a solid volume. You may have to clean up any internal geometry that could have been left behind accidentally from booleans.

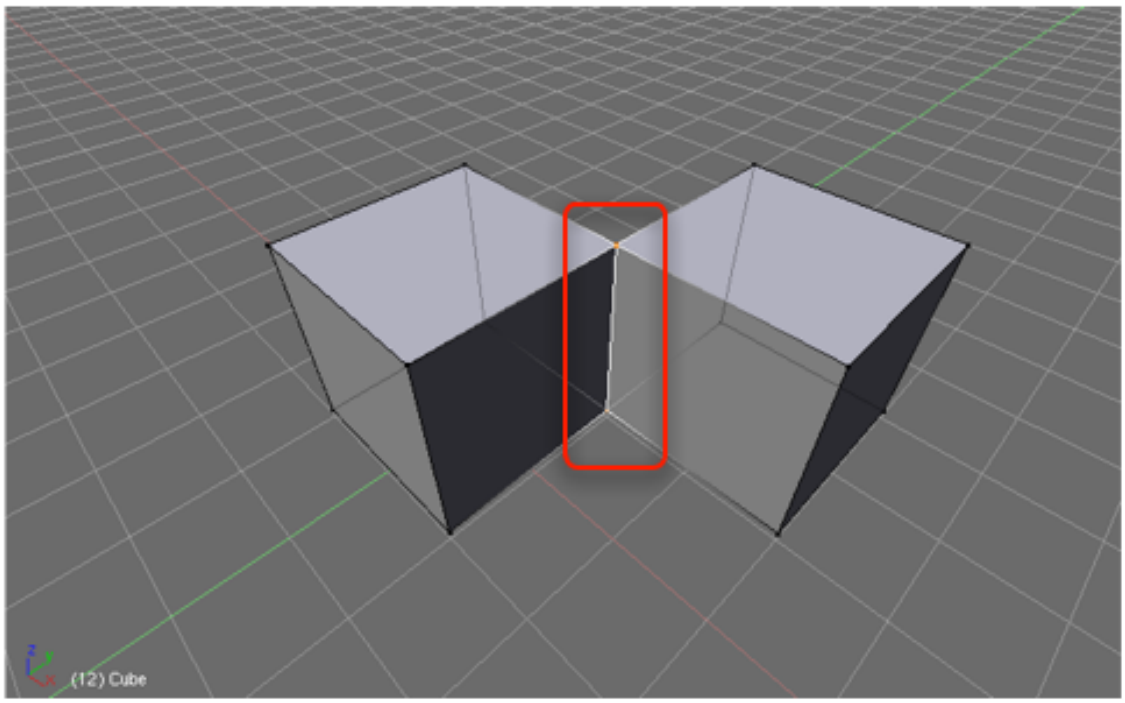


Removing Non-Manifold Geometry

Non-Manifold geometry can be defined as any edge shared by more than two faces. Non-Manifold geometry may occur when a face or edge is extruded but not moved, resulting in two identical edges directly on top of one another.

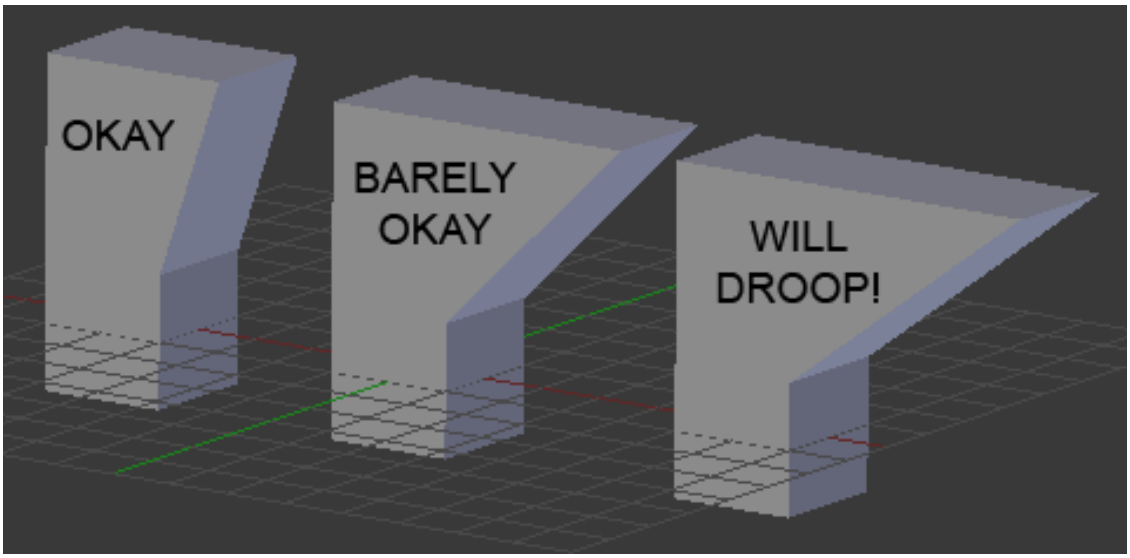
When Non-Manifold geometry is found in a model, 3D printers will have issues reading the file.

In this example, two cubes have one edge in common. Non-Manifold geometry exists because one edge is shared by four faces. Moreover remember to keep the **minimum wall thickness above 2mm**.



45 Degree Rule: Design To Avoid Using Support Material

Remember 45 degree rule, overhangs that are greater than 45 degrees will need support material or you need to use clever modeling tricks to get the model to print. Although support algorithms are improving all the time, support material can leave ugly marks on the outside of your prints. Support material can also be time consuming to remove. Design your models so that they are 3D printable without support.



Orient for the Best Resolution

Always orient your model for the best resolution possible for that particular model. Models can be sliced into pieces if necessary and then re-assembled.

Blender 3D Printing ToolBox

The Print 3D Toolbox panel is an add-on to Blender which helps in finding errors in your mesh.

Installing the Print3D toolbox

- Open up the User Preferences panel by selecting File » User Preferences.
- Select Addons from the menu at the top.
- Select the Mesh button at the lower-left corner
- Click on 3D Printing ToolBox and enable it

Checking for errors

The Print3D toolbox is located in Tool Shelf on the right-hand side of the 3D View window. From here you can check:

- Solid: This checks that the mesh has an inside/outside (is manifold), and connected faces are not flipped in opposing directions. This check is important because other tools rely on the mesh being solid.
- Intersections: The check reports faces that pass through other faces in the mesh (a self intersecting mesh). While intersecting geometry may print in some cases its normally something to avoid.

- Degenerate: This check reports zero length edges and zero area faces, against a user defined threshold. This may work in some cases but can cause problems depending on the tools used.
- Distorted Faces: This check reports non-flat faces which may print with undefined tessellation.
- Thickness: Checks the 'Wall Thickness', a physical limitation with many 3D printers is the thickness of a surfaces that can be printed.
- Sharpness: Similar to 'Wall Thickness', especially sharp areas may create areas which are too thin to print.
- Overhang: When printing without support material, there is a limit to the extent to which faces can overhang. This is a physical limitation which varies depending on the material and the printer but 45 degrees is a common default.

Note

The tool is quite strict: some errors do not prevent from printing! Just try to fix the as many errors as you can.

Export from Blender

In order to 3D print your sculpture, you will need to export your .blend file into a .STL file format. This will allow you to import your mesh into a 3D slicer tool (such as Slic3r).

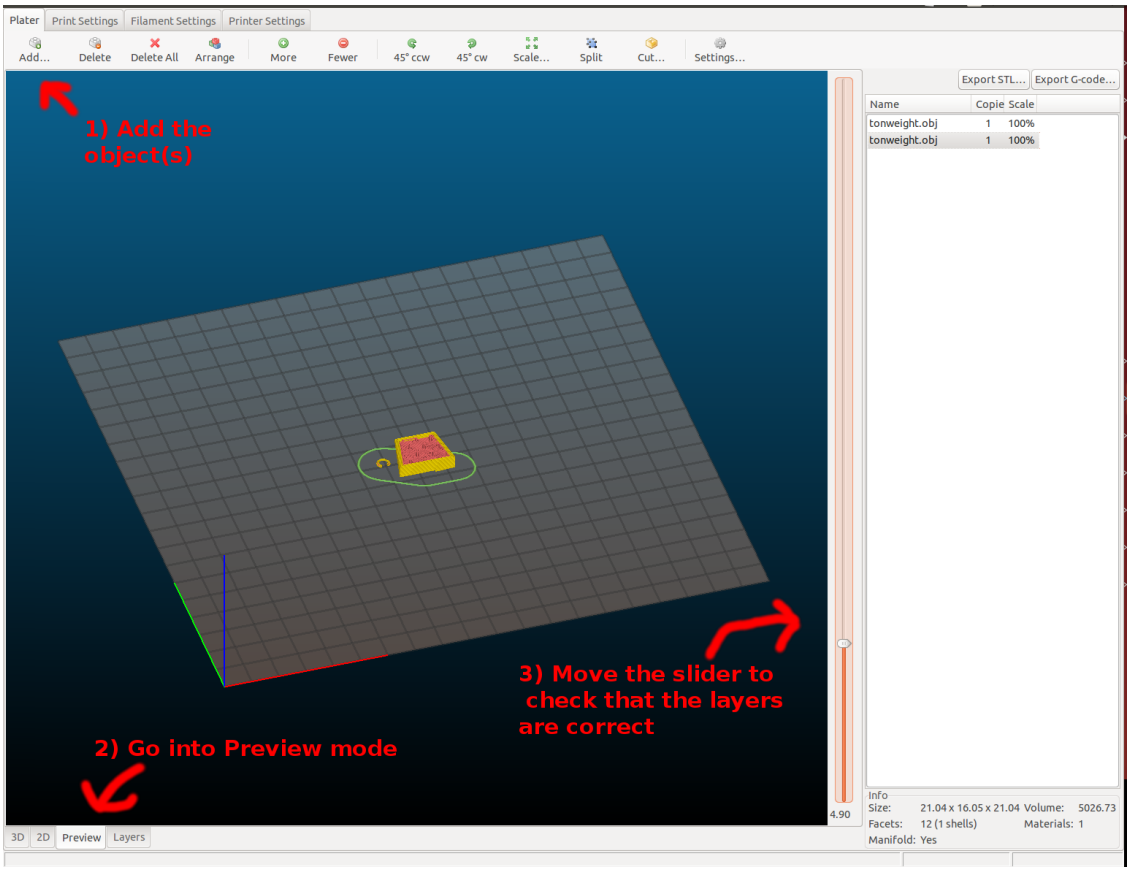
Slic3r

Slic3r is the tool you need to convert a 3D model into printing instructions for your 3D printer. It cuts the model into horizontal slices (layers), generates toolpaths to fill them and calculates the amount of material to be extruded. We are going to use Slic3r for testing if our mesh is printable.

Note

Install Slic3r from `http://slic3r.org/`

Open Slic3r and just accept the default settings from the wizard. To check that your object is printable, just do as below:



Other Resources and Credits

3D Printing Guidelines
http://makezine.com/2013/12/11/top-ten-tips-designing-models-for-3d-printing/
http://www.shapeways.com/tutorials/things-to-keep-in-mind