TinkerCAD User Manual

1. Starting with TinkerCAD - Access

Tinkercad is a very simple and intuitive 3D modelling tools available as online. It runs in your web-browser with a combination that makes it the go-to software for beginners. To start with the application, the user needs to go to: https://www.tinkercad.com/ and create an account by clicking 'Start Tinkering' blue button (Figure 1). Ane-mail and a password to create an your account shall be delivered.
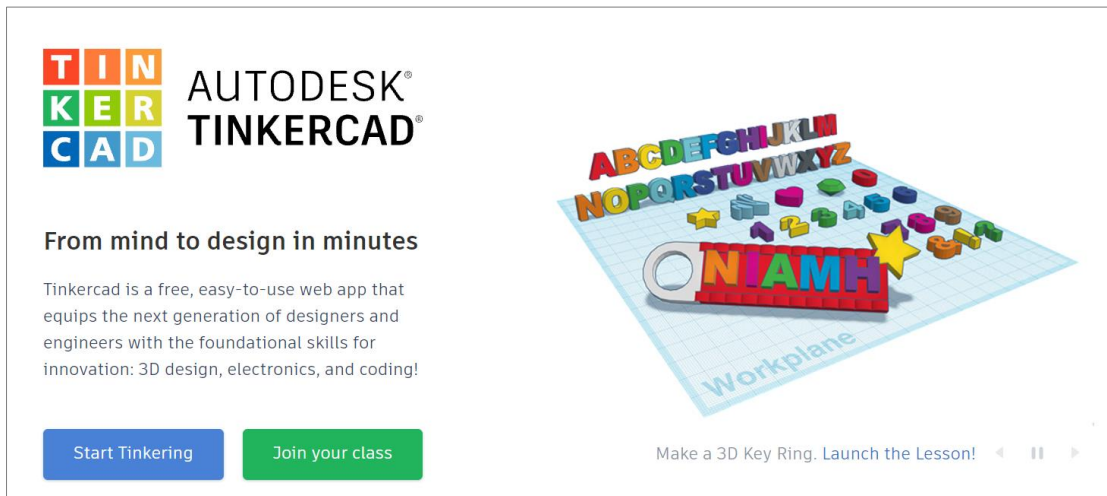

Figure 1 : Start-up screen of TinkerCAD

After several steps for registration, the user reachs the main panel of TinkerCAD. The usercan either go on with an online tutorial or start designing immediately. Skip this for now by closing it and instead go to the home page and click "Create new design".

In order to learn the basics of TinkerCAD, a simple wrench will be designed in this manual (Figure 2). The wrench is used for tightening nuts and bolts. This should be a familiar tool with a modest shape that can easily be 3D printed later on.

Figure 2 : A simple wrench for tightening nuts and bolts (Source: All3DP)

1. Setting Up the Project

In Figure 3, the user can see what a project's main interface looks like. As it would be expected, the very center is the "Workplane", upon which models are created and manipulated. The interface is also composed of the following:

A. Top toolbar, with more general tools like "Copy", "Paste", and "Delete" on the left and design operations like "Group" and "Align" on the right.

B. **Navigation tools** for basic orientation like zoom in and out. Note that these specific tools can also be used via mouse wheel.

C. **Shape Panel**, containing all building blocks for designing, including basic shapes like cubes, spheres, and text. (This can be a fun box to explore, as it has many interesting and sometimes customizable options!)
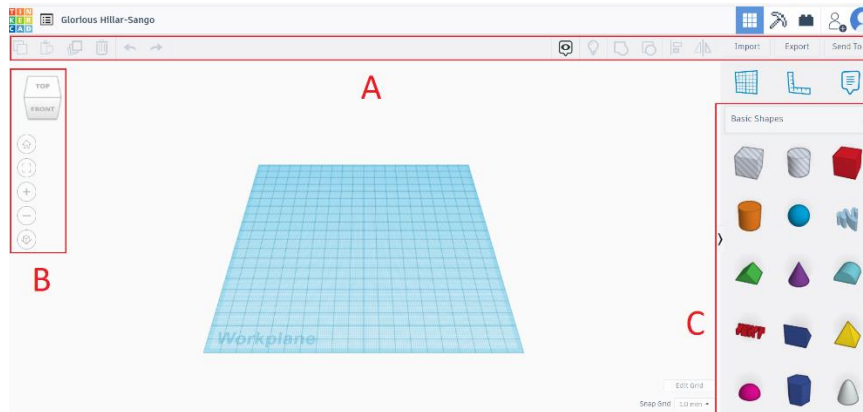

Figure 3 : Main Interface of TinkerCAD

You can zoom in and out with the mouse wheel, and move around by pressing the wheel. By right click and move, you will rotate the workplane as 3D.

2. Dividing Into Pieces

Dividing into pieces is a key principle in 3D modelling. Basically, it tells us to break a part down into smaller shapes and tackle them individually. These parts will then be assembled together to create the final object. This technique comes in especially handy when modelling complex real-world objects. For our wrench model, let's divide it into three basic units: the ring, the handle, and the jaw. (Figure4)

Figure 4 : Dividing the wrench into 3 parts (Source: All3DP)

These units will be designed in the next three steps, and after that, they'll be joined together. In the sixth and final step, we'll add some text to the wrench so that its size can be quickly identified.

3. Designing the Ring

The ring of a wrench can be approximated by a solid disk with a hollow circular area in the middle, something like a very thick washer. To create this shape, first drag a cylinder from the "Basic Shapes" panel to the workplane. Clicking on the shape presents its options under "Shape" (Figure 5). While selected, you'll notice the cylinder has five white squares, or "handles", four of which are in the shape's bottom plane and one of which is above the cylinder.
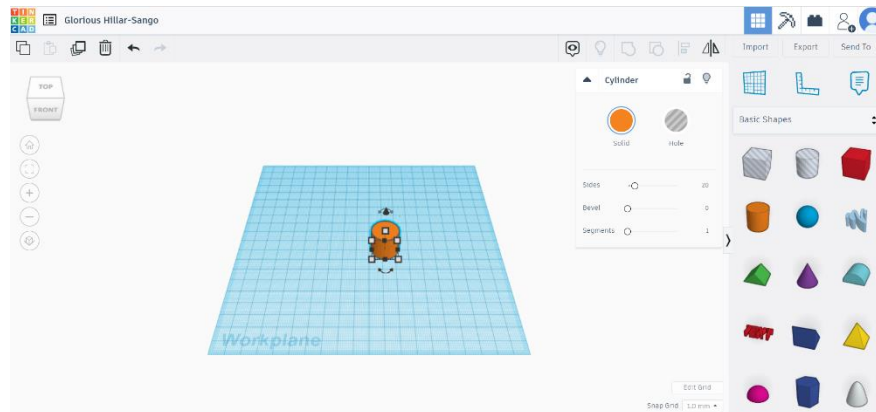
Figure 5 : Dragging shapes onto the workplane

3.1 Adjusting Dimensions

The four handles in the bottom plane can be used for adjusting the cylinder's length and width. We want our ring to have a diameter of 25 mm. Once you click on a white square handle, a small box with the current dimensions is displayed. Either drag the square handles till the cylinder has a length and width of 25 mm or enter the values by clicking on the dimension box and hitting Enter. (TinkerCADwill by default snap any dragging to whatever value appears at the bottom right, e.g. "1 mm".) The handle above the cylinder adjusts its height. We want our wrench to have a height of 4 mm, so adjust it in the same way as you did with the length and the width.

3.2 Boolean Operations and Holes

TinkerCADoperates on the principle of Boolean design, which allows the subtraction of objects to create empty areas and the addition of objects to form new shapes. (Note that these terms aren't explicitly used.) For this step, we'll use the subtraction operation to create the circular hole in our ring. First, drag another cylinder to the workplace and adjust both its length and width to 12 mm (Figure 6), which is the diameter of the hole we want. With this new cylinder selected, click on "Hole" in the upper right window. This tells TinkerCAD to treat this shape as something to be subtracted.
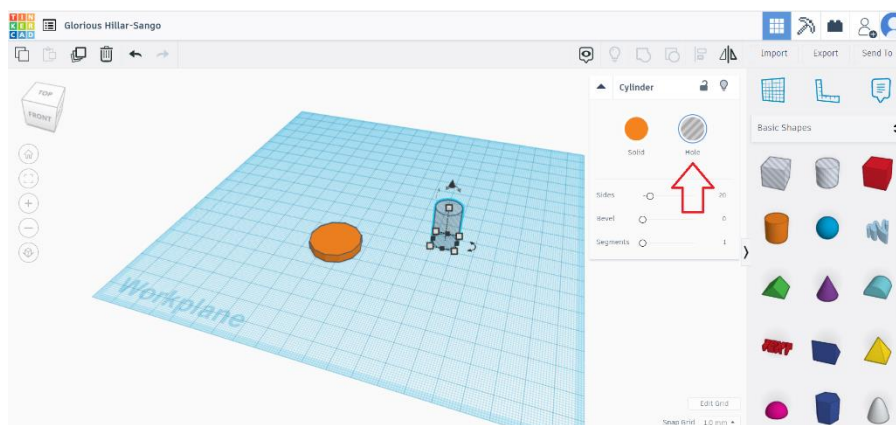

Figure 6 : Second Cylinder on workplace for subtraction

3.3 Aligning

Next, we need to position the cylinder-shaped hole in the middle of the solid disk. For this, we'll use the alignment tool in the top toolbar. First, we need to select the objects that need

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to be aligned. Left-click the solid disk and, pressing Shift, left-click the cylinder hole. If you do this correctly, both objects should be simultaneously selected and highlighted. Press the "Align" button in the top toolbar. You'll note that black round handles will show up on the workplane. Click on the bottom middle handles to align the objects along their central axes in both the X and Y directions.
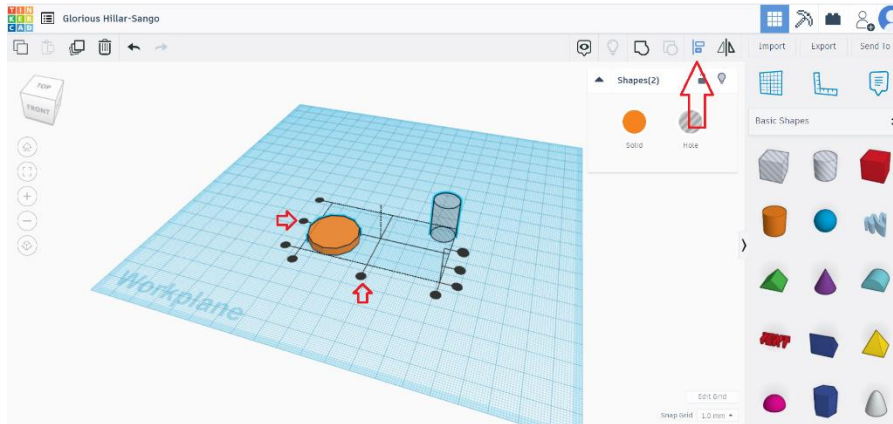

Figure 7 : Aligning cylinders in X and Y directions

3.4 Grouping

Once the two shapes are aligned, press the "Group" button in the top toolbar. You'll see that only one shape will remain. This tool essentially joins two shapes together, but in the case of one being a hole, it removes material. If all went well, you should have the finished ring ready.

4. Designing the Handle

The handle is probably the easiest part of our wrench. We basically just need a rectangle with the right dimensions. Start off by dragging and dropping a box from the basic shapes panel into the workplane. Set the length of the box to 100 mm and the width to 10 mm using either of the two procedures described earlier. Then, set the height to 4 mm so that it matches the ring.
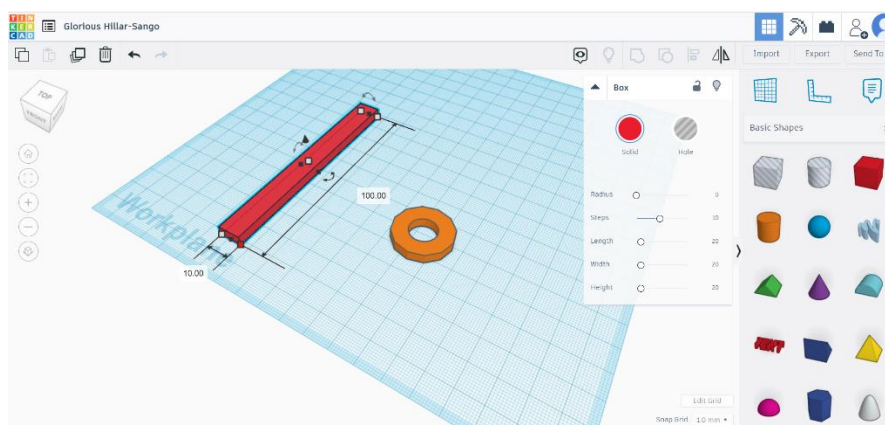

Figure 8 : Designing the handle

5. Designing the Jaw

The jaw is also a circular disk, but this time with a hexagonal hole carved out at an angle. Set your disk's diameter to 30 mm and its height to 4 mm. To create the jaw-like shape, we'll

use the "Polygon" basic shape. Dragging the polygon to the workplace, you'll see that it's by default a hexagon shape. This is exactly what we need but know that we could change the polygon's number of sides by adjusting the corresponding option in the Shape window. To position the hexagon, you can use the align tool once more to position the hexagon along one of the disk's central axes. (In other words, center it in one, but not two directions.) Select both shapes, hit the "Align" button, and click on the black round handle as before. (Figure 9)

With both shapes now properly positioned, we can set hexagon dimensions. We'll create a 15 mm wrench, but you are free to play around with it and go for whatever size you want. (Note that the rest of the wrench may not be appropriately dimensioned for values too far from 15 mm and a value of 22-23 mm is logical). Select both the hexagon hole and the disk and press the "Group" button in the top menu
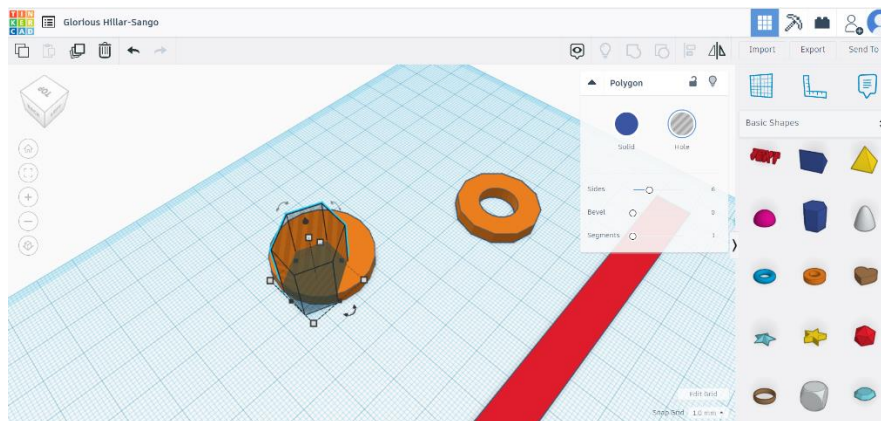

Figure 9 : Designing the jaw

6. Putting It All Together

Align both shapes just like we did in the last step – that is, just in the "long" direction. Then, hold Shift and move the ring in a straight line towards the handle. Make sure to overlap the two shapes just a little bit (using the image below as a reference), and when that's done, go ahead and group them. This should merge the handle and the ring into one continuous unit. Notice that it now has a single colour instead of two. In case the jaw is facing the wrong direction, we need to fix that by rotating it. First, select the jaw by left-clicking on it. In addition to the white handles, note the three curved handles with double arrows. Hovering your mouse over the one in the bottom (XY) plane will cause an angle value to pop up. To rotate the jaw, either click and drag the handle around the shape or enter a value manually. (Figure 10)
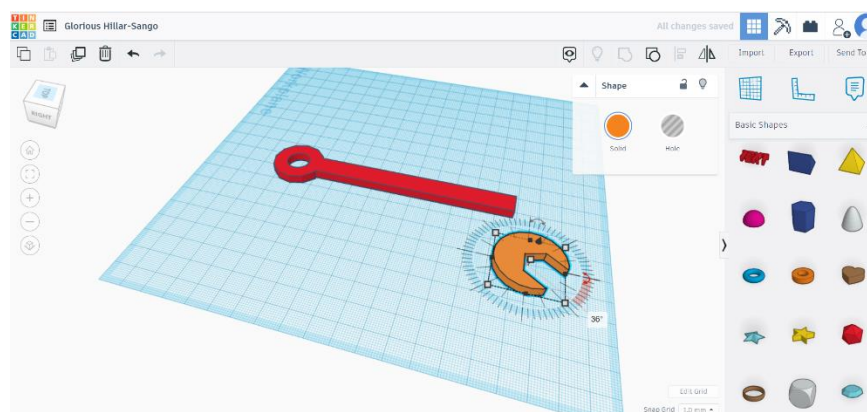

Figure 10 : Rotating the jaw

To join all, align both shapes first and then drag the jaw while holding Shift towards the wrench. Once more, position them so that they overlap a bit. Finally, group them. To change the colour, select the finalized wrench and click on the "Solid" button on the Shape window. A broad selection of colours will be displayed. Click on the colour that you want. (Figure 11)
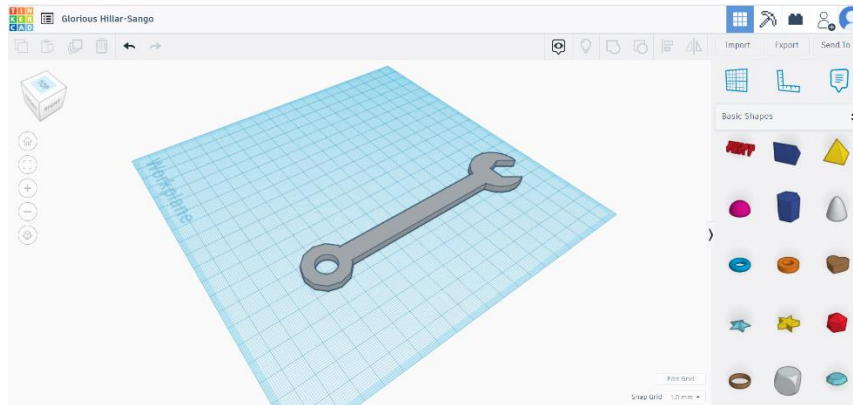

Figure 11 : Changing the colour

7. Adding Text

To add a text, look for "Text" in the basic shapes panel and drag it to the workplane. Text works like any other shape, but you can change its content. Since we want to "carve" the text out of the handle, make it a hole. In the Shape window, click on the "Text" field and replace it with "15 mm", which is the size of our wrench. The shape will automatically change. Now, it's just a matter of sizing the text and merging it with the wrench. Adjust its length and width to be 32 and 7 mm, respectively. To position the text over the handle, use once more the alignment tool. Select both shapes and hit "Align". Since we want the text to "dig" into the top of the wrench, we'll select the top round handle along the vertical dimension.

This will make both shapes' top surfaces flush. Finish placing the text by clicking on the middle round handles in the XY-plane. All that's left now is to carve the text by merging the shapes. (Figure 12)
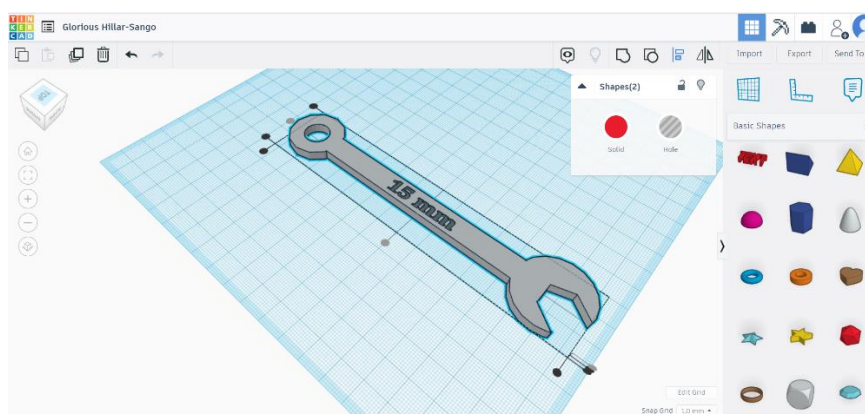

Figure 12 : Adding text as engraved

8. 3D Printing your Model

Now, the user is are ready to 3D print tis first TinkerCADpart. If you have a 3D printer, you can print the wrench, use it in your daily life, and show off your maker skills to all your friends. To get there, first click on "Export" and choose either the STL or OBJ file format. This should promptly open a download window. Make sure to choose a durable material and set a high infill percentage if you're planning on using the wrench to tighten anything. If you don't have a 3D printer, you can submit your design to a 3D printing service, who will print and ship the model to you.
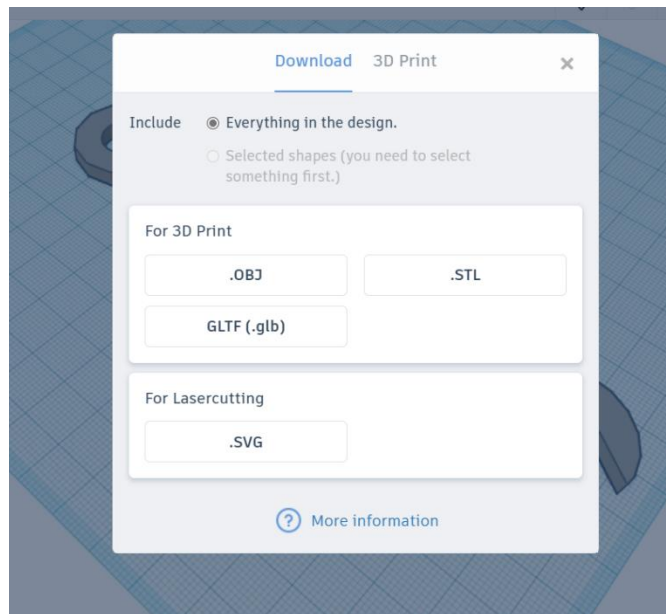

Figure 13 : Exporting the design for3D printing.

References:

1) https://www.tinkercad.com/
2) https://all3dp.com/2/tinkercad-tutorial-easy-beginners/