Basics

Camera Controls

View mode

Selection

Specifics

Reorient your model

Smooth your mesh

Make your model Solid

Create a circular hole

Prepare for printing

MeshMixer Visual Guide
BASICS

Camera Controls

The default camera control system in Meshmixer is the Fusion 360 camera system. This is a very simple system where shift+middle mouse button tumbles (rotates), the middle button pans, and the scroll wheel zooms, as shown below.

![Camera Controls Diagram]

The Hotbox

The third option for controlling the camera is to use the Hotbox. You'll find more info about the Hotbox on its manual page, but the basic usage is that by holding down the spacebar, a temporary window will appear which is dismissed when you release the spacebar. The hotbox gives access to some frequently-used controls, including:

![Hotbox Controls Diagram]
View Mode

When you position the mouse pointer over the ViewCube, regions of the cube appear highlighted. The highlighted areas of the cube (faces, corners, and edges) indicate the view that will be selected when you click your mouse.

Perspective and Orthographic Views

By default Meshmixer uses a Perspective Projection, so things in the distance get smaller. This is more natural for viewing 3D models, and tends to agree with our intuitions about 3D space. However, in many technical CAD programs, an Orthographic Projection is used instead, which does not distort angles. Under the View menu you will find an Orthographic View checkable menu-item, which will toggle between Perspective and Ortho projections. Rather than simply show you pictures explaining the difference between Perspective and Orthographic views, All pictures in this guide will be with the Orthographic view selected so if what you see on your screen seems different somehow then you likely have this setting turn off.

Selection

The main panel is shown on the left. At the top you can toggle between Brush and Lasso mode. Most of the time, you will use the Brush mode, where you will be “painting” a set of selected faces. Next we have the Size slider, which just changes the size of the Brush/Sphere. Note that you can also change the brush size by using the Hotbox [Spacebar].

- **ctrl/cmd + g - Create Facegroup.** This hotkey assigns a new facegroup to the selected triangles.
- **double-clicking** will append the entire facegroup that is under the cursor to the selection.
- **ctrl/cmd+a - Select All.** Note that this combo works outside of the Select tool, as long as you have a single mesh object selected.
- **Delete.** This will simply delete any selected faces.
Lay your model flat

The Align tool is used to position selected objects relative to either the ground plane, or to other (unselected) objects in the scene.

A basic example of the Align tool is shown below. This is the default setting, where the bottom-center of the objects bounding box is shifted to the world origin point which is 0 on the X, Y and Z Axes. The object is rendered opaque at its new position, and semi-transparent at its original position. A subset of the Transform tool's 3D widget is shown, allowing you to translate and rotate the object in the alignment plane.

If your model has a flat surface that you would like to lay flat – for instance the surface on which it was scanned – a simple way to achieve this is by selecting Surface Scribble in the Source drop-down menu and drawing a circle on the flat surface by holding the left mouse button. The result will be an average along the drawn path meaning that it will not always be perfect but usually good enough for our purposes. If the model appears upside-down after letting go of the left mouse button, check the Flip checkbox, otherwise you can select Accept to finalize its new position.
**Clean-up your model**

When you first import a scanned 3D model, it will likely include objects and surfaces captured by the scanner that are not part of the object you made. In the previous section I used the flat surface that the object was stood upon in order to make sure that my model was oriented perfectly flat, and I recommend doing the same, but having no more use for it I want to remove it.

**The Plane Cut tool**

The easiest way to do this is with the **Plane Cut tool**. The Property Panel for the this tool is quite simple. There are two main settings. The **Cut Type** determines what the cutting plane will do to your mesh. And **Fill Type**, whether or not the the opening will be filled-in after the resulting cut. Since the surface I want to remove is oriented flat against the groundplane I simply move the cutting plane down using the **cutting plane** down using the transformation widget (circled on the left). The **blue arrow** (circled on the right) indicates which side of the plane will be affected by the cut. By clicking on the arrow its direction can be reversed.

Once I am satisfied with the placement I click **Accept**. Since I left the **Fill Type** at **Remeshed Fill**, this will not only delete most of the bottom surface, it will also create a bottom for the model. This new part of the model is a different colour from the rest, denoting that it belongs to its own **Face Group** making it easy to reselect later, something which will become convenient later on. In this case there are still some parts of the surface left. These can be easily remove by going into the selection tool and double clicking the undesired parts each in turn and pressing the **Delete key**. There may still be some small parts left but those are better dealt with in a later step.
Before moving to the next step I will once again use the **Plain Cut tool**, this time I will remove a part of the top so as to give me a edge. However, unlike last time I will now select **No fill** in the **Fill Type** drop-down menu, this way the top will be left open. As you may have noticed unlike the outside of the model, the inside is not a solid colour. This signifies that we are seeing the **backside** of an infinitely thin surface, meaning that the model has no **volume**. An other way of saying this is that the mesh is **non-manifold** and because of that it cannot be 3D-printed. In later steps we will amend this but for now I bring it up mainly to highlight a problem with the model. As you can see in the pictures below not all of the inside is described, this is because the scanner managed to capture the transparent plastic mug. If I had more foresight when scanning the object, I could have covered the top of the model to avoid this, it is however easily amended. What I do is simply select the **Plane Cut tool** to remove part of the top edge, making sure to select **No Fill** as the **Fill Type**.

Having made the cut the undesired part of the inside is no longer connected to the main model, and I can then for simply activate the **Select tool** allowing me to **double click** and delete this part, much like I did previously with the leftovers from the surface. Even though my model looks pretty good at this point there is one final tool that is good use once you have clean-up the more obvious parts of your model.
The Inspector Tool

The Inspector Tool is used to repair a few common problems. It is focused on errors that occur in 3D scans, rather than all issues that might need to be resolved to create a model suitable for 3D-printing. The example below shows an imported scan. Running Inspector highlights all the issues, with color-coded stick/ball indicators. When using the Inspector, you can left-click on the balls to attempt to repair the indicated problem, alternatively, by clicking the Auto Repair All button Inspector will attempt to repair each detected issue sequentially. This can take some time, and, in this case, it would not be desirable as demonstrated in the image on the next page. Not understanding what I have in mind, Inspector will simply attempt to fill any holes it perceives defeating the entire point of the Pencil holder example.
Below is a better demonstration of **The Inspector tools** use case. Unlike the Pencil holder the Lampshade example is not supposed to have any holes on its sides. Zooming-in it becomes apparent that there are some tinny holes easily missed. Clicking on the associated **blue sphere**, this can be automatically filled-in. In this case before filling any holes, I changed **Hole Fill Mode** to **Smooth Fill** for a... Smother result.

![Inspector tool](image)

Going back to the Pencil holder, **The Inspector tool** is still useful to find and remove any residual parts that remain separate from the main model. Such parts will be associated with **pink spheres** and can be removed by clicking them each in turn. Having done that I click **Accept** and you should have a reasonably clean model!

**The Inspector tool** is very useful and as demonstrated previously it can be used to close large holes as well, even though it was not desirable in this case. Even after having done this initial cleaning-up I will use it frequently to make sure that no problems crop up as I work with my models.

This part mainly used the Pencil holder as an example and it was mainly chosen because it involved the most work. No additional tools were used cleaning up the other two models and in-fact neither required much cleaning-up at all.
Smooth your model

Smoothing your model is largely going to depend on what kind of result you have in mind. If you are making something akin to the pen holder then you will likely want to make sure that the mesh is as cylindrical as possible and that the cut-out pattern along the sides does not have overly jagged edges. You will also want to make sure that the structure of the model is as uniform as possible (more on that later). Conversely, if you are making something akin to the lamp-shade and the scanned object was made out of a textured material like fabric then you will likely want to preserve the roughness and not smooth your model at all. If you are making something ergonomic like the tablet grip which unlike the other two examples is already a manifold mesh, then chances are that it was already smoothed in the scanning software, and if it was not then you will likely only need to do some very slight smoothing.

Regardless, there are numerous methods for smoothing your model, but the two most common methods involve either using the Sculpt tool to smooth or deform smaller parts of your model with different Brushes, or by selecting segment of the model by way of the Select tool and using the automated Smooth tool. The Sculpt tools is one of Meshmixers most flexible tools and much like graphical editing software such as Photoshop it revolves around so called Brushes. I will not go into this tool as I did not use it for any of these examples, but all the same I suggest that you experiment, especially if you

The Smooth Tool

The property panel for Smooth is shown below. There are not many controls, and in most cases, you will find that you only need one. The actual Smoothing slider is not one you will likely ever need to change. Instead Smoothing Scale, is the parameter you are most likely to interact with. Similarly, Constraint Rings only has a few specific uses.

To reach the Smooth tool you need to have activated the Select tool, as well as having selected a part of your model. Note that by double clicking the grey part of the model I have selected everything except for the bottom which belongs to its own Face Group (the green part of the model) of my model. My reason for not selecting the entire model is because I already know the bottom to be perfectly flat having previously used the Plane Cut tool. Were I to select the entire model, the edge at the bottom would have been smoothed as well and since we will be 3D-printing our model we want a perfectly flat bottom.
The Remesh tool

For this step we will be using the Remesh tool. This tool is mainly relevant for the pen holder example in preparing it for the upcoming step, going over how to give you infinitely thin mesh some actual volume. The Remesh tool is quite complex but surmise to say that for our purposes it will make the structure of the model more regular and thus easier for Meshmixer to work with. By pressing \textit{W} the outline of the model's polygon structure will become visible, highlighting what the Remesh tool actually does.

The pictures to the right show the ‘before’ and ‘after’ using the Remesh tool on the pen holder and as you can see, before using the tool the model is nowhere near as neatly structured, especially around the intentional holes. It is mainly because we want to keep these cut-out that the Remesh tool is so important, as I will later also demonstrate.

Mush like the Smooth tool, the Remesh tool is reached by activating the selection tool and selecting a part of the mesh.

This time I will simply press \texttt{ctrl/cmd+a} to select the entire model before activating the Remesh tool. See the image on the next page for the settings I used, these should work well for most objects with cut-out patterns similar to the pen holder. Note that each time you change something the tool will start to calculate, and it can be easier to simply write the values you want manually.
Make your model ‘Watertight’

On a few occasions I have mentioned that in order for a 3D-model to be suitable for 3D-printing, it has to be what is called Solid or Watertight. This refers to an aspect inherent to polygon modeling which is that a polygon can only have one side. Looking back in this guide you have no doubt noticed that some parts of the models are grey where as others are a white-pink zebra pattern. This pattern is the backside of the polygon structure and whilst Meshmixer may render it visible for our convenience, in actuality we should be seeing strait through the model from that side. The pictures to the right of the lampshade attempt to demonstrate this. The first one is as describe a non-manifold mesh and thus cannot be 3D-printed. The second one however has the bottom covered and with no smaller holes in the model it is now a watertight solid. This was achieved simply by using the Plane Cut tool to remove the scanned surface with the settings at default where as I changed Fill Type to None for the uppermost one. Often this will be all the thought you need to put into this but since I want this to be a lamp shade the final model cannot be covered like this. What I want is something akin to the third picture. As you can see the bottom is now open but unlike the first picture the inside now has a solid colour denoting that the model now has some actual thickness.

Change the size of you model

Before you make your model watertight however you need to decide how large you would like your 3D-printed model to be. If you want the printed object to be the same size as the physical object you scanned, then you do not have to change anything but if you do want to change its size then you can simply press T to bring up the Transform tool. From here you can move, rotate and scale your model much in the same way as the Plane Cut tool using the transformation widget, but you can also type in precise values which is what I recommend you do if you want to change the scale of your model. You can change the size on all three axis separately or uniformly by ticking the Uniform Scaling checkbox. I changed the Y axis to 94mm to match the height of the lampshade I want to replace. Once happy click Accept.
The **Offset tool**

The first method I will demonstrate is the **Offset tool**, this method is suitable for objects like the Lampshade, that you want to be open at the top or bottom, but unlike the pen holder does not have a cut-out pattern on the sides. As should be familiar at this point the **Offset tool** is reached via the selection tool and requires that you have your entire model select (ctrl/cmd + a).

The settings for the **Offset tool** are fairly straight forward with the most central options being **Distance**, **Accuracy** and the **Connected** checkbox. Since this can be a fairly slow tool I suggest the first thing you change be **Accuracy**, this value determines how closely the offset wall will match the original and lowering it will make changing values a bit less time consuming. Having done that the star of the show is the **Distance** value. As you may have figured out this value determines the distance between the original and the offset wall, or with other words, the thickness that you would like the walls of your 3D-printed model to be.

The Offset tool

<table>
<thead>
<tr>
<th>Offset</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Distance</td>
<td>-1.6 mm</td>
</tr>
<tr>
<td>Accuracy</td>
<td>0</td>
</tr>
<tr>
<td>Connected</td>
<td>✔</td>
</tr>
<tr>
<td>Preserve Boundary</td>
<td></td>
</tr>
<tr>
<td>Preserve Groups</td>
<td></td>
</tr>
</tbody>
</table>

At this point it becomes relevant to know a little bit about the 3D-printer you will be using, more specifically, the **diameter of its nozzle**. This determines how thick the lines it print will be and ideally the thickness of your model should be an even multiple of this diameter. If you do not know what the diameter is on the nozzle and if you cannot ask someone then assume that it is **0.4mm** as this is a very common nozzle diameter. Knowing how thick the lines will be the next thing to know is that the thicker the walls the easier your model will be to print, and the sturdier the final result will be. Knowing that my lampshade will be printed with a 0.4mm nozzle I want the walls on my model to be **1.6mm** thick, which ends up being 4 lines if the nozzle is 0.4mm or 2 lines if it were to be 0.8mm. One final note about the **Distance** value is that a positive value will offset the new wall outwards whereas a negative value will offset it inwards. By offsetting it inwards the size and surface details remains unchanged and since I want to capture the texture and detail of the scanned fabric, I set the **Distance** to **-1.6mm**. Finally check the **Connected** checkbox to bridge the gap in-between the original and the offset wall. One final thing I will do is to use the **Plane Cut tool** to make sure that the part where the inner and other walls are now connected is perfectly flat.
The Extrude tool

Another simple tool for giving your model volume is the Extrude tool. In practice the Extrude tool works almost exactly the same as the Offset tool but the way that it works is much simpler than the Offset tool. Whereas the Offset tool can work with very complex polygon structures such as scanned fabric, the Extrude tool needs a more regular polygon structure to work well. This is why the Remesh tool was so important for the pen holder in the previous section. If I skip the Remesh tool then I get a bad result from using the Extrude tool as can be seen in the picture below. Using the Extrude tool on a model that has been properly smoothed however and we get a much better result as can be seen on the next page.

As mentioned the Extrude tool is fairly similar to the Offset tool, only simpler. The settings to pay attention to is the Offset value and the Direction dropdown menu. Offset corresponds to Distance and same as previously we now need to consider the thickness of the model in relation to the nozzle diameter on the 3D-printer. It is important to also consider that the cut-out pattern on the pencil holder will make it more challenging to 3D print, and in order to work around this the thickness of the walls will have to be quite a bit thicker than the lampshades. I have found 3.2mm or 8 walls with a 0.4mm nozzle to be a good thickness. As with the lampshade I want the walls to be offset inwards so I give it a negative value.

At this point things may not look as you would expect but that is likely because you have not changed the Direction to Normal. Having done so you should have something akin to the picture below. Click Accept once you are done. At this point the cut-outs will appear quite sharp and to amend this I select the entire model (ctrl/cmd + a) and use the Smooth tool. I set the value for Smoothing Scale to 1.5 as I do not want to make the walls too thin in the process. At this point all that is left for the pen holder is to prepare it for printing.
Create a circular hole

Up until this point we have only focused on making a scanned model printable, and in the case of the Pencil holder, that is all there really is to it. For objects such as the light shade and the Tablet grip however we have to actually make some changes to the models. Starting with the Tablet Grip, at this point all that I have done is reorient it and remove some excess parts, same as with the other two models. The only other thing I will do with it before preparing it for print is to cut out a rectangular shape so that it will fit my particular tablet. Before doing so I will press T to bring up the Transform tool and make some fine adjustments just to make sure that the grip is lined up straight. The red line is added to highlight which part should be lined up against the grid.

Meshmix Menu

Next you will want to bring a new object into the work area. This object will be used to create a gap in the model where in this example the tablet will fit. Since my tablet is more or less just a rectangular box all I need is a cube that I can change into the right size. Thankfully, Meshmixer has a built-in library of simple shapes which you can find under the Meshmix menu. These can be included simply by dragging and dropping the shape you want onto the work area (make sure to not drop it onto your actual model).

By doing so you should see something like this. Note that once you bring a new object onto the work area the Transform tool will be automatically selected. Since it is already selected I will change the size of the cube on the Z-axis to 7mm, making sure that the Uniform Scaling checkbox is not filled-in before doing so. My tablet is only about 6.5mm thick but adding about 0.5mm should result in a good fit. I then change to the front-view by clicking the corresponding face on the ViewCube and from this perspective I move the box, lining it up the same way I would like the tablet to fit into the 3D-printed model. I also make sure that the board that I modelled my grip to is completely encompassed by the box, after which I click Accept.

As you likely noticed, when adding the box onto the workspace, the original model turned a darker shade of grey whereas the cube became the same lighter shade that we usually see. This signifies that the two models are separate entities and that currently it is the newly added cube that is selected. By clicking on a darker shaded model, it will become the currently selected one. This is important to know as any tool you activate will only affect the currently selected model.
The Boolean Divide Tool

The next thing I will do is to actually use the created box to remove the part of the model where my tablet will fit. I will do so using the **Boolean Divide tool**. This tool is fairly complex, and I will not go into much detail but along side **Boolean Union** and **Boolean Intersection** these tools are used to combine two models into one, or in this case, to remove the part of one model being overlapped by another. To access these tools first click on the main model and then hold down shift and click on the model that should be used to tell the tool which parts to remove from the main model (this will turn them both the same shade of grey). Doing so should bring up the menu in the picture, from there select **Boolean Difference**. If you selected the models in the correct order, then you should see something similar to this. Note that when you activate this tool what you see is not always what you will get once you press the **Accept** button. The settings I used can be seen below and if you are doing something similar then I recommend trying the same. If not, then you may have to experiment to get the desired result. At this point the grip is more or less done and all that remains is to prepare it. The final part will cover how I made my scanned fabric pyramid to fit as a lamp shade. This process is only one way to achieve the same thing and with this in mind think of this more as a reference and less as a strict set of instructions for printing.

Centering the model

The first thing I do is to make sure that the model is centred on the workspace. I do this by selecting the Top view and activating the Selection tool. I then click what I deem to be the centre, giving me a circular selection, I then press **ctrl/cmd + g** to create a **Face Group** out of my selection.

Next I select the **Create Pivot tool** which allows you to create reference points on your model. This is useful for when you need objects to line up, in my case I need to be able to create a cylindrical hole through the centre of the model. By changing **Placement mode** to **Snap To Group Centre** and **Coordinate Frame** to **World Frame** I can simply click the newly create **Face Group** to create a pivot point at its centre. Before clicking **Done** I have to click **Drop Pivot**, otherwise my pivot would not be saved.
With my new pivot point I can now select **The Align tool** and change the Source to Pivot, after which I can simply click the pivot to align my model to the centre of the workspace.

With my model centered I will now create a cylinder and change its diameter to 42mm, similarly to how I created and resized the box in the previous section. This is going to be the flat top of my lampshade and so 42mm is the diameter of the part of the lamp socket it is supposed to line-up with. By ticking the Enable Snapping checkbox in the transform tool it becomes easy to moving the cylinder to the centre of the workspace as the centre will always be 0. Before closing the Transform tool I will make sure that the cylinder is low enough to poke through my entire model, this is important since there has to be a flat area on the inside for the lamp shade to be screwed in place properly.

Next I want to combine the two shapes and same as last section I will first select the main model by clicking on it and hold down shift before selecting the cylinder as well. This time I will use the **Boolean Union tool** which has the same settings as **Boolean Difference** which we used last time. The only thing I will change is to **uncheck** the **Auto Reduce Result** before clicking **Accept**. Both models now combined I use the **Plane Cut tool** to remove some of the cylinder sticking out of the top.
The last step is to cut a whole through the centre so that the lampshade can actually be screwed into place. I do this in the same way as I did the previous cylinder, this one should only be 30mm in diameter however. I move it to the centre, turning on Enable Snapping in the Transform tool. I also make sure that the cylinder is tall enough to cut a hole all the way through.

Once again, I use the **Boolean Divide tool** to remove the overlapping part of the mesh same as I did for the Tablet Grip. (making sure to select the main model first and the cylinder second). I use the same settings as I did for the Tablet Grip. With that the lamp shade is now done.

---

**Preparing For Print**

With all three models complete all that remains is to prepare them for 3D-Printing. Each model will require its own preparations, but none will be particularly challenging.

First, we will look at the Pencil holder. This is a fairly complex model to print particularly because of the cut-out pattern. No 3D-printer can print in mid-air and so we will have to add a so-called support structure to make things easier for the printer. As it happens Meshmixer can do this for us.
The Overhang Tool

The tool you want is called Overhang and you can find it under Analysis. As you can see there are a lot of settings to change here but we can simply select Ultimaker 2 in the Preset dropdown knowing that this is the same as the printers available at Sliperiet. After that there is only one more thing to consider and that is the Layer Height that you will be printing at. This value should correspond to the settings we select in the upcoming step and it is the single settings that will have the most impact on the quality of your printed model. It is unfortunately also the factor that will contribute the most to the time it will take to print your model. You may want to ask for some advice setting this value but for a model like this Pencil holder 0.15 – 0.2 mm should give you satisfactory results. Having set that value, click the Generate Support button and once it has finished computing you should see something akin to the picture above. The resulting branching structure will greatly improve the final print in case where parts of the model are hanging in mid-air. Whilst they may look intrusive they are designed to snap-off with minimal effort. Next click Done and your mesh is ready for export.
Cura

Support structure added, it is time to export your model and opening it in Cura for slicing. Cura is what is referred to as a Slicer software, and its job is to ‘slice’ your 3D model into 2D layers that can be understood by the 3D-printer. Cura can be a bit overwhelming but the point of this guide is to make this process as simple as possible, and in the case of the Penmug there are only two settings that you will need to change.

In order to alter those settings, you will first have to make sure that you press the button label Custom. Then can select the Normal profile (selecting discard if you are prompted by a dialog window). Now for the two settings, the first being Layer Height. Again, this should correspond to the value you selected when generating the supports in Meshmixer, in my case I selected 0.2mm. Next is the Wall Thickness which should be roughly the same as the value you selected when you made your model into a solid. If you did not make a note of this then simply exaggerate the value to something like 5mm and things should turn out fine. At this point your work is done and all that is left is to save the file to an SD card and put it in the printer.