INTRODUCTION TO 123D DESIGN

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1 Introduction

1.1 What is 123D Design?

123D Design is a powerful, easy and user-friendly 3D application that can create amazing and complex objects starting from basic shapes or sketches. These objects can then be 3D printed or manufactured with CNC, laser cutters, water jet cutters, etc. The ability to start from content reduces the learning curve and avoids the need of the level of abstraction required when starting from a black canvas and sketches

1.2 How does 123D Design work?

When we started the challenge of creating a simple but powerful application for 3D modeling and 3D printing, some decisions had to be made about which workflows to enable. The initial and core target audience was design enthusiasts and hobbyists, who may not want to spend too much time training on a more professional application. Soon we found Design getting used by a much wider audience, so we also expanded the use cases.

That said, we really wanted to stay as simple and intuitive as possible, while adding some extra power on different areas of the product. Using existing content seemed to be a great starting point, as opposed to starting with a blank canvas and forcing people to work from sketches. Sketching requires an additional level of abstraction, which we definitely wanted to support, but still give extra emphases to workflows were you can start with primitives and content. With that in mind, adding content to the scene and making it interact nicely with other content was very relevant. This is the reason why any primitive from the toolbar or content from the Parts Bin will try to align to existing content on the scene, and you will see it moving in the screen in real time. We call that action cruising. Cruising can also happen when you drag an existing part with snapping turned on in the navbar (by default snapping is always turned off). At that point, the part will try to use whatever you picked (a face, an edge, a vertex) to cruise and align with other faces or edges.

Most of the interactions try to be as natural as possible, and sometimes quite different to mechanical cad applications. For example, upon clicking any object, we select the object itself (instead of going directly into faces or edges). If you want to select edges or faces, you will need to drill down with a second selection. This interaction has proved to be much simpler, and also allows users to immediately click and drag an object. This would be less desirable in mechanical cad, but our target is far from that one anyways.

Another interaction we added from the beginning (and that you can disable if you don't like it) is that as soon as you snap an object with another one, they are automatically grouped. Why is that? Let's say that you add a cylinder on top of a box. Then you move the box. Would you expect to move the cylinder? There's a high chance this is the case. And for those situations we created the implicit grouping that you can see on 123D Design. Of course, there are as many people that love it than the ones that hate it. This is the reason we added a toggle for this behavior.

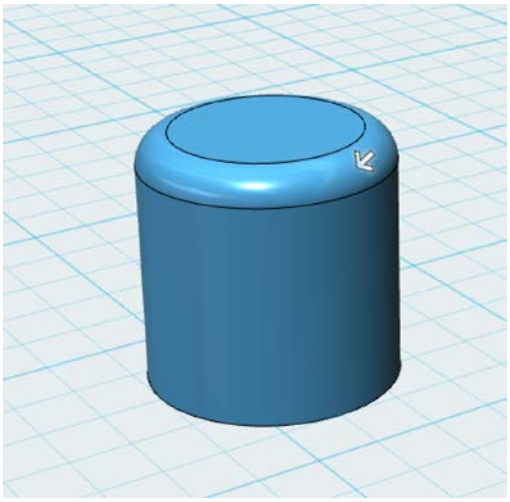
Grouping works in similar ways to other products. When we started thinking about how to build this product, we tried to make it as familiar as possible with known behaviors from 3D and 2D applications. For example, if you double click on a group, you will select the object of the group under your mouse, and will be able to edit this object without ungrouping. Something similar happens in PowerPoint, for example.

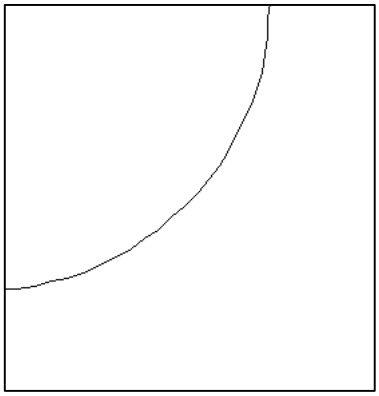
Smart Scale now brings familiarity into a new level, since it offers editing using the very same vertex and midpoint grips that you can see (again) in PowerPoint, or in 123D Design's cousin, Tinkercad. So we bring familiarity not only to users of 2D applications that are entering the 3D world,

but also to Tinkercad users that may want to start using a more powerful application for solid modeling while keeping some of the interactions they learned and love in Tinkercad (like editing and aligning).

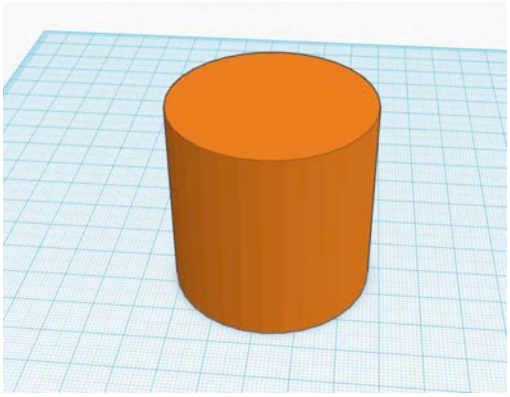
1.3 What is a solid and what is a mesh?

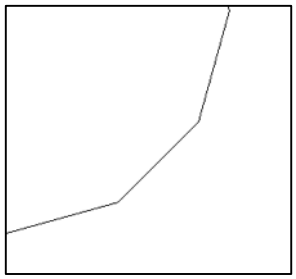
3D modeling solutions often work with two different types of models: meshes and b-rep solids. It is important to understand the difference between both, since it will help you have more control in 123D Design. The app was originally (and still is) a b-rep modeler. This means that you will have the most accurate representation of the objects you are working on. This approach is mostly seen on mechanical cad and some generic ones (Inventor, Fusion 360, AutoCAD). For example, if you build a cylinder in 123D Design (image below), it will be the extrusion of a circle. The surface will be smooth and extremely precise.

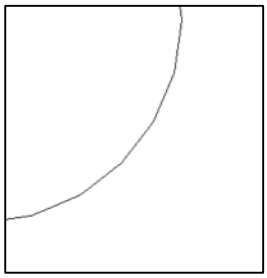




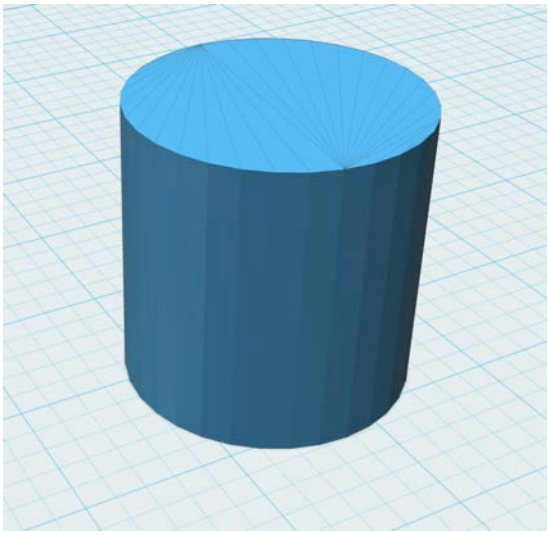
When you work with an application that builds meshes (like Tinkercad – image below -, or 3ds Max), a cylinder will contain a faceted circle and faceted side. So it is not as accurate as the previous one (especially if you were looking for a perfectly smooth side). You can always increase the resolution of a mesh and get closer to the real curve, but at the expense of really huge models that will slow down your system. The two images below show a section of a cylinder with 12 and 24 facets. The last one looks more like a smooth circle, but you would probably need many more facets to try to match the circle in the image above.







123D Design introduced meshes in a previous release, where we allowed you to have some level of interaction between them. That said, not all options that work on a solid work on a mesh. Let's see why not. Take a fillet, for example. A fillet can be defined as the blending of two or more adjacent faces by adding a radius on the edge, so you can get a smoothed edge. This is an easy operation in a solid, since you have one edge between two faces. Let's go back to the example of the cylinder. If you see a cylinder made out of a mesh (image below), the edge will be a collection of segments and there will be several faces converging there. Even worse, in some cases you will have a triangular face converging by a vertex. Smoothing an edge becomes a much more complex operation. It could be solved on a single fillet, but if you try to fillet or chamfer multiple edges, the solution is quite difficult when doing meshes.



B-rep solids are created in 123D Design or can be imported as SAT, STEP and SMB files. You can also insert a 123dx file that might contain solids.

Meshes are always imported into 123D Design, and can be either STL or OBJ files. You may also have a 123dx file containing meshes.

You can then do operations like Merge, Subtract and Intersect between solids and meshes. You can move, rotate and scale meshes. Some commands will only work in b-rep solids: fillet, chamfer, split body, split face, extrude, sweep, revolve, loft, press/pull and shell. If you want to perform any of these operations in a mesh, you will need to convert it into a solid, which we'll explore later in this document.

1.4 What is direct modeling?

Now that we understand what kind of objects can be used in 123D Design, let's mention another important detail: 123D Design is a direct modeling application. So what does this mean? It means that at every moment, you have access to editing faces and edges of any object, and that after every step, the b-rep solid gets recomputed. So, for example, when you do a subtraction between two solids and get a result, the only way to get back is with Undo. You can't access one level deeper in the operation and edit the original objects, because they are already gone. This is a fundamental different with other applications that keep a tree and are also normally parametric.

Direct modeling makes things remarkably easier for entry-level users. Of course there's a lot of power on parametric mechanical modeling, and Autodesk has pretty good solutions on the professional side for this (if you want to go along this path, you must try Autodesk Inventor).

1.5 Why can't I extrude, sweep, loft or revolve an open profile?

123D Design only works with solids and meshes, and doesn't support surfaces, which is what you would get when trying to extrude, sweep, loft or revolve an open profile. The reason we chose not to support surfaces is that 123D Design was mainly thought for 3D printing and other methods of manufacturing, and getting a surface ready for production takes some level of understanding that would be exceeding the scope of this product. There are great solutions for surface modeling, like Fusion 360, Alias Studio, AutoCAD, Maya, etc. No need to add yet another tool that does the same, at the cost of making the experience more complicated for an entry level user.

1.6 Can we use a coordinate system in 123D Design?

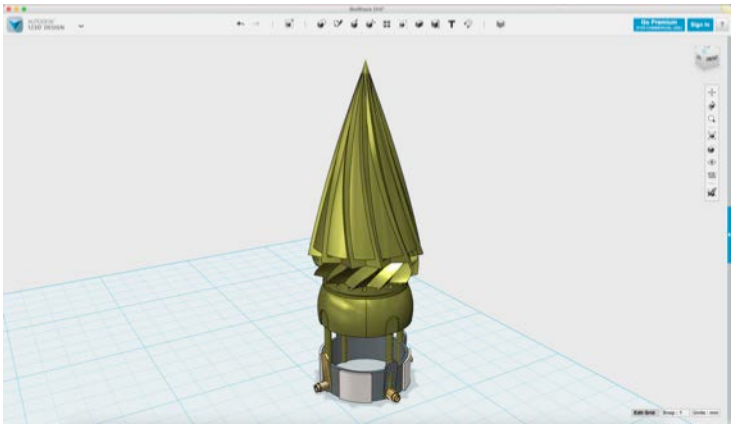
The answer is Yes and No. 123D Design does not expose a coordinate system (universal or user defined), but it somehow fulfills similar needs with the Ruler. You will be able to insert the Ruler at any part of the scene. If you insert it at the origin and then select an object, you are actually using a universal coordinate system.

If you snap the Ruler onto any object in the scene and select another one, you will be controlling relative position between these two objects, which is one of the main purposes for a relative coordinate system anyways.

1.7 What can I do with 123D Design?

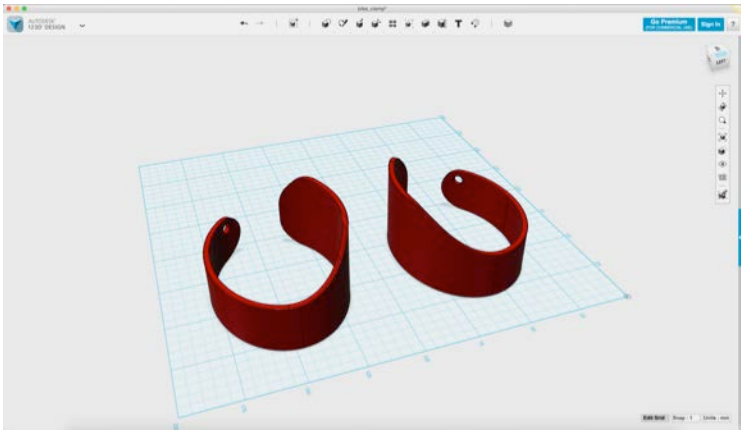
Just a look at the Gallery can show you all the amazing work of our users.

Let's take a look at some examples.

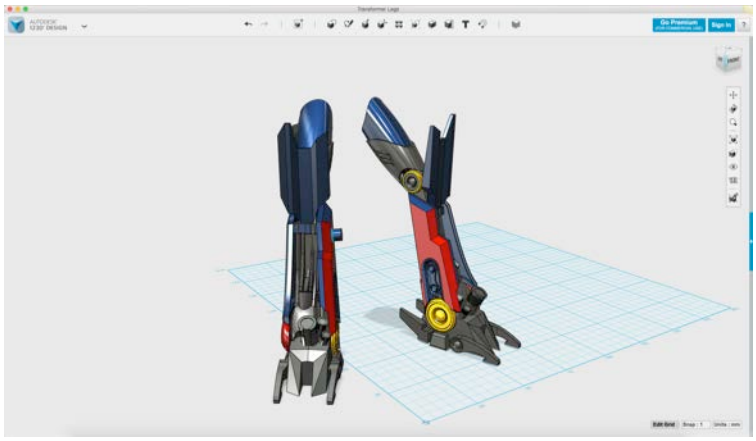


Great use of circular pattern for the drill.

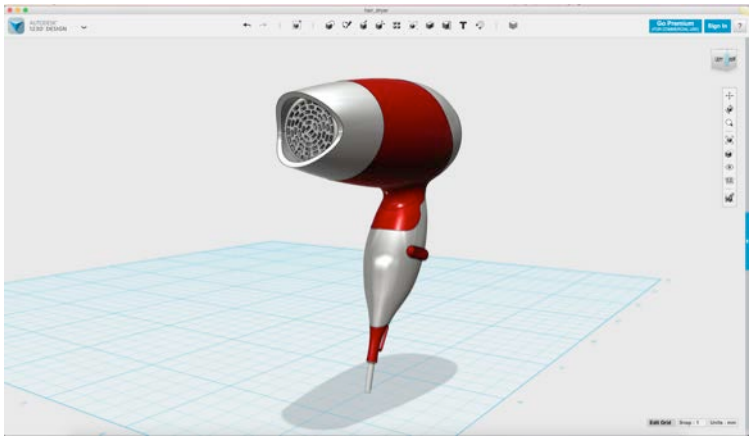
The thin layer of metal next to wood was achieved with projection of the boundary of the wood, and then this projection was offset in order to extrude the result.



This model looks complex, but is basically an extruded sketch and then the body gets split with a spline.

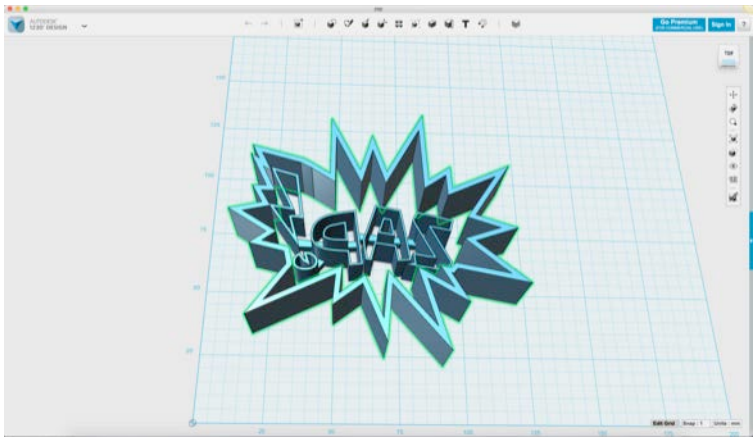


Great example of how you can create extremely complex models out of aggregating simple parts.

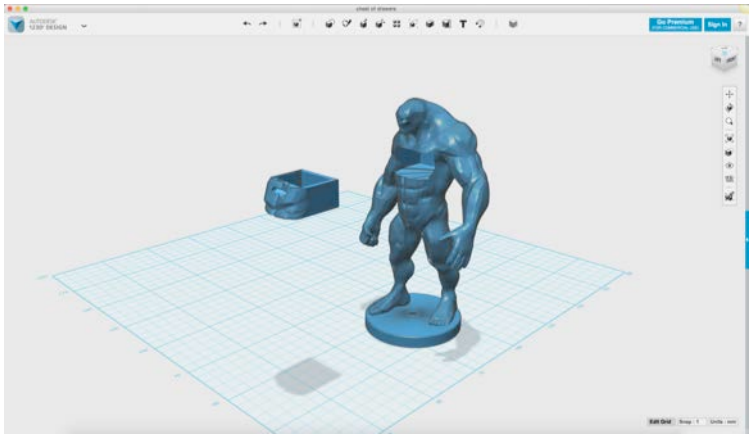


You can create interesting models by understanding how to use loft of several profiles.

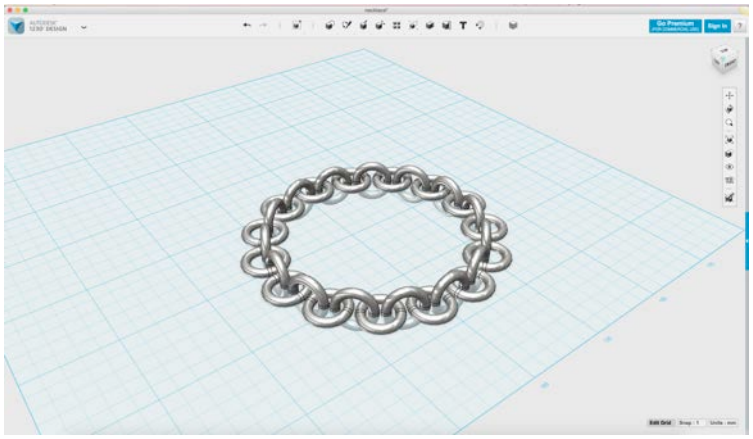
This model requires some sketching and projection of sketches, and some understanding around manipulations of sketches before lofting.



Importing an SVG file from a comic book can become a cookie cutter in minutes.



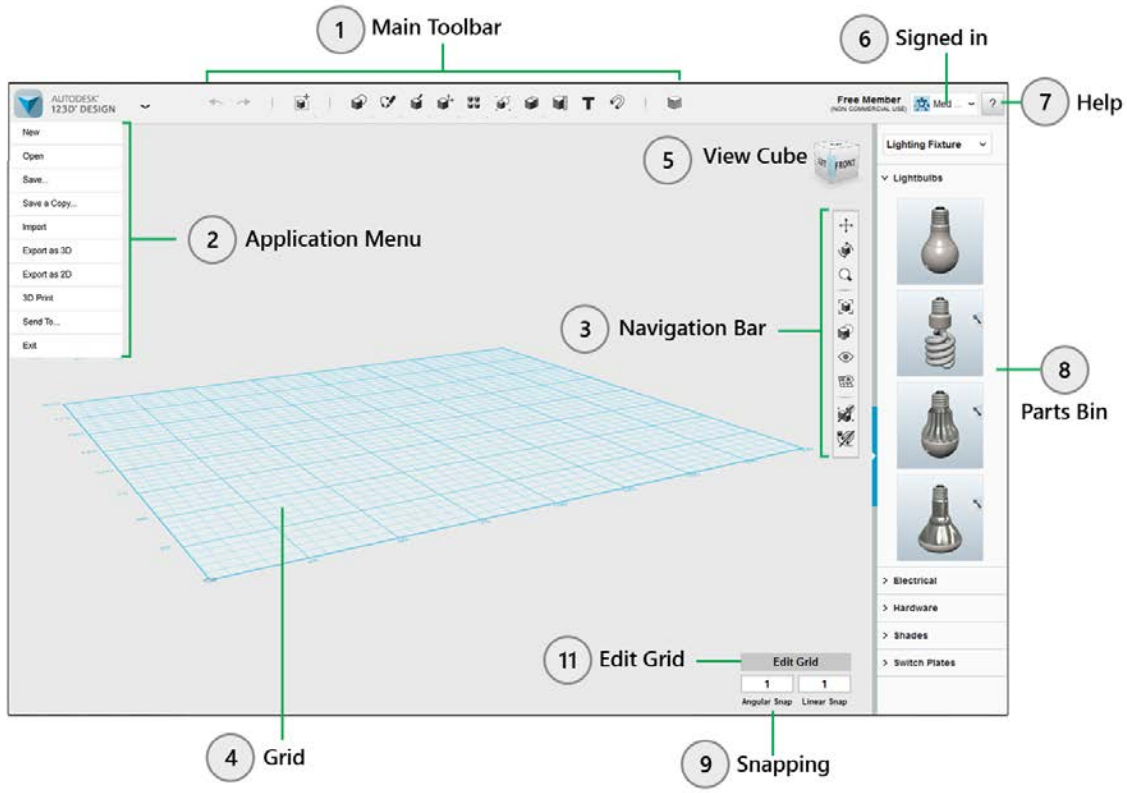
This example shows how an imported mesh (in this case an OBJ from 123D Creature) can allow you to do fun things in 123D Design. The drawer was cut on the mesh using a subtraction against a box.



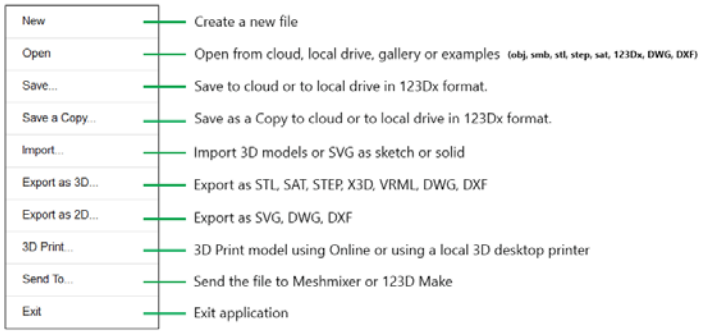
Another good example of a circular pattern. There's no end to creativity with these tools!

2 Getting Started

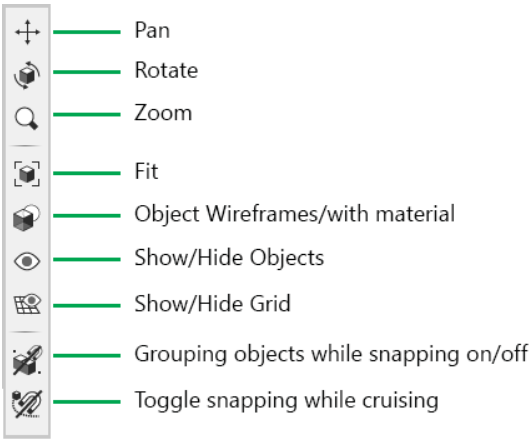
Let's take a look at all the UI components:



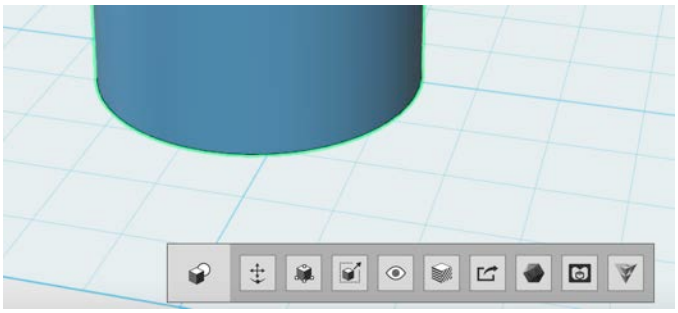
1. **Main Toolbar** – Start creating 2D shapes or 3D models using the tools in this toolbar. Transform, construct and modify objects using tools.
2. **Application Menu** – Create new, open, save, import and export files from here. Directly send the files to different applications like 123D Make and Meshmixer or simply send your model to 3D Print to your desktop 3D Printer or to Online Print Service Bureaus.

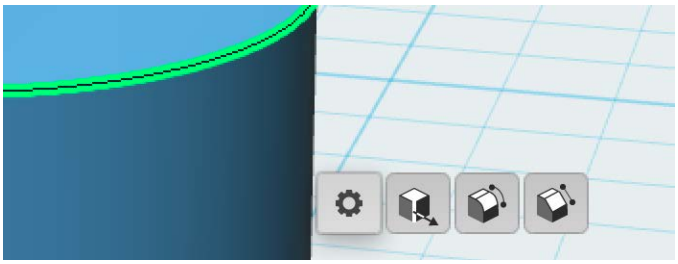


3. **Navigation Bar** –



4. **Grid** is the place to build the model. Switch on/off the grid with the icon on the Navigation bar.
5. **Viewcube** is the easy way to navigate on the screen. Just by clicking on different angles, view the model in different angles.
6. **Sign in** – Log on to check your projects, models and social connections from your account.
7. **Help** – Find shortcut keys information. Learn more about the product through the videos, tutorials, forums and website..
8. **Parts Bin** – Choose from the wide library of Kits that contain several useful models that you can use and modify in 123D Design.
9. **Snapping** – nudging is easy way to bring precision while creating. Use arrows on the keyboard to nudge object.
10. **Edit Grid** – Change the grid size to the preset sizes or specify your own custom grid size.


Context menu when selecting objects


Context menu when selecting edges or faces of objects

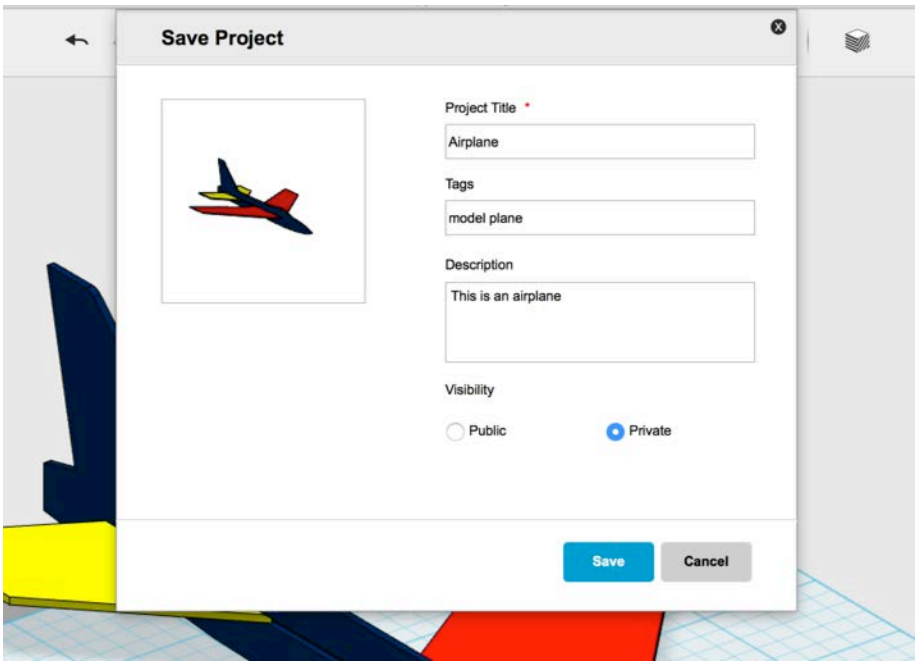
2.1 Opening a file from the cloud or from local drive

As a 123D user, you have access to save and open projects from My Projects, which are hosted in the cloud. You can also save locally on your drive. When you save a 123dx file in the cloud, we'll also create an STL file that you can download from anywhere, or you can directly submit to a 3D printing service from within your account in 123dapp.com. Of course you can also export an STL file from the application itself. You definitely have a lot of choices.

1. Select the Open (Ctrl/Cmd+O) option from Application Menu.
2. Choose from 'My projects', local computer, 123D Gallery or Examples.
3. Login to www.123dapp.com and view your models under My Projects.

2.2 Saving a file into the cloud or local drive

123D Design allows you to save either to your local drives or into the cloud. In order to save to the cloud, you need to save to My Projects. Once you select the option to Save or Save Copy into My Projects, you'll need to sign in (if you hadn't signed in yet) and add info on the requested fields.

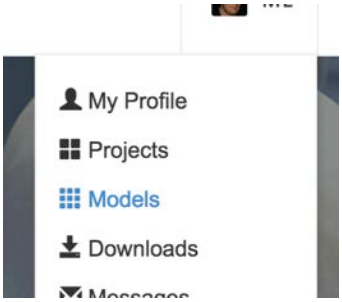


You also have options for saving Public or Private. When a model is saved to Public, it can be searchable by others and appears in the Gallery. Other users will also be able to download the model.

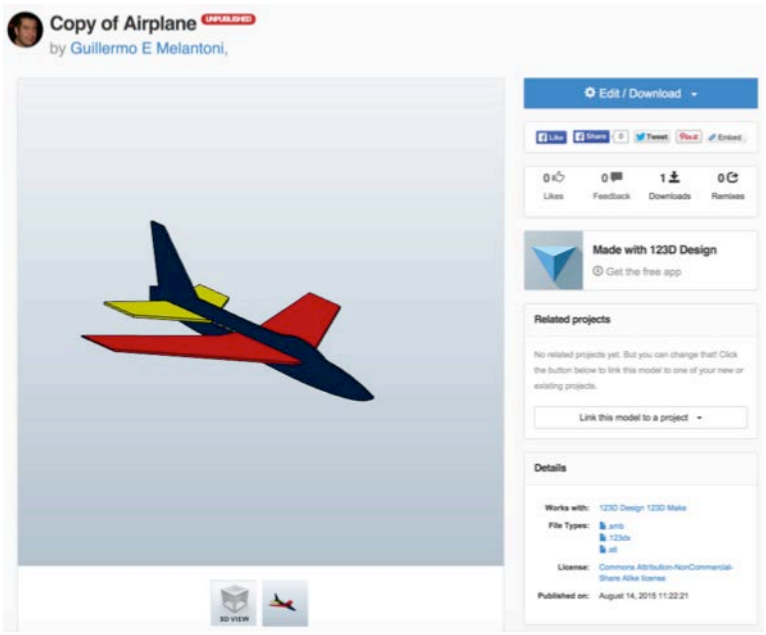
One of the benefits of saving to My Projects is that you get the file translated to different formats that you may find useful, especially if you are not at your computer where 123D Design is installed, and you want to browse your models, show them to someone else, or even launch 3D printing from outside of the application.

Let's see how the model we just saved looks like in My Projects.

You will need to visit www.123dapp.com, sign in and go to Models (it's under your profile)



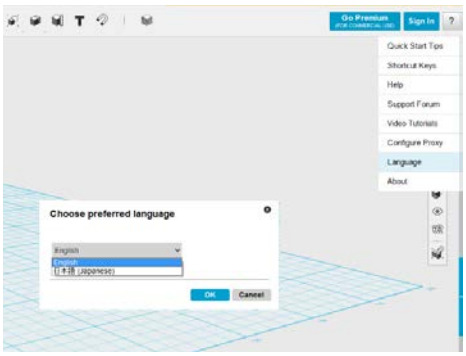
You will get to your gallery of models coming from all 123D apps (if you use Catch, Make or Meshmixer) and when you select the model's thumbnail, you will see several options.



From here you can download all translated assets (STL file, SMB –solid- file) and also launch services like 3D printing and creation of 2D layout.

You can also choose to add this model into a Project. A Project is made out of one or more models (from any app), and can also host pictures.

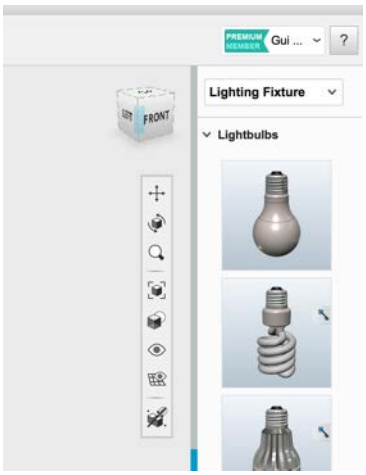
123D Design supports 2 languages at present. English and Japanese. You will get the option of choosing the language while you are installing. (on Windows). You can also choose the language from the help menu. After selecting the language you need to restart the application to view the change.



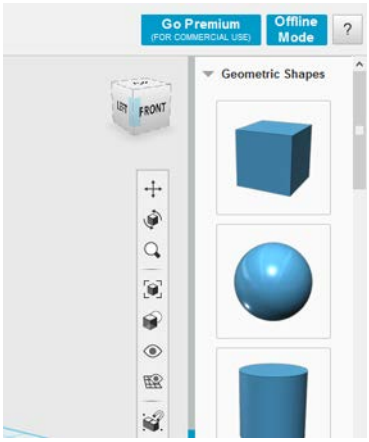
2.3 Online and offline modes

123D Design can work either online or offline. If you are in offline mode, you won't be able to access any of the online features (like opening and saving from/to Gallery, creation of 2D layout and send print to service bureaus). The content of the Parts Bin is also loaded from the web, so once you go to offline mode, you will get a reduced version of shapes to work with.

You will be notified you are in offline mode by a message next to Help on the top right part of the screen.



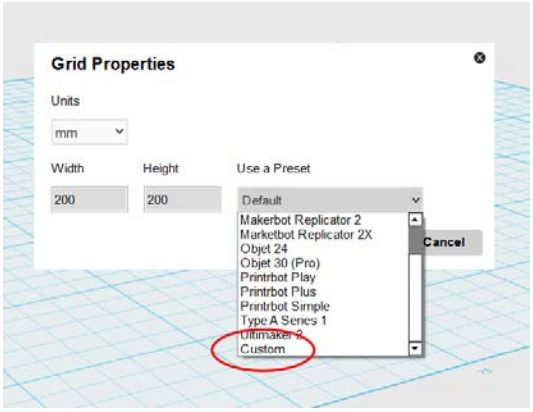
Online mode



Offline mode

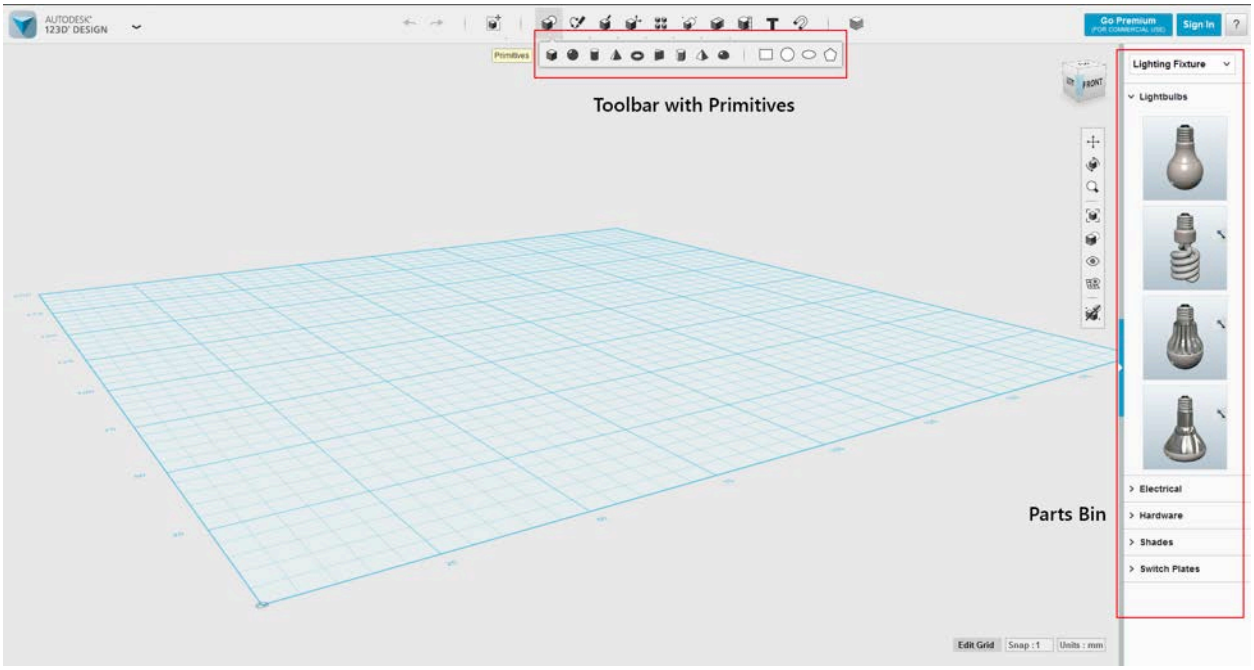
2.4 How to change the grid size?

In 123D Design you can select grid size of your choice form a wide variety of preset dimensions. If you want to prepare your file for 3D printing, you can select grid size of your 3D Printer's print bed. You can actually create your own custom grid size and save it for later use.



3 Using content in 123D Design

Starting with predefined content and creating models by manipulation of these shapes is much more intuitive than sketching. 123D Design has a set of shapes available in the toolbar, which we call primitives. It also has a panel with content on the right side, which we call Parts Bin.



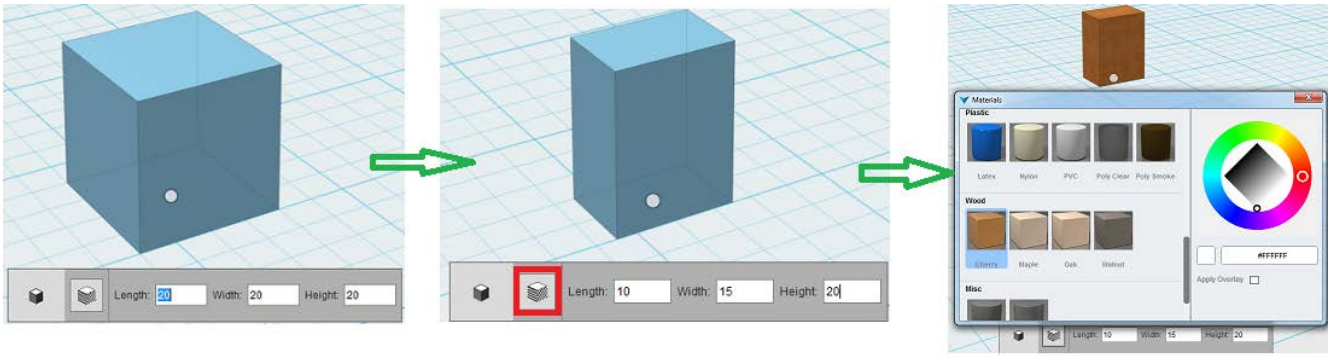
3.1 Concept of primitive

A primitive is a simple shape that can become a building block for different models. In 123D Design, we have 2D and 3D primitives. The 3D primitives are Box, Sphere, Cylinder, Cone, Torus, Wedge, Prism, Pyramid and Hemisphere. All of these contain some parameters that will be available for editing before inserting them on the scene. Once they are in the scene, manipulation of size should be done with other methods.

3.2 How to insert a 3D primitive in a scene

The easiest way to start a project is by dragging content into the scene. This can be done through the Parts Bin that appears on the right, or by adding primitives from the toolbar. Primitives allow you to edit their size before you place them on the scene. They will also try to align with other existing objects in the scene.

Let's see how to insert a Box in the scene.



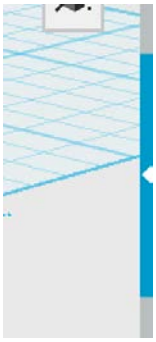
- Select a Box on the main toolbar under 'Primitives' menu
- Input values for Length, Width and Height in the glyph
- Place box in a grid/face, and then click outside with the mouse
- Click 'Material' to select a material e.g. Cherry
- While inserting a new primitive for the first time, you can snap it to any other solid by dragging it closer to another solid.

Note: Insert Sphere, Cylinder, Cone, Torus, Pyramid, Hemisphere, Wedge, and Prism in a similar way.

3.3 Using content from the Parts Bin

The Parts Bin is a repository of different kits of objects. You will find several categories that can help you in different projects (hardware, lighting fixtures, gears, toys, etc). Insert content by dragging and dropping into the canvas.

The Parts Bin may be collapsed by default. You will find a small control to the right of your screen, and you can use that same control for collapsing the Parts Bin.



When the Parts Bin is collapsed, you will see this control to expand it.

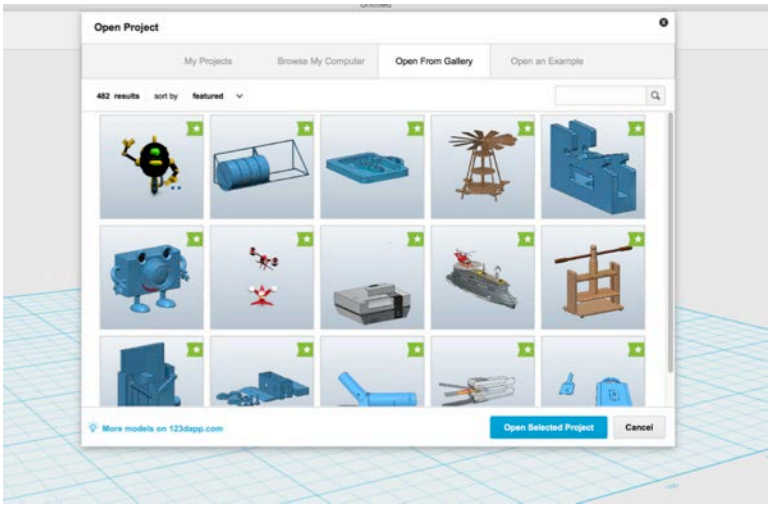
If the Parts Bin is expanded, you can use this control to collapse it.

There is a subtle difference between inserting primitives from toolbar compared to dragging models from the Parts Bin. The primitives from the toolbar will cruise against other objects (trying to align to faces), whereas the models dragged from the Parts Bin will always be inserted on the grid.

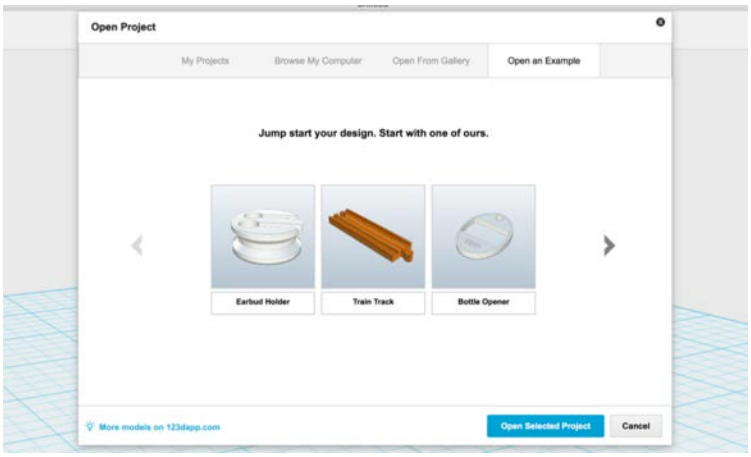
3.4 Opening content from the Gallery

You can also access a lot more content by opening or inserting content from the 123D Gallery. Since 123D Design can open both solid and mesh models, most of the content in the Gallery will be usable. You can browse the Gallery and sort content in different ways.

Just as a reminder, any content coming from an application based on mesh modeling may not be watertight (meaning that there may be gaps in the mesh which will prevent you from doing conversion to solid or 3D print successfully). We'll cover some ways of solving this problem later in this document.



123D Design also has a small gallery of examples that can help you understand the potential of the product.

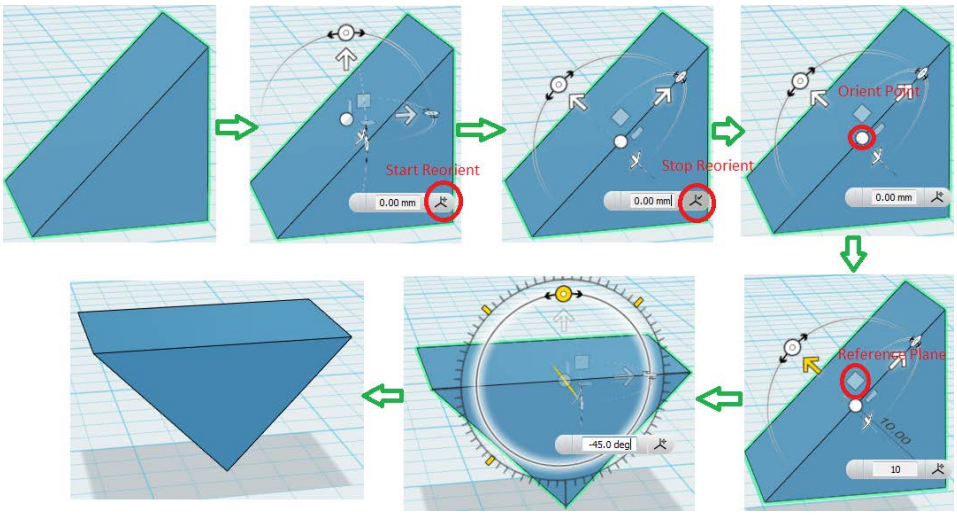


4 Manipulation

Now that we know how to insert content in the scene, let's see how to manipulate this content. Manipulation can happen at the object level, or at the sub-object level (faces, edges and vertices).

4.1 How to move/rotate an object/sketch

Move and rotate are done using the same Transform tool (if you want to do direct manipulation). You can also move by nudging with the arrow keys and rotate with the X, Y and Z keys. But if you want the whole experience, you will probably use the Transform tool. You can then move and input a distance, or rotate and input an angle. The Transform widget will by default be shown respecting universal coordinates. Sometimes you will need the direction of the widget to be aligned with something else in the scene. In those cases, you will use the option Reorient and the widget will align with any face you choose and also snap to the right place. Snapping to a new origin will allow you to use the widget as a pivot for rotation.



- Select solid/mesh/sketch.
- Select (Ctrl/Cmd+T)Move/Rotate feature on the main toolbar under 'Transform' menu
- Select Arrow manipulator, and then input the exact value or drag the Arrow manipulator to move
- Hold Rotate manipulator, and then input exact angle or drag the Rotate manipulator to rotate. If you press Shift while dragging, you will constrain the rotation every 45 degrees.
- If you want to reorient the center point, click 'Start Reorient', and move the manipulator to the appropriate point. Click 'Stop Reorient' to fix the point.

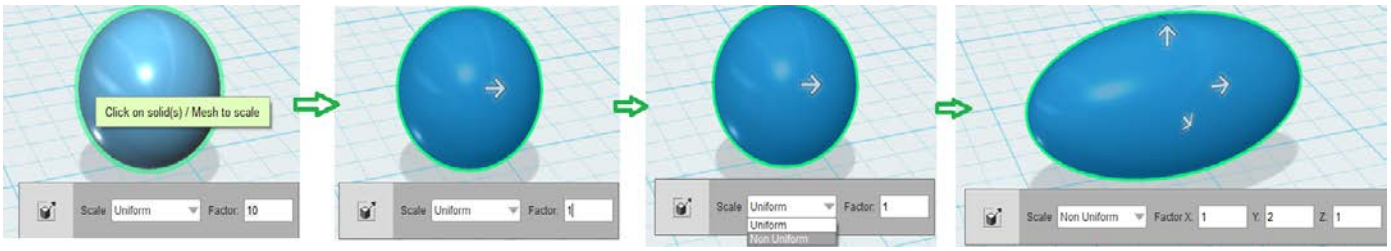
- Press ‘Enter/Return’ to execute the command or just click outside with the mouse.

Note: If the Orient Point is selected, then the object can be moved in three directions; if the Reference Plane is selected, then the object can be move in two directions.

You can also rotate solids and meshes after you select them without invoking the transform widget. You can basically press X, Y or Z and the models will rotate 45 degrees clockwise along those axis. By pressing Shift, you can rotate counterclockwise.

4.2 How to scale an object/sketch

Scaling in 123D Design can be done through two different options. The traditional one implies the use of a scale factor. Whenever you want to make something twice as big, or a third of the size, even if you don’t know the actual size, this is the option for you.

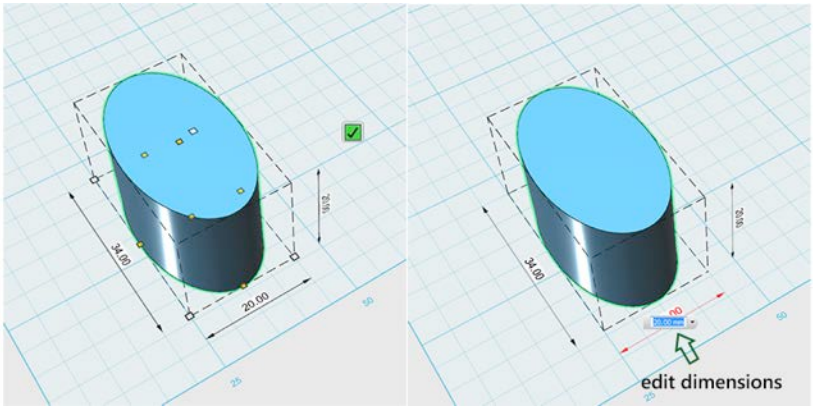


- Select Scale feature(S), on the main toolbar under ‘Transform’ menu
- Select a solid/mesh/sketch
- Uniform scale and Non Uniform Scale are the 2 different ways of scaling an object
- Select ‘Uniform’ to scale the object proportionately
- Select ‘Non Uniform’ and input the X,Y, and Z factor in the glyph, you get the flexibility of changing the values for all 3 directions
- Then, click outside with the mouse.

4.3 How to use Smart Scale

The second method for scaling is very similar to the one in Tinkercad. It’s very useful for direct manipulation or when you know exactly the final size you want on something, either on one axis or in the three.

- Select the object. Launch the Smart Scale feature (Ctrl/Cmd+B) from the Transform Menu on the toolbar.
- You will see manipulators at the corners of the object or the bounding box. Drag them to scale the object non-uniformly.
- Other way is to click on the dimension arrows or the digits to modify the dimensions of the objects. The object will resize assuming the origin on the center, so it will get bigger towards both sides. If you want to control the direction an object scales, you will do a similar interaction but use the Ruler to define an origin (Ruler will be explained later)



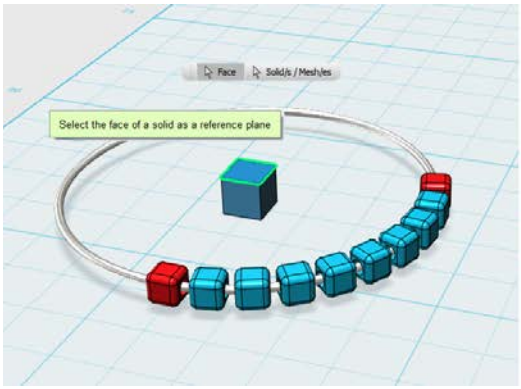
- Press Alt or Option in Mac to scale the object in one direction.
- Press Shift to scale the object uniformly.

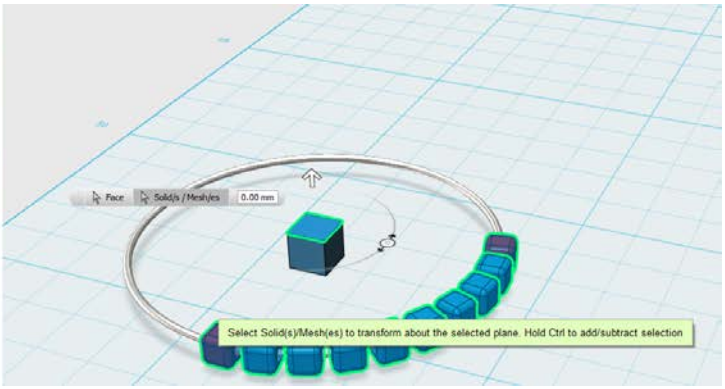
- You can press Alt (Option) and Shift together and scale uniformly from the center.

4.4 How to use Smart Rotate

With smart rotate you can move/rotate selected objects about the selected plane. It is very effective if you want to transform objects around a selected face.

- Select a solid preferably with a planar face. And then select solids/meshes that need to be transformed.
- Select Smart Rotate feature on the main toolbar under 'Transform' menu.
- First select a face of a solid.
- Then select solids or meshes that need to be transformed. You will get the manipulators to rotate and transform the objects.
- Hold rotate manipulator and then input exact angle or drag the rotate manipulator to the appropriate point. You can also move the objects if you wish to.
- Press 'Enter/Return' to execute the command or just click outside with the mouse.





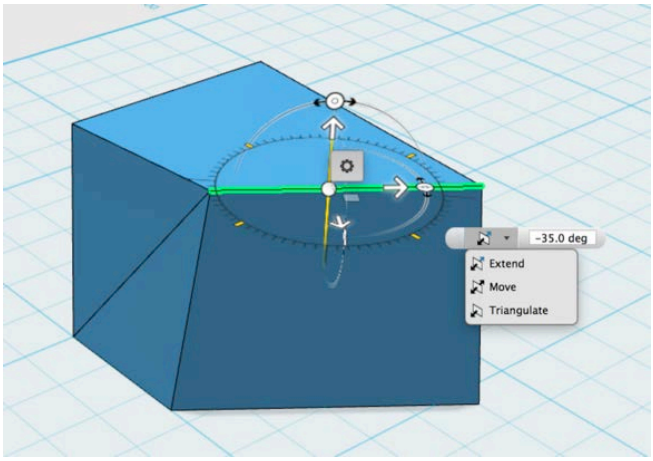
4.5 How to manipulate faces and edges

Once a solid is selected, you can now select a face, edge or vertex. There are several cases in which working at this level gives you much more flexibility than simply doing operations with different objects. This is one of the main advantages of direct modeling. If you had the chance to keep the history of Boolean operations (merge, subtract or intersect), you would lose the history when moving an edge.

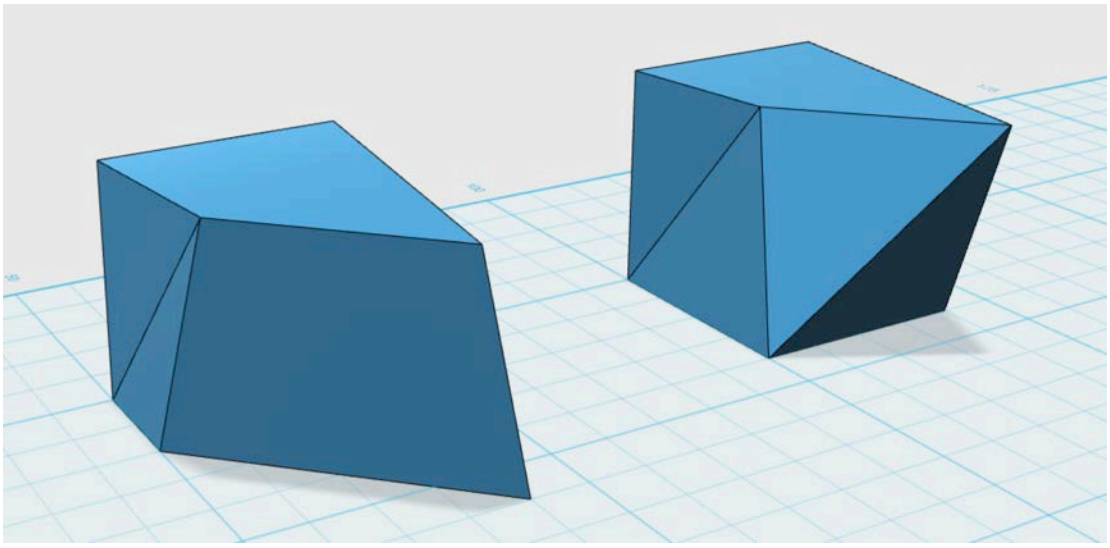
Once you select the object, 123D Design needs to understand if you want to select a face or an edge, if you select the object directly on the face you want to select later, you will need to move the mouse out of the face and back in.

Once you make the selection of a face or edge, you will see some contextual options (Tweak, Press/Pull and Shell for faces, and Tweak, Fillet and Chamfer for edges). Tweak will invoke the same Transform widget for moving and rotating objects, and you will use it on a very similar way.

In case of rotating edges, you will see some options next to the widget. This will help 123D Design understand some of your design intent. For example, when you move an edge you will obviously affect the two adjacent faces. And you will also affect the adjacent faces to these. Try the options shown in the drop down, and see what you exactly need.

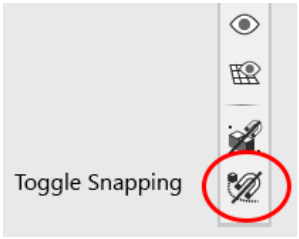


In the two examples below, the main difference is the option (Extend on the first and Triangulate on the second). Triangulate keeps the edge at the bottom of the front face in the same position. This is possible since we are triangulating faces in order to keep the edge unmoved. Extend also rotated that edge when rotating the face (since we were somewhat extending the influence of the rotation to the edge below).



4.6 How to snap solids

When you move objects, there's a high chance you will want to move them next to something else in the scene. Snapping in 123D Design can be done with the Snap tool or simply by dragging objects. Snap tool is explained later in this section. Let's discuss about the snapping behavior while cruising. You may find this option on the Navbar.

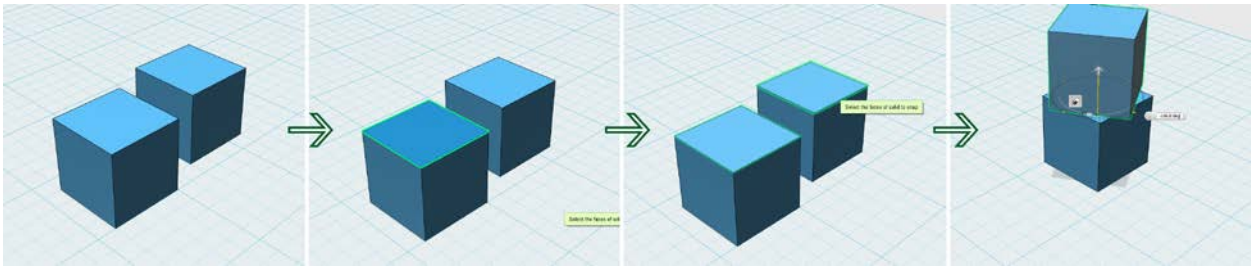


By default snapping of objects is turned 'OFF'. So even though you drag the objects close to each other, they won't snap. But you can set it 'ON' to snap objects to each other when they are close. As soon as it snaps to the face of the target object, you will get transform manipulators on

the snapped face. Use these manipulators to adjust and arrange the object properly. Toggling snapping brings the required flexibility of turning snapping ON/OFF while placing objects and building a model.

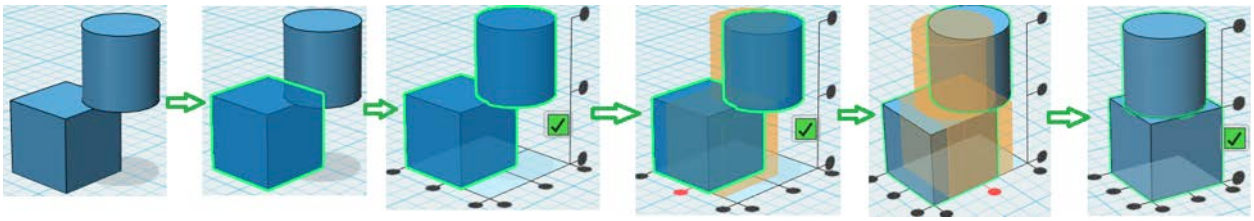
Another way to locate objects precisely with reference to other objects is to use the Ruler. You can drag the ruler, align it and snap it against a reference object, and then select the target object. Once you do this, the ruler will display distances between reference and target objects. Simply click on the values and edit them. If you use 0 as value, it will be like snapping to that object.

Now let's see how the Snap Tool works.



- Launch Snap feature(;) from the main toolbar
- Select a face of a solid to snap
- Select a face of another solid to snap. Once the objects are snapped you can use the manipulators to adjust the position according to the requirement.
- When you drag a new primitive from the toolbar, it automatically snaps to the solid closer to it.

4.7 How to align objects



- Select Align feature (A) on the main toolbar under 'Transform' menu
- Click on the solid/mesh to select
- Press 'Ctrl'/'Cmd' to add selection
- Hover on the round handles to check how the alignment works and click on the handle to align the objects finally.
- Press 'Enter/Return' to execute the command or just click the green ticker.

5 Sketching

5.1 How to sketch in 123D Design

Sketching in 123D Design is somehow a hybrid from what you may have seen in products like AutoCAD and parametric products like Inventor, or even Fusion 360.

Let's see the differences. When you draw in AutoCAD, all elements live in the same space, and anything that's coplanar may be used for a 3D operation like extrude, sweep, loft or revolve.

On the other hand, products like Inventor and Fusion 360 have the concept of Sketch mode. Each part of an assembly has its own sketch. You can also copy sketches out of a part, but basically only segments living in the same sketch mode can interact between each other. So you may have four segments that seem to form a closed square, but if at least one of these is not in the same sketch mode, you won't have a square.

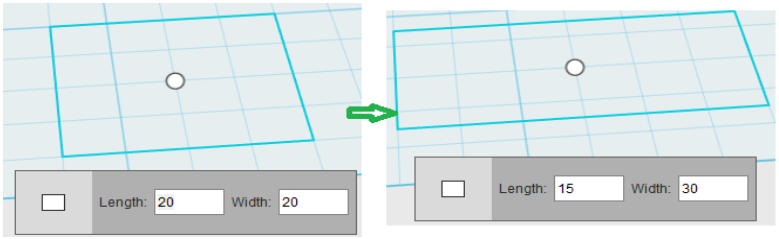
123D Design still has an implicit sketch mode, although no sketch belongs to a part in particular. This is why it can be called a hybrid.

The way to make several lines to be part of the same sketch is to first click on any part of an existing sketch before start sketching. It's as simple as that. Even if you think that two lines are in the grid (and are coplanar), this doesn't mean they will be in the same sketch.

5.2 How to insert a 2d sketch in the scene

123D Design not only contains 3D primitives (Box, Sphere, etc.) but also contains some pre-made 2d sketches. You can either use these or start a sketch from scratch. You can also import a sketch as an SVG file (we'll cover this later).

A nice feature when inserting 2D sketches from the toolbar is that they can align and snap to faces and edges. This can be very useful when just started in the world of 3d modeling, since you don't have to think much about how to align objects manually.



- Select Rectangle sketch on the main toolbar under 'Primitives' menu
- Input values for Length and Width in the glyph. Use Tab to cycle through the values (since if you hit Enter after the first one, it will already finish the operation)
- Place rectangle on the grid/face, and then click outside with the mouse.

Note: Insert Circle, Ellipse, and Polygon sketches in a similar way.

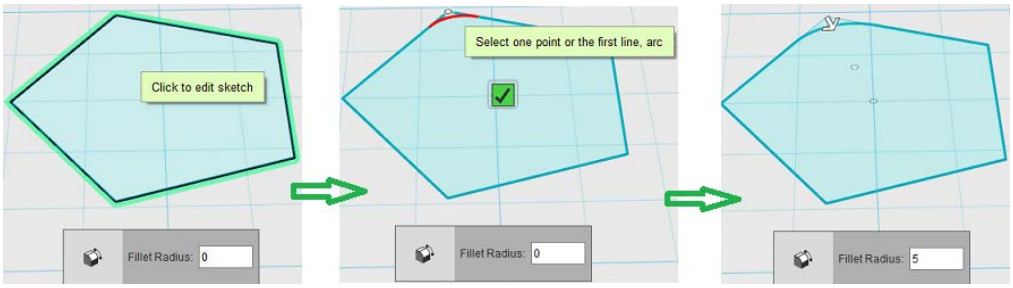
5.3 How to sketch a Rectangle

Before starting to sketch

- Select Sketch Rectangle on the main toolbar under 'Sketch' menu
- Select a grid/sketch/solid face to start sketching
- Click to specify first corner of rectangle
- Input the length and width of rectangle
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

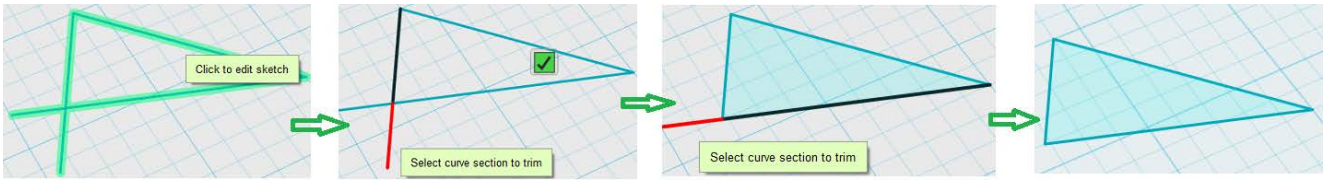
Note: Similarly sketch Circle, Ellipse, Polygon, Polyline, Spline, Two Point Arc, and Three Point Arc on the Grid/Sketch or Solid face.

5.4 How to add Fillet to a sketch



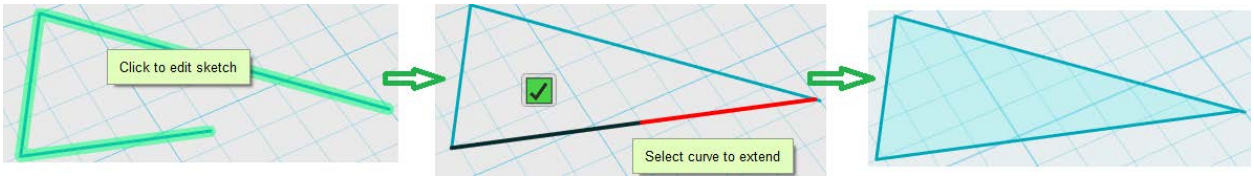
- 'Sketch fillet' adds a fillet to the sketch that already exists in the scene.
- Select Sketch Fillet feature on the main toolbar under 'Sketch' menu
- Select a sketch to edit
- Select one point
- Input the fillet radius or drag the Arrow manipulator to set the fillet radius
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

5.5 How to Trim a sketch



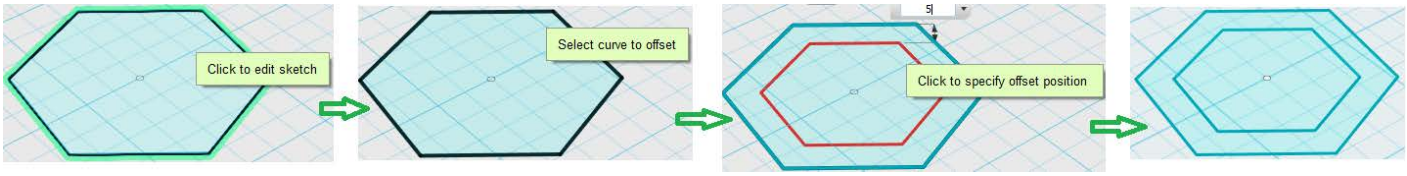
- Select Sketch Trim feature on the main toolbar under 'Sketch' menu
- Select a sketch to edit
- Hover on the line to see the projection. Click on the line to accept it
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

5.6 How to Extend a sketch



- Select Sketch Extend feature on the main toolbar under 'Sketch' menu
- Select a sketch to edit
- Hover on the line that you want to extend. You will see a projection. Click on the line to accept it
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

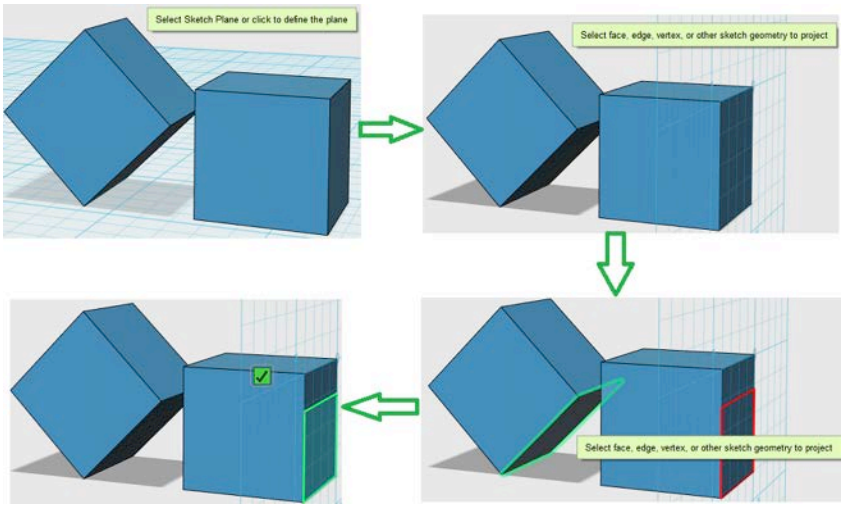
5.7 How to Offset a sketch



- Select Sketch Offset feature on the main toolbar under ‘Sketch’ menu
- Select a sketch to offset
- Input the offset value or move the mouse to set it
- Press ‘Enter/Return’ to execute the command or just click outside with the mouse.

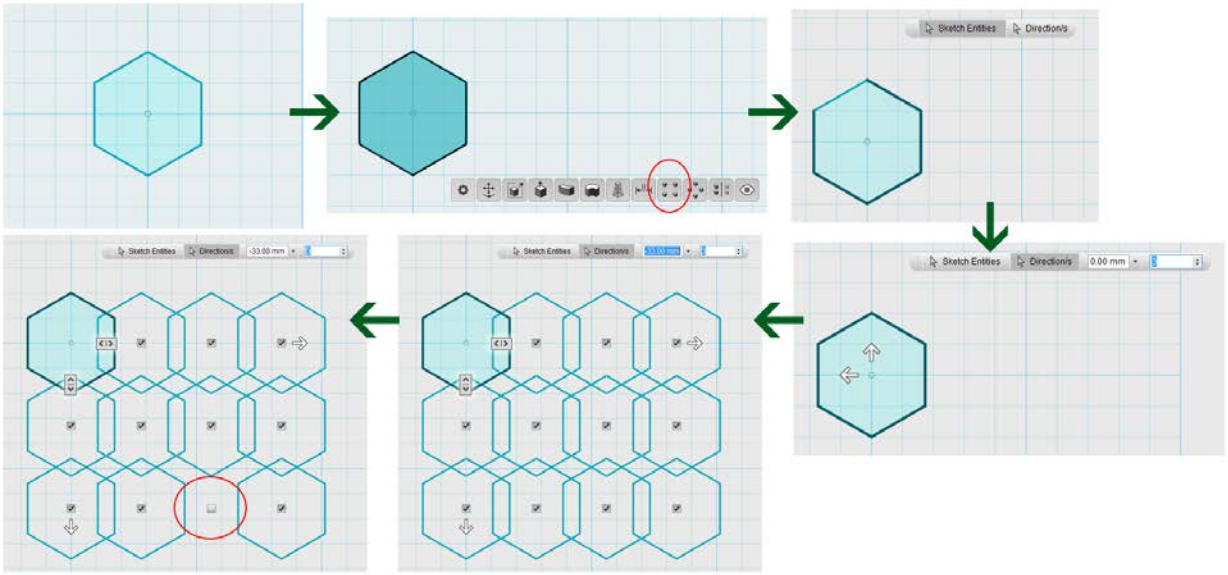
Note: A sketch can offset inside and outside.

5.8 How to Project a sketch



- Select Sketch Project feature on the main toolbar under ‘Sketch’ menu
- Select a sketch plane/grid/face of solid as project plane
- Select face/edge/vertex/sketch geometry to project
- Press ‘Enter/Return’ to execute the command or just click outside with the mouse

5.9 How to Rectangular Pattern with sketches?

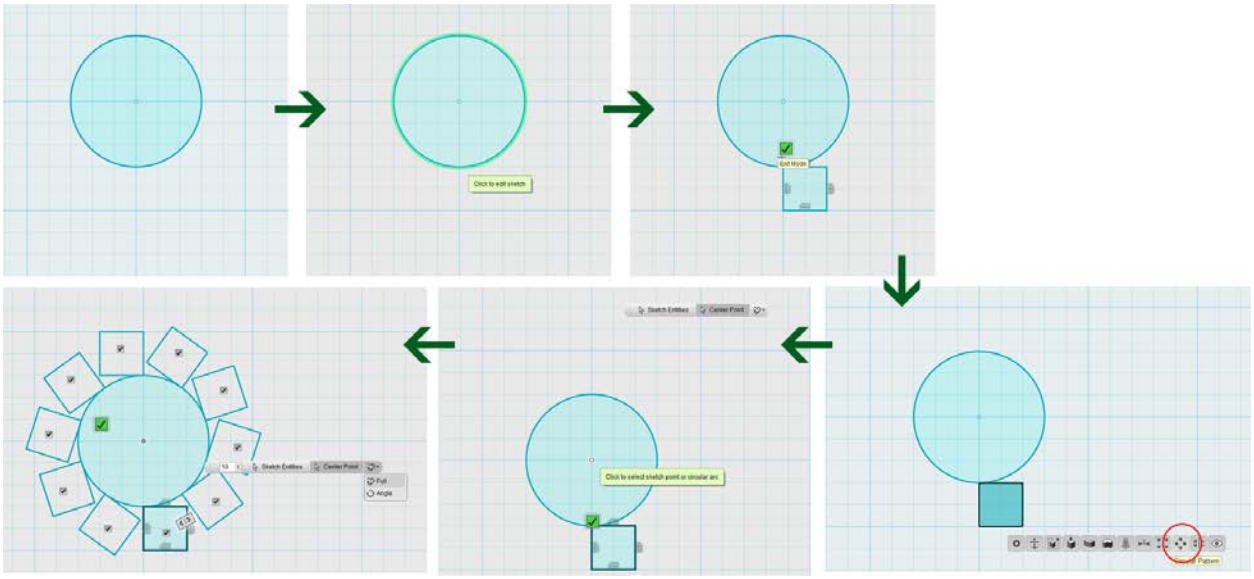


- Insert a Sketch from the toolbar.

- Select the sketch and choose Rectangular Pattern from the contextual menu. Pattern command for sketches can be launched from the contextual menu only. Main toolbar pattern command works only on solids
- From the glyph choose 'Sketch Entities' and start selecting the sketch (entities) that you want to pattern
- Click on 'Direction/s', and then select one or two lines to assign directions
- Input distance and quantity in glyph or drag the Arrow manipulator
- Select another direction, and input the distance and quantity again in glyph or drag the Arrow manipulator
- Press 'Enter/Return' to execute the command or just click outside with the mouse

Note: Multiple selected sketches can also be patterned at once. The number of patterned sketches can be added/ subtracted by moving the slider. To hide a sketch in a pattern, just uncheck the checkbox placed on it.

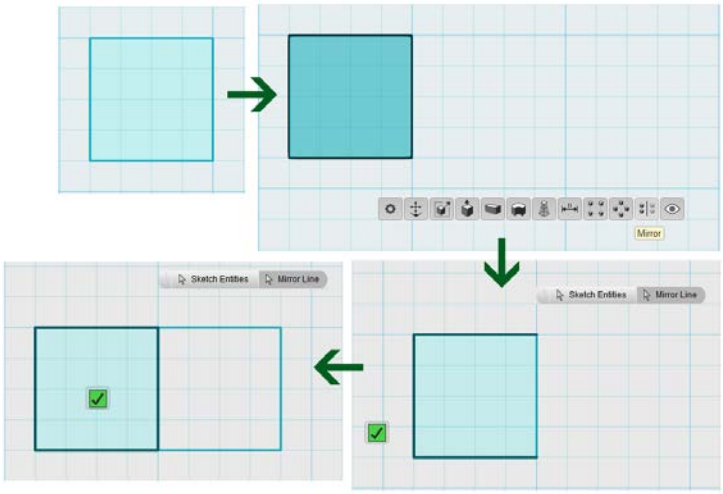
5.10 How to Circular Pattern with sketches?



- Insert a circle sketch primitive from the toolbar.
- Now choose a rectangle sketch tool from the toolbar and first click on the circle to make it one single sketch. Here it would be good to clarify what a single sketch means. If we draw a circle and exit sketch command and then draw a rectangle on the grid then, those become 2 different sketches. But now if we draw a circle, select it then draw another sketch primitive, then those two become a part of one single sketch.
- To use pattern command (rectangular, circular, mirror) both the sketches have to be a part of single sketch
- Select a sketch that needs to be patterned and from the contextual menu choose circular pattern
- Select sketch (entities) that needs to be patterned. Then select the center point.
- Input quantity and select options for full or semicircular pattern from the glyph.
- Press 'Enter/Return' to execute the command or just click outside with the mouse

Note: Multiple selected sketches can also be patterned at once. The number of patterned sketches can be added/ subtracted by moving the slider. To hide a sketch in a pattern, just uncheck the checkbox placed on it.

5.11 How to Mirror sketches?

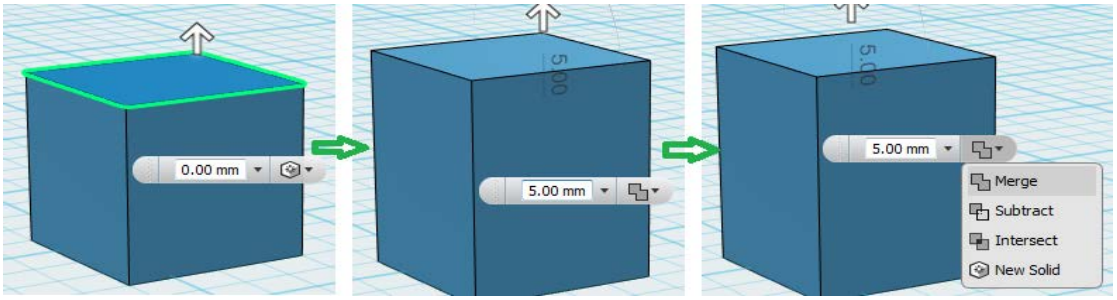


- Insert a rectangle sketch primitive from the toolbar
- Select the sketch and choose Mirror command from the contextual menu
- Select the sketch (entities) that need to ne mirrored. Now choose the mirror line. They all have to be a part of single sketch(explained in circular pattern)
- Press 'Enter/Return' to execute the command or just click outside with the mouse

6 How to create solid geometry

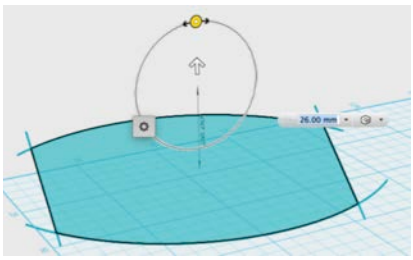
Solid geometry can be created through four commands: extrude, sweep, revolve and loft. It can be done based on closed profiles or existing faces of solids. Let's take a look at some details for each method.

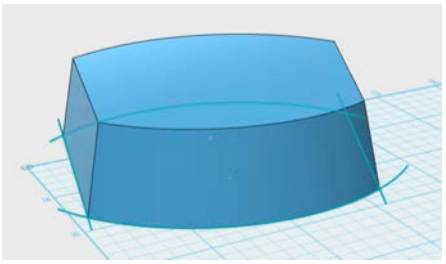
6.1 How to use Extrude feature



- Select Extrude feature(U) on the main toolbar under 'Construct' menu
- Select the top face of a solid
- Input exact value in the glyph or drag the Arrow manipulator on the top of the solid. You can also click on some face that is at a specific height, and 123D Design will use that height as input for value of Extrusion.
- Select the right Boolean method-e.g. Merge
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

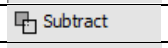
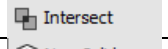
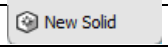
You can also extrude based on a closed profile. In both cases, you will also find a control at the top of the Extrusion widget that allows you to control the taper angle. This makes the extrusion not only be perpendicular to the profile or face, but also have an angle of extrusion. If you plan to make a mold for fabricating your object, this draft angle (or extrusion angle in our case) is critical for removing the part from a mold.





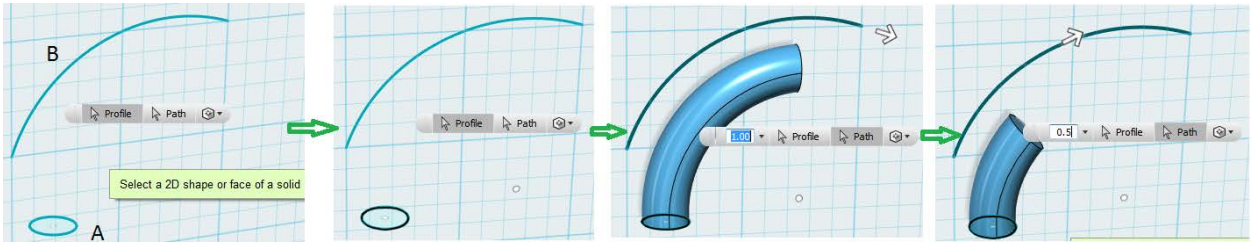
When you are making an extrusion and there's another object in the path of that extrusion, you will notice that it changes into red. This means that it will subtract whatever it finds along its way. If you don't want this to happen, you will be able to change the parameter from Subtract to Merge (will combine with intersecting object), Intersect (will only keep the area of intersection between objects and remove the rest), or New Solid (you will get a new part that will overlap with intersecting model).

Parameter Boolean:

Boolean	Image	Description
Merge		It is used to merge between the new part and original part
Subtract		It is used to subtract between the new part and original part
Intersect		It is used to intersect between the new part and original part
New Solid		It is used to create a new solid.

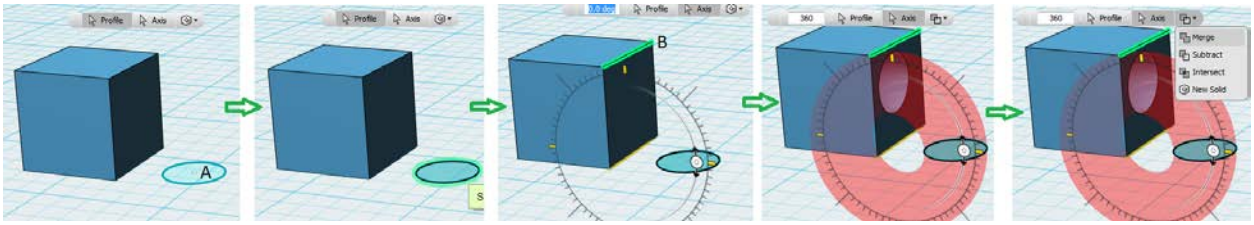
Note: Boolean works in the same way, for all of the features below. Use it exact the same way as you use while extruding.

6.2 How to use Sweep feature



- Select Sweep feature(W) on the main toolbar under 'Construct' menu
- Select closed 'Profile' A
- Click 'Path' button, and then select the path B
- Input exact value in the glyph or use manipulator to sweep
- Select the right Boolean method - e.g. New Solid (as explained in Extrude Feature)
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

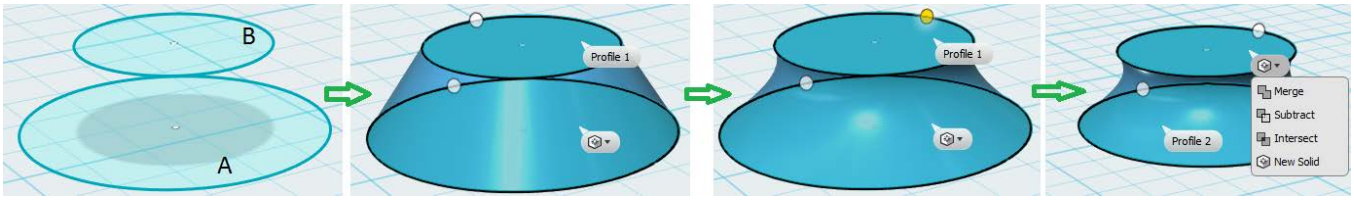
6.3 How to use Revolve feature



- Select Revolve feature (V) on the main toolbar under 'Construct' menu
- Select closed 'Profile' A

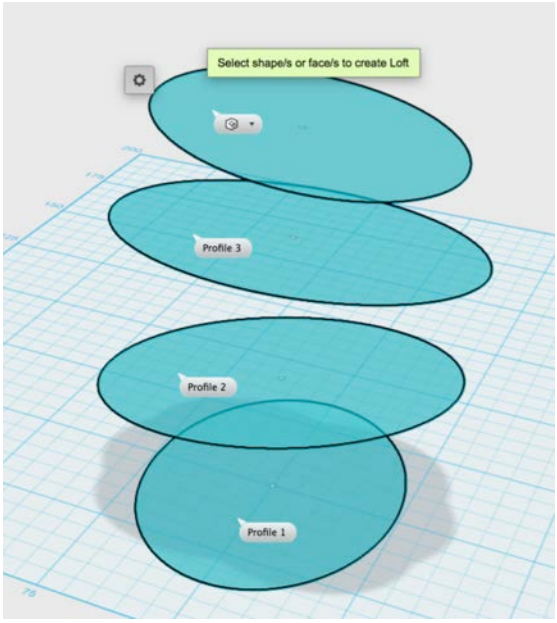
- Click ‘Axis’ button, and then select axis B
- Input exact value in glyph or use manipulator to revolve
- Select the right Boolean method - e.g. Merge (as explained in Extrude Feature)
- Press ‘Enter/Return’ to execute the command or just click outside with the mouse.

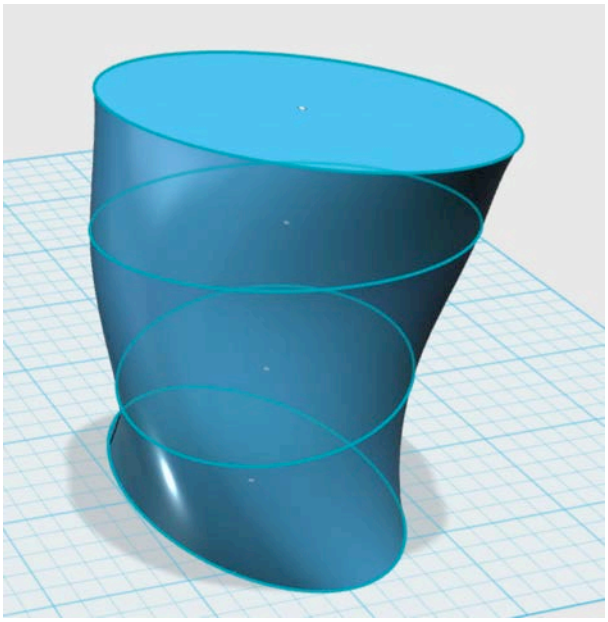
6.4 How to use Loft feature



- Create two closed sketches
- Select Loft feature (L) on the main toolbar under ‘Construct’ menu
- Select the first sketch A, and then click sketch B
- Drag round handles to modify the shape
- Select the right Boolean method-New Solid (as explained in Extrude Feature)
- Press ‘Enter/Return’ to execute the command or just click outside with the mouse.

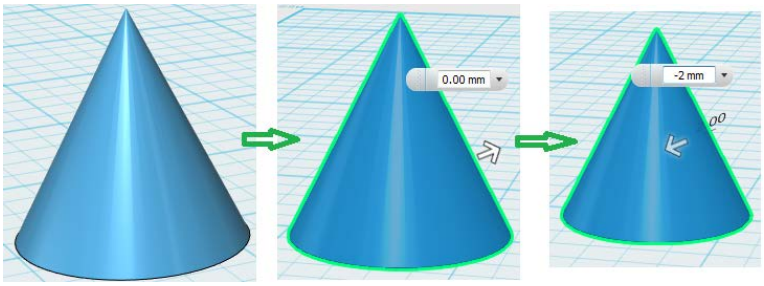
You can also loft multiple profiles. You simply need to click on Ctrl (on PC) or Command (on Mac) while you select profiles. You won’t see the progress of the loft until you press Enter and commit, since the application can’t assume the curvature on the profile before knowing what profile is coming next.





7 How to edit solid geometry

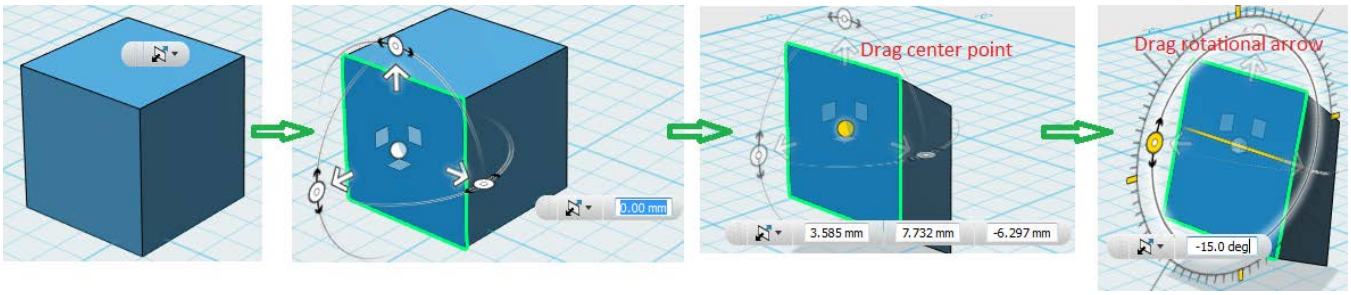
7.1 How to use Press Pull feature



- Select Press Pull feature(P) on the main toolbar under 'Modify' menu.
- Select a surface/plane of a solid
- Input exact value in glyph or pull the Arrow manipulator.
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

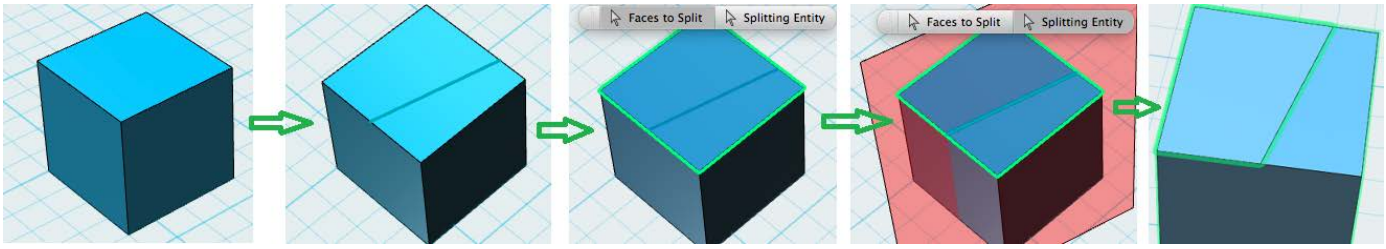
Note: if the input value is negative, then the 'Press' function is implemented. If the input value is positive, then the 'Pull' function is implemented.

7.2 How to use Tweak feature



- Select Tweak feature (K) on the main toolbar under 'Modify' menu.
- Select a 'face' or 'edge' or 'vertex' of a solid
- Drag center point to distort the solid in X, Y, and Z direction
- Drag rotational Arrow Manipulator to distort the solid in a degree
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

7.3 How to use Split Face feature

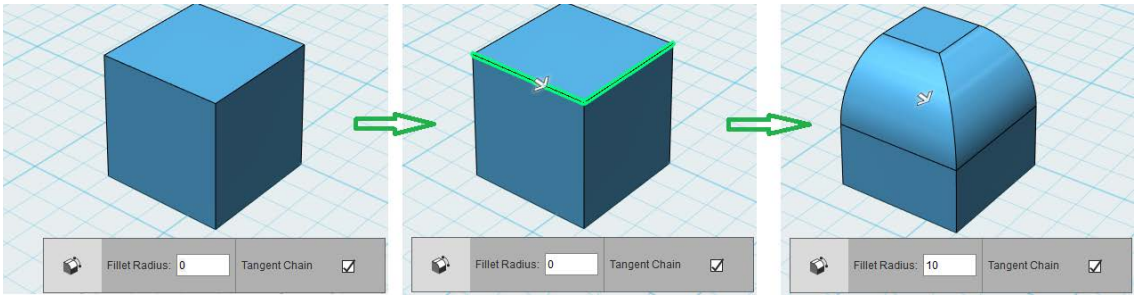


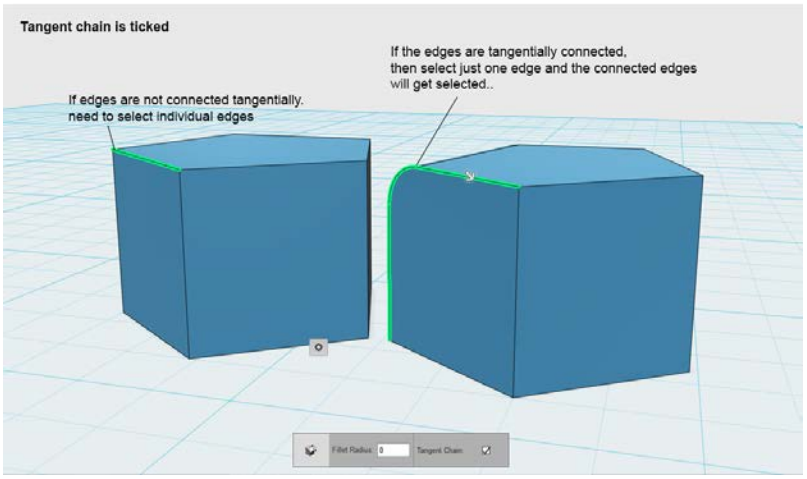
- Create a solid

- Create a sketch line on the top of 'face'
- Select Split Face feature (B) on the main toolbar under 'Modify' menu
- Click 'Face to Split' and then select the top face of a solid
- Click 'Splitting Entity', and then select the sketch line
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

Note: A face can be split by using a solid.

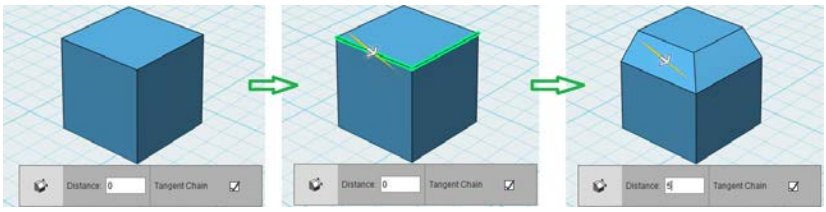
7.4 How to use Fillet feature





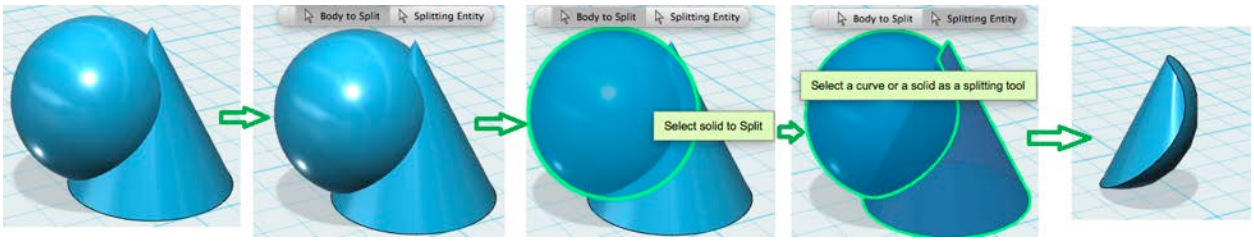
- Select Fillet feature (E) on the main toolbar under 'Modify' menu
- Select an edge or multiple edges of a solid
- Input exact radius value in glyph or drag the Arrow manipulator. You can also click on a previous fillet and 123D Design will take the value of that fillet, and use it as input for radius.
- If Tangent Chain is checked on then all the edges that are **tangentially connected**, will be selected at once. Otherwise you will need to individually select each edge.
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

7.5 How to use Chamfer feature



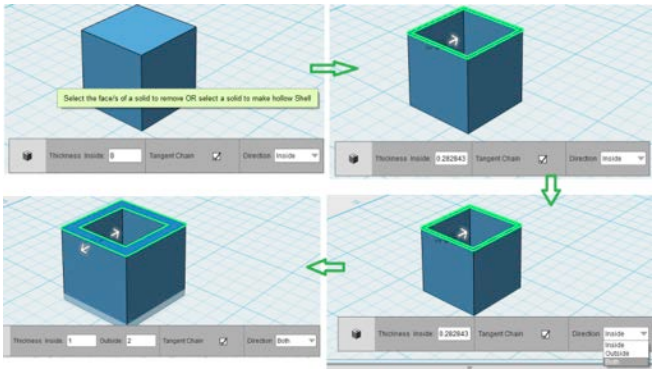
- Select Chamfer feature(C) on the main toolbar under 'Modify' menu
- Select an edge or multiple edges of a solid
- Input exact value in glyph or drag the Arrow manipulator
- Tangent chain result depends on the model. If your model has edges that are tangentially connected only then you will be able to see a visible effect on the model.
- If Tangent Chain option is checked on, then all the edges that are **tangentially connected**, will be selected at once. Otherwise you will need to individually select each edge.
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

7.6 How to use Split Solid feature



- Create 2 solids, for e.g. a Sphere and a Cone
- Select Split Solid feature (Alt+B) on the main toolbar under 'Modify' menu
- Click 'Body to Split', and then select the Sphere to be split.
- Click 'Splitting Entity', and then select the Cone as splitting tool
- Press 'Enter/Return' to execute the command or just click outside with the mouse.

7.7 How to use Shell feature

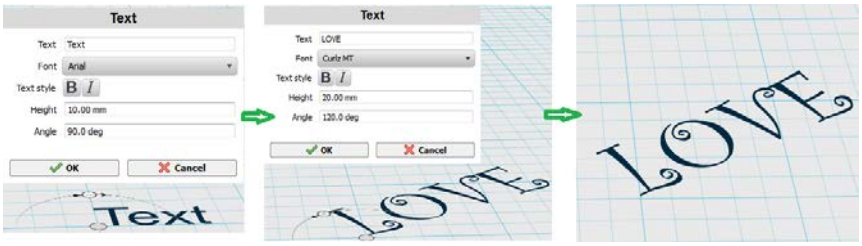


- Select Shell feature (J) on the main toolbar under 'Modify' menu
- Select the top face of solid
- Select 'Both' from the 'Direction' option
- Input the values of thickness for the 'Inside' arrow and the 'Outside' arrow, or just drag the Arrow manipulator
- Press 'Enter/Return' to execute the command or just click outside with the mouse

Note: A solid can be made hollow by using Shell.

8 How to use Text feature

Having a smart way of creating text was a huge request in 123D from the first version. The Text feature currently in 123D Design empowers you to not only create a text easily, but also to do some more sophisticated operations.

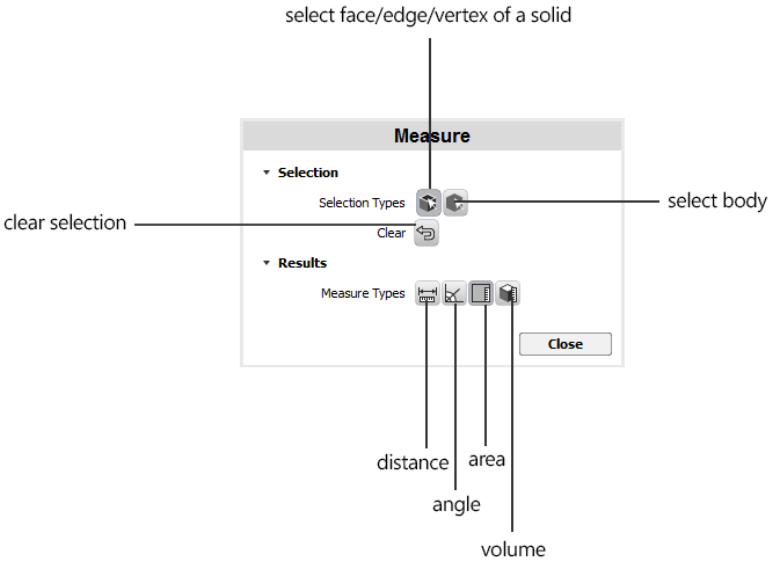


- Launch Text **T** feature (T) from the main toolbar
- Click on grid, sketch or solid face to add text. Specify text position and type the required text in the input box. Select font, text style, height and angle
- Click 'OK'.
- When you select the text on the grid, you will be able to see the contextual menu. Options like edit, move, extrude and explode will be available in the contextual menu.

The option to Explode the text will expose the sketch. At that point, you can work with each letter separately.

9 How to measure distances and sizes in 123D

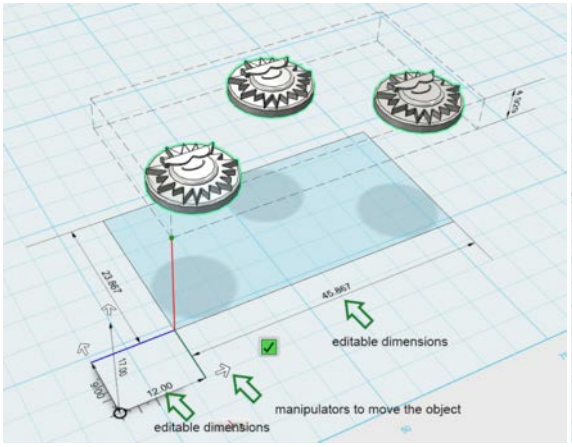
9.1 How to use Measure tool

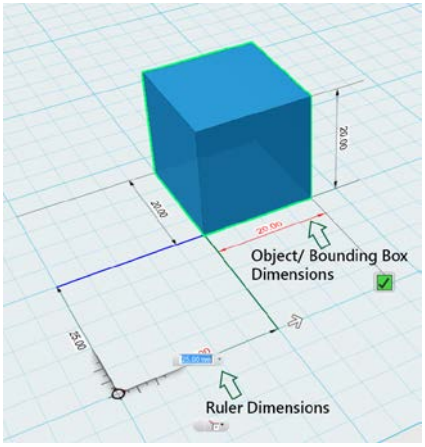


- Select Measure feature (Shift+M) on the main toolbar under main menu
- Select 2 different faces/edges or vertices of the same solid or 2 different solids.
- You will be able to see distance and angle between the 2 items.
- Clear the existing selection before you want to select the 1 or 2 different bodies.
- You will be able to see the results below.
- Click 'Close' to end the Measure feature.

9.2 How to use Ruler

The Ruler is a feature that 123D Design borrowed from Tinkercad, in an effort to converge the user experience in the areas where it made sense (like with Align, Smart Scale and overall look and feel). It does introduce some additional power, since the Ruler in 123D Design can also cruise along faces and snap to endpoints and midpoints of edges and center of faces. You can also have more granularity in selection of what to measure (minimum distance, midpoint or maximum distance).





- Select Ruler feature (R) on the main toolbar under 'Transform' menu
- Click on grid or solid/mesh to place the Ruler
- Click on solid/mesh/2D Shape to select
- Select Ruler type e.g. Midpoint from the glyph
- Click on Manipulator and drag it to move the selection along respective directions or input the exact values by clicking on the dimension arrows or digits.
- Press Ctrl to choose multiple objects.
- Press 'Enter/Return' to execute the command or just click the green ticker.

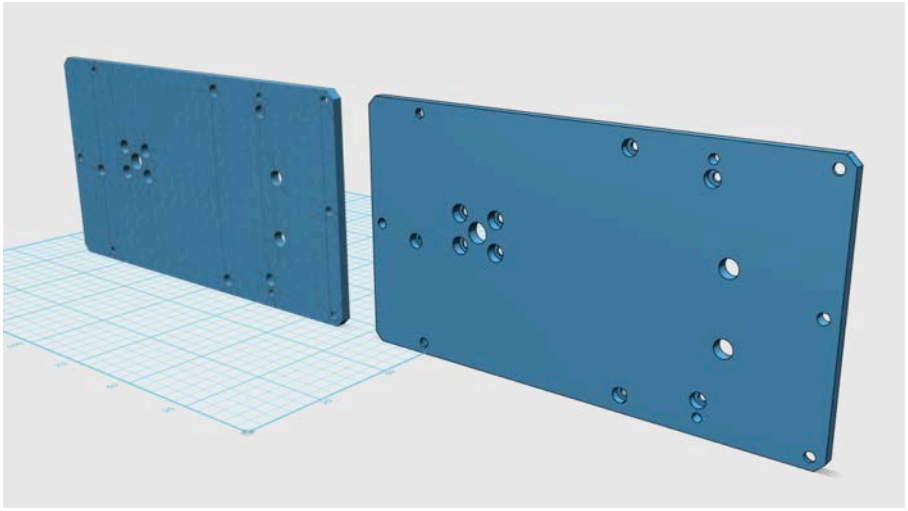
Note: You cannot edit dimensions of a sketch or a bounding box containing 'sketch and 3D object' in the Ruler command. But at the same time you can edit ruler dimensions.

Note: Press 'Ctrl'/'Cmd' to subtract selection.

10 How to convert a Mesh into Solid

123D Design 1.7 introduces an option called Convert to Solid, which takes a mesh and tries to optimize it for using solid modeling options after the conversion. For example, all triangles forming the face of an object that are coplanar will be optimized into a single face in the new solid. So this gives you the chance to then make a fillet or a chamfer to the edge between optimized faces. Sounds cool, right? Since you also have one single face, you can also extrude or press/pull that face, without ending up pulling only one triangle from the original mesh.

Remember that if you convert to solid a series of non-planar faces, you will end up with a solid with many triangles, which will lose the value of converting to solid in the first place. Meshes with organic shapes are a very bad idea if you want to use this command. Stay with meshes that have clearly been created from a CAD application.



- Select a Mesh that has planar or cylindrical surfaces.
- Click on the icon (M) from the Context Menu.
- If the part is complex, it will take a while to convert the Mesh into Solid.
- Once done, then perform any commands from the toolbar on the newly created Solid.

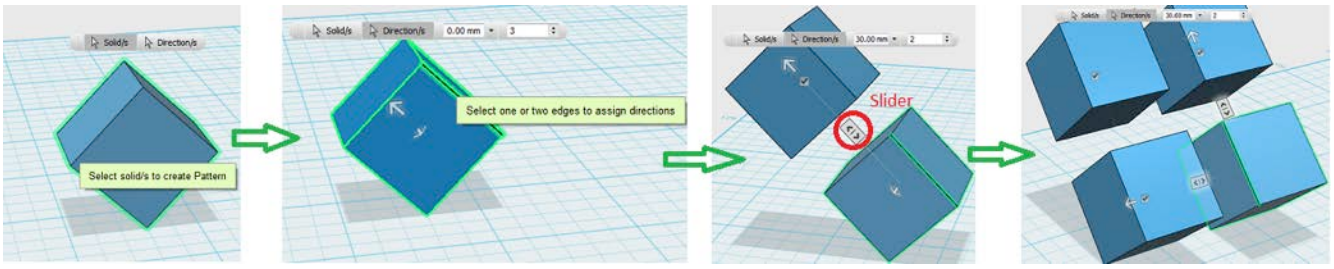
If the mesh is too complex, the conversion to solid may fail. When you make Booleans between solids and meshes, we are actually converting mesh to solid for making the Boolean, and then back to mesh (and you won't ever notice). If you don't need too much complexity on the mesh,

it is a good idea to reduce its complexity and make sure it's watertight and all normals are pointing in the right direction. You can do this in Meshmixer (some more details later).

11 How to create patterns

Patterns are a great mechanism for adding complexity on a model with very simple operations. You will only need an object of origin, and some auxiliary geometry (which can be an existing edge, or a sketch). Patterns can also be done on sketches, which can be very powerful too.

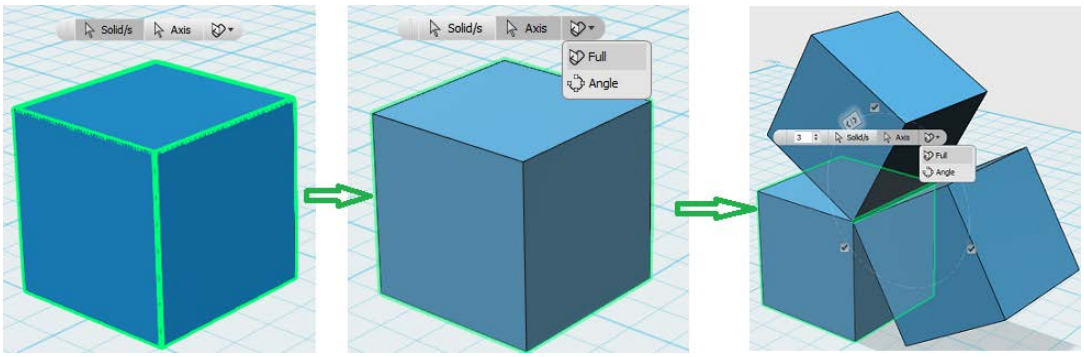
11.1 How to use Rectangular Pattern



- Select Rectangular Pattern feature (N) on the main toolbar under 'Pattern' menu
- Select a solid
- Click on 'Direction/s', and then select one or two edges to assign directions
- Input distance and number in glyph or drag the Arrow manipulator
- Select another direction, and input the distance and number in glyph or drag the Arrow manipulator
- Press 'Enter/Return' to execute the command or just click outside with the mouse

Note: Multiple selected solids also can be patterned at once. The number of patterned solids can be added/ subtracted by moving the slider. To hide a solid in a pattern, just uncheck the checkbox placed on it.

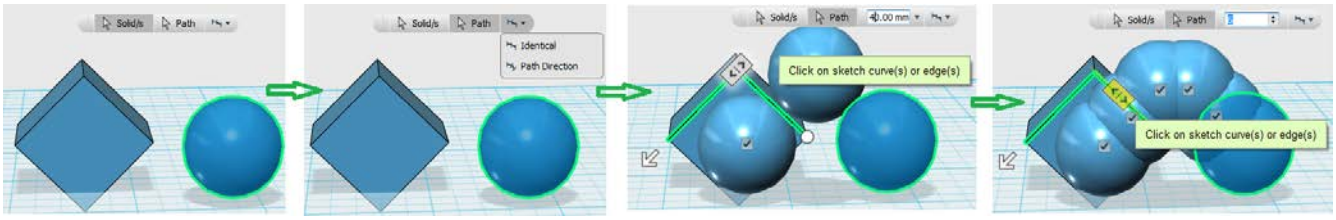
11.2 How to use Circular Pattern



- Select Circular Pattern (Shift+N) feature on the main toolbar under 'Pattern' menu
- Select a solid for e.g.: Box
- Click 'Axis', and then select circular type e.g. Full
- Select an edge of a solid or sketch, and then input number in glyph or drag the slider to set the number
- Press 'Enter/Return' to execute the command or just click outside with the mouse

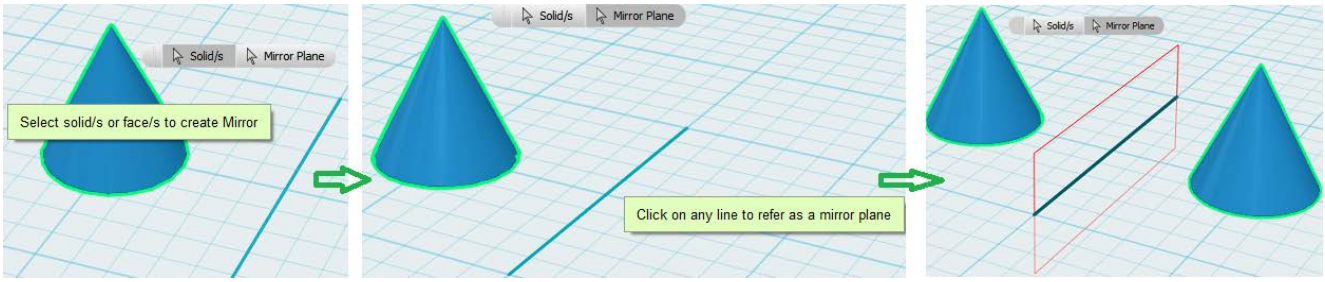
Note: Edge, face and line can be used to refer as axis. A face is selected, and then the normal of face is referred as axis. Line or edge is selected, and then it is used as axis.

11.3 How to use Path Pattern



- Select Path Pattern (Alt+N) feature on the main toolbar under 'Pattern' menu
- Select a solid
- Click 'Path', and select a path type e.g. Identical
- Select a path, e.g. the two edges of Box or a sketch, and then input the distance value or drag the Arrow manipulator
- Click the slider, and then input the number of solids or move the slider to set the number
- Press 'Enter/Return' to execute the command or just click outside with the mouse

11.4 How to Mirror solids



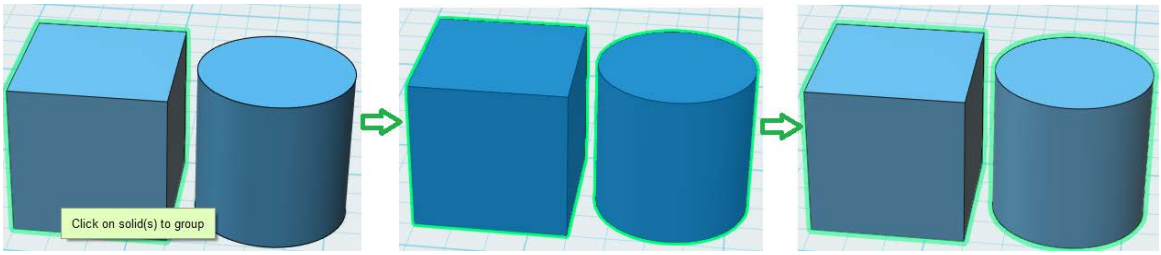
- Select Mirror feature on the main toolbar under 'Pattern' menu
- Select a solid e.g. Cone
- Click 'Mirror Plane'
- Select a line to refer to as a mirror plane
- Press 'Enter/Return' to execute the command or just click outside with the mouse

Note: A face also can be referred as a mirror plane.

12 How to group and combine objects

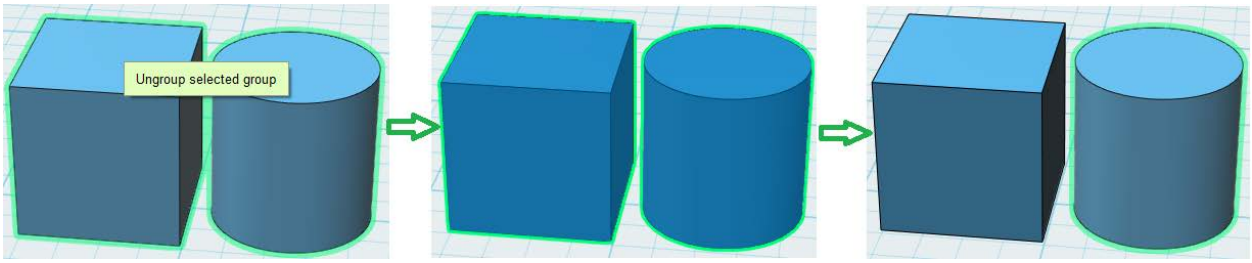
Grouping and combining are two very different operations.

12.1 How to Group solids



- Select Group feature (Ctrl+G/Cmd G) on the main toolbar under 'Grouping' menu
- Select multiple solids to group
- Press 'Enter/Return' to execute the command or just click outside with the mouse

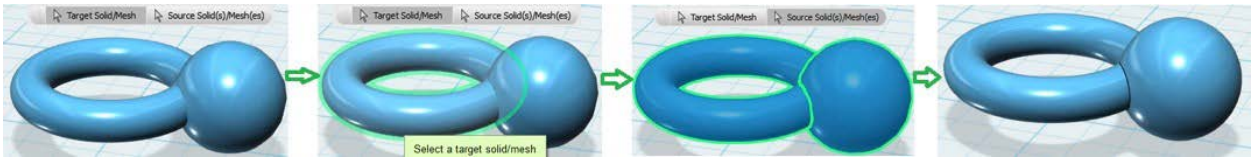
12.2 How to Ungroup solids



- Select Ungroup feature (Ctrl+Shift+G/Cmd+Shift+G) on the main toolbar under 'Grouping' menu
- Select the grouped solids.
- Press 'Enter/Return' to execute the command or just click outside with the mouse

Note: 'Ungroup All' command is similar to 'Ungroup'. 'Ungroup all' command ungroups all the sub groups in a selected group.

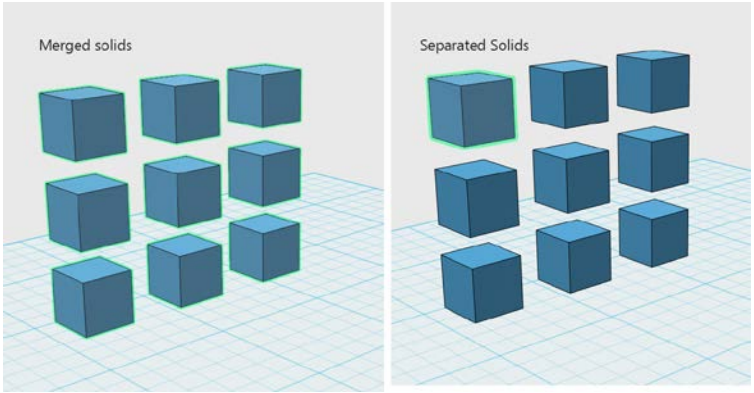
12.3 How to Combine objects



- Select Merge feature([) on the main toolbar under 'Combine' menu
- Select a target solid/mesh
- Select a source solid/mesh
- You can select and merge solids/meshes that are not intersecting. They will still combine together to form a single object.
- Press 'Enter/Return' to execute the command or just click outside with the mouse

Note: Follow similar steps for 'Subtract'(]) and 'Intersect'(\) commands

12.4 How to Separate objects



Separate feature works only on solids. If there are solids that are merged together and are not intersecting, you can separate them with this feature.

- Select a solid that has multiple parts (lumps) in them.
- Select the Separate feature (Shift+P) from the toolbar under Combine.
  The solids get separated as soon as the command is initiated.

13 How to import geometry into 123D Design

There are several ways of getting geometry into 123D Design:

- Solids (as SAT, STEP, SMB, 123DX, DWG, DXF)
- Meshes (as STL, OBJ)
- Sketches (as SVG)

We already know what to do with solids and meshes. Let's take a look at SVG files.

13.1 Importing SVG files

SVG files stand for Scalable Vector Graphics. You can get a lot of these in the Internet, although not all will work well. You need SVG files that have been created by vectors. There are several applications that can work with vectors, from very professional to quite amateur.

Once you get an SVG file, you also need to understand how it will work in 123D Design. You can import the SVG as a sketch, and then use it for the different commands we covered, or import them as solids. In this case 123D Design will extrude them. If the SVG file has several profiles containing islands (profiles inside profiles), you will get separate extrusions for each profile. You can then choose to keep the ones you want. Some other apps (like Tinkercad) add some logic where they try to detect internal islands, but this logic can fail depending on the SVG, so for now we chose to extrude all profiles and allow you to select the ones you don't want, and delete them.

This is a simple SVG extruded as a solid.

This SVG had internal islands (the letters), which were deleted.

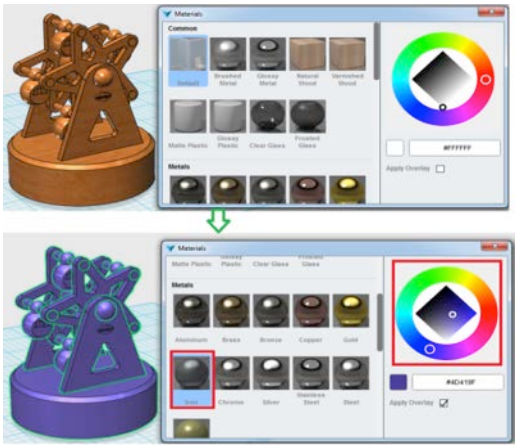
14 How to add Material to an object

You can apply materials per object in 123D Design. This means you can't apply materials to individual faces in a model. This has a lot of sense in a rendering application, but not much when thinking about how to manufacture an object. In that case, materials per object make more sense.

If you merge two objects with different materials, they will adopt the material of the first selection.

123D Design offers materials that you may eventually use for fabrication (3D printing or CNC, mostly). We offer some types, and then you can add a color overlay in order to achieve the right color.

When you are working with materials, please make sure that the Apply Overlay option below the color wheel is not checked (unless you want to override the color of the preview you can see on the thumbnails at the left of the color wheel). In many cases it may default to a blue tint on top of every material, so it is important to understand the reason.



- Select solid/s or mesh/es
- Launch Material feature (`) from the main toolbar
- Select a material e.g. Iron
- Select a color in the color disk
- Tick the 'Apply Overlay' checkbox. The material gets applied to the selected objects
- Click outside with the mouse or click the 'Close' button in window Material to exit the command.

15 How to export from 123D Design

123D Design has a quite complete set of export formats. Let's take a look at them and understand which one we would use for which reasons.

- Export 3D solid formats: DWG, DXF, 123DX, SAT, STEP (from Export Selection only)
- Export 3D mesh formats: STL, OBJ, VRML, X3D
- Export 2D vector formats: SVG, DWG, DXF

15.1 Export 3D solid formats

The main reason you may want to use solid formats is when you either want to share or use that file for certain manufacturing methods (molding would be one of them, but could also be CNC), or when you know that this file may get used on professional mechanical cad applications in order to add more details.

SAT format is a great format for exchange with other mechanical cad applications (like Inventor or Fusion 360).

STEP is mostly sought when you need to take your model into manufacturing, and especially when molding is involved. It contains similar info than SAT, and is created according to ISO standards.

When you create a DWG layout, you also get a 3D **DWG** file in the Model Space (if you open it in AutoCAD), so you can also see this feature as a way of getting not only documentation but also getting the 3d model. Once in AutoCAD, you can make more complete documentation out of your 3D model by using the Layout tools, or simply add dimensions to the sheets created with Create 2D Layout, which you can find in AutoCAD's Paper Space (also known as Layout). Now you can export solids as **DWG/DXF** locally. This can be done in the offline mode.. Make sure that you have only solids while exporting using this file format. Meshes won't get exported with it.

You can also export selections as 123DX files. This can allow you to then insert a part of a model into another model.

15.2 Export 3D mesh formats

There are several reasons for exporting as a mesh format. You may want to get your model into 123D Make for slicing and laser cutting, or import it into 3ds Max or Maya for rendering, or into Meshmixer or Mudbox for sculpting.

STL files are the most common until now when thinking about 3D printing. This format only describes the surface geometry of the object, and can't store properties like color or texture. This is generally ok when printing on one color, since you normally set the material on the 3D printing software. But if you plan to use several colors or textures (there are some 3D printers that support these scenarios), then STL won't be the right format. You can find both binary and ASCII representations of STL. 123D Design writes Binary version by default. But while exporting you get an option to export it as ASCII. STL files have no units, so you also need to make sure that you get the right size when exporting into other applications. There is a standard around assuming STL files in mm, and this is what 123D Design does. So a 2x2x2cm cube will be converted into a 20x20x20 STL. Since most applications will tend to read the STL file as coming in mm, this object will show up as a 20x20x20mm cube.

OBJ files represent geometry and also add UV position of textures. This is the format we use for saving 3d models captured with 123D Catch, or sculpted in 123D Sculpt+. That said, 123D Design won't support textures in a model. OBJ also lack units, but has similar conventions as STL for solving this issue.

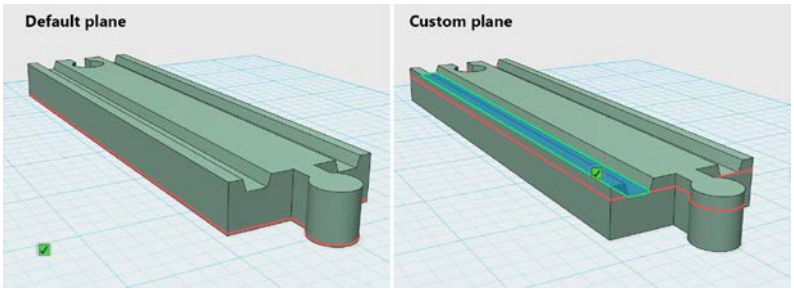
VRML and **X3D** are formats that store color. VRML was created with web in mind, and X3D is its evolution. Some printers that can print color will accept these formats.

15.3 Export 2D vector formats

123D Design has three ways of exporting 2D formats.

There are 2 ways of exporting DWG file. One is by choosing 'Create 2D Layout' (in cloud) that is explained later in this section and second is by exporting locally. Let's see how the offline option works.

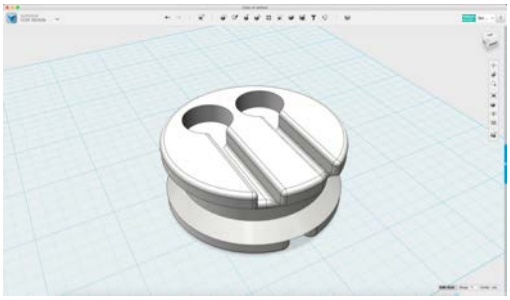
Select **DWG/DXF** from the 'Export as 2D' option in the application menu. By default it will create a cross section of the objects positioned on the grid. You can change the plane by clicking on any other planar face of a solid. If you want to select/deselect the plane you need to press 'Ctrl'. Once you feel comfortable with your selection you can then save the cross section by clicking on the tick mark and saving the file.

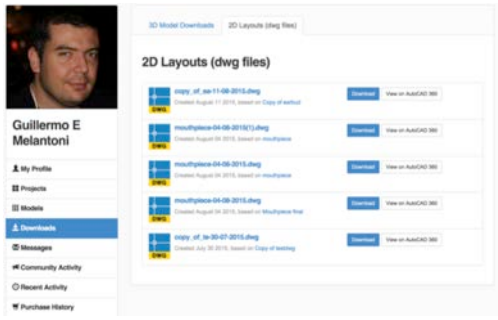


In the same way you can export a 2D file as **SVG**. Follow the steps above and save the file as SVG. You can use the SVG file as an input for a laser cutter, or even for CNC. You can also take an SVG file into other applications like Tinkercad.

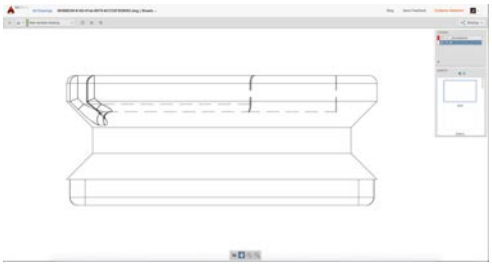
DWG files can be created using 'Create 2D Layout'. 123D Design will create on DWG file containing the 3D model in Model Space and a series of sheets on Layout or Paper Space. You can then open this in AutoCAD, AutoCAD 360, or any other application that can read a DWG file.

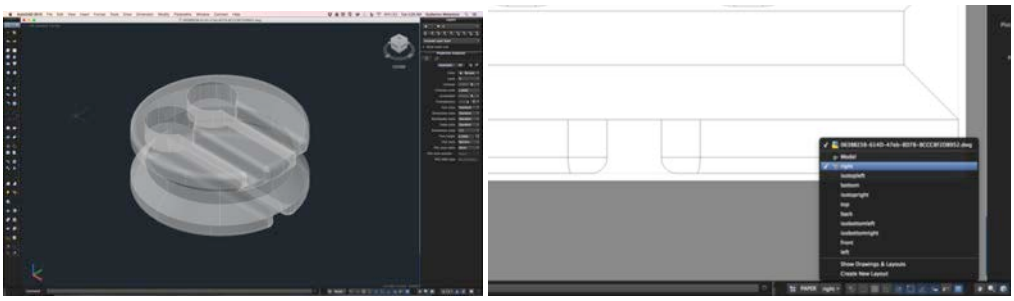
You will need to first save the model into My Projects, since the conversion to DWG happens in the cloud. So once you save the file, you can use the option Create 2D Layout, and you will get a notification when the conversion is ready (via email). Then you can go to www.123dapp.com into your profile, and get the DWG file under Downloads > 2D Layout.


Model in 123D Design ready for Create 2D Layout



This is the location where you will get the DWG


This is how it looks when opened through AutoCAD 360



You can also download the file and open it in AutoCAD. Here you can see both Model Space with the 3D model, and one of the sheets with the layout. If you check on the sheets, you have one per view (Top, Left, Front, Bottom, Right, Back), and four isometric views.

16 Working with the entire 123D ecosystem

123D Design is a very powerful application, and it gets even better when working with the rest of the apps in the ecosystem.

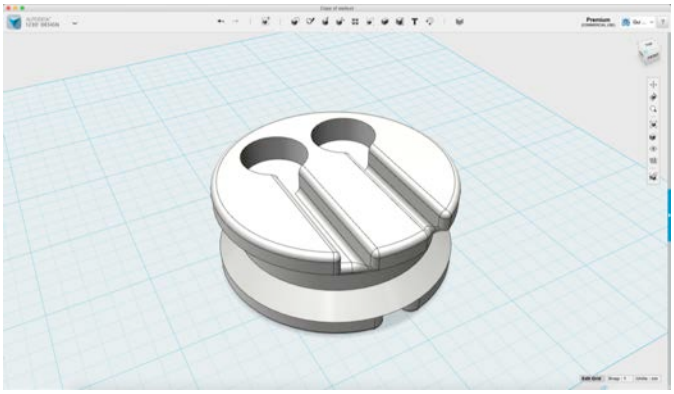
Let's talk a little bit about how to interoperate with some of these apps.

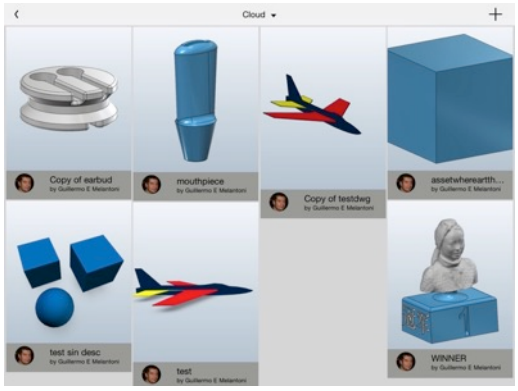
16.1 123D Design (desktop) and 123D Design iOS

You may be aware that apart from the desktop 123D Design app, we also have an iOS version of this product. The iOS version has a limited set of features, and can only open solids, but if you are ok with these constraints, it is a great standalone tool and also a perfect companion for the desktop version.

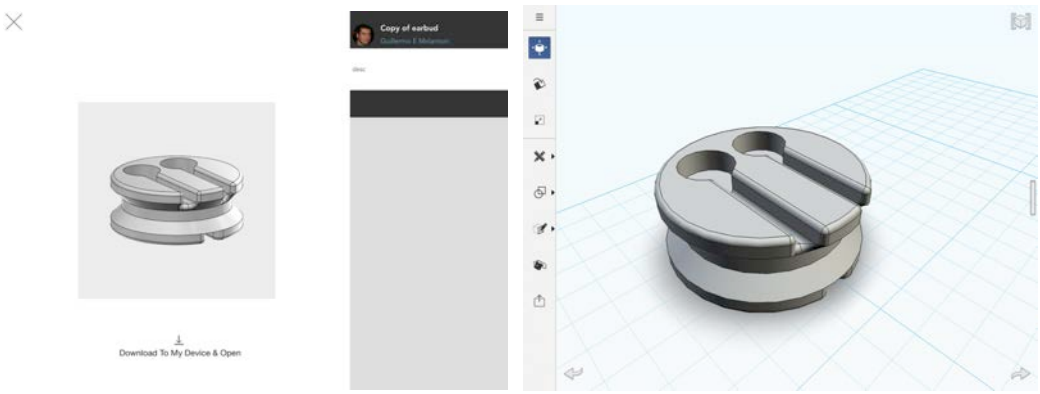
You can start work while you only have an iPad with you, save to My Projects, and continue working on 123D Design desktop. Or open what you did on the desktop app in your tablet and polish the design.

One particularly interesting feature in 123D Design iOS is the chance to make a composite between the model and a photograph. This is a particularly effective way to show your design in context, without the need to spend too much time on image editing software. The positioning of the model in the picture is done very intuitively, using same interactions than when moving and orbiting a model on the iPad. And you can also apply filters to make the image more interesting.

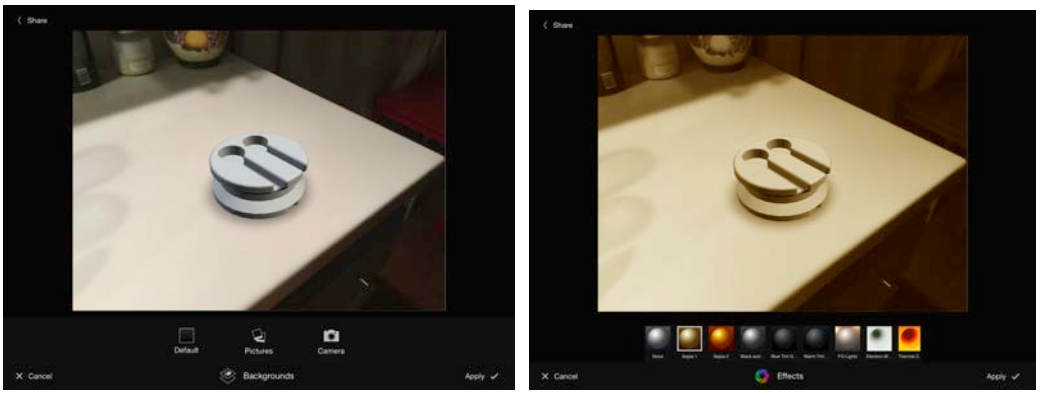




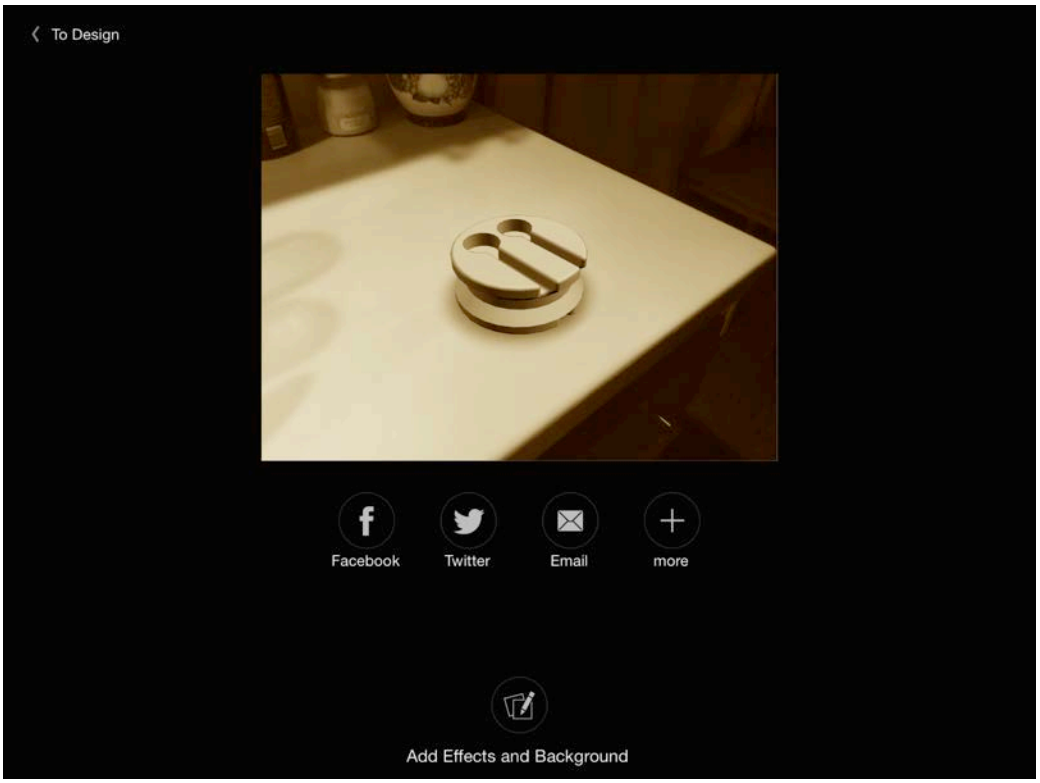
This model was built in 123D design desktop, and after getting saved to My Projects, you can see it on your gallery of projects in Design iOS.



You can check the file and then decide to download from My Projects into the iPad, and start work on it. As you can see, look and feel is familiar between both apps.



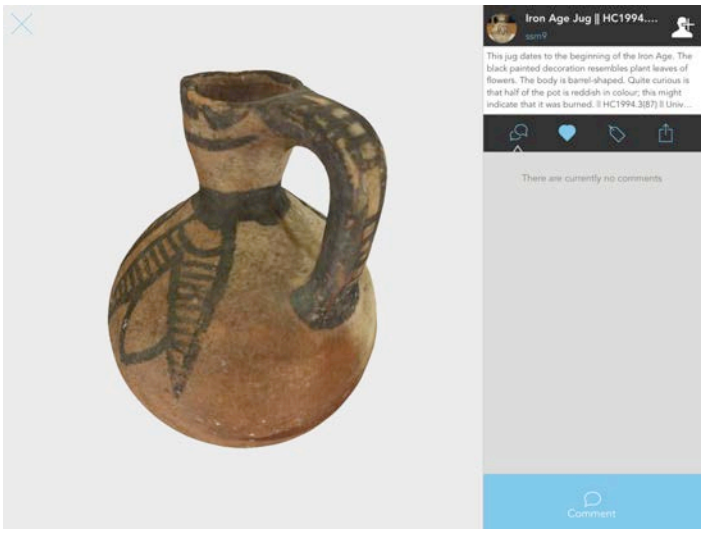
The Share option allows you to compose the model in real time against a background. In this case, it's a picture using the iPad's camera. Then you can add filters to make the image look more interesting.



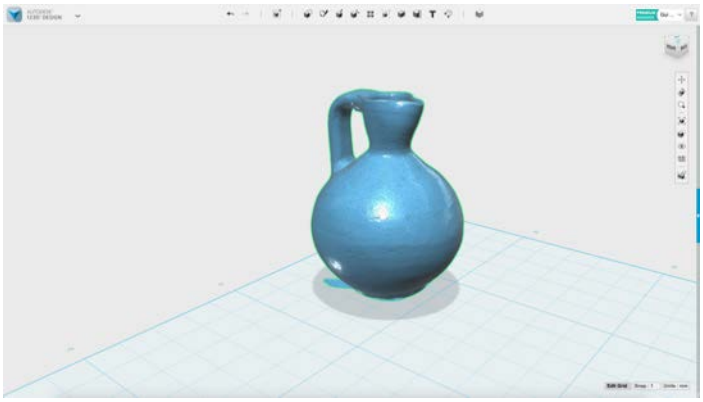
After you are done, the image is ready for sharing either through social media or via email.

16.2 123D Design and 123D Catch

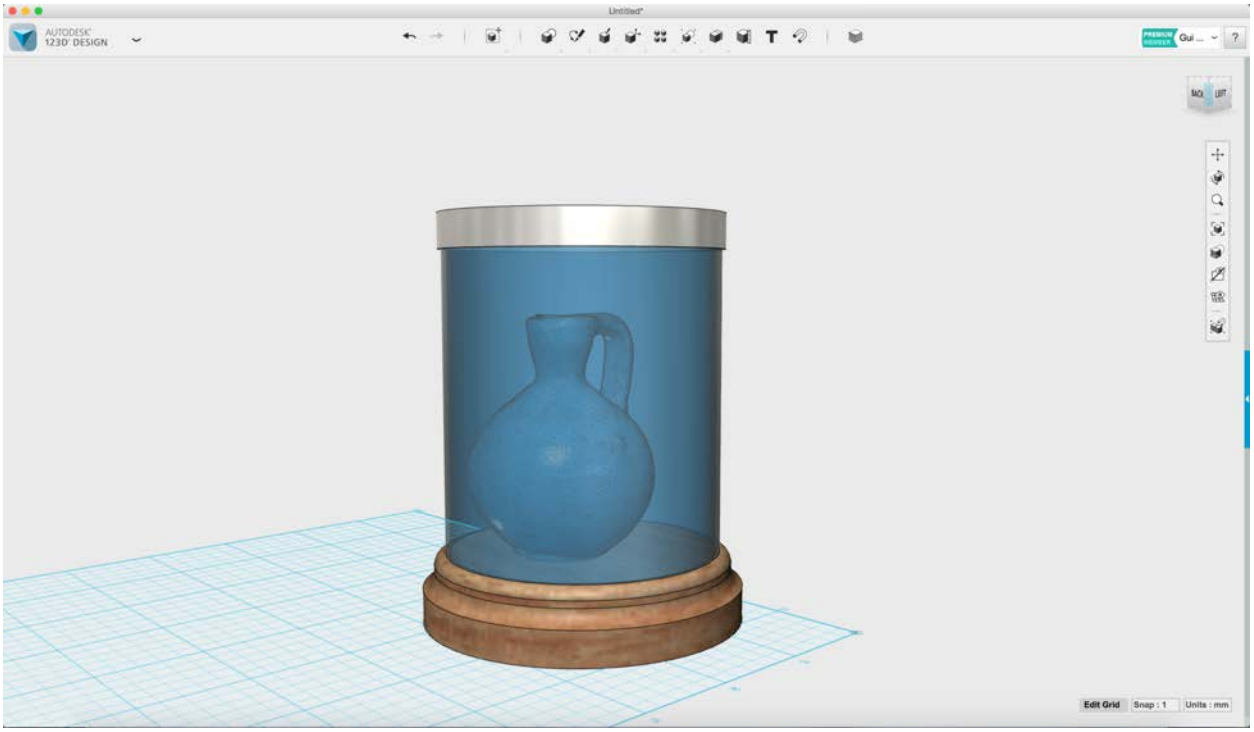
123D Catch was the first of the 123D apps to be released. It allows you to get a 3D model out of a series of pictures. 123D Catch is available for all mobile platforms, and is also a great app in itself and an amazing companion for 123D Design. You may want to capture something and then use it as reference for a design, or actually work on the mesh itself. As a reminder, in order to do Boolean operations with meshes, you will need to make sure that these meshes are watertight. 123D Catch will never produce a watertight mesh since there's always part of the model that won't be captured (for example the part that's against the floor/table). You will want to use Meshmixer to fix this (more on that later).



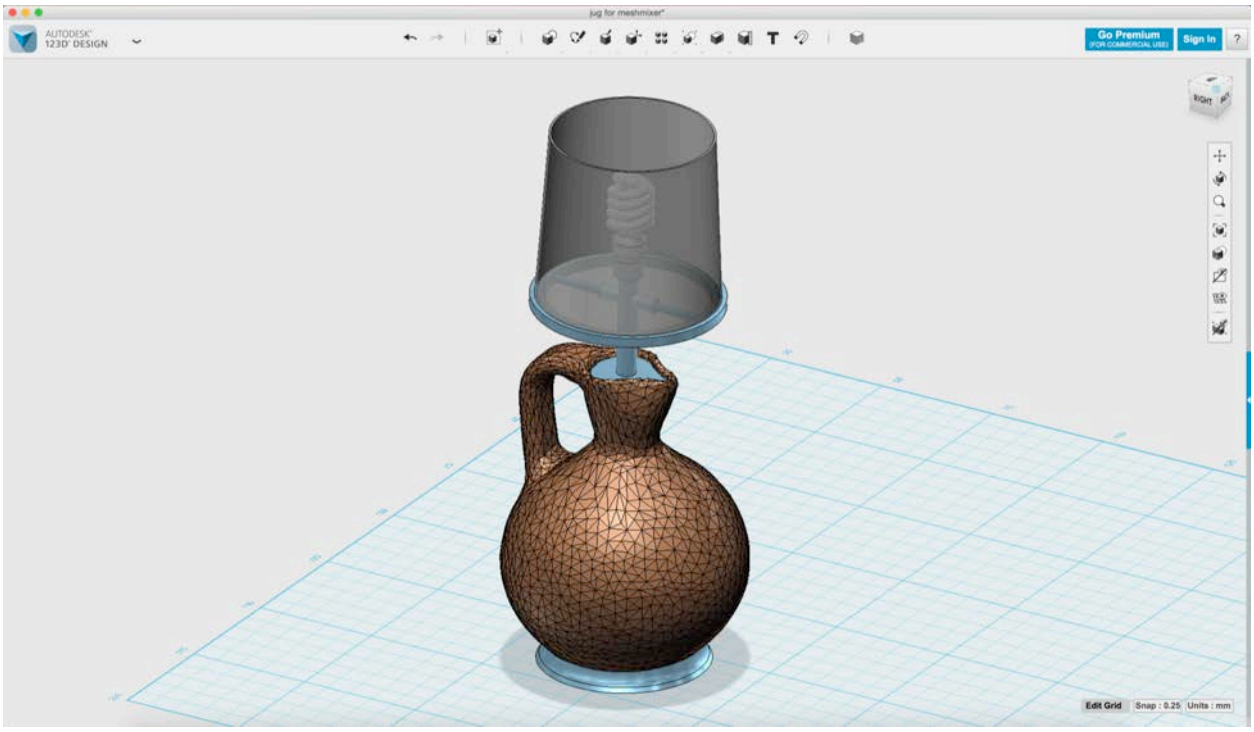
This is a capture of a jug made with 123D Catch. I can get this mesh into 123D Design using My Projects (since all 123D apps save to the same repository) and then work on my design.



Textures are not represented in 123D Design, so the material you see in the image above is the default representation of a mesh in 123D Design. You can assign material for the whole mesh with the Materials feature.



In the image above you can see the mesh taken as a reference for a design. If you end up with a design like this, you can then use Create 2D layout to send documentation for a wood shop and a metal shop, and also order the glass. Or if you can get a CNC, you will be able to build the base quite easily.



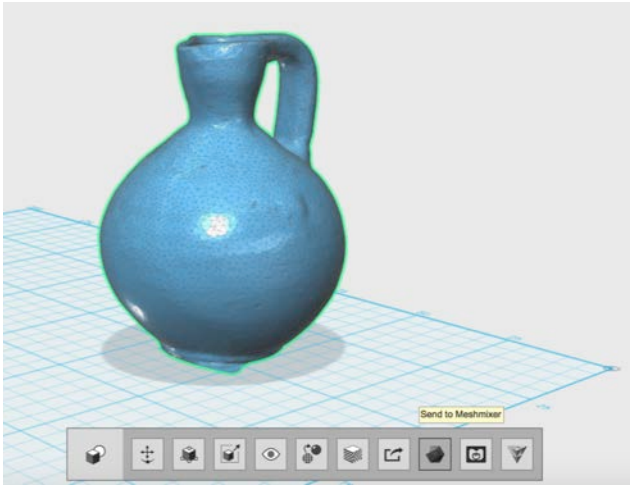
The example above shows the mesh converted to solid, simply because the project required some manipulation and interaction with other parts, in order to convert this jug into a lamp.

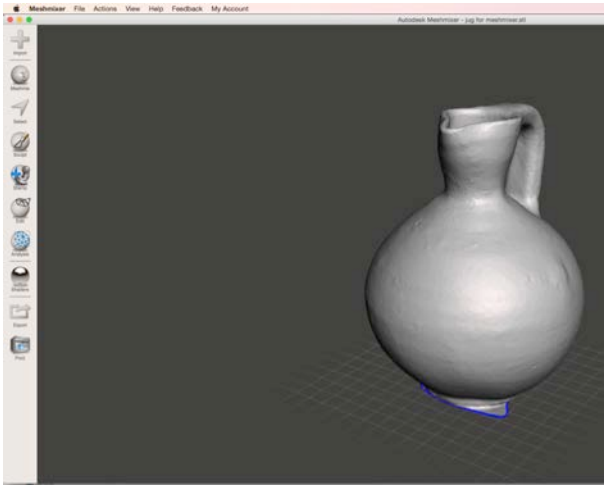
16.3 123D Design and Meshmixer

Meshmixer is a great application for editing meshes (sculpting, mash ups, patterning, etc) and preparation for 3D printing. There is a lot you can do between 123D Design and Meshmixer. And one of the advantages is that you don't need to even think about exchange formats. Let's see an example of what can be achieved with Meshmixer and 123D Design together. The lamp that came from Catch had a hole against the floor, and was also too dense. If we want to do conversion to solid, then it's critical to have a healthy mesh (watertight and no issues with normals)

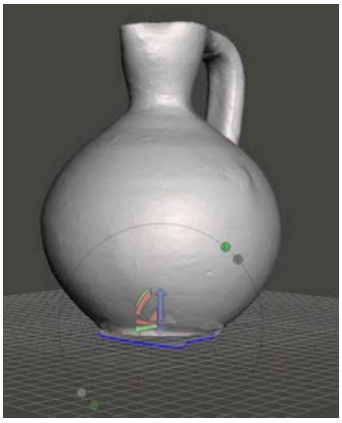
When you select an object in 123D Design, you get several options on the context menu at the bottom. One of them is Send to Meshmixer.

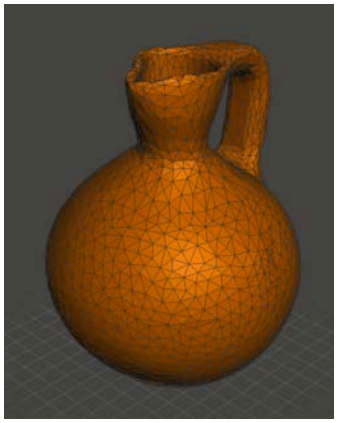
Once you click here, you will be redirected to Meshmixer (provided it has already been installed) and the model will automatically open in Meshmixer. Once you finish everything you need, simply Export from Meshmixer and Import back into 123D Design. It will show up in the exact same location, provided you have not modified its location in Meshmixer.



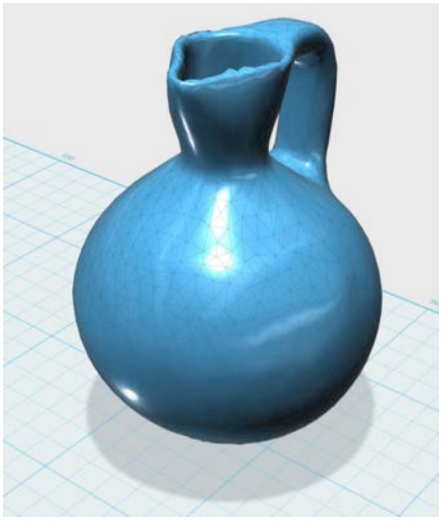


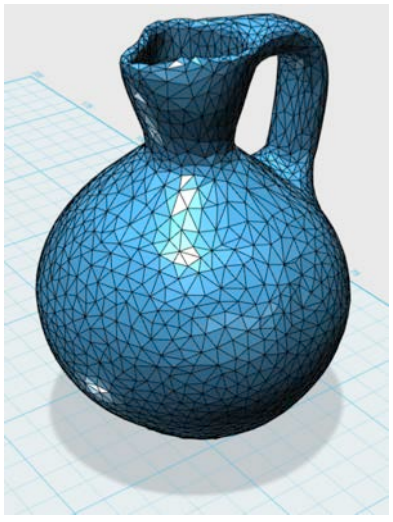
You can do several operations in Meshmixer. In this case, we'll cut the bottom to get a flat surface, by using Plane Cut. We'll also Remesh the model in order to get a better structure, and Reduce mesh, so it's easier to manipulate in 123D Design if we want to convert into solid (in order to make Boolean operations there).

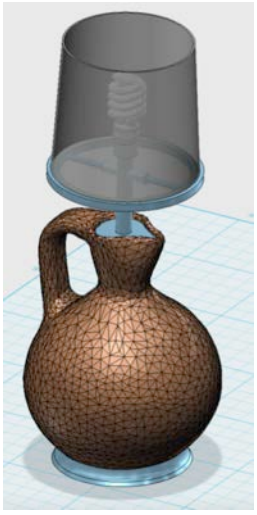




Now we are ready to take it back into 123D Design. After some modeling, you can get an interesting new model based on something you extracted from a picture.

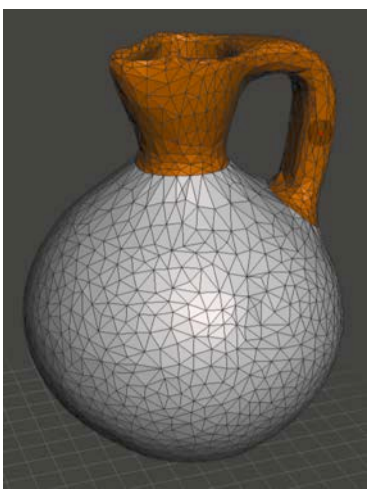
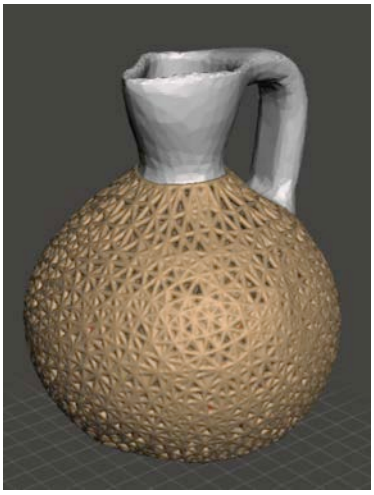




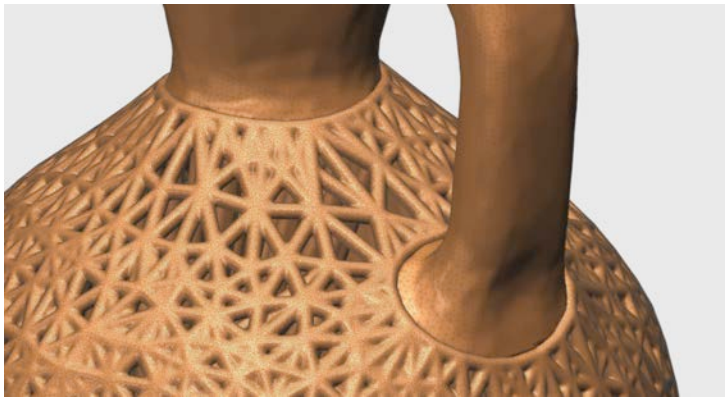


Let's take it back into Meshmixer for some more fun. The jug still looks a lot like the original, and we may like to deconstruct it a little more.

You can select the part of the jug you may want to work on, and then apply some patterns to it. This is how the process looks like. For more information on how to work in Meshmixer, you can check within Meshmixer app help.

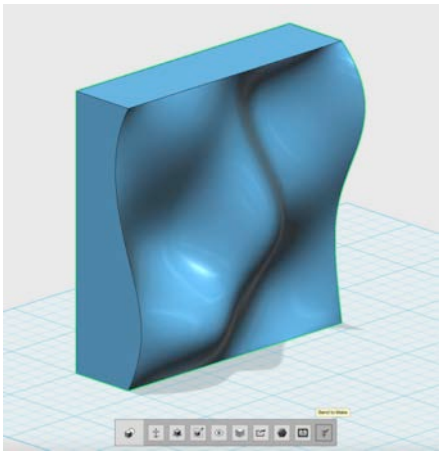
Pattern applied in Meshmixer



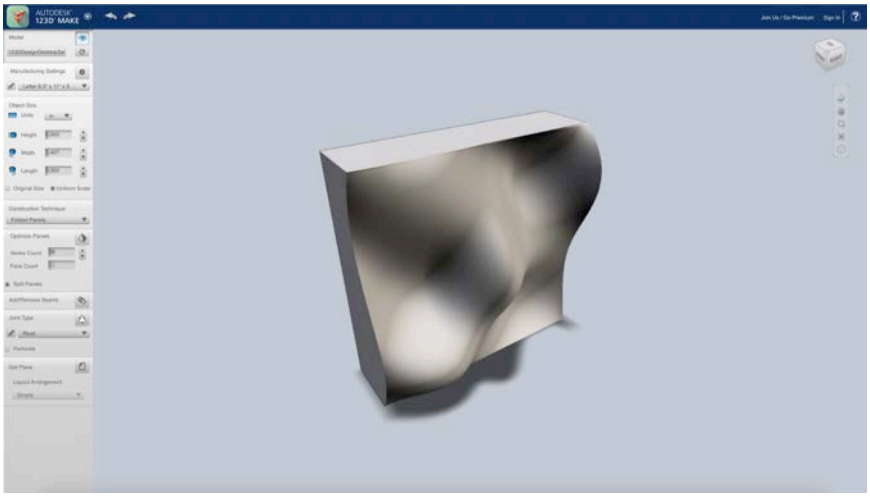
Final details in 123D Design

16.4 123D Design and 123D Make

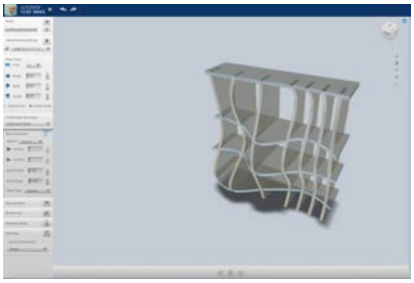
123D Design and 123D Make are also connected directly through the context menu (for selections) and from the Application Menu (for whole model). 123D Make will provide you tools for slicing model into parts that can be laser cut, water jet cut, manufactured through a CNC, and even printed in a traditional 2D printer in order to get a mold you can use for cutting manually. 123D Make can also panelize models, which can end up in a 3d print, or cutting of individual panels.

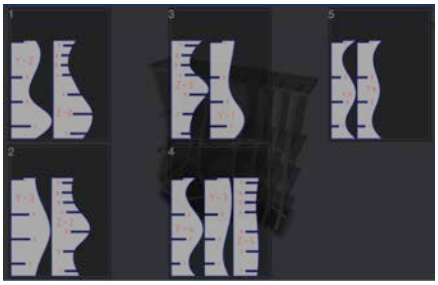


After sending the model into 123D Make, you need to define the Construction Technique and the material to use.

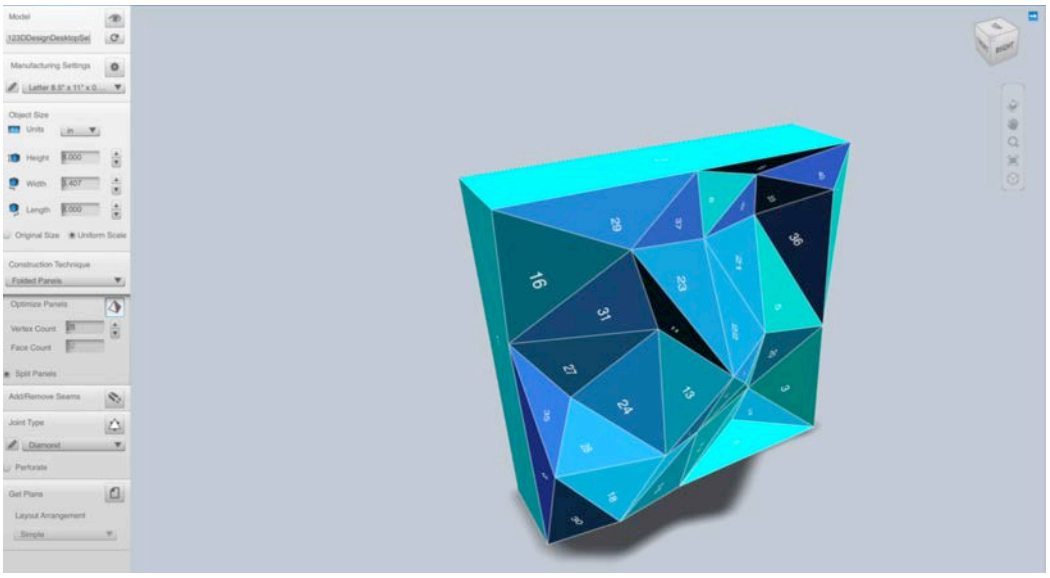


Interlocked Slices will provide a result like the one in the image below. This could be the starting point for an interesting bookshelf. 123D Make will also solve the joints and also allow you to tweak these slices manually. As soon as you are done, you will have the chance to export the 2D linework that can be leveraged by several cutters.





Panelization is another interesting process that can provide a series of panels along with different seams. You can use the same model to build that fancy boulder for climbing on your backyard!



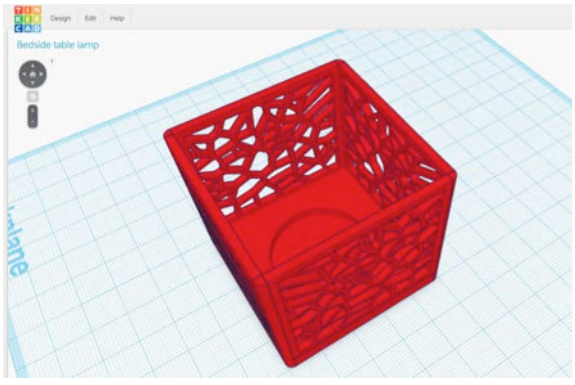
16.5 123D Design and Tinkercad

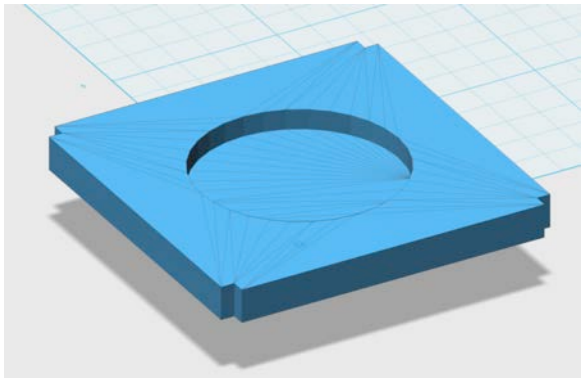
123D Design and Tinkercad are converging in a lot of different aspects. Any interaction where it makes sense to be similar and we are not limited by different technologies will be converging more and more. For example, Smart Scale adopted a similar behavior than the one you use in Tinkercad for scaling. Another example is Align, or the Ruler, or the export formats. Of course there are also divergences, most of them because of the additional power of 123D Design and the fact that Tinkercad works exclusively with meshes and 123D Design works with both meshes and solids.

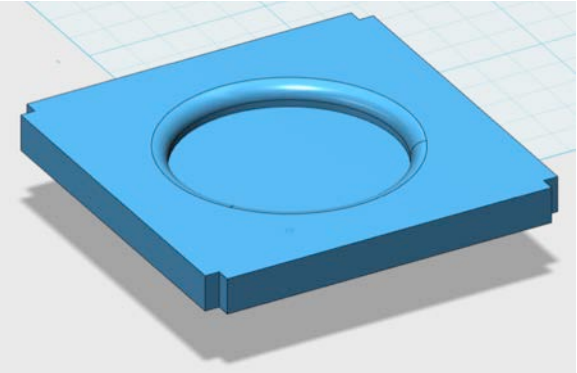
If you want any model or parts of models from 123D Design to be used in Tinkercad, simply export an STL, and import it into Tinkercad.

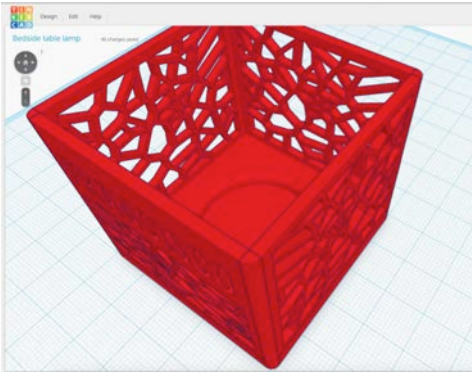
Or if you want a part from Tinkercad to be continued in 123D Design, export STL and import into 123D Design. Depending on the part, if you convert to solid, you can actually get a part you can do pretty powerful editing on.

In the example below, we start with an existing model in Tinkercad, which is using the app to its full potential with a Voronoi shape generator on the sides of this bedside lamp. We now want to have a smooth edge on the place where you put the light, and smoothing an edge is not something Tinkercad will do that easily. So you can export selection into 123D Design and use Convert to Solid. Then you can fillet the edge and export selection into Tinkercad. Use Align in Tinkercad in order to have the right placement, and you just added some extra interest to your Tinkercad model thanks to 123D Design!









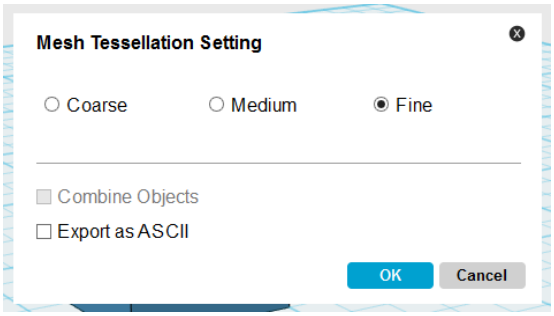
17 3D printing from 123D

This section doesn't try to be an in depth explanation around how to 3D print, but simply how to get from 123D Design into a 3D printer or a 3D printing service bureau.

17.1 Exporting for 3D printing

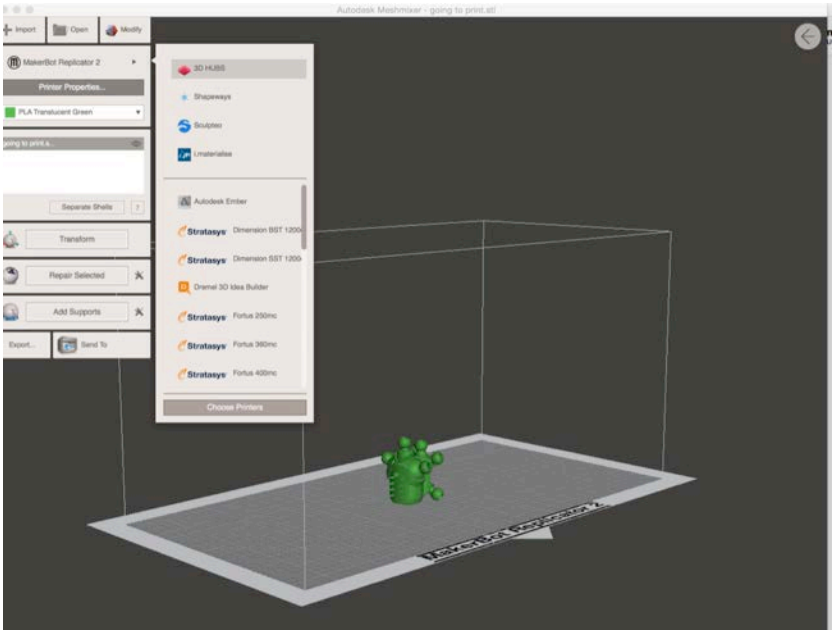
We already explained all the formats supported in 123D Design. Let's get into some of the options when exporting STL files.

You will see three presets for resolution of the STL file (which will impact on the size of the file) and there's also an option for combining objects. Some application that import STL files will only read the first body in an STL file. If you are exporting several objects in a single STL, you may find applications that only display one if you don't select Combine elements. That said, if you want to keep some level of control and maybe you want to assign different materials to different bodies within an STL file, then the answer is not to combine, since you want to access each object. STL files can be classified into 2 categories ASCII (text) and Binary. If 'Export as ASCII' is not checked then by default the file is exported as Binary STL.

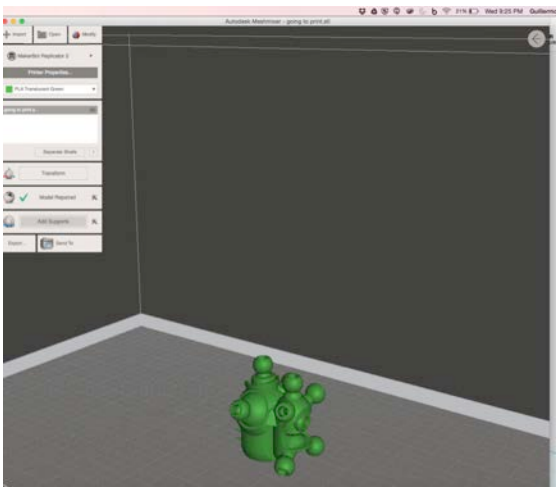


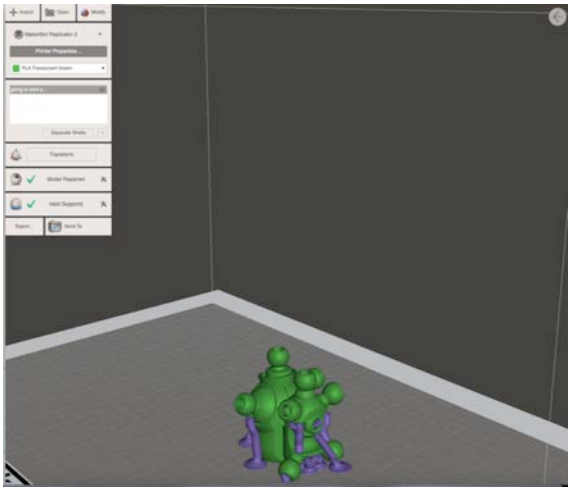
17.2 Using Meshmixer for 3D printing

Meshmixer is not only a great application for editing and playing around with meshes. It also provides some great options for 3D printing preparation. Preparation can imply from healing meshes to adding supports. You can send directly from the canvas or even from a selection into Meshmixer's 3D printing menu, and then decide if you want to print locally or use a service bureau.



In any case, you will need to make sure that the model is healed and that it has support material in order to prevent collapse from overhanging parts.

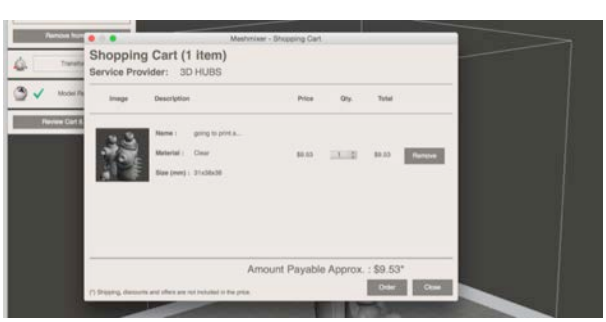


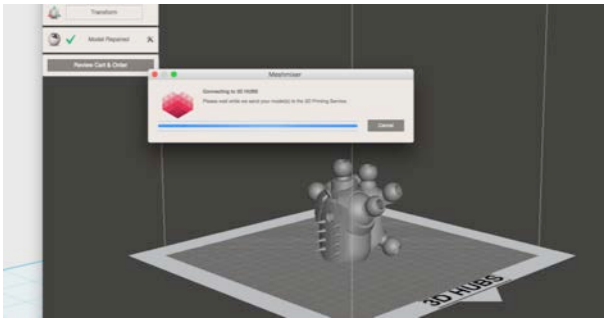


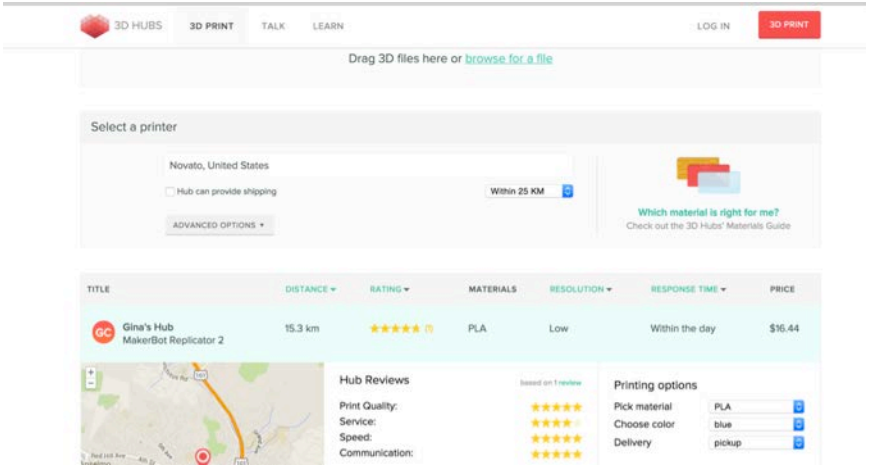
Once you are done, you can either print locally or use a service bureau. If you print locally, depending on the printer, we can send directly to the printer or at least to their printing software, and in other cases you will need to export the STL from Meshmixer and open it on your printer's software.

When you print through a service bureau, you will get an estimate of the printing price based on the volume of material. Once you are done with the parts you want to print, you will go to the shopping cart and start the process. Depending on the service, you may get a request for sign in immediately, or only when you have decided to print and need to pay.

The example below shows 3D Hubs integration. When the file gets uploaded to their site, they may already track your location and offer you users that own printers around your location. Then it's just a matter of defining material and start the process.

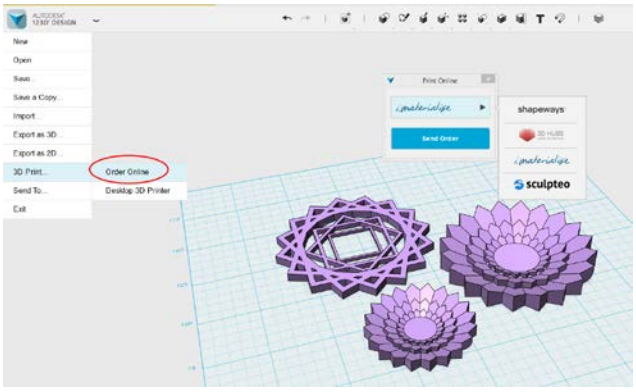




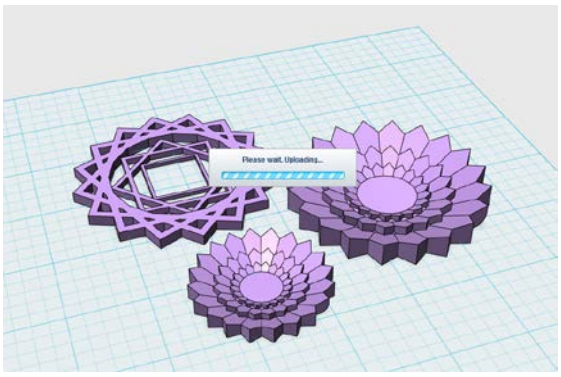


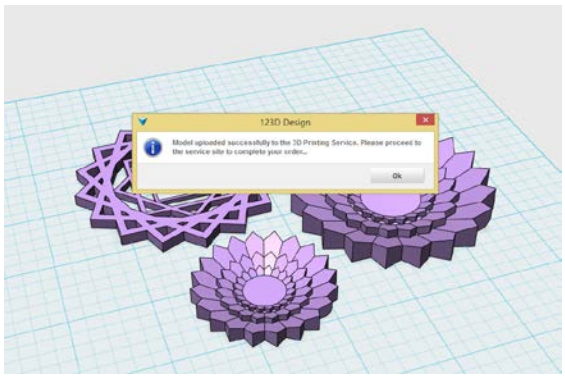
17.3 Sending to 3D Print Bureaus directly from 123D Design

It is easy to send your model to 3D Print Service Bureaus directly from 123D Design. Once you have created a model in the application, select 'Order Online' under '3D Print' from the Application Menu. You will see a small window where you need to select the Service Bureau of your choice from the dropdown. Your choice will stay default till you select another option.



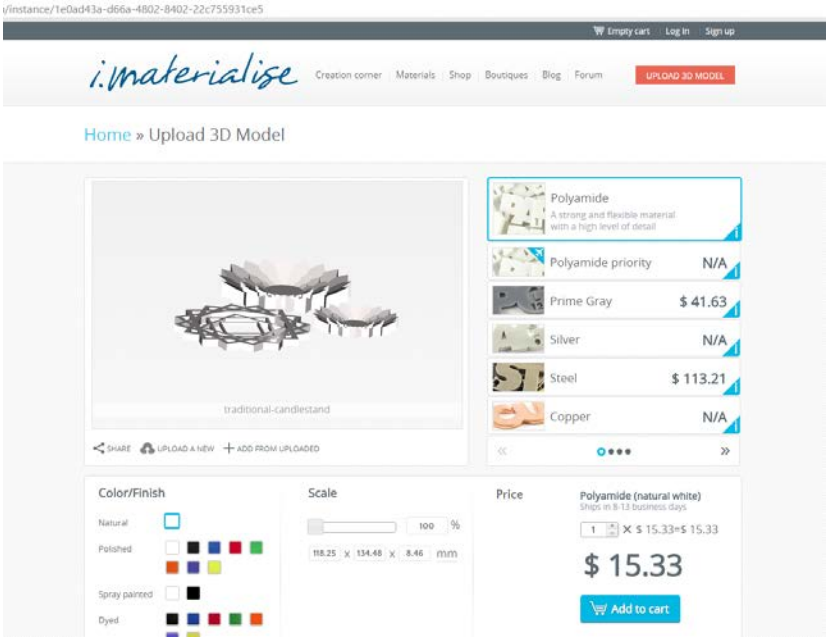
Click 'send order' to send the model in the scene to Service Bureaus' website. You should be signed in to send your model for 3D printing.





If you want to send just a part of your model and not the whole model to the Service Bureau, then the best way to do that is to hide parts that you don't want to send for printing. Hidden parts won't be sent to 3D Printing.

File will be sent to the selected Service Bureau, imaterialise in this case below.



18 Surface Pro 3 and 4 support for 123Design 2.0.

Here is the good news! Now you can install Design 2.0 on Surface Pro 3 (recommended – Models having Intel Core i5 and i7 processors with 150% default display scaling) and Surface Pro 4. Install normally as you would install on your desktop. Use keyboard to enhance the quality of interactions within the application.