CP11801

3D Printing with Inventor

Diego Valdes
Mechanical Engineer

Learning Objectives

- Learn how to identify what methods of rapid prototyping exist today (FDM, SLS, SLA)
- Learn how to identify what type of geometry is best suited for 3D printing
- Learn how to orient models in space for printing and subdivide the model if necessary
- Learn how to prepare your model with Print studio, by adding support material and repairing geometry

Description

This class will guide the student through the process of optimizing a few example models to be ready for 3D printing using Inventor software and its 3D printing tools.

Your AU Experts

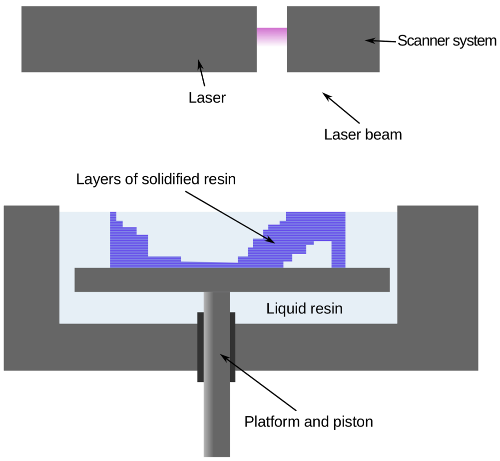
Diego Valdes is an aerospace engineer with 8 years of industry experience. Outside of work he is an avid maker, specializing in television and movie prop replicas as well as cosplay and costuming. Over the years he has made several costumes and props, with different fabrication techniques, such as hand sculpting, pepakura, 3D printing, and woodworking.

3D printing or Additive manufacturing (AM)

SLS SLA FDM Polyjet STL

First some definitions

SLA



[1]

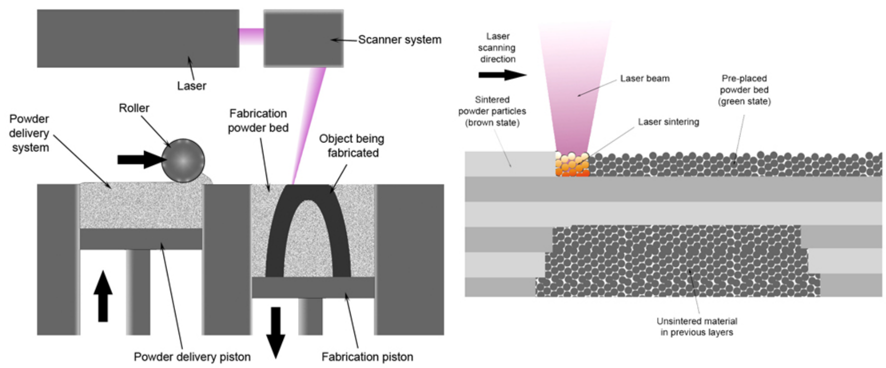
Stereolithography is an additive manufacturing process that works by focusing an ultraviolet (UV) laser on to a vat of photopolymer resin. With the help of computer aided manufacturing or computer aided design software (CAM/CAD), the UV laser is used to draw a pre-programmed design or shape on to the surface of the photopolymer vat. Because photopolymers are photosensitive under ultraviolet light, the resin is solidified and forms a single layer of the desired 3D object. This process is repeated for each layer of the design under the 3D object is complete.[2]

---

[1] <https://upload.wikimedia.org/wikipedia/commons/thumb/9/99/Stereolithography_apparatus_vector.svg/2000px-Stereolithography_apparatus_vector.svg.png>

[2] "Stereolithography - Wikipedia, the free encyclopedia." 2011. 20 Nov. 2015 <https://en.wikipedia.org/wiki/Stereolithography>

SLS

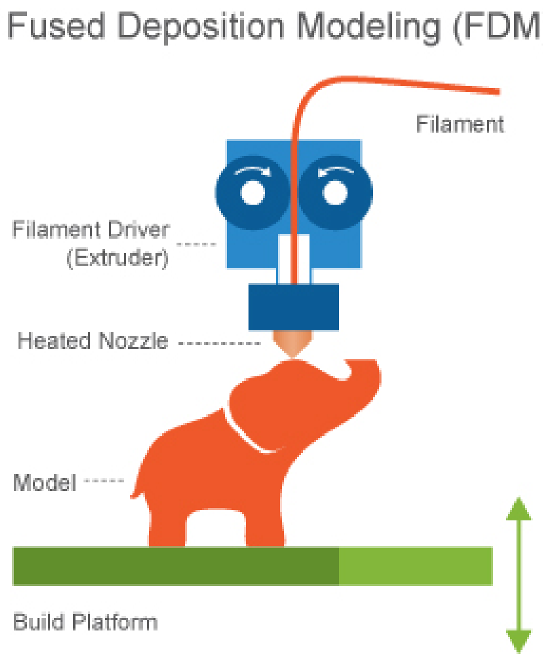


3

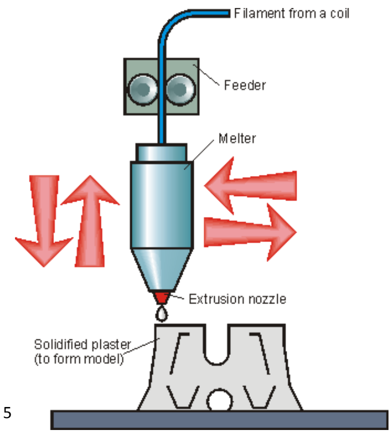
An additive manufacturing layer technology, SLS involves the use of a high power laser (for example, a carbon dioxide laser) to fuse small particles of plastic, metal, ceramic, or glass powders into a mass that has a desired three-dimensional shape. The laser selectively fuses powdered material by scanning cross-sections generated from a 3-D digital description of the part (for example from a CAD file or scan data) on the surface of a powder bed. After each cross-section is scanned, the powder bed is lowered by one layer thickness, a new layer of material is applied on top, and the process is repeated until the part is completed.[4]

[3] <https://upload.wikimedia.org/wikipedia/commons/3/33/Selective_laser_melting_system_schematic.jpg>
[4] "Selective laser sintering - Wikipedia, the free encyclopedia." 2011. 20 Nov. 2015 <https://en.wikipedia.org/wiki/Selective_laser_sintering>

FDM



[5]



[6]

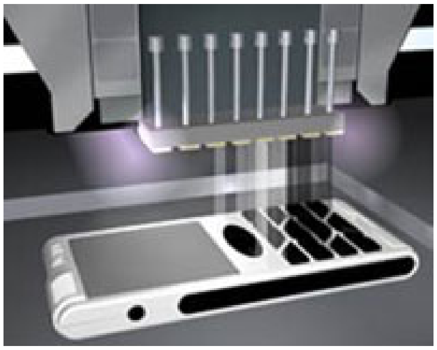
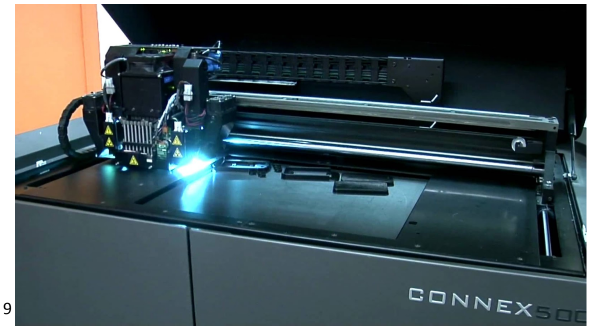
Fused deposition modeling (FDM) is an additive manufacturing technology commonly used for modeling, prototyping, and production applications. It is one of the techniques used for 3D printing. FDM works on an "additive" principle by laying down material in layers; a plastic filament or metal wire is unwound from a coil and supplies material to produce a part.[7]

PolyJet

PolyJet 3D printing is similar to inkjet printing, but instead of jetting drops of ink onto paper, PolyJet 3D Printers jet layers of curable liquid photopolymer onto a build tray. [8]

---

[5] <http://3devo.eu/wp-content/uploads/2015/08/FDM.jpg>
[6] <http://www.uni.edu/~rao/rt/fdm.gif>
[7] "Fused deposition modeling - Wikipedia, the free encyclopedia." 2011. 20 Nov. 2015 <https://en.wikipedia.org/wiki/Fused_deposition_modeling>
[8] "PolyJet Technology | Stratasys." 2014. 25 Nov. 2015 <http://www.stratasys.com/3d-printers/technologies/polyjet-technology>

 [9] 

[10]

PolyJet 3D Printing technology offers many advantages for rapid tooling and prototyping, and even production parts including astonishingly fine detail, smooth surfaces, speed and precision.

- Create smooth, detailed prototypes that convey final-product aesthetics.
- Produce short-run manufacturing tools, jigs and assembly fixtures.
- Produce complex shapes, intricate details and smooth surfaces.
- Incorporate color and diverse material properties into one model with the greatest material versatility available.[11]

STL

To generate the toolpaths for FDM, SLA or STL, the 3D model needs to be converted into a generic 3D model. The industry standard is a file extension of .stl, or STL file, which consists of a model containing only polygonal data (Triangles only in a watertight mesh).
From wikpedia:

STL (STereoLithography) is a file format native to the stereolithography CAD software created by 3D Systems.STL has several after-the-fact backronyms such as "Standard Triangle Language" and "StandardTessellation Language". This file format is supported by many other software packages; it is widely used for rapid prototyping, 3D printing and computer-aided manufacturing. STL files describe only the surface geometry of a three-dimensional object without any representation of color, texture or other common CAD model attributes. The STL format specifies both ASCII and binary representations. Binary files are more common, since they are more compact [12]

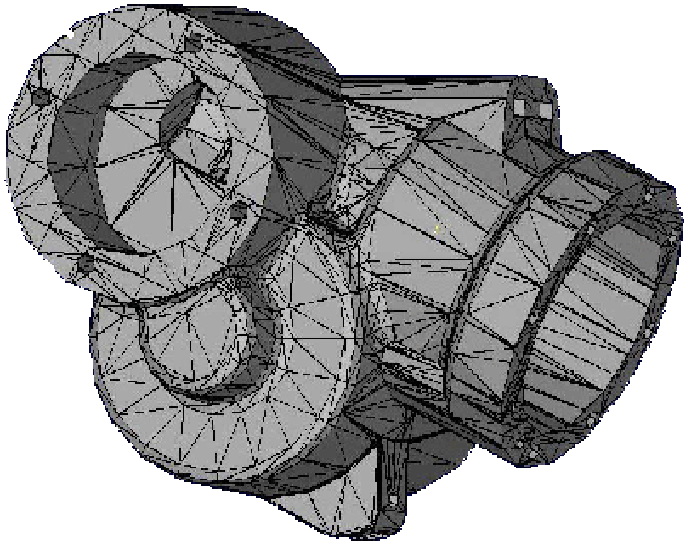
---

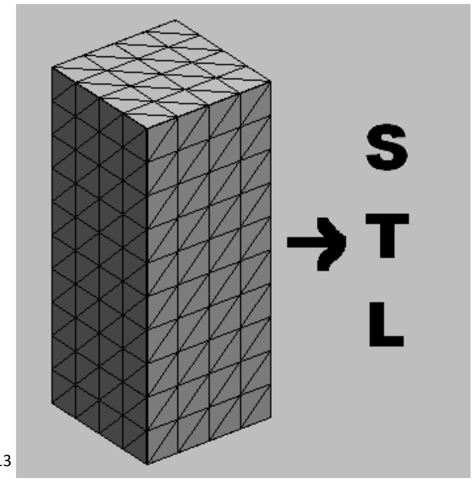
[9] <http://usglobalimages.stratasys.com/en/3D%20Printers/Technology/PolyJet%20Technology/polyjet_print_head_drawing.jpg?v=634998972713459201>

[10] <https://i.ytimg.com/vi/D4Yq3glEyec/maxresdefault.jpg>

[11] "PolyJet Technology | Stratasys." 2014. 25 Nov. 2015 <http://www.stratasys.com/3d-printers/technologies/polyjet-technology>

[12] "STL (file format) - Wikipedia, the free encyclopedia." 2011. 20 Nov. 2015 <https://en.wikipedia.org/wiki/STL_(file_format)>

 [13]



[14]

When Should you chose 3D printing vs Traditional manufacturing?

Good candidates for printed parts:

- Composite (fiberglass/carbon fiber) layup that requires new tooling
- Formed/molded plastic parts that require new tooling
- Complex multi-surface or internal features
- Curved surfaces

Stick with traditional manufacturing:

- Simple, flat machined features
- Parts that already have tooling fabricated
- Simple extrusions

Consider AM if you can answer YES to any of these questions:

- Does the part require tooling to build?
- Are there several components that make up the assembly that can be incorporated into a single part (flanges, hard points, fasteners, etc.)?
- Are there complex surfaces or internal features?
- Do you need a quick prototype?
- Is the part expensive or long lead time?

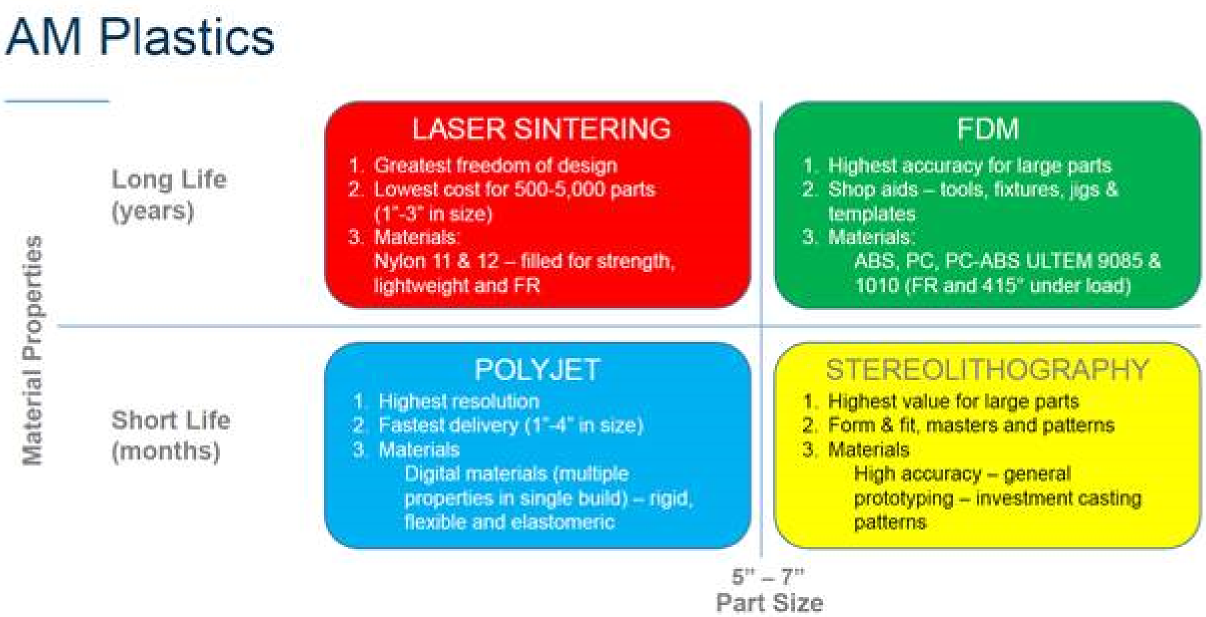
---

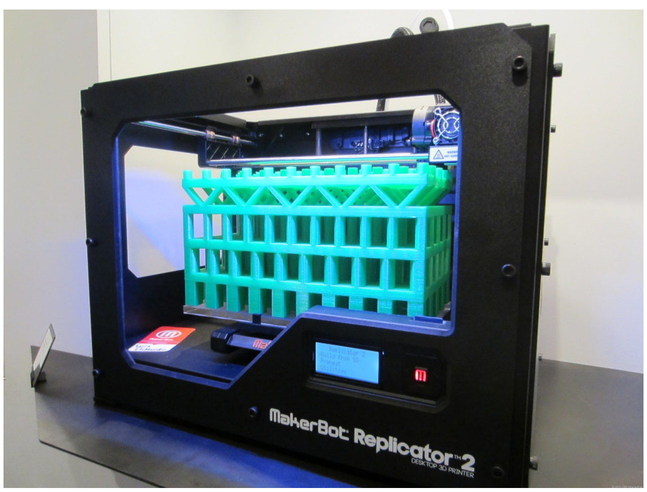
[13] <http://www.stereolithography.com/images/export/1_mesh.gif>

[14] <http://www.mathworks.com/matlabcentral/mlc-downloads/downloads/submissions/20922/versions/9/screenshot.jpg>

What technology is best suited for your project?

Consider the following chart for material selection





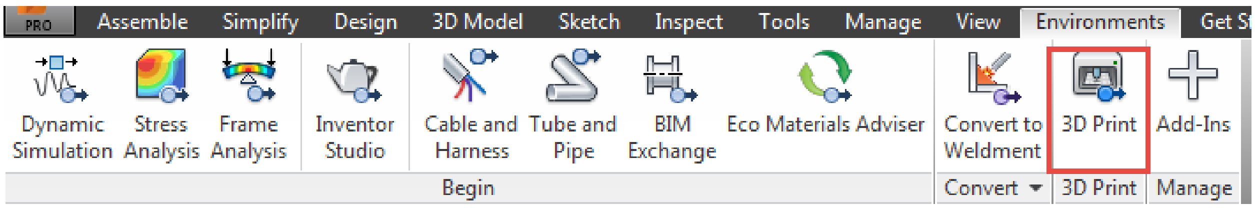
[15]

[15] <https://i.ytimg.com/vi/0wWG_3MeyHk/maxresdefault.jpg>

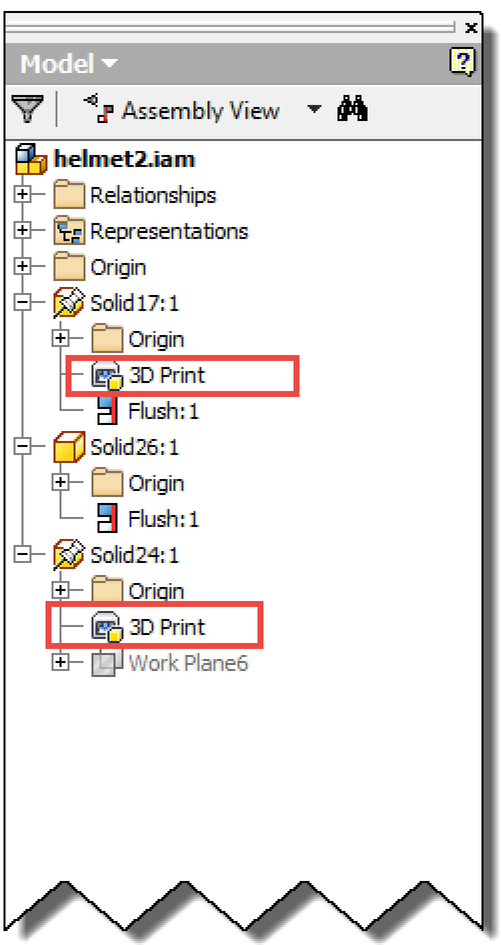
Inventor's 3D printing environment

Why do I need a 3D printing environment?

I can just export an STL file and be done with it, right?
Yes



But, Inventor 2016 comes with a new 3D printing environment that provides more fine control over the export process, and allows for part editing, orientation changing and scaling without affecting the native file.



It does this by creating a print object that is derived from the geometry of the model, so any changes to the 3D print environment won't be reflected back to the source geometry, but any changes to the source geometry will affect the 3D print.

Inventor's 3D print environment is mostly tailored to be used with at home **FDM** 3D printing, for a home user or small business. It includes a number of preset printers or build spaces, and can be customized to include more.

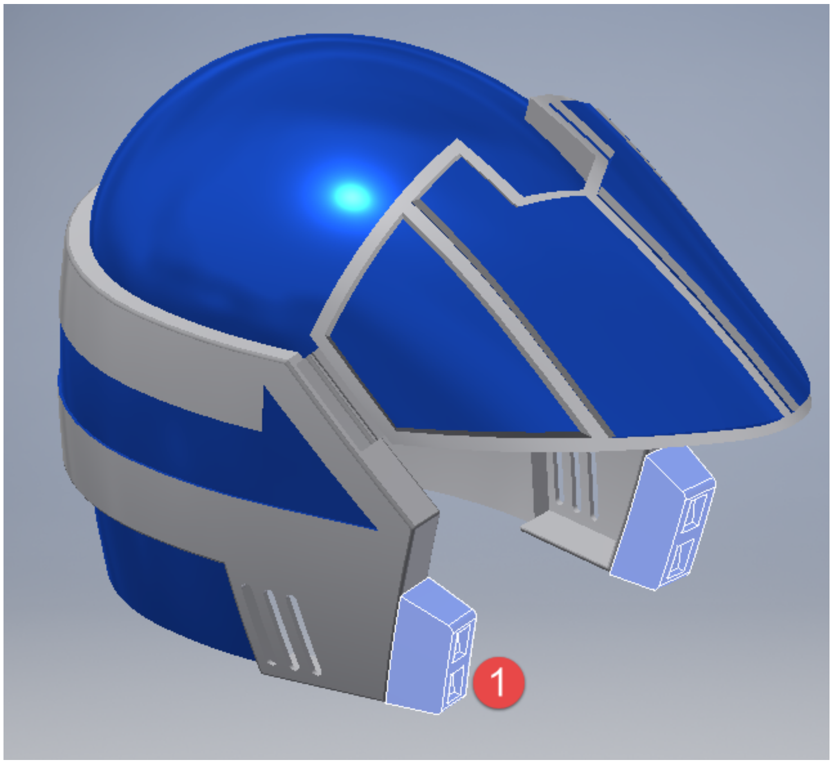
The build spaces are intended to give the user a sense of scale and to allow the optimization of the orientation of the part inside of the build volume, before exporting the STL format. Because in 3D printing, as with any CAM (computer aided manufacturing) workflow, orientation matters.

3D printing workflow

Inventor

Identify part to be 3D printed

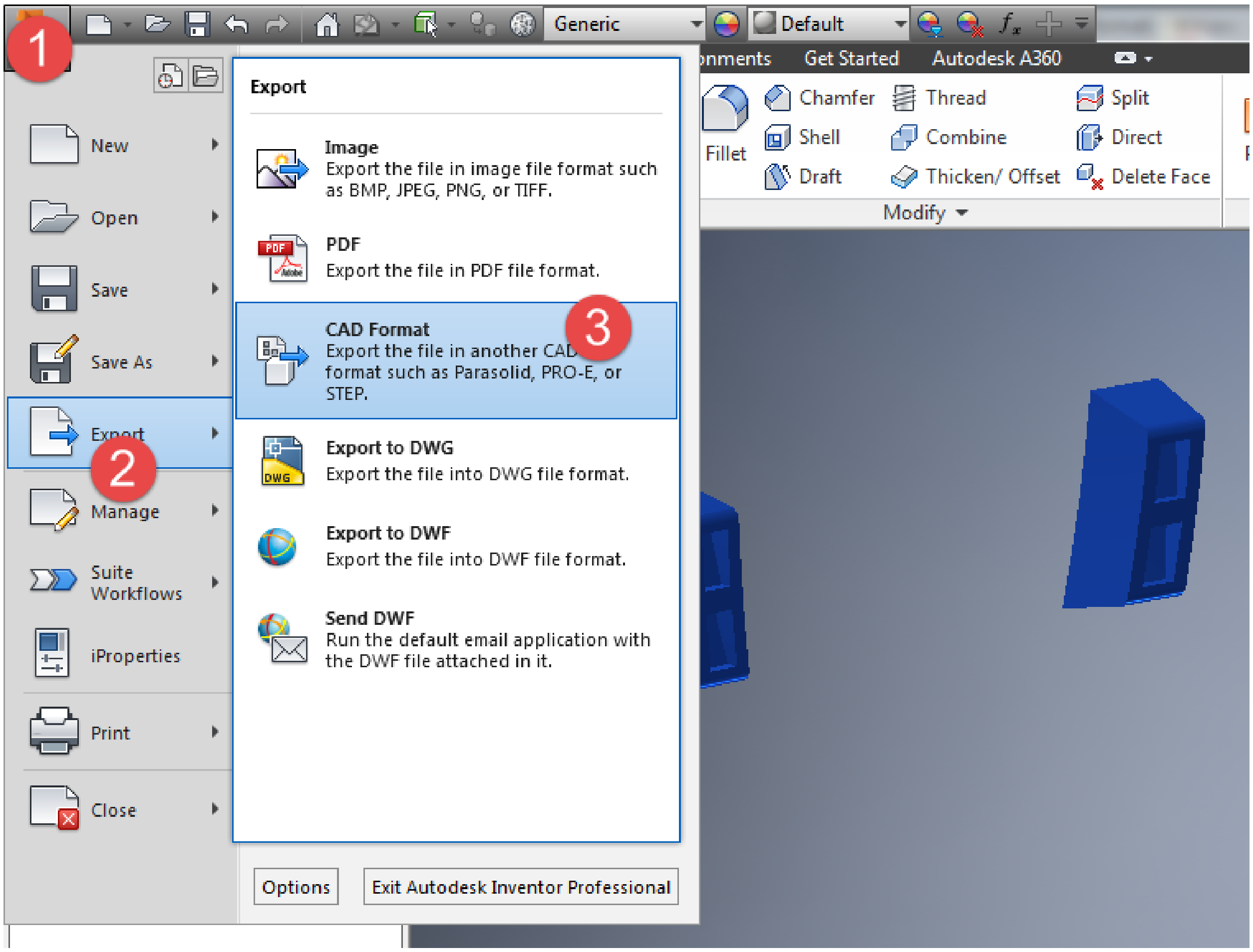
Based on the criteria from the previous section, and whether or not you can afford to produce the part using conventional subtractive manufacturing, open the model of the part you want to print.



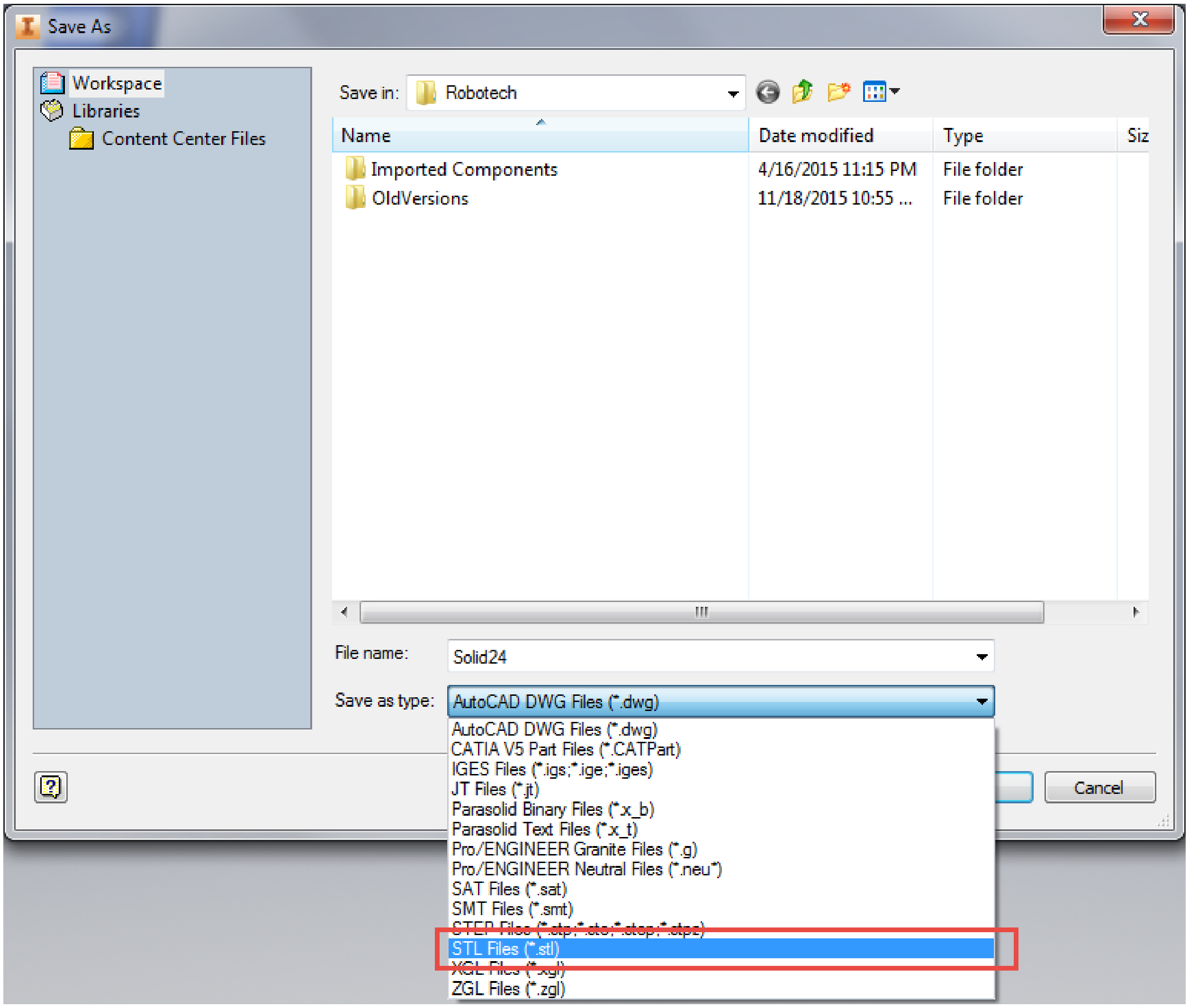
Export as STL

At this point, the part can be saved as an STL file, as you would in Inventor 2015 and prior.
The STL file can be sent to a 3D print service, or to a 3rd party program to manipulate, or to be used by the printing program that came with your 3D printer.
To do this:

1. click on the I button
2. select Export
3. Select CAD format

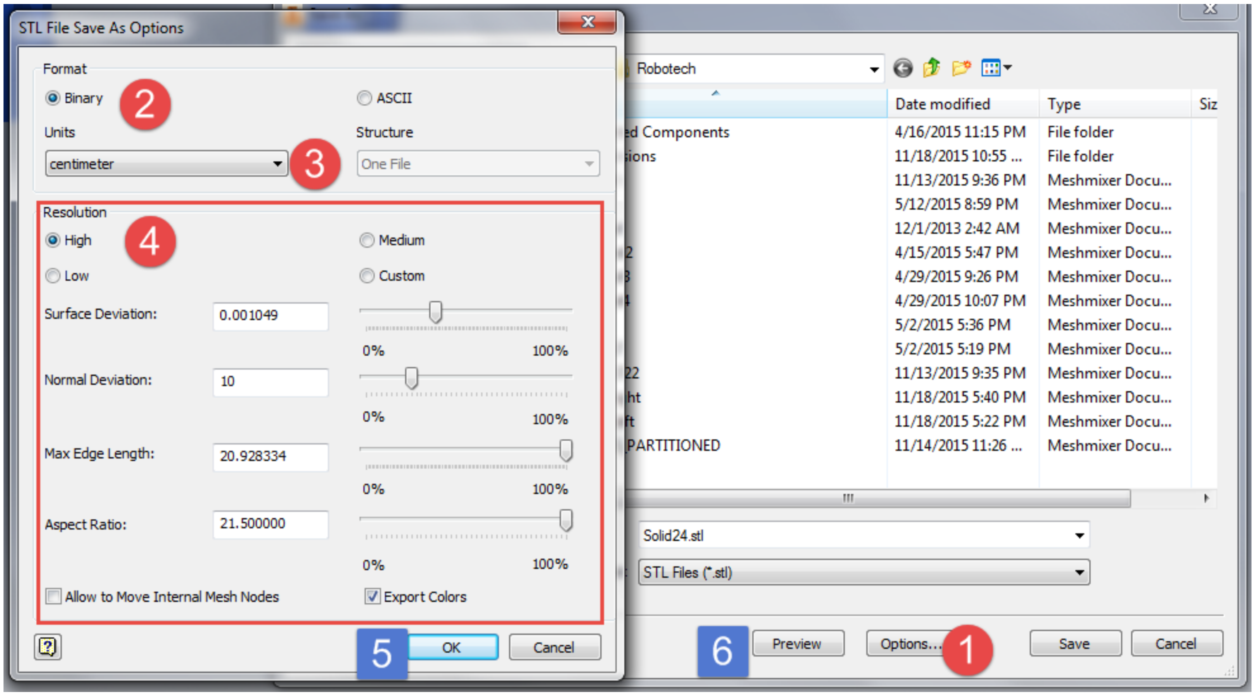


On the Save As dialogue, select STL files as the file type.



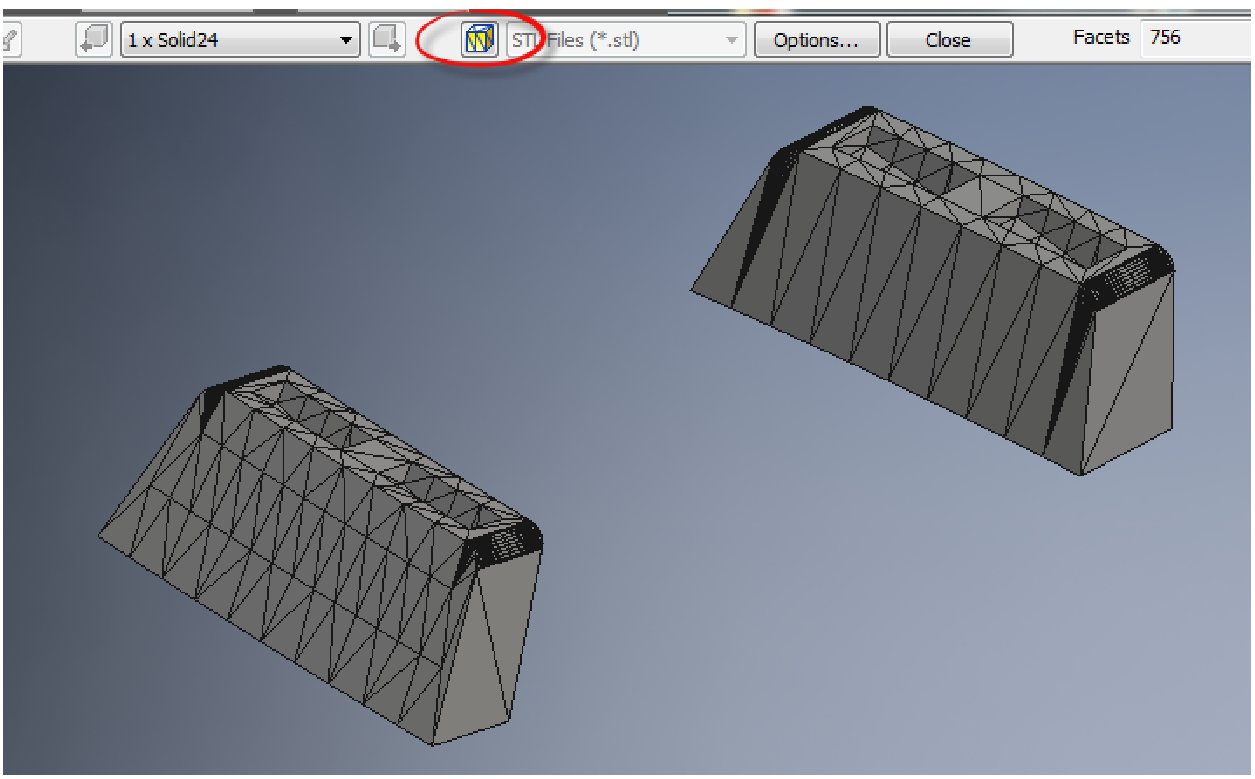
After selecting STL files, click on the

1. Click on ptions button
2. Select the type of file, Binary or ASCII
3. Select units
4. Choose the resolution (number of triangles describing the geometry)



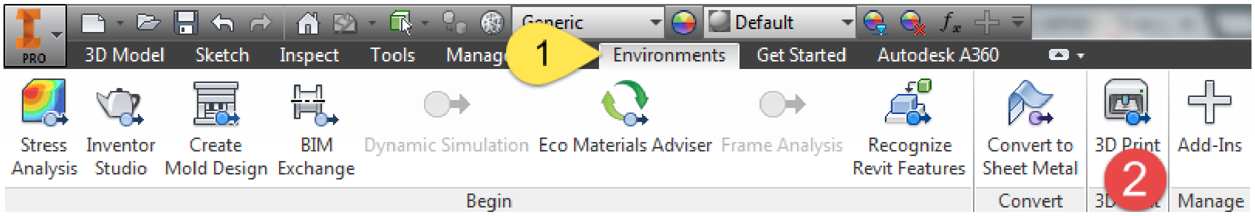
5. Click OK
6. Click Preview to verify your selection
7.

Toggle mesh view to see vertices, and ensure that they match the expectations for resolution.



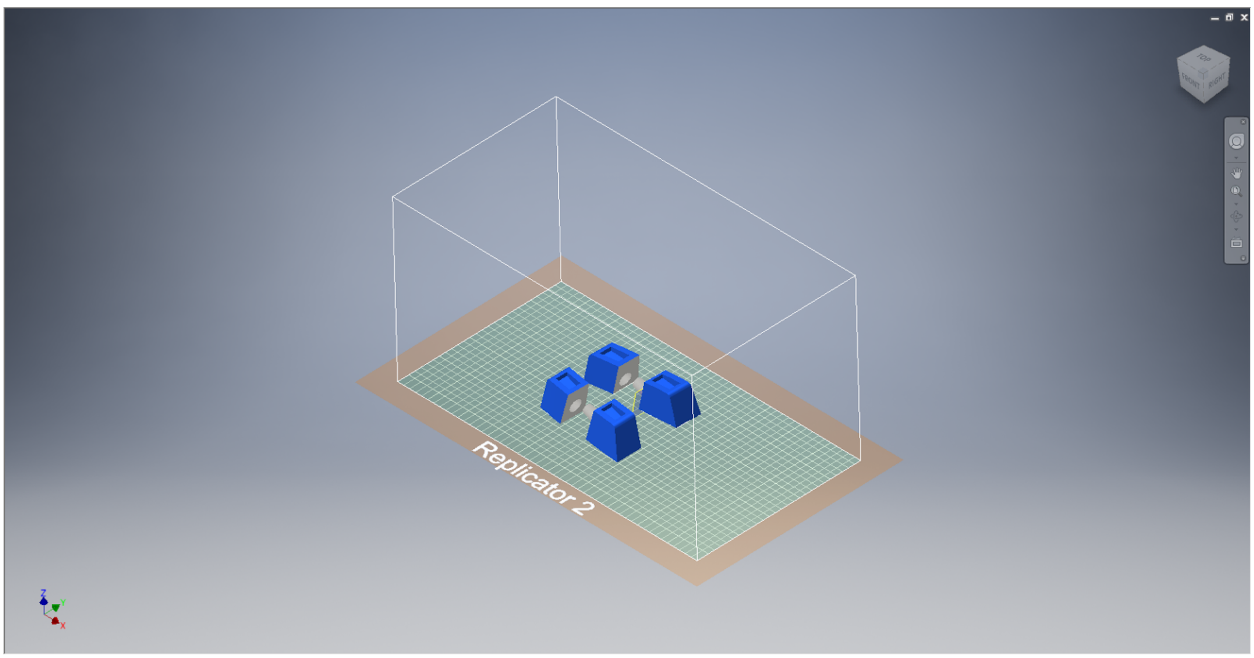
Alternatively:

Enter the 3D printing environment

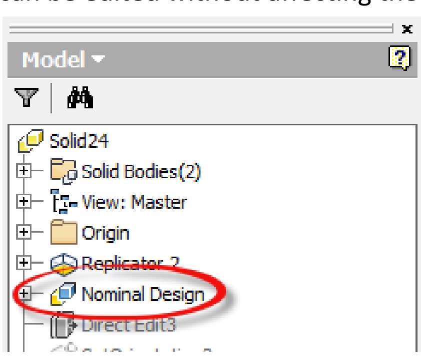


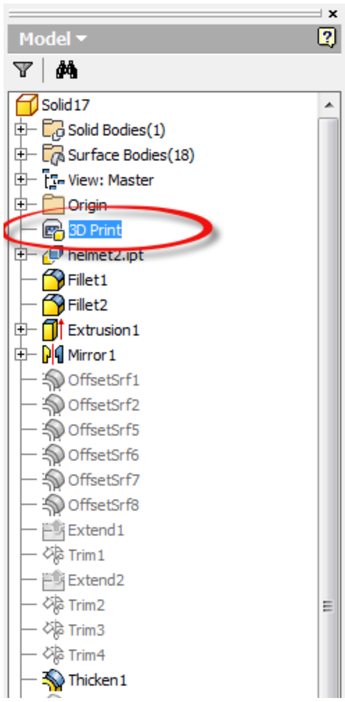
To enter the 3D print environment, click on the environments tab, and then on the 3D print button.

This brings up the 3D print environment.



The 3D print environment creates a derived solid from the original geometry, which is linked, and which can be edited without affecting the original part.





Whatever edits are made to the part in order to optimize it for 3D printing, will be contained inside a 3D print object inside the model tree of the original part

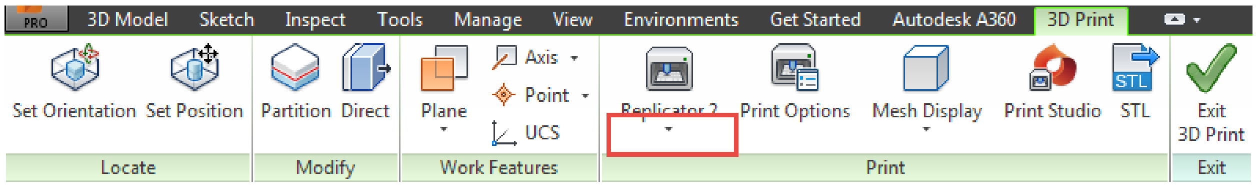
That way, the part can be edited at will without fear of affecting the original part, or without having to do a save as of the the part to be 3D printed.

This is particularly useful, if the design changes and we want the changes to be reflected in the 3D print environment, without having to do the edits again.

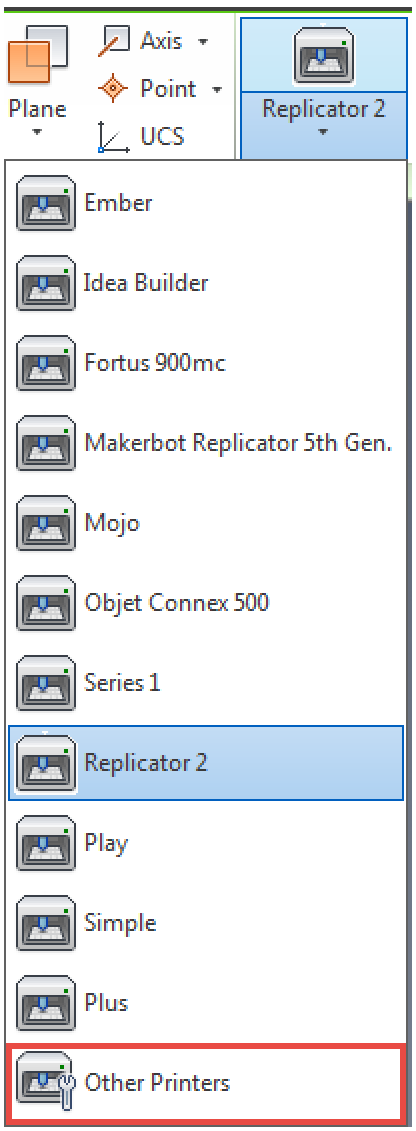
In other words, the 3D print environment and the original part maintain one-way associativity.

Select a printer

Inventor 2016 comes pre-loaded with specifications for a number of 3D printers, most of which are FDM printers. The most common ones can be found under the 3D printer drop down.

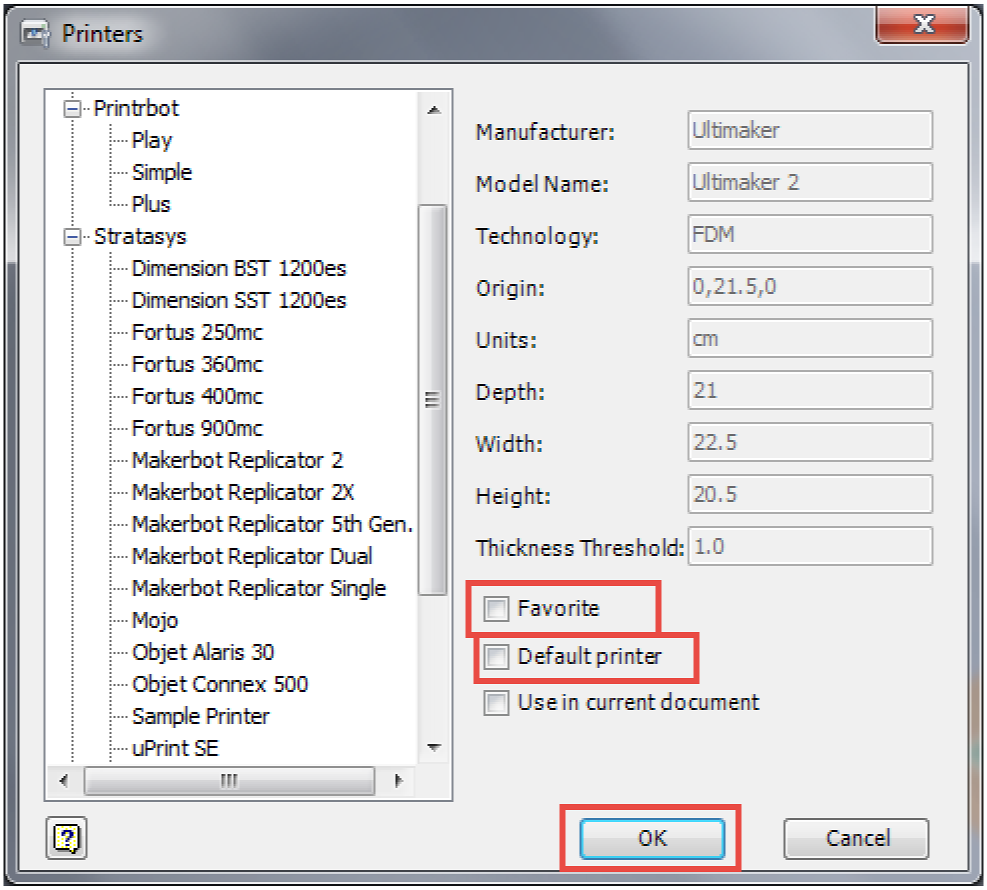


If your printer is not in the list, click on the Other printers button.



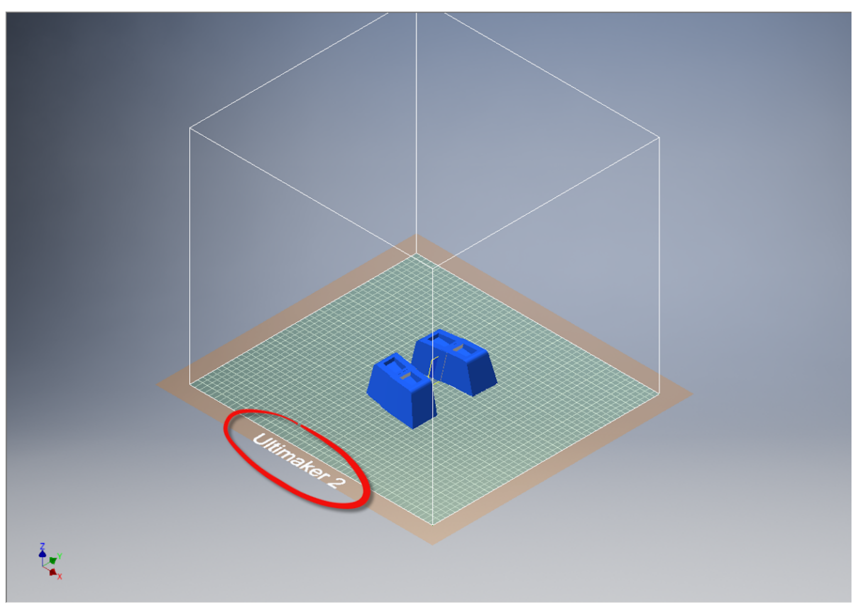
Add a printer

To add a printer to the drop down list, simply select it from the left hand column, which has an extensive list sorted by manufacturer.



Selecting it as Favorite, will add it to the drop down list, selecting it as Default printer will load it as default every time the 3D print environment is accessed, and Use in current document, replaces the printer currently in use.

The title of the printer, as well as the dimensions of the print bed should match inside the model window.

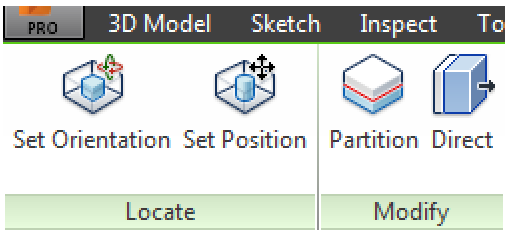


Edit the part

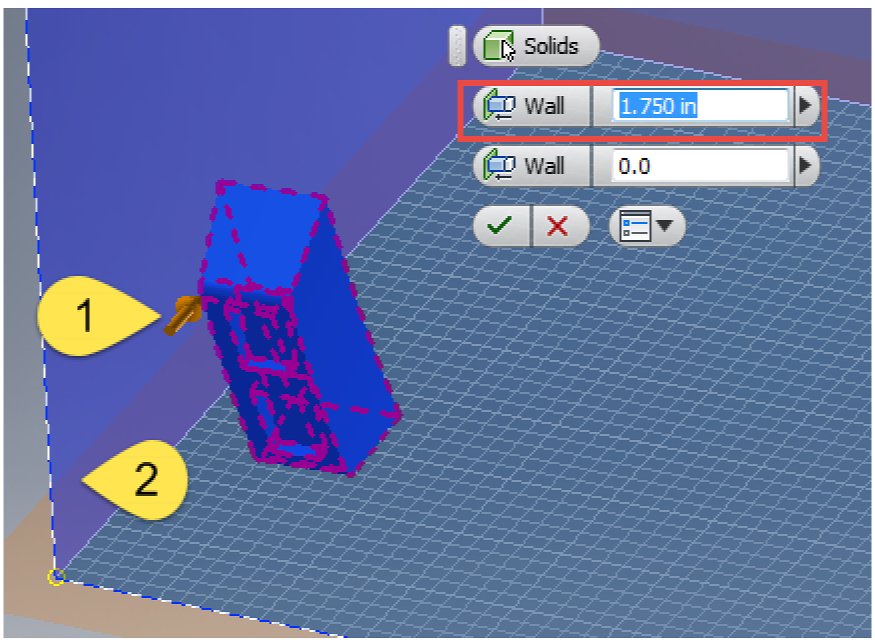
The edits made to the part in this environment won't affect the original geometry, so edit away!

Move

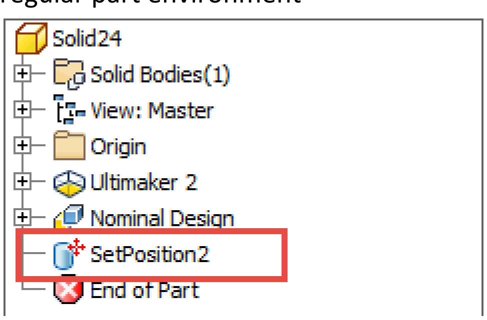
Use the set position tool to locate the part within the print volume.



Select a printer wall

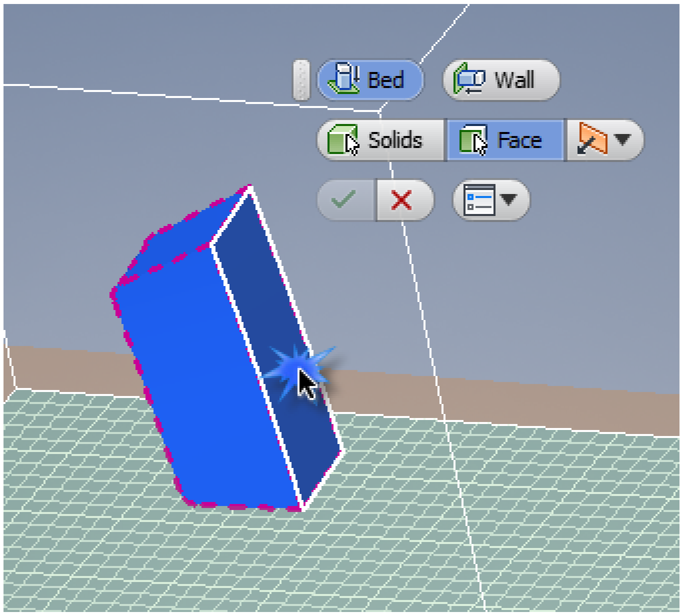


And specify an offset, or use the manipulator to move the part
You can select more than one wall.
The edits will be saved in the model tree of the 3D print environment, just like the model tree in the regular part environment



Set orientation

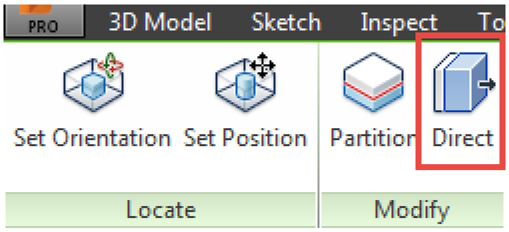
Set orientation works similarly
The printer Bed is the default base location.



Position the cursor over the edge of the wall you want to use as the base. When the wall highlights and the wall name displays, click to select it. This will re-orient the part so that the face selected will mate with the printer bed.
Walls and model edges can also be used to orient the model. If not flat edges exist, use the model's internal planes to orient the part.

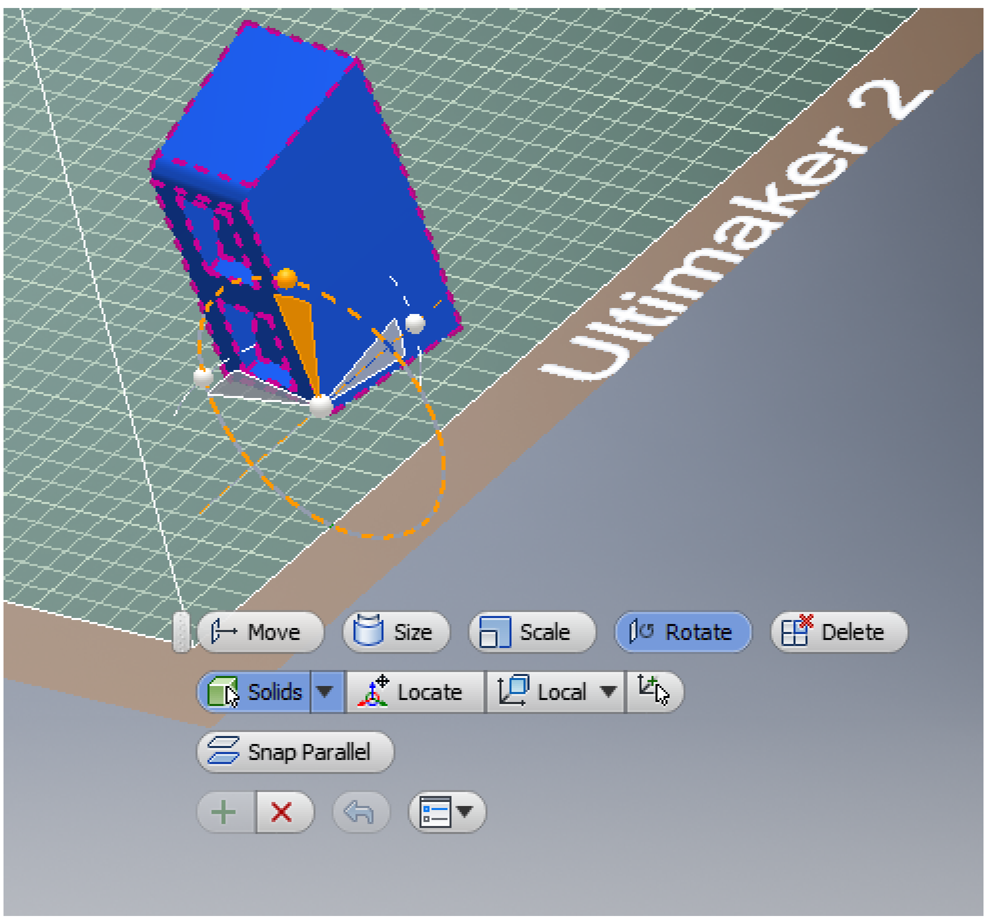
Direct edit

Direct edit gives finer control over position, orientation and scale. It is the same direct edit tool that exists inside of the regular modeling environment.



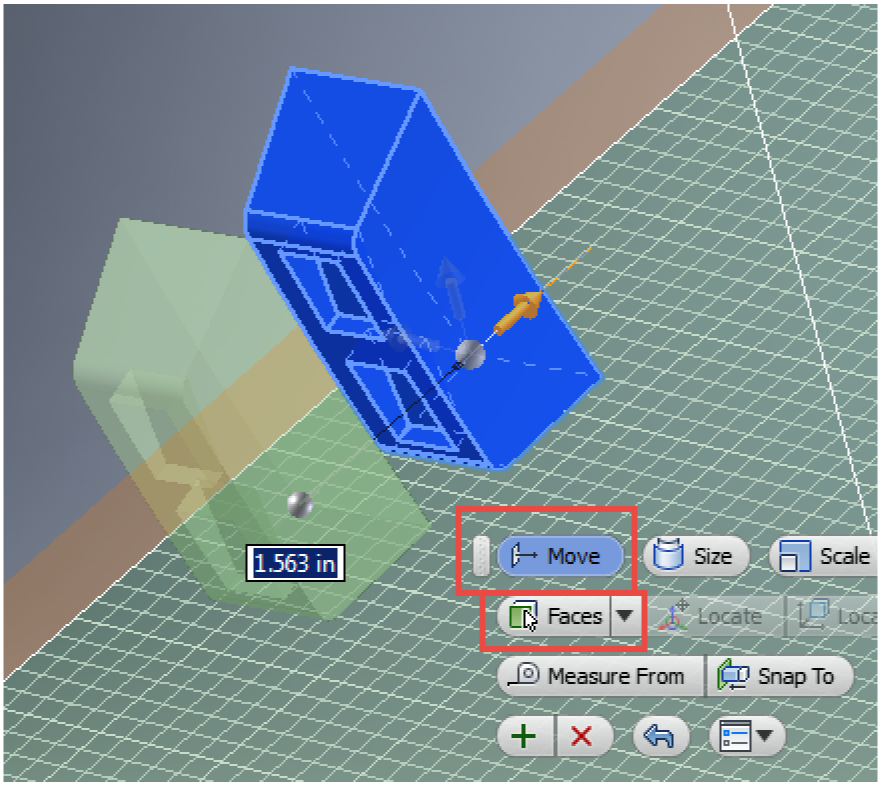
Rotate

Select the rotate button inside of the direct edit tool, and select a solid to rotate. Rotate by using the manipulators



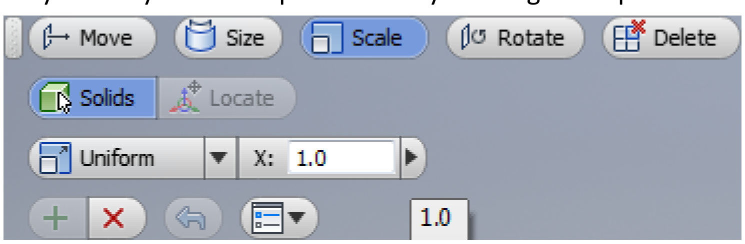
Move

Similarly to the rotate command, select solids or surfaces to move, and move using hte maniuplators



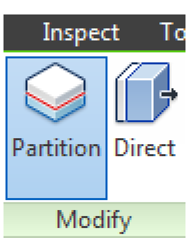
Scale

Why would you scale a part? Didn't you design it to precise measurements already?

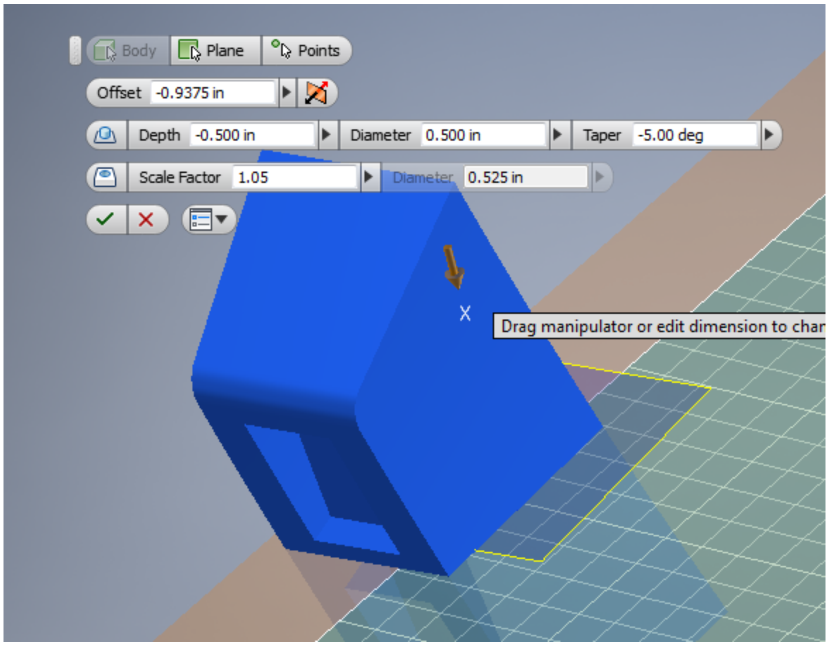


Same options for scaling. Use the manipulators, uniform or non-uniform scaling.

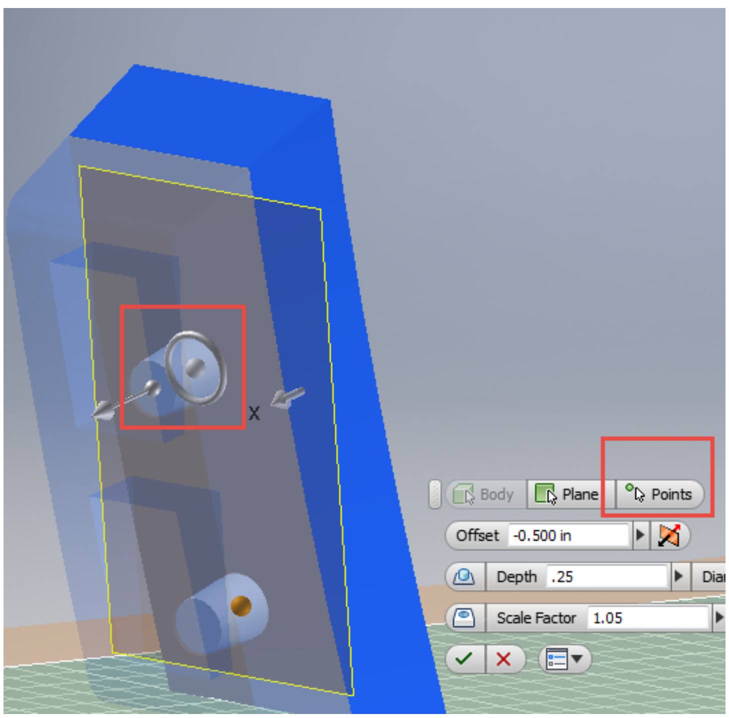
Partition



If the part will not fit within the print volume, one of the more useful features of Inventor allows for splitting a model into two solids and adding posts and holes in one go, without having to "engineer" them in the original component.

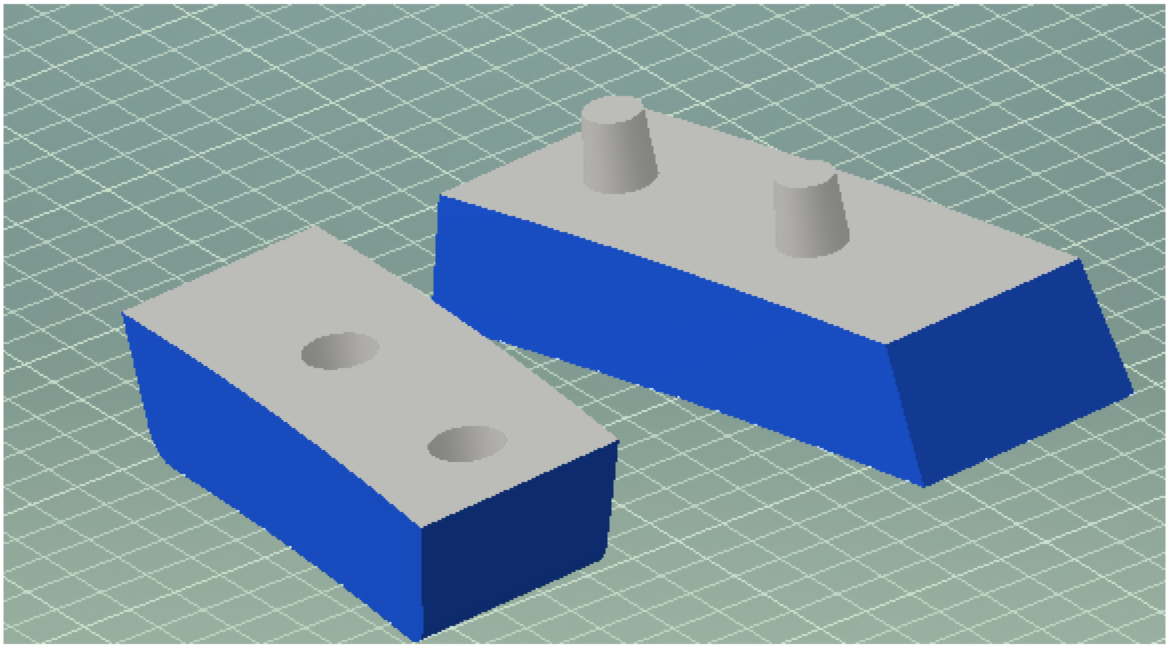


Select a cutting plane and move the manupulator to specify the offset from such plane.
Click on the Points button to add points where the posts and holes will be inserted.

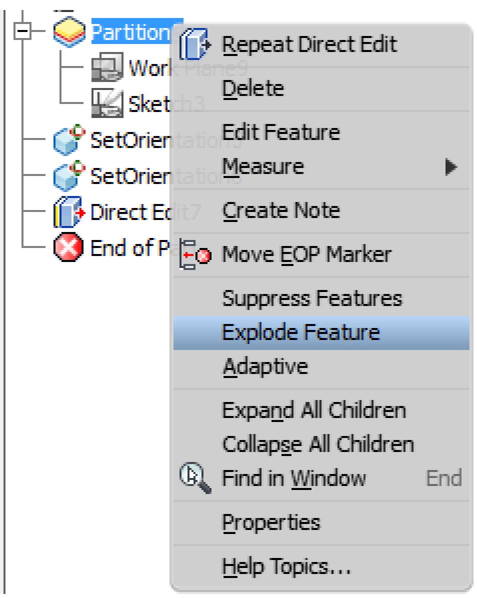


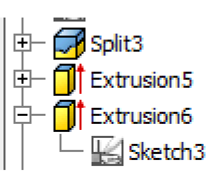
This will create a peg, or post and a corresponding hole in the opposite side. The scale factor between the hole and the post, as well as taper, depth and diameter of the posts can be manipulated here. This will also divide the part into two solid bodies.

It is a good idea to move the bodies apart for printing after this step.

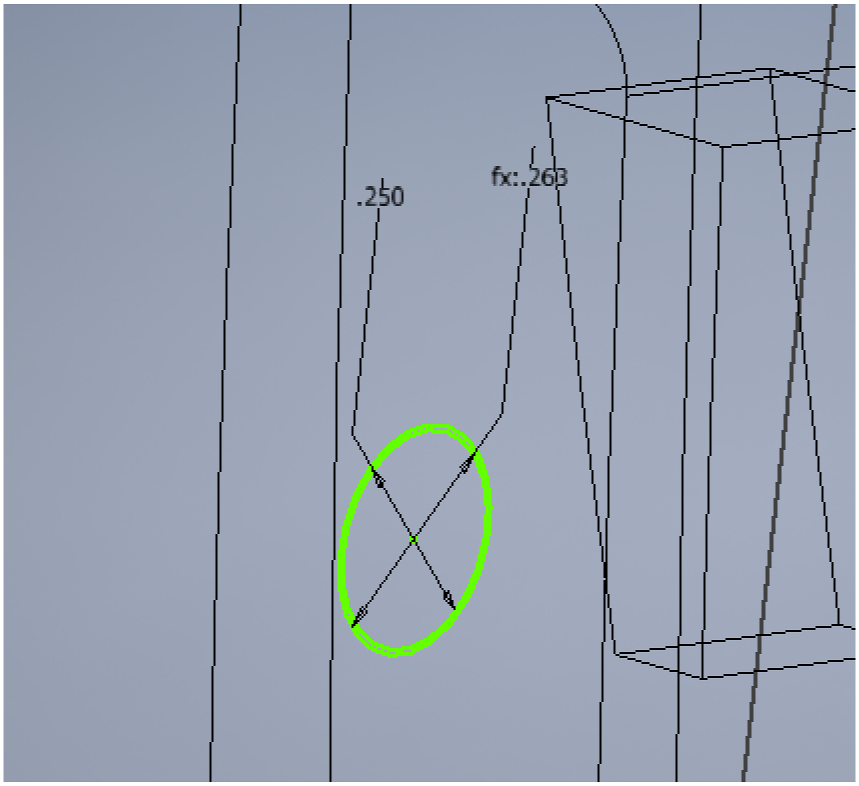


Furthermore, the partition feature can be exploded, for even finer control.



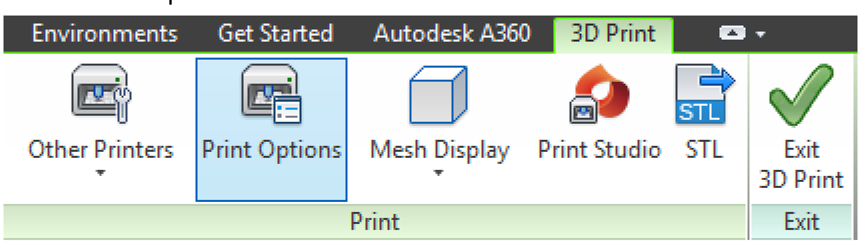


Which is nothing more than a split solid and two extrusions; one hole and one protrusion, driven by the same sketch.

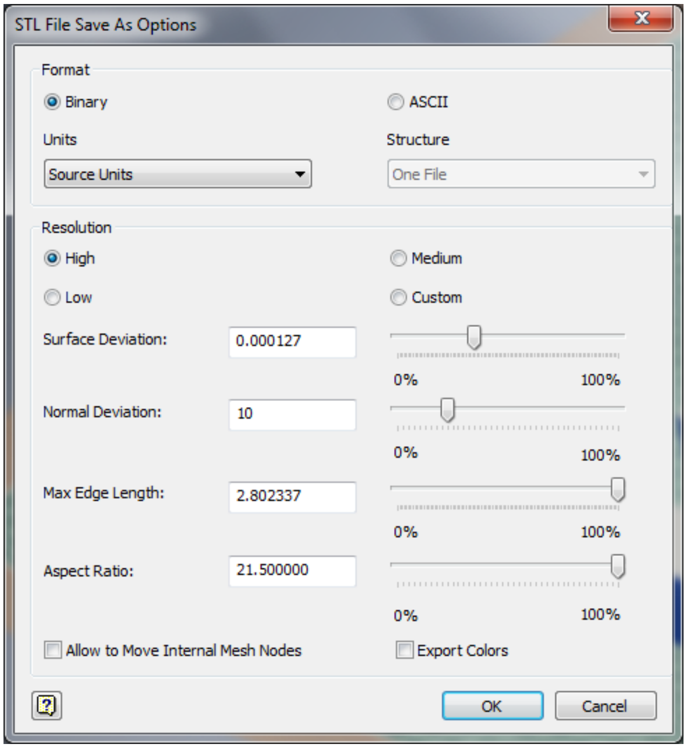


Print options (STL options)

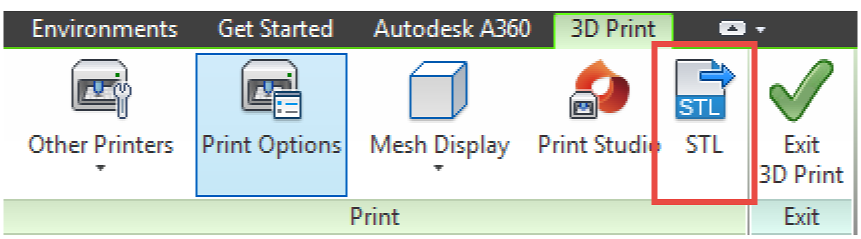
Now that the model has been prepared for printing, we can review the STL options once more from within the 3D print environment



This brings up the same dialogue box as the STL export options from the previous section



Export to STL

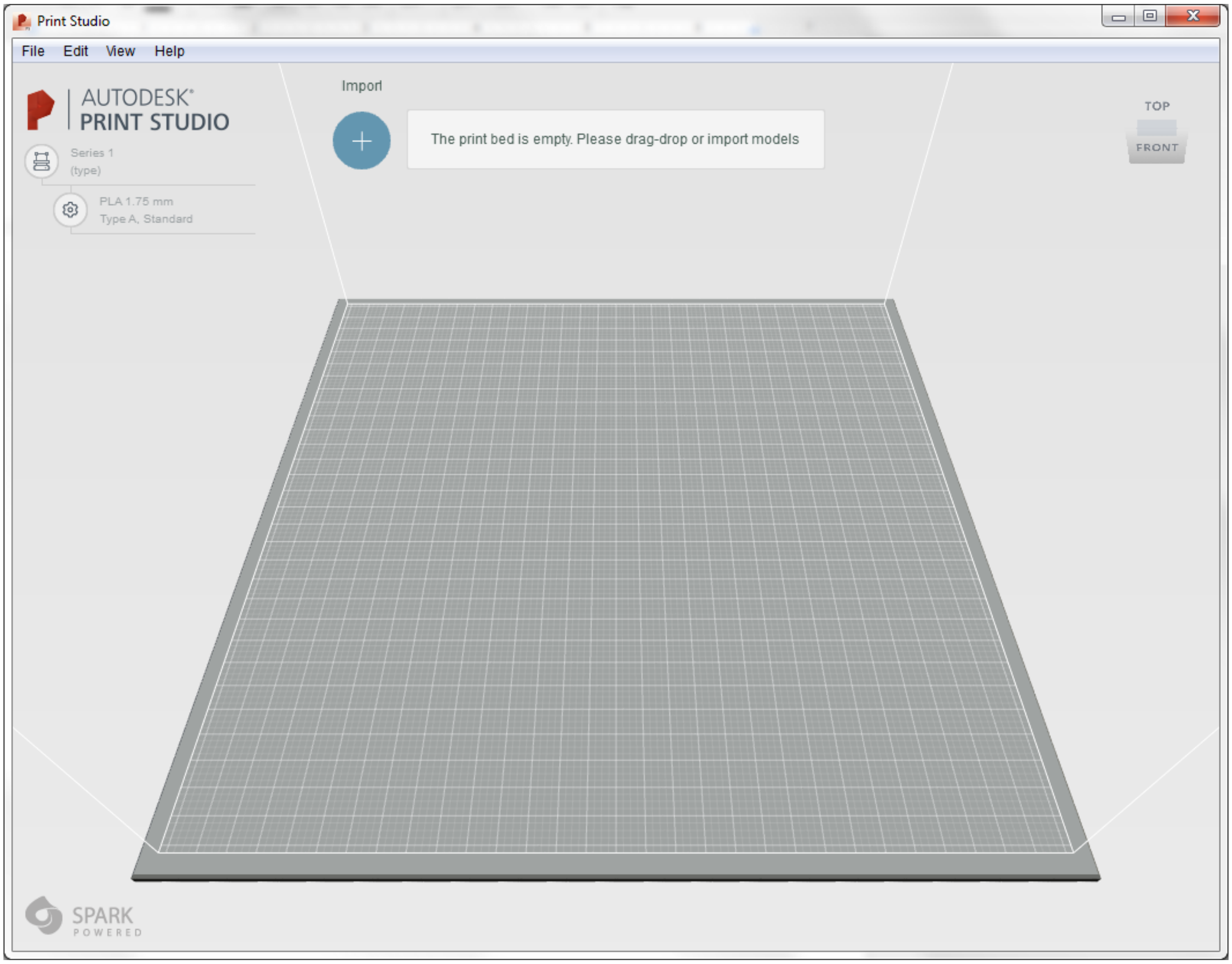


This simply saves the file with the current settings as an STL file, scaled appropriately.

3D printing workflow

Autodesk Print studio

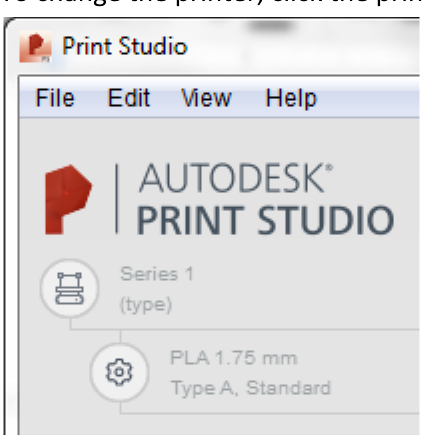
With 3D print studio, we can further edit the part to ready for print. Besides positioning, scaling and rotating the part, 3D print studio lets us add support material, verify errors in the mesh, edit the mesh, and even connect directly printer and allow for a 3D print. It can also generate the instructions (G Code) as a file, for a number of supported printers, for later printing.
To begin, we're presented with an empty workspace, and a default printer bounding box.



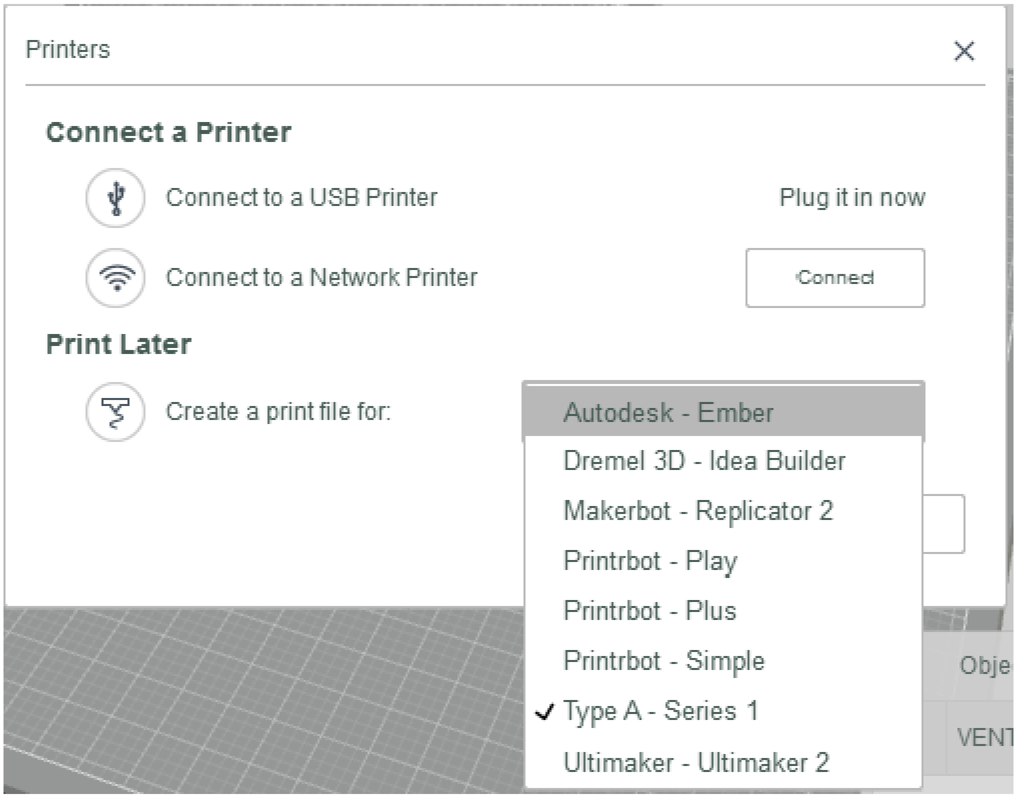
The mouse and keyboard controls are different than from inventor.

- To rotate, simply click the right mouse button.
- To pan, click the mouse wheel
- To zoom in and out, scroll the mousewheel.

To change the printer, click the printer icon on the top left.



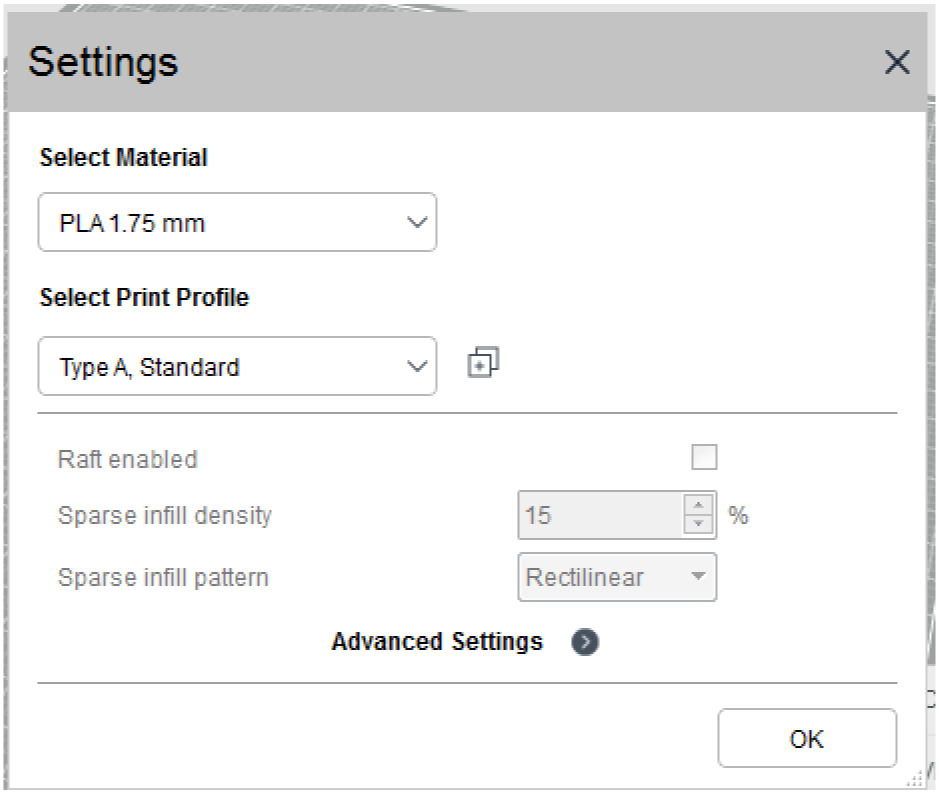
This will bring up the following dialogue box with the following printers available



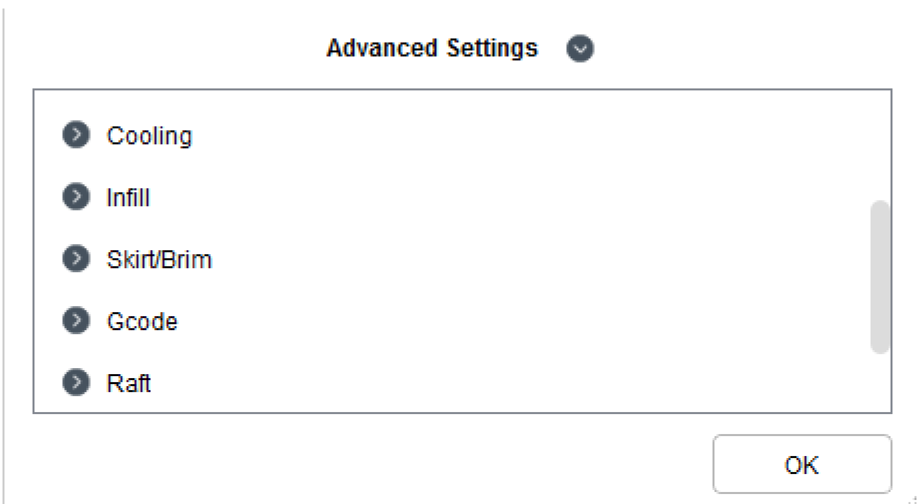
Note that the selection is smaller in this case - that's because these are also the printers with drivers supported by 3D print studio.

Next, select the type of material to be used by clicking on the settings icon

The following dialogue box appears, where we can select the material and the print profile. The print profile has predetermined settings for the sparse infill density and pattern definition.

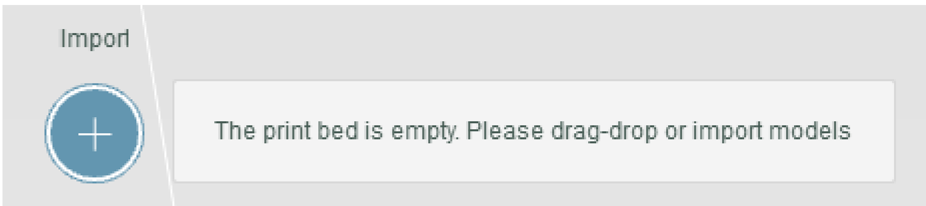


Clicking on advanced settings does just that - it brings up advanced settings for each printer type, such as cooling, Skirt/Brim options, Gcode and Raft

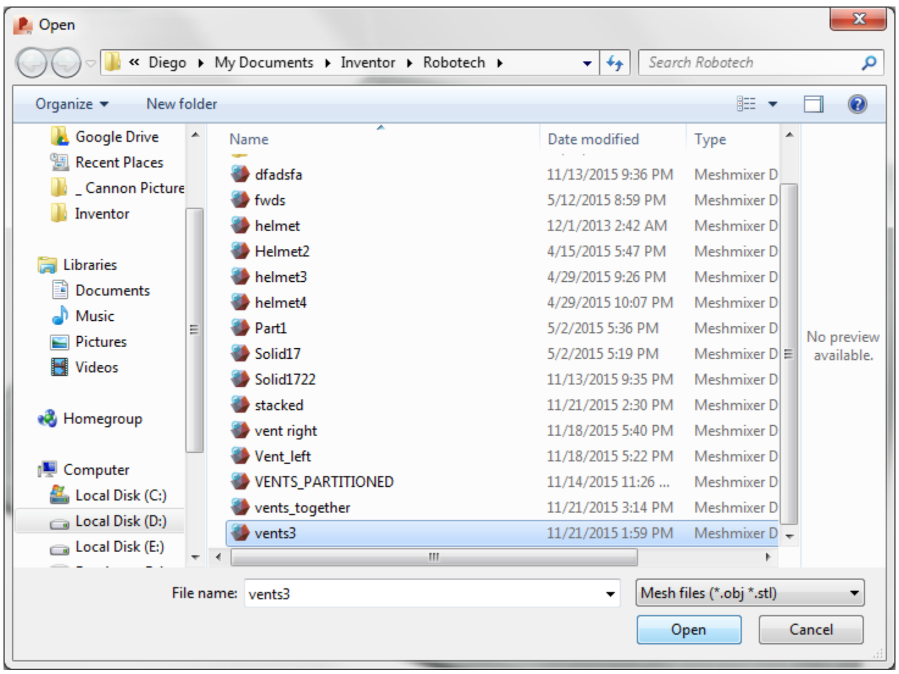


Import

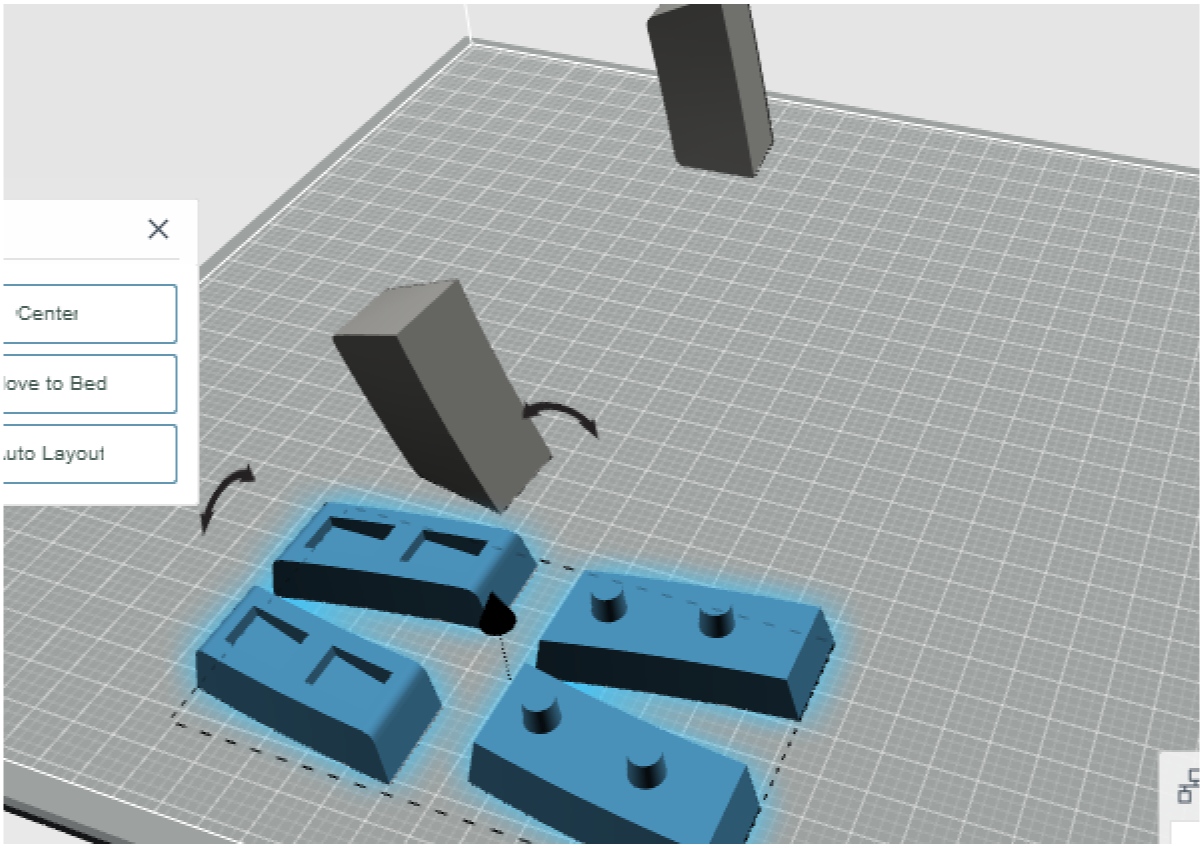
To begin, load the model by clicking the "import" plus sign on the top of the screen



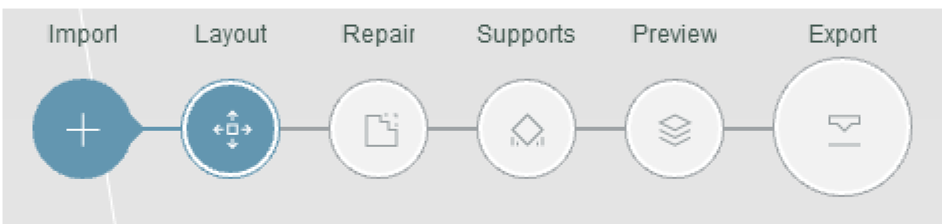
Which brings up the open dialogue box



Select the file you want to open and click Open.
You can load more than one model at a time.



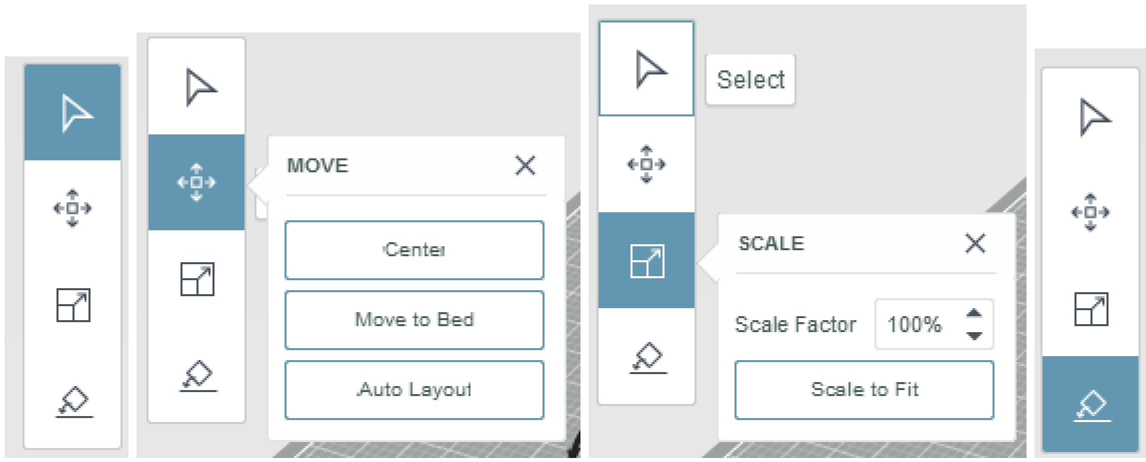
At this point, the steps necessary to verify and print the model are presented in a timeline icon progression.



The steps don't need to be completed in that specific order but the presentation is logical

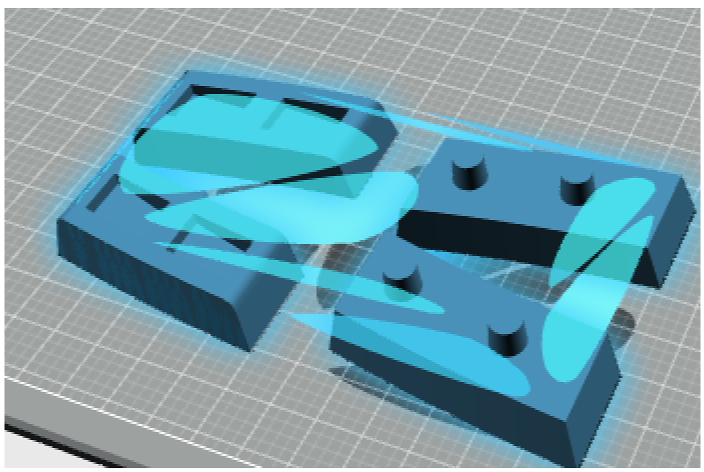
Layout

Layout allows for the repositioning of the model by selecting any of the options for movement from the left hand menu
These include Select
Move, which has further options such as center, move to bed or auto layout



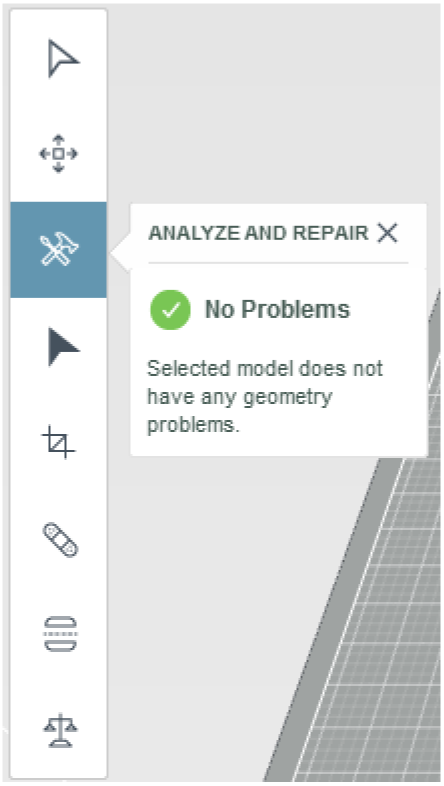
Scale, where a scale factor can be selected, or the option to scale to fit the print bed allows for resizing to fit

And lay flat, which analyzes the model and picks a surface to mate with the print bed

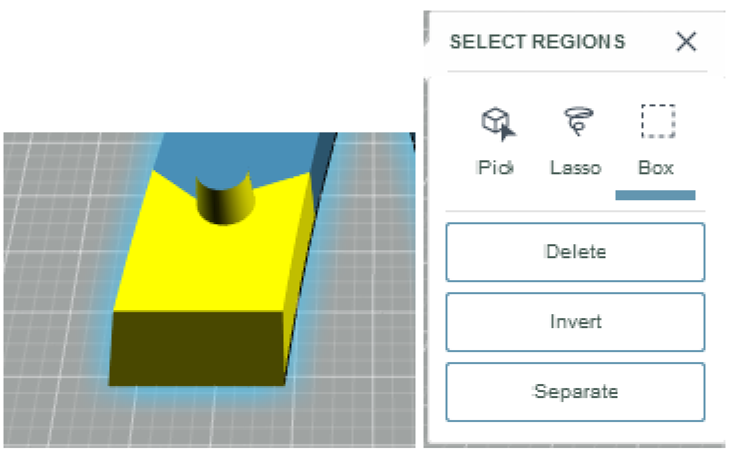


Each of the teal regions is a suggested lay flat position. In the case of the model pictured above, the model was already in the best orientation.

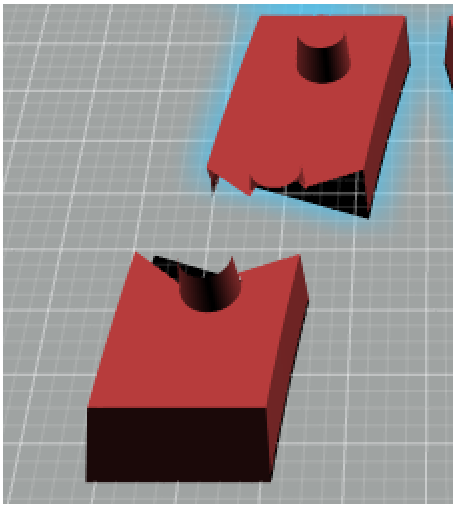
Repair

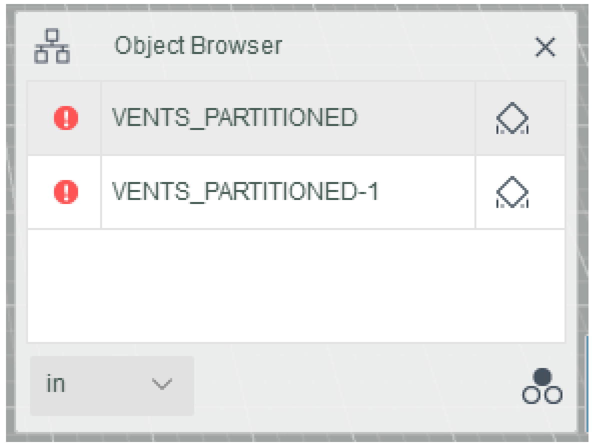


Since our model came from inventor, it is very unlikely that it will need repairing.
However, the tool allows for various edits to the mesh.
These include Moving, selecting regions:

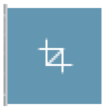


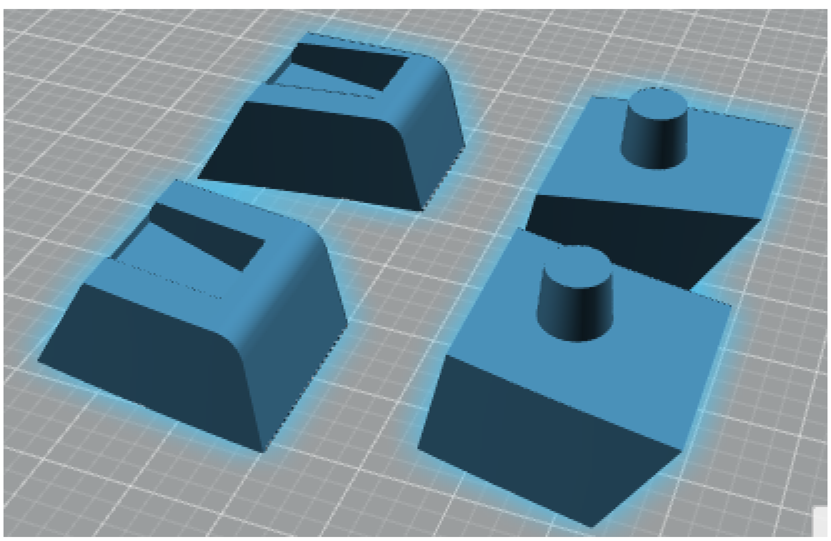
Selecting regions allows us to delete triangles in that region, invert the selection and separate (into two bodies)





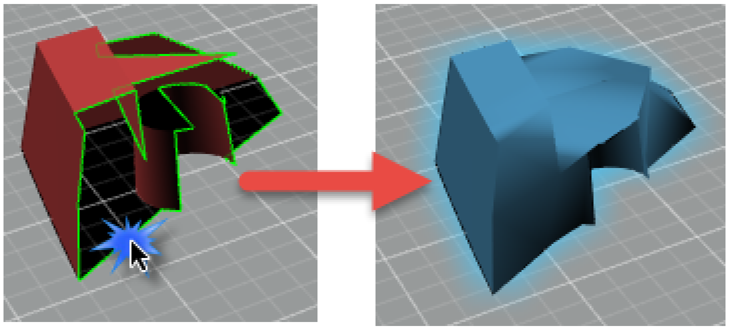
The object browser in the bottom right will show two solids now, where there was one. The exclamation mark indicates that there is a problem with our mesh (it is broken!)

The crop tool  does exactly that - it will allow you to create a bounding box that can cut the object (and often break it as well, although it tries to build walls where the cut occurred)

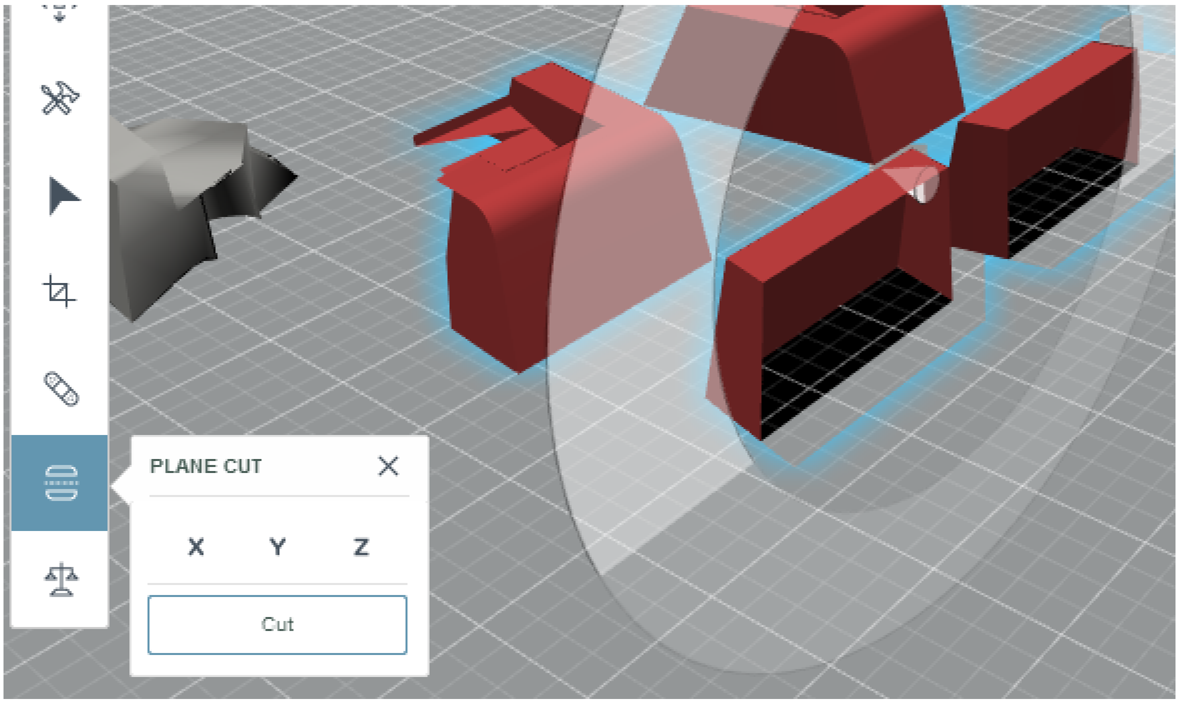


The patch holes command allows you to do exactly that - close holes that were created by moving or deleting faces.

Select the desired patch type, and click on the affected region:

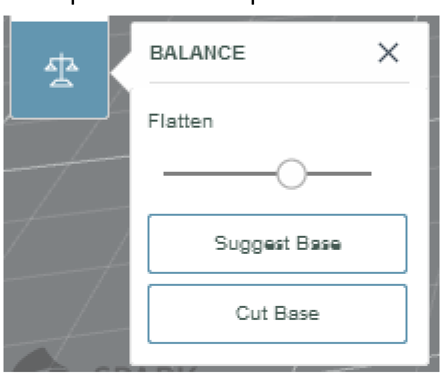


The plane cut tool allows for even more fine tune cutting and dicing of your model

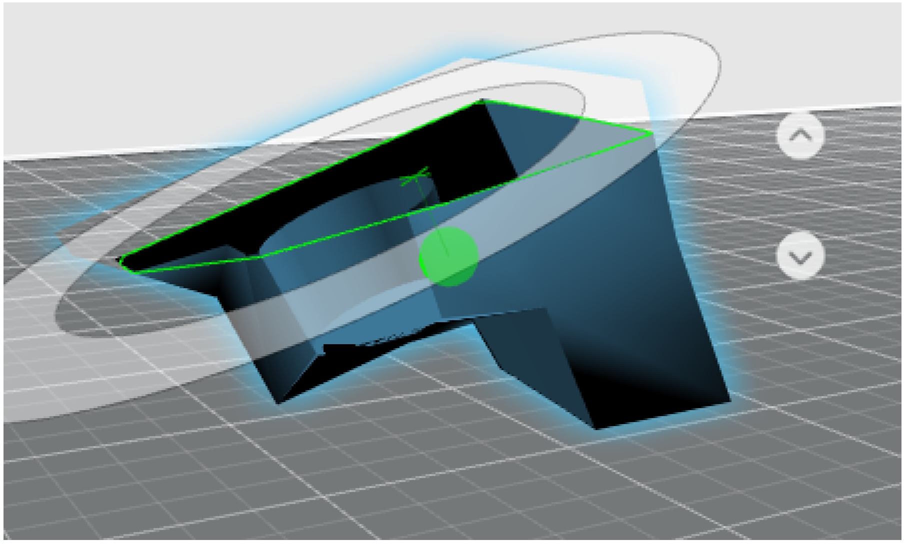


Select the plane about which you want to make the cut, drag the arrow to position your plane, and click on cut.

This operation attempts to also rebuild the cut face.



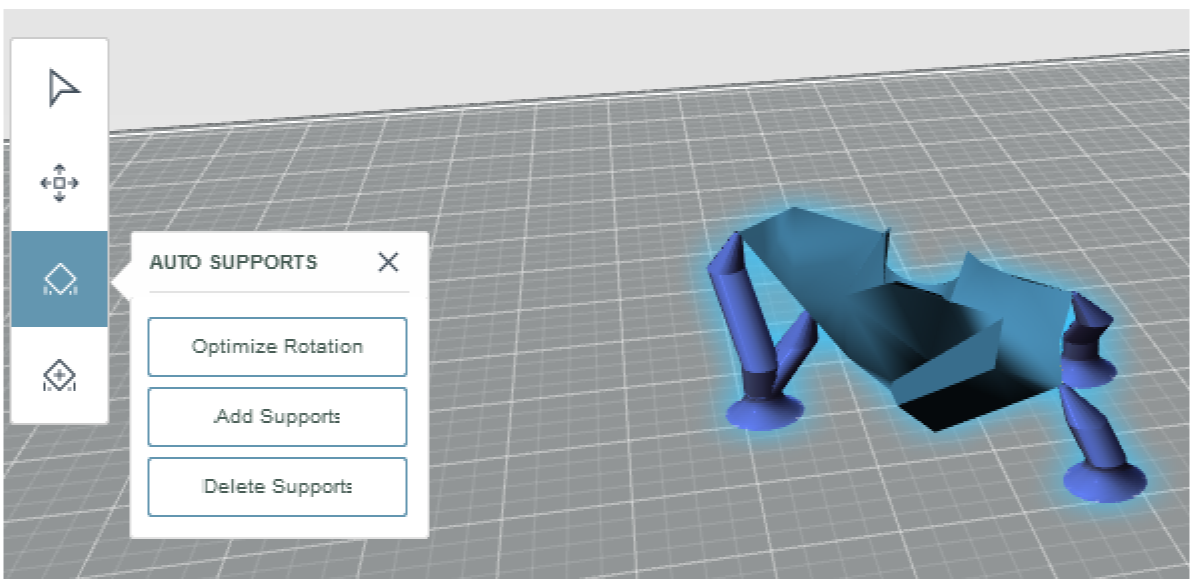
Finally, the balance tool will display the CG of the part as it gets build, displaying a green (or red ball) where the CG will be.



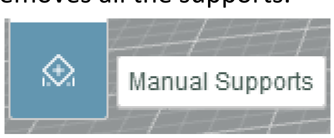
Use the suggest base to view the distribution from another angle. If you piece doesn't have a flat base, use the cut base command to make a region flat. (and cut the model in the process

Supports

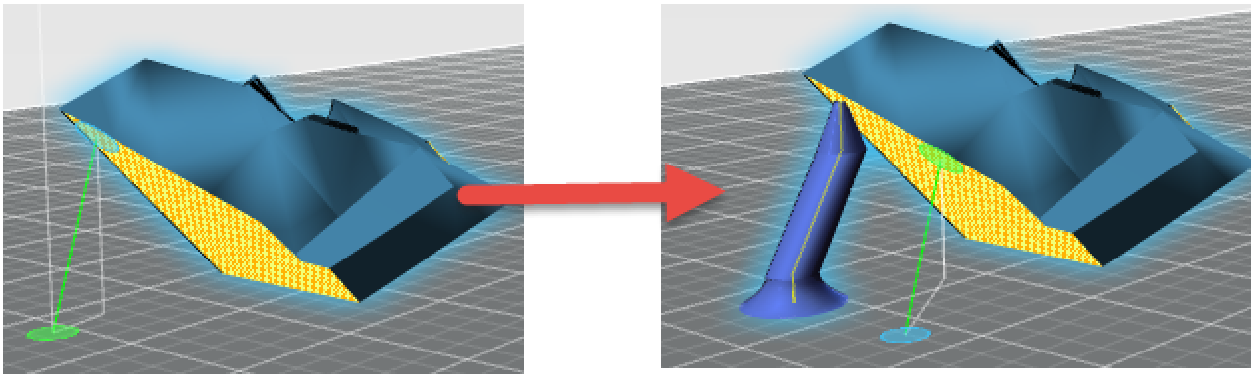
The supports section will has additional editing tools to add support and orient the piece.
Rotation optimization will rotate the part so that it has the least amount of overhangs.
Add supports will generate the supports of the tree type seen in the picture below



Delete supports removes all the supports.

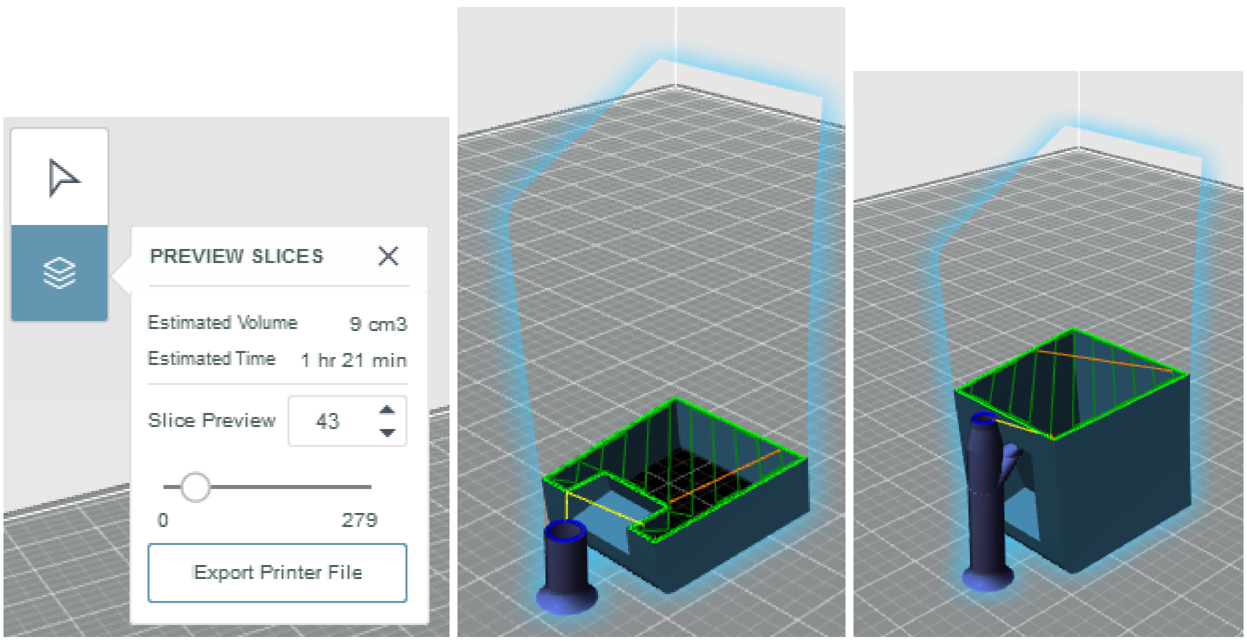
Manual supports 

Allows for the placement of supports manually. Guides appear to show what the corresponding vertical projection of the support is, as well as what areas need support (colored in yellow)



Preview

Preview allows the viewing of the toolpaths



Drag the slider to cycle between the different slices
This view also displays the estimated print time and volume

Export (gcode)

This simply saves the printer file in gcode file format, ready to be read by the printer.