PixiRite®

PixiRite Tutorials

The Tutorials are designed for PixiRite users，This course is appropriate for beginners and advanced students likewise.

Software Version:5.2.1

Preface

PixiRite is a software to improve your 3D Files for 3D Printing . This Manual will guide you to finish the Installation and Activation step by step.

PixiRite® is a trademark of MicroJet Technology Co., Ltd. , This software version is customized by netfabb GmbH.

PixiRite is able to used in certain 3D Printers, to provide full-color editing, repairing and modification.WRL(VRML 2.0) file format is easy to used in ComeTrue® , addwii® or certain 3D printers.

content

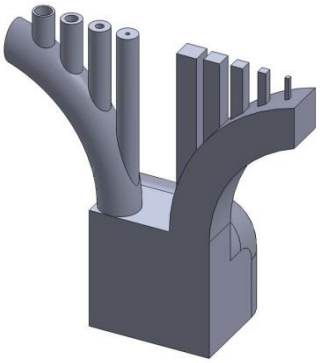
The PixiRite tutorials will give you an overview of PixiRite's most important features. To do the exercises, please use the files from this zip-file (available on the disk). Learn how to edit the part, improve its surface, make it fit into your buildspace and everything else you need for successful additive manufacturing and 3D Printing. For more information on netfabb's functionality, please take a look at manual on http://wiki.netfabb.com/

Content

B) becoming a pro

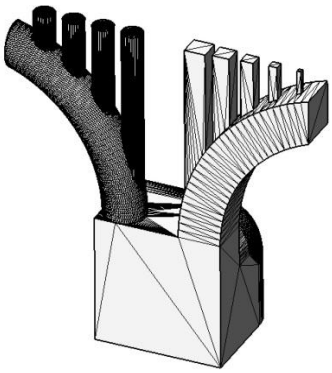
From CAD and STL - 3D printing terms

Constructing models in CAD



The surfaces of CAD files is determined by mathematical formulas. This works very well for constructing parts, but machines for additive manufacturing (AM) can't handle these formulas. That's why they have to be converted into another format.

From CAD to STL



STL for example has proved itself as a very reliable file format and is now a standard in manufacturing. It does not consist of formulas but of triangles that are connected to a mesh. (Other triangle mesh formats for AM are e.g. OBJ, PLY or AMF.)

When converting from CAD to STL there often happen rounding errors and so it can happen that some models have to be repaired, edited and improved before printing. PixiRite bridges this gap between CAD and machine.

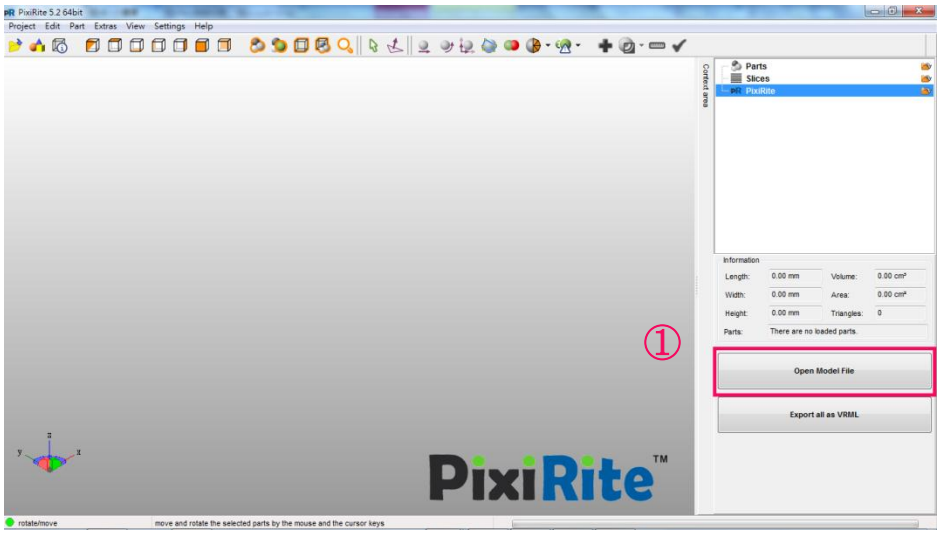
Basic vocabulary

Additive Manufacturing (AM)	Term for all production processes that build up parts in layers. (e.g. sintering, 3D printing, stereolythography)
CAD	**C**omputer **A**ided **D**esign，for designing and constructing 3D models.
Converting CAD to a triangle mesh	All CAD formats need to be converted into a triangle mesh to be repaired and edited.
Layer, slices, toolpath	Last step before printing: Every part has to be sliced into layers to be built up from bottom to top. For example of FDM, a toolpath additionally includes info about printhead speed, temperature etc.
STL	**S**urface **T**esselation **L**anguage, triangle mesh, reliable and most common 3D format for AM.
Triangle mesh	All formats for AM base on a mesh of triangles (consisting of three points and edges), before they can be further converted to slices.

Easy to repair in 4 steps

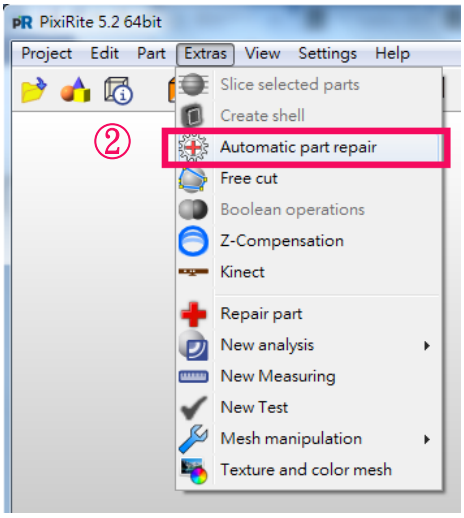
Content

This tutorial will teach you a quick start for repair parts by using PixiRite in 4 steps.

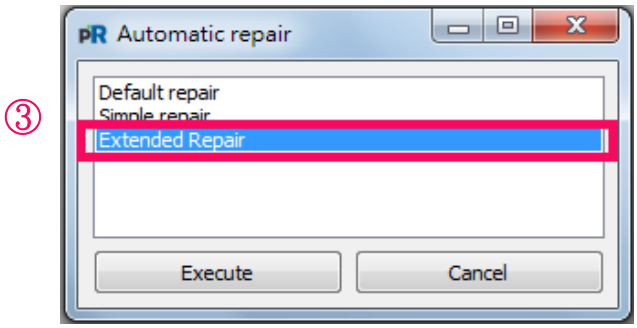


Open PixiRite and click **Open Model File** on the bottom of right column to open parts.

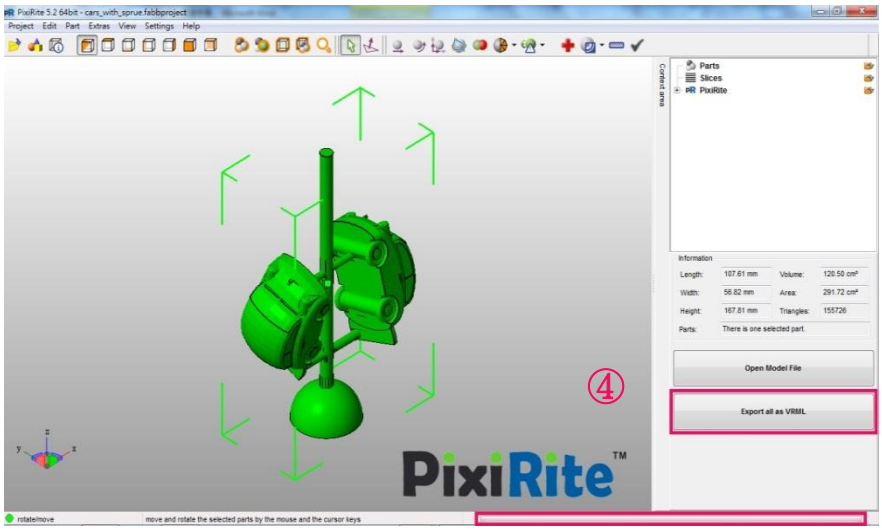
Note: You can refer to chapter 1 of this tutorial to open parts in other methods.



For Automatic repair the part, go into **Extras menu** and select **Automatic part repair**



Choose **Extended Repair** and click **Execute.**



When the part has been repaired (you could refer the schedule bar at the bottom right), Choose **Export all as VRML** to export the part which is compatible of ComeTrue ® series 3D Printer.

Note: Apply **Export all as VRML** to export the part, PixiRite will Automatic merge all shells to one. If you want to export one of the parts in project tree, please refer to chapter 5 in this tutorial.

If you want to confirm further the further status of the parts, please refer to chapter 6 : **complex parts,** chapter 9a **error overview** and chapter 14.

Note：

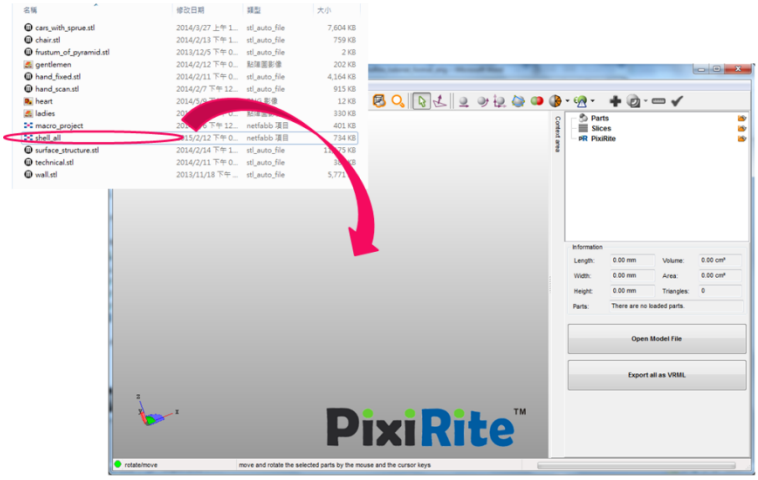
The function of automatic part repair can fix errors of mesh at parts automatically, change them into single shell and be compatible of 3D Printers. But PC (or we say PixiRite) can not recognize common complex shapes as like human's brain. For example of the hand in chapter 9b of this tutorial, PC has no the ability to know the texture, exterior and the numbers of finger what user really want. So, User NEED to be familiar with PixiRite and to comprehend the limitation of your 3D printer, then you will have good printing quality.

1. Open & view parts

Content

This tutorial will teach you how to open view several parts in PixiRite

Open parts



Open part of *shell_all.fabbproject*, open the folder where you have saved the tutorial file, select the part and pull it into the PixiRite windows (The usual way via the Open/Add parts icon is possible too, of course)
Make the **platform** visible: go to **View > Show platform** ◦

View parts (perspectives and zoom)

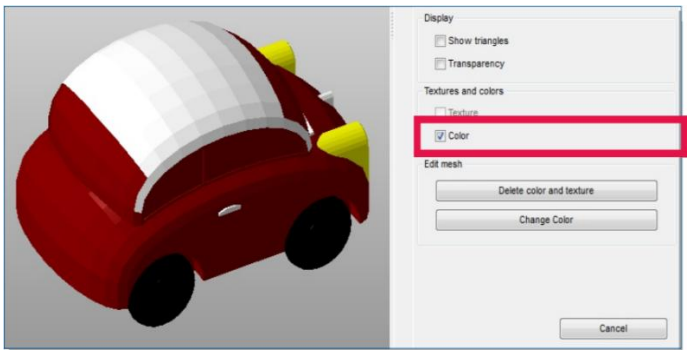
Select a part

Use standard views or hold left mouse button and move mouse to **rotate the view**.

Hold mouse wheel and move mouse to **shift the view.**

Use zoom options or role mouse wheel to **zoom** in and out.

Mesh viewer – display colors and textures



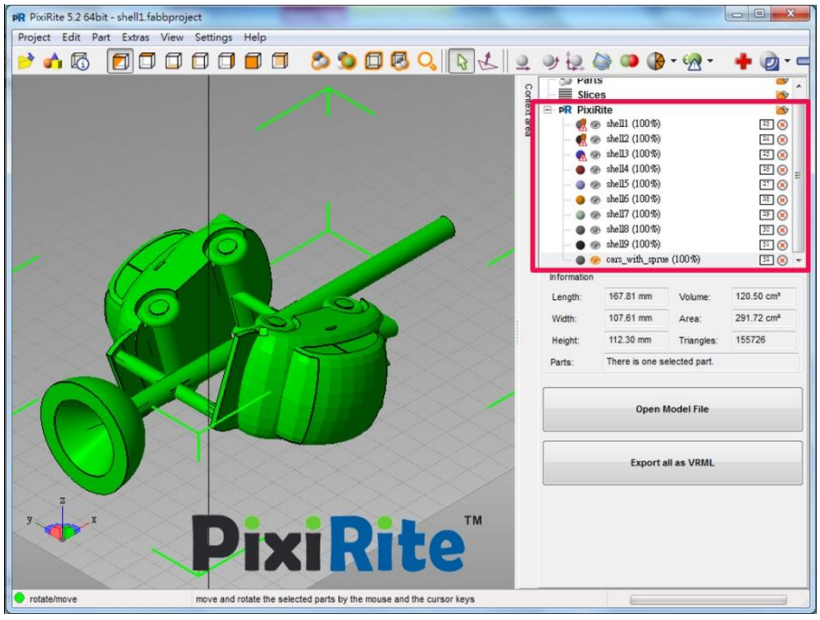
In this project the cars were originally designed with colors. Usually, STLs can't save and display color details, but this comparatively new format, called colorSTL, Display the color with the **Mesh viewer** (in the **Extras** menu). After viewing, you can **cancel** this module.

2. Automatically repair several parts

Content

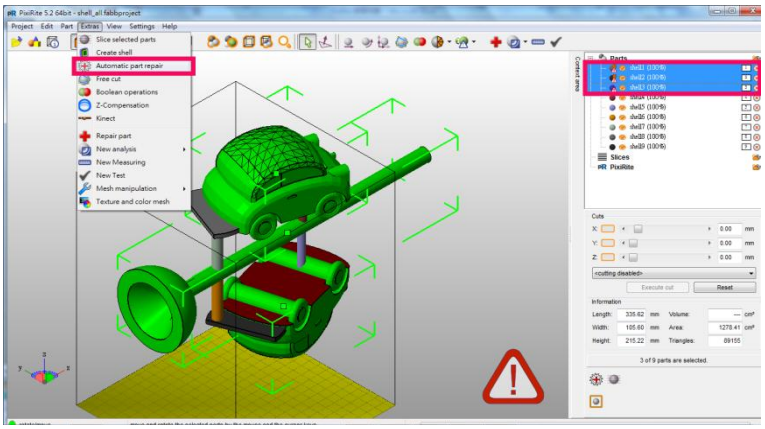
This tutorial will teach you how to find and select broken parts and how to repair them.

Signals for broken parts



On the top right , the **project tree** lists all the parts, Click on the little ⊞ to the left of the **Parts** section to expand the parts list. The small warning signs to the left of the part name indicate, which part is broken.

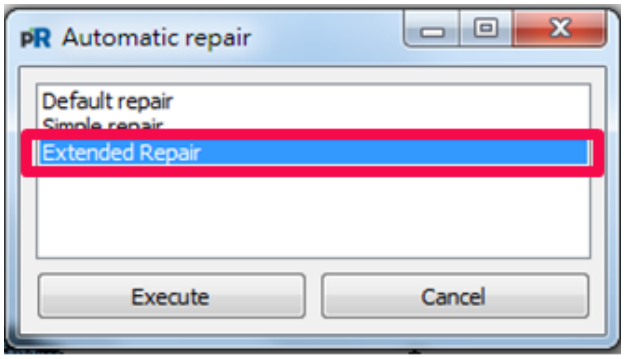
Select parts



Select the broken parts *shell1* by clicking on it the project tree. Hold *Ctrl* and click on *shell2* and *shell3* to Selected 3D models are marked green in the viewing screen.

Then go into **Extras manu** and select **Automatic part repair**

Automatically repair with powerful script



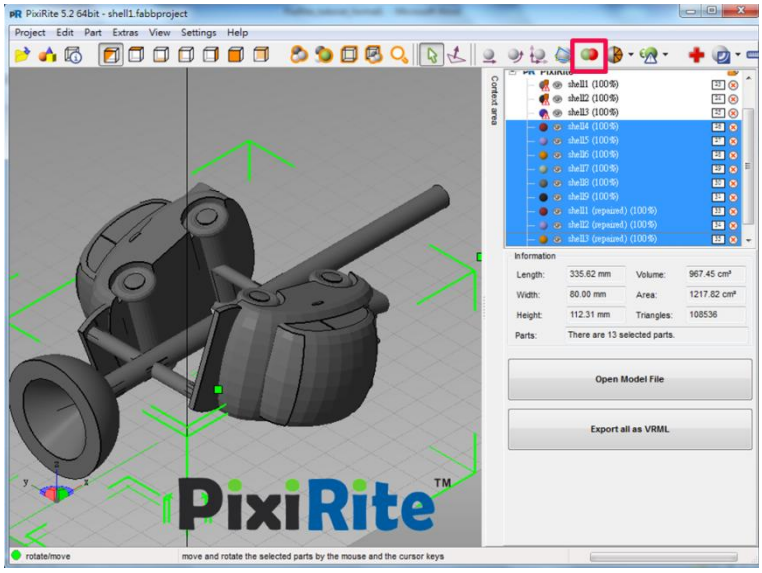
The **Extended Repair** is the most powerful repair script. Choose and execute this one to automatically repair all selected files. After a few seconds, the parts are fixed and replace the original files.

3. Merge parts (Boolean Operation)

Content

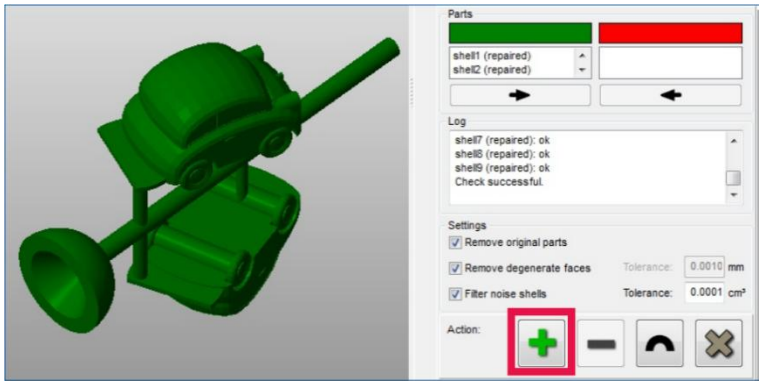
All shells will be merged on shell with the Boolean Operation.

Select all parts



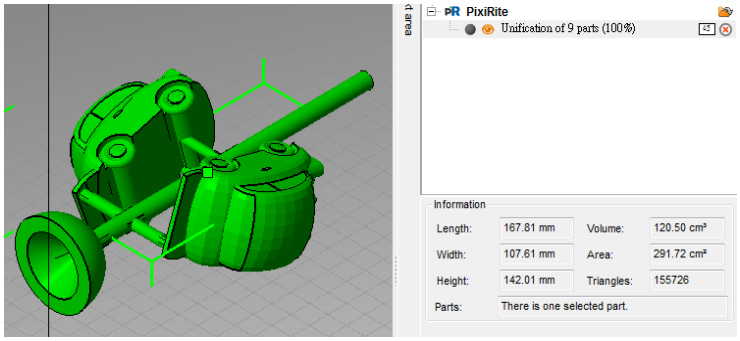
So far, the parts are saved as single shells (a shell consists of one close mesh). In order to merge them and make one shell out of it, **select** them and open the module for the **Boolean Operation.**

Boolean Operation-merge parts



All parts can be **merged** in the module. Click on the **Green plus** to unify them.

After a few seconds, the calculations are finished. Apply them by clicking on the **green check** mark.



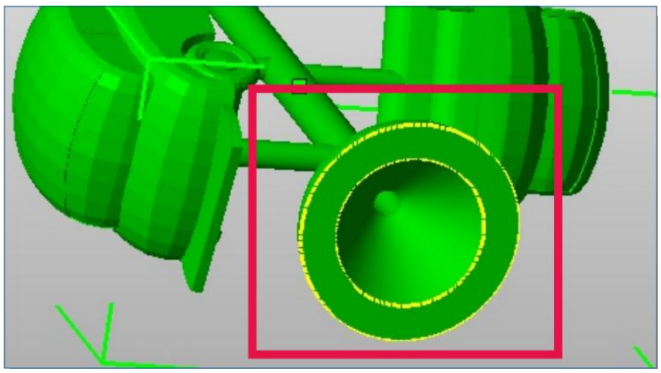
The originally 9 parts are now merged to one. This new part was automatically given a new name： *Unification of 9 parts* (see project tree)。

4. Simple part handling

Content

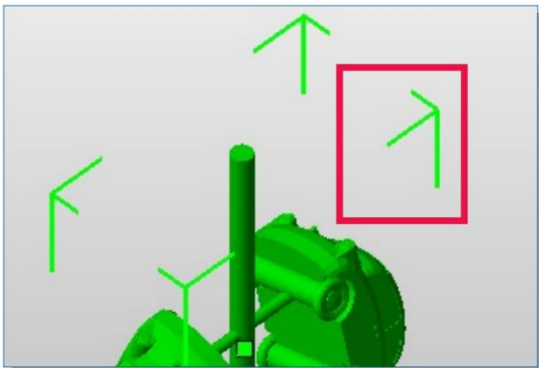
Learn how to align, rotate, move and scale a part.

Align



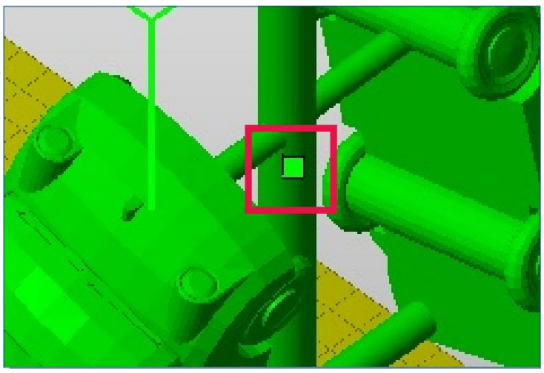
The part is now lying horizontally in the platform. To make it stand on its socket, choose the **Align to bottom plane** icon in the toolbar and **double-click on the socket's surface.** It will then stand upright.

Rotate



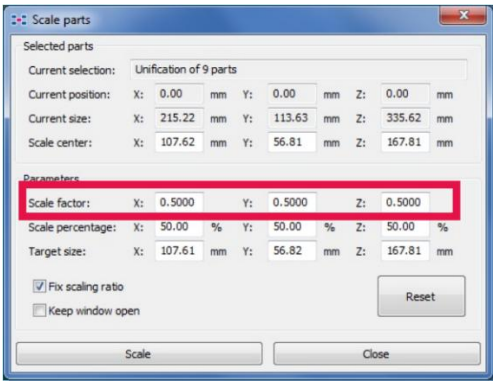
Alternatively, a part can also be rotated by holding and draging the green corner brackets of the selected part or by entering exact values in the **rotate** dialog (in toolbar)

Move



Move the part to the origin of the platform. You can do this by holding the green little square in the middle of the selected part and moving the mouse. Or to be more exact, open the  **move** dialog and choose **to origin** and **translate.**

Scale



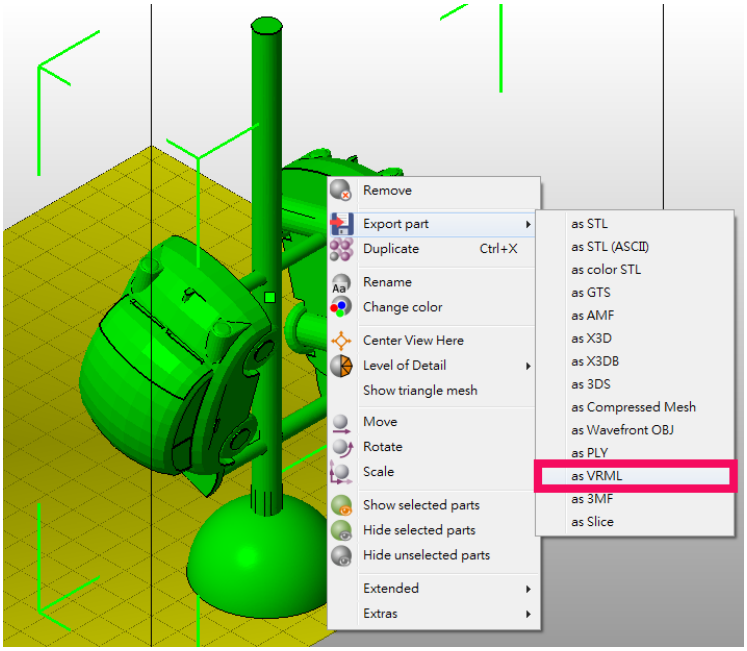
The part is relatively large. To make  it smaller, open to **scale dialog** (in the toolbar) and scale it down by **factor 0.5** – it'll be half as big as before (You can also scale by percentage or give it a certain size.)

5. Save and export

Content

Saving a part can happen via the **project** menu and **Save as.** It'll be saved as a netfabb project and can again be opened and further edited by PixiRite. But it's also to export the part directly into another 3D format:

Export part to VRML



To save a part, **Select** it, open the context menu with right-click, choose **Export part** and **as VRML** (Export as this format, the data of car's colors will be exported too).

Enter the **new file name** *cars_with_sprue1.wrl* in the export dialog and **Save.** (There is already a file called *cars_with_sprue1.stl* – it can be used if someone wants to leave out tutorials 1-5.)

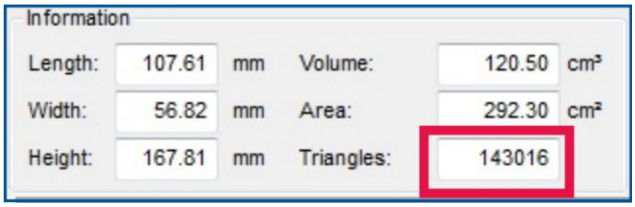
You have now completed the beginner's project. The advanced tutorials will continue on the next page.

6. Make huge part workable – reduce file size

Content

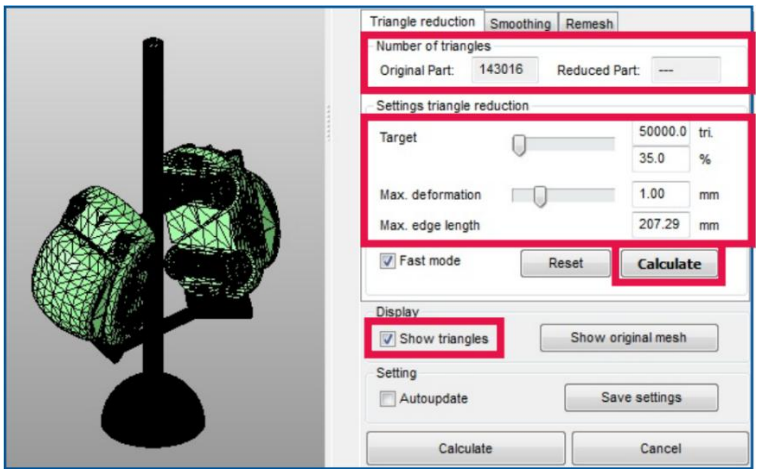
Huge parts are those, that are not necessarily large in dimensions, but have lot of triangles ("a lot of" is an elastic term here, but you might say that it starts at about 100,000 triangles). The more triangles there are, the longer all calculations last. Besides, many machines have a limitation when it comes to file size or triangles count. With PixiRite, it's easy to reduce triangles and the file size and still preserve the surface quality.

Complex parts



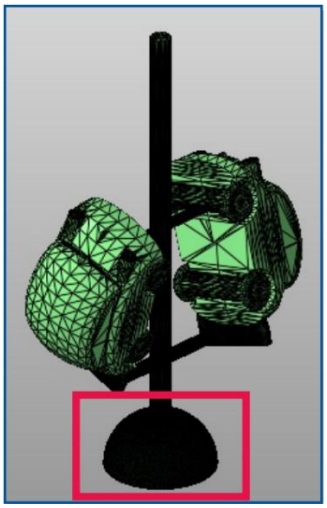
Load the part *cars_with_sprue.stl* and take a closer look at the **information box** on the lower right corner of PixiRite. The number of triangles amounts to about 143016 triangles (this result will be various with different versions). That we want to reduce.

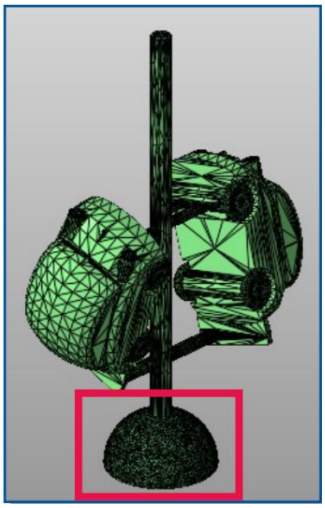
Set target for reduction and calculate



Select the part, open the **Extras** menu and choose the **triangle reduction.** On the top of the new tabsheet, you see the **original number of triangles**. In the **Target** box below, you can determine to how many triangles you want to reduce the part. Enter *50,000* tri, there. Then you can enter the max. deformation. This means, by how much the original shape may be changed. Enter *1.00* mm there. Activate **Show triangles.** Click on **calculate** and you'll see, that the target of 50,000 triangle was reached by PixiRite.

Compare parts





When you press and hold the button **Show original mesh,** you can compare original and changed mesh. Click on **OK** on the bottom of the tabsheet to apply the calculations and when you are asked to **remove the old part,** click yes.

To compare the meshes technically and in detail, please take a look at the **Mesh Compare** feature (chapter 6.10 in the manual).

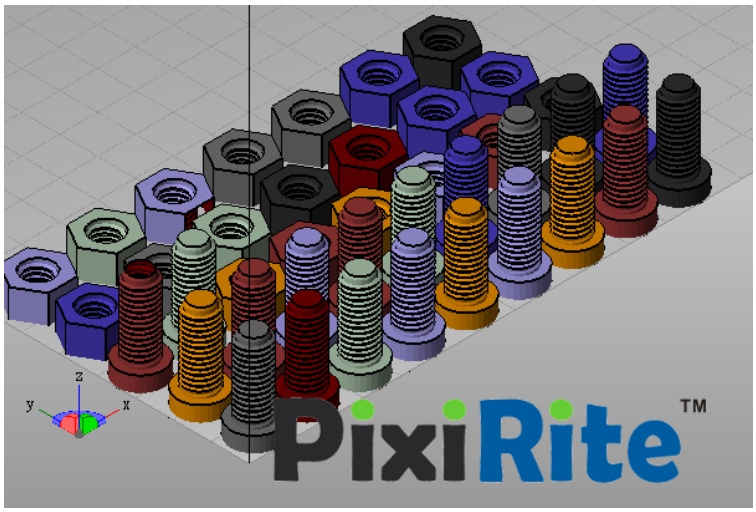
After this tutorial, you can either save or delete the part. We won't continue working with it.

7.Define your own macros

Content

If your parts always require the same treatment (e.g. repair, scale, rename), it is possible to define your own macros in PixiRite. Once created, such a script can simply be applied on all parts and PixiRite does the rest automatically. In the tutorial, we'll adjust an existing script for a fictional customer's project, which can be applied later on any other job.

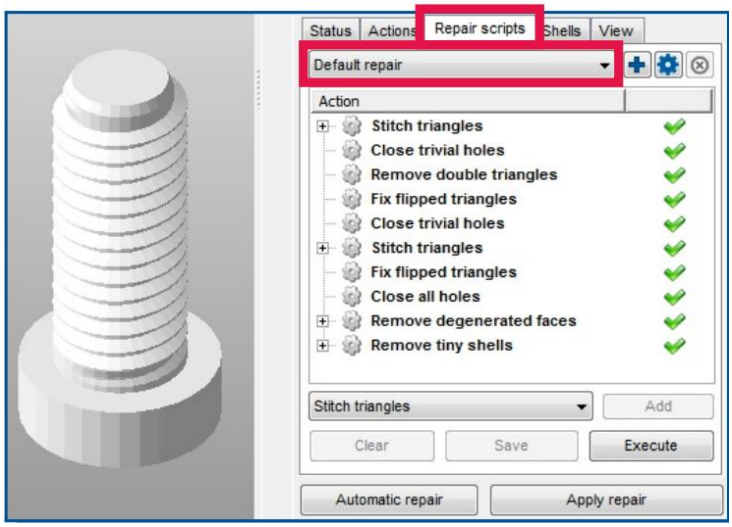
Open a fabbproject



Open the file *macro_project.fabbproject.* We will use this project to define a macro that will automatically repair all files and rename them. Like this, we will assign them to the project of our fictional customer called "Jones Design".

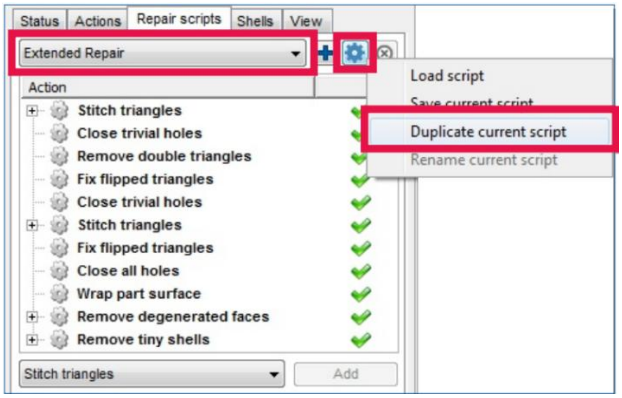
To define a macro, select the first file in your project tree and go into the **repair module** via the red cross in the toolbar.

Duplicate existing macro (=script)



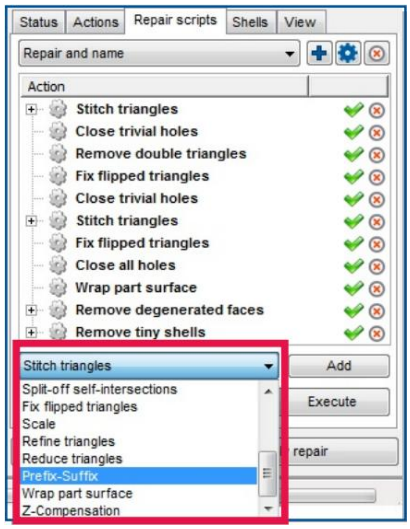
You're now in the repair mode. The toolbar on the top and the tabsheet on the right have changed – we'll go through that module later in the repair tutorials. For now, go to the **Repair scripts tab** in order to create your own macro (here called script).

Expand the first dropdown menu. There are three predefined scripts: Default, Simple and Extended Repair. They're all capable of repairing a part depending on its complexity. The Extended Repair is the most powerful one.



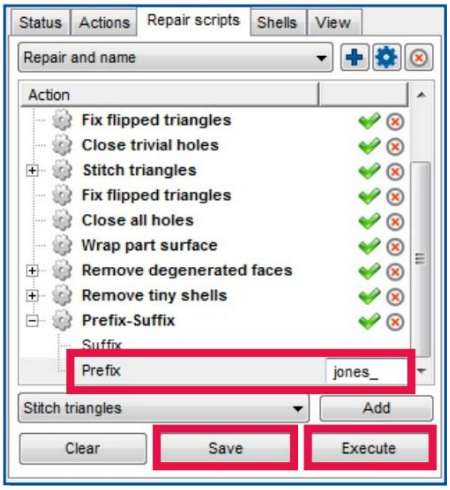
Choose the **Extended repair** and click on the little **gear** next to it in order to **duplicate** it. That way, we use the predefined powerful repair and simply add the action that we need. Name the new script *Repair and name* and press **OK**.

Adjust it to your needs



Open the dropdown menu below the actions list. Now scroll down to **Prefix-Suffix**, choose it, then press **Add**.

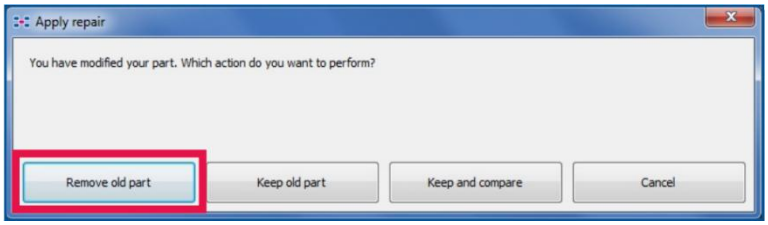
Specify the new macro



Click in the empty space behind the Prefix section and type in *Jones_* into the new field. Apply with enter.

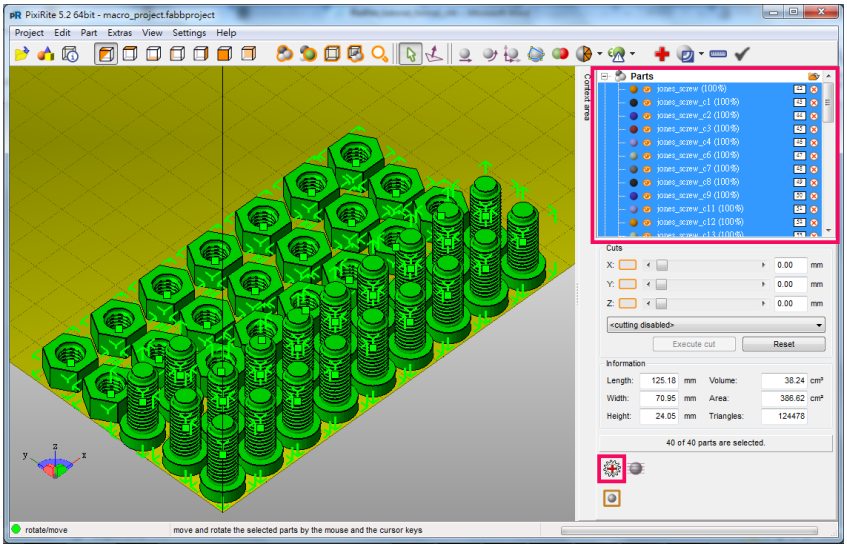
Press **save** (to apply the changes) and **execute** (to apply this new script to the selected part). Then **apply the repair** with the respective button on the bottom of the tabsheet.

Remove the old part

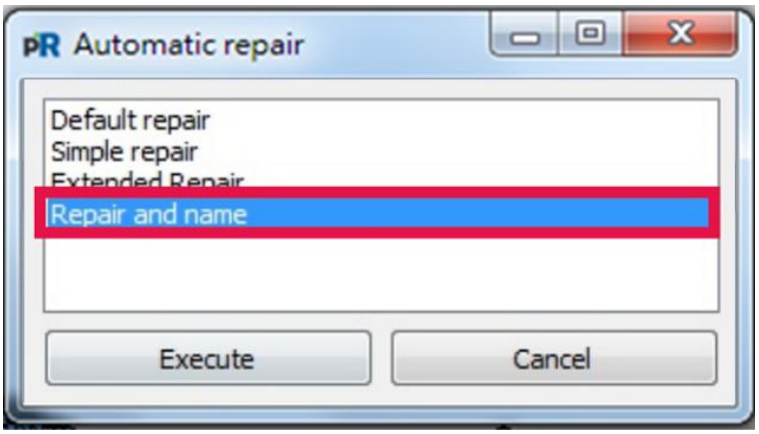


After that, you'll be asked what to do with the original part. Press **Remove old part** to delete the original and replace it by the repaired one. You can now see in the project tree, that the file was renamed to *Jones_screw*.

Apply the repair script to all the other parts



Select all the other parts (in the project tree, click on the top part, scroll down, hold Shift and click on the down part) and choose the **Automatic Part Repair** on the tabsheet.



Choose the new script that we've just created **Repair and name** and **Execute** it.

After a few seconds, all parts will be repaired and have the prefix *Jones_*. Like this, it'll be easy to later identify what part belongs to what customer project.

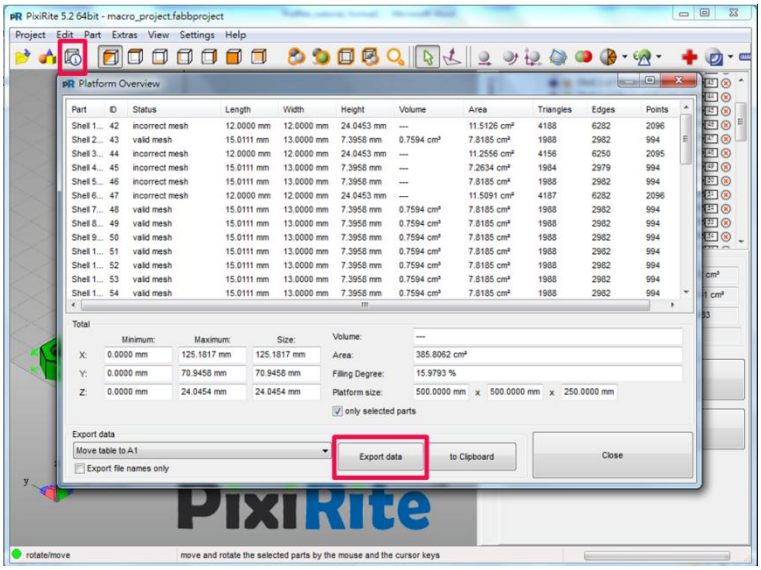
You can always adapt your scripts to new project or edit the parameters or=f the repair actions. Move info in our manual chapter **7.6 Automatic Rapair.**

8. Quote parts

Content

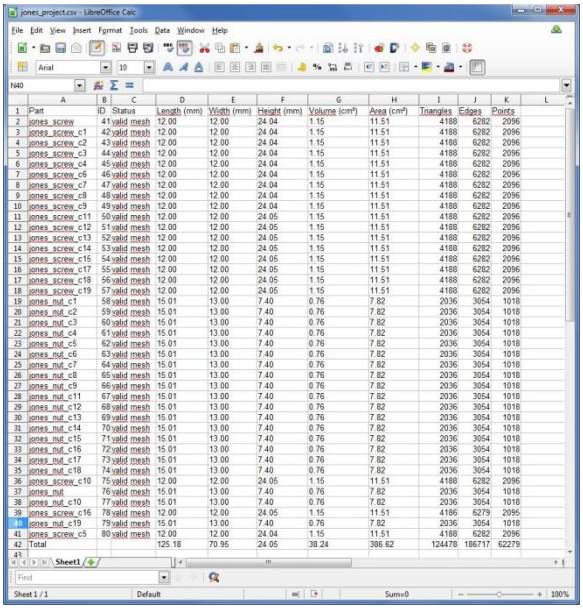
PixiRite supports you in creating quotes. Either with spread sheets for your calculations or with reports (templates) that you can hand out to customers.

Export part data to a spread sheet



Open or continue working with the *macro_project.fabbproject* (it's not necessary to complete Tutorial 2 before). Click on **overview** (in the toolbar) to open the dialog with all relevant details of the parts.

With a click on **Export data**, a .csv file will be created containing all important information on your platform. Save the file as *partinfo.csv*.



Open the partinfo.csv. You can easily copy these details into your own quoting tool or implement your calculation formulas.

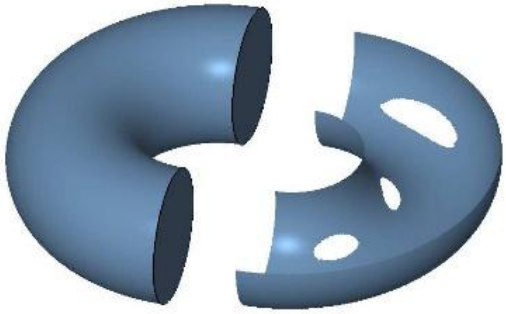
9. Repair and improve parts

Content

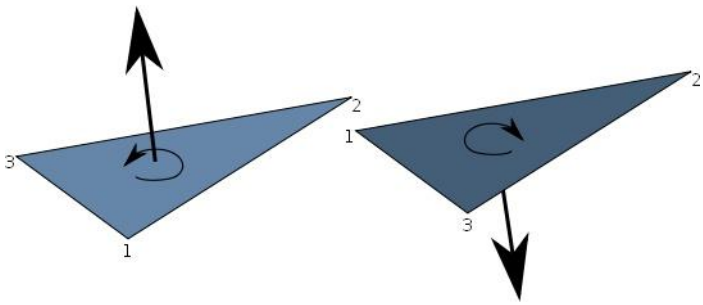
As computers are not as powerful as human brains and can't recognize shapes like we can, you sometimes have to give a little assistance to PixiRite. Parts, e.g. scans, can be so broken, that it's necessary to improve the surface manually. This tutorial will first give you an overview of possible errors in a triangle mesh and then go through two examples of complexly broken files.

Possible errors in a triangle mesh

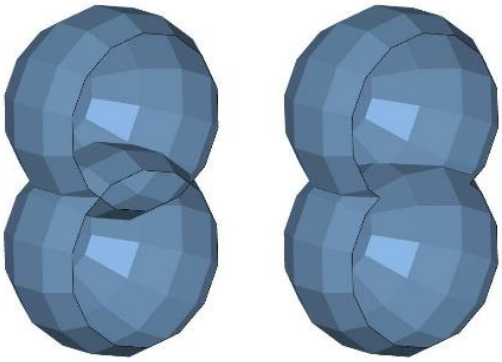
As described in the beginner's tutorial, errors can occur during the conversion from CAD to STL (and all other triangle mesh formats). Due to these rounding errors, there are three main issues:



Holes: Parts can only be used for 3D printing, if their surface is closed (=watertight). This means they may not have any holes or open border edges.



Invalid Orientations (invalid normals): For the conversion of triangle meshes into slice data ready for 3D printing, it is vital that the parts are oriented correctly. The orientation defines the outside and the inside of a part. In PixiRite, the inward orientation is always colored red.



Self-intersections: The surface of a solid body should not contain any self-intersections. These occur whenever triangles or surfaces of one part cut through each other.

In a vast majority of cases, the automatic repair in PixiRite is sufficient to make the parts printable. But there are sometimes files, that need special treatment, for example many scans. Learn how to repair them manually in the next chapters of this tutorial.

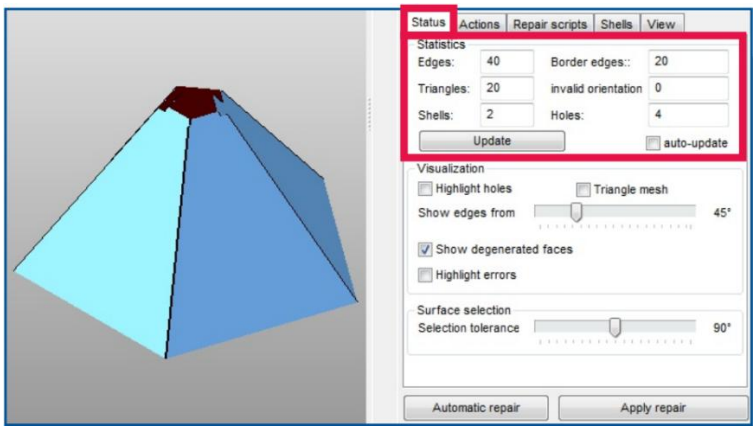
Correct orientation and hole closing

9a. Correct orientation and hole closing

Content

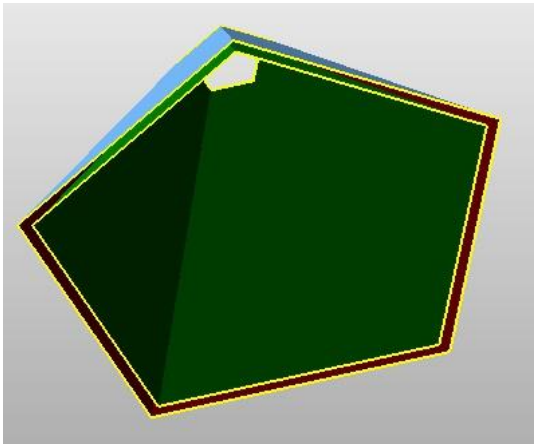
As described above, all triangles must have the correct orientation and the shells may have no holes. Learn how to flip shells and fix holes manually.

Error overview

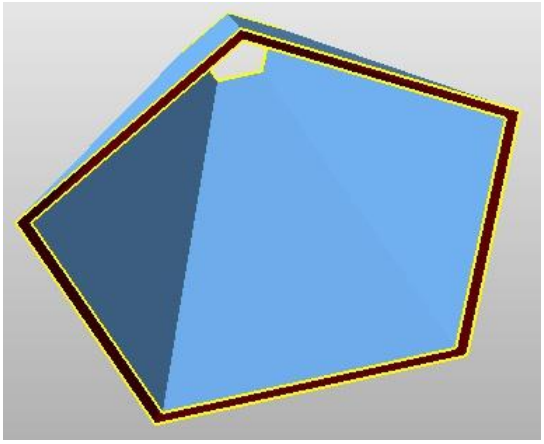


Open the file *frustum_of_pyramid.stl* and go into the repair mode ( in the toolbar).

In the new tabsheet **Status**, you'll see the **Statistics** of the part. Click on **Update** to refresh. Edges and triangles are simple information on the part. Border edges (open edges that are not connected to any other triangle), invalid orientations and holes are true errors. So we want to get rid of the 20 border edges and the 4 holes. A part with more than one shell can cause troubles, but doesn't always. Here we do need to unify the shells anyway, because we want to have the frustum of pyramid as only one part.

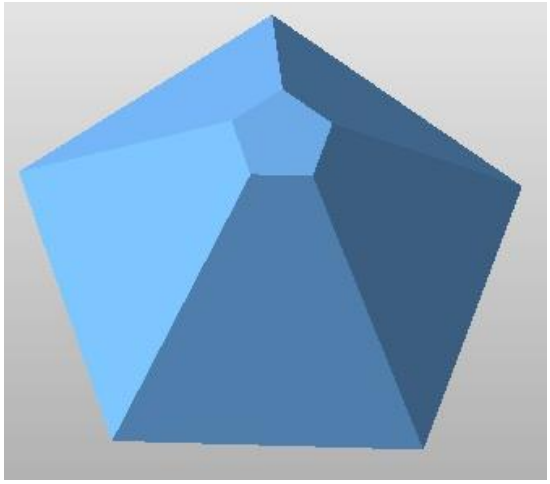


First, rotate the part so you can look at it from below. Then select the inner shell: Click on  the **Select shells** button in the toolbar and click on the inner red shell. It then turns dark green and is selected.

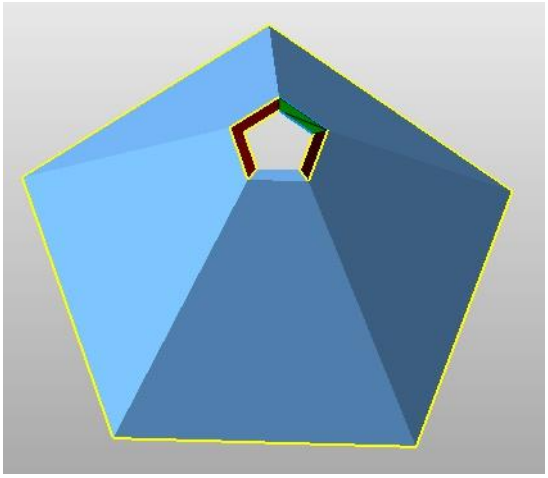


Then click on **Flip selected triangles**, then click next to the part to unselect it. The shell has now the correct orientation and turns light green. Deselect it and will be displayed blue.

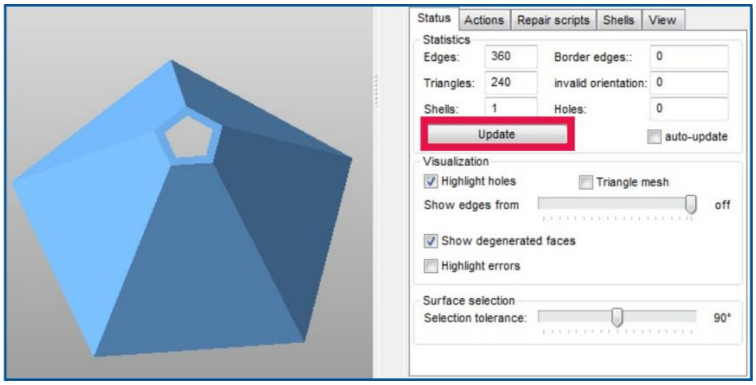
Closing the gap between the two shells



NOTE: After every manual repair step, you can try the automatic repair - in order to check if the rest can be done by the software. But an automatic repair still doesn't make sense for this part, Just to show you what's going wrong: here's the screenshot of it. All holes have been closed completely - though we only want to close the gap between the two shells. To undo a repair step, just press **Ctrl + Z**.



Choose the **Add triangles** button in the toolbar and move your mouse to an upper open edge. This edge will turn blue - click on it. Now move the mouse to the opposite edge and click on it, too. Two triangles will be added which connect both shells. Do the same for another edge on the upper hole and also twice for the hole on the bottom. This will be a help for the automatic repair to recognize which hole should really be closed. Now perform the **automatic repair**. The **Default repair** is just fine for that, as we only need very simple repair steps for this.



After the repair, click on **Update** again to see that there are no errors left. You have successfully repaired the part. Now press **Apply repair** and then **Remove old part**.
Save or discard the repaired frustum of pyramid to complete this chapter. We won't continue working with it.

9b. Repairing scans and improving surfaces

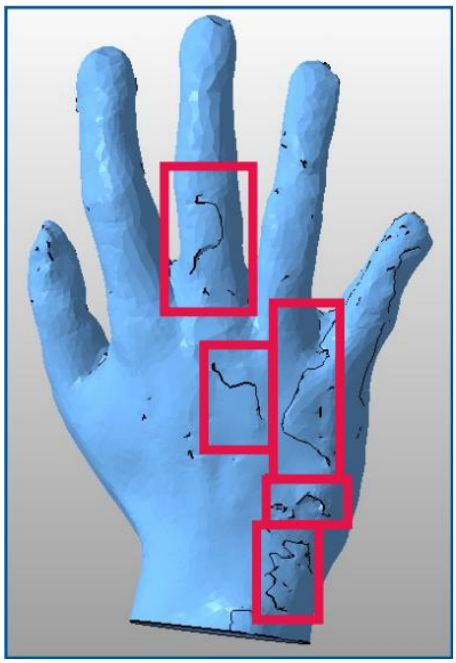
Content

Scans are often seriously broken. It's not always enough to close the holes, but you also have to cut out irregularities and smoothen the surface. Learn in this tutorial how to repair and improve a models surface quality.

Error overview and automatic repair

Open the file *hand_scan.stl* and go into the **repair mode**.
In the **Statistics** (click on **Update** to refresh), you see that this scan has 15 shells, 1894 border edges, 9349 invalid orientations and 150 holes, Choose the **Automatic repair** > **Extended repair**. After the calculations, click on **Update** to refresh. You see that the part is fixed.

Bad surface quality

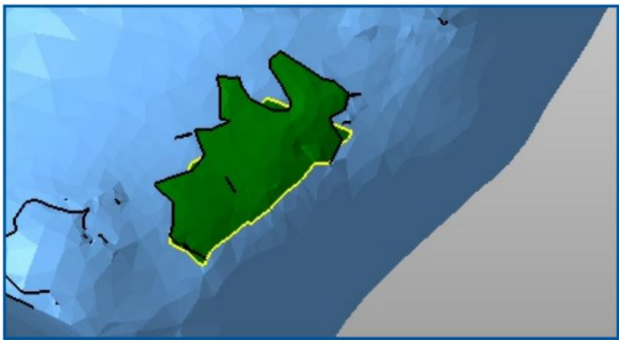


The scan is now fixed and watertight. But take a closer look at the areas that are marked on this screenshot. There are irregularities standing out of the part which give a terrible surface. The easiest way to get rid of them is to cut them out.

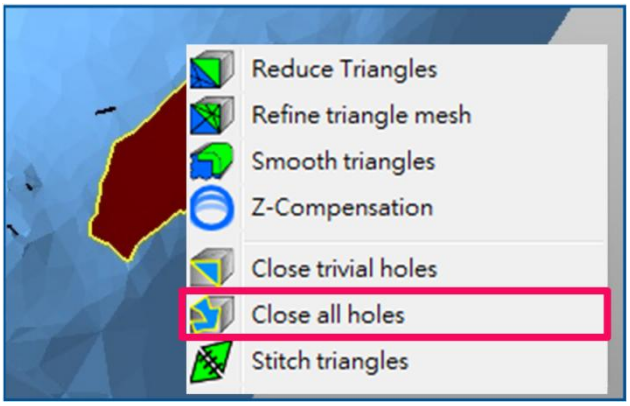
Cutting out irregularities

Select **Cut surfaces** and draw a line around a bump with a few mouse clicks. Try to cut out a surface as little as possible. Then

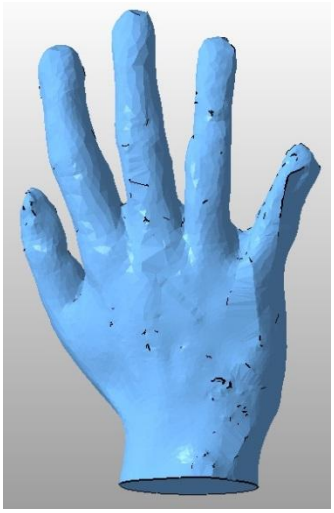
right-click next to the cutting line and choose **Insert to mesh**. This way, the cut out area becomes a new shell.



Choose the Select **Shells** mode (in the toolbar) and click on the new shell. Delete it with the **DEL key**. There's now a hole in the surface.

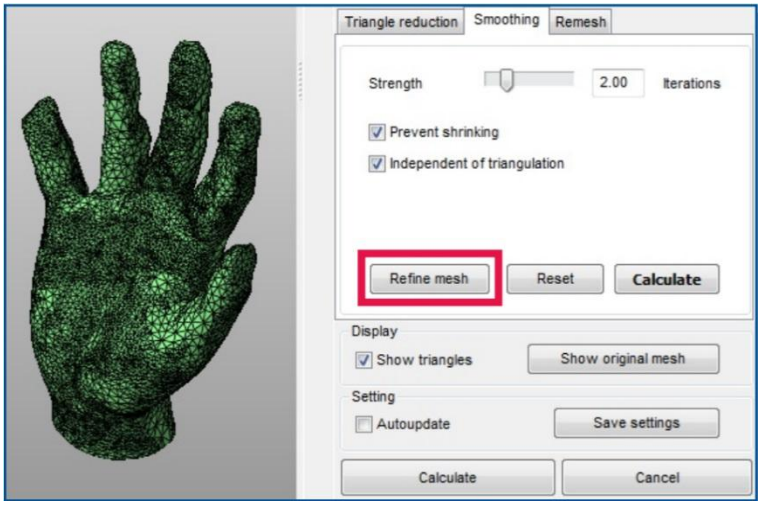


Right-click close to the hole (= very close to the yellow line) to open the context menu. Then choose **Close all hole**. The new triangles make a relatively flat surface. We'll later improve that. But first, **cut out all the edgy parts** as marked in the screenshot on the page before.



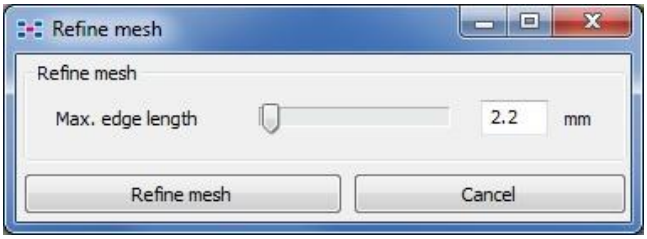
When you're done, the hand should look similar to this. There are still a few bumps left, but this shall be okay for now. **Apply the repair** and **remove the old part**.

Improve surface

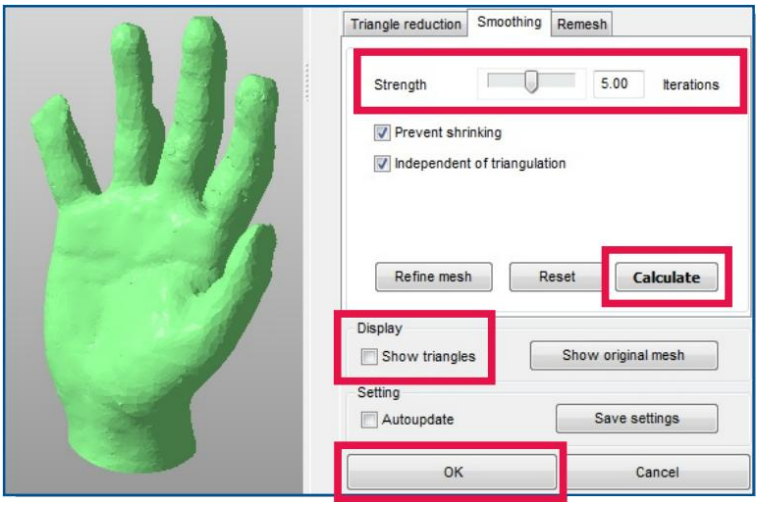


When you're back in the default mode, **select the hand**, open the **Extras** menu and select **Mesh smoothing**.

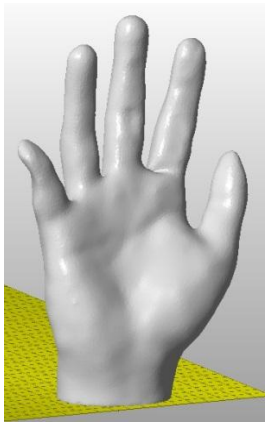
Take a look at the model and how large or small the triangles are. You can see that the mesh is pretty irregular, Choose **Refine mesh**.



In the new dialog, pull down the maximum edge length to *2.2mm*. All edges from all triangles won't be longer than 2.2mm then. The smaller the edges, the stronger the smoothing. Apply with **Refine mesh**.



In order to have a better view of the part, **uncheck the Show triangles box**. Then set the **Strength** bar to *5.00* Iterations (the more iterations, the stronger the smoothing) and press **calculate**. You now get a preview of the smoothened mesh. With pressing **Show original mesh** you can compare the original and the new mesh. Apply the smoothing with **OK** on the bottom of the tab and **Remove the old part**.



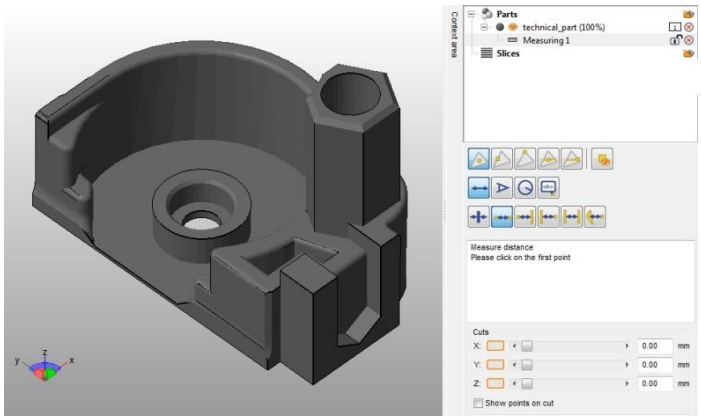
The surface of the hand has been improved a lot. You see that you sometimes have to combine automatic and manual repair to get the best results. Further information on the different repair modes and actions can be found in the netfabb Professional manual on. After this, you can save or discard the result, as we'll continue with another part.

10. Ouality assurance- measuring & analyzing

Content

The quality of your parts does not only depend on the correctness of the models, but also on your manufacturing process. Control the quality with PixiRite's measuring and analyzing tools.

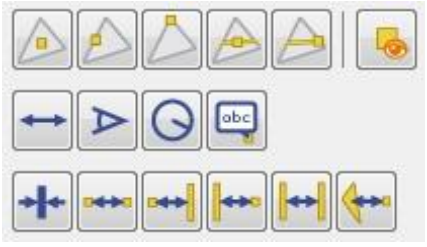
Measurement mode



Load the part *technical.stl* and open the **measurement mode** (in the toolbar, All new measurements will be listed in the project tree.

In PixiRite, you can set exact anchor points and therefore make very precise measurements.

Set the anchorpoints



Define the anchor points via the buttons in the tabsheet:

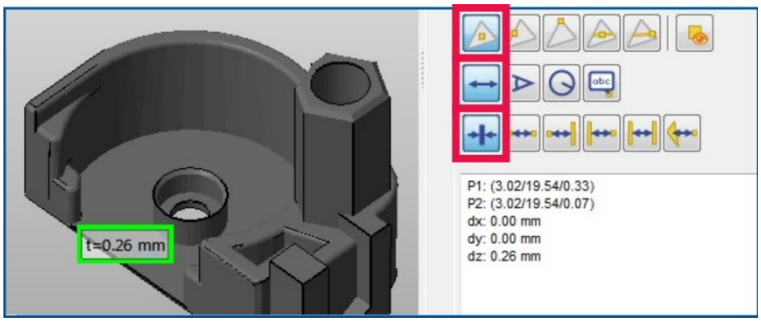
first row: define where the measured points should be placed: on a surface, a line, a corner, a cutting line or on a corner of a cutting line.

second row: measure a distance, an angle, a radius or if you want to place a note on the part.

third row: measure a wall thickness, from a point to another point, from a point to a line (or the other way round), line to line or from a surface to a point.

To set a measuring anchor point, always go through the lines from top to bottom. But let's go through it in a few examples.

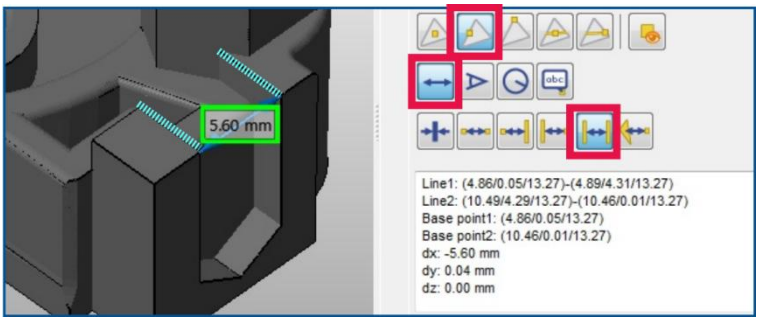
Measurement examples



Wallthickness

1st row: point on surface (as it can be on any point of the bottoms surface)
2nd row: measure distance
3rd row: wall thickness

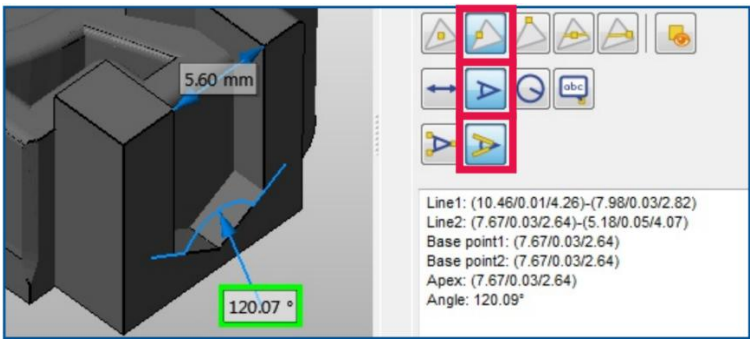
Then click on the bottom of the part to measure it's thickness. It's *0.26 mm*.



Distance of two edges

1st row: point on edge
2nd row: measure distance
3rd row: line-line (as it should not be just any point on the edge but the whole line)

Click on the edges that are marked blue in the screenshot. Rotate the view so you have a topview on the edge to hit it easier. The distance is *5.60mm*.

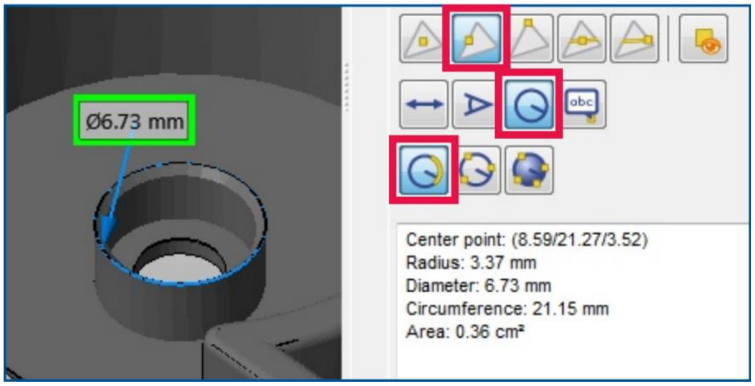


Angle

1st row: point on edge
2nd row: measure angle
3rd row: 2 edges

Click on the two edges as marked on the screenshot.

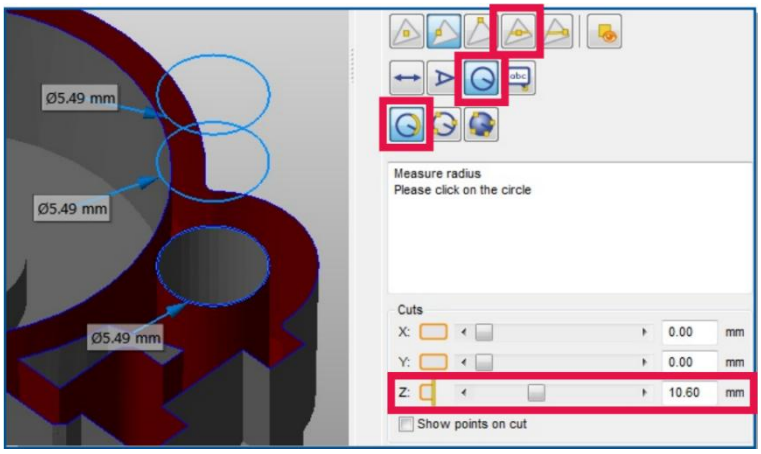
The angle is 120.07º. Press and drag the new measurement result (marked green) to pull it away from the part and to get a better view.



Radius

1st row: point on edge
2nd row: measure radius
3rd row: circle arc

Click on the outer circle as marked on the screenshot. The diameter is *6.73 mm*, the radius *3.37 mm* (see info in the tabsheet).



Measure self defined edges

If you want to measure the inside of a part, you can use the cutting lines as a help.

1st row: point on cut
2nd row: radius
3rd row: circle arc

In the **Cuts** box below, move the slider from the z-axis to the right until you see a blue outline on the tube. Click on the outline to make a measurement. Repeat this in another z-height, just like in the screenshot.

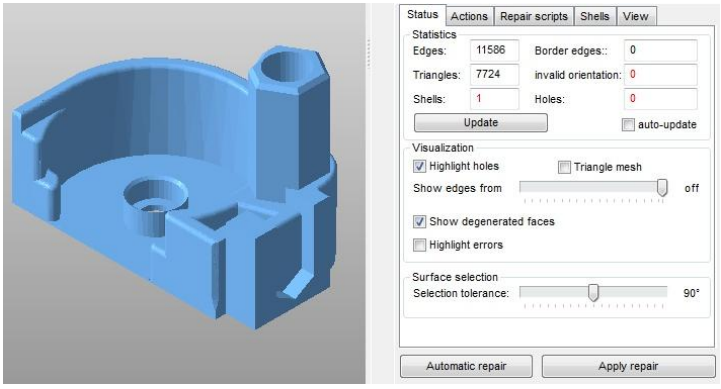
You have measured the part manually now, For standard analyses, you can also let PixiRite do the work automatically for you. See below.

11. Wall thickness-offset and hollowing

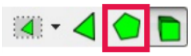
Content

Some parts have a too thin wall to be printed, some parts don't even have a wall but would be printed as a solid. Learn in this tutorial how to thicken walls to hollow them out.

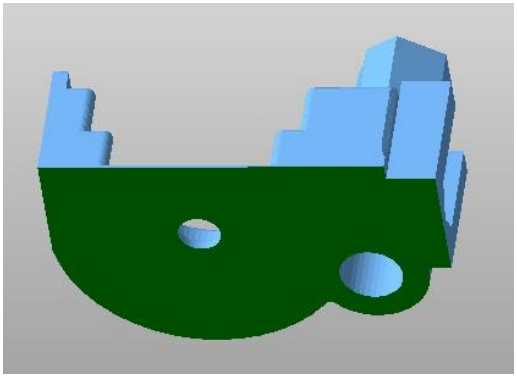
Extrude a flat surface



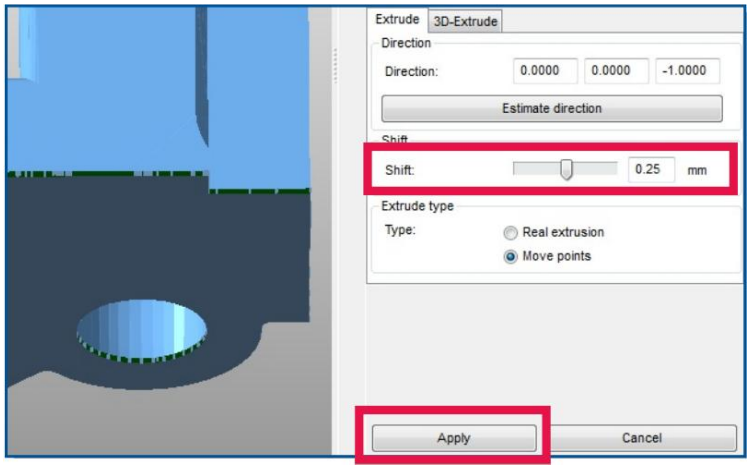
Continue working with *technical.stl* and go into the **repair mode**. To thicken a surface, you first have to select them. First choose **Select surfaces**.



Tip: By setting the selection tolerance (on the bottom of the tabsheet), you can define the maximum angle between triangles becoming part of a surface and the triangle you click on. You can test how it works with clicking on the back wall of the part and changing the angle of the selection tolerance consistently. This is not necessary for fixing this part, though.

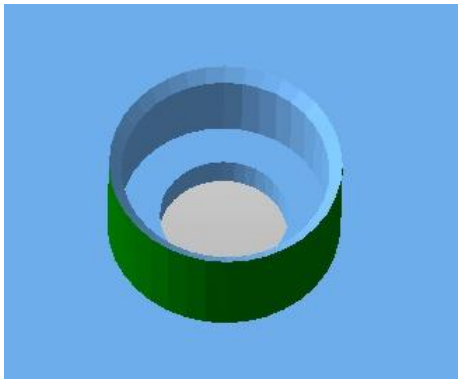


As you have seen above, the bottom was one of the areas that was too thin. It only had 0.26 mm. Rotate the view and **select the bottom**. Then go into the  **Extrude** mode.

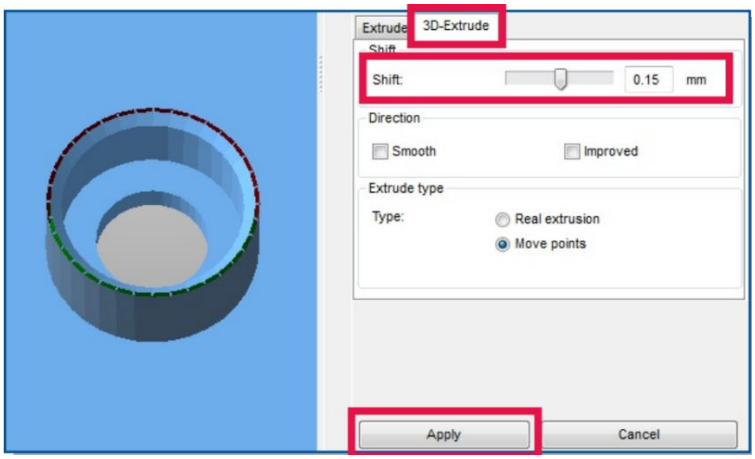


We measured 0.26 mm for the bottom wall thickness, so to make it 0.5 mm in the end, type in *0.25 mm* (a little more than needed to be safe) in the **Shift** field on the right. Then **Apply**. The bottom is now thickened.

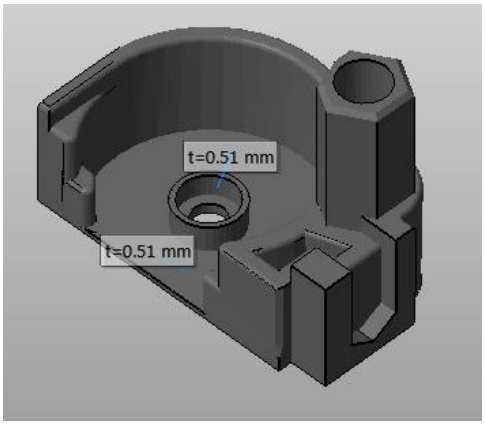
Extrude a round surface



Now select the round surface in the middle of the part. Choose  **Select Surfaces** again, **click on a triangle** of the surface, **rotate the view** (right mouse button), press **Ctrl** and **click on one of the other triangles** on this surface. Do that **until the whole surface is selected**. After  that, go to the **extrusion mode** again.

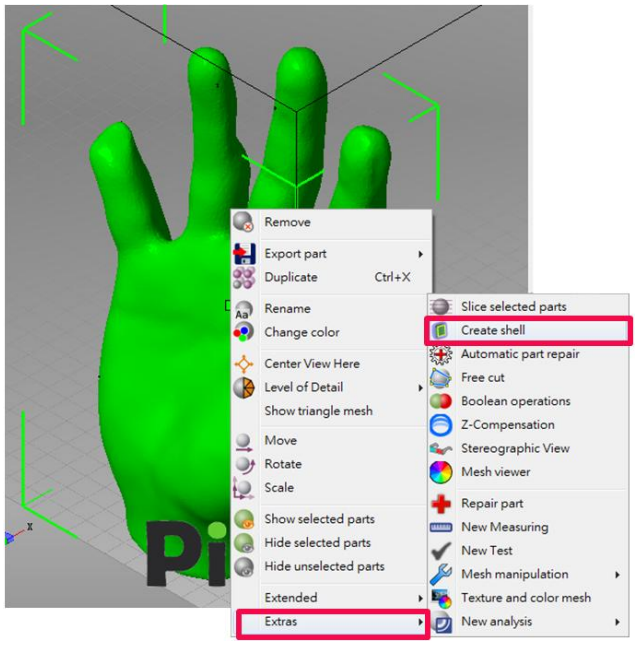


This area has a wall thickness of 0.36 mm. To extrude round surfaces, you have to switch to the **3D-Extrude tab**. Then type in *0.15 mm* (a little more than needed to be safe) in the **Shift field** and click **Apply**. Then choose **Apply repair** and **remove old part**.

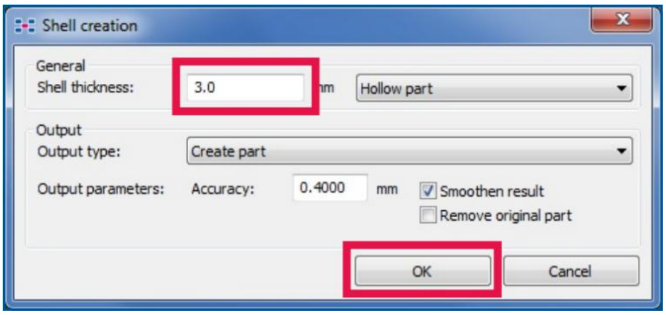


You can check if the wall thicknesses are ok now if you go to the measuring mode again. All values are at least 0.50 mm now. The part will also pass the wall thickness analysis successfully. After this, you can save or discard the result, as we'll continue with another part.

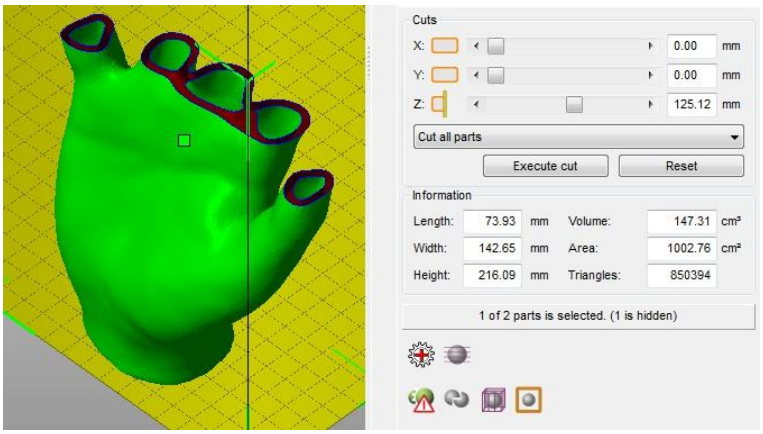
Hollow out a part



Load the part *hand_fixed.stl* and open the **context** menu (with a right-click on the part) > **Extras** > **Create shell**.



We want to create a wall thickness of *3.0mm*. Enter that in the **Shell thickness** filed in the Shell creation dialog and apply with **OK**.



Take a look inside the part with the cuts on the right. There are two shells now with the opposite orientation: the part is hollowed.

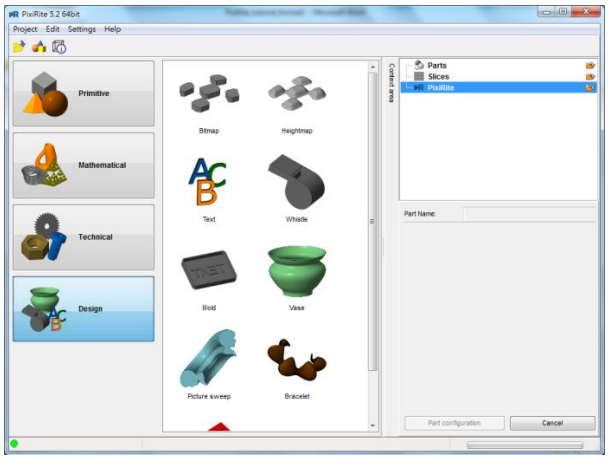
Tip: In the Shell creation dialog you can also make inner or outer offsets for the hole part. Just choose the respective option in the drop down menu of the Shell Creation dialog. See chapter **6.3 Create Shell** in the manual for further information. After this, you can save or discard the result, as we'll continue with another part.

12. Create own shapes with part library

Conent

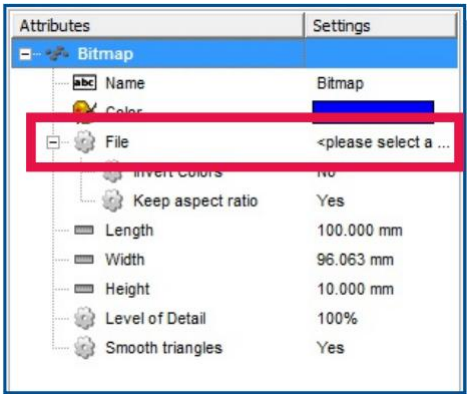
In PixiRite, you can use and edit predefined shapes (e.g. boxes, screws, bricks) to make completely new models. You can also load an image file and convert it to a triangle mesh. In this tutorial, you'll learn how to create your own wc signs.

The parts library- Bitmap



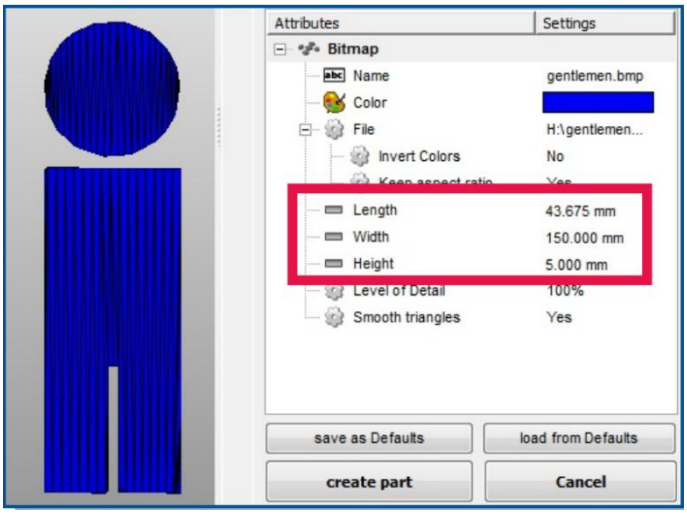
If you have other STLs left open, delete them in order to have a free new platform. Then open the **parts library** (in the toolbar) and go to the **Design** section, Open the **Bitmap** primitive with a double-click.

Select an image



Click on the text **<please select..>** and open the image *gentlemen.bmp*.

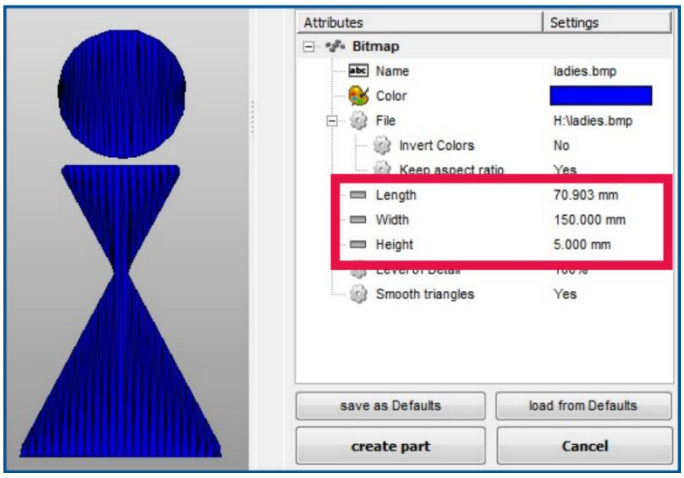
Set parameters



Change the following parameters by clicking on the numbers, entering the new ones and pressing enter.

Width: *150.000* mm (the length is adjusted automatically)
Height: *5.000* mm

Then press **create part**.

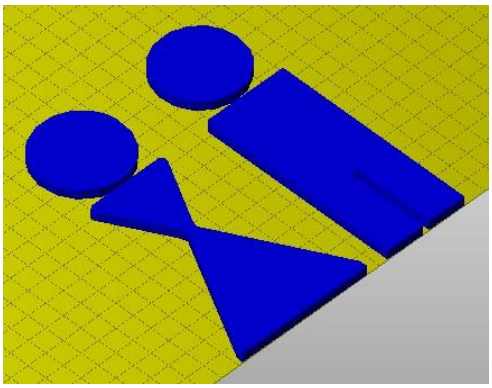


Load the image ladies.bmp and use the same parameters as above:

Width: *150.000* mm (the length is adjusted automatically)
Height: *5.000* mm

Then press **create part**.

Result



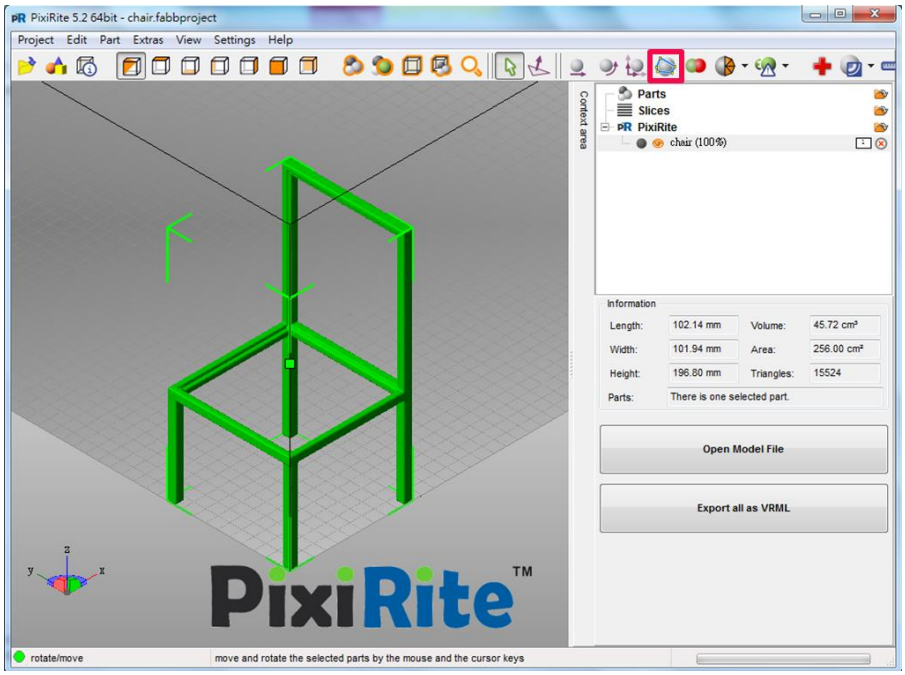
The images have become 3D models. Pull them a little a part from each other to get a better view. Now you can export them (see beginner's tutorial) and send them to your printer.

13. Cut parts

Content

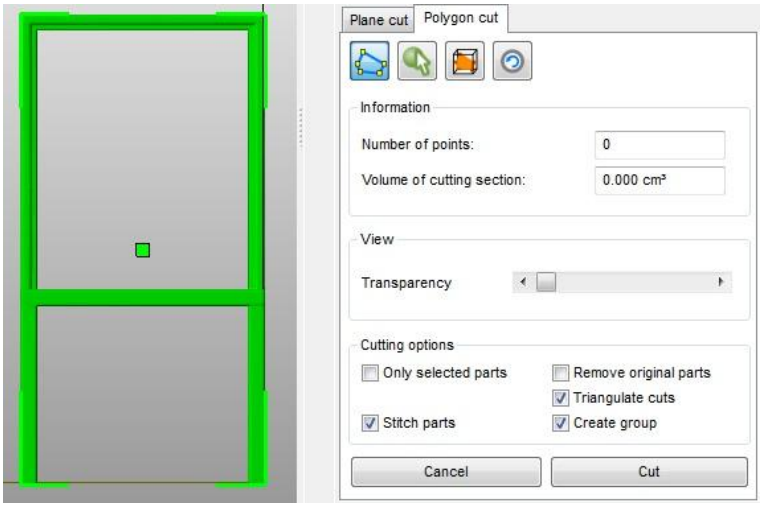
Parts often need to be cut - either to make them fit into the build space or to save support material. In PixiRite you can simply cut with the cutting box in the right tabsheet of the default mode. But to be more exact, cut with planes or polygons.

Open the cutting module

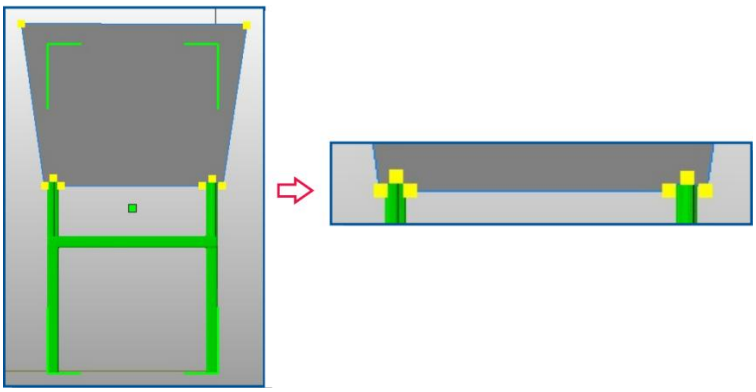


Load the file *chair.stl* and open the **cutting module** (in the toolbar).

The polygon cut

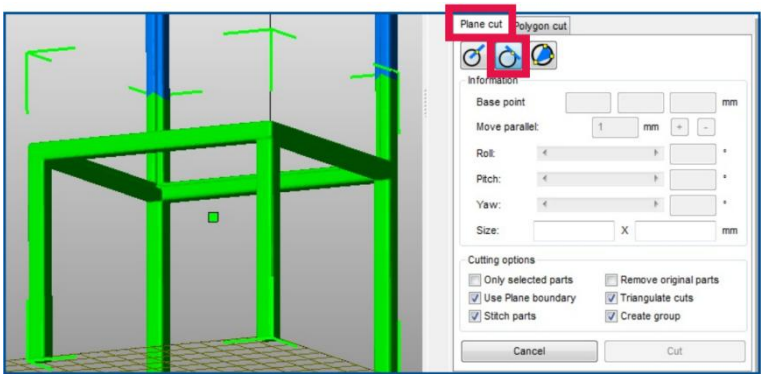


In the polygon cut it's very important to work with the **standard perspectives**, as the cut will always be placed in the same direction as you are looking on the part. Choose the view from the **left**.

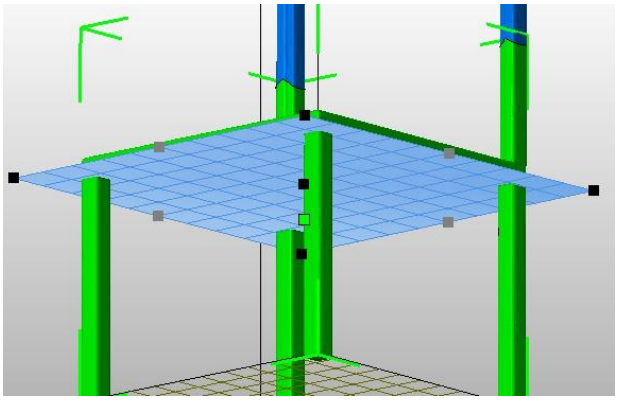


Set cutting anchors similar to the screenshot: Draw a frame around the upper third of the chair. On the bottom line of this chair, add two more anchors on each side so that two small bumps are cut out of the cutting area. When you print this chair then, these bumps will help you fit the parts perfectly together again. The notches on the bottom will make the two parts fit back together perfectly after printing. Then press **cut** to apply.

The plane cut



The upper part of the chair is cut off. Open the **cutting module** again, but his time, go into the **Plane cut** tab and choose the option **Create cutting plane tangential to surface**. Shift the view so you can see the chair from below.



Click somewhere on the bottom of the seat and a blue cutting plane appears parallel to the bottom surface, Scale and rotate the plane until it looks similar to the screenshot:

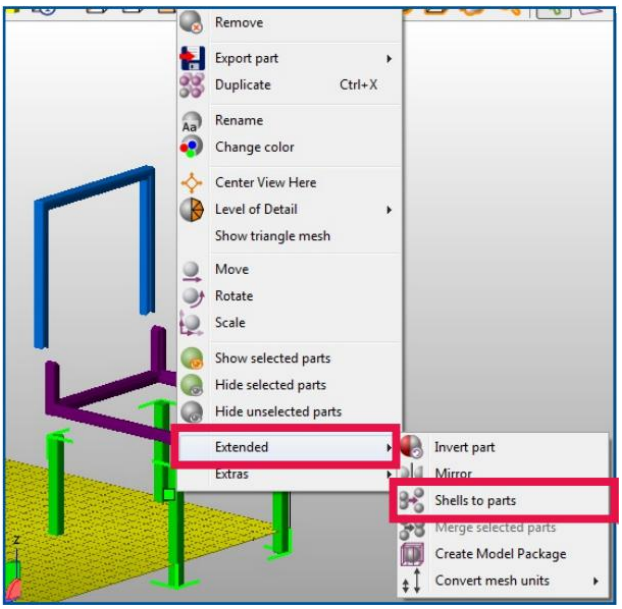
Click and hold the black/grey little squares on the edges of the plane


to rotate


to scale

Then press **cut** to apply.

Separate shell to parts



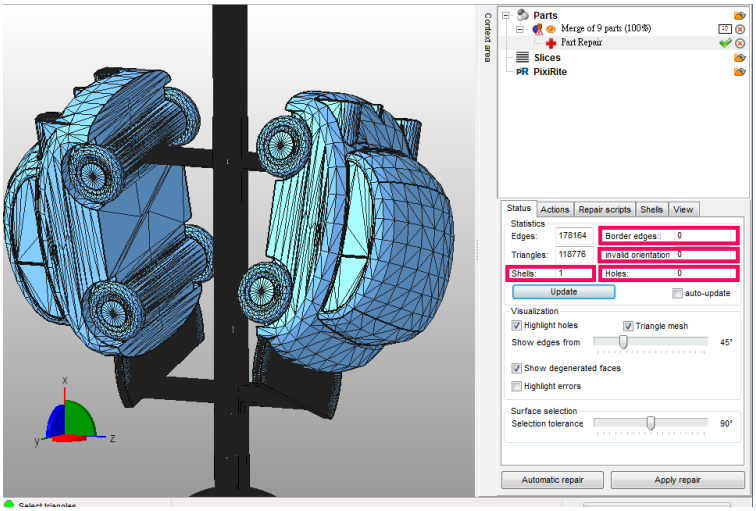
Move the cut off parts a little apart from each other to get a better view. You'll see that the legs of the chair are separated into 4 shells, but they are still saved as one part. To separate them from each other, open the **context menu** with a right-click on a leg > **Extended** > **Shells to parts**. All shells are now independent parts, can be moved independently and can be saved separately. After this, you can save or discard the result, as we'll continue with another part.

14. Checking the repair parts

Content

For having good printing quality and checking the correctness of the part, PixiRite offer a simple way to inspect and confirm your part.

Check the Mesh and shell



Please choose the part that you want to confirm and click the **Repair** (in the toolbar) into the Repair mode. Please click the **auto-update** in the **Statistics** first, then you will find that the red numbers on statistics would change and turn into black.

As the picture on the left, the correct single part will show the numbers of **Border edges, Invalid orientation** and **Holes** to be 0, and the number of **Shells** would close to 1. If

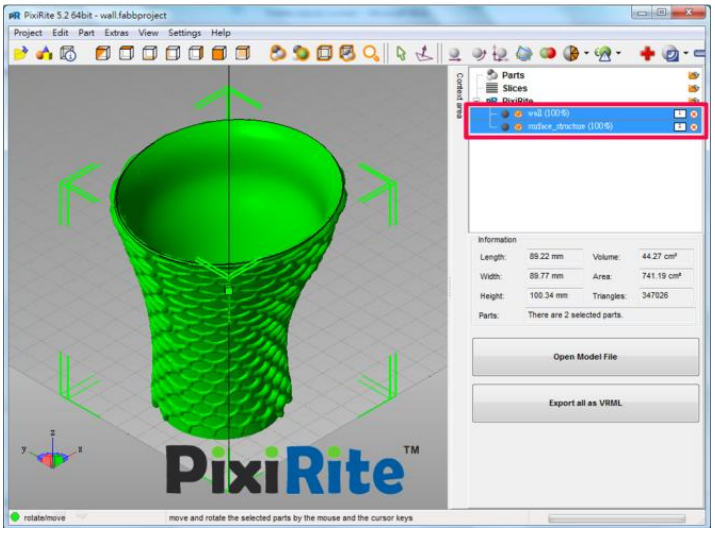
there is some cases not correspond to above circumstance, please press **Automatic repair** and click **Extended Repair** to repair.

15. Slicing

Content

For many machines, the 3D models have to be converted to slices. Learn in this tutorial, how to give the model a filling and how to define contours and hatches.

Open parts

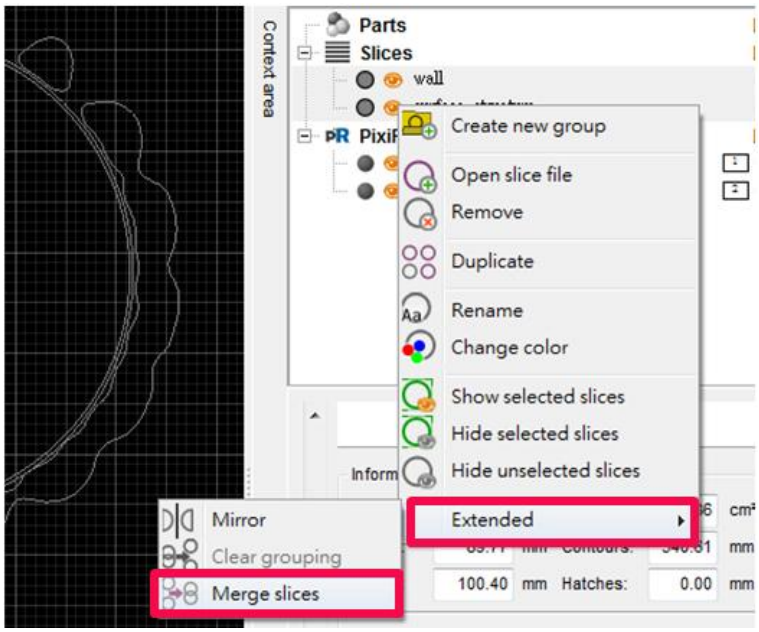


Open the parts *wall.stl* and *surface_structure.stl*. Select them both in the project tree and pull them up to the **Slices section**.

Slicing parameters

A dialog will pop up. Enter a **Layer size** of *0.08* mm.

Merge slices



Now we need to merge the outer shell and the filling. We won't use the Boolean Operation, because this will make them inseparable later. We only want to combine them and save them as one file, but keep the possibility to edit each one independently, **Select** the outer shell and the filling > **context menu** > **Extended** > **Merge slices**.

You have now completed PixiRite's advanced tutorials. You have reached knowledge in all modules of PixiRite - but there are still many features to experience. Take a look at netfabb professional user manual for more details: http://wiki.netfabb.com

If you want to buy PixiRite, please visit our website: http://www.cometrue3d.com/