CSS 1.2

AM with the Caligma 200

using Hot Lithography

"CSS 1.2 - AM with the Caligma 200".

July 2020.

Copyright

Contact Address

CUBICURE GMBH

Gutheil-Schoder-Gasse 17

1230 Vienna

Austria

Table of Contents

Preamble

The ‘CSS 1.2 - AM with the Caligma 200’ document gives an introduction on how to prepare a printable data from a given CAD-Model.

Therefore, 3D objects are prepared for 3D-printing and finally converted into 2D slices in a given layer thickness.

First the installation procedure is covered, afterwards the CSS Interface will be explained in detail as well which commands are crucial to get the data preparation working.

The aim of this manual is to guide the user through all important steps to obtain a working file for your printer setup.

1 Installation

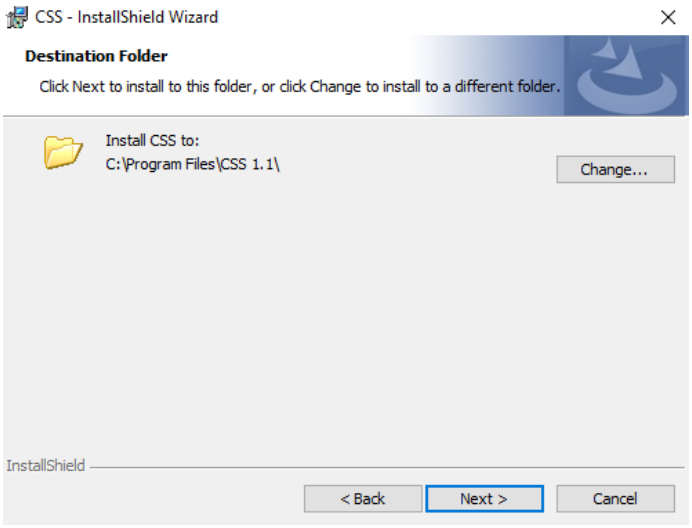
Welcome to CSS 1.2 Manual. First and foremost, we will guide you through the Installation Wizard and how to request a license for your package.

1. Double click the installation file:

2. When asked if you want to run the .exe, click ‘OK’.
3. Next a new window opens:

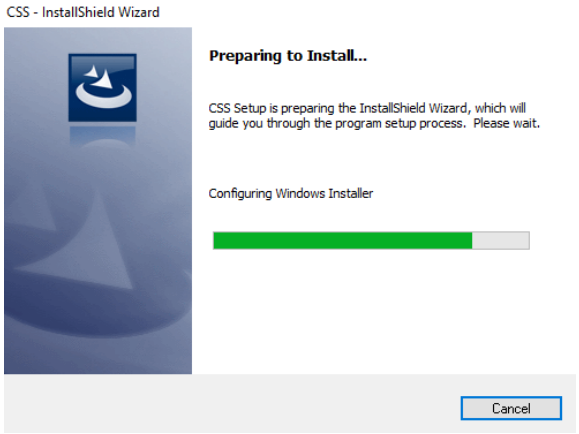
Once you carefully read the license agreements, select the checkbox ‘I agree’ and go to the next step.

4. Select the destination folder for the installation with the browse function.



When done so please click ‘Next’ to start the installation.

5. Now the installation should be running:



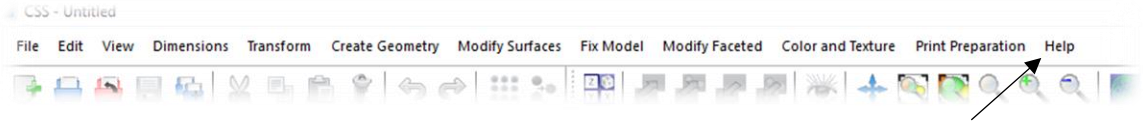
After the installation is finished, complete the process by clicking ‘Finish’.

2 Licensing

After you successfully installed the CSS software, a license will be needed in order to use the software to its full potential.

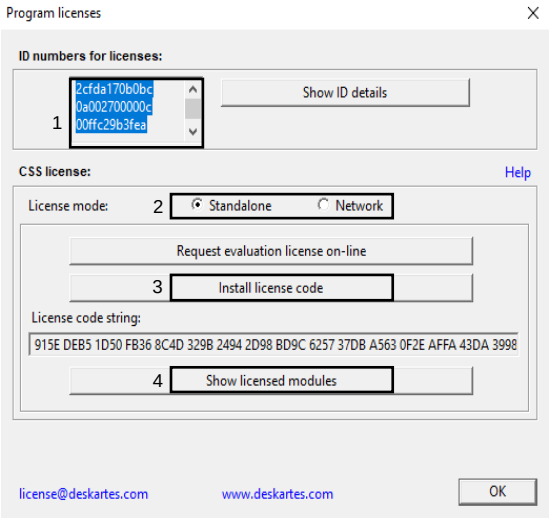
Therefore, you start CSS via the desktop symbol

and select the tab ‘Help’ as shown here:



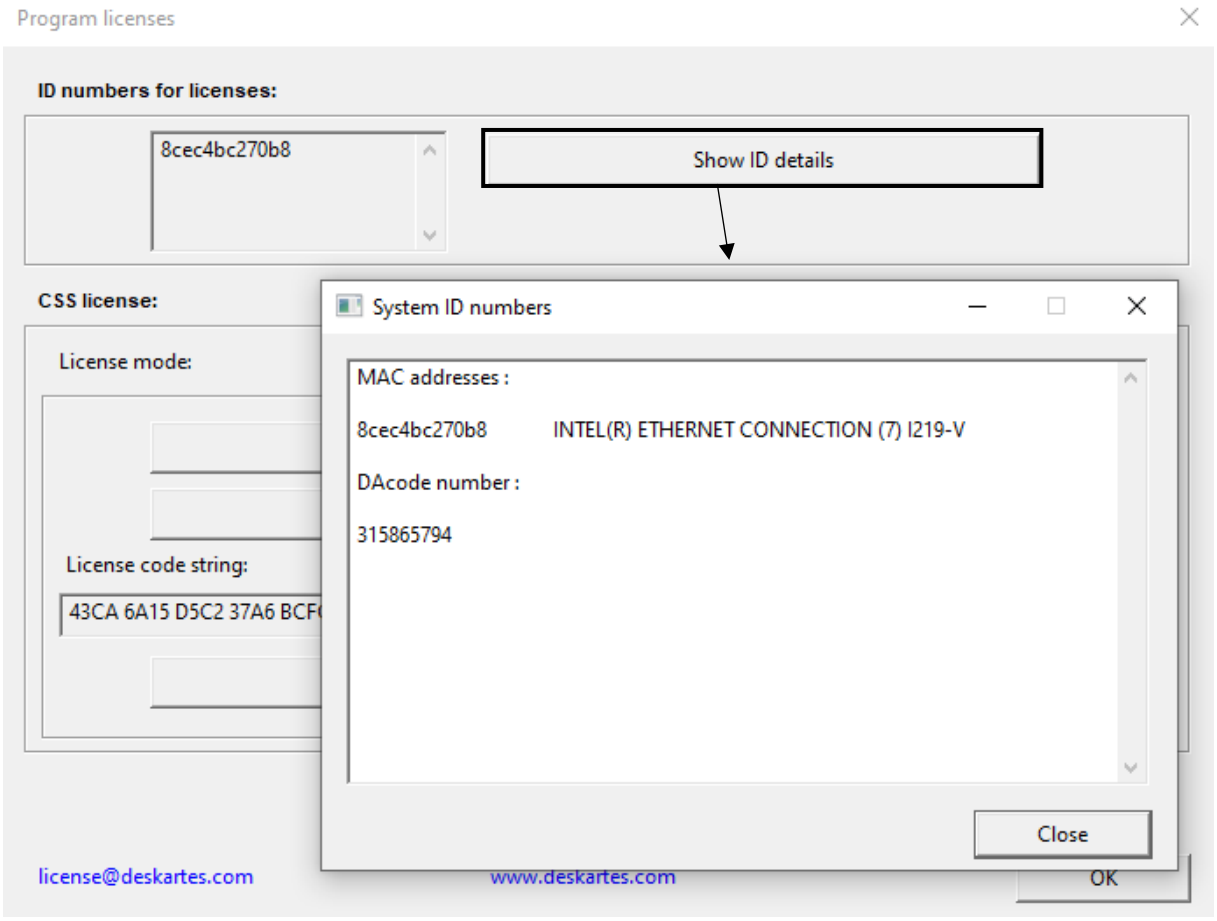
and click on ‘Licenses…’

A window opens showing all CSS license information:



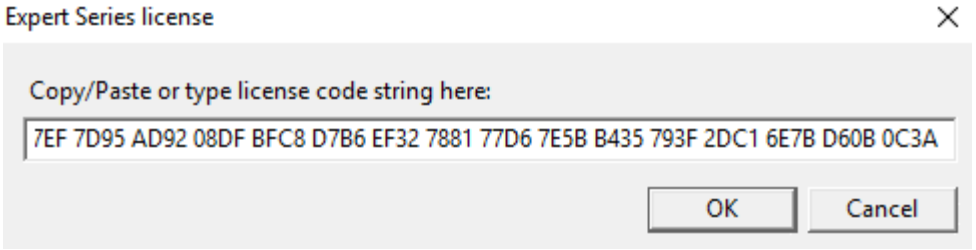
1) System ID (License is tied to the ID)
2) Standalone or Floating License
3) The received license code string is implemented through ‘Install license code’
4) All included functionalities, features as well as additional data inputs formats can be displayed here

To obtain either a testing license or an official license for extended use of all functions and no time limitation contact the Cubicure support via info@cubicure.com and provide the 'System ID numbers'.



Afterwards, you will receive a 15-day evaluation license or official license usually within 24 hours.

When the license arrives, press the button 'Install license code' in the 'Program licenses' dialog and Copy/Paste *the license string* into the input field that appears:



Then press 'OK' as requested to accept the license. Finally, restart the software. You are ready to start!

3 Quick Access

3.1 Create New Project


Create a new job project

3.2 Open Project


Open a job project

3.3 Close Project


Close job project

3.4 Save Project


Save a job project

3.5 Insert Object


Insert an object

3.6 Import of 3D-Data

Various file types can be loaded into CSS.

Import options in CSS base package:

STL, VRML, ZPR, OBJ, DXF, PLY, 3DS, Collada, FBX and 3DE Files

Import options through additional import translater expansion packages:

IGES, STEP, Catia 4, Catia 5, Catia 6, Catia CGR, SolidWorks, SolidEdge, Unigraphics, Parasolid, Creo/ProE, Inventor, JT

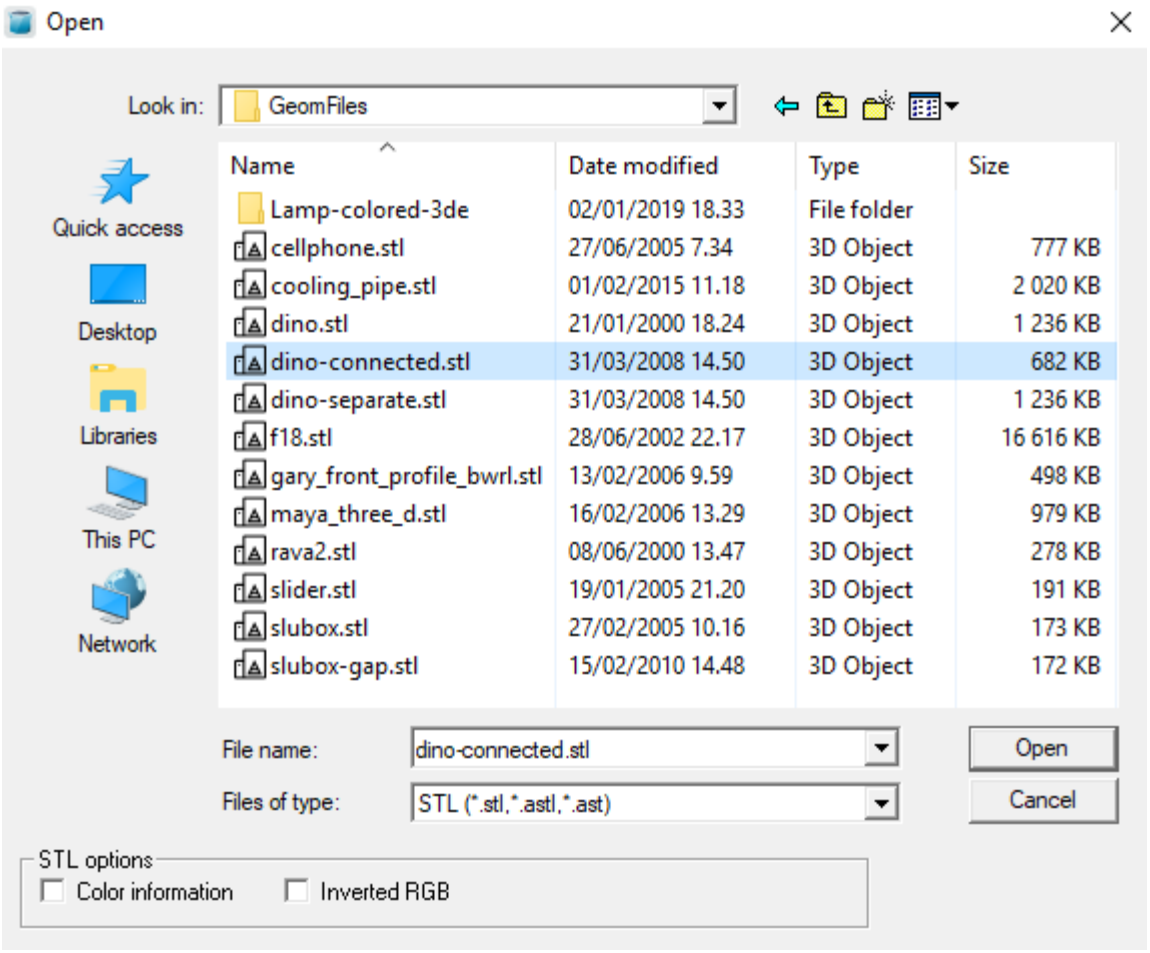
These optional input formats can be implemented on demand. For further information, please contact info@cubicure.com

4 Auto Repair process for STL files

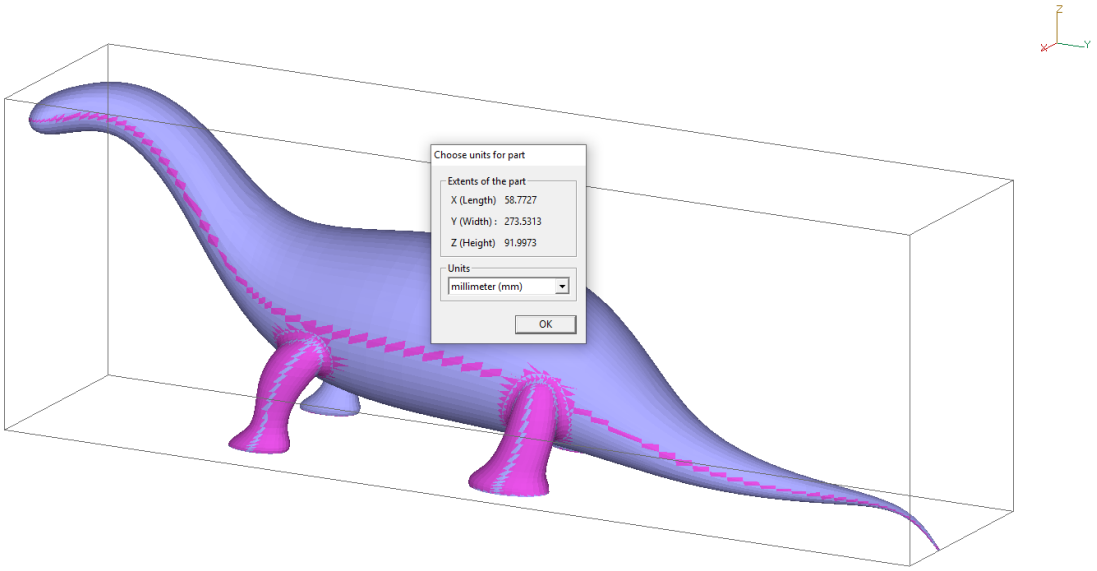
When inputting a triangulated model into CSS software, an automatic analysis and repair process starts. For the everyday use you can verify and correct most of the models using the Auto Repair process.

In the CSS manual the single steps are demonstrated with the STL-file ´dino-connected.stl´ as an example. For your own process please use your required STL-files.

First open the *dino-connected.stl* file in the *Tutorials/GeomFiles* directory with the *File > Open* command. Make sure the type of files is *STL*:



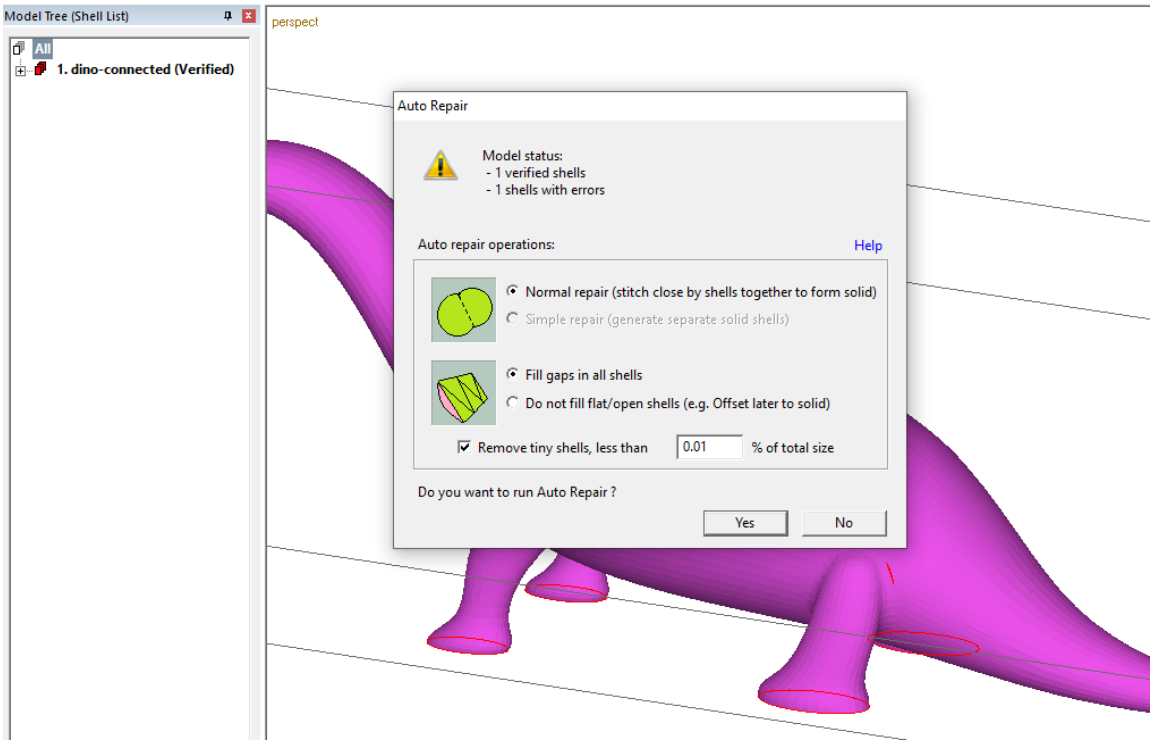
Press *'Open'* to bring the model into the software. You will be prompted for the units, accept the proposed units 'millimeter (mm)':



Press 'OK' to continue and you will be prompted for the Operation Mode. CSS has four main operation modes: *Fix Model*, *Paint & Texture Model, Print Preparation*, and *View Model* mode. The different modes are used for specific tasks:

- *'Fix Model'* takes you to automatic model verification and repair, and contains shortcuts to the main tools used for repair work
- *'Paint & Texture Model'* displays the tools for painting and texturing. These tools check the model for errors and set it to "what you see is what you get" display color mode
- *'Print Preparation'* shows tools for model manipulation, such as hollowing and splitting. These tools also check the model for errors
- *'View Mode'* minimizes the Operation Mode window and allows you to view the model and then return to Operation Mode selection when clicking the minimized window again.

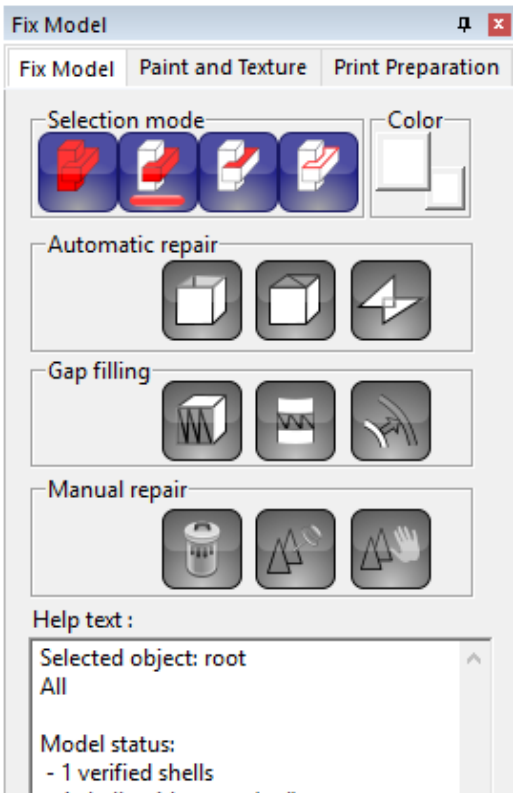
The normal way to begin repairing a model is to start with the 'Fix Model' mode. To start fixing **press the 'Fix Model' button** in the Operation Mode dialog**.** CSS will start the automatic model verification (*Fix Model > Verify Shells* command) and a progress bar appears. When the analysis is ready, the 'Auto Repair' dialog with 'Model status' message is displayed:



The model is verified for errors and divided into separate shells during verification. In this case we only get one shell (*1. dino-connected (Verified)*), which also contains errors. The display shows red gap curves through the surfaces. You can use different viewing commands when 'Auto Repair' dialog is visible (see page 12). Gaps are an error type you must fix during the repair process of *dino-connected.stl*.

The shell has errors (i.e. gaps), which is indicated by the red icon in the Model Tree (correct shells would have green icons).

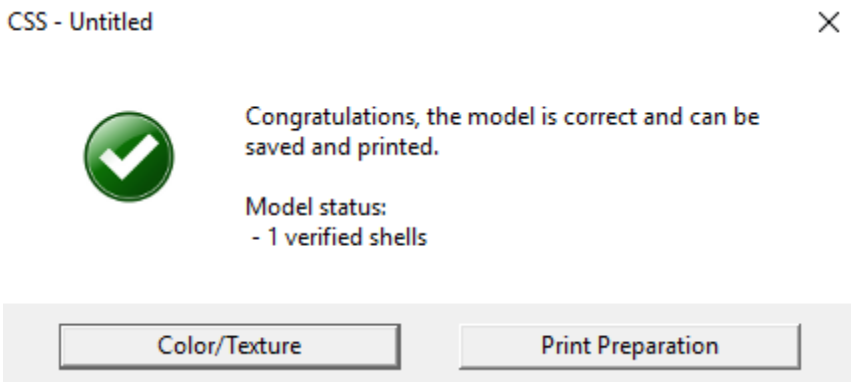
On the right side of the graphics area you will see the *Tools Window*. *Fix Model* tab on the *Tools Window* provides shortcuts to the commands available for repair, like Repair Shells, Offset, and Fill Gaps commands. The *Help text* window at the lower part of the *Tools Window* also contains information for required repair actions. Command icons are greyed out while in *Auto Repair* process. We will learn more about these commands in the Tutorials, now we will follow the *Auto Repair* process.



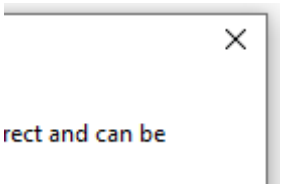
Auto Repair is the best path to follow when repairing everyday models. It will attempt to fix all shells with errors in the Model Tree. Especially when you only have one shell, like now, you should always try *Auto Repair* first. The parameters are automatically set based on the model properties. *Normal Repair* is set because there is only one shell to repair and *Fill gaps in all shells* is set because there are no open/flat shells in the result (which may require offset or some other fixing methods available in Expert Series).

Press the 'Yes' button to initiate the automatic repair process.

When *Auto Repair* is started each shell is run through the automatic repair command (*Fix Model > Repair Shells* command) using automatically calculated parameters. For each shell, a progress bar shows the progress of the repair. After finishing the process the following model status message is displayed:

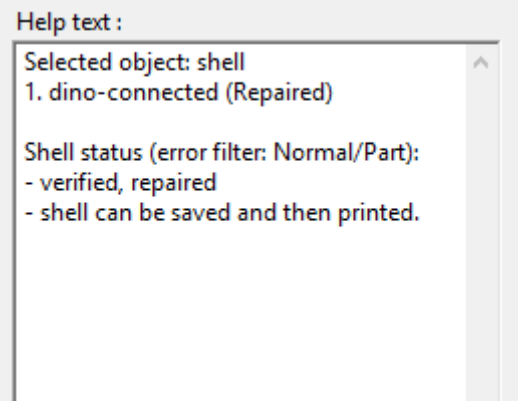


This time we want to learn more about the viewing commands, not about changing model coloring or print preparation, so you can close the status message box by **closing the dialog** on the upper right corner.



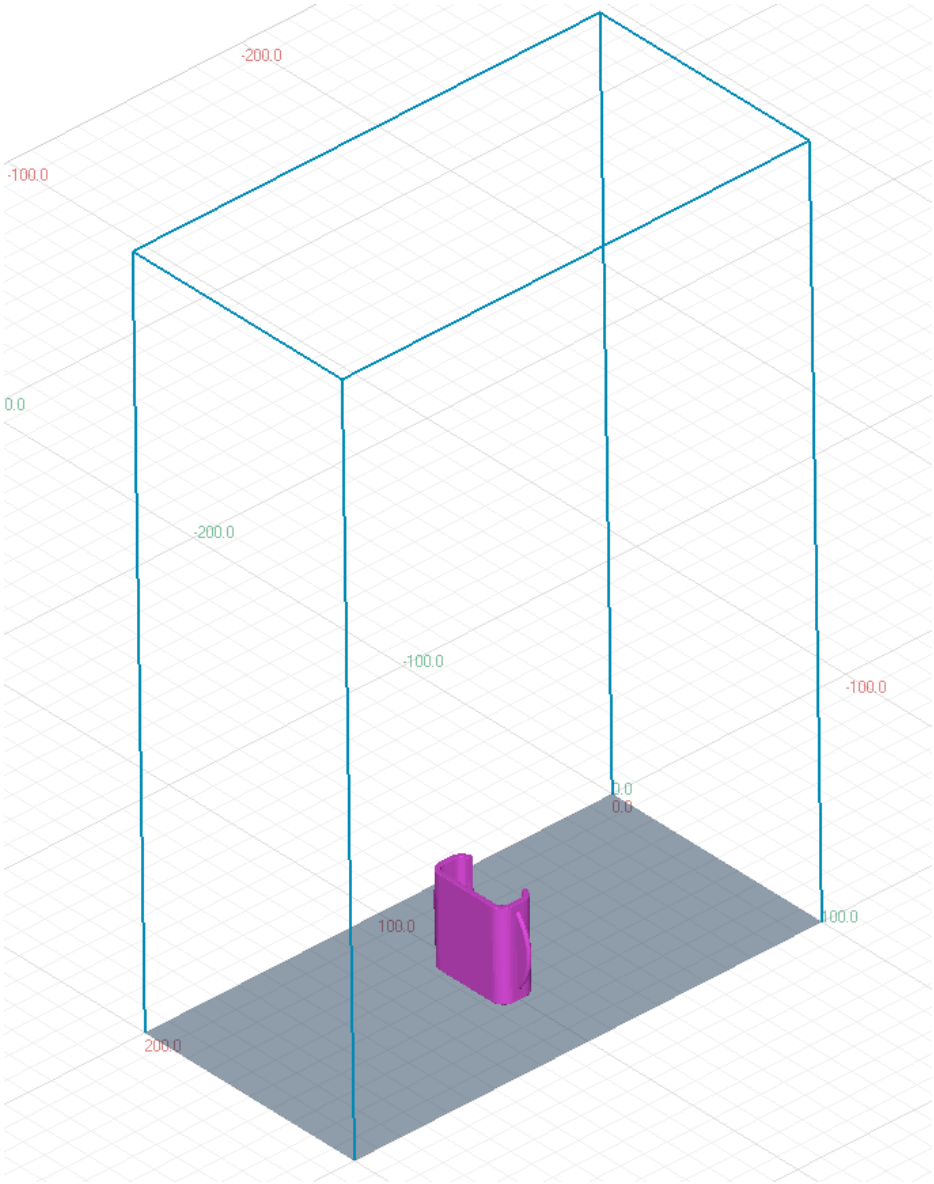
During *Auto Repair*, CSS fills all gaps and removes non-manifold triangles (overlapping triangles, multi-edge triangles, duplicate triangles, triangles having less than three neighbors, etc.) from the model. Overlapping and duplicate surfaces are analyzed and connected in the optimal way. When you want to learn more about triangle errors, please see the*Triangle errors* page in the Online Help chapter *Handling Faceted Files*.

The *dino-connected.stl* model only has simple errors, like inverted triangles and gaps, which are all fully repaired during the automatic repair process. The model is now free of defects and can be 3D printed (or taken into simulation software), as is indicated by the status information in the *Help text* window below the '*Fix Model*' tab on the right side of the screen.



4.1 **Workspace Caligma_200**

When you open a file in CSS, it will be placed automatically into the workspace Caligma 200 printer. The predefined workspace for CSS is the Caligma 200 and can be shown or hidden manually by clicking  .



If you want to use a different workspace, you can define and select other ones under the Tab *Print Preparation > Platforms*. The predefined and selected workspace, when starting CSS, is the Caligma 200 platform.

4.2 Basic viewing commands

You have just fixed your first model with CSS software. Now try the following commands to become familiar with viewing operations.

Note that the commands below do not alter the model's true 3D coordinates. They only alter the eye-point location:

- ***Rotate*** the model on the screen by pressing the *middle mouse* (*MMB*) and moving the cursor. MMB rotation is always available. *Left mouse button* (*LMB*) rotation is available most of the time except when working with painting, texturing, curve editing, and transformations (LBM is mostly used for selecting objects as shown later).
- ***Rotate*** the model around a selected surface point by pressing the *right mouse button* (*RMB*) and moving the cursor. A blue cross indicating the rotation middle point appears on the part surface if the cursor is on top of the model when the *RMB* is pressed.
- ***Pan*** the model by dragging with the *Shift* + *MMB* (or *LMB* most of the time) (*Shift* + *MMB* means pressing the Shift key and dragging with the mouse).
- ***Pan*** *with rubberband* by dragging with *Shift* + *RMB*.
- ***Zoom in/out*** by rotating the *mouse wheel.*
- ***Zoom in/out with*** *Ctrl + MMB (or LMB).*
- ***Zoom to box*** *with Ctrl + RMB.*
- ***Hide all*** by pressing the  icon with LMB *(or F5)*
- ***Show all*** by pressing the  icon with LMB *(or F6)*
- ***Fit the model*** by pressing the  icon with LMB *(or F7).*

The main viewing command icons reside in the *Viewing Toolbar*:

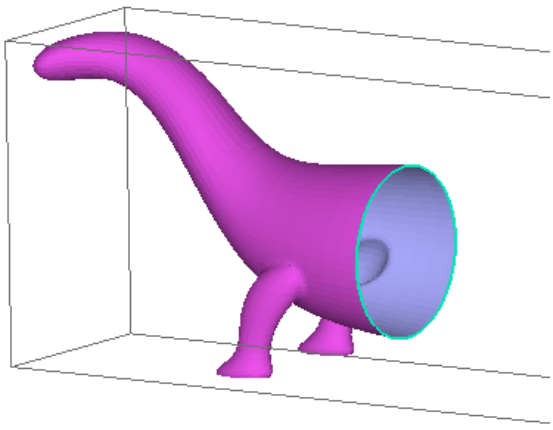
Try them out. The tooltips provide a short description of each icon (let the cursor remain on top of the icon for a couple of seconds to see the tooltip)*:*

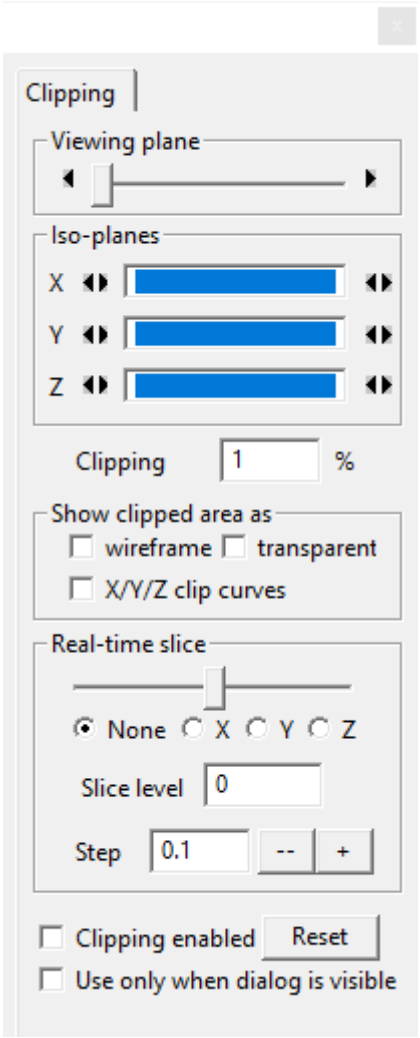
Also, the *Viewing Planes* toolbar allows you to select any projection direction as the viewing direction with *LMB*:

You can show the grey selection box around the selected object by pressing the *Show Bounding Box* icon in the '*Viewing*' toolbar: 

Try also clipping functionality through the *Clipping* tab to look inside the model. The tab is opened with the  icon. The clipped area can be displayed with different ways, like with wireframe or transparent.

Clipping also enables to extract real time slice information on the model. *Real-time slice* can be used to visually check the model quality or be copied to the *Model Tree* for internal model dimensioning with the *Edit > Save Clip curves* command.

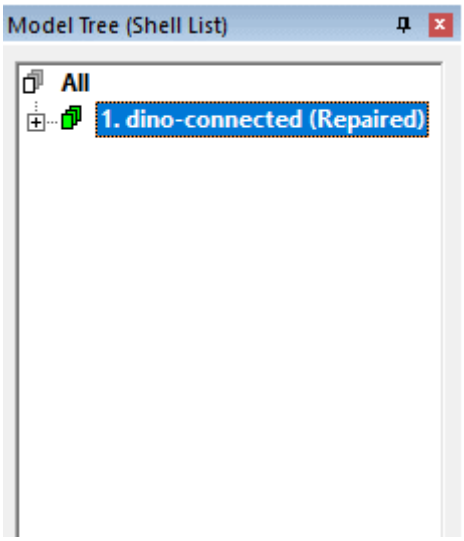




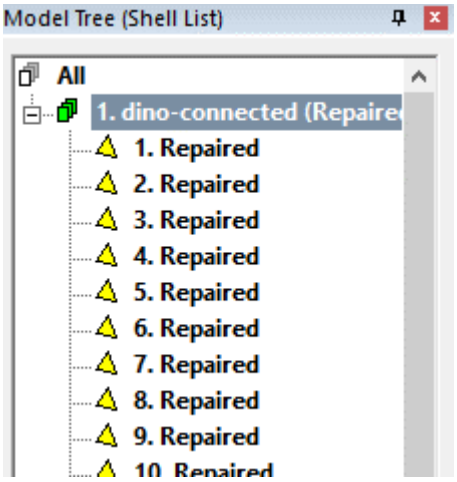
4.3 Selecting objects

On the left side of the CSS software window you can see the *Model Tree* tab.

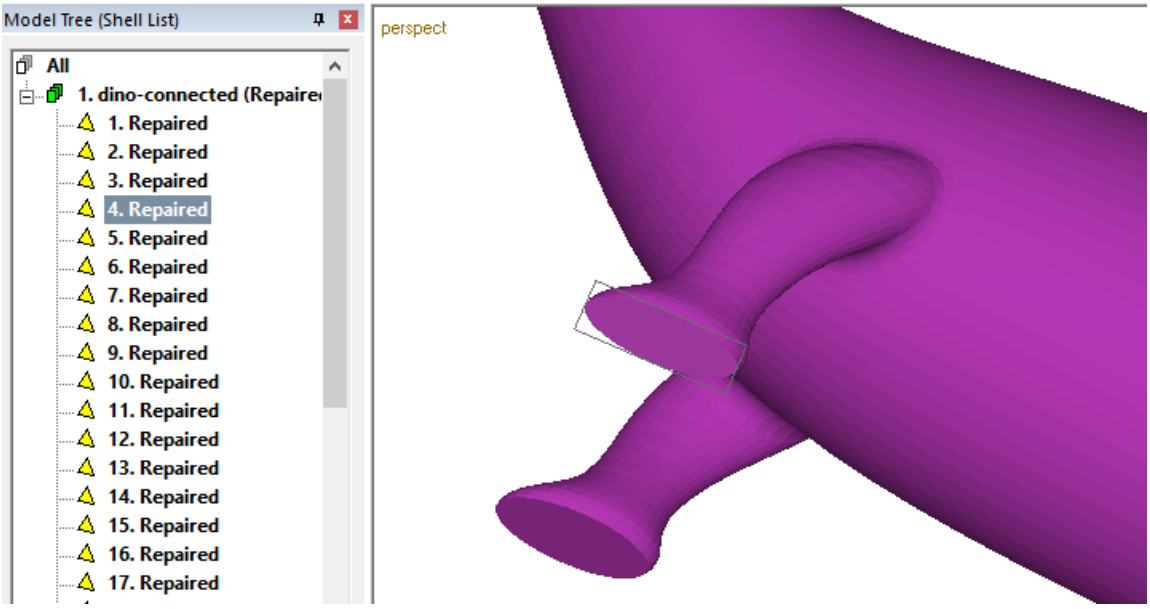
This tab tells you the current contents of the program database. With this example you can see one *shell* (aka node, group) *1. dino-connected (Repaired)*. The node 1. includes the correct dino model, as indicated by the green icon with the shell. The *(Repaired)* text describes the last command run on the surfaces, which in this case is repair.



If you click the *+*-icon at the shell 1. it will open and display the separate triangle surfaces the *dino* model is made of. During the repair process the model is subdivided into separate surfaces along sharp edges. Triangle surfaces are indicated by a yellow triangle icon and the last operation name, i.e. *1. Repaired* for the first and largest area triangle surface in the model and so forth.



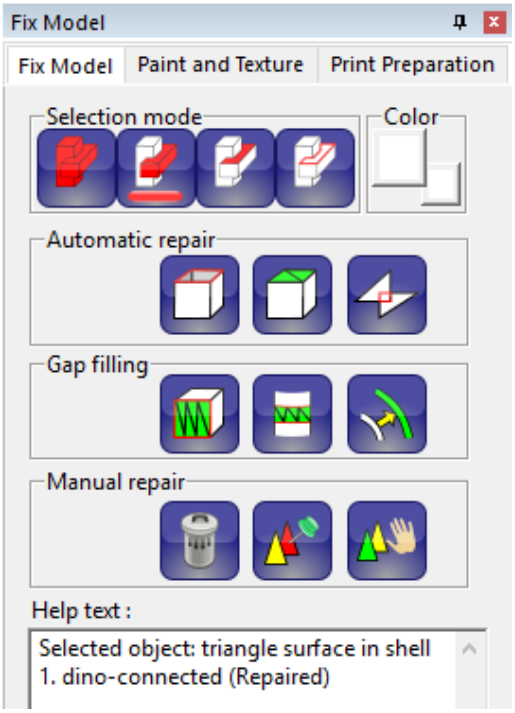
You can select objects at different levels either directly from the *Model Tree* with LBM click or graphically from the display area. For example, the surface 4 selected from the *Model Tree* corresponds to the sole of the front left leg of the *dino* model. This is indicated with highlight and green bounding box around the selected surface in the display area:



Different objects can be selected also graphically from the display area with LMB. On the right side of the screen you can see the Tools Window which includes the *Selection mode* icons.

The *Selection mode* icons allow you to set the graphical selection level to *All, Shell, Surface* or *Gaps*. The default mode is shown in the image right and it is the *Shell* level. If you click on the dino model on the screen with the LMB, the shell *1. dino-connected (Repaired)* will be selected in the *Model Tree*.

Generally, the Tools Window contains handy shortcuts to the available commands in the selected Operation Mode as well as *Help text* window for the selected object or for the command in progress.



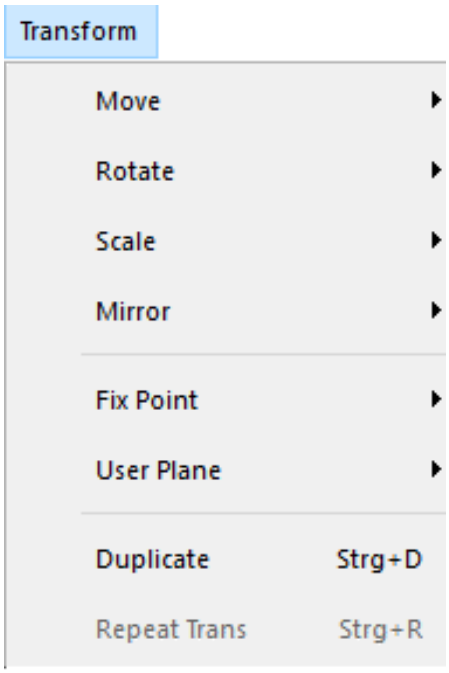
5 General Workflow

After we learned about fixing an STL-model and knowing how to navigate and use the CSS - workspace, we will now go through the general workflow coming from said STL-model and resulting in a ready-to-print CLI-file.

5.1 Basic Transformation

After the completion of the repair and verification cycle, the next thing we want to focus on are basic transformation commands and how to use them.

A quick overview is given below:



Move an object (possible in direction of X, Y and Z axis)

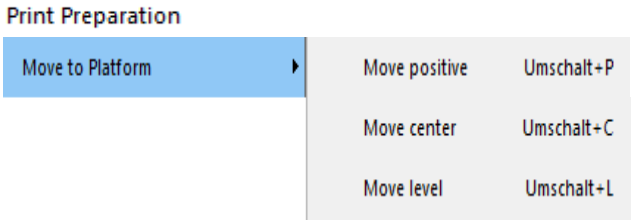
Rotate an object

Change the dimensions of the part (see Chapter 5.2.1)

Mirror an object over a selected axis

In order to make transformations when already preparing a print job for your machine, there are specific transformations possible, which are directly affecting the position depending on your selected building platform.

These are found in *Print Preparation* and shown below:



Move part into positive coordinate system

Move part into centre of building platform

Move part in Z-axis

5.2 Pre-print Preparation

After your part is positioned and has the correct alignment, it is necessary to make a few additional adjustments to ensure that the part is printed correctly.

5.2.1 Scaling

Because of specific influences (over polymerization, post-curing shrinkage), which are connected to printing photopolymers, these deviations must be compensated by scaling the digital part beforehand.

Meaning, due to the effects mentioned above the part you want to print, will not be printed in the exact measurements given in the digital version.

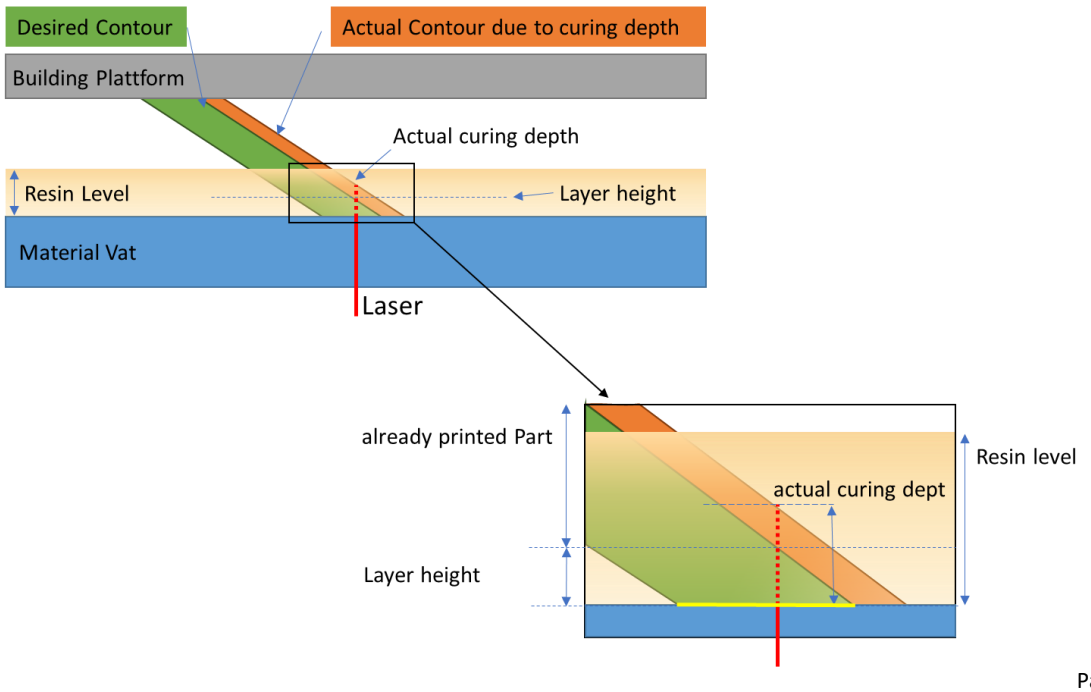
For a recommendation of appropriate scaling factors or values for later described *Tool* as well as *Z-Compensation* for the processing of Cubicure resins please get in contact with Cubicure staff.

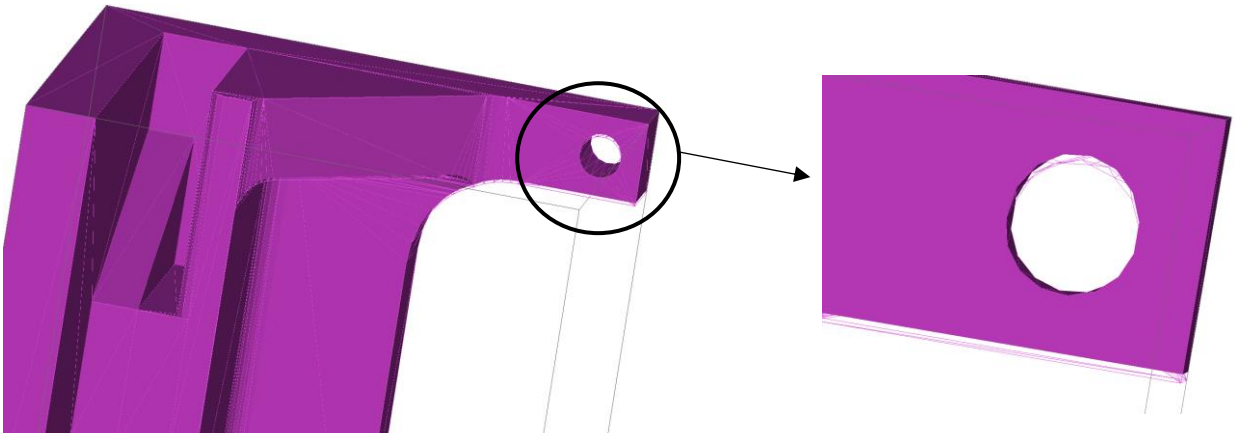
> Note that calculated compensations (especially scaling) will not always result in the desired measurements. Depending on the size of an object the necessary adjustments may differ. For specific cases, an iteration may be necessary.

5.2.2 Z-Compensation

Every polymer has a specific curing depth which is usually higher than its adjusted layer thickness during printing. Thus, when printing a part with down-facing surfaces, these surfaces must get compensated, meaning being moved upwards.

This is explained figurately in the *picture below.*





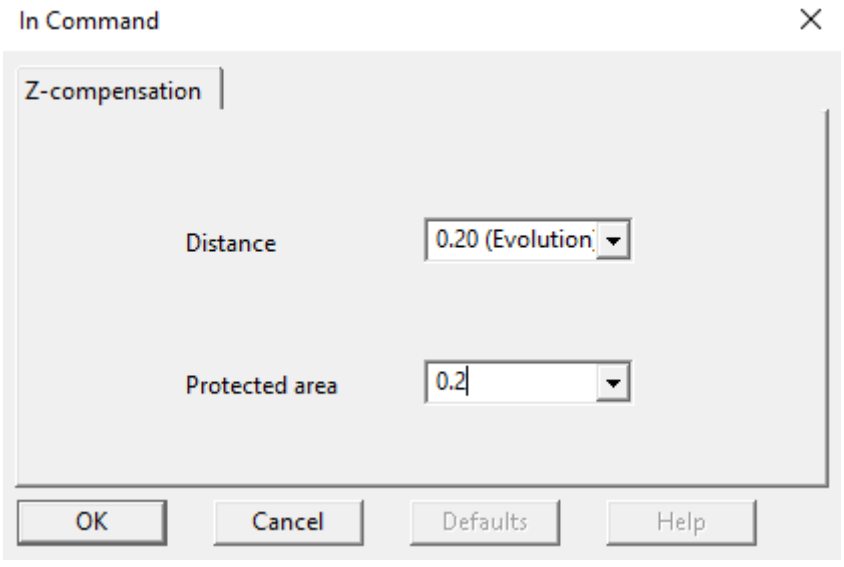
In the *pictures above* the applied Z-compensation is shown through a 'wireframe' part, which has all downfacing surfaces compensated.

Every polymer provided by Cubicure has a specific Z-compensation value, which is specified individually.

The protected area value should be the same as the compensation value itself, to guarantee that the base of the part is not compensated but has still contact to the building platform.

Select the part and use the Z-compensation command which can be found in the tab 'Print preparation'.

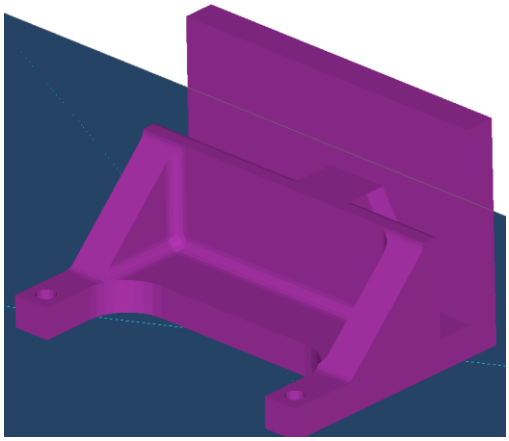
The following window will appear with preselected parameters depending on the material.



Note that the Z-compensation needs to be applied after the correct alignment and before "generating supports".

5.2.3 Generating Support Structures

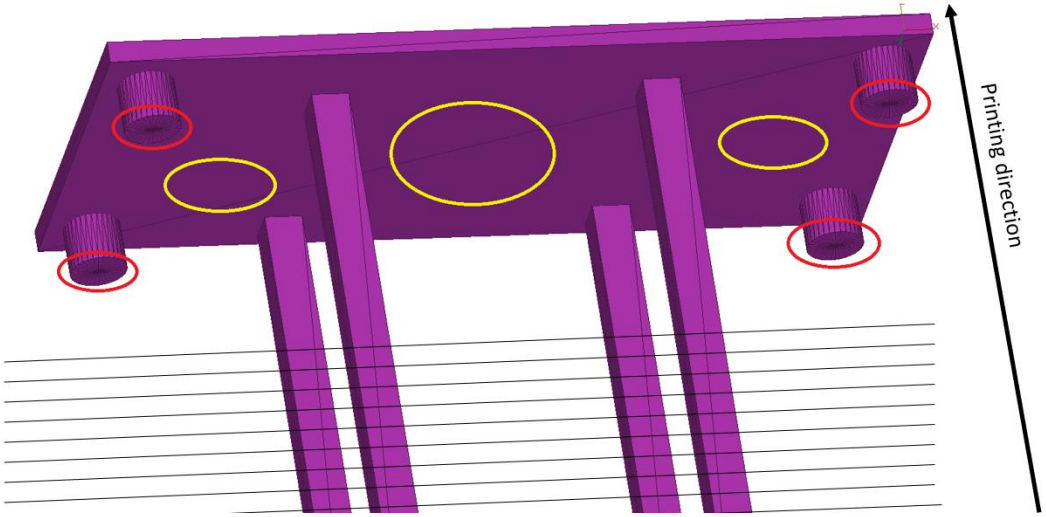
Now your part is correctly compensated, scaled and aligned in the building area like shown below:



Support generation is not always needed and therefore a few guidelines are helpful to determine if this is the case for your part.

For example, the part depicted on the right can be printed completely without support structures if aligned correctly.

Because of the principle of layer-printing, one layer always needs a solid connection to the previous layer.



If you want to print an object like shown above, there are a few marked areas in red which would lead to problems because of a non-existent connection to its previous layer. Therefore, these areas must be supported beforehand to guarantee their connection to the part.

Few areas are also marked in yellow which depicts that these are areas have indeed a connection to the part in that specific layer, but this connection is not enough to guarantee that the geometry is built correctly.

Therefore, an evaluation beforehand is necessary to generate the part´s geometry precisely.

5.2.3.1 *Different types of support geometry – automatic support generation*

To generate the support structures, we select the part and hit the command

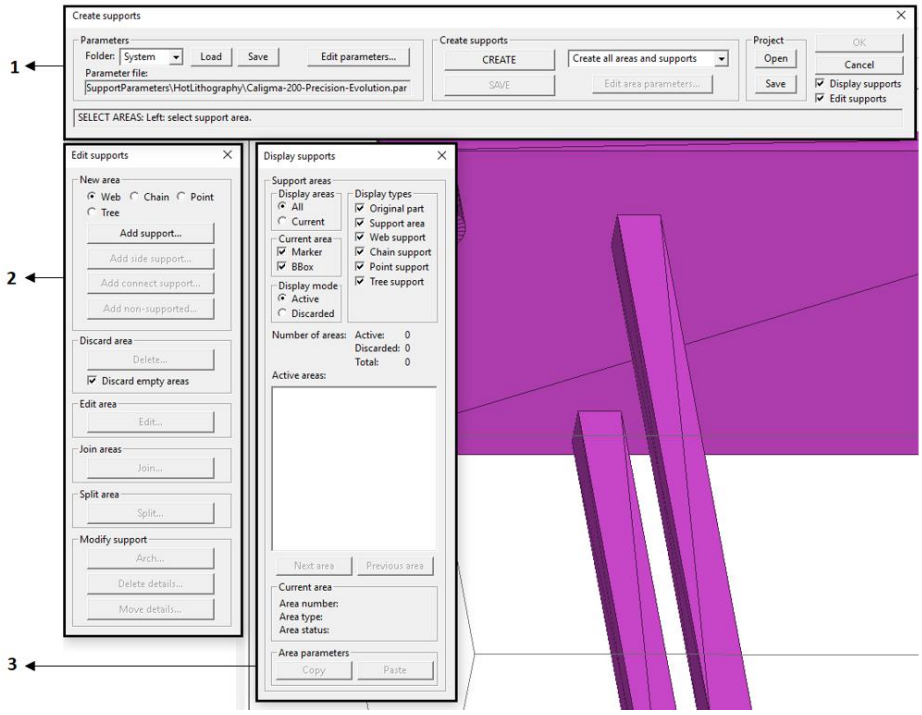
´create generate support' 

The command is also available under the tab '*Print preparation*'.

Next the parameter-set for the support generation can be selected.

When using a specific material, the corresponding parameter set should be selected if available. This guarantees i.e. that the connection between support and part itself will be strong enough while at the same time as tiny as possible to achieve best possible surface quality.

After selecting the parameter set, the *support generate* interface will show up:



1. *Create support window*
   Here you can load/safe different parameter sets and change specific parameters or the set as a whole
2. *Edit support window*
   Tool for area creation/editing/deleting which are then used for support generation; modifications of the support can be done
3. *Display support window*
   This window is used for displaying which areas have been created and where to find them; differencing through support types is possible here

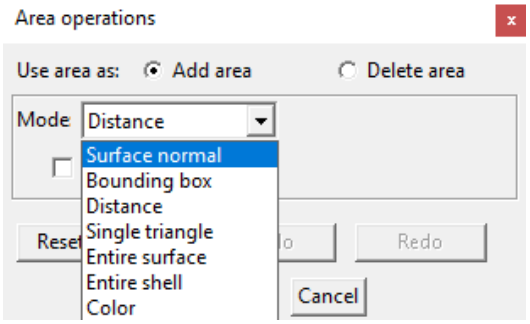
Before generating actual support structures, an area must be created.

Four main types of area can be created through the command *Add support…*, which is in the *Edit support* window.

Web - Grid structure; generally good usability for all types of surfaces

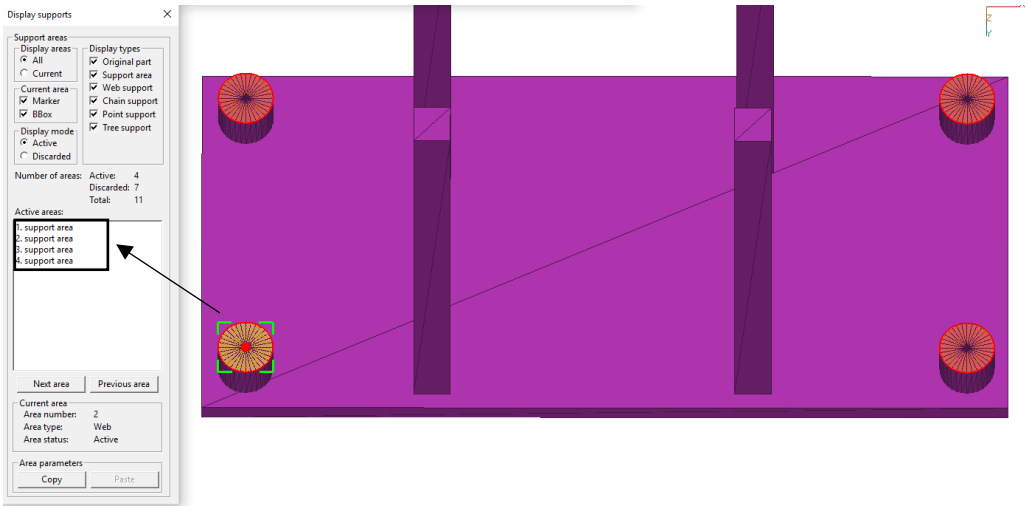
Chain - Chain structure; points must be selected manually

Point - Single Points; points must be selected manually



When trying to create a support area, a few different possibilities are selectable:

Select *Surface normal*, click on the surface which needs support and accept it with 'OK'.



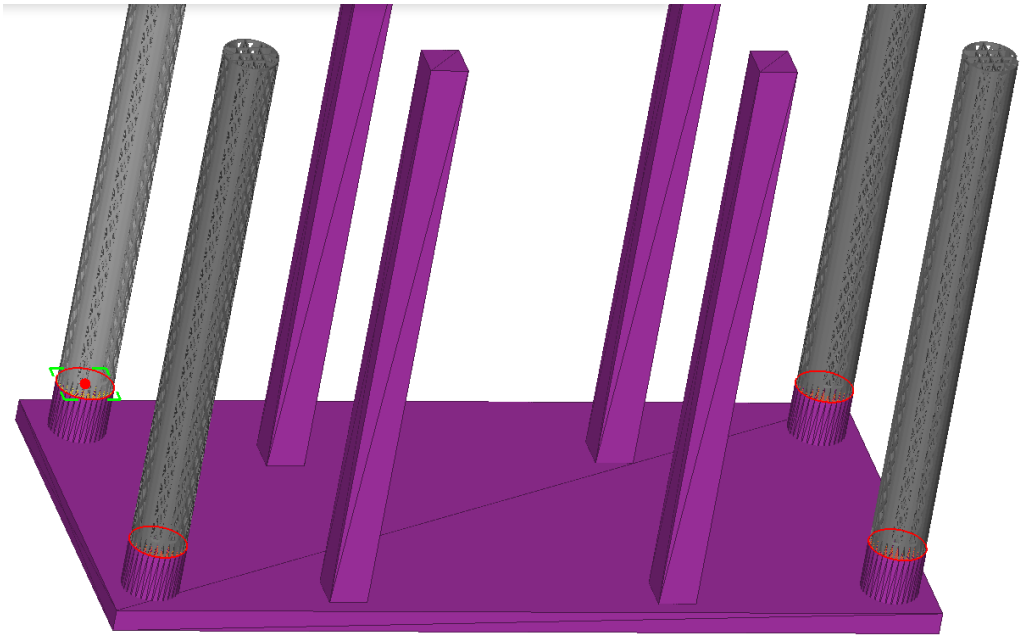
As displayed above, the created and now active support area is highlighted with a green bounding box and a center point.

Beside *Surface normal* there are a few other modes for selecting support area which can come in handy for specific applications.

For automatic procedure four possibility are given:

- **Create all areas and supports**
  - *with the selected parameters, areas are automatically determined and supported*
- ***Create all areas***
  - *with the selected parameters, areas are automatically determined and shown*
- ***Create supports for all areas***
  - *create supports automatically for all areas either automatically or manually created*
- ***Create supports for current area***
  - *create supports for the chosen area*

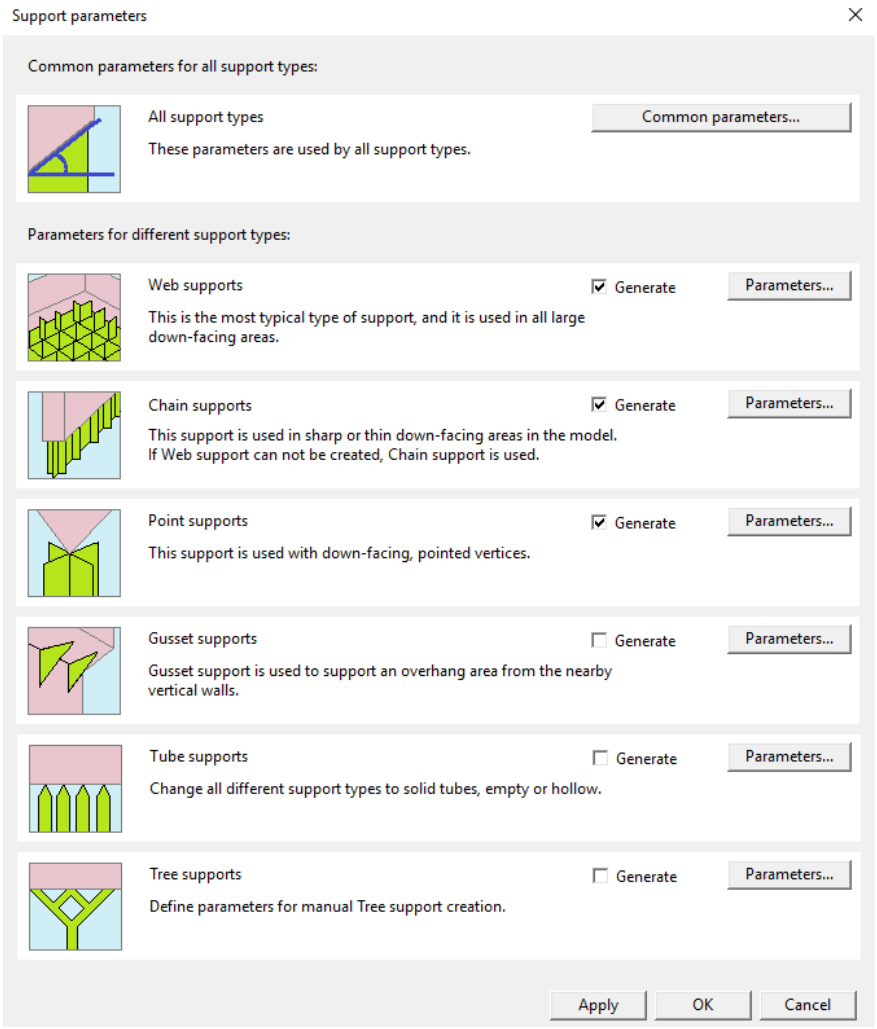
All 4 shown areas are supported by use of ‘create supports for all areas`.



5.2.3.2 ***Support parameters – manual support adaptations***

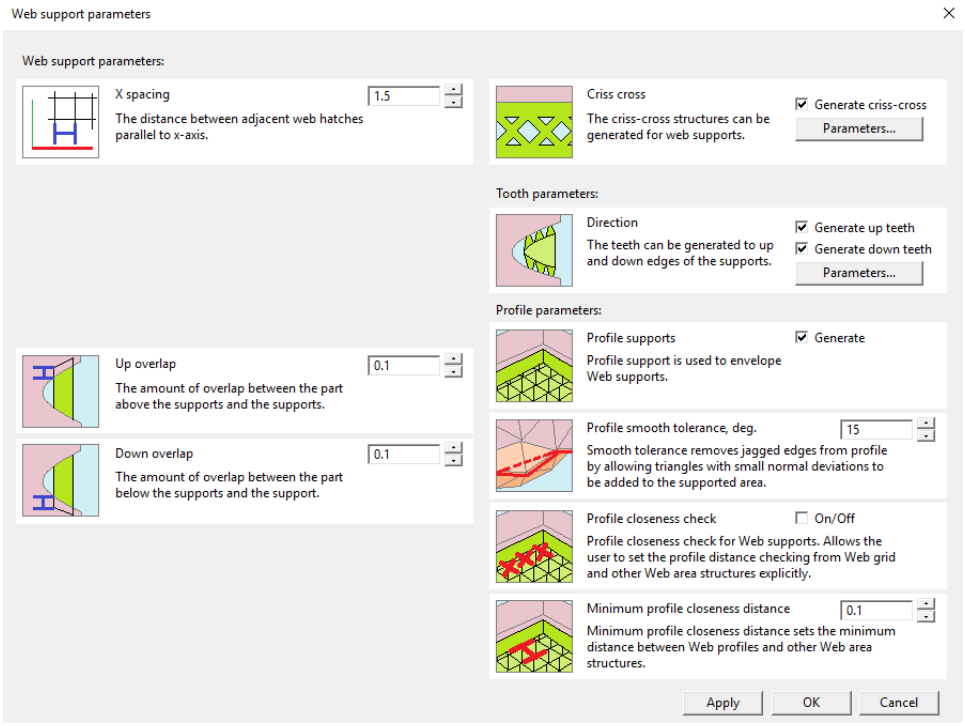
If the situation arises that the used parameter set does not produce the desired result, the possibility to change specific single settings might be necessary.

To do so, click on *Edit parameters* or *Edit area parameter* to open the configuration window for support generation displayed below.



Here the different support types can be manually altered to gain the desired outcome. For more precise explanations, the web support parameters will be shown in the subsequent window.

When opening the *web support parameters*, the following possibilities are displayed:



Different parameter settings are provided which enables the user to configure the general outcome of the web structure.

For a better understanding of the features an explanation is given below.

By clicking on ´Apply´ the changed settings will overwrite the current ones and recreate all corresponding support structures again.

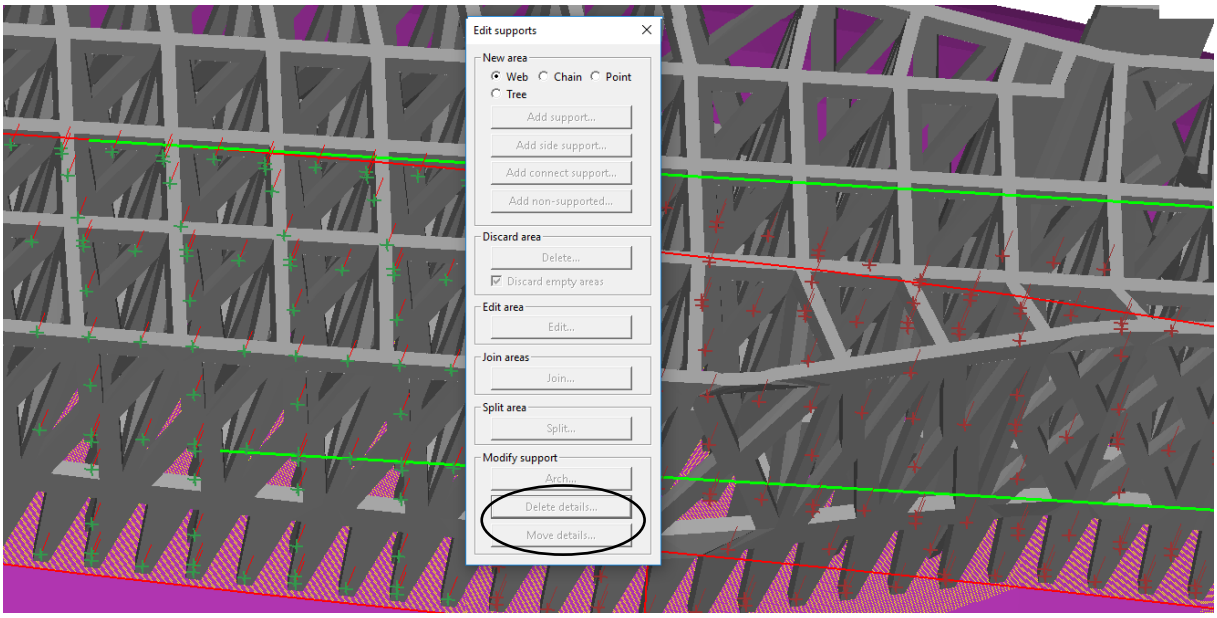
Move/delete support details

For even more configuration possibilities, the commands *Move details* as well as *Delete details* are available in the *Edit supports* window displayed below.

When opening this, little details in green/red will show up.

With a simple “drag & drop”, support details can be either deleted or moved to an exact location.

Finish the support modification by clicking the RMB.



If you want to finish the support generation click ‘OK’.

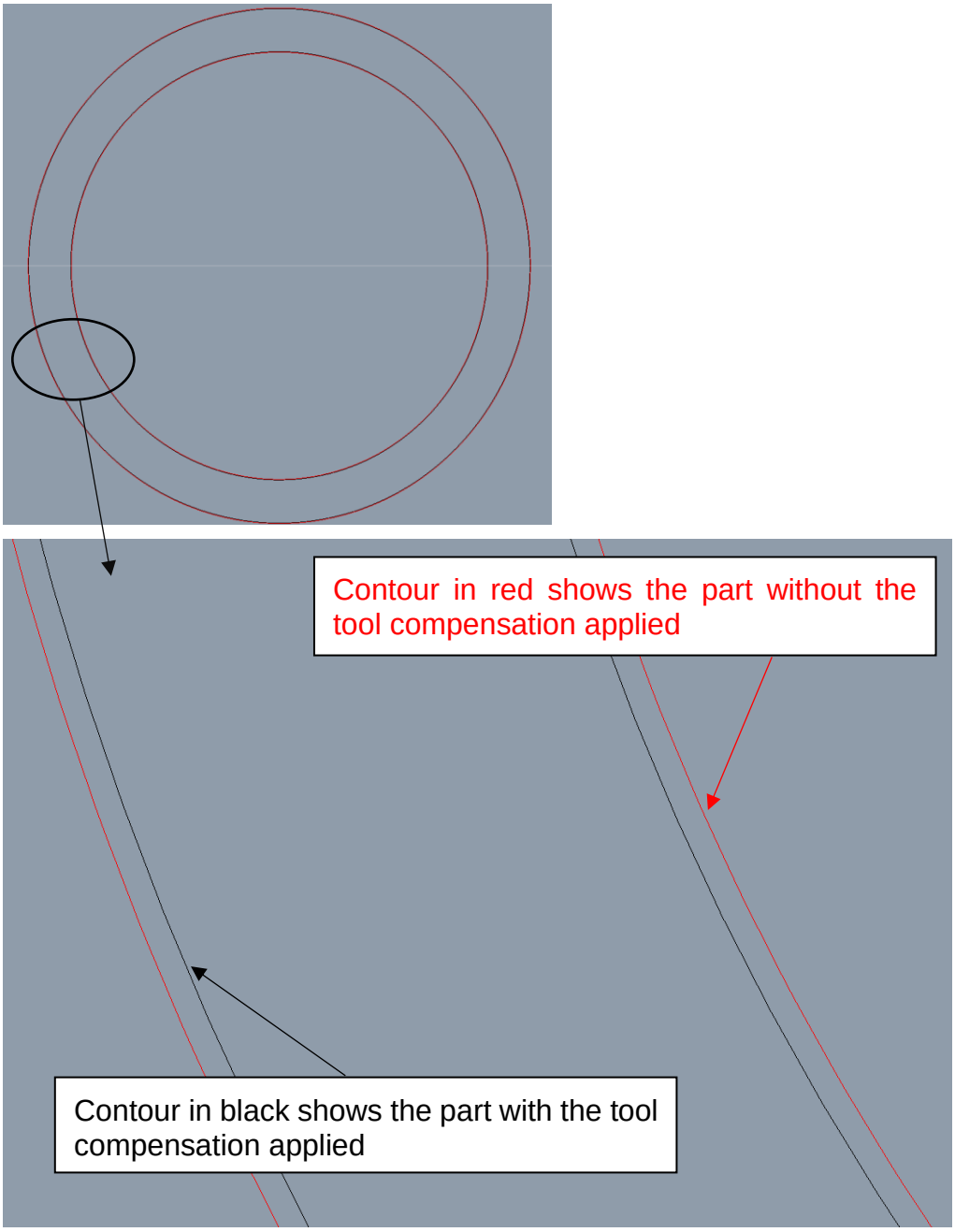
5.2.1 Tool Compensation

This feature will be applied when slicing your ready-to-print part.

The tool compensation is necessary due to over polymerization which occurs while printing. The effect of over polymerization affects both the inside and outside measurements of the part. Therefore, scaling alone will not result in the desired output.

The tool compensation takes any outer contour of the given part and reduces them by an absolute amount which is necessary to get both inner and outer contour measurements correctly.

This is shown by the comparison of two CLI´s, one with applied tool compensation and one without it.



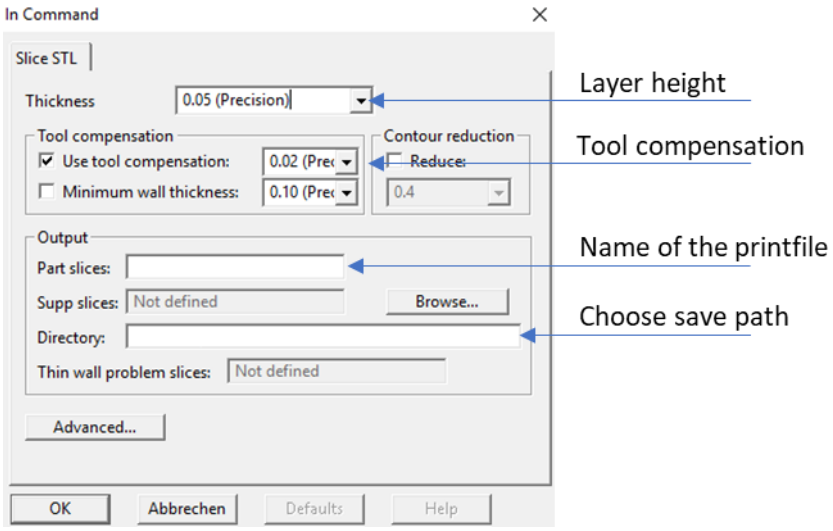
5.2.1.1 ***Slicing***

When the STL model is finished, it is ready to be sliced.

Therefore, the model is selected in the model tree, and you press the slice command which is in the *Print preparation* window as well as in the tab *Print preparation*. 

The following window gives the possibility to select the desired layer thickness, tool compensation & minimum wall thickness and how the output procedure will be performed.

The minimum wall thickness, when turned on, will give a warning if some slices would generate walls which are thinner than the defined ´minimum wall thickness´. If this happens in the slicing, the contour segments are reported in the ´TWP´-File (Thin wall problem slices file).



For Cubicure resins, recommended default values are given which should be used to get the desired results.

With Cubicure resins all corresponding parameters are automatically selected.

The possibility to use different values are given, which can be freely used.

Under *Output* a name can be given to the CLI which then will be exported in the named directory.

Congratulations, you created your first printable CLI!

5.2.2 Additional features and tools

5.2.2.1 Project File

The project file (.3de_project) concept is implemented in CSS. The first use of the project file is to store intermediate stages of the *Generic Support* command. This allows the user to store and go back to the previous support generation process with a later software session.

To save a project file, the 'save' as well as an 'open' button is in the *Create support* window in the support interface.

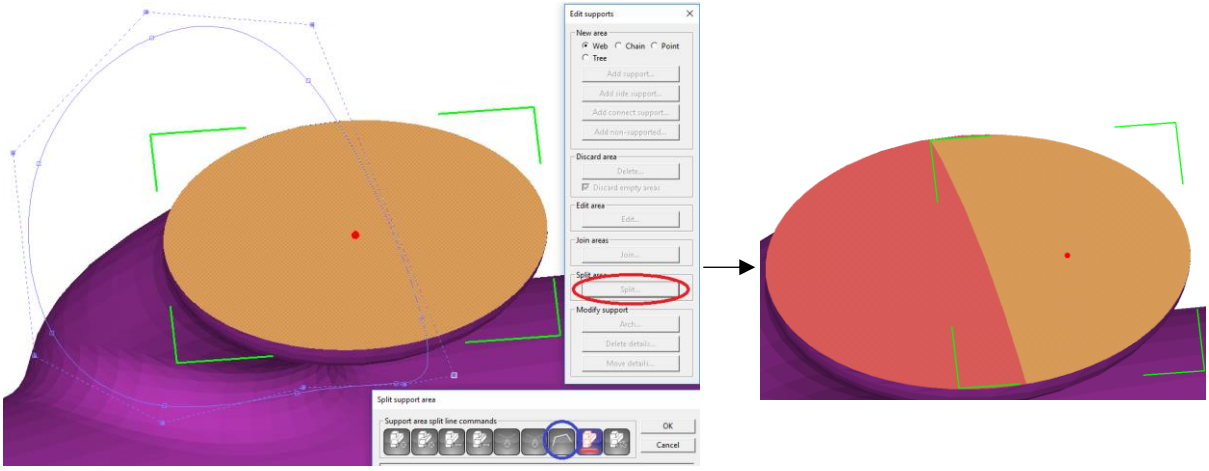
A project file can be accessed right after the start of the software by pressing 'Open…' and then selecting a project file.

5.2.2.2 Draw support area splitline command

This command allows to split the support area with free form curve into multiple areas.

The image below illustrates how to use the command:

- First select the support area from the model with RMB or by selecting from the active areas list (indicated with orange surface and green box corners in the image).
- Then give command 'Split' in the *Edit support* dialog (red circle).
- Then press the 'Draw support area splitline' command button in the *Split support area* toolbar (blue circle).
- Now it is time to select two or more points to define the split curve. In the image below we have drawn a direct light blue split line with two points. The start and end point must be outside the selected area to form a closed split line on the area. The points are given with LMB (left mouse button). If you make a mistake you can always cancel the last point with ESC button.

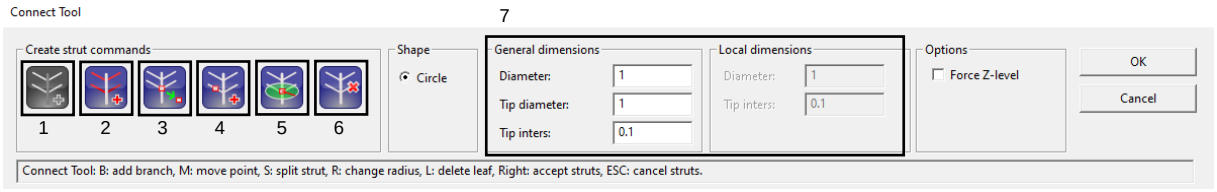


5.2.2.3 ***Connect Tool***

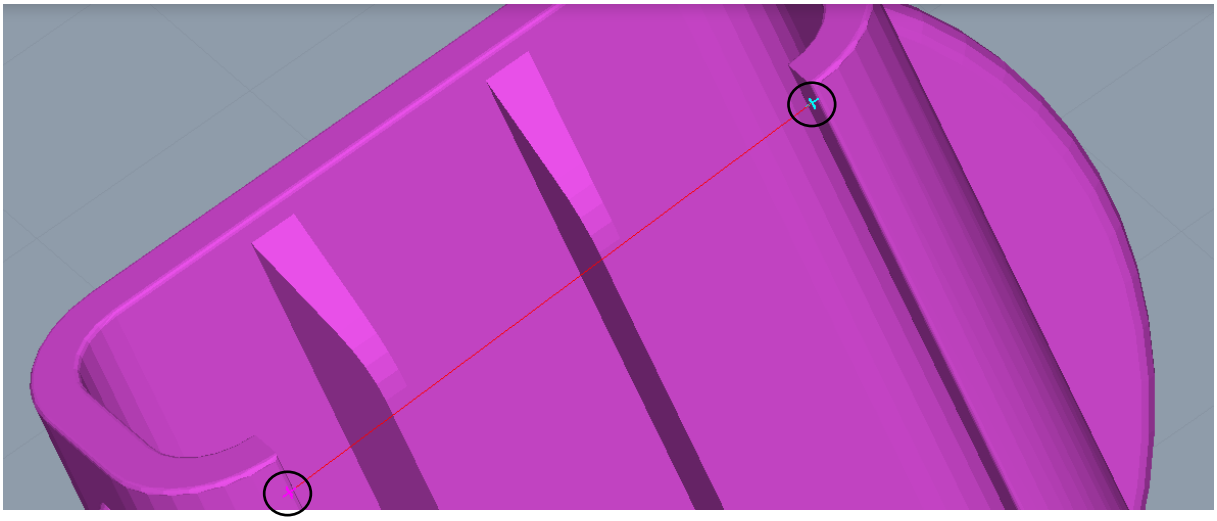
This tool makes it possible to connect either build parts or support parts to each other to ensure built quality and stability.

By creating a handmade strut which can be placed at a chosen position, these positions can be additionally supported.

The tool is in the tab ´*Modify Faceted*´ and will open the window shown beneath.



The first step is to select the ´add trunk´ and choose a suitable position for the trunk, which is shown in the following picture.

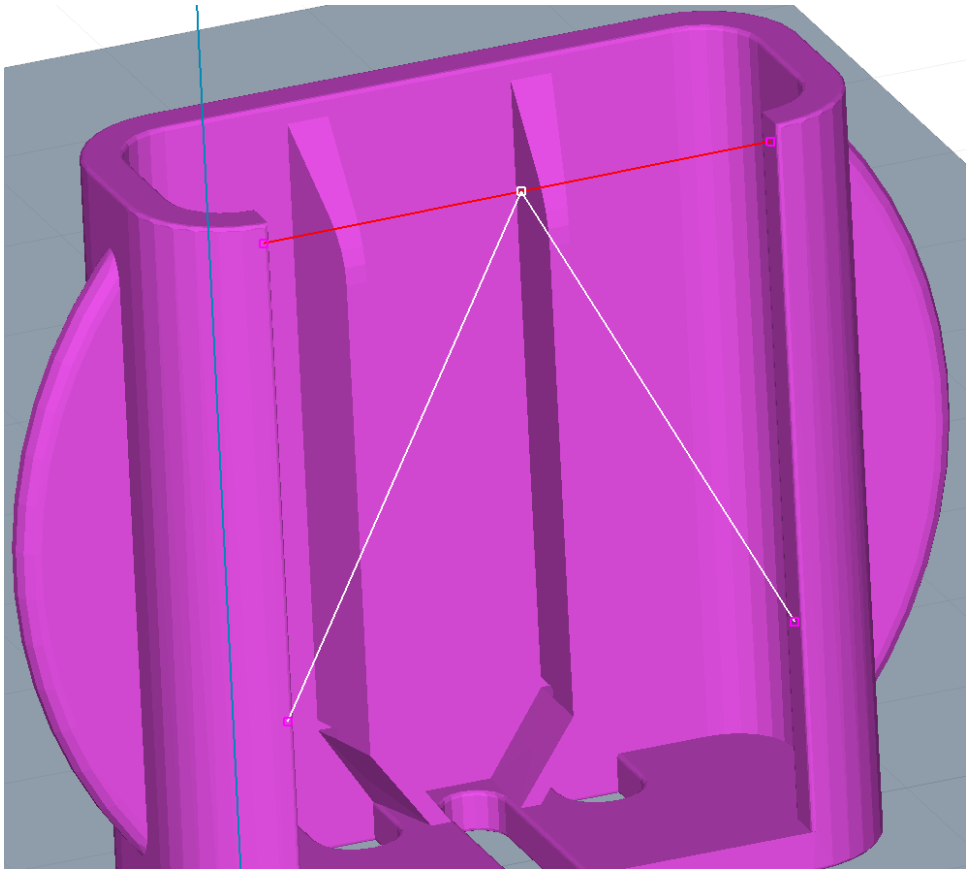


From that point several additional options are given to further optimize the created trunk.

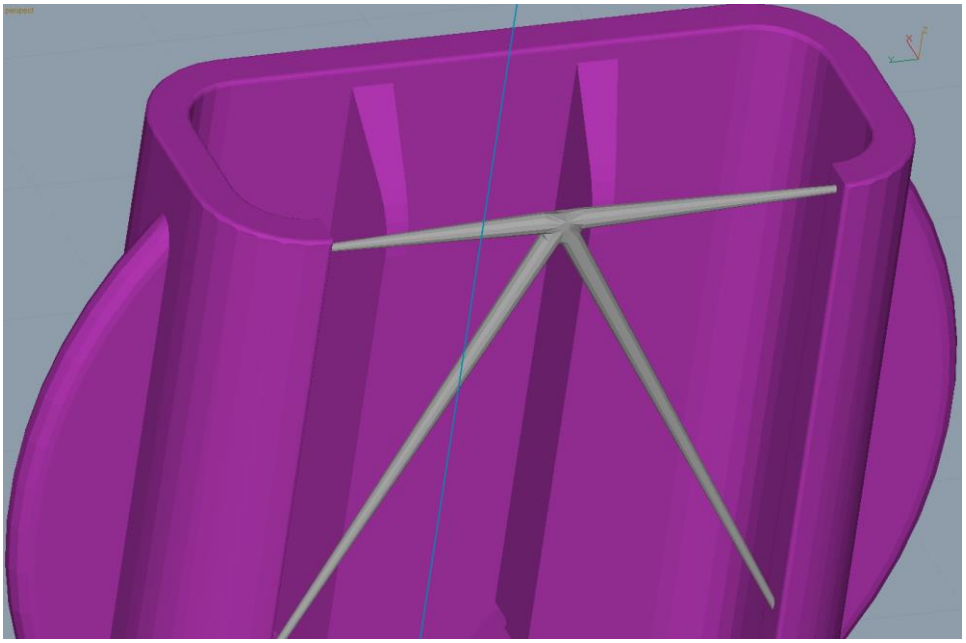
1. Add a main trunk for stability
2. Add additional branches to the trunk
3. Move single struts points
4. Add additional points to split the trunk
5. Change the diameter of a specific trunk point
6. Delete created leaf branches

Also, the general dimensions as well as the local dimensions of specific points can be changed in the shown tabs in the menu beside the manual commands (Point 7).

After the main trunk is set, as depicted in the picture above we will add to side branches as seen below.



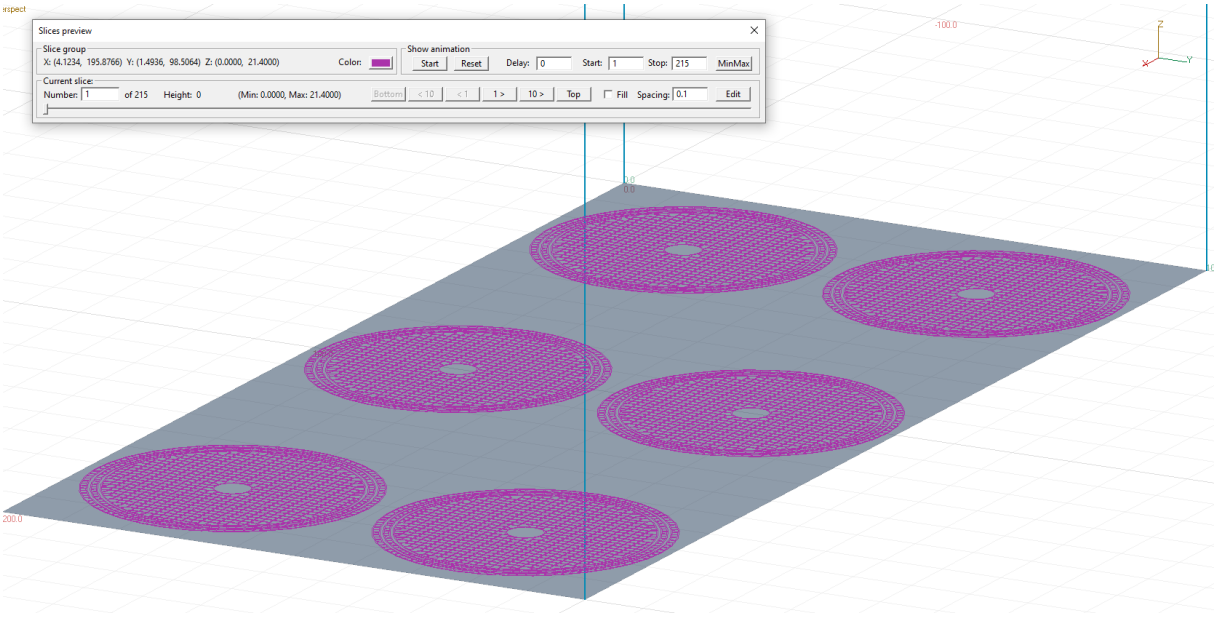
When generating a main trunk with two additional added side branches, it will look like this:



5.2.2.4 ***Preview of Slice Data***

To check your Slice Data for any errors or just to control if everything will work the way intended, CSS has now a tool for exactly that. After finishing slicing of your data, the resulting CLI file can be loaded like a conventional STL file.

When this is done, the following picture shows that the CLI is loaded correctly.

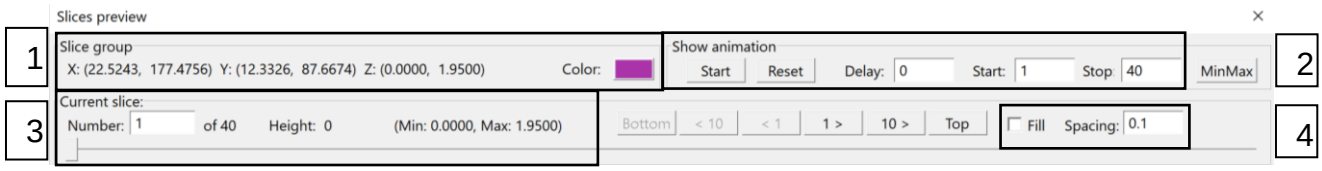


Now the dimensions of the parts will be displayed, to continue hit the ‘OK’ button.

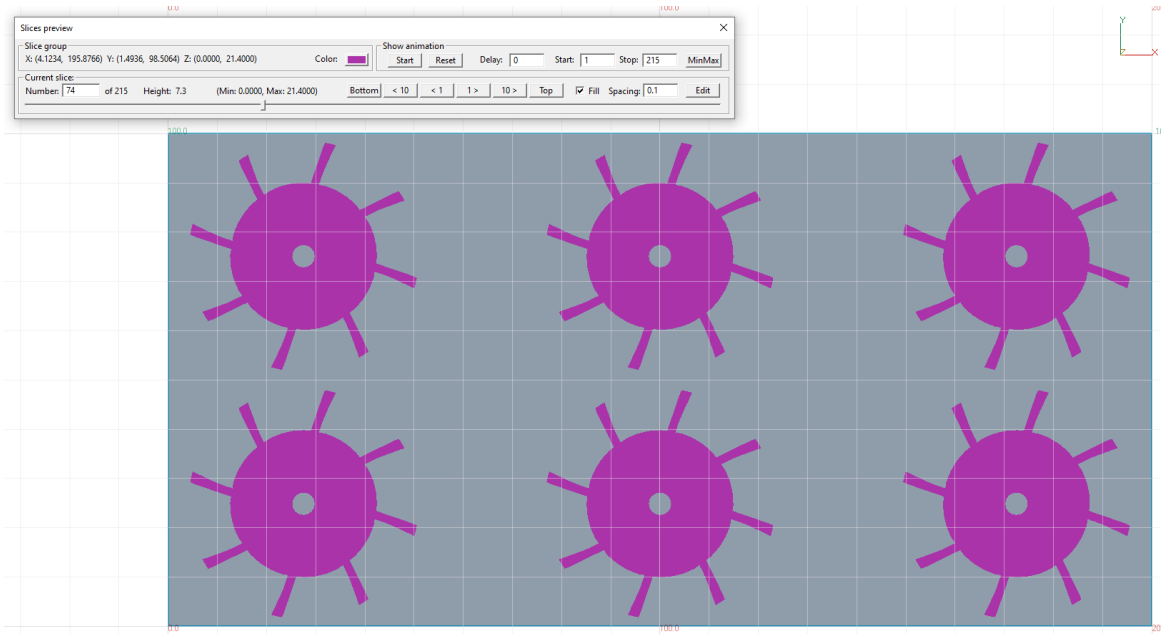
Normally, the best way of viewing at the slices is through the Z axis direction which can be done by clicking the  icon to change the current viewpoint.

In this slicing box different tools can be used to check for errors as well as to get an overall impression of how the data is prepared and how the print job will work out.

In the picture, which is depicted below, the most important features of this slicing box are shown and labelled:



1) Basic information of the slice group is shown: XYZ start and end coordinates
2) With this part of the tool an automatic animation can be reviewed. The delay as well as start and end point are open for configuration. The delay determines the time in between shown layers when the slice group is automatically animated.
3) With *Current slice* you can manually "scroll" through all slices to precisely investigate all specific layers.
4) With the fill/spacing function it is possible to simulate the actual hatching, therefore it can be used to check for non-closed contours or something similar (depicted below)



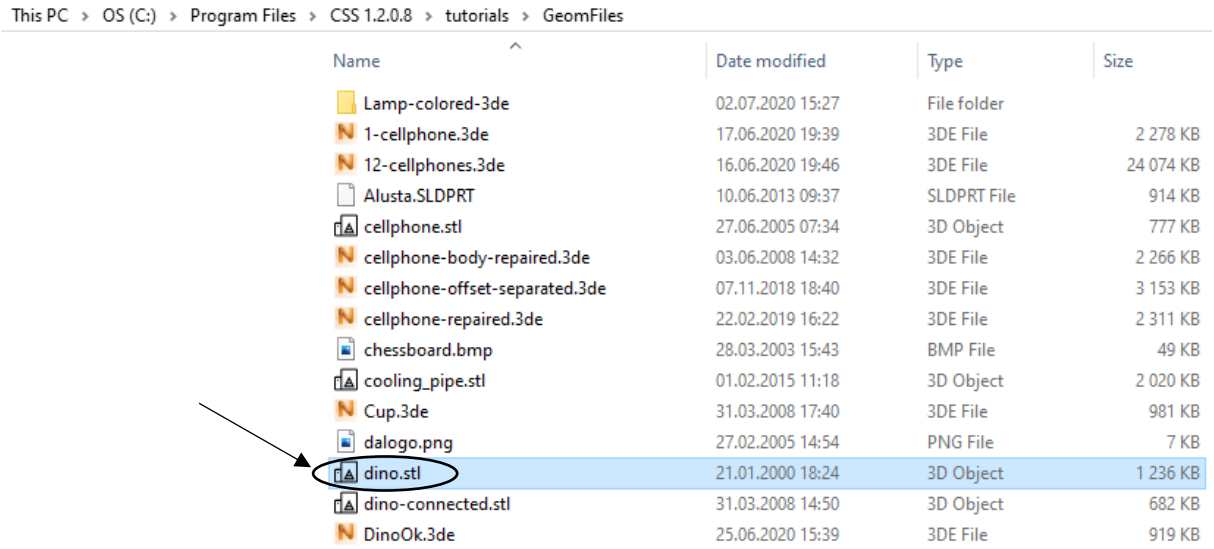
Note Attention: When using a low spacing value, the time to generate the hatches can increase. Therefore, adjust this value on demand.

If you want to compare or investigate several slice groups, you can use multi-selection to preview all of them at the same time.

We will investigate that in detail by using the dino.stl to create a CLI for that purpose.

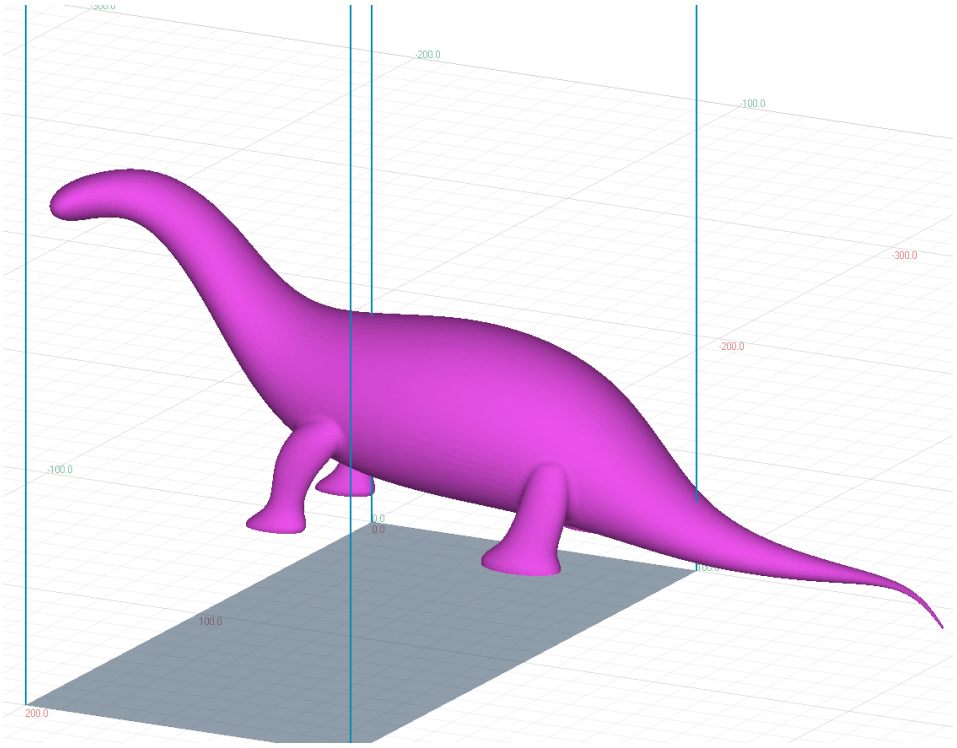
First, load the dino.stl into CSS by either using the command `open` or drag & drop.

The file is in the CSS directory: *C:\Program Files\CSS 1.2.0.8\tutorials\GeomFiles.*

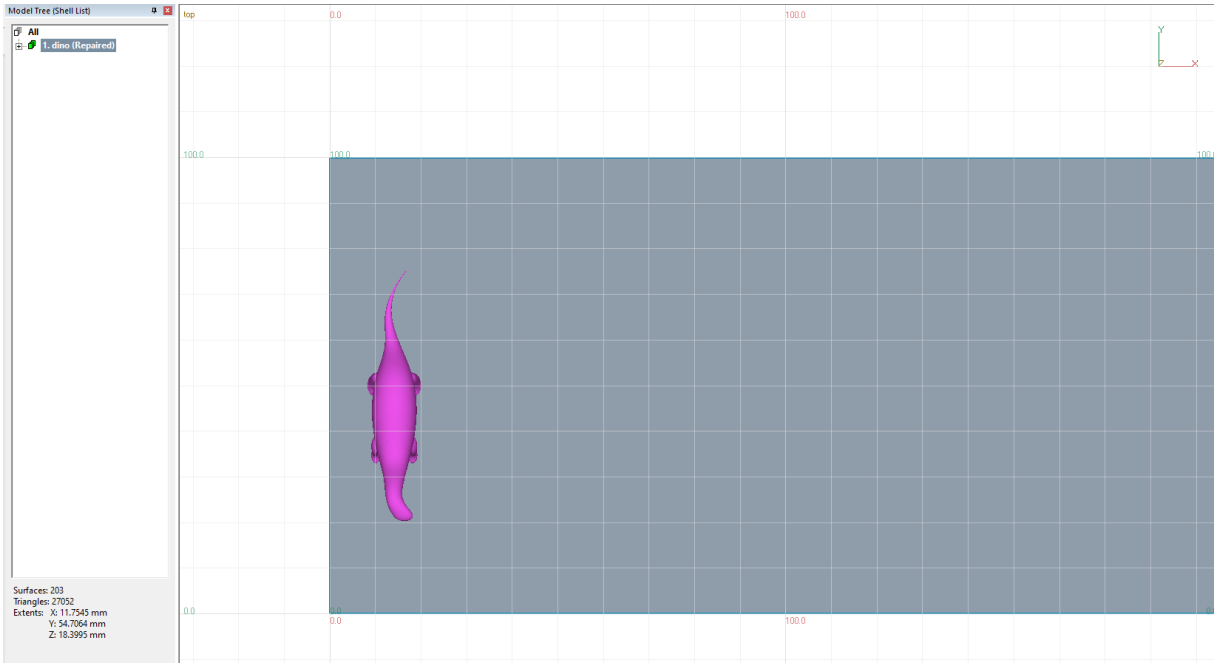


When loading the file, the process of verifying and repairing will take place and should leave you with one verified dino shell which is to big for the building space in the Y-direction and therefore must be scaled.

It should look like this:



We now uniform scale it to 20% and use the shortcut *Shift+P* to move it into the positive building area. It can now manually be moved with  until it is to your liking, which will kind of look like this:



Now we will create an array with the dino. For that have the dino selected and click the array-icon  and the following window will appear:

Create the array with the following parameters and make sure to have the original shell removed.

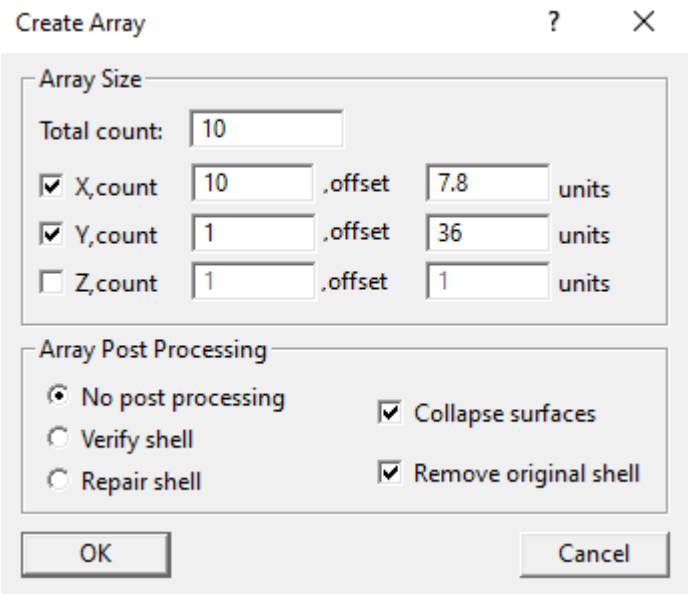
With that the array should be ready to be sliced.

Make sure to have the array selected in the Model Tree and click the icon 

Finally slice the generated array with the given parameters and select the desired location for it to be stored.

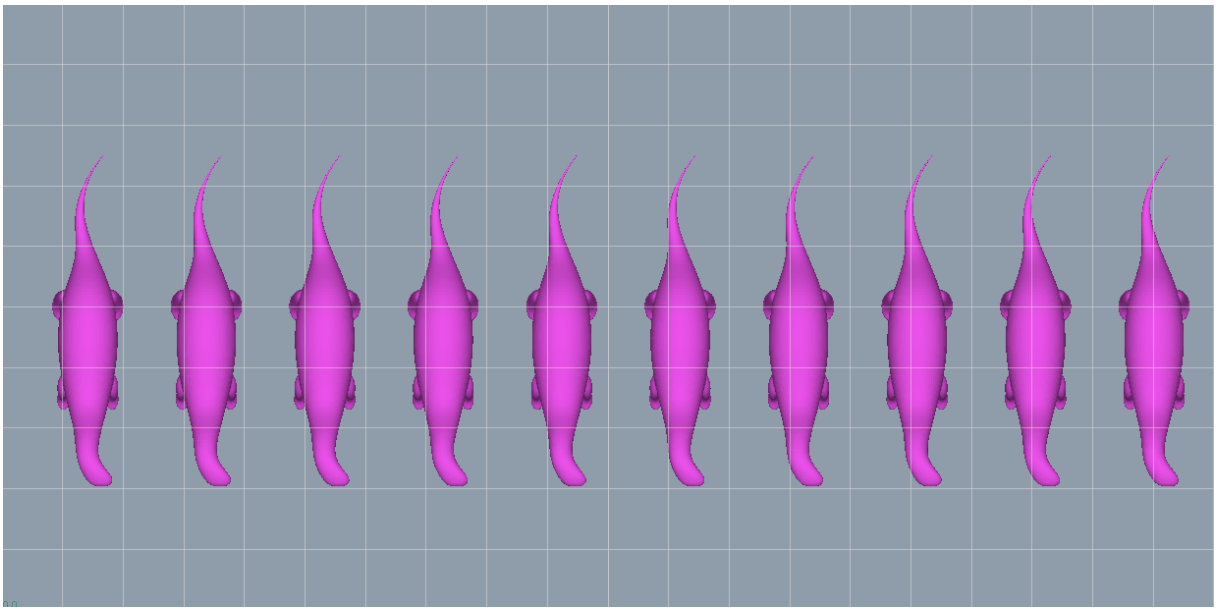
Also make sure to save the array as STL or 3DE file for further usage.

When finished, the resulting CLI can now be loaded into CSS via. `open` or drag & drop, which

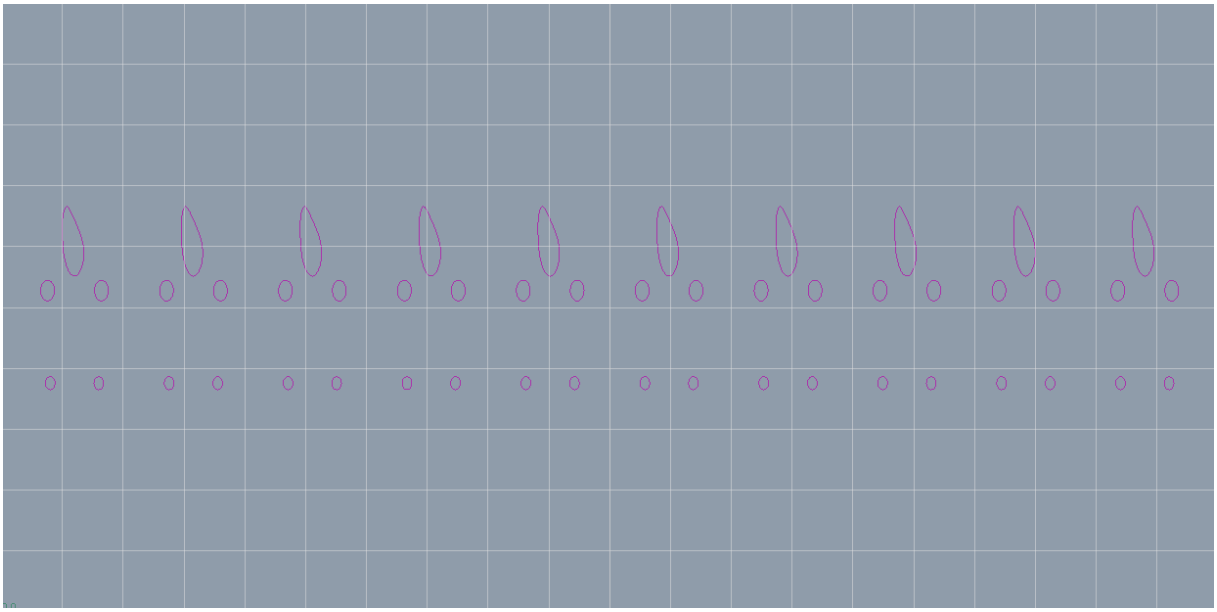


should look like the depictions below:

Ready to be sliced array:



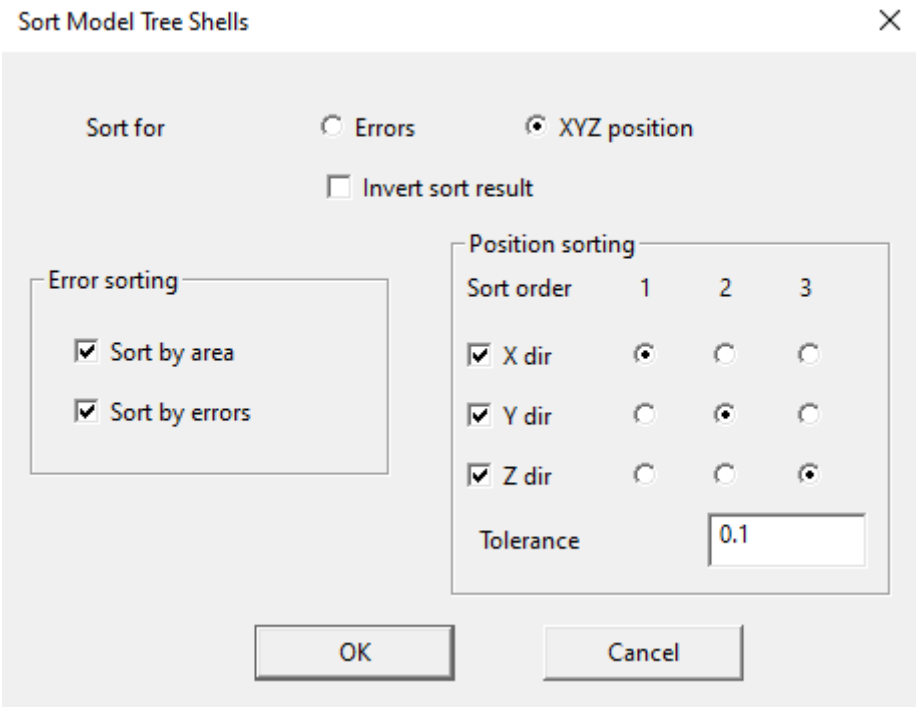
Sliced array (Layer 35) in the CSS slice preview:



5.2.2.5 ***Sort model tree***

To prepare a following automatic labelling step of parts it can be necessary to perform a sorting of parts inside your building area. For this reason, the command *Sort model tree* is given to ensure that when labelling an array or multiple parts, this happens correctly.

To sort your building area select under *Fix model* > *Sort model tree* and the following window will be shown:



With this it is possible to sort your parts either regarding their XYZ position or also by area or errors.

Sort by area: The shells in the model tree will be sorted in descending order by their surface area. The largest shell will be listed first in the model tree.

Sort by errors: Error shells will be listed first into the model tree. This can help to locate and identify error shells easier.

With that you can also freely decide which order (XYZ) as well as tolerances you want to apply for the sorting of the parts.

X dir sets the X axis direction sorting

Y dir sets the Y axis direction sorting

Z dir sets the Z axis direction sorting

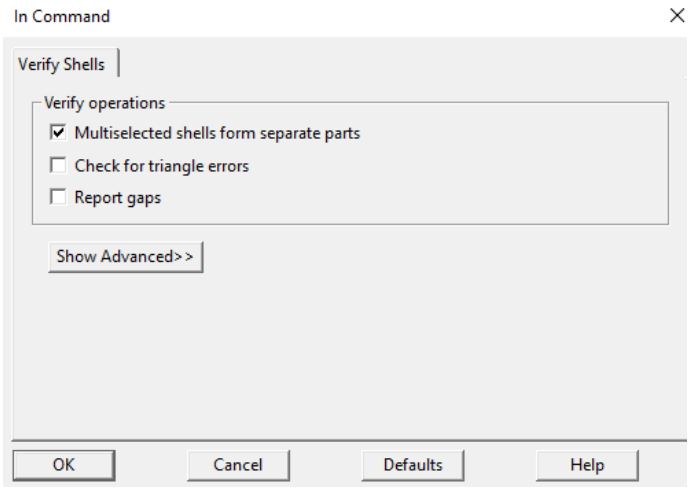
The **sort order numbers** (**1**,**2** or **3**) for each axis direction defines which axis direction is sorted first, second and last.

Tolerance defines the maximum distance between the minimum point coordinate values of the bounding box, which are still considered equal.

We will use that to sort the array we created in 5.2.5.4.

Therefore, reload either STL or 3DE, which we saved earlier for this.

Select the array in the model tree and use the command *Verify shells* under *Fix model*, which results in the following window:

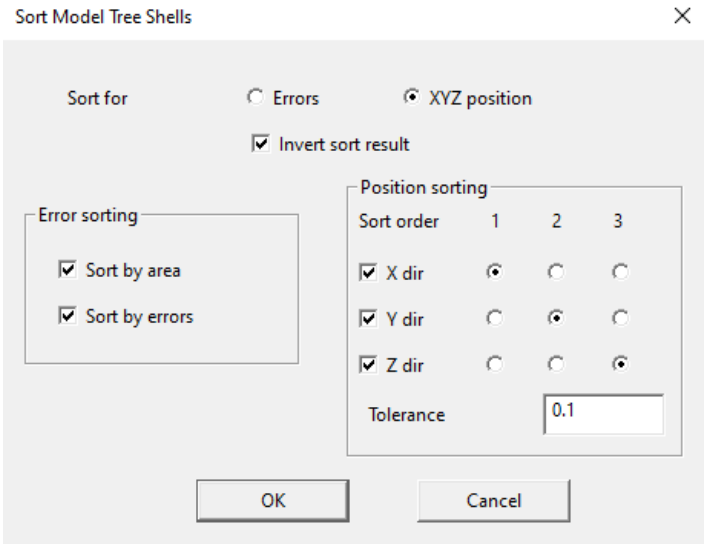


Make sure to have only *multiselected shells form separate shells* enabled and run the command.

Now all dinos should be separated in single shells and therefore ready to be sorted and then labelled accordingly.

Multi-select all ten shells in the model tree and run the command *Sort model tree* under *Fix model*, which opens the sort window shown in 5.2.5.5.

For a correct order have the parameters selected as shown in the picture below:



After running the command, all shells should be sorted correctly for the labelling in the next step.

5.2.2.6 ***Labelling Tool***

With the labelling tool you can select a certain area on the surface of the to-be-printed part which should be labelled for identification purposes.

When using this for a serial production print (several similar parts), you can label all parts individually to be able to distinguish them later.

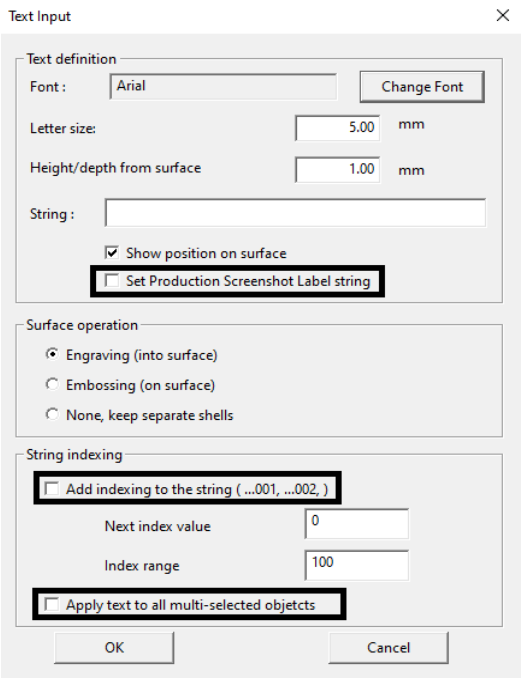
For this multiselect all parts which you want to label and select under *Create Geometry* > *Create 3D Text* and the following window will be opened.

A name for the string must be given, then click on 'Set Production Screenshot Label string'.

If an indexing with the string for the multiselected objects is desired, choose this function as well.

How the index values and range is defined you can change in the boxes below.

As the last point click 'Apply text to multi-selected objects' to project the string on all selected shells/parts.



Note Strings are added in the reverse order in which they are stored in the model tree. The last part in the model tree gets the first string and so forth.

For that we will continue with the array we created and sorted in point 5.2.5.4 and 5.2.5.5.

Note Make sure to have the viewing angle and how close you are to the object set beforehand in a way that the area which should be labelled is shown accordingly.

Multiselect the ordered shells in the model tree and select *Create 3D Text* under *Create Geometry*, which opens the text input window shown above.

Here the font, letter size, height/depth of the labelling as well as type of labelling (engraving or embossing) can be selected.

Make sure to have "*Set Production Screenshot Label string*", "*Add indexing to the string*" and "*Apply text to all multi-selected objects*" checked on.

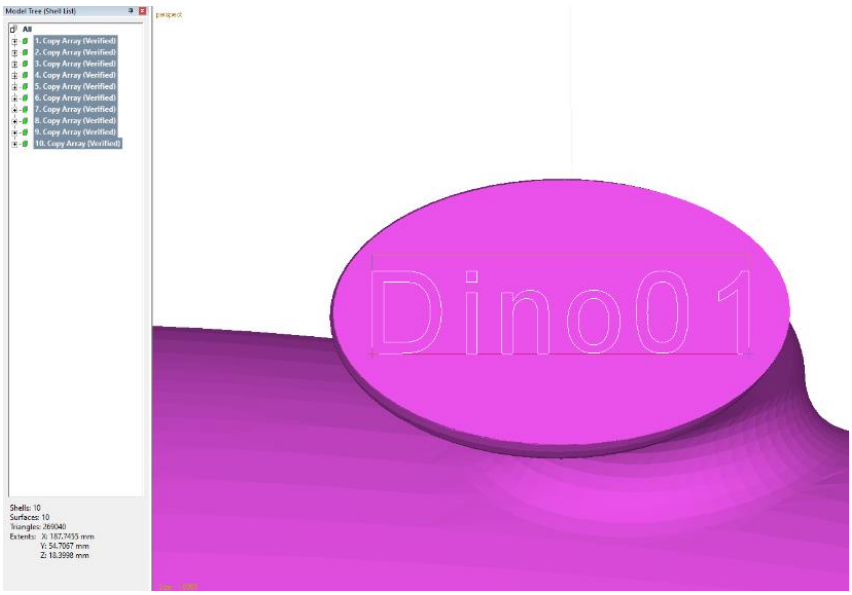
The index value and the index ranged can also be defined to meet the given requirements.

When all parameters are defined, we click 'OK' and define the area which should be labelled.

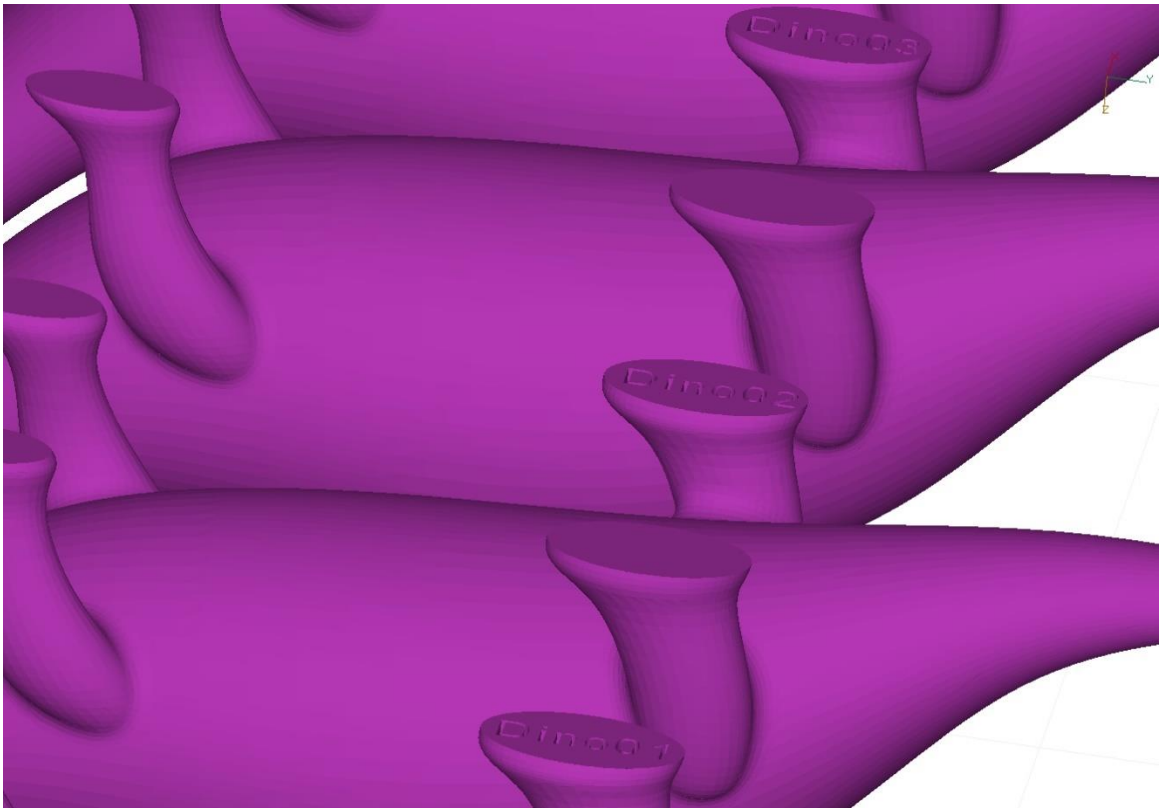
The labelling will be shown as the first-string number.
It is possible to move it by dragging it with the ***LMB*** and changing the size through scrolling with ***MMB***.

When the position and size is decided the labelling is done by finishing the operation with the ***RMB***. All shells which were multi-selected before will now get a successive number through Boolean merging the number shells onto the original shells.

Note Make sure to use a flat surface for the engraving/embossing if possible.



When all models were labelled correctly, it should look like this:



5.2.2.7 ***Production Screenshot***

The Production Screenshot feature makes it possible to record your final print status by saving a screenshot from a desired viewpoint of the building platform.

This is meant for generating information which is helpful for documentation of identification and definition of given parts.

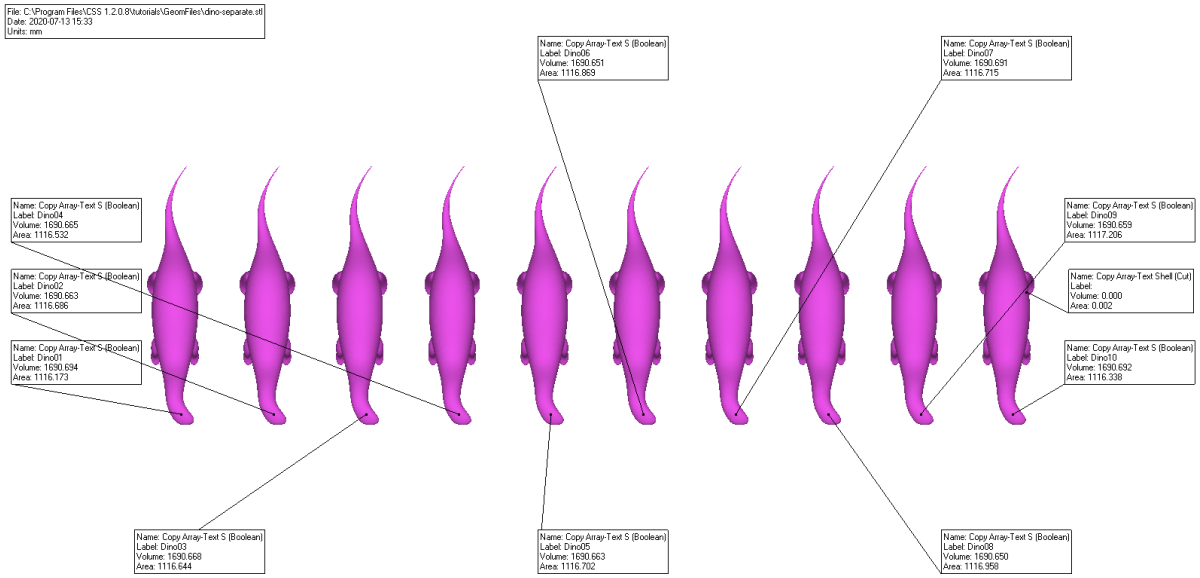
When all parts are edited, orientated and supported, the next step would be to slice the parts to generate the desired slice file.

Now before doing that we can set up a production screenshot to have all informations documented. For that, select under *Manufacturing > Production Screenshot*.

Now your print area will be viewed from the Z-axis with additional boxes which are connected to all parts present on the platform (means in the model tree). Those give details regarding their areas and volume values as well as their label-tag and name and date, for now.

Note The information given will be extended and can be changed in the future.

How this looks like is depicted in the picture below:



The given boxes can be selected and dragged to another position if desired.

After adjusting the settings of the screenshot, click ‘Export as image’. Now you select the location, where the image should be saved on your hard disk and click ‘Save’.