cara®
Application Guide

cara Print system

Application Guide for cara Print 4.0 device, CAM software, dima Print materials, cleaning- and post-curing process.

Version 06.2021

Giving a hand to oral health.

KULZER
MITSUI CHEMICALS GROUP

Contents

1. Review of STL files

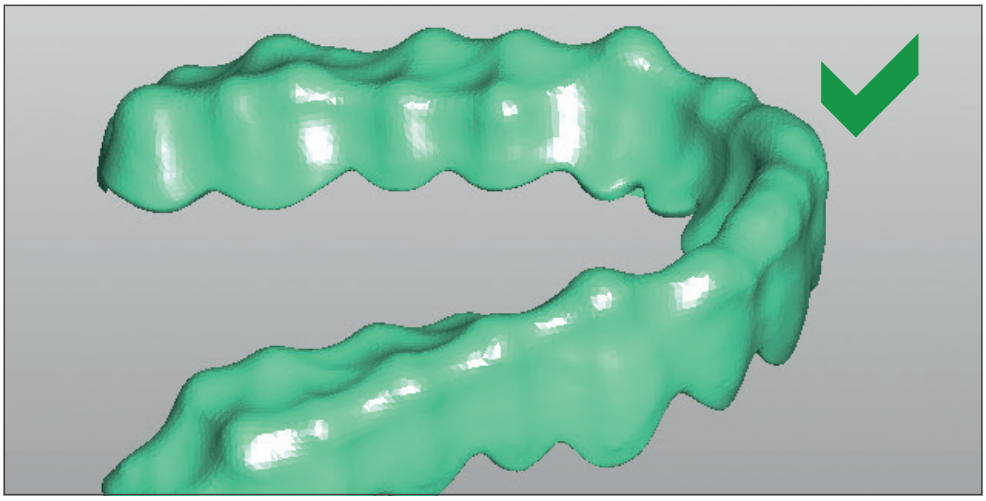
THE 3SHAPE MATERIALS FILE FOR DIMA PRINT MATERIALS IS AVAILABLE VIA DOWNLOAD:

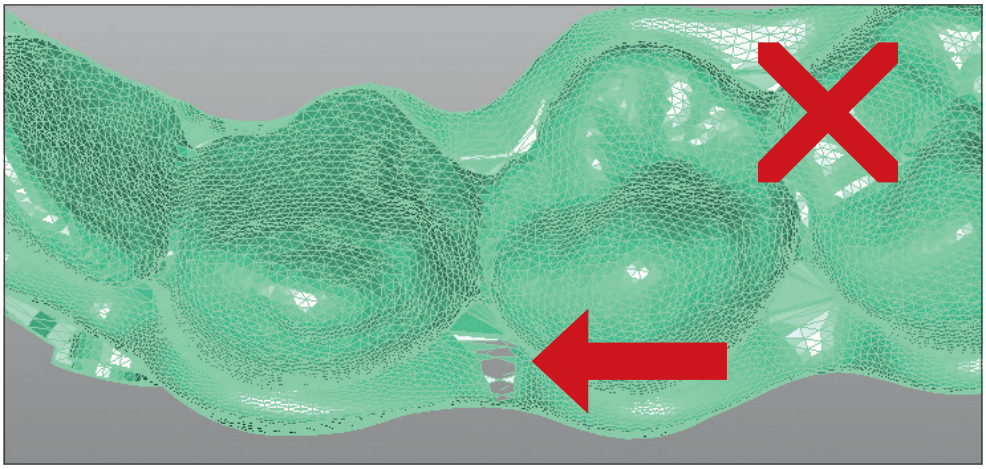
www.kulzerUS.com/ cara-print-3shape-dme

Construction and design guidelines:

It is critically important to review all files prior to printing and to ensure that they are completely free of errors, visible or otherwise. Please always check the mesh visually to ensure that the structure is truly “closed”. If it is not, the file must be adjusted using your CAD software, or using the “Solidify scan” function in the cara Print CAM software.

The surfaces of structures to be printed must be as smooth as possible. If there are any mesh problems (i.e. self-intersections, open faces or inverted triangles), please repair them or, if needed, smooth the surface again.





2. Positioning of objects on the building platform

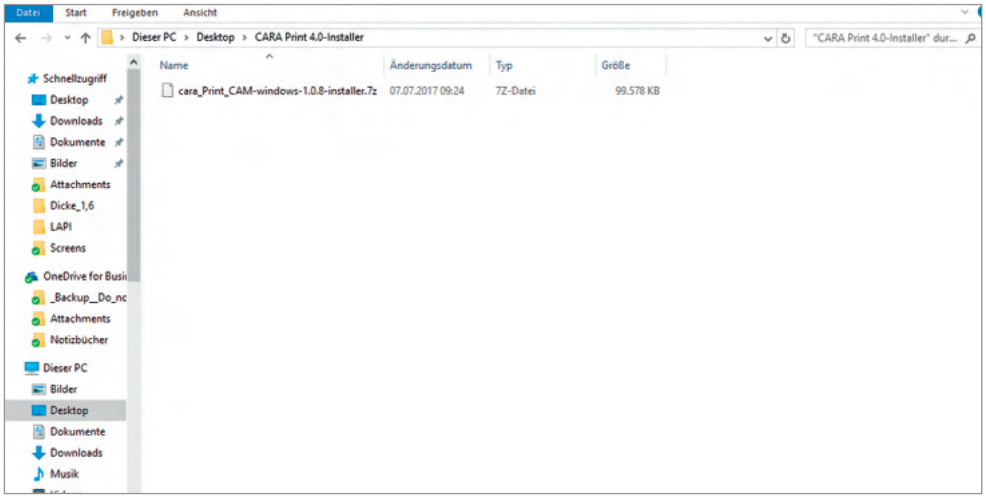
How does one best position objects on the building platform and what is the optimal orientation for objects to be printed?

- It is almost impossible to give a general recommendation regarding the “best possible” position within the building platform. However, you should always “think in layers” when it comes to the additive process of 3D printing, as opposed to the subtractive process of milling.
- This means that you should focus on the “self-supporting” nature of layers both individually and in combination. As a result, the location of the object to be printed is not important, assuming an equal level of precision at all points of the building platform.
- Overlapping areas should be strictly avoided.

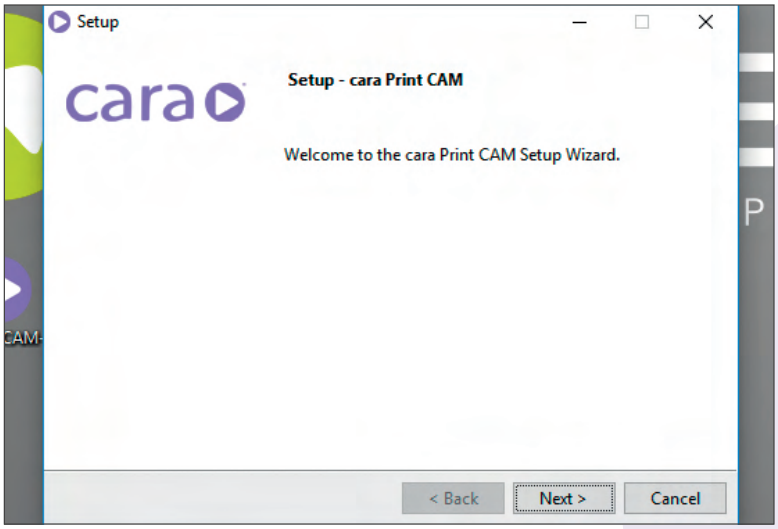
3. cara Print CAM

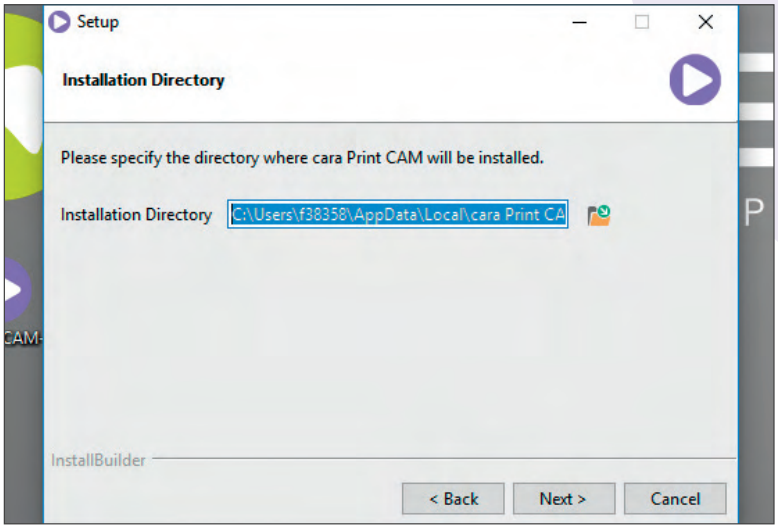
3.1. Installing cara Print CAM software

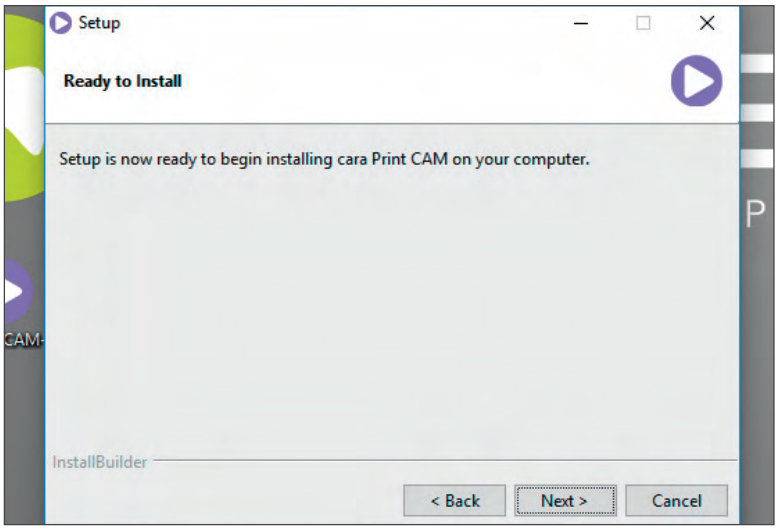
cara Print CAM – the so called slicing software – is part of your start-up package. Please install it on a separate computer using the USB stick provided. It creates printable files based on the STL file of your CAD program. Double-click on the most recent installation file to start the installation.

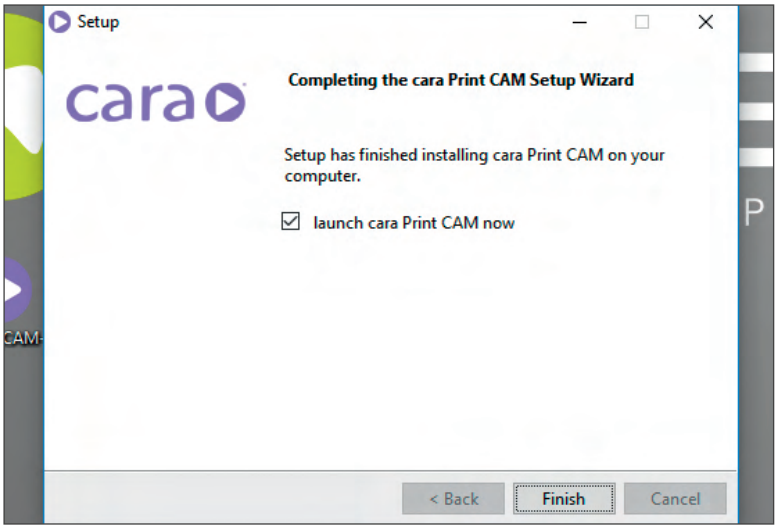


Follow the directions on your screen.


Setup
Setup - cara Print CAM
Welcome to the cara Print CAM Setup Wizard.
< Back
Next >
Cancel


Setup
Installation Directory
Please specify the directory where cara Print CAM will be installed.
Installation Directory
C:\Users\f38358\AppData\Local\cara Print CA
InstallBuilder
< Back
Next >
Cancel


Setup
Ready to Install
Setup is now ready to begin installing cara Print CAM on your computer.
InstallBuilder
< Back
Next >
Cancel

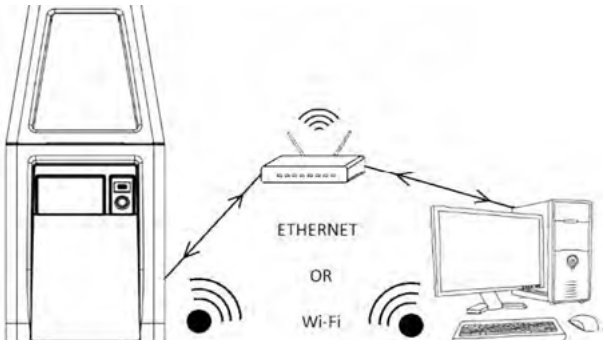


Once the installation is complete, you will find a shortcut on your desktop.

3.2. Connecting to cara Print 4.0

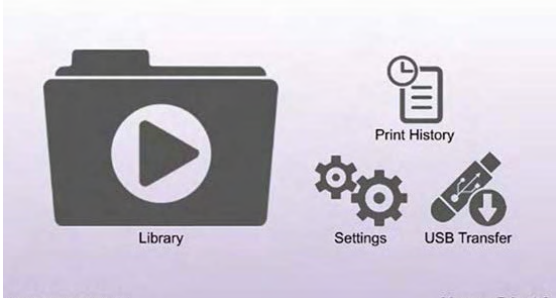
Networked Configuration

The cara Print 4.0 can be networked with a computer or laptop through a common router.



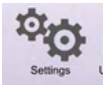
Primary Menu

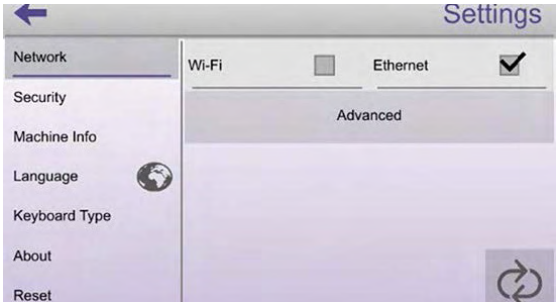
This page is the main starting point for all actions and inputs for the operator. From here, the pages for starting a print, transferring files, changing the settings, and viewing the printer's history can all be accessed.



Wi-Fi/Ethernet Selection

Press the Settings icon on the Primary Menu page. The touch-screen will display the **Settings Menu**.



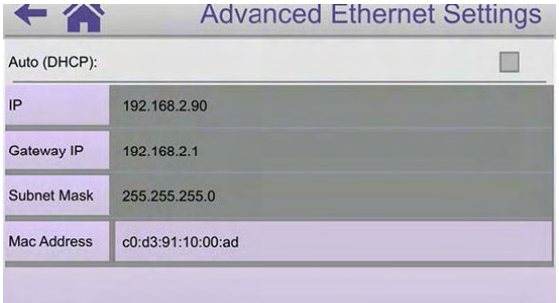


Select either **"Wi-Fi"** or **"Ethernet"** by touching the box to the right of whichever network connection type is desired. Initializing the printer for the selected network connection type may take a few moments. Once the network connection type has been initialized, the operator may press the **Advanced** field on the touchscreen to display additional information about the network connection type that was selected.

Advanced Ethernet Network Connection

Pressing the Advanced field after selecting the Ethernet box will display the Advanced Ethernet Settings page. Many Ethernet settings are displayed. The operator can select each setting field if desired and change them if needed. Once the proper settings are displayed, press Apply.

- It is often best to press the box to the right of "Auto (DHCP)" and "Apply" to allow the network to establish the IP address.
- For Ethernet to work properly, an Ethernet cord must connect the 3-D printer to the network on which the cara Print 4.0 desktop software is running.



Advanced Wi-Fi Network Connection

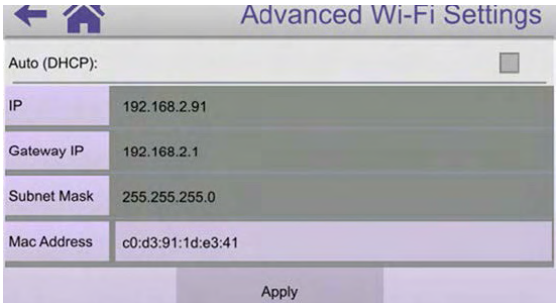
If a Wi-Fi Network is selected, the Wi-Fi Network and Wi-Fi Password fields will be displayed on the Settings page. The operator must enter a valid Wi-Fi Network name in the Wi-Fi Network field.

A list of active Wi-Fi networks will be displayed if the operator presses the field to the right of Wi-Fi Network on the Settings page. Once the desired Wi-Fi Network is found on the list, the operator can select it by touching the name on the touchscreen. The selected Wi-Fi Network will be displayed in the field to the right of Wi-Fi Network on the Settings page.

If the network selected is a secure network, a password must be provided. To enter a password, press Wi-Fi Password field and enter the password.

By selecting Advanced on the Settings page, the Advanced Wi-Fi Settings page will be displayed.

On this page, the operator may manually enter the IP address, the Gateway IP address and the Subnet Mask.

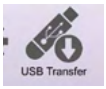


- Instead of manually entering the addresses and information in the setting fields, it is often best to press the box to the right of “Auto (DHCP)” and “Apply” to allow the network to establish the IP addresses.
- For Wi-Fi to work properly, there must be a wireless network to which the printer can connect. Additionally, the desktop software must be running on a computer that is connected to the same network.

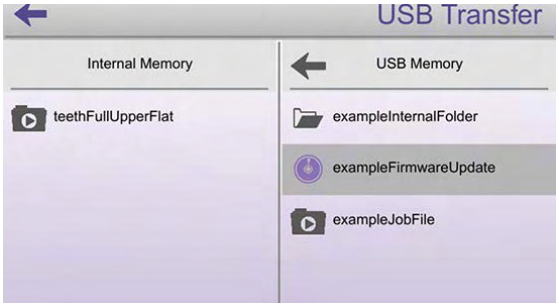
Adding a Print Object Via USB Device

Print objects can be added to the cara Print 4.0 printer via USB Device whether the printer is being used in either the “Stand Alone” or “Networked” Configuration.

Insert a USB Device in the USB port on the front of the cara Print 4.0 printer.

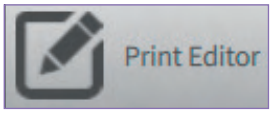


Press the USB Transfer icon on the Primary Menu page. The touchscreen will display the USB Transfer Menu. Files that are located on the USB drive will display under USB Memory. Select the file to transfer. A pop-up menu will ask if you want to transfer the selected file. Select “Yes”.

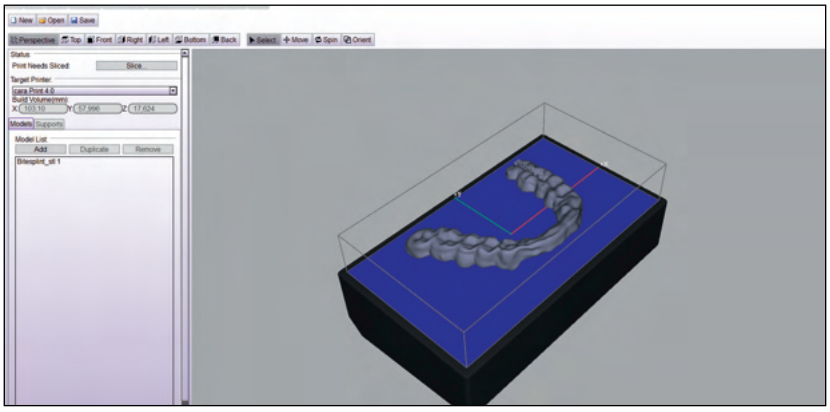


3.3. Positioning and slicing the CAD structure

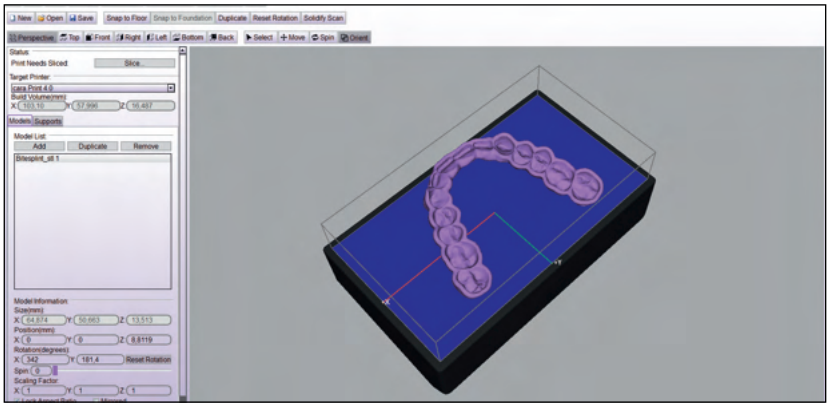
Unlike the process of preparing digital files for milling, the 3D printing process requires the division of the digital structure into individual (print) layers. To do this, please open the “Print Editor”.



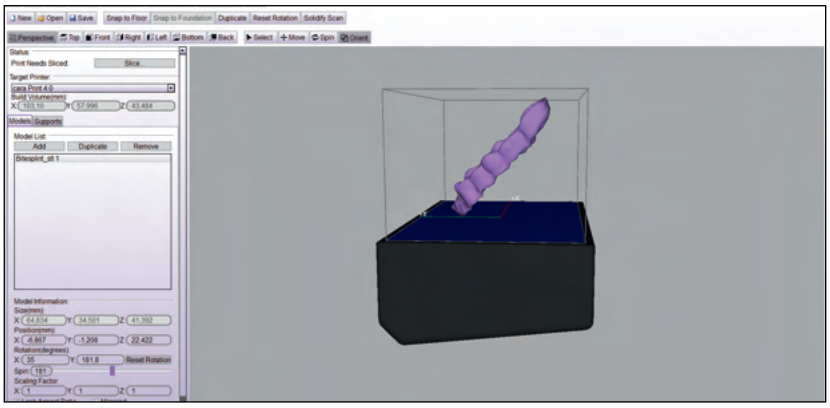
By clicking “Add” in the menu, you can load your flawless (see Section 1) dental appliance file (STL) in the Print Editor. This will be displayed as a grey object within the possible print area (blue).
The dental appliance is now ready for positioning. The available print area is shown as a transparent block with grey borderlines:



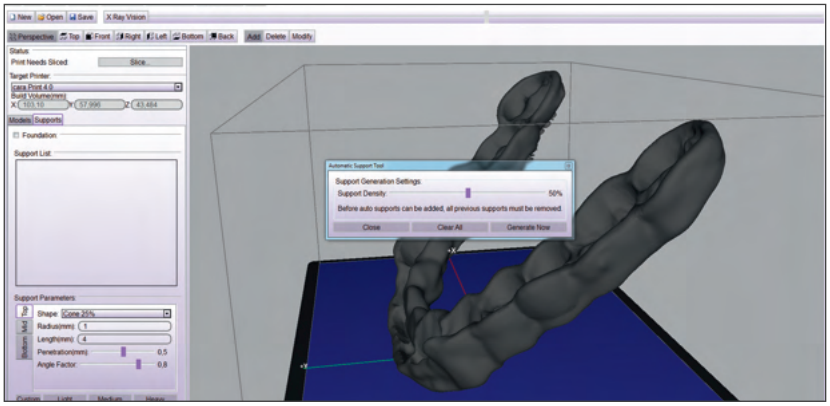
This picture with a splint (dima Print Ortho) represents a special case. The inner cavity (the fitting surface, not the outer surface) of the splint must face upwards. This will allow the support structures to be added to the other side at a later point in the process. It will be easier for you to remove the support structures from the smooth outer side of the splint.



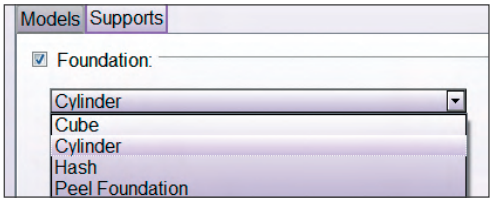
Click “Orient” in the menu to angle the appliance. This saves space and allows you to print several appliances at the same time. Additional appliance files can be added – or duplicated – within the same building platform by selecting “Add” in the menu.

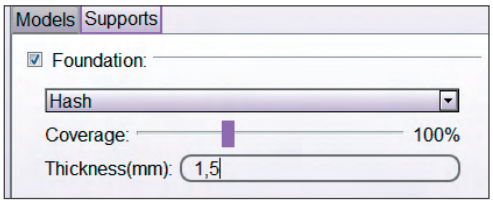


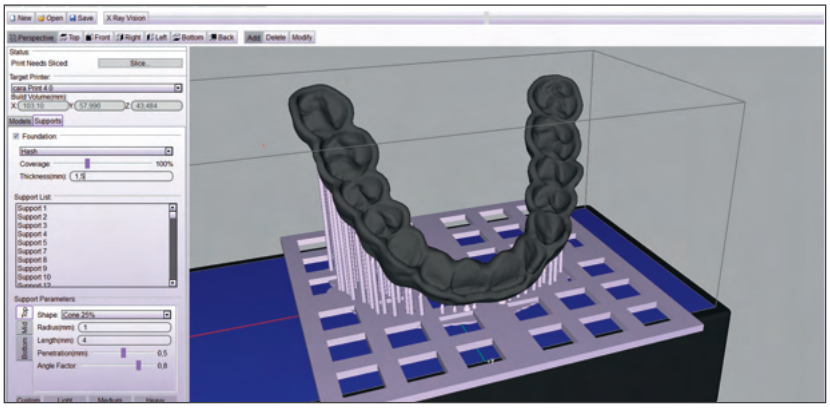
After you are finished positioning, click “Support tool” in the menu to automatically generate the supports. The overall number and spacing of support structures depends on the material and indication.



Check “Foundation” to open a list with different types of foundations. Choose “Hash” to create a grid type foundation, which provides for better stability. The thickness of the foundation should be set to 1.5 mm. See Section 5 for more details according to the specific indication.

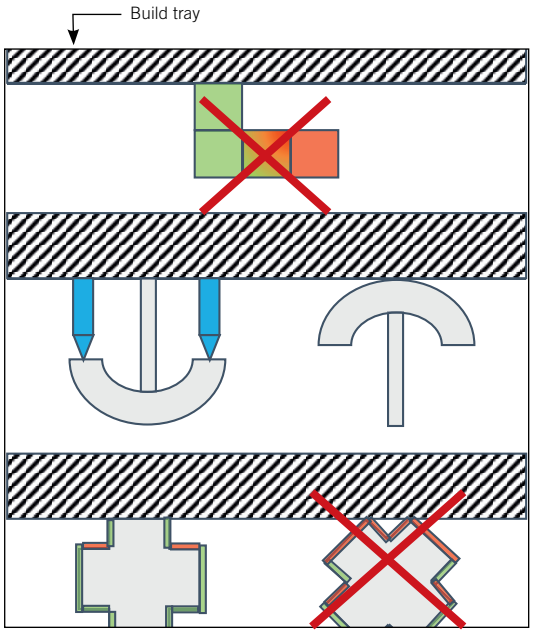






3.4. Creating and arranging support structures

Ideally, the individual print layers should provide one another with mutual support. However, you can add additional supports if this is not the case (according to the x-ray function in cara Print CAM). If supports are not added, the unsupported parts could later be left floating in the monomer solution. Unsupported parts can be shown in red by using the "View Slices" function. Please refer to chapter 3.5.



How many supports are needed and where?

- It is very important to add supports to the deepest points of the object to be printed.
- The right number and spacing of supports depends on what you are printing.
  For more information, please see the descriptions of various dental appliances in Section 5.

Arranging supports:

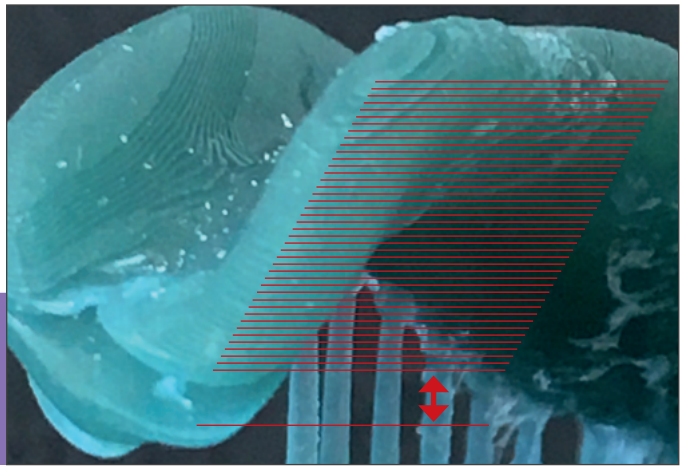
It is generally best to use the software's auto-function, which calculates the support structure requirements based on an algorithm and positions them on the virtual object.

However, please pay special attention to the arrangement of supports, as this always depends on the type and angle of the object you are printing.

Additional supports:

Be space saving when adding additional supports, and be very careful of overlapping objects. When you add additional supports, it is possible for them to interfere with neighbouring parts. Check before each print to ensure this does not happen.

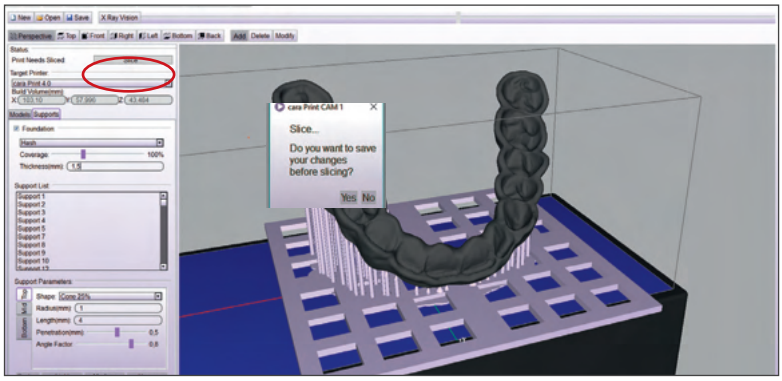
In some cases, fragile parts and extremities might need additional supports (e.g. impression trays for the lower jaw with ascending ramus):



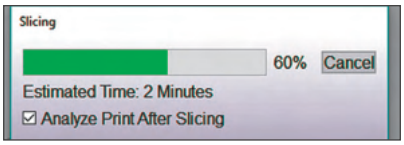
! **IMPORTANT: MISSING SUPPORTS CAN CAUSE DELAMINATION. THIS DETAIL IS VERY IMPORTANT!**

3.5. Creating the print file (slicing)

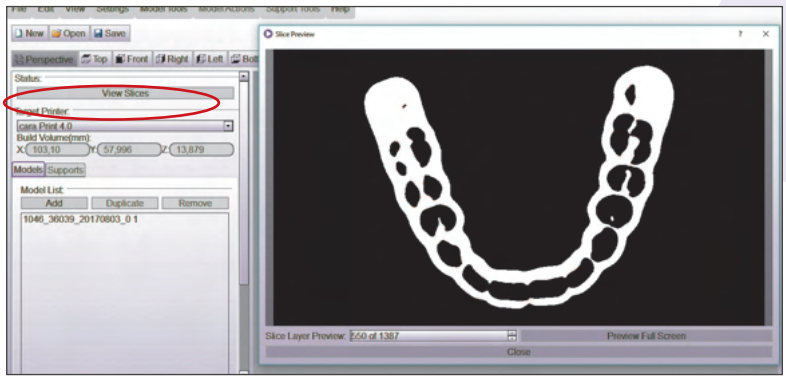
After creating and arranging the necessary supports, the next important step involves dividing up the virtual object into individual, horizontal print layers, also known as “slicing”. Start this process by clicking “Slice”.



You will then be asked to select a name and location for saving the resulting .cpj file. Save it using a file name and location of your choosing. Later, you will need this file to start the 3D printing process with cara Print 4.0. The slicing process can take several minutes depending on the size of the object.

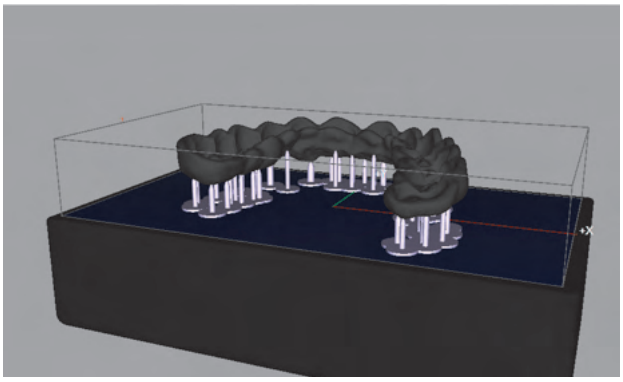


Afterwards, you can select “View Slices” to see each slice individually in a cross-sectional view. This lets you review the object in detail and ensure that it is sufficiently supported.

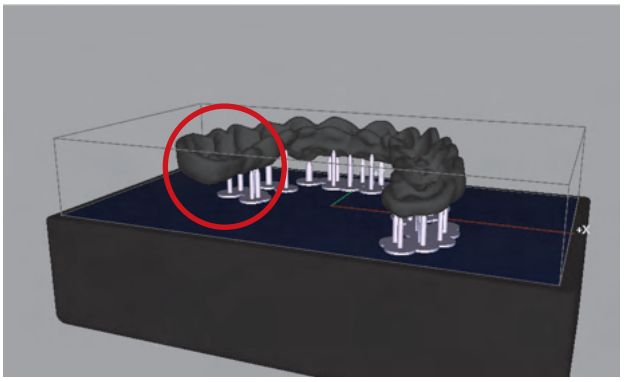


If portions of the object are not supported sufficiently, this function will display the edges of the receptive layer in red to make it visible.
Scroll through the layers and identify where the red areas evolve from.
There, additional supports are needed.

Supports sufficient



Supports not sufficient



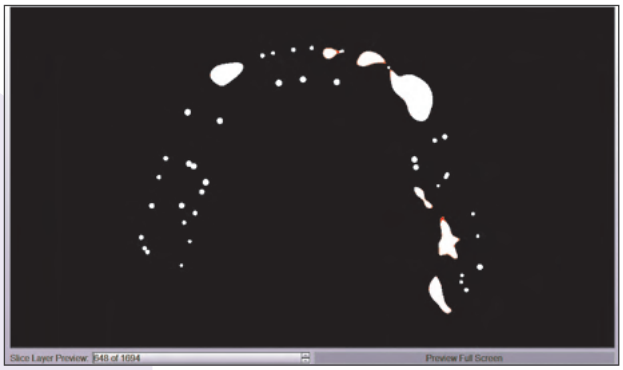
The two cases shall illustrate the function “view slices” that can help to find unsupported areas.
On the left hand side the splint is well supported, on the right hand side essential supports have been deleted to show the function.

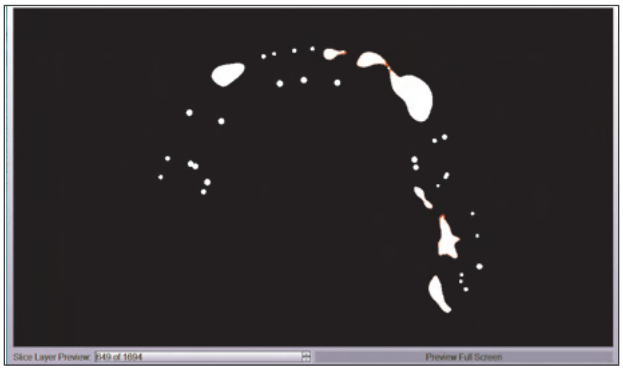
Slice views: scrolling through the slices starting from 0 to higher numbers.

When scrolling through the slices there are three options to do so:
- Mouse scroll wheel: up and down 12 slices
- Page up/down: up/down 10 slices
- Arrows up/down: up/down 1 slice

Visible are the actual images the projector will illuminate on the build plane. White is where curing will take place.
Right now the supports are visible.

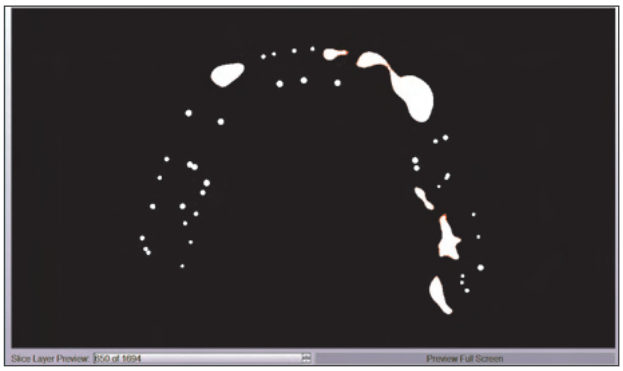
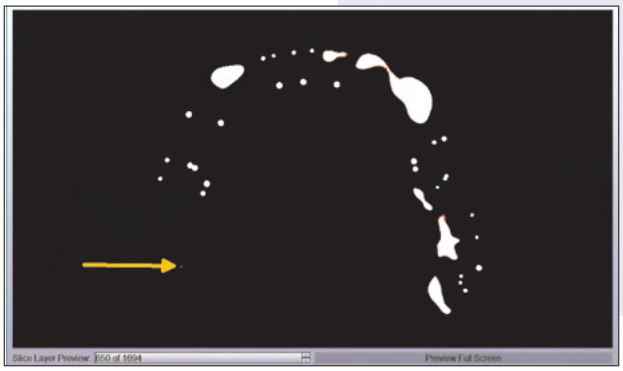
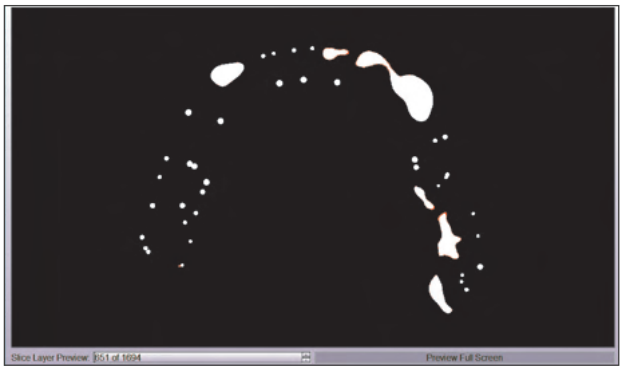
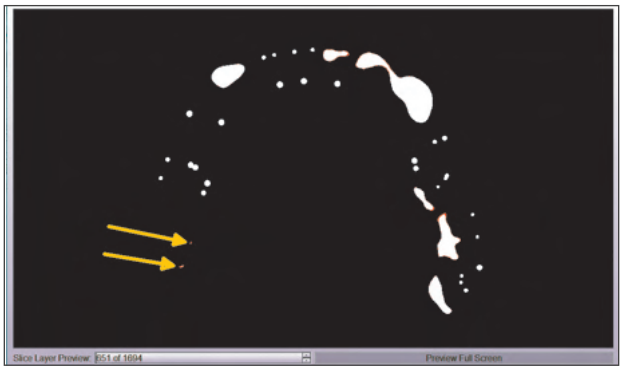
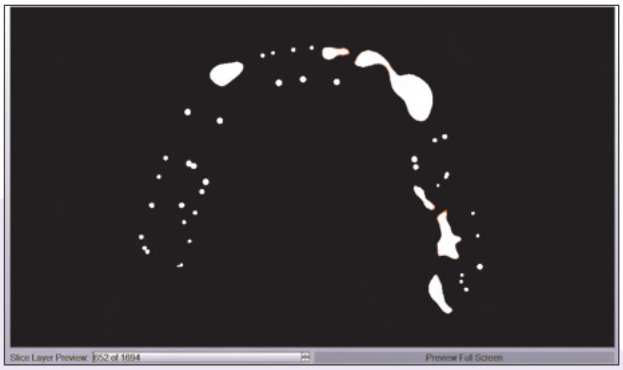
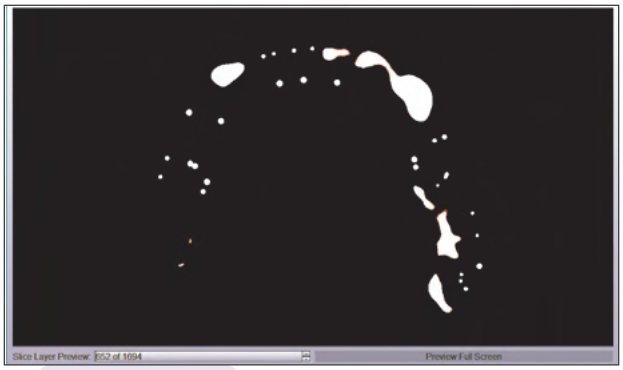
When scrolling through eventually the splint body will emerge as white areas to be illuminated.
Wherever a white area emerges completely out of black, a support is most likely missing.





Now try to find the slices where the unsupported part begins to evolve.

! **REMEMBER: MOUSE WHEEL 12 SLICES, PAGE UP/DOWN 10 SLICES, ARROWS UP/DOWN 1 SLICE.**

! **REMEMBER, THE POSITION WHERE THE SLICE SHOWS THE RED DOT OR SMALL AREA IS WHERE THERE NEEDS TO BE A SUPPORT.
THE AUTOMATIC SUPPORT FUNCTION WILL MOST LIKELY ENSURE THAT THERE IS A SUPPORT SET.
IF NOT, TRY INCREASING THE DENSITY UP BY 10 %.**

4. The 3D print process
4.1. Filling the resin tray

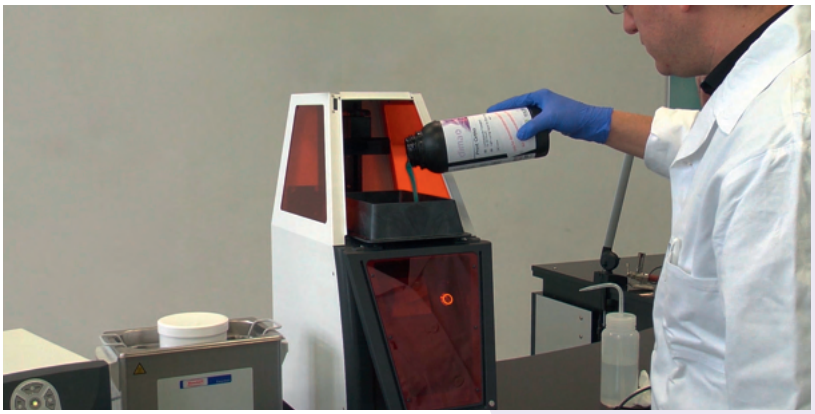
THE ENTIRE 3D PRINT PROCESS IS ALSO AVAILABLE AS A VIDEO.

kulzerUS.com/cara-print-process

Please always check how much resin you have in the tray:

Never use too much or too little of a monomeric liquid. Always fill the liquid up to the MAX line.

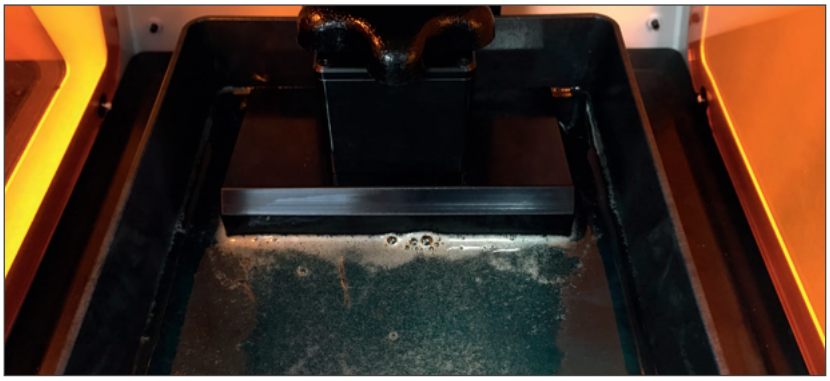
SHAKE THE DIMA PRINT MATERIAL VIGOROUSLY FOR 5 MINUTES BEFORE USE.



SAFETY WARNING:
ALWAYS USE PROTECTIVE NITRILE GLOVES, PROTECTIVE EYEWEAR AND APPROPRIATE PROTECTIVE CLOTHING WHEN WORKING WITH MONOMERIC LIQUIDS AND ISOPROPANOL. DO NOT INHALE THE VAPORS AND AVOID SKIN CONTACT.

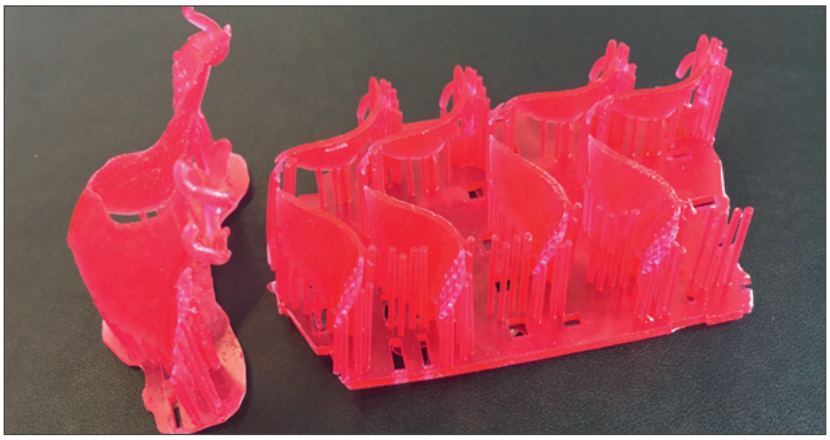
IF LEFT STANDING FOR LONG PERIODS OF TIME, MONOMERIC LIQUIDS CAN SEDIMENT IN THE BOTTLE AND IN THE RESIN TRAY. THIS CAN ADVERSELY AFFECT THE QUALITY OF PRINTS. IT IS NECESSARY TO ENSURE THAT MONOMERS ARE THOROUGHLY MIXED PRIOR TO EVERY PRINT.

Adding a sufficient amount of liquid will help to ensure that bubbles are pushed out of the way when the building platform sinks down to the required level.

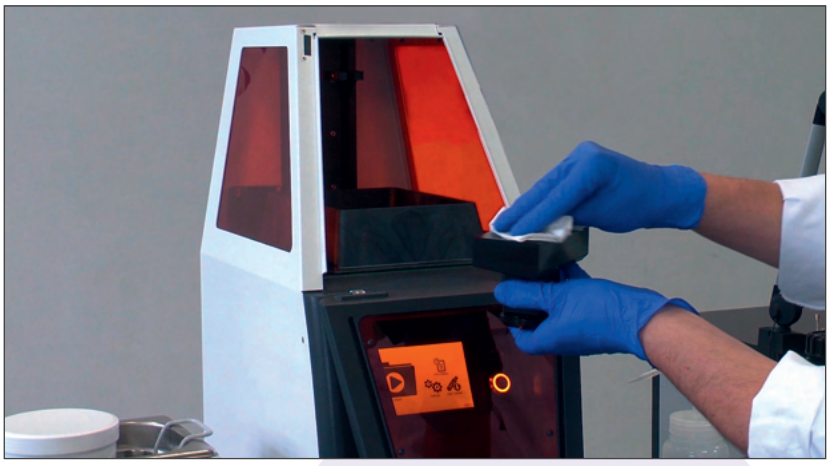


If the tray does not have enough liquid, the printed object will contain faults or the process will be interrupted, resulting in an incomplete print:

Left: desired object
Right: incomplete print



If there is too much liquid in the tray, the top of the building platform could be submerged in the liquid. Before starting a print, it is always necessary to check the building platform and the tray. Clean the building platform thoroughly using isopropanol to remove all monomeric residues. Furthermore, always check that the monomer used is homogeneous and, if necessary, stir it using the silicone spatula provided.



! **If any polymerized fragments are left in the tray or on the building platform from a previous print, this can damage the foil. This foil, which is located on the top of the projector window, can be easily scratched. It should be cleaned using the utmost care, and should only be touched in exceptional circumstances. It will no longer be possible to make perfect prints if this layer gets damaged, and the entire tray will have to be replaced.**

4.2. Changing print material and cleaning the print tray

The two resin trays provided with cara Print 4.0 can be filled and used with different materials. It is possible to store the light-curing material inside the tray on a temporary basis by closing it completely with the black silicone cover. If a resin has to sit for more than 12 hours, we recommend pouring it back into its original bottle using the supplied funnel and attached sieve.

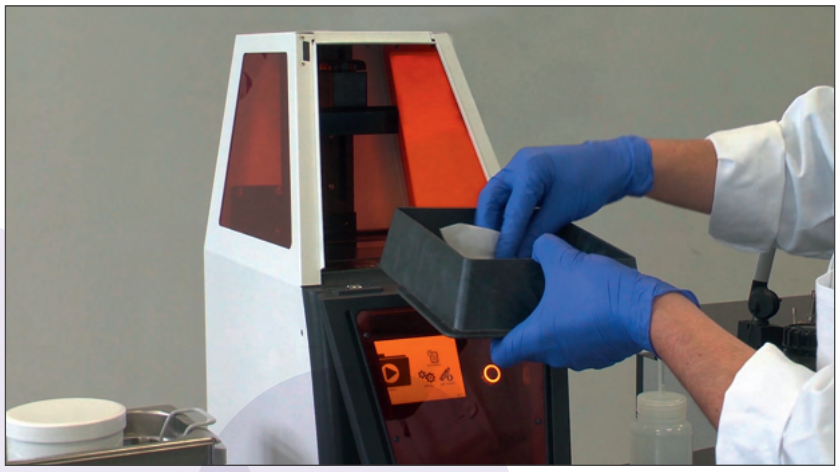
KEY PRINCIPLE: THE INSIDE OF THE VAT AND THE SURFACE OF THE ILLUMINATION WINDOW, WHICH IS HIGHLY SENSITIVE TO SCRATCHING, MUST ONLY BE TOUCHED USING THE PROVIDED SILICON SPATULA AND LINT-FREE, SOFT PAPER TOWELS SOAKED IN ISOPROPANOL, OR A CLEAN MICROFIBER CLOTH.

Recommended procedure:

1. Use one of the top corners of the vat to pour the remaining resin back into the appropriate dima Print bottle. Use the provided sieve in the funnel to filter the resin. Remove as much of the resin as possible using the silicone spatula.

It is very important to avoid leaving monomer residue from the previous material within the tray and on the projector window. Check both sides of the window carefully.

2. Remove the remaining resin from the vat using a lint-free, soft paper towel soaked in clean isopropanol. Repeat this step several times until the vat is completely free of resin. You may also use the microfiber cloth provided with cara Print 4.0 to clean the print tray window.

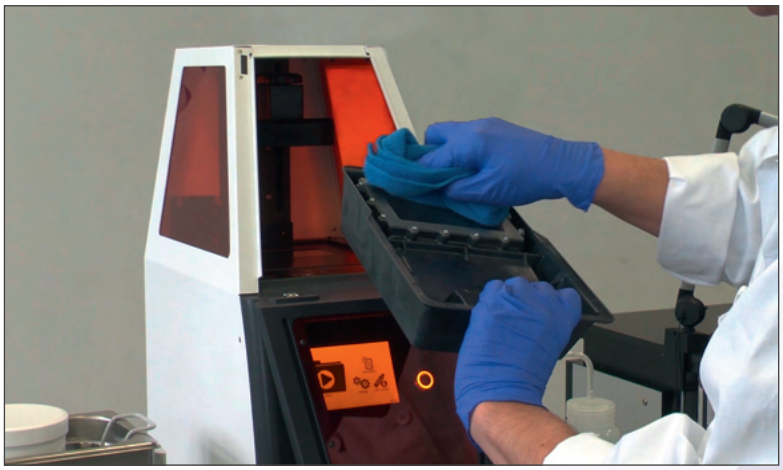


3. Next, lay the vat on a flat, even surface and wet the illumination window completely with isopropanol. Gently lift the print tray to drain the isopropanol over one of the upper corners. The illumination window should then be free of resin. If needed, blow off any remaining isopropanol using compressed air. Do not forget that the underside of the illummination window must also remain completely streak-free. Handle and clean it with equal care.

ATTENTION:
THE TOP AND BOTTOM SIDES OF THE PROJECTOR WINDOW ARE VERY EASY TO SCRATCH.

BE VERY CAREFUL IN SELECTING APPROPRIATE CLEANING MATERIALS.

You may also continue using a clean microfiber cloth if you prefer. The clean microfiber cloth should only be used after all the monomer residue is completely removed. The microfiber cloth is used to help ensure streak-free cleaning. We recommend holding the window up to a light to ensure that there are no streaks, smudges or residue. If you do not check this, the projected layers may get blocked or distorted, which would create faults in the printed object.

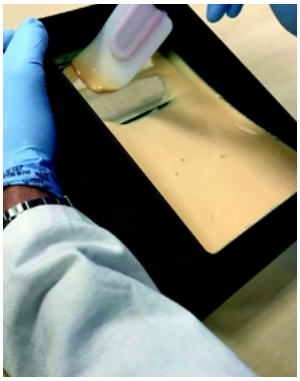


4.2.1. Intermittent review and cleaning of the illumination window

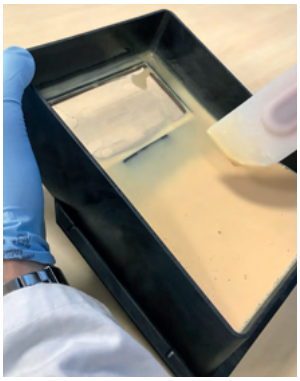
If you notice unexpected inconsistencies in your printed object, this might be the result of an unclean illumination window. Particularly when using highly pigmented resins, such as dima Print Stone (beige, gray or teal), dima Print Gingiva mask or dima Print Impression blue, we recommend checking and cleaning the illumination window after every print:

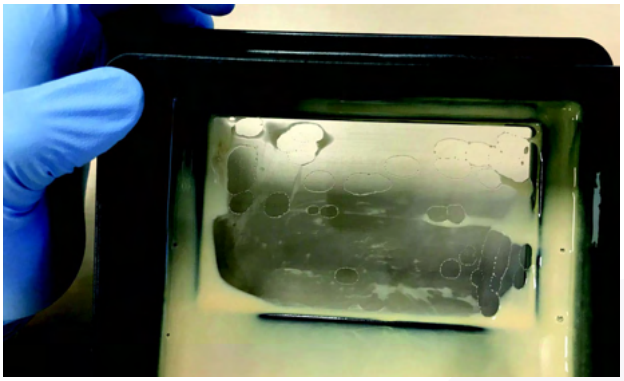
- Tilt the tray to clear the resin and have a clear view of the illumination window (Caution: don't spill resin over the edge.)
- Wipe the window carefully several times using the silicon spatula provided to remove any attached resin. Any remaining residue will drain from the window due to a pearling effect. Please wait a moment until the illumination window is fully visible so that you can check it for residue or damage.
- Return the tray to its normal position.
- Stir the remaining resin carefully with the silicone spatula for about five seconds.

If you noticed any residue attached to the illumination window or the film, please clean it as described in Chapter 4.2 (Cleaning the Print Tray). If the film is damaged, please call the Kulzer Hotline.



Carefully wipe away any remaining residue.





Pearling effect on the illumination window.

What should be done if printed objects are sticking to the illumination window?

Recommended procedure:

1. Drain the vat described in step 1 of the cleaning procedures (Page 18).
2. Fill the vat with enough isopropanol to cover the smudged illumination window.
3. Close the vat with its lid and let it sit for approx. 10 minutes. Drain the isopropanol over one of the upper corners.
4. Try to remove any stuck material with the silicone spatula or, if needed, compressed air. Protect your eyes!
5. Continue as described in Points 2 and 3 of the cleaning procedures.

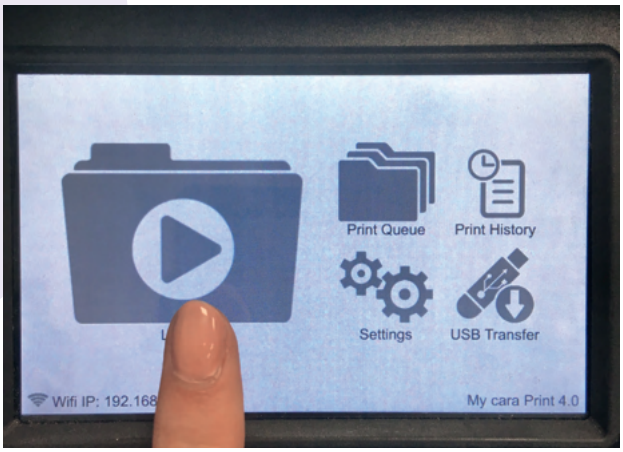
KEY PRINCIPLE: THE INSIDE OF THE VAT AND THE SURFACE OF THE ILLUMINATION WINDOW, WHICH IS HIGHLY SENSITIVE TO SCRATCHING, MUST ONLY BE TOUCHED USING THE PROVIDED SILICON SPATULA AND LINT-FREE, SOFT PAPER TOWELS SOAKED IN ISOPROPANOL, OR A CLEAN MICROFIBER CLOTH.

*The microfiber cloth must be clean and free of any hardened residual micronomer before using it on the print tray. Using a dirty cloth could scratch the window if hardened specs of cured resin are present.
*You may clean your microfiber cloths in a standard washing machine with warm water.
*When drying, please tumble dry the cloths on low or delicate setting and **DO NOT** use fabric softener or dryer sheets!

4.3. Storage space in cara Print 4.0 and deleting jobs

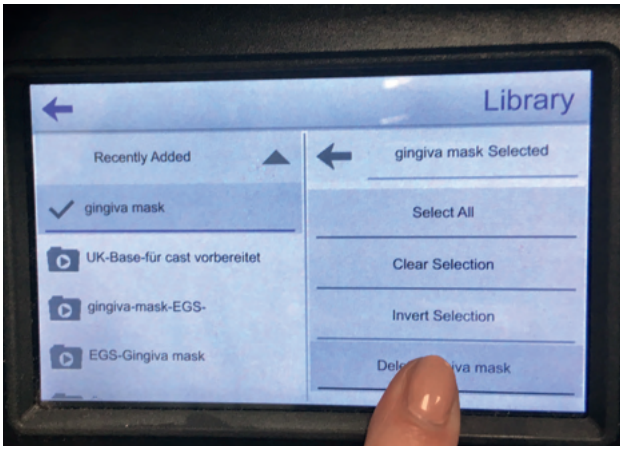
cara Print 4.0 has an internal storage capacity of round about 70 print jobs depending on their size. If the printer runs out of memory, you can free up additional space as follows:

1.

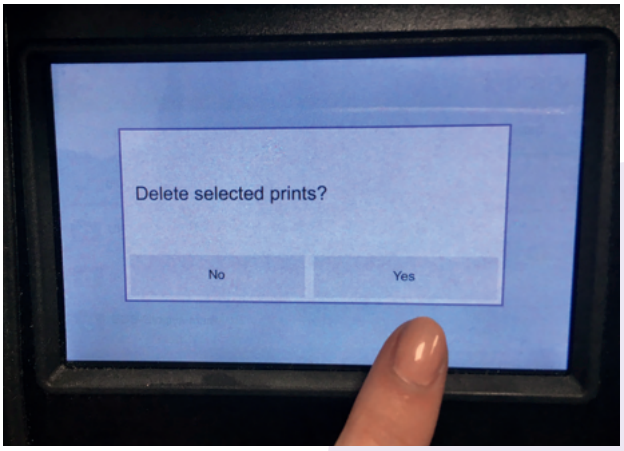


2.

3.



4.



Highlight the print jobs you want to delete with a long click on the touch display, then select “Delete”. Confirm by pressing “Yes”.

4.4. Starting a 3D print

Transfer your print file to cara Print 4.0 via Ethernet (recommended for best stability), USB stick or Wi-Fi.
See Section 3.2 Connecting to cara Print 4.0 for more details.

The following quality specifications (layer thicknesses) can be selected:

- 30 µm (high quality) up to 150 µm (low quality)

Your selection will depend on the material used and the dental appliance you intend to print.
Remember: as the layers get thinner, the accuracy increases but you have to deal with longer printing times.

(Example: approximate printing times for a splint made with dima Print Ortho)

- High quality = 19 min
- Low quality = 10 min

After selecting the desired quality,
tap “Print” to start the process.

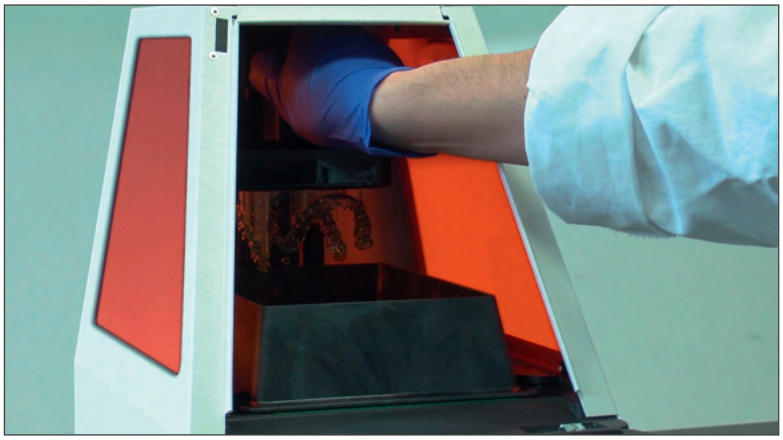
ATTENTION:
KEEP THE DOOR CLOSED DURING THE ENTIRE PRINT PROCESS.

4.5. What to do after printing

4.5.1. Removing objects from the platform

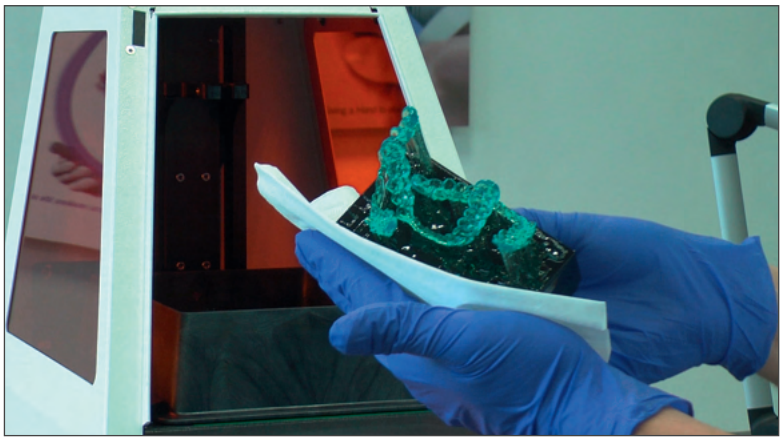
After printing, the build table will rise up just enough for the printed appliance to clear the print tray for removal.

Unscrew the top to detach the building platform. There is no need to completely remove the clamping knob.



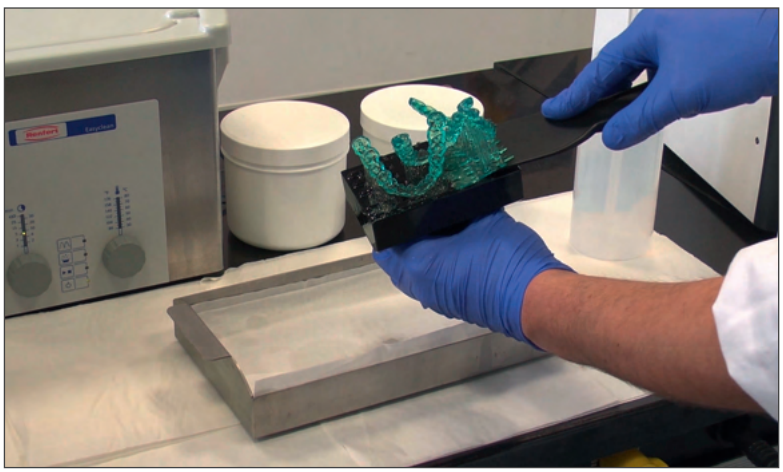
SAFETY WARNING:
ALWAYS USE PROTECTIVE NITRILE GLOVES, PROTECTIVE EYEWEAR AND APPROPRIATE PROTECTIVE CLOTHING WHEN WORKING WITH MONOMERIC LIQUIDS AND ISOPROPANOL. DO NOT INHALE THE VAPORS AND AVOID SKIN CONTACT.

Using the provided IPA spray bottle, rinse off the printed appliance with IPA to remove any excess print material prior to removing from the build table. We recommend using the supplied plastic spatula to remove objects from the building platform. This will help protect the black anodization on the aluminum build table or building platform.



Please take great care when removing objects from the building platform. Directly after printing, objects are still very soft, and the risk of damage is high. We recommend the following method: set the spatula flat against the surface of the building platform next to the base grid and slowly work it under the base bit by bit, until it is completely removed from the platform.

After this, you can clean the building platform and printed object with isopropanol. The building platform should be cleaned thoroughly leaving no residues (monomeric liquid, polymerized fragments, cleaning towel fragments, etc.)



4.5.2. Cleaning of printed objects cara Print Clean

We generally recommend using cara Print Clean to wash printed objects. For more information, see the instructions for the cleaning unit. Please also take notice of the information in the cara Print Clean Application Guide.

Be sure to complete two separate cleanings: First wash in Isopropanol for 3 mins, then wash in fresh Isopropanol for 2 minutes, either using a second cara Print Clean wash unit or a second wash unit container.

PLEASE TAKE NOTICE OF THE INFORMATION IN THE CARA PRINT CLEAN APPLICATION GUIDE:

kulzerUS.com/
cara-print-clean

You can still use an ultrasonic dental cleaner together with two separate containers for the isopropanol cleaning solution (one for pre-cleaning, one for post-cleaning). Avoid any mixing of cleaning solutions used for different materials (cross-contamination, medical devices).

Do not close the cleaning containers completely. Just lay the lids loosely on top. We recommend carrying out every step of the cleaning process using a good fume hood or ventilated cabinet.

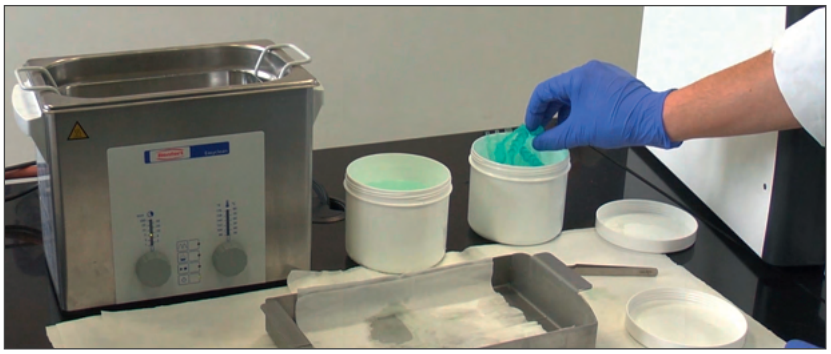
Please use 91%-99% IPA. 99% is preferable.

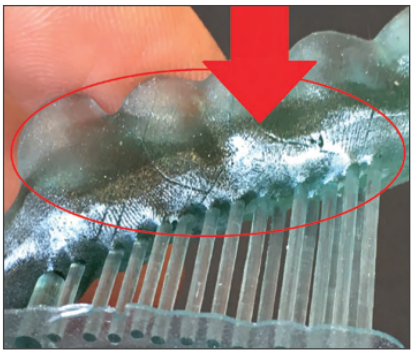
If the pieces contain long, thin indentations, the cleaning process can take longer.

1. Carefully blow the pieces with pressurized air.
2. 3 minute ultrasonic cleaning in container no. 1 (pre-cleaning).
3. Short drying/cleaning with pressurized air to remove any remaining monomer.
4. 2 minute ultrasonic cleaning in container no. 2 (post-cleaning in fresh isopropanol).
5. Turn the pieces several times within the container for the best cleaning results.
6. Dry with pressurized air.

Avoid letting the isopropanol get warm. You can avoid this by regularly replacing the water in the ultrasonic cleaner with cold water. If any residue remains on the object after cleaning, we recommend repeating the cleaning process from step 4 above, using fresh isopropanol. The pieces should not remain submerged within the isopropanol for longer than described above. If left in isopropanol for too long, the pieces could begin to dissolve and absorb isopropanol (see picture below).

PLEASE FOLLOW STANDARD SAFETY PRECAUTIONS WHEN USING, STORING AND DISPOSING OF ISOPROPANOL AND ISOPROPANOL BASED SOLUTIONS. IF YOU HAVE ANY QUESTIONS RELATED TO STORAGE AND DISPOSAL, CONTACT YOUR ISOPROPANOL SUPPLIER.





After cleaning, it is necessary to post-cure the printed pieces. Post-curing (post-exposure) is important, because it ensures the hardness and stability needed for dental applications and their biocompatibility.

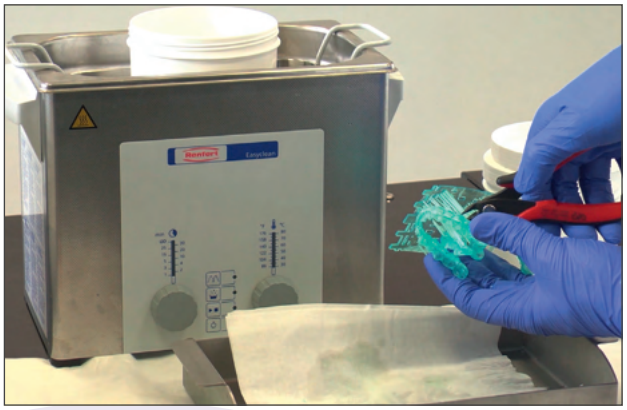
4.5.3. Post-curing printed objects

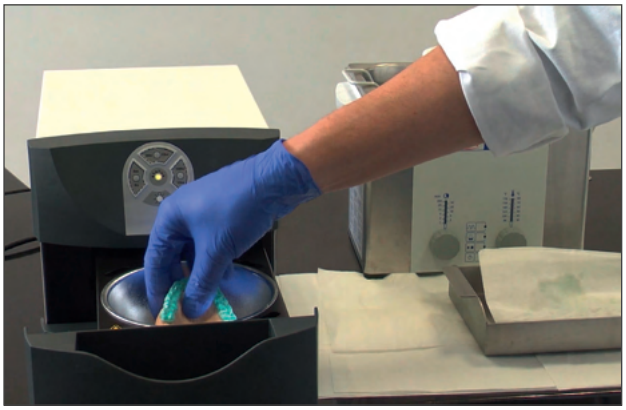
For post-curing, Kulzer recommends using the light-curing unit HiLite® Power 3D.

IMPORTANT: COOL DOWN BETWEEN CURING CYCLES

The HiLite Power 3D is designed for 60% operation time (flash on/off ratio) – but this is a technical value to be observed in printing sequences. Because the HiLite Power 3D is not designed for continuous operation, it needs some time to cool down after each cure. We recommend a cool-down time of e.g. 3 minutes after 2 x 15 minute cycles or 2 minutes after 2 x 10 minute operational cycles. Cooling time recommendations are also noted inside the HiLite Power 3D IFU.

Before you post-cure your objects, the support structures must be removed. We recommend using the flush-cut clipper that is part of your accessory kit.





SAFETY WARNING:
ALWAYS USE PROTECTIVE GLOVES, EYEWEAR AND APPROPRIATE PROTECTIVE CLOTHING.

3.1 Working Rules

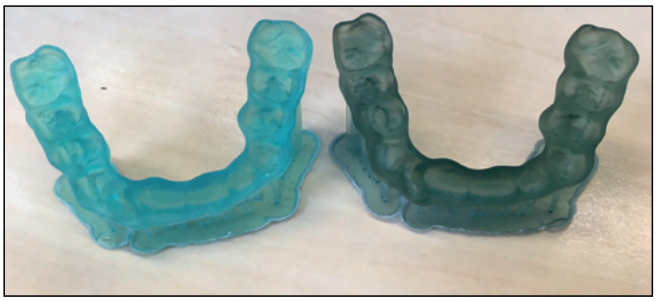
CAUTION
Remove any jewellery and put on the necessary personal safety clothing, e.g. hand, face, and body protective clothing.

CAUTION
Do not use the machine for the jobs listed below:

- Do not use the machine to warm food.
- Do not operate the machine without first inserting a filter, and do not place objects on the filter disc.
- Do not use the machine with a heavily-soiled light reflector and/or if the bottom holes are closed- otherwise, the heat will not be dissipated.

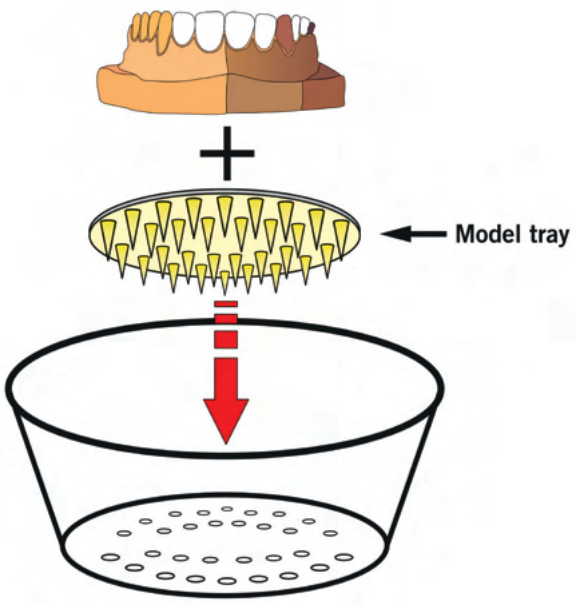
* We recommend that you do not use the machine for 30 seconds to 60 seconds between each polymerisation operation of 90 seconds or 180 seconds, for example, so that the machine may cool down (ventilator running).

Changes in color are normal. Here is an example of dima Print Ortho. (Left = before post-curing; Right = after post-curing)



In order to avoid warping and to ensure a good fit, we recommend post-curing objects on the original plaster model. For exact post-curing times and other important details, please see the individual guides or the respective section of this document covering the desired dental application.

We recommend using the “model tray”. This allows for the effective utilization of the full volume of the post-curing unit and for good ventilation. Carefully check the total height to ensure that the door can close securely.



Post Cure Times:

Material	Total Post Curing Time	Procedure
dima Print Ortho	10 min	Turn over after 5 min
dima Print Impression blue	10 min	Turn over after 5 min
dima Print Guide	10 min	Turn over after 5 min
dima Print Cast	10 min	Turn over after 5 min
dima Print Cast ruby	3 min	Turn over after 1.5 min
dima Print Gingiva Mask	10 min	Turn over after 5 min
dima Print Stone* beige, gray, teal	10 min	Turn over after 5 min
dima Print Model*	6 min	Turn over after 3 min
dima Print Splint Clear	6 min	Turn over after 3 min
dima Print Denture Base Try-in**	20 min	Turn over after 10 min
dima Print Denture Base**	20 min	Turn over after 10 min
dima Print Denture Teeth**	20 min	Turn over after 10 min
dima Print C&B Temp**	20 min	Turn over after 10 min

* Dies and Models must be separated during post-curing.

Recommended 4 90-second cures. Turn over after 180 Seconds.

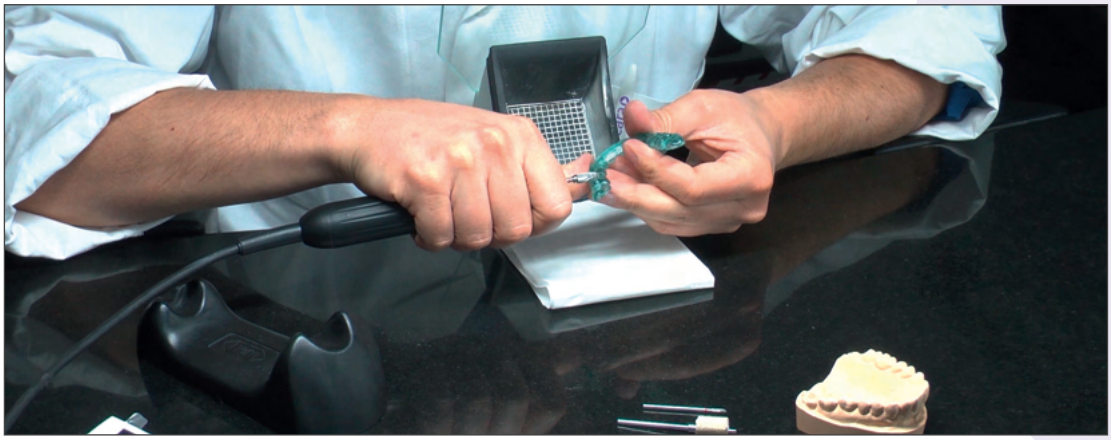
** Prior to post-curing dima Print Denture Base and dima Print Denture Teeth, see post-curing and finishing steps listed at the end of Sections 5.8 and 5.9.

** Prior to post-curing dima Print C&B Temp, see post-curing and finishing steps listed in Section 5.11.

4.5.4. Post-process for printed and post-cured objects

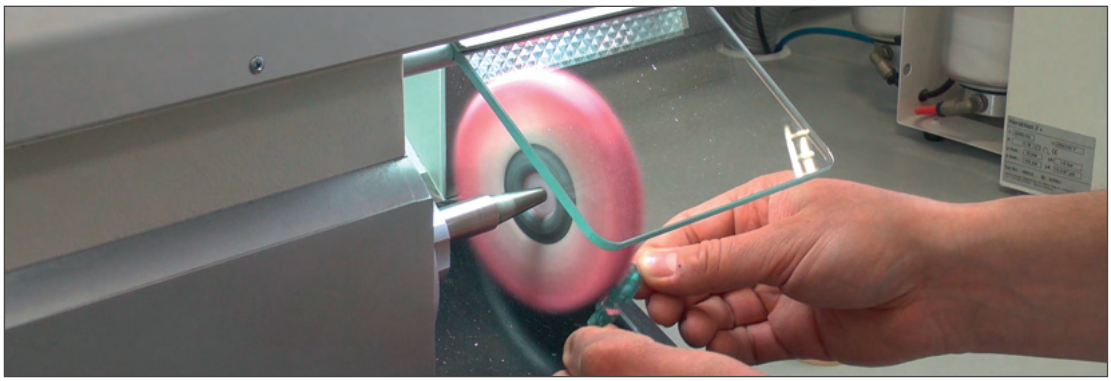
SAFETY WARNING:
ALWAYS USE PROTECTIVE GLOVES, EYEWEAR AND APPROPRIATE PROTECTIVE CLOTHING. TAKE PRE-CAUTIONS TO AVOID INHALING POLYMER DUST

Any sharp edges that remain after removing the support structures must be filed down. Please use standard cross-cut hard metal cutters.



Polishing:

Surgical drilling guides made of dima Print Guide and (bite) splints made of dima Print Ortho or dima Print Splint clear require additional polishing after the supports are removed. We recommend preparing the pieces with fine sandpaper before polishing. They can be polished using either a handheld or a benchtop polishing lathe. Recommendation: for pre-polishing use a pumice solution; for high-gloss polishing use a high-gloss polishing paste.



4.5.5. Cleaning cara Print 4.0

We recommend 91% - 99% isopropanol for cleaning both the 3D printer and the printed pieces.

SAFETY WARNING:
ALWAYS USE PROTECTIVE NITRILE GLOVES, PROTECTIVE EYEWEAR AND APPROPRIATE PROTECTIVE CLOTHING WHEN WORKING WITH MONOMERIC LIQUIDS AND ISOPROPANOL. DO NOT INHALE THE VAPORS AND AVOID SKIN CONTACT.

When changing the resin, it is extremely important to ensure complete, residue-free cleaning of the tray. The monomeric liquids must not be mixed. This would have a negative impact on final material properties and biocompatibility, and could result in faulty prints. We recommend using the provided silicone spatula and lint-free, soft paper towels soaked in isopropanol. Remove the tray from the printer to facilitate cleaning.

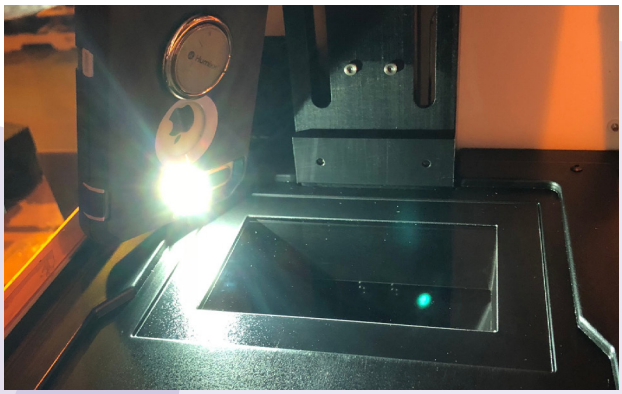
Take extra care to ensure that the tray window is absolutely clean and streak-free on both sides.
This helps to ensure error-free illumination when printing.

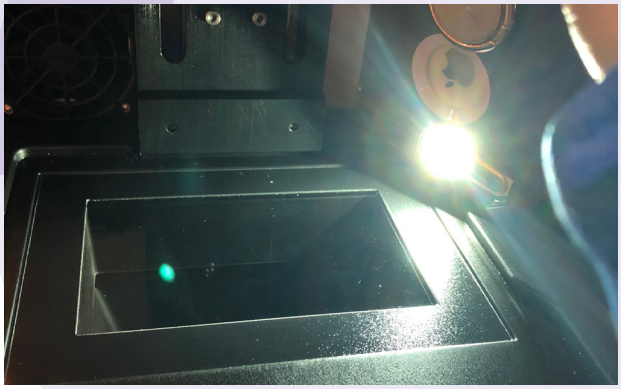
*Always inspect the projector window inside Cara Print 4.0 prior to printing. To determine glass cleanliness, shine a flashlight at the back of the optical glass as shown below. Shine the light across the surface of the glass from the back side of the glass. Examine the entire window. Very small smears or haze can significantly impact print quality.

If any liquid has been spilled on the glass of the projector window inside cara Print 4.0, this should also be cleaned, leaving no residue or streaks.

Placement of light source.

Shine the light from the back of the printer facing front, on both angles of the glass window.





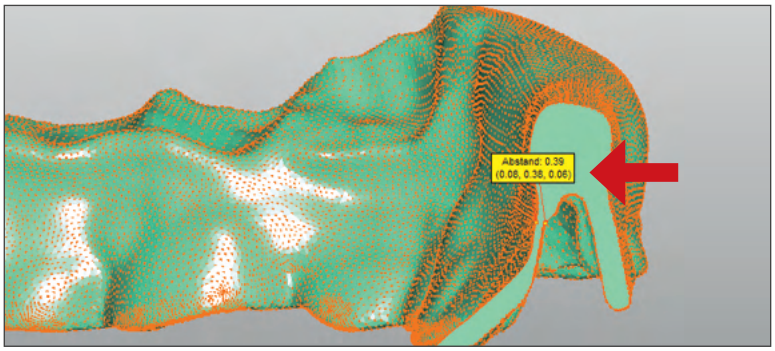
5. Indication-specific information and tips

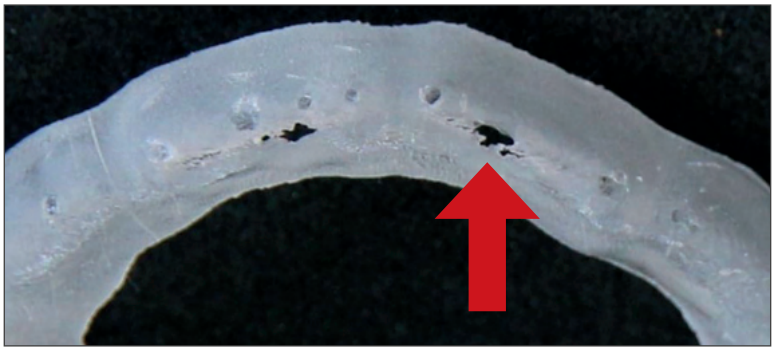
5.1. dima Print Splint clear

Design notes:

Recommended settings:

- Minimum thickness: 1.3 mm. If portions of the splint are thinner than 1.3 mm, dimensional stability cannot be guaranteed, and holes may appear as a result.
- Offset (e.g. 3shape SplintDesigner™): 0.00 mm
- Drill compensation (drill radius setting): 0.6 mm



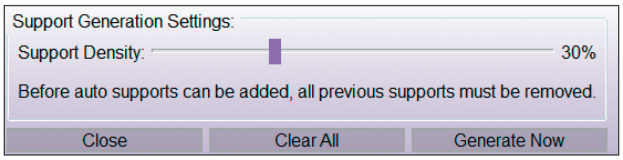


Support design notes:

The supports must be positioned with sufficient distance between one another:

- Spacing not less than 0.5 mm
- Support density: 20–40 % (depending on size of splint)

This helps to avoid the retention of monomeric liquid between the supports, which would make cleaning more difficult.

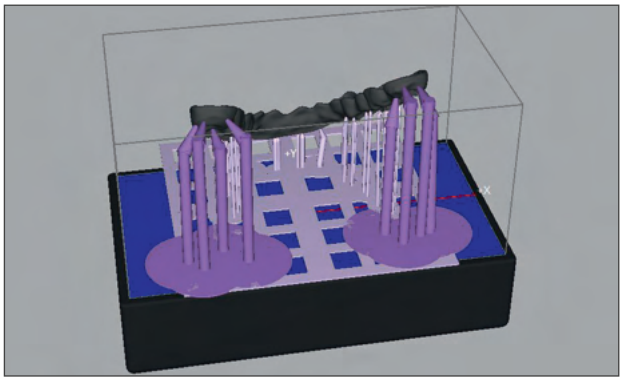


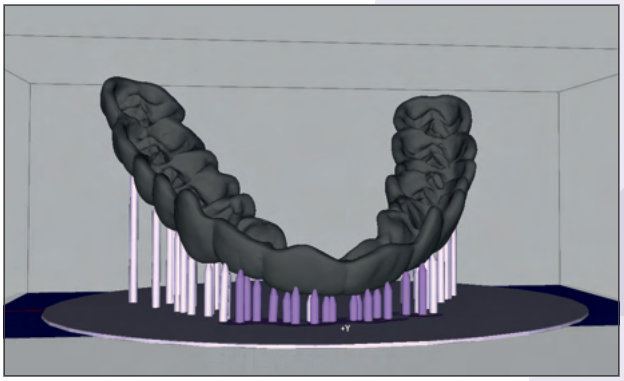
Angle (print positioning):

- 45° recommended

Supports:

- Center of the support down to the grid base: 0.5 mm to 1.0 mm (radius); Form: “cylinder”
- Form of the tip on the printed piece: “Cone 25 %” (for easier removal after printing)





- Front area: type “Light”
- Front area: please add more supports manually (three per tooth) after using the autosupport function
- Back area: type “Medium”

Foundation:

- “Hash” or peel foundation
- Thickness: 1.5 mm
- Distance of splint to building platform: 10 to 15 mm

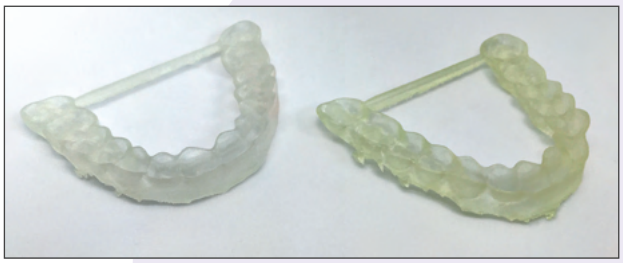
Coloring

3D-printed, transparent objects generally have a slight yellowish hue. The ultimate visual impression of printed and post-cured splints made with dima Print clear depends on the ambient light (artificial lighting vs. sunlight, etc.), as well as on the thickness of the material in the finished splint.

Post-curing recommendations for dima Print Splint clear

Before post-curing splints, make sure that the filter glass is in place within the HiLite Power 3D. Please do not use the device without a filter glass. For more information, see chapter 7.5 of the Operating Instructions for HiLite Power 3D.

Post-curing without the filter glass will result in discoloration.



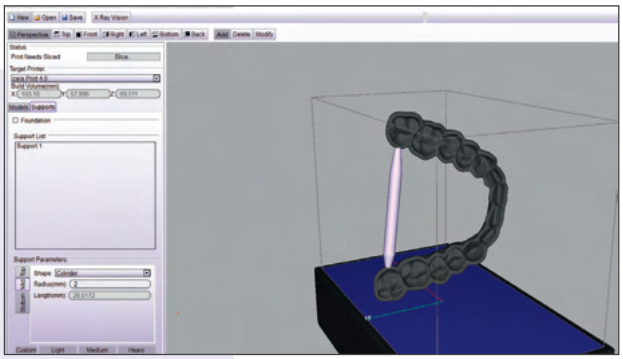
Unpolished dima Print Splint clear splints after post-curing
(left **with** filter glass, right **without** filter glass)

dima Print Splint clear splints after post-curing and polishing
(left **with** filter glass, right **without** filter glass)

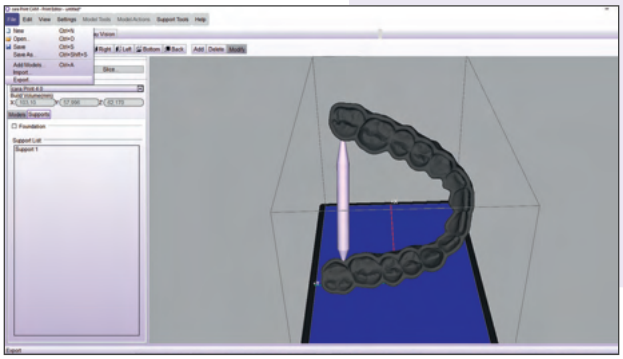
Post-curing without a model

If you don't have a plaster model when making a splint (i.e. 100 % digital workflow), it is still possible to ensure a perfect fit after the splint is post-cured. The only way to do this, however, is by integrating specific elements to stabilize the structure of the splint. You will need to use cara Print CAM to integrate horizontal braces at the end molars.

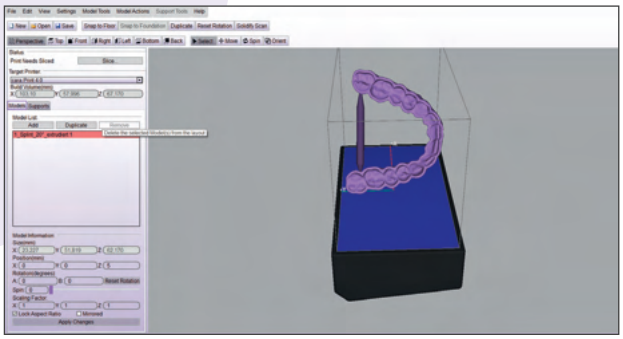
1. Orient the Splint in a vertical direction. Add a support with diameter "heavy" on the back part of the splint between the quadrants.



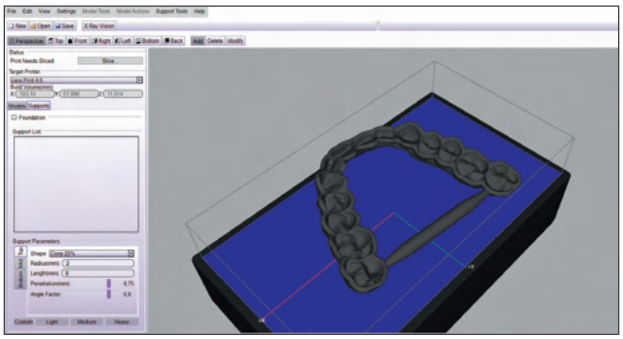
2. After support is placed, click on model tab and select to highlight appliance on build table.



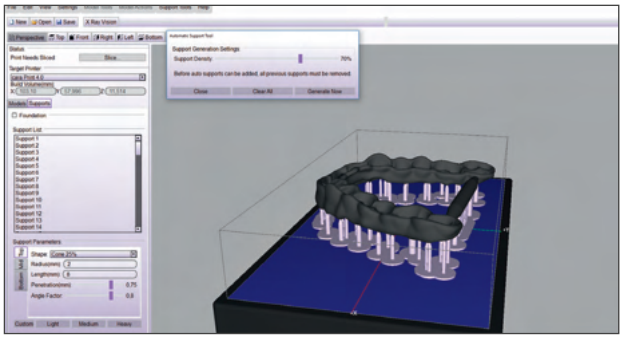
3. Export the edited stl file on your desktop or on a another storage device.



4. Create new layout and discard the old project. Then reimport the edited stl file into your cara Print CAM Software.



5. Make sure to put enough supports on the surface (see chapter splints, page 30) and print it.

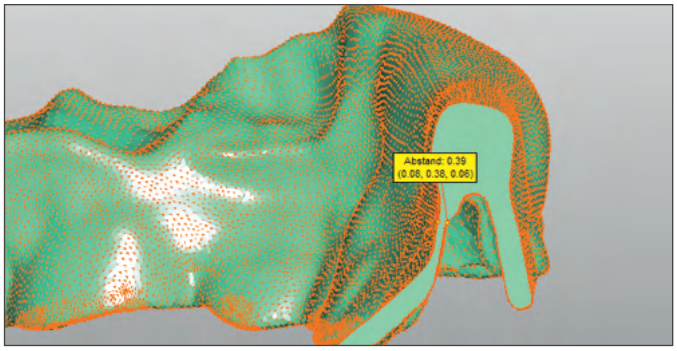


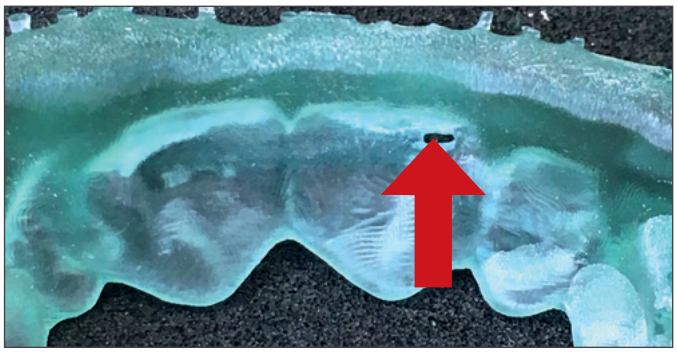
The length of the braces in the STL file must be adjusted manually to fit each splint.
To do this, change the "z" scale variable until a connection is made that does not protrude out of the other side. Afterwards, you can position the recommended support structures as usual.

5.2 dima print Ortho

Recommended settings:

- Minimum thickness: 1.5 mm. If the walls of a splint are thinner than 1.5 mm, dimensional stability cannot be guaranteed and holes may appear as a result.
- Offset (e.g. 3shape SplintDesigner™): 0.03 mm
- Drill compensation (drill radius setting): 0.6 mm





Angle (print positioning/orientation):

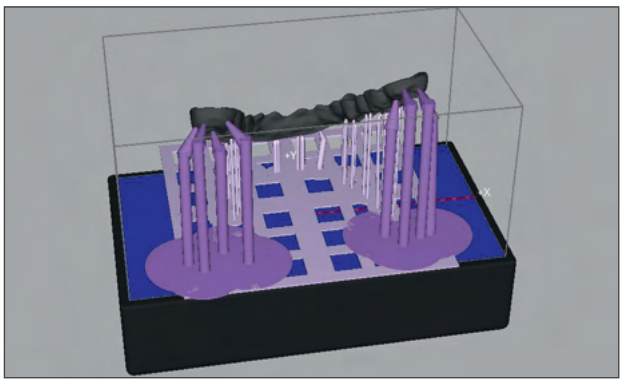
- 45° maximum
- After your orientation is set, highlight the appliance and hit the “Snap to Floor” button at the top of your CAM screen. This will snap the appliance to the floor of the build table.

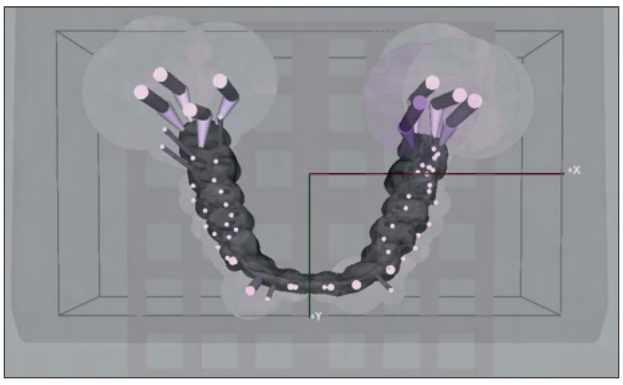
Supports:

The supports must be positioned with sufficient distance between one another:

- Spacing not less than 0.5 mm.
- Support Density: 20–40 % (depending on size of splint).

This helps to avoid the retention of monomeric liquid between the supports, which would make cleaning more difficult.





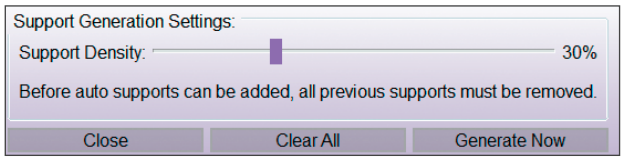
- Front area: type “Light”
- Back area: type “Medium”

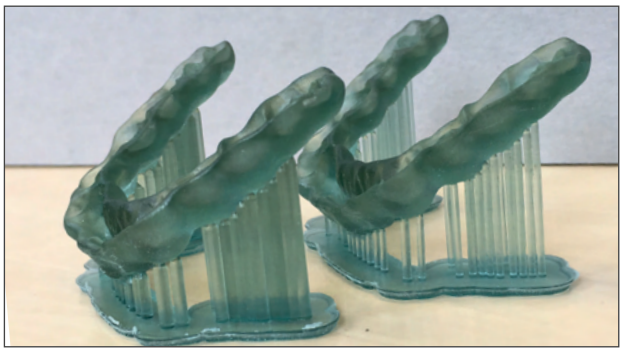
Support design notes:

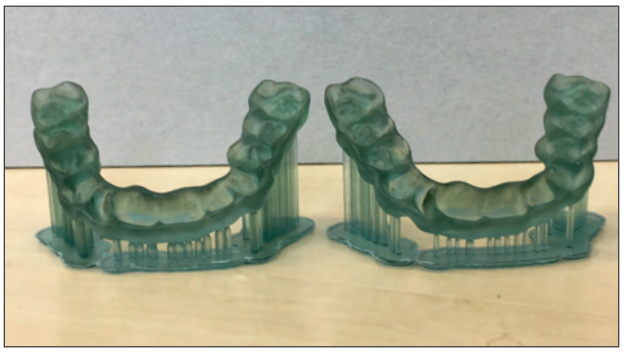
The supports must be positioned with sufficient distance between one another:

- Spacing not less than 0.5 mm.
- Support Density: 20–40 % (depending on size of splint).

This helps to avoid the retention of monomeric liquid between the supports, which would make cleaning more difficult.







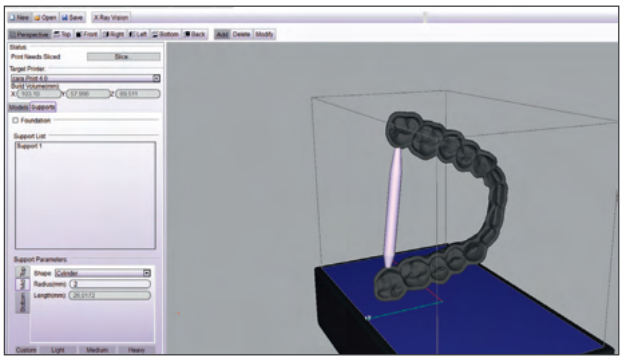
Foundation:

- “Hash” or peel-off
- Thickness: 1.5 mm
- Distance from splint to building platform: 10 to 15 mm
- If adding a foundation, once it’s set, please hit the “Snap to Foundation” button at the top of your CAM screen. This will snap the appliance to the foundation on the build table.

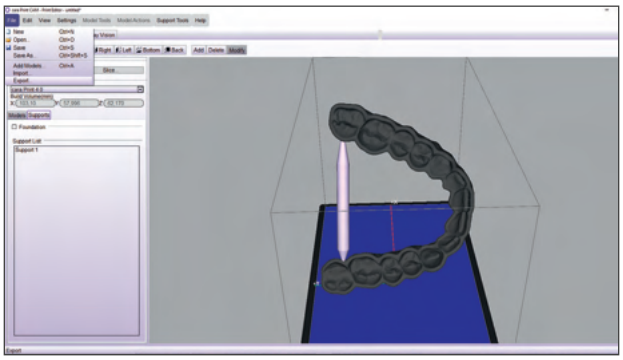
Post-curing without a model:

If you don't have a plaster model when making a splint (i.e. 100 % digital work flow), it is still possible to ensure a perfect fit after the splint is post-cured. The only way to do this, however, is by integrating specific elements to stabilize the structure of the splint. You will need to use cara Print CAM to integrate horizontal braces at the end molars.

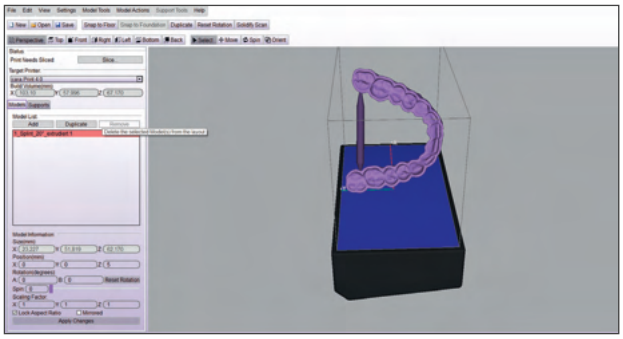
1. Orient the Splint in a horizontal direction. Add a support with diameter "heavy" on the back part of the splint between the quadrants.



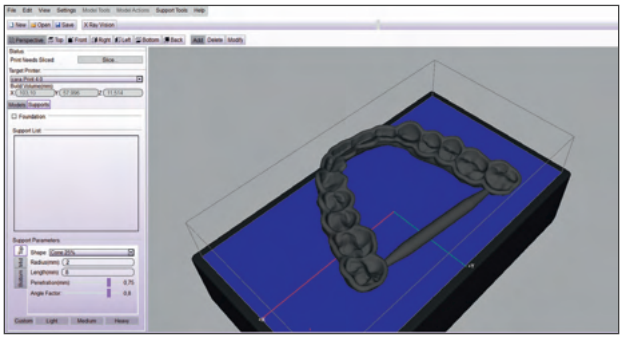
2. After support is placed, click on model tab and select to highlight appliance on build table.



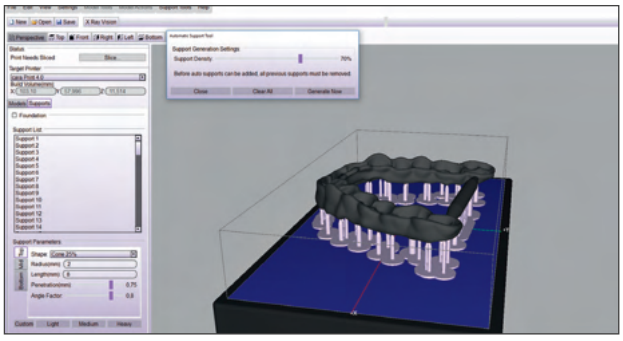
3. Go to file/export and rename supported appliance. Then save it as a new STL file.



4. Delete the old file from the build table and then re-import the new supported STL to your cara CAM.



5. Add all remaining supports, slice, and print your appliance.



The length of the braces in the STL file must be adjusted manually to fit each splint.
To do this, change the "z" scale variable until a connection is made that does not protrude out of the other side. Afterwards, you can position the recommended support structures as usual.

5.3. dima Print Impression blue

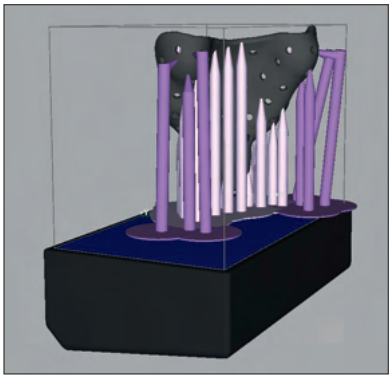
Design notes:

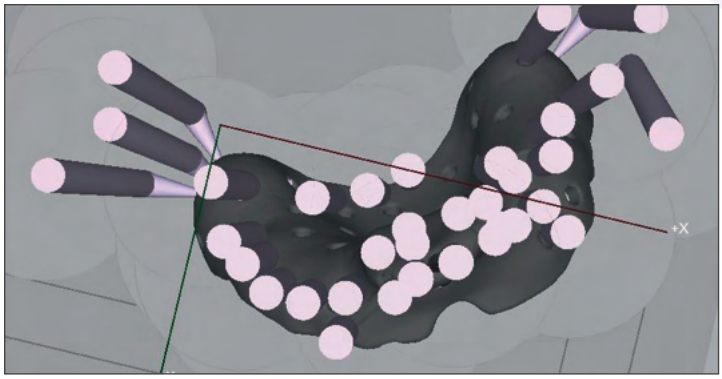
Recommended settings:

- Minimum thickness: 2.0 mm. If the walls of the impression tray are thinner than 2.0 mm, dimensional stability cannot be guaranteed.
- When designing impression tray handles, it is important to ensure that they are strong enough to withstand the force needed to remove the completed impression from a patient's mouth. Some CAD systems offer libraries with pre-designed handles. If you design the handle independently, you may run the risk of making it too thin.
- Retention holes should be no larger than 1.5 mm in diameter.

Supports:

- Spacing not less than 0.5 mm
- Density of supports: 10–20 %, adjust manually if needed
- Type for all supports: "Heavy"

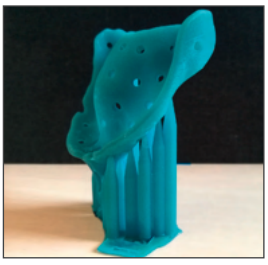


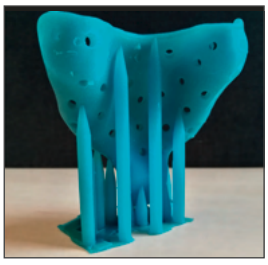


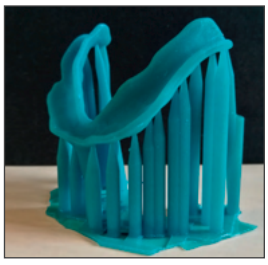
- Form of the tip on the printed piece: "Cone 25 %" (for easier removal after printing)

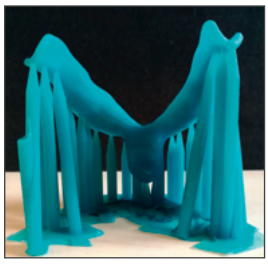
Angle (print positioning/orientation):

- 75°–80° maximum
- After your orientation is set, highlight the appliance and hit the "Snap to Floor" button at the top of your CAM screen. This will snap the appliance to the floor of the build table.
- Due to the viscosity and density of dima Print Impression blue, the support structures must be designed to be especially stable.









Foundation:

- "Hash" or peel-off
- Thickness: 1.5 m
- Distance from tray to building platform: 10 – 15 mm
- If adding a foundation, once it's set, hit the "Snap to Foundation" button at the top of your CAM screen. This will snap the appliance to the foundation on the build table.

Special notes:

- Due to the strong pigmentation of the material, it is especially important to ensure that all residue is removed from the print tray after cleaning – particularly from the critically important projector window.
- If the material in the print tray is not used for a while, mix it very well before each print with the silicone spatula. If it needs to be stored for longer periods (>12 hours), pour it back into the original bottle.

5.4. dima Print Guide

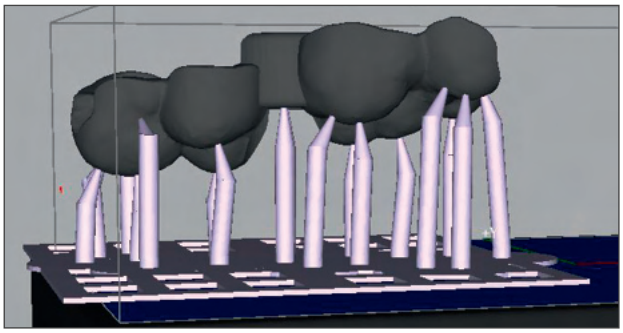
Design notes:

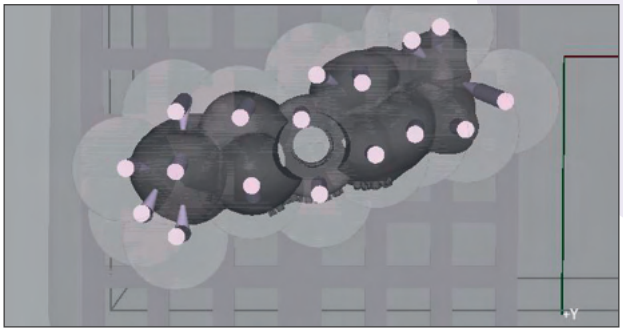
Recommended settings:

- Minimum thickness: 2.0 mm. If the walls of a surgical drilling guide are thinner than 2.0 mm, dimensional stability cannot be guaranteed and holes may appear as a result.
- Settings (based on 3shape Surgical Guide):
  - Thickness: 2.0 mm
  - Offset from teeth: 0.075 mm
  - Offset from sleeve: 0.04 mm
  - Retention amount: 0.025 mm

Supports:

- Spacing not less than 0.5 mm.
- Density of supports: 20–30 % (supports can be positioned manually for smaller objects, but always on the highest points of the object).
- Center down to the grid base: type “Medium”.
- Form of the tip on the printed piece: “Cone 25%” (for easier removal after printing).





Foundation:

- “Hash” or peel-off
- Thickness: 1.5 mm
- Distance from drill guide to building platform: 10–15 mm
- If adding a foundation, once it’s set, hit the “Snap to Foundation” button at the top of your CAM screen. This will snap the appliance to the foundation on the build table.

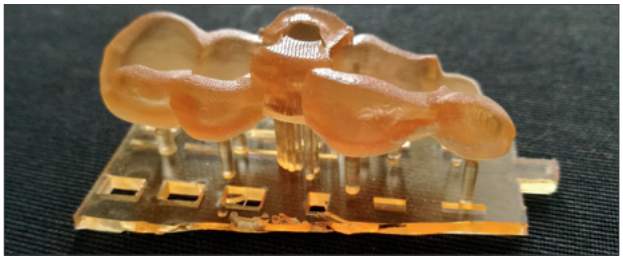
Angle (print positioning/orientation):

- The best results are obtained when printed flat (0° or not angled, respectively).
- Exceptions can be made for larger objects, which should not be angled by more than 45°.
- After your orientation is set, highlight the appliance and hit the “Snap to Floor” button at the top of your CAM screen. This will snap the appliance to the floor of the build table.

Usage notes for cara Print CAM:

How should surgical drilling guides be positioned within the virtual print space?

- Always position the drilling guide hole along the z-axis.



3D-printed and cleaned drilling guide before removing supports and before placing the drilling sleeve.

Integration of the guide sleeve:

How do I insert the guide sleeve securely into the surgical drilling guide?

- The sleeve is inserted after cleaning the printed drill guide and removing the supports but before the entire guide is placed into HiLite Power 3D for post-curing. The shrinking associated with polymerization will ensure the sleeve is fixed solid and securely within the drill guide.

Color change due to post-curing and autoclaving

Left: post-cured and polished drill guide; Right: After autoclaving

The finished surgical drilling guide with the guide sleeve (fixed via final polymerization/post-curing) must always be fully polished (high gloss) before autoclaving. Skipping this polishing step can lead to micro tearing as a result of the autoclaving process. The drilling template may only be sterilized in standard-issue sterilization bags. This also prevents the occurrence of micro-fissures.

Recommended sterilization method:

- Please make sure that the surgical guide is fully post-cured and polished before sterilization! This is important with regard to form stability.
- Polish the guide to high gloss (see chapter “4.4.4 Post-process for printed and post-cured objects”).
- Place the polished surgical guide in a standard sterilization pouch.
- Apply sterilization by placing the pouch containing the guide in an autoclave for 15 minutes at 121°C or 3 minutes at 138°C. Make sure no mechanical forces are applied to the guide during sterilization.
- Let the guide cool down to room temperature before using the guide. Make sure no mechanical forces are applied to the guide during cooling down.

5.5. dima Print Cast ruby

The CAD-to-Cast material dima Print Cast ruby is suitable for the production of various indications using metal alloys and press ceramics. The printed objects deliver optimal fits on their respective anatomic models and ensure equally perfect fits of the final restoration with smooth surfaces, whether they are made using the cast or press method.

5.5.1. Partial Dentures

Design notes

- The pre-designed standard wax profiles available in the 3Shape software cannot be used due to insufficient thickness! Please use the provided dima Print profile. Please manually adjust the wall thickness to match the values below for clasps, occlusal rests, retention wires, small connectors, large connectors and so-called “Ney clasps”, or use our DME file that can be found at **kulzer.com/cara-print-3shape-dme.**
- CAD-to-Cast models should generally be printed lying flat to ensure optimal fit.
- Clasps and frames should be provided a sufficient amount of supports, particularly the tips of braces.
- We recommend round supports.

Minimum material thickness	
Clasps	1.0 mm
Occlusal rests	0.6 mm
Retention mesh, overlaying reinforcement wire	0.6 mm 1.0 mm
Small connector	1.5 mm
“Ney clasps”	0.7 mm

Minimum material thickness	
Palate plate (large connector) terminating at	1.0 mm
	0.3 mm
Sublingual bar	1.5 mm
horizontal reinforcement bars, round profile	1.0 mm

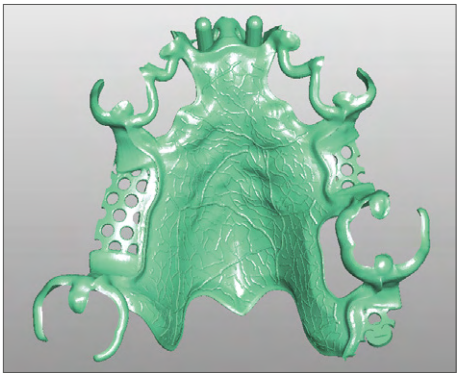
Reinforcement bars

To ensure that the printed object fits perfectly, please use your CAD or cara Print CAM to add at least one horizontal reinforcement bar. This bar stabilizes the printed partial framework to prevent distortion during light curing or manual shaping. The horizontal reinforcement bar should be used especially for delicate cast structures with a thin palatal plate (1 mm) and for structures with a smaller expansion of the base plate (transverse connection). The reinforcing bar can also be left on the object as a stabilizing element during investment.

When should a stabilizing bar be used and when not necessary?

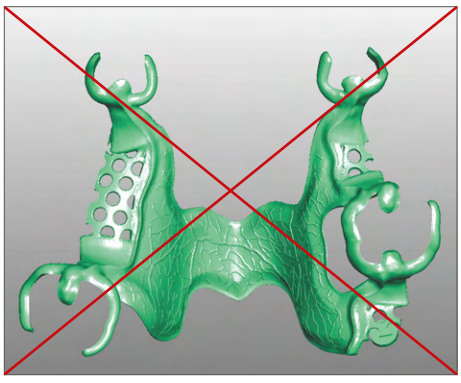
- large connection with extensive expansion, covering almost the entire palate
- thickness > 1.0 mm

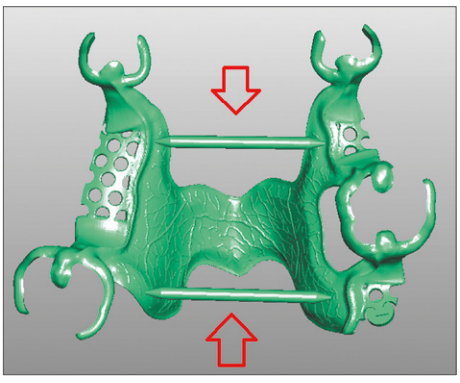
→ **No** stabilizing bar is needed.



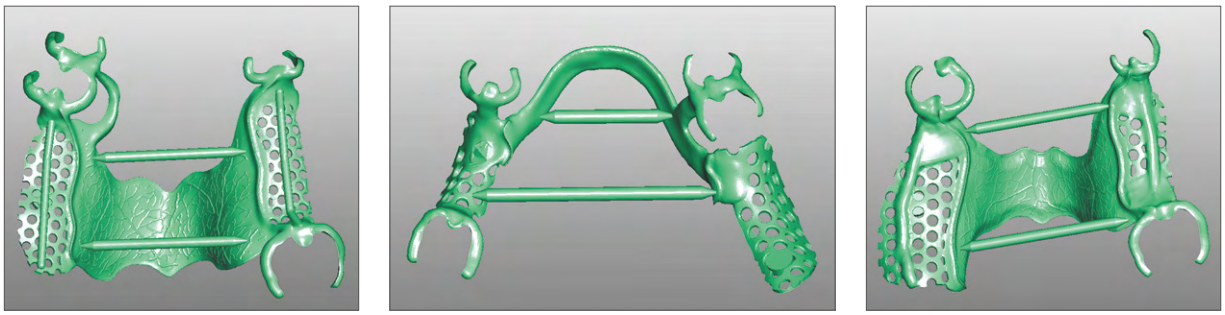
- thin, fragile transverse connection
- base plate covers only a part of the palate
- approx. 1.0 mm thick

→ We recommend to at least **one** bar. The more fragile the design, the more bars are needed.





The position of the brace is based on the respective design of the partial denture framework.
The following examples show a possible way to attach it in three different designs.



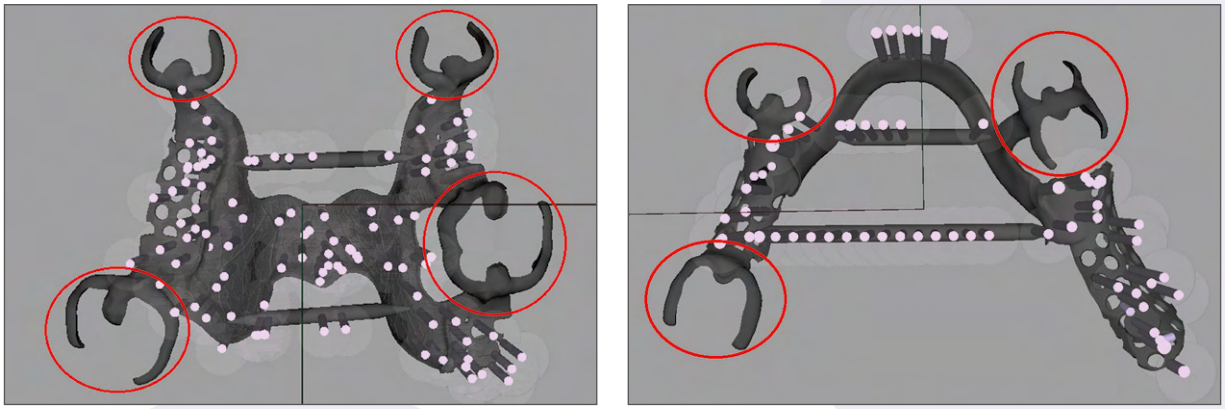
Distance and density of support structures

- Spacing of approx. 0.5 mm minimum
- Density of the supporting structures: upper jaw 20–30 %, lower jaw 60–70 %

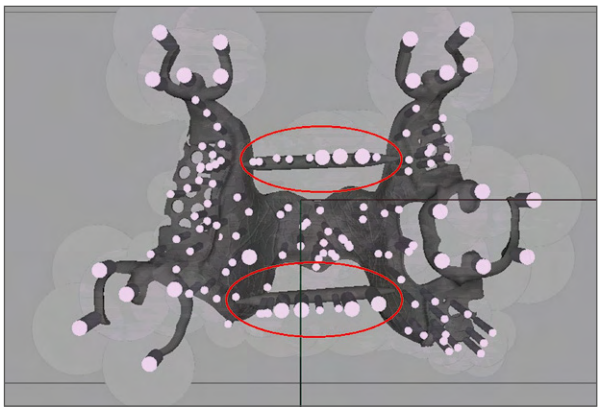
Thickness of support structures

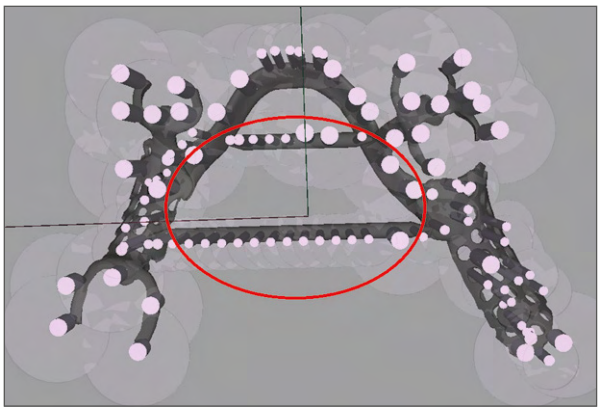
- Center down to the grid base: “fine”
- On clasps: “normal”
- On reinforcement bars: “normal”

For small, fine portions of a cast structure (especially clasps), the automatically generated fine supports should be deleted and manually replaced with thicker ones (normal size).

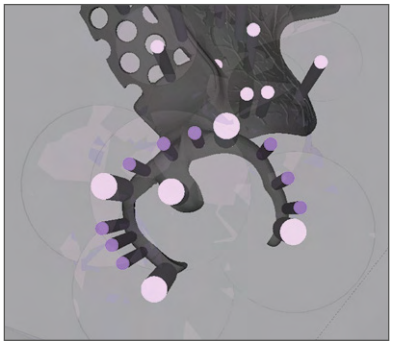


Always place the new and thicker supports on fragile parts of the clasp (occlusal support, clasp tip, connection pieces from clasp to clasp, lingual/palatal arm of the clasp):





Five thicker support structures per clasp are sufficient. Make sure that none of the placed supports penetrate the cast framework. For each occlusal support, we recommend one support structure (normal size) placed centrally on the upper side of the support (left picture).

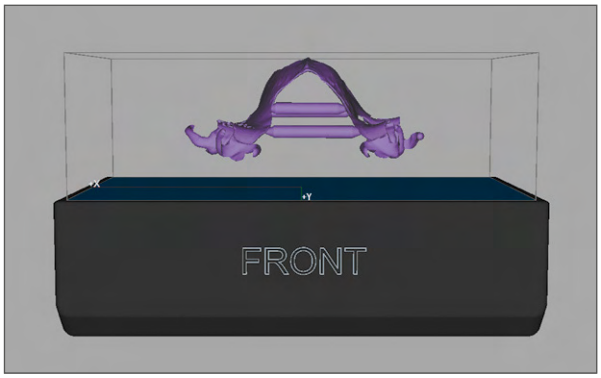


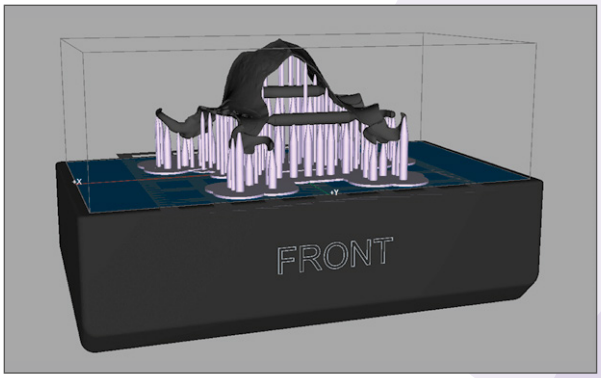
Areas on the clasps that do not yet have supports should be filled in with thinner (“Fine”) supports (image right).

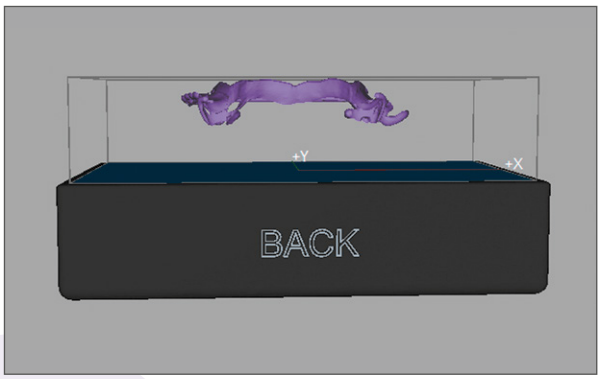
Tips of the supports

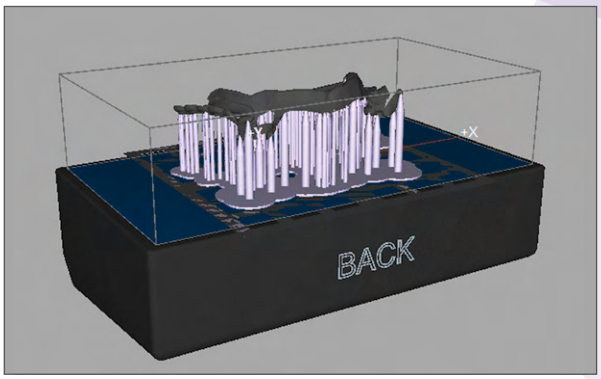
- 0.3 mm (form: “cone 25%” for better separation from the object later)

Alignment of objects in the cara Print CAM



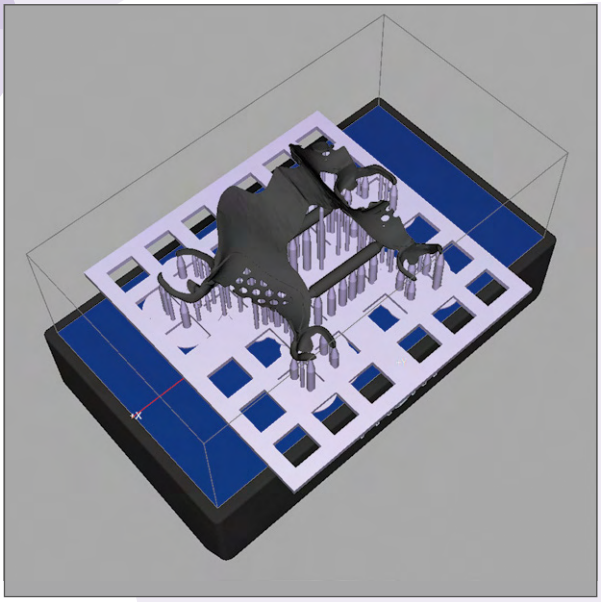




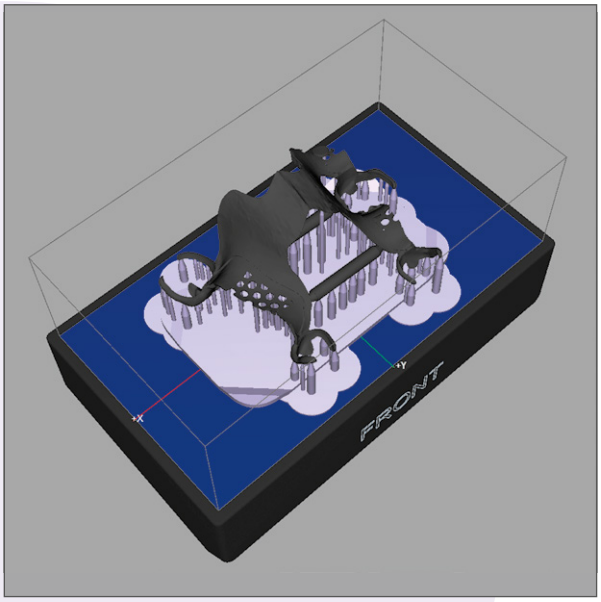


Foundation

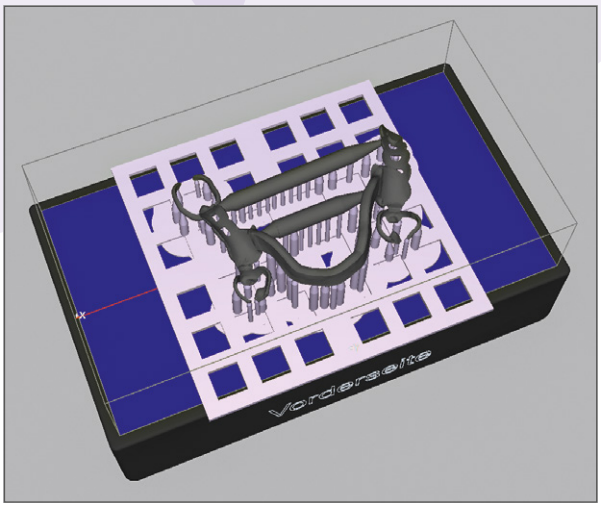
- Form: “Hash” or “peel off”-base
- Thickness of the base: 1.5 mm
- Distance of cast object to building platform: 15 to 10 mm

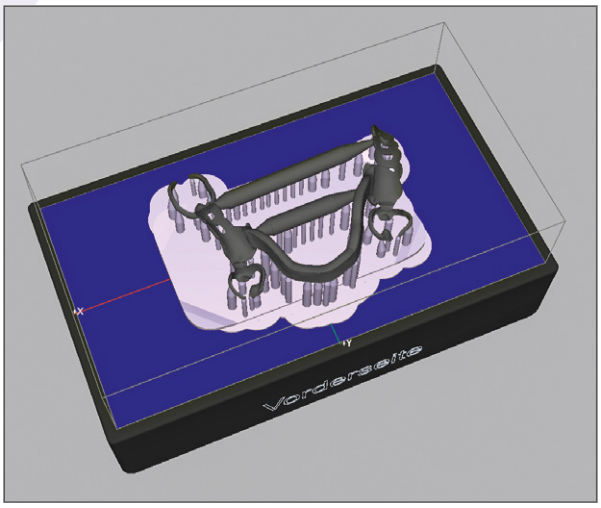


“Hash” foundation



“Peel off”-base



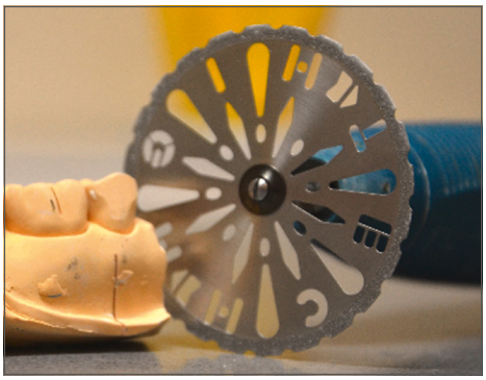


Changes/Special notes

- **Recommended print resolution: 50 µm**
- **NEW cleaning times: 1 min pre-cleaning in iso-propanol and 1 min post-cleaning in fresh iso-propanol**
- **NEW post-curing times: 180 s (90 s front side, 90 s back side)**

Separation of the supports

To ensure that no surrounding parts of the printed object are damaged when the supports are cut off, we recommend using a cutting disc (for gypsum). With cast clasps, you can significantly reduce the risk of clasps breaking by carefully separating the supports from the base first.



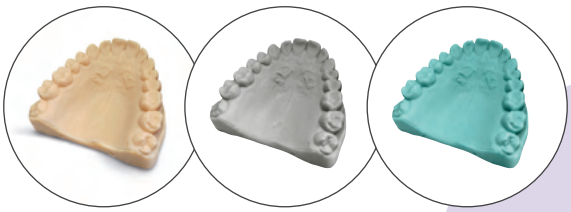
Spruing and investing

Please invest the printed objects **immediately** after post-curing. The accuracy of fit deteriorates with increasing storage time. If, contrary to expectations, immediate investing and further processing is not possible, the printed object can be temporarily stored on the corresponding gypsum model, protected from light, for 24 h at room temperature.

If too many printed casting objects are placed too close together in only one investment ring, this increases the risk of cracks and flaws in the cast objects or, in the worst case, of investment ring fractures during heating. It is preferable to divide the printed objects between several investment rings instead of placing too many objects in one large investment ring.

NOTE:
IN GENERAL, WE DO NOT RECOMMEND THE USE OF SURFACE TENSION REDUCING AGENTS (DEBUBBLIZER/ SURFACTANT) IN COMBINATION WITH DIMA PRINT CAST RUBY. THEY INCREASE THE RISK OF POROSITY IN THE SURFACE OF THE CAST OBJECT.

5.6.1. dima Print Stone



Color options

The model material dima Print Stone is available in three colors. The color **beige** is closely aligned to common dental gypsum to facilitate an easy transition to the production of digital models.

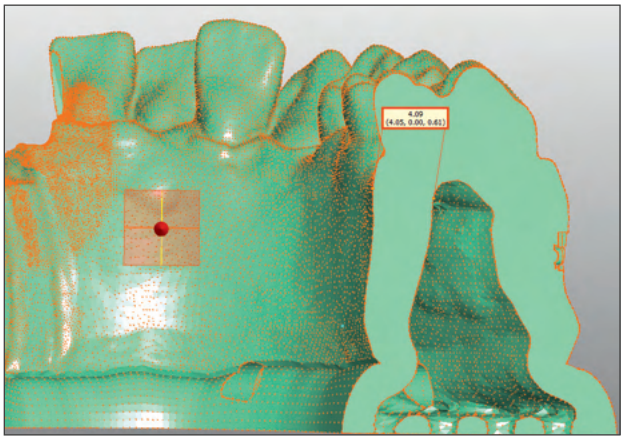
The color **gray** is particularly well suited to the aesthetic needs of realistic ceramic veneering jobs. The neutral color of dima Print Stone gray helps avoid any visual color distortion when preparing ceramic veneers.

dima Print Stone **teal** is geared towards the preparation of orthodontic appliances.

Design notes

The following recommended settings and design tips apply only to Model Builder™ from 3Shape. You can download the complete parameters as a DME file to import into 3Shape in the cara download area: **kulzer.com/cara-print-3shape-dme**

With regard to the parameters for the settings of the material please see the cara setting parameters document for:
cara CAD: **kulzer.com/cara-print-cad**
exocad: **kulzer.com/cara-print-exocad**
Dental Wings: **kulzer.com/cara-print-DentalWings**

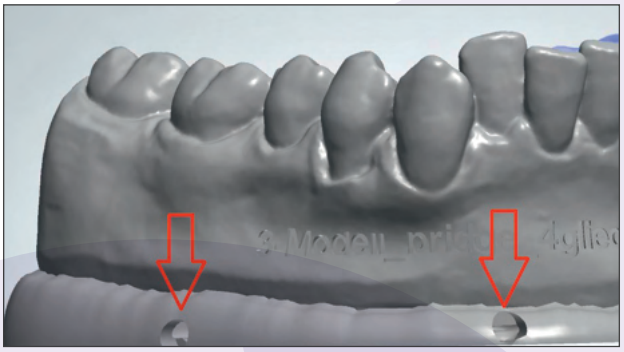


- Build type: hollow (Solid objects tend to become too heavy and detach from the platform while printing. They also require more energy for illumination, which also increases printing time).

Minimum wall thickness for models:

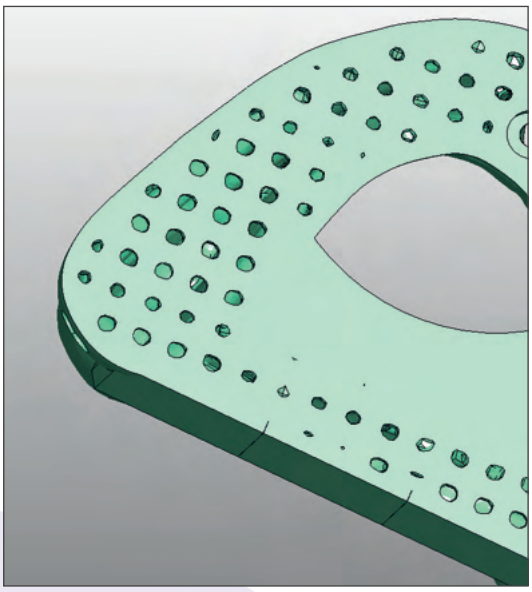
- Partial/quadrant models (positioned flat): 2 mm
- Full models (positioned flat, if there is enough space – as well as inclined): 3–4 mm
- When positioning/orienting models flat on the build table, once the model is oriented properly, highlight the appliance and hit the “Snap to Floor” button at the top of your CAM screen. This will snap the appliance to the floor of the build table.

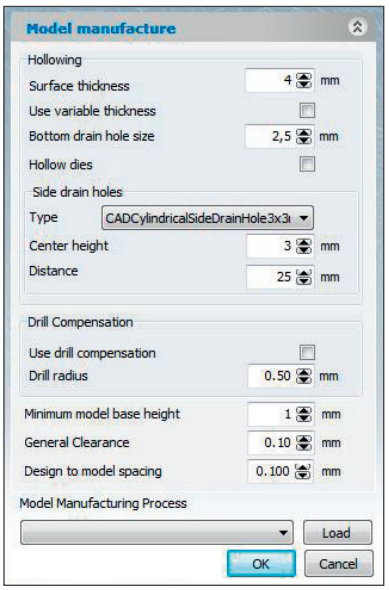
5.6.2. Anatomic models



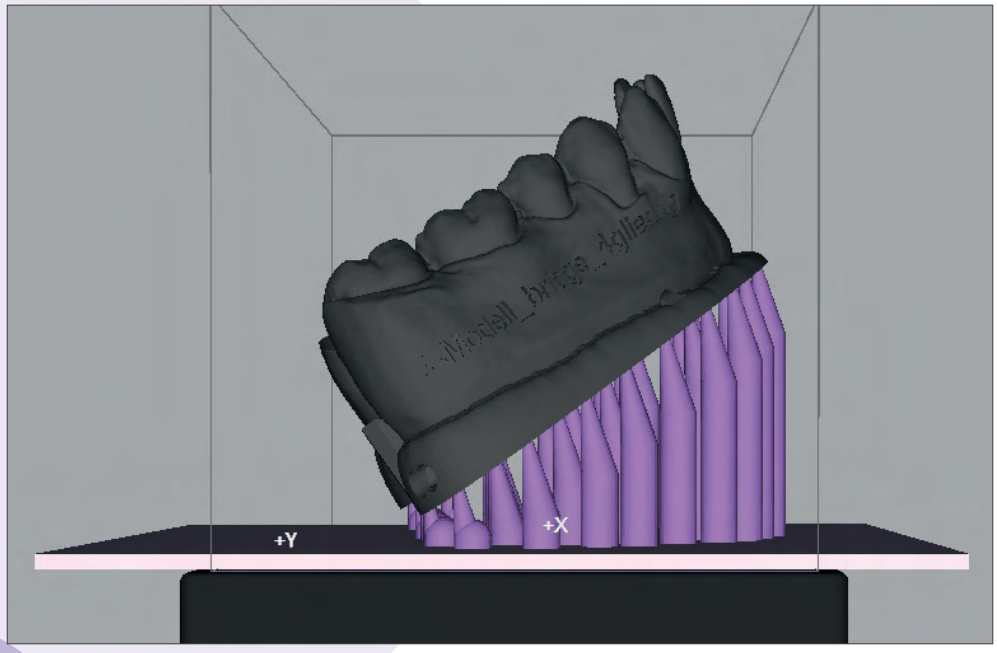
While printing, the hollow cavity of the model will fill up with liquid resin unless it is allowed to flow out continuously. To ensure this happens, we recommend adding a sufficient number of drainage holes to the base and sides of the model.

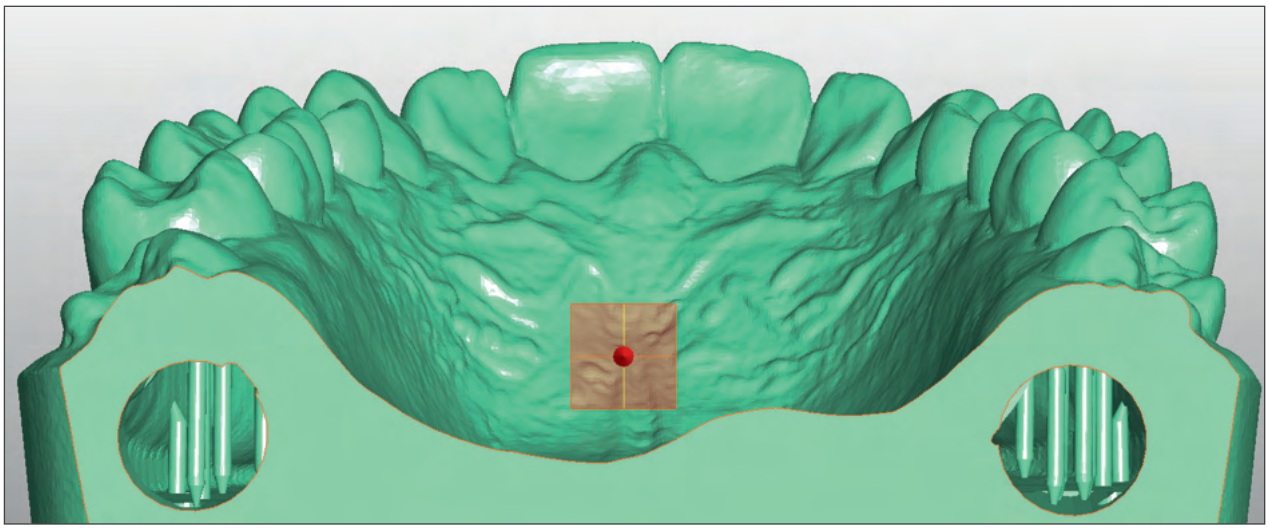
- Drainage holes on the side: 3 mm in diameter, positioned every 25–30 mm depending on the size of the model (partial/quadrant models: approximately distance 10–15 mm)
- Height (centered height): the center of the drainage hole should be positioned 1–3 mm above the model’s base
- Drainage holes on the base: min. 2.5 mm in diameter (these are automatically distributed across the entire surface of the base)



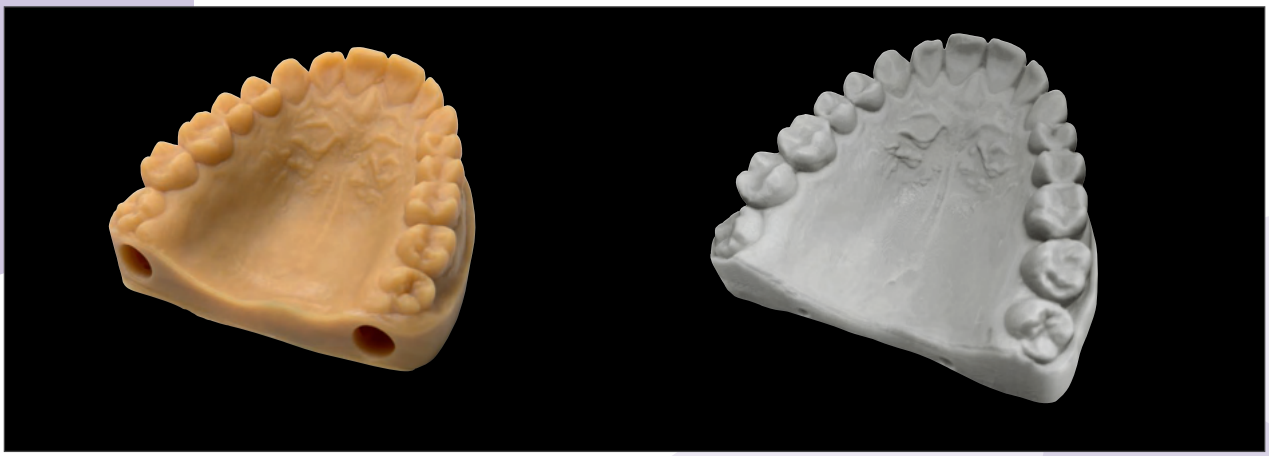


- If a model must be positioned at an angle due to its size (optimal angle: 50°–60°), please use supports for better stability. We recommend the “Heavy” form to ensure the best stability.
- The base (“Cube”) should be at least 1.5 mm thick.
- Form of the support tip on printed piece: “Cone 25 %” (for easier removal after printing)





Cleaning recommendations
Before the usual cleaning (see Section 4.5.2), we recommend using compressed air to blow out any remaining liquid monomer. Please use extreme care when removing the remaining resin from the hollow areas of the model.



dima Print Stone for
5.6.3. Die models

Color options

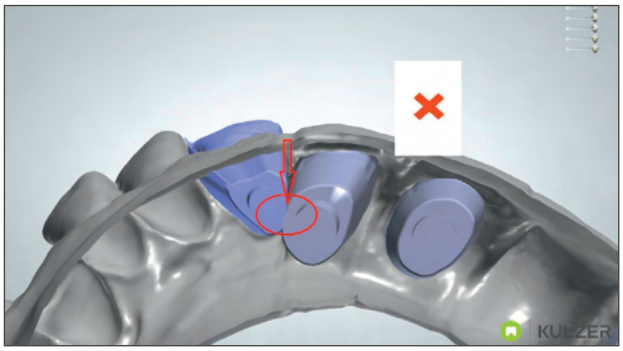
Both **dima Print Stone beige** and **dima Print Stone gray** are equally well-suited to printing die models.

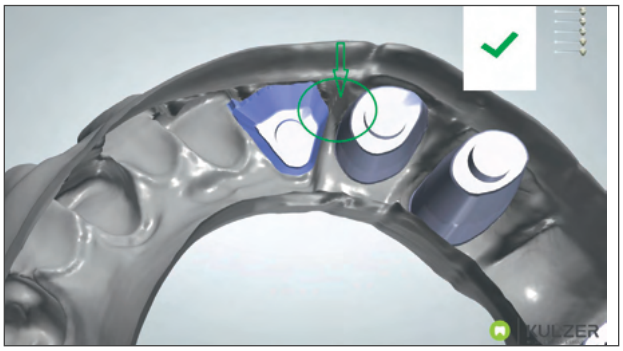
Design notes

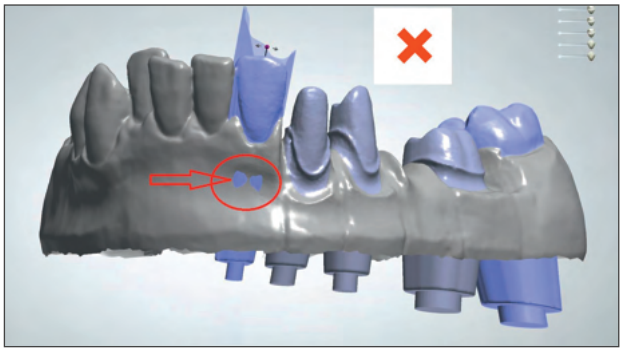
- The recommended settings and design tips apply only to Model Builder™ from 3Shape.
- In the order entry, please select the option “Digital Impression” to activate Model Builder. Two CAD settings are available: “large dies” for posterior teeth and “small dies” for anterior teeth. If both tooth types are present in equal numbers in a construction, please use the “small dies” library.

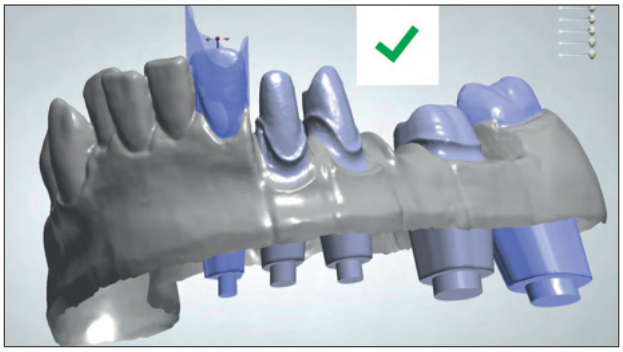
- It is important to always inspect the model visually in order to prevent a die from exiting the side of the model, or to prevent undesired contact between the dies.

We recommend a safety distance of 1 mm between dies, as well as between individual dies and the exterior of the model (see following pictures).



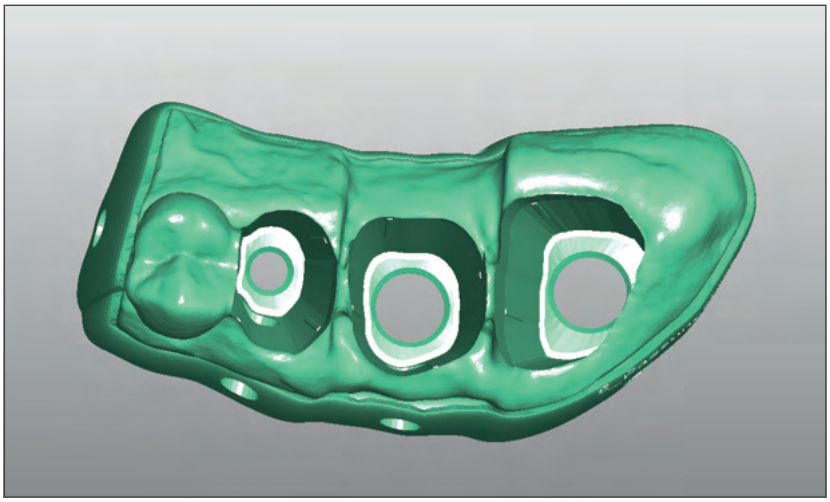


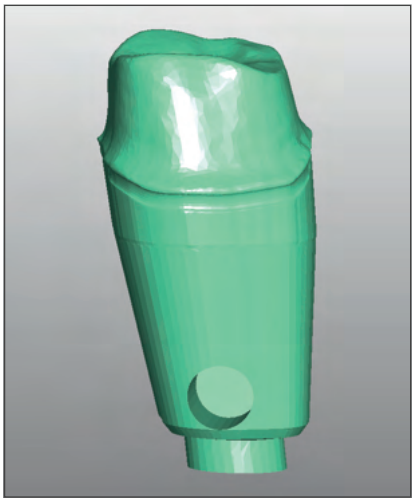




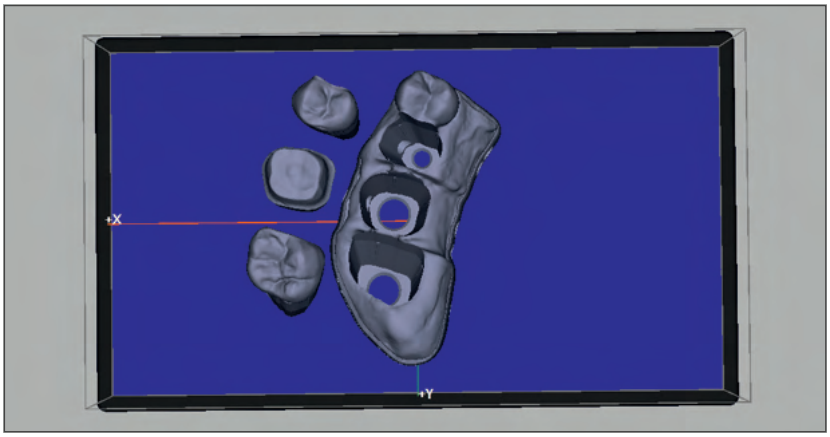
Partial models

We recommend printing smaller models parallel with the building platform, i.e. “flat”, as shown in the following image.



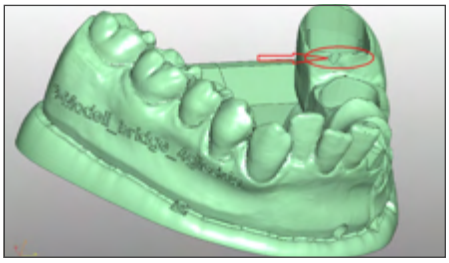


Important:
The dies must be aligned in the same position (z-axis) as the die fixture. This helps ensure a smooth surface in the areas most relevant to fit. Later, this will also guarantee a better fit for the die within its fixture.



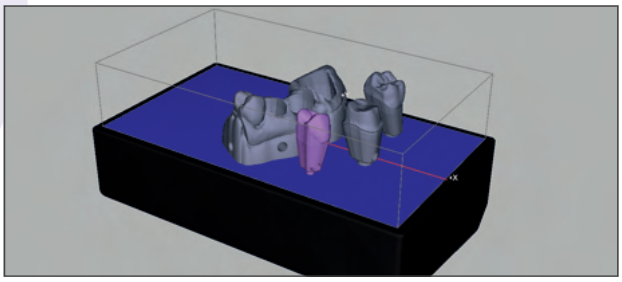
Full models

Thin areas, which may include areas that taper off to 0 mm, must be adjusted in 3shape ModelBuilder™, because they are not printable.

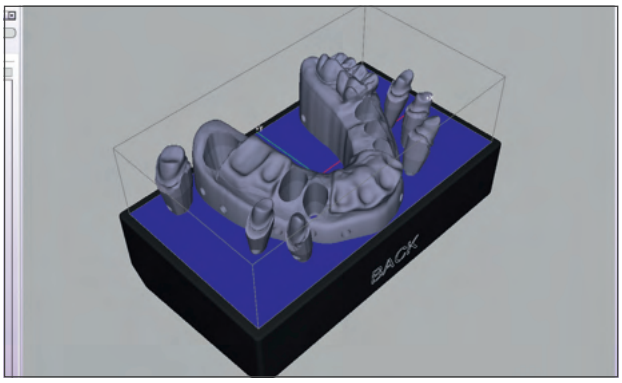


Positioning within the building platform

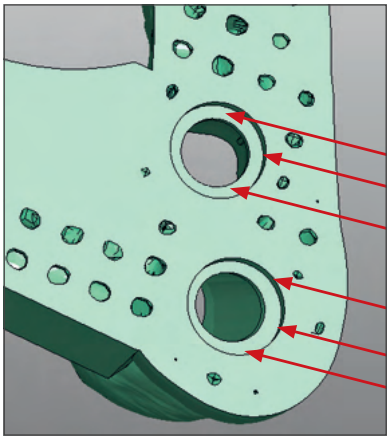
Partial model



Full model



- If a model has to be positioned at an angle due to its size (optimal 50°–60°), please ensure that you add enough supports (as the piece will be heavy).
  We recommend using the form “Heavy” to ensure sufficient stability.
  Note: the precision of the printed model decreases at higher angles.
- When placing supports, make sure that no supports are positioned around the ring area below the places where dies are located on the model. Delete any supports located within the spaces indicated by the red arrows in the picture.
- The foundation (“Cube”) should be at least 1.5 mm thick.



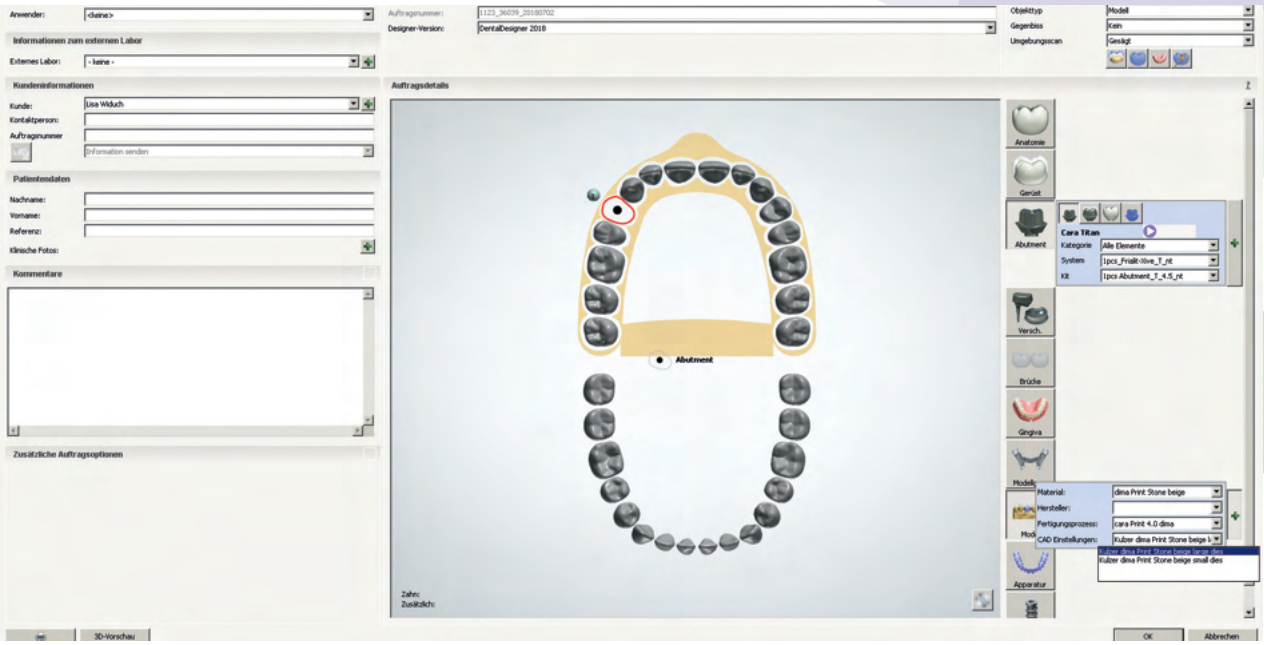
dima Print Stone for
5.6.4. Implant models

You can easily and reliably print implant models using cara Print 4.0 to produce form-fitting abutments. The 3D-printed implant model allows each patient situation to be transferred into a precise working model. And **dima Print Gingiva Mask** (Chapter 5.7.), a material which remains soft, makes it possible to realistically re-create the patient situation.

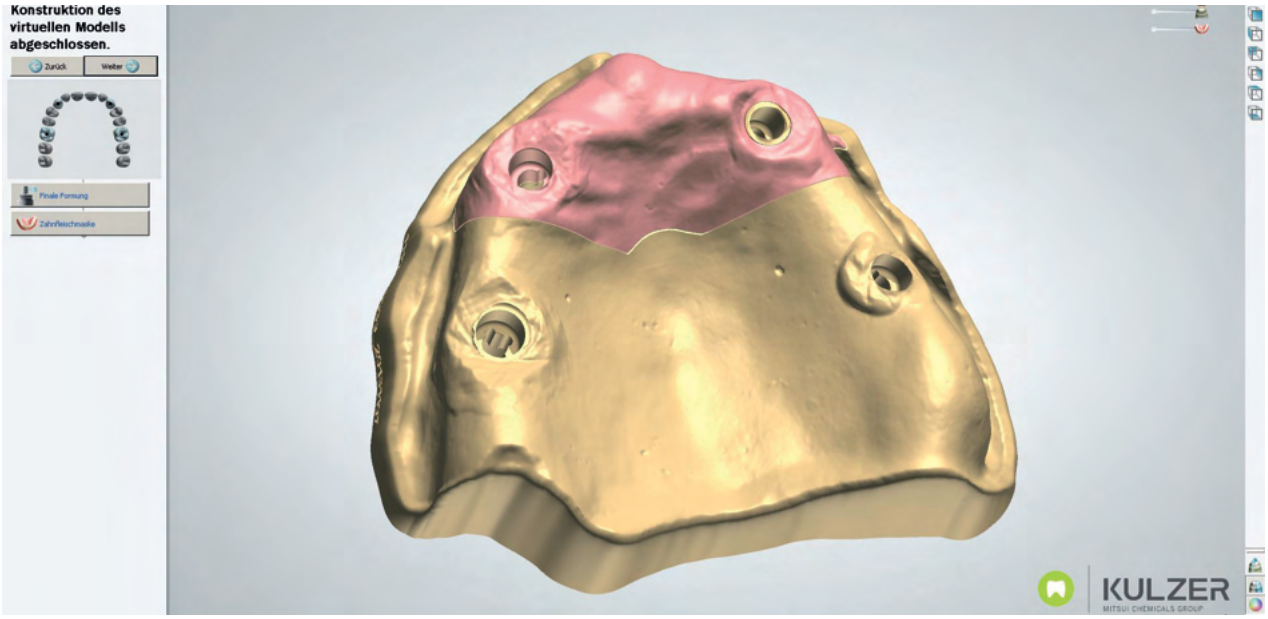
Construction information:

- The adjustment parameters and design recommendations are based exclusively on the use of Model Builder™ from 3Shape.
- Please note that before beginning the design, the library for each implant system is to be imported in the 3shape Control Panel. Restart the DentalManager afterwards if necessary.
- Before you start with the actual design of the implant model for 3D printing, the CAD design for the desired abutment must be concluded.

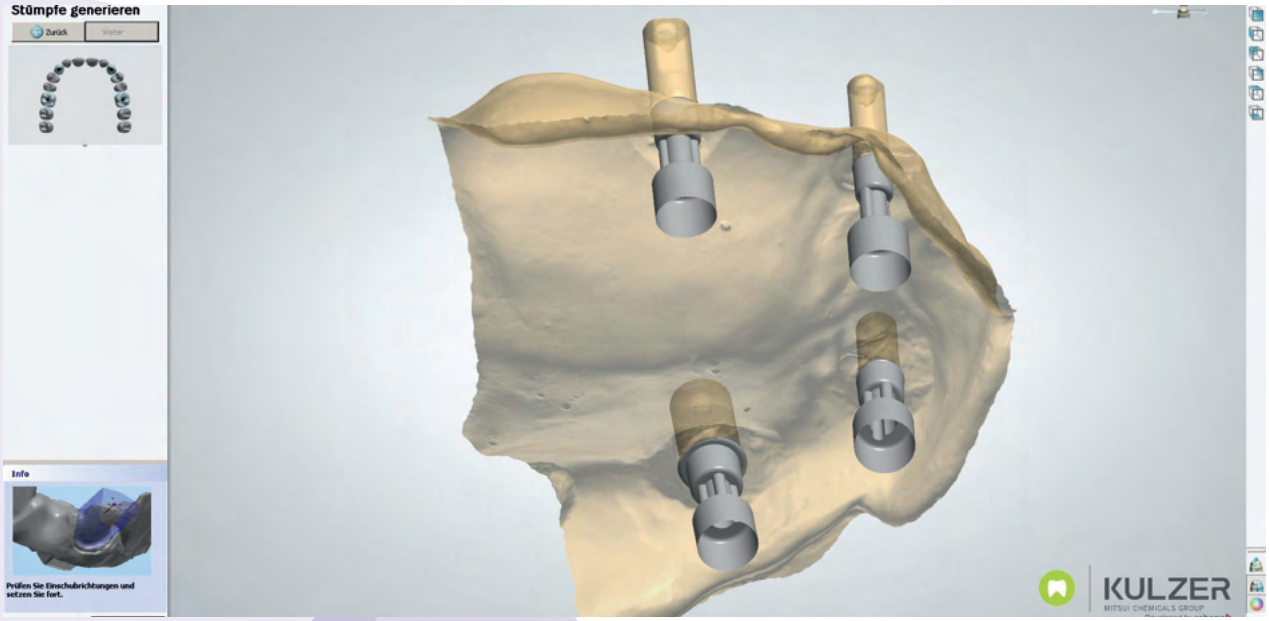
The order entry for the 3Shape model then appears in the DentalManager as shown here:



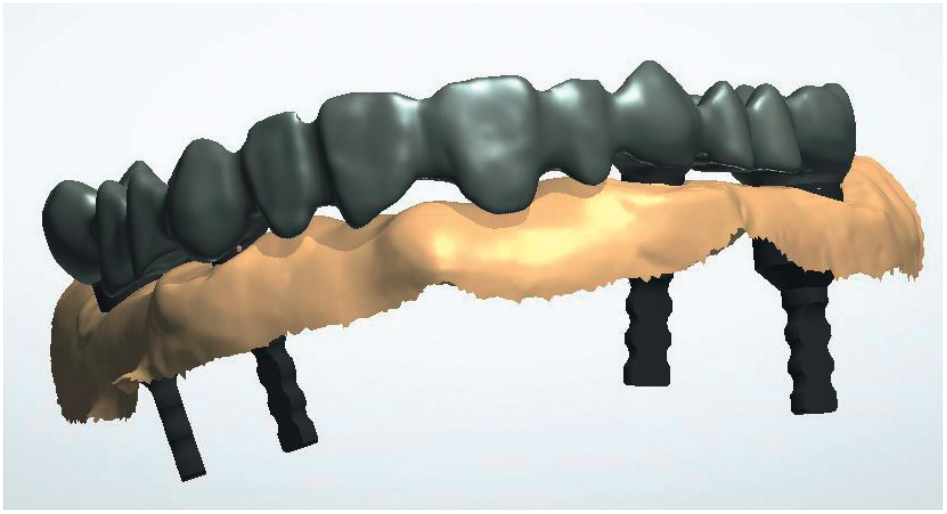
If you follow the indicated series of steps, you will receive the following image once the step “prepare scans” is completed and after the intraorally scanned mouth situation has been successfully imported:

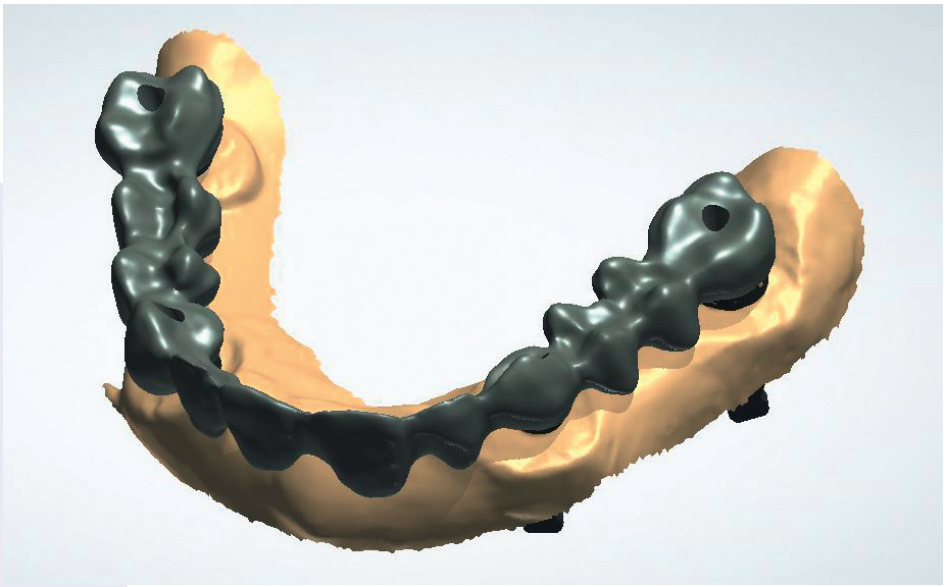


Based on the implant position provided by the scan body, the so-called DIM analog (Digital Implant Model) is virtually inserted at the correct position in the implant model:
If desired, at this point you can define a certain area around the implant as “gingiva”.

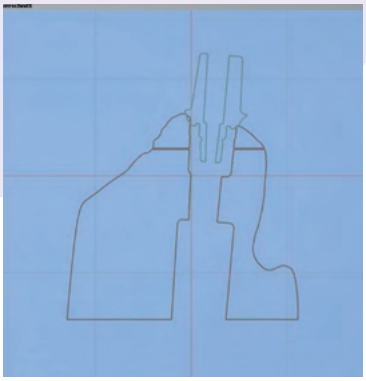


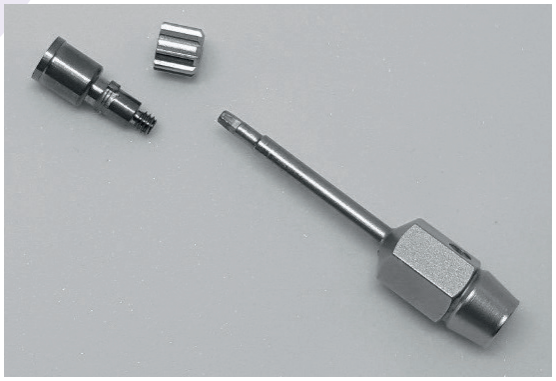
This allows you to later produce a removable soft gingiva mask.
After finalizing the model design, the result should look like this example:



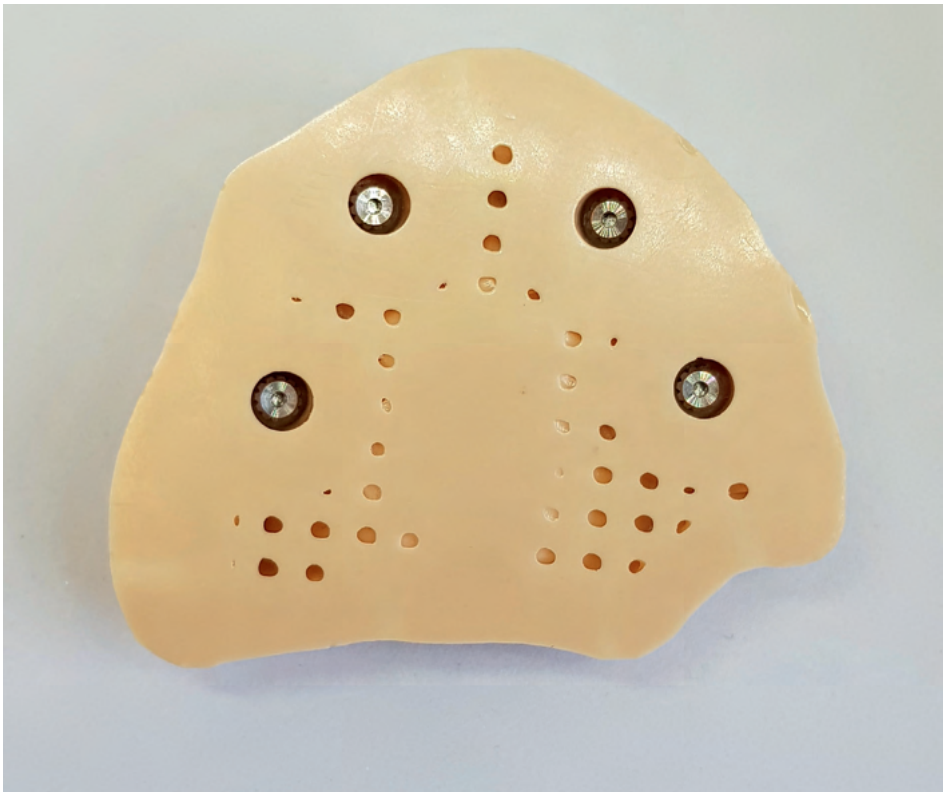


On the left, you will see a cross section through the positioned DIM analog. On the right, you see the actual DIM analog with locking nut and respective screwdriver.





Basal view with screwed-in DIM analogs:



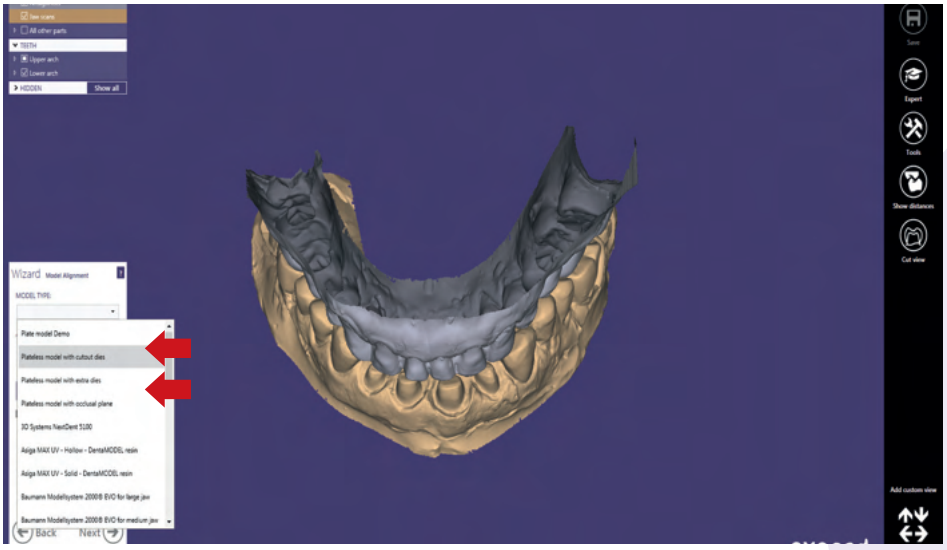
5.6.5. Models with exocad

We have also developed setting parameters as well as CAD design recommendations for exocad for you.
Setting parameters in general can be found in download section on our homepage: **kulzer.com/cara-print-exocad**

! **PLEASE KEEP IN MIND THAT ALL MODELS MUST BE DESIGNED IN HOLLOW MODE AND THEREFORE ALSO HAVE TO BE PRINTED HOLLOW.**

Design notes:

- Please start designing your model in exocad “Model Creator” by selecting either “plateless model with cut dies” (die model) or “plateless model with occlusal plane” (anatomic model).
- exocad software does not allow placing necessary drainage holes within your model for waste printing material. Therefore all models to be printed must be equipped with a foundation in cara CAM software afterwards.



- Please use our recommended setting parameters for die fitting based on our published table:

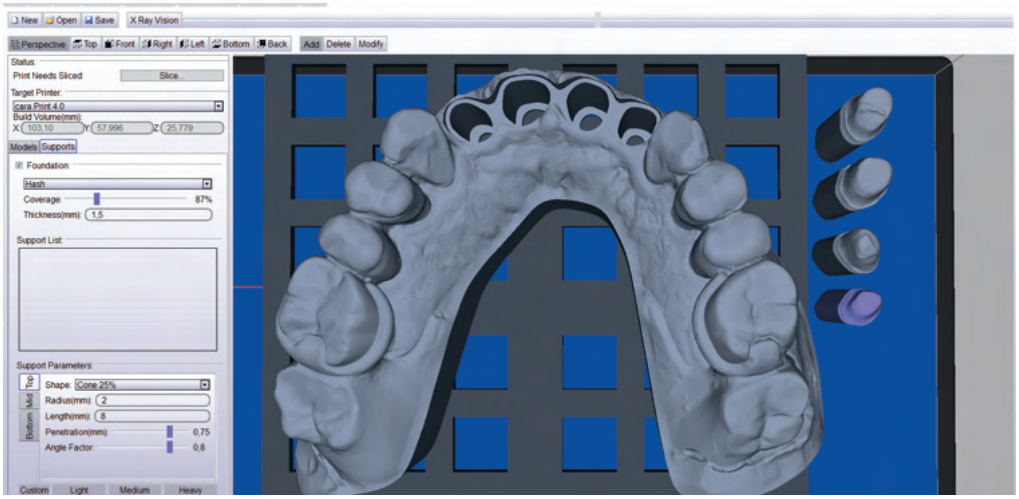
Software Setting Options

As of 07.2019

dima Print Stone beige					
Plateless Model Design					
DIES	Pin height	Preparation margin	Seating width	Omit die pin	Add concavity
	2.5 mm	0 mm	1.5 mm	non-activated (Checkbox)	activated (Checkbox)
Settings					
Base	Horizontal shaft	Vertical shaft gap			
	0 mm	0 mm			
	Ditch	Ditch width	Ditch depth avg.		
	activated (Checkbox)	0.5 mm	0.5 mm		
	Add pin grooves	Groove width	Groove depth		
	activated (Checkbox)	1 mm	0.5 mm		
	Hollow model	Wall thickness	Cavity fill diameter		
	activated (Checkbox)	3 mm	0 mm		
	Anticipate milling				
	non-activated (Checkbox)				

Please note: Use the support structure Hash to close the model basal.

Now the exocad model STL file needs to be processed with cara CAM as usual to generate a print file. Please choose foundation type “hash”. This type of foundation (or support structure) ensures that waste material can drain after the 3D print. In addition, the grid base (foundation) gives the printed model enhanced stability, especially during later removal from the platform.



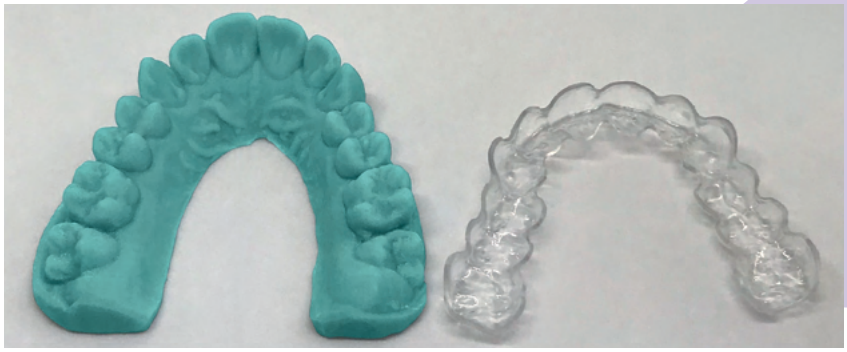
Please note that all designed models together with their attached foundation must be placed directly onto the virtual building platform in cara CAM (menu item: “snap to floor”) as the final step.

Information for implant models:

It is not yet possible to generate perfectly fitting models with DIM analogs with current exocad version (DentalCAD 2.3 Matera, Engine build 6990). So far there is no possibility to change a specific fit parameter in exocad unfortunately.

5.6.6. dima Print Stone teal – orthodontics

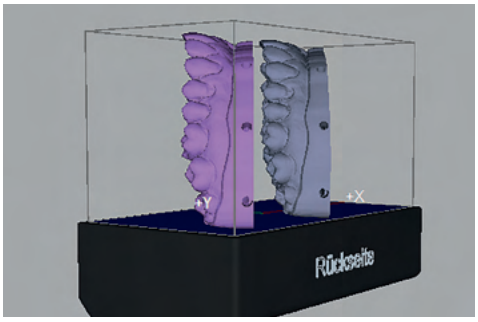
Due to its good heat resistance, dima Print Stone teal is suited to the production of orthodontic appliances (e.g. aligners, activators, bionators double-plates, etc.). Production is simplified by the relatively short printing time that is made possible with dima Print Stone teal.

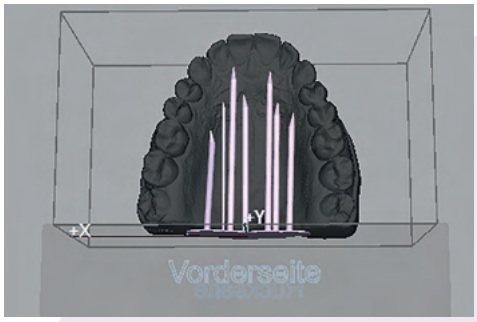


Design notes:

The recommended settings and design recommendations here only apply when used in combination with Model Builder™ from 3Shape. You can download the complete parameters as a DME file to import into 3shape in the cara download area: **kulzer.com/cara-print-3shape-dme**

- Build type: generally hollow with a minimum thickness of 3 to 4 mm.
- Build type: solid, if the maximum object height is smaller than 15 mm (for models that will later be used for deep drawing foil)
- Due to space restrictions, models (hollow or solid) for deep drawing splints may also be printed at an angle of 90° in deviation from the previous recommendation (chapter 5.1.1.).





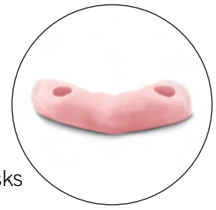
For the following topics:

- drainage holes on the side to drain excess resin,
- drainage holes on the model base,
- support design, and
- special cleaning recommendations,

please refer to the contents and descriptions in Chapter 5.1.1. Anatomic models.

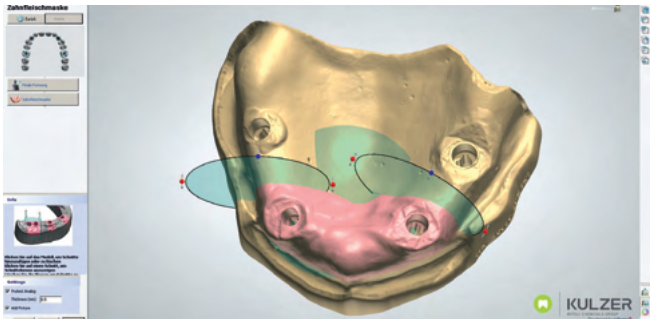
! DIMA PRINT STONE TEAL IS NOT SUITABLE FOR PRINTING DIE AND IMPLANT MODELS.

5.7. dima Print Gingiva Mask



With dima Print Gingiva Mask, a material which remains soft, you can create realistic gingiva masks for your printed model. There are no special settings needed for your CAD software.

Users can select and establish their respective dimensions for the gingiva mask on an individual basis.



Design & print notes:

Minimum thickness: 0.5 mm.
Applicable resolutions: 100 µm and 150 µm

To ensure an accurately fitting object, we recommend removing the printed gingiva mask from the platform of cara Print 4.0 using an extended box cutter very carefully.

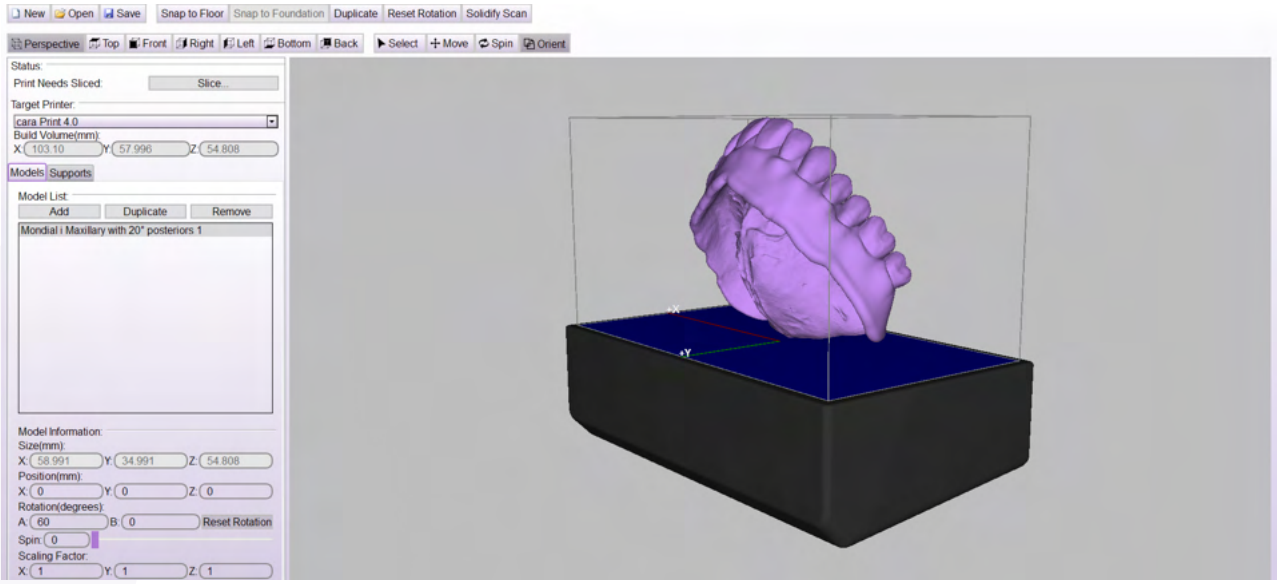
! TAKE GREAT CARE NOT TO DAMAGE THE SENSITIVE PRINT PLATFORM.

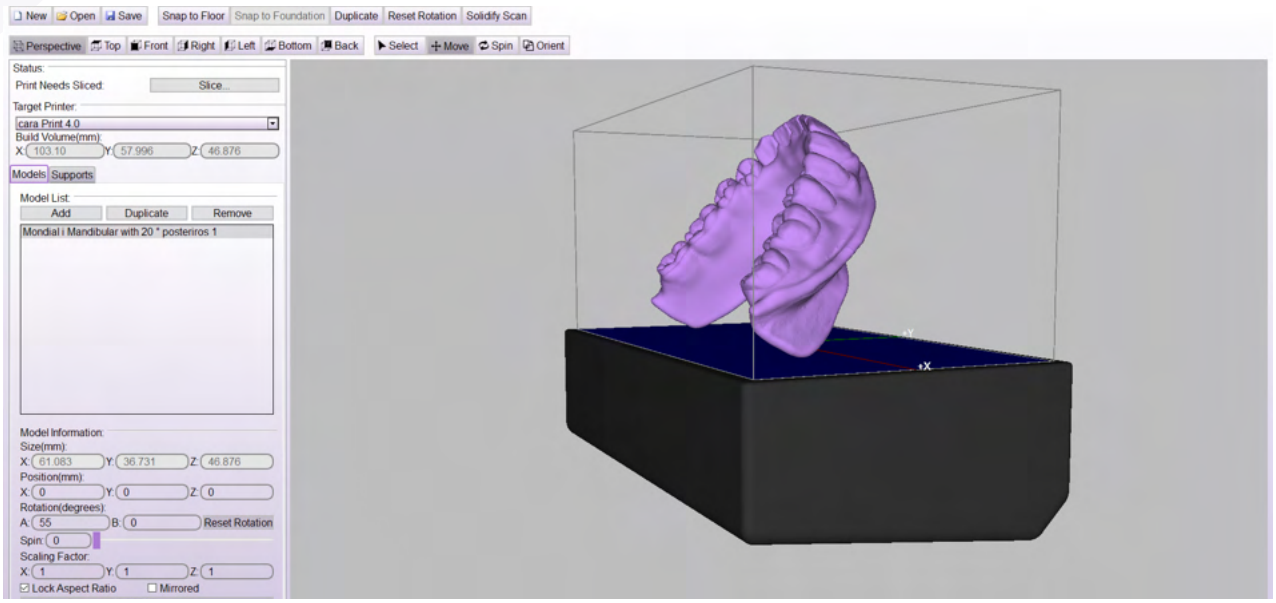
The gingiva mask can be adjusted well using a cross-cut burr.

5.8. dima Print Denture Base Try-in (white)

Recommended settings:

- Minimum Thickness: 2.5mm
- The recommended orientation is a tilted orientation such as space diagonal from 55° to 60° angle.
  If auto-orientation is not satisfied, rotate manually to make optimal position.





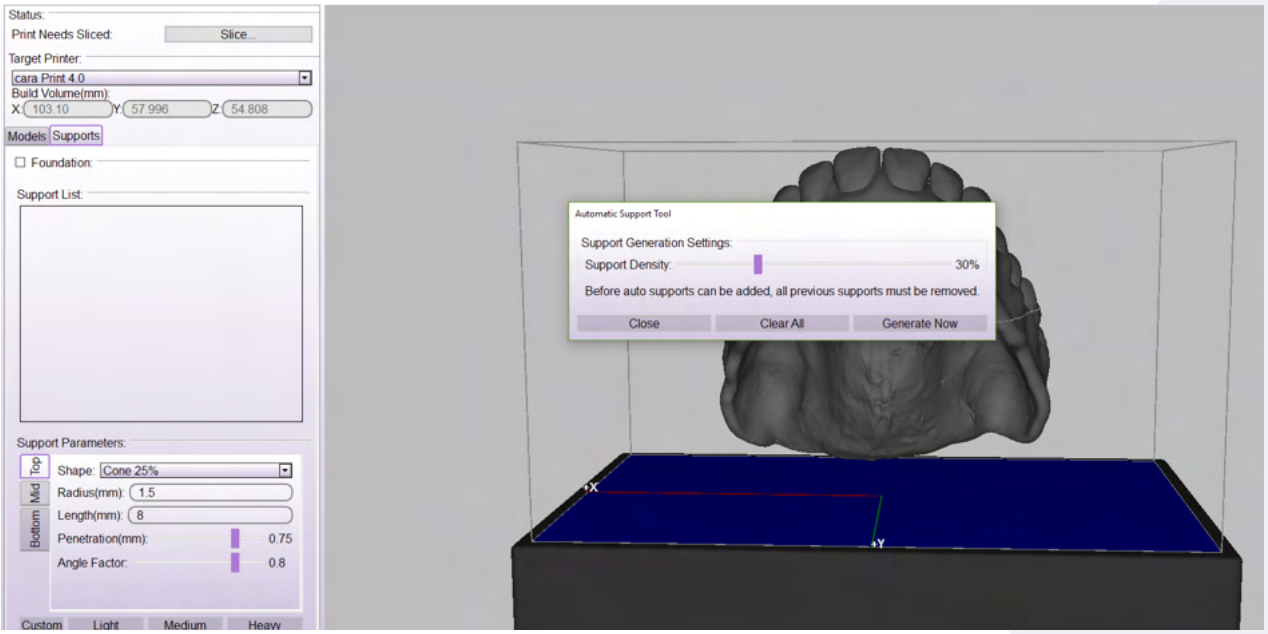
Angle (print positioning/orientation):

After the STL file is imported, rotate the appliance to fit on the build table in cara CAM as shown in the previous screen-shots.

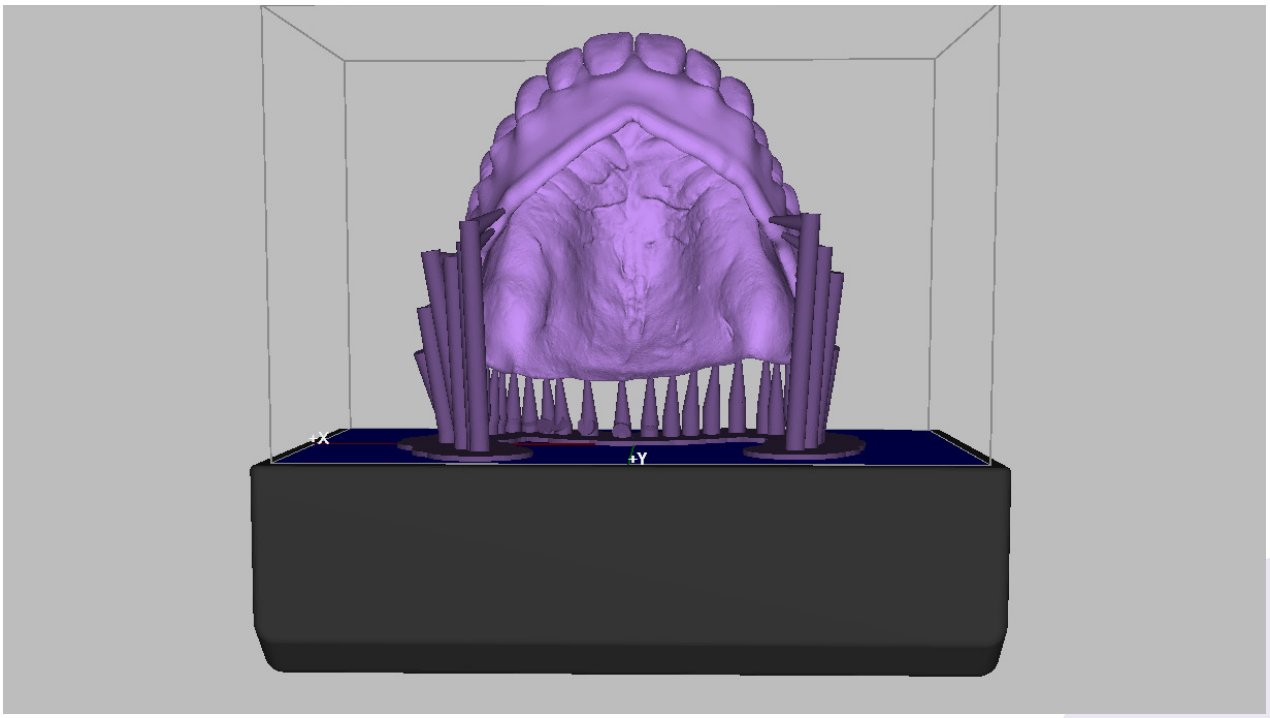
- Recommended angles: For maxillary, 60° (you can orient manually or by entering angle degree data in the rotation degrees boxes A or B under Model Information). For mandibular, 55° (you can orient manually or by entering angle degree data in the rotation degrees boxes A or B under Model Information).
- May need to adjust angles depending on the arch size and number of arches printing on the build table.
- Place the intaglio surface down towards the build plate (as shown below).
- After your orientation is set, highlight the appliance and hit the “Snap to Floor” button at the top of your CAM screen. This will snap the appliance to the floor of the build table.

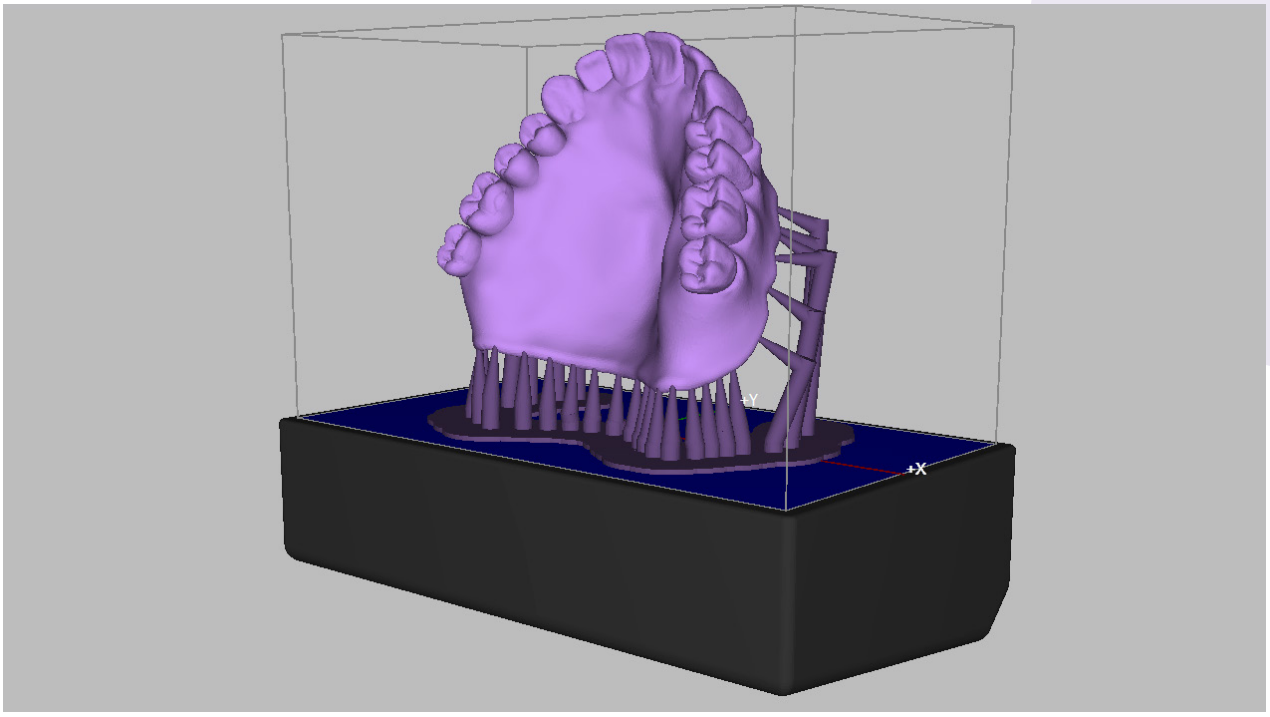
Supports:

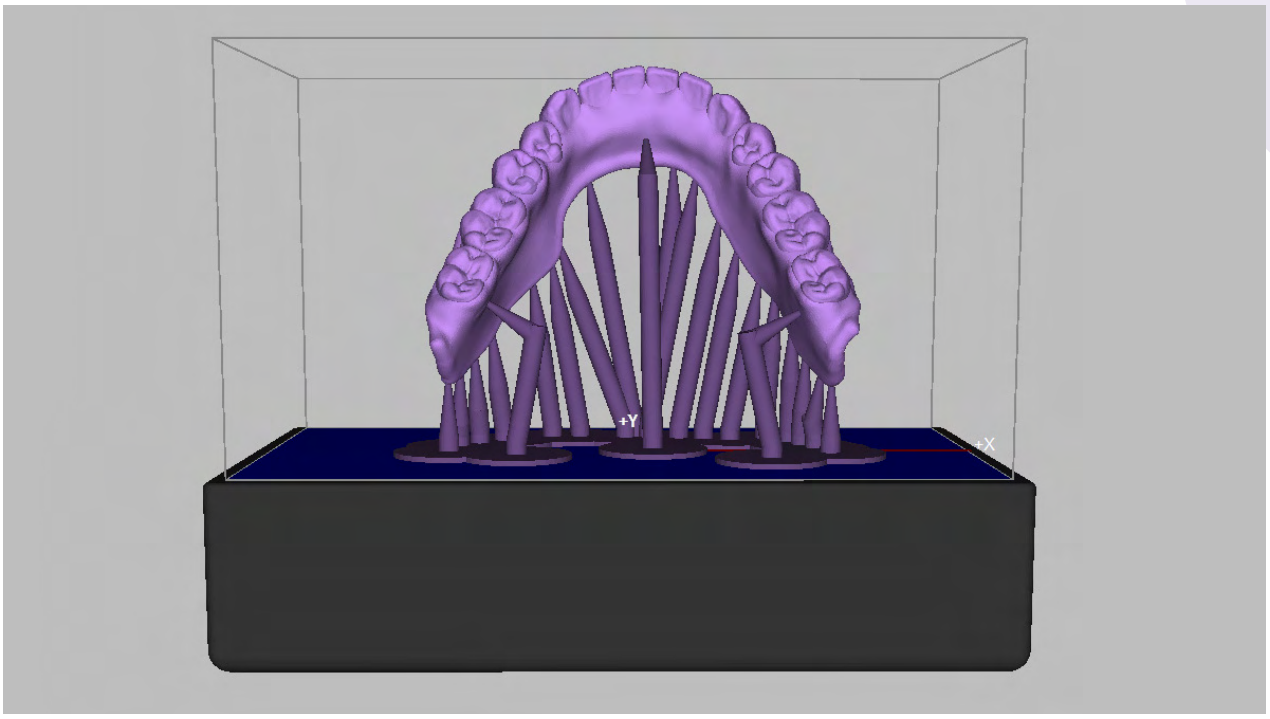
- We recommend manually placing the supports as shown in the photos on the next page. Supports will be on the border of the denture only and will be carried 2/3 up the border of the denture.
- Spacing with “heavy supports” no less than 1mm.
- Density of supports: Between 25%-40%, adjust manually if needed.
- Support point size: 0.4 – 1.0mm.
- Type for all supports: “Heavy”
- If you automatically generate supports (as shown below), you may adjust the supports as needed. You must remove any auto-generated supports effecting the intaglio surface.
- You can always add in supports, adjust the density of supports, and/or remove supports if auto-generation bunches them too closely together.

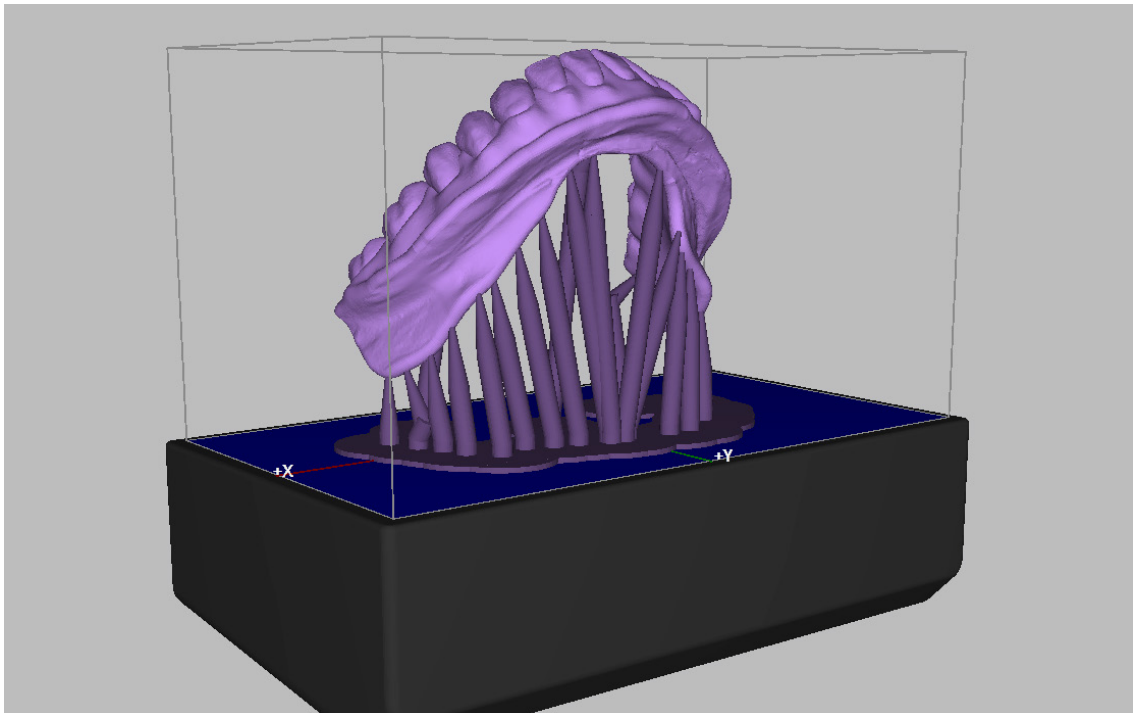


Examples shown below of where to manually place supports:





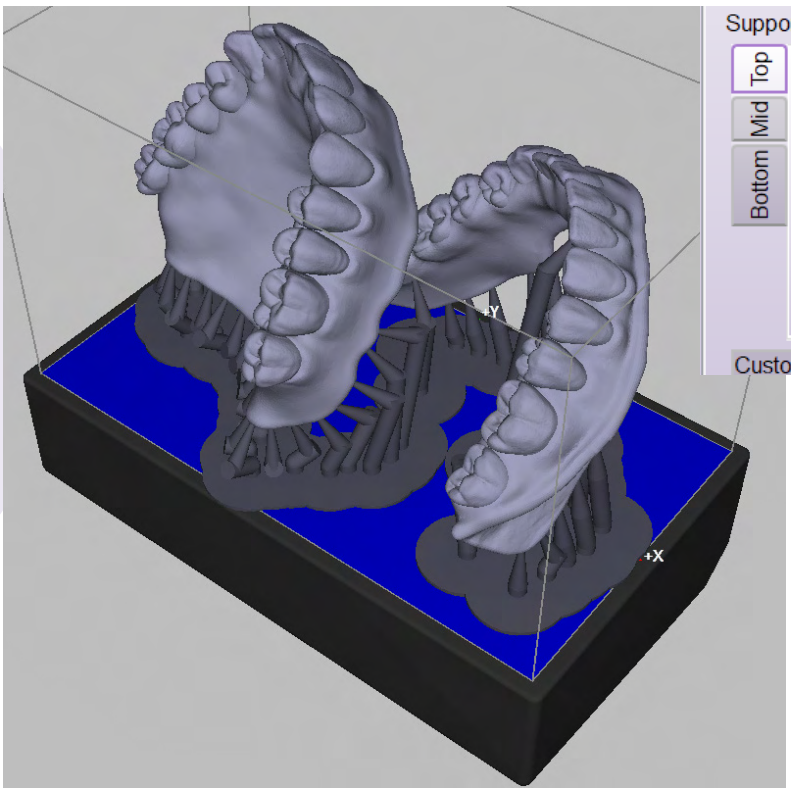


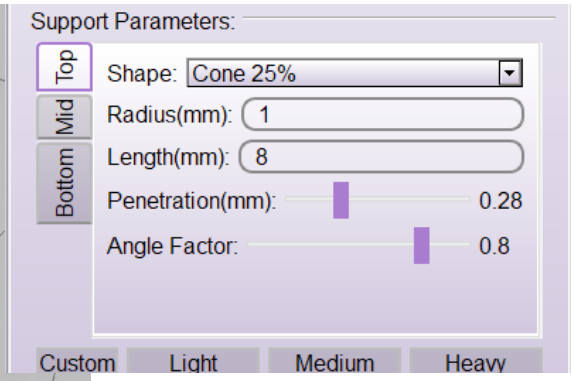


Support Parameters:

Although not required, you may also consider adding a foundation to your denture model.
This will help support the model during production. If adding a foundation, it is recommended to use the peel or hash foundation at a thickness of 1.5mm.

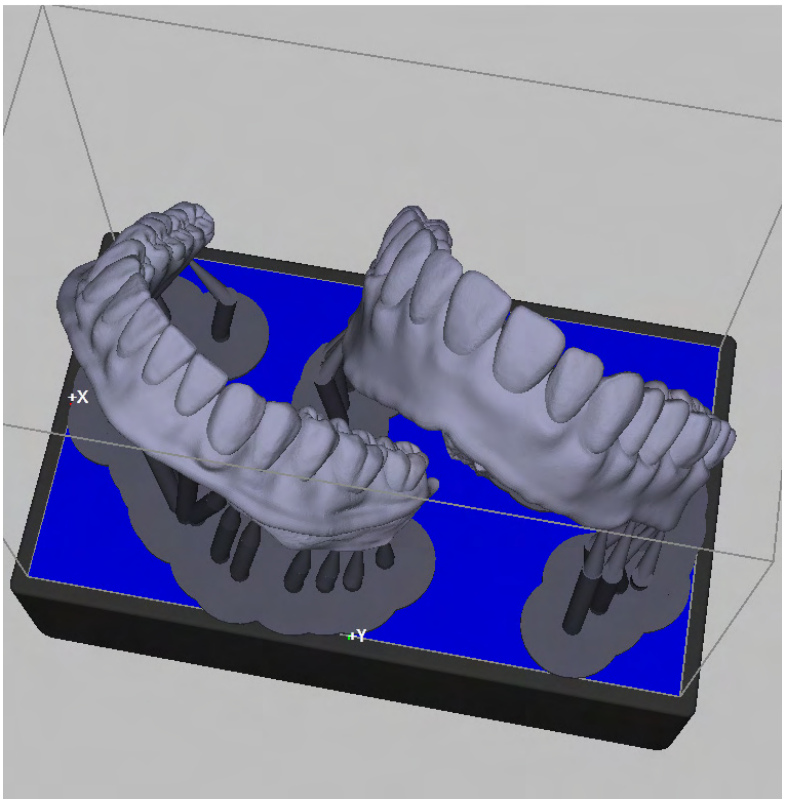
If printing 2 arches on the build table simultaneously, we recommend the below orientation.

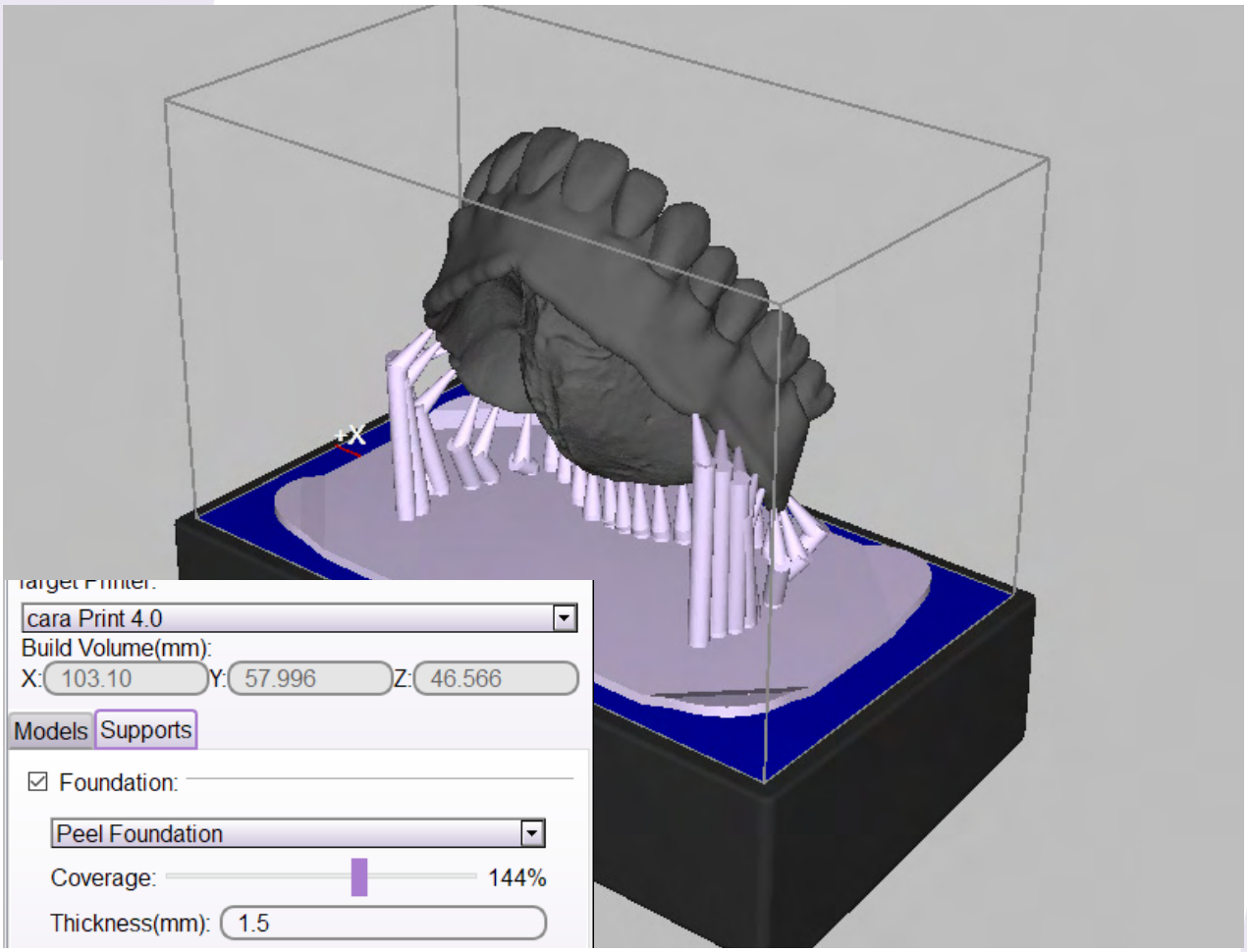




Printing parameters:

- Layer thickness: 100µm
- Optimal orientation: 50 – 60 degree tilted orientation
- Support density: Between 25%-40%, adjust manually if needed.
- Fill the resin tray at least 1/3 full slowly to cover 85% of the resin tray surface and allow it to spread evenly. **Do not** overfill the resin tray. Max fill line should not be exceeded.





Foundation:

Not required, but if you choose a foundation we recommend the following settings:

- Hash or Peel foundation
- Thickness: 1.5mm
- If adding a foundation, once it's set, hit the "Snap to Foundation" button at the top of your CAM screen. This will snap the appliance to the foundation on the build table.

Post-Curing and Finishing:

- Place the printed denture base try-in into the HiLite® Power 3D for a total of 20 minutes (curing 10 minutes on each side).
- Cure the denture base try-in in a glycerin bath. The glycerin acts as an oxygen inhibitor. To do so, simply fill a small glass bowl with glycerin (we recommend a Pyrex 1-cup bowl, 3.5" x 1.7"), place the appliance into the glycerin bowl, and place the bowl onto the model tray inside the reflector pot of the HiLite (see page 59). Close the curing department door and begin post-curing (only put enough glycerin in the bowl to completely submerge the appliance).
- Once curing is complete, wipe down the try-in with a paper towel to remove the glycerin and use compressed air to blow out any sockets or crevices where the glycerin may remain. You may also rinse the cured appliance under water to remove any residual glycerin after post-curing is complete.
- Grind the support spots and borders of the fabricated try-in to make smooth.
- Polish the try-in denture with polishing unit.

*Post-curing in glycerin

CAUTION: THE GLYCERIN WILL BE VERY HOT AFTER CURING. PLEASE USE AN INSTRUMENT TO FLIP THE APPLIANCE AND TO REMOVE THE APPLIANCE FROM THE HOT GLYCERIN ONCE CURING IS COMPLETE!

Special Notes:

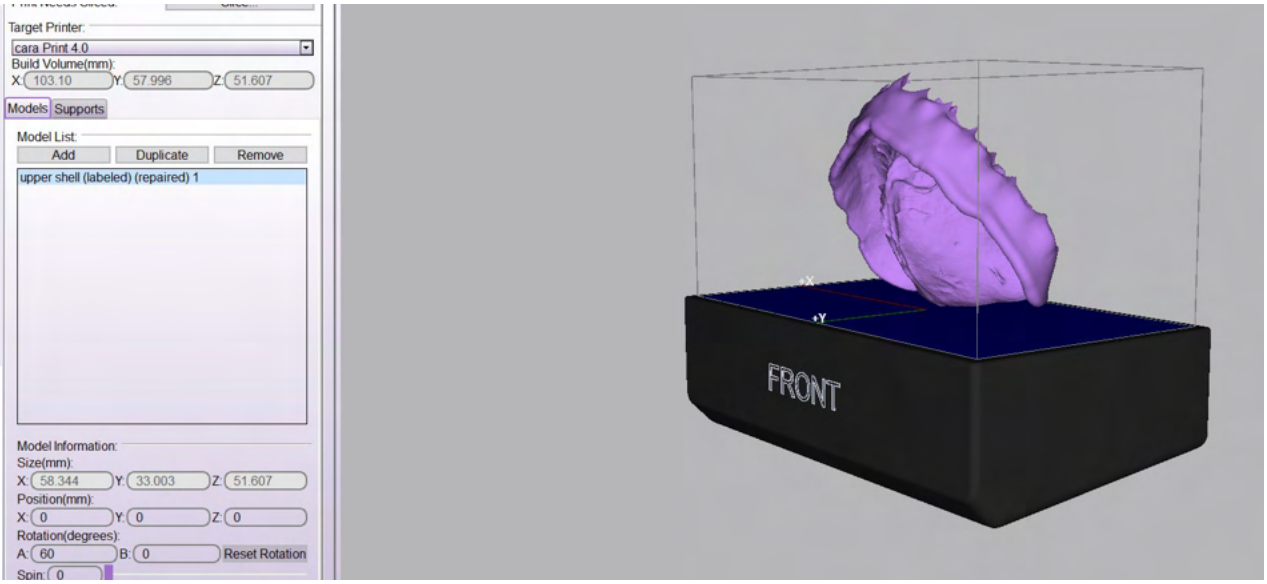
- Stir print material in the cara Print 4.0 print tray thoroughly after each print with the silicone spatula included with the printer.

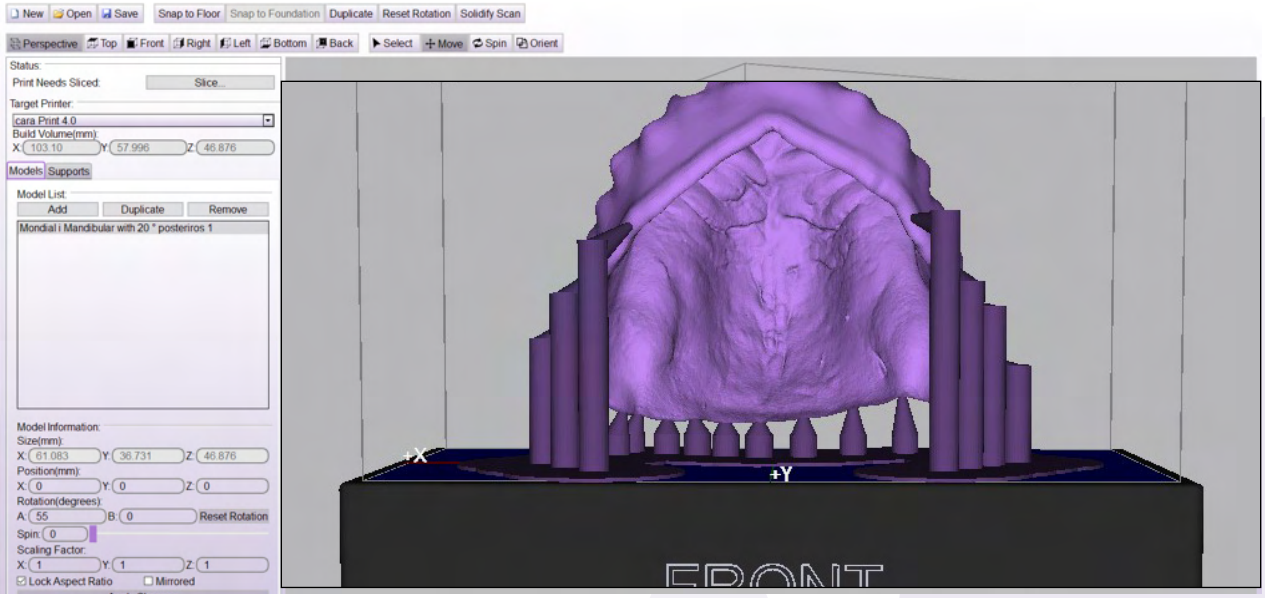
*dima Print Denture Base and dima Print Denture Teeth materials should all be post-cured with an oxygen inhibitor (we recommend glycerin) because the green state of these printed objects have unreacted functional groups. If these printed objects further cure inside our HiLite power 3D at normal environmental air temperatures, the oxygen generates the soft surface layer at the printed surface after curing. Curing in glycerin will prevent this soft surface layer from blocking the oxygen. Also, in glycerin, the unreacted functional groups reactivity will increase.

5.9. dima Print Denture Base

Recommended settings:

- Minimum Thickness: 2.5mm
- The recommended orientation is a tilted orientation such as space diagonal from 55° to 60° angle (See “Angle print positioning” instructions). If auto-orientation is not satisfied, rotate manually to make optimal position.





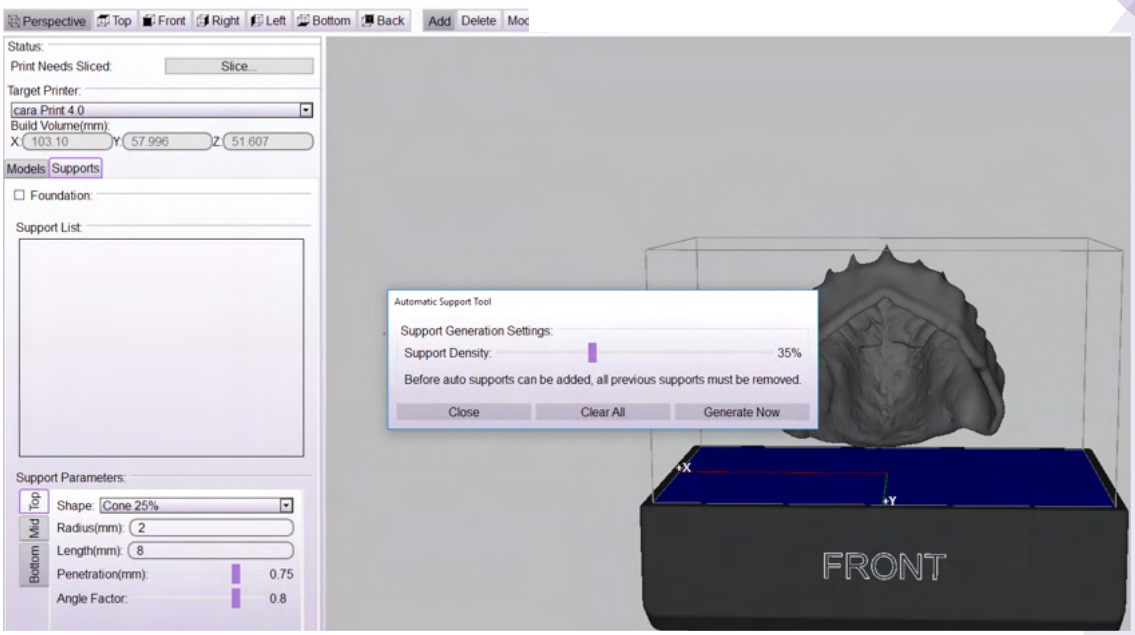
Angle (print positioning/orientation):

After the STL file is imported, rotate the appliance to fit on the build table in cara CAM as shown in the previous screen-shots.

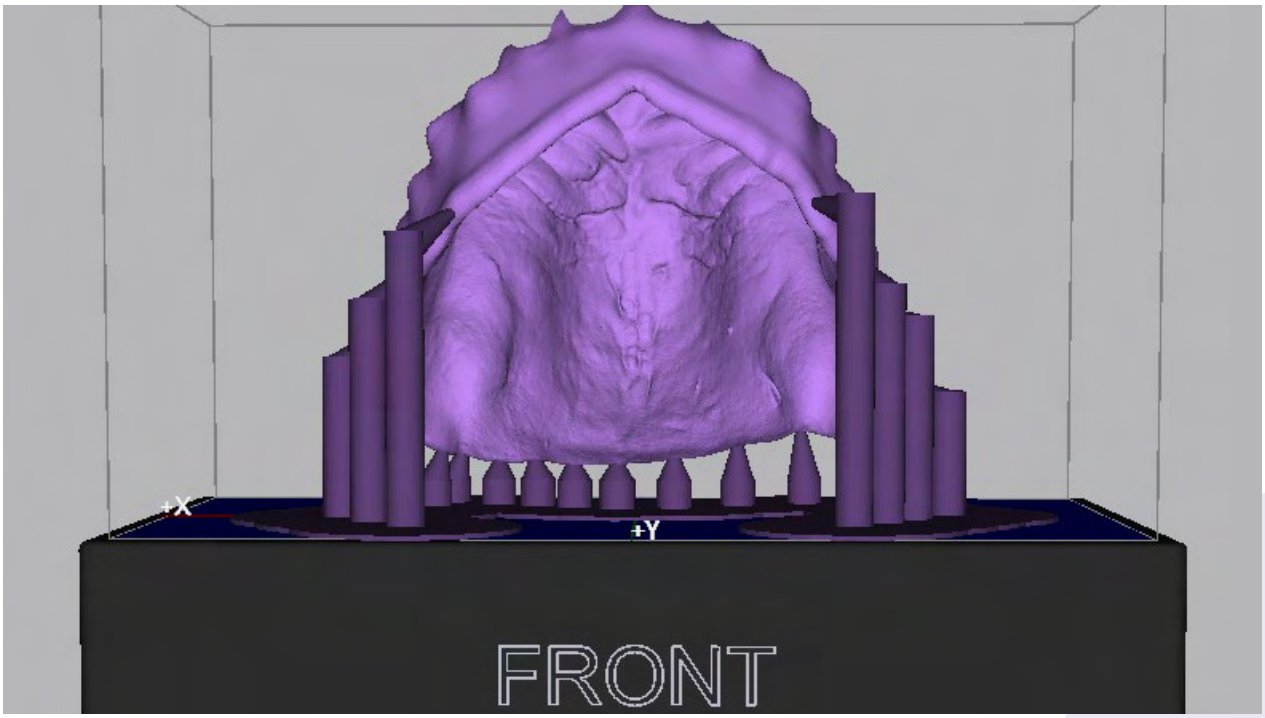
- Recommended angles: For maxillary, 60°. (You can orient manually or by entering angle degree data in the rotation degrees boxes A or B under Model Information). For mandibular, 55°. (You can orient manually or by entering angle degree data in the rotation degrees boxes A or B under Model Information).
- May need to adjust angles depending on the arch size and number of arches printing on the build table.
- Place the intaglio surface down towards the build plate (as shown below).
- After your orientation is set, highlight the appliance and hit the “Snap to Floor” button at the top of your CAM screen. This will snap the appliance to the floor of the build table.

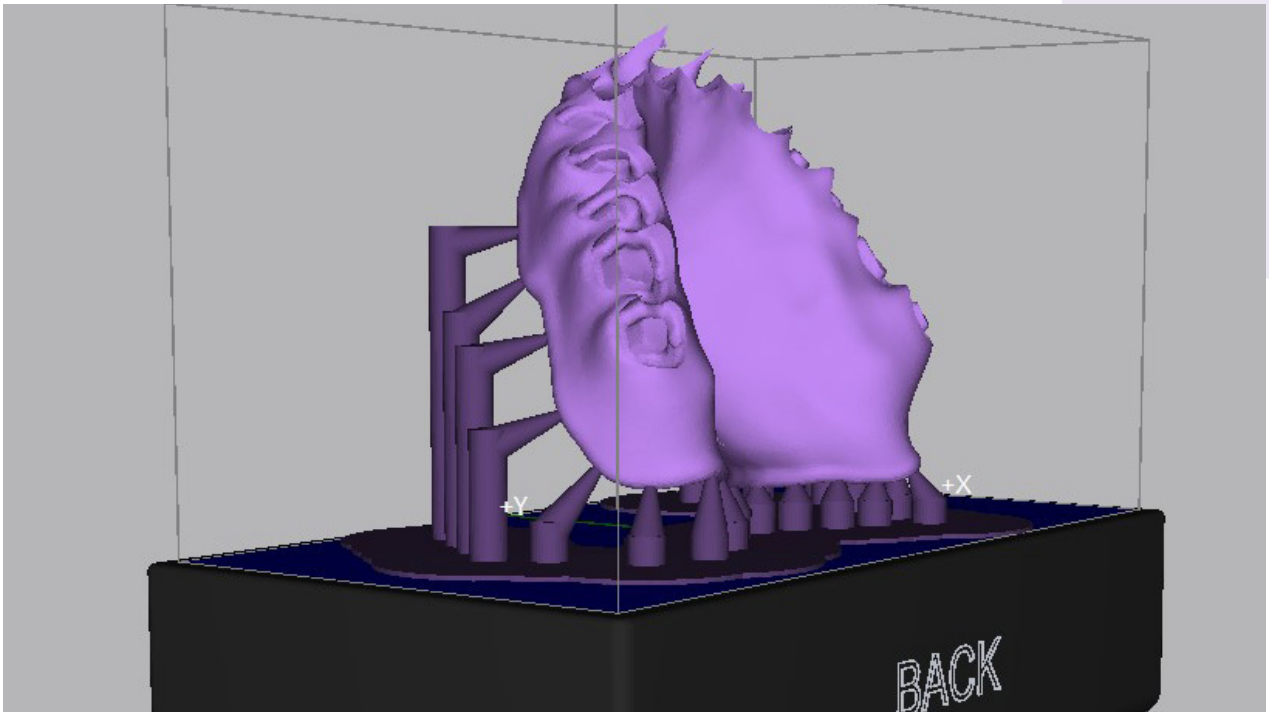
Supports:

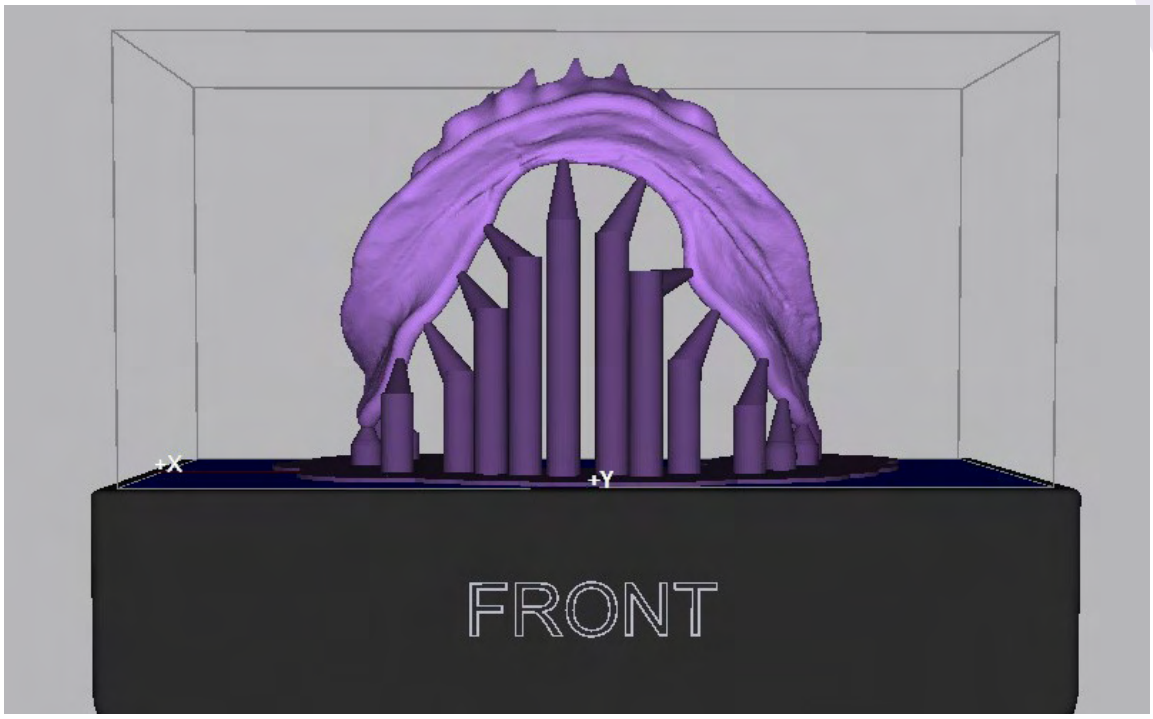
- We recommend manually placing the supports as shown in the photos on the next page. Supports will be on the border of the denture only and will be carried 2/3 up the border of the denture.
- Spacing with “heavy supports” no less than 1mm.
- Density of supports: Between 25%-40%, adjust manually if needed.
- Support point size: 0.4 – 1.0mm.
- Type for all supports: “Heavy”
- If you automatically generate supports (as shown below), you may adjust the supports as needed. You must remove any auto-generated supports affecting the intaglio surface.
- You can always add in supports, adjust the density of supports, and/or remove supports if auto-generation bunches them too closely together.

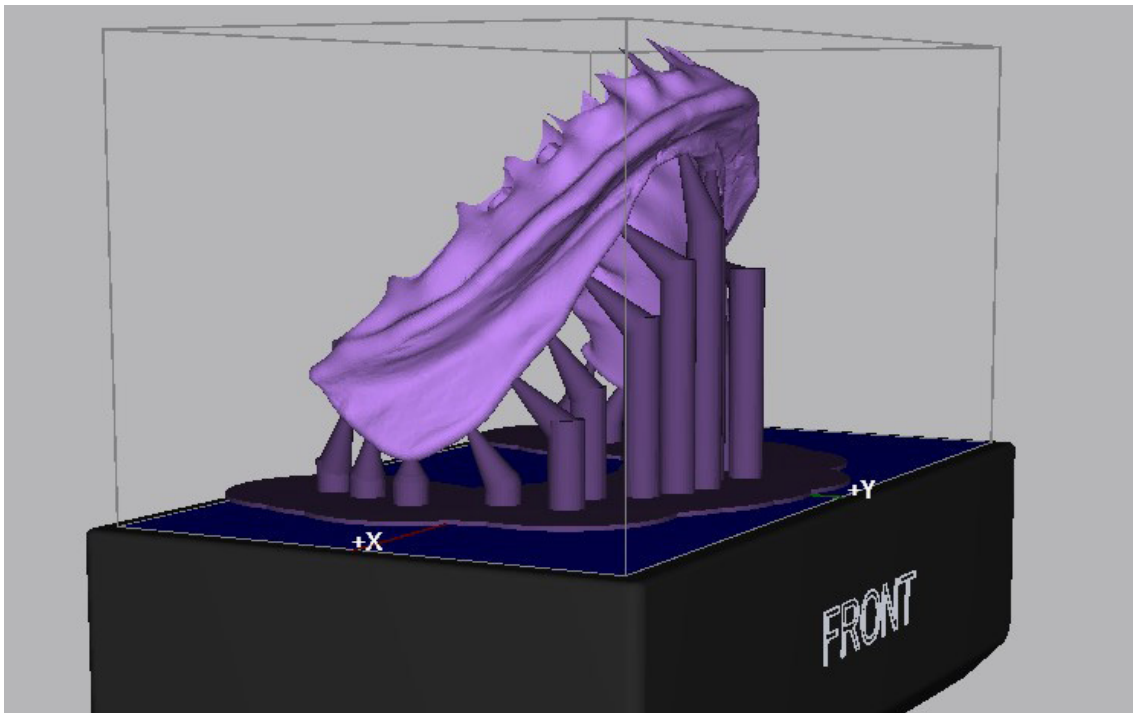


Examples shown below of where to manually place supports:





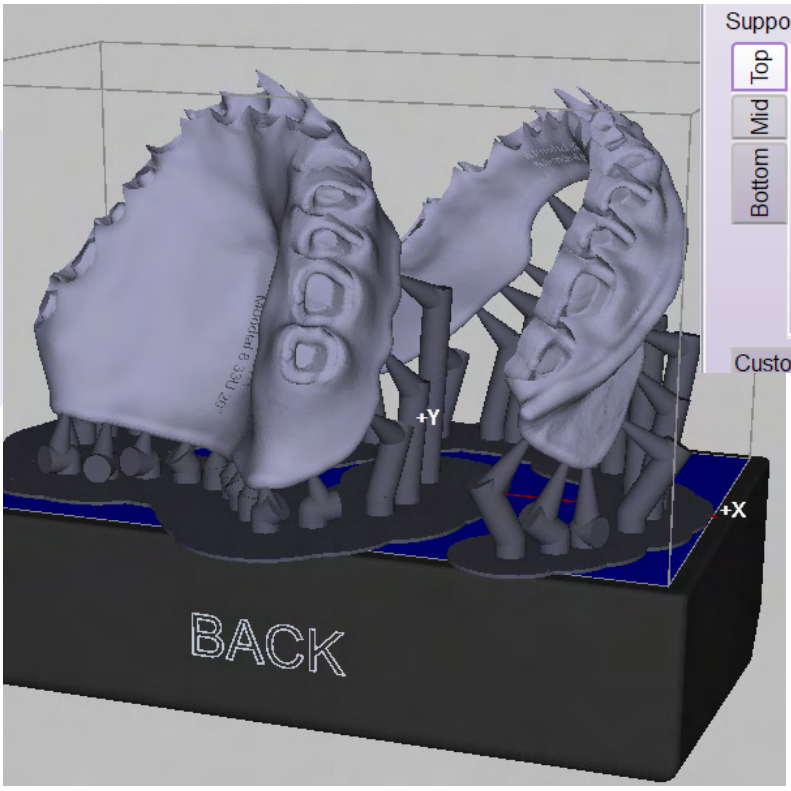


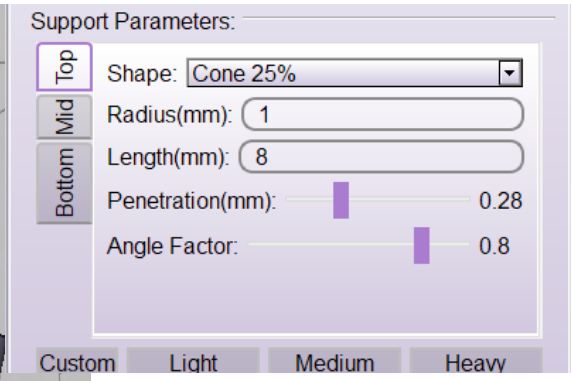


Support Parameters:

Although not required, you may also consider adding a foundation to your denture model. This will help support the model during production. If adding a foundation, it is recommended to use the peel or hash foundation at a thickness of 1.5mm.

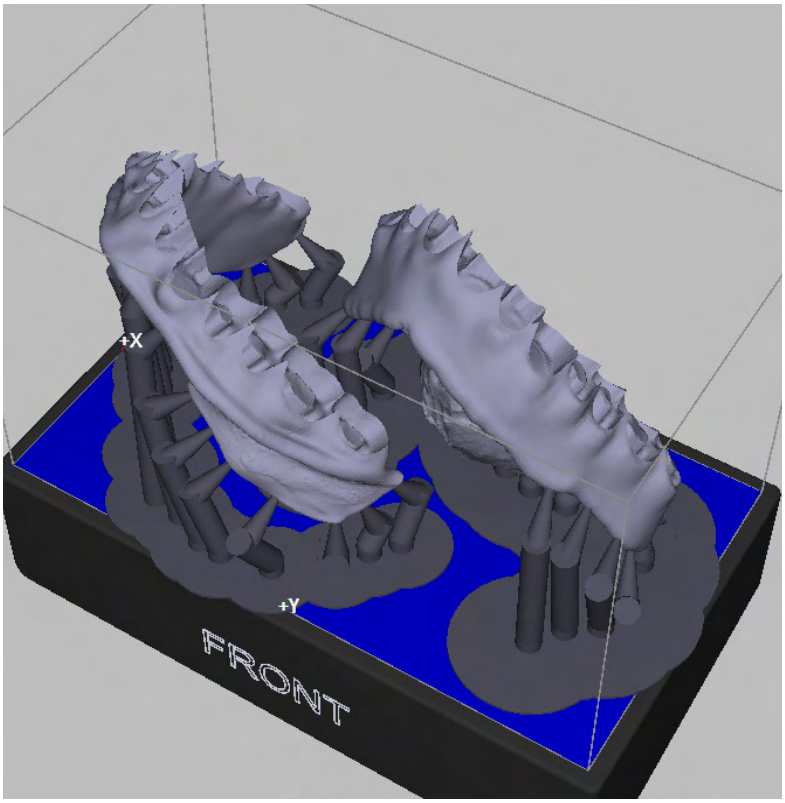
If printing 2 arches on the build table simultaneously, we recommend a similar orientation as shown below.

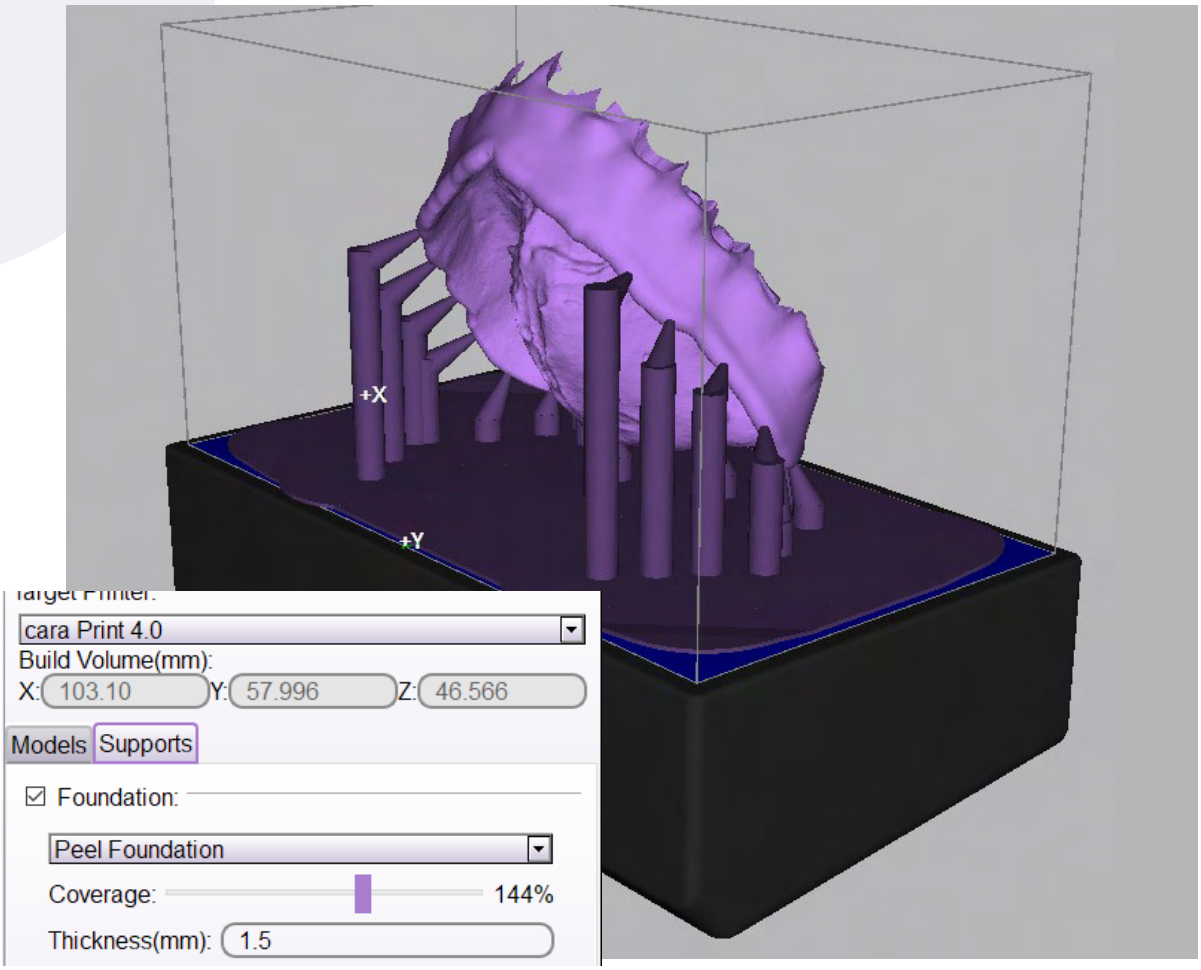




Printing parameters:

- Layer thickness: 100µm
- Optimal orientation: 50 – 60 degree tilted orientation.
- Support density: Between 25%-40%, adjust manually if needed.
- Fill the resin tray at least 1/3 full slowly to cover 85% of the resin tray surface and allow it to spread evenly. Do not overfill the resin tray. Max fill line should not be exceeded.





Foundation:

Not required but if you choose a foundation we recommend the following settings:

- Hash or peel foundation
- Thickness: 1.5mm
- If adding a foundation, once it's set, hit the "Snap to Foundation" button at the top of your CAM screen. This will snap the appliance to the foundation on the build table.

Procedures to fabricate a Denture using printed denture base:

Bonding the printed teeth to the printed denture base:

- Denture Base: After removing from the build table and completing the cleaning process as noted in Step 4.5, leave the denture in its green state (uncured) and remove the supports.
- You are now ready to bond the dima Print Denture Teeth to the dima Print Denture Base. Do not post-cure the base or teeth prior to bonding step. Bonding the teeth to the base is initially done in green-state.
- Prior to post-curing, prepare the printed teeth and printed denture base with socket shapes to receive printed teeth (tooth

sockets).

- Place the dima teeth into the corresponding tooth sockets on the printed denture base and check teeth fitting. Then remove the teeth.
- Apply the small amount of light curable adhesive or resin into the tooth sockets using a small brush or applicator. We recommend you use the same dima Print Denture Base shade for bonding.
- Place the dima Print Denture Teeth into the corresponding tooth sockets on the printed denture base until they are fully seated. Then wipe off any excess material with a paper towel or tissue.
- Use a small flat-headed brush to clear any additional excess base material away from the teeth and gingiva junction (CEJ).
- Lute-in all anterior and posterior teeth on the maxillary and mandibular base using this method.

Post-Curing the Denture base and Teeth Combined:

- Cure the fully printed denture (base and teeth combined) in a glycerin bath. The glycerin acts as an oxygen inhibitor. To do so, simply fill a small glass bowl with glycerin (we recommend a Pyrex 1-cup bowl, 3.5" x 1.7"), place the appliance into the glycerin bowl, and place the bowl onto the model tray inside the reflector pot of the HiLite (see images on page 59). Use enough glycerin to fully submerge the denture.
- Place the denture base with teeth into the glycerin bath inside your HiLite Power 3D with the teeth facing up.
- Set your initial cure time for 10-minutes.
- After the first 10-minute cure ends, open the drawer of the HiLite and let it cool for approximately 3-minutes. Then flip the denture over (teeth facing down) and set your second 10-minute cure cycle (for a total cure time of 20-minutes).
- Once curing is complete, wipe down the denture with a paper towel to remove the glycerin and use compressed air to blow out any sockets or crevices where the glycerin may remain. You may also rinse the cured appliance under water to remove any residual glycerin after post-curing is complete.

Finishing the Denture base and Teeth Combined:

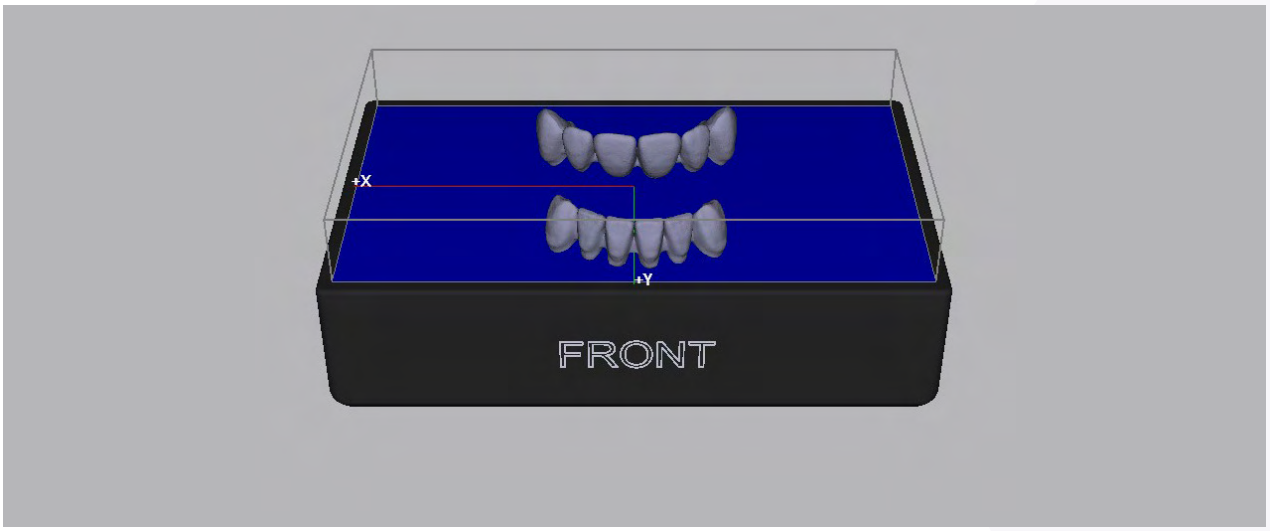
- Once post-curing is complete, grind the support spots and borders of the fabricated denture and teeth to make it smooth.
- Pumice and polish the denture with a polishing unit using desired pumice.
  We recommend using Pala® Polish for achieving a high shine for the fully printed denture.
- To further characterize the denture gingiva (festooning, stippling, etc.), we recommend using Pala® cre-active.

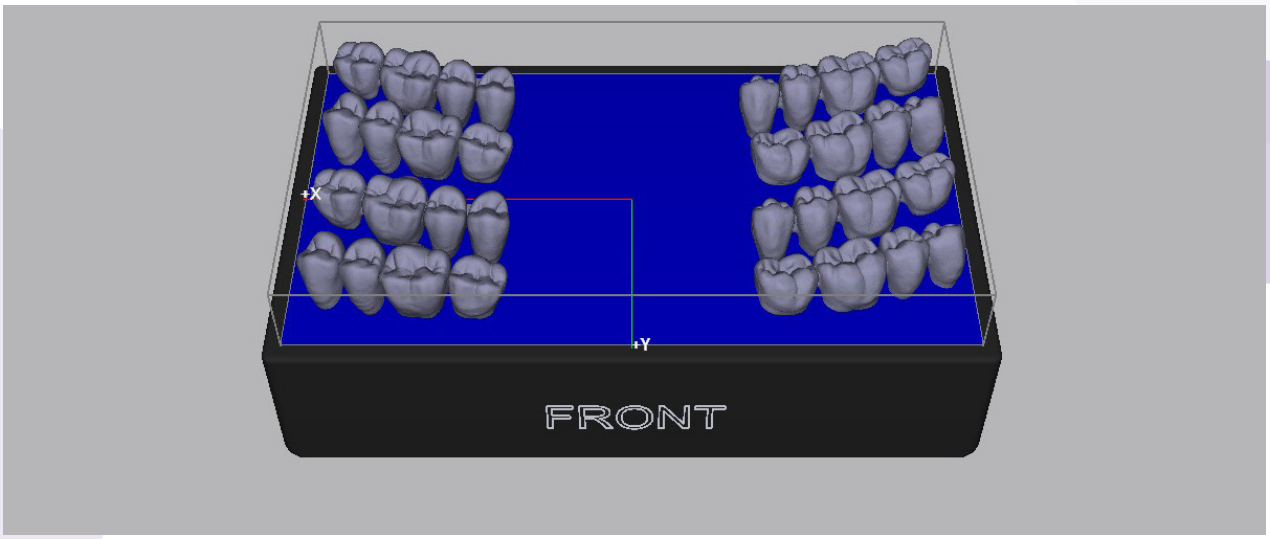
! CAUTION: THE GLYCERIN WILL BE VERY HOT AFTER CURING. PLEASE USE AN INSTRUMENT TO FLIP THE APPLIANCE AND TO REMOVE THE APPLIANCE FROM THE HOT GLYCERIN ONCE CURING IS COMPLETE!

5.10. dima Print Denture Teeth

Recommended settings:

- Minimum Thickness: No minimum thickness required for denture teeth.
- Position the anterior and posterior teeth on the build table with the basal surface facing down toward the build table (see below images for reference).
- The recommended orientation for anterior and posterior teeth are shown below.
- After you've oriented your teeth on the build table, hit "Snap to Floor" using the Snap to Floor button at the top of your CAM screen.



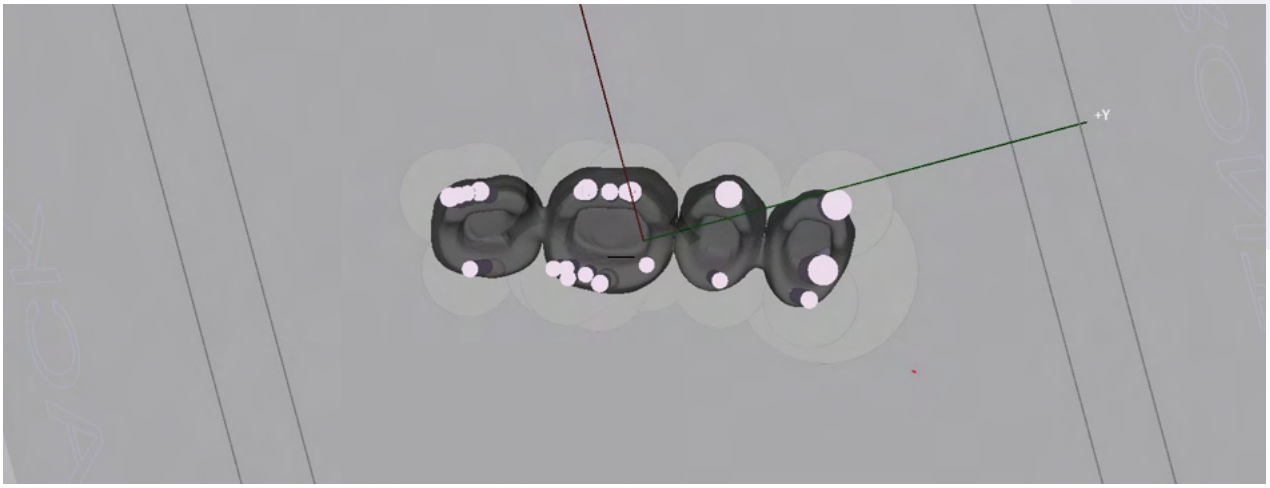


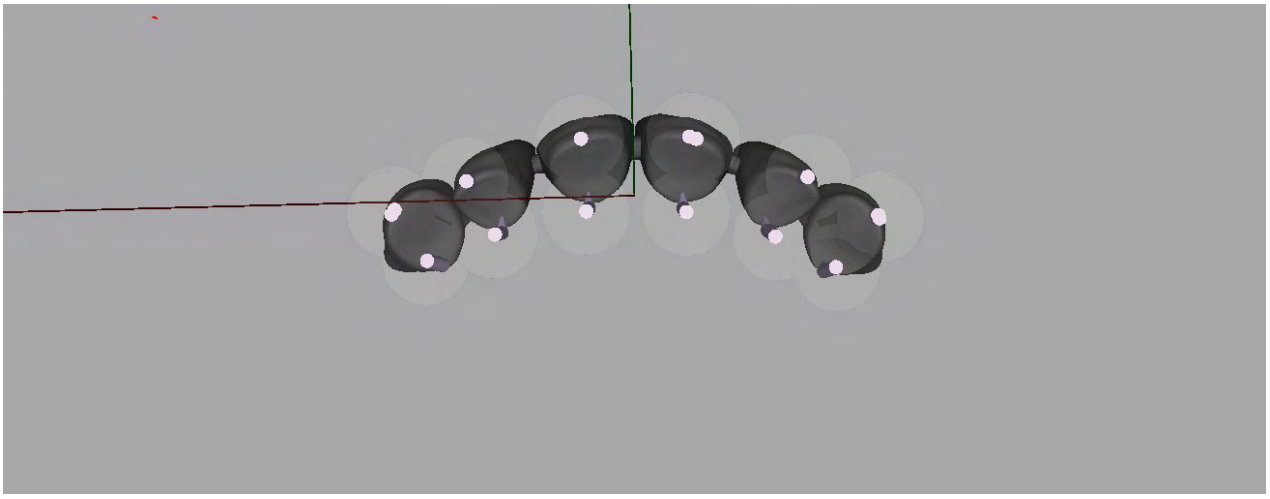
Angle (print positioning/orientation):

- After the STL file is imported, rotate the appliance to fit on the build table in cara CAM, utilizing recommended settings listed above.

Supports:

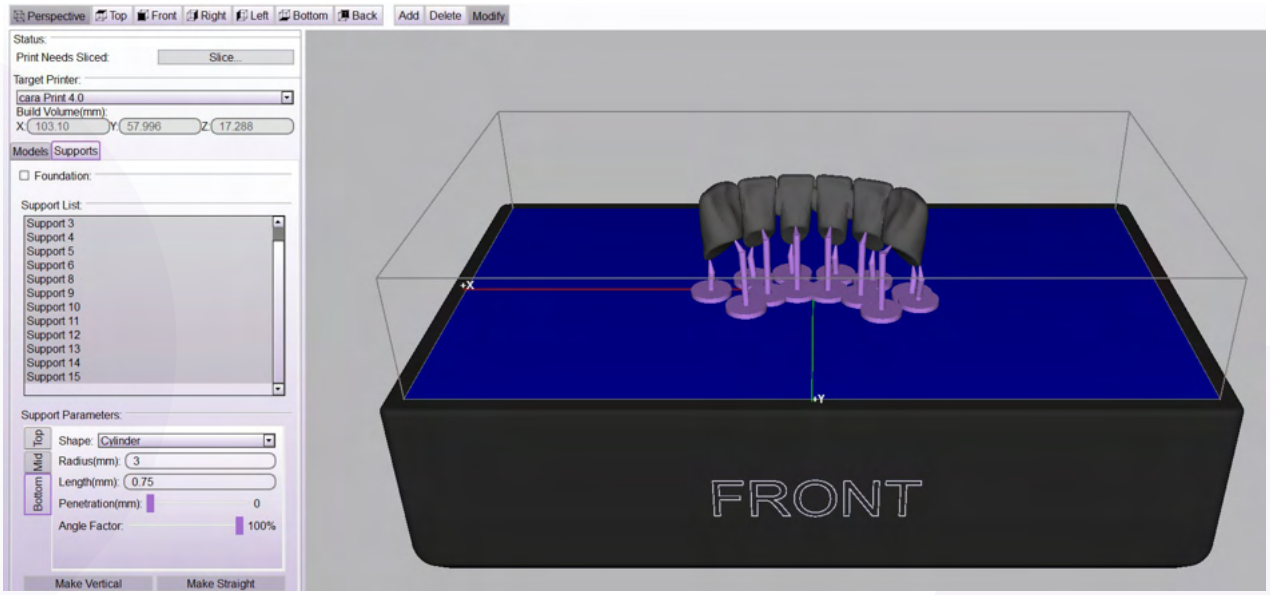
- For both anterior and posterior teeth, place “light” supports all around the border of the Basal surface of the teeth (see partial support example photo below).
- We recommend placing supports individually using the add support feature in cara CAM.
- If you choose to auto-generate supports, we recommend you use 30% support density with light supports.

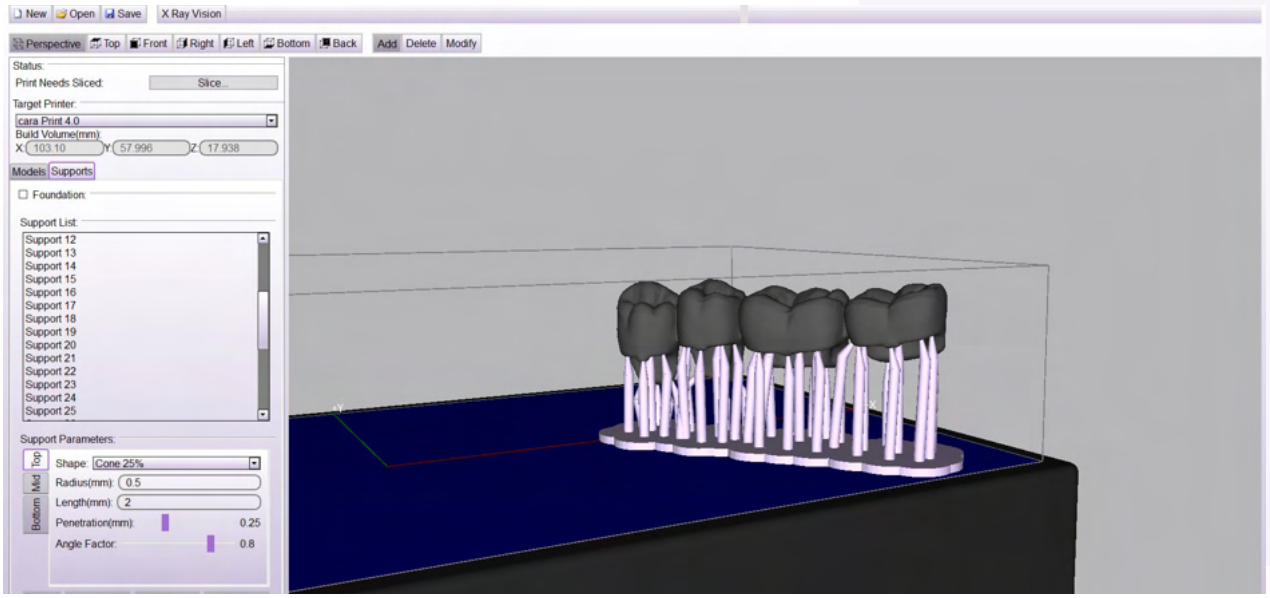


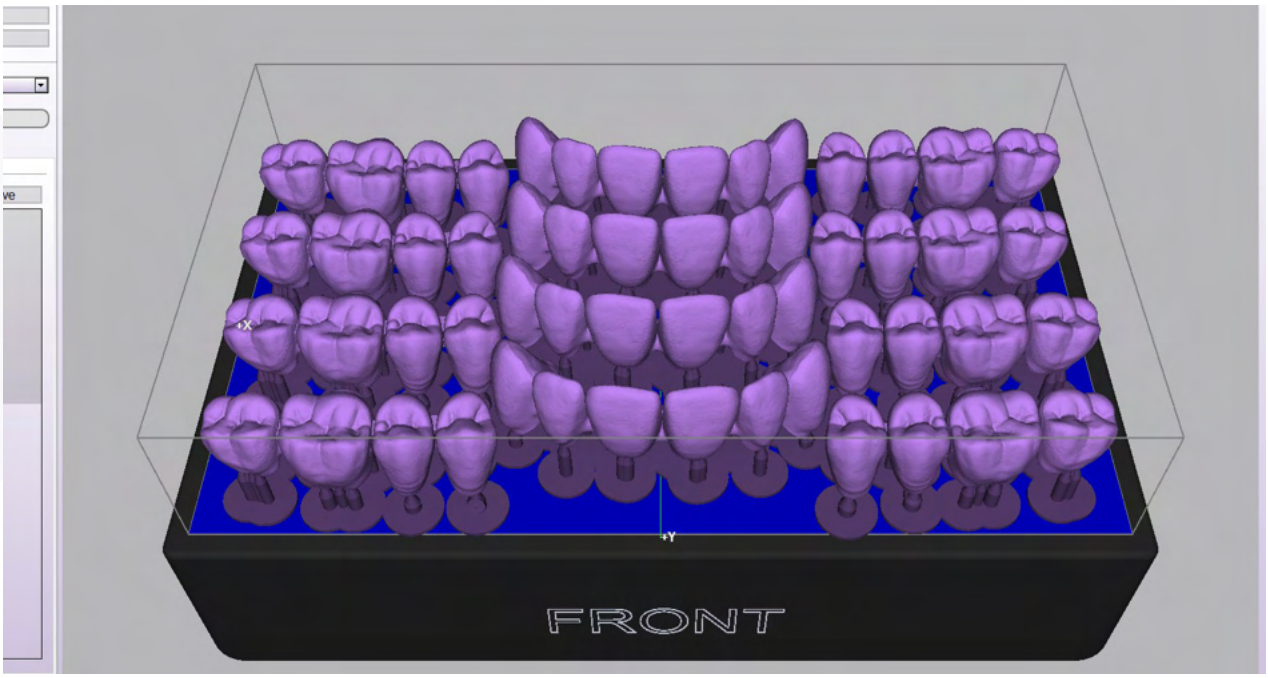


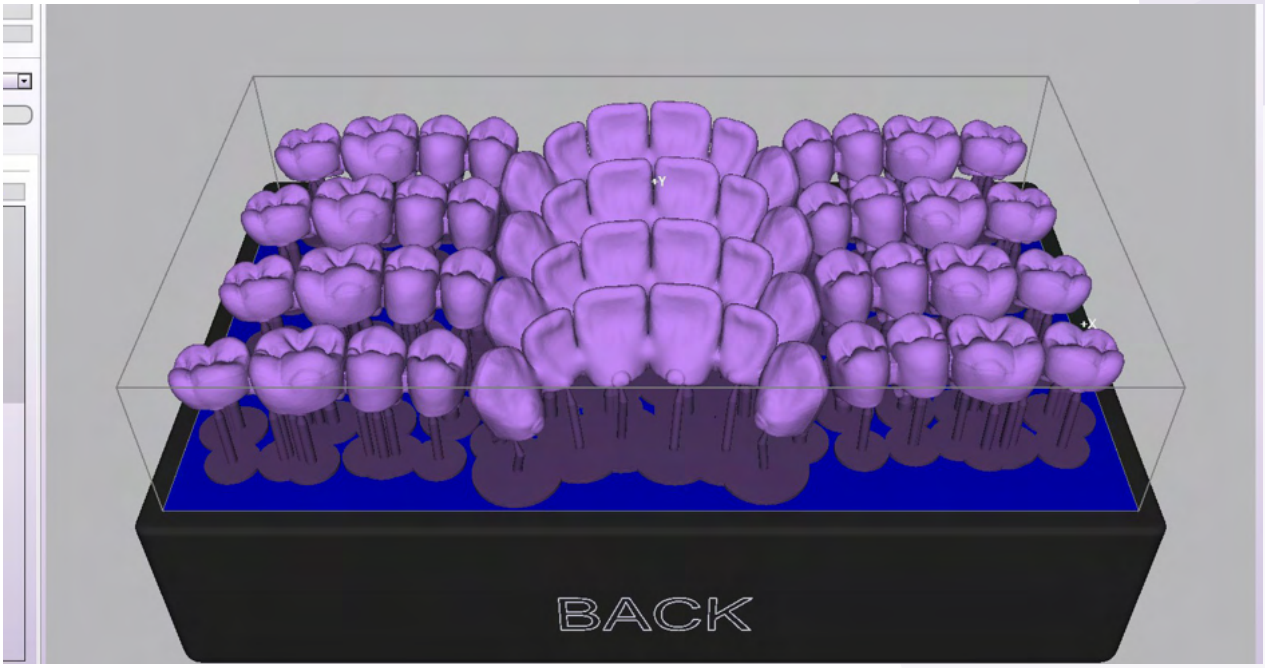
Support Parameters:

- Form of the tip on the printed teeth” Cone 25%” (for easier removal after printing).
- Only “light supports” needed.
- Adding a foundation is not necessary for printed teeth.
- We recommend you click the “bottom tab” under Support Parameters, highlight all of your supports in your support list, and increase the length of the teeth bases to 0.75mm (see top image on page 65).


Perspective
Top
Front
Right
Left
Bottom
Back
Add
Delete
Modify
Status:
Print Needs Sliced:
Slice...
Target Printer:
cara Print 4.0
Build Volume(mm):
X: 103.10
Y: 57.996
Z: 17.288
Models
Supports
Foundation:
Support List:
Support 3
Support 4
Support 5
Support 6
Support 8
Support 9
Support 10
Support 11
Support 12
Support 13
Support 14
Support 15
Support Parameters:
Top
Mid
Bottom
Shape: Cylinder
Radius(mm): 3
Length(mm): 0.75
Penetration(mm): 0
Angle Factor: 100%
Make Vertical
Make Straight
FRONT


New
Open
Save
X Ray Vision
Perspective
Top
Front
Right
Left
Bottom
Back
Add
Delete
Modify
Status:
Print Needs Sliced:
Slice...
Target Printer:
cara Print 4.0
Build Volume(mm):
X: 103.10
Y: 57.996
Z: 17.938
Models
Supports
Foundation:
Support List:
Support 12
Support 13
Support 14
Support 15
Support 16
Support 17
Support 18
Support 19
Support 20
Support 21
Support 22
Support 23
Support 24
Support 25
Support Parameters:
Top
Mid
Bottom
Shape: Cone 25%
Radius(mm): 0.5
Length(mm): 2
Penetration(mm): 0.25
Angle Factor: 0.8


FRONT


BACK

Post-Curing the Denture base and Teeth Combined:

- **Denture Teeth:** After removing from the build table and completing the cleaning process as noted in Step 4.4, leave the denture teeth in its green state (uncured) and remove the supports
- Following the bonding steps listed at the end of the Denture Base section, use the same dima Print Denture Base shade as the printed base to paint into the sockets of the printed denture
- Place the dima printed teeth into the socket of the denture until they are fully seated. Then wipe off any excess material with a paper towel or tissue.
- Use a small flat-headed brush to clear any additional excess base material away from the tooth and Gingiva junction (CEJ)
- Cure the fully printed denture (base and teeth combined) in a glycerin bath. The glycerin acts as an oxygen inhibitor. To do so, simply fill a small glass bowl with glycerin (we recommend a Pyrex 1-cup bowl, 3.5" x 1.7"), place the appliance into the glycerin bowl, and place the bowl onto the model tray inside the reflector pot of the HiLite (see images on page 59). Use enough glycerin to ensure the denture is fully submerged.
- Place the denture base with teeth into the glycerin bath inside your HiLite Power 3D with the teeth facing up.
- Set your initial cure time for 10-minutes.
- After the first 10-minute cure ends, open the drawer of the HiLite and let it cool for approximately 3 minutes. Then flip the denture over (teeth facing down) and set your second 10-minute cure cycle (for a total cure time of 20 minutes).
- Once curing is complete, wipe down the denture with a paper towel to remove the glycerin and used compressed air to blow out any sockets or crevices where the glycerin may remain. You may also rinse the cured appliance under water to remove any residual glycerin after post-curing is complete.
- Grind the support spots and borders of the fabricated denture to make smooth.
- Pumice and polish the denture with polishing unit using desired pumice. We recommend using Pala Polish for achieving a high shine for the denture.

! CAUTION: THE GLYCERIN WILL BE VERY HOT AFTER CURING. PLEASE USE AN INSTRUMENT TO FLIP THE APPLIANCE AND TO REMOVE THE APPLIANCE FROM THE HOT GLYCERIN ONCE CURING IS COMPLETE!

Post curing and Finishing dima Print Denture Teeth only:

- Curing time for printed teeth sets is 10 minutes per side (20 minutes total) in a glycerin bath.
- After curing, remove the supports and grind away any remaining support spots.
- Pumice and polish the denture teeth with a polishing unit using your preferred pumice. We recommend using Pala Polish for achieving a high shine on your printed teeth.

dima Print C&B temp

With dima Print C&B temp you can produce temporary crowns and bridges

1. Design notes

The recommended settings and design recommendations here only apply when used in combination with 3Shape. You can download the parameter as a DME file to import into 3Shape in the cara download area:

kulzer.com/cara-print-3shape-dme

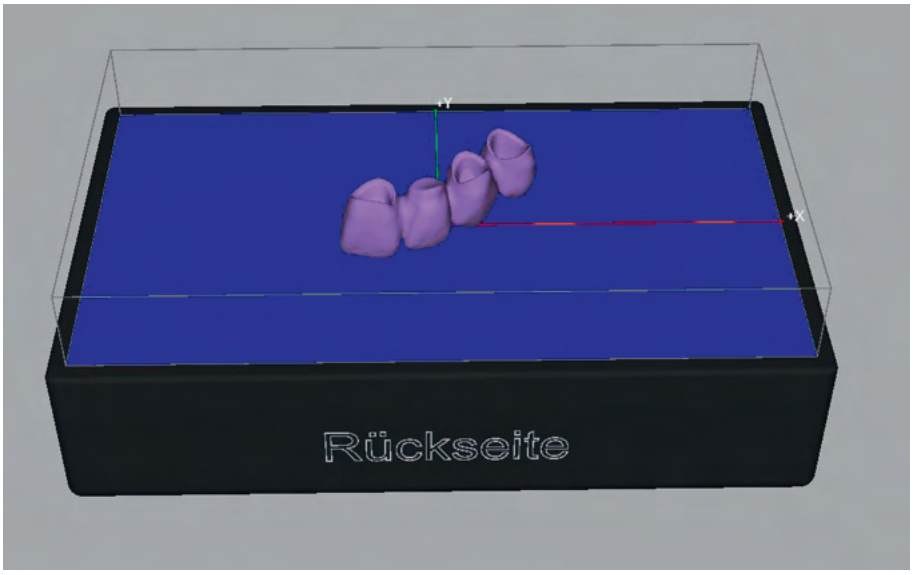
Register and get your free CAM licence!

2. Recommended setting

- Minimum wall thickness: 2 mm
- To maintain a certain wall thickness is especially recommended in the occlusal area and the connection area of units; The edge of the crowns can be designed tapering thinly
- Anterior bridges up to 6 units and posterior bridges up to 4 units

3. Print position (angle and orientation)

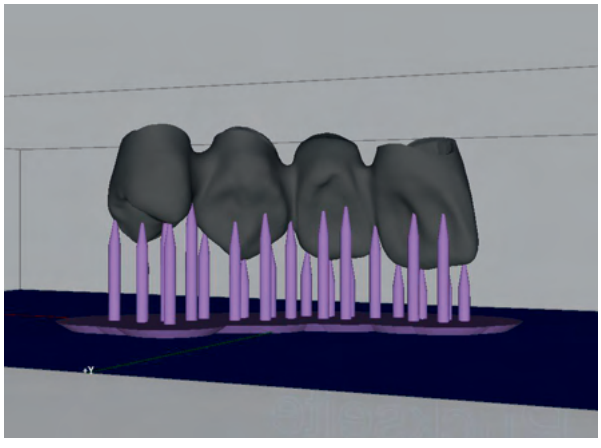
- Position crowns/bridges on the build table with the basal surface facing up toward the build table (see below images for reference).
- Distance from appliance to building platform min. 5 mm

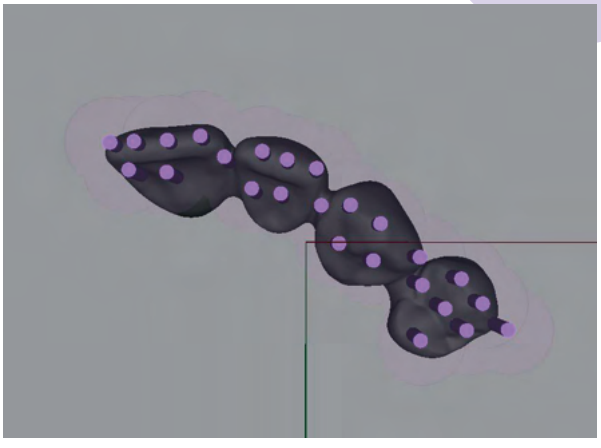


Place the intaglio surface upwards as shown

- We recommend to then manually add supports, also on the connector of each unit (5–8 in total, depending on the size and shape of the tooth)
- Supports Top shape should be ”Cone 25 %” and support bottom “Cone 75 %” (for easier removal after printing)

Light support parameters	
Top	
Shape	Cone 25 %
Radius (mm)	0.5
Length (mm)	2
Penetration (mm)	0.35
Angle Factor	100 %
Mid	
Shape	cylinder
Radius (mm)	0.5
Bottom	
Shape	Cone 75 %
Radius (mm)	5
Length (mm)	0.75
Penetration (mm)	0
Angle Factor	100 %

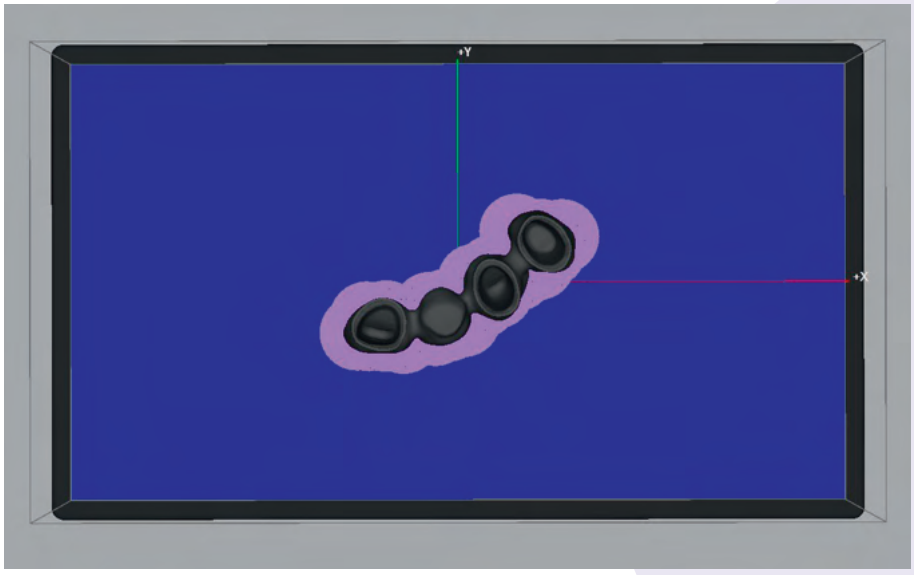




Full anatomic anterior bridge with fine supports, seen from the side and below

4. Foundation

- Adding a foundation is not necessary for printed crowns and bridges.



Full anatomic 4 unit bridge with fine supports, seen from above

5. Printing

- Files can be printed in 50 µm and 100 µm layer thickness

6. Cleaning and Postcuring

Cleaning recommendations:

- Before the usual cleaning, we recommend using compressed air to blow out any remaining liquid monomer inside the cavities.
- Remove supports before postcuring.

Curing recommendations:

- dima Print C&B temp must be post-cured with an oxygen inhibitor (glycerin) to remove unreacted functional groups and create harder surfaces. We recommend using 99 % glycerin.

	Cleaning time in Isopropanol (cara Print Clean or ultra sonic bath)	**Post-curing time** **(HiLite Power 3D)**	**Post-Curing time LEDcure** (no turning of object necessary)
dima Print C&B temp	pre-cleaning: 3 minutes post-cleaning: 2 minutes	20 minutes (10 minutes front side +10 minutes reverse side) in Glycerin	C&B Temp program, cure in Glycerin

Using glycerin in Kulzer light curing devices:

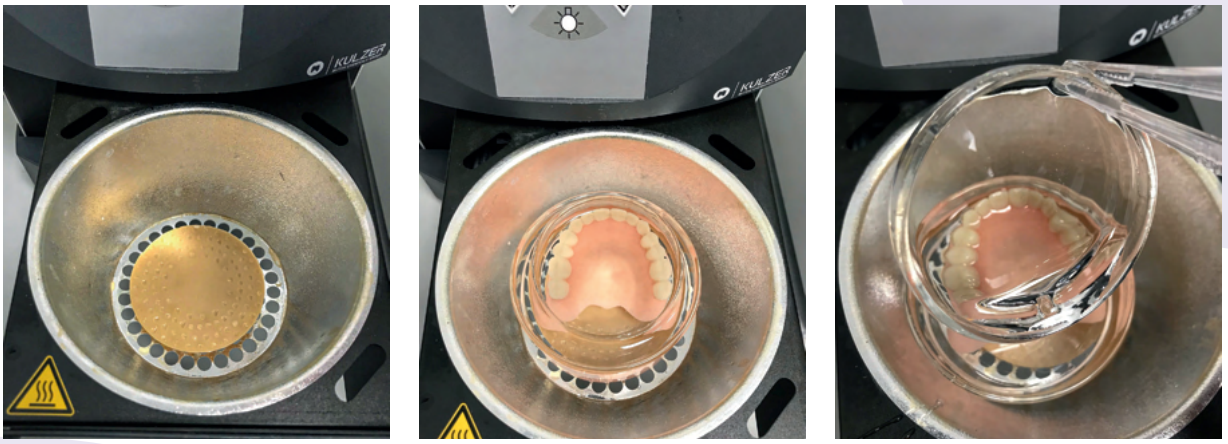
Glycerin is a non-flammable liquid that can be used in Kulzer light curing devices when handled with care:

- To cure the objects in glycerin you need an up to 100°C heat-resistant glass bowl with a glass lid that fits in your light curing device. You can use microwave glassware, Pyre glas or tempered glass for laboratory equipment but also most of normal glass table ware will be sufficient as well.
- The printed object must be covered with glycerin completely
- Glycerin can be reused and only needs to be changed when getting foggy

Special information for HiLite power 3D users:

- Never forget to use the model tray of your HiLite power 3D below the glass bowl. This prevents the curing device from overheating
- Open and close the curing department door carefully so that no glycerin spills inside the device
- We recommend a cool-down time for the HiLite Power 3D of 2–3 minutes after each 10 minutes operational cycle.
- Maximum height of the glass bowl 5 cm, maximum width 10 cm at the bottom

CAUTION: THE GLYCERIN WILL BE VERY HOT AFTER CURING. PLEASE USE AN INSTRUMENT TO FLIP THE APPLIANCE AND TO REMOVE THE APPLIANCE FROM THE HOT GLYCERIN ONCE CURING IS COMPLETE!



Use the model tray to carry the glass bowl for optimal ventilation

- Once curing is complete, wipe down the appliance with a paper towel to remove the glycerin and use compressed air to blow out any sockets and crevices or rinse the cured appliance under water.

7. Finishing

- Once post-curing is complete, grind the support spots and borders of the restorations.
- Polish the external surface of the crowns/bridges with pumice; then switch to a dry lathe disc and polishing paste for a glossy finish

8. Cementation

- Crowns and bridges printed with dima Print C&B temp can be cemented using standard dental temporary cements
- If the crown/bridge requires adaptions for fitting, it can be easily grinded, but also relined with conventional auto cure acrylic

Available shades	
dima Print C&B temp	
7 shades	
	A1
	A2
	A3
	A3.5
	B1
	B2
	BL1

Stay competive, deliver quality:

with Kulzer's full range of 3D-Printing solutions.

kulzer.com/cara-print

dima Print Digital Denture:

Stress-free digital dentures with 3D printing

dima Print Stone beige:

3D print all dental model types with
a traditional look & feel.

dima Print Stone teal:

The color for orthodontics: less eye strain
and high heat resistance.

NOTES

Distributed by:
Kulzer, LLC.
4315 S. Lafayette Blvd
South Bend, IN 46614
(800) 431-1785

Manufactured by:
Kulzer GmbH
Leipziger Straße 2
63450 Hanau, Germany
cara-service@kulzer-dental.com

CARA_AG_1 06/2021

kulzerUS.com/caraprint