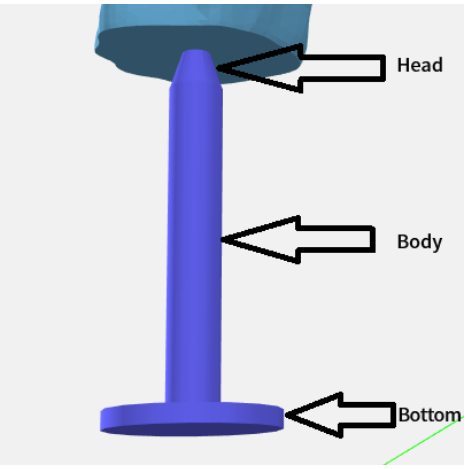
Some relationship between SLA/DLP/LCD printing success rate and support

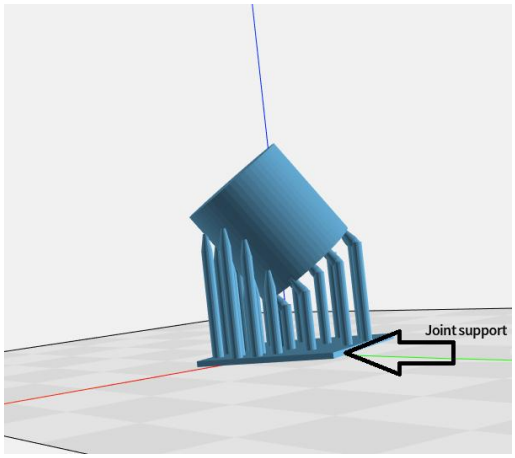
In 3D printing, we usually add support structures to the dangling and cross-bridge to improve the print success rate. Setting the right support parameters is the half success of printing. But, you know the resin 3D printing (SLA/DLP/LCD), how to set the supporting parameters is the most reasonable? For different slicing software, the name of support parameters is different. In general, they are all similar. In order to grasp the support settings better, take GEEETECH WEBGUI as an example to analyze parameters related to the supporting structure step by step.

The GEEETECH WEBGUI support parameter name is shown below:



I. Joint support:

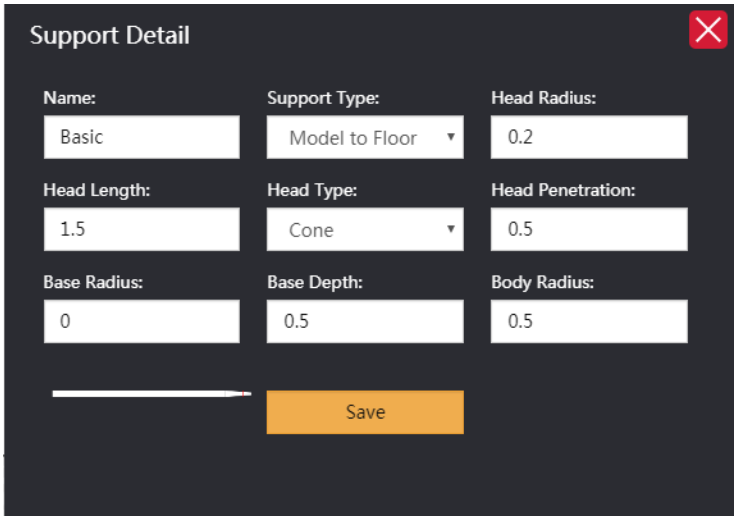
The joint support is first cured in printing and its primary purpose is to increase adhesion to the forming platform. The bottom raft is printed directly on the forming platform, followed by supports and models. The exposure time of the bottom raft is recommended to be at least 8 times that of the normal exposure time, so that any possible gap between the forming platform and the trough contact surface is sufficiently cured to make sure the first layer is firmly attached to the forming platform.



Support Type:

Model to Floor

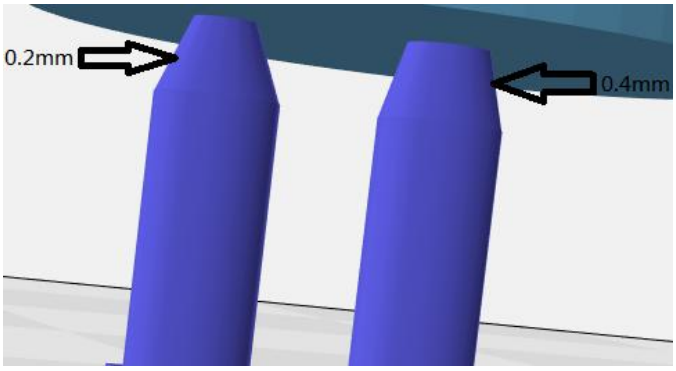
- **Head:** The head refers to the top of the support structure. The top is a key part of the connection model and support, and strength is important. If the strength of the top of the support is not sufficient, the 3D model may tear the model off the support due to the separation force generated by the release film on the model.



1. Head Radius

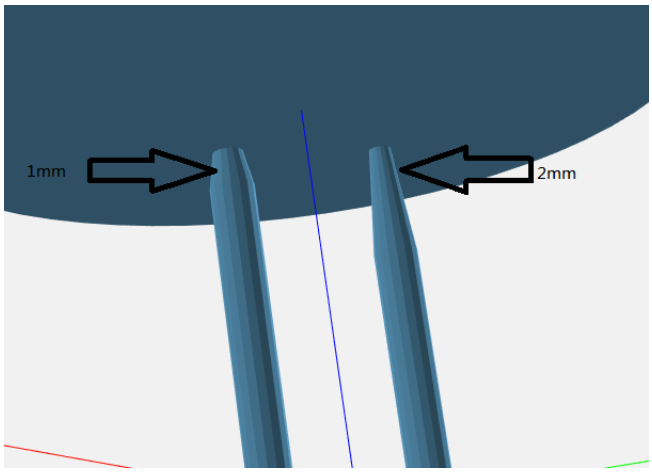
Directly reacts on the contact area of the model and support. The longer the diameter of the contact shape, the larger the contact area between the support head and the model,

and it is more troublesome to cut off the support with sharp-nose pliers.



2. **Head Length**：The head length can be adjusted.

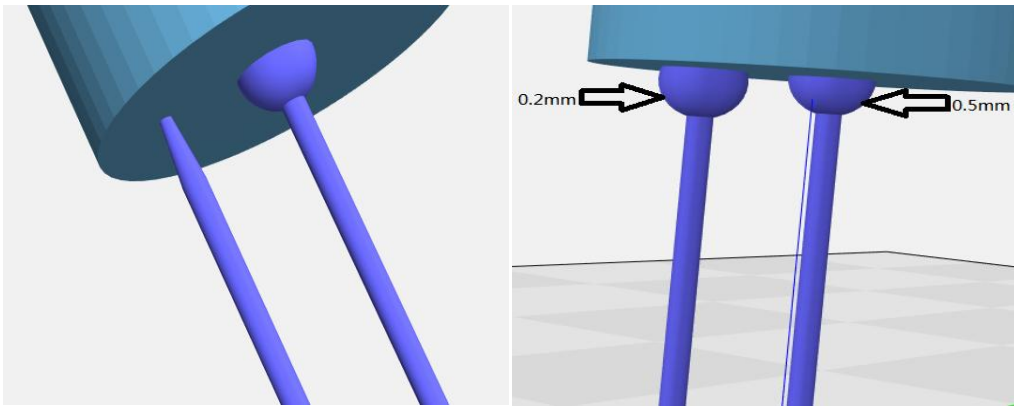
Connections of the right length make the support top more stable and easier to remove.



3. **Head Type**

The spherical contact is a small sphere that is used to increase the joint between the model and the support. In addition, when the support is removed, the scorpion can be cut from the ball and the support connection, so that the model is not easily broken. Some rigid resins are more brittle, and when cut with tweezers, they are easily attached to the surface of the model, leaving holes in the model.

Head Penetration：The deeper the penetration, the deeper the depth of the support head into the model.



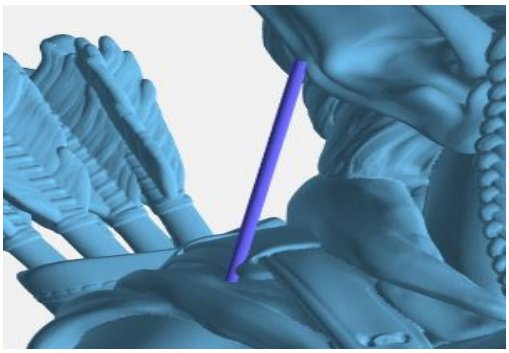
（Left: cone, right: sphere）

- **Body :** Support Size

The middle of the support is like the spine of the human body. The strength of the middle has a great influence on the strength of the support structure.

Body Radius: Support Size. As the "spine" of the support, the thicker the body diameter, the higher the support strength. In the same way, the more material consumption will be. It should also be noted that the middle part needs to maintain a certain proportion of harmony with the top, not too thick or too thin.

Head Penetration：The deeper the penetration, the deeper the depth of the support head into the model



(Bridge: model to model support)

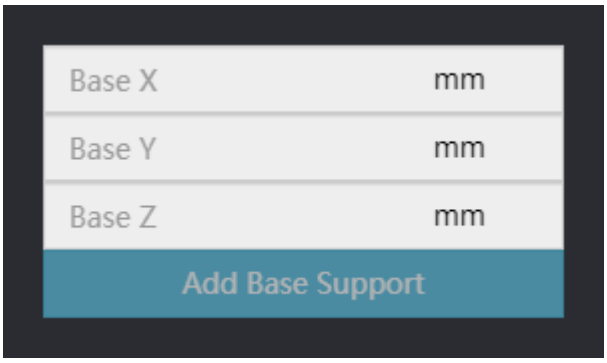
- **Base Support:**

The bottom is the base of the support. In the absence of joint support, the bottom is the layer that is used to improve the adhesion of the platform first. In addition, when it is necessary to generate a support on the model body, the effect of the bottom is large.

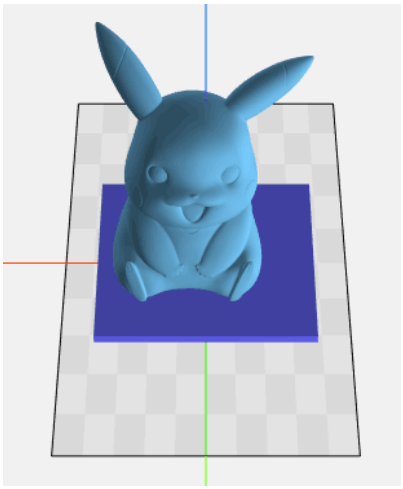
Base size: Base X, Base Y

The bottom dimensions are similar to the joint support and are used to increase adhesion on the forming platform, However, the larger the attachment area, the greater the difficulty in material consumption and eradication.

Base Depth (Base Z): Bottom thickness of the model support

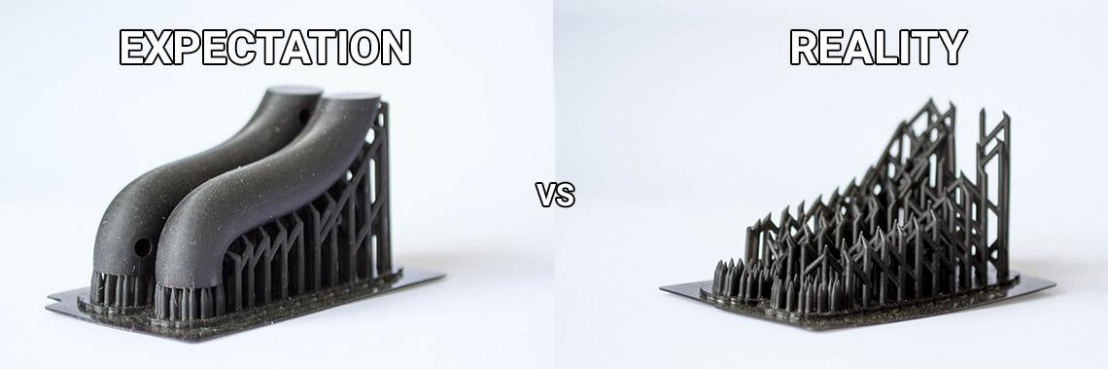


Clicking Add Base Supports after entering the value in the input box will form the corresponding bottom support on the preview platform



II. Some relationship between print success rate and support:

We found that for most "pull-up" LCD (stereoscopic light curing) 3D printer beginners, there is a common problem. That is, in 3D printing, only the support is printed on the molding platform, not the model. Today, we will explore this very common problem when using LCD 3 D printers. This problem is common for beginners, but for some experienced 3D printing users, it will also appear



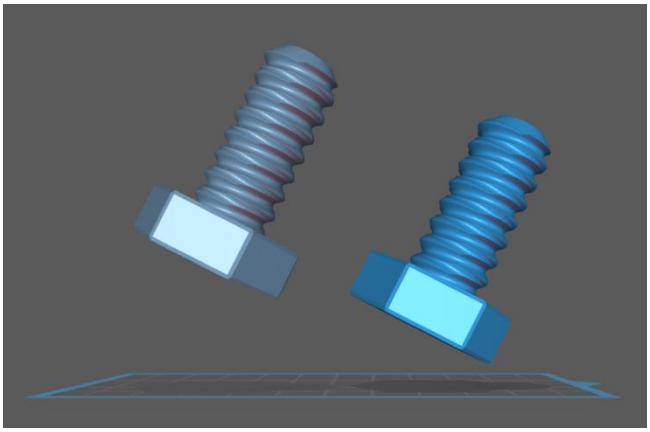
By understanding the underlying causes of this phenomenon, we can easily solve this problem with simple techniques. Now let's discuss it in more detail. Why do you need support?

Before delving into this topic, we first discuss why you should add support when using a photo-curing 3D printer. There are some details that are worth knowing. These are not all, but according to our research, we have listed the most important ones.

- **Suspended condition:** Obviously, support is needed. This is common in most 3D printing, not just light curing.
- **Some of the bottoms are unique in geometry:** Over-cure of the underlying layer of the model is very common in order to better adhere to the forming platform. However, the molding quality may vary due to the increased exposure time. In order to avoid these problems, the model will generally be lifted and supported, and the bottom of the model will not be destroyed when the model is taken.
- **Keep the cross-sectional area uniform when the print direction changes:** It is important to maintain a uniform transition of the cross-sectional area of the

layer. This will help avoid the apparent striations. The increase in the cross-section of the support helps to alleviate the adverse effects of sudden changes in the area and also contributes to a more stable formation.

- **Appropriate placement angle**: The popular "pull-up" printer DP200 (Geeetech) has FEP (Fluorinated ethylene propylene) or PDMS (polydimethylsiloxane) resin trays. After each layer is cured, the separation force generated by FEP or PDMS applies a higher tensile force to the model. May cause various deformations, size errors, faults, etc. By choosing the right placement angle and adding finer support points, you can reduce the possibility of failure.

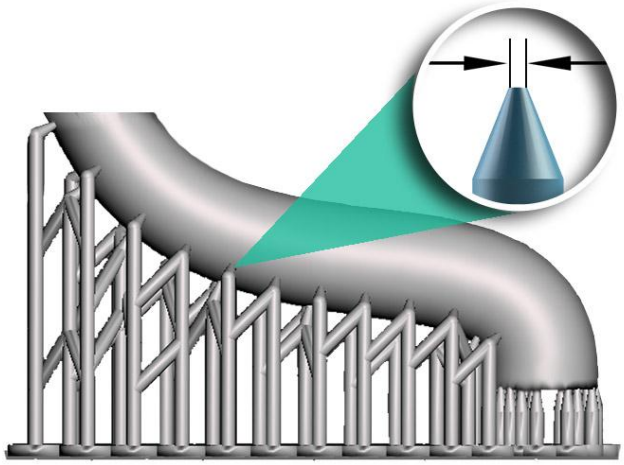


So, when you take all the factors into consideration, you might generate a series of support around the model.

III. Potential risks and problems:

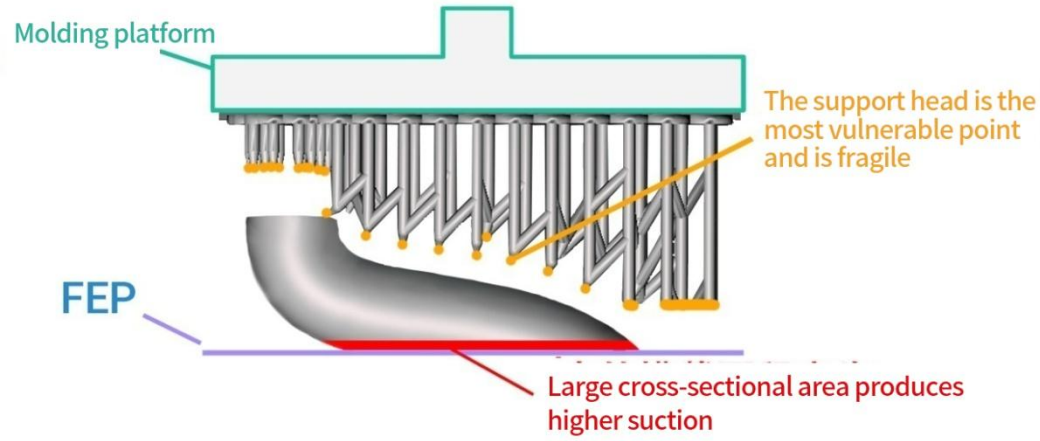
In the techniques mentioned earlier, there is a potential risk of failure even if the entire model is placed on a support. In addition to the many parameters associated with a single

support, the key parameters are usually the diameter and depth of the support head.



The diameter of the support head (usually less than 0.3 mm) is smaller, making it easier to remove the support and leaving fewer marks on the surface, but also more fragile. Conversely, thicker support heads (usually greater than 0.4 – 0.5 mm) are more powerful, but leave more visible marks after removing. The size of the support head is usually set by the model itself, That is, to peel off the newly generated printing layer from the FEP film. The forces that occur during the stripping are very large. They depend to a large extent on the cross-sectional area of the layer: the larger the area, the greater the force of the peeling and the larger the diameter of the support head.

If the printed model is peeled off from the FEP film, the force of the peeling is much greater than the force that holds the model on the support, and this fails. Keep in mind that during 3D printing, the model is formed by layer-by-layer stacking, and sometimes there may be thousands of layers, each layer exerting a certain amount of pressure on all layers previously printed. See below.



This is what we have to avoid. Look at some possible solutions.

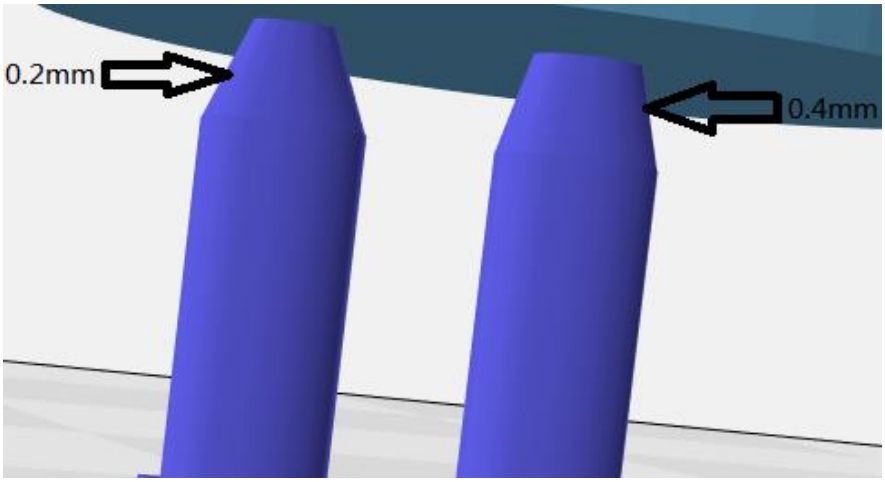
IV. Some relationship between print success rate and support:

Based on the causes of these problems mentioned above, it is natural to think of possible ways to solve this problem. It can be considered that increasing the density of the support or the diameter of the support head helps to avoid printing failure. This is partly true, but the answer is not so simple.

It is worth mentioning that most people choose their own way based on past experience and personal preferences. There is no universal or truth-like approach here. Every 3D print model has a slight difference and requires careful analysis to achieve high success rates. Therefore, we will also share our experience and possible suggestions. However, these also need to be self-identified to suit different situations. Let's look at the solution from a different perspective.

1. Thicker support head

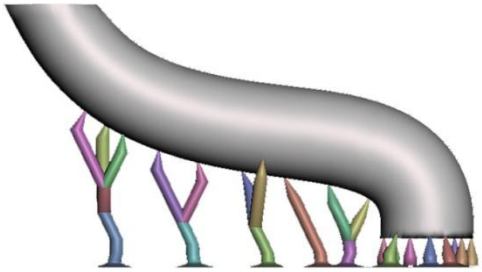
We usually recommend thicker support heads, which are about 0.4-0.5 mm on average, rather than thinner (usually 0.2-0.3 mm). According to our experience, if you want to print quickly with a higher lifting speed, it is more secure with a thicker support head.

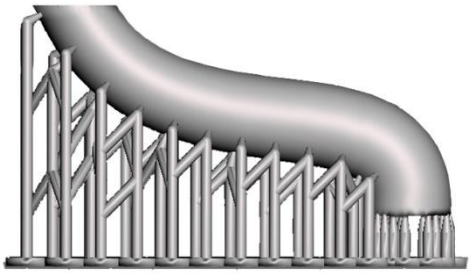


However, in some cases, a thinner support head is still needed. For example, the model has many small hanging points, and the thicker support head will affect the forming quality of the contact point.

2. High/medium density support

If you use a support, the surface must have visible marks. Then why not need more support? If you want the surface to be smooth, you can polish it. Therefore, sometimes adding more support does not add too much grinding work to yourself, but the success rate can be significantly improved. At the same time, you can add more support to surfaces that are less demanding on smoothness. In this way, the success rate will be greatly improved. However, as the density increases, the consumption of materials will also increase, and it will be more troublesome to remove the support.





3. Consider resin properties

The choice of resin is also an important part. For example, the use of harder resins can provide better model details. These resins are commonly used to print models with complex details, and the support required for these models is fine and the support head is smaller. Harder 3D printing resins do not undergo severe deformation during FEP film separation. This rigidity ensures that even if the model features are small, they can be retained. In the same way, the requirements of support and exposure of elastic materials are often different from ordinary resins. On the other hand, flexible resins also have different needs. This resin will work well in the 3D printing process for retaining the elasticity of the support and the thick support head. In addition, if it is combined with the interconnected support, a considerable success rate and excellent print quality can be obtained.

4. High quality FEP (Geeetech FEP)

The last thing to say is the FEP film. The new FEP film works better than it has been for a while. If there are some unexpected failures, you can check if it is caused by a damaged FEP film. If it is highly scratched, matte or twisted, it is recommended to replace

it in time. Also remember to use a silicon scraper to clean the FEP film and avoid using any sharp tools.

Always remember that the biggest resin cost is caused by a print failure, not to add some "extra" security support to avoid printing failures.