Voxeldance Additive

User manual

Content

1 Software Overview

1.1 About Voxeldance Additive

Voxeldance Additive is a data preparation software for Additive Manufacturing, 3D printing, prepare and optimize the parts quickly and easily, ensure to print high quality parts. To prepare for printing, it converts three-dimensional files into two-dimensional slice files, consisting of a list of two-dimensional slice layers for printing. It can view, edit, repair and analyze three dimensional files in several formats.

Voxeldance Additive mainly provides the following functions: Part import, file repair, smart 2D/3D placement, part editing, support generation, slicing and so on.

1.2 System Requirements

1.2.1 Hardware

CPU

- Intel Core i5/i7/i9
- AMD Phenom II X4/X6 at 3.0GHZ or higher with SSE2

Memory

- 16GB RAM or higher

Free Disk Space

- 2GB of free disk space

Display

- 11920 x 1080 resolution or higher
- 2560 x 1440 is recommend

Video Card

- NVIDIA Geforce GTX 1060 or AMD Radeon RX 480 or better

- At least 1GB of memory
- At least a memory interface width of 192-bit (256-bit is recommended)
- Any Intel GPU chipsets is not recommended

1.2.2 Operation System

Voxeldance Additive is only supported on:

- Windows 10 (64 bit) (recommended)
- Windows 8 / 8.1 (64 bit)
- Windows 7 (64 bit)

Voxeldance Additive is recommended：

- Windows Professional edition
- Windows Enterprise edition

Voxeldance Additive is not supported on following system：

- Windows Server edition
- Virtualization system such as VMWare

2 Program Overview

The user interface of this program consists of a graphic display window, a menu bar, a toolbar, and a part view window, as shown in Figure 2.1.

The graphic display window consists of a platform and a ruler, occupying most of the space of the interface.

The Part Navigator displays a list of parts or a list of slices as a tree list. The Details and the Clipping are used to display the basic data information of the part and control the internal observation of the cutting section, respectively. Click the icon on the right of the menu bar is to switch the display status of the menu bar.

The functions and operations can be found in the menu bar, toolbar, right-click menu or through shortcut keys.

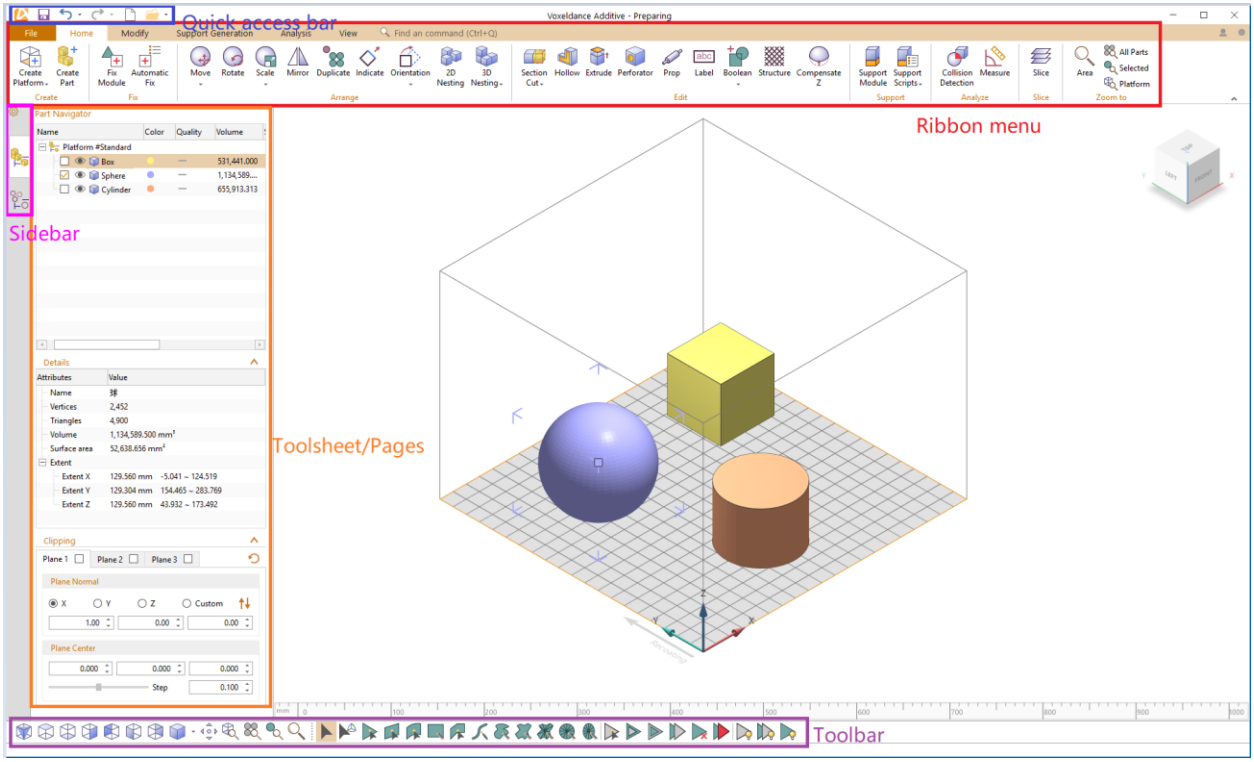


Figure 2.1 Voxeldance Additive Standard Interface

2.1 Part List

The Part Navigator displays all parts on each platform in a tree-like directory structure, as shown in Figure 2.2. Select the part by checking the check box in front of the part. You can also select multiple parts at the same time by using the Shift or Ctrl key, and then check in front of any part to select multiple parts.

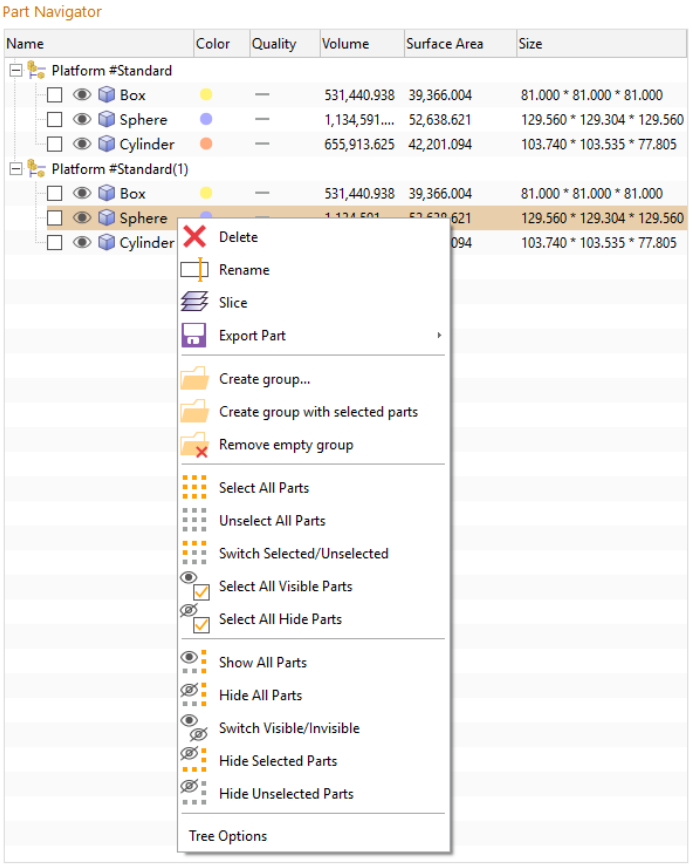


Figure 2.2 Part View window

Double-click the circular icon " " on the right side of the part to change the color of the part; click the eye icon " " on the left side of the part to control the part to be shown or hidden. Double-click the icon " — " under the Quality to check the quality of the part.

In addition, by right-clicking each item in the directory, you can find the corresponding functions and operations in the right-click menu. For example, you can rename part through the right-click menu or double-click the part name; you can create a group through the right-click

menu, and add one or more parts to the group by dragging and dropping, which is convenient for unified management and operation, especially for multiple parts .

You can add multiple platforms in the Parts Navigator, and you can drag one or more parts from one platform to another platform with the left mouse button. In addition, you can choose to display all platforms or only the current platform through the tree option in the right-click menu, as shown in Figure 2.3.

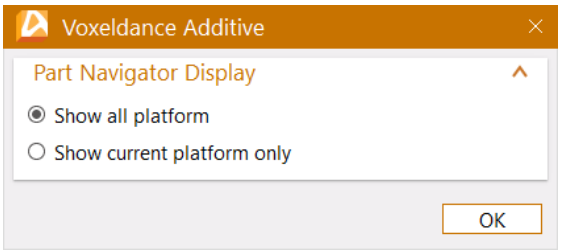


Figure 2.3 Platform display options

2.2 Graphic Display

The graphic display window provides a three-dimensional visual representation of the platform and the parts. A platform is a three-dimensional display area that represents the actual machine print space with the borders. The coordinate system in the upper right corner of the display window is used to indicate the current viewing angle.

The graphic view can be adjusted by mouse operation. By holding down the right button and moving the mouse, you can rotate the viewing angle; by holding down the scroll button and moving the mouse, you can move the platform and parts without changing the angle of view; by sliding the scroll wheel, you can control the zooming of platforms and parts.

By selecting and dragging a part, you can move or rotate the part without changing the position and orientation of the platform and other parts. After selecting a part, left-click and hold the square icon in its center position and move the mouse to change the position of the part, as shown in Figure 2.4.

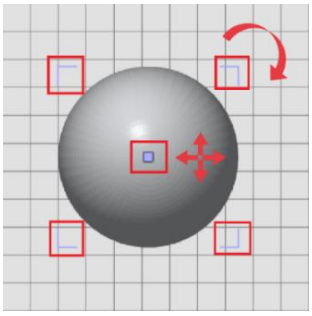


Figure 2.4 Select part to move or rotate by mouse

2.3 Software Functions

Apart from the basic operations in the above-mentioned display window, most of the remaining functions of the software can be realized through the following ways.

Firstly, right-click the part name in the part list, the part in the display window, or the blank area in the display window, the corresponding context menu will be displayed with available functions and actions, as shown in Figure 2.5.

Secondly, the toolbar at the bottom of the display window provides various available functions and operations according to the current interface or function module.

The following sections will introduce the function modules in the File, Home, Modify, Support Generation, Analysis, and View menus in later chapters.

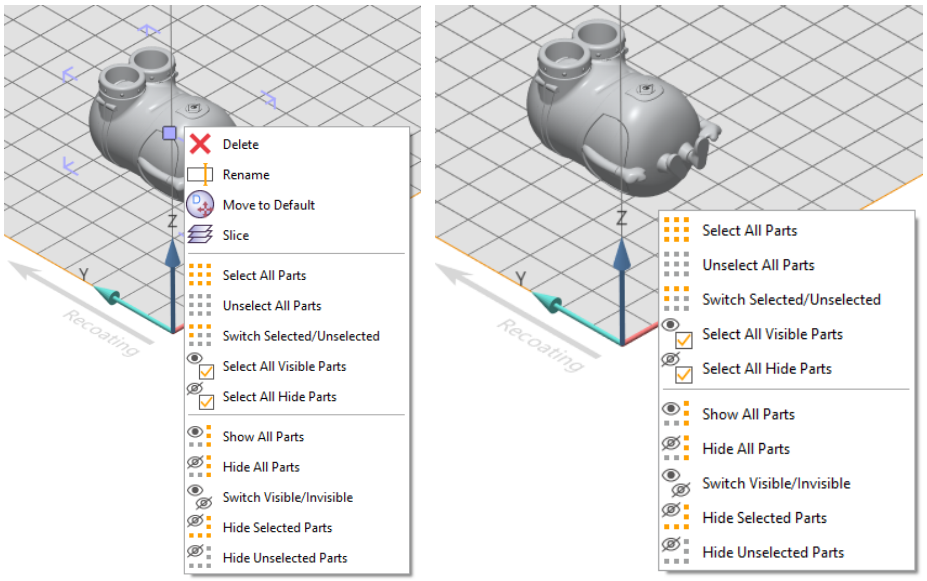


Figure 2.5 The right-click menu of part (left) and blank area (right)

In addition, some functions and features associated with each module will be presented in the context window located in the lower right part of the interface. The Details page shows the information of the selected part. The Clipping page provides multi-section cutting to observe the internal structure of the part, as shown in Figure 2.6.

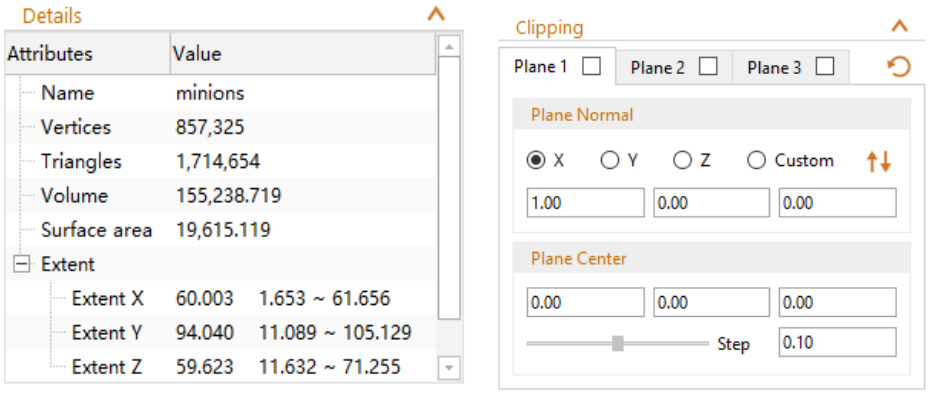


Figure 2.6 Basic information page

2.4 All Shortcut keys

Voxeldance Additive can be controlled with shortcut keys. See the following table for an overview:

Operation	Shortcut key	Operation	Shortcut key
New Project	Ctrl + N	Display Mode	Ctrl + G
Open	Ctrl + O	Zoom in	A
Save Project	Ctrl + S	Zoom out	Q
Cut	Ctrl + X	Zoom to All Parts	F2
Copy	Ctrl + C	Zoom to Selected Parts	F3
Paste	Ctrl + V	Zoom to Platform	F4
Undo	Ctrl + Z	Panning	Ctrl + M
Redo	Ctrl + Y	Perspective	Space

Select All Parts	Ctrl + A	View Right	G
Highlight Selected Parts	H	View Front	V
Switch Selected/Unselected	Ctrl + I	View Bottom	B
Hide Selected Parts	Ctrl + H	View Top	V
Find a command	Ctrl + Q	View ISO	F
Create Platform	Ctrl + W	Wall Thickness	Shift + W
Platform Definitions	Ctrl + P	Volume Estimation	F12
Create Part	Shift + N	Move selected parts in positive X direction	Shift + Right
Move	T	Move selected parts in negative X direction	Shift + Left
Move to Default	Ctrl + F	Move selected parts in positive Y direction	Shift + Up
Move to Default Z	Ctrl + E	Move selected parts in negative Y direction	Shift + Down
Put on Platform	Ctrl + Shift + Down	Move selected parts in positive Z direction	Shift + PgUP
Pick & Place part	F9	Move selected parts in negative Z direction	Shift + PgDown
Rotate	R	Move view up	Up
Scale	S	Move view up slightly	Alt + Up
Mirror	M	Move view down	Down
Duplicate	Ctrl + D	Move view down slightly	Alt + Down
Indicate	Shift+ B	Move view right	Right
Orientation	Shift + U	Move view right slightly	Alt + Right

2D nesting	Shift + A	Move view left	Left
Section cut	C	Move view left slightly	Alt + left
Perforator	Shift + P	Rotate selected parts clockwise around X direction	X
Label	Shift + L	Rotate selected parts anticlockwise around X direction	Shift + X
Boolean	Ctrl + B	Rotate selected parts clockwise around Y direction	Y
Support module	Shift + I	Rotate selected parts anticlockwise around Y direction	Shift + Y
Support script	Ctrl + Shift + I	Rotate selected parts anticlockwise around Z direction	Z
Collision Detection	Shift + K	Rotate selected parts clockwise around Z direction	Shift + Z
Measure	Shift + M	Rotate view clockwise	Ctrl + PgUp
Fix Module	Shift + R	Rotate view clockwise slightly	Ctrl + Alt + PgUp
Mesh Reduction	Shift + T	Rotate view anticlockwise	Ctrl + PgDown
Transparent Parts	N	Rotate view anticlockwise slightly	Ctrl + Alt + PgDown
Exit Repair/Support	Shift + Q	Rotate view up	Ctrl + Up
Mark Triangle	F5	Rotate view up slightly	Ctrl + Alt + Up
Mark Shell	F7	Rotate view down	Ctrl + Down

Mark Plane	F6	Rotate view down slightly	Ctrl + Alt + Down
Unmark All	F8	Rotate view right	Ctrl + Right
Delete Selected/Marked	Del	Rotate view right slightly	Ctrl + Alt + Right
Show Platform	F11	Rotate view left	Ctrl + Left
Restore the default interactive mode	Esc	Rotate view left slightly	Ctrl + Alt + Left
Exit	Alt + F4		

3 Project Management

3.1 Open Files

New

With the "New" command in the File menu or the "  " icon on the quick access toolbar, you can create a new empty project. Warning: Any changes made on previously opened projects will be discarded. You can add platforms or open files to the new project and perform operations.

Open

You can open Voxeldance Additive project files, STL files, CAD files, 3D graphics files in other formats, or previously saved slice files through the Open command in the File menu or the “ ” icon on the quick access toolbar, as shown in Figure 3.1 . If you import a CAD file, Voxel will automatically convert it to STL. The dialog box will allow you to specify the transform parameters, and you can also specify the placement of the part in the workspace. Besides, you can open a file and add it to the current project by dragging the valid file directly to the interface window.

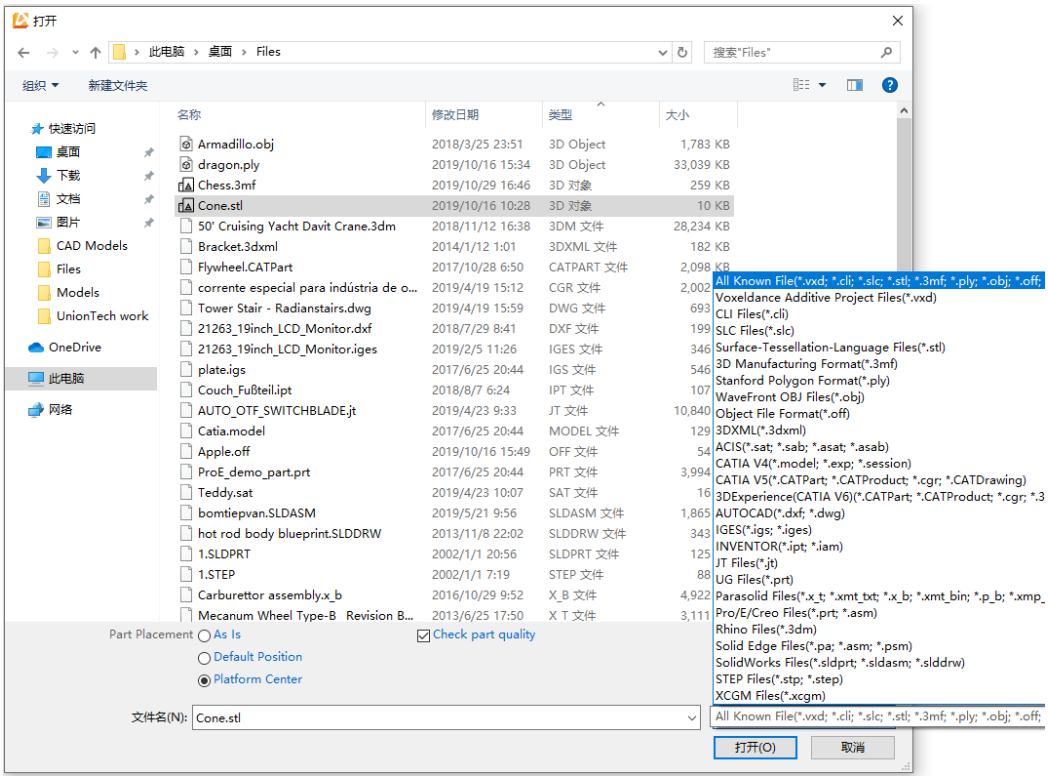


Figure 3.1 Open file

Preview File

File Preview provides a quick preview of the part library without having to open each file. By clicking the name of any part file or slice file, you can preview the part in the display window, as shown in Figure 3.2. In the preview mode, you can also perform the basic view operations such as movement, zooming, and view switching. Then by clicking the Open File button, you can add the selected file to the current platform.

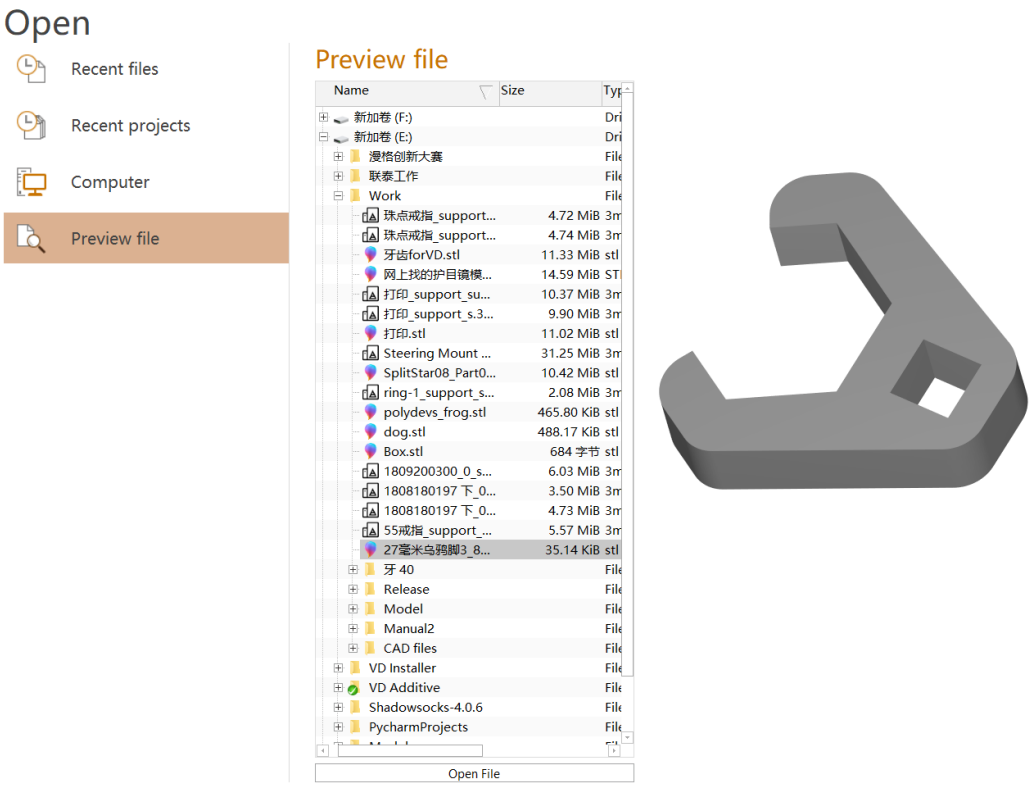


Figure 3.2 Preview file

3.2 Save Files

Save

Through the Save command by the icon “ ” on the quick access toolbar, the current project will be saved and the previous files will be overwritten.

Save As

Click the "Save As" command in the File menu, you can define the save type, name and path for the file.

Export Part

If you need to export the part separately, select the part, and click the "Export part" command in the File menu or the right-click menu of the part in the part list, and the sub-menu of output format options will appear. Select a format, and the Save dialog will pop up, where you can save the part in the selected format.

3.3 Quick Search

Search Function in Voxeldance Additive. As shown in Figure 3.3, all relevant features will be immediately displayed by entering the name of function in the search box. Clicking on the desired function can be launched directly.

Figure 3.3 Search box

4 Viewing Options

The view to the display screen can be altered in multiple ways. The intuitive mouse operation has been introduced in Section 2.2. Besides, the view can be adjusted by a series of standard commands in the VIEW menu, as shown in Figure 4.1.

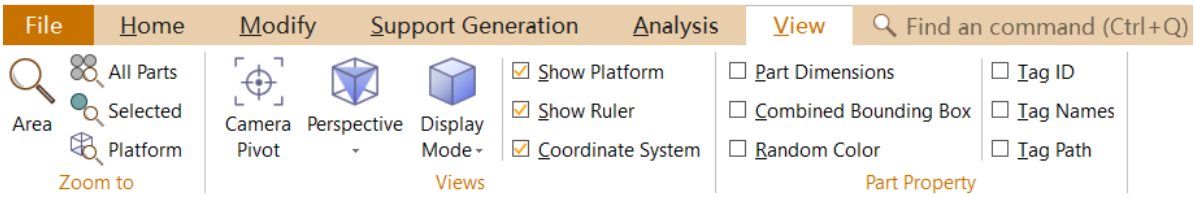


Figure 4.1 VIEW menu

4.1 Perspectives

The perspective refers to the direction and angle of viewing a part and it is indicated by the cubical coordinate system in the top right of the display window, with the X-Z plane of the coordinate representing the front panel. There are seven standard perspectives, as viewed from front, back, top, bottom, left, right, and isometry respectively. The isometric perspective is a view from the front-left-top-corner of the platform, forming a three-dimensional observation of the entire platform.

There are three ways to switch between the seven standard perspectives:

1) The view can be switched by the sub-menu commands of the 'Perspective' command in the View menu or the corresponding view icons in the toolbar, as shown in Figure 4.2.

Figure 4.2 Seven standard viewing icons in the toolbar

2) By clicking the six faces, eight vertices, and twelve edges of the coordinate cube, you can specify the viewing angle referring to the coordinate system to switch between different perspectives, as shown in Figure 4.3.

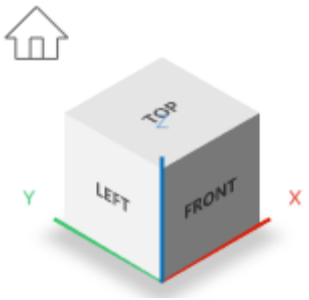


Figure 4.3 Directional cube

3) Press the Space key to switch the view backwards or the Shift + Space key to switch the view forwards. If you do not change the perspective, the default mode is the isometric view.

4.2 Moving and Zooming

As described in Section 2.2, by holding the scroll wheel and moving the mouse, the entire platform can be shifted; and by selecting the part and dragging its central position, you can move the part.

The scroll button of the mouse can be used to zoom in and out. Additionally, there are four default zooming control commands, as shown in Figure 4.4, which can be found in the Zoom menu or in the toolbar.

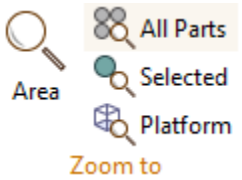


Figure 4.4 The Zoom commands in the toolbar

Zoom to Select

After clicking the "Zoom to Select" command in the Zoom submenu or toolbar, you will enter the mode of waiting for the users to select the area. At this time, by holding the left mouse button and dragging the mouse, you can create a rectangle selection area. After releasing the left button, the display window will zoom into the selected area, as shown in Figure 4.5.

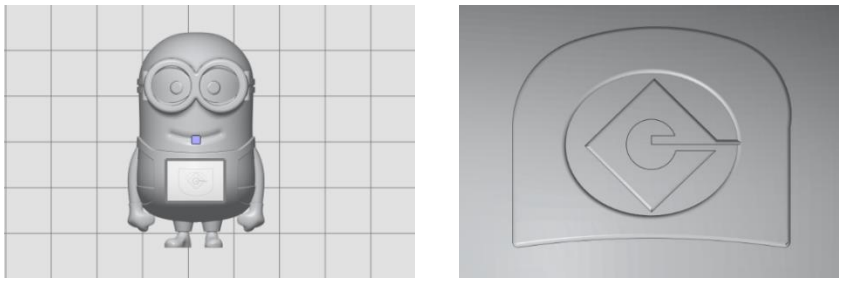


Figure 4.5 Select area to zoom in

Zoom to All Parts

After clicking the "Zoom to All Parts " command in the Zoom submenu or the toolbar, the display window will be automatically zoomed to display the entire platform and all parts in a suitable scale.

Zoom to Selected Part

After selecting one or more parts, by clicking the "Zoom to Selected Part " command in the Zoom submenu or the toolbar, the display window will be automatically zoomed to display all the selected parts in a suitable scale.

Zoom to Platform

After clicking the "Zoom to platform " command in the Zoom submenu or the toolbar, the display window will be automatically zoomed to display the entire platform frame.

4.3 Displaying Options

4.3.1 Hide/Show Parts

With clicking " " icon on the left side of the part name in the part list, you can control the display or hide of the part. And you can also use "Hide Selected Parts" command in the right-click menu of part, or the shortcut keys Ctrl + H to hide the selected parts. When a part is hidden, the icon " " on the left of the part name turns " ". Besides, all other parts other than the selected parts can be set hidden by using the "Hide Unselected Parts" command in the right-click menu of the part.

By using the “Show All Parts/Hide All Parts” command in the right-click menu of any blank area on the platform, you can control the displaying and hiding of all the parts on the platform. By using the “Switch Visible/Invisible” command, you can perform the switch between the displaying and hiding of all parts.

When several parts are displayed on the platform and block each other, select the part you want to observe, press and hold the ‘h’ key to display the part in the foreground, and release the ‘h’ key so it will return to the original display state.

4.3.2 Display Mode

There are five display modes for parts on the platform, the shadow mode, the triangular mesh mode, the wireframe mode, points mode and bounding-box mode, as shown in Figure 4.6. Through the corresponding command under the Display mode sub-menu in the VIEW menu, or the drop-down options of the “  ” icon in the toolbar, or the shortcut keys Ctrl + G, you can switch between these display modes for all the parts on the platform, with the Shade mode in default.

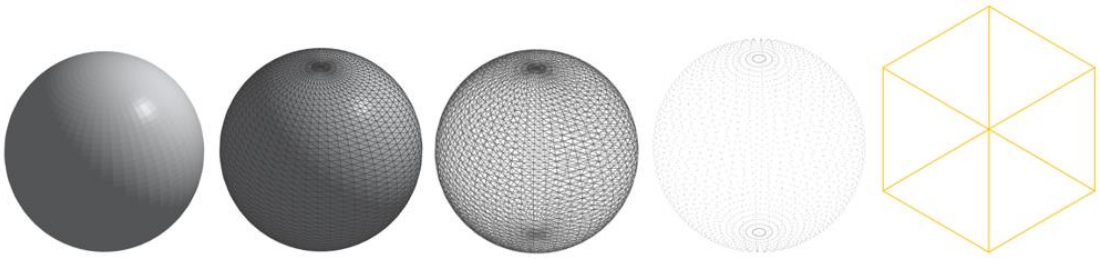


Figure 4.6 Three display modes: shadow mode, triangular mesh mode, wireframe mode, point mode, bounding-box mode

4.3.3 Other Display Options

Camera Pivot

Select the “Camera Pivot  ” option in the menu, and click any point on the part surface, to make this point the center point of the camera.

Panning

With the “Panning ” option in the toolbar, or the shortcut keys Ctrl + M, you can enter the panning view mode and move the view through the left mouse button, and then click the ESC key to exit the mode.

Transparent Parts

With the “Transparent Parts” option in the Display Mode submenu or the toolbar, or the shortcut key N, you can control whether the transparent view of the parts is displayed.

Show Part Boundary

With the “Part Boundary” option in the Display Mode submenu or the toolbar, you can control whether the sharp edges of the part on the platform are displayed with a black border.

Show Part Projection

With the “Show Part Projection” option in the Display Mode submenu or the toolbar, you can control whether the part projection on the platform is visible, as shown in Figure 4.7.

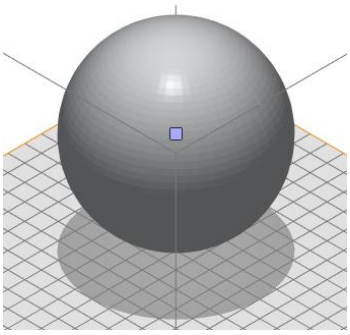


Figure 4.7 Show part projection

Show platform

With the “Show Platform” option under the VIEW menu, you can control whether the three-dimensional platform is visible in the display window.

Show Ruler

With the “Show Ruler” option under the VIEW menu, you can control whether the ruler in the display window is visible.

Show Coordinate System

With the “Coordinate System” option under the VIEW menu, you can control whether the orientation indicator of the platform in the display window is visible.

Show Part Dimensions

With the “Part Dimensions” option under the VIEW menu, you can control whether the part dimensions of the part are displayed.

Show Combined Bounding Box

With the “Combined Bounding Box” option under the VIEW menu, you can control whether the bounding box size of all selected part is displayed.

Show Random Color

With the “Random Color” option under the VIEW menu, you can control whether to set the random color of the part.

Tag ID

With the “Tag ID” option under the VIEW menu, you can control whether the parts’ number is displayed.

Tag Names

With the Tag Names option under the VIEW menu, or clicking the icon  *under the View tab in the lower right corner of the main interface, you can control whether the parts’ name is displayed.*

Tag Path

With the Tag Path option under the VIEW menu, or clicking the icon  under the View tab in the lower right corner of the main interface, you can control whether the parts’ file path is displayed.

4.4 Clipping View

In the "Clipping" on the left tool page, you can use section cut to observe the inside of the part, as shown in Figure 4.8.

Click the selection box of the “Plane” to turn on / off the section, and then select X / Y / Z / in the “Plane Normal” to define how to create the section. There are several options for defining sections:

1) perpendicular to the X, Y or Z axis

2) Customize the position of the section. Enter the exact value in the "Custom" setting.

Move the slider below to make the section move to one side. In the "Step" setting, you can enter a value to adjust the step size.

Click the " " icon to reset the crop view. Click the “” icon to switch the hidden part of the part.

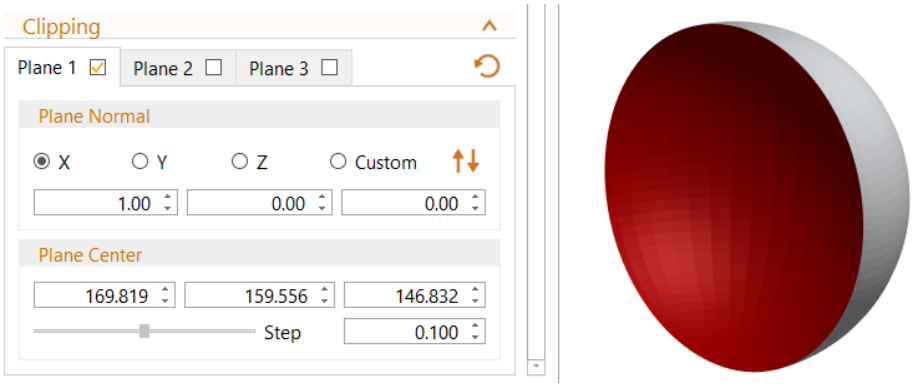


Figure 4.8 Clip view

5 Part management

5.1 Create Parts

By creating part functions, you can create some simple 3D primitive parts. Click the "Create Part" command in the Home menu, a series of simple 3D parts will be listed, as shown in Figure 5.1. Although it is only some basic geometry, you can create various required parts by changing the detailed parameter values.

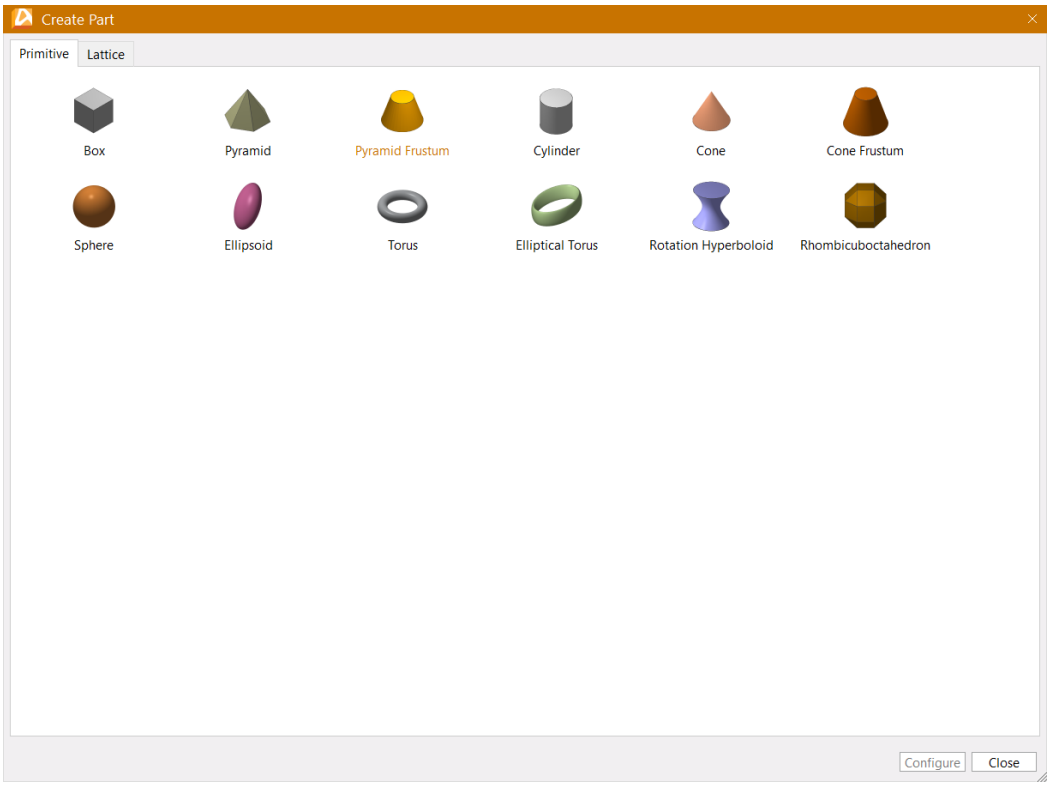


Figure 5.1 Part library

Double-click the part to enter the parameter setting interface. These settings can be saved as the default settings by clicking the "Save as Default" button. Clicking the "Load from Defaults" button can restore the default settings after parameter changes. Clicking the "Preview" button can preview the part after changing part parameters, check the "Auto Preview" selection box to automatically preview the part after the part parameters are changed. At the same time, you can perform basic view operations such as moving, zooming, and switching the view. Finally, insert the part into the project by clicking “Create” button.

In the following table, a brief description of some parameters is listed:

Parameter	Features
Name	The name of the part, which displays in the part list
Length	The measured length of the X axis
Width	The measured length of the Y axis
Height	The measured length of the Z axis
Radius	The radius / size of the part
Edges	The number of edges of the part surface
Tolerance	The larger tolerances make the roundness of the part less accurate, and cause fewer triangles
Height Partitions	In the Z-axis direction, the triangle is divided by a certain number
Rotation Angle	Bending angle (hyperboloid of revolution)

5.2 Select Parts

When you manage or edit a specific part, you must first select it.

a. Select in the parts list

Select the part by checking the selection box in front of the part name in the part list. You can also select multiple parts at the same time by using the Shift or Ctrl key, and then check the selection box in front of any part to select multiple parts. Use the Ctrl + A keys can check all the selection boxes in front of all parts name to select all parts.

b. Select in the display window

Click the "Select Part" icon on the toolbar, then click the part in the display window to select it. You can also hold down the Shift or Ctrl keys to select multiple parts at the same time. With Ctrl + A keys, you can select all parts.

Besides, there are several selection commands in the right-click menu of the blank area on the platform, including: select all parts, unselect all parts, switch selected/unselected, select all visible parts, and select all hide parts.

5.3 Undo and Redo

Through the “Undo/Redo  ” icon in the quick access toolbar, or shortcut keys Ctrl + Z, Ctrl + Y, you can undo and redo some operations.

5.4 Copy and Paste

Select the parts that need to be cut or copied, click the "Cut / Copy" command in the Modify menu, or the shortcut key Ctrl + X / Ctrl + C, then select the platform to be placed, and then click the "Paste" command in the Modify menu, or the shortcut key Ctrl + V, can copy and paste the parts.

5.5 Convert Units

STL files generally work with inches or in millimeters, but do not store which unit is being used. Therefore, you may import a part that seems too small or too large. This will usually trigger a warning that your part is smaller or larger than a certain value, asking whether you would like to convert from inch to mm or vice versa, as shown in Figure 5.2.

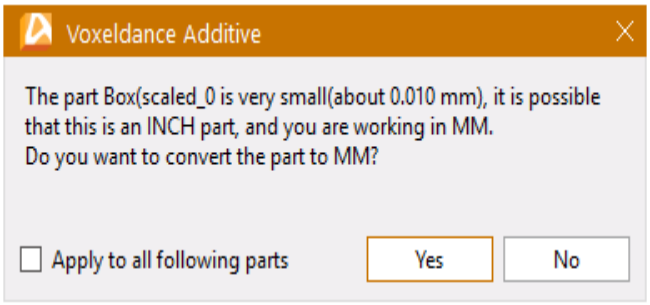


Figure 5.2 Unit conversion dialog

If this dialog does not pop up while opening a file, you can always convert units using “Convert inch to mm” or “Convert mm to inch” functions in Modify Ribbon.

And you can change the parameters for automatic unit conversion from Options, as shown in Figure 5.3.

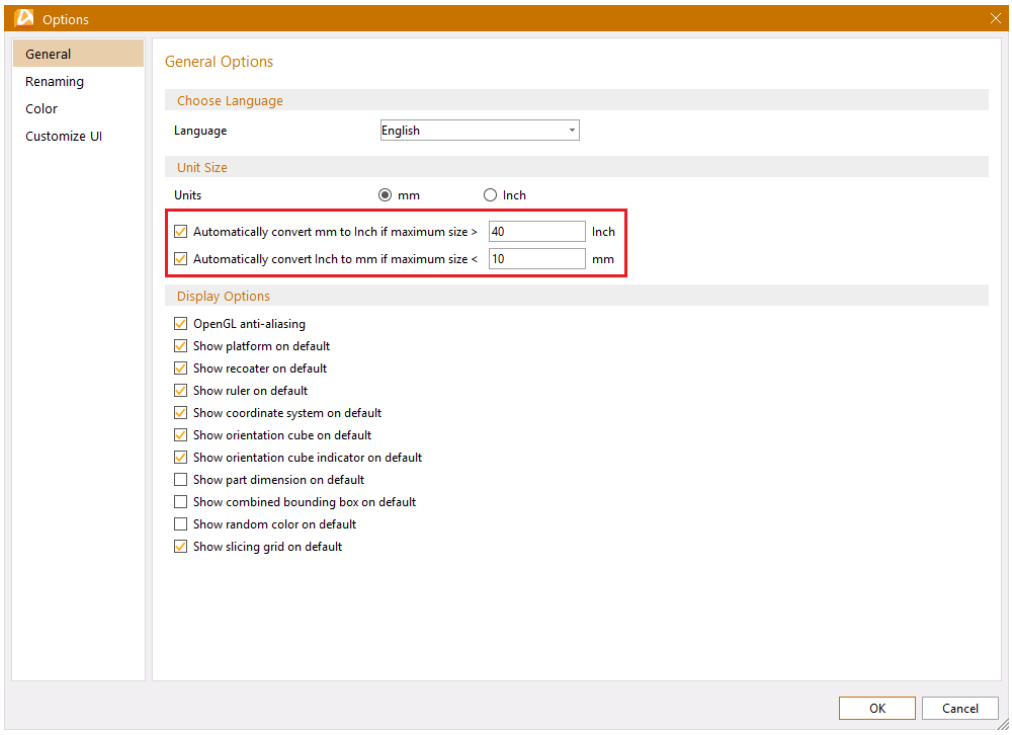


Figure 5.3 Unit conversion settings

6 Part Position

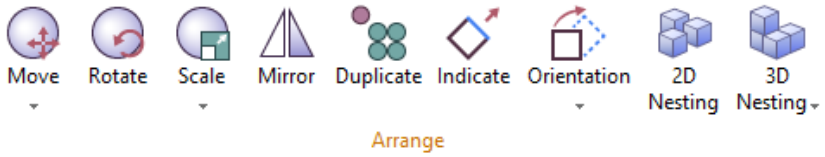


Figure 6.1 Part Position menu

6.1 Move Parts

Moving the part can be implemented by dragging its center position directly. Additionally, select the part, clicking the “Move” command in the Home menu, or press the shortcut key T, the “Move Parts” dialog for accurately setting movement parameters will pop up in the display window, and the part enters the movement mode of the control dimension, and the selected part displays the three-dimensional coordinate system, as shown in Figure 6.2.

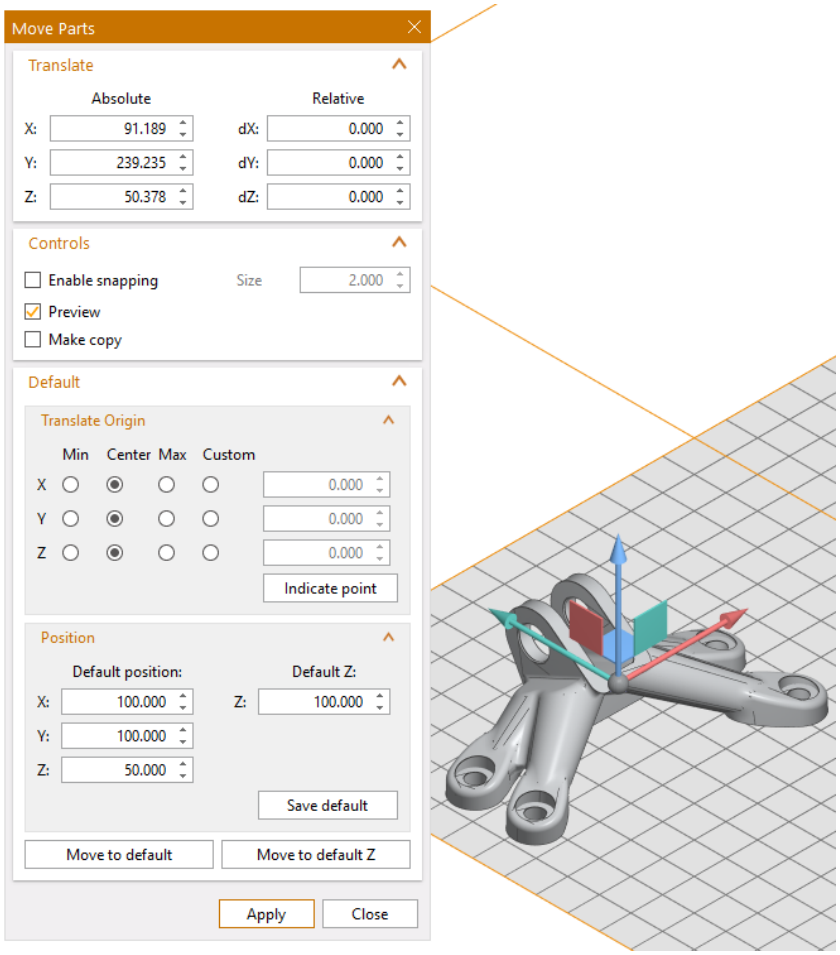


Figure 6.2 Move parts setting

The top of the settings dialog shows absolute position and relative position information: Absolute page for setting the absolute position coordinates X, Y, Z (relative to the origin X = Y = Z=0 coordinates); Relative page for setting dx, dy, dz parameters relative to the current coordinates. After checking the Enable snapping checkbox, “Size” is in editable state, and you can set the step when adjusting the position parameters. If you check the “Preview” below, you can preview the part movement while setting the parameters; if you check the “Make copy”, a copy can be created at the original position after the part is moved. After setting, click Apply button to apply without exiting the module, and subsequent settings can be made.

When the mouse is moved to any axis (X or Y or Z) or any coordinate plane (X-Y or X-Z or Y-Z), this direction will be marked as yellow highlighting. When holding down the left mouse button and moving the mouse, you can control the part to move only in the marked direction, that is, only in one of the X, Y, or Z direction or in one of the X-Y, X-Z, and Y-Z planes. If you hold down the left mouse button and move the mouse while pressing the Shift key, the part will be moved by integer steps.

In the middle of the setting dialog, you can set the position of the starting point of the three-dimensional coordinate system of the control dimension. There are three main ways: Click on the current three-dimensional coordinates range (Min, Center, Max) of the part to set; Click Custom of any X, Y, Z axis, you can change the parameters value in the corresponding direction to set. Click the “Indicate point” button, you can set the position of the selected part with the mouse.

The default parameters can be set at the bottom of the dialog, including the default position and the default Z axis height. After setting the corresponding default parameters, you can save these values by clicking the “Save default” button. After the default parameters are saved, you can move the part to the specified position by clicking “Move to default”, and move the part to the specified height by clicking “Move to default Z”.

After clicking the “Confirm” button to exit the module, and changing the location of the part, then you can directly move the part to the default location by clicking the “Move to Default” or “Move to default Z” command in drop-down menu or the right-click menu of part in the display

window. Click "Move to platform center" to move the part to the center of the platform, click "Put on Platform" or the shortcut key Ctrl + Shift + Down to directly place the bottom of the part on the platform, click "Select & place part" to enter the control dimension Mobile mode.

6.2 Rotate Parts

Rotating part can be implemented by directly dragging the square brackets around it. You can also use the shortcut keys X/Y/Z to realize the clockwise rotation around X, Y, Z axis or use the shortcut keys Shift + X/Y/Z to realize the counterclockwise rotation around X, Y, Z axis. Besides, select the part, click the “Rotate” command in the Home menu, or press the shortcut key “R”, you can set the rotation parameters accurately in the Rotate Parts dialog displayed in the view window. When the dialog box is opened, the part enters the rotation mode of the control dimension, and the selected part displays the three-dimensional coordinate system, as shown in Figure 6.3.

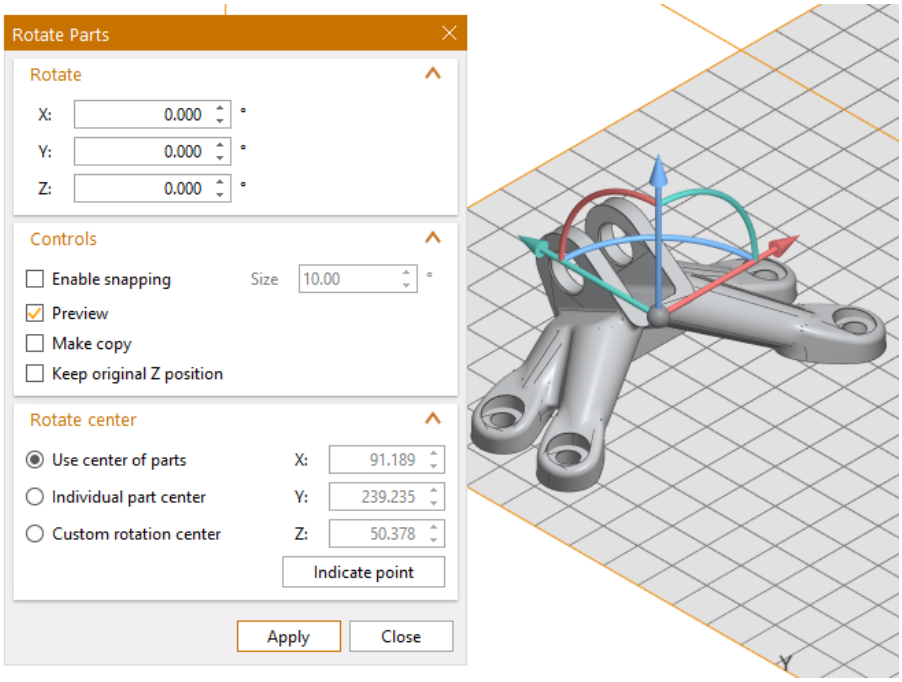


Figure 6.3 Rotate parts setting

You can set the rotation angles around the X, Y, and Z axis respectively in the Rotate angles. After checking the “Enable snapping” checkbox, “Size” is in editable state, and you can set the step when adjusting the rotation angle parameters. If you check the “Preview” below, you can preview

the part rotation while setting the parameters; if you check the “Make copy”, a copy can be created at the original position after the part is rotated. After setting, click Apply button to apply without exiting the module, and subsequent settings can be made.

The coordinates of the rotation center can be set in the Rotate center drop-down menu at the bottom of the dialog, there are main four ways: Rotate around the center points of multiple parts; Rotate around the center of each part; Define part center point to rotate; Click Indicate point button, then specify a position on the part as the center point to rotate.

Besides, in the control dimension mode, when the mouse is moved to the circular area of any axis (X or Y or Z), the circular area in the direction will be marked as yellow highlighting. At this point, holding down the left mouse and dragging, the part can be controlled to rotate only in that direction (that is only in one of the X, Y, or Z direction).

6.3 Scale Parts

Select the part, click the “Scale” command in the Home menu, or press the shortcut key “S”, you can set the scale parameters accurately in the “Scale Parts” dialog displayed in the view window, as shown in Figure 6.4.

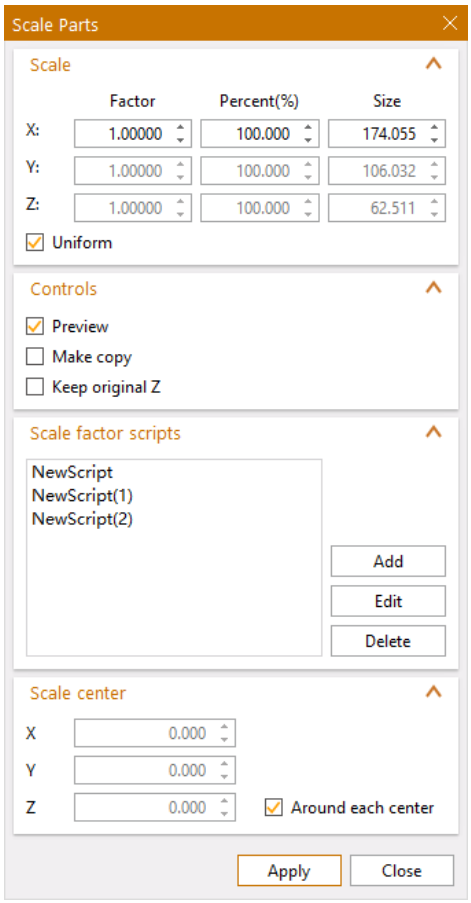


Figure 6.4 Scale parts setting

You can set the scale and size of the X, Y, or Z dimension respectively in the top of the dialog. If the “Uniform” check box at the bottom is checked, and if the parameter in any dimension is changed, the other dimension data will be changed in the same proportion to maintain the original shape. Otherwise, independently setting the three-dimension parameters may cause deformation or stretch effects; If you check the “Preview” below, you can preview the part scale while setting the parameters; if you check the “Make copy”, a copy can be created at the original position after the part is scaled. If you check the “Keep original Z” box, the original Z axis does not change after the part is scaled.

The scale center can be set in the Scale center drop-down menu at the bottom of the dialog. After checking the “Around each center” box, all parts will be scaled around their respective center points.

The scale factor scripts can store the scale factor. You don't need to open the “Scale Parts” dialog, you can directly click on the drop-down button of the Scale menu and select a script to quickly scale the part. Click the "Add" button to create a new scale factor script, as shown in Figure 6.5. Select an added script, click the "Edit" button to edit the script name and scaling parameters, and click the "Delete" button to delete the selected script.

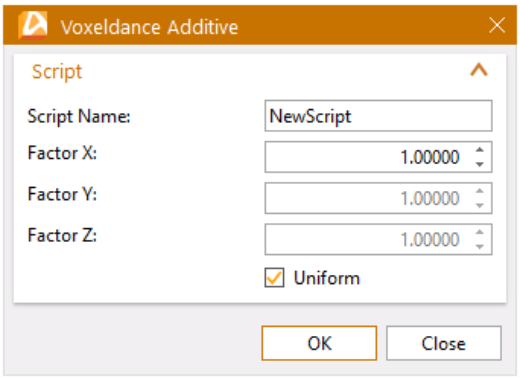


Figure 6.5 Scale Part

6.4 Mirror Parts

Symmetric mirroring transformation of the part can be realized with the Mirror Parts command. Select the part, click the “Mirror Parts” command in the Home menu, or press the shortcut key “M”, the Mirror Parts parameter setting dialog will pop up in the view window, as shown in Figure 6.6. You can choose the XY or XZ or YZ plane in the Mirror panel tab. The default symmetry plane is the three middle axis planes of the part (that is, on their own symmetry). If you check the “Preview” below, you can preview the mirroring of the part while setting the parameters; by checking the “Make copy”, you can create a new symmetric copy, otherwise the symmetry transformation is performed only on the original part. By checking the “Use center parts” box in the Plane Position tab, you can change the default symmetry plane position.

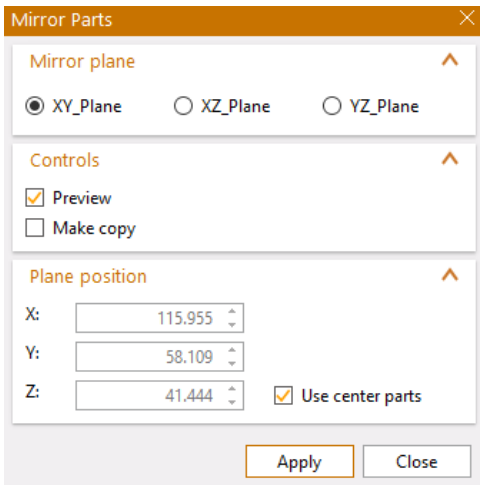


Figure 6.6 Mirror Part setting

6.5 Duplicate Parts

Select the part, click the "Duplicate Parts" command in the Home menu, or press the shortcut key “Ctrl + D”, and the Duplicate Parts parameter setting dialog box will pop up in the view window, as shown in Figure 6.7. Enter the total copy of parts (> = 1, including original parts), and set the required number and spacing in the three directions of X, Y, and Z respectively. Check the "Preview" selection box to preview the duplicated parts while setting the parameters. Check the "Fit to platform" selection box to place the duplicated parts in the platform.

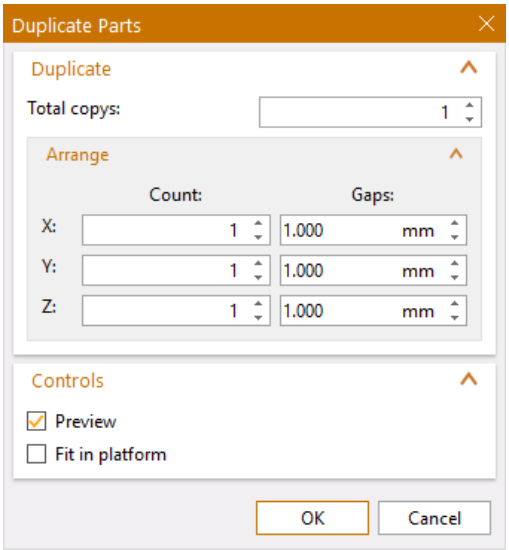


Figure 6.7 Duplicate parts setting

6.6 Indicate Plane

Select the part, click the "Indicate" command in the Home menu, or press the shortcut key “Shift + B”, and the Indicate Bottom/Upper Plane parameter setting dialog box will pop up in the view window, as shown in Figure 6.8.

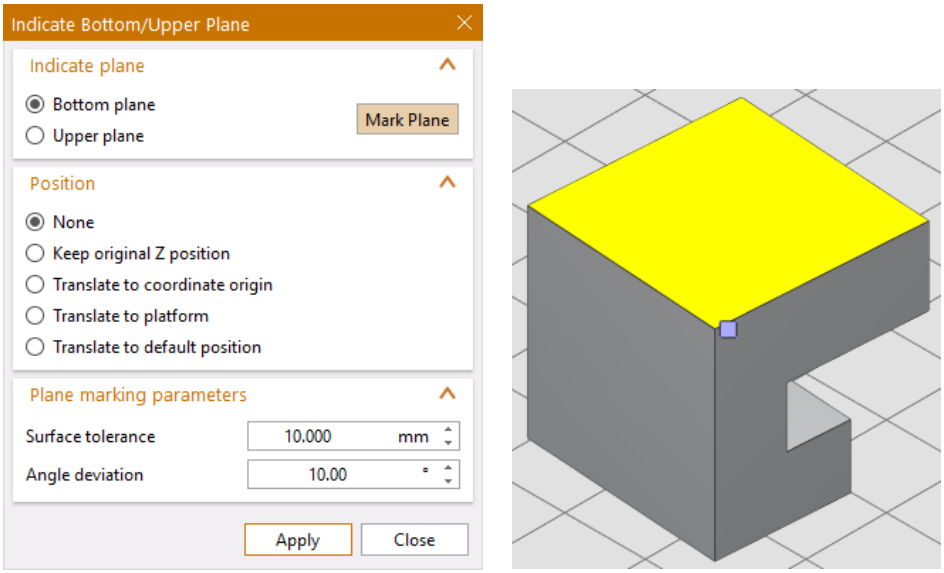


Figure 6.8 Indicate Bottom/Upper plane setting (left) and plane selection (right)

If the "Mark Plane" button in the dialog is in orange, it indicates a selecting state; otherwise, click this button to turn it orange. Then select a plane on the part surface, the selected plane will be marked as yellow, and then click the Apply button at the bottom, you can realize the flip of the part, so that the selected plane is parallel to the platform surface.

The Bottom Plane and Upper Plane options in the Indicate panel field are used to place the selected plane face down or up, that is, and 180-degree flip. More transformation options are available in the Position field, including Keep Original Z Position, Translate to coordinate origin, Translate to platform and Translate to default position.

The Plane Making parameters field is used to set the error range of the selected surface, including the Surface Tolerance and Angle Deviation. The higher the set value is, the larger the selected surface range will be.

6.7 Orientation

6.7.1 Orientation

Using the Orientation Optimizer function can adjust the parts to the optimal placement position, which is beneficial to the later support operations.

Select the part, and click the “Orientation Optimizer” command in the Orientation drop-down menu under the Home menu. The Orientation Optimizer Setting dialog will appear in the display window, as shown in Figure 6.9.

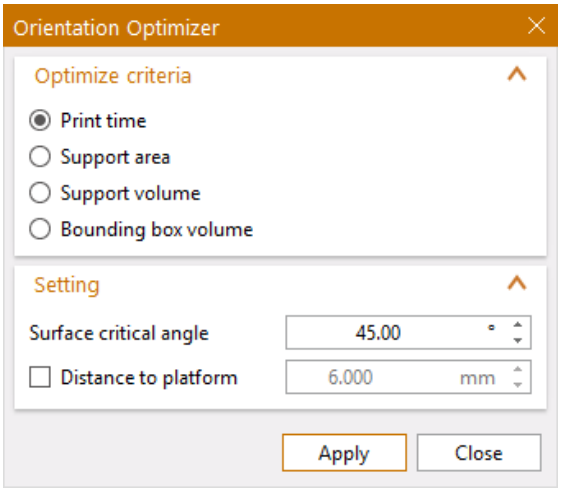


Figure 6.9 Orientation Optimizer setting

The dialog provides four optimization criteria: Print time, Support area, Support volume, Bounding box volume.

Print time: adjust the part to the proper position to make it has the least print time after supporting.

Support area: adjust the part to the proper position to make it has the minimal support bottom area after supporting.

Support volume: adjust the part to the proper position to make it has the minimal support volume after supporting.

Bounding box volume: adjust the part to make its boundary bounding box volume smallest.

At the same time, in the setting, you can set the size of the surface critical angle when adding the support and the distance from the lowest point of the part to the platform.

After the setting is completed, click “Confirm”, the selected part completes the automatic placement according to the setting.

6.7.2 Orientation Comparator

The Orientation Comparator function is to adjust the parts to the optimal placement position in a more intuitive way, which is conducive to later support operations.

Select the part, and click the “Orientation Comparator” command in the Orientation drop-down menu under the Home menu, or press the short keys “Shift + U”, the Orientation Comparator Setting dialog will appear in the display window, as shown in Figure 6.10.

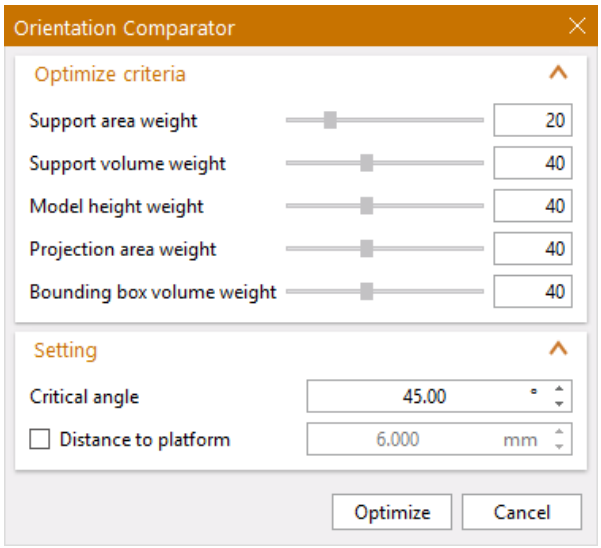


Figure 6.10 Orientation Comparator setting

Orientation comparator provides five optimized standard parameter settings for part placement, including Support area weight, Support volume weight, Part height weight, Projection area weight, and Bounding box volume weight. These five parameters can be set individually by sliding the scroll bar or directly entering specific values in the parameter box.

Support area weight: adjust the support area when the part is added with support, the larger the weight, the smaller the support area;

Support volume weight: adjust the support volume when the part is supported, the greater the weight, the smaller the support volume;

Part height weight: adjust the height of the part, the greater the weight, the lower the part's position, the better to add support;

Projected area weight: adjust the projected area of the part, the larger the weight, the smaller the projected area of the part.

Bounding box volume weight: adjust the volume of the bounding box of the part boundary, the larger the weight, the smaller the bounding box volume;

The critical angle in the Setting field is the same as the angle of the critical surface in the support operation (see Chapter 9); check the "Distance to platform" checkbox to set the distance from the lowest point of the part to the platform.

After setting all parameters, click the "Optimize" button to display a comparison table of the 12 orientation of part, as shown in Figure 6.11. Select each orientation of part, the part will do the corresponding operation. In these 12 orientations, click the column headings to sort in ascending or descending order, and click on different rows to see the suggested orientations and the support required in that orientation. The more green mark in a row, the better the orientation of part. the orientation that is sorted by 0 is selected by default.

After selecting an orientation, and click "OK", the selected parts will complete the placement according to the selected orientation.

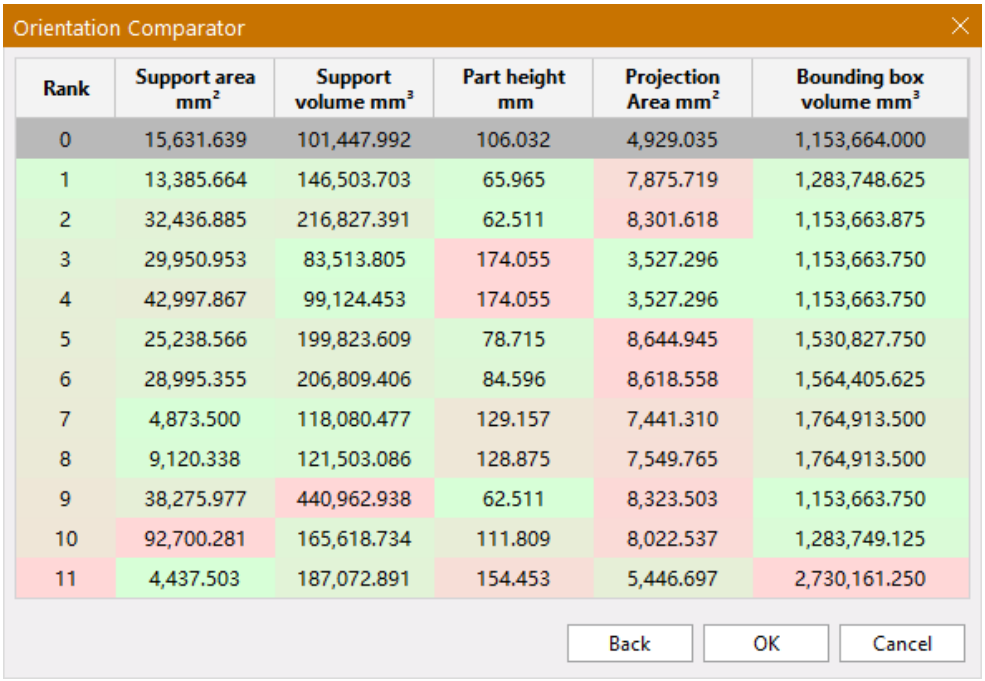


Figure 6.11 Orientation comparator table

6.7.3 Minimize Bounding Box

This feature minimizes the bounding box by rotating the part, but the size of the part itself does not change.

Select the part, and click the "Minimize Bounding Box" command in the Orientation drop-down menu under the Home menu. A dialog box for setting the parameter of the minimized bounding box will pop up in the display window, as shown in Figure 6.12.

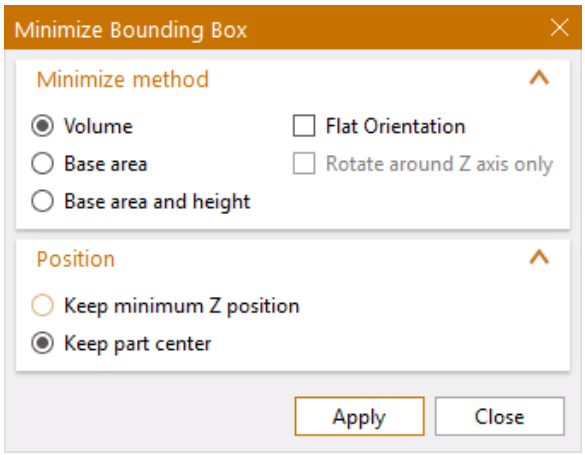


Figure 6.12 Minimize Bounding Box setting

The setting dialog provides three parameter setting modes for minimizing the bounding box of the part, which are Volume, Base area, Base area and height.

Volume: The bounding box has the smallest volume and chooses whether to set the longest side to be horizontal.

Base area: The base area of the bounding box is the smallest, and whether to rotate only around the Z axis.

Base area and height: The base area and height of the bounding box are the smallest at the same time.

Besides, the position of the part can be set at the bottom of the dialog. You can choose whether to keep the minimum Z position or keep the part center.

After the setting is completed, click "Apply", and the selected parts will be placed according to the settings, as shown in Figure 6.13.

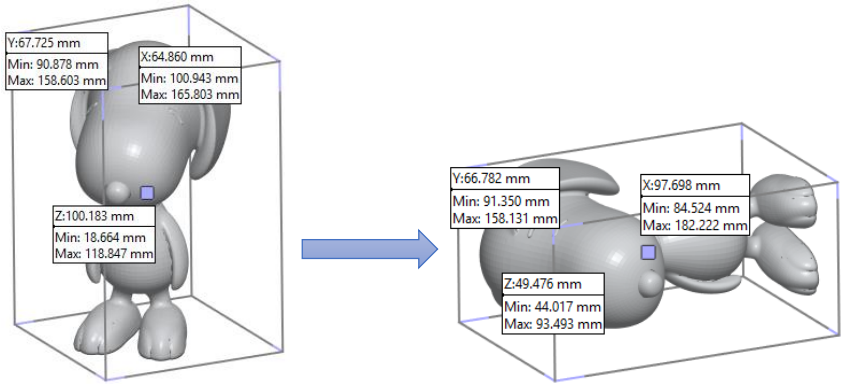


Figure 6.13 The effect of minimal bounding box volume

6.7.4 Orientation Optimizer for Teeth

This function is mainly for the placement of multiple teeth, so that their crowns are automatically placed upwards.

Select the teeth part, and click the "Orientation Optimizer for Teeth" command in the Orientation drop-down menu under the Home menu. A dialog box for setting the teeth orientation optimizer parameters will pop up in the display window, as shown in Figure 6.14.

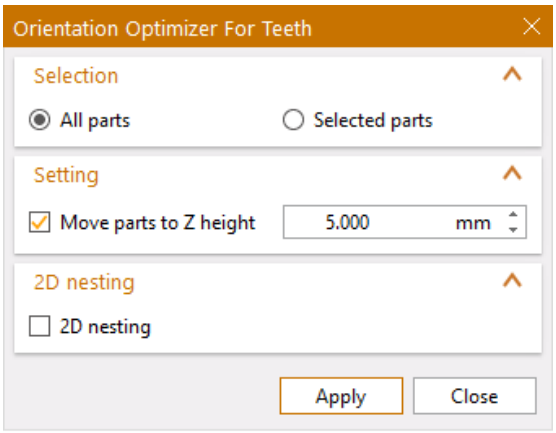


Figure 6.14 Orientation Optimizer for Teeth setting

If "All parts" is selected, the orientation optimization is performed for all the tooth parts on the platform. If "Select parts" is selected, the orientation optimization is performed only for the selected tooth parts.

If "Move part to Z height" is checked, the tooth part will be moved to the set Z height for orientation optimization. If it is not checked, the orientation is optimized at the original position of the part.

If 2D nesting is checked, the parts can be placed automatically and orderly while the orientation is optimized. For details, refer to Chapter 6.8.

After the setting is completed, click "Apply", the tooth parts on the platform will be optimized for orientation according to the settings, and the optimized picture is shown in Figure 6.15.

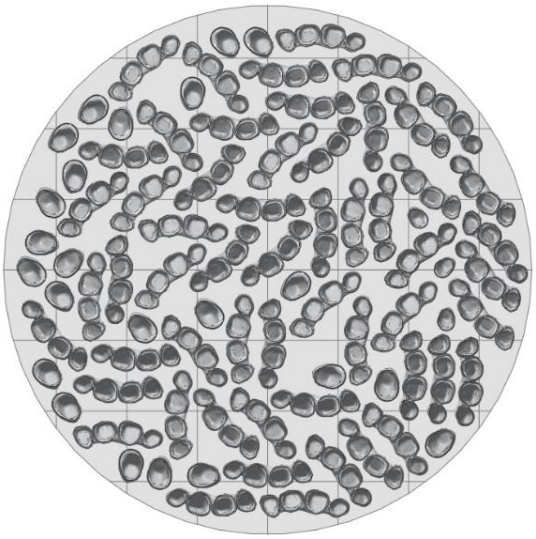


Figure 6.15 Dental crown automatically placed upwards effect

6.8 2D Nesting

When multiple parts are placed on the platform, the parts can be automatically arranged by the 2D nesting function.

Select the part, and click the “2D Nesting” command under the Home menu, or press the short keys “Shift + A”, the parameter setting dialog box will pop up, as shown in Figure 6.16. The setting dialog mainly include five parts: Selection setting, Accuracy setting, Space setting, Rotation setting and Nesting solution.

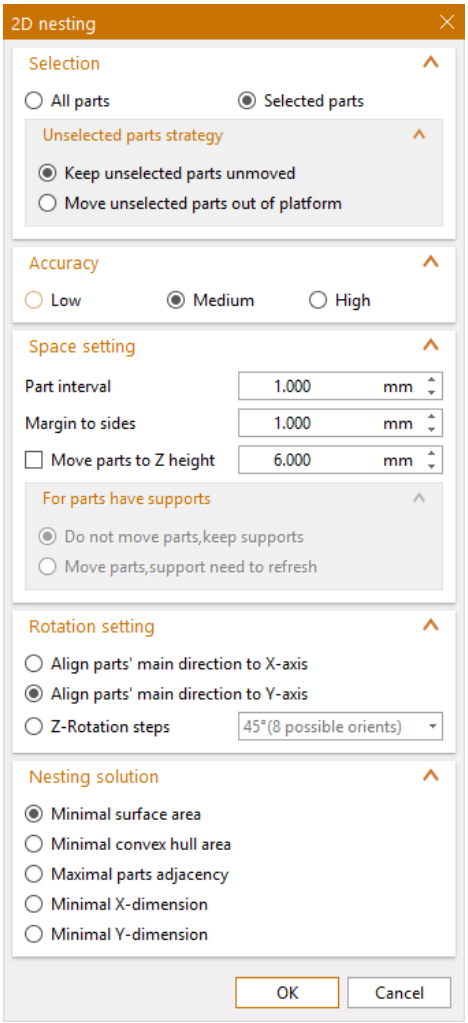


Figure 6.16 2D nesting setting

If "All parts" is selected, all parts on the platform will be placed in two dimensions. If "Selected parts" is selected, only the selected parts will be placed, and there are two placement modes for unselected parts. One is to keep the unselected parts in the original place, and the other is to move the unselected parts outside the platform.

In the accuracy setting section, you can choose the placement accuracy. The higher the accuracy, the longer the placement time.

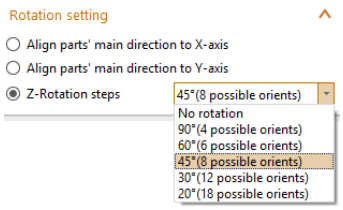
In the space setting section, you can clearly define the part interval and the distance from the part to the platform boundary. The specific parameter explanation is shown in Table 6.1.

Table 6.1 Space setting parameters

Part interval	Minimum distance between two parts
Margin to sides	Minimum distance from part to platform boundary
Move parts to Z height	Move the part to the specified Z-axis height for placement. If the part has support, you can choose not to move the part to retain the support, or move the part but refresh the support

In the rotation setting part, the specific parameters are explained in Table 6.2.

Table 6.2 Rotation setting parameters

Align parts' main direction to X-axis	If checked, during part placement, rotate the part so that its main direction is parallel to the X axis
Align parts' main direction to Y-axis	If checked, during part placement, rotate the part so that its main direction is parallel to the Y axis
Z-Rotation steps	Provides multiple step sizes to rotate around the Z axis  No rotation: the part does not rotate around the Z axis 90 °: The rotation step is set to 90 °, which provides four possible placement positions during placement 60 °: The rotation step is set to 60 °, which provides six possible placement positions during placement 45 °: The rotation step is set to 45 °, which provides eight possible placement positions during placement 30 °: the rotation step is set to 30 °, which provides twelve possible placement positions during placement 20 °: The rotation step is set to 30 °, which provides eighteen possible placement positions during placement

In the Nesting solution section, specific parameter explanations are shown in Table 6.3.

Table 6.3 Nesting solution parameters

Minimal surface area	Minimal total surface area for parts
Minimal convex hull area	Minimal convex hull area for parts
Maximal parts adjacency	Place the parts next to each other best
Minimal X-dimension	Minimal total surface area in the X direction
Minimal Y-dimension	Minimal total surface area in the Y direction

After the setting is completed, click "OK", the parts on the platform will be placed in two dimensions according to the settings. After placement, the effect picture is shown in Figure 6.17.

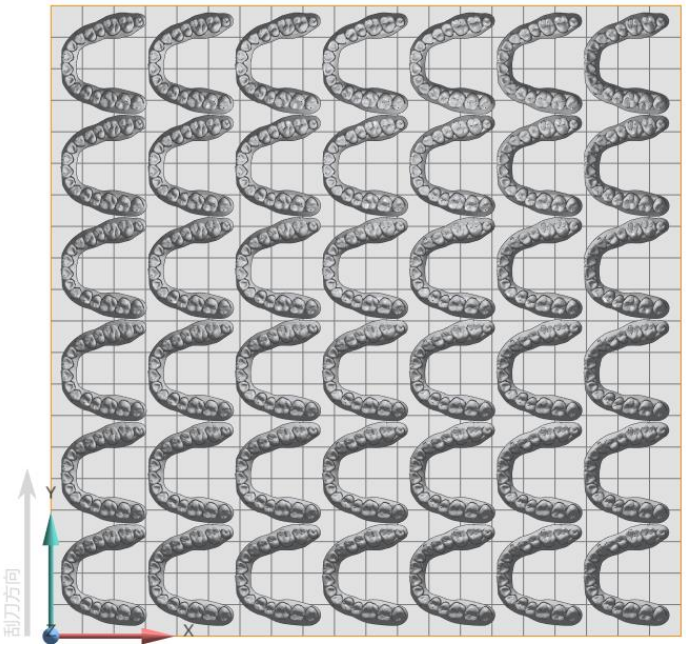


Figure 6.17 Two-dimensional nesting effect

6.9 3D Nesting

6.9.1 3D Nesting

The 3D nesting function ensures an optimal load of the sintering machine(s) when using the selective laser sintering technology. Considering the geometry of the parts, the software

maximizes the construction and packaging of parts on the platform and minimizes build time. At the same time, the software ensures that no collisions will occur between any parts after placement.

Select the part, and click the "3D Nesting" command in the 3D Nesting drop-down menu under the Home menu. the parameter setting dialog box will pop up in the display window, as shown in Figure 6.18. The settings dialog includes four sections: Selection setting, Accuracy setting, Space setting and Rotation setting.

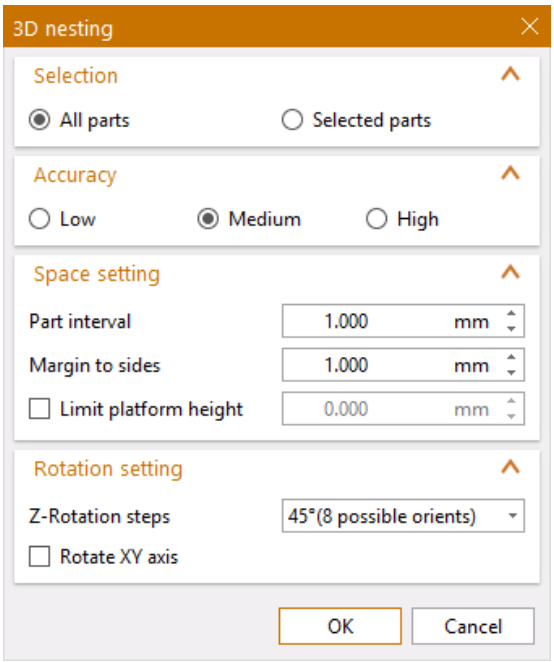


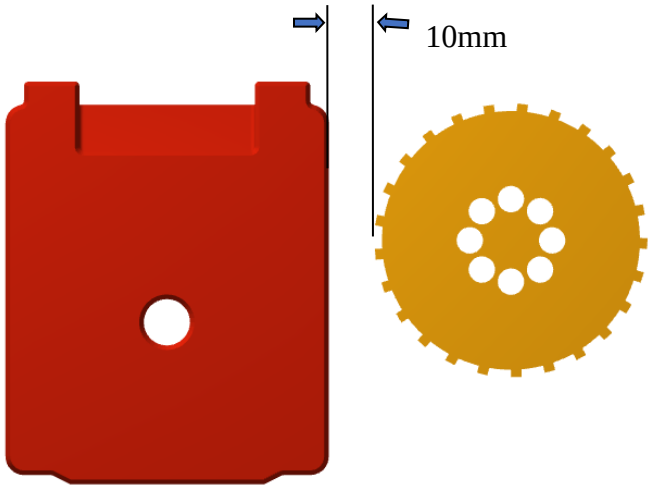
Figure 6.18 3D nesting setting

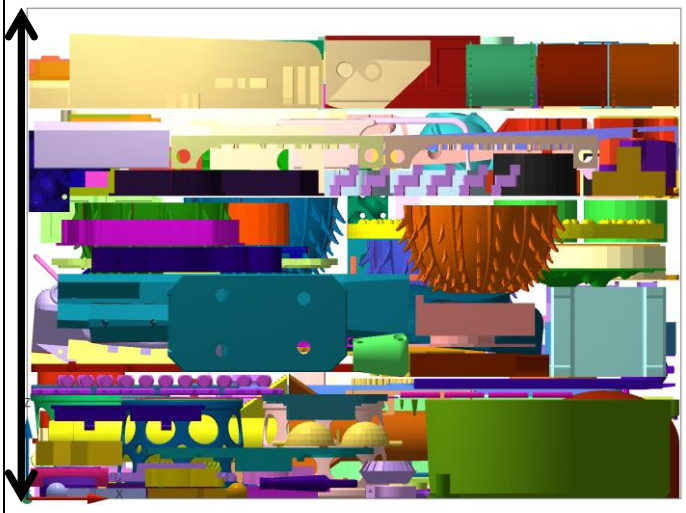
If "All parts" is selected, all parts on the platform will be placed in three dimensions. If "Selected Parts" is selected, only selected parts will be placed, and unselected parts will be placed outside the platform.

In the accuracy setting section, you can choose the placement accuracy. The higher the accuracy, the longer the placement time.

In the space setting section, you can clearly define the part spacing and the distance from the part to the platform boundary. The specific parameter explanation is shown in Table 6.4.

Table 6.4 Space Settings parameters

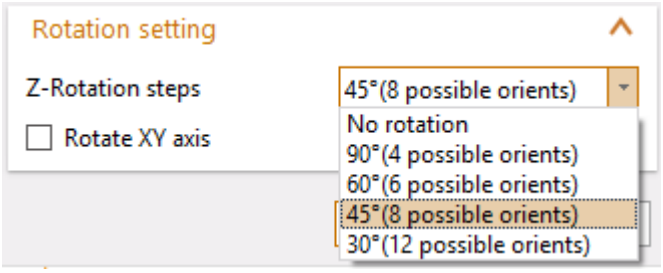
Part interval	This value refers to the minimum distance between two pre-processed parts. 
Margin to sides	The minimum distance from the parts to the platform boundary after packing. 

Limit platform height	Check the box to set the maximum height of the part placed on the platform. If not checked, the default is placed within the entire height range of the platform. Do not check the selection box:  The platform height is limited to 200mm:  The platform height is limited to 100mm:

In the Rotation settings section, the specific parameters are explained in Table 6.5.

Table 6.5 Rotation Settings parameters

Z-Rotation steps	Provides a variety of step sizes for rotation around the Z axis  No rotation: The part does not rotate around the Z axis. 90° : The rotation step is set to 90° , and four possible placement positions are available during the placement process. 60° : The rotation step is set to 60° , and six possible placement positions are available during the placement process. 45° : The rotation step is set to 45° , and eight possible placement positions are available during the placement process. 30° : The rotation step is set to 30° , and twelve possible placement positions are available during the placement process.
Rotate XY Axis	After checking, the part is rotated around the XY axis during the placement process and by default, it is checked.

After the setting is completed, click “Confirm”, the parts on the platform will be placed in three dimensions according to the settings, and the effect picture is shown in Figure 6.19.

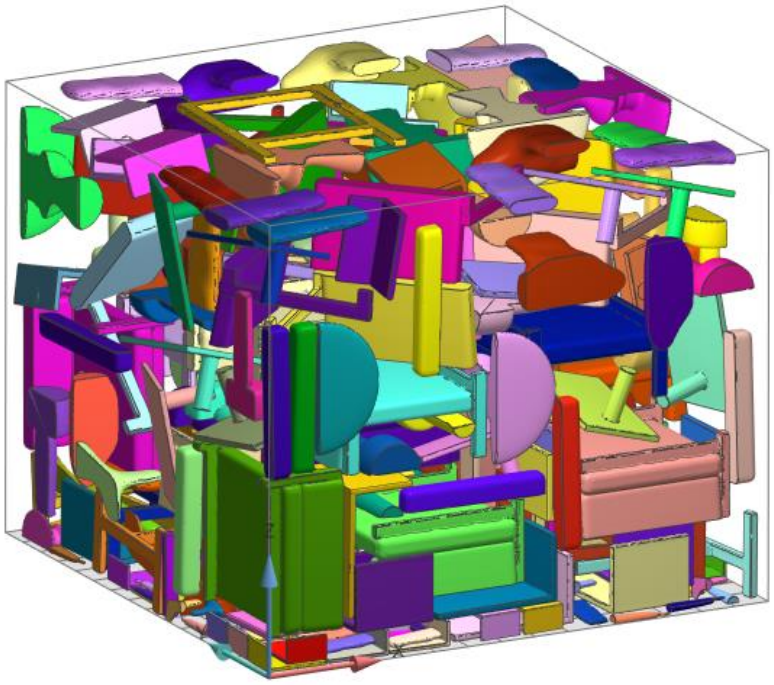


Figure 6.19 3D nesting effect

After the parts are placed, you can check whether there is a collision between the placed parts by using the Collision Detection command under the Analysis menu (refer to Section 10.1).

6.9.2 Sub-nester

Sub-nesting is pre-nesting the selected parts before creating the sintering box. In this way, small and fragile parts can be nicely nested well.

Select some parts, and click the "Subnester " command in the 3D Nesting drop-down menu under the Home menu, and the sub-nesting parameter setting dialog will pop up in the display window, as shown in Figure 6.20.

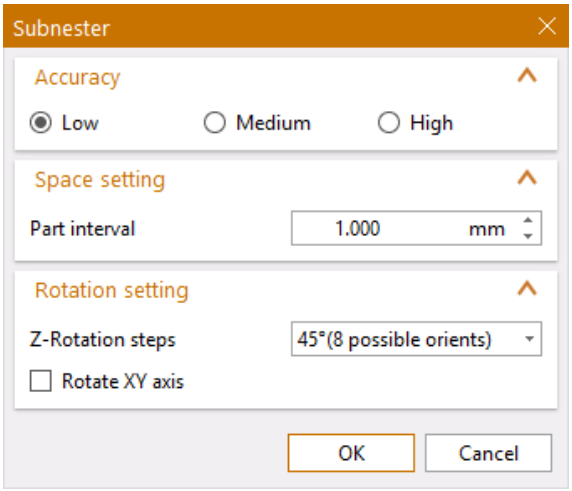


Figure 6.20 sub-nester setting

In the Accuracy setting section, you can choose the placement accuracy. The higher the accuracy, the longer the placement time.

In the Space settings section, you can define the minimum distance between parts.

In the Rotation setting section, you can set the rotation step around the Z axis and whether to rotate the XY axis.

After the setting is completed, click "OK". The selected parts on the platform will be placed in sub-nested according to the settings. The effect picture after the placement is shown in Figure 6.21.

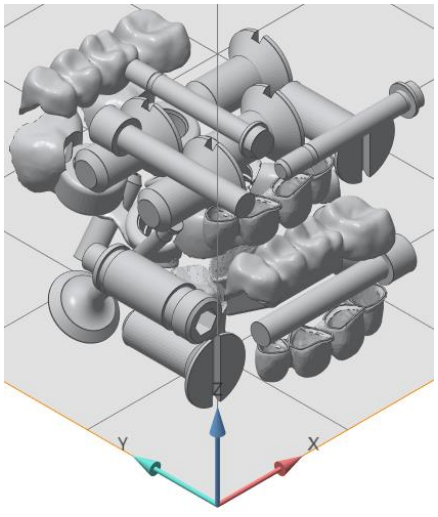


Figure 6.21 Sub-nester setting

6.9.3 Rectangle SinterBox

Using the rectangle sinter box function, small and fragile parts can be protected in a small box for easy removal from the powder later.

Select some parts, click the "rectangular sintering box" command in the 3D Nesting drop-down menu under the Home menu, and the dialog for setting the parameters of the rectangle sinter box will pop up in the display window, as shown in Figure 6.22. The setting dialog include three parts: Space setting, Perforations and Tag.

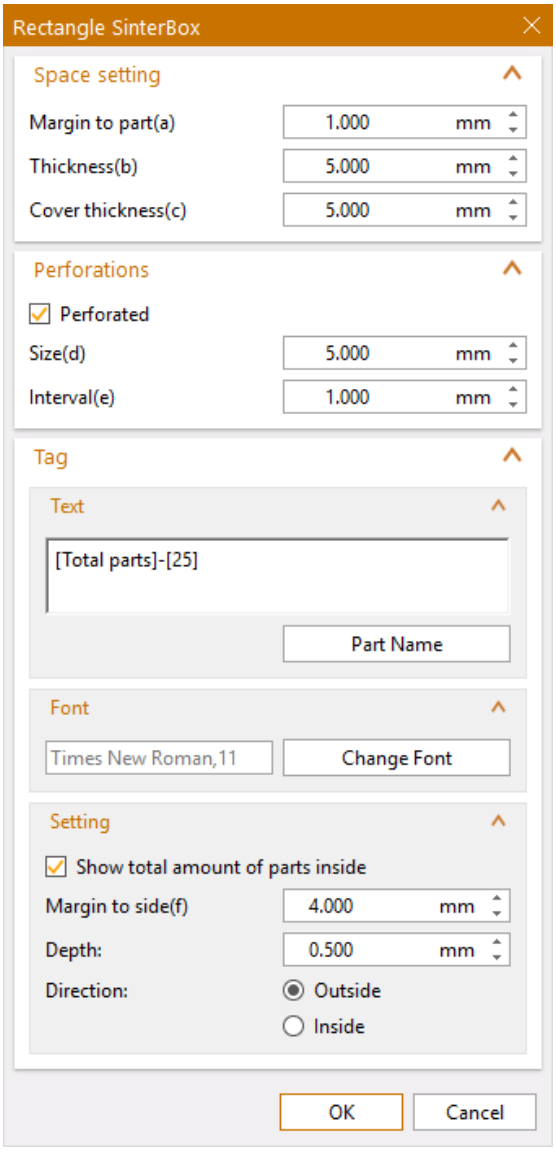


Figure 6.22 Rectangular sinter box setting

In the Spacing setting part, you can clearly define the part gap and the thickness of the sintering box. The specific parameters are explained in Table 6.6.

Table 6.6 Space setting parameters

Margin to part	The minimum distance between (the bounding box of) a part and the edge of the sinter box
Thickness	The wall thickness of the box, except the cover thickness
Cover thickness	The wall thickness of the cover of the box

In the Perforation setting section, the specific parameter explanation is shown in Table 6.7.

Table 6.7 Perforation setting parameters

Perforated	If checked, the sintering box can be perforated, so that it is easier to let the powder flow out before removing the parts, if not checked, no perforation
Size	Define the size of the perforation
Interval	The distance between two perforations

In the label setting section, in order to easily find which parts are located in the sinter box, you can create labels. The specific parameter explanations are shown in Table 6.8.

Table 6.8 Label setting parameters

Text		Type here the text you'd like to have on the Sinterbox.
Part Name		This function will enter the first part names as text to the label.
Change Font		The font can be changed in a dialog when you push the Change Font button.
Show total amount of parts inside		The total amount of parts inside the Sinterbox is displayed on the label.
Margin to side		Distance between text and edge of the box.
Depth		The label will be put on the part in relief with the given depth.
Direction	Inside	The letters will be engraved in the Sinterbox.
	Outside	The letters will be on the Sinterbox

After the setting is completed, click "OK". The selected parts on the platform will build a rectangular sinter box according to the settings. The effect picture is shown in Figure 6.23.

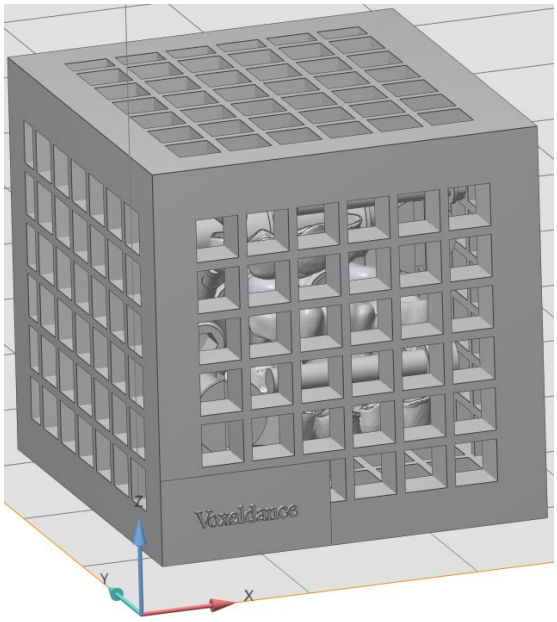


Figure 6.23 Effect of rectangular sinter box with perforations

7 Part Edit

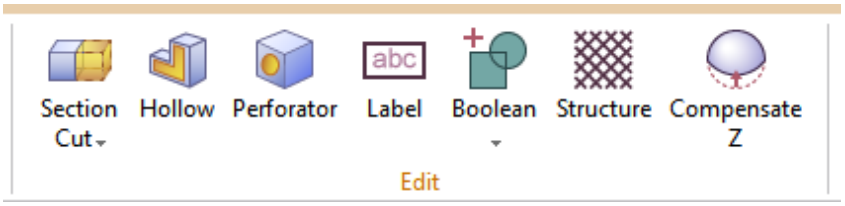


Figure 7.1 Part Edit menu

7.1 Section Cut

7.1.1 Section Cut

Click the "Section Cut" command in the Section Cut drop-down menu under the Home menu, and the section cut setting dialog box will pop up in the display window, as shown in Figure 7.2.

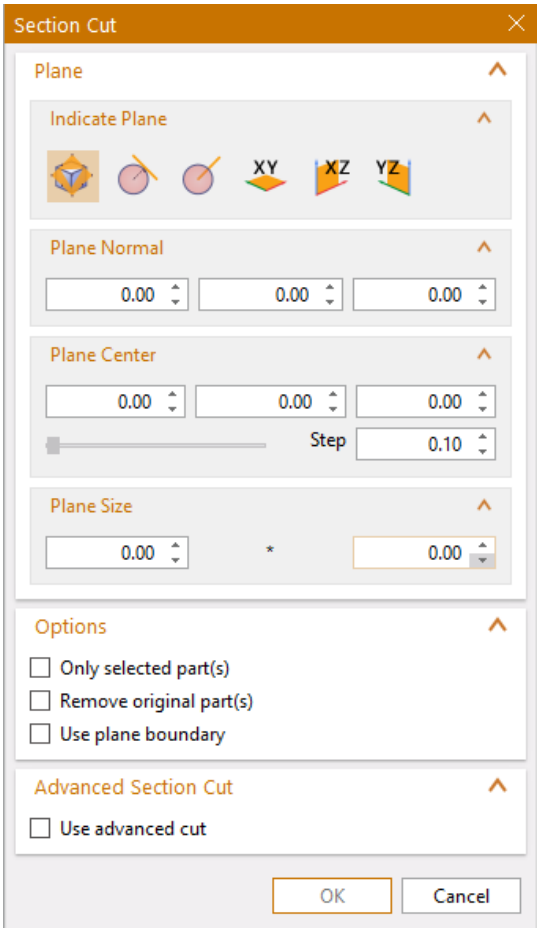


Figure 7.2 Section cut setting

If "All Parts" is selected, all parts on the platform will be cut. If "Selected Parts" is selected, only the selected parts will be cut, even if the cutting plane is inserted on unselected parts, do the corresponding cutting.

To place the cutting plane, there are six options: clicking three points on the part to define the plane and then places it, placing the plane along the direction of the part surface, placing the plane perpendicular to the surface direction of the part, and placing the plane parallel to the X-Y, X-Z, Y-Z plane directions. By default, the cutting plane is blue and is marked by dots at its center, corner and edge center.

You can select the four ways to insert the cutting plane by clicking the following icons respectively:

 : Creating a cutting plane by three points. After clicking on any three points on the part, and inserting the cut plane defined by these three points, then you can preview the plane when you place the third point. The last point selected is the center of the plane, as shown in Figure 7.3.

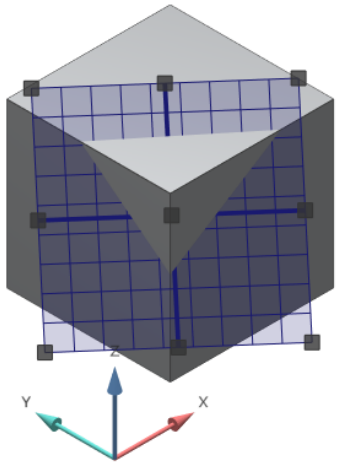


Figure 7.3 Create a cutting plane at three points

 : Creating a tangent plane that is tangent to the surface. After clicking on any point on the part, you can create a cutting plane parallel to the triangle where the point is located, as shown in Figure 7.4.

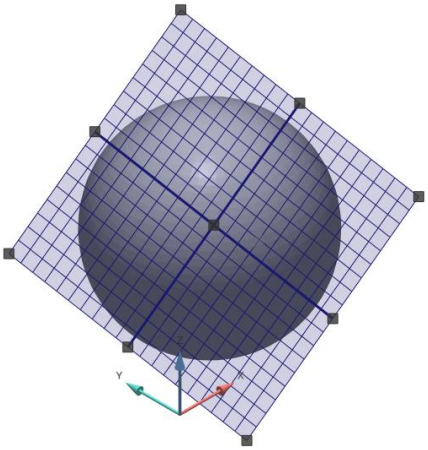


Figure 7.4 A cutting plane tangent to the plane

 : Creating a cutting plane perpendicular to the surface. After clicking on any point on the part, you can create a cutting plane perpendicular to the triangle where the point is located, as shown in Figure 7.5.

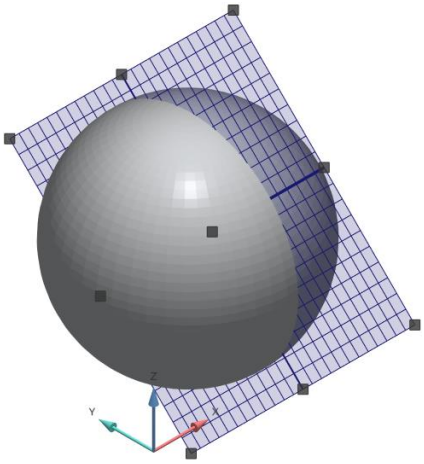


Figure 7.5 A cutting plane perpendicular to the plane

: Creating a cutting plane parallel to the XY, XZ, and YZ planes. Taking an example parallel to the X-Y plane, after clicking on any point on the part, you can create a cutting plane parallel to the X-Y plane where the point is located, as shown in Figure 7.6.

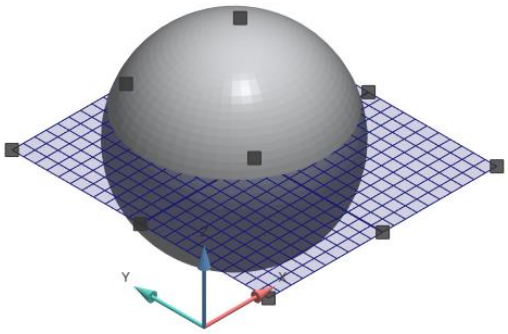


Figure 7.6 A cutting plane parallel to the XY plane

The plane can be moved and rotated and its size can also be changed by dragging and dropping the mouse through the points on the plane.

If you use the mouse to click and drag the center point of the plane, as shown in Figure 7.7, you can move the entire plane freely; if you click the points on the edge and the corners, you can rotate the plane; if you hold down the Ctrl key, then click and drag the points on the edges and the corner, you can change the size of the plane.

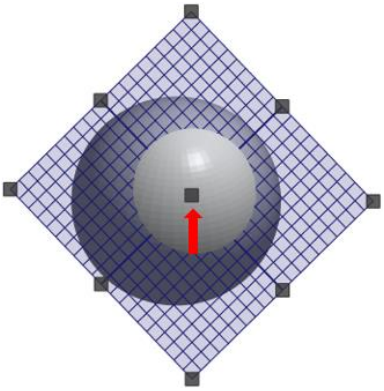


Figure 7.7 Plane center point

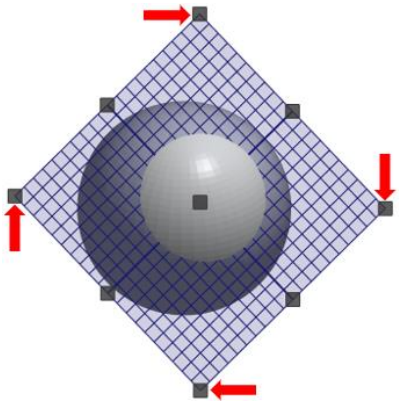


Figure 7.8 Plane corner point

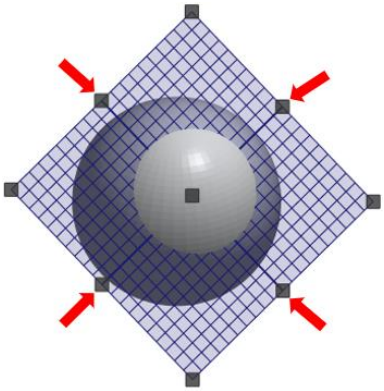


Figure 7.9 Edge center point

If you move the point on the corner, as shown in Figure 7.8, the plane will rotate about the axis perpendicular to the plane, and the plane stays in its original position, only its edges and corners move. If you move the center point on the edge, as shown in Figure 7.9, the plane will rotate along the clicked edge axis paralleled to the edge. You can adjust the cutting surface to any position by rotating its each side.

After inserting the cutting plane, you can also set the value of the cutting plane in the settings dialog, so that you can more accurately fine-tune and position the plane.

In the Options setting, there are 2 options, as shown in Figure 7.10. You can enable or disable these functions by checking or unchecking them.

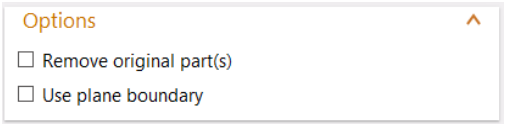


Figure 7.10 Plane information parameter setting

If "Remove original part(s)" is checked, you can delete the original part and replace it with the cut part. If you do not remove them, they appear behind the cut part after cutting.

If "Use plane boundary" is checked, you can only cut along the outer edge of the plane. If this option is not checked, the part will be cut completely along the plane, regardless of the size of the displayed cutting plane.

In the Advanced Section Cut section, check "Use advanced cut" to enable this advanced cutting function, as shown in Figure 7.11.

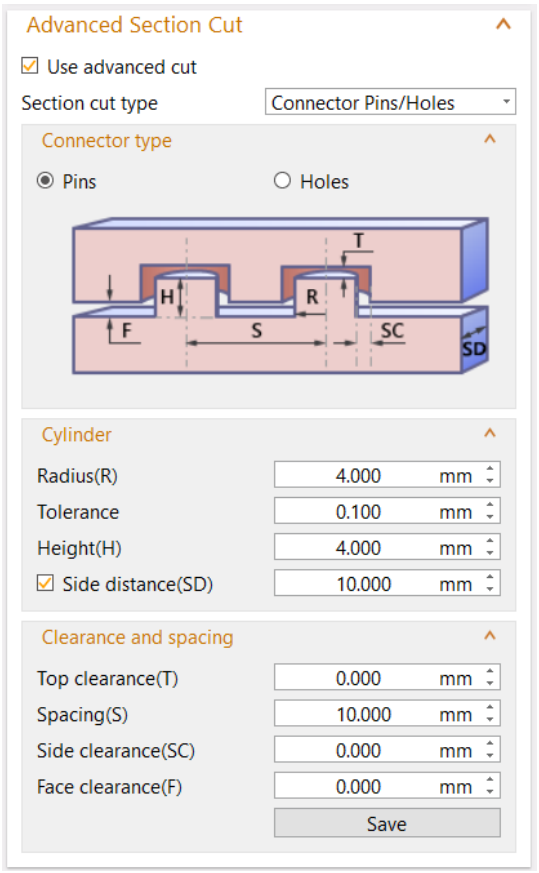
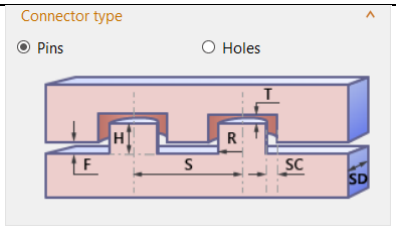
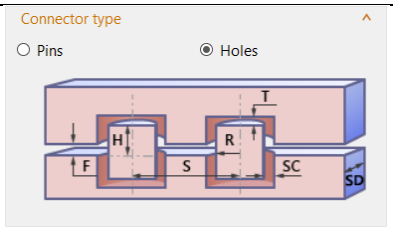


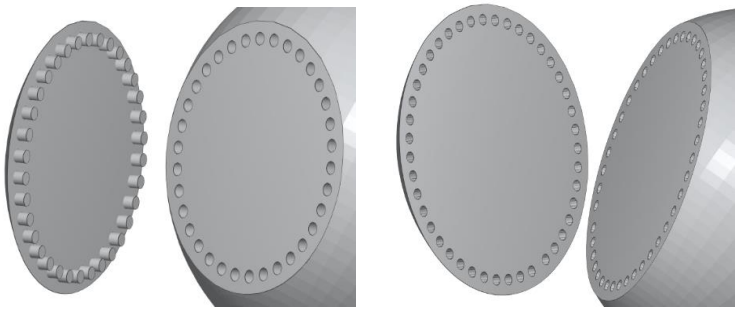
Figure 7.11 Advanced section cut setting

There are two types of section cut, pins/holes setting and lap joint setting.

a. Pins/ Holes type setting

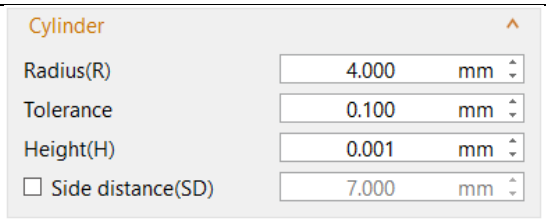
Table 7.1 Pins/Holes type

	
Pins：The connection ways of the two sides after the section cutting are different, one side is pin connection, and the other is hole connection Holes：The connection ways of the two sides after the section cutting are same, both sides are hole connection	



Pins cut　　Holes cut

Table 7.2 Cylinder settings of the Pins/Holes cut

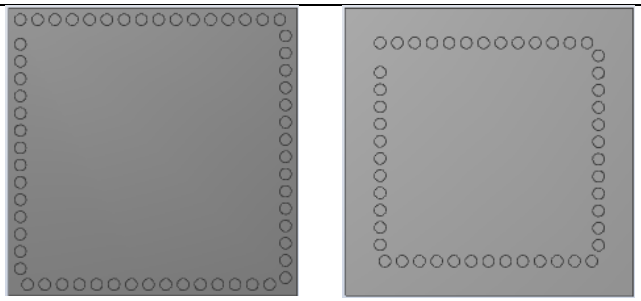


Radius：The radius of pins after cutting

Tolerance：The tolerance parameter has a great influence on the formation of the triangles after cutting

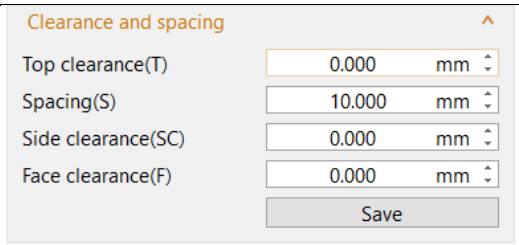
Height：The height of the pins after cutting

Side distance：The distance from the part boundary to the pin



Side distance =4mm　　Side distance =12mm

Table 7.3 Clearance settings of the Pins/Holes cut

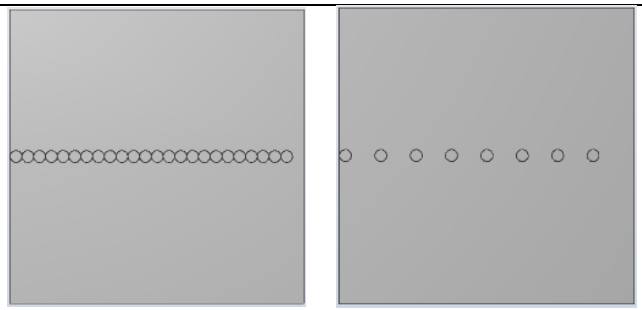


Top clearance：The top clearance of the pins

Spacing：The distance of the pins

Side clearance：The side clearance of the pins

Face clearance：The face clearance along the cutting direction and it facilitates rapid recovery of the part after cutting

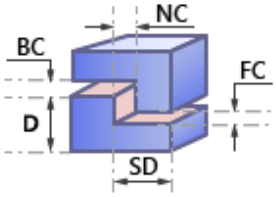


Spacing =4mm　　Spacing =12mm

After setting the parameters, click the “Save” button to save the parameter settings.

b. Lap joint setting

Table 7.4 Lap Joint type



After cutting, the two sides of the connection are differently. One side is convex connection and the other side is concave connection.

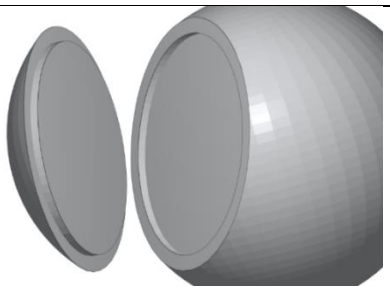
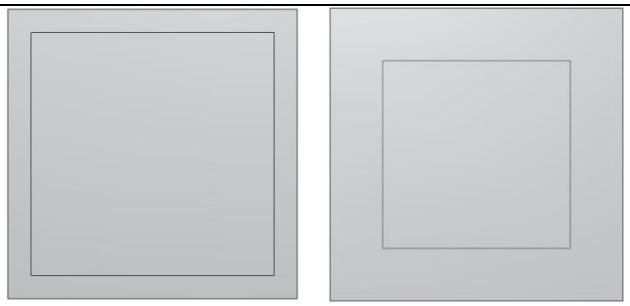


Table 7.5 the Parameter settings of the Lap Joint

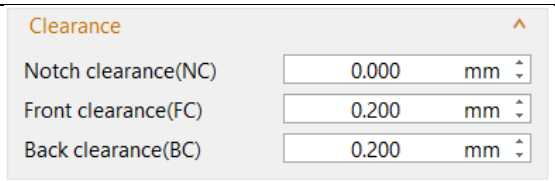
Notch depth: Defines the height between the top and bottom incision plane.

Side distance: The distance between the wall of the part and the step in the cutting surface.



Side distance =10mm Side distance =20mm

Table 7.6 the Clearance settings of the Lap Joint



Notch clearance: Make a little gap along the cut. This way the two parts can easily slide into each other if you have to assemble them.

Front clearance: Clearance added in the direction perpendicular to the section. Back clearance: Clearance added in the direction parallel to the section.

Finally, click OK to perform the part cutting. The cut part name is displayed in the parts list on the left.

7.1.2 Polygon Cut

Click the "Polygon Cut" command in the Section Cut drop-down menu under the Home menu, and the Polygon Cut Settings dialog box will pop up in the display window, as shown in Figure 7.12.

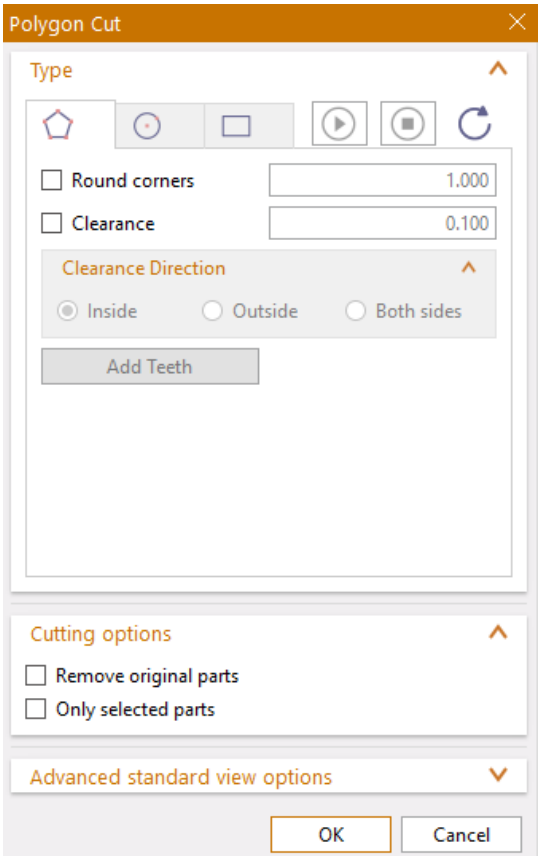


Figure 7.12 Polygon cut setting

If "All Parts" is selected, all parts on the platform will be cut. If "Selected Parts" is selected, only selected parts will be cut. Even if the cutting plane is inserted on non-selected parts, it will not do the corresponding cut.

In the first polygon mode  , you can create and edit polygons. With left-clicks on the part, you set corner points for the cutting line. And in the default color setting, the points appear as a black square and the cutting line is red. When you set the third corner point, the first and last point are connected, so that the cutting line becomes a triangle. More corner points make it a polygon, and the added corner points are always connected to the two corners points attached to the closest line. Left click on the corner point and dragging the mouse can change the location of the corner point, and right click on the corner point can delete the point in the pop-up menu.

In the following parameter settings, there are Round corners and Clearance settings. If “Rounded corners” is checked, the corners of the polygon will be smoothed. The input value defines the radius of the rounded corners. If the “Clearance” is checked, you can define its thickness, and then you can choose whether to add space inside, outside or both sides of the cutting line. By adding a clearance, the space between the two cutting lines can also be cut off, which is helpful for that the contours swell up a little in the 3D printing. The effects of Round corners and Clearance settings are shown in Figure 7.13.

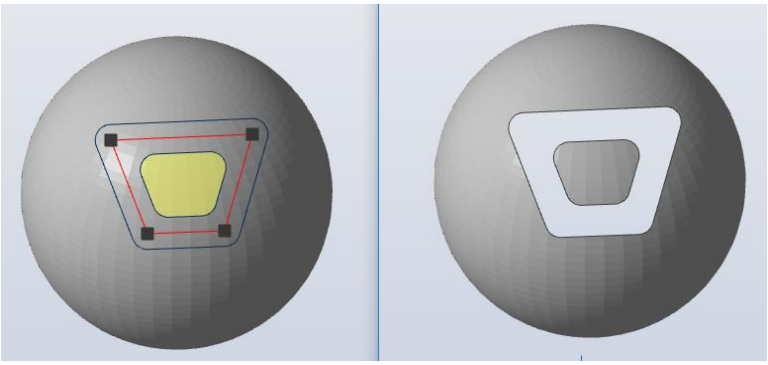


Figure 7.13 Left: Rounded corners on both sides, Right: The cut part

After the polygon is created on the part, the “Add-teeth” button is in an editable state. Click the “Add Teeth" button to enter the boundary shape setting module. The setting dialog pops up in the display window, as shown in Figure 7.14.

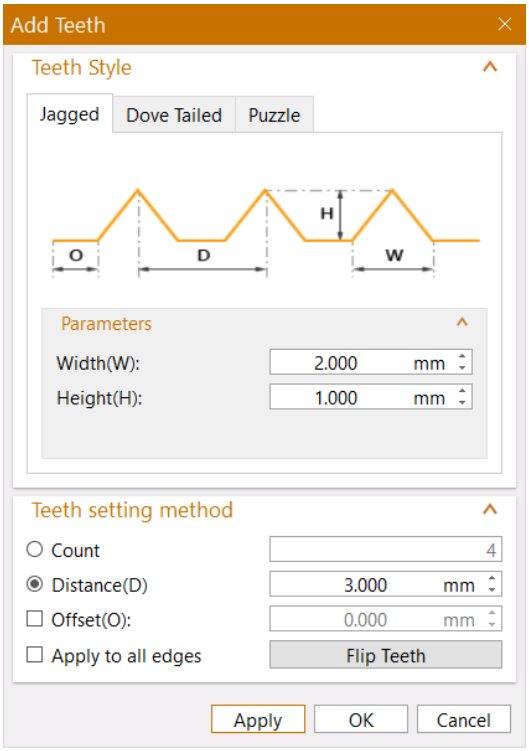
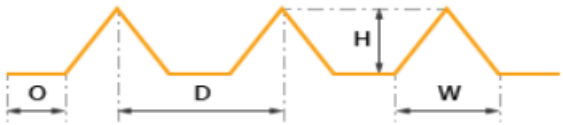


Figure 7.14 Add teeth setting

Under the Teeth style tag, you can select the type of teeth line. There are three main types, and each type and its corresponding parameters are explained in Table 7.7.

Table 7.7 "Add Teeth" parameter settings

Teeth line type	Jagged	
	Dove tailed	

	Puzzle	
Width	The width of the teeth structure	
Height	The height of the teeth structure	
Count	Setting the specific number of teeth structures on a boundary	
Distance	Distance between two teeth structures	
Offset	The distance from the beginning of the cutting boundary to the boundary of the first toothed structure	
Flip Teeth	After clicking Flip Teeth, the tooth profile on the cutting boundary is inverted	
Apply to all edges	When checked, all cutting edges are toothed	

After selecting the appropriate teeth type and setting the corresponding parameters, you can add a teeth line to a cutting line selected by the left mouse, then click the Apply button to apply. The tooth addition of each cutting line is independent of each other. Right-click the teeth line and click "Delete selected teeth" to delete the teeth structure added to this cutting line. Click "Delete all teeth" to delete the teeth structure on all cutting lines. Finally, click the Confirm button to exit the add teeth module, as shown in Figure 7.15.

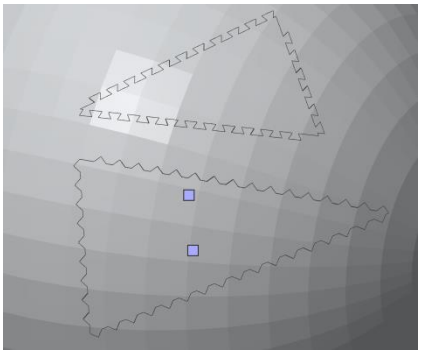


Figure 7.15 Effect of adding different teeth style

In the above plane cutting and polygon cutting operation, after clicking a cutting icon, the mouse will carry the corresponding icon, such as  . During the execution of this function, the mouse will always carry the icon until it switches to other functional operation icons.

With a click on the circle icon  , the predefined shape of a circle cut will be previewed on the part. Then you can adjust its size by entering the exact radius value, or by dragging the round edge line with the mouse. The tolerance value determines the deviation between the circle drawn by the program (a polygon) and a real circle. It is the distance along the radius perpendicular on and through the middle of one of the edges of the polygon between the cross point of the radius with the polygon and the cross point of the radius with the circle. The higher the tolerance, the bigger the deviation. Besides, Notch and Clearance can be set, as shown in Figure 7.16.

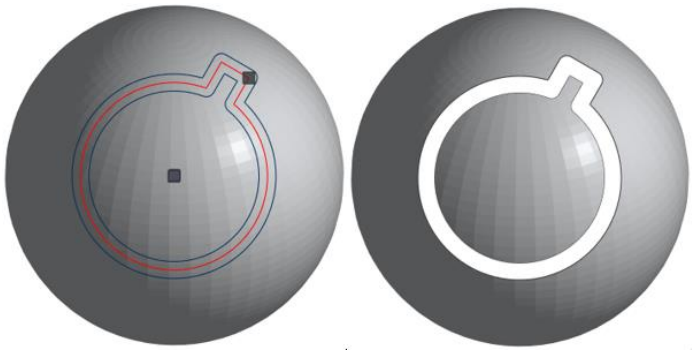


Figure 7.16 Left: Notch and clearance on both sides, right: After cutting

With a click on the rectangle button  , the predefined shape of a rectangle cut will be previewed on the part. Then you can adjust its size by entering the exact length and width, or by dragging the center point displaying on the rectangle edge with the mouse. You can also set Notch and Clearance, the effect is shown in Figure 7.17.

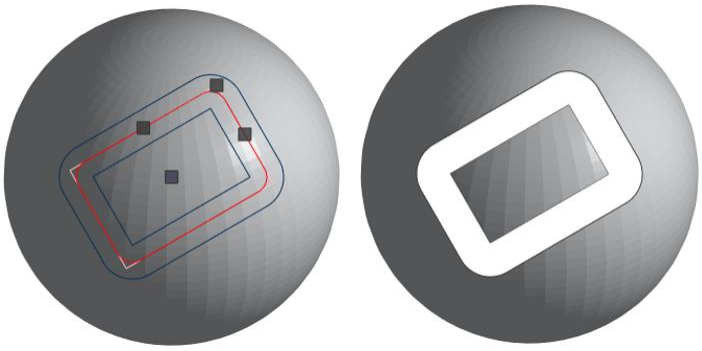


Figure 7.17 Left: Round corner and clearance on both sides, right: After cutting

With a click on the Delete button , the added cut corners will be deleted and automatically return to the edit polygon mode.

In order to observe the cutting effect more clearly, you can click the preview button to preview the cutting effect. The part to be cut is in yellow color and clicking the stop button to stop the preview operation. Click the reset button to clear all cutting line segments on the part. After setting the cutting point, you can use the right mouse button to rotate the perspective to know the shape of the cutting line at each viewing angle.

In the Cutting option, if you check "Remove original part", you can delete the original part and replace it with the cut part. If you do not remove them, they will appear in front of the cut part after the cut.

Clicking the “Advanced standard view options” drop-down menu at the bottom of the dialog, you can enter the advanced standard view options settings interface. After clicking one of the six standard viewing angles: front, rear, top, bottom, left, right, the parameter settings in advanced standard view options are displayed as editable. Take clicking on the top view for example, as shown in Figure 7.18. After checking the “Cut till Z=” box, you can only cut to the specified Z axis height; after checking the “Show coordinate” box, the coordinate information of all the cutting corners will be displayed, and the coordinate information under the “Move selected point to” is in an editable state, you can change the coordinates of a selected corner point; after clicking the “Add Point” button, the Coordinates dialog will pop up, then you can add corner points at the specified position, as shown in Figure 7.19.

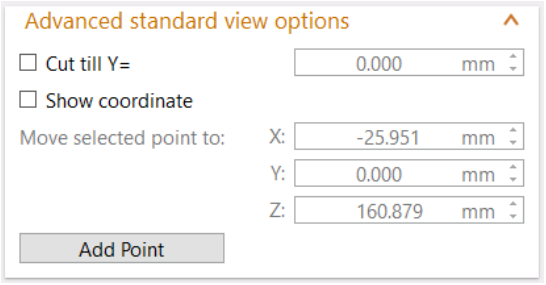


Figure 7.18 Advanced standard view options Settings

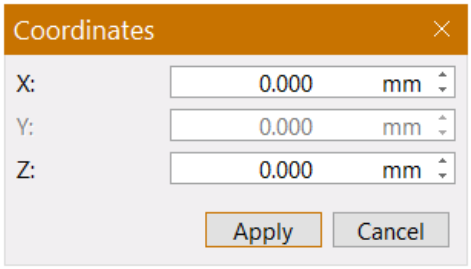


Figure 7.19 Add corner points at the specified position

7.2 Hollow

7.2.1 Hollow

By creating a concentric (identical shape) new shell at a certain thickness inside or outside the part and merging with the original part, you can create a hollow part with the same appearance.

Select the part, and click the "Hollow" command in the Hollow drop-down menu under the Home menu, and a dialog box for setting parameters will pop up, as shown in Figure 7.20. The Wall Thickness parameter specifies the distance over which the triangles of the original shell get an offset in order to generate a hollow part. The Accuracy parameter specifies the hollow accuracy, and the Accuracy value is lower, the number of triangular mesh is greater, and the graphics accuracy, the computational complexity is higher. The Direction option specifies where the new shell is created: Inside refers to create an inner shell inside surrounded by the original part; Outside refers to creating the outer shell on the outside and enclosing the original part. Finally click the Confirm key to complete the hollow operation, the original part will be overwritten.

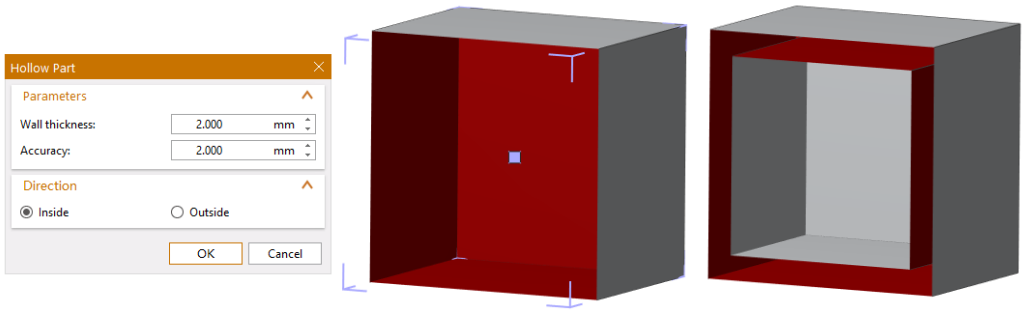


Figure 7.20 Hollow Operation (left) and Result (right)

7.2.2 Shell

Mark the surface on the part to delete, the rest of part will be hollowed out.

Select the part, mark the surface to be deleted, and click the "Shell" command in the Hollow drop-down menu under the Home menu, and a dialog box for setting parameters will pop up, as shown in Figure 7.21. The Wall Thickness parameter specifies the distance over which the triangles of the original shell get an offset in order to generate a hollow part. The Accuracy parameter specifies the hollow accuracy, and the Accuracy value is lower, the number of triangular mesh is greater, and the graphics accuracy, the computational complexity is higher. Finally, click the “OK” button to complete the shell operation, and the original part will be overwritten.

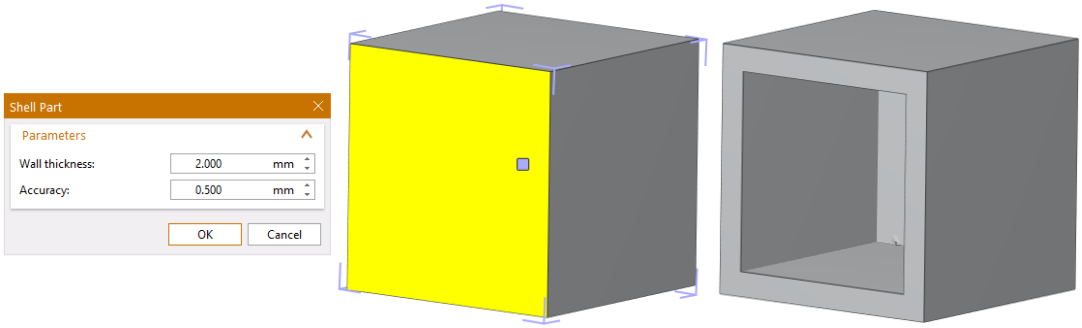


Figure 7.21 Shell Operation (left) and Result (right)

7.3 Extrude

Use the extrude function to extrude triangles in the certain direction, allowing you to quickly reshape parts without using CAD software.

Select the part, and click the "Extrude" command in the Home menu. A dialog box for setting parameters will pop up, as shown in Figure 7.22.

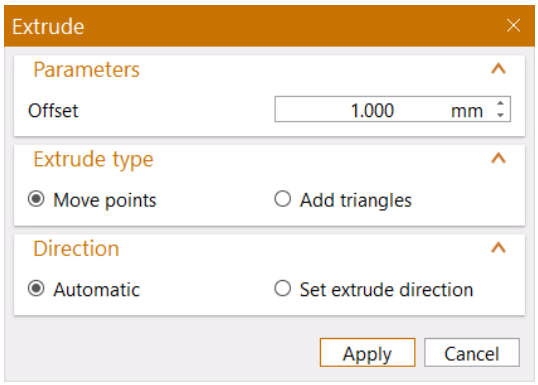
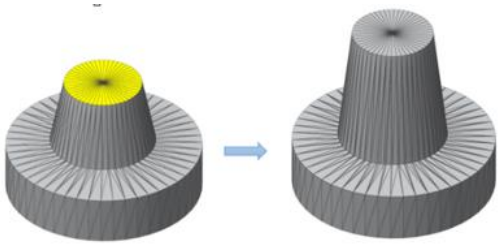
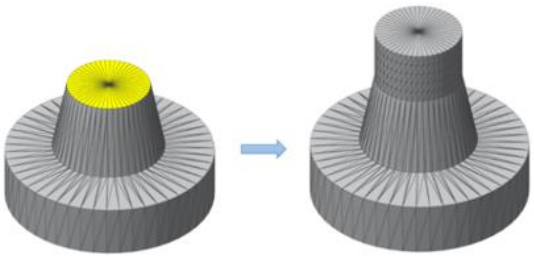


Figure 7.22 Extrude setting

Table 7.8 Extrude parameters setting

Offset		Specify the extrude offset.
Extrude type	Move points	The triangles adjacent to the selected triangles will be redrawn. They are stretched like shown in the figure below. The common points are moved. The area of the surface formed by the triangles that are extruded will stay the same after extrusion. The slope of the adjacent triangles will change a bit. 
	Add triangles	The triangles adjacent to the triangles that are extruded stay the same. The common points of the selected and adjacent triangles remain on their position. The gap between the latter and the triangles that have undergone

			an offset is filled with new triangles. This is shown in the figure below. 
Direction	Automatic		The program will make the best choice of extrude direction.
	Set extrude direction	X、 Y、 Z	You can give an extrude direction coordinates in the X, Y and Z field.
		Indicate line	You can click the Indicate Line button and afterwards click on a line. The extrude offset will be in the direction of the line.
		Indicate triangles	You can click the Indicate Triangle button and afterwards click on a triangle. The extrude offset will be the direction of the normal of the triangle.

7.4 Offset

The Offset function allows offsetting the whole part or some selected triangles. The triangles are moved along a direction proper to the normal of that triangle over a defined distance, which is the same for all triangles.

7.4.1 Global offset

Select the part, and click the "Global Offset" command in the Offset drop-down menu under the Home menu. A dialog box for setting parameters will pop up, as shown in Figure 7.23.

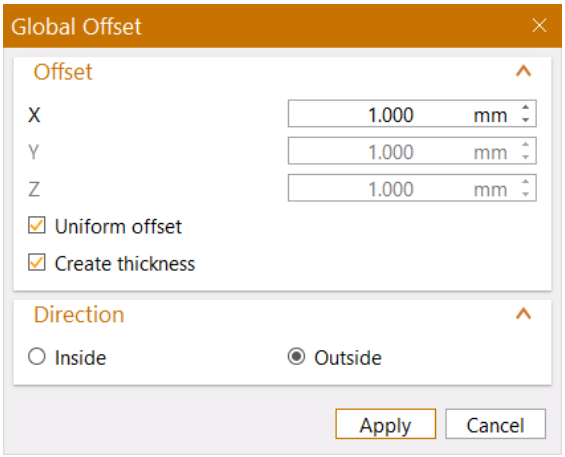


Figure 7.23 Global offset setting

Offset X, Y, Z: Enter the offset value, and each triangle will be moved along the direction of its normal over a distance set by the offset value.

Uniform offset: If you want to apply the offset to X, Y, and Z uniformly, check "Uniform offset". If you do not check "Uniform offset", you can set different offset values in the X, Y, and Z fields.

Create Thickness: Preserve the original shell and create thickness.

Direction: Choose whether the offset is inward or outward.

7.4.2 Local offset

Select the part, and click the “Local Offset” command in the Offset drop-down menu under the Home menu. A dialog box for setting parameters will pop up, as shown in Figure 7.24

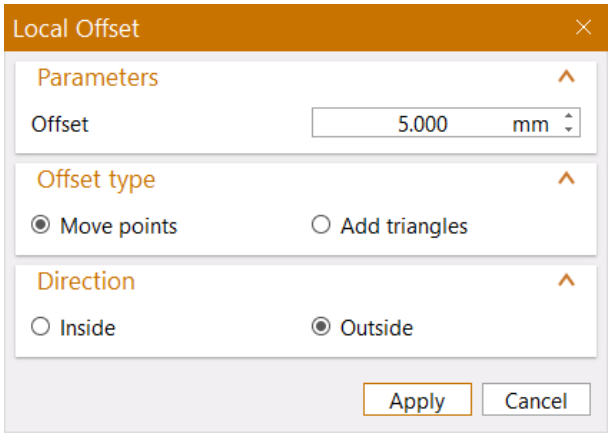


Figure 7.24 Local offset setting

Offset: Select (mark) triangles on the part and enter offset values. These triangles will be moved by the distance set by the offset value.

Move point: The triangles adjacent to the selected triangles will be changed. The common points are moved. The area of the triangles selected remains the same after the offset. The slope of the adjacent triangles will change a bit.

Add triangle: The common points of the selected and adjacent triangles remain on their position. The gap between these triangles and the triangles that have undergone an offset is filled with new triangles.

Direction: Choose whether the offset is inward or outward.

The effect of moving the points outwards and adding triangles is shown in Figure 7.25.

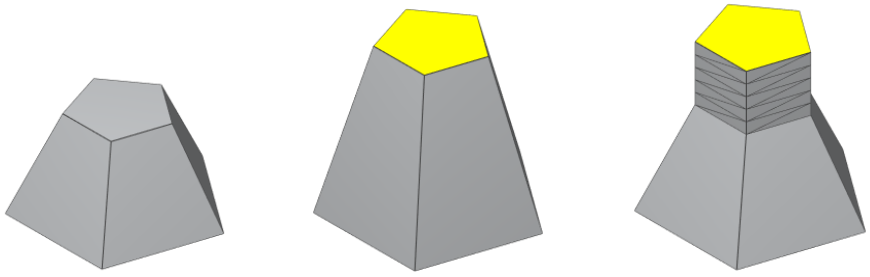


Figure 7.25 Original part (left)， Move points (middle)， Add triangles (right)

Remark: The local offset differs from the Extrude operation. In an extrude operation all triangles are moved along the same direction. In the local offset operation, the direction of offset depends on the triangle and its neighbors.

7.5 Perforator

Click the "Perforator " command under the Start menu, or the shortcut key Shift + P to open the Add Holes dialog box, as shown in Figure 7.26.

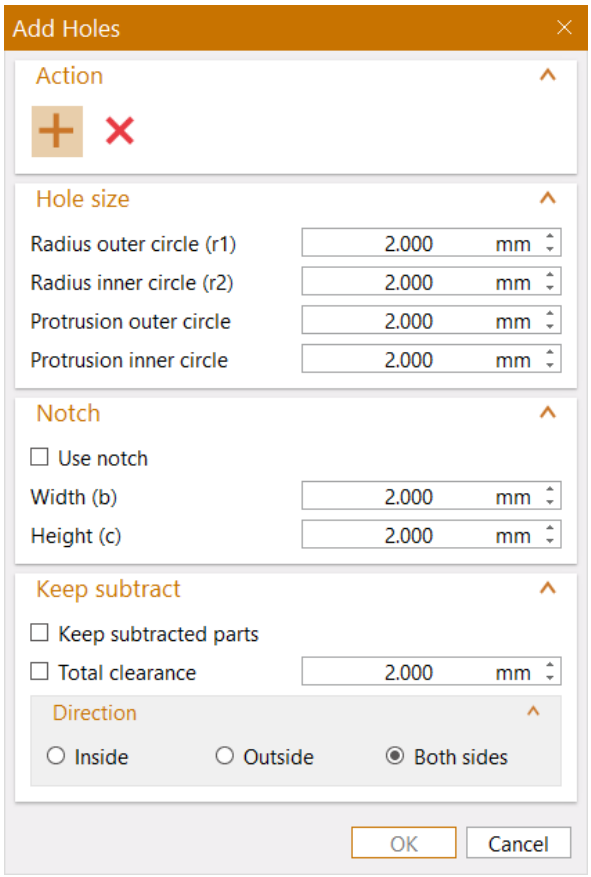


Figure 7.26 Add holes parameter setting

By clicking the "+" button at the bottom of the settings dialog, you can add a perforation part body on the part, as shown in Figure 7.27. By default, the part body is bright yellow, and after placing the part body in the location to be perforated, then clicking the left mouse button, you can add the part body on the part, and by clicking "×" button, you can delete the added part body, then clicking "Apply" button, you can achieve the perforator operation of the part and a hollow part is generated.

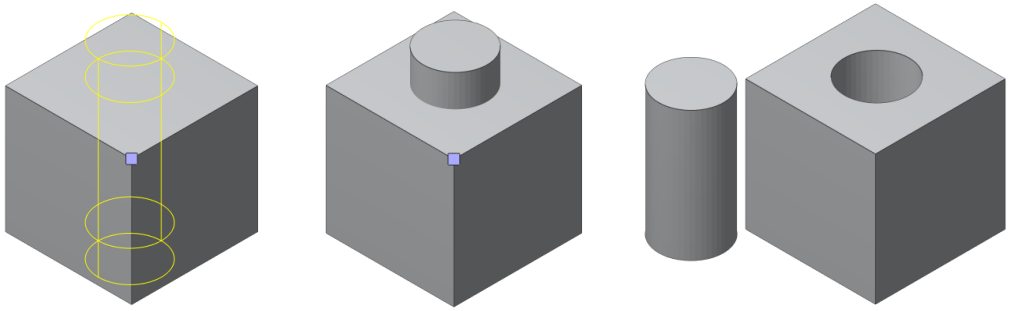


Figure 7.27 Perforator operation (left, middle) and result (right)

In the Hole size setting, you can set the radius of the outer circle, the radius of the inner circle, the protrusion length of the outer circle, and the protrusion length of the inner circle, as shown in Figure 7.28. If you check the "Use notch", you can add a notch in the perforation part body, as shown in Figure 7.29.

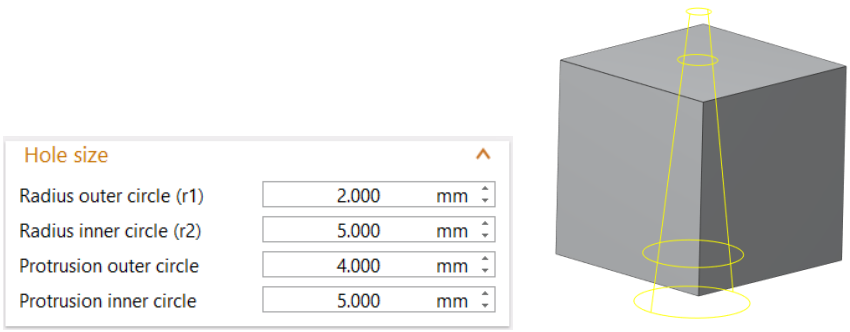


Figure 7.28 Hole size parameter setting

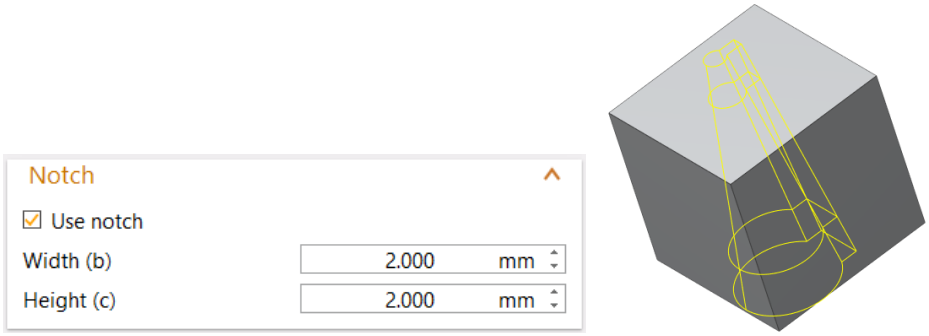


Figure 7.29 Use notch setting

For the "Keep subtracted parts" checkbox, you can choose whether to retain the perforation part body and whether to add clearance between the part and the perforation part body. By default, the checkbox is not checked, and after the perforator operation, the perforation part body is removed directly from the part and a hollow part is generated; if you need to retain the part body, check on it. If you want to make a clearance between the two sections after the perforation, you can add a gap distance, there are three types: Inside, Outside, Both sides. Inside refers that the clearance is reduced from the perforation part body; Outside refers that the clearance is reduced from the part; Both sides refers that the gap is reduced from both sides, the perforation part

body reduces half of the gap, and the part reduces half of the gap. The result of these three types are shown in Figure 7.30.

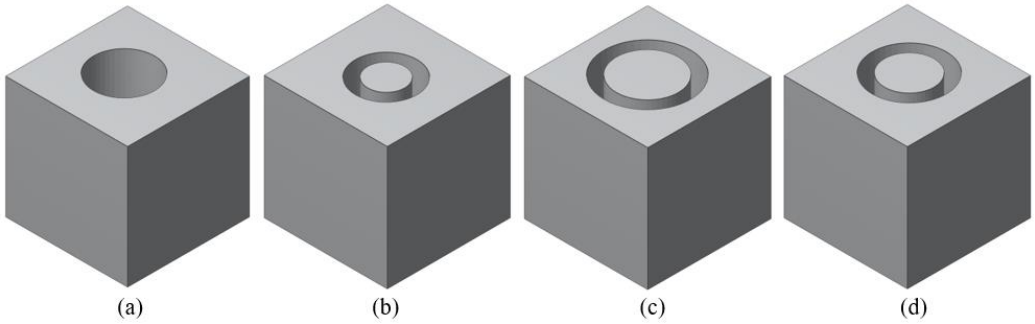


Figure 7.30 Effect after perforator: Not retained(a)， Inside(b)， Outside(c)， Both sides(d)

7.6 Prop

To avoid distortion on your part during building you can create “props” to make sure that the shape of the part is kept.

Select the part, and click the "Prop" command in the Home menu. A dialog box for setting parameters will pop up, as shown in Figure 7.31.

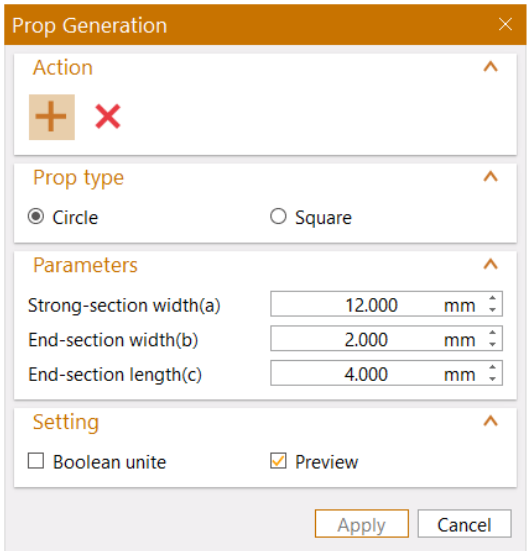


Figure 7.31 Prop generation setting

: When you click this add icon, prop can be drawn on the part, but not yet created. To better see where the prop is placed, you can check “Preview”.

✕ : Click this delete button, you can delete unnecessary prop.

Prop type: Circle / Square. The shape of the prop can change between circles or squares.

Strong-section width (a): The width of the circle/ square connector, as shown in Figure 7.32.

End-section width (b): The width of the connection (between the strong section and the part).

End-section length (c): The length of the connection between the strong section and the part.

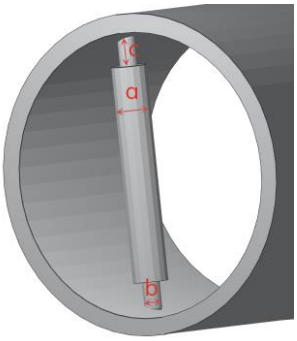


Figure 7.32 Parameter diagram

Boolean Merge: The part and prop(s) are united into one file and all surfaces are trimmed to make one shell of both parts.

Preview: While adding the props a preview is created of how the prop will be placed on the part, as shown in Figure 7.33.

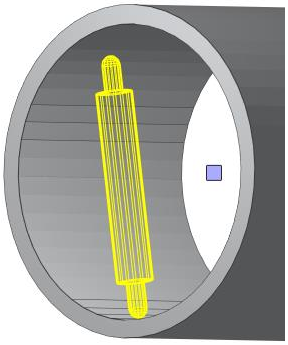


Figure 7.33 Preview of adding prop

7.7 Label

By using the Label function, you can merge the text into the part.

Select the part, and click the “Label” command in the Home menu, or press the shortcut keys Shift + L, A dialog box for setting parameters will pop up, as shown in Figure 7.34.

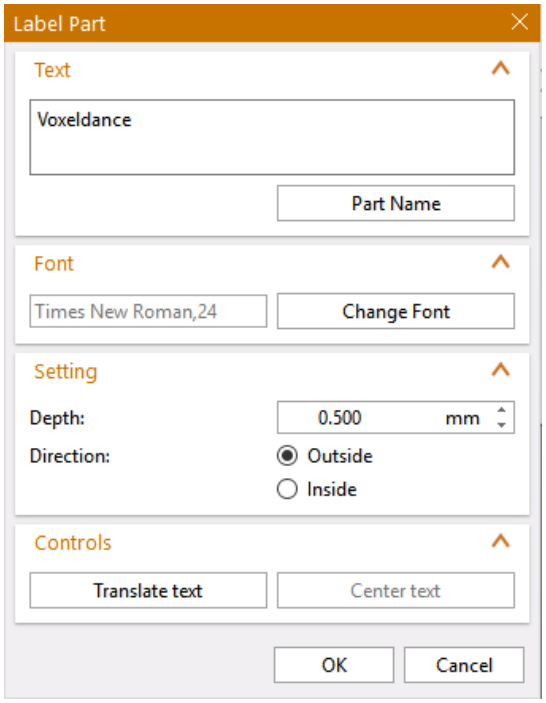


Figure 7.34 Label part setting

Under the Text tab, you can add the text directly, or click the “Part Name” button to use the part name as the added text. Click the “Change Font” button can change the text font and size.

In the Setting tab, you can select the texture direction, that is, the outside surface or the inside surface, and set the depth of the texture.

The added text will be displayed in the screen with the original part, and you can hold down the mouse wheel to move the part to reach the desired label effect. Finally, click the “OK” to complete the label.

In the Controls tab, click “Translate text” to move the text to the appropriate position, and click “Center text” to move the text to the center of the part. Besides, you can use the mouse to move the part so that the text is placed in the appropriate place on the part.

The effect of adding the text "Voxeldance" on the part is shown in Figure 7.35.

Figure 7.35 Add label

7.8 Boolean Operation

7.8.1 Boolean Operation

The Boolean operation module provides three Boolean operations for the part, including merging part, extracting overlapping portions, and removing overlapping portions from the part.

To conduct the Boolean operation, firstly you should move the parts (at least two) together to overlap each other, that is, each part does not exist independently. Select all parts you want to use, click the “Boolean” command in the Boolean drop-down menu under the Home menu, or press the shortcut keys Ctrl + B, to open the Boolean operation setting dialog, as shown in Figure 7.36.

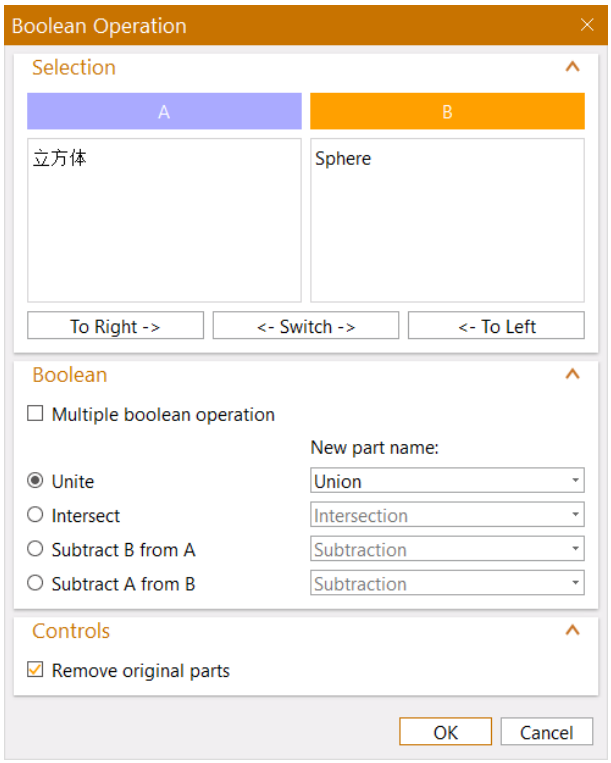


Figure 7.36 The Boolean Operation setting

There are A and B operating areas in the dialog, and the part in area A is purple, and the part in area B is orange. Users can adjust the operating area of each part by pressing the “to right” and “to left” buttons; users can also switch the parts of the two regions via the switch key, but each area must contain at least one operating part. After the adjustment, you can perform three Boolean operations for the part in the A, B regions respectively:

(1) Union: All parts of the two operating areas are merged, as shown in Figure 7.37.

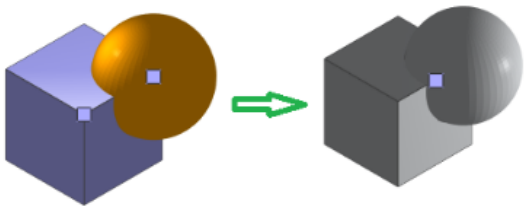


Figure 7.37 Union operation

(2) Intersect: Extracting the parts of the two operating regions where the parts overlap each other, as shown in Figure 7.38.

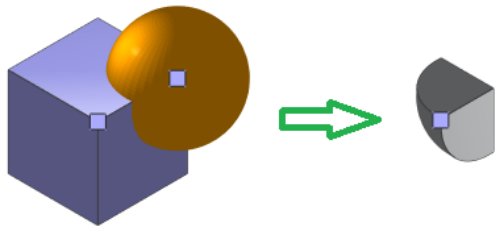


Figure 7.38 Intersect operation

(3) Subtract

a) Subtract B from A: Combine all parts in area A, but remove the part that overlaps the part in B region, as shown in Figure 7.39.

b) Subtract A from B: Combine all parts in area B, but remove the part that overlaps the part in A region.

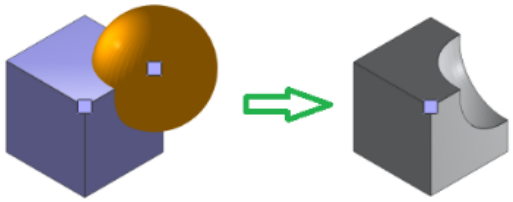


Figure 7.39 Subtract operation

If you check the "Multiple Boolean Operations" option on the Boolean operation tab, you can perform multiple Boolean operations at the same time and specify which Boolean operations to use. This option is OFF by default.

If you select the "Remove original part" option on the “Controls” tab, you cannot keep the original part.

7.8.2 Merge Parts

Select two or more parts, and use the "Merge Parts" command in the Boolean drop-down menu under the Home menu to merge them into one part. The merged part contains the entire shell of the selected part, as shown in Figure 7.40. Clicking this "Merge Parts" command will pop up a dialog asking if you want to keep the original part.

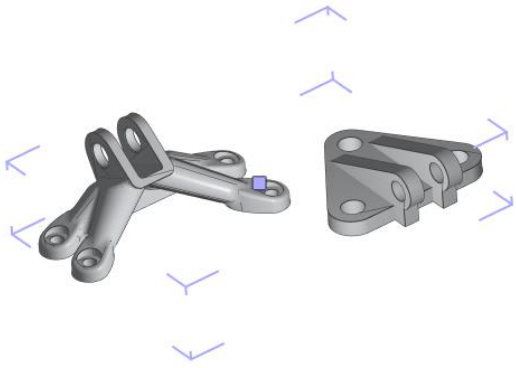


Figure 7.40 Merge parts

7.8.3 Shells to Parts

If the selected part contains more than one shell, each shell can be split into independent parts through the "Shell to Parts" command in the Boolean drop-down menu under the Home menu, as shown in Figure 7.41. Clicking this command will pop up a dialog asking if you want to remove the original part.

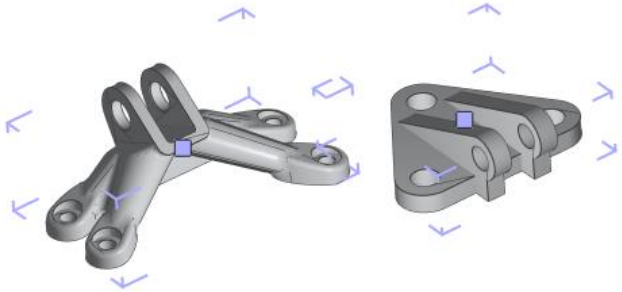


图 7.41 Shell to parts

7.9 Structure

The Structures Module enables you to swiftly make complex structures within a certain part. These structures allow you to reduce the consumption of material and weight of the part, without losing strength. The Structures wizard will guide you to through a few steps in order to hollow the part, and fill them with lightweight structures. And you can easily add holes to allow the powder to escape.

Select the part, and click the “Structure” command under the Home menu, the parameters setting dialog for creating structure will pop up in the display window, as shown in Figure 7.42.

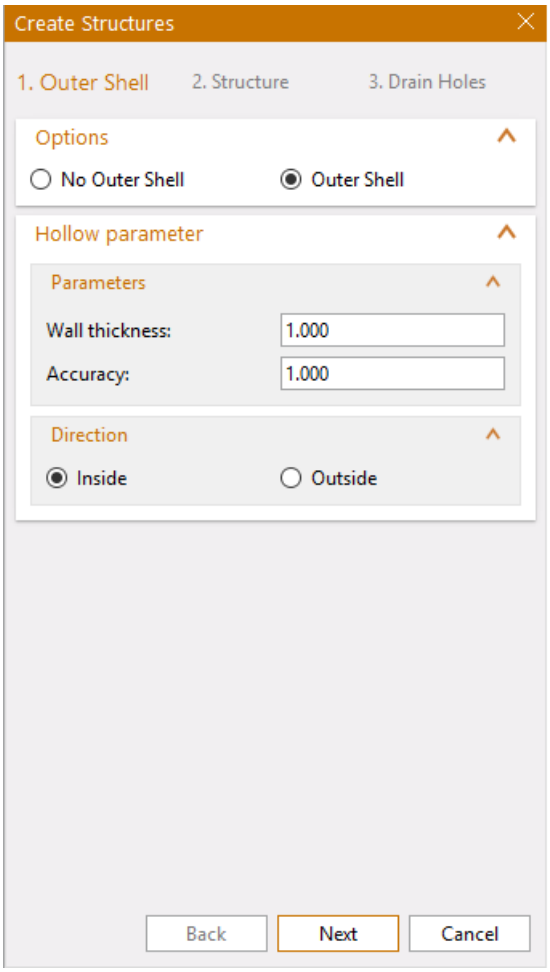


Figure 7.42 Create structure setting

The parameter setting dialog consists of three main steps: define Outer Shell、choose Structure、add Drain Holes. In the definition of the shell operation, you can choose “No Outer Shell” or “Outer Shell”. If you select “No Outer Shell”, structure will be completely replaced by lattice, no shell; if you select “Outer Shell”, the part will add the shell firstly, then add the lattice in the internal structure, and the part has a shell after structure (Part hollow refer 7.2.1).

When the “Outer Shell” operation finishes, click Next to enter the “Structure” operation, such as figure 7.43.

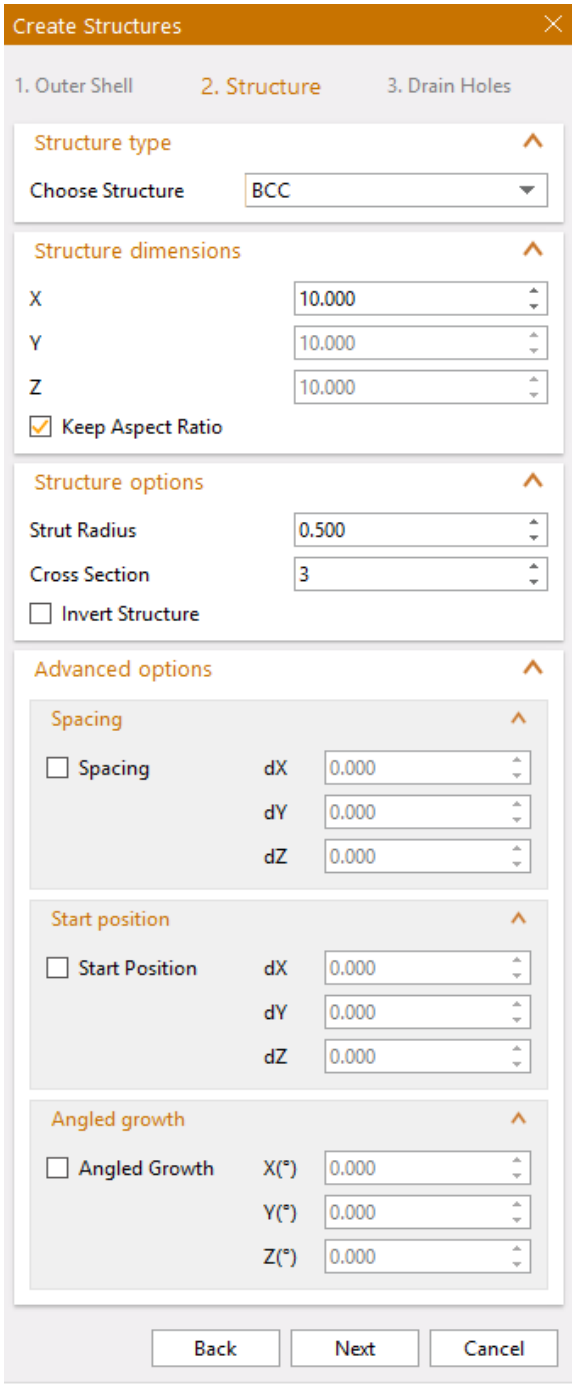


Figure 7.43 Choose structure setting

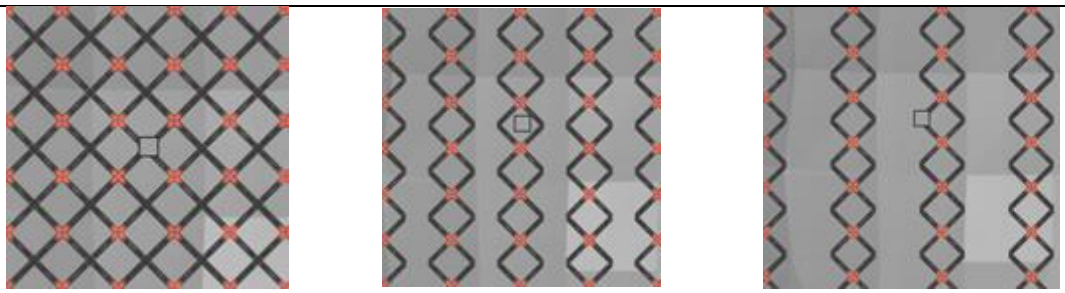
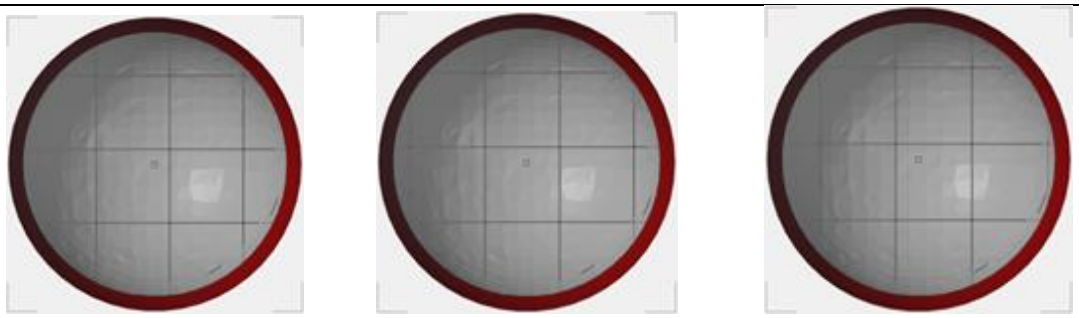
There are 9 structural types and corresponding structure preview provided at the top of the "Choose Structure" settings page. The corresponding parameters in the middle part are explained in Table 7.9.

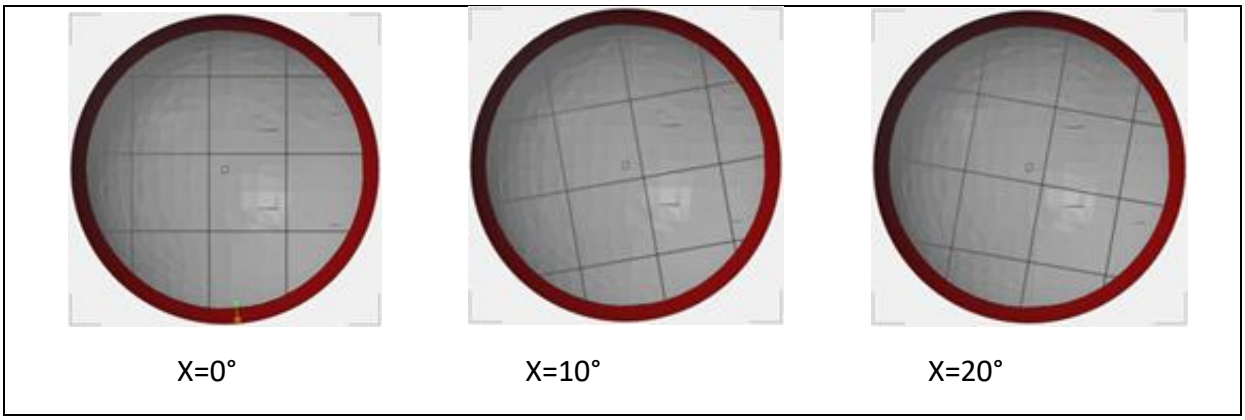
Table 7.9 Choose Structure parameter settings

Structure Dimensions	Set the length in X, Y, and Z of the unit structure
Strut Radius	The radius of the structural strut
Cross Section	The number of cross section of structural struts
Keep Aspect Ratio	If selected, the Y and Z length will be rescaled uniformly with the X length
Invert Structure	After selection, the surface of the lattice structure is inverted

The “Advanced Options” setting provides some advanced settings. The corresponding parameter settings are explained in Table 7.10.

Table 7.10 Advanced Options parameter settings

Spacing：Spacing between cell structure bounding boxes
 dx=0mm　　dx=5mm　　dx=10mm
Start Position：The start position of structure can be defined
 dx=0mm　　dx=12mm　　dx=25mm
Angled Growth：Generating lattice at a specified angle



After the “Structure” operation finishes, you can add the lattice to the part and enter the “Drain Holes” operation by clicking the Next button, as shown in Figure 7.44.

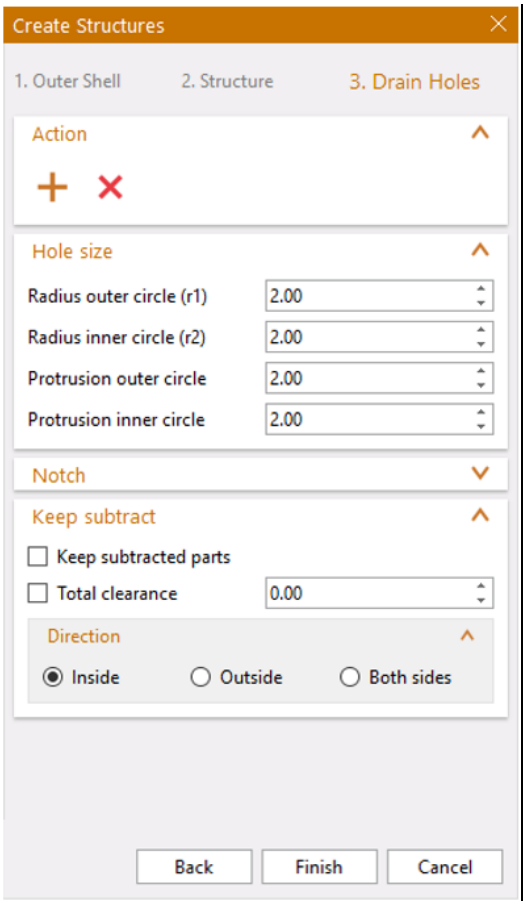


Figure 7.44 Add drain holes setting

In the Add Drain Holes operation, a hole can be added in the part body (See 7.5 for the Perforator operation).

After the perforator operation finish, click “Finish” button to complete the structure.

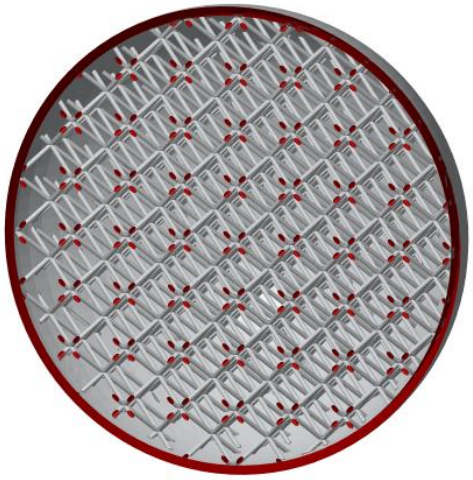


Figure 7.45 Create structure effect

7.10 Z-Compensation

In the actual printing process, the machine may cause the thickness of the first printing layer higher than other layers. That is, the bottom of the Z-axis may stretch, causing the deformation of the entire printed part. This effect can be eliminated by the Z compensation function.

Select the part, and click the “Z-Compensation” command in the Home menu, and the Z-axis compensation parameter setting dialog will pop up, as shown in Figure 7.46.

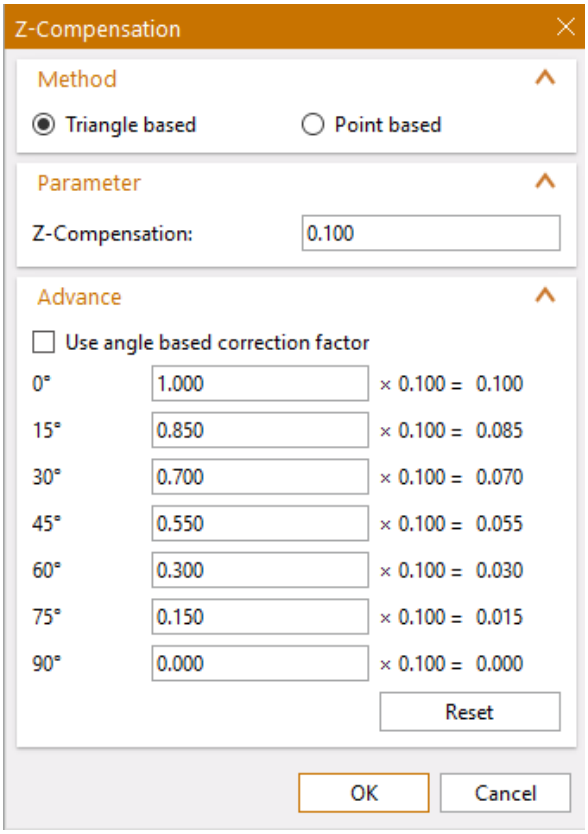


Figure 7.46 Z compensation setting

If "All Parts" is selected, Z compensation will be performed for all parts on the platform. If "Selected Parts" is selected, Z compensation will be performed only for the selected parts.

The user should set the first-layer Z-axis height in the Z-Height compensation adjustment box according to the characteristics of the printer used, and the bottom layer thickness of the part will be adjusted to that value.

Triangle based: This algorithm will detect all downward facing triangles and move them in the Z direction by a distance determined by the input Z compensation value.

Point-based: The algorithm will detect all downward facing triangles and move the points of these triangles in the Z direction by a distance determined by the input Z compensation value.

Moreover, by checking the "Use Angle based Correction factor" checkbox, you can set compensation parameters according to the different angles between the bottom of the part and the platform. If you click the reset button, the compensation parameters of each angle will

restore the default values. After all the settings, click the Confirm button to complete the operation, the effect is shown in Figure 7.47. After the Z-axis compensation is performed, the updated Z-axis coordinates will be displayed in the Box property under the Basic Info page.

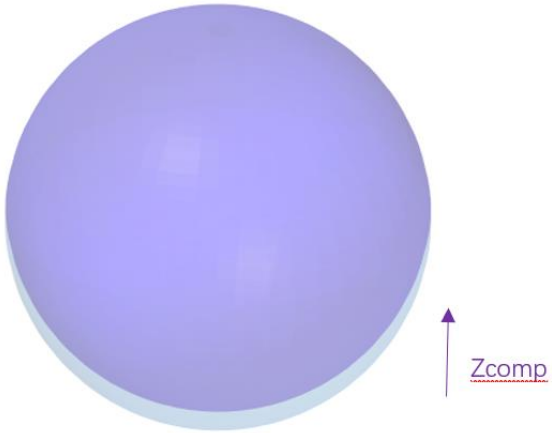


Figure 7.47 Z-Compensation

8 Part Repair

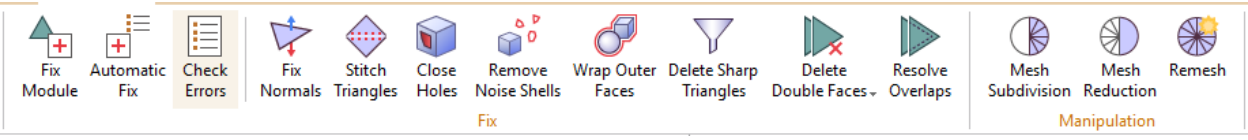


Figure 8.1 Part Repair menu

Any damaged part can't be used for printing, and you can repair it with the repair module. Select the part, and click the "Fix Module" command in the Home menu or the Modify menu, or the shortcut keys Shift + R, will enter the repair mode and the selected part will be displayed in the triangular mesh mode. Meanwhile, the Repair Part control page will open at the left part of the interface, as shown in Figure 8.2

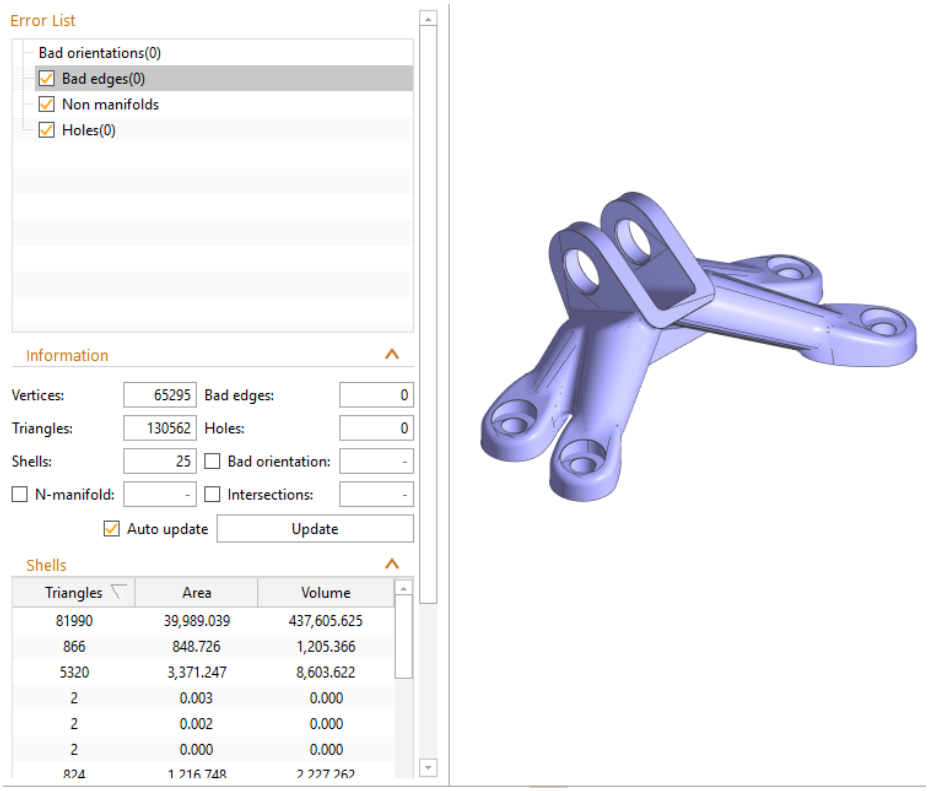


Figure 8.2 Repair mode interface

The Repair Part page is mainly divided into four functional sub-pages such as Error list, Information, Shells and Holes. Errors List area lists each type of damage and the exact location of each damage. The Information area mainly shows the part information and damage information,

including Vertices, Triangles, Shells, N-manifolds, Bad edges, Holes, Bad orientation, Intersections. After each operation, you can update the above information by clicking the “Update” button below the info area. If the “Auto Update” checkbox is checked, the part status is automatically detected and updated. Shells page displays shell information. Holes page displays hole information.

8.1 Automatic Fix

Click the "Automatic Fix" command in the Home menu or the Modify menu, or in the Fix module, you can perform automatic repair for the part.

After the repair is completed, without entering the repair module, you can perform the error check of the repaired part by modifying the "Check Errors" command under the Modify menu. Click this command, Check Errors dialog will pop up in the display window, as shown in Figure 8.3, the display content of this dialog is consistent with the content of the information displayed in the Information area under the repair module, including Vertices, Triangles, Shells, N-manifolds, Bad edges, Holes, Bad orientation, Intersections. Click the “Update” button at the bottom of the dialog to begin to check the errors on the part.

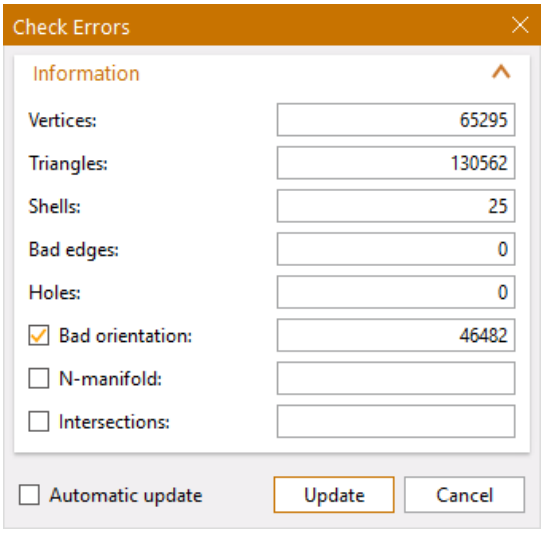


Figure 8.3 Check error

8.2 Semi-Automatic Repair

Semi-automatic repair is usually targeted for specific damage or error type. Enter the repair module, the corresponding menu is shown in Figure 8.4.

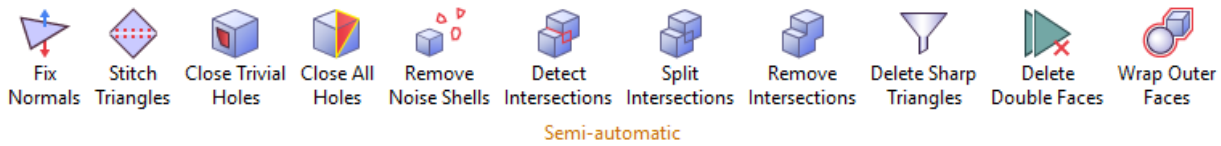


Figure 8.4 The semi-automatic repair menu

8.2.1 Fix Normals

In an STL-file, the outside of a triangle is defined. For further processing/slicing of the STL-file, it's important that the outside is defined correctly, if not, these flipped triangles must be inverted.

8.2.2 Stitch Triangles

When a gap (hole) exists between adjacent triangles, the adjacent vertices and edges can be connected with the Stitch Triangles function, thereby forming new triangles and stitching the voids.

Click the "Stitch Triangles" button, a dialog for setting Tolerance will pop up. It allows you to set the maximum spacing of the triangles to be stitched, thus preventing the part distortion caused by stitching. The stitching effect as shown in Figure 8.5.

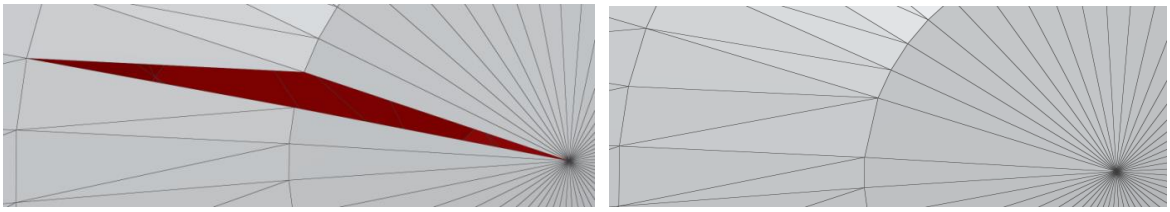


Figure 8.5 Before stitching (Left) and after stitching (right)

8.2.3 Close Holes

By clicking the “Close Trivial Holes” button, individual triangles can be inserted directly and appropriately into all the simple polygon holes in the part to get them closed. By clicking the “Close All Holes” button, all holes in the part are closed. Although this is the most straightforward way to repair holes, it may not achieve the best repair effect when dealing with complex holes associated with the part structure or direction.

In addition, detailed information about each hole is displayed under the Holes area in the left part of the interface, including the number of edges and the perimeter. After clicking on any item in the list, you can select the corresponding hole, marked with red borders, and it can be observed on the display graphics, as shown in Figure 8.6. Once selected, it can be closed after you click the Fix selected button below the list, and the corresponding item will be removed from the list.

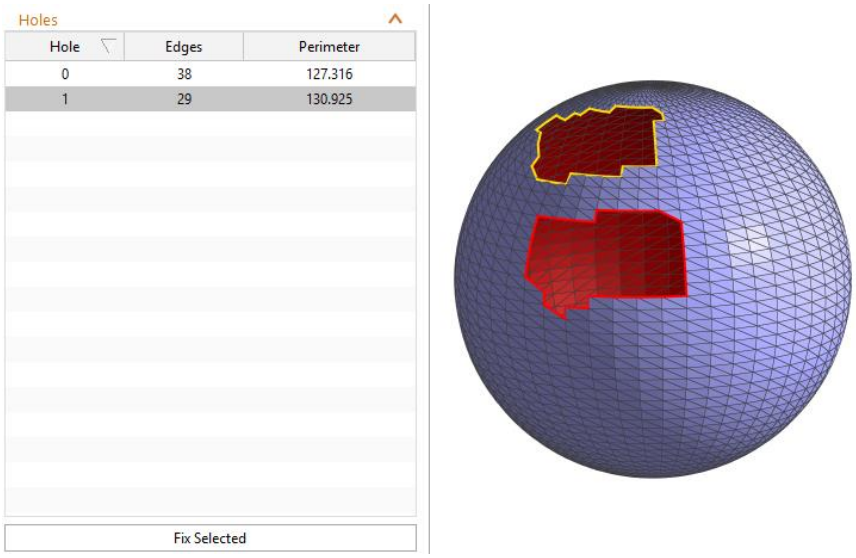


Figure 8.6 Holes page and holes selection

8.2.4 Remove Noise Shells

Some shells are just noise and make no geometrical sense. It's easier to fix the part when they are removed in an early stage of fixing. Click the “Remove Noise Shells” button, all noise shells in the part will be removed.

And the Shells area shows the details of each shell, including the number of triangles, area and volume, and. By clicking on any item in the list, you can select the corresponding shell, which is marked as yellow, and it can be observed on the display graphics, as shown in Figure 8.7. Once selected, the shell will be deleted after you click the Delete selected button below the list, and the corresponding item will be removed from the list. By clicking the Delete noise button below the list or the Delete Noise Shell button in the Action sub-page, you can delete all the noise shells directly and leave only the main shells.

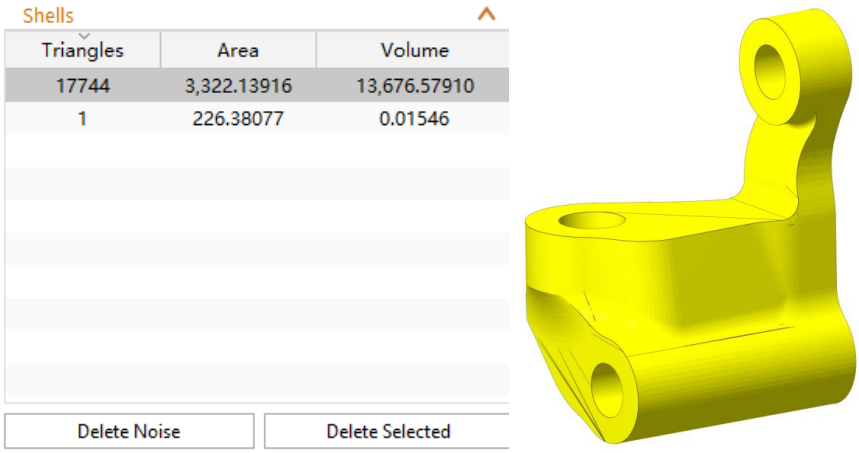


Figure 8.7 Shells page and shells selection

8.2.5 Remove Intersections

The fix intersection area provides the operations of Detect Intersection, Split Intersection, and Fix Intersection.

In the information area on the left, check the “Intersections” option, and then click “Update” button. If a non-zero number of intersections is displayed, it indicates that there is a self-intersection in the part. Click the “Detect Intersection” button, the intersection lines will be marked red on the part. Click the “Split Intersections” button, the intersection surface will be cut off / stripped along the dividing lines, producing some Non-Manifolds. After clicking the "Fix Intersections" button, all the internal shells and the overlapping surfaces will be removed and the outer surfaces will be connected to each other to form a single shell, as shown in Figure 8.8. The Fix operation should be done with no holes on the part or after all holes are repaired.

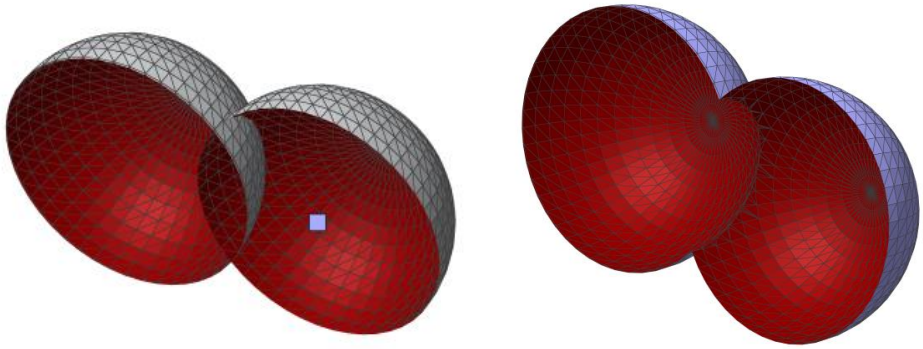


Figure 8.8 Self-intersecting before (left) and after the repair (right)

8.2.6 Delete Sharp Face

When parts have thin triangles, this function is to deletes them. Click the "Delete Sharp Triangle" button, and the setting dialog will pop up, as shown in Figure 8.9.

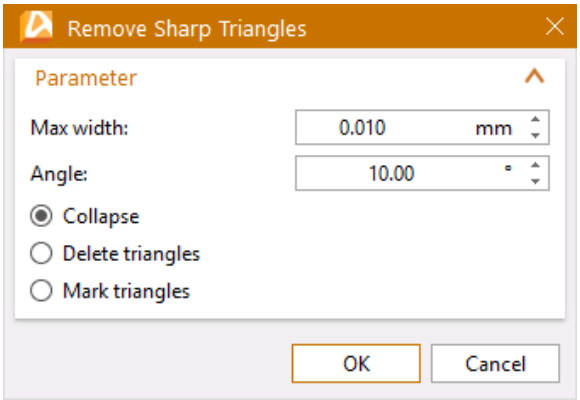


Figure 8.9 Remove sharp triangles setting

Max width: Triangles smaller than this distance will be marked or deleted, depending on your choice

Angle: The thin triangle will only be selected when the angle it makes with its neighbours is bigger than the given angle. This is easy to filter only thin triangles of folds and leave thin triangles of curves untouched.

Collapse: The thin triangle will be removed and its neighbours will be connected to each other.

Delete triangles: The thin triangles will be deleted, bad edges will appear

Mark triangles: The thin triangles will be marked.

8.2.7 Delete Double Face

Click the “Delete Double Faces” button, all the overlapping triangles or overlapping faces in the part will be removed. After this operation is executed, the updated number of triangles will be displayed in the Information field.

8.2.8 Wrap Outer Faces

By clicking the “Wrap Outer Faces” button you can remove all meshes or shells inside the part and leave only the outer surface of the part. This function is typically used to remove the inner shells of the part.

8.2.9 Resolve Overlaps

Under the Modify menu, click the “Resolver Overlaps” button. A dialog will pop up to set the tolerance value, that is, the maximum distance to repair the overlap. After setting, it can be executed to repair the triangles that touch, overlap, or close to each other in the part as single plane or shell.

8.2.10 Create Bridge

Under the Modify menu, click the "Create Bridge" button, and then select the two contour lines on the part where in between a bridge needs to be created. A preview is shown of the bridge before creation. This function is used to help fixing complex holes.

8.3 Manual Repair

8.3.1 Add a New Triangle

When a small number of missing triangles or holes exist, click the "Add a New Triangle" button to enter the vertex selection state. In this mode, when you select any vertex, a red circle and a number will appear on the vertex to indicate that it is selected. After three vertices are selected, a new triangle will be added, as shown in Figure 8.10.

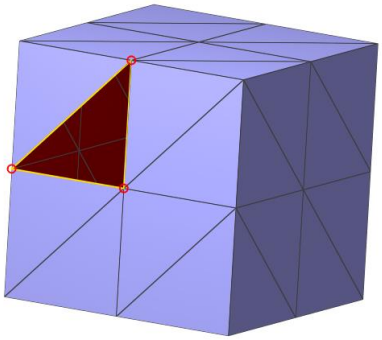


Figure 8.10 Add a triangle manually

8.3.2 Delete Marked

When a mark currently exists, click the "Delete Mark" button or select the "Delete Mark" triangle command from the right-click menu of any triangle, you can delete the marked triangle

8.3.3 Flip Marked

When a mark currently exists, click the "Flip Marker Triangle" button, or select the "Flip Marker Triangle" command from the right-click menu of any triangle, the marked triangle will be flipped.

8.4 Manipulation

8.4.1 Mesh Subdivision

When the file is too small or the accuracy of the part is too low, you can increase the number of triangles by subdividing the triangles to improve the accuracy of graphics. After selecting the part, click the “Mesh Subdivision” command under the Modify menu, and the parameter setting dialog of the subdivision operation will pop up, as shown in Figure 8.11.

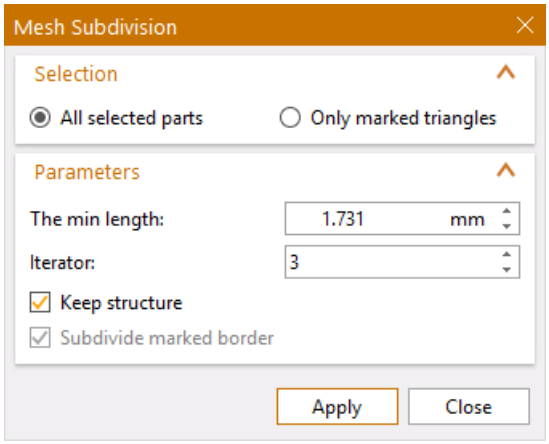


Figure 8.11 Mesh subdivision setting

If you select “All selected parts”, the corresponding mesh operation will be performed on the whole part. If you select “Only marked triangles”, you need to select an area on the part where the corresponding operation will be performed.

The user can set the minimum length of the triangle edges through the min length spin-box and thus limit the number of triangles by limiting the size of the triangles. he iteration adjustment box is used to set the number of iterations, that is, the number of times to perform the subdivision operation repeatedly. When the iteration value is larger, the number of subdivided triangles increases exponentially. By checking the “Keep structure” checkbox below, you can also perform the subdivision process without changing the original structure of the part. In the case of " Only marked triangles", "Subdivide marked border" can be edit. If checked, it means that the boundary of the selected triangle will also be subdivided.

After the part is subdivided, the updated number of triangles will be displayed in the Part Details field on the left of interface.

8.4.2 Mesh Reduction

When the file is too large or the part is too complicated, you can merge triangles with mesh reduction function to reduce file size or speed up calculations. After selecting the part, click the "Mesh Reduction " command under the Modify menu, or the shortcut keys Shift + T, a parameter setting dialog for the operation will pop up, as shown in Figure 8.12.

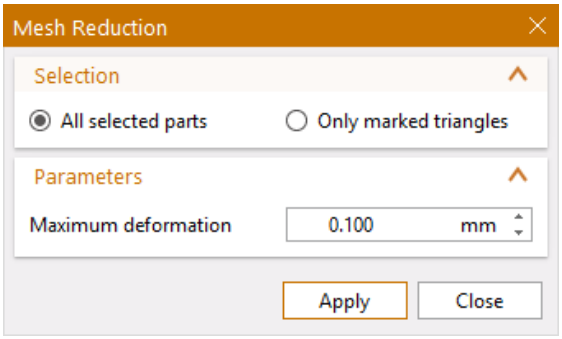


Figure 8.12 Mesh reduction setting

If you select “All selected parts”, the corresponding mesh operation will be performed on the whole part. If you select “Only marked triangles”, you need to select an area on the part where the corresponding operation will be performed.

Maximum deformation: If 2 triangles are replaced by one triangle, there may be a little deviation in position. The tolerance indicates the maximum deviation allowed between the original surface and the new one.

After the part is reduced, the updated number of triangles will be displayed in the Part Details field on the left of interface.

8.4.3 Remesh

When the part has a lot of irregular meshes, you can refactor smoother triangles through the Remesh function. After selecting the part, click the “Remsh” command in the Modify menu, and the Remesh setting dialog will pop up, as shown in Figure 8.13. In the setting box, the tolerance represents the maximum value of the mesh surface deviation distance before and after remeshing, and the lower this value is, the more similar to the original part and the finer the triangle mesh will be, and the larger this value is, the surface is simpler. The minimal length and the maximal length represent the shortest and longest sides of the reconstructed triangle, and the dihedral angle means that the triangle whose angle is greater than this value in the original part will retain its feature.

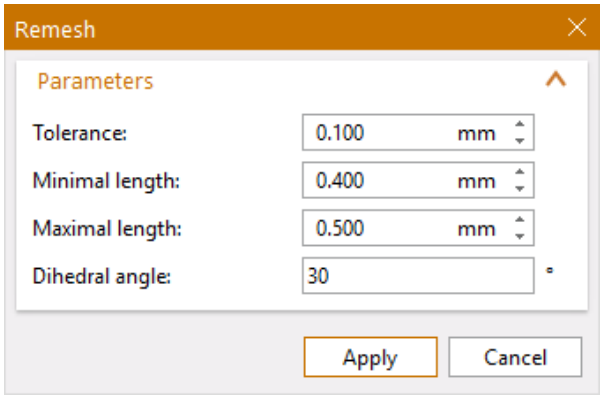


Figure 8.13 Remesh setting

8.5 Mark Triangles

The function of marking triangles exists in the toolbar by default.

To repair the damaged STL file, the user can mark the triangles of the selected part. You need to select the part before marking the triangle. The selected triangle is marked (default) yellow on the part.

1) Mark single triangle: Triangles can be marked one by one. Marked triangles can be unmarked by indicating them with the Mark single triangle cursor again.

2) Mark Plane: By clicking one triangle, a whole plane can be selected or unselected.

3) Mark Surface: By clicking one triangle, a surface can be selected or unselected.

4) Mark Window: Click this icon, you can mark all triangles within a rectangular area.

5) Mark Shell: A single shell is marked by clicking on any triangle in the shell. Click the triangle again to unmark it.

6) Mark Brush: Click this icon, you can draw a path on the part to mark all the passed triangles.

7) Mark Freedom: Click this icon, you can draw a curve on the part to mark all the triangles inside.

8)  Mark Polygon: Click this icon, you can draw a polygon on the part to mark all the triangles inside.

9)  Mark Polygon and Remesh: Click this icon, you can draw a polygon on the part to mark the triangles inside, and at the same time, the triangular face on the edge of the polygon is remeshed.

10)  Mark Circle and Remesh: Click this icon, you can draw a circle on the part mark the triangles inside, and at the same time, the triangular face on the edge of the circle is remeshed.

11)  Mark Ellipse and Remesh: Click this icon, you can draw an ellipse on the part to mark the triangles inside, and at the same time, the triangular face on the edge of the ellipse is remeshed.

12)  Unmark All： All triangles will be unmarked.

13)  Expand Marked: The group of selected triangles will become bigger. All unmarked triangles lying next to a marked triangle will be marked.

14)  Shrink Marked: The group of selected triangles will become smaller. All triangles lying at the border of the group will be unmarked.

15)  Invert Marked: Click this icon, you can invert the marked triangles, all unmarked triangles turn yellow (or the defined marked triangle color) and vice versa.

16)  Delete Marked: The marked triangles are deleted.

17)  Separate Marked: The marked triangles are separated from the original part and stored in a separate part.

18)  Copy Marked: The marked triangles are copied; and a new part is created in the part list.

19)  Flip Marked: Flip the normal of the marked triangle.

20)  Invert Part: Flip the normal of all triangles.

21)  Hide Marked: The marked triangles are hidden.

22)  Invert Triangles Visibility: Make invisible triangles visible and vice versa.

23)  Unhide All: Make all triangles visible.

9 Support Operation

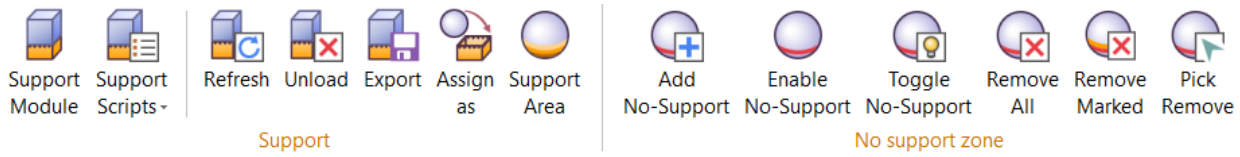


Figure 9.1 Support Generation menu

The part needs to be added support structure before printing, there are five main types of support: point support, bar support, line support, volume support and smart support, as shown in Figure 9.2.

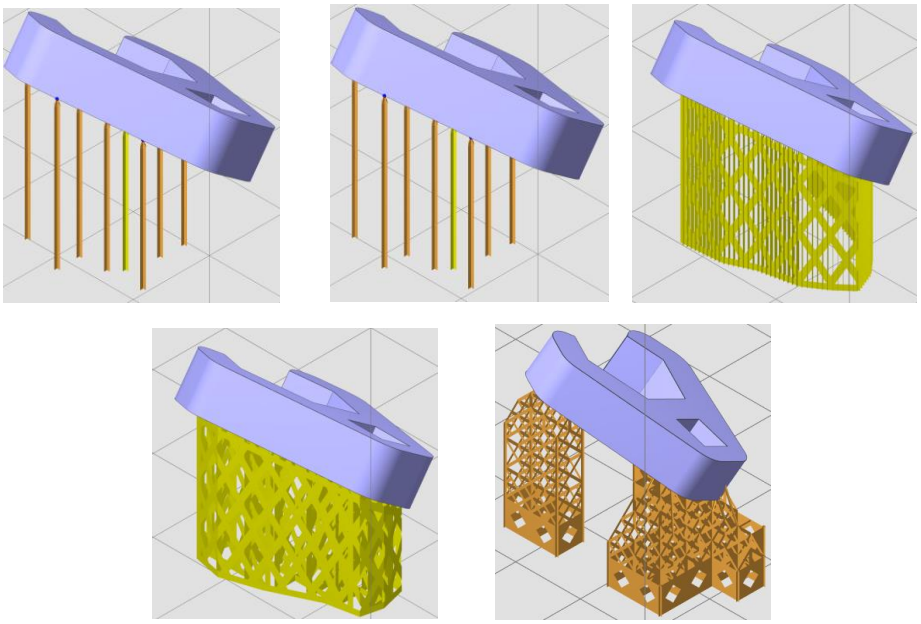


Figure 9.2 From top to bottom, from left to right: point support, bar support, line support, volume support and smart support

9.1 Support Module

After selecting the part that is undamaged or has been repaired, then clicking the “Support Module” command in the Support Generation menu, or pressing the shortcut keys Shift + I, users can enter the support operation module.

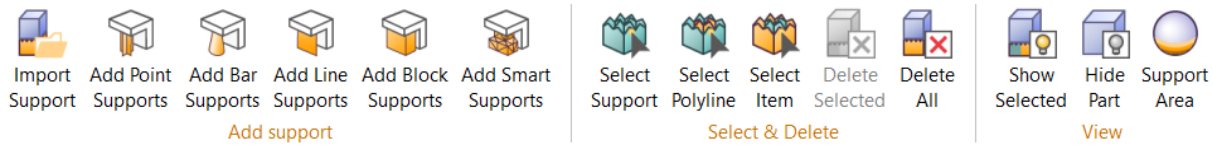
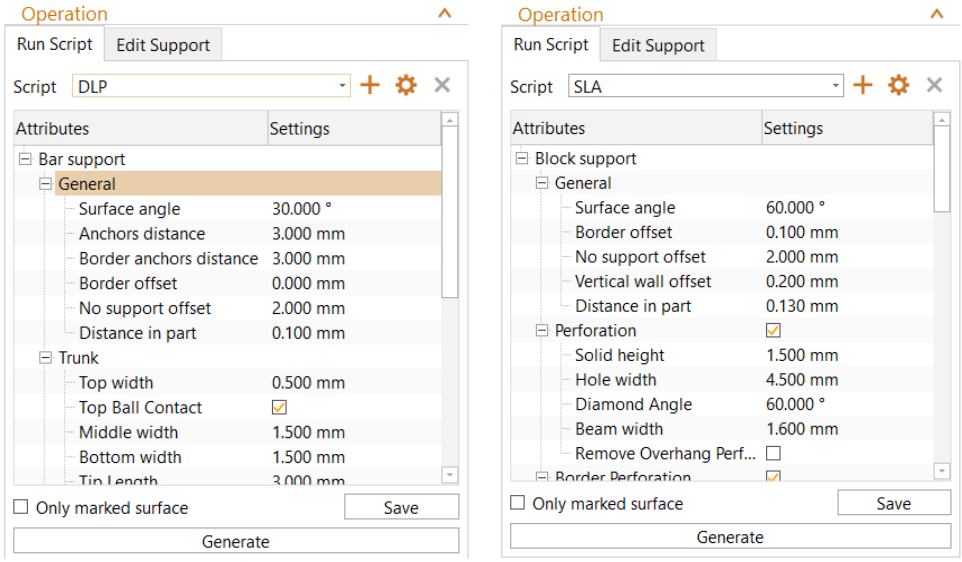


Figure 9.3 Support Module menu

9.1.1 Automatic Support

Many parts do not only contain one support area, and may contain many unconnected support areas. Automatically supporting the entire part or each support area can be completed in one step.

In the current support module, there are three default predefined scripts for creating automatic support: DLP (for Bar support), SLM (for Line support and Volume support) and smart support, and they perform support operations in a predetermined order. After entering the support module, the following operation page will appear in the left lower part of the interface, as shown in Figure 9.4.



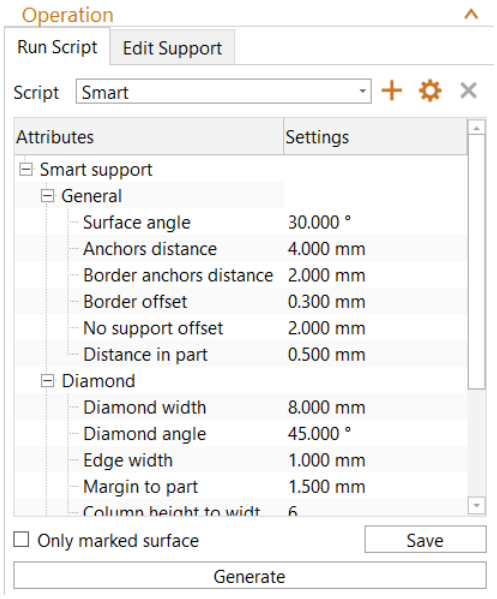


Figure 9.4 Default support script

In this operation page, you can customize, edit and save new support scripts. The above drop-down menu shows the used script, and you can create a new script by clicking the "+" icon button. If you want to delete the added support script, you can click " ✕ " icon button to delete, and if you want to set the default script, import/export script, you can click " ⚙ " icon button to set. The default DLP, SLA and Smart cannot be edited and deleted.

The following attribute area shows the script order in which the support is executed. Some of the functions in the script have a sign ⊟ on the left, and you can open, view and edit the detailed parameters of these functions by clicking it. When the mouse is hovered over each parameter name, a graphical explanation of the parameters will pop up, as shown in Figure 9.5.

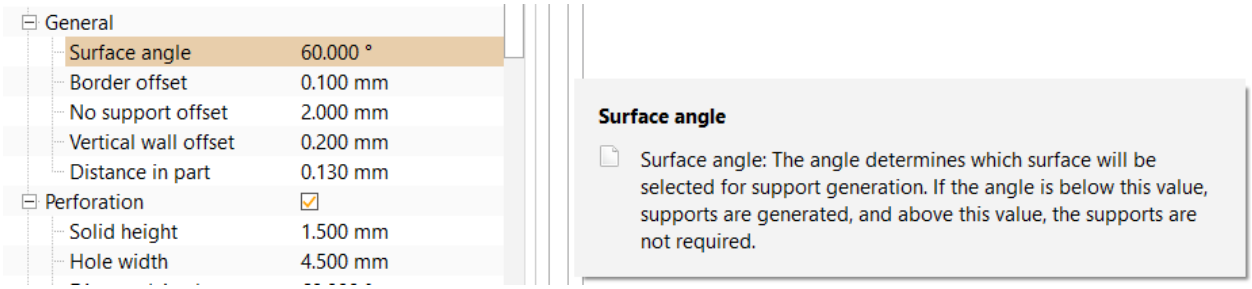


Figure 9.5 The graphical explanation of the Surface angle parameter

Currently, you can add two three script types, including bar support, block support and smart support, as shown in Figure 9.6. Select a support type, set the name and click "Confirm" to add its function parameters to the current script. Click "Save" button to save the added new script settings to this program.

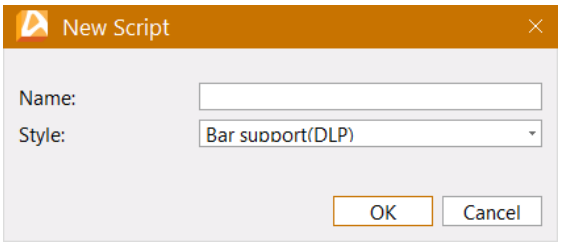


Figure 9.6 Add new support script

Select a support script, set the required parameter values, and click "Generate support" to complete the automatic addition of the support structure, and at the same time, the support region boundary contour line will be formed at the support boundary. If the “Only marked surface” selection box is checked, support is only generated in the marked area, as shown in Figure 9.6.

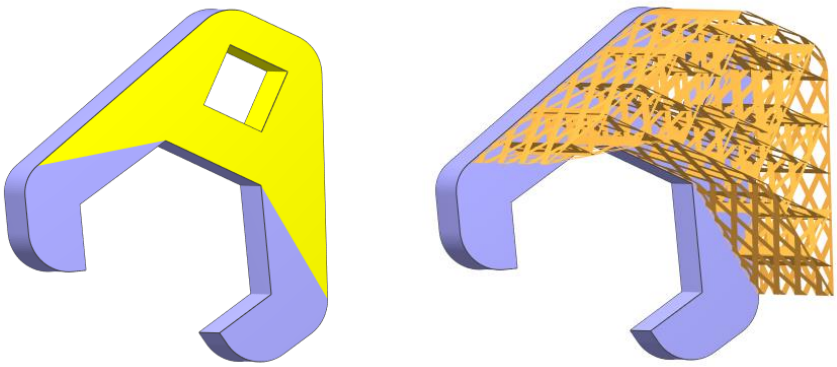


Figure 9.6 Generating support in marked area

After adding the appropriate support, click the “Exit Support” in the menu, a query operation dialog will appear. If you click "Apply Operation” button, the support structure will be added to the part and the support module is exited; if you click "Discard changes" button, the part will be restored to the one before entering this module, and the support module is exited.

A detailed description of the bar support and block support is shown in Tables 9.1, 9.2 and 9.3.

Table 9.1 Bar support settings

Attributes	Meaning
General	
Surface angle	The angle between the support surface and the horizontal plane The larger the critical angle, the greater the area that needs to be supported
Anchors distance	Define the distance between internal anchor points
Border anchors distance	Define the distance between border anchor points
Border offset	The distance from support to the support area boundary
No support offset	Overhangs less than this offset value will not generate support
Distance in part	The length of the support bar inserting into the part
Trunk	
Top width	The width at the top of the support bar
Top Ball Contact	Whether the top of the support bar is spherical
Middle Width	The width at the middle of the support bar
Bottom width	The width at the bottom of the support bar
Tip length	Distance between part surface and contact point widened to middle width
Tip type	Direction of support contacting parts
Bar edge number	The number of the support bar
Cross type	The cross type between two support bar
Base	Whether to generate a support base
Base offset	Relation to the base type, if the base is convex hull and the minimum area, it is the distance between the support point and the base boundary; if the base is rectangular, it is the distance between the base border and the critical bounding box
Base height	The thickness of the base

Base type	The support base type has three types: rectangle, convex hull and minimum area. Rectangle: The projection of the part's bounding box in the XY plane as the base; minimum area: The minimum base that connects each support base; convex hull: The convex shape composed of each support bar as the base
Circle Base	Support base with the minimum area
Circle radius	The radius of the support base with the minimum area

Table 9.2 Block support settings

Attributes	**Meaning**
General	
Surface angle	The angle between the support surface and the horizontal plane The larger the critical angle, the greater the area that needs to be supported
Border offset	The distance between the support and the support area boundary
No support offset	Overhangs less than this offset will not be supported
Vertical wall offset	The distance support and the vertical plane
Distance in part	The length of the support bar inserting into the part
Perforation	Whether to generate a diamond-shaped structure on the internal support
Solid Height	Boundary thickness in the diamond area
Hole width	The right and left heights of the diamond structure
Diamond angle	The angle of diamond edge and horizontal diagonal
Beam witch	Diamond outer thickness
Remove Overhang Perforation	Whether to remove holes with overhanging edges (complete holes are diamond-shaped. If the sides of incomplete holes have horizontal edges, then this edge is an overhanging edge, and this hole is an overhanging edge)
Border Perforation	Whether to generate a diamond-shaped structure on the border support

Solid Height	Boundary thickness in diamond area
Hole width	The right and left heights of the diamond structure
Diamond angle	The angle of diamond edge and horizontal diagonal
Beam witch	Diamond outer thickness
Remove Overhang Perforation	Whether to remove holes with overhanging edges (complete holes are diamond-shaped. If the sides of incomplete holes have horizontal edges, then this edge is an overhanging edge, and this hole is an overhanging edge)
Teeth (Upper)	
Top width	The top width of teeth in the upper support
Base width	The base width of teeth in the upper support
Height	The height of the teeth in the top support
Base interval	The spacing between the teeth in the upper support
Teeth (Lower)	
Same as upper teeth	Using the same parameters value for upper and lower teeth
Top width	The top width of teeth in the lower support
Base width	The base width of teeth in the lower support
Height	The height of the teeth in the lower support
Base interval	The spacing between the teeth in the lower support
Is filling	
Hatching	Whether the filling line is added to the support
X hatching	Filling line space in the X axis direction
Y hatching	Filling line space in the Y axis direction
Rotate angle	The angle of X-axis fill line and horizontal
Fragmentation	Whether to fragment on the support area
X interval	The width of the x-axis interval after adding interval support
Y interval	The width of the y-axis interval after adding interval support

Separation Width	The width between intervals
Divide support block	Whether to generate support blocks

Table 9.3 Smart support settings

Attributes	Meaning
General	
Surface angle	The angle between the support surface and the horizontal plane The larger the critical angle, the greater the area that needs to be supported
Anchors distance	Define the distance between internal anchor points
Border anchors distance	Define the distance between border anchor points
Border offset	The distance from support to the support area boundary
No support offset	Overhangs less than this offset value will not generate support
Distance in part	The length of the support bar inserting into the part
Diamond	
Diamond width	The right and left heights of the diamond structure
Diamond angle	The angle of diamond edge and horizontal diagonal
Edge width	Diamond outer thickness
Margin to part	Distance from support structure to part edge
Column height to width ratio	Represents the number of diamonds in a column before expanding the grid
Teeth	
Top width	The width of the contact point between the support and the part
Base width	The width of the support leading to the contact point of the part

Height	Distance between part surface and contact point widened to connection width

9.1.2 Manual Support

With clicking the "Edit Point Supports" icon  , the corresponding point support attributes will appear in the support operation page in the left side. You can create an anchor point by clicking any point on the lower surface of the part, its size can be adjusted by the anchor radius scroll bar in the right setting. After adding any anchor point, you can delete the anchor point by clicking the left mouse button at the anchor point; you can move the anchor point by selecting anchor point and dragging the left mouse button to change the anchor point position. After creating an anchor point, you can add a point support structure at the anchor point. And the property of “Rib Num” is to define the number of ribs that make up the Point support.

With clicking the "Edit Bar Supports" icon  , the corresponding bar support attributes will appear in the support operation page in the left side. You can create an anchor point by clicking any point on the lower surface of the part, its size can be adjusted by the anchor radius scroll bar in the right setting. After adding any anchor point, you can delete the anchor point by clicking the left mouse button at the anchor point; you can move the anchor point by selecting anchor point and dragging the left mouse button to change the anchor point position. When you create at least one anchor point, then click right-click the mouse, a bar support structure at the created anchor point can be added.

With clicking the "Edit Line Supports" icon  , the corresponding line support attributes will appear in the support operation page in the left side. You can create an anchor point by clicking any point on the lower surface of the part, its size can be adjusted by the anchor radius scroll bar in the right setting. After adding any anchor point, you can delete the anchor point by clicking the left mouse button at the anchor point; you can move the anchor point by selecting the anchor point and dragging the left mouse button to change the anchor point position. When you create at least two anchor points, then right-click the mouse, a line support structure at the created anchor point can be added. In this support structure, if you select the anchor point and drag the

left mouse button to any position, the line support structure will be updated in real-time. And if you hold Ctrl button when selecting the anchor point and drag the left mouse button, the real-time update of the line support structure is not executed, until you release the Ctrl button, the line support structure will be updated. This operation can adjust the line support structure of multiple anchor points faster.

With clicking the "Edit Volume Supports" icon  , the corresponding volume support attributes will appear in the support operation page in the left side. You can create an anchor point by clicking any point on the lower surface of the part, its size be adjusted by the anchor radius scroll bar in the right setting. After adding any anchor point, you can delete the anchor point by clicking the left mouse button at the anchor point; you can move the anchor point by selecting the anchor point and dragging the left mouse button to change the anchor point position. When you create at least three anchor points, then right-click the mouse, a volume support structure at the created anchor point can be added. In this support structure, if you select the anchor point and drag the left mouse button to any position, the volume support structure will be updated in real-time. And if you hold Ctrl button when selecting the anchor point and drag the left mouse button, the real-time update of the volume support structure is not executed, until you release the Ctrl button, the volume support structure will be updated. This operation can adjust the volume support structure of multiple anchor points faster.

With clicking the "Edit Smart Supports" icon  , the corresponding bar support attributes will appear in the support operation page in the left side. You can create an anchor point by clicking any point on the lower surface of the part, its size can be adjusted by the anchor radius scroll bar in the right setting. After adding any anchor point, you can delete the anchor point by clicking the left mouse button at the anchor point; you can move the anchor point by selecting anchor point and dragging the left mouse button to change the anchor point position. When you create at least one anchor point, then click right-click the mouse, a smart support structure at the created anchor point can be added.

After clicking the "Select Support" icon  , you can select the support in the area by clicking any point in the support region boundary contour line. After clicking the "Select Polyline" icon  , you can select a single support polyline, and clicking it again will deselect it. After you click the “Select Item” icon  , you can select a single support item, and clicking it again will deselect it. And with clicking the "Delete selected supports" icon  , you can delete the selected support, and you can also remove the selected supports and all supports in the right-click menu.

After selecting a contiguous support, and clicking the "Show Select" icon  , the unselected support of the part is hidden, and only the selected support area is displayed. Click the "Hide Parts" icon  , the parts are hidden and only the supports are showed. Click the “Support Surface” icon  , and a preview of the support area will be displayed according to the settings in the dialog.

In the above operation, after clicking an icon in the edit support, the mouse will carry the corresponding icon. During the execution of this function, the mouse will always carry the icon until it switches to other functional operation icons.

9.1.3 Edit Support

After entering the support module, the support list tab page is automatically added in the Part View. The added tree support, point support, line support, block support and smart support are displayed in the Support List, as shown in Figure 9.8. Double-click on a support in the Support List, the support will be selected to enter editable state.

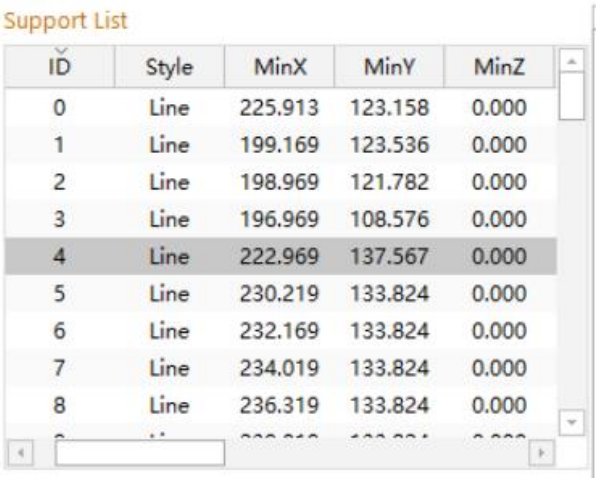


Figure 9.8 Support List

In the Edit tab, you can edit the selected support with common settings and advanced settings.

Under the common setting tab, you can change the support type of the selected support. After selecting a support, the support type automatically jumps to the support type of it, as shown in Figure 9.9. Under the support type, you can change the relevant parameters in the attribute settings, click the "Regenerate" button to regenerate the support with the support type; you can also select another support type in Support Type list, and set the parameter, then click the "Regenerate" button to implement the type conversion of the selected support, “None” indicates that there is no support in support area.

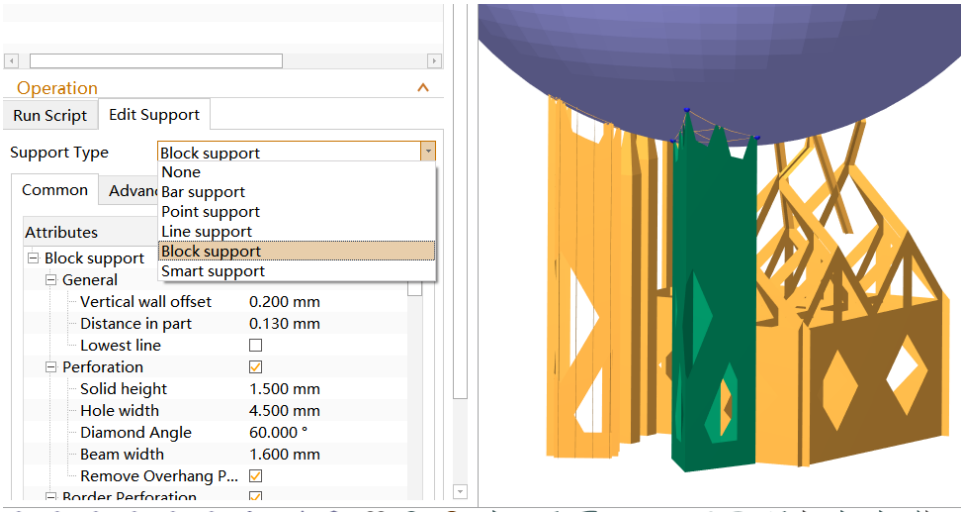


Figure 9.9 Matching the support type with the selected support

Under the Advanced settings tab, you can set the angle support and scale support for the selected support.

1) Angle Support Settings

For point support, line support and block support structures, if the bottom of the support is on the top surface of the part, to protect the part surface from being damaged, the support position can be changed by adjusting the support angle so that the support can fall on the platform as much as possible.

After adding the corresponding support, clicking the "Select Support" in the menu, and then select a section of the connected support. In the advanced settings in the Edit tab, check the “Angle support” selection box and click the “Interactive” button to add the direction manipulator in the support, as shown in Figure 9.10. In this mode, you can drag the support in the corresponding direction by moving the mouse to any axis (X, Y or Z) or any coordinate plane (X-Y, X-Z or Y-Z). If you check on “Auto generate”, the support will be automatically regenerated after dragging; if not, you are request to click “Regenerate” button to regenerate the support. If you check on “Show current support” the current support is showed during the dragging; if not, the current support is hide, and only the moving shade is showed.

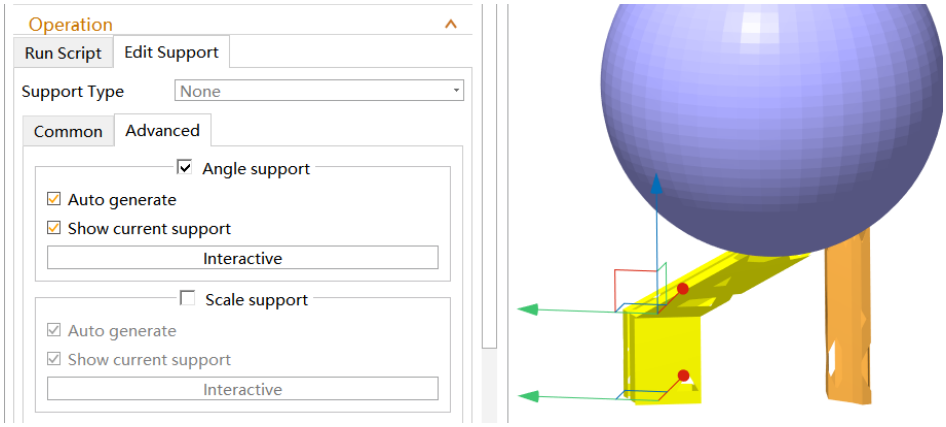


Figure 9.10 Move support

2) Scale Support Settings

For point support, line support and block support structures, the support is stronger only if the bottom of the support is wider than the top. In order to get a more solid support structure, the bottom support can be made wider by scaling the bottom support to change the support size.

After adding the corresponding support, you can click the "Mark Support" in the menu, and then select a section of the connected support. In the advanced settings in the Edit tab, check the “Scale support” selection box and click the “Interactive” button to add the support scale mode of the control direction in the support, as shown in Figure 9.11. In the scale mode of the control direction on the support, you can scale the support in the corresponding direction by moving the mouse to any axis (X, Y) or X-Y coordinate plane. If you check on “Auto generate”, the support will be automatically regenerated after scaling; if not, you are request to click “Regenerate” button to regenerate the support. If you check on “Show current support”, the current support is showed during the scaling; if not, the current support is hide, and only the scaling shade is showed.

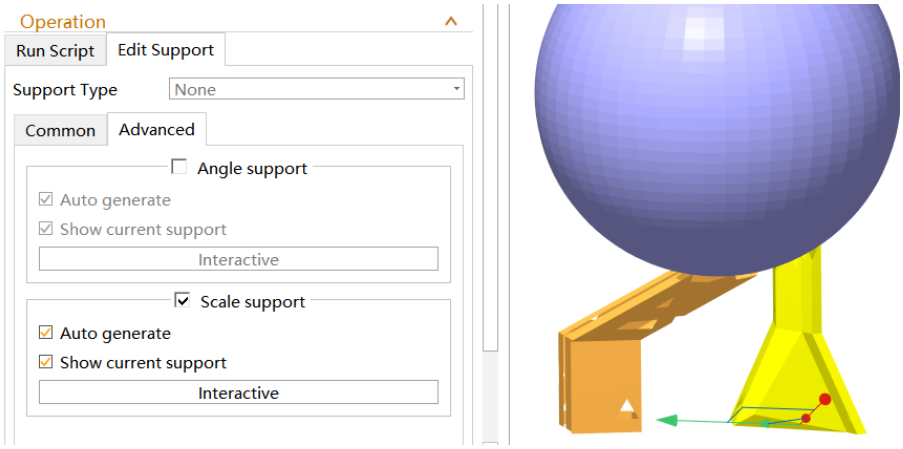


Figure 9.11 Scale support

In addition, in the Edit Support tab, 2D editing dialog can be opened by clicking “2D Edit” button, and support structure can be changed by editing. In 2D editing, you can do the operation of adding point support, line support, enhance support, and rectangle deletion and polygon deletion, flip view, zoom to select, zoom to selected part, as shown in Figure 9.12. Supports added in 2D Edit are also displayed in the Support List.

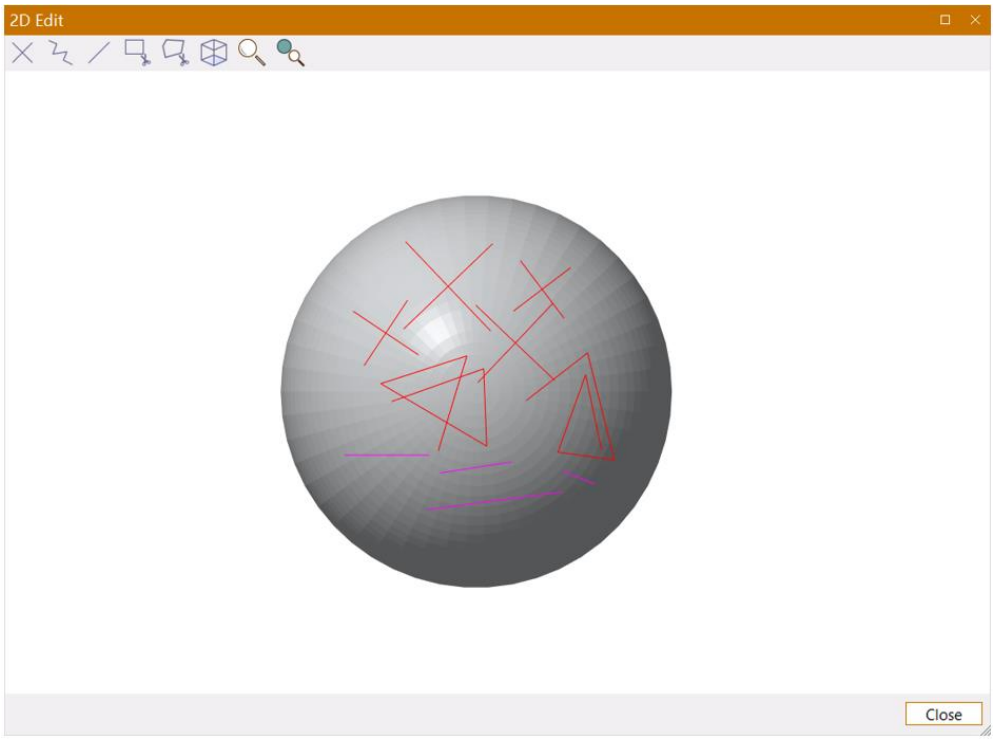


Figure 9.12 2D Edit dialog

About the detailed parameters in the 2D edit dialog, as shown in Table 9.4.

Table 9.4 2D Edit Parameters

	Draw a point support
	Draws one cross line support
	Draws a reinforcement line
	Click and drag on the 2D image. All selected supports in Support List and inside the window selection will be removed
	Draw points on the 2D image. Click the right mouse button to exit the command. All selected supports in Support List and inside the polygon selection will be removed
	Switch top and bottom view
	Zoom in the selected area

	Zoom to selected part

9.1.4 Import Supports

Under the support module, after clicking the “Import Support” command  , the file selection path box will pop up, and you can select the appropriate STL file as a support to import to the support module.

9.2 Quick support script

After setting the default support script, and selecting the part, you can directly add support to the selected part by clicking the “Support Script” command in the Support Generation menu, or pressing the shortcut keys Ctrl + Shift + I, without entering the support module, or selecting the corresponding run script in the drop-down menu of the Support Script command.

9.3 Refresh Attached Support

When moving the part with support, a prompt dialog will pop up, as shown in Figure 9.13. After the part is moving to the final position, then click the “Refresh” command in the Support Generation menu, the support will be adaptively updated with the height of the part, as shown in Figure 9.14.

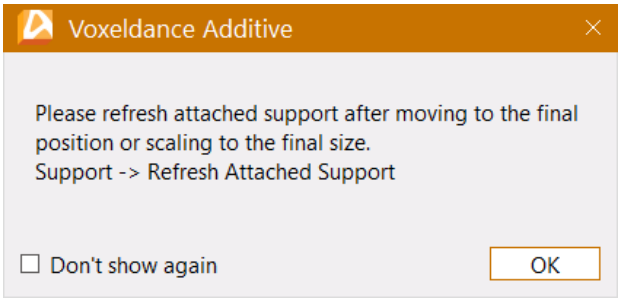


Figure 9.13 Prompt dialog

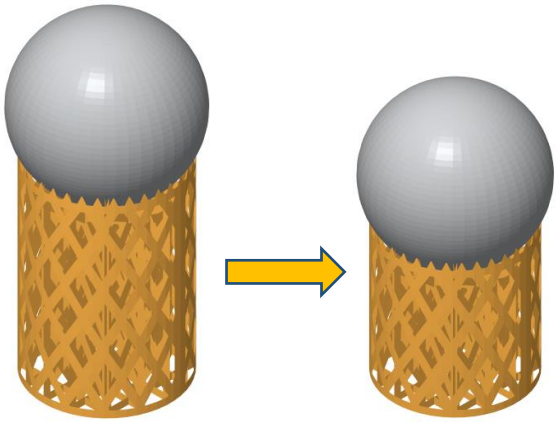


Figure 9.14 Move the part with support

When scaling the part with support, a prompt dialog will pop up, as shown in Figure 9.13. After the part is moving to the final size, then click the “Refresh” command in the Support Generation menu, the support will be adaptively updated with the size of the part, as shown in Figure 9.15

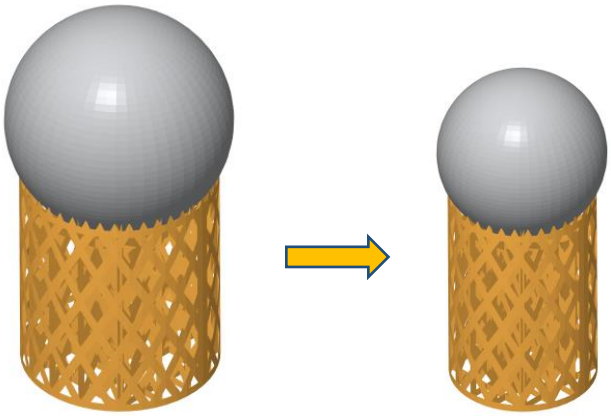


Figure 9.15 Scale the part with support

After the part is added with support, click the part placement icon  in the toolbar to enter the part movement mode of controlling direction, then move the mouse to the yellow circle in the z-axis direction and rotate, the part with support can be rotated, as shown in Figure 9.16.

Figure 9.16 The rotation around Z direction of the part with support

9.4 Unload Supports

After selecting one or more parts with supports, then clicking the “Unload” command in the Support Generation menu, you can remove the support.

9.5 Export Supports

After selecting one or more parts with supports, then clicking the “Export” command in the Support Generation menu, the Export Support operation settings dialog box will pop up, as shown in Figure 9.17. Click the icon “ ··· ” to set the file save path, and select whether to export it as an STL file including the part.

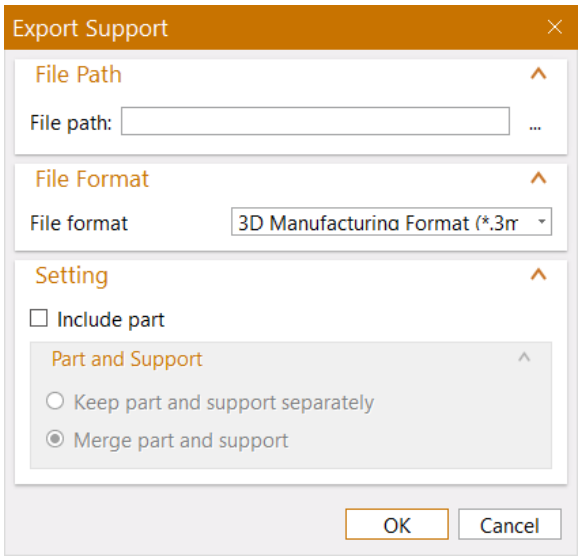


Figure 9.17 Export support operation settings

9.6 Assign Part as Support

This operation can add any selected part as a support to another part. After selecting one or more parts that need to be designated as supports, click the “Assign as” command in the Support Generation menu, then click the part on the platform that needs to be supported, the part that has been designed as support is added to the part, and the support part is displayed as yellow, as shown in Figure 9.18.

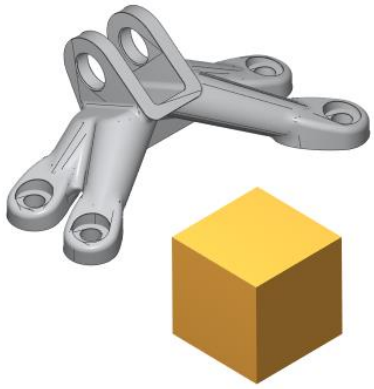


Figure 9.18 The effect of assigning the part as support

9.7 Supported Area Preview

Before generating your support, you can in advance visualize the areas that would need support. Click the “Supported Area” command under the Support Generation menu, the support area preview settings dialog will pop up, as shown in Figure 9.19.

Enter the surface angle value directly, you can adjust the size of the support area, and the support area is displayed in yellow; Check the “Highlight down-facing edges” option to highlight the lowest line of the part; When checking the ‘Show support preview’ checkbox, a preview of the support is displayed, as shown in Figure 9.20. At the same time of the preview, you can change the surface angle or re-position your part based on the analysis of the surfaces and edges. Check the “Clear the marked area while closing” option, the marked support area in yellow will be clear when closing the dialog box, and if this option is not checked, the marked support area in yellow will still be displayed on the part after the dialog is closed.

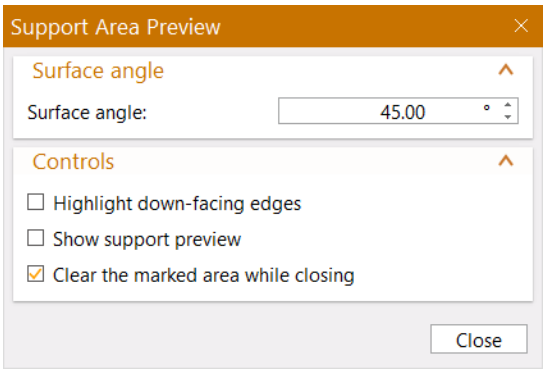


Figure 9.19 Support area preview settings

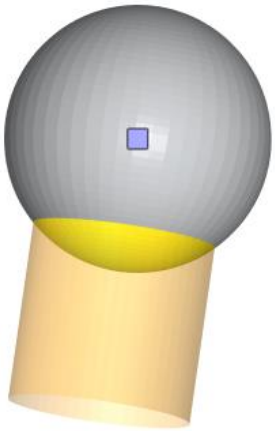


Figure 9.20 Support area preview

9.8 Add No-Support zones

It is difficult to add support on the surface of some parts, and it increases the difficulty for subsequent post-processing operations. In this case, it may be preferential to prevent supports from being generated in these areas, especially when achieving a higher quality surface finish requirement outweighs a possible deformation due to lack of supports.

The steps to create an unsupported area are as follows:

1. Mark the triangle area on the part where no support is added.

2. Click the “Add No-Support” command in the Support Generation menu, the no-support zones are displayed in orange, as shown in Figure 9.21.

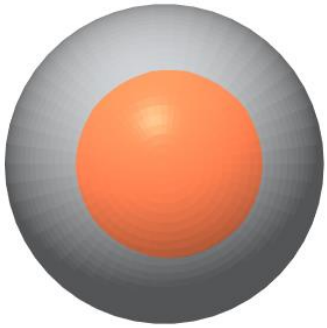


Figure 9.21 No-support zones are shown in orange

9.9 Enable No-Support zones

The “Enable No-Support” command in the Support Generation menu is enabled by default. If it is not enabled, click this command to enable it, and support will not be generated in the unsupported area when performing quick support operation or automatically adding support in the support module. If not enabled, support will still be generated in the unsupported area when adding support, as shown in Figure 9.22.

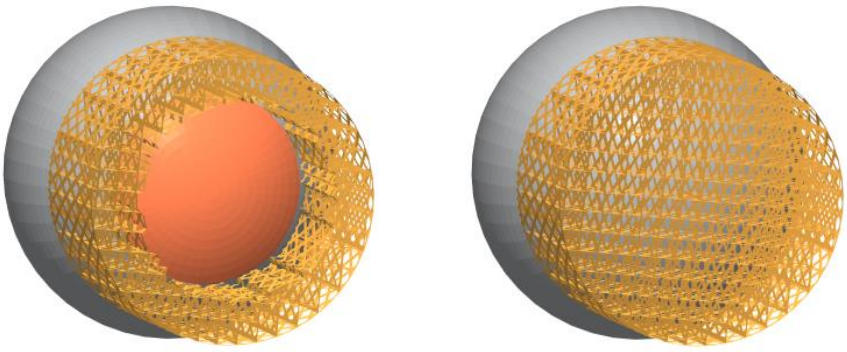


Figure 9.22 Enable no-support areas (left), disable no-support areas (right)

9.10 Toggle No-Support zones

By clicking the “Toggle No-Support” command under the Support Generation menu, you can switch the display and hiding of the unsupported areas of all the parts on the platform.

9.11 Remove No-Support zones

Remove all No-Support Zones

If no-support zones are added to the part, clicking “Remove All” command, all no-support zones on the part will be removed.

Remove Marked No-Support Zones

Firstly, mark one or more added no-support zones on the part, then clicking “Remove Marked” command, the marked no-support zones on the part will be removed.

Pick and Remove No-support Zones

Firstly, use the mouse to select a continuous no-support zones on the part, then clicking “Pick Remove” command, the selected no-support zones on the part will be removed.

10 Analysis

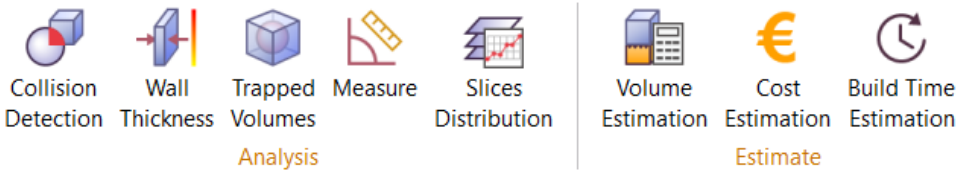


Figure 10.1 Analysis menu

10.1 Collision Detection

After clicking the “Collision Detection” command in the Analysis menu, or pressing the shortcut keys Shift + K, A collision detection dialog will pop up, as shown in Figure 10.2.

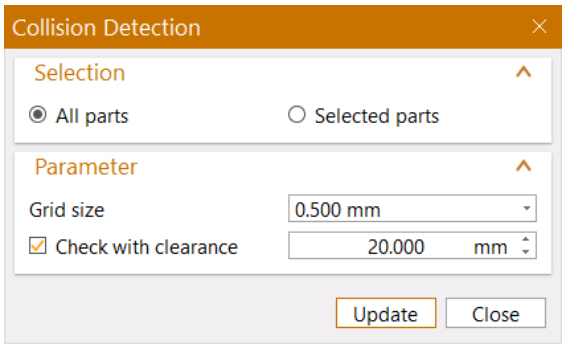


Figure 10.2 Collision Detection setting

If "All parts" is selected, collision detection is performed on all parts on the platform. If "Selected parts" is selected, collision detection is performed only on the selected parts.

Grid size: Determines the grid for checking collisions. The finer the grid, the more time the calculate takes.

Check with clearance: Define the spacing allowed between parts. If they are located within a distance smaller than this value, they will be identified as colliding parts.

After clicking “Update” button, the corresponding result will be displayed on the dialog according to whether there is a collision, as shown in Figure 10.3. Besides, in the display window, you can perform operations such as moving, rotating, and indicate plane to the selected parts. The collision detection will be performed again after each operation on the part.

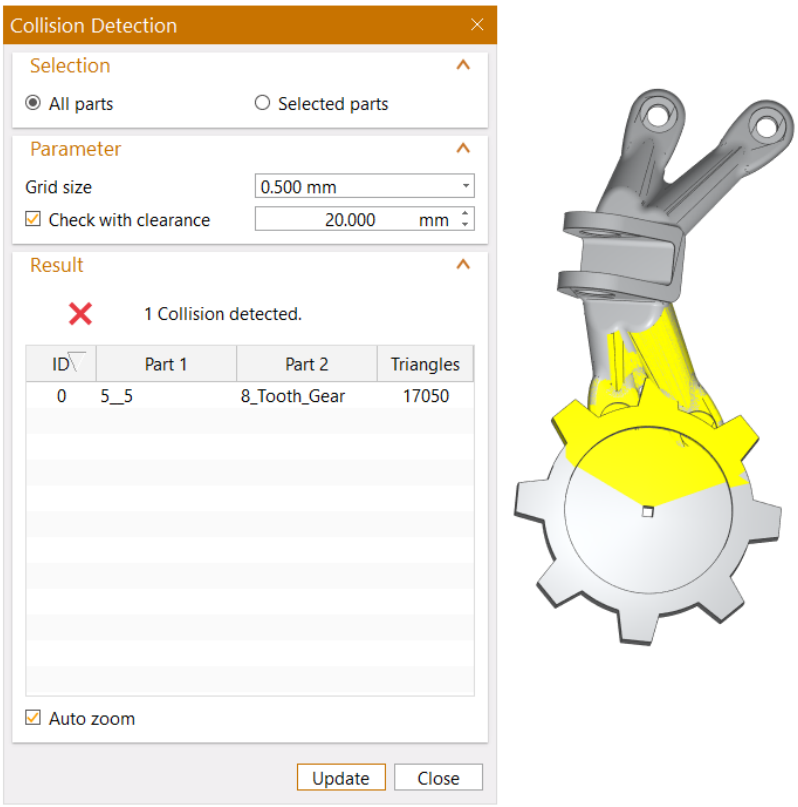


Figure 10.3 Collision Detection effect

10.2 Wall Thickness Analysis

Wall thickness analysis operation helps to detect small details, thin/thick walls, which is helpful for predicting where problems can pop up during building. This module can determine the local wall thickness for each triangle. If requested, you can divide bigger triangles in smaller ones according to the refine triangles parameters entered. This way, a more detailed figure of the wall thickness can be calculated. After clicking the “Wall Thickness” command under the Analysis menu, or the shortcut key Shift + W, the “Wall Thickness Analysis” dialog will pop up in the display window, as shown in Figure 10.4.

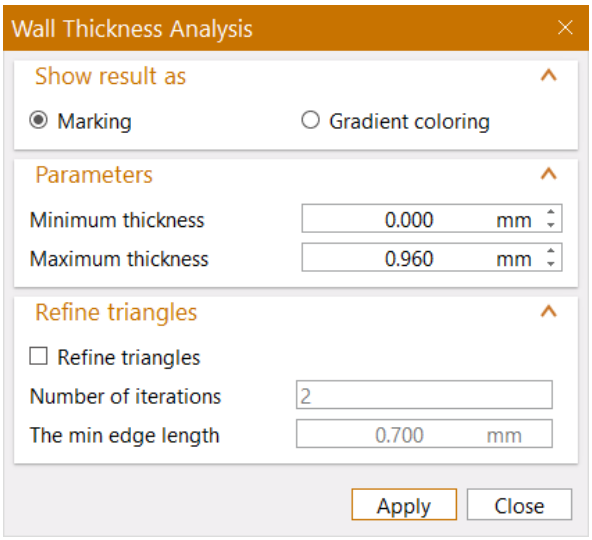


Figure 10.4 Wall Thickness Analysis setting

There are two display results for wall thickness analysis, Marking and Gradient coloring. The corresponding parameters are explained in Table 10.1.

Table 10.1 Parameter settings

Minimum thickness	Define the minimum wall thickness required. Every triangle with a local wall thickness smaller than the entered value, will receive the red color.
Maximum thickness	The maximum thickness is the wall thickness which you no longer expected to have problems. The local thickness bigger than the maximum thickness does not require special attention and will be marked as the yellow color. Triangles with local thickness situated between the minimum and the maximum thickness will have a color gradually changing from the red color (minimum) to the yellow color (maximum wall thickness) over the color spectrum. The minimum and maximum wall thickness also forms the borders of the Wall Thickness Color Legend
Refine triangles	If checked, the triangles that met the criteria are re-triangulated. The analysis is based on the newly created triangles.

Number of iterations	The number of iterations of the triangles that are re-triangulated.
The min edge length	The minimum edge length of the triangles after re-triangulation.

10.2.1 Marking

For the marking display mode, after setting the parameters and analyzing the part, a list is displayed, which contains the areas that match the set conditions, as shown in Figure 10.5. The view will shrink to a specific triangle area on the part when clicking on an item.

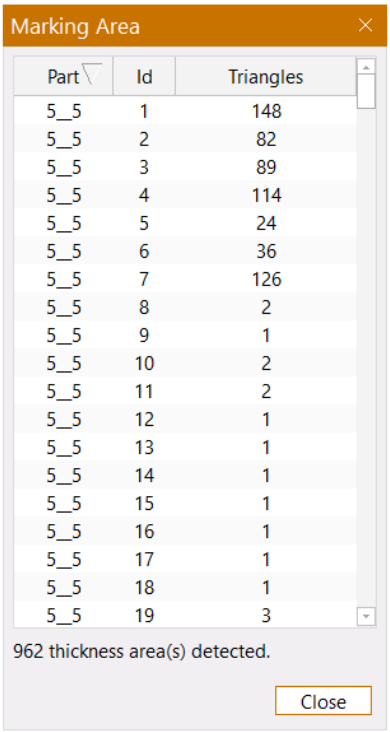


Figure 10.5 Marking area list

10.2.2 Gradient coloring

In the gradient coloring display mode, after setting the parameters, click “Apply” button to display the analyzed color legend, different wall thicknesses correspond to different colors, as shown in Figure 10.6.

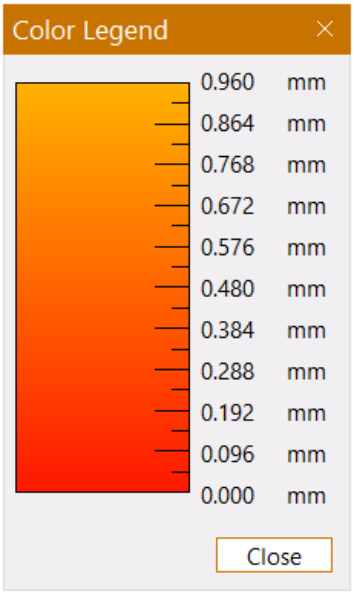


Figure 10.6 Color legend

10.3 Trapped Volumes

Because the height difference between the resin in the cavity and the part will cause the part to deform during the manufacturing process, the user usually needs to determine whether there is a cavity in the part before manufacturing the part.

This function makes it easy to detect the cavity on the part, and once these areas are detected, some perforator operations can be done to avoid part deformation, or to reposition the part to minimize the cavity.

There are two types of trapped volumes that can be detected on your part, which are open volumes and closed volumes.

Open trapped volumes: there is a hole in the cavity in the part, and after the manufacture is completed, the resin in the cavity flows out through the hole.

Closed volumes: the cavity exists inside the part and has no connection with the outside world. The resin in the cavity cannot be discharged after manufacture.

After selecting one or more parts on the platform, click the “Trapped Volumes” command under the Analysis menu, the Detect Trapped Volume parameter setting dialog will pop up in the display window, as shown in Figure 10.7.

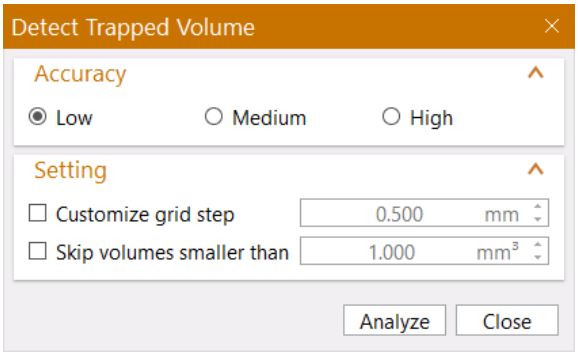


Figure 10.7 Detect trapped volumes setting

The corresponding parameters in the settings dialog are explained in Table 10.2.

Table 10.2 Parameter settings

Accuracy	The higher the accuracy, the longer the analysis time. The software offers low, medium and high precision options.
Customize grid step	If checked, perform analysis based on predefined or entered grid sizes.
Skip volumes small than	Eliminate cavities detection with volume less than the input value.
Analyze	The analysis of the trapped volumes is performed.

After setting the parameters, click “Analyze” button, the detection analysis report will pop up at the bottom of the dialog, as shown in Figure 10.8.

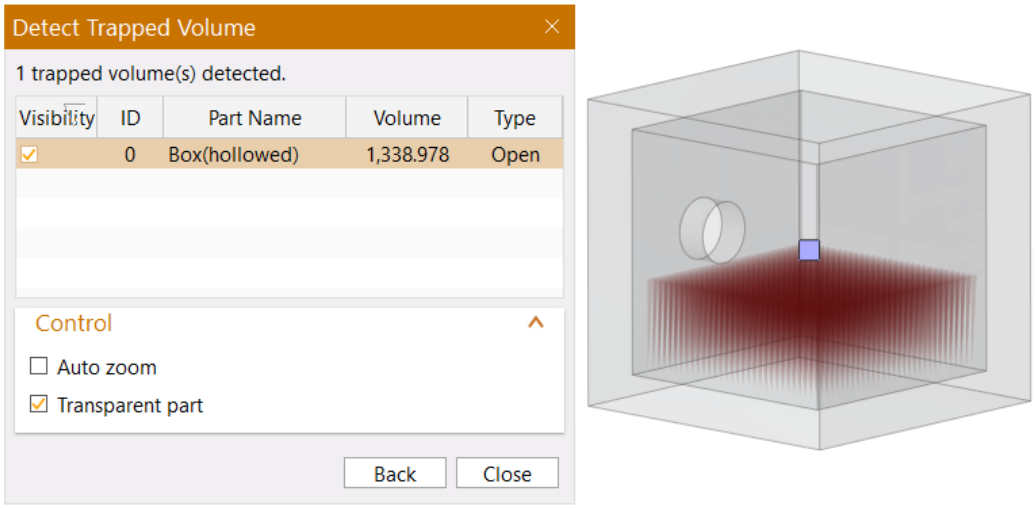


Figure 10.8 Trapped volume analysis report

The corresponding parameters in the report are explained in Table 10.3.

Table 10.3 Trapped volume analysis report parameters

Visibility	Check the check-box ☑ to control whether the cavity in the part is visible and the default is checked.
ID	Each detected cavity has a unique ID identifier.
Part Name	Show part name.
Volume	The volume of the cavity detected.
Type	The type of the cavity detected.
Auto zoom	Automatically zoom to the selected cavity position, and the default is checked.
Transparent part	After checking, the part is in transparent state and the default is checked.

10.4 Measurement

After selecting any part, and clicking the “Measure” command in the Analysis menu, or pressing the shortcut keys Shift + M, you will enter the measurement mode. Measurement parameters settings dialog will pop up, including four sub-pages of Information, Distance, Angle, and Radius, which provides a variety of measurements for the part.

10.4.1 Information

The Information page is used to measure the coordinate information of the basic graphic elements of the part, including points, lines, and triangles, as shown in Figure 10.9. The following three measurement options are available in the drop-down selection box:

(1) Point: Move the mouse to any vertex of any mesh on the graph. The vertex will be marked by a red circle and the XYZ coordinate information of the point will be displayed in the Info field of the page.

(2) Line: Move the mouse to any edge of the graph. The edge will be marked red, and the coordinate information of the two endpoints will be displayed in the Info field.

(3) Triangle: Move the mouse to any triangular area on the graph. The two faces of the triangle will be marked red, and the coordinate information of the three vertices will be displayed in the Info field.

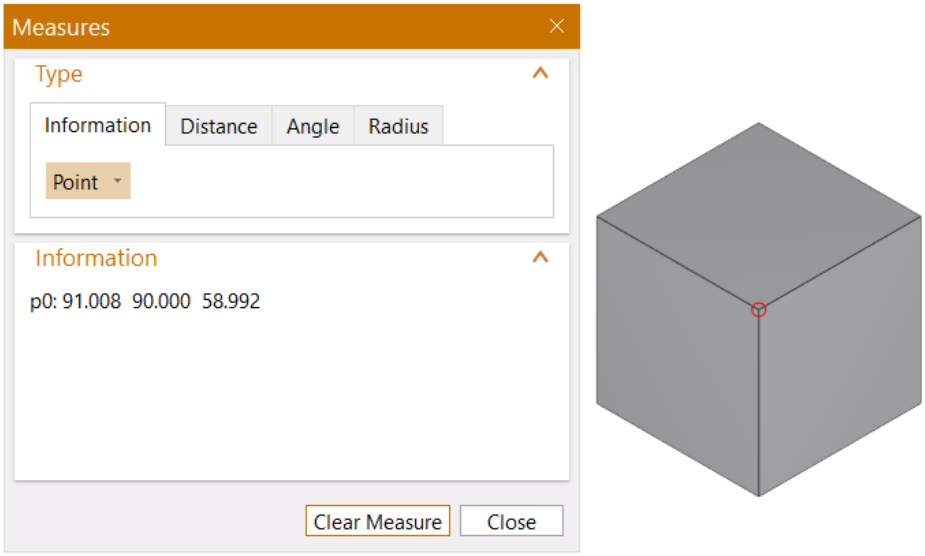


Figure 10.9 Information page and basic measurement

10.4.2 Distance

The distance page is used to measure distance information, and a second feature menu is added in the distance tab, as shown in Figure 10.10. Two feature menus provide a variety of distance types:

(1) Point to point: Measure the distance between two points. Clicks two vertices in turn, and the measurement distance will be displayed in the Info field and on the graph simultaneously.

(2) Point to line: Measure the distance from a point to a line. Clicks the vertex and the edge in order, and the measured distance will be displayed in the Info field and on the graph simultaneously.

(3) Point to triangle: Measure the distance from a point to a triangle. Clicks the vertex and the triangle in order, and the measured distance will be displayed in the Info field and on the graph simultaneously.

(4) Point to axis: Measure the distance from a point to the X axis, Y axis, or Z axis. Clicks a point on the part, and the measured distance will be displayed in the Info field and on the graph simultaneously.

(5) Point to the plane: Measure the distance from a point to the X-Y, X-Z, or Y-Z plane. Clicks a point on the part, and the measured distance will be displayed in the Info field and on the graph simultaneously.

(6) Line to line: Measure the distance between two lines. Clicks the two lines in turn, and the measured distance will be displayed in the Info field and on the graph simultaneously.

(7) Line to plane: Measure the distance from the line to the parallel plane. Clicks the line and the parallel plane, and the measured distance will be displayed in the Info field and on the graph simultaneously.

(8) Plane to plane: Measure the distance between two parallel planes. Clicks the two parallel faces, and the measured distance will be displayed in the Info field and on the graph simultaneously.

(9) Thickness: Measure the thickness of a plane. Clicks any point on the measured plane, and the measured thickness will be displayed in the Info field and on the graph simultaneously.

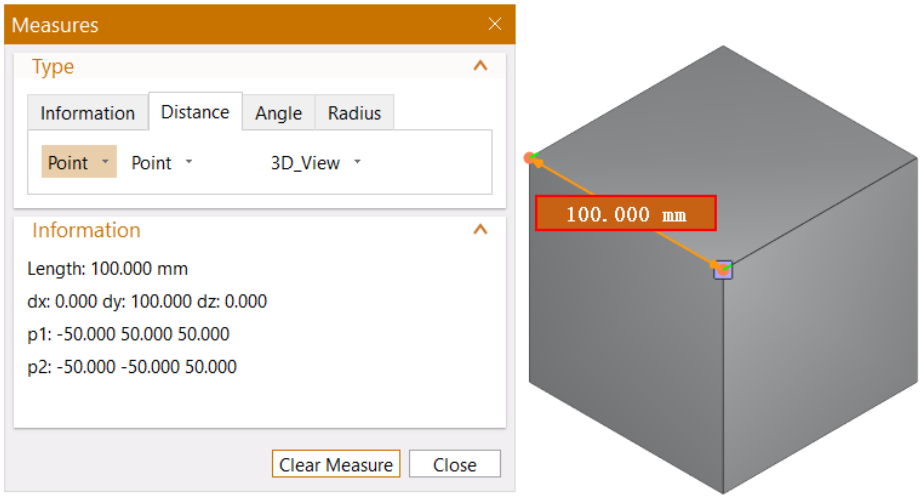


Figure 10.10 Distance page and the distance measurement

(10) 3D_view：European distance in three-dimensional space.

(11) 2D_XY：The distance parallel to the X-Y plane.

(12) 2D_XZ：The distance parallel to the X-Z plane.

(13) 2D_YZ：The distance parallel to the Y-Z plane.

(14) 1D_X：The distance parallel to the direction of the X axis.

(15) 1D_Y：The distance parallel to the direction of the Y axis.

(16) 1D_Z：The distance parallel to the direction of the Z axis.

10.4.3 Angle

The Angle page is used to measure angle information, and a second feature menu is added in the angle tab, as shown in Figure 10.11. The two feature menus provide a variety of angle measurement types:

(1) 3 Points: Clicks the three vertices in turn, and then you can measure the angle with the second vertex as the center. The angle information will be displayed in the Info field and on the graph simultaneously.

(2) Line to line: Clicks the two lines in turn, and then you can measure the intersection angle of the two lines. The angle information will be displayed in the Info field and on the graph simultaneously.

(3) Line to plane: Clicks on the line and the non-parallel plane, and then you can measure the intersection angle between the line and the non-parallel plane. The angle information will be displayed in the Info field and on the graph simultaneously.

(4) Line to axis: Measure the intersection angle between the line and the X axis, Y axis, or Z axis. Clicks on a line on the part, the angle information will be displayed in the Info field and on the graph simultaneously.

(5) Line to plane: Measure the intersection angle between the line and the X-Y, X-Z, or Y-Z plane. Clicks on a line on the part, the angle information will be displayed in the Info field and on the graph simultaneously.

(6) Plane to plane: Clicks the two triangular facets in turn, and then you can measure the intersection angle of the two faces. The angle information will be displayed in the Info field and on the parts simultaneously.

(7) Plane to axis: Measure the intersection angle between the plane and the X axis, Y axis, or Z axis. Clicks on a plane on the part, the angle information will be displayed in the Info field and on the graph simultaneously.

(8) Plane to plane: Measure the intersection angle between the plane and the X-Y, X-Z, or Y-Z plane. Clicks on a plane on the part, the angle information will be displayed in the Info field and on the graph simultaneously.

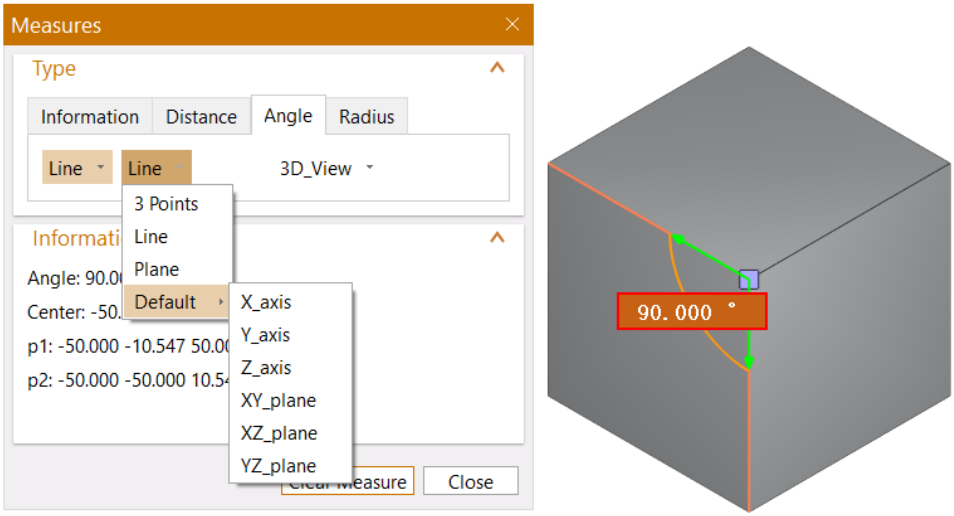


Figure 10.11 Angle page and angle measurement

(9) 3D_view：The angle in three-dimensional space.

(10) 2D_XY：The angle parallel to the X-Y plane.

(11) 2D_XZ：The angle parallel to the X-Z plane.

(12) 2D_YZ：The angle parallel to the Y-Z plane.

10.4.4 Radius

The Radius page is used to measure the radius of a selected circle or an arc, as shown in Figure 10.12. The following two measurement options are available in the drop-down selection box:

(1) 3 Points: Click three vertices which are not collinear in turn, and the three vertices will determine a circle. The circle with its center will show on the graph, and the radius information will be displayed in the Info field and on the graph simultaneously.

(2) Circle: In the show Part Boundary mode, click any circle or arc on the boundary lines of the part. The circle with its center will show on the graph, and the radius information will be displayed in the Info field and on the graph simultaneously.

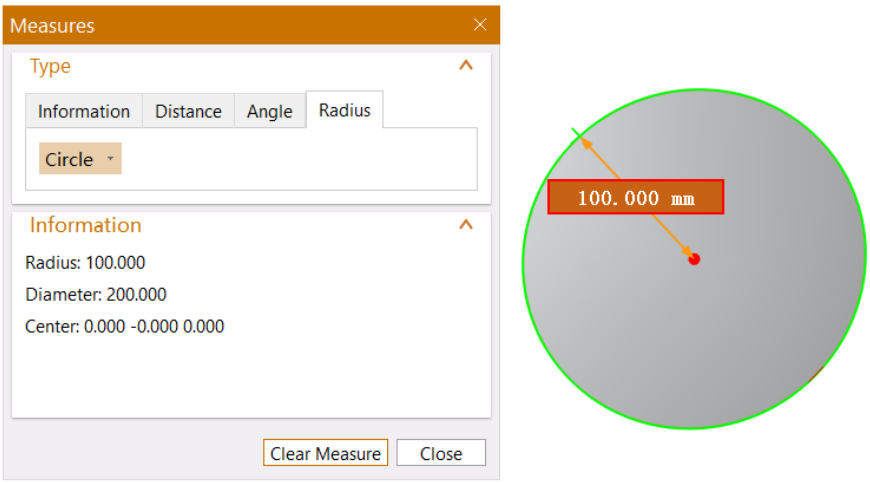


Figure 10.12 Radius page and radius measurement

If you need to measure for multiple times, you can clear the existing measurement data in the Info field and the marks on the part by clicking the “Clear Measure” button at the bottom of the dialog.

10.5 Slices Distribution

Click the “Slice Distribution” command in the Analysis menu, and the slice area distribution map of all parts will appear, as shown in Figure 10.13. The slice distribution graph gives you the possibility to analyze the surface area for each layer and the distribution within the platform.

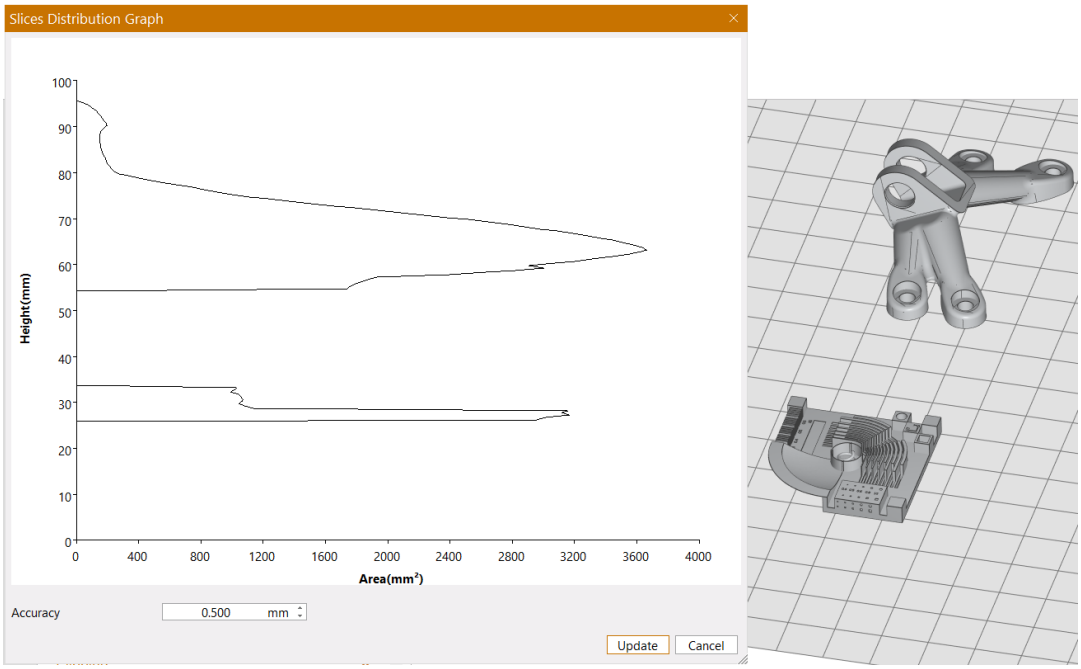


Figure 10.13 Slice distribution graph

Some operations can change the covered area, such as moving, rotating parts, and so on. In this case, you need to click the "Update" button to recalculate.

10.6 Volume Estimation

Show the volume estimation of non-solid support. To calculate the volume estimate, you need to set the laser parameters of the platform attributes in the Platform Definition firstly (for parameter settings, see Chapter 12.2). After clicking the “Volume Estimation” command under the Analysis menu, a message prompt box will pop up, as shown in Figure 10.14. Click “Parameters” button to directly go to laser parameters setting dialog. Click OK to close the dialog. And the volume estimation for selected parts and supports will be displayed in the lower right corner of the interface window.

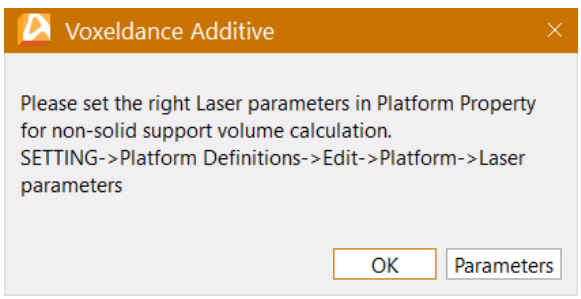


Figure 10.14 Message prompt dialog

10.7 Cost Estimation

Show the cost estimation of the materials consumed for printing part(s). To calculate the cost estimation, you need to set the unit cost price, material density and currency in the Platform Definition firstly (for parameter settings, see Chapter 12.2). After clicking the “Cost Estimation” command under the Analysis menu, a message prompt box will pop up, as shown in Figure 10.15. Click Parameters button to directly go to cost parameters setting dialog. Click OK to close the dialog. And the cost estimation for all parts will be displayed in the lower right corner of the interface window, and the cost estimation for each part will be displayed beside each part name in the Parts List.

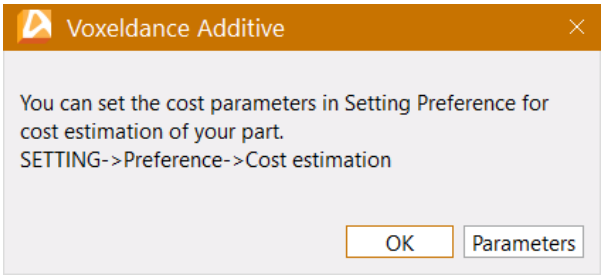


Figure 10.15 Message prompt dialog

10.8 Build Time Estimation

Show an estimation of the build time of the part(s). To calculate the time estimation, you need to set the laser parameters and build parameters of the platform attributes in the Platform Definition firstly (Parameter setting see chapter 13.4). After clicking the “Build Time Estimation” command under the Analysis menu, a message prompt box will pop up, as shown in Figure 10.16. Click Parameters button to directly go to laser parameters and build parameters setting dialog. Click OK to close the dialog. And the total build time estimation for all parts will be displayed in the lower right corner of the interface window.

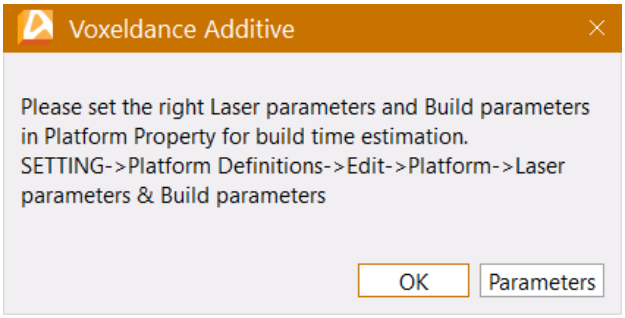


Figure 10.16 Message prompt box

11 Slice

Select the part, and click the “Slice” command in the Home menu. A dialog box for slice operation will pop up in the interface window, as shown in Figure 11.1.

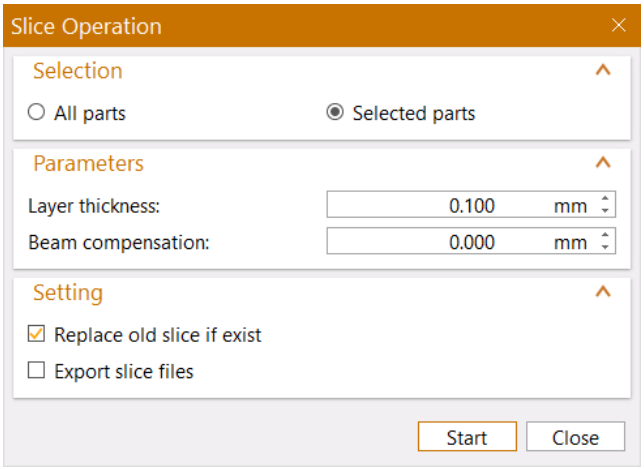


Figure 11.1 Slice operation setting

If "All Parts" is selected, all parts on the platform will be sliced. If "Selected Parts" is selected, only selected parts will be sliced. Set the slice layer thickness and beam compensation according to the characteristics of the machine. If you select the "Export slice files" check box, you can export the slice file to a local folder.

After setting the parameters, click the "Start" button to slice the file, and you can enter the slice window, or you can click the "  " icon button on the sidebar to enter the slice window, as shown in Figure 11.2, the slice file is added to slice list on the left side. The slicing mode also provides zooming operations, as well as controlling the display or hiding of grids, platforms, rulers, and coordinate systems.

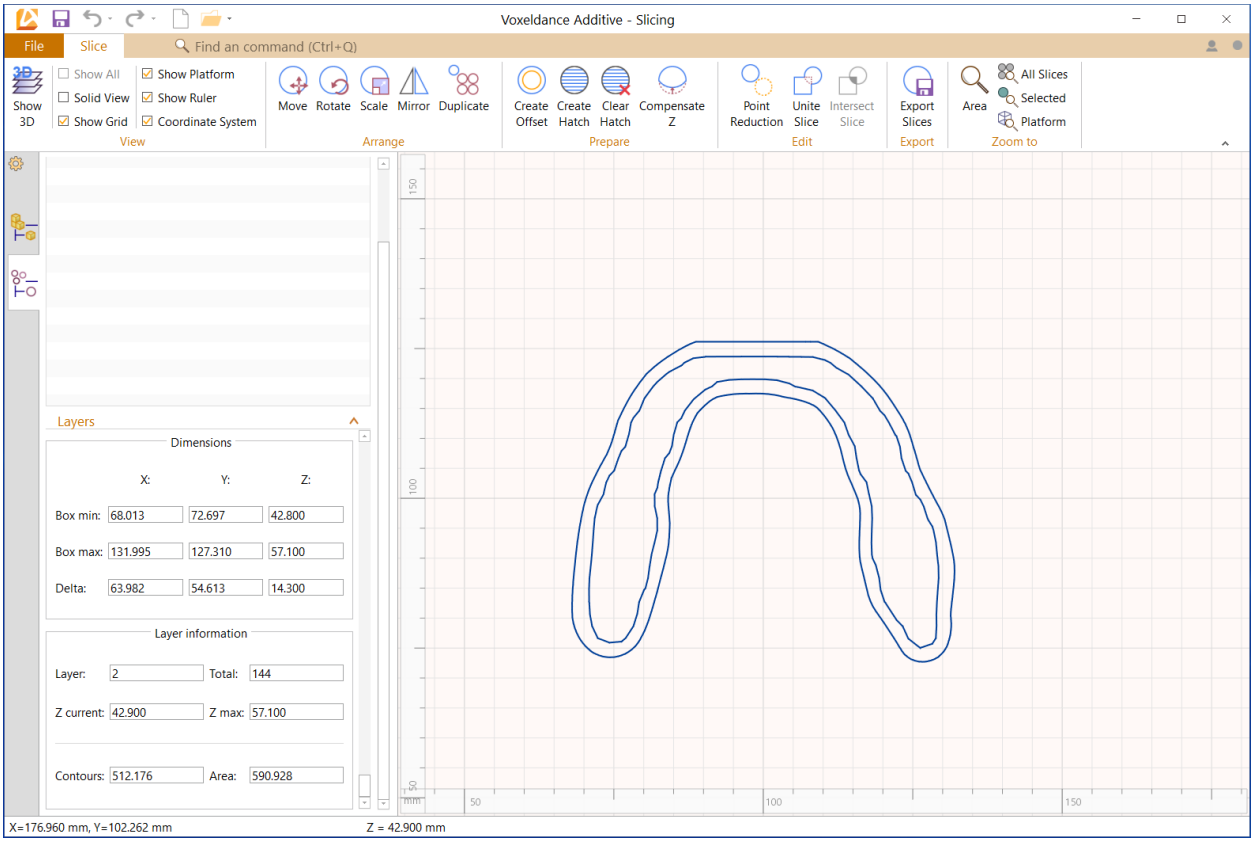


Figure 11.2 Slice window

The Layers field displays the detailed information for the selected slice file, including Dimensions and Layer information. The right scroll bar is used to change the Z-axis coordinate to observe different layers, and the information of the current layer is displayed in the Layer Information field, including Layer, Total, Z current, Z max, Contours, and Area.

In the Slice menu, click the "Show 3D" command to switch between 2D and 3D views. In the 3D view, if you check the "Show All" selection box, you can observe the whole part in the sliced state. If you check the "Solid View" selection box, you can observe the solid view of each slice layer to better distinguish it from the supported slice layer. Moreover, the Z-axis coordinates of the current slice and the two-dimensional (X-Y) coordinates of the cursor position is shown at the bottom of the display window.

Move Slices:

Select the slices, and click the "Move" command on the menu. A dialog for moving the slices will pop up, as shown in Figure 11.3.

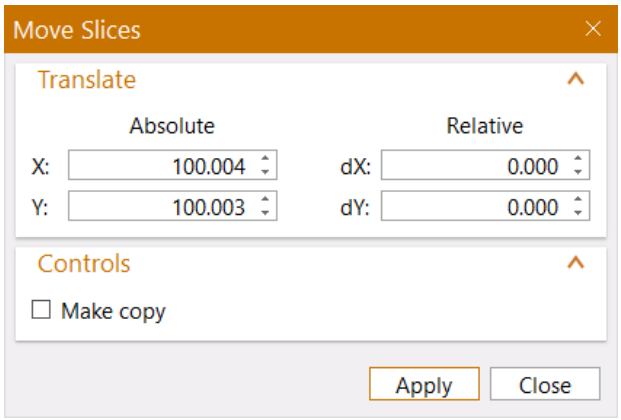


Figure 11.3 Move slices setting

Set the distance to move and whether you need to duplicate this slice, then click the “Apply” button.

Rotate Slices:

Select the slices, and click the "Rotate" command on the menu. A dialog for rotating the slices will pop up, as shown in Figure 11.4.

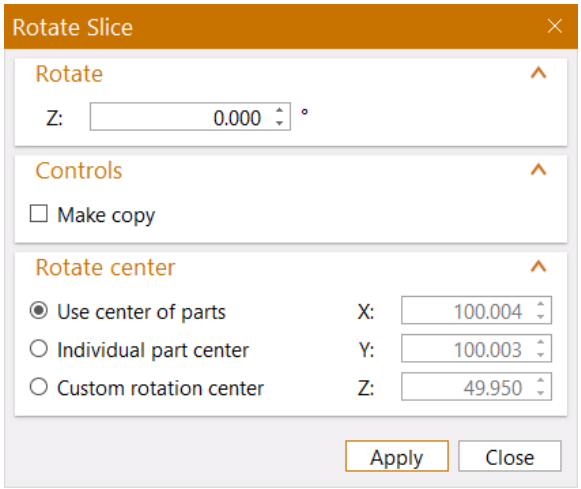


Figure 11.4 Rotate slices setting

Set the rotation angle, rotation center, and whether you want to copy this slice, then click the "Apply" button.

Scale Slices:

Select the slices, and click the "Scale" command on the menu. A dialog for scaling the slices will pop up, as shown in Figure 11.5.

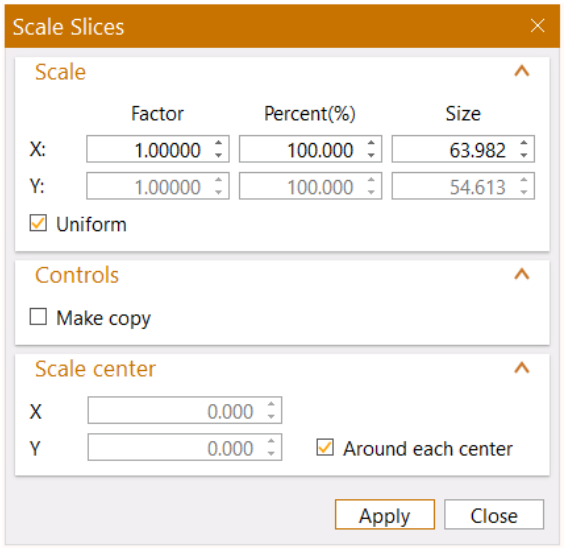


Figure 11.5 Scale slices setting

Set the zoom factor, zoom center, and whether you want to copy this slice, click the “Apply” button.

Mirror Slices:

Select the slices, and click the "Mirror" command on the menu. A dialog for mirroring the slices will pop up, as shown in Figure 11.6.

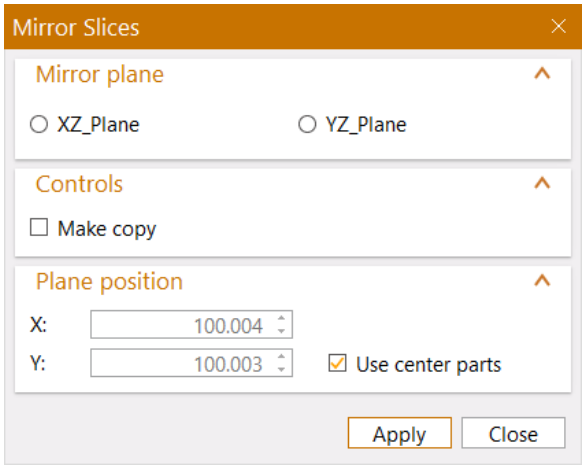


Figure 11.6 Mirror slices setting

Set the mirror plane, plane position, and whether you want to copy this slice, click the "Apply" button.

Duplicate Slices:

Select the slices, and click the "Duplicate" command on the menu. A dialog for duplicate slices will pop up, as shown in Figure 11.7.

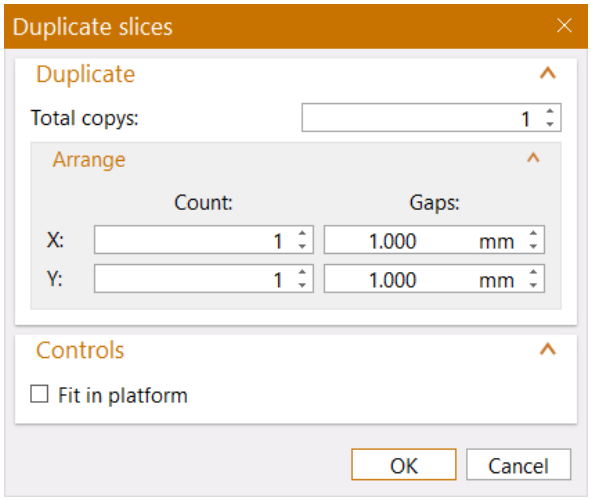


Figure 11.7 Duplicate slices setting

Set the copy number, gaps, and whether you need to fit the platform, click the "OK" button.

Offset Slices:

Select the slices, and click the "Create Offset" command on the menu. A dialog for offset slices will pop up, as shown in Figure 11.8.

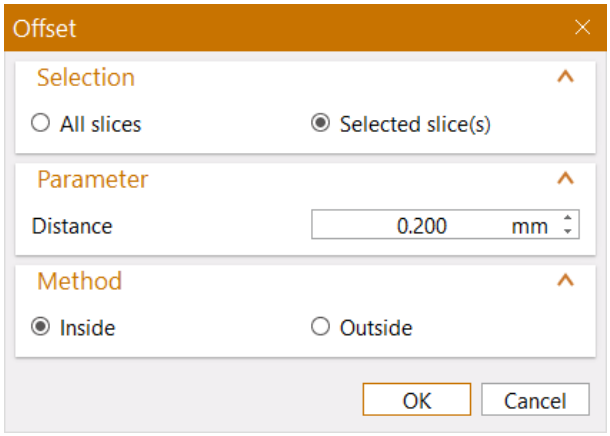


Figure 11.8 Offset slices setting

If "All slices" is selected, all slices on the platform will be offset. If "Selected slices" is selected, only selected slices will be offset. Then set the offset distance and method (inside or outside), click the "OK" button, a new slice path parallel to the original slice path and offset by a certain distance will be generated.

Create Hatch:

Select the slice, and click the "Create Hatch" command on the menu. A dialog box for creating hatch will pop up, as shown in Figure 11.9.

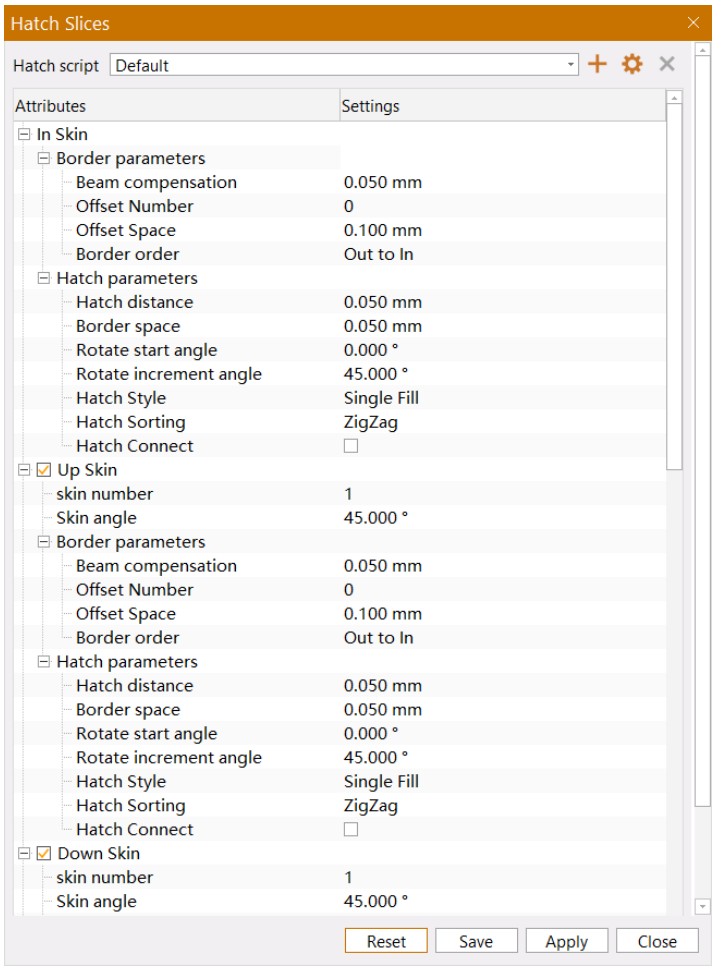
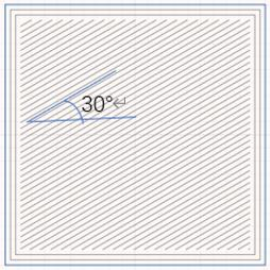
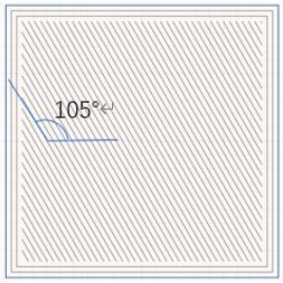
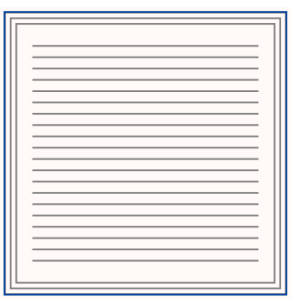
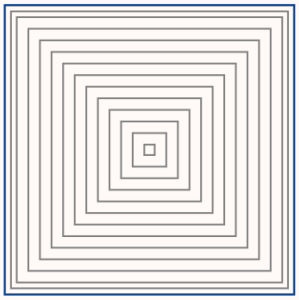
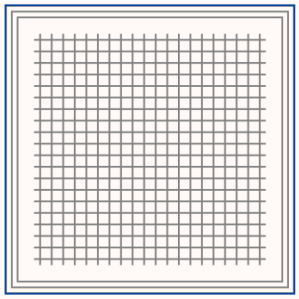
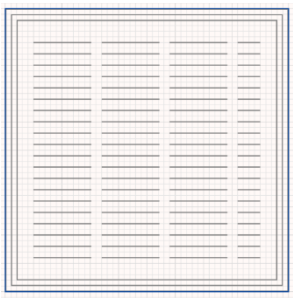
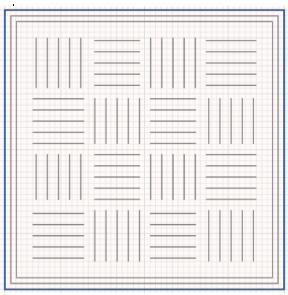


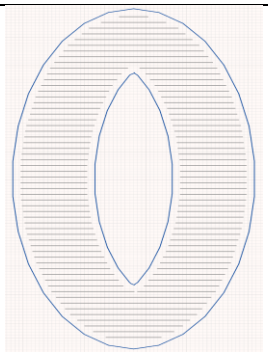
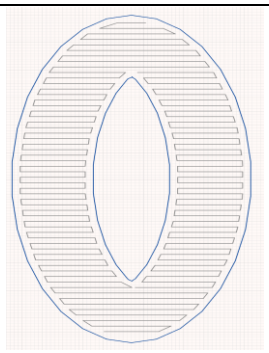
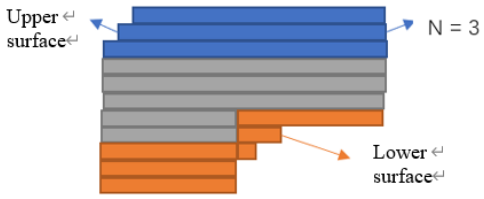
Figure 11.9 Hatch slices setting

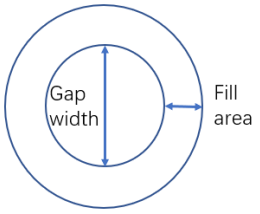
Parameter introduction is shown in Table 11.1

Table 11.1 Parameters explanation

In Skin, Up Skin, Down Skin、Z Connect Skin, Hole/Slot Skin	Border parameters	Beam compensation	Set the beam compensation value for the slice border line
		Offset number	Define the number of offset borders, enter "1" to generate a border, enter "2" to generate two borders, and so on
		Offset Space	Distance between two adjacent borders
		Border order	From inner border to outer border, or from outer border to inner border
	Hatch parameters	Hatch distance	Distance between two adjacent hatch lines
		Border space	Distance from hatch line to innermost border
		Rotate start angle	The hatch line of each layer can be rotated at a set incremental rotation angle, and the starting angle defines the initial rotation angle 
		Rotate increment angle	Defines the rotation angle increment of each layer, for example, enter "45", which means that the rotation angle of each layer increases by 45 degrees 
		Hatch style	Define the hatch style Single fill: Generates a parallel straight scan path Offset fill: Generates an offset closed contour scan path starting from the border Cross fill: Generates straight scan paths that are perpendicular to each other

			Stripe fill: Generate stripe hatch, gap can be set between stripes Chess fill: Generates chess hatch, gap can be set between each rectangular area   Single fill Offset fill   Cross fill Stripe fill  Chess fill
		Hatch sorting	Two type, zigzag fill (no jump) and block sorting (jump)

			  Zigzag (no jump) block sorting (jump)
		Hatch connect	Defines whether single path with hatch lines end to end (only valid for block sorting jump mode)
Up Skin, Down Skin		Skin number	Define the number of layers on the upper and lower surfaces, for example, enter "3" to start from the lower surface. The overlapping area of the three layers above is the lower surface 
		Skin angle	Define the recognition angle of the upper and lower surfaces. When the surface slope angle is less than the set value, the upper and lower surfaces are recognized.
Z Connect Skin		Area factor	The area of the connection area / the area of the current layer (The Z connection area refers to the connection area (overlapping part) of the current layer and the previous layer. If the area ratio is less than the set value, it is the Z connection area, and then the upper and lower N layers corresponding to the identified connection area are also as Z Connection area.)
		Upper number	When the area change ratio of the upper and lower layers is less than the set value (the default is 0, 1), the current layer is the Z-axis joint connection layer. The number of upper recognition layers defines the number of layers in the plane of the N corresponding layers up to the current connected layer.
		Lower number	The number of recognition layers on the lower surface defines the number of layers in the plane of the N corresponding layers down to the current connected layer.
Hole/Slot Skin		Gap width	Defines the threshold for identifying the gap in holes and slot, in mm. When the diameter of the hole or the maximum gap width of the slot is smaller than this value, it is identified as a slot. For example, if the gap width is set to

			1.0mm, a hole with a diameter less than 1.0mm is identified as a hole feature, and the maximum gap width of the slot is less than 1.0mm is identified as a slot feature.
			
		Hatch range	Define the extended range of the generated fill path for holes

In the "Hatch Slices" dialog, click the "+" icon button to add a new hatch script, click "⚙" to set the default hatch script, and export / import the parameter file of the hatch script

Clear Hatch:

Select the slices with hatch, and click the "Clear Hatch" command on the menu, the hatch in the slice will be cleared.

Z Compensation:

Select the slices, and click the "Compensation Z" command on the menu. The Z-compensation dialog will pop up, as shown in Figure 11.10.

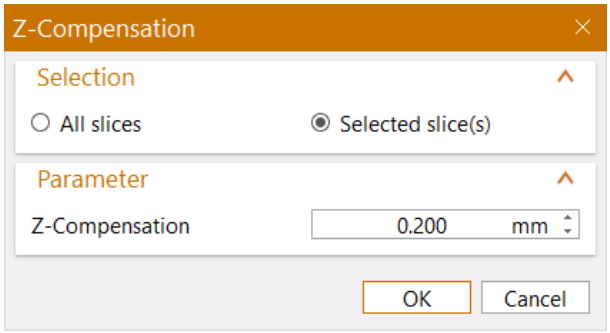


Figure 11.10 Z-Compensation setting

If "All slices" is selected, Z-compensation will be performed on all slices on the platform. If "Selected slices" is selected, Z-compensation will only be performed on the selected slices. Then set the Z-compensation distance, and click the "OK" button.

Point Reduction:

Select the slice, and click the "Point Reduction" command on the menu. A dialog box for reducing points will pop up, as shown in Figure 11.11.

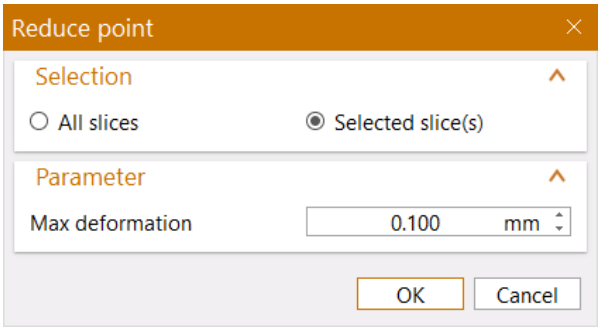


Figure 11.11 Reduce point setting

If "All slices" is selected, the point reduction is performed on all slices on the platform. If "Selected slices" is selected, the point reduction is performed only on the selected slices. Then set the maximum deformation value, and click the "OK" button. This operation reduces the amount of points needed to maintain the original shape while adhering to a specifiable tolerance of deformation. Be mindful not to specify too large a value for the maximum deformation as this may impair the quality of rounded contours to unacceptable levels.

Unite Slices:

Select the slice, and click the "Unite Slices" command on the menu. A Slices union dialog will pop up, as shown in Figure 11.12

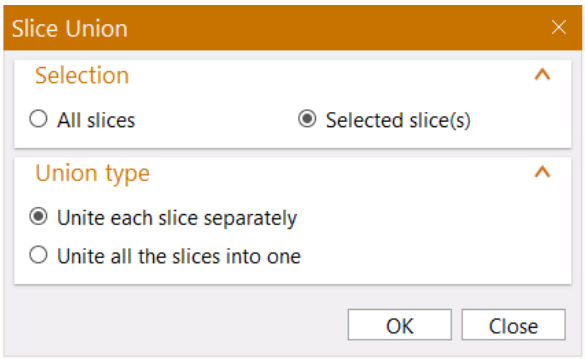


Figure 11.12 Slices union setting

If "All slices" is selected, the union operation is performed on all slices on the platform. If "Selected slices" is selected, only the selected slices will be union. Then set the union type, and click the "OK" button.

There are two union type. The first type, "Unite each slice separately," is to remove the self-intersecting part of each slice file. As shown in Figure 11.13, the second type, "Unite all the slices into one" is to merge two or more slice files into one file, and also remove the self-intersecting part.

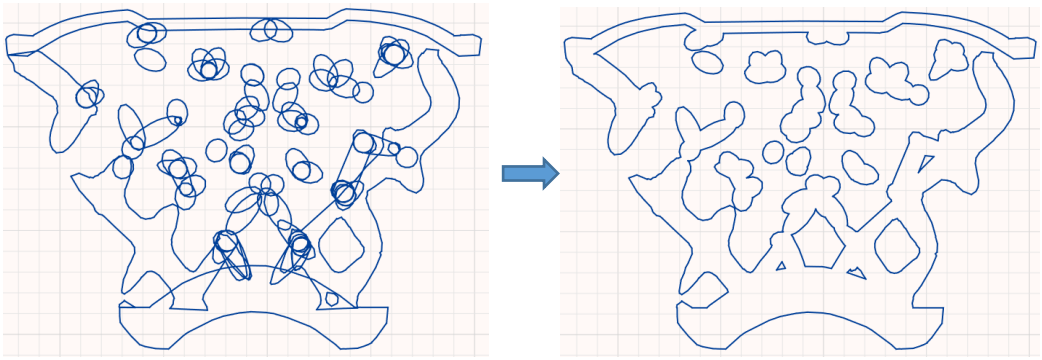


Figure 11.13 Unite one slice

Intersect Slices:

Select two or more slices, and click the "Intersect Slice" command on the menu. Only the common area of all selected slices is retained.

Export Slices:

Select the slice, and click the "Export Slices" command on the menu. A dialog for exporting the slice file will pop up, as shown in Figure 11.14.

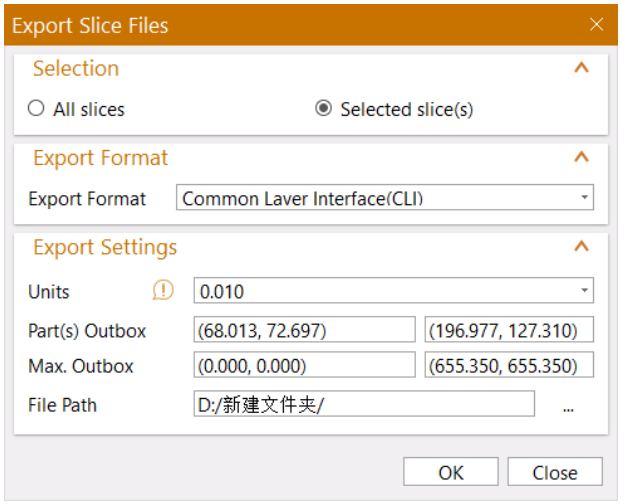


Figure 11.14 Export slice setting

If "All slices" is selected, all the slices on the platform will be exported, if "Select slices" is selected, only the selected slices will be exported. Then set the export format, the related parameter settings of this format, and the exported file path, and click the "OK" button.

12 Setting

12.1 Option Setting

Click the "Options" command under the File menu or the "  " icon button on the sidebar, and the option setting dialog will pop up, as shown in Figure 12.1.

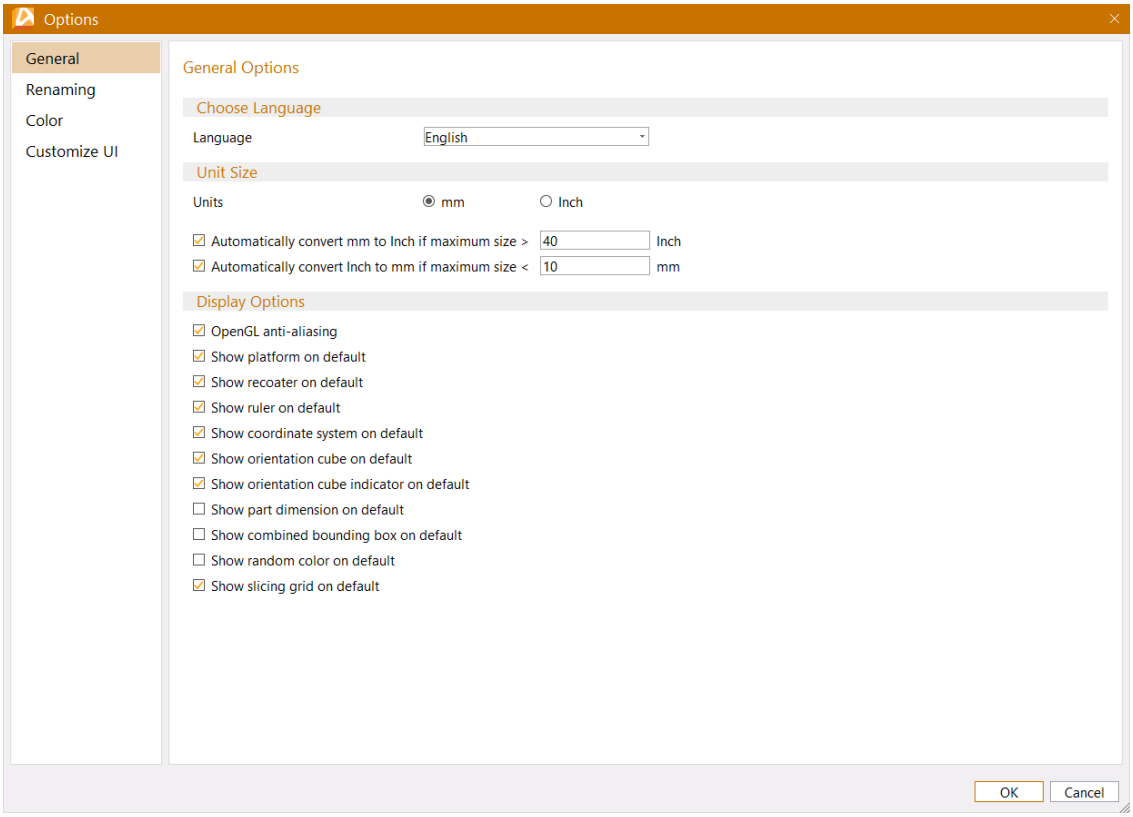


Figure 12.1 Options setting

General setting:

1）Language

The currently supported languages include Chinese, English, Japanese, German, Spanish, and Korean.

2）Unit Size

The imported part looks too small or too large, and a dialog box asks if you want to convert units.

3）Display Options

OpenGL anti-aliasing: The smooth edge of the part brings better visual effects.

Show platform on default: Set whether to display the platform after launching the software.

Show recoater on default: Set whether to display the recoater after launching the software.

Show ruler on default: Set whether to display the ruler after launching the software.

Show coordinate system on default: Set whether to display the coordinate system after launching the software.

Show orientation cube on default: Set whether to display the orientation cube after launching the software.

Show orientation indicator on default: Set whether to display the orientation indicator after launching the software.

Show part dimension on default: Set whether to display the part dimension after launching the software.

Show combined bounding box on default: Set whether to display the combined bounding box after launching the software.

Show random color on default: Set whether to display the random color after launching the software.

Show slicing grid on default: Set whether to display the slicing grid after launching the software.

Rename setting:

In the Options dialog, click the Renaming option on the left, and the page for setting the rename function will open, as shown in Figure 12.2.

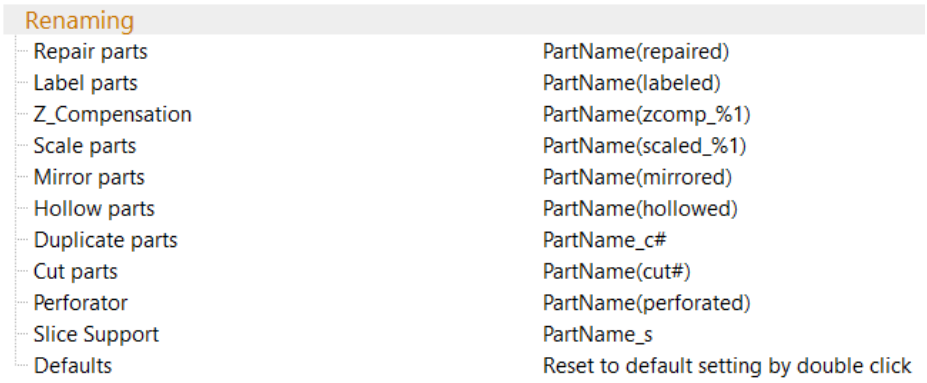


Figure 12.2 Rename setting

After performing repairing, labeling, z-compensation, scaling, mirroring, hollowing, duplication, cutting and perforation on the original part, a new part or several new parts will be created, such as “part1_repair” and “part2(hollowed)”. The renaming format of the new created part(s) can be set here. Double clicked on a function in the list that needs a change (such as Repair Parts) in its renaming style, an editing dialog appears for users to choose from the predefined renaming or to design custom templates, as shown in Figure 12.3. You can insert what will be added before or after the part name in the text fields left and right of “part name”. Below, you can see a preview of how your parts will be named. "Part name" always refers to the name of the original part. After all the input, click the “+” icon on the right of Preview so that the custom design will be added to the predefined templates to choose from. With the button "Set to defaults" at the bottom of the dialog, you can restore the default predefined naming for that particular function.

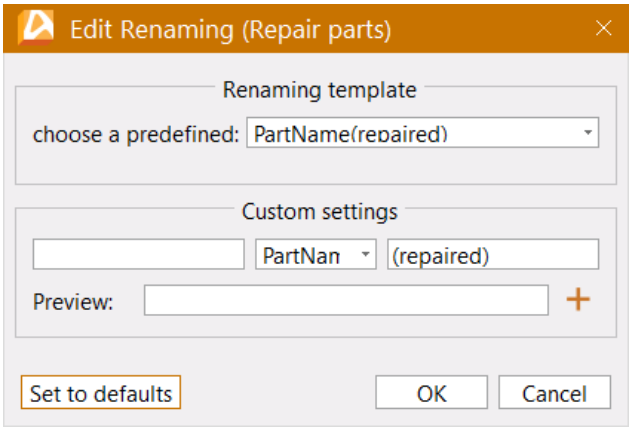


Figure 12.3 Rename format edit setting

After setting the naming format of these functions, double-click " Reset default settings by double click" to restore the default naming format of all functions.

Color Setting:

In the Options dialog, click the Color option on the left, you can open the page for setting the color of parts, slices, and some objects in repair and support, as shown in Figure 12.4.

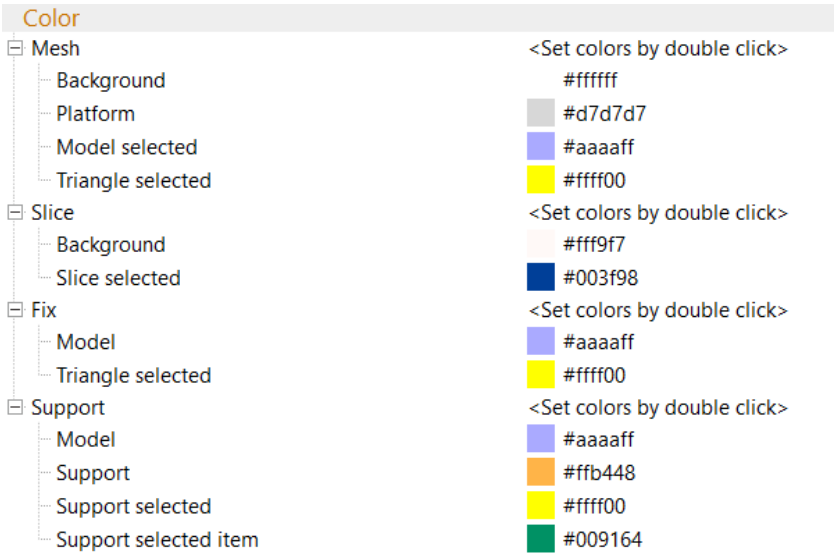


Figure 12.4 Color setting

The color dialog provides the entire spectrum of each color, as shown in Figure 12.5.

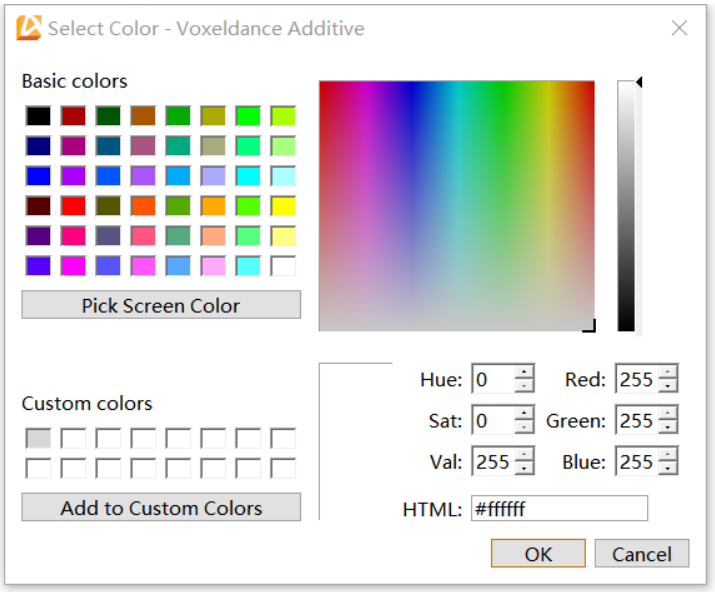


Figure 12.5 Select color

In addition, for part and slice, you can also change the color in the displayed color dialog by double-clicking the color icon on the right side of the part or on the left side of the slice.

Customize UI Setting:

In the Options dialog, click the Customize UI option on the left, you can open page for setting the user interface, as shown in Figure 12.6. Users can customize shortcut keys, ribbons, toolbars, and right-click menus if need.

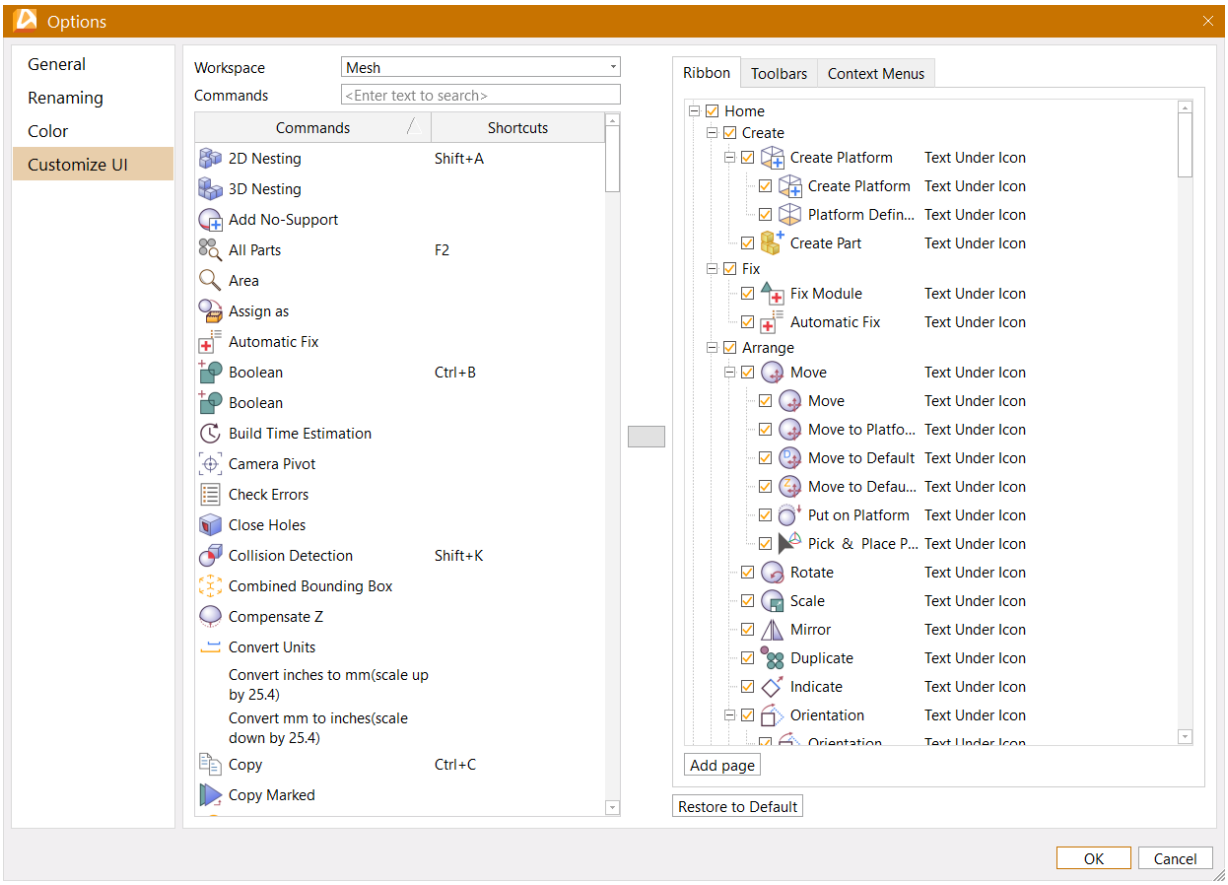
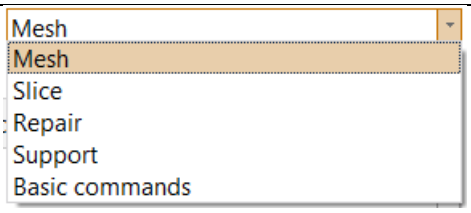
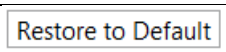


Figure 12.6 Customize UI setting

The corresponding function parameters in the dialog are explained in Table 12.1.

Table 12.1 Interface function

Workspace	 Different interface modes defines their own toolbars and context menus. The basic commands are to set shortcut keys for the function without the corresponding toolbar command.
Commands	A list of all commands in Voxeldance Additive. If a command has the keyboard shortcuts, it is shown next to the command. Scroll through the list, or use the search box to find a specific command
	Search box for the Commands.
Keyboard shortcuts	To assign or delete a keyboard shortcut, double-click the shortcut area and enter or delete the shortcut. 
Ribbon	The ribbon tab contains all the ribbons and their available functions. The visible ribbons and functions have a check. Different interface mode corresponds to different default toolbars.
Toolbars	The toolbar tab contains all the toolbars and their available functions. The visible toolbars and functions have a check. Different interface mode corresponds to different default toolbars.
Context Menus	The context menu tab contains all context menus and their commands. Different interface mode corresponds to different context menus.
	Restores the default settings of the ribbon, toolbar, and right-click menu.

1) Ribbon:

In Voxeldance Additive， users can customize the ribbon according to their needs. At present, there are default ribbon and custom ribbon. The default ribbon cannot be deleted or renamed, but the custom ribbon can be added, named and deleted by the user.

Take the main interface mode as an example, as shown in Figure 12.7, the corresponding commands in the ribbon can be operated by clicking the function button below, or by the right-click menus of the page, the group or a command.

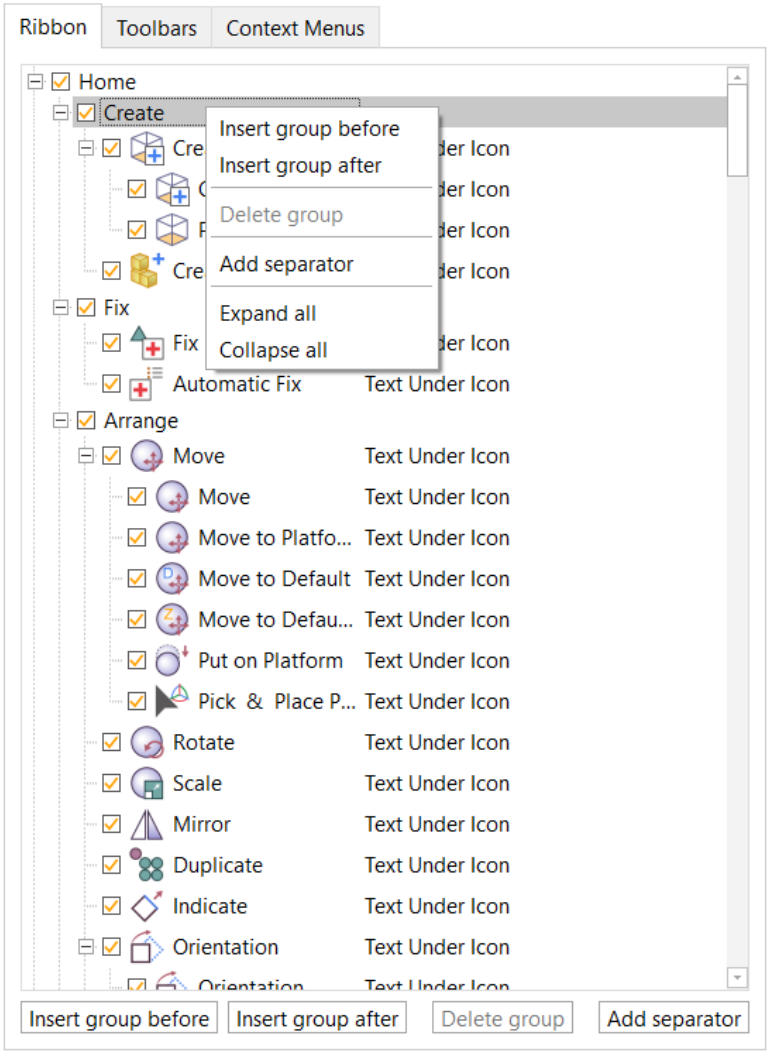
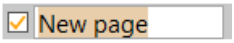
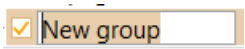


Figure 12.7 Ribbon setting

To create a custom ribbon, proceed as Table 12.3.

Table 12.3 Create a custom ribbon

1.	After selecting the workspace where the page needs to be added, select a page in the ribbon tab and then click Insert group before or Insert group after to add a new page.
2.	Enter a name for the new page. 
3.	Click Add group to add a new group。
4.	Enter a name for the new group. 
5.	Select the command on the left and drag it to the new group to add the command. A separator can be added between the added group commands.
6.	Click “Save” to finish the creation of the ribbon. The new ribbon is visible in the selected interface mode.

2) Toolbars：

In Voxeldance Additive， users can customize the toolbars as required. Currently, there are default toolbars and custom toolbars. The default toolbar cannot be deleted or renamed, but the custom toolbar can be added, named, and deleted by the user.

Take the main interface mode as an example, as shown in Figure 13.6, the corresponding commands in the toolbar tab can be operated by clicking the function button below, or by the right-click menus of the toolbar or a command.

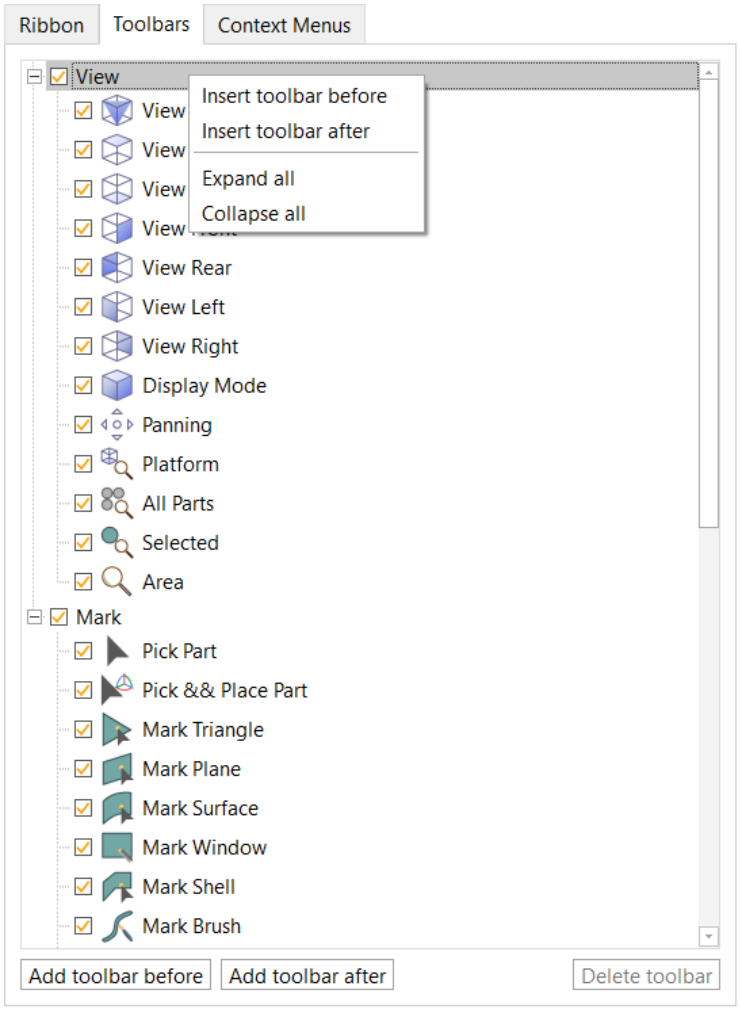
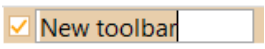


Figure 12.8 Toolbars setting

To create a custom toolbar, proceed as Table 12.4.

Table 12.4 Create a custom toolbar

1.	After selecting the workspace where the toolbar needs to be added, select a toolbar in the toolbar tab and then click Add toolbar before or Add toolbar after to add a new toolbar.
2.	Enter a name for the new toolbar. 

3.	Add commands by selecting the command in the left side and dragging it to the new toolbar. By adding a separator, you can make different groups within your toolbar.
4.	Click “Save” to finish the creation. The new toolbar is visible in the selected interface mode, and dragging the toolbar can change its position.

3) Context Menus：

In Voxeldance Additive， the user can customize the context menus as required. Currently, there are default context menu and custom context menu. Users cannot delete or rename the default menu, but can delete or add a command in the default menu. The custom context menus are created, named, and defined by the user and can be deleted.

Take the main interface mode as an example, as shown in Figure 12.9, the corresponding commands in the context menu tab can be operated by clicking the function button below, or by the right-click menus of the context menu or a command.

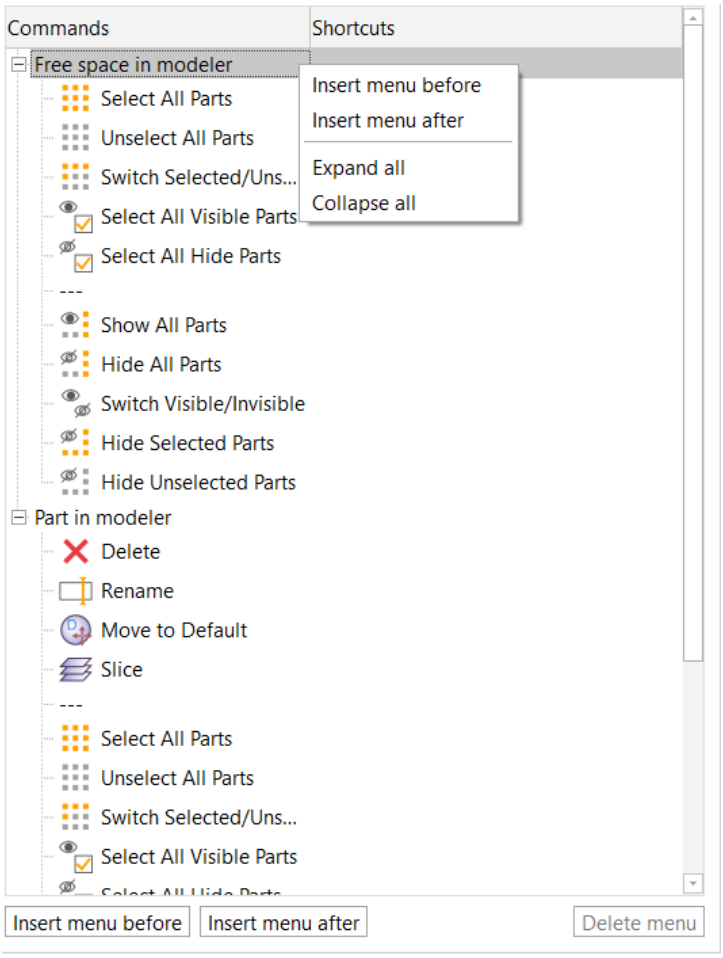


Figure 12.9 Context menus setting

To create a custom context menu, proceed as Table 12.5.

Table 12.5 Create a custom context menu

1.	After selecting the workspace where the context menu needs to be added, select a context menu in the context menu tab and then click Insert menu before or Insert menu after to add a new context menu.
2.	Enter the name and the specified shortcut key for the new context menu. New Menu Assign shortcut

3.	Add commands by selecting the command in the left side and dragging it to the new context menu. You can add menu groups, separators in the context menu commands.
4.	Click “Save” to finish creating the creation. The method of the new context menu is visible in the selected interface mode: press the specified shortcut key with clicking on the right mouse button, then the pop-up menu is the new context menu.

12.2 Platform Setting

In the Home menu, click the “Create Platform - Create Platform” command to create a new default platform. Click the “Create Platform - Platform Definition” command to open the platform settings dialog, as shown in Figure 12.10.

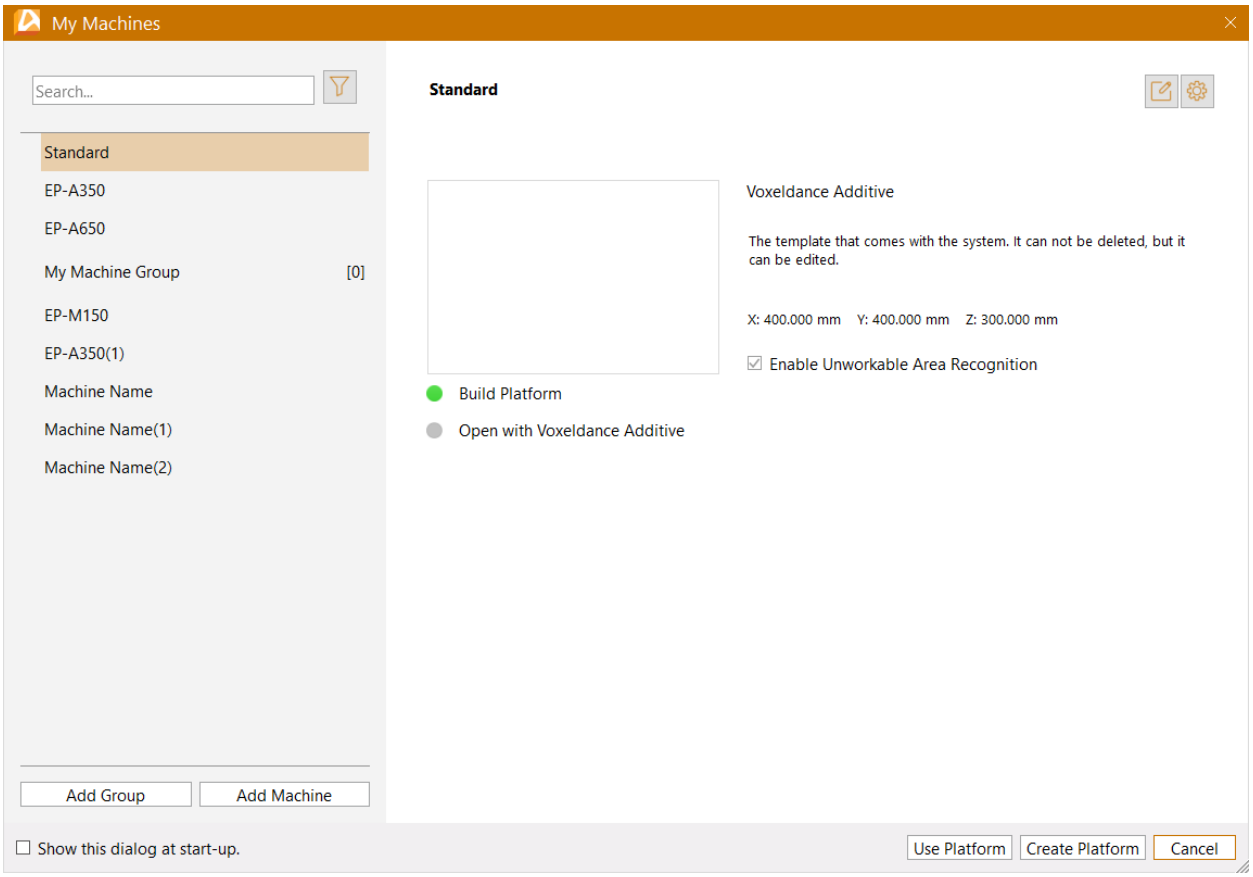


Figure 12.10 Platform setting

1) In the "My Machines" dialog, click the "  " icon button, you can change the 3D dimensions of the platform by adjusting the X, Y, and Z values. With check "Enable Unworkable Area Recognition", you can set some areas on the platform where parts cannot be placed. Click "Edit Unworkable Area Parameters", a dialog box for setting the unworkable area will pop up, as shown in Figure 12.11.

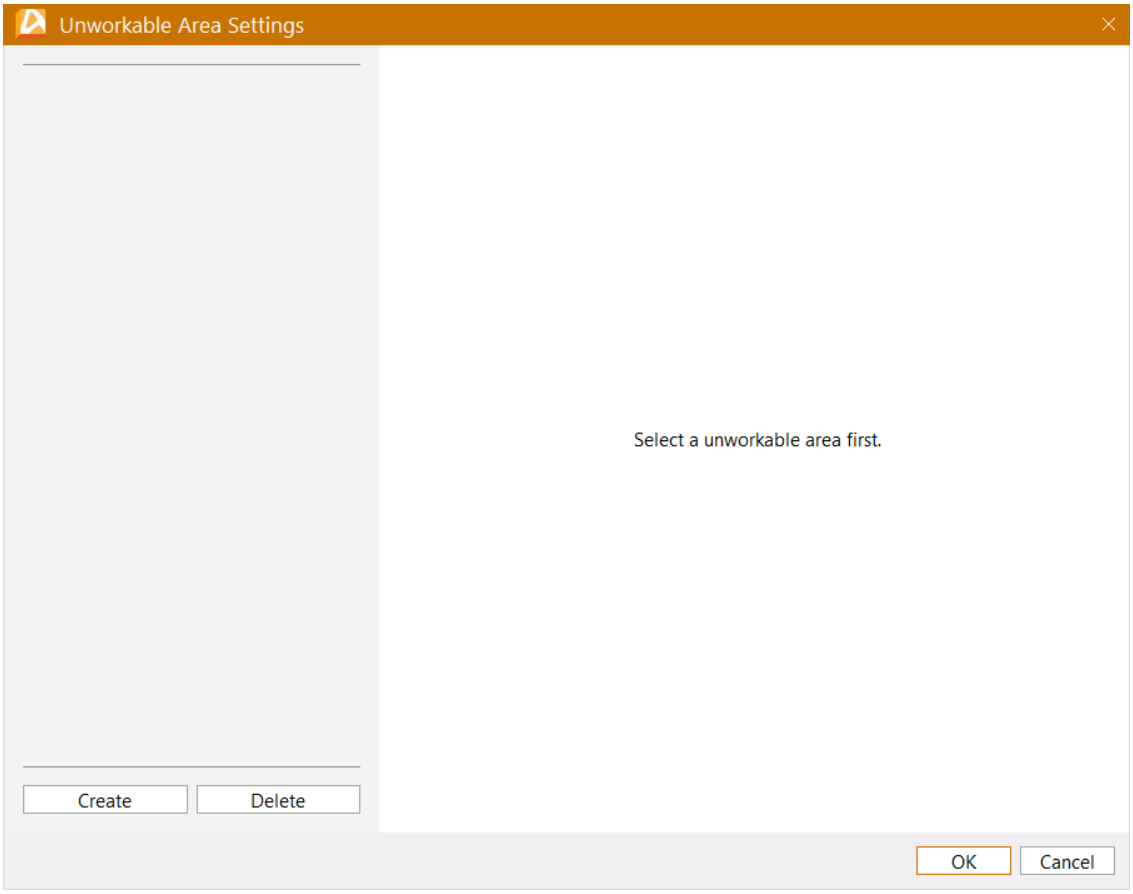


Figure 12.11 Unworkable area setting

Click the "Create" button, you can create and edit a circular or polygonal unworkable area, as shown in Figure 12.12. The unworkable areas of circle and polygon can be defined by setting their respective parameter values. Besides, if "Enable 3D unworkable area" is checked, the Z-axis height of the unworkable area is defined by setting the height Z. Clicking "OK" will save this unworkable area and list it on the left. Clicking the "Delete" button will delete the unworkable area selected on the left list.

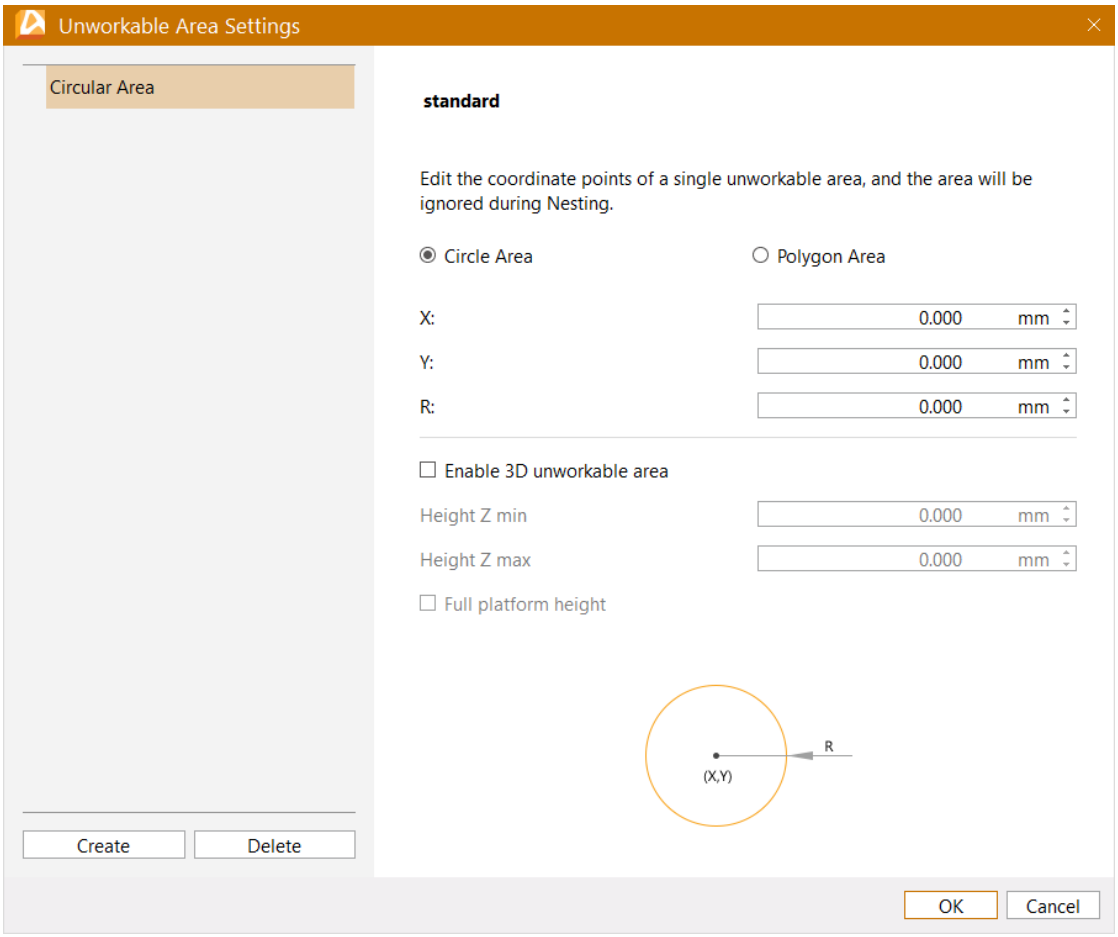


Figure 12.12 Edit unworkable area

The set unworkable areas are marked in red on the platform, and the effect is shown in Figure 12.13.

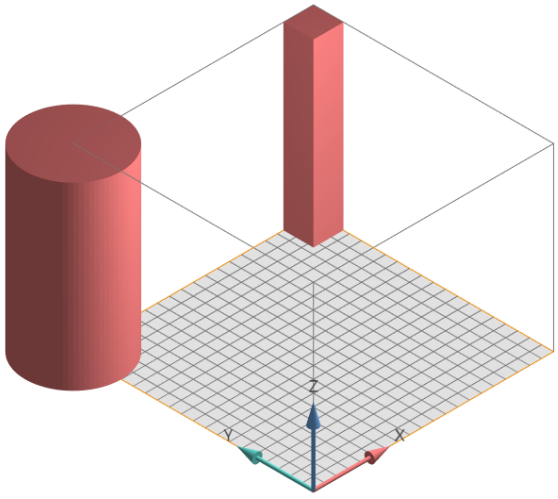


Figure 12.13 Platform with unworkable area

2) In the "My Device" dialog, click the " " icon button, and the setting page of the platform attributes will pop up, as shown in Figure 12.14.

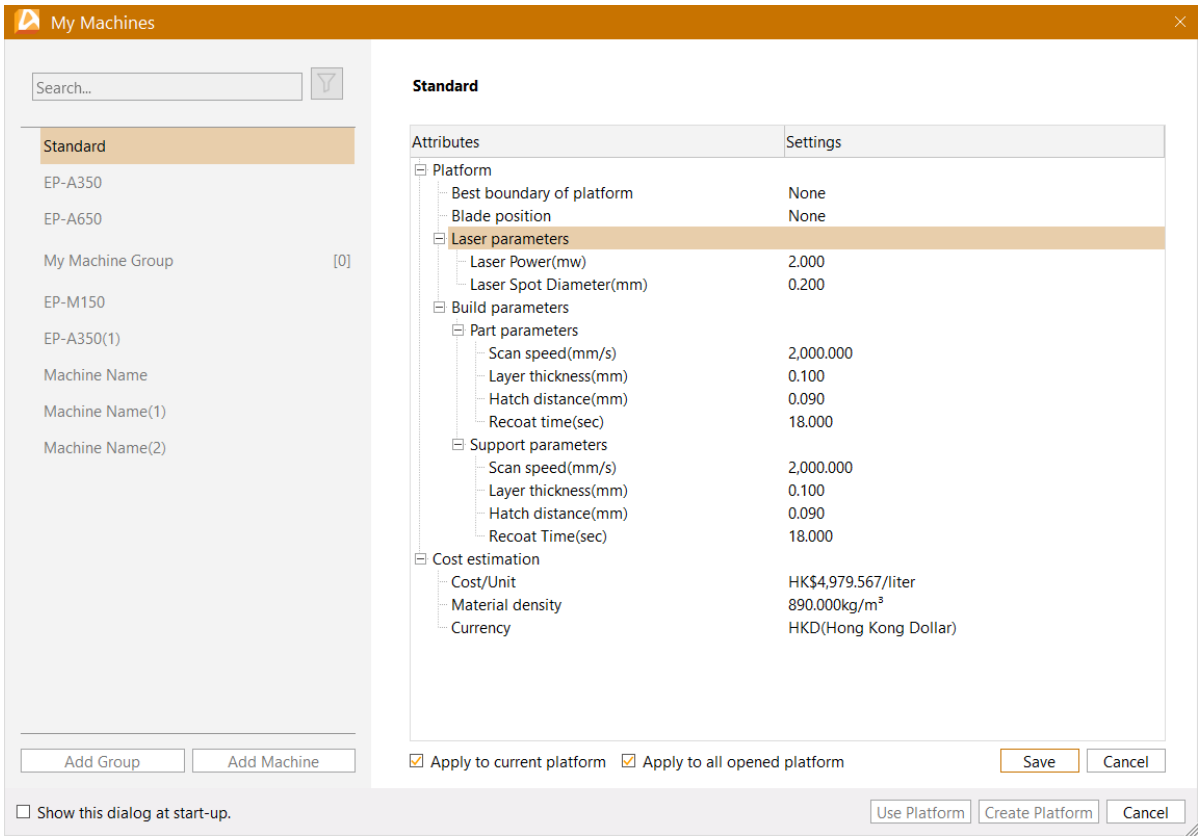


Figure 12.14 Platform attributes setting

The detailed parameter explanation is shown in Table 12.8.

Table 12.8 Platform parameters

Best Boundary of platform	**Best Boundary in 2D and Best Boundary in 3D**
Margin to X side	The distance between print area to border X
Margin to Y side	The distance between print area to border Y
Margin to Z side	The distance between print area to border Z (Visible when selecting “Best Boundary in 3D)
Blade position	Displayed blade direction position relative to the platform：

	None: Do not show blade direction Front side of the platform：Blade direction is displayed in front of the platform Back side of the platform：Blade direction is displayed behind the platform Left side of the platform：Blade direction is displayed on the left side of the platform Right side of the platform：Blade direction is displayed on the right side of the platform
Laser parameters	
Laser Power	Laser intensity (mW)
Laser Spot Diameter	The diameter of the laser spot
Build parameters	
Part parameters	
Scan speed	Laser scanning speed
Layer thickness	Slice thickness
Hatch distance	Spacing distance when filling
Recoat time	The time required to coat a new resin/powder layer on an already made structure
Support parameters	
Scan speed	Laser scanning speed
Layer thickness	Slice thickness
Hatch distance	Spacing distance when filling
Recoat time	The time required to coat a new resin/powder layer on an already made structure
Cost estimation	

Cost/Unit	Set the cost unit price of materials used (/liter, /kg, /g)
Material density	Set the density of the material used (g/cm^3、kg/m^3)
Currency	Select the currency type that corresponds to the cost unit price

3) In the "My Machines" dialog, click the "Add Group" button, you can create a group to place the added devices. Click the "Add Device" button, there will be some machine libraries provided by the manufacturer, as shown in Figure 12.15, you can select the required machine. You can also enter the machine name in the "Search" box to find it, or click the "  " icon to filter by manufacturer and technology. Click "Create Custom Parameters", you can set the shape of the platform (rectangle or circle), as well as the size and unworkable area.

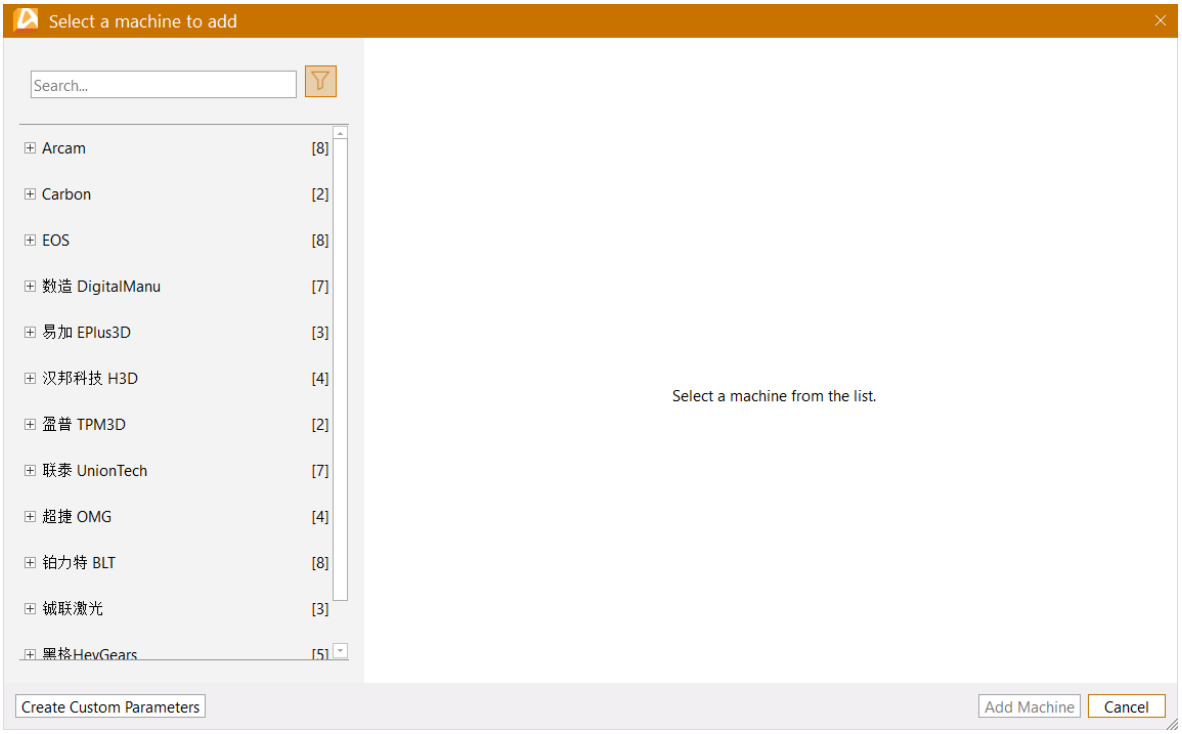


Figure 12.15 Select and define machine

4) In the "My Device" dialog, right-click the machine item, the menus of "Set Default", "Edit" and "Duplicate" will appear, as shown in Figure 12.16.

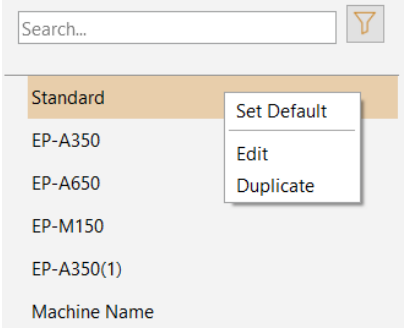


Figure 12.16 The context menu of machine

If you select the "Set Default" item, this platform will be used as the default platform when creating a new platform. If you select the "Edit" item, you will go to the page for setting the platform size and unworkable area. If you select the "Duplicate" item, this platform will be copied and added to the current list.