

# DeskArtes

## Expert Series 12.1

Generating Supports for AM with  
3Data Expert

## Expert Series 12.1: Generating Supports for AM

---

### ***First Edition***

Ismo Mäkelä: "Expert Series 12.1: Generating Supports for AM".

June 2020.

### ***Copyright***

© 2020 DESKARTES. All rights reserved.

DESKARTES reserves the right to revise this publication and to make changes from time to time without the obligation to notify any person of such revisions and changes.

### ***Trade Marks***

The DESKARTES name and the logo are trademarks of DESKARTES Oy. Other brand and product names are trademarks and registered trademarks of their respective owners.

### ***Contact Address***

DESKARTES OY

Asentajankatu 5

00880 Helsinki

FINLAND

<http://www.deskartes.com>

---

## Table of Contents

FOREWORD.....	4
LESSON 1 – LEARNING THE GRAPHICAL USER INTERFACE.....	7
<i>Generating supports automatically</i> .....	7
<i>Examining the first supports</i> .....	12
<i>Optimizing parameters for a selected support area</i> .....	15
LESSON 2 – EDITING SUPPORT GEOMETRIES MANUALLY.....	26
<i>Adding supports manually</i> .....	26
<i>Deleting supports manually</i> .....	35
<i>Applying Arch command to support geometry</i> .....	40
<i>Adding Side support</i> .....	44
<i>Adding Tree support</i> .....	48
LESSON 3 - LEARNING SUPPORT PARAMETERS.....	57
<i>Opening and saving support parameter files</i> .....	57
<i>Editing Common support parameters</i> .....	59
<i>Editing Web support parameters</i> .....	65
<i>Editing Chain support parameters</i> .....	71
<i>Editing Point support parameters</i> .....	77
<i>Generating Tube supports</i> .....	81
<i>Generic Support improvements with 12.1 version</i> .....	85
LESSON 4 - OUTPUTTING AND SLICING SUPPORTS FOR AM.....	88
<i>Saving supports STL file</i> .....	88
<i>Saving support and part slices</i> .....	88
<i>Slicing supports and parts into one file</i> .....	95
<i>Previewing Slice data</i> .....	98
<i>Fixing slice data</i> .....	101
LESSON 5 – ADDITIONAL TOOLS FOR SUPPORT GENERATION.....	104
<i>Center of Gravity calculation</i> .....	104
LESSON 6 – CONNECT TOOL.....	105
<i>Connect Tool command</i> .....	105
LESSON 7 – SAND SUPPORTS.....	110
<i>Sand Support generation</i> .....	110

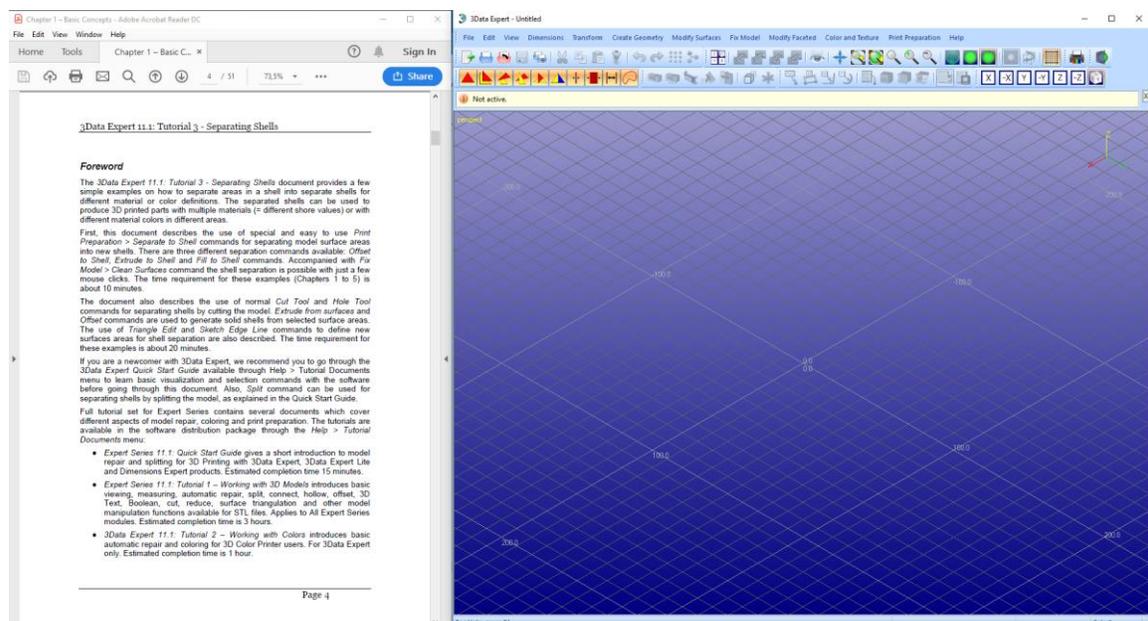
## Foreword

The *Expert Series 12.1: Generating Supports for AM* document provides a tutorial to support generation module in 3Data Expert 12.1 and later versions. The new *Generic Support* command enable the generation of supports for various AM processes, like metal, stereolithography and DLP systems.

Additionally, the last Lesson shows how to generate supports for sand based systems.

There is a wide variety of support geometries you can generate with the 3Data Expert support generation commands. Supports can be either open or solid (e.g. with volume). Solid supports can be generated for heat dissipation and warpage prevention with metal based AM. Open and delicate support structures can be generated for DLP or stereolithography parts. Web, Chain and Point supports can be used to support different areas in the part, like large down-facing surfaces, sharp edges and local minimum points in the part. Supports can be even converted to small tubes for special medical model manufacturing, like teeth implants. Side supports and Arch supports can be used to prevent part movement during the build or to avoid hitting part surface area below the areas needing supports. Tree supports can be generated for parts with special geometries, like rings and other jewelry products in metal. Supports can be generated both fully automatically as well as optimized with manual commands.

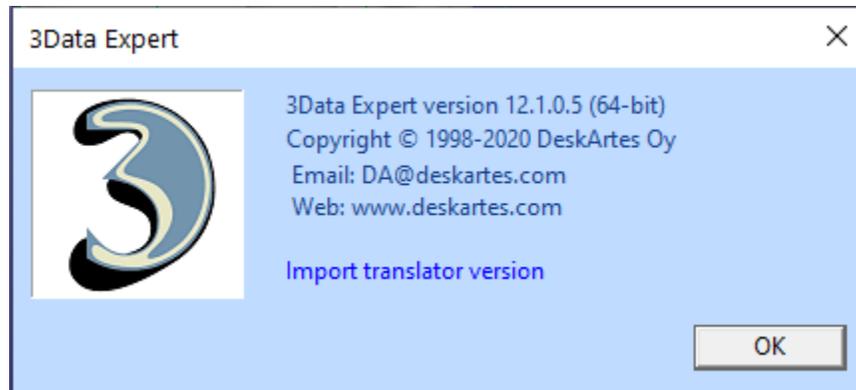
To get the most of the tutorial lessons you should go through the examples with 3Data Expert 12.1. If no hard copy of the tutorial is available, we recommend you to open 3Data Expert and tutorial document side by side on the computer screen:



## Expert Series 12.1: Generating Supports for AM

---

You will need a licensed version of 3Data Expert for the lessons. The functionality described in this tutorial can be tested with DeskArtes 3Data Expert software release version 12.1.0.5 or later:



You can also use DeskArtes 3Data Expert to generate supports for sand process parts. More on Sand Support command can be found in the application Help and Tutorial 1 – Working with 3D models.

Full tutorial set for Expert Series contains several documents which cover different aspects of model repair, coloring and print preparation. The tutorials are available in the software distribution package through the *Help > Tutorial Documents* menu:

- *Expert Series 12.1: Quick Start Guide* gives a short introduction to model repair and splitting for 3D Printing with 3Data Expert, 3Data Expert Lite and Dimensions Expert products. Estimated completion time 15 minutes.
- *Expert Series 12.1: Tutorial 1 – Working with 3D Models* introduces basic viewing, measuring, automatic repair, split, connect, hollow, offset, 3D Text, Boolean, cut, reduce, 3D nesting, surface triangulation and other model manipulation functions available for STL files. Applies to All Expert Series modules. Estimated completion time is 3 hours.
- *3Data Expert 12.1: Tutorial 2 – Working with Colors* introduces basic automatic repair and coloring for 3D Color Printer users. For 3Data Expert, 3Data Expert Lite and 3DEdit Pro. Estimated completion time is 1 hour.
- *3Data Expert 12.1: Tutorial 3 – Separating Shells* describes the use of Shell Separation commands in 3Data Expert. These commands enable separating a part into several shells for different material (shore value) or color definitions in a single part. For 3Data Expert only. Estimated completion time is 30 minutes.
- *3Data Expert 12.1: Tutorial 4 – Generating Supports for AM* explains the use of new Generic Supports command and Sand Supports command. The support generator can be used to generate supports for DLP, metal,

## Expert Series 12.1: Generating Supports for AM

---

stereolithography and sand based processes. Adding extra connectors to prevent parts from warping and bending during the build is explained. Also, new slicing, slice preview and slice filling is demonstrated. Estimated completion time 1 hour.

- *Expert Series 12.1: STL and VRML repair examples* document includes more advanced model repair operations with 3Data Expert. Applies to 3Data Expert module and mostly to 3Data Expert Lite, Dimensions Expert and Sim Expert modules. Estimated completion time with 3Data Expert 2 hours.

The tutorial documents and the geometry files for the tutorial examples are found in the *Tutorials* folder and its sub-folders (*GeomFiles*, *Tutorial 1*, *Tutorial 2*, etc.) in the 3Data Expert (or product in question) installation directory.

### **Lesson 1 – Learning the Graphical User Interface**

The Generic Support command is operated through a set of toolbars. The toolbars allow both generating the supports automatically based on predefined parameter files as well as manual editing of supports for improved surface quality or increased robustness of the build.

#### **Generating supports automatically**

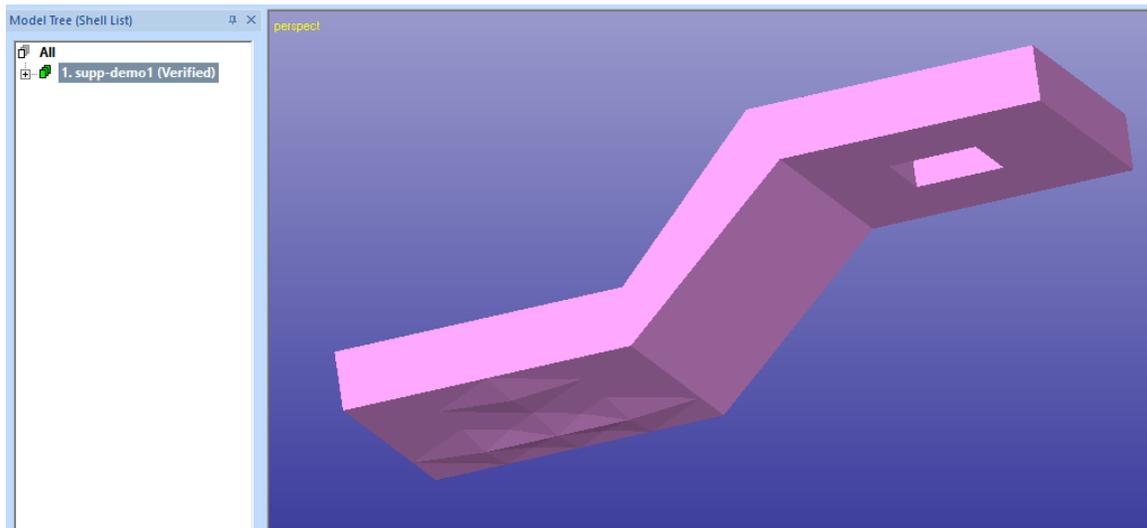
With this example we will just generate the supports using the automatic support generation process: We select the predefined support parameter file and generate the supports in “black box” mode.

The support generation is started by inputting a file needing supports into the Model Tree. First we will

- **open a test file *supp-demo1.3de*** into 3Data Expert with *File Open* command. The file is stored in the *tutorials/GeomFiles* directory under the 3Data Expert installation in Program Files directory.

↑  > This PC > Windows (C:) > Program Files > DeskArtes 3Data Expert 11.1 > tutorials > GeomFiles

- **accept** the proposed units by **pressing OK**
- **press Print Preparation** in the Operation Mode window and you will see the model stored in the Model Tree



The example file contains a model with hole, surfaces with different surface angle as well as some local minimums. The support generation will create sufficient support at the flat areas, around area edges as well as below the local minimum points.

Let's now start generating supports to the example model:

## Expert Series 12.1: Generating Supports for AM

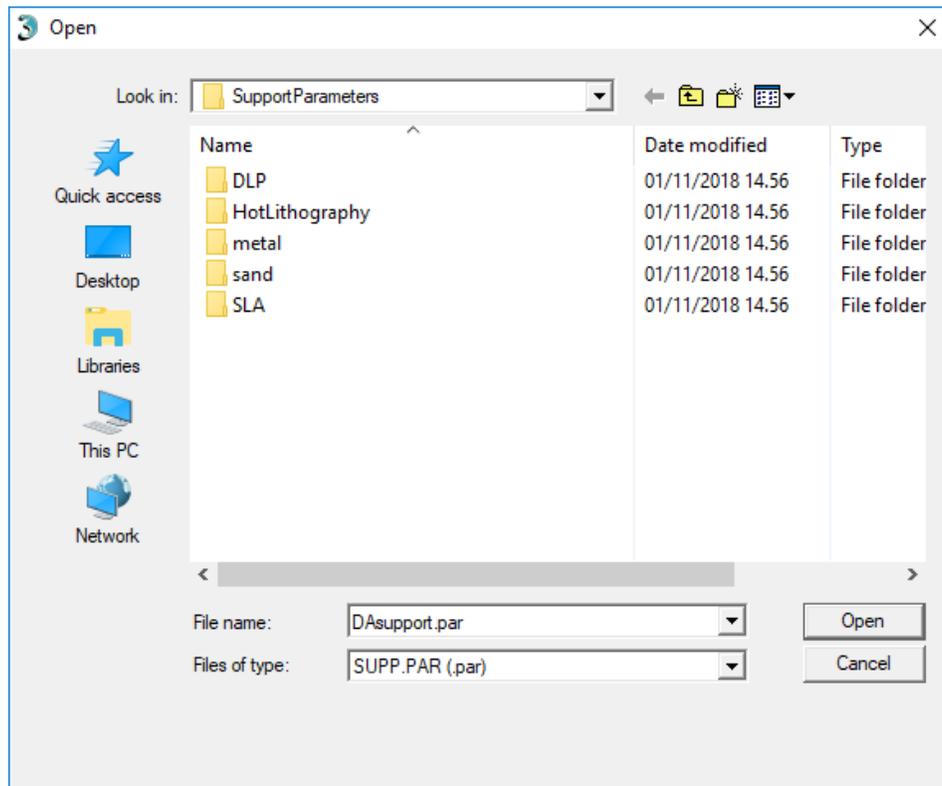
---

- **select the model** in the Model Tree
- **Give the command *Print Preparation > Supports > Generic Support*** or press Supports button in the Print Preparation Tab



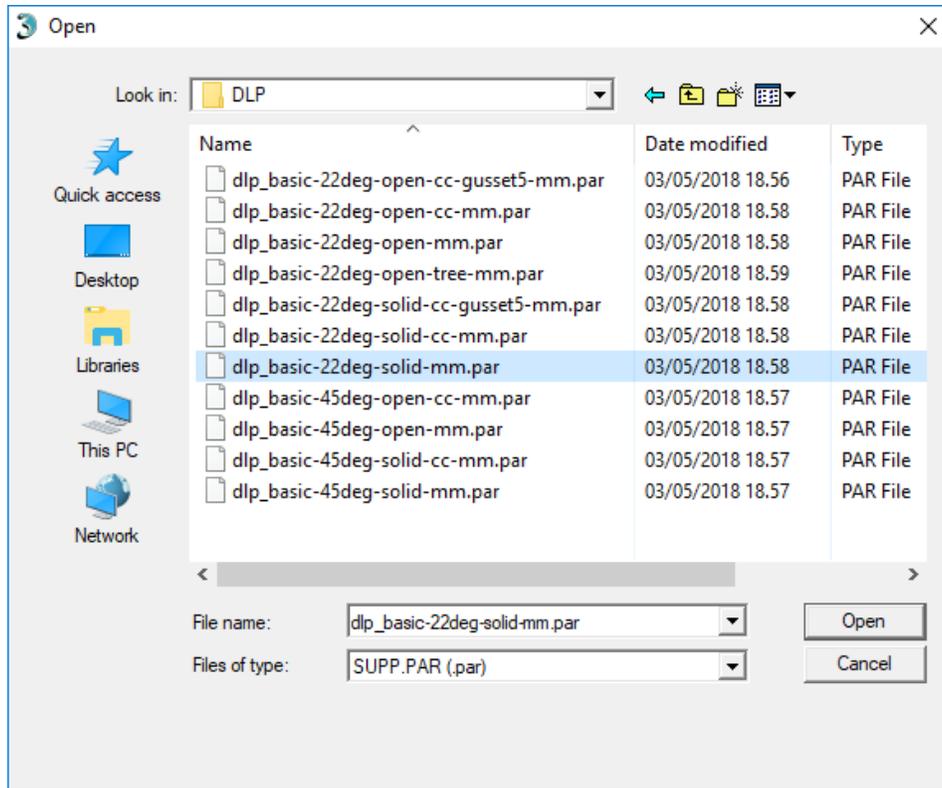
- a parameter dialog with different parameter directories appears

The system comes with a set of predefined support parameter directories for different AM processes.

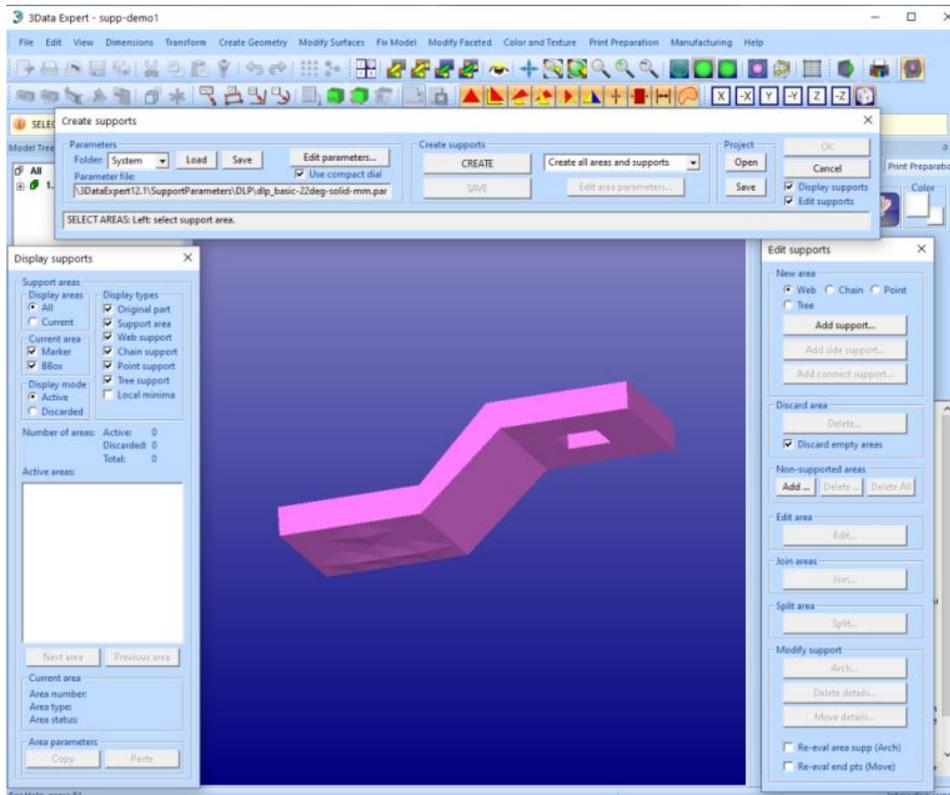


- **open the directory** for intended AM process, in this example we will use ***DLP*** directory
- **select the suitable parameter file** for your system, in this case we will select the ***dlp\_basic-22deg-solid-mm.par*** and press ***Open***

## Expert Series 12.1: Generating Supports for AM



The Graphical user interface toolbars for support generation appear.



## Expert Series 12.1: Generating Supports for AM

We recommend you to position the different support generator toolbars as given in the image above. This will leave the Viewing Toolbar and especially the Viewing Planes Toolbar (X, Y, Z views) visible.

There are three toolbars: *Create supports*, *Display supports* and *Edit supports*. The functions are divided between the different toolbars according to their names.

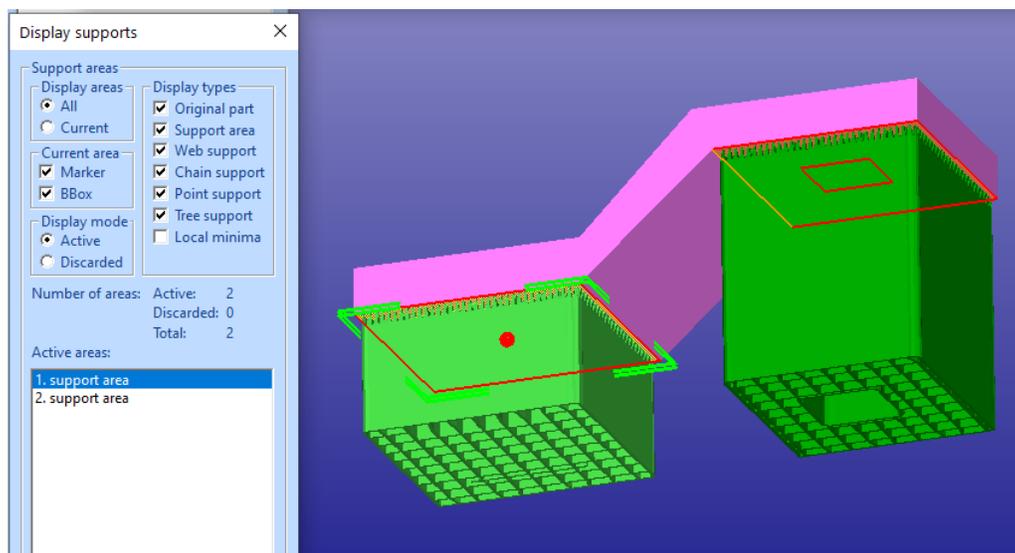
Anyhow, this time we will just generate the supports using the automatic support generation. The supports are generated according to the parameters given in the *dlp\_basic-22deg-solid.par* file. Then

- **Press Create button** in the Create supports toolbar to create all areas and supports for the plate model



First the command checks that all parameters are consistent with each other. **Press Ok and Yes** to the appearing two report dialogs, if appear, to continue the support generation. If necessary, the support parameters are adjusted by the system to ensure concise STL structure generation.

After a while the supports are generate and appear in the graphics area.



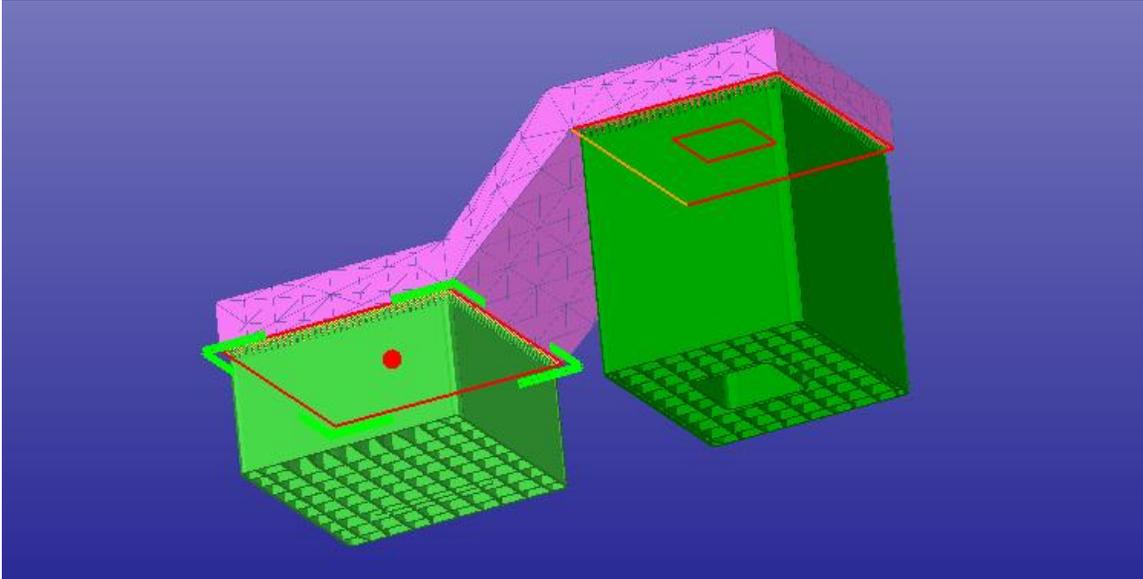
You can investigate the support visually with the *middle mouse rotation*, *middle mouse zoom* and other graphical viewing commands. The red edge line around

## Expert Series 12.1: Generating Supports for AM

---

a supported area means that this edge is not supported by the part itself, orange edges are supported by the part. Depending on the supported/not supported edge the support itself can be further or closer from the given edge.

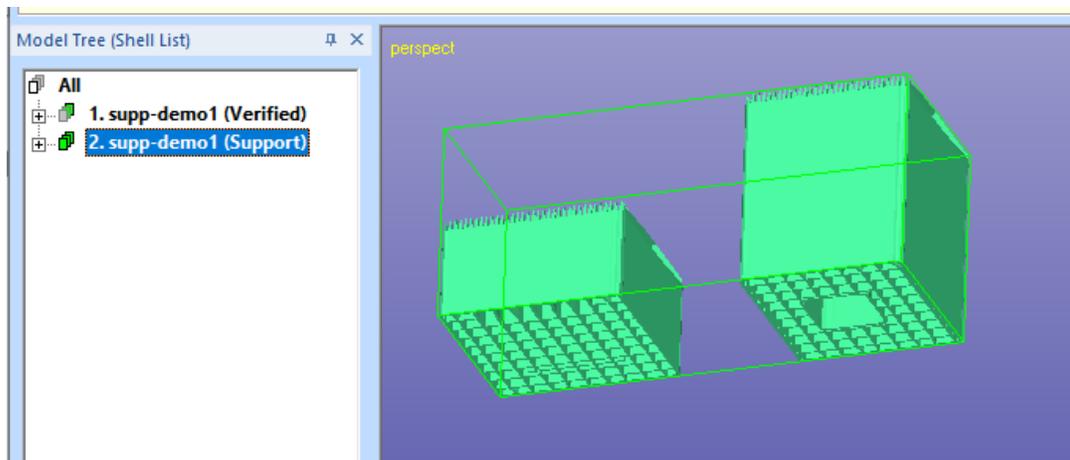
The support area with lighter green color and marked with red point is currently selected in the support area list (1. support area).



Finally, we store the generated supports in the Model Tree.

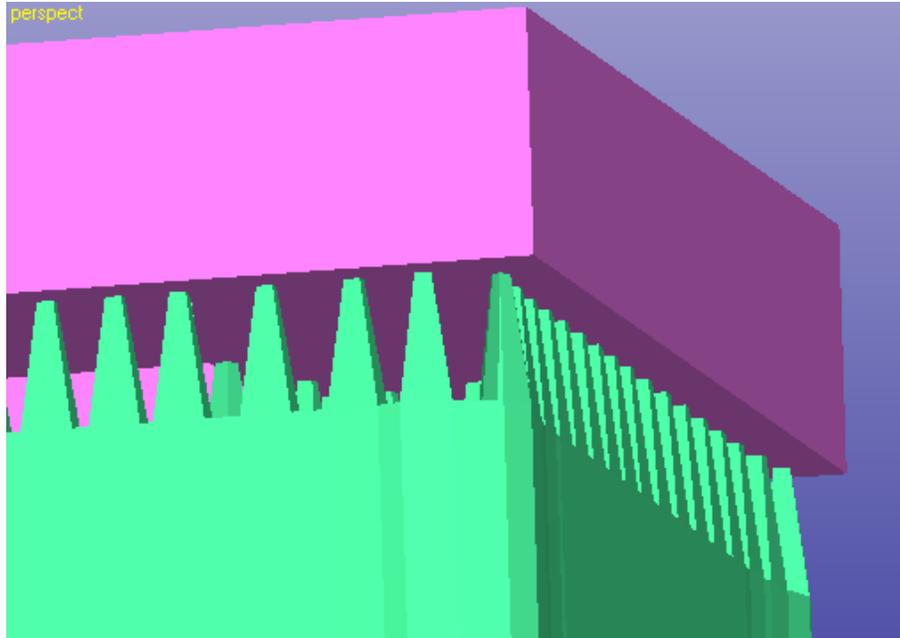
- **Press OK button in the Create supports toolbar** to store the supports into the Model Tree (in green circle below)

After a while the supports are verified and stored in the Model Tree:

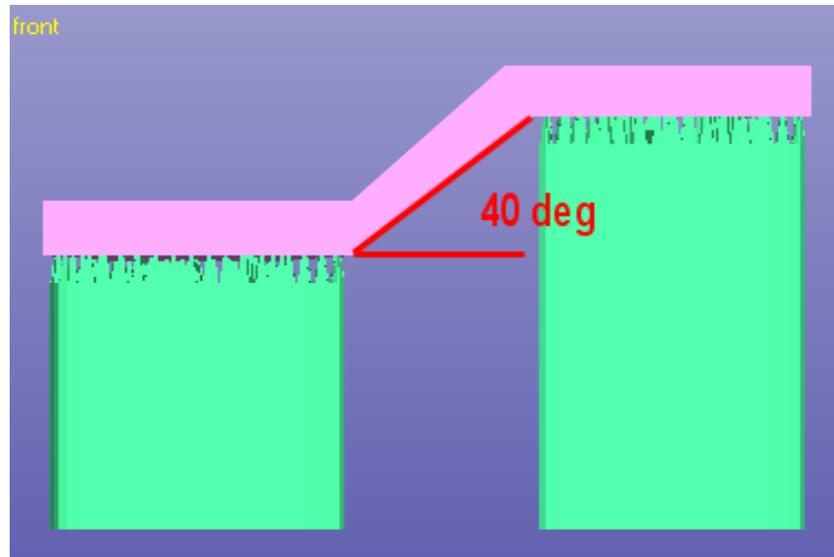


### Examining the first supports

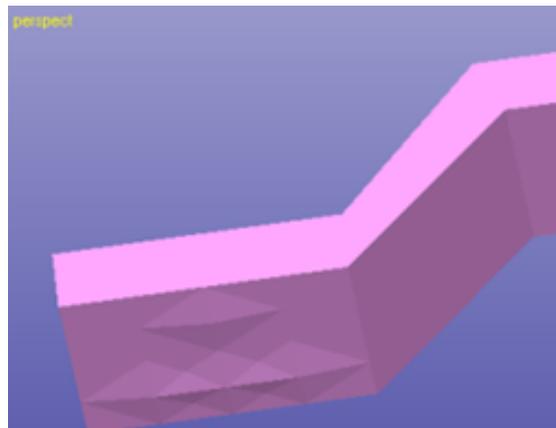
The generated supports have several properties which are important for correct build of a part with an AM system. The selected parameter set produced supports with the following properties:



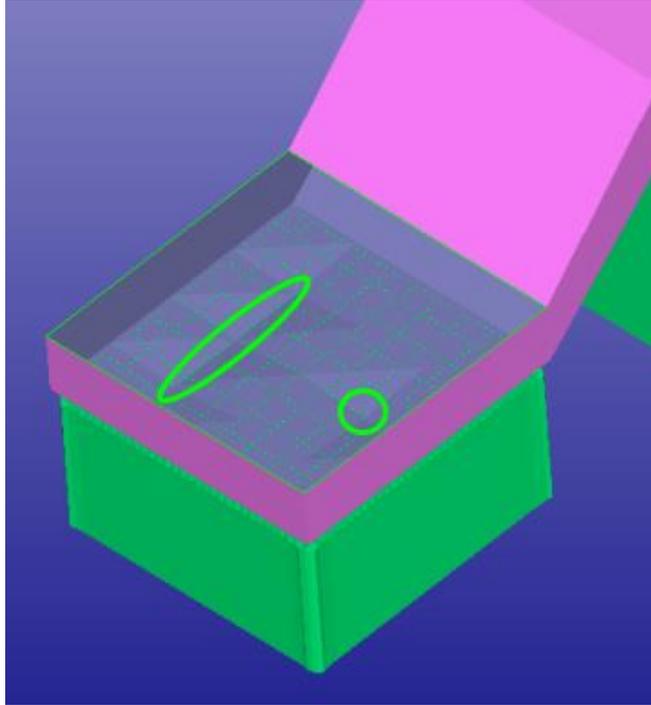
- The supports are solid. This allows to use supports for any AM system (DLP, metal, stereolithography) by providing enough strength and/or heat dissipation from the part with metal processes. Open (zero thickness) support structures can also be generated by simply modifying a parameter via the parameter editor (to be shown later).
- The supports are connected to the part via teeth. This makes it easier to remove the supports from the part after manufacturing. The user has full control to the teeth size via the parameter editor. Additionally, dovetail shape can be generated on the teeth for controlling the breaking point for the teeth accurately.
- The supports follow the profiles of the supported areas. This prevents delamination at the support area edges with minimum amount of supports
- The supports are only generated to areas requiring supports. With this example, *dlp\_basic-22deg-solid-mm* parameters, we only generate supports to surfaces with smaller than 22 degree angle to the vertical plane. The diagonal area in the middle has 40 degree angle with the horizontal plane, thus not needing supports, please see below.



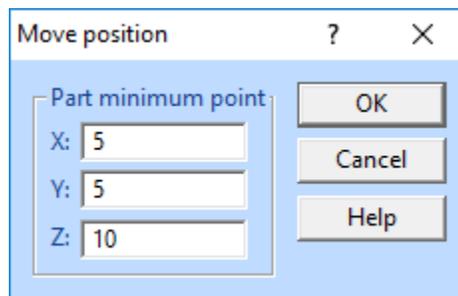
- The supports teeth are generated under the local minima. In the original parts some areas of the down facing surfaces were slightly lower than the surrounding down facing area:



In the final result we can see that teeth tips are located exactly below the local minimums (in green circles below, top surface Hidden). It is essential to support the part from the lowest points to guarantee correct manufacturing of the part.



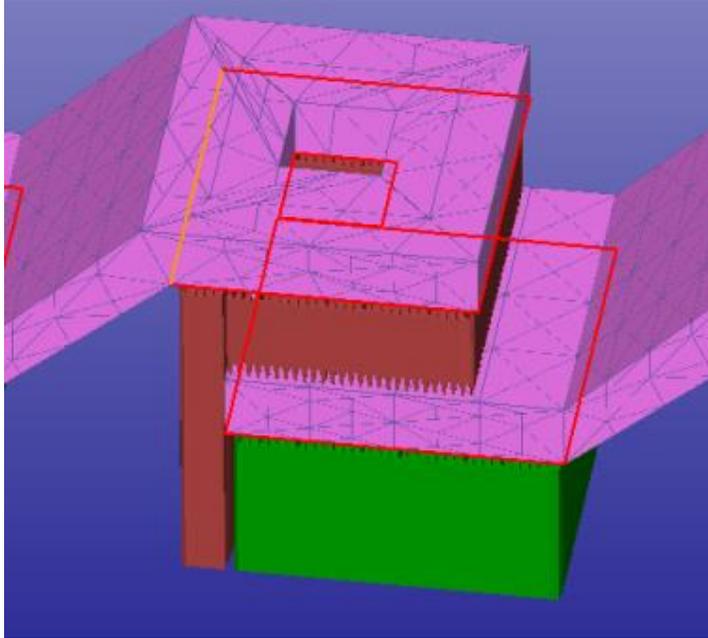
- The supports are generated from the part to the vat bottom  $Z = 0$  level. You should always leave sufficient room between the part and the vat bottom for the supports. Also, the part must be in the positive space when supports are generated. If necessary, you can use *Print Preparation > Move to Platform > Move positive* command to position the part correctly before support generation:



The supports for *supp-demo1* can now be outputted for AM or sliced and then send directly for AM. These steps are described in the last *Chapter 6. Outputting and Slicing Supports for AM.*

Additionally, supports

- will separate from vertical areas with user given tolerance. This will ensure better surface quality
- teeth can also be generated on the bottom of the support when an overhang area needs to be supported from the part surface below the overhang area. This will also improve surface quality and make support removal easier.



In the example above the support area hitting the part below is show with red color while in support generation mode. Sometimes you can try to avoid hitting the part below by using the *Edit supports* dialog *Arch* command. With *Arch* command you can manually bend the supports to avoid hitting the part below.

### Optimizing parameters for a selected support area

The normal process for support generation is described above where the user selects the parameter file and then automatically generates the supports. Anyhow, there may be situations where the automatic support generation does not generate optimal result. Some areas may require removal of extra support, supports for some area may require different solid thickness, another area may need to be split into separate areas and some model edges may require extra support. This and more can be achieved by using the interactive tools available with Generic support command toolbars.

In the next example we will change the parameters for a specific area and then regenerate the area supports only. Once again,

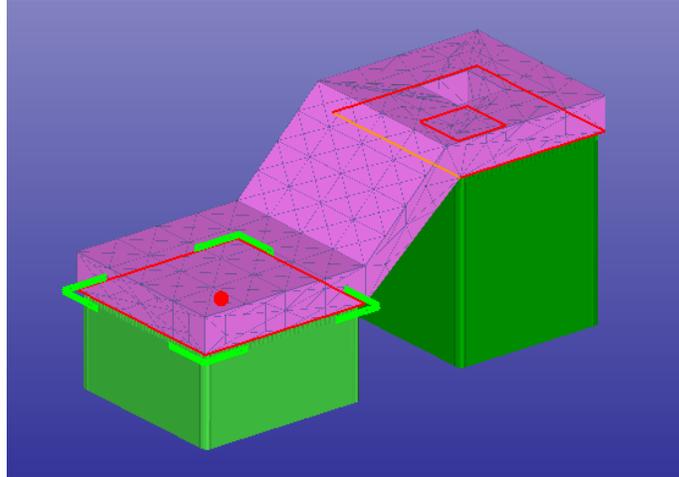
- **open the *sup-demo1.3de* model** and

## Expert Series 12.1: Generating Supports for AM

---

- give **Generic Support** command with **dlp\_basic-22deg-solid-mm.par** file.
- press **Create** button in **Create supports** group

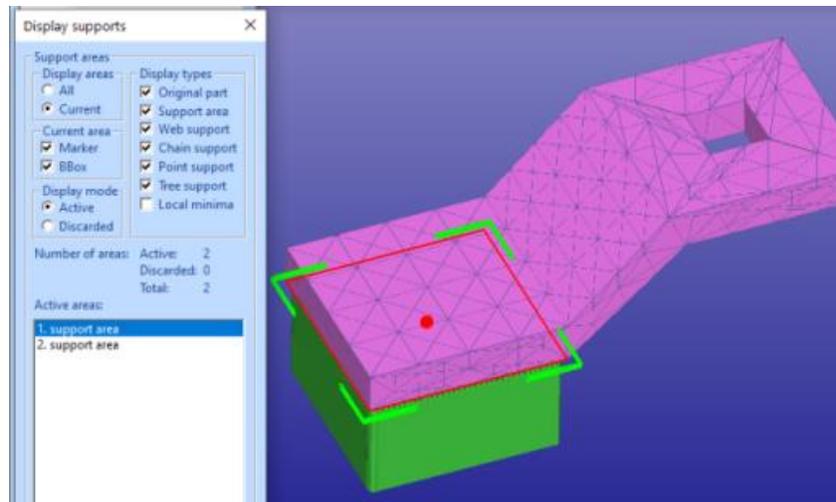
The supports are now visible in the graphics area.



We will now take a quick look at the different display operations for the *Support areas*. The *Display areas* group will allow you to show either *All* areas or the *Current* area. All areas is the default value and by selecting the *Current* area you will see only the area which is selected in the support area list. Below we have

- **selected the *Current* area** display
- then select **the *first are* in the area list.**

Only the first area is now displayed on the graphics area:



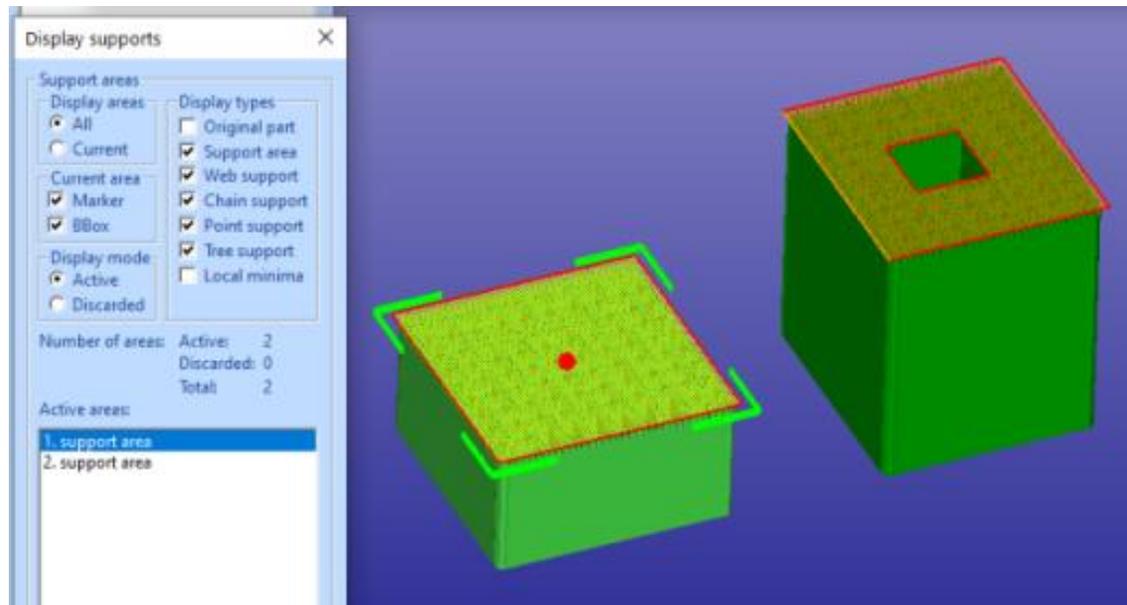
- **select the *All areas* again**

The *Current area* group lets you select if the current area is shown with red point marker and/or bounding box corners.

The *Display mode* group allows to show either Active areas or Discarded areas. Discarded areas are support areas which do not have any support or which have been deleted by the user.

The next group, *Display types*, allows you to set on display of different objects in the support generation. For example, you may only want to see the supported areas and supports, then

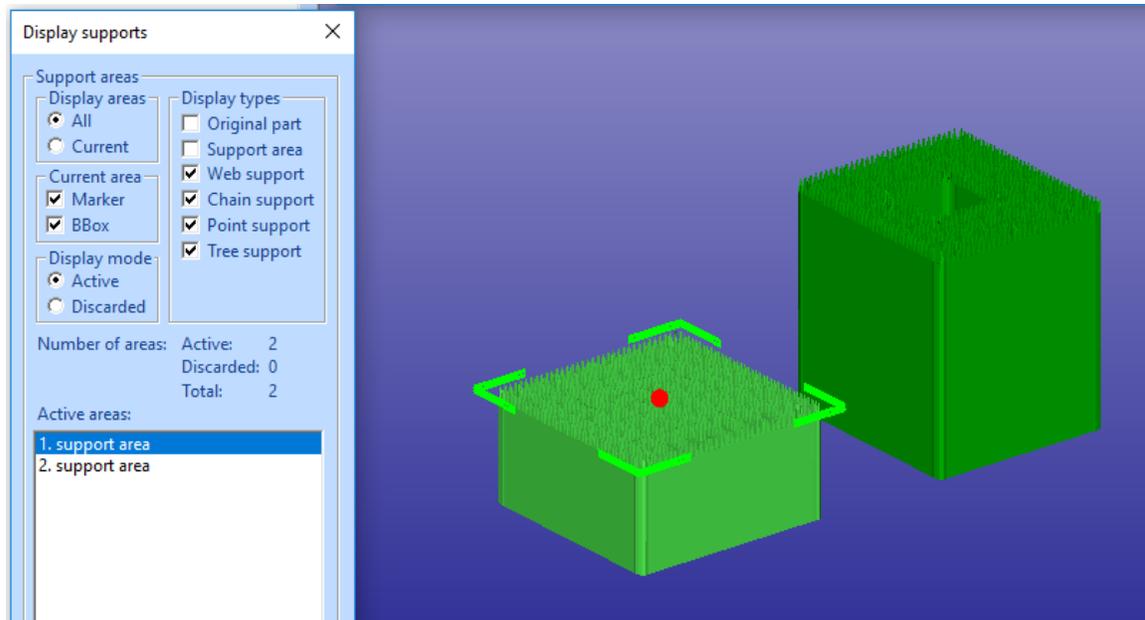
- **click Off the *Original part* display type selection:**



You will see the support area triangles with orange hue and the area edges with red edge polygon.

Or you may only want to see the supports, then

- **tick Off the *Support area* selection:**

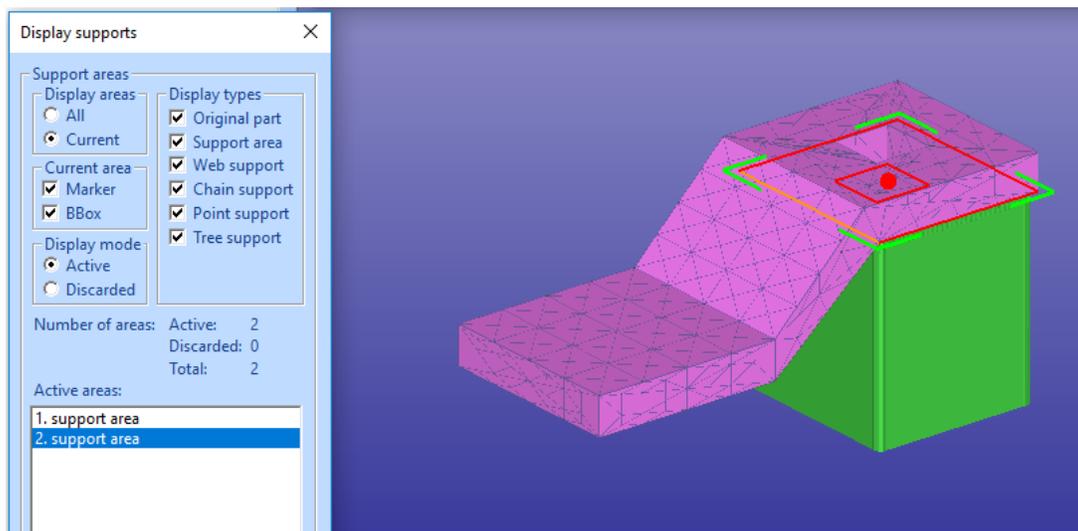


Furthermore, you can only select to show specific support types, like *Web*, *Chain* or *Point* supports. We do not have an example of the *Chain* and *Point* types available now, but different visualization types will allow you to effectively investigate, edit and select areas for optimal support generation.

Now, let's select the second area from the support area list and change the parameters for that area.

- Turn **On** all *Display types* and
- select the *Display area* selection **Current**. Then
- select the **2. support area** in the list with the **LMB** (left mouse button):

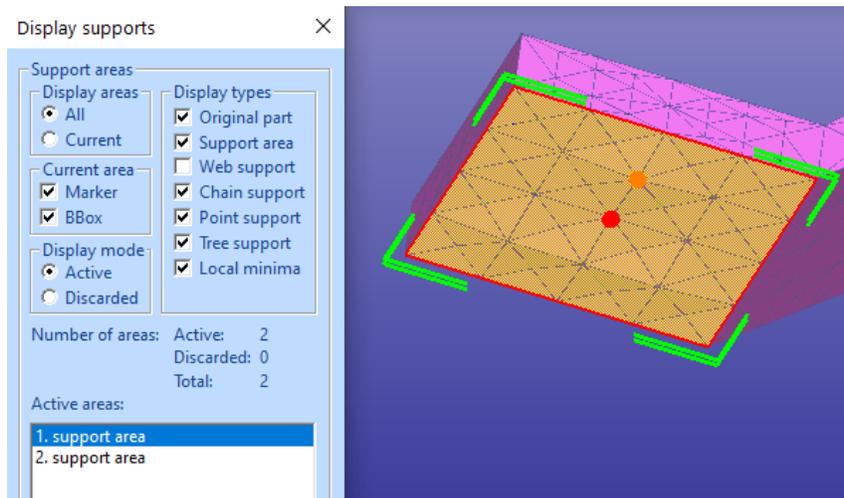
We will now see the part and the supports for the second area in the graphics area.



## Expert Series 12.1: Generating Supports for AM

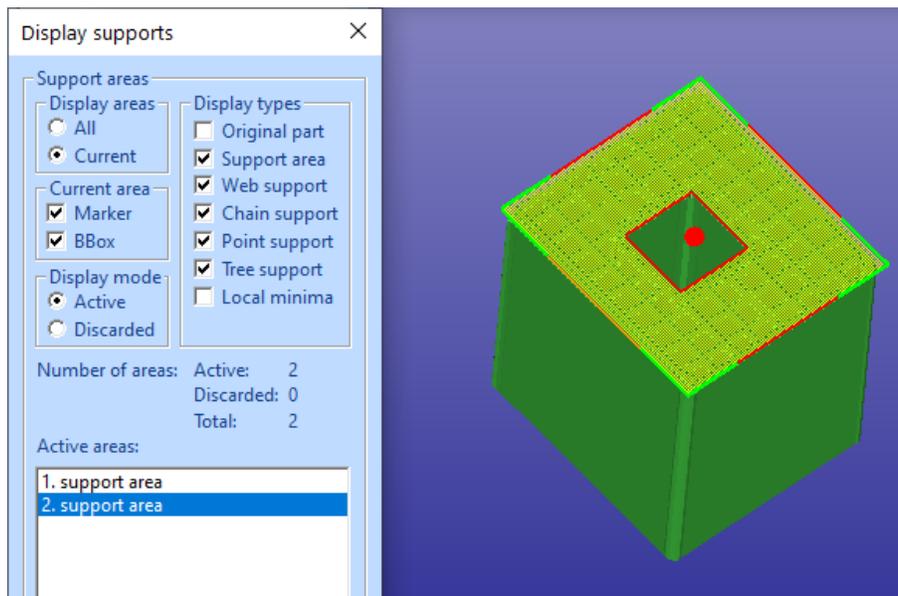
---

Finally, the last *Display type* selection **Local minima** shows the local minimum points with orange dot on the model. This is helpful if supports are added manually to ensure all minimum points are supported correctly:



Let's now have a closer look at the supports in the second area.

- **set Off the display of the *Original part***
- **zoom into** the second support area in the graphics area



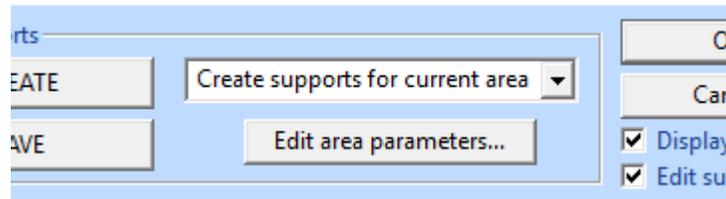
We now want to change the density of the web with the second area. To start the area parameter editing we first must select the correct Create supports type in the Create supports toolbar:

- **Open the *Create supports* dropdown list and**
- **select the *Create supports for current area* selection:**

## Expert Series 12.1: Generating Supports for AM



The *Edit area parameters* button now becomes active, as seen below:



- **Press the *Edit area parameters* button** to open the Support Parameters parameter editor dialog.

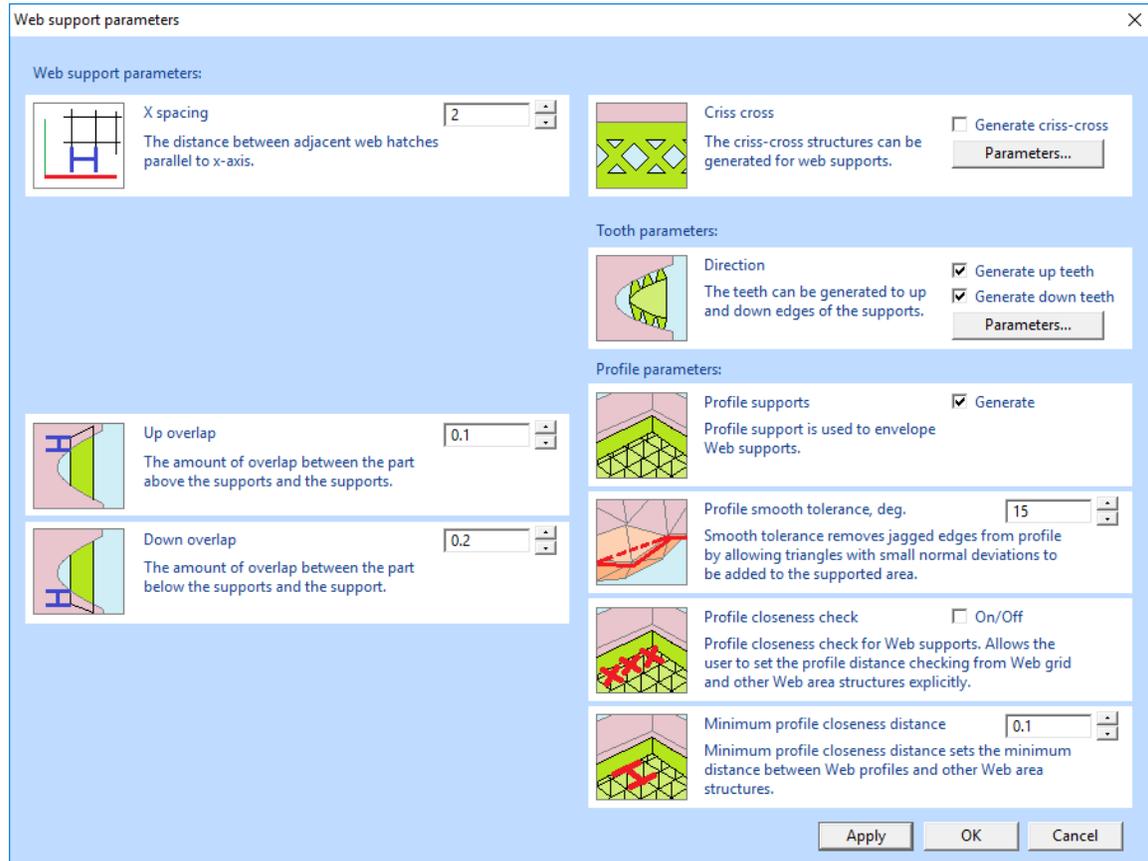


## Expert Series 12.1: Generating Supports for AM

The *Support parameters* dialog includes several options to edit both common parameters for all supports or to edit parameters for specific support types. This time we will just edit one parameter for Web support types.

- Press the **Parameters...** button in the *Web supports* group.

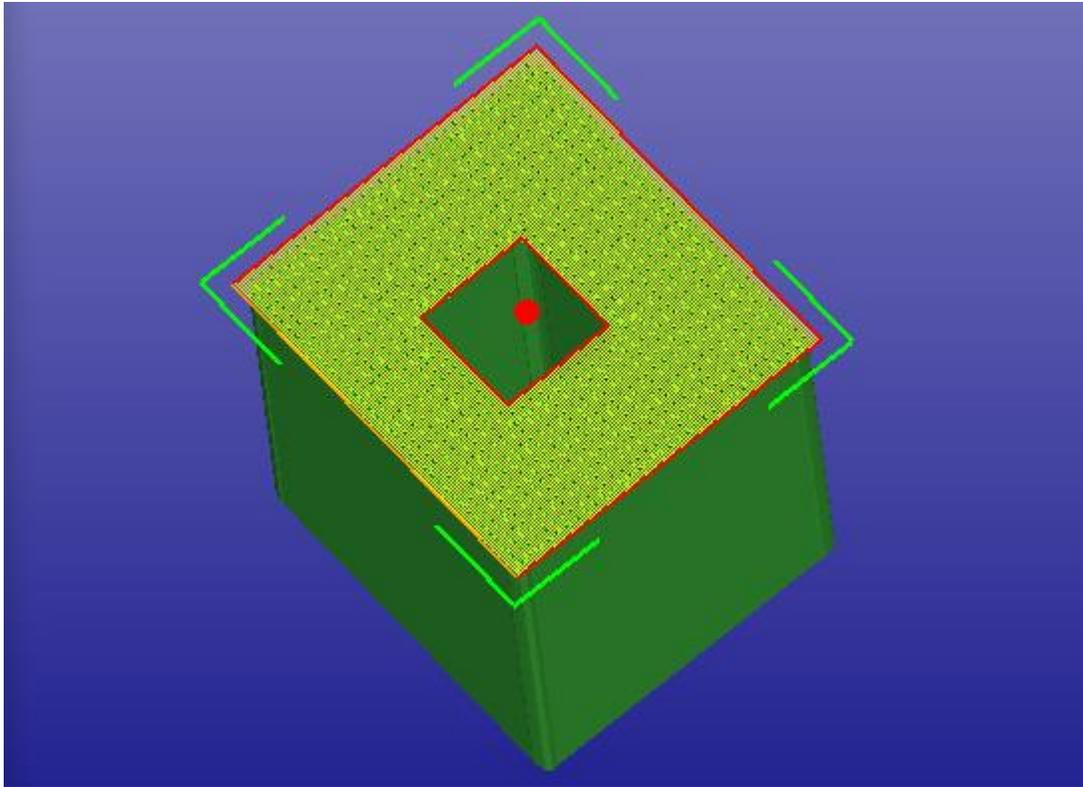
The *Web support parameters* dialog becomes visible.



There are different options to tune the Web supports but now we just want to make the supports more dense for the second area.

- **change the X spacing** value to **1 mm**
- **Press Apply** button to “Create supports for the current area” as set earlier in the *Create supports toolbar*.

We can immediately see that the supports for the second area change and become more dense:



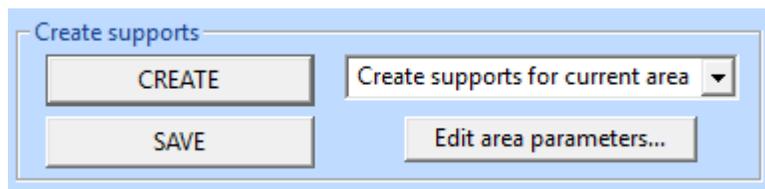
You can edit any parameter, press the **Apply** button and see the change immediately on the screen (you may need to do some panning/zooming to see the support area). This will help you to quickly find and tune the correct parameters setting for different manufacturing processes and areas in a model.

Alternatively, you can also close the dialogs one by one:

- **press OK to close** the *Web support parameters* dialog
- **press OK to close** the *Support parameters* dialog

We are back at the Create supports toolbar and can

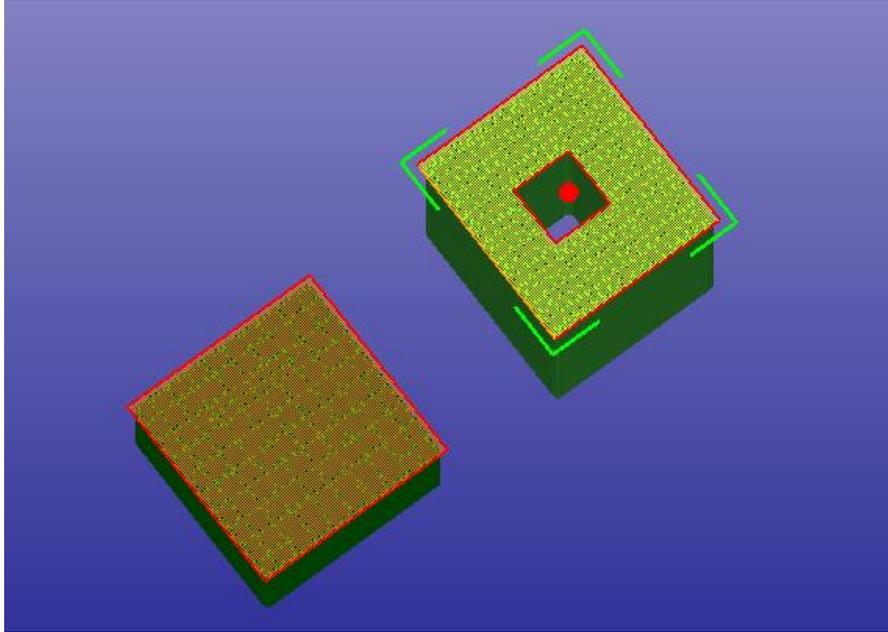
- **press the Create button** to *Create supports for the current area* again.



If we now

- **turn on the Display areas type All**

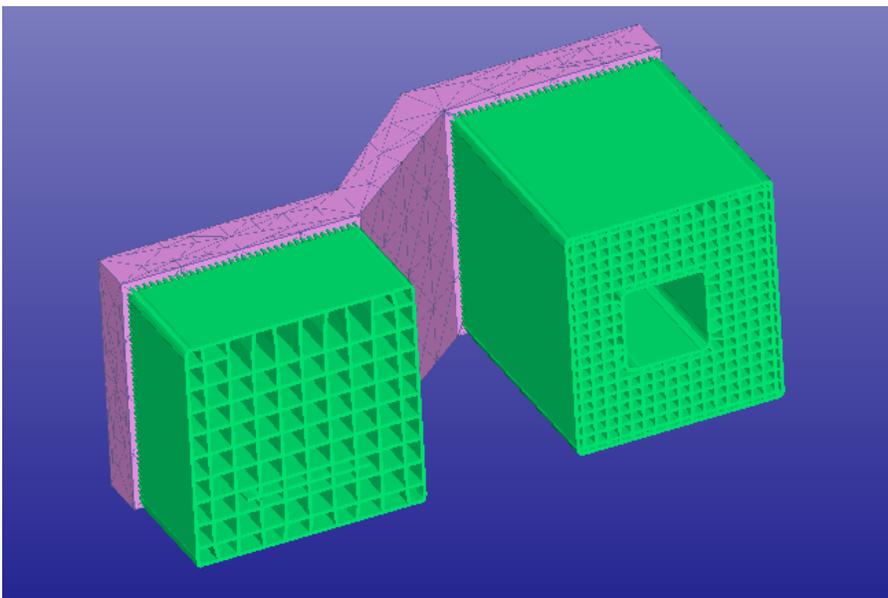
we can see that the two areas now have different density for the supports.



You can now

- **press *OK* in the *Create supports toolbar*** to save the supports in the Model Tree

You will see the two areas generated with different Web density:



The *Apply* button in each parameter edit dialog allows you to test the changes to the parameters immediately after changing them in the parameter dialogs. This will enable quicker parameter tuning without the need to go back to *Create* button in the *Create supports toolbar*. The use of *Apply* button is explained in more detail in the *Chapter 2 – Learning the support parameters*.

Generally, you can generate supports for one or all areas at a time. Also, you can only generate areas and add the supports later, automatically or by adding them manually with support editing. The *Create support* selections have the following meanings:

***Create all areas and supports***

recreates everything automatically, default choice.

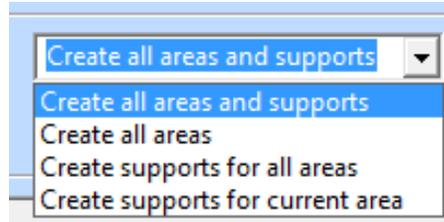
***Create all areas*** recreates the areas requiring supports but leaves the actual support generation later.

***Create supports for all areas***

recreates the support for all areas available in the support area list

***Create supports for current area***

recreates the supports for the area selected in the support area list.

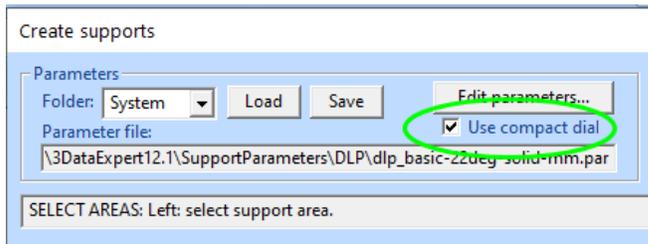


*Note:* If Side or Arch supports are manually created in the model (see chapter Arch and Side supports), they will disappear if *Create all areas and supports* are generated. If *Create supports for all areas* is generated a warning is issued. Arch supports may be straightened but Side supports are not affected when *Create support for all areas* is performed.

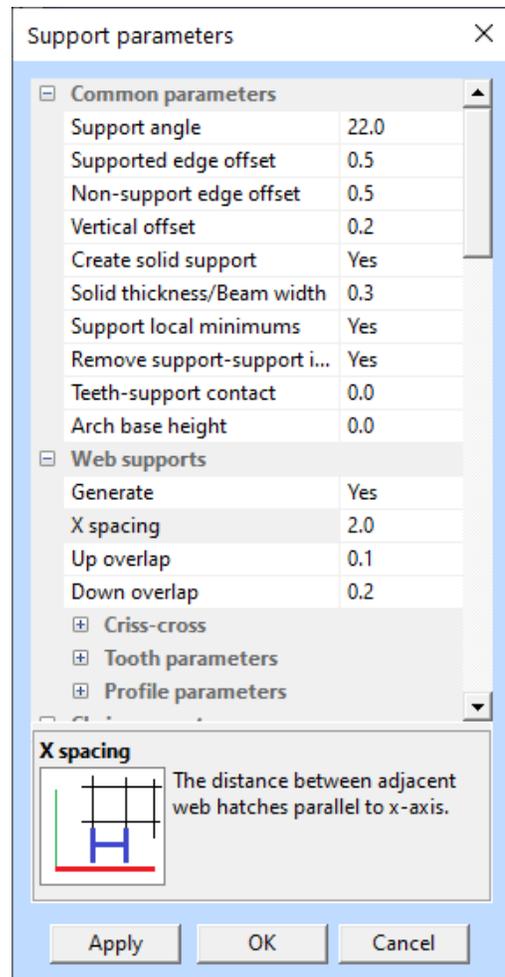
*Note:* You can also use the *compact Support parameters* dialog when you get more familiar with the system and its parameters. With *compact Support parameter* dialog less graphics area is covered and 3D models can be more easily viewed.

## Expert Series 12.1: Generating Supports for AM

Compact Support parameter dialog is used when **Use compact dial** is *On*.



In the image right the X spacing is selected for editing. When a parameter is selected in the dialog, the lower window shows the explanation text as well as the parameter image.



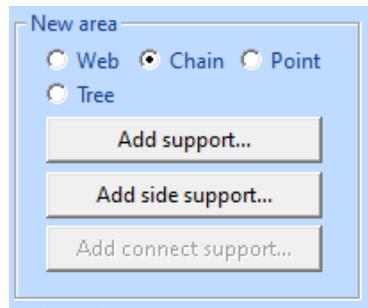
## ***Lesson 2 – Editing support geometries manually***

The Generic Support command with 3Data Expert can generate supports to most models

### **Adding supports manually**

Sometimes it might be necessary to add supports manually. With Generic Supports command the user can add four types of supports manually: Web, Chain, Point and Tree supports.

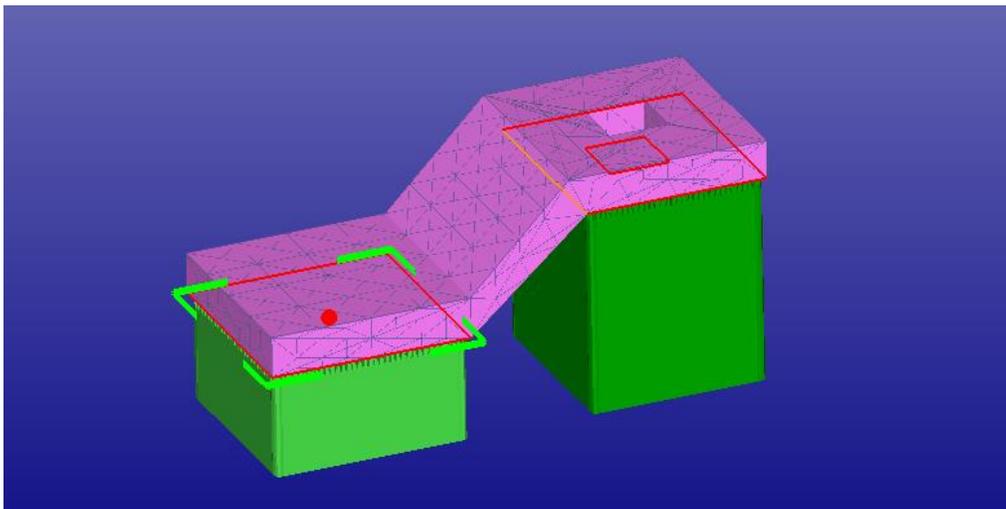
First the area type to add is selected with the *New area* group on the *Edit supports* toolbar.



Once again,

- **open the *supp-demo1* model and**
- **give *Generic Support* command with *dlp\_basic-22deg-solid-mm.par* file.**
- **press *Create* button in *Create supports* group**

The all supports are now visible in the graphics area.

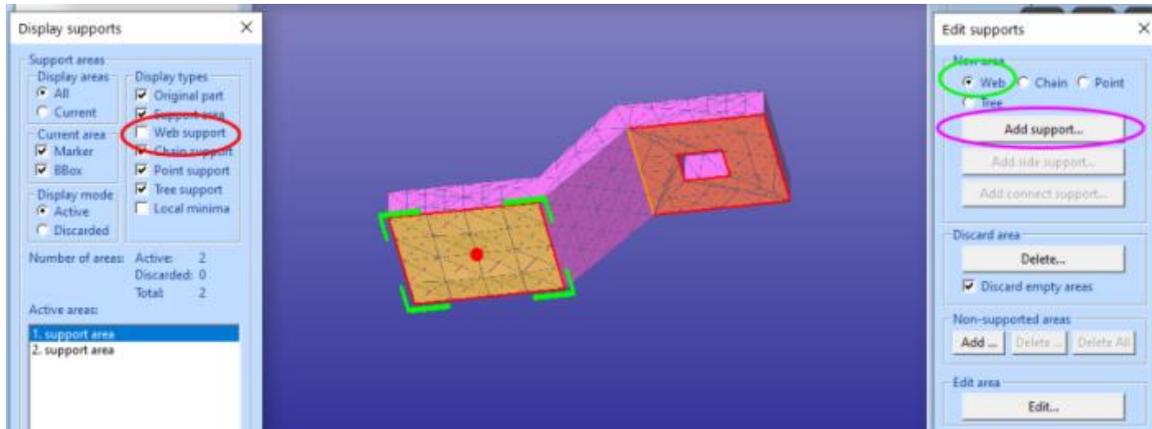


## Expert Series 12.1: Generating Supports for AM

The slanted area between the two horizontal support areas does not have any support generated due to the 22 degree surface angle limit.

Let's now take a closer look to the non supported area and hide the created supports. In the image below we can see the result after

- **Web support display is turned Off** (in red circle below)
- **the shaded+wireframe display mode is set on via  icon in the Viewing toolbar**

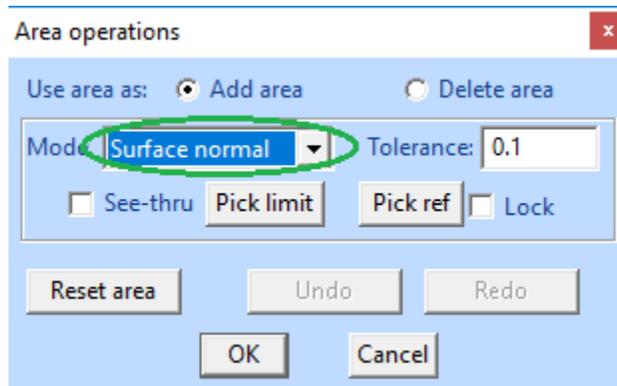


We will first generate a new Web support area. For this

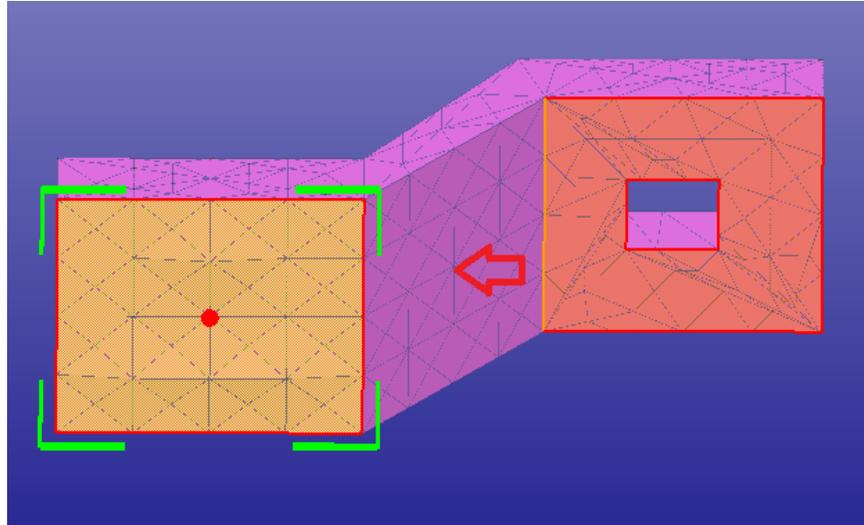
- **select Web from the New area group** (in green circle above)
- **Press Add support button to start adding areas** (in pink circle above)

An area selection tool *Area operation* window becomes visible. We will use Area operations to select the triangles for new support area generation. Area operations allow you to select the area in many different ways, please see the *Fix Model > Edit Triangles* command Online help for more information. This time we will use the *Surface normal* method available in the Mode selection list.

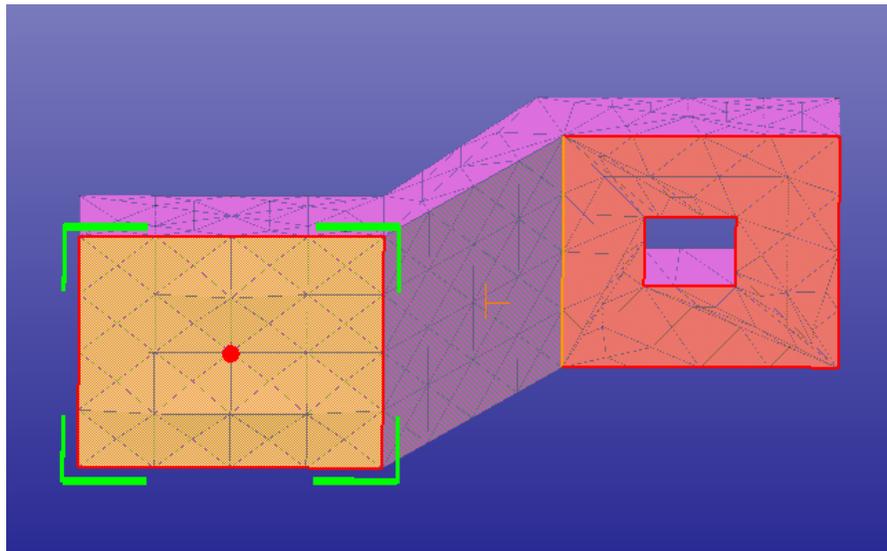
- **select the Surface normal mode** (circled in green below)



- **click on a triangle in the middle of the area with LBM (below)**



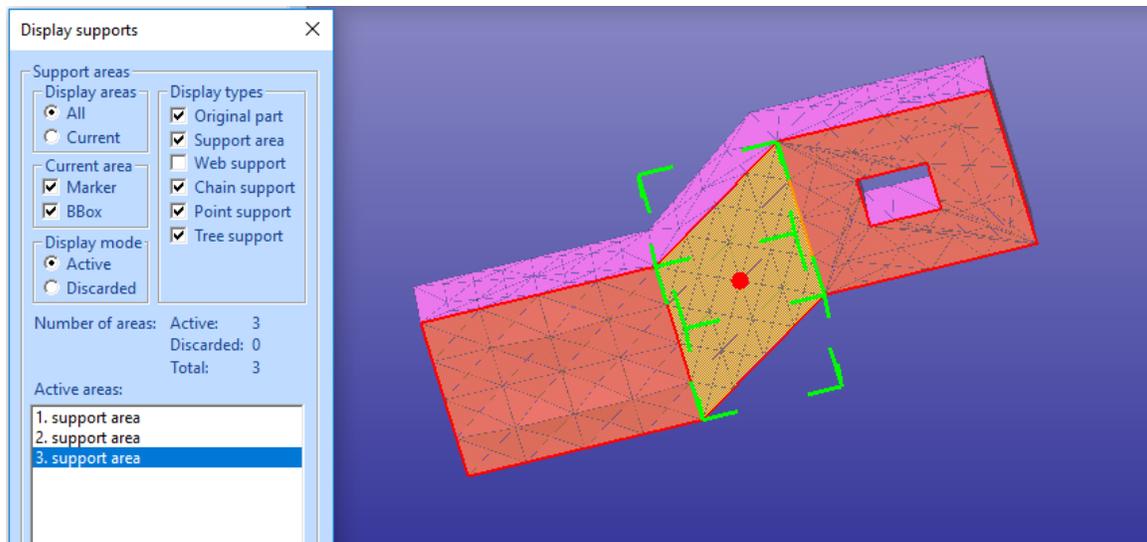
The whole slanted surface becomes greyed out and will show the triangles selected by the *Surface normal* criteria.



These triangles will form the new area when we press the OK button in the Area operations dialog.

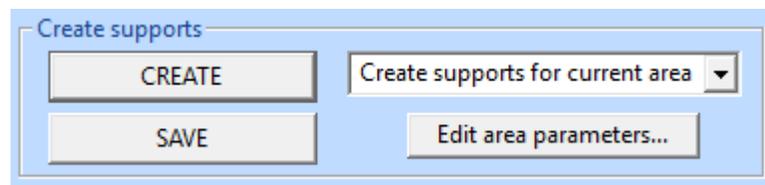
- **press *OK* in the *Area operations* dialog**

A new area appears in the support area list and the triangles in the slanted area are shaded with support area color. The new area is also selected in the area list.



Now we are ready to generate the supports for the newly created area.

- **select *Create supports for current area*** in the *Create supports* group

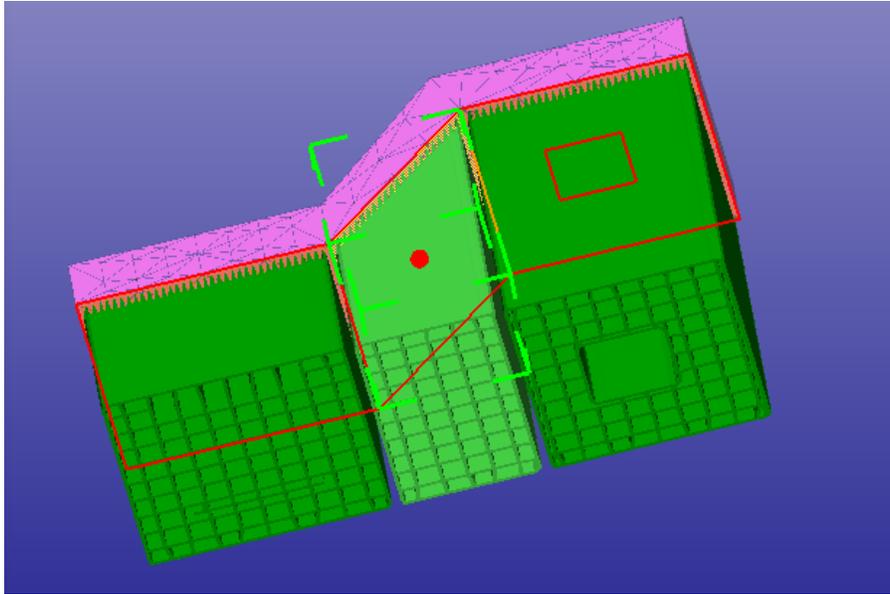


- **press *Create* button** in the same group

And the supports are created for the new area using the currently available support parameters for Web area. Anyhow, they are not immediately visible because the *Web support* in *Display types* is *Off*.

- **turn *On* the *Web support* type** in *Display types* group, if *Off*

You will see the new support below the slanted area, as seen in the next image. (Note: you may need to increase the *Support angle* parameter to 40 deg, described later to get full grid below the middle area)



Let's now add some *Chain* supports and a *Point* support on the slanted area. You can now press *Cancel* in the Create supports toolbar to quit this exercise.

Then restart again with

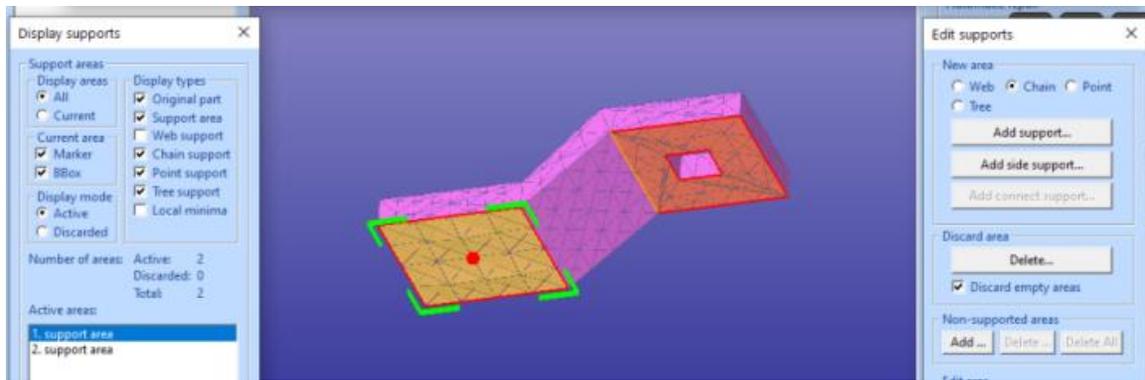
- **open the *supp-demo1.3de* model** (or continue with already inputted model) and
- **give *Generic Support* command with *dlp\_basic-22deg-solid-mm.par* file.**
- **press *Create* button in *Create supports* group with *Create all areas and supports* selected**

The *support toolbars* and all supports are again visible in the graphics area.

Let's again take a closer look to the non supported area and hide the created supports. In the image below we can see the result after

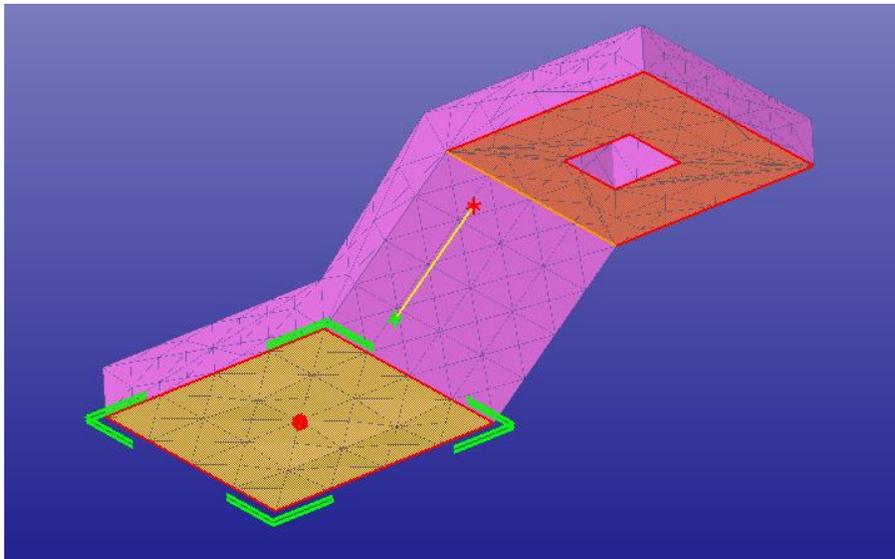
- ***Web support display is turned Off***
- ***the shaded+wireframe display mode is set on via  icon***

## Expert Series 12.1: Generating Supports for AM



We will now generate new *Chain support* areas. For this

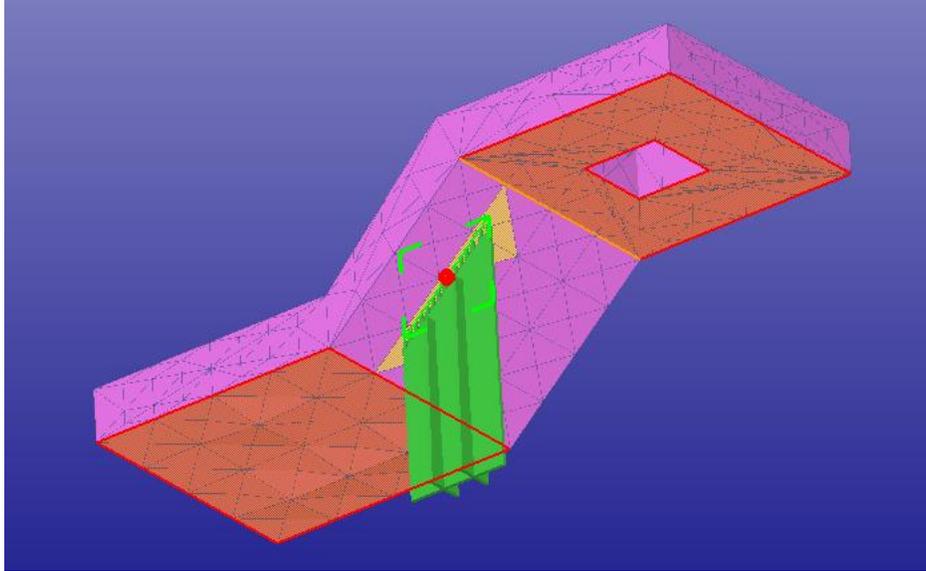
- **select *Chain* from the *New area* group**
- **Press *Add support* button to start adding areas**
- **use the *hair cross* and *LMB* on the graphics area to select two points on the surface as seen below (Alt+LMB would select a triangle corner point). If you make an error, you can undo with ESC button.**



Also, the real-time Help window gives advice on what mouse button and keyboard key commands are available:

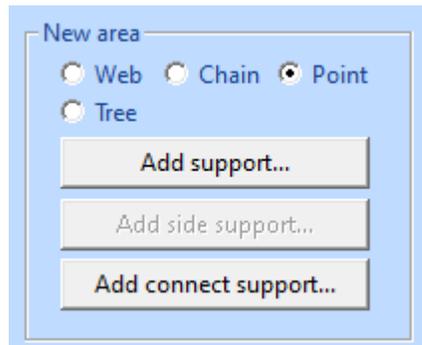
ADD AREAS: (CHAIN) : 0 point(s). Left: select chain support start point (Alt+ Left: select corner), ESC: cancel add operation

- **use the *RMB* button to accept the new Chain support definition**
- **and the Chain support is generated**

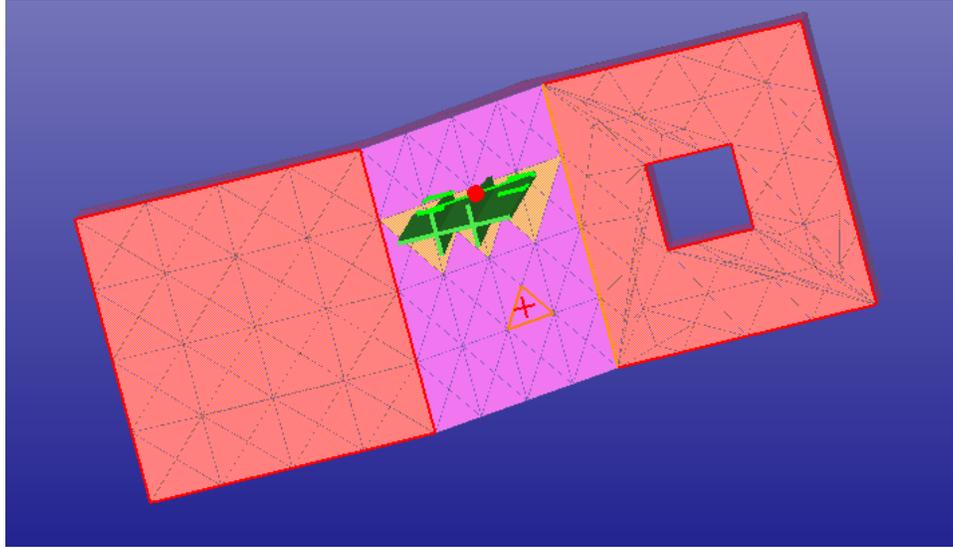


We can also add one Point support area in the slanted area.

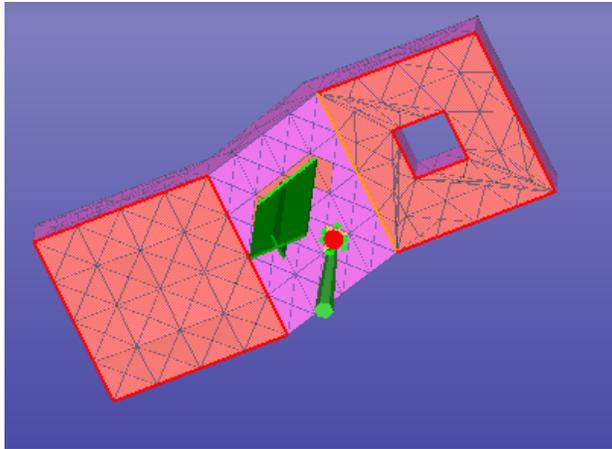
- select **Point support type** in the **New area group**



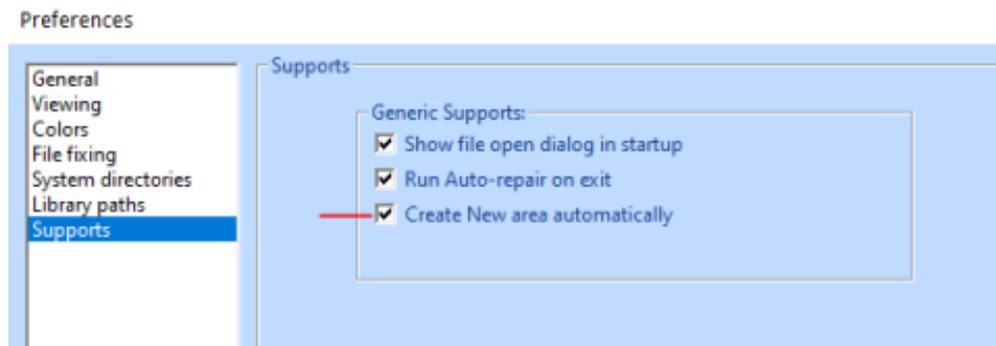
- press **Add supports** to start adding **Point** supports. Select a point in the middle of the slanted area with LMB as seen below (Alt+LMB would select a triangle corner point)



- when ready, press **RMB** to add the **Point** support area into the support area list



Now we have defined two new areas in the support area list and they are automatically generated when RMB is pressed. This happens when the *Create New area automatically* is set in the *Edit > Preferences > Supports* tab:



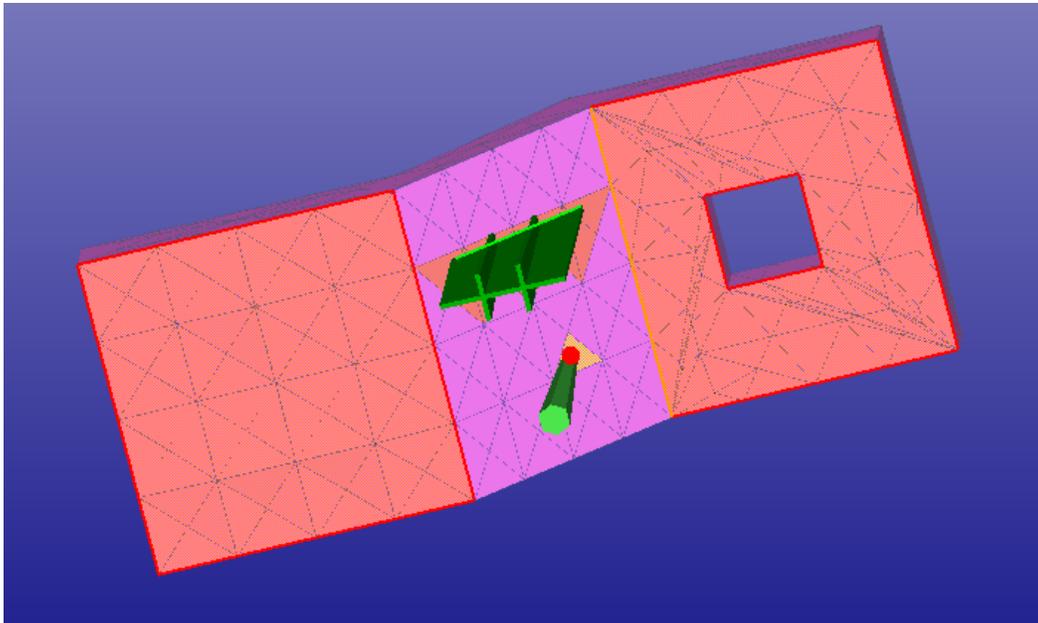
## Expert Series 12.1: Generating Supports for AM

---

If not set to generate automatically, you need to generate the new support areas separately. For this we will select the *Create supports for all areas* in the *Create supports* group, if not already selected.



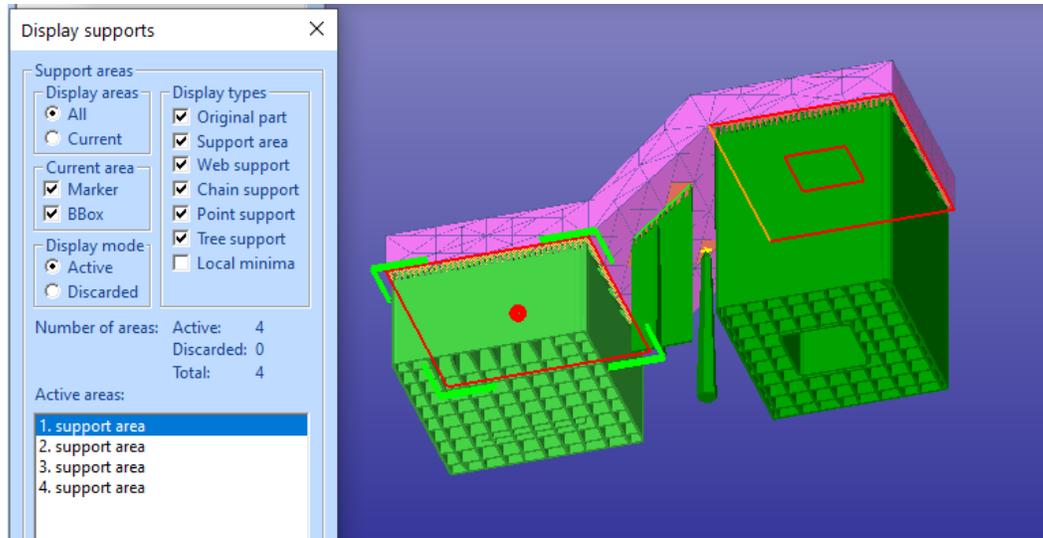
- **select *Create supports for all areas*** in the Create supports list
- **press *Create button*** and supports for new areas become visible



Now supports for all areas are regenerated. Currently we can see only the new *Chain* and *Point* supports because *Web supports* is *Off* in the *Display types* group.

- **set *Web supports type On*** in the *Display types* group

and you will see all the supports in the graphics area.

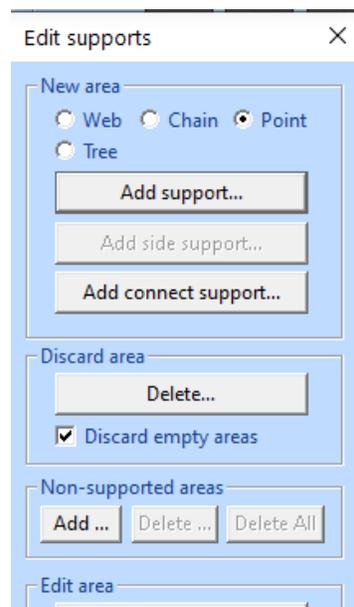


The green lines indicate the location of the Chain supports and yellow cross the location of the Point support.

### Deleting supports manually

There are also some other tools available in the support toolbars which allow you to *Delete Whole areas* and *Area details*, *Join* and *Split* areas.

In this exercise we will delete whole support area as well as details inside a support area.

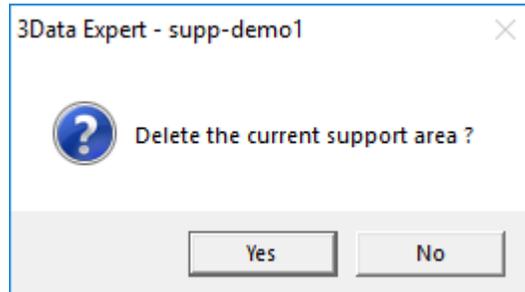


## Expert Series 12.1: Generating Supports for AM

---

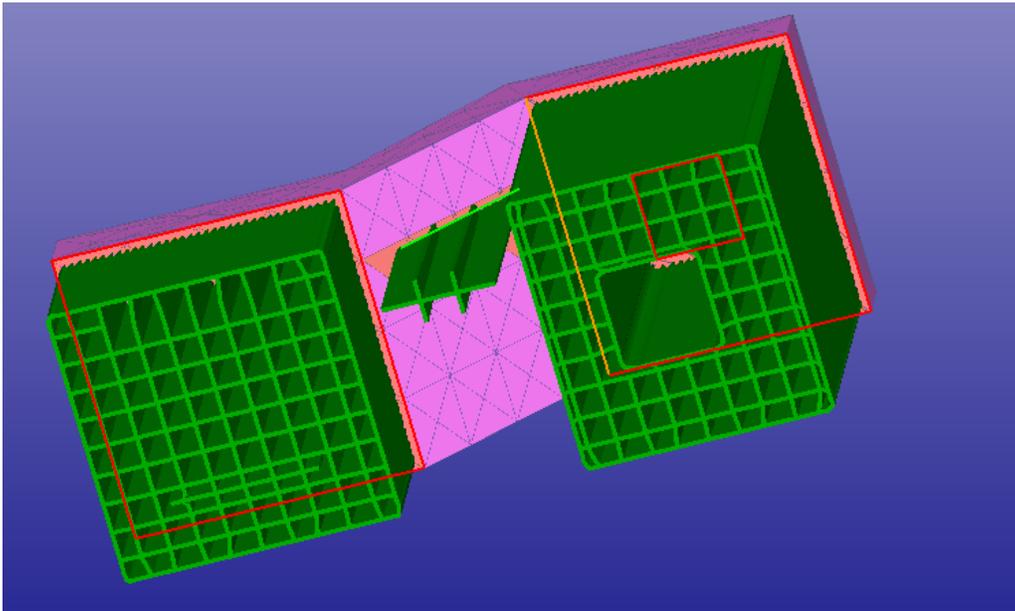
Let's now do a couple deletions from the supports generated in the previous chapter. First we will delete the last newly added area, the Point area.

- **select the 4. support area in the area list** (either from the list or by clicking the area triangle on the model with LBM)
- **press *Delete* button** in the Delete area group (as in the image above)



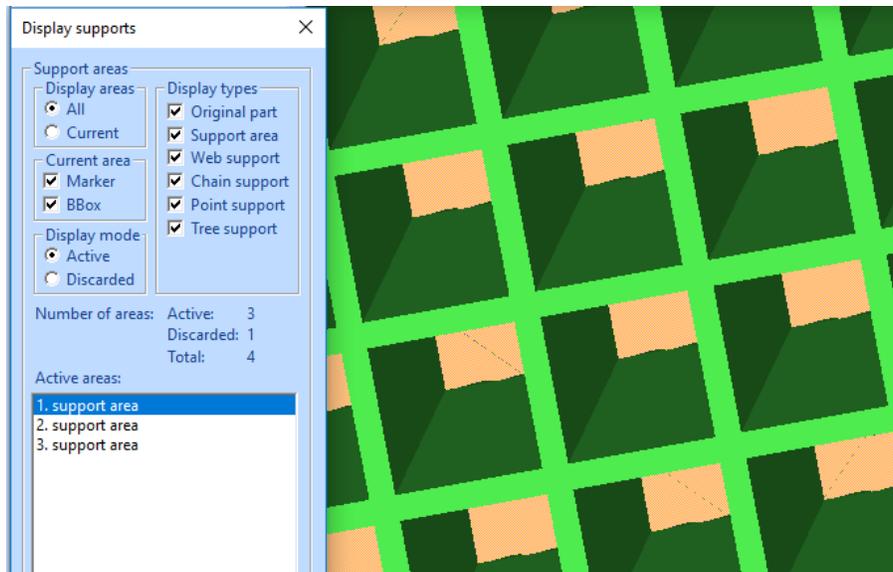
- **press *Yes*** to the *Delete the current support area* request

The Point support disappears from the support area display. Anyhow, it remains in the support area list for subsequent recovery, if necessary.



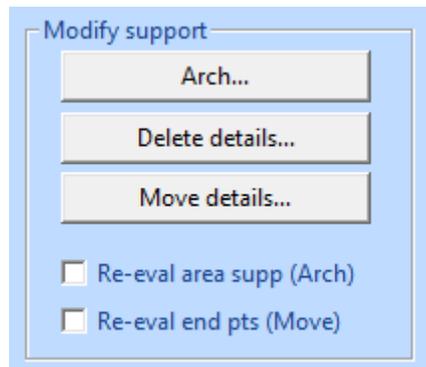
Now let's delete some details from the existing support areas.

- **First select the 1. support area** and zoom closer to it.

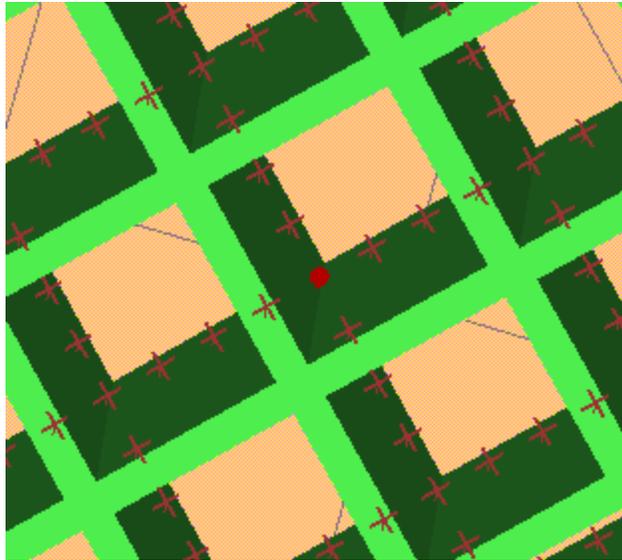


Then

- Press **Delete details** button in *the Modify support group*



- then press **Delete button** and the software goes into *support detail deletion state*. You can **select** as many web points indicated with small red cross as needed **with the LBM**. Selected points are indicated with small dots and line (1 selected in the image below). If necessary, press **ESC** to cancel a selection.



- When all necessary web points are selected, **press *RMB* to delete** the points from the support geometry.

The selected points are immediately removed from the support geometry and the system returns from detail selection state.

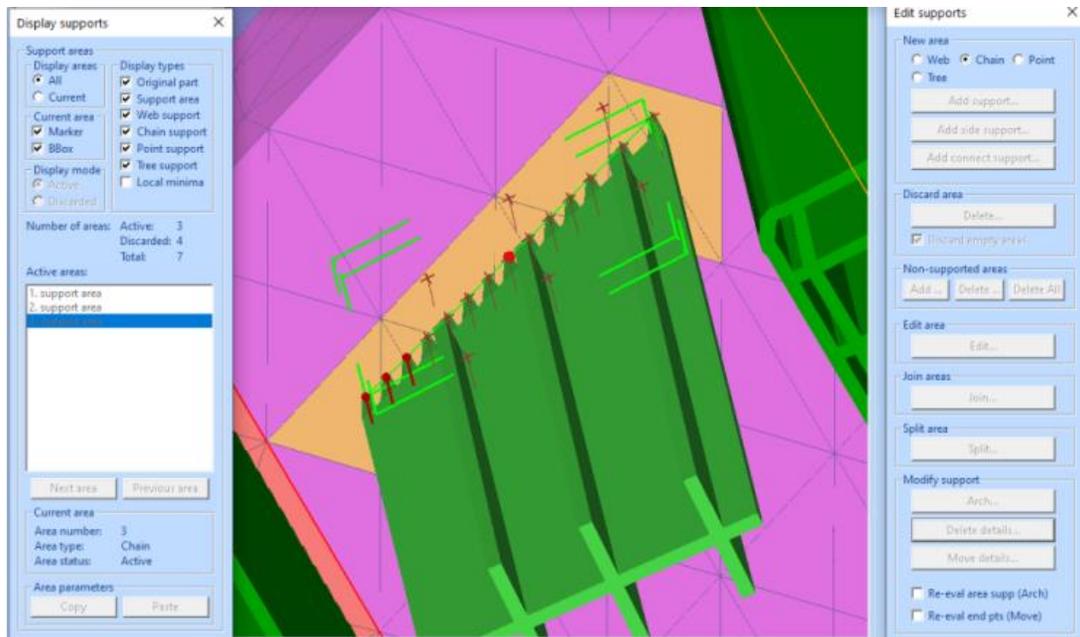


After the selected points are removed there is no possibility to Undo the deletions anymore. If necessary, the supports for the given area must be Created again.

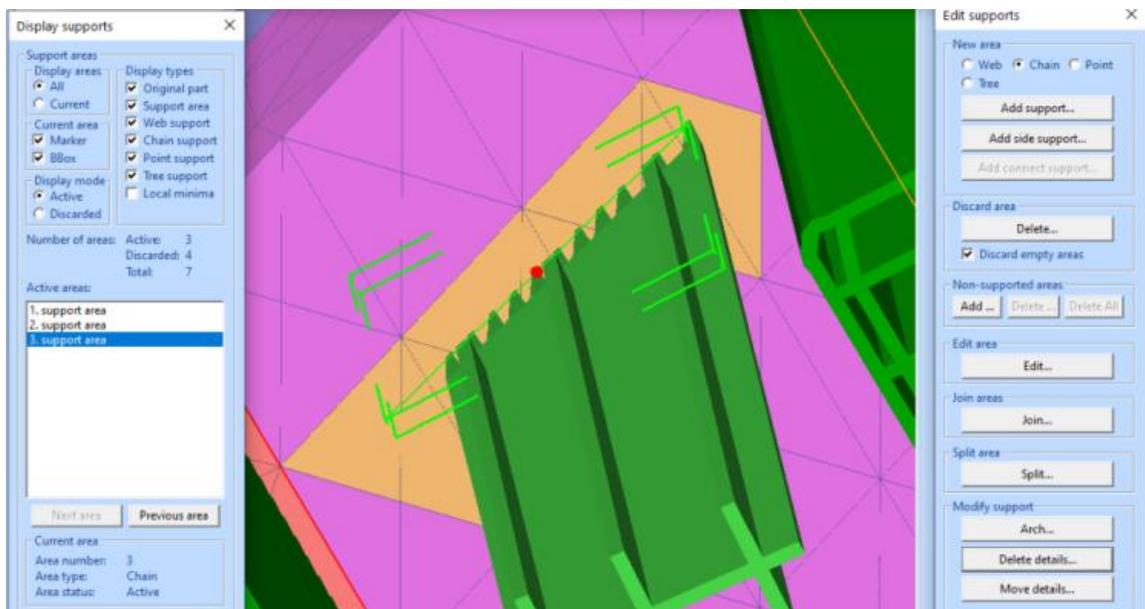
Similarly, you can delete points from the newly added Chain supports:

- **select the 3. support area** from the area list (display on the Current area for clarity)
- **press *Delete* button with *Area detail selection***
- **select Chain teeth points with *LMB*** by clicking a part triangle close to an edge point

## Expert Series 12.1: Generating Supports for AM



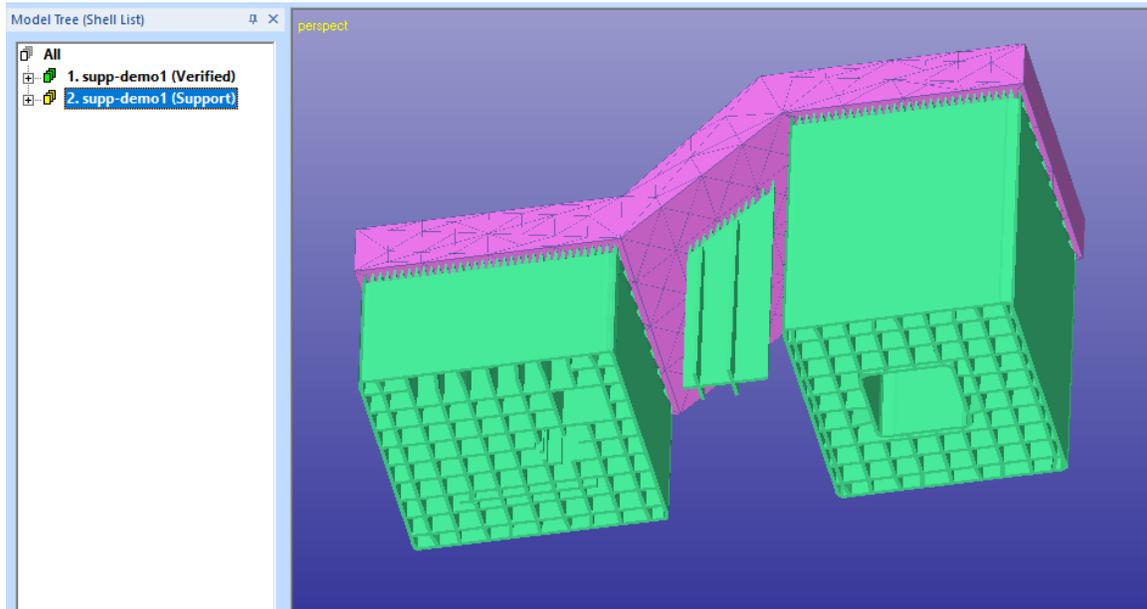
- **press RMB to delete** the selected points. The selected points are immediately removed from the support geometry



Delete area details help to clean the support geometry in case of unnecessary points or to prevent surface areas below the supported area to be destroyed by the supports. Of course, you must be careful not to delete too many support points to enable correct building of the part.

Finally,

- **press OK in the Create supports dialog to store** the supports into the Model Tree



### Applying Arch command to support geometry

When building complex parts with AM, the supports many times must be built starting from the actual part. Supports added to an overhang area must be built on top of an up-facing surface in the part. This will lower the surface quality on areas where the supports are built.

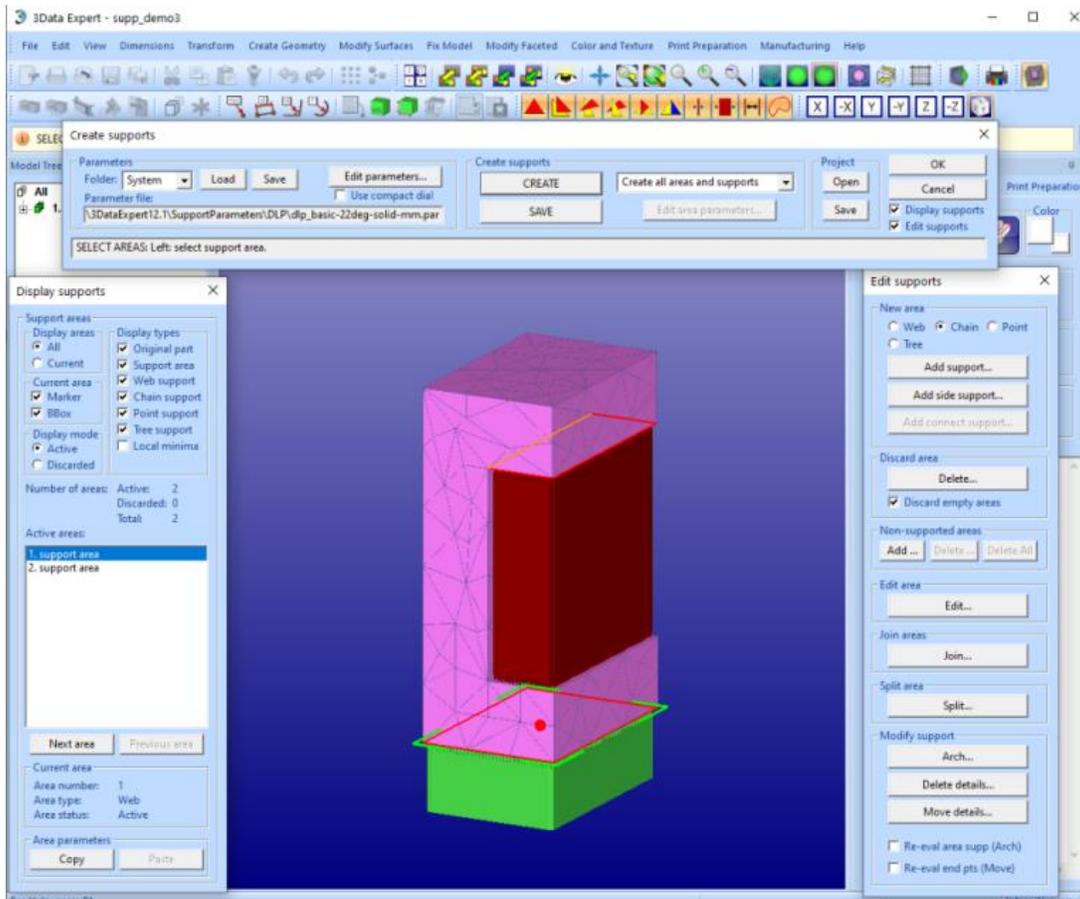
Generic Support *Arch* command helps to move the supports away from up-facing areas in the part. The user can graphically drag the support geometry into new location away from the part or to a less critical area of the part.

In this exercise we will use a simple part to demonstrate the use of Arch support. Start by

- **input the test model supp\_demo3.3de**
- **start Generic Support command**
- **select the usual dlp\_basic-22deg-solid-mm.par parameter file**
- **press Create button** in the Create supports dialog

You should see the following result on the screen:

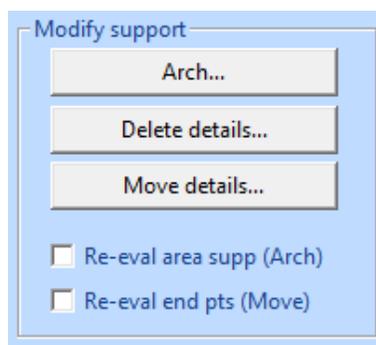
## Expert Series 12.1: Generating Supports for AM



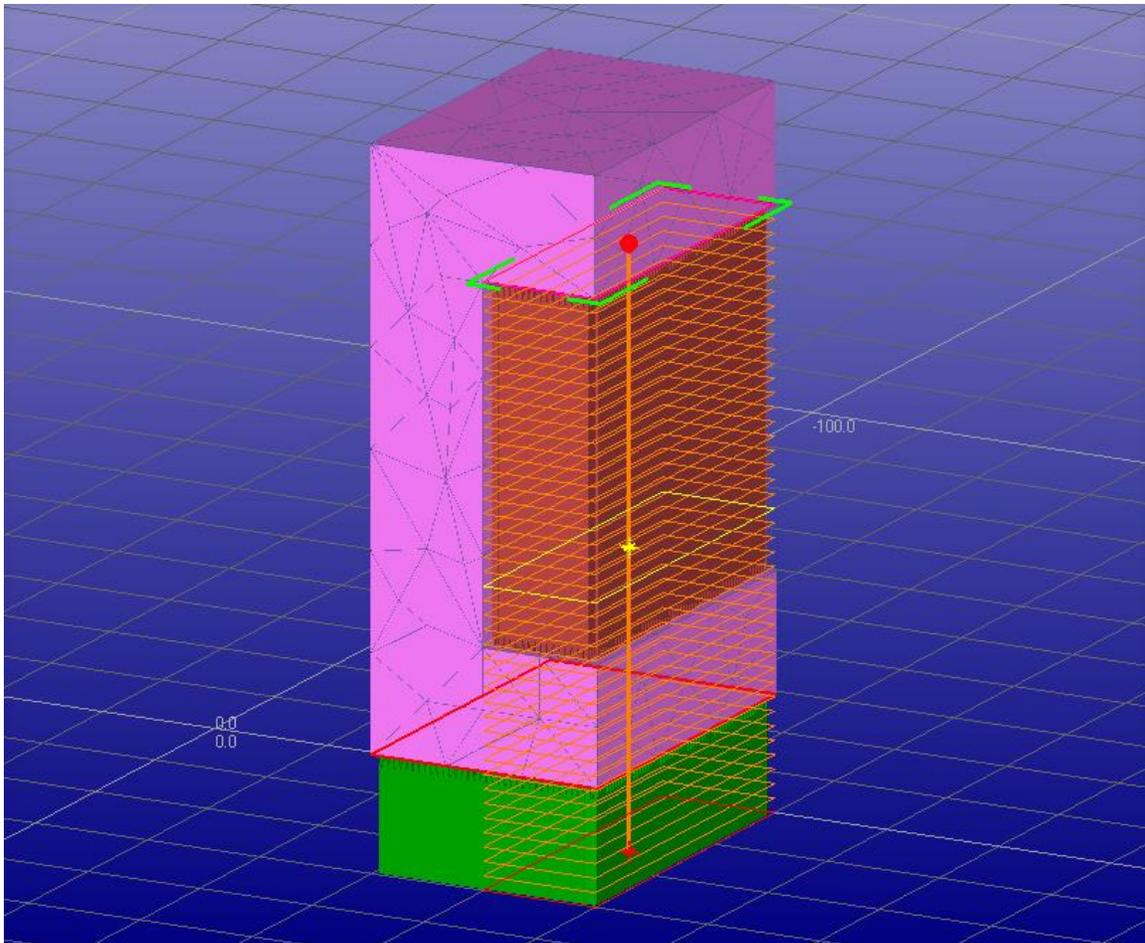
The support for the upper overhang area is hitting the part surface below. This is indicated by *red* support color.

We will now move the support away from the up-facing surface. First

- **select the 2. support area** (red)
- **press the *Arch* button** in the Edit supports dialog



The software will display the *arch support preview*:

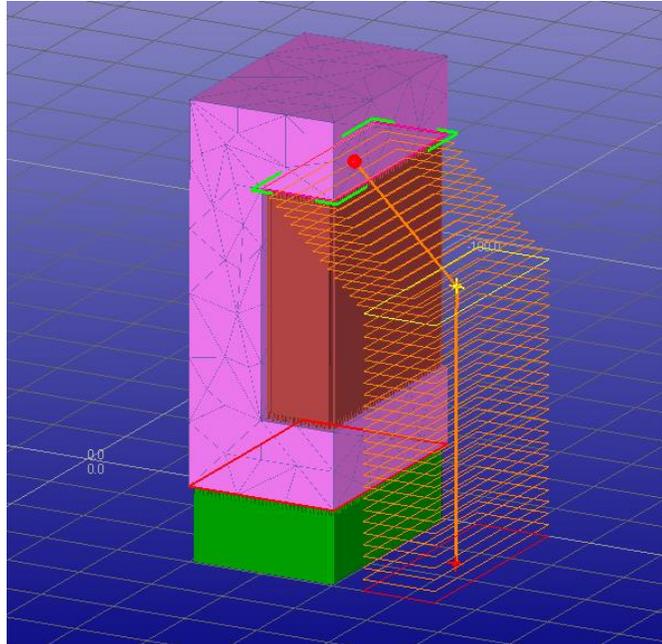


The preview shows the outline of the support geometry. The outline will help to position the Arch support correctly. The Create support dialog *Help* window will give *context sensitive* help on the possible operations:

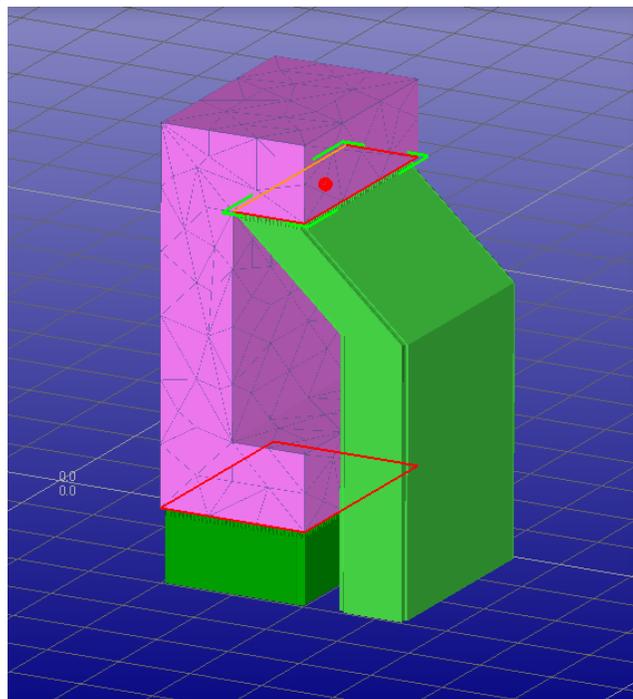
BEND SUPPORT: Left: Select arch point and move, Right: End arch operation, ESC: Cancel arch operation

First we will select the arc point, i.e. the bend point of the Arch support. The point is in the middle by default and it can be located anywhere on the vertical orange line.

- **first click on the bend position on the line with *LMB***, preferably higher than the given middle point
- **move graphically the bend point using *LMB*** until suitable position is found. The Grid displayed in the image will help to position the support root at the XY plane.



- when a suitable position is found, **press RMB to recreate** the support in the given location



In the new location the support will not hit the actual part. The surfaces in the finished part will be smooth without support marks.

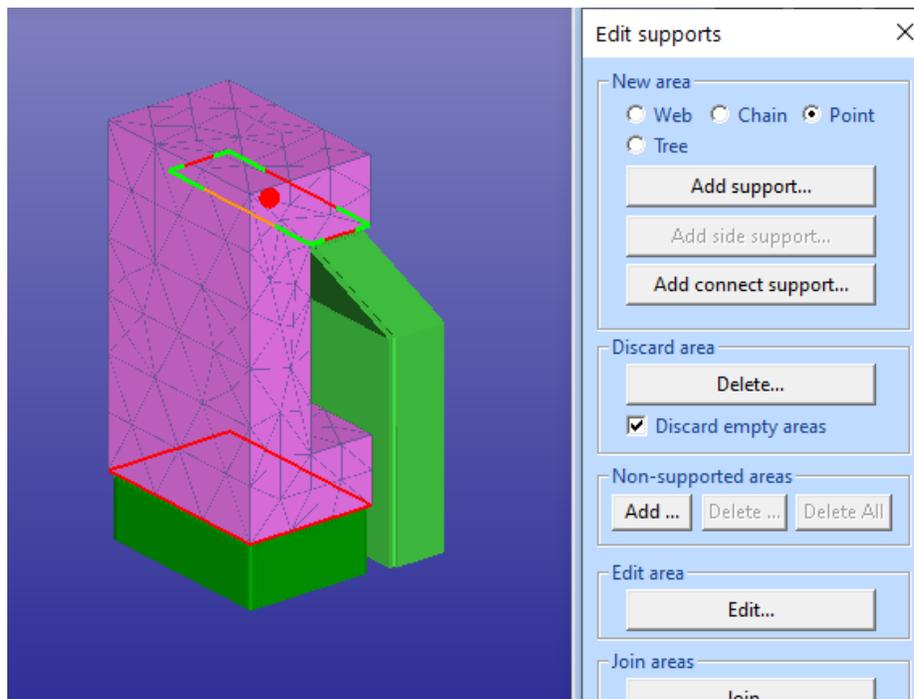
Note: Do not quit the Generic Support command with this model yet, we will continue the next exercise with the current situation.

### Adding Side support

The part supported in the previous exercise is relatively high when compared to the thickness. The support of the upmost overhang area starts to support the part only in the final phases of the build. This may cause bending or other defects in the part caused the recoating process required for the layer based manufacturing.

*Side support* can be used to support the part from vertical or slanted walls not normally requiring supports. These supports will add the robustness of the build assembly (part + supports) and/or dissipate heat more quickly from metal processes.

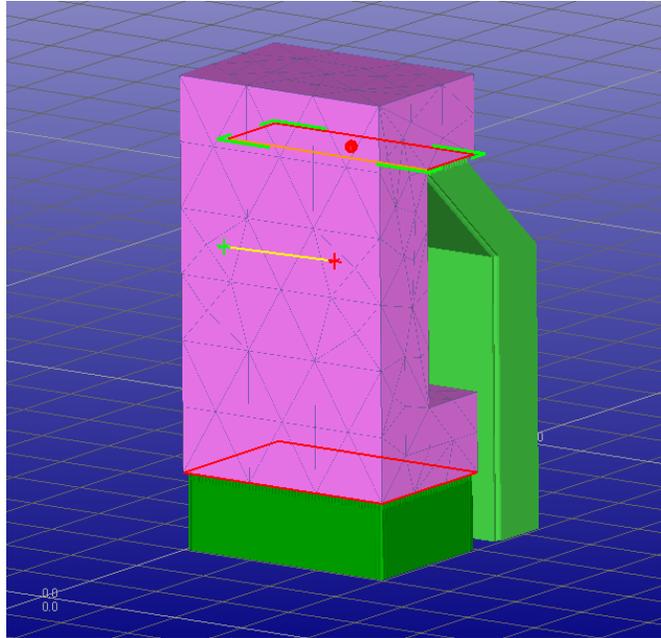
We will now continue with the result of the previous exercise. First take a view from the other side of the part and turn on *shaded+wireframe* display:



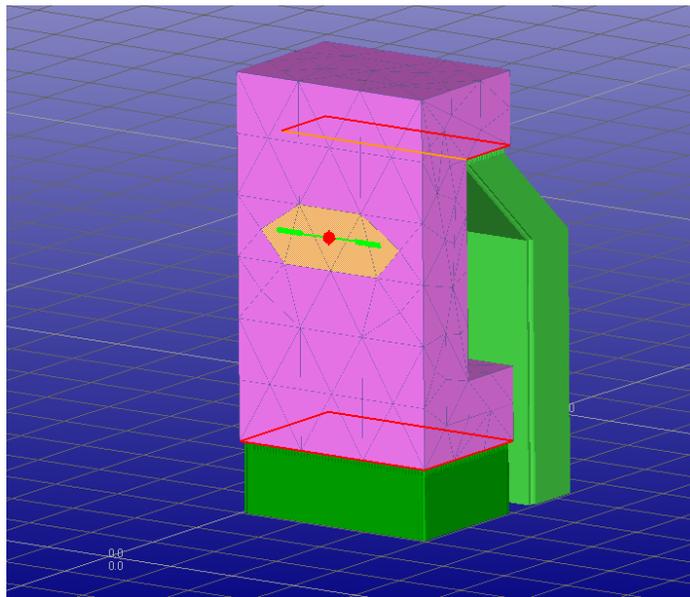
Also, you need to select the *Chain* support type in the *New area* group (Web and Point types implemented soon), as seen in the image above.

Now we are ready to make the Side support structure using Chain support type.

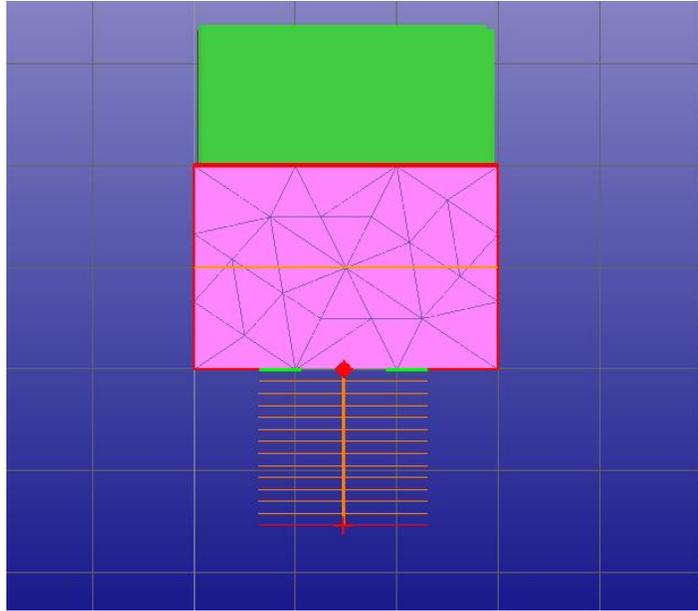
- **press the *Add side support*** button
- **select Chain support polyline from the part surface** as seen in the image below. Click on the consecutive triangle edges with *LMB*. Press *ESC* to undo in case of errors.



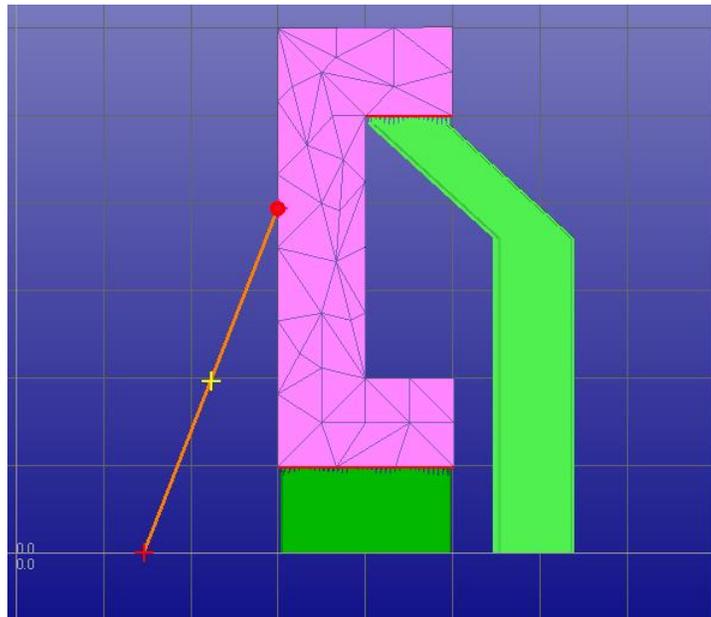
- when the polyline is correct, **press RMB to accept the polyline**. The side support polyline and triangles are displayed on the part



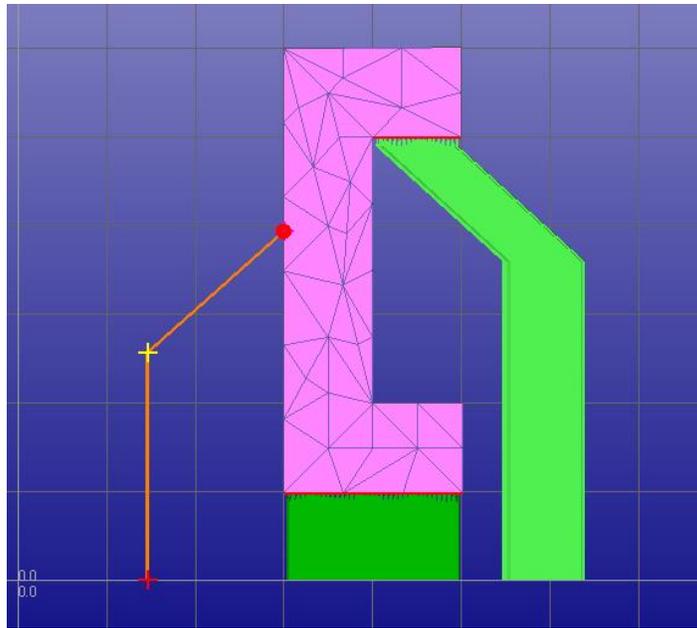
- **press Z projection button** (  ) to take a view from up
- **show the Side support root point with the LMB** on the XY plane. The *side support preview* is displayed on the graphics area



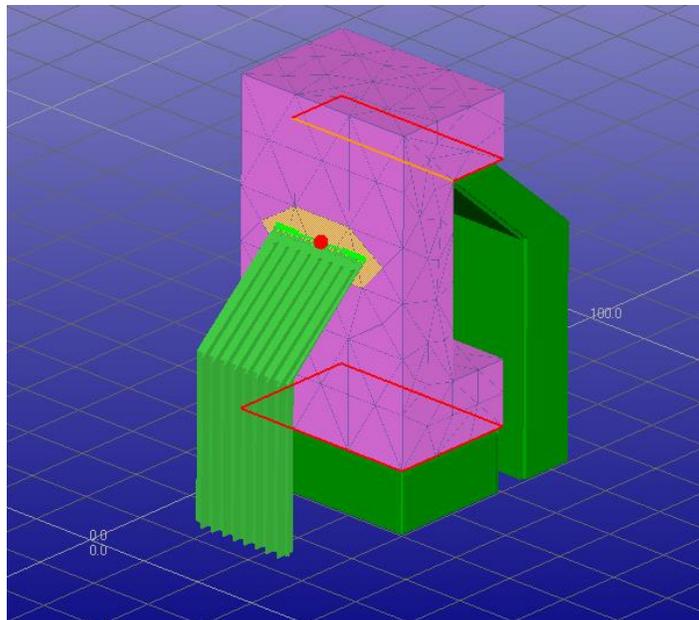
- **press RMB to accept the bottom point** (or *ESC* to undo and reset)
- **press X projection button** () to take a view from front



- **press with *LMB* on the orange preview middle line** to indicate the bend point for Side support
- **and move to correct location with *LMB+dragging***



- press **RMB** when ready to generate the Side support



The added Side support geometry will increase the robustness of the build significantly.

During the Side support drawing we were using Z and X projections. The drawing and dragging can also be done in 3D view but many times it is more accurate to take projection views for support drawing operations.

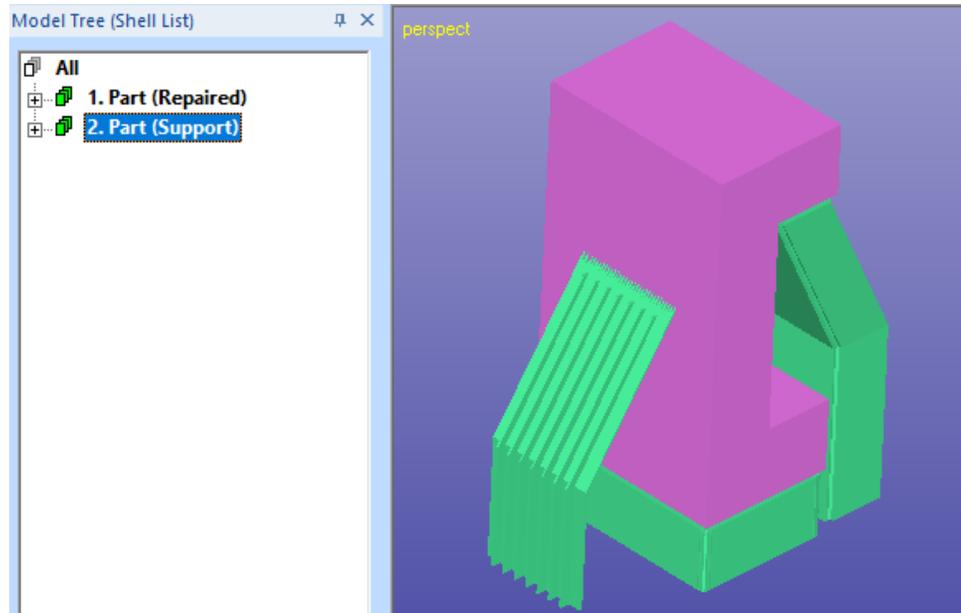
You can now

- **save the supports into Model Tree by pressing OK** in the Create supports dialog

## Expert Series 12.1: Generating Supports for AM

---

*Note: When the supports are stored into the Model Tree they are checked for errors when solid supports are used. This may take some time due to the Arch supports which cause more triangle-triangle intersection checks than normal supports.*



In the above example Side supports were based on Chain support type. Chain supports are normally created to down-facing sharp areas and lay relatively horizontal. Thus it is a good practice to draw the Side support when using Chain type horizontally, otherwise the supports may have some artifacts in the final result (not proper solid).

### **Adding Tree support**

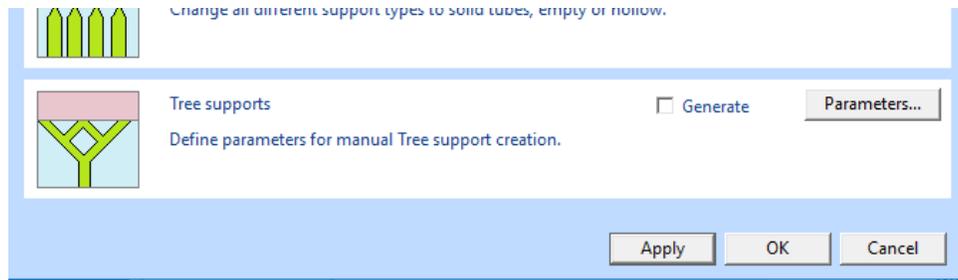
Let's now have a look at the last support type, *Tree supports*. Tree supports are created by drawing a tree like net with tree root and branches.

The tree support is generated in phases: First we will select the top of the tree on the part surface. Then the tree root is show on the XY plane. After that we can add branches, move points and tune the tree thickness in different levels.

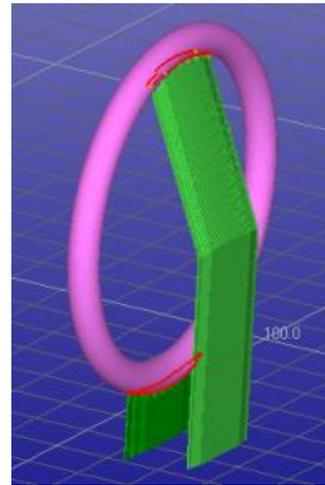
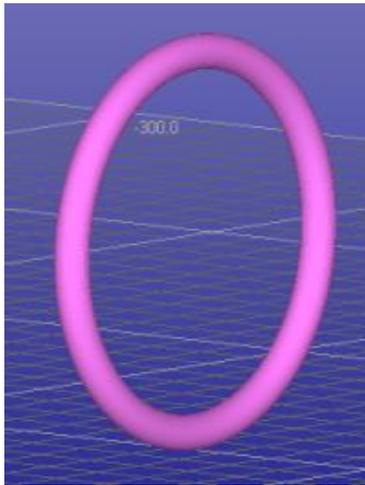
The Tree parameters are set through Tree supports parameter dialog accessible via Support parameters dialog:

## Expert Series 12.1: Generating Supports for AM

---



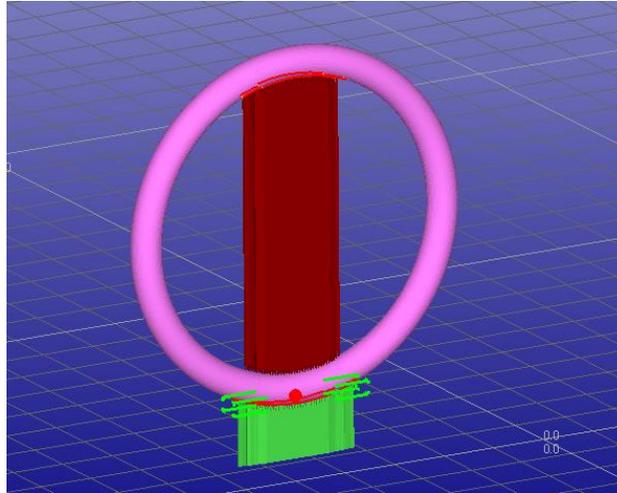
As an example we will use a simple ring like geometry. The supports for this model could also be done using the Arch support, as seen in the image below but this is left as an exercise to the reader.



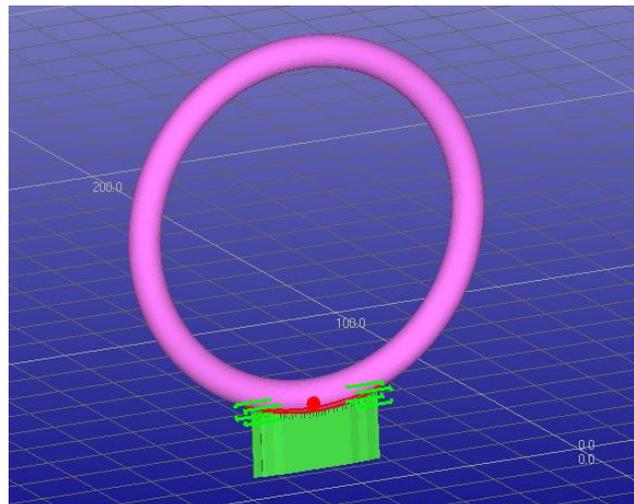
We start the Tree support generation by inputting the ring model.

- **input the test model `supp_demo4.3de`**
- **start `Generic Support` command**
- **select the usual `dlp_basic-22deg-solid-mm.par` parameter file**
- **press `Create supports` to generate normal automatic support**

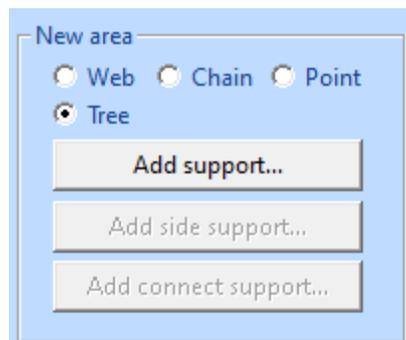
This time we will delete the 2. *support area* from the part and generate a less invasive Tree support instead. The 2. area is hitting the surface below and we want to avoid that. Also, Tree support will use potentially less material. Sometimes it could be sufficient to use Arch support to bend the support away from the surface below.



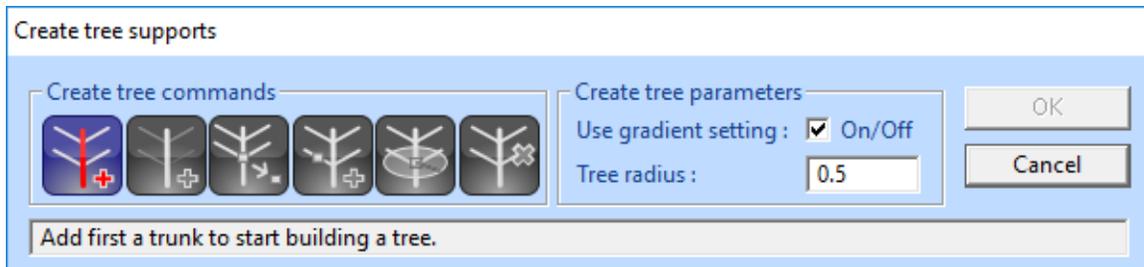
- select **2. support area** and delete it using **Delete whole area** command in Edit supports dialog



- select **Tree type** in **New area** group



- and press **Add support** button and **Create tree supports** dialog appears:



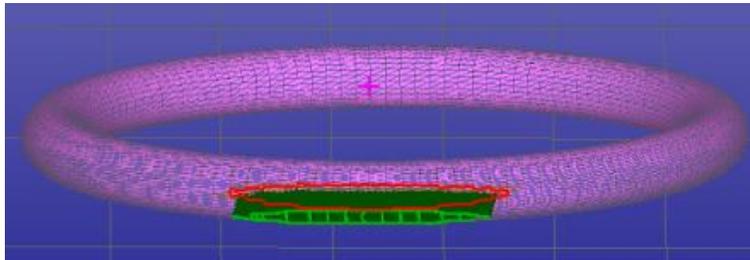
This dialog has all necessary commands to draw a Tree support. Also, the help text window gives context sensitive help to take through the tree generation easily.

It is a good idea to first take a “slanted” view to the model. This will help to position the tree tip to the correct position.

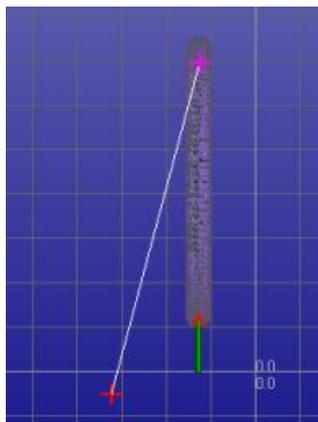
- **rotate the view** so that it is visible from the bottom (almost, see below)
- **press Add trunk icon** to define the tree trunk



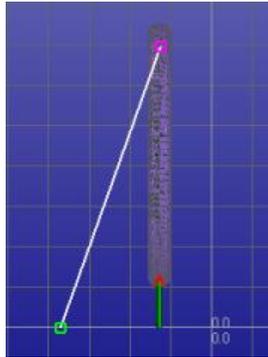
- **show the location of the tree tip on the model with *LMB*** (red cross in the image below)



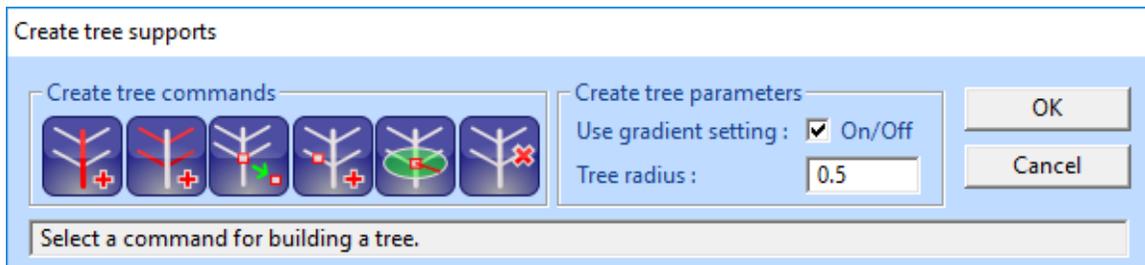
- **take a view from the Y axis** direction with 
- **show the location of the trunk root with *LMB***, preferably below the XY plane, as seen in the image below



- press **RMB** to accept the root location (or ESC to reposition)



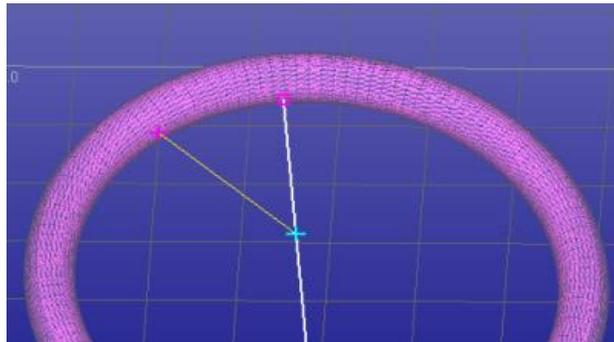
Now all tools in the Create tree supports toolbar become active and we can start adding branches to the tree.



We will generate two new branches to the tree. Once again we take a view almost from the bottom to be able to position the trunk tips correctly.



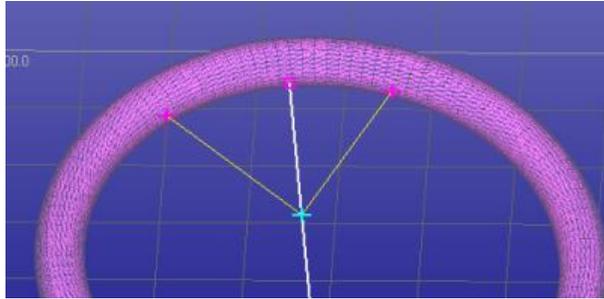
- first press **Add branches** icon
- then click on the model surface with **LMB** (red cross in the image below)
- then click on the tree trunk with **LMB** to add the branch point (blue point in the image below)



## Expert Series 12.1: Generating Supports for AM

---

Now we have added one branch on the tree. The second branch is added similarly, first click on the model surface then on the same location on the tree as the previous branch point on the trunk. You should have something similar to the image below. You can always *Undo* by pressing *ESC*.



- **when all branches are added press *RMB*** to accept the new branches

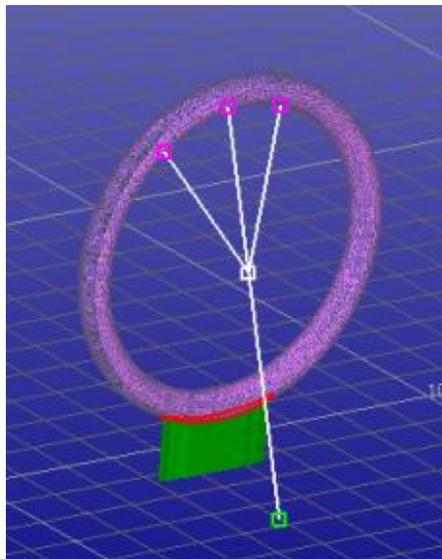
You should have a tree similar to the image below. Now you can either move the points with



icon or you can add wanted thickness to selected nodes in the tree with



icon.

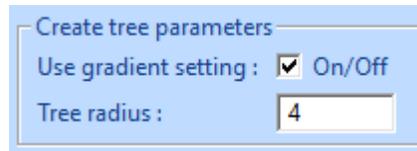


## Expert Series 12.1: Generating Supports for AM

---

The Tree thickness in the loaded parameter set is very small, 0.5 mm. We want to adjust the tree thickness in different nodes for more robust build.

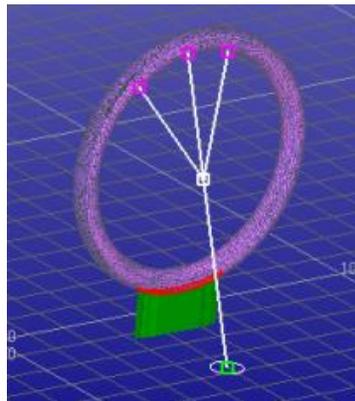
First set the Tree radius to 4 mm



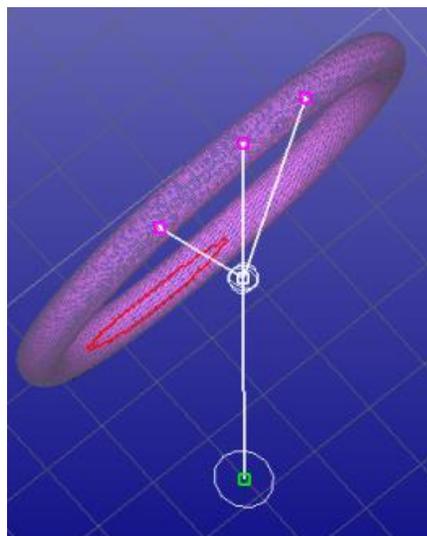
Then



- **press the Change tree radius at point icon**
- **and select the tree root node.** The new radius for the tree root will be displayed on the tree preview



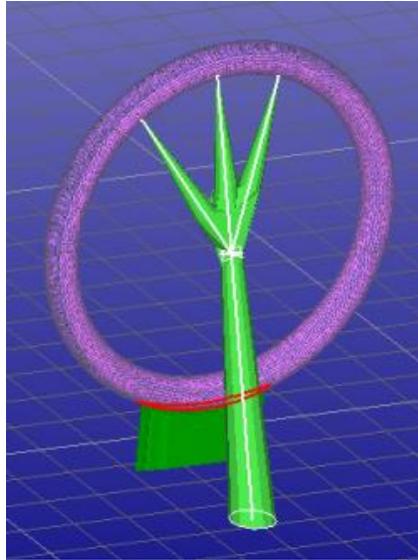
- **change the Tree radius to 2.0 mm**
- **and click the middle branch with *LMB*.** The middle branch will have a new diameter 2.0 mm



The new radius values will be used for the tree generation instead of the original 0.5 mm set through the Tree support parameter dialog.

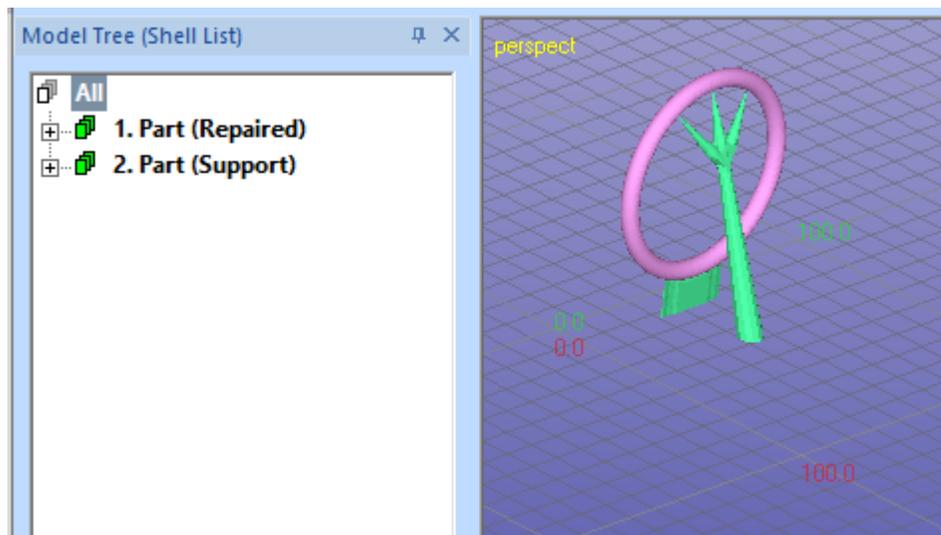
- **press *RMB* to quit changing tree radius**
- **press *OK* in the *Crate tree supports* dialog (or *RMB*) to store the new tree in the support area list and to generate the supports geometry**

And the Tree support is ready.



- **press *OK* to store the supports in the Model Tree**

The ring and the supports are now ready to be manufactured with suitable AM system.



## Expert Series 12.1: Generating Supports for AM

---

The above examples were all done using solid support structures. This was done mostly for better viewing experience, zero volume open support plates do not perform visually well. All the above examples can be done using the corresponding “open” parameter files available in the installation package.

### **Lesson 3 - Learning Support Parameters**

The Generic Support command with 3Data Expert can generate supports to most models automatically. For the automatic process to succeed with different processes, like metal or DLP, a suitable set of parameters need to be given to the software. The software installation comes with several default parameter files tuned towards different processes. Anyhow, the user must check that the given parameters and generated support geometries are suitable for the receiving AM system.

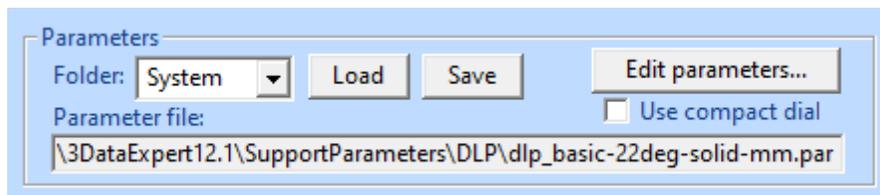
#### **Opening and saving support parameter files**

Always when the support command is started, a parameter file must be selected according to the used AM system. This ensures that the correct set or parameters is always used before starting the support generation.

Now let's start with the same example as before. Once again,

- **open the *supp-demo1* model** and
- **give *Generic Support* command with *dlp\_basic-22deg-solid-mm.par* file.**

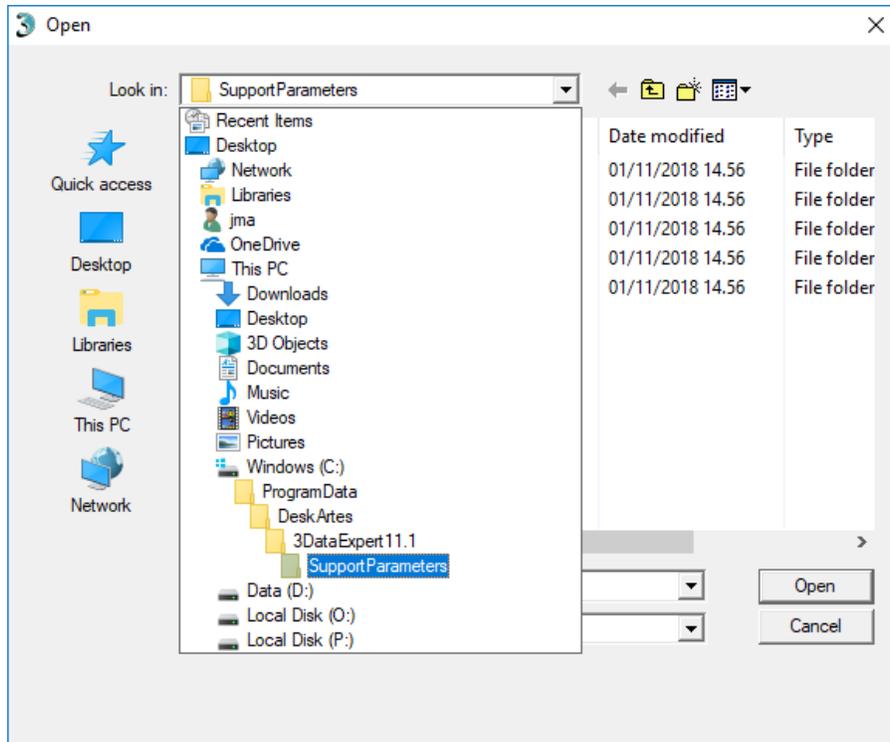
The first *Parameters* group in the Create supports toolbar includes file *Load* and *Save* options:



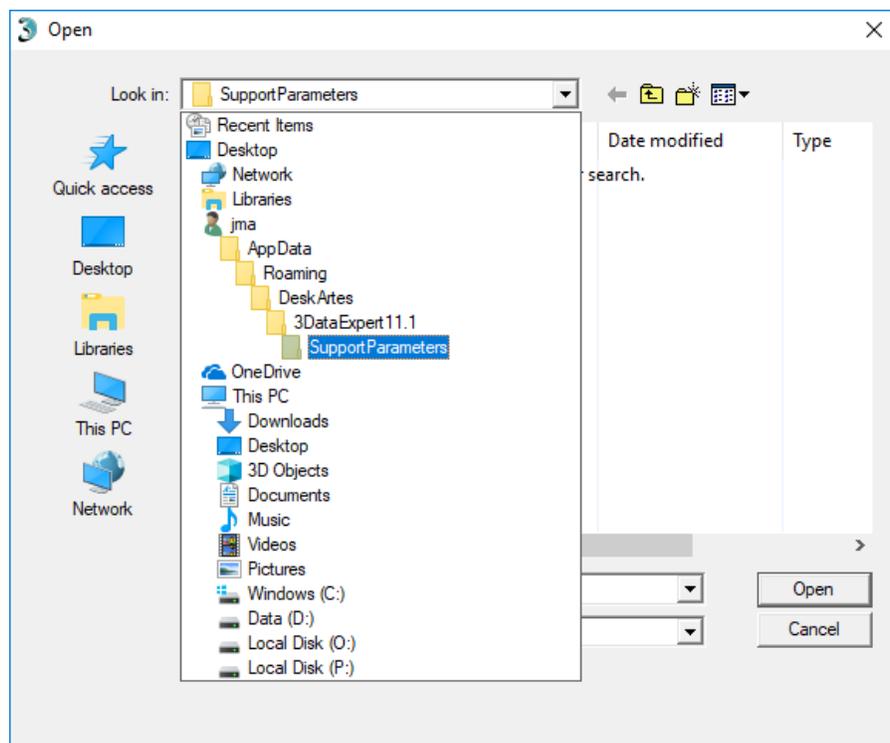
The predefined parameter files which come with the installation are installed in the System folder. These files cannot be overwritten without administrator rights.

- **press the *Load* button** when the ***Folder* is set to *System***

The file open dialog shows the path to the *SupportParameters* directory located under the *ProgramData* directory. These parameter files are accessible to all users of the given PC. You can store parameter files with new file names to this Folder to be accessed by all users.



- **press *Cancel*** not to load any parameter file at this time
- **change *Folder* setting to *Personal***
- **and press *Load* again**

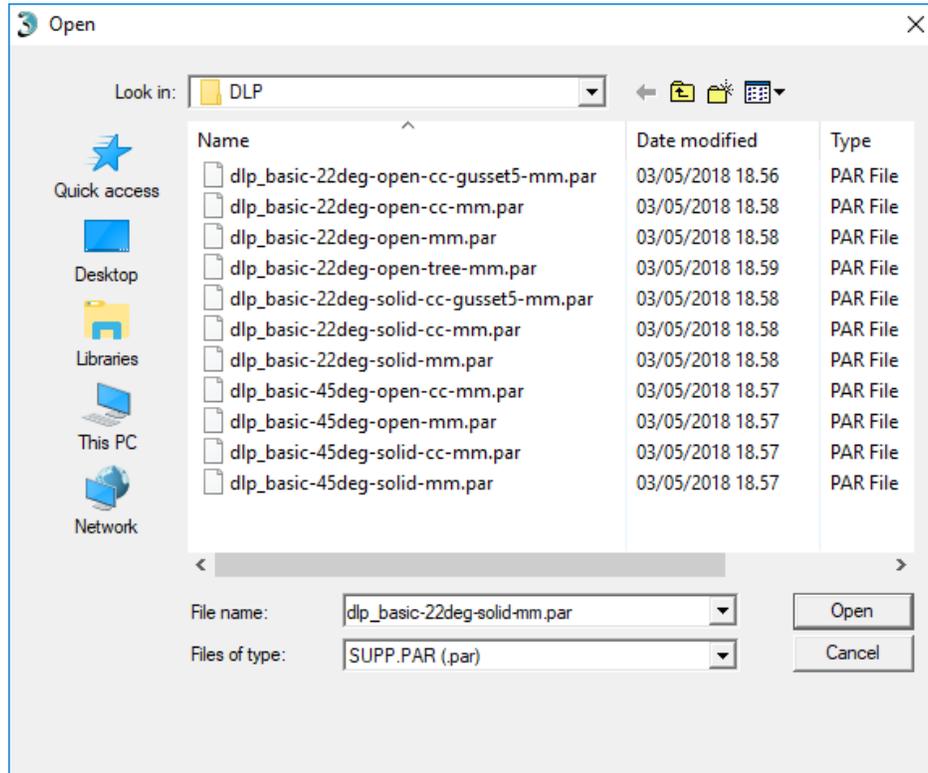


## Expert Series 12.1: Generating Supports for AM

---

Now the *SupportParameters* directory selected is under your personal user account. You can save and manipulate parameter files in this Folder for your personal use.

- **press *Cancel*** not to load any parameter files this time
- **the change the *Folder* setting to *Recent***
- **and press *Load***



This selection will take you directly to the last used support parameter directory.

- **press *Cancel*** not to load any parameter file this time

Similarly, the *Save* command with *Generic supports* command toolbars takes you automatically to the *System*, *Personal* or *Recent* directory for support parameter file save.

- **press *Cancel* in the *Create supports toolbar*** to exit the support generation without generating any supports

### Editing Common support parameters

The support parameter editing is done via a set of parameter dialogs. There are currently 6 different dialogs for different support types. The first, *Common parameters*, affect all support types. Three support specific parameter dialogs,

*Web, Chain and Point support parameters*, affect the different support types respectively. If set, the *Tube supports* and *Tube support parameters*, allows to convert all above support types to tubes. The Tubes supports are useful for medical applications, like titanium teeth implants. Tree support parameters currently define the parameters for manual Tree support addition (later an automatic conversion of all supports to Tree is implemented).

All support parameter dialogs include both image and textual explanation on each parameter. This makes it very easy to understand and modify the parameters for an optimal support result. In the sequel we will mostly rely on the parameter explanations given in the dialogs and concentrate to describe the effect of each parameter change.

The effect of a parameter change is best observed by pressing the *Apply* button on the bottom of each parameter dialog. Pressing the *Apply* button all parameter changes are updated immediately to the support preview in the graphics area. You can use different viewing commands (pan, zoom, rotate) when the parameter windows are open to take a better view to the part and supports.

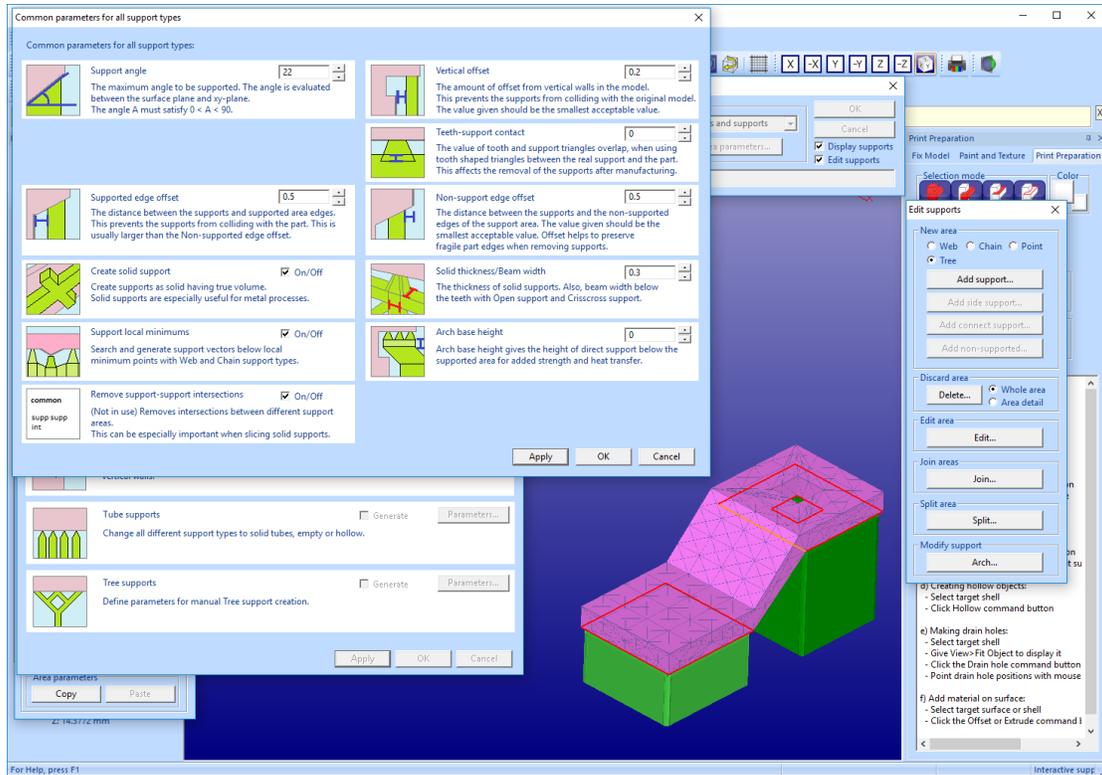
With low resolution screens it might be better to press OK on each parameter window and execute the *Create* command when the graphics window is fully visible. Some of the parameter dialogs are quite large and may hide the part and supports.

Now start again with the familiar *supp-demo1.3de* file:

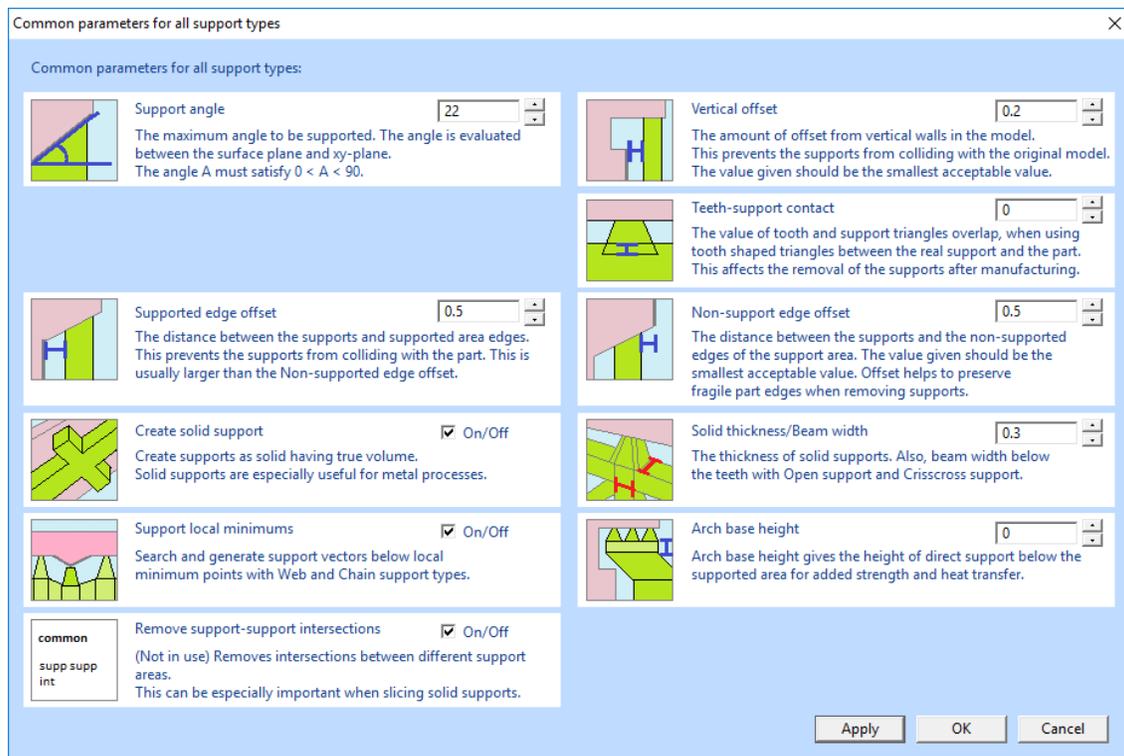
Once again,

- **open the *supp-demo1* model** and
- **give *Generic Support* command with *dlp\_basic-22deg-solid-mm.par* file.**
- **press *Create* button in *Create supports* window**
- **set *Create supports* setting to *Create all areas and supports*, if not already set**
- **press *Edit Parameters* button in the *Create supports* toolbar**
- **press *Common parameters* in the *Support parameters* dialog**
- **zoom in/pan** to have unobstructed view to the part and supports, as seen in the image below

# Expert Series 12.1: Generating Supports for AM

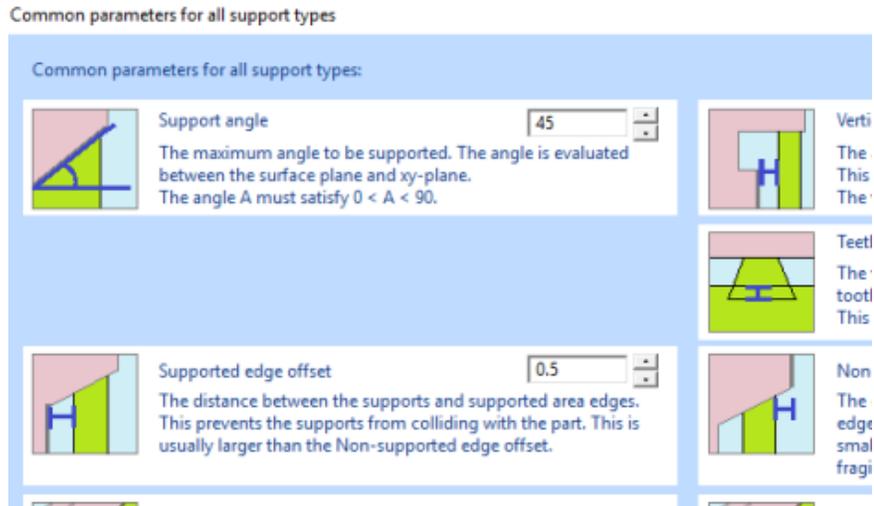


Your screen should now look more or less as seen above, the model and supports on the right hand side of the screen and support parameter dialogs on the left hand side of the screen.

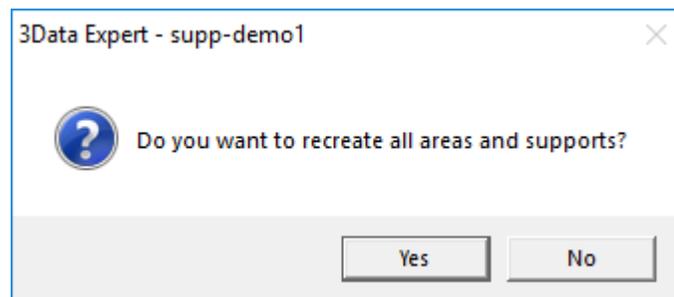


We will first demonstrate the effect of the most important *Common support* parameters. *Support angle* parameter defines which areas will be supported or not. If the angle between the surface and horizontal plane is bigger than the given angle, the part is considered self-supporting and no support is generated under the surface.

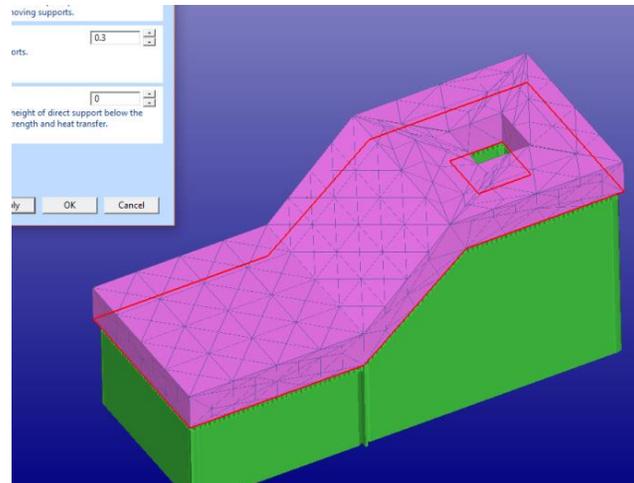
- **set the *Support angle* parameter to 45 degrees**



- **press *Apply* button** in the *Common parameters* dialog
- **press *Yes*** to the recreate request, if appears

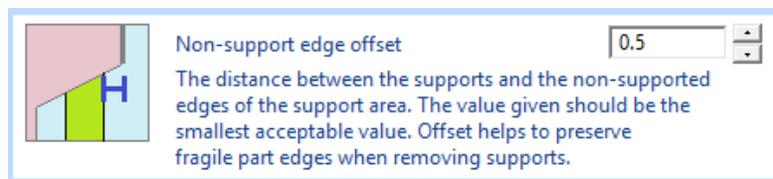


The result on the graphics changes immediately, as seen in the image below. The slanted area in the between the two horizontal areas is no longer considered self-supporting and it is added to support areas. Now there is just one large support area below the part. Minimum 45 degrees is a recommended support angle for most metal based AM processes.



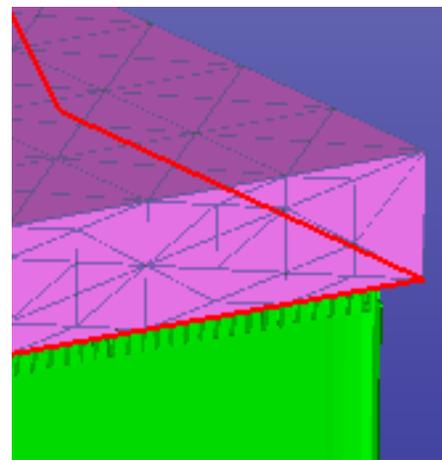
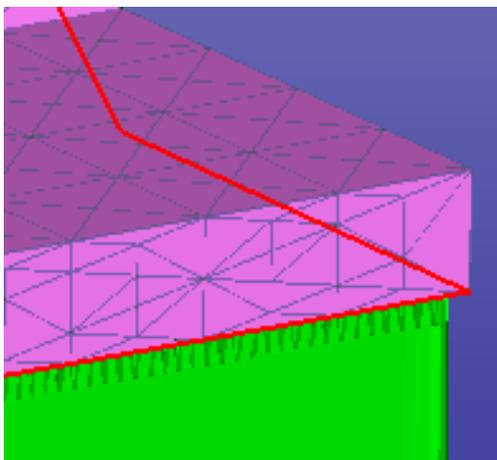
Now let's try the *Non-supported edge offset* parameter. This value defines how far the support following the area edge can be located from the outer edge of the support area. The default value is 0.5 mm,

- **change the *Non-supported edge offset* value to 1.0 mm**



- **press *Apply* and *Yes* to the recreate request**

The support structure outer profile is now moved further away from the support area edge. In the left hand image below we can see the 0.5 mm *Non-supported edge offset* result and on the right hand side the 1.0 mm offset. The edge offset value depends heavily on the receiving AM system, varying between 0.1 to 1.0 mm. Metal processes tend to require smaller values than other AM processes due to strong forces induced to the part during the cooling. If the support profile is too far from the edge, delamination or cracking may occur in the final part.

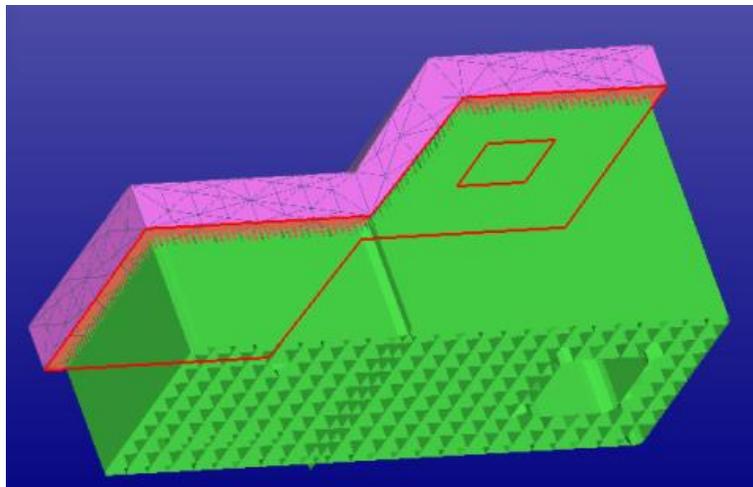


Finally, let's change all supports to open supports. By default, the supports generated with *dlp\_basic-22deg-solid.par* file are solid, i.e. form a watertight body. Some processes may not require solid supports and suffice with open support structure.

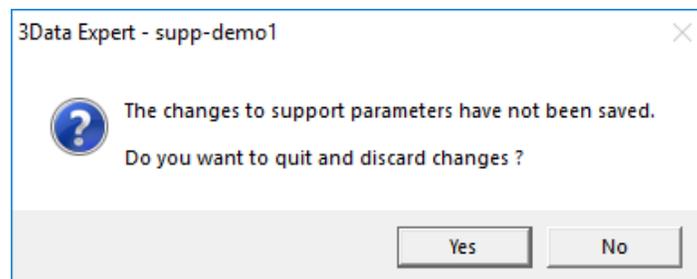
- **tic Off the *Create solid support setting***, as seen in below



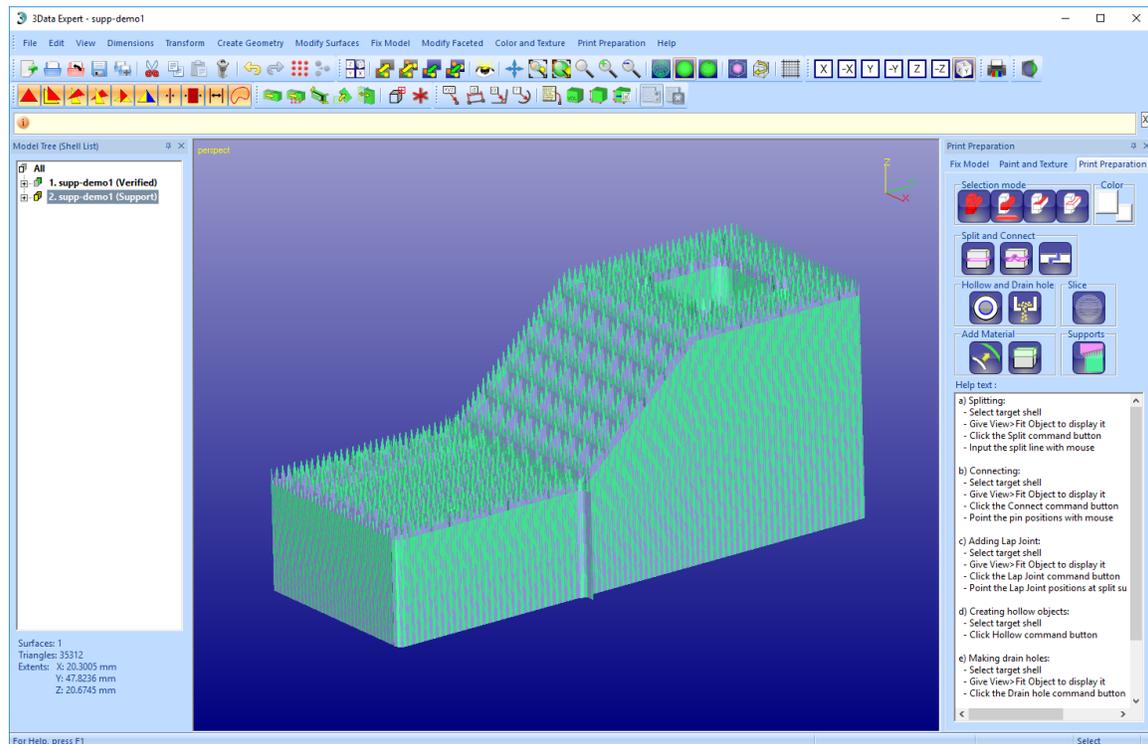
- **press *Apply* button**
- **press *OK*** to *Common parameters dialog*
- **press *OK*** to *Support parameters dialog*
- **rotate the result** to see the open support structure as seen in the image below



- **press *OK*** to *Create supports toolbar* to exit and store the supports to the Model Tree
- **press *Yes*** to the warning dialog, we do not want to save the parameters this time



You will soon see the open support structure stored in the Model Tree.



### Editing Web support parameters

Web supports create a regular grid shaped support structure below the supported area. The grid size, thickness and teeth size can be tuned with the support parameter dialogs. Also, a profile which follows the support area edges at a given distance can be generated around the grid support structure. The latest version allows to convert Web and Chain supports into crisscross structure.

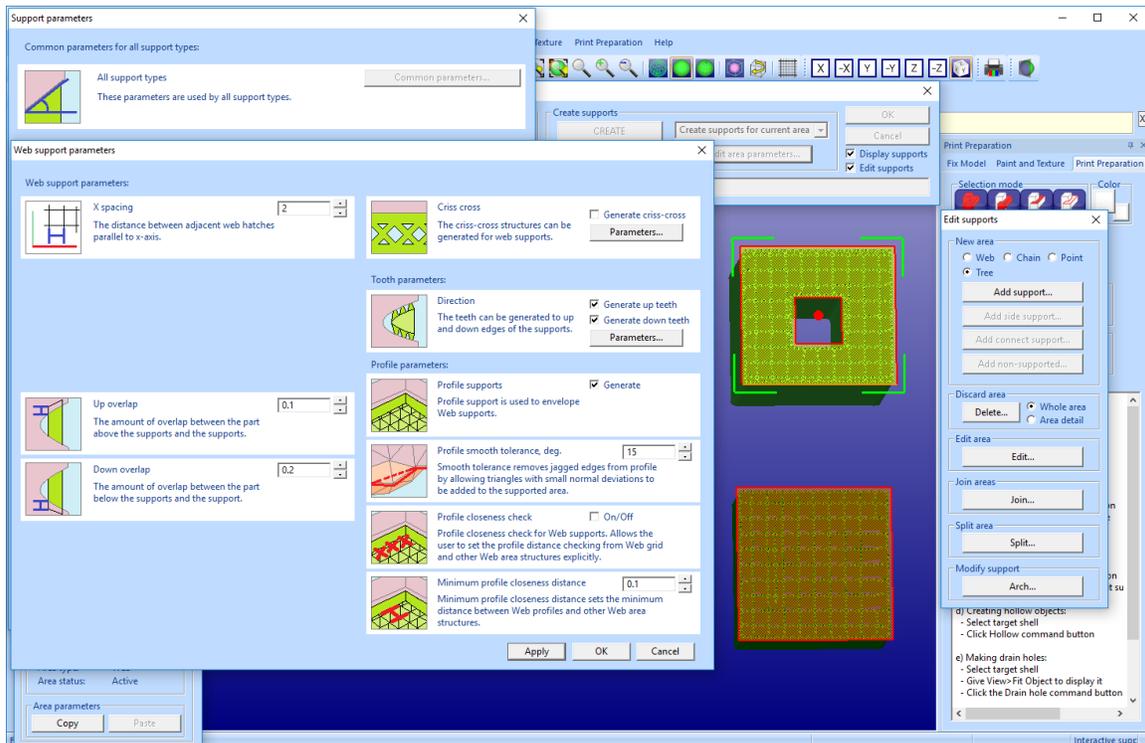
Now start again with the familiar *supp-demo1.3de* file. Once again,

- **open the *supp-demo1* model** and
- **give *Generic Support* command with *dlp\_basic-22deg-solid-mm.par* file.**
- **press *Create* button in *Create supports* group**
- **zoom into** the higher end of the support structures, make sure you have unobstructed view to the model higher support area (with a hole)
- **select the *2. support area* from the area list with *LMB***
- **set *Create supports* setting to *Create supports for current area*, if not already set**

## Expert Series 12.1: Generating Supports for AM

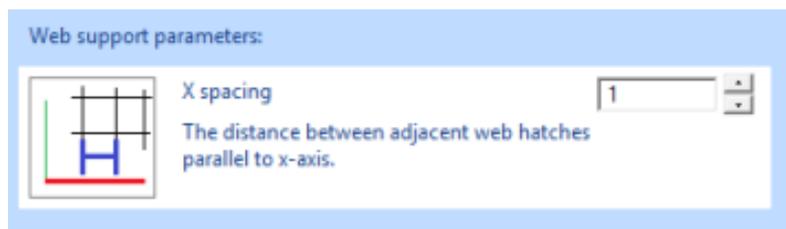
- press **Edit area parameters** button in the **Create supports** group (in green circle below)
- press **Parameters** button in the **Web supports group** in the **Support parameters** dialog

Your screen should now look more or less as seen below, the model and supports on the right hand side of the screen and support parameter dialogs on the left hand side of the screen. You can pan, zoom and rotate the view if necessary.



First we will modify the density of the Web support grid.

- change the **X spacing** to **1 mm** (applies currently to Y spacing too)

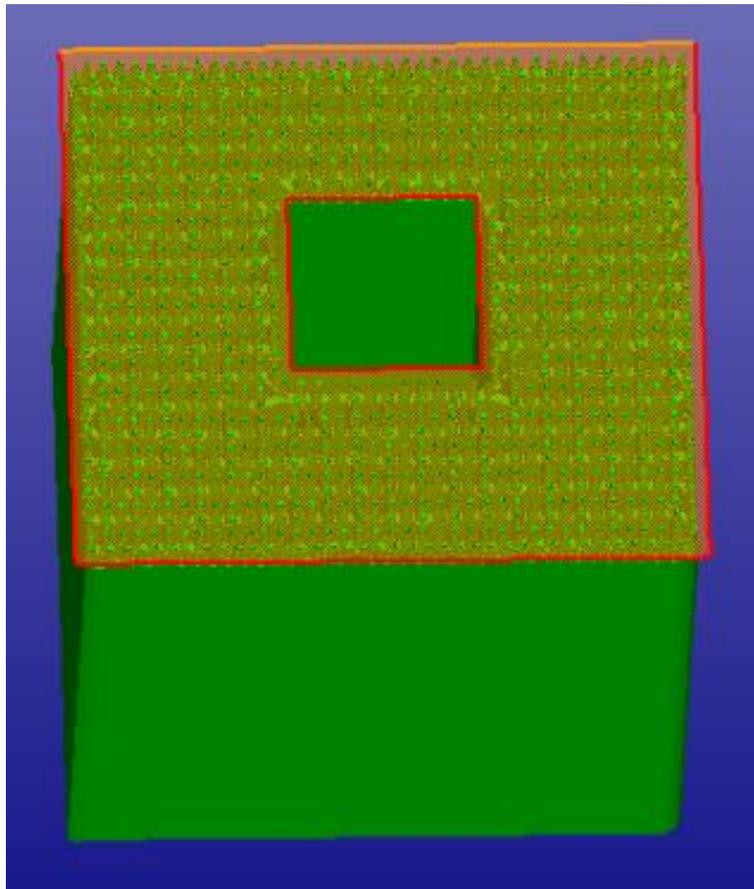


- press **Apply** button in the bottom of the Web support parameter dialog

## Expert Series 12.1: Generating Supports for AM

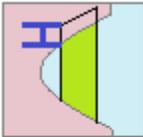
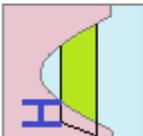
---

The support grid immediately becomes more dense. Now the grid will provide a stronger support which may be necessary to avoid bending of the part or to dissipate heat from the part during the AM process.



Let's now change the how the teeth intersects the part.

- **set *Up overlap* and *Down overlap* to 0.0 mm**

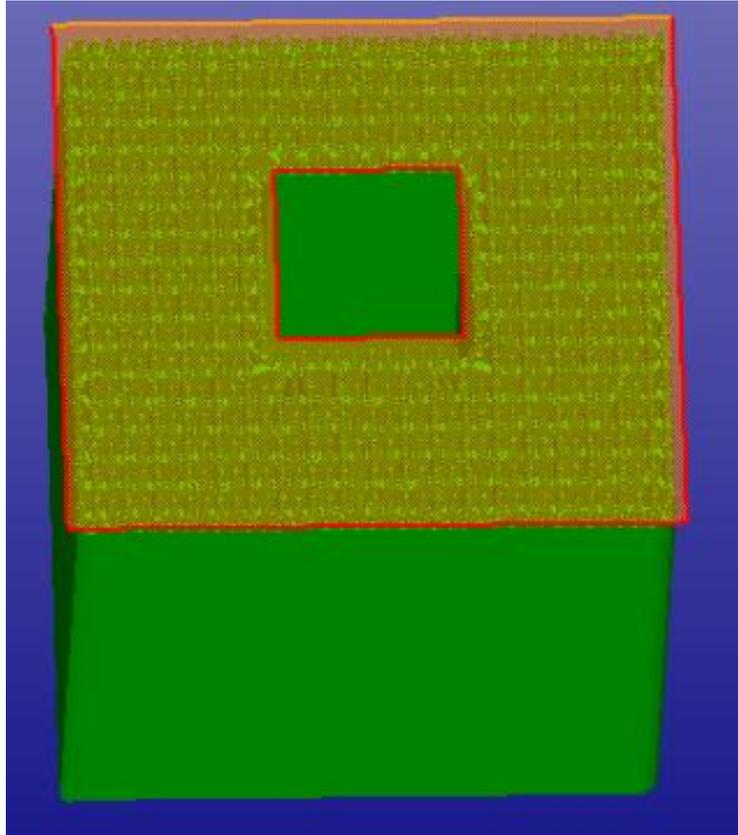
	<b>Up overlap</b> The amount of overlap between the part above the supports and the supports.	<input type="text" value="0"/>	<input type="button" value="▲"/> <input type="button" value="▼"/>
	<b>Down overlap</b> The amount of overlap between the part below the supports and the support.	<input type="text" value="0"/>	<input type="button" value="▲"/> <input type="button" value="▼"/>

- **press *Apply* button**

## Expert Series 12.1: Generating Supports for AM

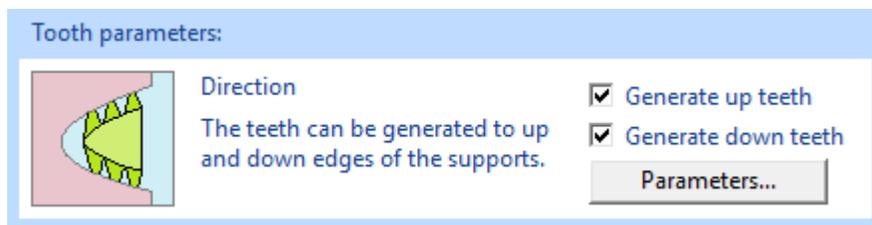
---

The support teeth do not anymore intersect the part surface. With some processes it may be necessary for the supports to intersect the part to get strong enough contact between the part and the supports. On the other hand, some processes may produce better result if the support just touch the surface, or even with small distance if overlap value is set to  $< 0.0$  mm (i.e. negative value).



Let's now make a sparser grid with bigger teeth.

- **first set the *X spacing* to 3 mm**
- **then press *Apply* button** in the ***Web support parameters*** dialog to update the grid
- **then press the *Parameters* button** in the ***Tooth parameters*** group

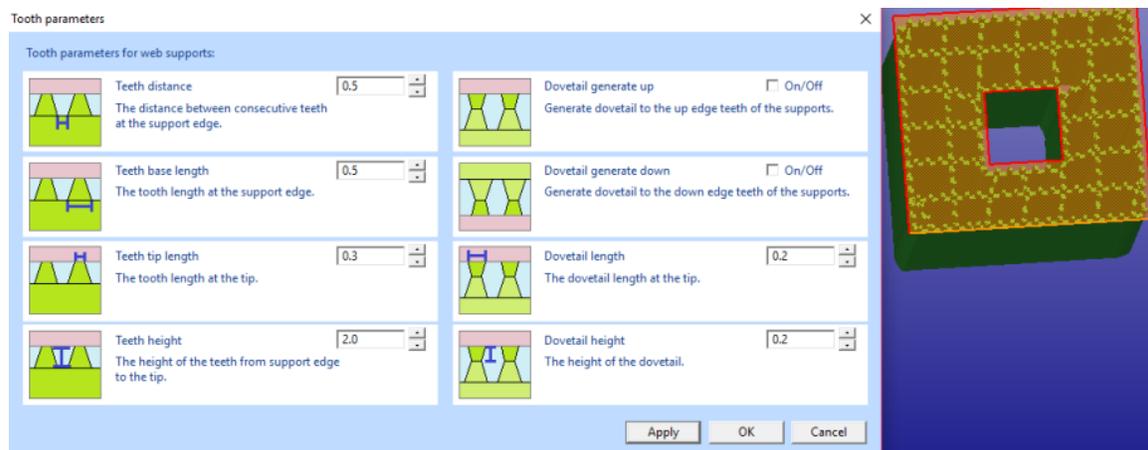


You should have the screen as shown below. The Tooth parameter dialog is visible and the more sparse grid is shown on the graphics area.

Now we will fill in new values to teeth parameters.

- **set *Teeth distance*** to **0.5 mm**, as seen in the image below
- **set *Teeth base length*** to **0.5 mm**
- **set *Teeth tip length*** to **0.3 mm**
- **set *Teeth height*** to **2.0 mm**
- **press *Apply*** in the ***Teeth support parameters*** dialog

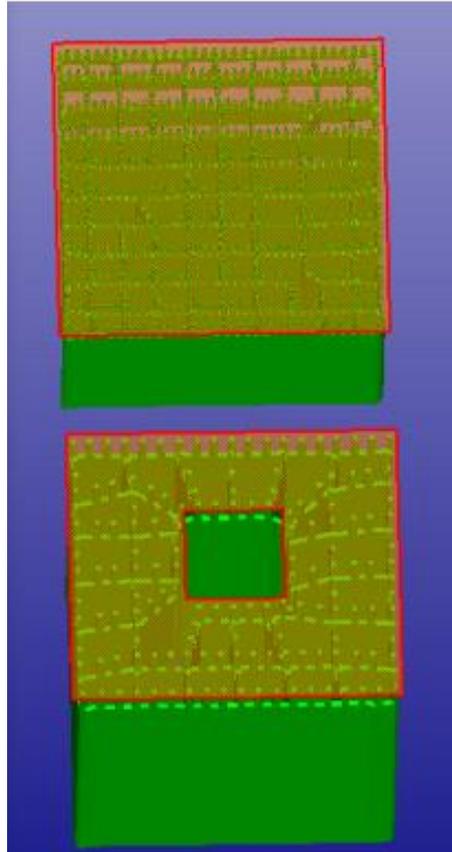
You should see updated teeth structure on the sparse grid structure. The software automatically calculates and modifies the most suitable teeth distance against the grid size and generates as dense teeth as possible.



You can now exit all support parameter dialogs by pressing OK. This will store the modified parameters for the area 2. support area we selected when starting the parameter editing.

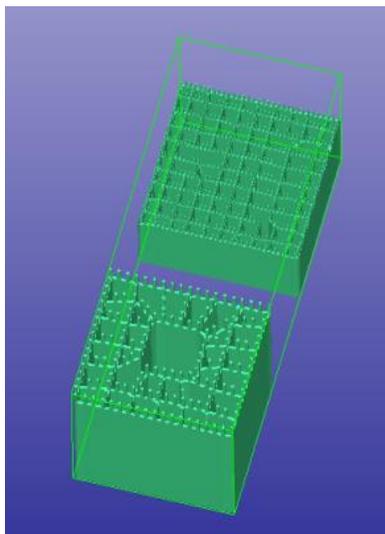
- **press *OK*** to Tooth support parameters dialog
- **press *OK*** to Web support parameters dialog
- **press *OK*** to Support parameters dialog
- **zoom and rotate** to see both support areas for the part

The two support areas now have different support structure parameters applied. In the start we selected the 2. *support area* and pressed the *Edit area parameters* button. All changes are now done only to the area 2. and the area 1. has remained in the original state.

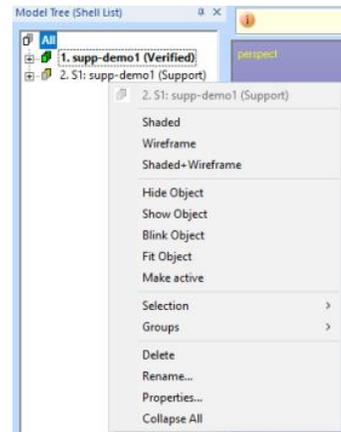


- **press OK** to save the supports into Model Tree
- **press Yes** to discard support parameter changes

With 3Data Expert it is easy to define different support properties for different areas in the part.



**Note:** You can also save the current support version to the Model Tree by pressing **Save** button in the Create supports dialog. This will not exit the support generation, it only stores the current supports in the Model Tree for subsequent comparisons. The saved result is named with “Sn:” prefix and made passive not to become visible. After exiting the support generation you can make the saved support object visible with *RMB Make active* command.

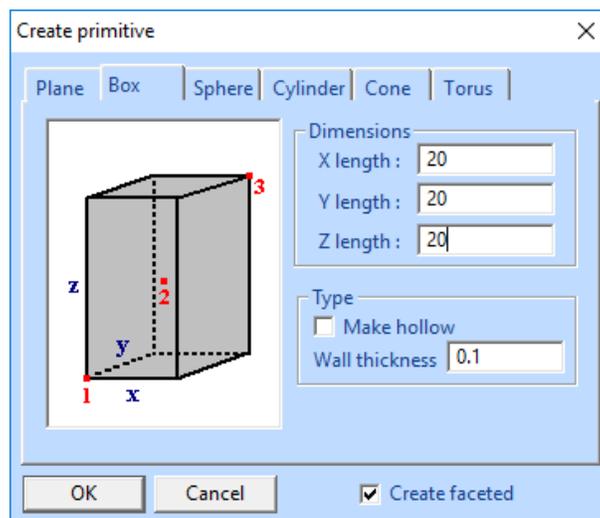


### Editing Chain support parameters

Chain supports are used to support thin or sharp downfacing edges in the part. With the support parameters we can tune the teeth size and cross plate distance and cross plate distance from the support top.

To investigate Chain support parameters we will first generate a simple cube and rotate it for Chain support generation at the sharp downfacing edges. Also, we will Refine the box for some manual Chain support additions.

- **empty the *Model Tree*** with File > New command
- **create a box geometry with *Create Geometry > Box* command** with the parameters below (20x20x20mm box):

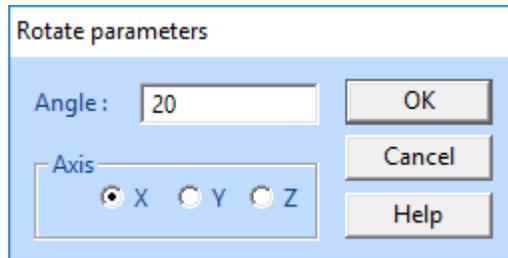


- **press *OK*** to create the box into the Model Tree

## Expert Series 12.1: Generating Supports for AM

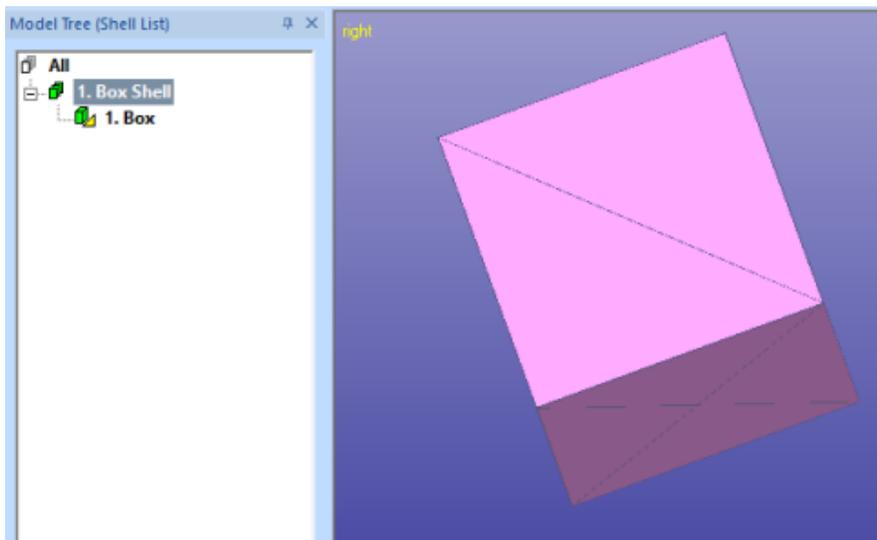
---

- select the **Box Shell** in the Model Tree
- give command **Transform > Rotate > Numeric to** rotate along X axis 20 degrees, press **OK**

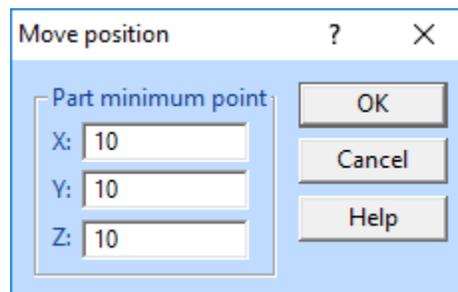


- give command **Transform > Rotate > Numeric to** rotate along Y axis 20 degrees, press **OK**

You should have the following box in the Model Tree (viewed from Y axis direction):



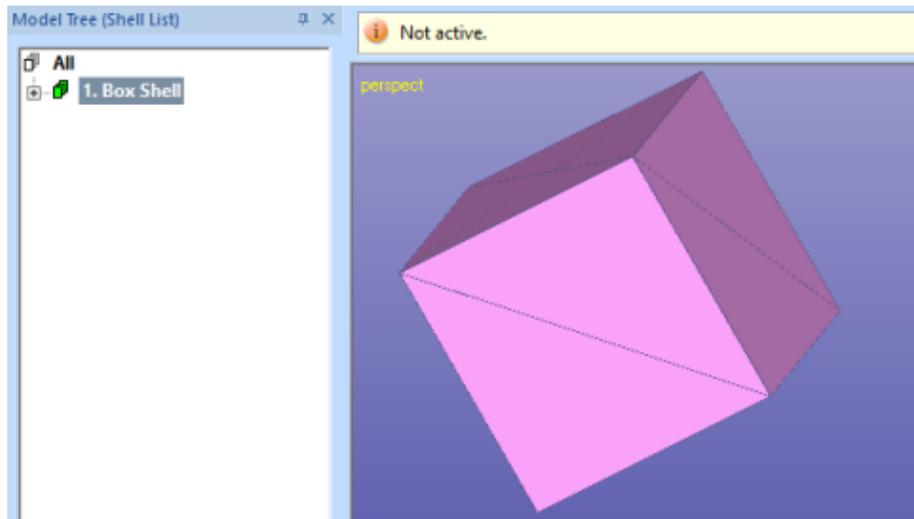
- move the part to the positive space with **Print Preparation > Move to Platform > Move positive** command using the parameters below



## Expert Series 12.1: Generating Supports for AM

---

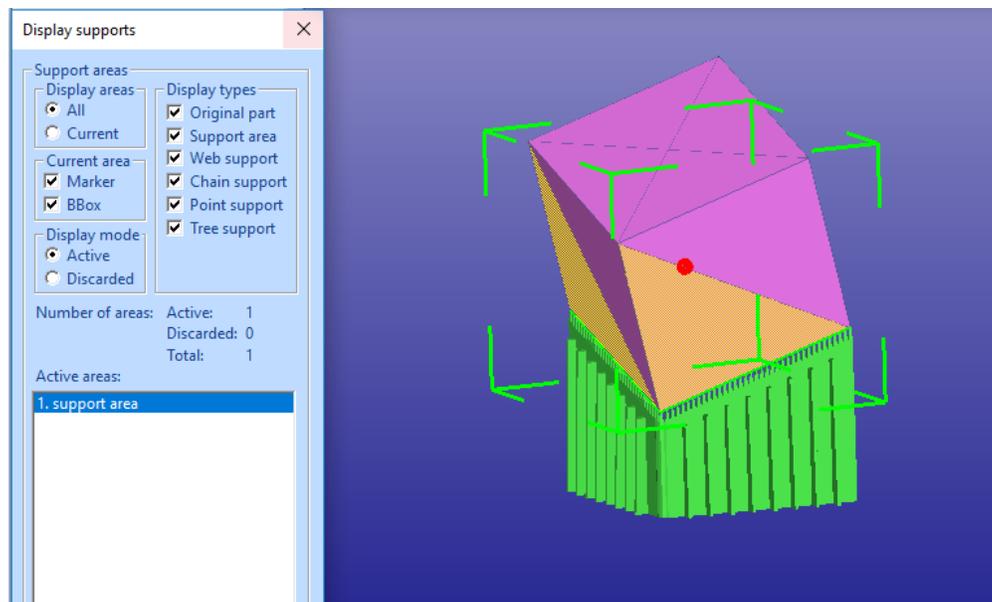
You should now have the box in the right place for support generation.



Let's now generate the *Chain supports* below the sharp edges of the box.

- select the **Box Shell** model in the Model Tree
- give command **Print Preparation > Supports > Generic Supports**
- select the **dlp-basic-22deg-solid-mm.par** file for the supports
- press **Create** button in the *Create supports* group

You will see the support result in the support area list (one Chain support area) and in the graphics area:

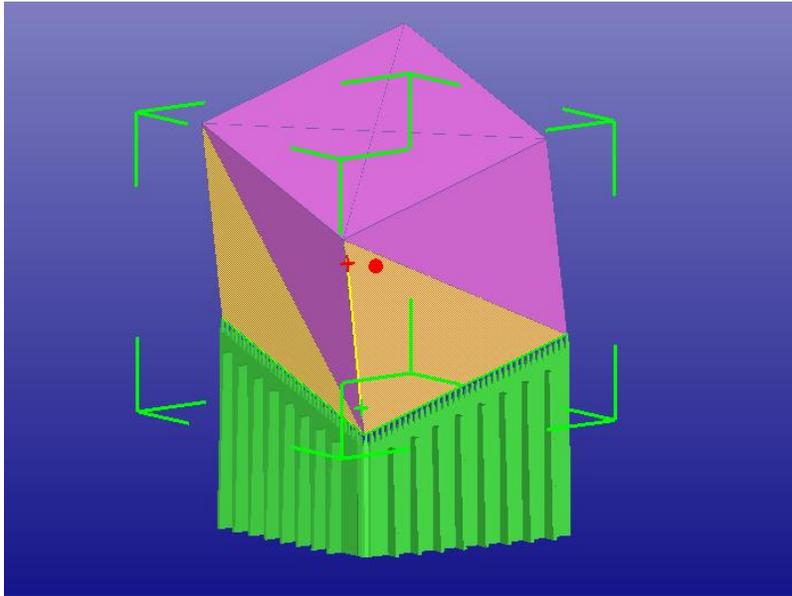


## Expert Series 12.1: Generating Supports for AM

---

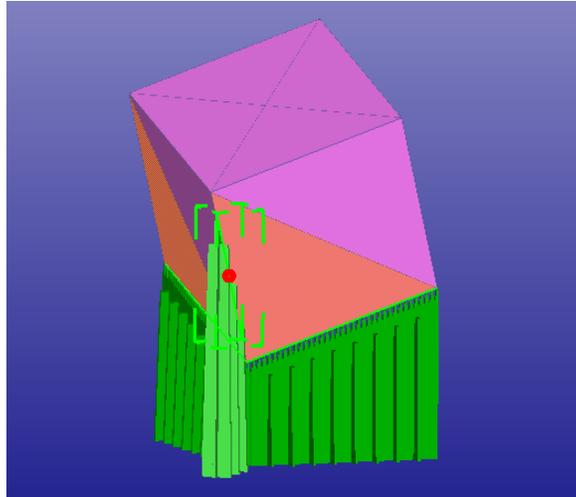
Before going to edit the *Chain support* parameters, let's add some stability to the build by adding manual support to the third slanted edge.

- now select the **Chain** type in the **New area** group,
- then press **Add support...** button to select two points from the box edge as seen below. Makes sure you select the points as closet to the edge as possible. If you make a mistake, you can **Cancel** each step with the **ESC** key.



- when ready press **RMB** to save the new Chain area to the support area list and to generate the support geometry

Both areas are now regenerated and the result is displayed on the screen.



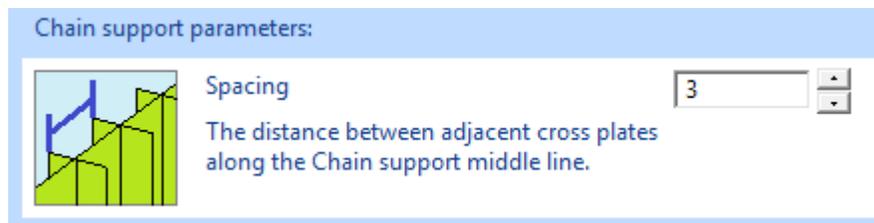
With this example we show how you can add more support to increase the stability of the part build.

Now rotate and zoom the model on the graphics area so that you will have an unobstructed view to at least one of the support areas. Then

- **press the *Edit Parameters* button** in the *Parameters* group to start editing parameters for all support areas at once
- **press the *Parameters* button** in the *Chain supports* group (in green circle below)



- **change the value of the *Spacing* to 3 mm** in the *Chain support parameters* dialog



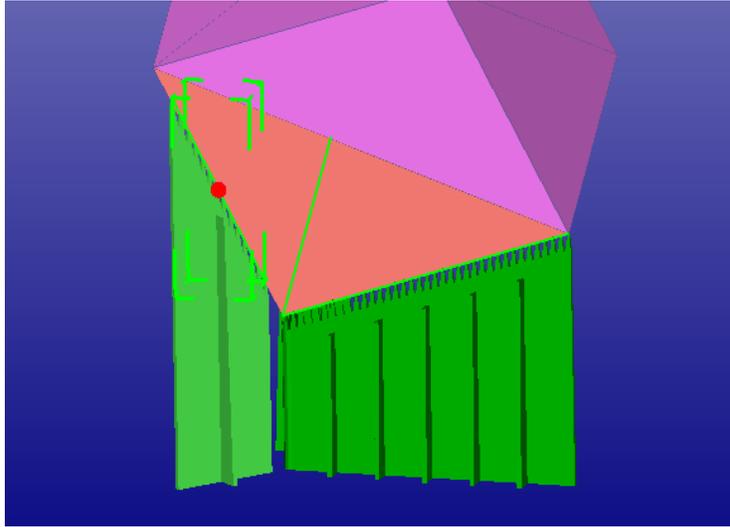
- **press *Apply*** in the bottom of the parameter dialog

The distance of the cross plates at the Chain support has increased. This parameter allows to create stronger (more dense) or weaker (more sparse) Chain supports, depending on the receiving AM process. Especially metal processes should have dense enough spacing to avoid support bending while

## Expert Series 12.1: Generating Supports for AM

---

creating them. Similarly, increasing the *Cross length* parameter in the parameter dialog will add more stiffness to the supports.

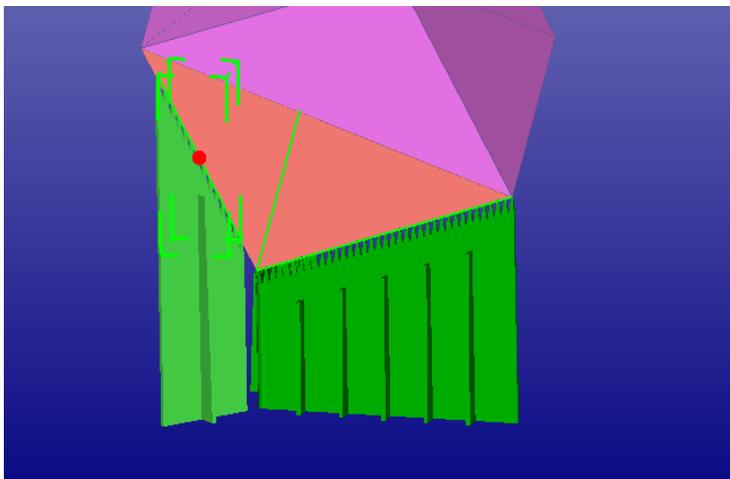


- now change the *Distance from support top* value to 2 mm



- press **Apply**

The distance of the *Cross plate top* is now increased.



The value depends once again on the receiving AM process. Some processes can build higher (DLP, stereolithography) thin wall supports than some other processes (metal) without bending them. The smaller the distance from the top the stronger the support will be.

## Expert Series 12.1: Generating Supports for AM

---

*Up overlap*, *Down overlap* and *Tooth parameters* work the same way as the corresponding support parameters with *Web support* type, described in the previous chapter.

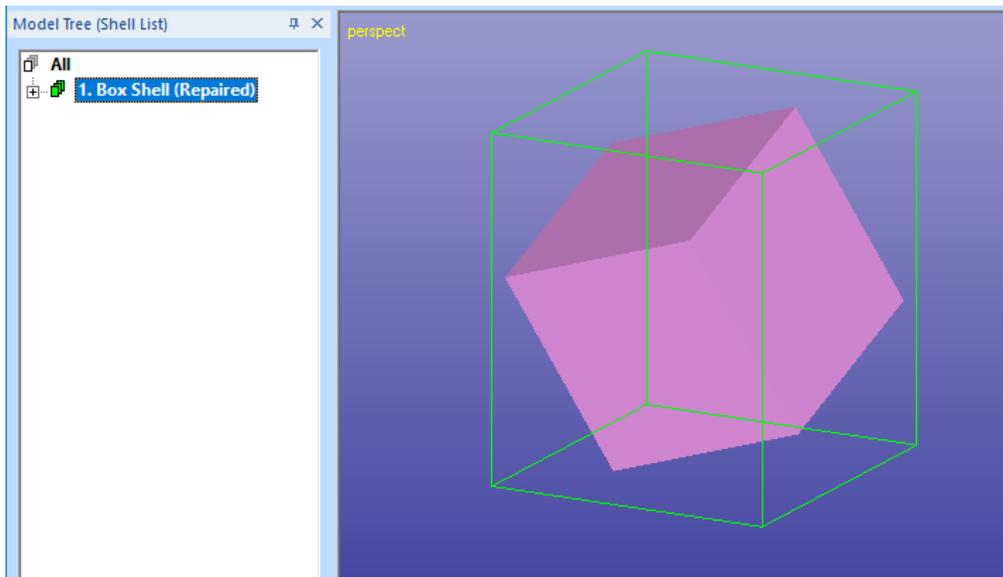
*Note:* When you are adding the points for Chain support, make sure you always pick them from downfacing surfaces. Otherwise the results may be unpredictable.

### Editing Point support parameters

We will continue Point support parameter editing with the model created in the previous example for Chain supports.

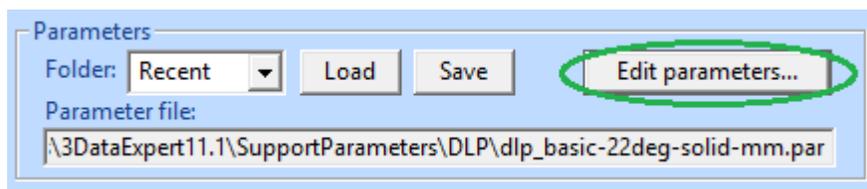
- **restart with supp-demo2.3de file**

You should have only the box in the Model Tree now:



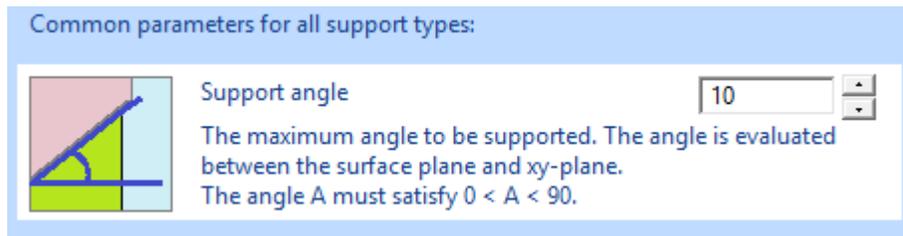
Now start the support generation with Generic Support command

- **give command *Print Preparation* > *Supports* > *Generic supports***
- **open *dlp\_basic-22deg-solid-mm.par* file**
- **open the *Support parameters* dialog via *Edit parameters* button in the *Create supports toolbar***

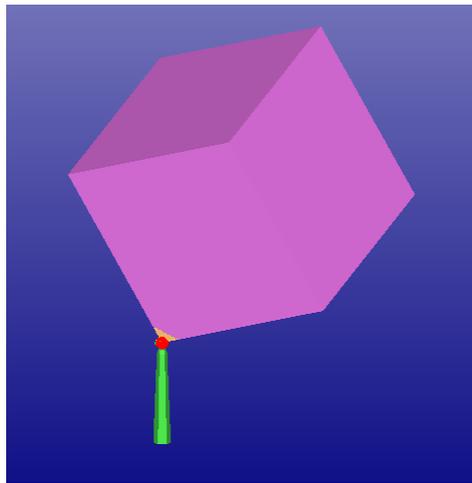


- **press *Common parameters* button in the *Support parameters* dialog**

- **edit *Support angle* parameter to value *10 degrees***



- **press *Apply* button** in the bottom of the *Common parameters dialog*



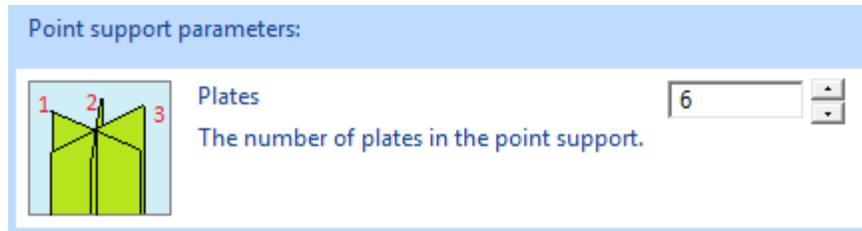
You will now see a single *Point support* below the Box Shell. Due to the *10 degree Support angle* no other area below the Box Shell is considered to need supports this time.

We will now go through some basic *Point support* parameters.

- **press *OK*** to close the *Common support parameters dialog*
- **in *Support parameters dialog* press the *Parameters* button** in the *Point supports* group

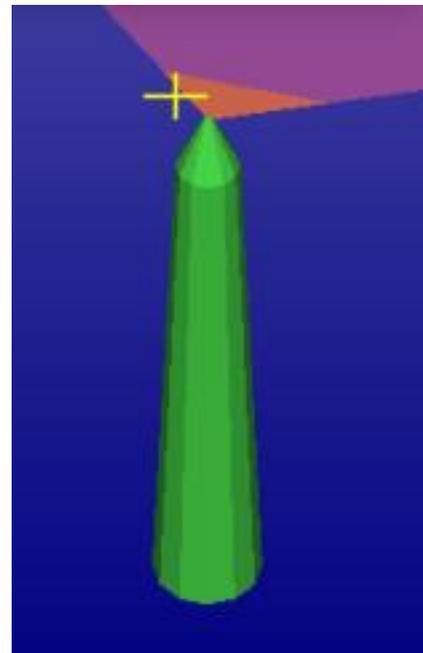
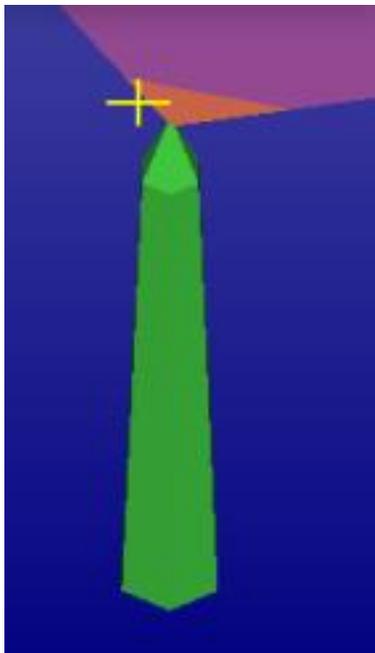


- **first change the number of *Plates* parameter to *6***



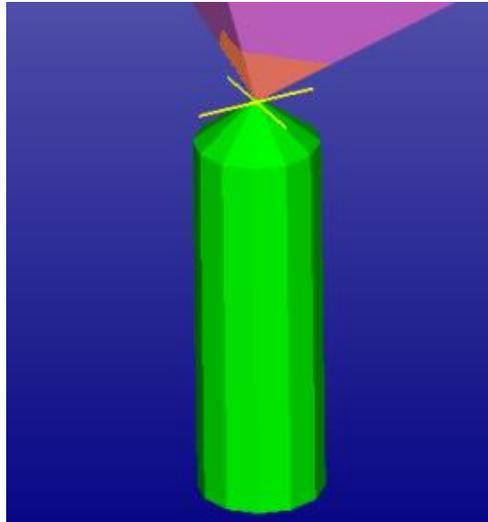
- **press *Apply* button**

You will see the Point support shape to change in the graphics area. In the right hand image we you can see the original Point support with 3 plates. Because the *Solid supports* in *On* in the *Common support parameters* the support is a solid tube instead of three open plates crossing each other. When we increase the number of Plates, the point support tube becomes smoother due to more plates, i.e. more accurate subdivision of the tube outer surface.

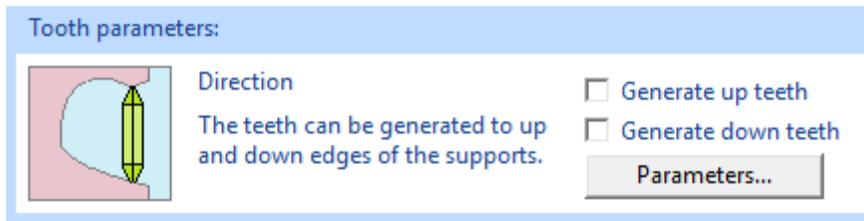


- **now change the *Plate length* to 3mm**
- **press *Apply* button**

The Point support is now 3 mm in diameter.



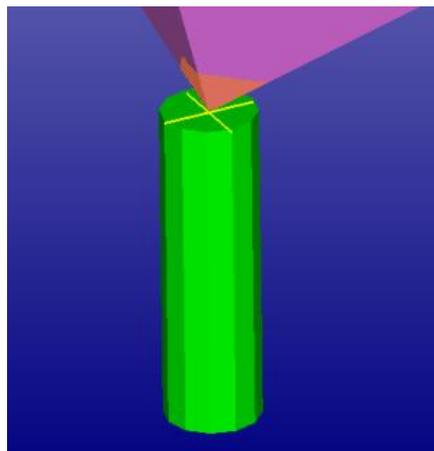
- now turn **Off *Generate up teeth*** and ***Generate down teeth***



- **press *Apply* button**

You will see the tooth on top of the Point support disappear, as seen in the image below.

*Tooth parameters, Up overlap and Down overlap* parameters work in the same fashion as was described in the *Web support* parameter editing chapter previously.



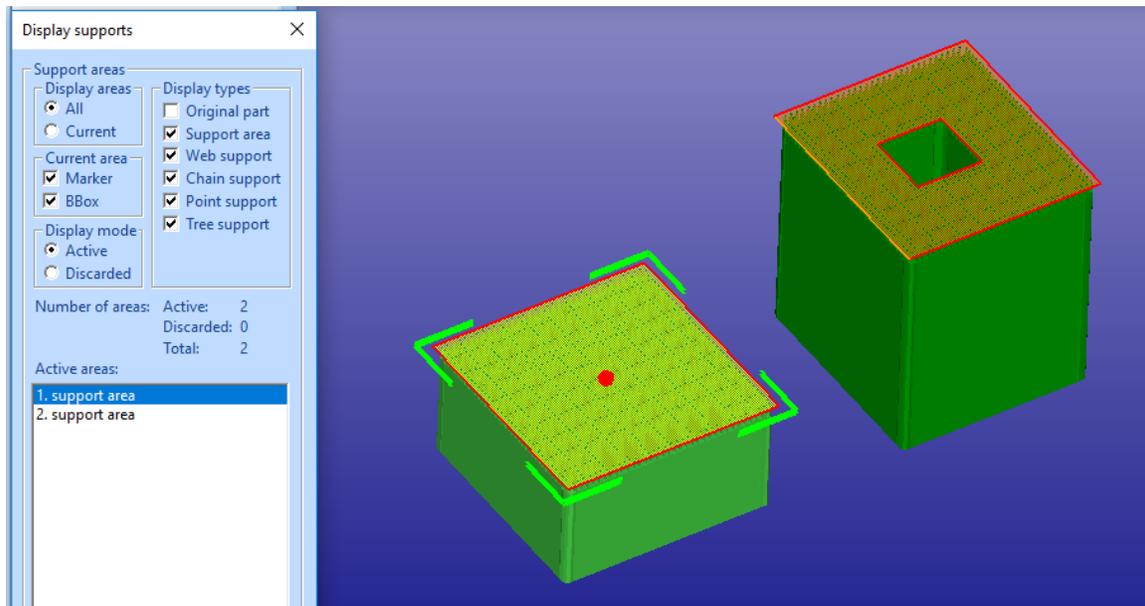
### Generating Tube supports

Tube supports allow you to convert the Web, Chain and Point supports to small tubes. This kind of supports have several uses, like supporting teeth implants for medical applications.

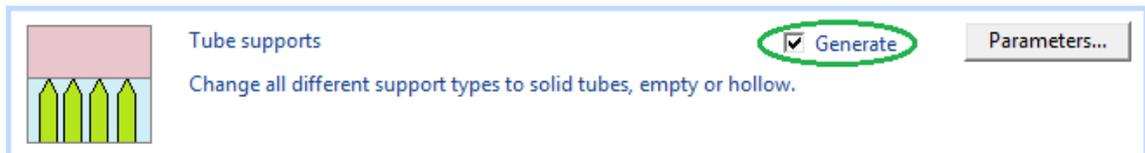
Start again with the original `supp-demo1.3de` file. Once again,

- open the ***supp-demo1*** model and
- give ***Generic Support*** command with ***dlp\_basic-22deg-solid-mm.par*** file.
- press ***Create*** button in ***Create supports*** group

You will soon see the familiar support structure to appear on the graphics area:

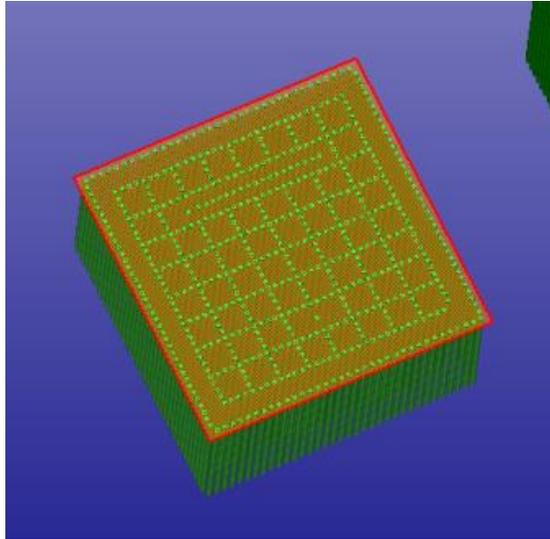


- Now press the ***Edit Parameters*** button
- Then tick ***On the Generate*** selection in the ***Tube supports*** group at the bottom of the dialog

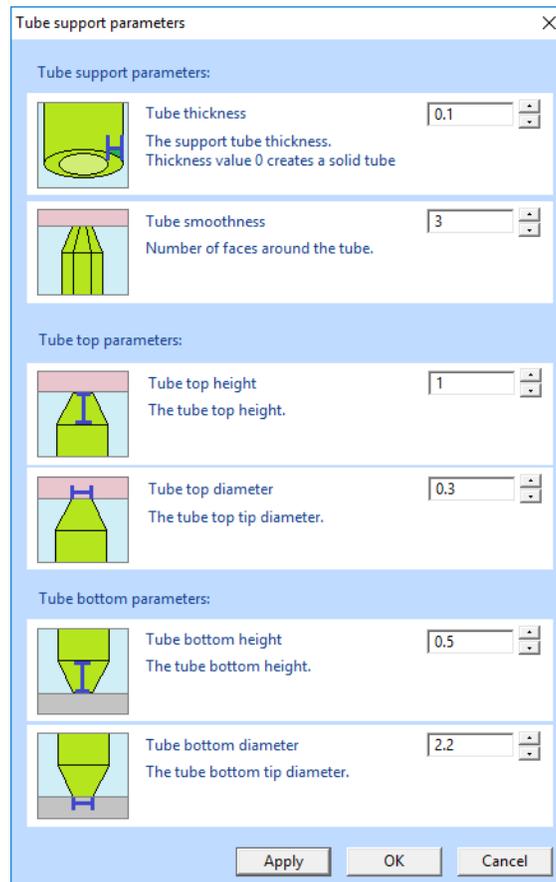


- Press ***OK*** to close the ***Support parameters*** dialog
- Press ***Create*** button in the ***Create supports*** group

Soon you will see the Web, Chain and Point supports changed to tube like structures, as seen in the image below. The Tubes are generated more or less to the same locations where the teeth to the first support were located. The Tubes are disconnected from each other and also stretched to the wat bottom.

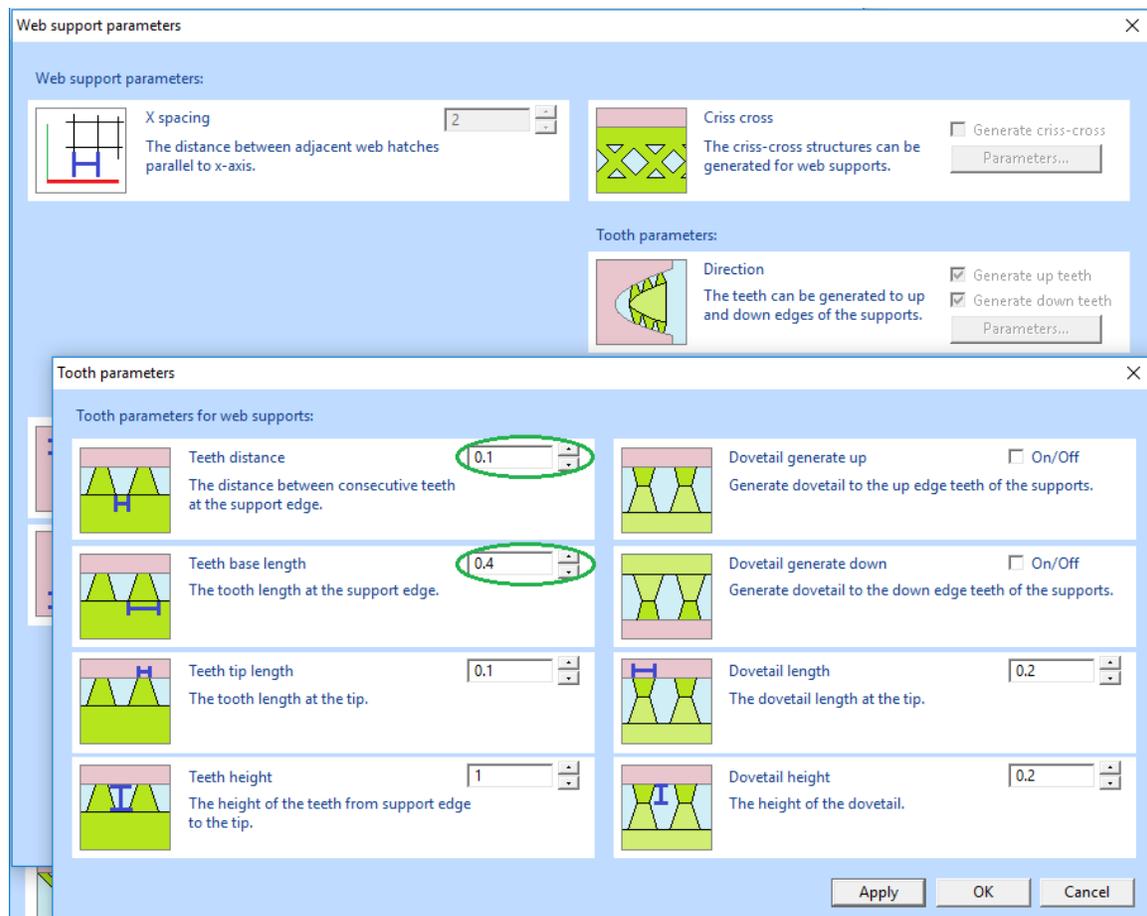


The tube parameters are set through the *Parameters* button in the *Tube supports* group. The parameters allow you to change the *Tube wall thickness*, *Tube smoothness*, and *Tube top and bottom size parameters*, as seen in the image below. If the *Tube thickness* is set to 0, a solid tube is generated (closed at both ends).



## Expert Series 12.1: Generating Supports for AM

The Tube diameter and distance is set according to the teeth size defined for the Web supports. Teeth base length will define the diameter of the tube and Web supports Teeth distance will define the average distance between tubes (the distance value is modified to fit optimal amount of teeth between the Web grid points and Profile edge points). You must be careful not to use too big *Tube thickness* when compared to the  $Diameter = Teeth\ base\ length$  or *Top/Bottom diameter*, otherwise you might generate self-intersecting support structures.



You may now start again the Tube support generation and change the different values to see how it affects the generated Tube supports.

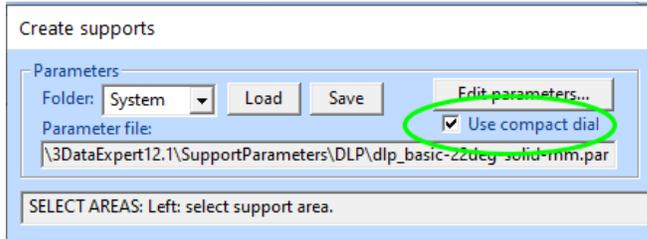
- **When ready, press *OK*** to the *Create supports toolbar* to save the tube supports into the Model Tree

**Note:** You can also use the *compact Support parameters* dialog when you get more familiar with the system and its parameters. With *compact Support*

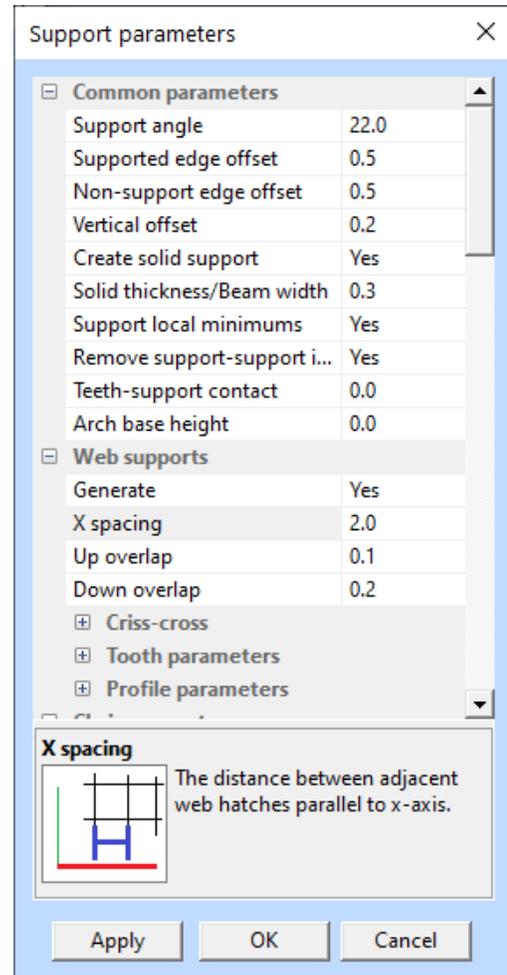
## Expert Series 12.1: Generating Supports for AM

*parameter* dialog less graphics area is covered and 3D models can be more easily viewed.

Compact Support parameter dialog is used when **Use compact dial** is *On*.



In the image right the X spacing is selected for editing. When a parameter is selected in the dialog, the lower window shows the explanation text as well as the parameter image.

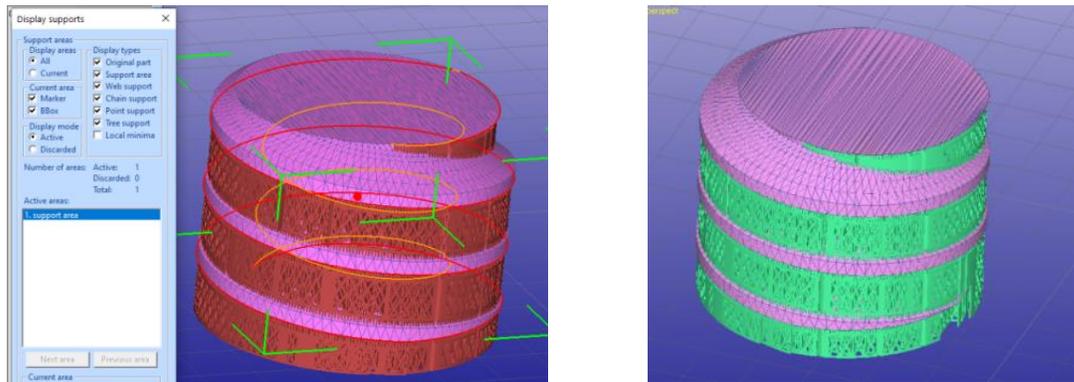


## Generic Support improvements with 12.1 version

Generally, there have been several improvements within the Generic Support command internal functionality: better area filtering, vertical wall separation, smoother profile edges etc. Additionally, some new functionality has been added:

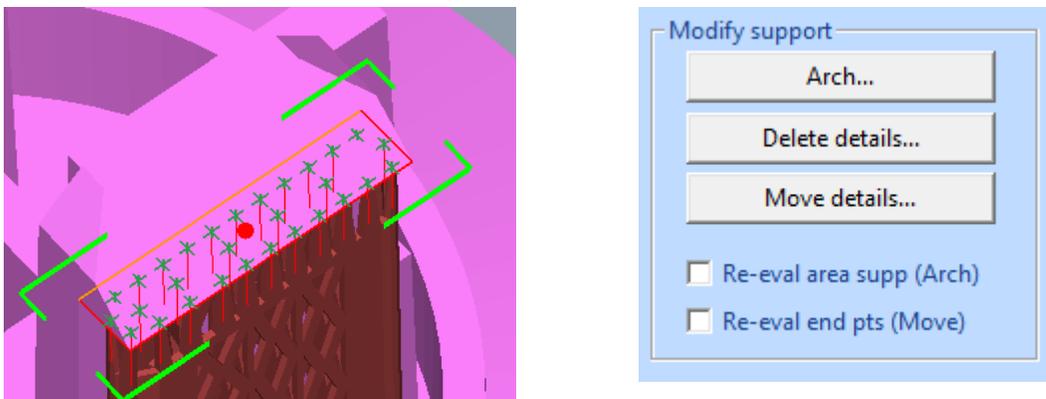
### Helical area supports

Helical parts and areas can now be supported normally without splitting the area in two.



### Move details end point re-evaluation

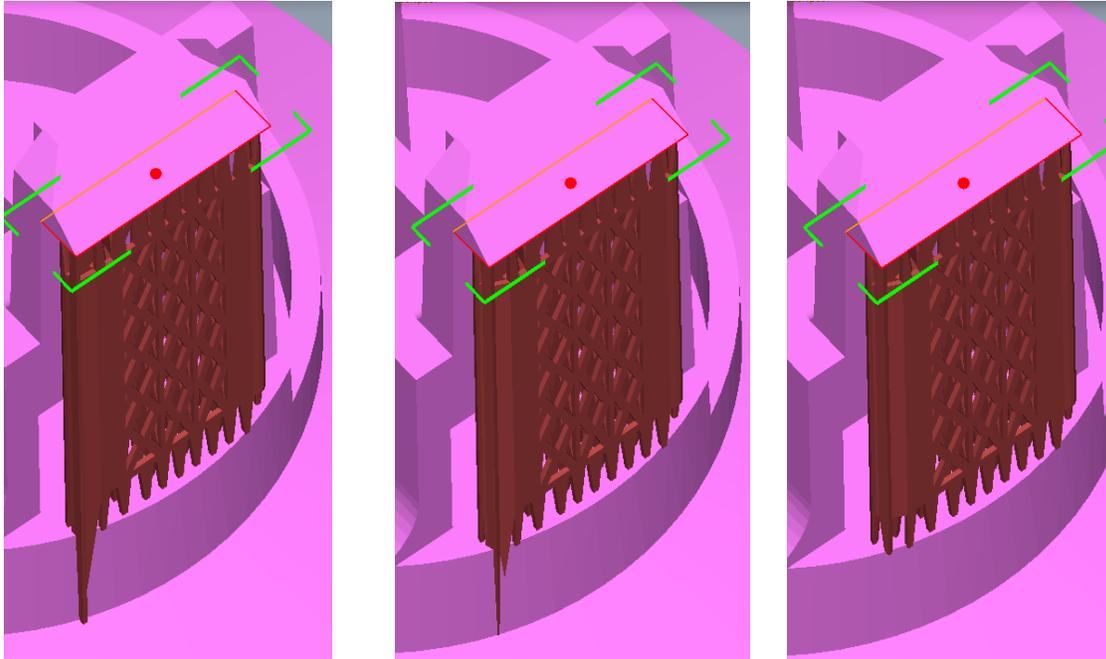
The Move details command in Edit Supports dialog has been improved. The support point selection is now less crowded and an option to re-evaluate the end points of the moved support point has been added. In the left hand image below the available selection points area show with green crosses. In the right hand image the new *Re-eval end pts* parameter is shown.



## Expert Series 12.1: Generating Supports for AM

---

The effect of re-evaluating the end points is shown below. The automatic support generation produces the result in the leftmost image below. The support goes along the vertical support at the bottom of the image. To create better surface quality the support point at the area border can be moved manually. The middle image shows the movement without re-evaluating the moved support vector end points. There is still some overlapping of the supports and the part in the lower areas. When the *Re-eval end pts* is used in the right hand image the support does not affect the vertical surface in the bottom anymore.

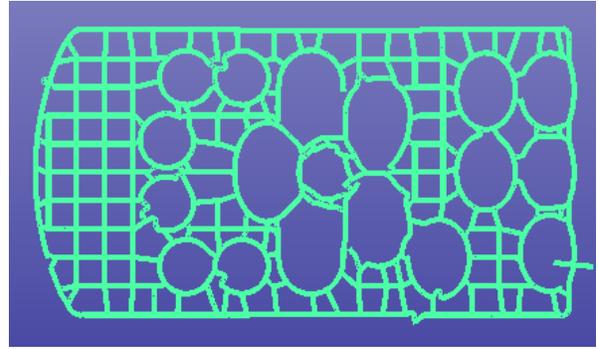
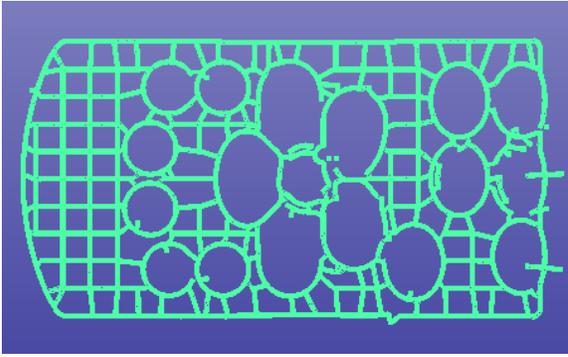


Similarly, with *Arch support* modification the original support geometry is used by default. This enables you to delete some details from the support and then perform the Arch operation and keep the modified support geometry. If *Re-eval area supp (Arhc)* is set, all support vectors are re-evaluated during the Arch generation and previous edits are lost.

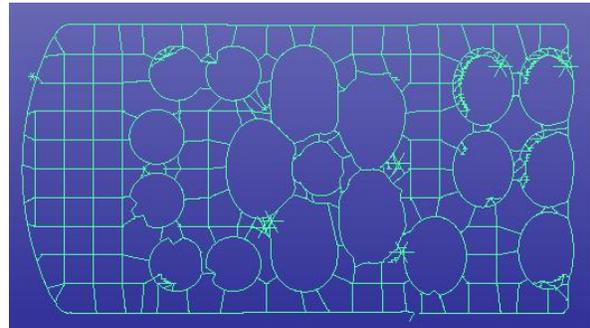
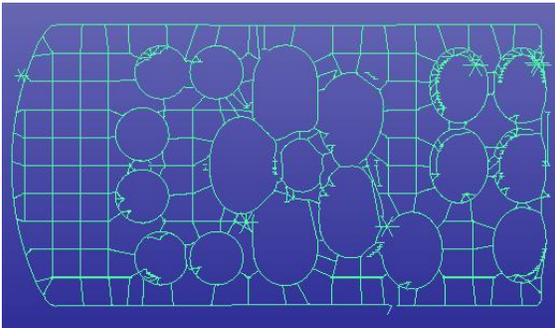
### More consistent support geometry

There are less “stray” single point supports and the supports are generally smoother and better connected than with previous versions. This will help to keep the used material clean and allow better reuse of the material.

In the image below we have version 12.0 solid supports on the left and new 12.1 solid supports on the right hand image:



The same applies to open support structures, less stray components in the supports. Version 12.0 on the left and 12.1 on the right hand image:



### **Lesson 4 - Outputting and slicing supports for AM**

There are two ways to output the generated supports for AM. Either both part and the supports can be stored as STL files on the hard disk or both can be sliced and outputted as slice files for AM. Current slice file output formats include .cli (standard), .sli (EOS format), .ssl and .slc formats.

#### **Saving supports STL file**

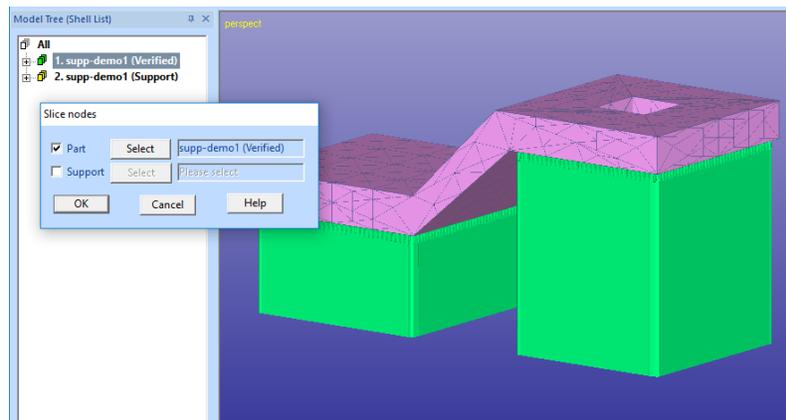
The STL file output is simply done with *Edit > File Save As* command. Select both the part and the support shell in the Model Tree one at a time and save them on hard disk. You should name the parts clearly to separate supports from the part, like *plate.stl* and *plate\_s.stl*, where “\_s” suffix denotes the support file for the plate. STL files can later be given to AM system software for slice generation.

#### **Saving support and part slices**

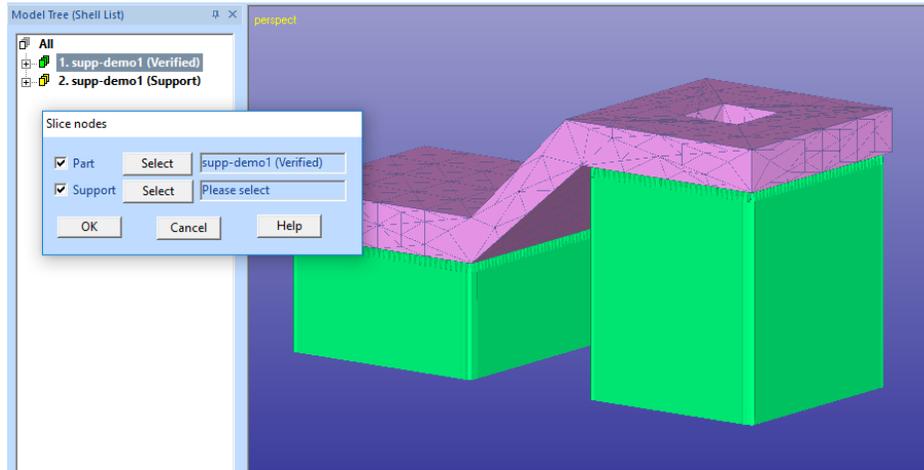
Both the part and the supports can be sliced in 3Data Expert and then sent to receiving AM system software. This involves the use of two different commands, *Print Preparation > Slices* and *Print Preparation > Fix Slices* commands.

If you do not have a supported model available, please generate support to the simple-plate automatically as shown in the Chapter 2. After the supports are generated, the slicing is started by selecting the part to be sliced in the Model Tree:

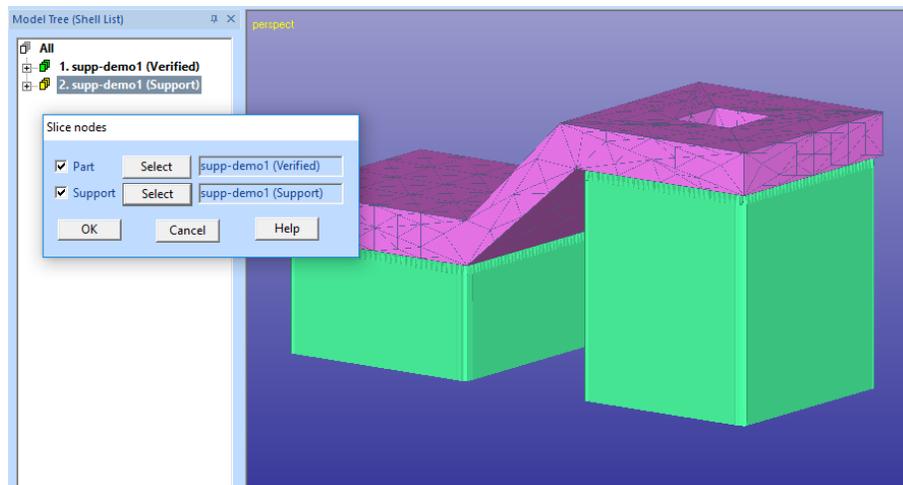
- **Input 1. *Supp-demo1 (Verified)*** in the Model Tree
- **Create supports with Generic Support** command using *DLP/ dlp\_basic-22deg-solid-mm.par* parameter file
- **give command *Print Preparation > Slices*** and a parameter dialog appears:



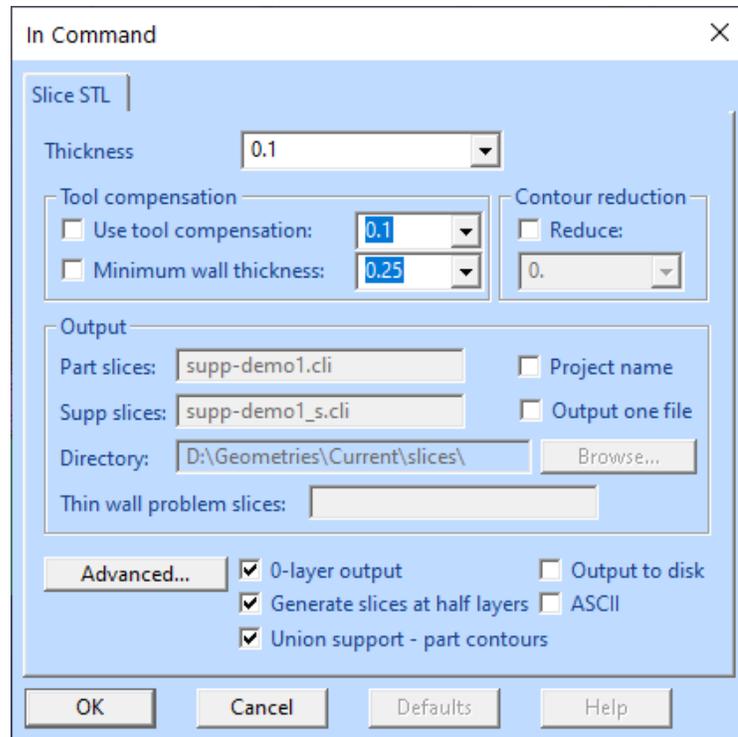
- **click the Support On** and select the **2. supp-demo1 (Supports)** in the Model Tree



- **then press Select** to set the support shell as support data

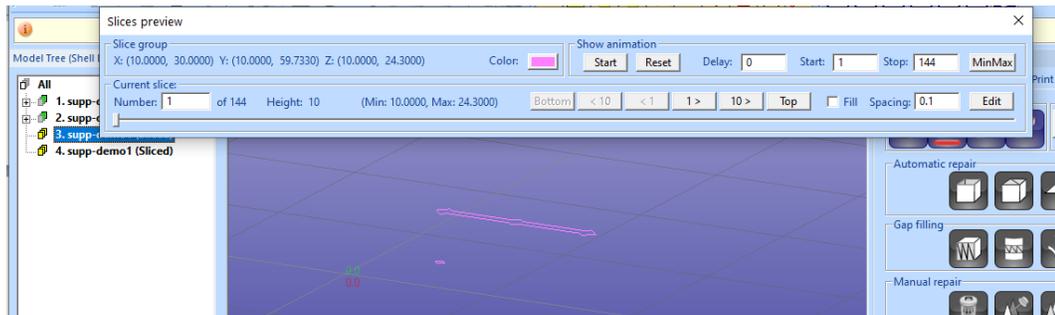


- **then press OK** to go to slice parameters
- **fill in the values shown below:** *Thickness 0.1 mm, No Reduce, No Output to disk, etc...* Especially, make sure the *Output to disk* is *Off*.



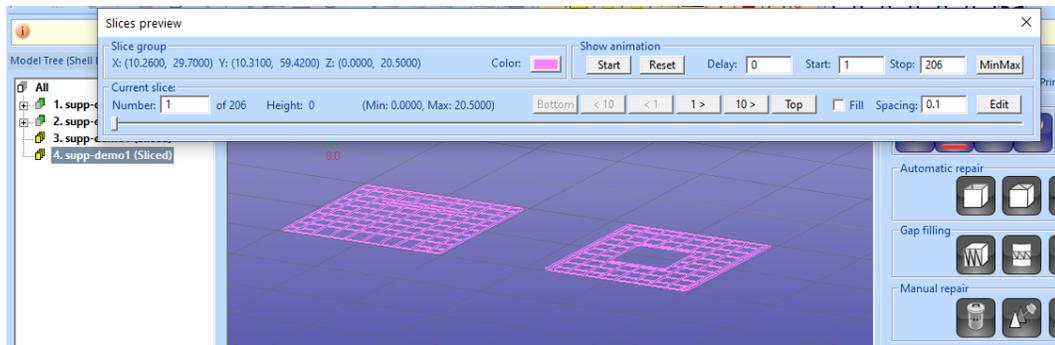
- **press *OK*** to start the slicing

After a while you will see the part slices in the graphical area displayed with the Slice preview tool:



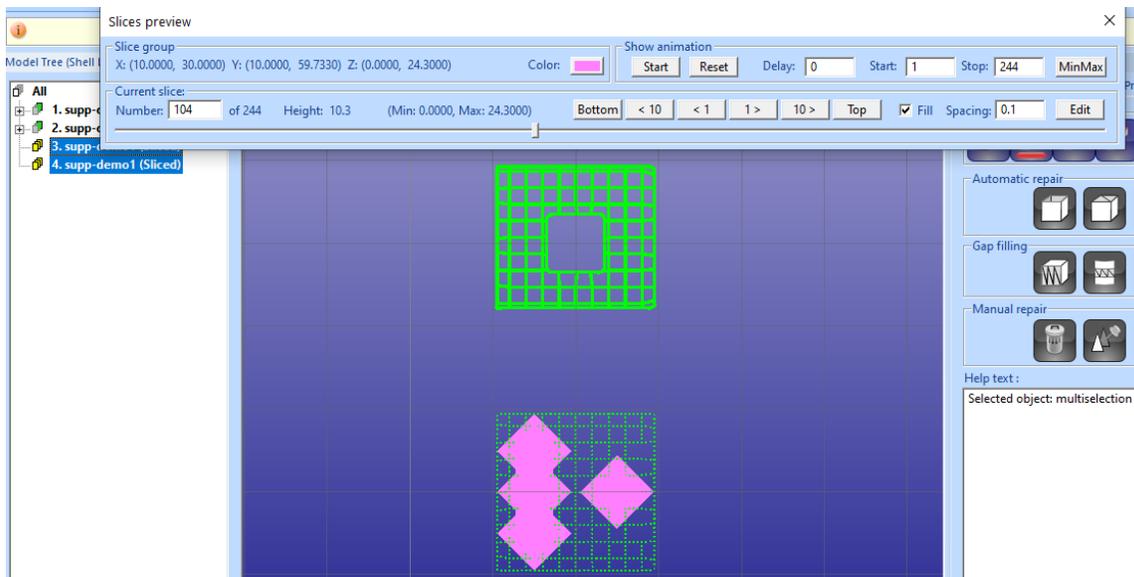
And the support slices are in the last shell.

## Expert Series 12.1: Generating Supports for AM

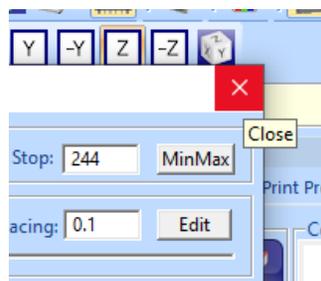


We have now generated slices for both the part and the support with slice thickness 0.1 mm. You can preview the slices using the Slice preview toolbar commands: Change the color of the slices, use the vertical *slider* to manually go through the slices, *Show animation* to see all slices automatically and go to specific slice level with *Current slice group* settings.

Visual checking shows that the slices are ok and should be ready for output to AM processes, as seen in the image below. We have selected both slices for display at once and also use the 0.1 mm filling to ensure the contours are closed.



You can now close the Slice preview toolbar with the upper right hand corner Close icon.

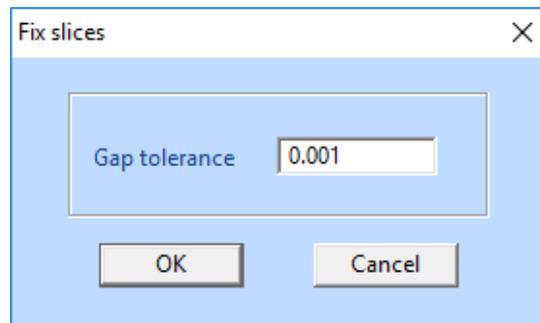


## Expert Series 12.1: Generating Supports for AM

---

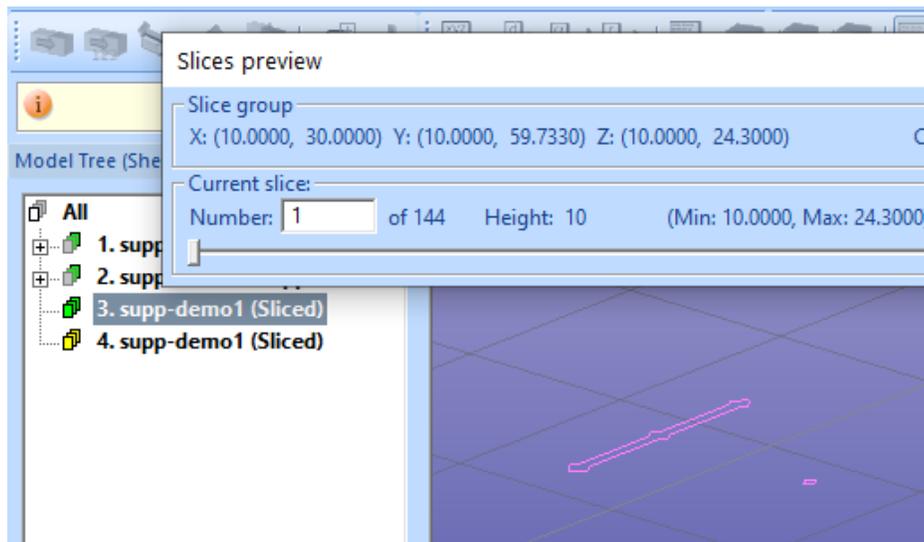
Originally, we have generated solid supports with `dlb_basic-22deg-solid-mm.par` file which produces solid support data. For this reason we want to make sure all slices form closed polygons too. This is checked and fixed with **Print Preparation > Fix Slices** command. We will fix both the part and the support slices.

- **select the part slice shell** in the Model Tree
- **give command *Print Preparation > Fix slices*** and parameter dialog appears

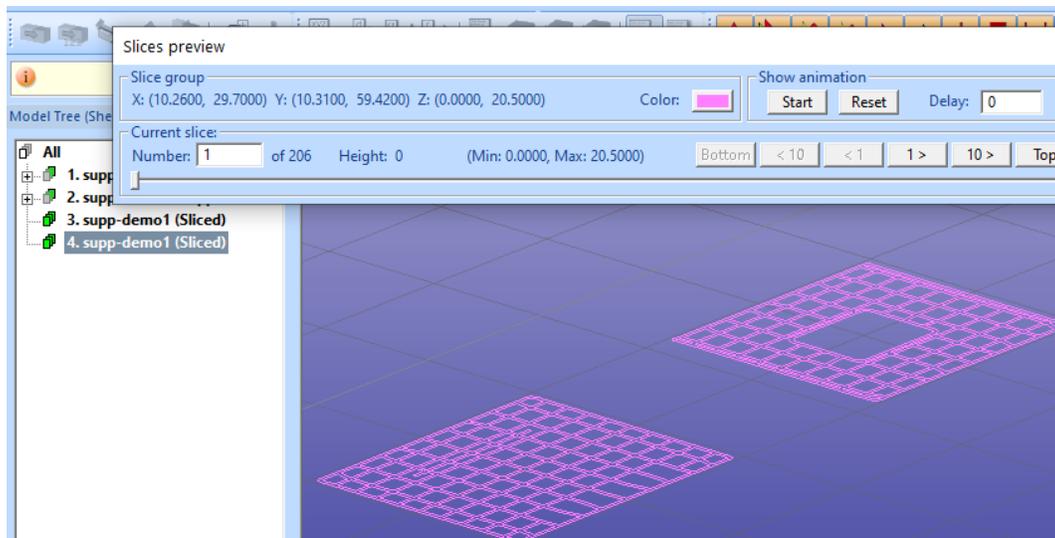


- **accept the default tolerance and press OK**

After a while you will see the slices checked for gaps (no gaps, the Model Tree icon turns green).



Close the Slice preview tool and repeat the **Fix slices** command for the support slice shell too. You will see similar result in the end of the Model Tree and in the graphics area:

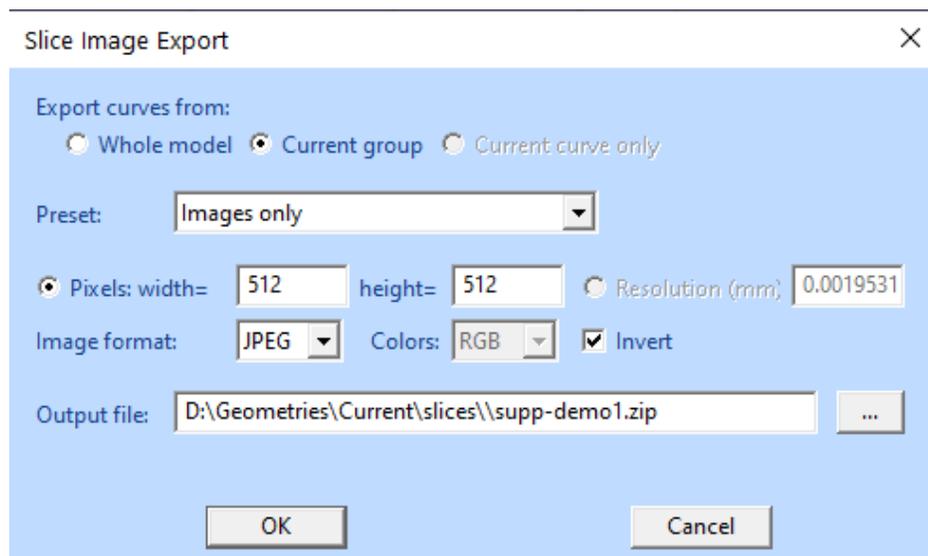


You are now ready to output the slice data for AM. There are two ways to store the data, either as contour data (.cli, .sli) or as bitmap data (.bmp, .png, .jpg).

*Note: When you select the slice shell from the Model Tree the Slice preview toolbar is always displayed. To be able to use other commands, like Fix slices or Save as, you will need to close the toolbar first (via upper right hand Close icon).*

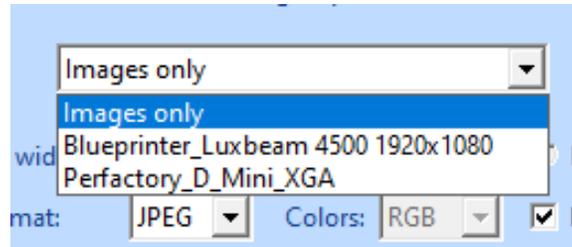
The contour data is saved simply by using the *Edit > File Save As* command. Select both slice shells separately and output them in suitable slice format, CLI or SLI (EOS format). Remember to name the part and support slice files so that they can be separated from each other, like part.sli and part\_s.sli, where “\_s” suffix depicts the support slice data.

For bitmap output you need to generate separate bitmap file. This is done with *Print Preparation > Export slice bitmaps* command. When the command is given, the *Slice Image Export* dialog appears:



With the dialog you can set several properties of the output data:

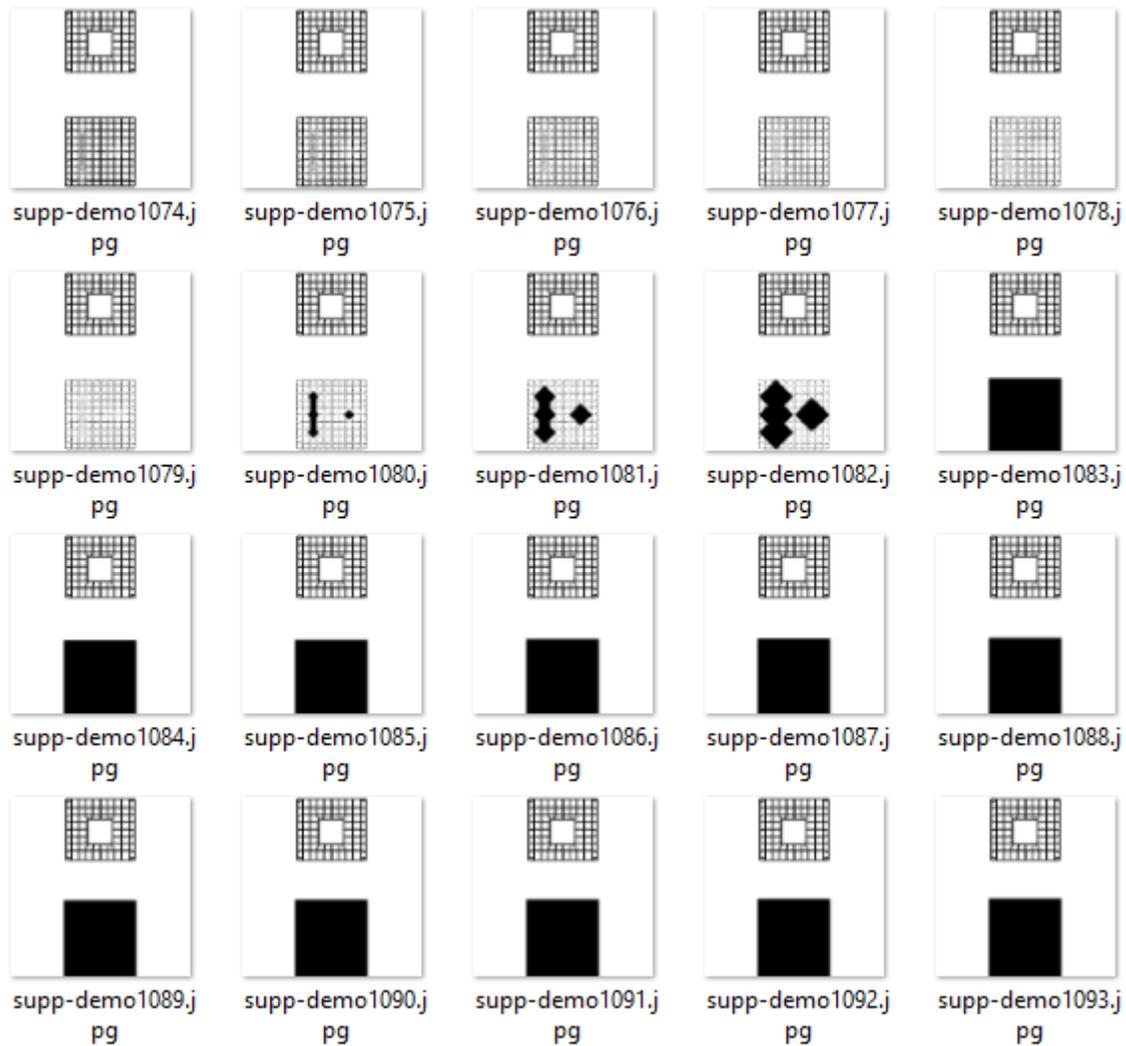
- **Output *Whole model* vs. *Current Group*.** Normally you will select *Whole model* to output both part and support slices into the same job file (.zip, .job)
- Select a suitable preset of parameters for specific AM system or just output standard image files



- set the slice file resolution in pixels
- select image type PNG, JPG or BMP
- select RGB or grayscale image type
- invert inside vs. outside color (dark inside/white outside or vs.)
- select the output directory and output file name (.zip, .job)

Some of the above parameters depend on the receiving AM system, like image type, inverted vs. not inverted, RGB vs greyscale.

When pressing OK the slices are stored into the given directory and file:



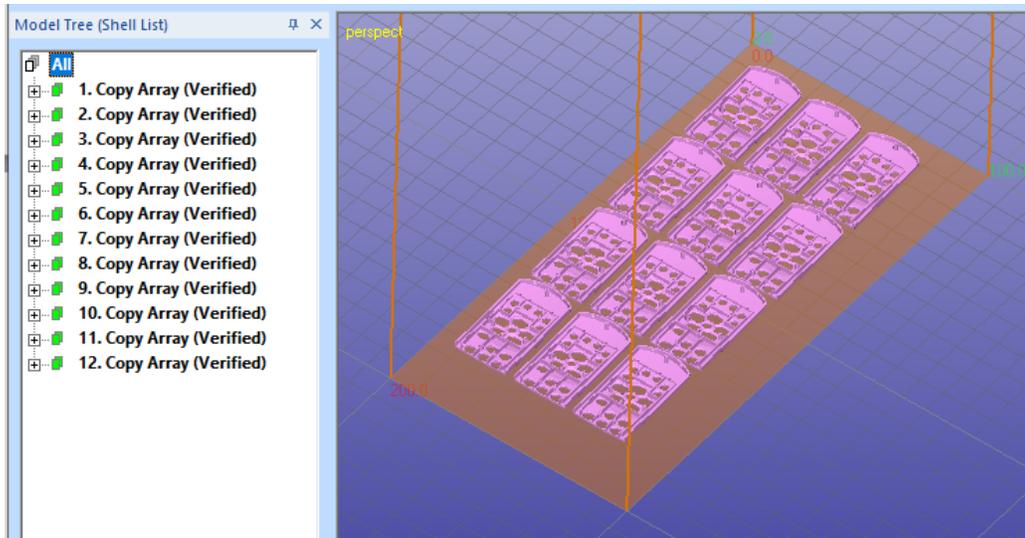
The preview images of the slices above show the support as a web like structure and the part as solid black area.

### Slicing supports and parts into one file

With earlier 3Data Expert version it was necessary to merge all part and support shells into one shell before the *Print Preparation > Slices* command if both part and support slices needed to be stored in a single file. The new 3Data Expert 12.1 allows to slice several part and support shells at once into the same slice file. Actually, it is recommended not to merge the components as that would remove the Label text string attached to the separate shells for Production Screenshot, for example.

We will start with the familiar *12-cellphones.3de* file:

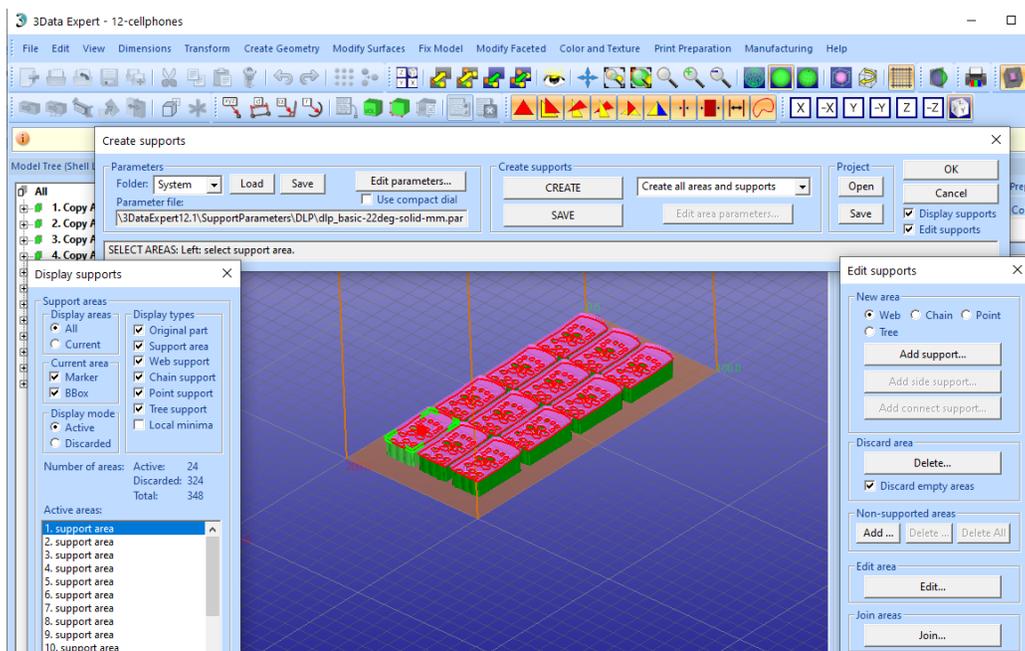
## Expert Series 12.1: Generating Supports for AM



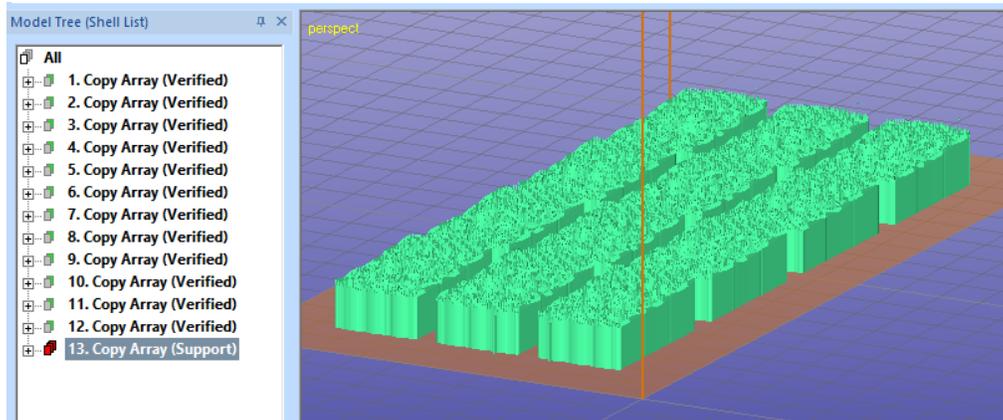
The file includes a workspace for a specific stereolithography system. If you wish, you can turn On/Off the platform display with  icon.

First we will create supports to the cellphone models.

- Select Model Tree root All
- **Create supports with Generic Support** command using *DLP/dlp\_basic-22deg-solid-mm.par* parameter file:



- **Press OK** to save the supports into the Model Tree
- After a few minutes you will have all supports checked and verified in one shell in the Model Tree:



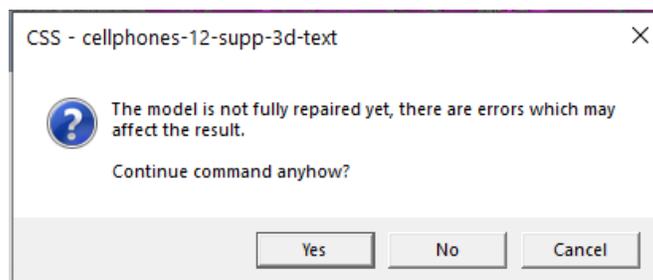
The support geometry has some minor intersection errors which will not affect the slicing process.

Now you can

- **first select root *All*** and Slice all components at once.
- **give command *Print Preparation > Slices***.

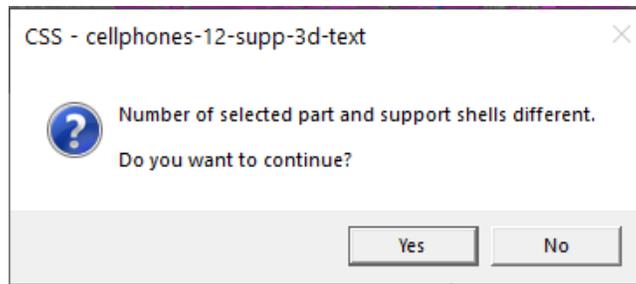
First you will see a warning about the errors in the Support group. We know these are tiny intersections and will not affect the result, so

- **press *Yes*** to continue anyhow:

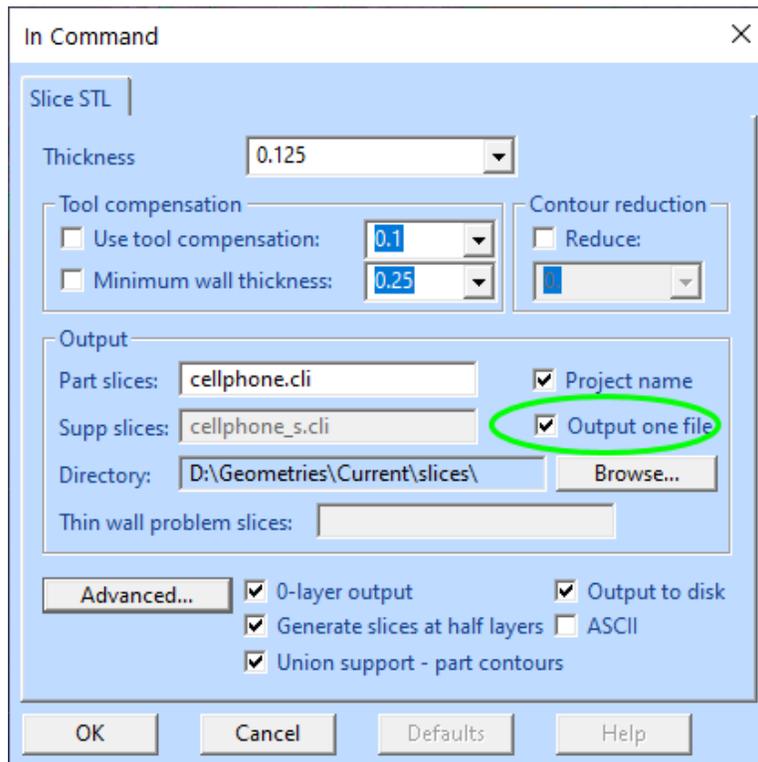


After that software warns that there are different number of part and support shells selected. This is correct because we have generated all supports at once to all separate parts earlier.

- **press *Yes*** to continue:



Finally we will see the Slice STL dialog. There is one new parameter *Output one file* (green circle below):



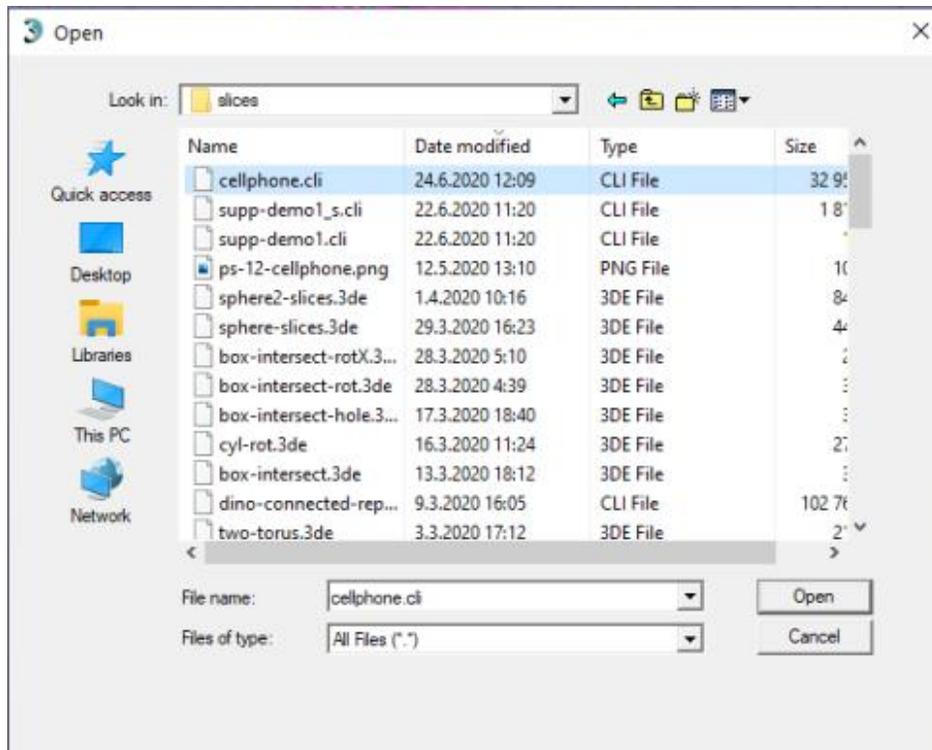
This will cause both the part and support slices to be stored in the same output file *cellphone.cli* (the *Supp slices* field is greyed out). This also allows to use **Union support – part contours** option.

### Previewing Slice data

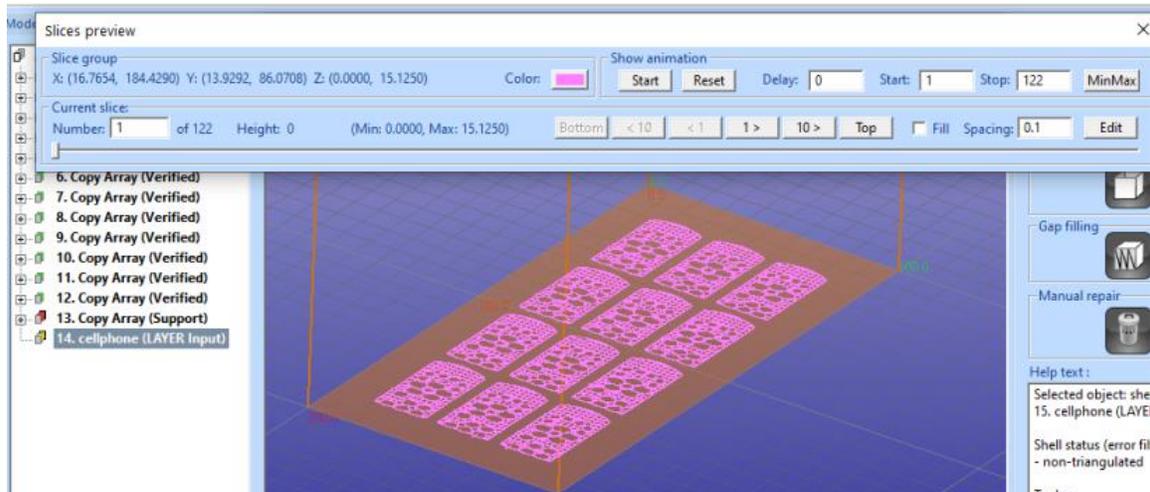
We can now input the slice file into the 3Data Expert for inspection. This is now possible due to redesign of slice data structures in the software.

**We will use the *File > Insert*** command to input the slices into the Model Tree and still keep the original 3D model.

## Expert Series 12.1: Generating Supports for AM

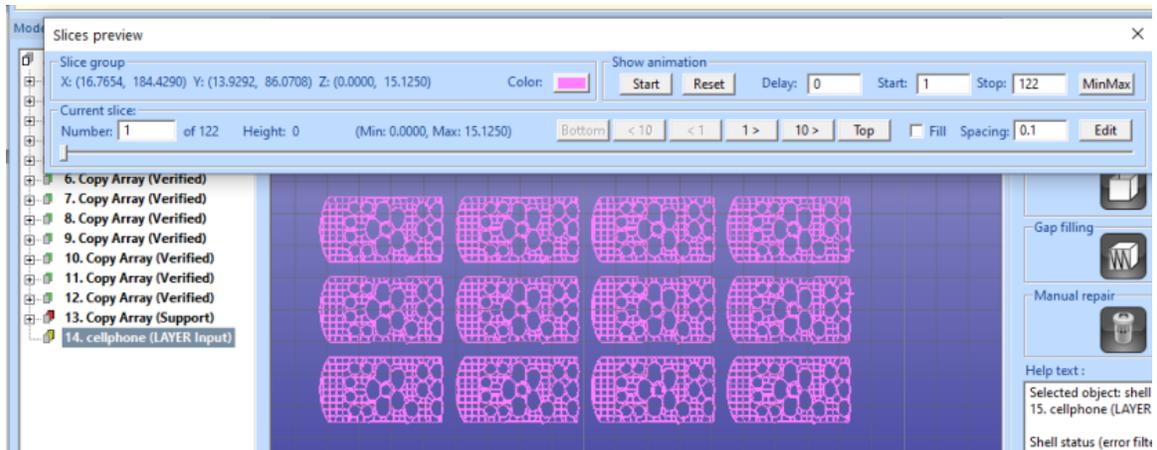


Press Open to insert the slice data in the Model Tree. After a while we can see the new Slice preview tool open and first slice level on the graphics area:



Normally it is best to look at the slices from positive Z axis direction and you can press **Z** icon to change the view point (you may need to fit the slices by double clicking on the slice shell in the Model Tree):

## Expert Series 12.1: Generating Supports for AM

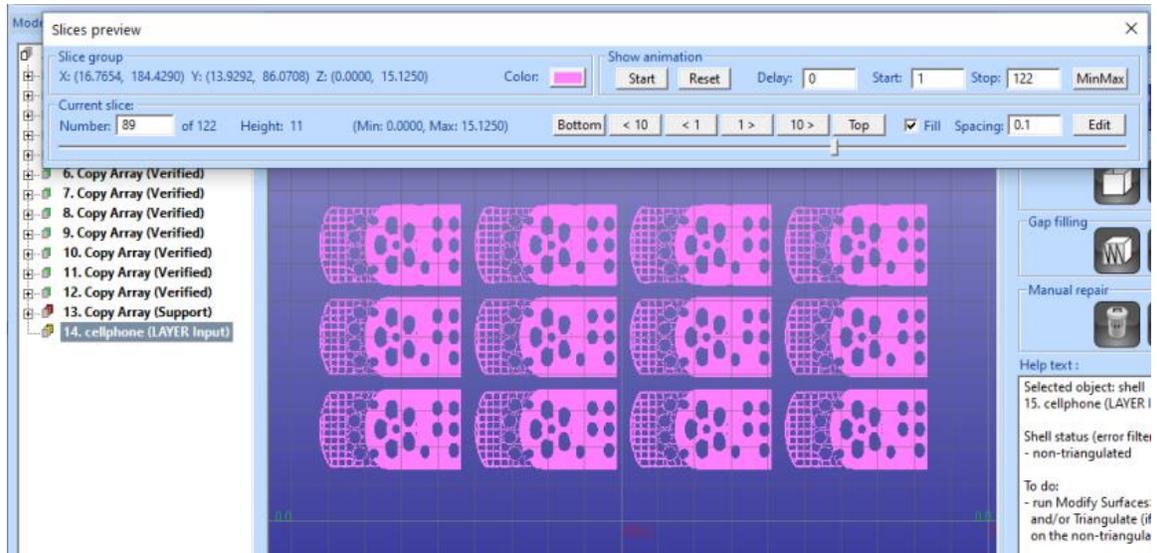


We have also turned the platform display off in the image above.

Now you can start looking at the slices either individually with the slider in the *Current slice* group or start animation in the *Show animation* group to see through all slices automatically.

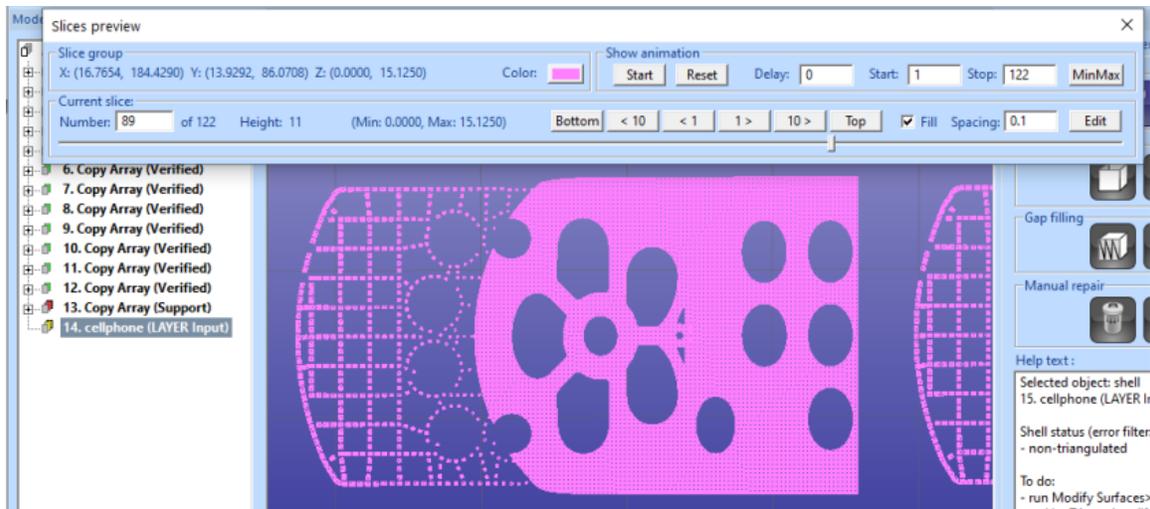
To check the quality of the slicing it is possible to use *Fill* and *Spacing* parameters to simulate the hatching process with AM system software. You can set the density of the hatchign freely but remember that the smaller the spacing between the hatches the longer it takes to generate them.

Below is an example of a level 116 with Fill and 0.1 mm Spacing:

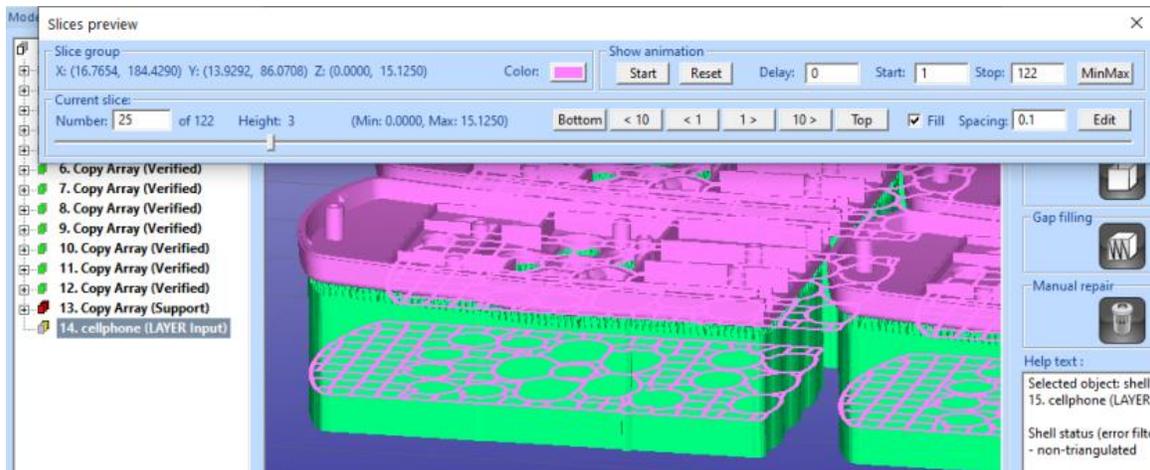


You can use all viewing commands like Zoom to look closer to any region on the slices:

## Expert Series 12.1: Generating Supports for AM



Even 3D viewing with the original 3D model is possible. First select Model Tree root *All* to show the 3D model and then re-select the slice group. Slice colors can be changed to differ from the part colors. All viewing commands are available, like zoom, pan, rotate etc.



*Note: the Slice preview tool is turned on automatically when you select slices (LAYER Input) shell from the Model Tree and turned off when something else is selected in the Model Tree. You can also close the Slice Preview tools window on the right hand- up corner cross. After that you can Copy/Paste/Delete etc. the slice shell.*

### Fixing slice data

The *Print Preparation > Fix Slices* command is also adapted to new slice structure handling. It can now process very large slice data sets in a reasonable time.

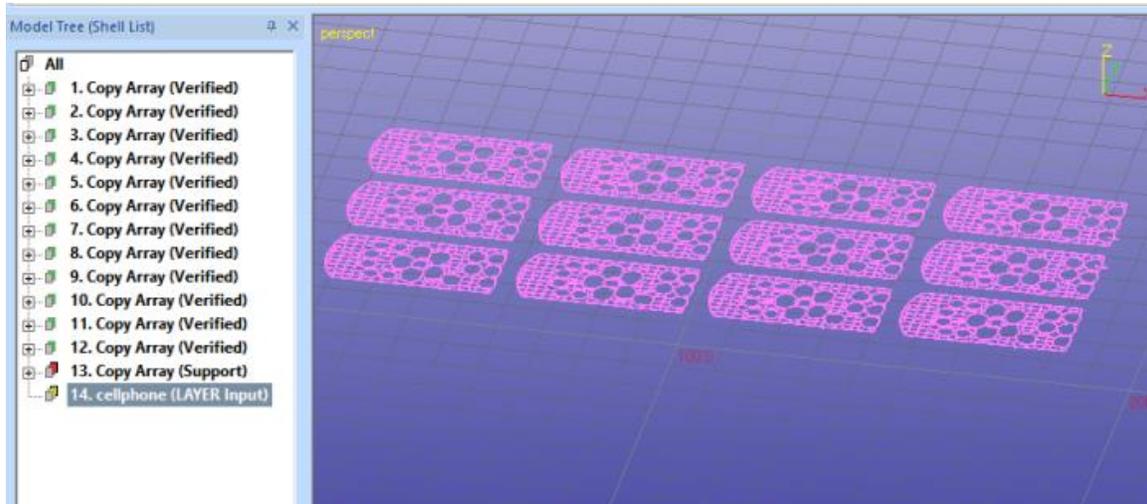
## Expert Series 12.1: Generating Supports for AM

---

You should use Fix Slices command to close gaps in the slice contours if you see stray hatches during the Slice Preview. These stray hatches may cause struts in the final part.

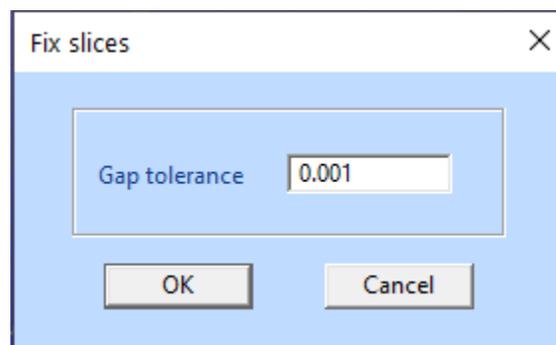
To run Fix Slices

- **first select a slice shell** and
- **close the Slice Preview tool** which appears (from the upper right corner Close icon).

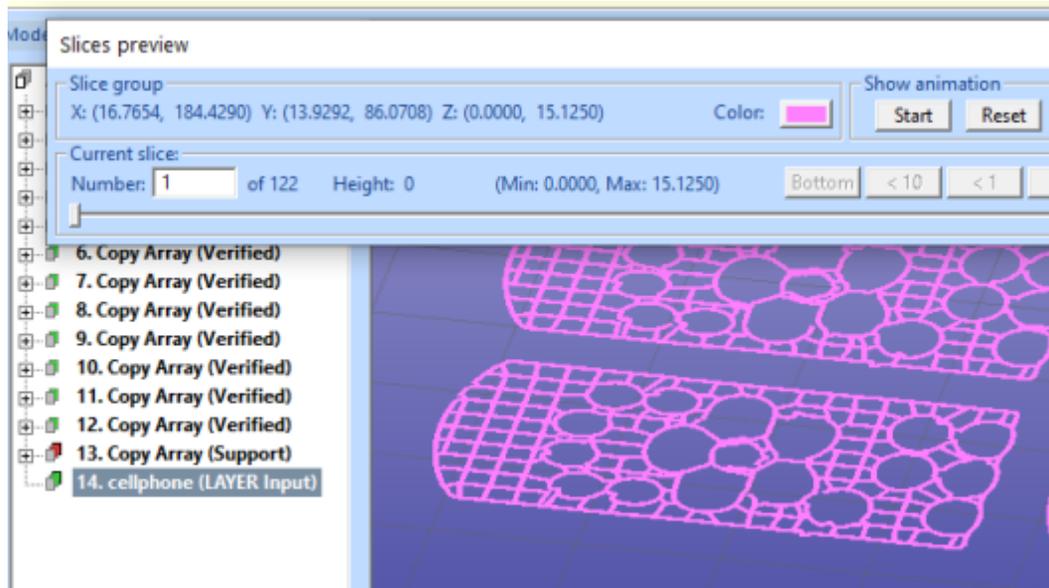


Now you are ready to

- **give the *Print Preparation > Fix Slices*** command. A simple parameter dialog with *Gap tolerance* parameter for the maximum size of gaps to fill appears:



Press OK to run the command and after a few minutes the result can be seen on the Model Tree, the cellphone sell has a green icon:



Green icon with slice shell indicates that there are no slices with gaps in the model. So, it is fairly safe to say that the hatching will produce a correct result during the manufacturing.

If there still were gaps in the slice data after the Fix Slices command the icon would be red and the slice with gaps would be indicated with red color when viewed with Slice Preview. Of course, a larger *Gap tolerance* could be tried to fill the remaining gaps too.

*Note: Fix Slices does not check for intersecting or overlapping slice segments.*

If you have more than one slice group in the Model Tree you can use multi-selection to preview both slice sets the same time for comparison. Anyhow, you should change the color of the slice shells with **Color** button in the *Slice preview* toolbar first.

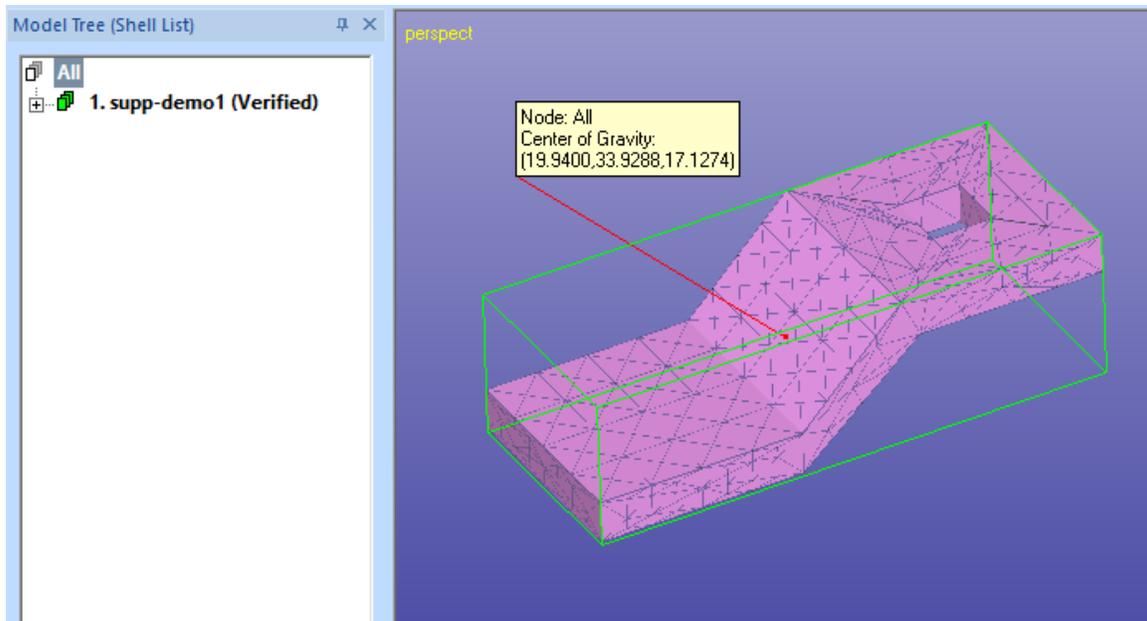
New *Slice preview* toolbar even allows editing each slice with Edit command for removing and fixing stray contour segments in the slice file, for example. For more details, please have a look at Help pages for Slices command.

### ***Lesson 5 – Additional tools for support generation***

There is a set of additional tools to aid the generation of optimal support for a part. These tools include Transformation commands, like Rotate and Move, as well as Center of Gravity calculation tools. For example, a part can be rotated so that the selected surface is facing exactly upwards for best surface finish or the Center of Gravity is set to the lowest point to minimize the bending forces to the part during the build.

#### **Center of Gravity calculation**

The *Dimensions > Center of Gravity* command is used to calculate the corresponding point in 3D. For the calculation the bounding box of the part is divided into 100x100x100 cells and the calculation is based on this subdivision.



The *Center of Gravity* command can be run either on a single shell or root All to calculate common center of gravity for multiple shells. If multi-selection is used, the point is calculated to the first selection.

### Lesson 6 – Connect Tool

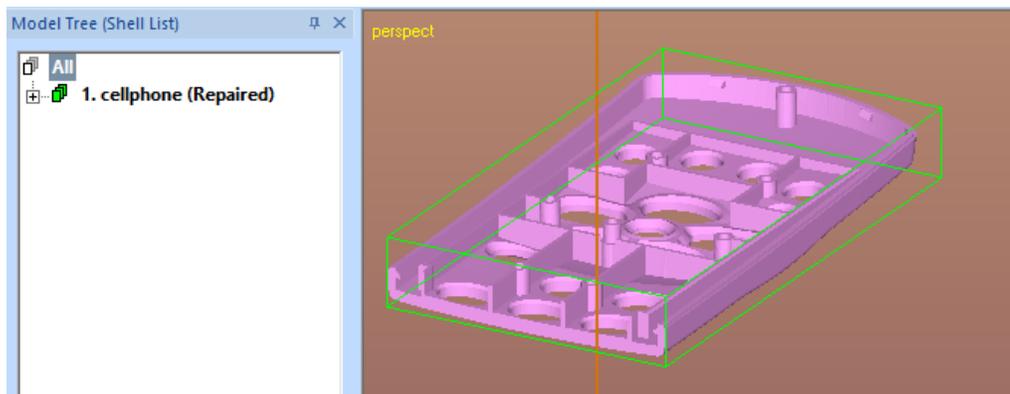
The Connect Tool is used to add support struts between component details for added strength. Struts will prevent the model from warping during the AM or post-processing.

■ Free View   ■ Dimensions   ■ Lite   ■ 3Data   ■ Sim

#### Connect Tool command

Connect Tool in Modify Faceted menu will allow to add connecting struts and tree like structures between component surfaces. The connecting struts will help to keep the shape of the part during support removal and post-processing. For this example we are using file *1-cellphone.3de* file.

- **Input 1-cellphone.3de** from Tutorial/GeomFiles directory
- **Bypass Fix Model etc.**



- **Select the cellphone shell** from the Model Tree and
- **start the *Modify Faceted > Connect Tool* command.**

The Connect Tool command toolbar appears:



First you must set the *General Dimensions* of the connectors, overall *Dimension* 1.0 and *Tip dimension* 0.5. These will be stored to the connector tip and bend points during the connector creation.

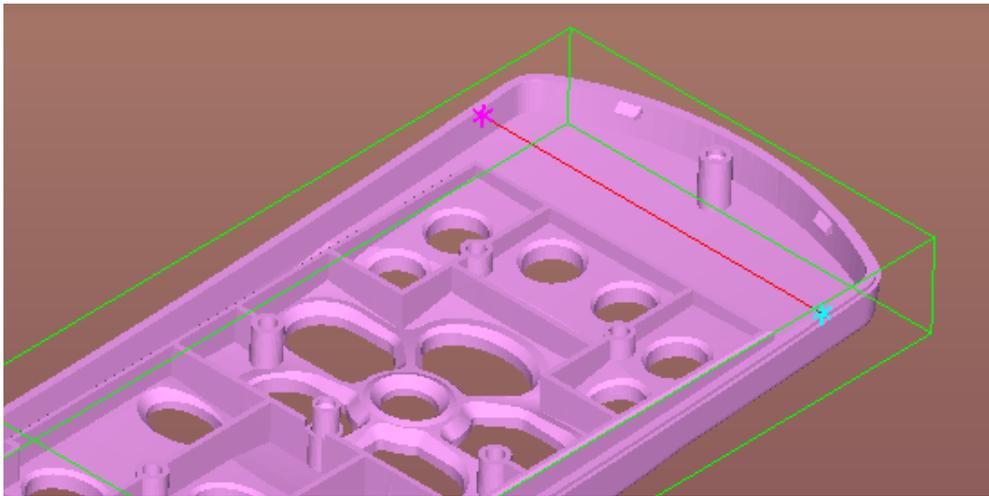
When ready

- **push the active *Add strut trunk* button** .

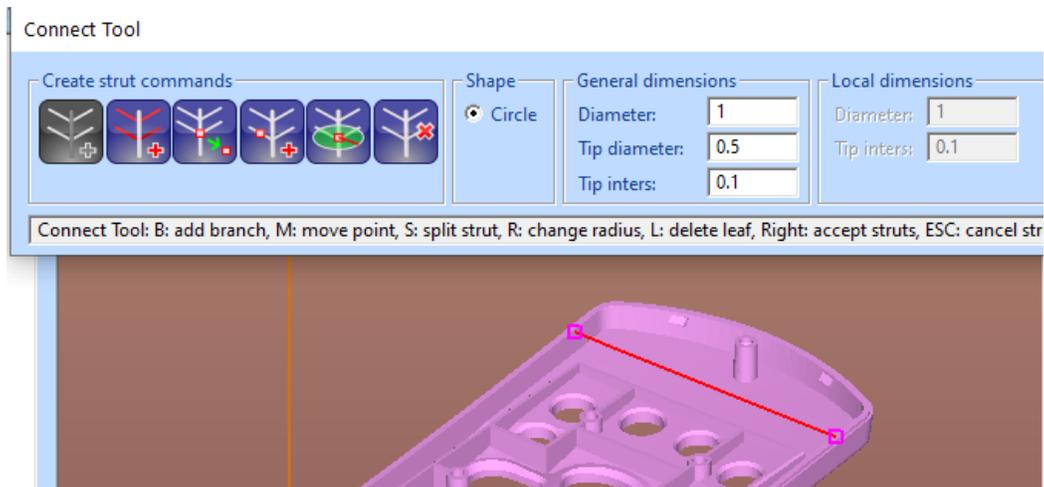
You will go into strut point addition mode.

- Show the *strut start point* (red cross) and *end point* (blue cross) on the component walls with **LMB**. Then press **RMB**.

You can rotate the view during the point addition with the MMB (middle mouse button).

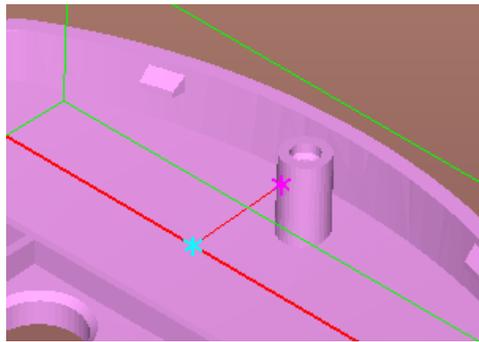


- **After the *RMB* is pressed**, the connection strut main trunk is ready:



Now you can accept and generate the single strut by pressing RMB or OK button in the Toolbar. Or you can continue to add branches to the strut tree. We will do that.

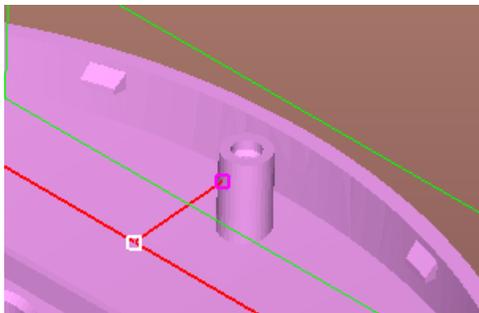
- Now press the active **Add branches to strut**  button.
- Then **show first** point on the component feature with **LMB** (red cross) **and the second point** on the strut main trunk with **LMB**, as seen in the image below:



When ready

- **press RMB** to accept the branch. Note, you can *always cancel the last operation by pressing ESC button*.

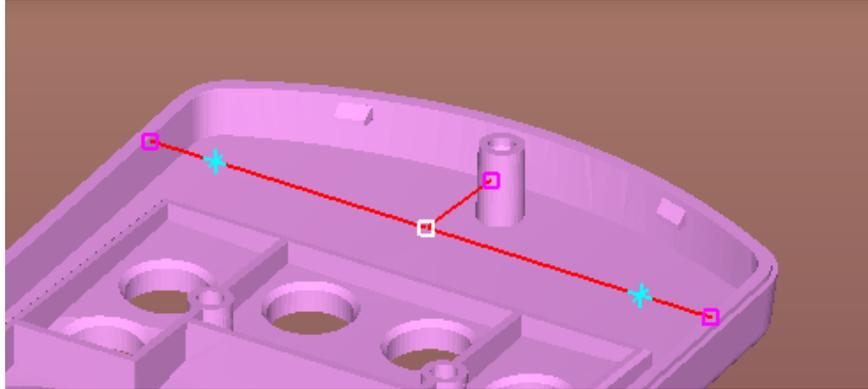
You will see the branch added to the trunk structure.



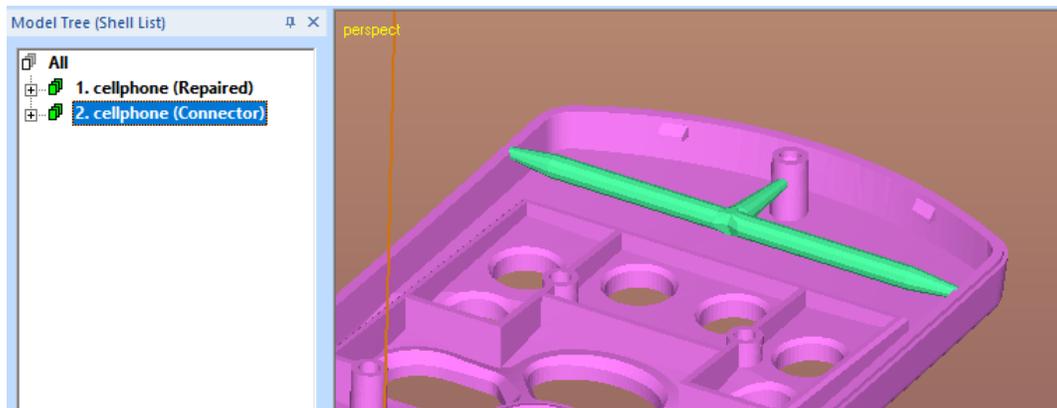
We will finally add two additional split points to separate the trunk end points from the trunk body. This is needed to have the tip diameter separated from the main trunk diameter more clearly (otherwise there would be continuous radius change from the back point in the middle towards the end points).



- Press **Add point to split strut** button to **Add point to split strut**. Show two points with the **LMB** as seen in the image below (blue crosses). When ready adding points **press RMB**:

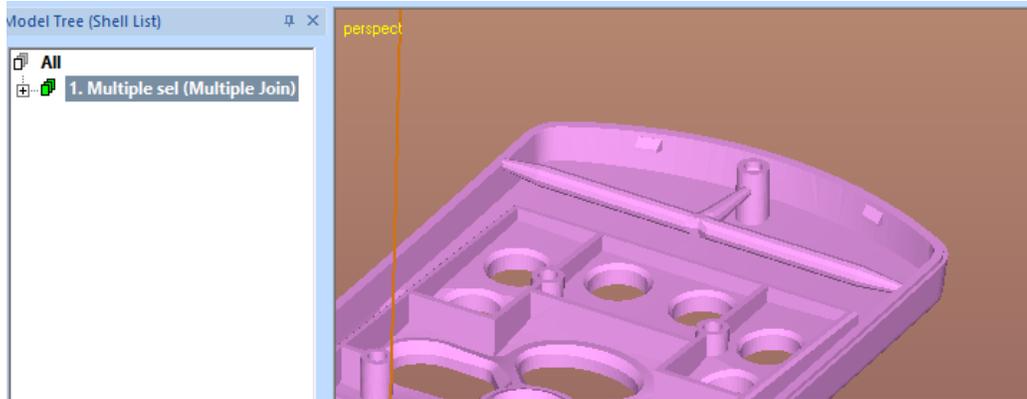


We are now ready creating the connect strut structure and will **press RMB** again or **OK** button in the toolbar to actually generate the triangle model for the struts.



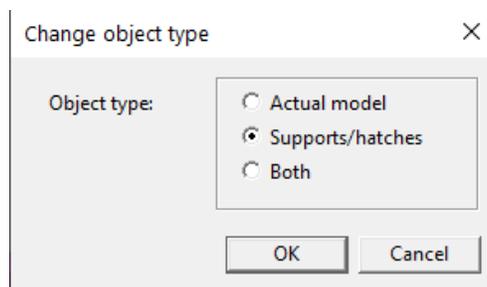
The connector strut is now stored in the Model Tree. It is a separate shell from the part component and there are different ways to use it with the cellphone part.

One normal way is to use Boolean commands, like *Modify Faceted > Multiple Join* to combine cellphone and strut into one component. This way the connector will be produced with the part building parameters on the AM machine and possibly have supports generated to along the part support generation.



If necessary, supports can be generated below the connector.

Another way is to mark the connector as being part of support structures (with *Print Preparation > Supports > Mark Supports* command, please see dialog below). This way the connector will be slice and processed as part of the support structures.



### Lesson 7 – Sand Supports

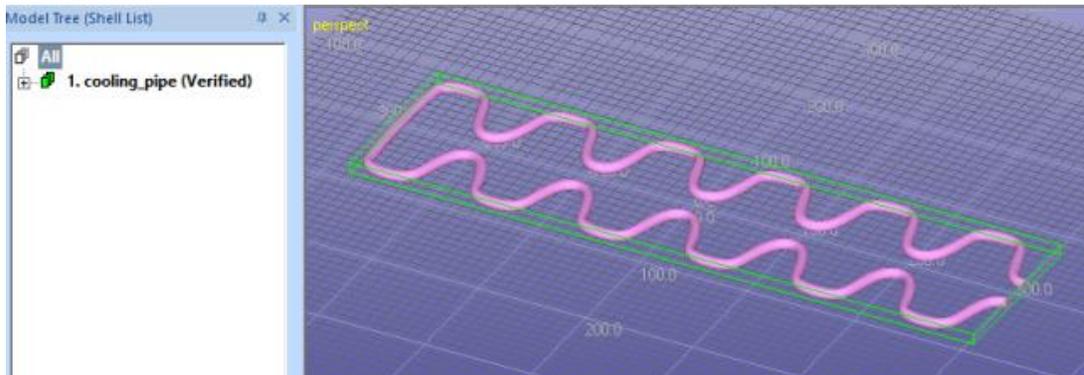
The *Print Preparation > Supports > Sand Support* is used to generate sand support structures for sand based AM systems. Sand supports generate a supporting plate below the part to avoid breaking the part during clean-up and transportation to customer site.

■ Free View   ■ Dimensions   ■ Lite   ■ 3Data   ■ Sim

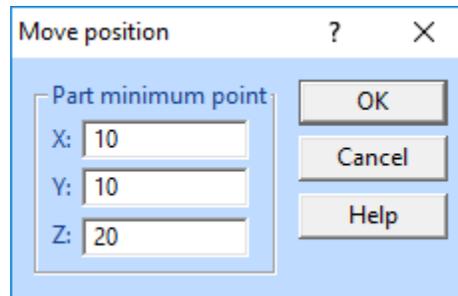
### Sand Support generation

In the sequel we will explain how to make a supporting plate to a thin cooling pipe geometry. We will use *cooling\_pipe.stl* file as an example.

- input the *cooling\_pipe.stl* model from *Tutorials / GeomFiles* directory.
- run through the **Fix Model and Auto Repair** process
- **Go directly to the Print Preparation mode.**



- **Move the cooling\_pipe into correct position** (positive space) with *Print Preparation > Move to Platform > Move Positive* command with the parameters below:

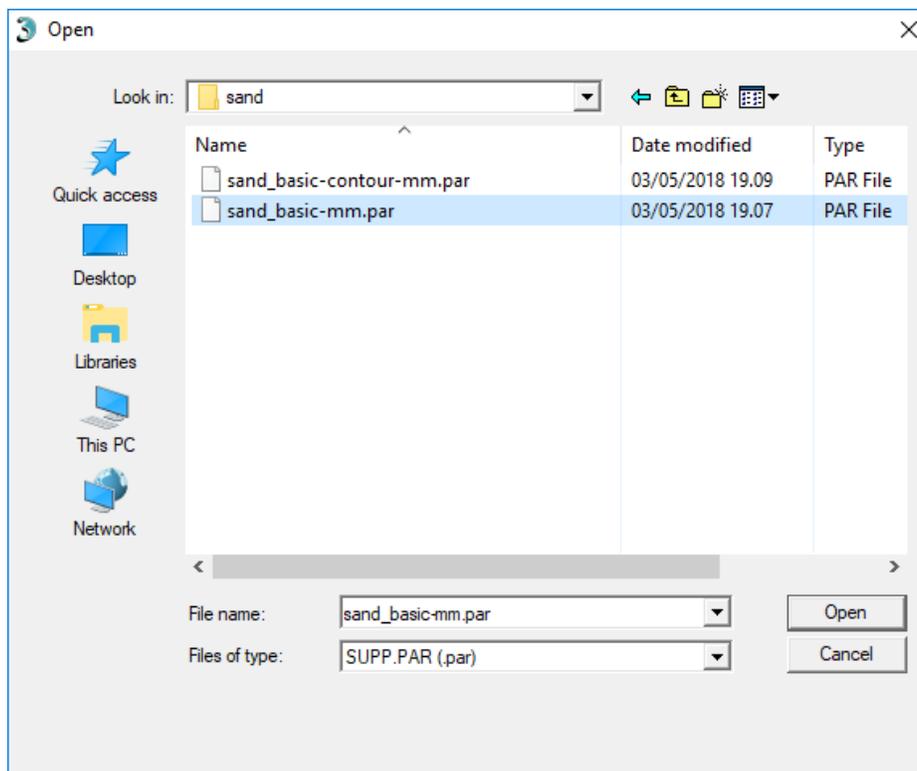


The part where the supports are to be generated must be selected before the command.

- **Select the shell *cooling\_pipe* from the Model Tree.**
- **Start support generation with the command *Print Preparation > Supports > Sand Support*.**

The File Open window opens to select the proper support file for the different support operations.

- Go to sand-directory and **select the *sand\_basic-mm.par* file**

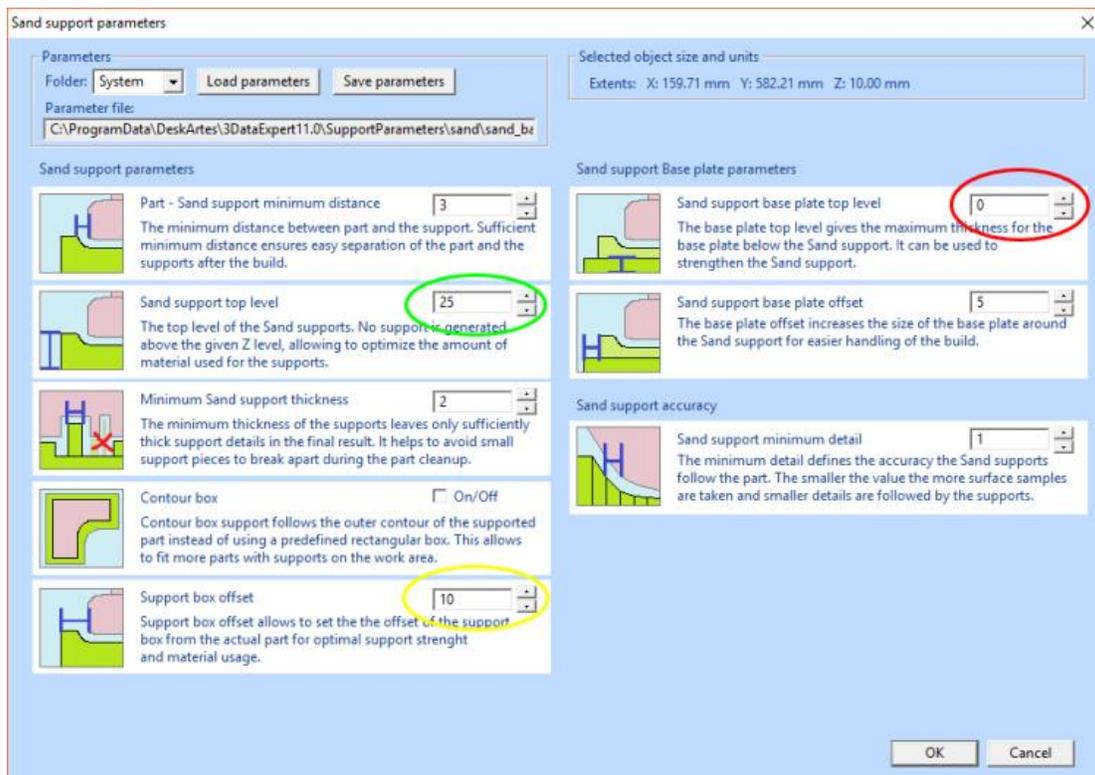


The dialog box for *Sand Support* parameters now opens. Each parameter includes a comprehensive text and image explanation.

## Expert Series 12.1: Generating Supports for AM

First you will have to set the values for the support generation before executing the command. In the image below you can see the proper values to run the command for *cooling\_pipe.stl* file. Especially, set the

- **Sand support top level to 25 mm**, in the middle of the pipe part in Z direction,
- **Support box offset to 10 mm and**
- **Sand support base plate top level to 0 mm** (this is no longer needed)



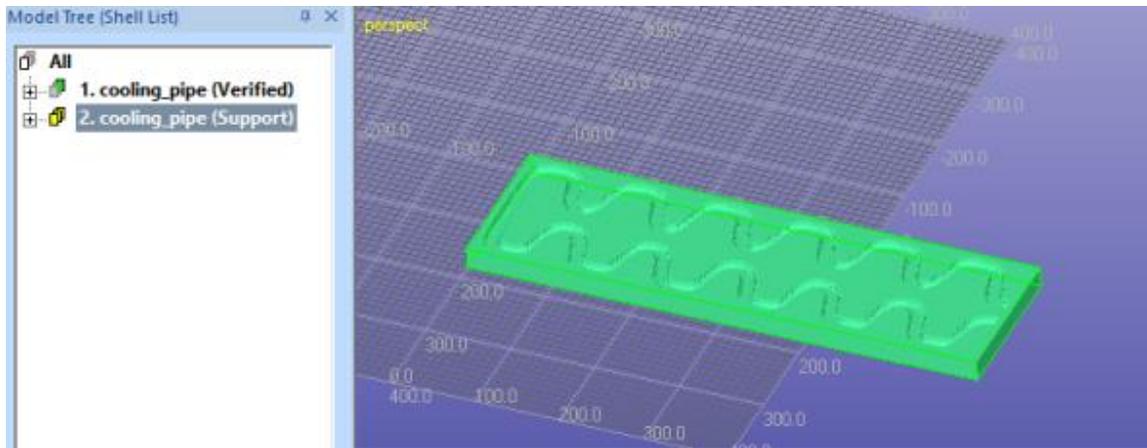
This set of parameters should get you started with the support generation. Anyhow, the exact values depend on the machine and the used resins. Testing, your own expertise and machine documentation will tell the correct values for each machine/material type.

- **Start the support generation by pressing OK.**

A percentage bar appears, and soon you will see the supports displayed on the screen.

## Expert Series 12.1: Generating Supports for AM

---



The part and the supports can now be selected with *multi-selection* and outputted into a single file for slicing with the machine vendor's software.

