

DeskArtes

3Data Expert 12.1

Tutorial 3 - Separating Shells
for different material and color definitions

3Data Expert 12.1: Tutorial 3 - Separating Shells

First Edition

Ismo Mäkelä: "3Data Expert 12.1: Tutorial 3 – Separating Shells".

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Foreword

The *3Data Expert 12.1: Tutorial 3 - Separating Shells* document provides a few simple examples on how to separate areas in a shell into separate shells for different material or color definitions. The separated shells can be used to produce 3D printed parts with multiple materials (= different shore values) or with different material colors in different areas.

First, this document describes the use of special and easy to use *Print Preparation > Separate to Shell* commands for separating model surface areas into new shells. There are three different separation commands available: *Offset to Shell*, *Extrude to Shell* and *Fill to Shell* commands. Accompanied with *Fix Model > Clean Surfaces* command the shell separation is possible with just a few mouse clicks. The time requirement for these examples (Chapters 1 to 5) is about 10 minutes.

The document also describes the use of normal *Cut Tool* and *Hole Tool* commands for separating shells by cutting the model. *Extrude from surfaces* and *Offset* commands are used to generate solid shells from selected surface areas. The use of *Triangle Edit* and *Sketch Edge Line* commands to define new surfaces areas for shell separation are also described. The time requirement for these examples is about 20 minutes.

If you are a newcomer with 3Data Expert, we recommend you to go through the *3Data Expert Quick Start Guide* available through *Help > Tutorial Documents* menu to learn basic visualization and selection commands with the software before going through this document. Also, *Split* command can be used for separating shells by splitting the model, as explained in the Quick Start Guide.

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

- *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, 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.

This document uses *cellphone-repaired.3de* model as an example. The file comes with the 3Data Expert installation in the *tuorial / GeomFiles* directory.

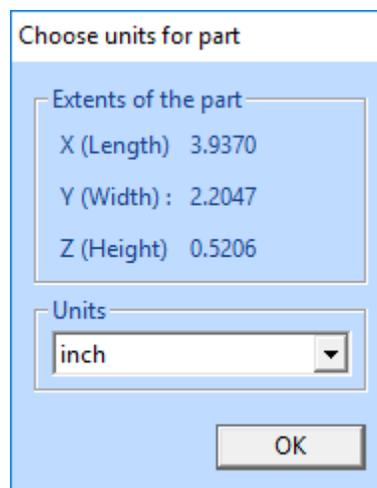
Chapter 1 – Preparing for shell separation

Shell Separation commands allow you to separate surface areas and other components from a repaired shell into separate shells. The separate shells will lay exactly on top of the part they are separated from and they can be used for defining different material properties (shore value) or different colors in a single part.

In the sequel we will first prepare a file with low quality surfaces for shell separation operation. Of course, this is most of the time not needed, but shows how to work on low quality data too.

Input cellphone.3de

First open the tutorials > **Geom Files > cellphone-repaired.3de** file with the *File > Open* command. Accept the proposed units (inch)

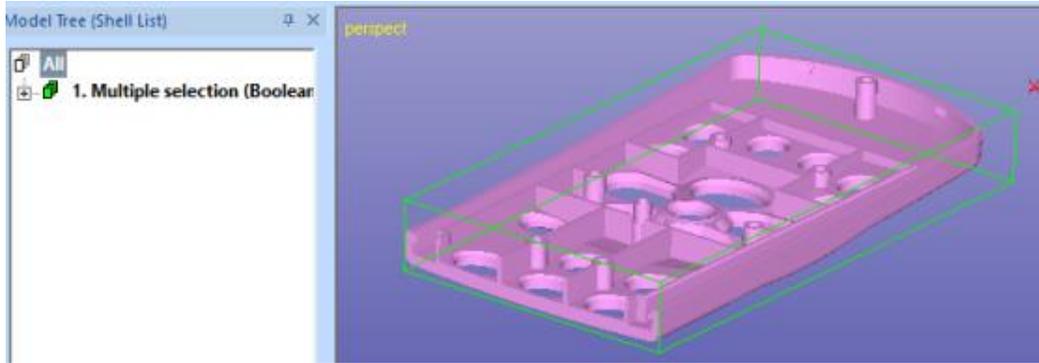


and close the Operation Mode dialog by pressing the *close box* in the upper right corner of the dialog or press *Print Preparation* icon. The model is already fixed so we do not need to perform any fixing.



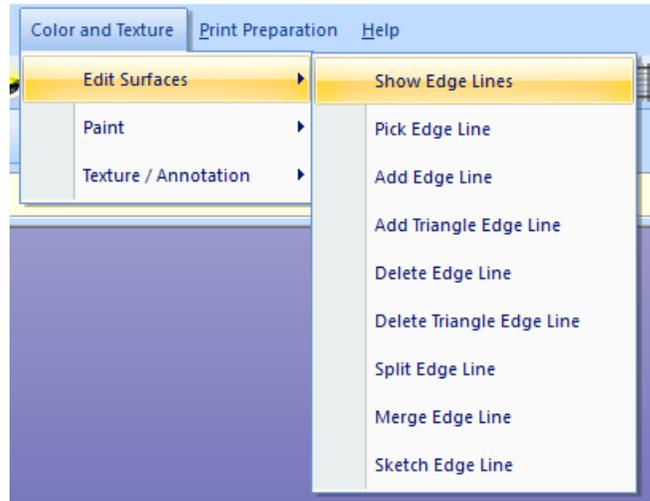
After the input the cellphone model is visible on the graphics area:

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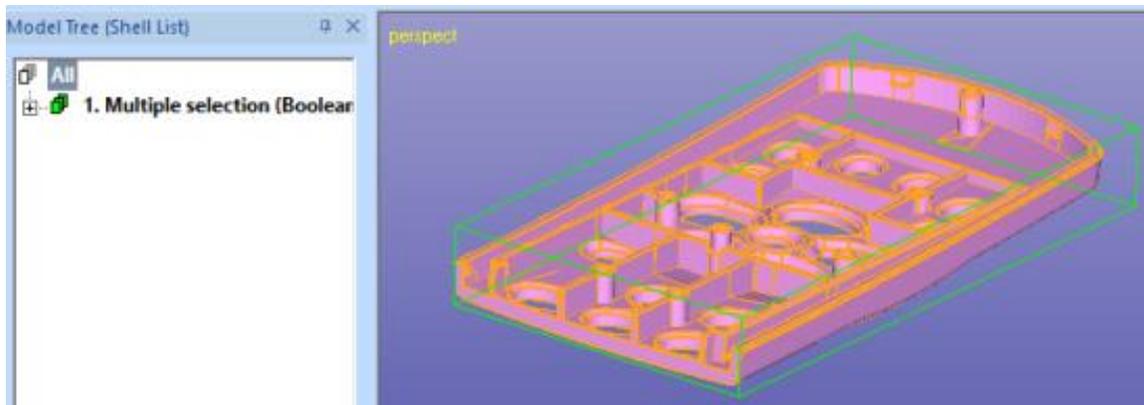


The new shell generation commands are based on the surface subdivision of the triangle model. A set of surfaces is first selected and then a solid shell is created from them.

Before we go on with the shell separation, let's check the surfaces on the cellphone model first. Now **turn on the surface edge display with *Colors and Texture > Edit Surfaces > Show Edge Lines* command:**



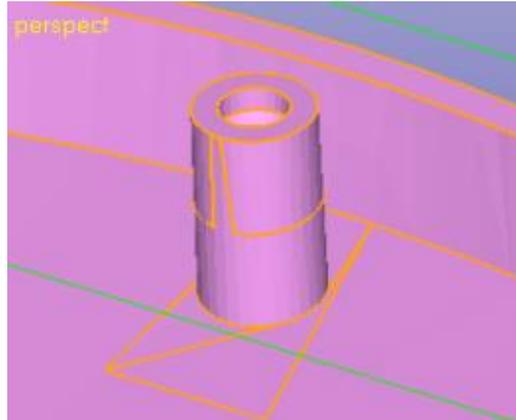
You will immediately see the surface edges displayed with *orange* edge lines:



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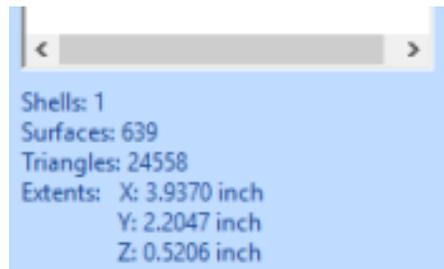
Closer investigation shows that there are excess amount of surfaces and surface edges. The surface edges have originated from the original input model which had several separate shells which were then stitched into one shell (as described in *Tutorial 1 - Working with 3D Models* document).

For example, the largest cylinder fixture on the round end of the part is made of several surfaces, as seen in the image below:



Because we want to separate surfaces on the cylinder into separate shells, lets simplify the surfaces subdivision first.

The number of shells, surface, triangles etc. on the selected Model Tree object is always displayed below the Model Tree:



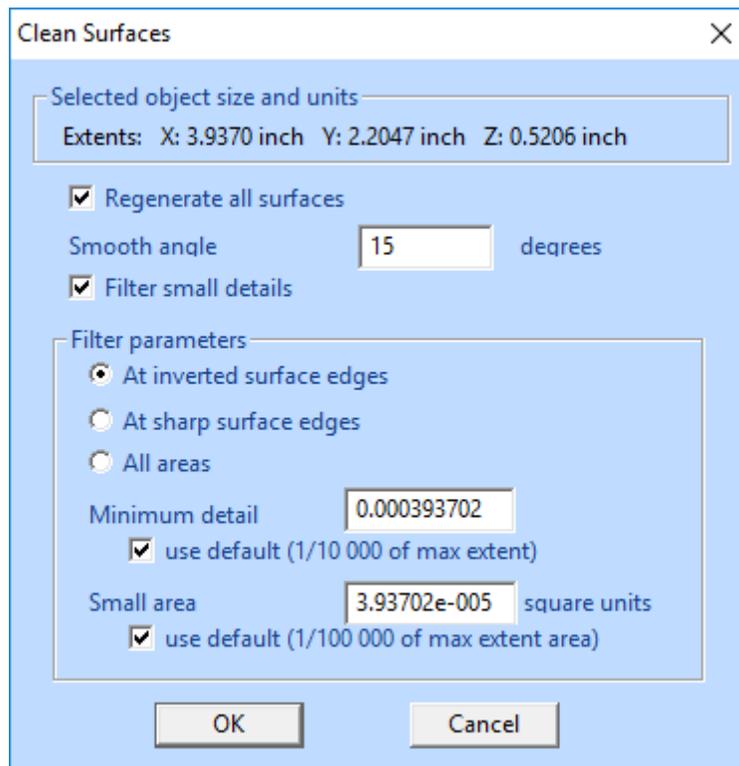
Note: When you need more info on how to create new surfaces manually by adding edge lines on the model, please see 3Data Expert 10: Tutorial 2 – Working with Colors.

Chapter 2 – Simplifying with Cleaning Surfaces command

The *Clean Surfaces* command is used to simplify surface subdivision on the part.

- **select first shell** in the Model Tree
- **give command *Fix Model > Clean Surfaces...***

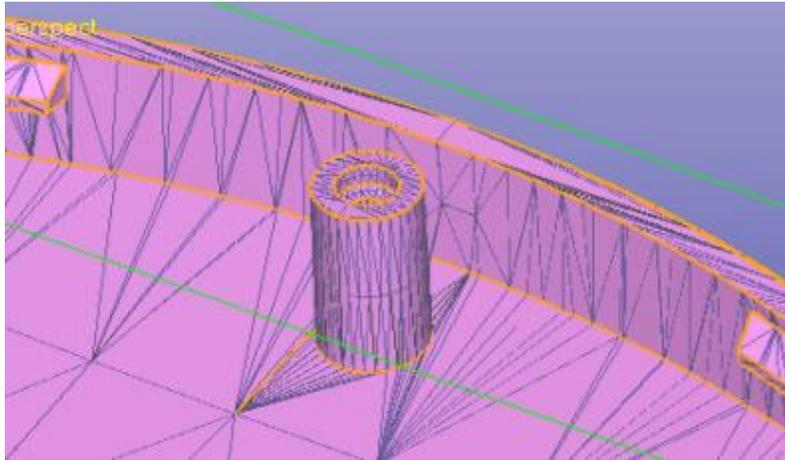
A parameter dialog appears filled with default parameters:



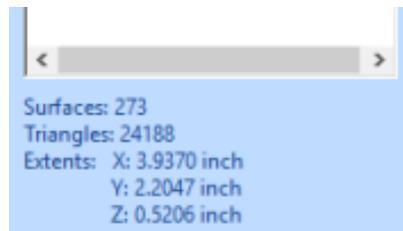
The parameters are automatically calculated based on the part size. This time we will just accept them and reduce the number of surfaces with default settings. The explanation to the parameters can be found in the Help pages.

- **Press *OK*** to start the command

After a while the surfaces are reduced into more meaningful set:



Now we can easily select the surfaces in the cylinder fixture for shell separation. Also, the total number of surfaces is reduced from > 600 to < 300, as see in the bottom of the Model Tree:



Chapter 3 – Using Offset to Shell command

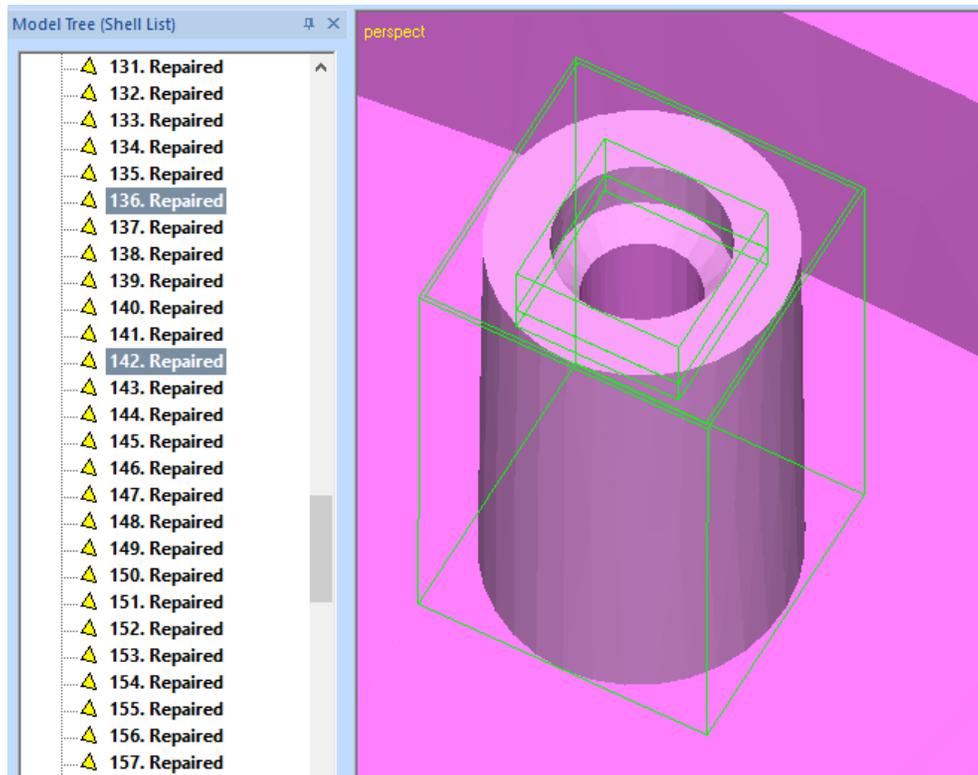
First we will use offset function to separate shells from the cellphone model. *Offset to Shell* command will offset all multi-selected surfaces and create new shells from them.

First **set on the Surface selection mode** through the *Selection mode* buttons:



Let's now select surfaces for shell separation:

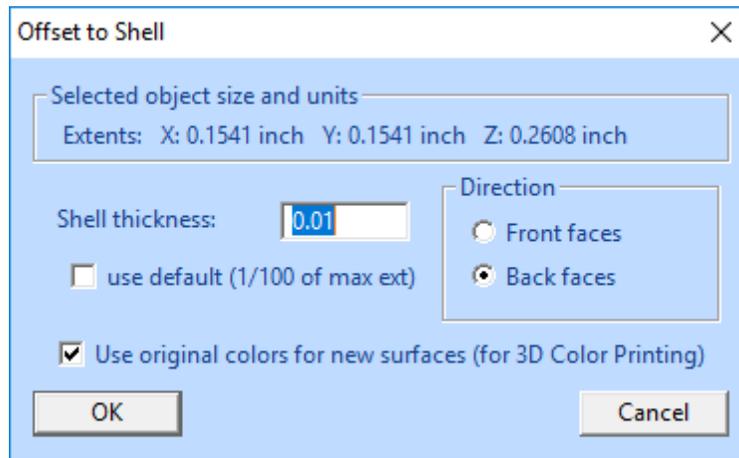
- **rotate and zoom** the view to see the surfaces to select
- **use graphical multi-selection** (CTRL + left mouse button) to select the four (4) adjacent surfaces as seen in the image below



- give command *Print Preparation > Separate to Shell > Offset to Shell*

3Data Expert 12.1: Tutorial 3 - Separating Shells

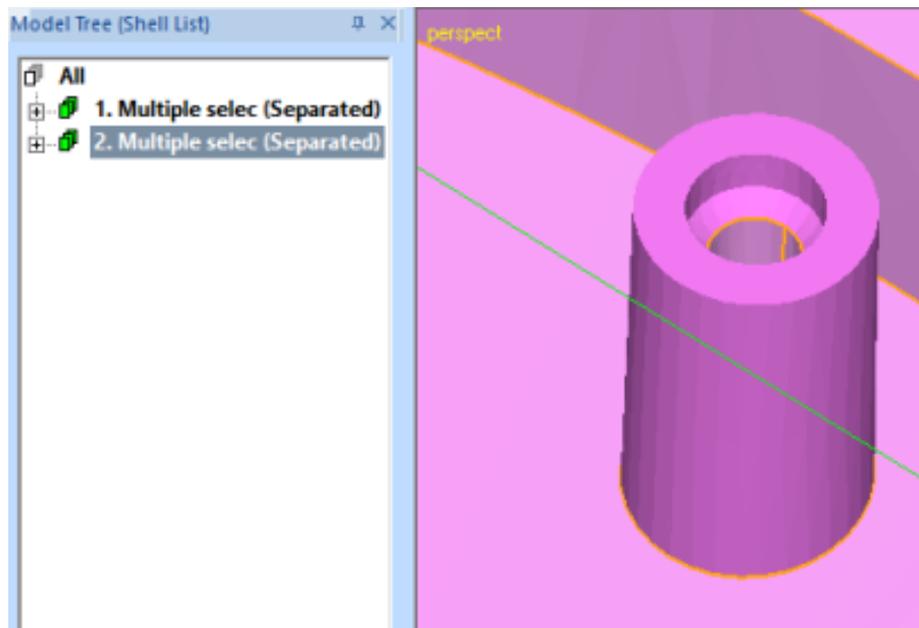
After the command is given a parameter dialog is displayed with the default parameter values. **Fill in the values given below (0.01 inch)** and *uncheck* the *use default* check box:



The given settings will produce a solid offset of the selected surfaces. The offset will be inside the mode (*Back faces*) and the thickness is 0.01 inch. Normally, you should use as small thickness as possible to avoid potential errors with the offsetting.

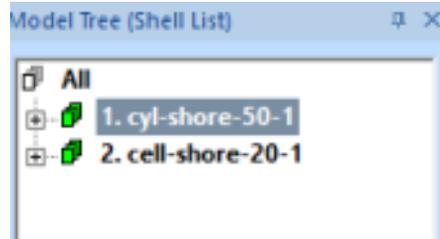
- **Now press *OK* to generate the separate shell** from the selected surfaces.

After a while you will see the result in the Model Tree. The original shell is now selected and stored as the last shell in the Model Tree and the separated smaller shell(s) is just in front of the original shell. *Separated* suffix is added to both shells.

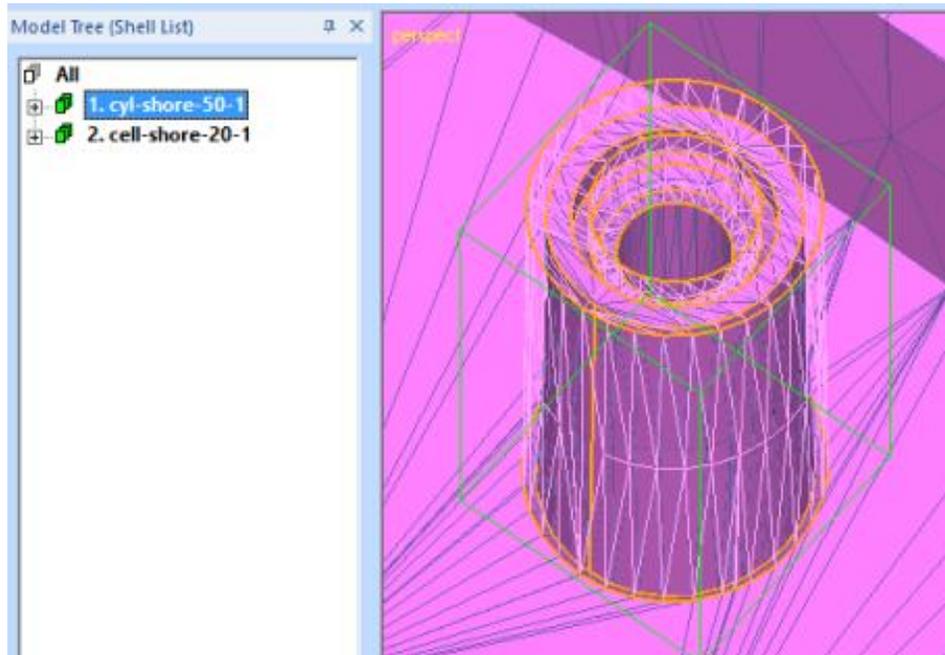


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For clarity, lets now rename the shells with names describing their material composition and to output them correctly to disk files later. We will rename the first shell (from the cylinder) as *cyl-shore-50-1* and the original part with *cell-shore-20-1*, as seen in the image below (shore value is used to describe the “rubberness” of a material):



The *1. cyl-shore-50-1* shell is a fully separate solid shell laying on top of the original model with shrunk cylinder. The thickness of the shell is approximately 0.01 inch and it follows accurately the cylinder left in the original model:

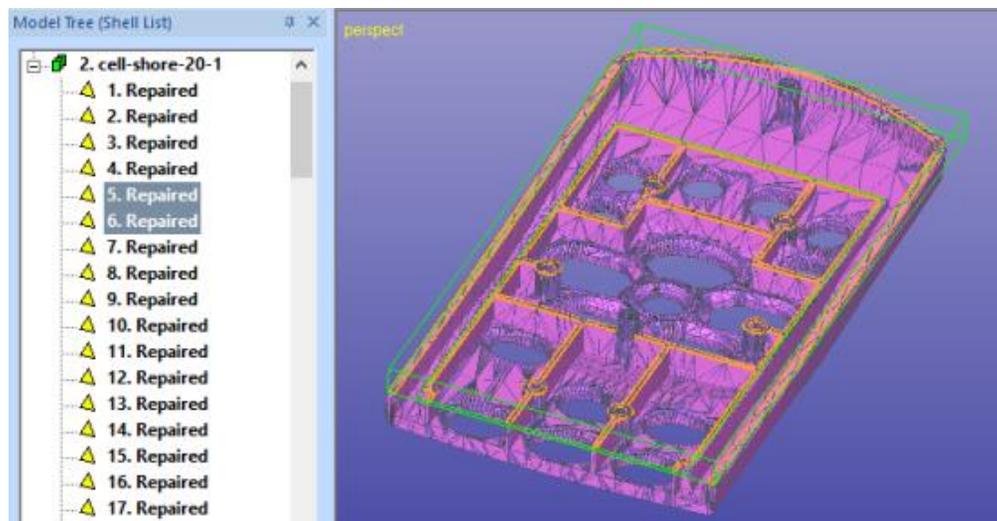


Chapter 4 – Using Extrude to Shell command

Now we will use extrude function to separate shells from the cellphone model. *Extrude to Shell* command will extrude all multi-selected surfaces in a model and create new separated shells.

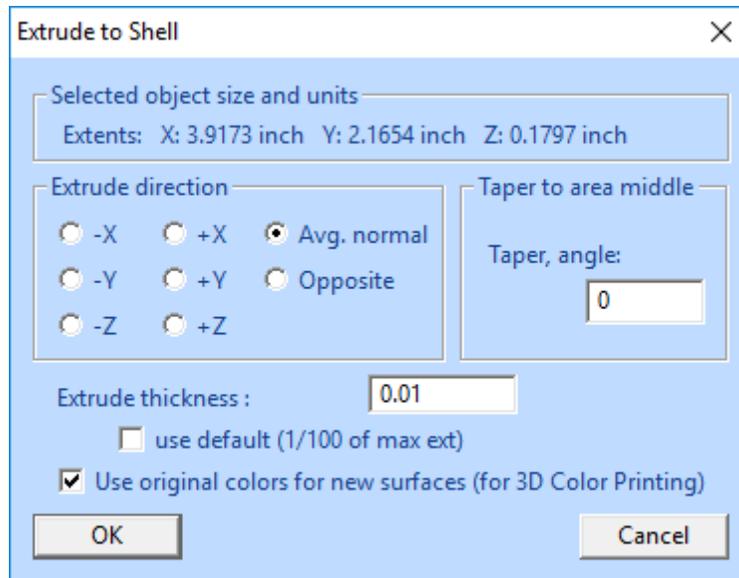
First we will select suitable surfaces for shell separation in the model:

- **set *Surface selection*** mode on
- rotate and zoom to correct view point
- **use graphical multi-selection** (CTRL + left mouse button) to select the tow long and thin surfaces going around the model as seen below



- **give command *Print Preparation > Separate to Shell > Extrude to Shell***

After the command is given a parameter dialog is displayed with default parameter values. **Fill in the values given below** (*thickness 0.01 inch*), direction *Avg. normal* (average surface normal) and *uncheck* the *use default* check box:



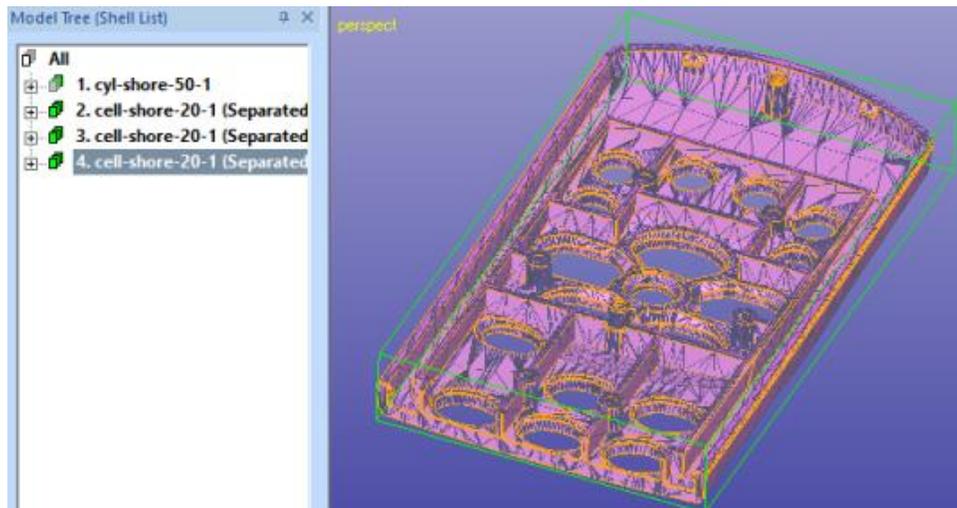
Our goal is to create thin separate shell on top of the original model walls. *Extrude to Shell* can be used especially when separate shell is created by adding material on top of the original model.

Note: Tapering is useful when extruding convex areas towards the inside the model to avoid surface intersections at the extrude area edges. Normally you should use Offset to Shell when separating shells towards the inside of a part, as seen in the previous example.

When ready,

- **press OK** to generate the new shell on top of the thin walls.

After a while you will see the results in the Model Tree. The original shell is once again the last shell in the Model Tree and the two new shells just in front of the original last shell. *Separated* postfix is added to all resulting shells.



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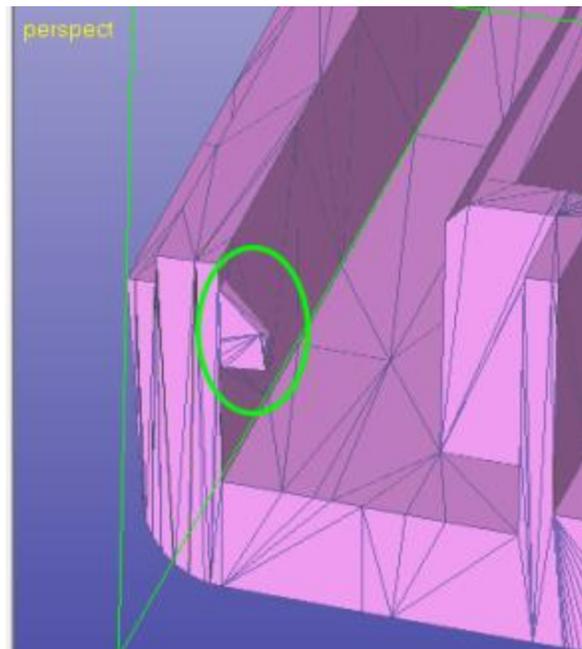
Rename the new shells with unique names, *edge-shore-50-1* and *edge-shore-50-2* as seen in the image below:



Chapter 5 – Using Fill to Shell command

The final special shell separation command is the *Fill to Shell* command. This command will separate and fill all multi-selected surface set and create new separated solid shells.

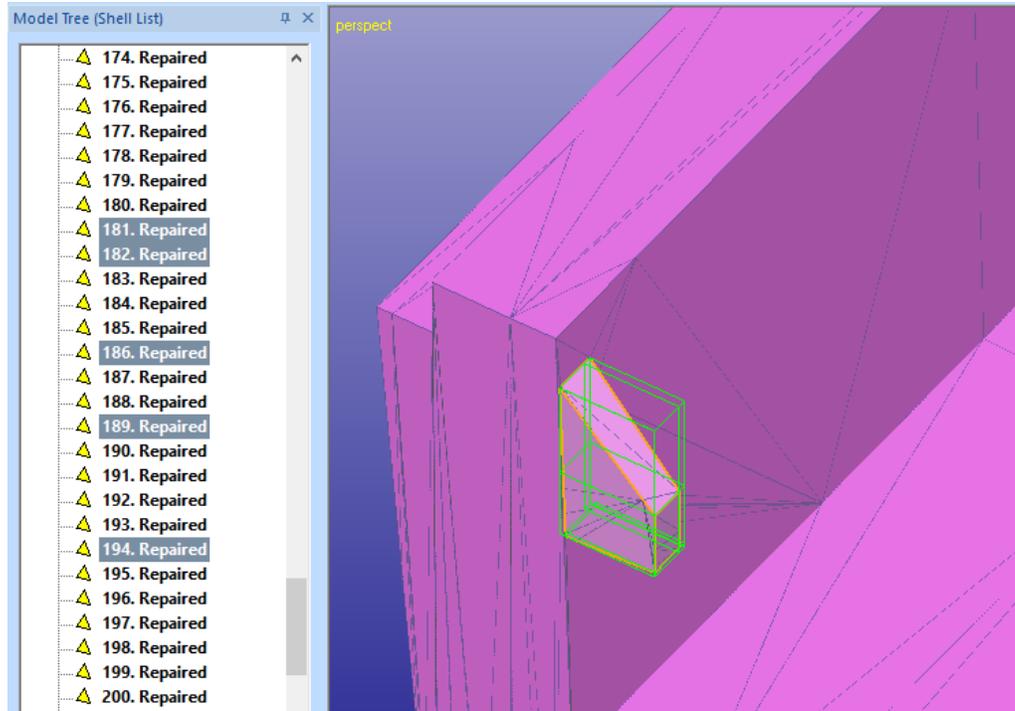
The *Fill to Shell* command is best applied to a set of surfaces which create a nearly closed volume or where the gaps created at the separation can be filled with normal gap fill. For example, the four small fixtures in the cellphone model meet this criterion. One suitable fixture is circled below:



Now let's separate the surfaces making the fixture into a separate shell:

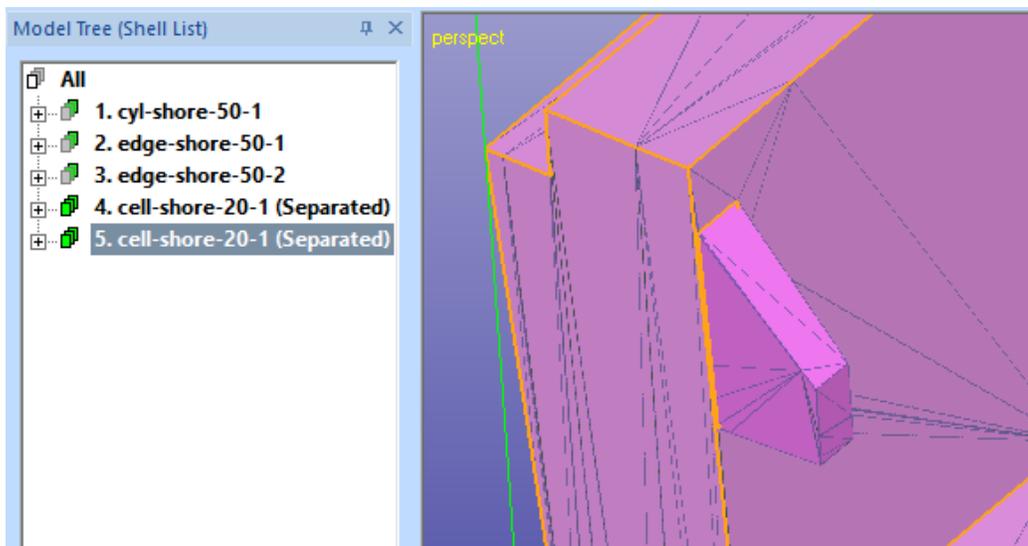
- **set *Surface Selection* mode on**
- **use graphical selection to select the surfaces** show in the image below (*Ctrl + LMB* or *Alt + LMB* for rubberband selection)

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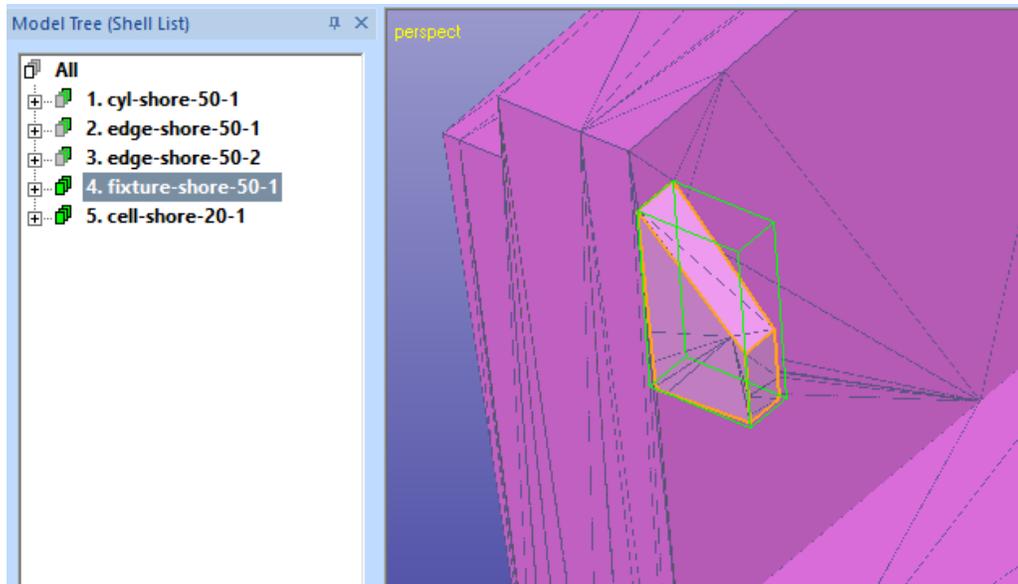
- give command **Print Preparation > Separate to Shell > Fill to Shell**

After a while you will see the fixture separated and the solid result in the Model Tree. The last shell in the Model Tree will be the original part and the shell before that is the small separated fixture:



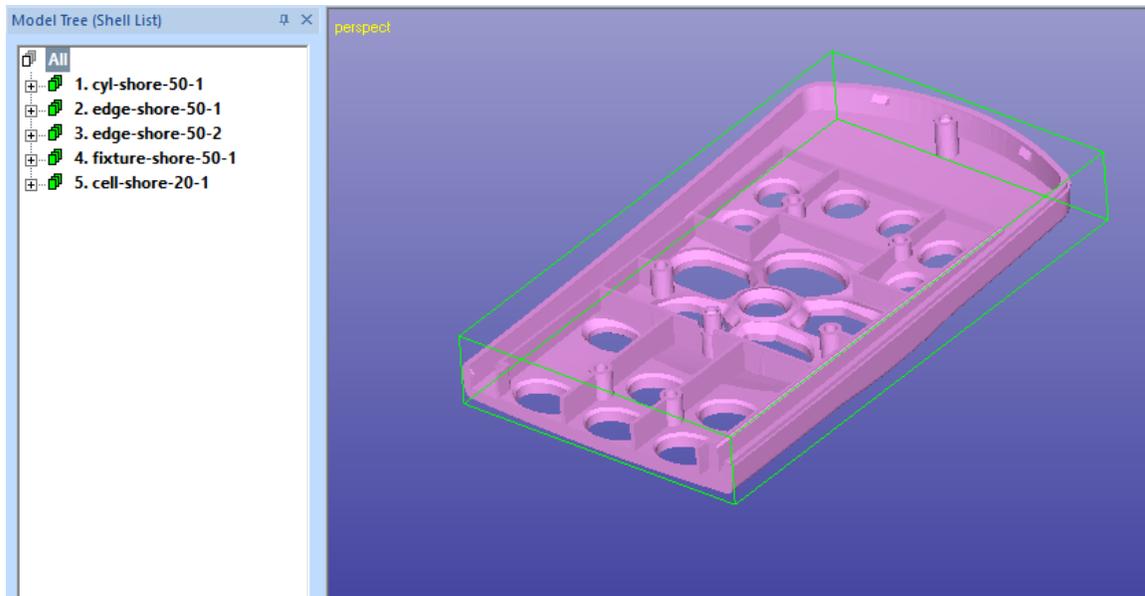
Rename the fixture with a suitable unique name (*fixture-shore-50-1*) for output later:

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Outputting separate STL files

To finish the shell separation process we want to output the shells into separate STL files. We have already named each shell with a unique name which will be used for the disk file names too, as seen in the image below:

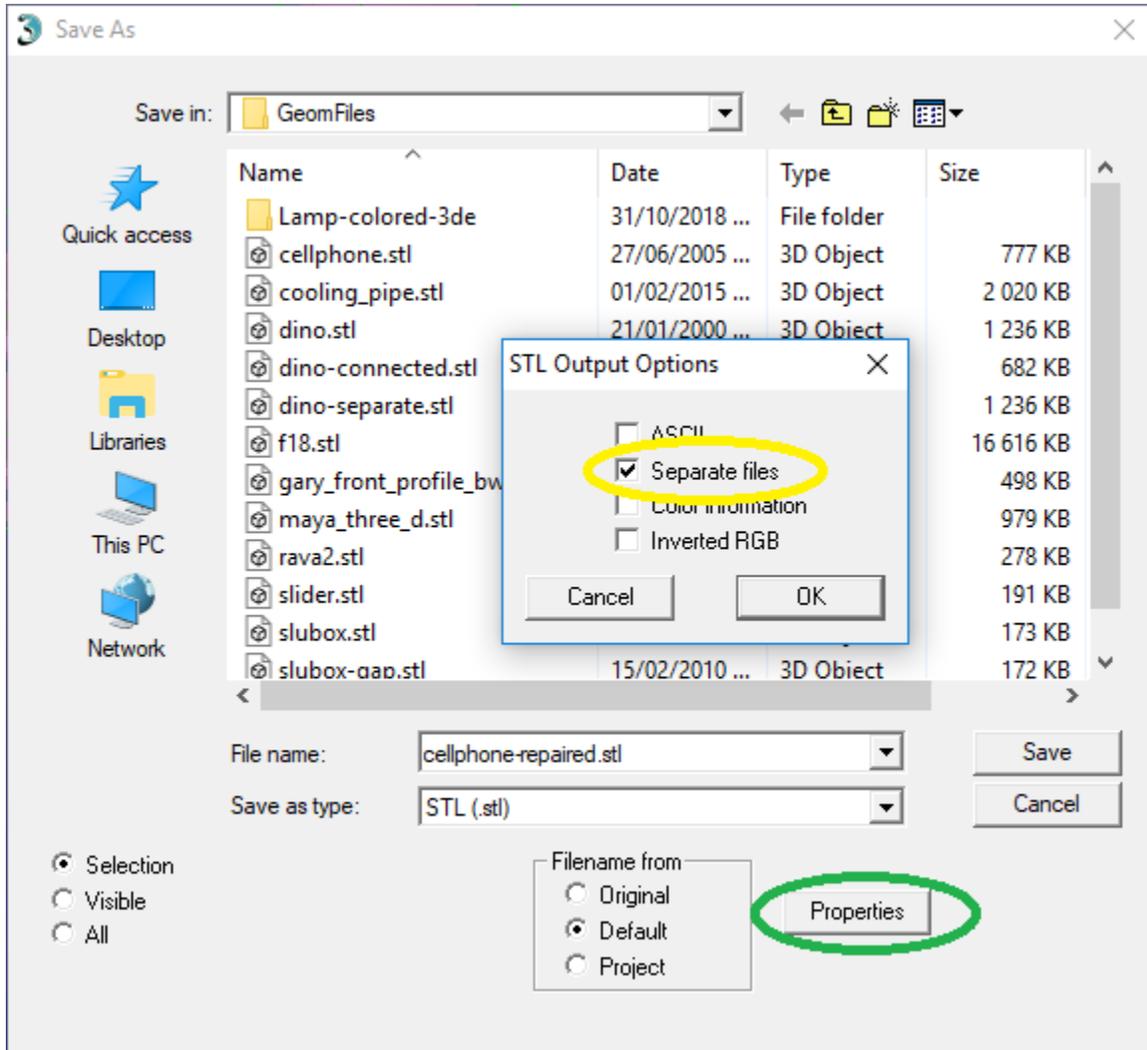


To output the files

- **select Model Tree root *All***
- **give command *File > Save As...***
- **make sure *.stl* file type is selected**

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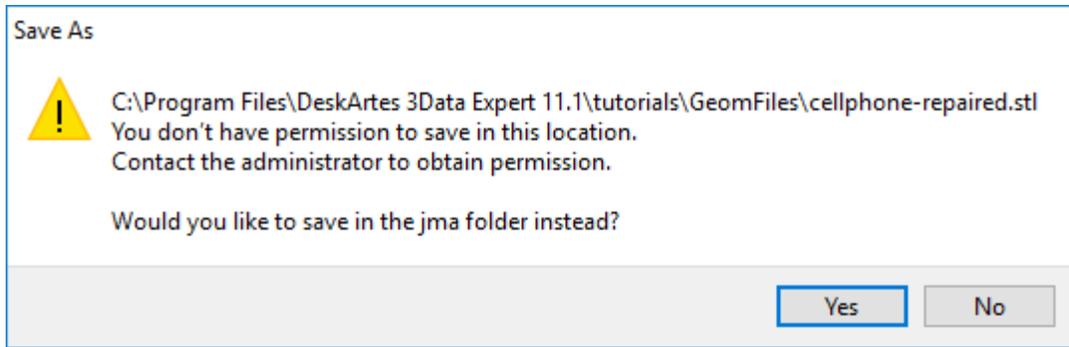
- **press *Properties*** (circled in green below) **button** in the Save As dialog
- **select *Separate files*** (circled in yellow below) **setting in the *STL Output Options* dialog**



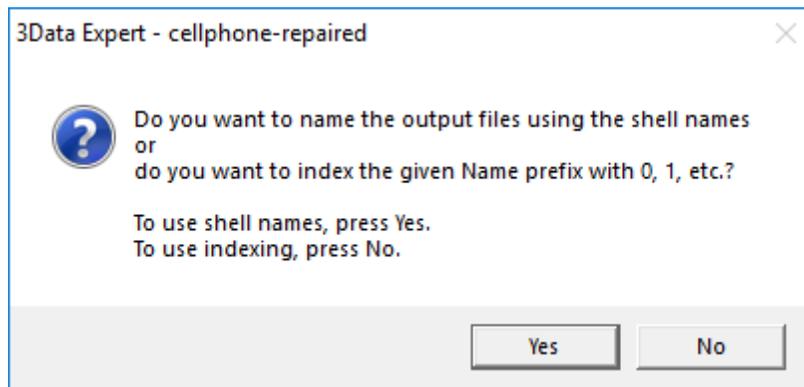
- **press *OK*** in the STL Output Options dialog
- **press *Save*** in the Save As dialog

If you get a following dialog complaining the permission to save, **press Yes** and select a folder where your user account has write access.

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Then a query dialog appears:



- **press Yes** to use the shell names for file names during the output

The shells are written in separate disk files with the shell names. These files can be inputted to receiving 3D printing system for different color or material settings.

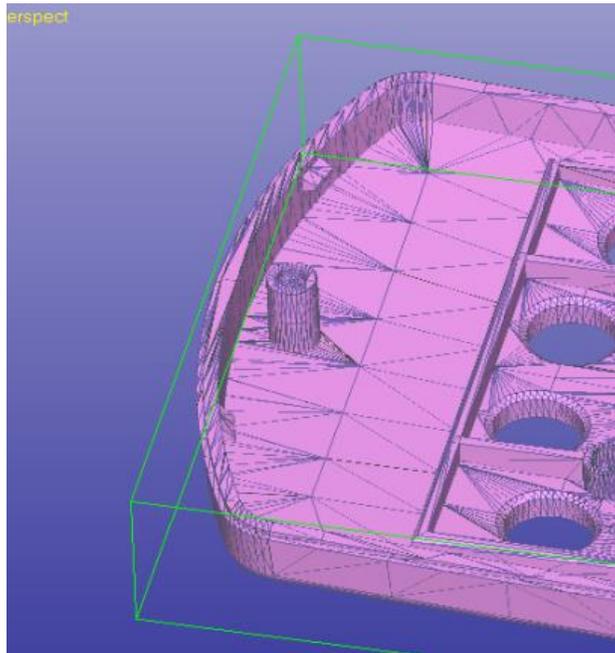
Chapter 6 – Using Cut Tool to separate shells

The easiest and most recommended way to generate separate shells from a part is described in Chapters 1 to 5. In this and following chapters we will take a look at some other efficient but not so straightforward methods to create separate shells from a connected model.

One efficient way to generate separate shells from a solid shell is to use *Modify Faceted > Cut Tool* command. This Chapter shows you how to use Cut Tool command available with 3Data Expert for separating model areas into separate shells.

First open the cellphone-repaired.3de file with the *File > Open* command. Accept the proposed units (inch) and close the Operation Mode dialog by pressing the close box in the upper right corner of the dialog. The model is already fixed so we do not need to perform any fixing.

We will first separate the largest fixture in the round end of the cellphone model:



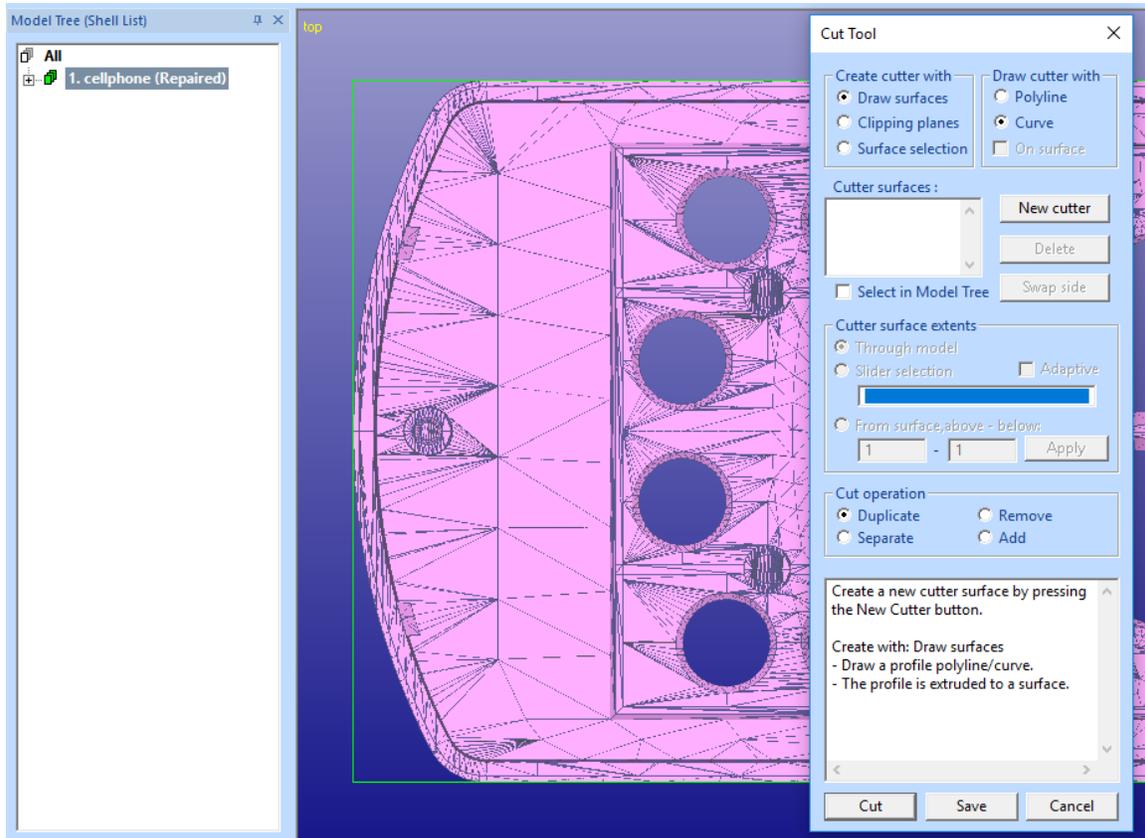
Separating the largest fixture

To separate the fixture, start with the following commands:

- **take a view** from the Z axis direction
- **select the 1. cellphone (Repaired)** shell from the Model Tree
- **give command *Modify Faceted > Cut Tool***, a Cut Tool dialog appears
- make sure ***Draw surfaces selection is set*** in *Create cutter with group*

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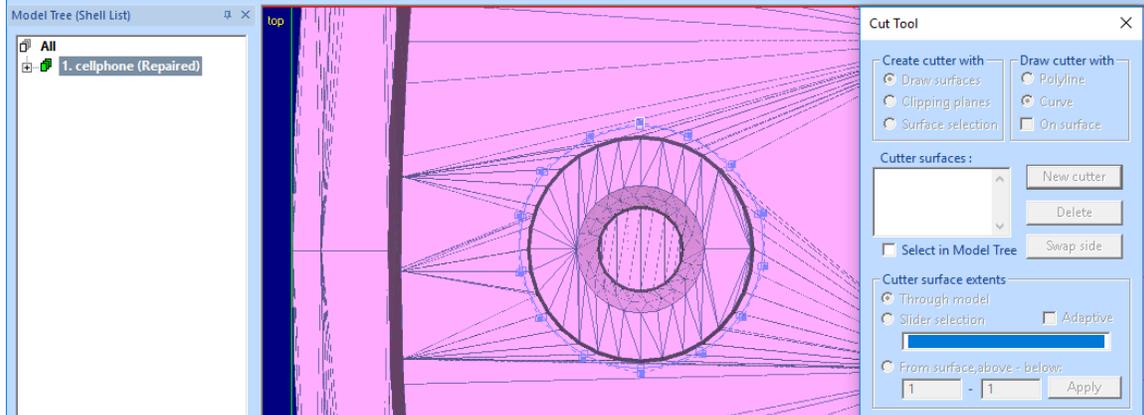
- **select *Curve type*** in *Draw cutter with* group because now we want to draw a smooth curve around the separated area instead of a polygon



Then we continue to draw the separated area:

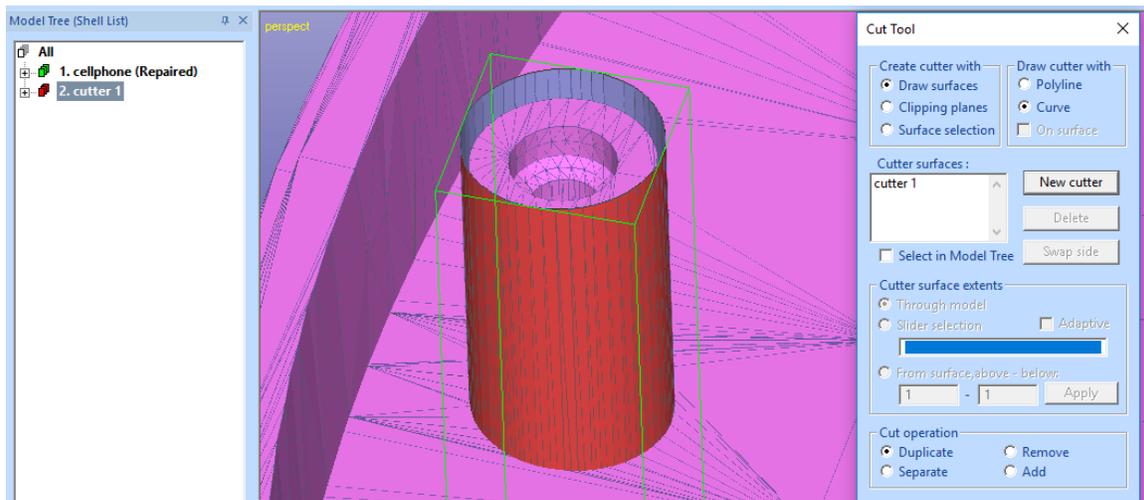
- zoom closer to the model, if necessary
- **press *New cutter* button** to start inputting curve around the fixture
- **continue adding curve points** with the crosshair icon evenly around the fixture so that the curve does not overlap the vertical walls of the fixture
- when ready, **close the curve by picking the first point again** with the crosshair icon
- you may **select and move the curve points with LMB (left mouse button)** so that the curve is evenly around the fixture

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Now the curve is drawn correctly around the fixture. To finish the cut tool generation

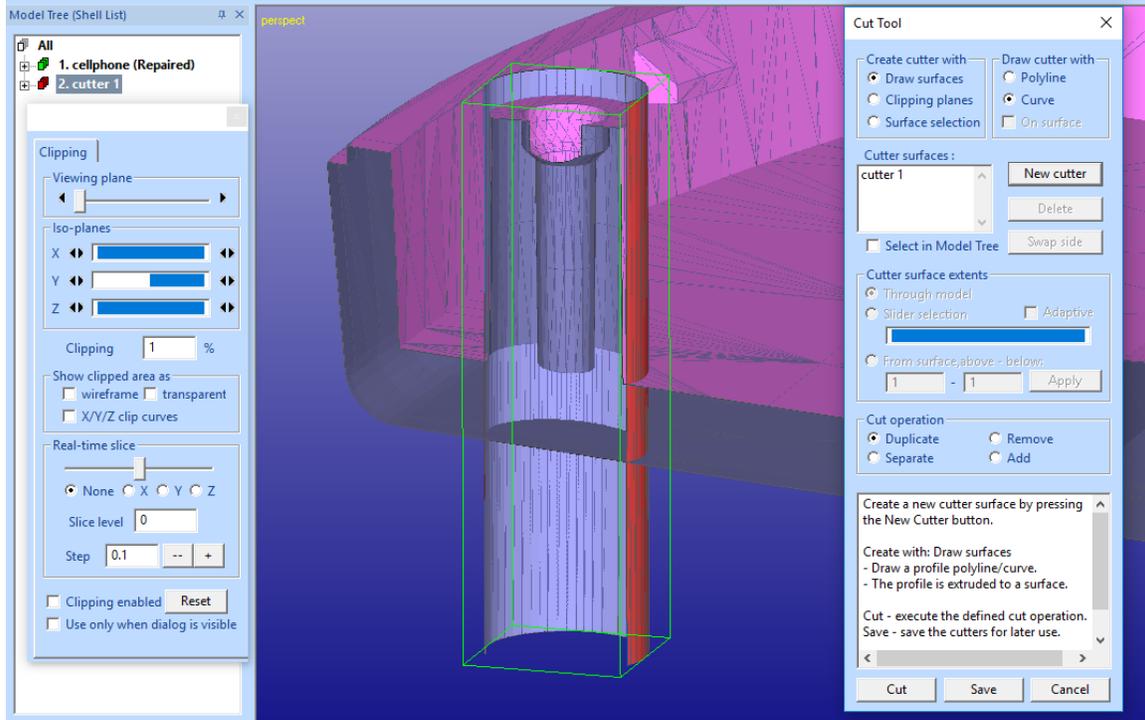
- **press RMB (left mouse button)** to finish the curve edit
- the ***cutter 1 tool appears*** in the Cutter surfaces list and Model Tree and is displayed on the graphics area (rotate the view to see it)



Now we will tune the cut tool to correct depth:

- **select the *cutter 1*** in the *Cutter surfaces* list, this will allow us to fine tune the tool depth
- **select *Slider selection*** in the *Cutter surface extents* group
- **open Clipping Tab** by pressing  icon
- **use Y clip plane** to see the cut tool to go through the whole model

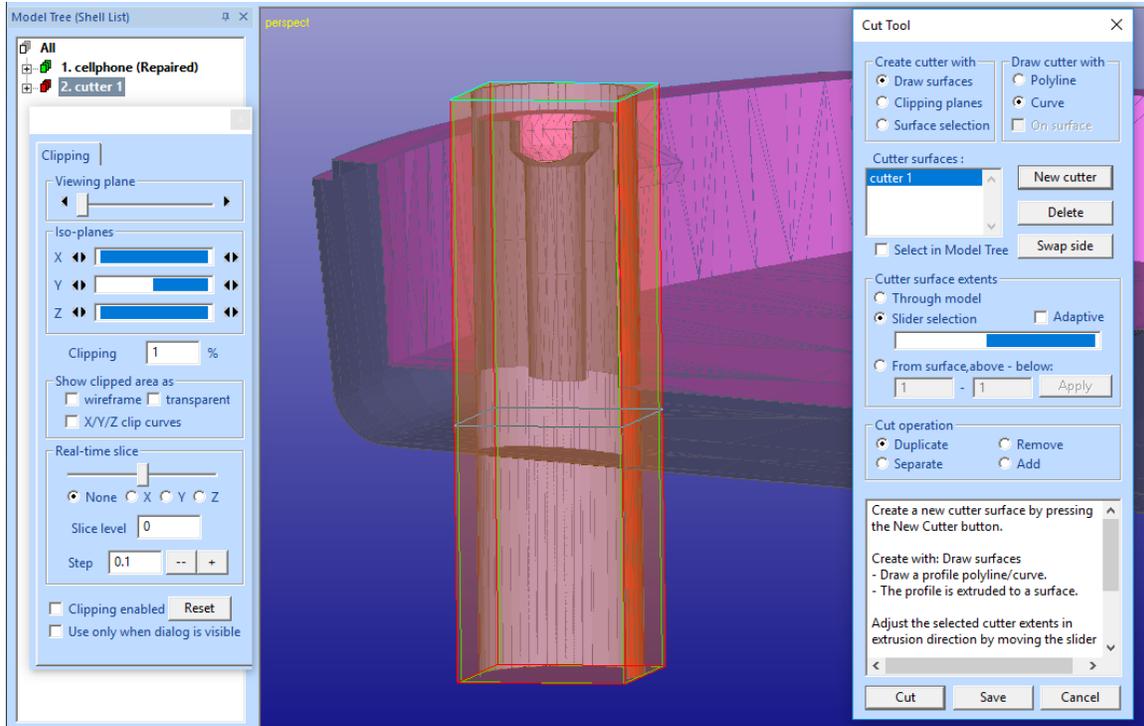
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Now you can set the correct depth for the cut tool. We do not want to go through the whole model, we only want to separate a shell with sufficient contact to the original part:

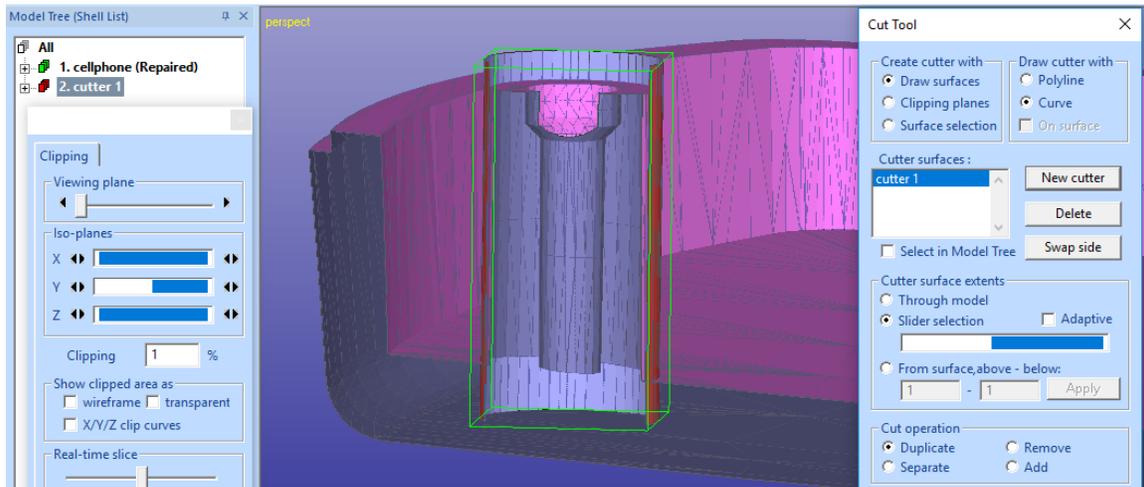
- **start moving the slider** in the *Cutter surface extents* group
- you will see a bounding box with the maximum extent of the cut tool displayed
- **move the left end** of the slider towards the right end with LMB
- the bottom plane for the cut tool is displayed in the bounding box and the tool is cut accordingly, please see on the image next page.

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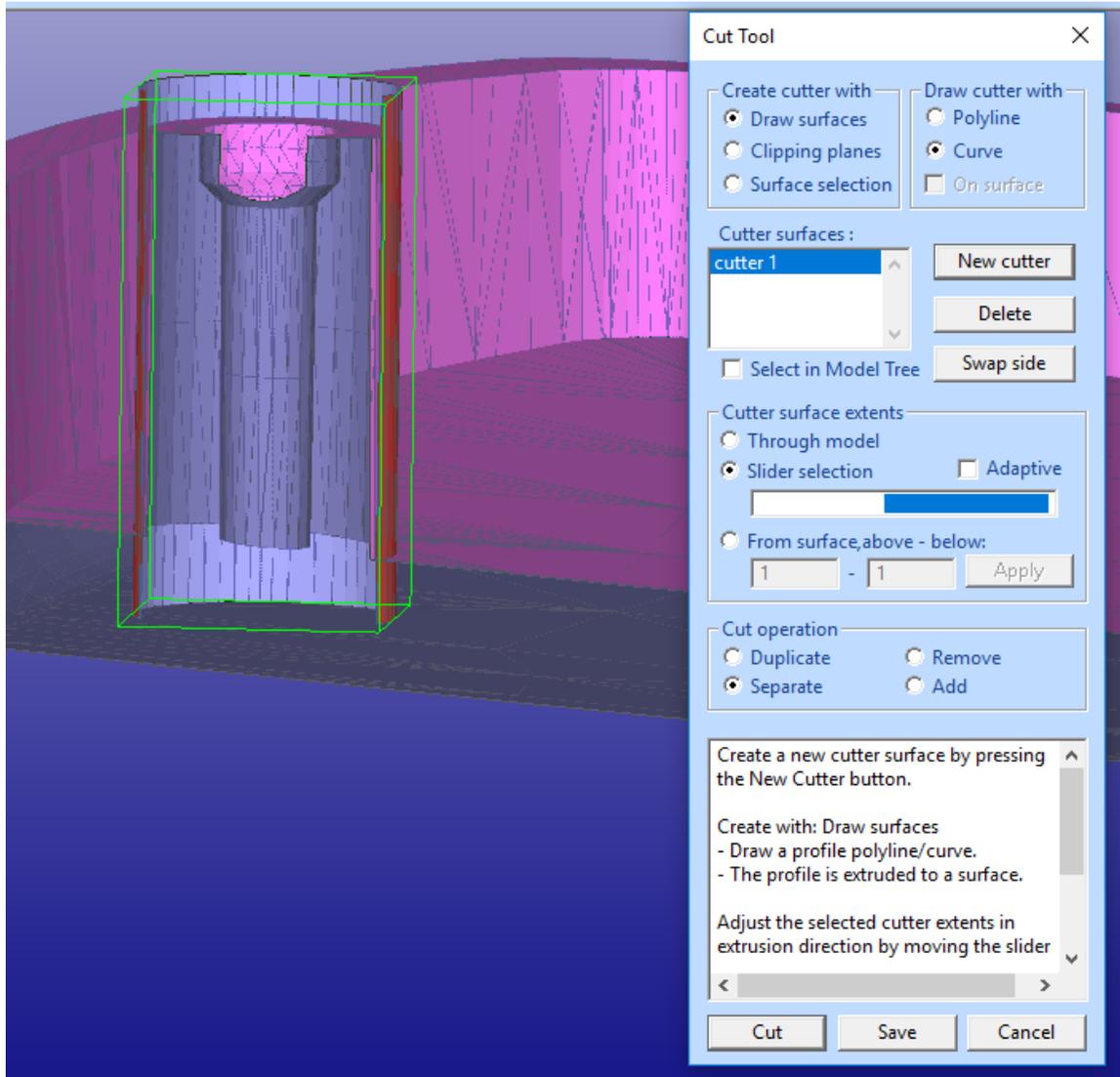
Fine tune the cut tool depth:

- **move the slide *left end*** until the cut tool bottom is somewhere in the middle of the cellphone model, as seen in the image below. The cut tool must be fully intersected by the surfaces in the cellphone body:



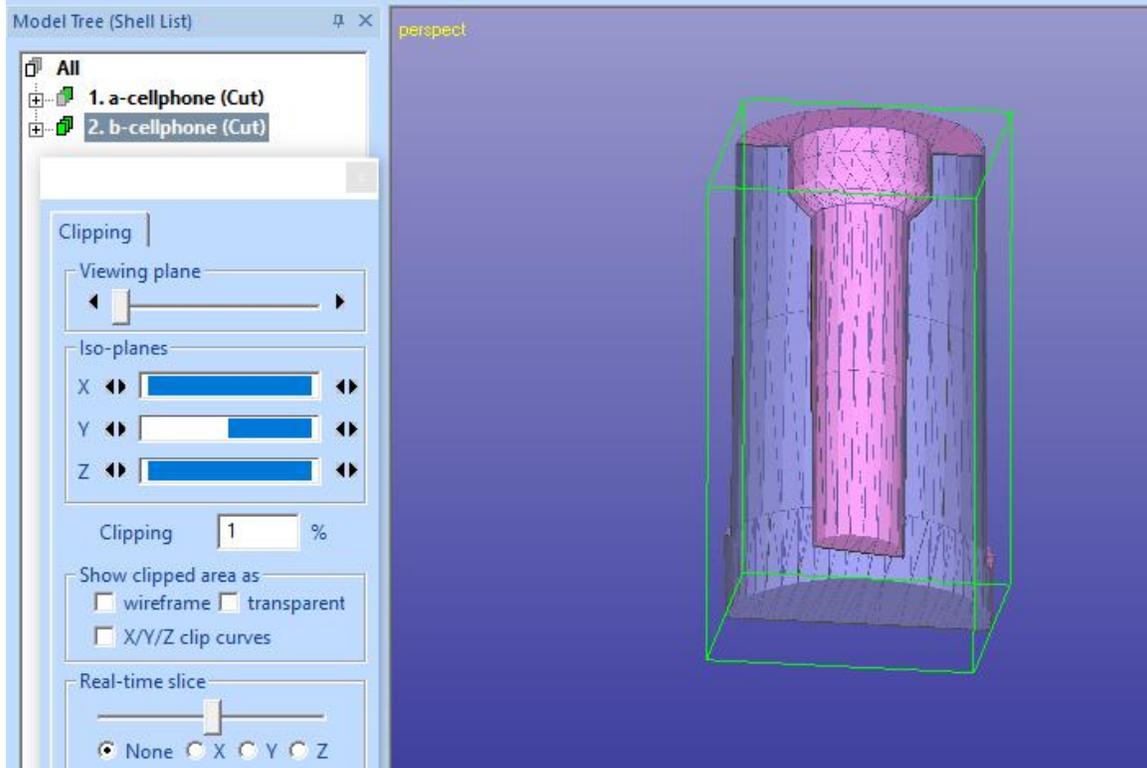
Now we are ready to select the correct operation and to apply the cut tool:

- **select *Separate* operation** in the *Cut operation* group
- **press *Cut* button** to apply the *Separate* operation

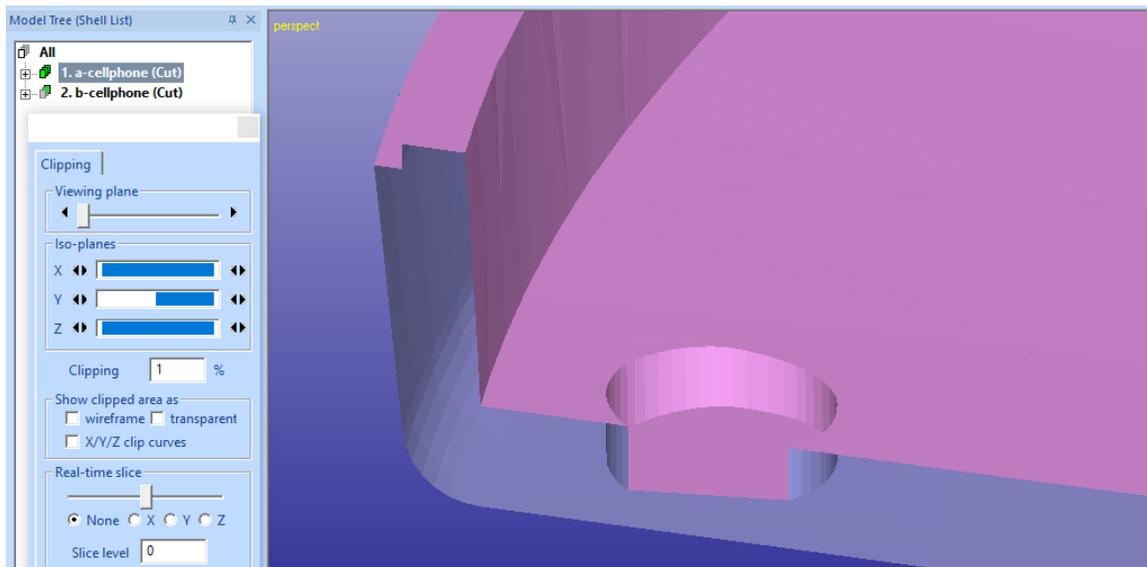


Soon you will see the cut fixture on the graphics area:

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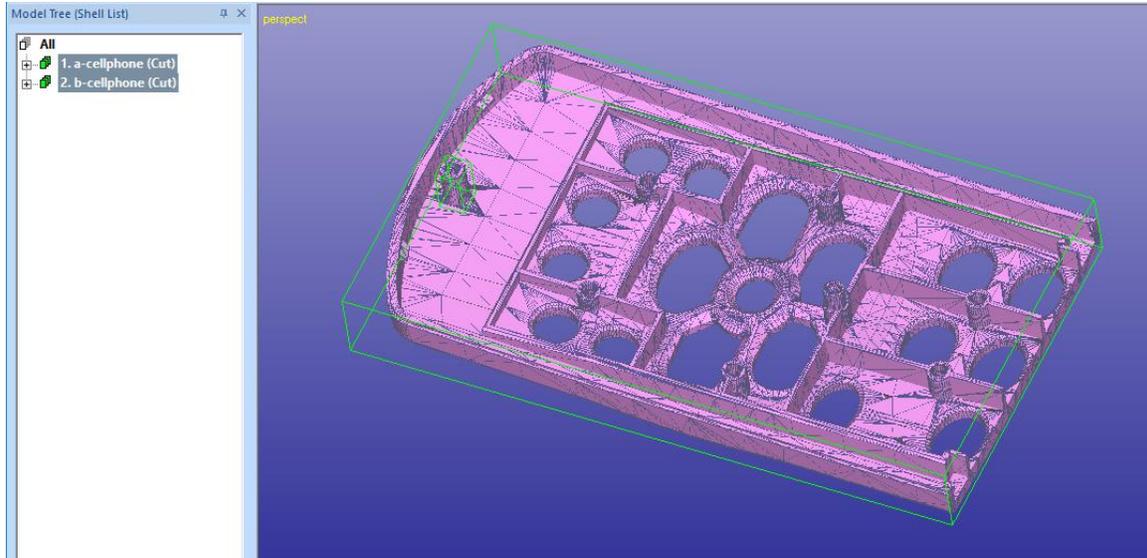


And the cellphone body has corresponding hole:



You may need to run Fix Model > Repair to the body to get rid of minor gap generated to the body during Boolean operation topology generation. Finally, you will have the two separate shells available for different sore value definitions.

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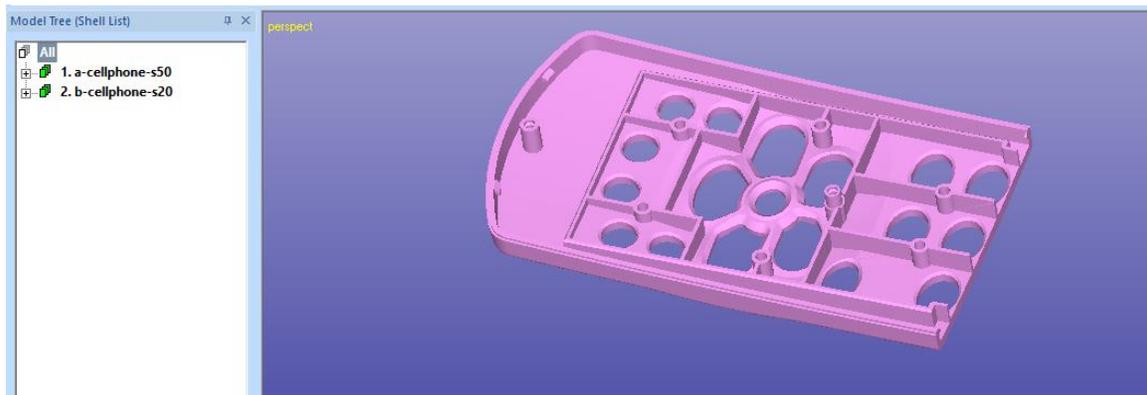


Renaming shells and outputting STL files

You may select and store each shell into a separate STL file just by selecting them in the Model Tree one by one, giving *File Save As* command and defining suitable file name during the file save.

You can also output each shell into separate STL file with one command and in more controlled fashion. For example, the following process would allow you to plan and indicate the correct shore value for each shell before output:

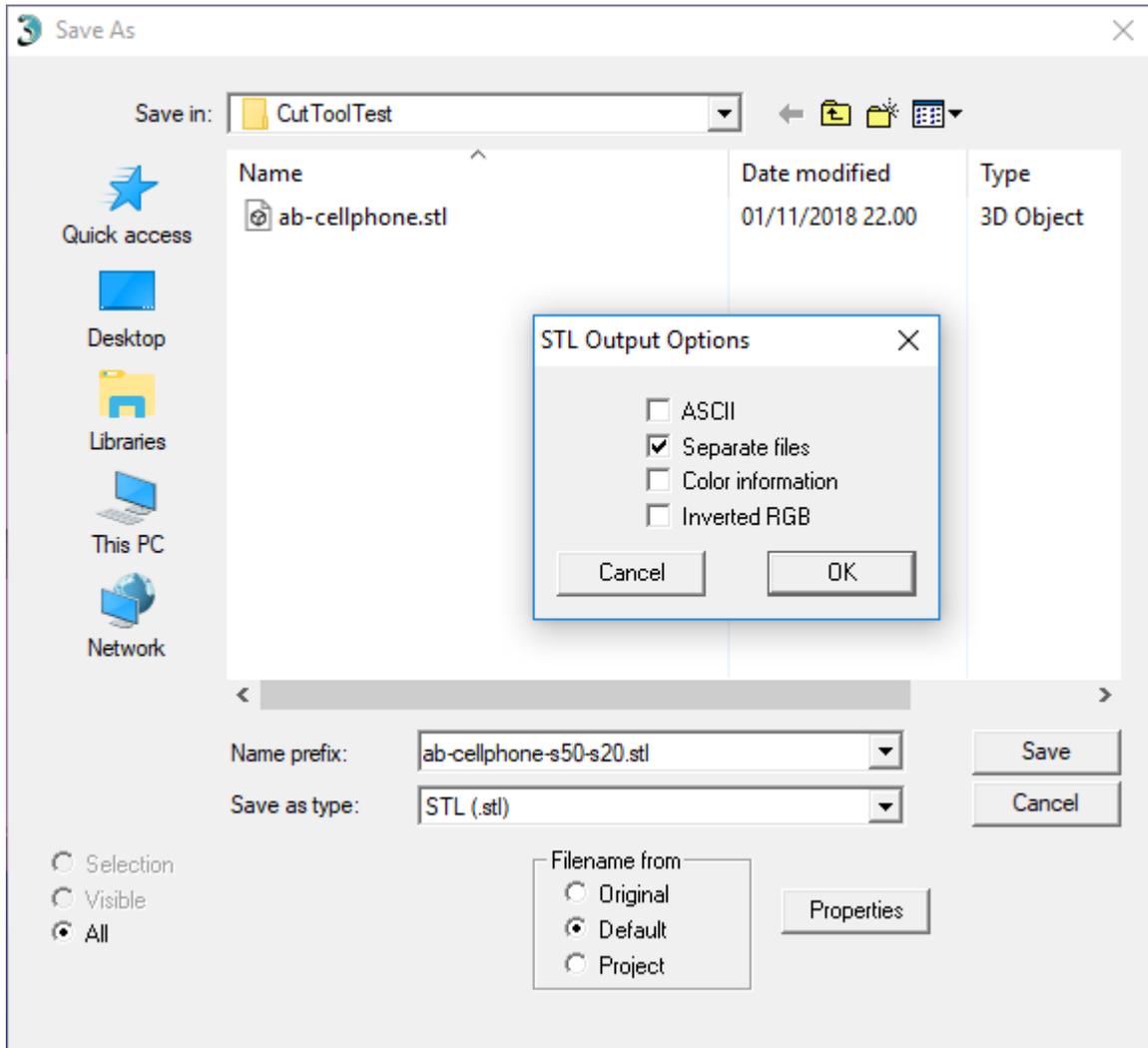
- **rename the two shells** through RMB menu on the Model Tree with indication on the desired shore value, *a-cellphone-s50* and *b-cellphone-s20*, for example. You must make sure that all shell names are different.



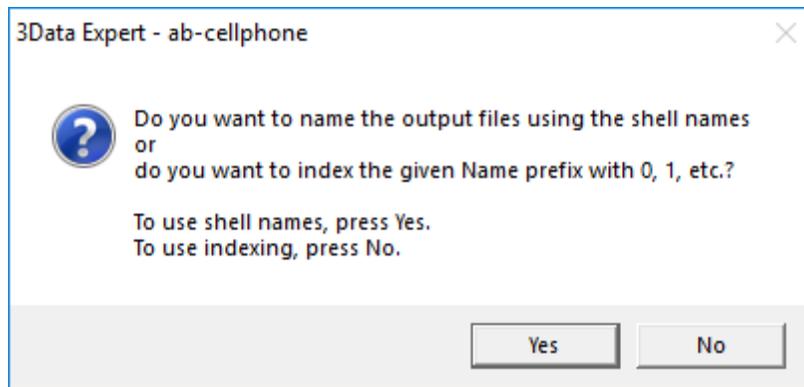
- **give *Edit > File Save As*** command
- **select Save as type: *STL***
- **select *All*** shells for output

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- select Filename from **Default**
- press **Properties** button for **STL Output Options**, select **Separate files**
- press **OK** to close the output options



- press **Save** to start file write, a question box appears



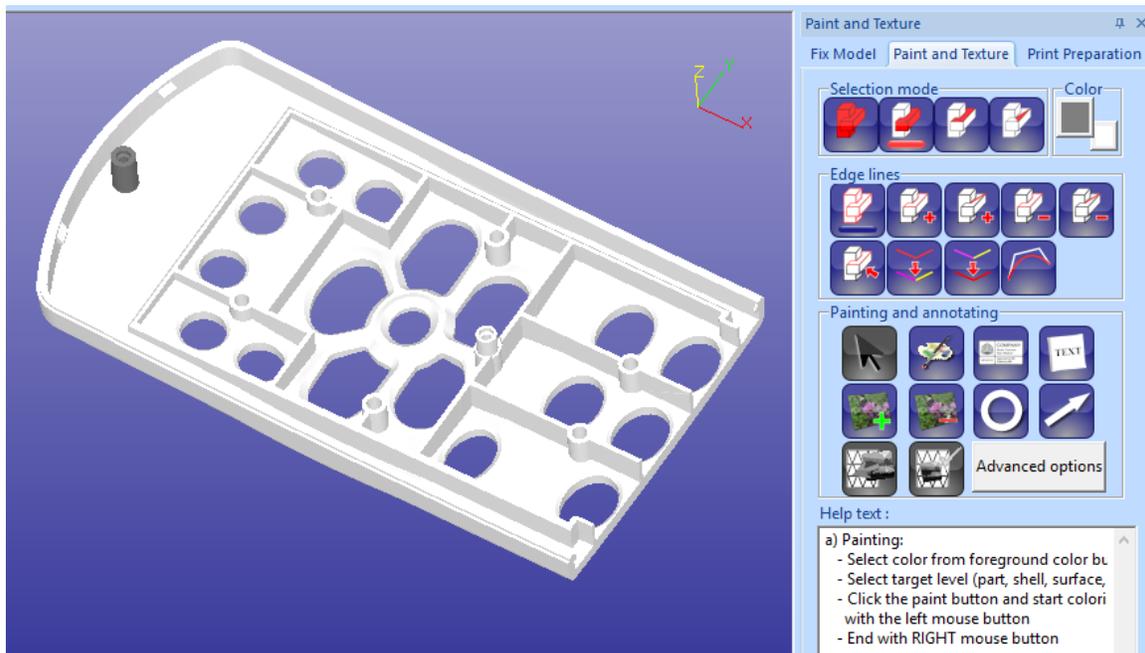
- press Yes to use the shell names for file names

And you are ready, the files with names *b-cellphone-s20.stl* and *a-cellphone-s50.stl* are stored on the hard disk.

Emulating material properties with colors

You can actually emulate the final print result by setting different colors to the shells using the Color and Texture tab commands. Please see *Tutorial 2 – Working with Colors* for more details on painting objects. Colors can be seen

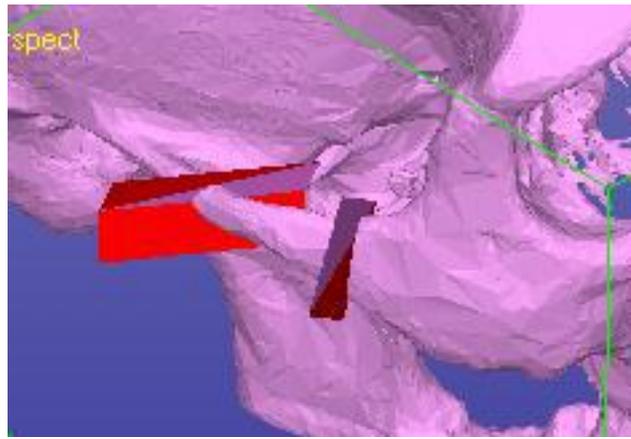
when the 3D Color Printer mode is set through  icon (Note: only colors are available, transparency cannot be defined).



More cutting planes

Cut Tool allows you to define *any number* of cutter planes and they do not need to be closed tubes as was used in the example above. You just need to make sure cutter planes are drawn in consistent order for consistent inside/outside separation, please see more at Help pages for Cut Tool command. Generally, the inputted points for different cut planes should be given either in clockwise or counter clockwise order around the middle of the area to be separated.

Multiple cutter planes can be used to separate geometries in very complicated topologies accurately, like in medical models:

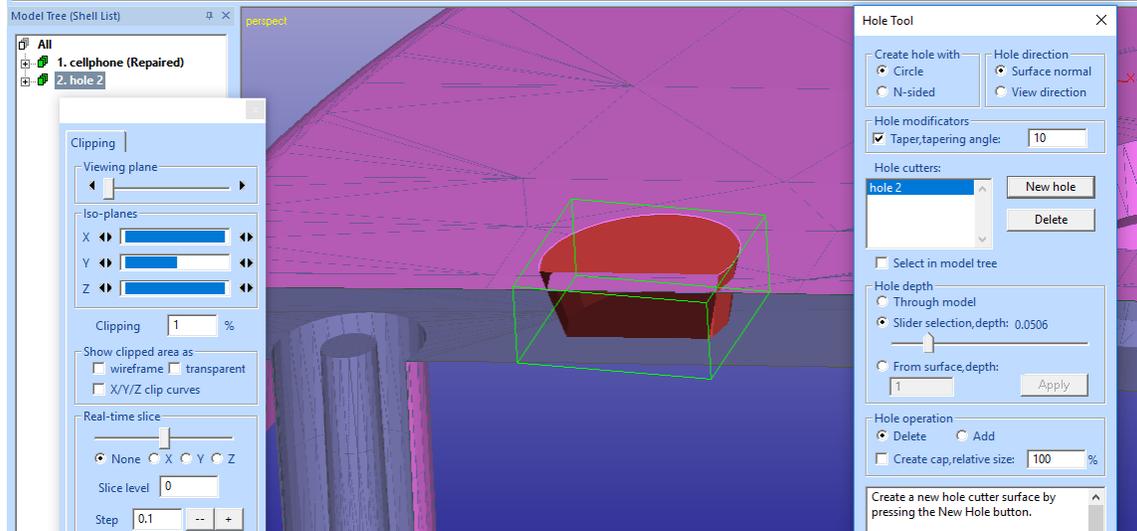


Using Hole Tool to separate small flat surface areas

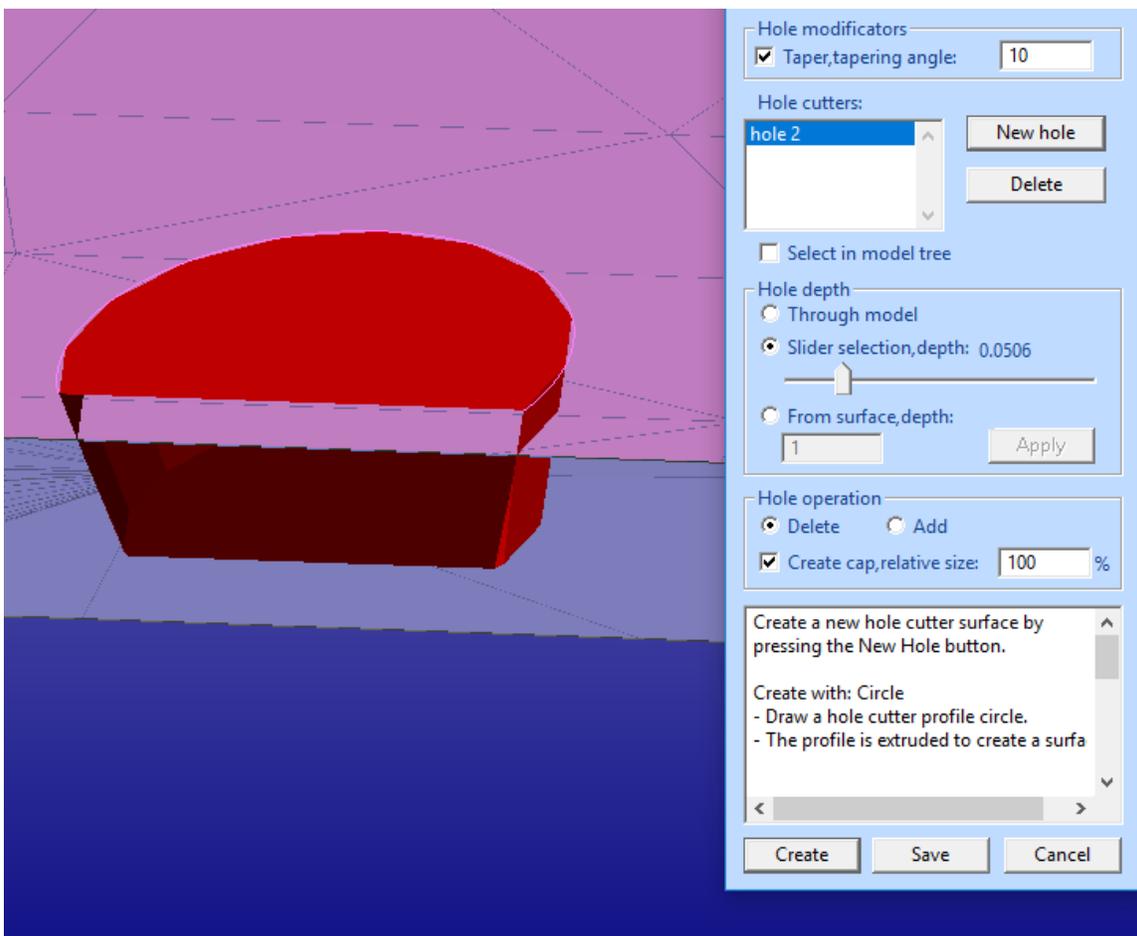
The *Modify Faceted > Hole Tool* command can also be used to separate simple areas from a solid STL model. It works in similar fashion as Cut Tool command, you first define a tool to apply on the model and then separate the desired area. With Hole Tool you can only create circular (smooth or polygon) tools on top of the surfaces.

In the image below we have generated and tuned a circular area on the other side of the cellphone model for separation:

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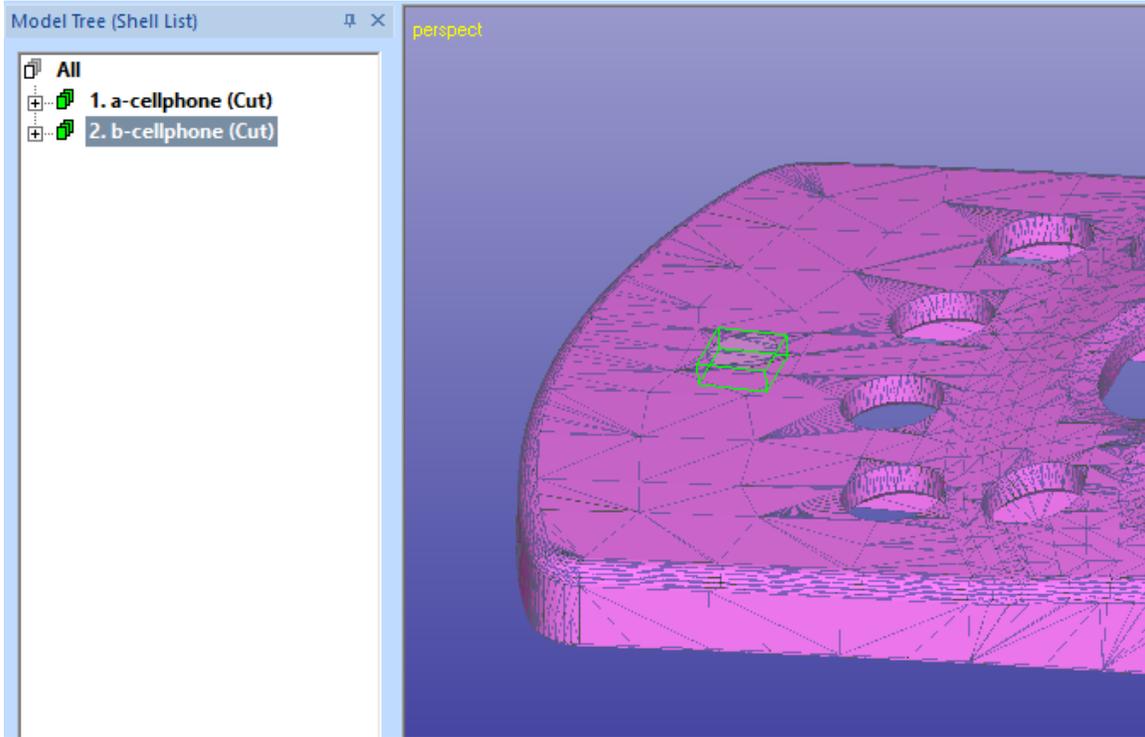


When the depth for the separated round area is correct, select the correct operation parameters (*Delete* and *Create cap*) and **press *Create*** button:



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After a while you will have a round are separated into a new shell:



In case of errors in one of the separated shells, please run *Fix Model > Repair Shells* command on error shells (*1. a-cellphone (Cut)*) to get fully correct components.

Chapter 7 – Using Extrude to separate shells

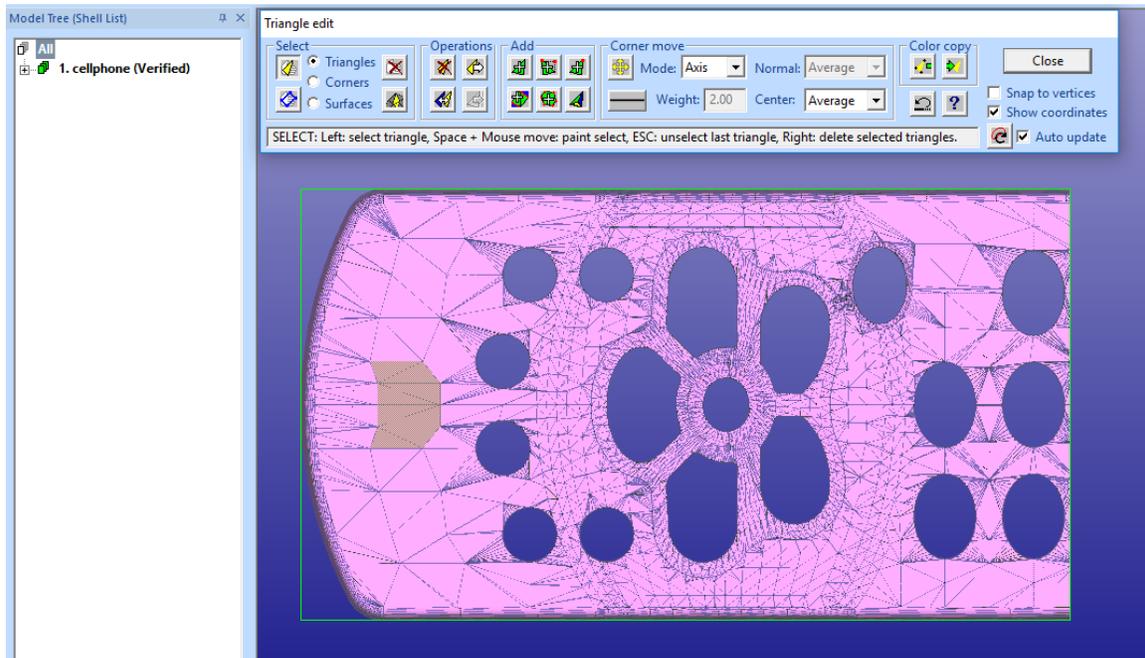
This Chapter shows you how to use *Fix Model > Edit Triangles* to create new surfaces for shell separation with *Create Geometry > Extrude Surface > from Surface* command.

Creating surfaces with Edit Triangles command

Edit Triangles command allows you to select any triangles and separate them into a new surface for Offset or Extrude command. We will use the same cellphone-repaired model for this example too.

Start by:

- input the **cellphone-repaired.3de** model, continue through the units and Operation Mode dialogs
- take a view from **-Z direction** (bottom)
- select Model Tree root **All**
- give **Fix Model > Edit Triangles** command
- use **LMB** to select desired triangles from the model, as seen below, grayed

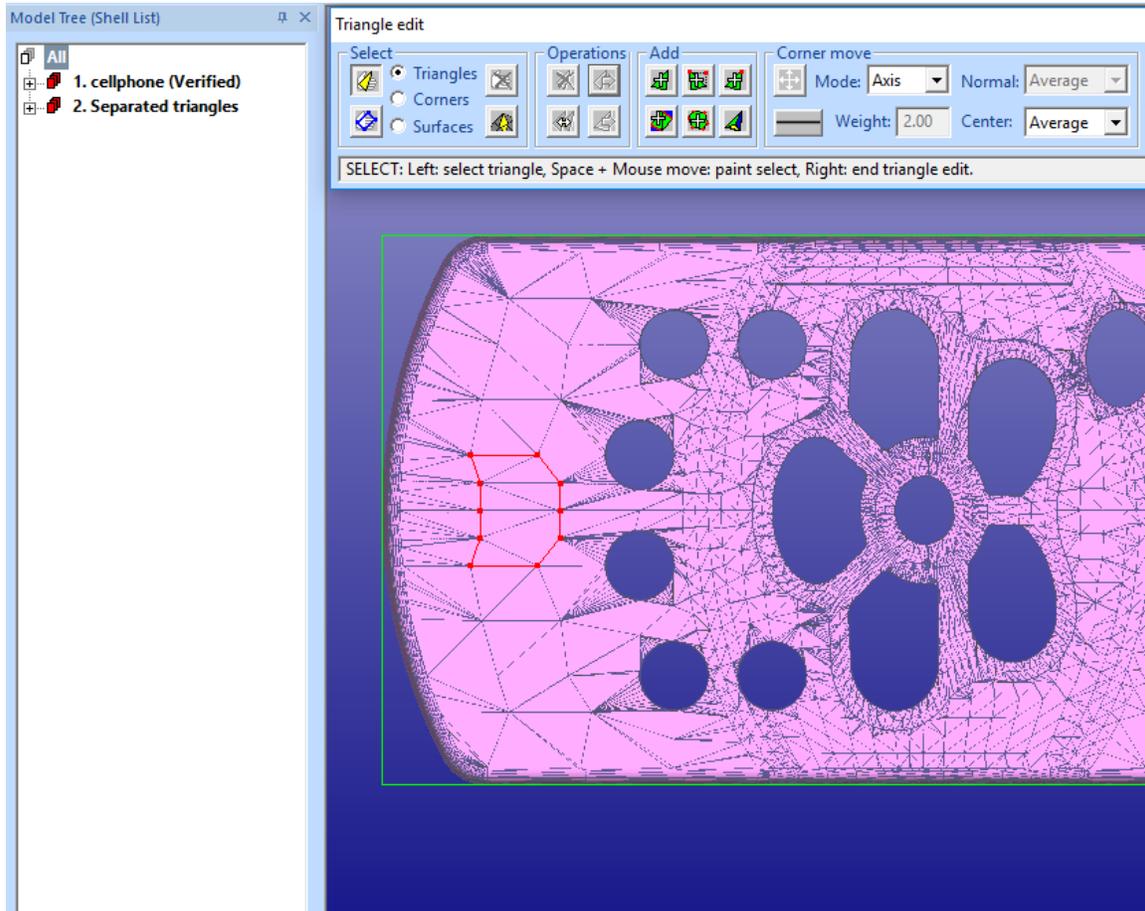


When you have selected the triangles, it is time to separate them into a new shell. We have the Model Tree *All* selected, this causes the separated triangles to be stored automatically into a new shell:

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- press the  icon to create new shell for the triangles

A new shell is created in the Model Tree, 2. Separated triangles. Also, gaps shown on the graphical area indicate that the two shells are no longer solid.



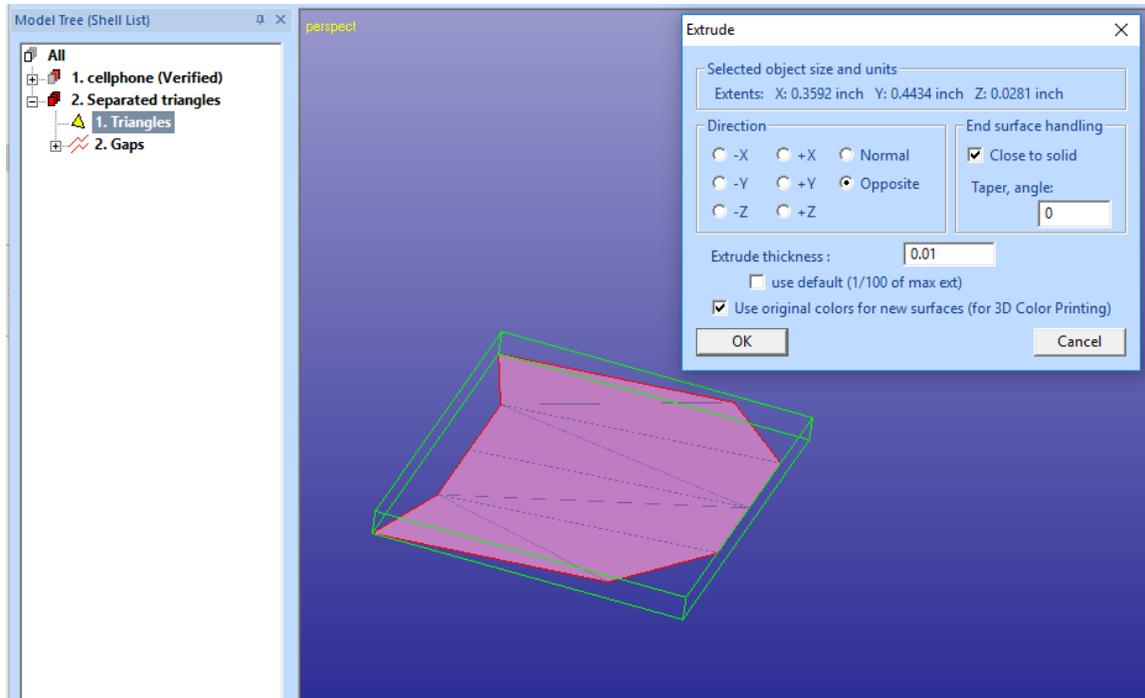
- and then press **Close button** to end Edit Triangles

Using Extrude from Surfaces command

Extrude from Surfaces command can be used to extrude surfaces from solid shells or flat/open shells. We can either select the 2. Separated shell or the surface in it for the extrude command. Let's do the following:

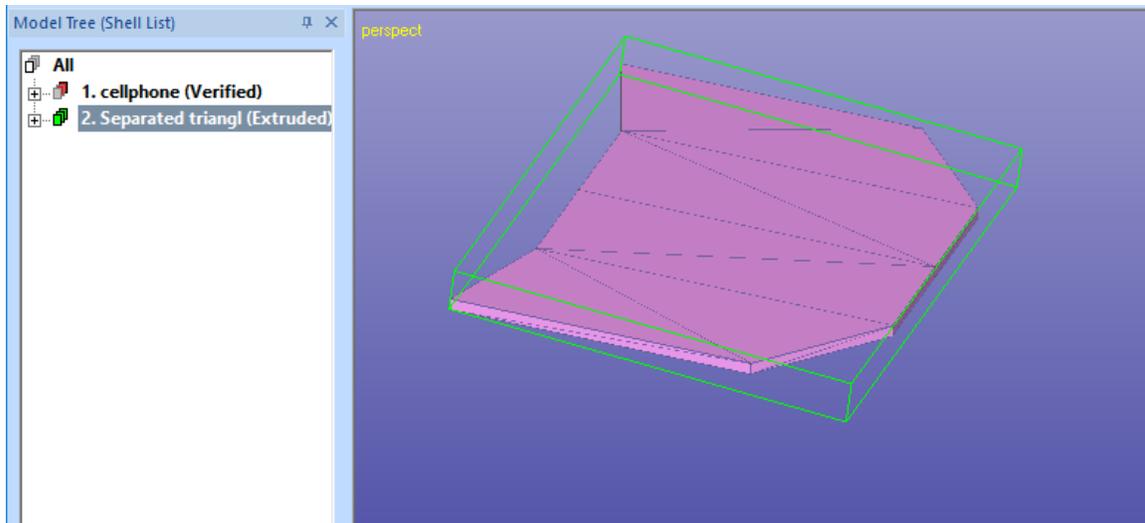
- select the **1. Triangles surface** in the **2. shell**
- give **Create Geometry > Extrude Surface > from Surface** command
- fill in **parameters** as seen in the image next page (*Direction Opposite* and *thickness 0.01 inch*)

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We will generate an extrude towards the inside of the original model (Opposite) and with 0.01 inch thickness.

- **press OK** to create the extrude



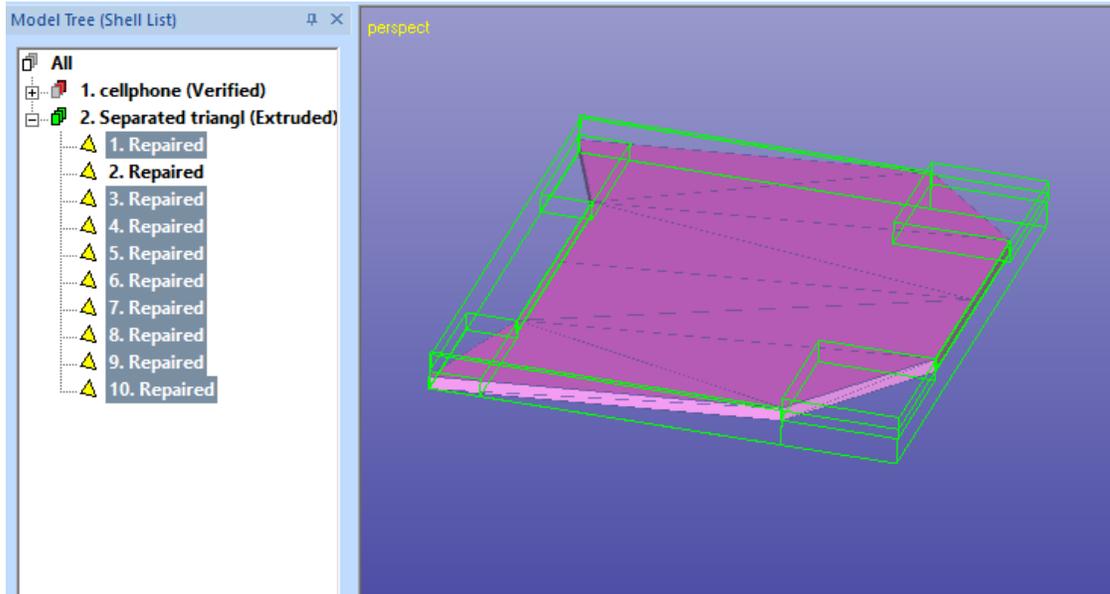
The result is a solid shell with given thickness.

The original cellphone shell is still open and missing the corresponding blind hole on the surface. We now need to move the new surface generated by the Extrude command from the 2. shell into the 1. shell.

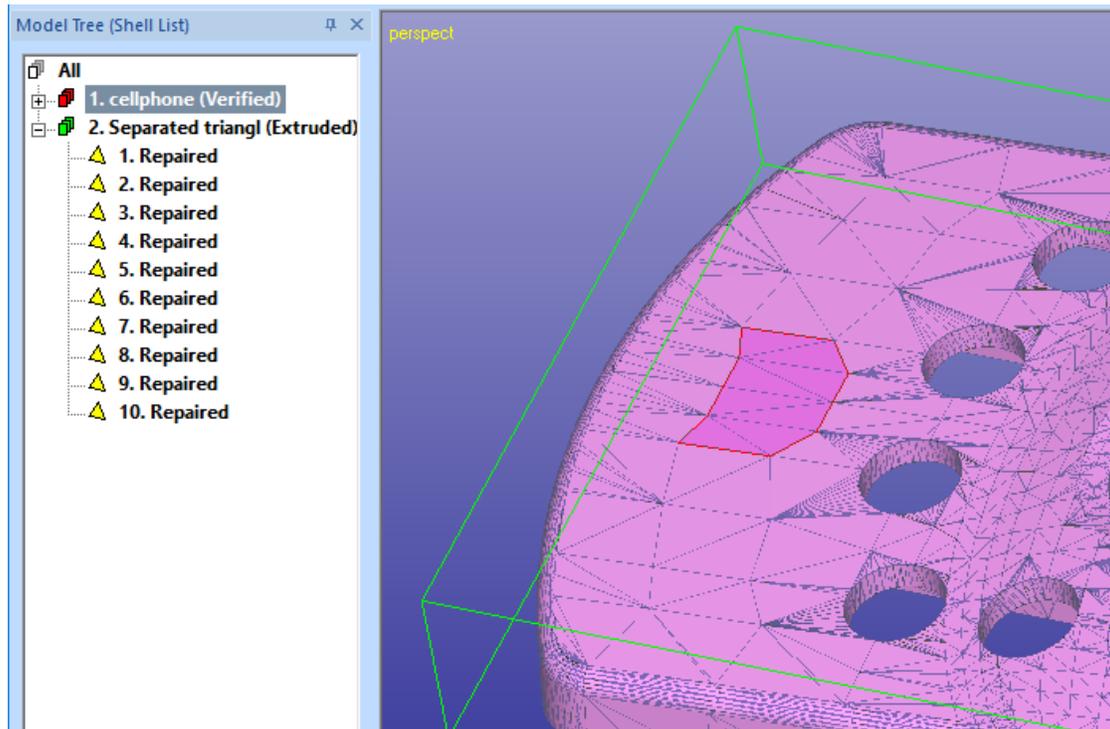
Copying Extruded surfaces to original shell

We will select all surfaces generated by Extrude and copy them into the original shell to make it solid again:

- **select all surfaces except the original surface** taken from the cellphone model

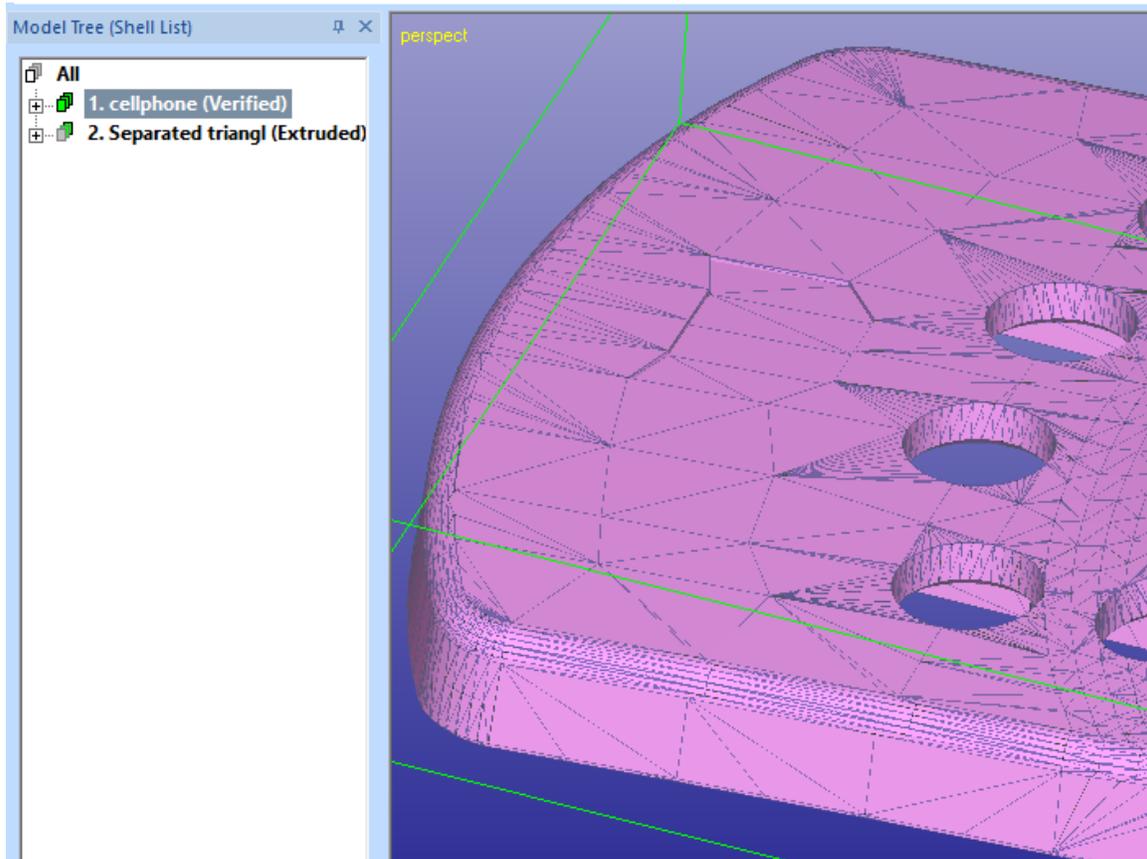


- **give Edit > Copy command**
- **select the 1. cellphone shell** from the Model Tree



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- give **Edit > Paste** command
- and you will have a correct *1. shell* in the model tree too
- **hide the 2. Separated triangl shell** to see the corresponding blind hole on the cellphone surface



You can now rename and output the shells into separate STL files as was done in the chapter *Renaming shells and outputting STL files* (see page 29).

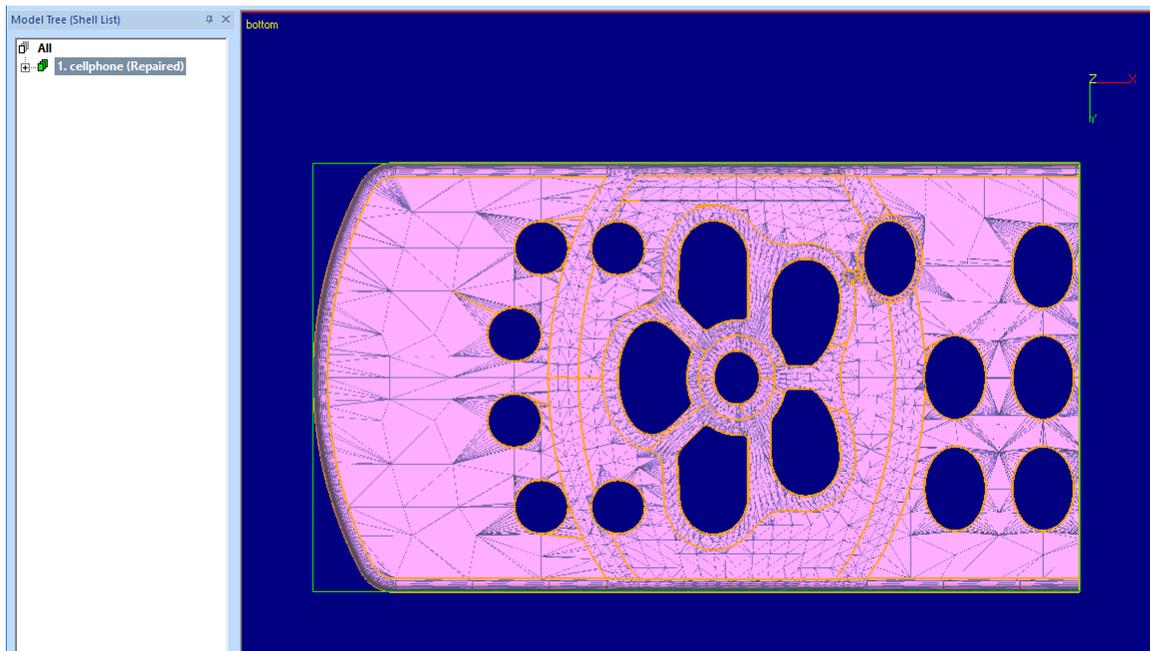
Chapter 8 – Using Offset to separate shells

In a similar way as Extrude command, the Print Preparation > Offset command can also be used to generate solid shells from separated surface areas. In this example we will first separate a surface area with Sketch Edge Line command and then separate and Offset it to solid shell.

Creating surfaces with Sketch Edge Line command

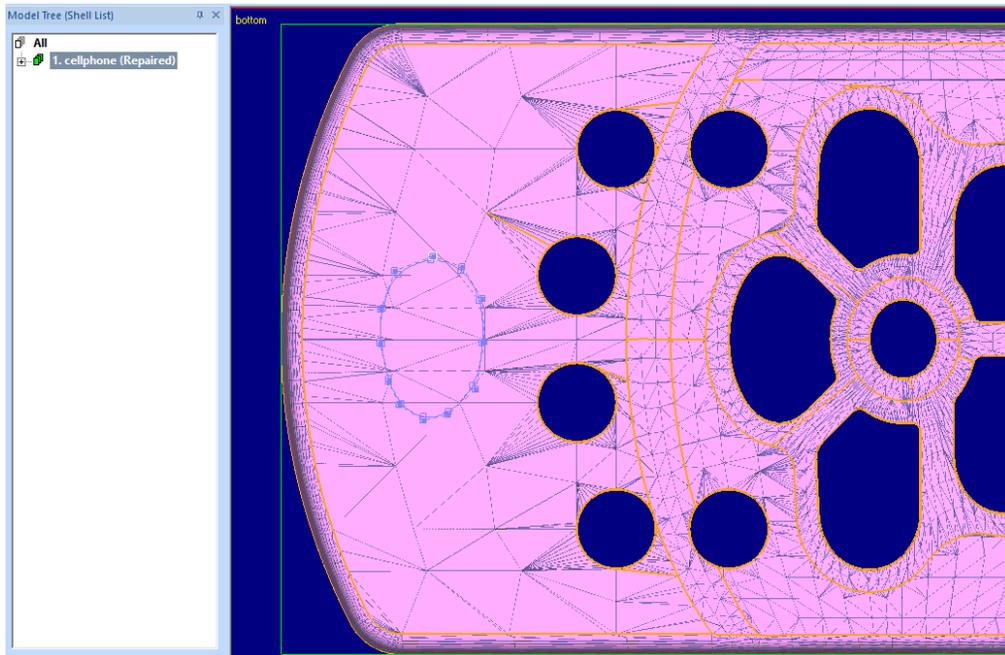
Let's start again with cellphone-repaired.3de model:

- **input the *cellphone-repaired.3de*** model, continue through the units and Operation Mode dialogs
- take a **view from -Z direction** (bottom)
- **select 1. cellphone (Repaired) shell** from the Model Tree
- **give Color and Texture > Edit Surfaces > Sketch Edge Line** command, the system goes into *curve input* mode and surface edges are displayed:



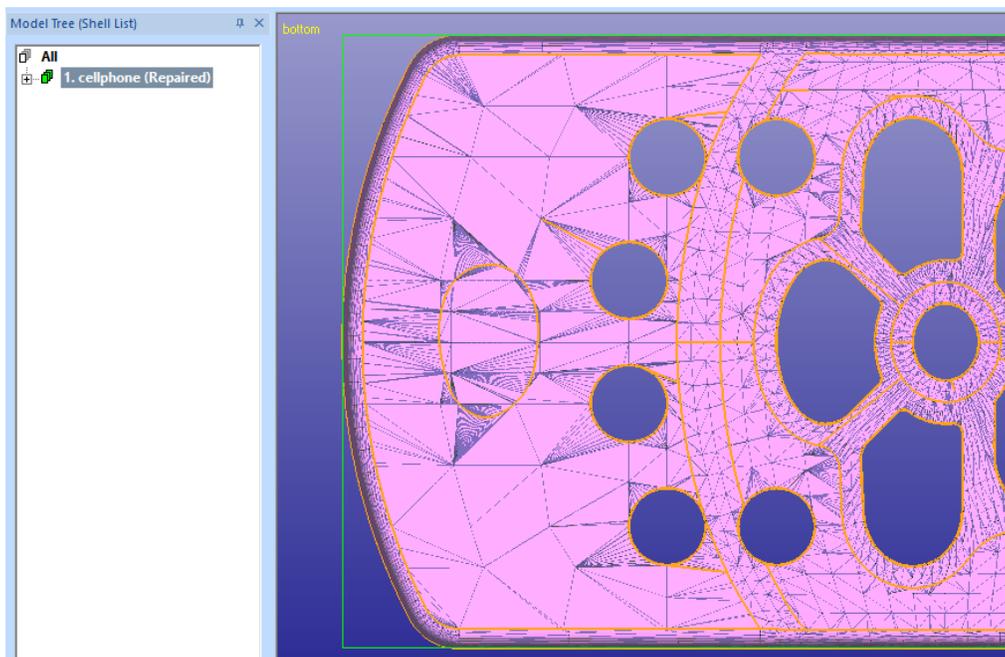
The system now waits for you to draw the new surface edge on the model:

- **add points on the model using the LMB**
- to quit adding points **select the first point again**, i.e. create a closed curve, as seen on the image below
- you **may select and reposition the points** until the curve shape is correct



Now our surface edge line is ready and we will generate the new surface on top of the cellphone model.

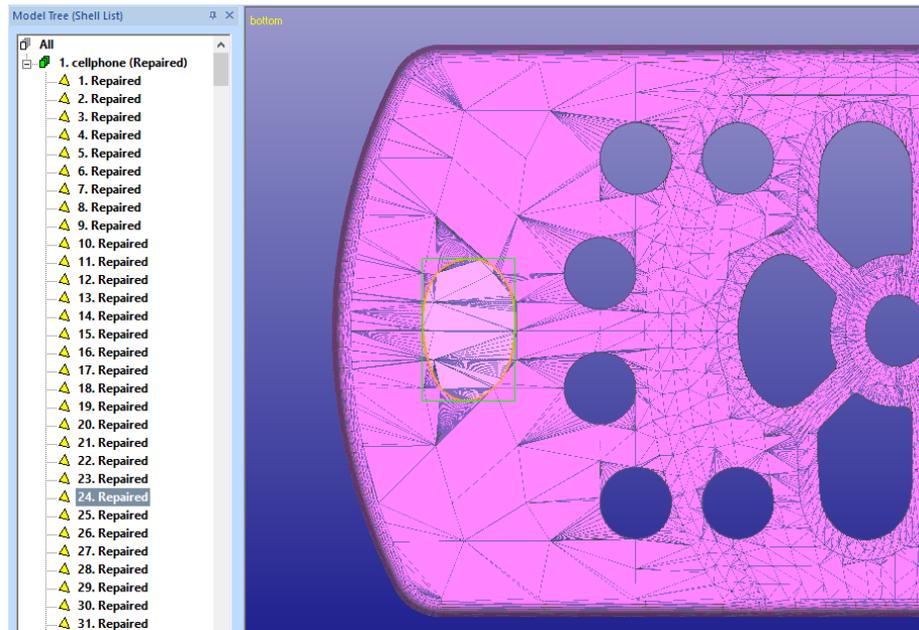
- **press RMB to add the new surface** on the model
- **run Fix Model > Repair Shells** to fix minor errors in the model (and to avoid seeing hidden surface edges)
- and you will see the new surface:



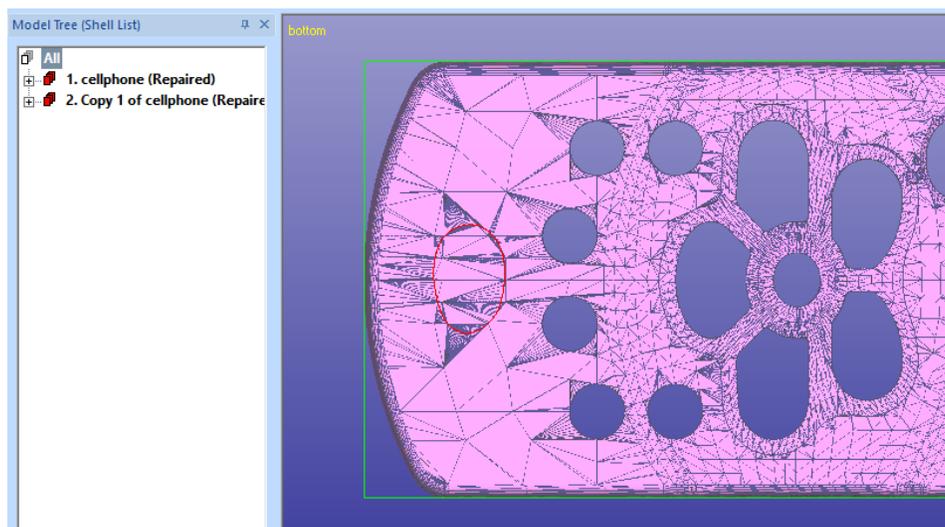
Separating the sketched surface

The new surface must be separated to a new shell before it can be Offset to solid model.

- **select the sketched surface** from the graphical area or from the Model Tree



- **give Edit > Cut** command
- **select Model Tree root All**
- **give Edit > Paste** command
- **turn off surface edge line display** via *Color & Texture > Edit Surfaces > Show Edge Lines* setting.



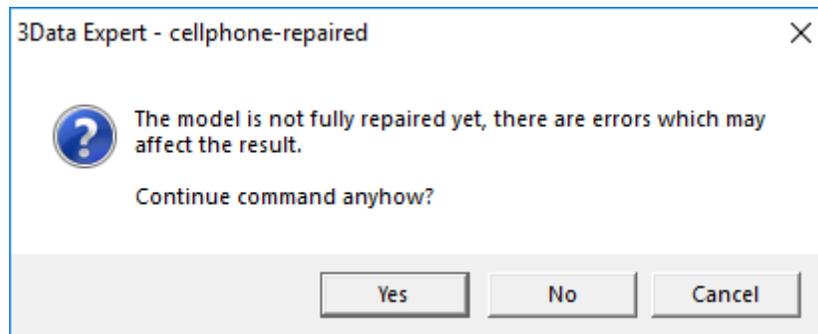
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The Model Tree now contains two shells, one with the original cellphone mode without the delete sketched surface and the sketched surface in shell 2. *Copy 1 of cellphone (Repaired) shell.*

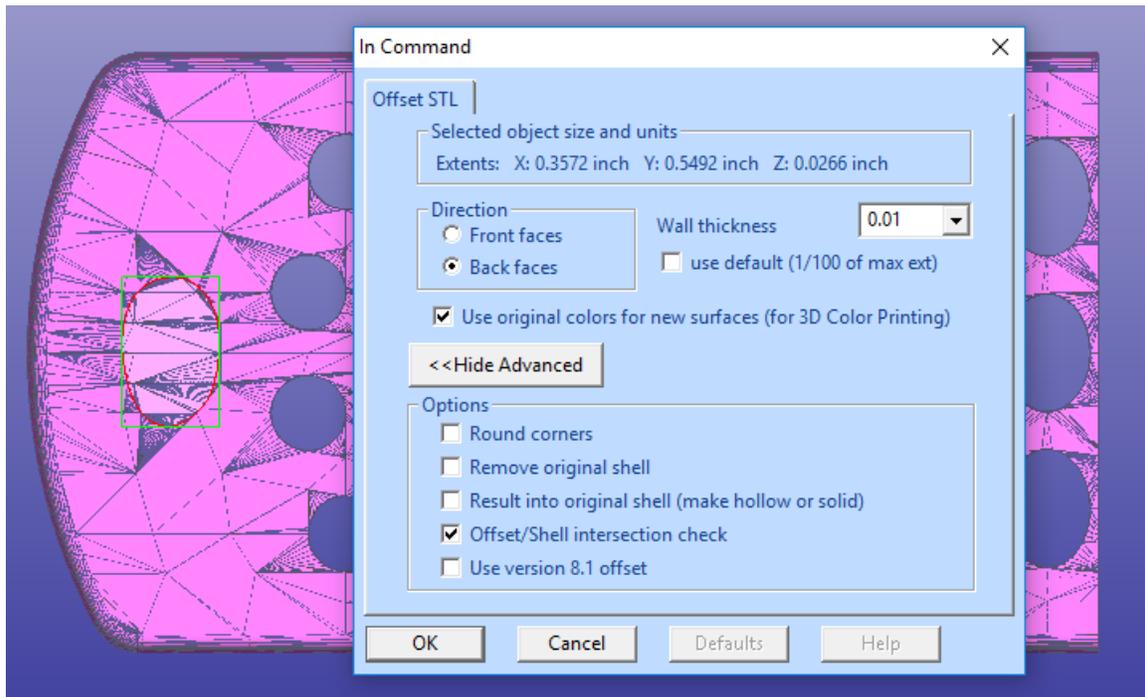
Using Offset command

We are now ready to use the Offset to generate the components for the new shells.

- **select 2. Copy 1 of cellphone shell** from the Model Tree
- **give Print Preparation > Offset command**
- if you get a warning about the model not being fully repaired **press Yes to continue**

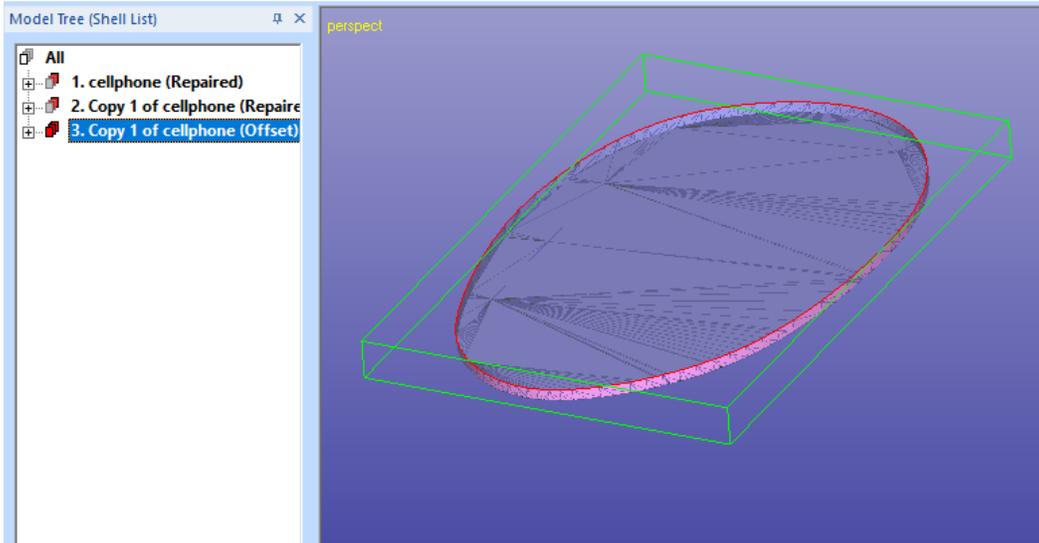


- **fill in the parameters** given in the image below, especially **Result into original shell** must **not** be set!



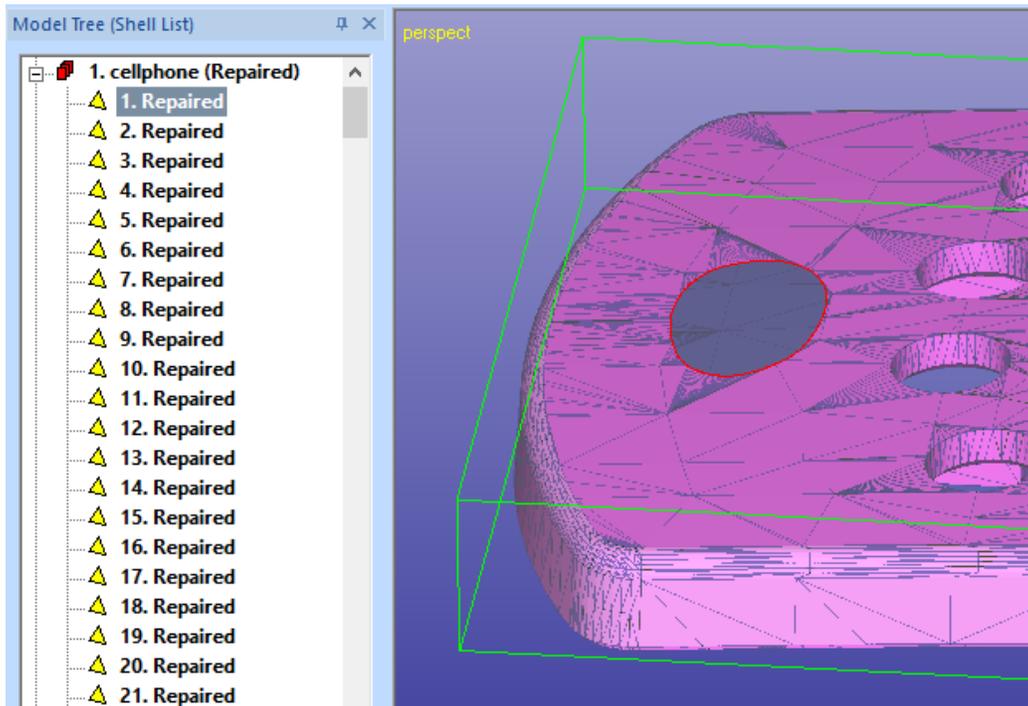
3Data Expert 12.1: Tutorial 3 - Separating Shells

- **press OK** to create the offset
- **fit the Offset result 3. Copy 1. of cellphone (Offset)** to see the result



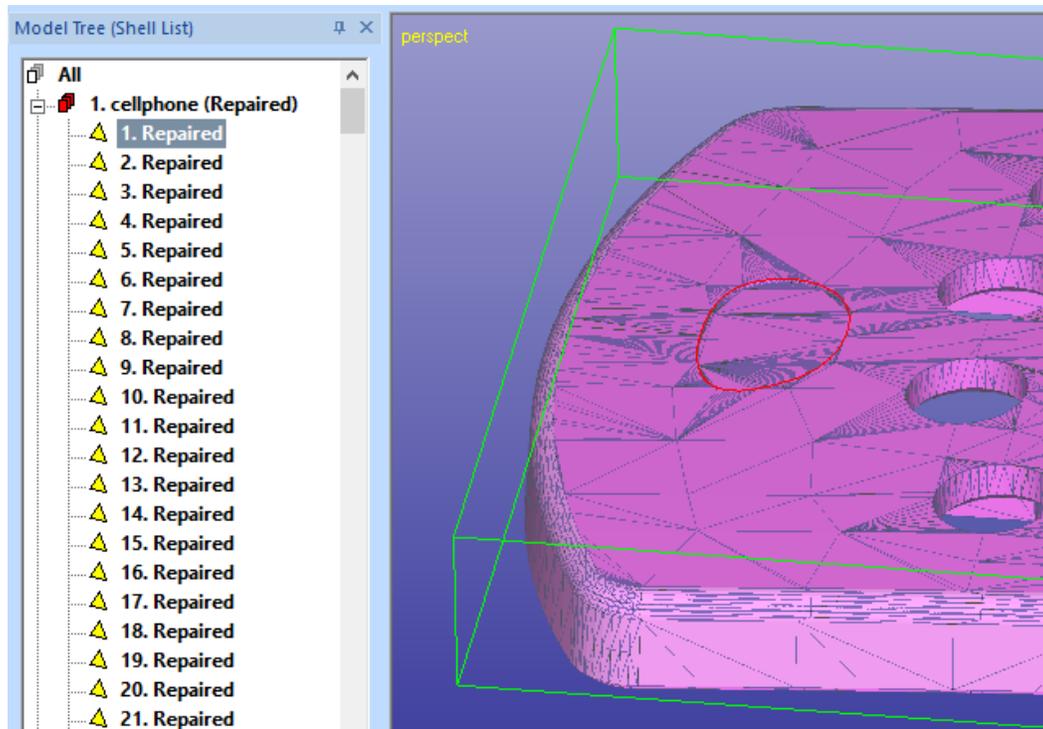
You now have all necessary parts in the Model Tree for creating the tow solid shells. Do the following:

- **select the third shell 3. Copy 1 of cellphone (Offset)** in the Model Tree (offset result)
- **give *Edit > Copy* command**
- **select the 1. Repaired surface** of the first shell *1. Cellphone (Repaired)* in the Model Tree (original model)

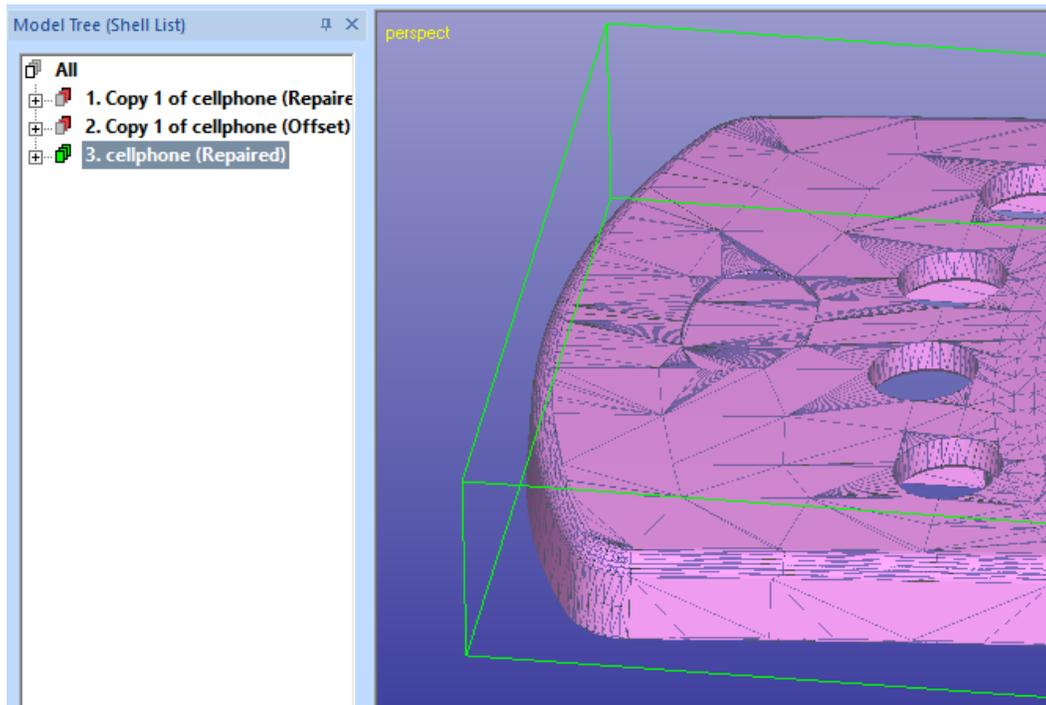


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- give **Edit > Paste** command to add the copied Offset result to the shell
- you will see a correct 1. shell to appear in the Model Tree. The original shell now has the blind hole added to it.



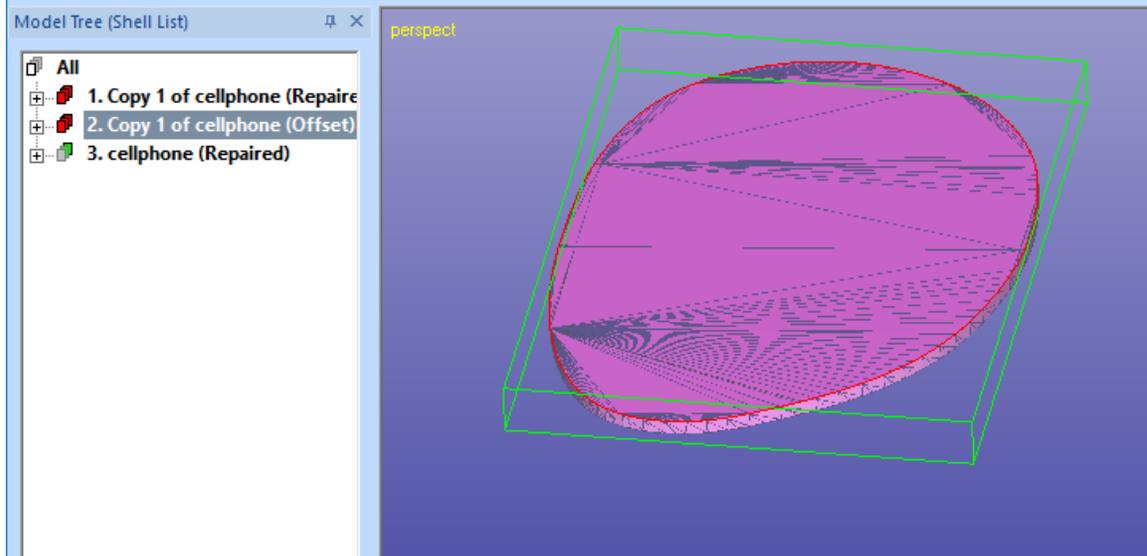
In case of errors in *1. cellphone (Repaired)*, please run *Fix Model > Repair Shells* command with default parameters on that, to get fully correct component.



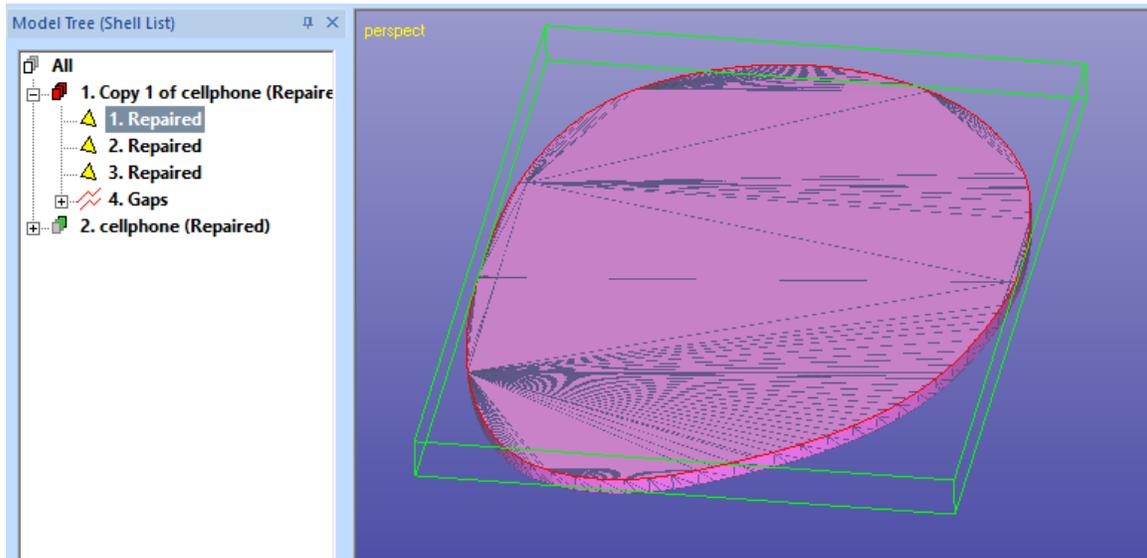
3Data Expert 12.1: Tutorial 3 - Separating Shells

Now we need to join the two remaining red shells in the Model Tree into one shell to get correct solid shell. For example:

- select the **Copy 1 of cellphone (Offset)** shell
- give **Edit > Cut** command



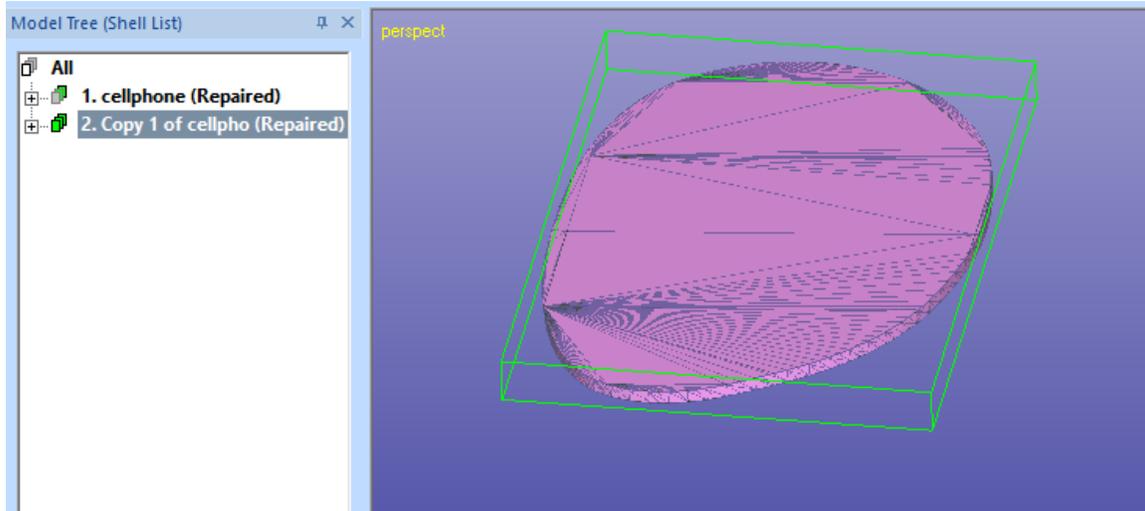
- select the first surface in the **Copy 1 of cellphone (Repaired)** shell
- give **Edit > Paste** command to paste the cut shell into the that shell



In case of errors in **1. Copy 1 of cellphone (Repaired)**, please run **Fix Model > Repair Shells** command with default parameters on that, to get fully correct component.

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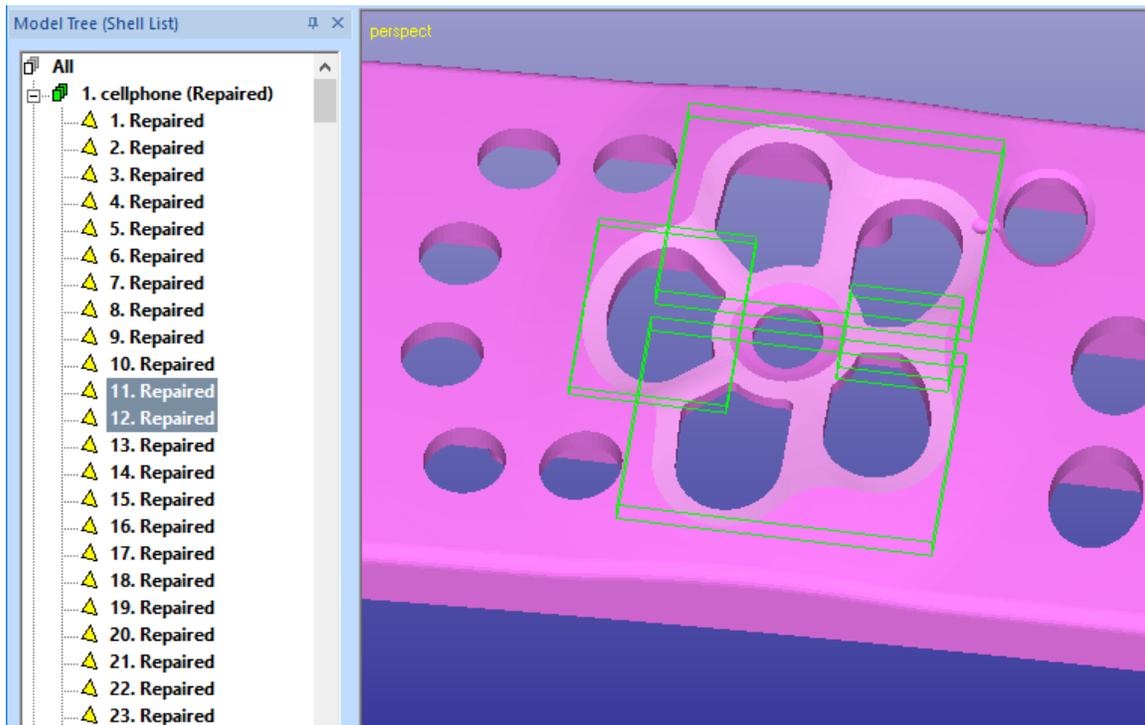
Now you will have the two solid shells in the Model Tree for output, please see chapter *Renaming shells and outputting STL files* at page 11.



Selecting multiple surfaces for Offset or Extrude

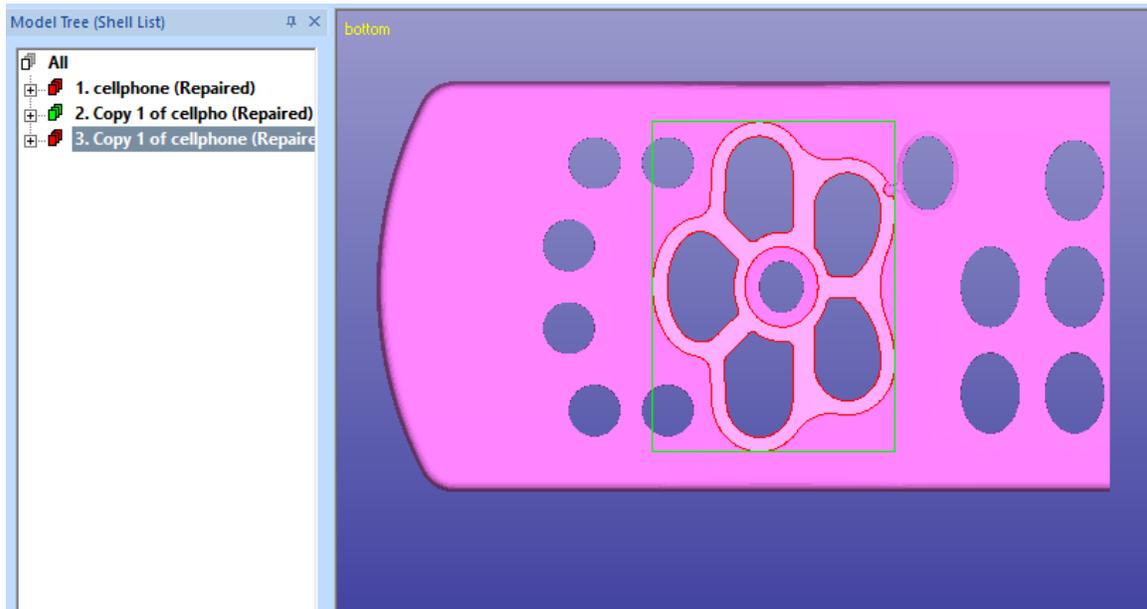
Of course, it is possible to use the existing surfaces for separation and subsequent Offset or Extrude. For example, you may want to try to separate the surfaces around the keypad for separate shell creation.

First select the surfaces show in the image below (Ctrl + LMB):

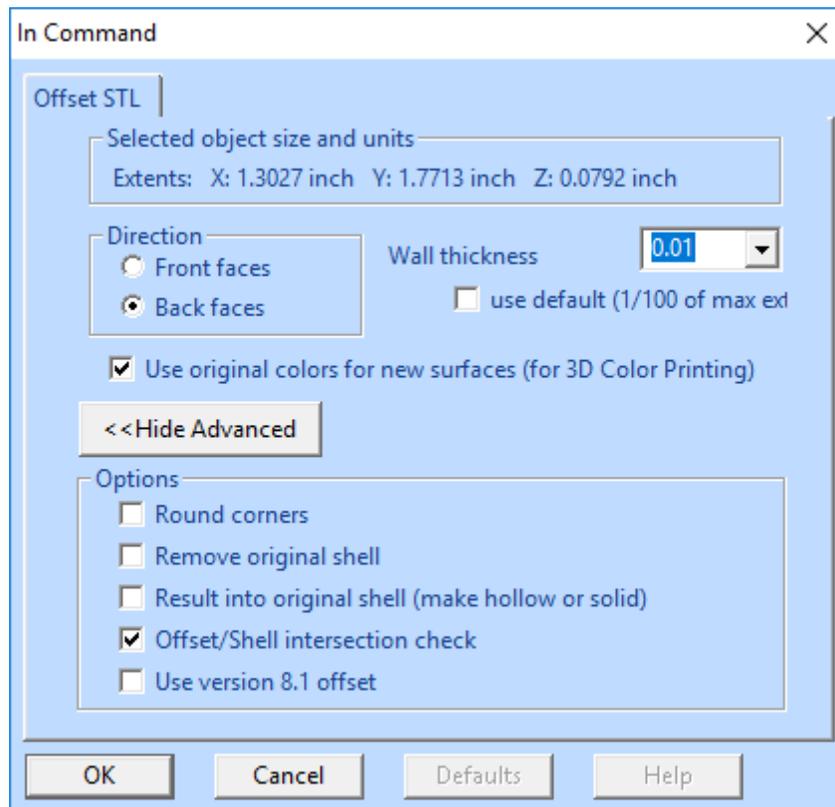


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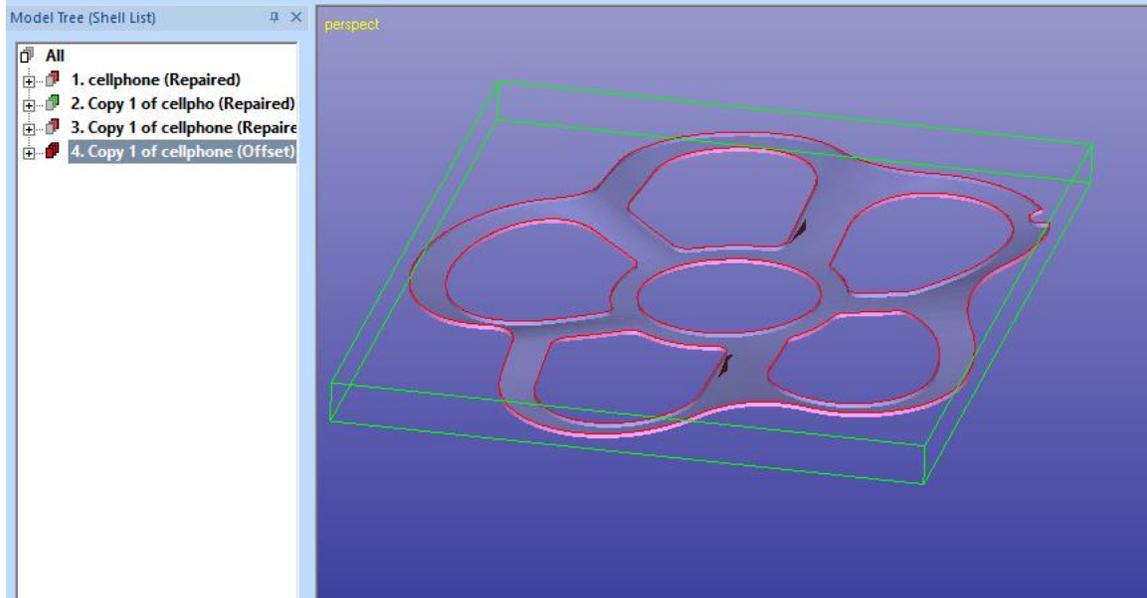
Then Cut and Paste them into Model Tree:



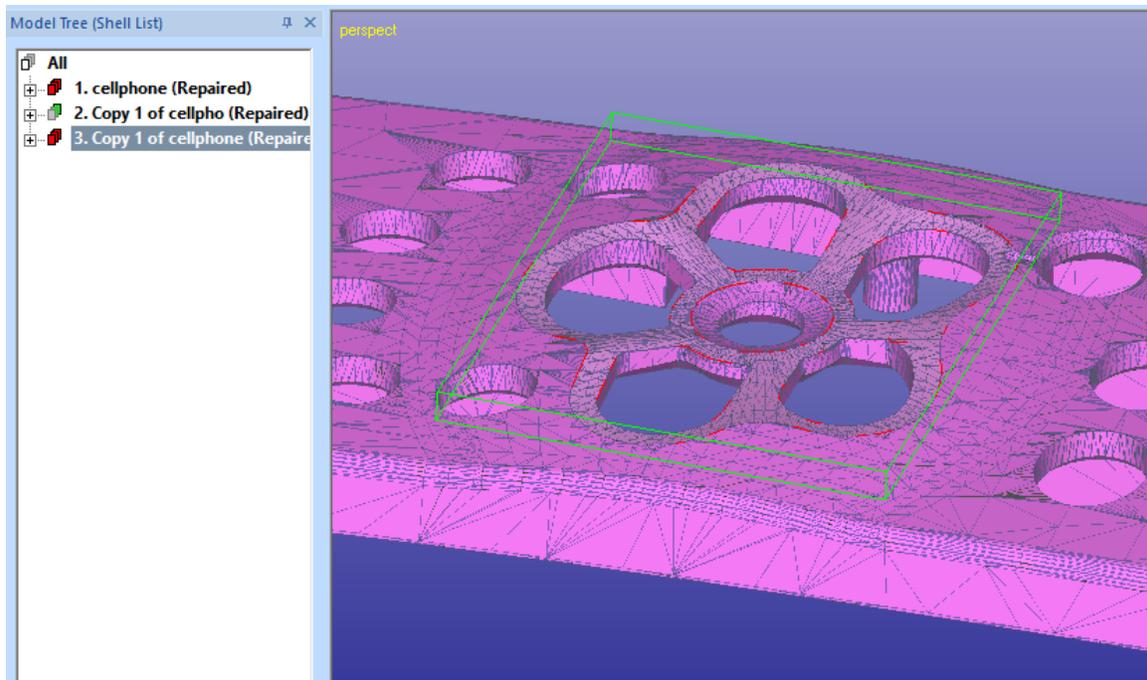
Now you can use Offset to inside direction again (Opposite direction, result not in the same shell as the offset shell):



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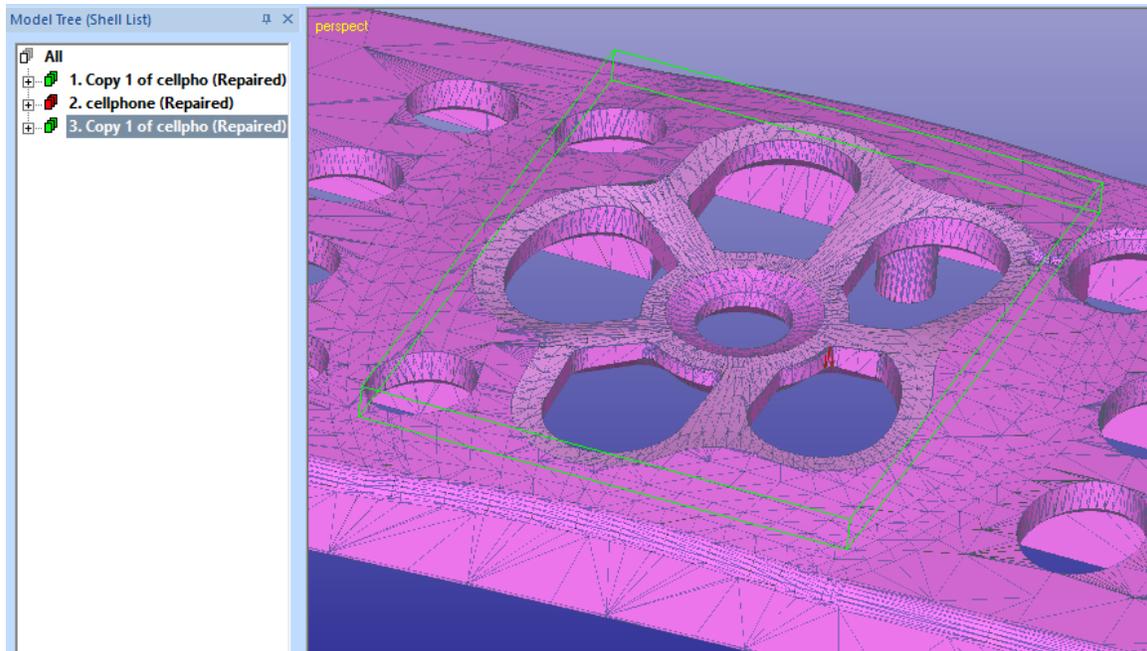


Then make the two shells solid again by Copying the Offset result to the original shell and then connect the separated shell and the offset shell into a solid shell:

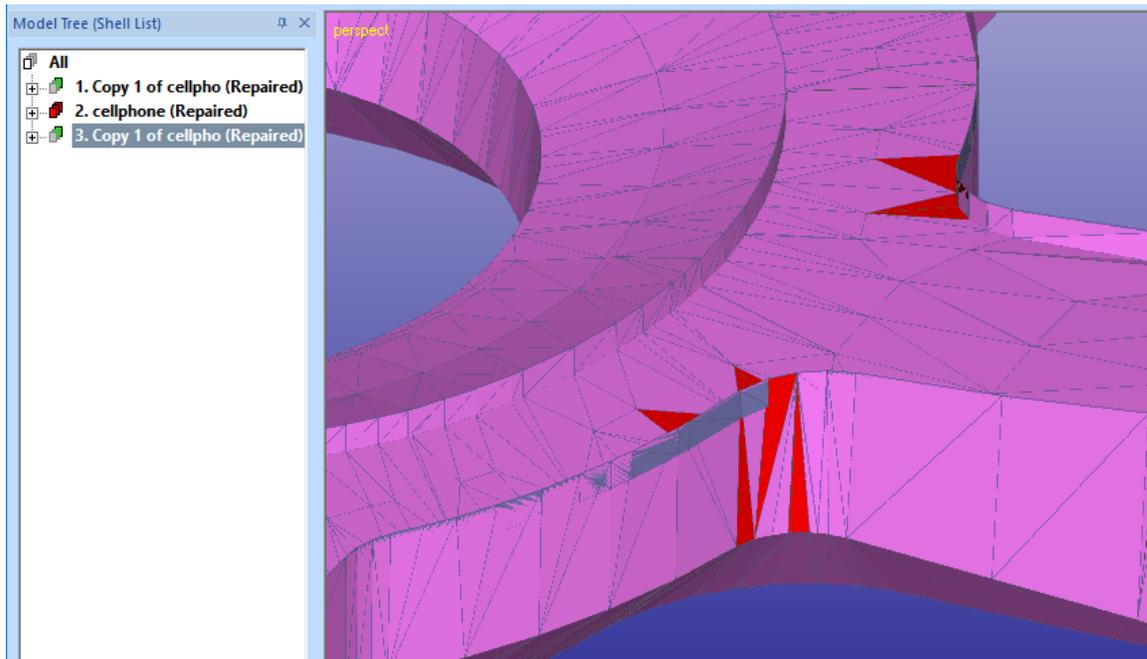


There are some errors in the generated shells, run them both through the Fix Model > Repair Shells command. The separated parts turn out to be correct but there are still some errors in the original cellphone model:

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The errors are caused by some zero volume walls and intersecting triangles in them. These are created when the Offset result is added to the original model. The Offset direction follows the vertical walls of the holes in the original model:



The seriousness of these errors with different 3D printing systems is not known at the time of the writing, but must be taken into consideration when creating separate shells at areas where surfaces meet perpendicularly at the extrude area edges. This result is found in *Tutorials\GeomFiles* folder (*cellphone-offset-separated.3de* file). These problems should be solved automatically when the implementation of Extrude to Shell and Offset to Shell commands is completed.

Conclusions

This *Separating Shells* document has given you the skills to separate shells from a solid STL model for systems with digital material properties or capable to produce parts with multiple colors.