

DeskArtes

3Data Expert 14.1

Tutorial 1 - Working with 3D Models

First Edition

Ismo Mäkelä: “3Data Expert 14.1: Tutorial 1 - Working with 3D Models”.

February 2023

Copyright © 2023 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.

Trademarks

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

Takomotie 8

FIN-00380 Helsinki

Finland

<https://www.deskartes.com>

Table of Contents

FOREWORD.....	4
<i>Modules.....</i>	5
LESSON 1 – BASIC USER INTERFACE PROPERTIES	7
<i>The Free module Free.....</i>	9
<i>Sandbox mode exceptions</i>	9
<i>File input.....</i>	9
<i>Model Tree.....</i>	14
<i>Basic viewing commands.....</i>	15
<i>Selecting objects.....</i>	16
<i>Selecting shells, surfaces and triangles graphically</i>	17
<i>Display Area menu.....</i>	19
<i>Object properties</i>	19
<i>Dimensions and Annotations</i>	20
<i>Repairing the slubox model.....</i>	23
LESSON 2 – FULLY AUTOMATIC STL FIXING	24
<i>Auto Repair process for STL files</i>	24
LESSON 3 – STL FIXING.....	27
<i>Verifying and repairing the STL model.....</i>	27
<i>Verifying and repairing, take two</i>	28
<i>Using Repair Shells command.....</i>	29
LESSON 4 - SPLITTING AND COMBINING STL	33
<i>Splitting the dino model</i>	33
<i>Creating the pins.....</i>	36
<i>Saving the result.....</i>	39
<i>Using Lap Joint to connect thin parts.....</i>	40
<i>Adding Lap Joint.....</i>	42
LESSON 5 – POSITIONING PARTS.....	46
<i>Print Configuration.....</i>	46
<i>Moving parts into the platform</i>	49
LESSON 6 – NESTING	52
<i>3D Nesting regular components</i>	52
<i>3D Nesting irregular components.....</i>	55
LESSON 7 – ADDING 3D TEXT	60
<i>Creating 3D Text.....</i>	60
<i>Adding 3D Text to multiple parts.....</i>	62
<i>Adding 3D Text on curved surfaces</i>	65
LESSON 8 – GENERATING AM REPORTING	68
<i>Production Screenshot command.....</i>	68
<i>Generate Report command</i>	69
LESSON 9 – PREPARING FOR 3D PRINTING	71
<i>Making Hollow models</i>	71
<i>Hollowing with Scan Offset.....</i>	73
<i>Creating a 3D Lattice</i>	76
<i>Making Drain Holes</i>	79
<i>Making tapered plugs with Hole Tool.....</i>	81
<i>Adding thickness by Offset</i>	85
<i>Combining shells using Boolean commands.....</i>	88
<i>Generating advanced cuts with Cut Tool.....</i>	90
LESSON 10 – PREPARING FOR SIMULATION APPLICATIONS	93
<i>Starting the reduction</i>	93
<i>Reducing the rava2 model.....</i>	95
<i>Which reduction method to use?</i>	100
<i>Improving the triangle aspect ratio.....</i>	101
CONCLUSIONS	104

Foreword

The *3Data Expert 14.1: Tutorial 1 - Working with 3D Models* document introduces the basic 3D model handling with 3Data Expert 14.1: 3D model viewing, measuring, repair, splitting, connecting, adding lap joint, hollowing, drain hole generation, Boolean operations, support generation, surface triangulation, data reduction and separating shells for different material or color definitions in one part.

DeskArtes 3Data Expert 14.1 is available as a 64-bit implementation, allowing you to handle huge models for Additive Manufacturing.

The full tutorial set for 3Data Expert 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:

- *3Data Expert 14.1: Quick Start Guide* gives a short introduction to model repair and splitting for 3D Printing with 3Data. Estimated completion time 15 minutes.
- *3Data Expert 14.1: 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. Estimated completion time is 3 hours.
- *3Data Expert 14.1: Tutorial 2 – Working with Colors* introduces basic automatic repair and coloring for 3D Color Printer users. Estimated completion time is 1 hour.
- *3Data Expert 14.1: 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. This tutorial is meant only for users who have a *Base* module license. Estimated completion time is 30 minutes.
- *3Data Expert 14.1: 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.
- *3Data Expert 14.1: STL and VRML repair examples* document includes more advanced model repair operations with 3Data Expert. 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 installation directory.

Additional information and help on other STL manipulation tools can be found in the Handling Faceted Files section of the On-line Help pages. These pages contain information on verifying, repairing, splitting, offsetting, hollowing and draft checking for tooling applications.

The operations given in **boldface** in this document are commands which affect the geometry of the part in question or are otherwise important and must be run by the user in the given order to get the results show in this document. Several other commands, like viewing and clipping, are not given in boldface but should be used to enable generation of a correct result. Terms given in *italics* refer to file names, commands, shell names, surface names and parameters for different commands.

The pictures in this document can differ from the version you get. Most of the pictures have been grabbed with the *classic* theme, while the *dark* theme has been used for some of them. The theme can be changed from *Edit > Preferences*.

Modules

The lessons in this tutorial are organized in such a way that that the required software functions increase along the lessons.

DeskArtes 3Data Expert software suite consists of several products targeted from home and DIY use to education and professional Additive Manufacturing:

- *3Data Expert Free* is an entry level tool targeted for viewing, verifying, repairing, dimensioning, and communicating 3D geometry information. It allows you to input triangle models, create consistent topology, check errors, orient normal vectors, repair gaps and self-intersections, edit triangles and output STL data for 3D printing, without any cost. In the tutorials, when information relevant to the *Free* module is handled, it is marked with **Free**.
- *AM Expert* module (an option to *Free* and *Base*) provides automatic AM process selection based on part requirements (material, size, accuracy, tensile strength, and post processing). The *AM Expert* database currently contains data for 129 professional AM systems, and it is regularly updated.
- *3Data Expert Dimensions* is a simple, economical, and user-friendly tool for STL repair and 3D model splitting for machines with a smaller build area. The tools include Repair, Splitting and Connect with Pins or Lap Joint. Information relevant to *Dimensions* will be marked with **Dime**.
- *3Data Expert Lite* includes all standard 3D model preparation tools for 3D Printing and Additive Manufacturing. Available tools include geometry generation, Boolean and 3D markup commands, Offset, Hollowing, Coloring and Texturing, Solidify (aka “shrink wrap”), as well as Slicing and Tool compensation. Optional 3D Nesting makes *3Data Expert Lite* a perfect 3D data preparation tool for powder-based AM systems. Information related to *Lite* will be marked with **Lite**.

- *3Data Expert Base* is a professional tool for preparing 3D models for Additive Manufacturing and Simulation applications. There are several commands for manipulation of both 3D surface models and 3D faceted models. These commands include Surface triangulation, STL verification and repair, STL Offsetting, Splitting, Connecting, Decimation, Smoothing, Solidify, Boolean, Part subdivision, 3D Nesting, Z Compensation, Slicing and Tool Compensation as well as STL Coloring and Texturing. Supports can be generated for DLP, metal and stereolithography processes. Special Sand Support command creates supporting plates for delicate sand parts with a push of a button. Information relevant to Base is marked with **Base**. However, most of the tutorial contents are applicable with *Base*.
- *3Data Expert Color* (**Color**) contains a subset of *3Data Expert Base*, especially meant for handling color models. A variety of color and texture specific input formats are handled: VRML, ZPR, 3DS, OBJ, PLY. *3Data Expert Color* has previously been labeled as *DeskArtes 3DEdit Pro*.

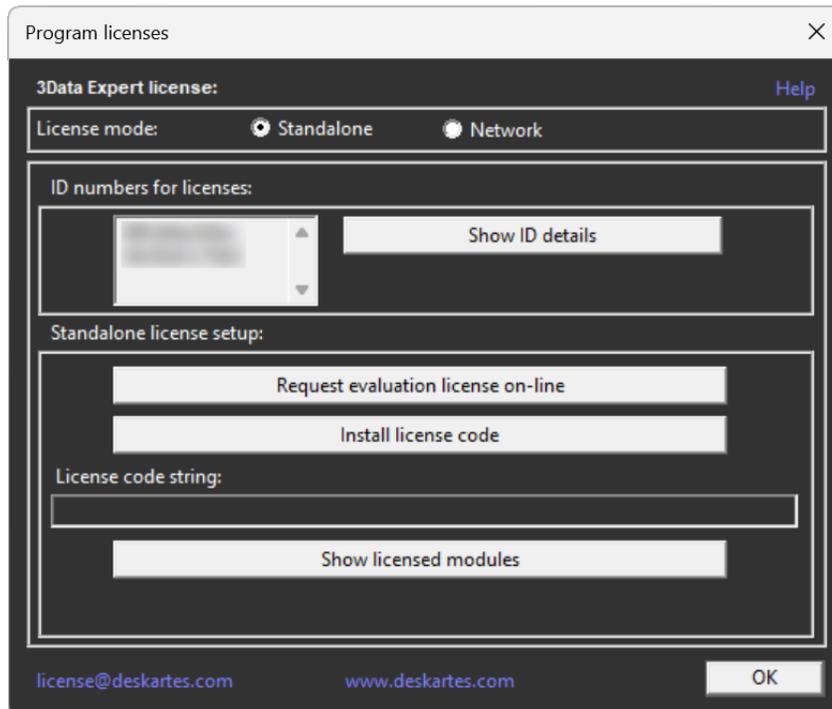
Lesson 1 – Basic User Interface properties

This lesson shows you how to do basic file input, viewing, dimensioning and verification of surface data with 3Data Expert.

You will need a license to go through the lessons except the ones marked with **Free**. Before starting the lessons, please make sure you have received and installed the software evaluation license from DeskArtes. If you do not have a license yet, it is very simple to request a license. First start the **DeskArtes 3Data Expert** (or Dimensions Expert, etc.) by double-clicking on the icon created on the desktop during the installation.



Then go to the *Program licenses* dialog through *Help > Licenses* command. There you can find the *Request evaluation license on-line* button.



Press it and it will open the DeskArtes license request page on your favorite web browser. Fill in the required fields and press *Submit evaluation request* button to send the license request to DeskArtes.

3Data Expert 14.1: Tutorial 1 - Working with 3D Models



Evaluation license request for: 3Data Expert 14.1

Please fill your email address:

Your name:	<input type="text"/>	
Your email address:	<input type="text"/>	
Verify email address:	<input type="text"/>	
Select optional input formats:	<input type="checkbox"/> Catia 4	
	<input type="checkbox"/> Catia 5	
	<input type="checkbox"/> Catia 6	
	<input type="checkbox"/> STEP	
	<input type="checkbox"/> NX Unigraphics	
	<input type="checkbox"/> SolidWorks	
	<input type="checkbox"/> Solid Edge	
	<input type="checkbox"/> Inventor	
	<input type="checkbox"/> Creo/ProE	
	<input type="checkbox"/> JT	
	<input type="checkbox"/> Catia CGR	
	<input type="checkbox"/> Parasolid	
	<input type="checkbox"/> IGES	
System ID list	Value	Description
		(REALTEK USB GBE FAMILY CONTROLLER #2)
		(INTEL(R) WI-FI 6 AX200 160MHZ)
Comments:	<input type="text"/>	
<input type="button" value="Submit evaluation request"/>		

You will receive the 15-day evaluation license within 24 hours.

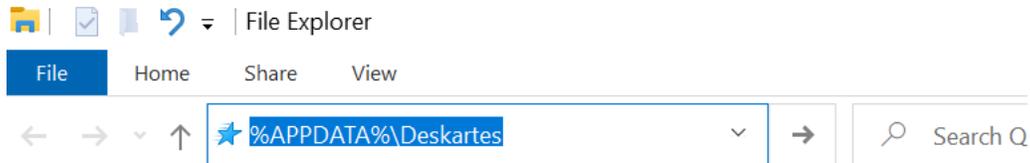
When the license arrives, press the *Install license code button* and *Copy/Paste the license string* into the input field:

The dialog box titled "Expert Series license" contains a text input field with the prompt "Copy/Paste or type license code string here:". Below this is a section for "License file save location" with three radio button options: "Machine ProgramData folder (common for all users)", "User AppData folder (only for the current user)", and "User HOME folder (only for the current user)". At the bottom right are "OK" and "Cancel" buttons.

Then press *OK* to accept the license. Finally, *restart* the software. Now you are ready to go ahead with the lessons!

If you wish to install network licenses, please have a look at the On-line Help (*Help > Help Topics* command) page *Installation / Licenses* section.

If you have used 3Data Expert 14.1 earlier in your computer you may want to delete the settings file (settings3DE141.das) in your *AppData* folder: open Windows Explorer, type "%APPDATA%\DeskArtes" into the address field, locate the settings files and delete it, finally restart the program. This initializes the system into its basic state and ensures that your results will not differ from the lesson results. Please see the On-line Help *Installation / Settings* page for the location of the settings file.



The Free module Free

Some of the functionality of 3Data Expert does not require a license: you can input several faceted file formats (STL, PLY, ZPR, VRML, ...), create topology for the model, check errors, repair 3D models automatically and manually, as well as output the checked data for 3D Printing (STL, VRML, PLY, ZPR). For more serious repair work and model manipulation, like split, connect, hollow, drain hole, support structure generation etc., a suitable license is required.

Sandbox mode exceptions

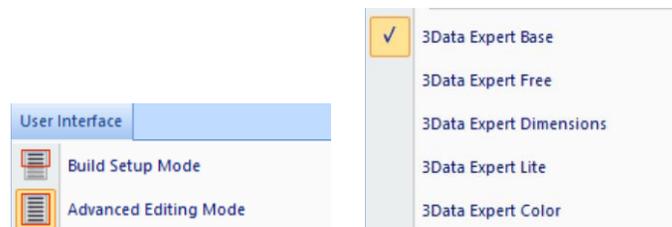
Certain 3Data Expert functions, like *Drain Hole* and *Hatch Supports*, are created using a separate background process started from 3Data Expert. In this case some antivirus software products may try to restrict the background process executable into "Sandbox" mode. This will cause the command to fail. To fix the problem set an exception for the background process executable. This is normally done through the antivirus software interface where you can change the run mode from "Sandbox" to "Normal".

File input

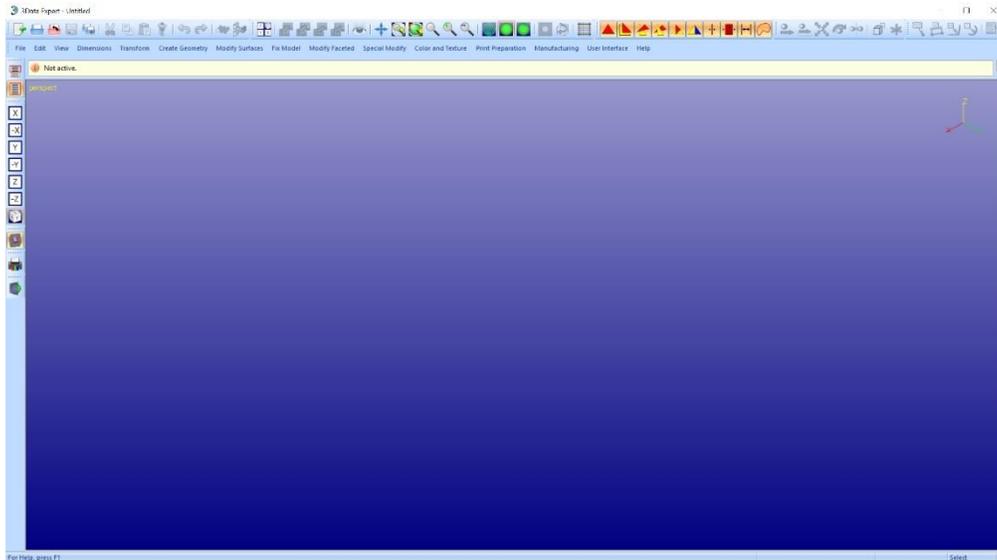
You can start the **3Data Expert** by clicking on the icon created on the desktop during the installation. If no desktop icon is created, use the Windows Start menu to search and launch the software.



After starting the system and closing the Message dialogs you will see the basic configuration of the 3Data Expert on your screen: menus, different toolbars, and the graphics area. Initially, the menus of the *Base* module are shown. You can, however, change between menu setups if you wish to see less commands (like hiding those you don't have a license to run); also, in 3Data Expert Base, you can switch between *Build Setup* and *Advanced Editing* modes:



3Data Expert 14.1: Tutorial 1 - Working with 3D Models

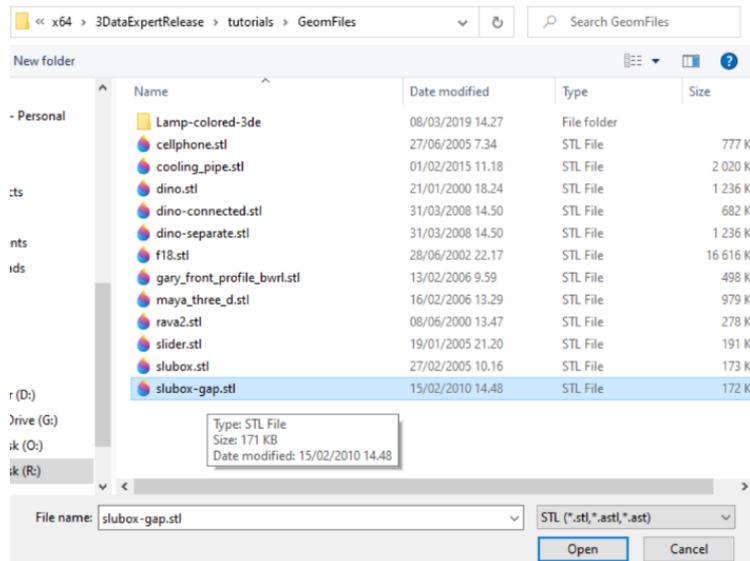


File input is normally done *File > Open* command (which deletes any existing models from program memory), or you can drag a geometry file into the graphics area (which actually performs the *File > Insert* command).¹

- **Start the File > Open** and **select the Tutorials / GeomFiles** folder under the software installation directory.

The file type is chosen from the *Files of type* dropdown list.

- **Select type STL files (*.stl)** and you will see the list of STL files in the *GeomFiles* directory.
- **Select *slubox-gap.stl*** from the file list and press the **Open** button.



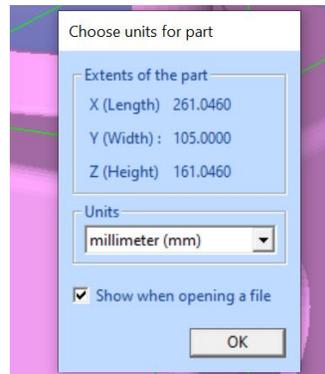
After a while you should see the *slubox* geometry on your screen.

¹ For both file input and output, there are options which can be controlled via *File > File Options*. These will be discussed later.

3Data Expert 14.1: Tutorial 1 - Working with 3D Models

When a model is inputted 3Data Expert allows to check and set the units to the part. For example, STL files do not have any unit information in them.

- **Press OK** to accept the proposed units, *millimeters (mm)*.



You will next be prompted for the *Operation Mode*². 3Data Expert has six main operation modes: *Fix*, *Modify*, *Position*, *Print*, *Color* and *View* mode. You may have a different set of Modes available depending on which module you are running.



The different modes are used for different tasks:

- *Fix* takes you to automatic model verification and repair and contains shortcuts to the main tools used for repair work.
- *Modify* lets you change the model geometry with features which are necessary or useful for production.
- *Position* contains tools which help you optimize the positioning of components on the printing platform.
- *Print* shows tools for preparing a repaired and positioned model for your AM printer, such as creating support structures and slicing.
- *Color* displays the tools for painting and texture image addition. This mode also checks the model for errors when selected and sets the display into “what you see is what you get” display color mode.
- *View Mode* allows you to temporarily view the model and then return to Operation Mode selection later (or to close the minimized window).

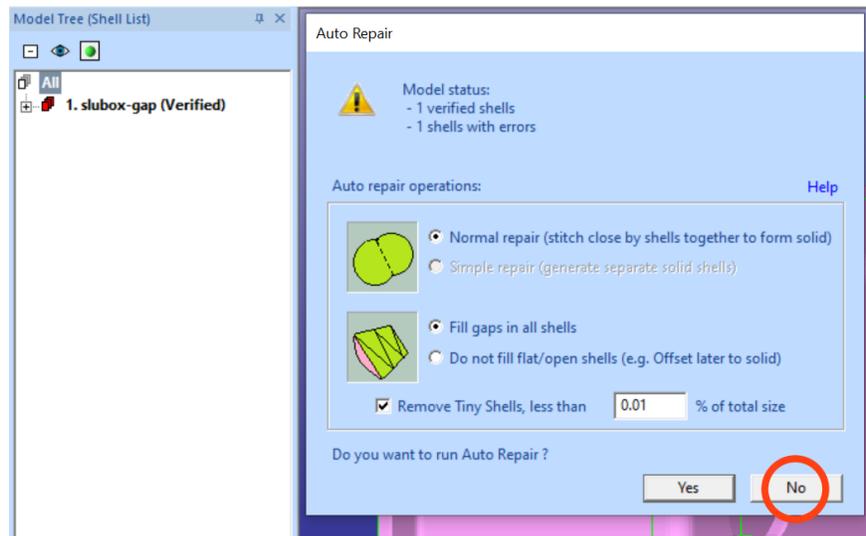
You can also simply close the Operation Mode via the top-right close button.

² In *Edit > Preferences*, you can turn off automatic features, including the Operation Mode display.

The normal way to begin repairing a model is to start with the Fix mode. To start fixing

- **press the Fix button** (the leftmost one) in the Operation Mode dialog.

3Data Expert will begin automatic model verification (*Fix Model > Verify Shells* command). The screen is erased, and a progress bar appears. When the analysis is ready, *Auto Repair* dialog with *Model status* message is displayed:



You will also see the model structure in the *Model Tree* on the left-hand side of the graphics area (see the image below). The model is divided into separate shells during verification. In this case we only get one shell (*1. slubox-gap (Verified)*). The shell has errors (i.e. gaps), which is indicated by the red icon  in the Model Tree; correct shells would have green icons with  or without  a warning sign (depending on defined warning settings), non-verified shells with yellow icons .

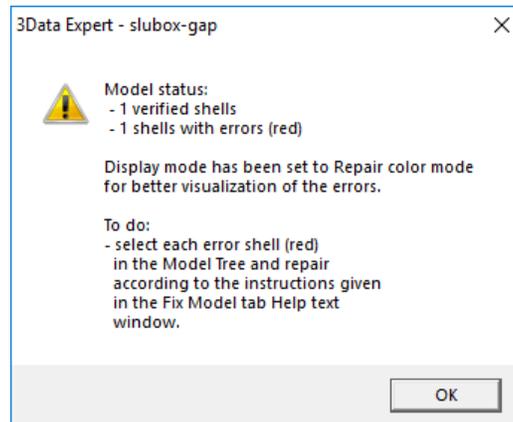
The *Model status* message indicates that there is one verified shell, which contains errors. The display shows red *Gap curves* through the surfaces. You can use the viewing commands when the Auto Repair dialog is visible, zoom (*mouse wheel*) and pan (*Ctrl + left mouse button*) if you cannot see the gap.

On the right-hand side of the graphics area, you will see the *Tools Window*. *Fix Tab* on the *Tools Window* provides shortcuts to the commands available for repair, like Repair Shells and Fill Gaps commands. The *Help text* window at the lower part of the Tools Window also contains information useful for the required repair actions.

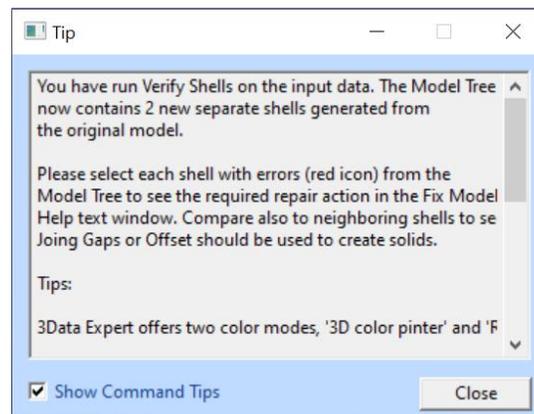
Auto Repair is normally the best path to follow when repairing 3D models. The different parameters for Auto Repair are set automatically, you should normally accept them as they are. We will later see examples where the automatically set parameter values are changed. Anyhow, this time we are not going for immediate full repair (this is done in the Quick Start Guide), instead we will investigate the Model Tree, the Tools Window, and the selection mechanisms available in the software.

- **Press No** (in the red circle) to start viewing the model.

A status window appears:



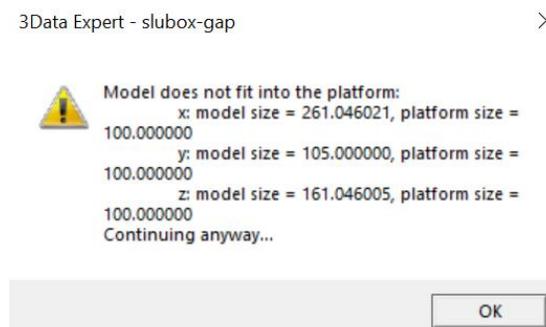
- Press **OK** to continue to see the *Tip Window* with useful tips on viewing and selection mechanisms.



- After reading the tip you can **Close** the window. You may also prevent it from displaying in the future by ticking off the *User Interface > Show Command Tips* setting.

Workspace and Print Configuration

When you open a file in 3Data Expert, a workspace, as defined in the current Print configuration is automatically added to the model and shown in the graphics window. When opening the slubox-gap.stl, you get a warning:

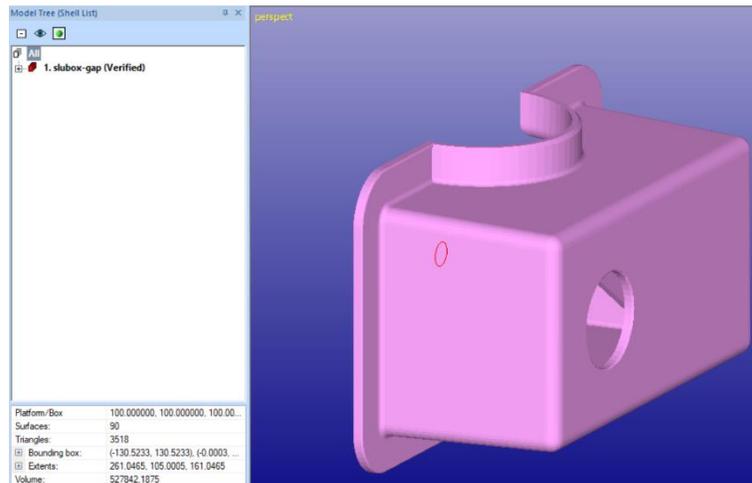


3Data Expert 14.1: Tutorial 1 - Working with 3D Models

This happens because the slubox-gap model does not fit in the workspace in the x- and y-directions. **Press OK** to see the model in the workspace.

The workspace is not needed for the lessons in this tutorial, so you can hide it using the *View > Show Platform* command (or the  icon). You can also hide the bounding box with the  icon and change to shaded view with  icon.

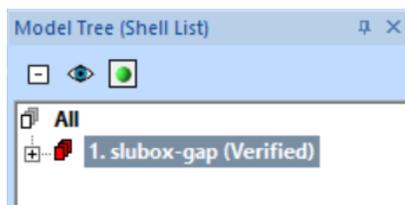
After closing the Tip Window, you will see the model in the graphics area:



The model icon is red to indicate that it has errors (gaps). The different colors of the object (pink and blue) on the graphics area are caused by the gap (drawn as a red curve in the model). Through the gap we can see inside the part. In the default *Repair color* mode, the outside (normal side) of the surfaces is displayed with *pink* color and the inside (inverted normal side) is displayed with *blue* color.

Model Tree

On the left-hand side of the 3Data Expert window you can see the Model Tree.



In the current contents you can see one shell, *1. slubox-gap (Verified)*. The shell icon is red indicating errors in the model. The *(Verified)* text indicates the last command run on the model. Above the tree you can see a line of context-dependent buttons. Three buttons are always visible: collapse tree, view/hide object, and fit object.

Name	1. slubox-gap (Verified)
Surfaces:	90
Triangles:	3518
Bounding b...	(-130.5233, 130.5233), (...
Extents:	261.0465, 105.0005, 16...
Volume:	527842.1875
Area:	214773.2188
File name:	slubox-gap.stl
Directory name	R:\DA1107\projects\Ex...

Below the Model Tree you can see information about the selected part, for example the number of triangles, part extents, volume, as well as the original file name and path. We will later take a more careful look at the contents of the Model Tree.



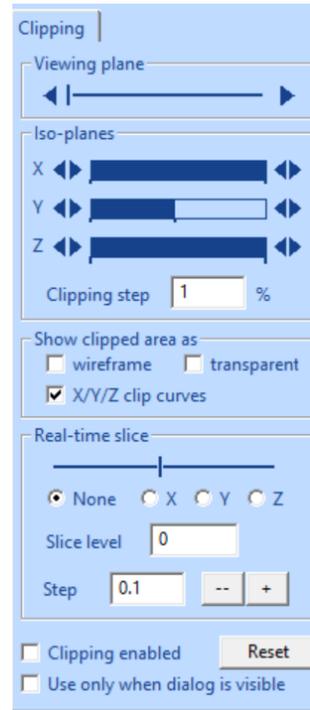
You can show the green selection box around the selected object by pressing the Show Bounding box icon in the Viewing Toolbar:



Try also Clipping functionality through the Clipping Tab to look inside the model.

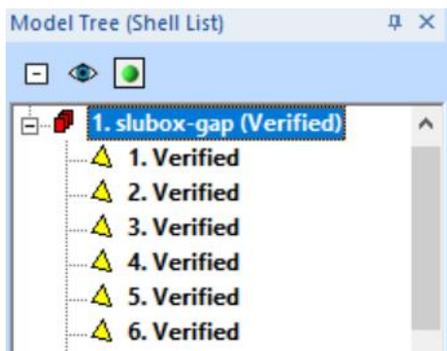
The Clipping Tab is opened with the  icon. The clipped area can be displayed in different ways, like as wireframe or transparent.

Clipping also enables to extract real time slice information from the model. *Real time slices* can be copied to the Model Tree for internal model dimensioning with the *Edit > Save Clip curves* command.



Selecting objects

Object selection can be done either from the display area or from the Model Tree. **Erase the screen and then display the object node “1. slubox-gap (Verified)” again** by clicking it in the Model Tree. Now we have done one selection at the shell level. This level can also be called as the part or component level, depending on the context. Click on the little box with the '+' icon besides the part name. The result should look like below:

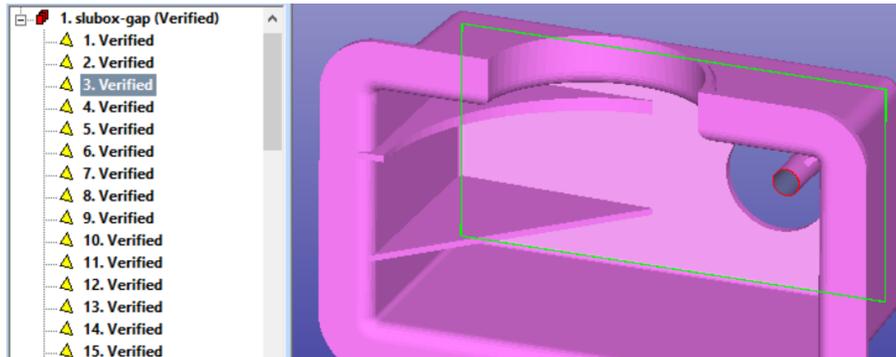


The node “1. slubox-gap (Verified)” has opened and shows the contents of the *shell*. Each shell consists of several leaves, *surfaces* (1. Verified, ...), which all together make up the whole list. Triangle surfaces are identified with the  icon. An STL model is subdivided into separate surfaces along sharp edges during the Verify operation, as done earlier.

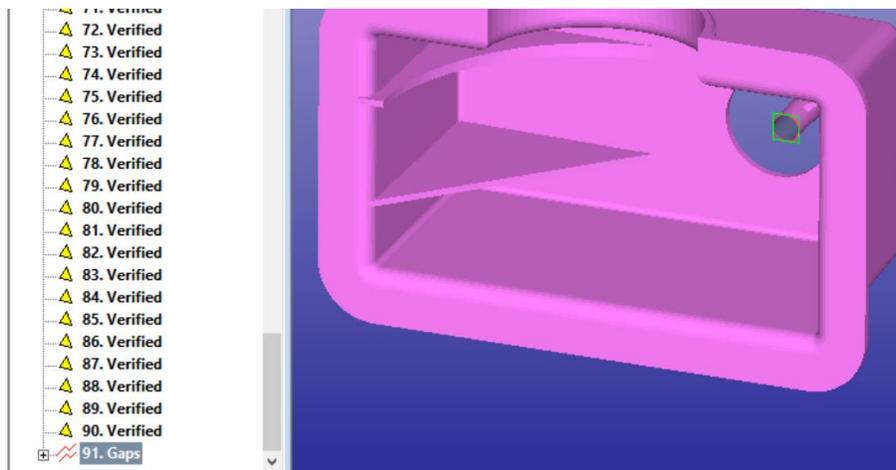
Select surface number 3 from the model tree by clicking it with the *left mouse button (LMB)*. You can see one of the surfaces displayed with a green Bounding Box around it. Now this surface is your current selection (make sure the bounding

box display is set with the  button.

The selected surface is also highlighted on the screen:



The model contains errors, one gap. The gap data is also located in the Model Tree after the surfaces. Similarly, gaps can be selected from the Model Tree for different operations, like *Fill Gap* command (to be done later).



Selecting shells, surfaces and triangles graphically

You can also select shells or surfaces from the display area by pointing with the *left mouse button (LMB)*. First

- **display the Tools Window by pressing Shift+F8** (if not already visible):

The Tools Window includes shortcuts to frequently used repair, coloring and print preparation tools as well as Help text to aid with the repair work. You will later learn more about the use of the Tools Window.

In the following we will use the *Selection mode* buttons to set the Selection Mode to correct level.

You can use *Shift + F8* to open and close the Tools Window.



The selection is done with the *left mouse button (LMB)*, and the selection target is controlled by the *Selection mode* buttons in the Tools Window:



The four *Selection mode* buttons refer to (from left to right) selecting the *whole model*, *one shell*, *one surface*, *one gap/triangle*. Now try to select a shell from the display area, such as the “1. slubox-gap (Verified)”.

If the *Selection mode* is not *Shell* (which is the default) **press** the  **button** to turn on *Shell* selection. Then **click on the model with LMB** and the shell will be selected.

Then **set the Selection Mode to Surfaces** with  **and click with LMB on surface 3** which now gets selected. If you wish to select objects behind *surface 3* just re-click with LMB (without moving the mouse. Instead of LMB, you can use the *SPACE* key, also without moving the mouse (this can help especially if mouse movement causes problems with a high-resolution display).

Then **press** the  **button** to turn on the *Gap* selection and click close to the gap on the graphics area to have it selected in the Model Tree.

Note: *Triangle selection* mode is available through the Color tab, and it is reserved for painting the model; please see “*Tutorial 2 – Working with Colors*” for more information. Otherwise, triangles are selected and manipulated through Fix Model > Edit Triangles command.

Multiple selection can be done by pressing the *Ctrl* key down while selecting objects from the graphical area or from the Model Tree with the LMB. The *Shift* key can be used to select objects between the first selection and the latest selection in the Model Tree. Different combinations *Alt*, *Shift*, *Ctrl + LMB* on the graphics area create, add and delete objects from the multi-selection, please see Online Help page *Selection / Selecting objects*. Multiple selection can be used with several commands, like Transformation or Fix Model menu commands.

Display Area menu

The *right mouse button (RMB)* has a special meaning on the display area.

- Press the **Surface selection mode** button  again.
- Clicking the **RMB** on top of a displayed object now brings up the **Display Area Menu** (not available in *Triangle/Gap* selection mode):

When you click the RMB on the model while in *Surface selection mode* the surface which is nearest to the cursor is indicated on the first menu item.

- Click RMB **on top of surface 3 again**.
- With the **Next/Previous** command you can indicate surfaces which are behind *surface 3* along the line of sight at the mouse position.



You can also change the shading and visibility of the selected object with the commands in the menu. If you wish to select the indicated surface, **press Select**.

The *1487: OK triangle* entry in the RMB menu indicates the input number of the triangle below the cursor. If this menu item is selected, more information about the triangle is displayed. This can be helpful especially when working with triangles with errors.

The RMB menu of the graphics area is shorter in other modes than *Surface selection* and, for example, in *Shell selection* gives the name of the shell at the mouse location as the first item.

A similar popup menu is available on the Model Tree with the RMB click, please try that too.

Object properties

Information about the properties of the currently selected object is given to the user in two different ways.

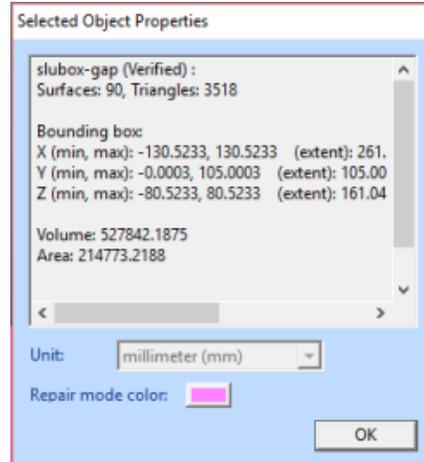
The basic information about the current selection is given below the Model Tree window. Much of the information is same as in the *Object properties* dialog (as explained below), but the information is structured in a different manner. There are also differences, like *shell history* which is shown more clearly in this list.

Name	1. slubox-gap (Verified)
Surfaces:	90
Triangles:	3518
Bounding b...	(-130.5233, 130.5233), (...)
Extents:	261.0465, 105.0005, 16...
Volume:	527842.1875
Area:	214773.2188
File name:	slubox-gap.stl
Directory name	R:\DA1107projects\Ex...

There are three places to display the *Selected Object Properties*:

- The menu command **View > Object Properties** or
- the Model Tree RMB popup menu command **Properties** item
- or the Display Area popup Menu **Properties**

The *Object Properties* window contains a couple of items which are not available in the information below the Model Tree, for example the options to temporarily choose the object's display color, and at root level, change the unit of the model.



The exact position of the part is given as bounding box values both in the information field and the *Object Properties* window. Also, the extents, volume and the surface area of the object are calculated.

Note: with this example the *Volume* is not correct. This is because the model is not solid. The *Volume* is reliable with Repaired faceted models without gaps and for surface models with consistently oriented surface normal. The *Area* value is always valid both for surfaces and faceted models.

Dimensions and Annotations

For communication and documentation purposes it is often important to include dimensions and annotations to be stored with the model. The Dimensions menu gives us the tools to add dimension information to the model.

- **Set Shell Selection mode** with the  shortcut button.

Measuring and annotation tools can be started from the Dimensions menu or through the toolbar:

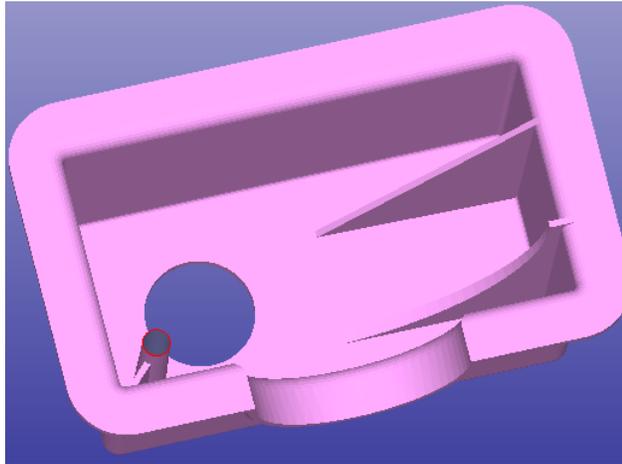


If the toolbar is not visible,

- give command **User Interface > Toolbars > Dimensions Toolbar**.

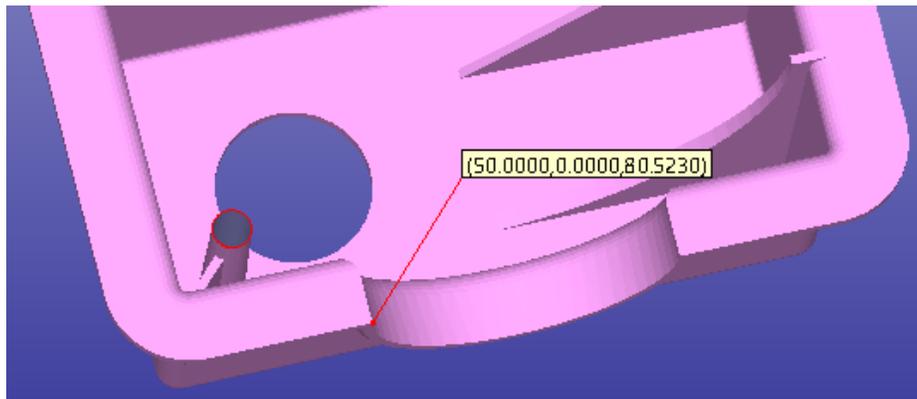
Let's first take an absolute point value from the *slubox-gap* model.

- **Rotate and zoom the model** so that you can see it as shown here (you can also change the viewing and clipping during the dimensioning to get a good view of what you are measuring):



- Give command **Dimensions > Point Value** (the  icon) .

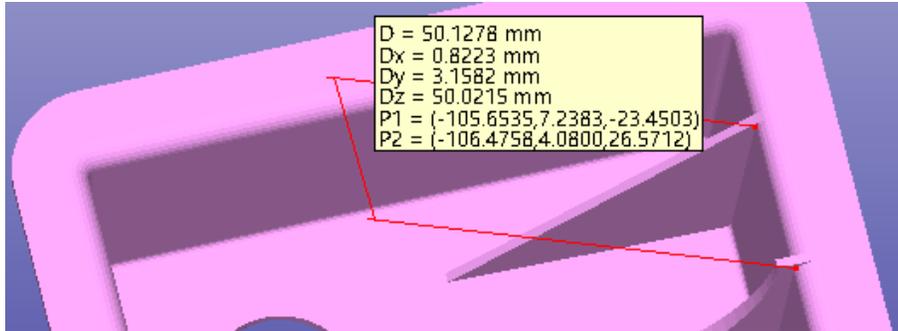
The *Crosshair* icon is shown on the display area and the program is waiting for you to select a point on the model for coordinate values. Select one point from the model and you will see a result like below (cropped):



Let's now try distance between two points. This is done with

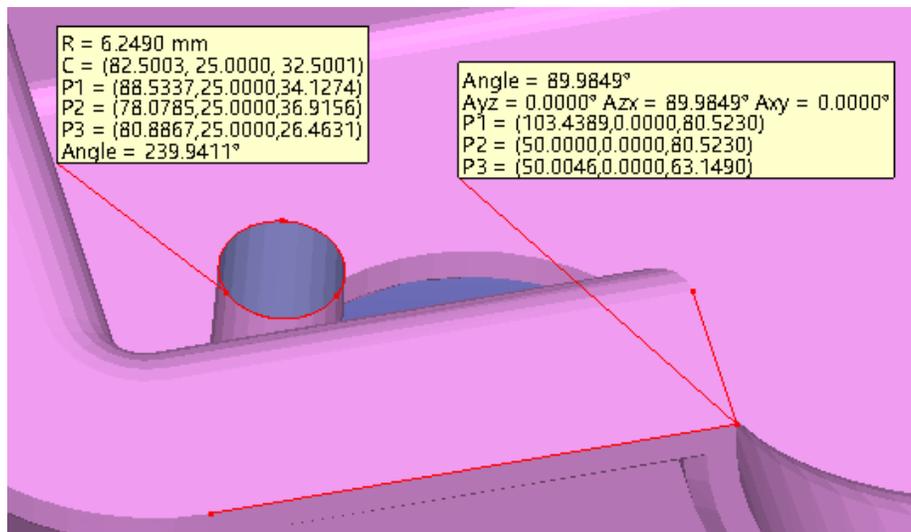
- command **Dimensions > Distance** or the  icon.

After you give the command, you will see a new *Crosshair* icon. The number *1* beside the crosshair icon tells you that it is waiting for the first point for the point distance evaluation. Now select the first point and the second point (crosshair with number *2* beside it) so that the result would look like the one on the previous image. Press *RMB* to accept or *ESC* to quit or cancel a selected point.



In the same way you can take *Angle* and *Radius* using the Dimension menu. Try to repeat the example shown on the next page. Keep in mind that you can *Zoom*, *Pan* and *Move* the eye point using the mouse and keyboard shortcuts. *Dimensions* > *Wall Thickness* allows to measure the wall thickness at a selected surface point.

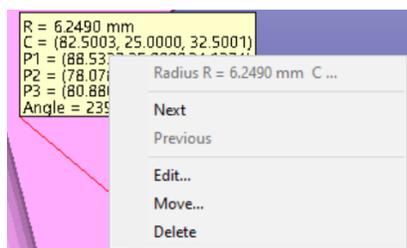
Pressing *LMB* only selects the closest surface point below the cursor for dimensioning. Pressing *ALT + LMB* causes the closest *triangle corner point* to be selected instead of the surface point below the crosshair icon. This will enable selecting accurate triangle corner points for the measurements:



Now you are ready to edit and move the annotations (or measurements).

- Click with the **right mouse** on top of an annotation.

You should see the *Annotations popup* menu below. If you see the Display area *RMB* popup menu select the *Next* command in the popup menu until you get menu associated with the annotation:



This menu allows you to *Edit, Move and Delete* existing annotations. Try these commands on the selected annotation. The *Edit* operation only allows you to change the text contents: to re-measure the target points, you must make a new measurement (and possibly delete the original annotation). The *Move* operation similarly only moves the text box, not any of the measured points.

You can also add free annotation texts and messages with *Dimension > Mark-up* command ().

You can display model information on the screen by simply giving the command *Dimensions > Object Info* ().

To copy the object info into the cut buffer, give command *Dimensions > Copy Object info* (). From the cut buffer it is easy to paste the information to Microsoft Excel, for example.

Dimensioning can be made *Sticky* through the Preferences tab (see *Edit > Preferences > General Tab*), you can then give as many consecutive *Point Value, Point Distance* etc. commands as needed without the need to repeatedly restart the command. In the Sticky mode you should press *ESC* or *RMB* to quit when you are ready. You may try it now but remember to turn *Off* the Sticky mode before continuing.

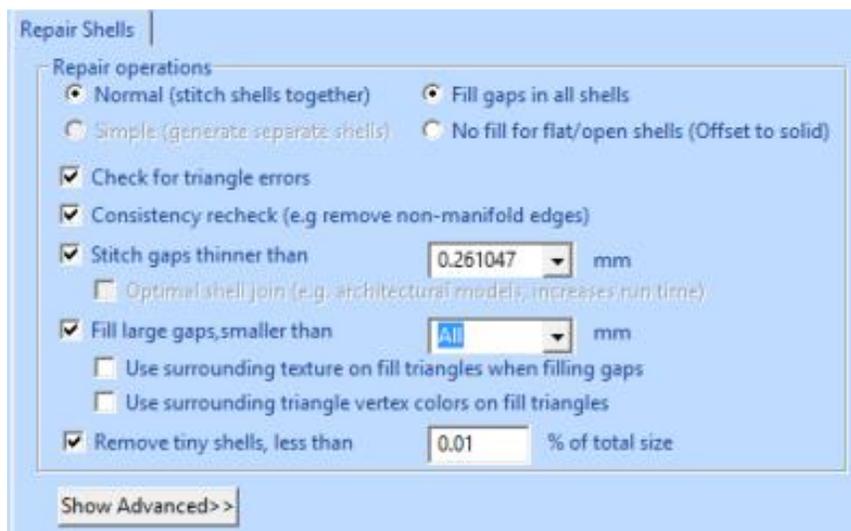
You can now delete all annotations by pressing the  icon.

Repairing the slubox model

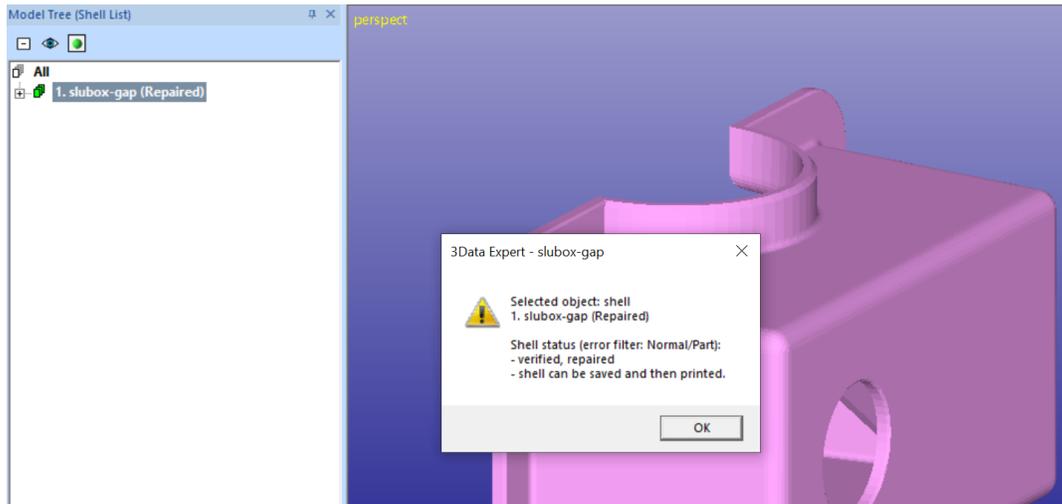
Now it is time to repair the gap in the slubox-gap model.

Make sure the shell “1. slubox-gap (Verified)” is selected in the Model Tree. Then

- **give the *Fix Model > Repair Shells* command** and you will see the following parameter dialog:



- Just **press OK** when the parameter dialog appears, and you will have a correct slubox model in a second:



- **Press OK** to close the Status message dialog.

This was the first lesson covering viewing, dimensioning, popup menus, object selection and other basic software usability issues. In the sequel we will show the use of the actual repair and 3D model manipulation commands.

Lesson 2 – Fully Automatic STL fixing

This lesson shows you how to fix an erroneous STL model for 3D Printing or simulation software use using the automatic tools with 3Data Expert software.

Clean up the software workspace before continuing by

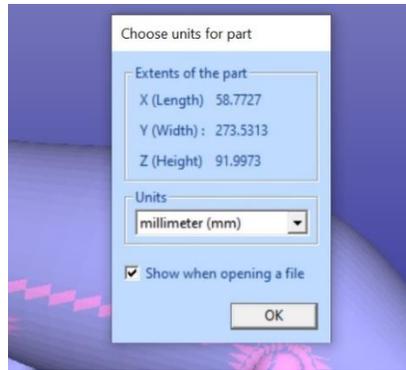
- **giving the *File > New* command**. Do not save the result of the previous example.

Auto Repair process for STL files

When inputting a triangulated model into 3Data Expert an automatic analysis and repair process is started. In normal everyday use you can correct most of the models using the Auto Repair process.

- **First open the *dino-connected.stl* file** in the *Tutorials / GeomFiles* directory.

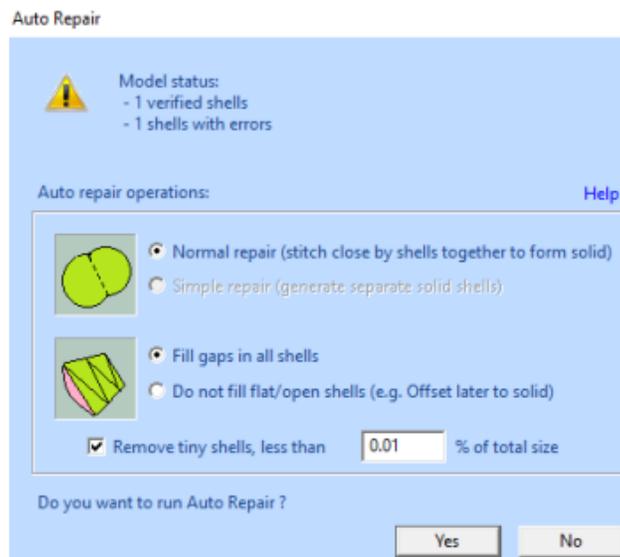
You will be prompted for the units, accept the proposed unit, *millimeters (mm)*:



- **Press OK** to continue and

you will be prompted for the Operation Mode. To start fixing **press the Fix button** in the Operation Mode dialog now.

3Data Expert will begin automatic model verification (the *Fix Model > Verify Shells* command). The screen is erased, and a progress bar appears. When the analysis is ready, the *Auto Repair* dialog with *Model status* message is displayed:



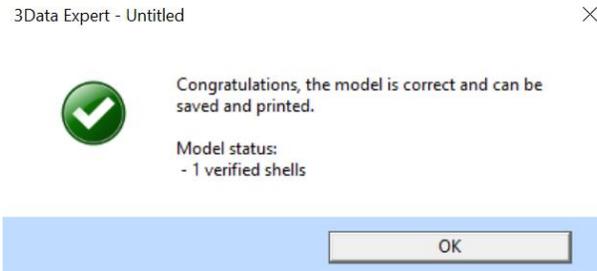
The message indicates that there is one verified shell, which also contains errors. The display shows red *Gap curves* through the surfaces. Gaps are an error type you must fix during the repair process.

- **Press the Yes button** to initiate Auto Repair. The parameters are automatically set correctly.

Auto Repair will attempt to fix all shells with errors. When you only have one shell in the Model Tree you should always try Auto Repair first.

When Auto Repair is started the shell is run through the automatic repair command (*Fix Model > Repair Shells* command) using automatically calculated parameters. For each shell, a progress bar shows the progress of the repair. After the Auto Repair process finishes the following message is displayed for the *dino-connected* model:

3Data Expert 14.1: Tutorial 1 - Working with 3D Models

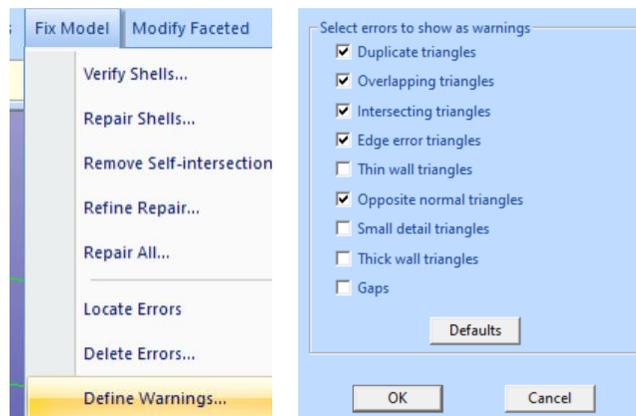


- **Press the Close button in the upper right corner** to close the window. Also close the possibly appearing tips and platform fit warning message.

The model is now ready for output and 3D printing.

During Auto Repair, 3Data Expert attempts to fill all gaps and remove non-manifold triangles (overlapping triangles, multi-edge triangles, duplicate triangles, triangles having less than three neighbors, etc.) from the model. Also, duplicate and overlapping surfaces are identified and optimally joined. To see more about triangle errors, please access the "Triangle errors" page in the Online Help chapter "Handling Faceted Files".

With certain models, you may see a different result icon for a shell in the Model Tree: . This means that some errors have been found in the model, but they do not negatively affect the 3D Printing. The Help text below the Tools Window shows what errors remain in the model. What is considered a significant error, depends on the properties of the 3D Printing process. You can also change yourself the settings of what is an error and what is a warning, through *Fix Model > Define Warnings*:



The *dino-connected.stl* model only has simple errors, like inverted triangles and gaps, which are all fully repaired during the Auto Repair process. You can now continue to save and print the model, as indicated by the status information in the *Fix* tab Help text window.

Lesson 3 – STL fixing

This lesson shows you how to fix slightly more erroneous STL model for 3D Printing or simulation software by using the automatic tools with 3Data Expert.

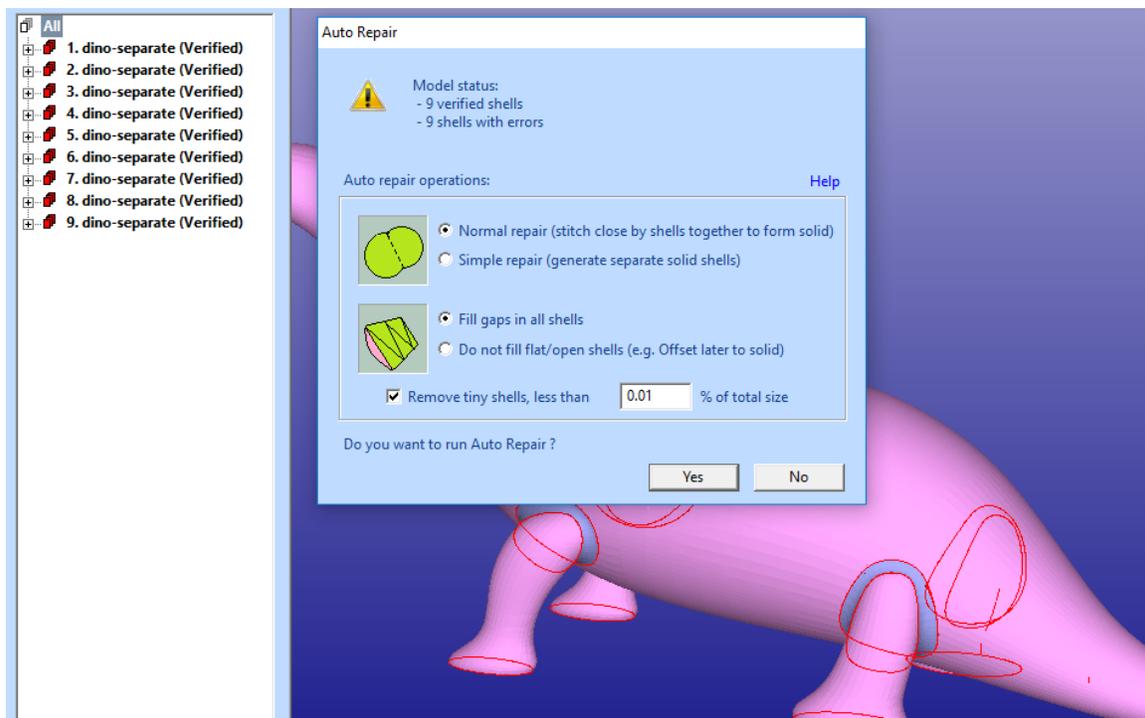
Verifying and repairing the STL model

Start this lesson by

- **File > New command** to clean up the workspace. Do not save the result of previous example.
- **File > Open the STL model *Tutorials > GeomFiles > dino-separate.stl*.**

The software will prompt you for the Units and Operation mode as usual.

- **Accept millimeters (mm)** units and
- **start the Fix mode.** After a while the software displays the Auto Repair dialog with Model status message:



This is the result of the automatic analysis (a separate verification of the input model with *Fix Model > Verify Shells* would yield the same result). We can see that the 9 shells in the Model Tree are all separated by thin gaps.

The automatic verification and fixing functions in 3Data Expert have now analyzed the model and propose the following repair actions:

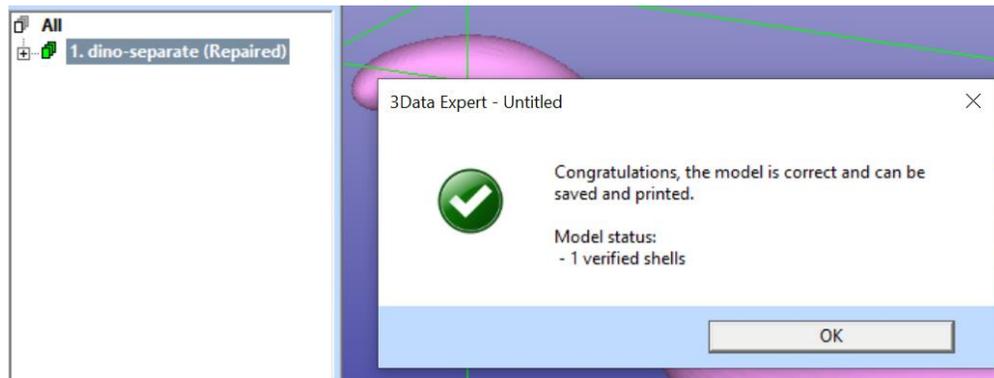
3Data Expert 14.1: Tutorial 1 - Working with 3D Models

- *Normal repair* to stitch separate shells into one shell. This is done when more than 70% of the open edges between the different shells can be joined
- *Fill gaps in all shells* is proposed as the model contains open/flat shells in the result which may require Offset or Extrude for proper fixing into a solid shell.
- Remove tiny shells, less than 0.01 % of the total size, is proposed. This will ignore all insignificant shells smaller than the given size in the result. Minor shells can be the result of needle triangles and other minor errors in the model.

We will now repair the model with complex gap errors into one single shell with the Auto Repair process.

- **Press Yes to run** the automatic repair on the model

After a while we will have a fully repaired model in the Model Tree, ready for 3D Printing.



Close the dialog by pressing the Close button in the upper right corner of the message dialog.

Verifying and repairing, take two

In very old versions of DeskArtes 3Data Expert (10.0 and earlier) it was necessary to run the automatic repair for this model through *Fix Model > Repair Shells* command. In the following, the old way to fix *dino-separate.stl* file is repeated learn some basic concepts of model errors.

Restart the lesson by

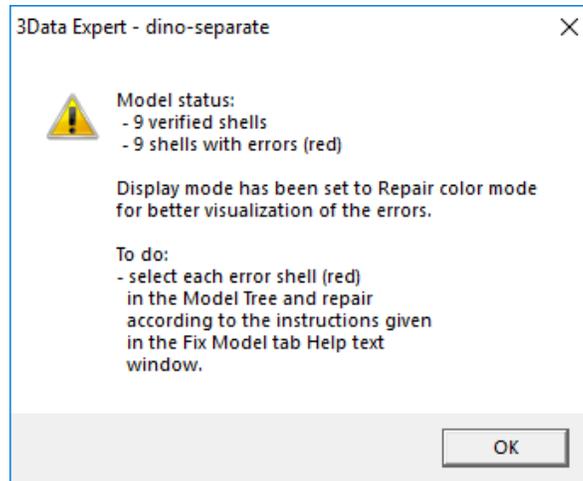
- **File > New** command
- **File > Open** the dino-separate.stl
- Accept millimeters
- start Auto Repair by **pressing Fix** in the Operation Mode window

You will again see the Auto Repair dialog with the status message.

- **press No** to quit Auto Repair process



A status dialog appears:

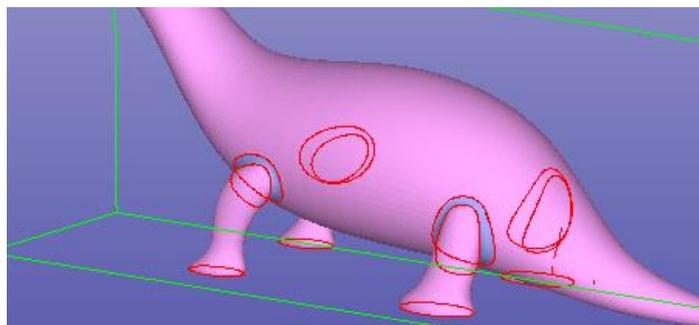


- When the status dialog is displayed, **press OK** to close it. Close the Tip and does-not-fit-into-platform windows as well.

The *dino-separate.stl* part is split into nine shells (components) which together will make the full part.

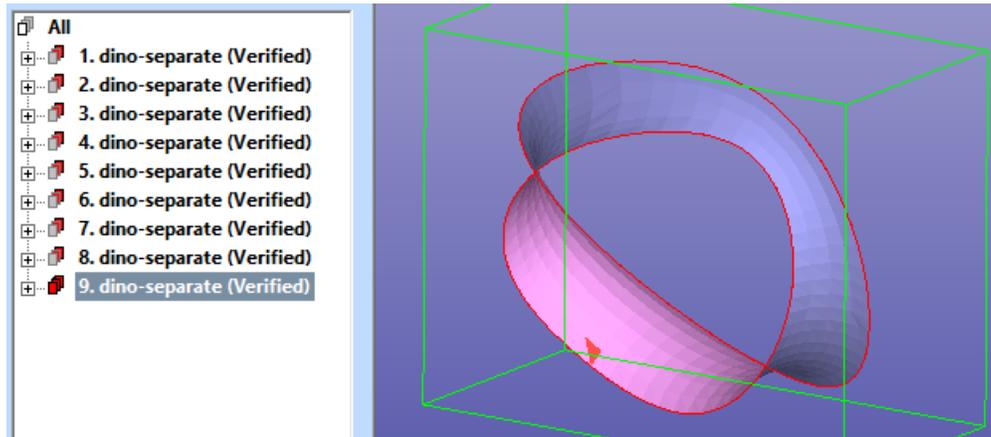
Using Repair Shells command

Thin gaps between the surfaces in a triangulation is normally caused by an erroneous surface triangulation for 3D Printing; The surfaces are triangulated separately without connecting the faces at the trim curves (or surface edges). Larger gaps may be due to missing surfaces in the original model. With DeskArtes 3Data Expert you can easily repair these STL model errors:



In the Model Tree, after the *Fix Model > Verify Shells* command, the *dino-separate* model components are sorted according to surface area. This helps us to first delete unnecessary minor components from the model, by just checking the last small components in the Model Tree.

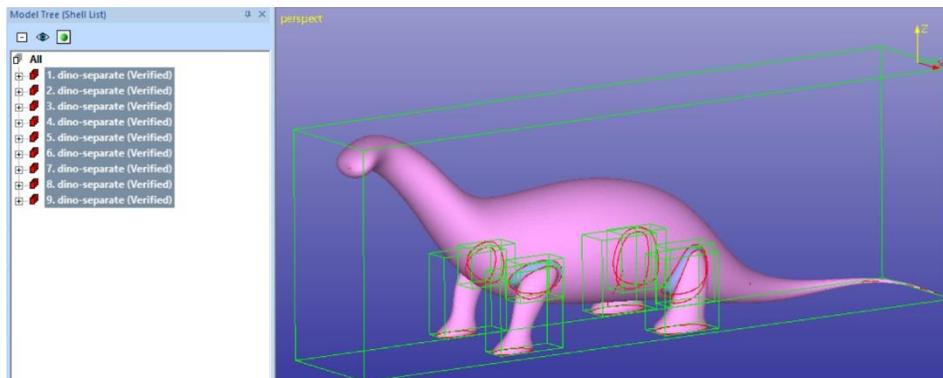
- **Select the last component 9. *dino-separate (Verified)* and Fit it on the screen:**



We can immediately see that this is an important component and must not be deleted. The components before the 9th are even bigger and must thus remain in the model. So, in this case we won't immediately delete anything.

The next step is to combine the separate *dino* components into one connected part.

- **Use multiselection on the Model Tree** to select all components (or select Model Tree root):

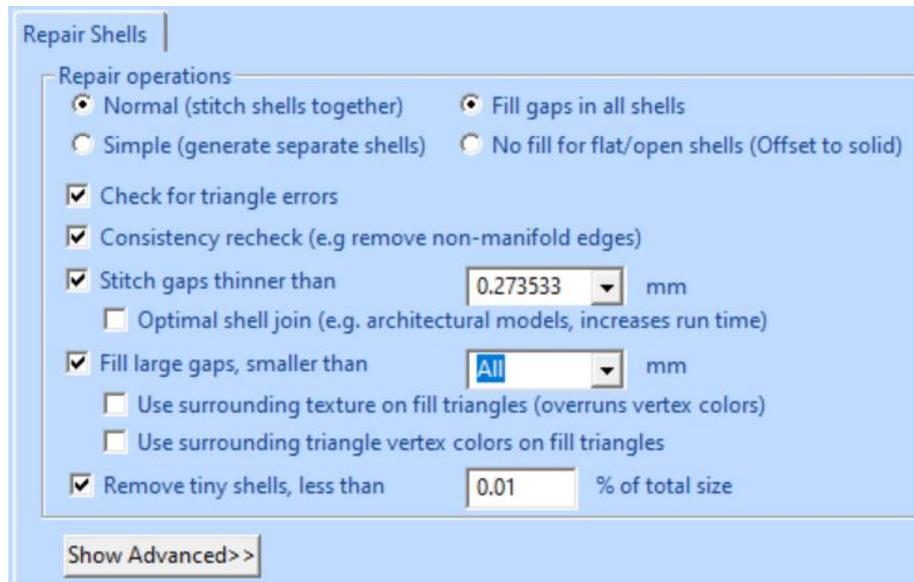


- Then **start the *Fix Model > Repair Shells* command** to stitch the components into one part and to fill in the missing soles of the *dino*.

The command can be started from the Fix Model Tab shortcut button, too:



Use the default parameters:

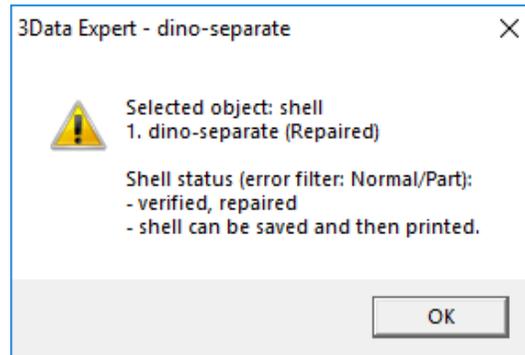


There are several parameters for the Repair Shells command. The parameters are automatically calculated for each model and can normally be accepted as is:

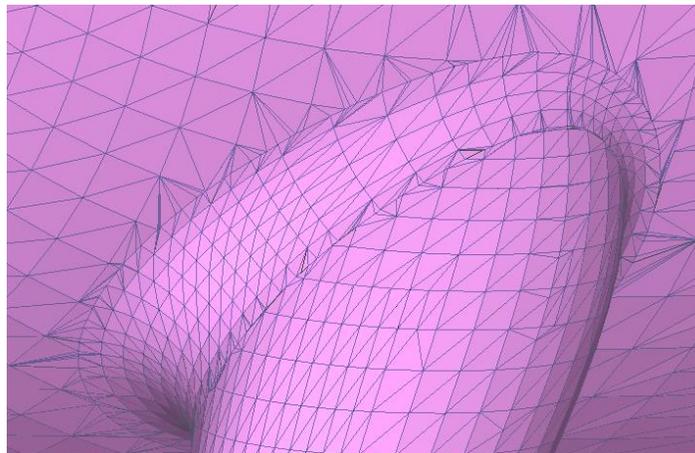
- *Normal vs. Simple repair:* *Normal* will stitch separate shells into one shell, if possible. *Simple* repair repairs each shell separately and does not connect them into one.
- *Fill gaps in all shells vs. No fill for flat/open shells:*
 - *Fill gaps in all shells* fills all gaps in all shells, regardless of the shell shape, according to the *Fill large gaps* parameters.
 - *No fill for flat/open shells* leaves flat shells unfilled. These shells may be more suitable to be filled with *Offset* or *Extrude surface* commands.
- *Check for triangle errors:* to report intersections, multi-edge and overlap conditions
- *Consistency recheck:* to check duplicate and overlapping surfaces and use optimal connection algorithms between these surfaces to form a solid shell.
- *Stitch gaps thinner than:* to connect separate components where the distance between open edges is less than given value
- *Fill large gaps:* to close all gaps smaller than given value. Value *All* is applied to close all soles of the dino. For textured and colored models you may want to apply *Use surrounding texture* and *Use surrounding triangle vertex colors* to keep the original colors in the result, otherwise the default foreground color is used for fill triangles (the foreground color is set in the **Color** operating mode tab).
- *Remove tiny shells:* shells whose area/volume is less than 0.01 % of the total area/volume are removed.

- Check that parameters are as in the image above and then **press OK** to start the Repair Shells command

After a while the *Repair Shells* operation will finish and the status is displayed:



The software reports a fully fixed model which can be printed with a 3D Printer. A close-up at the leg/body connection shows how the triangles are stitched at the gap area (displayed in shaded+wireframe mode):



We have now fixed two models with relatively simple errors. For more information on how to fix more difficult STL and VRML models, please see the *3Data Expert 14.1: STL and VRML repair examples* document. The document covers the fixing of several difficult models for 3D printing and Additive Manufacturing.

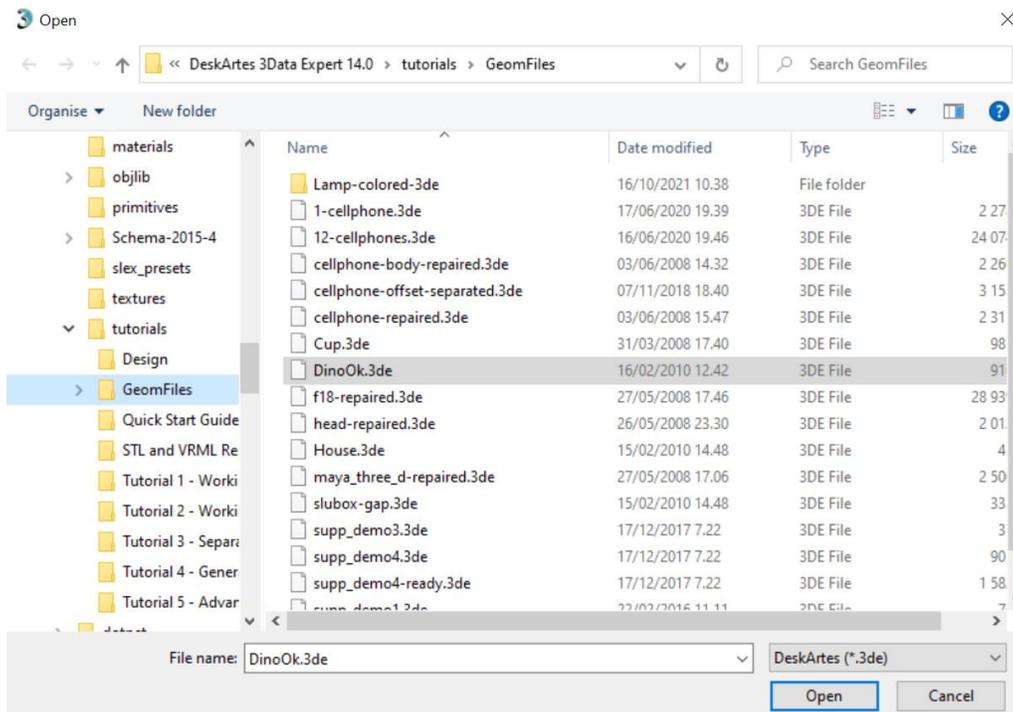
Lesson 4 - Splitting and Combining STL

This lesson shows you how to split large STL models for 3D printing and Additive Manufacturing. Building a very tall part may be more expensive than building two or more lower parts simultaneously. Also, the building area may not be large enough for a very big part. Correct joining of the split parts is ensured by adding guiding pins to the models.

This lesson is not applicable to 3Data Expert Free. 

Splitting the *dino* model

- Start by inputting the DeskArtes geometry file *DinoOk.3de* In the *Tutorials / GeomFiles* directory.
- Set the *file type to .3de* to input DeskArtes proprietary files:

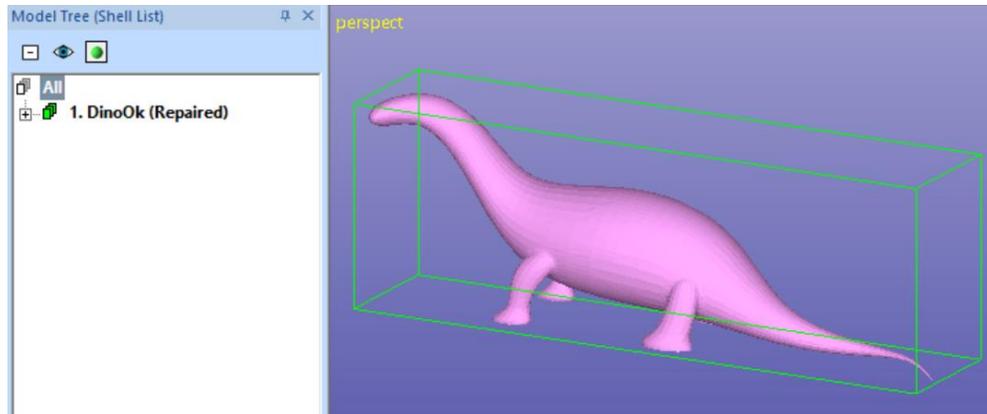


The *DinoOK.3de* file includes a correct triangulation for the familiar dino model.

You should accept the proposed units and **go directly to the *Modify* mode** by pressing the



button in the Operation Mode Window. You should see the following dino model:



with the Modify Tab on the right-hand side of the screen.

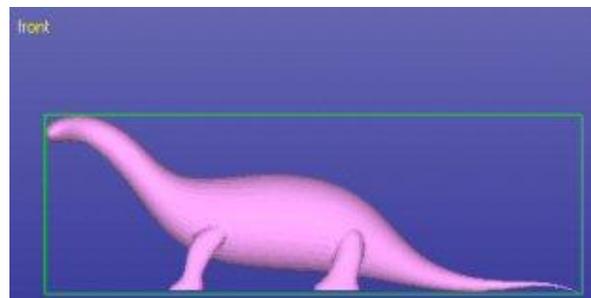
The Modify Tab gives shortcuts to the commands most used for adding or changing geometry before the final print preparation, like split, connect, add lap joint, hollow, offset, drain hole, 3D text. The Help text provides context sensitive help to the commands.



- **Select node *1. DinoOk (Repaired)*** by clicking it with the left mouse in the Model Tree.

Before splitting we must take a correct view to the model because the splitting plane is generated by extruding a user drawn split polygon away from the viewer. With 3D Printing the correct way normally is to split the model perpendicular to the z-axis to make the part lower. Thus,

- **take a view from the x-axis direction with *View > Ortho View > Front (+X)*** (+ LMB).

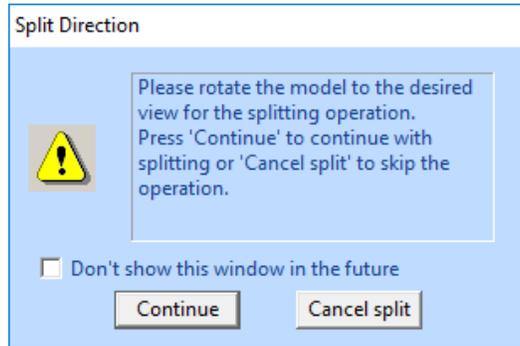


The splitting is

- **started now with command *Modify Faceted > Split* or with icon**



The system prompts you to take the correct view (in case you haven't set it yet):

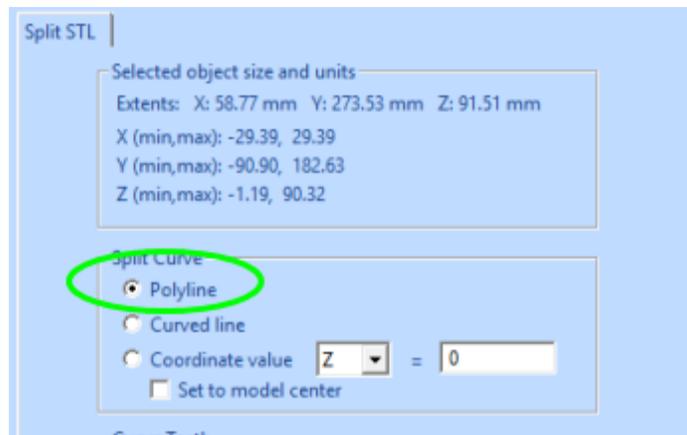


When the view is correct you can

- **press *Continue*** or *Cancel* to quit. You may hide this window permanently when not needed anymore.

The system asks you if you wish to use a *Polyline (linear)*, *Curved* split line or exact *Coordinate value*.

- **Select *Polyline***

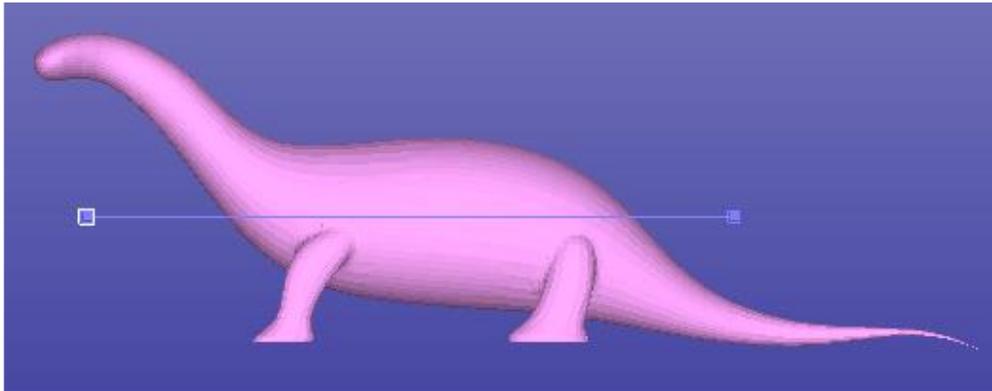


- **press *OK*** and the system goes into the curve edit mode where you can define the shape of the splitting surface.

Now you can draw the shape for the splitting surface.

- **Draw a horizontal line by giving two points with the LMB clearly outside of the *dino* model**

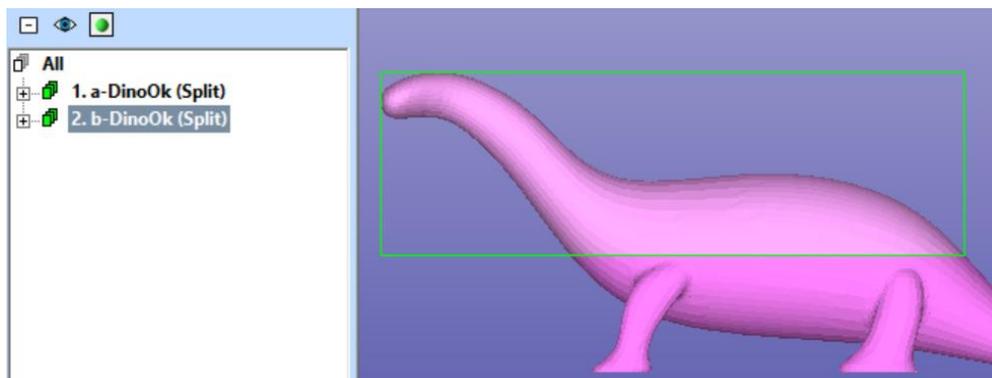
as seen here (the line is exactly horizontal when a dashed help line extends as a continuation of the line):



The curve editing is finished by

- **pressing the *RMB*** (or *ESC* key to cancel if you need to redraw)

When finished the model is split along the plane defined with the user drawn polyline, Select the second shell in the Model Tree to highlight the top half:



The two halves are grouped and named with *a-* and *b-* prefixes to distinguish between the shells above the split line and below the split line.

Note: If you are splitting models with more than one shell you can select the *a* or *b* part with the *Groups* command in the Model Tree popup menu.

Creating the pins

The separate parts of the model could now be stored into *.stl* files for 3D Printing. However, precise joining of the ready parts would be difficult or impossible. To be able to position the parts accurately after the build, we shall add pins and holes in the shells to connect them together when they come ready from 3D Printing.

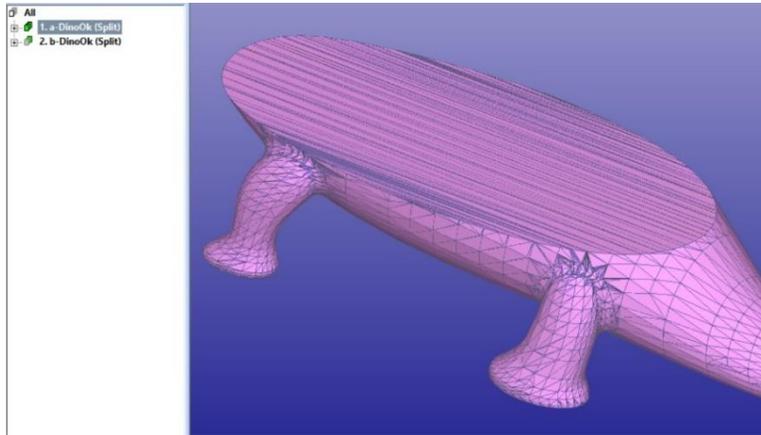
We will first select and show the locations of the primitives required for connecting the two parts together. For that we will get a nice view for pointing the pins on the

3Data Expert 14.1: Tutorial 1 - Working with 3D Models

screen. Here we are also using the shaded+wireframe view mode through the  icon in the Viewing Toolbar.

- **Rotate the model** to position shown in the image below
- **erase the screen with**  **and**
- **select the lower part of the dino and Fit it**  **on the screen**, so that it is displayed properly on the screen
- **set on shaded+wireframe mode with the**  **icon**

Zoom close to the model to be able to accurately position the pins:

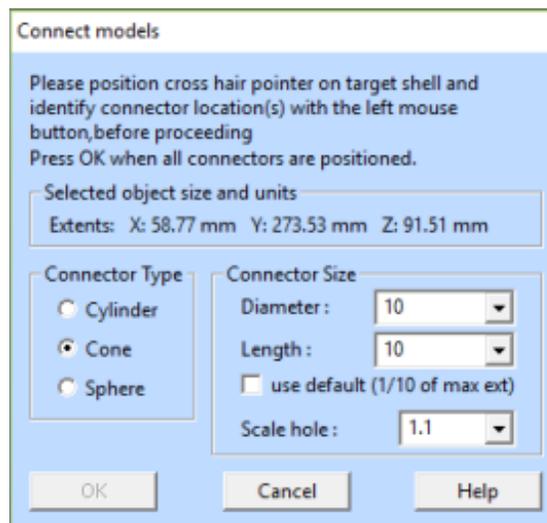


The pins are now

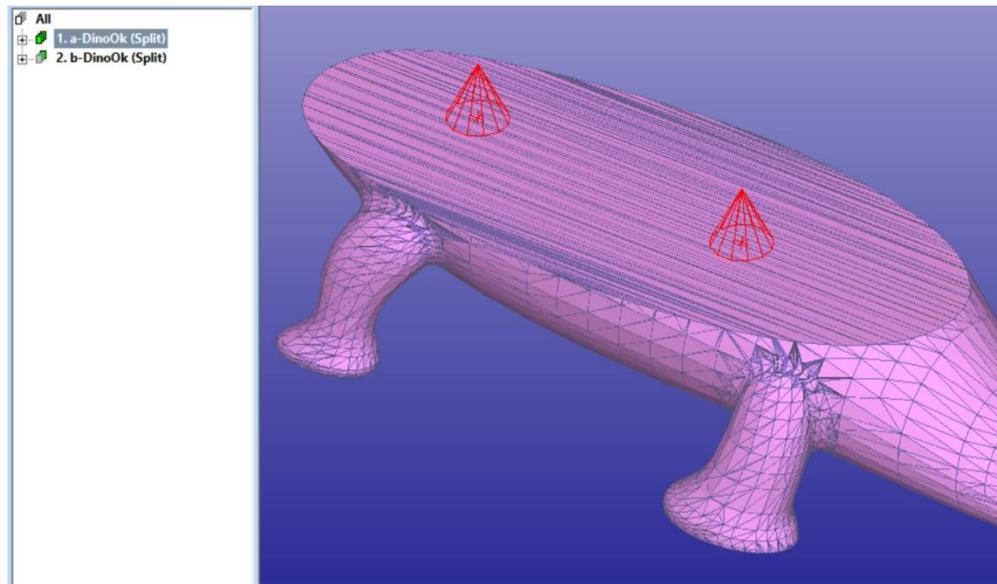
- **added with the *Modify Faceted > Connect* command** or using the Modify Tab shortcut



After the command is given, it is possible to give the *shape* and the *size* for the pins to be created. Set *Diameter* and *Length* both to 10.0, as seen below:



The positions for the pins should look like the this:

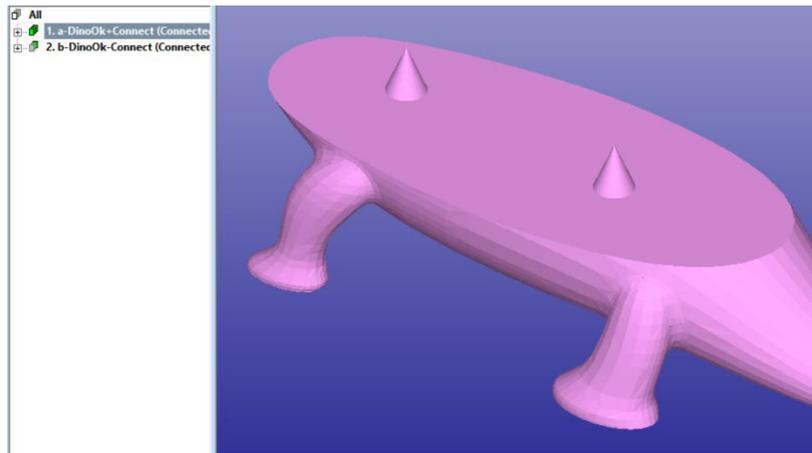


If a pin is incorrectly located on the shell, you can remove it by clicking the red indicator cone again with the LMB. Then reposition the pin again with an LMB click.

When ready

- **press *OK* to generate the connectors.**

After a while you will see the connectors added to the selected shell (in shaded mode again with the  icon in the Viewing Toolbar):



Corresponding holes are automatically created into the *b-dino-Ok* shell. You should move the shells apart from each other with the Transform menu commands prior to outputting the model for 3D Printing.

For a robust Boolean operation between the selected surface and the pins, the connector is positioned on the surface so that 10% of the height is below the pointed surface. 90% of the pin length is above the surface and will be cut into the opposite part.

Saving the result

You can save both files into the same STL file by selecting the Model Tree root *All*. Anyhow, you normally want to build and position the parts separately in the 3D Printer. For that select each shell separately and

- **output** it to a proper STL file **with *File > Save As* command**.

If you wish to save the two parts into one file, you should move the shells apart from each other with the Transform or Print Preparation > Move Platform menu commands prior to outputting the model for 3D Printing. To output one file, select the Model Tree root All before the Save As command.

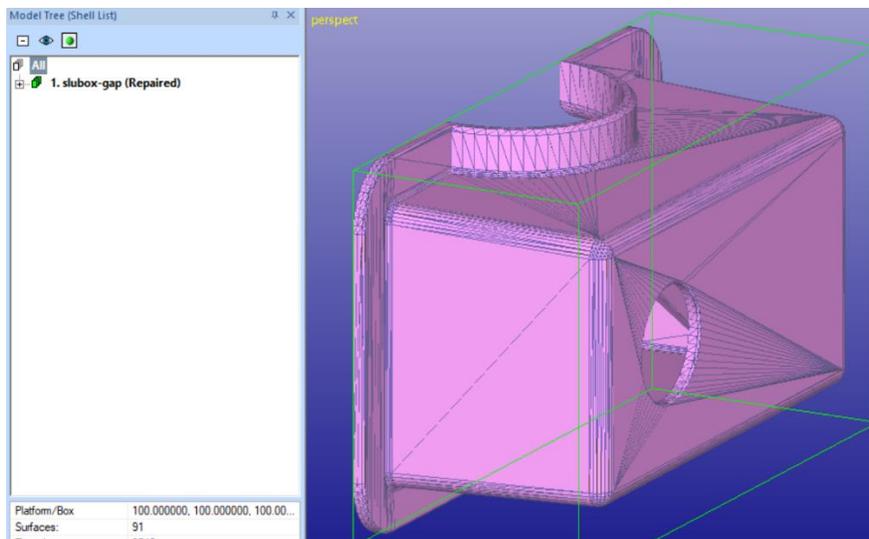
Using Lap Joint to connect thin parts

With this example we will split a part with thin walls and connect it with a *lap joint* connection. A lap joint is a small overlapping structure on both sides of the wall along the split surface edge. With Lap Joint command you can generate strong bond even along very thin walls.

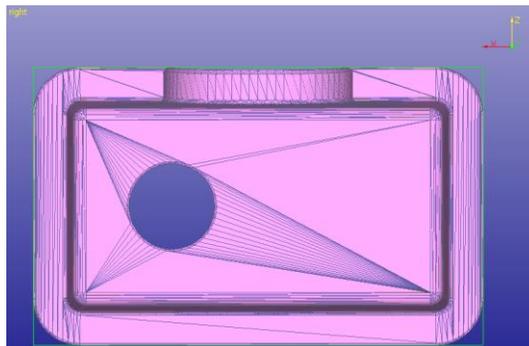
We will first input the familiar *slubox-gap.stl* file and repair it for Splitting:

1. Input file *slubox-gap.stl* file with **File > Open**.
2. Accept millimeters as the unit.
3. **Choose Fix** in Operation Mode to go to the Auto Repair process.
4. When asked “Do you want to run Auto Repair” **press Yes**.
5. And after a while you will get “Congratulations, the model is correct and can be saved and printed” message.
6. Press Close button in the upper right corner to close the status message window. Also close other possibly appearing messages.

You now should have the *slubox* model fully repaired for splitting:



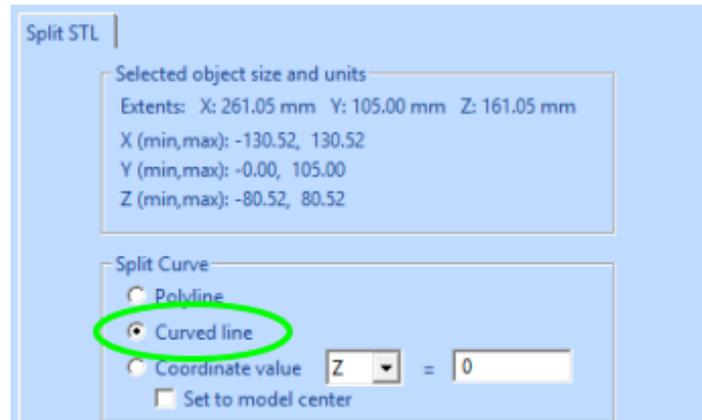
- Take a suitable **view direction +Y for splitting** by pressing the  icon:



Start model splitting with

- **Modify Faceted > Split** command now and
- **select Split curve type *Curved line***, as seen in the next page.

This time we want to make a split line which is not straight, i.e. it has teeth, and the split line corners are slightly rounded. This will allow us to create a connection with no freedom of movement between the split parts as well as to avoid small holes between the connected halves due to too sharp corners in the split plane.

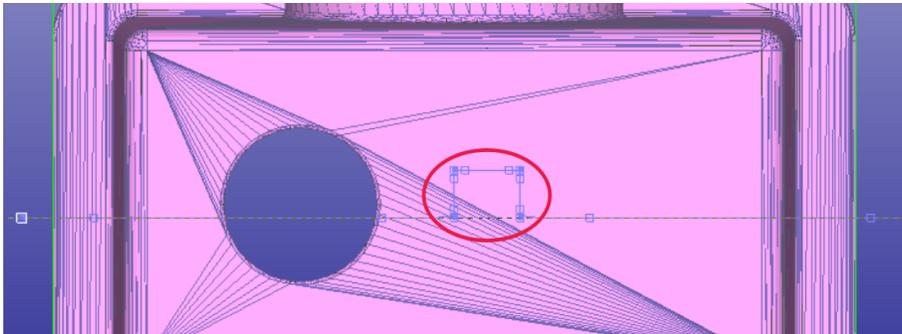


- **Press *OK*** to start adding curve points.

For each split curve point at the curve corners below

- **press twice for each point with the left mouse** (points inside the red ellipse below).

This will generate a rounding with small radius at each corner (single click would generate a “wavy” split line):

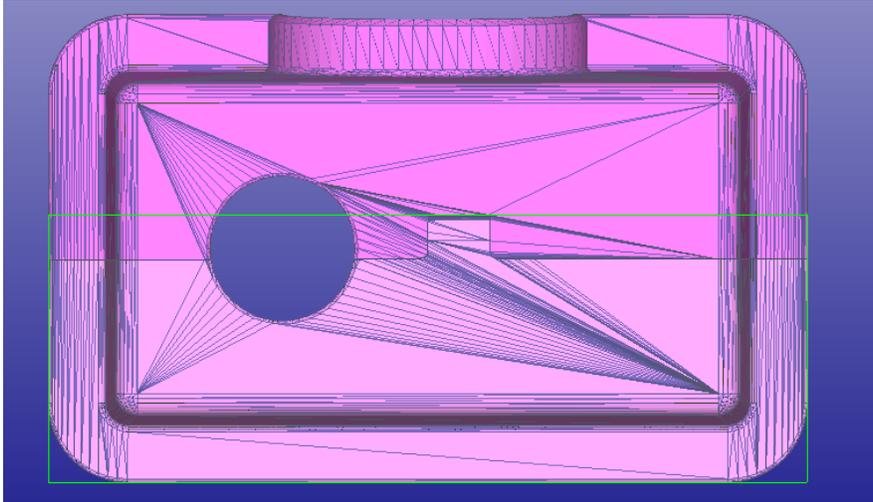


Or if even a bigger radius is needed at each corner, create two points close to each other, not double points³. This will increase the radius at each corner and allow to use bigger Lap Joint Height values later, for example.

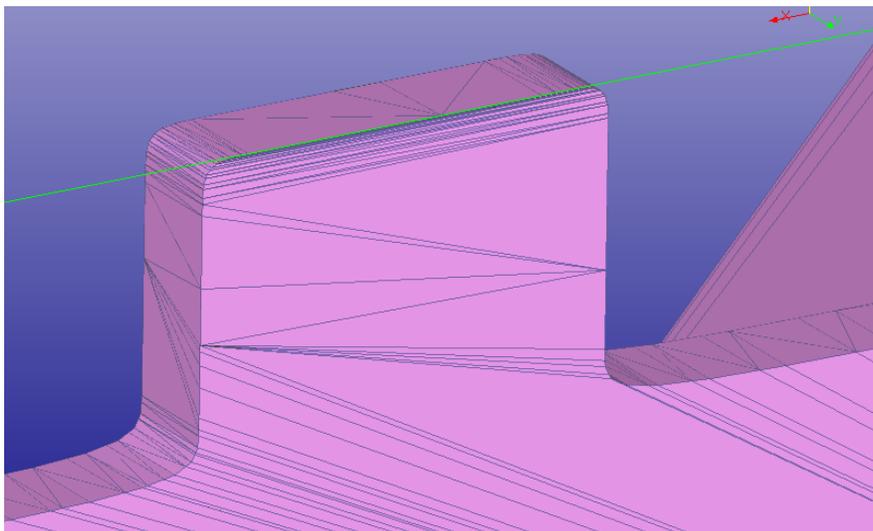
When ready drawing the split line,

- **press *RMB*** to split the model:

³ The curve is actually a Bézier curve.



If you now display the selected half of the model and zoom closer to the tooth, you can see that there is a small rounding to enable more accurate building of the part without cracks at the sharp corners:



Adding Lap Joint

You are now ready to start generating the Lap Joint.

- Give the **Modify Faceted > Lap Joint** command.

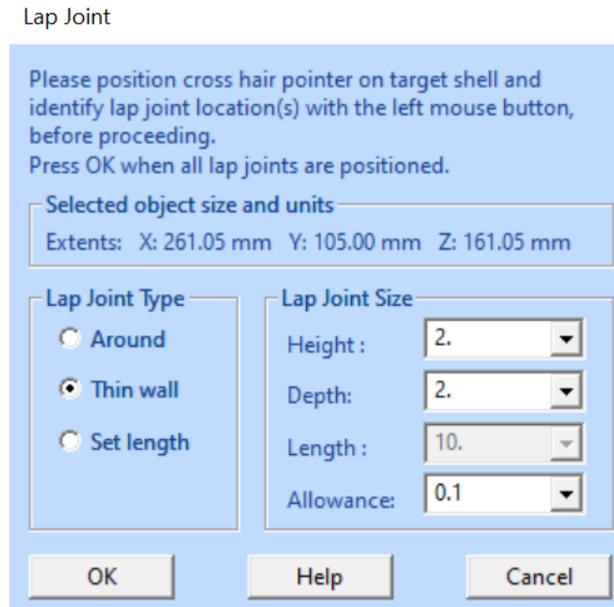
You can also start it by pressing the *Lap Joint icon* in the Tools Window Modify tab:



Let's now fill in the parameters for the Lap Joint command. It is important to use parameters suitable for the split part. The Lap Joint *Depth* (2 mm) must not be too deep when compared to the wall thickness (about half) and *Height* (2 mm) should

be suitable to create a strong bond between the two halves after the build. There are also three different *Lap Joint Types* available, this time we will

- use **Thin wall type** to create the lap joint structure just on one side of the part.
- **Fill in the parameters** seen below:



The *Allowance 0.1 mm* will leave a small void between the lap joint halves for the parts to fit easily and to allow for adding adhesive between the lap joint halves.

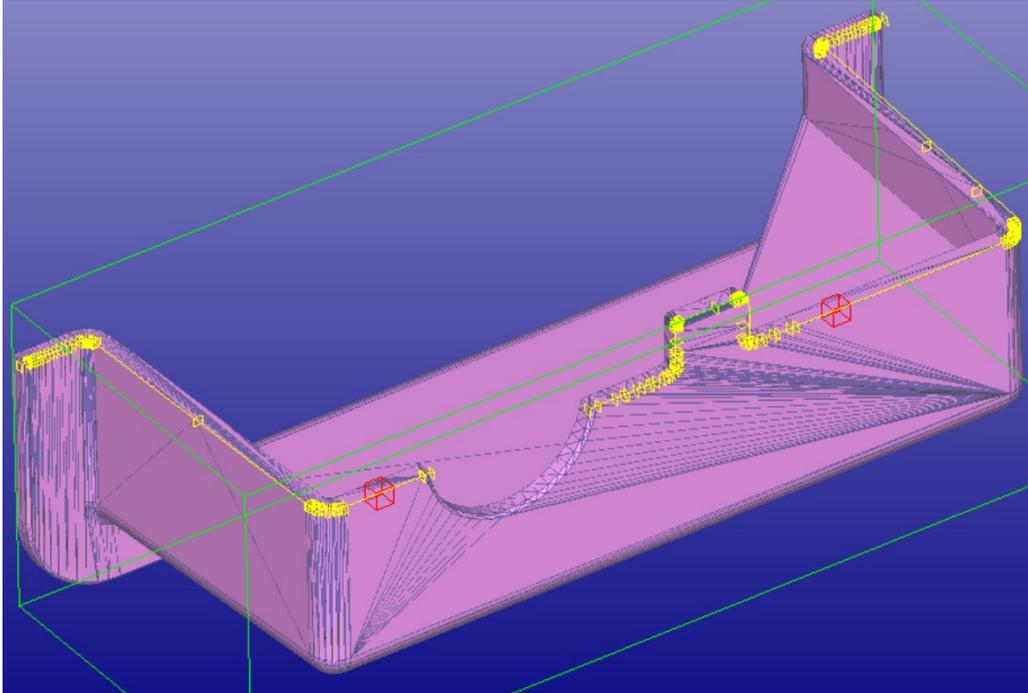
You are now ready to select the Lap Joint locations. Locations must be selected along the split surface edge. When a lap joint location is selected, a red indicator box appears on the edge (see picture below) and the lap joint area is shown with yellow indicator boxes. The yellow indicator boxes allow you to check that the lap joint is correctly located as well as that the *Depth* and *Height* are reasonable. Also, if the yellow boxes intersect each other, the generated lap joint may not be valid. To redefine a lap joint, click the red indicator box and the lap joint disappears. Modify the parameters or press another point to change the location, if necessary.

You can use *mouse wheel zoom* and *middle mouse rotation* during the Lap Joint command to visually check the lap joint area.

As in the picture below

- **select two locations** indicated with red boxes.

The yellow boxes show the size and location of the corresponding lap joint structures:



When ready defining the lap joints,

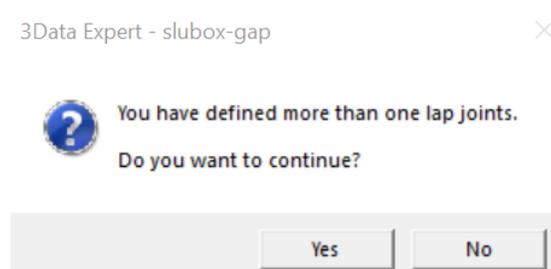
- **press OK to create them.**

The lap joint structure is automatically deleted from the displayed part and added to the other split half (thus you must not move the split parts before adding lap joints between them).

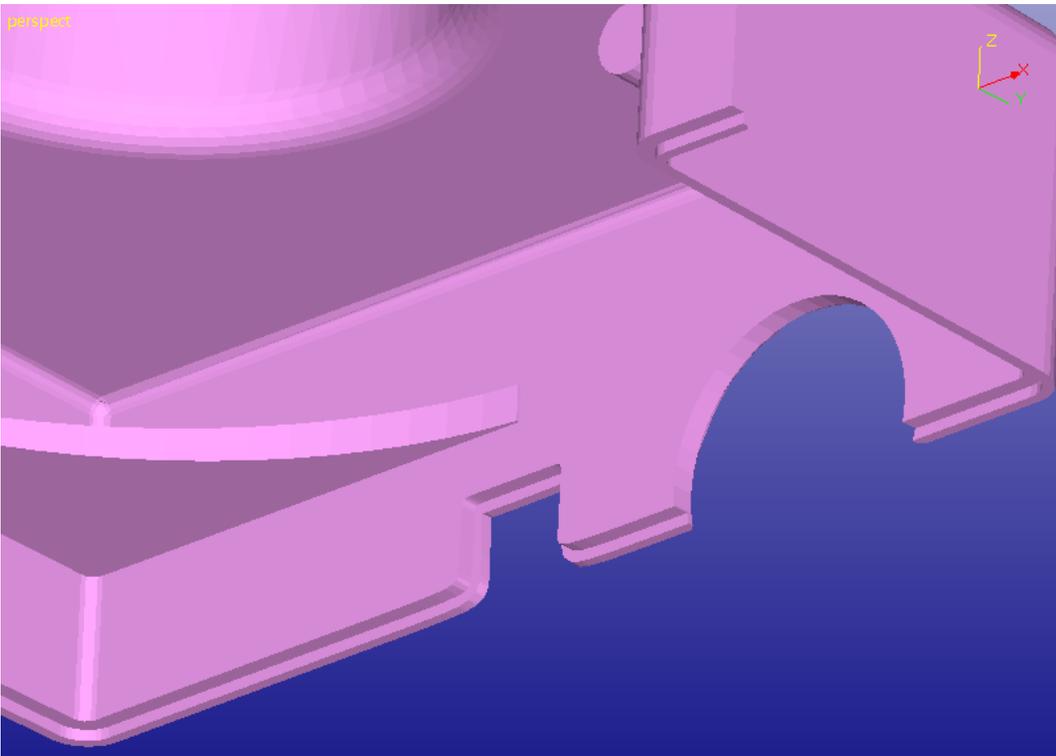
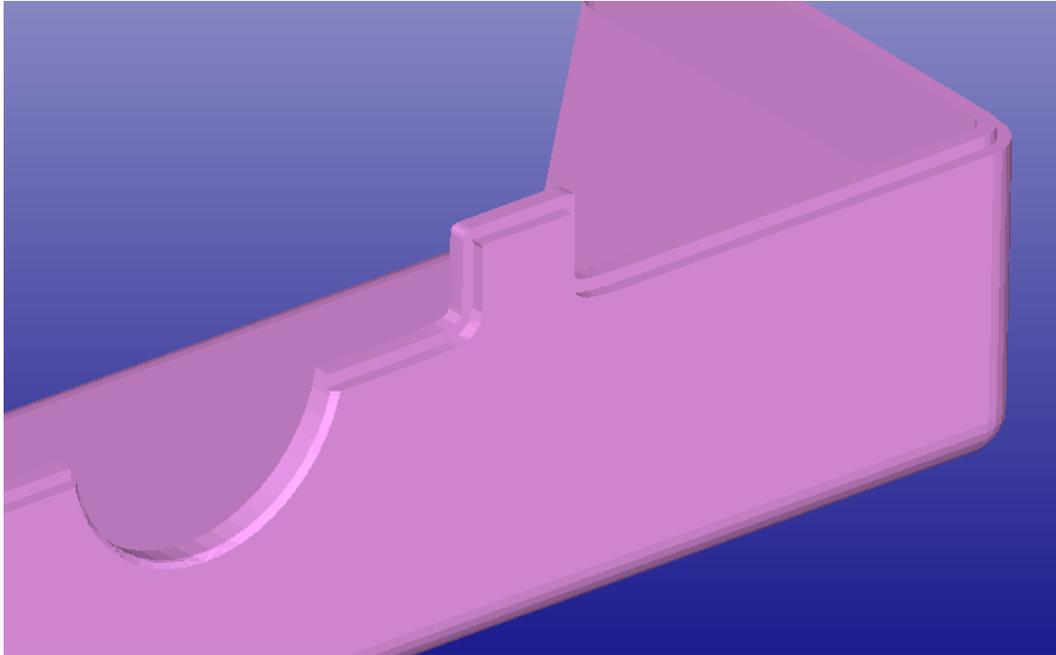
Note also, that the lap joint positions (red boxes) are selected at the edges of well behaving triangles (not thin), this will ensure the best possible result with the lap joint generation.

After pressing OK, the software checks if more than one lap joints should be generated.

- **Press Yes** to generate both areas.



After a while you will see the result, a lap joint structure deleted from the other part and added to the other half:



The parts can now be outputted and built separately and then accurately joined and glued back together.

Lesson 5 – Positioning Parts

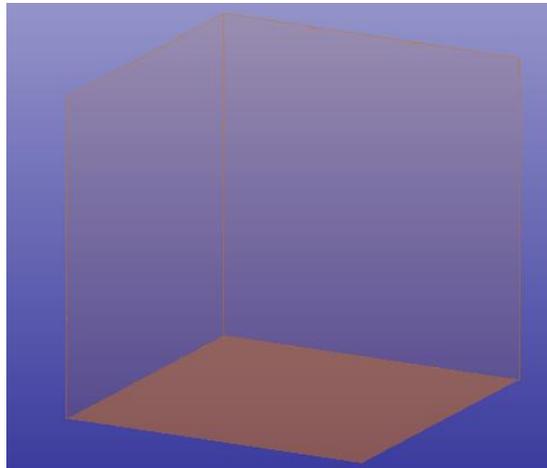
This lesson shows you how to use the 3Data Expert commands to position parts before transferring to Additive Manufacturing machines. The *Platform* definition and *Move to Platform* commands are accessed through the *Print Preparation* menu.

This lesson is not applicable to 3Data Expert Free, where part positioning must be done manually. 

Note: Most 3D Printing systems will reposition the part when inputted into the 3D Printer control software.

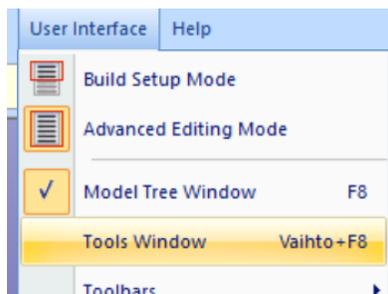
Print Configuration

When you start 3Data Expert, you will see a workspace⁴:



When 3Data Expert is started the first time, a default Print Configuration (*GenericDLP*) is loaded, along with an associated workspace. We will now show how to load another configuration set saved during the 3Data Expert installation. Note: this lesson is only meant to be a guideline, you may have a suitable Print configuration pre-installed and loaded.

First, let's make the *Tools Window* visible via **User Interface > Tools Window** (or Shift + F8).



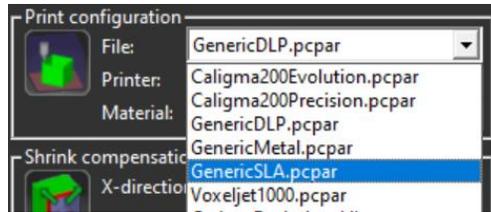
⁴ If the workspace is not shown, click the  button to make it visible

3Data Expert 14.1: Tutorial 1 - Working with 3D Models

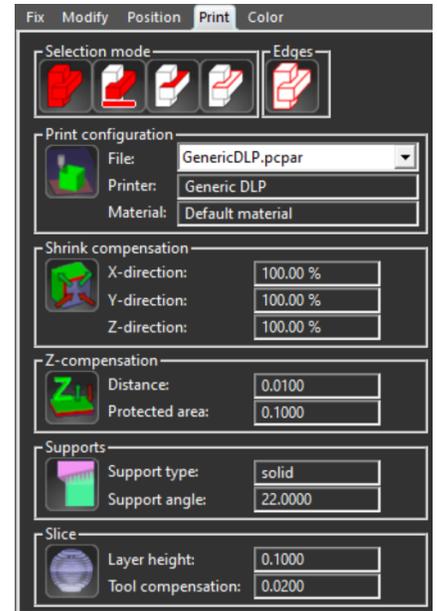
Then choose the **Print** tab. The *Print Configuration* file interface is displayed as its first group.

Below the Print configuration group, the most often used parameters (which are often modified during print preparation) are shown.

From the *File* drop-down list, you can choose a configuration from the preset configurations (or a previously used one):



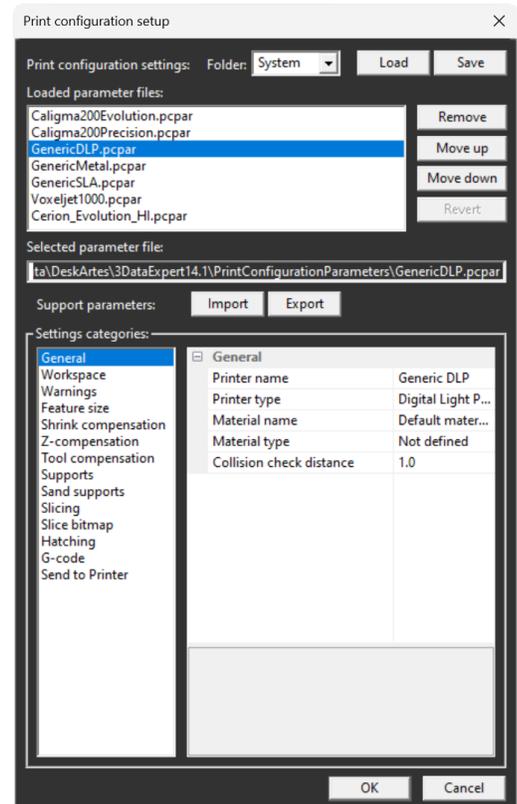
Click the  button now to get a closer look at how to choose and tune different Print Configurations.



The Generic DLP parameters are shown in the *Print Configuration* and shown in the dialog. You will see the *Print Configuration Setup* interface (picture on the right). On top, there is the Print configuration settings file management area. Below it, you can see the parameter categories and category-specific parameter values.

From the settings file management area, you can:

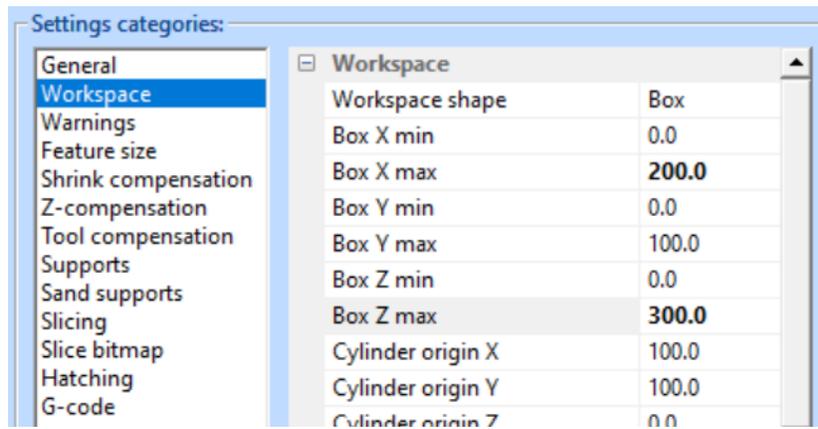
- Load settings from a parameter file (for example, to open a Print Configuration tuned to better match your 3D printing requirements).
- Save current settings into a parameter file.
- Change the order in which Print Configurations appear in the drop-down menu (useful if you have many of them).
- Select where to browse the parameter files, either system or personal folders
- Import and export support parameter files (a subset of the setup) for use outside the *Print Configuration* context.



The parameters themselves are divided into several categories. For simplicity, let's only look at the workspace size. **Click the "Workspace" category.** The type and

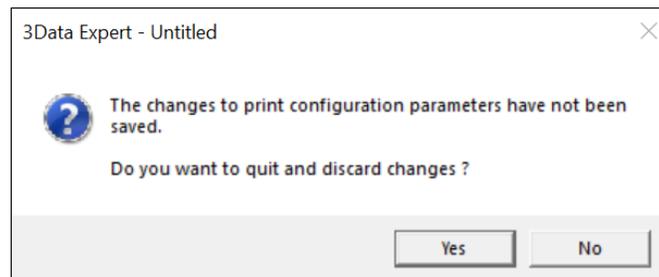
3Data Expert 14.1: Tutorial 1 - Working with 3D Models

size of the workspace are displayed, as well as additional parameters required for a specific printer. Change the workspace box dimensions: X_{max} to 200.0 and Z_{max} to 300.0:

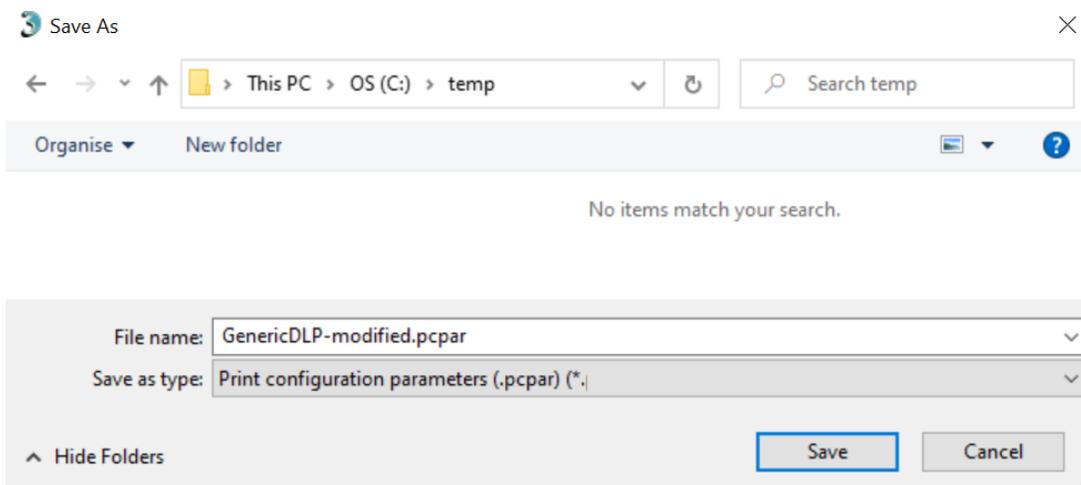


Then **click OK** to close the Print configuration parameter edit window and start using the new parameter set.

Let's now try to exit from 3Data Expert. A warning is shown that the print configuration has been changed. **Click No** in the message window, because we want to save our modified Generic DLP parameter set for later use.



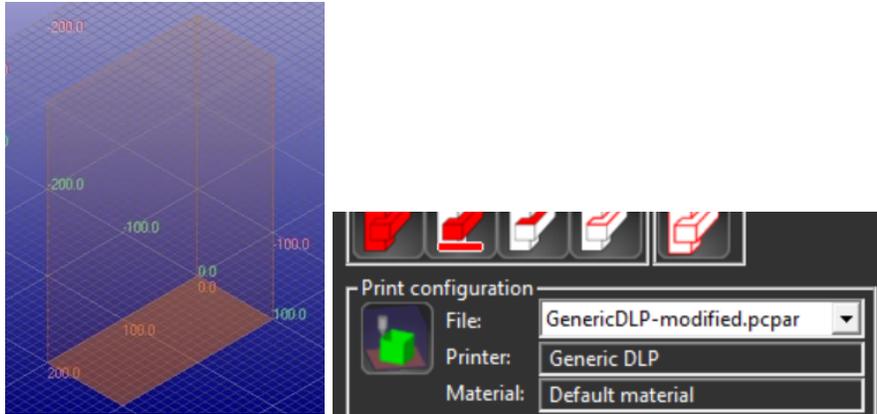
Now, **re-open** the **Print configuration setup** window with the  button, click **Save** to select where and with which name to save the parameter set:



3Data Expert 14.1: Tutorial 1 - Working with 3D Models

In the file dialog, go to the folder where you want to save the print configuration setup, type the new parameter file name, and finally click Save.

Now you can quit 3Data Expert. The next time you start it, the print configuration parameters last used are automatically loaded:

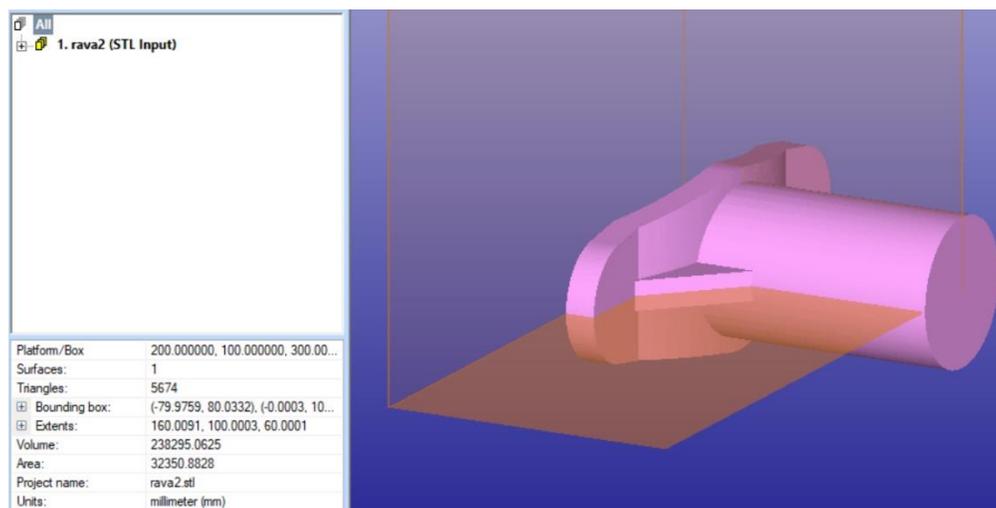


Moving parts into the platform

Now we will see how the *Print Preparation > Move to platform* commands are used for positioning the part into the workspace. The move platform commands will now use the minimum/maximum x,y,z values of this platform when computing the default values for the movement.

Let's first

- **input the model *Tutorials/GeomFiles / rava2.stl* using the *File > Open command***
- **simply close the *Operation Mode* window by pressing its close button.**

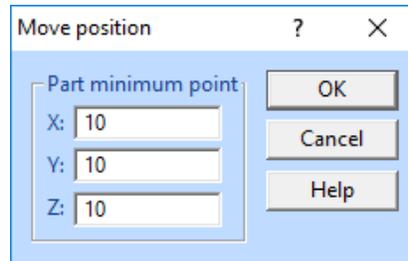


We can see that the part is not fully inside the build area, both visually and from the model properties window at bottom left (platform size, bounding box, extents).

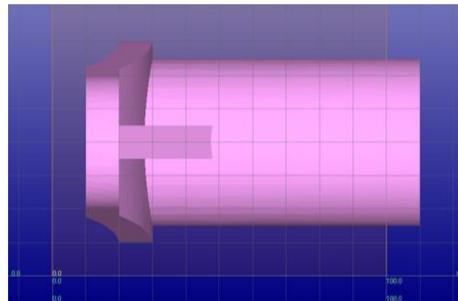
- **Select part *rava2* from the Model Tree**, then
- **move the part into the area with the command *Print Preparation > Move to Platform > Move positive***.

The command suggests the default values which are the same as the minimum bounding box values for the workspace.

- **Fill in the values below and press *OK***.



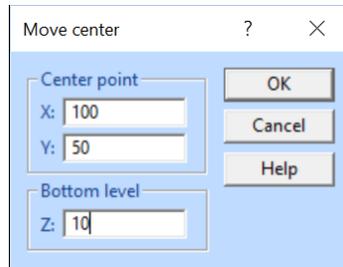
This means that the minimum corner point of the part bounding box is set to the given values. You can check this by turning the grid on  and taking a view from the +X direction:



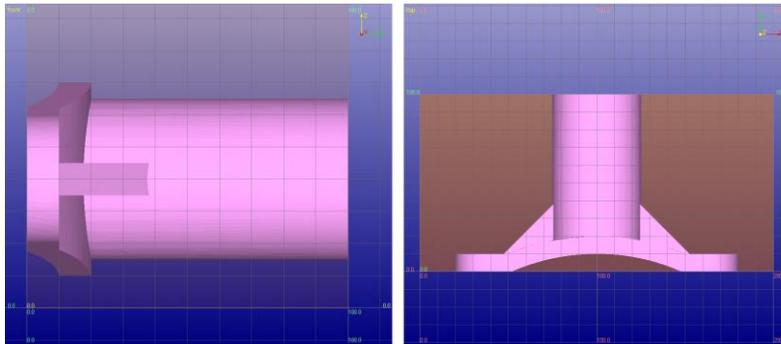
Then **give command *Print Preparation > Move to Platform > Move center*** (or click the  button in the *Position* tab).

Move Center computes the center point of the selected platform and uses this as the default for x and y values. The xy center point of the object is moved to this point. You still must give the z distance from the bottom of the vat (to allow for supports, for example).

- **Fill in the level value below and press *OK***.



You should get the result as view here from X axis (left) and Z axis (right). Note that the model barely fits into the workspace and would require either splitting or scaling:



The *Move to Platform* > *Move Level* command only moves the part in z-axis direction to add sufficient space for supports.

You may want to try the normal transformation commands also. They are accessible from either the *Transform* menu, the *Transform* toolbar or the *Position* tab:



The toolbar includes commands for part moving, scaling, rotation, and mirroring. The Transform menu gives you more commands, like numeric transformations, scale from *MM to Inch* and the other way around. The Position tab contains the most used transformation commands and additional tools from other menus: Duplicate object, Merge shells, and Copy to array, Move to platform center, Move to platform level.

Lesson 6 – Nesting

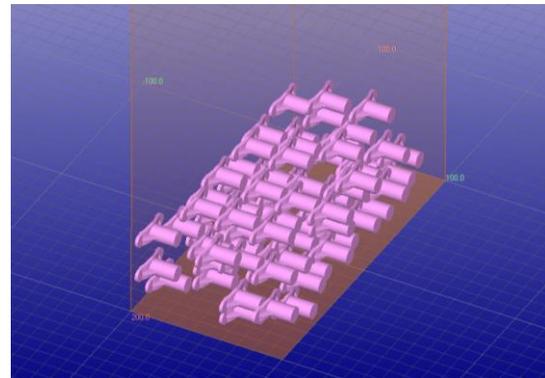
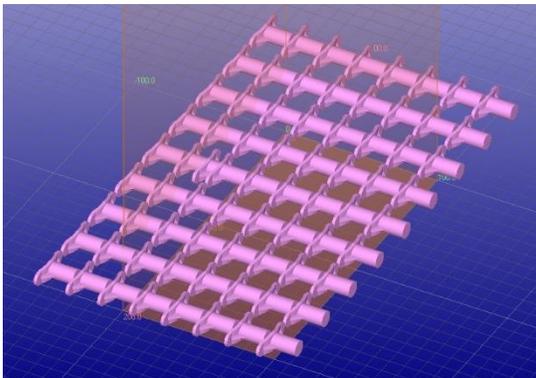
This lesson shows you how to use the 3Data Expert commands to position parts automatically using 3D Nesting. This is done with *Print Preparation > Nesting Tools > Nesting* command. Currently 3D Nesting is available for efficient part merging for powder-based system which do not require supports during the manufacturing.

This lesson is only applicable with the *Base* and *Lite* modules [Base](#) [Lite](#) .

3D Nesting regular components

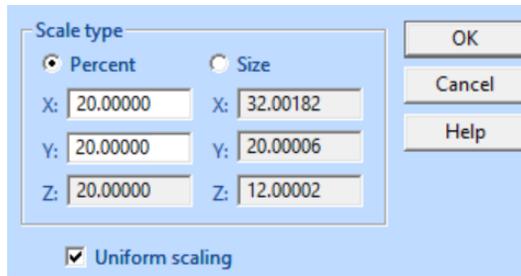
In this example we will pack a set of consistently ordered similar components into the work area, aka *bin* with nesting. The goal is to organize 73 scaled *rava2* parts into the bin as effectively as possible.

In the images below we have 73 original components show in the left-hand image and nested 73 components in the right-hand image:

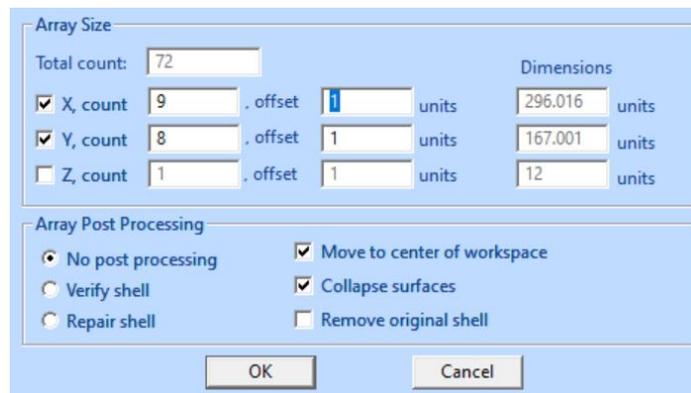


Before nesting we need to generate the 73 components to nest:

- **Input *rava2.stl*** and run **Fix Model** process on it
- **Use *Transform > Scale > Numeric*** to scale it to 20% of the original size



- Create an array of objects with **Edit > Copy to Array** command or, in the Position tab, click the  button

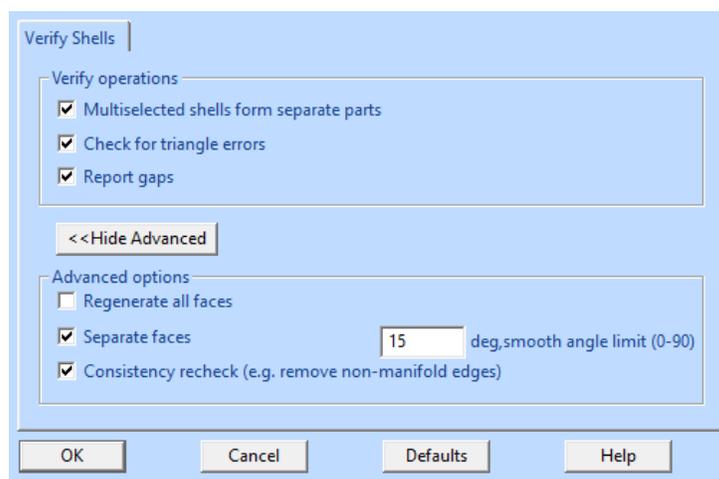


- Click OK. After a while you will see an array of 72 components added to the Model Tree

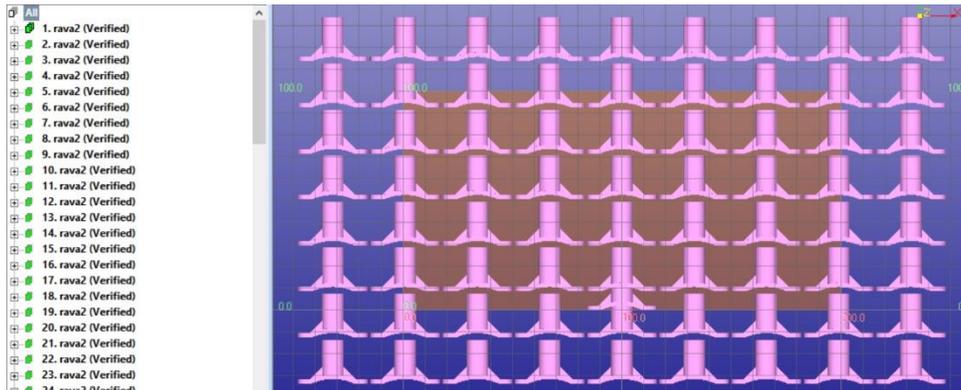


The 72 new copies are in a single shell. For Nesting we must separate each copy into its own shell.

- Select the **"2. Rava2 (Copy Array)" shell** and run **Fix Model > Verify Shells** command on all components to separate the copies



- After a while you will have the original and 72 separate new shells in the Model Tree



We are now ready to start nesting the parts. You may want to read the Help pages for the Nesting command to generally learn about the different parameters available for the Nesting command. In the sequel we will make some specific changes to the default parameter set to get the best possible nesting result for the rava2 array.

We will start the nesting by

- **Select the Model Tree root *All***
- **Give the *Print Preparation > Nesting Tools > Nesting* command and you will see the Nesting parameters dialog.**

Fill in the parameters below, if not already set. Especially note the *Orient algorithm = orig-dir* and *Init algorithm = bin split*. The component directions will not be changed during the nesting and the bin initialization is done by splitting the bin into boxes which fit each component. A one-minute run time is sufficient because most of the nesting is performed by the initialization algorithm (bin split).

Extents: X: 296.02 mm Y: 167.00 mm Z: 12.00 mm

Number of parts:

Part minimum distance:

Nesting max time: h min sec

Minimum density: %

Bin type: Box Cylinder

BBox: X min: X max:
Y min: Y max:
Z min: Z max:

Nesting mode: 2D nesting 3D nesting

<< Hide Advanced

Orient algorithm:

Vib loops:

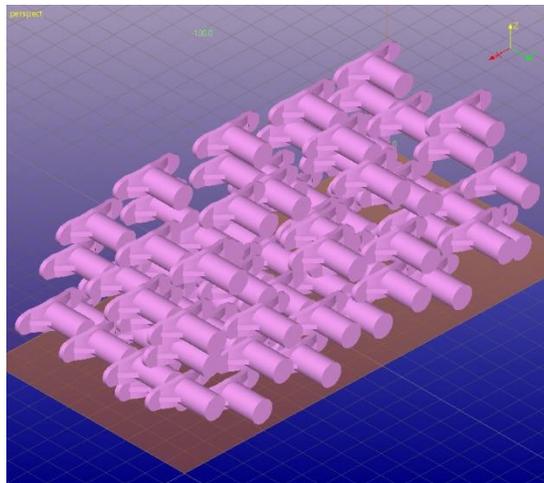
Geom level:

Init algorithm:

Nesting algorithm:

Eval algorithm:

- Press **OK** to start the nesting
- About one minute later you will have the result in the Model Tree:



Generally, it is useful to position the components so that the longest edge is along the longest edge of the bin. If the *Orient algorithm* is set to *z-rot* or *xyz-rot*, the software will try to find the longest component edge and align it to the longest bin edge before the initial bin fill.

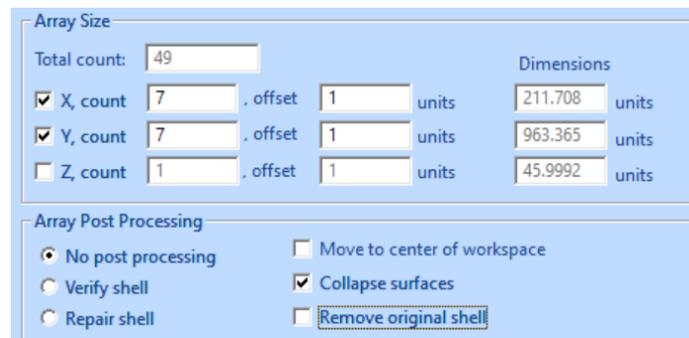
3D Nesting irregular components

When the components are more irregular and/or different sizes, different initialization and orientation algorithms can be used to achieve better results than

simple *z-rot/bin split* algorithms would give. We will give a couple of examples here and also show how the allowed nesting time affects the results.

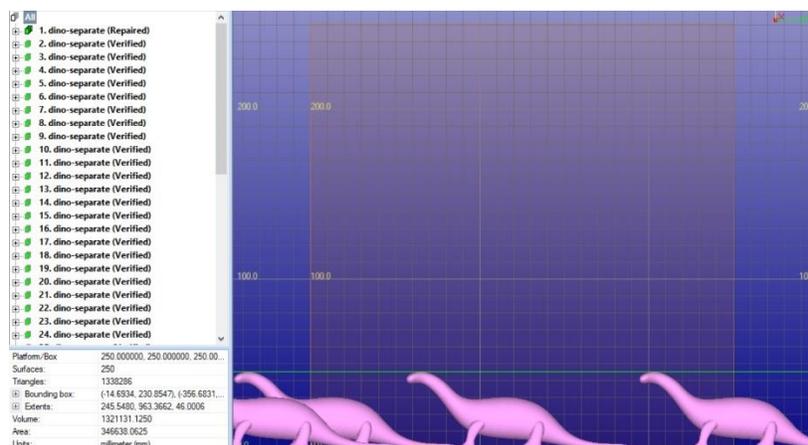
We will first prepare a set of dino components to nest:

- **Input *dino-separate.stl*** and run it through the **Fix Model process**
- **Change the workspace size to 250 * 250 * 250 cube (*Print Preparation > Print Configuration > Workspace*)**
- **Scale the model to 50%** of its original size
- **Create Array of 49 components**



- **Run Verify Shells** to separate the Copy Array result to shells

Now you should have a set of 50 *dino* shells in the Model Tree. For easier comparison during later steps, take a view from the X axis, fit the workspace into the view, with some extra space above:



Next, we will run the shells through the same parameters as was done with the *rava2* example above. We will use a bit longer run time, 1 min 30 sek.

- **Select Model Tree root *All***
- **Give command *Nesting*** with the following parameters (*orig-dir, bin-split*):

Extents: X: 245.55 mm Y: 963.37 mm Z: 46.00 mm

Number of parts:

Part minimum distance:

Nesting max time: h min sec

Minimum density: %

Bin type: Box Cylinder

BBox: X min: X max:
Y min: Y max:
Z min: Z max:

Nesting mode: 2D nesting 3D nesting

<< Hide Advanced

Orient algorithm:

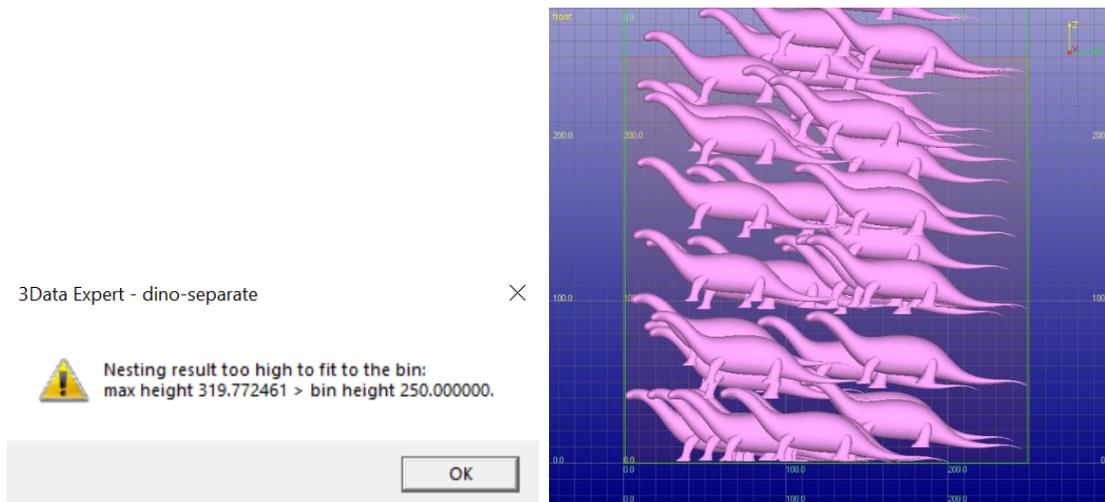
Vib loops:

Geom level:

Init algorithm:

Nesting algorithm:

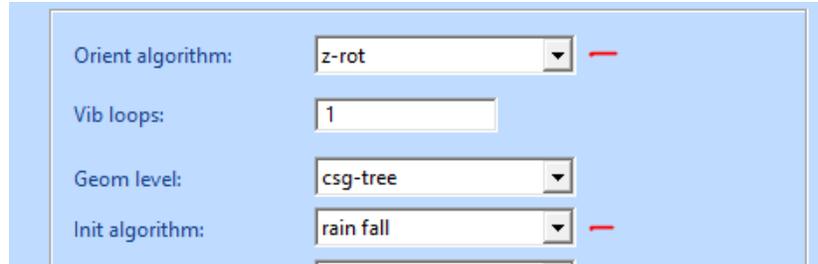
- **Press OK** to start nesting
- After a while the result is available in the Model Tree:



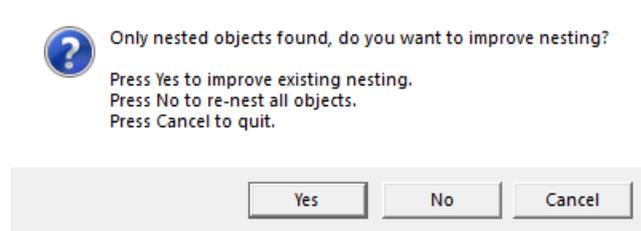
This time the shells did not quite fit into the workspace. Anyhow, with more dense merging the shells should fit there.

Let's try improving this nesting with the *z-rot* and *rain-fall* methods:

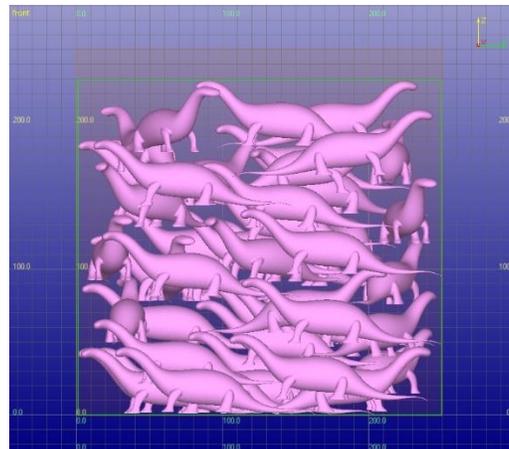
- **Start the Nesting** command again with the following changed parameters:
Orient algorithm = z-rot and *Init algorithm = rain fall*.



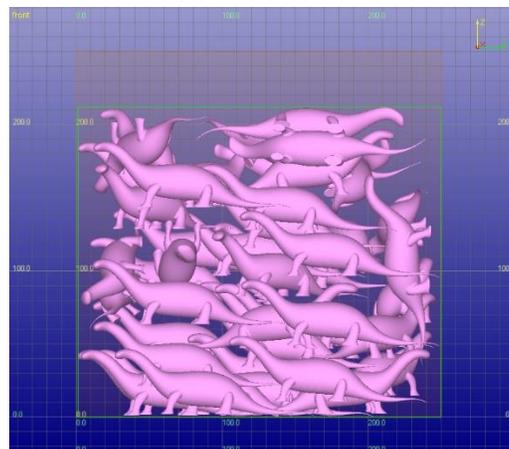
- **Answer Yes** to the question asked after the **OK** is pressed. This will continue with the existing nesting and improve it using the nesting algorithm:



This time the nesting should be lower or even fit into the bin. In the image below is one example of the new nesting result:

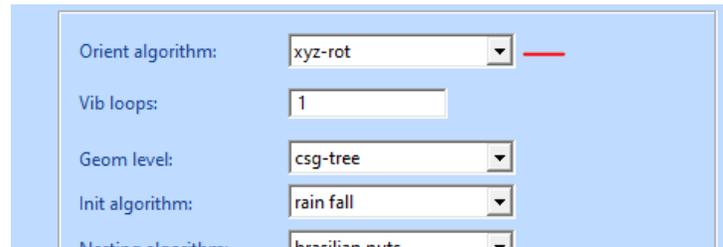


After a couple of improvement runs, we will get a more compact nesting:

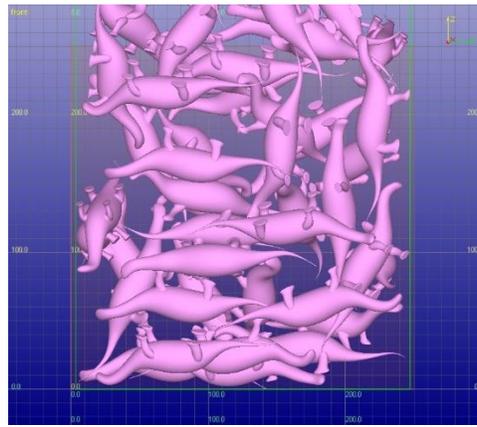


3Data Expert 14.1: Tutorial 1 - Working with 3D Models

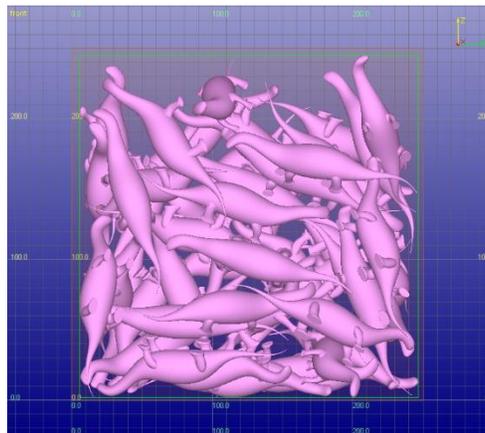
You can still try another *Orient algorithm*, *xyz-rot* (marked with red line below), where the shells can be rotated freely during the nesting.



The result will be similar in height than the previous example, and can be improved by rerunning the command on the result.



After a couple of iterations, some of them using *z-rot*, some *xyz-rotz*, we may achieve a result like this:



Above we were using 1 min 30 sec run time. With some model compositions using a longer run time, like 30 minutes, yields the best result. However, a better packing can be often be achieved using shorter run times and alternating between *z-rot* and *xyz-rot* parameter settings.

Lesson 7 – Adding 3D Text

Sometimes it is necessary to add decorative texts or tag parts with individual texts to tell them apart after the build. The 3D Text and Boolean commands make marking parts easy and effective.

This lesson applies to *Base*, *Lite* and *Color* modules only. 

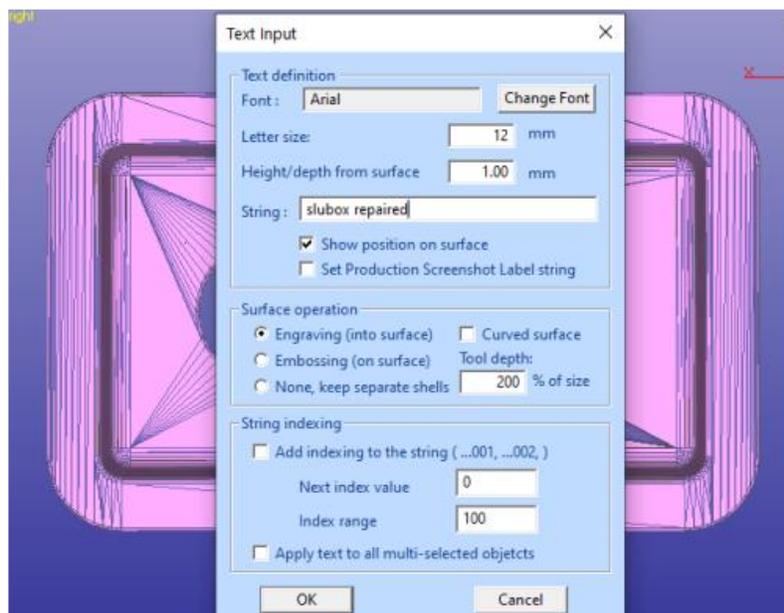
Creating 3D Text

Normally you want to add the 3D text on a part to be built. To start the lesson, first input and repair slubox-gap.stl: we will first input the STL file and repair it for adding text:

1. **Input file slubox-gap.stl** file with File > Open.
2. Accept millimeters as units.
3. Press **Fix Model** with Operations Toolbar to **go to the Auto Repair** process.
4. When asked “Do you want to run Auto Repair” **press Yes**.
5. And after a while you will get “Congratulations, the model is correct and can be saved and printed” message.
6. **Press Close button** to close the status message, as well as all other similar message windows.

You should now have the slubox model fully repaired for adding 3D text. Take a

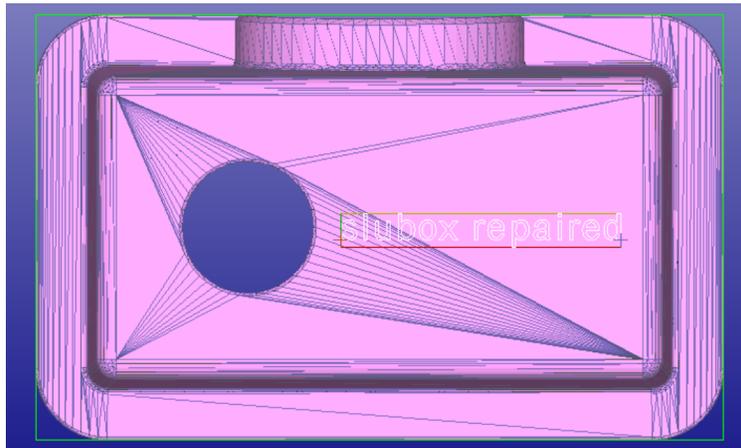
- **view from Y axis direction and**
- **give the command *Create Geometry > Create 3D Text***:



Fill in the values given in the dialog above: *Size 12 mm* (vertical height), *Height 1.2 mm* (=text height from the surface) and finally *String "slubox repaired"*. The font is preset to *Arial*, well use that. *Show position on surface* is set. When we

- **select *Surface operation Engraving***.
- make sure ***Show position on surface*** is set.
- **press OK** the program asks us to show a line on the surface along which the text will be generated.
- You can now **position the text with LMB** on the correct location.

When positioning the text just make sure the whole text fits on the flat surface area on *slubox*:

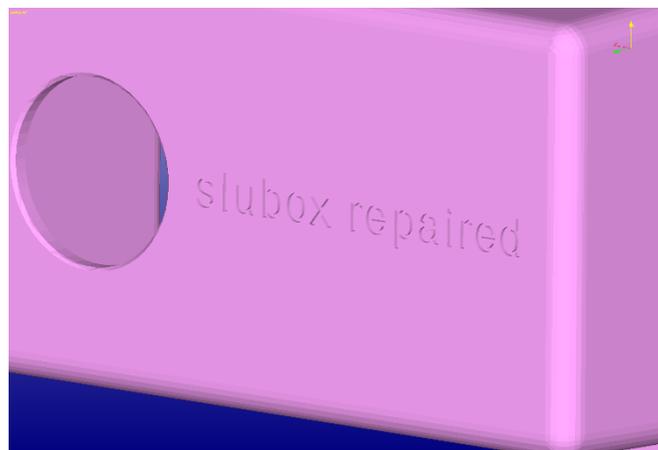


If necessary, you can scale and rotate the text on the graphics area to better fit it on the *slubox* model. The yellow Help text window on top of the graphics area shows the different text positioning options available:

 3D Text positioning: LEFT: move, Shift+LEFT | Wheel: scale, Ctrl+LEFT: rotate, RIGHT: done, ESC: cancel positioning

When positioning is done, click RMB to start text engraving.

Soon you will see the text engraved on the *slubox* surface (turn wireframe off and rotate the view to better see the text):



Generally, 3D Text can be attached on both flat and curved surfaces. If the surface is curved, you should use the *Curved surface* option to reliably attach the text on the surface. If not used and the surface is too curved along the given text path, some letters may not be attached to the part or may be fully inside the part.

Adding 3D Text to multiple parts

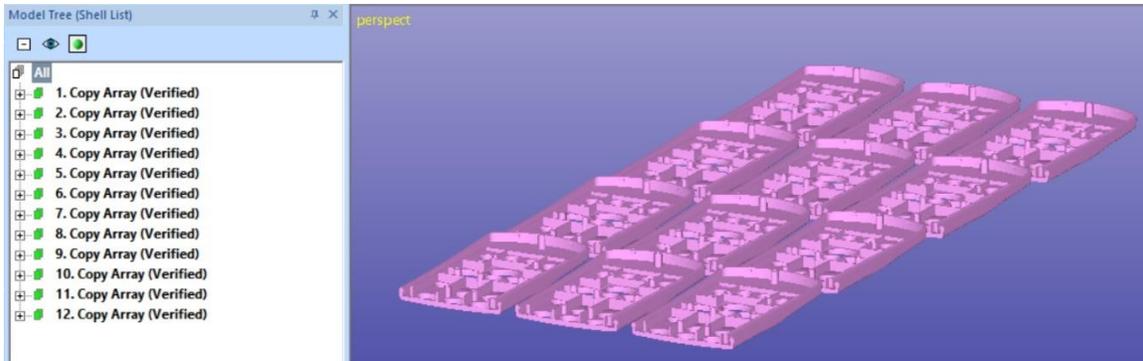
Sometimes it is necessary to add text and label strings to multiple objects at once. For example, quality checking may require that every object on the printing platform can be uniquely identified.

An indexed *Label string* can be added to the shell onto which the 3D text is added (engraved or embossed); this will help component identification, together with the *Production Screenshot* image or the *Generate Report* documentation.

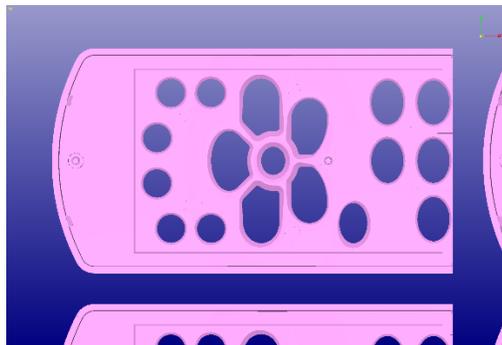
Note: The parts which will be added with the same text must be rather similar. The text position is defined in relation to the bounding box middle point of each component. If the shape of the components is very different, the text in each shell may end up in wrong locations.

For this example, we are using the *12-cellphones.3de* file in the *GeomFiles* directory.

- **Input the *12-cellphones.3de* file** (closing the Operation Mode dialog with the upper right corner X)



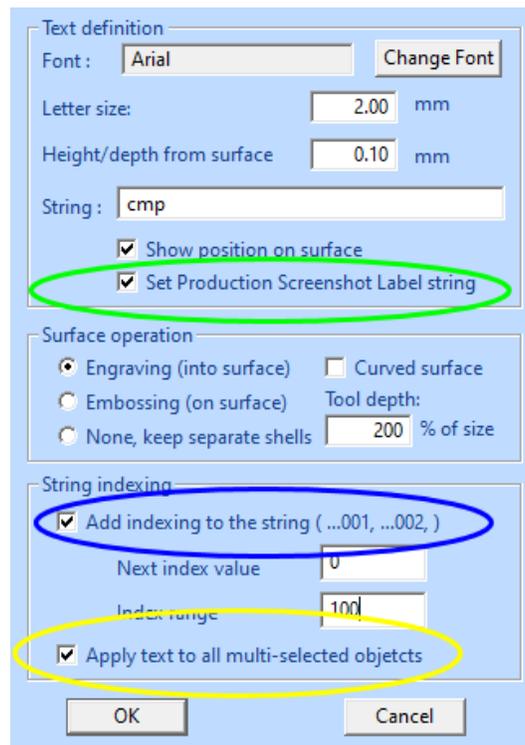
- Take a view from the Z direction
- Zoom into one of the copes to define it as a reference shell:



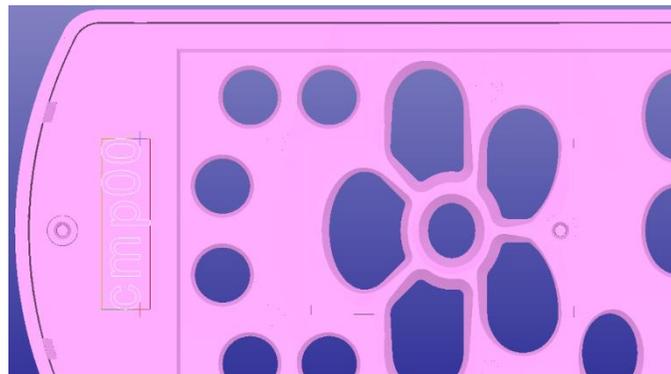
3Data Expert 14.1: Tutorial 1 - Working with 3D Models

The goal is to add an identifying string to each of the components for identification after the build. This is achieved with the 3D Text dialog fields *Set Production Screenshot Label string*, *Apply text to all multi-selected objects* and *Add indexing to the string*.

- Keep Model Tree root **All selected**
- **Give** the command **Create Geometry > Create 3D Text**
- The **Text Input** dialog appears. Make sure all the values are as seen in the image next page: *Letter size*, *Height*, *Show Position...*, *Set Production ...*, *Engraving*, *Add index...*, *Apply text to...* etc. The shell is rather thin, so especially the *Height/depth* must be very small.



- **Press OK** to start positioning the text on the reference shell currently in view
- Position the text on top of the reference shell. **Move** the text with **LMB**, **rotate** with **Ctrl+LMB**, **resize** with **Shift+LMB**:



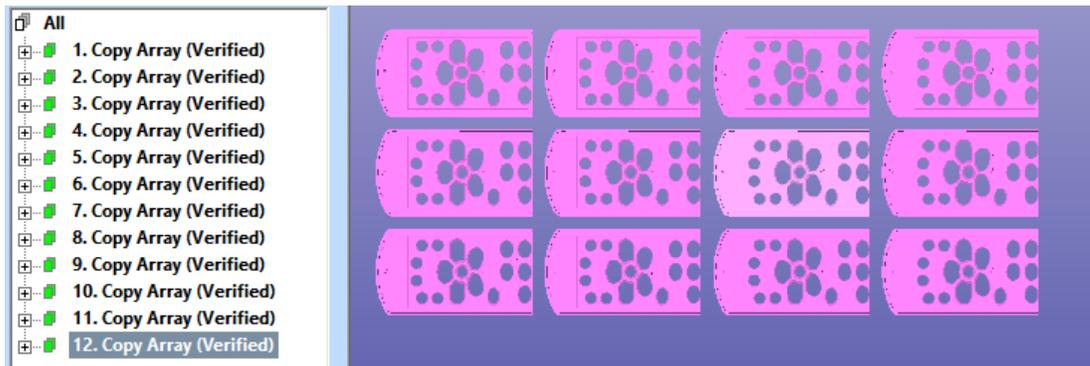
3Data Expert 14.1: Tutorial 1 - Working with 3D Models

When the text is at the correct location on reference to the upper left shell,

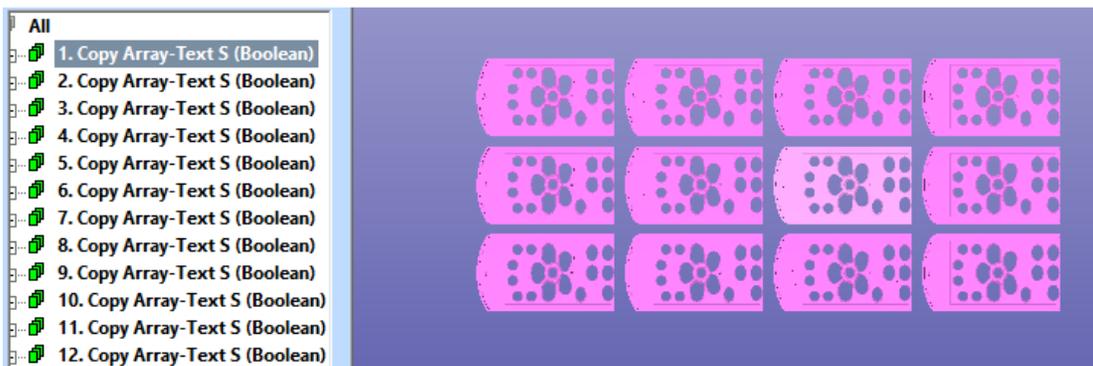
- **Press *RMB*** to add the text to all selected shells
- After a while the result is visible at the Model Tree, each component has the text with increasing index added:



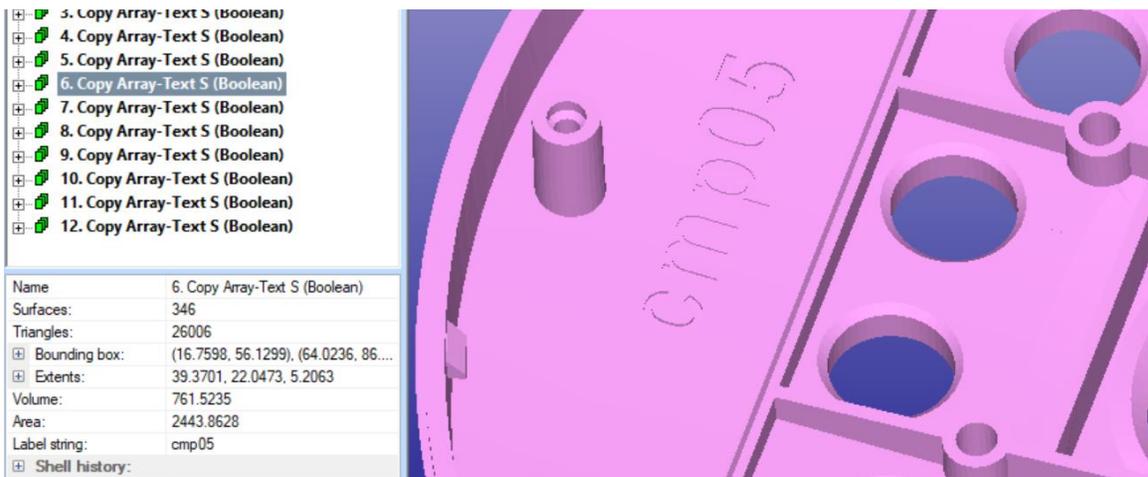
The Text is added starting from the last selected shell in the Model Tree before the command was given and continues towards the shells higher in the Model Tree. Before the command the highlighted shell was the last in the Model Tree and will get the text “cmp00”:



After the command the last shell is the first in the Model Tree with the “cmp00” text:



The *Label string* can be seen in the properties area below the Model Tree or with the *Object Properties* command:



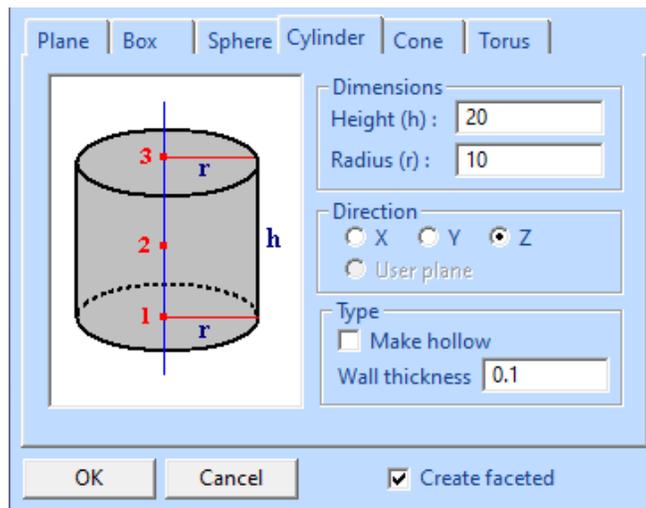
- **Save the resulting model as 12-cellphones-3D-text.3de** for the later lesson on Production Screenshot command.

Adding 3D Text on curved surfaces

3D text can be added on curved surfaces too. However, you need to take care with the parameters to ensure that the 3D text is visible on all desired locations.

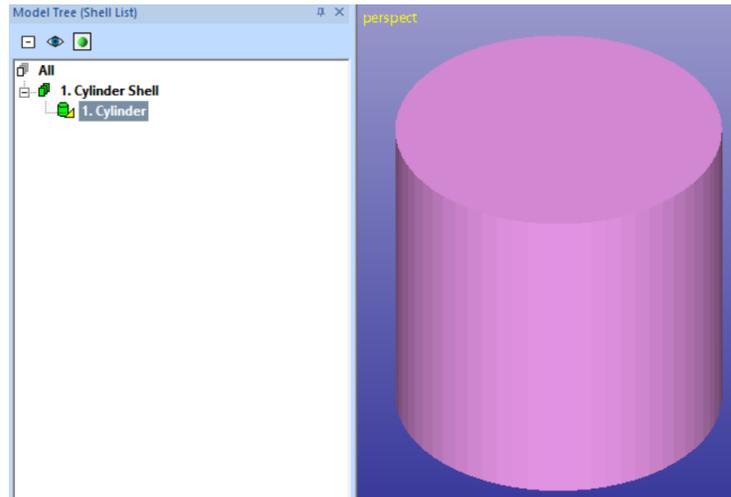
Let's create a primitive cylinder with 20 mm height and 10 mm radius for this lesson.

- **File > Close** the current model to get an empty model
- **Create a cylinder with Create Geometry > Cylinder** command



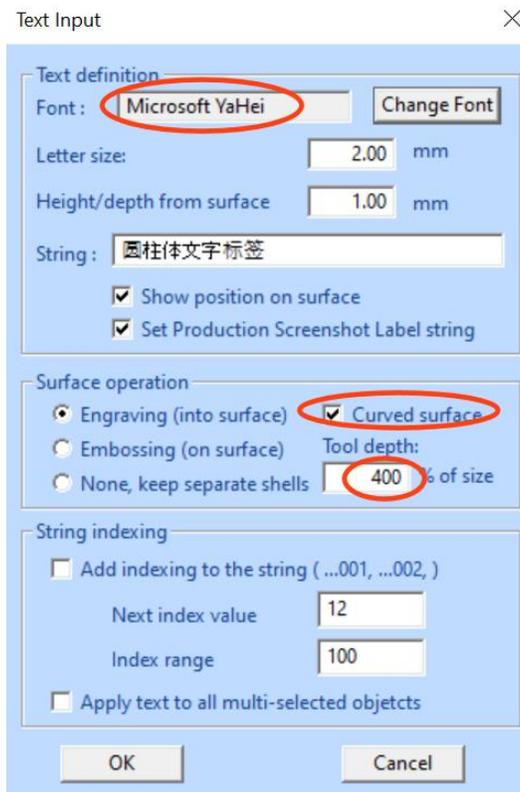
3Data Expert 14.1: Tutorial 1 - Working with 3D Models

- **Press OK** to save the cylinder; you should automatically get a clean cylinder without the need to repair it:

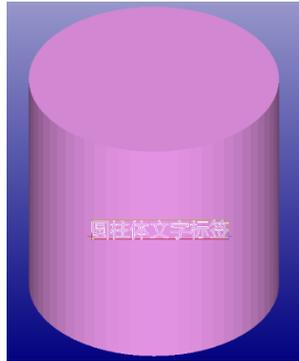


We will now add simple text on top of the curved cylinder vertical side.

- **Start the *Create 3D Text*** command and fill in the following parameters, especially note the *400% Tool depth*. This will use up to 8 mm (4*2mm) high letter primitives to cut the text on the curved surface. Feel free to use the font and text of your preference.



- **Press OK** to show the location on the model



- **Scale the text** (Shift+LMB) to fit on the surface as shown in the above picture.
- You can see a quick glimpse of the temporary engraving tool surfaces (used for the Boolean operation) before the actual result appears:

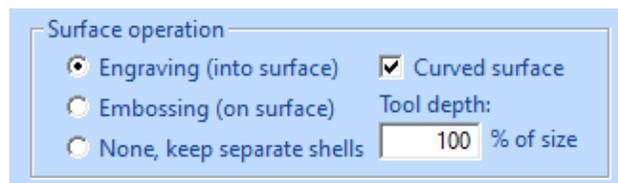


- And finally, the correct engraving:

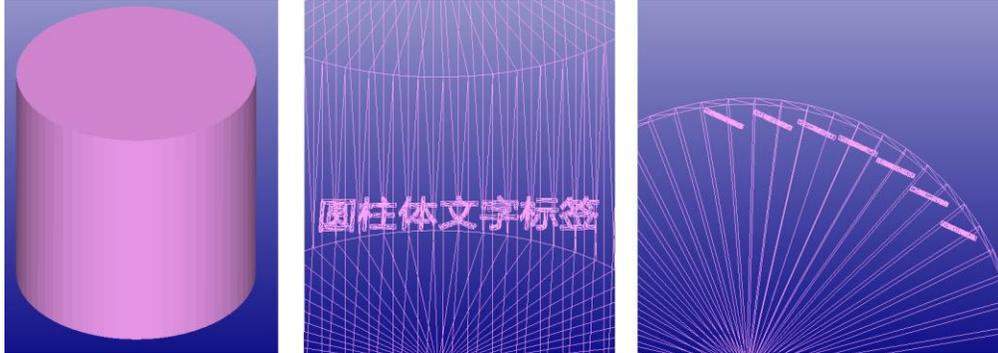


The text is correctly cut against the model. Let's try the operation with a smaller engraving tool height.

- **Undo** the last command
- **Give Create 3D Text** with 100% tool height (try an even smaller percentage if the result is still correct after the Boolean operation)



- **Press OK** to add the text
- The end result is a model where the text is not properly attached to the cylinder model, as can be seen in here (left: shaded, middle: wireframe zoomed, right: wireframe looked from almost Z axis):



So, it is important to use a big enough tool height with highly curved surfaces. Also, to get the most accurate text projection on the surface, the viewing should be as directly towards the average normal of the text insert surface, or anomalies may appear.

Lesson 8 – Generating AM reporting

This lesson shows you how generate different reports for AM production and quality control.

Production Screenshot command

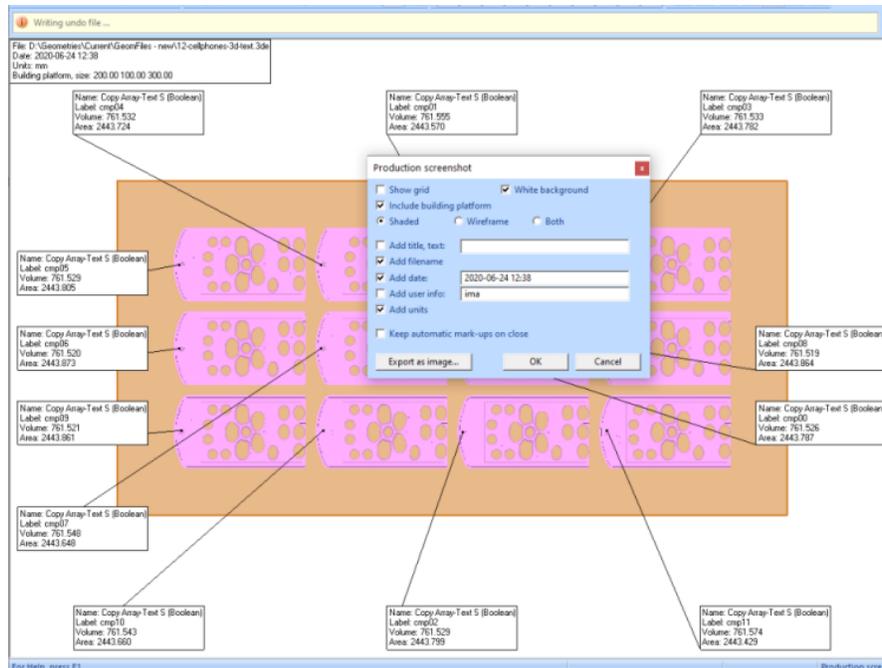
The *Manufacturing* menu in 3Data Expert 14.1 is intended to contain commands which are closer to the ready built AM part. Currently it includes commands to produce documentation for post-processing operations and quality control.

The Production Screenshot command is meant for generating documentation to help identifying the parts on the platform after the build. First the user sets up the suitable part positioning and orientation on the build area for the parts and supports. When ready to go for slicing it is time to produce the Production Screenshot. Here we are using as an example the result from the previous example, 12-cellphones-3d-text.3de file.

- **Input the 12-cellphones-3d-text.3de**
- Skip the Auto Repair process (simply close the Operation Mode dialog)
- **Give the command *Manufacturing > Production Screenshot*.**

You will soon see a view from the positive Z axis with additional yellow info boxes for each component on the platform.

3Data Expert 14.1: Tutorial 1 - Working with 3D Models

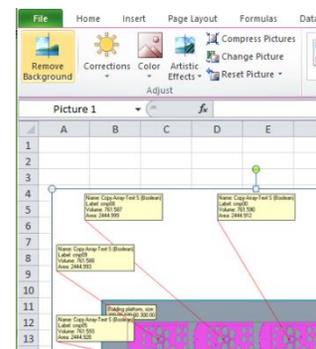


Each info box is connected to the corresponding component with a red line (close to the middle of each component). In this case the *Label strings* created with the 3D Text command are visible in both the info boxes and as engraved into the components. This will make identification of each component very easy after the build. The properties (size) of the building platform are also shown on the upper left corner of the platform; if the platform is defined but not visible, tick the *Include building platform* check box.

To minimize info box cluttering you can rearrange them with LMB. The info box target points can be re-positioned similarly.

The *Export as image* button allows saving the image (as a PNG file) with a desired name on the hard disk. The exported .png file can be inputted or copy/pasted to different Microsoft Office documents (for example) for part identification and commenting after the build.

Clicking OK will store the screenshot into the model for later viewing with Manufacturing > *Show Production Screenshot* (when the model is saved in either .3de or .3dez format).



Generate Report command

The Generate Report command in Manufacturing menu will allow automatic transfer model and geometry related info into MS Office documents for quality control purposes. The transferred data will include model name, volume, surface

3Data Expert 14.1: Tutorial 1 - Working with 3D Models

area, size, dimensions, etc. and can be fully defined by the user. For everyday use, the data transfer is based on predefined templates.

For the prerequisites and a detailed description of report generation, please see the application Help page *Menus and commands > Manufacturing menu > Generate Report*.

Report generation Word example

STL file:	D:\Geometries\Current\GeomFiles - new\12-cellphones.3de
Model name:	cellphone
Volume:	9141,3
Area:	28286,6
Vertical area:	15747,6
XY projection area:	7843,6
Extents:	166.3, 72.14, 5.20634
Bounding box:	(18.7597, 13.9290, 9.8998) ... (183.2, 86.07, 15.2064)
Center of gravity:	100.81, 48.147, 5.90970
Model units:	mm

Bounding box as separate coordinates: 18,7597 13,9290 9,8998 183,2 86,07 15,2064

Bounding box as points: 18.7597, 13.9290, 9.8998 183.2, 86.07, 15.2064

#	Label	Bounding box	#triangles
1		(143.9, 13.93, 10.00) ... (183.2, 36.0, 13.21)	23874
2		(59.1, 13.93, 10.00) ...	23874

Generate Report Word output example

Label	Model name	STL file	Area	Volume	Extents	Bounding box	Area/Volume
18 cmp00	1. Copy Array-Text		2443,8	761,5	39.4, 22.0, 5.20(101.5, 39.0, 10.00) ...	(140.9, 61.0, 10.00)	3,205
19 cmp01	2. Copy Array-Text		2443,6	761,6	39.4, 22.0, 5.20(101.5, 64.0, 10.00) ...	(140.9, 86.07, 10.00)	3,205
20 cmp02	3. Copy Array-Text		2443,8	761,5	39.4, 22.0, 5.20(101.5, 13.93, 10.00) ...	(140.9, 13.93, 10.00)	3,205
21 cmp03	4. Copy Array-Text		2443,8	761,5	39.4, 22.0, 5.20(143.9, 64.0, 10.00) ...	(183.2, 86.07, 10.00)	3,205
22 cmp04	5. Copy Array-Text		2443,7	761,5	39.4, 22.0, 5.20(59.1, 64.0, 10.00) ...	(98.5, 86.1, 10.00)	3,205
23 cmp05	6. Copy Array-Text		2443,8	761,5	39.4, 22.0, 5.20(16.76, 64.0, 10.00) ...	(56.1, 86.1, 10.00)	3,205
24 cmp06	7. Copy Array-Text		2443,9	761,5	39.4, 22.0, 5.20(16.76, 39.0, 10.00) ...	(56.1, 61.0, 10.00)	3,205
25 cmp07	8. Copy Array-Text		2443,7	761,5	39.4, 22.0, 5.20(59.1, 39.0, 10.00) ...	(98.5, 61.0, 10.00)	3,205
26 cmp08	9. Copy Array-Text		2443,9	761,5	39.4, 22.0, 5.20(143.9, 39.0, 10.00) ...	(183.2, 61.0, 10.00)	3,205
27 cmp09	10. Copy Array-Text		2443,9	761,5	39.4, 22.0, 5.20(16.76, 13.93, 10.00) ...	(56.1, 36.0, 10.00)	3,205
28 cmp10	11. Copy Array-Text		2443,7	761,5	39.4, 22.0, 5.20(59.1, 13.93, 10.00) ...	(98.5, 36.0, 10.00)	3,205
29 cmp11	12. Copy Array-Text		2443,4	761,6	39.4, 22.0, 5.20(143.9, 13.93, 10.00) ...	(183.2, 13.93, 10.00)	3,205

Generate Report Excel output example

Lesson 9 – Preparing for 3D Printing

There are several issues to consider when sending 3D models to 3D printing. Most processes are more effective when printing bulky models hollowed rather than solid. Also, being able to drain out excess material from the hollow models gives remarkable material savings. Open flat surfaces must be offset to a certain minimum thickness and too thin walls require material addition to print properly. Some processes require external supports to build correct models.

3Data Expert offers effective tools for preparing 3D models for optimal print results through its *Modify Faceted* and *Print Preparation* menus. For this and following lessons 3Data Expert software is mostly required.

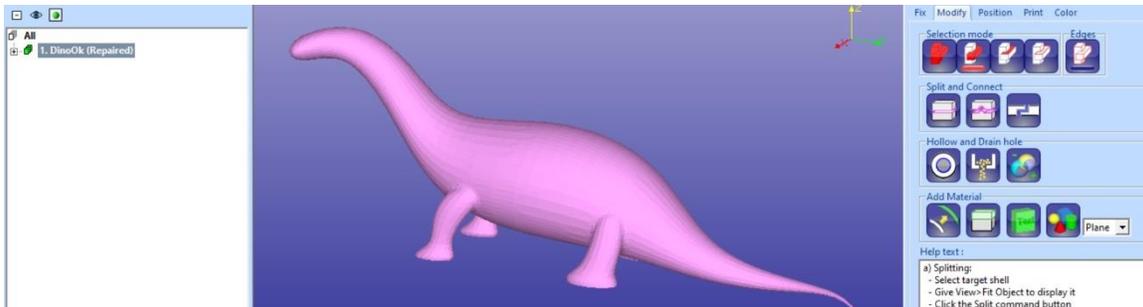
This lesson is only applicable with the *Base*, *Lite* and *Color* modules. 

Making Hollow models

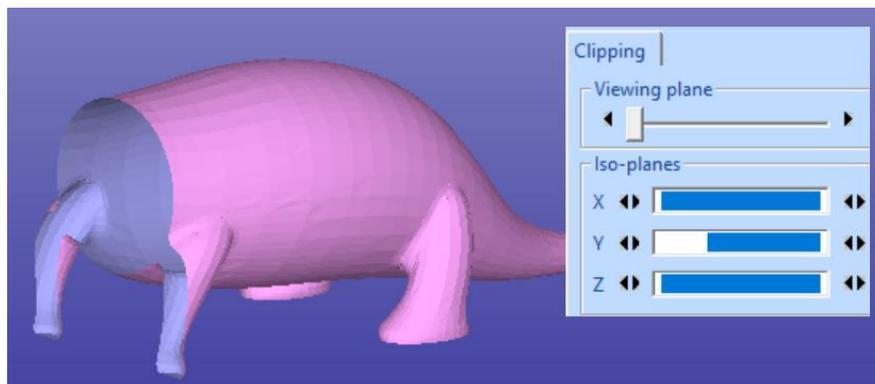
Building large and bulky models may require a long time to print or they consume large amounts of material. If these models are made hollow, the processing time, as well as the material usage, will be minimized.

Start this exercise by

- emptying the 3Data Expert work space with **File > New** and then
 - input the model *Tutorials / GeomFiles / DinoOk.3de*.
- **Go directly to the Modify Mode**



Enable clipping from negative Y direction to see the inside of the dino:



The dino model is a bulky model suitable for hollowing.

- Give the command **Modify Faceted > Hollow** or use the



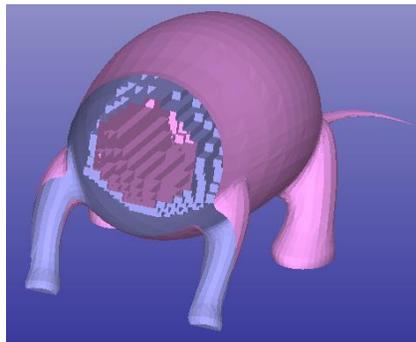
shortcut in the Modify tab. When the command is given the following parameter dialog appears (change the values to those shown):

A screenshot of the 'Hollow' parameter dialog box. The dialog has a light blue background and a title bar that says 'Hollow'. It contains several fields and controls: a text box for 'Selected object size and units' showing 'Extents: X: 58.77 mm Y: 273.53 mm Z: 91.51 mm'; a 'Minimum wall thickness' dropdown menu set to '4'; a '<<Hide Advanced' button; a checkbox for 'use default (1/20 of max ext)' which is unchecked; an 'Accuracy' dropdown menu set to 'default'; a 'Direction' section with two radio buttons, 'Inside' (checked) and 'Outside' (unchecked); and four buttons at the bottom: 'OK', 'Cancel', 'Defaults', and 'Help'.

The dialog gives the extents of the selected part on the *Selected object size and units* group. The default *Minimum wall thickness* is calculated based on the maximum extent of the selected object (13.67 mm in this case). The default value is too large for this model: change the *Minimum wall thickness* value to 4 mm as seen in the image above. According to the default *Advanced* settings we will make the hollowing surface *Inside* the Dino model and use the default *Accuracy*.

- **Fill in the parameters above** and
- **Press OK** to start the hollowing.

After a while the Hollowing is ready. To see the actual result, you can use *Clipping* to look inside the model. As seen in the image below, the model is now made hollow by adding a new surface inside the original surface.



The hollowing is done by filling the inner void of the model with empty boxes. The size of the boxes is defined by the *Minimum wall thickness* and *Accuracy* settings in the parameter dialog. The inner surface triangles are generated where the distance of an empty box wall from the original surface is at least the given

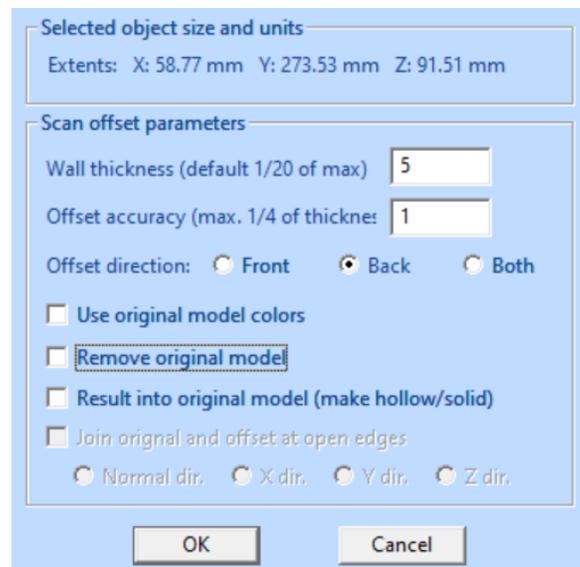
Minimum wall thickness. This guarantees that the wall thickness is at least the given value.

The *Hollowing* command can only be used with solid models. If you wish to make an open surface a solid you should use the *Modify Faceted > Offset* command.

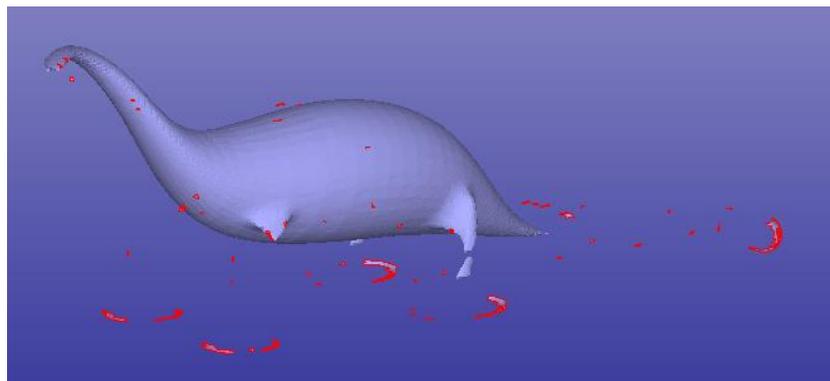
Hollowing with Scan Offset

The basic hollowing method described above removes a voxel-type volume from inside the model. We will next use another hollowing method which produces a smoother inside cavity, although with more steps and calculation time: the *Scan Offset* method.

Let's again start with the same Dino model. After reading the model, select shell *DinoOk (Repaired)*, then start command ***Modify Faceted > Scan Offset***. A parameter dialog appears. In the dialog, set *Wall thickness* to 5.0 and *Offset accuracy* to 1.0, *Offset direction* to "Back", uncheck all the checkboxes, and then click *OK*.

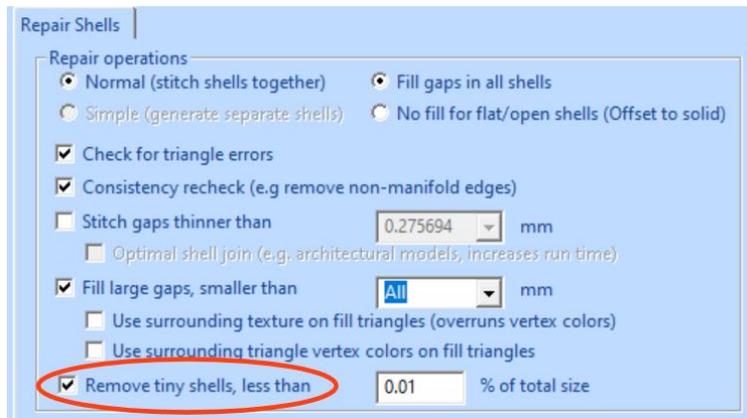


After a while, the scan offset result appears. It contains errors which need fixing:

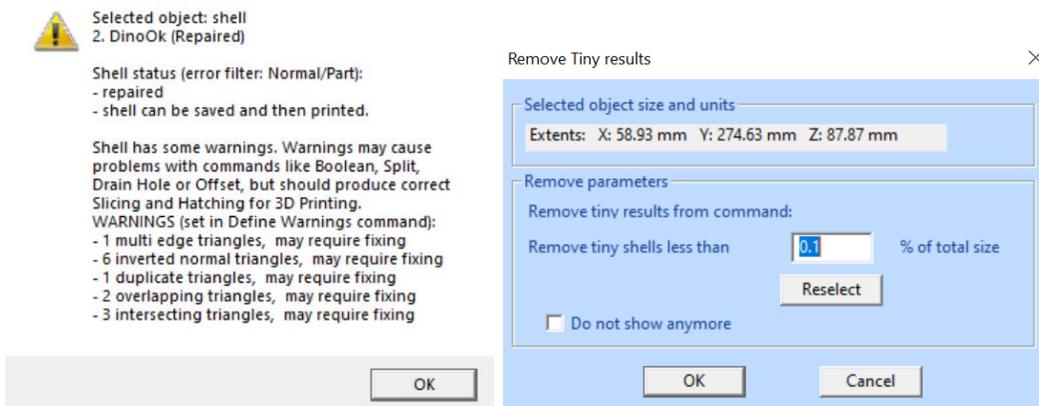


3Data Expert 14.1: Tutorial 1 - Working with 3D Models

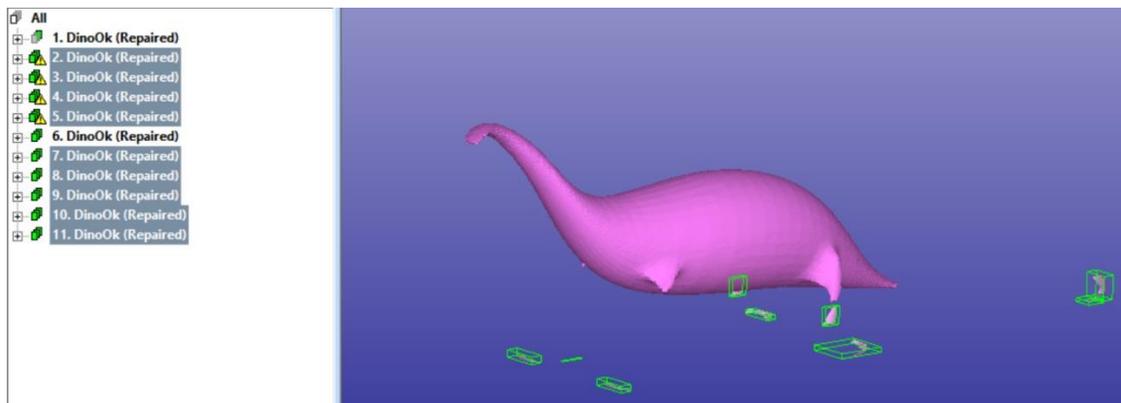
We will first run the *Fix Model > Repair Shells* command on the *DinoOk (Solid)* shell, with the following parameters (in particular, *Remove tiny shells* should be checked):



After the repair, statistics will be shown (click OK), then the *Remove Tiny Results* dialog (use the values suggested and click OK):

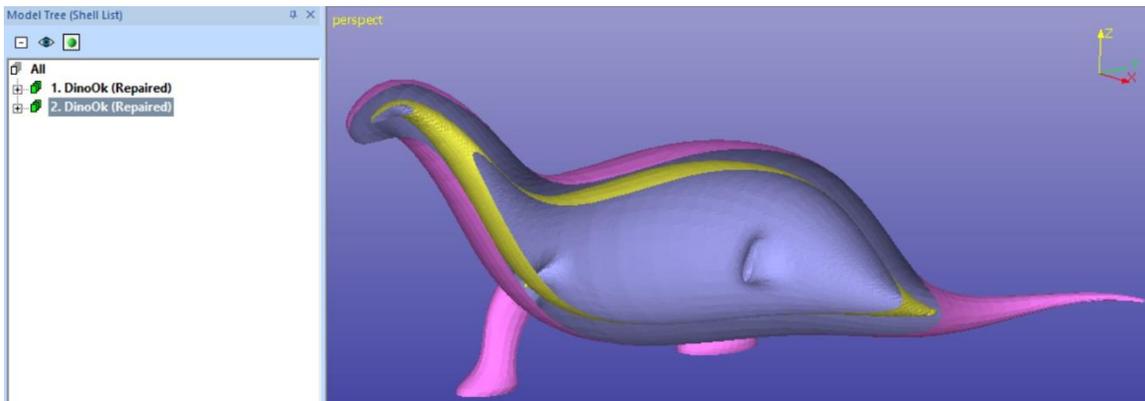


Even if most undesired small details are removed, some remain and must be removed manually. From the Model Tree, select all other shells than the first one (the original Dino model), the graphically unselect the biggest *Scan Offset* result part (Ctrl + pick the part; showing bounding boxes clearly shows what has been selected):

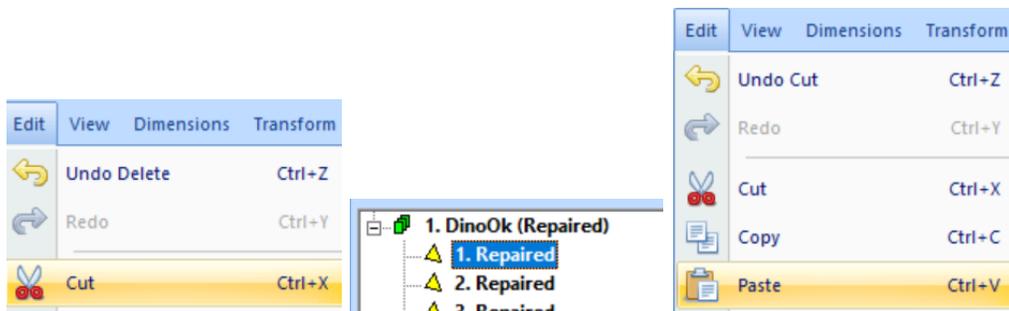


3Data Expert 14.1: Tutorial 1 - Working with 3D Models

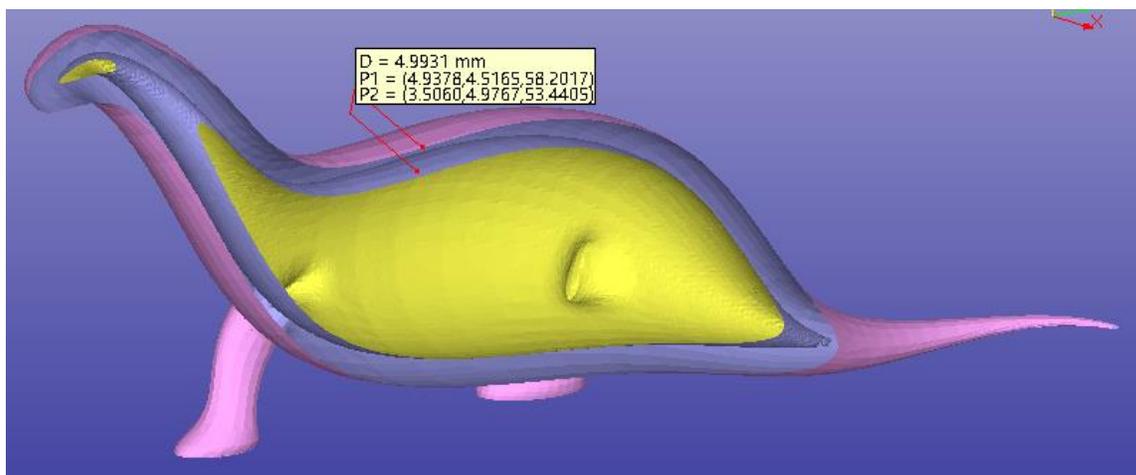
Delete the selected small parts. Now (for clarity) change the model color of the internal part to yellow (in *Edit > Properties*), select *All* and apply clipping to see that there are two shells:



To convert the inner shell into a cavity inside the outer shell, select the inner shell, give command *Edit > Cut*, select the first surface of the outer shell and give command *Edit > Paste*:



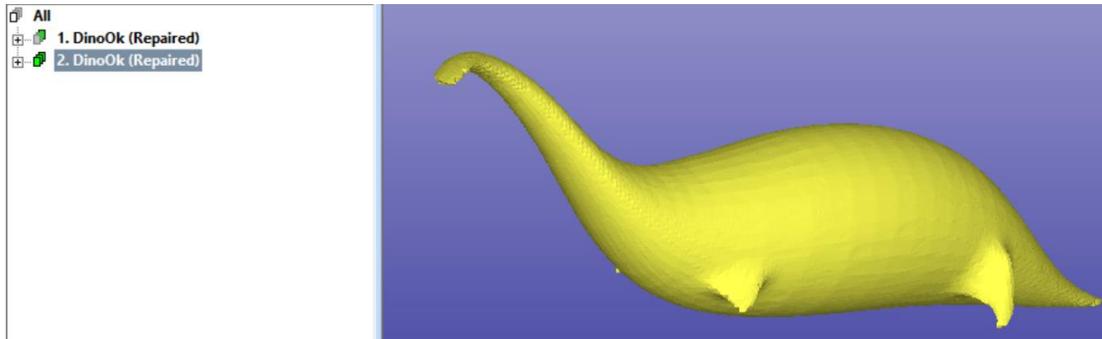
Now the inner shell is topologically correctly defined as a cavity. We can measure the wall thickness at a suitable point to see that the thickness is close to 5 mm:



Creating a 3D Lattice

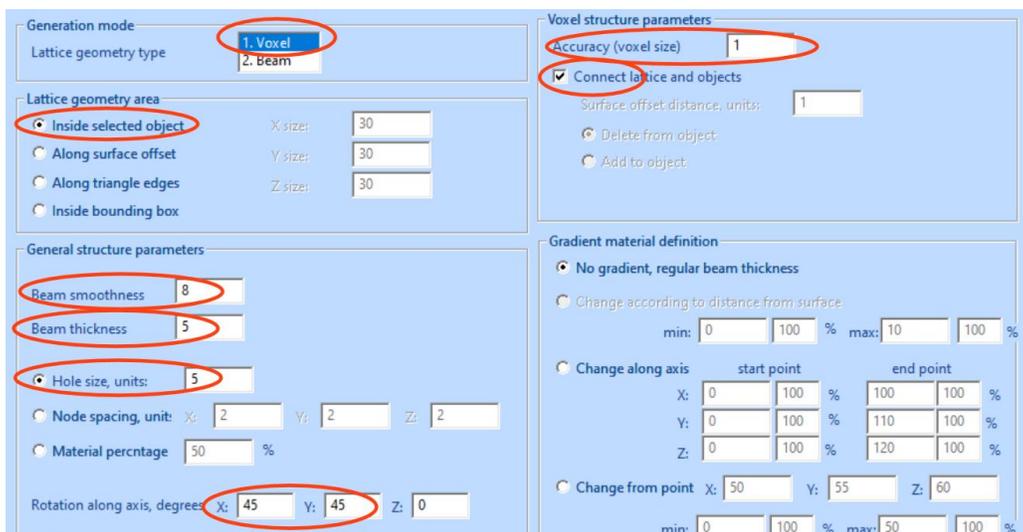
A large cavity is often not a structurally feasible means for saving material. In the following we will take the temporary *Scan Offset* result and convert it into a 3D Lattice structure, and finally convert the lattice structure into a cavity inside the Dino model.

First **undo** the *Scan Offset* steps **until you see two *DinoOK (Repaired)* shells**, then select the innermost one and hide the original Dino:



Then start command *Modify Faceted > 3D Lattice*. The parameter dialog for 3D lattice generation is shown. For a concise description of the parameters, please look at the 3Data Expert Online Help (F1). As seen in the picture below, set the following parameters:

- Lattice geometry type: Voxel
- Lattice geometry area: Inside selected object
- Accuracy: 1.0 (using a small value would significantly increase calculation time)
- Turn on parameter *Connect lattice and objects*
- Beam smoothness: 8
- Beam thickness: 5
- Hole size: 5
- Rotations along axes: X and Y: 45 degrees, Z: 0 degrees

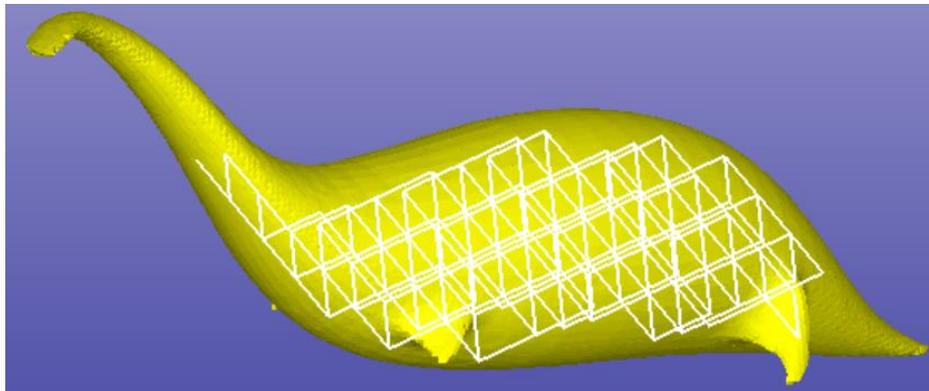


3D Lattice dialog buttons do the following:

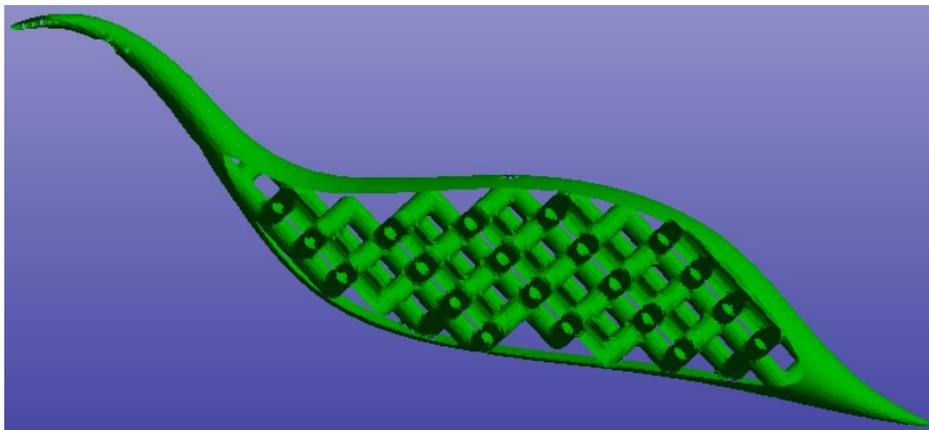
- *Show Net*: display a quick preview of the lattice structure
- *Create Triangles*: calculate a triangle model of the lattice. This is still a temporary preview of the final 3D lattice, but allows you to investigate the details of the structure
- *OK*: Create the lattice triangles and add them into the Model Tree.



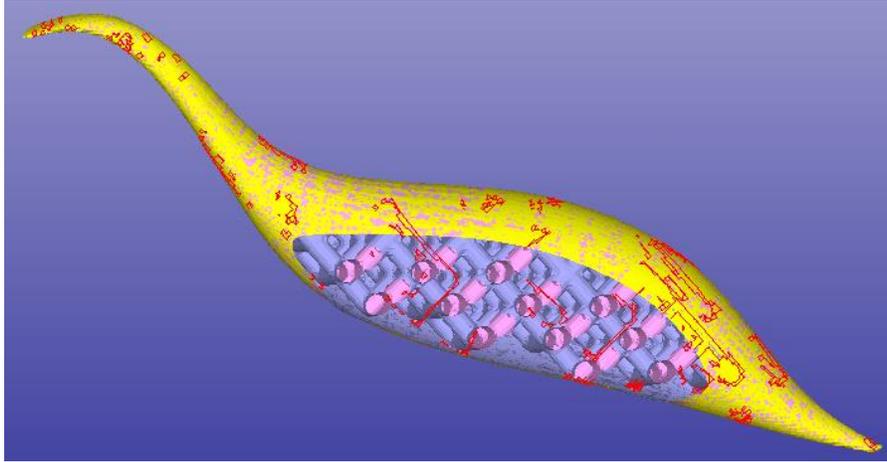
Let's first click *Show Net* to get a quick preview of the lattice generated from these parameters:



Next we will click the *Create Triangles* button. To better see the effect, we hide the original shell in the Model Tree and apply clipping from both sides of the X axis:

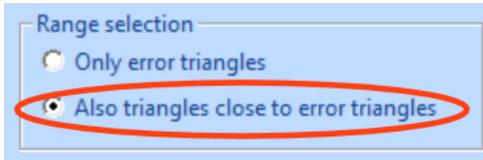


Next click *OK* to add the lattice into the model tree. The lattice will be automatically repaired and tiny details removed (click *OK* in the *Remove Tiny Results* dialog). Even after this the model needs more cleaning:



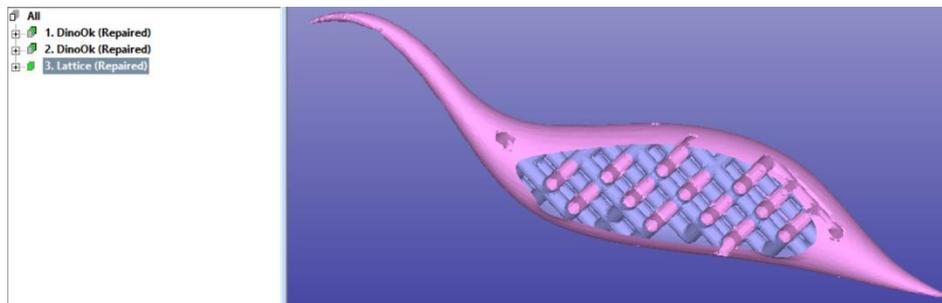
Remove first any remaining small *Lattice (Offset)* shells from the Model Tree. To fix remaining (big) *Lattice (Offset)* shell, run (probably several times):

- *Fix Model > Delete Errors* (with parameter *Also triangles close to error triangles* set!)

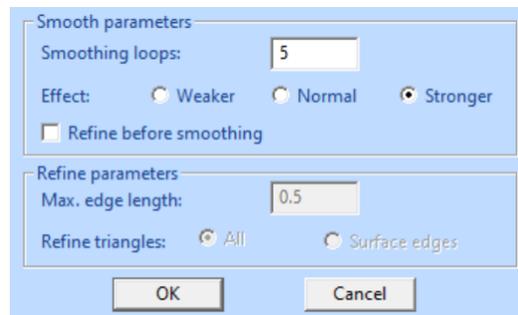


- *Fix Model > Repair Shells*

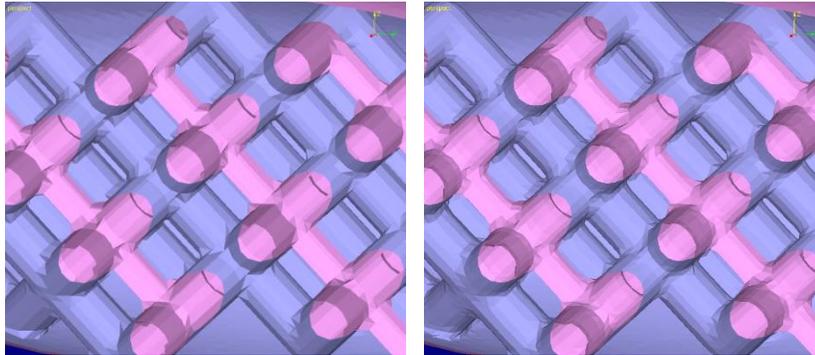
until you get a clean enough lattice shell:



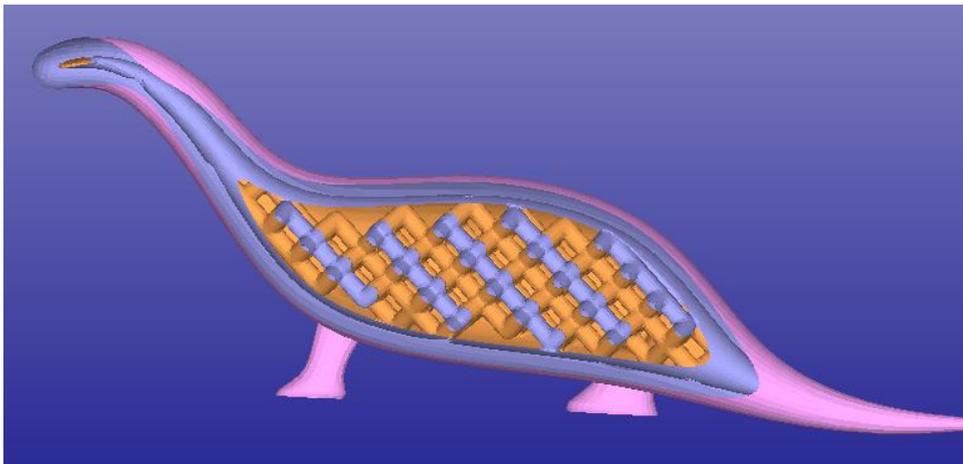
Next, we will smoothen the lattice. Select the *Lattice (Repaired)* shell and start command *Modify Faceted > Smooth*. Click Yes to the collapsed-surfaces warning message, and set *Smoothing loops* to 5 and *Effect* to “Stronger” in the smoothening dialog:



Here's a detailed view of the lattice before and after smoothing:



Let's again change the color of the smoothed lattice to orange. Finally, *Edit > Cut* the *Lattice (Smoothed)* shell, select the first surface of the original outer shell of the Dino model and *Edit > Paste*. The model now contains an internal lattice structure:

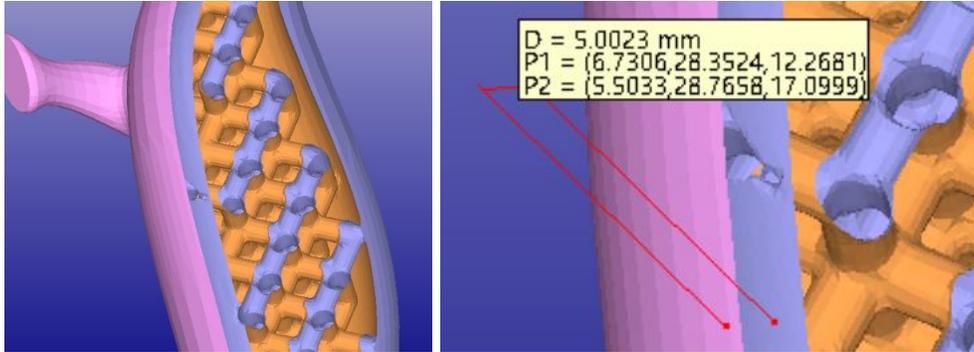


Making Drain Holes

Drain holes are needed when building Stereo lithography or powder based hollow parts with large, trapped volumes. The *Print Preparation > Drain Holes* command gives an easy way to make holes of desired sizes in the STL models.

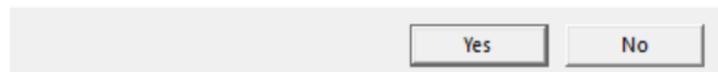
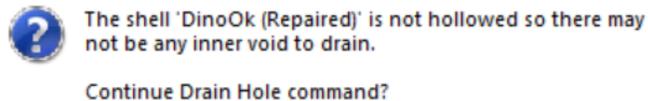
We will continue with the result of the previous example, hollowed Dino model. We will keep the same clipping settings to verify how the drain hole will interact with the 3D lattice.

- **First take a suitable view** of the *dino* model to see the underside of the model by pressing the  icon with the *RMB*.
- Rotate the view so that you can see both the dino belly and the internals
- Take a measurement of the **wall thickness** at a suitable position



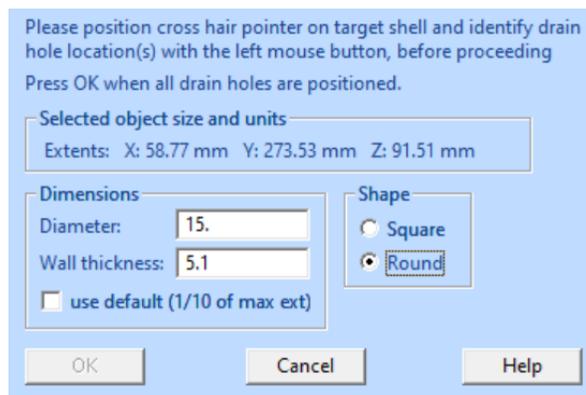
– Give the **Modify Faceted > Drain Hole** command or press the  button to start the *Drain Hole* command.

Depending on how the hollowing was made, you may get a warning that the model is perhaps not hollow. Just accept it with **Yes**:



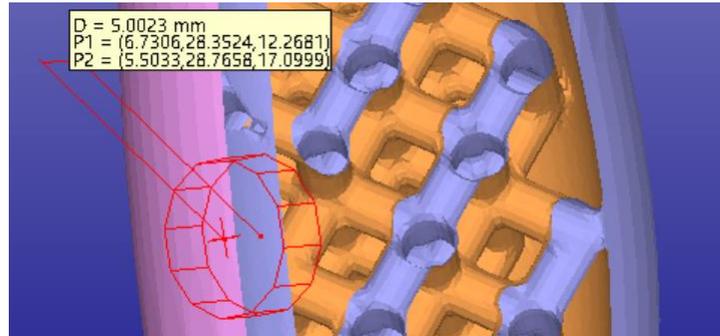
A parameter dialog appears; the default values are again calculated based on the maximum extent of the selected object.

- **Modify the *Diameter* and *Wall Thickness* values**, as seen in the parameter dialog image below (15 diameter and 5.1 mm wall thickness, slightly bigger than the measured wall thickness):
- **Choose Round shape**



The holes (one or more) are positioned with *LMB* clicks on the display area on top of the dino model. Give the hole location in the middle of the bottom surface. Note that you can use the *MMB* zoom and *MMB* rotation to change the viewpoint when locating the connectors.

- **Position a drain hole with LMB in the middle** of the bottom surface, near the wall thickness measurement point, as seen here:

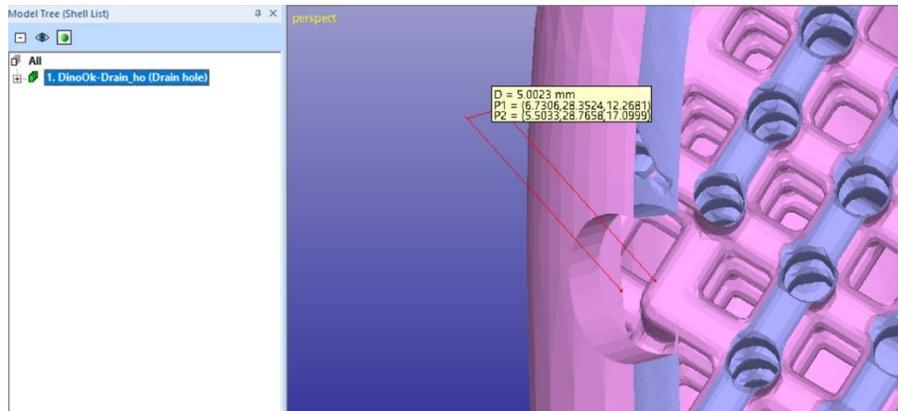


If the hole location is not correct you can remove it by clicking on the red indicator box again. Then reposition with another LMB click on the correct location.

When ready

- **Press *OK* to start the Drain Hole generation.**

After a while, the Drain Hole is generated parallel to the surface normal at the given location:



Drain Holes can be generated on any wall, they are always created orthogonal to the surface at each given location.

Making tapered plugs with Hole Tool

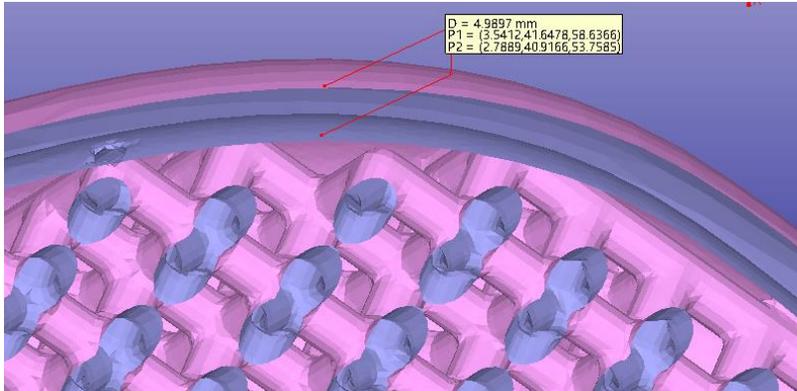
The *Special Modify > Hole Tool* is meant for generating holes with different shapes and depths for various applications. It can also be used to generate a drain hole for hollowed parts with a tapered plug. The tapered plug can be printed separately and glued to the printed part to close the hole accurately.

If you cannot see the *Special Modify* menu, turn on the *Advanced Editing* mode from the  toolbar button or from *User Interface > Advanced Editing Mode*.

3Data Expert 14.1: Tutorial 1 - Working with 3D Models

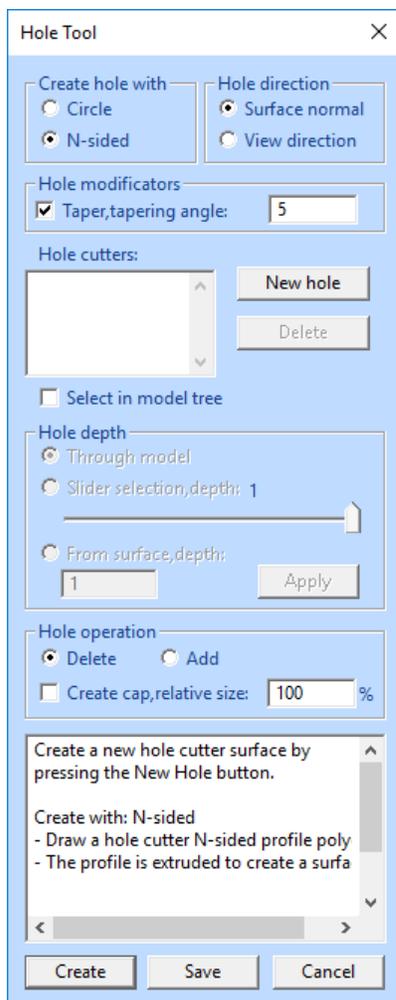
We will continue with the hollowed dino created in the previous example. Now take a view from the top of the model, apply clipping from +X direction (almost to the middle of the model).

Then measure the wall thickness at a suitable point at the top of dino's back:



Write down the thickness. Then **start the *Special Modify > Hole Tool* command.**

A Hole Tool dialog is displayed:



The Hole Tool dialog contains many parameters for hole generation. Now we want to generate a hole which is drilled in the surface normal direction. We also want to generate a plug for the hole at the same time. The plug will have tapered geometry, to prevent it from falling into the part. Also, the plug will be scaled slightly smaller than the hole, to enable easy gluing of the plug into the hole.

Start the hole generation by

- **select *N-sided*** hole
- **use *Surface normal*** direction for the hole direction
- **set *Taper angle*** to 5 degrees

as seen in the image on the left.

The hole location is selected graphically with the LMB on the model. When the above parameters are set, we are ready to

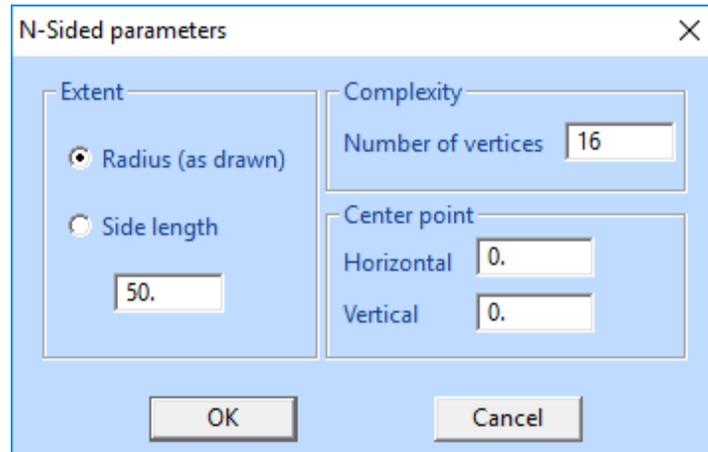
- **press the *New hole* button** to show the location and size of the hole.

A crosshair icon will appear.

- **Click near the wall thickness measurement point to set the hole middle point.**

Then

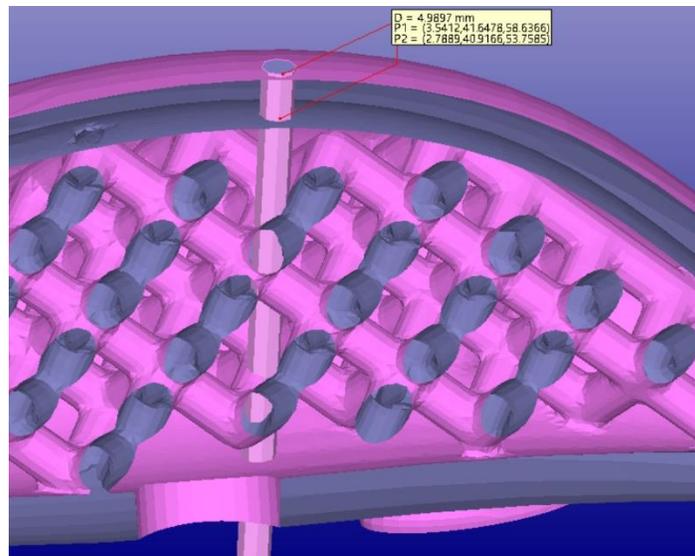
- **draw a box by dragging** with the LMB to estimate the size of the hole to generate.
- **When the mouse is released**, a parameters dialog appears



Note: Center point does not apply here, it was selected with the mouse.

Then

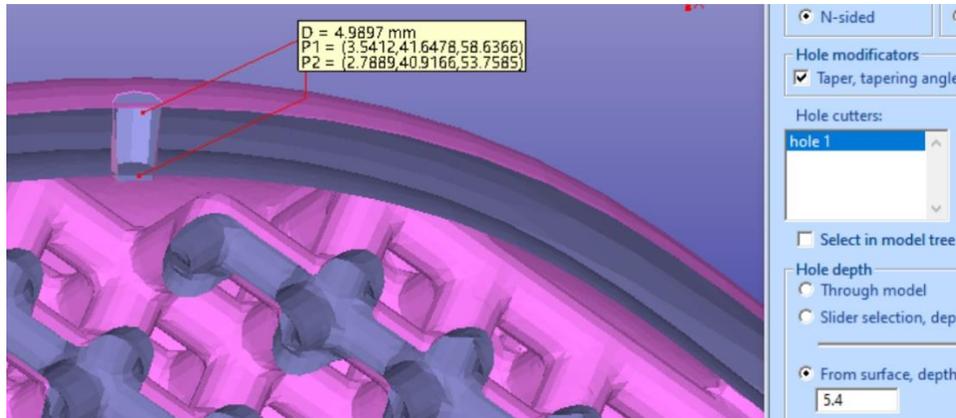
- **Select *Side length*** and set the value to *4.0*. Also
- **set the *Number of vertices*** to *8* to generate a plug with distinguishable corners for easier positioning after the build.
- **Then press OK** to generate the plug indicator on the model, as seen in the image below (note how it extends far down through the dino):



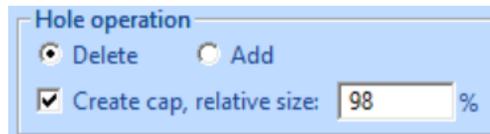
Hole 1 will be generated into the Hole cutters list. Now we will set the length and scaling for the plug.

Then

- **Select “hole 1”** in the cutters list.
- For *Hole depth* **select the *From surface, depth*** and set the value to 5.4 (slightly bigger than the measured wall thickness).
- **Press *Apply*** to really change the hole depth after setting the depth value. Note how the hole tool now extends a little outside the dino outer and inner surface, but does not interfere with the lattice structure:



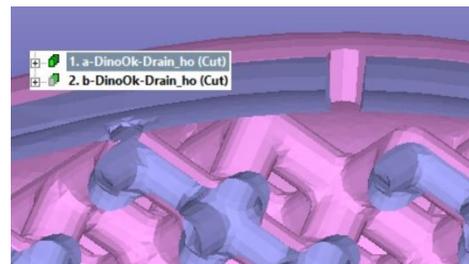
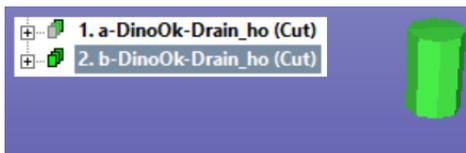
- Make sure the *Hole operation* is **set to *Delete***.
- **set the *Create cap, relative* size 98%** to make the plug slightly smaller than the hole:



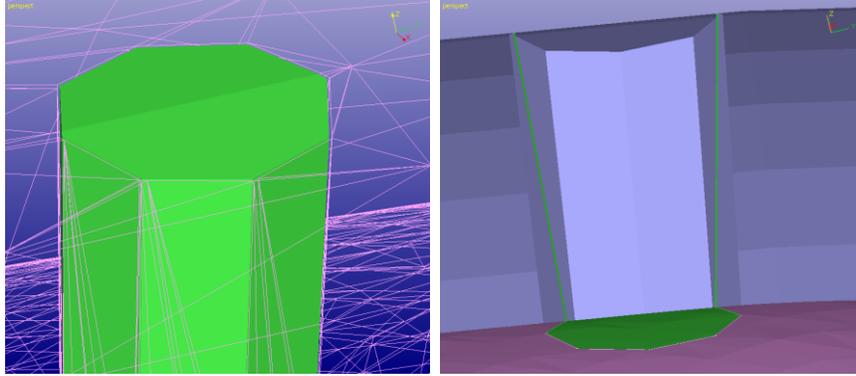
When ready,

- **press the *Create* button.**

The created hole cutter is deleted from the original model and in a few moments, you will see the plug fitted in the middle of the screen. The dino model has a corresponding hole in the middle of the back (shown with clipping and plug color changed to green):



If you zoom closer to the hole and change the dino body to wireframe mode (with graphics area RMB popup menu) you can see that the plug is slightly tapered and smaller than the hole. Another closeup with clipping on the right.



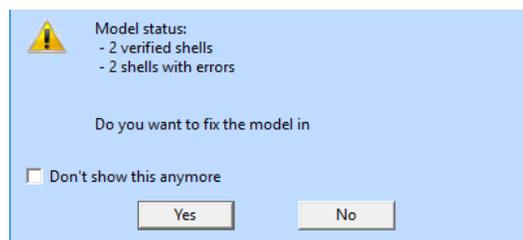
These two shells can now be printed separately. After the body is drained from non-processed material or any internal support structures are removed, the two pieces can be glued back together for a nice-looking part.

Adding thickness by Offset

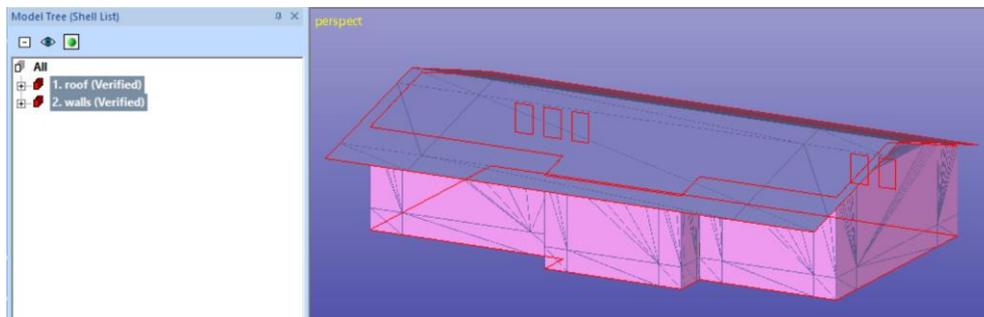
There are several software products which do not automatically produce solid STL data. For example, many architectural software products output open models made of open surfaces unless constructed very carefully. These open models must be made solid before inputting to 3D Printing.

3Data Expert *Modify Faceted* > *Offset* command is used to generate solid models from open surface models. Open shells can be offset to user defined thickness for correct build.

- **Initialize your system with *File* > *New*** and
- **input the *House.3de* model from *Tutorials / GeomFiles* directory.**
- **Go directly to the *Modify* mode** and when the repair question appears:



- **press *No*** to continue without repair.

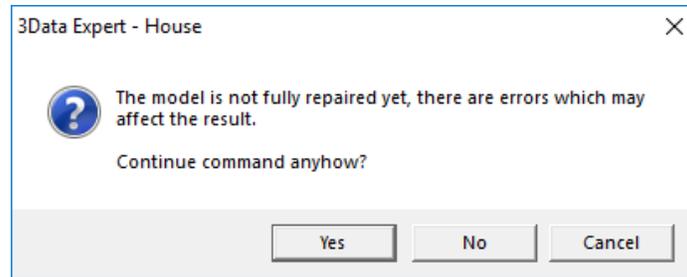


3Data Expert 14.1: Tutorial 1 - Working with 3D Models

The House model contains two open shells, the roof and the walls. The open surface edges are drawn with red gap curves.

We will use the Offset command to give thickness to the open surfaces.

- **Select the *roof shell*** and
- **give the *Modify Faceted > Offset* command** or use the  button. The program will notify that there are errors in the model after you give the command.

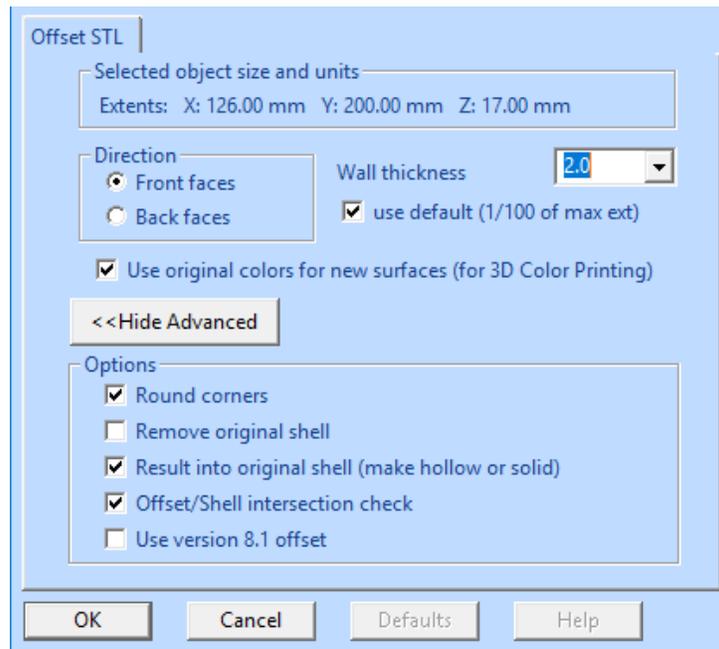


- **Press *Yes*** to continue the command (the errors will be automatically repaired after the offset):

You will see the *Offset STL* parameter dialog. As usual, the default offset distance is calculated based on the maximum extent of the roof shell.

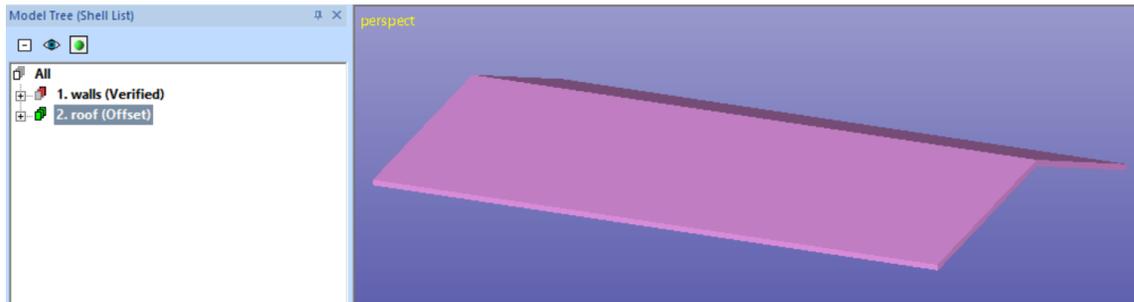
- **Change the *Wall thickness* to 2 mm** and make sure the other parameters are as shown on the image below.

Otherwise, we can accept the default values for *Advanced* parameters.

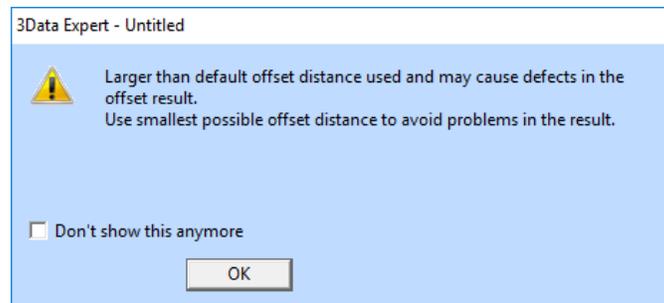


3Data Expert 14.1: Tutorial 1 - Working with 3D Models

- **Press OK** to start the command. After a while you will have a correct solid roof in the Model Tree:

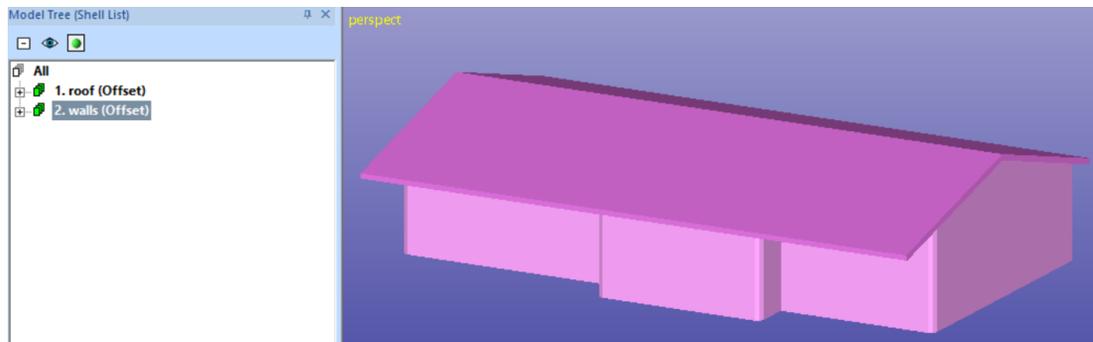


- **Now select the wall shell** and
- **repeat the Offset with the 2 mm Wall thickness** again. You may see the warning



- **Press OK** to close the dialog and start the offsetting by
- **pressing OK** again in the Offset parameter dialog.

After a while the offset is ready, and you should have two correct shells in the Model Tree:

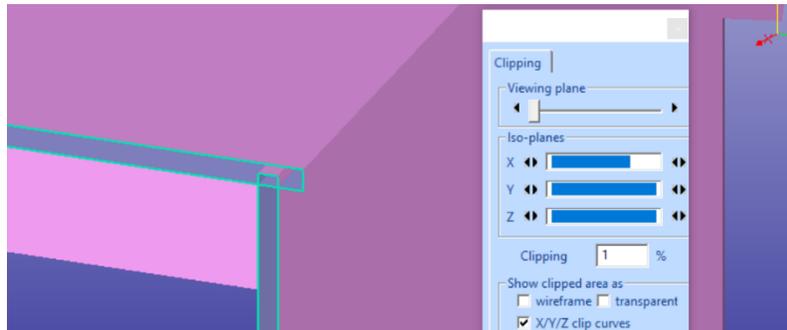


There are some 3D Printing systems which can produce a correct result from these two shells directly. Some systems require the two shells to be connected into one shell to produce a correct result. In the next example we will join the two shells into one shell using the Boolean commands.

Combining shells using Boolean commands

The two shells generated in the previous example are correct, but may not print properly if, before outputting to 3D Printing, they are not connected into one shell. We will use the Boolean commands to generate a single shell from the two shells.

Let's first check how the two shells intersect each other. If you zoom in closer to the part and use the Clipping to look inside the shells (with X/Y/Z clip curves checked), you will notice that the shells truly intersect:



Note: If the roof and the walls do not intersect you may have generated the roof offset to *Back faces* direction instead of *Front faces* or used too small offset thickness. If so, please start the lesson again and follow the instructions carefully.

Reset and close the clipping window before the next command.

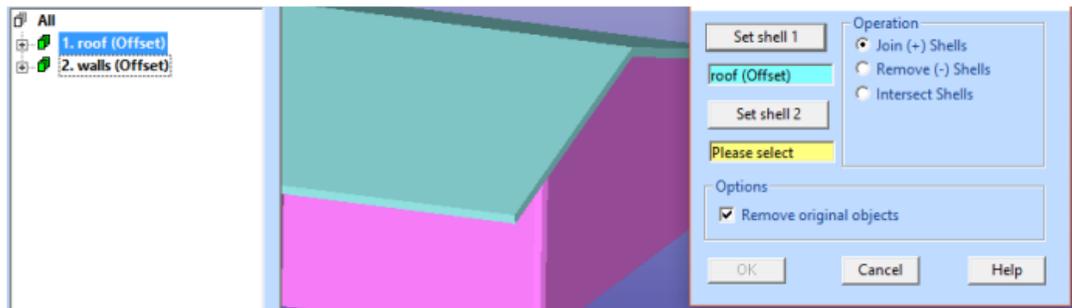
We will use the Boolean command to connect the two shells.

- **Start the *Modify Faceted > Boolean* command now.**

The Boolean parameter dialog will appear. It allows you to select the shells from the Model Tree, as well as define the Boolean operation to perform between the selected objects.

First

- **select the *1. roof* shell in the Model Tree and then**
- **press the *Set shell 1* button.**



The roof turns light blue in the graphics are and its name is displayed in the shell 1 text filed. Then

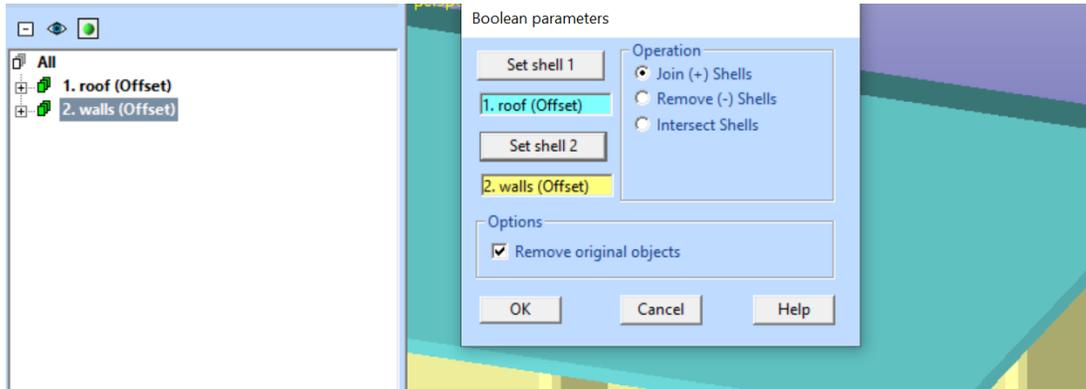
- **select the *2. walls* shell in the Model Tree and**
- **press *Set shell 2* button.**

3Data Expert 14.1: Tutorial 1 - Working with 3D Models

The walls turn yellow and its name is stored in the shell 2 text field. There are several different operations available between the selected objects but this time we simply want to

- **Join** (default) them into one shell.

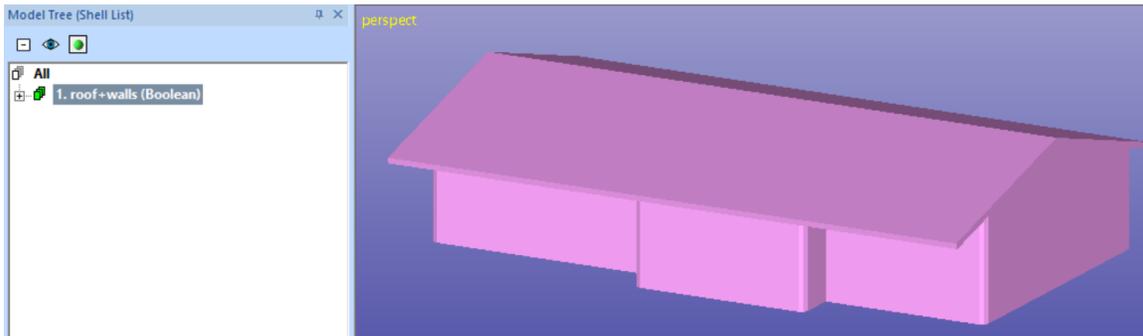
Make sure the dialog looks as seen below:



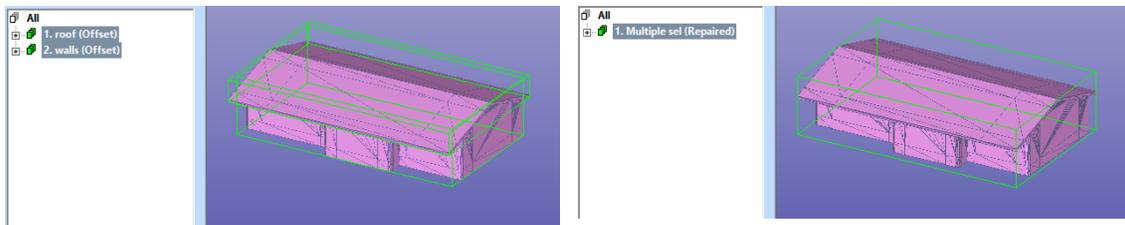
Then

- **press OK to join the two shells into one.**

After a while you will see the two shells connected into one correct shell:



You may also generate the same Boolean result by multi-selecting the two shells and by running the *Modify Faceted > Multiple Join* command on them:



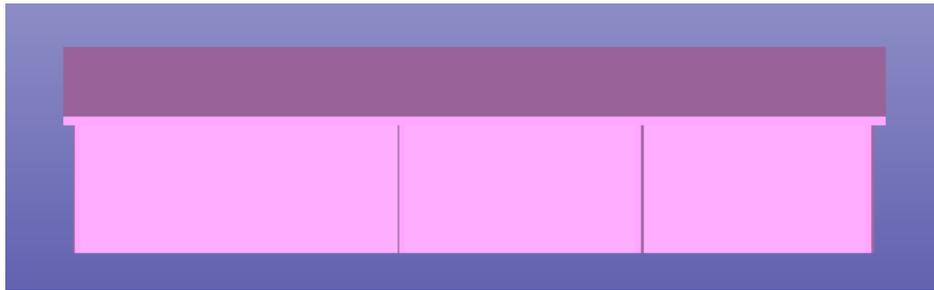
Generally, there are some issues which need to be considered when using the Boolean command. Especially important is to make sure that the two shells really intersect each other. Tangential or overlapping shells may behave improperly when sent to the Boolean operation. In these cases, you should slightly move one of the shells or scale them a little. The amount of move or scale should be very

small (like 0.001 - 0.01 mm for a model this size), to not affect the accuracy of the printed model.

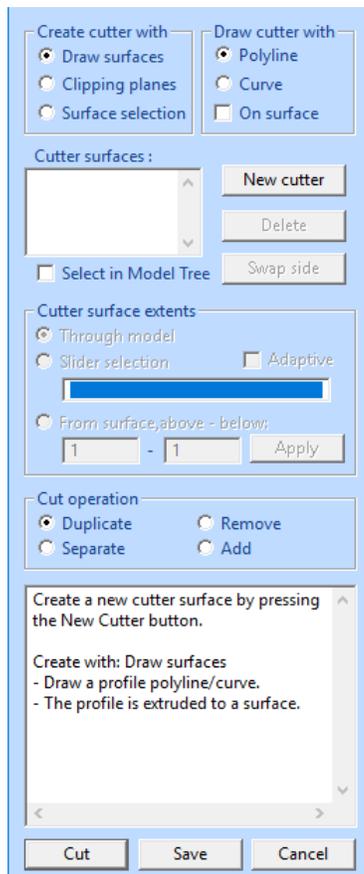
Generating advanced cuts with Cut Tool

Cut Tool is meant for advanced cutting for various applications. For example, it can be used to cut specific details inside a 3D model without disturbing the surface area in the close vicinity of the cutaway detail. We will now use it to cut a door opening to the house model in the previous example.

First take a view from the x axis direction by pressing the X icon with the LMB.



Then give the **Special Modify > Cut Tool** command to open the Cut Tool dialog:



The Cut Tool dialog will be used to define a Cutting surface to cut away the door opening to the model. We will start by selecting

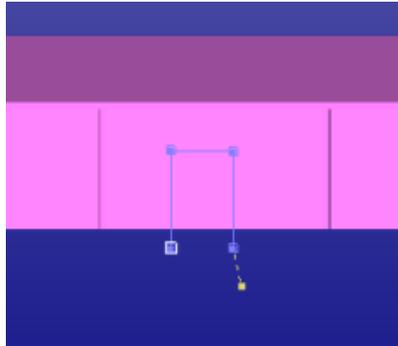
- Create cutter with **Draw surfaces**
- Draw cutter with **Polyline**

The profile for the cutter surfaces will be generated with the curve editing tool already used with the Split command.

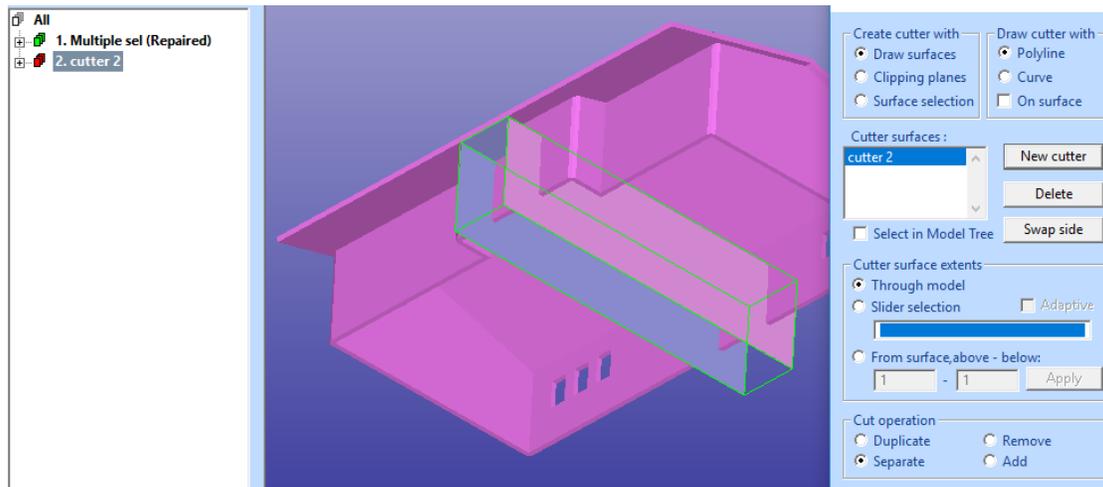
- **press New cutter button** to start drawing the door profile on the house.

Then

- input the curve points with the **LMB** and
- press **RMB** when ready with the shape below:



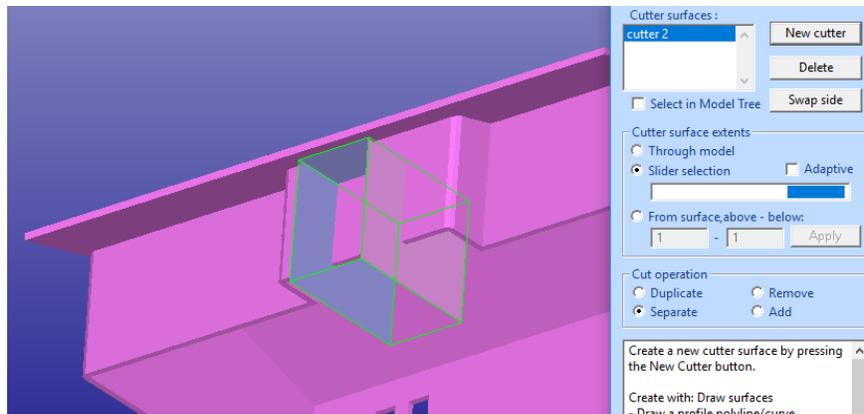
A new cutter surface is generated into the *Cutter surfaces* list:



The next step is to set the *Cutter surface extents* so that it will only cut the front wall of the house model. For that

- **select the created cutter** in the Cutter surfaces list.
- **select the Slider selection** and use the slider to resize the cutter surface to correct depth, as seen in the image below

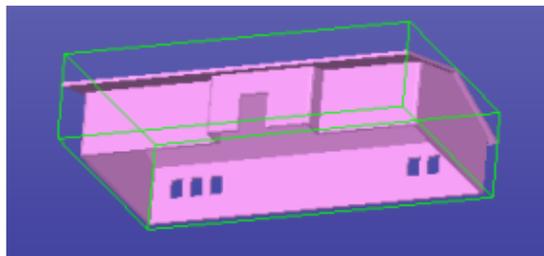
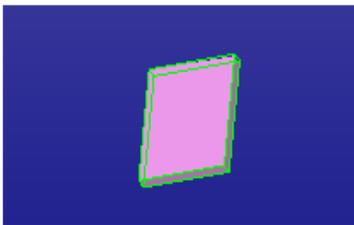
A colored bounding box is used to indicate the cutter surface size while moving the slider with the **LMB**. The yellow bounding box indicates the maximum size of the cutter surface, red bounding box indicates the current size of the cutter surface and moving blue lines indicate the new start and end locations to be used when the LMB is released.



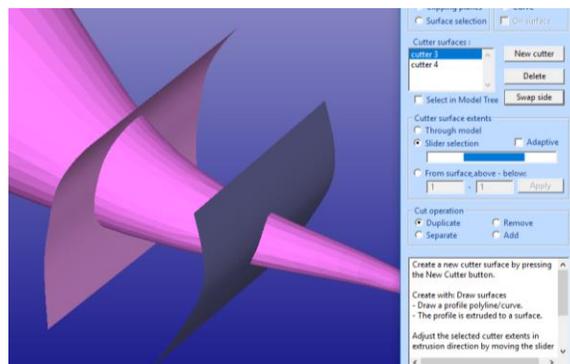
After rescaling the cutter surface to the depth seen above

- make sure that the *Cut operation* is set to **Separate**.
- Then **press Cut button**.

After a while you will see the result on the screen, the cut part (left image below). When fitting the house, we see that there is a door opening in the front wall.



The Cut Tool gives you unlimited possibilities to cut models. You can also use several cutting surfaces simultaneously to generate even more complicated cuts for various applications. For example, in the next picture we can see two cutting planes designed to remove a piece from the dino tail, using cutting planes with different cutting directions:



Note: When using multiple cutting planes make sure that the surface normal directions are consistent for the cut operation: For two adjacent cutting surfaces the normal must either be directed towards each other or away from each other. If it is not, the normal direction for a cutter surface can be changed with the *Swap side* button after selecting the surface in the *Cutter surfaces* list.

Lesson 10 – Preparing for simulation applications

This lesson shows you how to use the Reduce and Refine commands in the Modify Faceted menu to reduce the number of triangles in an STL model as well as how to reshape the triangles for a better aspect ratio. For some simulation software systems, it is important not to have too many triangles in the model to speed up the simulation. Some simulation applications require triangles with a good aspect ratio to produce the most reliable results.

The 3Data Expert *Reduce* command can also be used for preprocessing data for Augmented Reality (AR), Virtual Reality (VR) and game software systems. If required, the reduction will keep the textures and colors on the inputted VRML or ZPR model as accurately as possible.

This lesson is applicable to the *Base* module only.

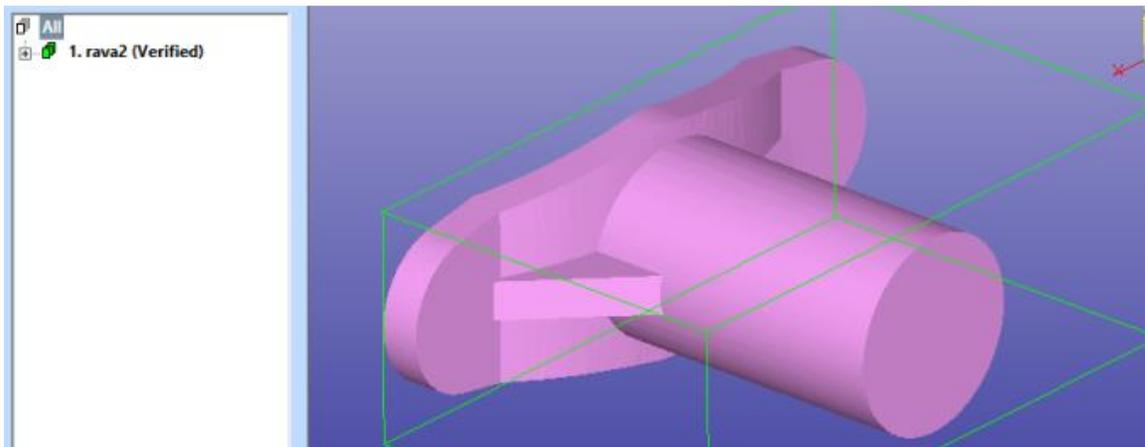
Starting the reduction

Let's first

- **input the model *Tutorials / GeomFiles / rava2.stl* using the *File > Open* command.**

Accept the proposed units and when the Operation Mode must be chosen,

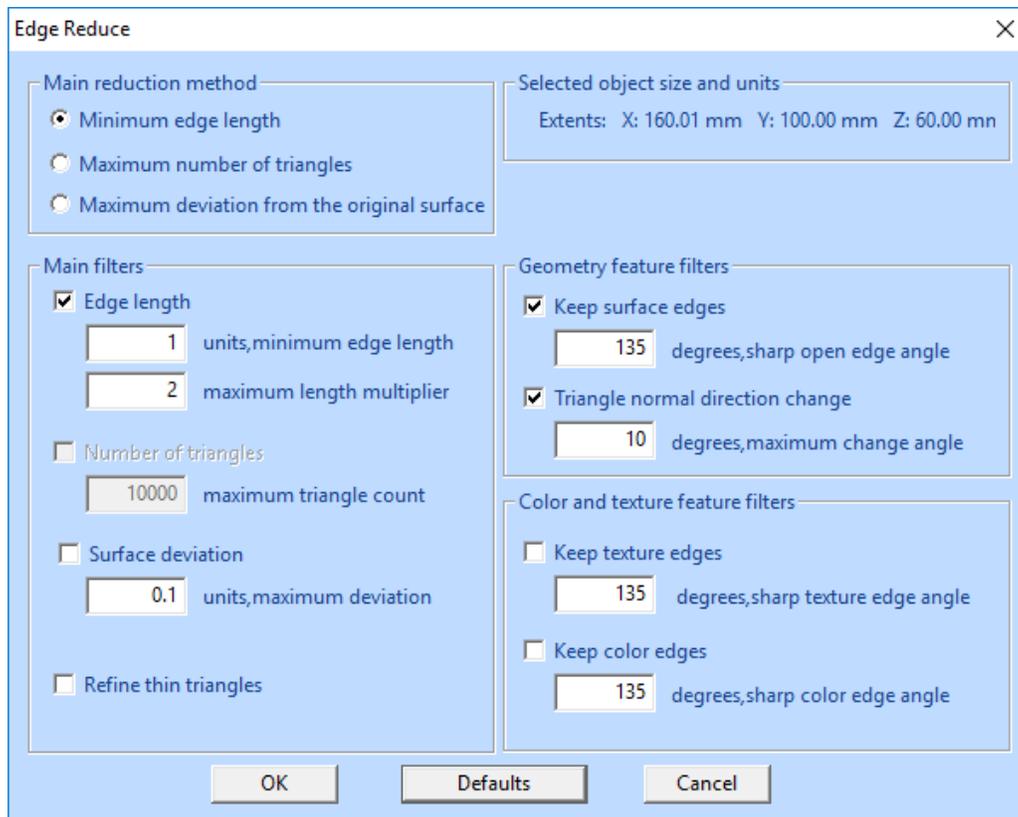
- **run the basic *Auto Repair* process** on the model by pressing the *Fix* button in the Operation Mode Window
- After a while you will have a correct model:



Now select the rava2 (Verified) in the Model Tree and

- **give the command *Modify Faceted > Reduce*.**

The Reduce parameter dialog will appear:



There are three different main reduction methods, and you can affect the result by selecting the most suitable one for your application:

- **Minimum edge length method** will allow you to set the minimum edge length to be left in the model, all triangle edges shorter than the given minimum edge length are removed.
- **Maximum number of triangles** method will remove triangles until the number of triangles reaches the given maximum triangle count.
- **Maximum deviation from the original surface** will remove all triangle edges which do not cause the surface points to move more than the given maximum deviation.

Whichever method is used, the triangles are reduced so that the given limits are respected. Anyhow, the command does not reduce triangles so that erroneous geometry would be generated (i.e., non-manifold geometry with more than two triangles at an edge, overlapping triangles etc.).

The limits for the reduction method are given in the **Main filters** group. Each reduction method has its own Main filter which must be used with the method.

- **Edge length** filter will set the **minimum edge length** remaining in the reduced model for **Minimum edge length method**. Maximum length multiplier sets the upper limit to the edge length change during the reduction, this is important when producing triangles for simulation applications which require a good triangle aspect ratio. **Note:** *With Minimum*

*edge length method you can, in addition to the basic Edge length filter, use **Surface deviation filter** to have even more control over the surface shape change.*

- **Number of triangles** filter will give the **maximum triangle count** allowed in the model after the reduction.
- **Surface deviation** filter will give the maximum distance of the reduced surface from the original surface with the **maximum deviation limit**.

Geometry feature filters will add special rules to preserve the geometric features in the model, i.e., the surface edges and triangle normal directions.

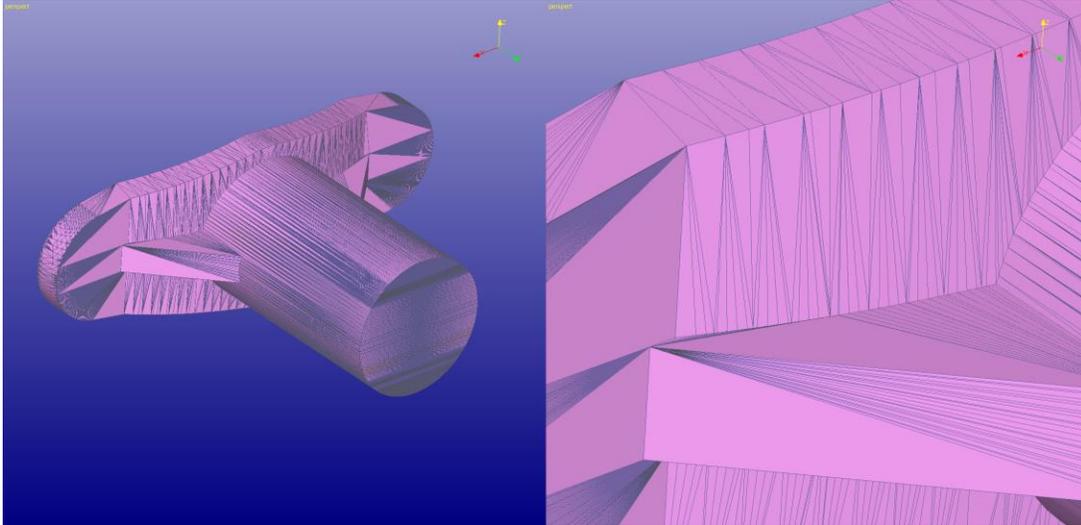
- If **Keep surface edges** is set, the surface edges generated during the *Auto Repair*, *Verify Shells* or *Repair Shells* commands will be respected and not deleted from the model. Triangles can be reduced along the edges but not over the edges, effectively preserving at least a few triangles from each surface in the model. Also, sharp corners at open edges (at the gap curves) are preserved within the user defined limit, all corners sharper than the given limit are kept. For example, a square with 90-degree sharp corners will remain a square after the reduction with the default 135 degrees **sharp open edge angle** limit ($90 < 135$).
- **Triangle normal direction change** will limit the change of triangle normal to the given degree value. This will very effectively prevent large changes in the model shape. For example, if a cylindrical hole needs to be reduced to minimum number of triangles (i.e., a triangle shaped hole), you must increase the **maximum change angle** to 45 degrees or more, by default only 10 degrees changes are allowed, therefore cylindrical holes may remain densely triangulated.

Color and texture feature filters will add special rules to preserve the textures and colors in the model. These rules are similar to the rules applied to the surface edges above: triangles are not reduced over texture or color boundaries, but they may be reduced along a texture boundary. This will always leave a reduced representation of the texture (or color) area in the reduction result. Color and texture feature filters are particularly useful when reducing data for AR, VR or game applications. **Sharp texture edge angle** and **sharp color edge angle** work in the same fashion as **sharp open edge angle** above.

Generally, the more filters you are using, the less reduction may happen and the longer it may take to run the command. On the other hand, filters are very important for preserving desired features in the model.

Reducing the rava2 model

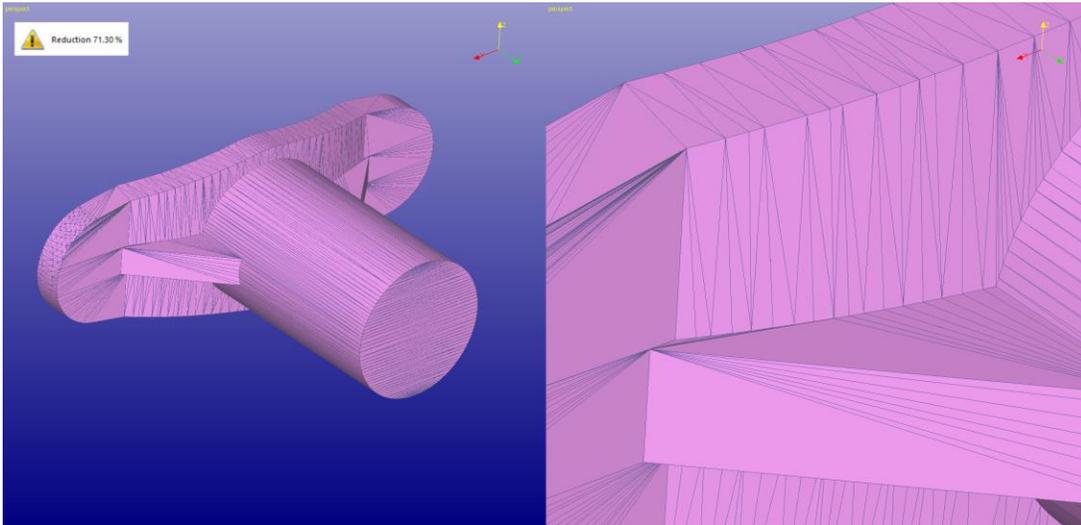
Now let's try to reduce the rava2 model with the default parameters. Drawn as wireframe and shaded, the original model looks like this (with a close-up on the right):



For reduction, you can make sure the default parameters are in use by

- **pressing the *Defaults*** button in the dialog.
- Then **press OK button**.

You will soon see the result with 71,3% reduction:



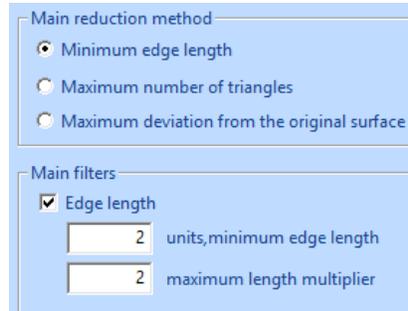
If you compare the detail in the original model with the reduced model, you can see that the geometric features are very well kept while a serious reduction is applied to the model.

The model still looks the same but now there are only 1628 triangles left from the original 5674 triangles.

Generally, there are no edges shorter than 1.0 units left in the model anymore. The new edges originally shorter than 1.0 units are now between 1.0 and 2.0 units (1.0*upper limit multiplier = 2.0).

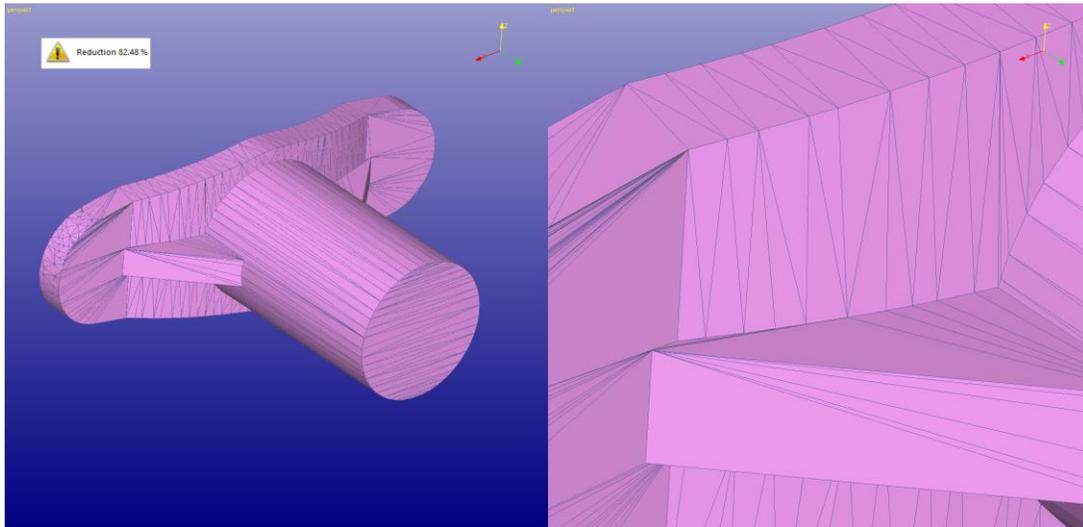
Let's try another reduction with a bigger minimum edge length.

- **Press Edit -> Undo Reduce** and then
- **Modify Faceted -> Reduce** again and set the minimum edge length to 2.0 as seen below:



Then **press OK**

and after a short while you will see the result with 82,5% reduction:



You can clearly see how more triangles are reduced from the mesh and still the model shape remains accurate.

There are some areas where you can still find triangles with shorter edges than 2.0 mm, as seen here (note the small green bounding box in the top-right corner):

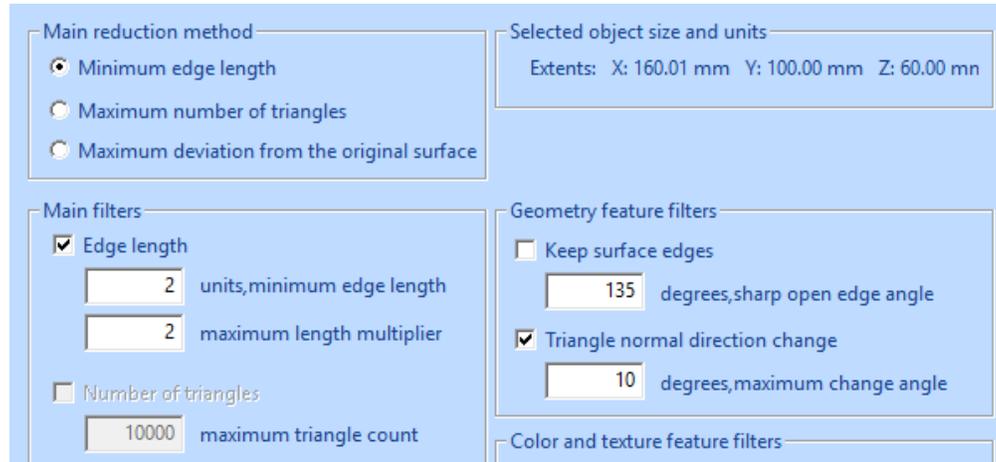
- ▲ 29. Verified
- ▲ 30. Verified
- ▲ 31. Verified
- ▲ 32. Verified
- ▲ 33. Verified
- ▲ 34. Verified
- ▲ 35. Verified
- ▲ 36. Verified
- ▲ 37. Verified
- ▲ 38. Verified
- ▲ 39. Verified
- ▲ 40. Verified
- ▲ 41. Verified
- ▲ 42. Verified
- ▲ 43. Verified
- ▲ 44. Verified
- ▲ 45. Verified
- ▲ 46. Verified



3Data Expert 14.1: Tutorial 1 - Working with 3D Models

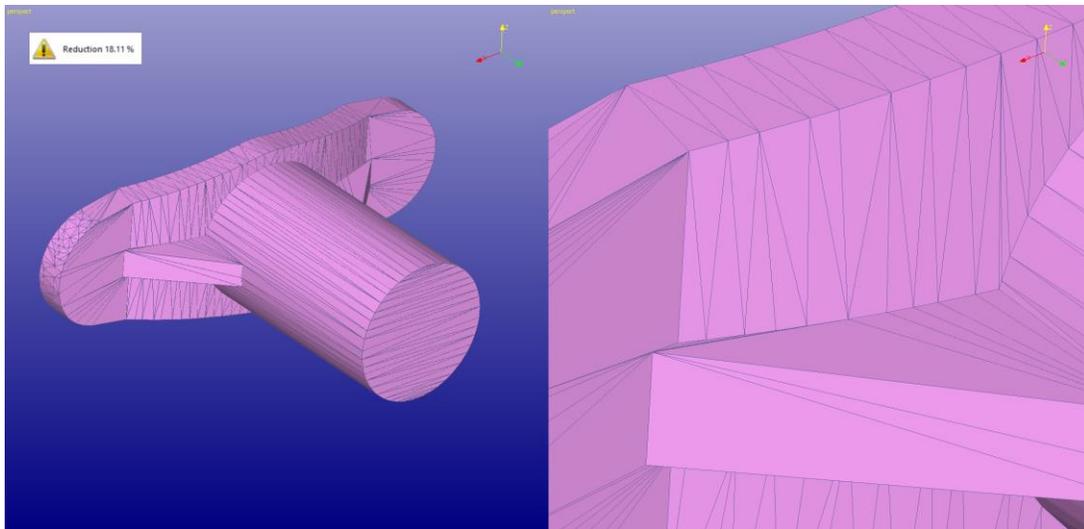
These areas are due to small one triangle surfaces in the original model (in green bounding box above), **Keep surface edges** filter prevents us from deleting any surfaces fully from the model. To remove minor surfaces like this we can make another run with the **minimum edge length** 2.0 units but

- turn **Off** the **Keep surface edges** filter, as seen below:



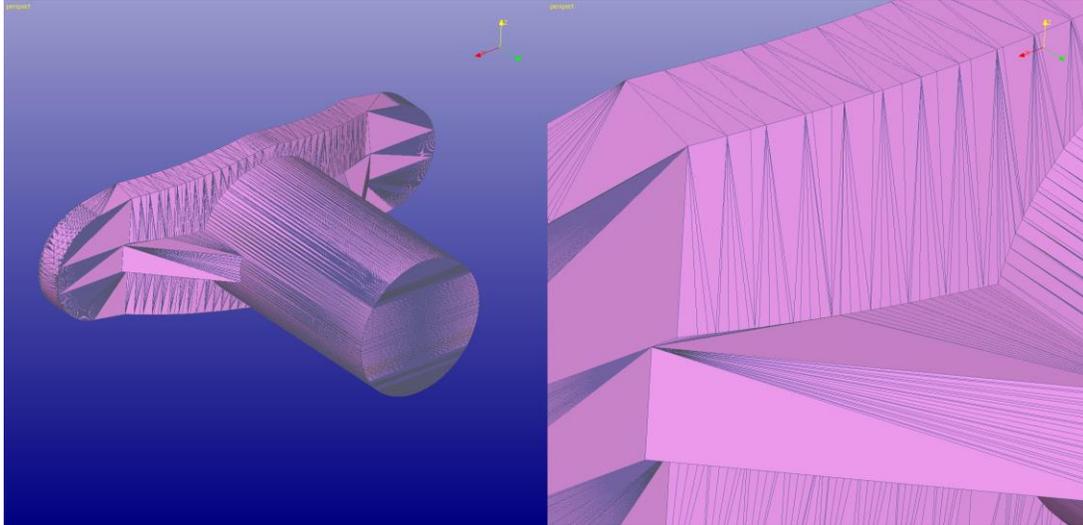
- **press OK to rerun the Reduction.**

This will produce another 18,1% reduction by removing some small surfaces altogether.



Let's now try the other reduction methods.

- **Press Edit > Undo** until you have the original *rava2 (Verified)* model available again (here in shaded + wireframe display mode).

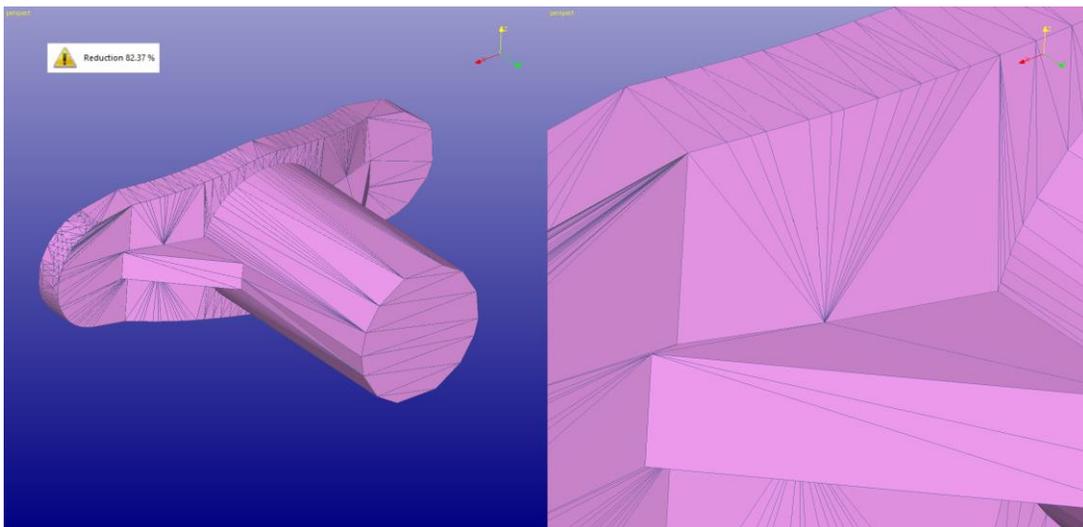


Then give the **Modify Faceted > Reduce** command

- press the **Defaults** button and give the following parameters
- reduce to maximum **1000 triangles**:

<input type="radio"/> Minimum edge length	Extents: X: 160.01 mm Y: 100.00 mm Z: 60.00 mm
<input checked="" type="radio"/> Maximum number of triangles	
<input type="radio"/> Maximum deviation from the original surface	
Main filters	
<input checked="" type="checkbox"/> Edge length	
<input type="text" value="1"/> units, minimum edge length	
<input type="text" value="2"/> maximum length multiplier	
<input checked="" type="checkbox"/> Number of triangles	
<input type="text" value="1000"/> maximum triangle count	
Geometry feature filters	
<input checked="" type="checkbox"/> Keep surface edges	<input type="text" value="135"/> degrees, sharp open edge angle
<input checked="" type="checkbox"/> Triangle normal direction change	<input type="text" value="10"/> degrees, maximum change angle
Color and texture feature filters	

- Press **OK** to see the following result with 82,4% reduction and 1000 triangles:



There is still one method to go, *Maximum deviation from the original surface*.

- Give **Undo** command, then
- start **Reduce** and give the following parameters after pressing **Default** button:

Main reduction method

- Minimum edge length
- Maximum number of triangles
- Maximum deviation from the original surface

Selected object size and units

Extents: X: 160.01 mm Y: 100.00 mm Z: 60.00 mm

Main filters

- Edge length
 - 1 units, minimum edge length
 - 2 maximum length multiplier
- Number of triangles
 - 1000 maximum triangle count
- Surface deviation
 - 1 units, maximum deviation

Geometry feature filters

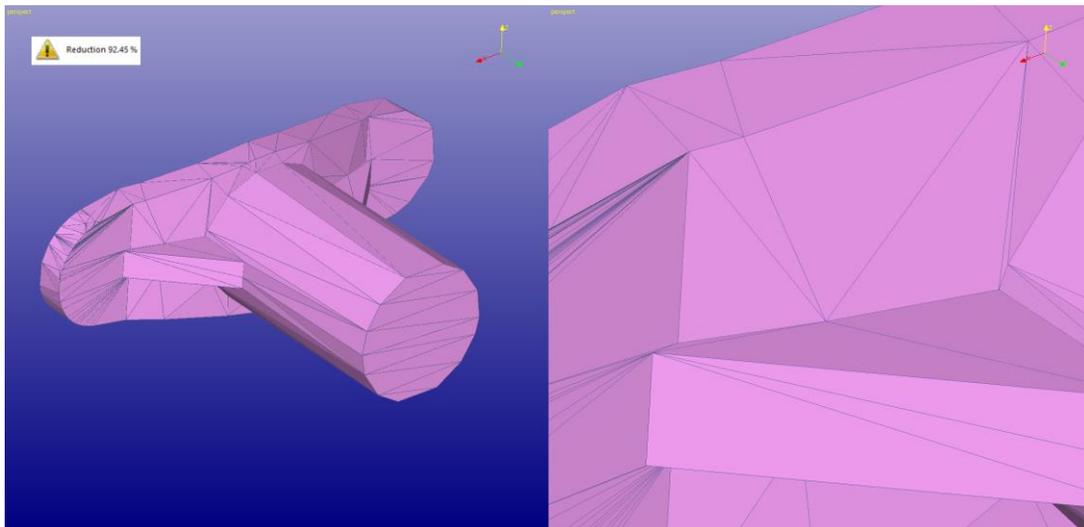
- Keep surface edges
 - 135 degrees, sharp open edge angle
- Triangle normal direction change
 - 10 degrees, maximum change angle

Color and texture feature filters

- Keep texture edges
 - 135 degrees, sharp texture edge angle

Now the maximum allowed distance of the resulting surface is 1 mm from the original surface.

- Press **OK** and you will end up with 92,5% reduction and 428 triangles:



Which reduction method to use?

As you can see, there are different ways to get to a reduced result, depending on the requirements defined for the result. Generally, **Minimum edge length** method should be used if a reduced triangulation with evenly shaped triangles is required.

Maximum number of triangles method should be used if there is some definite upper limit to the triangles accepted for the receiving application. **Maximum**

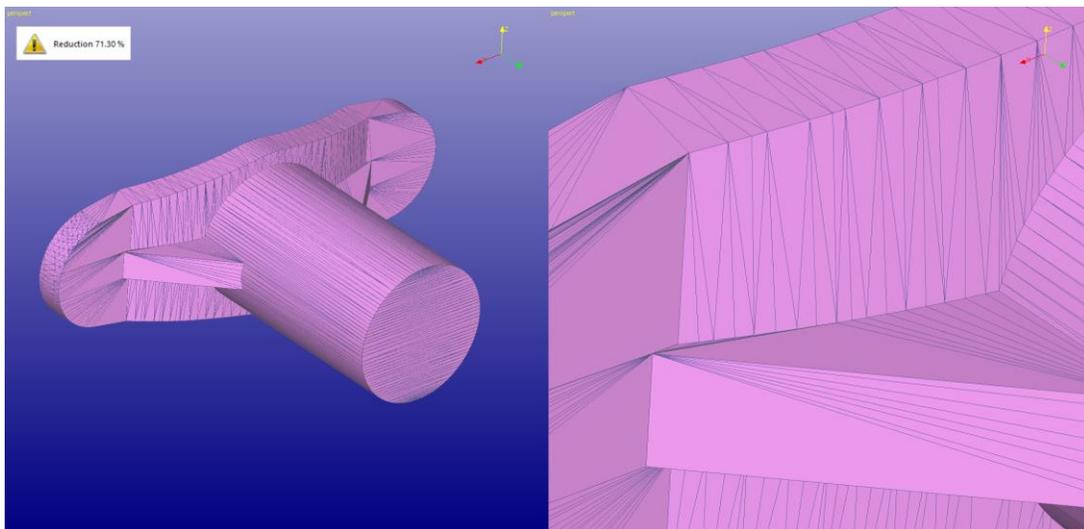
deviation from the original surface should be used if keeping the model shape is the most important requirement; some simulation systems as well as 3D printing applications may require that.

Improving the triangle aspect ratio

The triangle reduction command can also be used to create a triangle mesh with a better aspect ratio, i.e., to remove very thin triangles from the mesh. This is an important feature for some simulation applications, because better-shaped triangles will generate more accurate simulation results. A better aspect ratio is achieved by combined use of the *Modify Faceted > Refine* and *Reduce* commands.

We will start this exercise by

- **reopening the rava2.stl** file and
- **running the Auto Repair** on it through the Operation Mode Fix Model button. Then repeat the first reduction done in the chapter *Reducing rava2 model* (i.e., press *Defaults* button)
- **Select the 1. rava2 (Verified) shell** in the Model Tree,
- **give the command *Modify Faceted > Reduce*** and press **Default** button in the Reduce parameter dialog.
- **The press OK** to get the first reduce result:

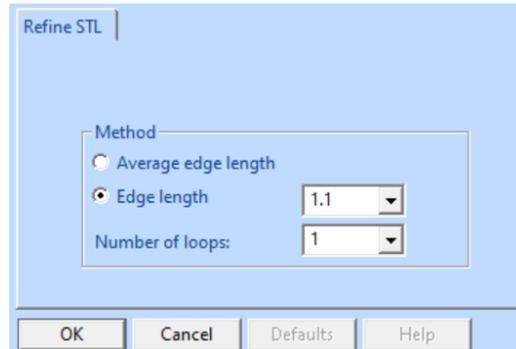


We have now removed excess triangles from the mesh. We will next run the *Refine* command to generate new triangle corner points into the mesh. These new points will be used to create new, evenly shaped triangles during the next reduction run.

First you should estimate the minimum size of the triangles you would like to see in the resulting mesh. Let's assume we want to use triangle size of approximately

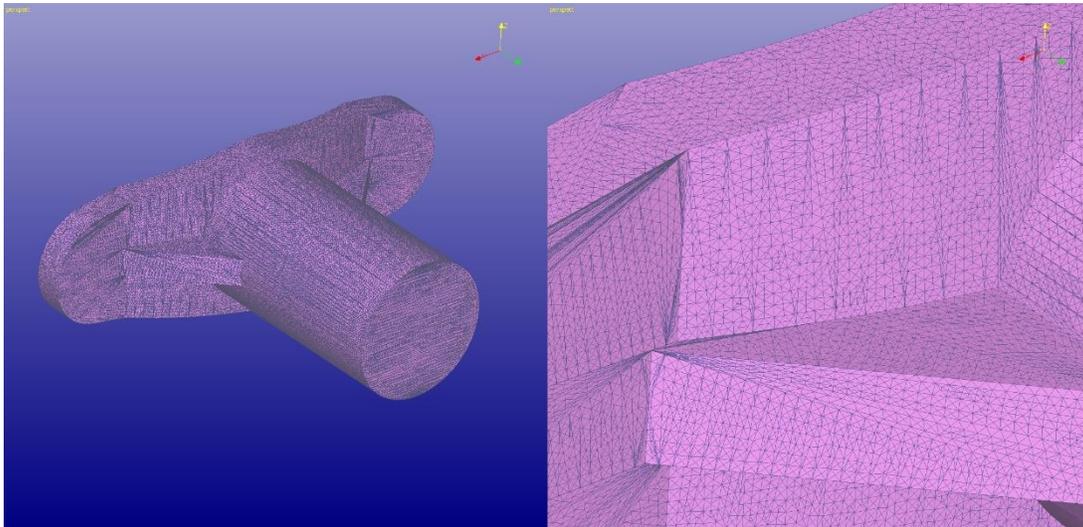
1.0 mm. In this case we will first refine the mesh to maximum 1.1 mm edges and then reduce it to 1.0 mm edges to get to the desired result.

- **Select the *rava2 (Reduced)* shell** from the Model Tree.
- **Give the command **Modify Faceted > Refine****, then fill in the following parameters (choose *Edge length* and set it to 1.1):



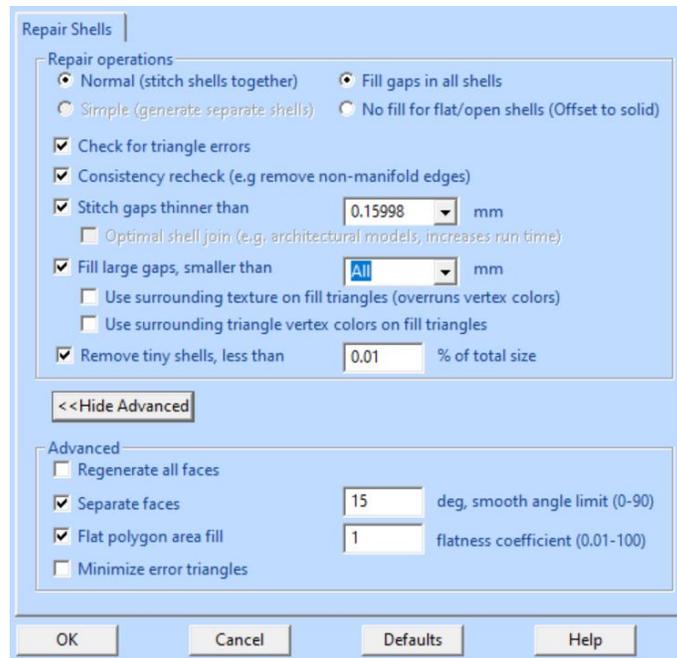
- **Press **OK** to start the **Refine** command**, after a while you will see the following result on the screen (next page).

The result now contains approximately 172 000 triangles. It is full of very thin triangles with bad aspect ratios. Anyhow, we now have a suitable set of new triangle corner points for further reduction, to generate new, evenly shaped triangulation (a close-up detail on the right).



The refine result does not contain surface information in it. To be able to produce more accurate reduction results with the Keeps surface edge filter we must regenerate the surface edge information to the model. To do this you should

- **run the **Fix Model > Repair Shells** command** with the following **default** parameters:

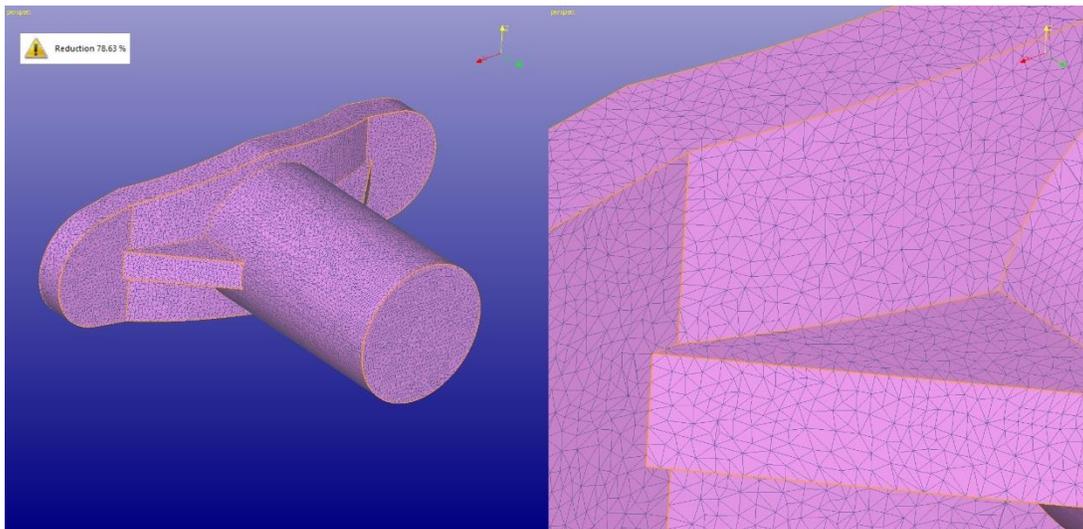


Also turn on *View > Show Edge Lines* to see the surface edges.

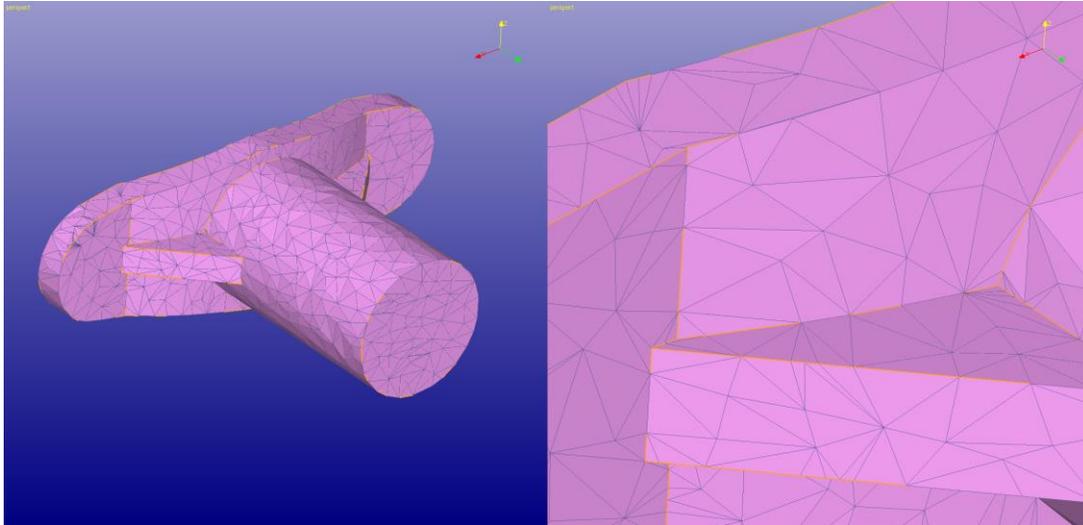
After **pressing OK** you will see the shell subdivided to surfaces again and we are ready to simply

- run the **Reduce** command on the **rava2** shell with the **Default** parameters.

After a while you will have the result with 78,6% reduction (you can run the result through *Fix Model > Verify Shells* to recalculate the edge lines). The result will contain approximately 37 000 triangles. If you look closer, the triangle mesh is well shaped all over the model ensuring accurate simulation results:



If you wish to get even less triangles but to keep the good aspect ratios, just rerun the Reduce command with a bigger ***minimum edge length*** value. For example, value 5.0 gives the following result of further 93.6% reduction with approximately 2300 triangles:



For the last reduction, you would achieve optimal results if you would use *Edge length* value 5.5 mms in the first Refine but this example shows that you can use very strong edge reduction on any kind of model and still get good results for simulation applications.

Conclusions

Thank you for going through these Tutorial lessons for 3Data Expert software. You can now use the 3Data Expert software modules for everyday 3D printing work or prepare surface and STL data for simulation software packages.

The 3Data Expert software contains a vast number of features not covered in this document. Please refer to the On-line Help and its examples for more information on the available functionality.

To learn more about the use of the STL and VRML handling available in the 3Data Expert software, please have a look at the advanced repair document: *3Data Expert 14.1: STL and VRML repair examples*. This document is available through the Help > Tutorial Documents command and the demo files are stored in the *Tutorials / GeomFiles* directory under the software installation directory.

We hope you will have a productive time with the DeskArtes 3Data Expert software in your everyday data preparation work. If you have any questions or comments on the use of the DeskArtes software products or this manual, do not hesitate to contact us at support@deskartes.com.