

DeskArtes

Expert Series 10.1

Tutorial 1 – Working with 3D Models for

- 3Data Expert
- 3Data Expert Lite
- Dimensions Expert
- Sim Expert
- View Expert

First Edition

Ismo Mäkelä: “Expert Series 10.1: Tutorial 1 – Working with 3D Models”.

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Foreword

The *Expert Series 10.1: Tutorial 1 - Working with 3D Models* document provides an introduction to the basic 3D model handling with Expert Series software products: 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.

Expert Series 10.1 is available as a 64 bit implementation allowing you to handle huge models for Additive Manufacturing. This and the ability to connect directly to AM processes via user defined plugins make your everyday data preparation as easy as you can imagine!

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

- *Expert Series 10.1: Quick Start Guide* gives a short introduction to model repair and splitting for 3D Printing with 3Data Expert, 3Data Expert Lite and Dimensions Expert products. Estimated completion time 15 minutes.
- *Expert Series 10.1: Tutorial 1 – Working with 3D Models* introduces basic viewing, measuring, automatic repair, split, connect, hollow, offset, 3D Text, Boolean, cut, reduce, surface triangulation and other model manipulation functions available for STL files. Applies to All Expert Series modules. Estimated completion time is 3 hours.
- *3Data Expert 10.1: Tutorial 2 – Working with Colors* introduces basic automatic repair and coloring for 3D Color Printer users. For 3Data Expert only. Estimated completion time is 60 minutes.
- *3Data Expert 10.1: Tutorial 3 – Separating Shells* describes the use of Shell Separation commands in 3Data Expert. These commands enable separating a part into several shells for different material (shore value) or color definitions in a single part. For 3Data Expert only. Estimated completion time is 15 minutes.
- *Expert Series 10.1: STL and VRML repair examples* document includes more advanced model repair operations with 3Data Expert. Applies to 3Data Expert module and mostly to 3Data Expert Lite, Dimensions Expert and Sim Expert modules. Estimated completion time with 3Data Expert 2 hours.

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

Additional information and help on other STL manipulation tools can be found in the Handling Faceted Models 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 be able to generate a correct result. Terms given in *italics* refer to file names, commands, shell names, surface names and parameters for different commands.

Lessons

The lessons in this tutorial are organized in such a way that that the required software functions increase along the lessons. The lessons can be started with View Expert, continued with Dimensions Expert or 3Data Expert Lite and finally requiring 3Data Expert to complete the last two lessons (or Sim Expert, specializing for simulation applications). Of course, it is best to use 3Data Expert from the first lesson, if available.

The first lesson with this tutorial shows you how to do basic file input, viewing and dimensioning with Expert Series. The first lesson shows the functionality common to all Expert Series modules and can be done with any of them, like View Expert.

The second lesson fixes an erroneous STL model for Additive Manufacturing using the Auto Repair process available with Expert Series after the model input. All modules except View Expert.

The third lesson fixes an erroneous STL model for 3D Printing using Repair Shells command with Expert Series. All modules except View Expert.

The fourth lesson describes methods to split and combine STL parts for Additive Manufacturing. Large parts must be split and pins or lap joints have to be added to be able to attach the manufactured parts correctly with Expert Series. All modules except View Expert and Sim Expert.

The fifth lesson describes the platform definition for different 3D Printing systems and moving parts into platforms. All modules except View Expert and Sim Expert.

The sixth lesson shows how to add 3D Text on a part. Suites to 3Data Expert or 3Data Expert Lite modules.

The seventh lesson describes various tools in the Expert Series Print Preparation menu. These tools allow you to make z-compensations, hollowing, offset, drain holes and supports for stereolithography systems. This lesson requires 3Data Expert module but can be partly followed with 3Data Expert Lite.

The eight lesson explains how to perform surface verification and triangulation to produce a solid STL and surface file with 3Data Expert. This lesson requires 3Data Expert or Sim Expert module.

The ninth lesson will help you to prepare STL files for simulation applications. This includes reducing the number of triangles as well as improving the triangle aspect ratios. The color and texture preserving reduction command can also be used for visualization applications with high real time requirements, like Augmented Reality, Virtual Reality and games. This lesson requires 3Data Expert or Sim Expert module.

DeskArtes Expert Series

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

- *View Expert* is entry level tool targeted for viewing, verifying, dimensioning and communicating 3D geometry information. Free View Expert allows you to input triangle models, create consistent topology, check errors, orient normals and output STL data for 3D printing without any cost.
- *3Data Expert* is 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, shell separation and boolean as well as STL coloring and texturing. Basic supports can be generated for both Stereolithography and metal processes.
- *Dimensions Expert* is a simple, economical and user friendly tool for STL repair and splitting for machines with a smaller build area.
- *3Data Expert Lite* adds primitive generation, boolean and 3D markup commands to Dimensions Expert functionality for connecting STL models as well as adding/engraving 3D text on STL parts.
- *Sim Expert* is a dedicated tool for preparing 3D models for Simulation applications, with triangle count reduction and triangle aspect ratio improvement functions.
- *Design Expert* is a tool for free form conceptual design for jewelery, ceramics and conceptual design.

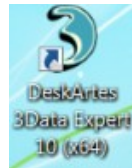
Lesson 1 – Basics User Interface properties

This lesson shows you how to do basic file input, viewing, dimensioning and verification of surface data with the Expert Series. This lesson can be performed with View Expert, Dimensions Expert, 3Data Expert Lite, 3Data Expert or Sim Expert module.

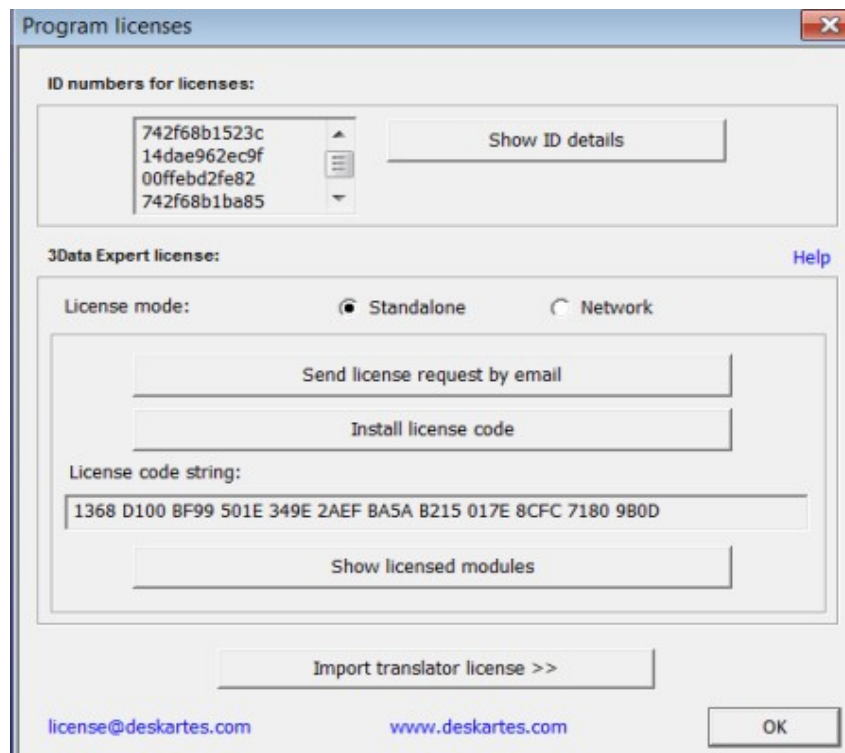
Green box indicates if the lesson is applicable to the given module.

 View  Dimensions  Lite  3Data  Sim

You will need a license to go through the lessons. 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 clicking on the icon created on the desktop during the installation.



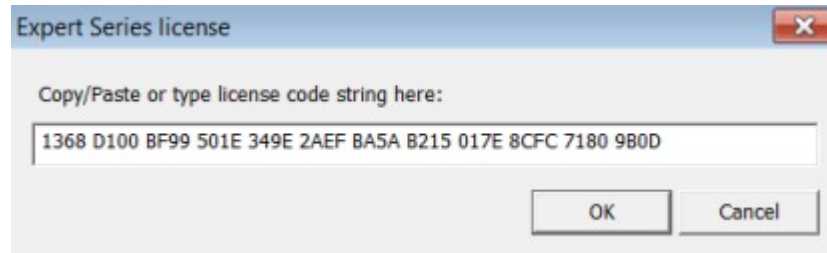
Then go to the *Program licenses* dialog through *Help > Licenses* command. There you can find the *Send license request by email* button. Press it and it will open your favorite email application to send the license request to DeskArtes.



Expert Series 10.1: Tutorial 1 – Working with 3D Models

You will receive the 30-day evaluation license within 24 hours.

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



Then press *OK* as requested 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 the Expert Series 10.1 earlier in your computer you may want to delete the settings files (settings3DE9010das, settignsDIME10.das, ... depending on the application in use) in the user home directory and then 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 for settings file for each Expert Series software.

Note: Free View Expert functionality does not require a license. With Free View Expert you can input several faceted file formats (STL, PLY, ZPR, VRML, ...), create topology for the model, check errors, repair surface normals and output the checked data for 3D Printing (STL, VRML, PLY, ZPR). Or you can communicate the errors with the other Free View Expert users or users of other Expert Series modules, like 3Data Expert. For more serious repair work and model manipulation, like split, connect, hollow, drain hole, support etc. with other Expert Series modules a license is required.

Sandbox mode exceptions

Some Expert Series functions, like *Drain Hole* and *Hatch Supports*, are created using a separate background process started from Expert Series. In this case some anti virus 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 anti virus software interface where you can change the run mode from “Sandbox” to “Normal”.

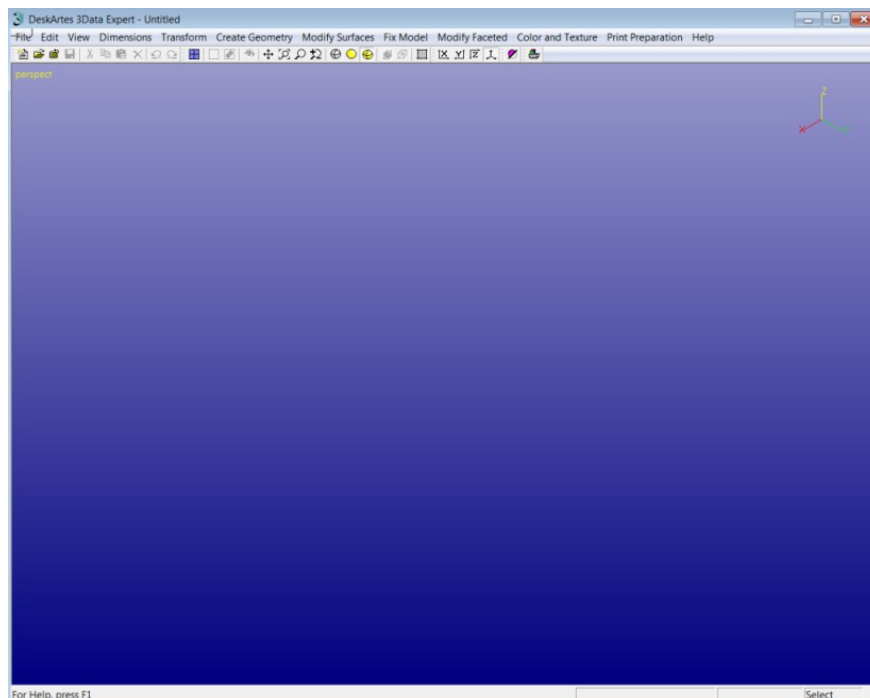
File input

In the sequel we will use 3Data Expert as the example program. For other modules, like Dimensions Expert, the file paths are identical except for the module name, where 3Data Expert is replaced with Dimensions Expert, for example.

You can start the **DeskArtes 3Data Expert** by clicking on the icon created on the desktop during the installation. If no desktop icon is created, use 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 including menus, different toolbars and the display area. The menu selection varies depending on the Expert Series software in use.



The file input is normally done through standard Microsoft File Open dialog with *File > Open* command.

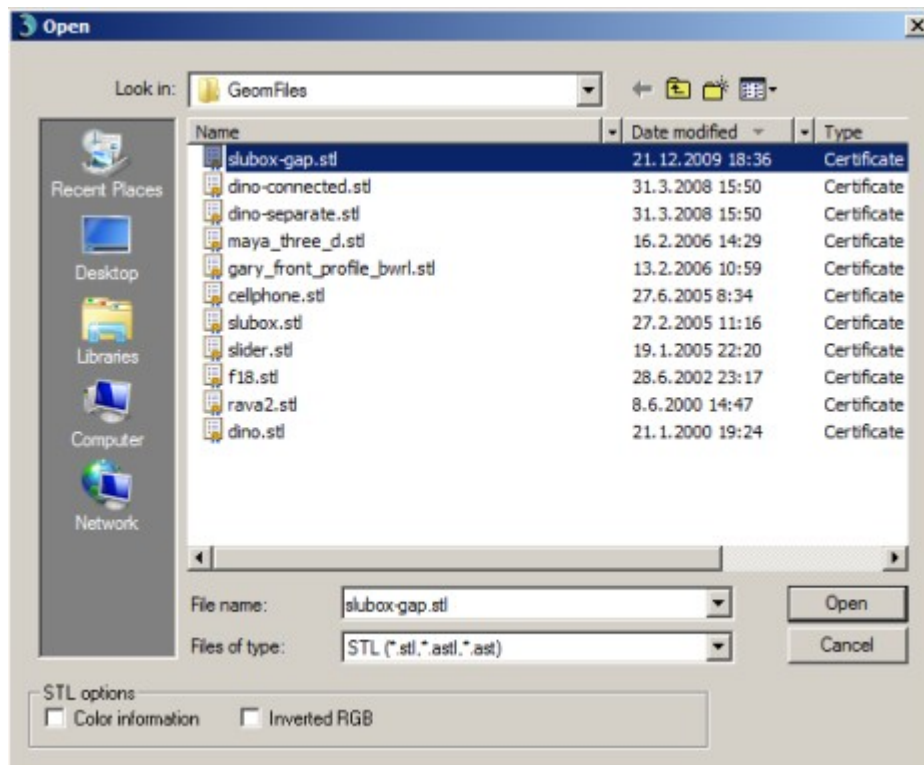
Expert Series 10.1: Tutorial 1 – Working with 3D Models

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

Commands given in **boldface** must be run to get the desired result for the lesson.

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

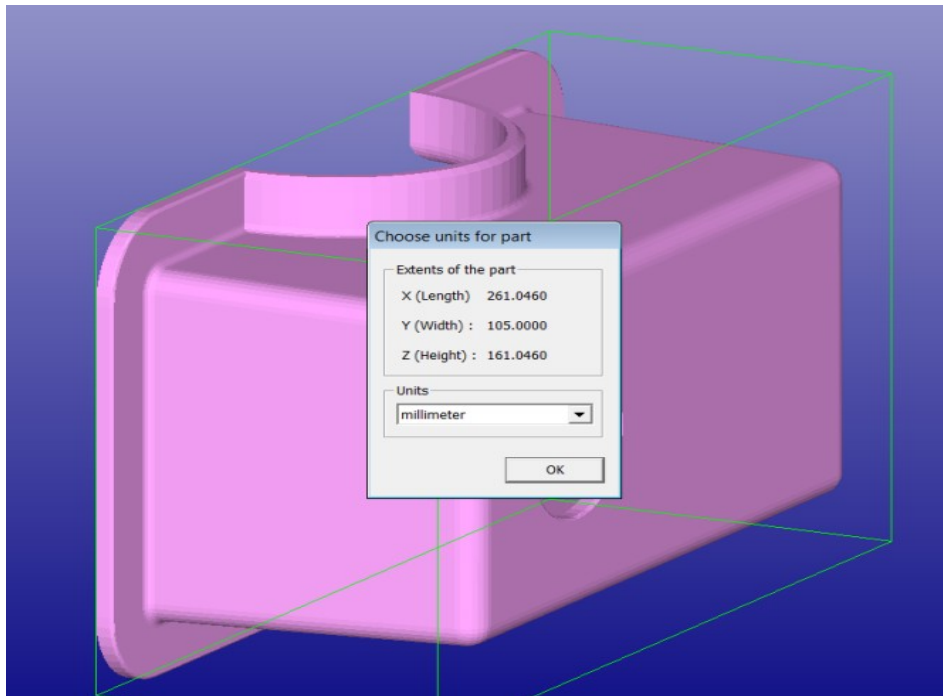
- **Select type STL files (*.stl)** and you will see the list of STL files in the *GeomFiles* directory.
- **Select *slubox-gap.stl*** from the file list and press the **Open** button.



After a while you should see the *slubox* geometry on your screen, as seen on the next page.

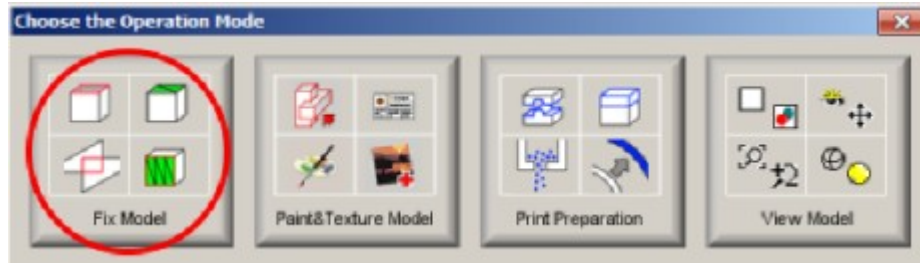
When a model is inputted 3Data Expert allows to check and set the units to the part. For example, STL files do not have any unit information in them. Possible choices are *Inches*, *Millimeters*, *Centimeters* and *Meters*.

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



Next you will be prompted for the *Operation Mode*. 3Data Expert has four main operation modes: *Fix Model*, *Paint & Texture Model*, *Print Preparation*, and *View Model* mode. You may have different set of Modes available depending which module you are running. The different modes are used for different tasks:

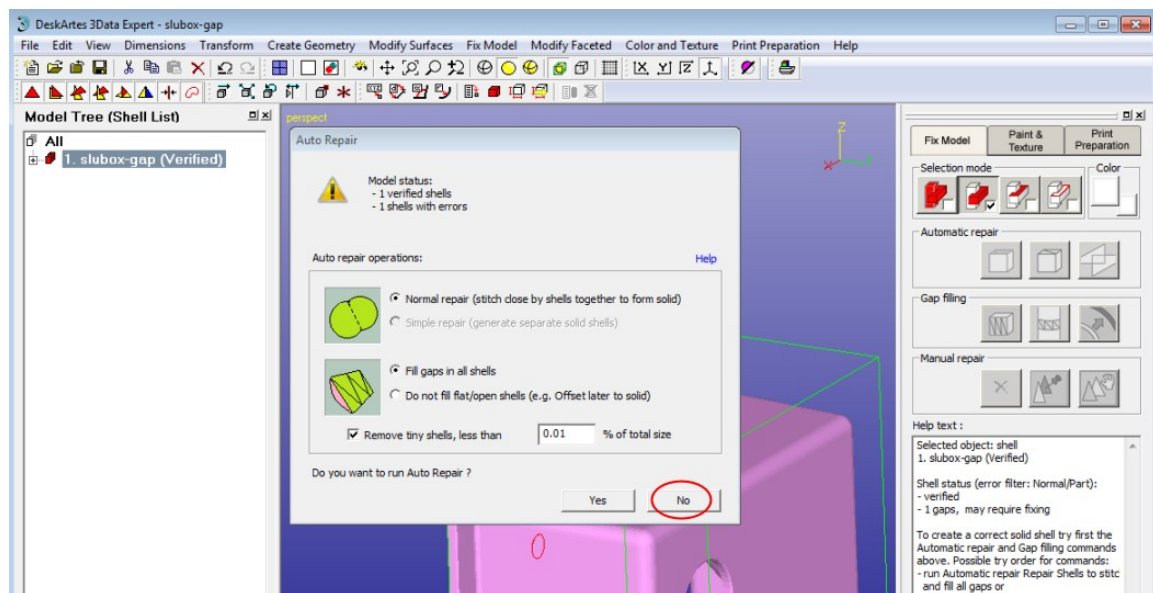
- *Fix Model* takes you to automatic model verification and repair, and contains shortcuts to the main tools used for repair work;
- *Paint & Texture Model* displays the tools for painting and texture image addition. This mode also checks the model for errors when selected and sets the display into “what you see is what you get” display color mode.
Note: this mode is only available with 3Data Expert;
- *Print Preparation* shows tools for model manipulation, such as hollowing and splitting. This mode also checks the model for errors when selected.
Note: this mode is not available in Sim Expert;
- *View Mode* allows you to temporarily view the model and then return to Operation Mode selection later (or to close the minimized window).



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

- **press the Fix Model button** (in red circle above) in the Operation Mode dialog now.

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, non verified 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 can not see the gap.

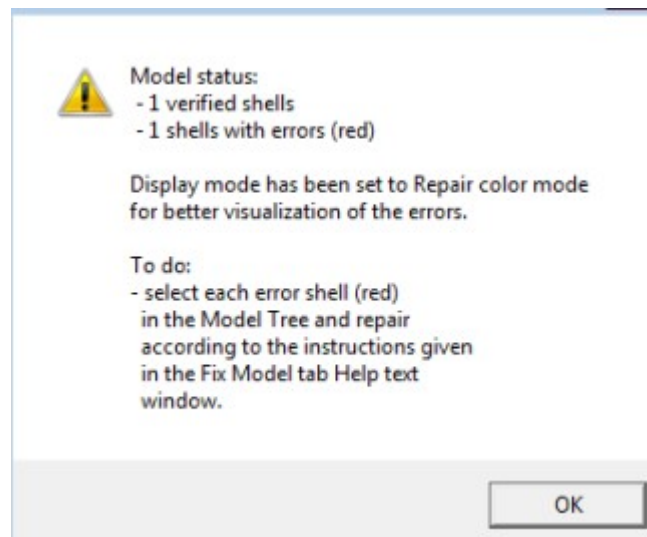
On the right hand side of the graphics area you will see the *Tools Window*. *Fix Model Tab* on the *Tools Window* provides shortcuts to the commands available for repair, like Repair Shells, Offset, and Fill Gaps commands. The *Help text* window at the lower part of the Tools Window also contains information for 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 normally should accept them as they are. Later we will see examples where the automatically set parameter values are changed. Anyhow, this time we are not going for full repair immediately (this is done in the Quick Start Guide), instead we will now investigate the Model Tree, Tools Window and selection mechanisms available in the software.

So

- **press the No** button (in red circle above) to start viewing the model now.

A status window appears:

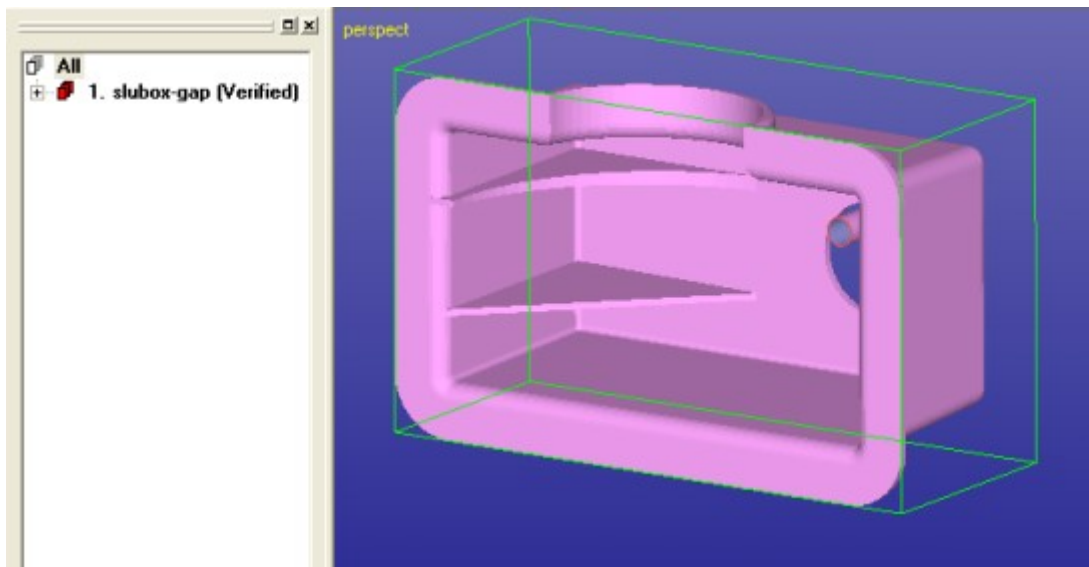


- **Press OK** to continue to see the *Tip Window* with useful tips on viewing and selection mechanisms.



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

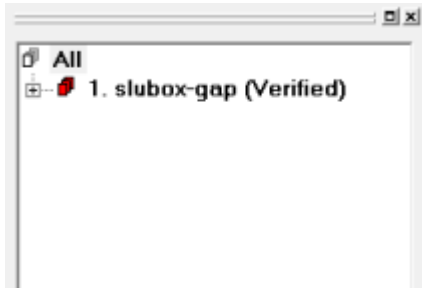
After closing the Tip Window you will see the model on the graphics area:



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. Through the gap we can see inside the part. The outside (normal side) of the surfaces is displayed with *pink* color and the inside (inverted normal side) is displayed with *blue* color in the default *Repair color* mode.

Model Tree

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



This tab tells you the current contents of the program database. With this file 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.






At the bottom of the Model Tree you will see the number of Shells, Surfaces and Triangles in the model as well as the extents along different axis. We will later take a more careful look at the contents of the Model Tree.

You can press *F8* to open and close the Model Tree window.

Basic viewing commands

Try the following commands to become familiar with viewing operations. Note that the commands below do not alter the model's true 3D coordinates. They only alter the eye-point location:

- **Rotate** the model on the screen by pressing the *middle mouse (MMB)* and moving the cursor. MMB rotation is always available. *Left mouse button (LMB)* rotation is available most of the time except when working with painting, texturing, curve editing, and transformations (LBM is mostly used for selecting objects as shown later).
- **Rotate** the model around a selected surface point by pressing the *right mouse button (RMB)* and moving the cursor. A blue cross indicating the rotation middle point appears on the part surface if the cursor is on top of the model when the *RMB* is pressed.
- **Pan** the model by dragging with the *Shift + MMB* (or *LMB* most of the time) (*Shift + MMB* means pressing the Shift key and dragging with the mouse).

- **Pan** with rubberband by dragging with *Shift + RMB*.
- **Zoom in/out** by rotating the *mouse wheel*.
- **Zoom in/out with** *Ctrl + MMB (or LMB)*.
- **Zoom to box** with *Ctrl + RMB*.
- **Hide all** by pressing the  icon with LMB (or *F5*)
- **Show all** by pressing the  icon with LMB (or *F6*)
- **Fit the model** by pressing the  icon with LMB (or *F7*).


The main viewing command icons reside in the Viewing Toolbar:



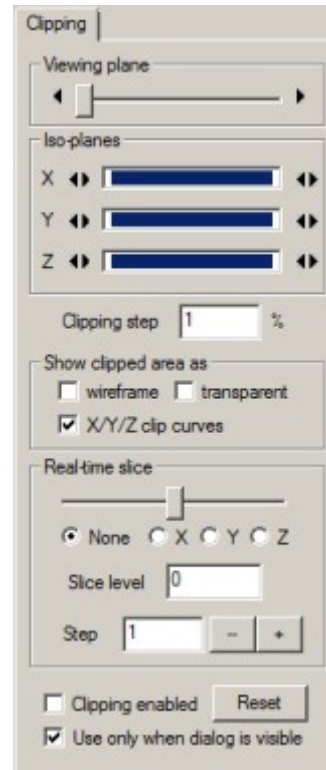
Try them out now. The tooltips provide a short description of each icon. Let the cursor remain on top of the icon for a couple of seconds to see the tooltip. Please note that some icons will perform different operations when pressed with *RMB* or *LMB*:



Also, the Viewing Planes x/y/z toolbar above allows you to select any projection direction as the viewing direction (try both *LMB* and *RMB* clicks).

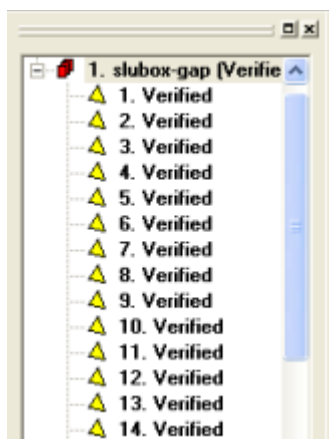
Try also Clipping functionality through the Clipping Tab to look inside the model. The Clipping Tab is opened with the  icon. The clipped area can be displayed with different ways, like with wireframe or transparent.


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

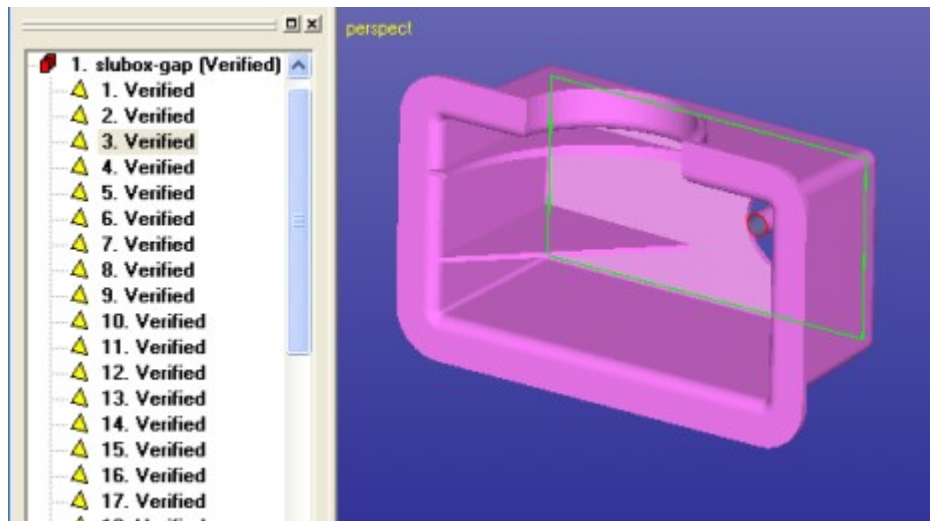
The object selection can be done either from the display area or from the Model Tree. **Erase the screen and then display the object node *1. slubox-gap (Verified)* again** by clicking it in the Model Tree. Now we have done one selection at the shell level. This level can also be called as the part or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



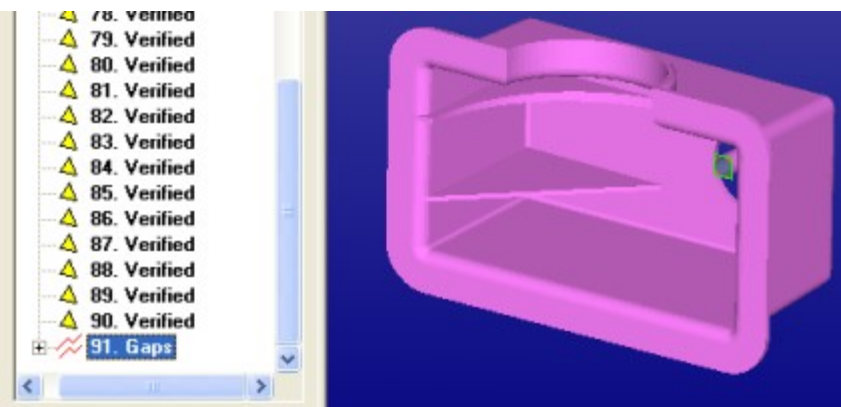
The node *1. slubox-gap (Verified)* has opened and shows the contents of the *shell*. Each shell (*1. slubox-gap (Verified)*), consists of several leaves, *surfaces* (*1. Verified, ...*), which all together make up the whole slider. Triangle surfaces are identified with the icon . STL model is subdivided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

The selected surface is also highlighted on the screen. The surface number 3:



The model contains errors, one gap. The gap data is also located in the Model Tree after the surfaces (please see next image). 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

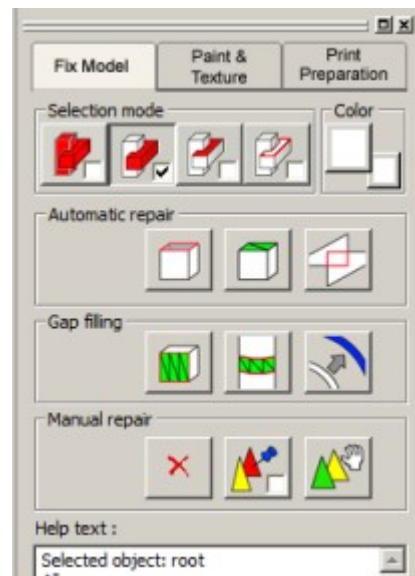
- **display the Tools Window by pressing Shift+F8** (if not already visible):

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.

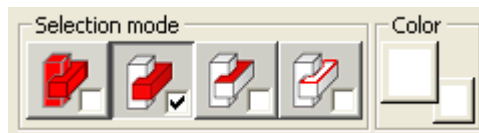
In the sequel we will use the *Selection mode* buttons to set the Selection Mode to correct level.

Note: The Tools Window is not available with View Expert software, you should use appropriate *Edit > Selection Mode* command from the Edit menu to set the correct Selection mode. The default selection mode is *Shell*.

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



During model repair, you can select target objects (whole model, individual shells or surfaces, triangles and gaps) to work on from the display area. 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 selecting the *whole model*, *one shell*, *one surface*, or *one gap/triangle*, respectively. Now try to select a shell from the display area, such as the *1. slider (IGES Input)*. The *Selection mode* is on *Shell selection* by default but if not **press** the



button to turn on *Shell* selection. Then **click with the LMB on the model** and the shell is 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.

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 available through the Paint and Texture tab are 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.

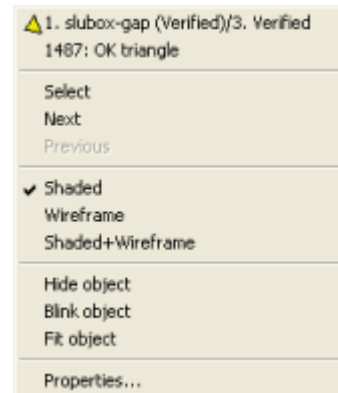
- **Press the *Surface selection mode*** button again.



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

When you click the RMB on the model while in the *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.

If other Selection mode, like Shell selection is in use, the RMB menu on graphics is shorter and gives as the first item the name of the shell at the mouse location.

Similar popup menu is also 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. Here we have a shell level selection, with some surfaces and triangles.

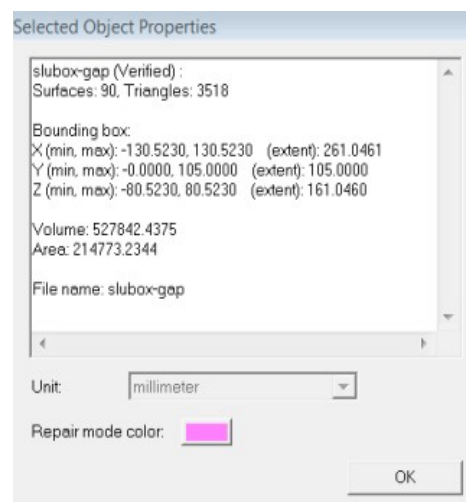


Shells: 1
Surfaces: 90
Triangles: 3518
Extents: X: 261.0461 mm
Y: 105.0000 mm
Z: 161.0460 mm

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**

Properties menu item opens a dialog with more accurate information of the selected part in the Model Tree.



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

Note: with this example the *Volume* is may not be correct. This is due to the fact that the model is not solid. The *Volume* is reliable with Repaired faceted models without gaps and for surface models with consistently oriented surface normals. The *Area* value is always valid both for surfaces and faceted models.

Dimensions and Annotations

For communication and documentation purposes it is many times 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 the **Shell Selection mode** for the next lessons with the



shortcut button.

Measuring and annotation commands area available through the Dimensions menu or through the toolbar:

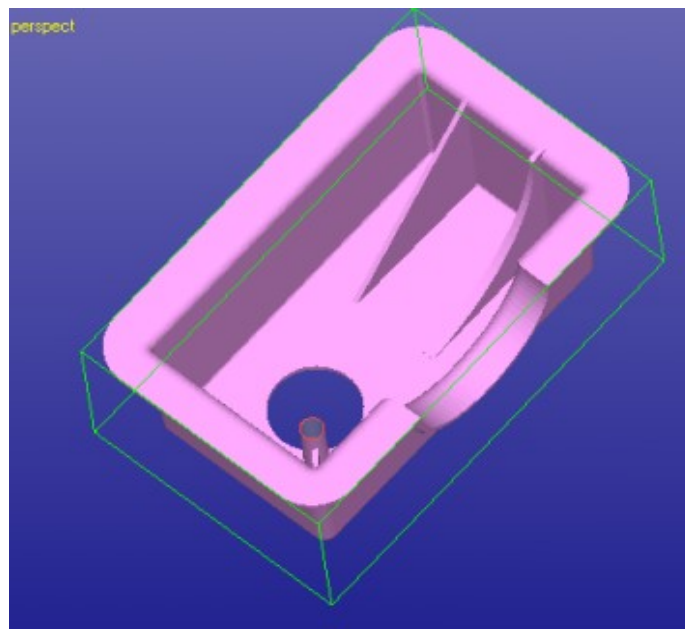


If the toolbar is no visible,

- give the command **View > Toolbars > Dimensions Toolbar**.

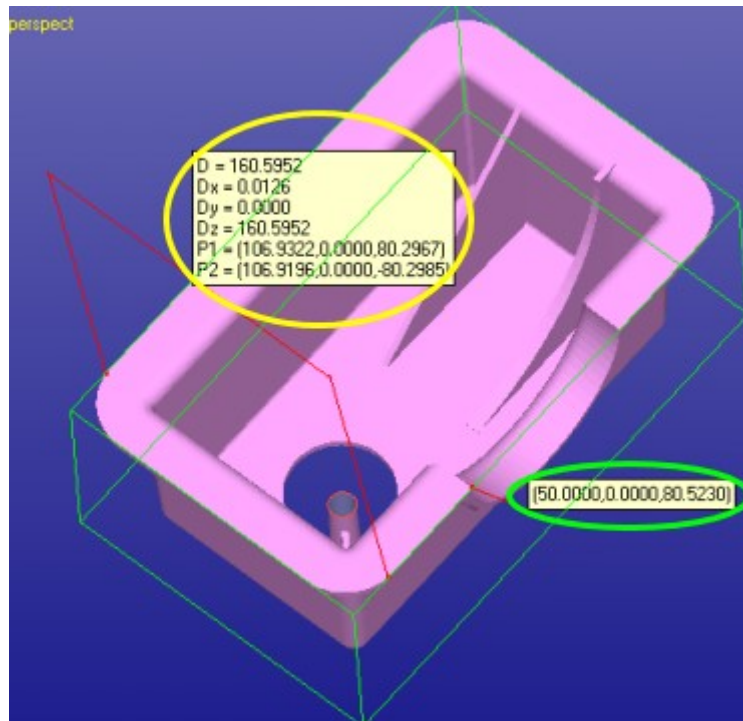
Let's first take an absolute point value from the *slubox-gap* model.

- **Rotate and zoom the model** so that you can see it as shown on the below:



- Now give the command **Dimensions > Point Value** ( 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 (in the green circle):



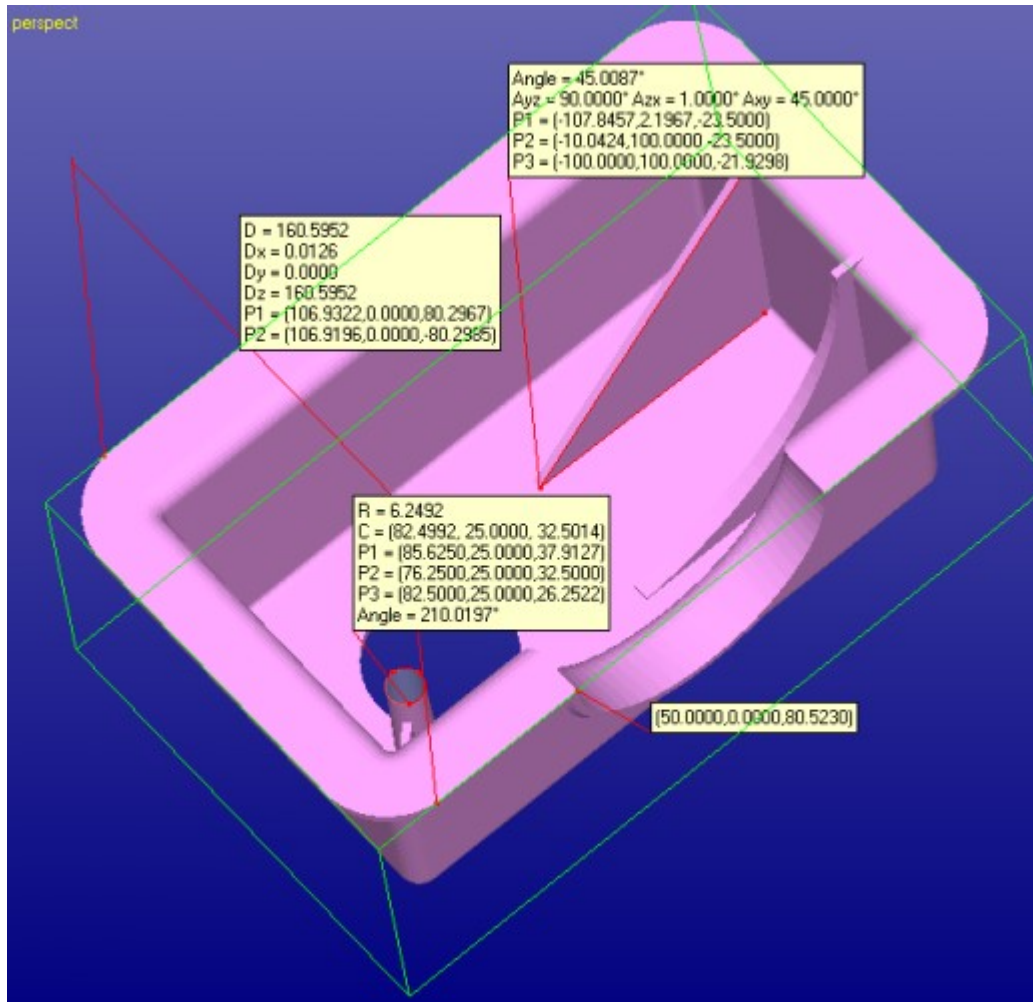
Let's now try distance between two points (yellow circle below). This is done with

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

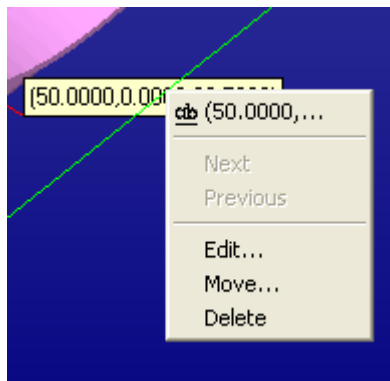
Pressing *LMB* only selects the closest surface point below the cursor for dimensioning. Pressing *ALT + LMB* causes the closest *triangle corner point* to be selected instead of a the surface point below the crosshair icon. This will enable selecting accurate triangle corner points for point value,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



Now you are ready to edit and move the annotations.


- 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, as seen below:




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This menu allows you to *Edit, Move and Delete* existing annotations. Try these commands on the selected annotation.

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 Excel, for example.

The dimensioning can be made *Sticky* through the Preferences tab (see *Edit > Preferences > General Tab*) and you can give as many *Point Values, 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.

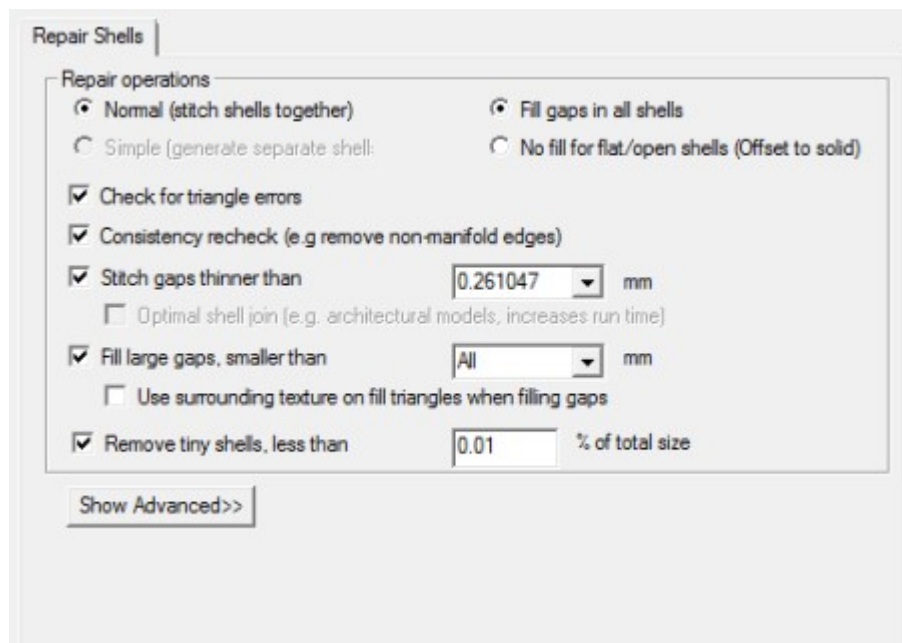
Repairing slubox model

Now it is time to repair the gap in the slubox-gap model. **Note:** this repair can not be done with Free View Expert, a licensed higher level Expert Series module is required.

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

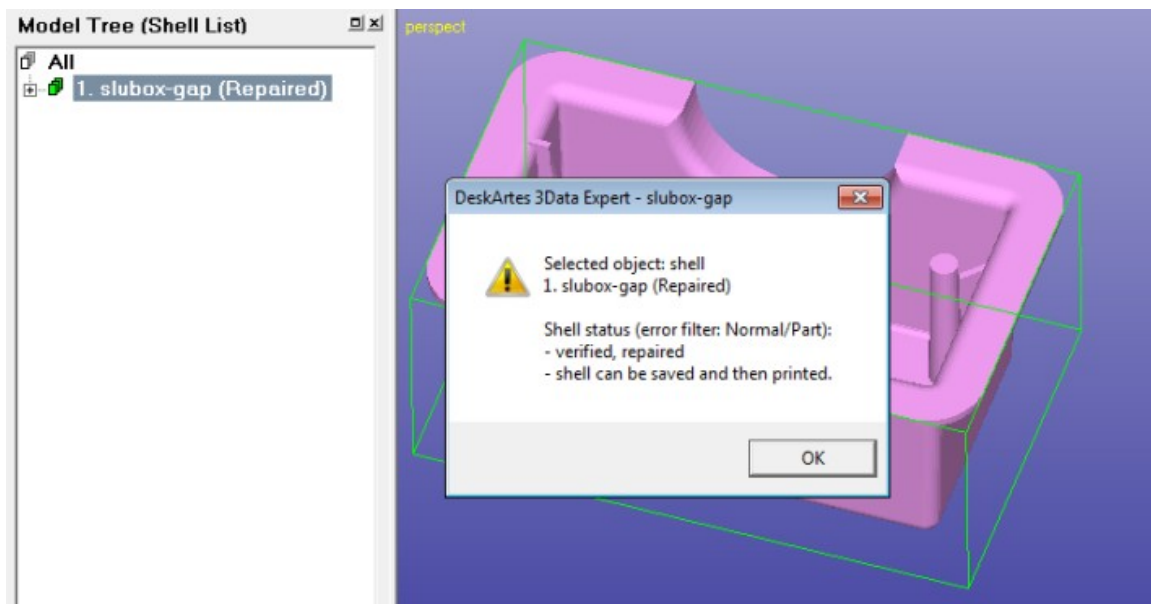
- **give the *Fix Model > Repair Shells* command and**

you will see the following parameter dialog:



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- Just **press OK** when the parameter dialog appears and you will have 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 Expert Series software.

Green box indicates if this lesson is applicable to given module:

■ View ■ Dimensions ■ Lite ■ 3Data ■ Sim

Clean up the software workspace before continuing by

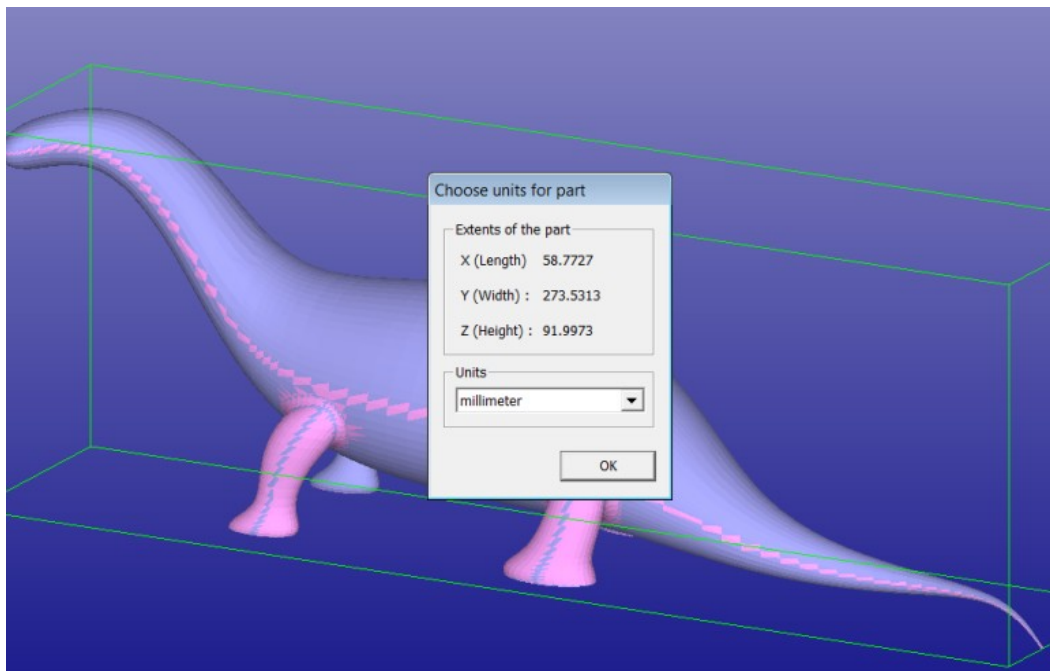
- giving the **File > New** command. Do not save the result of the previous example.

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.

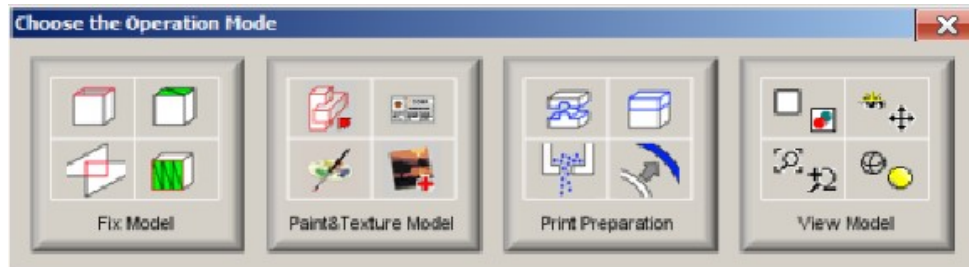
- **First open the *dino-connected.stl* file** in the *Tutorials / GeomFiles* directory.

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



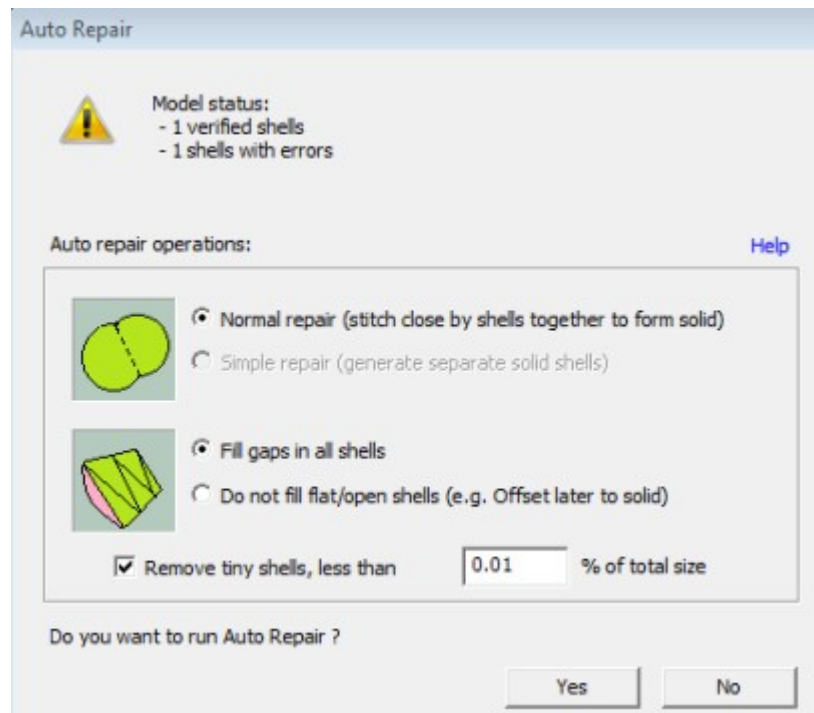
- **Press OK** to continue and

you will be prompted for the Operation Mode. 3Data Expert has four main operation modes: *Fix Model*, *Paint & Texture Model*, *Print Preparation*, and *View Model* mode (Operation Mode window may look different if you are running Dimensions Expert or some other module):



- To start fixing **press the Fix Model button** in the Operation Mode dialog now.

Expert Series will begin automatic model verification (*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

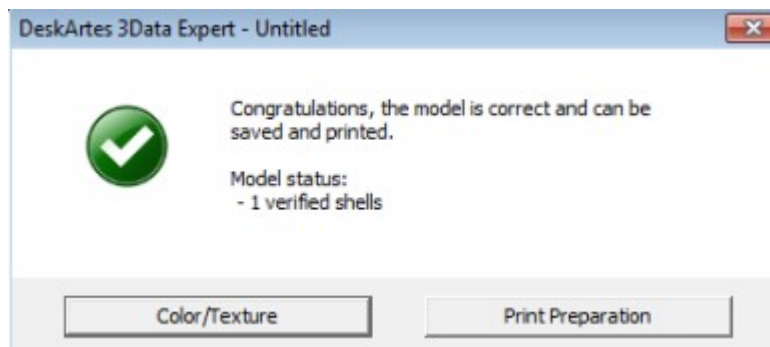


The message indicates that there is one verified shell, which also contains errors. The display shows red *Gap curves* through the surfaces. Gaps are an error type you must fix during the repair process.

- **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:



- **Press red Close button in the upper right corner** to close the window.

The model is ready for output and 3D printing.

A *Tip Window* with useful tips on viewing and selection mechanisms may be displayed after Auto Repair. After reading the tip you can **Close** the window:



During Auto Repair, Expert Series 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 Models”. 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 Model 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 Expert Series.

Green box indicates if this lesson is applicable to given module:

View Dimensions Lite 3Data Sim

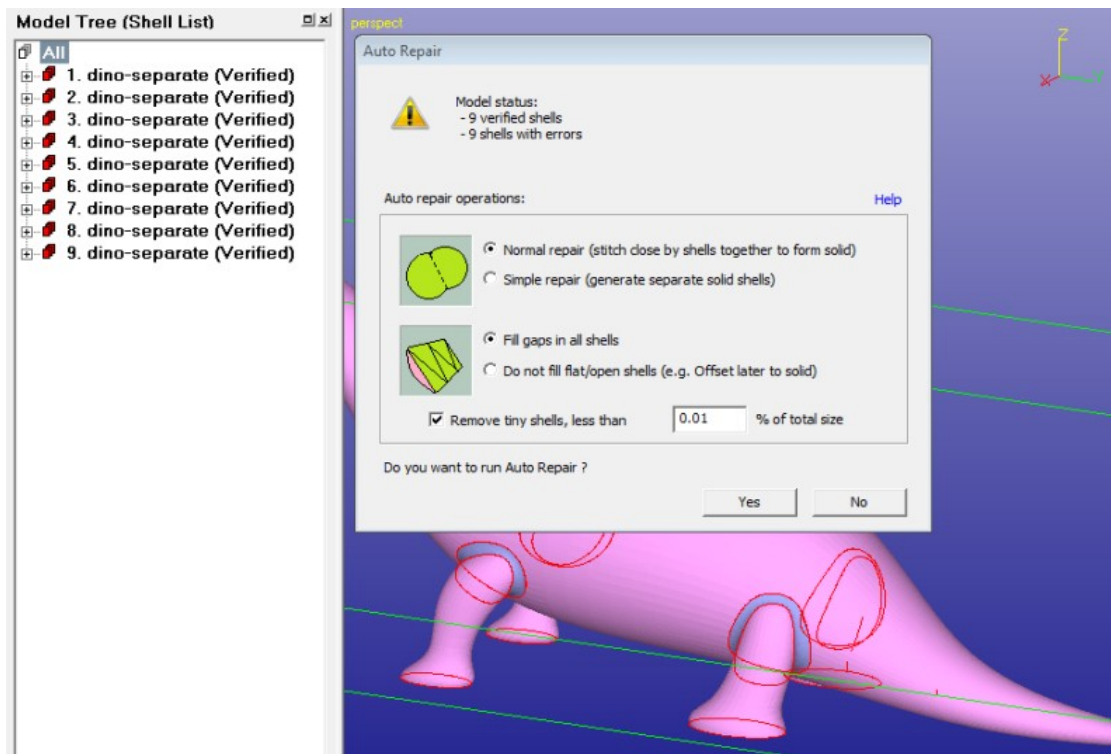
Verifying and repairing the STL model

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 *Tutorial > 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 Model mode.** After a while the software displays the Auto Repair dialog with Model status message:



Expert Series 10.1: Tutorial 1 – Working with 3D Models

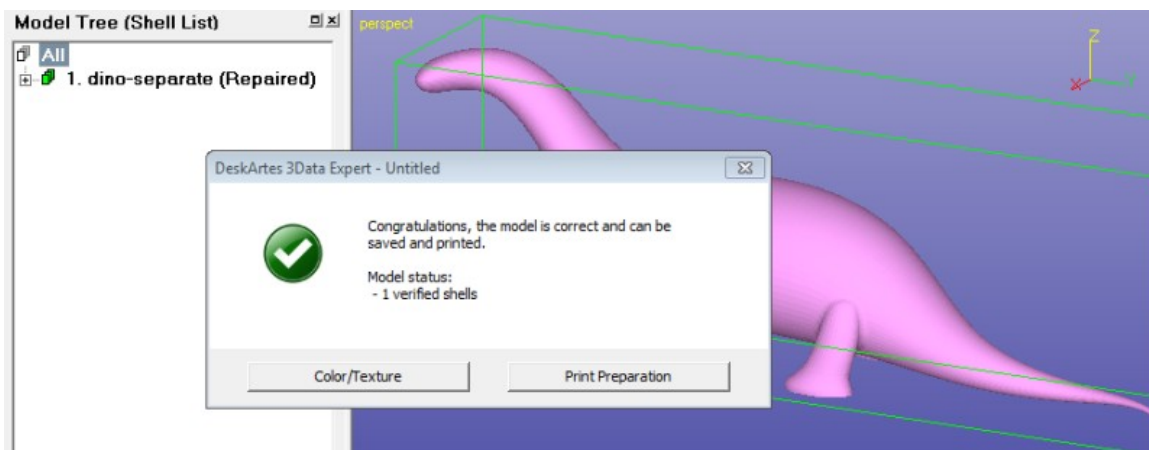
The latest automatic verification and fixing functions in Expert Series have now analyzed the model and propose the following repair actions:

- *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 there are now open/flat shells in the result which may require Offset or Extrude for proper fixing to 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 final 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

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

- **Press OK to close the model status window.**

Verifying and repairing, take two

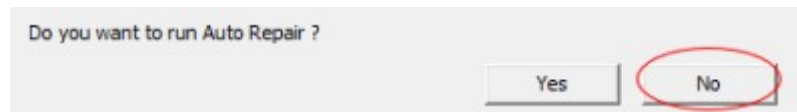
In the previous versions (10.0 and earlier) it was necessary to run the automatic repair for this model through *Fix Model > Repair Shells* command. Below the old way to fix *dino-separate.stl* file is repeated to emphasize the development with new Auto Repair process show above as well as to learn some basic concepts on model errors.

Restart the lesson by

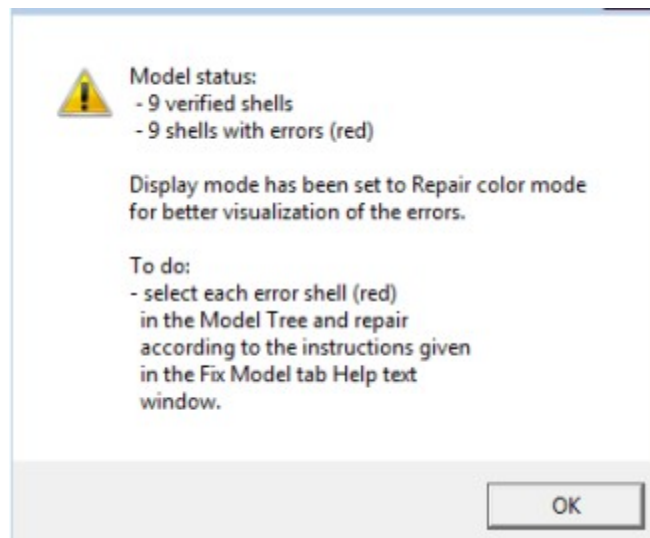
- **File > New** command
- **File > Open** the *dino-separate.stl*
- Accept millimeters
- start Auto Repair by **pressing Fix Model** in the Operation Mode window

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

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



A status dialog appears:

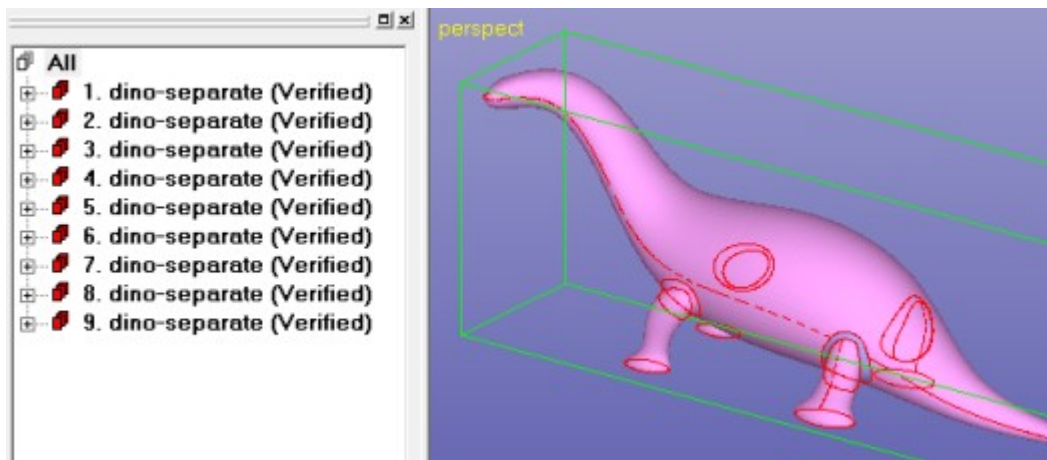


- When the status dialog is displayed, **press OK** to close it.

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

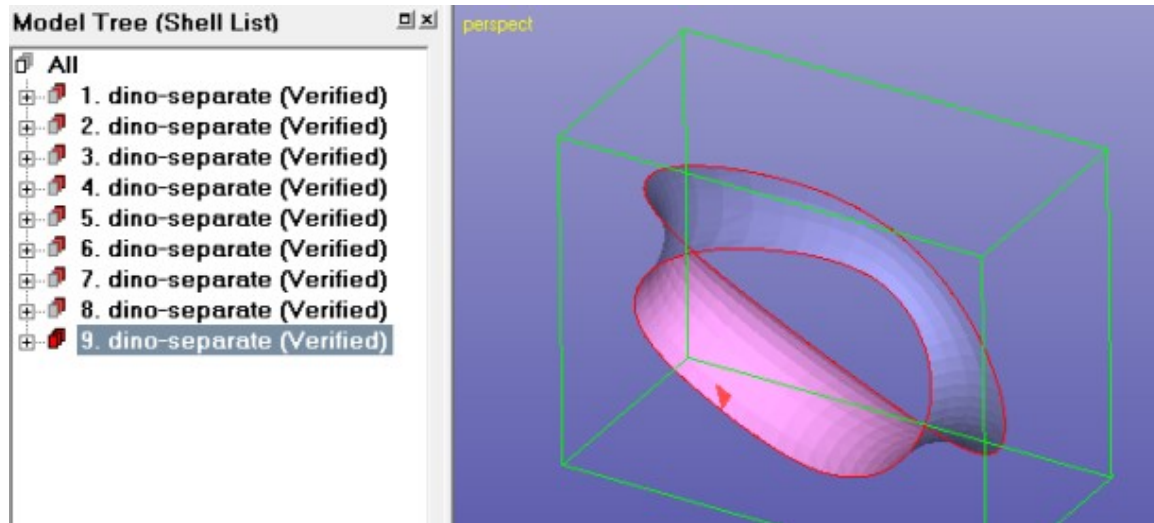
Using Repair Shells command

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 Expert Series you can easily repair these STL model errors:



The *dino-separate* model components are sorted according to the surface area in the Model Tree after the *Fix Model > Verify Shells* command. This helps us to delete minor obsolete components from the model before further operations by just checking the last small components in the Model Tree.

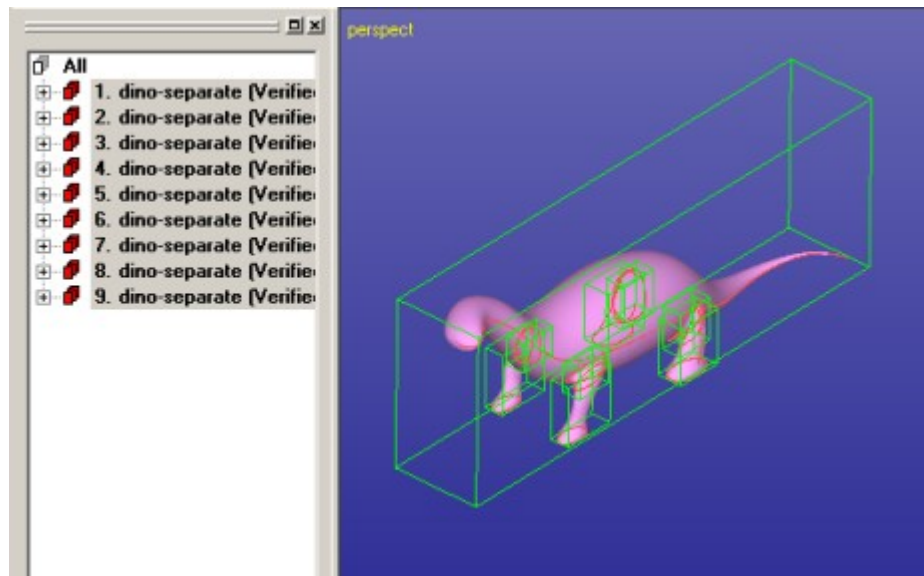
- **Select the last component 9. *dino-separate (Verified)* and Fit it on the screen:**



We can immediately see that this is an important components and must not be deleted. The components before the 9th are even bigger and thus must remain in the model.

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):



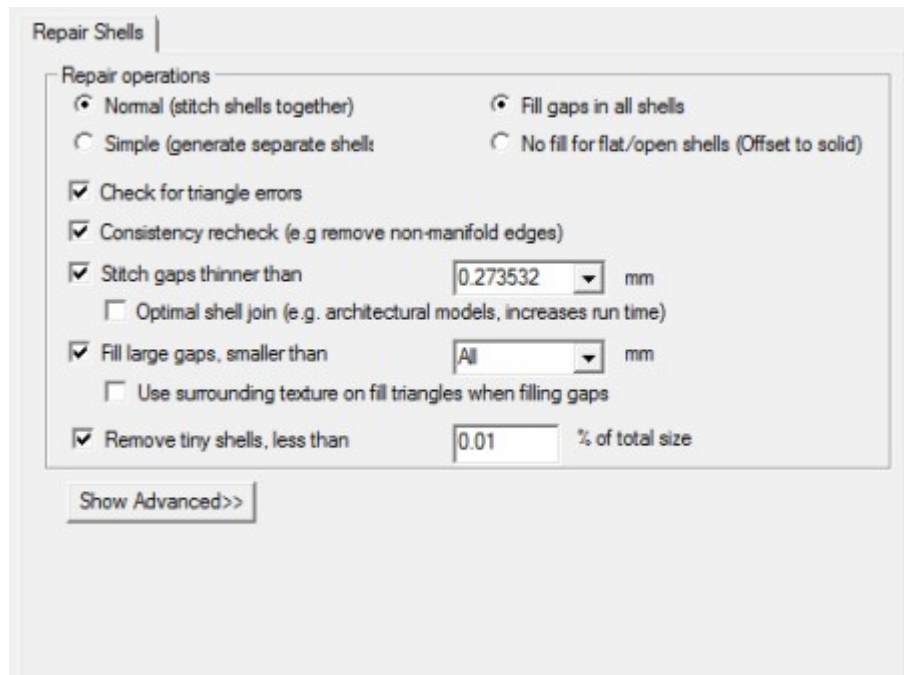
Then

- **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 below:

A screenshot of the 'Repair Shells' dialog box. The dialog has a title bar 'Repair Shells' and a 'Repair operations' section. It contains several radio buttons and checkboxes. The 'Normal (stitch shells together)' radio button is selected. The 'Fill gaps in all shells' radio button is also selected. There are checkboxes for 'Check for triangle errors', 'Consistency recheck (e.g remove non-manifold edges)', 'Stitch gaps thinner than' (with a value of 0.273532 mm), 'Fill large gaps, smaller than' (with a value of All mm), and 'Remove tiny shells, less than' (with a value of 0.01 % of total size). There is also a 'Show Advanced>>' button at the bottom.

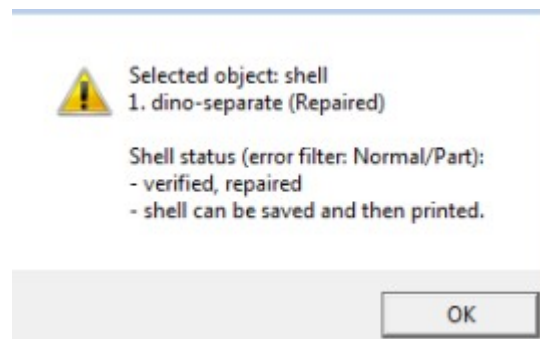
There are several parameters for the Repair Shells command. The parameters are automatically calculated for each model and can normally be accepted as is:

- *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 gap parameters. *No fill for flat/open shells* leaves flat shells unfilled. These shells may be better to fill with Offset or Extrude surface commands.
- *Check for triangle errors* to report intersections, multi-edge and overlap conditions
- *Consistency recheck* to check duplicate an overlapping surfaces and use optimal connection algorithms between these surfaces to form a solid shell.

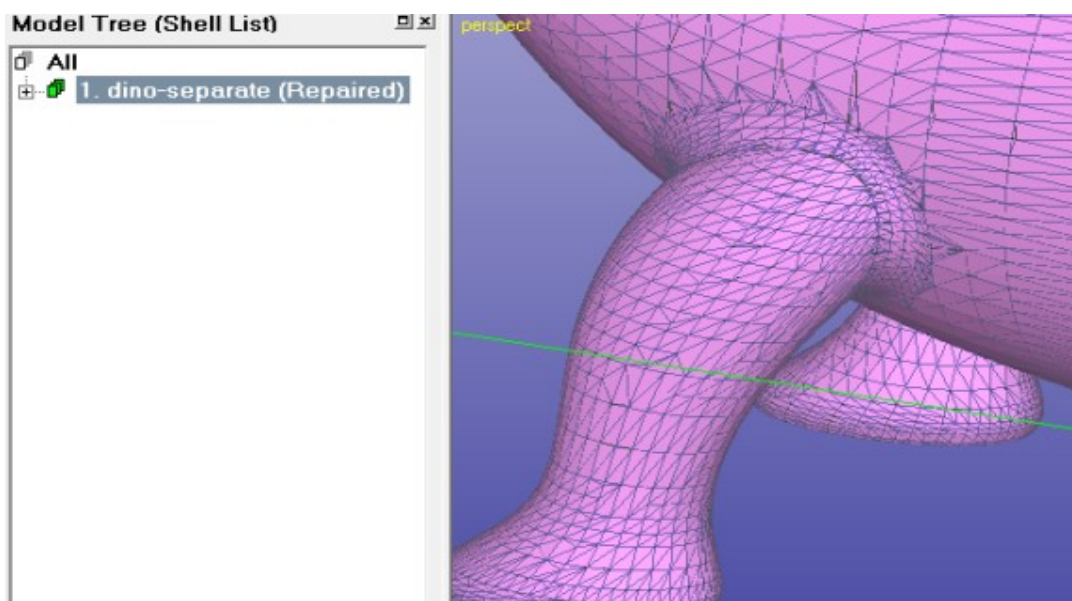
Expert Series 10.1: Tutorial 1 – Working with 3D Models

- *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. All to close all holes of the dino.
 - *Remove tiny shells* where the area/volume is less than 0.01 % of the total area/volume.
- check that parameters are as in the image above and then **press OK** to start Repair Shells command

After a while the *Repair Shells* finishes and the status is displayed:



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



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 10.1: STL and VRML fixing* 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 Rapid Prototyping. 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.

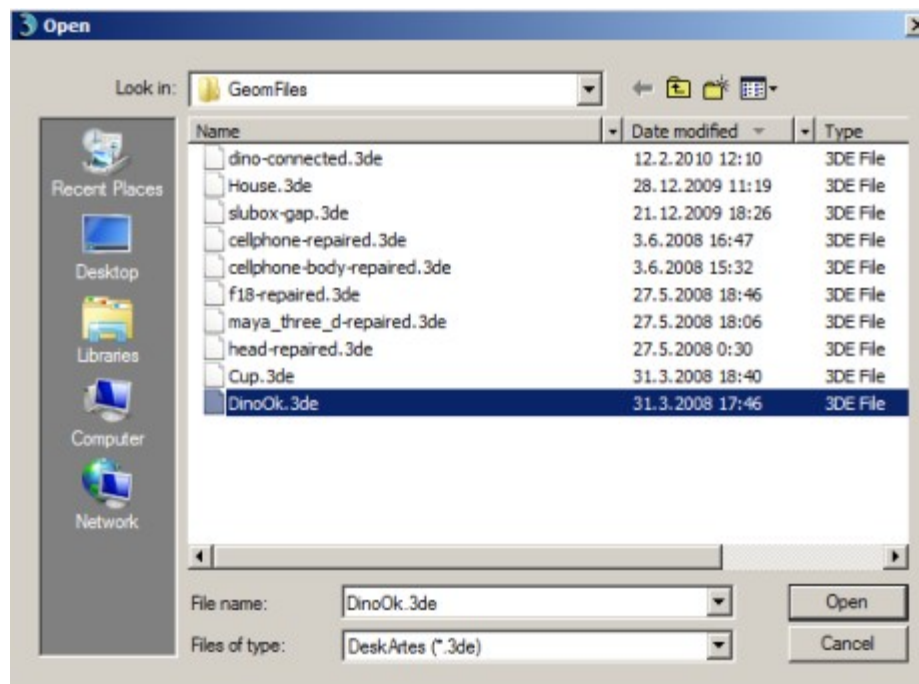
Green box indicates if this lesson is applicable to given module:

■ View ■ Dimensions ■ Lite ■ 3Data ■ Sim

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:

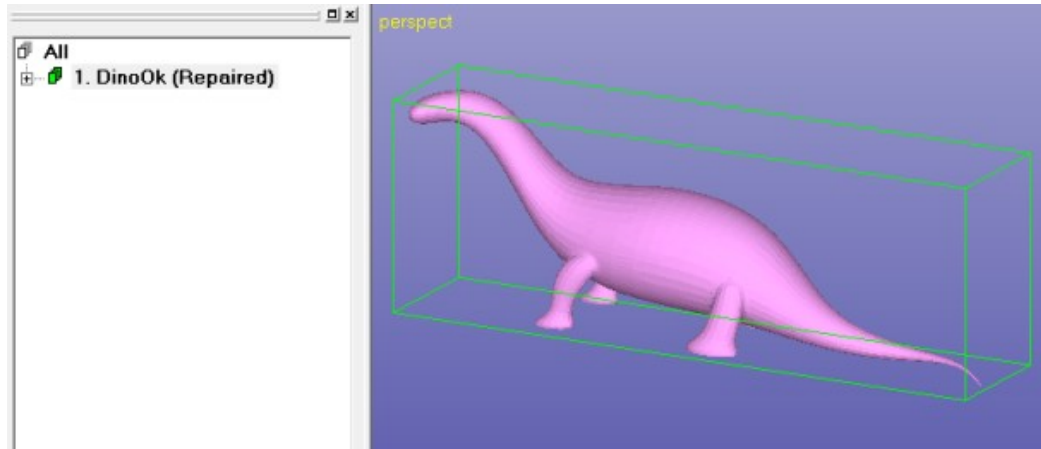


The *DinoOK.3de* file includes a correct triangulation for the familiar dino-model. You should accept the proposed units and

- go directly to the **Print Preparation** mode by pressing the



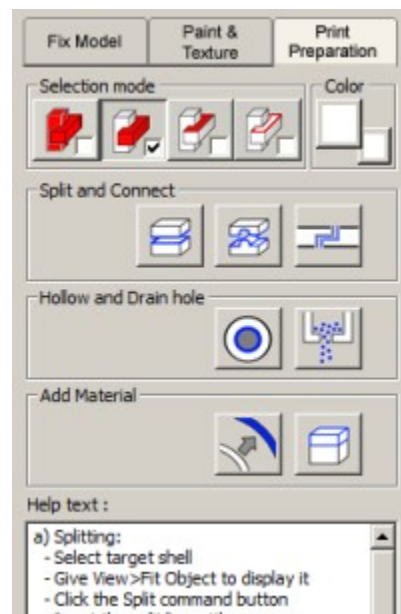
button in the Operation Mode Window. You should see the following dino model:



with the Print Preparation Tab on the right hand side of the screen.

The Print Preparation Tab gives shortcuts to the mostly used print preparation commands, like split, connect, add lap joint, hollow, offset and drain hole. The Help Text provides context sensitive help to the commands.

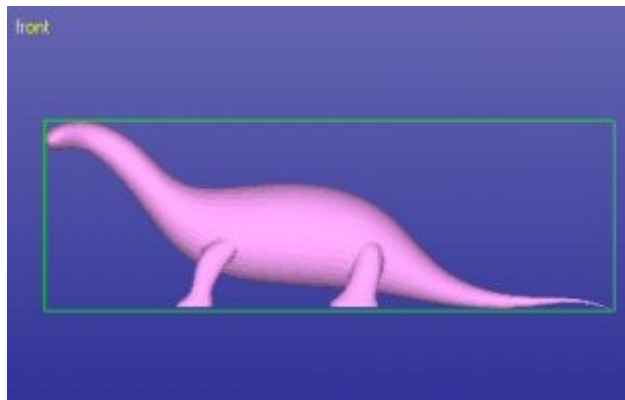
Note: The contents of the Print Preparation tab depend on the Expert Series module in use.



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

Before the 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,

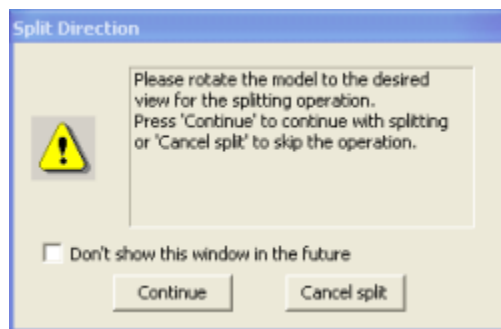
- **take a view from the x-axis direction with *View > Ortho View > Front (+X)*** ( + LMB).



The splitting is

- **started now with command *Print Preparation > Split*.**

The system prompts you to take the correct view direction if no yet set:

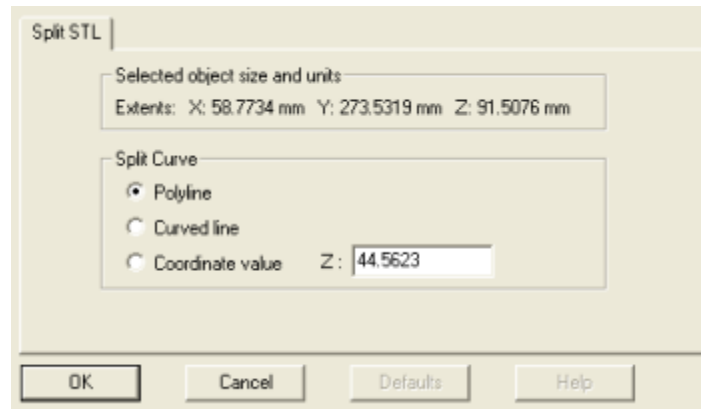


When the view is correct you can

- **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***

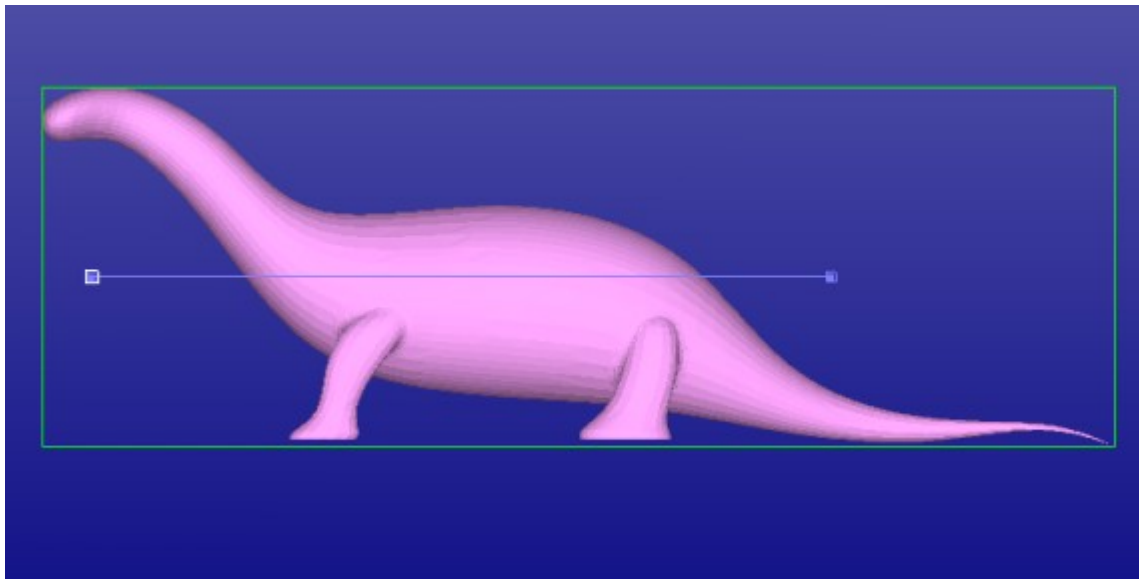


- **press OK** and the system goes into the curve edit mode where you can define the shape of the splitting surface.

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**

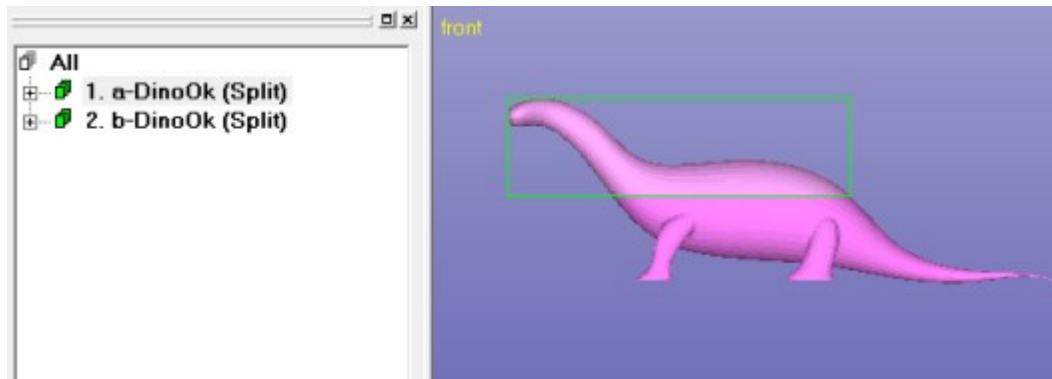
as seen in the image below:



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 first shell in the Model Tree to highlight the up halve:






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.

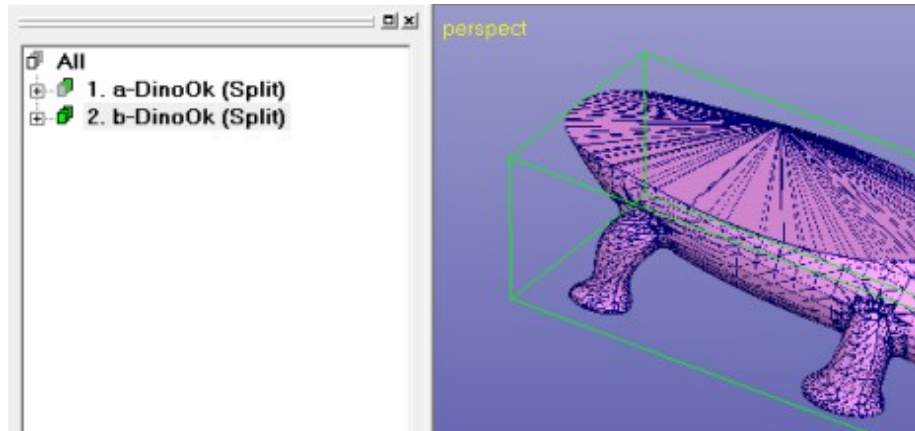
Creating the pins

The separate parts of the model could now be stored into .stl files for 3D Printing. However, the 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 ready from the 3D Printing.

First we will select and show location to the necessary primitives which will connect the two parts together. First we will get a nice view to point the pins on the 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, shell 2. *b-DinoOk (Split)***, it is displayed on the screen
- **set on shaded+wireframe mode with**  **icon**

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

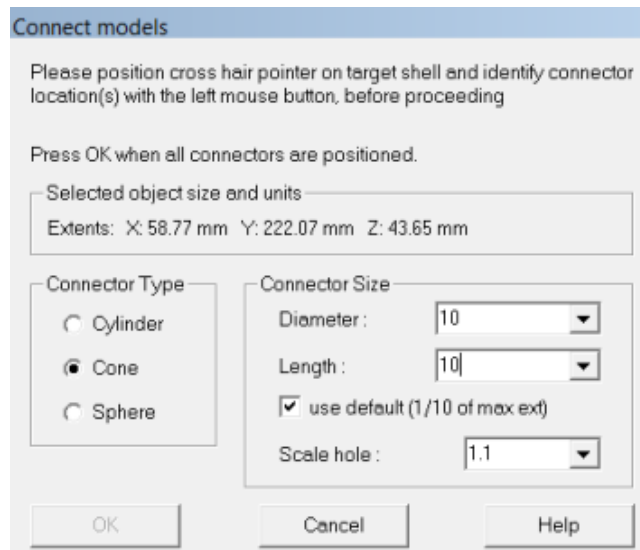


The pins are now

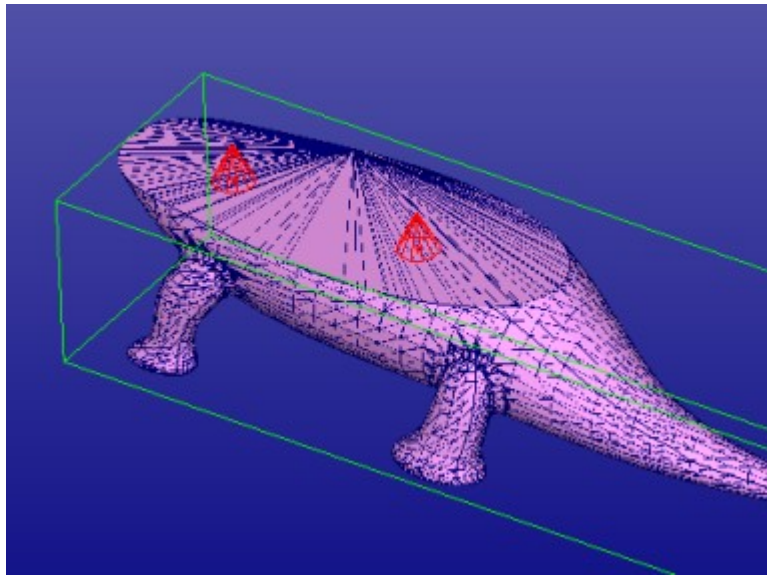
- added with the **Print Preparation > Connect** command or using the Print Preparation Tab shortcut



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




The positions for the pins should look similar to the image below.

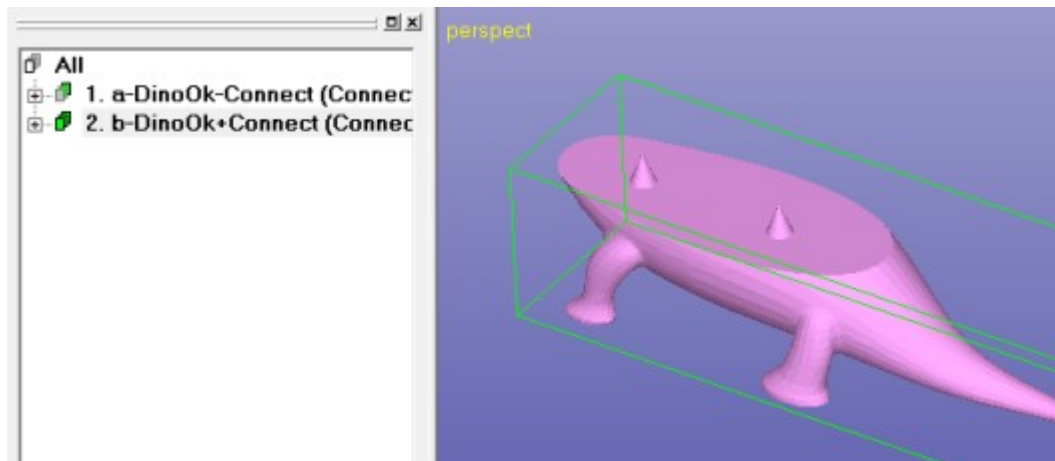


If a pin is incorrectly located on the shell you can remove it by clicking the red indicator cone again with the LMB. Then reposition the pin again with the LMB click.

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  icon in the Viewing Toolbar):



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

The connector is positioned on the surface so that 10% of the height is below the pointed surface for robust Boolean operation between the selected surface and the pins. 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.

Transferring to 3D Printing via Plugin

Also, if you want to use the **Print Preparation menu plugins** to transfer the two halves directly to another AM application, like some 3D Printer control software, it is good to position the halves correctly before the plugin execution.

To define and use plugins from DeskArtes software products, please see On-line Help *Menus and Commands > Print Preparation > New Plugin* page. This page explains how to create connection to 3D Systems ZPrint, Stratasys CatalystEX and Objet Studio software products, for example.

Using Lap Joint to connect thin parts

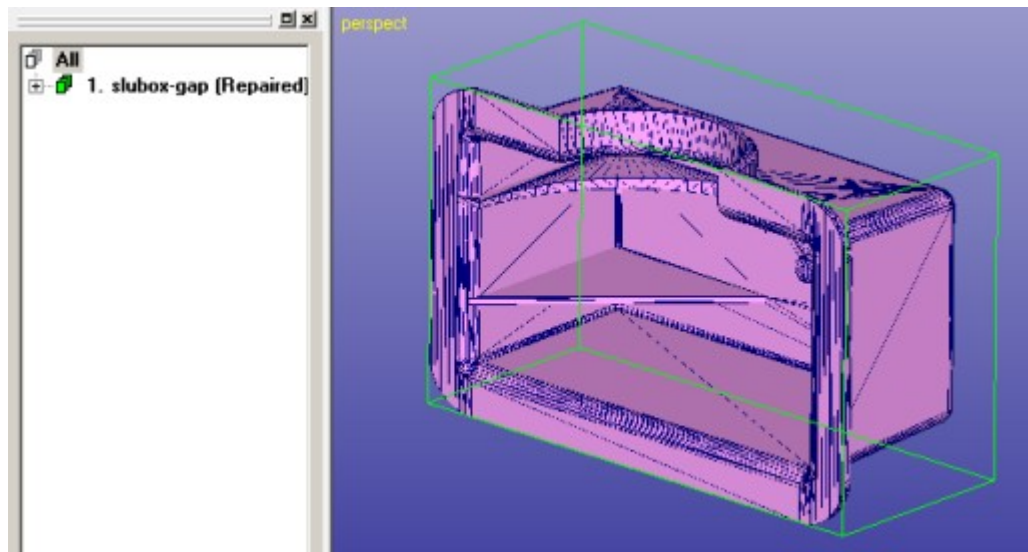
With this example we will split a part with thin walls and connect it with a *lap joint* connection. A lap joint is a small overlapping structure on both sides of the wall along the split surface edge. With Lap Joint command you can generate strong bond even along very thin walls.

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

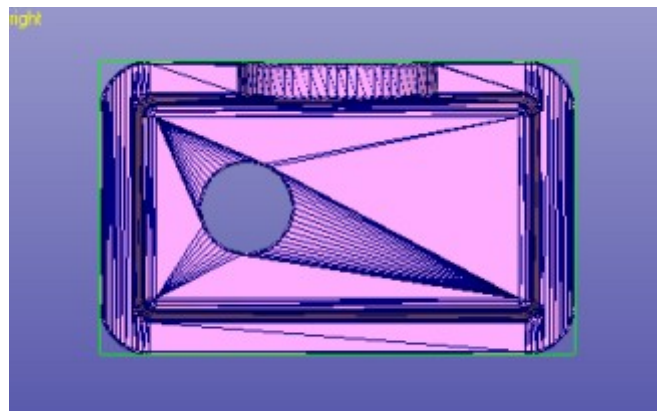
1. Input file *slubox-gap.stl* file with **File > Open**.
2. Accept millimeters as units.
3. **Press Fix Model** with Operations Toolbar to go to the Auto Repair process.
4. When asked “Do you want to run Auto Repair” **press Yes**.

5. And after a while you will get “Congratulations, the model is correct and can be saved and printed” message.
6. Press Close button in the upper right corner to close the status message window.

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



- Now take a suitable **view direction +Y for splitting** by pressing icon: 



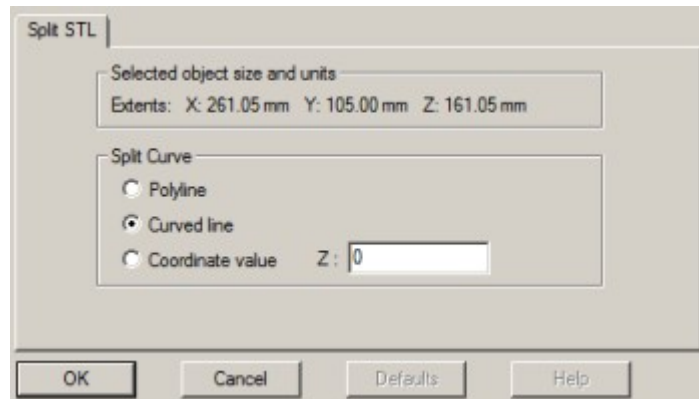
Start the model splitting with

- ***Print Preparation* > 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

Expert Series 10.1: Tutorial 1 – Working with 3D Models

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.

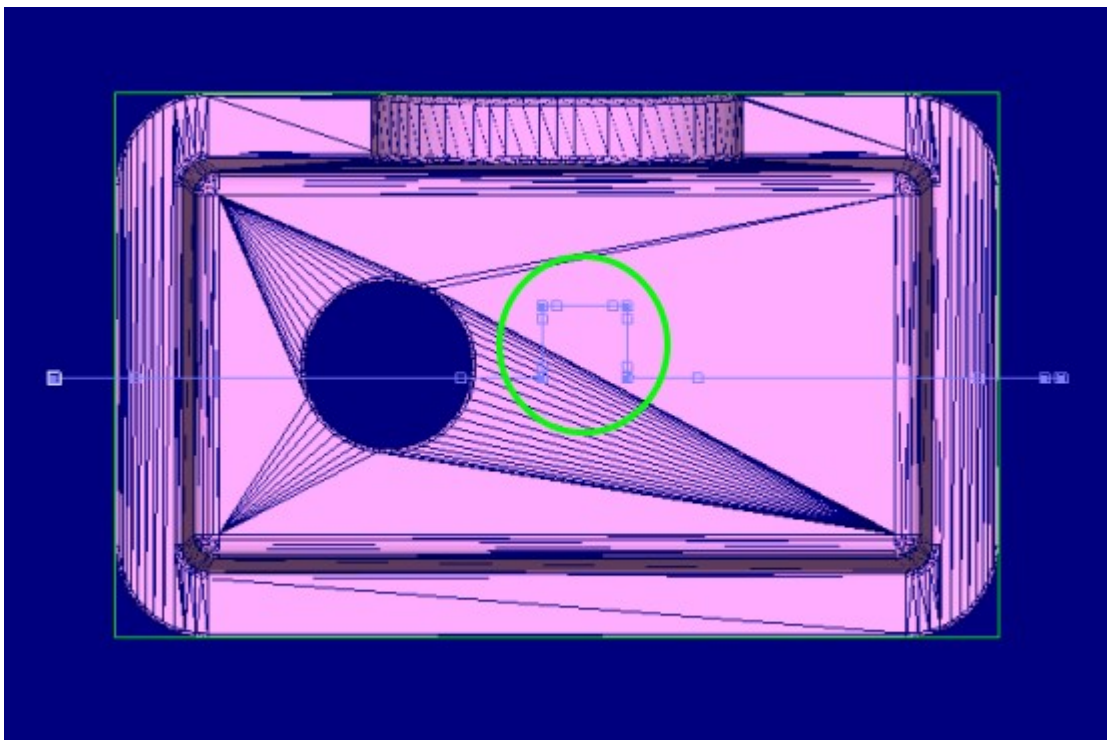


- **Press *OK*** to start adding curve points.

For each split curve point at the curve corners below

- **press twice with the left mouse** (points inside the green circle below).

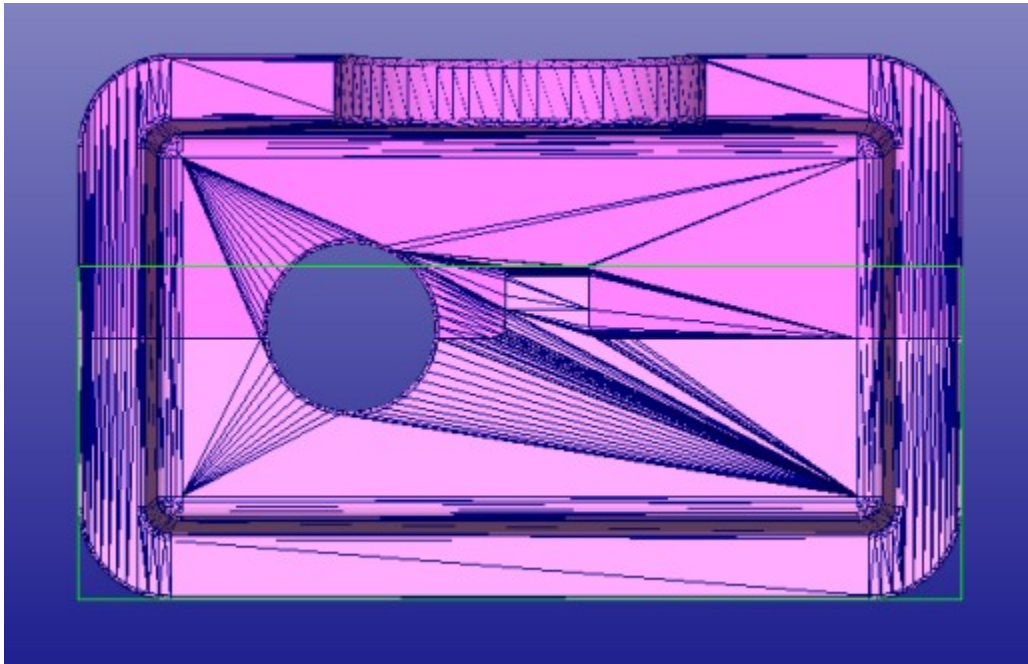
This will generate a rounding with small radius at each corner (single click would generate a “wavy” split line):



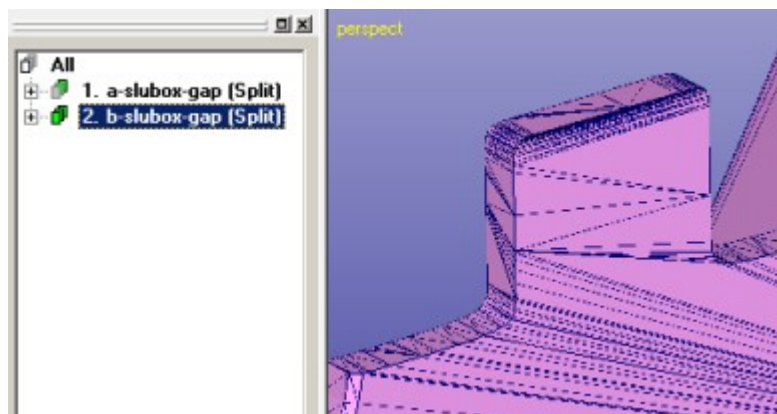
Or if even a bigger radius is needed at each corner, create two points close to each other, not double points. This will increase the radius at each corner and allow to use bigger Lap Joint Height values later, for example.

When ready drawing the split line,

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



If you now display the selected half of the model and zoom closer to the teeth, 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.

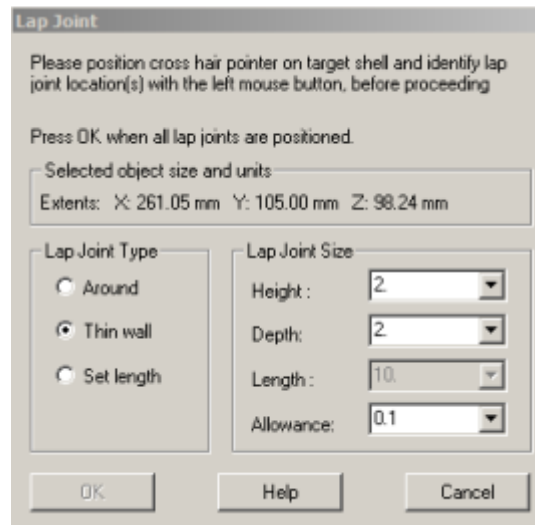
- Give the **Print Preparation > Lap Joint** command.

You can also start it by pressing the *Lap Joint icon* in the Tools Window Print Preparation 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 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

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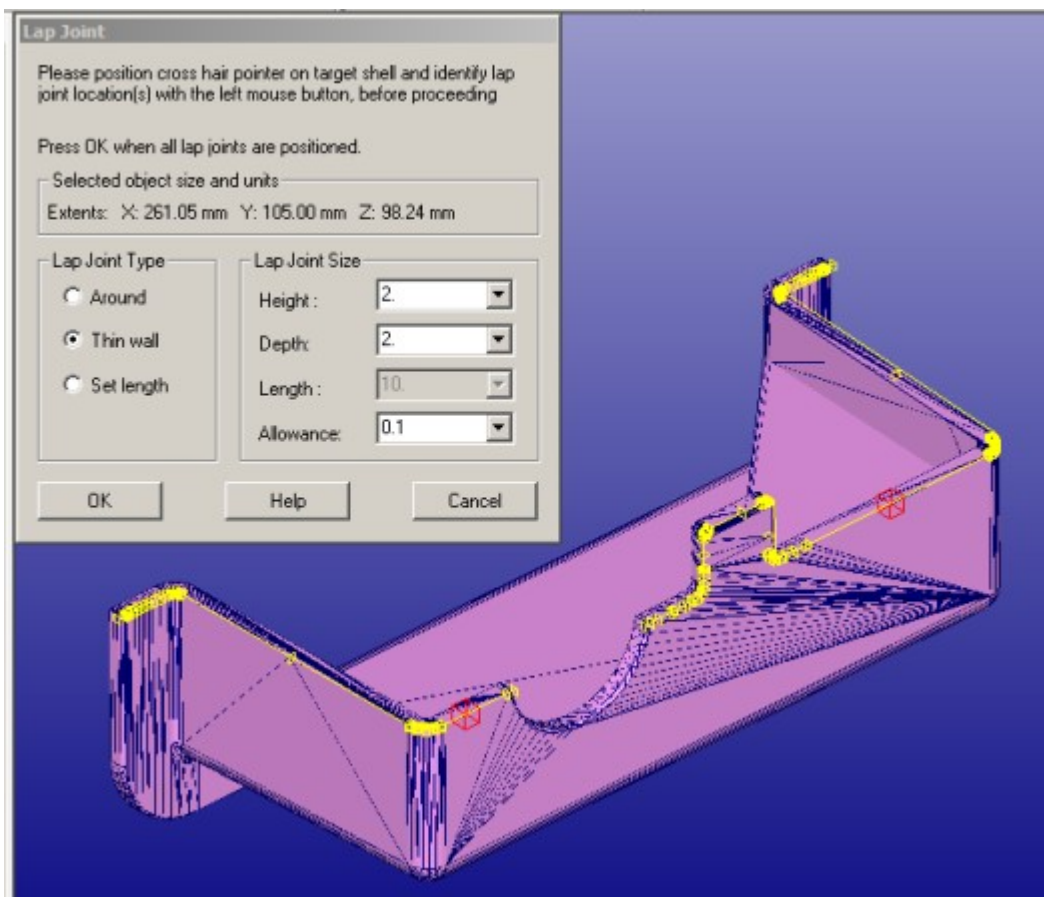
lap joint, click the red indicator box and the lap joint disappears. Modify the parameters or press another point to change the location, if necessary.

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 image on the next page

- **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.**

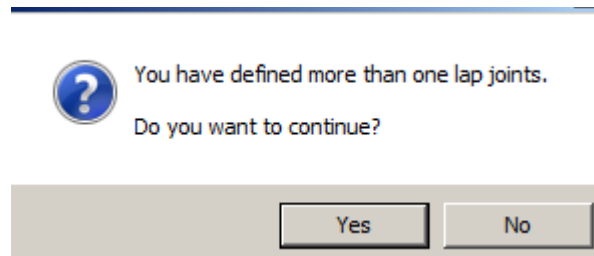
The lap joint structure is automatically deleted from the displayed part and added to the other split half (thus you must not move the split parts before adding lap joints between them).

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

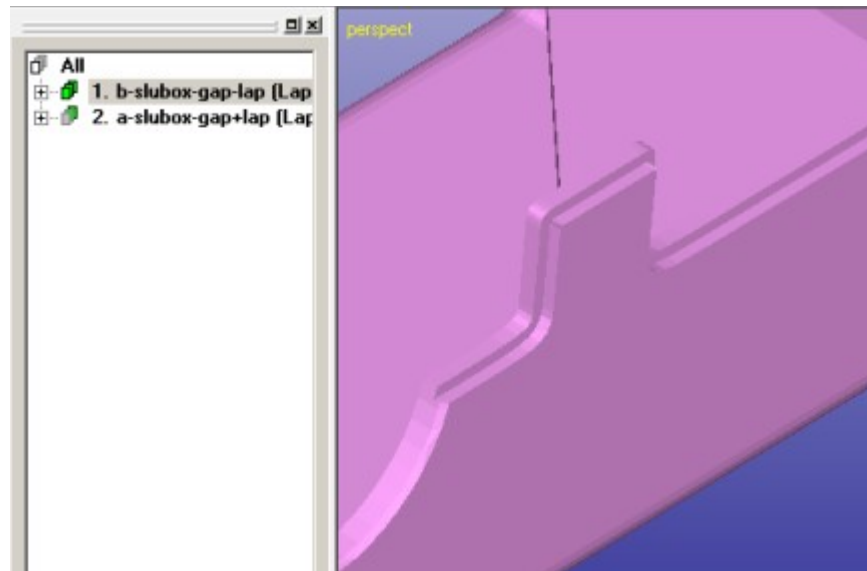
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After pressing OK the software checks if more than one lap joints should be generated.

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



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



Parts can now be outputted and build separately and then accurately joined and glued back together.

Lesson 5 – Positioning Parts

This lesson shows you how to use the Expert Series 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.

Green box indicates if this lesson is applicable to given module:

■ View ■ Dimensions ■ Lite ■ 3Data ■ Sim

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

Defining Platforms

The DeskArtes Expert Series has a user friendly interface for defining and manipulating the available RP platforms. Restart the 3Data Expert software or clean up the workspace with File > New command.

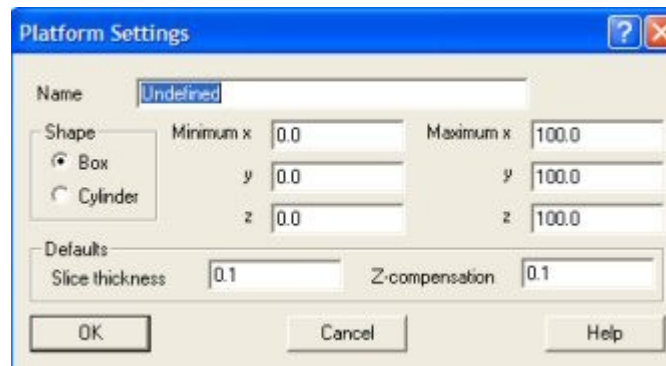
- **Give the command *Print Preparation* > *Platforms*.** You will see the following dialog:



This dialog includes the commands to manipulate the set of available platforms. Let's make our first 3D Printer workspace

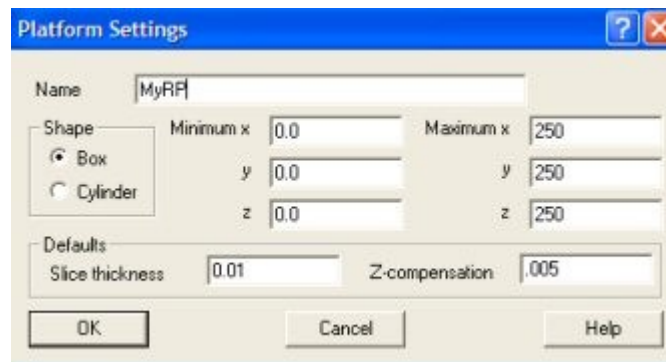
- **by pressing *New* button.**

The *Platform Settings* dialog will appear.



The *shape*, *size*, *slice thickness* and *z-compensation* values you will find in the machine specifications. Here we will use an imaginary *MyRP* machine as an example.

- **Please fill in the values as in the figure below:**

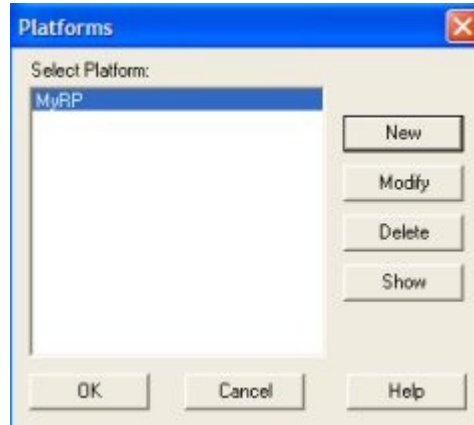


In this example we have selected the *size* to be 250x250x250 mm. The *slice thickness* is set to 0.01. This value depends on the specifications of your RP machine. The slice thickness here is used with the *Print Preparation > Slice* command automatically. The *Z-compensation* value is used with the *Print Preparation > Z-compensation* command automatically.

Now we are ready and you can

- **press OK to save the platform** and to close the dialog.

Now we have our first platform definition in the *Select Platform* list.



More platforms can be defined in similar way. When you are preparing data for a specific platform you do not need to remember different parameters for the machine anymore, just select the correct platform from the list.

If you are not happy with the definition of a platform you can either *Modify* it or generate delete it with the *Delete* command.




Note: Since there are no units given in the platform definition window you must name the platforms according to their units, like MyRP-mm or MyRP-inch, to distinguish them from each other.

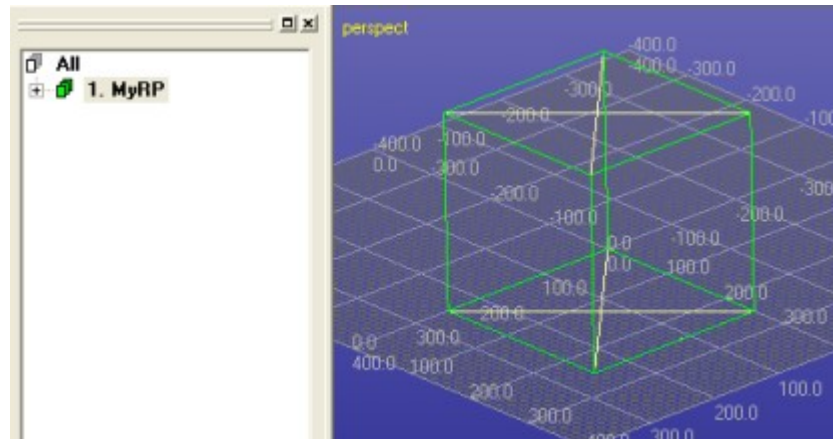
To be able to see the platform on the graphics area,

- **press the *Show* button** in the Platform dialog

This generates an object with the given dimensions to the Model Tree and displays it. Now you can use this object to check the position of your parts in the vat visually and use it to guide the positioning of the parts in the work space.

- **Press *Cancel* to close** the Platform dialog.

You should see the model tree and drawing area as seen below. If the grid is not displayed, go to *Edit > Preferences > Viewing* tab and turn *On* the *Show Grid* toggle . If the platform changes into the shaded mode press  icon to get back to wireframe viewing mode. Below we are also displaying the grid with Display Grid command,  icon.



Moving parts in the platform

Now we will see how the *Print Preparation > Move to platform* commands are used for positioning the part into the used platform. We have selected the *MyRP* platform as the current machine definition earlier. The move platform commands will now use the minimum/maximum x,y,z values of this platform when computing the default values for the moves.

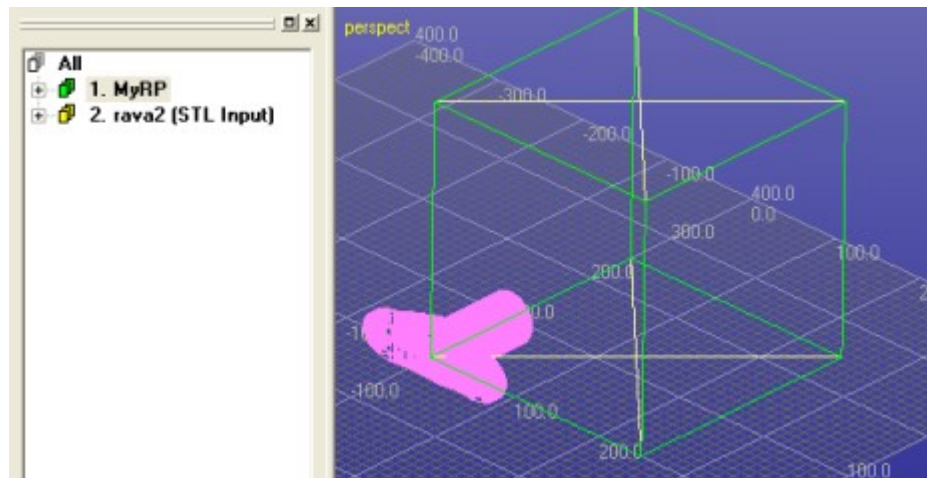
Let's first

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

(we insert to avoid emptying the Model Tree with Open command). When the Operation Mode must be chosen,

- ***select View Model mode* to avoid unnecessary file analysis and fixing at this point.**
- **Close the miniature Operation Mode Window by pressing its close button.**

Also ensure that the MyRP platform and the Model Tree are visible:

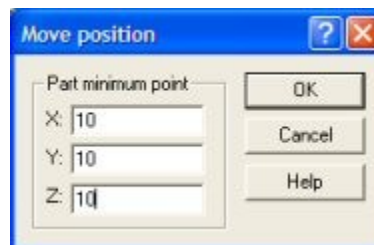


We can see that the part is not inside the build area.

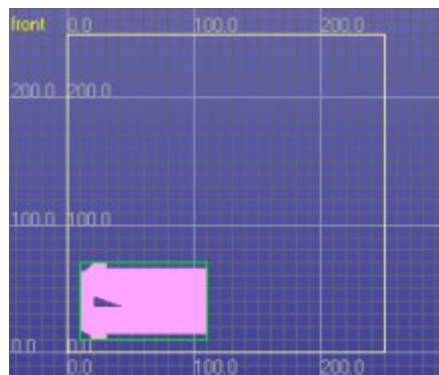
- **First select the part *rava2* from the Model Tree and then**
- **move the part into the area with the command *Print Preparation > Move to Platform > Move position*.**

The command suggest the default values which are the same as the minimum bounding box values for the selected platform *MyRP*.

- **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 taking a view from the +X direction:

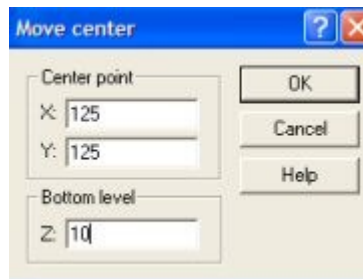


Then

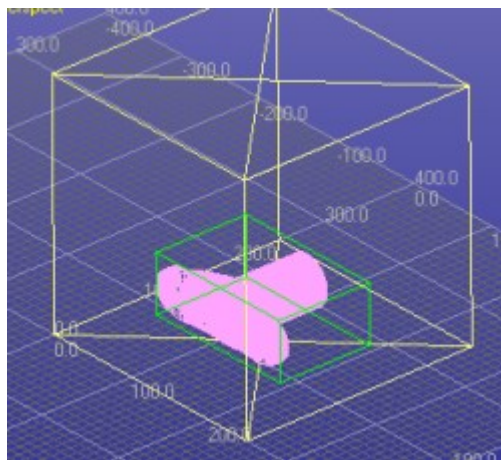
- give **Print Preparation > Move to Platform > Move center** command.

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 have to 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 have the result show on the next image:



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. For that make sure that the Transformation toolbar is visible by giving the command *View > Toolbars > Transform Toolbar*



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This toolbar includes commands for part moving, scaling, rotation and mirroring. Note that clicking the icon with LMB allows you to move parts graphically, clicking with RMB opens a dialog for numerical transformations.

There are even more commands in the Transform menu, like scale from *MM to Inch* and the other way around.

Lesson 6 – Adding 3D Text

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

Green box indicates if this lesson is applicable to given module:

 View  Dimensions  Lite  3Data  Sim

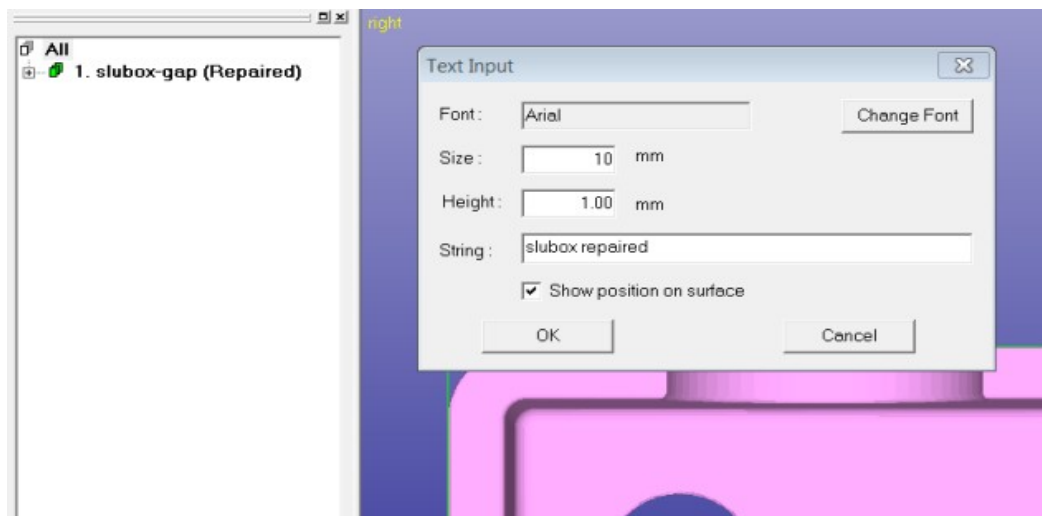
Creating 3D Text

Normally you want to add the 3D text on a part to be build. To start the lesson, first input and repair slubx-gap.stl: First we will input the familiar slubox-gap.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 window.

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

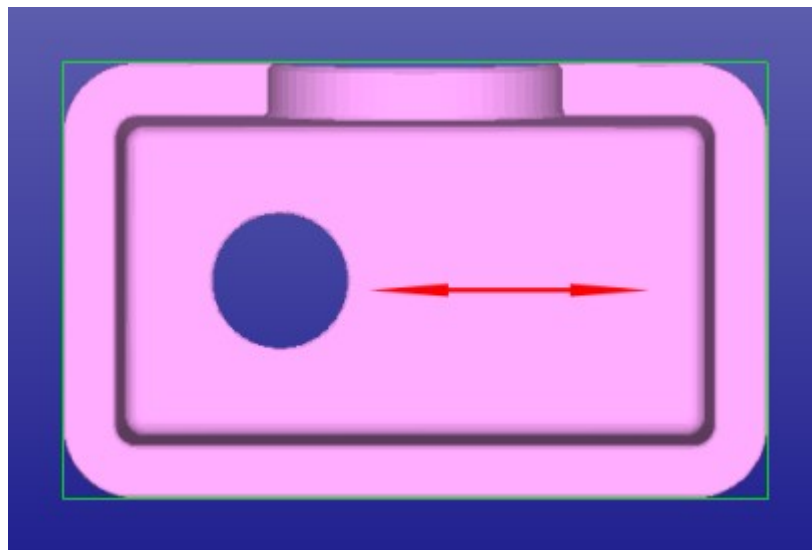
- **view from Y axis direction and**
- **give the command *Create Geometry > Create 3D Text*:**



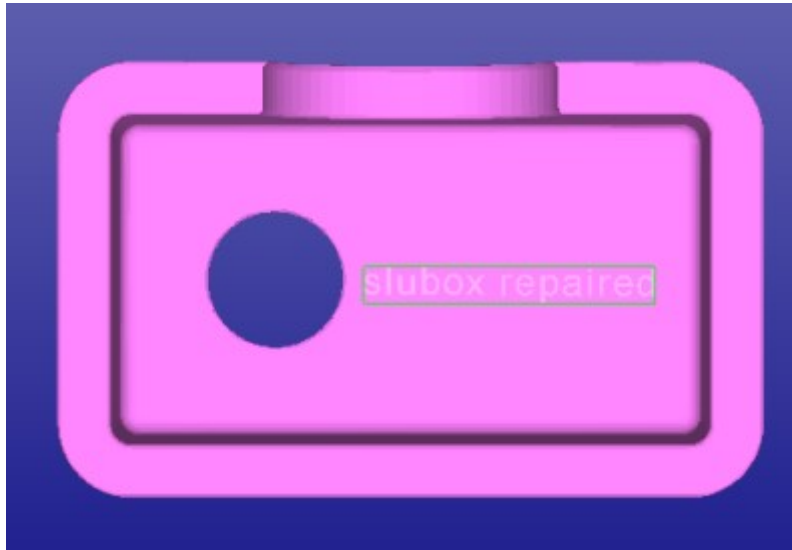
Fill in the values given in the dialog above: *Size 10 mm* (vertical height), *Height 1 mm* (=text height from the surface) and finally *String “slubox repaired”*. The font is preset to *Arial*, well use that. *Show position on surface* is set. When we

- **press OK** the program asks us to show a line on the surface along which the text will be generated.

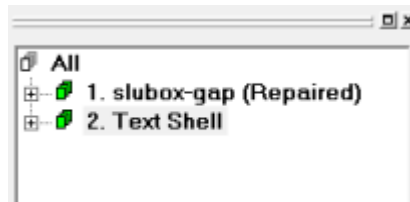
Click with the mouse on two locations (red arrow heads, left to right) to define the line on which the text is generated on surface, as indicated below:



Soon you will see the text generated on top of the slubox surface:



The “slubox repaired” text is generated into a separate *Text Shell* in the Model Tree:

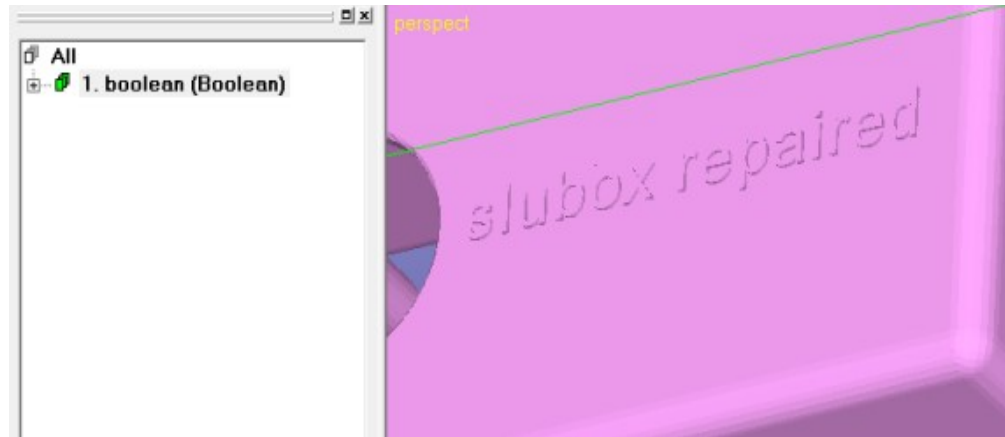


Adding 3D Text to the part

The text is automatically positioned so that 10% of the text *Height 1 mm* is below the selected surface, 90% is on top of the selected surface. Now we can use the *Multiple Join* command to add the text to the part. First

- **Select** Model Tree root **All**, then
- give the command **Modify Faceted > Multiple Join**.
- Accept *Remove original parts* and **press OK**

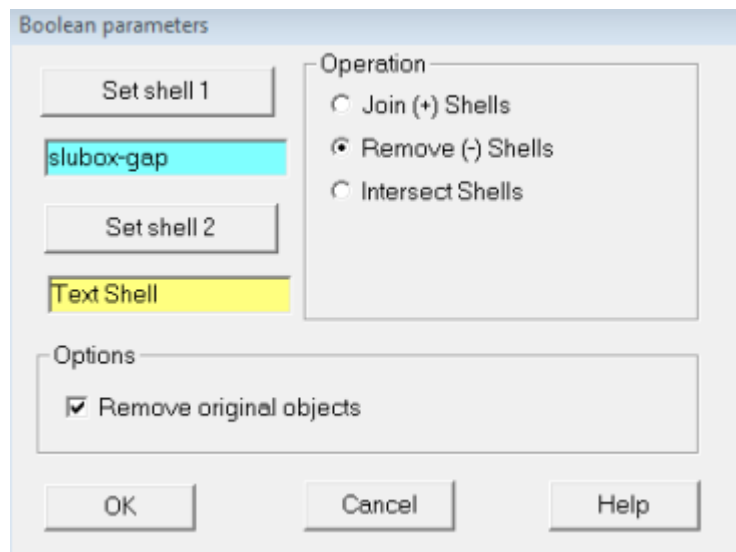
After a few moments you will find the combined result in the Model Tree and visible on the graphics area:



You may want to rename the result shell *1. boolean (Boolean)* to something more descriptive, like *1. slubox-rep*.

Note: If there remains some errors in the Multiple Join results, try using Transform > Move Interactive commands to move the text into more feasible location on the surface and rejoin.

If you wish to engrave the text into the part, you should use the *Modify Faceted > Boolean* with *Remove (-) Shells* option, as seen on the next page. Also, to tune the depth of the text or the final location on the surface, you can use the *Transform > Move Interactive* or *Move Numeric* or *Rotate* commands before the actual boolean operation.



Generally, 3D Text can be attached on a relatively flat surfaces. When the text location is given with the two mouse clicks on the surface, a direct line is drawn from the first point to the next and the text is generated along the direct line. If the surface is too curved along the given line, some letters may not be attached to the part or may be fully inside the part.

Lesson 7 – Preparing for 3D Printing

There are several issues to take into account when sending 3D models for 3D printing. Most processes are more effective when printing bulky models hollowed. Also, being able to drain out the excess material from a bulky hollow model enables remarkable material savings. Open flat surfaces must be offset to a certain minimum thickness to print properly 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 result through its *Print Preparation* menu. For this and following lessons 3Data Expert software is mostly required.

Green box indicates if this lesson is applicable to given module:

■ View ■ Dimensions ■ Lite ■ 3Data ■ Sim

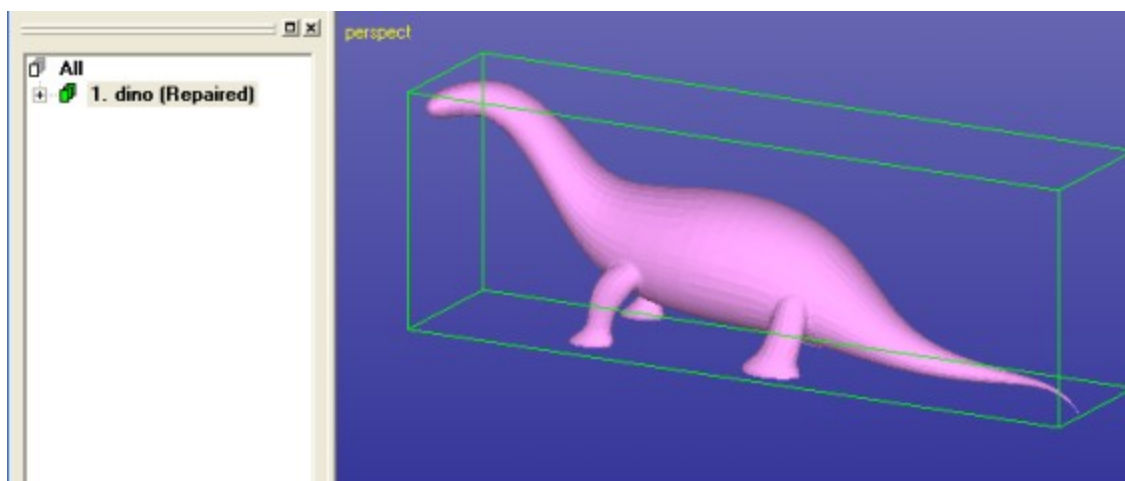
The Hole Tool, Cut Tool and Boolean examples can be partly followed with 3Data Expert Lite software too.

Making Hollow models

Building large and bulky models may require long time to print or consume large amount 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 Print Preparation Mode**

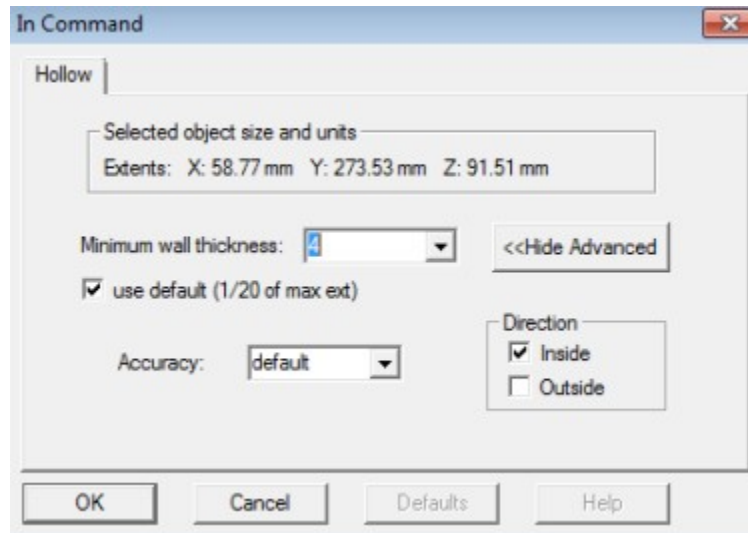


Dino model is a bulky model suitable for hollowing.

- Give the command **Print Preparation > Hollow** or use the



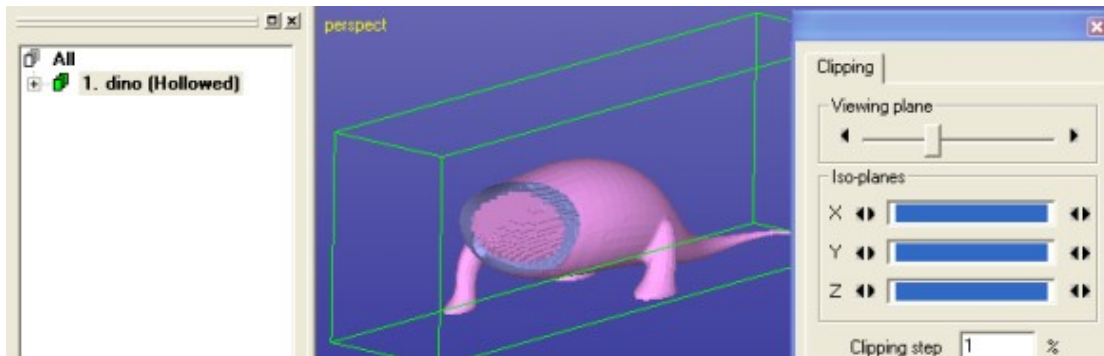
shortcut in the Tools Window. When the command is given the following parameter dialog appears:



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 default *Minimum wall thickness* value to 4 mm as seen in the image above. According to default *Advanced* settings we will make the hollowing surface *Inside* the Dino model and use 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 the *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 a 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 *Print Preparation > Offset* command.

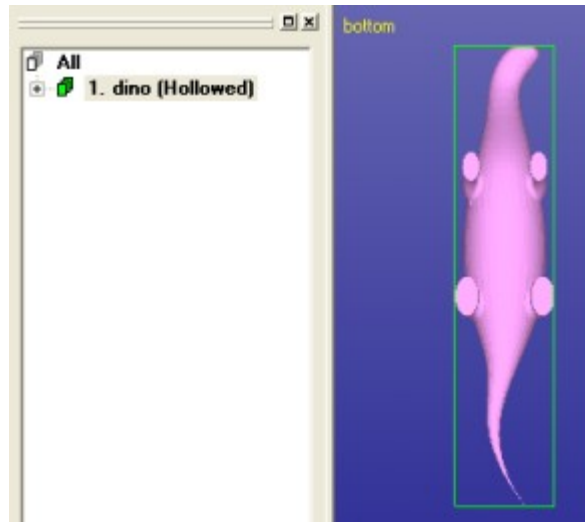
We will use the result of the hollowing in the next chapter where we make a drain hole to the hollow model.

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 size in the STL models.

We will continue with the result of the previous example, hollowed Dino model.

- **First take a suitable view** of the dino model to see the underside of the model by pressing the  icon with the *RMB*:

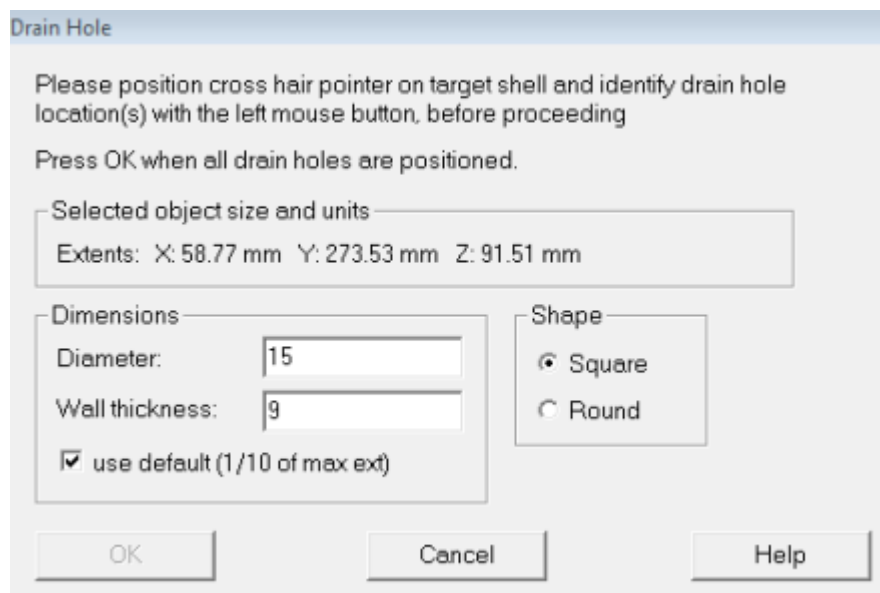


- Give the **Print Preparation > Drain Hole** command or press



shortcut to start the *Drain Hole* command. A parameter dialog appears (on the next page). This time the default values are also calculated based on the maximum extent of the selected object.

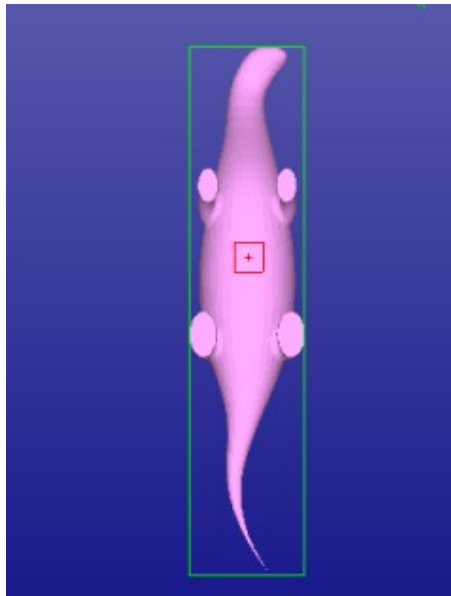
- **Modify the *Diameter* and *Wall Thickness* values**, as seen in the parameter dialog image below (15 and 9 mm respectively):



The *Wall thickness* parameter value depends on the wall thickness of the part in the position we want to put the drain hole. Here we will create the hole in the middle of the part bottom where the wall thickness is about 4 mm, but not less. A bigger value, like 9 mm, is used to make sure that the hole will go through the hollowed surface.

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

- **Position a drain hole with LMB in the middle** of the bottom surface as seen below:

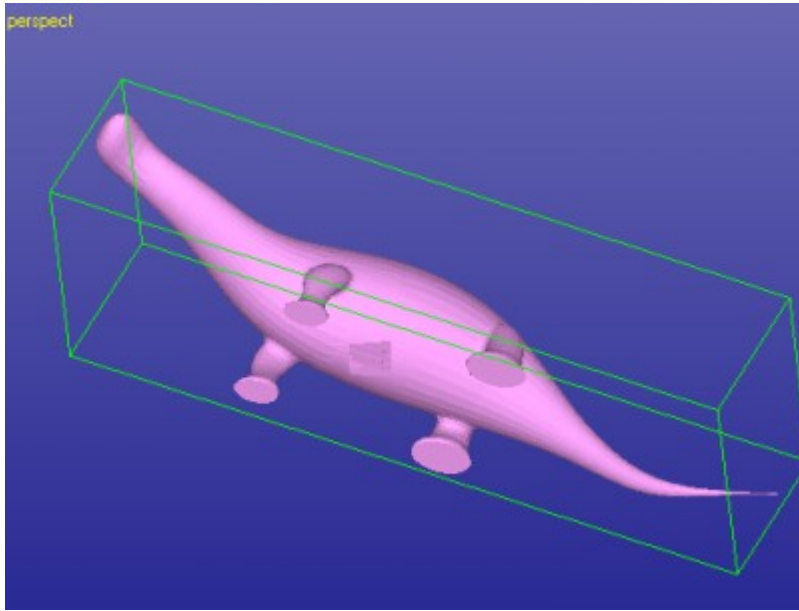


If a hole location is not correct you can remove it by clicking on the red indicator box again. Then reposition with another LMB click on the correct location.

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.

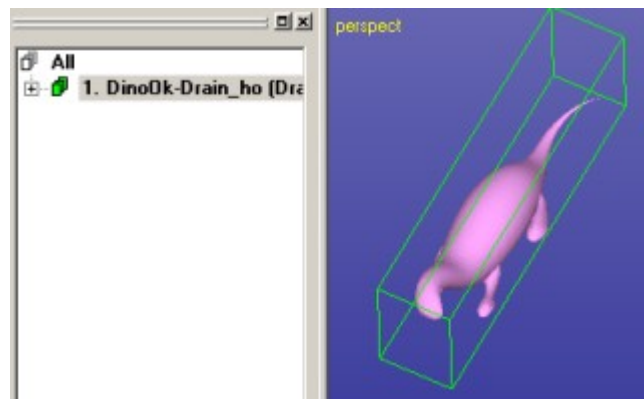


Drain Holes can be generated on any wall, it is always created parallel to the surface normal at the given location.

Making tapered plugs with Hole Tool

The new *Modify Faceted > 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.

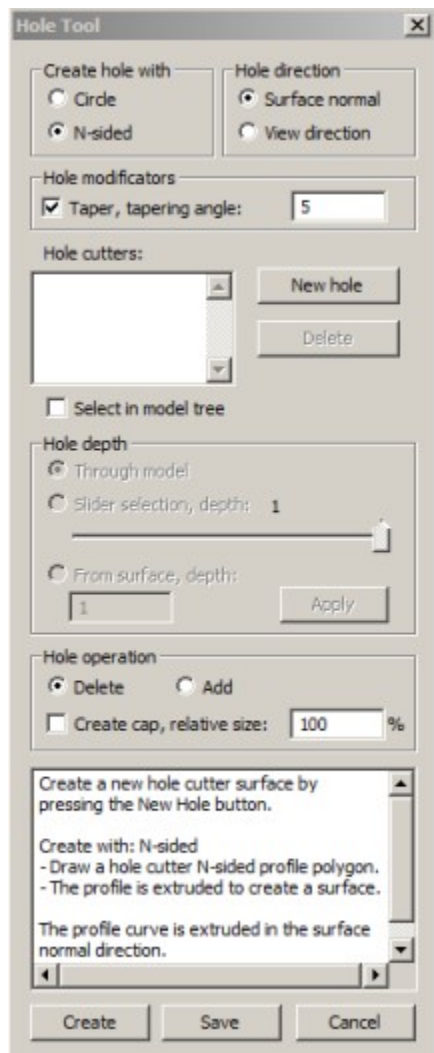
We will continue with the hollowed dino created in the previous example. Now take a view from the top of the model:



Then

- start the **Modify Faceted > Hole Tools** command.

A Hole Tool dialog is displayed:



Hole Tool dialog contains several 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 the same time. The plug will have a tapering to prevent it falling into the part. Also, the plug will be done slightly smaller than the hole to enable easy gluing of the plug into the hole.

Start 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 beside.

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

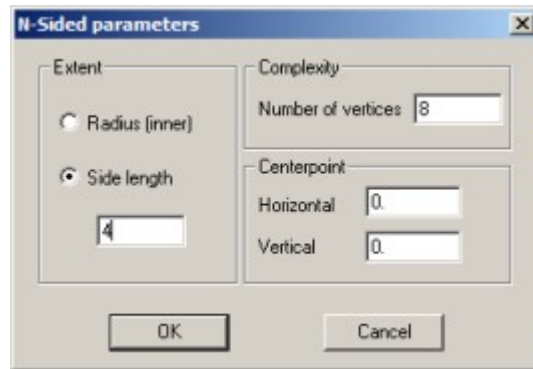
- **press the *New hole* button** to show the location and size of the hole.

A crosshair icon will appear.

- **Click in the middle of the direction back to set the hole middle point.**

Then

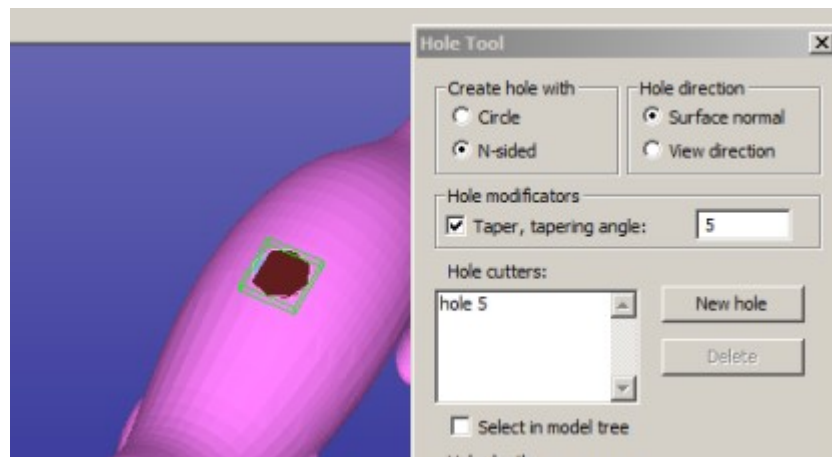
- **draw a box by dragging** with the LMB to estimate the size of the hole to generate.
- **When the mouse is released**, a parameters dialog appears:



Note: Centerpoint 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:



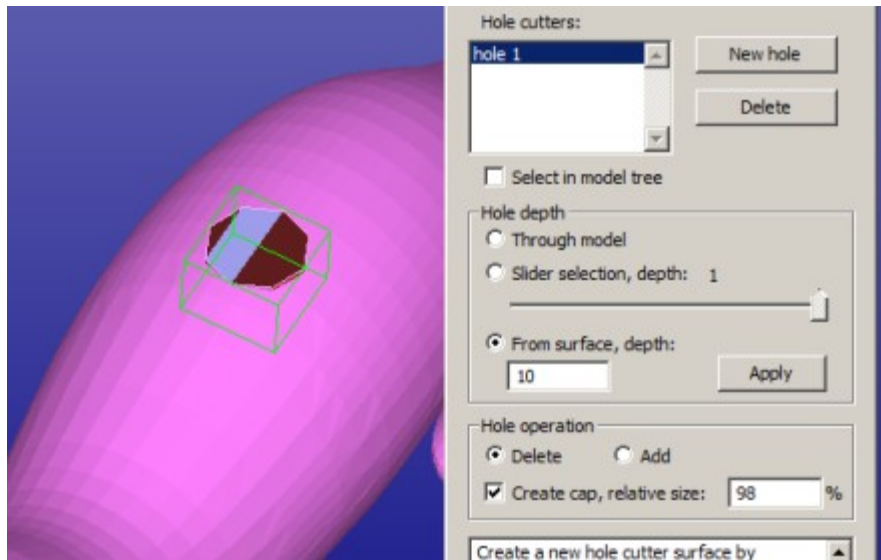
Hole 1 will be generated into the Hole cutters list. Now we will set the length and scaling for the plug.

Then

- **Select the *Hole 1*** in the cutters list with the *LMB*.
- For *Hole depth* **select the *From surfaces, depth*** and set the value to *10.0* (to be sure to go through the *dino skin*).
- **Press *Apply*** to really change the hole depth after setting the value to *10.0*.
- Make sure the *Hole operation* is **set to *Delete***.

Expert Series 10.1: Tutorial 1 – Working with 3D Models

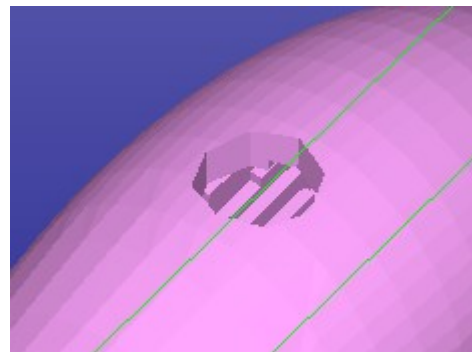
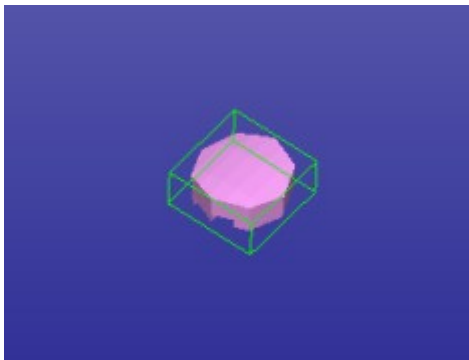
- **set the *Create cap, relative* size 98%** to make the plug slightly smaller than the hole, as seen in the image below:



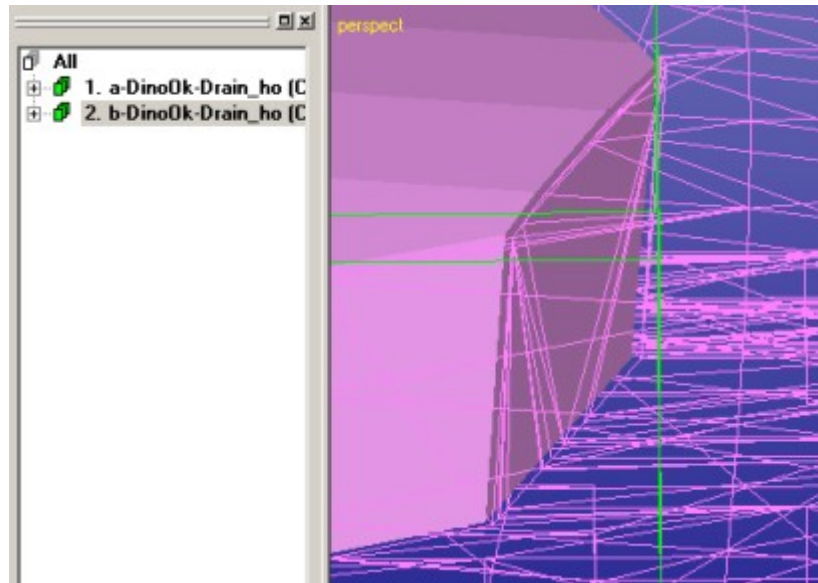
When ready,

- **press *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.



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:



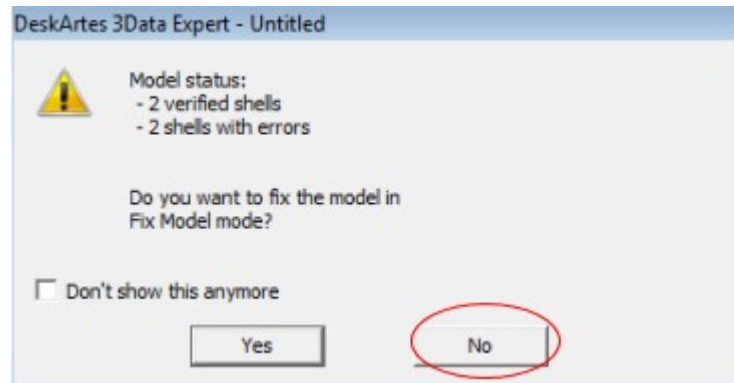
These two shells can now be printed separately. After the body is drained from non processed material or any internal support structures are removed, the two pieces can be glued back together for a nice looking part.

Adding thickness by Offset

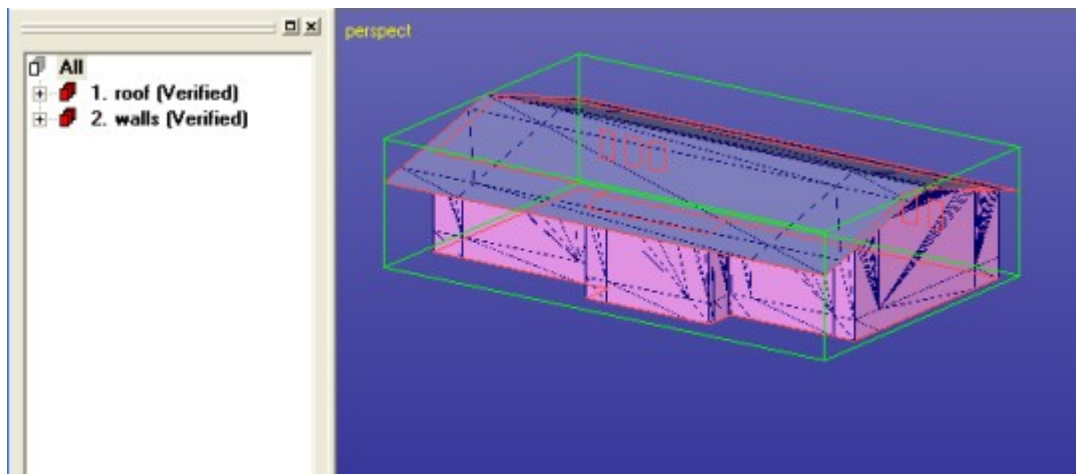
There are several software products which do not automatically produce solid STL data. For example, many architectural software products output open models made of open surfaces if not constructed carefully. These open models must be made solid before inputting to 3D Printing.

3Data Expert *Print Preparation* > *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 *Print Preparation* mode** and



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



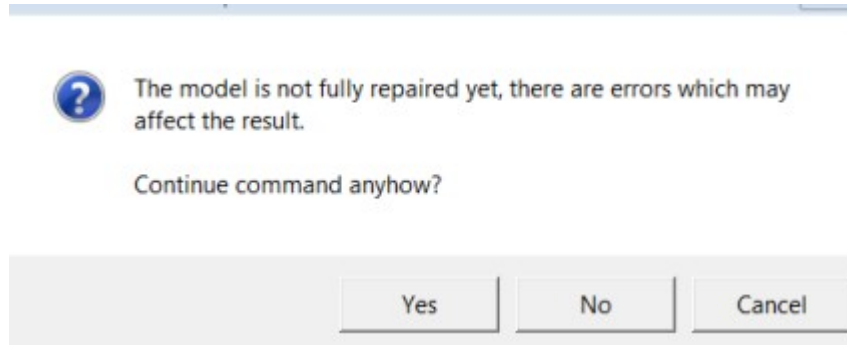
The House model contains two open shells, the roof and the walls. The open surface edges are given with red gap curves.

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

- **Select the *roof* shell** and
- **give the *Print Preparation > Offset* command** or use the



shortcut. The program will notify that there are errors in the model after you give the command.

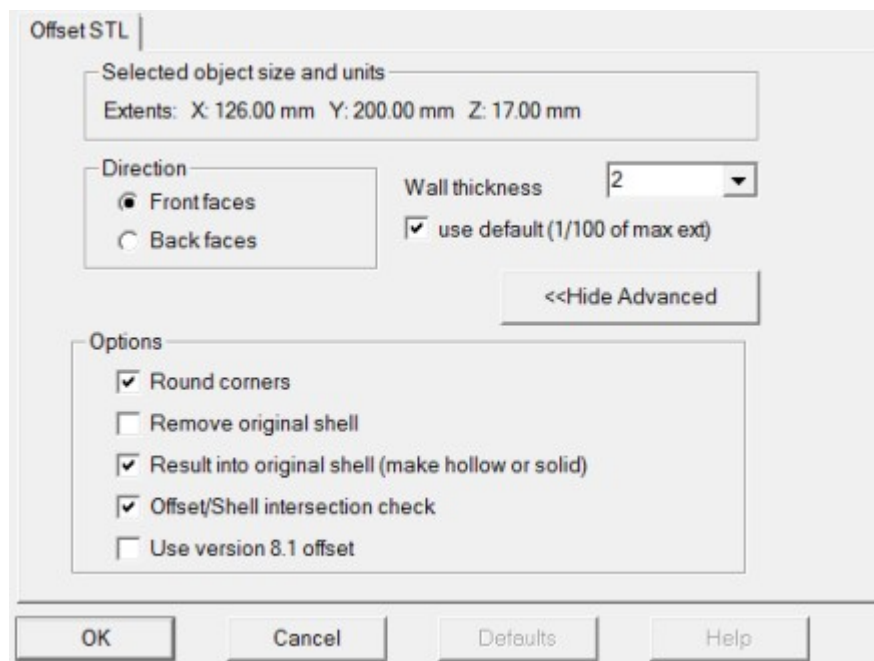


- **Press Yes** to continue the command anyhow (we will repair the errors by offsetting):

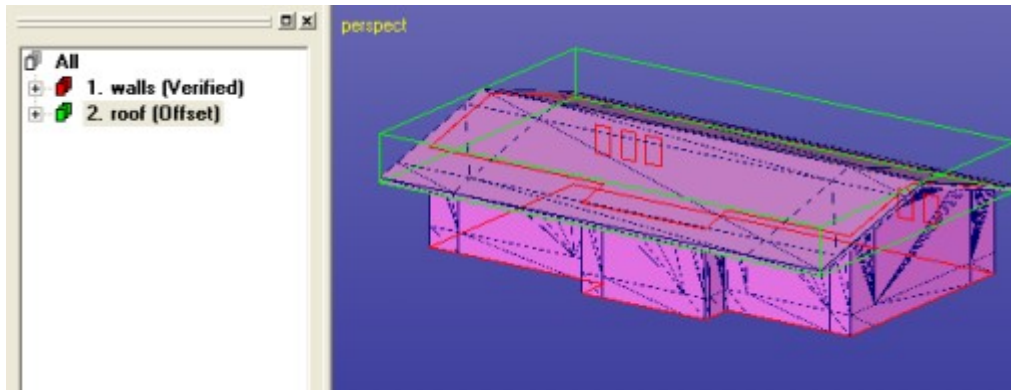
You will see the *Offset STL* parameter dialog. As usual, the default offset distance is calculated based on the maximum extent of the roof shell.

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

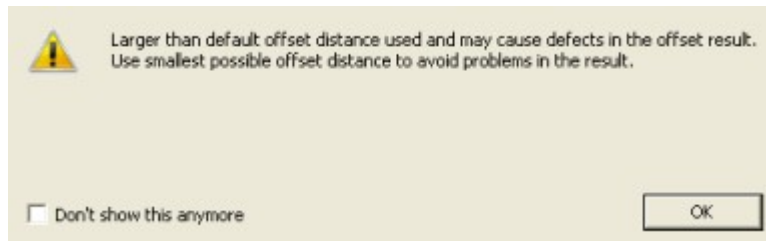
Otherwise we can accept the default values for *Advanced* parameters.



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

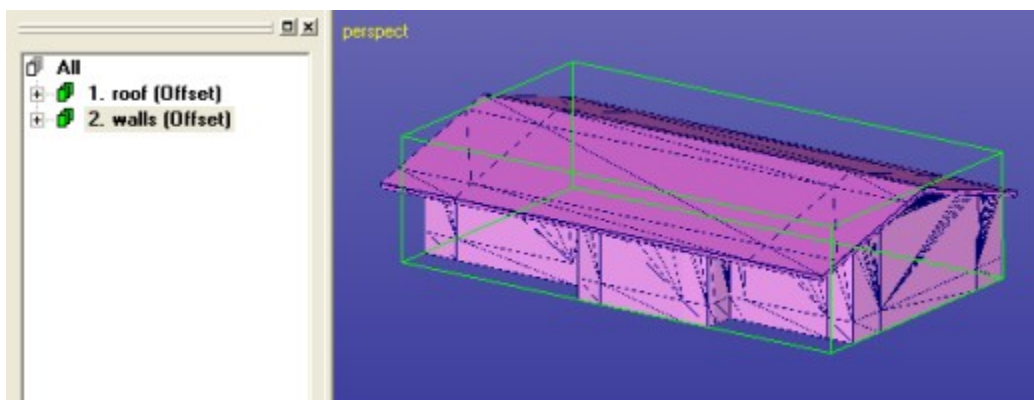


- Now select the **wall shell** and
- repeat the **Offset** with the **2 mm Wall thickness** again. You may see the warning



- 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:



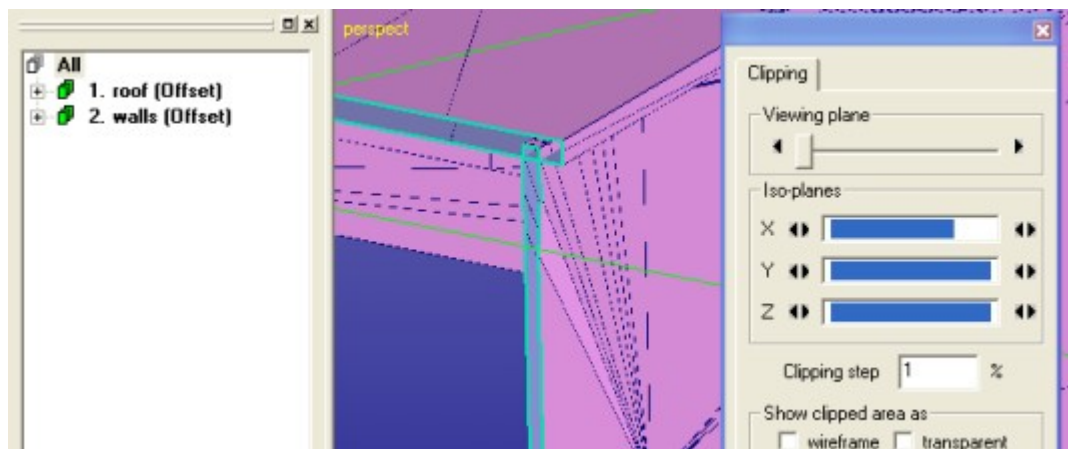
There are some 3D Printing systems which can produce a correct result from these two shells directly. Some systems require the two shells to be connected into one shell to produce a correct result. In the next example we will join the two shell into one shell using the Boolean commands.

Combining shells using Boolean commands

The two shells generated in the previous example are correct but may not print properly if not connected into one shell before outputting for 3D Printing. 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 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.

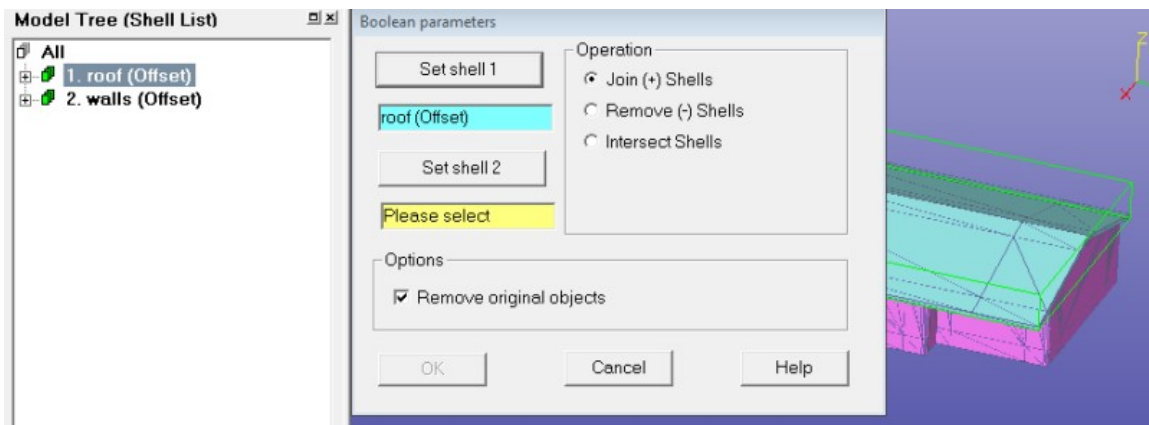
We will use the Boolean command to connect the two shells.

- **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 command to perform between the selected objects.

First

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



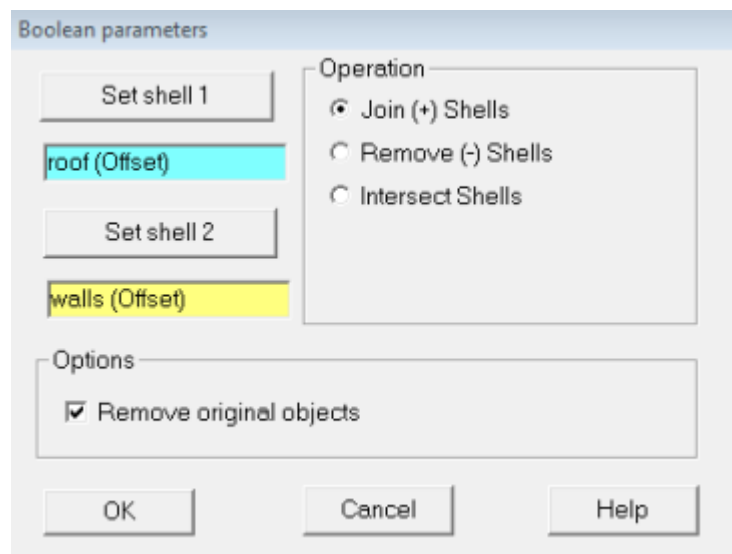
The roof turns light blue in the graphics area and its name is displayed in the shell 1 text field. Then

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

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.

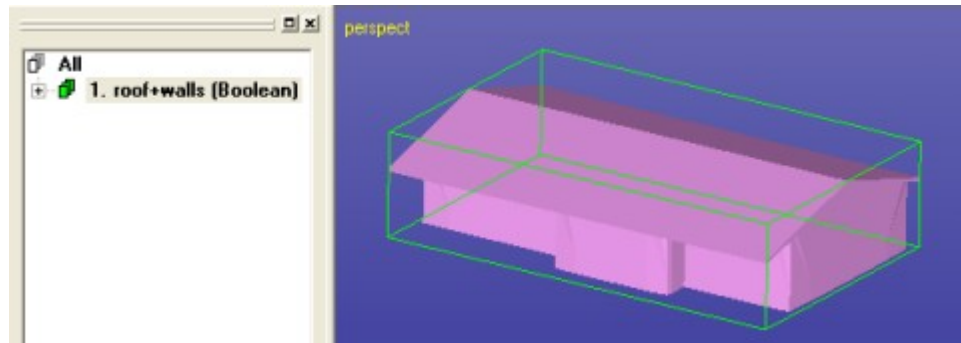
Make sure the dialog looks as seen below:



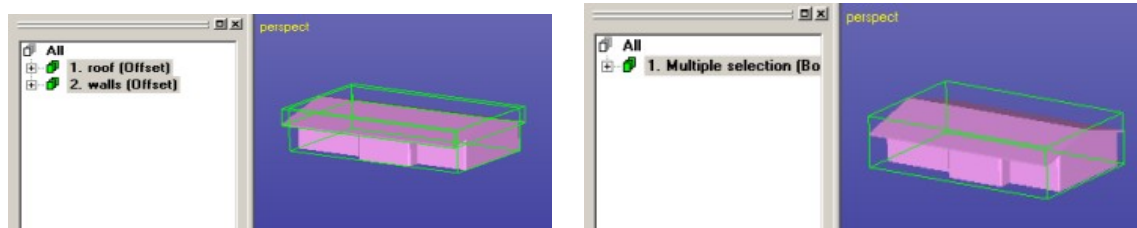
Then

- **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 multiselecting the two shells and by running the *Modify Faceted > Multiple Join* command on them:

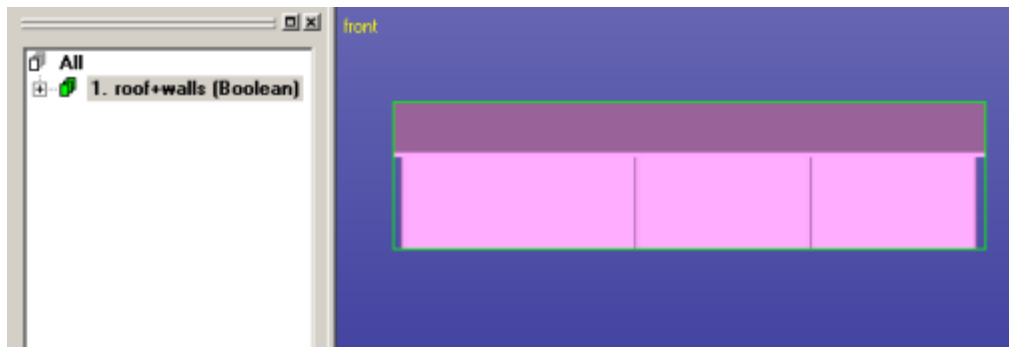


Generally, there are some issues which need to be considered when using Boolean command. Especially important is to make sure the two shells really intersect each other. Tangential or overlapping shells may not behave properly when sent to the Boolean operation. In these cases you should move one of the shells slightly or scale them slightly. The amount of move or scale should be very small (like 0.001 - 0.01 mm for model this size) not to affect the accuracy of the printed model.

Generating advanced cuts with Cut Tool

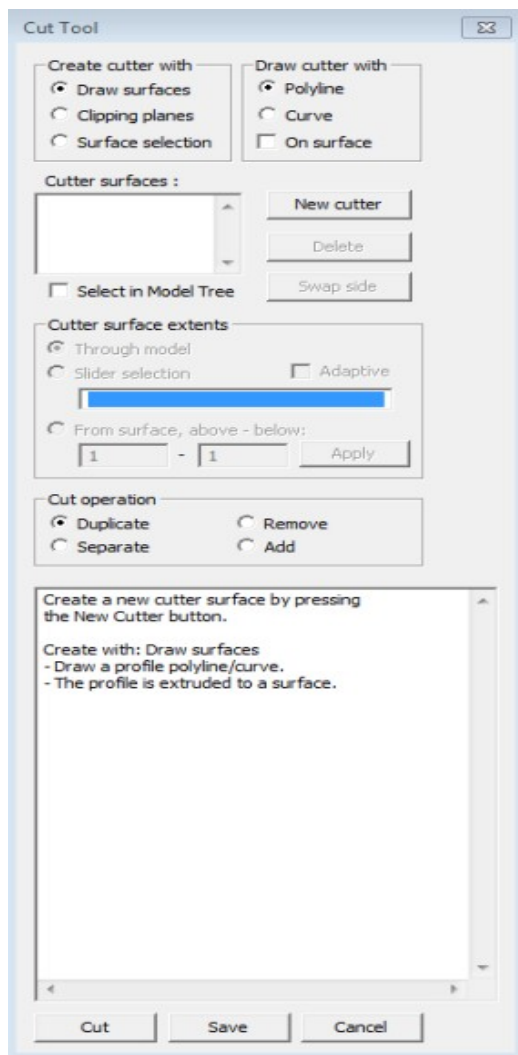
Cut Tool is meant for advanced cutting for various applications. For example, it can be used to cut specific details inside a 3D model without disturbing the surface area in the close vicinity of the cutaway detail. We will now use it to cut a door opening to the house model in the previous example.

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



Then

- give the **Modify Faceted > 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**

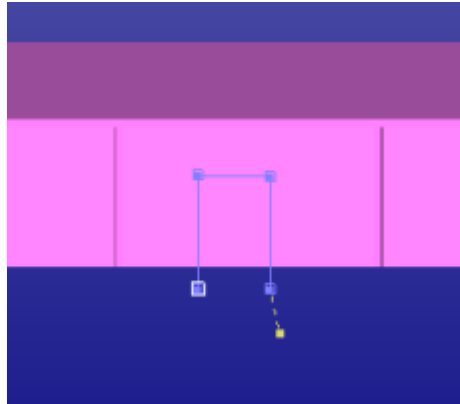
The profile for the cutter surfaces will be generated using the familiar curve editing tool available with the Split command.

When ready

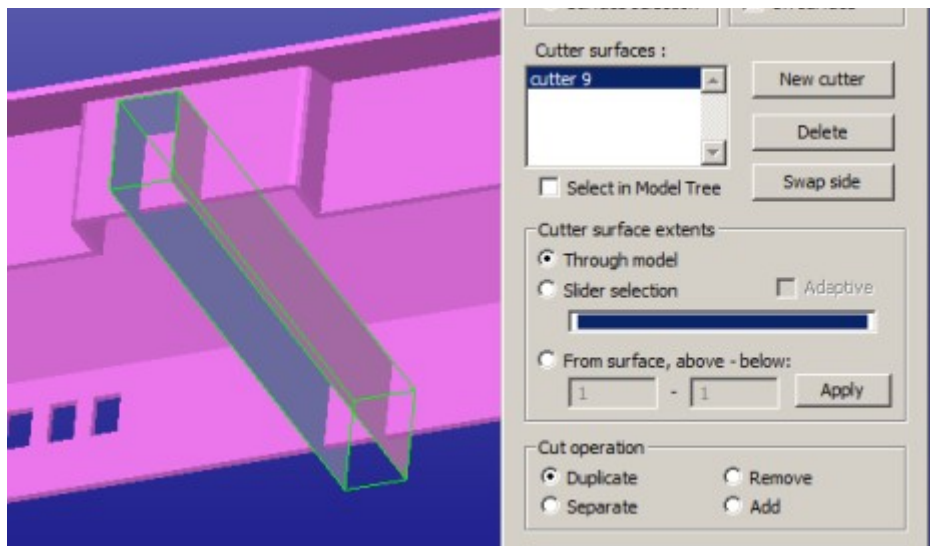
- press **New cutter** button to start drawing the door profile on the house.

Then

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



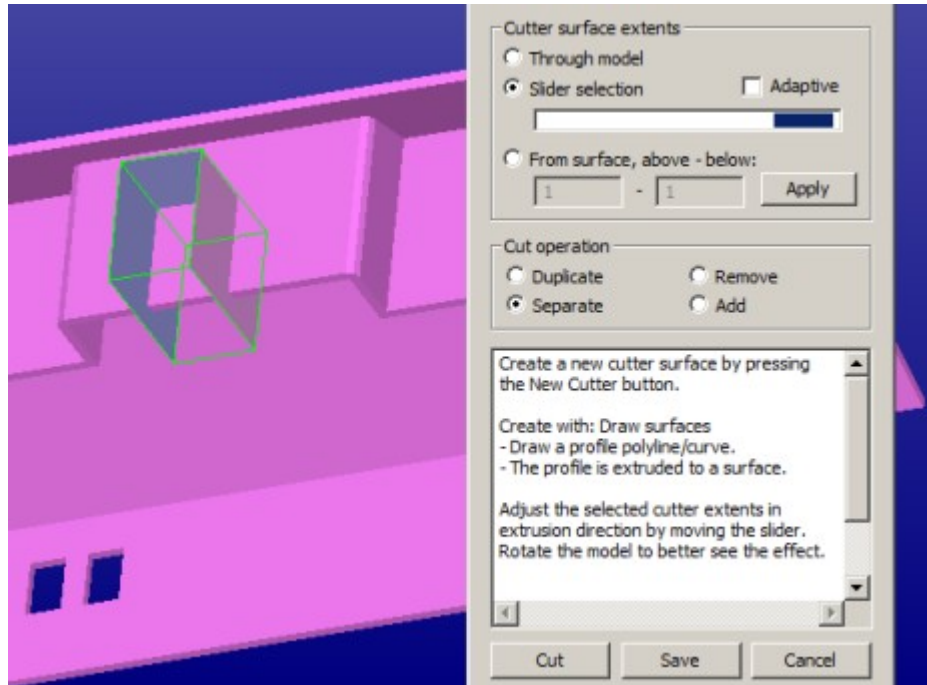
A new cutter surface is generated into the *Cutter surfaces* list:



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 *Slider selection*** and use the slider to resize the cutter surface to correct depth, as seen in the image below.

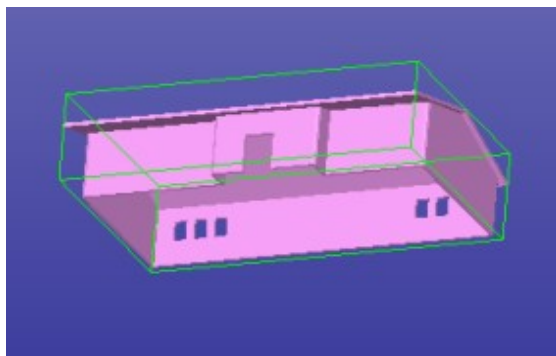
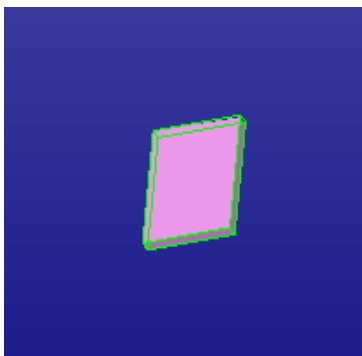
A colored bounding box is used to indicate the cutter surface size while moving the slider with the *LMB*. The yellow bounding box indicates the maximum size of the cutter surface, red bounding box indicates the current size of the cutter surface and moving blue lines indicate the new start and end locations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



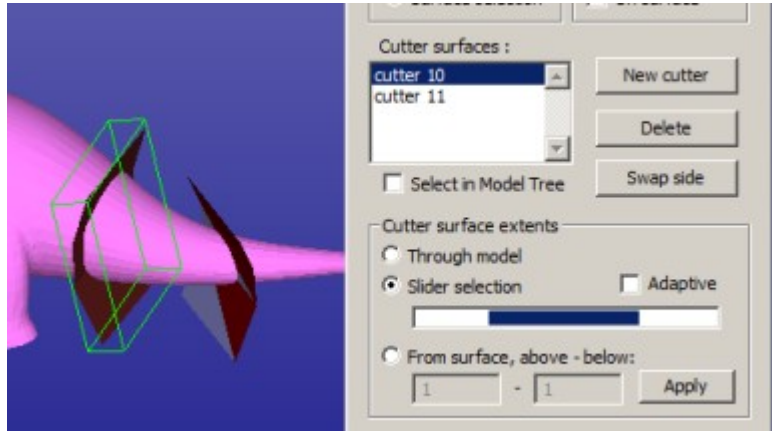
After rescaling the cutter surface to the depth seen above

- 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 new Cut Tool command 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 image we can see two cutting planes ready to remove a part from the dino tail, using cutting planes with different cutting directions:



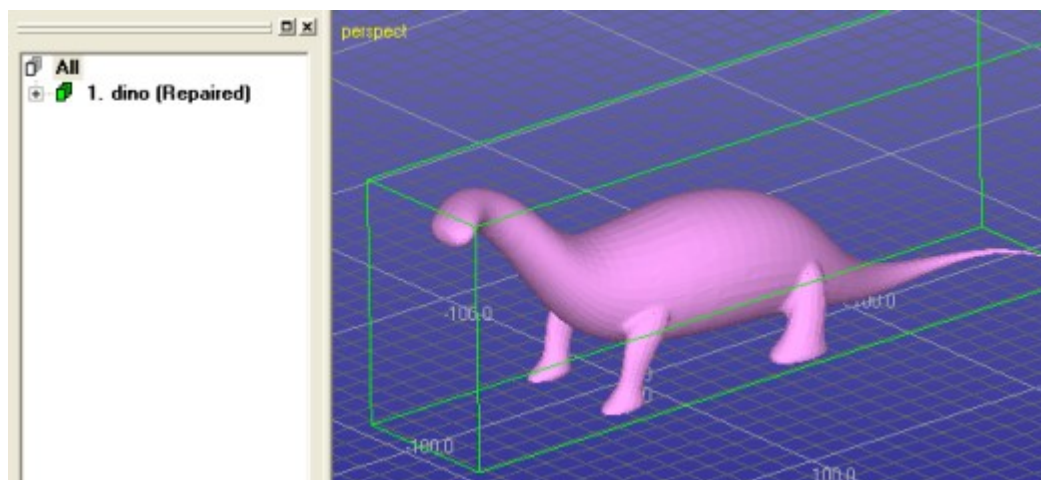
Note: When using multiple cutting planes make sure that the surface normal direction are consistent for the cut operation: For two adjacent cutting surfaces the normals must either be directed towards each other or away from each other. If not the normal direction for a cutter surface can be changed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

Generating SLA supports

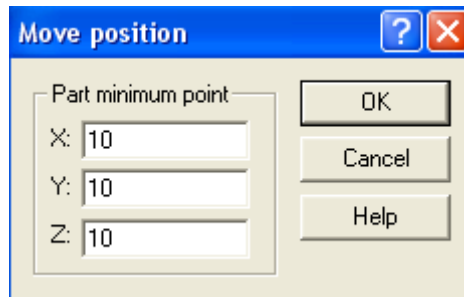
The *Print Preparation > Supports > Hatch Support* is used to generate hatch support structures for Stereo lithography (liquid) based systems.

Here we will shortly describe the process of generating supports for liquid based systems. You should also carefully read the On-line Help pages for the *Hatch support* command. Those pages include a detailed description of the different support parameters used with hatch supports.

- **Initialize your system with *File > New*** and
- **input the DinoOK.3de model *Tutorials / GeomFiles*** directory.
- **Go directly to the Print Preparation mode.**



- **Move the dino into correct position** (positive space) with **Print Preparation > Move to Platform > Move Position** command with the parameters below:



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

- **Select the shell *dino* from the Model Tree.**
- **Start support generation with the command *Print Preparation > Supports > Hatch Support*.**

The dialog box for *Hatch Support* parameters now opens. This dialog has several tabs to define the properties of different kinds of support types exactly. The different types are explained in the On-line Help pages for Hatch support generation.

First you will have to get the values for the support generation before executing the command. We have included one default settings file *DAsupport.par* into your installation directory (same place where the main program *DArtes.exe* is located). You can use the *File Open dialog* which is shown after

- **pressing the Open... button.**
- **Input the default support parameter file *DAsupport.par* with the Open command (under the installation directory)**

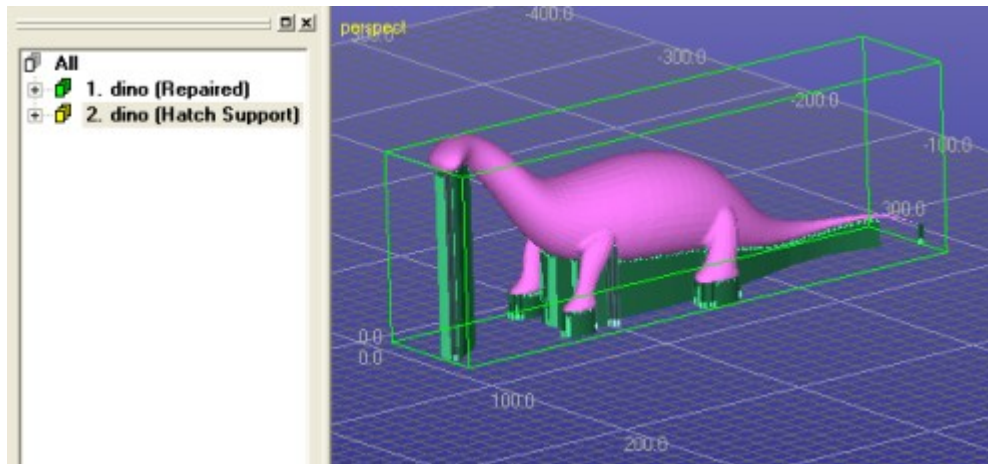


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This set of parameters should get you started with the support generation. Anyhow, the exact values depend on the machine and the used resins. Testing , you own expertise and machine documentation will tell the correct values for each machine/material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

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

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



The part and the supports can now be outputted for slicing with the machine vendor's software, or they can be sliced with the *Print Preparation > Slices* command.

Lesson 8 – Surface Triangulation

This lesson shows you how to do surface verification and triangulation to produce a solid STL file form IGES surface data with Expert Series. The lesson also explains how to fix open faceted model. For this and following lessons 3Data Expert or Sim Expert software is required.

Green box indicates if this lesson is applicable to given module:

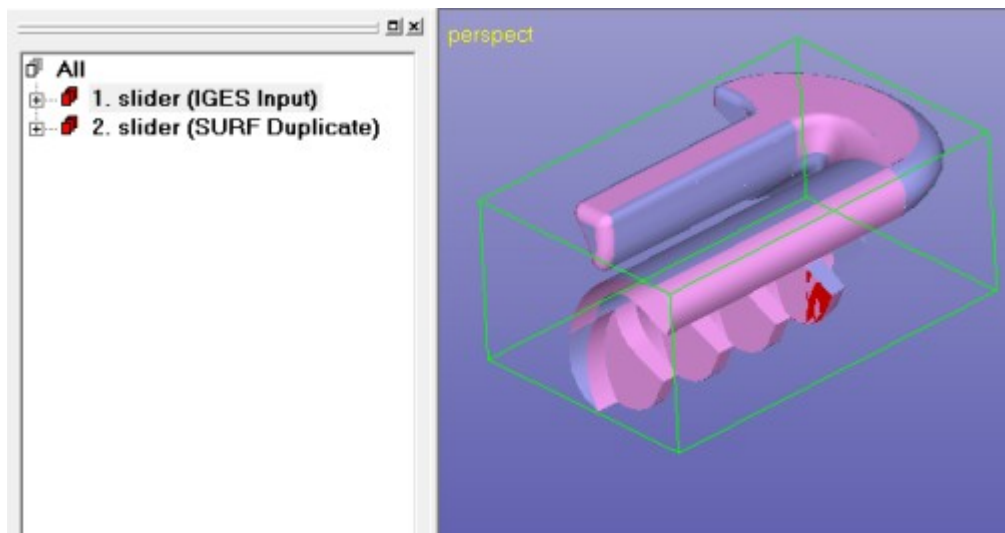
■ View ■ Dimensions ■ Lite ■ 3Data ■ Sim

Verifying the surface

In the previous lesson we learned how to input IGES files and how to verify it. Start this lesson by repeating the verify operation:

- **clean up the workspace with *File > New* command and**
- **read in the file *Tutorials / GeomFiles / slider.igs***


You should see the the slider on the display area and two components in the Model Tree, as seen below:



The outside of the surfaces are shown pink and the inside is shown in blue color.

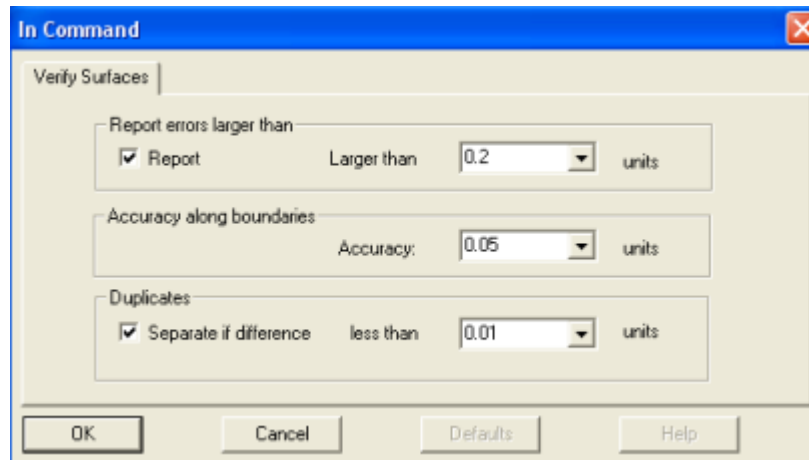
The two nodes in the Model Tree describe what has been found in the input file. The first component *1. slider (IGES Input)* stores the surfaces and the second node *2. slider (SURF Duplicates)* includes the erroneous duplicate surfaces. For triangulation purposes only the correct input surfaces are needed.

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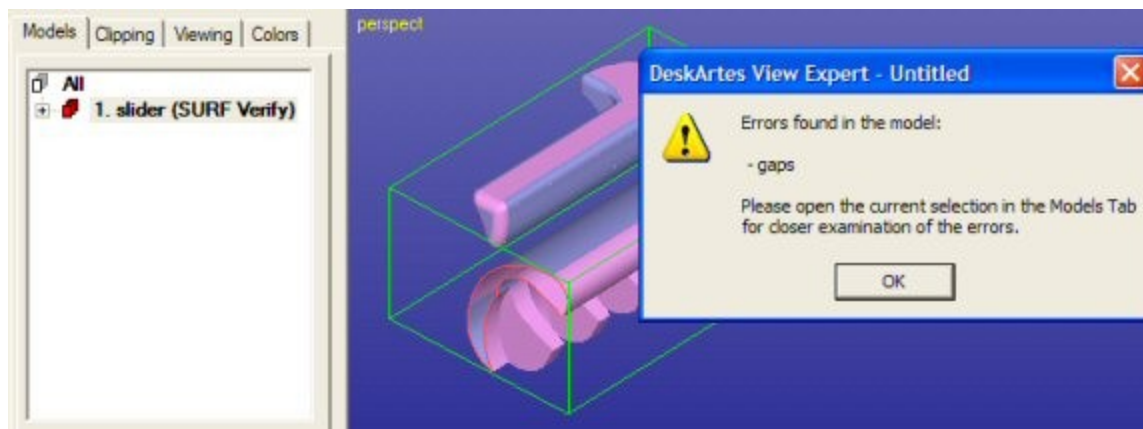
- Just **RMB** click on the **duplicates** and then use **Edit > Delete** command () to remove them.

The verification of surfaces is done through the **Modify Surfaces > Verify Surfaces** command.

- **Start the *Modify Surfaces > Verify Surfaces* command** and **press OK** with the following (default) parameters:

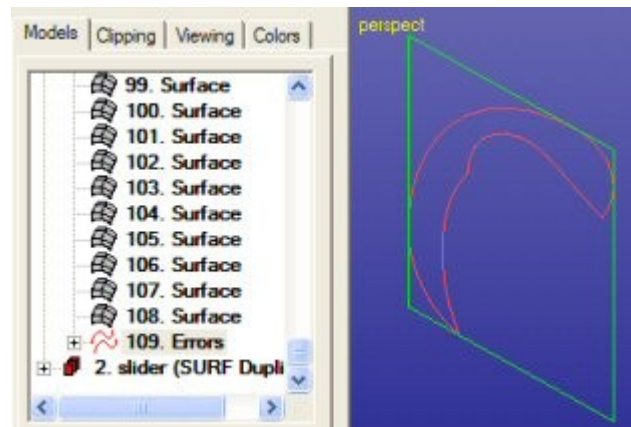


After a while you will see the following result:



The folder icon in front of the *slider* component is red. This, as well as the result dialog, points out that there are errors in the model. The *slider* model has gaps. To display these gaps

- **first erase the screen with the *View > Hide All* command** and then click on the '+' box in front of the *slider* node.
- After that **select the 109. Errors** element. The graphics area and the Model Tree should look as below:



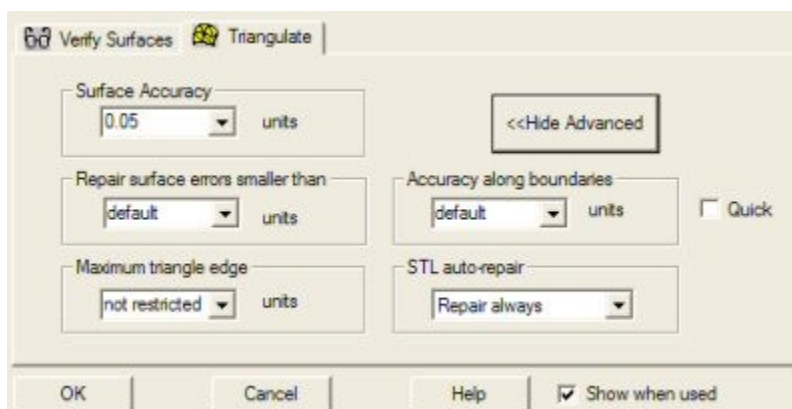
We can see that the end of the rod in the *slider* is open. Short visual investigation shows that the gap is flat and will be easily repaired after the triangulation with *Fix Model > Repair Shells* command which will be done later.

Triangulating the surface

Now

- **Triangulate the slider part with the *Modify Surfaces > Triangulate* command.**

The triangulation can only be done at the shell level and the program automatically proposes shell *1. slider (SURF Verify)* as the target object if a *surface* or *error gap* leaf is selected. If this happens, **press OK** to accept it shell selection You will see the *Modify Surfaces > Triangulate* settings (if not all parameters shown, press *Show Advanced* button):



These parameters control the accuracy and error repair options of the triangulation. The most important value is the *Accuracy* which gives the

maximum distance of the triangles from the original surface model. See the On-line Help *Menus and commands/Modify Surfaces/Triangulate* page and its settings for a detailed description of other parameters.

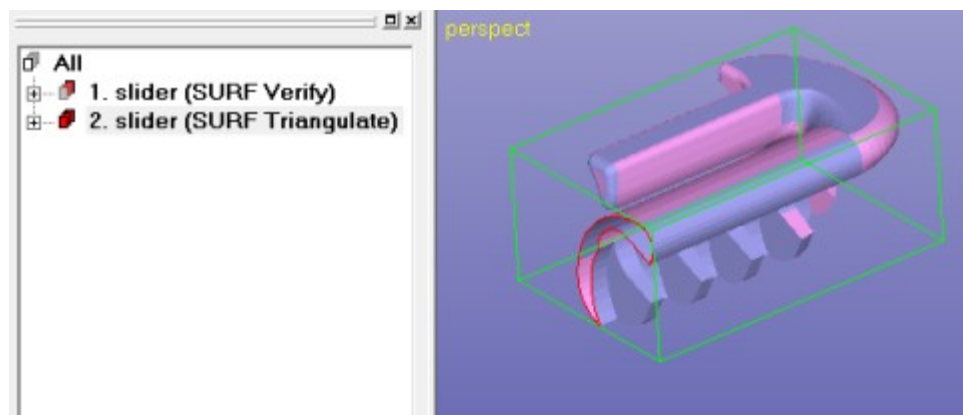
Before continuing let's


- **change the *STL auto-repair* setting to *Do not repair*.**

We do not want to run the triangle repair automatically after the triangulation this time.

- **Press *OK* to start the triangulation.**

After a while you should see the following result on the display area and the Model Tree:



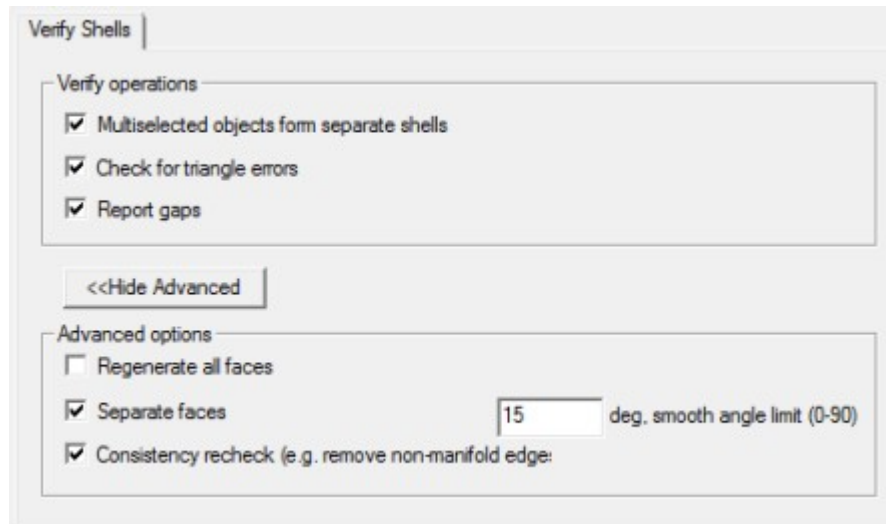
Inspecting the display and the Model Tree shows us the same gap as after the *Modify Surfaces > Verify* command. Also, some triangles are shown in pink and some triangles in blue color. The triangle outside is shown in pink and inside in blue the same way as it was with the surfaces. You may change the shading to temporarily to *Shaded + Wireframe* mode () to see the individual triangles.

Verifying triangulated models

We will continue with triangle verification to check any errors in the triangulated model.

- **Give the command *Fix Model > Verify Shells* for the 2. slider (*SURF Triangulate*) component**

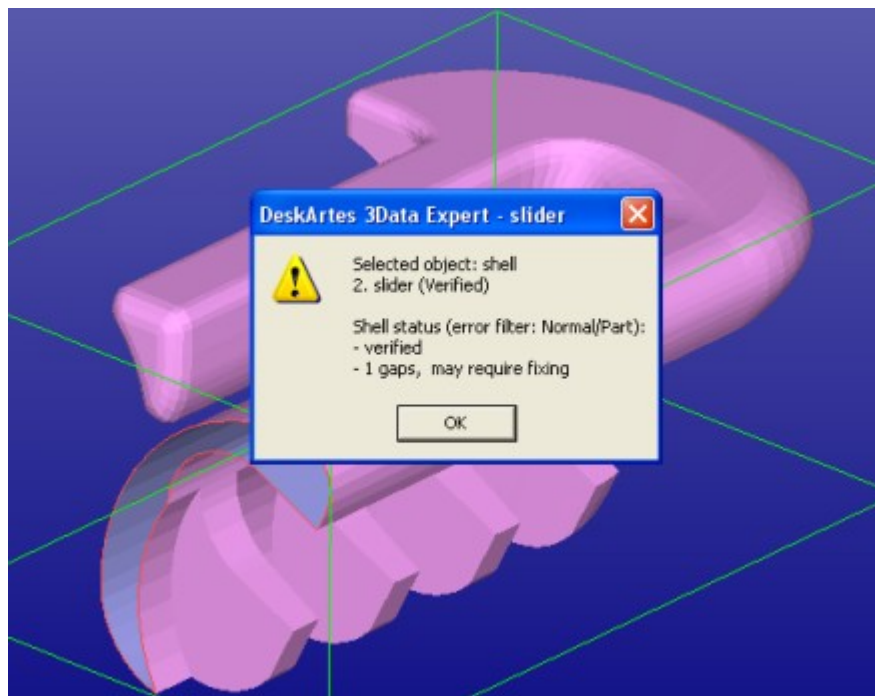
and you will see the Verify Shells settings dialog (if you do not see all the parameters, press *Show Advanced* button):



We want to know all the gaps and check for possible intersections in the triangulated model.

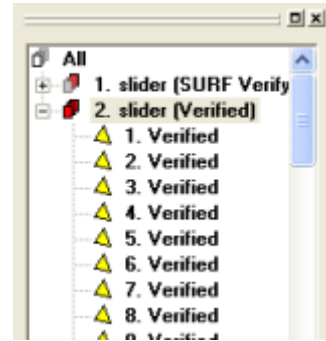
- Please **check that your settings are the same as in the picture above**.

Then **press OK to start the processing**. You should see the following result



which reports us the same gaps as with the previous commands. The drawing area will show you the properly oriented *slider* with *gaps*. The Tools Window with Fix Model Tab is opened if not already open.

A closer look at the Model Tree will show us the resulting *Verified* triangle surfaces. During the verification the model is subdivided into separate triangle surfaces according to the parameters given to the *Fix Model > Verify Shells* command. For more details please have a look at the On-line Help of the *Verify Shells* command. The resulting Model Tree component will have *265 Triangle* surfaces and one *Gaps* element in the end of the component.



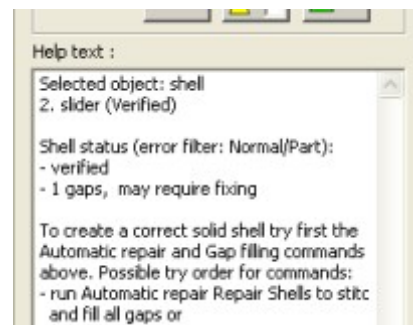
The model outside is now displayed with pink color. This is due to the fact that the verification also orients the triangle normals consistently, as seen in the picture on the previous page.

The surface subdivision of the verified (or repaired) model will allow us to use the different surfaces efficiently for subsequent repair work, if necessary. For example, separate surface can be extruded or cut against each other with *Modify Faceted > Boolean* command.

Repairing triangulated model

Now it is time to produce a solid STL model for 3D Printing. We could have done this immediately after the triangulation, but for educational purposes we performed the Verification of the triangle data first.

Now you should have a look at the *Help Text window* in the *Fix Model* tab. The model status is given, as well as help on what command to run next to produce a correct model from the selected shell.

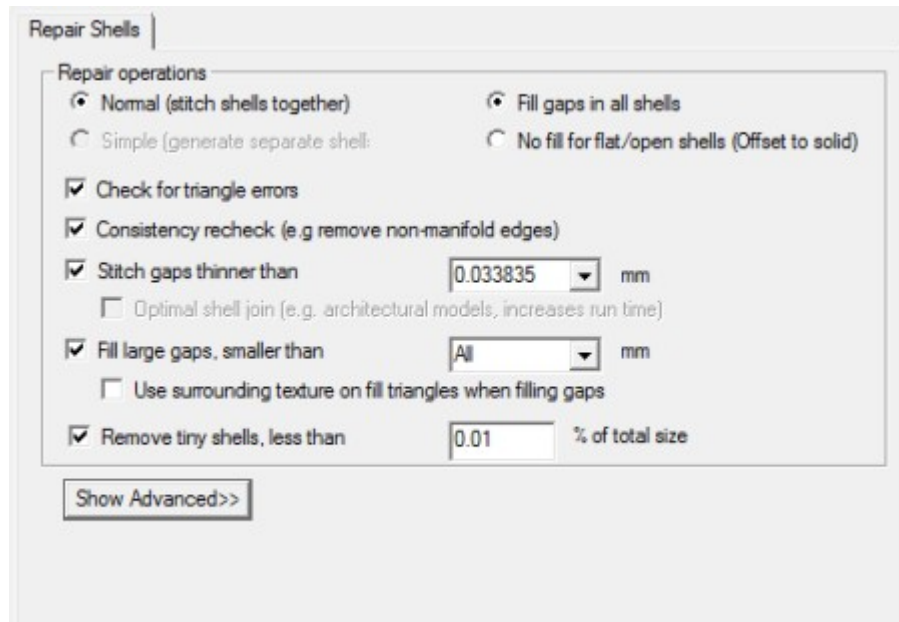


- Start the command ***Fix Model > Repair Shells*** with the *2. slider (STL Verify)* node.

You can also use the shortcut in the Fix Model Tab:



The Repair Shells settings dialog will appear after the command is given:

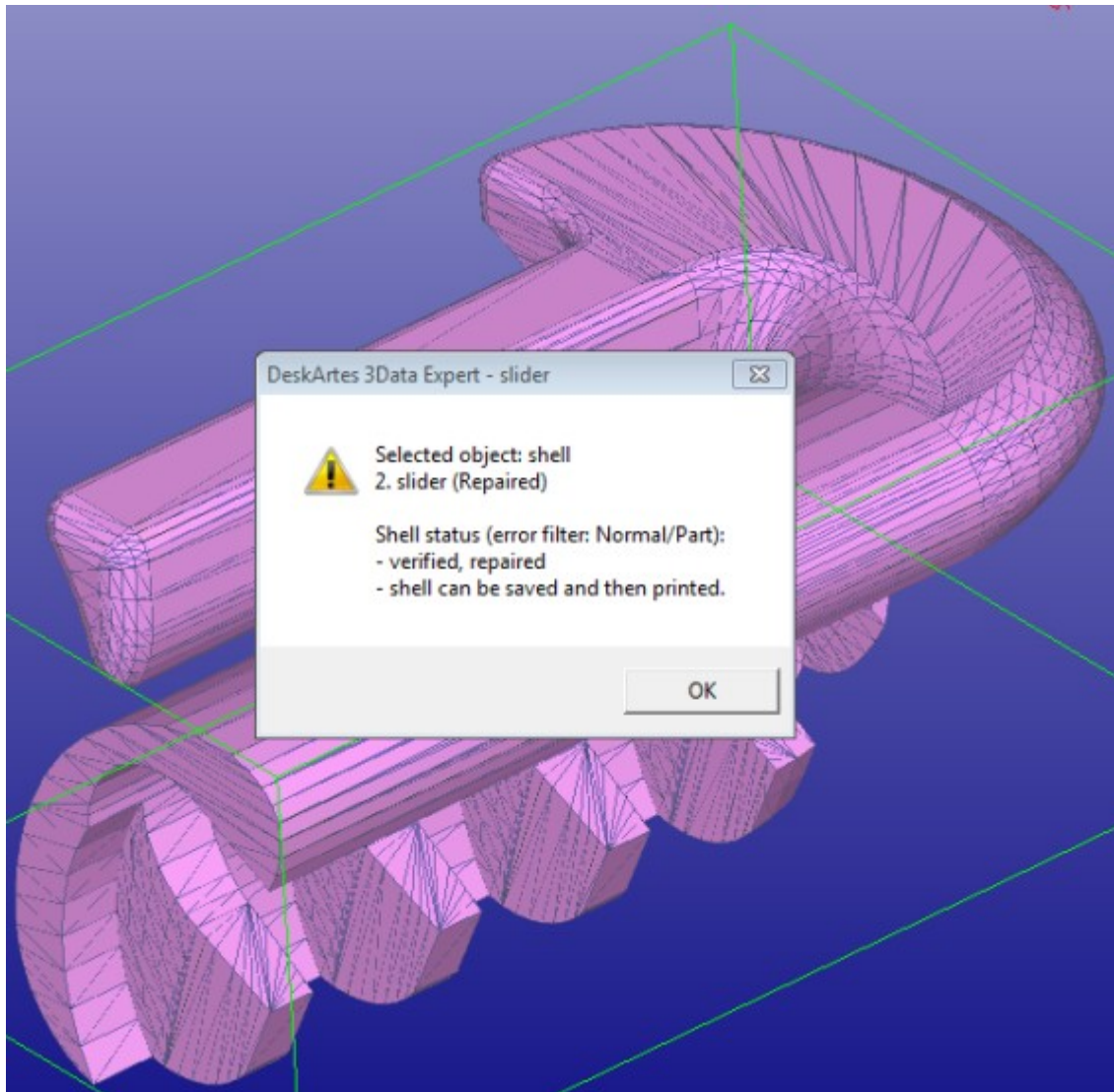


Again, we will *Check for triangle errors* in the data. Now we will also close gaps in the model. The *Stitch gaps smaller than* parameter gives the upper limit to sew together thin gaps between the surfaces (automatically calculated for each model). The *Fill gaps smaller than* parameter defines the maximum size for filling individual large gaps. Here we want to close *All* of them.

- Make sure you have selected the parameters above and **start the command by pressing OK**.

Note: The Advanced parameters are needed when you are working with models containing difficult errors. You can learn to work with difficult models by downloading the “STL and VRML repair examples” document and associated example files through the Help > Tutorial Documents command.

The result will be a correct STL model. The display area shows a pink model without any gaps. This means that we only see the outsides of the triangles, i.e. all triangles are correctly oriented.



The Model Tree indicates a correct model with the green icon in front of node 2. *slider (1. Repair)*. If you open the Model Tree and select individual surfaces, you will see that the surfaces are sorted by the surface area. The first *Triangles* surfaces is the largest and the last surface is the smallest. You can also use graphical selection to show individual surfaces by clicking on them with the left mouse button.

The message and green icon in the Model Tree indicates that the shell is now correct and can be printed with a 3D Printer. **Press OK** to close the status dialog.

Lesson 9 – 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 better aspect ration. 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 better aspect ratio to produce most reliable results.

3Data Expert *Reduction* command can also be used for preprocessing data for Augmenter 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 accurate as possible after the reduction.

Green box indicates if this lesson is applicable to given module:

■ View ■ Dimensions ■ Lite ■ 3Data ■ Sim

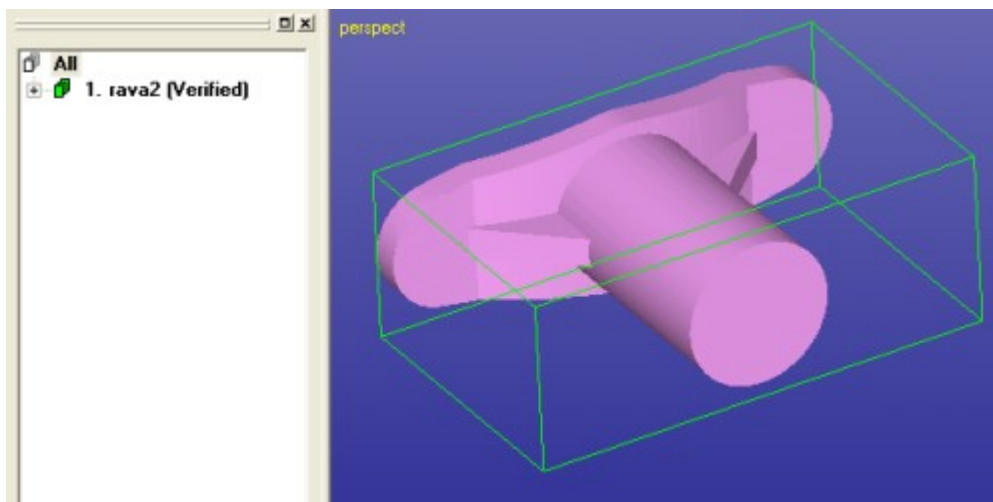
Starting the reduction

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 Model* 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**.

Reduce parameter dialog will appear:

Main reduction method

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

Main filters

- ☒ Edge length
 - units, minimum edge length
 - maximum length multiplier
- ☐ Number of triangles
 - maximum triangle count
- ☐ Surface deviation
 - units, maximum deviation

Geometry feature filters

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

Color and texture feature filters

- ☐ Keep texture edges
 - degrees, sharp texture edge angle
- ☐ Keep color edges
 - degrees, sharp color edge angle

OK Defaults Cancel

There are three different main reduction methods and you can affect the result by selecting the most suitable method 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.

Which ever method is used, the triangles are reduced so that the given limits are respected. Anyhow, at all times 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 requiring better triangle aspect ratio. **Note:** *With Minimum edge length method you can use **Surface deviation filter** in addition to the basic Edge length filter to have even more control over the surface shape change.*
- **Number of triangles** filter will give the **maximum triangle count** left 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 with the user defined limit, all corners shaper 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 normals to the given degree value. This will very effectively prevent large changes in the model shape. For example, if a cylindric 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 degree changes are allowed and thus cylindric 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, i.e. 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 useful when reducing data for Augmented Reality, Virtual Reality or game applications. **Sharp texture edge angle** and **sharp color edge angle** work in the same fashion as **sharp open edge angle** above.

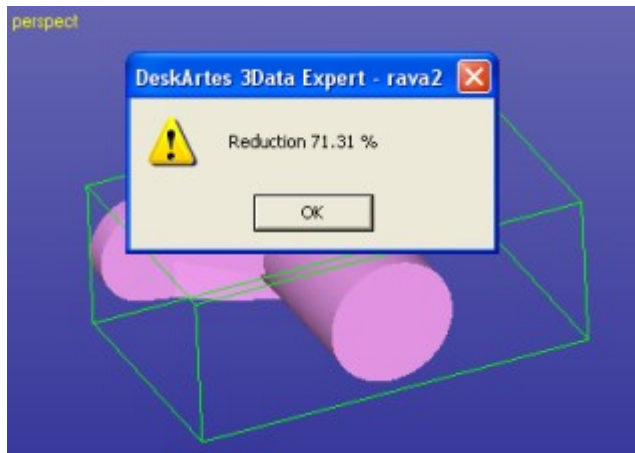
Generally, the more filters you are using, the less reduction may happen and the longer it may take to run the command. On the other hand, filters are very important for preserving desired features in the model.

Reducing rava2 model

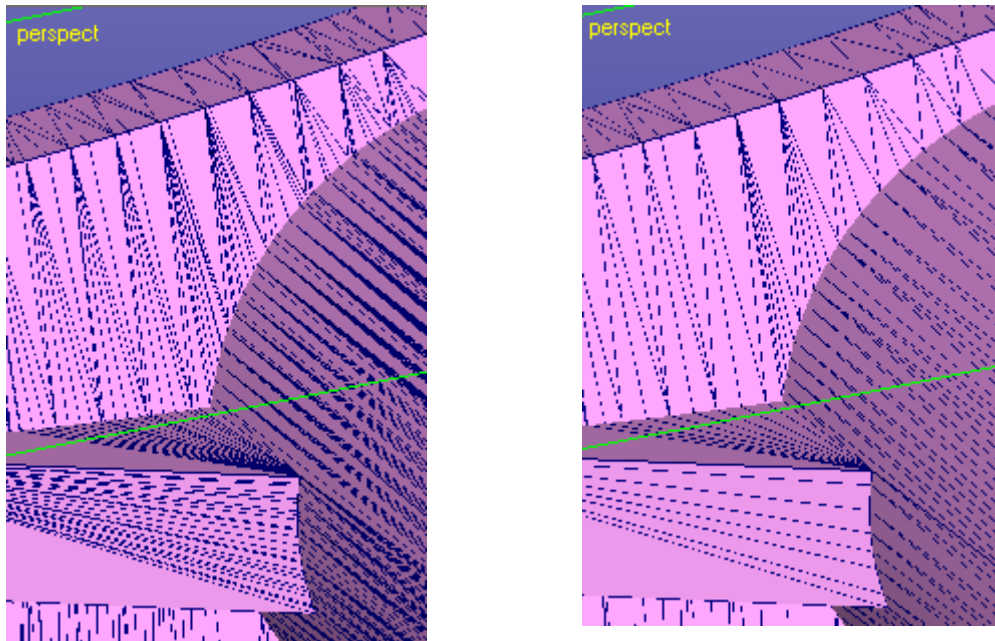
Now lets try to reduce the rava2 model with the default parameters. You can make sure the default parameters are in use by

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

You will soon see the result with 71% reduction:



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

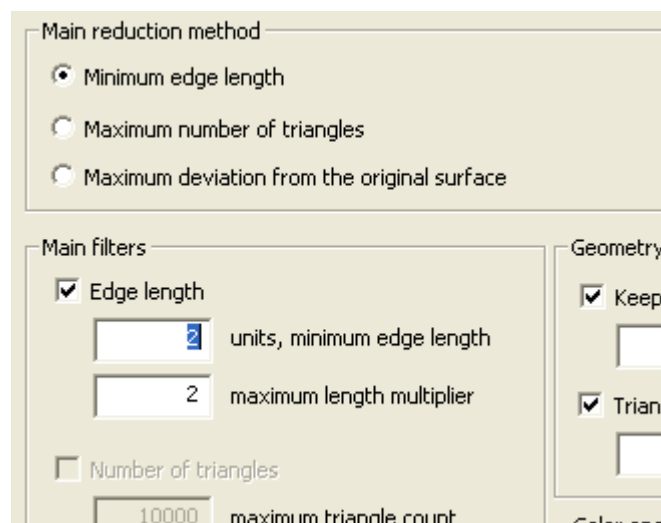


The model still looks the same but now there are only 1628 triangles left from the original 5674 triangles.

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 ($1.0 \times \text{upper limit multiplier} = 2.0$).

Let's try another reduction with 2.0 minimum edge length.

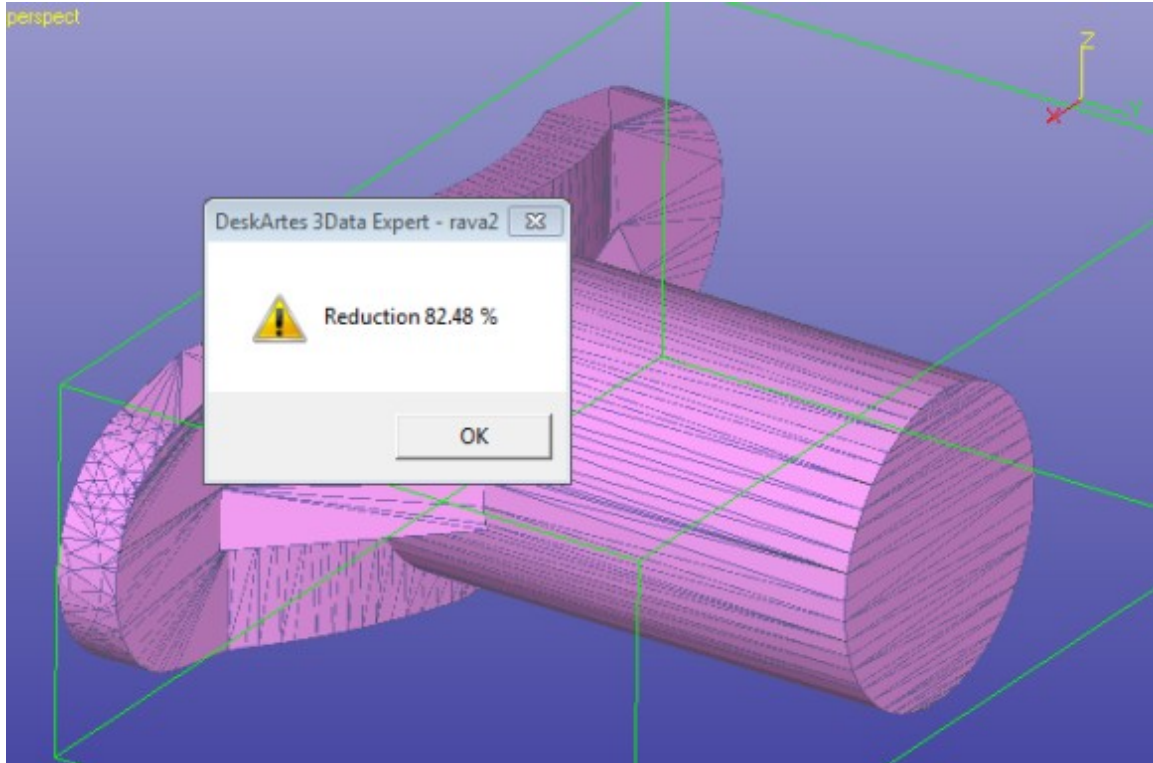
- **Press Edit -> Undo Reduce** and then
- **Modify Faceted -> Reduce** again and set the minimum edge length as seen below:



Then

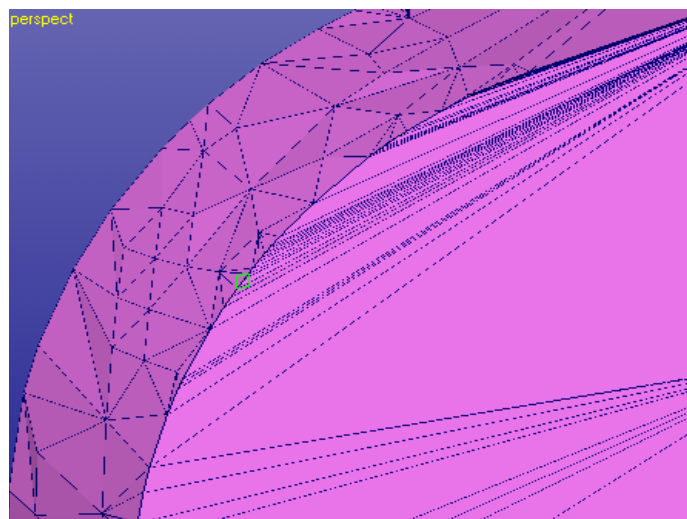
- **press OK**

and after a short while you will see the result:



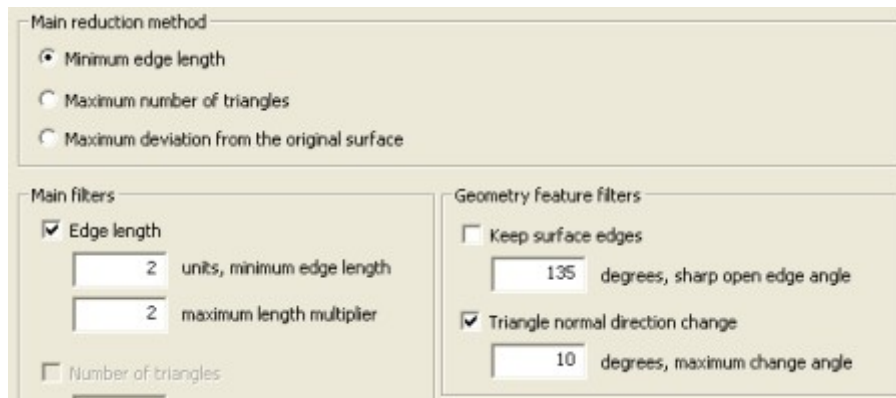
You can clearly see how more triangles are reduced from the mesh and still the model shape remains accurate.

There are some some areas where you can still find triangles with shorter edges than 2.0 mm's, as seen in the image below:



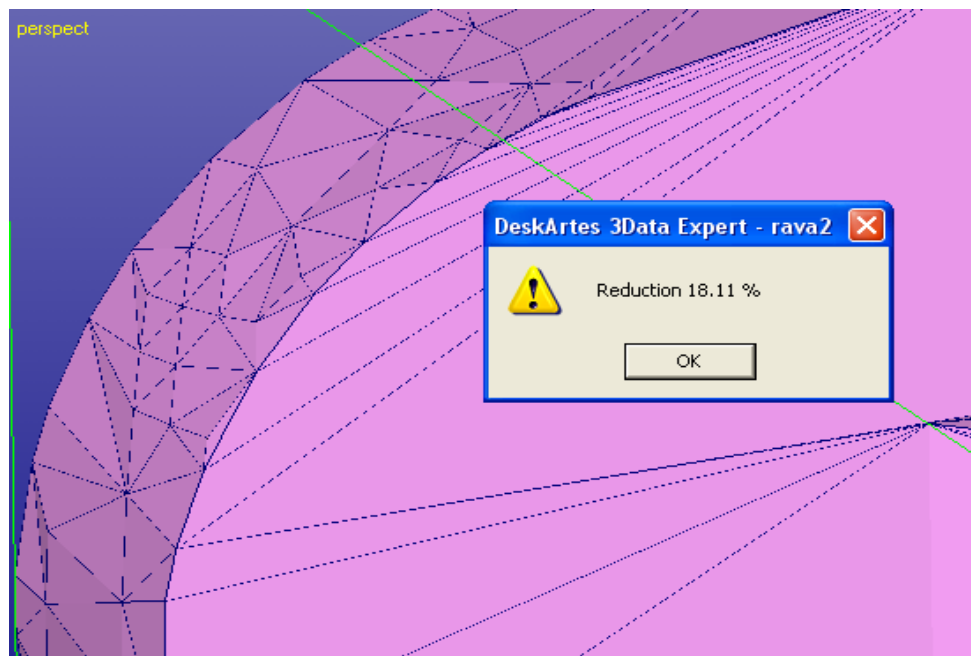
These areas are due to small one triangle surfaces in the original model (in green bounding box above), **Keep surface edges** filter prevents us from deleting any surfaces fully from the model. To remove minor surfaces like this we can make another run with the **minimum edge length** 2.0 units but

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



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

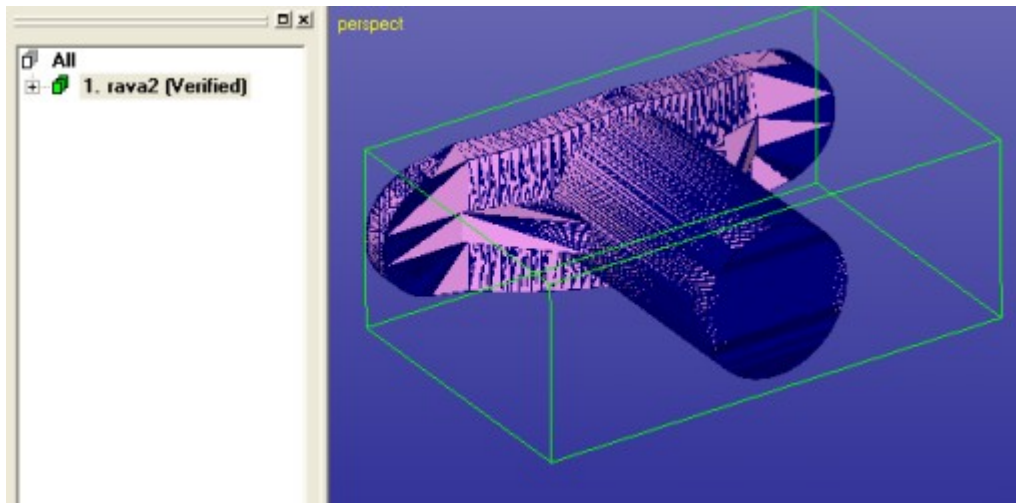
This will produce another 18% reduction by removing some small surfaces altogether.



Let's now try the other reduction methods.

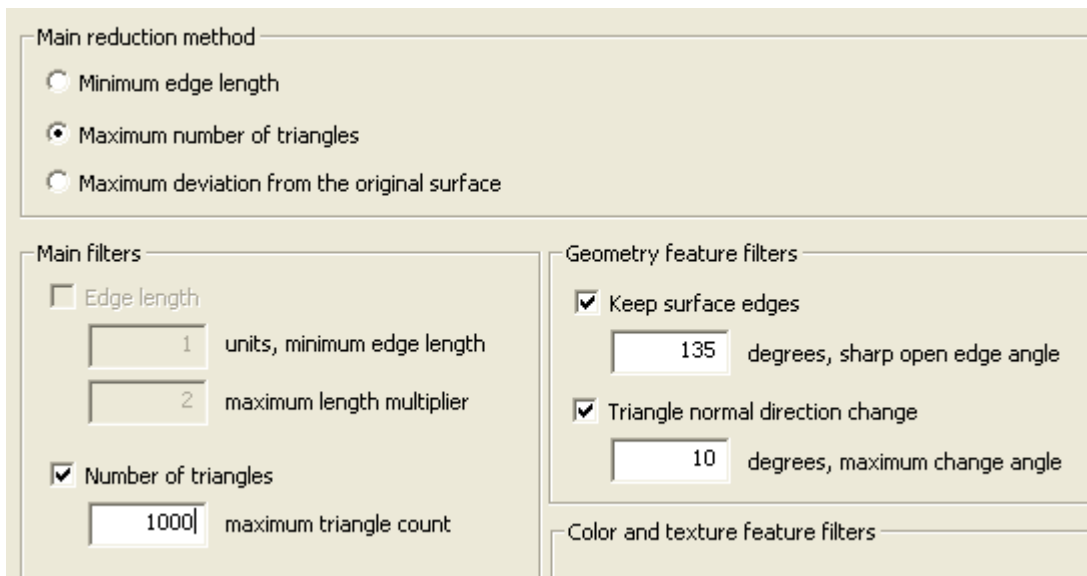
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- **Press Edit > Undo** until you have the original *rava2 (Verified)* model available again (here in shaded + wireframe display mode).

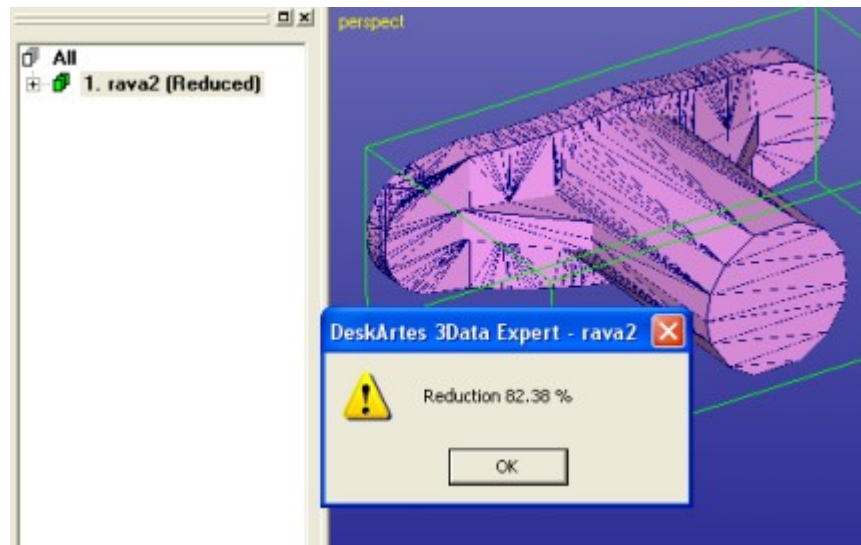


Then

- give the **Modify Faceted > Reduce** command,
- **press the Default button** and give the following parameters,
- i.e **reduce to maximum 1000 triangles**:

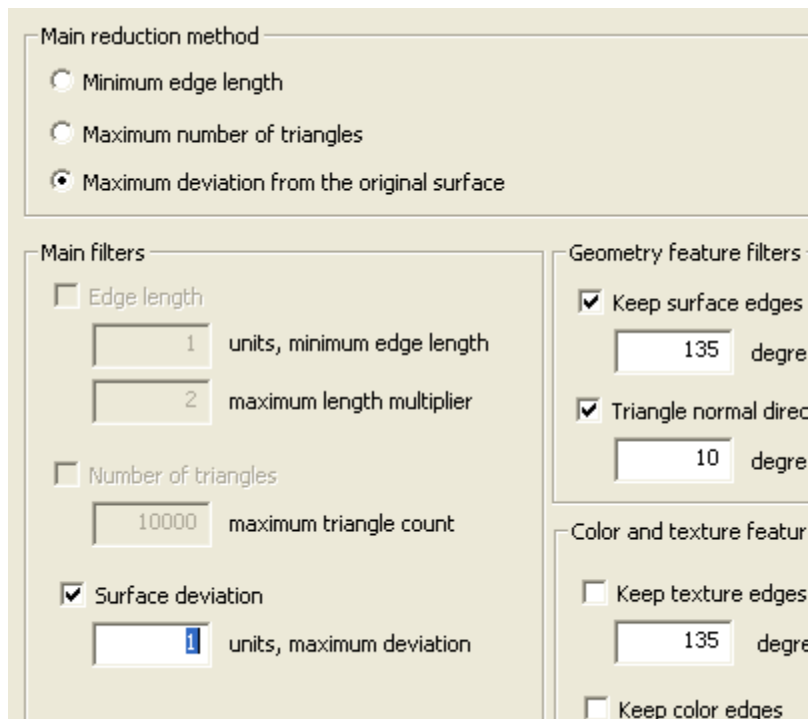


- **Press OK** to see the following result with 82% reduction and 1000 triangles:



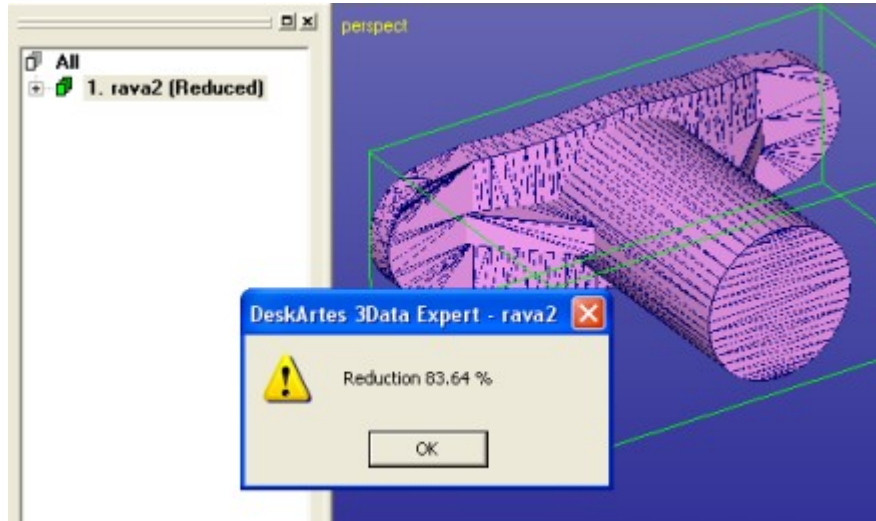
There is still one method to go, *Maximum deviation from the original surface*.

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



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

- **Press OK** and you will end up with 928 triangles:



Which reduction method to use?

As you can see there are different ways to get to reduced result, depending on the requirements set to the end 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.

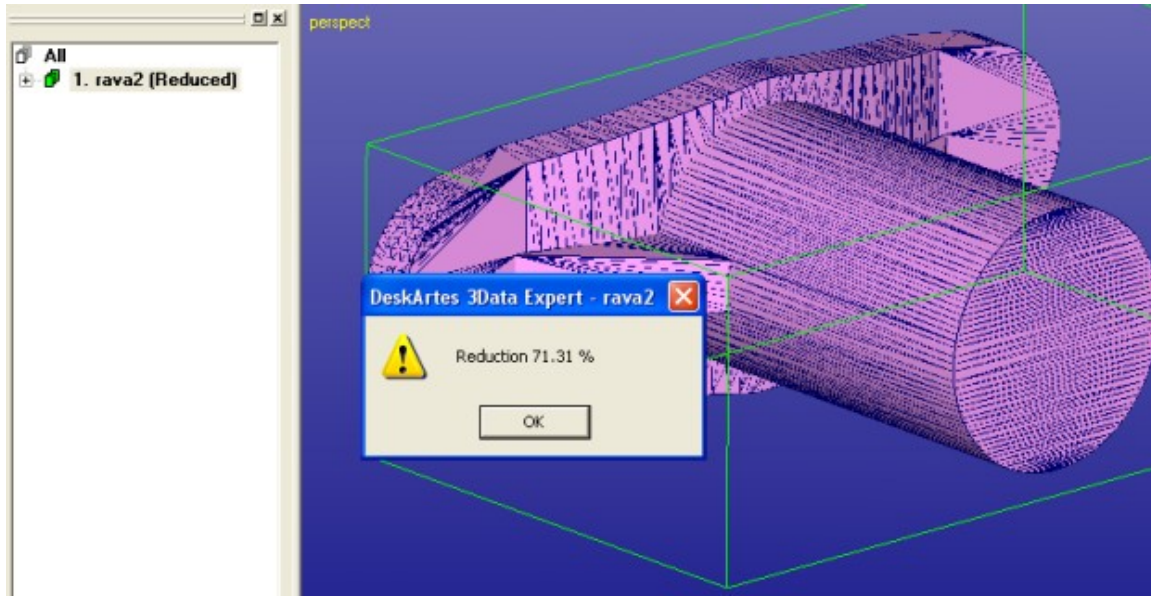
Improving the triangle aspect ratio

Triangle reduction command can also be used to create 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, better shaped triangles will generate more accurate simulation results. Better aspect ration 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,

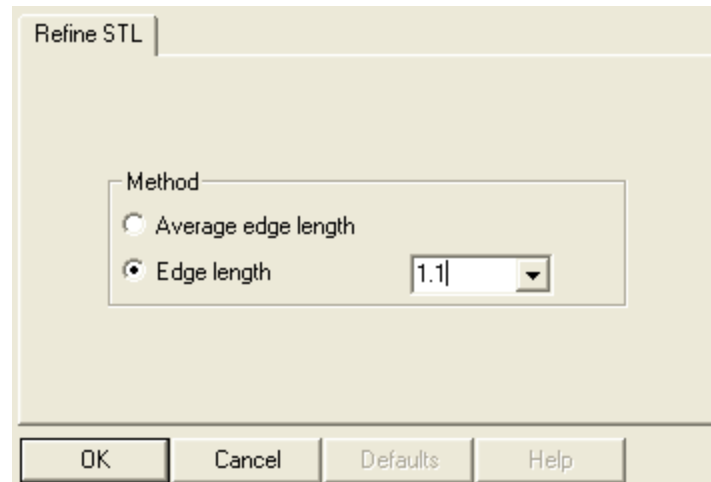
- give the command **Modify Faceted > Reduce** and press **Default** button in the Reduce parameter dialog.
- The **press OK** to get the first reduce result:



We have now removed excess triangles from the mesh. Next we will 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 run to the reduction.

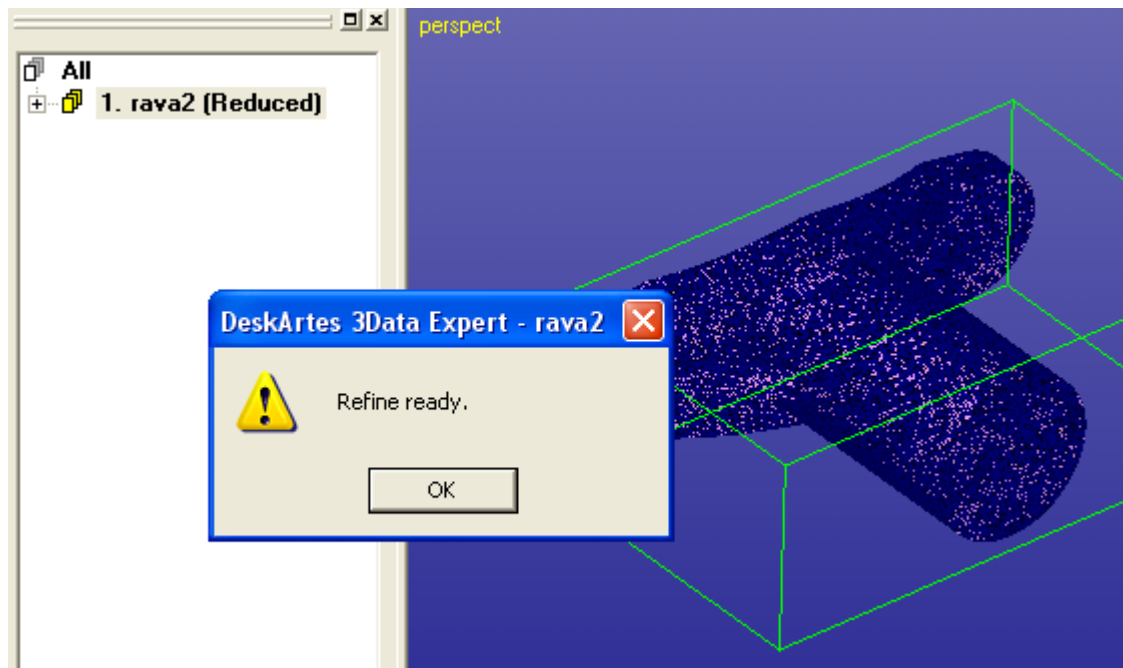
First you should estimate the minimum size of the triangles you would like to see in the resulting mesh. Let's assume we want to use approximately 1.0 mm triangle size. In this case we will first refine the mesh to maximum 1.1 mm edges and then reduce it to 1.0 mm edges to get to the desired result.

- **Select the *rava2 (Reduced)* shell** from the Model Tree.
- **Give the command *Modify Faceted > Refine***, then fill in the following parameters:



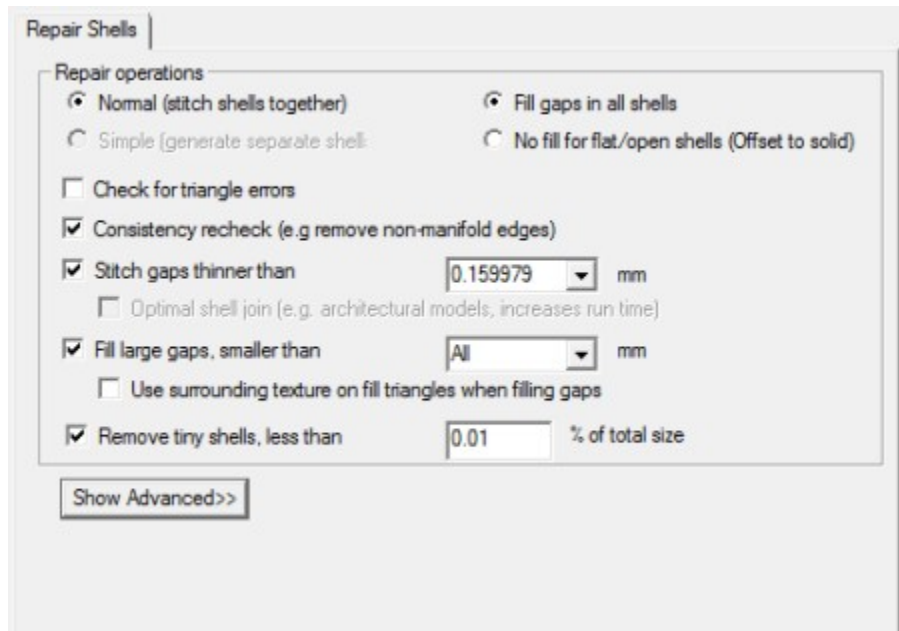
- Press **OK** to start the **Refine** command, after a while you will see the following result on the screen (next page).

The result now contains approximately 172 000 triangles. It is full of very thin triangles with bad aspect ratios. Anyhow, now we have a suitable set of new triangle corner points available for the reduction to generate new evenly shaped triangulation.



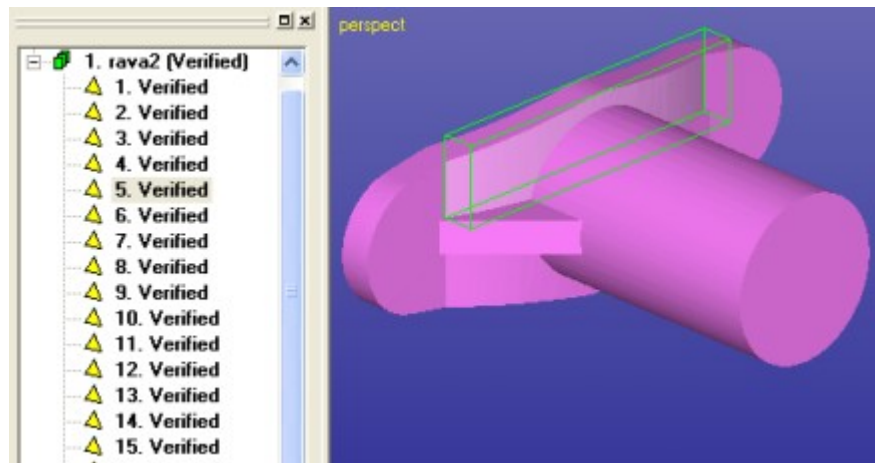
The refine result does not contain surface information in it. To be able to produce more accurate reduction results with the Keeps surface edge filter we must regenerate the surface edge information to the model. To do this you should

- run the **Fix Model > Repair Shells** command with the following parameters (for quick execution):



After

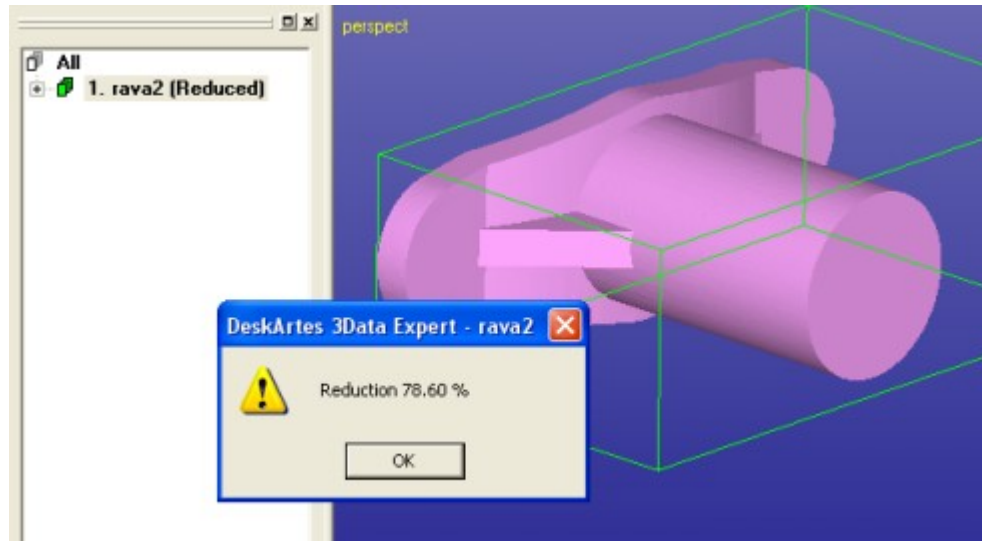
- pressing **OK** you will see the shell subdivided to surfaces again:



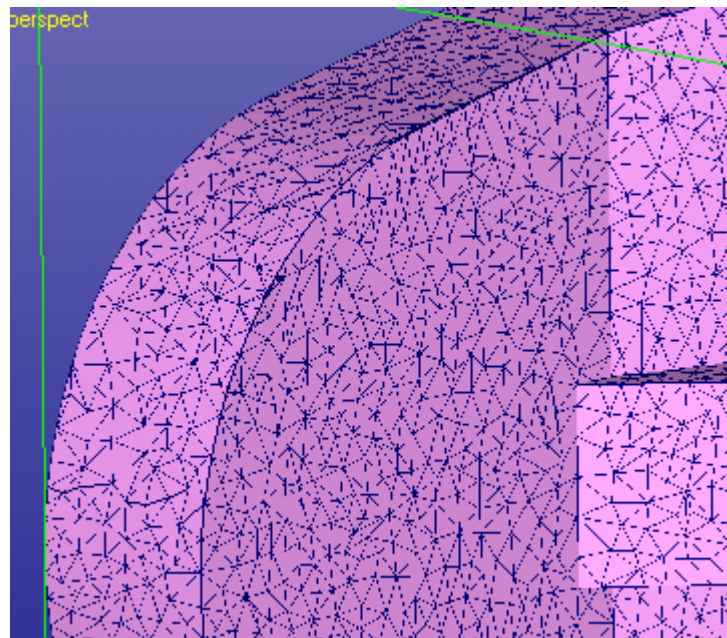
No we are ready to simply

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

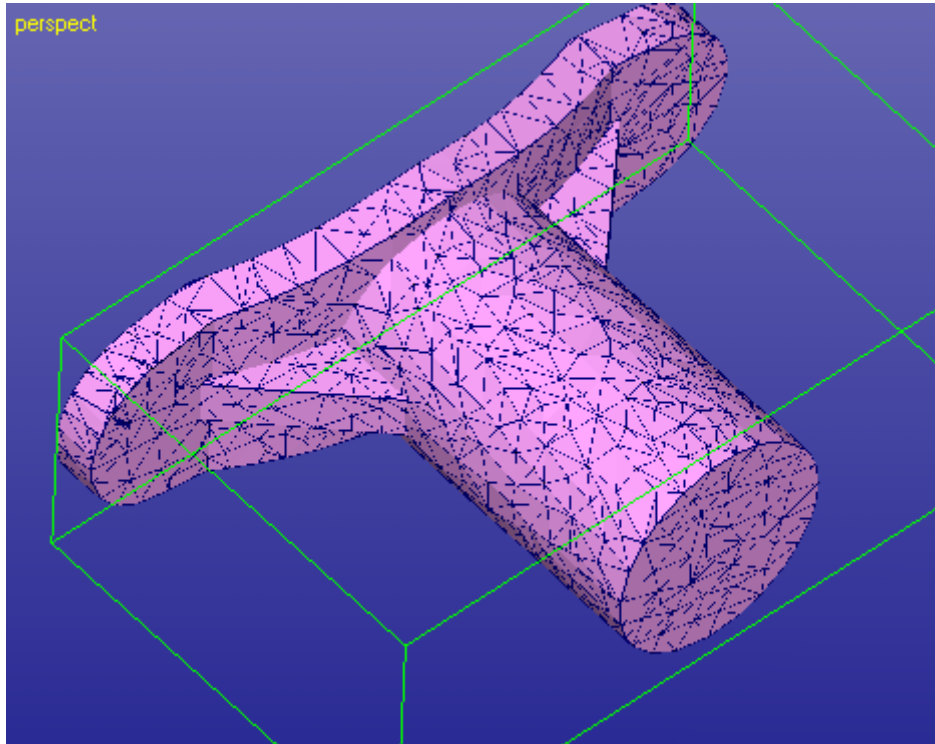
After a while you will have the result with 78% reduction. The result will contain approximately 37 000 triangles.



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 still 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 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 Expert Series software. You can now use the Expert Series software modules for everyday 3D printing work or prepare surface and STL data for simulation software packages.

The Expert Series 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 Expert Series software, please have a look at the advanced repair document: *Expert Series 10.1: 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 Expert Series 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.