DeskArtes
Expert Series 10.0

STL and VRML Repair Examples for

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

Second Edition

DeskArtes: "Expert Series 10.0: STL and VRML Repair Examples".

March 2012.

Copyright

© 2012 DESKARTES. All rights reserved.

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

Trade Marks

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

Contact Address

DESKARTES OY

Olarinluoma 7

02200 Espoo

FINLAND

http://www.deskartes.com

Expert Series 10.0: STL and VRML Repair Examples

Chapter 1 – Foreword

DeskArtes Expert Series software suite consist of several software modules: 3Data Expert, 3Data Expert Lite, Dimensions Expert, Sim Expert, Design Expert and View Expert. The software suite covers everything from 3D CAD design through photo realistic renderings and CAD viewing to STL/VRML data transfer for 3D printing, Additive Manufacturing and simulation software packages.

DeskArtes 3Data Expert 9.1: STL and VRML Repair Examples document describes the use of the latest improvements done for the faceted model repair with the Expert Series software. The use of the commands will be shown through several examples and most commands for faceted model repair are covered in the document:

- *Fix Model > Verify Shells* and *Repair Shells* commands for automatic model verify and repair,

- *Fix Model > Edit Triangles* command for manual model editing,

- *Modify Faceted > Boolean* and *Multiple Join* commands for part combination and cutting,

- *Great Geometry > Extrude Surface* to add material on surfaces,

- STL, VRML and ZPR output for 3D printers,

- and several other supporting commands.

The examples contain both STL and VRML models. The shown techniques will apply to any STL, VRML and ZPR model similarly because all are used to define faceted (i.e. triangulated) models. The repairing of STL, VRML and ZPR models is identical, the only difference is the additional color and texture handling available for VRML and ZPR models. New textures and colors can be added on any faceted model with 3Data Expert.

Most of the examples will require the use of the 3Data Expert software equipped with full set of repair, manipulation (repair, boolean, extrude, edit triangles, …) and texturin/coloring commands for triangulated models.

The repaired models include models with variable level of difficulty. These models are selected due to the errors in them. Normally you should not meet model with such large amount of errors if designed and outputted correctly from the sending CAD system. The models include a cell phone, a human head and Maya demo part are included in the installation package *tutorials/GeomFiles* directory.

All commands and their parameters are explained in detail in the On-line Help of the Expert Series software. If in doubt please check the meaning of each parameter from the On-line Help.

If you are using Expert Series the first time, you should reserve about four hours to go through all examples.

The operations given in **boldface** in this document are commands which affect the geometry of the part in question 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 must be used to be able to generate a correct result. Terms given in *italics* indicate file names, commands, shell names, surface names and parameters for different commands.

The full set of tutorials for Expert Series include and it is recommended to go them through in the given order:

- *Expert Series 10.0: 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 10.0: 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 10.0: 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.

- *Expert Series 10.0: 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.

Chapter 2 – The cell phone model

The first model to repair is a cellphone model *(tutorials/GeomFiles/cellphone.stl)* You will need the 3Data Expert, 3Data Expert Lite or Sim Expert to complete this exercise in about 10 minutes. You can also use Dimensions Expert until the paragraph *Combining the cellphone components* at page 25, which requires Boolean commands*.*

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

🟥 View   🟩 Dimensions   🟩 Lite   🟩 3Data   🟩 Sim

Now

- **input the STL file with *File > Open*** command you can see the model on the graphics area.

For each inputted file the software will ask for the units if not available in the input file.

- Select default *Millimeter* units and **press *OK:***

:
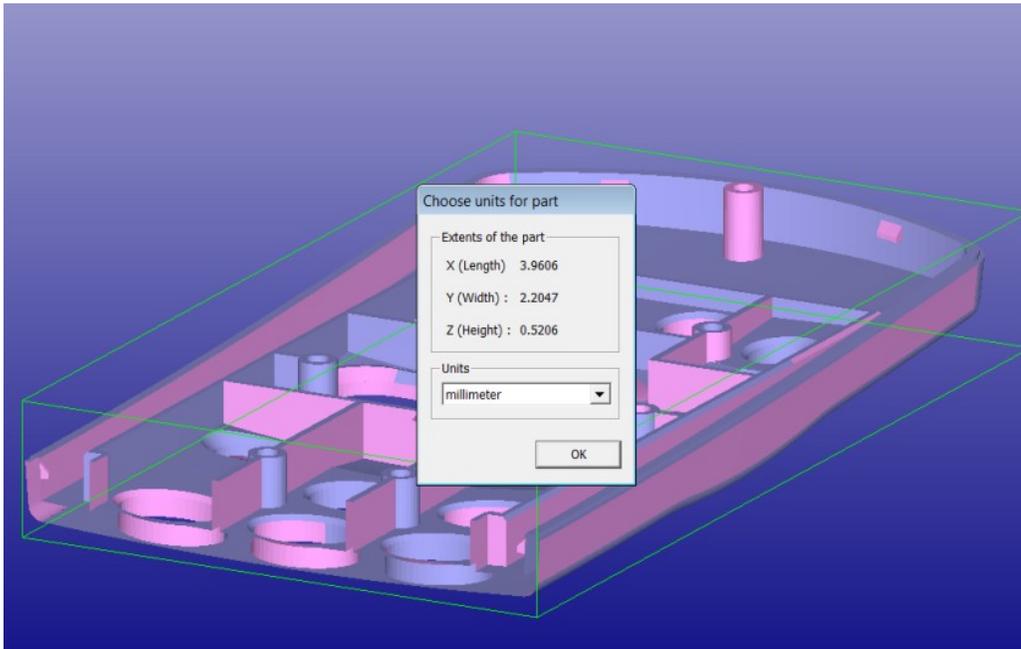
When the units are set the software will ask for the *Operation Mode*:
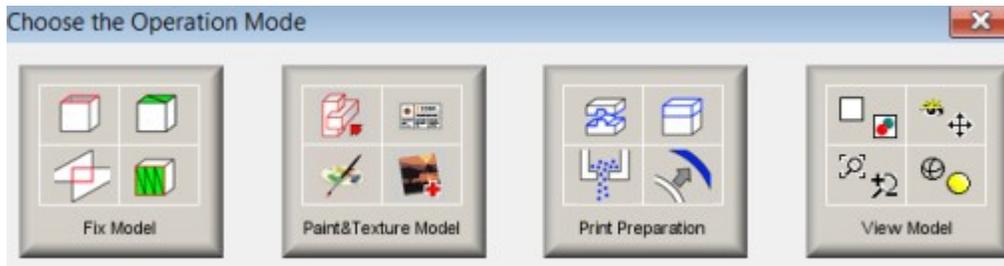
Operation Mode Window gives a reminder on the possible modification processes you can perform on a faceted model and allows you to select the correct set of tools for any operation. **Note**: if you do not see the Operation Mode window, please make sure the *Show operation mode dialog when opening a file* is *On* in the *Edit > Preferences* General Tab:
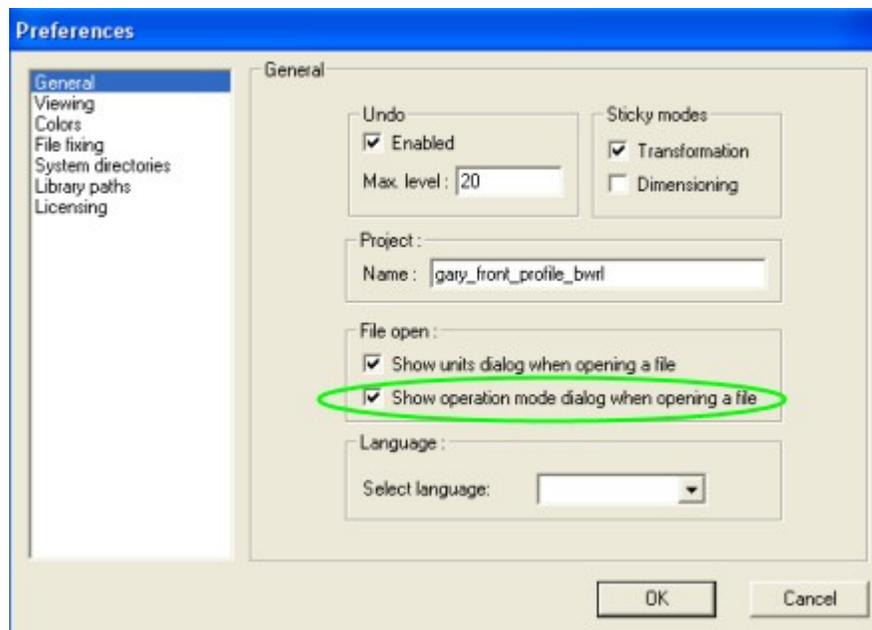
The differences in the triangle colors on the display (blue vs. pink) are due to the fact that the triangle normals are not consistently oriented. The blue colors marks the inner side (inverted normal) and pink color marks the outer side (normal) of the triangles.

Accept

- **Fix Model mode** and let the software run until you will see the following message:
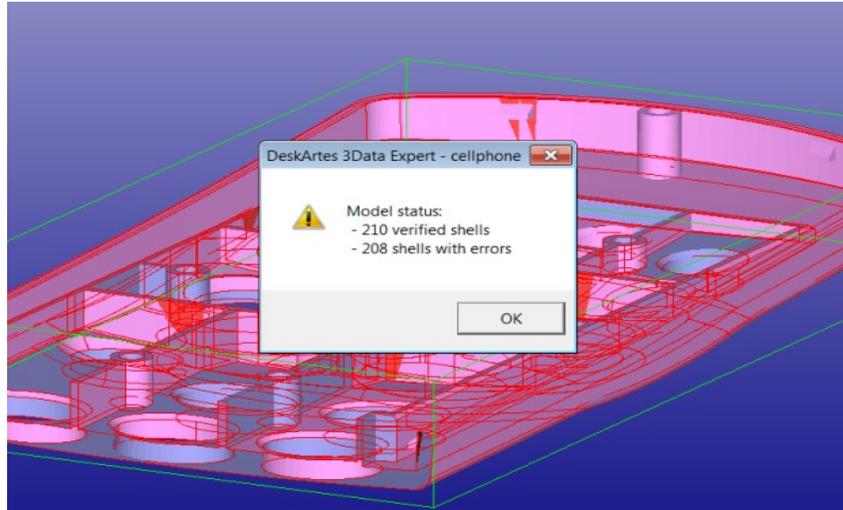
This is the result of the automatic analysis before the Auto Repair (the same result can be achieved through *Fix Model > Verify Shells* command if Operation Mode Window is not used).

We can see that most of the 210 shells are separate from each other (red gap curves) but obviously should be connected into one single shell. It is up to taste either to run the Auto Repair with this model as the open flat shells are left non repaired for more controlled repair operations (to stitch with neighboring shells) later. Anyhow, running Auto Repair may generate solid shells from shells which should be connected to neighboring shells.
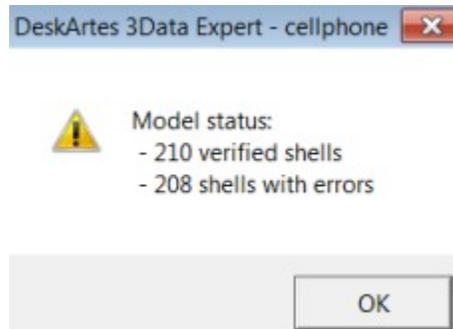

Now

– **Press No** to the *Do you want to run Auto Repair* question. After pressing No the status dialog appears:

This indicates that most shells have errors and we should use Model Tree and Help Text to repair the model.

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

You may see the Command Tip window (if not seen give the command *Help > Show Command Tips* before running the analysis again):
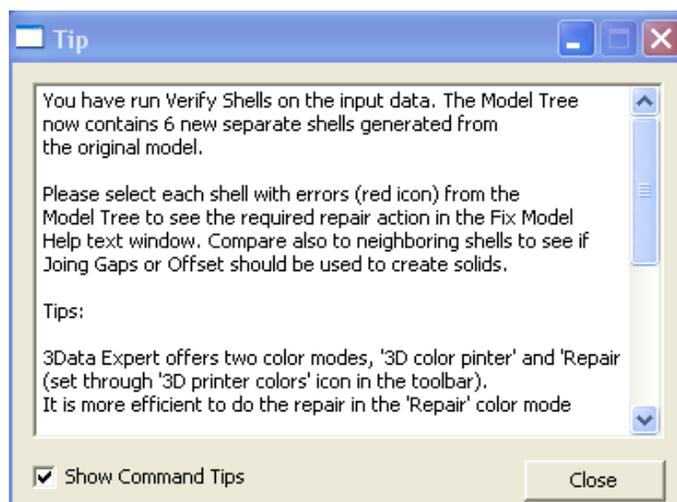
The Tip window gives useful information after different commands. For example, this tip window proposes to use the Fix Model Help Text window as well as visual checking to decide the next repair actions.

- **Press Close** when ready reading the tip.

The analysis shows that the part is made of many (210) non connected components or shells. The first task will be connecting these components into meaningful part(s). The repair is started by visual inspection due to the large number of disconnected shells in the model. We can also see that some triangle normals are incorrectly oriented (blue inner side is displayed) and a surface is missing from the flat end of the part.

After the verification the components are listed according to their surface area in the Model Tree. If you take a look at the end of the Model Tree and display the last component *210* (F4) on the screen and then the whole model you can see that this part is an very thin triangle stripe and thus

> – shell 210 can be **deleted** (this is not absolutely necessary but can be done for practice).
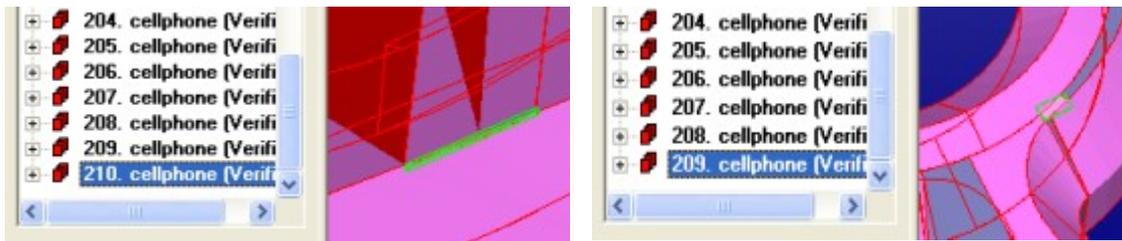
The next component to check is the 209. The component *209* is already a meaningful square surface needed to generate a good part.

With a short visual inspection we can also see that there are alt least two bigger open shells (or surfaces) that we can remove (shells *107* and *182*).

> – **Remove first the component *182*:**
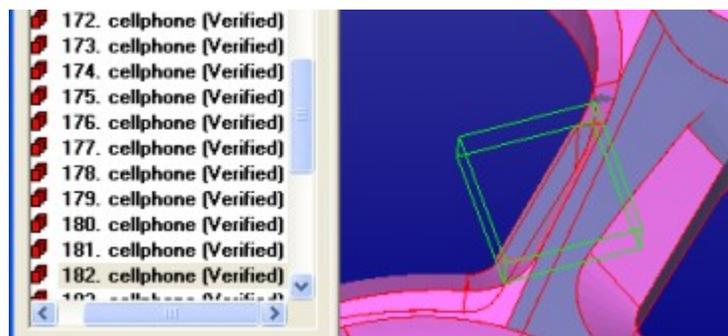
Note: If you did not remove these open shells now, you would notice these error areas after the first Repair run and could use Undo to get back to delete and rerun the Repair.

> – **Then remove the shell 107**, the gaps below the shell will be filled automatically later:
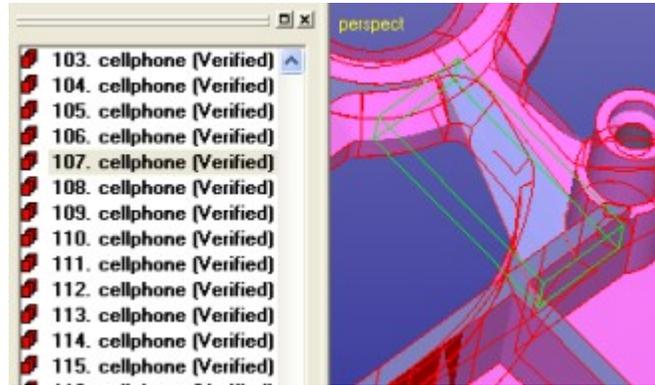
You may find more surfaces with errors (red areas indicating overlapping, intersecting, duplicate etc. triangles). You may sometimes remove more small shells or surfaces if you think are not needed when creating the correct model. Generally, very thin or flat surfaces can be removed, they will be fixed with gap stitching and gap filling later. Anyhow, with this exercise we will remove remaining errors using the help of *Fix Model > Locate Errors* command later.

Useful Toolbars

Before continuing the exercises you may want to turn on the Transform Toolbar:

and the Errors Toolbar:

and the Dimensions Toolbar:

All are turned on via the View > Toolbars menu and contain shortcuts to useful commands for repair.

## Repairing the *cellphone* for the first time

Now we are ready to run the first *Fix Model > Repair Shells* command on the model to connect the separate shells (surfaces) into meaningful shell or shells. First

---

- **select** the shell *1* from the Model Tree wih LMB (left mouse button) and then select the last component *207* while keeping the *Sift* button down.

This will create a multi-selection containing all separate components:
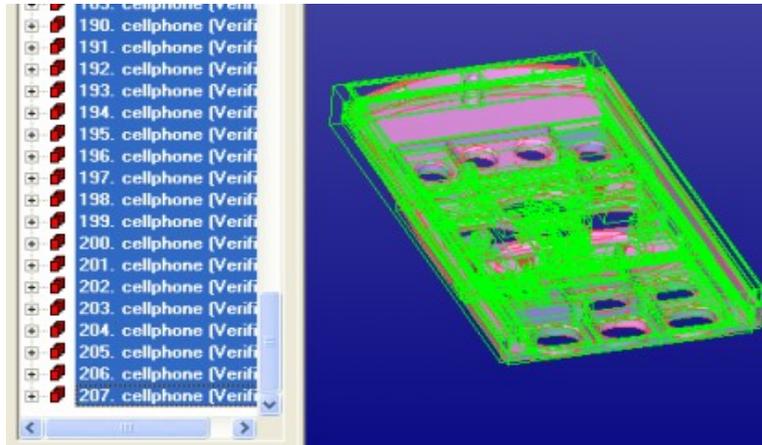
You may also select the Model Tree root *All* before running the command to get all shells selected automatically for the command.

Now

- **Run the *Fix Model > Repair Shells*** command

As mentioned before, the first task is to connect or stitch the open shells into meaningful closed shells. For that we will use the *Stitch gap* function in the repair command to connect disjoint open shells (surfaces). We will use the automatically calculated default parameters values now, especially

- make sure that the ***Multiselected shells form separate parts*** is ***not*** set,

this will ensure that the stitching is done between the edges of separate shells. Your parameter dialog should look as seen below before continuing:
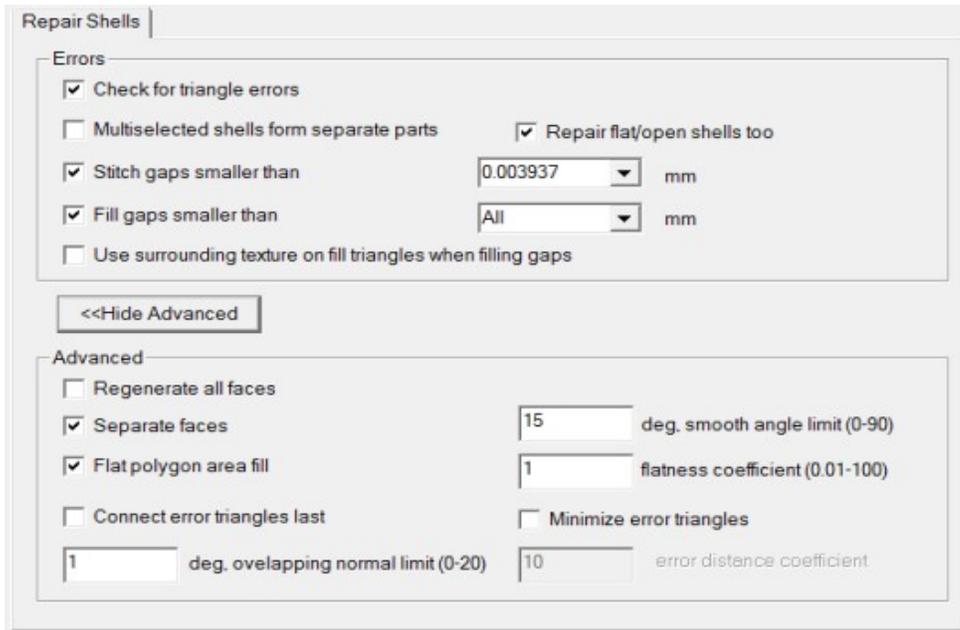
Flat open shells may be sometimes better to fix using Offset (or Join Gaps) to generate solid shells. Normal Auto Repair process does not try to fix flat open shell for that reason. But now we want to fix all shells we have selected i.e. stitch them to neighboring shells, so you make sure "*Repair flat/open shells too*" is taicked and

– **Press OK** to start the command.

After a while the command finishes and the status is displayed (for the first shell). Now we have significantly fewer components left in the Model Tree, as seen below. The components are sorted in the order of decreasing area and the status of the last selected shell is displayed:
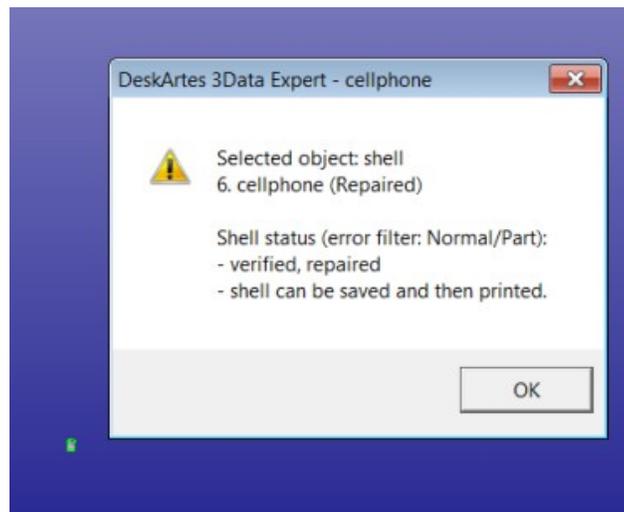
- **Press OK** to close the status dialog.

You should have 6 shells in the Model Tree.

- **Now use the multi-selection** in the Model Tree to see each shell bounding box:
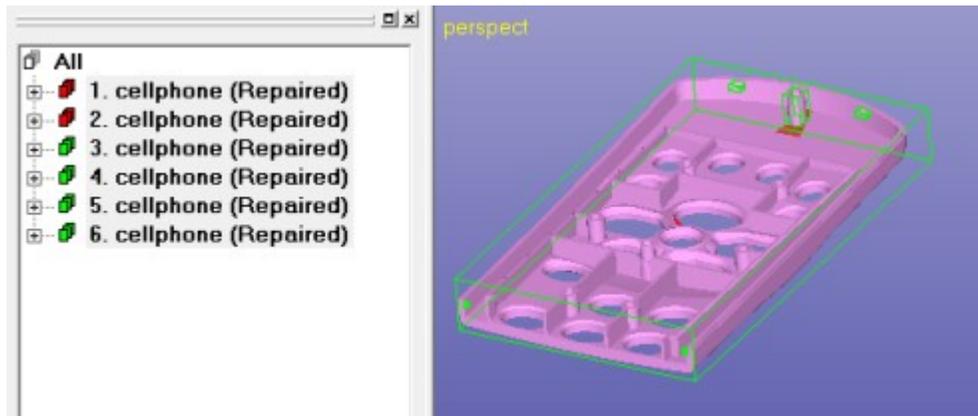

We have now manged to combine over 200 separate (open surface) components into 6 useful components. The next step is to repair each component separately to a correct STL file and then use the Boolean operations to form one final part. The next step is to first to repair each shell into green state and then use the Boolean commands to join them into one component.

Note: if you have more than 6 shells in the Model Tree, you may delete the the small extra shells caused by overlapping surface areas in the model.

Locating and removing errors in the body

Let's now give a more descriptive name to each component: *body*, *cylinder*, u*pbox1, upbox2, downbox1* and *downbox2*. The first task is to repair the *body* component*:*
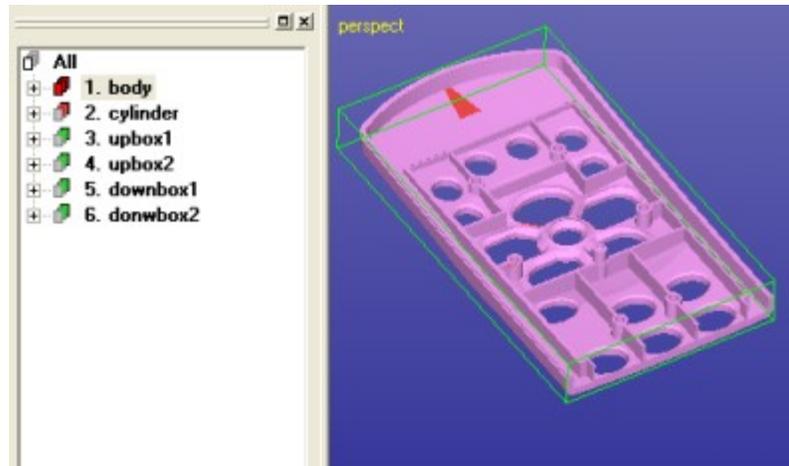
The *body* component has several errors due to the bad quality of the input data.

– **Select** the *body* shell now

After selecting the body shell in the Model Tree, the *Help Text* window on the right hand side of the window gives a textual description on the errors.

In the image above we can see the red areas on the top of the component, these are caused by the other components intersecting the *body* (*cylinder*). Also, in the image next page we can see areas which have both error triangles and complex gaps caused by duplicate and overlapping surfaces in the original input model (if you removed more duplicate/overlapping surfaces in earlier you may have less errors now).
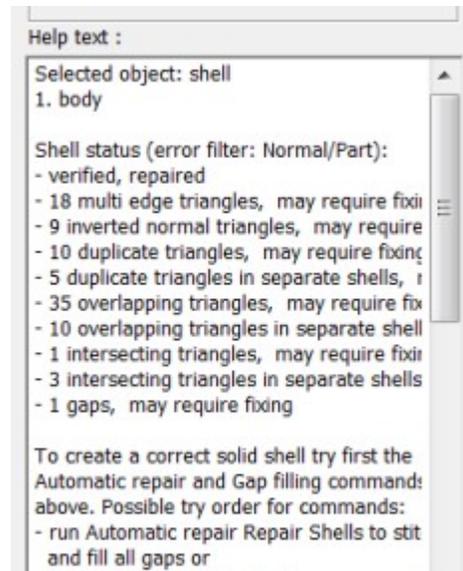
– **give the command *Fix Model > Locate Errors*** or press the shortcut

button. You will see the Locate Errors dialog appear and error triangles highlighted in the *body* shell:
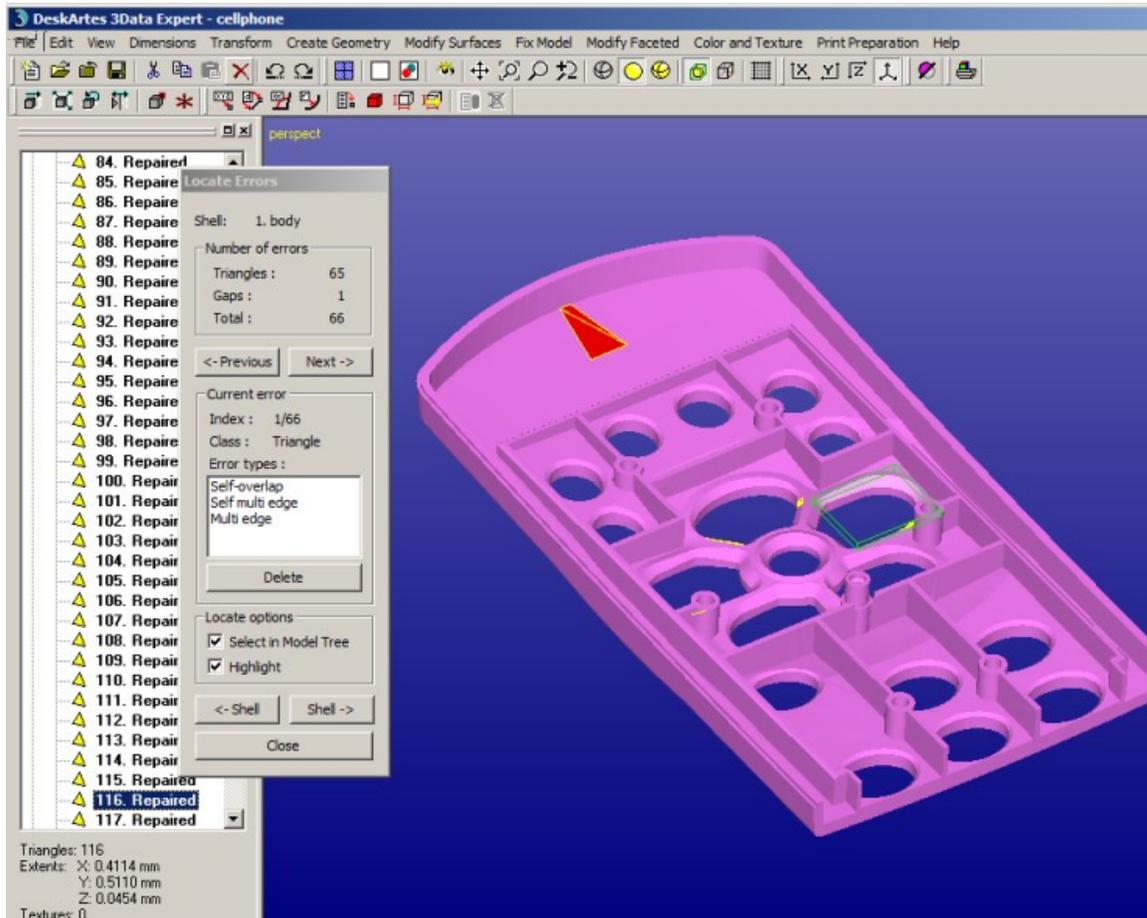
The Locate Errors dialog gives exact information on the currently highlighted error triangle (or gap), *Current error*. Also, the surface containing the error triangle is automatically selected in the Model Tree, as can be seen in the image above.

We will now use the *Previous* and *Next* buttons to find each area with error triangles and use surface *selection and deletion* as well as *Fix Model > Triangle Edit* to remove or add triangles to enable correct fixing with the next call to *Fix Model > Repair Shells* command. We will not use the *Delete* button available with Locate Errors tool this time, now it is better to use the red error triangles to indicate the error areas for surface selection and deletion.

- **Press *Next->* button** in the dialog and you should see the following error fitted in the middle of the screen:
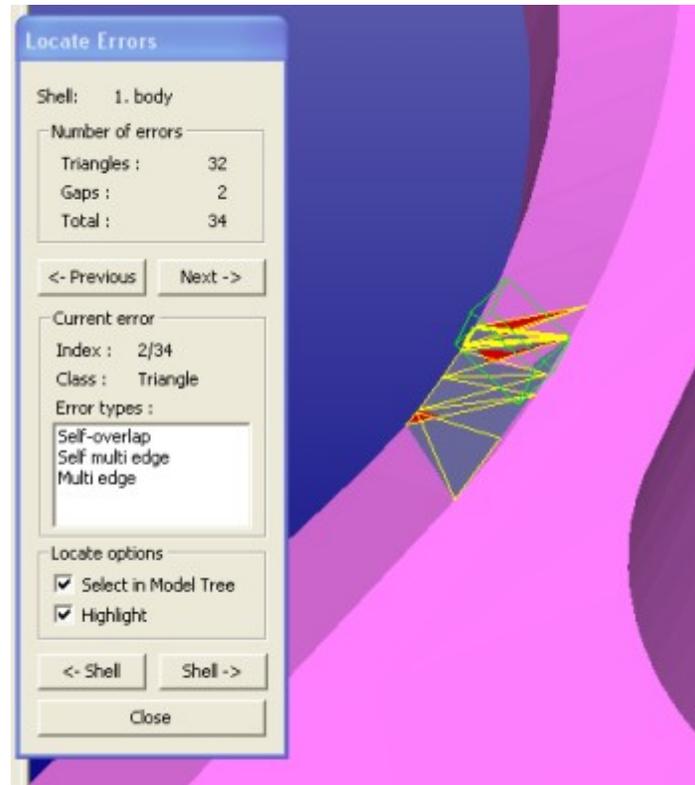
You can use the *mouse wheel zoom* to zoom closer/farther to the area pointed with the mouse and *middle mouse rotation* to change the view point.

This error area is caused by some overlapping surfaces in the model. You can have a more detailed explanation on each error (red) triangle in the screen by

  – **clicking it with the *right mouse button* on the screen on an error triangle**:
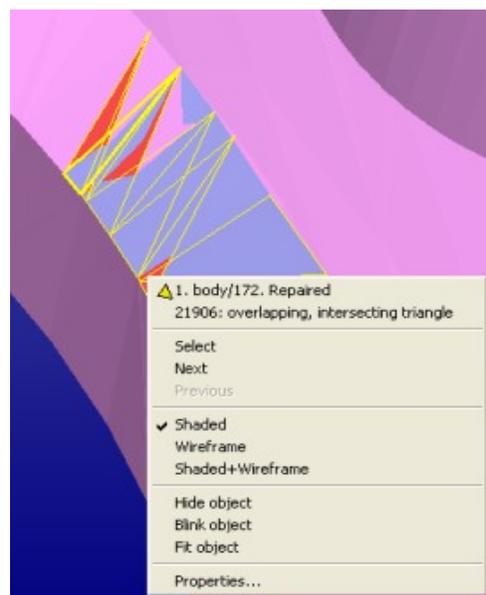
Then

> – **select the triangle index menu item *21906: overlapping, intersecting triangle* (or similar)** to get more detailed explanation on the error triangle:
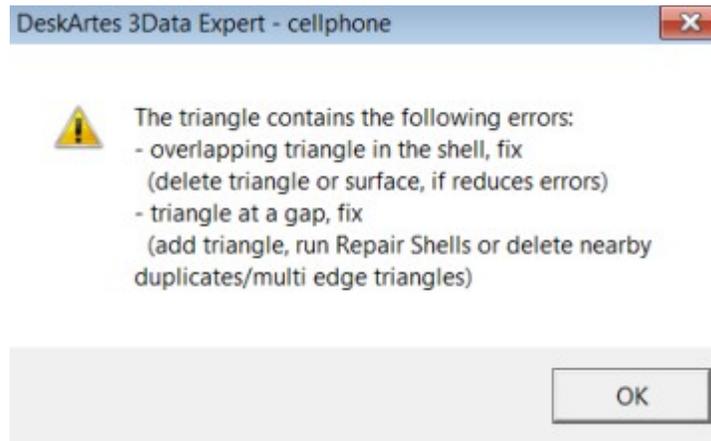
The dialog appearing will give more detailed information on the errors in the clicked triangle. We will follow the first help to remove triangles or surfaces to reduce errors.

Let's now start deleting bad surfaces and triangles in the error area. When we delete small overlapping surfaces, the number of errors diminish and soon we will only have gaps which can to be fixed automatically.

Make sure that

> – the **Selection Mode is set to *Surfaces*** now:

You may turn on the shaded+wireframe view mode by pressing the icon with the l*eft mouse button (LMB) and also make sure the selected object bounding bo is displayed by clicking* *icon*.

Then

> – **click with the LMB (left mouse button)** on the middle of the error area

to get an error surface selected:
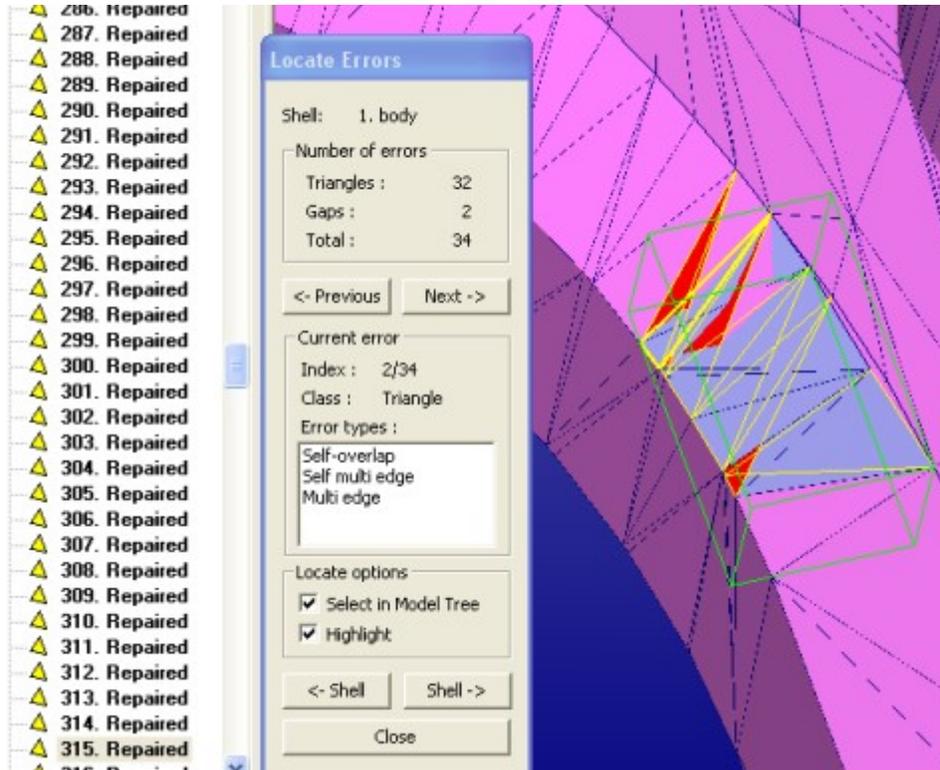
Then

– **give the command Edit > Delete** to remove the bad surface or press
  the shortcut

button. Then reselect a new surface at the bad area. When removing surfaces
always try to remove the smallest error surfaces at a given error area first. Also,
you should try to remove surfaces surrounded with red gap curves first. You may
need to click several times on the same mouse location to figure out how many
small surfaces are laying in the same area. You will notice that the error status is
updated after each deletion:
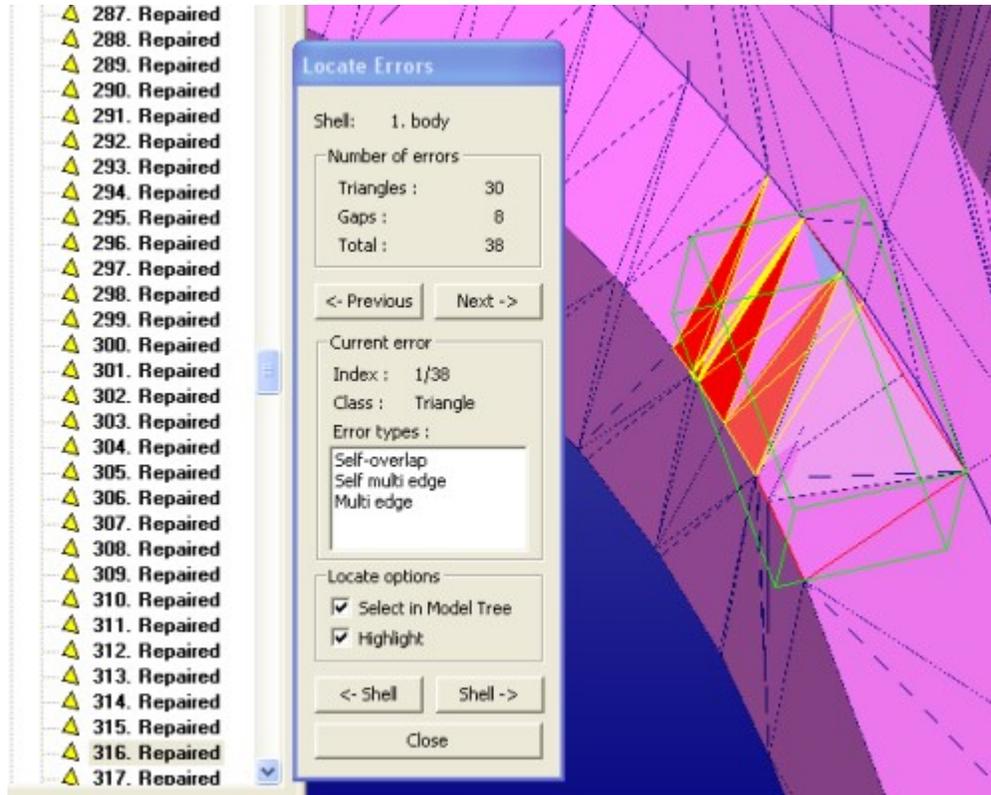
Now

- – **repeat selection and delete combination** until all overlapping triangles are removed:
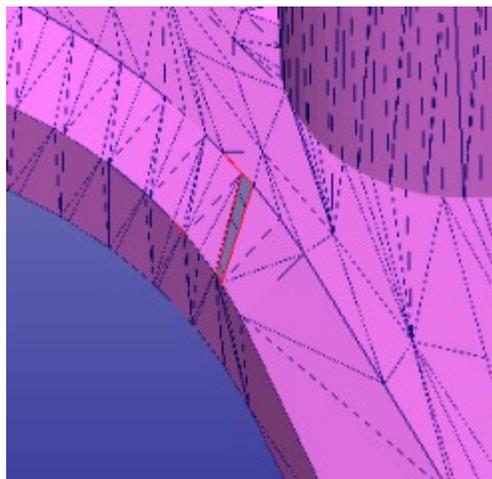
A gap or gaps are left in the error area. These gaps will be fixed during the next run to the *Repair Shells* command done later. Now we will go through the rest remaining error areas to perform similar bad overlapping or intersecting surface removal.

You can easily find the next error area by pressing the Previous or Next buttons in the Locate errors dialog. Or you can select the next area visually. For example you can select the following surface in the middle section of the part:
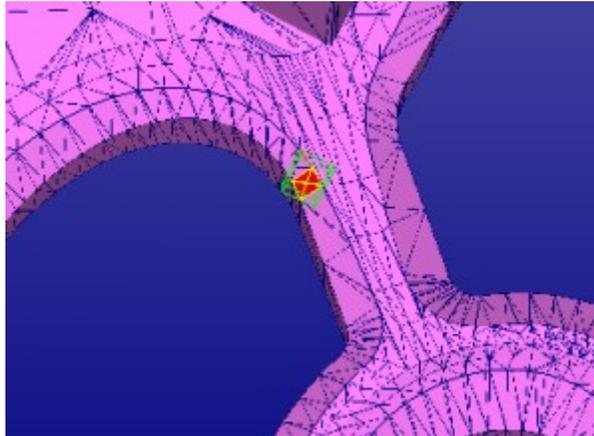

You will need to remove two small error surfaces in this are to get the result suitable for subsequent automatic repair shells command, i.e. model with small gaps:
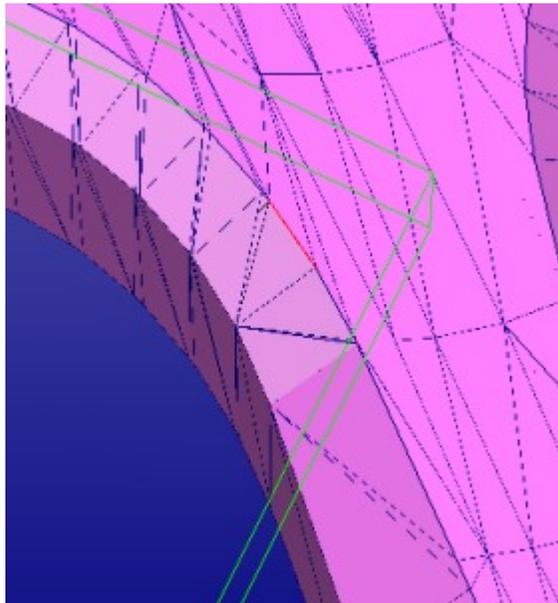
Now go to the next error area:
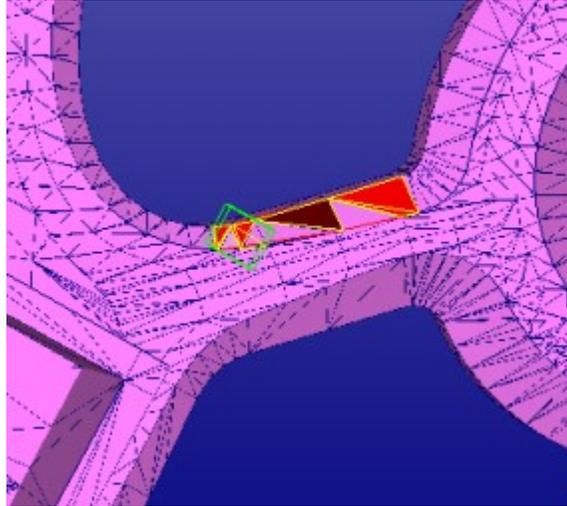
Select and remove surfaces until you have removed all bad surfaces in the area. You may also generate a larger gap at the area but do not worry, this will be filled with subsequent *Repair Shells* command:
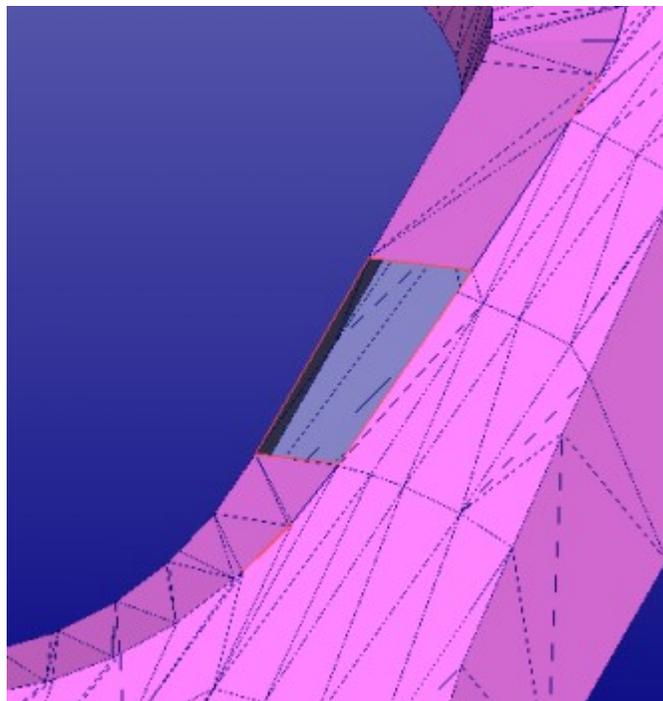
There is still one area with red triangles around. The body – cylinder intersection has created red triangles on top of the body part. If you use the right mouse

button on top of the red triangles you will see the error status. This indicates that the error should be removed by using boolean operation between the *body* and the *cylinder*:
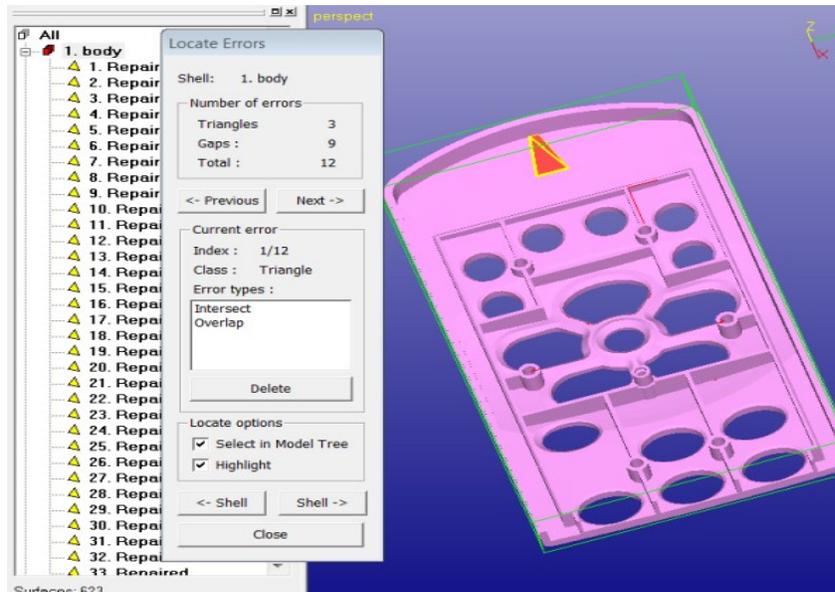

You can **close** the Locate Errors dialog now.

The Boolean operations will be done after each shell is fixed into a correct STL shell. Now we will

- **select the *body* shell and**

- **run *Fix Model > Repair Shells* command on to** fill in the gaps generated during the surface removal.

You can use automatically proposed parameters:
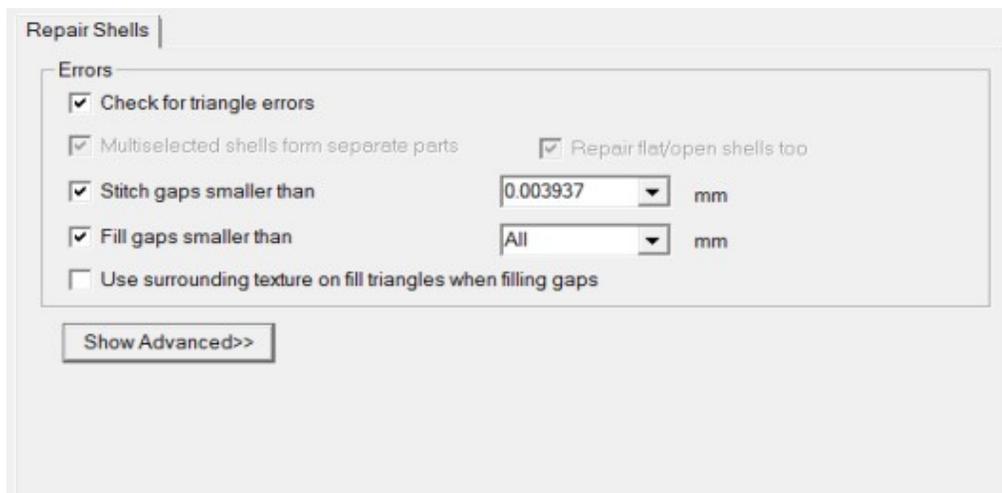
- **Press OK to start** the command.

After the command is run we have a *body* shell without any errors:
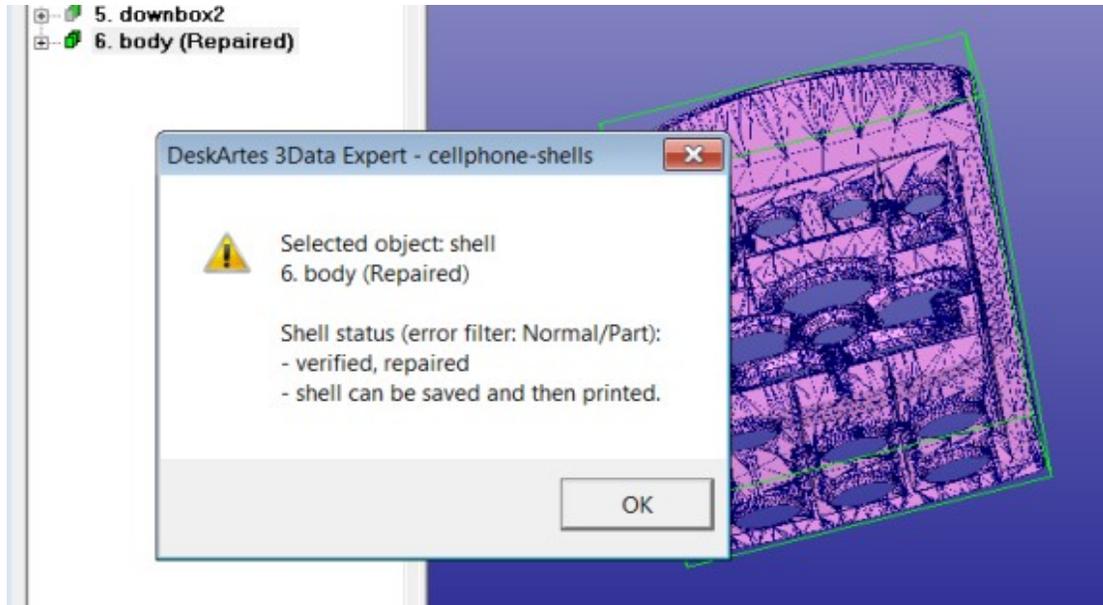
If you had problems producing the correct body, you can continue with the file *tutorials/GeomFiles/cellphone-body-repaired.3de* stored in the distribution package.

Note: If you are still left with some errors and seem not to get forward, you may try the *Fix Model > Edit Triangles* command to remove individual triangles from areas with errors. Just start the command, click on triangles with LMB and finally RMB to remove them to make clean gaps for the automatic gap filling with Repair Shells command.

Repairing the small cellphone components

Now we will have a look at the remaining cellphone components, *cylinder*, *upbox1, upbox2, downbox1* and *downbox2*. Before we can use Boolean operations to combine them into the body we need to make sure that the components are correct. We will **use multiselection** on shells 1 to 5**:**
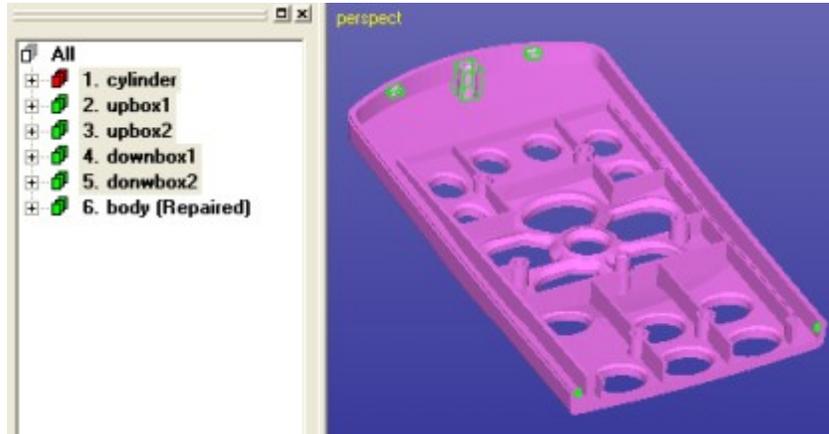
and run *Fix Model > Repair Shells* on all of them with the following parameters. Especially, the *Multiselected shells form separate parts* should be set *On* to keep the names for the results correct (and to avoid trying to stitch them together):
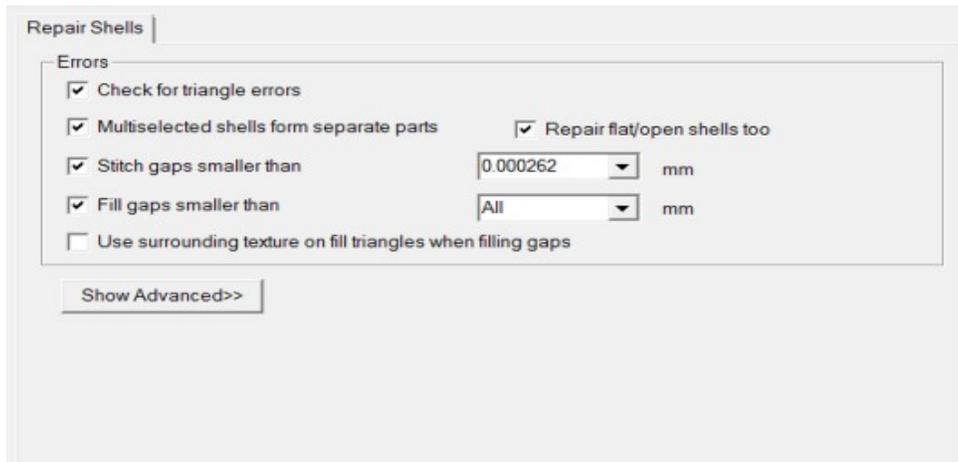

The result is 5 correct components:
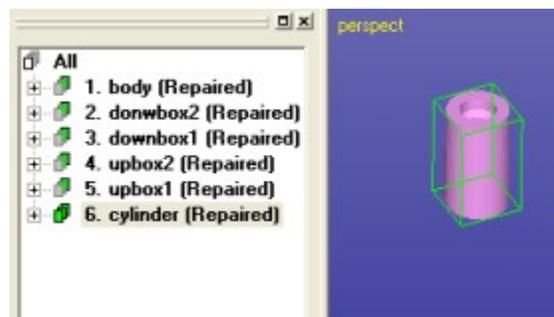
Combining the cellphone components

The *Modify Faceted > Boolean* command is used to combine the 6 components into one correct STL file. You will need 3Data Expert, 3Data Expert Lite or Sim Expert to finish the cellphone example as Dimensions Expert does not include Boolean commands.

First we will combine the *cylinder* with the *body*. As we can see in the image below (using clipping with ![icon] icon) the two components are slightly intersecting:
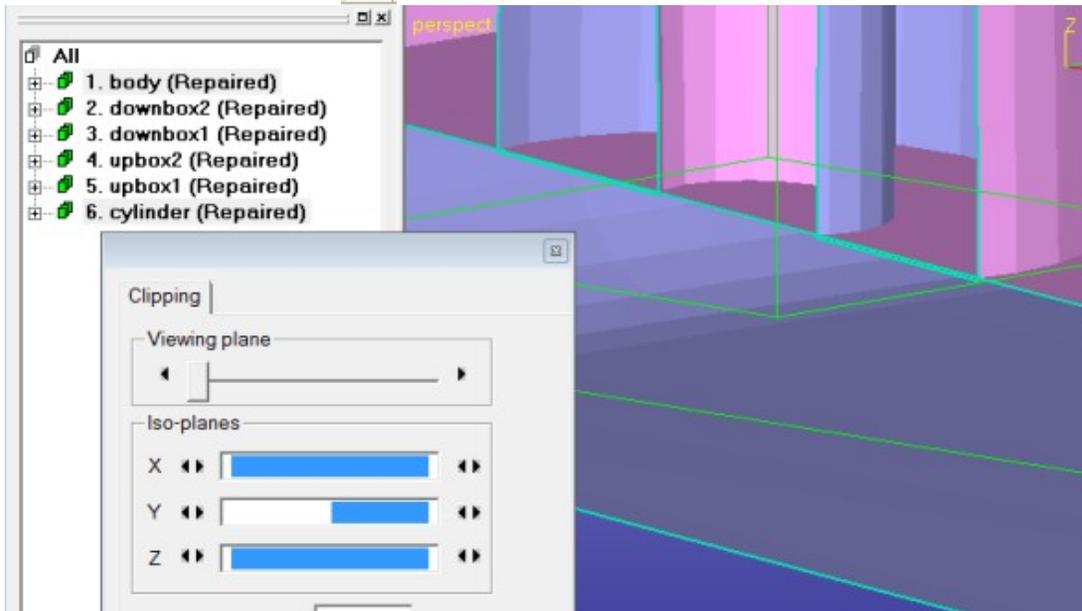
We can connect these two shells into one just by using *Modify Faceted > Multiple Join* or with *Modify Faceted > Boolean* command. We will use Multiple Join command.

- **Select the *shell 1. body* and *shell 6. cylinder* as shown in the image above.**

- **Give command Modify Faceted > Multiple Join**

Soon you are asked to *Remove original parts*:
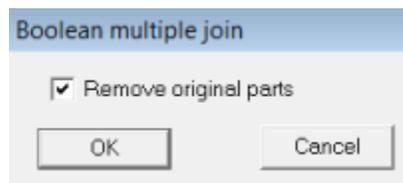
- **Press Ok** to *Remove original parts* and to commence the Boolean operation.

Soon the result will appear, the two shells are connected and the result is stored in a new shell, *body (Multiple Join)*:
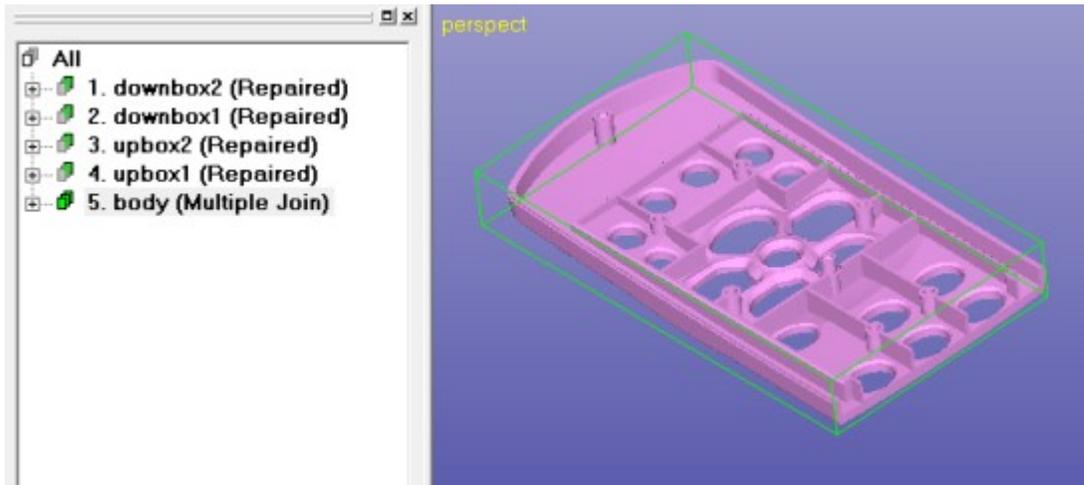
Now we will connect the *upbox1* and *upbox2* shells to the *body*. The fixtures are too far from the *body*, so we either need to move them towards the body or we need to generate extra material on the surfaces facing the body. Now we will generate more material on the surfaces. Extra material is added with *Create Geometry > Extrude Surface > from Surface* command.

Note: With 3Data Expert Lite or Sim Expert you can use the Transformation commands to move the fixture shells slightly (0.01 mm) towards the body shell.

With 3Data Expert we will add material on the *upbox1* shell, selected below in the Model Tree:
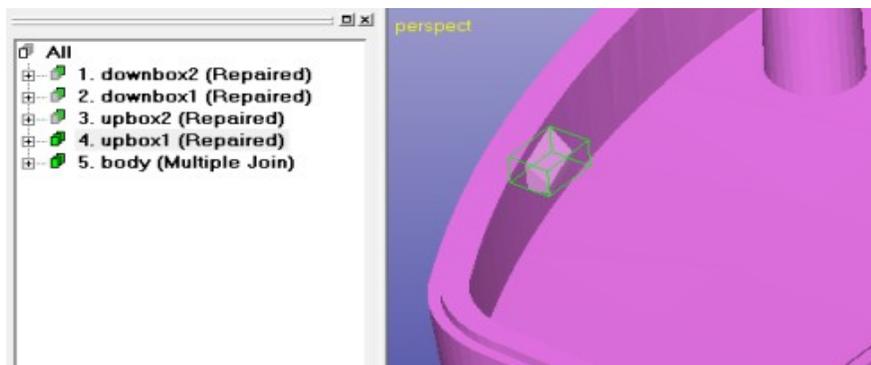

To be able to create a boolean result, we will add material to the surface facing the *body* shell.

- – **Fit** the shell with LBM + [🔍] icon
- – *m*ake sure you are in the **Surface** selection mode:

- – Then **rotate eye point** to see the surface facing the *body* shell
- – and **select surface** graphically
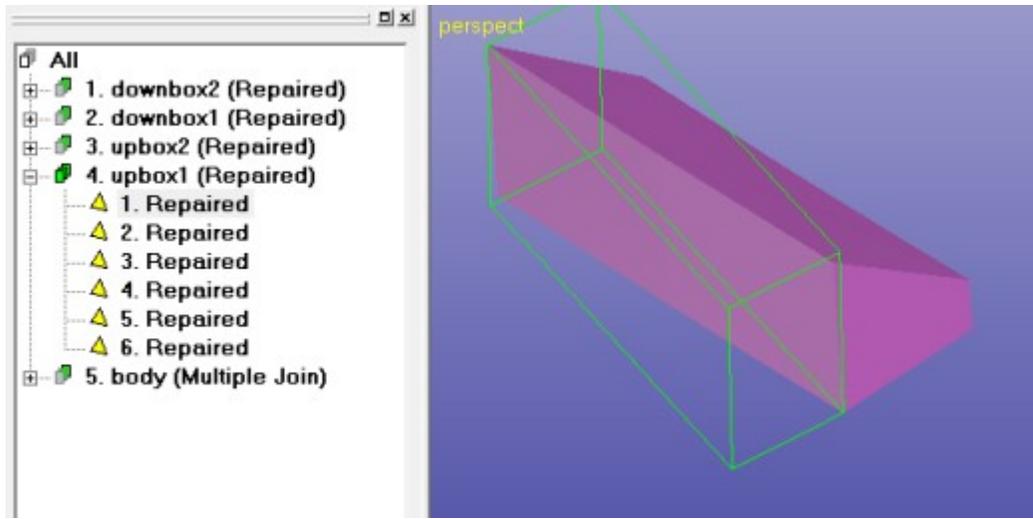
Then

- – **give the *Create Geometry > Extrude Surface > from Surfaces*** command

You will see the following parameter dialog:
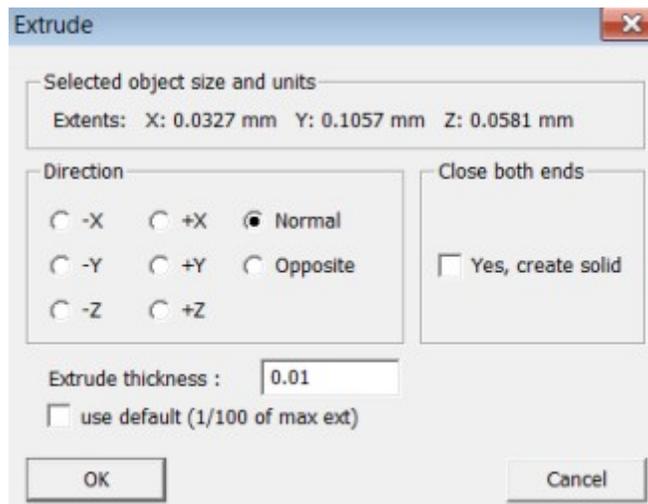
- **fill in the values above** (*thickness* 0.01 mm and *untick* the *use default)*

- **press OK**

After a while you will see the extrude result:
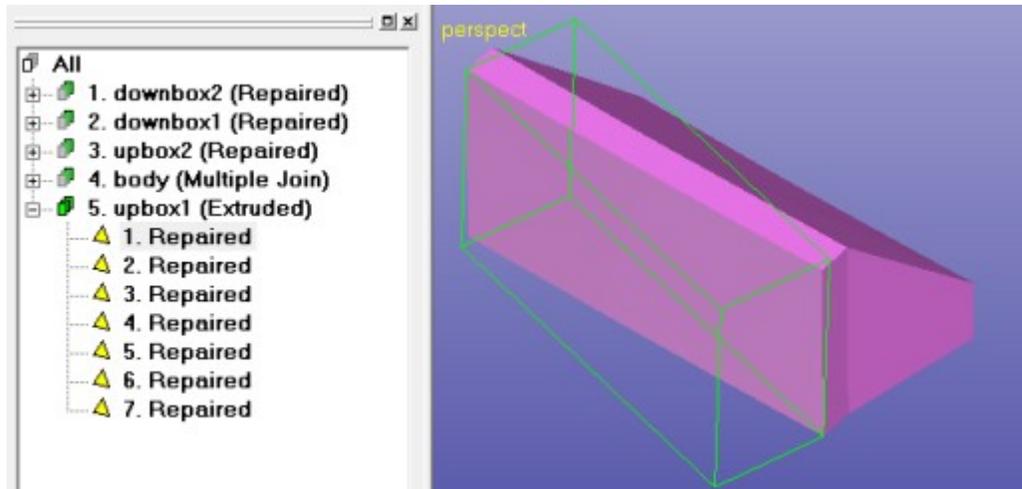
The command moves the selected surface to the (average) normal direction of the selected surface(s) and creates new edge triangles to keep the fixture solid. Now

- **use the multi-selection (*Ctrl + LMB*) to select** the *body* and the *upbxo1* shell
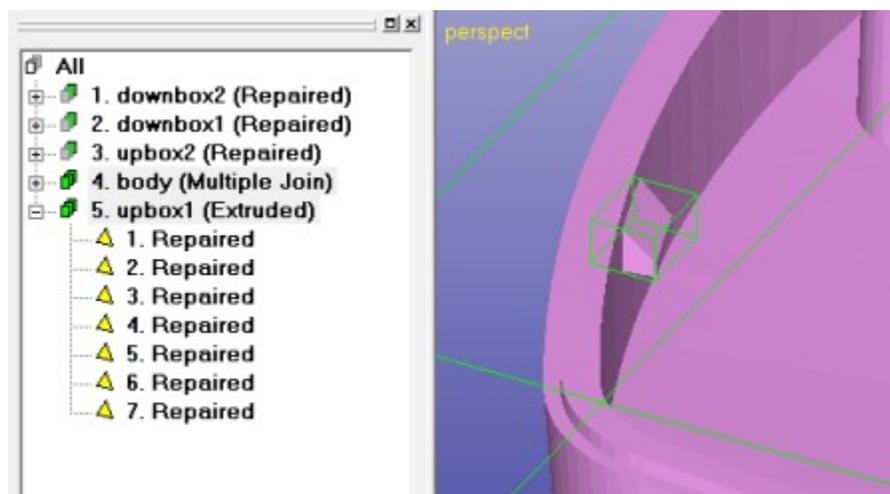
- **give the command** *Modify Faceted > Multiple Join* and **press** *OK (twice):*
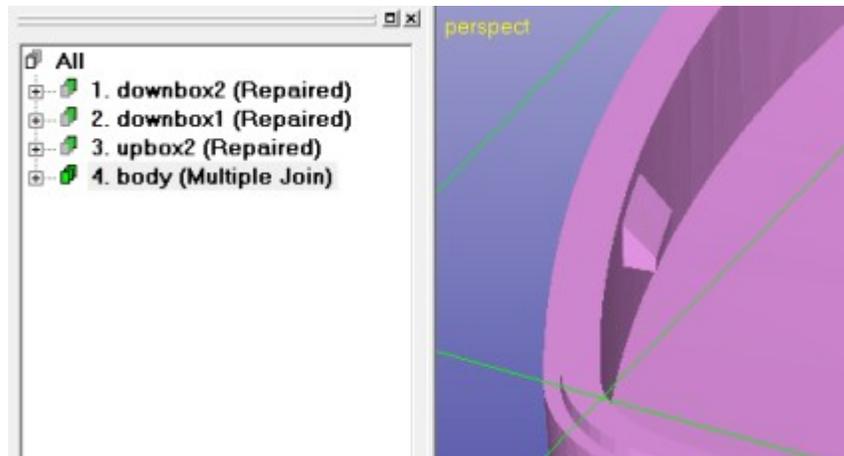
The result now has the two shells connected into one shell 4. body (Multiple Join). Connect the upbox2 shell to the body in similar fashion,

- **select the surface** facing the *body*

- **extrude material with *Extrude > from Surface*** with the same parameters used in the previous extrude

- **run Multiple Join** to connect the shell to the body

After connecting the *upbox2* shell you should have the following composition in the Model Tree:
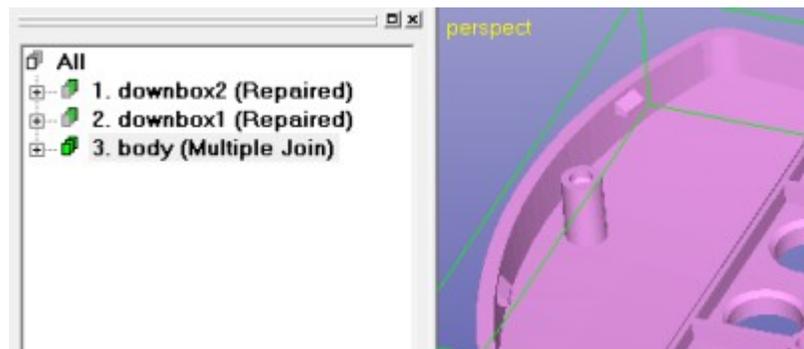
In the last Boolean step we will connect the *downbox1* and *downbox2* to the *body*.

- Use multi-selection to select all three shells
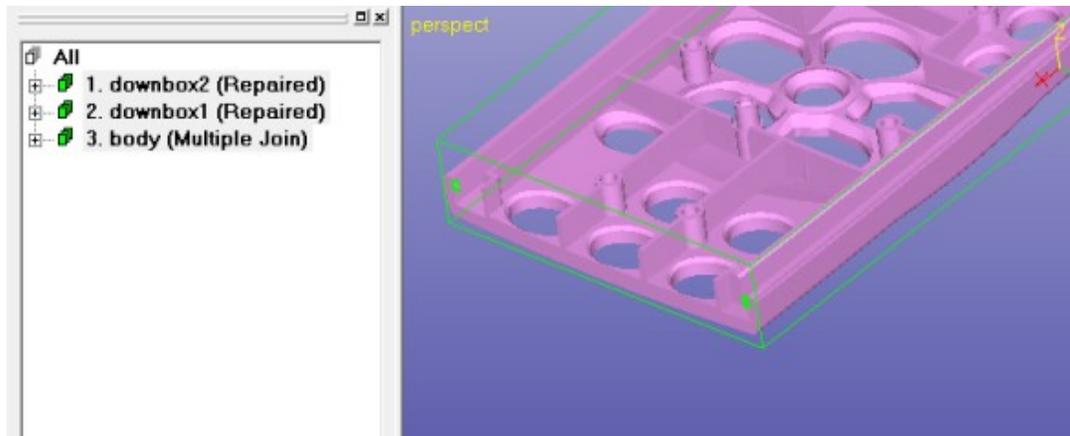
If you do some visual checking with Clipping tool, you will see that the components are tangential:
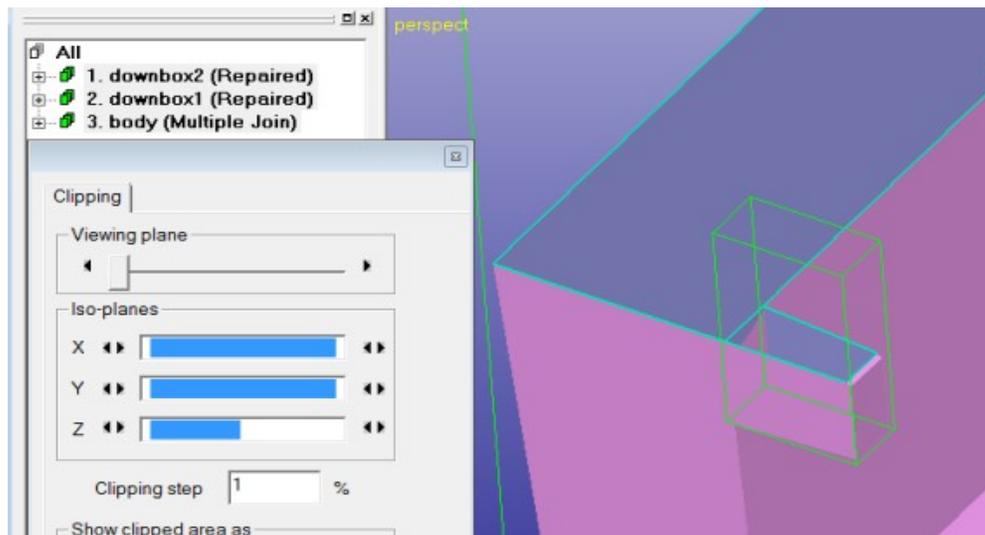
The new Boolean operations with Expert Series software can most of the time handle tangential surfaces, i.e. to connect them correctly. Just proceed with

- **command Modify Faceted > Multiple Join** for the three shells

After a while you will see the three shells connected into one *body* shell:
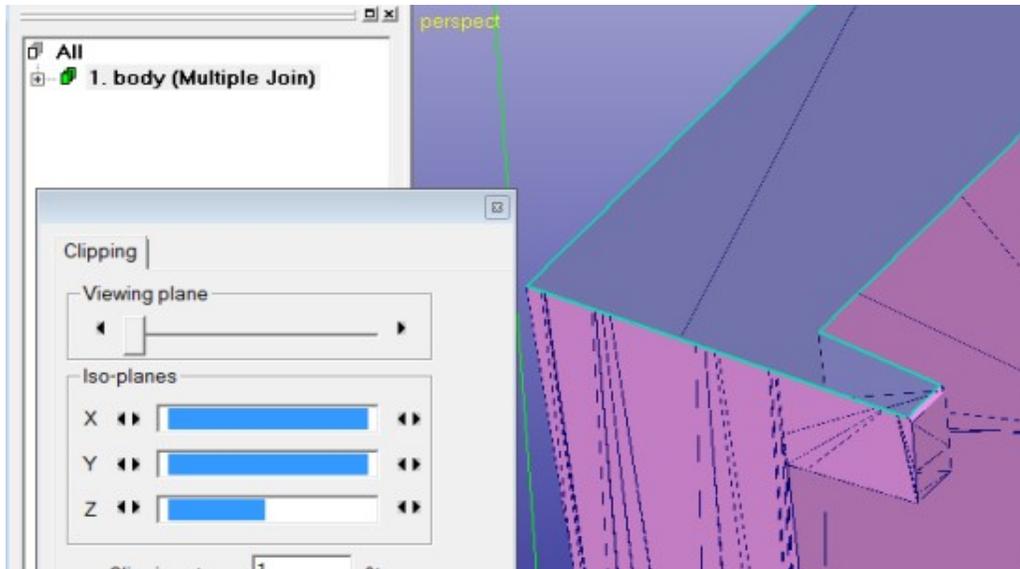
The correct model is stored in the cellphone-repaired.3de file in the distribution package.

Transferring to 3D Printing via Plugin

You amy want to use the **Print Preparation menu plugins** to transfer the cellphone model directly to another Additive Manufacturing application, like some 3D Printer control software.

To define and use plugins from Expert Series 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.

Conclusions with the cellphone model

The cellphone model can be repaired using the Expert Series software within 5 - 10 minutes by an experienced user. Stitching gaps, adding material and using Boolean allow us to fix the cellphone file in a few minutes. The output result is a fully fixed STL model for 3D Printing or Additive Manufacturing.

Chapter 3 – A human head

The second example will be a VRML model, a human head, file *tutorials / GeomFiles / gary_front_profile_bwrl.wrl* in the example package. The head model can be repaired using 3Data Expert, 3Data Expert Lite or Sim Expert modules with less than 5 minutes. The extensive use of Boolean operations make this example unsuitable for Dimensions Expert.

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

🟥 View　　　🟥 Dimensions　　　🟩 Lite　　　🟩 3Data　　　🟩 Sim

Anyhow, Dimensions Expert can be used until paragraph "*Repairing and combining the head and the eyes*". You may also want to try the self-intersection removal in paragraph "*Repairing and combining the head and the teeth*" on page 44 if running Dimensions Expert.

Start by

- **inputting the VRML** file *gary_front_profile_bwrl.wrl* with *File > Open* command

you can see the model on the graphics area. For each inputted file the software will ask for the units if not available in the input file.

- Select *Millimeter* units and **press OK**.
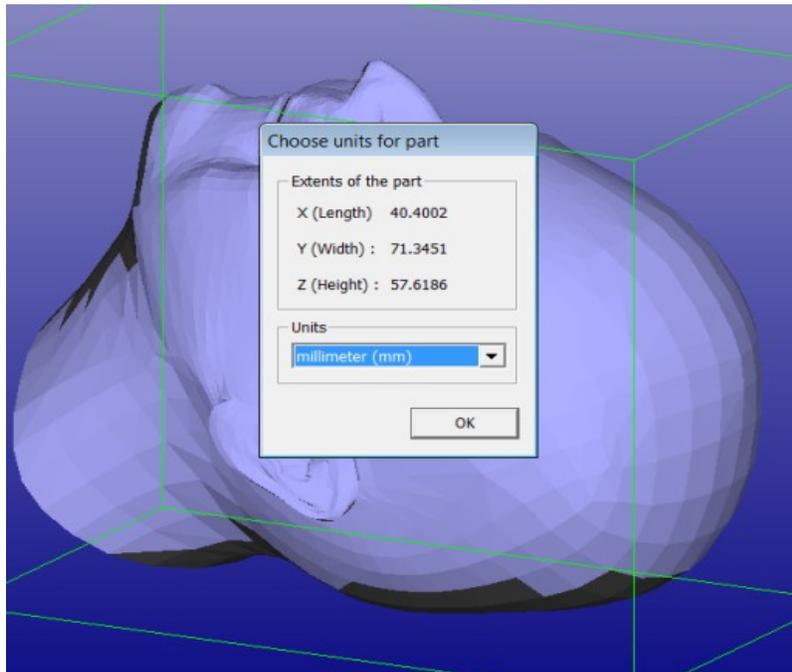
When the units are set the software will ask for the Operation Mode:
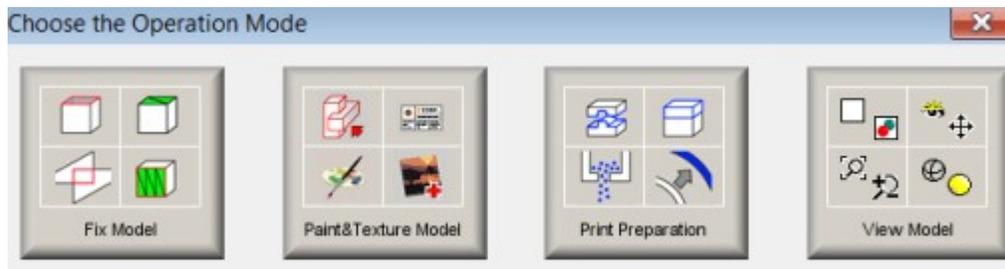
Analyzing the head model automatically

We will now

- **select the *Fix Model* mode**, this will automatically analyze the model for errors as well as orient the normals consistently.

After a while the analysis result is shown:
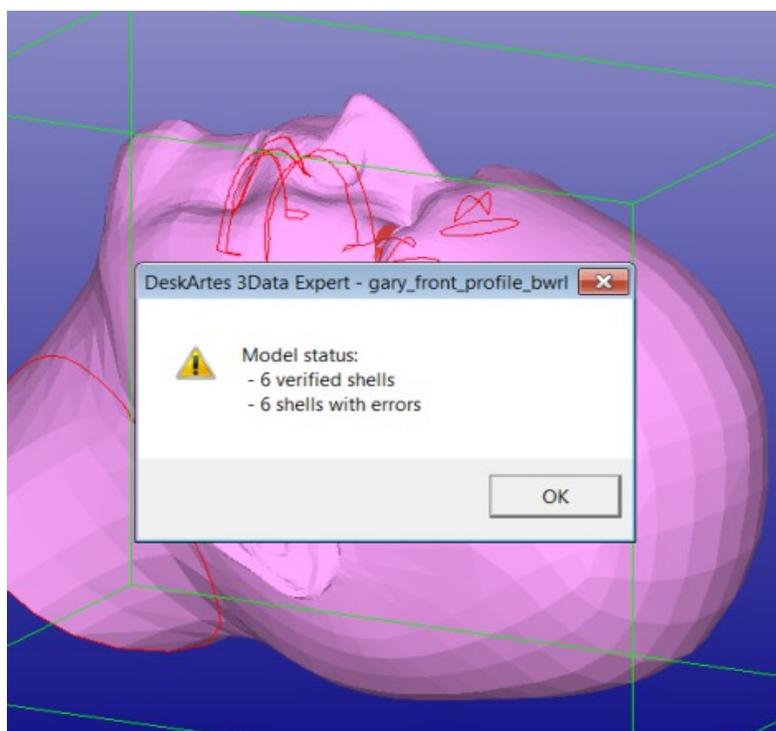
After the analysis the surface normals are corrected and the outer surface color is changed to pink, the inner surface color remains light blue.

The model has 6 shells each with errors.

- **Press OK** to close the dialog and

to see the Model Tree and the Tools Window with Fix Model Tab and repair Help text window. The software asks you if you wish to run the Auto Repair immediately:
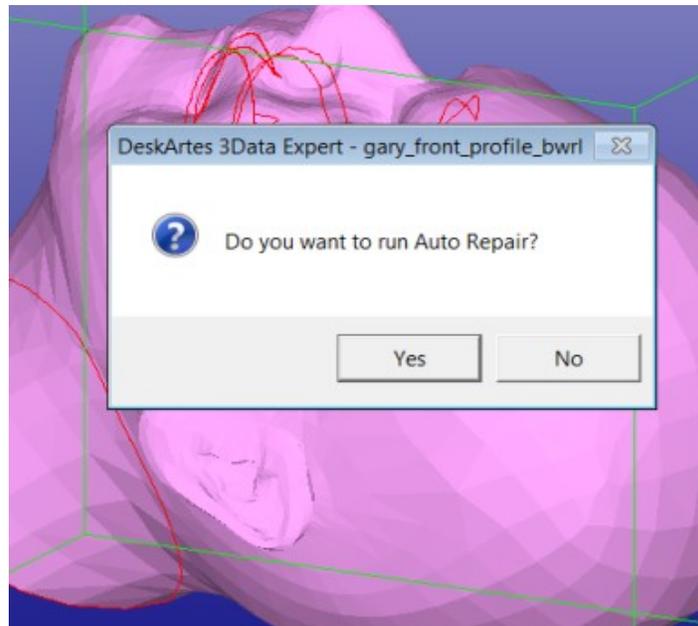
The visual inspection of the errors (gaps) in the model suggest that we should not run the Auto Repair. Auto Repair will repair each open shell (6) separately into a solid shell. The gaps in this model at the mouth / lip region show that closing them may generate separate shells where shells should be connected to each other at open edges (lips vs. mouth). Anyhow, you may do a faster repair just by running the Auto Repair now and skipping to the chapter *Repairing the teeth* on page 44.

- **Press No Auto Repair** to start a more careful examination of the errors.

You will see the status window appearing:
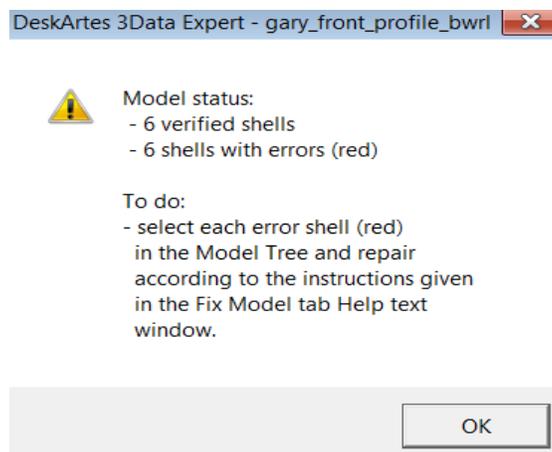
This indicates that each shell has errors and we should use Model Tree and Help Text to repair the model.

- **Press OK to close the window** and

to see the Command Tip window (if not turned off earlier):
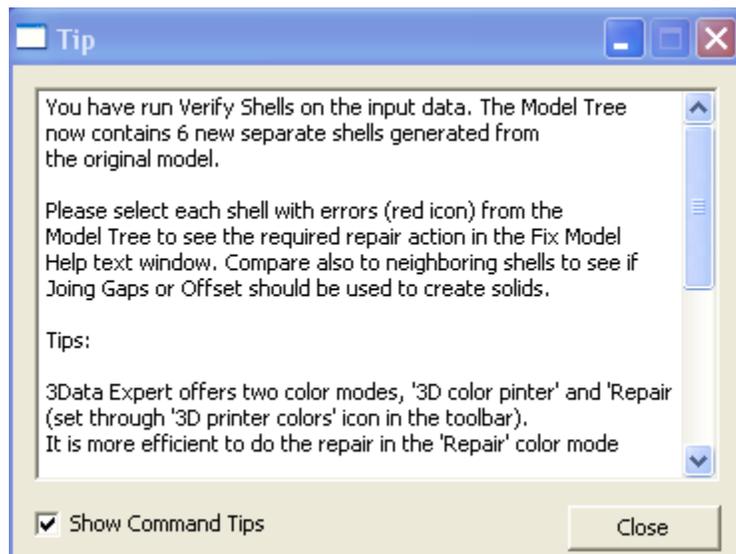
The Tip window gives useful information after different commands. For example, this tip window proposes to use the Fix Model Help Text window as well as visual checking to decide the next repair actions.

- **Press Close** when ready reading the tip.

Before going to more controlled use of the repair commands you may want to take a look at the real colors in the VRML file. For that you should turn on the *3D Color Printer* mode for a second. Press the ![icon] icon to see the original VRML texture map colors:
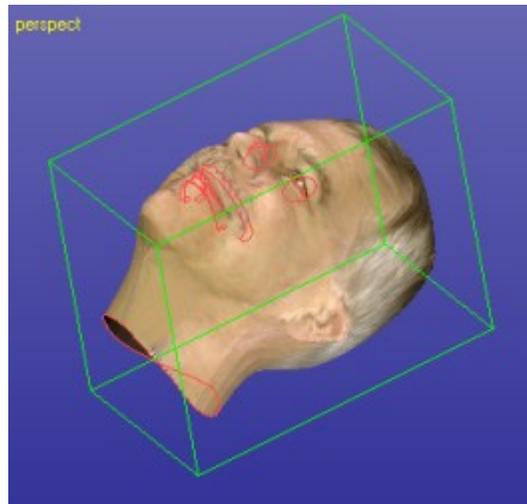
3Data Expert can repair and manipulate fully colored and textured models for 3D Color Printing. Now press the [icon] icon again to get back to the *Repair color* mode.
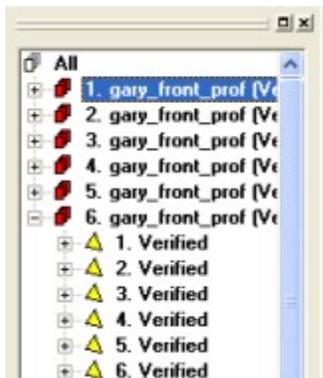
The Model Tree on the left hand side of the display area contains the directory to the geometrical information for the model. This model contains 6 *shells* (*e.g. 1. gary_front_prof (Verified), ...*). Each shell contains *Verified* triangle surfaces (press +-icon to see the 1. *Verified, 2. Verified. etc.* surfaces inside the first shell). The texture information for the VRML data (*.jpg* images) is stored after each surface.

If you select each shell separately you will see that they represent the eyes, teeth and mouth for the head model. After the initial verification the shells are ordered by surface area in descending order.

Now select different shells in the Model Tree and see the Help text window in the bottom of the Fix Help Tab on the right hand side of the screen. Select the *1. gary_front_prof (Verified)* shell with the left mouse button from the Model Tree:

The message in the Fix Model Tab Help text window tells that there are several *intersecting triangles* (about 153) and *gaps* (4) in the selected shell. Gaps are shown as red curves on the graphics area and stored in the *Gaps* element in the end of each shell in the Model Tree (for example, open the *1. gary_front_prof (Verified)* component and use the vertical scrollbar to scroll to the end of the component to see the *Gap* element). The intersecting triangles are show with red color in the display area. The are mostly generated by intersections between the eyes and mouth parts with the 1. shell.

Below the status information the set of possible commands to fix the model are presented. There may be one or more commands and the first is the most likely action to take. Anyhow, you should also use visual inspection to finally decide the command to run.
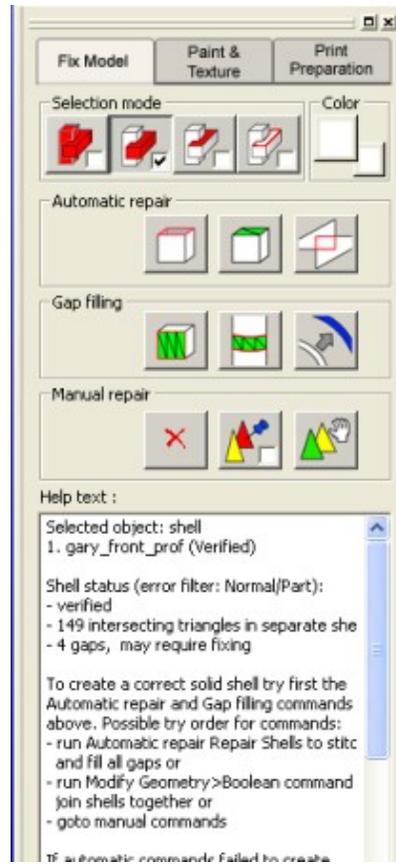
The input file is divided into 6 different shells in the Model Tree; the original head consist of these disconnected components (eyes, teeth, mouth …). In the sequel, all gaps will be closed and the components will be connected into one correctly oriented part.

Analyzing the head model visually

Some visual inspection very quickly shows us that the 1. shell (head) should be connected with the inner mouth shell (2. shell). In the image below the two shells are displayed (non displayed shells are grayed out in the Model Tree) and clipped to display the inner structure of the model (the Clipping Tab can be displayed by pressing the 🖌 icon):
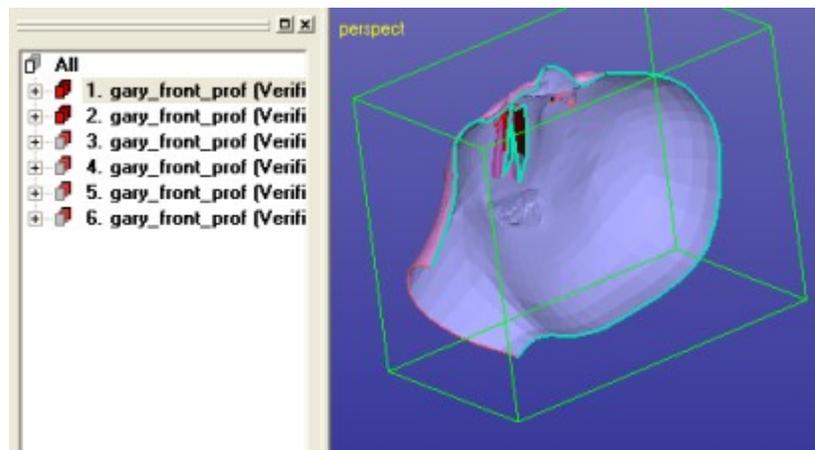
We can use different visual aids to define the order how to fix and combine the different components together (see the image above). Several viewing commands are available in the *View* menu as well as the Viewing Tab and Clipping Tab functions do determine the best way to go forward. For example, we can have a look at the model using clip form +X direction and a thick light blue clip curve, as seen above.

Investigating the verified model visually will show us that one possible way to go is to repair and combine the parts in the following order:

1) stitch head and mouth into one correct shell

2) repair eye shells

3) repair teeth shells

4) join all shells with Multiple Join command into one shell


Before starting the repair operations, lets

 – **rename the different shells** with more descriptive names

with right mouse popup menu on the Model Tree item + *Rename* command. This will help us to keep better track on different parts during the repair process. Below we can see the renamed components.
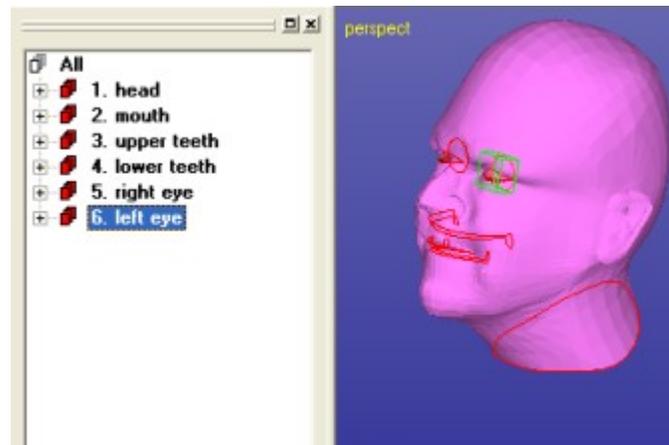
Repairing and combining the head and the mouth

With the head and the mouth we will use the automatic *Fix Model > Repair Shells* command to combine the mouth to the head at the lip area. Stitching the mouth edges into the lips (see below) will produce a desirable result.
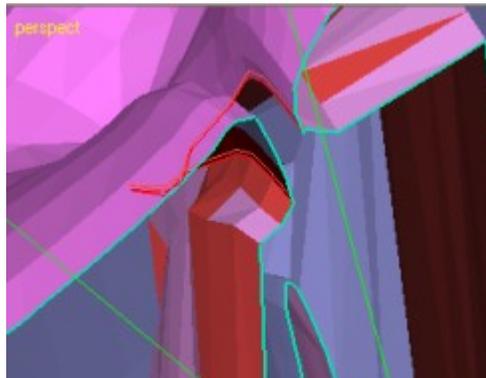
We can see that both the disjoint shells are more or less at the same distance from each other along the open edge. The approximate distance can be measured using the *Dimensions > Distance* command and it will give approximately 0.05 units (you can also use the [icon] icon to start the *Distance* command).

Before running the *Fix Model > Repair Shells* command we will use the multi selection (*Ctrl + left mouse button (LMB)*) to select both *head* and *mouth* components in the Model Tree.

Now

- – **select the two shells** with multi-selection, first the *mouth*, then the *head* (the last selected shell will give the name to the result)
- – **run the *Fix Model > Repair Shells*** command from the menu or use the command Fix Model Tab shortcut

button. You will see the following parameter window:
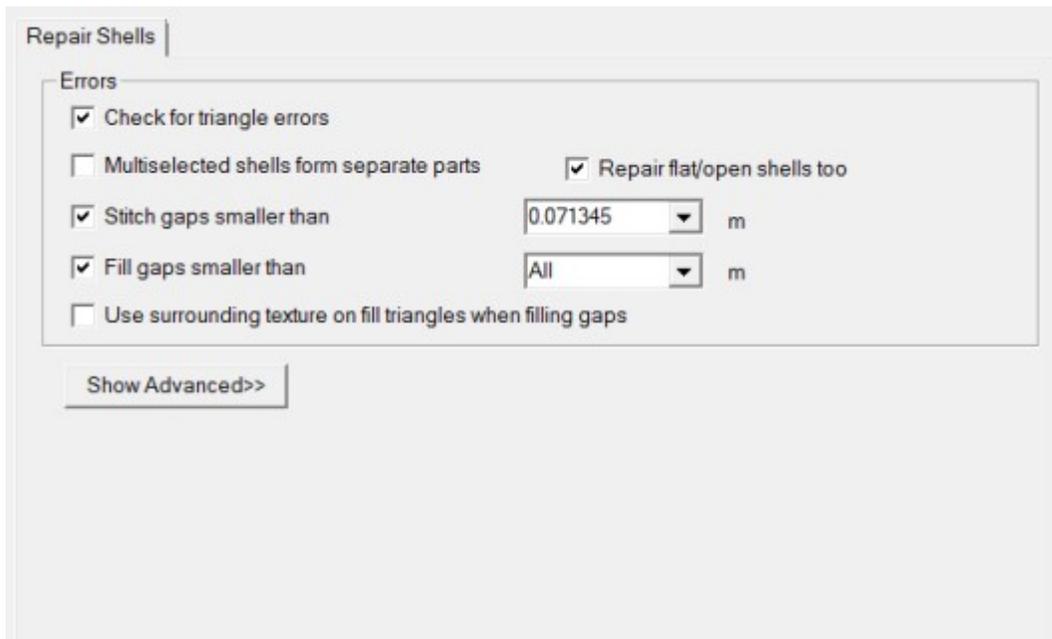
To get good results, we must give correct parameter values for the repair process. As seen above, normally we want *Check for triangle errors* as we are changing the geometry and we want to know the possible errors after the repair.

- Turn **Off** the **Multiselected shells form separate parts** setting.

Stitch gaps smaller than 0.071345 is automatically calculated according to the size of the selected shells. According to our measurements earlier this will connect the shells together by stitching the edges together, i.e. all segments which are closer than *0.071* units from each other will be combined. *Fill gaps smaller than* and value *All* will fill the remaining large gaps (neck and eye slots) to form a solid model.

Note: you must be careful when filling the *Stitch gaps smaller* than value. If a too large value is given some edges may be connected to edges too far away, causing new errors.

The *Advanced* parameters are not needed with this example.

With the head model we will accept the default parameters seen in the image above and

- **press OK** to start the **Repair Shells** command

After a while we will see the correct result and the message
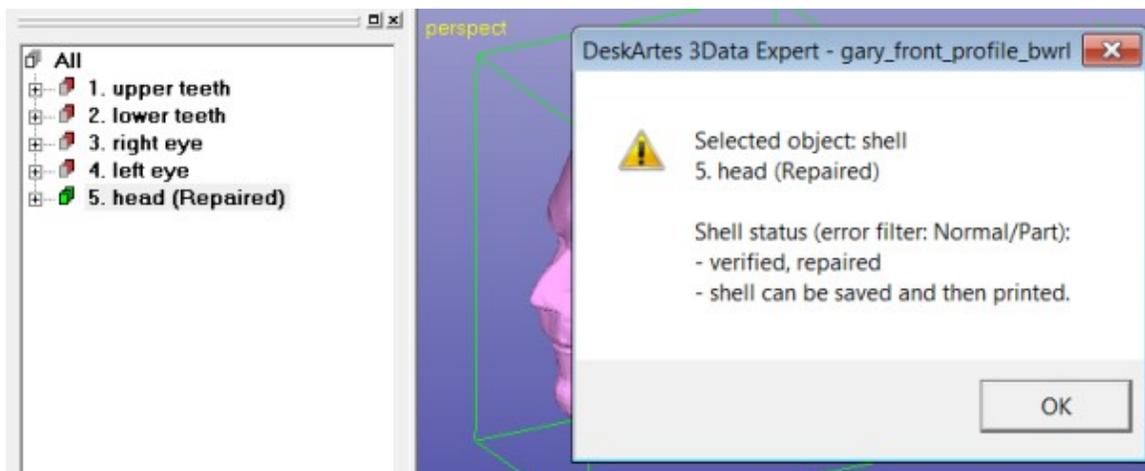
The original parts will be deleted from the Model Tree and the result will be stored in the end of the Model Tree, *5. mouth (Repaired)* (should be renamed to *head* immediately).

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

Repairing the eyes

In the next step we will repair the *eye ball* shells to prepare for combining them with the head shell later in the process.

First we will

- **select both eye shells**, *left eye* and *right eye*,

for the second *Fix Model > Repair Shells* command. The selected sells are fitted on the graphics area in the image below:
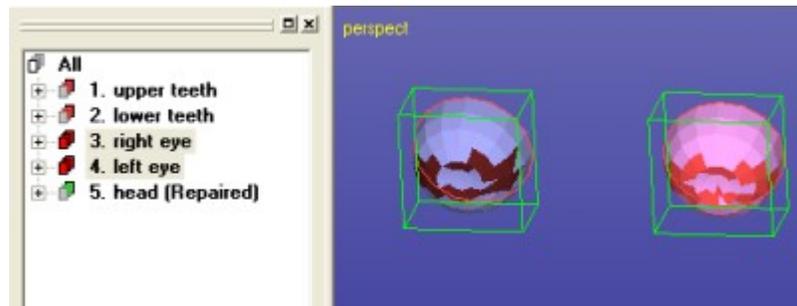
– **Give the command *Fix Model > Repair Shells***

The parameter dialog is displayed.

We will use the default parameters except that we will

- **set** the ***Multiselected shells form separate parts*** to ***On***

*a*s we do not intend to join the two eye balls with each other (they would be too far from each other in any case)*:*
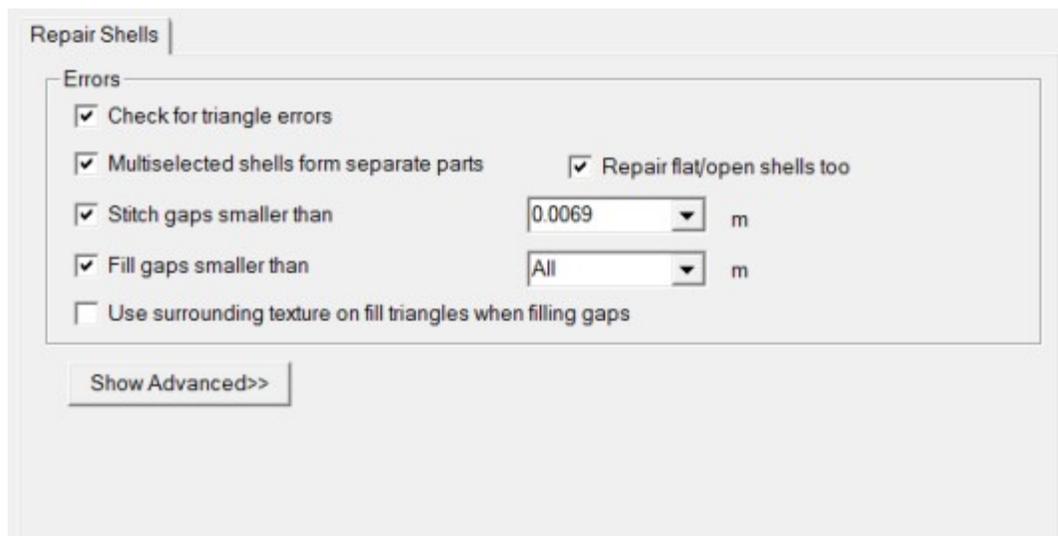
– **Press OK** to run Repair Shells command
– **Press OK to close the following status** dialog

And you will see two correct eye balls in the end of the Model Tree:
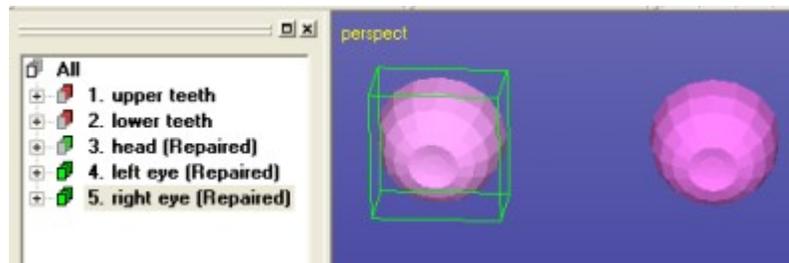
Repairing the teeth

In the next repair the teeth shells to be connected to the head shell later. First we will run *Repair Shells* command individually on both teeth set, *upper teeth* and *lower teeth*, to get a solid STL data for the following Boolean operations.

- – **Select the *upper teeth* shell** and
- – **run the *Fix Model > Repair Shells*** on the selection*.*

Use the automatic parameters. **Note**: *Repair Shells* is the second preferred command in the Help Text window in the Fix Models tab (first proposal *Join Gaps* can not be used as there are no similar gaps close by)**.**
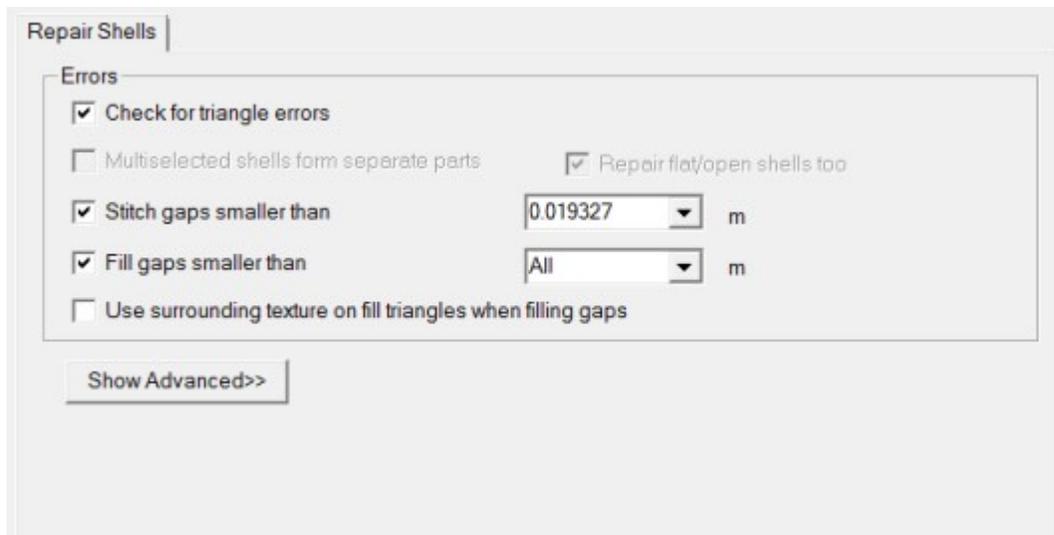

- – **Press OK** to start the Repair Shells command

After a while you will see the result
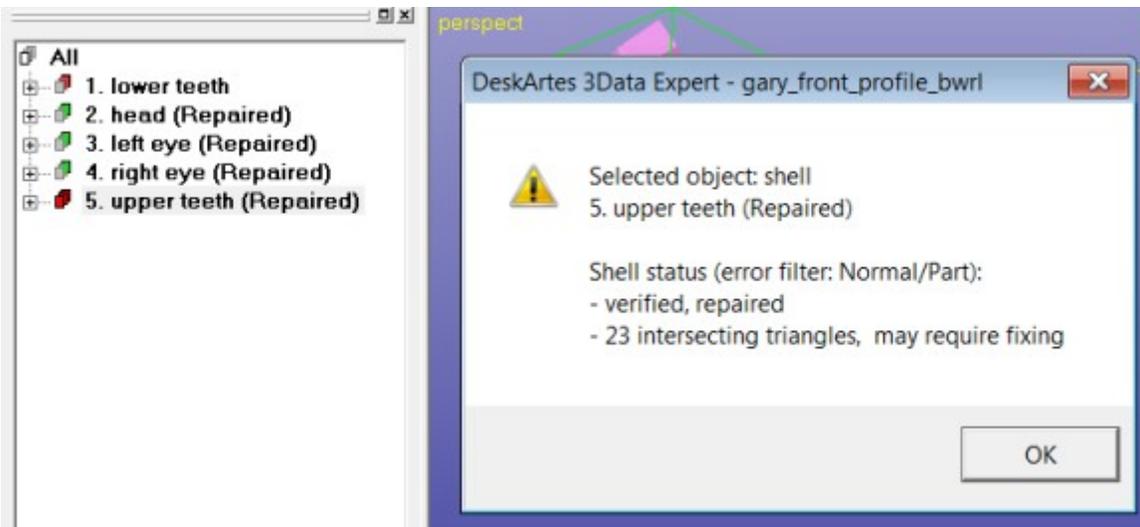
The software reports 23 *Intersecting triangles* error. A closer investigation (red triangles in the image below) shows that the intersecting triangles are are piercing through the outer surface.
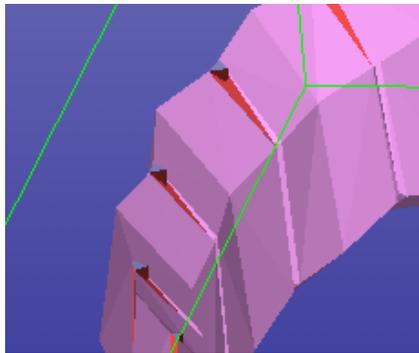

In the Fix Model Tab Help text window the most recommended repair action is to run *Remove Self-intersections* command.

- **Run the *Fix Model > Remove Self Intersections*** command on the *upper teeth* shell now.

The command can also be started from Fix Model Tab shortcut

button. After a while the self-intersection result is displayed with Tip window, if not turned off earlier:
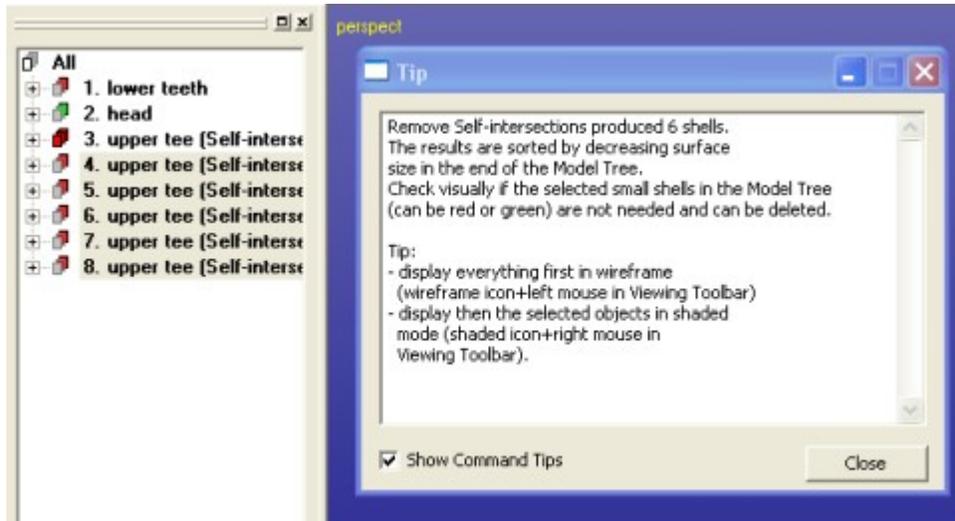
The Tip window indicates that the shells preselected after the command is run can be deleted. These are the small pieces generated by the self-intersection removal and can normally be deleted from the result; the largest shell is the one before the preselected shells and is normally the correct result. Anyhow, you should always visually check that the correct extra shells are removed. With this example the visual check also supports the proposed removal of the shells 6 to 10, as seen on the next image:
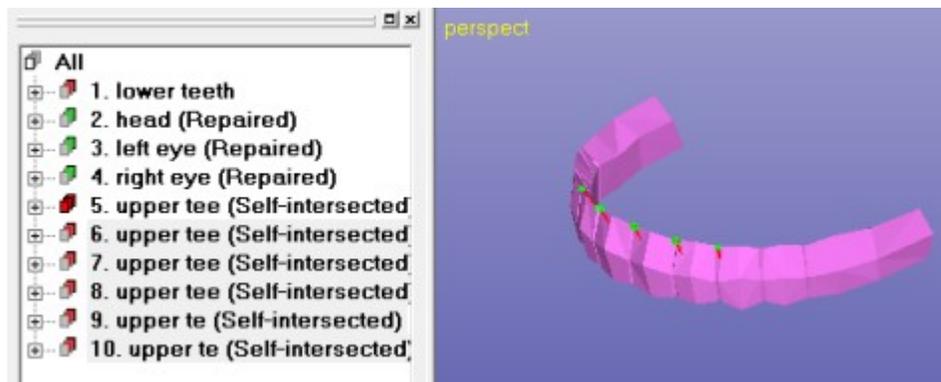
– **Delete** the extra shells with *Edit > Delete* **command**

or by pressing the Fix Model Tab delete shortcut

button.

The software now recommends you to run the Repair Shells to fill the gaps generated during the Remove Self-intersections command.

‒ **Run *Fix Model > Repair Shells* command** with the automatic parameters now to generate a solid *upper teeth* shell:
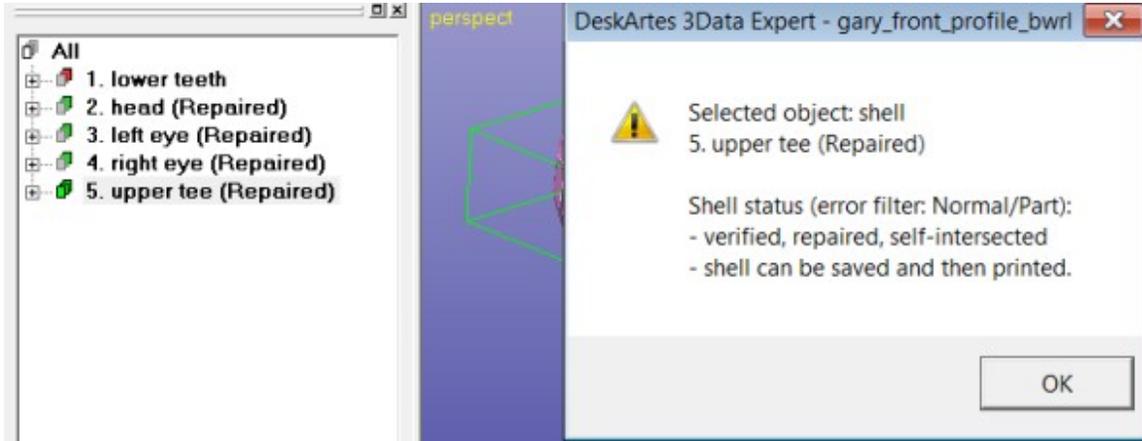

The *lower teeth* is handled in the similar manner. First

‒ **run the *Fix Model > Repair Shells* command to close all gaps**

on the *lower teeth* component to make it solid. Once again, there are self intersecting triangles which are removed by

‒ **running the *Fix Model > Remove Self-intersecting* command**.

After the self intersection removal the extra small shells 4 to 12 are again deleted:
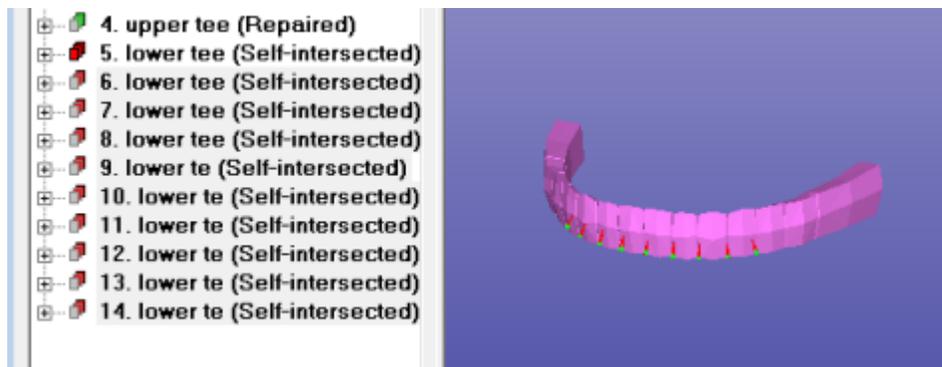

‒ **Delete the preselected small shells** now**.**

‒ **Run *Fix Model > Repair Shells*** on the shell 5. to create a solid shell.

‒ **Press OK** to close the status window

‒ **give View > Fit for the root node All**

Now we will have a model with 5 correct shells to be connected into one final shell.
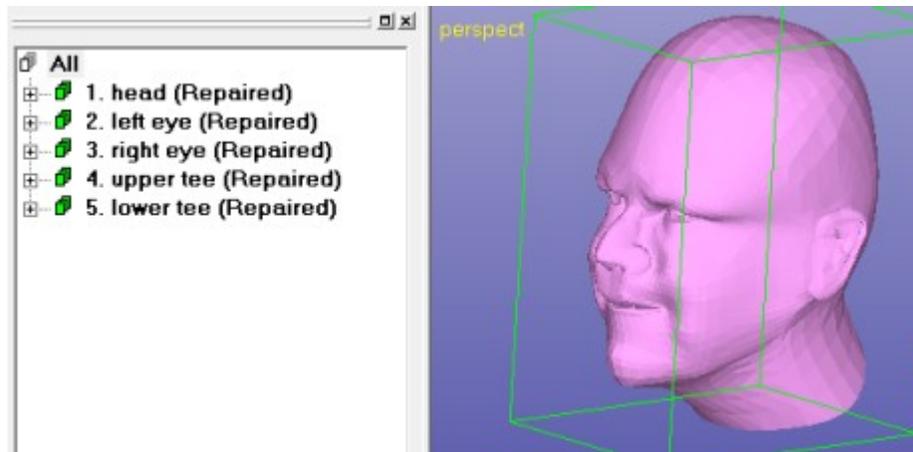
Now we are ready to combine the three remaining components, *head, upper teeth* and the *lower teeth* into one solid STL part. First we will use multiple selection to select all parts and then run the *Modify Faceted > Multiple Join* command on them.

  – **Multi-select all shells** in the Model Tree and

  – **Run *Modify Faceted > Multiple Join*** on the selected shells:
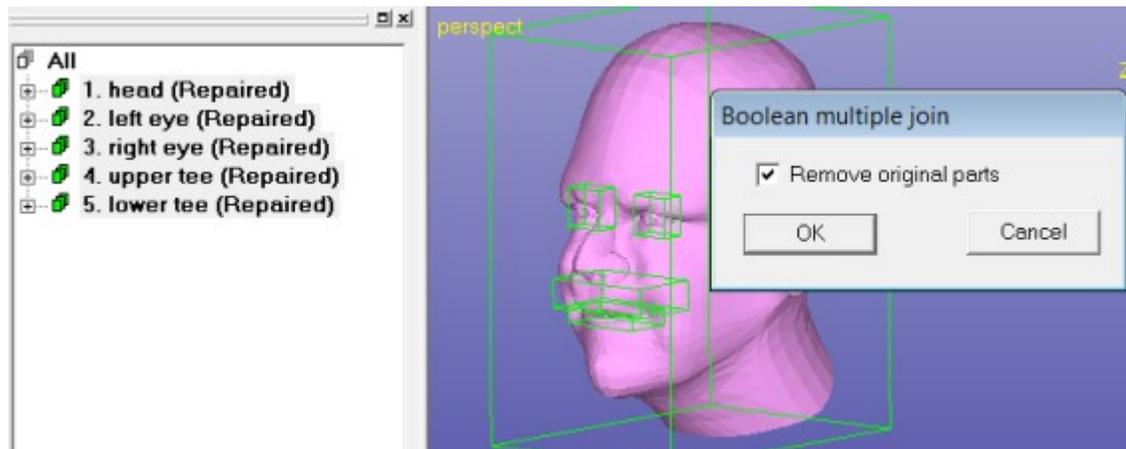
  – **Press OK to *Remove original parts***

A progress bar is shown for each shell to connect (1/5, 2/5, … 5/5) and soon you will see the correct head (if necessary, rename the mouth shell to head shell):
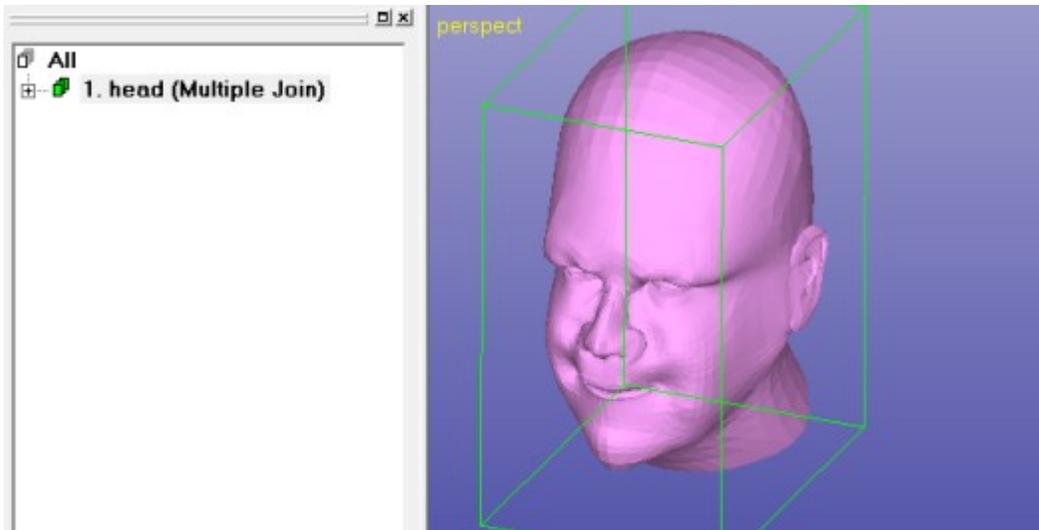
The Fix Model Tab Help texts shows that the model is correct too:
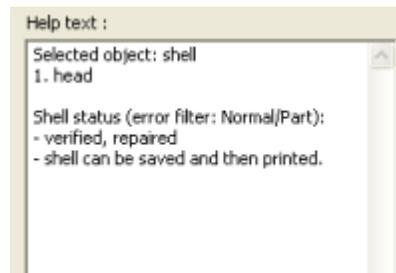
Outputting head STL and VRML files

The STL output of the file is straightforward, just select the model and **give *File > Save As*** command with the STL option. Also, outputting to 3D Color Printing can be done easily by selecting either VRML or ZPR file formats.

The correct file *tutorials / GeomFiles / head-repaired.3de* can be found in the installation package.

Chapter 4 – The Maya model

The second model to fix will be a test part generated with Maya software (*maya_three_d.wrl* in the example package). The maya model fixing should not take more than 20 minutes with 3Data Expert or other Expert Series module.

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

■ View   ■ Dimensions   ■ Lite   ■ 3Data   ■ Sim

Start and

– **Open the file *maya_three_d.wrl*** with File Open dialog.

After the input we will immediately see that the model is not oriented correctly, there is a lot of blue inside (inverted normal) color visible amongst the material colors (normal side) inputted from the VRML file.
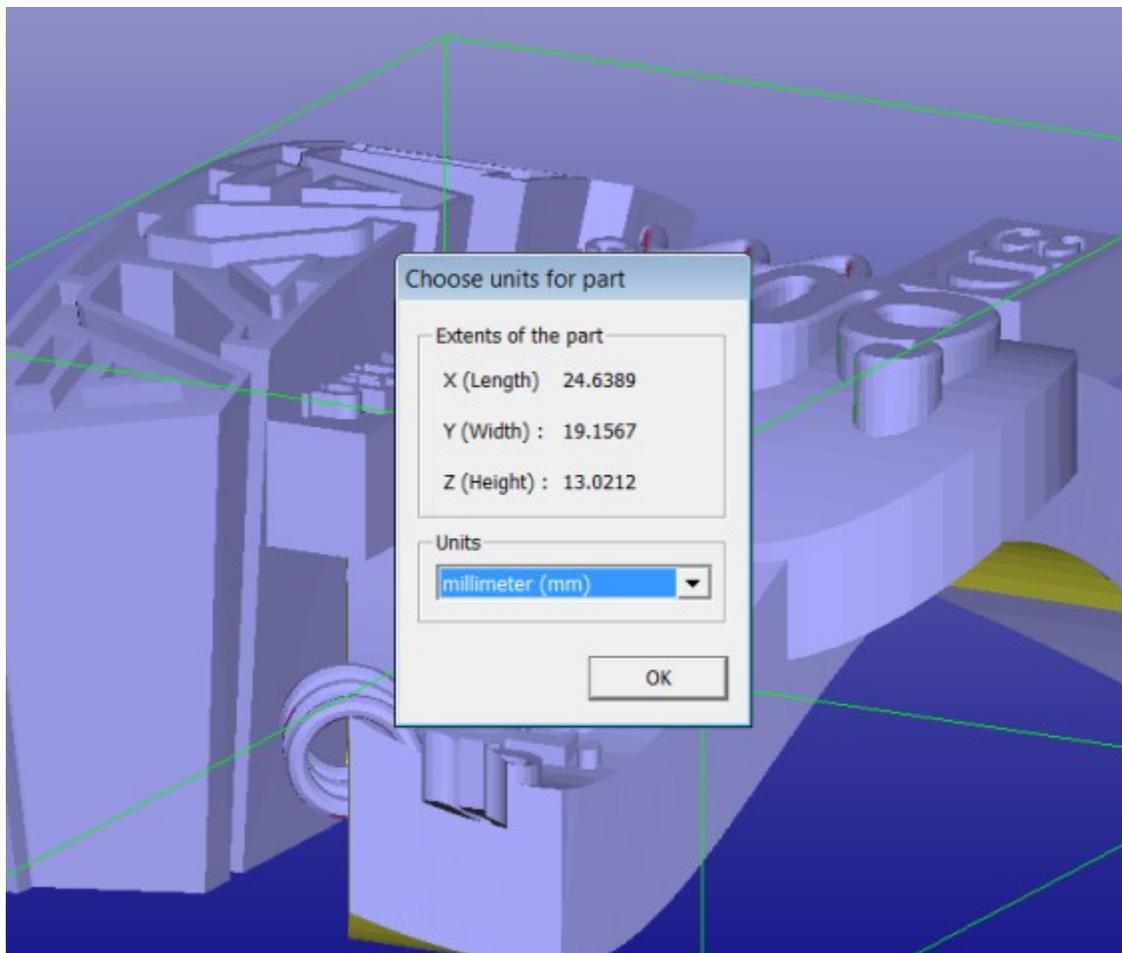
- **Set the *Units* to *Millimeters* and press *OK*** to continue to Fix Model mode:
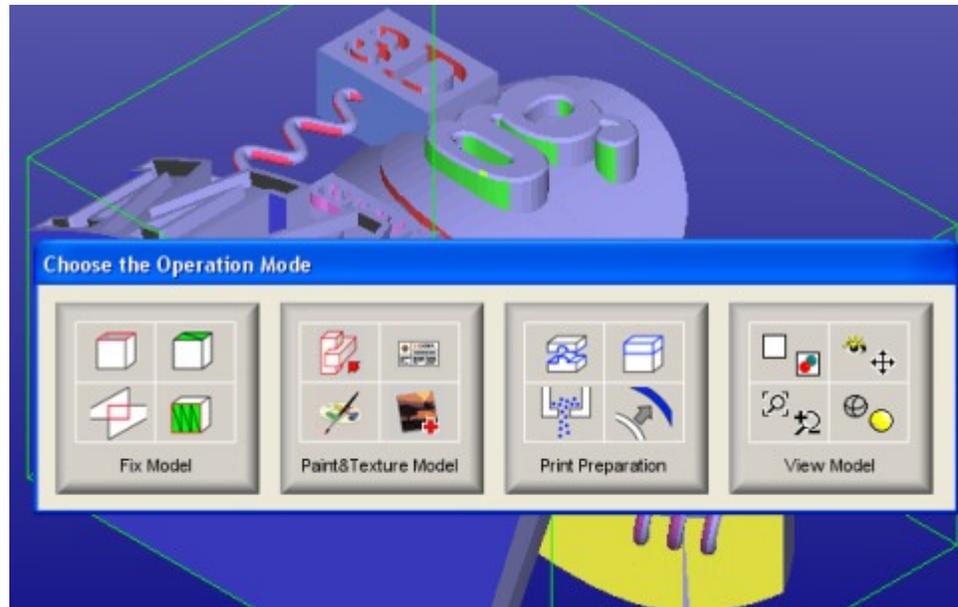
- **Press Fix Model to start the analysis** and after while you will see the error message:
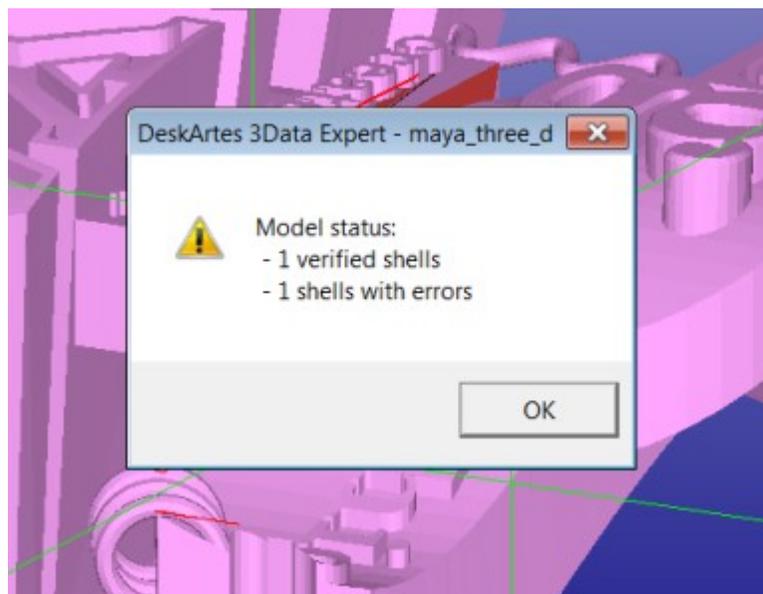
The normals are now consistently oriented and also the repair color is set to the normal sides of the triangles (the original material colors are stored for the 3D

Color Printer mode). There are also a few gaps visible on the screen. We can see that there are errors close to the "created with" text. Running Auto Repair may join the small details in the letters in a way which may cause difficulties later with the Boolean operations, so we will

- **press *No*** to the Auto Repair question:
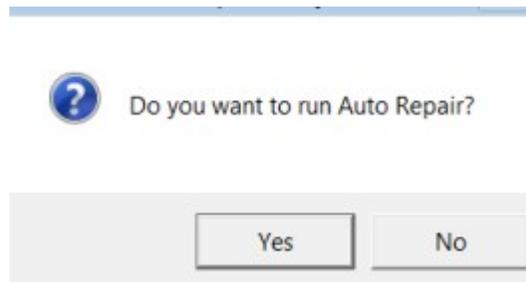
Familiar status message appears:
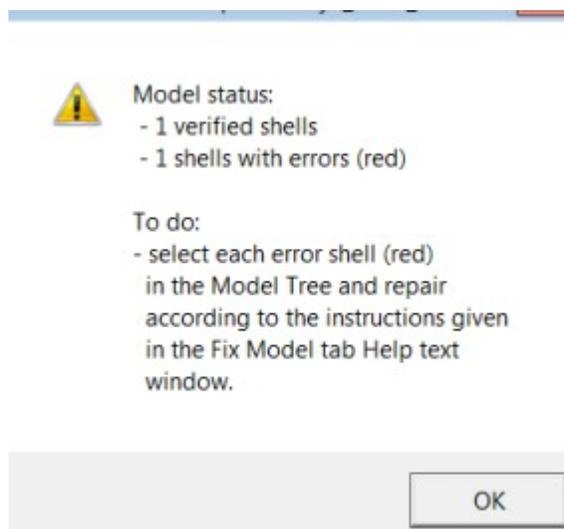
- **Press OK** to close the window and also press
- **Close** to the Tip window appearing if set On through the Help menu.

Now we will select the shell *1. maya_three_d (Verified)* from the Model Tree and have a look at the Help text in the Fix Model Tab.

The indicated errors, 137 overlapping triangles are typically the most difficult to fix. This may require removing surfaces manually and then adding some triangles to help correct filling for the gap. Also, re-trimming of some surfaces against each other is sometimes required.

The 10 intersecting triangles are either fixed by Remove Self-intersecting triangles command or Delete triangles / Repair Shells pair. Anyhow, now we must first remove the overlapping triangles to see the real situation concerning the intersecting triangles.

To start, let's rename the shell to shorter name, *maya*. If you now now open the Model Tree for the *maya* part, we can see no textures as was the case with the previous head model. Now the model colors are given through the material definitions in the VRML file.

You may now try to press the *3D Color Printer* icon to see the true colors of the model:
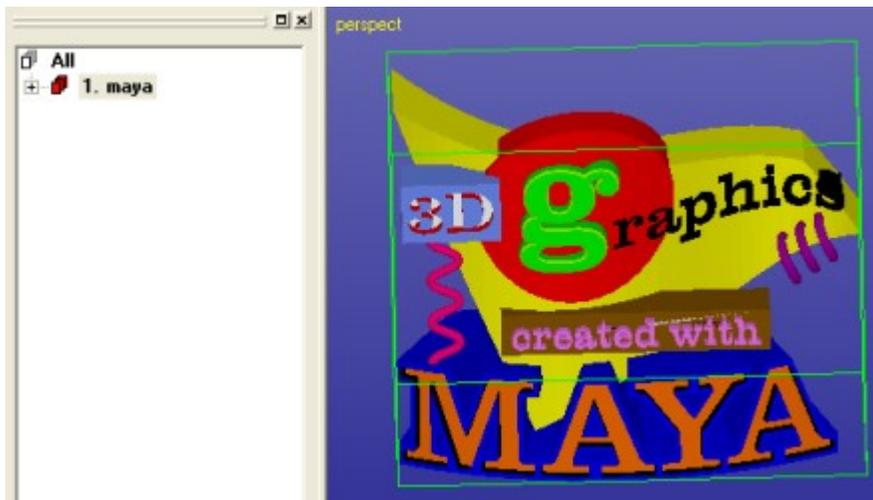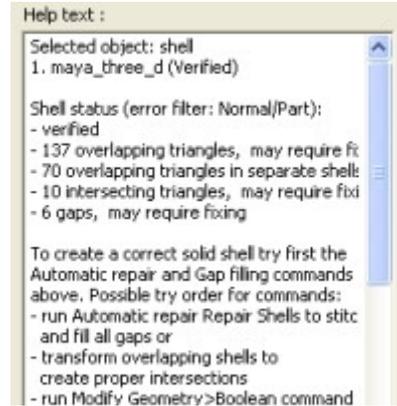
Return to *Repair* color mode by pressing the icon again before continuing.

Repair steps for the Maya model

The errors in the model are caused by erroneous triangulation of surfaces at the intersections of different model components. First you should investigate visually how serious the errors are.

A short investigation reveals that we can not just delete the erroneous triangles with *Fix Model > Delete Errors > All error triangles* command. Instead, we need to do some

1) The "*created with*" text requires to be re-connected to the part for correct STL/VRML file generation.

2) The box below the "*created with*" is not correctly trimmed to the curved surface attached to it.

3) Manual triangle removal and addition at the string "*graphics*" character borders to help the automatic repair to produce the correct result in the end.

4) Gaps should be filled with automatic repair.

Let's start with the worst problem, re-connecting the "*created with*" string.

Connecint the "created with" string

The string "*created with*" needs to be re-connected to the maya shell first. If you take a look from X axis and use clipping, you will see that the text is on a flat surface. In this case we can use the new *flat polygon area fill* functionality available with the Repair Shells command. In the clipped image below you can see that the gaps are on the flat plane:
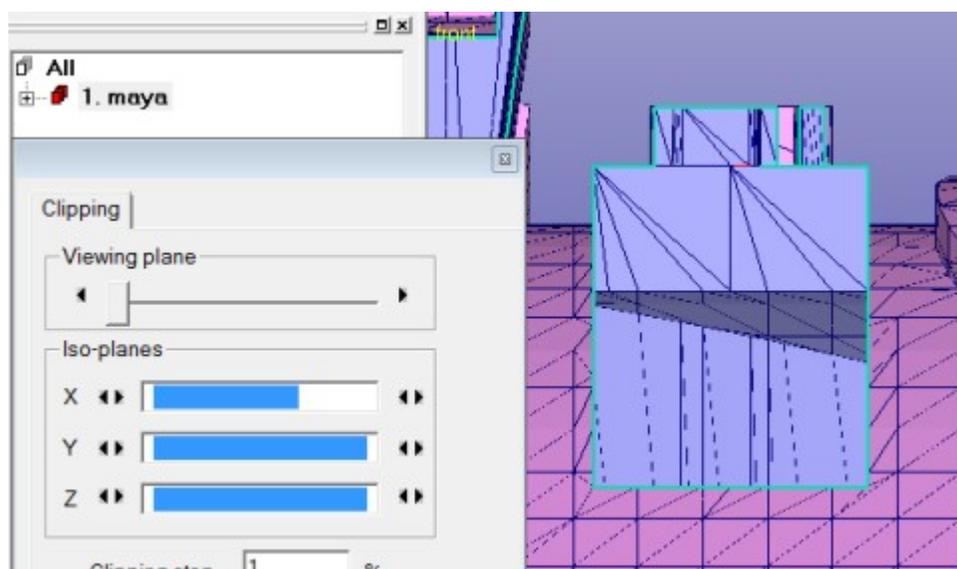
We will first clean up the planar area surfaces the text is attached to. Before selection make sure the *Selection mode* is in the *Surface* selection mode:

Then

- **take a view from X** axis direction
- **use Alt + LMB rubberband selection** to select the surfaces in the flat plane as seen below
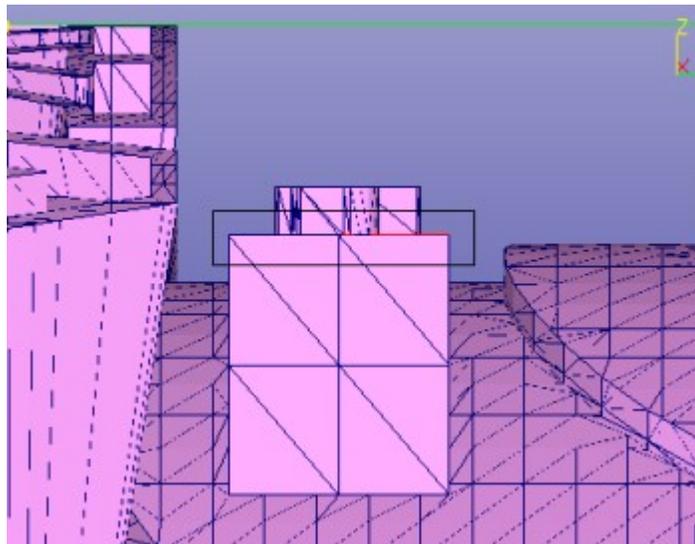
When the mouse is released, the surfaces at the plane are selected:
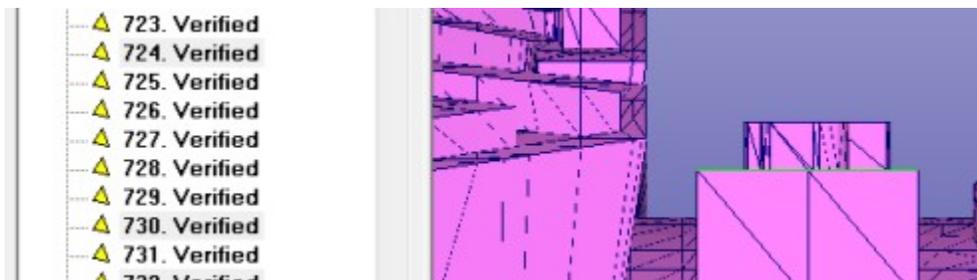
And from another view point:
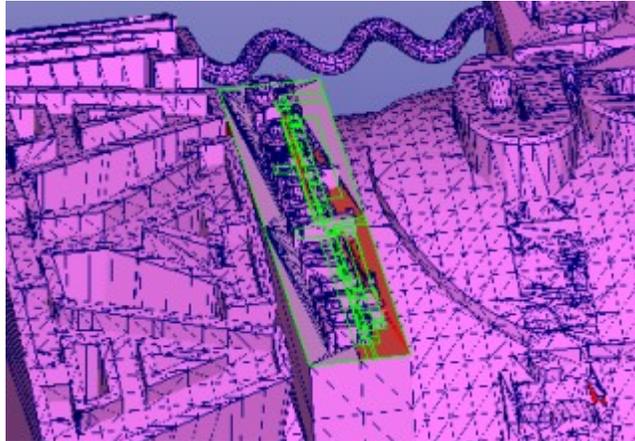
Then

  - give Edit > Delete command to delete all selected surfaces

Now

  – **select the maya shell**
  – **give *Fix Model > Repair Shells* command** to fill the gap between the text and the surrounding area

Soon you will see the parameters dialog:
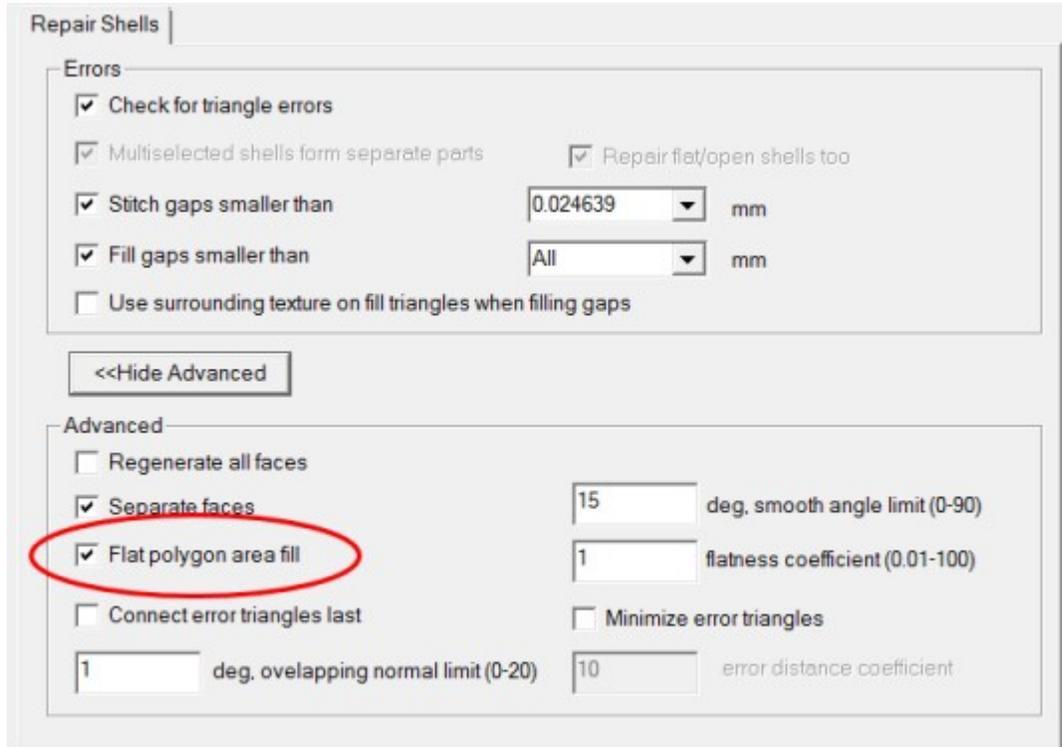
The new Flat polygon area fill, set on by default, will connect the flat gap polygons at the letter bottom edges to the surrounding box edges. All polygons lay in the same plane and the software use a special algorithm to fill between the separate polygons automatically.

- **press OK to run Repair Shells** on the maya shell.

The result on the text area is a prefect triangulation between the letters and from the letters to the surrounding square gap.

Fast visual inspection to the result will show as that the "*created with*" string is correctly connected to the *maya* component.

The *maya* model would be printable with any 3D Printing system with the current state. Anyhow, there are still a few *overlapping* and *intersecting* triangles and we will fix them for the completeness shake in the sequel. Also, some thin triangles may have caused inverted surface area to appear. This will also be fixed in the sequel.

Help text :

Selected object: shell
1. Multiple selection (Boolean)

Shell status (error filter: Normal/Part):
- verified, repaired
- 7 overlapping triangles, may require fixin
- 3 intersecting triangles, may require fixin

To create a correct solid shell try first the
Automatic repair and Gap filling commands
above. Possible try order for commands:
- run Automatic repair Remove Self-interse

Repairing the text base box

To locate the remaining errors you can either look for the red triangles on the screen or use the Locate Errors command to highlight the error triangles with yellow edge color and to fit them on the middle of the screen. Locate Errors tool will help you to pinpoint any error triangle or gap in the model.

– **Give *Fix Model > Locate Errors* command** or press the Locate Errors

button in the Fix Help Tab.

The Locate Errors Tab is displayed and error triangles highlighted on the screen. The dialog will give info on the total number of errors in the selected shell, *Previous* and *Next* buttons to select and fit the next error on the display area, error type for the currently selected error triangle as well as buttons to go to the next or previous *Shell* with errors
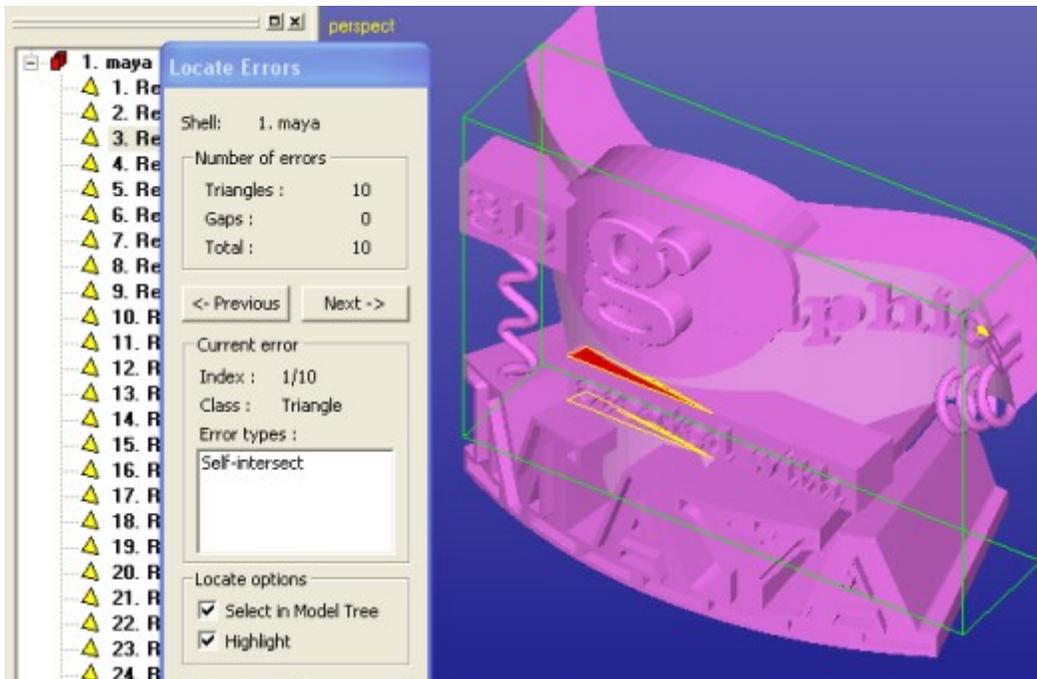
Press the *Next* button on the Locate Errors dialog until the following error triangle is highlighted on the scree:
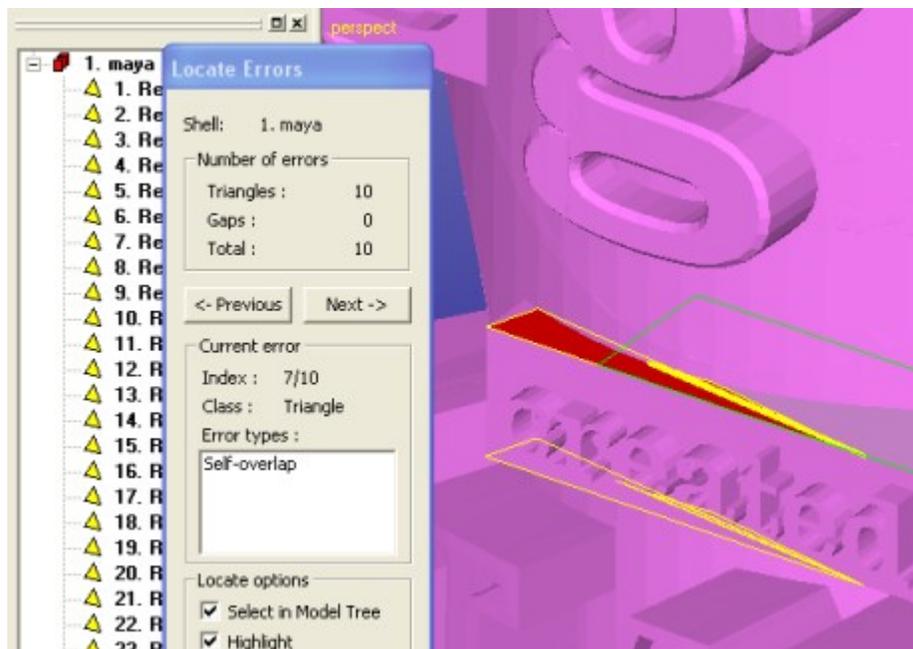
We will now clean up this overlapping triangle and the area surrounding it. You can

- **close the Locate Errors dialog now by pressing *Close* button**.

The box feature in the model has some overlapping triangles on the upper face, as seen in the image below:
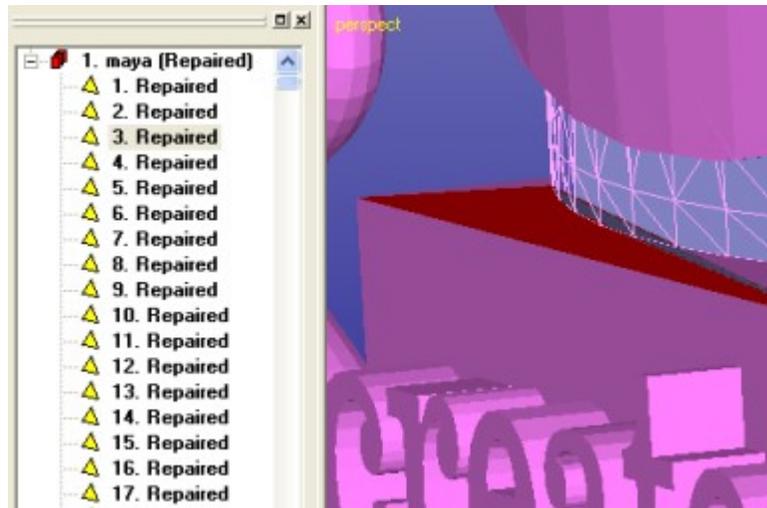
These errors can not be removed 100% reliably using automatic Remove Self-intersections command due to the tangential intersection conditions. Now we will remove a few triangles manually and then let the *Fix Model > Repair Shell* command to *Fill all* created gaps.

    – **Start triangle removal with *Fix Model > Edit Triangles* command** or using the Fix Model Tab shortcut


button**.**

Remove enough overlapping triangles from the error area, as seen in the next image. You could also use Surface selection mode and LMB selection with Delete to get rid of extra surfaces in the area.
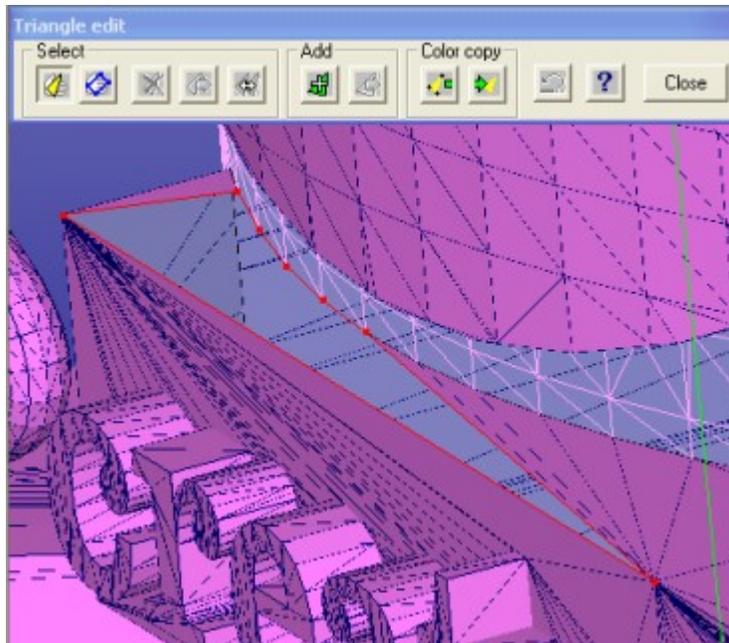
To finish the fixing at the "*created*" string, have a look at the bottom of the base.

- – **Select all red triangles and delete** them also on the down surface of the box to allow the automatic repair to fill the gap correctly:
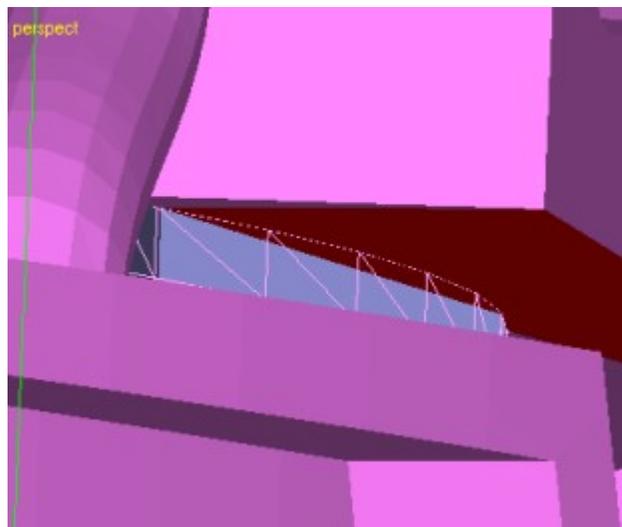
You may use the *mouse wheel zoom* and *middle mouse rotation* during the Edit Triangles command to get a better view to the delete triangles if necessary. After the removal the gap should look as seen here:
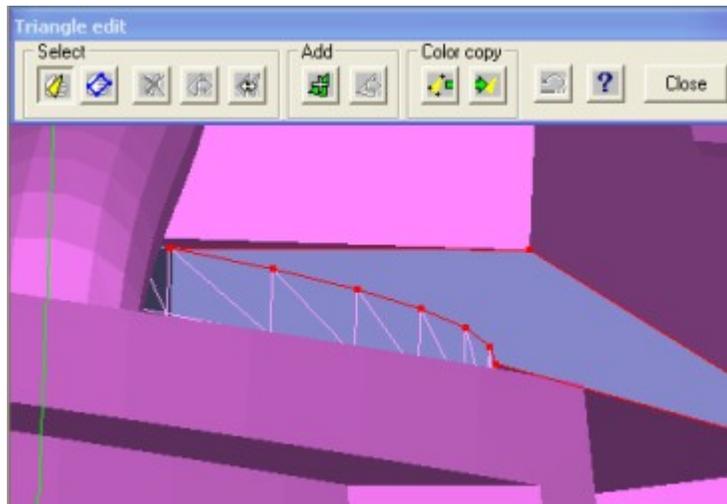
The run

- *Fix Model > Repair Shells* **command** and the corrected area is displayed:

Repairing the "graphics" string

To locate the last remaining errors we should use the *Fix Model > Locate Errors* tool again.

- **Give the command *Fix Model > Locate Errors*** to open the Locate Errors dialog or press the Fix Model Tab shortcut

button. Error triangles in the model will be highlighted with yellow color as seen on the image next page:
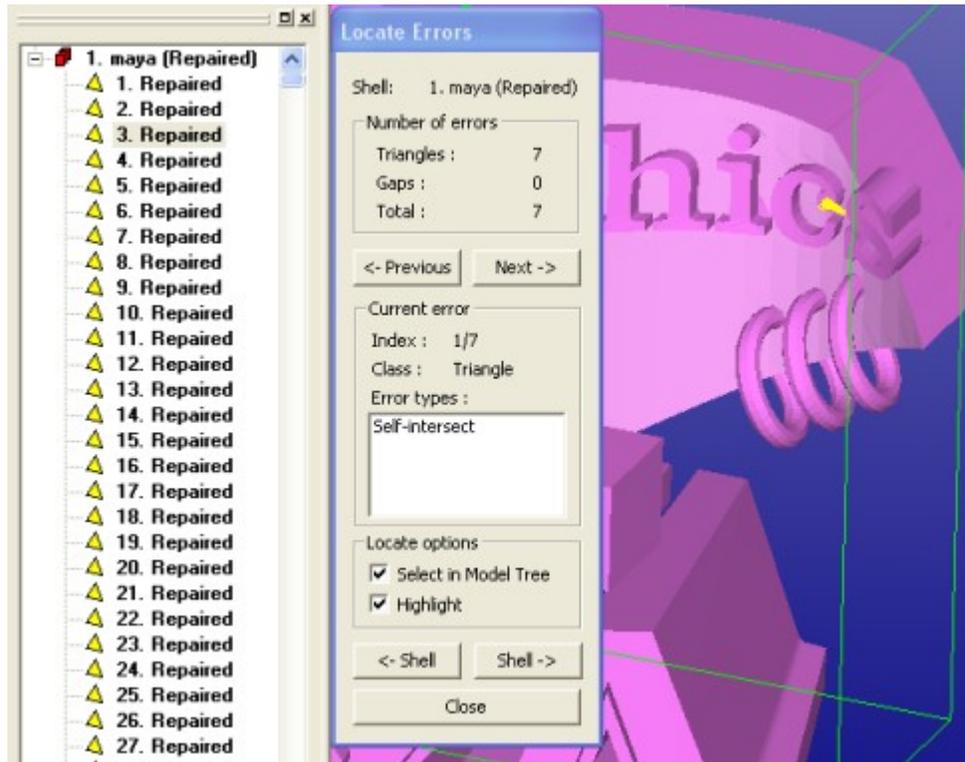
The last errors are at the letter 'c' in the graphics string. **Press *Next*** or Previous to zoom into the error area and the **close the Locate Errors dialog**. The errors around the letter 'c' are similar to the previous two errors we have fixed:

- Open *Fix Model > Edit triangles tool* and

- **remove the overlapping triangles with LMB selection + RMB delete**

The give

- *Fix Model > Repair Shells* **command** for a gap filling and you will have a correct *maya* model.
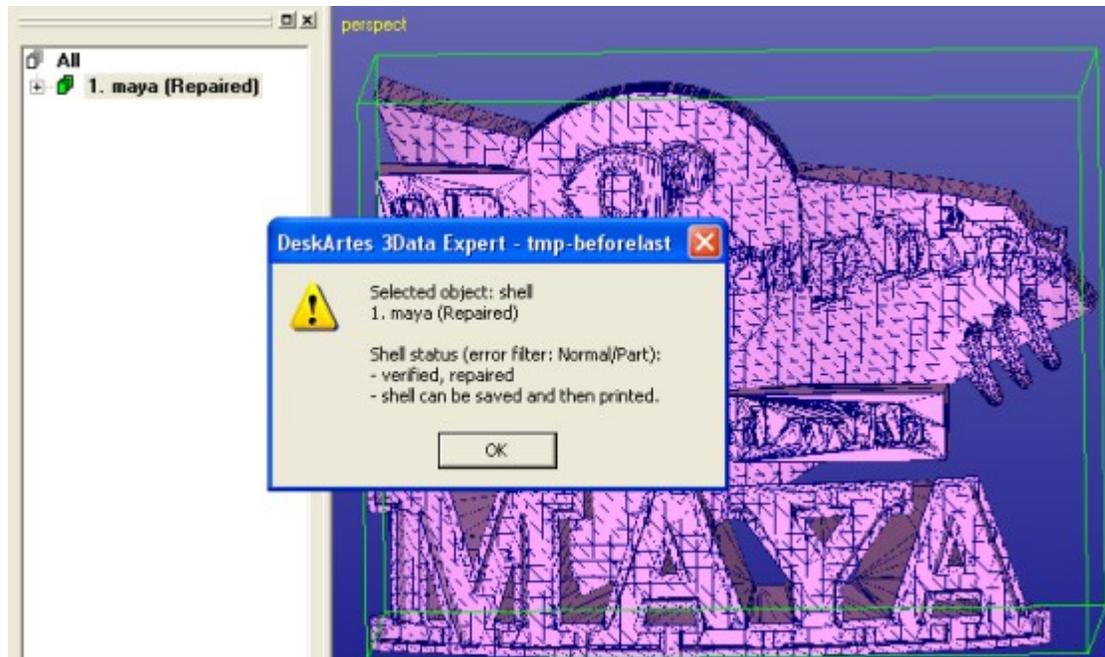
Note: You may find a new area or areas at the letter 'p' with similar errors after the last repair, just remove the *overlapping or intersecting* triangles in that case and rerun the Repair Shells once more. Or use *Fix Model > Edit Triangles* to clean up the error triangles, simplify the gaps and rerun the *Fix Model > Repair Shells* until there are no errors.

Outputting the Maya model

Once again, STL, VRML and ZPR output is straightforward *File > Save As* operation.

The correct file *tutorials / GeomFiles / maya_three_d-repaired.3de* is available in the example package.

If your goal is to take the model to 3D Color Printing, you will need to fix some color errors created during the topology fixing. Model coloring is described in detail in the "*Working with Colors" (*through the Help menu*)*.

Chapter 6 – Conclusions

Congratulations for reaching the end of this repair example document! By going through the examples you have learned a lot of the advanced STL and VRML model fixing functions in the DeskArtes Expert Series software. The handled models are difficult and in the everyday work you should not need all the skills acquired to prepare parts to be build. Anyhow, when a difficult model comes you will now know what do with it!

We have shown that the DeskArtes Expert Series, especially the 3Data Expert can be efficiently used to fix even most difficult geometrical problems. Investment in the 3Data Expert software will pay itself back very fast due to increased throughput of the 3D Printing system or seamless STL model transfer to simulation softwares. New Expert Series Plugins allow you to connect to other software products with ease.

There are still several other commands available in the DeskArtes Expert Series software suite for generating and handling triangle models. To know more about these commands we recommend you to go through the example documents available through the *Help > Tutorial Documents* command.

Thank you for spending you time with these exercises. We wish you pleasant and productive moments with the software in the future and also ask you not to hesitate to contact us at DeskArtes (support@deskartes.fi) if you have any questions concerning this manual or the Expert Series software in general.