3Data Expert 7.1

Tutorial

First Edition

Ismo Mäkelä: “DeskArtes 3Data Expert 7.1: Tutorial”.

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Credits

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Foreword

The purpose of these tutorial lessons is to help you to use the DeskArtes 3Data Expert software (version 7.1 or later) functionality efficiently and to solve the most common problems you will meet while handling 3D geometry data.

The Expert Series consist of several products: View Expert, Render Expert, Design Expert, 3Data Expert, Dimensions Expert and Spectral Expert. In this Tutorial we will cover the use of the View Expert and 3Data Expert software. The View Expert is targeted for viewing, verifying, dimensioning and communicating 3D geometry information. The 3Data Expert is used for manipulation of 3D data, including surface triangulation, triangulation repair and modification, like offsetting and boolean operations. Render Expert and Design Expert are for targeted for photo realistic rendering and free form surface design. Dimensions Expert is a tool for STL repair and splitting for machines with smaller build area, like Dimension from Stratasys or SD 300 from Solidimension. Spectral Expert allows you to add textures on your STL data for Z Corporation 3D color printers, like Spectrum system. ***Note****: Dimension Expert and Spectral Expert functionality is a subset of the 3Data Expert functionality.*

You need different licenses for DeskArtes Expert Series to go through the lessons. We recommend you to get a full evaluation license from DeskArtes before starting these lessons.

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

The second lesson explains how to perform surface verification and triangulation to produce a solid STL and surface file with 3Data Expert. The final part is positioned for Rapid Prototyping.

The third lesson fixes an erroneous STL model for Rapid Prototyping using both automatic and manual tools with 3Data Expert.

The fourth lesson describes methods to split and combine STL parts for Rapid Prototyping. Large parts must be split and pins have to be added to be able to attach the manufactured parts correctly with 3Data Expert.

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

The fifth lesson describes the tools in the DeskArtes Expert Series RP-module. These tools allow you to define platforms, move parts into platforms, make z-compensations, drain holes and different supports. You can also slice the model and do slice checking.

The sixth lesson shows you how to add textures on STL model and how to handle VRML models with 3Data Expert.

Lesson 1 – Basic for Expert Series

This lesson shows you how to do basic file input, viewing, dimensioning and verification of surface data with the View Expert. The verification information can then transferred into other CAD systems and/or users.

This lesson requires the Expert Series license for Verification and IGES input, i.e. View Expert license.

If you have used the Expert Series before in your computer you may want to delete the settings files in the user home directory and then then restart the program. Setting files are **SettingsVE.das** and **Settings3DE.das** for View Expert and Render Expert, respectively (names may contains version numbers). This initializes the system into its basic state and ensures that your results will not differ from the lesson results.

File input

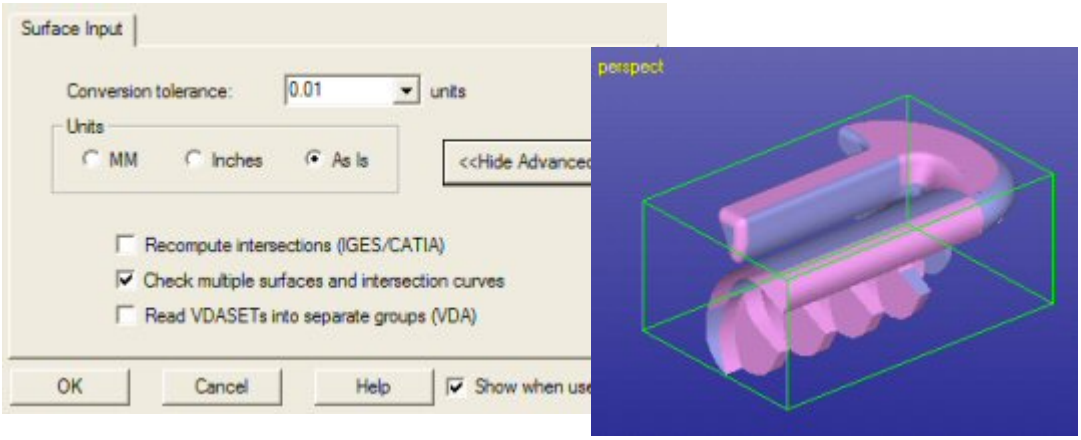
Start the View Expert program from the **1) Start/Programs/DeskArtes View Expert 7.1/Deskartes View Expert.** You can also use the DeskArtes View Expert icon created on the desktop during the installation. This lesson can also be done with 3Data Expert, there will be just more menus and toolbars available.

After starting the system and closing the Tips dialog you will see the basic configuration of the View Expert on your screen including menus, different toolbars and tabs and the display area.

The file input is always done through easy to use standard Microsoft File Open dialog with *File/Open* command. **2) Start by selecting** the *Tutorials/GeomFiles* directory under the Expert Series installation.

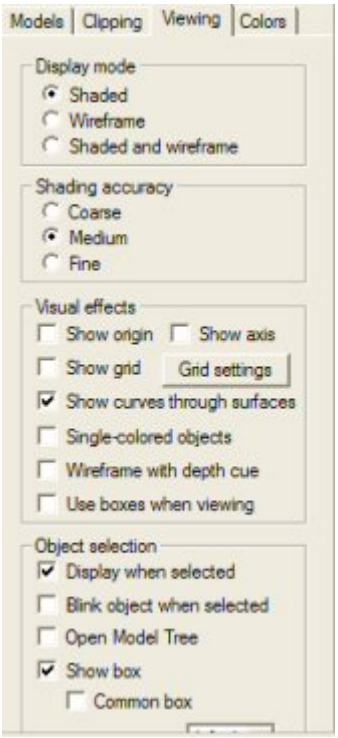
The file type is chosen from the Files of type dropdown list. **3) Select IGES** files (**.igs*) and you will see the list of IGES files in the GeomFiles directory. **4) Select the *slider.igs*** from the file list and press Open button.

The File Settings dialog is shown. With this dialog you can change the input properties for IGES files, like the units to use:



5**) Accept the default settings** by pressing OK. After a while you should see the slider geometry on your screen, as seen above.

The different colors of the object slider are caused by the inconsistent orientation of the surfaces in the input file. The outside (normal side) of the surfaces is displayed with pink color and the inside is displayed with blue color. You should not worry about this at the moment. You can also use the *Single colored objects* setting in the Viewing Tab to shade everything with one color. The green bounding box around the object shows the currently selected object. You can toggle it on/off with *Show box* setting in the Viewing Tab on the left had side of the program window. For more information on the different setting pelase see the Online Help *Model Window/Viewing Tab* chapter.



Model Tree

On the left hand side of the View Expert window you can see the Models tab.

This tab tells you the current contents of the DeskArtes Expert Series program database. With this file you can see two nodes, *1. slider (IGES Input)* and *2. slider (SURF Duplicate).* The first node includes the correct surfaces and the second node includes the duplicate surfaces in the input model. The default input mode separates the duplicate surfaces from the input data. We will later see how to inspect this information more carefully.

The first node in the Model Tree is now highlighted and this means that it is the currently selected object. Transformation, Solid and Modify menu and many other commands apply only to the currently selected object.

There are sevelal different ways to select objects from the Model Tree and the graphics area. For more information please see the *Modes/Selection/Selecting mode* page on the Online Help.

Moving the Eyepoint

The default mode for the Expert Series is *Move viewpoint* mode (),which enables you to look at the model from different directions. You can see the Move viewpoint icon when the mouse is on the display area. **6) First press the left mouse on the screen and drag the mouse**. You can see that the model rotates around the middle of the screen. Try this both in vertical and horizontal direction.

7) Then try pressing the right mouse button. You can see a *blue cross* appearing on the screen near by the move eyepoint icon arrow head. This is the *View Center Point* and it is now interactively selected. Drag the mouse and notice how the eyepoint moves around the newly selected *View Center Point*.

Panning

8) Select the command *View/Pan (*).Now the Expert Series changes into Pan mode. The mouse operations on the display area no longer move the eyepoint around the model but change the model position on the screen. Try the left mouse for continuous pan and right for rubber band pan.

For more information on the View Modes pleas see the Online Help *Modes/Viewing/Viewing* modes page.

Zooming

The Zoom mode is started either with *View/Zoom/Continuous* (left mouse) or View/Zoom/Rubber band command (ght mouse). 9) While in Zoom mode press the left mouse on the display area and drag it up and down and you will see the part changing smaller and bigger on the screen, respectively.

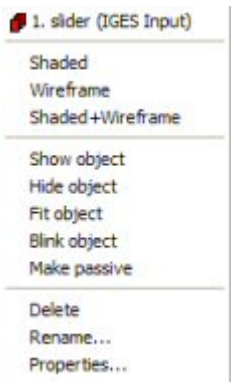
The right mouse performs the rubber band zoom. **10) Select a corner for the rubber band box and drag the mouse** to define the whole box where to zoom.

For more information on Zooming please see the *Modes/Viewing/Zoom mode* page on the Online Help.

Displaying and Erasing

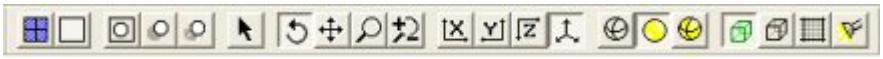
There are many ways to display and erase data on the screen. Lets now **11) erase the screen with *View/Show All/Erase*** command (+ left mouse). Now the screen is empty. 1**2) Click with the left mouse** on the node *1.slider (IGES Input)* in the Model Tree. The node becomes selected and the surfaces, i.e. the familiar slider, is displayed on the screen. If the model is not displayed, please check the Viewing Tab *Display when selected* setting.

13) Now move the mouse again on the *1. slider (IGES Input)* node and press the right mouse button. You will see the Model Tree right mouse menu appearing:

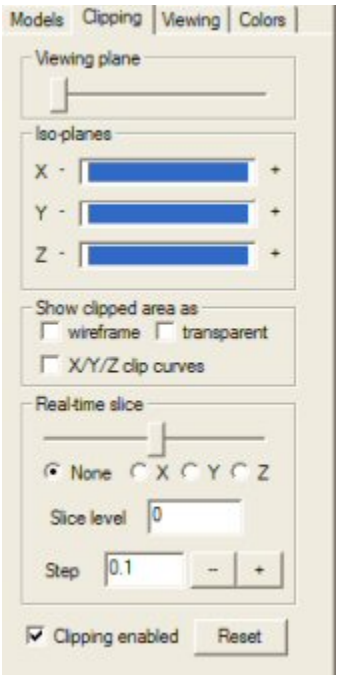


14) Try the popup command *Hide object* and see the slider model disappear.
15) Select the node with the right mouse again and give the popup command ***Show object*** and see it appearing on the screen again. Try also the popup commands ***Fit object*** and ***Blink object***.

16) Now try all different shading modes for the *slider* model. Try commands *Shaded, Wireframe* and *Shaded+Wireframe* and see the differences on the display. The shading modes can also be set through Viewing Tab and View toolbar:



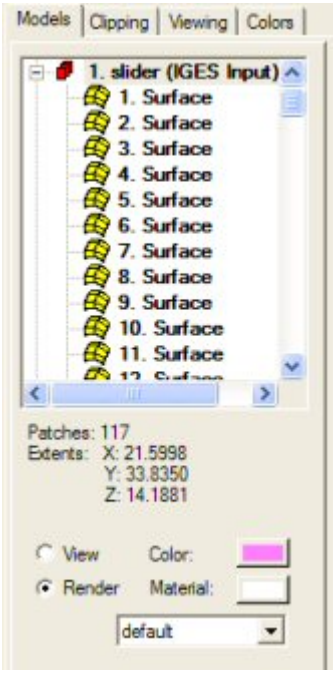
17) Try also Clipping functionality throgh the Clipping Tab to look inside the model. The clipped area can be displayed with different ways, like wireframe or transparent. Clipping also enables to extract real time slice information on the model. *Real time slices* can be copied to the Model Tree for dimensioning with the *Edit/Save Clip curves* command, for example.



Selecting objects

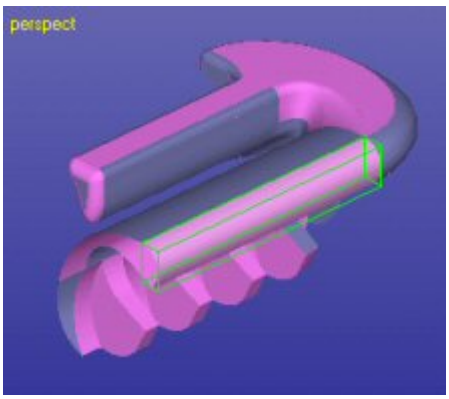
The object selection can be done either from the display area or from the Model Tree. **18) Erase the screen and then display the object node *1. slider (IGES Input)* again** by clicking it in the Model Tree. Now we have done one selection at the model level. This level can also be called as the part level or component level, depending on the situation. Click on the little box with the '+' icon besides the part name. The result should look like below:

The node *1. slider (IGES Input)* has opened and shows the contents of the node. The node consists of several leaves, Surfaces, which make up the whole slider. Surfaces are always identified with the icon.



19) Select the surface number *89* from the model tree by clicking it with the left mouse. You can see one of the surfaces displayed with a green Bounding Box around it. Now this surface is your current selection and some commands will apply to this surface only. **20) Try *View/Show Object/Blink*** ( ), for example.

The surface number *89*:



You can also select nodes and leaves from the display area by pointing with the left mouse. First **21) close the *slider* part** by pressing the box with '-' token, **select the node *slider*** from the Model Tree and **give command *View/Show Object/Fit*** (+right mouse) . **22) Now set the system into *the Selection mode*** by pressing the *Selection mode* icon (or use *Edit/Select mode* or *Esc* key).

23) Now click on the surface *89* on the display area with *the left mouse*. You can see that the surface once again becomes brighter and the surface is shown highlighted in the Model Tree also. Click on the neighbor surface on the display area. Now this becomes selected and highlighted. Click again on top of the same *surface*. Now the whole *slider* model becomes selected and highlighted again.

When the system is in the selection mode the right mouse has a special meaning on the display area. **24) Clicking the right mouse on top of an displayed object brings up the *Display Area Menu***:

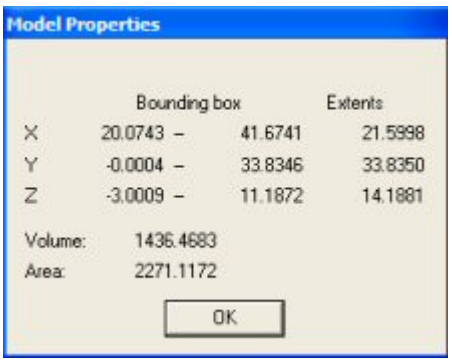
By giving the Select command the surface which is nearest to the cursor is selected as the current object. **25) Do this on top of the *surface 89* again**. With the *Next/Previous* command you can select surfaces which are behind the *89* surface along the line of sight at the mouse position. You can also change the shading and visibility of the selected object with the commands in the menu.

Multiple selection can be done by pressing the *Ctrl* key donw while selecting objects from the graphical area or from the Model Tree. The *Sift* key can be used to select objects between the first selection and the latest selection in the Model Tree. By pressing the *Alt* key wihle making a selection in the graphics area will automatically select the component level object instead of the surface below the mouse. Multiple selection can be used with several commands, like the Transfomation menu commands.

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, as seen on the Model Tree image on the page 8. There we can see the number of surface patches (117) and the extents of the model (X: 21.5998, Y:33.8350, Z:14.1881).

26) The command *View/Properties* and the Model Tree popup menu command *Properties* invokes a dialog with more accurate information of the part:



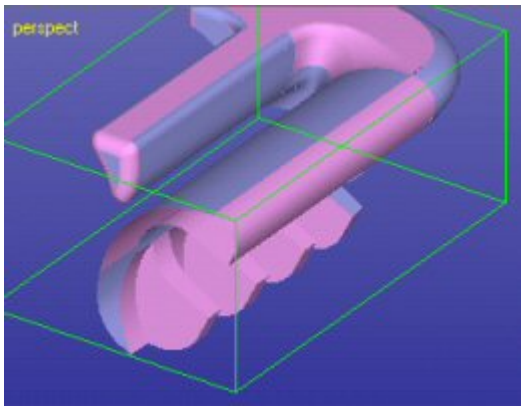
The exact position of the part is now given as bounding box values. Also, the volume and the surface area of the object is calculated.

Note: with this example the *Volume* is not correct. This is due to the fact that the model is not solid and the surfaces are not oriented consistently which can be seen from the different surface colors, pink and blue, on the *slider* component. The *Volume* is reliable with Repaired STL models wihtout gaps or consistently oriented surface models without gaps. The *Area* value is always valid both for surfaces and STL models.

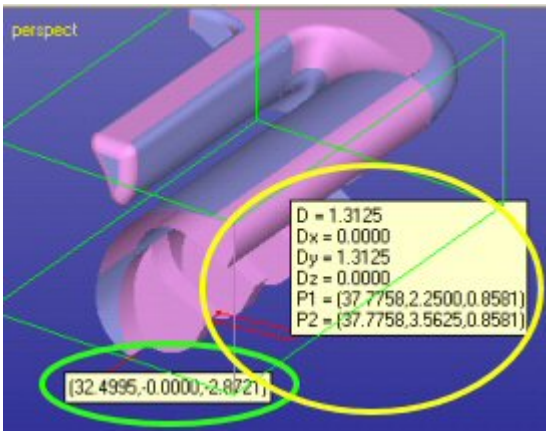
Dimensions and Annotations

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.

Lets first take an absolute point value from the slider. **27) Rotate and zoom the model** so that you can see it as show below:



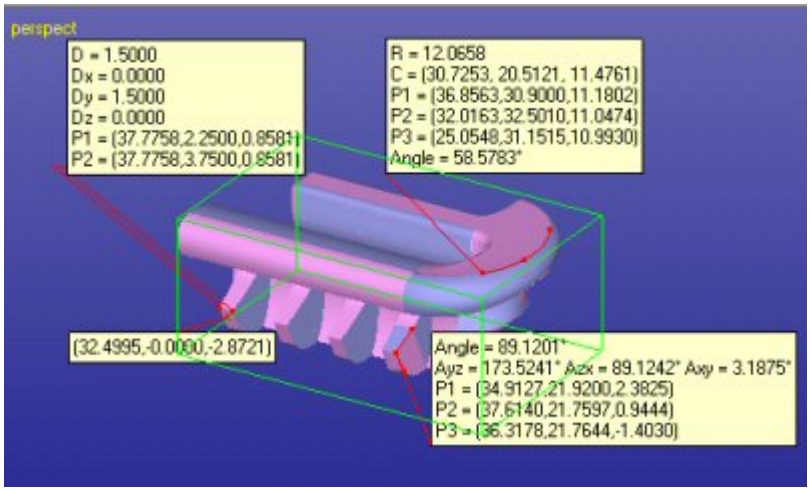
28) Now give the command *Dimensions/Point Value* () . The Crosshair icon is shown on the display area and the program is waiting for you to select a point on the model for co-ordinate values. Select one point from the model and you will see a result like below (in the green circle):



The program is now in the *Annotation* mode (if set *Sticky*, see *Edit/Customize dialog*) and you can give as many *Point Values* as needed. Let's now try distance between two points (yellow circle below). This is done with **29) command Dimensions/Distance**. After you give the command you will see a new *Crosshair* icon. The number *1* beside the 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 above.

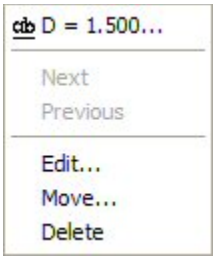
Now you are in a *Distance* evaluation mode and you can make as many distance evaluations as needed. If you accidentally select a wrong point on the screen you can step back by pressing the *ESC* key on the keyboard.

In the same way you can take *Angles* and *Radius* using the Dimension menu. Try to repeat the example shown below. Remember that you can *Zoom*, *Pan* and *Move* the eyepoint using the keyboard shortcuts while in the *Annotation* mode. Try pressing *Ctrl*, *Sift* and *Ctr+Sift* buttons while moving the mouse on the screen. For more infomation on the keyboad shorctus for please see the Online Help page *Viewing/Shotcuts*.



30) Now clear the workspace with *File/New* command and **input** the Expert Series geometry file *Tutorial/GeomFiles/SliderDimensions.da*. You should see similar information as in the figure above. If you do not see the annotations give the command *Dimensions/Show Annotations* or create one new annotation yourself.

Now you are ready to edit and move the annotations. **31) Set the View Expert into *Selection mode*** (press *ESC* key) and **click with the *right mouse* on top of an annotation**. You should see the Annotations popup menu below. If you see Display area selection menu select the Next command until you get the wanted menu.



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 ( ).

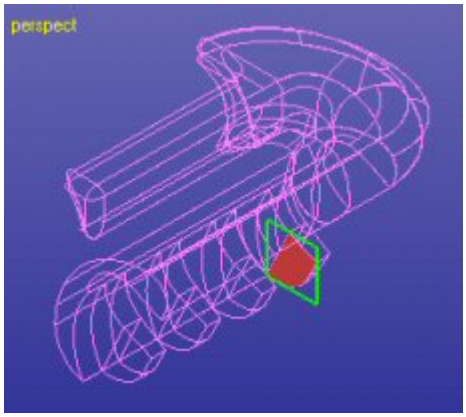
If you are in some other than the *Selection* mode you can still get the graphics area Selection menu or the Annotation menu on the screen by pressing the Menu key on the keyboard.

You can also display model information and copy it into the cut buffer by simply giving the command *Dimensions/Object Info* ( ) and *Dimensions/Copy Object info* ( ), respectively. From the cut buffer it is easy to paste the information to Excell, for example.

Verifying the geometry

In this section we will verify the correctness of the *slider* part. For this purpose **32) clean the workspace again with the *File/New* command and input the *slider.igs* again**.

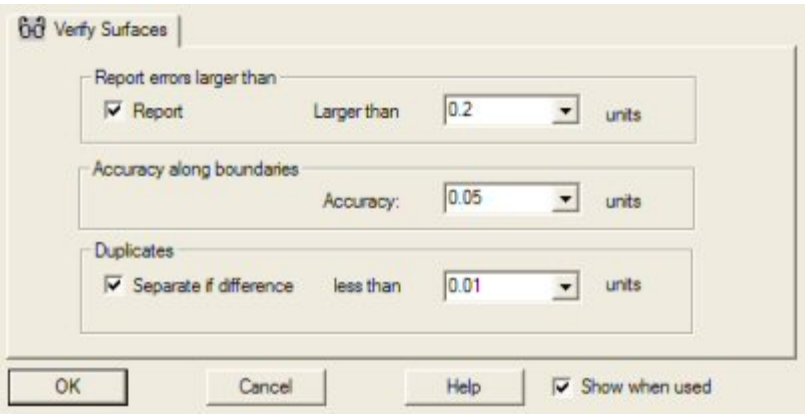
Once again you can see the Model Tree with two nodes, *1. slider (IGES Input)* and *2. slider (SURF Duplicate)*. This already reveals us that there has been an error in the input file, a duplicate surface is found. Now **33) change the display mode of the first component *slider* to Wireframe** and then display the second component in shaded mode. You should see the following information on the



graphics area:

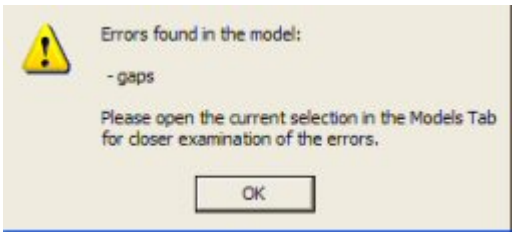
The duplicate surface is shown in red in the lower part of the display area. Now you could send this problem geometry to the designer of the slider with email, for example. Select the *File/Send mail* command to send the whole Expert Series geometry to the wanted recipient. The designer can then have a look at the problem with the free version of the Expert Series.

Let's try the main verification method with the View Expert. **34) Hide all and select the component *1. slider (IGES Input)* again. 35) Give the command *Solid/Verify***. You will see the the *Verify Surfaces* parameter dialog:



The program will check the *slider* component using the given tolerances. *All gaps* which are l*arger than 0.2 units* are reported. The *Accuracy* controls the handling of surface boundaries and *Duplicate* surfaces which are closer than *0.01 units* from each other are reported. **Note**: For this model we have performed duplicate checking already during the input.

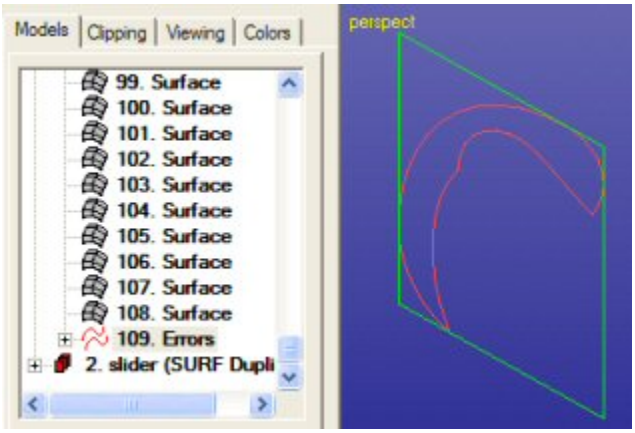
36) Start the verification by pressing the *OK* button. After the verification the program will show an error message dialog



This tells that there are gaps in the result. **37) Press *OK* to close the dialog**.

Let's now take a closer look at the gap curves. Now the name of the first node has changed to *1. slider (SURF Verify)*. **38) Open this component** and scroll to the bottom of the list. **39) Select the leaf *109. Errors.* 40) Hide all and use**

View/Show Object/Fit (F4) command to fit the selected object in the display area. You should see the following result:



We have verified the surface model and noticed that there is a gap in the model. You can aplify the gap by selecting and showing the *slider* component in shaded mode (+ right mouse).

To report the error to the designer of the slider model we can write the gap curve in IGES format to a disk file and send this file back to the designer. **41) Select the 109. Error curves** from the Model Tree and use **42) *File/Save As*** to write the disk file. Select *Save as type* to IGES (*.igs) and give the file a suitable name, like *slidererrors.igs.*

View Expert can only output error curves into IGES format. Outputting surface data in IGES format is available with the 3Data Expert and Design Expert software packages.

This concludes the first lesson. This lesson was done with the View Expert software, and it could have peformed with any Expert Series software package. Anyhow, to continue to the lesson 2 you will need the 3Data Expert software.

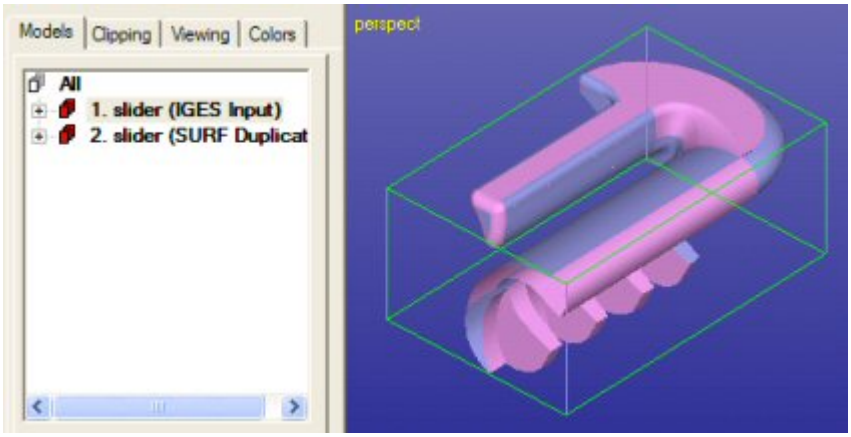
Lesson 2 – Surface Triangulation

This lesson shows you how to do surface verification and triangulation to produce a solid STL file with 3Data Expert and how to position it for manufacturing. The lesson also explains how to generate a closed surface model from an open surface model.

This lesson requires the 3Data Expert Base and IGES/VDA module licenses.

Verifying the surface

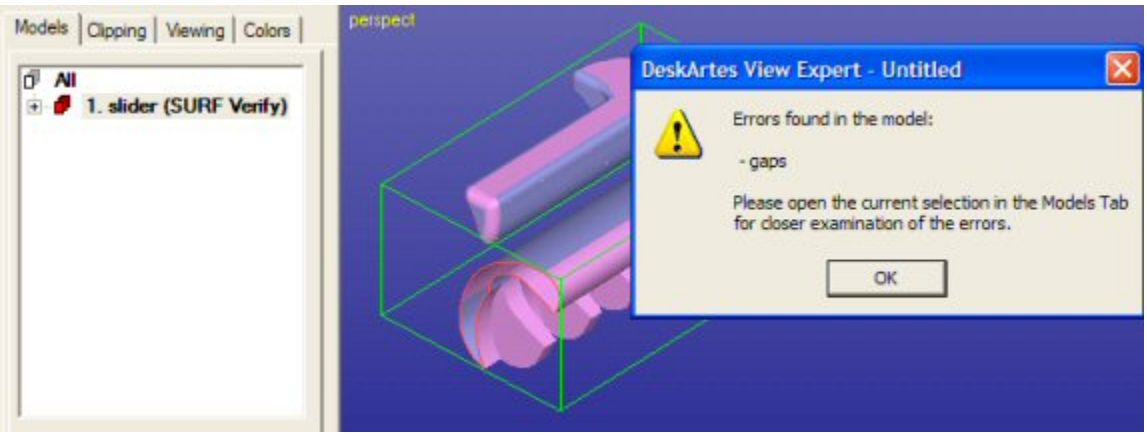
In the previous lesson we learned how to input IGES files and how to verify it. Start this lesson by repeating the verify operation: **1) clean up the workspace** with *File/New* command and the **2) read in the file *Tutorial\GeomFiles\slider.igs*** as shown in the first lesson**.** You should see the the slider on the display area and tow components in the Model Tree, as seen below:



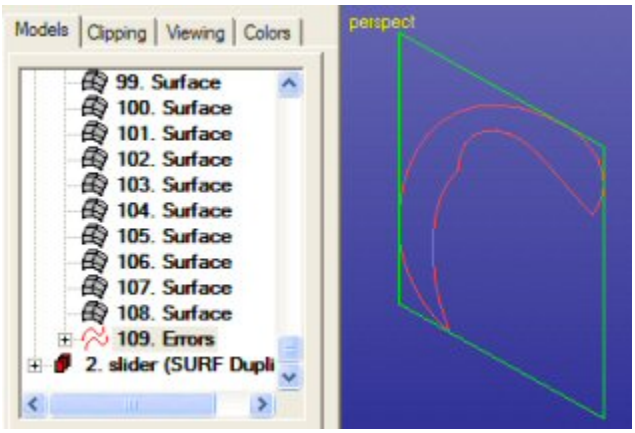
The outside of the surfaces are shown pink and the inside is shown in blue color.

The two nodes in the Model Tree describe what was 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. **3) Just click on the *duplicates* and then use *Edit/Delete* command ( ) to remove them**.

The verification of surfaces is done through the Solid/Verify command . **4) Start the *Solid/Verify* command** ( ), check that you are using the parameters shown on the page 16 and **5) press *OK*.** After a while you will see the following result:



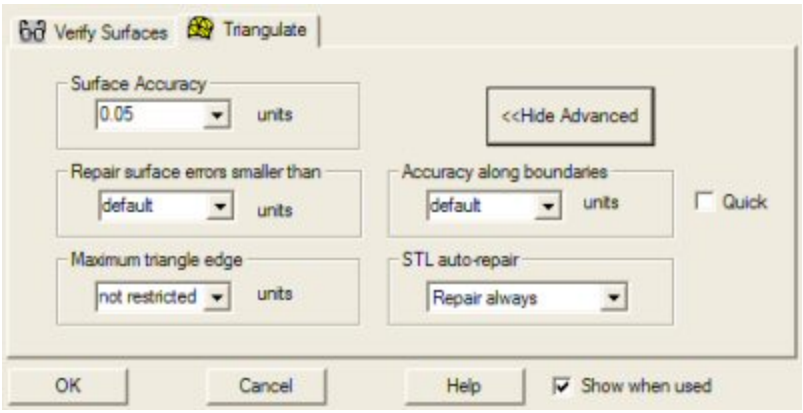
The folder icon in front of the *slider* component is red. This as well as the result dialog point out that there are errors in the model. The *slider* has gaps. To display these gaps **6) erase the screen first with the *View/Hide All*** command and then click on the '+' box at the *slider* node. After that **select the 109th E*rror* curves** 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 *Solid/Repair* command . Repairing will be done later.

Triangulating the surface

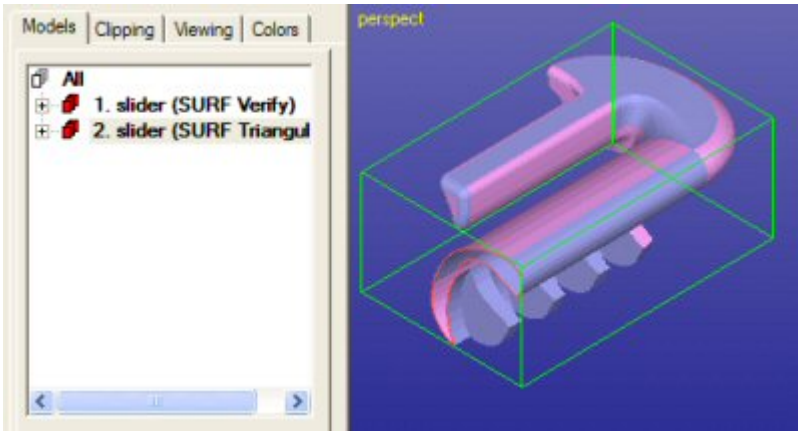
7) Triangulate the slider part with the command *Solid/Triangulate* ( ). The triangulation can only be done at the node level and the program automatically proposes the node *1. slider (SURF Verify)* as the target object if a *surface* or *error gap* leaf is selected. If this happens, press OK to accept it. You will see the *Solid/Triangulate* settings:



These parameters control the accuracy and error repair with the triangulation. The most important is the *Accuracy* which gives the maximum distance of the triangles from the original surface model. See the Online Help *Menus and commands/Solid/Triangulate* page and its settings for more details on other parameters.

Before we continue let's **8) change the *STL auto-repair* setting to *Do not repair*.** We do not want to run the triangle repair automatically after the triangulation at this time. Then **9) press *OK* to start the triangulation.**

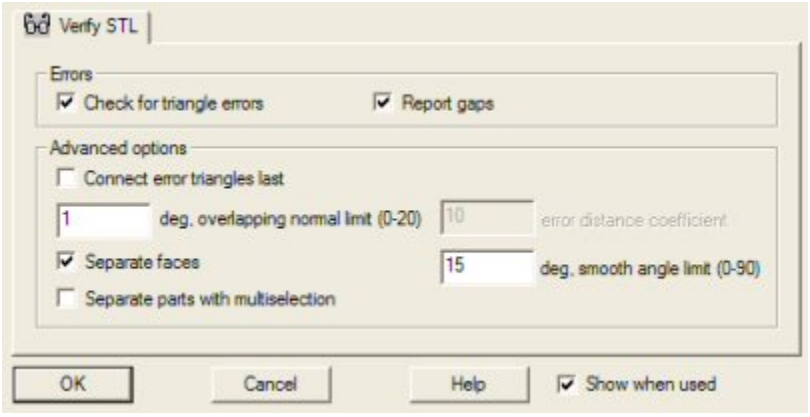
After a while you should see the following result on the display area and the Model Tree:



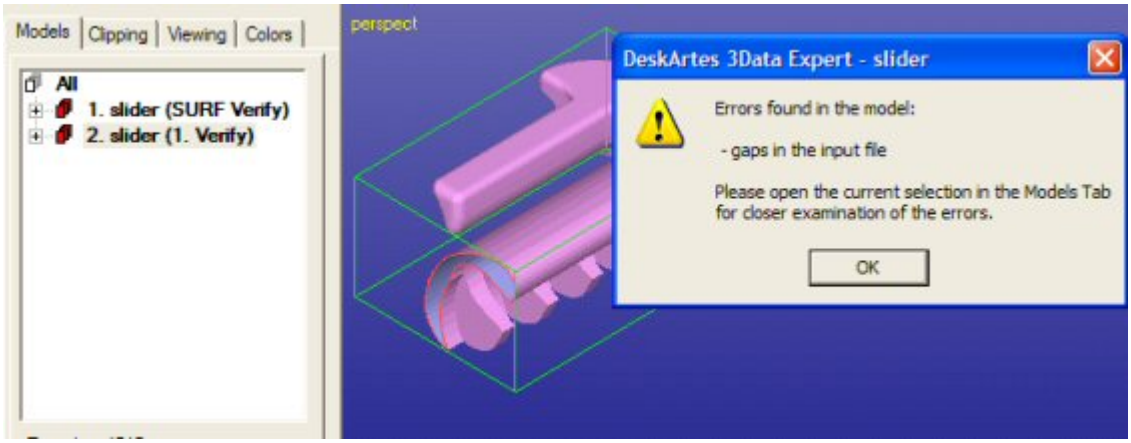
Inspecting the display and the Model Tree shows us the same gap as after the *Solid/Verify* command did. Also, some triangles are shown in pink and some triangles in blue color. The triangle outside is shown in pink and inside in blue the same way as it was with the surfaces. **10) Change the shading to *Shaded+Wireframe*** mode ( ) to see the individual triangles.

STL Verify

We will continue with triangle verification to check any errors in the triangulated model. **11) Give the command *Solid/Verify*** again for the *2. slider (SURF Triangulate)* component and you will see the Verify STL settings dialog (the software automatically recognizes the type of the verified object):

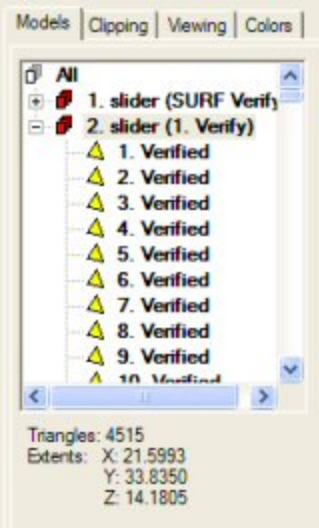


We want to know all the gaps and check for possible intersections in the triangulated model. Check that your settings are the same as in the picture above, especially set the *Check for triangle errors on* if not set. **12) Then press *OK* to start the processing**. You should see the following result



which reports us the same gaps as with the previous commands. The drawing area will show you oriented *slider* with *gaps.*

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



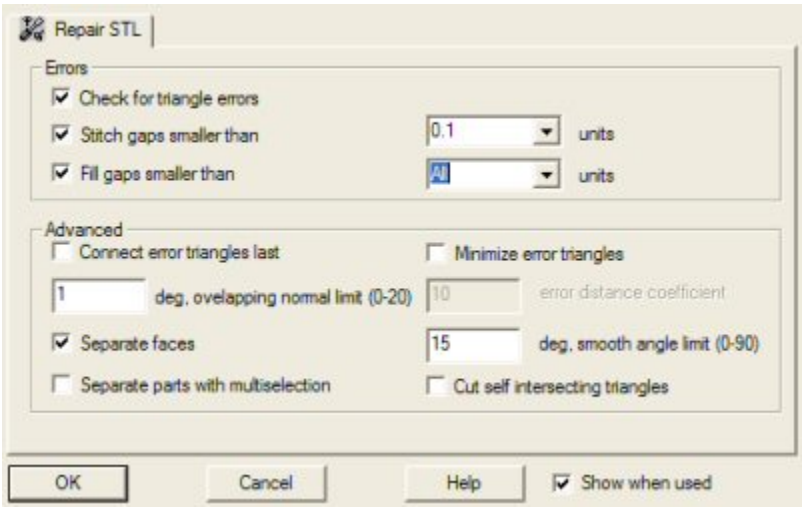
The model outside is now all pink. This is due to the fact that the verification also orients the triangle normals consistently, as seen in the picture on the previous page.

The surface subdivision of the verified (or repaired) model will allow us to use the different surfaces efficiently for subsequent repair work, if necessary. Separate surface can be extruded or cut against each other, which will be shown later in the lessons.

STL Repair

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

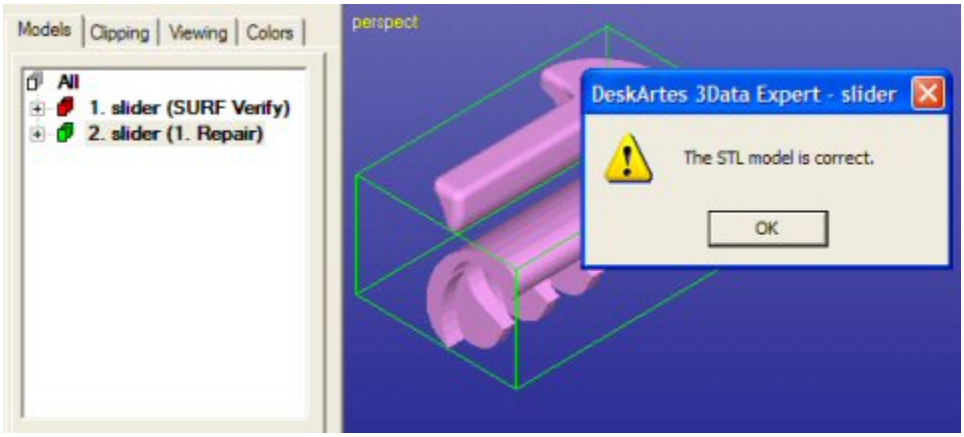
13) Start the command *Solid/Repair* with the *2. slider (STL Verify)* node. The Repair STL settings dialog



will show you the parameters in use. Again, we will *Check for intersections* in the data. Now we will also close gaps in the model. *Stitch gaps smaller* than parameter gives the upper limit to sew together thin gaps. The *Fill gaps smaller* than parameter will give the maximum size for filling individual large gaps. Here we want to close *All* of them. Make sure you have the parameters above and **14) start the command by pressing *OK***.

Note: *The Advanced parameters are needed when you are working with models with difficult errors. You can learn to work with difficult models by downloading the "Expert Series 7.1: STL and VRML repair" document and associated example files from the DeskArtes web pages.*

The result will be a correct STL model. This is indicated with the green box in front of the node *2. slider (1. Repair)* and the result dialog:

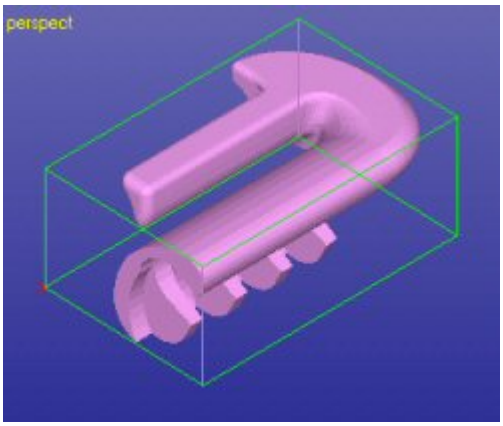


The display area shows a pink model without any gaps. This means that we only see the outside of the triangles i.e. all triangles are correctly oriented.

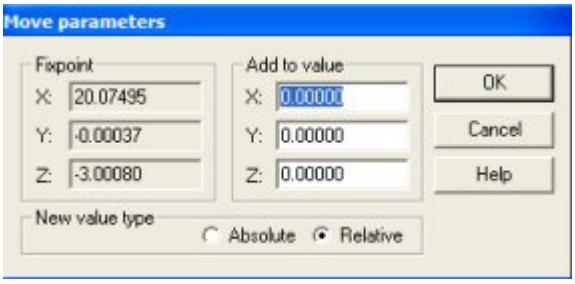
The Model Tree indicates a correct model with the green icon. If you open the Model Tree and select individual surfaces, you will see that the surfaces are sorted by the surface area. The first *Triangles* surfaces is the largest and the last surface is the smallest. You can also use graphical selection to select individual surfaces by showing with the mouse can clicking on them.

Positioning parts

Now it is time to position the part in the correct place for a Rapid Prototyping system. The part can be positioned for example with numerical commands through the *Transform* menu. For that we first have to set the Fix point for the node *1. slider (1. Repair)* into its minimum bounding box value. **15) Set the Fix point with the command *Transform/Fix point/Interactive*** () to the minimum x,y,z corner, seen in the image below (if you do not see the Fix point, use the  icon to make it visible).

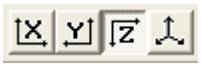


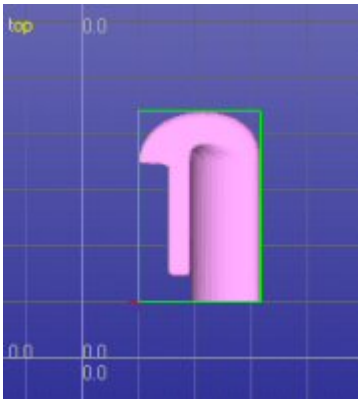
16) Move the part to the wanted position with the *Transform/Move/Numeric command.* The command brings up a dialog box for *Move parameters*:



The *Fix point* is shown in the left hand side of the dialog. Now we will set this value absolutely to some other position (**select *Absolute***), i.e. move the part. **17) Lets fill *New value* fields with value 10, 10 and 10 and press OK**.

The part is now moved to the wanted position. This can be checked by **18) displaying the grid ( ) and using different view directions X, Y and Z**





.

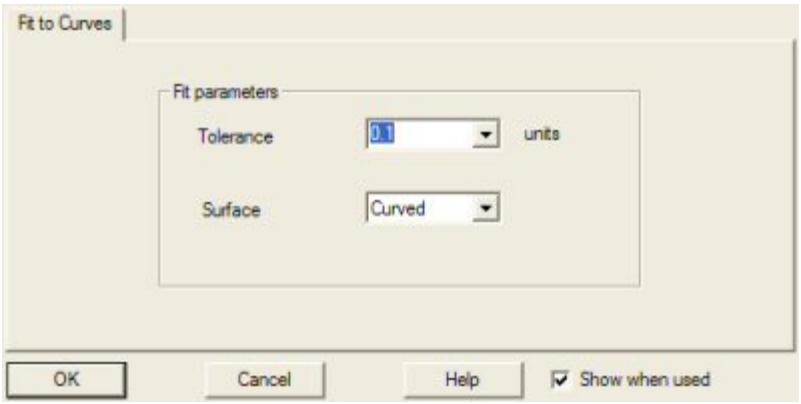
It is also possible to use the *Interactive move* and the grid to position the part. The *Grid size* can be set through Viewing Tab (or + right mouse).

Prototyping/Move to Platform command gives some specialized functions to move the part into the right place in the work area. They work together with the selected Platform. Please see the Online Help for more details on these commands.

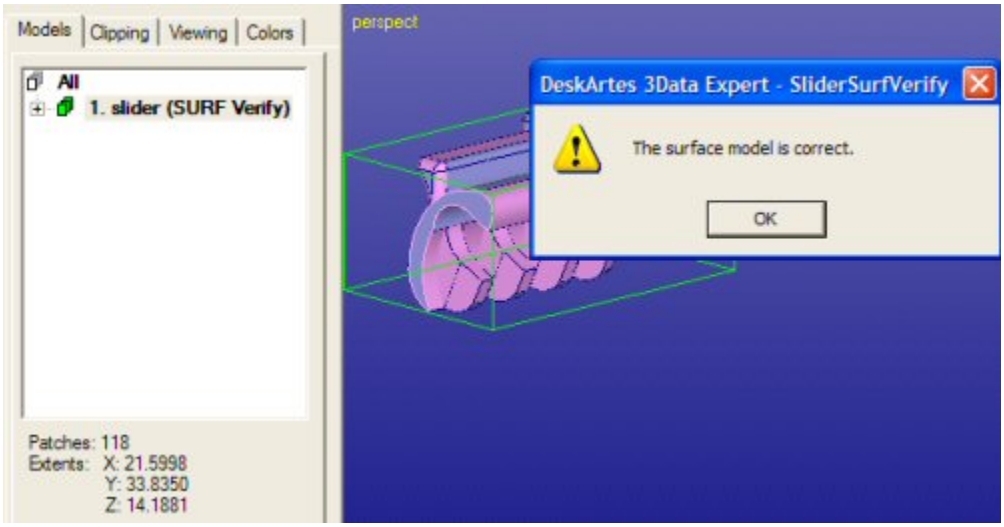
Creating correct surface model

If the purpose of the verification and repairing is to generate correct IGES surface data without missing surfaces, the 3Data Expert gives you tools to do that also. To generate new surface **19) clean the workspace** and **20) input the model *Tutorial\GeomFiles\SliderSurfVerify.da***.

21) Select the leaf *109. Errors* which includes the gap curves. Now we want to use these curves to generate a surface to fill the gap. **22) Give the command *Solid\Fit to Curves***. The Fit to Curves settings dialog is shown



23) Press *OK* to start the surface generation. After a while you will see the newly generated surface which fits exactly to the gap. You can now **24) verify the surface model with the *Solid\Verify*** command (use the default parameters) and the result will be correct surface model as shown with the result:



25) Output the surface model in IGES format with *File\Save As* command. Give a suitable name and then set *Save type* as to *IGES (*.igs).*

It is also very straightforward to run the *Solid/Triangulate* and then the *Solid/Repair* to generate a good STL data from this surface model. This is left as an exercise for the reader.

Lesson 3 - STL fixing

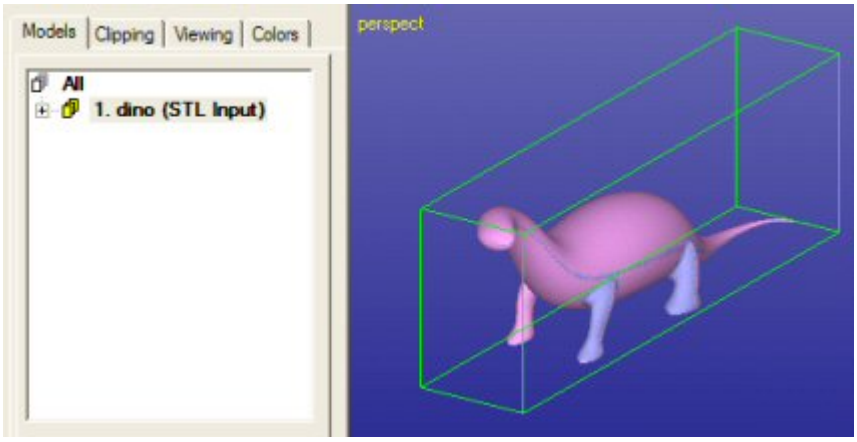
This lesson shows you how to fix an erroneous STL model for Rapid Prototyping or simulation software use using both the automatic and manual tools with 3Data Expert.

This lesson requires the 3Data Expert Base license for STL handling.

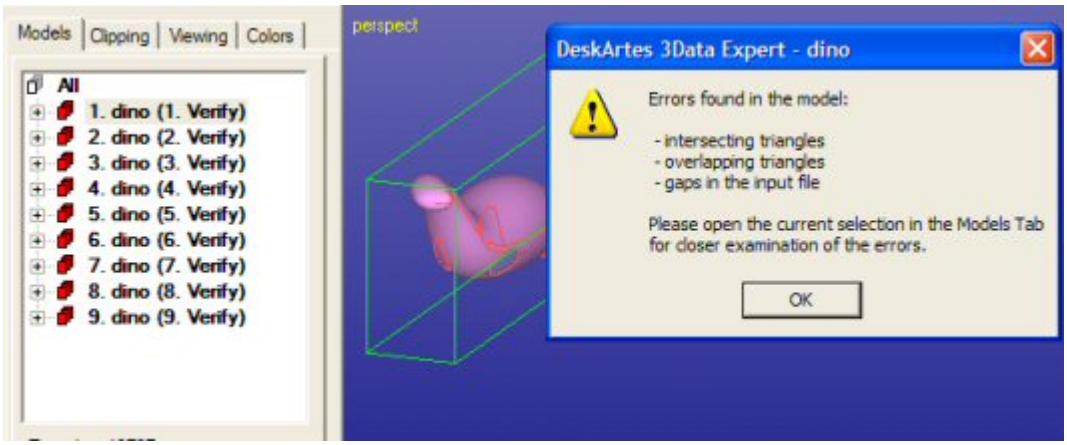
For information on how to fix both difficult STL and VRML models, please see the in the *Expert Series 7.1: STL and VRML fixing* document. The document covers the fixing of several difficult models for 3D printing and 3D color printing.

Verifying the STL model

Start this lesson by **reading in the STL model *Tutorial\GeomFiles\dino.stl*.** You should see the following figure on the display area and the Model Tree should have the following data:



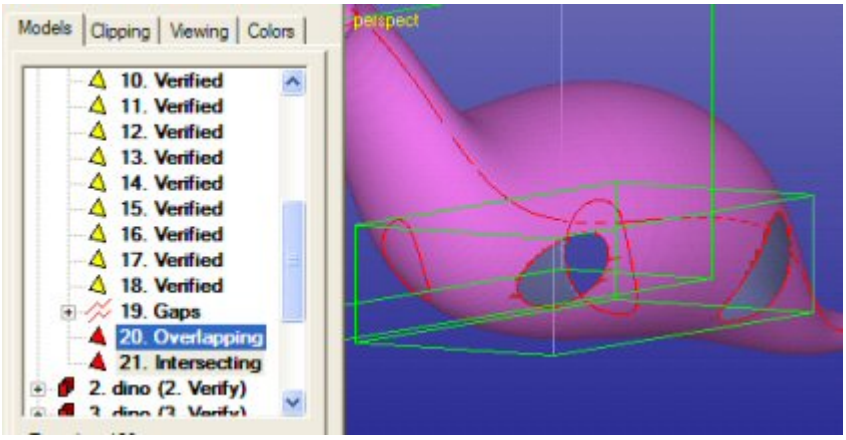
Verify the model with the *Solid/Verify* command . The result has many gaps, intersecting and overlapping triangles:



The dino part is also split into several components which together will make the full part. We will soon show how to repair the separate components into one connected part. To investigate the errors, **Display the error triangles by *erasing* the screen.** Now the error triangles in the selected component are still left visible on the screen if the respective error triangle display is set *on* through the Display Toolbar (icons inside the green circle):

Try selecting different components and using different shading modes (wireframe or shaded) to find the error triangles.

You can also use the *Solid/Separate Errors* menu commands to create disjoint triangle surface elements from the error triangles. Try ***Solid/Separate Errors/All Error Triangles*** with the *Group errors* setting *on* on the 1. dino (1. Verify) component:



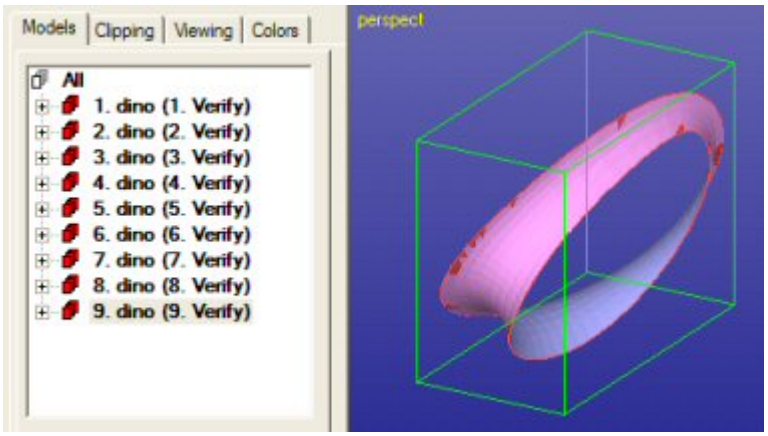
This will allow you to separate, select and display the error triangles as any other triangle surface in the model.

The gap curves are clearly shown if the *Show curves through surface* setting is *on* in the Viewing Tab.

Repairing the STL model

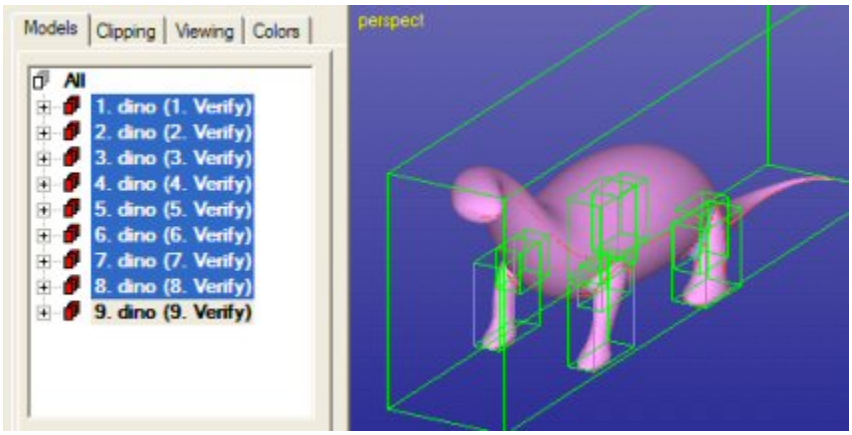
Thin gaps between the surfaces in a triangulation is normally caused by erroneous surface triangulation for Rapid Prototyping. The surfaces are triangulated separately without connecting the faces at the trim curves. Also, larger gaps may be due to missing surfaces in the original model. DeskArtes 3Data Expert can repair these STL problems with ease.

The *dino* model components are sorted by the surface area in the Model Tree after the *Solid/Verify* command. This helps us to delete minor obsolete components from the model before further operations just by checking the last small components in the Model Tree. **Select the last component *9. dino (9.Verify)* and Fit it** on the screen:

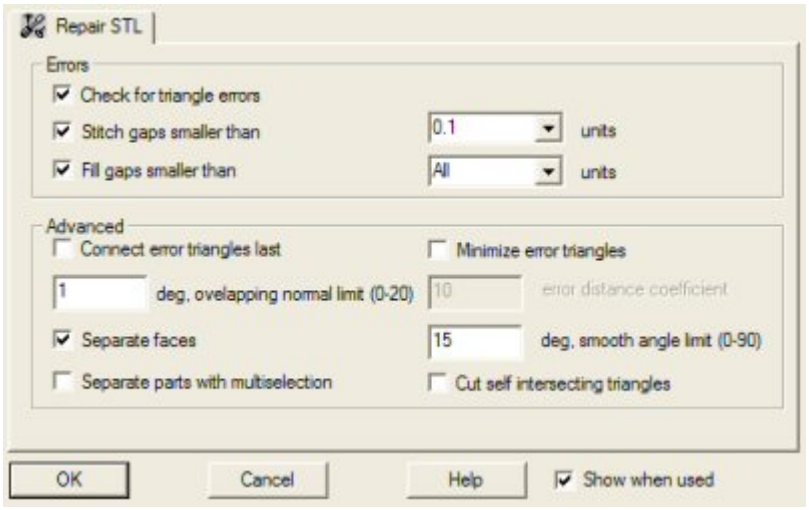


We can immediately see that this is an important components and must not be deleted. The components before the 9th are even bigger and thus must remain in the model.

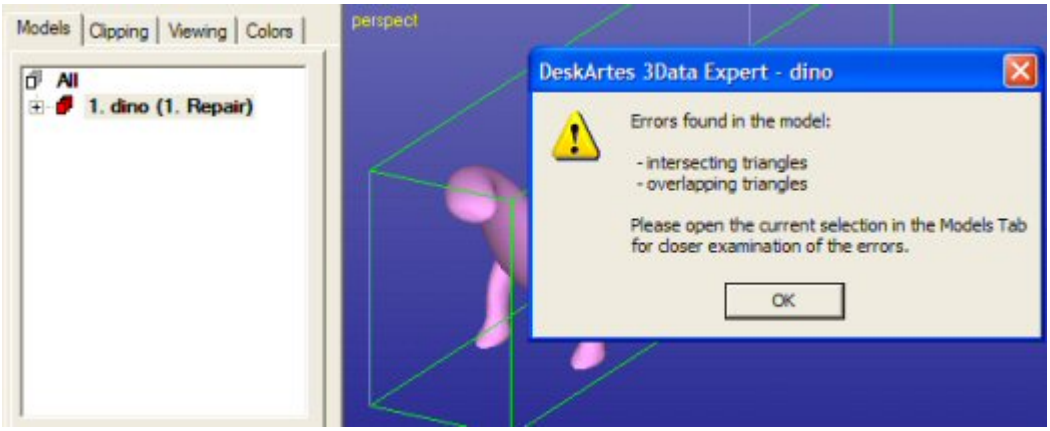
The next step is to combine the separate *dino* components into one connected part. **Use multiselection on the Model Tree** to select all components.



Then **give the *Solid/Repair* command** ( ) to stitch the components into one part and to fill the missing soles of the *dino.* Use the parameters below:

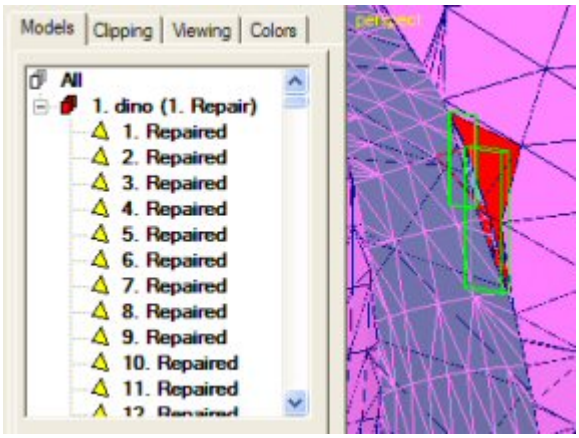


We will *Check for triangle errors, Stitch gaps* to connect separate components and *Fill gaps* to close the soles of the *dino*. After a while the software reports some intersecting and overlapping errors and shows the result below. Notice that all gaps are filled and we only have one *dino* component left:



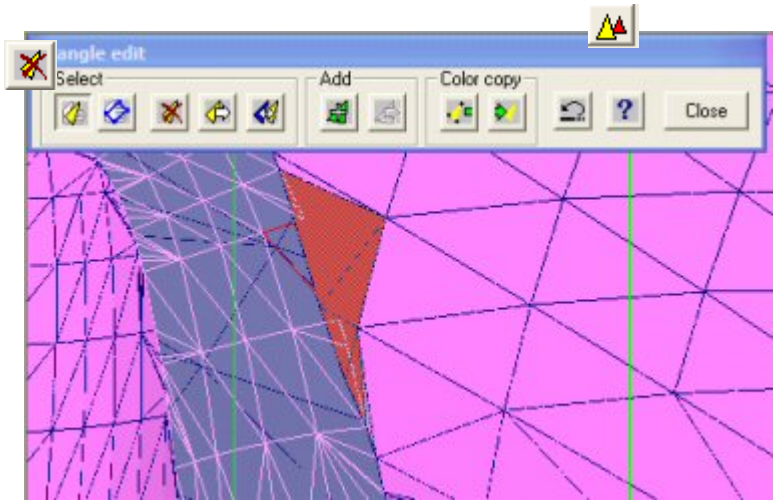
Advanced parameters are needed when there are serious errors in the model. You can learn how to use these parameters by going through the *Expert Series 7.1: STL and VRML Repair* document.

Now find the intersecting triangles in the model and display them on the screen. It is rather easy to locate the larger error triangle area at the join of the left back foot and the body. You can either look for it visually (*Erase screen* + display of error triangles) or use the *Solid/Separate Error Areas* command to find the explicitly in the Model Tree. Anyhow, the investigation shows that we have a thin overlapping surfaces at the area. Below you can see the area with two overlapping surfaces already selected graphically from the screen (rubberband selection with the left mouse in the selection mode) and also the surface in front shown in wireframe mode for better visibility:



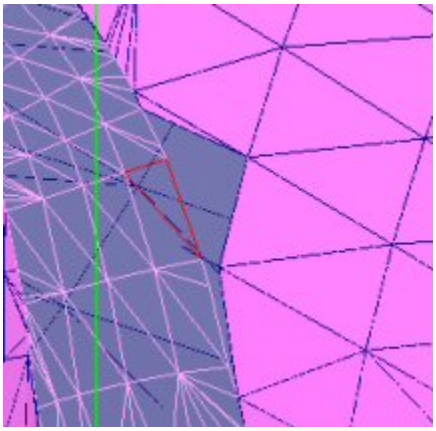
We must delete the selected surfaces to avoid them causing errors in the next runs to *Solid/Repair* command. We also should remove some of the intersecting red triangles to help the automatic repair to generate correct result on

the next run. Use the Solid/Edit Triangles command ( ) to select ( ) and remove ( ) the triangles show grayed below:

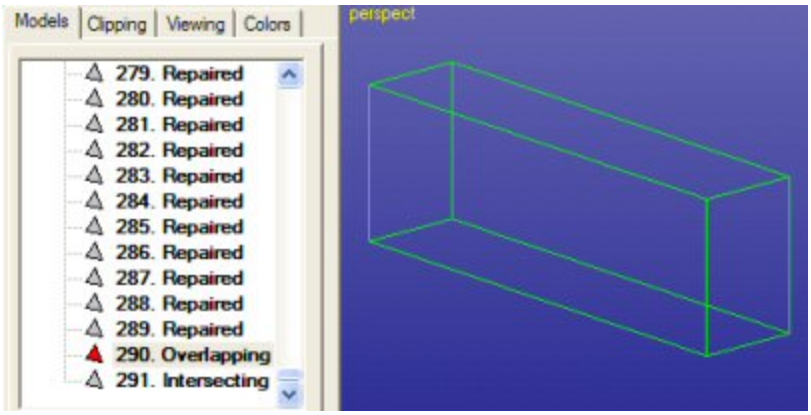


You should end up with a gap similar to the image on the next page. The automatic gap filling can close these gaps and generate correct triangulation on this error area on the next run to the *Solid/Repair* command.

Sometimes you may need to add triangles to simplify the gap curves in the model before the final automatic repair. Adding triangles ( ) is also done through the *Add* commands in the Triangle Edit Toolbar.



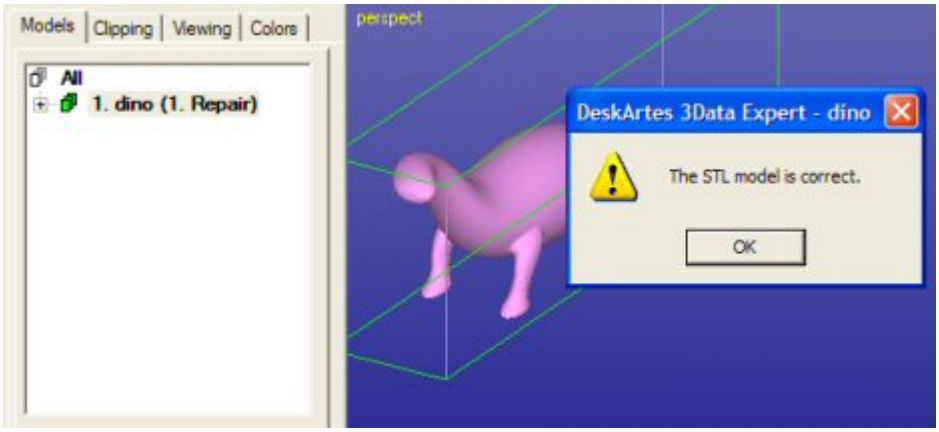
To find the rest of the errors we need to **use the *Solid/Separate Errors/All Error Triangles*** with *Group errors* setting *on*. After the command we can display the *overlapping* triangles in the model:



The overlapping triangle errors are so small that it is difficult to see them even when separated into disjoint surface. In normal operation for Rapid Prototyping we would leave these errors in the model but we will remove them for completeness shake for this exercise. By investigating at the corners of the bounding box we can find that there is a minor error area on the right back foot – body connection area. In the left hand image below **graphical selection is used to select minor surfaces** at the area of the overlapping triangles. We **can delete these *Triangle* surfaces** and let the automatic repair to stitch the gap correctly in the next run:

Note: the Selection Window on the right hand image above can be used to show the selected objects and to display and hide them from the screen. When you press the  icon the currently selected elements are stored and shown in the list. This window is useful when handling models with complicated errors.

Similar minor errors can be found at the tip of the *dino* nose. The thin triangle surface can be deleted. The run the Solid/Repair to generate fully correct STL model:



The intersection checking with the 3Data Expert is very strict, it tries to report all errors in the model. This is sometimes important for simulation software systems as well as some very accurate Rapid Prototyping systems. Anyhow, with most 3D Printers the intersecting and overlapping errors above would not have caused any problems.

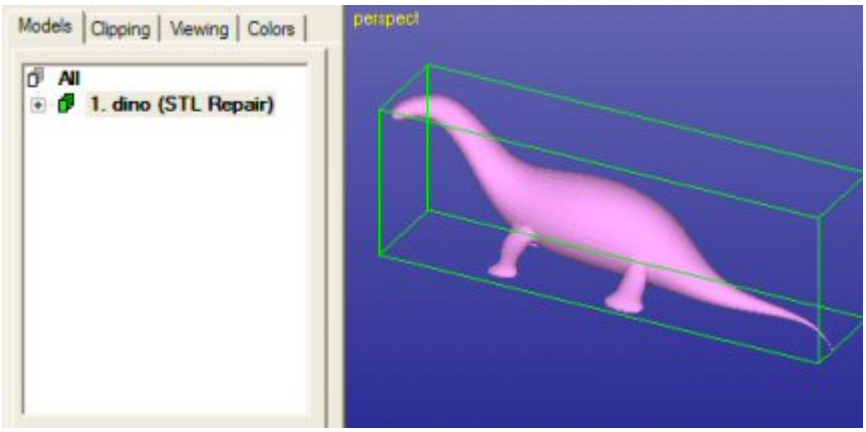
Lesson 4 - Splitting and Combining STL

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

This lesson requires the 3Data Expert Base licenses.

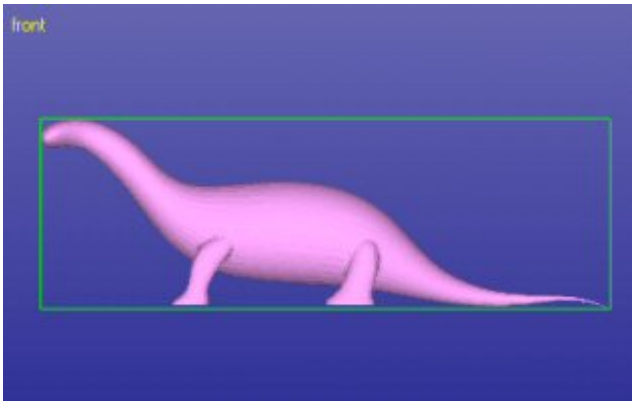
Splitting the *dino*

Start by **inputting the DeskArtes geometry file *Tutorial/GeomFiles/DinoOk.da***. The file includes a correct triangulation for the familiar dino-model.

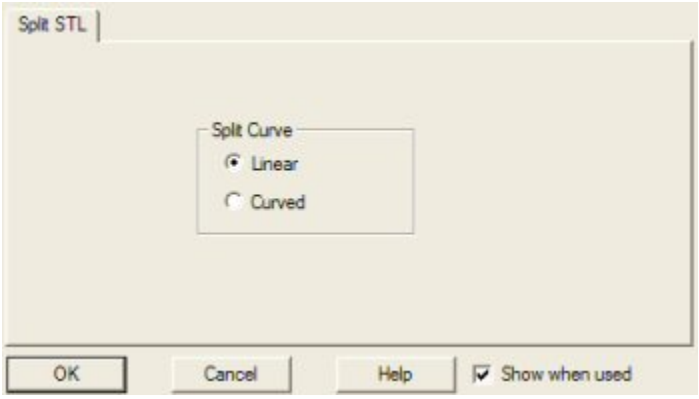


Then **select the node *1. dino (STL repair)*** by clicking it with the left mouse in the Model Tree.

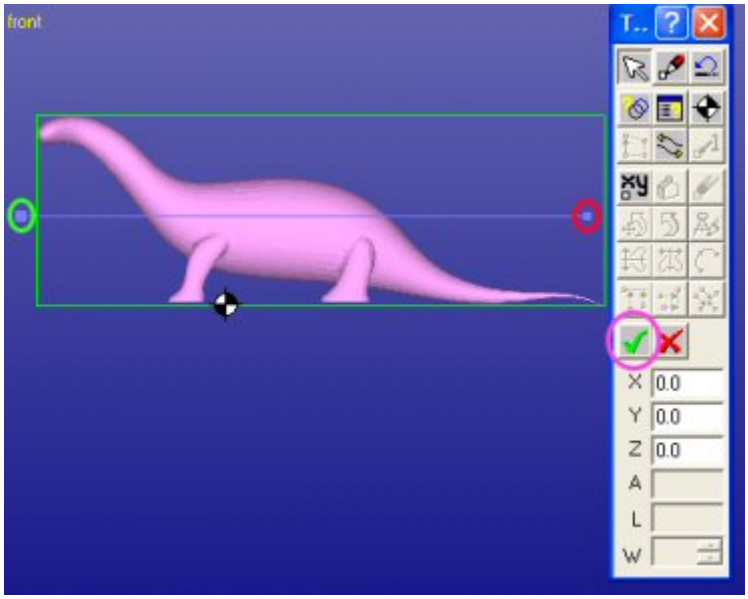
Before the splitting we must take a correct view to the model because the splitting plane is generated by extruding a user defined split polygon away from the viewer. With Rapid Prototyping the correct way normally is to split the model perpendicular to the z-axis to make the part lower. Thus, **take a view from the x-axis direction with *View/Move Viewpoint/Ortho/+X*** (+ left mouse).



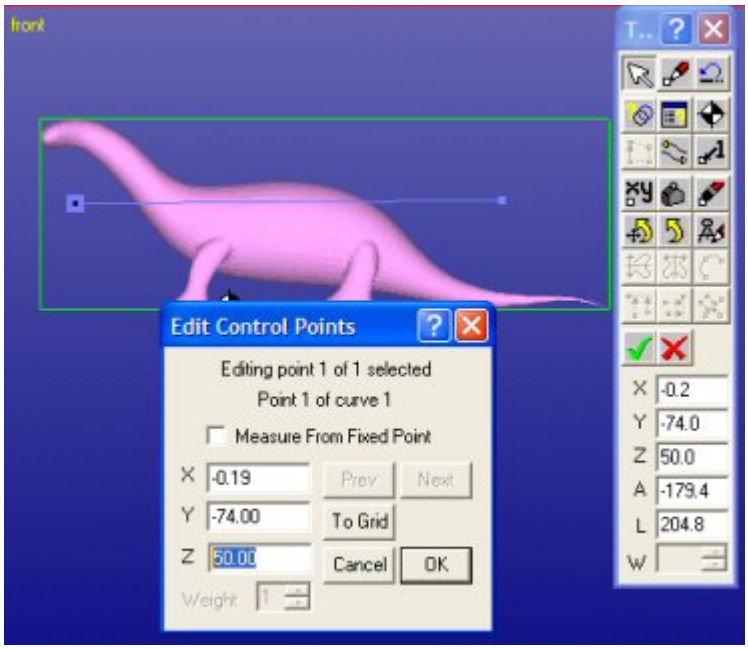
The splitting is started with command *Modify/Split.* The system asks you if you wish to use a *linear* or *curved* split line, select *linear*. After that the system goes into the curve edit mode where you can define the shape of the splitting surface.



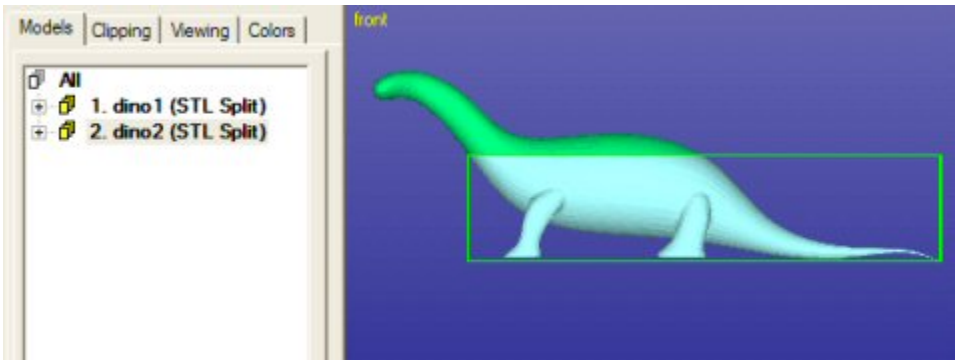
Now you can draw the shape for the splitting surface. You can see a crosshair icon on the graphics area and the Curve Edit Toolbar. You can **start inputting the polygon points by clicking the *left mouse* on the graphics area.** Input two points, both clearly outside of the *dino*-model, as shown in the figure below. For example, first input the point inside the green circle and the the point inside the red circle. **The input mode is finished by pressing the *right mouse* on the graphics area**.



Now you are in the *edit mode* indicated by the arrow icon when you can select points and move them around. Try clicking on the blue points and *drag* them with the left mouse. Try *rubberband selection* with the *left mouse* and then move both points the same time in z direction with the left mouse. You can also delete selected points with  . You can go back to the input mode (or draw mode) by click  the pen icon and start adding points to the polygon. Selected points can be given absolute values with the Edit Control Point dialog, available by pressing the  icon (Edit points). Give z value 50.0 to both points:



Now you can perform the actual splitting by pressing the *ok*  icon (pink circle on the first image on the previous page) in the Curve Edit Toolbar. Soon you should have the result visible:



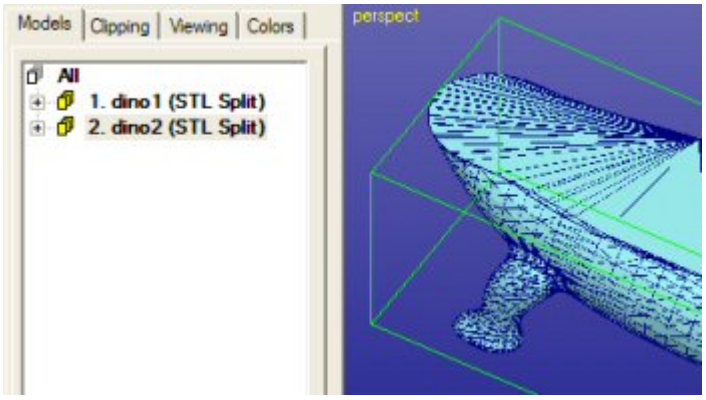
We can see the nicely split *dino*-model. The Model Tree shows us two separate nodes, one for the upper part of the body (*1. dino1*) and one for the lower part of the body (*2. dino2*). Although the ordinal number of the nodes identify the two parts already, numbers are also added to the end of the name dino to distinguish the two nodes. This is done because the ordinal number can change if the node is copied and pasted to somewhere else in the Model Tree.

Creating the pins

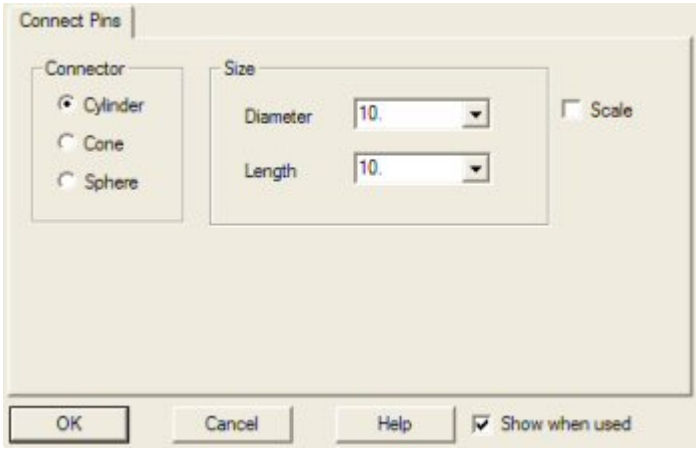
The separate parts of the model could now be positioned and stored into .stl files for manufacturing. Anyhow, the precise joining of the ready parts would be difficult or impossible. To be able to position the parts accurately we will add pins in the models to connect them together after finishing the build.

First we will create the necessary primitives which will connect the two parts together. First we will get a nice view to point the pins on the screen.

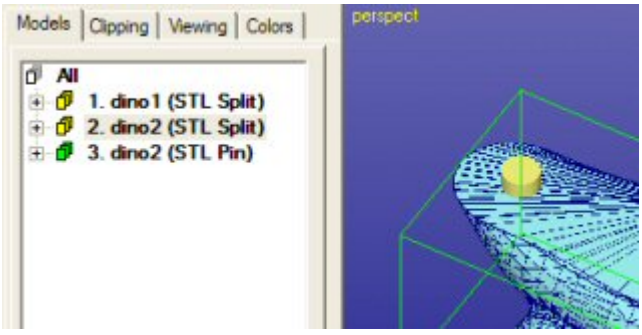
Rotate the model, erase the screen and display the lower part of the dino, node *2. dino2 (STL Split)*, in shaded+wireframe mode. Zoom in to the front end of the model to be able to accurately position the primitive:



The pins are now **created with the *Prototyping/Connect/Create Pin* command**. After the command is given, it is possible to give the *shape* and the *size* for the pin to be created. The S*cale* button allows you to scale the size of the pin graphically after the positioning on the screen.

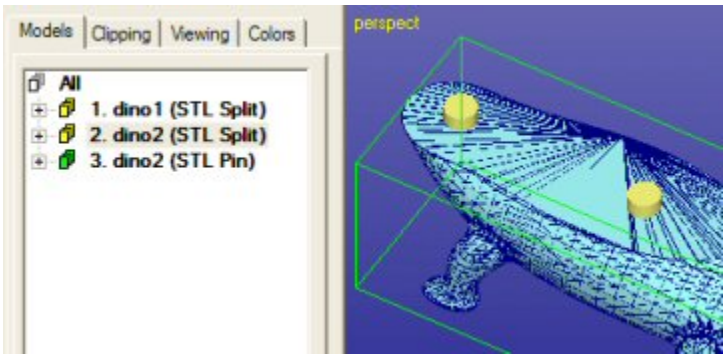


Press *OK* to generate the first pin. Now you can show the position of the pin on the drawing area. Click the left mouse on the wanted place on the *dino2* part. The result should look as below:



You have created your first pin. A new component *3. dino2 (STL Pin)* is created after the currently selected model to store the pin objects.

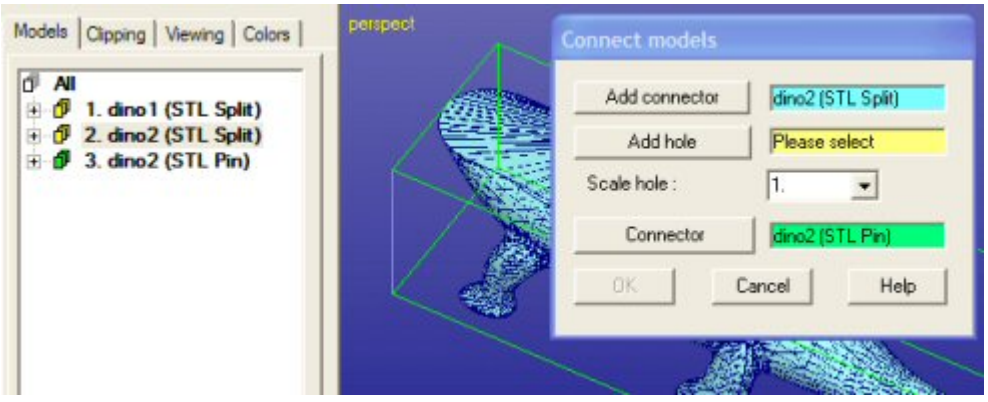
To be able to position the parts correctly after the manufacturing you will need two pins. Create the second pin the similar manner as the first was created. The result should look more or less the same as seen below.



Note: you must keep the *dino2 (STL Split)* component selected during the pin creation command. This ensures that both pins will be stored in the same *dino2 (STL Pin)* component.

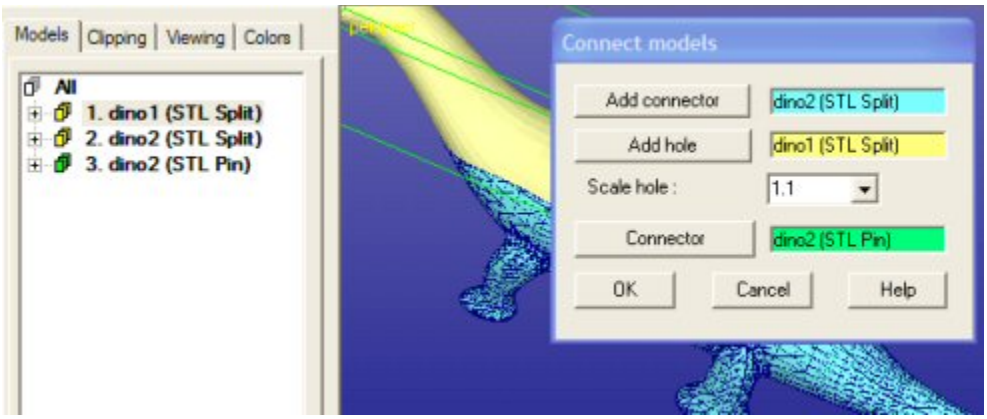
Combining the parts

To add the newly created pins into the *dino2* and to make corresponding holes into the *dino1* component we use the *Prototyping/Connect/Apply Pins* command. **Select the *dino2* component** from the Model Tree and **give the *Modify/Connect* command**. This brings up the Connect-dialog box as seen in the image next page.



The currently selected *dino2* node is automatically set as the part receiving the pins and the color is changed to blue, as indicated by the background of the *Add connector* field. The *dino2 (STL Pin)* node is set as the Connector object automatically.

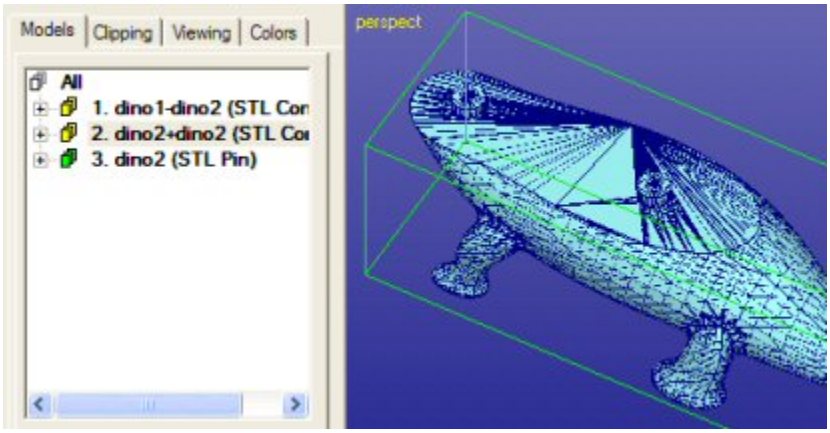
Now you need to specify the part which will receive the holes. **Click on the *dino1* node in the Model tree.** See how this becomes the *Add hole* object and is also shown on the screen with the yellow color:



You can easily change the *Add connector* and *Add hole* -nodes. Just select the wanted object from the model Tree or from the drawing area with the left mouse and then press either the *Add connector* or *Add hole* button. The selected object becomes the wanted object and the not selected object becomes the other object.

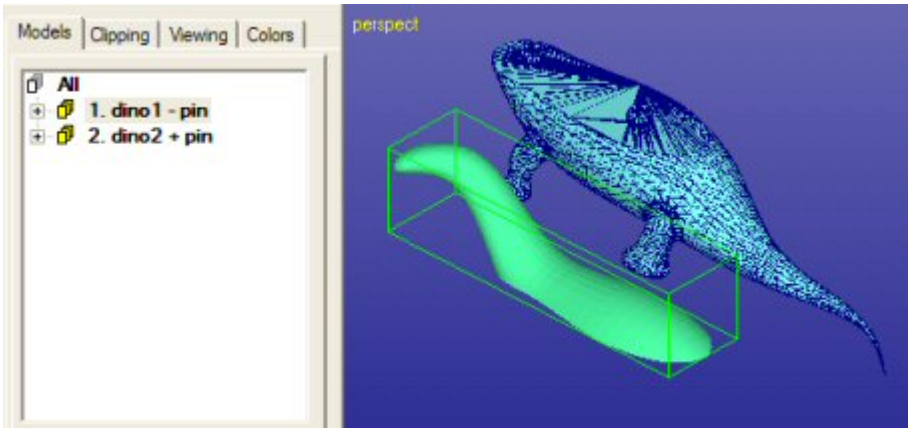
For the two manufactured parts to fit together easily, it is important to scale the hole a bit larger than the pin. **Type *Scale hole value 1.1* into the given field,** as seen above.

Then press *OK* to start the command. After a while you can see the result, as show on the image next page.



Now the name of the nodes have changed to *dino1-dino2* and *dino2+dino2.* It may be a good idea to **change the names** to something more descriptive, like *dino1-pin* and *dino2+pin*. You can now **remove the *dino2 (STL Pin)* node**.

You still have to position the parts for the manufacturing. Numerical positioning is explained in the *Lesson 2, Positioning the part*. You can use also use the Interactive Transformation commands available in the Transform menu to position the parts approximately. As an exercise try to get a result which looks like the figure below with a few *Transform/Move/Interactive* commands. Use first *view from x* direction to move the *dino1-pin* bottom to the same level as the *dino2+pin* bottom. Then *view from z* direction and move the *dino1* besides the *dino2* componnent.



You can find the final result in the file *SplitDinoOk.da.*

This concludes the model splitting and combining lesson.

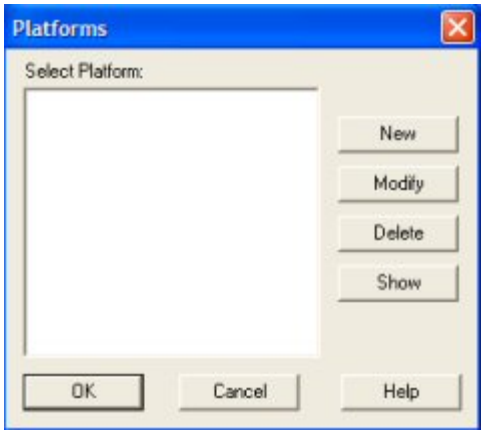
Lesson 5 - Using the RP-module

This lesson shows you how to use the RP-module commands for setting up runs for 3D printers and Rapid Prototyping machines. The RP-module commands are accessed through the *Prototyping* menu.

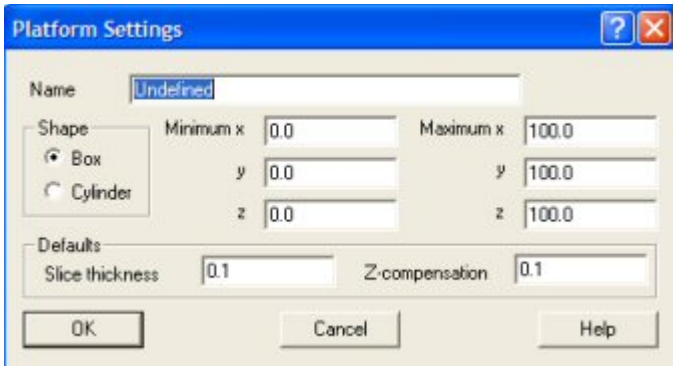
This lesson requires the Expert Series license For 3Data Expert Base.

Defining Platforms

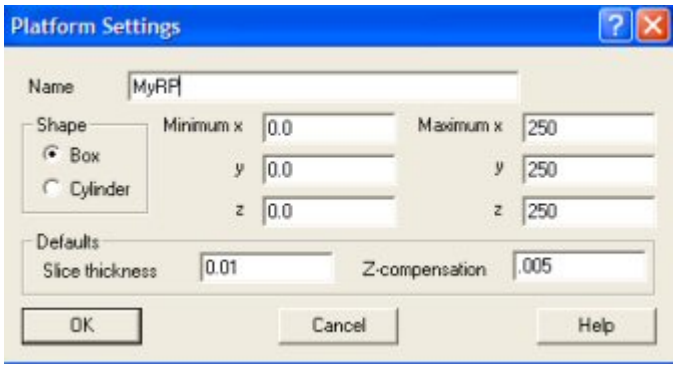
The DeskArtes Expert Series has a user friendly interface for defining and manipulating the available RP platforms. Restart the 3Data Expert software and **give the command *Prototyping/Platforms*.** You will see the following dialog:



This dialog includes the commands to manipulate the set of available platforms. Let's make our first RP machine workspace **by pressing *New* button**. The *Platform Settings* dialog will appear.

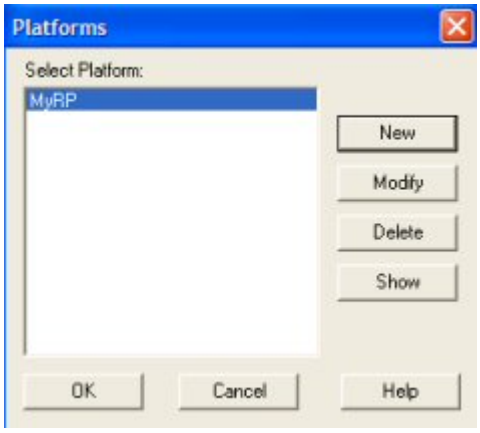


The *shape*, *size*, *slice thickness* and *z-compensation* values you will find in the machine specifications. Here we will use an imaginary *MyRP* machine as an example. Please fill in the values as in the figure below:



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 *Prototyping/Slice* command automatically. The *Z-compensation* value is used with the *Prototyping/Z-compensation* command automatically.

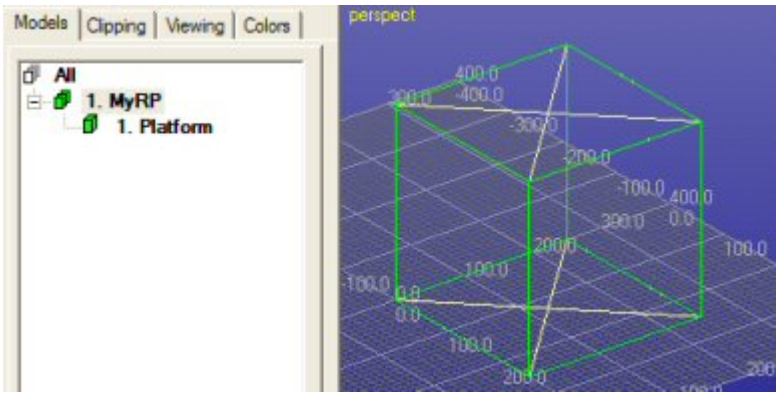
Now we are ready and you can **press *OK* to save the platform** and to close the dialog. Now we have our first platform definition in the *Select Platform* list.



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.

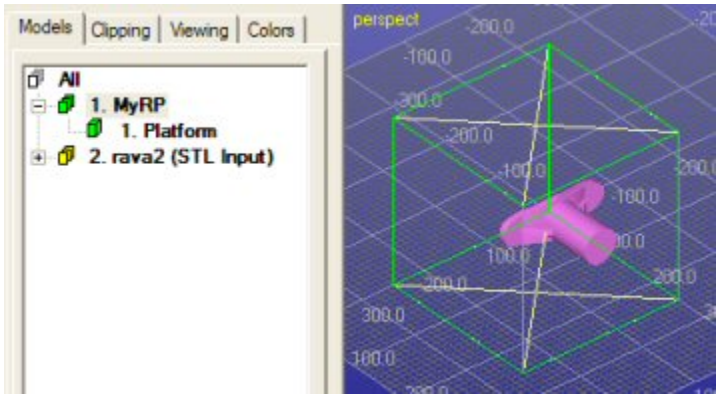
To be able to see the platform on the graphics area, **press the *Show* button.** This generates an object with the given dimensions to the Model Tree and displays it. Now you can use this object to check the position of your parts in the vat visually and use it to guide the positioning of the parts in the work space. **Press *Cancel* to close the Platform dialog**. You should see the model tree and drawing area as seen below. If you do not have the grid on, press  icon.



Moving parts in the platform

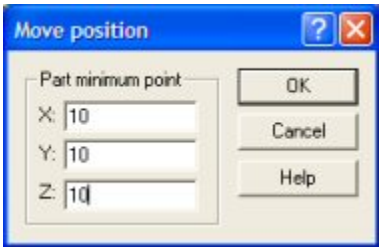
Now we will see how the *Prototyping*/*Move to platform* commands position the part into the used platform. We have selected the *MyRP* platform as the current machine definition earlier. The move platform commands will now use the minimum/maximum x,y,z values of this platform when computing the default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

Let's first **input the model *Tutorial/GeomFiles/rava2.stl.*** Display also the MyRP platform.

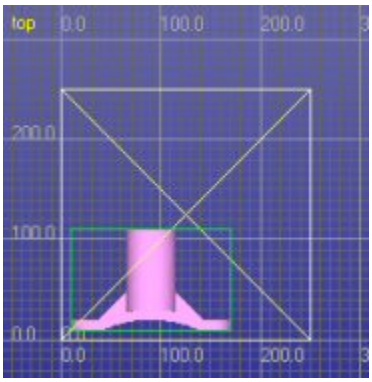


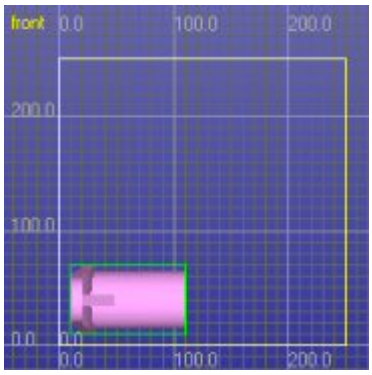
We can see that the part is not inside the build area. **First select the part *rava2* from the Model Tree** and the **move the part into the area with the command *Prototyping*/*Move to Platform/Move positive***. The command suggest the

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*.**

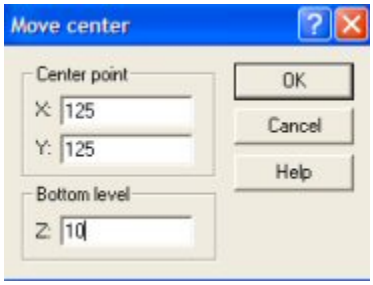


This means that the minimum corner point of the part bounding box is set to the given values. You can check this by taking a view from the z and x directions:

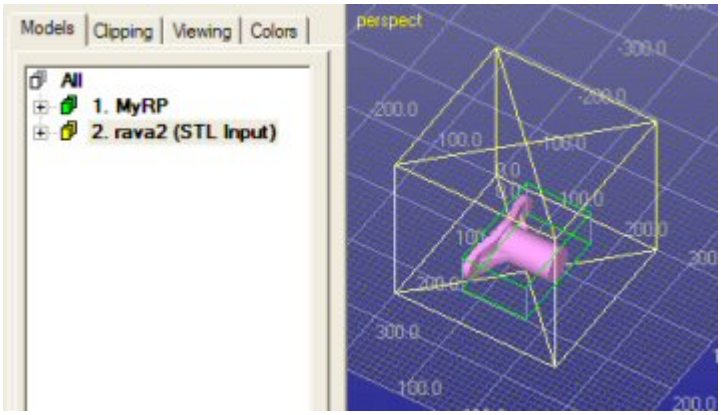




Then give *Prototyping/Move to Platform/Move center* command. *Move Center* computes the center point of the selected platform and uses this as the default for x and y values. The xy-center point of the object is move into this point. You still have to give the z distance from the bottom of the vat (to allow for supports, for example). **Fill in the level value below and press *OK.***



You should have the result show on the next page:

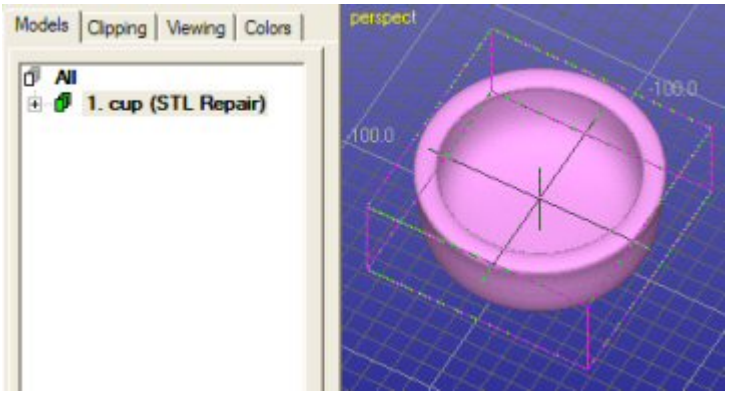


The *Move to Platform/Move Level* command only moves the part in z-axis direction.

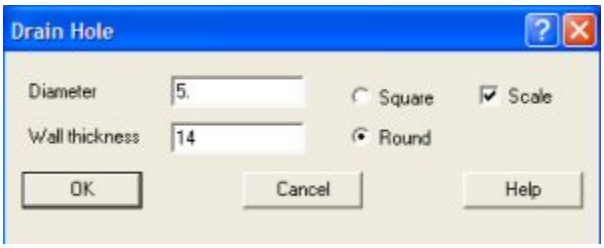
Making Drain Holes

Drain holes are needed when building Stereo lithography parts with large trapped volumes. The *Prototyping/Drain Holes* command gives an easy way to make holes of desired size in the STL models.

Start this exercise by emptying the 3Data Expert work space with *File/New* and then **input the model *Tutorial/GeomFiles/Cup.da***.

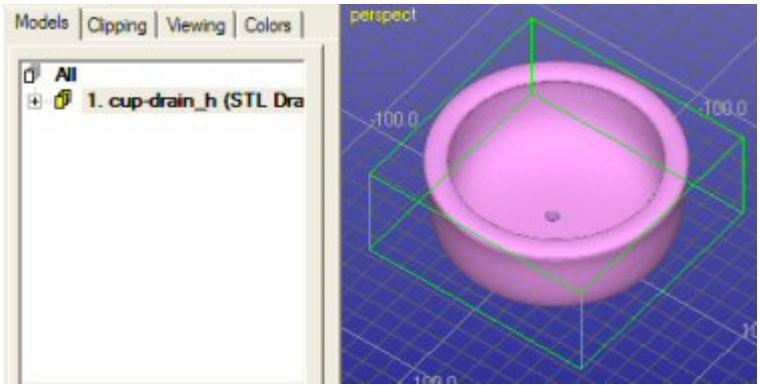


This is a simple and large cup like model, potentially needing a drain hole. **Select the *1. cup (STL Input)* part and start the *Drain Holes* command**. The *Drain Hole* dialog with default values appears. **Modify the values to**



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 7 mm. Double, *14* mm, is used to make sure that the hole will go trough the wall.

Toggling the *Scale* on allows us to scale the hole bigger or smaller than the given *5* mm in the xy-plane after we have positioned it graphically. **Press *OK* to start the computation**. First you will show the place of the hole center with the right mouse using the cross hair pointer. After that the system allows you to scale the cylinder shown on the scree with the left mouse again. When releasing the left mouse after the scaling the drain hole is generated:



The Drain Hole is generated parallel to the surface normal at the given drain hole location. So, Drain Holes can also be generated on vertical or diagonal walls.

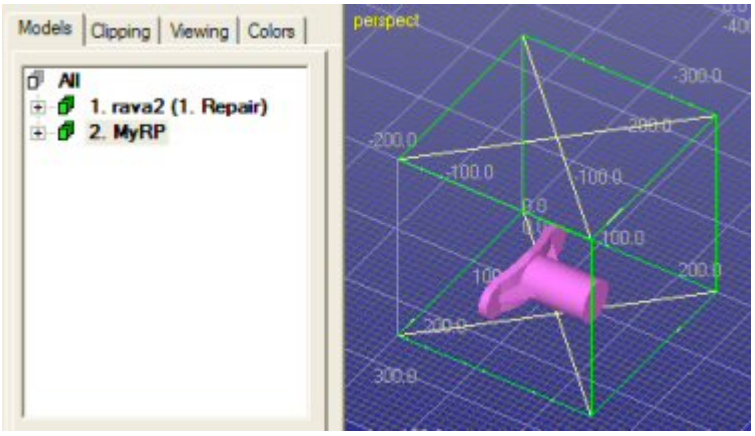
Generating SLA supports

The 3Data Expert RP-module has two main support generation functions. The *Prototyping/Supports/Hatch Support* is used to generate hatch support structures for Stereolitography (liquid) based systems. The *Prototyping/Supports/Volume Support* is used to generate volume supports for sand based systems.

Here we will shortly describe the process of generating supports for liquid based systems. You should also read carefully the Online Help pages for the *Hatch*

support command. Those pages include a detailed description for the different support parameters used with hatch supports.

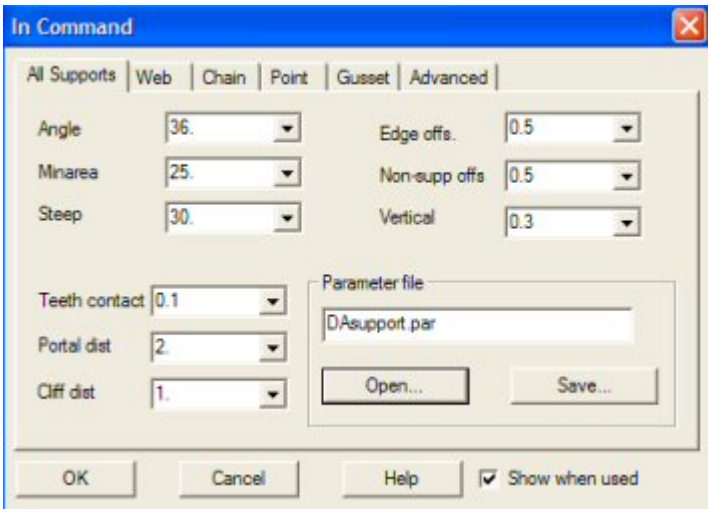
Initialize your system with *File/New* and **input the model *Tutorial/GeomFiles/rava2pos.da*.** It includes the *rava2* model positioned correctly for support generation. Also, **select the *MyRP* platform with the *Prototyping*/*Platforms* command** (generated earlier in this lesson).



Select the model *rava2* from the Model Tree. The part where the supports are generated must be selected before the command. **The support generation is started with the command *Prototyping/Supports/Hatch Support***. First you will see a warning for orientation, but you can ignore it now. We know that the model is properly oriented, so you can just **press *NO*** to continue with the command.

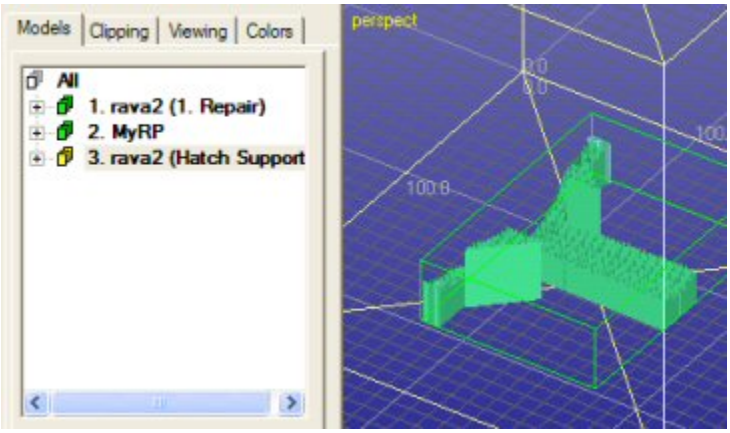
After that the dialog box for Hatch Support parameters appear. This dialog has several tabs to define the properties of different kind of support types exactly. The different types are explained in the Online Help pages for Hatch support generation.

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 DA.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.



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 expertize and machine documentation will tell the correct values for each machine.

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 vendors software, or they can be sliced with the *Prototyping/Slices* command.

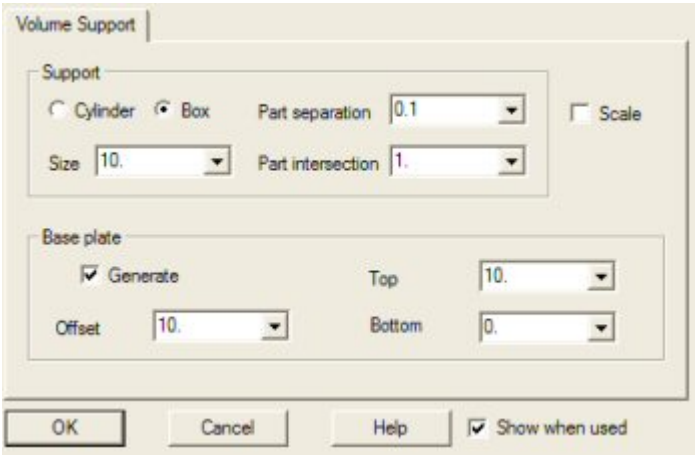
Generating Volume supports

Especially the sand based require support which will hold the green part in position during the careful removing of the extra powder. At the same time the support should not intersect the part but there must be a tiny gap between the

part and the support. The volume supports are normally build using the same strategy as the actual part. With 3Data Expert this kind of supports are generated with the *Prototyping/Supports/Volume Support* command.

Initialize your system with *File/New* command and **input the model *Tutorial/GeomFiles/rava2pos.da***. It includes the rava2 model positioned correctly for support generation. Also, we have selected the MyRP platform with the *Platforms* command, just as in the start of the Hatch Support generation on page 48.

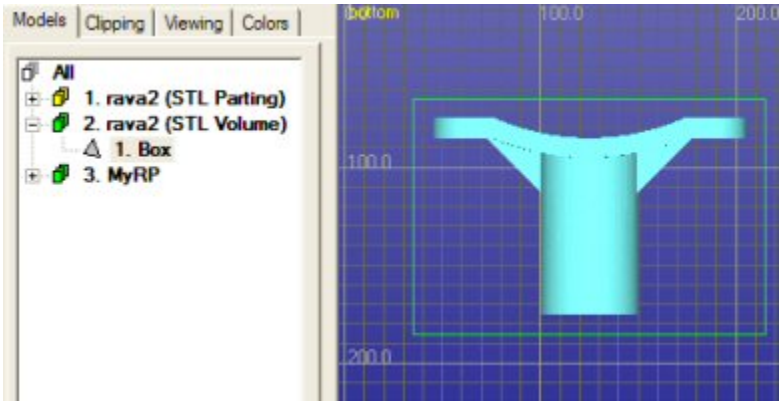
Select the model *rava2* from the Model Tree. The support **generation is started with the command *Prototyping/Supports/Volume Support***. First you will see a warning for orientation, but we know that the model is properly oriented, so you can **just press *NO***. You will see the default Volume support dialog:



This dialog allows you to change the Volume S*upport* parameters. Also, you can select if you want to generate the *Base plate*. *Base plate* is build below the part and the volume support elements to enable easy removal of the whole package from the build volume.

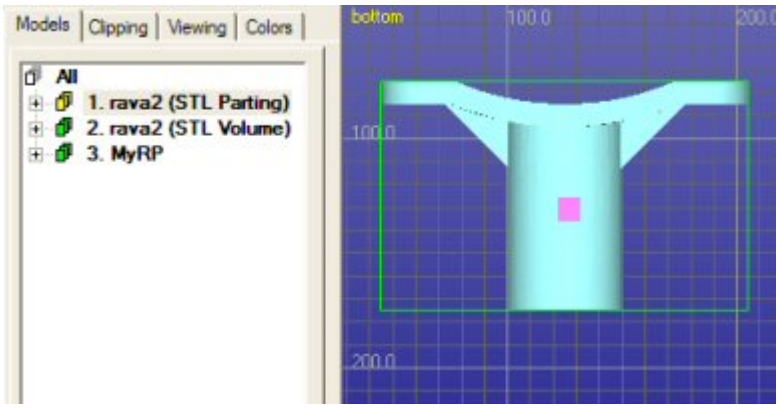
Let's now accept the default values and generate the first volume support element by pressing *OK.*

You are now in the mode where you can point the support location with the left mouse. This is indicated by a cross hair icon when the mouse is on the drawing area. Also, the view is changed to -Z, you are looking the part from the bottom. The volume supports must always be generated below the part and the volume supports must not touch vertical or upfacing areas.



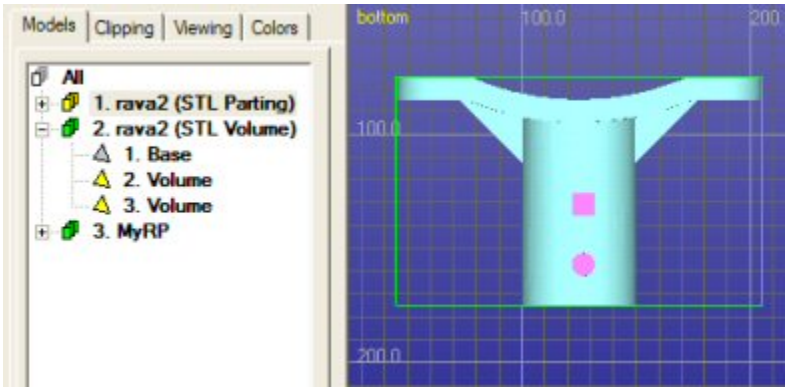
In the Model Tree you can see that the original *rava2* model is split into *downfacing* and *upfacing* surfaces *2. rava2 (STL Parting)*. Of course, at the momen you can only see the *downfacing* areas. If you want to change the view before picking the place for the support, it can be done with the viewing shortcuts.

Show the place for the first support *with the left mouse* in the middle of the part.

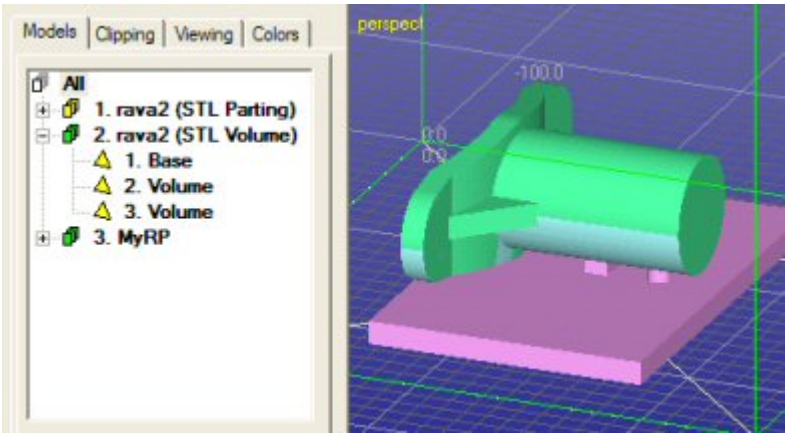


After a while you can now see the first volume support element on the scree. By opening the components in the Model Tree you can see that the volume support element is stored into the *3. rava2 (STL Volume)* element. The base plate is not show automatically because it would obscure the part and the volume element (make sure it is hidden before you continue).

Make another volume support element with the command *Prototyping/Supports/Volume Support*. The new element is automatically stored in the same volume support node as the first one (Make sure you select the *2. rava2 (STL Parting)* node before the giving the command).



Now rotate the view and display the whole result.



Adding some more volume support elements for the balance is left as an exercise for the reader.

The part and the supports can now be outputted for slicing with the machine vendors software, or they can be sliced with the *Prototyping/Slices* command for slice output.

Lesson 6 - Working with Textures

This lesson will explain you how to add new textures to triangle models inputted from STL or VRML file. We will also learn how to separate a STL model into faces for more accurate texture painting.

In the previous lessons we have learned how to repair and split a model for Rapid Prototyping. Now we will start adding textures on triangle models. You can start from a non textured STL model and add any number of textures on it and output is as VRML file to your 3D color printer. Or you can start from a Z Corporation .zpr file or VRML model, repair it for RP, add textures, remove textures or change textures on selected areas.

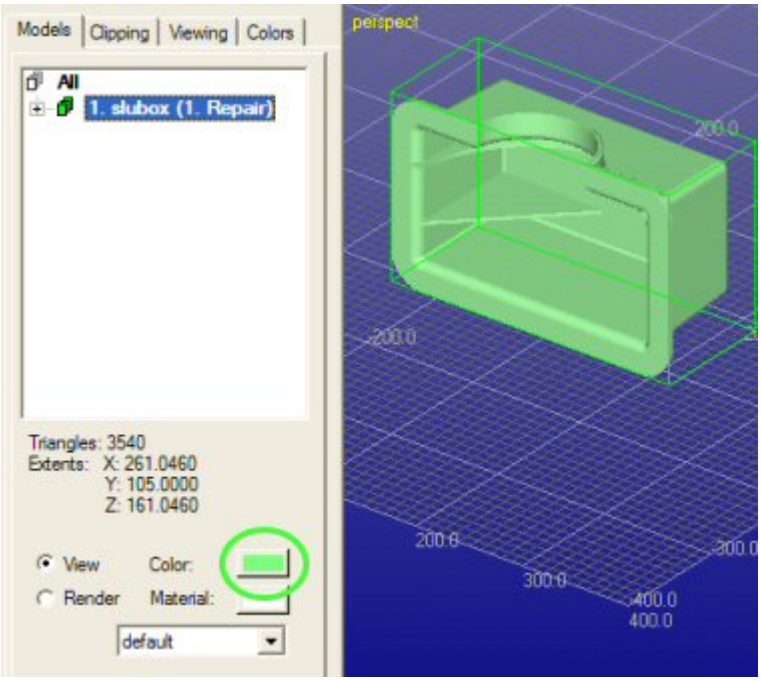
DeskArtes 3Data Expert also enables restoring textures into a repaired VRML or ZPR model. This technique is explained in detail in the *Expert Series 7.1: STL and VRML fixing* document. The document covers the fixing of several difficult models for normal 3D printing and 3D color printing.

Adding a logo

Adding a logo on a STL model to be build with a 3D printer may be the most common operation you will use the *Render/Define STL Texture* command.

We will start by **inputting the model *slubox.stl* from the Tutorial/GeomFiles**. If you are no sure that the model is correct, it is a good practice either to run *Solid/Repair* or *Solid/Verify* to see that the model is correct starting to add textures on it. Let's **run the *Solid/Repair* on the model now with the default parameters**. It is correct.

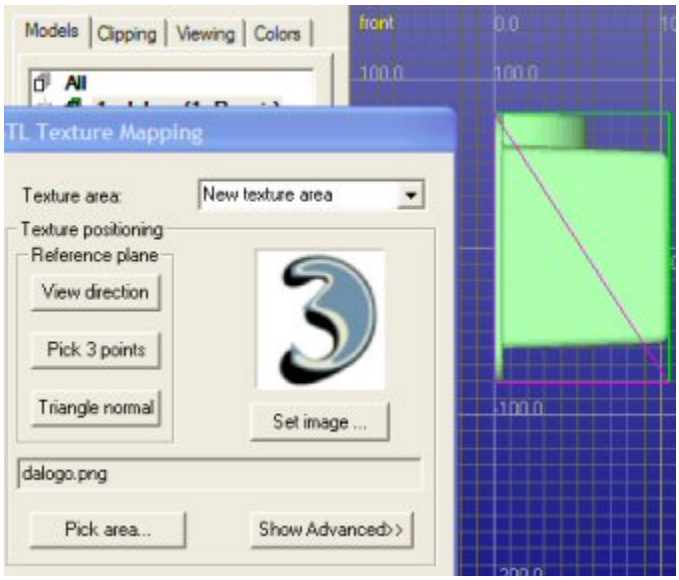
Before we continue to add the logo we can set the basic color for the part. When a STL model is inputted into Expert Series a default pink color is attached to it. **Let's change the color to a shade of green with the *Color* button in the Model Tree window** (in the green circle on the image on the next page):



Now we will add the DeskArtes logo in the other end of the *slubox.stl*. **Give the command *Render/Define STL Texture***. You will see the following dialog:

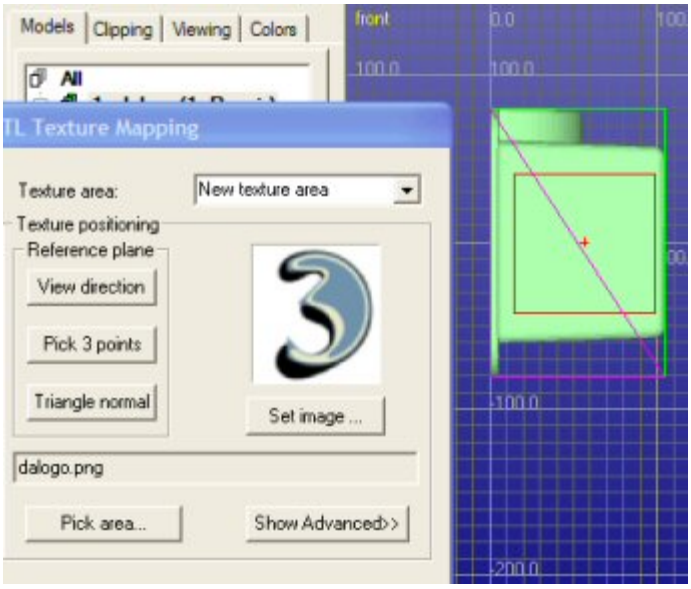
For details on the different buttons please see Define STL Texture command help. Now we will simply select the image, projection direction and effective depth of the texture. **First take a view from the x-direction** and fit the *slubox.stl* in the middle of the screen (pleas see the image on the next page)

First **select the texture projection plane by pressing the *View direction* button**. Now the system uses x-axis direction to project the texture on the STL part. A pink wireframe triangle is shown on the screen to indicate the projection plane. **Then select the texture image by pressing the *Set image* button**. This opens a normal File open window in the Expert Series Textures directory. **Open the file *dalogo.bmp.*** After pressing *Open* you should see the DeskArtes logo in the image window:

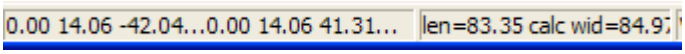


You can also see that the *Pick area...* button has turned from gray to active. Now you are ready to select the area for the texture on the STL part.

Press *Pick area...* button to start defining the texture area on the screen. All command buttons in the dialog turn inactive. **Move the mouse on the screen and *pick a point with the left mouse***. A red cross appears to indicate the starting point of the texture area definition. **Then 10) *move the mouse* on the part** (no buttons pressed) and see how a rubberband box is following the mouse moves on the screen. This is the area where the texture will appear later. Also, the line from the mouse position to the red cross initial position corresponds to the left hand edge of the texture image in the STL Texture dialog. When you are happy with the size of the image **press *with the left mouse to fix the size* and initial position**. You screen should look like this:



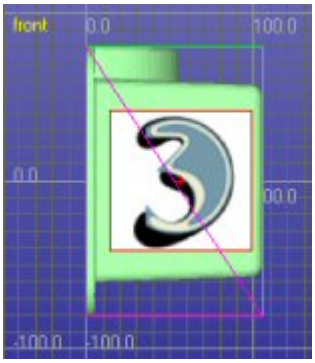
The red box shows the position of the image. If you are not happy with the position, just press the *Pick area...* button again and redefine the area. The start and the end point of the left edge of the texture are and the size of the area can be seen in the status bar below the graphical area:

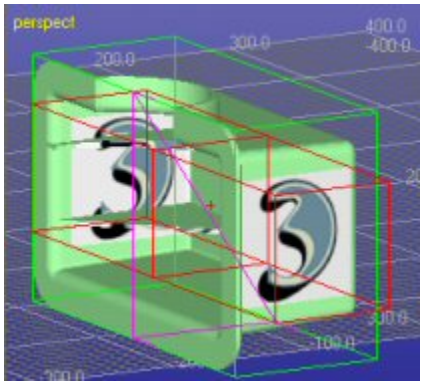


The len-field gives the length of the left edge of the image in 3D. calc wid -field gives the width for the texture area. This is calculated based on the length of the left edge (give with the mouse) and the aspect ratio of the actual texture image.

When you are happy with the position of the texture you must **press *Apply* in the STL Texture Mapping dialog** to apply the image on the STL part. If you are change the position of the texture you can now *press arrow buttons* on your keyboard to move the texture, *Sift + arrow* to scale the texture.

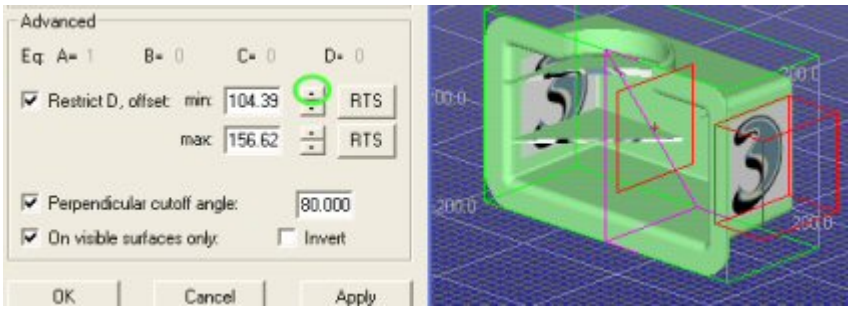
On the left hand image below you can see the textured slupart after the *Apply* operation.



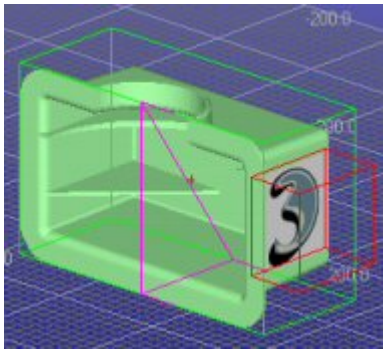


If you change the eyepoint by rotating a little you can see that the effective texture area extends through the full model (by default) on the right hand side image.

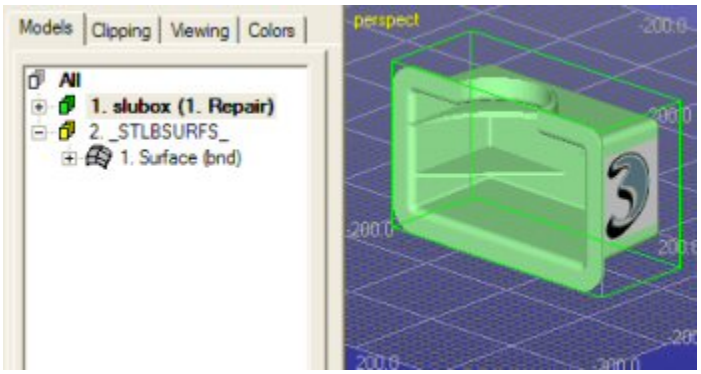
In this example we only want to put the texture on the face closest to us. **Press the Advanced button to set the depth for the texturing effect**. You will see several new parameters to control the behavior of the texture projection, see the image below. Now we will use the circled up arrow to change the minimum coordinate value for the texture effect box. **Press the *up arrow* (green circle) until you have reached the situation as seen on the right hand side of the image below** (the back end of the box should be a bit farther than the front face with the texture). You may need to rotate and zoom into the part as well as press the down arrow to get the correct result.



Press *Apply* again. Now you will see that the texture is only applied on the nearest face and the result is correct. If the result is not correct use the arrow buttons (up/down) to set the texture effect area correctly and press Apply again until the texture is correctly set.



You can now **press *OK* t**o finish defining the texture. You should see the situation below (after opening the Model Tree node):



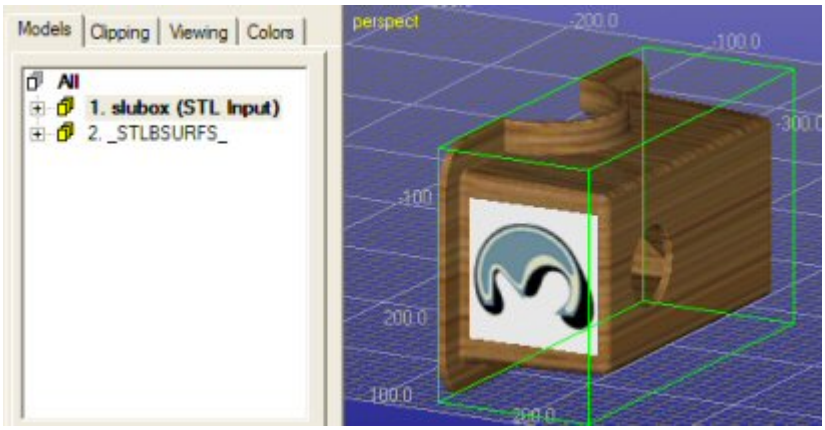
If you open the *slubox* component in the Model Tree each Triangles surface has a *Texture* definition after it. Also, a new node *_STLSURFS_* is generated. This node includes the texture definitions (images, projection direction, texture effect box etc.). *You must not modify or delete this node*, it is updated automatically by the *Define STL Texture* command.

Make sure that the *1. slubox (STL Repaired)* node is selected and **give the File Save command with file type *.zpr*.** The file will be written into the .zpr file according to the current viewing settings, i.e. it will look the same in the Z Print software.

Or you can give command Render/VR Export to generate a VRML file for more general color file output. Use the *Browse* button in the VRML dialog to set the output file name and directory, otherwise use the default settings in the dialog (make sure the *Export texture file* type is *JPEG*). The outputted VRML file, a box with a logo, is now ready for animation or printing with 3D color printer.

You can define any number of textures on a single STL model. You may want to restart the Define STL Texture command and to create a new texture area by selecting the "New texture area" in the dialog dropdown list:

You can set several textures on top of each other. The textures which are added later will obscure the earlier textures. For example, to generate a box with a wooden background texture and a logo you must first set the wooden background texture and then the logo texture, as seen in the image below (*Tutorial/GeomFiles/slubox_tex2.da or slubox_tex2.wrl*):



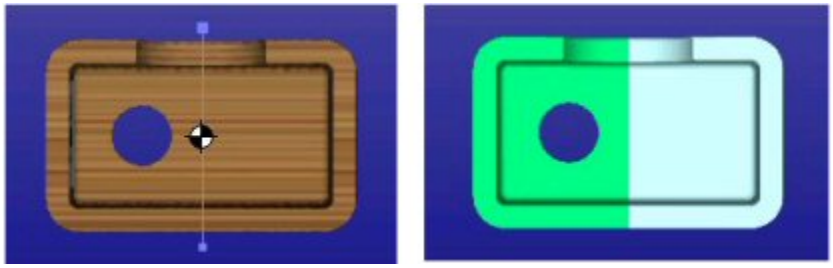
You can also freely add new textures on top of inputted VRML models with textures using the Render/Define STL Texture command as shown above.

Restoring the Texturing

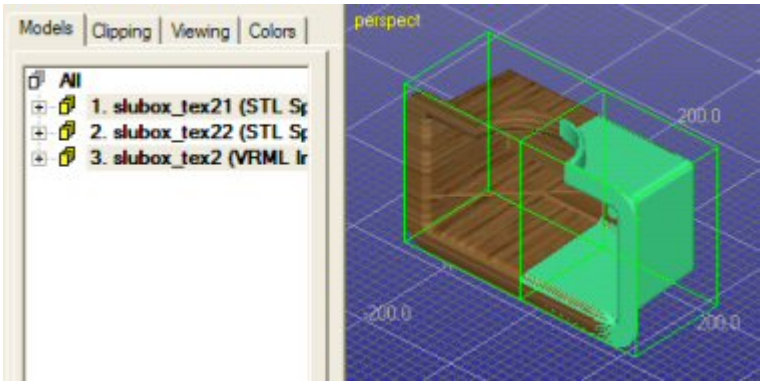
If you are running a *Solid* or *Modify* menu commands on a textured STL part or on an inputted VRML file, the texturing information will disappear. Anyhow, the

textures can be resumed after the operations with *Resume VRML Textures* command. Let's now input a VRML model, split it and then restore the textures.

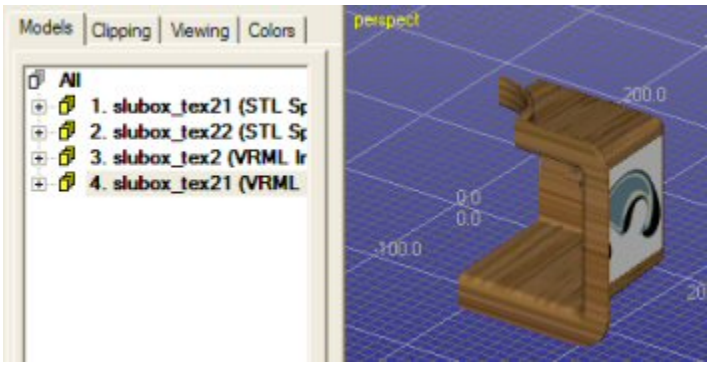
Start by inputting the *slubox_tex2.wrl* in the Tutorial/GeomFiles directory. **Then take a view from the Y direction and split the model using the Modify split command, as done in the Lesson 3: Splitting and Combining STL** . You should see the textured model before the split to turn into two non textured pieces, as seen below:



With real parts to build you may want to add pins between the two parts, but let's now just concentrate on the texture restoring. **Now input the original *slubox_tex2.wrl* again**. Then **use multiple selection to first select *one of the split* parts and then *the newly inputted VRML* model.** You should have the following situation displayed on the screen:



Give the command *Render/Restore VRML Textures*. After a while you should see the following result:



You can repeat the command to the other split part, if you wish. Now the resulting parts have the suffix *'(VRML Restore)'* and can be outputted as .zpr file for 3D color printing.

If you are texturing STL parts inside the software and want to split them after the textures are set, you must remember to make a copy (or save on disk) of the original part before splitting to be able to use the copy as the second selection in the Restore VRML Textures command.

Conclusions

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

There are still a wast number of functionality not covered in this document available in the DeskArtes 3Data Expert software. Please refer to the Online Help and its examples for more information on the available functionality.

The first lesson in this document applies to all Expert Series softwares. Also, the basic repair and splitting operations can be achieved with the Dimension Expert and Spectral Expert softwares, and with the Design Expert software. Anyhow, for full surface triangulation, STL/VRML repair with boolean, reduction, refining and other operations you will need the 3Data Expert software.

To learn more on the use of the STL and VRML handling available in the 3Data Expert software, please have a look at the advanced repair document: *Expert Series 7.1: STL and VRML fixing*. This document as well as the necessary demo files are available for download at www.deskartes.com at the Documents section.

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