The goal of this document is to learn how to start using 3DReshaper.

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### INTRODUCTION

... (omitted for brevity)

### REVERSE ENGINEERING

#### GENERALITIES ABOUT REVERSE ENGINEERING IN 3DReshaper

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INTRODUCTION

Technodigit is pleased to introduce you to 3DReshaper, the 3D Scanner Software.

Your Beginner’s Guide...

This Beginner’s guide will walk you through some typical process using 3DReshaper. All samples used in this guide are present in the directory “My Documents” (by default C:\Users\Public\Documents\3DReshaper 2016 MR1 (x64)). You can also access to the samples through the windows start menu, in the 3DReshaper Application folder.

To go further...

Some practical exercises are available on our website (www.3dreshaper.com), and do not hesitate to refer to the 3DReshaper help files (press F1 in 3DReshaper) to find specific information about all the features. You can also contact your local reseller in order to have a complete training.

3DReshaper typical workflows

Here are the typical 3DReshaper workflows:
Generalities about Reverse Engineering in 3DReshaper

Introduction

The Reshaper polygonal mesh modeling generates models, made up with hundreds or thousands of non-continuous triangles. These 3D meshed models are ready for rapid prototyping, tool path generation, simulation, analysis, etc.

However, a “continuous” model is sometimes required by CAD-CAM software. This process of making a CAD model is also called “reverse-engineering” because you generate a continuous model, also called “exact model” from scattered data sets: mesh and point cloud.

This module allows you to make CAD Surface reconstruction starting from a mesh. CAD Surface means NURBS and BSpline surfaces that are fitted on your original mesh. Finally, these surfaces can be exported into IGES or STEP files and/or used to process inspections.

The process to create surfaces from a 3D mesh is divided in 3 parts:

- First, you have to create a polylines network in order to delimit the different zones having similar curvature properties on your mesh: fillet, planar zone. These lines must lie “on” the mesh.
- Then this network of lines is used to create NURBS/BSpline curves using an automatic tolerance which can be modified after for a specific curve or for all curves.
- Finally we create NURBS/BSpline surfaces using previous BSpline curves. These surfaces are fitted on the mesh. The patches are displayed with different colors, which evaluate the quality of the result.

Figure 1: The three steps in order to create surfaces
Rules to make a good polyline network

Our “Reverse-engineering” process is based on a polyline’s network that you must achieve first. So, before having your CAD objects (curves and surfaces), you must create delimitation with polylines on the mesh. To make a good delimitation you must follow these rules:

- Make borders on zones having same curvature characteristics: lines along a small radius, line along sharp edges, etc.
- Create polylines that intersect so that Reshaper can easily determine the accurate intersection.
- Make contours with 4 sides (as far as possible).
- The lines that are created must lie “on” the mesh; otherwise some surface reconstruction may fail.

Make smooth polylines along curvature discontinuity

The goal of the surface reconstruction is to divide the complete surface of the model into elementary surfaces called “patch” or “face”. The most interesting property of the NURBS / BSpline mathematical definition is that the surface is continuous. “Continuous” means that the shape changes smoothly from a point to another point of the same face.

Some discontinuities may exist in a surface but they are always located on the border between two patches. This never occurs inside one patch. You may have two types of discontinuity:

- Tangency discontinuity: typically this occurs when you have a sharp edge on your model.
- Curvature discontinuity: typically this occurs when you have a fillet on your model.

When you create your polylines network you must follow the discontinuities on the surface as shown on the Figure 2.

When you have a fillet, it is important to consider that there is one curvature discontinuity on each side. This means that you should have 2 curves: one on each side on the fillet like on the right picture and not only one on the top of the fillet like on the left picture. We will see further in paragraph 3.3 that Reshaper has a command called “Extract Feature Line” to make automatically and very easily these 2 lines.

Make polylines intersecting each other

You must have intersecting lines in order to create a real network. From a network of intersecting lines, Reshaper will automatically calculate the intersections and trim irrelevant parts. Every time a valid polyline contour is detected, Reshaper will automatically transform polyline pieces into NURBS curves and fill a surface patch inside the contour.

Figure 2: polylines along curvature discontinuity; lines should be as on the right, they are smooth and correctly positioned on the discontinuities.

Figure 3: On the left, lines are not intersecting -> the patch creation will fail. On the right the patch will be successfully created.
**Make contours with 4 borders**
The mathematical definition of a NURBS surface has 4 and only 4 borders. Then, it is better to make as much as possible rectangular contours with 4 borders when designing patches.

When Reshaper analyzes the borders, several situations may occur:

- If you have 4 sides, this is the ideal situation and this is the reason why the resulting patch will be displayed in green color.
- If you have less than 4 sides, Reshaper will create a “degenerated face”, which means that the mathematical definition will keep 4 borders but some border(s) will have a null length. The resulting patch will also be displayed in green color.
- If you have more than 4 borders, Reshaper will analyze whether some borders are made of several continuous pieces of polylines.
  - If Reshaper can successfully merge continuous borders and can find 4 sides at the end of the process, a 4 side face will be created. However, the resulting patch will be displayed in orange color because a bad accuracy may occur on the side coming from the merge of different pieces.
  - If no merge is possible Reshaper will create a bigger patch with 4 sides and limit the valid zone of the face on the contour. A face like this is called “trimmed face” or “restricted patch”. This situation is the worst case. This is why the resulting face is displayed in red color. When you get such a face, you can access to the base face using the command “Explode” into CAD sub-objects.

**Generate patch on a mesh**
The main command to create surface patches is CAD\Generate Patch. This command is very powerful because it drives you directly from a set of polylines to the set of patch in different steps that are explained further.

The philosophy of this command is that at each step you can decide:

- To continue automatically to the next step. This is very interesting if you want to make rapid surfacing because the execution of this command takes only a few minutes.
- To stop execution at a certain step to make manual control and (or) modifications. This is very interesting if you want to have an optimized result.

You can launch the command with:
- A list of polylines + a mesh, or a Set of polyline and a mesh. Then the step 1 is displayed.
- A list of BSpline and a mesh. Then the step 2 is launched. The surface computation starts directly.

**Exercise: Reverse engineering on a screw**
Open the file “ReverseEng.rsh”. This file contains a mesh “Screw” and some polylines created with the following commands:

- Freehand Sections
- Projected Contour
- Extract Contour / Hole
- Radial Sections
- Extract Feature Line
Computing intersection

Select the mesh and all the polylines and then go to **CAD\Generate Patch**. It will compute first all intersections between lines. An information message displays the number of segments detected and the module sets a random color for each of them. Each segment is a part of initial lines.

After that you have 2 choices:

- Go to the **next step** creation. We will take this option to compute BSpline **curves**.
- You may also exit the command to manually edit polylines. If you exit to make some corrections (move line extremity, delete lines ...) on the network, you will be able further to re-enter into the “Patch Creation” command. For this, you just have to select all polylines and the mesh, and then restart the process.

Note: It may happen that you have a warning message to tell you that you have some potential dangerous lines that can introduce troubles when you will make BSpline surfaces. Our advice is to check (and correct if needed) these lines before running the next step. You can see these polylines into the object explorer (they appear in first position in the list). You may need to “**Zoom on**” these objects on then make with the contextual menu.

Computing NURBS curves

When your network of polylines is clean, you can start the “**next step**” in order to compute NURBS curves. In this process, lines of the network are approximated by BSpline curves, which are a type of NURBS, with an automatic tolerance.

The tolerance is a very important parameter because it has an influence on:

- The smoothness of the curve.
- The number of control points, which means the complexity of the curve. In other words, higher is the number of control points, longer is the computing time.
- The distance between the curve and the measurement, represented by the mesh.

A low tolerance needs a lot of control points to create the curve. If we apply a tolerance of 0.0, the resulting curve will have a large number of control points and may follow the noise of the mesh: in this case, at the end, CAD surfaces will not be relevant for you and not useful because not smooth enough.
By default Reshaper tries to find the best compromise and you can see the average deviation error for all curves and the maximum error for all curves. When the process is done, you can manually modify tolerance for all curves or just for some selected curves. To modify tolerance for only one curve, you just have to select it (Deviation values changes in the dialog box) and adjust tolerance with the slider.

The function manages also the tangency constraint when 2 polylines have similar directions. You can “Modify Tangent Constraints”. By convention, tangent constraints are represented in green while non-tangent constraints are red. You can click on one tangency constraint or on two non-tangency constraints to respectively lock or unlock the constraint.

At this step, you can either exit command (to edit manually the curves with the command CAD\Create BSpline) or go to the next step, which is the final step: BSpline surfaces creation.

Computing BSpline surfaces

When you have adjusted correctly the tolerance of the BSpline curves, you are ready to create the final surfaces by clicking on the next step button. This step is fully automatic. The curve network will become patches borders. These surfaces will also be fitted on the mesh: you can see in the dialog box the standard and maximum error deviation between the mesh and surfaces.

Note that after validation, some artifact issues may appear. This can be easily solved by entering a 5µ deflection to the surface representation. This whole process is detailed in the paragraph “Solving surface display artifact issues”.

At the end of the process you can have 3 types of surface regarding the surface quality:

- Green BSpline surfaces: These surfaces are made with 3 or 4 borders (BSplines curves). With this kind of surface, you have G0 (G-zero) constraint between surfaces (they share the same curve border).
- Orange BSpline surfaces: These surfaces are delimited by 4 border curves. The main difference with the green surface is that we have merged some curves that have the same tangency vector at extremity. With these surfaces we have no guarantee on G0 continuity; however the error is generally very small. We can see that the border line is very near from the other curves.
- Red BSpline surfaces: These surfaces have more than 4 border curves that cannot be merge to have only 4 curves (tangent vector at extremities are different). In this case, we make a greater surface (which has 4 borders), we fit it on the mesh and we make restriction on this surface in order to have the final surface. The problem of this type of surface is that we are not G0 with neighbor surfaces. The restriction is made with the projection of the BSpline curve on the great surface.
Depending on the result and surface quality you want, you may have to modify the initial line network or BSpline curves. All the intermediate polylines and curves coming from the previous steps can be found inside the folders “Contour Group” and “CAD group” of your object explorer.

**Improve Surfaces**

**Improve continuity between surfaces**

It is possible to apply tangency continuity (G1 constraint) between neighbor patches sharing a common edge. This command will align, as much as possible, tangency on the two patches common border.

To apply tangency constraint, select two neighbor surfaces and go to **CAD\Tangency Constraint**. When the execution starts, in the dialog box, you can see values corresponding to the current angle between surfaces (standard deviation angle and maximum angle in degree). In the 3D scene, you also see normal vector for each patch along the common edge.

In the dialog box, you can decide to move only one surface or both. Select the option “Both surfaces” and apply the constraint with the “Preview” button, new values of the angle between the patches are displayed on the right, inside the Dialog Box. In terms of angle value, you will never obtain exactly zero but a certain error will remain. However, if the resulting error is “small” the surfaces can be considered as tangent. The threshold value to consider two surfaces as “tangent” is in relation with the surface quality that you want to output:

- If you obtain an angle less than 3 degrees, this may be acceptable in most cases.
- If you obtain an angle less than 1 degree, this can be considered as a good quality.
- If you obtain an angle less than 0.5 degree, this can be considered as a very good quality with a severe criterion.

Note: You can also apply tangency constraint with orange BSpline patches. In this case G1 constraint is applied on the common border but the G0 constraint is not modified (if you have gaps before, you will still have gaps after tangent modification). You cannot apply tangency constraint on red surfaces (restricted patches).

You can also enter in the command **CAD\Tangency Constraint** without making any selection. In this case, the function will work like a paintbrush. Every time you pass over a common border between 2 patches with the left button pressed, the tangency continuity will be improved.

**Make holes / restriction on surfaces**

Sometimes it is interesting to make hole directly on a BSpline patch: this is useful if you want a smaller number of patches or if you want to avoid tangency problem between surfaces.

To do this you have to create the polyline network of lines without taking care of the hole. The best is to have a large rectangle around the hole. The resulting NURBS patch will completely cover the hole.
After that, you have to:

- Extract the hole with the command Polyline\Contour / Hole to get a polyline around the hole.
- Select the contour polyline and launch the command CAD\Create BSpline to transform the contour into a NURBS curve.
- With the patch covering the hole and the closed curve representing the hole you can use the command CAD\Hole/Restriction you can make a restricted patch. Do not forget to select the “Hole” option inside the dialog box before clicking the “Preview” button.

Making one surface only from patches
If you have followed this tutorial, you will obtain at the end a set of NURBS patches. However, in most situations, you want to get only one surface:

- This is easier to select and to handle.
- This is absolutely necessary if you want to make a comparison between the surface and something else because the command Measure\Compare / Inspect needs:
  - One and only one reference object (and not a set of patches) as reference entity.
  - One and only one object to compare (and not a set of patches).

To do this:

- Select all the patches that you want to group together in a unique surface.
- Launch the command CAD\Create compound.

Exporting your model
When you have reconstructed your CAD curves and surfaces, you may wish to use them with other CAD-CAM software.

You have the export function, which allows you to export your CAD objects into IGES or STEP files. Select objects that must be exported and go to the Main button\Export\Export CAD objects.

Solving surface display artifact issues
Sometimes, you may see that some patches have not a good aspect because the reflection of the light inside the patch is not smooth. This is an artifact due to the technique used to represent a NURBS surface.

A NURBS surface cannot be displayed “as is”. It requires some transformation so that your graphic board can make the representation. This transformation is called “discretization” and consists in sampling the continuous surface with “discrete” points.

In this process, the surface is “simplified” with a certain error called “deflection”. By default, Reshaper takes a deflection of 0.05 but in some cases this value is not low enough to get a good representation.

To change this deflection parameter, you have to select the patch(es) or surface(s) and launch the command Edit\Colors and Aspects. You will find a tab called “CAD” that contains the deflection of your model and you can change this value. You can also decide to change the default parameter so that the future patches are created with this new value.

Note: If this deviation is smaller, the representation will be better. However, you must be prudent and avoid giving a too small value because:

- The number of “discrete” points will increase.
- The “weight” of your model in terms of RAM memory will increase.
- Response time will be longer.
In the case of this exercise, you can enter 0.005 (5µ). However, this value must be adapted to the object size. In other words if you reshape a ship, a value of 5µ is not relevant!