

rhinoreverse™

Version 3 (for Rhinoceros® 5)

User Manual



30 August 2013

Note: Please watch the tutorial video on www.rhinoreverse.icapp.ch as well!

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rhinoreverse™ is a plug-in application for use with the Rhinoceros® CAD system (www.rhino3d.com). It was designed to enable the user to create NURBS surfaces efficiently based on given triangle meshes. With **rhinoreverse**, control point meshes can be converted to NURBS surfaces. Boundary curves are sketched onto the mesh. All calculations for the approximation and adaptation of the surface transitions are performed automatically, resulting in a very simple and intuitive workflow. **Rhinoreverse** offers the user the advantage of being able to represent large complex shapes very accurately with only a few smooth surfaces. **rhinoreverse** can handle the following input formats:

***.stl, *.pol, *.wrl, *.vrml, *.af, *.nas**

rhinoreverse is easy to use and is very cost-effective compared to other reverse engineering programs. **rhinoreverse** is a product of iCapp GmbH.



Load mesh



Sketch surfaces



Import surfaces



Settings



Healing



Patch



Distance measurement



Redesign curves

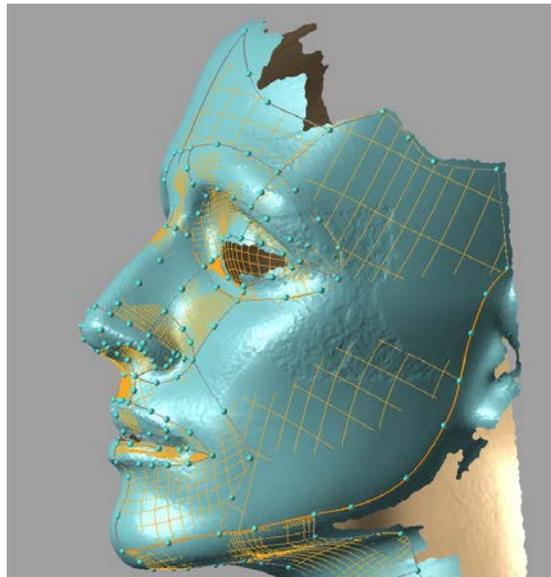
New in Version 3

All significant results and improvements from our development up to now have been incorporated in Version 3. And there's more:

- Distance measurements from the point cloud to the surface
- Geometry-based curve redesign

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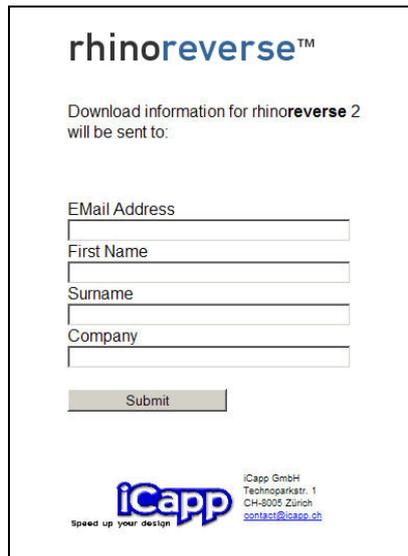
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Polylines (lines specified with a sequence of points) are drawn to divide a given mesh into individual NURBS surfaces. The partitions should be placed as close as possible to the curvature transitions of the surface. A NURBS surface will be automatically calculated for each closed cell.

1. Download

1. Ensure that Rhino 5 is installed on your computer before installing rhinoreverse.
2. Download the installation file of rhinoreverse from the web-site www.rhinoreverse.icapp.ch. A request form window will appear:



The image shows a web form titled "rhinoreverse™". Below the title, it says "Download information for rhinoreverse 2 will be sent to:". There are five input fields: "EMail Address", "First Name", "Surname", and "Company". Below these fields is a "Submit" button. At the bottom left is the iCapp logo with the tagline "Speed up your design". At the bottom right is the contact information: "iCapp GmbH, Technoparkstr. 1, CH-8005 Zürich, contact@icapp.ch".

3. Please enter your email address, first name, last name and company and click "Submit". The Internet page address to download the software installation package from will be sent to you immediately:

```
Please download RhinoReverse 2 Installer RhinoReverse_2b1.exe at
http://www.icapp.ch/iCapp-Daten/http-Download/RhinoReverse\_2b1.exe

For further information visit http://rhinoreverse.icapp.ch/
Please do not hesitate to contact mailto:rhinoreverse@icapp.ch
if you have any questions or suggestions

Enjoy!
```

4. Click on the provided link and save the rhinoreverse.zip file to the C: drive (hard disk) of the computer, you plan to install the software on.

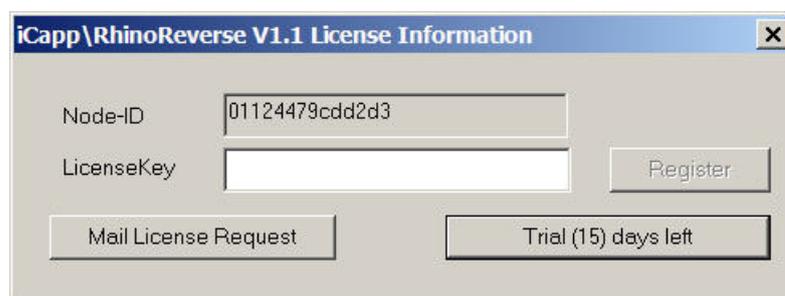
Note: The installation may fail if the installation files are run from any other drive letter, or if you try to run the installation across the network.

2. Install the plug-in

Launch the installation file. The software will be installed automatically, provided you've followed the installation instructions on the screen. The following files will be installed:

- **Program files** – The program file rhinoreverse.rhp is copied to the plug-in directory of Rhino 5.
- **Sample files** – These will be copied to the directory “Install-path\RhinoReverse\Example_Data”.
- **User manual** – The file “UserManual_eng.pdf” will be copied to “Install-path\RhinoReverse\Tutorial”. The user manual is accessible via the Rhino Help Menu and can be opened using the path Help/Plug-ins/Rhinoreverse in Rhino 5.

When you launch Rhino 5, the rhinoreverse plug-in will be automatically initialized and the toolbar will be displayed. The License Information window will be displayed:



rhinoreverse license request

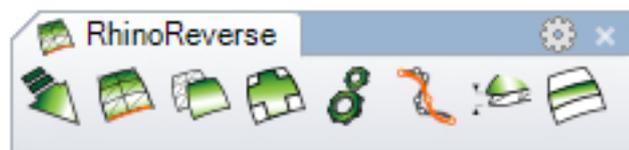
If you wish to try the software, click on the button **Trial (..) days left**. You may use rhinoreverse for the period indicated on the button. If you want to buy a license, please click on **Mail License Request**. An e-mail containing your node ID (the number of your computer determined by rhinoreverse) will be automatically generated. Please add any specific questions and suggestions you may have and send the e-mail directly to rhinoreverse@icapp.ch. iCapp will then generate a

(paid) license key valid for use on your computer only and send it to you. Please enter the issued license key in the **LicenseKey** field of the Licence Information window and click on **Register**.

Note: *You need to log in with administrative rights to install the software, because the license key must be written to the Windows registry file. If the registration fails, a message will be displayed. Log in as an administrator and repeat the registration procedure.*

3. Launch Rhino 5

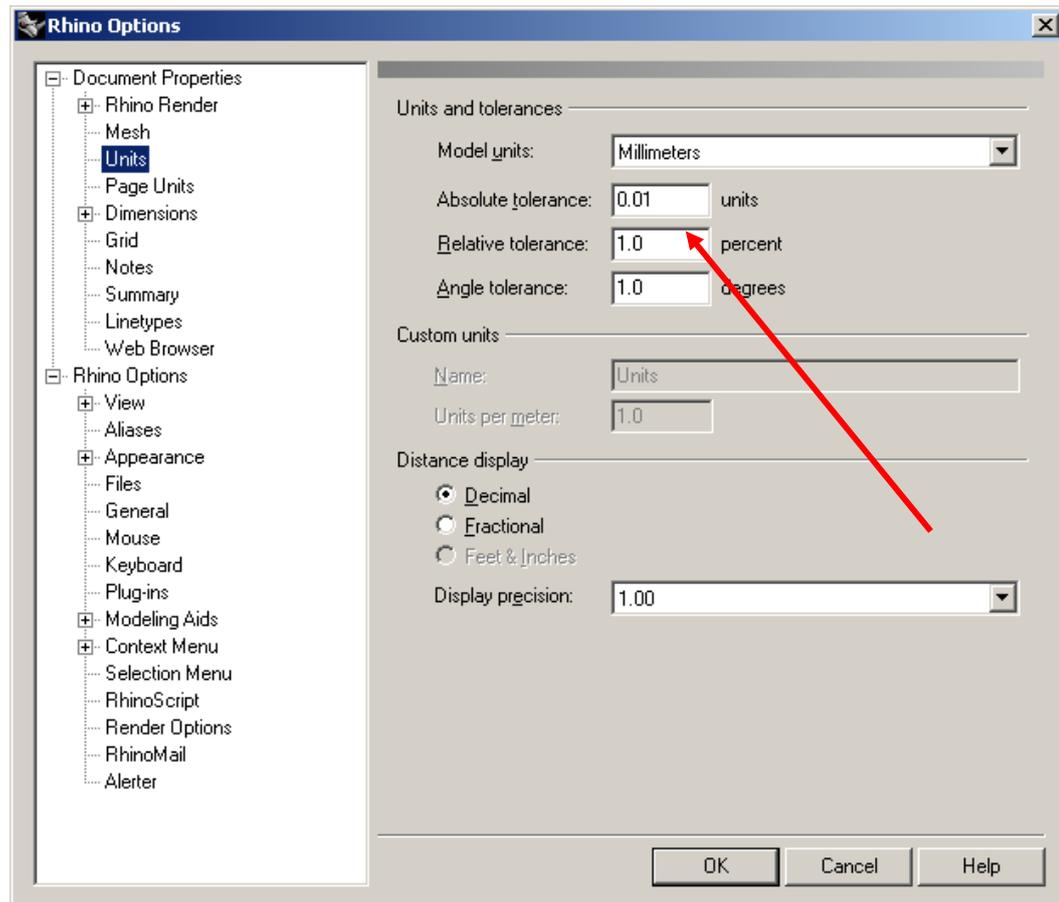
Launch Rhino 5 as you would any other software, e.g., via the Windows “Start” menu. rhinoreverse will be launched automatically and the toolbar will appear.



rhinoreverse toolbar

Once rhinoreverse function has been carried out, the menu item **iCapp Tools** will be initialized and displayed in Rhino’s main menu.

Now make sure the command prompt window is shown. In the main menu select **Tools/Options**, then select **Rhino Options/Appearance** and check the option to show **Command prompt**.



Check Absolute Tolerance

Then check Rhino's **Absolute Tolerance**. To do so, select **Tools/Options** and then **Document Properties/Units**. Adjust the tolerance setting to match the size of your current model. If it is too small, rhinoreverse will assign a high level of flexibility to the NURBS surfaces in order to reach the transitional tolerance between them. This will lead to increased amount of data to process, longer times required to calculate new surfaces and larger file sizes.

We recommend setting absolute tolerance value to 0.01 when working with models with dimensions around 1000 mm (see figure above).

4. Load mesh data

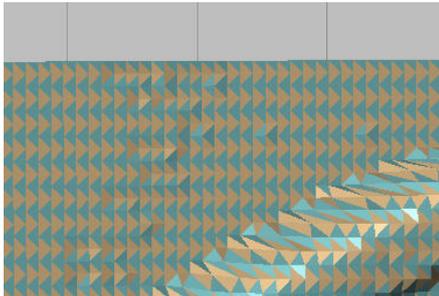
Note: The functions of **rhinoreverse** can be accessed via the toolbar, the **iCapp Tools** menu and the Rhino command window. The toolbar includes only the most important commands. If you wish to work with the command window, enter “**RR**” to display the list of all available **rhinoreverse** commands. The full list appears at the end of this document.

In the menu, click **LoadMesh** (or enter the command **RRLoadMesh**) to open a mesh file. Select any of the available Rhino mesh objects and press “Enter”, or press the “**f**” key to read a mesh from a file. The following file formats are available: *.stl (Stereo-Lithography), *.pol (PolyWork Version 1), *.wrl and *.vrml.

The option “**Delete input**” is automatically set to YES. This is to avoid duplication when displaying the mesh after converting it from Rhino to **rhinoreverse**.

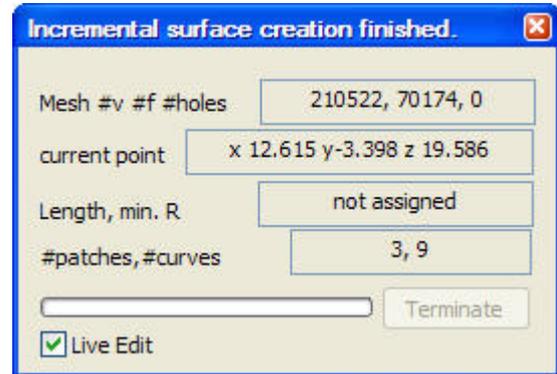
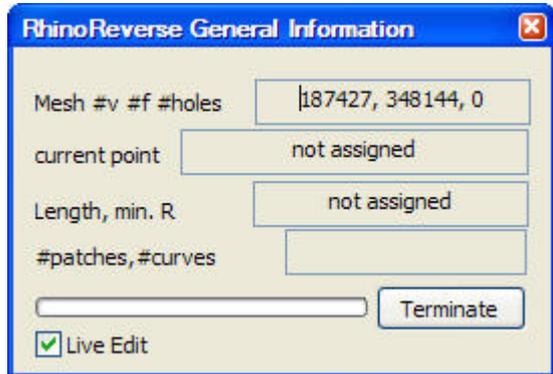
Please note the following important points:

- a. The given **point cloud must be meshed**. Point data without any connections between the points (mesh data) cannot be used.



- b. The picture shows a defective mesh. The color of each triangle represents the direction of its normal. **Neighboring triangles must have identical normals** and thus be the same color as well.
- c. **Large amounts of data** can slow down data display and processing. In such cases (> 1 million triangles), **rhinoreverse** will automatically use a simplified display.
- d. **Using inches as Model unit**. The default setting for the “RMS of Mesh” tolerance is 0.1. This value is correct if the model units used are millimetres and for models with dimensions of 100-1000 mm. If a model dimensions

are to be defined in inches (3.94 – 39.37 inches), the RMS tolerance must be adjusted to 0.005.



The "General Information" window appears after a mesh has been loaded

The "General Information" window shows the current status of the "sketch"

The "General Information" window appears after a mesh has been loaded. The first line of the window displays the number of vertices (#v), the number of facets (#f) and the number of holes (#holes) of the current mesh. The second line shows the coordinates of the current 3D mouse position on the mesh. In the third line, the length of the active polyline, shown in orange (or green, if you are currently modifying it with the mouse) is calculated. The fourth line shows the number of patches and curves generated. In the last line you will find a **Terminate** button to cancel the automatic calculation process.

Close this window to exit the rhinoreverse editing mode. The command **RREditGrid** can also be used for this purpose.

Live Edit. If you do not want to set new points to define the surface boundary curves, you can switch off the preview and thus the setting of points using the **Live Edit** button.

Note: The dialogue title shows the status of the current calculation.

5. Sketching surface boundaries

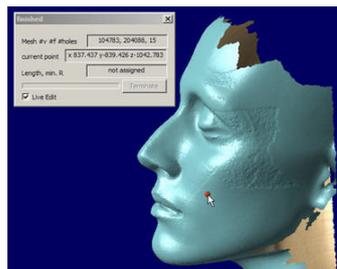
First, if working with mesh data, generated by measurement systems, please adjust the rhinoreverse measuring tolerance parameters. If the tolerance value is set too low, rhinoreverse may create a surface with "wrinkles. If the tolerance value is set too high, the surfaces will not follow the mesh closely enough. Please launch **RROptions** and adjust the **RMS** tolerance (RMS: root mean square distance) accordingly (see also section 8).

Sample: "Head" The mesh data

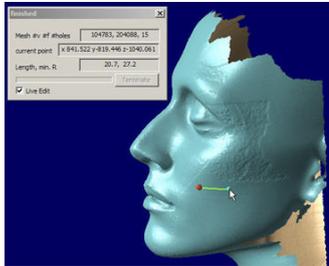
("Install-path\RhinoReverse\Sample_Files\Head_start.pol"), come from a laser scanner; the measurement accuracy is approximately 0.25 mm. In this case the RMS must be set to 0.25 (default is 0.1).

Sketch polylines

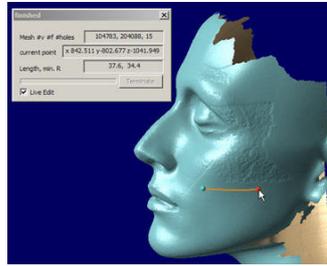
- a. Make sure that the edit mode of rhinoreverse is active: The "General Information" window (see section 4) must be open. The editing mode should be activated immediately after mesh data are read. If it is not activated, click on the menu item **Edit Grid** (command: **RREditGrid**) to activate the edit mode.
- b. **Click** anywhere on the mesh. The first point of a polyline (surface boundary curve) will be generated and shown as a small red sphere.



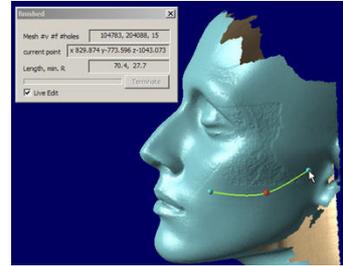
- c. **Moving the mouse** will generate a preview of the active polyline. The preview shows the polyline, which would be generated if you set the next point by left-clicking.



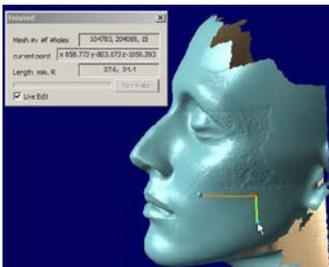
The green line shows a preview of the curve. When the user clicks at the cursor position...



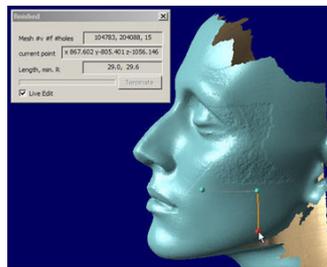
... the second point is set. The curve has two points now. It automatically becomes active (orange).



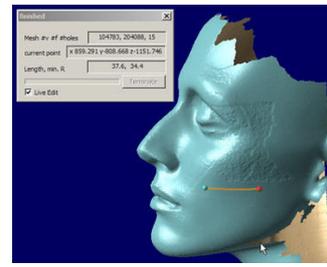
Option 1: When the mouse is moved again a preview of the whole curve is shown (green).



Option 2: If the angle of the mouse movement is more than 60 degrees, a new curve (green) will be started at the last point of the first one.

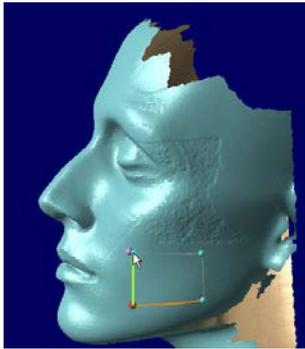


Clicking will set the second point of the new curve. The new curve is now active (orange) the old one inactive (gray).

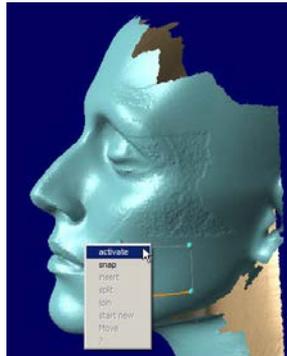


Option 3: If the distance between the cursor and the last point is greater than the combined length of the last two curve segments, left-clicking creates a new curve.

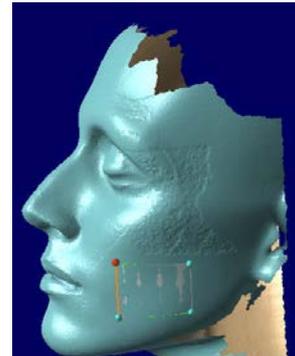
How do I close a loop?



If the cursor is close to an existing point, the point will be highlighted...



... If the user then clicks the left mouse key, a menu will showing all available options will be displayed.

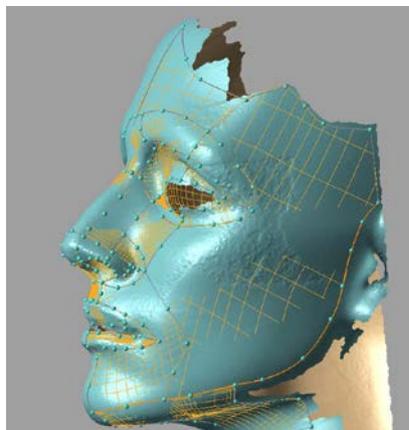


... In this case, select "Snapping" to close the loop and start the calculation of the first surface (gray).

Note: If polylines are not visible...

When the polylines are being drawn, an internal depth test mechanism (see section 8) decides what is displayed and what is hidden. If a curve or a part of the curve is behind the mesh, that part will not be displayed.

A complete grid is shown here.



Every closed loop of curves defines one surface (max. 25 curves).

When a closed loop is ready, the loop will be filled with one surface. Polylines with only one point will be deleted automatically.

Note: Curves that cross each other on the mesh surface without having a common defined vertex could lead to problems when calculating a new surface. Care must be taken to avoid crossing curves.

Note: The polylines that are used to define a closed loop must be connected with common points. The respective start and end points may not be adjacent to each other.

Note: Sometimes the displayed starting surfaces are not positioned exactly over the mesh data. They will be pulled into their final position on the mesh in the last approximation step (see section “Calculate surfaces”).

Edit polylines

- a. **Activate** any polyline by clicking on it, or on a point of the polyline, with the left mouse button. The active polyline is highlighted in orange.

Note: If you wish to activate the polyline, without activation any of its points, please click on the line segment between the points.

- b. **Move** single points of the active polyline by clicking and dragging them using the mouse. When you release the mouse key, the point is dropped and defined on the mesh.
- c. **Delete points.** Activate the point to be deleted by left-clicking it (the point is highlighted in red) and press the **Delete** button on your keyboard. Pressing Delete again deletes the entire polyline.
- d. **Delete polylines.** Activate the polyline (but not the point) by clicking on the line segment between the points and press the **Delete** button. If no point was activated (highlighted in red), the active polyline (highlighted in orange) will be deleted.
- e. **Join/split polylines.** Right-click the common point of the two polylines to join them. Click on any point to split the polyline again. This is important particularly when the surface boundary is to have sharp corners.

6. Calculate surfaces

NURBS surfaces can be calculated only when the polylines form a closed loop. Then a starting surface is generated and displayed.

When transferring intermediate surface data to Rhino, use the menu item **Commit** (command: **RRCommit**) to approximate these surfaces on the mesh. The transitions between the surfaces will be automatically adjusted to the specified tolerance and the surface then transferred to the Rhino database. The command **RRRelief** is used to generate a relief surface; see the section entitled “Additional Tools”.

Note: *Sometimes the display of the surfaces and the mesh is not optimal for the current task. For this reason, various display modes are available. Select **RROptions** to change the display options. See more about the options in the section “Additional Tools”.*

Transitions between surfaces. Rhino offers several options for analyzing transitions between surfaces (e.g. in a typical zebra plot). If the resulting quality is not satisfactory, use rhinoreverse’s **Heal** function (command: **RRHeal**) on a selected set of surfaces to improve the transitions.



Zebra plot in Rhino

7. Calculate curves/curve networks

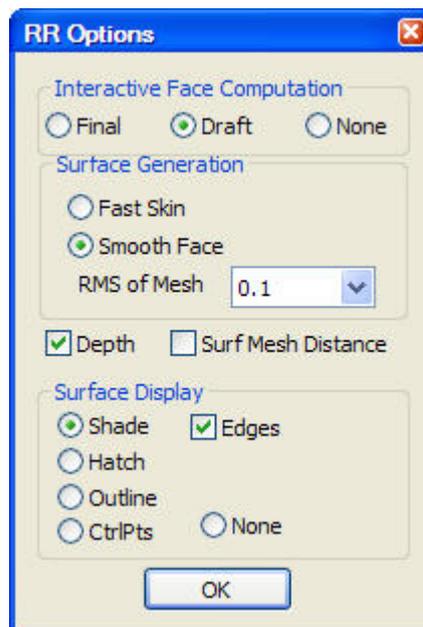
In addition to calculating surfaces, the user also has the option to import the generated polylines as curve entities to the Rhino database. To do this, use the corresponding option of the **Commit** function (command: **RRCommit**.)



Wireframe curves in Rhino, defined by polylines in rhinoreverse

8. Additional tools

- a. **Options**. A dialogue box with specific options for rhinoreverse (command: **RROptions**) is displayed.



rhinoreverse Options

- **Final.** The approximation of the surfaces is done continuously, so that the user can evaluate the final form of the surfaces at any time. This setting requires a lot of computing power and may delay the display.
- **Draft.** In the standard setting, the initial surfaces are automatically roughly approximated to the mesh. However, the final form of the surfaces is calculated only when the user selects the “Commit” function.
- **None.** No surface calculations are done. The surfaces are calculated only when the user selects the “Commit” function.
- **Surface Generation - Fast Skin / Smooth Face.** “Fast Skin” is a fast and robust, but less precise method for calculating surfaces. It uses interpolation rather than approximation. This method allows generation of more complex surfaces, but it is less smooth than the “Smooth Face” method, which uses an approximation step to carry out the final surface refinements. Note: An additional **RRRelief** surface generation option is available. See more details below.
- **RMS of Mesh.** Expected measuring precision or “white noise” of the mesh; this depends on the measuring device used to define the points for this mesh. The “measurement precision” is the tolerance range of the deviations of the measured points from the real surface. In rhinoreverse the RMS setting limits the approximation iterations up to a value corresponding to this internal tolerance. This setting is also used as a basis for distance calculations.

The RMS value should always match the measuring precision of the mesh!
An approximation calculation with excessively high precision can result in extremely long calculation times and wavy surfaces.
- **Depth Test - on -.** Switches the depth test of the display on. Hidden surfaces and polylines will not be displayed.
- **Surf Mesh Distance.**

Note: *This is an experimental option. Save your model before using this option.*

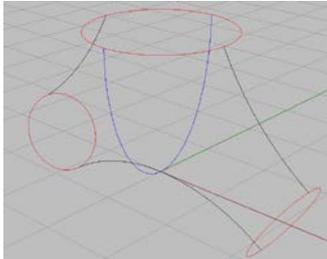
This option is used to calculate the distance between surfaces and the mesh online. Please note that the “Final” option must also be activated. When a polyline is edited, the associated surfaces are recalculated and the distances to the mesh are displayed in colour. The colour shade depends on the current setting for the RMS value of the mesh, as it is assumed that the surface is approximated to the mesh up to this distance. The colour goes from green to yellow to red. Green indicates the range between +/- 1 x RMS, yellow between +/- 2x RMS and red +/- 3 x RMS.

- **Surface display – Shade.** – The initial surfaces are displayed with shading.
- **Surface display – Hatch.** -. The surfaces are represented only by their iso-curves, allowing a transparent view of the mesh and the polylines.
- **Surface display – Outline.** - Only the outlines are shown with an offset to the inside. This option helps check whether all loops are closed as required.
- **Surface display – CtrlPts.** - The control polygons (control points and control mesh) of the NURBS surfaces are displayed.
- **Surface display – none.** – The surface display is completely switched off.
- **Edges.** Switches the display of face edges on or off.

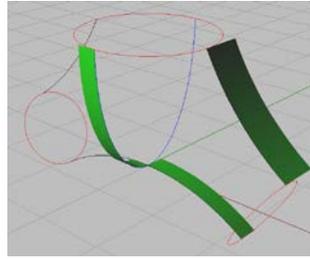
b. **Special NURBS commands**

1. Heal. All transitions of the selected set of surfaces are checked and the absolute and angular precision values are adjusted automatically to match those set within Rhino’s (command: **RRHeal**).

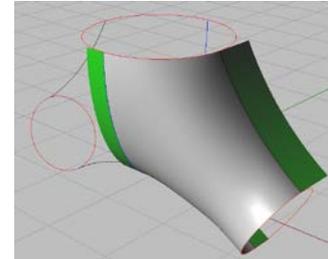
2. Patch. Patch may be used to close a hole between surfaces. There is an optional setting within the patch command to match tangency to adjacent surfaces at the boundary curves.



Curve network

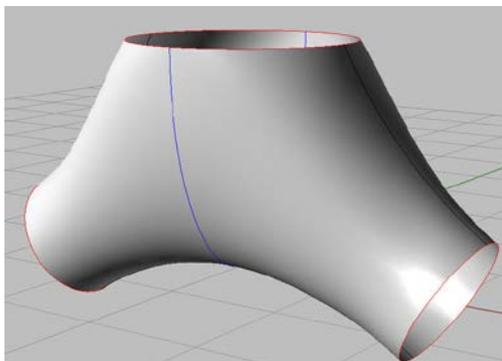


Support faces to define tangential boundary condition

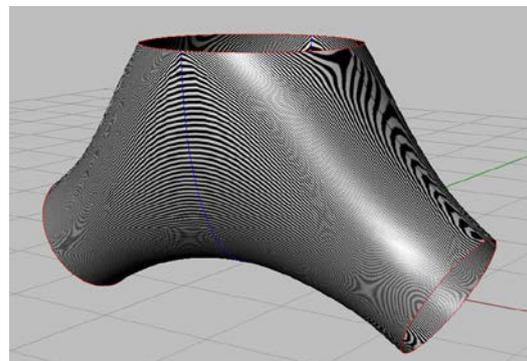


Result of RRPatch

When designing surface models, users frequently need to fill in gaps between already finished surfaces with a tangential surface. The menu item **Patch** (command: **RRPatch**) enables such gaps to be closed automatically. After selecting the function, select the corresponding boundary curves of the adjacent surfaces. If these do not form a closed loop, the opening is closed in linear fashion.



Result of RRPatch (4 times)

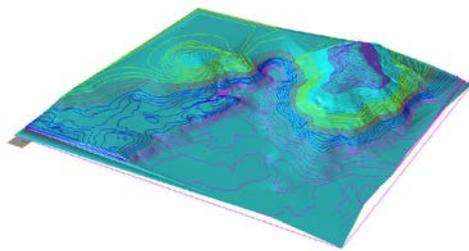


Zebra plot to display transition quality

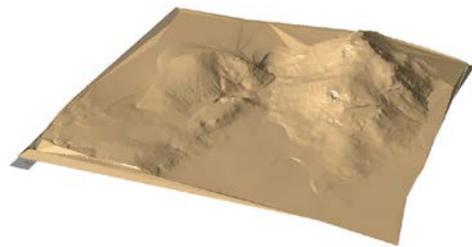
3. Relief. In geography and architecture, large relief surfaces are frequently available only in STL format. The function **RRRelief** was developed to convert

them into NURBS. This function works in a similar fashion to Rhino's "Drape" function. The crucial difference is that steep surface areas can also be accurately mapped: the control points are evenly distributed throughout the surface.

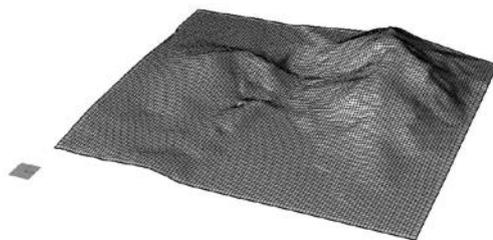
The boundary curves (polylines) of the desired patch must be sketched on the STL data before the **RRRelief** command can be launched and the number of control points set for the edges to control the mapping accuracy.



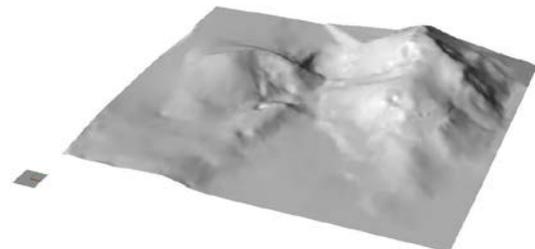
Given relief structure



Given STL surface



*Resulting NURBS (200 control points
in the long edge)*



Resulting NURBS (shaded view)

4. Mesh-face Distance. The approximated surface is analyzed at a distance to the reference "point cloud". The result of the calculation is displayed in a false color rendering. Green areas match the point cloud, red areas represent the negative triple RMS value, and blue areas represent the positive triple "RMS" value. White and black areas are outside the calculated tolerances.

Topological details can be analyzed in greater detail by changing the RMS settings and recalculating.

Several areas can be selected for distance measurement. In this case a separate mesh is calculated in order to display each surface.

The false color rendering is a separate mesh. It can be moved to a separate level for documentation if necessary.

5. Redesign curves. The redesign of curves is always strictly geometry-based. The maximum allowed distance can be freely selected. The best possible result is determined by the control point distribution. The objective is always the largest possible reduction of control points.

9. Tables

Commands relating to the mesh and the generation of polylines:

Tool	Menu Item	Command	Description
	Load Mesh	RRLoadMesh	Imports a Rhino mesh object or opens a mesh file
	Edit Grid	RREditGrid	Switches on the display for the polylines belonging to the current mesh. "Edit Mode" is activated.
	Commit	RRCommit	The surfaces are approximated and the transitions adjusted. Then the surfaces or curves are transferred to the Rhino database.
	Options	RROptions	Sets tolerances and display options.
-	Hide Mesh	RRHideMesh	Hides or shows the mesh object
-	ImportGridV1	RRImportGridV1	Enables the import of polylines generated and saved with rhinoreverse version 1
-	Relief	RRRelief	Fast method of "draping" a specific area of a given point mesh with one NURBS surface

Additional tools which work on NURBS:

	Heal	RRHeal	All transitions of a selected set of surfaces are adjusted automatically to conform to Rhino's "Absolute Tolerance" and "Angular Tolerance"
	Patch	RRPatch	Covers a hole enclosed by an open or closed loop of selected surface boundary curves.
	Mesh-Face Distance	RRMeshFaceDis tance	Calculates the distance from the approximated surface to the given mesh.
	Curve	RRCurve	New approximation of the curve depending on the given geometry