

# rhinoreverse™

Version 2 (for Rhinoceros® 4, SP 1)

## Tutorial



31. October 2008

**Note:** Please also see the video tutorial on [www.rhinoreverse.icapp.ch](http://www.rhinoreverse.icapp.ch)

rhinoreverse™ is a plug-in application for use within the Rhinoceros® CAD-system ([www.rhino3d.com](http://www.rhino3d.com)). It has been designed to enable the user to create clean efficient surfaces (NURBS) based on available mesh data. Input from the user has been reduced to a minimum. Boundary curves are sketched onto the mesh. The program then automatically calculates the surfaces, transitions and curvature matching. This leads to very simple and intuitive handling of the software and offers the user the substantial advantage of being able to represent large complex shapes by only a few surfaces. rhinoreverse will handle the following input formats:

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

rhinoreverse is easy to use and represents favourable cost savings compared to alternative reverse engineering applications. rhinoreverse is a product of iCapp GmbH.

## **New in Version 2**

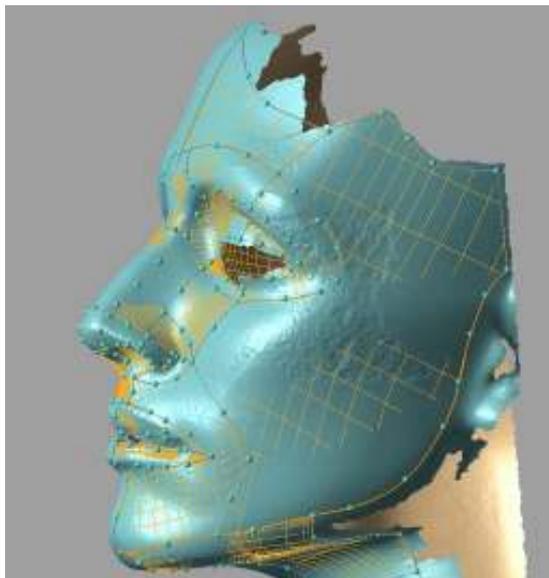
Version 1 has been significantly developed to provide improvements incorporated in to Version 2:

- **Greater design flexibility and higher data quality.** Surfaces are no longer restricted to a maximum of 4 boundary curves. The user is free to sketch up to 25 boundary curves for a single surface. The resulting trimmed surfaces are created automatically and the seams are adjusted to Rhinos stitching tolerance. The resulting surfaces show close conformity to the mesh. They are highly accurate and display smooth transitions between adjacent surfaces. You will be able to modify the surfaces with normal Rhino commands.
- **Ease of use.** rhinoreverse works completely inside Rhino 4 enabling the same ease and speed of use. You can use the command line and mouse controls just as you would in Rhino 4.
- **Simplified data management.** All data is now stored within Rhinos' 3dm-file. Extra files are not needed to store rhinoreverse's mesh and grid information.

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*The purpose of “curves” (curves with small balls along them), is to mark the mesh surface as a grid. The curves should be placed near curvature changes. Each cell of the grid (completely bounded by curves), will be closely approximated, automatically by one surface.*

## 1. Download

1. Rhino 4 must be installed on your computer before installing rhinoreverse.
2. Download the installation version of rhinoreverse from the web-site [www.rhinoreverse.icapp.ch](http://www.rhinoreverse.icapp.ch). A request window will appear:



The image shows a web form for downloading rhinoreverse 2. It includes the rhinoreverse™ logo at the top, followed by the text "Download information for rhinoreverse 2 will be sent to:". Below this are four input fields: "EMail Address", "First Name", "Surname", and "Company". A "Submit" button is located below the fields. At the bottom of the form, there is the iCapp logo and contact information: "iCapp GmbH", "Technoparkstr. 1", "CH-8005 Zürich", and "contact@icapp.ch".

3. Please enter your email address, first name, surname and company. The download location will be send to this email address:

```
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 given link and save the rhinoreverse-zip-file to the C partition (hard disk), of the computer where you want to install the software.

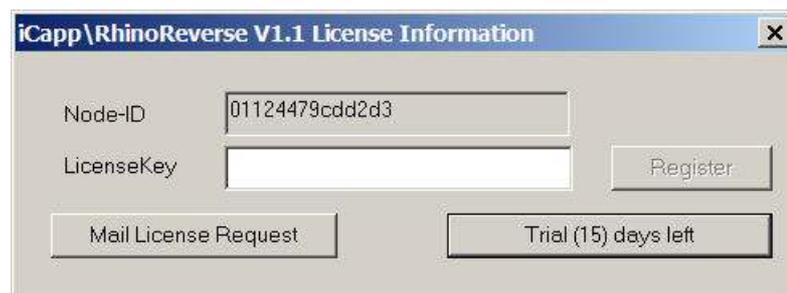
**Note:** The installation can fail if the installation files have been stored on any other disk or if you try to install across a network link.

## 2. Install the plug-in

Unzip the installation file and run file "Setup.exe". Follow the installation process displayed on the screen. The installation of the plug-in is done automatically. If this fails to install the rhinoreverse plug-in when Rhino starts, use Rhinos plug-in manager. The following files will be installed:

- **Program files** - rhinoreverse.rhp is copied to the plug-in directory of Rhino (Install-path).
- **Example data** - will be copied to "Install-path\RhinoReverse\Example\_Data".
- **Tutorial** - Tutorial\_eng.pdf will be copied to "Install-path\RhinoReverse\Tutorial". The tutorial can be opened by calling Rhinos menu function: Help/Plug-ins/Rhinoreverse in Rhino 4.

When starting Rhino, after finishing the installation process, rhinoreverse will be set up automatically and the toolbar will be shown. You will also be asked for a license:



*rhinoreverse license request*

With clicking on the button **Trial (15)...** you can test the software as long as your trial period is valid (number of remaining days is displayed in brackets). If you want to buy a license, please click on **Mail License Request**. An automatically created email including your Node-ID will be prepared. Send the mail directly to iCapp GmbH (rhinoreverse@icapp.ch). iCapp will generate a

valid license key for use on your computer only. Please enter the issued license key to the field of the dialogue box and click on **Register**.

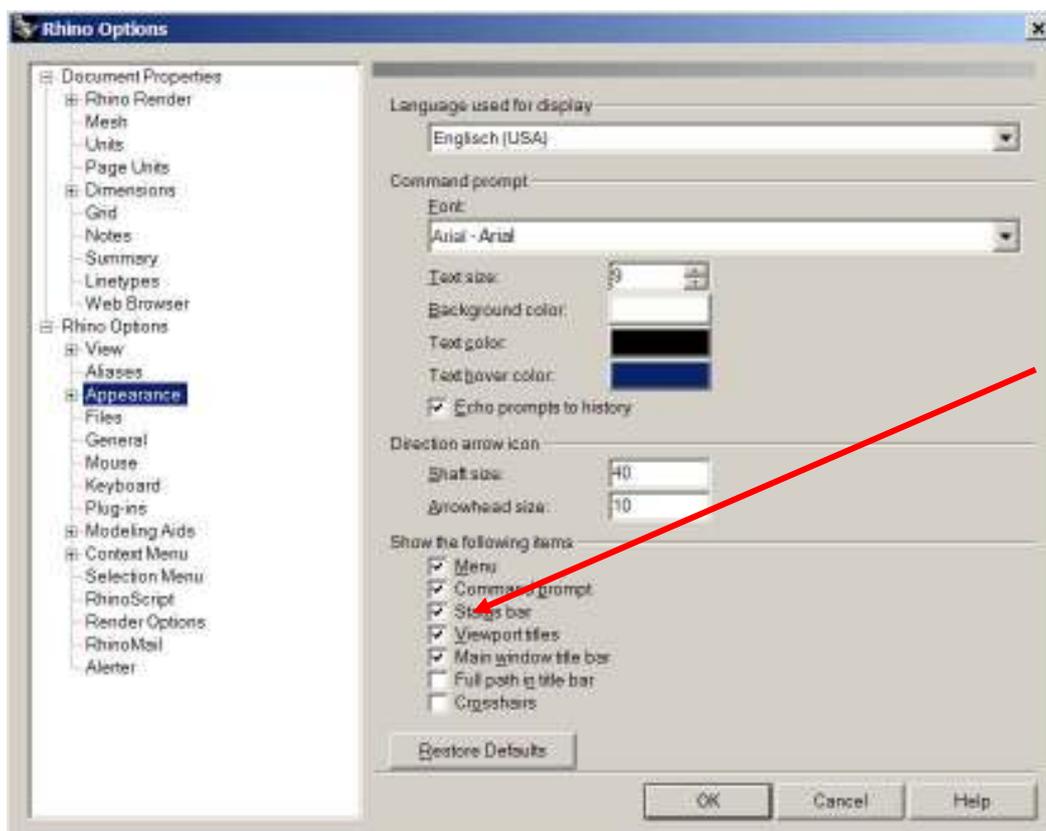
**Note:** When registering the code, you must have the permission to modify the registry file of your windows system. If the registration does not work, a message will appear. You will need to log in as administrator and repeat the registration procedure.

### 3. Start Rhino 4

Start Rhino 4. rhinoreverse will be started automatically and the toolbar will appear.



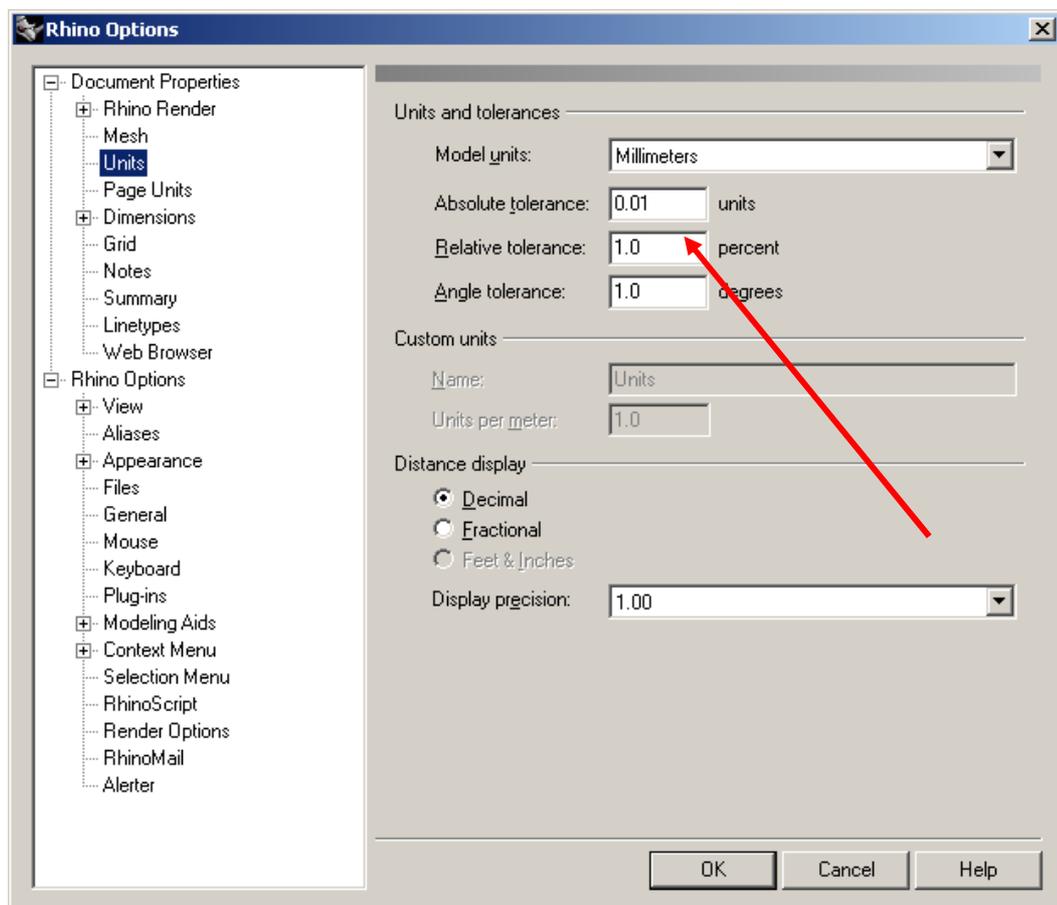
*rhinoreverse toolbar*



*Activate the command prompt*

After calling any of rhinoreverse 's commands, the menu-entry **iCapp Tools** will also be shown in the Rhino options dialogue box under 'Appearance'.

- a. If you want to use the command mode, make sure that the command prompt of Rhino is active. Click on the menu entry **Tools/Options** to display the option-set **Appearance** and activate the option **Command prompt** (see figure above).



#### *Check Absolute Tolerance*

- b. Check Rhinos default **Absolute Tolerances** in the menu Tools/Options/Units. Please adjust the tolerance setting for the size of your current model. If it is too small, rhinoreverse will take longer to calculate new surfaces and produce larger file sizes. Experience shows,

that a tolerance value of 0.01 is suitable for a model space of 1000 mm (see figure above).

#### 4. Load mesh data

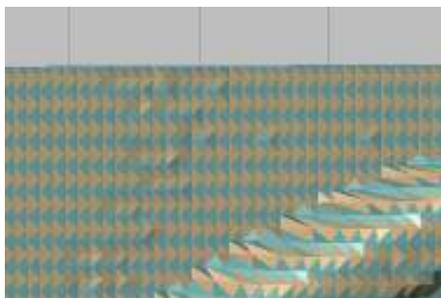
**Note:** All functions are located in the menu **iCapp Tools**. They can be used by entering them on the Rhino command line. The commands have the same name, but use the additional prefix **“RR”**. If you type **“RR”** in the command line, you will get a list of all available **rhinoreverse** commands. The full list appears at the end of this document.

Click on the menu-entry **LoadMesh** (command: **RRLoadMesh**), to open any mesh file. Select any available Rhino mesh object and press enter. Or press **option “f”** to read a mesh from a file. The following file formats are available: \*.stl (Stereo-Lithography), \*.pol (PolyWork Version1), \*.wrl and \*.vrml.

The **option “delete input”** is automatically set to YES. This is to avoid any double displaying of the mesh, when converting a Rhino mesh to **rhinoreverse**.

**Please pay attention to the following very important notes:**

- a. The given **point cloud must be meshed**. Point clouds composed of independent points cannot be used.

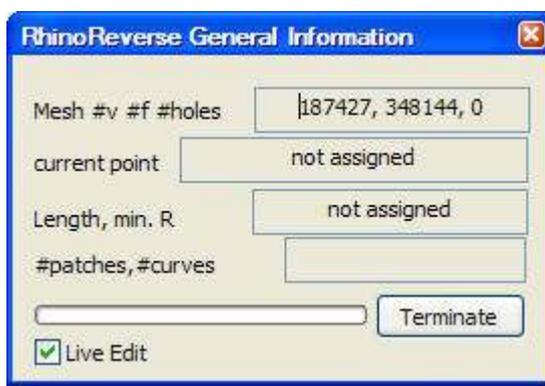


- b. The picture shows a wrong mesh. The color of each facet depends on the facet normal. Neighbouring faces must be displayed in the same color to indicate correct normals.

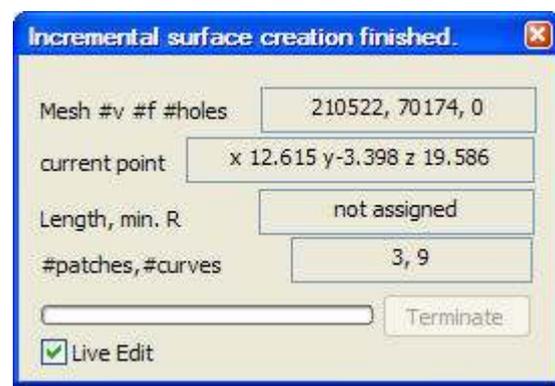
- c. **Large amounts of data** lead to a sluggish display and slow performance. Try to optimise the amount of data to achieve sufficient

*accuracy and performance. rhinoreverse will automatically minimise the amount of data shown dynamically if necessary (> 1 million facets).*

- d. **Model unit Inches.** The default tolerance setting "RMS of Mesh" is 0.1. This value is good, if meshes are given in mm and if they have a size of approx 100-1000 mm. If any model with the same size, but given in inches (3.94 – 39.37 inches) should be processed, the tolerance has to be adjusted to 0.005.*



*"General Information Window" appears after loading any mesh*



*"General Information Window" after creating 3 patches with 9 curves*

The "General Information" window of rhinoreverse appears after loading the mesh. Closing this window will finish the editing mode of rhinoreverse. The command **RREditGrid** does the same. 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 gives the coordinates of the current mouse position on the mesh. The third line calculates the length of the active curve, shown orange (or green, if you are currently modifying it with the mouse). The fourth line shows, how many patches have been completed and how many curves created. In the last line you will find a **Terminate** button to cancel any unnecessary calculation.

**Live Edit.** If you do not want to set new curve points, you can switch off the preview using option **Live Edit**.

**Note:** *The headline of the dialogue changes while editing the grid. It is used to display the current calculation state.*

## 5. Define surfaces by sketching curves

Check the tolerance/resolution and measurement deviation parameters for any scanner or digitizer used to produce source data for the meshes.

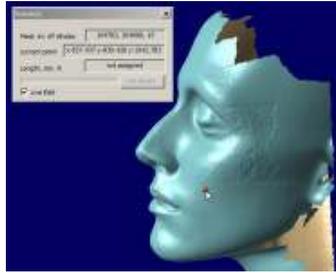
rhinoreverse could produce wrinkles if tolerances are too small. If too high, surfaces may be generated with large deviations from the source mesh. Call **RROptions** and adjust the **RMS** parameter (see also section 8).

**Example “Head”:** *The given mesh data*

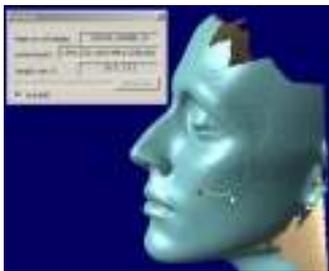
*“Install-path\RhinoReverse\Example\_Data\Head\_start.pol”, comes from a laser scanner, the “measurement deviation parameter” was approx. 0.25 mm. In this case the RMS was set to 0.25 (default is 0.1).*

### Sketch curves

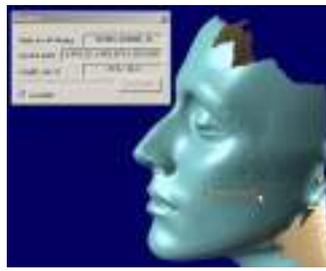
- a. Make sure that the edit mode of rhinoreverse is active: The “General Information Window” (see chapter 4) must be open. The window will appear after loading the mesh. If not, click on the menu-entry **Edit Grid** (command: **RREditGrid**) to activate the edit mode.
- b. **Click with the left mouse button** somewhere on the mesh. The first point of a curve will be set and drawn as small red ball.



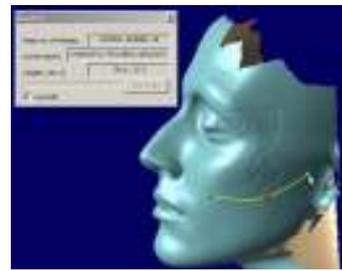
c. **Moving the mouse** will preview all options to set additional points:



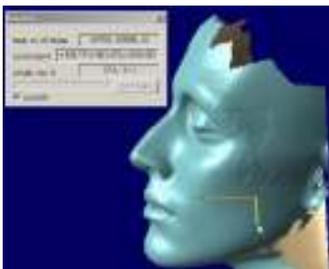
*The green line gives a preview of the curve. When the user clicks at the cursor position the...*



*... second point will be set. The curve now has two points. It automatically becomes active (orange).*



*Option1: When the mouse is moved again a preview of the whole curve appears (green).*



*Option 2: If the angle is more than 60 degrees a new curve (green) will be started at the last point...*

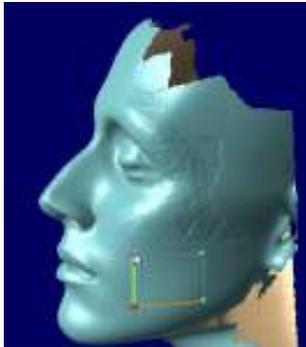


*Clicking will set the second point of the new curve. The new curve is now active (orange) the old one inactive (grey).*



*Option 3: If the distance between the cursor and the last point is bigger than the length of the last two curve segments, an independent new curve can be started by a left mouse click.*

How to close a loop?



*If the cursor is near to any existing point, the point will be highlighted...*



*... If the user then pushes the left mouse button a menu will appear and show all available options.*

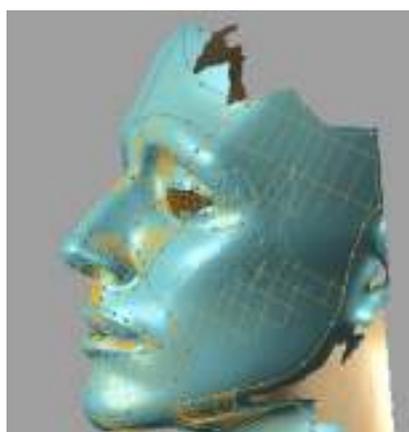


*... In this case snapping will close the loop and start the calculation of the first surface (grey).*

**Note:** *If curves are not visible...*

*When displaying the curves an internal depth test mechanism decides what is in front and what is behind. If a curve or a part of the curve is behind the mesh, that part will not be displayed.*

A complete grid of curves 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.

Curves that consist of only one single 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 calculating a new surface. Care must be taken to avoid crossing curves. Instead users may construct 'junctions' in the curve grid by ensuring curves meet exactly at their end points at those positions.*

**Note:** *All curves that are used to define a loop must be drawn with end snaps active, snapping to the end points of adjacent curves to form a completely closed loop.*

**Note:** *Sometimes starting surfaces are not positioned exactly on the mesh. The final position will be calculated when finishing and leaving the edit-mode (see section "Calculate surfaces").*

## Edit curves

**Activate** any curve by clicking on it with the left mouse button. The active curve is highlighted in orange.

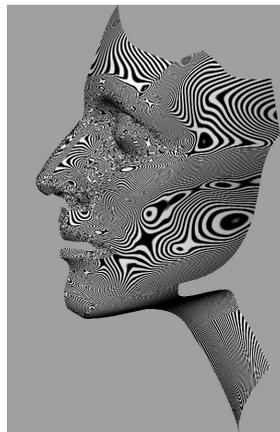
- a. **Move** single points of the active curve, by selecting and moving them with the left mouse button.
- b. **Delete points.** Activate the point with the left mouse button and press the **Delete** key on your keyboard. Pressing the delete button again will delete the corresponding curve.
- c. **Delete curves.** Activate the curves and press the **Delete** key. If no point is shown in red the active line (shown orange), will be deleted.
- d. **Join/split curves.** Click with the right mouse button the curve point (ball), where you want to join or split.

## 6. Calculate surfaces

Define a grid of curves, which covers the loaded mesh. Starting surfaces will be calculated automatically and displayed. When transferring intermediate surface data to Rhino, using the menu entry **Commit** (command: **RRCommit**) an automatic healing function will adjust the transitions (positions and tangents), between the surfaces. If you want to create a relief surface please see option **RRRelief** in section "Additional Tools".

***Note:** Sometimes the display mode will not appropriate for the current job. Call **RROptions** to change the display options. See more about the options in the section "Additional Tools".*

**Transitions between surfaces.** To check the transitions between the surfaces several Rhino-Commands are available (e.g. zebra-plot). If the resulting quality is not satisfactory, use rhinoreverse´s heal command on a selected set of surfaces to improve the transitions. (**RRHeal**)



*Zebra plot in Rhino*

## 7. Calculate curves

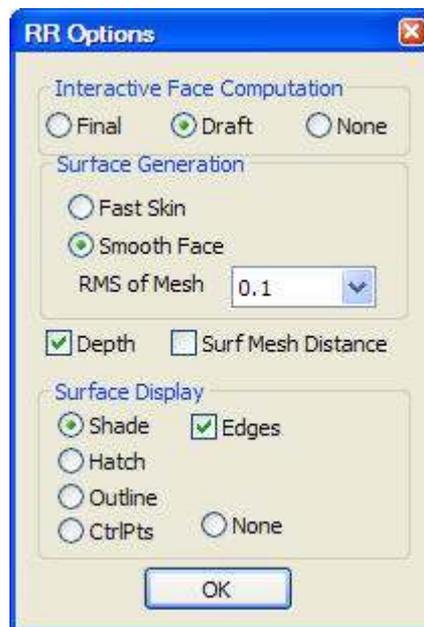
The user also has the option to transport the designed curves as curve entities to Rhino. Please use menu-item **Commit** (command: **RRCommit**) and choose the option 'Curves'.



*Wireframe curves in Rhino, defined by curves in rhinoreverse*

## 8. Additional tools

- a. **Options.** Opens a dialogue box to set some specific options for rhinoreverse (command: **RROptions**).



*rhinoreverse Options*

- **Final.** Face generation will be done online in final quality mode.
- **Draft.** Face computation will be done online in draft mode.

- **None.** Online face computation is switched off. The faces will be calculated when calling the commit command.
- **Surface Generation - Fast Skin / Smooth Face.** “Fast Skin” is a fast and robust skinning method, which may be used for fast calculation of “rough” surfaces. It is an interpolation method. This method is able to create more complex surfaces, but it is less smooth than the “Smooth Face” method, which uses an approximation step to make the final refinement in the surface. Note: There is another additional function for surface generation **RRRelief**. See more details below.
- **RMS of Mesh.** Expected measurement uncertainty or ‘white noise’ of the mesh, is dependent on the tolerance/resolution/deviation of any measurement device used to define the source data. How this is set limits the interval used to sample data when a surface is created. Care should be taken to set an appropriate value. Adjust the setting to the “white noise” of your mesh.
- **Depth Test - on -.** Hidden objects will not be displayed.
- **Surf Mesh Distance.**

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

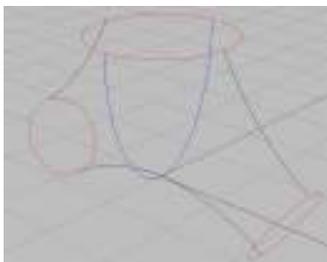
This option is used to calculate the distance between new surfaces and the mesh. Note: Option “Final” must be set active. When modifying any curve point the associated surfaces will be rebuilt and a colour plot will appear. The colour scaling depends on the current setting for the ‘white noise’ of the mesh (RMS). The colour scaling goes from green over yellow to red. Green colours the range between +/- 1x RMS, yellow between +/- 2xRMS and red +/- 3xRMS).

- **Surface display – Shade.** - Surfaces will be shown in shaded view.
- **Surface display – Hatch.** -. The surfaces will be displayed by their iso-curves only, allowing a clear view of the mesh and the grid.
- **Surface display – Outline.** - Only the outlines will be shown with an offset to the inside. This option helps when editing or checking closed and open loops.
- **Surface display – CtrlPts.** - The control polygons of the surfaces will be displayed.
- **Surface display – none.** - Switches off displaying surfaces.
- **Edges.** Switch displaying of face edges on or off.

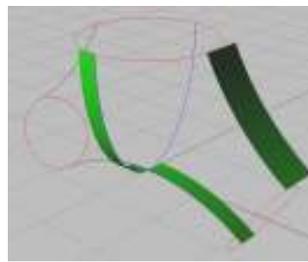
#### b. Special NURBS commands

**1. Heal.** All transitions of a selected set of surfaces are adjusted automatically to conform to Rhinos “Absolute Tolerance” and “Angular Tolerance” (command: **RRHeal**).

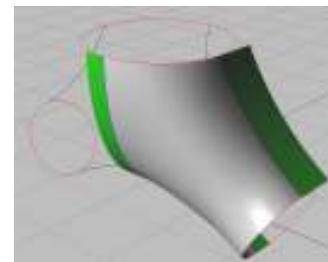
**2. Patch.** Patch may be used to close a hole between several surfaces. There is an optional setting within the patch command to match tangency to adjacent surfaces at the boundary curves.



*Curve network*

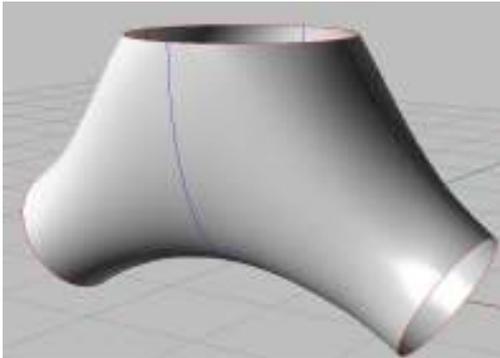


*Support faces to define tangential boundary condition*

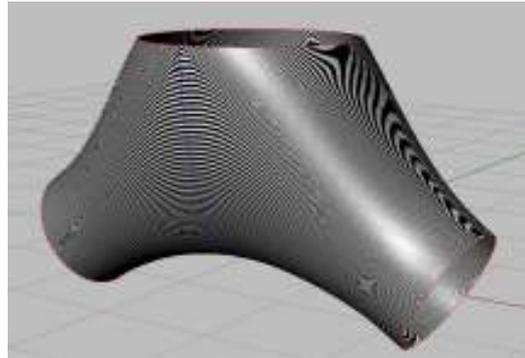


*Result of RRPatch*

The menu-item **Patch** (command: **RRPatch**) enables holes to be closed automatically. After calling the command, the user selects the boundary curves of the surfaces. They do not need to form a closed loop at their boundary.

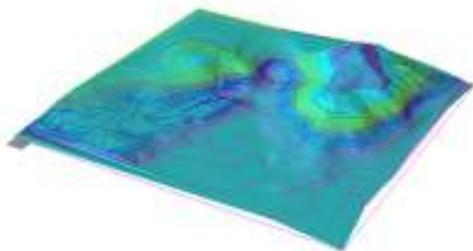


*Result of RRPatch (4 times)*



*Zebra plot to display transition quality*

**3. Relief.** In geography and architecture, often large relief surfaces are available in STL format. The method **RRRelief** was designed to transform them into NURBS. This function works similar to Rhinos “drape” command. The main improvement is that steep surface areas are taken into account. The boundary curves of the desired patch have to be sketched first. Then command **RRRelief** has to be entered. The user can set/adjust the number of control points 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)*

## 9. Tables

Commands that deal with rhinoreverse mesh and grid structure:

Tool	Menu Entry	Command	Description
	<b>Load Mesh</b>	<b>RRLoadMesh</b>	Load any mesh data file
	<b>Edit Grid</b>	<b>RREditGrid</b>	Display the grid, which belongs to the selected mesh. Enable modifications
	<b>Commit</b>	<b>RRCommit</b>	Adjust the transitions of rhinoreverse surfaces and transport the surfaces and curves to the Rhino model
	<b>Options</b>	<b>RROptions</b>	Set tolerance and display options
	<b>Hide Mesh</b>	<b>RRHideMesh</b>	Hide or show the mesh object
	<b>ImportGridV1</b>	<b>RRImportGridV1</b>	Enables importing grid files, that have been designed with rhinoreverse version 1
	<b>Relief</b>	<b>RRRelief</b>	Fast method to “drape” a specific area of a given point mesh with one NURBS surface

Additional tools, which work on NURBS:

	<b>Heal</b>	<b>RRHeal</b>	All transitions of a selected set of surfaces are adjusted automatically to conform to Rhinos "Absolute Tolerance" and "Angular Tolerance"
	<b>Patch</b>	<b>RRPatch</b>	Covers a hole enclosed by an open or closed loop of selected surface boundary curves.