

VTK Export for Rhino

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VTK Export for Rhino Help

Welcome to *VTK Export for Rhino* Help!

This help file provides all the information you need to understand and use *VTK Export for Rhino*. Click the links below or use the Contents tab on the left to browse help topics. You can also scroll the topics on the above right hand pane.

The *VTK Export for Rhino* help file is divided into the following sections:

- [Introduction](#) - Introduction to *VTK Export for Rhino*.
- [Command Reference](#) - Quick reference of all the commands added by *VTK Export for Rhino*.
- [Tutorial](#) - Tutorial to get you started in the shortest possible time.

If you need any additional information please [contact](#) SYCODE.

Note:

- Please refer to the section "Loading Instructions" in the [ReadMe](#) file to load *VTK Export for Rhino* plug-in file into the Rhino environment.
- Please read the [License Agreement](#) before using the software.

Introduction

VTK Export for Rhino is a Visualization Toolkit VTK file export plug-in for Rhino. This plug-in gives Rhino the ability to export geometric data from Rhino to ASCII and Binary VTK files.

VTK Export for Rhino extracts facet data from solids, surfaces and meshes in the active Rhino model and exports them to polygon mesh objects in a VTK file.

VTK Export for Rhino is designed to work with the following versions of Rhino:

- VTKExport_VC80x64.rhp -> Rhino 5.0 (64 Bit)
- VTKExport_VC80.rhp -> Rhino 4.0, 5.0 (32 Bit)
- VTKExport_VC60.rhp -> Rhino 3.0

VTK Export for Rhino is designed to be extremely user friendly and easy to understand. Once installed, it automatically loads itself into the Rhino environment. If automatic loading fails there are manual loading instructions in the ReadMe file. Once loaded it adds new commands to Rhino's existing commands and a new submenu called "VTKExport" to the Rhino menu containing these newly added commands.

VTK Export for Rhino comes with detailed documentation as well as a [tutorial](#) to get you started in the shortest possible time.

If you need any additional information please [contact](#) SYCODE.

The VTK File Format

The Visualization ToolKit (VTK) is an open source, freely available software system for 3D computer graphics, image processing, and visualization used by thousands of researchers and developers around the world. VTK consists of a C++ class library, and several interpreted interface layers including Tcl/Tk, Java, and Python. The Toolkit stores geometric data in a file format known as VTK. Very few CAD applications have the ability to read and write VTK files.

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ReadMe

This is the *VTK Export for Rhino* ReadMe file.

VTK Export for Rhino is a plug-in for Rhinoceros. VTK Export for Rhino adds several new commands to Rhino. It also adds a new menu to the Rhino menu called "VTKExport" which contains these newly added commands.

Automatic Loading:

=====

VTK Export for Rhino has to be loaded into the Rhino before it can be used. The installer does this automatically. The next time you run Rhino you will see the "VTKExport" menu added to the Rhino menu. Due to an installation problem, if the "VTKExport" menu does not appear then you will have to manually load the plug-in into Rhino.

Manual Loading:

=====

The VTK Export for Rhino plug-in files are installed into the VTK Export for Rhino installation folder (typically "C:\Program Files\SYCODE\VTK Export for Rhino\"). The plug-in files have an extension of ".rhp". There is one plug-in file for each version of Rhino. The plug-in file name indicates the version of Rhino that it has been designed for.

The list of plug-in files along with the version of Rhino is shown below. Please be sure to use the plug-in file that corresponds to your version of Rhino.

VTKExport_VC60.rhp -> Rhino 3.0
VTKExport_VC80.rhp -> Rhino 4.0, 5.0 (32 Bit)
VTKExport_VC80x64.rhp -> Rhino 5.0 (64 Bit)

To load VTK Export for Rhino into Rhino please follow the steps listed below:

- (1) Start Rhino
- (2) Type "PlugInManager" at the command prompt and press "Enter"
- (3) Click the "Install" button.
- (4) Browse for the plug-in file for your version of Rhino in the VTK Export for Rhino installation folder.
- (5) Click the "Open" button.
- (6) Click the "Close" button.

Usage:

=====

VTK Export for Rhino adds a new menu to Rhino called "VTKExport". All the VTK Export for Rhino commands can be invoked from this new menu.

Support:

=====

For support, please E-Mail support@sycode.com

Contact

| | |
|-----------------|--|
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| Fax | +91 832 2416615 |
| E-Mail | info@sycode.com |
| Web Site | http://www.sycode.com |

Command Reference

VTK Export for Rhino adds new commands to Rhino and a submenu called "VTKExport" which houses them. The new commands are:

| | |
|-----------------------------------|--|
| VTKExport | Exports a VTK file |
| VTKExportHelp | Displays the <i>VTK Export for Rhino</i> help file |
| VTKExportRegister | Registers your copy of <i>VTK Export for Rhino</i> |
| VTKExportAbout | Displays the <i>VTK Export for Rhino</i> about box |

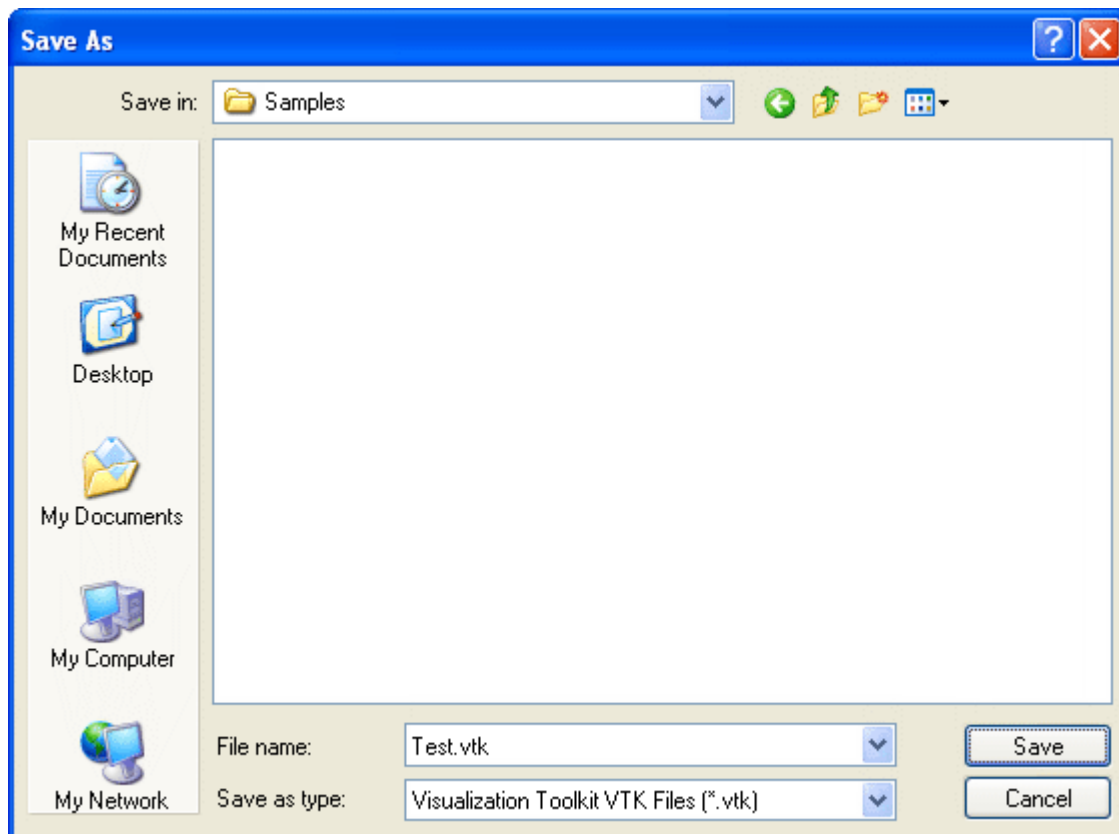
VTKExport

| | |
|--------------------|--------------------|
| Menu Button | VTKExport > Export |
| Command | VTKExport |

This command exports geometric data from AutoCAD to ASCII and Binary VTK files.

To export a VTK file

1. Select *Export* from the "VTKExport" menu or type *VTKExport* at the Rhino command prompt and press *Enter*.
2. At the *Select objects* prompt, select the objects that you wish to export. Press *Enter* when done.
3. The Rhino "Save As" dialog box is displayed. Enter the name of a VTK file and click *Save*.



4. The "VTK Export Options" dialog box is displayed. Set the options as required and click *OK*.

VTKExportHelp

| | |
|--------------------|------------------|
| Menu Button | VTKExport > Help |
| Command | VTKExportHelp |

This command displays the help file. The [Welcome](#) page is a good place to start.

VTKExportRegister

| | |
|--------------------|----------------------|
| Menu Button | VTKExport > Register |
| Command | VTKExportRegister |

This command is used to register your copy of *VTK Export for Rhino*.

You need a license key to start using the software. A license key can be of two types: Trial or Permanent. A trial key is issued to you instantly and completely free of cost so that you may try the software before you purchase a license. A permanent key is issued after you have purchased a license.

How do I get a Trial Key?

Simply click the 'Request Key' button and select 'TRIAL KEY' as the type of key being requested. Fill the form and submit it. A trial key will be immediately sent to you by email.

How do I get a Permanent Key?

To get a permanent key you need to have already purchased a license. If you have not yet purchased a license you can do so by clicking the 'Buy Online' button. After you place your order click the 'Request Key' button and select 'PERMANENT KEY' as the type of key being requested. Fill the form and submit it. Your order will be verified and a permanent key will be sent to you by email within one business day.

IMPORTANT:

(1) Please ensure that your computer is connected to the internet before you request for a key.

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- (2) Please be sure to enter a valid email address as the key will be sent to you by email.
- (3) To avoid disruption in your work-flow we urge you to place your order before the trial period ends.

If you need anything else please send an email to register@sycode.com

VTKExportAbout

| | |
|--------------------|-------------------|
| Menu Button | VTKExport > About |
| Command | VTKExportAbout |

This command is used to display the *VTK Export for Rhino* About box. The About box lists the software and its version



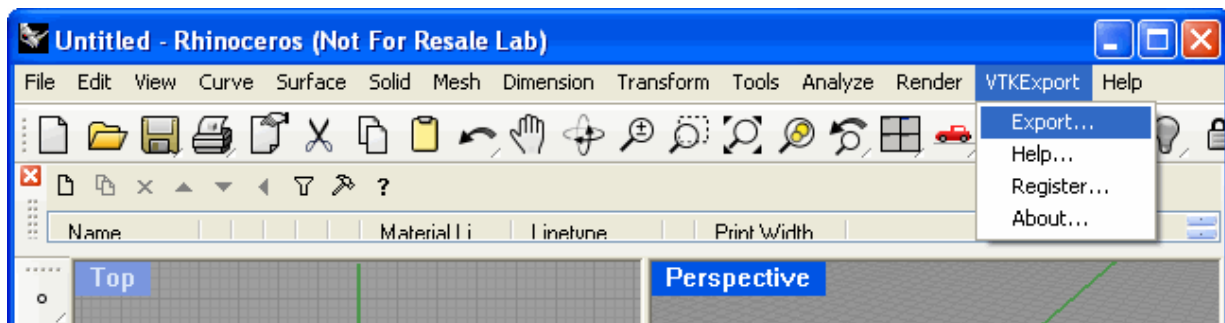
Tutorial

This tutorial provides the necessary information and hands-on experience to help you get started with *VTK Export for Rhino*.

Step 1: Start Rhino.

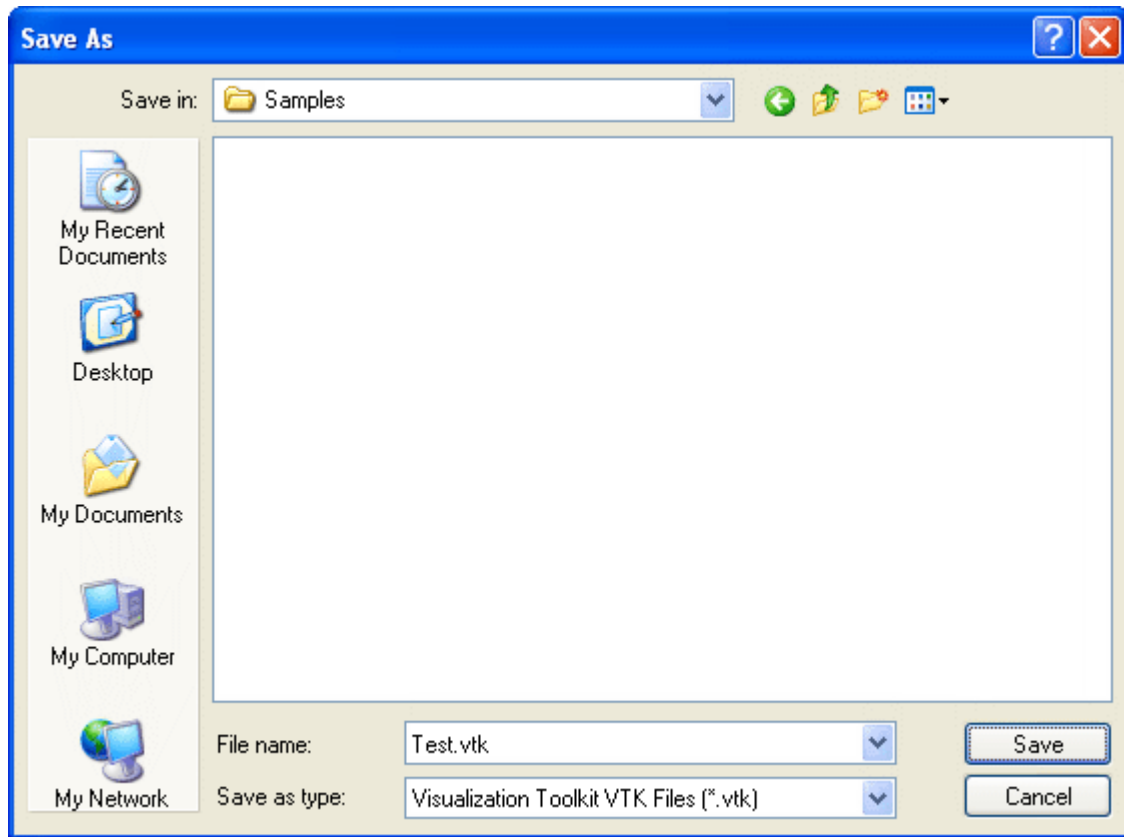
Step 2: Open the file. The Rhino "Open" dialog box is displayed. Select the file C:\Program Files\SYCODE\VTK Export for Rhino\Samples\Sample 1.3dm and click *Open*.

Step 3: Select *Export* from the "VTKExport" menu or type *VTKExport* at the Rhino command prompt and press *Enter*.

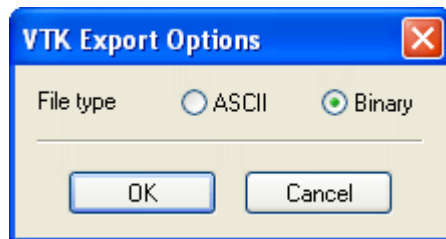


Step 4: At the Rhino "Select objects" prompt, select the objects you wish to export. Press *Enter* when done.

Step 5: The Rhino "Save As" dialog box is displayed. Enter the name of a VTK file and click *Save*.



Step 6: The "VTK Export for Rhino Options" dialog box is displayed. Set the options as required and click *OK*.



The objects will be exported to the VTK file you specified.