ZSTools

Version 1.0.0.2 (July 9th 2019)

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Introduction

Overview

ZSTools is a toolbox of functions for Rhino. It is designed to enhance productivity of meshing by enabling interaction between Rhino and ZSoil Preprocessor.

ZSTools allows objects created in Rhino 6 to be exported to ZSoil .inp file format. To this end the Rhino objects need to be selected and the appropriate Rhino-command has to be called.

ZSTools has two main functionalities:

- *Exporting geometric objects*: Points and curves can be exported to ZSoil Preprocessor as geometric objects.
- *Exporting finite elements*: Geometric objects can be exported to ZSoil Preprocessor directly as finite elements. This step requires definition of the type of finite element prior to exporting, as surfaces and lines can be the basic shape of several types of elements: Shells, membranes and 2D continuum elements for 4-noded surface objects, and beams or trusses for curve objects. Optionally, finite element properties such as material, existence and load functions and replacement material numbers can be defined in Rhino and exported to ZSoil as properties attached to the finite elements.

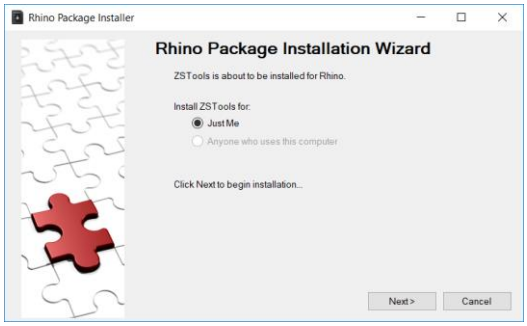
Features

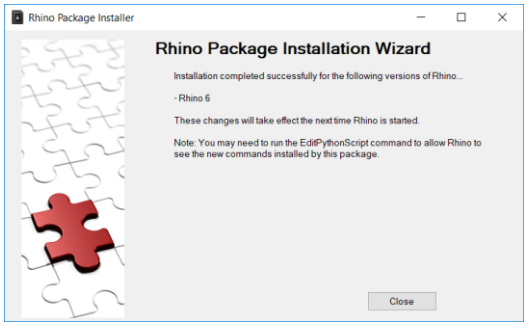
ZSTools includes the following functions:

ExportGeomObjects	Exporting Points and Lines to ZSoil .inp file format
ExportElements	Exporting Finite Elements with attached properties to ZSoil .inp file format
SetElementTypes	Setting finite element types of geometric shapes (Volumics, Shells, Membranes, Volumics2D, Trusses and Beams)
SetObjectProperties	Setting material, existence, load function and replacement material numbers for selected finite elements and thickness for single layer shells

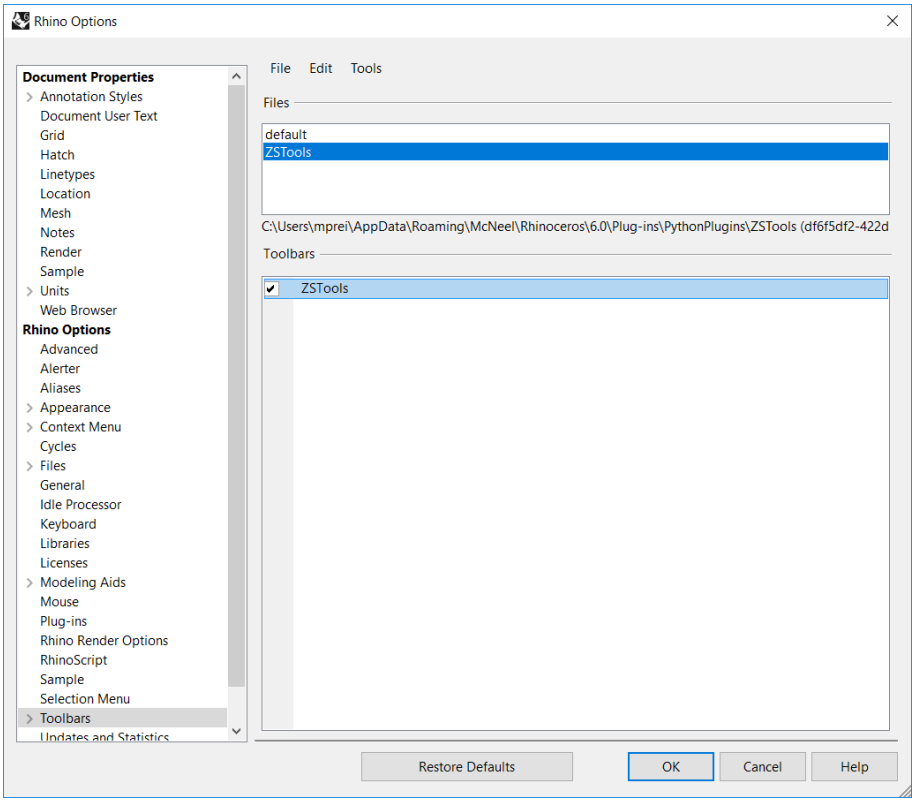
Installation

1. Double-click the file ZSTools.rhi or drop it into an open Rhino window.
2. Follow the instructions of the installation dialog:

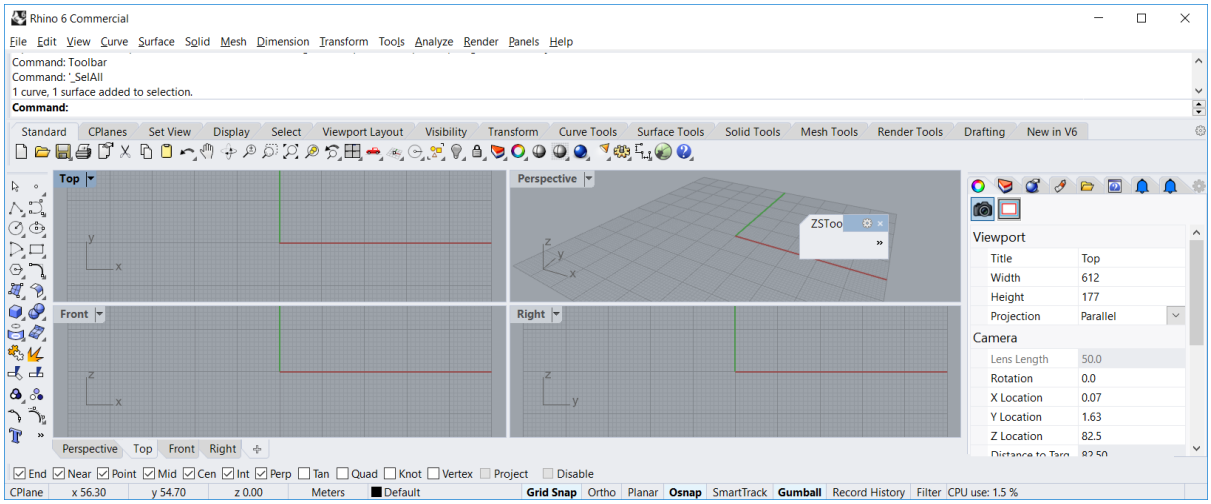


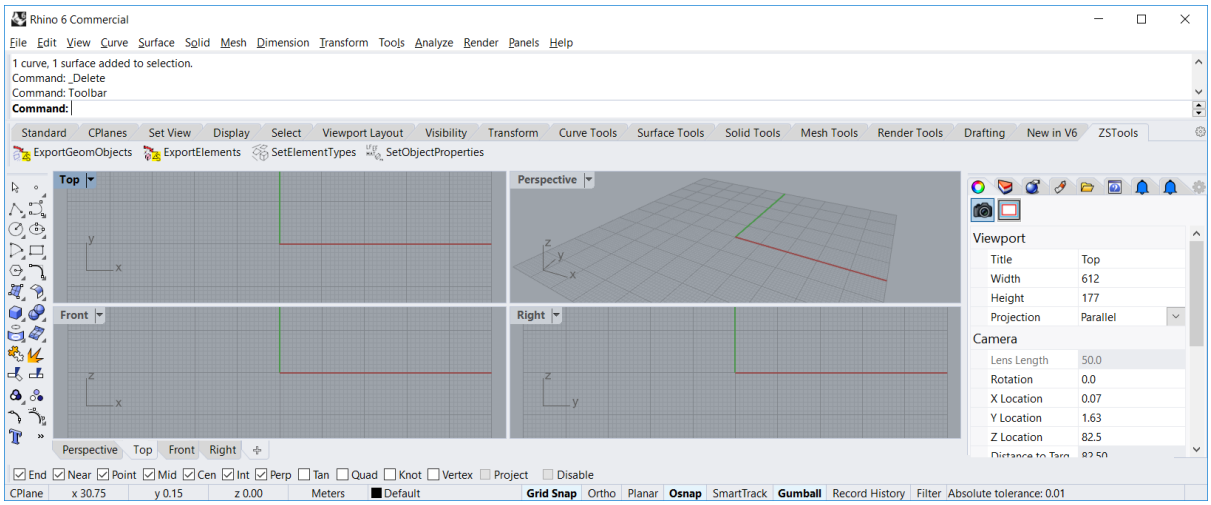


3. Open the folder %AppData%/Roaming/McNeel/Rhinoceros/6.0/Plug-ins/PythonPlugins/ZSTools (xxx..)/version number
4. Grab the file ZSTools.rui with the mouse pointer and drop it into the open Rhino window.
5. Type Toolbar into the command line.
6. Under “Files” there should be an item “ZSTools”. Select it and check the box next to ZSTools under “Toolbars”. Close the dialog by clicking OK:



7. You can grab the ZSTools toolbar that has appeared on the screen by clicking on the gray tab and slide it onto the standard toolbar above the drawing canvas:



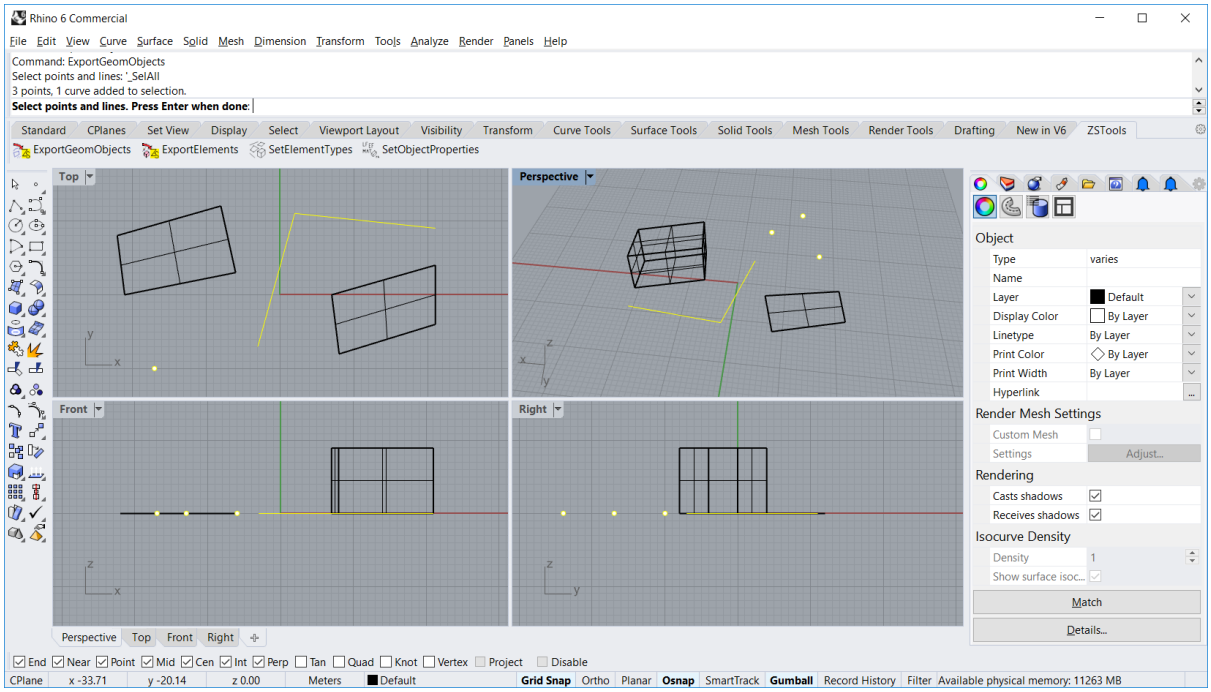


8. Type EditPythonScript into the command line. Close the window immediately.
9. Close Rhino and restart it.
10. The ZSTools commands should now be accessible by command line or by clicking on the ZSTools toolbar.

Command reference

ExportGeomObjects

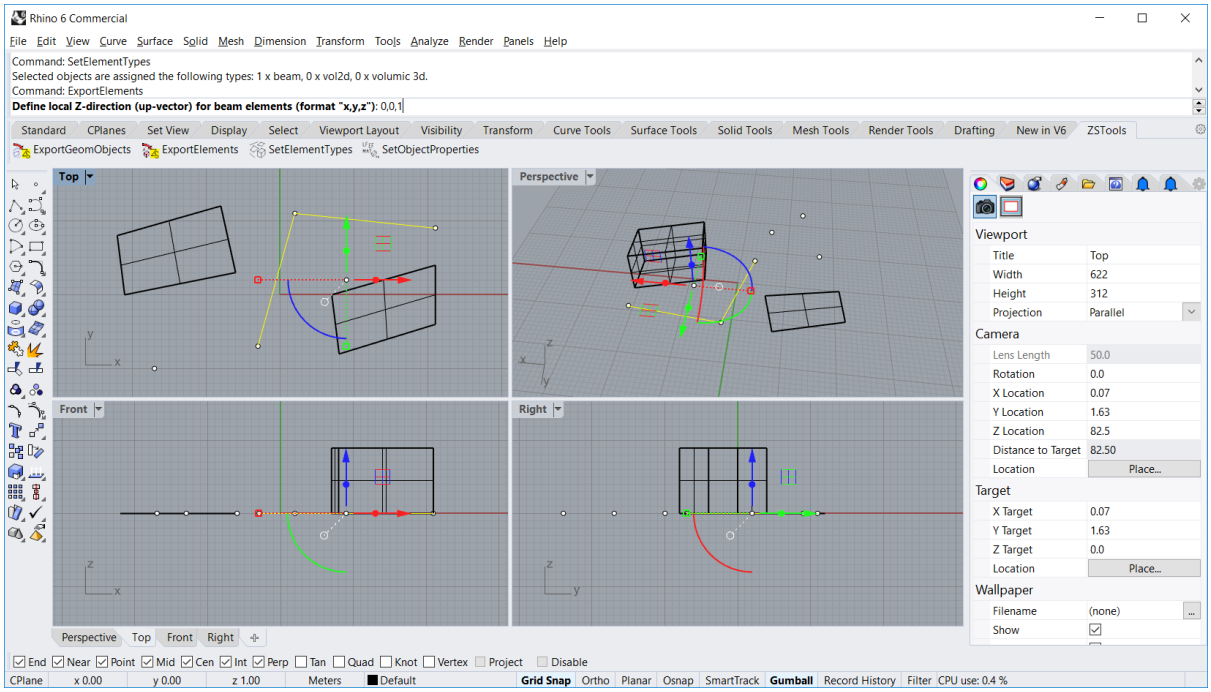
1. Draw points and curves in Rhino, using the method of your choice (direct drawing on canvas or by intersection of objects, projection, duplication etc.). Straight lines and splines are supported.
2. Select the geometric objects you wish to export.
3. Type the command *ExportGeomObjects* into the Rhino command line.



4. “Convert Splines to Polylines?” Accept by pressing Enter or change by typing C or clicking on the option in the command line.
    a. Do NOT Convert to Polylines: The curves will be exported as cubic splines to ZSoil. Note: Currently ZSoil .inp file format requires splines to be defined by fit points. Splines are exported with with 4 fit points as cubic splines. Exporting higher order splines is currently not supported.
    b. Do Convert to Polylines: The user is prompted to give the number of equidistant subdivisions of each spline. These points are connected by a Polyline, that is exported to .inp.
5. “Enter file name:” Enter a file name for exporting. The file will be saved to the working folder. In order to change the working folder, call the Rhino command *SetWorkingFolder* prior to exporting.
6. Open the .inp file in ZSoil 2018 or higher
7. Optional: ZSoil object labels can be created in Rhino with the command *SetUserText*. After typing the command into the command line, type “name” as key and click enter, then enter the object label (if name contains whitespace, include it within quotation marks). The object name for lines can be seen in the .inp file only, ZSoil Preprocessor currently does not allow to access it.

ExportElements

1. Draw curves, quadrangular surfaces and hexahedral polysurfaces or extrusions.
2. Assign element types (see *SetElementTypes*).
3. Optional: Assign element properties (see *SetObjectProperties*).
4. Select objects you wish to export as finite elements.
5. Type the command *ExportElements* into the command line.
6. For Beam elements only: The user is prompted to enter a direction vector indicating upward direction of the beam section (Z-axis). The direction has to be entered in the command line, with the 3 components separated by commas:



7. “Enter file name:” Enter a file name for exporting. The file will be saved to the working filder. In orderto change the working folder, call the Rhino command *SetWorkingFolder* prior to exporting.
8. Open the .inp file in ZSoil 2018 or higher.

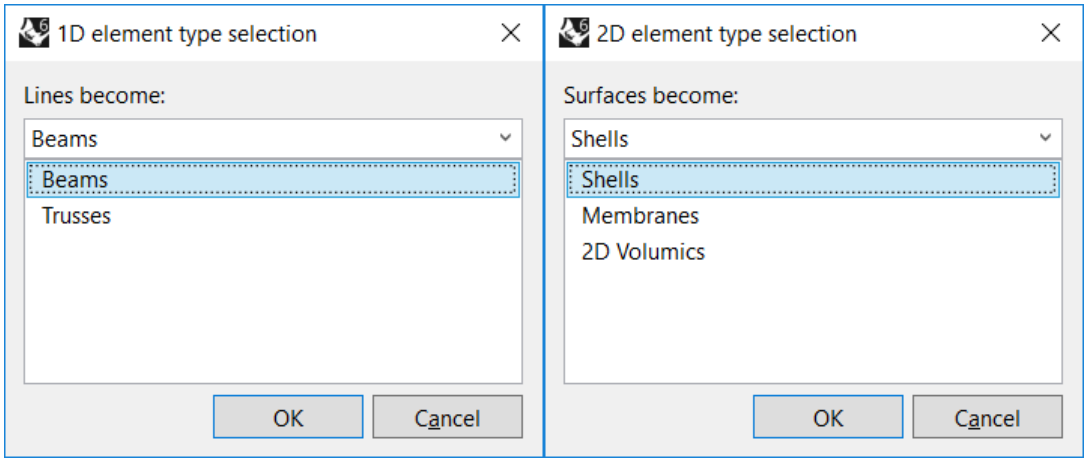
Remarks:

- All lines are exported as straight beams or trusses, connecting their end points with one segment. In order to discretize curves such as arcs and splines they must be split into several segments in Rhino.
- The command supports 4-node elements for surface objects and 8-node elements for polysurfaces. When trying to export surfaces or polysurfaces with a different number of corners, elements might be invalid.

SetElementTypes

Prior to exporting geometric shapes to .inp the objects must be assigned an element type. This step is required as objects such as lines and quadrangular surfaces can be interpreted as more than one element type: Lines can be either trusses or beams, quadrangular surfaces can be 2D continuum elements (volumics), membranes or single layer shells.

1. Select objects you wish to assign an element type.
2. Type the command *SetElementTypes* into the command line.
3. For lines and surfaces select the element type from the drop-down list. For hexahedras the element type 3D volumic is assigned automatically:



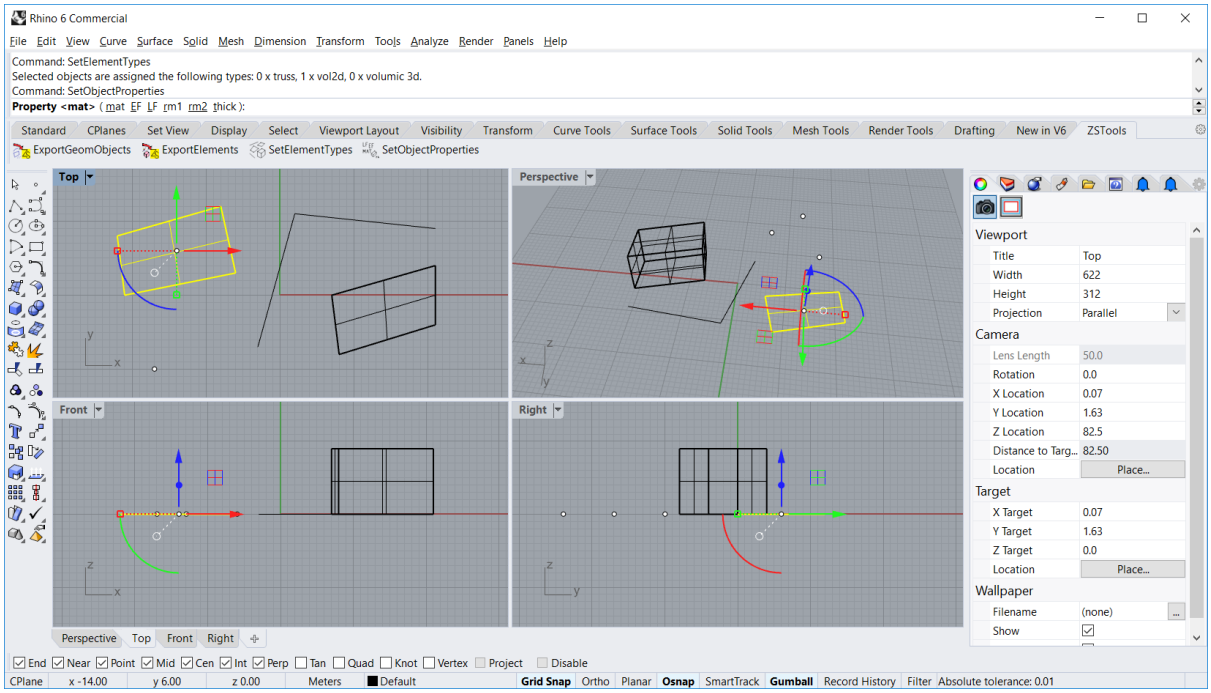
SetObjectProperties

The following finite element properties can be assigned to Rhino objects:

- Material number (mat)
- Existence function number (EF)
- Load/Unloading function number (LF)
- Replacement material 1 (rm1)
- Replacement material 2 (rm2)
- Thickness (single layer shells only) (thick)

The properties are saved as Attribute User Text in Rhino.

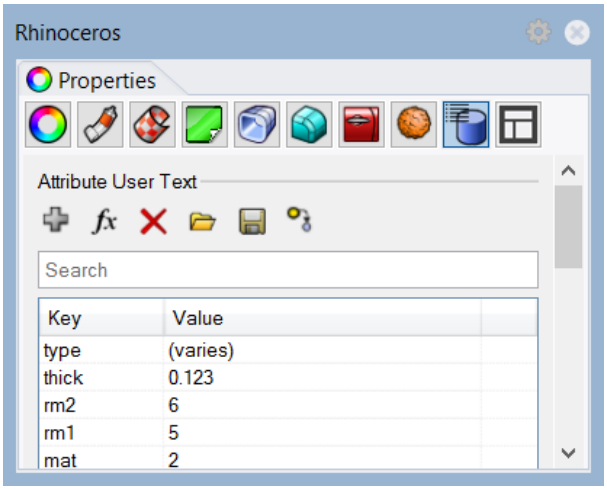
1. Select objects you wish to assign object properties.
2. Type the command *SetObjectProperties* into the command line.
3. Select the property you wish to assign from the options shown in the command line (by clicking or by typing the underlined letters):



4. Type the value for the chosen property and press Enter.

Remarks:

- Object properties can be inspected for a selected object in the Attribute User Text page.



- Several standard Rhino commands can be useful for handling Attribute User Text items:
  - *SelKey*: Select objects by User Text key (e.g. selecting all objects on which a property has been defined, by typing for instance “EF”).
  - *SelValue*: Select objects by User Text value (e.g. selecting all objects with any property equal to “2”).
  - *SelKeyValue*: Select objects by User Text key and value (e.g. selecting all objects with Existence function equal to “2”).