RhinoBIM Structure

RhinoBIM Structure is one tool in the suite of RhinoBIM products in development for architects, designers and engineers to conceptualize, refine, analyze and release structural design models built within the 3D Rhinoceros workspace.

The key implementation of RhinoBIM Structure is its capability of building large design models with a very small modeling data size. RhinoBIM Structure incorporates the use of a Light Weight Extrusion element new to Rhino V5 allowing users to build larger models than ever before.

RhinoBIM Structure program has functionality to BUILD and EDIT structural members that are selected from an extensive library of international standards and catalogs. Attached to structural members are various materials, attributes, and process treatments that are associated with the database. This is available for extracting to BIM reports for downstream applications, including structural analysis, detailing, cost analysis, scheduling, fabrication and staging.

Along with the associated libraries, RhinoBIM Structure has the ability for users to add their own cross sectional shapes into a user defined library. RhinoBIM can export and import structural libraries to coordinate with multiple users within a project.

RhinoBIM has a Clash and Clear function that can detect a hard clash or a soft clearance of a set of selected Rhino entities against another set of entities. The clearance function allows the user to specify a clearance zone offset from the selected structural members and then detect any soft clashes that occur within that zone. This function is useful for allowing a design criteria of clearance around the structural members for such treatments as fireproofing on specified members. Images in jpg format can be created for each interference issue found and stored with generated reports. With this analysis there is no need to export the geometry to another program to perform this function. As clash and clearance issues are detected within a design model, the fix can be performed within the same working model at that moment.

With RhinoBIM a project team can build a Data Set of alpha numeric information. Users can then attach the information to any Rhino entity within a model and generate reports of the Rhino entities of the attached information and the geometry it is associated with.

RhinoBIM Structure has industry standard and IFC translators available for importing and exporting structural models to other design packages as well as downstream detailing and fabrication work.
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Start RhinoBIM Structure by selecting the icon in the toolbar command line or by entering the command RhinoBIM_STR

RhinoBIM Structure main menu has five function tabs:

BUILD, EDIT, ANALYSIS, MANAGE and HELP.

Tab selection follows the design process flow to BUILD the structural frame, EDIT existing structural members to refine the model, ANALYSIS for detecting clashes and clearances and generate quantity take off information, MANAGE for adding user defined profiles, create a user defined information data base and to release the design transferring to detailing and fabrication. The HELP tab directs the user to manuals and information.

BUILD function builds structural members within Rhino 3D modeling space. Select the standard library you will use – Imperial or Metric, select from the standard catalog, structural shape, size and treatment if necessary.

User selects a standard

ANSI Imperial or ISO metric

Catalog:

AISC

Shape:

Wide Flange W, S, HP, M
Wide Flange Tee WT, ST, MT
Channel C, MC
Angles L, 2L
Pipes
Tubs
Plate

Specific size

Treatment

Or Match existing member in the model

The structural members can have treatments assigned to each individual member. This information is used in BIM reports for total cost estimating.

Galvanized
Fireproof
Painted
Untreated

Add your own treatments with Manage / Create User Profiles function.
Highlight Members function highlights all instances of the member in the model that match the selection list under the arrow at the menu’s side. All Data would list what is in the Geometric Parameters display menu. All other selections such as, Finish, Shape Parameters and Orientation Parameters would Highlight only that information that is in the Geometric Parameters display menu.

Change Parameters function allows you to change any parameter of any structural members in the model.

First set the new desired parameters in the database parameters menu. Then select the structural members in the model. After selecting the members, select the Change Parameters button.

Multi-selection options can be used with this function by selecting arrow next to the menu button for the options.

Search by ID displays the unique identifier name of the structural member and will highlight it’s locate within the model.

Select a member in the model and it will display the ID name.

Enter an ID number for any known member in the menu and select the Search by ID button to highlight that member in the model.

Information Tool Tip displays all the attribute information about any structural member when the tool tip is turned on and the cursor is held over the member. This is a great function for auditing a design. Hold the tool tip over a member, if it is correct select it and then hide it.

There are four methods for placing a structural member within the model space: PT to PT, Lines, Curves and Surface. Each function allows you to place a structural shape; from a point to point, on lines in the model, on curves in the model or on a surface oriented with its vector.

Line and curve elements within the Rhino model can be selected using standard Rhino selection methods. Another method for selecting multiple lines or curves at one time is to select the arrow within the function buttons to display multi-selection options: by color, by layer or by group.

Point to Point builds structural members within the model by selecting or indicating a point location for the first end of a member and a second point for the other end. Continue selecting any two points within the model space until you press Spacebar or Enter Key to end the routine.

In the command line are the following options:
**Vertical mode** builds vertical columns from first point selected.

- Select **PT to PT** function
- Select first point in the model
- Select **Vertical** option command line
- Key value for length or drag to desired length

**Chain mode** builds a chain of consecutive structure members from last point selected.

- Select **PT to PT** function
- Select first point in the model
- Select (Chain) in command line or Enter C on keyboard to begin **Chain mode**

When finished with **Chain mode**, press Spacebar or Enter Key to return to the **PT to PT** mode

**Miter** builds a chain of consecutive structure members and trims the connections with miters

**Lines** function allows a structural member to be placed on lines within the model space.

- There is no limit to the number of structural members that can be placed on lines at one time.
- Select lines within the model to place structural members on
- Select **Lines** button
  - or
- Select **Lines** button first
- Select lines within the model
- Press Spacebar or Enter key

**Curves** function acts the same as **Lines** function but works only for curved members.

When selecting lines or curves in the model space, RhinoBIM Structure can detect the difference between a line, a planer curve or a non-planer curve.

- If a planer curve is detected, the profile is oriented with the flanges of the structure parallel to the plane of the curve.
- If a non-planer curve is detected, the program prompts to: **Pick orientation direction** of the profile. Pick in space and drag the orientation line or select an end of a line to align the profile.

**Curves** function allows a structural member to be placed on multiple curves at one time. There is no limit to the number of curve selections at one time.
Select **Curves** button within the model to place structural members on

or

Select Curves within the model

Press **Spacebar** or **Enter key**

**Surface** function allows for placing multiple structure elements on one planar surface at one time, and orienting the structure 90 degrees to the surface. This is beneficial for placing mullions on glazing.

Select **Surface** button

Select the surface the structure will be placed on

Select the curves on the surface that the structure will be built on

Press **Spacebar** of **Enter Key**

Option to flip the structure       **Flip beam’s normals (Yes No)**
The **Surface** function can be used to sweep a profile along a non-planar curve that lies on a compound surface. The profile orientation is controlled by the surface normals along the curve.

Select **Surface** button

Select the controlling surface

Select the curve that lies on the surface.

Press **Spacebar** or **Enter Key**

Option to flip the structure — **Flip beam’s normals (Yes No)**

After creating structural members with the **PT to PT**, **Lines** or **Curves** functions you can flip or rotate members by selecting functions in the command line.

**Press Enter when done (Flip EnterAngle Rotate90 ByGeometry Gumball)**

- Selecting **Flip** mirrors C channels, L angles and user defined shapes about their axis
- Selecting **EnterAngle** to enter a specific angle of rotation
- Selecting **Rotate90** rotates the member in 90 degree increments
- Selecting **ByGeometry** selects two lines that defines the angle
- Selecting **Gumball** displays an arc. Select and drag to the desired angle

The amount of rotation is displayed in lower left corner of the Rhino window.

There is a new **Osnap** developed for RhinoBIM structure members only, called **vertex**. This **Osnap** is located at the end points of structural member’s hang points.
Cursor **Tool Tip** displays a window near the tip of the cursor listing the parameters of the structure member. This function helps to audit the structure model.

To turn this function on:

Select the arrow to the right of the **Information Tool Tip** button

Select ON
EDIT tab has all functionality to make changes to structural members after they are built. Submenus within EDIT are Attributes and Members.

All standard Rhino transformation functions and the Gumball work on all RhinoBIM structural members as well.

Drag End can extend or retract the length of selected members. Arrow option for multi-selection can select multiple members.

Select Drag End button
Press Spacebar or Enter Key
Pick end of members to be stretched
Use cursor to drag to desired length

Length amount is displayed in lower left corner of the Rhino window.

You may key in a specific length before dragging the cursor. Then drag the cursor to either increase or decrease the length of the member to that amount.

To Geometry stretches selected members to selected geometry.

Select To Geometry button
Press Spacebar or Enter Key
Select intersecting geometry

Real Time Rotate rotates selected structural members to a desired angle.

Select Real Time Rotate button
Select members to rotate
Press Spacebar or Enter Key
Select the arc and drag rotating member to desired angle of rotation.

The degrees of rotation is displayed in lower left corner of the Rhino window.
**By Geometry Selection** rotates selected structural members to a desired angle by selecting two lines that define desired angle.

- Select **By Geometry Selection** button
- Select members to rotate
- Press Spacebar or Enter Key
- You are prompted to pick a reference object that defines the first line that describes angle
- Select second line to define angle

**Flip** flips flanges of a member about its axis. This is used in case of C channels, L angles and user defined profiles to invert profile.

**Enter Angle** allows you to key in a specific angle to rotate a structural member.

- Key in number of degrees to rotate
- Select **Enter Angle** button
- Select members in model to rotate
- Press Spacebar or Enter Key

**Move Node** moves selected node points of the structural members.

- Select **Move Node** button
- Select members to move
- Press Spacebar or Enter Key

A popup menu gives two options: **By End Point** or **By Axis**

- **By End Point** - moves the node from one point to another point
  - Pick reference point to begin move and pick point to move to.

- **By Axis** – moves the node along an axis or on a plane.
  - Select one of the axis vectors or one of the plane symbols and drag the node.
  - or
  - Key a value in the window then select a vector or plane to drag the node.
From Planer Surface can trim selected structural members to planer surface.

- Select **From Planar Surface** button
- Select members to trim
- Press Spacebar or Enter Key
- Select a planer cutting surface
- Select a point on the side of the structural member to trim away

Plane from three points can trim selected structural members to three points.

- Select function **Plane from Three Points**
- Select members to be trimmed
- Press Spacebar or Enter Key
- Select three points that define the cutting plane
- Select a point on the side of the structural member to trim away

Plane from normal determines a cutting plane that is normal for a line.

- Select **Plane from Normal**
- Select members to trim
- Press Spacebar or Enter Key
- Select a point
- Select end point to define a line that is normal to the cutting plane
- Select a point to which side of the structural member to trim away

Miter miters two members to each other.

- Select **Miter** button
- Select end of first member to miter
- Select end of second member to miter
Copy Attributes copies attributes of a curved member to a single curve or multiple curves.

Select curves to be copied to

Press **Spacebar** or **Enter key**

Select curved structural member to copy its attributes
**Clash and Clear** compares two sets of geometry looking for clashes and clearances between any two objects in each set. Specifying a clearance value checks where objects protrude within the defined clearance zone.

The program generates and displays a report listing hard clashes and soft clearances. Within the display window you can select an incident and ask to zoom into the selected incident and take a snapshot image of the incident.

Select objects for the first set
Select objects for the second set


To check with a clearance zone enter a value

A value of 0 in the clearance checks for hard clashes only
Select Calculate button

Once the program has completed the analysis, a window displays a report listing each incident of Hard Clash or Soft Clear.

In a Soft Clear case the report lists the distance the two objects actually clear.

The report also lists the unique Identifier of the element name in each set.
Selecting an incident number will highlight the elements involved.

Zoom into the location of the model where the incident occurs and save a snapshot image.

Right click incident number in the menu

Select Zoom to
Rotate view for better image.

Right Click Select Snapshot

To easily identify the saved images the name of the image defaults to the incident number, if it's a Hard Crash or Soft Clear, the amount of clearance and Unique Identifier name of the two objects.

For example 14 Soft Clear 1.01 11 17.jpg

Selecting Export will create an excel spread sheet of the entire report.
**Quantity Take Off** is found under the **Analysis** tab. This function creates a report listing the length, surface area, volume and weight of structural objects or surface objects selected. Each selection creates a set that can be saved and then loaded back in RhinoBIM and have new structural members appended to the set to generate a report.

Click the **Select Object** button or select the right arrow for the pull down menu for other selection options.

Press the **Return** button or **Spacebar** to accept the selection. Select the **Complete Report** button to compute the report.

Select a member listed in the report and press the right mouse button. This will bring up a pop up menu that allows to **Zoom** to that member and then take a **Snapshot** jpg image.
Select **Export** to save an xml spread sheet.

```
Name | Area (sq. centimeters) | Length (centimeters) | Volume (cubic centimeters)
STR01 | 16641.75 | 130.05 | 7326.83
```

Select **Export** to save an excel spread sheet

```
<table>
<thead>
<tr>
<th>Name</th>
<th>Material</th>
<th>Finish</th>
<th>Shape Size</th>
<th>Length (sq. centimeters)</th>
<th>Area (centimeters)</th>
<th>Volume</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>STR126</td>
<td>AISC</td>
<td>Painted</td>
<td>W12x33</td>
<td>6334.05</td>
<td>340.63</td>
<td>1590.01</td>
<td>1590.01</td>
</tr>
<tr>
<td>STR128</td>
<td>AISC</td>
<td>Painted</td>
<td>W14x22</td>
<td>69953.8</td>
<td>932.16</td>
<td>2200.05</td>
<td>2200.05</td>
</tr>
<tr>
<td>STR189</td>
<td>AISC</td>
<td>Painted</td>
<td>W14x22</td>
<td>38713.34</td>
<td>803.72</td>
<td>17110.5</td>
<td>17110.5</td>
</tr>
</tbody>
</table>

Total: 120071.79  1816.51  58398.56  58398.56
```

The **Save selection** button is also found in the main menu. Selecting the Load selection loads previously saved report sets.
**Create User Profiles** function creates new profile shapes within a library. You can create new Standards, Catalogs, Shapes, Sizes and Treatments. You can also delete any items.

Each item (Standard, Catalog, Shape, Size, Finish) has the option to create a new or delete.

To change the name of an item

- Select the arrow to the right of the named button
- Select the name to edit
- Key in a new name in the command line

To **Create** a new item

- Select the far right arrow next to the named button and select Create
- Key the name of the new item

To **Delete** an existing item

- Select the arrow to the far right of the named button
- Select the name to be deleted
- Select the far right arrow and select Delete
- Select Yes in the pop up window
Creating a custom user profile.

Create the custom profile geometry in the Rhino modeling space.

Select the far right arrow next to the Size button and select Create.
If the profile will be stored in a new Standard, Catalog or Shape name, make sure these are created before.

Key in the name for the custom profile in the command line

Select the outer profile geometry

Press Spacebar or Enter key

Select the inner profile geometry if it applies

Press Spacebar or Enter key

If you want to add analysis parameters type or select Yes or No

User created library files can be exported and imported to coordinate multiple users on a project.
BIM Reports generates a listing of the structural members selected in the model.

Select structural objects
Select BIM Report button

Report information is an excel spreadsheet ready to be saved.

CIS/2 translates out structural members selected in the model into the CIS/2 format.

Select structural members
Select CIS/2 button under the RhinoBIM Manage tab

The translated information is saved in a user specified file
Translation back into Rhino is under development

IFC translates out structural members selected in the model into the IFC 2X3 format.

Select the structural members
Use the Rhino File function and select the option Export Selected

In the next menu enter the file name
In the File Type select the IFC file type
Select the Save button in the menu
Convert to NURBS function converts selected structural members into joined NURBS surface entities. It allows users to save in a Rhino 3dm format or translate to other formats that support NURBS such as STEP, DWG or IGES.

Select structural members

Select **Convert to NURBS** button under the RhinoBIM Manage tab

In Save As window select file format type

Enter a file name
User Data function allows for the creation of a database of any information a project requires. The information can then be attached to any Rhino geometry in a model. The function is found under the Manage tab.

Building a User Data Base

Select Create/Edit button

In the pop up window double click on New Dataset

In the pop up window key in the Data set name

Select OK or press the Return Key
In the pop up window select the **Add** button to add information in the data set that will listed under the data set name.

The **Remove** button removes column names.

To attach information to Rhino entities

Select **Attach** button

Select any Rhino entity use the Rhino Edit Select

Or

Select the arrow at the far right of the **Attach** Button

Select by **Color, Layer** or **Group**

Select a Rhino entity in that **Color, Layer** or **Group**
A pop up window displays the information and a list of the entities it is attached to

To generate an excel spread sheet report

Select **Report** button

Select any Rhino entity or use the **Rhino / Edit / Select Objects**

Or

Select the arrow at the far right of the **Attach** Button

Select by **Color, Layer** or **Group**

Select a Rhino entity in that **Color, Layer** or **Group**