



GRASSHOPPER TO AXISVM ADD-ON VERSION 5.0

USER'S MANUAL

AXISVM X6R2* / RHINO 6 OR 7 FOR WINDOWS

MARCH 2022

DOCUMENT REVISION 5.0.1

*) Restricted functionality is available with versions AxisVM X5R4 – X6R1.
For full functionality use AxisVM X6R2.

Copyright © 1991-2022 Inter-CAD Kft. of Hungary. All rights reserved. No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording or otherwise, for any purposes.

This page is intentionally blank.

TABLE OF CONTENTS

1. INSTALLATION GUIDE.....	5
2. UPGRADE PROCESS (FROM V4.1 / V4.2 TO V5.0)	6
3. COMPONENTS.....	7
31. INPUTS	7
32. PARAMS	14
33. ELEMENTS	15
34. LOADS	31
35. SEND.....	40
4. KNOWN ISSUES	42

This page is intentionally blank.

1. INSTALLATION GUIDE

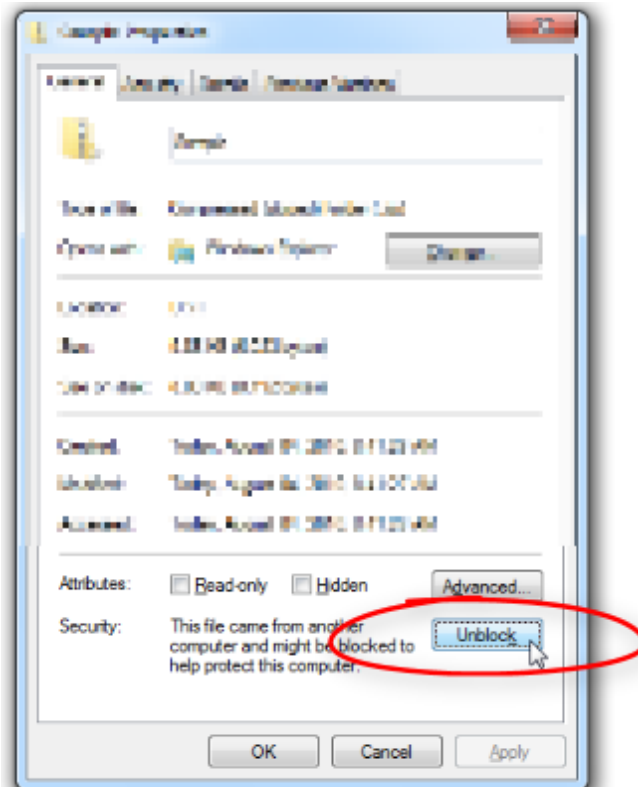
The GrasshopperToAxisVM add-on is compatible with Rhino 6 and Rhino7 as well.

AxisVM X5R4 or newer version is supported.

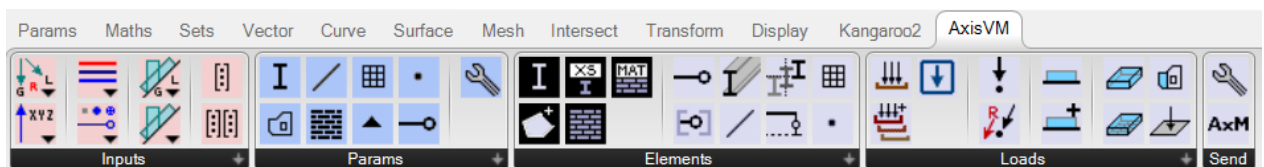
The add-on works only on Windows operating system.

The Grasshopper Components folder must contain one single version of this add-on, otherwise it will not work.

- 1) Download the latest *.gha file from the food4rhino.com website.
- 2) Open the folder where the file is saved. Right-click the file, open the File properties and (if required) check the Unblock option (see image below), then click OK.



- 3) Copy the *.gha file into the GH Components folder (...\AppData\Roaming\Grasshopper\Libraries). You can reach this directly from file browser or via Grasshopper → File menu → Special Folders → Components Folder
- 4) Open the local installation folder of your AxisVM version that you suppose to use together with Grasshopper (e.g. .../Program Files/AxisVM...). Search for the Interop.AxisVM.FW4.dll file.
- 5) Copy the *.dll file beside the *.gha file into the GH Components folder.
- 6) Start (or restart) Rhino and Grasshopper.
- 7) The AxisVM tab must appear at the Component palettes.



Note: after each update of the installed AxisVM software it worth to check the date of modification of the *.dll file and replace the *.dll file in the Grasshopper Components Folder in case of changes.

3. COMPONENTS

The GrasshopperToAxisVM V4.1 and V4.2 offered 16 AxisVM components. The current version, V5.0 introduces numerous new possibilities and consists of 82 AxisVM components in total. These are grouped into five subcategories under the AxisVM tab in Grasshopper: Inputs, Params, Elements, Loads and Send. This section contains the description of each component that is available in these subcategories. The component input and output parameters are listed with information about the type, content and meaning of them. If you need further information or details about the AxisVM functionality, check the AxisVM User's Manual.

3.1. INPUTS

The Inputs menu provides a bunch of dropdown menus (derived from the ValueList component of Grasshopper) that contain some lists of options related to AxisVM components. For example: line types, distribution types etc.

These components are available not only in the Inputs menu but they can be created from the pop-up menu of the components (right-click on the component). In the second way they are going to be immediately connected to the appropriate input parameter.

Note: these inputs are convertible from integer and textual inputs as well.

The components of the subcategory are listed in alphabetic order. The default values are underlined.



Auto Eccentricity Type

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Custom / Top / Mid / Bottom.
Type of auto eccentricity for ribs.
The output is an integer representing the selected item of the list.



Axis

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* X / Y / Z / XX / YY / ZZ.
Directions as value list.
The output is an integer representing the selected item of the list.



Beam Rib Distribution Type

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Length / Projected.
Beam / rib load distribution type.
The output is an integer representing the selected item of the list.



Cross-section Process

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Other / Rolled / Welded / Cold Formed.
Cross-section manufacturing process.
The output is an integer representing the selected item of the list.



Degree Of Freedom

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Free / Xfix / Yfix / Zfix / XXfix / PlateYZ / YYfix / PlateXZ / FrameXY / ZZfix / PlateXY / FrameXZ / FrameYZ / TrussAndMembraneXZ / wfix.
Contains predefined settings to fix one or more degrees of freedom of the structure.
wfix means: no warping. Taken into account only for nodes, the diaphragms will ignore it.
The output is an integer representing the selected item of the list.
- Hint* The constant values assigned to the combined constraints are calculated by adding the constants of the basic constraints. E.g.: PlateXY = Xfix+Yfix+ZZfix=1+2+32= 35. Do not modify these constants.



Distribution Type

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Global / Local / Projected / Edge Local.
Mesh-independent surface load distribution type.
The output is an integer representing the selected item of the list.



Domain Composite Rib Eccentricity Type

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Top / Midplane / Bottom / Custom.
Rib alignment to the domain (for composite rib domain).
The output is an integer representing the selected item of the list.



Domain Eccentricity Type

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* None / Constant / One Direction / Two Directions / Top Surface / Bottom Surface.
Types of domain eccentricity.
The output is an integer representing the selected item of the list.

**Domain Variable Thickness**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* None / One Direction / Two Directions.
Types of domain variable thickness.
Type "None" means, constant thickness is applied.
The output is an integer representing the selected item of the list.

**Gap Type**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Active In Tension / Active In Compression.
Gap element type.
The output is an integer representing the selected item of the list.

**Group Combination Type**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Old (not used) / Exclusive / Additive.
Behaviour of load cases of groups when determining the critical combination.
The output is an integer representing the selected item of the list.

**Hollow Hole Type**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Circular / Rectangular.
Hole geometry for hollow domains.
The output is an integer representing the selected item of the list.

**Line Element Eccentricity Alignment Point**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Top Left / Top Center / Top Right / Center Left / Center Center / Center Right / Bottom Left / Bottom Center / Bottom Right.
Alignment points (main points of the cross-section envelope box) for line element cross-sections.
The output is an integer representing the selected item of the list.



Line Element Eccentricity Type

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* None / Custom Offset / Alignment Point / Group / Ref / Rib Domain.
Types of line element eccentricities.
The output is an integer representing the selected item of the list.



Line Nonlinearity

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Tension And Compression / Tension Only / Compression Only.
Types of nonlinear behaviour for truss elements, line supports, surface supports.
The output is an integer representing the selected item of the list.



Line Support Type

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Edge Global / Edge Relative / Rib Elastic Foundation / Beam Elastic Foundation / Edge Reference.
Line support type. (Note: Direction of nonlinearity, stiffness and resistance depends on this.)
The output is an integer representing the selected item of the list.



Line Type

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Truss / Beam / Rib / Spring / Gap / Edge / Hole / Simple Line / NN Link / LL Link.
Structural type of lines. Simple line means only geometry.
The output is an integer representing the selected item of the list.



Load Combination Type

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Other / SLS1 = SLSChar / SLS2 = SLSFreq / SLS3 = SLSQuasi / ULS1 = ULS / ULS2 = ULSSeismic / ULS3 = ULSExceptional / ULSALL / ULSab / ULSa / ULSb / ULSALLab / ULSA1 / ULSA2 / ULSA3 / ULSA4 / ULSA5 / ULSA6 / ULSA7 / ULSA8 / ULSAIISE1 / ULSAIISE2 / ULSAIISE3 / ULSAIISE4 / ULSAIISE5 / ULSAIISE6 / ULSAIISE7 / ULSAIISE8 / SemiAutoSLS1 / SemiAutoSLS2 / SemiAutoSLS3.
Load combination type. (Some values are referred by two different names.)
The output is an integer representing the selected item of the list.

**Load Distribution Type**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Constant / Linear.
Mesh-independent surface load distribution types.
The output is an integer representing the selected item of the list.

**Load Group Type**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Permanent / Incidental / Exceptional / Seismic / Prestress / Moving / Imperfection / Snow / Snow Except / Wind / Manual Seismic / Manual Prestress / Fire.
Load group types.
The output is an integer representing the selected item of the list.

**Member Local X Direction**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* ij / ji.
Local X orientation with respect to the node with the lower index.
The output is an integer representing the selected item of the list.

**Release Position Type**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Auto / Ratio / Length.
Position type of the release interface.
The output is an integer representing the selected item of the list.

**Release Type**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Rigid / Hinged / Semi Rigid / Plastic / Pushover.
Type of beam end release.
The output is an integer representing the selected item of the list.

**Spring Direction**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Global / Geometry / Point Reference / Vector Reference / Element Relative / Node Relative / Seismic Isolator.
Coordinate system for spring stiffness components.
The output is an integer representing the selected item of the list.

**Surface/Domain Load Distribution Type**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Surface / Projected.
Surface/domain load distribution type.
The output is an integer representing the selected item of the list.

**Surface Type**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Hole / Membrane Stress / Membrane Strain / Plate / Shell.
Type of surface elements.
The output is an integer representing the selected item of the list.

**System**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Global / Local / Reference.
Coordinate system of load components.
The output is an integer representing the selected item of the list.

**XLAM Moisture Content**

- Usage* Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.
- Input* There is no Grasshopper input for this component.
- Output* Class 1 / Class 2.
XLAM moisture content (service class) for XLAM domains.
The output is an integer representing the selected item of the list.

**XLAM Top Layer Direction**

Usage Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.

Input There is no Grasshopper input for this component.

Output Local X / Local Y.
Grain direction of the XLAM panel's top layer.
The output is an integer representing the selected item of the list.

**XY Direction**

Usage Click the arrow on the right side of the component and select the required value from the dropdown menu by one mouse click.

Input There is no Grasshopper input for this component.

Output X / Y.
Axis direction.
The output is an integer representing the selected item of the list.

**XYZ**

Usage This component compresses three values into an array. It can collect for example Fx, Fy, Fz force components and serve as input for other components.

Input

- X: first numerical input (floating point number). Default value: 0
- Y: second numerical input (floating point number). Default value: 0
- Z: third numerical input (floating point number). Default value: 0

Output XYZ parameter that consist of X, Y, Z inputs compressed into an array.

Hints Convertible from Vector, considering X, Y, Z coordinates as X, Y, Z inputs.
Convertible from Point, considering X, Y, Z coordinates as X, Y, Z inputs.
Convertible from a number, considering the same number at each of X, Y, Z inputs.
Convertible to Point, considering X, Y, Z inputs as X, Y, Z coordinates.
Convertible to Vector, considering X, Y, Z inputs as X, Y, Z coordinates.

**XYZ.XXYYZZ**

Usage This component compresses six values into an array. It can collect for example the support stiffnesses in six direction (translational in X, Y, Z direction and rotational about X, Y, Z axis) and serve as input for other components.

Input

- X: first numerical input (floating point number). Default value: 0
- Y: second numerical input (floating point number). Default value: 0
- Z: third numerical input (floating point number). Default value: 0
- XX: fourth numerical input (floating point number). Default value: 0
- YY: fifth numerical input (floating point number). Default value: 0
- ZZ: sixth numerical input (floating point number). Default value: 0

Output XYZ.XXYYZZ parameter that consist of X, Y, Z, XX, YY, ZZ inputs compressed into an array.

Hints Convertible from a number, considering the same number at each of X, Y, Z, XX, YY, ZZ inputs.

32. PARAMS



Usage The Params subcategory contains the parameters for each AxisVM component (except of the parameters of the AxisVM value list inputs and for the sender component). Parameters may be useful to organize data of a given type inside a script.

Input The input can be any data source with the same data type that the parameter can store.

Output The output of the parameter has the same data type that the parameter stores. Parameters never modify the data.

Hints To store the data permanently, right-click on the parameter and choose Internalize data.

The parameters never modify the input data itself, however they can change the data structure using the Reverse / Flatten / Graft / Simplify options.

AxisVM Line, Line extended, Gap and Spring share the same data type, thus they all can be stored inside AxisVM Line parameter.

AxisVM Simple Domain, Ribbed Domain, Domain With Custom Stiffness Matrix, Trapezoid Domain, XLAM Domain, Composite Rib Domain and Hollow Core Domain share the same data type, thus they all can be stored inside AxisVM Domain parameter.

AxisVM Domain Area Load and Panel Area Load share the same data type, thus both can be stored inside the AxisVM Domain/Panel Area Load parameter.

AxisVM Domain Polyline Load and Panel Polyline Load share the same data type, thus both can be stored inside the AxisVM Domain/Panel Polyline Load parameter.

33. ELEMENTS

Components are listed in the same order that they appear in the Elements subcategory.



Cross-Section

- Usage** Double-click on the left side (icon) of this component in order to open the AxisVM Cross-Section Library form. On the left side of the form: extend the list of the relevant cross-section shape, select a cross-section table. On the right side: select the required cross-section. Close the form with OK in order to store the selection.
- Input** There is no Grasshopper input for this component.
- Output** Selected cross-section as Cross-Section parameter. If cross-section is not selected, it returns an empty parameter.
- Hints** Graphical user interface is available for selecting a cross-section. If a cross-section has already been stored, it is highlighted upon re-opening the form. The components keep and reload the stored data if the Grasshopper file is saved and closed, then re-opened.
- Loading of cross-section tables and cross-sections happens in real-time, and that is why after clicking on elements the loading may take 1-2 seconds
- The contents of the Catalog depend on the language settings of AxisVM. If the language is changed during the connection between Grasshopper and AxisVM, the transfer of the Cross-Sections might fail. In this case, user need to place a new Cross-Section component to the canvas, and maybe the re-opening of Rhino and/or AxisVM (connecting a new GrasshopperToAxisVM component) is also necessary.



Cross-Section – Thick Polygon

- Usage** This component creates cross-sections from custom thick polygons and imports them into the current AxisVM model.
- Input**
- **Name:** text input, required name of the cross-section inside the AxisVM model.
 - **Polyline:** Closed polyline curve(s) with the required shape and dimensions. Counterclockwise vertex order denotes the outer contour of the cross-section; clockwise vertex order denotes the openings.
 - **Process Type (optional):** Cross-section Process Type value list input (Other / Rolled / Welded / Cold Formed). Default value: Other
- Output** Selected cross-section. If cross-section is not selected, then it returns empty parameter.
- Hints** The cross-section is added to the current AxisVM model, but NOT to the AxisVM Cross-Section Library.
- The component is NOT applicable to create parametric cross-sections (e.g.: I, C, U-shape etc.).



Cross-Section From Text

- Usage** This component creates an AxisVM Cross-Section from textual input.
- Input**
- **CS Shape (optional):** text input, accepted with and without suffix, e.g.: L or L-Shape.
 - **CS Table (optional):** text input, the exact name of the cross-section table is required, as it is stored in the AxisVM Cross-Section Library. Integer values (table IDs) are also accepted as input, if the CS table ID is known by the user. If this input is empty, the first matching cross-section table is chosen based on the CS Name input.
 - **CS Name:** text input, the exact name of the cross-section is required, as it is stored in the AxisVM Cross-Section Library. E.g.: I 80
- Output** Cross-section parameter. If cross section is not selected, then it returns empty parameter.
- Hints** The inputs are case-sensitive and sensitive for amount of space characters. If the component is not able to identify the cross-section you are looking for, copy the cross-section name from the AxisVM Cross-Section Library. (E.g.: by displaying the output of a Cross-section component in a Panel and

use the “Copy data only” command from the pop-up menu of the Panel.)

The contents of the Catalog depend on the language settings of AxisVM. If the language is changed during the connection between Grasshopper and AxisVM, the transfer of the Cross-Sections might fail. In this case, user need to place a new Cross-Section component to the canvas, and maybe the re-opening of AxisVM (connecting a new GrasshopperToAxisVM component) is also necessary.



Material

Usage Double-click on the left side (icon) of this component in order to open the AxisVM Material Library form. Extend the list of the relevant National Design Code, then select a material. Close the form with OK in order to store the selection.

Input There is no Grasshopper input for this component.

Output Selected material as Material parameter. If material is not selected, it returns an empty parameter.

Hints Graphical user interface is available for selection process. If a material has already been stored, it is highlighted upon re-opening the form. The components keep and reload the stored data if the Grasshopper file is saved and closed, then re-opened.

The contents of the Catalog depend on the language settings of AxisVM. If the language is changed during the connection between Grasshopper and AxisVM, the transfer of the Materials might fail. In this case, user need to place a new Material component to the canvas, and maybe the re-opening of AxisVM (connecting a new GrasshopperToAxisVM component) is also necessary.



Material From Text

Usage This component creates an AxisVM Material from textual input.

- Input**
- National Design Code (*optional*): text input, the exact name of the National Design Code is required, as it is stored in the AxisVM Material Library. E.g.: EuroCode or Dutch_NEN. If this input is empty, the first matching material table is chosen based on the Material Name input.
 - Material Name: text input, the exact name of the material is required, as it is stored in the AxisVM Material Library. E.g.: S 235

Output Material parameter. If material is not selected, then it returns empty parameter.

Hints The inputs are case-sensitive and sensitive for amount of space characters. If the component is not able to identify the material you are looking for, copy the material name from the AxisVM Material Library. (E.g.: by displaying the output of a Material component in a Panel and use the “Copy data only” command from the pop-up menu of the Panel.)

The contents of the Catalog depend on the language settings of AxisVM. If the language is changed during the connection between Grasshopper and AxisVM, the transfer of the Materials might fail. In this case, user need to place a new Material component to the canvas, and maybe the re-opening of AxisVM (connecting a new GrasshopperToAxisVM component) is also necessary.



Beam End Release

Usage This component can set up a beam end release, thus it provides input for the Beam End Releases component.

- Input**
- Release Type: requires a Release Type value list as input, valid inputs are: Rigid / Hinged / Semi Rigid. Default value: Rigid.
 - FunctionID (*optional*): integer input, required only for 7DOF beam with semi-rigid (spring param) type. The functions can be defined inside AxisVM at the moment, the indices start with 1.

Output Selected cross section. If cross section is not selected, then it returns empty parameter.

Hints Right-click on the component and click Create list of Release Types to add the value list to Release Type input.



Beam End Releases

Usage This component organizes the beam end release settings for the degrees of freedom of a beam end release. Only those inputs must be filled, at which degrees of freedom a beam end release need to be applied.

- Input**
- X (*optional*): Beam End Release parameter, translational release along the X axis.
 - Y (*optional*): Beam End Release parameter, translational release along the Y axis.
 - Z (*optional*): Beam End Release parameter, translational release along the Z axis.
 - XX (*optional*): Beam End Release parameter, rotational release about the local X-axis
 - YY (*optional*): Beam End Release parameter, rotational release about the local Y-axis
 - ZZ (*optional*): Beam End Release parameter, rotational release about the local Z-axis
 - Warping Release (7DOF) (*optional*): warping release, valid only for 7DOF elements.

Output AxisVM Releases parameter.

Hints Rigid and hinged type is accepted for every input parameter. 7DOF accepts the semi-rigid release type as well.



Line

Usage This component is able to define AxisVM line elements. Beams, ribs or trusses. It is also possible to define simple lines without material and cross-section assignment.

- Input**
- Lines: list of lines.
 - Material (*optional*): Material parameter. (Material or Material From Text component.)
 - Cross-section (*optional*): Cross-Section parameter, (Cross-Section, Cross-Section From Text or Cross-Section – Custom Thick components are accepted as input.)
 - Line Type (*optional*): Line Type value list (valid inputs: Truss / Beam / Rib / Simple Line). Default value: Beam.

Output AxisVM Line parameter.

Hints The component cannot handle the following line types: Spring / Gap / Edge / Hole / NN Link / LL Link. Types Gap and Spring have their separate, own components.

The lines are not grouped into structural members, they are transferred as separate line elements.



Line - Extended

Usage This component is able to define AxisVM Lines. Beams, ribs, trusses or simple lines. Lines can have variable cross-section, releases, eccentricity etc.

- Input**
- Line: list of lines.
 - Material (*optional*): Material parameter. Material or Material From Text component as input.
 - Moisture content (*optional*): integer value, service class (moisture content) of timber, Class 1, 2 or 3.
 - k_def (*optional*): floating point value, deformation factor [-] for timber members. Default values: for Class 1 k_def = 0.6, for Class 2 k_def=0.8, for Class 3 k_def = 2.0.
 - CS_start (*optional*): Cross-Section parameter. Cross-Section, Cross-Section From Text or Cross-Section – Custom Thick components are accepted as input.
 - CS_end (*optional*): Cross-Section parameter. Cross-Section, Cross-Section From Text or Cross-Section – Custom Thick components are accepted as input.
 - Line Type: Line Type value list (Truss / Beam / Rib / Simple Line). Default value: Beam.
 - Beam7DOF: Boolean input, valid only for beam line type. 7DOF beam (warping) is considered if this input is True. Default: False.
 - Local X Orientation: Local X Orientation value list. Local X direction with respect to the node with the lower index. Default value: ij.
 - Reference ID (*optional*): integer value, local Z reference index. Indices start with 1 in AxisVM. Use 0 for auto. Note: not all Z-references are valid for a given line.

- Truss NonLin Type (*optional*): Line Nonlinearity value list (Tension And Compression / Tension Only / Compression Only). Default: Tension And Compression.
- Resistance (*optional*): floating point value, the axial resistance (absolute value), valid only for Truss NonL Type "Tension Only" and "Compression Only".
- Eccentricity (*optional*): AxisVM Line Eccentricity parameter, eccentricity settings.
- AxDomain1 (*optional*): AxisVM Domain parameter; required if a rib is connected to a domain
- AxDomain2 (*optional*): AxisVM Domain parameter; required if two domains are necessary to determine the rib's orientation, or as a backup domain (in case of Domain1's deletion)
- k_x (*optional*): floating point value, friction resistance [kN/m/m] between rib and surface. Valid only in case of RibDomain eccentricity setting and Top or Bottom auto eccentricity of rib.
- Start Release (*optional*): AxisVM Releases parameter to be considered at the beginning of the line. Default setting: no release.
- End Release (*optional*): AxisVM Releases parameter to be considered at the end of the line. Default setting: no release.
- Material Color (*optional*): Color parameter (RGB), color override for the material color. For no override use input -1 (default).
- Contour Color (*optional*): Color parameter (RGB), color override for the contour color. For no override use input -1 (default).

Output AxisVM Line parameter.

Hints The component cannot handle the following line types: Spring / Gap / Edge / Hole/ NN Link / LL Link. Types Gap and Spring have their separate, own component.

The lines are not grouped into structural members, they are transferred as separate line elements.

If the line does not have variable cross-section, it is sufficient to feed the CS_start input. In this case CS_end is set automatically to the same value.

The value lists can be created from the component pop-up menu: Create list of Line types / Local X directions / Truss nonlinearity types.



Line Eccentricity

Usage This component must be used to manage the line eccentricity settings.

- Input**
- Start Eccentricity Type: AxisVM Line Element Eccentricity Type value list input (None / Custom Offset / Alignment Point / Group / Ref / Rib Domain), eccentricity type at the lower index node. In case of Group type: the alignment is specified in the Start Alignment Point field. Default value: Alignment Point.
 - End Eccentricity Type: AxisVM Line Element Eccentricity Type value list input (None / Custom Offset / Alignment Point / Group / Ref / Rib Domain), eccentricity type at the higher index node. Default value: Alignment Point.
 - Start Alignment Point (*optional*): AxisVM Line Element Eccentricity Alignment Point value list input (Top Left / Top Center / Top Right / Center Left / Center Center / Center Right / Bottom Left / Bottom Center / Bottom Right), alignment point on the cross-section envelope box at the lower index node. Considered only for eccentricity types: AlignmentPoint and Group. Default value: Top Center.
 - End Alignment Point (*optional*): AxisVM Line Element Eccentricity Alignment Point value list input (Top Left / Top Center / Top Right / Center Left / Center Center / Center Right / Bottom Left / Bottom Center / Bottom Right), alignment point on the cross-section envelope box at the higher index node. Considered only for eccentricity types: AlignmentPoint and Group. Default value: Top Center.
 - Rib Auto Eccentricity Type (*optional*): AxisVM Auto Eccentricity Type value list input (Custom / Top / Mid / Bottom), eccentricity type of ribs, only used when Start Eccentricity Type and End Eccentricity Type is RibDomain. Default value: Bottom.
 - Start Eccentricity Y (*optional*): floating point value, custom eccentricity [m] in local y direction at the lower index node. Considered only for eccentricity types: RibDomain with

custom AutoEccentricity and CustomOffset.

- Start Eccentricity Z (optional): floating point value, custom eccentricity [m] in local z direction at the lower index node. Considered only for eccentricity types: RibDomain with custom AutoEccentricity and CustomOffset.
- End Eccentricity Y (optional): floating point value, custom eccentricity [m] in local y direction at the higher index node. Considered only for eccentricity types: RibDomain with custom AutoEccentricity and CustomOffset.
- End Eccentricity Z (optional): floating point value, custom eccentricity [m] in local z direction at the higher index node. Considered only for eccentricity types: RibDomain with custom AutoEccentricity and CustomOffset.
- Eccentricity Group Index (optional): integer value, eccentricity group index. Considered only for eccentricity type Group.
- Start Reference AxLine (optional): AxisVM Line parameter, reference line at the lower index node. Considered only for eccentricity type Ref. The reference line must already have an AxisVM ID when it is considered as reference, therefore it must be connected to the GrasshopperToAxisVM sender component before the current line element, otherwise the eccentricity settings fail.
- End Reference AxLine (optional): AxisVM Line parameter, reference line at the higher index node. Considered only for eccentricity type Ref. The reference line must already have an AxisVM ID when it is considered as reference, therefore it must be connected to the GrasshopperToAxisVM sender component before the current line element, otherwise the eccentricity settings fail.
- Reference Start Alignment Point (optional): AxisVM Line Element Eccentricity Alignment Point value list input (Top Left / Top Center / Top Right / Center Left / Center Center / Center Right / Bottom Left / Bottom Center / Bottom Right), alignment point on the cross-section envelope box at the lower index node reference line. Considered only for eccentricity type Ref. Default value: Top Center.
- Reference End Alignment Point (optional): AxisVM Line Element Eccentricity Alignment Point value list input (Top Left / Top Center / Top Right / Center Left / Center Center / Center Right / Bottom Left / Bottom Center / Bottom Right), alignment point on the cross-section envelope box at the lower index node reference line. Considered only for eccentricity type Ref. Default value: Top Center.
- Start Eccentricity Release (optional): AxisVM Eccentricity Release parameter, eccentricity release settings at the lower index node. Considered only for eccentricity types CustomOffset, AlignmentPoint, Group and Ref.
- End Eccentricity Release (optional): AxisVM Eccentricity Release parameter, eccentricity release settings at the higher index node. Considered only for eccentricity types CustomOffset, AlignmentPoint, Group and Ref.

Output AxisVM Line Eccentricity parameter.

Hints The value lists can be created from the component pop-up menu: Create list of Line ecc. types / Line ecc. alignment points / Auto ecc. type of ribs / Line ecc. alignment points (Ref).

If the Ref eccentricity settings do not appear in the transferred AxisVM model, cross-check the order of line elements connected to the GrasshopperToAxisVM sender component. The reference lines must precede the current line element (that wants to refer to them) in the list of line elements to be transferred.



Line Eccentricity Releases

Usage This component organizes the line eccentricity release settings for each degree of freedom. Only those inputs must be filled, at which degrees of freedom an eccentricity release need to be applied. Only Rigid and Hinged release types are accepted.

- Input**
- Type X (optional): Release Type value list, type of translational release along the X axis. Valid types: Rigid and Hinged. Default value: Rigid.
 - Type Y (optional): Release Type value list, type of translational release along the Y axis. Valid types: Rigid and Hinged. Default value: Rigid.
 - Type Z (optional): Release Type value list, type of translational release along the Z axis.

Valid types: Rigid and Hinged. Default value: Rigid.

- Type XX (optional): Release Type value list, type of rotational release about the X axis. Valid types: Rigid and Hinged. Default value: Rigid.
- Type YY (optional): Release Type value list, type of rotational release about the Y axis. Valid types: Rigid and Hinged. Default value: Rigid.
- Type ZZ (optional): Release Type value list, type of rotational release about the Z axis. Valid types: Rigid and Hinged. Default value: Rigid.
- Position type (optional): Position Type value list (Auto / Ratio / Length), position type of the release interface. Auto is valid only for Ref eccentricity type. Default value: Length.
- Position (optional): floating point value. Position of the release interface, required only for Ratio or Length. For Ratio, it is between [0..1], for Length, it is as value in [m].

Output Line Eccentricity Releases parameter. This serves input for the AxisVM Line Eccentricity component.

Hints The value lists can be created from the component pop-up menu: Create list of Release types / Release position types.



Mesh

Usage This component transfers a Rhino mesh into an AxisVM mesh. The mesh can be transferred to AxisVM as edges, as surface elements (shell elements) or as domains.

- Input**
- Mesh: Rhino mesh
 - Thickness (optional): floating point value, thickness [m] for surface elements or domains. Default: 0.025 m.
 - Material (optional): Material parameter for edges, surface elements or domains. Default: EuroCode / S235.
 - Cross-section (optional): Cross-Section parameter for edges.
 - Line Type (optional): Line Type value list (Beam / Rib / Truss / Simple Line) for edges. Default value: Beam.

Output AxisVM Mesh parameter.

Hints The value list can be created from the component pop-up menu: Create list of Line types.
The nodes need to be in-plane according to a strict AxisVM criteria.



Point

Usage This component can dedicate a point to be an AxisVM point. Note: if nodal support or nodal load need to be created in the AxisVM model, the appropriate AxPoint must also be connected to the GrasshopperToAxisVM sender component.

- Input**
- Points: list of points that need to be transferred to AxisVM and get an AxisVM ID..

Output AxisVM Point parameter.



Domain

Usage This component can set up AxisVM domains with constant or variable thickness and constant or variable eccentricity. Holes, surface element type and color settings can also be defined.

- Input**
- Polyline: closed, planar polyline curve. This polyline is the contour of the domain.
 - Surface Type: Surface Type value list. Membrane Stress / Membrane Strain / Plate / Shell input is valid. Default value: Shell.
 - Thickness: constant or variable. Constant thickness [m] can be defined by using a floating point number as input. Variable thickness can be defined providing an AxisVM Domain Variable Thickness parameter. Default: 0.025 m.
 - Material: AxisVM Material parameter. Default: EuroCode / S235.
 - Eccentricity (optional): constant [m] or variable eccentricity. Constant eccentricity can be defined using a floating point number as input. Variable eccentricity can be defined

providing an AxisVM Domain Variable Eccentricity parameter. Default: no eccentricity.

- Holes (*optional*): list of closed, planar polyline curves, inside the contour of the domain. This/these polylines define the contour(s) of the hole(s). The outline of the holes must be in the plane of the domain otherwise no hole will be created. If the hole outline intersects the domain outline, the hole is deleted.
- Material Color (*optional*): Color parameter (RGB), color override for the material color. For no override use input -1 (default).
- Contour Color (*optional*): Color parameter (RGB), color override for the contour color. For no override use input -1 (default).

Output AxisVM Domain parameter.

Hint The value list can be created from the component pop-up menu: Create list of Surface types.



Domain – Composite Rib

Usage This component can set up composite ribbed domains. The ribs can run in local x or y direction but not both. Their material can be different from that of the domain.

- Input**
- Polyline: closed, planar polyline curve. This polyline is the contour of the domain.
 - Surface Type: AxisVM Surface Type value list. Plate / Shell input is valid. Default: Shell.
 - Constant Thickness: floating point value, defines the constant thickness of the domain [m]. Default value: 0.025 m.
 - Material: AxisVM Material parameter. Default: EuroCode / S235.
 - Domain Constant Eccentricity (*optional*): floating point value, defines the constant eccentricity of the domain [m]. Default: no eccentricity.
 - Holes (*optional*): list of closed, planar polyline curves, inside the contour of the domain. This/these polylines define the contour(s) of the hole(s). If the hole outline intersects the domain outline, the hole is deleted.
 - Origin: point parameter, origin of the rib grid. Set coordinates in [m]. Default: [0; 0; 0]
 - Directions: AxisVM XY direction value list input (X / Y), defines whether the ribs go into X or Y direction. Default: X.
 - Distance: floating point value, distance between the ribs. This can not be smaller than the width of the rib cross-section.
 - Rib Material: AxisVM Material parameter. Default: EuroCode / S235.
 - Rib Cross-Section: AxisVM Cross-Section parameter, cross-section of the ribs.
 - Rib Eccentricity Type: Domain Composite Rib Eccentricity value list input (Top / Midplane / Bottom / Custom), type of the rib eccentricity, the position of the rib related to the plane of the domain. Default: Bottom.
 - Rib Constant Eccentricity (*optional*): floating point value, constant, custom eccentricity of the ribs. Relevant only if rib eccentricity type is Custom.
 - Actual Ribs (*optional*): Boolean parameter, if True, actual ribs will be created. If False, logical ribs are considered, no finite elements will be created. It must be False, in case of steel ribs cast partially or entirely in concrete. Default value: False.
 - Has Custom Shear (*optional*): Boolean parameter, set to True if the Shear connection is Elastic and custom shear value must be defined. If False (default): shear connection is Rigid.
 - Custom Shear (*optional*): floating point value, value of the custom shear for elastic shear connections [kN/m/m]. This input is considered only if Actual Ribs input is True and Has Custom Shear input is True. Default: 0 kN/m/m.
 - Material Color (*optional*): Color parameter (RGB), color override for the material color. For no override use input -1 (default).
 - Contour Color (*optional*): Color parameter (RGB), color override for the contour color. For no override use input -1 (default).

Output AxisVM Domain – Composite Rib parameter.

Hints The value list can be created from the component pop-up menu: Create list of Surface types / Directions / Rib eccentricity types.

If the domain parameters are modified using Number sliders, AxisVM may freeze during the redraw process. Possible countermeasures: 1) type discrete values instead of pulling the number slider, 2) use the Lock solver option in Grasshopper during the tuning the Number sliders.



Domain – Custom Stiffness Matrix

Usage This component can set up a domain with custom stiffness matrix. This provides full control over the relations between internal force components and strains.

- Input**
- **Polyline:** closed, planar polyline curve. This polyline is the contour of the domain.
 - **Surface Type:** AxisVM Surface Type value list. Membrane Stress / Membrane Strain / Plate / Shell input is valid. Default: Shell
 - **Constant Thickness:** floating point value, defines the constant thickness of the domain [m]. Default value: 0.025 m.
 - **Material:** AxisVM Material parameter. Default: EuroCode / S235.
 - **Constant Eccentricity (optional):** floating point value, defines the constant eccentricity of the domain [m]. Default value: no eccentricity.
 - **Holes (optional):** list of closed, planar polyline curves, inside the contour of the domain. This/these polylines define the contour(s) of the hole(s). If the hole outline intersects the domain outline, the hole is deleted.
 - **A (optional):** 3x3 matrix input, membrane stiffness matrix [kN/m]; (normal forces – relative strain)
 - **B (optional):** 3x3 matrix input, coupling stiffness matrix, eccentric effects [kNm/m]; (normal forces – bending or moments – relative strain)
 - **D (optional):** 3x3 matrix, plate flexural (bending) stiffness matrix [kNm]; (moments – bending)
 - **S (optional):** 2x2 matrix, adjusted shear stiffness matrix [kN/m]; (shear forces – relative shear strain)
 - **Material Color (optional):** Color parameter (RGB), color override for the material color. For no override use input -1 (default).
 - **Contour Color (optional):** Color parameter (RGB), color override for the contour color. For no override use input -1 (default).

Output AxisVM Domain – Custom Stiffness Matrix parameter.

Hints Membrane elements require matrix A as input.

Plate elements require matrices D and S as input.

Shell elements require matrices A, B, D and S input.

The value list can be created from the component pop-up menu: Create list of Surface types.

If the domain parameters are modified using Number sliders, AxisVM may freeze during the redraw process. Possible countermeasures: 1) type discrete values instead of pulling the number slider, 2) use the Lock solver option in Grasshopper during the tuning the Number sliders.



Domain – Hollow Core

Usage This component can set up a hollow core, concrete domain. The voids can run in local x or y direction but not both.

- Input**
- **Polyline:** closed, planar polyline curve. This polyline is the contour of the domain.
 - **Surface Type:** AxisVM Surface Type value list. Plate / Shell input is valid. Element type of the domain. Default: Shell.
 - **Constant Thickness:** floating point value, the constant thickness of the domain [m]. Default: 0.025 m.
 - **Material:** AxisVM Material parameter. Concrete material is required.
 - **Constant Eccentricity (optional):** floating point value. Constant eccentricity [m] of the domain. Default: no eccentricity.
 - **Holes (optional):** list of closed, planar polyline curves, inside the contour of the domain.

This/these polylines define the contour(s) of the hole(s). If the hole outline intersects the domain outline, the hole is deleted.

- **Origin:** point parameter, origin of the void grid. Set the global coordinates in [m]. Default value: [0; 0; 0]
- **Direction:** AxisVM XY Direction value list (X / Y), the voids can run in local x or y direction along a grid. Default: X.
- **D distance:** floating point value, distance between the centers of the voids.
- **Hole Type:** AxisVM Hollow Hole Type value list (Circular / Rectangular). Default: Circular.
- **Fi diameter (optional):** floating point value, the diameter of the tubular (circular) voids.
- **b width (optional):** floating point value, width of the rectangular voids.
- **h height (optional):** floating point value, height of the rectangular voids.
- **Material Color (optional):** Color parameter (RGB), color override for the material color. For no override use input -1 (default).
- **Contour Color (optional):** Color parameter (RGB), color override for the contour color. For no override use input -1 (default).

Output AxisVM Domain – Hollow Core parameter.

Hints The value list can be created from the component pop-up menu: Create list of Surface types / Directions / Hollow hole types.

If the domain parameters are modified using Number sliders, AxisVM may freeze during the redraw process. Possible countermeasures: 1) type discrete values instead of pulling the number slider, 2) use the Lock solver option in Grasshopper during the tuning the Number sliders.



Domain - Ribbed

Usage This component can set up a parametric ribbed plate (ribbed domain). The ribs can run in local x and/or y direction. The same, concrete material is considered for the plate and the ribs.

- Input**
- **Polyline:** closed, planar polyline curve. This polyline is the contour of the domain.
 - **Surface Type:** AxisVM Surface Type value list. Plate / Shell input is valid. Element type of the domain. Default: Shell.
 - **Constant Thickness:** floating point value, the constant thickness of the domain [m]. Default: 0.025 m.
 - **Material:** AxisVM Material parameter. Concrete material is required.
 - **Domain Constant Eccentricity (optional):** floating point value. Constant eccentricity of the domain [m]. Default: no eccentricity.
 - **Holes (optional):** list of closed, planar polyline curves, inside the contour of the domain. This/these polylines define the contour(s) of the hole(s). If the hole outline intersects the domain outline, the hole is deleted.
 - **Origin:** point parameter. Origin of the rib grid. Define the global coordinates in [m]. Default value: [0; 0; 0]
 - **xh (optional):** floating point value, height of the x directional ribs (including slab) [m]. Default value: 0.
 - **xb (optional):** floating point value, width of the x directional ribs [m]. Default value: 0.
 - **xd (optional):** floating point value, spacing of the x directional ribs [m]. Default value: 0.
 - **Auto Eccentricity Type x (optional):** AxisVM Auto Eccentricity Type value list, the eccentricity of the x directional ribs. Custom / Top / Bottom are valid inputs. Default: Bottom.
 - **Rib Eccentricity x (optional):** floating point value, custom eccentricity [m] of the x directional ribs. Considered only if Auto Eccentricity Type is Custom.) Default value: 0.
 - **yh (optional):** floating point value, height of the y directional ribs (including slab) [m]. Default value: 0.
 - **yb (optional):** floating point value, width of the y directional ribs [m]. Default value: 0.
 - **yd (optional):** floating point value, spacing of the x directional ribs [m]. Default value: 0.
 - **Auto Eccentricity Type y (optional):** AxisVM Auto Eccentricity Type value list, the eccentricity of the y directional ribs. Custom / Top / Bottom are valid inputs. Default: Bottom.

- Rib Eccentricity y (*optional*): floating point value, custom eccentricity [m] of the y directional ribs. Considered only if Auto Eccentricity Type is Custom.) Default value: 0.
- Actual Ribs (*optional*): Boolean parameter, if True, actual ribs will be created. If False, logical ribs are considered (no finite elements will be created). Default: False.
- Material Color (*optional*): Color parameter (RGB), color override for the material color. For no override use input -1 (default).
- Contour Color (*optional*): Color parameter (RGB), color override for the contour color. For no override use input -1 (default).

Output AxisVM Ribbed Domain parameter.

Hints The value list can be created from the component pop-up menu: Create list of Surface types / Eccentricity types of ribs (x) / Eccentricity types of ribs (y).

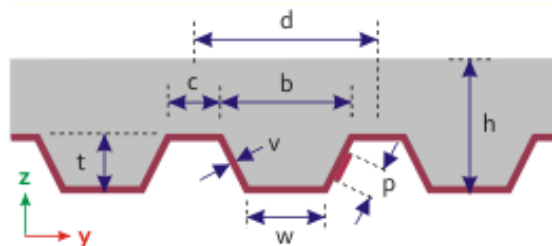
If the domain parameters are modified using Number sliders, AxisVM may freeze during the redraw process. Possible countermeasures: 1) type discrete values instead of pulling the number slider, 2) use the Lock solver option in Grasshopper during the tuning the Number sliders.



Domain - Trapezoid

Usage This component can set up a domain with trapezoidal steel deck (a composite slab where a steel deck is filled with concrete).

- Input**
- Polyline: closed, planar polyline curve. This polyline is the contour of the domain.
 - Surface Type: AxisVM Surface Type value list. Plate / Shell input is valid. Element type of the domain. Default: Shell.
 - Infill Material (*optional*): AxisVM Material parameter. Material of the concrete infill of the trapezoid domain.
 - Plate Material: AxisVM Material parameter. Material of the steel plate. Default: EuroCode / S235.
 - Constant Eccentricity (*optional*): floating point value. Constant eccentricity of the domain [m]. Default: no eccentricity.
 - Holes (*optional*): list of closed, planar polyline curves, inside the contour of the domain. This/these polylines define the contour(s) of the hole(s). If the hole outline intersects the domain outline, the hole is deleted.
 - Origin: point parameter, origin of the rib system. Set the global coordinate in [m]. Default value: [0; 0; 0]
 - Direction: AxisVM XY Direction value list, (X / Y). The direction of the ribs. The ribs can go to local x or y direction but not both. Default: X.
 - h: floating point value; dimension "h" on the image [m]. Default value: 0.2 m.
 - t: floating point value; dimension "t" on the image [m]. Default value: 0.04 m.
 - v: floating point value; dimension "v" on the image [m]. Default value: 0.005 m.
 - d: floating point value; dimension "d" on the image [m]. Default value: 0.2 m.
 - b: floating point value; dimension "b" on the image [m]. Default value: 0.14 m.
 - w: floating point value; dimension "w" on the image [m]. Default value: 0.07 m.
 - p: floating point value; dimension "p" on the image; width of rolled mechanical embossments [m]. Default value: 0 m.



- eta: floating point value; effectiveness of the shear connection [-]. Default value: 1.
- Material Color (*optional*): Color parameter (RGB), color override for the material color. For no override use input -1 (default).

- Contour Color (*optional*): Color parameter (RGB), color override for the contour color. For no override use input -1 (default).

Output AxisVM Domain – Trapezoid parameter.

Hints The value list can be created from the component pop-up menu: Create list of Surface types / Directions.

If the domain parameters are modified using Number sliders, AxisVM may freeze during the redraw process. Possible countermeasures: 1) type discrete values instead of pulling the number slider, 2) use the Lock solver option in Grasshopper during the tuning the Number sliders.



Domain - XLAM

Usage This component can set up an XLAM (cross-laminated timber) domain based on XLAM panels created in AxisVM GUI.

- Input**
- Polyline: closed, planar polyline curve. This polyline is the contour of the domain.
 - Surface Type: Surface Type value list. Membrane Stress / Membrane Strain / Plate / Shell input is valid. Default: Shell.
 - Material: AxisVM Material parameter. Timber material must be selected.
 - Constant Eccentricity (*optional*): floating point value. Constant eccentricity [m] of the domain. Default: no eccentricity.
 - Holes (*optional*): list of closed, planar polyline curves, inside the contour of the domain. This/these polylines define the contour(s) of the hole(s). The outline of the holes must be in the plane of the domain otherwise no hole will be created. If the hole outline intersects the domain outline, the hole is deleted.
 - XLAM Index: integer parameter; ID in the AxisVM library. Indexing starts with 1.
 - Top Layer Direction: AxisVM XLAM Top Layer Direction value list (Local X / Local Y). Top layer direction relative to the local coordinate system of the domain. Default: Local X.
 - Moisture content: AxisVM XLAM Moisture Content value list, (Class 1 / Class 2). This is a classification based on the moisture content of the material and the relative humidity. Default: Class 1.
 - k_{def}: floating point value from the interval [0; 1], deformation factor. Default: 0.8.
 - k_{sys}: Boolean parameter, if True, the system strength factor is taken into consideration when calculating the strength properties of the member. Default value: False.
 - k_{fin}: Boolean parameter, if True, the flexural strength is taken into account with reduced value. Check it on if the lamellas are finger-jointed. Default value: False.
 - Material Color (*optional*): Color parameter (RGB), color override for the material color. For no override use input -1 (default).
 - Contour Color (*optional*): Color parameter (RGB), color override for the contour color. For no override use input -1 (default).

Output AxisVM Domain – XLAM parameter.

Hints The value list can be created from the component pop-up menu: Create list of Surface types / XLAM top layer directions / XLAM moisture contents (service classes).

XLAM module is required in AxisVM to use this domain type.

XLAM panels need to be created in AxisVM GUI.

If the domain parameters are modified using Number sliders, AxisVM may freeze during the redraw process. Possible countermeasures: 1) type discrete values instead of pulling the number slider, 2) use the Lock solver option in Grasshopper during the tuning the Number sliders.



Domain Variable Eccentricity

- Usage** This component provides an AxisVM Domain Variable Eccentricity parameter that may be the input of domain components that can handle variable eccentricity.
- Input**
- **Eccentricity Type:** AxisVM Domain Eccentricity Type value list (None / Constant / One Direction / Two Directions / Top Surface / Bottom Surface). The type of domain eccentricity. Type None means: no eccentricity. Default: Constant.
 - **Point 1 (optional):** point parameter, reference point #1, required for eccentricity types One Direction and Two Directions.
 - **Point 2 (optional):** point parameter, reference point #2 required for eccentricity types One Direction and Two Directions.
 - **Point 3 (optional):** point parameter, reference point #3 required only for eccentricity type Two Directions.
 - **Eccentricities (optional):** AxisVM XYZ parameter; eccentricity values [m] in each reference point. First value is considered for eccentricity type Constant. The first two values are considered for eccentricity type One Direction. All of the three values are considered for eccentricity type Two Directions. (This input is irrelevant for eccentricity types Top Surface and Bottom Surface.)
 - **Eccentricity Group ID (optional):** integer value. Index of the group with same eccentricity parameters. Required for eccentricity types Top Surface and Bottom Surface. Group index starts with 1. The group member domains will be aligned with each other (top / bottom surface).
- Output** AxisVM Domain Variable Eccentricity parameter.
- Hints** If Type is None, the usage of the Domain Variable Eccentricity as input is unnecessary, because domains have zero eccentricity anyway by default.
The value list can be created from the component pop-up menu: Create list of Domain ecc. types.
As many eccentricity groups can exist as many domains are defined in the AxisVM model.



Domain Variable Thickness

- Usage** This component provides an AxisVM Domain Variable Thickness parameter that may be the input of domain components that can handle variable thickness.
- Input**
- **Variable Thickness Type:** AxisVM Domain Variable Thickness Type value list (None / One Directions / Two Directions). Type None means constant thickness. Default: None.
 - **Point 1 (optional):** point parameter, reference point #1, required for variable thickness types One Direction and Two Directions.
 - **Point 2 (optional):** point parameter, reference point #2 required for variable thickness types One Direction and Two Directions.
 - **Point 3 (optional):** point parameter, reference point #3 required only for variable thickness type Two Directions.
 - **Thicknesses:** AxisVM XYZ parameter; thickness values in each reference point. First value is considered for variable thickness type None. The first two values are considered for variable thickness type One Direction. All of the three values are considered for variable thickness type Two Directions.
- Output** AxisVM Domain Variable Thickness parameter.
- Hints** If Type is None, user may connect a floating point value directly into the Thickness input of the domain component instead of this Domain Variable Thickness component in order to set the constant domain thickness.
The value list can be created from the component pop-up menu: Create list of Domain variable thickness types.



Line Support

Usage This component can set up a line support.

- Input**
- **AxLine:** AxisVM Line parameter, the line(s) to place the support on. If the line support must lie on the edge of a domain, the edge need to be transferred as Simple Line into AxisVM (through AxisVM Line component), and the Check Double option must be activated on the Settings component.
 - **AxDomain1 (optional):** AxisVM Domain parameter; required if the line support has to be placed on a domain edge. Considered only for line support types: Edge Global / Edge Relative / Edge Reference.
 - **AxDomain2 (optional):** AxisVM Domain parameter to define the second domain connected to the line support. Input must be empty if only one domain is connected to the support.
 - **Line Support Type:** AxisVM Line Support Type value list (Edge Global / Edge Relative / Rib Elastic Foundation / Beam Elastic Foundation / Edge Reference). Type of the line support. Default values: Beam Elastic Foundation for beams, Rib Elastic Foundation for ribs, otherwise Edge Global.
 - **Stiffnesses:** AxisVM XYZ.XXYZZ parameter; stiffnesses of the support for each degrees of freedom: translation in X, Y, Z [kN/m/m] and rotation about X, Y, Z [kNm/rad/m].
 - **Shear Stiffness (optional):** floating point value; shear stiffness for Pasternaks [kN]. If set to 0 (default), a traditional Winkler support will be created.
 - **Nonlinearity:** Boolean parameter; if True: nonlinear parameters (nonlinearity type and resistance) are considered. Default value: False.
 - **Nonlinearity Type (optional):** AxisVM XYZ.XXYZZ parameter; type of nonlinearity may be set for each degrees of freedom: translation in X, Y, Z and rotation about X, Y, Z. User may apply AxisVM Line Nonlinearity value lists as inputs of the XYZ.XXYZZ component, or connects integer values to the inputs of the XYZ.XXYZZ component according the following rule: 0 = Tension And Compression / 1= Tension Only / 2 = Compression Only. These settings are considered only in nonlinear analysis.
 - **Resistances (optional):** AxisVM XYZ.XXYZZ parameter; resistance values for each degrees of freedom: Fx, Fy, Fz [kN/m] and Mx, My, Mz [kNm/m]. For beam and rib elastic foundation only Fx, Fy and Fz values are considered. The resistance values are considered only in nonlinear analyses.
 - **Reference ID (optional):** integer value; required only for line support type Edge Reference. The index of the reference edge. Indexes start from 1. Reference must be created in AxisVM.

Output AxisVM Line Support parameter.

Hints If the same stiffness / nonlinearity / resistance need to be applied for every degrees of freedom, it is sufficient to connect one single value to these input parameters (instead of XYZ.XXYZZ components). Thus the same value is going to be assigned to every DOF.

The value list can be created from the component pop-up menu: Create list of Line support types.

To get the edge of a domain, user may use the Explode component after a Curve or Polyline component and take the appropriate List Item afterwards from the list of Segments.



Nodal Support

Usage This component can set up a nodal support

- Input**
- **AxPoint:** AxisVM Point parameter; the list of points (nodes) to apply the support settings on.
 - **Stiffnesses:** AxisVM XYZ.XXYZZ parameter to collect the stiffness values of the nodal support for each degrees of freedom.

Output AxisVM Nodal Support parameter.

Hint If the same stiffness need to be applied in each direction, connect one single value instead of an XYZ.XXYZZ component to Stiffnesses input parameter. Thus each stiffness value is going to be the same.



Surface Support

Usage This component is used to set up a surface support.

- Input*
- AxMesh (*optional*): AxisVM Mesh parameter, the mesh (mesh surface or mesh domain) to apply the surface support on.
 - AxDomain (*optional*): AxisVM Domain parameter, the domain(s) to apply the surface support on.
 - Stiffnesses: AxisVM XYZ parameter; stiffnesses of the elastic foundation in local X, Y, Z direction of the surface [kN/m/m²]. Ensure that the surface support has a non-zero stiffness in each direction.
 - Shear Stiffness (*optional*): floating point value; shear stiffness of Pasternaks [kN/m]. If set to 0 (default), a traditional Winkler support will be created.
 - Nonlinearity: Boolean parameter; if True: nonlinear parameters (nonlinearity type and resistance) are considered. Default value: False.
 - Nonlinearity type (*optional*): AxisVM XYZ parameter; nonlinear behaviour of the the elastic foundation. User may apply AxisVM Line Nonlinearity value lists as inputs of the XYZ component, or connects integer values to the inputs of the XYZ component according the following rule: 0 = Tension And Compression / 1= Tension Only / 2 = Compression Only. These settings are considered only in nonlinear analysis.
 - Resistances (*optional*): AxisVM XYZ parameter; resistances of the elastic foundation in local X, Y, Z direction of the surface [kN/m²].

Output AxisVM Surface Support parameter.

Hint If the same stiffness / nonlinearity / resistance need to be applied for every degrees of freedom, it is sufficient to connect one single value to these input parameters (instead of XYZ components). Thus the same value is going to be considered in each direction.
Either AxMesh or AxDomain input must receive some data.



Supports

Usage This component collects every AxisVM Nodal Support parameter, AxisVM Line Support parameter and AxisVM Surface Support parameter that need to be transferred to the model. The AxisVM Supports component must be connected to the GrasshopperToAxisVM sender component.

- Input*
- AxNodalSupports (*optional*): AxisVM Nodal Support parameter; list of nodal supports.
 - AxLineSupports (*optional*): AxisVM Line Support parameter; list of line supports (on beam / rib / domain edge / surface edge).
 - AxSurfaceSupports (*optional*): AxisVM Surface Support parameter; list of surface supports.

Output AxisVM Supports parameter.



Diaphragm

Usage This component can set up diaphragm(s) on the selected lines. The diaphragms are special rigid bodies where the relative position of the element nodes remain constant in a global plane. Diaphragms represent plates totally rigid in their planes.

- Input*
- AxLine: AxisVM Line parameter; one or more closed loop of lines to be considered as diaphragm. Note: unconnected groups form separate diaphragms.
 - Working plane: AxisVM Degree Of Freedom value list; plane of operation. Valid inputs are PlateXY / PlateXZ / PlateYZ. The relative position of element nodes remain constant in the working plane. E.g.: for rigid plates in the X-Y plane choose PlateXY. Default: PlateXY.

Output AxisVM Diaphragm parameter.

Hints The value list can be created from the component pop-up menu: Create list of Working planes (DOF).



Edge Hinge

- Usage** This component can set up edge hinge (edge connection). It can be defined between domain edges (on common edge of domains) or between a rib and a domain edge.
- Input**
- **AxLine:** AxisVM Line parameter; rib that should be part of the edge hinge. (Consider to set up RibDomain eccentricity using the AxisVM Line Extended element.)
 - **AxDomain:** AxisVM Domain parameter; domain(s) that should have an edge hinge.
 - **Stiffnesses:** AxisVM XYZ.XXYZZ parameter; hinge stiffness in the local system of the edge of the selected domain. The values may be set for each degrees of freedom: translation in X, Y, Z [kN/m/m] and rotation about X, Y, Z [kNm/rad/m].
 - **Resistances (optional):** AxisVM XYZ.XXYZZ parameter; resistance values may be set for each degrees of freedom: translation in X, Y, Z [kN/m] and rotation about X, Y, Z [kNm/m]. These values are considered only in nonlinear analysis.
- Output** AxisVM Edge Hinge parameter.
- Hint** Nonlinearity type Tension and Compression is considered as nonlinearity type for each DOF in nonlinear analyses.
- To define the edge hinge on the common edge of two domains, connect both domains, and connect the domain edge as Simple Line.
- Activate the Check Double option on the AxisVM Settings component in order to eliminate the duplicated lines.



Rigid Element

- Usage** This component is used to define rigid elements on the selected lines. The selected lines that have common nodes define the same rigid element.
- Input**
- **AxLine:** AxisVM Line parameter; line(s) to be considered as rigid elements.
- Output** AxisVM Rigid Element parameter that consist of the AxisVM Rigid Element definition.
- Hints** The rigid elements and diaphragms have a common input on the GrasshopperToAxisVM sender component.



Gap (Line)

- Usage** This component is used to set up gap elements. The gap element is used to model point-to-point contact. It has two states: active (contact achieved) and inactive (no contact).
- Input**
- **Line:** line parameter; list of Rhino lines, line geometry of the gap elements.
 - **Local X Orientation:** AxisVM Member Local X Direction value list (ij / ji). Sets the direction of the local X axis from the lower to the higher node index or the opposite way. Default: ij.
 - **Gap Type:** AxisVM Gap Type value list; (Active In Tension / Active In Compression). select whether the gap should be active in tension (bolt tension) or compression (contact of plates). Default: Active In Compression.
 - **Active Stiffness:** floating point value; active stiffness [kN/m], stiffness is several orders of magnitude higher than in inactive state (simulates that a contact is achieved). It must be at least 1E+6 kN/m. Default value: 1E+8 kN/m.
 - **Inactive Stiffness:** floating point value; inactive stiffness [kN/m], small stiffness value (simulates that no contact is achieved). Default value: 1E+0 kN/m.
 - **Initial Opening:** floating point value; initial opening [m] for gap type active in compression, initial penetration [m] for gap type active in tension. Positive or zero value. While the initial opening does not close, the gap is considered inactive. Default value: 0 m.
- Output** AxisVM Gap (Line) parameter.
- Hint** The value lists can be created from the component pop-up menu: Create list of Local X directions / Gap types.
- If the gap element is used in an analysis different from a nonlinear static analysis, the element will be taken into account as a spring with a stiffness corresponding to its initial opening.

**Spring (Line)**

- Usage* This component is used to set up spring elements. The spring elements are not real structural elements, they connect each nodal degrees of freedom of two nodes of the model one-by-one. There is NO geometric coupling between the different DOFs (translation-rotation, force-moment).
- Input*
- Line: line parameter; list of Rhino lines, geometry of the spring element.
 - Spring Type: AxisVM Spring Direction value list; valid types: Global / Seismic Isolator. Default: Global.
 - Spring Characteristics (*optional*): AxisVM XYZ.XXYYZZ parameter; required only for spring type Global. Set the spring characteristics indexes for each degree of freedom from the AxisVM spring characteristics library. (Default library: 1 – soft translational, 2 – rigid translational, 3 – soft rotational, 4 – rigid rotational etc.) X,Y,Z inputs of the XYZ.XXYYZZ component represent the translational degrees of freedom; XX, YY, ZZ inputs represent the rotational degrees of freedom. New spring characteristics can be created inside AxisVM if necessary.
 - Seismic Isolator Index (*optional*): integer value; required only for seismic isolator. An applicable spring characteristic must be prepared inside AxisVM and then the ID must be referred here. You find the index in the Table Browser of AxisVM under Spring characteristics.
 - Isolator D2 (*optional*): floating point value; required only for seismic isolator, maximum design displacement at ULS [m].
- Output* AxisVM Spring (Line) parameter.
- Hint* The value list can be created from the component pop-up menu: Create list of Spring directions.
The axis of seismic isolators are parallel with the global coordinate axes.

34. LOADS



Load Case

Usage This component is used to set up standard load case(s) including one or more loads. The load cases can be assigned to load groups and may be part of load combinations.

- Input**
- **Load Case Name:** text input; user-defined name of the load case. Default: ST1.
 - **AxisVM Load Group (optional):** AxisVM Load Group parameter. The load case is going to be attached to this load group in AxisVM.
 - **AxisVM Nodal Load (optional):** AxisVM Nodal Load parameter; connect the nodal loads that must be considered in the load case
 - **AxisVM Distributed Line Load (optional):** AxisVM Distributed Line Load parameter; connect the distributed line loads (both the basic and extended components) that must be considered in the load case
 - **AxisVM Domain Concentrated Load (optional):** AxisVM Domain Concentrated Load parameter; connect the domain concentrated loads that must be considered in the load case
 - **AxisVM Domain Polyline Load (optional):** AxisVM Domain Polyline Load parameter; connect the domain polyline loads that must be considered in the load case
 - **AxisVM Distributed Domain Load (optional):** AxisVM Distributed Domain Load parameter; connect the distributed domain loads that must be considered in the load case
 - **AxisVM Domain Area Load (optional):** AxisVM Domain Area Load parameter; connect the domain area loads that must be considered in the load case
 - **AxisVM Distributed Surface Load (optional):** AxisVM Distributed Surface Load parameter; connect the distributed surface loads that must be considered in the load case
 - **AxisVM Load Panels (optional):** AxisVM Load Panel parameter; connect the load panels that must be considered in the load case
 - **AxisVM Panel Concentrated Load (optional):** AxisVM Panel Concentrated Load parameter; connect the panel concentrated loads that must be considered in the load case
 - **AxisVM Panel Polyline Load (optional):** AxisVM Panel Polyline Load parameter; connect the panel polyline loads that must be considered in the load case
 - **AxisVM Panel Area Load (optional):** AxisVM Panel Area Load parameter; connect the panel area loads that must be considered in the load case
 - **AxisVM Self-Weight Load (optional):** AxisVM Self-Weight parameter; connect the self-weight loads that must be considered in the load case
 - **AxisVM Thermal Load (optional):** AxisVM Thermal Load parameter; connect the thermal loads that must be considered in the load case

Output AxisVM Load Case parameter.



Load Combination

Usage This component can set up load combinations from load cases, considering user-defined factors. It is not necessary to create load combinations to determine the critical combination.

- Input**
- **Name:** text input; user-defined name of the load combination.
 - **Load Combination Type:** AxisVM Combination Type value list; (Other / SLS1 = SLSChar / SLS2 = SLSFreq / SLS3 = SLSQuasi / ULS1 = ULS / ULS2 = ULSSeismic / ULS3 = ULSExceptional / ULSALL / ULSab / ULSa / ULSb / ULSALLab / ULSA1 / ULSA2 / ULSA3 / ULSA4 / ULSA5 / ULSA6 / ULSA7 / ULSA8 / ULSAIISE1 / ULSAIISE2 / ULSAIISE3 / ULSAIISE4 / ULSAIISE5 / ULSAIISE6 / ULSAIISE7 / ULSAIISE8 / SemiAutoSLS1 / SemiAutoSLS2 / SemiAutoSLS3), select the required combination type from the dropdown list. Default: ULS.
 - **Load Cases:** AxisVM Load Case parameter; list of load cases to be considered in the current load combination.
 - **Factors:** floating point value(s); list of factors that need to be assigned to the load cases.

Output AxisVM Load Combination parameter.

- Hints* It is recommended that the lists provided for the Factors and Load Cases input have the same length. If this is not the case, double-check whether the values are paired as supposed.
- The value list can be created from the component pop-up menu: Create list of Load combination types.



Load Group

- Usage* This component is used to set up load groups. Load groups are used when generating of critical (design) values of the results. The load group component needs to be used as input for each load case that must be a part of the current load group.
- Input*
- **Name:** text input; user-defined name of the load group. Each load group must have unique name.
 - **Load Group Type:** AxisVM Load Group Type value list; (Permanent / Incidental / Exceptional / Seismic / Prestress / Moving / Imperfection / Snow / Snow Except / Wind / Manual Seismic / Manual Prestress / Fire.) Select the required load group type from the dropdown menu. Default: Permanent.
 - **Group Combination Type (optional):** AxisVM Group Combination Type value list; Exclusive / Additive. Select the required behaviour when determining the critical combination. Default value: Additive.
- For permanent and prestress load groups:**
Exclusive: When determining critical combination only the most unfavourable load case will be taken into account from the load group with its upper or lower safety factor.
Additive: all load cases will be taken into account simultaneously when determining the critical combination.
- For incidental load groups:**
Exclusive: only one load case of the group can be included into the critical combination.
Additive: multiple load cases can be included into the critical combination.
- **Simultaneous (optional):** Boolean parameter; considered only for Incidental load group type; it tells whether the group can be simultaneous with load cases of exceptional groups. Default: True.
- Output* AxisVM Load Group parameter.
- Hints* The value lists can be created from the component pop-up menu: Create list of Load group types / Group combination types.



Nodal Load

- Usage* This component is used to set up nodal loads in AxisVM. Nodal load may be defined by its components (forces and moments related to x, y, z axes) or nodal load is applied in reference direction (in this case Fx and Mx components are taken as load magnitudes).
- Input*
- **AxPoint:** AxisVM Point parameter; list of points as acting point of the nodal load.
 - **Forces (optional):** floating point value; forces in x, y, and z direction [kN]. If load is applied in reference direction, the Fx value is considered as force magnitude.
 - **Moments (optional):** floating point value; moments about x, y and z axes [kNm]. If load is applied in reference direction, the Mx value is considered as moment magnitude.
 - **Reference ID (optional):** integer value; if set to 0, global coordinate system is applied, else the load direction is set by the selected reference. Indexes start from 1. Reference must be created in AxisVM.
- Output* AxisVM Nodal Load parameter.
- Hint* Either Forces or Moments must receive any input data.



Distributed Line Load

Usage This component is used to set up a basic distributed line load (distributed force or moment). The load can be applied on AxisVM Lines or AxisVM Mesh Edges. Load can act in one direction (either in global or local coordinate system), over a segment of the line/edge. The segment can be defined by length or by ratio.



- Input**
- AxLine (optional): AxisVM Line parameter; the list of lines which the loads must act on. Line type must be Beam or Rib.
 - AxMesh (optional): AxisVM Mesh parameter (mesh edges are accepted); the list of mesh edges which the loads must act on. Line type must be Beam or Rib.
 - Coordinate system: AxisVM System value list; (Global / Local are valid inputs); choose the required coordinate system. Default: Global.
 - Direction: AxisVM Axis value list (X / Y / Z / XX / YY / ZZ); choose the direction of the distributed load. X, Y and Z are applicable for forces, XX, YY and ZZ are applicable for moments. Default: Z.
 - Position 1: floating point value, this value specifies the starting point of the loaded segment. If By ratio is True: starting position must be in [0; 1]. If By ratio is False, the starting position must be in [0; L] where L [m] is the length of the line element. Default: 0 (start of the line)
 - Position 2: floating point value, this value specifies the end point of the loaded segment. If By ratio is True: end position must be in [0; 1]. If By ratio is False, the end position must be in [0; L] where L [m] is the length of the line element. Default: 1 (end of the line)
 - Intensity 1: floating point value; this value specifies the load intensity at the starting point of the loaded segment. Distributed force: [kN/m], moment: [kNm/m]. Default: 0 kN/m
 - Intensity 2: floating point value; this value specifies the load intensity at the end point of the loaded segment. Distributed force: [kN/m], moment: [kNm/m]. Default: 0 kN/m
 - By Ratio: Boolean parameter; if True, the starting and end position of the loaded segment must be defined as the ratio of the line element length; if False, the positions must be defined in length dimension [m]. Default: True.

Output AxisVM Distributed Line Load parameter.

Hints The value lists can be created from the component pop-up menu: Create list of Coordinate systems / Directions.

Either AxLine or AxMesh (edge) requires input data.

Either Intensity 1 or Intensity 2 must be non-zero in order to create the distributed load.

To define more complex distributed line loads, use the extended component.



Distributed Line Load - Extended

Usage This component is used to set up complex distributed line loads (distributed forces and moments). You can define load components in each direction and trapezoid or triangle load profile can also be used.



- Input**
- AxLine (optional): AxisVM Line parameter; the list of lines which the loads must act on. Line type must be Beam or Rib.
 - AxMesh (optional): AxisVM Mesh parameter (mesh edges are accepted); the list of mesh edges which the loads must act on. Line type must be Beam or Rib.
 - Coordinate system: AxisVM System value list; (Global / Local are valid inputs); choose the required coordinate system. Default: Global.
 - Distribution type: AxisVM Beam Rib Distribution Type value list (Length / Projected). Select the required distribution type. Default: Length.

- **Position 1:** floating point value, this value specifies the starting point of the loaded segment. If By ratio is True: starting position must be in [0; 1]. If By ratio is False, the starting position must be in [0; L] where L [m] is the length of the line element. Default: 0 (start of the line).
- **Position 2:** floating point value, this value specifies the end point of the loaded segment. If By ratio is True: end position must be in [0; 1]. If By ratio is False, the end position must be in [0; L] where L [m] is the length of the line element. Default: 1 (end of the line).
- **Intensities 1:** AxisVM XYZ.XXYZZ parameter; the six floating point value inputs of the XYZ.XXYZZ component collects the six load components to be applied at the starting point of the loaded segment: forces in X, Y, Z direction [kN/m] and moments about X, Y and Z axis [kNm/m]. Default: 0 kN/m.
- **Intensities 2:** AxisVM XYZ.XXYZZ parameter; AxisVM XYZ.XXYZZ parameter; the six floating point value inputs of the XYZ.XXYZZ component collects the six load components to be applied at the end point of the loaded segment: forces in X, Y, Z direction [kN/m] and moments about X, Y and Z axis [kNm/m]. Default: 0 kN/m.
- **By Ratio:** Boolean parameter; if True, the starting and end position of the loaded segment must be defined as the ratio of the line element length; if False, the positions must be defined in length dimension [m]. Default: True
- **Trapezoid:** Boolean parameter; select whether the trapezoid load distribution should be used. Default: False.

Output AxisVM Distributed Line Load parameter.

Hints The value lists can be created from the component pop-up menu: Create list of Coordinate systems / Distribution types.

Either AxLine or AxMesh must receive any input data.

Either Intensities 1 or Intensities 2 must have at least one non-zero component in order to set up distributed load.

Triangle distribution can be created by using Trapezoid = True setting and assigning the same intermediate position for Position 1 and 2 and the same load intensity for Intensities 1 and 2.

To define more simple distributed line loads, user may take the basic AxisVM Distributed Line Load component.



Domain Concentrated Load

Usage This component is used to set up concentrated loads inside domains.

- Input**
- **AxMesh (optional):** AxisVM Mesh parameter; list of meshes (as domains) where the load will be applied.
 - **AxDomain (optional):** AxisVM Domain parameter; list of domains where the load will be applied.
 - **Coordinate System:** AxisVM System value list; (Global / Local / Reference). Select the coordinate system in which the load will be defined and applied. Default: Global.
 - **Position:** point parameter; the acting point of the load. Global coordinates in [m]. The point must be inside the domain.
 - **Forces (optional):** AxisVM XYZ parameter; the three floating point values of the XYZ component define the force components in X, Y and Z directions [kN]. If load is applied in reference direction, the Fx value is considered as force magnitude.
 - **Moments (optional):** AxisVM XYZ parameter; the three floating point values of the XYZ component define the moment components about the X, Y and Z axes [kNm]. If load is applied in reference direction, the Mx value is considered as moment magnitude.
 - **Reference ID (optional):** integer parameter; the AxisVM ID of the chosen reference; indexes start from 1. If this input is set to zero, load components are in global directions, otherwise the load direction is set by the chosen reference. Reference must be created in AxisVM upfront the index is referred here.

Output AxisVM Domain Concentrated Load parameter.

Hints The value list can be created from the component pop-up menu: Create list of Coordinate systems.

Either AxMesh (as domain) or AxDomain must receive input data.

Either Forces or Moments must have at least one non-zero component in order to set up the load.

If user connects one single value to the Forces / Moments input (instead of the XYZ components), the same load magnitude will be considered for each load component. (E.g.: force will have the same magnitude in X, Y and Z direction.)



Domain Polyline Load

Usage This component is used to set up distributed load inside one or more domains along a polyline. The domain must be defined by a normal vector at the moment. The force and moment components can be defined for the starting and end point of the polyline.

- Input**
- AxMesh (optional): AxisVM Mesh parameter; the mesh domains that need to be loaded along a polyline that lies inside the outer mesh boundaries. All mesh faces must be in-plane for this purpose.
 - AxDomain (optional): AxisVM Domain parameter; the domain(s) that need to be loaded.
 - Normal Vector (optional): vector parameter; the Rhino vector defines the components of the normal vector of the domain(s) to be loaded.
 - Polyline: curve parameter; planar polyline is expected. The distributed load will be applied along this polyline.
 - Distribution Type: AxisVM Distribution Type value list (Global / Local / Projected / Edge Local). Select the required distribution type from the dropdown menu. Default: Global.
 - Intensities 1 (optional): AxisVM XYZ parameter; the three floating point values of the XYZ component define the load intensities [kN/m] at the starting point of the loaded polyline in X, Y and Z directions.
 - Intensities 2 (optional): AxisVM XYZ parameter; the three floating point values of the XYZ component define the load intensities [kN/m] at the end point of the loaded polyline in X, Y and Z directions.
 - Distributed Moment 1 (optional): floating point value; the distributed torsional moment value [kNm/m] about the local X axis of the loaded polyline at the starting point.
 - Distributed Moment 2 (optional): floating point value; the distributed torsional moment value [kNm/m] about the local X axis of the loaded polyline at the end point

Output AxisVM Domain Polyline Load parameter.

Hints The value list can be created from the component pop-up menu: Create list of Distribution types.

Either AxMesh, AxDomain or NormalVector must receive input data. AxisVM can identify the domains that need to be loaded also based on the normal vector input.

Either Intensities 1 or 2, or Distributed Moment 1 or 2 must have at least one non-zero component in order to set up the load.

If user connects one single value to the Forces / Moments input (instead of the XYZ components), the same load magnitude will be considered for each load component. (E.g.: force will have the same magnitude in X, Y and Z direction.)



Distributed Domain Load

Usage This component is used to set up distributed load over one or more complete domains. The distributed load can act in one selected direction with constant intensity.

- Input**
- AxMesh (optional): AxisVM Mesh parameter; list of meshes (as domains) where the load will be applied.
 - AxDomain (optional): AxisVM Domain parameter; list of domains where the load will be applied.
 - Coordinate System: AxisVM System value list; (Global / Local). Select the coordinate system in which the load will be defined and applied. Default: Global.
 - Distribution type: AxisVM Surface Domain Distribution Type value list; (Surface / Projected) Select the type of the distributed load. Default: Surface.
 - Direction: AxisVM Axis value list (X / Y / Z / XX / YY / ZZ); choose the direction of the

distributed load. Default: Z.

- Intensity: floating point value; set the constant intensity of the distributed load [kN/m²].
- Window Load: Boolean parameter; if True: load falling on an opening is distributed along the edges of the opening, otherwise it is neglected. Default: False.

Output AxisVM Distributed Domain Load parameter.

Hints The value list can be created from the component pop-up menu: Create list of Coordinate systems / Distribution types / Directions.

Either AxMesh or AxDomain must receive input data.

To define more complex distributed domain loads, use the AxisVM Domain Area Load component.



Domain Area Load

Usage This component is used to set up complex distributed loads over one or more domains. User can set either constant intensity up to three directions or set linear load distribution in one selected direction. The load can be applied over a complete domain or inside a restricted area (defined with a polyline contour).

- Input**
- AxMesh (*optional*): AxisVM Mesh parameter; list of meshes (as domains) where the load will be applied. Either AxMesh or AxDomain must receive input data.
 - AxDomain (*optional*): AxisVM Domain parameter; list of domains where the load will be applied. Either AxMesh or AxDomain must receive input data.
 - Polyline (*optional*): curve parameter; planar, closed polyline is expected. If this input is empty, the distributed load is applied over the complete domain(s). If this input is provided, the load is applied only inside this polyline area. The polyline must be in the same plane as the loaded domain.
 - Distribution Type: AxisVM Distribution Type value list; (Global / Local / Projected / Edge Local). Select the required coordinate system in which the load will be defined and applied. Default: Global.
 - Load Distribution Type: AxisVM Load Distribution Type value list; (Constant / Linear). Select the required load distribution type. Default: Constant.
 - Direction: AxisVM Axis value list; (X / Y / Z are valid inputs). Required if load distribution type is Linear. Select the direction of the distributed load. Default: Z.
 - Point 1 (*optional*): point parameter, global coordinates of reference point #1, required only for linear load distribution type.
 - Point 2 (*optional*): point parameter, global coordinates of reference point #2, required only for linear load distribution type.
 - Point 3 (*optional*): point parameter, global coordinates of reference point #3, required only for linear load distribution type.
 - Intensity 1 (*optional*): floating point value; if load distribution type is constant, this value represents the load magnitude [kN/m²] in X direction; if load distribution type is linear, this value represents the load magnitude in the reference point #1, in the selected direction.
 - Intensity 2 (*optional*): floating point value; if load distribution type is constant, this value represents the load magnitude [kN/m²] in Y direction; if load distribution type is linear, this value represents the load magnitude in the reference point #2, in the selected direction.
 - Intensity 3 (*optional*): floating point value; if load distribution type is constant, this value represents the load magnitude [kN/m²] in Z direction; if load distribution type is linear, this value represents the load magnitude in the reference point #3, in the selected direction.
 - Window Load: Boolean parameter; if True: load falling on an opening is distributed along the edges of the opening, otherwise it is neglected. Default: False.

Output AxisVM Domain Area Load parameter.

Hints The value list can be created from the component pop-up menu: Create list of Distribution types / Load distribution types / Directions.

Either Intensity 1, 2 or 3 must receive input data in order to set up the load.

To define more simple distributed domain loads, user may take the AxisVM Distributed Domain Load component (constant load distribution in one specified direction).



Distributed Surface Load

- Usage** This component is used to set up distributed load over surface elements in one selected direction with constant intensity.
- Input**
- **AxMesh:** AxisVM Mesh input (mesh surface); list of surface elements where the load will be applied.
 - **CSys:** AxisVM System value list (Global / Local are valid inputs). Select the coordinate system in which the load will be defined and applied. Default: Global.
 - **Direction:** AxisVM Axis value list. (X / Y / Z / XX / YY / ZZ). Select the direction of the distributed load. Default: Z.
 - **Intensity:** floating point value, set the constant intensity of the distributed load [kN/m²].
- Output** AxisVM Distributed Surface Load parameter.
- Hints** The value list can be created from the component pop-up menu: Create list Coordinate systems / Axis.



Load Panel

- Usage** This component is used to set up load panels that are used to transfer nodal loads, distributed line and surface loads to the structural elements. They have no structural role, the only function of a load panel is to distribute loads over the domains, beam and rib elements.
- Input**
- **AxLine (optional):** AxisVM Line parameter; the line elements that define the contour of the load panel.
 - **AxMesh (optional):** AxisVM Mesh parameter; the mesh edges / surfaces / domains that need to be included in the load panel. (The load panel is defined on the outer contours of the complete mesh.)
 - **AxDomain (optional):** AxisVM Domain parameter; the domain(s) that need to be covered with load panel(s); separate load panel is created over each domain. To combine more domains in one load panel, try to add the contour as AxisVM simple lines (then activate the Check Double option on the Settings component in order to remove duplicated lines.)
- Output** AxisVM Load Panel parameter.
- Hint** Either AxLine, AxMesh or AxDomain must receive input data.



Load Panel Concentrated Load

- Usage** This component is used to define concentrated load onto load panels.
- Input**
- **AxLoadPanel:** AxisVM Load Panel parameter. The load panel that includes the acting point of the concentrated load.
 - **CSys:** AxisVM System value list; (Global / Local / Reference). Select the coordinate system in which the load will be defined and applied. Default: Global.
 - **Position:** point parameter; global load position, acting point of the concentrated load.
 - **Forces (optional):** AxisVM XYZ parameter; the three floating point values of the XYZ component define the force components in the X, Y and Z directions [kN].
 - **Moments (optional):** AxisVM XYZ parameter; the three floating point values of the XYZ component define the moment components about the X, Y and Z axes [kNm].
 - **Reference ID (optional):** integer value; if set to zero, global coordinate system is applied, else the load direction is set by the selected reference. Indices start from 1. Reference must be defined in AxisVM upfront the load is transferred.
- Output** AxisVM Panel Concentrated Load parameter.
- Hint** The value list can be created from the component pop-up menu: Create list Coordinate systems. Reference vector must be created in AxisVM.
At least one input value of Forces or Moments must be non-zero in order to set up the load.



Load Panel Polyline Load

- Usage** This component is used to set up distributed load inside one or more load panels along a polyline. The load panel must be defined by a normal vector at the moment. The force and moment components can be defined for the starting and end point of the polyline.
- Input**
- **Normal Vector:** vector parameter; the Rhino vector defines the components of the normal vector of the load panel(s) to be loaded.
 - **Polyline:** curve parameter; planar polyline is expected. The distributed load will be applied along this polyline.
 - **Distribution Type:** AxisVM Distribution Type value list (Global / Local / Projected / Edge Local). Select the required distribution type from the dropdown menu. Default: Global.
 - **Intensities 1 (optional):** AxisVM XYZ parameter; the three floating point values of the XYZ component define the load intensities [kN/m] at the starting point of the loaded polyline in X, Y and Z directions.
 - **Intensities 2 (optional):** AxisVM XYZ parameter; the three floating point values of the XYZ component define the load intensities [kN/m] at the end point of the loaded polyline in X, Y and Z directions.
 - **Distributed Moment 1 (optional):** floating point value; the distributed torsional moment value [kNm/m] about the local X axis of the loaded polyline at the starting point.
 - **Distributed Moment 2 (optional):** floating point value; the distributed torsional moment value [kNm/m] about the local X axis of the loaded polyline at the end point
- Output** AxisVM Load Panel Polyline Load parameter.
- Hint** The value list can be created from the component pop-up menu: Create list Distribution types. At least one component of Intensities 1, 2 or Distributed Moment 1, 2 must receive input data in order to set up the load.



Load Panel Area Load

- Usage** This component is used to set up complex distributed loads over one or more load panels. User can set either constant intensity up to three directions or set linear load distribution in one selected direction. The load can be applied over a complete load panel or inside a restricted area (defined with a polyline contour).
- Input**
- **AxLoadPanel:** AxisVM Load Panel parameter; list of the load panels where the load will be applied.
 - **Polyline:** curve parameter; planar, closed polyline is expected. If this input is empty, the distributed load is applied over the complete load panel(s). If this input is provided, the load is applied only inside this polyline area. The polyline must be in the same plane as the loaded load panel.
 - **Distribution Type:** AxisVM Distribution Type value list; (Global / Local / Projected / Edge Local). Select the required coordinate system in which the load will be defined and applied. Default: Global.
 - **Load Distribution Type:** AxisVM Load Distribution Type value list; (Constant / Linear). Select the required load distribution type. Default: Constant.
 - **Direction:** AxisVM Axis value list; (X / Y / Z are valid inputs). Required if load distribution type is Linear. Select the direction of the distributed load. Default: Z.
 - **Point 1 (optional):** point parameter, reference point #1, required only for linear load distribution type.
 - **Point 2 (optional):** point parameter, reference point #2, required only for linear load distribution type.
 - **Point 3 (optional):** point parameter, reference point #3, required only for linear load distribution type.
 - **Intensity 1 (optional):** floating point value; if load distribution type is constant, this value represents the load magnitude [kN/m²] in X direction; if load distribution type is linear, this value represents the load magnitude in the reference point #1, in the selected direction.
 - **Intensity 2 (optional):** floating point value; if load distribution type is constant, this value

represents the load magnitude [kN/m²] in Y direction; if load distribution type is linear, this value represents the load magnitude in the reference point #2, in the selected direction.

- Intensity 3 (*optional*): floating point value; if load distribution type is constant, this value represents the load magnitude [kN/m²] in Z direction; if load distribution type is linear, this value represents the load magnitude in the reference point #3, in the selected direction.
- Window Load: Boolean parameter; if True: load falling on an opening is distributed along the edges of the opening, otherwise it is neglected. Default: False. NOTE: this option is considered only if Polyline input is filled (load is applied inside a closed polygon).

Output AxisVM Panel Area Load parameter.

Hint The value lists can be created from the component pop-up menu: Create list of Distribution types / Load distribution types / Directions.

Either Intensity 1, 2 or 3 must have a non-zero input in order to set up the load.



Self-Weight

Usage This component can set up self-weight load on the selected structural elements.

- Input**
- AxLine (*optional*): AxisVM Line parameter; line elements of which the self-weight must be considered.
 - AxMesh (*optional*): AxisVM Mesh parameter; the mesh edges / mesh surfaces / mesh domains (AxMesh elements transferred as edges / surface elements / domains to AxisVM) of which the self-weight must be considered.
 - AxDomain (*optional*): AxisVM domain parameter; domains of which the self-weight must be considered.

Output AxisVM Self-Weight parameter.

Hints Either AxLine, AxMesh or AxDomain must receive input data in order to set up the load.



Thermal Load

Usage This component is used to set up thermal load over line elements, surface element or domains.

- Input**
- AxLine (*optional*): AxisVM Line parameter; list of line elements to apply the thermal load on.
 - AxMesh (*optional*): AxisVM Mesh parameter; list of mesh elements to apply the thermal load on.
 - AxDomain (*optional*): AxisVM Domain parameter; list of domains to apply the thermal load on.
 - T_ref: floating point value; reference temperature [°C]
 - T0 (*optional*): floating point value; relevant only for truss elements: T0 is the actual truss temperature [°C]
 - T_top (*optional*): floating point value; for beams / ribs: top cord temperature [°C] in the selected direction (local Y or Z); for surfaces / domains: temperature on the top side according to the local Z axis;
 - T_bottom (*optional*): floating point value; for beams / ribs: bottom cord temperature [°C] in the selected direction (local Y or Z); for surfaces / domains: temperature on the bottom side according to the local Z axis;
 - Direction (*optional*): AxisVM Axis value list (Y / Z are valid inputs). direction of temperature variation, only for line elements and only in local Y or Z direction. Default: Z.

Output AxisVM Thermal Load parameter.

Hint The value list can be created from the component pop-up menu: Create list of Directions.

Either AxLine, AxMesh or AxDomain must receive input data.

35. SEND



Grasshopper ToAxisVM

- Usage** This component handles the connection and data transfer between Grasshopper and AxisVM. It is the sender component. The items connected to this component are tried to be transferred to the AxisVM model. If transfer is not successful, a textual error message appears at the output parameter of the component.
- Input**
- AxisVM Settings: AxisVM Settings parameter; this is a mandatory input, this tells to the sender component whether a new file or an existing one should be opened; are there any special settings to be used during data transfer (e.g.: send just point coordinates / send only properties). This input tells whether a linear static analysis should be performed. No data transfer happens until an AxisVM model is not opened via COM server.
 - AxisVM Point (optional): AxisVM Point parameter; the list of points (nodes) to be transferred to the current AxisVM model.
 - AxisVM Line (optional): AxisVM Line parameter; the list of line elements (simple line / beam / rib / truss / gap / spring) to be transferred to the current AxisVM model.
 - AxisVM Mesh Surface (optional): AxisVM Mesh parameter; the list of mesh items to be transferred to the current AxisVM model as surface elements.
 - AxisVM Mesh Domain (optional): AxisVM Mesh parameter; the list of mesh items to be transferred to the current AxisVM model as domains.
 - AxisVM Mesh Edges (optional): AxisVM Mesh parameter; the list of mesh items to be transferred to the current AxisVM model as edges (line elements).
 - AxisVM Domain (optional): AxisVM Domain parameter; the list of domains to be transferred to the current AxisVM model.
 - AxisVM Load Cases (optional): AxisVM Load Case parameter; the list of load cases to be created in the current AxisVM model.
 - AxisVM Load Groups (optional): AxisVM Load Group parameter; the list of load groups to be created in the current AxisVM model.
 - AxisVM Load Combinations (optional): AxisVM Load Combination parameter; the list of load combinations to be created in the current AxisVM model.
 - AxisVM Supports (optional): AxisVM Supports parameter; the list of supports to be transferred to the current AxisVM model. The AxisVM Supports component need to be connected that has already collected the nodal, line and surface support settings.
 - AxisVM Rigid Element or AxisVM Diaphragm (optional): AxisVM Diaphragm parameter; the list of rigid elements and diaphragms to be created in the current AxisVM model.
 - AxisVM Edge Hinge (optional): AxisVM Edge Hinge parameter; the list of edge hinges to be created in the current AxisVM model
- Output** AxisVM Error parameter.
Connect a Panel to the output in order to visualize the error messages in case they appear.
- Hints** Sender component does not handle data trees. Data trees must be flattened at every input parameter into data lists in order to transfer them correctly. The AxisVM Settings input requires one single Settings component as input, because the COM server can handle one single communication channel in a reliable way.
- If nodal supports or nodal loads need to be created, not only the nodal support (boxed into AxisVM Supports) and nodal load (boxed into AxisVM Load Cases) but also the nodes must be transferred (via AxisVM Point input of the sender component) in order to get back the node ID. The node ID is required for Grasshopper to know where to apply the nodal support or nodal load.
- Only those data and elements can be transferred to AxisVM that has a connection to the sender component (directly or through other AxisVM components).



AxisVM Settings

- Usage** This component is used to define the settings of the data transfer between Grasshopper and AxisVM. As a default every single item is always deleted and re-send to AxisVM in case of model changes (this is valid for the data sent through Grasshopper!). The Settings component offers some other solutions for special cases to speed up the data transfer of huge models. This component is a mandatory input of the GrasshopperToAxisVM sender component.
- Input**
- NEW: Boolean parameter; if **True**, a completely new, clean AxisVM model is opened. Must be set to **False**, if user wants to open an AxisVM model from an existing file.
 - AxisVM Model File Path (*optional*): text input; this input works only if NEW input is False. This input requires a valid file path. This is accepted from text input (e.g.: from Panel) or – as a more comfortable solution – from a Grasshopper “File Path” parameter.
 - Just Point Coordinates (*optional*): Boolean parameter; if **False**, the default data transfer process is applied: every data is cleared and re-sent to AxisVM. If **True**, just the point (node) coordinates are transferred and modified in the AxisVM model. The user need to guaranty that the connections of the nodes and the element properties are unchanged, because these information are not updated if the parameter is True. Default value: False.
 - Send Only Properties (*optional*): Boolean parameter; if **False**, the default data transfer process is applied: every data is cleared and re-sent to AxisVM after each modification. If **True**, only the modified properties of the structural elements are transferred and updated in the AxisVM model. The user must guaranty that the node coordinates and the connections of the nodes are unchanged, because these information are not updated if the parameter is True. (Thus it can lead to disjoint of objects, multiple sent or unsend objects.) Supports, loads, load cases, load groups, load combinations, rigid elements, diaphragms and edge hinges are deleted and re-sent regardless of the Boolean value of this input. Default value: False.
 - Check Double (*optional*): Boolean parameter; if **True**, removes duplicate elements and nodes from the AxisVM model. (It is useful, for example, if RibDomain eccentricity is set or edge hinge is defined between rib and domain etc. The lines are transferred more than once in such cases.) If **False**, there may remain some duplicated nodes or elements in the model. Default value: False.
 - Intersect Lines (*optional*): Boolean parameter; if **True**, it intersect lines and creates common nodes and sections where applicable. If **False** (and auto intersect is not active in AxisVM), every node and element relation remains as they are defined in Rhino-Grasshopper. It is recommended to activate this option only if Just Point Coordinates and Send Only Properties are False in order to avoid mistakes in data transfer due to renumbering of objects in AxisVM model. Default value: False.
 - Run Analysis (*optional*): Boolean parameter; if **True**, a linear static finite element analysis is started after the data transfer. Before the analysis starts the user is asked to save the model. If **False**, no analysis is started, just the data transfer is proceeded. Default value: False.
- Output** AxisVM Settings parameter.
- Hints** Other option to speed up data transfer of huge models: right click or press middle mouse button on the Grasshopper canvas, select Lock solver. Perform the modification of the inputs during the locked state, then switch off the Lock solver option. This prevents AxisVM trying to transfer every intermediate status of the model.
- It is meaningless to activate the Just Point Coordinates and Send Only Properties options together, because they implement mutually exclusive processes.
- Run Analysis should be activated if the entire model is set up and transferred.

4. KNOWN ISSUES

- **Multiple AxisVM opens:** Loading Grasshopper projects that contain *GrasshopperToAxisVM* components may cause multiple AxisVM applications opened parallel. User may keep only that single AxisVM window open that is relevant for the current data flow (usually the last opened window). Other windows can be closed, the COM Server Warning is irrelevant in their case (it does not destroys or influences the current, active dataflow).
- **Inactive/blind AxisVM window opens:** inactive/blind AxisVM window might open during the start of Grasshopper. (The reason is that Grasshopper connects to the AxisVM Catalogs of cross-sections and materials during the loading process of the add-on.) This window is not necessary for the add-on operation, it can be closed. Even the "COM Server Warning" message box that appears can be closed by clicking on OK, since the COM connection used for data transfer is set up just at the moment when the GrasshopperToAxisVM sender component receives a valid Settings input.
- **Re-opening of AxisVM after closing:** If the AxisVM is closed, re-opening is possible only if a new sender component is placed onto the canvas and connected with the inputs. Reason: the sender component initializes itself only once.
- **Special domains freeze from Number sliders:** If the parameters of the special domains (e.g. ribbed, hollow core, composite rib, trapezoid etc.) are modified using Number sliders, the AxisVM may freeze during the redraw process. Possible countermeasures: 1) type discrete values instead of pulling the number slider, 2) use the Lock solver option in Grasshopper during the tuning the Number sliders.
- **Checking model progress bar freeze:** The usage of Check Double option on the Settings component may cause that the progress bar "Checking model..." freeze in AxisVM. Usually, the software operates normally in spite of this, but sometimes it is required to open a new AxisVM (by replacing the GrasshopperToAxisVM component in the script with a new one, pulled from the component palette).
- **Line element eccentricity (Ref):** the alignment points of the line element remain Center Center in spite of any other setting. API functionality review is in progress.
- **Nodal support intermediate spring characteristics:** if the nodal support stiffness values are changed via Number sliders, many intermediate spring characteristics are automatically generated and stored in AxisVM. Countermeasure: use Panel to set the nodal support stiffness.
- **If there are objects created directly inside the AxisVM model by the user,** that can cause renumbering of nodes, elements, supports etc. and this may leads to disjoint of objects, multiple sent or unsent objects. It is recommended to make any modifications on the model directly inside AxisVM when the data transfer from Grasshopper is finished and no more parameter change is expected.