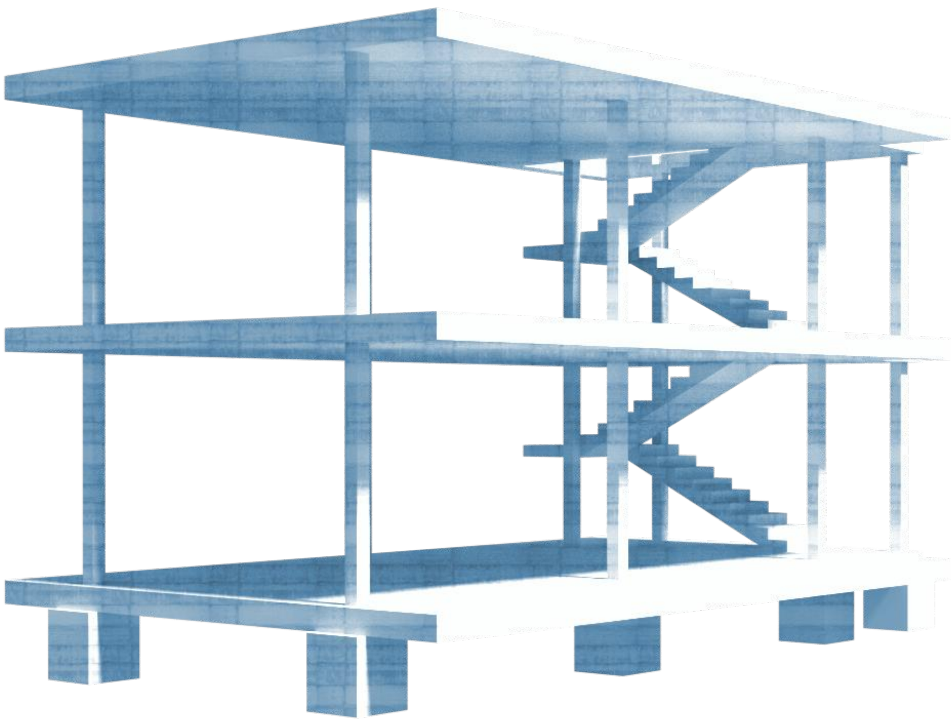
Bullet Constraints Builder

User Manual

Version 2.36
for BCB v3.49

11. July 2021



Laurea University of Applied Sciences
Kai Kostack & Oliver Walter

Table of Contents

I INTRODUCTION

The Bullet Constraints Builder (BCB) is at this stage first and foremost suitable to simulate the effects of major structural deficiency in composite reinforced concrete structures such as incapacitated load bearing elements e.g. beams or pillars. While the BCB's formulas for the strength evaluation of reinforced concrete (RC) elements have been approved by civil engineers during the three yearlong R&D project INACHUS at LAUREA, University of Applied Sciences, the formulas for the strength of steel members are not yet approved.

The BCB has been validated by comparison in a few collapse cases in which the simulation results showed a good affinity with the real-world collapse shapes. However, virtual collapse simulations in general can't claim absolute authenticity and the BCB results should be used with careful consideration in critical applications.

A rich set of functionalities allow specialists to setup building models, define the interdependencies between the structural elements or define collapse scenarios. Nevertheless, the tool can also be used by amateurs. With a few simple steps the user can load predefined models, setup collapse scenarios and start the simulation to observe the effects.

Technically this software is installed as an add-on from within Blender. It extends Blender's basic physics functionality with a sophisticated toolset to simulate load dynamics in building structures. It establishes constraining connections between loose rigid bodies that incorporate real world parameters.

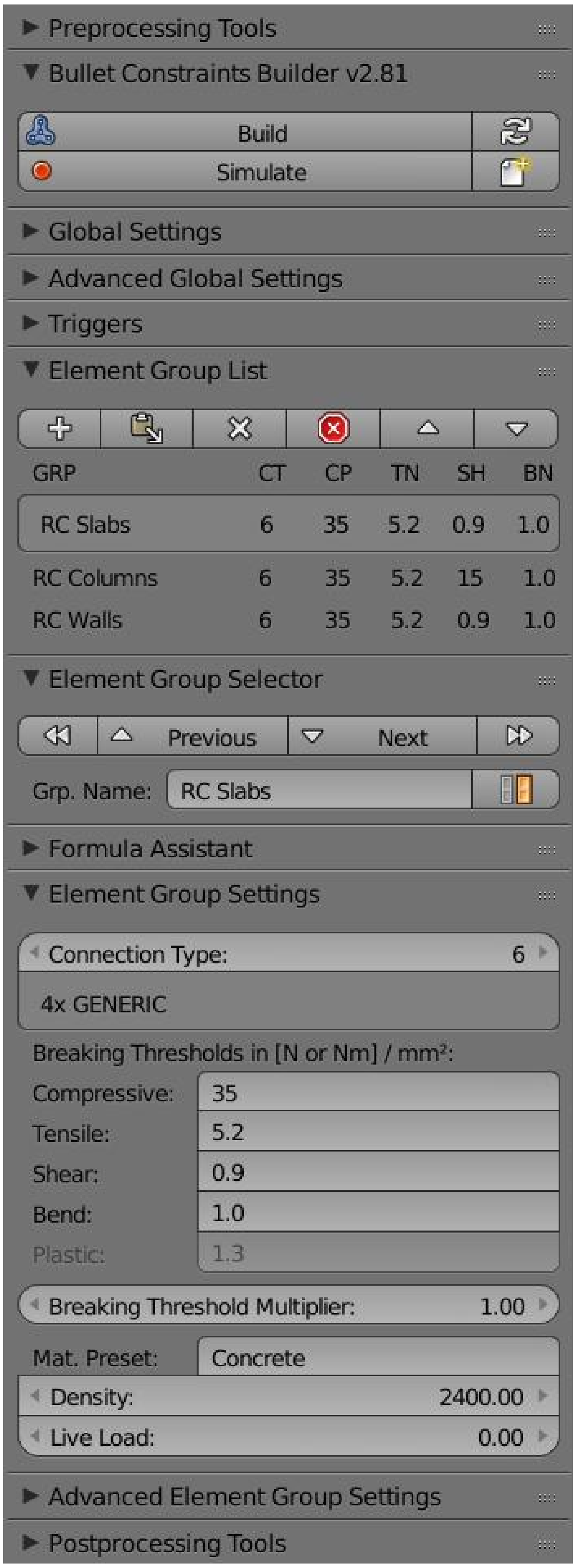
Refer to the BCB Installation Guide for further assistance in this regard.

In this document all individual settings and options of the BCB are explained.

To learn more about the workflow we recommend to read our tutorials.

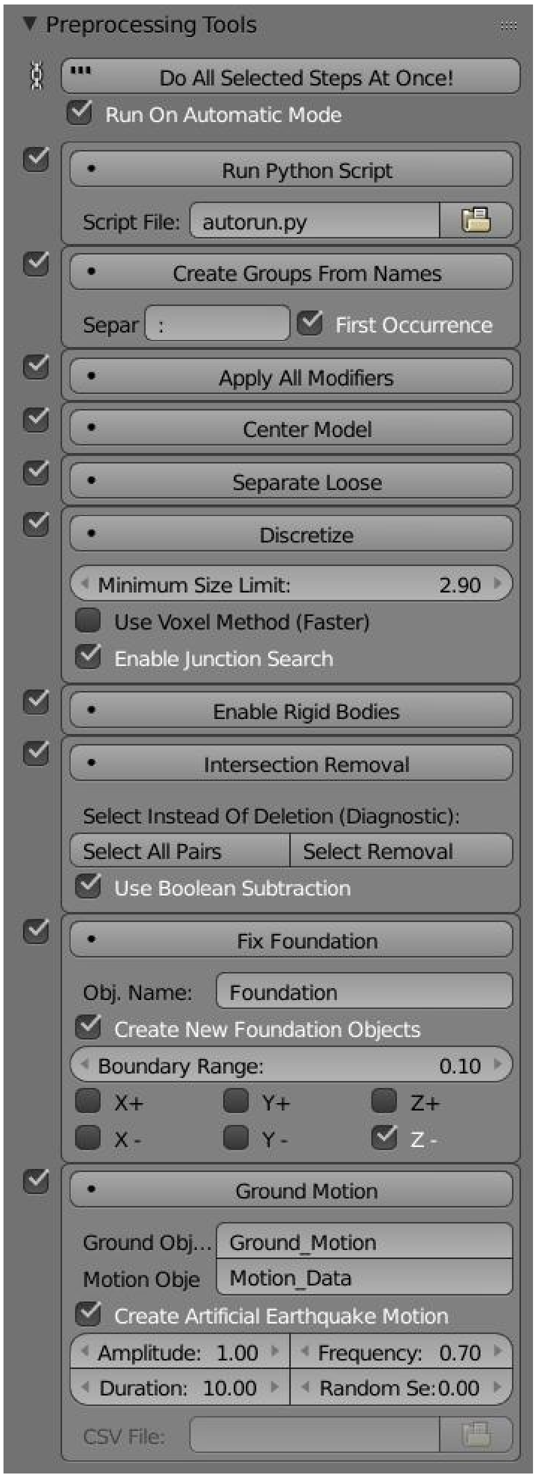
All documention can be found here:
https://github.com/KaiKostack/bullet-constraints-builder/tree/master/doc

II USER INTERFACE



1. Preprocessing Tools

The *Preprocessing Tools* are tools to prepare the mesh topology of a model for simulation. They provide the entire workflow from the imported CAD model (or others) to the completely setup and discretized model ready to be simulated.



1.a Do All Selected Steps At Once!

Executes all selected tools in the order from top to bottom. Select those tools you want to be invoked during the batch process by ticking the checkboxes in the left column. These checkboxes are also taken into account for automatic mode.

Run On Automatic Mode

Enables that preprocessing will be performed on *Automatic Mode*. To avoid accidental double execution, this will be disabled whenever a preprocessing tool is activated manually, but it can be activated again at any time.

1.b Run Python Script

Executes a user-defined Python script for customizable automatization purposes (e.g. batch import or general scene management).

Script File

Enter the filename of an existing Python script, either within the .blend file or extern.

1.c Create Groups From Names

Creates groups for all selected objects based on a specified naming convention and adds them also to the element groups list.

Separator

Defines a key character or string to derive the group names from the object names in the scene. Example: An object name 'Columns:B4' with separator ':' will generate a group named 'Columns' containing all objects with this phrase in their names.

First Occurrence

Enables first occurrence search of the separator within an element name for cases when there are more than one separator included, if disabled the last occurrence is used.

1.d Apply All Modifiers

Applies all modifiers on all selected objects.

1.e Center Model

Shifts all selected objects as a whole to the world center of the scene.

1.f Separate Loose

Separates all loose (not connected) mesh elements within an object into separate objects, this is done for all selected objects.

1.g Discretize

Discretizes (subdivides) all selected objects into smaller segments by splitting them into halves as long as a specified minimum size is reached.

Minimum Size Limit

Discretization size in m this tool tries to reach by discretization. To enforce regularity at all times, elements afterwards can deviate in size to some extent from the target size. For booleans (default method): The minimum dimension value serves as limit for an element still being considered for subdivision, at least two dimension axis must be above this size. After discretization no element will be larger than this value anymore, although they can be smaller up to 50%.

Use Voxel Method (Faster)

Use Voxel Method (Faster)", default=0, description="Enables the voxel based discretizaton method and geometry is converted into cuboid-shaped cells. While this method has the disadvantage that it can't keep mesh details such as curved surfaces, round columns or mural reliefs, it is extremely fast compared to the default boolean based method and can create thousands of new elements within seconds. Also note that this method is limited to odd subdivision level numbers [1,3,5,7..], so you basically can't split an element into two for instance but only into three, five and so on.

Enable Junction Search

Tries to split cornered walls at the corner rather than splitting based on object space to generate more clean shapes.

1.h Enable Rigid Bodies

Enables rigid body settings for all selected objects.

1.i Intersection Removal

Detects and removes intersecting objects (one per found pair). Intesecting objects can be caused by several reasons: accidental object duplication, forgotten boolean cutout objects, careless modeling etc.

Select All Pairs

Selects element pairs intersecting each other, even those which could have been resolved automatically by the algorithm, for review by user.

Select Removal

Selects elements meant to be removed by the algorithm without removing them for review by user.

Use Boolean Subtraction

Uses boolean operations to resolve overlapping elements. Their geometries will be subtracted from each other and the collision shapes will be switched to 'Mesh'. (For accurate simulations it is strongly recommended to resolve such intersections manually and leave this option disabled.)

1.j Fix Foundation

Either uses name based search to find foundation objects or creates foundation objects for all objects touching the overall model boundary box. These foundation objects will be set to be 'Passive' rigid bodies.

Object Name

Enter a name (or substring) for elements which should be set to 'Passive' in rigid body settings.

Create New Foundation Objects

Enables generation of additional rigid body objects to serve as anchors adjacent to the selected model objects.

Boundary Range

Internal margin in m for the model boundary box to include also objects within a certain distance from the outer border. This value should always stay smaller than Discretization Size divided by 2 because otherwise foundation elements can overlap user elements.

X+ X- Y+ Y- Z+ Z-

Enables this side of the overall model boundary for which fixed foundation objects will be created.

1.k Ground Motion

Attaches all selected passive rigid body objects to a specified and animated ground object. This can be useful for simulating earthquakes through a pre-animated ground motion object like a virtual shake table.

Ground Object

Enter the name of a ground object here and the passive foundation objects will automatically be attached to it. If it is not existing it will be created at the underside of the active rigid body boundary box.

Motion Object

Enter the name of an optional motion data object here and the ground object will automatically be attached to it. This can be useful in case animation data should be manageable completely separate from the ground object.

Create Artificial Earthquake Motion

Enables generation of artificial ground motion data based on noise functions, this can be useful if there is no real world ground motion data available.

Amplitude

Amplitude of the artificial earthquake to be generated in m (because of the random nature of the noise function this should be taken as approximation).

Frequency

Frequency of the artificial earthquake to be generated in Hz (because of the random nature of the noise function this should be taken as approximation).

Duration

Duration of the artificial earthquake to be generated in seconds.

Random Seed

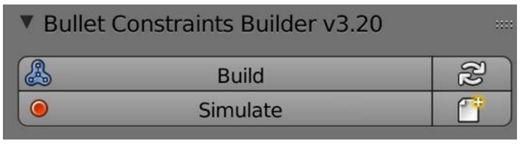
Seed number for the random noise function used to generate the artificial earthquake, modification will change the characteristics of the motion.

CSV File

Enter filename or search for earthquake time history file as plain ASCII text with comma-separated values (.csv). File structure: 4 columns: t [s], X [m/s²], Y [m/s²], Z [m/s²]. Lines starting with '#' are skipped.

2. *Main Menu*

2.a Build / Update



- <u>If no old BCB constraint settings are being loaded:</u>
  Starts building process and adds constraints to selected elements.
- If old BCB constraint settings are being loaded:
  Update old constraints with the new settings (faster).

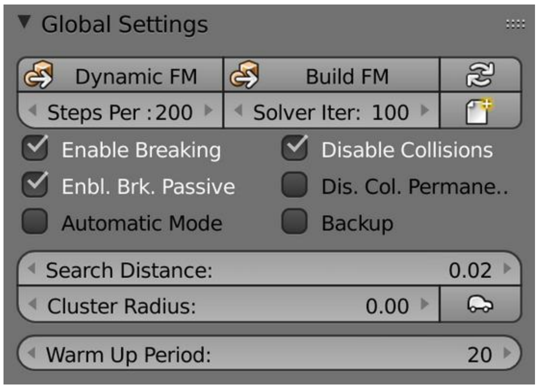
2.b Store / Get / Delete Scene Settings

- <u>If no connections are being built yet:</u>
  Stores actual configuration data in current scene.
- <u>If BCB settings are stored within scene:</u>
  Get settings from scene and update GUI.
- <u>If BCB and constraint settings are stored in scene:</u>
  Get settings and constraint data from scene and update GUI. One can update constraints after that instead of rebuilding constraints from scratch.
- <u>If BCB and constraint settings are already loaded:</u>
  Removal of all BCB related data from scene, including constraints, scaling and bevel changes, and additional facing meshes.

Simulation

Starts the rigid body simulation and creates a *bake* of it for later real-time playback. A *build* is invoked beforehand if not already done. Use of this button instead of the regular Blender baking is crucial because BCB constraints require to be monitored on per frame basis for the entire simulation.

3. Global Settings



Build FM (Fracture Modifier)

Builds and simulates with help of the Fracture Modifier (special Blender version required). 'Simulate FM' will simulate scientifically like 'Simulate'.

Dynamic FM

Builds and simulates with help of the Fracture Modifier (special Blender version required). But also enables dynamic geometry shattering for more realistic appearance but is actually non-scientific.

Import BCB Configuration Data

Imports BCB config data from an external file located in the render output folder.

Export BCB Configuration Data

Exports BCB config data to an external file located in the render output folder.

Steps Per Second

Number of simulation steps taken per second (higher values are more accurate but slower and can also be more instable).

Solver Iterations

Number of constraint solver iterations per simulation step (higher values are more accurate but slower).

Enable Breaking

Enables breaking for all constraints.

Disable Collisions

Disables collisions between connected elements until breach.

Disable Collisions Permanently

Disables collisions between initially connected elements permanently. This can help to make simulations with intersecting geometry more stable at the cost of accuracy.

Automatic Mode

Enables a fully automated workflow for extremely large simulations (object count-wise) were Blender is prone to not being responsive anymore. After clicking *Build* these steps are being done automatically:

1. Building of constraints
2. Simulation (and baking)
3. Clearing constraint and BCB data from scene

Backup (for Automatic Mode)

Enables saving of a backup .blend file after each step for automatic mode, whereby the name of the new .blend ends with `_BCB´.

Search Distance

Search distance to neighbor geometry based on the boundary box of the elements in m.

Cluster Radius

Search distance to neighbor constraints. Close constraint objects will be bundled into clusters in m. This can be important if connection types other than 1 are used to ensure rotation is possible as the cluster serves as pivot point. See also the *Technical Details* section of this document.

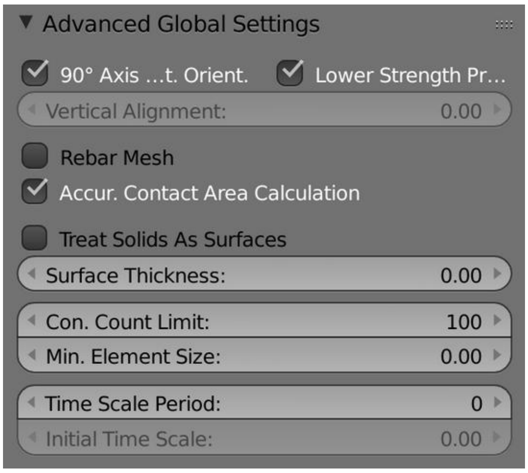
Estimate Cluster Radius

To automatically estimate an appropriate Cluster Radius from the selected elements in the scene the button next to the *Cluster Radius* field can be used (even if you already have built a BCB structure only selected objects are considered).

Warm Up Period

Disables breakability of constraints for an initial period of the simulation (frames). This is to prevent structural damage caused by the gravity impulse on start.

4. Advanced Global Settings



90° Axis Snapping for Constraint Orientation

Enables axis snapping based on contact area orientation for constraints rotation instead of using center to center vector alignment (old method).

Lower Strength Priority

Gives priority to the weaker breaking threshold of two elements from different element groups with same Priority value to be connected, if disabled the stronger value is used for the connection.

Vertical Alignment

Enables a vertical alignment multiplier for connection type 4 or above instead of using unweighted center to center orientation (0 = disabled, 1 = fully vertical).

Rebar Mesh

Enables creation of a rebar mesh on build or export execution using the settings from the Formula Assistant. This mesh is meant for diagnostic purposes only, it is not required nor used for the simulation. It is also not very accurate for very small elements as the rebar count is converted from the definition to the actual element size with a minimum limit of 4 bars per element.

Accurate Contact Area Calculation

Enables accurate contact area calculation using booleans for the cost of an up to 20x slower building process. This only works correct with solids i.e. watertight and manifold objects and is therefore recommended for truss structures or steel constructions in general. If disabled a simpler boundary box intersection approach is used which is only recommended for rectangular constructions without diagonal elements like reinforced concrete buildings.

Treat Solids As Surfaces

Enforces treatment of solid elements as surface elements. This has impact on discretization and mass calculation.

Surface Thickness

Artificial thickness for non-manifold elements (surfaces). If the element is solid this value will be added as extra margin to the detected contact area.

Connection Count Limit

Maximum count of connections per object pair (0 = unlimited).

Minimum Element Size

Deletes connections whose elements are below this diameter and makes them parents instead. This can be helpful for increasing performance on models with unrelevant geometric detail such as screwheads.

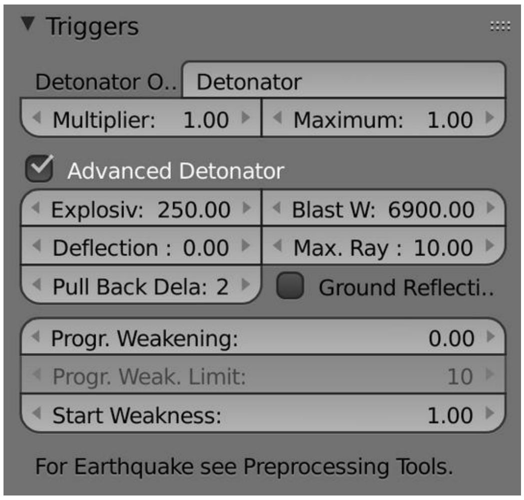
Time Scale Period

Use a different time scale for an initial period of the simulation until this many frames has passed (0 = disabled).

Initial Time Scale

Use this time scale for the initial period of the simulation, after that it is switching back to default time scale and updating breaking thresholds accordingly during runtime.

5. Triggers



Triggers are used to trigger the collapse of a structure in a specific way. However, in this panel only triggers which affect the structural building routine are located, ***Earthquake*** for instance can be found under *Preprocessing Tools* since it requires a ground object.

Detonator Object

Enter name of an object to be used to simulate the effects of an explosion. This feature replicates the damage caused by such an event by weakening the constraints within range of the object. It is recommended to use an Empty object with a sphere shape for this. The damage is calculated as gradient of the distance mapped to the size, from 200% weakening at center to 0% at boundary.

Multiplier

Multiplier the weakening gradient strength derived from the *Detonator* object space will be multiplied with.

Maximum

Detonator influence maximum to limit the weakening.

Advanced Detonator

Enables advanced detonation blast wave simulation using animated force fields and the *Time Scale Period* functionality. If enabled the detonator object will be used as origin to place the force fields but is ignored otherwise.

This feature also requires a *Time Scale Period* > 0 to be set to allow the blast wave to travel the desired distance. Also notice that too large *Initial Time Step* values can have a negative impact on blast wave simulation accuracy.

We are using the blast wave model by Kinney, G.F. and K.J. Graham, 1985. Explosive Shocks in Air. 2nd Edn., Springer Verlag, ISBN-10: 0387151478

Explosive Mass

Mass of the explosive in kg TNT equivalent.

Blast Wave Velocity

Velocity with which the blast wave is traveling in m/s (depends on frame rate).

Deflection Multiplier

Detonation deflection multiplier for open/confined spaces. Larger values will increase the blast wave impact on revealed structures such as outside walls compared to protected elements inside of buildings (0 = no difference).

Maximum Ray Distance

Maximum distance in m for ray casting for confined space detection.

Pull Back Delay

Delay in frames until the negative pressure wave follows the positive one (pressure will be divided by this value to keep overall pressure consistent).

Ground Reflection

Enables reflection of the blast wave from the ground by adding duplicate force fields beneath the ground level.

Progressive Weakening

Enables *Progressive Weakening* of all breaking thresholds by the specified factor per frame (starts not until timeScalePeriod and warmUpPeriod have passed). This can be used to enforce the certain collapse of a building structure after a while.

Progressive Weakening Limit

For progressive weakening: Limits the weakening process by the number of broken connections per frame. If the limit is exceeded weakening will be disabled for the rest of the simulation.

Start Weakness

Start weakness as factor all breaking thresholds will be multiplied with. This can be used to quick-change the initial thresholds without performing a full update.

6. Element Group List

Element groups can be used to define different material properties to certain groups of objects.

The controls within the list box can be used to add, move and select element groups as desired. All settings below the list box belong to the selected group. A short overview of those settings is displayed in the list box as well.

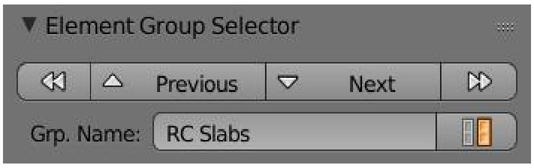
Further, there is the option to pick materials from a selection of preset materials which can be modified after addition to the list. Every preset can be created from scratch by the user, there is no need to use presets to initialize specific materials.

For a connection by default only the weakest evaluated breaking threshold per degree of freedom from both element group settings is used, however, it is possible to invert this behavior by enabling *Lower Strength Priority.*

In case two elements of different element groups needs to be connected the order of the list defines the priority for conflicting connection settings. It is recommended to avoid such configurations.

7. Element Group Selector

In order to keep the element group list collapsable the selector is located in an extra panel. Use the arrow buttons to navigate through the element group list above.



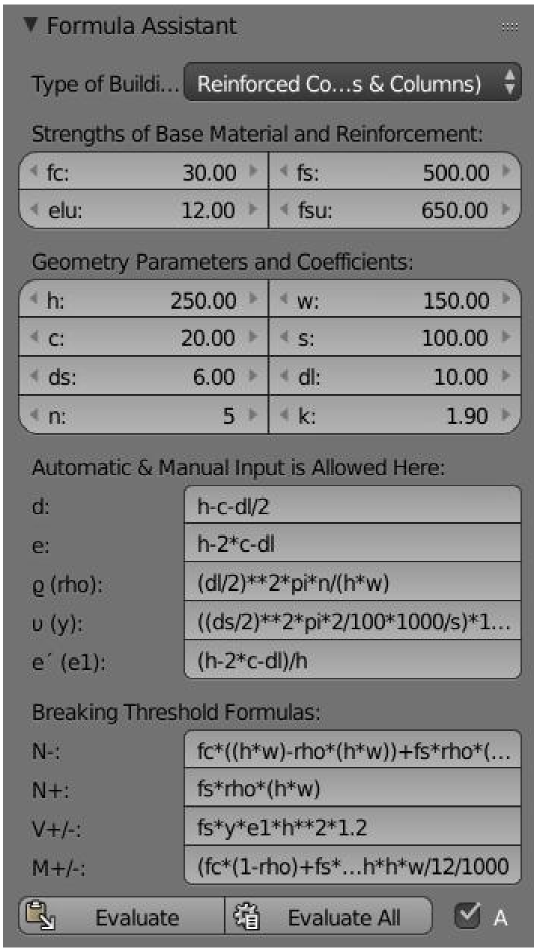
Group Name

The name of the chosen element group.

Select Group

Selects objects belonging to this element group in viewport.

8. Formula Assistant



The *Formula Assistant* provides an convenient way for structural engineers to enter their building configurations.

Type of Building Material

Select a formula assistant for a specific type of structural element. Formulas and UI might change depending on the requirements. Currently available types are:

- Reinforced Concrete (Beams & Columns)
- Reinforced Concrete (Walls & Slabs)

For reinforced concrete types the user interfaces of the formula assistants are identical except for the formula for V+/-.

Strength of Base Material and Reinforcement

fc

Yield strength of concrete (N/mm²).

fs

Yield strength of reinforcement irons (N/mm²).

elu

Ultimate elongation of reinforcement irons (%).

fsu

Ultimate breaking strength of reinforcement irons (N/mm²).

Density Concrete

Density for concrete (kg/m³).

Density Reinforcement

Density for reinforcement / steel (kg/m^3).

h

Height of element (mm). Leave it 0 to pass it through as variable instead of a fixed number.

w

Width of element (mm). Leave it 0 to pass it through as variable instead of a fixed number.

c

Concrete cover thickness above reinforcement (mm).

s

Distance between stirrups (mm).

ds

Diameter of steel stirrup bar (mm).

dl

Diameter of steel longitudinal bar (mm).

n

Number of longitudinal steel bars.

d

Distance between the tensile irons and the opposite concrete surface (mm).

e

Distance between longitudinal irons (mm).

ϱ (rho)

Reinforcement ratio = As/A.

υ (y)

Shear coefficient (asw*10/d) (% value).

e' (e1)

Distance between longitudinal irons in relation to the element height: e/h (% value).

N- (Advanced)

Compressive breaking threshold formula.

N+ (Advanced)

Tensile breaking threshold formula.

V+/- (Advanced)

Shearing breaking threshold formula.

M+/- (Advanced)

Bending or momentum breaking threshold formula.

Evaluate

Combines and evaluates above expressions for constraint breaking threshold calculation. It is recommended to choose a Connection Type with 7x Generic constraints to get the best simulation results.

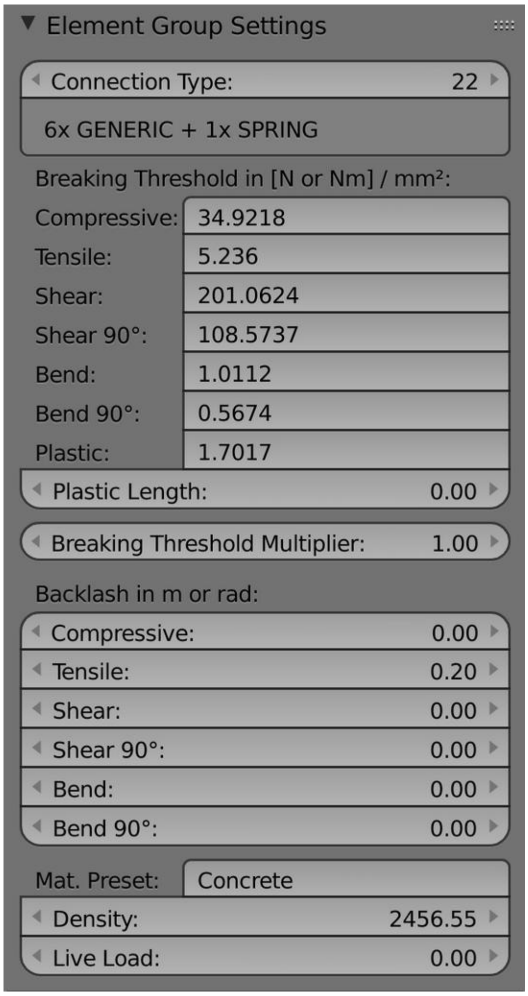
Evaluate All

Combines and evaluates expressions for every element groups with active Formula Assistant. Warning: Use this with care as it will overwrite also manually changed breaking thresholds for these element groups.

Advanced

Shows advanced settings and formulas.

9. Element Group Settings



Connection Type

Connection type ID for the constraint presets defined by this script, see the *Technical Details* section of this document.

Compressive Breaking Threshold

Real world material compressive breaking threshold in N/mm².

Tensile Breaking Threshold

Real world material tensile breaking threshold in N/mm² (not used by all connection types).

Shearing Breaking Threshold

Real world material shearing breaking threshold in N/mm² (not used by all connection types).

Shearing Breaking Threshold 90°

Breaking threshold when height and width of the reference element in the *Formula Assistant* is flipped. This is used to adapt the reference dimensions to the actual geometry.

Bending Breaking Threshold

Real world material bending breaking threshold in N/mm² (not used by all connection types).

Bending Breaking Threshold 90°

Breaking threshold when height and width of the reference element in the Formula Assistant is flipped. This is used to adapt the reference dimensions to the actual geometry.

Plastic Breaking Threshold

Real world material ultimate tensile breaking threshold in N/mm^2 (only used for spring connection types).

Plastic Length

Length of the springs used for plastic deformation in m. If 0 is entered the distance between the element´s centroids is used (default).

Breaking Threshold Multiplier

Multiplier to be applied on all breaking thresholds for constraint building. This can be useful for quickly weaken or strengthen a given element group without changing the original settings.

Backlash Compressive Axis

Maximum distance in m through which any connected element may be moved in compressive direction without applying force or motion to the next element.

Backlash Tensile Axis

Maximum distance in m through which any connected element may be moved in tensile direction without applying force or motion to the next element.

Backlash Shearing Axis

Maximum distance in m through which any connected element may be moved in shearing direction without applying force or motion to the next element.

Backlash Shearing Axis 90°

Maximum distance in m through which any connected element may be moved in shearing direction, with h and w swapped (rotated by 90°), without applying force or motion to the next element.

Backlash Bending Axis

Maximum angle in rad through which any connected element may be moved in bending direction without applying force or motion to the next element.

Backlash Bending Axis 90°

Maximum angle in rad through which any connected element may be moved in bending direction, with h and w swapped (rotated by 90°), without applying force or motion to the next element.

Required Vertex Pairs (obsolete, hidden in UI and disabled)

How many vertex pairs between two elements are required to generate a connection.

Material Preset

Preset name of the physical material to be used from Blender's internal database. See Blender's Rigid Body Tools for a list of available presets.

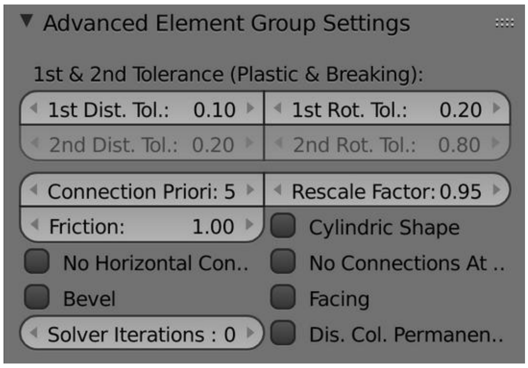
Material Density

Custom density value (kg/m³) to use instead of material preset (0 = disabled).

Live Load

Additional weight representing live load which will be added to the total mass with respect to floor area (kg/m²).

10. Advanced Element Group Settings



1st Distance Tolerance

First deformation tolerance limit for distance change in percent for connection removal or plastic deformation (1.00 = 100 %).

1st Rotation or Bending Tolerance

First deformation tolerance limit for angular change in radian for connection removal or plastic deformation.

2nd Distance Tolerance

Second deformation tolerance limit for distance change in percent for connection removal (1.00 = 100 %). Smaller or greater distances will force the connection to detach completely.

The Formula Assistant might calculate this setting automatically on evaluation, it will appear greyed out then.

2nd Rotation or Bending Tolerance

Second deformation tolerance limit for angular change in radian for connection removal. Smaller or greater angles will force the connection to detach completely.

The Formula Assistant might set this to 0 which means that this tolerance will be calculated later during the constraint building phase individually for each connection using Formula Assistant settings, there is no need to change it back then.

Connection Priority

Changes the connection priority for this element group which will override that the weaker breaking threshold of two elements is preferred for an connection. Lower Strength Priority has similar functionality but works on all groups, however, it is ignored if the priority here is different for a particular connection.

Rescale Factor

Applies scaling factor on elements to avoid `Jenga´ effect (undesired stability increase caused by incompressible rigid bodies). This has no influence on breaking threshold and mass calculations.

Friction

Coefficient of friction for the given material (dimensionless).

Cylindric Shape

Interpret connection area as round instead of rectangular (ar = a *pi/4). This can be useful when you have to deal with cylindrical columns.

No Horizontal Connections

Removes horizontal connections between elements of different element groups. This can be useful for masonry walls touching a framing structure without a particular fixation.

No Connections At All

Removes connections between elements of different element groups. This can be useful for rigs with predefined constraints where groups should stay completely detached from another even when they are actually touching or overlapping.

Bevel

Enables beveling for elements to avoid `Jenga´ effect (undesired stability increase caused by incompressible rigid bodies). This uses hidden collision meshes and has no influence on breaking threshold and mass calculations.

Facing

Generates an additional layer of elements only for display (will only be used together with bevel and scale option, also serves as backup and for mass calculation).

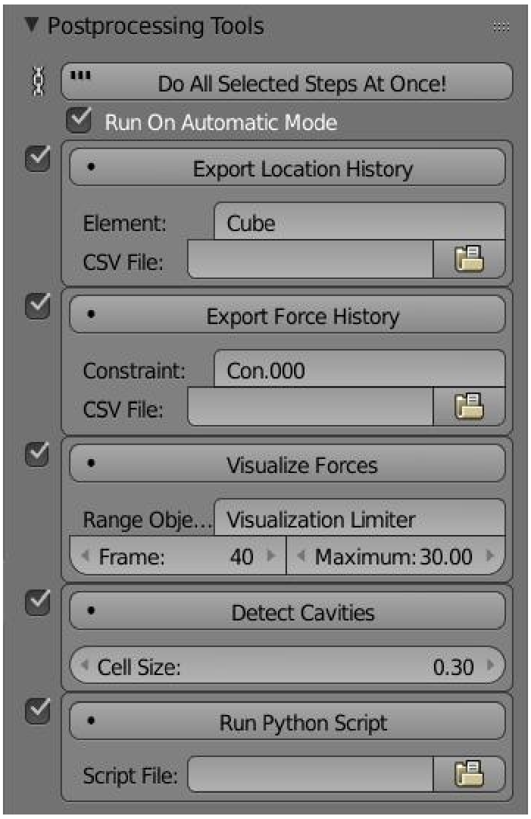
Solver Iterations Override

Overrides the *Constraint Solver Iterations* value of the scene for constraints of this element group if set to a value greater 0. Higher numbers can help to reduce solver induced deformation on elements bearing extreme loads.

Disable Collisions Permanently

Disables collisions between initially connected elements of this element group permanently (overrides global setting).

11. Postprocessing Tools



The *Postprocessing Tools* are tools to analyse, visualize and export simulation results. They expect a simulation to be performed already.

11.a Do All Selected Steps At Once!

Executes all selected tools in the order from top to bottom. Select those tools you want to be invoked during the batch process by ticking the checkboxes in the left column. These checkboxes are also taken into account for automatic mode.

Run On Automatic Mode

Enables that postprocessing will be performed on *Automatic Mode*. To avoid accidental double execution, this will be disabled whenever a postprocessing tool is activated manually, but it can be activated again at any time.

11.b Export Location History

Exports the location time history of an element centroid into a .csv file.

Element

Enter the name of an element for which the location time history should be exported.

CSV Folder

Enter a path or search for folder for data export as plain ASCII text with comma-separated values (.csv).

11.c Export Force History

Exports the force time history for a constraint into a .csv file.

This tool requires the Fracture Modifier for which a special Blender version is needed. Visit graphicall.org/1148 for the FM-enabled Blender build.

Constraint

Enter the name of a constraint for which the location time history should be exported.

CSV Folder

Enter a path or search for folder for data export as plain ASCII text with comma-separated values (.csv).

11.d Visualize Forces

Visualizes forces for constraints in form of spheres generated within the scene whereby each sphere's radius is normalized to the corresponding maximum strength of the connection (each constraint is normalized individually). Accurate values can be found in each sphere's properties.

This tool requires the Fracture Modifier for which a special Blender version is needed. Visit graphicall.org/1148 for the FM-enabled Blender build.

Range Object

Enter the name of a helper object whose dimensions will be used to define for which connections forces should be visualized, i.e. all within its boundary range. For instance an empty object can be placed and scaled accordingly to fit a specific area of interest.

Limit To Foundation Connections

Limits visualization to connections with foundation / passive elements.

Frame

Frame number at which the visualization snap-shot of forces will be taken.

Normalize To Breaking Threshold

Normalizes the visualizer to the breaking thresholds of the constraints, so that red color always means close to failure.

Maximum

Maximum force to be expected, actual forces will be normalized accordingly. This will only influence the appearance of the visualizer, but the readout value stored within the visualizer's properties will not be modified.

11.e Detect Cavities

Visualizes cavities on the selected mesh in form of a cell grid where each cell represents an air pocket large enough to contain the cell.

Cell Size

Cell size for cavity detection algorithm to consider empty spaces (smaller values are more accurate but take longer to compute).

11.f Run Python Script

Executes a user-defined Python script for customizable automatization purposes (e.g. batch import or general scene management).

Script File

Enter the filename of an existing Python script, either within the .blend file or extern.

III TECHNICAL DETAILS

1. Update / Rebuild

Most changes in material properties or in the element groups are covered by the update functionality. There are however changes that will make a rebuild of the whole structure necessary. The remove button has to be pressed before you can rebuild.

Here is a not conclusive list of changes that require a full rebuild:

- Adding or removing elements to or from the structure (it is however possible to put elements into another layer to make them invisible for the simulation)
- Adding or removing constraints to or from the structure

These changes outside of the BCB panel require an update:

- Changing of the simulation steps rate in the Rigid Body World panel of the scene properties, as it influences the breaking thresholds (but the constraint iteration rate can be changed)

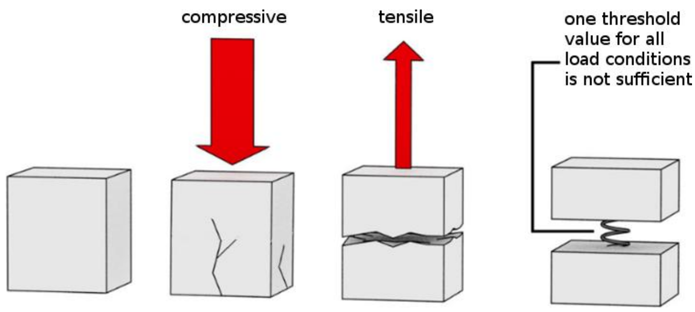
2. Simulate

Since the introduction of multiple constraints per connection a custom simulation (or *bake* in Blender terms) option has been implemented. As constraints shared by a connection have no awareness of each other it has become necessary to manage these during the simulation and baking. Per each frame an event handler is invoked which is checking all connections if at least one constraint per connection has been detached and then it automatically detaches all other constraints within the particular connection as well. Individual left over constraints otherwise would heavily confuse the simulation, which can lead to very unstable behavior like exploding structures especially with higher step rates.

Detached connections are detected indirectly by measuring element to element relationships. More precisely, this is done by verifying changes in distance between elements and changes in their relative rotation to each other. Occurring differences are then compared to the *Tolerance* value. A change in distance which is only or mainly caused by a change in rotation will be recognized by the system as the tolerance threshold will be expanded accordingly to take this into account, so undesired detaching at least in this case should not happen.

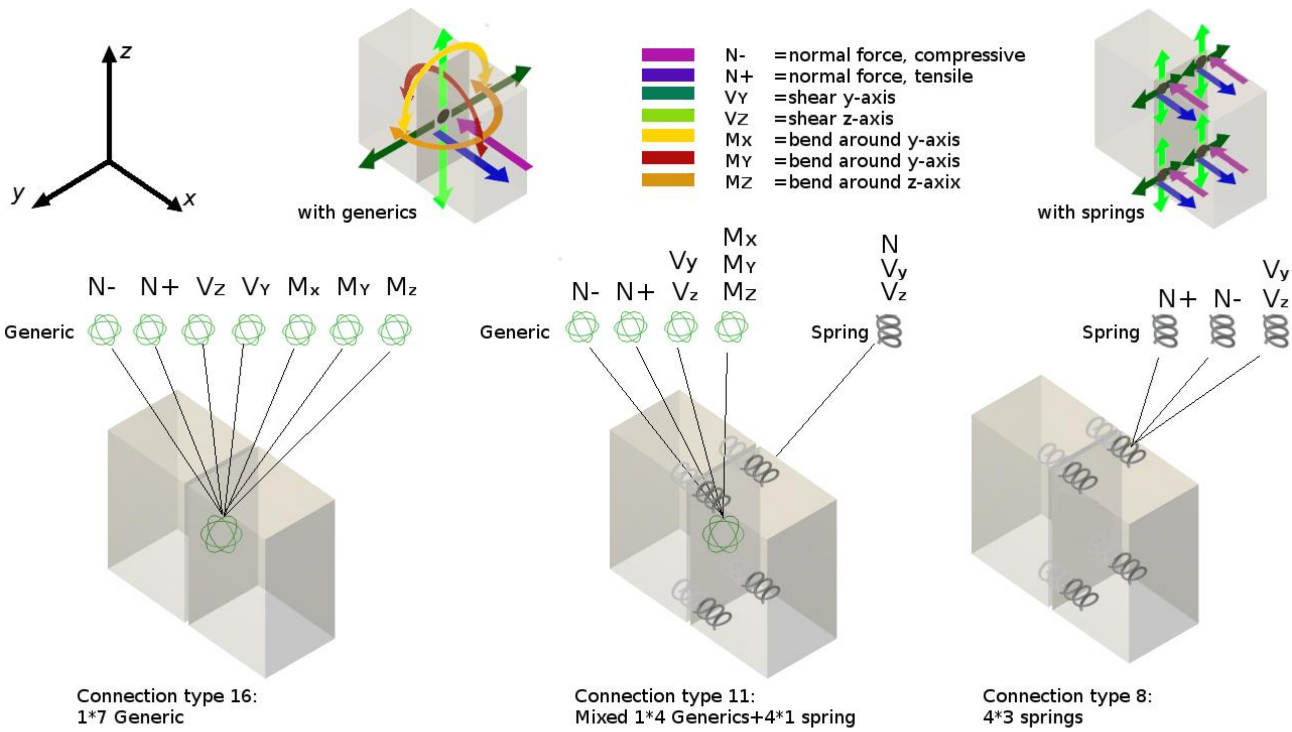
3. Connection types (CT)

The key to realistic collapse simulations is the accurate description of connections between elements. To accomplish this target the BCB is using following principles:



Different strain directions require separate constraints

1. Multiple constraints per element pair to represent the relevant degrees of freedom (DOF)
2. Accurate constraint placement taking into account the contact surface of two elements
3. Evaluation of breaking thresholds based on physical structural properties of the building element



Three connection types in the BCB with generic and spring constraints

Constraints in connections are defined by presets, only the ID from the following list needs to entered into the *Connection Type* field to enable the respective connection type:

CT ID	Name in BCB interface	Const. Count	Short Description
0	PASSIVE	0	Passive (all other connection types will have priority over it)
1	1x FIXED	1	Linear omni-directional + bending breaking threshold
2	1x POINT	1	Linear omni-directional breaking threshold
3	1x POINT + 1x FIXED	2	Linear omni-directional and bending breaking thresholds
4	2x GENERIC	2	Compressive and tensile breaking thresholds
5	3x GENERIC	3	Compressive, tensile + shearing and bending breaking thresholds
6	4x GENERIC	4	Compressive, tensile, shearing and bending breaking thresholds
7	3x SPRING	3	Linear omni-directional breaking threshold with plastic deformability
8	4x SPRING	4	Linear omni-directional breaking threshold with plastic deformability
9	1x FIXED + 3x SPRING	4	Linear omni-directional + bending breaking threshold with plastic deformability (2nd mode)*
10	1x FIXED + 4x SPRING	5	Linear omni-directional + bending breaking threshold with plastic deformability (2nd mode)*
11	4x GENERIC + 3x SPRING	7	Compressive, tensile, shearing and bending breaking thresholds with plastic deformability (2nd mode)*
12	4x GENERIC + 4x SPRING	8	Compressive, tensile, shearing and bending breaking thresholds with plastic deformability (2nd mode)*
13	3 x 3x SPRING	9	Compressive, tensile and shearing breaking thresholds with plastic deformability
14	3 x 4x SPRING	12	Compressive, tensile and shearing breaking thresholds with plastic deformability
15	6x GENERIC	6	Compressive, tensile, shearing XY and bending XY breaking thresholds
16	7x GENERIC	7	Compressive, tensile, shearing XY and bending XY and torsion breaking thresholds
17	6x GENERIC + 3x SPRING	9	Compressive, tensile, shearing XY and bending XY breaking thresholds with plastic deformability (2nd mode)*
18	7x GENERIC + 3x SPRING	10	Compressive, tensile, shearing XY and bending XY and torsion breaking thresholds with plastic deformability (2nd mode)*

19	1x FIXED + 1x SPRING	2	Linear omni-directional + bending breaking threshold with plastic deformability (2nd mode)*
20	1x PNT + 1x FXD + 1x SPR	3	Linear omni-directional and bending breaking thresholds with plastic deformability (2nd mode)*
21	4x GENERIC + 1x SPRING	5	Compressive, tensile, shearing and bending breaking thresholds with plastic deformability (2nd mode)*
22	6x GENERIC + 1x SPRING	7	Compressive, tensile, shearing XY and bending XY breaking thresholds with plastic deformability (2nd mode)*
23	7x GENERIC + 1x SPRING	8	Compressive, tensile, shearing XY and bending XY and torsion breaking thresholds with plastic deformability (2nd mode)*
24	1x SPRING	1	Linear omni-directional + bending breaking threshold with plastic deformability
25	1x POINT + 1x SPRING	2	Linear omni-directional and bending breaking threshold with plastic deformability
26	1x HINGE	1	Linear omni-directional + bending XY breaking threshold

* 2nd mode means that spring constraints are not simulated simultaneously with the main/generic constraints of the connection, only when those break the connection activates springs to simulate plastic deformation (second mode).

4. Constraint placement specifications

4.a Location

Constraints are placed at the center of the contact area boundary box. If bundling into clusters is enabled (Cluster Radius > 0) then all constraint locations will be bundled within a post process based on their previous locations.

4.b Orientation

Constraints will be aligned to the center to center line of both connected elements.

The *Vertical Alignment* setting can be used to make sure skewed but mostly vertically oriented constraints will be rectified. This is done by reducing the X and Y components of the directional vector by a custom factor in range from 0 to 1. This way horizontal oriented constraints will stay horizontal and more diagonal ones will be corrected towards upright. This can help for buildings where one wants to avoid undesired diagonal oriented constraints.

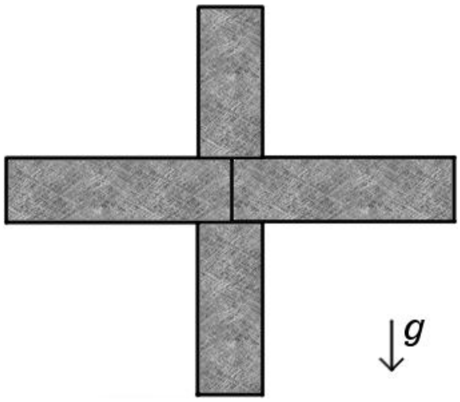
IV MODELING TIPS & CONSIDERATIONS

1. Simulation precision

Coherent looking simulations cannot be expected from coarse building models such as presented in the Dom-Ino House example. We have to bear in mind that single rigid body elements are not breakable! Imagine a column in the middle of a rather large ceiling slab, this column will not be able to punch through that slab element. Breaks are only possible at the joint of two elements. To evaluate forces as authentically as possible object edges must be available close to the occurrence of the highest forces, this is increasingly the case in building models with higher *discretization*.

2. The `Jenga´ effect

By `Jenga´ effect this document refers to describes the characteristic of rigid bodies of not being able to give in or break away if they were stacked upon each other like the wooden blocks of that popular game.



Bad: Elements will not collapse even if all constraints are broken

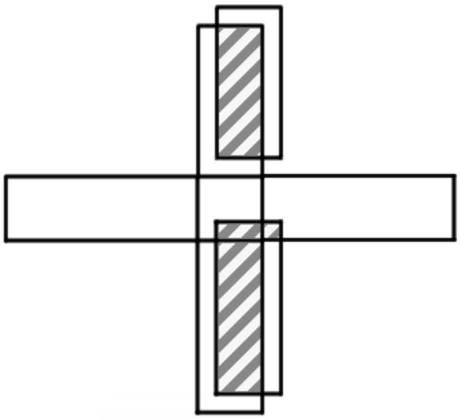
Rigid bodies are by design incompressible and indestructible so one needs to make sure they being hold together only by the connection constraints and that they are able to move away and to collapse, especially if that is the desired outcome of the simulation.

To deal with this issue in a convenient way BCB offers two options, beveling and rescaling, to change the collision shape accordingly. This can help to give all elements of the corresponding group more freedom of motion in all directions.

It is also possible to add an optional facing layer of element duplicates which will hide the visible changes of the underlying collision objects.

3. Element intersections

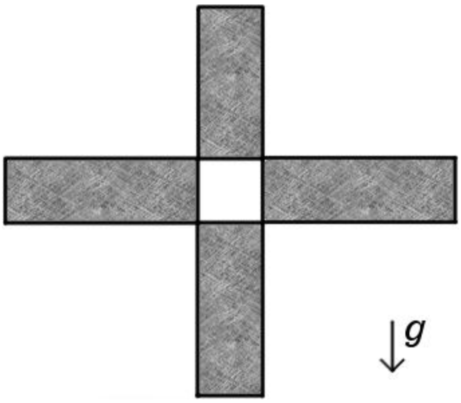
It is important that no active rigid body elements are intersecting each other, this would lead to an explosion like repelling reaction during simulation. Passive rigid body elements however can intersect each other, like for footings of a building that can intersect with a ground plane.



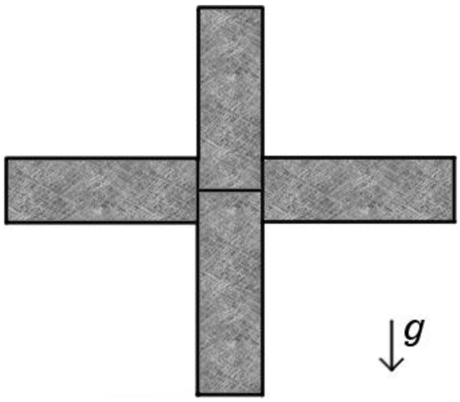
Bad: Elements overlapping others

4. Connection evaluation

The search for connections starts by finding boundary box intersections of all mesh elements. Then for every found element pair the contact area will be calculated and only the contact area is > 0 then a connection is created. You can have gaps between the elements as long as the distance between surfaces lies within *Search Distance*, all boundary boxes will be extended by this value.



Bad: No contact area



Optimal structure modeling