BES322440-L

Dealing with the Structural Analytical Representation in Revit – Hands-On for beginners

Catalin Lang
Autodesk

Learning Objectives

- Learn how to create the analytical representation of the structural elements in Revit
- Learn how to automate the relation between physical and analytical representations using Dynamo
- Learn how to complete the analytical model for structural analysis purpose
- Learn how to integrate the structural analysis in the intelligent models

Description

In order to create an analytical representation of a structural model ready for analysis, there are some transformation, changes, and assumptions that need to be applied. In Revit software, the analytical model is created automatically as the physical model is built. Some structural configurations are not suitable for direct integration with analysis and design software. Adaptive adjustment is required. This could be tedious and time consuming.
In this class, you'll learn how to take full advantage of the analytical representation features in Revit, and how to integrate the structural analysis in the Building Information Modeling (BIM) models.

Speaker(s)

Catalin is a former Autodesk customer, currently Autodesk employee, working in constructions field for over 19 years, going through several branches of the industry, from junior unskilled worker to formwork specialist, storekeeper, project coordinator, project manager, CAD designer, structural designer. Joined Autodesk since 2014 as Quality Analyst. Currently, he is Product Owner for one of the teams that develops Revit and Advance Steel structural features.
He is specialized in structural modeling and detailing.

Create the Analytical Representation of the Structural Elements in Revit

About Structural Analytical Modeling in Revit

The physical model carries information about geometry.
The analytical model is a simplified 3D representation of the structural physical model. It consists of those structural components, geometry, material properties, and loads, that together form an engineering system.
In Revit software, the analytical model is created automatically as the physical model is built.

Setting-up Revit Environment

Visibility and Graphics

Visualization of the analytical model depends on the project template, view template, view parameters, and object styles. The structural analysis project template contains two structural analytical plan views and a 3D analytical view.

Show/Hide Analytical Model

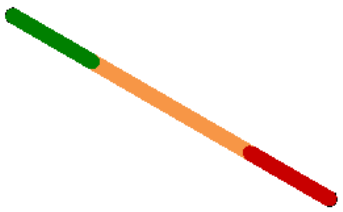
Analytical Model can be display in any View.
In the View Control Bar

- Click Show Analytical Model to display it. It will be displayed with the settings specified in the Visibility/Graphics Override dialog.
- Click Hide Analytical Model to hide it.

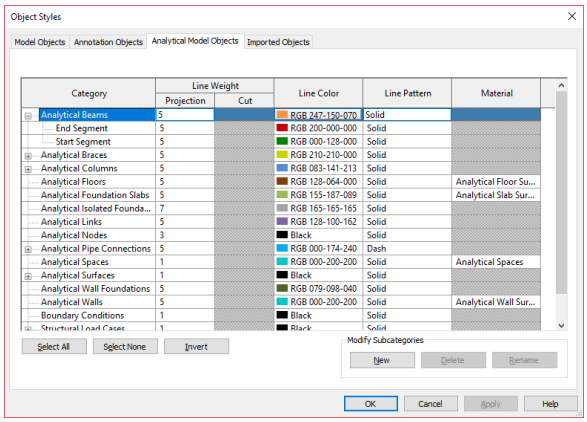
Representation Styles Settings

In the Object Styles dialog you can specify different colors, line weights, or patterns for the beginning and end analytical model segments of columns, beams, or braces.

- Manage tab ➤ Settings panel ➤ Object Styles.
- Go to the Analytical Model Objects tab in the Object Styles dialog.
- Expand the analytical element category ➤ Line's Color/Weight/Pattern

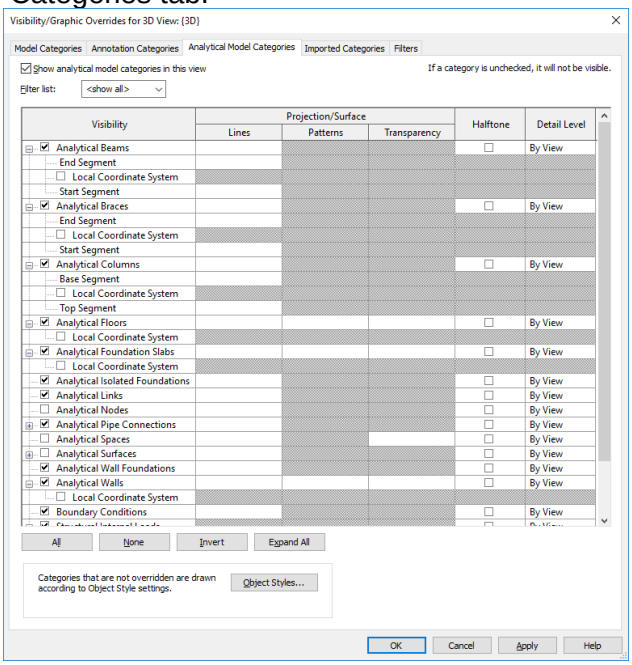


For the linear analytical elements, you can also set the line settings for the Start/End representation.



OBJECT STYLES DIALOG
ANALYTICAL MODEL OBJECTS TAB

The representation settings can be overridden from the Visibility/Graphics Overrides in Analytical Model Categories tab.



VISIBILITY/GRAPHICS OVERRIDES DIALOG
ANALYTICAL MODEL CATEGORIES TAB

Also, to view the Local Coordinate Axis, there is an option that enables its visibility.

Tips:

Analytical Node visibility is disabled by default

Highlight the Correspondent Analytical/Physical Elements

1. Select a structural element.
2. Click Modify <Element> tab ➤ Analytical panel ➤ Highlight Analytical.

The analytical model will highlight.

Similarly, you can highlight the physical model when adjusting analytical models.

Highlighting the physical model

1. Select an analytical model element.
2. Click Modify <Element> tab ➤ Analytical Model Tools panel ➤ Highlight Physical.

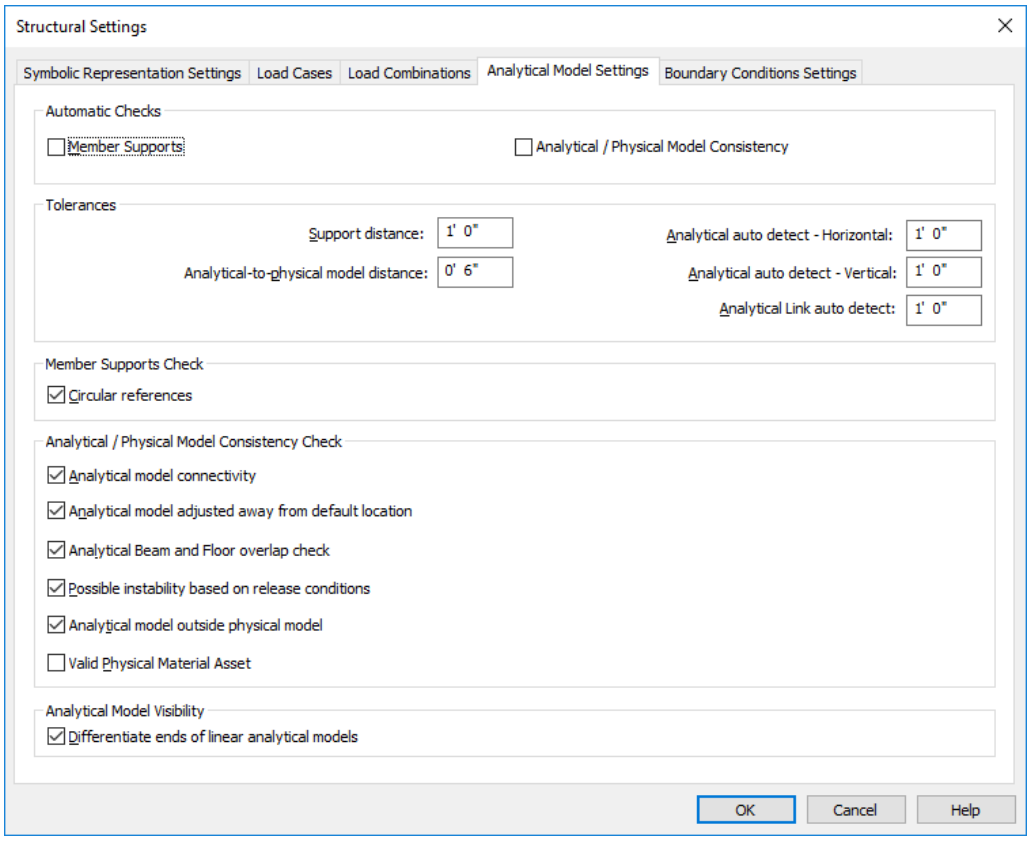
The physical model will highlight.

Analytical Model Settings

The Analytical Model Settings are used to adjust how Revit performs certain tasks on the analytical model.

1. Manage tab ➤ Settings panel ➤ Structural Settings
2. On the Structural Settings dialog ➤ Analytical Model settings

These are project-specific settings, stored within the project.



There are three groups of settings:

Automatic Checks

Automatic analytical model checking provides a warning when the analytical model of your project may not be correct.

1. **Member Supports**.
   - Provides a warning when a member is not supported during model creation or modification. Circular References are specified in the Member Supports Check section of this dialog.
2. **Analytical/Physical Model Consistency**. Provides warnings during element creation or modification for:
   - all unsupported structural elements
   - all inconsistencies found within the analytical model
   - all inconsistencies between the analytical and physical models
   - all analytical elements without a Physical Material Asset assigned.

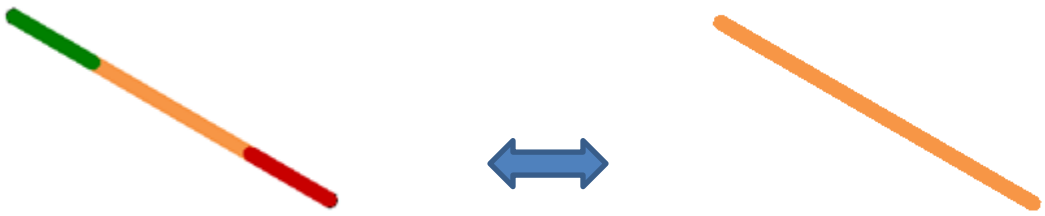
Tolerances

The Tolerances options set tolerances for both the Analytical/Physical Model Consistency check and the Auto-Detect of the analytical model.

- **Support distance** - Specifies the maximum allowable distance between the physical model of an element and the physical model of the supporting element. If it exceeds this tolerance, a warning is issued during a consistency check.
- **Analytical-to-physical model distance** - Specifies the maximum allowable distance between the analytical and physical model. If it exceeds this tolerance, a warning is issued during a consistency check.
- **Analytical auto detect** – **Horizontal** - Specifies the maximum horizontal distance between the analytical and physical model. .
- **Analytical auto detect** – **Vertical** - Specifies the maximum vertical distance between the analytical and physical model.
- **Analytical Link auto-detect** - Specifies the minimum distance in 3D space (horizontal or vertical) in which an automatic analytical link will be created. Analytical Links provide rigidity to the analytical model without adding physical geometry. This tolerance does not take the physical model into account when calculating links.

Analytical Model Visibility

- **Differentiate ends of linear analytical models** – Enable/Disable the markup for the Strat/End segments of the linear analytical elements.



Analytical Model Elements

Structural Elements Representatives in the Analytical Model

There are nine **types** of analytical elements associated with physical model:

- **Analytical Column**
  - Is the representation of the structural column
  - Is derived from the physical column
  - Has a 1:1 relationship with the correspondent structural column
  - Depends on the structural column's transformations
  - Linear straight element
  - It's not differentiated based on vertical or slanted
- **Analytical Beam**
  - Is the representation of the structural framing - beam
  - Is derived from the physical beam
  - Has a 1:1 relationship with the correspondent structural framing - beam
  - Depends on the structural beam's transformations
  - Linear straight or curved element
  - Curved elements can be discretized
- **Analytical Brace**
  - Is the representation of the structural framing - brace
  - Is derived from the physical brace
  - Has a 1:1 relationship with the correspondent structural framing - brace
  - Depends on the structural brace's transformations
  - Linear straight element
- **Analytical Floors**
  - Is the representation of the structural floor
  - Is derived from the physical floor
  - Has a 1:1 relationship with the correspondent structural floor
  - Depends on the structural floor's transformations
  - Planar or curved surface
- **Analytical Walls**
  - Is the representation of the structural wall
  - Is derived from the physical wall
  - Has a 1:1 relationship with the correspondent structural wall
  - Depends on the structural wall's transformations
  - Planar or curved surface
- **Analytical Isolated Foundation**
  - Is the representation of the structural foundation - isolated
  - Is derived from the physical isolated foundation
  - Has a 1:1 relationship with the correspondent structural foundation - isolated
  - Depends on the structural isolated foundation's transformations
  - Point element
- **Analytical Wall Foundation**

- Is the representation of the structural foundation - wall
- Is derived from the physical wall foundation
- Has a 1:1 relationship with the correspondent structural foundation - wall
- Depends on the structural wall foundation's transformations
- Linear straight or curved element

▪ **Analytical Foundation Slab**
- Is the representation of the structural foundation - slab
- Is derived from the physical slab foundation
- Has a 1:1 relationship with the correspondent structural foundation - slab
- Depends on the structural slab foundation's transformations
- Planar or curved surface

▪ **Analytical Line within In-Place Family**
- a linear element consisting of one or more segments
- defined inside the In-Place family by
  - choosing a structural family category for the In-Place component
  - creating a Model Line and assign it from Modify| Line tab ➤ Subcategory panel ➤ Analytical Model […]

Analytical elements **instance parameters** are grouped in:

➢ ***Analytical Model***
- *Analyze as* - Indicates the property type for structural analysis (e.g. column contribution to lateral analysis)
- *Analytical Links* - for beams and columns
- *Discretization parameters* – for beam

➢ ***Analytical Properties***
- *Family Type* - The family type of the element -inherited from the correspondent physical object
- *Physical Material Asset* - The name of a physical asset assigned to the correspondent physical element material
- *Length* – for analytical column, beam, brace, wall
- *Area* – for analytical floor and foundation slab
- *Cross-Section Rotation* – for analytical column, beam, brace, wall
- *Perimeter*– for analytical floor and foundation slab

➢ ***Analytical Alignment*** - Each physical element has optional locations for its analytical projection plane. These projection plane locations are relative either to the levels of the structure or to the structural element itself. Change the location by adjusting the Vertical Parameter on the Analytical Model section of the Properties palette.

➢ ***Release conditions*** – available only for the linear elements. Release conditions are defined by six components:
- Fx - axial force (X direction)
- Fy - shear force (Y direction)
- Fz - shear force (Z direction)

- Mx - torsion force (X direction)
- My - bending moment (Y direction)
- Mz - bending moment (Z direction)

The release conditions are defined for each end.
Release conditions are Fixed, Pinned, Bending Moment, or can be User-defined.

- ***Member End Forces***

Nodes

Nodes are **created** automatically at the linear element's ends and inflection points or at the surface's corners.

The nodes connect analytical elements.

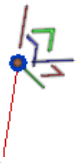
There are two **types** of nodes:

- End Nodes - at the linear element's ends and inflection points
- Intermediate Nodes – at the intersection of a bar end with an analytical element, unless the intersection point coincides with an end node

Visibility and Graphics
- the Nodes visibility is disabled by default in views.
- In Analytical Adjust editor the Nodes are always visible.

Nodes position can be **modified** only from Analytical Adjust editor.
Nodes position can be adjusted by:

- Move (MV)
- Drag
- Drag using the node's coordinate system



- Picking a new host. The new host can be:
  - Another analytical element
  - Face
  - Work Plane

Analytical Node **instance parameters** are grouped in:

- ***Structural***
  - *Connection Status*

Boundary Conditions (Supports)

There are three **types** of supports:

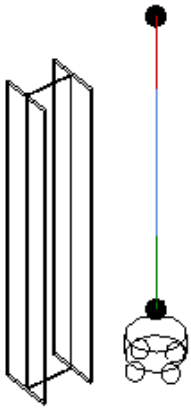
- (Point) – on nodes
- (Line) – on lines
- (Area) – on surfaces

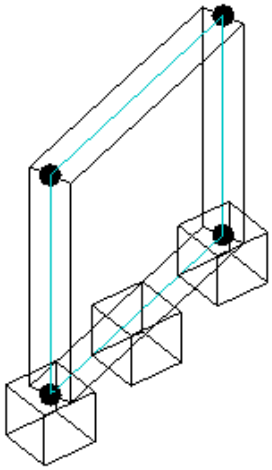
To **create** a boundary condition:

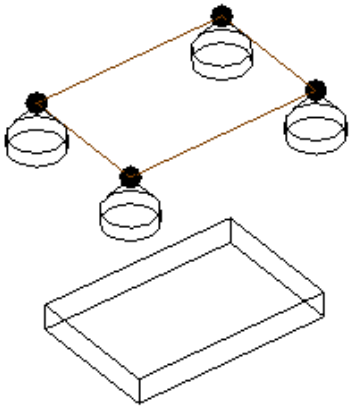
- go to Analyze tab ➤ Analytical Model panel ➤ (Boundary Conditions) To access the boundary condition tools.
- Click Modify | Place Boundary Conditions tab ➤ Boundary Conditions pane ➤ Pick a boundary condition type

Use the Boundary Condition **Settings** tab of the Structural Settings dialog to specify family symbols and adjust the spacing four each boundary condition representation.

- Click Manage tab ➤ Settings panel ➤ Structural Settings
- Select the Boundary Conditions Settings tab
- Under Family Symbol, select a symbol for the Fixed, Pinned, Roller, and User Defined boundary condition states.
- Specify the desired distance in the Area and Line Symbol Spacing field to complete the Boundary Conditions settings.







Boundary Conditions **instance parameters** are grouped in:

- ***Structural Analysis***
  - *Orientation* – Project or Host Local Coordinate System
  - *Type* – point, line or area
  - *State* – predefined or user defined
- ***Translation parameters*** – specifies the conditions applied for translation on X/Y/Z
- ***Rotation parameters*** – specifies the conditions applied about the X/Y/Z rotation

Analytical (Rigid) Links

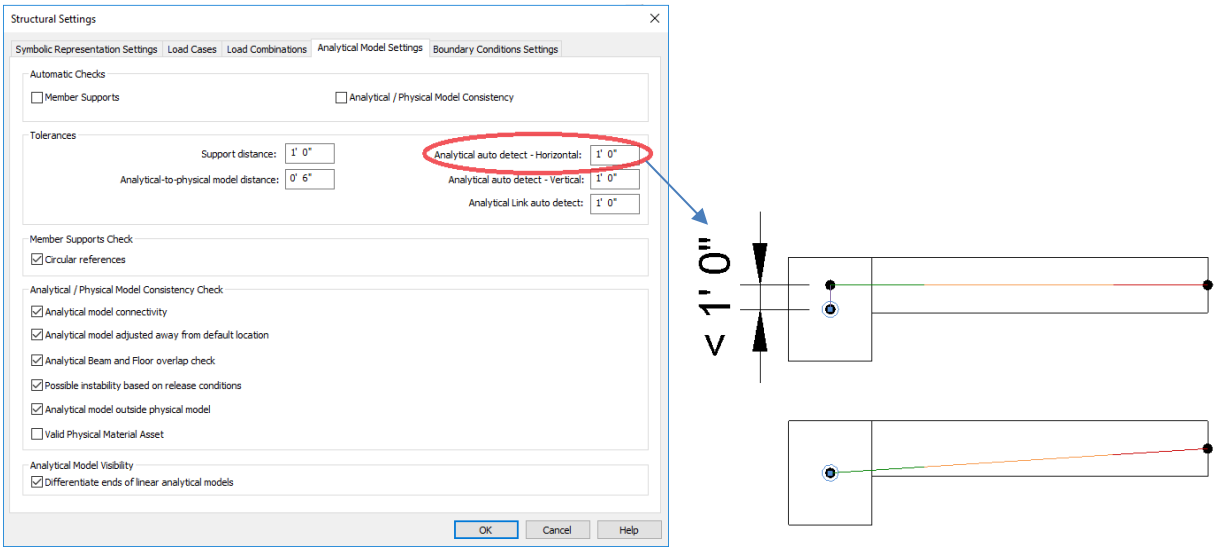
An analytical link is an element connecting two separate analytical nodes.

To **create** an Analytical Link:

1. Click Analyze tab ➤ Analytical Model Tools panel ➤ Analytical Adjust
2. Click Edit Analytical Model panel ➤ Analytical Link
3. Click 2 nodes to be connected by the analytical link.
4. Press Esc or click Modify to exit the tool.
5. Click Finish Edit Mode to save changes.

Analytical Links are also created automatically when:

1. Connecting columns and beams
    - Depends on the:
        - Analytical Column links parameter
            - Analytical Column instance parameters list, in the Properties Palette ➤ Analytical Model group of parameters ➤ Analytical Links parameter must be checked
        - Analytical Beam links parameter
            - Analytical Beam instance parameters list, in the Properties Palette ➤ Analytical Model group of parameters ➤ Analytical Links parameter must Yes or Column



(See also Revit Help documentation - https://help.autodesk.com/view/RVT/2018/ENU/?guid=GUID-4ED9521E-E61D-49D7-AB94-6BC4DCABDF20)

Enable/Disable the Analytical Representation of the Structural Elements

The analytical representation of a structural element can be enabled by:

- Checking Enable Analytical instance parameter. It is available in the Properties Palette ➤ Structural group

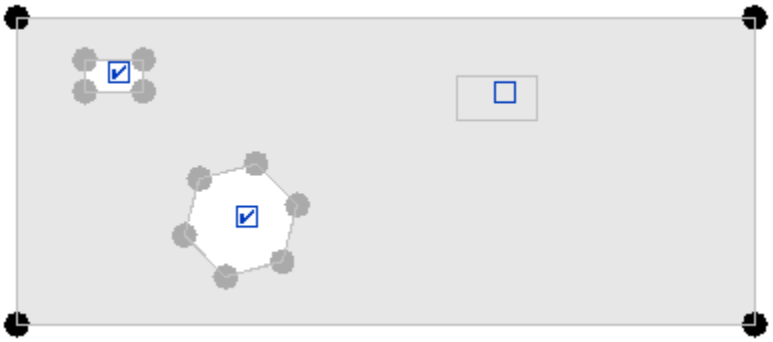
The analytical representation of a structural element can be disabled by:

- Selecting the structural element ➤ Properties Palette ➤ Structural group ➤ Uncheck Enable Analytical instance parameter
- Selecting the analytical element ➤ Modify| Analytica <element> contextual tab ➤ Analytical Model Panel ➤ Disable Analytical

Analytical Surface Opening

To consider openings in analytical model:

1. Click Analyze tab ➤ Analytical Model Tools panel ➤ Analytical Adjust.
2. Click Edit Analytical Model panel ➤ Openings to reveal check boxes for each opening.
3. Clear the check box to exclude the opening from the analytical model. The opening boundary fills with the surface color.



4. Openings with their check boxes selected are included in the analytical surface.
5. Press Esc or click Modify to exit the tool.
6. Click Finish Edit Mode to save changes.

Analytical Model Adjustments

Auto-Detect Adjustment

Automatic adjustment is performed on a structural element, in relation to a neighboring structural element.

Revit can automatically adjust the analytical model for beams, braces, structural columns, structural walls, structural floors, and foundation slabs so that they align more accurately. This behavior is based on the instance parameters of the elements and tolerance settings.

For auto-detection to take place, the analytical Adjustment Methods instance properties must be set to Auto-Detect for an element and its individual ends. This is the default justification method for all analytical structural elements.

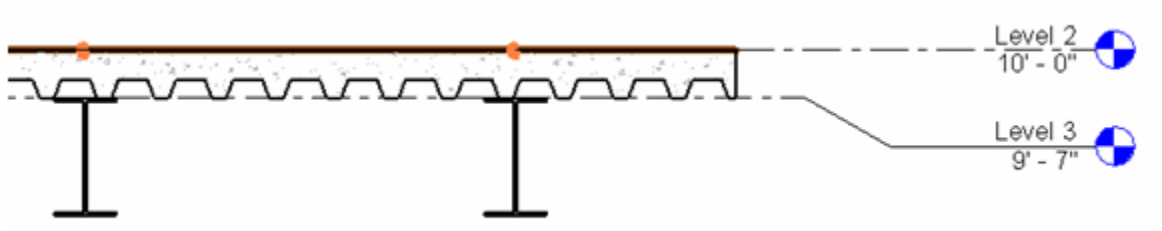
Analytical Alignment	
Top Alignment Method	Auto-Detect
Top y Projection	Location Line

ANALYTICAL COLUMN INSTANCE PARAMETER FOR TOP ANALYTICAL ASSIGNMENT

Auto-detect tolerances are specified on the Analytical Model Settings tab of the Structural Settings dialog. *(see Analytical Model Settings > Tolerances chapter)*

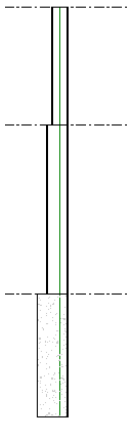
The auto-detect feature automatically adjusts the analytical model when creating the following structural elements within a project:

- **Beams**



Analytical Auto-Detect – Vertical setting from the Structural Settings dialog ➤ Analytical Model Settings tab ➤ Tolerances define the distance between levels for which auto-detect adjust the analytical beam position.
The distance between elements is not considered.

- **Walls**



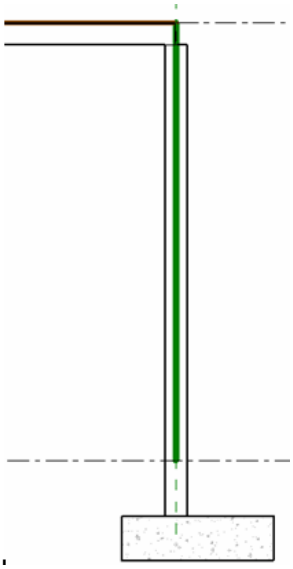
VERTICAL WALLS ALIGNMENT

The auto-detect feature aligns the vertical and horizontal analytical projection plane of walls, despite any variation in wall thicknesses or projection plane location.

Tips:

If walls are overlapping on the same level, the auto-detect tool will not adjust the analytical wall position.

- **Walls with Floors**



If a wall and floor are joined, the top or bottom plane of the wall's analytical model will coincide with the floor analytical model.

- **Floor-to-Floor (Wall-to-Wall)**
  For elements of the same structural category (such as floor-to-floor or wall-to-wall) auto-detect is based on the order of creation, with the highest priority given to the element created first.

Projection Adjustment

Projection references for linear elements are defined as horizontal and vertical in relation to the local beam coordinate system.

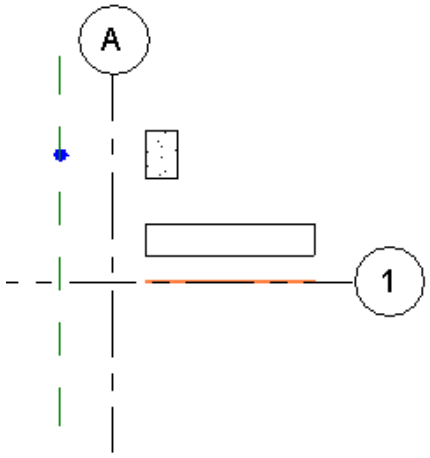
Horizontal plane (y-direction) projection references include grids, sides and center of a beam.
Vertical plane (z-direction) projection references include levels, top of beam, middle of beam and bottom of beam.

Named reference planes are included in the horizontal and vertical projections where appropriate.

All sloped planes are included in each projection list. If both projection planes refer to a sloped reference plane the projection point is perpendicular to the sloped plane passing through the location line.

- **Horizontal Projection**

Linear analytical models can project horizontally to a specific reference plane or grid.



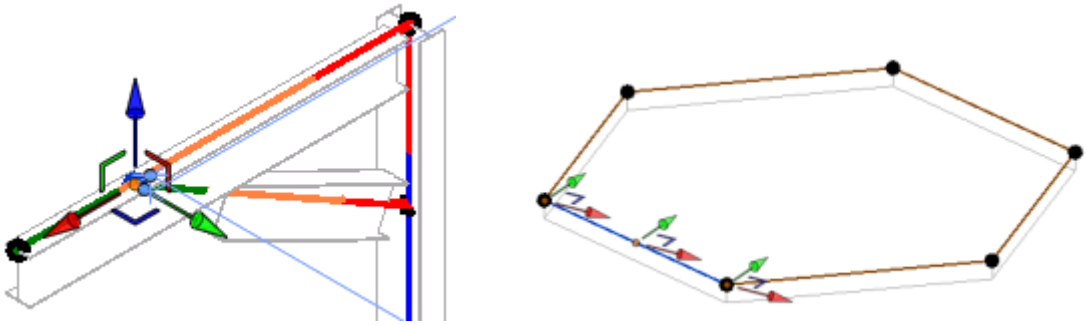
ANALYTICAL COLUMN ALIGNED TO A REFERENCE PLANE
ANALYTICAL WALL ALIGNED TO A GRID

- **Vertical Projection**

Top and bottom vertical projection planes for walls and columns can be adjusted to the analytical projection of a structural floor.

Manual Adjustment

Some structural configurations are not suitable for direct integration with analysis and design software. Adaptive adjustment is required before a structural model is input into the analysis and design software.



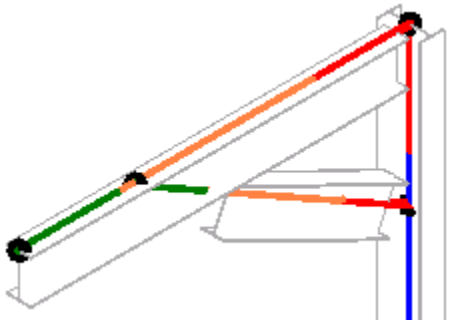
Use the Analytical Adjust tool to prepare the analytical model for various analysis applications.

1. Click Analyze tab ➤ Analytical Model Tools panel ➤ Analytical Adjust to manually adjust the analytical model.
2. In the drawing area, move and adjust analytical nodes and edges by snapping them onto analytical model geometry, nodes, grids and reference planes.
3. Click Analyze tab ➤ Edit Analytical Model panel ➤ (Finish) to exit the analytical edit mode and save changes to the analytical model or (Cancel) to exit without saving.

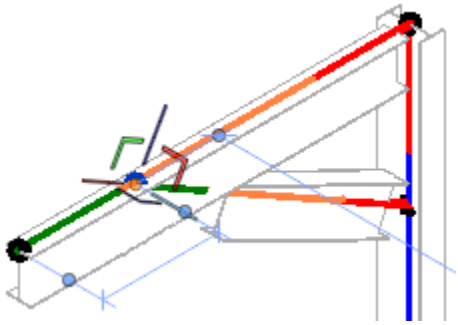
If the analytical model is modified, its Analytical Alignment parameters will specify Manually Adjusted.

- **Linear Elements**

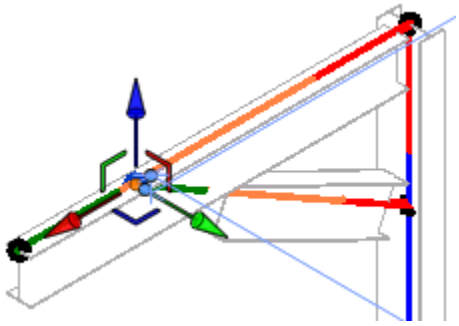
Linear analytical model elements can be manipulated by the analytical nodes at both ends.



Click a node to reveal a 3D control to move the model end in the local coordinate system.



Press Spacebar to flip the control to the global coordinate system.

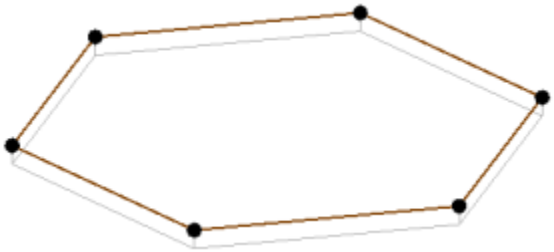


Move and adjust analytical nodes by snapping them onto analytical model geometry.

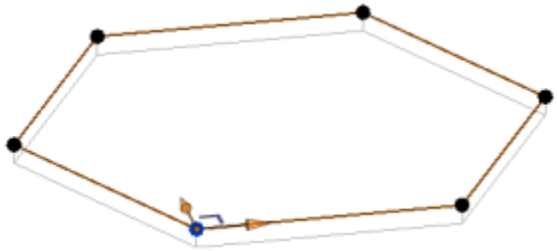
If a node is placed where it cannot be hosted, an offset will be maintained from the element's location line.

- **Surface Elements**

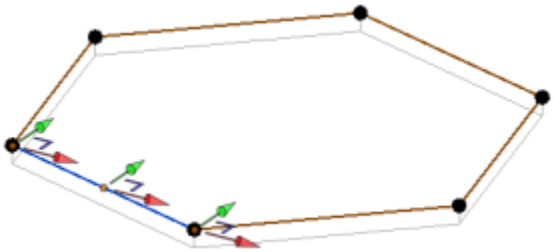
Surface analytical model elements can be directly manipulated by their nodes and edges in the analytical model plane.



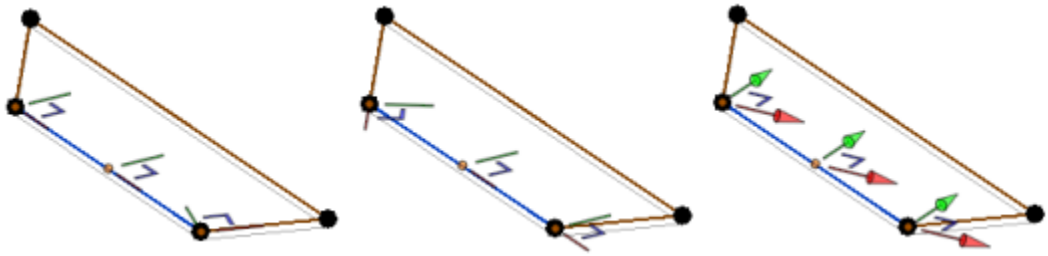
Click a node to reveal its 2D controls. Drag the node as needed.



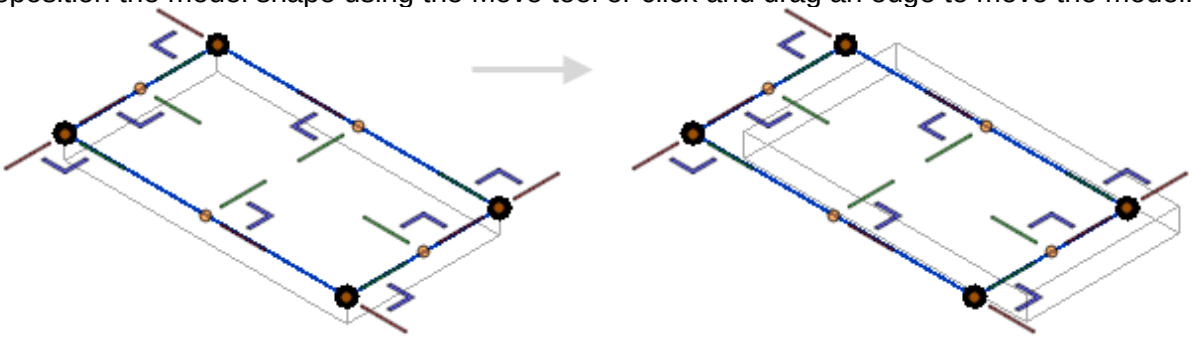
Tab select an edge to reveal its 2D controls. Drag the edge as needed.



Press Spacebar to flip the 2D controls from the local to the global coordinate system. The following 3 coordinate systems are available: local with respect to the edge adjoining the node, local with respect to the other edge adjoining the node and global.



Reposition the model shape using the Move tool or click and drag an edge to move the model.



Analytical Wall Vertical Edges Adjustment

Analytical Wall Adjustment tool is used to adjust the analytical wall to:

- Another analytical wall
- Analytical column
- Analytical beam
- Analytical node of an analytical floor or an analytical foundation slab.

1. Click Analyze tab ➤ Analytical Model Tools panel ➤ Analytical Adjust.
2. Click Edit Analytical Model panel ➤ Wall Adjustment.

Select an end edge of the analytical wall to adjust as the source analytical element. You can select only wall vertical end edges as the source analytical element.
Select the target analytical element. Elements that you can select as target analytical elements highlight when you place the cursor over them.

3. Click Select panel ➤ Modify to accept changes and finish the Wall Adjustment tool. You are still in the Analytical Edit mode. You can continue adjusting analytical elements.
4. Click Finish Edit Mode to save changes and exit the Analytical Edit mode. To exit the Analytical Edit mode without saving changes, click Cancel.

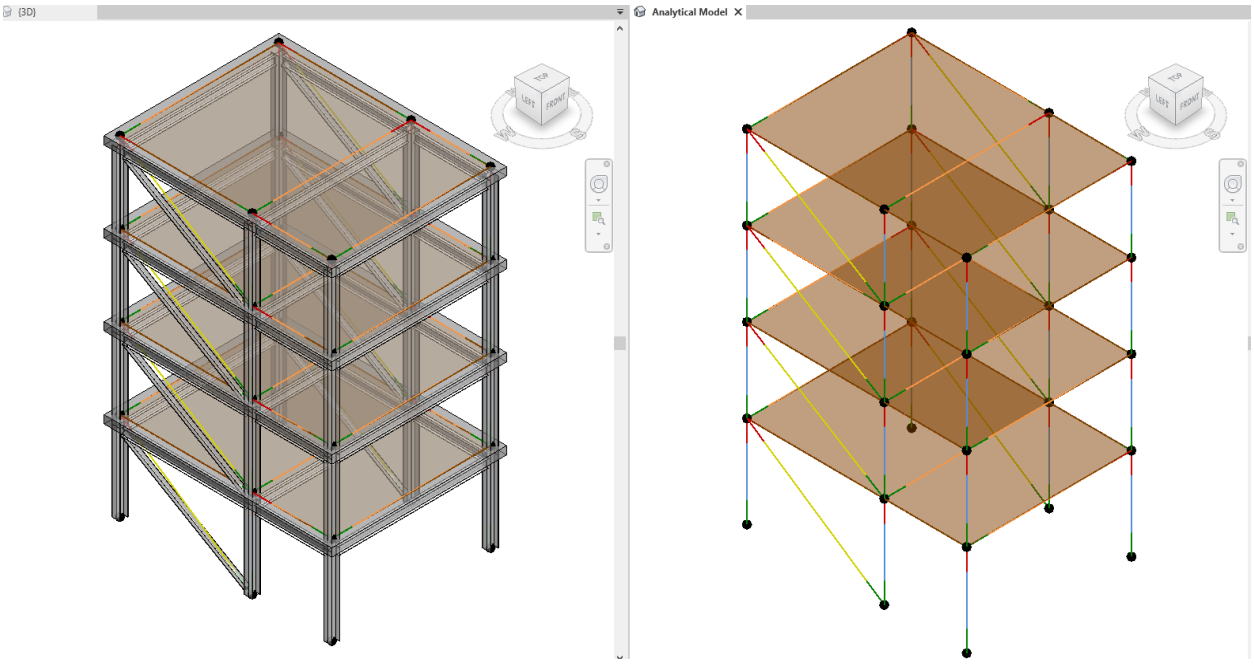
(see also Revit Help documentation: http://help.autodesk.com/view/RVT/2020/ENU/?guid=GUID-0FB45CBB-2808-45B5-9D2C-2626BB19E31B)

Reset the Analytical Representation

Restore the analytical model of an element to its default position.

1. Click Analyze tab ➤ Analytical Model Tools panel ➤ Analytical Reset.
2. Select the element to reset the selected structural element analytical model back to its original shape or location, relative to its corresponding physical model.
3. Optionally, If you are currently editing the analytical model, select the element and click Modify | <Element> tab ➤ Analytical Model Tools panel ➤ Analytical Reset.

Exercise 1 | Create the Analytical Representation for the Structural Elements of a Steel Structure in Revit



In this exercise, we'll create a structural model while dealing also with the analytical representation of it. We'll create a simple structure with steel frames and concrete decks. In the end the model will be ready to be consumed by structural analysis solvers.

Set-Up Revit Environment

1. Open Exercise_01_StartPoint.rvt
2. Open Level 2 structural view and Analytical Model 3D view
3. Go to View tab ➤ Windows panel ➤ Tile Views
4. Go to Manage tab ➤ Settings panel ➤ Structural Settings dialog ➤ Analytica Model Settings tab ➤ Make sure that:
    - Analytical-to-Physical Model Distance is set to 400mm
    - Analytical Auto-Detect – Horizontal is set to 300mm
    - Analytical Auto-Detect – Vertical is set to 600mm

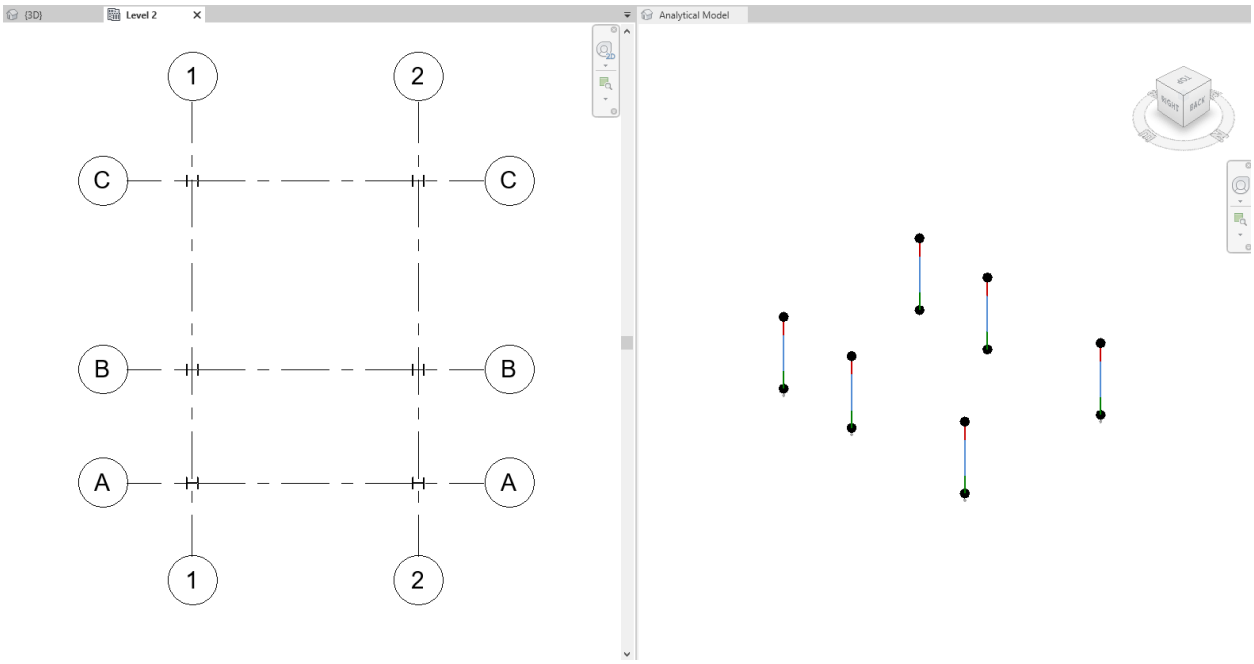
Create the Structural Elements for the First Level

1. Go to Level 2 structural view ➤ create UC305x305x97 steel columns at the axis intersection.

   - *Structure tab ➤ Structure panel ➤ Column*
   - *Modify | Place Structural Column tab ➤ Placement panel ➤ Vertical Columns*
   - *Modify | Place Structural Column tab ➤ Multiple panel ➤ At Grids*
   - *Select all the grids. A preview of the column placement is displayed.*
   - *Press Space bar once. The column position is with the web along A axis.*
   - *Modify | Place Structural Column >At Grids Intersection tab ➤ Multiple panel ➤ Finish*
   - *Align the column web with grid A.*

Tips:

Press the Space bar while columns are in creation or selection.

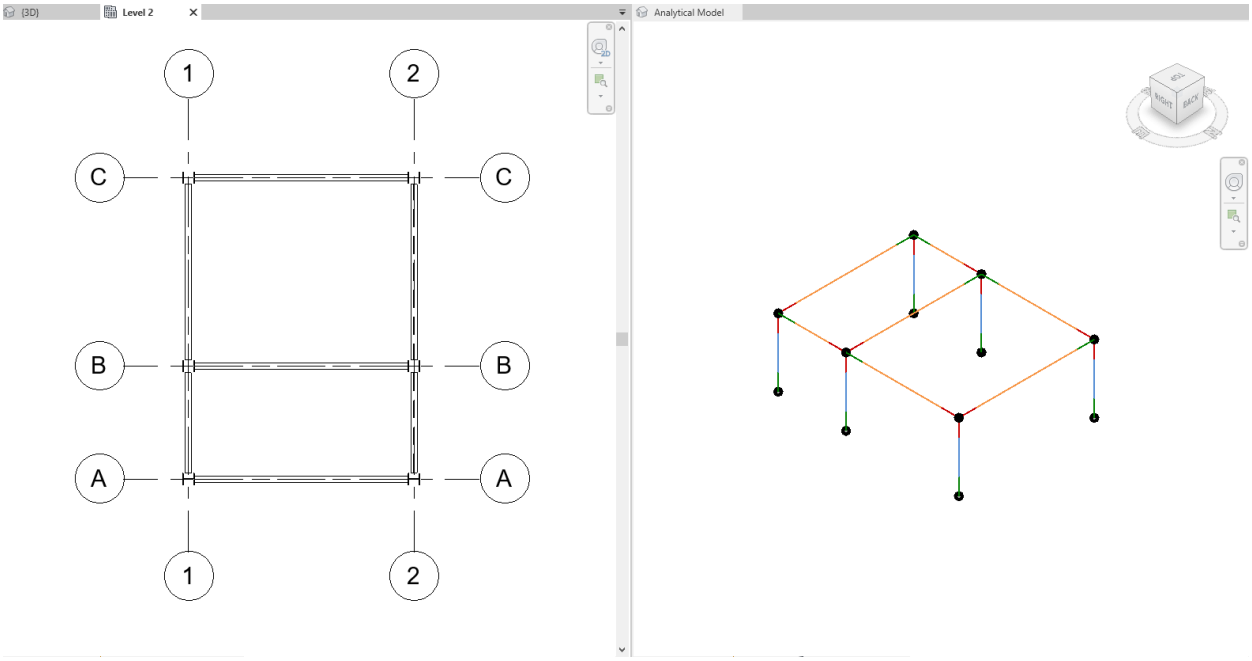


2. Create the beams between columns. Choose UB305x165x40 section type for the beams.

Tips:

Use On Grids option from the Beam contextual menu.

- *Structure tab ➤ Structure panel ➤ Beam*
- *Modify| Place Beam tab ➤ Multiple panel ➤ On Grid*
- *Select the grids and the columns*
- *Modify | Place Beam > On Grid Lines ➤ Multiple panel ➤ Finish*

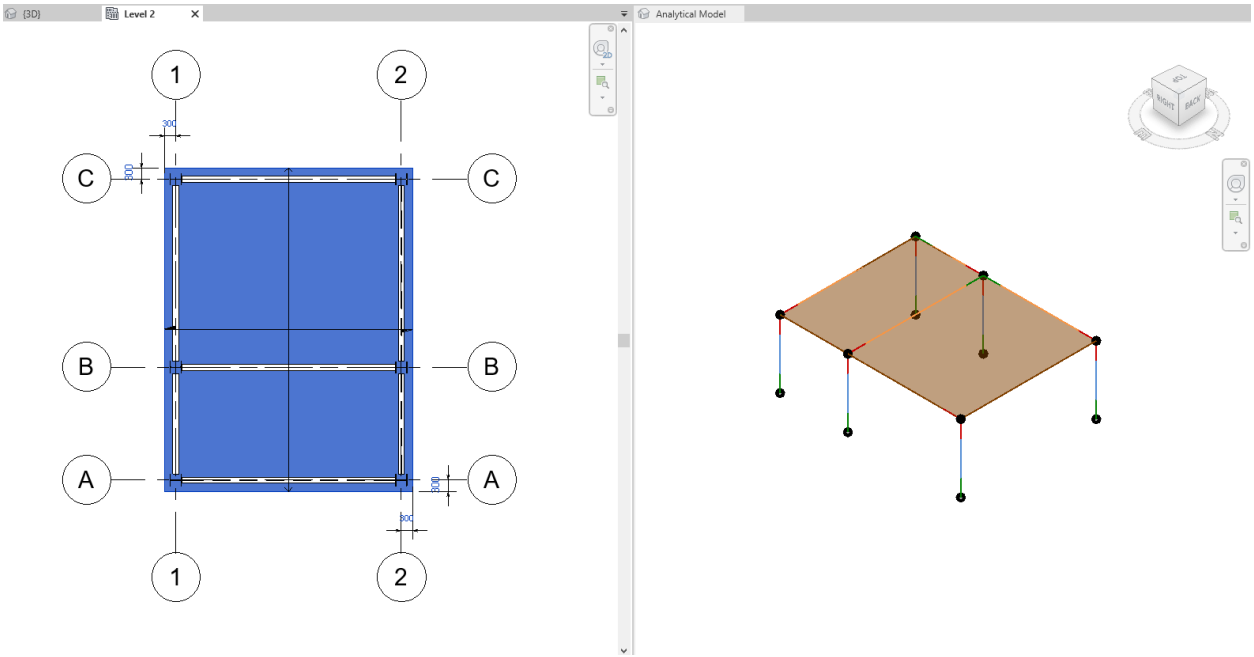


3. Create a floor at Level 2. Choose Floor Generic 300mm type. The floor offsets from the axis will be 300mm.

Tips:

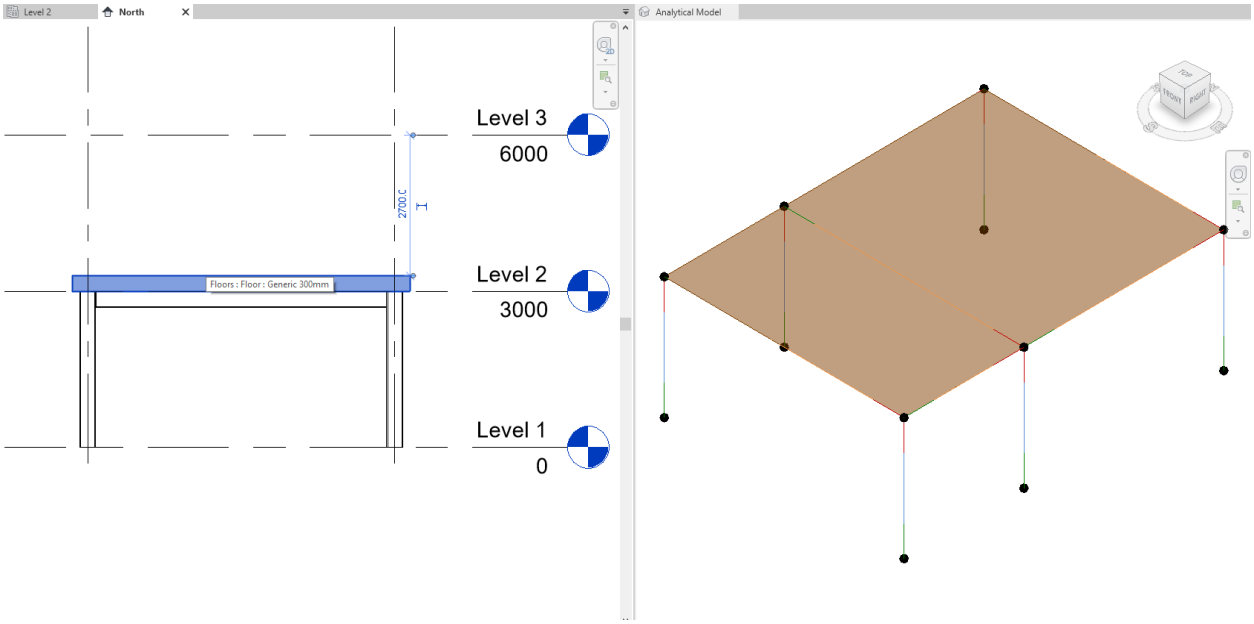
Use as few edges as possible to define a floor. All these edges will reflect also in the analytical floor definition.

- *Structure tab ➤ Structure panel ➤ Floor*
- *Modify | Create Floor Boundary*
- *Draw floor's sketch.*
- *Mode panel ➤ Finish*



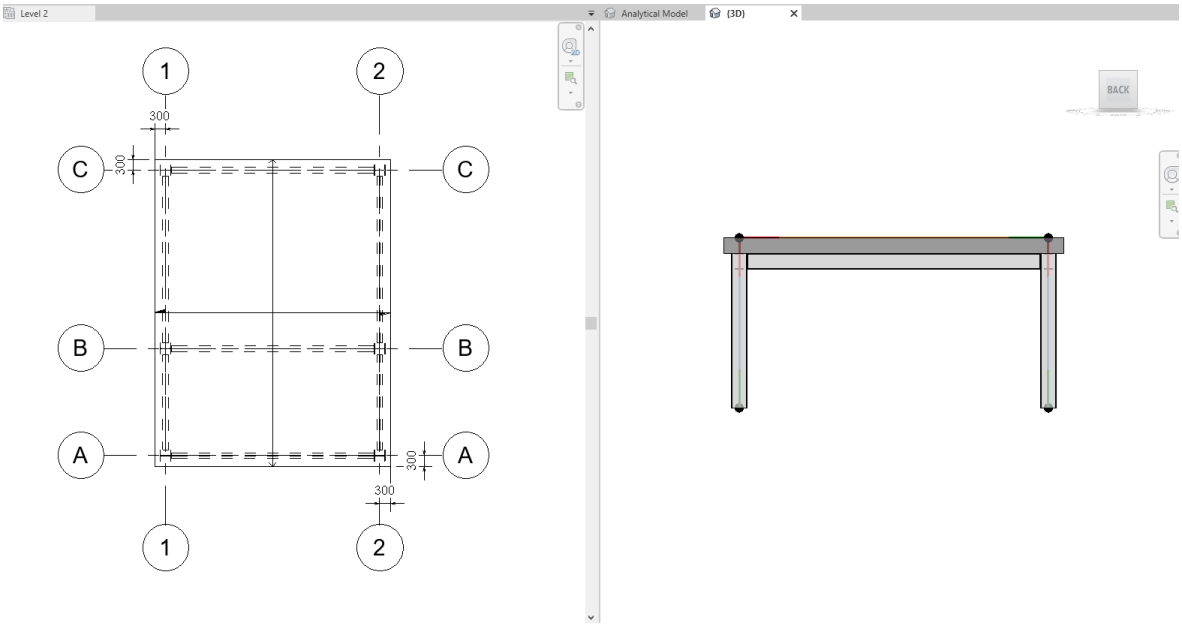
4. Set the floor's offset 300mm above Level 2.

   - *Select the floor*
   - *In the Properties Palette ➤ Constraints group of parameters ➤ Set the Floor's Height Offset From Level parameter to 300mm.*
   - *Apply*



5. Set the analytical floor alignment at the top face of the physical floor.

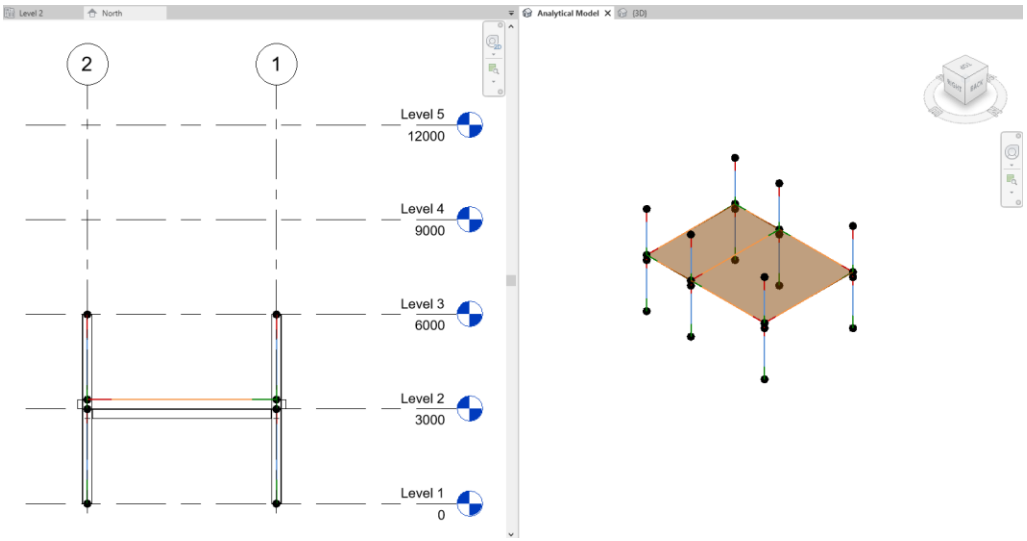
   - *Go in Analytical Model 3D view ➤ select the Analytical Floor*
   - *With the Analytical Floor selected ➤ Properties Palette ➤ Analytical Alignment group of parameters ➤ set Alignment Method to Projection ➤ set the z Projection parameter to Top of the element.*
   - Apply.



Copy the Elements to the Other Levels

1. Select Level 2 elements and copy them to Level 3.

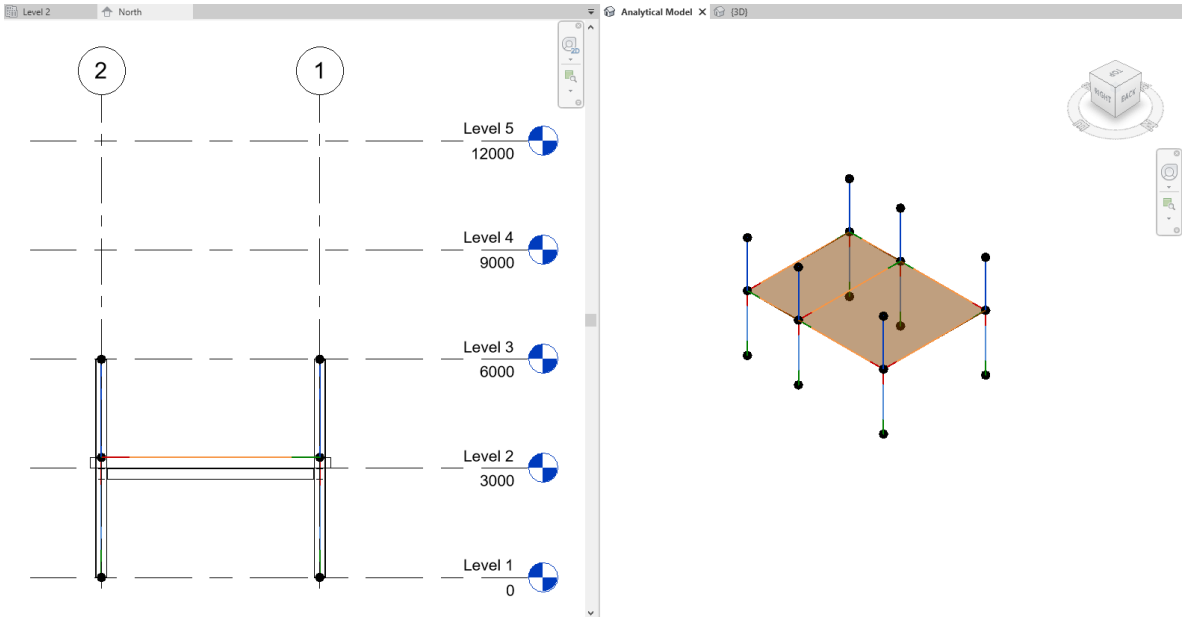
   - *Select Level 2 columns*
   - *Modify| Multi-Select tab* ➤ *Clipboard panel* ➤ *Copy to Clipboard*
   - *Paste drop-down* ➤ *Aligned to Selected Levels* ➤ *Select Level 3* ➤ *OK*



Notice that the Level 3 columns bottom nodes are not joined with the Level 2 nodes.

2. Adjust the analytical columns bottom alignment.

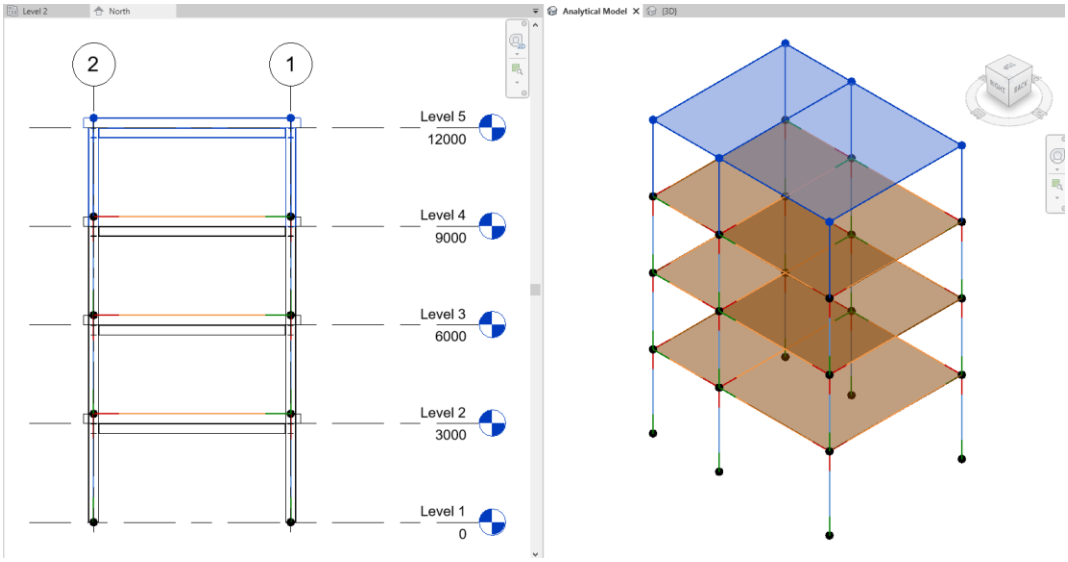
   - *Select all the Level 3 analytical columns*
   - *With the analytical columns selected ➤ Properties Palette ➤ Analytical Alignment group of parameters ➤ set both Base Alignment Method and Base Extension Method parameters to Auto-Detect*
   - *Apply.*



Notice that the bottom of all the analytical columns from Level 3 are automatically adjusted. The physical columns keep its base offset while the analytical column extends to the analytical floor.

3. Copy the elements from Level 3 to Level 4 and Level 5

   - *Select all Level 3 elements*
   - *With the elements selected ➤ to Modify| Multi-Select tab ➤ Clipboard panel ➤ Copy to Clipboard*
   - *Paste drop-down ➤ Aligned to Selected Levels ➤ Select Level 4 and Level 5 ➤ OK*



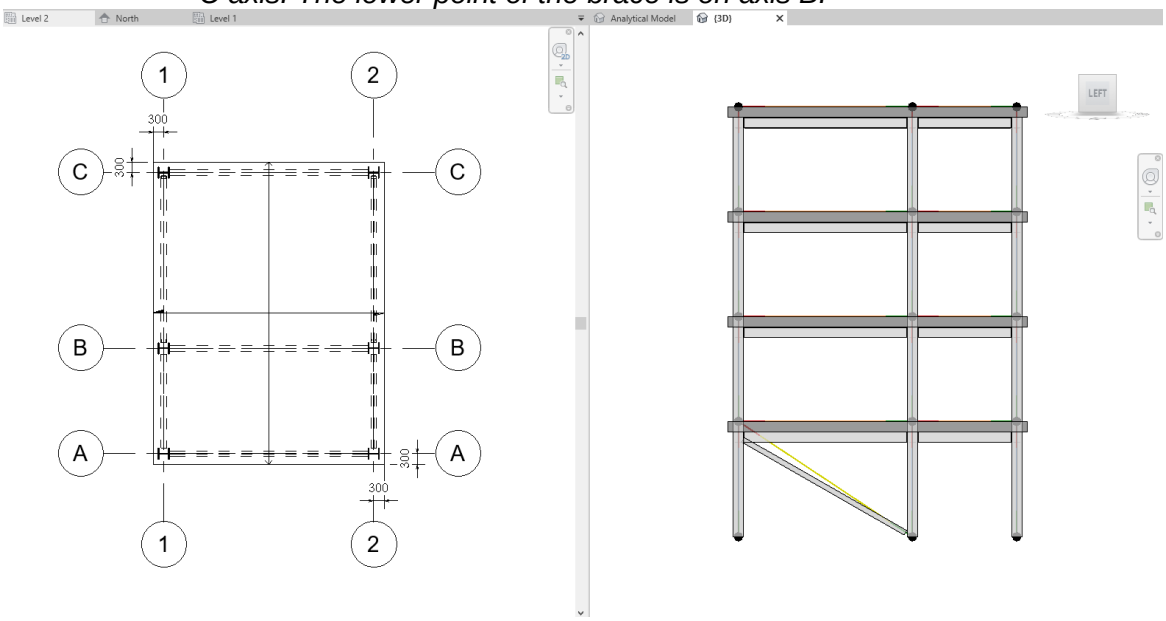
4. Select all the beams and columns and make sure all the Analytical Alignment parameters are set to Auto-Detect

Create Braces

Go to Level 2 structural view

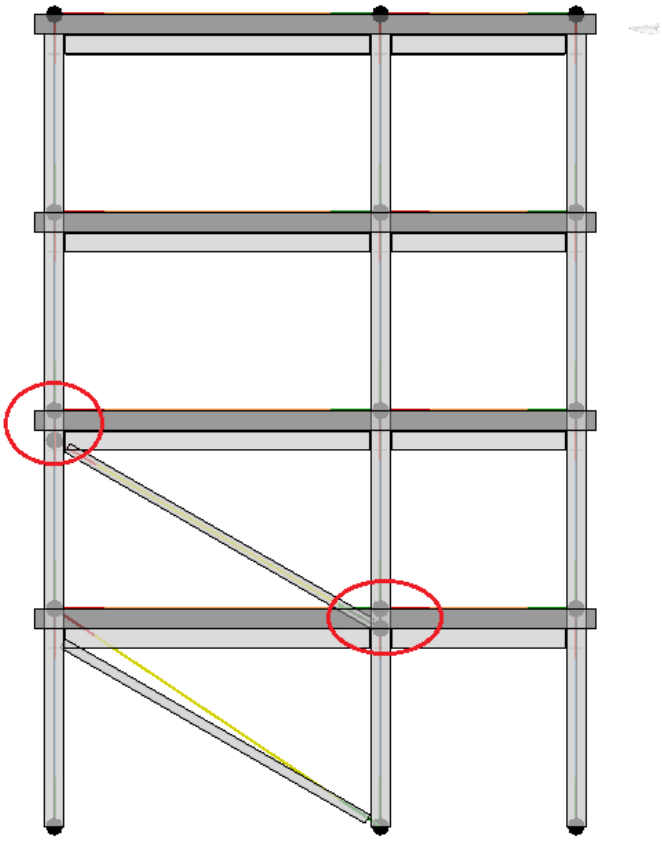
1. Create vertical brace.

   - *Structure tab ➤ Structure panel ➤ Brace*
   - *Create bracings of type HSS152.4x152.4x9.5, on axis 1 and 2, between B and C axis. The lower point of the brace is on axis B.*



2. Copy the created bracing on Level 2.

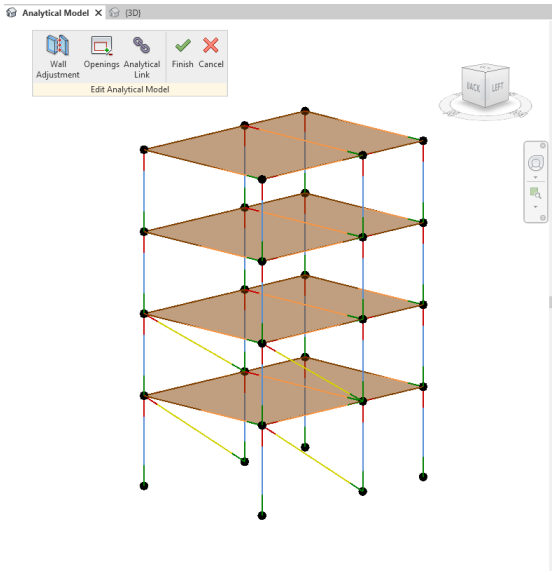
- *Select the two braces*
- *Modify| Multi-Select tab* ➤ *Clipboard panel* ➤ *Copy to Clipboard*
- *Paste drop-down* ➤ *Aligned to Selected Levels* ➤ *Select Level 2* ➤ *OK*



Notice that the nodes are not correctly positioned.

3. Adjust the Level 2 analytical brace's position.

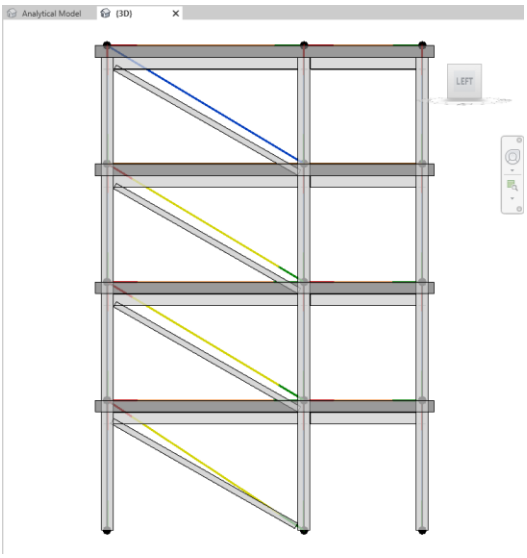
   - *Go to Analytical Model 3D view*
   - *Analyze tab ➤ Analytical Model Tools panel ➤ Adjust ➤ Analytical Adjust mode will open*
   - *Drag each bracing node in the correct position.*
   - *Finish the analytical adjustment.*



Notice that the Analytical Alignment parameters changed to <Manually Adjusted>

4. Copy the braces to Level 3 and Level 4

   - *Select the two adjusted bracings*
   - *Modify| Multi-Select tab ➤ Clipboard panel ➤ Copy to Clipboard*
   - *Paste drop-down ➤ Aligned to Selected Levels ➤ Select Level 3 and Level 4 ➤ OK*

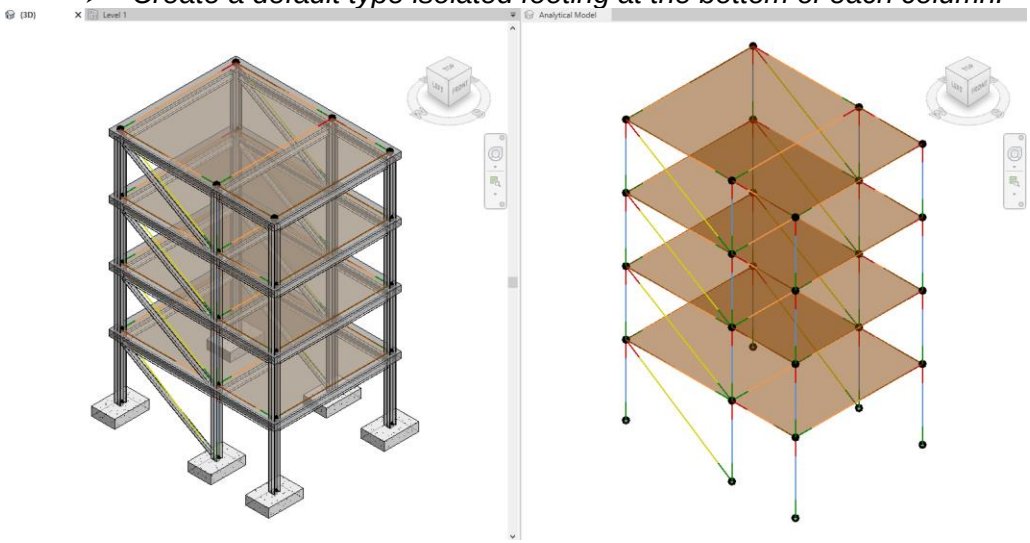


Create the Isolated Footings for the Columns

Go to Level 1 structural view

1. Create the isolated footings

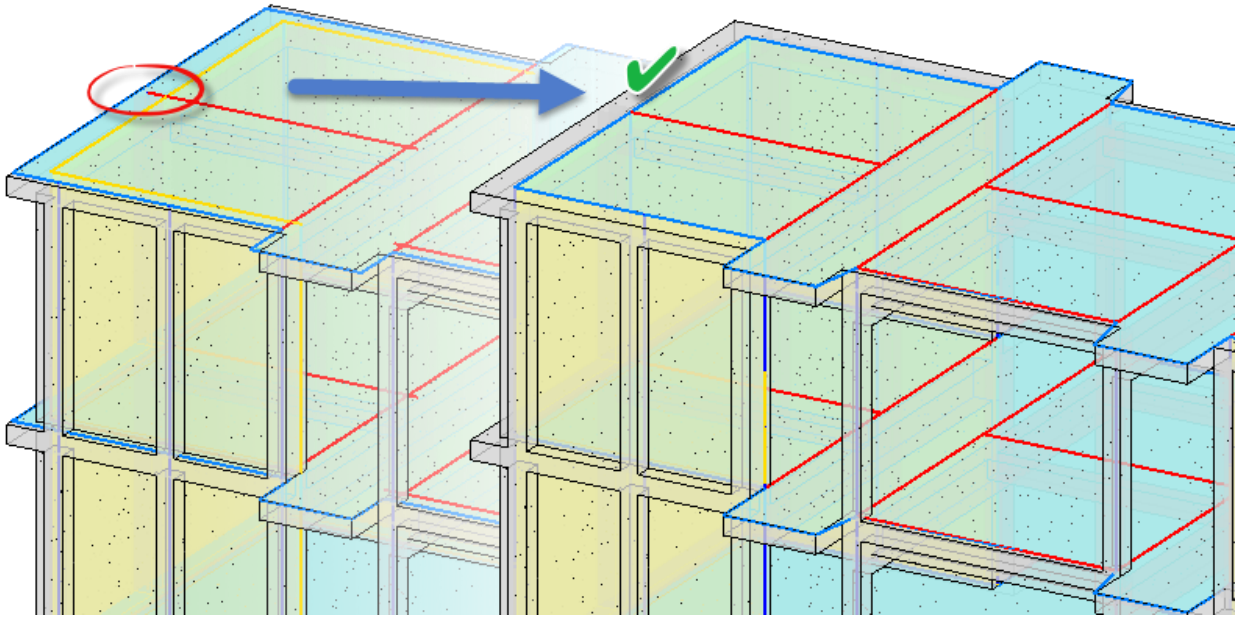
   - *Structure tab ➤ Foundation panel ➤ Isolated*
   - *Create a default type isolated footing at the bottom of each column.*



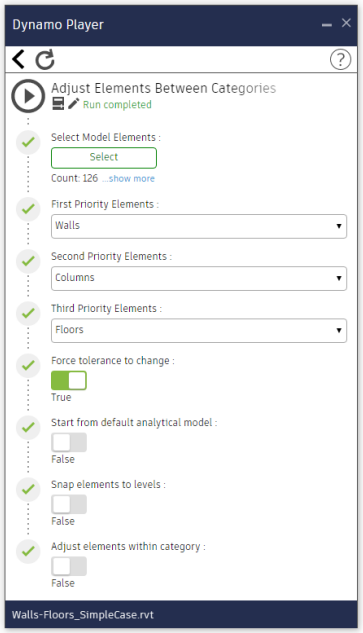
Automate the relation between physical and analytical representations using Dynamo

Autodesk Analytical Modeling 2020 Dynamo Package

The Autodesk Analytical Modeling 2020 Dynamo package helps structural engineers better control and automate the creation and adjustment of analytical models in Revit. Create a complete and consistent analytical model with parametrization, definition of logical assumptions, priorities, and customizable scripting. Customizable logic helps you create different rules-based analytical models for diverse types of buildings and multiple analytical model variants for similar building structures, and allows you to use similar patterns across projects.



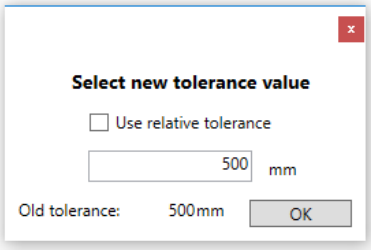
Dynamo scripts created with the package take a Revit design model and attempts to generate a consistent and connected analytical representation that corresponds to the geometric shapes of the model. The generated analytical model is based on user configurations such as relative prioritization of elements categories (for example, column location is more important than beam location) and other geometrical characteristics. The scripts can be executed from Dynamo Player interface as well as from the Dynamo for Revit interface.



There are three stack ranked priorities to be specified. Choose the first, second, and third priority elements following the general premise that the first priority elements act as master elements and snap second and third priority elements to themselves. Second priority elements will also attract nodes of elements of the third priority category.

The disconnected analytical elements will connect with each other if their vertices are within the tolerance range in units determined by the project unit settings.

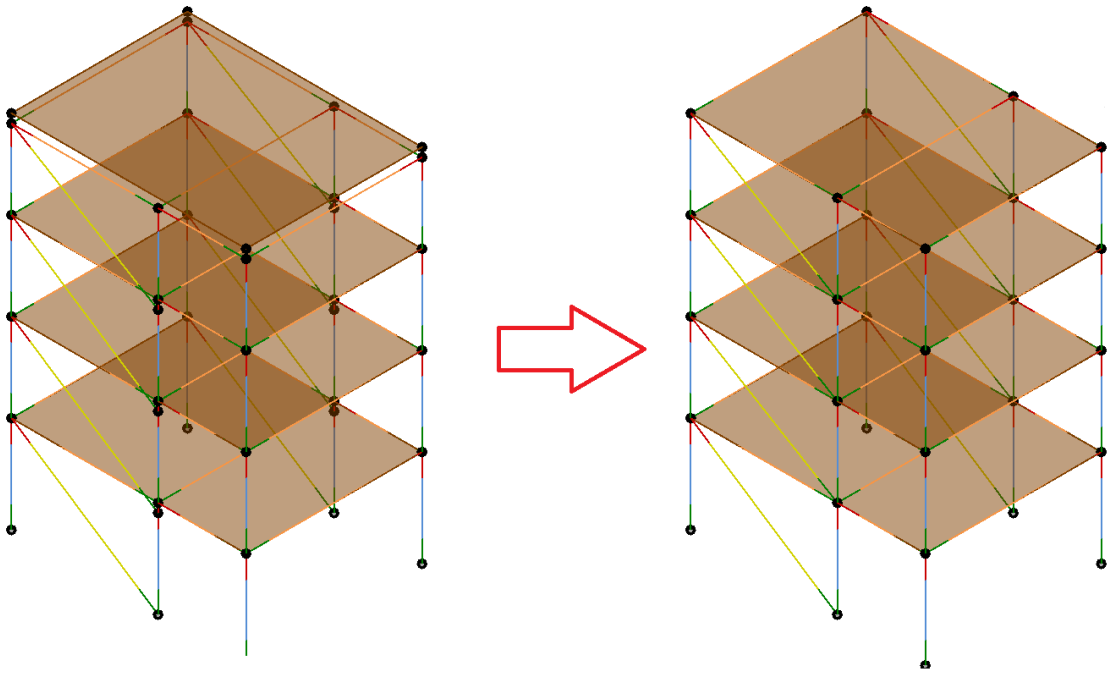
The tolerance can be absolute or relative; absolute tolerance is measured in model units and equal for all of elements, whereas when it is relative it is stated as percentage of each element biggest cross-section dimension (height or width for linear elements and thickness for planar elements) . The tolerance relative value can be customized:



The Snapping elements to level option determines if all selected analytical elements should be snapped to closest levels defined in Revit project.

When the Adjust elements within a category option is turned on, nodes of elements within the same category will be connected to each other, for example, beam nodes will be join other beam nodes.

Exercise 2 | Adjust the Analytical Representation of a Structural Model using Dynamo



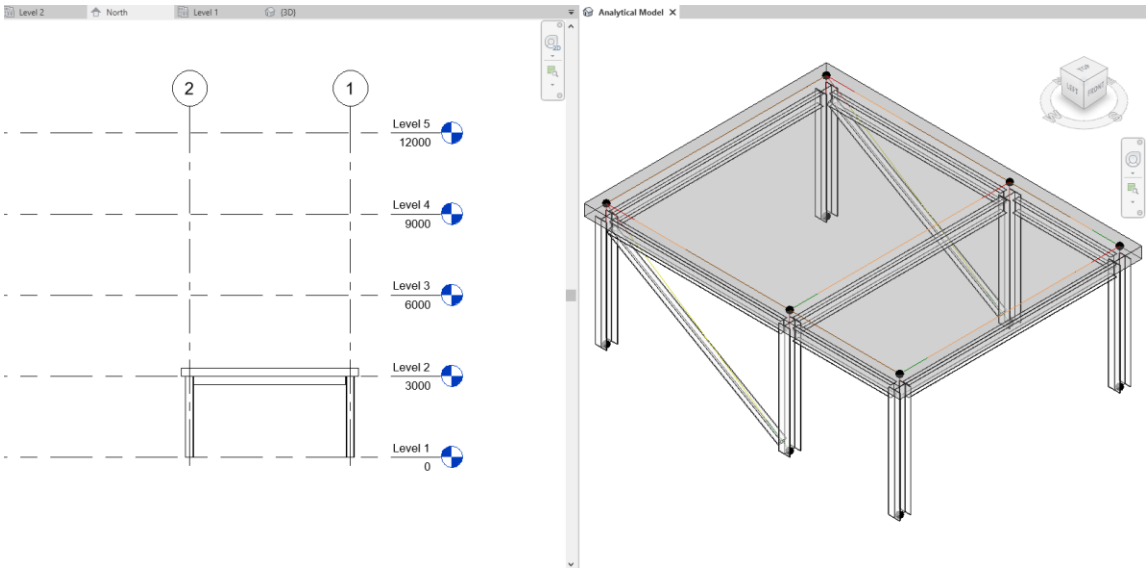
In this exercise, we'll adjust the analytical representation of the same structure using a Dynamo tool. Autodesk Analytical Modeling Dynamo package attempts to generate a consistent and connected analytical representation that corresponds to the geometric shapes of the model.

Set-up Environment

1. Add Autodesk Analytical Modeling 2020 Dynamo package to your collection.

   - *In Revit, Manage tab ➤ Visual Programming panel ➤ Dynamo*
   - *Dynamo will open.*
   - *In Dynamo, Packages tab ➤ Search for Packages…*
   - *Online Package Search will open.*
   - *Search for Autodesk Analytical Modeling 2020 Dynamo.*
   - *Install the latest version.*

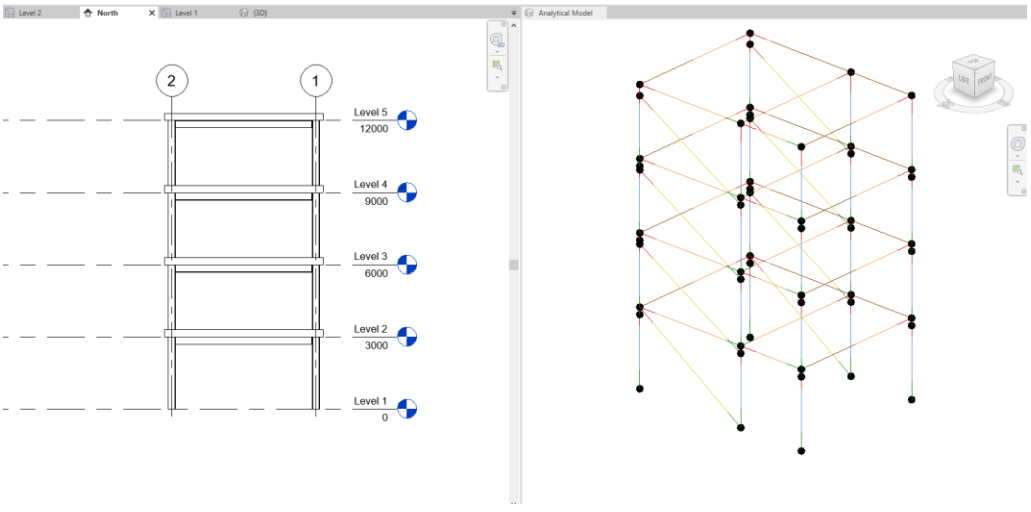
2. Open Exercise_02_StartPoint.rvt



3. Copy all elements from Level 2 to Level 3, Level 4 and Level 5

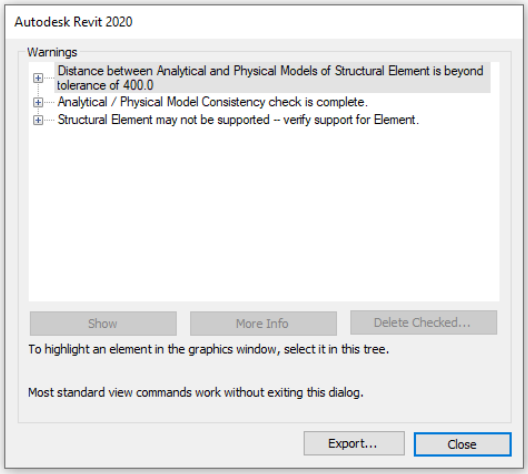
- *Select all Level 2 elements, except the braces.*
- *With the elements selected ➤ to Modify| Multi-Select tab ➤ Clipboard panel ➤ Copy to Clipboard*
- *Paste drop-down ➤ Aligned to Selected Levels ➤ Select Level 3, Level 4 and Level 5 ➤ OK*

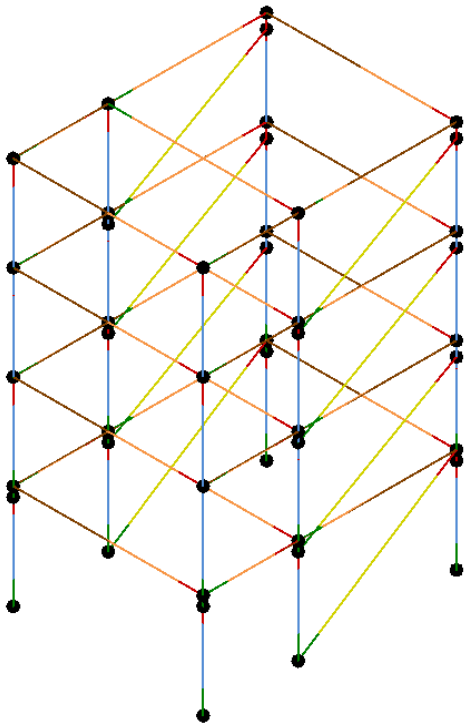
- *Select the braces from Level 1.*
- *With the elements selected ➤ to Modify| Structural/Analytical Braces tab ➤ Clipboard panel ➤ Copy to Clipboard*
- *Paste drop-down ➤ Aligned to Selected Levels ➤ Select Level 2, Level 3 and Level 4 ➤ OK*



Verify Analytical Model Consistency

1. Go to Manage tab ➤ Settings panel ➤ open Structural Settings Dialog ➤ go to Analytical Model Settings tab ➤ check the Analytical/Physical Model Consistency and Member Supports
2. OK



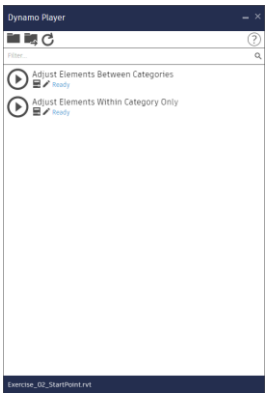


3. Explore the warning messages.

Adjust the Analytical Representation Using Dynamo

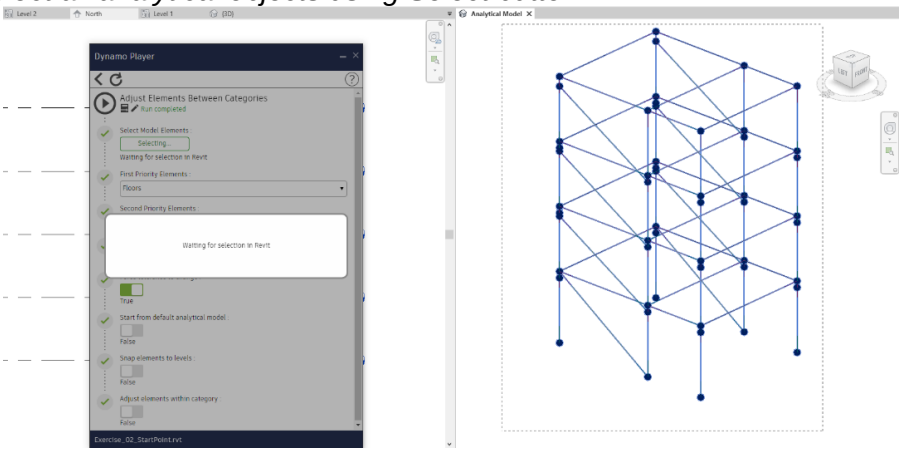
1. Load analytical modeling scripts in Dynamo Player

   - *In Revit, Manage tab ➤ Visual Programming panel ➤ Dynamo Player*
   - *In Dynamo Player, Browse to folder*
   - *The installed package and its scripts can be found in: C:\Users\<username>\AppData\Roaming\Dynamo\Dynamo Revit\2.2\packages\Autodesk Analytical Modeling 2020 Dynamo*



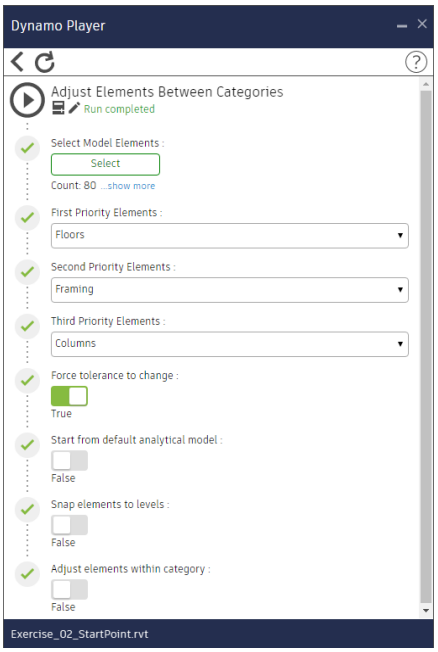
2. Adjust the elements between categories

   - *On the Adjust the Elements Between Categories script, select Edit Inputs. Select all analytical objects using Select button*

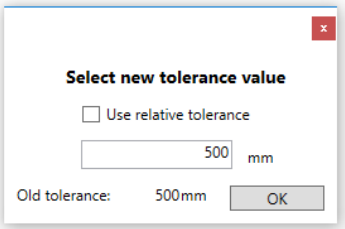


   - *Set the elements priorities:*
     - *First Priority Elements: Floors*
     - *Second Priority Elements: Framings*
     - *Third Priority Elements: Columns*

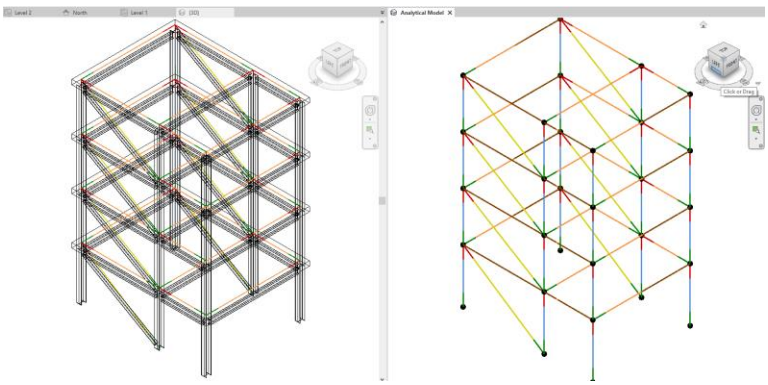
- *Set Force Tolerance to Change to True*



- *Run*
- *When Dynamo prompt ask for the new tolerance value, un-check Use Relative Tolerance and Set the tolerance to 600mm*



- *OK*



Fix All Warnings

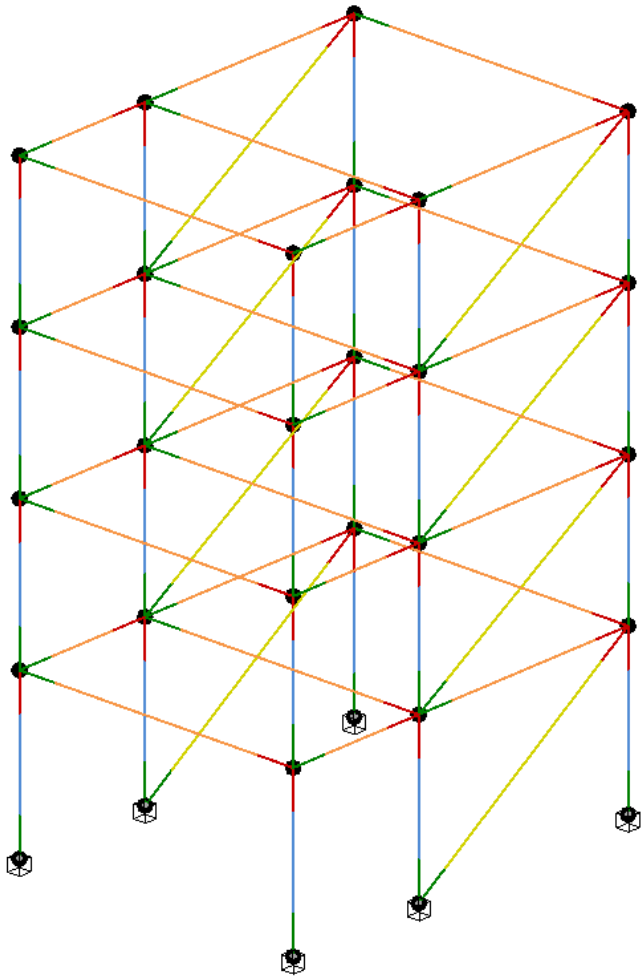
Go to Level 1 structural view

1. Create the isolated footings

   - *Structure tab ➤ Foundation panel ➤ Isolated*
   - *Create a default type isolated footing at the bottom of each column.*

2. Attach boundary conditions for columns support.

   - *Analyze tab ➤ Analytical Model panel ➤ Boundary Conditions*
   - *Modify | Place Boundary Conditions tab ➤ Boundary Conditions panel ➤ Point ➤ Assign Point boundary condition to each analytical isolated foundation.*



Complete the Analytical Model in Revit for Structural Analysis Purpose

Loads, Load Cases and Load Combinations

Loads

You can apply point, line, and area loads to your analytical model.

1. Click Analyze tab ➤ Analytical Model panel (Loads) to access the load tools.
2. Click Modify | Place Loads tab ➤ Loads panel

- Un-hosted loads
  - Point Load
  - Line Load
  - Area Load
- Hosted Loads
  - Hosted Point Load
  - Hosted Line Load
  - Hosted Area Load

Each of these six load geometries is a family that contain instance and type parameters.
You can edit load force and moment parameters before or after placing loads.
You can also apply load combinations to your model.

➢ **Point Load**

Place an un-hosted point load along structural elements in an analytical view.

1. Open a structural plan level - analytical view with a framing plan model similar to the one shown.

Note**:** Place loads in the analytical view of a structural plan level.

2. Click Analyze tab ➤ Analytical Model panel ➤ (Loads).
3. Click Modify | Place Loads tab ➤ Loads panel ➤ (Point Load).
4. On the Properties palette, select a value for Load Case.
5. For the Orient to parameter, select Project or Work Plane.
6. Place point loads as appropriate in your model, such as at the center of a beam.

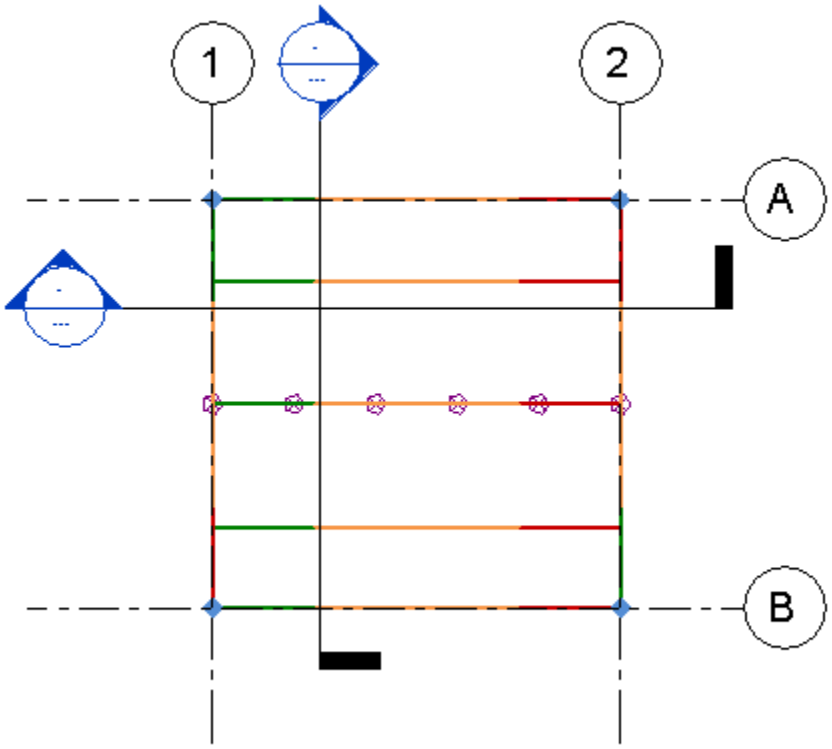
Place a hosted point load at end points of beams, braces, and columns

1. Click Analyze tab ➤ Analytical Model panel ➤ (Loads).
2. Click Modify | Place Loads tab ➤ Loads panel ➤ (Hosted Point Load).
3. On the Properties palette, select a value for Load Case.
4. For the Orient to parameter, select Project or Host Local Coordinate System.
5. Place hosted point loads on appropriate end points in your model.

- **Line Load**

Sketch an un-hosted line load along structural elements in an analytical view.

1. Click Analyze tab ➤ Analytical Model panel ➤ (Loads).
2. Click Modify | Place Loads tab ➤ Loads panel ➤ (Line Load).
3. On the Properties palette, select a value for Load Case.
4. For the Orient to parameter, select Project or Work Plane.
5. Sketch line loads as appropriate in your model, such as along a joist.



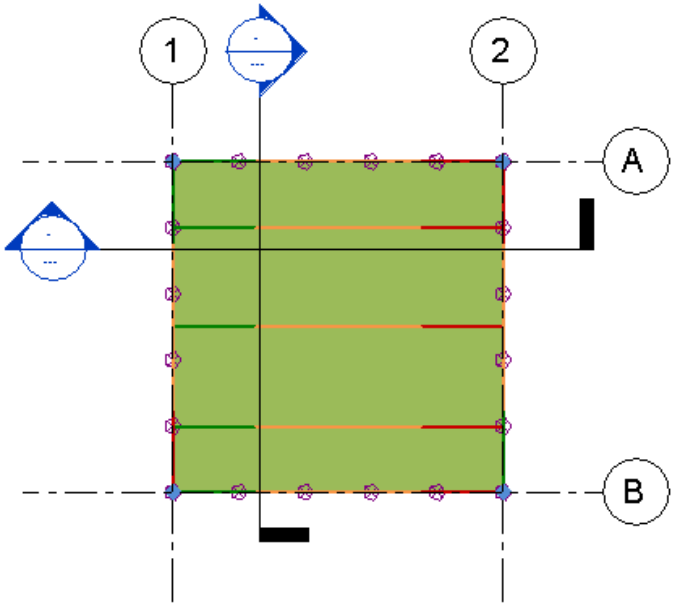
Place a hosted line load along structural wall or floor edges, and structural framing elements

1. Click Analyze tab ➤ Analytical Model panel ➤ (Loads).
2. Click Modify | Place Loads tab ➤ Loads panel ➤ (Hosted Line Load).
3. On the Properties palette, select a value for Load Case.
4. For the Orient to parameter, select Project or Host Local Coordinate System.
5. Select a component along which you wish to place the load instance.

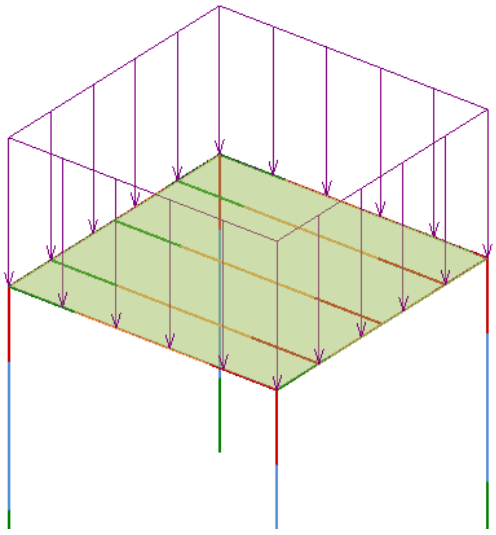
➢ **Area Load**

Sketch an un-hosted area load onto structural floors or structural walls.

1. Click Analyze tab ➤ Analytical Model panel ➤ (Loads).
2. Click Modify | Place Loads tab ➤ Loads panel ➤ (Area Load).
3. On the Properties palette, select a value for Load Case.
4. For the Orient to parameter, select Project or Work Plane.
5. Sketch area loads as appropriate in your model. For example, select the rectangle sketch tool and click the corners of a floor.



Note**:** Load symbols appear in 3D analytical views or in elevation as arrow lines, like the area load representation shown.



Place a hosted area load

Only the analytical model is available to be selected for placement of hosted loads. The physical geometry of components will not accept the load.

1. When you finish sketching, click Create Area Load Boundary tab ➤ Area Load panel ➤ (Finish Area Load).
2. Click Analyze tab ➤ Analytical Model panel ➤ (Loads).
3. Click Modify | Place Loads tab ➤ Loads panel ➤ (Hosted Area Load).
4. On the Properties palette, select a value for Load Case.
5. For the Orient to parameter, select Project or Host Local Coordinate System.
6. Select a component over which you wish to place the load instance, such as a structural floor. The area load applies over the extent of the element.

Load Cases

1. Click Analyze tab ➤ Analytical Model panel ➤ (Load Cases).
2. Click the Add button. New Case 1 is added as a table record and the Add button changes to Duplicate.
3. Click in the Name cell of this new load case, and enter a name, such as Mechanical Unit.

Note**:** The Case Number column of the table is read-only. Revit provides a unique number.

4. Click the Category cell of the new load case and select a category.

Note**:** You may also create a new load case by selecting an existing load case in the table, clicking Duplicate, and editing the new load case as needed.

The second table in the Structural Settings dialog is the Load Natures table. Use this table to add or delete load natures.

Load Nature

1. Click Analyze tab ➤ Analytical Model panel ➤ (Load Cases).
2. Click in the Load Natures table.
3. Click the Add button. A new load nature record is added to the table.

Note**:** When adding a Dead Load to the model, you must include an estimated load for the self-weight of the structure.

4. Click in the cell of the new load nature.
5. Change the name of load nature as appropriate.

Load Combinations

You can edit and add load combinations in the Structural Settings dialog.

1. Click Analyze tab ➤ Analytical Model panel ➤ Load Combinations.
2. Click in the Load Combination table and click Add.
3. Click in the Name field and enter a name.
4. Click in the Edit Selected Formula section and click Add in the same section.
5. Click in either the Case or Combination field to select a Case or Combination.
6. Click in the Factor field to enter a factor.

Note**:** Notice that the Name and Formula fields change in the Load Combination table.

7. Click Add again in the Edit Selected Formula section.
8. Click in the Case or Combination field to select a Case or Combination value.
9. Click in the Factor field to enter a factor.

Note**:** Notice that the Name and Formula fields change in the Load Combination section.

10. In the Type field of the Load Combination table, select either Combination or Envelope.

Note**:** Setting the load combination type to Combination, gives results (reactions and member forces) for a single load combination. Envelope gives maximum and minimum results on a group of load combinations.

11. In the State field of the Load Combination table, select either Serviceability or Ultimate.

Note**:** Setting the load combination state to Serviceability, reflects how a structure performs (deflection, vibration etc.) under normal or expected loading, whereas Ultimate states are based on the total capacity of a structure to safely resist extreme or 'factored' loads without failing (buckling, fracturing etc.).

12. Click in the Load Combination Usage field and click Add.

Note**:** The load combination usage parameter is user defined (either gravity, lateral or combined).

- Gravity Load Combinations include the vertical loads, both permanent or dead (self-weight of the structure - floor, beams, columns etc.) and live loads based on occupancy (people on an office floor, boxes in a storage room, snow on a roof, etc.).
- Lateral Load Combinations include the horizontal loads, both permanent or dead (soil resting up against a foundation wall) and live loads (wind against the face of a structure or the shaking of a structure from an earthquake).
- Combined Load Combinations include varying degrees of both gravity and lateral loads in order to account for cases when structures are both occupied and experiencing wind or earthquake loads.

13. Click the Load Combination Name field to select a Combination to which to add a new Load Combination Usage.
14. In the Load Combination field, select the Load Combination to which a new Load Combination Usage is to be applied. This is achieved by clicking anywhere in the row of the Load Combination.

15. In the Load Combination Usage field, check the new Load Combination Usage that you want.

Note**:** Notice that as soon as a Load Combination Usage is checked, it applies itself to the selected Load Combination.

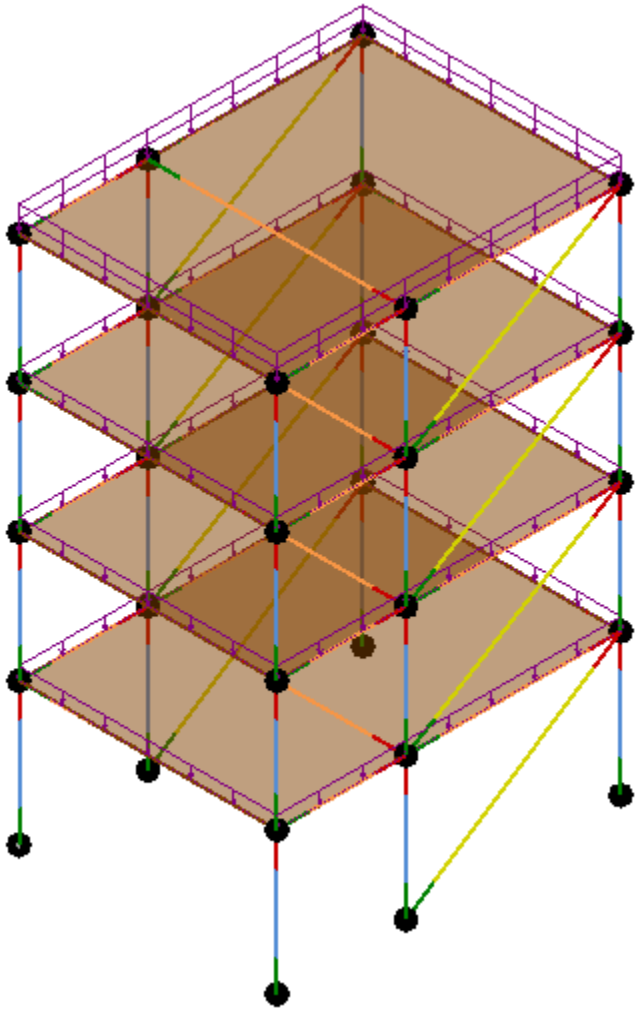
16. Click OK to exit the dialog.

 *Tips:*

By default, load combinations generated by add-ins are not displayed. To view them, select Show third-party generated combinations. However, they are not available for editing.

Exercise 3 | Apply Loads to the Structural Elements

Apply Hosted Loads



Goal: Assign Point, Line and Area Loads

Open Exercise_03_01_StartPoint.rvt

Linear Loads

1. Click Analyze tab ➤ Analytical Model panel ➤ (Loads).
2. Click Modify | Place Loads tab ➤ Loads panel ➤ (Hosted Line Load).
3. On the Properties palette, select a DL (1) value for Load Case.
4. For the Orient to parameter, select Project or Host Local Coordinate System.
5. Select top level's perimetral beams.

Area Loads

1. When you finish sketching, click Create Area Load Boundary tab ➤ Area Load panel ➤ (Finish Area Load).
2. Click Analyze tab ➤ Analytical Model panel ➤  (Loads).
3. Click Modify | Place Loads tab ➤ Loads panel ➤  (Hosted Area Load).
4. On the Properties palette, select DL for Load Case.
5. For the Orient to parameter, select Project or Host Local Coordinate System.
6. Select each floor to assign the area load

Use the predefined Visibility/Graphics Filters based on load cases to visualize easier each load case and assign new loads for each.

View tab ➤ Graphics panel ➤ open Visibility/Graphics dialog ➤ go to Filters tab

Integrate Structural Analysis Results in Revit Workflow

Robot Structural Analysis Toolkit for Revit

The Structural Analysis Toolkit for Autodesk® Revit® software is a suite of tools that supports the Building Information Modeling (BIM) process and allows structural engineers to analyze and check your structure from within the Autodesk® Revit® environment.

Using this toolkit structural designers and engineers can optimize their workflows by extending the Revit model to Autodesk® Robot™ Structural Analysis Professional software or supported third party analysis solutions. Once complete, analysis results can be easily stored and explored in the Revit environment.

Analysis Result Settings

After performing an analysis and applying an analysis display style to the view, you can modify view properties to change the visibility, display style, units and scale configuration of individual results.

On the Properties palette for the view, click the Edit button next to Analysis Display Settings under the Graphics heading.

In the Analysis Results Settings Dialog, each analysis configuration is listed with visibility options:
Show analysis in this view. Displays analysis as defined in the Analysis Results Settings dialog.
Name. A list of the results calculated by the add-in application.
Visible. Displays the result and legend in the current view.

Analysis Display Style. The analysis display style assigned to the individual result. Click  to override the default view.
Units. The available units for the result as returned by the add-in application.
Scale Units. The current model scale unit for the currently selected unit. This field is returned by the add-in application. It is editable for both Diagrams with text and Vectors with text display styles, otherwise it is represented by the default units of the document per analysis unit.
Max. The maximum range value for each result. This read-only field is returned by the add-in application.
Min. The minimum range value for each result. This read-only field is returned by the add-in application.
All/None/Invert. Switches the visibility setting for multiple results.

Accessing and Downloading Results Packages

After an analysis has been run, a result package is created. When the analysis is complete, you can use the **Results Manager** to access these results packages.

1. In Revit, click Analyze tab ➤ Structural Analysis panel ➤  Results Manager.
   The **Results Manager** dialog opens and shows the list of available results packages, as well as their status and location.
   Note**:** There are three locations where the results packages can be saved:
   - **In project**: i.e. in the Revit® project.
   - **Remote**: i.e. on the cloud server. In this case, the size of the package is also indicated.
   - **In project and remote**: in this case, the results package has already been saved in the Revit® project, and a copy remains on the server.
2. Select a remote result package, and then click **Download**.
3. Click **Explore** to visualize the content of the selected results package or close the **Results Manager** dialog.

Visualizing Structural Analysis Results in Revit

Results Explorer tool displays different types of results for static analysis and gravity analysis.

Depending on the analysis type, results are presented in the form of maps and diagrams using the settings available in Revit They are displayed on elements of the analytical model for which the analysis was performed.

When an analysis has been run, and results are available on the server, the **Results Explorer** tool  becomes active in the **Structural Analysis** panel of the **Analyze** tab.
Note**:** If the **Results Explorer** tool is not active, then it means that no results are available on the server. This can happen either because no analysis has been run, or because the analysis has failed.

If you selected model elements in the graphic pane, results display only for the selected elements. If you do not select model elements in the graphic pane, results display for the whole model.

➢ **Display structural analysis results in Revit**

1. On the View Control Bar, click  Show Analytical Model.
2. Click Analyze tab ➤ Structural Analysis panel ➤  Results Explorer.

The **Results** dialog opens.

3. At the top of the **Results** dialog, select the model, analysis, and load case for which you want to display results.
4. In the **Result** column, select the result you want to display.
5. At the bottom of the Results dialog, click **>** if you want to specify display settings such as the style, units, and scale for the selected result.

6. Specify if you want the **Local coordinate system for members** to be displayed.

Note**:** Select this option to enable display of the Local Coordinate System (LCS) widgets along Analytical Columns, Analytical Beams, and Analytical Braces. Selecting this option enables Local Coordinate System Widgets for these analytical elements in the Visibility/Graphics dialog.

7. Specify if you want the **Local coordinate system for surfaces** to be displayed.

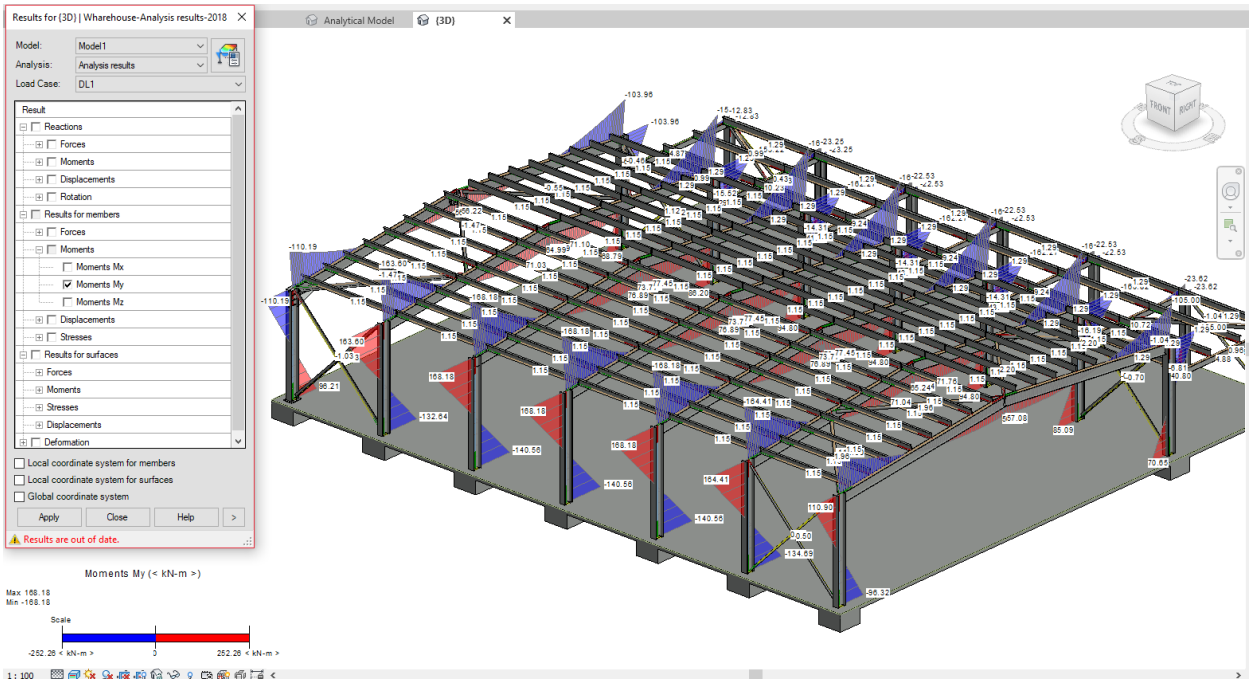
Note**:** Select this option to enable display of the Local Coordinate System (LCS) widgets along Analytical Floors, Analytical Slabs, and Analytical Walls. Selecting this option enables Local Coordinate System Widgets for these analytical elements in the Visibility/Graphics dialog.

8. Specify if you want the **Global coordinate system** to be displayed.
9. Click **Apply**.

The results are displayed on the graphic pane.

Note**:** When you click  (Results Explorer) the application checks if the current analytical model is identical to the model for which results were calculated. If you modify the analytical model or the load cases after the analysis has been run, a warning appears which notifies you that the results are out of date.

Exercise 4 | Integrate Structural Analysis Results using Robot Structural Analysis Toolkit



Open Exercise_04_StartPoint.rvt

1. Go to Analyze tab ➤ Structural Analysis panel ➤ Result Explorer
2. Explore the results