BUILDING INFORMATION MODELLING

Training Materials

Navisworks

Advanced Construction Information Development Ltd.

Table of Contents

1 Navisworks Manage File System

In this chapter:

Navisworks file types

Opening files directly

Exporting from Revit to Navisworks

Navisworks Appending and merging models

1.1 Navisworks file types

Autodesk Navisworks has three native file formats: NWD, NWF, and NWC.

1.1.1 NWD File Format

An NWD file contains all model geometry together with Autodesk Navisworks-specific data, such as review markups. You can think of an NWD file as a snapshot of the current state of the model.

NWD files are very small, as they compress the CAD data by up to 80% of the original size.

1.1.2 NWF File Format

An NWF file contains links to the original native files (as listed on the Selection Tree) together with Autodesk Navisworks-specific data, such as review markups. No model geometry is saved with this file format; this makes an NWF considerably smaller in size than an NWD.

1.1.3 NWC File Format (Cache Files)

By default, when you open or append any native CAD or laser scan files in Autodesk Navisworks, a cache file is created in the same directory and with the same name as the original file, but with an .nwc file extension.

NWC files are smaller than the original files, and speed up your access to commonly used files. When you next open file or append file in Autodesk Navisworks, the data is read from the corresponding cache file if it is newer than the original file. If the cache file is older, which means the original file has changed, Autodesk Navisworks converts the updated file, and creates a new cache file for it.

1.2 Opening files directly

In Autodesk Navisworks you can open files originated from a variety of CAD applications.

You can combine these files together, and create a single Autodesk Navisworks file with a whole-project view of your model. This file brings together geometry and data created by multi- disciplinary teams, and enables you to explore and review complex models in real-time.

1.3 Exporting from Revit to Navisworks

Autodesk Navisworks can read native Revit (RVT) files directly. Alternatively, you can use the file exporter to save your files in NWC format.

The file exporter is available for Revit 2018.

1.3.1 To export NWC files from Revit

1. In Revit, click Add-Ins External Tools Autodesk Navisworks.

Note: This option is not available in Demo/Viewer mode. If you are not in demo mode, but do not have access to the Autodesk Navisworks menu, check if editing view is set to normal, and the modify tool is selected (Modify).

2. In the Export Scene As dialog box, enter the name for the Autodesk Navisworks file, and browse to the desired storage location.
3. Click Save to export the file, or Cancel to return to the application without exporting it.

1.3.2 To adjust the options for the Revit file exporter

1. In Revit, click Add-Ins ➤External Tools ➤Autodesk Navisworks.
2. In the Export Scene As dialog box, click the Autodesk Navisworks Settings button.
3. Expand the File Exporters node in the Options Editor, and click the Revit page. Use the options on this page to adjust the settings for future exports of NWC files from Revit.
4. Click OK to save the changes and return to the Export Scene As dialog box.
5. Click Cancel to close the dialog box.

1.4 Appending and merging models

1.4.1 To append 3D models in a multi-sheet file

1. Open a multi-sheet file.
2. If the Sheet Browser window is not displayed, click on the Status bar.
3. Double-click the desired 3D model in the Sheet Browser to open it in the Scene View.
4. Use the Sheet Browser to select all 3D models that you want to append to the currently open model.
   Tip: To select multiple model, use SHIFT and CTRL keys.
5. Right-click the selection, and click Append to Current Model. Note: The Undo command is not available.

1.4.2 To merge files

1. Click New on the Quick Access toolbar.
2. Open the first of the files with the review markup.
3. Click Home tab Project panel Merge.
4. In the Merge dialog box, use the Files of Type box to select the appropriate file type (NWD or NWF), and navigate to the folder where your files you want to merge are located.
5. Select the required files, and click Open.
   Tip: To select multiple files, use SHIFT and CTRL keys.Command entry: CTRL + M

2 Navigating in a Model

In this chapter:

Looking at the navigation bar

Rotating your model with the Orbit tool

Looking around your model

Using the Gravity tool

Using the ViewCube

2.1 Looking at the navigation bar

The navigation bar is a user interface element where you can access both unified and product-specific navigation tools.

Unified navigation tools (such as Autodesk® ViewCube®) are those that can be found across many Autodesk products. Product-specific navigation tools are unique to a product. The navigation bar floats over and along one of the sides of the Scene View.

You start navigation tools by clicking one of the buttons on the navigation bar or selecting one of the tools from a list that is displayed when you click the smaller portion of a split button.

1. **View Cube** Indicates the current orientation of a model, and is used to reorient the current view of a model. Clicking this button displays the View Cube in the Scene View when it's not visible.
2. **Steering Wheels** Collection of wheels that offer rapid switching between specialized navigation tools.
3. **Pan tool**. Activates the pan tool and moves the view parallel to the screen.
4. Zoom tools. Set of navigation tools for increasing or decreasing the magnification of the current view of the model.
5. **Orbit tools**. Set of navigation tools for rotating the model around a pivot point while the view remains fixed.
6. **Look tools**. Set of navigation tools for rotating the current view vertically and horizontally.
7. **Walk and Fly tools**. Set of navigation tools for moving around the model and controlling realism settings.
8. **Select tool**. Geometry selection tool. You cannot navigate through your model during geometry selection.
9. **3Dconnexion**. Set of navigation tools used to reorient the current view of a model with a 3Dconnexion 3D mouse.
   Note: In a 2D workspace, only the 2D navigation tools (such as 2D Steering Wheels, Pan, Zoom, and the 2D Mode 3Dconnexion tools) are accessible.

2.2 Walking through your model

Set of navigation tools for moving around the model and controlling realism settings. These tools are not available in a 2D workspace.

The following tools are available:

- Walk. Moves through a model as if you were walking through it.
- Fly. Moves through a model like in a flight simulator.

2.1.1 Walk

The tool is activated by clicking Walk in the Walk/Fly drop-down on the navigation bar. By default, the tool behaves like the Walk tool on the Steering Wheels. You can customize the tool options in the Options Editor. You can also switch back to the legacy Walk mode.

2.1.2 Fly

The tool is activated by clicking Fly in the Walk/Fly drop-down on the navigation bar.

2.2 Rotating your model with the Orbit tool

Set of navigation tools for rotating the model around a pivot point while the view remains fixed. These tools are not available in a 2D

workspace. The following orbit tools are

available:

- **Orbit**. Moves the camera around the focal point of the model. The up direction is always maintained, and no camera rolling is possible.
- **Free Orbit**. Rotates the model around the focal point in any direction.
- **Constrained Orbit**. Spins the model around the up vector as though the model is sitting on a turntable. The up direction is always maintained.

2.2.1 Orbit

The tool is activated by clicking Orbit in the Orbit drop-down on the navigation bar. It behaves the same way as the Orbit tool on the Steering Wheels. You can use the Options Editor to switch back to the legacy Orbit mode.

2.2.2 Free Orbit

The tool is activated by clicking Free Orbit in the Orbit drop-down on the navigation bar. You can use the Options Editor to switch back to the legacy Examine mode.

2.2.3 Constrained Orbit

The tool is activated by clicking Constrained Orbit in the Orbit drop-down on the navigation bar. You can use the Options Editor to switch back to the legacy Turntable mode.

2.3 Looking around your model

2.3.1 Pan Tool

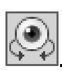
The pan tool moves the view parallel to the screen.

The tool is activated by clicking Pan  on the navigation bar. Pan behaves the same way as the pan tool available on the Steering Wheels.

2.3.2 Look Tools

Set of navigation tools for rotating the current view vertically and horizontally. These tools are not available in a 2D workspace.

The following look tools are available:

- Look Around . Looks around the scene from the current camera location.
- Look At . Looks at a particular point in the scene. The camera moves to align with that point.
- Focus . Looks at a particular point in the scene. The camera stays

where it is. Look Around

The tool is activated by clicking Look Around in the Look drop-down on the navigation bar. It behaves the same way as the Look tool available on the Steering Wheels.

Look At
The tool is activated by clicking Look At in the Look drop-down on the navigation bar. It behaves the same way as the Steering Wheels Look tool when you press and hold the SHIFT key.

Focus
The tool is activated by clicking Focus in the Look drop-down on the navigation bar.

2.4 Using the Gravity tool

To toggle gravity

When using the Walk tool, click Viewpoint tab ➤ Navigate panel ➤ Realism drop-down ➤ Gravity check box.

Command entry: CTRL + G

Note: This function only works in connection with collision.
Where collision gives you mass, gravity gives you weight. As such, you (as the collision volume) will be pulled downwards whilst walking through the scene.

Note: Gravity can only be used with the Walk navigation tool. This allows you to walk down stairs, for example, or follow terrain.

2.5 Using the ViewCube

The ViewCube tool is a persistent, clickable, and draggable interface that you use to switch between views of your model.

When you display the ViewCube tool, by default it is shown in the top-right corner of the Scene View over the model in an inactive state. The ViewCube tool provides visual feedback about the current viewpoint of the model as view changes occur. When the cursor is positioned over the ViewCube tool, it becomes active. You can drag or click the ViewCube, switch to one of the available preset views, roll the current view, or change to the Home view of the model.

Figure 6-1 View Cube

Tip: When the navigation bar is linked to the ViewCube, both can be moved around the Scene View.

2.5.1 Control the Appearance of ViewCube

The ViewCube tool is displayed in one of two states: inactive and active. When the ViewCube tool is inactive, it appears partially transparent by default so that it does not obscure the view of the model. When active, it is opaque and may obscure the view of the objects in the current view of the model.

In addition to controlling the opacity level of the ViewCube when it is inactive, you can also control its size, and the display of the compass. The settings used to control the appearance of the ViewCube are located in the Options Editor.

2.5.2 Use the Compass

The compass is displayed below the ViewCube tool and indicates which direction North is defined for the model. You can click a cardinal direction letter on the compass to rotate the model, or you can click and drag one of the cardinal direction letters or the compass ring to interactively rotate the model around the pivot point.

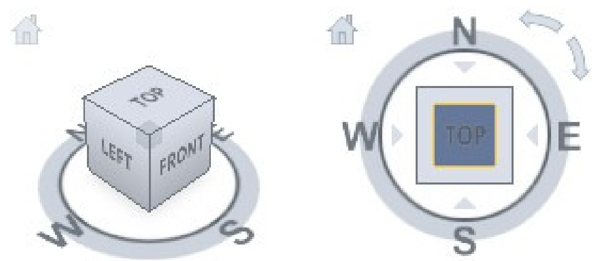


Figure 6-2 Compass View of View Cube

2.5.3 Drag or Click ViewCube

When you drag or click the ViewCube tool, the view of the model reorients around a pivot point. The pivot point is displayed at the center of the object that was last selected before using the ViewCube tool.

3 Working with Viewpoints

In this chapter:

Saving a viewpoint

Choosing render styles

Changing the background

Slicing your model using sectioning

Getting rid of lines and text

3.1 Saving a viewpoint

Viewpoints are snapshots taken of the model as it is displayed in the Scene View. They can be taken by clicking the Save Viewpoint button under the Viewpoint Tab. Importantly, viewpoints can be used for more than just saving information about the view of the model. For example, they can be annotated with redlines and comments, allowing you to use viewpoints as a design review audit trail. Viewpoints can also be used as links in the Scene View, so that when you click on and zoom to the viewpoint, Autodesk Navisworks also displays the redlines and comments associated with it.

The viewpoints, redlines and comments are all saved into an NWF file from Autodesk Navisworks, and are independent of the model geometry. So, if the native CAD files are changing, the saved viewpoints remain the same, appearing as an overlay on top of the base layer of model geometry. This enables you to see how the design has evolved. See Review Your Model for more information on links, comments and redlines and Native File Formats for more information on the NWF file format.

Viewpoints encompass a range of different information about the view of the model, navigation settings, and annotations in the form of redlines and comments. See Default Viewpoint Options for more information.

3.1.1 View of the Model

- Camera position, projection mode, field of view and orientation
- Lighting mode, render mode and toggles for the display of different geometry types (surfaces, lines, points)
- Sectioning configuration

Additionally, the following item overrides can be saved with the viewpoint (this is optional):

- Visibility (hidden / required)
- Appearance (color and transparency)

3.1.2 Navigation

- Linear and angular speeds of motion
- Realism settings (collision, gravity, third person, crouch)
- The currently selected navigation tool

3.1.3 Annotations

- Redlines
- Comments

3.1.4 Saved Viewpoint Window

The Saved Viewpoints window is a dockable window that enables you to create and manage different views of your model so that you can jump to preset viewpoints without having to navigate each time to reach an item.

Viewpoint animations are also saved with the viewpoints, as they are simply a list of viewpoints treated as keyframes. In fact, viewpoint animations can be made by dragging preset viewpoints onto an empty viewpoint animation. You can organize your viewpoints and viewpoint animations using folders.

Icons are used to represent different elements:

represents a folder which may contain all other elements (including other

folders). represents a viewpoint saved in orthographic mode.

represents a viewpoint saved in

perspective mode. represents a viewpoint

animation clip.

represents a cut inserted into a viewpoint animation clip.

You can select more than one viewpoint by either holding down the CTRL key and left-clicking, or by left-clicking the first item, and then clicking the last item while holding down the SHIFT key.

You can drag viewpoints around the Saved Viewpoints window, and reorganize them into folders or animations.

There are no buttons on this window, and commands are invoked through context menus.

Through these menus, you can save and update viewpoints, create and manage viewpoint animations, and create folders to organize these viewpoints and viewpoint animations. You can also drag and drop viewpoints or viewpoint animations onto viewpoint animations or folders.
Holding down the CTRL key during this operation will copy the element being dragged. This allows complex hierarchies of viewpoint animations and folders to be easily composed.

Viewpoints, folders and viewpoint animations can all be renamed by slow clicking (clicking and pausing without moving the mouse) the element, or clicking it and pressing F2.

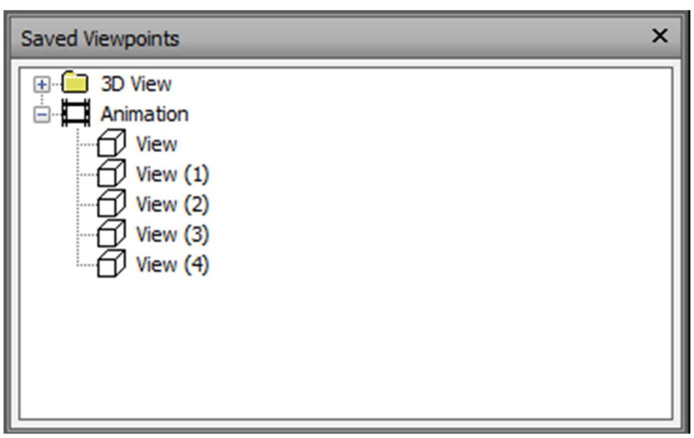


Figure 7-1: Saved Viewpoints Window

3.2 Choosing render styles

Use the Autodesk Rendering window to access and work with materials libraries, lighting, and environment settings.

The Autodesk Rendering window is a dockable window, which is used to set up materials and lights in your scene, as well as environments settings, and render quality and speed. It can be

accessed by pressing the Autodesk Rendering  button.

The Autodesk Rendering window contains the Rendering toolbar, and the following tabs:

- **Materials.** Allows you to navigate and manage material collections, known as libraries, provided by Autodesk, or create custom libraries for specific projects. Default material library Includes a variety of materials, which can be selected and applied to your model. You can also use this tab to create new materials, or customize existing materials.
- **Material Mapping.** Allows you to adjust the orientation of the texture to fit the shape of the object. This feature is recommended for advanced users only.
- **Lighting.** Allows you to view the lights already added to the model, and customize the lighting properties.
- **Environments.** Allows you to customize the Sun, the Sky, and Exposure properties.
- **Settings.** Allows you to change the Render Style presets. You can select from a range of default quality presets or customize your own render settings.

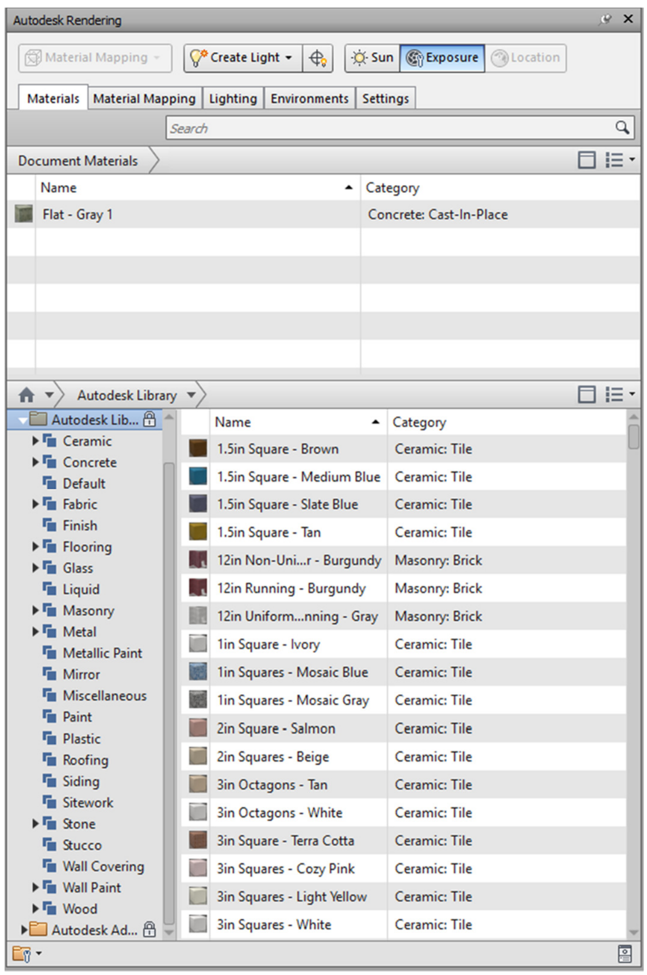


Figure 7-2: Autodesk Rendering Window

3.3 Changing the background

Use this dialog box to choose a background effect to use in the Scene View

3.3.1 Mode

Selects the type of background effect. Choose from:

- Plain
- Graduated
- Horizon

Note: Horizon mode and the associated colors are only available for 3D models.

3.3.2 Color

Sets the color for a plain background.

3.3.3 Top Color

Sets the top color in a graduated background.

3.3.4 Bottom Color

Sets the bottom color in the graduated background.

3.3.5 Sky Color

Sets the sky color (top) in a horizon background. This option is available for 3D models only.

3.3.6 Horizon Sky Color

Sets the sky color (bottom) in a horizon background. This option is available for 3D models only.

3.3.7 Horizon Ground Color

Sets the ground color (top) in a horizon background. This option is available for 3D models only.

3.3.8 Ground Color

Sets the ground color (bottom) in a horizon background. This option is available for 3D models only.

Ribbon: View tab ➤Scene View panel ➤Background 
Shortcut menu: Right-click a blank area in the scene, and click Background on the context menu.

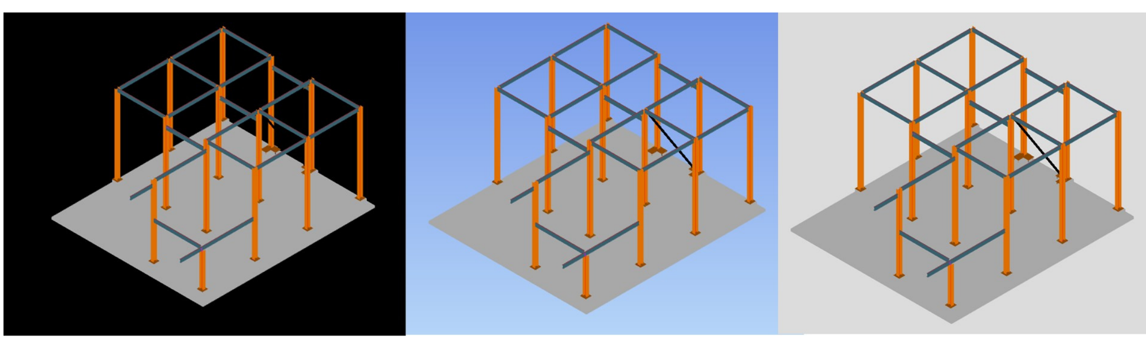


Figure 7-3: Backgrounds – Plain, Graduated, Horizon (Below Horizon)

3.4 Slicing your model using sectioning

Autodesk Navisworks enables you to turn on sectioning for the current viewpoint and to create cross sections of your model in a 3D workspace. The sectioning functionality is not available for 2D sheets.

A cross section is a cut-away view of a 3D object that enable you to see inside it. You can turn sectioning on and off for the current viewpoint by clicking Viewpoint tab ➤ Sectioning panel ➤ Enable Sectioning. When the sectioning is turned on, the Sectioning Tools contextual tab is automatically displayed on the ribbon.

There are two sectioning modes available from the Sectioning Tools tab ➤ Mode panel: Planes and Box.

Planes mode allows you to make up to six sectional cuts in any plane while still being able to navigate around the scene, enabling you to see inside models without hiding any item. By default, section planes are created through the center of the visible area of the model.

3.5 Getting rid of lines and text

3.5.1 To delete text

Right-click the text you want to delete, and click Delete Redline on the context menu.

3.5.2 To erase redlines

1. Click Viewpoint tab ➤ Save, Load & Playback panel ➤ Saved Viewpoints drop-down, and choose the viewpoint that you want to review.
2. Click Review tab ➤ Redline panel ➤ Draw drop-down, and click Erase.
3. Drag a box over the redline you want to delete, and release the mouse.

4 The Review Tab

In this chapter:

Getting measurements

Finding the shortest distance between objects

Creating redlines

4.1 Getting measurements

To access the tools, click Review tab Measure panel on the ribbon.

4.1.1 Measure Tool

- Measures the distance between two points.
- Measures the distance between a base point and various other points.
- Measures a total distance between multiple points along a route.
- Calculates the sum total of several point-to-point measurements.
- Calculates an angle between two lines.
- Calculates an area on a plane.
- Measures the shortest distance between two selected objects.
- Clears all measuring lines in the Scene View.
- Enables you to move or rotate an object.
- Converts the endpoint markers, the lines, and any displayed measurement values into redlines.

4.2 Finding the shortest distance between objects

Measure tools enable you to measure between points on items in the model. All measurements are made in display units. The measure tools are available from Review tab Measure panel.

Using measure tools is mutually exclusive to using navigation tools (see Product-Specific Navigation Tools), so that when you are measuring you cannot navigate and vice versa.

After selecting a measuring tool, select the points on the model that need to be measured. Navisworks will then show the information it has collected from that measurement.

Figure 4-1: Measure Tool

To clear a measurement, select **Clear Measurement** in the **Measure Panel**.

4.3 Creating redlines

The Redline Tools panel on the Review tab enables you to markup viewpoints and clash results with redline annotations. When you create a redline, the associated viewpoint is automatically saved.

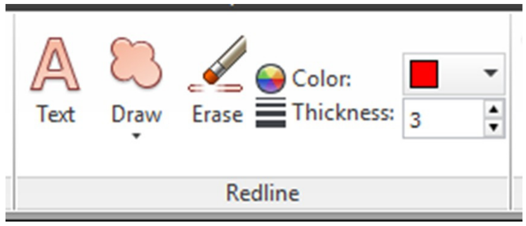


Figure 4-2: Reline Panel

The Thickness and Color controls enable you to modify the redline settings. These changes do not affect already drawn redlines. Also, thickness only applies to lines; it does not affect redline text, which has a default size and weight and cannot be modified.

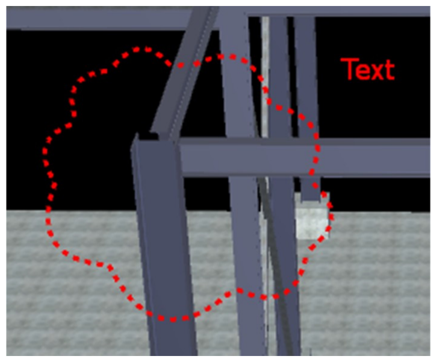


Figure 4-3: Redline Markup

You can also convert measurements to redlines with the **Convert To Redline** tool .

5 Dealing with Objects

In this chapter:

Selecting objects

Overriding color, transparency, and a transform

Hiding and turning on objects

Moving items

Rotating items

Scaling items

Adding links

Viewing properties and adding fields

Switching back to Revit

Holding objects

5.1 Selecting objects

With large models, it is potentially a very time-consuming process to select items of interest. Autodesk Navisworks makes this a much simpler task by providing a range of functions for quickly selecting geometry both interactively and by searching the model manually and automatically.

9.1.1 Interactive Geometry Selection

In Autodesk Navisworks, there is a concept of an active selection set (the currently selected items, or the current selection) and saved selections sets. Selecting and finding items makes them part of the current selection, so you can hide them or override their colors. At any time, the current selection can be saved and named for retrieval in later sessions with the **Save Selection**  button.

Selecting items makes them part of the current selection, so you can hide them or override their colors.

You can use several methods to interactively select items into the current selection. You can use the drop-down list in the Selection Tree, select items directly in the Scene View with the Select and Select Box tools, and you can select other items with similar properties to an existing selection using the selection commands under the **Home Tab** on the **Select and Search Panel**. Note: Right-clicking any item in the Selection Tree or Scene View opens a context menu.

You can also customize the level at which you select items (selection resolution), and modify the highlighting method for the items selected in the Scene View.

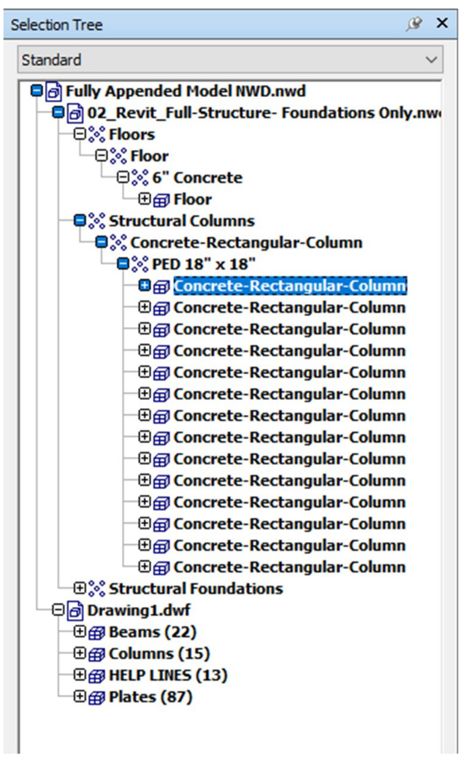


Figure 5-1: Selection Tree – Column Selected

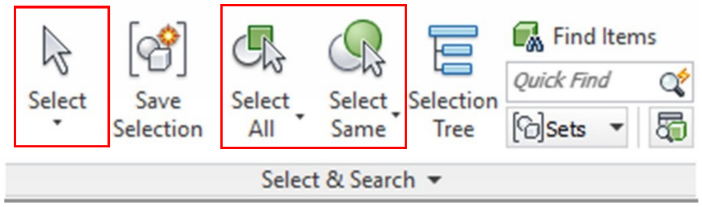


Figure 5-2: Home Tab - Select & Search Tools

5.2 Overriding color, transparency, and a transform

5.2.1 Manipulate Object Attributes

In Autodesk Navisworks, you can manipulate objects' transforms (translation, rotation, and scale), and also change appearance (color and transparency) of objects. All object manipulation is carried out in the Scene View.

Any changes that you make to object attributes are considered to be global, (as if they'd been changed in the original CAD model), and can be saved with Autodesk Navisworks files. You have an option of resetting object attributes back to the state they were in when imported from the original CAD files.

To transform objects, you can use three visual manipulation tools, or gizmos, available from the **Item Tools Tab, Transform Panel**. You can also transform objects numerically.

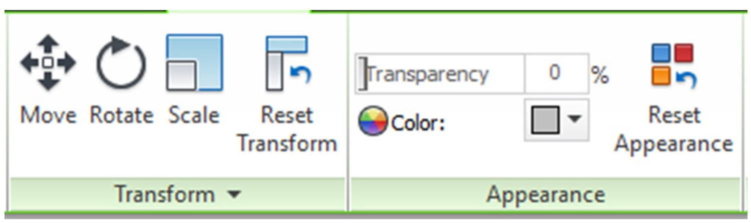


Figure 9-3: Item Tools – Transform and Appearance Panels

To get a clearer view of objects as you manipulate them, you can use the Options Editor to adjust the way in which the current selection is highlighted. For more information, see Set Highlighting Method.

5.2.2 Using Measure Tools to Transform Objects

You can use the Measure Tools functionality to move and rotate the currently selected objects.

5.2.3 Examples of Overrides

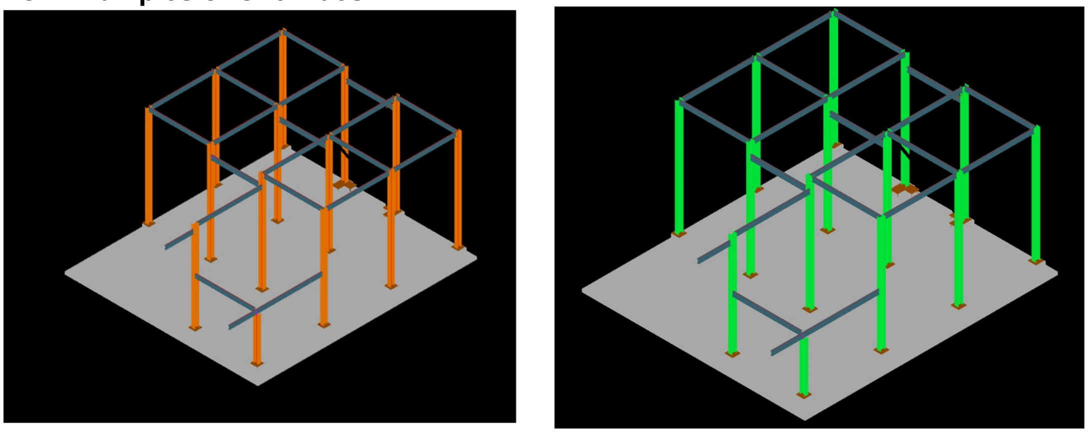


Figure 5-4: Color Change Override

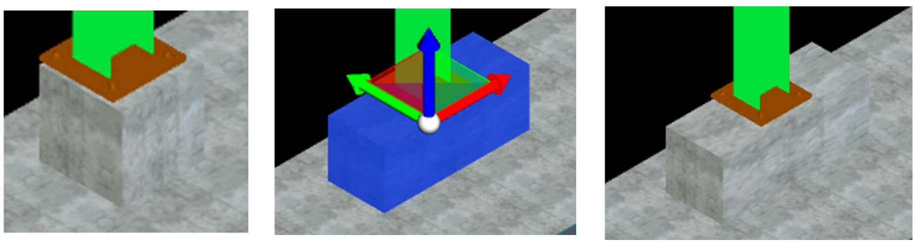


Figure 5-5: Scale Change Override

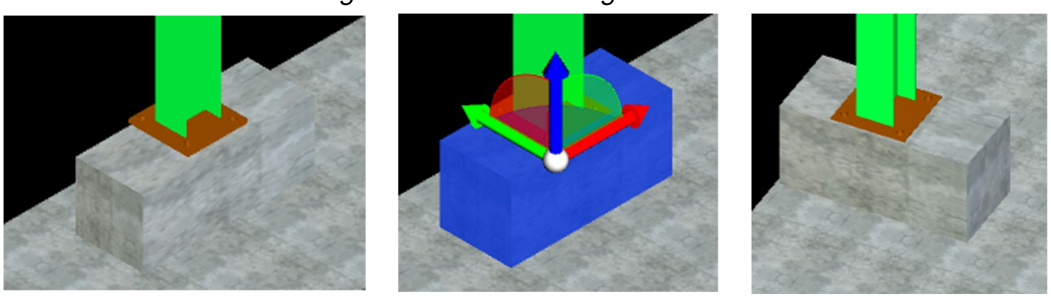


Figure 5-4: Rotate Change Override

5.3 Hiding and turning on objects

Autodesk Navisworks provides tools that can be used to hide and display objects or groups of objects. Hidden objects are not drawn in the Scene View.

5.3.1 Hide Selected Objects

You can hide the objects in the current selection so that they are not drawn in the Scene View. This is useful when you want to remove specific parts of the model. For example, when you walk down the corridor of building, you may want to hide a wall that occludes your view of the next room.

5.3.2 Hide Unselected Objects

You can hide all items except those currently selected so that they are not drawn in the Scene View. This is useful when you only want to see specific parts of the model.

5.3.3 To hide selected objects

1. In the Scene View, select all items you want to hide.
2. Click Home tab ➤ Visibility panel ➤ Hide . The selected objects are now invisible.

Clicking Hide again displays the invisible objects.

5.4 Moving items

5.4.1 To move an object numerically

1. Select the object you want to move in the Scene View.
2. Click the Item Tools tab, and slide out the Transform panel.
3. Type in numerical values into the manual entry boxes to move the object by the amount entered:
   - Position X, Y, Z represent translation distance in the current model unit.
   - Transform Center X, Y, Z represent the translation center point.

5.4.2 To move an object with a measure tool

1. Select the object you want to move.
2. Click Review tab ➤ Measure panel ➤ Measure drop-down ➤ Point Line .
3. Click the selected object to create the first point. This is the start point from which the repositioning will be calculated.
4. Click the second point in the scene. This is the point where the object will be moved to. There is now a line connecting the start point and the end point in the Scene View.
5. If you want to be able to move the object several times, create more points in the scene. Note: You can only select a point on another object in the scene. Selecting a point in 'space' is not a valid option. To reposition an object into 'space', you can either use a translation gizmo or, if you know the distance by which the object is to be moved, by overriding its transform.
6. Slide out the Measure panel, and click Transform Selected Items to move the object to the second point. If you have multiple points in the scene, each time you
   click Transform Objects the selected object is moved to the next point.

5.5 Rotating items

5.5.1 To rotate an object with gizmo

1. Select the object you want to rotate in the Scene View.
2. Click Item Tools tab ➤ Transform panel ➤ Rotate .
3. Use the gizmo to rotate the currently selected object:
   - Before you can rotate the currently selected objects, you need to position the origin (center point) of the rotation. To do this, place the mouse over the arrow at the end of the desired axis. When the cursor changes to , drag the arrow on

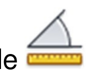
the screen to increase/decrease the translation along that axis. This will move the gizmo itself.
Dragging the ball in the middle of the rotate gizmo enables you to move it around, and snap it to points on other geometry objects.

- Once the rotate gizmo is positioned correctly, place the mouse over one of the curves in the middle, and drag it on the screen to rotate the selected objects. The curves are color-coded, and match the color of the axis used to rotate the object around. So, for example, dragging the blue curve between the X and Y axes, rotates the objects around the blue Z axis.
- To rotate the orientation of the gizmo to an arbitrary position, hold down the CTRL key while dragging one of the three curves in the middle.
- To snap the gizmo to other objects, hold the CTRL key while dragging the ball in the middle of the gizmo.

5.5.2 To rotate an object numerically

1. Select the object you want to rotate in the Scene View.
2. Click the Item Tools tab, and slide out the Transform panel.
3. Type in numerical values into the manual entry boxes to move the object by the amount entered:
    - Rotation X, Y, Z represent degrees of rotation in the current model unit.
    - Transform Center X, Y, Z represent the rotation center point.

5.5.3 To rotate an object with a measure tool

1. Select the object you want to move.
2. Click Review tab ➤ Measure panel ➤ Measure drop-down ➤ Measure Angle .
3. To rotate the object, click three points on the selected object to determine how the object is to be rotated.
4. Slide out the Measure panel, and click Transform Selected Items . This rotates the selected object from position A into position B (as shown in the previous diagram). Clicking this button again, rotates the object from position B into position C.

5.6 Scaling items

5.6.1 To resize an object with gizmo

1. Select the object you want to resize in the Scene View.
2. Click Item Tools tab ➤ Transform panel ➤ Scale .
3. Use the scale gizmo to resize the currently selected object:
    - To resize the objects across a single axis only, use colored arrows at the end of the axes. To resize the objects across two axes at the same time, use colored triangles in the middle of the axes. Finally, to resize the objects across all three axes at the same time, use the ball in the center of the gizmo.
    - You can modify the center of scaling. To do this, place the mouse over the ball in the middle of the gizmo, and hold down the CTRL key while dragging the ball on the screen.

5.6.2 To resize an object numerically

1. Select the object you want to resize in the Scene View.

2. Click the Item Tools tab, and slide out the Transform panel.
3. Type in numerical values into the manual entry boxes to move the object by the amount entered:
   - Scale X, Y, Z represent a scaling factor (1 being the current size, 0.5 half, 2 being double, and so on).
   - Transform Center X, Y, Z represent the scaling center point.

5.7 Adding links

You can add links that point to various data sources, such as spreadsheets, webpages, scripts, graphics, audio and video files and so on. An object can have multiple links attached to it, although, only one link, called default, is shown in the Scene View. The default link is the link that you add first, but you can mark a different link as default, if necessary.

5.7.1 To add a link to an object

1. In the Scene View, select the desired geometry item.
2. Click Item Tools tab ➤Links panel ➤Add Link .
3. In the Add Link dialog box, enter the name for the link in the Name box.
4. In the Link to File or URL box, type in the full path to the required data source or the URL address. You can also browse to the folder containing the desired external file.
5. Choose the category for your link from the Category drop-down list. by default, your link has hyperlink category.
   Tip: To create a custom category type, type its name directly into the Category box. when you save your link, the corresponding user-defined category is automatically created.
6. Optional: By default, your link is attached to the default center of the item's bounding box. If you want to attach your link to a specific point on the selected item, click the Add button. A cross-hair cursor appears in the Scene View, allowing you to select a point on the item where the link will be attached to.
   Note: If you made a mistake, clicking the Clear All button deletes all attachment points associated with this link and reverts to the link being attached to the center of the item's bounding box.
7. Click OK.

5.7.2 To add several links to the same object

1. In the Scene View, select the desired geometry item.
2. Right-click and click Links Add Link.
3. Use the Add Link dialog box to add the first link. This is the default link, and it's the only link that will be visible in the Scene View. You can choose a different default link later, if necessary.
4. Right-click the object again, and repeat the previous steps to add all required links.

5.8 Viewing properties and adding fields

5.8.1 To examine object properties

1. Select the object of interest in the Selection Tree, or in the Scene View.
2. Open the Properties window under the Home Tab, and use the tabs to navigate between the available property categories.

Note: If more than one object is selected, the Properties window only shows the number of selected items, and doesn't show any property information.

5.8.2 To add a custom property tab

1. Open the Properties window.
2. Select the object of interest in the Scene View or on the Selection Tree.
3. Right-click the Properties window, and click Add New User Data Tab. The new property category is added for the currently selected object. By default, the tab is called User Data.

5.9 Switching back to Revit

1. For Revit or products based on it, open the product in the usual manner, and initialize the Navisworks SwitchBack add-in:
   a. Open any existing project, or create a new one.
   b. Click Add-Ins tab External Tools Navisworks SwitchBack to enable it. You can now close the project, but don't close Revit.
2. Return to Autodesk Navisworks and open the desired file. As long as you are working with an NWC file exported from Revit, or a saved NWF or NWD file, you can SwitchBack to Revit.
3. Select an object in the Scene View, and click Item Tools tab SwitchBack panel SwitchBack . Revit will load the relevant project, find and select the item, and zoom to it. If SwitchBack was unsuccessful with the selected object, and you receive an error message, try selecting further up the Selection Tree in Autodesk Navisworks.
   Tip: Alternatively, in the Clash Detective window, on the Results tab, you can click the SwitchBack button. The Clash Detective feature is available for Autodesk Navisworks Manage users only.

Note: If you try to use SwitchBack and the RVT file is not in the same location as when it was saved, a dialog box will appear asking you to browse to the RVT file. When using SwitchBack for the first time to load a Revit file, a 3D view based on the chosen projection view mode is created in Autodesk Navisworks. The next time SwitchBack is used to load a Revit file, the same projection view mode loads, unless the projection view mode is changed
in Autodesk Navisworks.

5.10 Holding objects

1. Select the objects you want to hold either in the Scene View or in the Selection Tree.
2. Click Item Tools tab Hold panel Hold .
   The selected objects are now held and will move with you through the model when you use navigation tools, such as Walk, Pan and so on.
3. To release the held objects, click Hold on the ribbon again.
4. If you want to reset the objects to their original position, click Item Tools tab Transform panel Reset Transform .

6 Sorting and grouping

In this chapter:

6.1 The Selection Tree

The Selection Tree is a dockable window, which displays a variety of hierarchical views of the structure of the model, as defined by the CAD application in which the model was created.

Autodesk Navisworks uses this hierarchical structure to identify object-specific paths (from the file name down to a particular object).

For Revit 2014 and later (RVT or NWC) files, the model hierarchy can be structured to display category, family, type and instance below each level, and show the material from which the Revit object is made. For example, a door may be broken up into constituent parts such as glass or wood panel.

By default, there are four options on the drop-down list:

- Standard. Displays the default tree hierarchy, including all instancing. The hierarchy can be sorted alphabetically.
- Compact. Displays a simplified version of the hierarchy on the Standard option, omitting various items. You can customize the level of complexity of this tree in the Options Editor.
- Properties. Displays the hierarchy based on the items' properties. This enables simple manual searching of the model by item property.
- Sets. Displays a list of selection and search sets. If no selection and search sets have been created, this option is not shown.
  Note: The list of the items on the Sets option is exactly the same as the list on the Sets dockable window.

Additional customized Selection Tree options can be added by using the Autodesk Navisworks API.

Naming of items reflects the names from the original CAD application, wherever possible. You can copy and paste names from the Selection Tree. To do this, right-click an item in the Selection Tree, and click Copy Name on the context menu. Alternatively, you can click an item in the Selection Tree, and press CTRL + C. The name is now copied to the clipboard.

There are different tree icons representing the types of geometry making up the structure of the model. Each of these item types can be marked as hidden (gray), unhidden (dark blue) or required (red).

Note: If a group is marked as hidden or required, then all instances of that group are marked as hidden or required. If you want to operate on a single occurrence of an item, then you should mark the instanced group (the level above, or the “parent”, in the hierarchy) hidden or required.

6.2 Finding items

1. On the Selection Tree, click the items where you want to start searching from. For example, if you want to search the whole model, select the Standard option from the drop-down list, press and hold CTRL, and click all files that comprise the model. If you want to limit your search to a selection set, select the Sets option from the drop-down list, and click the required set.
2. Define a search statement:

a. Click the Category column, and select the property category name from the drop- down list, for example, ‘Item’.
b. In the Property column, select the property name from the drop-down list, for example, ‘Material’.
c. In the Condition column, select the condition operator, for example, “Contains”.
d. In the Value column, type in the property value to search for, for example, “Chrome”.
e. If you want to make your search statement case-insensitive, right-click it, and click Ignore String Value Case.

3. Define more search statements, if required.
   By default, all statements are ANDed. This means that they all need to be true for an item to be selected. You can make a statement use OR logic, by right-clicking it and clicking Or Condition. If you are using two statements, and marked the second one as ORed, this means that an item will be selected if either of these statements is true.
4. Click the Find All button. The search results are highlighted in the Scene View and the Selection Tree.

6.3 Saving selections as a set

The Sets window Sets is a dockable window, which displays both selection sets and search sets available in your Autodesk Navisworks file.

The selection sets are identified by this icon: , and the search sets are identified by this icon: .

Note: The list of the items on the Sets window is exactly the same as the list on the Sets drop- down option on the Selection Tree.

You can customize the names of selection and search sets, and add comments. You can copy and paste names from the Sets window. To do this, right-click an item in the Sets window, and click Copy Name on the context menu. Alternatively, you can click an item in the Sets window, and press CTRL + C. The name is now copied to the clipboard.

You can also show the selection and search sets as links in the Scene View. These links are created automatically by Autodesk Navisworks. Clicking on a link restores the geometry within the corresponding selection or search set to the active selection, and highlights it in the Scene View and on the Selection Tree.

You can use the buttons in the Sets window to create and manage selection and search sets in the Autodesk Navisworks file.

6.4 Using the Quick Find tool

1. Click Home tab ➤Select & Search panel.
2. In the Quick Find text box Quick Find , type in the string to search for in all item’s properties. This can be a word or a few words. The search is not case-sensitive.
3. Click Quick Find. Autodesk Navisworks finds and selects the first item in the Selection Tree that matches the entered text, selects it in the Scene View, and stops the search.

4. To find more items, click Quick Find again. If there are any more items that match the entered text, Autodesk Navisworks selects the next one in the Selection Tree, selects it in the Scene View, and stops the search. Subsequent clicks find next instances.

Command entry: To open the Quick Find dialog box: CTRL + F. To Find Next: F3

6.5 Using the Selection Inspector

The Selection Inspector is a dockable window which can be accessed in the Home Tab, which displays a list of all the selected objects and the Quick Properties associated with those objects.

You can inspect a selection from the Selection Tree or Scene View. Alternatively, choose a Selection Set or a Search Set. You can then zoom the selection to display it within the model, or modify it by deselecting objects and adding property definitions.

When you save a selection, it appears in the Sets window. You can then rename your selection set. See Sets Window for more details.

All the objects that are visible within the Selection Inspector window may be exported into a CSV file. You must deselect objects if you wish to exclude them from the exported file.

7 Clash Detection

In this chapter:

Creating a clash test and setting rules

Selecting objects to clash and adding clearances

Running the clash

Resolving and reducing clashes

Grouping and assigning clashes

Creating report; Viewpoints

Creating reports: HTML

7.1 Creating a clash test and setting rules

Exported clash tests can be used as a basis to define custom clash tests. If you have a common set of clash tests that you reuse on multiple projects, you can turn them into a custom clash test. Once installed as a custom clash test, all the tests can be selected and run directly from the Select Tab. The results from all tests are combined and presented as the results of the custom clash test. The name of each individual test in the custom clash test is displayed in the Description field of the results.

Custom clash tests are an excellent way to roll out a standardized set of tests across an organization. They allow the expertise of “power” users to be reused by everyone. They can also be seen as a way of implementing object intelligence. For example, a custom clash test could be written that checked for compliance with a local building code based on object information and properties defined in a particular CAD system.

7.1.1 To define and use a custom clash test

1. Export Clash Tests to an XML file. The name of the file is used as the default name for the custom test.
2. If you want, change the name of the custom test by editing the XML file directly. The top- level element in the XML file is called “batchtest”. The name of the custom test as displayed to the user is defined by the “name” attribute. The name of the custom test as saved in a file is defined by the “internal name” attribute.
3. To install the custom test, copy the exported XML file to the custom_clash_tests folder of one of the Autodesk Navisworks search directories, for example: C:\Documents and Settings\All Users\Application Data\Autodesk Navisworks 2018\custom_clash_tests. See Search Directories for more information.
4. Restart Autodesk Navisworks. On startup, Clash Detective checks these search directories for custom clash tests.
5. To use the custom test, open the Clash Detective window, and click the Select tab.
6. Select your custom test from the Type drop-down box.
7. Click the Start button. All other options and rules are specified by the custom test.

7.2 Selecting objects to clash and adding clearances

1. If the Clash Detective window is not already open, click Home tab Tools panel Clash Detective .
2. Click the Tests panel expand button, and select the test you want to configure.
3. Click the Select tab.
   There are two identical panes in this tab called Selection A and Selection B. These panes contain tree views of two sets of items that will be tested against each other during the clash test, and you need to select items in each. You can select the items by choosing an option from the drop-down list for each selection tree and manually selecting items from the tree hierarchies. Any selection sets in the scene are also available from the drop-down list, which is a quick and useful method of setting up items across sessions (see Interactive Geometry Selection).
   You can also transfer the current selection to one of the selection panes by selecting items in the usual way in the Scene View or Selection Tree and clicking the appropriate Use Current Selection button.

4. Optional: Select the appropriate Self Intersect button to test the corresponding set for self-intersection, as well as intersection against the other set.
5. Optional: You can include the clashing of points, lines, or surfaces in your test. Underneath each window there are three buttons that correspond to surfaces, lines and points. To toggle a button, click it.
   So, for example, if you want to run a clash test between some surface geometry and a point cloud, you can set up the geometry in the Selection A pane, and click the Point Cloud button under the Selection B pane. The Surface button under the Selection A pane is toggled on by default.
   Additionally, you could set the clash Type to Clearance with a Tolerance of 1 meter.
   Note: If the Type is set to Hard, lines and surfaces will actually need to intersect with any points to register a clash.

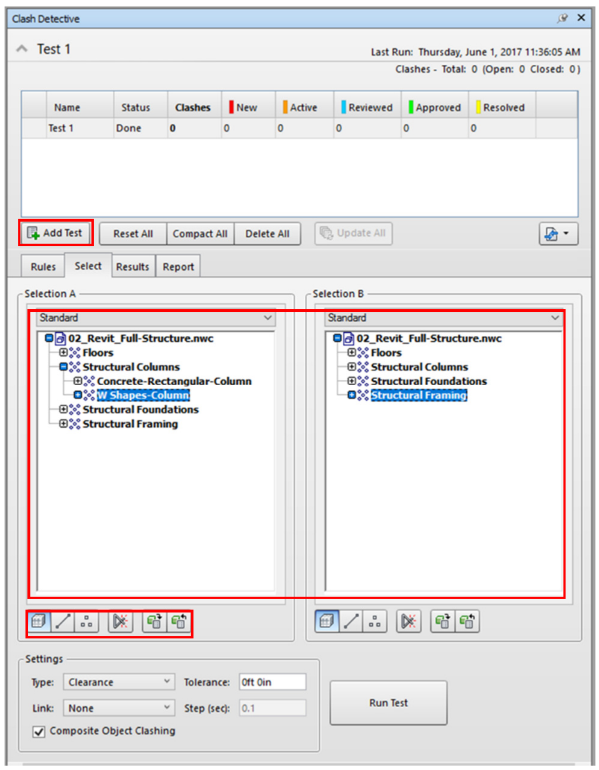


Figure 11-1: Clash Detective

7.3 Running the clash

If the Clash Detective window is not already open, click Home tab ➤ Tools panel Clash Detective .

1. Click the Tests panel expand button and select the test you want to run.
2. Click the Select tab and set the required test options.
3. Once the Selection A and Selection B items are selected, and the clash type and tolerance defined, click the Run Test button to start the test running.

Note: The progress bar shows how far through the test Clash Detective has got. If you wish to stop the test at any time, press the Cancel button. All clashes found up until the interrupt will be reported, and the test will be saved with a Partial status.
When the test is complete
the Results tab will open displaying the results of the test, unless there were no clashes found.

7.4 Resolving and reducing clashes

There are four default clash test types for you to choose from:

- **Hard**. Choose this option if you want the clash test to detect actual intersections between geometry.
- **Hard (Conservative).** This option performs the same clash test as Hard, however, it additionally applies a conservative intersection method.
- **Clearance**. Choose this option if you want the clash test to check for geometry within a specific distance from other geometry (see tolerance for more information). You can use this type of clash when, for example, pipes need to have space for insulation around them.
  Note: Clearance clashes are not the same as “soft” clashes. Clearance clashes detect for static geometry coming within a distance of other geometry, whereas soft clashes detect potential clashes between moving components. Clash Detective supports soft clash checking when you link it to Object Animation.
- **Duplicates**. Choose this option if you want the clash test to detect for duplicate geometry. You can use this type of clash test to check a model against itself to ensure the same part has not been drawn, or referenced twice, for example.

7.5 Grouping and assigning clashes

7.5.1 To group clashes

1. On the Results tab, select all clashes you want to group together and click Group.
2. Type in a new name for the group, and press Enter.
   Note: Clash groups are identified with the icon. When clashes are grouped, they are treated as a single clash in the count at the top of the Clash Detective window. Each clash group is counted as a single clash unless no clashes have been added to it, in which case it will not be included in the count.

When you click the created clash group, the two panes in the Items panel show all the clashing items contained within that clash group, and all corresponding clashes are shown in the Scene View.

7.5.2 To create a clash group

1. Click New Group on the Results tab. A new group called New Group X is added above the currently selected clash (or at the top of the list if nothing is selected).
2. Type in a new name for the group, and press Enter. Note: Clash groups are identified with the icon.
3. Select clashes you want to add to this group, and drag them into the group.
   Note: When clashes are grouped, they are treated as a single clash in the count at the top of the Clash Detective window. Each clash group is counted as a single clash unless no clashes have been added to it, in which case it will not be included in the count.

4. When you click the created clash group, the two panes in the Items panel show all the clashing items contained within that clash group, and all corresponding clashes are shown in the Scene View.

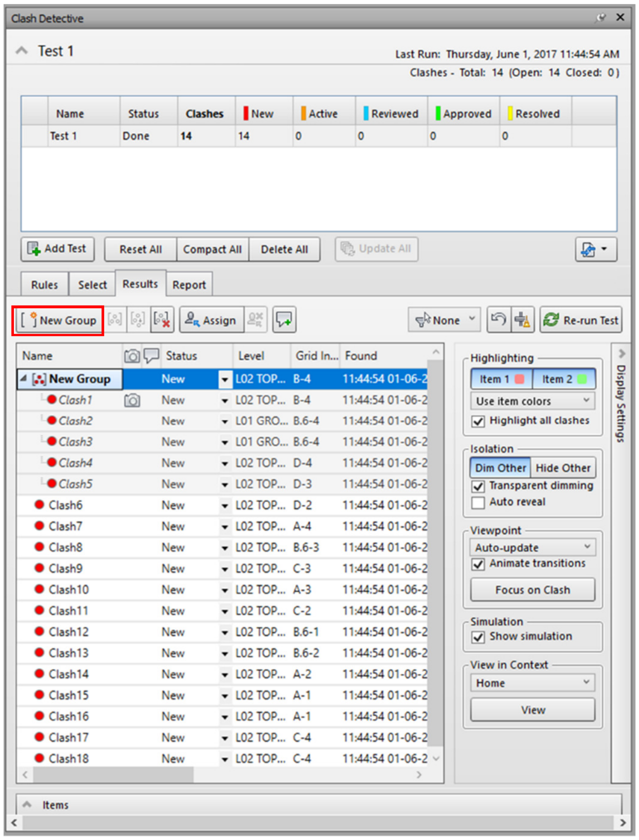


Figure 11-2: Results and Clash Groups

7.6 1Creating report; Viewpoints

11.6.1 To save viewpoints with clash results manually

1. In the Clash Detective window, click the Results tab.
2. Right-click in the Viewpoints column for a clash result and select Save Viewpoint. The position displayed in the Scene View will be saved as the viewpoint for the clash. If there is no existing viewpoint thumbnail then a new viewpoint thumbnail will be created in the Results grid.
   Note: Selecting Focus on Clash always returns to the clashes original default viewpoint.

7.6.2 To save viewpoints with clash results automatically

1. In the Clash Detective window, click the Results tab Display Settings show/hide button.
2. Select the Auto-Update option from the Viewpoint drop-down list.
   Now when you navigate away from the default viewpoint for a clash in the Scene View, the viewpoint for the clash will be updated to the new position. If there is no existing viewpoint thumbnail then a new viewpoint thumbnail will be created in the Results grid. Note: Selecting Focus on Clash always returns to the clashes original default viewpoint. Once redlining has been added, subsequent changes to the viewpoint due to navigation

will not be saved. In order to save a different viewpoint, the redlining must first be removed using the redline Erase tool, or by manually saving the viewpoint.

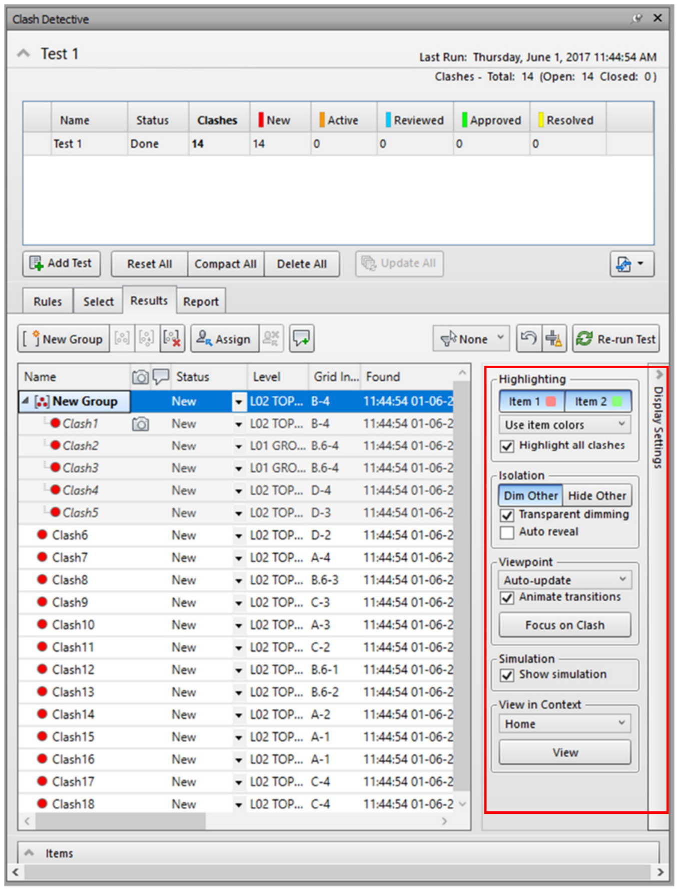


Figure 7-3: Clash Detective Results – Viewpoints

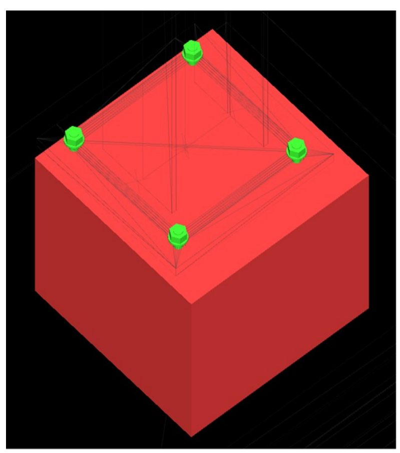


Figure 7-4: Clash Viewpoint Example

7.7 Creating reports: HTML

1. In the Clash Detective window, run the desired test. If you ran all tests in the Tests panel, select the test you want to view the results for.
2. Click the Report tab.
3. In the Contents area select the check boxes for the data you want to appear in the report for each clash result. This can include Quick Properties relating to the items involved in the clash, how to find them in the standard Selection Tree from root to geometry, whether images or simulation information should be included, and so on.
4. In the For-Clash Groups Include drop-down list, select how to display the clash groups in your report.
   Note: If your test does not contain any clash groups, this box is not available.
5. Choose from the following options:
   - **Group Headers Only** - your report will include summaries of clash groups and individual clashes not in groups.
   - **Individual Clashes Only** - your report will only contain individual clash results, and will not distinguish those results that have been grouped. For each clash that belongs to a group, an extra field, called Clash Group, can be added to the report to identify it. To enable this functionality, select the Clash Group check box in the Contents area.
   - **Everything** - your report will contain summaries of clash groups that you have created, clash results that are part of each group, and individual clash results. For each clash that belongs to a group, an extra field, called Clash Group, can be added to the report to identify it. To enable this functionality, select the Clash Group check box in the Contents area.
6. Use the Include Clash Types box to select the clash results that you want to report on.
7. Select the type of report in the Report Type box:
   - Current Test creates a single report file for the current test.
   - All Tests (Combined) creates a single file containing all results from all tests.
   - All Tests (Separate) creates a separate file for each test containing all results.
8. Select the format of the report in the Report Format box:
   - XML creates an XML file containing all the clashes and a jpeg of their viewpoints alongside their details. On choosing this option, you will need to select or create a folder for the files and enter a name for the XML file.
   - HTML will create an HTML file containing all the clashes and a jpeg of their viewpoints alongside their details. On choosing this option, you will need to select or create a folder for the files and enter a name for the HTML file.
   - HTML (Tabular) will create an HTML tabular file containing all the clashes and a jpeg of their viewpoints alongside their details. On choosing this option, you will need to select or create a folder for the files and enter a name for the HTML file. HTML tabular files can be opened and edited in Microsoft Excel 2007 onwards. Note: To customize the appearance or layout of the HTML / HTML (Tabular) file, you will need to edit the clash_report_html.xsl / clash_report_html_tabular.xsl file. The installed file is located in the stylesheets subdirectory of the Autodesk Navisworks install directory. You can copy the edited file to the stylesheets subdirectory of any of the Autodesk Navisworks search directories. See Search Directories for more information.

- Text creates a TXT file containing all the clash details and the location of a jpeg of each clash. On choosing this option, you will need to select or create a folder for the files and enter a name for the TXT file.
- As Viewpoints creates a folder in the Saved Viewpoints dock able window (this window is automatically displayed when the report is run), with the same name as the test. Each clash is saved as a viewpoint in this folder, with a comment attached containing the clash result details.
  Note: The Home view is the default camera orientation for clash viewpoints.

9. Click the Write Report button to write the report.
   Note: Reports will respect any sorting of the data that you have done, for example into ascending or descending order, on the Results tab.

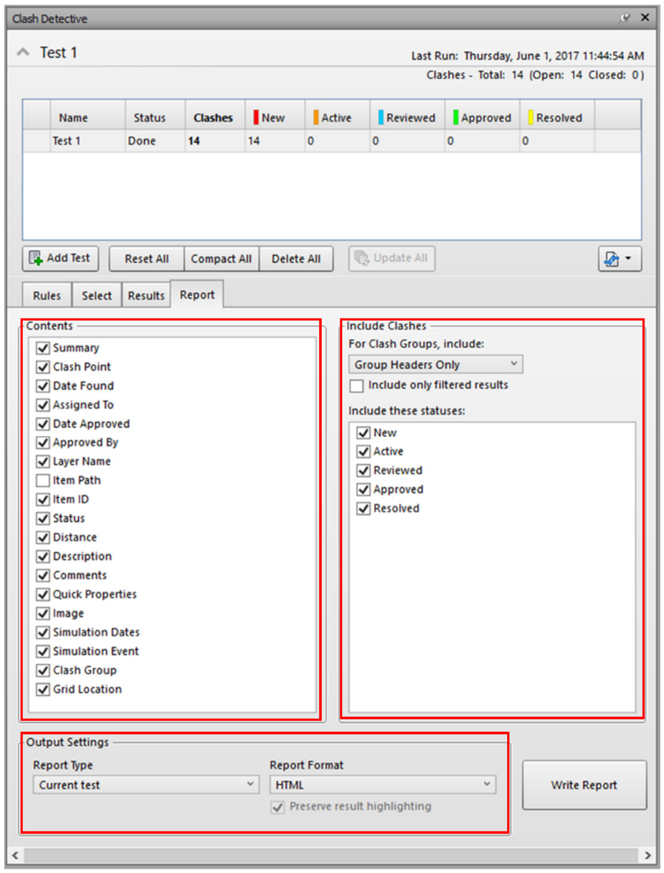


Figure 11-5: Clash Detective Report

8 The TimeLiner

In this chapter:

Configuring appearances

Creating tasks

Adding selection sets to a task

Adding multiple tasks and running the movie

Simulating settings

Exporting the TimeLiner

Adding a data source

8.1 Configuring appearances

The TimeLiner tool adds schedule simulation to Autodesk Navisworks. TimeLiner imports schedules from a variety of sources. You can then connect tasks in the schedule with objects in the model to create a simulation. This allows you to see the effects of the schedule on the model, and compare planned dates against actual dates. Costs can also be assigned to tasks to track the cost of a project throughout its schedule. TimeLiner also allows the export of images and animations based on the results of the simulation. TimeLiner will automatically update the simulation if the model or schedule changes.

You can combine the functionality of TimeLiner with other Autodesk Navisworks tools:

- Linking TimeLiner and Object Animation together enables the triggering and scheduling of object movement based on start time and duration of project tasks, and can help you with workspace and process planning. For example, a TimeLiner sequence may indicate that when a particular site crane moves from its start point to its end point over the course of a particular afternoon, a workgroup working nearby causes an obstruction along its route. This potential obstruction problem can be resolved before going to site (e.g., the crane can be moved along a different route, the workgroup moved out of the way, or the project schedule altered). See Add Animation to TimeLiner Schedules for more information.
- Linking TimeLiner and Clash Detective together enables the time-based clash checks on the project. See Clash Detective User Guide for further details on time-based clashing. Note: This feature is available for Autodesk Navisworks Manage users only.
- Linking TimeLiner, Object Animation, and Clash Detective together enables clash testing of fully animated TimeLiner schedules. So, instead of visually inspecting a TimeLiner sequence to make sure, for example, that the moving crane didn't collide with a workgroup, you can run a clash test.
  Note: This feature is available for Autodesk Navisworks Manage users only.

8.2 Creating tasks

1. Load your model into Autodesk Navisworks (see Open Files if you need help).
2. Click Home tab ➤ Tools panel ➤ TimeLiner, and click the Tasks tab in the TimeLiner window.
3. Click Add Task or right-click anywhere in the task view, and click Add Task on the context menu.

   Note: You can click on an existing task and select Insert Task to insert a task above the selected task.
4. Enter the name for your task, and press Enter. The task is now added to your schedule. Note: If you press Enter when the bottom task in the task view is selected, then a new task will be created below it.

8.3 Adding selection sets to a task

8.3.1 To add tasks based on the Selection Tree structure

1. If the TimeLiner window is not already open, click Home tab ➤Tools panel ➤TimeLiner 

2. In the TimeLiner window Tasks tab, click Auto-Add Tasks .

3. Click For Every Topmost Layer if you want to create tasks with the same names as each topmost layer in the Selection Tree.
   Click For Every Topmost Item if you want to create tasks with the same names as each topmost item in the Selection Tree. This can be a layer, a group, block or cell, or geometry, depending on how the model is constructed.

Note: Planned Start and End dates will be automatically created, starting from the current system date and incrementing by one day for each subsequent end and start date. The Task Type will be set to Construct.

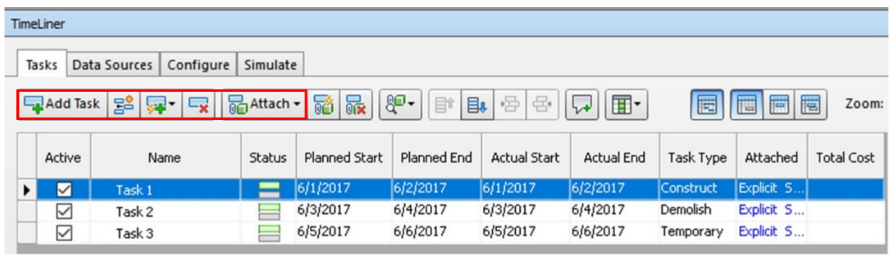


Figure 8-1: Adding Timeliner Tasks

8.3.2 To build tasks from a data source

1. If the TimeLiner window is not already open, click Home tab ➤ ➤
   Tools panel TimeLiner 
2. In the TimeLiner window, the Data Sources tab, click the Refresh button, select either Selected Data Source or All Data Sources then select Rebuild Task Hierarchy in the Refresh from Data Source Dialog Box and click OK.
   This will import all of the tasks from the project file into TimeLiner.
   Note: You can also right-click on a file in the Data Sources tab and select the Rebuild Task Hierarchy option.
3. Click the Tasks tab to view the created tasks. The task table is populated in accordance with predefined settings for the data source. You can make any necessary overrides in the Field Selector Dialog Box.

Note: Although the tasks have now been imported into TimeLiner, you still need to Attach Tasks to Geometry before you can run a simulation. The fastest way to attach imported tasks is by applying rules (see Use Rules to Attach Tasks).

8.4 Adding multiple tasks and running the movie

1. If the TimeLiner window is not already open, click Home tab ➤Tools panel ➤TimeLiner 
2. On the Tasks tab, click the task you want to add an animation to, and use the horizontal scroll bar to locate the Animation column.
3. Click the drop-down arrow in the Animation field, and select a scene or an animation set from a scene. When you select a scene, all animation sets recorded for this scene will be used.
4. Click the drop-down arrow in the Animation Behavior field, and select how the animation will play during this task:
   - Scale - the duration of the animation is matched to the duration of the task. This is the default setting.
   - Match Start - the animation starts when the task starts. If the animation runs past the end of the TimeLiner simulation, the end of the animation will be clipped.
   - Match End - the animation starts early enough so that it ends just when the task end. If the animation starts before the beginning of the TimeLiner simulation, the start of the animation will be clipped.

8.5 Simulating settings

The Settings button on the Simulate tab provides access to the Simulation Settings dialog box.

It is possible to override the Start and End dates that the simulation runs between. Selecting the Override Start/End Dates check box enables the date boxes and allows you to choose the start and end dates. By doing this, you can simulate a small sub-section of the overall project. The dates will be shown on the Simulate tab. These dates will also be used when exporting animations.

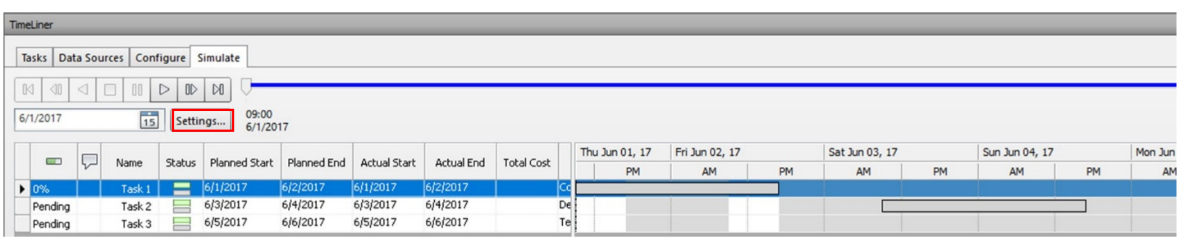


Figure 12-2: Simulate Tab

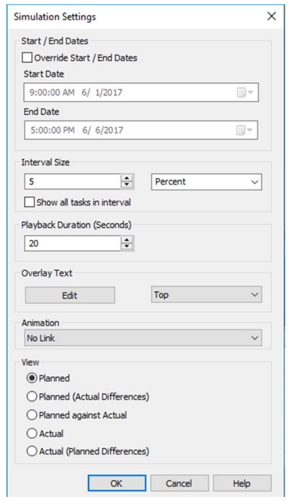


Figure 8-3: Simulate Settings

You can define the Interval Size to use when stepping through the simulation using the playback controls. The interval size can be set either as a percentage of the overall simulation duration or to an absolute number of days or weeks, and so on.

Use the drop-down list to select the interval unit, then use the Up and Down arrow buttons to increase or decrease the interval size.

It is also possible to highlight all the tasks that are being worked on during the interval. By selecting the Show All Tasks in Interval check box, and, for example, setting the Interval Size to 5 Days, all tasks being worked on during those 5 days will be set to their Start Appearance in the Scene View, including those that begin and end within the bounds of the interval. The Simulation slider will show this by drawing a blue line under the slider. If this check box is clear, tasks that begin and end within the bounds of the interval will not be highlighted in this manner, and will need to overlap with the current date in order to be highlighted in the Scene View.

You can define the overall Playback Duration for the complete simulation (the time needed to play it through from start to finish). Use the Up and Down arrow buttons to increase or decrease the duration (in seconds). You may also enter a duration directly into this field.

You can define whether the current simulation date should be overlaid in the Scene View, and if so whether it should appear at the top or bottom of the screen. From the drop-down, choose from None (to display no overlay text), Top (to display the text at the top of the window), or Bottom (to display the text at the bottom of the window).

You can Edit the information displayed in the overlay text using the Overlay Text Dialog Box. This dialog box also makes it possible to alter the Font Type, Style and Size by clicking on the contained Font button.

You can add animation to an entire schedule, so that during the TimeLiner sequence playback, Autodesk Navisworks will also play the specified viewpoint animation or camera.

The following options can be selected in the Animation field:

- No Link - no viewpoint animation or camera animation will be played.

- Saved Viewpoints Animation - links your schedule to the currently selected viewpoint or viewpoint animation.
- Scene X - Camera - links your schedule to a camera animation in the selected animation scene.

You can pre-record suitable animations for use with the TimeLiner simulation (see Record and Play Animations). Using animation also affects the Animation Export.

View area. Each view will playback the schedule depicting Planned and Actual relationships:

- **Actual**. Choose this view to simulate the Actual schedule only (that is, only use the Actual Start and Actual End dates).
- **Actual (Planned Differences**). Choose this view to simulate the Actual schedule against the Planned schedule. This view will only highlight the items attached to the task over the Actual date range (that is, between Actual Start and Actual End. See diagram below for graphical representation). For time periods where the Actual dates are within the Planned dates (on schedule), the items attached to the task will be displayed in the Task Type Start Appearance. For time periods where the Actual dates are early, or late in comparison to the Planned dates (there is a variance), then the items attached to the task will be displayed in the Task TypeEarly or Late Appearance, respectively.
- **Planned**. Choose this view to simulate the Planned schedule only (that is, only use the Planned Start and Planned End dates).
- **Planned (Actual Differences).** Choose this view to simulate the Actual schedule against the Planned schedule. This view will only highlight the items attached to the task over the Planned date range (that is, between PlannedStart and Planned End. See diagram below for graphical representation). For time periods where the Actual dates are within the Planned dates (on schedule), the items attached to the task will be displayed in the Task TypeStart Appearance. For time periods where the Actual dates are early, or late in comparison to the Planned dates (there is a variance), then the items attached to the task will be displayed in the Task TypeEarly or Late Appearance, respectively.
- **Planned Against Actual.** Choose this view to simulate the Actual schedule against the Planned schedule. This will highlight the items attached to the task over the entire Planned and Actual date range (that is, between the earliest of Actual and Planned Start dates and the latest of Actual and Planned End dates. See diagrams below for graphical representation). For time periods where the Actual dates are within the Planned dates (on schedule), the items attached to the task will be displayed in the Task Type Start Appearance. For time periods where the Actual dates are early, or late in comparison to the Planned dates (there is a variance), then the items attached to the task will be displayed in the Task Type Early or Late Appearance, respectively.

8.6 Exporting the TimeLiner

1. In the TimeLiner window, Simulate tab, click Export Animation . The Animation Export dialog box opens.
2. To export the currently selected viewpoint animation, select Current Animation in the Source box.
   To export the currently selected object animation, select Current Animator Scene in the Source box.
   To export a TimeLiner sequence, select TimeLiner Simulation in the Source box.

3. Set up the rest of the boxes in the Animation Export dialog box, and click OK. For more information, see Animation Export Dialog Box.
4. In the Save As dialog box, enter a new filename and location, if you want to change from those suggested.
5. Click Save.

8.7 Adding a data source

8.7.1 To import data from an external project schedule

1. If the TimeLiner window is not already open, click Home tab ➤ ➤ Tools panel TimeLiner
2. In the TimeLiner window click the Data Sources tab.
3. Click the Add button and choose the required option from a list of the project sources that may be connected to on the current PC.
   Note: For more information on which sources are typically available, see Supported Scheduling Software.
4. Use the standard Open dialog box to browse to and open the desired project file. TimeLiner accesses the selected file in accordance with the predefined parameters configured for the corresponding data source using a COM interface.
   Depending on the data source, the Field Selector Dialog Box may be displayed. You can use it to override some of the predefined data import options.
5. By default, your data source is called “New Data Source(x)”, where “x” is the latest available number. To make your data source more descriptive, right-click it,
   click Rename on the context menu, and enter a new name.

8.7.2 To import data from a Primavera

Note: This procedure shows how to import data from a Primavera P6 project. Follow the same steps to import data from a Primavera P6 V7, V8 or V8.2 project, selecting the Primavera P6 V7, V8, or V8.2 options where applicable.

1. If the TimeLiner window is not already open, click Home tab ➤ ➤ Tools panel TimeLiner .
2. In the TimeLiner window, click the Data Sources tab.
3. Click the Add button and click Primavera P6 (Web Services).
   Note: If you can’t see this option, you must set up a Primavera Web Server first. Refer to the Primavera P6 Web Server Administrator Guide (available in your Primavera documentation).
4. When the Primavera login dialog box is displayed, enter your user name, password, and the server address.
   Tip: The Server Address is the name of a machine on your domain or an IP address for the server.
5. In the Primavera P6 Database Instance Selection dialog box double-click the desired Instance ID to select it.
6. In the Primavera P6 Project Selection dialog box double-click the desired project file to open it.

7. Use the Field Selector Dialog Box to override some of the predefined data import options.
   TimeLiner connects to the selected project file.

8.7.3 To import CSV data

1. If the TimeLiner window is not already open, click Home tab ➤ ➤ Tools panel TimeLiner 
2. In the TimeLiner window click the Data Sources tab.
3. Click the Add button and click CSV Import.
4. Use the standard Open dialog box to locate the desired project file in CSV format, and click Open.
5. In the Field Selector Dialog Box, use the CSV Import Settings area to specify how your data should be imported into TimeLiner.
   Note: The CSV import settings, including field mappings, are remembered by the system and pre-populated when the dialog box opens. If you are linking to a different CSV file than was previously used, TimeLiner will attempt to map any columns to similarly-named columns in the CSV file.
   You should have a column in the CSV file containing unique data, for example an incrementing number, and map it to the Synchronization ID field. This unique data must remain the same once it has been set against a row in the CSV file to enable synchronization to enable you to rebuild and synchronize the data source link. The Task Name field must also be mapped.
   - Select the Row 1 Contains Headings check box if you want the first row of data in your CSV file to be treated as column headings. TimeLiner will use it to populate the External Field Name options in the grid.
     If the first row of data in your CSV file does not contain column headings, clear this check box.
   - Select the Automatically Detect Date and Time option if you want TimeLiner to attempt to determine the date/time format used in your CSV file.
     First, TimeLiner applies a set of rules to try to establish the date/time format used in the document; if it is not possible, it uses the local settings on your system.
     Select the Use Specific Date/Time Format option if you want to manually specify the date/time format that should be used. When this radio button is selected, you can enter the required format into the box provided. For the list of valid codes, see Field Selector Dialog Box.
     Note: If one or more date/time-based columns are found to contain fields where the data cannot be mapped to a valid date/time value using the manually- specified format, TimeLiner will 'fall back' and attempt to use the automatic date/time format.
6. By default, your data source is called "New Data Source(x)", where "x" is the latest available number. To make your data source more descriptive, right-click it,
   click Rename on the context menu, and enter a new name.

8.7.4 Timeliner Simulate Example

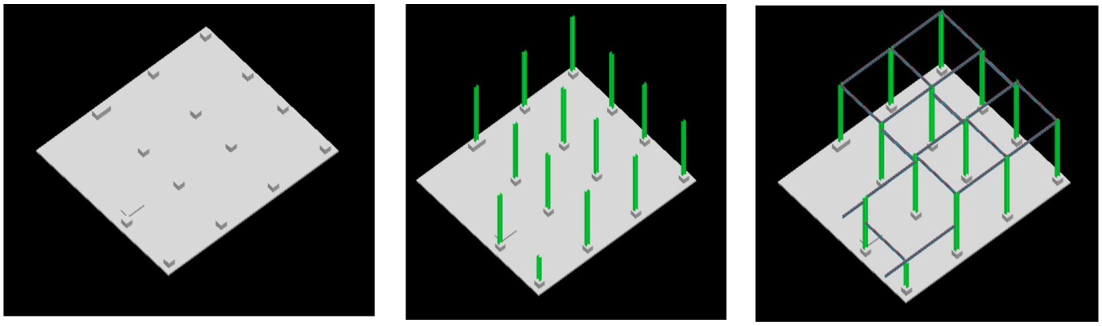


Figure 8-4: Timeliner Simulation Example

9 Quantification

In this chapter:

Setting up your workbook

Dragging items to the workbook

Taking off the entire model

Exporting the takeoff

9.1 Setting up your workbook

Quantification supports integration of three-dimensional (3D) and two-dimensional (2D) design data. You can combine multiple source files and generate quantity takeoff. Takeoff an entire building information model (BIM) and create synchronized project views that combine information from BIM tools such as Revit® and AutoCAD® software together with geometry, images, and data from other tools. You can also perform a virtual takeoff for items with no associated model geometry or properties.

Your takeoff data can then be exported to Excel for analysis and shared with other project team members in the cloud using Autodesk BIM 360® for optimized collaboration.

Quantification can count and measure item quantities associated with these disciplines:

- Civil (earth, road, drainage)
- Architecture (doors, walls, windows)
- Engineering (structural, mechanical, electrical,

plumbing) The available measurements (variables) are:

- Length
- Width
- Thickness
- Height
- Perimeter
- Area
- Volume
- Weight
- Count

9.1.1 Quantification workflow

A typical workflow begins with a design file created in Autodesk design applications, such as AutoCAD, AutoCAD Architecture, AutoCAD Civil 3D, and Revit. Before publishing a design file, an architect, engineer, or other designer, determines which features (model, layouts, layers, properties and so on) to include in the file. After determining the content, a designer publishes the drawing and shares the file with the Estimator. In some cases, designers scan or save drawings as image files and send them for estimation. When the files are received, the Quantification workflow begins.

1. In Navisworks, open a design data source file
2. Open the Quantification workbook
3. Set up a project
4. Create or select takeoff items
5. Hide unwanted items
6. Use measurement tools for items not in catalog (for virtual takeoff)
7. Organize takeoff items (change item order, create new items)
8. Edit formulas/parameters
9. Refresh model after changing data
10. Analyze and validate takeoff data
11. Output takeoff data to Excel XLSX format

9.2 Dragging items to the workbook

1. Open the Quantification window in the Home Tab.
2. Click and drag to the title bar at the top or side of the window.
3. Optional: to prevent a window from automatically docking while you drag it, hold down the CTRL key.

Note: The docking tool allows you to place windows in a specific relationship to the canvas areas.

9.3 Taking off the entire model

Virtual takeoff can be carried out when you want to add takeoff objects that are not linked to a model object or item, for example:

- where an object has geometry, but has no properties
- where an object does not have geometry, and has no properties

This could be the case if you did not save the properties from the original design application with your file, or that the object you want to takeoff does not exist in the model. In both cases, you can associate a viewpoint with the virtual takeoff object so you can navigate your way back to it during the takeoff process. Once your virtual object is taken off, you can begin to add properties by using formulas, which provides quantities for the object.

Note: You cannot refresh a virtual takeoff, since it is not linked to the model.

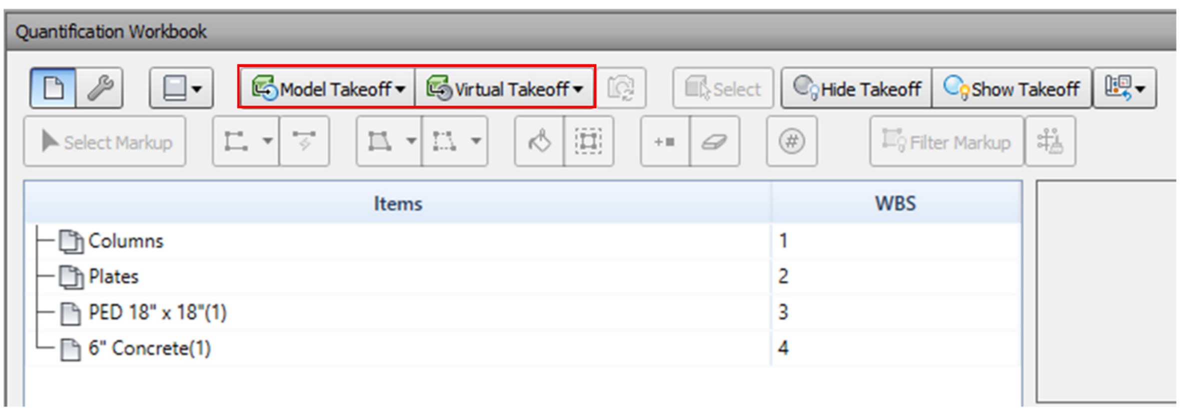


Figure 13-1: Quantification Take-off Tools

9.3.1 Takeoff objects with no properties

If you want to takeoff objects that have geometry in the model, but no associated properties, you can use a virtual takeoff. This may be the case if you did not save the object properties from the original design application, such as AutoCAD or Revit. In this instance, you can use measurement tools and formulas to add some property details to the Quantification workbook.
You can also associate a viewpoint with the takeoff object, to help reference its position in the model.

9.3.2 Takeoff unmodeled objects

You can capture the quantities of objects that do not appear in the model using a virtual takeoff. Virtual takeoff takes advantage of viewpoints, formulas, measurements, and manual overrides.

You can add a virtual takeoff to an existing Item, then associate a Viewpoint for that Item, which you can update and refer back to, make measurements and create redlines. The camera icon in the Viewpoint field denotes the associated Viewpoint. Click it to view the Item in the Scene.

9.3.3 Perform 2D takeoff

2D Takeoff allows you to measure lines, areas and counts on a 2D sheet. Rather than carrying out manual calculations on paper sheets, you can markup geometry and perform accurate calculations. The geometry is then taken off automatically, to sit alongside the 3D takeoffs in your Quantification workbook.

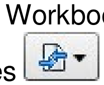
2D Takeoff supports native and scanned DWF files and non-native DWFs files such as PDF. See Supported file types.

2D Takeoff Workflow:

- Select or create an Item in the Items Catalog.
- Open the 2D sheet you want to use. If your project contains several sheets or models, use the Sheet Browser to select a 2D sheet.
- Select a Markup tool from the 2D Takeoff toolbar.
- Draw your markup on the sheet.
- Your Takeoff results appear in the Quantification workbook.

9.4 Exporting the takeoff

To export Quantification data

1. In the Quantification Workbook, click Import/Export Catalogs and Export Quantities 
2. From the drop-down list, click Export Quantities to Excel
3. Save the file to your preferred location

10 Presenting, Animating, and Exporting

In this chapter:

10.1 Animating saved viewpoints

There are two ways to create viewpoint animations in Autodesk Navisworks. You can either simply record your real-time walk through, or you can assemble specific viewpoints for Autodesk Navisworks to interpolate into a viewpoint animation later.

Viewpoint animation is controlled through the Animation tab and the Saved Viewpoints window.

It is worth remembering that you can hide items in viewpoints, override colors and transparencies and set multiple section planes and these will all be respected by a viewpoint animation. This way you can easily create powerful viewpoint animations.

Once a viewpoint animation is recorded, you can edit it to set the duration, the type of smoothing and whether it loops or not.

There is also nothing to stop you from copying viewpoint animations (hold down the CTRL key when dragging an animation on the Saved Viewpoints window), dragging frames off the animation into a blank space on the Saved Viewpoints window to remove them from the viewpoint animation, editing individual frames attributes, inserting cuts or dragging other viewpoints or viewpoint animations onto the existing one, to continue developing your animations.

10.1.1 Animation Cuts (Pauses)

Cuts in a viewpoint animation are simply points where the camera pauses for a while. They are inserted automatically when you click Pause during the interactive recording of a viewpoint animation, or you can insert them manually into an existing viewpoint animation.

10.1.2 To create a viewpoint animation in real time

1. Click Animation tab ➤ Create panel ➤ Record .
   To the far right of the Animation tab, notice that the Recording panel displays.
2. Navigate around in the Scene View while Autodesk Navisworks records your movement. You can even move the section planes through the model during your navigation, and this will be recorded into the viewpoint animation too.
3. At any point during the navigation, click Animation tab ➤ Recording panel ➤ Pause .
   This will pause the recording while you maneuver into a new position. To continue recording the viewpoint animation, click Pause again.
   The resulting viewpoint animation will contain a <u>cut</u> for the duration of the pause.
4. When finished, click Animation tab ➤ Recording panel ➤ Stop .
   The animation is saved automatically in the Saved Viewpoints window (click View tab ➤ Workspace panel ➤ Windows drop-down ➤ Saved Viewpoints). Your new viewpoint animation is called “AnimationX”, where “X” is the latest available number. The name will be editable at this point if you want to name it yourself. This viewpoint animation will also become the current active animation in the Available Animations drop-down list on the Playback panel of the Animation tab.

While the above method is useful for creating quick viewpoint animations on the fly, sometimes you need more control over the viewpoint camera. To do this in Autodesk Navisworks, you need to set up several viewpoints and add them to an empty viewpoint animation. When playing back the animation, Autodesk Navisworks will then interpolate between these viewpoints.

10.1.3 To create an animation frame by frame

1. If necessary, display the Saved Viewpoints window (click ➤ View tab ➤ Workspace panel ➤ Windows drop-down ➤ Saved Viewpoints).
2. Right-click the Saved Viewpoints window and select Add Animation.
   A new viewpoint animation is created, called “AnimationX”, where “X” is the latest available number. The name will be editable at this point if you want to name it yourself. There will be no plus sign next to the new viewpoint animation because it is empty.
3. Navigate to a position in the model you would like to add to your animation and save the new location as a viewpoint. (Right-click the Saved Viewpoints window and select Save Viewpoint.) Repeat this step as desired. Each viewpoint will become a frame for the animation. The more frames you have, the smoother and more predictable the viewpoint animation will be. See Save Viewpoints for more information on creating viewpoints.
4. When you have all required viewpoints, drag them onto the empty viewpoint animation you just created. You can drag them on one-by-one, or you can select multiple viewpoints using the CTRL and SHIFT keyboard keys and drag several on at once.
   If you drop them onto the viewpoint animation icon itself, then the viewpoints will become frames at the end of the animation, but you can drop the viewpoints anywhere on the expanded animation to put them where you wish.
5. At this point, you can use the Playback Position slider on the Playback panel of the Animation tab to move backward and forward through the viewpoint animation to see how it looks.
6. You can edit any of the viewpoints inside the viewpoint animation (see Edit Viewpoints for details on this), or you can add more viewpoints, delete them, move them around, add cuts, and edit the animation itself until you are happy with the viewpoint animation.

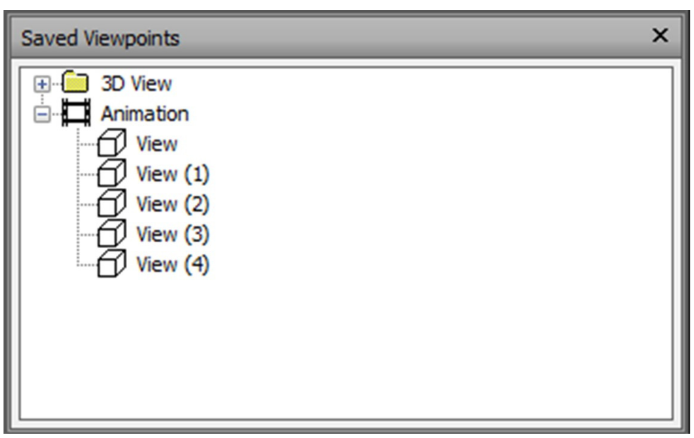


Figure 10-1: Animation and Related Viewpoints

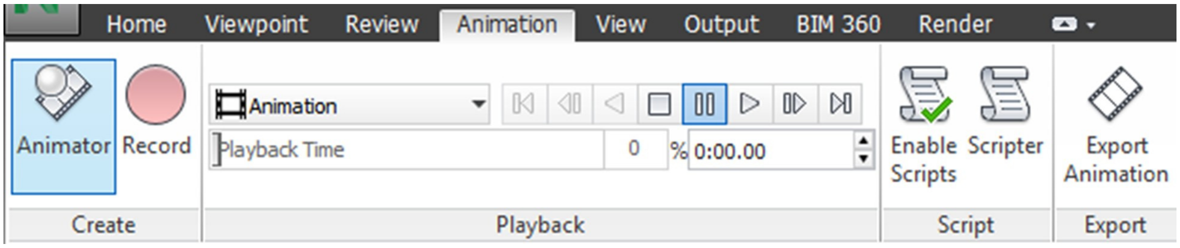


Figure 10-2: Playback Panel

Once you have several viewpoint animations, you can drag and drop them onto a master viewpoint animation to compose more complex combinations of animations, just like dragging and dropping viewpoints onto an animation as a frame.

10.2 Animating objects

Animation tab ➤ Create panel ➤ Animator

With the Animator, scenes can be created. Under those scenes, animations can be established. A scene could be a door and the animation would be that door opening and closing. To create an animation for a scene, select the object or set of objects to be animated. Attach them to the animation set by right-clicking the name of that animation set and selecting "Add Animation Set" and "From Current Selection". Then use the following tools to create the animation itself. To control time in the animation, use the right side of the Animator dialog box.

- Puts Animator into translation mode. The Translation gizmo is displayed in the Scene View, and enables you to modify the position of the geometry objects. This mode remains active until you select a different object manipulation mode from the toolbar
- Puts Animator into rotation mode. The Rotation gizmo is displayed in the Scene View, and enables you to modify the rotation of the geometry objects. This mode remains active until you select a different object manipulation mode from the toolbar.
- Puts Animator into scale mode. The Scaling gizmo is displayed in the Scene View, and enables you to modify the size of the geometry objects. This mode remains active until you select a different object manipulation mode from the toolbar.
- Puts Animator into color mode. A color palette is shown in the Manual Entry bar, and enables you to modify the color of the geometry objects.
- Puts Animator into transparency mode. A transparency slider is shown in the Manual Entry bar, and enables you to modify the transparency of the geometry object
- Takes a snapshot of the current change to the model as a new keyframe in the timeline view.

- Enables/disables snapping. Snapping only comes into effect when moving objects by dragging the gizmos in the Scene View, and has no effect on numerical entry or keyboard control.
- Scene 1 Selects the active scene.
- 0:10.00 Controls the current position of the time slider in the timeline view.
- Rewinds the animation back to the beginning.
- Rewinds one second.
- Plays the animation backward from the end point to the start, and then stops. This does not alter the direction that the animated elements face.
- Pauses the animation. To continue playing, click Play again.
- Stops and rewinds the animation to the beginning.
- Plays the animation forward from the starting point to the end.
- Plays the animation forward one second.
- Fast forwards the animation to the end.

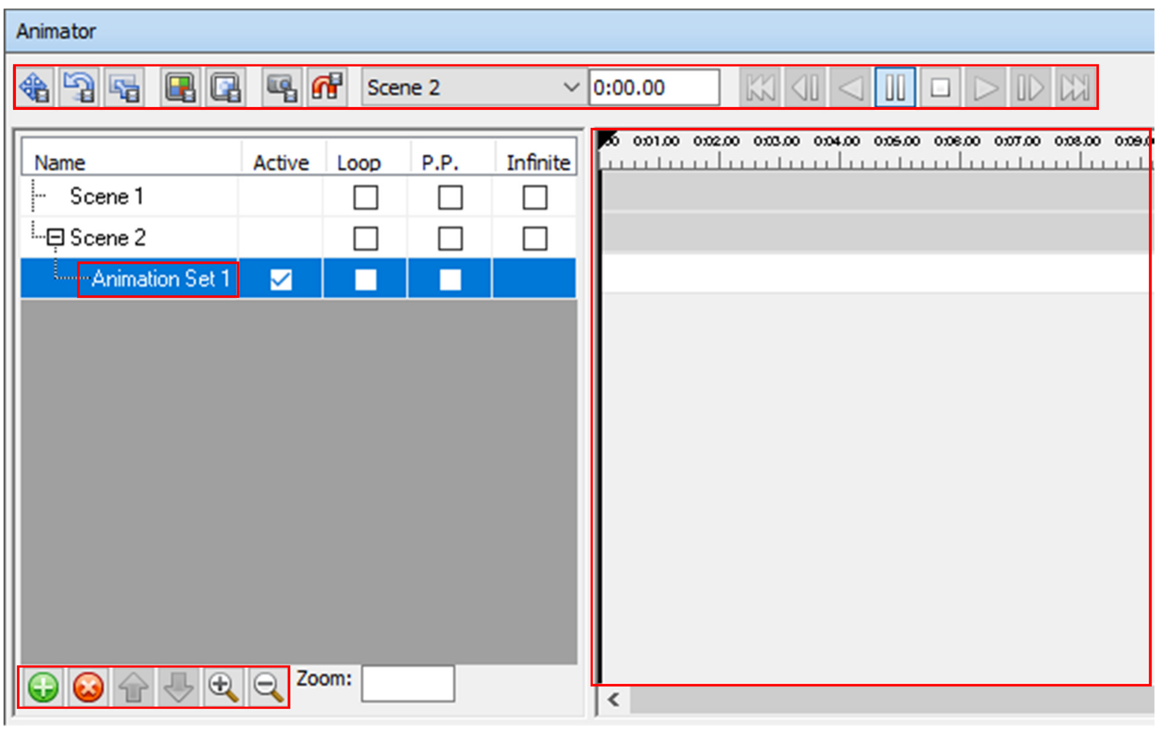


Figure 14-3: Animator Dialog Box

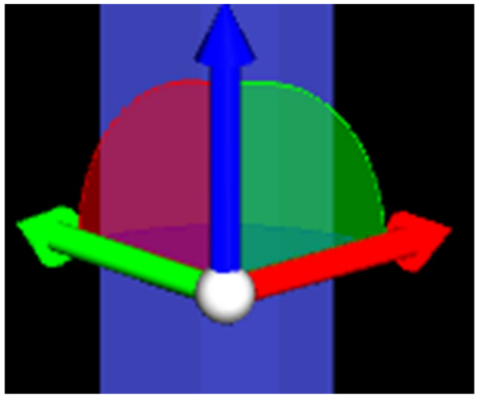


Figure 14-4: Example Animator Rotation Gizmo

10.3 Creating a script

Click Animation tab ➤ Scripts panel ➤ Scripter .

Add a script . Choose how it will exist in the model by adding events
A script will create a trigger for an animation. Scripts can be tied to hot-keys, hotspots in the model, and more. If triggered, the animation associated with it will play.

A script can be modified in the right side of the Scripter Dialog Box under

Properties. Click Animation tab ➤ Scripts panel Enable Scripts .

This will turn on scripts and allow a user to interact with the model. For example, if there is a script to open a door associated with pressing a specific key on the keyboard, pressing this key will open the door.

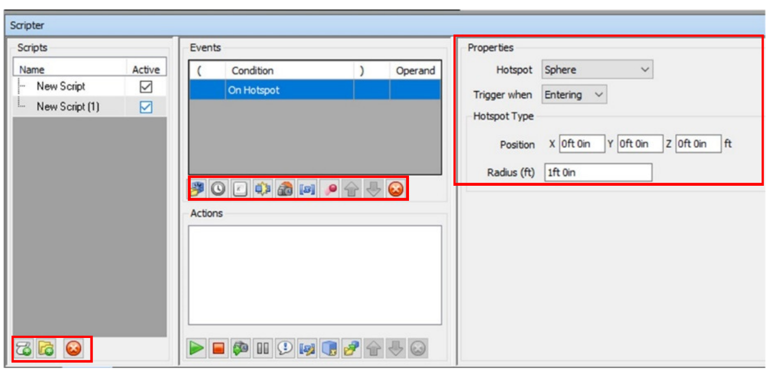


Figure 14-5: Scripter Dialog Box

11. Special topic: Rebar and Steel Connections Collaborations

3D reinforcement (rebar) can be modelled in Revit and then export to Navisworks for clash detective.

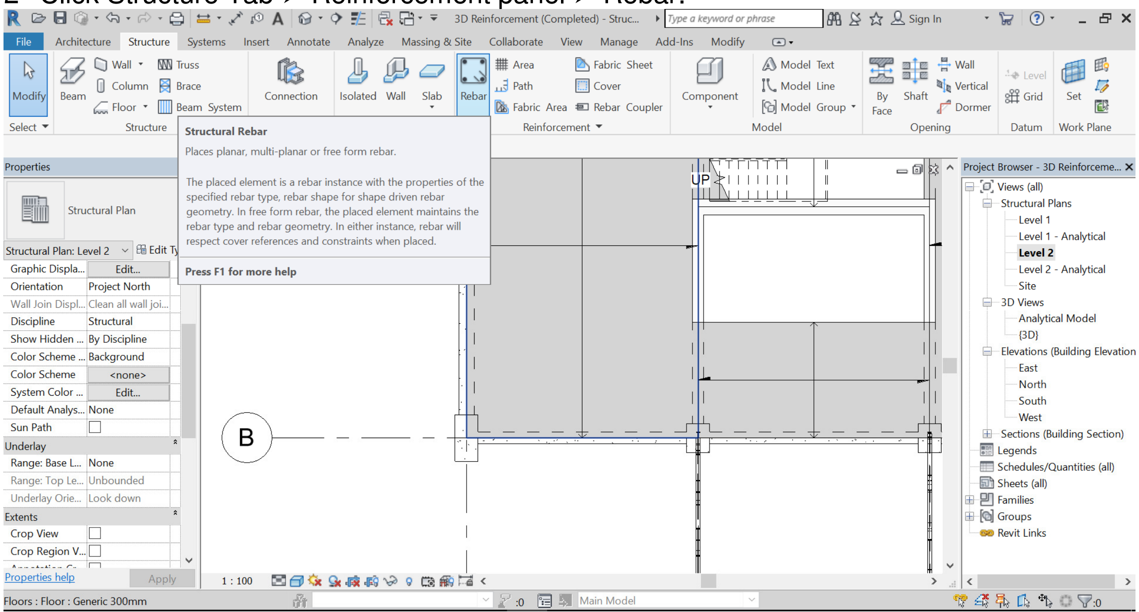
Reinforcement must be hosted by a valid host such as structural wall, column and floor. In this section, you will learn

- How to place reinforcement for Columns.
- How to place area reinforcement for Floors.
- How to control visibility of rebars.
- Various reinforcement settings in Revit.
- Export the model into Navisworks and check clashing

11.1 Modelling Reinforcement for Columns

1 Open from the course folder the “3D Reinforcement.rvt” file and go to Level 2 (Structural Plan).

2 Click Structure Tab ➢ Reinforcement panel ➢ Rebar.

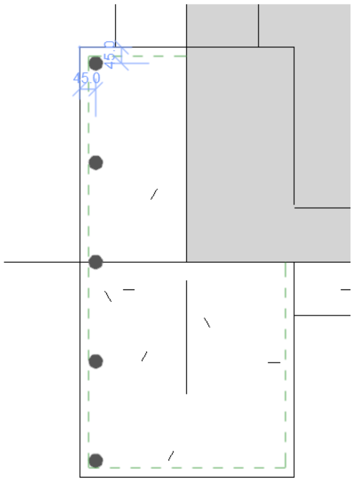


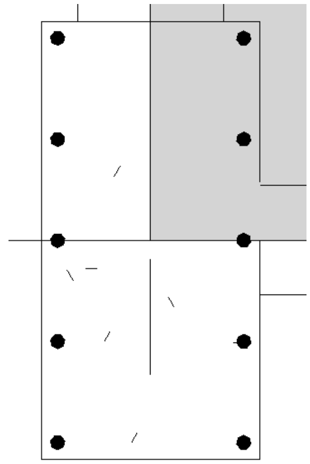
3 First, place the main (longitudinal) rebars with the following settings:

- Use default 40R rebar
- Placement Plane: Near Cover Reference
- Placement Orientation: Perpendicular to Cover
- Rebar Set: (Layout: Fixed Number and Quantity: 5)

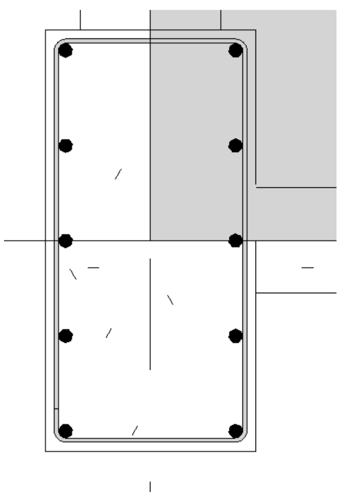
Hover over to the left side of the column at Grid-B-1 and place the rebars. Revit will show you boundary of the cover in green dotted lines.

4 Select the rebars just created and mirror (type “MM”) them to the other side of the column.

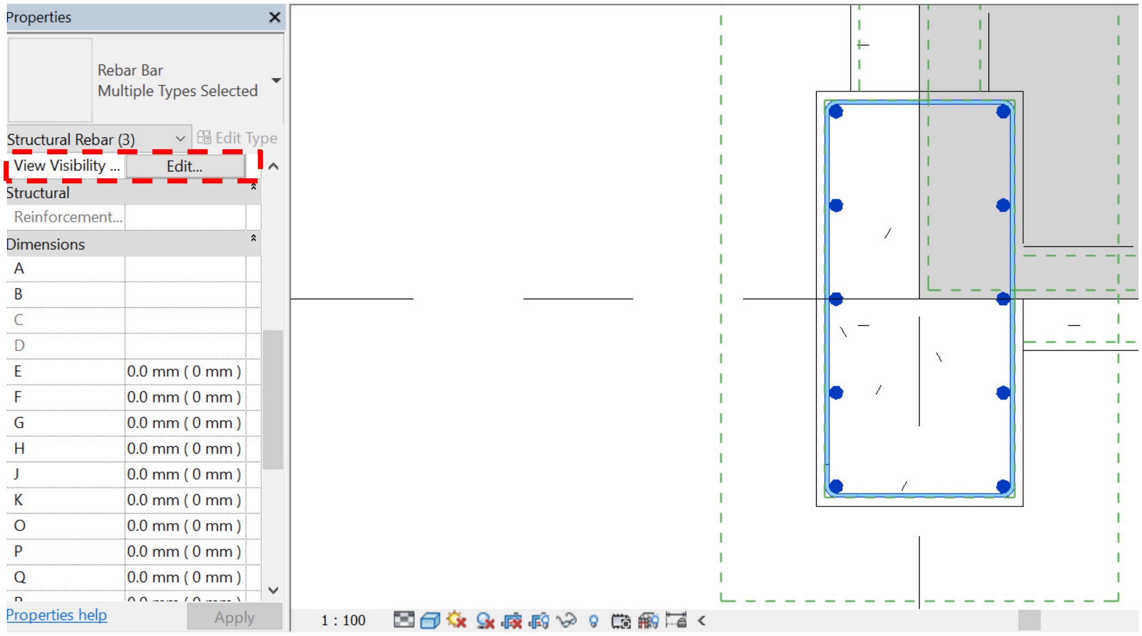


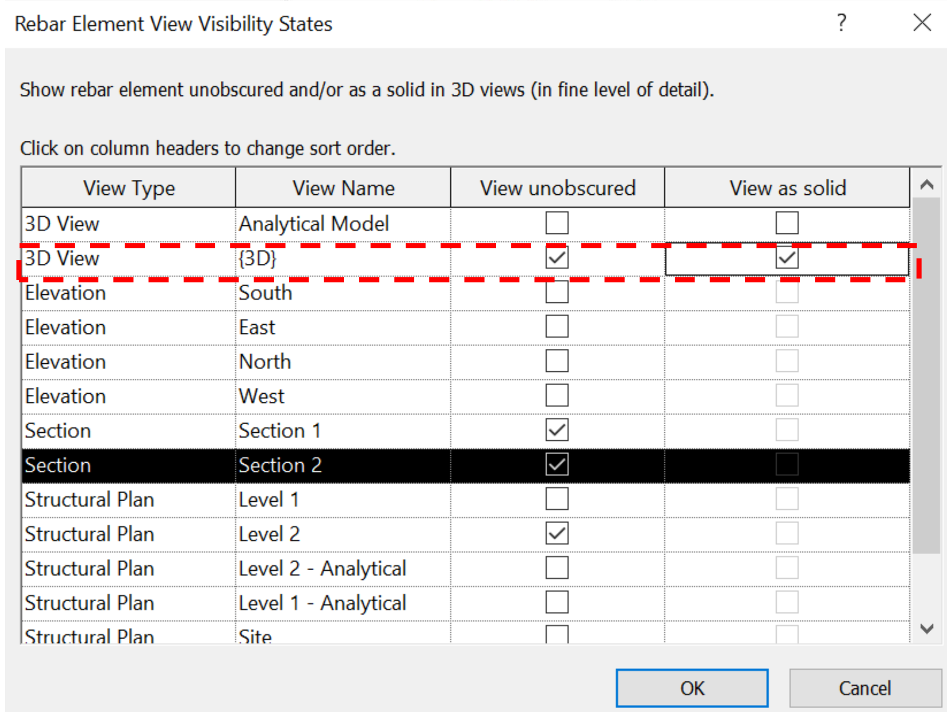


5 Next, place the links to the column. Click Structure Tab ➢ Reinforcement panel ➢ Rebar.
6 Select
- Rebar Shape: 51
- 12T rebar
- Placement Plane: Near Cover Reference
- Placement Orientation: Parallel to Cover
- Rebar Set: (Layout: Maximum Spacing and Spacing: 300mm)

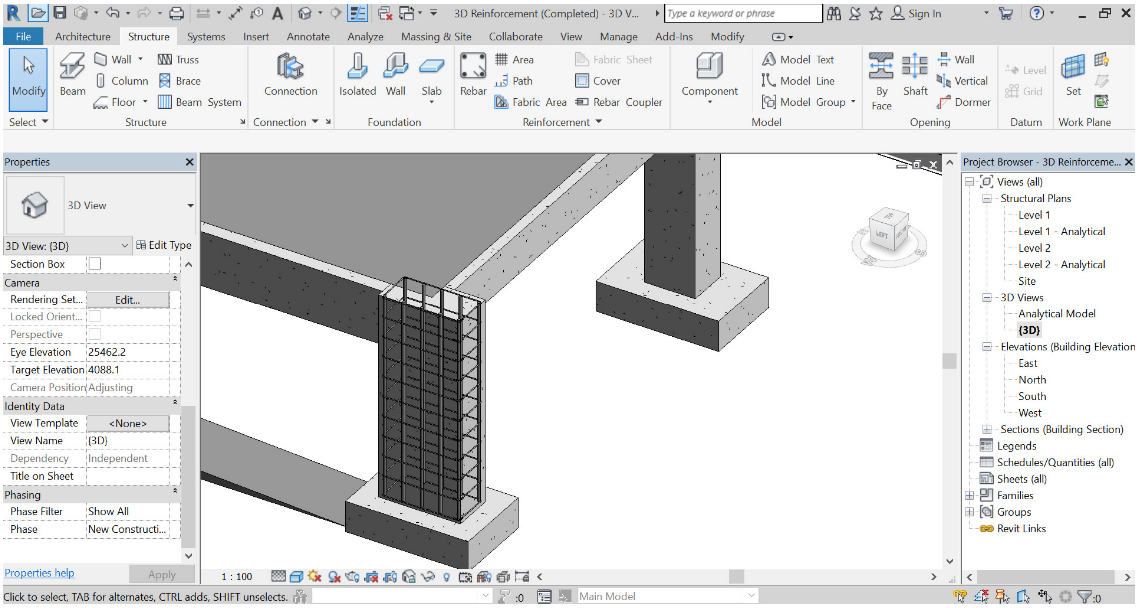


7 Now change the view visibility of these rebars. Select all the rebars and click “Edit” in View Visibility from the Properties Palette and make the rebars “View unobscured” and “View as solid” in 3D View {3D}. Then click OK.

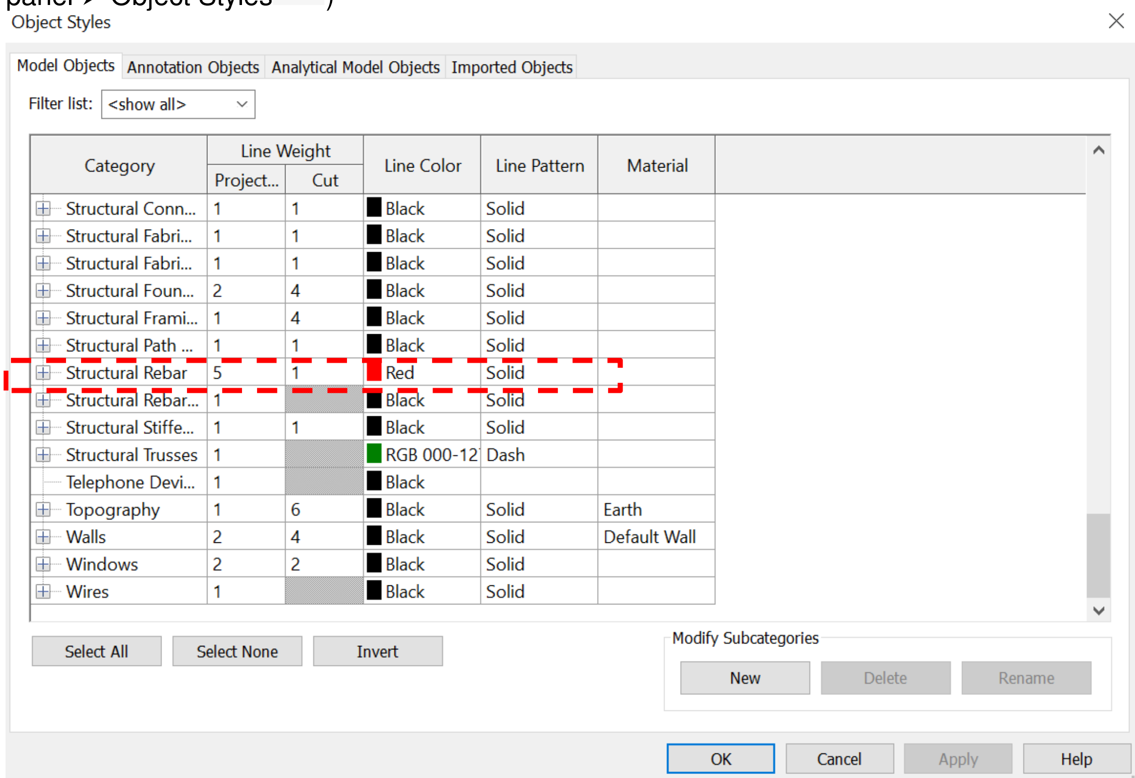


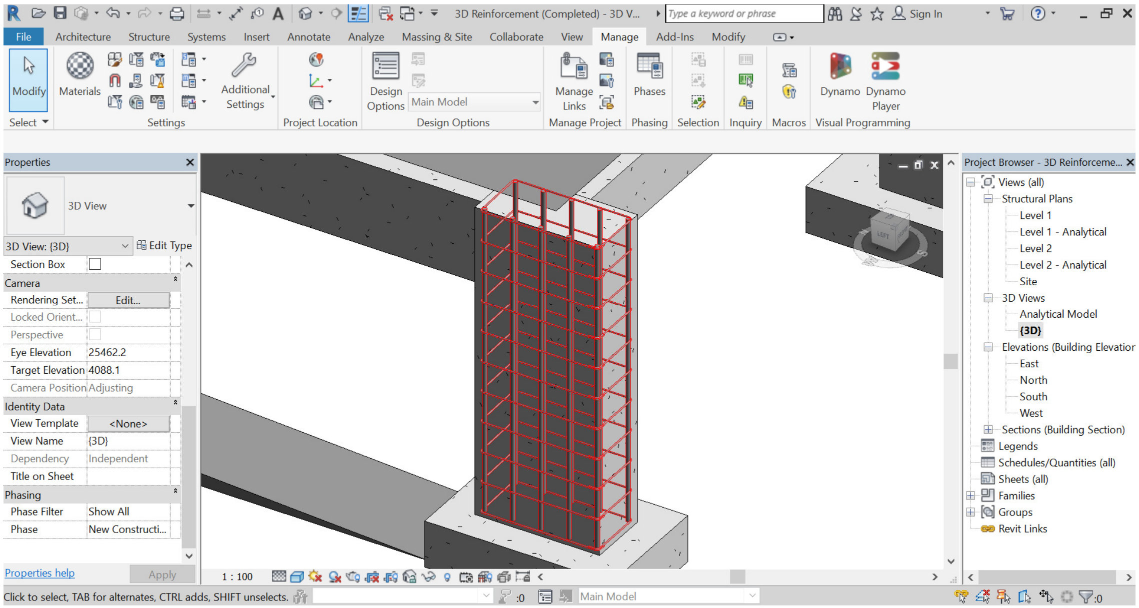


8 Go to 3D view and the rebars can be seen unobscured now.

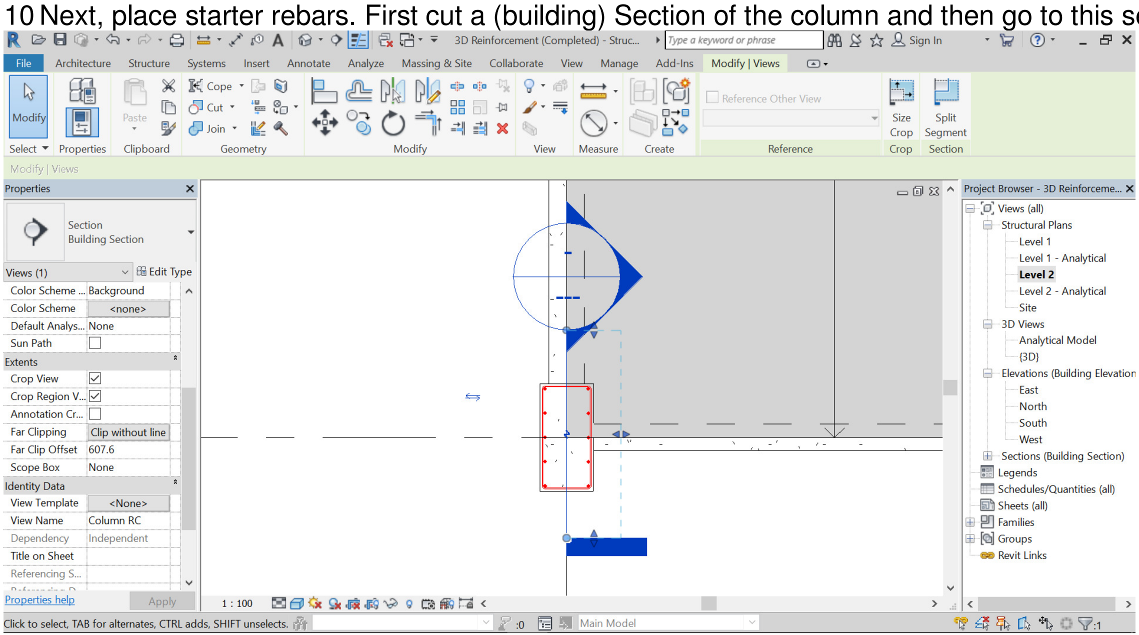


9 You can set the Structural Rebar Object Style and make it easier to view. (Click Manage Tab ➢ Settings panel ➢ Object Styles)





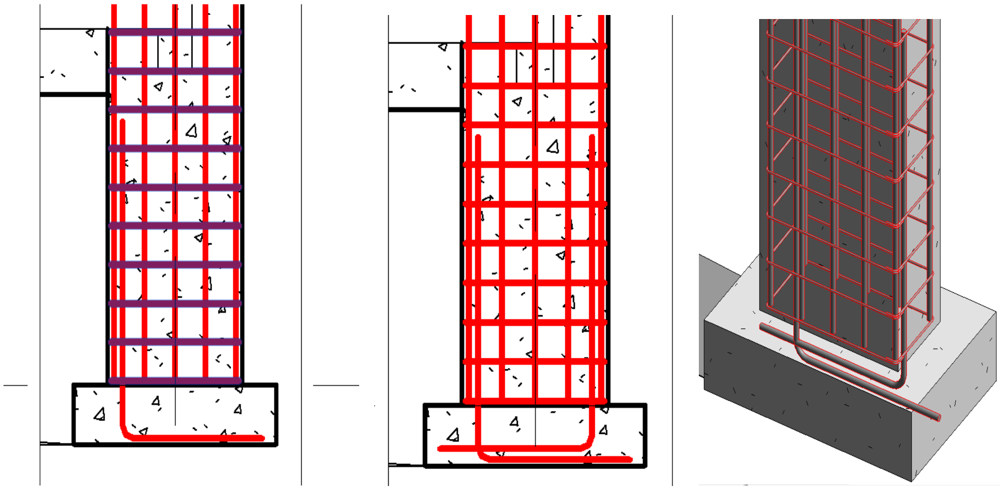
10 Next, place starter rebars. First cut a (building) Section of the column and then go to this section view.



11 Click Structure Tab ➢ Reinforcement panel ➢ Rebar. Select

- Rebar Shape: 00
- Placement Methods: Sketch Rebar and select the footing
- 40R rebar
- Placement Plane: Near Cover Reference
- Placement Orientation: Parallel to Cover
- Rebar Set: (Layout: Maximum Spacing and Spacing: 300mm)

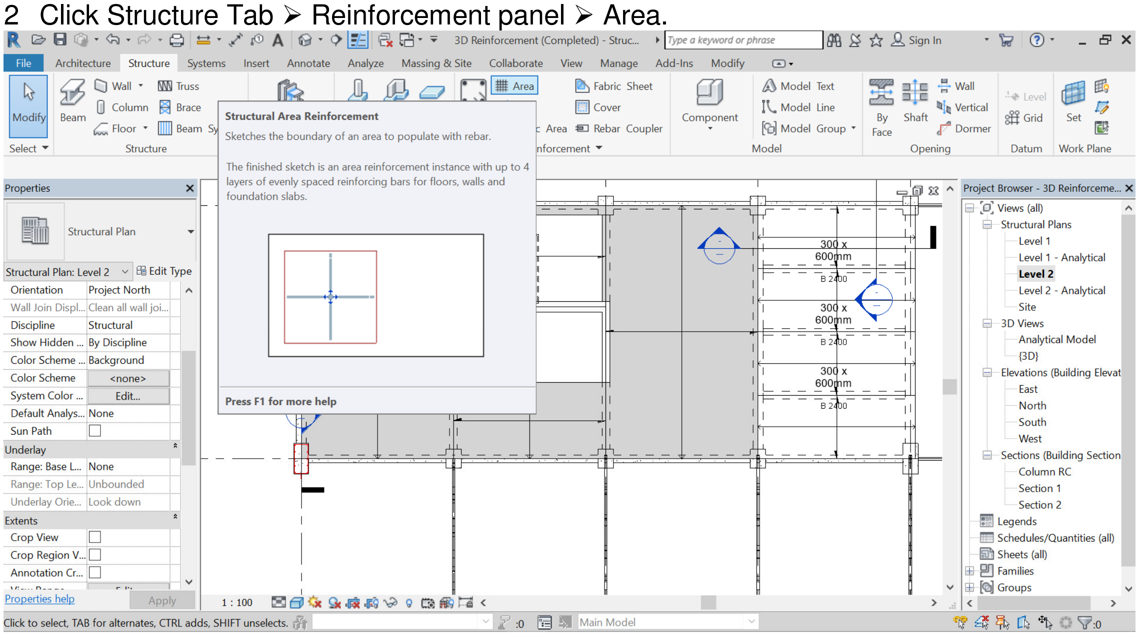
Sketch the starter rebar as shown below. Then mirror this starter rebar to the opposite face and move it slightly upward.



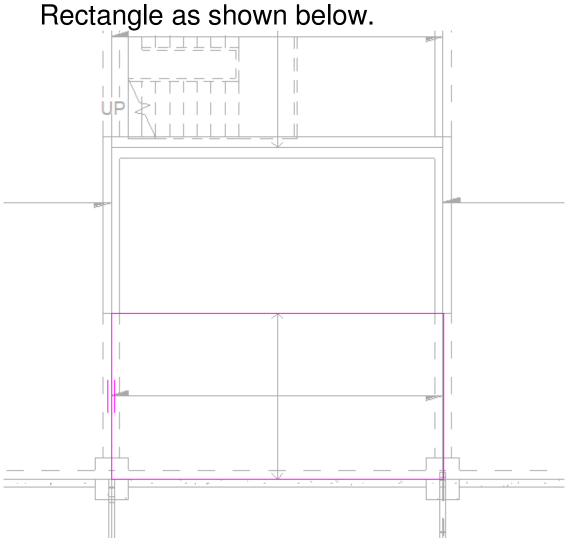
11.2 Placing Area Reinforcement for Floors (Slabs)

1 Go to Level 2 (Structural Plan).

2 Click Structure Tab ➢ Reinforcement panel ➢ Area.



3 Select the slab below the corewalls on plan. Then sketch the area of the reinforcement on plan using Rectangle as shown below.

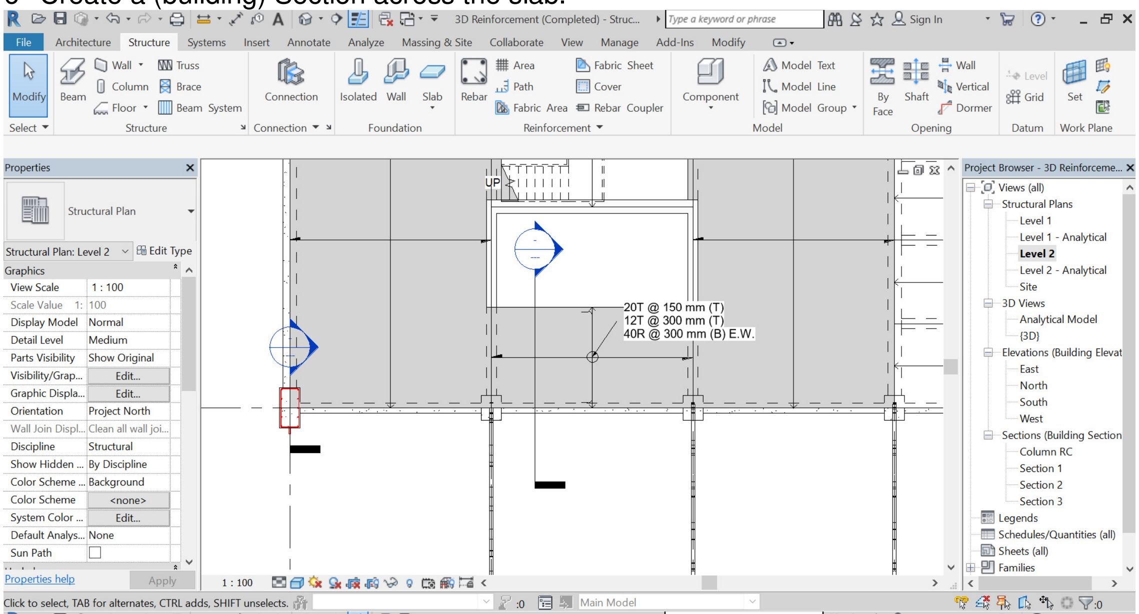


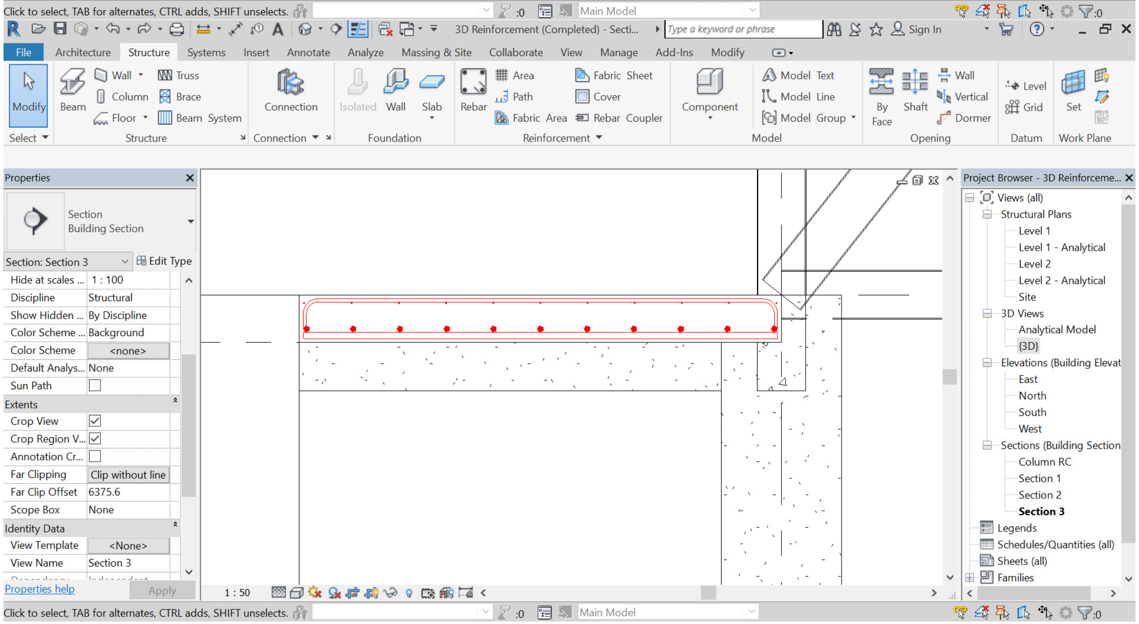
4 Change the area reinforcement properties as follow.

Layers	
Top Major Dir...	☑
Top Major Bar...	20T
Top Major Ho...	Standard Hook...
Top Major Ho...	Down
Top Major Sp...	150.0 mm
Top Major Nu...	2
Top Minor Dir...	☑
Top Minor Bar...	12T
Top Minor Ho...	None
Top Minor Ho...	Down
Top Minor Sp...	300.0 mm

5 Click Finish Edit Mode to add the area reinforcement.

6 Create a (building) Section across the slab.

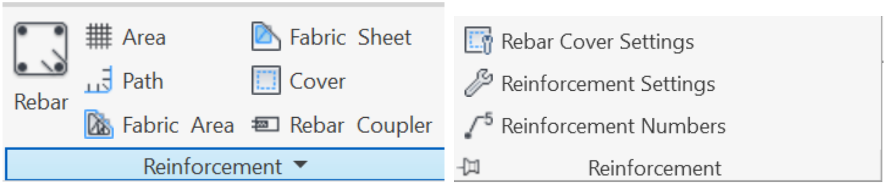


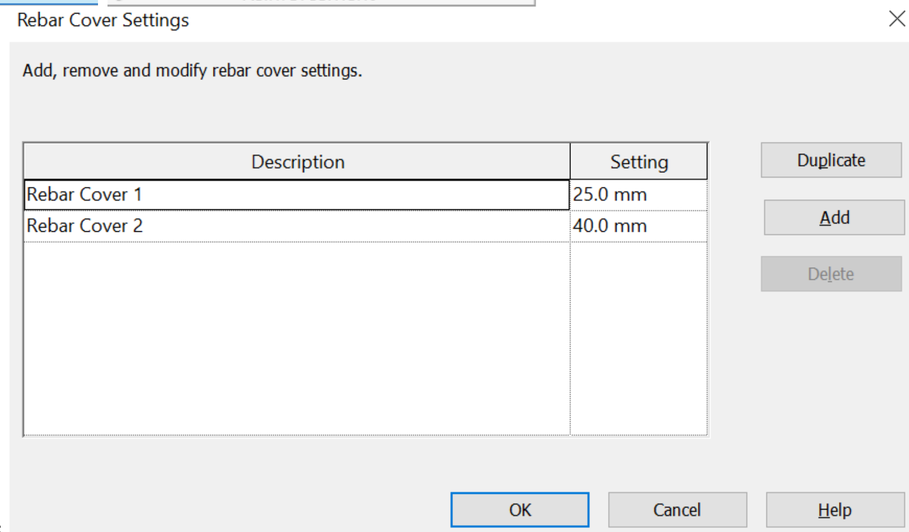


11.3 Reinforcement Settings

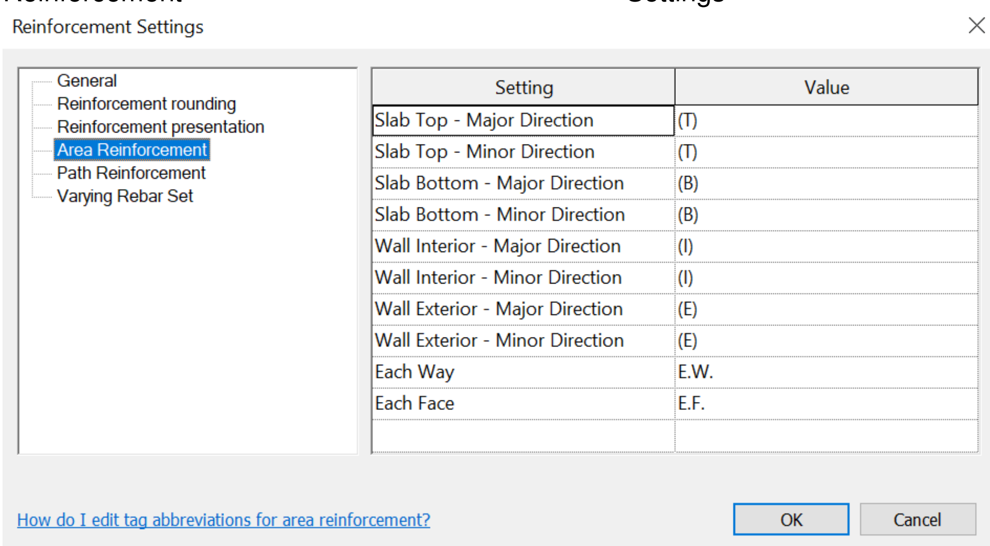
There are many parameters to control the reinforcement in Revit. To check/revise the setting, Page 72

Click Structure Tab ➢ Reinforcement panel

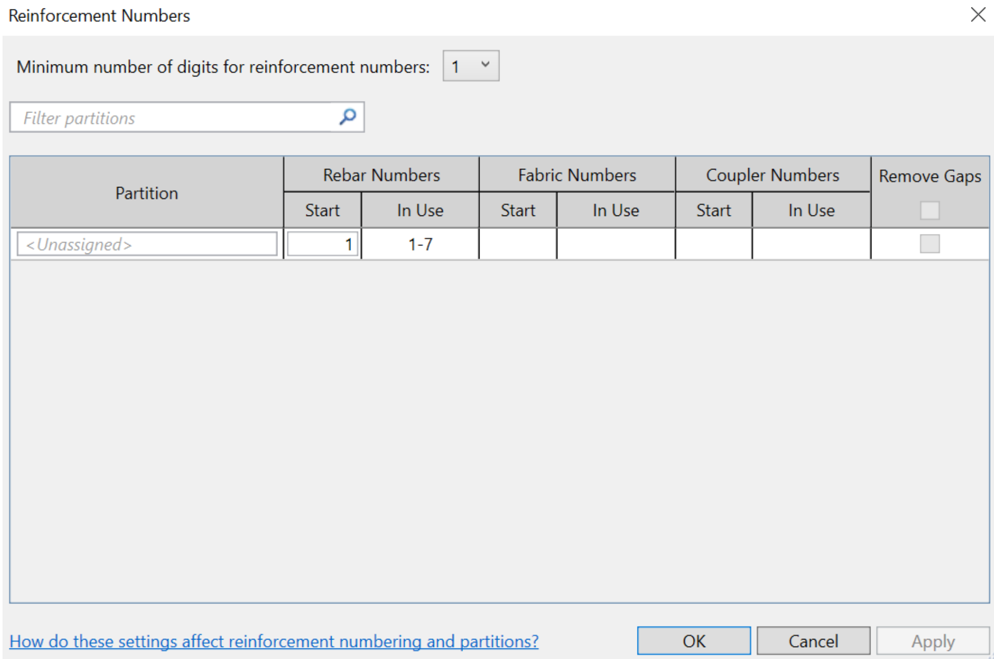




- Rebar Cover Settings
- Reinforcement Settings (e.g.



- Reinforcement Numbering

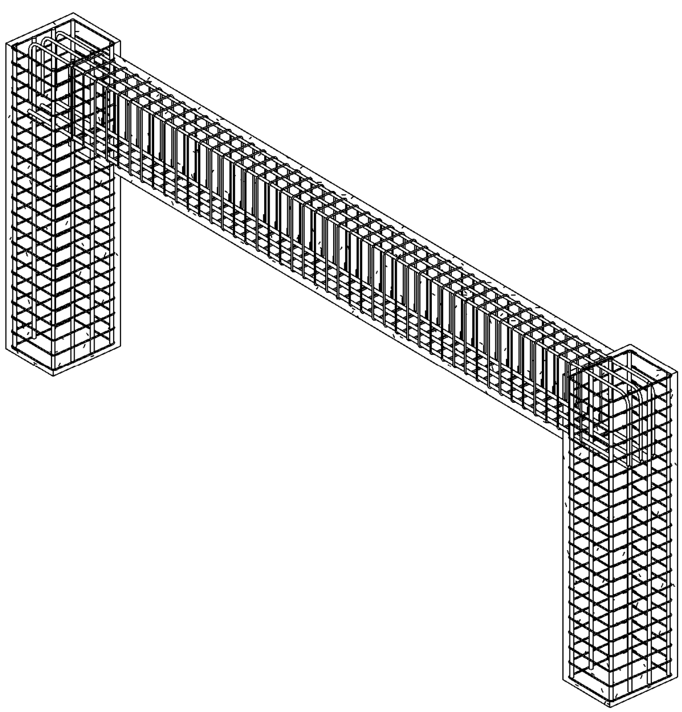


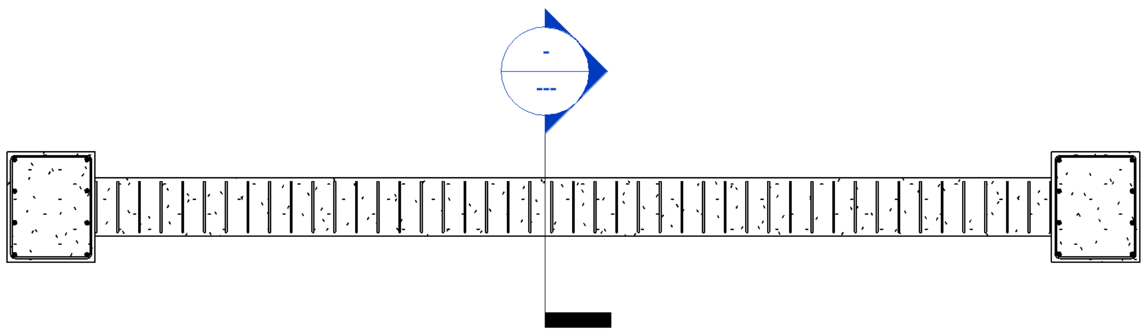
Exporting the Reinforcement model to Navisworks.

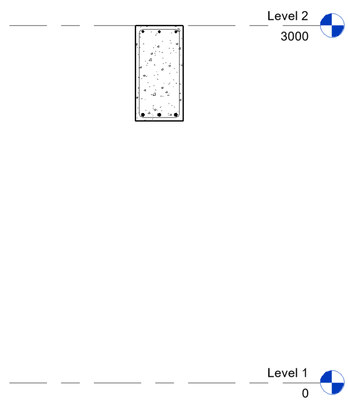
Open model “Rebar Model.rvt”. Create the model with following reinforcement.

Column: T10 -150 Links, 8 x T32 Longitude bars.

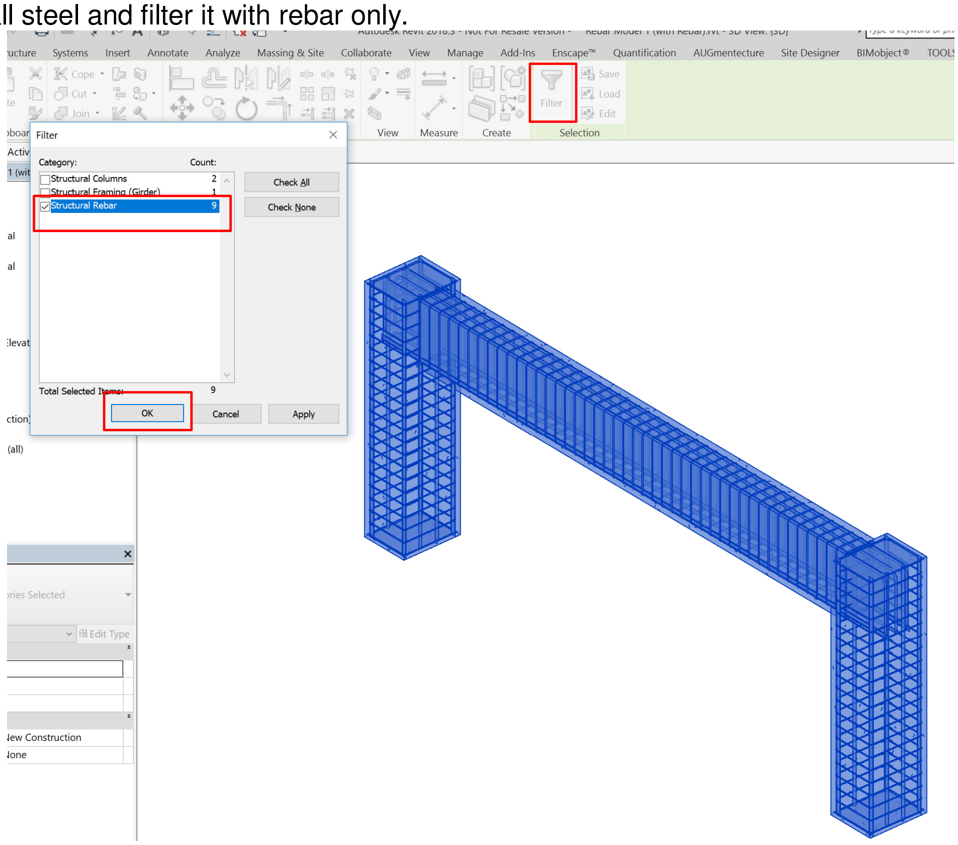
Beam: T12-150 Links, 3T25 Top, 3T32 Bottom.



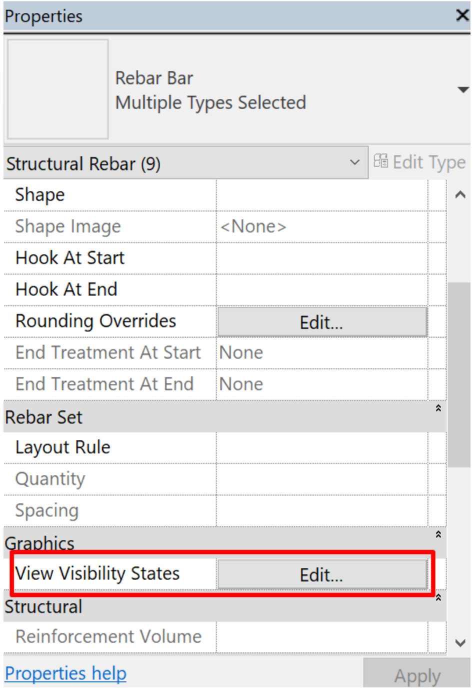


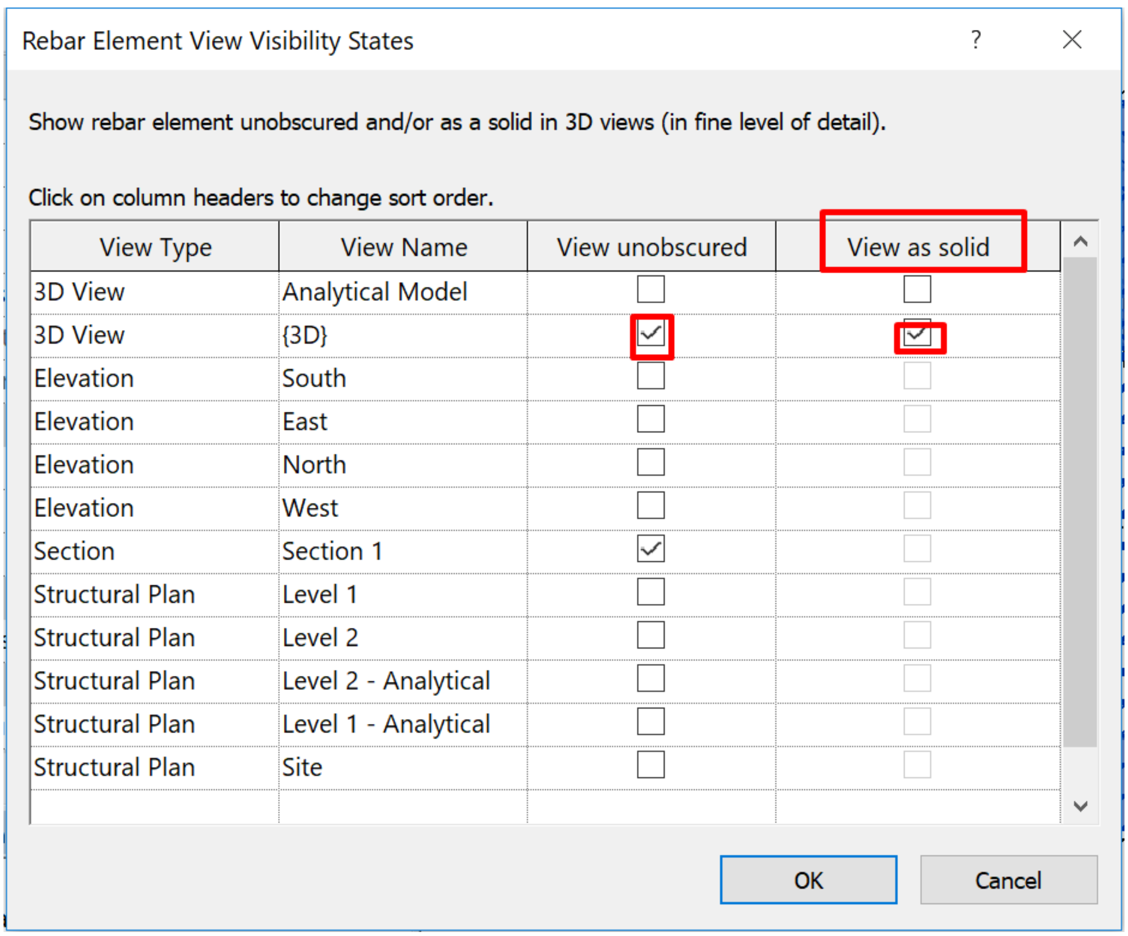


Highlight the all steel and filter it with rebar only.

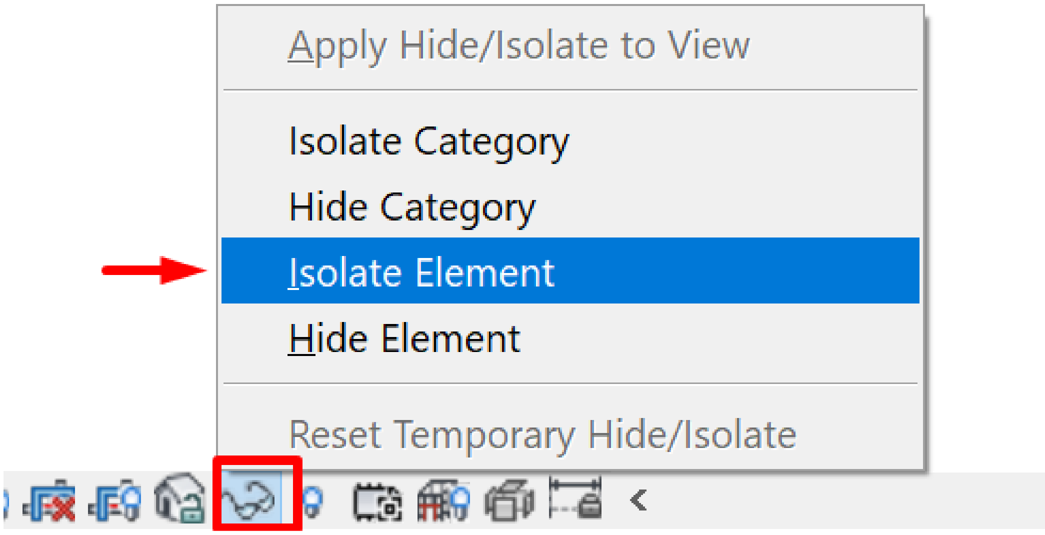


At properties, make sure to change the visibility states of bar to “solid”.

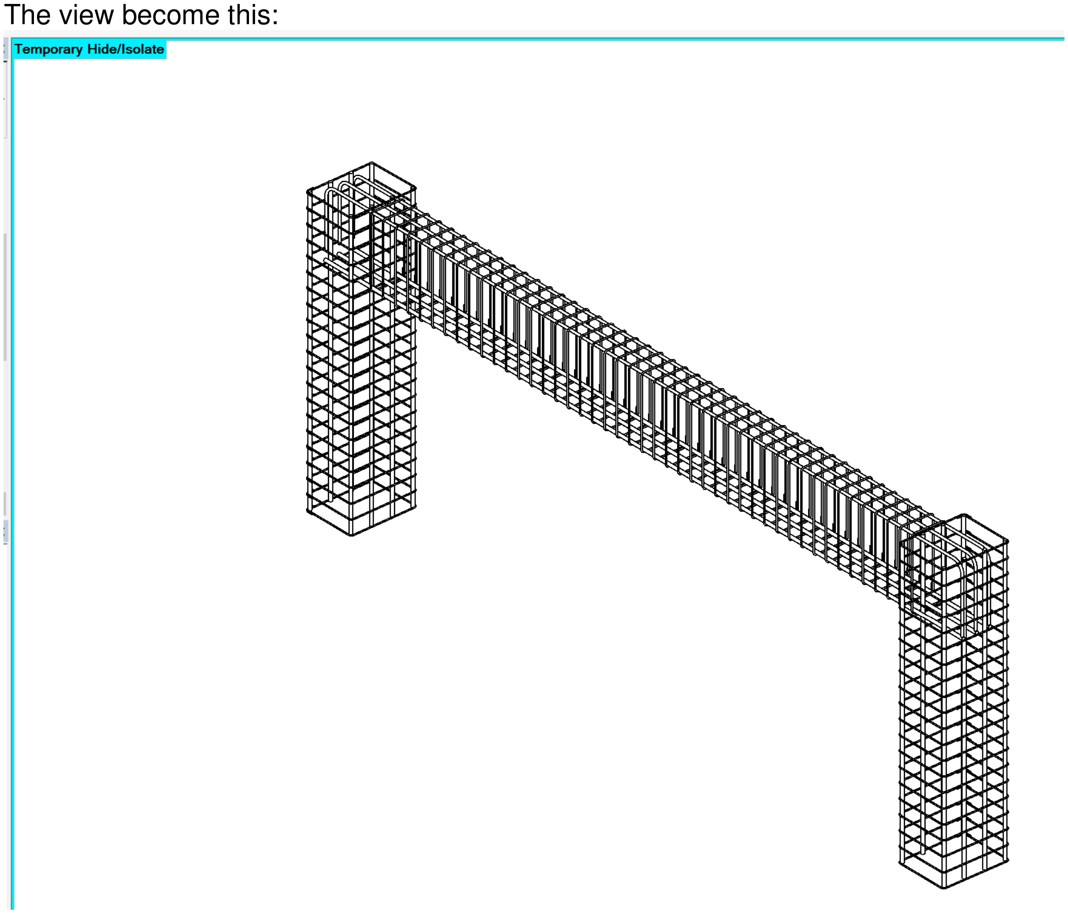




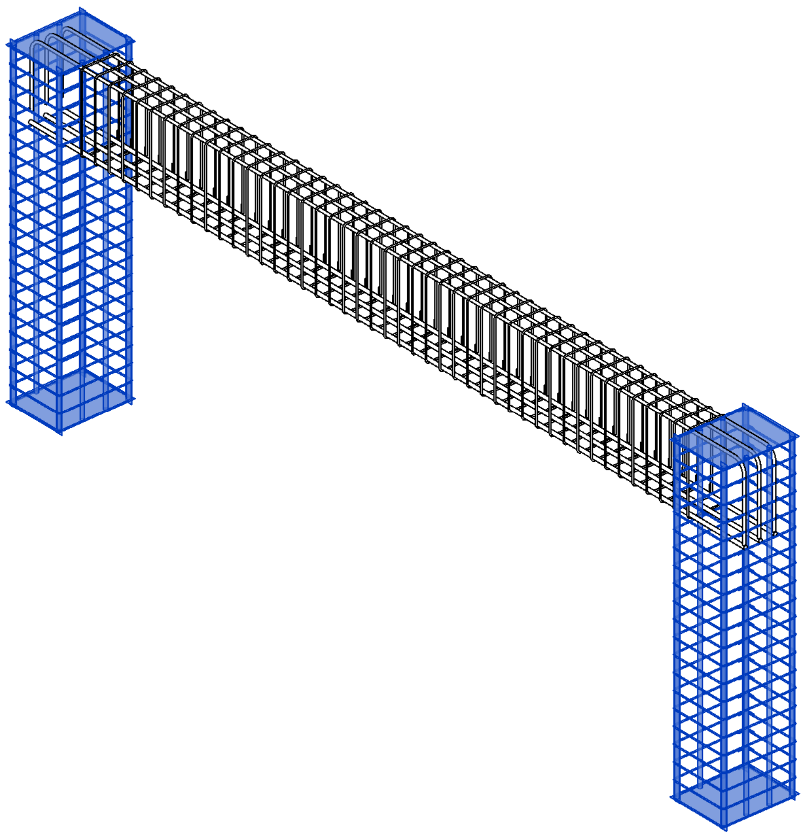
Isolate the rebar with the glass icon at bottom.



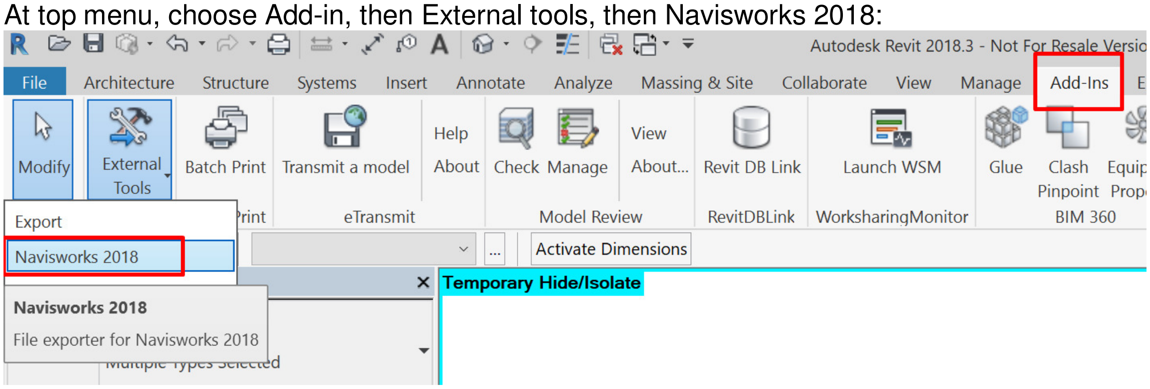
The view become this:



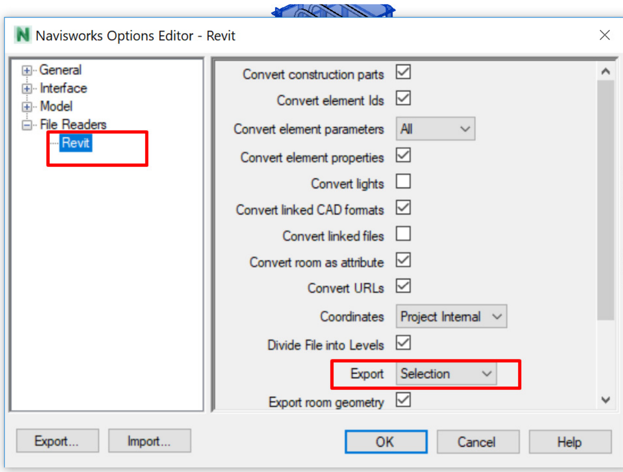
Highlight the column steel only:

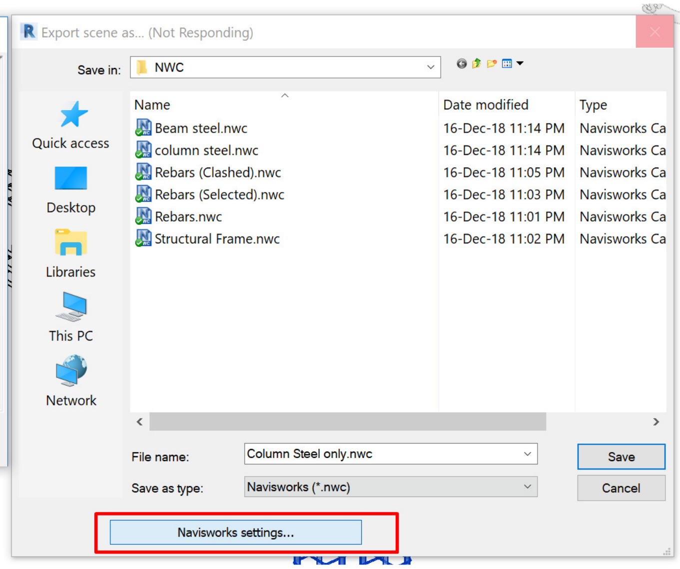


At top menu, choose Add-in, then External tools, then Navisworks 2018:



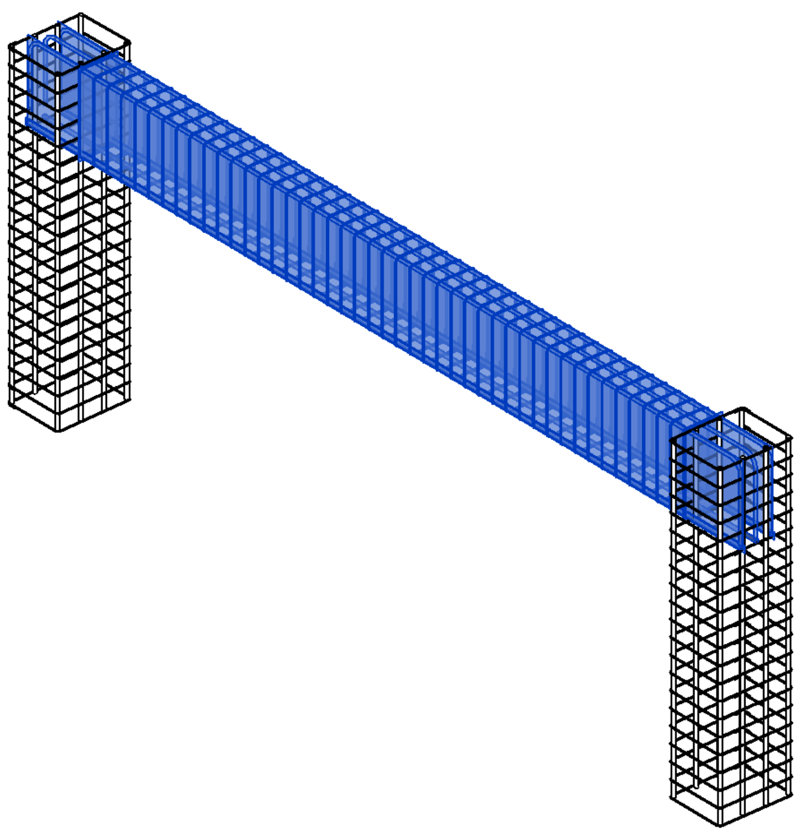
Choose Navisworks settings → Revit → Export to “Selection” as shown.





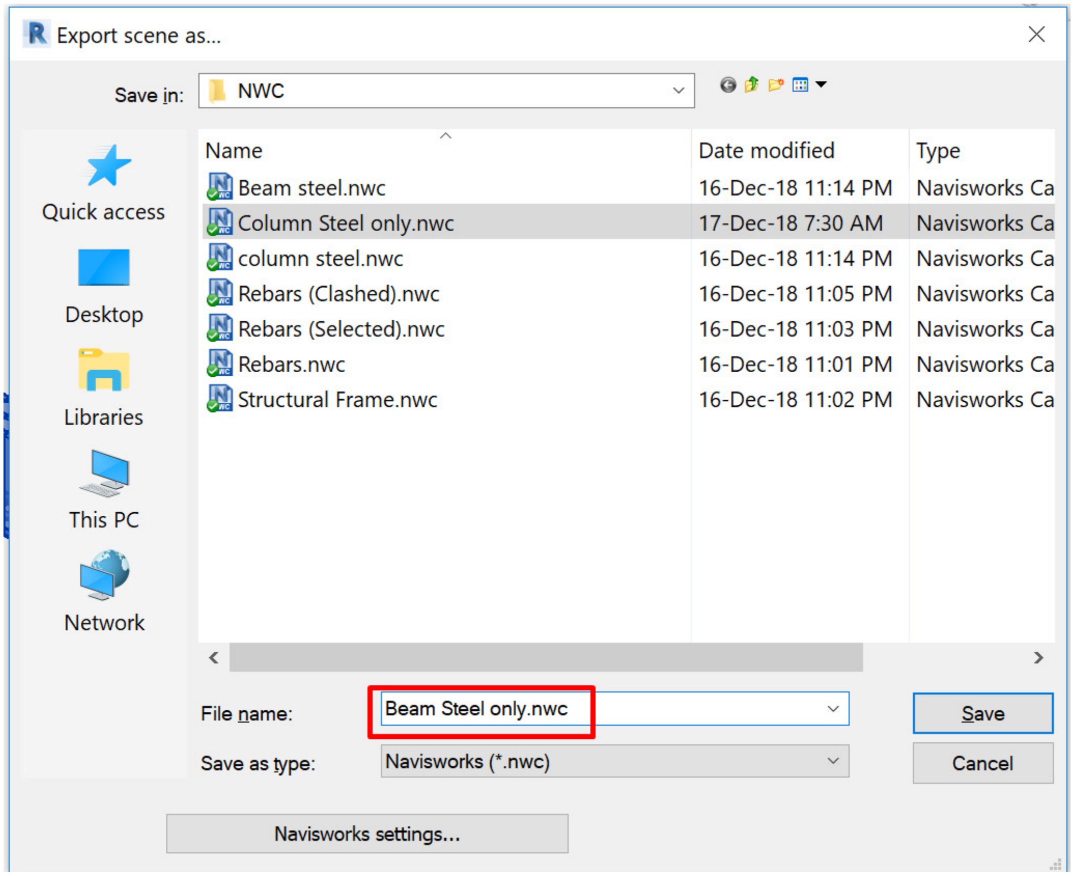
Save the file as " Column Steel only.nwc"

Repeat the same procedure for beams:

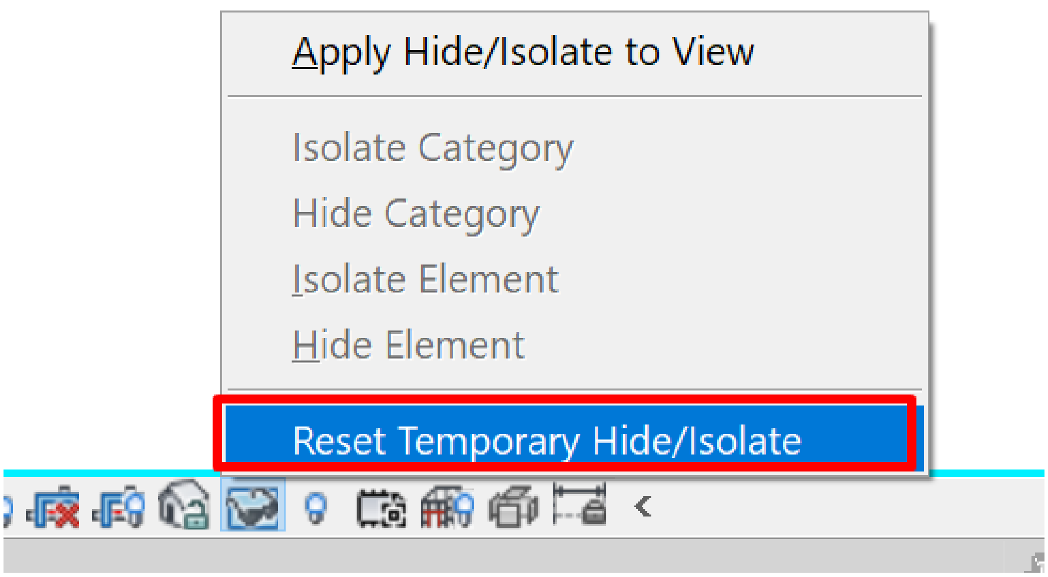


Export to Navisworks 2018.

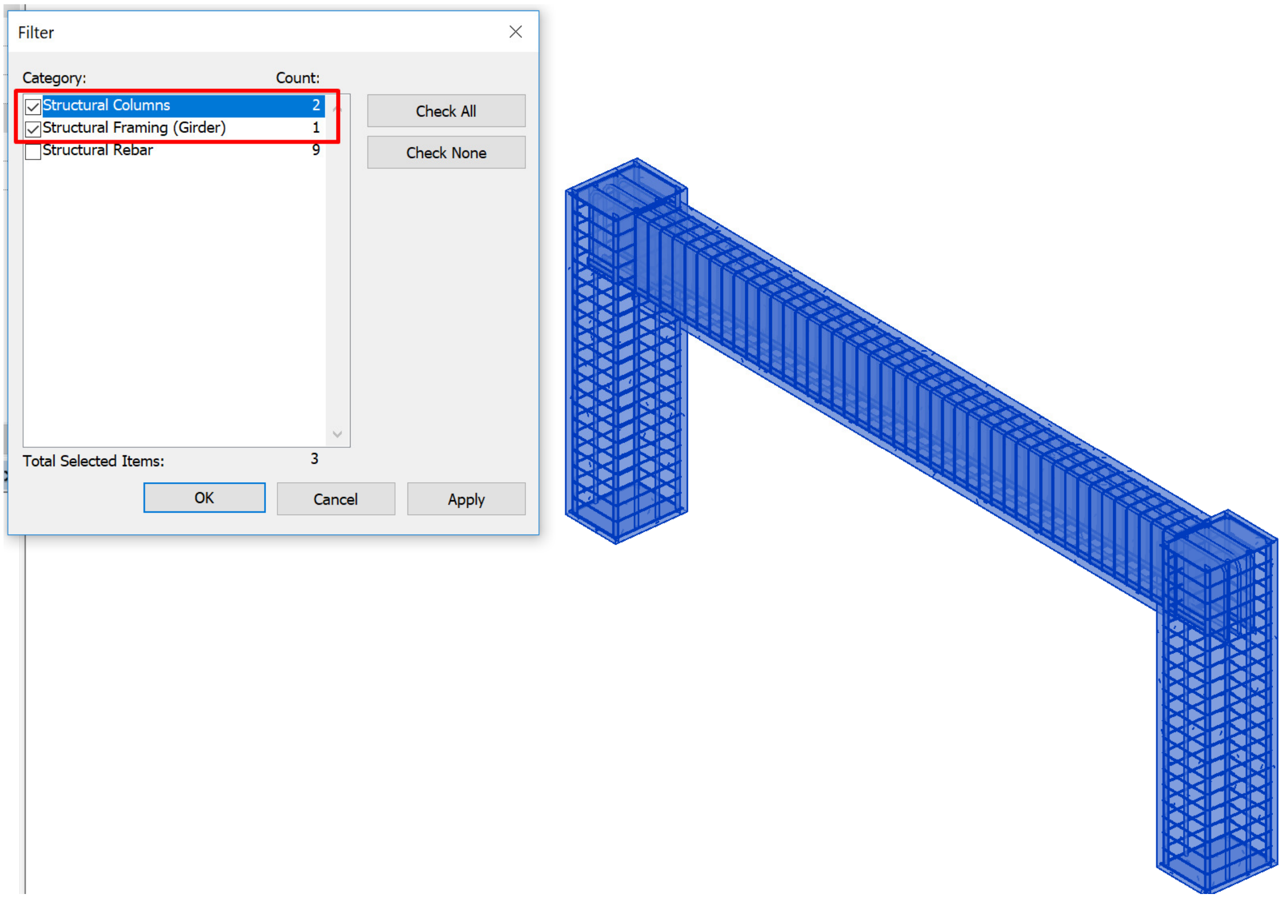
Save as "Beam Steel only.nwc"



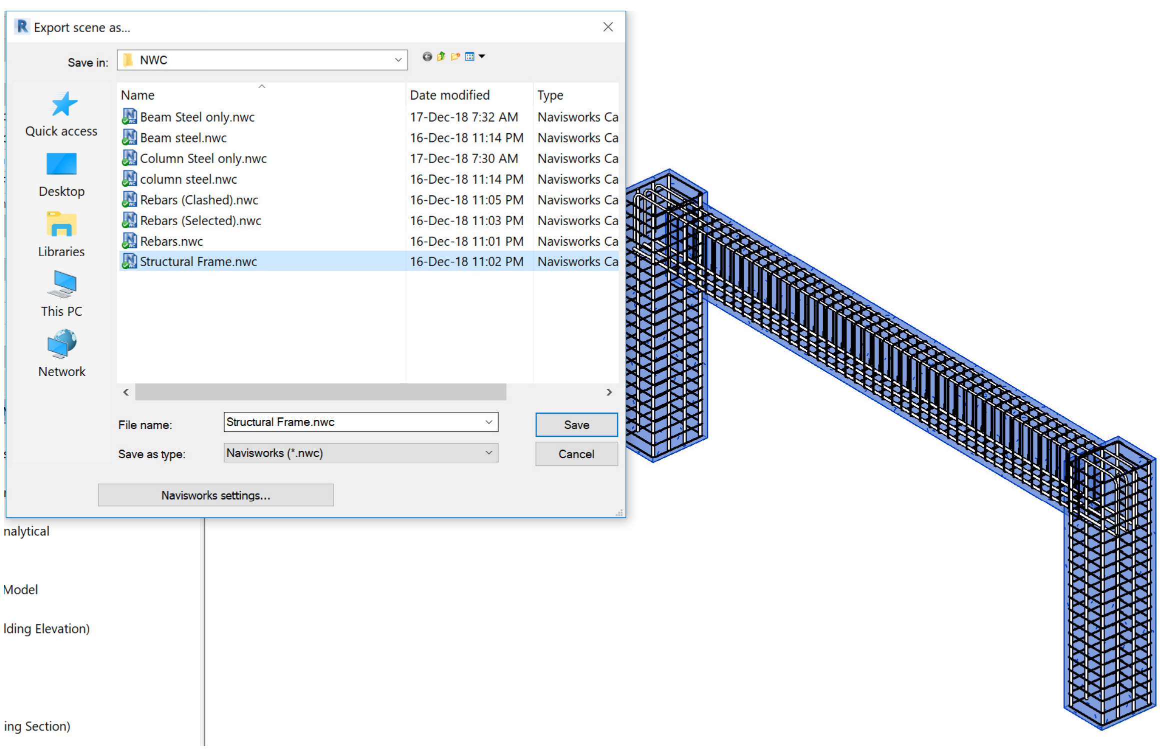
Reset temporary Hide / Isolate.



You can also isolate the column and beam elements with the same method.

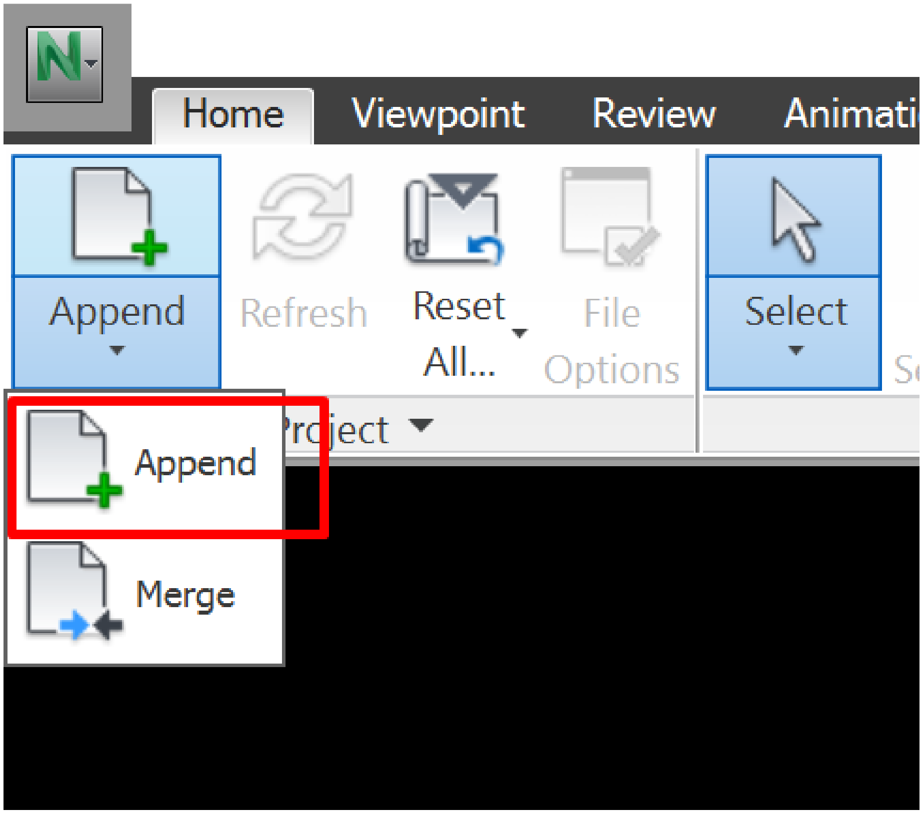


Export and Save it as “**Structure Frame.nwc**”

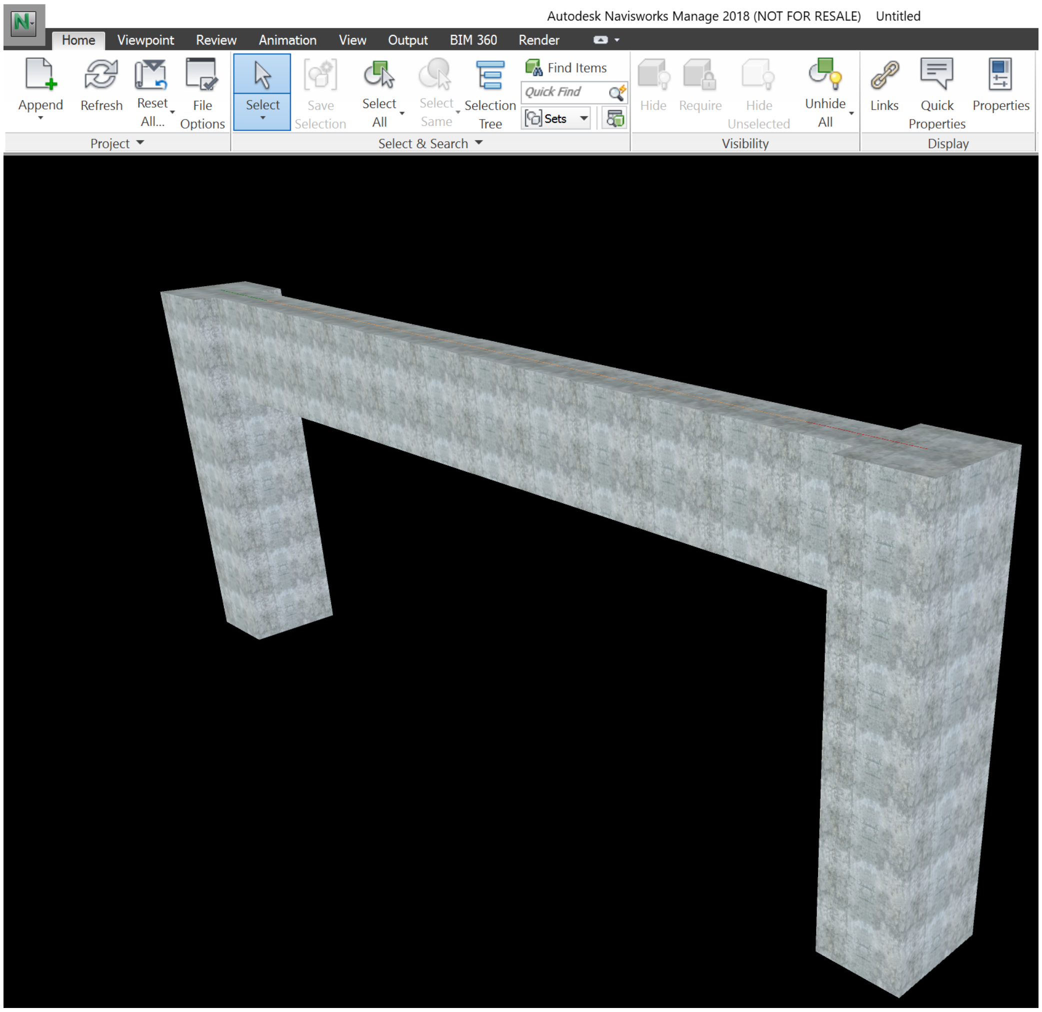


Open Navisworks Manage 2018.

At top Menu,choose Home, then Append.

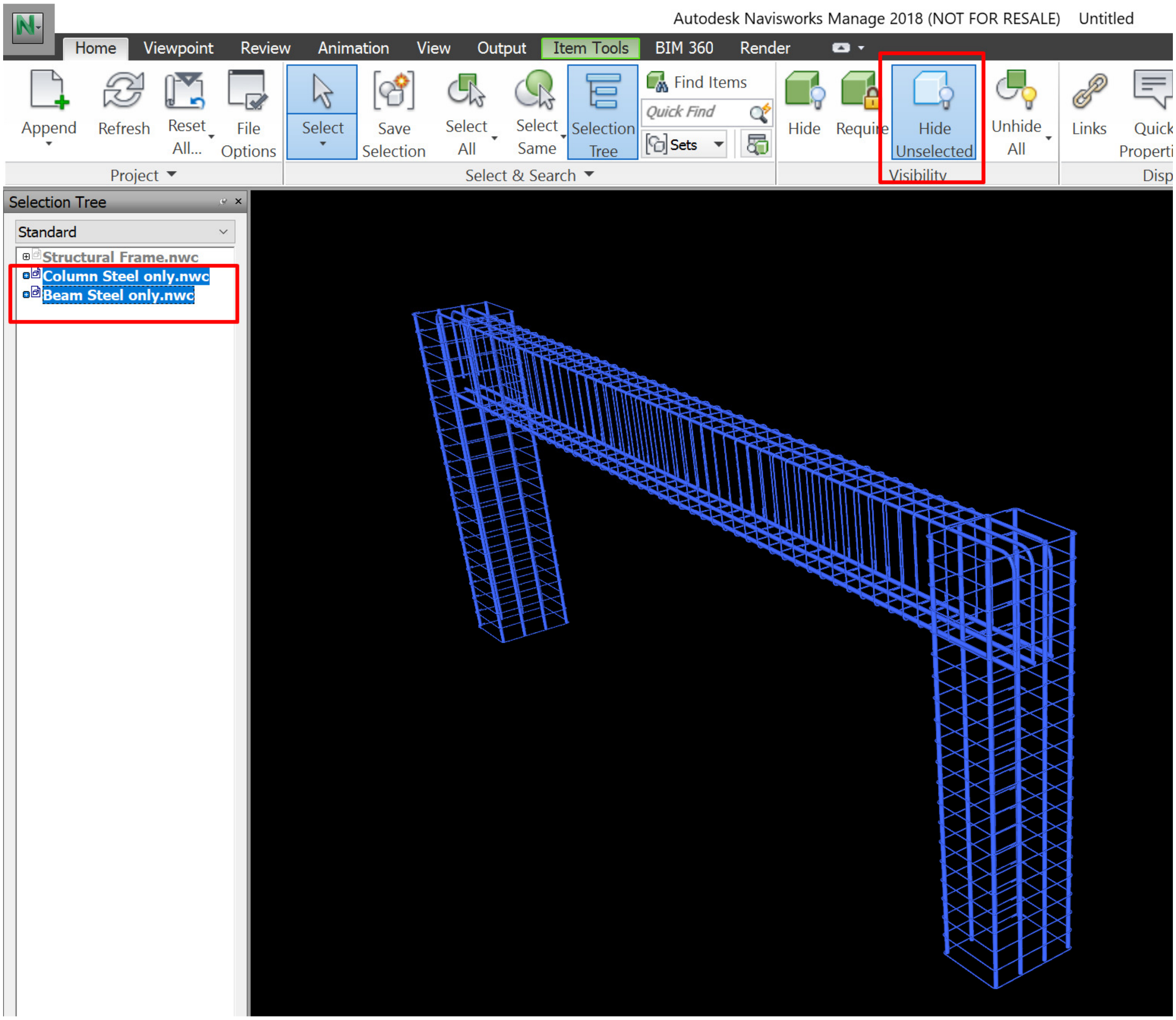


Put on the structure Frame.nwc first.

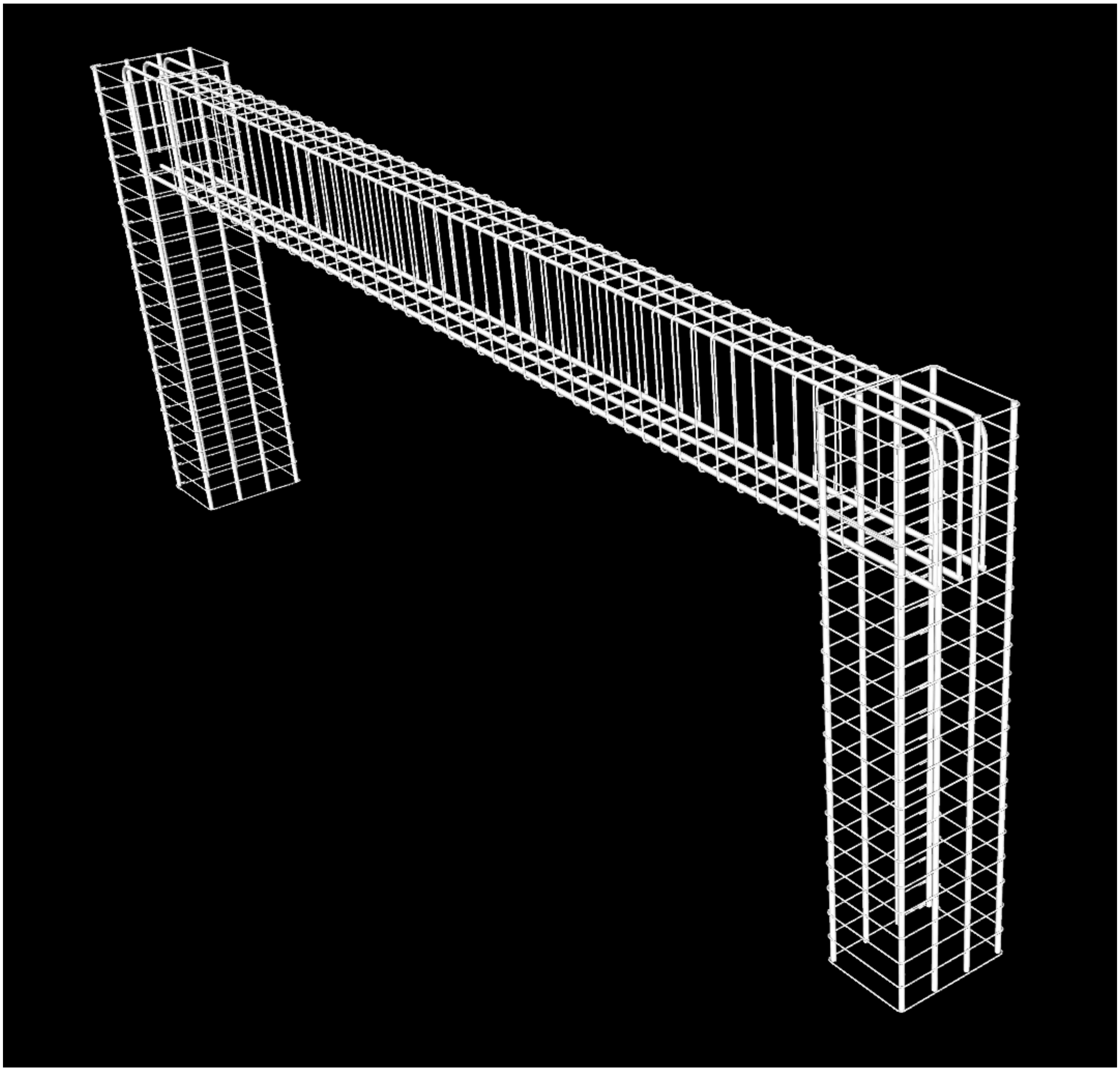


Append the Column steel and beam steel as well.

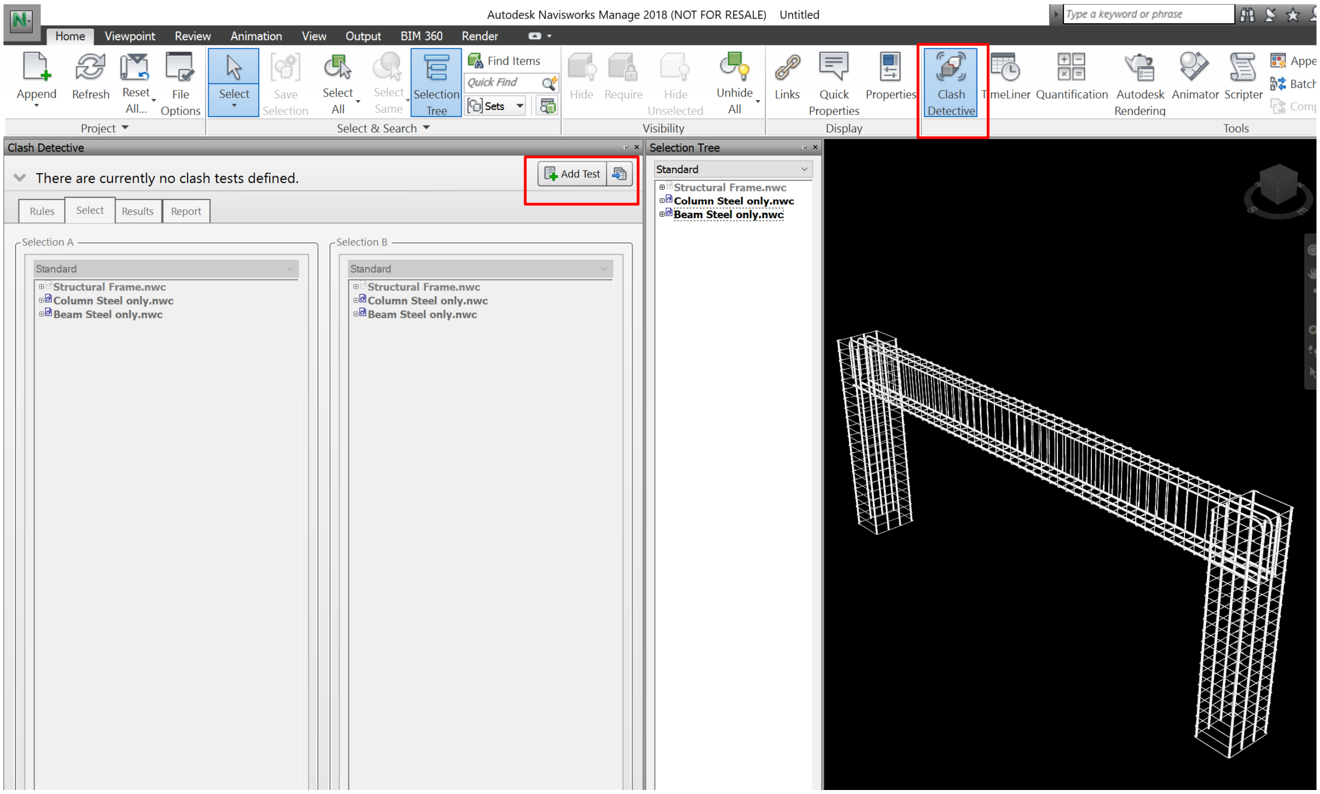
Filter the steel by choosing “Column Steel only.nwc” and “Beam Steel only.nwc” all then “Hide Unselected”:



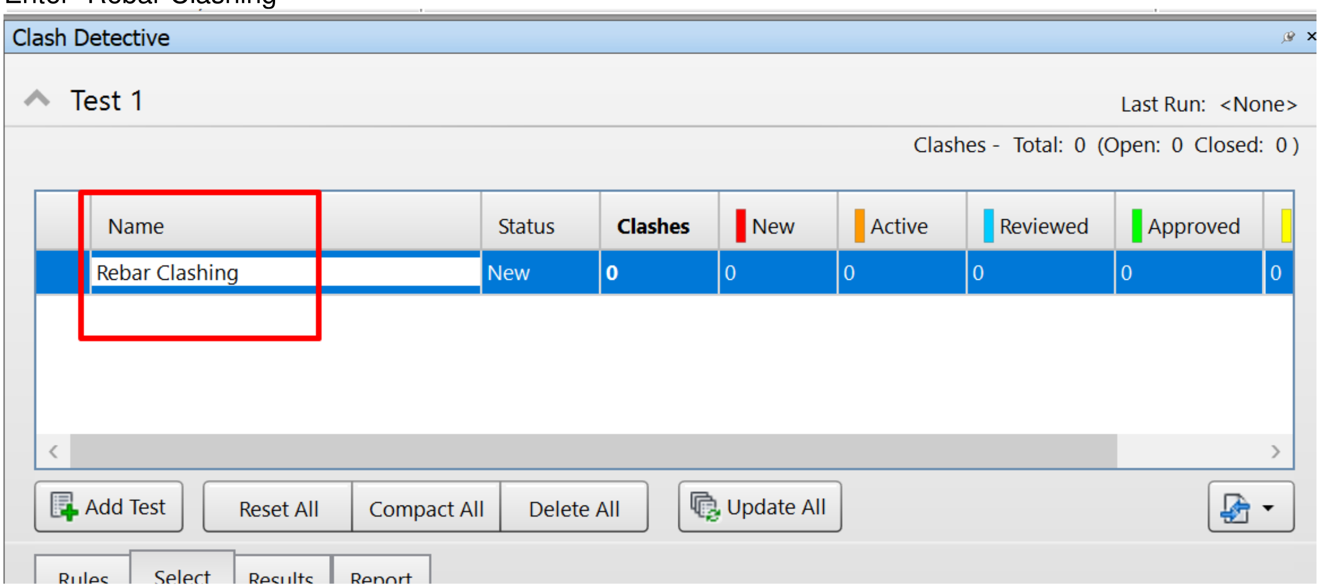
Press Escape few times.



On menu, press Clash detective and Add Test:

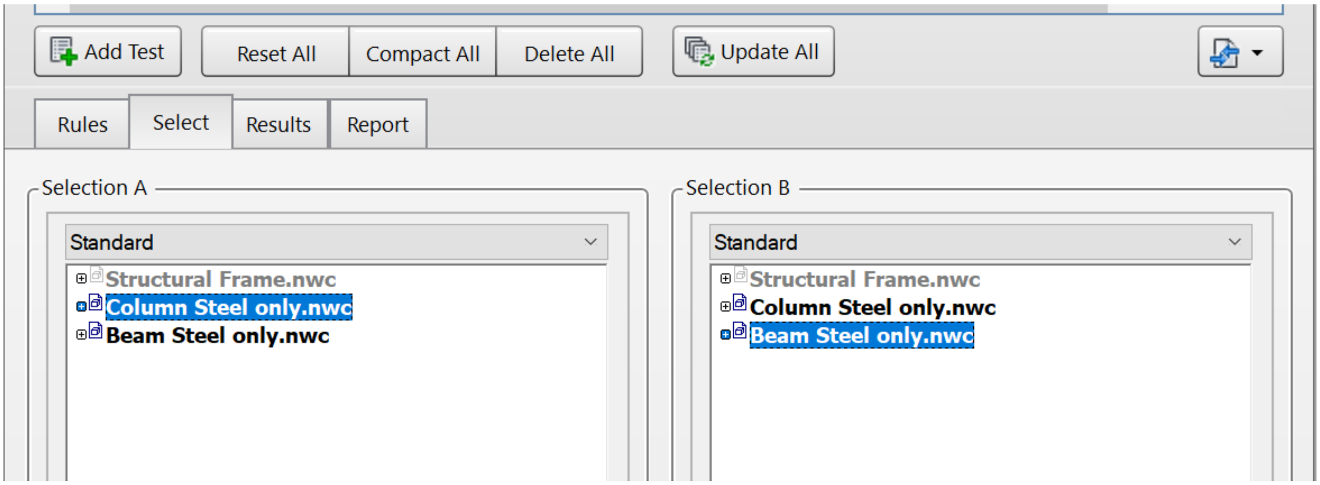


Enter “Rebar Clashing”

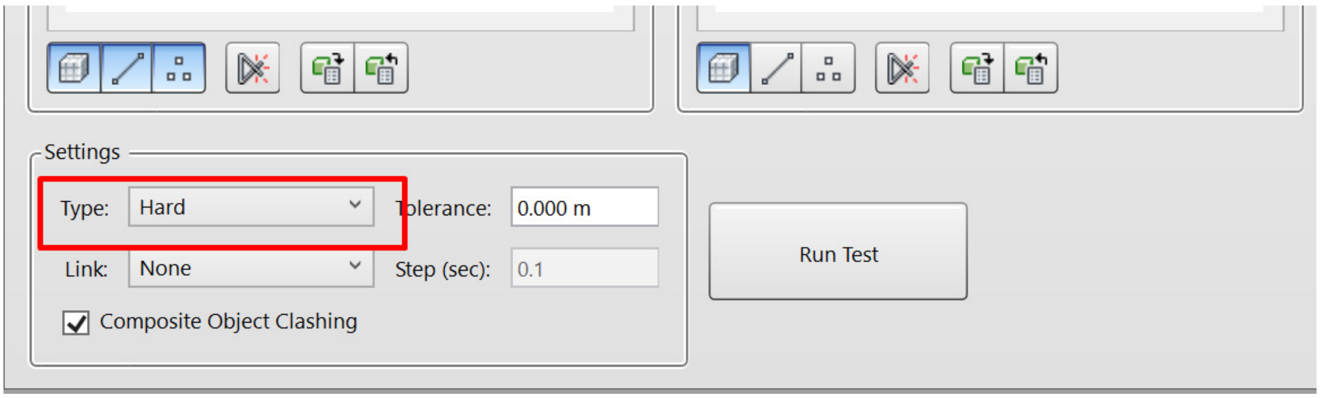


Choose Selection A – Column Steel only

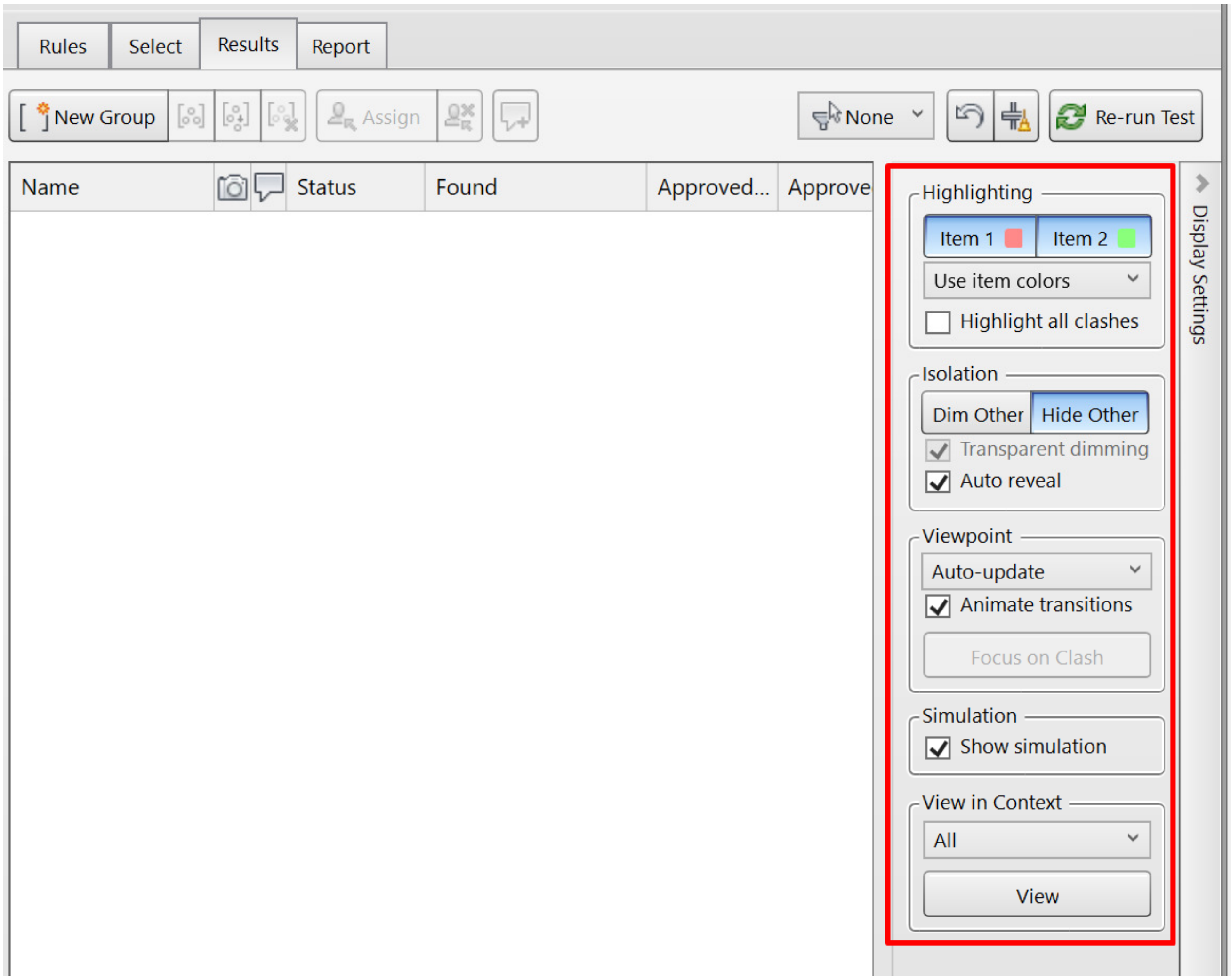
Choose Selection B – Beam Steel only



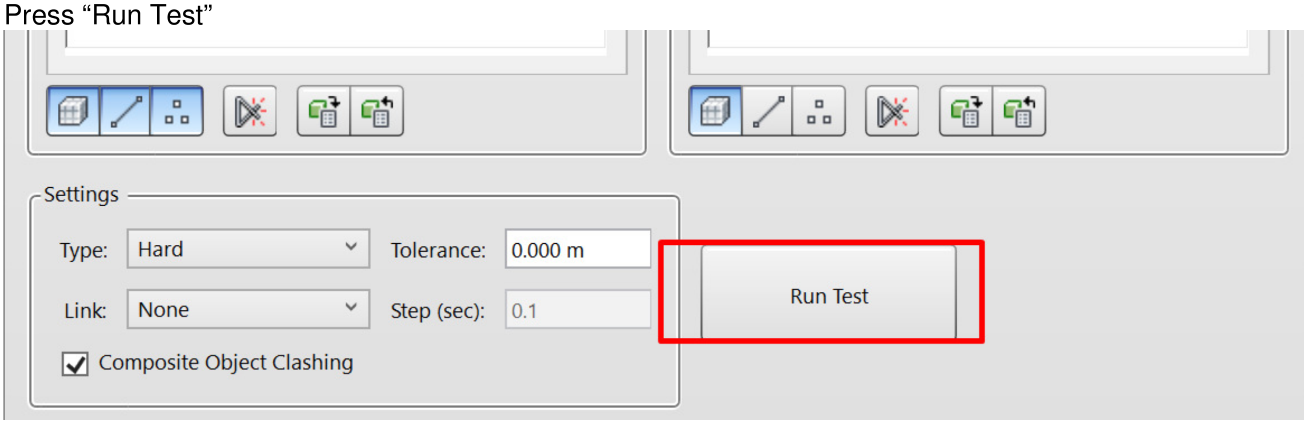
Select the Type of clash as "Hard Clash"



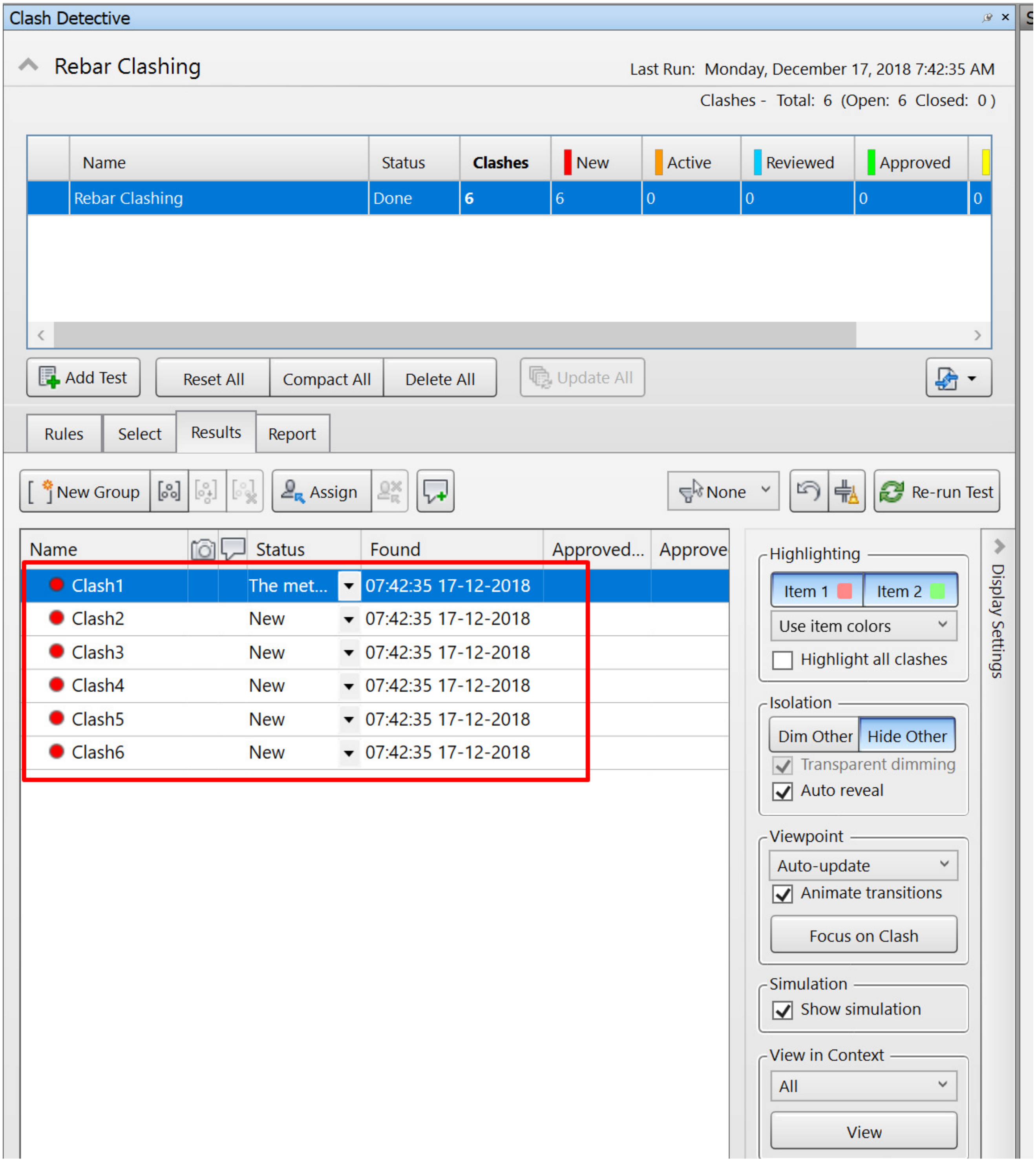
Result follow the following settings:



Press “Run Test”

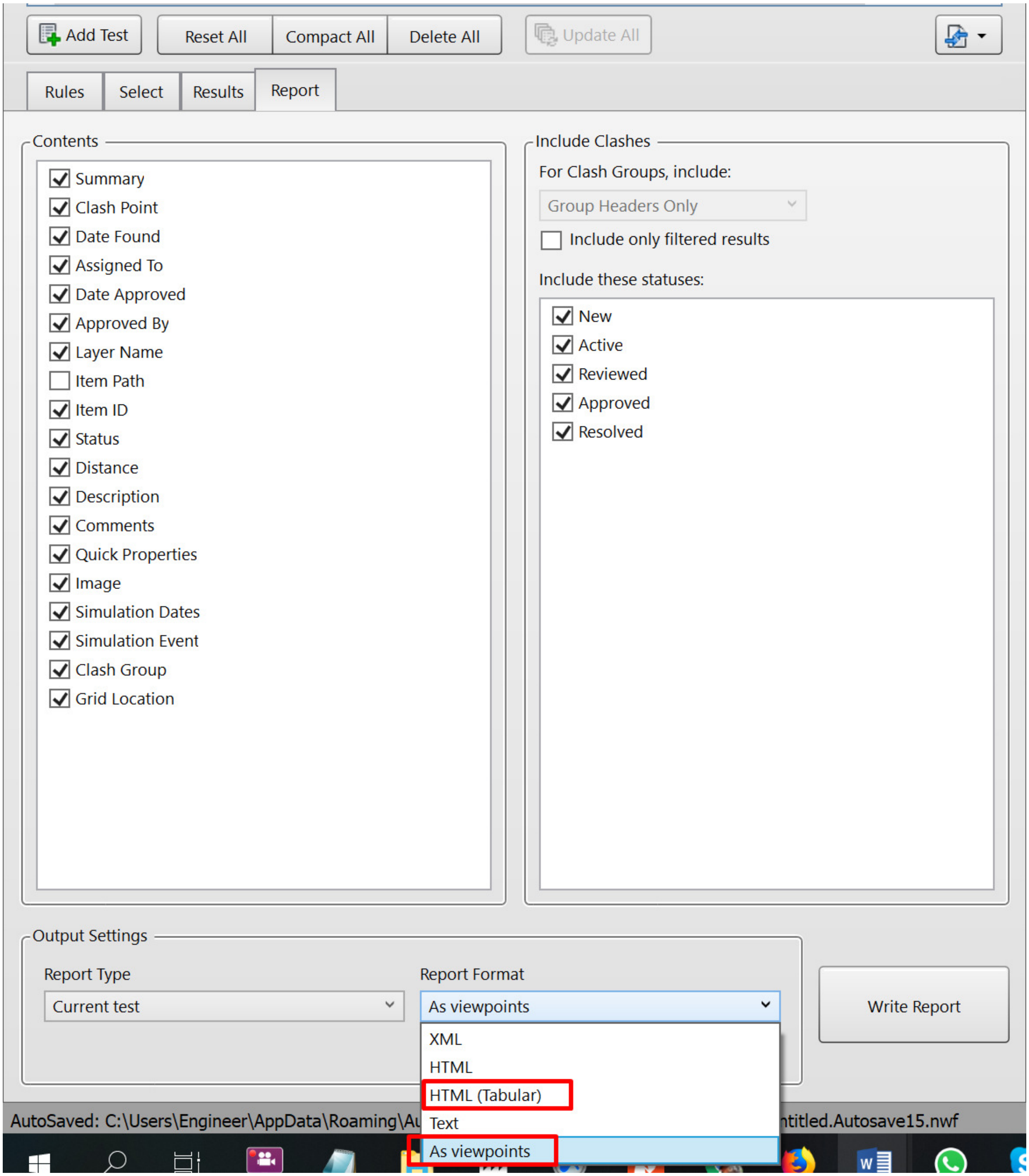


Clashes are highlighted.

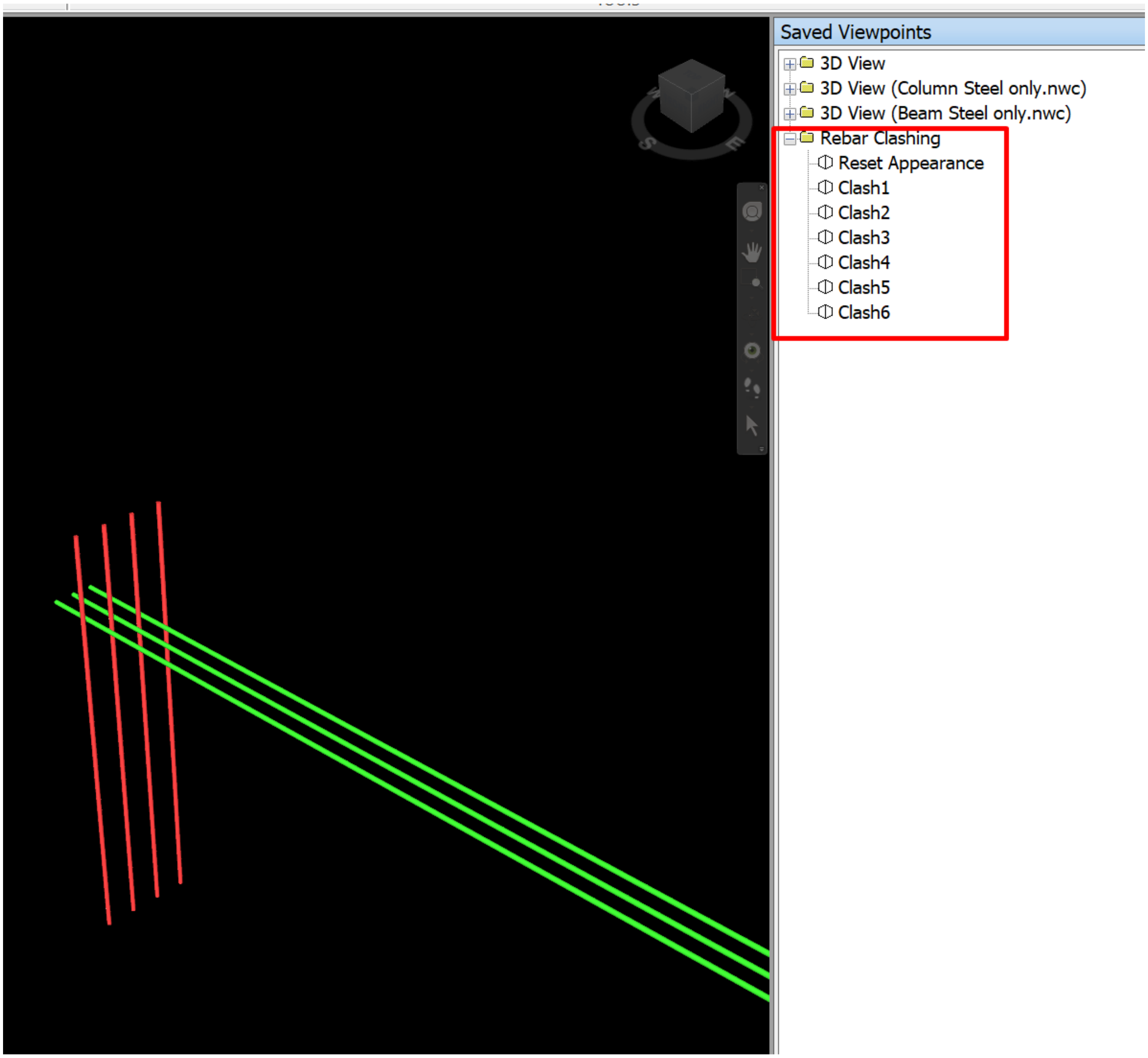


Examine the above clashes and then

Report can be formed as Viewpoint / HTML (Tabular)



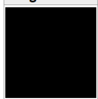
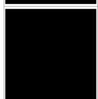
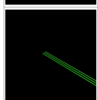
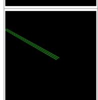
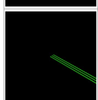
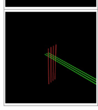
Report in Viewpoints:



Report in HTML:

AUTODESK® NAVISWORKS® **Clash Report**

Rebar Clashing	Tolerance	Clashes	New	Active	Reviewed	Approved	Resolved	Type	Status
	0.000m	6	6	0	0	0	0	Hard	Old

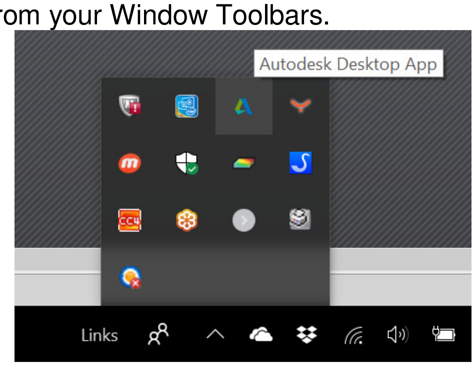
							Item 1				Item 2			
Image	Clash Name	Status	Distance	Description	Date Found	Clash Point	Item ID	Layer	Item Name	Item Type	Item ID	Layer	Item Name	Item Type
	Clash1	New	-0.005	Hard	2018/12/16 23:42	x:-6.313, y:-1.195, z:2.369	*Element ID*: 352979	<No level>	Rebar Bar	Solid	*Element ID*: 353891	<No level>	Rebar Bar	Solid
	Clash2	New	-0.004	Hard	2018/12/16 23:42	x:1.325, y:-0.917, z:2.518	*Element ID*: 352981	<No level>	Rebar Bar	Solid	*Element ID*: 353891	<No level>	Rebar Bar	Solid
	Clash3	New	-0.002	Hard	2018/12/16 23:42	x:-5.778, y:-0.936, z:2.239	*Element ID*: 352979	<No level>	Rebar Bar	Solid	*Element ID*: 354029	<No level>	Rebar Bar	Solid
	Clash4	New	-0.002	Hard	2018/12/16 23:42	x:0.785, y:-1.203, z:2.237	*Element ID*: 352981	<No level>	Rebar Bar	Solid	*Element ID*: 354029	<No level>	Rebar Bar	Solid
	Clash5	New	-0.001	Hard	2018/12/16 23:42	x:-6.290, y:-1.191, z:2.250	*Element ID*: 353054	<No level>	Rebar Bar	Solid	*Element ID*: 354029	<No level>	Rebar Bar	Solid
	Clash6	New	-0.001	Hard	2018/12/16 23:42	x:-5.797, y:-1.191, z:2.252	*Element ID*: 353151	<No level>	Rebar Bar	Solid	*Element ID*: 354029	<No level>	Rebar Bar	Solid

Steel Connections:

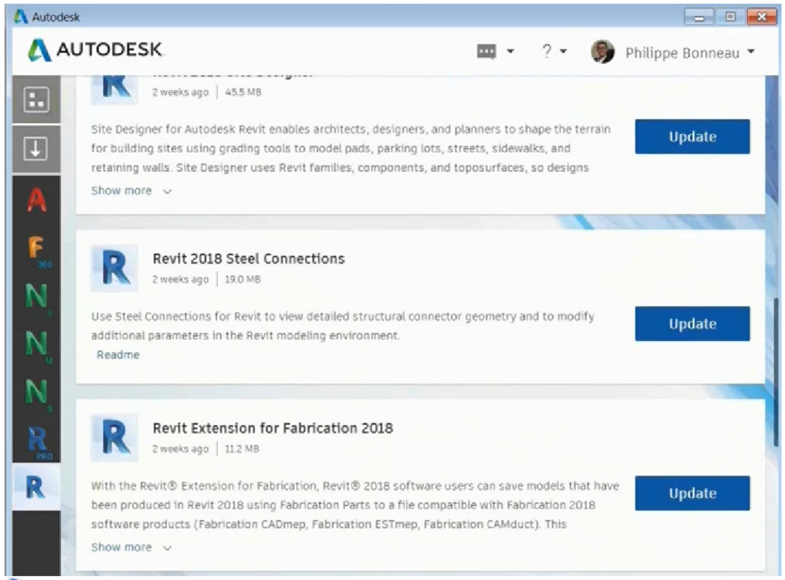
In Revit 2018, there is a library of Steel Connection components to help you model the connection between steel members. You will learn how to install and load them for use in a project. Two of the connections will be used as examples to demonstrate this powerful tool.

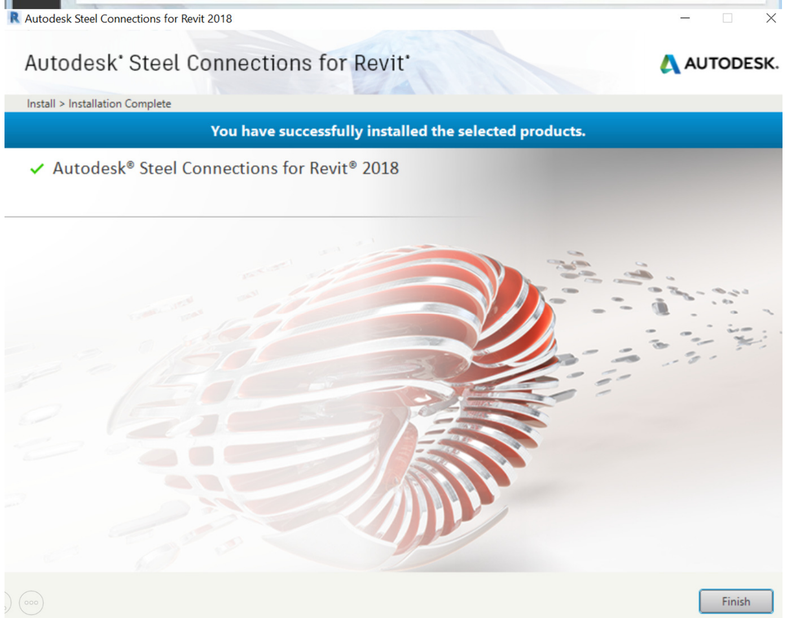
12.1 Install and Load Steel Connections

1 Open Autodesk Desktop App from your Window Toolbars.



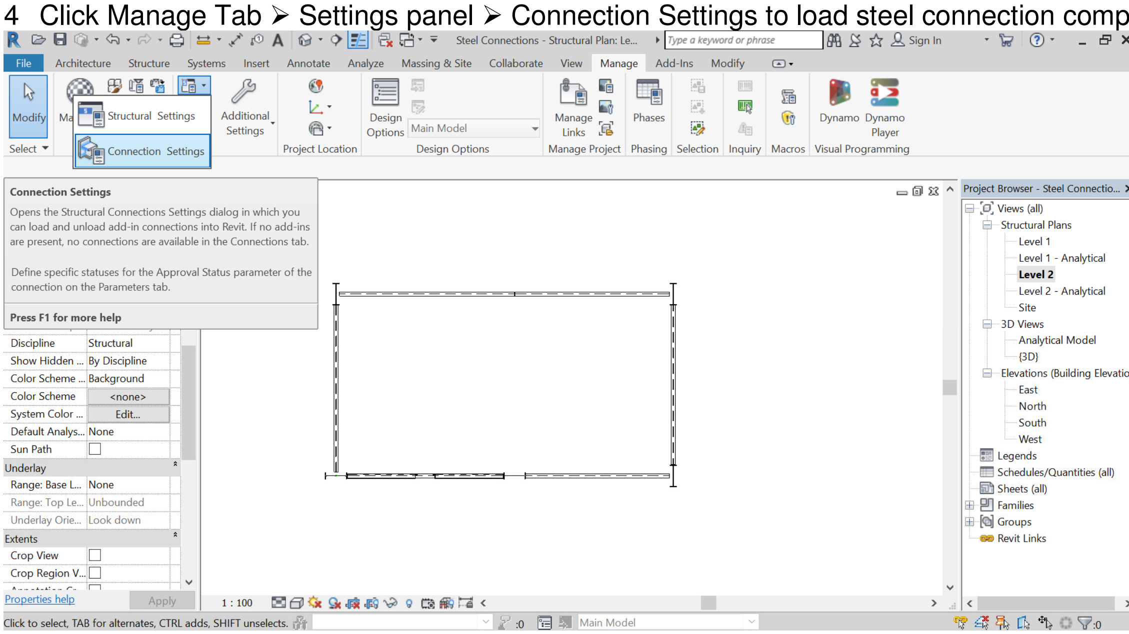
2 Scroll down until you find the Revit 2018 Steel Connections and click “Update” to install.





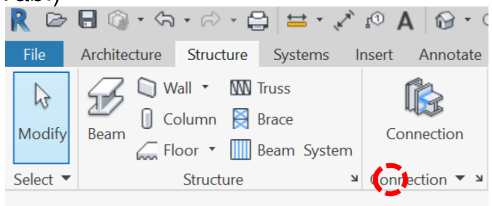
3 After installation, open from the course folder the “Steel Connections.rvt” file.

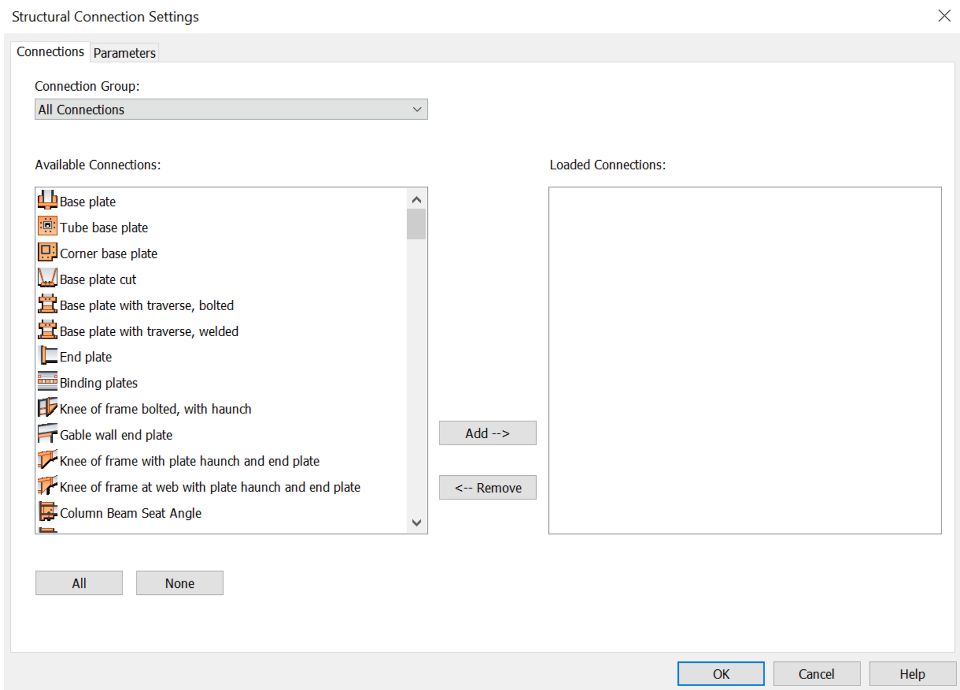
4 Click Manage Tab ➢ Settings panel ➢ Connection Settings to load steel connection components.



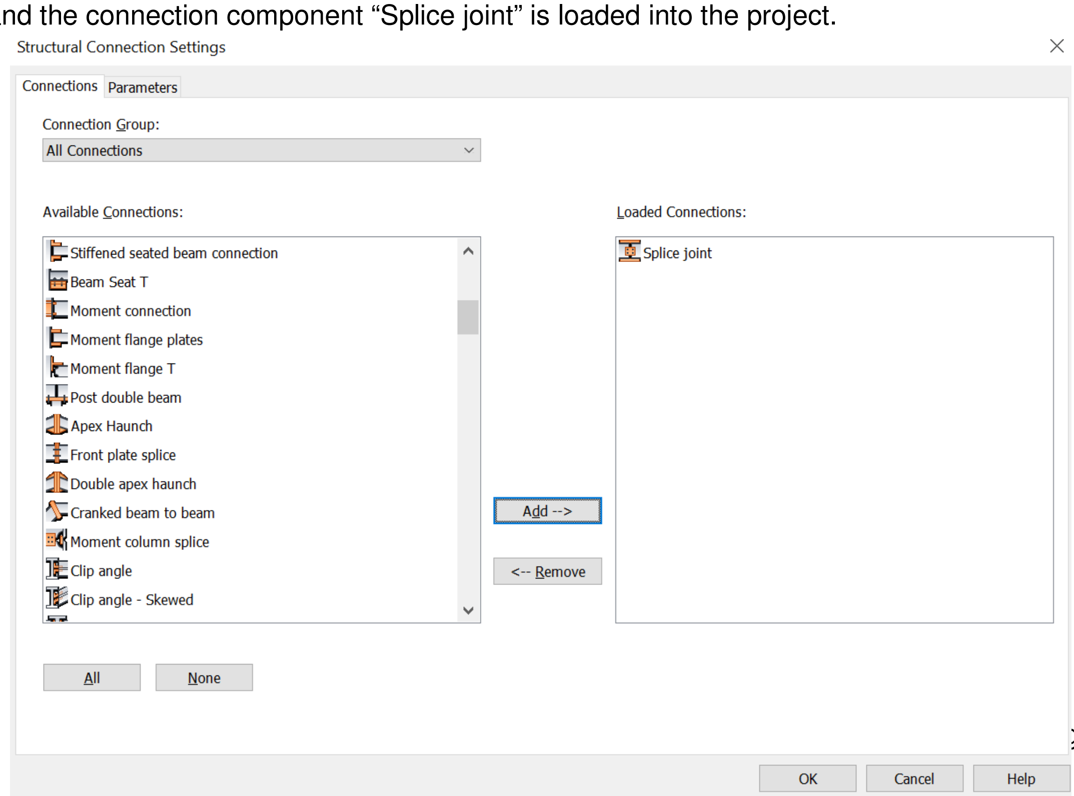
(*Alternately, you can open Connection Settings by clicking on the little arrow at the bottom right of the

Connection panel in Structure Tab.)



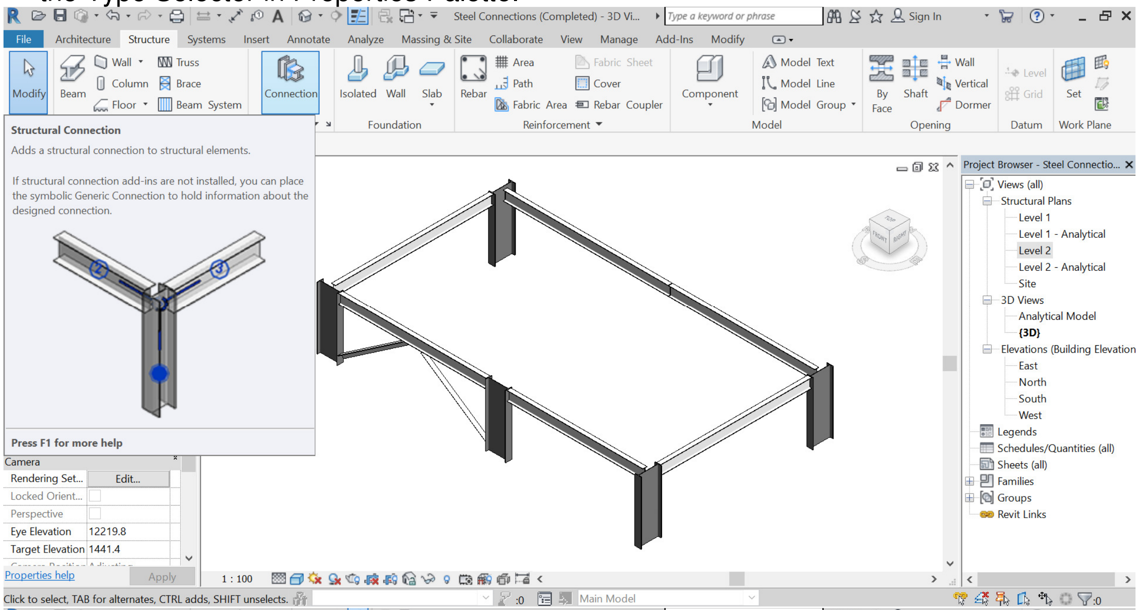


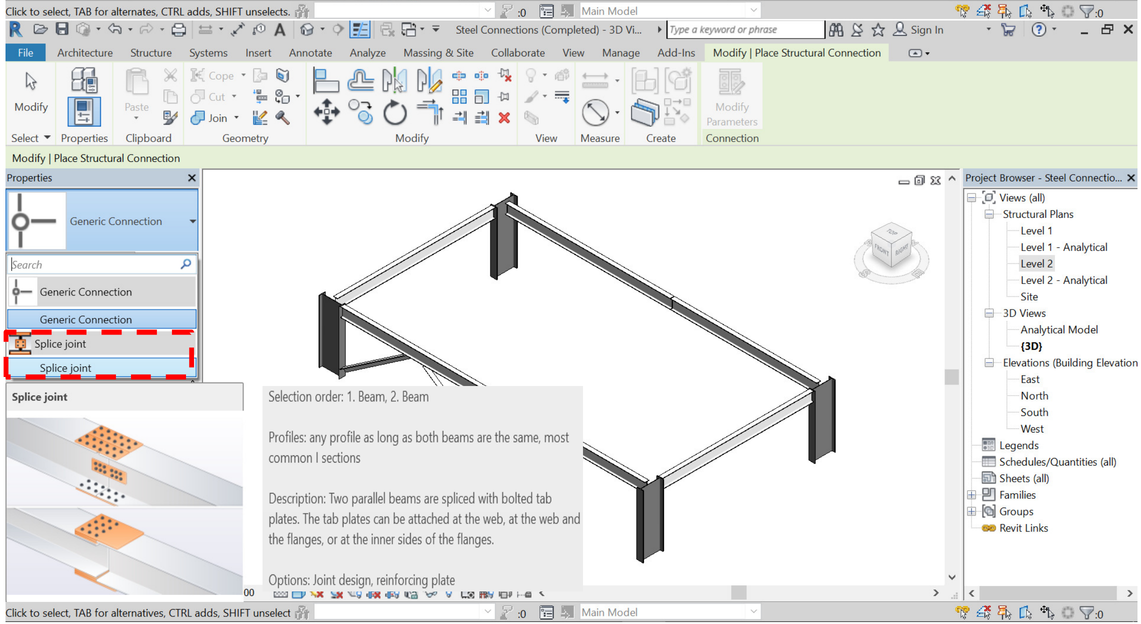
5 You can see there are many steel connection components such as “Base Plate” and “Column Beam Seat Angle”.
Scroll down the list of Available Connections and find “Splice joint”. Select it and click “Add”. Then click “OK” and the connection component “Splice joint” is loaded into the project.



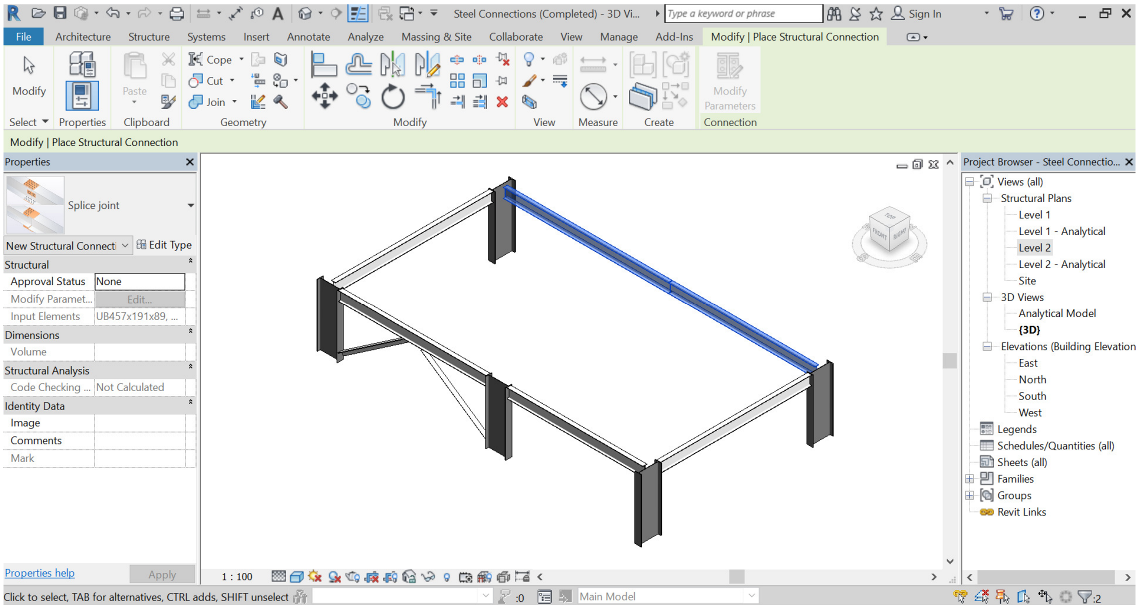
12.2. Create a Splice Joint (joining two steel beams)

1 Go to 3D view and click Structure Tab ➢ Connection panel ➢ Connection and select “Spice joint” from the Type Selector in Properties Palette.

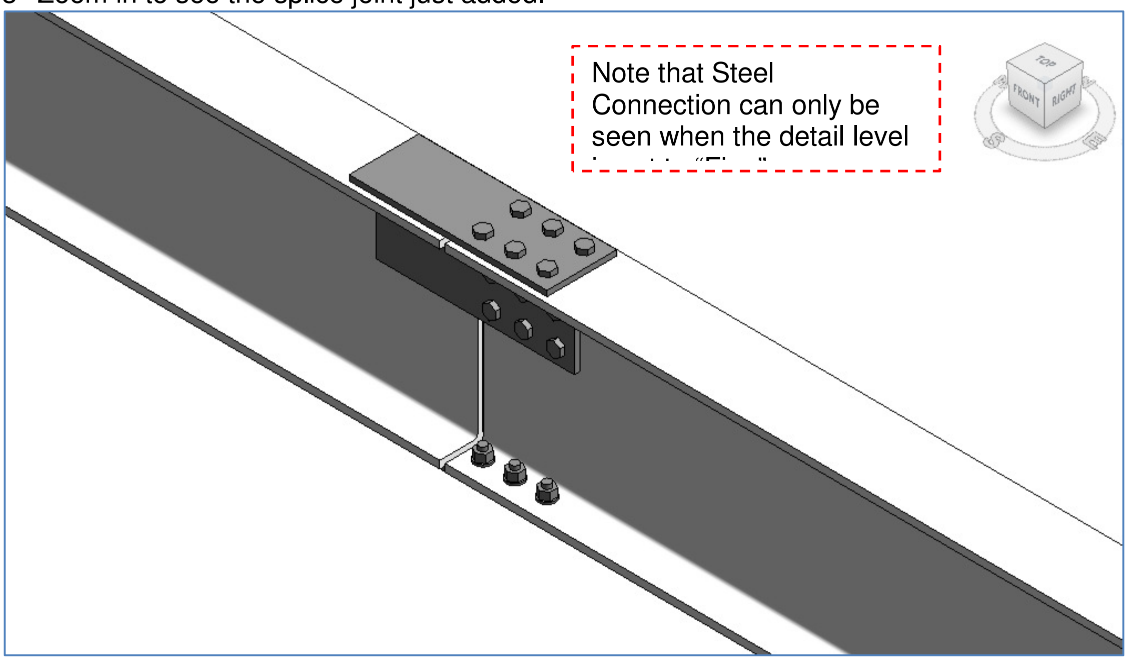




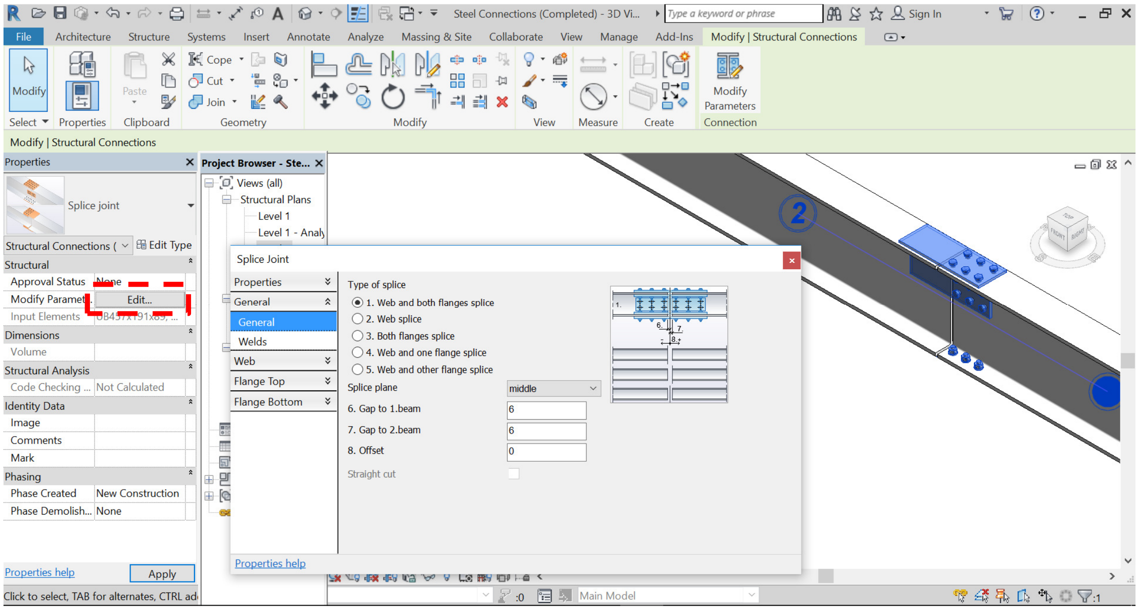
2 Select the two steel beams as shown below. (Hold down “ctrl” key while selecting the beams.) Then press “spacebar” and a splice joint is added to join the selected beams.



3 Zoom in to see the splice joint just added.



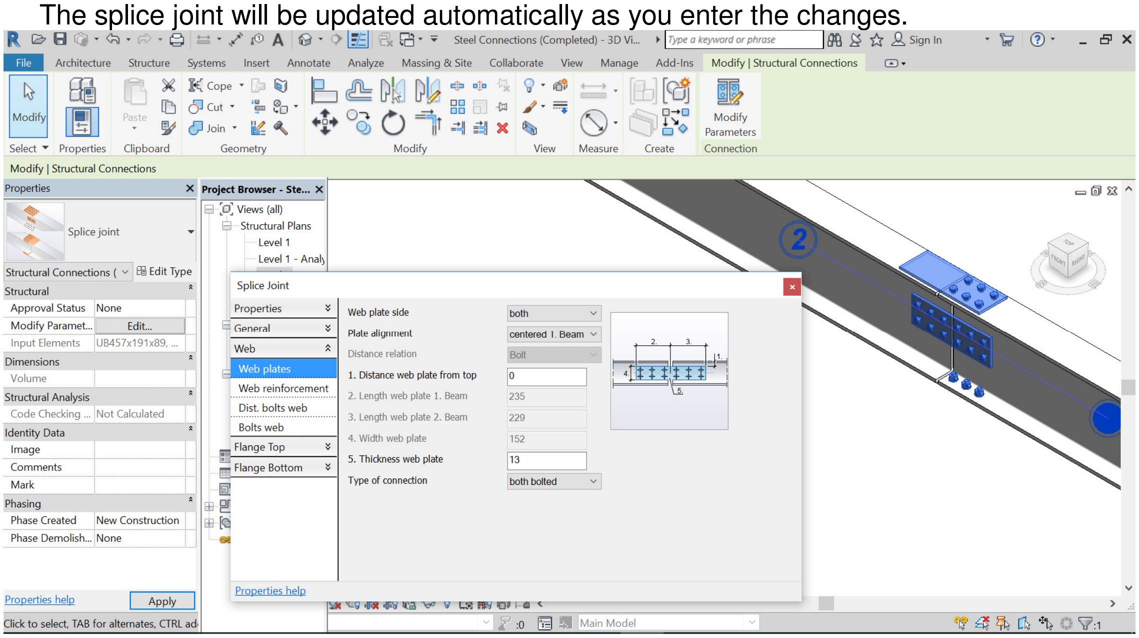
4 Select the splice joint and click “Edit” to modify the detail of this joint. Notice the number “2” on the left beam. This indicates that it is the “2.beam” in this joint configuration.



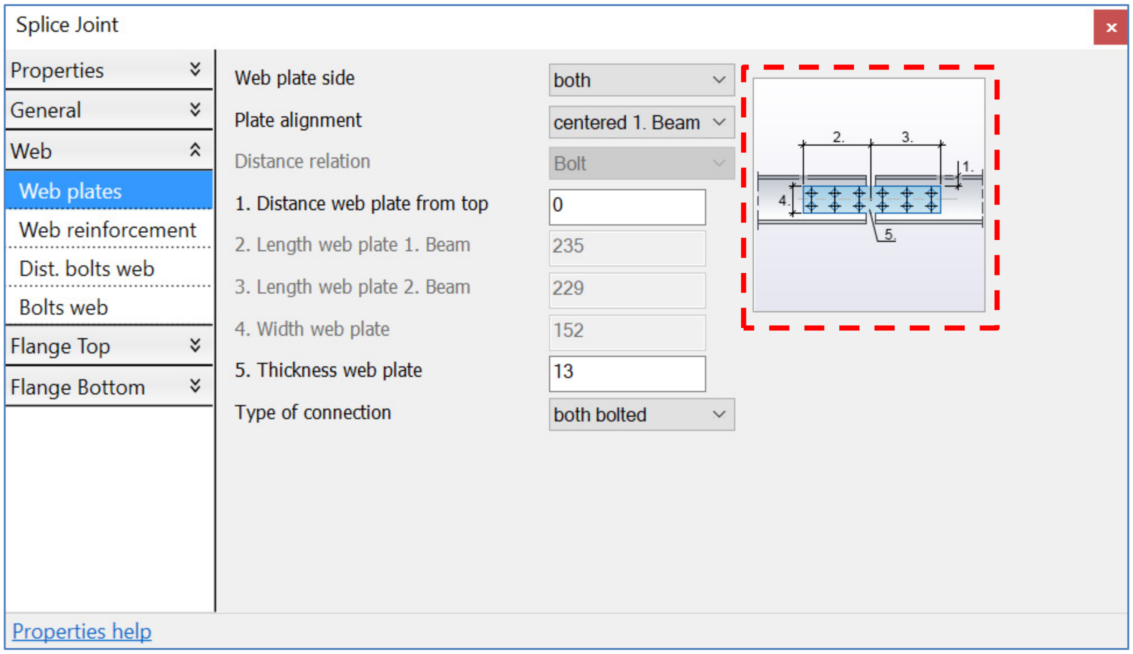
5 From the “Web” menu, select “Web plates” and set

- Plate alignment: “centered 1.Beam”
- Distance web plate from top: “0”
- Type of connection: “both bolted”

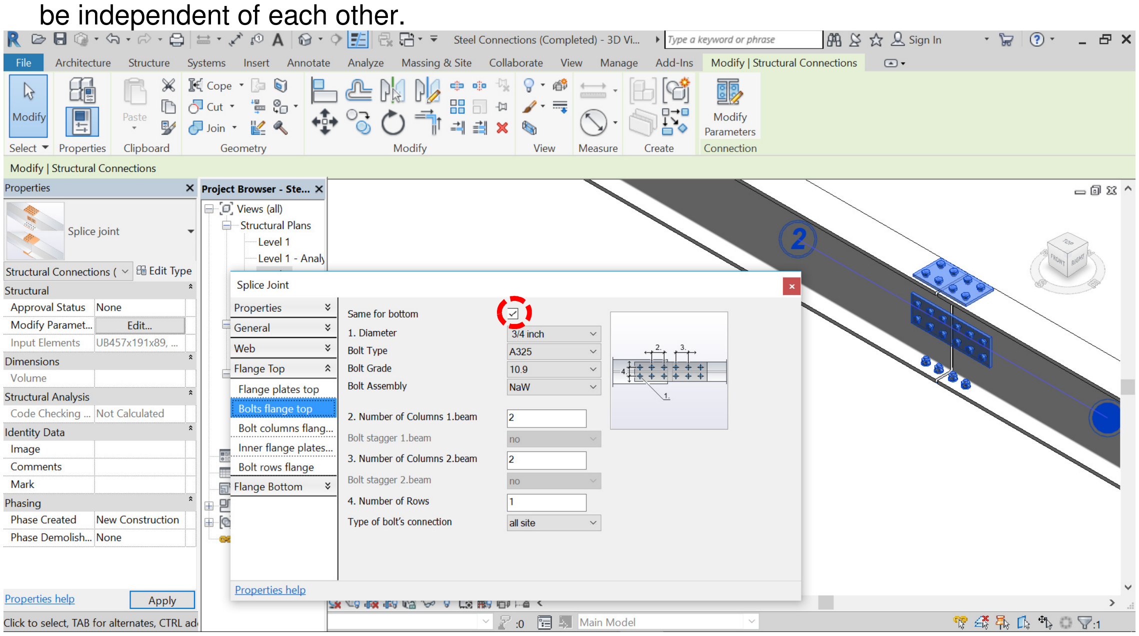
The splice joint will be updated automatically as you enter the changes.



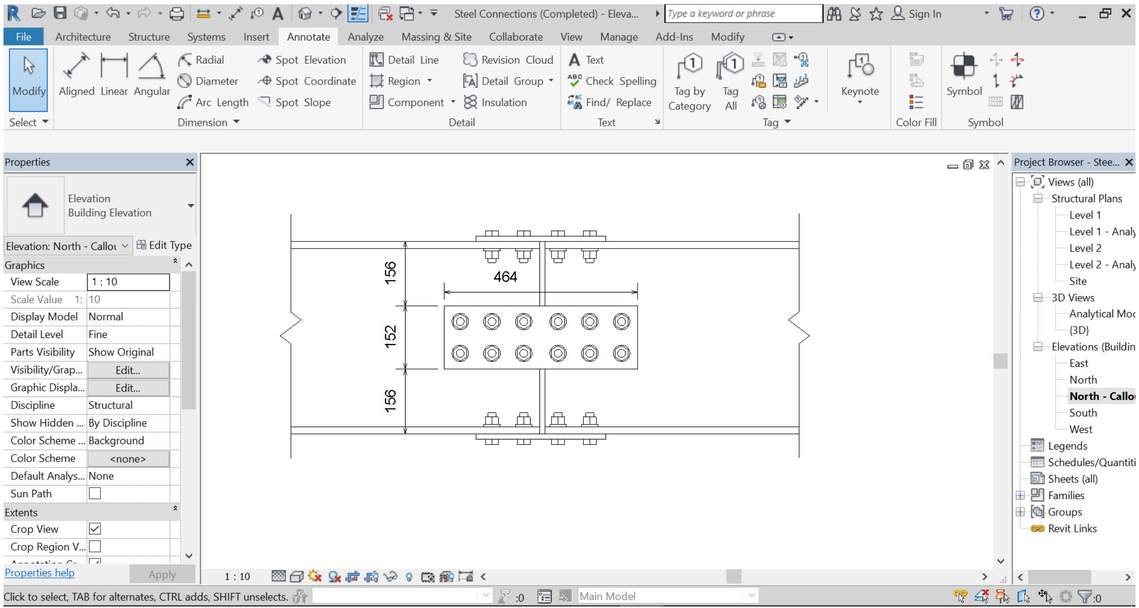
Note the little diagram with annotation to help you understand what parameter you are changing.



6 Next, from the "Flange Top" menu, select "Flange plates top" and change "Type of connection" to "both bolted". Then select "Bolts flange top" and change both "Number of Columns 1.beam" and "Number of Columns 2.beam" to "2".
Note the "Same for bottom" option. Uncheck it if you want the joint setting for top and bottom flanges to be independent of each other.

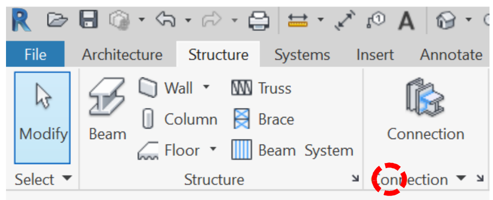


7 You can create a callout of the joint from the north elevation of the model. Then annotate the callout for drawing production.

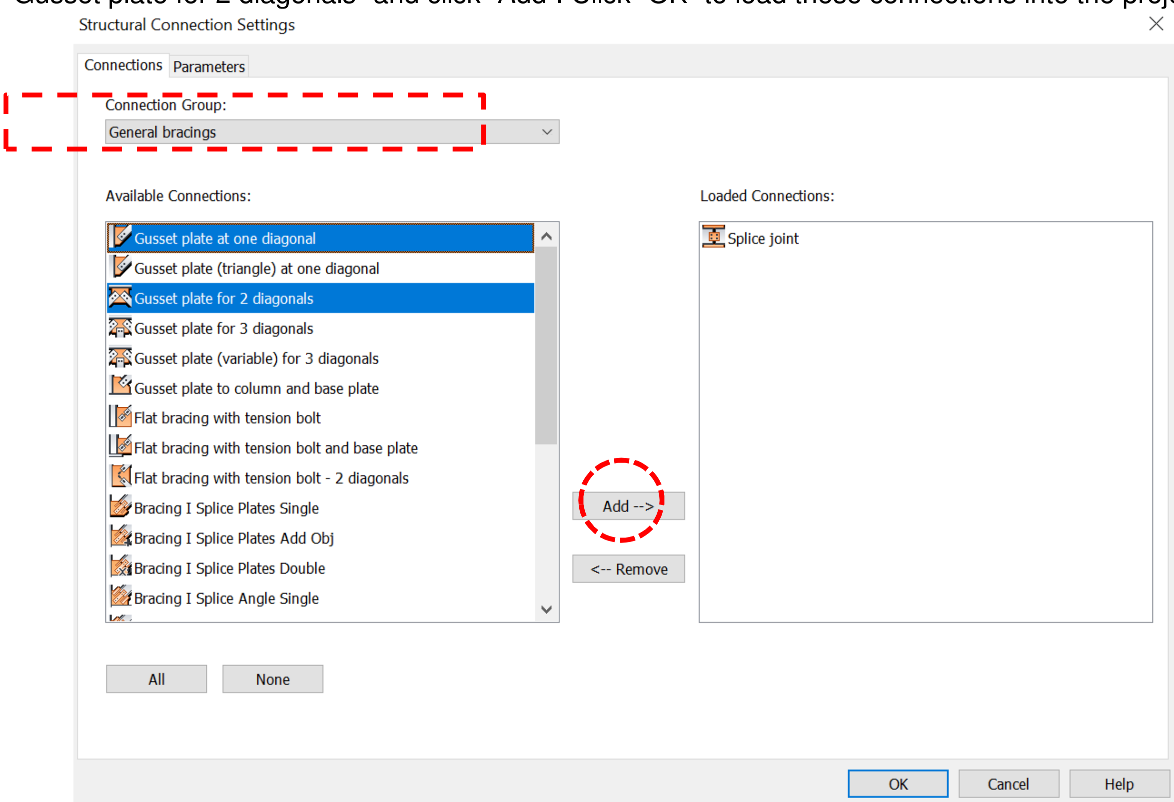


7.5.3 Create Gusset Plate joints for Bracings

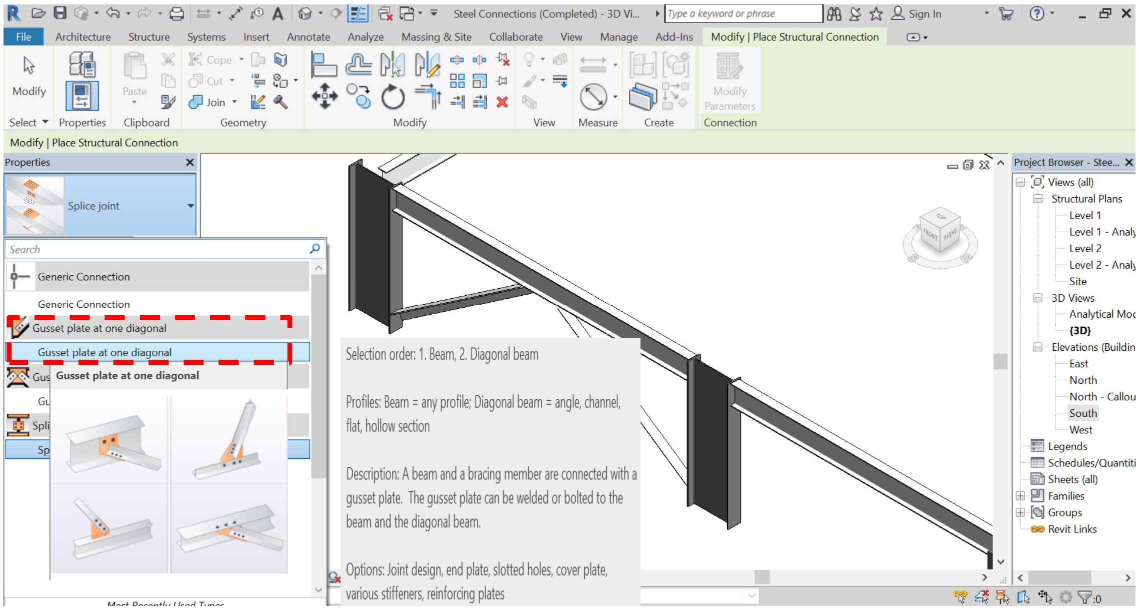
1 Go to 3D view and turn on the Connection Settings by clicking the little arrow in the Connection panel (Structure Tab).



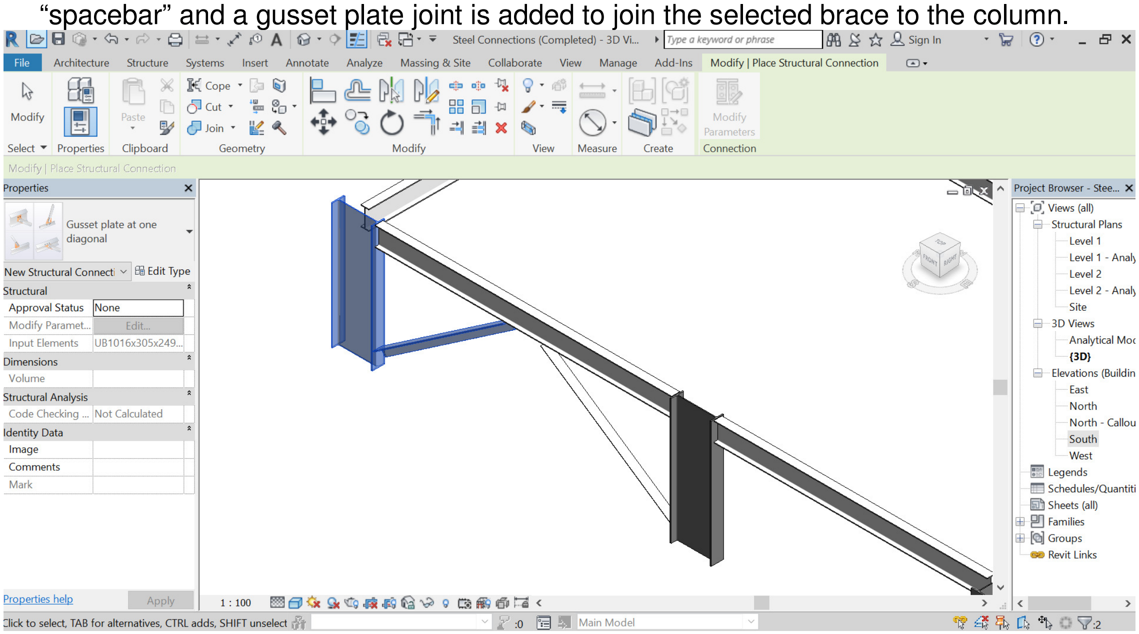
Choose “General bracings” in “Connection Group”. Then select “Gusset plate at one diagonal” and “Gusset plate for 2 diagonals” and click “Add”. Click “OK” to load these connections into the project.

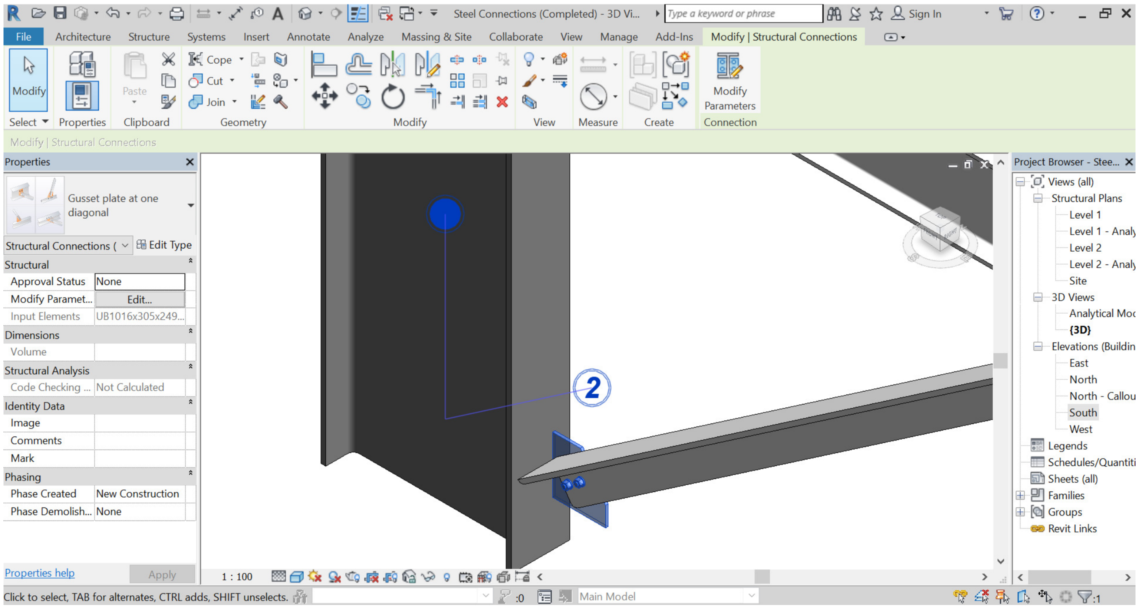


2 Click Structure Tab ➢ Connection panel ➢ Connection and select “Gusset plate at one diagonal” from the Type Selector in Properties Palette.

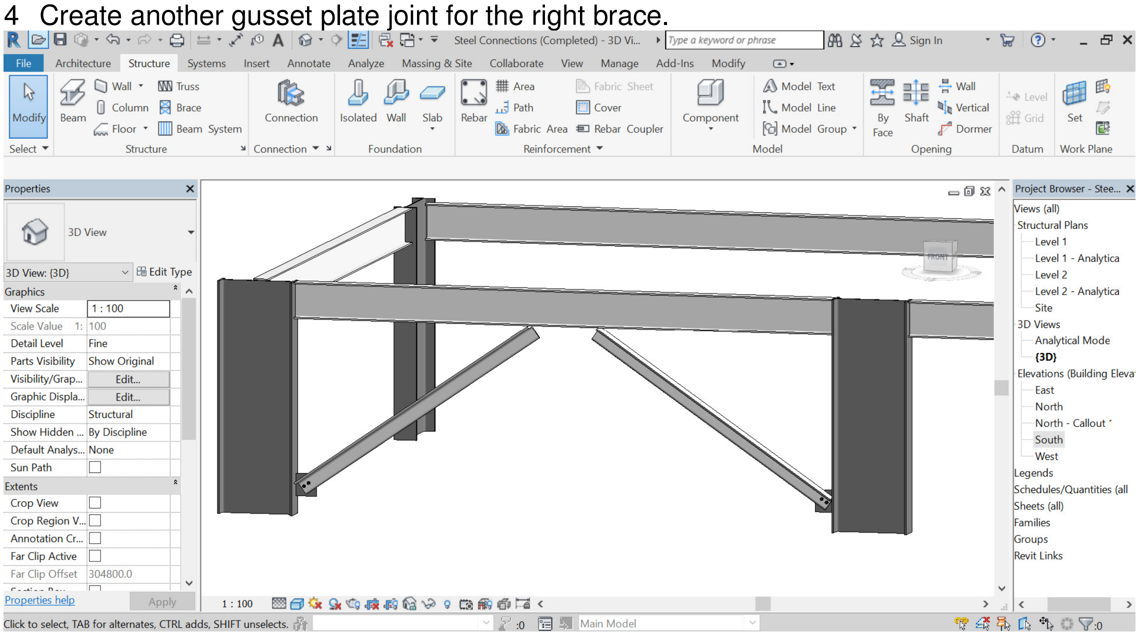


3 Select the left brace and the column as shown below. (Hold down “ctrl” key while selecting.) Then press “spacebar” and a gusset plate joint is added to join the selected brace to the column.

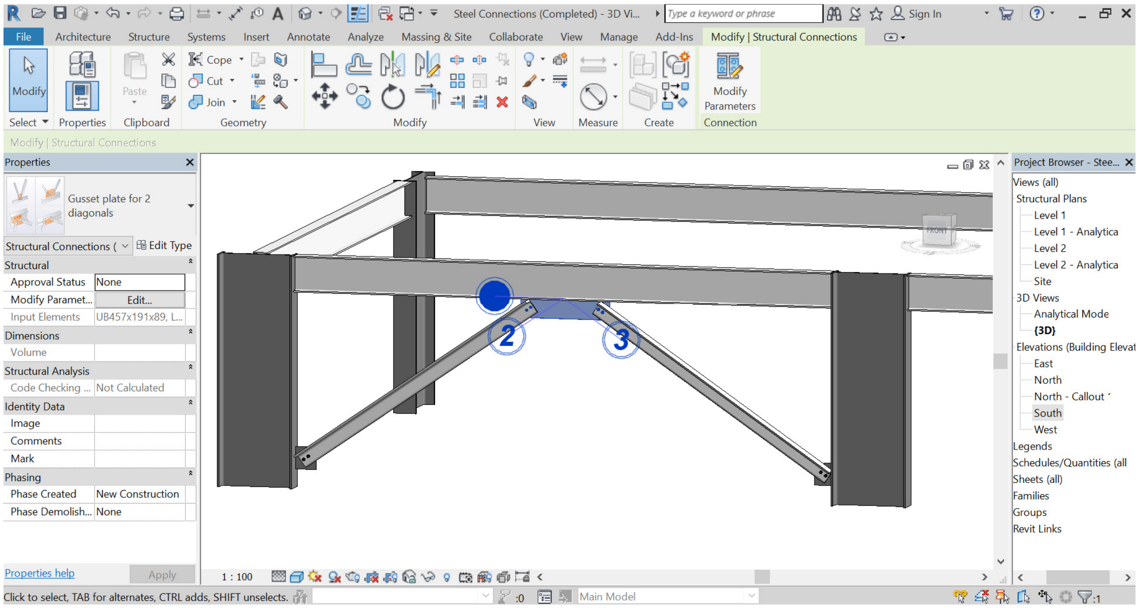




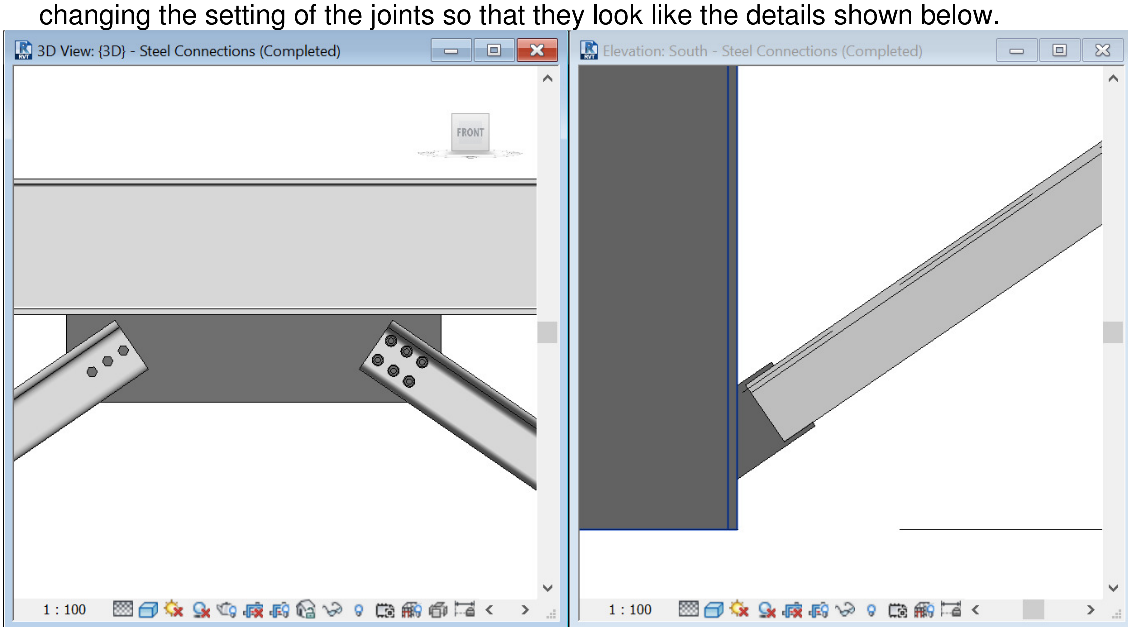
4 Create another gusset plate joint for the right brace.



5 Now, click Structure Tab ➢ Connection panel ➢ Connection and select "Gusset plate for 2 diagonals" from the Type Selector in Properties Palette.
Select the two braces and the horizontal beam (Hold down "ctrl" key while selecting). Then press "spacebar" and a gusset plate joint is added to join the selected braces to the column.



You can select any steel connections created and click “Edit” to modify the details of the joints. Try changing the setting of the joints so that they look like the details shown below.



Exporting the Model to Navisworks and collaborations.