324109

Robot Structural Analysis: A Solid Foundation for Practical Projects

Will Ikerd, P.E.
IKERD Consulting

Garrett Shahan, E.I.T.
IKERD Consulting

Michael Perdue
IKERD Consulting

Learning Objectives

- Explore Robot Structural Analysis software's key features and capabilities
- Learn how to use the different project types and layouts built into Robot Structural Analysis
- Learn how to implement best practices for deploying Robot Structural Analysis within a structural engineering office
- Learn how to use the capabilities of the Robot Structural Analysis, Revit, and Advance Steel interoperability

Description

This lab will use straightforward practical examples to teach best practices to new users of Robot Structural Analysis software. The lab is intended for those who have an understanding of structural analysis but are new to Robot Structural Analysis. It will introduce the basic functionality of Robot Structural Analysis, key features, and best office practices for teams working in Robot Structural Analysis. Topics will include, but are not limited to, concrete design, steel design, and composite deck design. Additional topics will address loadings and key strategies for collaboration between Robot Structural Analysis and other Autodesk applications, such as Revit software and Advance Steel software.

Speaker(s)

Garrett Shahan is a Graduate Engineer at IKERD Consulting, an internationally recognized consulting group in buildings, civil, and industrial construction markets specializing in using Building Information Modeling (BIM)-enabled Virtual Design and Construction (VDC). During his time at IKERD, he has been involved in a variety of projects involving engineering design and modeling, reality capture, software training, and BIM coordination. He has utilized and taught Autodesk products including but not limited to Revit, Robot, Advance Steel, Inventor, ReCap, and Navisworks for projects ranging from residential houses to multi-million square feet sport facilities.

Will Ikerd—PE, CM-BIM, LEED AP—is principal at IKERD Consulting, an internationally recognized consulting group in buildings, civil, and industrial construction markets specializing in using Building Information Modeling (BIM)-enabled Virtual Design and Construction (VDC). He serves as an expert consultant in design and construction cases involving BIM and VDC processes. Currently, he is on the board of directors of the national BIM Forum, and he was past chair of the Structural Engineering Institute’s national BIM Committee. He has won the Best Speaker award twice from the International Structures conference, and he was named Structural Engineering magazine’s “Top 10 Leaders in Structural Engineering,” Glass Magazine’s “Top 30 under 40,” and Building Design & Construction magazine’s “Top 40 Under 40.” He has served as PCI’s BIM consultant in that industry’s innovation initiatives.

1.0 Three Coordinates

Because we will be working in Revit, it is important to understand coordinates in Revit. There are three coordinates or origins that are relevant to working with Revit:

1.1 Revit's System Origin

With a Revit project open in a plan view, type "RH" or click the "Reveal Hidden Elements" button at the bottom left of the view window. This will reveal the location of both Revit's System Origin and the Project Base Point. If using a template, these items should appear near the center of the default elevation view locations.

Revit's System Origin cannot be moved. It serves as a constant reference point across all Revit projects.

1.2 Project Base Point

The Project Base Point is an automatically generated reference point like Revit's System Origin, but the Project Base Point may be moved like any other hidden element. This can be helpful if all trades wish to coordinate a uniform base point other than Revit's System Origin, but it is not designed to provide coordination across different software.

At any time, you can right-click the Project Base Point and select "Move to Startup Location" to reset the Project Base Point to Revit's System Origin.

The Project Base Point is "clipped" by default, which means that moving the base point will also move the other elements of the project with it. To move the Project Base Point independent of the other project elements, you can click the paperclip icon to toggle its clipping on or off.

1.3 Working Origin

It is often helpful to establish an origin for your project in addition to these two other origins provided by Revit. This working origin can be created and placed by the user to avoid certain complications such as drawing in negative space. Although the placement of this Working Origin is arbitrary, it is often helpful to place it at a sufficiently positive location relative to the Project Base Point so that no part of the drawing will be placed in negative space. For simplicity, we will use 100', 100', 100'

1. Expose Revit's System Origin and the Project Base Point (Figure 1)
    a. Create a new Revit project using the structural template.
    b. Open the Level 1 view
    c. Type "RH" or click the "Reveal Hidden Elements" button on the bottom left of the view window to expose Revit's System Origin and the Project Base Point.

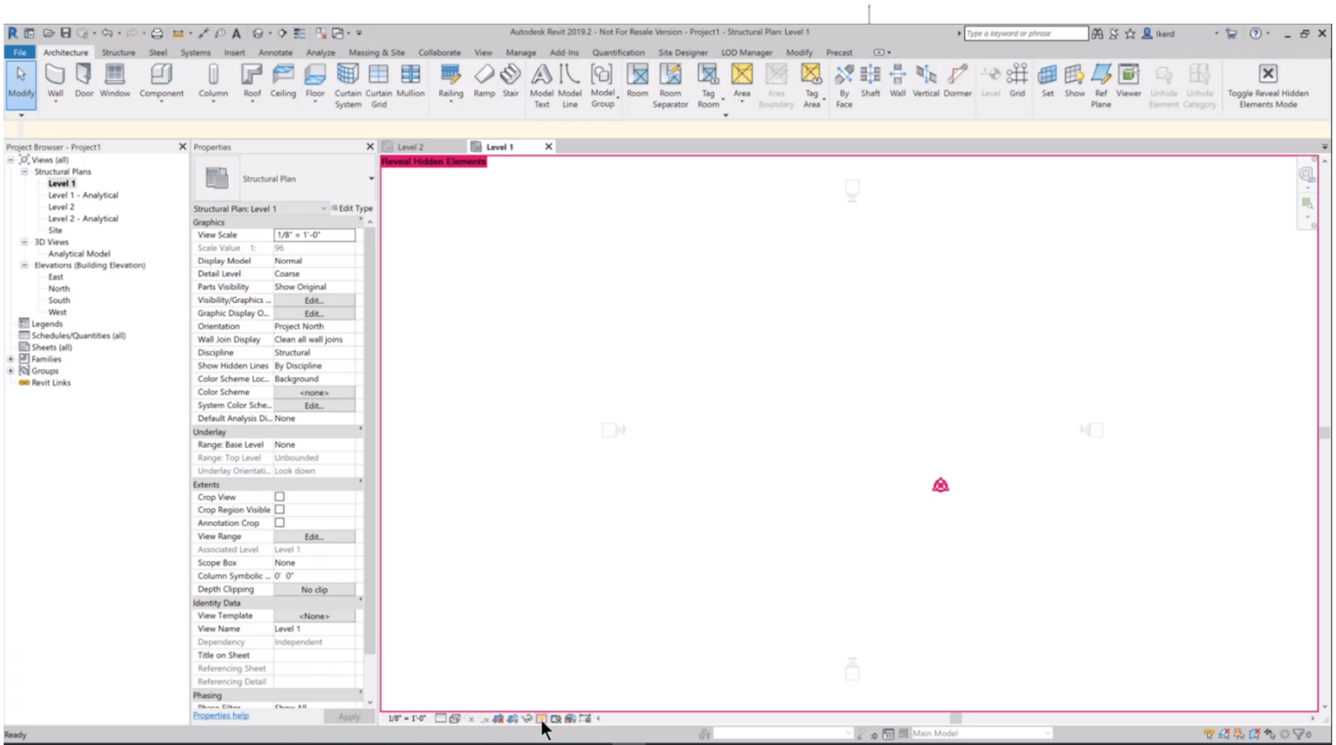


Figure 1 Exposing the Origin

2. Mark the X and Y grids based on the Project Origin (Figure 2)
    a. On the Architecture tab, on the Datum panel, click the Grid icon, or type “GR.” Draw a horizontal and a vertical gridline each intersecting the Project Base Point.
    b. Type “RH” or click the “Close Reveal Hidden Elements” button to toggle off hidden elements.
    c. Change the grid name of the horizontal grid to "XXX" and change the grid name of the vertical grid to "YYY".

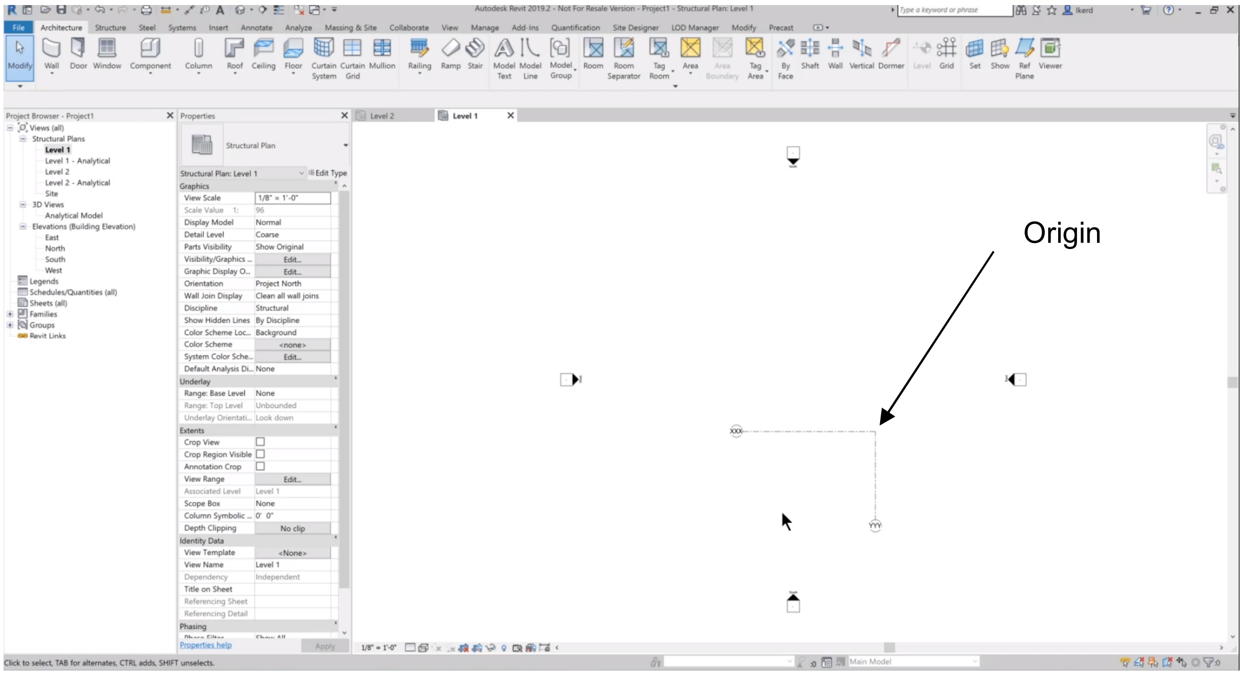


Figure 2 Gridlines

3. It is helpful to mark the Project Base Point with a set of model lines. The exact design is somewhat arbitrary, but the following instructions yield one helpful design (Figure 3):
    a. On the architecture tab, on the Model panel, select Model Line, or type “LI”. Draw a model line from the Project Base Point (now marked by the intersection of your gridlines) to the east 10’ and a model line from the Project Base Point to the north 10’.
    b. Create a circle that starts at the origin and is 5’ in diameter.
    c. Create a line from the intersection of the horizontal model line and the circle to the intersection of the vertical model line and the circle.

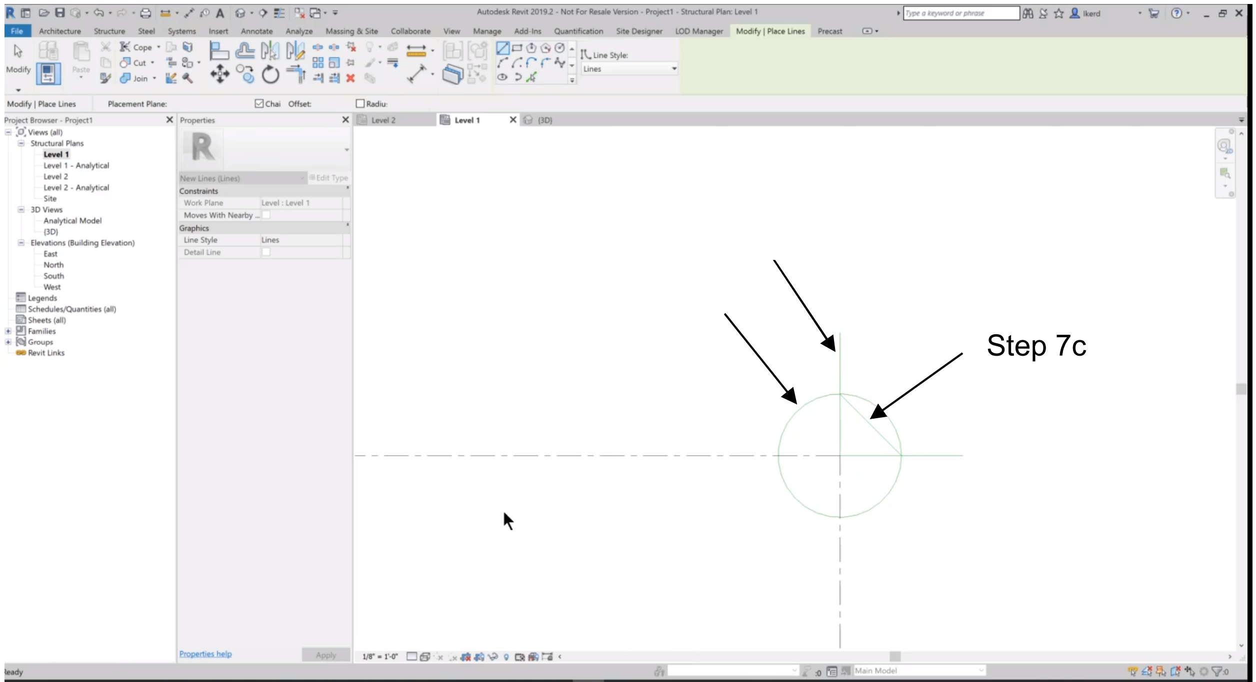


Figure 3 origin icon

d. On the View tab, on the Create panel, select the Framing Elevation in the Elevation drop down.
e. Select the "XXX" gridline to create the Framing Elevation, then click the Escape button on your keyboard.
f. Open the new elevation view by double clicking on the elevation tag.
g. In the structure tab select the model line on the Model panel, select Model Line.
h. Draw a 10' model line along the "YYY" gridline upward from Level 1 (Figure 4).

4. Create a new 3D view from the quick access toolbar to view the 3D origin (Figure 5).

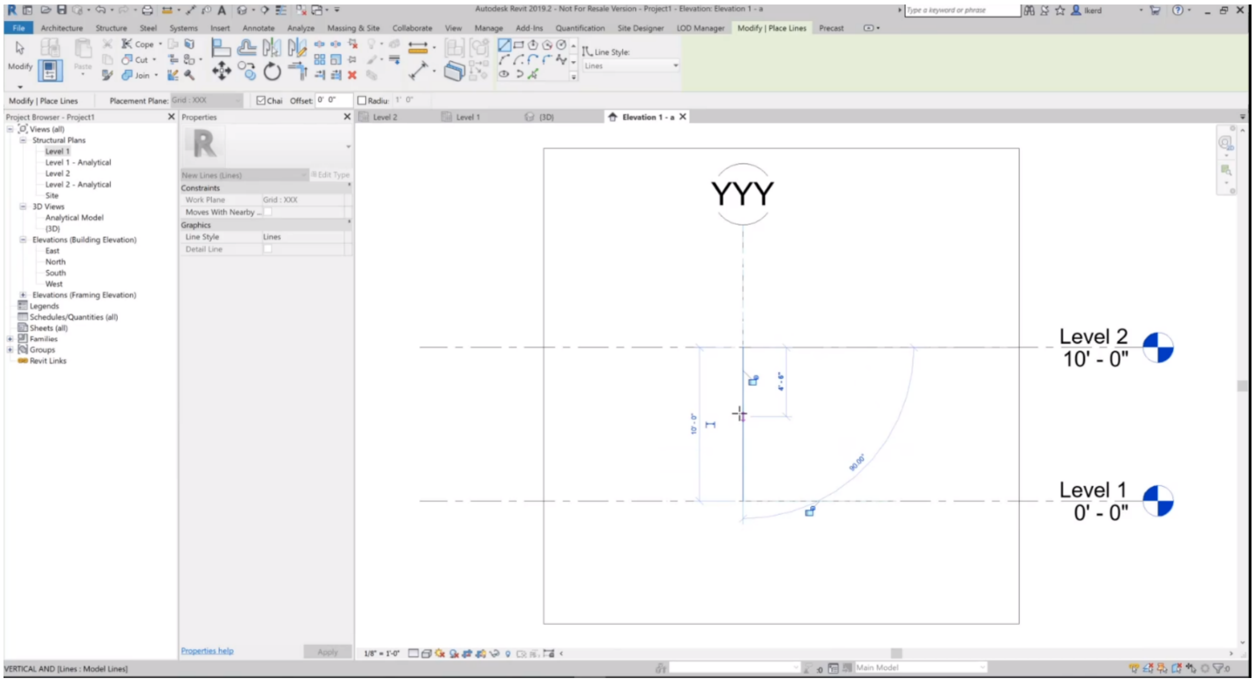


Figure 4

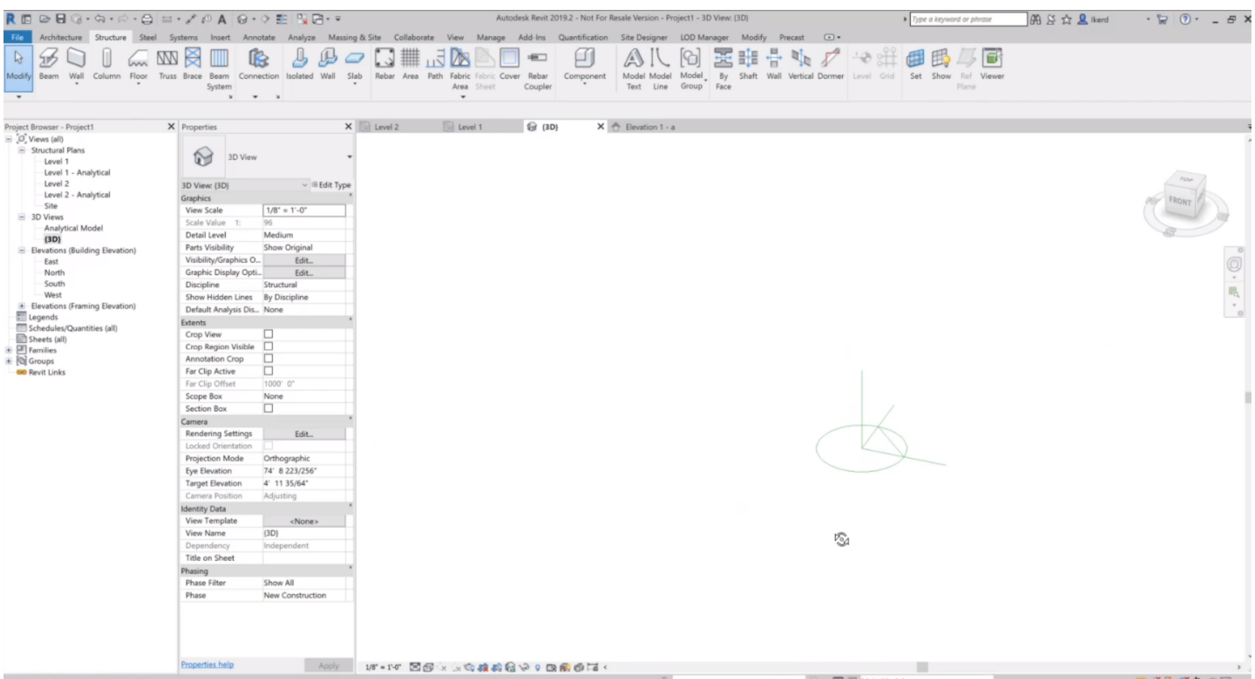


Figure 5

Now we will create the Working Origin.

5. Mark the X and Y grids based on the Project Origin (Figure 2)
    a. In the Level 1 view, select the "YYY" gridline.

b. In the modify grids tab, select copy, or type “CO”. Copy the gridline 100’ to the right (Figure 6).
c. Change the name of the new gridline to “Y1”.
d. Select the “XXX” gridline.
e. Copy the gridline upward 100’ as in step 5b.
f. Change the name of the new gridline to “X1”.
g. Extend gridlines “X1” and “Y1” until they intersect (Figure 7). The intersection of these new gridlines marks our Working Origin at 100’, 100’, 0’.

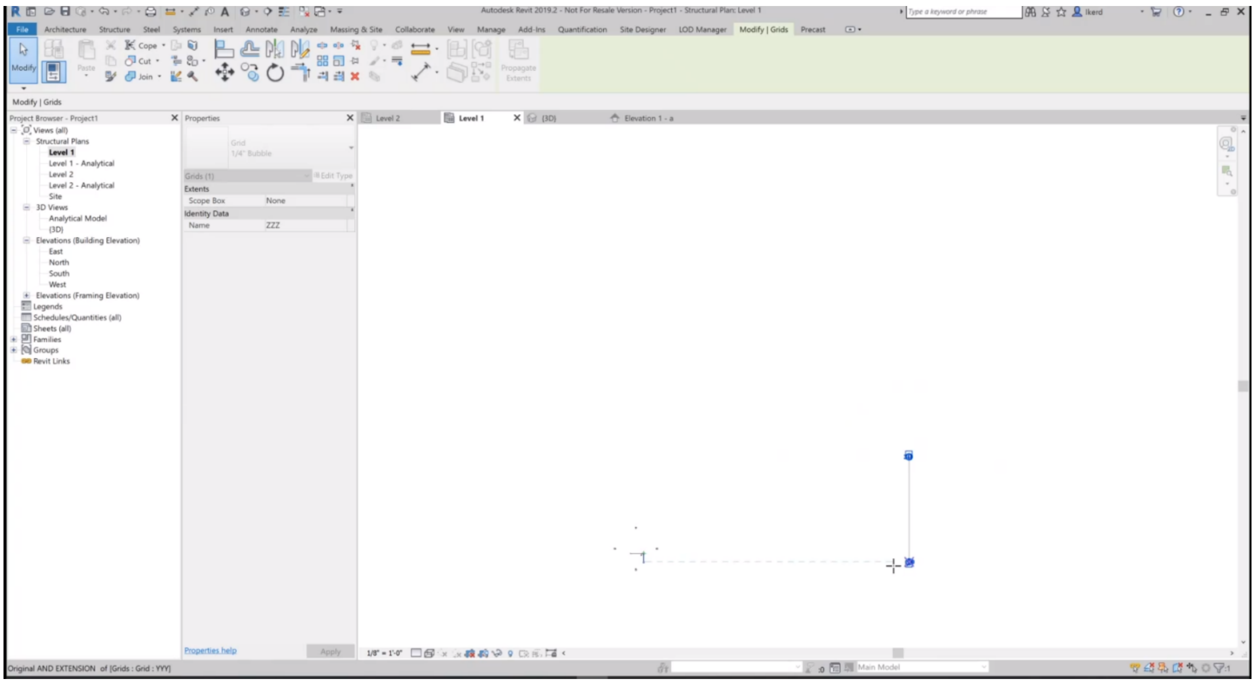


Figure 6

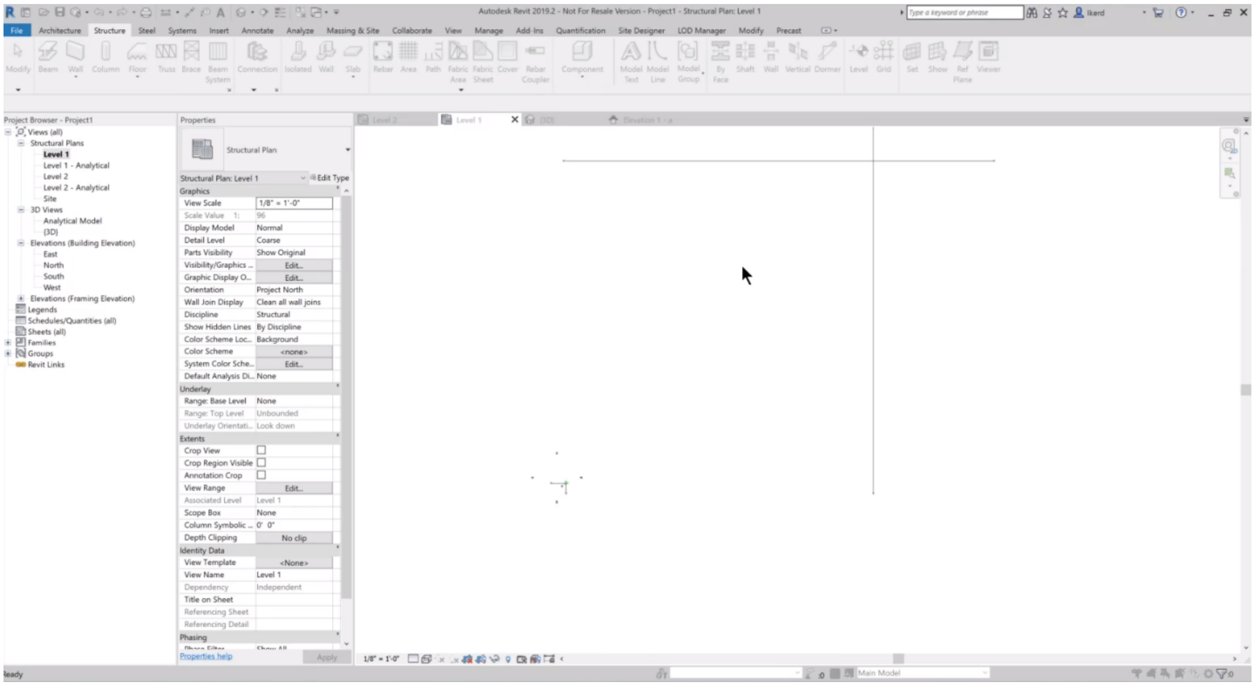
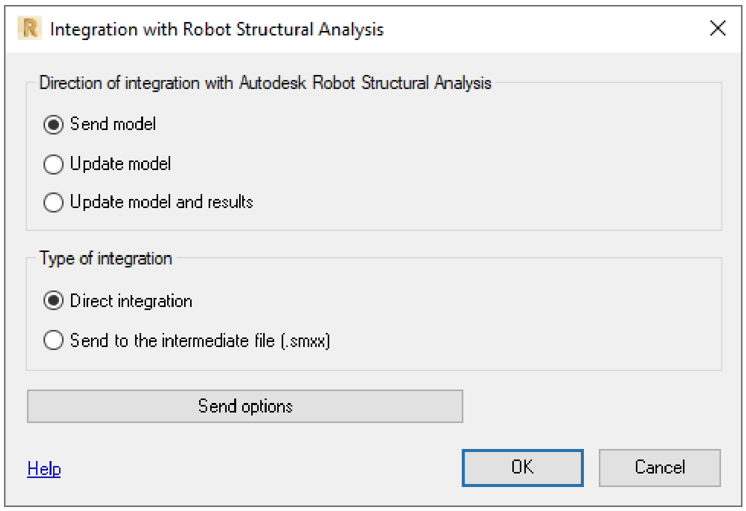


Figure 7

6. Modify levels to indicate the origin at 100’.
    a. Open an elevation view
    b. Change the name of the Level 1 level to “ORIGIN”. Click “Yes” if asked to confirm whether to rename corresponding views in a pop-up dialog box.
    c. Move the Level 2 level so it is 100’ above the “ORIGIN” level.
    d. Change the name of the Level 2 level to “LEVEL 01”.
    e. Move your gridlines so they are visible in the “LEVEL 01” view.
    f. Create a new level called “LEVEL 02” 10’ above “LEVEL 01”.
7. Mark the new working origin with a symbol of model lines as was done with the Project Base Point in step 3. Make sure the working origin marker is placed at LEVEL 01 since that will be the ground level. You may need to make another structural elevation view if you wish for the working origin to be 3D.
8. Save and close the Revit project.

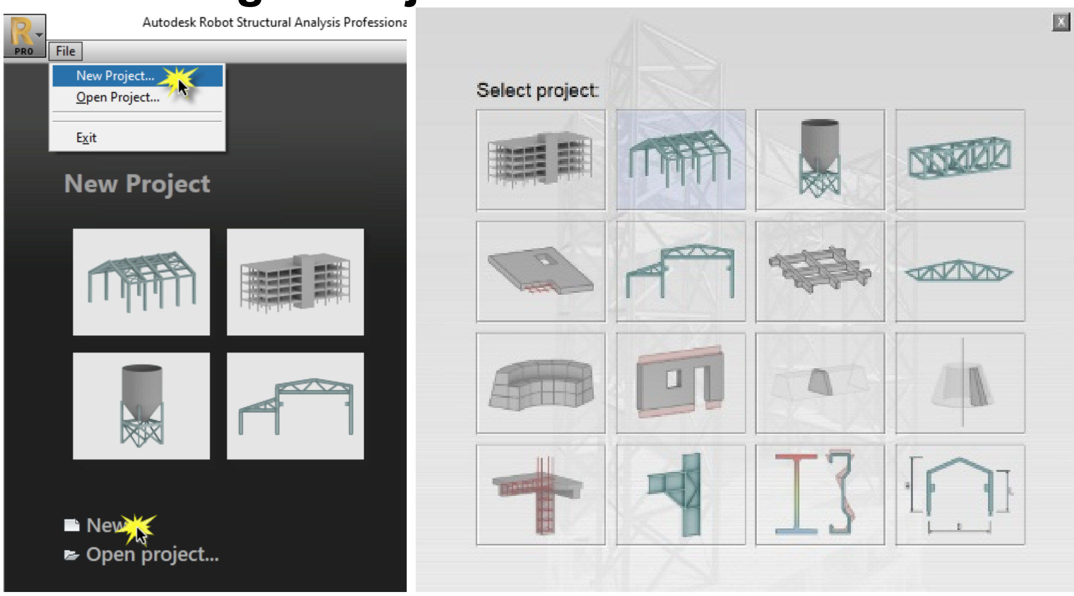
2.0 Importing Projects From Revit

1. Open the “Revit to Robot” file in Revit 2020.
2. Go to the Analyze tab. Under the Structural Analysis section, go to Robot Structural Analysis>Robot Structural Analysis Link.



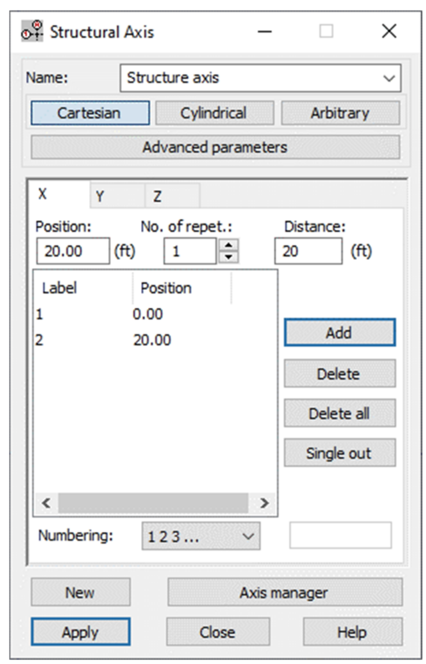
3. Using the default settings of **Send model** and **Direct integration** will send the model contents, grids, and levels over to Robot Structural Analysis Professional 2020. It does not matter what view is used to export as it will send all model content regardless. Sending the model will also automatically open Robot if it is not already and open the new export.

3.0 Starting A Project



Start a new project by selecting the **Frame 3D Design**  structure type or click new and select the **Frame 3D Design** structure type.

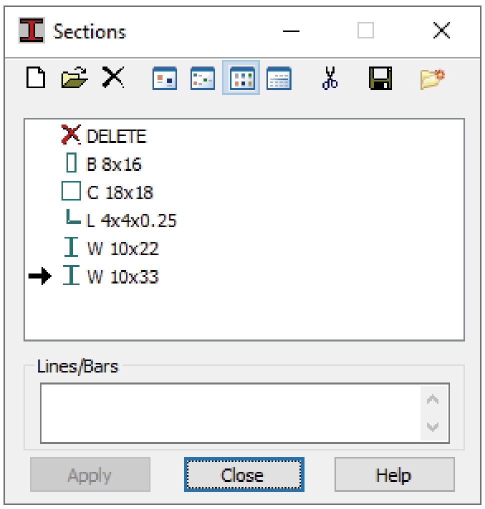
3.1 Structural Axis

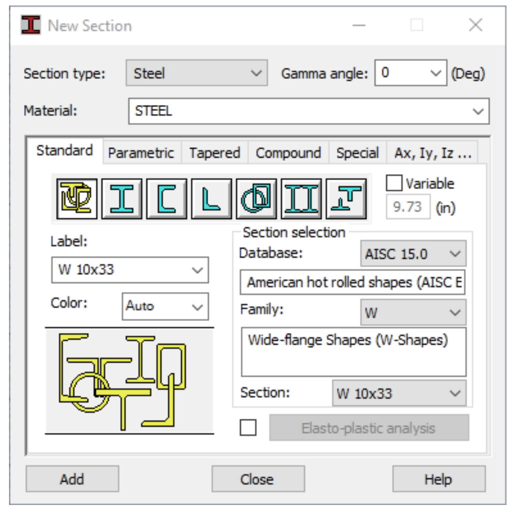


1. On the right-hand toolbar, select **Axis Definition**, or use **Geometry>Axis Definition**.
2. On the **X** tab in the **Structural Axis** dialogue box, set the number of **Repetitions to 1**, and the **Distance** to **20 feet**. Confirm that **Position** is still at **0 Feet** for the first axis. Set the **Numbering** to **Define** and label the axes **Y1** and **Y2.** Add the axes.
3. Repeat this on the **Y** tab by placing axes at both 0 ft and 20 ft. **Define** the axes as **X1** and **X2.**
4. Repeat this one last time on the **Z** tab by placing an axis at 0 ft, 1 ft 6 in (1.5 ft), and 11 ft 6 ¾ in (11.5625 ft). Set the number of **Repetitions** to 1 and the **Distance** to **1.5 feet**. **Add** the axis.

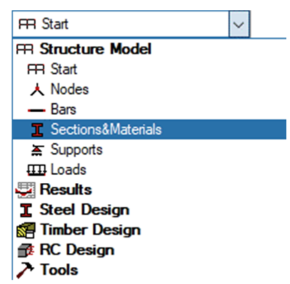
Change the **Distance** to **10.0625 feet** and **Add** it. **Define** the axes as **L01**, **L02**, and **L03**. Click **Apply**.

3.2 Adding Sections





5. Open the **Bar Sections** dialogue box on the right-hand toolbar or expand the layouts drop-down menu and select **Sections and Materials**.

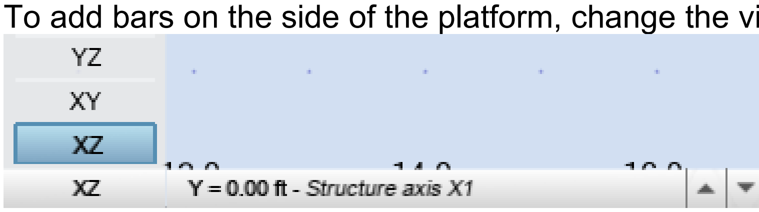


6. Click **New Section Definition**. On the **Standard** tab, set the **Database** to **AISC 15.0**, the **Family** to **W**, and the **Section** to **W10X33**. **Add** the section. Also add the **W10X22** and **L4X4X1/4** sections. Close the New Section dialogue box.

7. Close the **Section** dialogue box or expand the layouts drop-down menu and select **Start**.

3.3 Bar Placement

8. To add bars on the side of the platform, change the view to **XZ** on axis X1.



9. Open the **Bars** dialogue box on the right-hand toolbar or expand the layouts drop-down menu and select **Bars**.

10. Set the **Bar Type** to **Simple Bar** and the **Section** to **W10X33**.

11. Draw two **W10X33** along axes **Y1** and Y**2** from axes **L02** to **L03**. Draw a **W10X22** on axis **L03** from axes **Y1** to **Y2**. The result should resemble the image above.

12. Change the view axis to **X2**. Add the bars along the same axes for the other side of the platform.

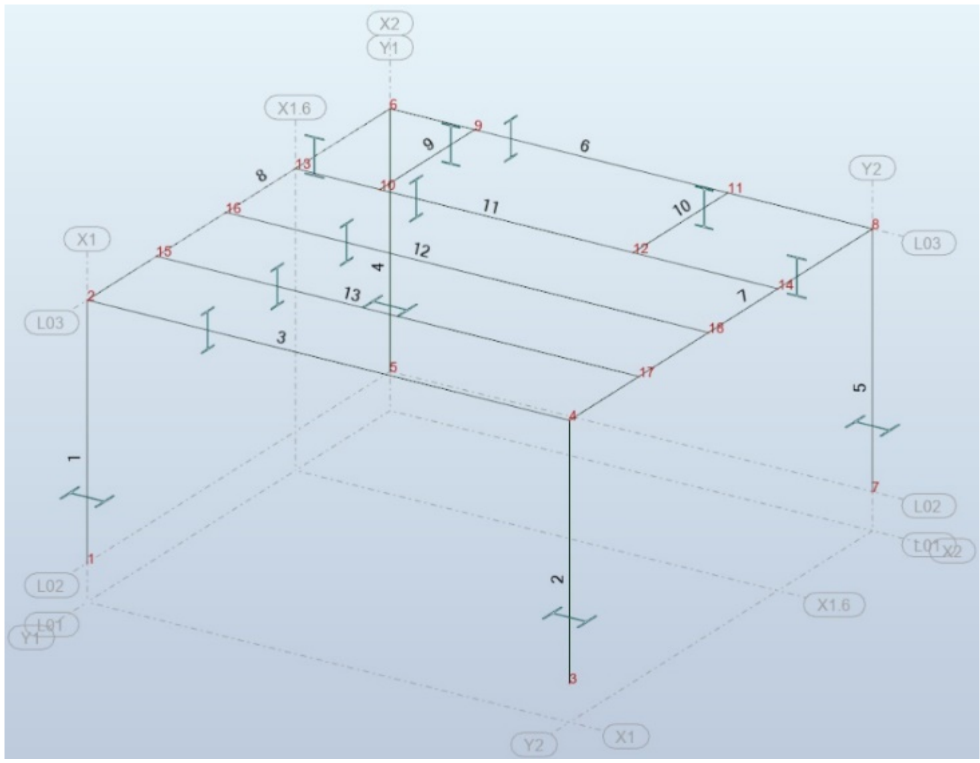
13. To add the other members on top change the view to **XY** and Structure axis to **L03**. Add the **W12X26** and **W8X10** sections though the **Sections** button or the **Sections and Materials** view mode.

14. Add the **W12X26** bars along axes **Y1** and **Y2** from **X1** to **X2**.

15. Open the **Structural Axis Definitions** dialogue and add axes **Y1.3** and **Y1.4** in the X tab at **Positions** of 3.5' and 14', respectively. Also add axis **X1.6** in the Y tab at the **Position** of 13.75'.

16. Using the **Bars** dialogue box add **W8X10** bars along **Y1.3** and **Y1.4** from **X1.6** to **X2**. Add a **W10X22** on axis **X1.6** from **Y1** to **Y2**.

17. Open the **Nodes** () dialogue or the **Nodes** view mode. Place nodes at the following locations:
    a. 0, 4.5833, 11.5625
    b. 0, 9.1666, 11.5625
    c. 20, 4.5833, 11.5625
    d. 20, 9.1666, 11.5625



18. Pressing the **Node Numbers** button at the bottom left of the view will make the nodes visible. With the nodes visible add **W10X22 Bars** from gridline **Y1** to **Y2** between the newly created nodes

19. Using the **Bar Sections** dialogue, add the last member we're using of a **L4X4X1/4.**

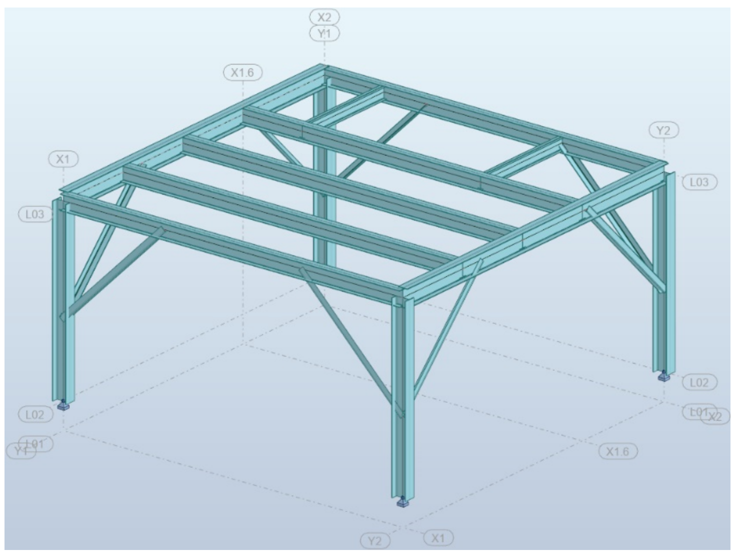
20. To get the last members added, add nodes at the following locations:
    a. 0, 0, 5.5625
    b. 6, 0, 11.5625
    c. 14, 0, 11.5625
    d. 20, 0, 5.5625
    e. 20, 6, 11.5625
    f. 20, 14, 11.5625
    g. 20, 20, 5.5625
    h. 6, 20, 11.5625
    i. 0, 20, 5.5625
    j. 0, 14, 11.5625
    k. 0, 6, 11.5625

21. With the nodes now placed, add the L4X4X1/4 **Bars** from the nodes on the columns to the nodes at L03.

3.4 Supports

22. Open the **Supports** tool by using the button on the right, Geometry>Supports, or switch to the **Supports** view mode.

23. Place pinned supports at the base of all four W10X33 columns by one of the following methods:
    a. Select the Pinned option and click the bases of the columns in the view.
    b. If using the **Supports** view or if you open the Supports table through View>Tables>Supports, enter the nodes of the base of the columns (1, 3, 5, 7) in the table in the **List of nodes** column and the Pinned row.
    c. Optional: select the **Section shapes** button ( )at the bottom left of the view window to get the appearance of the image below.



3.5 Loads

24. The loads we'll be applying will require a cladding in order to be applied.

25. Go to Geometry>Claddings.

26. Using the rectangle option, pick points along the top of the platform using either a XY view at L03 or a 3D view. Or use the point coordinates from the image on the right.

27. Move to either the **Loads** view mode or go to Loads>Load Types. Within the **Load Types** dialogue add a dead load by setting **Nature** to dead and click **Add**. Setting the nature will automatically set the **Name** and **Label** to **DL1**.

28. **Add** both a live load type and wind load type.

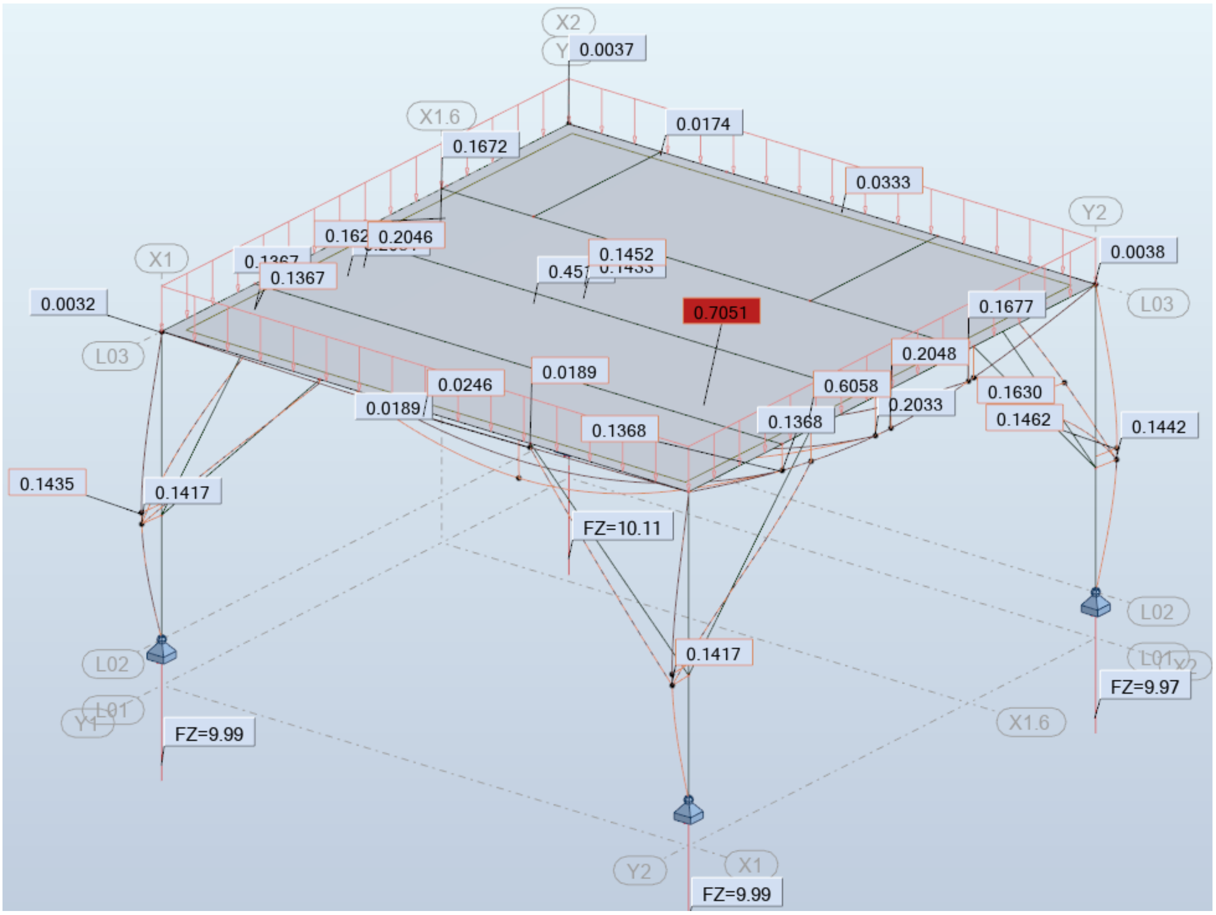
29. Use the Loads table through the **Loads** view mode or go to Loads>Load Table. Put the **Case** to **1:DL1**, the **Load type** to **surface on object**, **List** item 22 for the cladding. And lastly set PZ=-0.1.

30. In the next row set the **Case** to **2:LL1**, the **Load type** to **surface on object**, **List** item 22 for the cladding. And lastly set PZ=-0.1.

31. To run these newly defined loads, use the Calculations button () or go to Analysis>Calculations.

32. To see the effects of the loads, change the view mode to **Results** or open Results>Diagrams for bars.

33. Within the Diagrams dialogue go to the Deformation tab. Select the **Exact deformation for bars** option. Then on the Reactions tab check the options for FZ and Descriptions. And lastly on the Parameters tab change the **Diagram description** to the labels option. Click apply.



Note the units will be displayed in the title bar of the view.