ES17637-L

Integrating Structural Design and Analysis: The Basics of a Revit-Robot Structural Analysis Workflow

Aaron M. Vorwerk
AIA, NCARB, EIT, LEED AP BD+C

Exercise Guide

Please reference this document during our hands-on exercises. It has been formatted to position adjacent to your Revit/RSA/browser window.

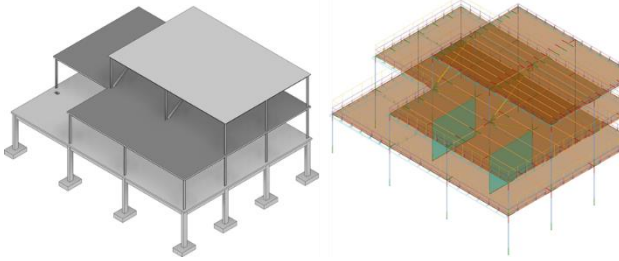
Exercise 1: Exploring the analytical model in Revit

In this exercise, we'll develop an understanding of some of the differences between the physical and analytical models in Revit. Then we'll look at the user-specified settings for analytical model verification.

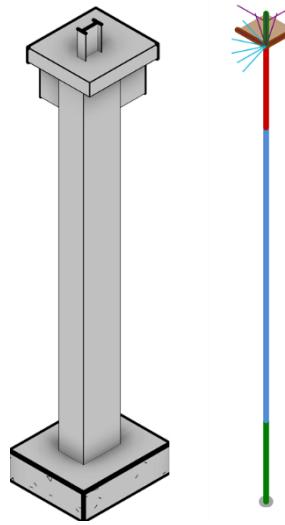
Exercise 1: Viewing the analytical model

1. Open **01 – Simple Building.rvt**.

2. Open the **View 1 – Analytical** view and tile side-by-side with the default 3D view.



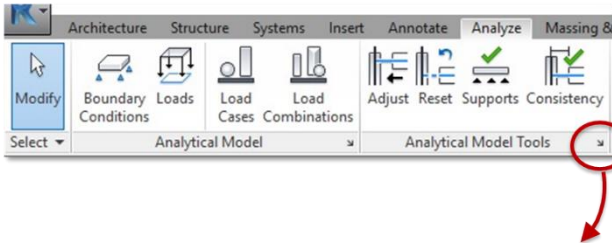
3. Select an element (e.g. a column) in the default 3D view and review its properties. Select the same element in the analytical view and note the differences.

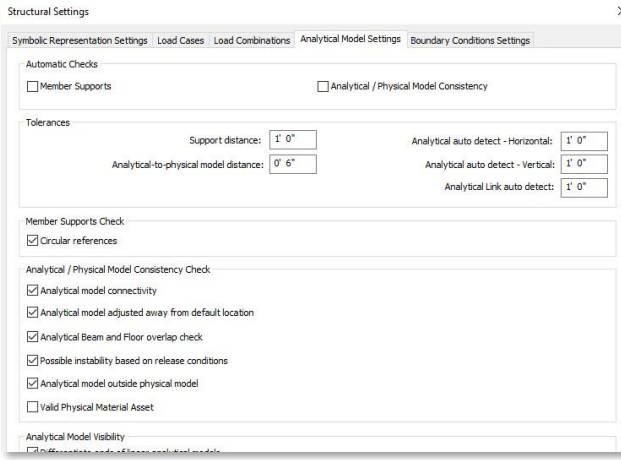


Exercise 1: Analytical Model Settings

4. Switch to the **Analyze** tab.

5. From the **Analytical Model Tools** panel, open the **Structural Settings**.





6. Notice the options available here, including **Automatic Checks** for supported elements and model consistency, as well as **Tolerances** defining user preferences.

7. Close the dialogue box and select the **Check Member Supports** button. Notice the ‘warning’ that appears to report that no unsupported elements have been detected.

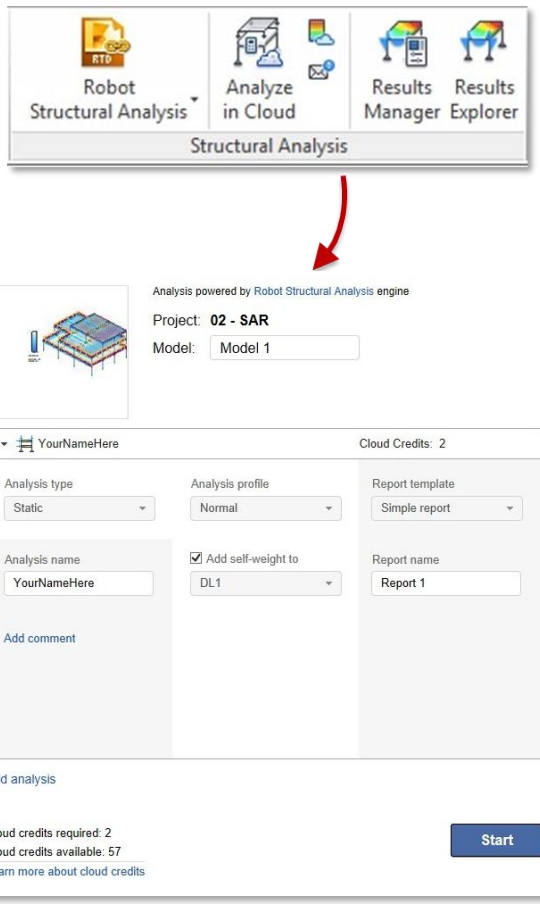
Exercise 2: Structural Analysis for Revit

In this exercise, we'll use the Structural Analysis for Revit tool to perform a static analysis in the cloud. We'll review the results both in a browser and in Revit.

Exercise 2: Performing a structural analysis using Structural Analysis for Revit

1. Open **02 - SAR.rvt** and locate the **Analyze** tab > **Structural Analysis** panel.

2. Select **Analyze in Cloud**; configure a **Static** analysis with the **Analysis name** and **Report name** of your choice and select **Start**. *Note: You will need an Autodesk ID with access to the Structural Analysis for Revit service and cloud credits to perform steps 2-3. If you don't have these, don't worry! You will be able to participate when we return to Revit.*



3. Open your browser and navigate to **structuralanalysis360.autodesk.com** to view the result.

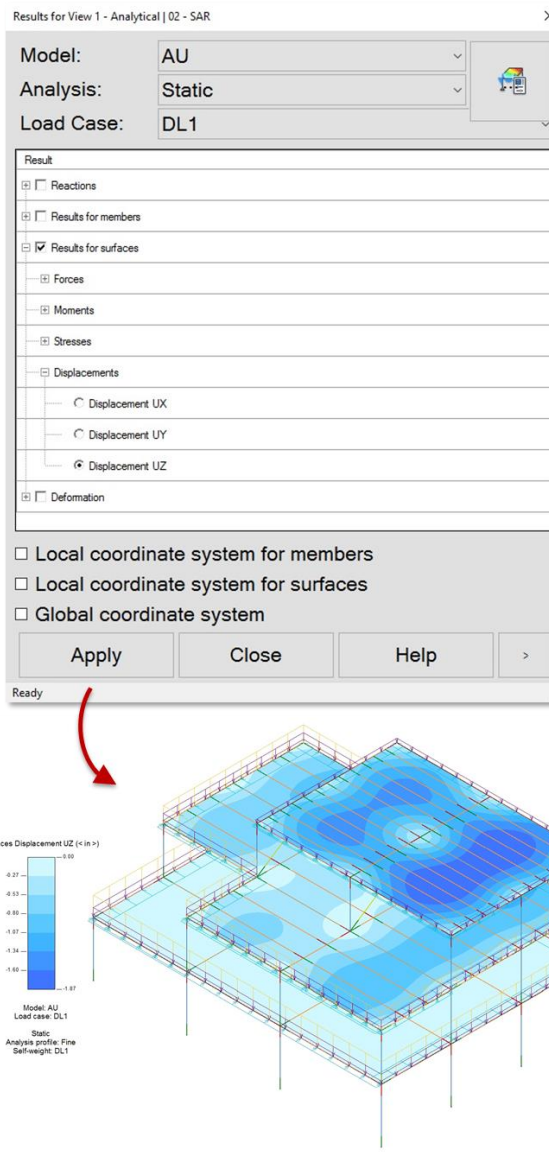
Exercise 2: Performing a structural analysis using Structural Analysis for Revit

4. In Revit, select **Results Manager** on the **Structural Analysis** panel.

5. Select the AU static analysis that is listed as “in project”.

6. Click the **Explore** button to open the **Results Explorer**.

7. Choose **Results for surfaces** > **Displacements** > **Displacement UZ** and select **Apply** to view results.



Exercise 3: Code group design for steel members using Revit and RSA

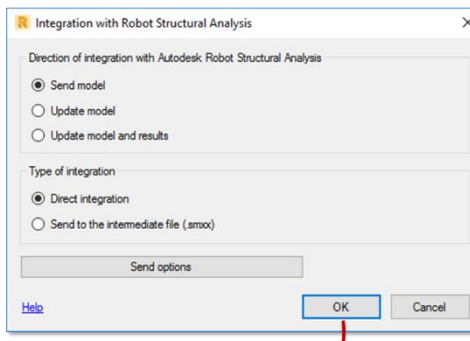
In this exercise, we'll start by sending our model from Revit to RSA. We'll then prepare and perform analysis on the structural model in RSA. We'll review the results of the analysis and perform code group design on a selected set of members. Finally, we'll send the model back to Revit and confirm that the round-trip was executed successfully.

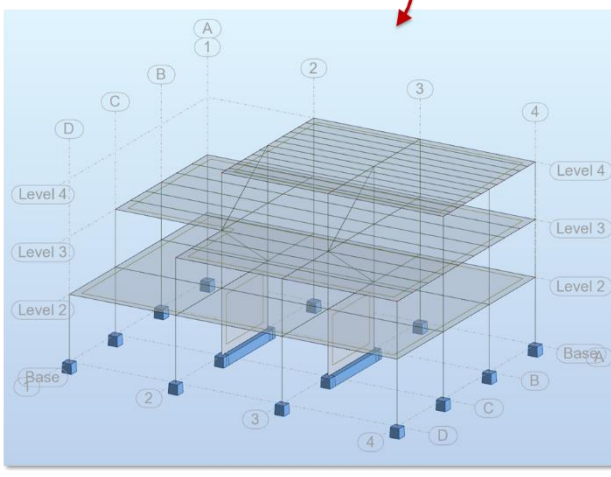
Exercise 3: Send the Revit model to RSA

1. Open **03 - Start.rvt** and locate the **Analyze** tab > **Structural Analysis** panel.

2. Select **Robot Structural Analysis** > **Robot Structural Analysis Link**.

3. Leave default options and select **OK**. RSA will open and begin importing the Revit model data.

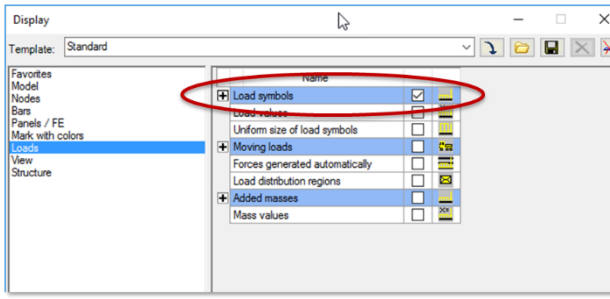




4. Click **Yes** to the pop-up dialog to view the Events Report; close the report after viewing.

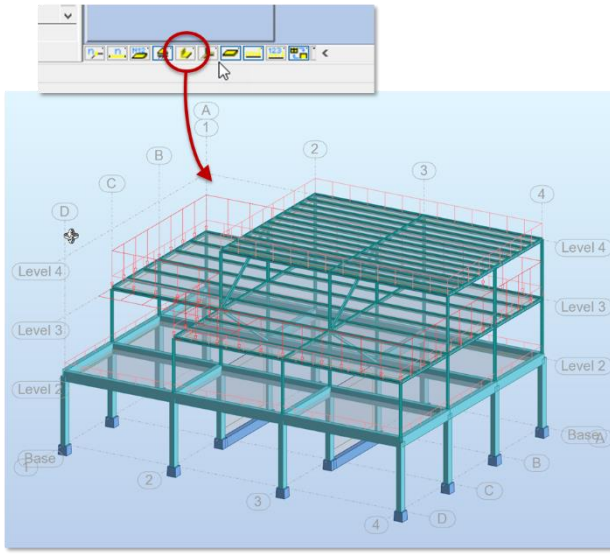
Exercise 3: Configure the model display in RSA

5. If load symbols are not displayed, go to **View** tab > **Display** > **Loads** and toggle **Load symbols** off and on again, clicking **Apply** each time.



6. Select **OK** to exit that dialog.

7. Element visibility may also be controlled using the toolbar at the bottom left edge of the drawing window, similar to the View Control toolbar in Revit. Use this toolbar to toggle the display of **Section shapes**.

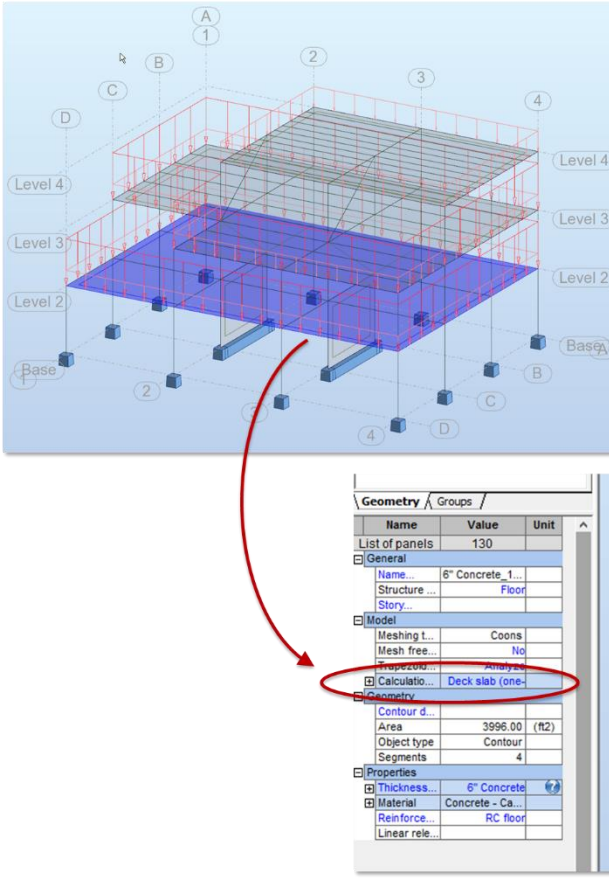


Exercise 3: Adjust analytical geometry

8. Select the analytical floor at Level 2.

9. In the Properties Inspector (similar to the Properties Palette in Revit) at the left side of the screen, change the **Calculation model** for this floor from **Shell** to **Deck slab (one-way)**.

10. Repeat for the floor at Level 3.

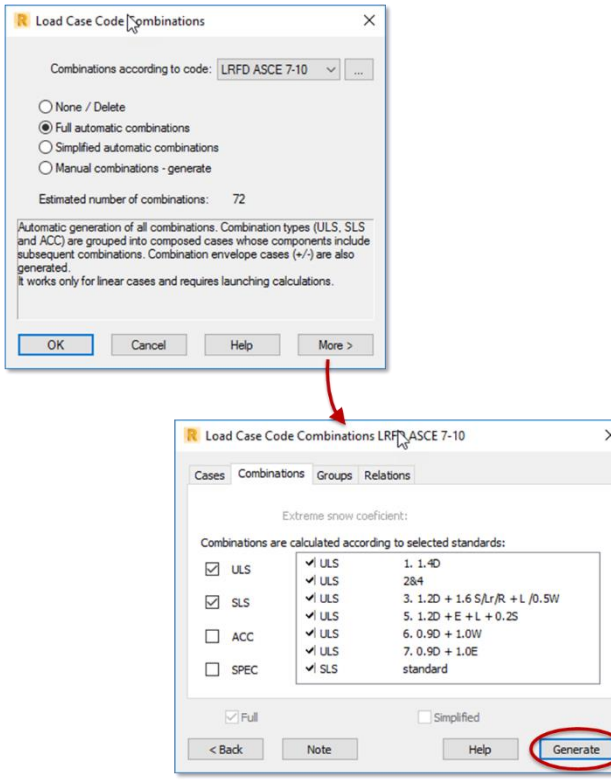


Exercise 3: Generate load case combinations

11. Select **Loads** tab > **Automatic Combinations** to open the Load Case Code Combinations dialog.

12. Select **Full automatic combinations**, then click **More** to view the combinations in more detail.

13. Select **Generate** to build out the load combinations list per ASCE 7-10.

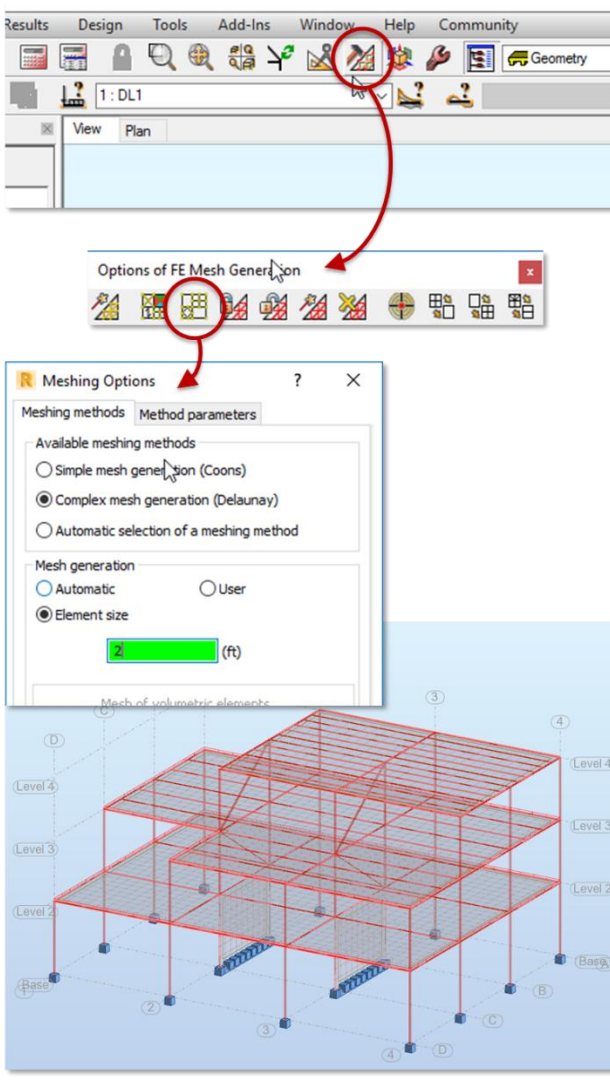


Exercise 3: Create finite element mesh

14. Click the **Options of FE Mesh Generation** icon to open this toolbar, then choose **Meshing Options** (select **Yes** to the pop-up message to select all panel elements).

15. Select **Complex mesh generation (Delaunay)** and set the **Element size** to 2 feet. Select **OK**.

16. Select **Generation of calculation model** to create the FE mesh.



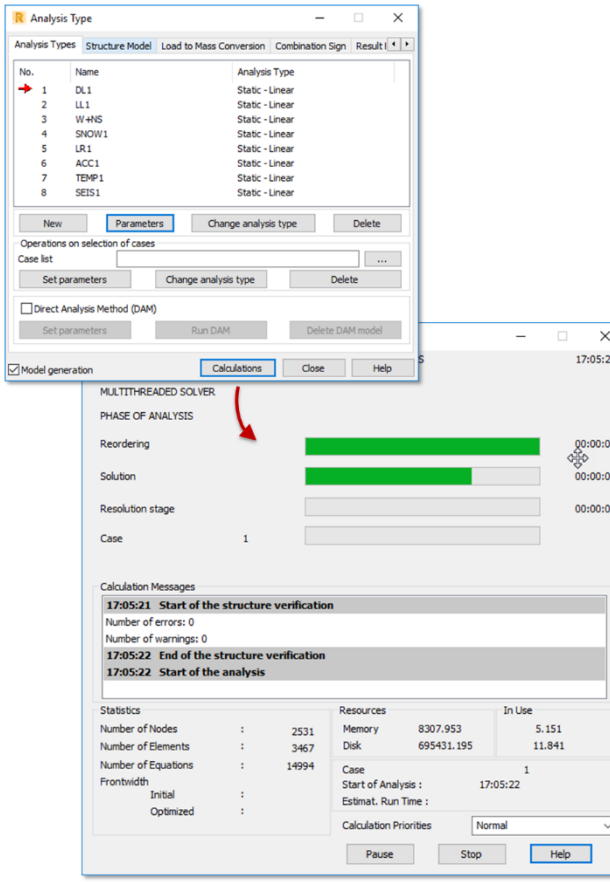
17. Select **Mesh Freeze** to store this mesh; then close the toolbar.

Exercise 3: Perform analysis

18. Time for analysis! If you've gotten lost along the way, open **04 – Analysis.rtd** to catch up.

19. Go to **Analysis** tab > **Analysis Types** to open this menu. Observe additional capabilities under the **New** and **Parameters** buttons.

20. Select **Calculations** to run the analysis. Once completed, a green light at the bottom of the screen indicates that current results are available.

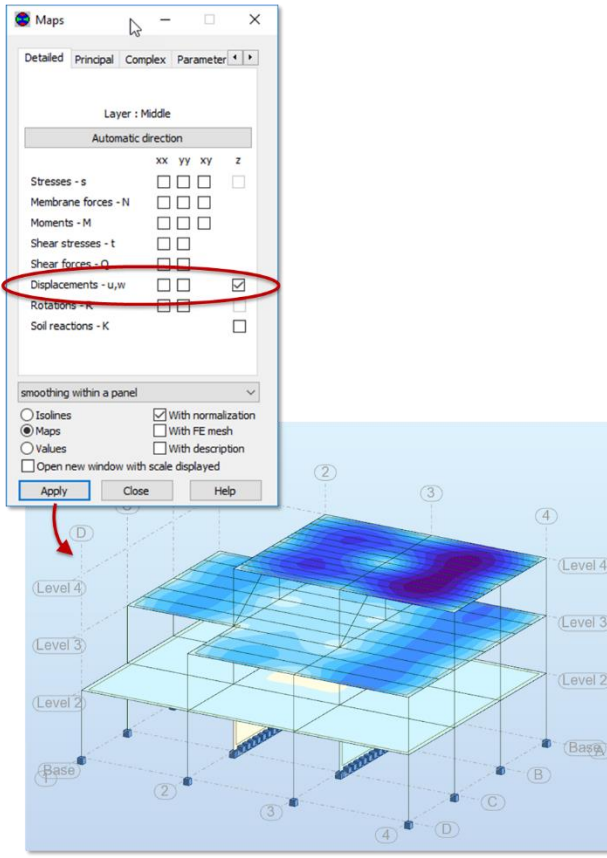


Exercise 3: View results

21. Select **Results** tab > **Maps** to open this dialog.

22. Select the **z** direction for **Displacements – u,w** and select **Apply**.

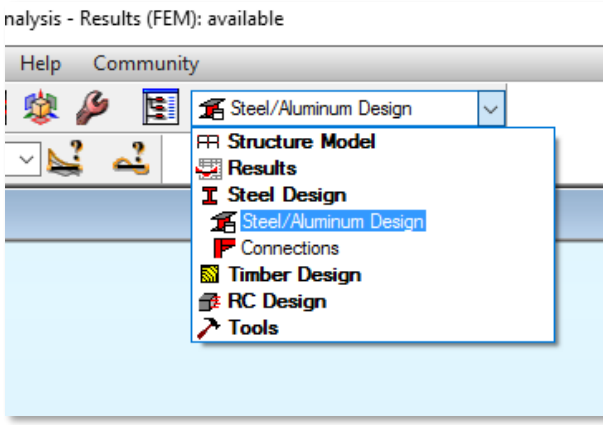
23. Note the color mapping in RSA is similar to the results previously explored in Revit. The interior beams of the top deck indicate the largest displacements; we'll make them the focus of this exercise.



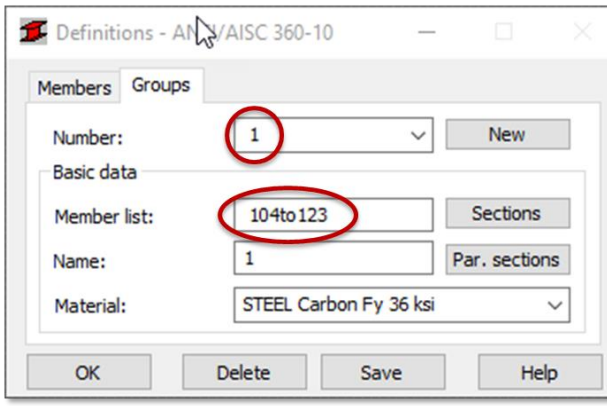
24. Deselect the **z** checkbox and select **Apply** again to remove the map.

Exercise 3: Configure code group for design

25. For this step, let's change our RSA layout. Locate the **Layouts** toolbar and change from **Geometry** to **Steel/Aluminum Design**.



26. Proceed to the **Groups** tab of the **Definitions** dialog. Click **New** to create a new code group and enter members **104 to 123** (the interior beams from the top deck) in the **Member list**. *Note: RSA offers many selection methods, but we are directly entering known bar numbers in this example.*

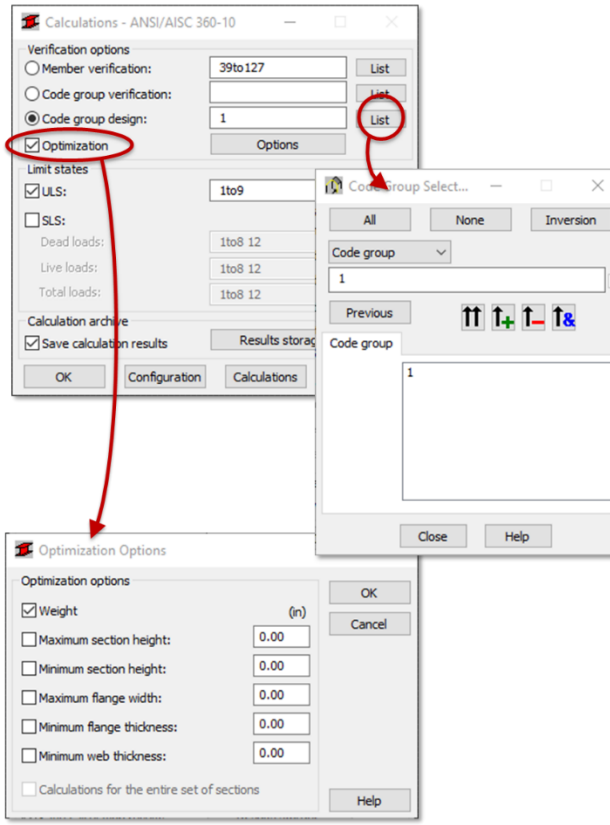


27. Select **Save.**

Exercise 3: Perform code group design with optimization

28. Proceed to the **Calculations** dialog and select the **Code group design** option. Enter **1** or use the **List** button to find and select group 1 using the “up” arrow icons.

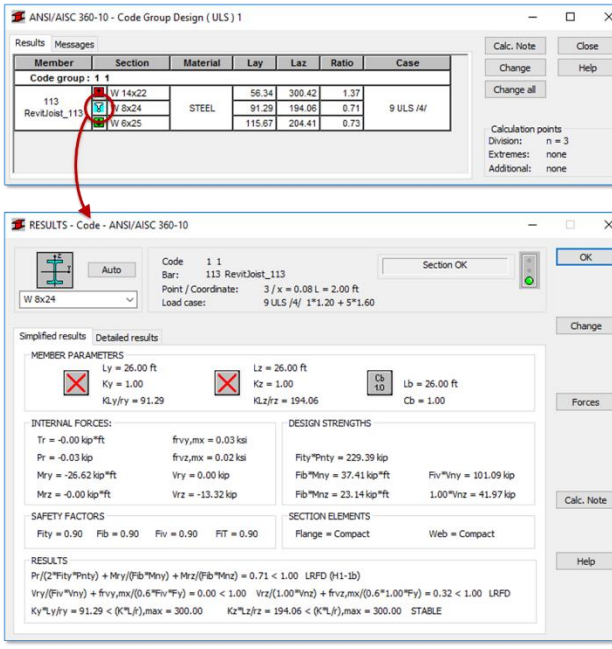
29. Select **Optimization** and check the **Weight** option. Select OK to close this dialog. If you’re behind, open **05 – Design.rtd** to catch up.



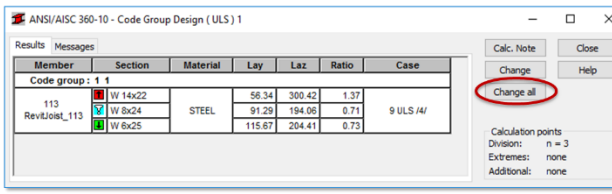
30. Select **Calculations** to perform code group design for the selected settings.

Exercise 3: Select optimal sections

31. The **Code Group Design** module highlights the optimal section for the group (W 8x24 here). Click on the icon next to this section to view results; select **OK** to return.



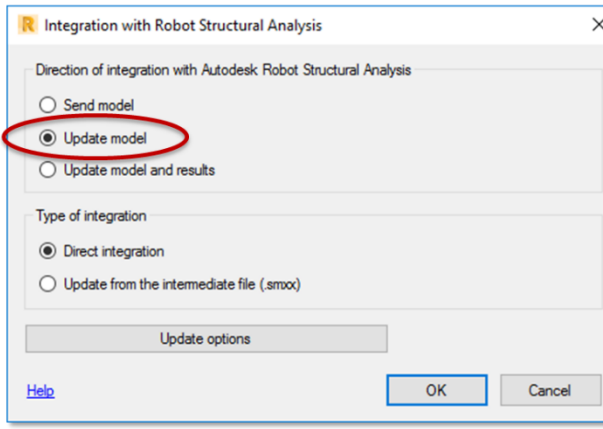
32. In the **Code Group Design** dialog, select **Change all** to resize the sections. **Close** the dialog and **Cancel** saving the calculation results.



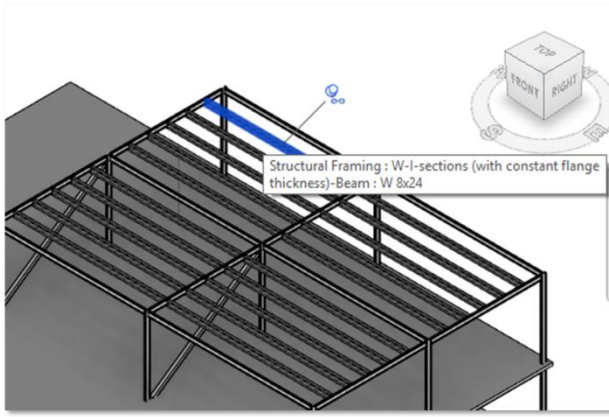
Exercise 3: Update the Revit model

33. Use Alt-Tab to switch back to Revit. As before, select **Robot Structural Analysis** > **Robot Structural Analysis Link**.

34. This time, choose **Update model** and click **OK**. Alternatively, instead of direct integration, you can choose **Update from the intermediate file** and select **06 – Update.rtd**. Ignore the events report.



35. Open the default **{3D}** view, hide the top floor slab (by selecting it and typing **HH**), and select one of the interior beams to confirm its new size in the Properties Palette.



Congratulations…YOU DID IT!!!