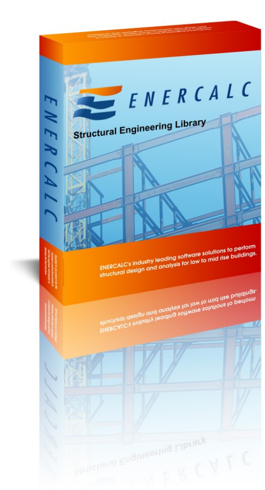
ENERCALC for Revit
User's Manual



ENERCALC, LLC

ENERCALC for Revit®

A product of ENERCALC, LLC

ENERCALC Link for Autodesk® Revit®
Build 20

Publisher

ENERCALC, LLC

Managing Editor

Seth Roswurm, S.E., P.E.

ENERCALC, LLC

400 W Broadway St
STE 101-515
Missoula, MT 59802
(949) 614-0689
(800) 424-2252

Sales: info@enercalc.com
Support : support@enercalc.com
Web : enercalc.com

ENERCALC for Revit User's Manual

Table of Contents

Part I

1 Introduction

Last Revised: 25 August 2022

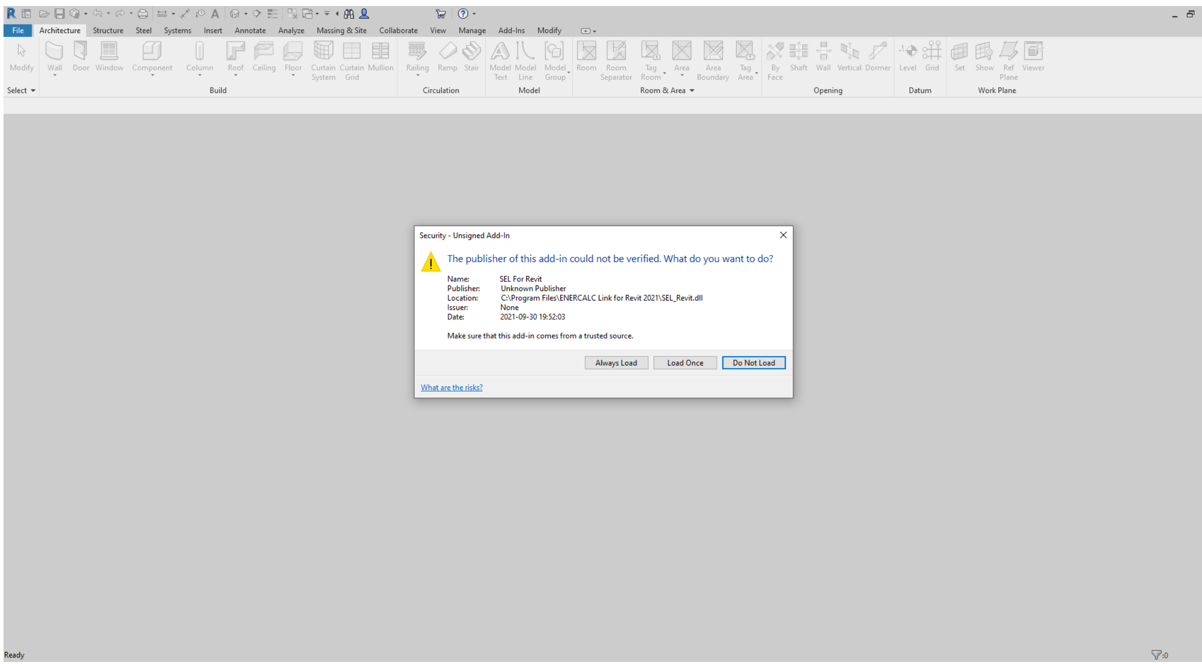
Note: Autodesk and Revit are registered trademarks of Autodesk.

Part II

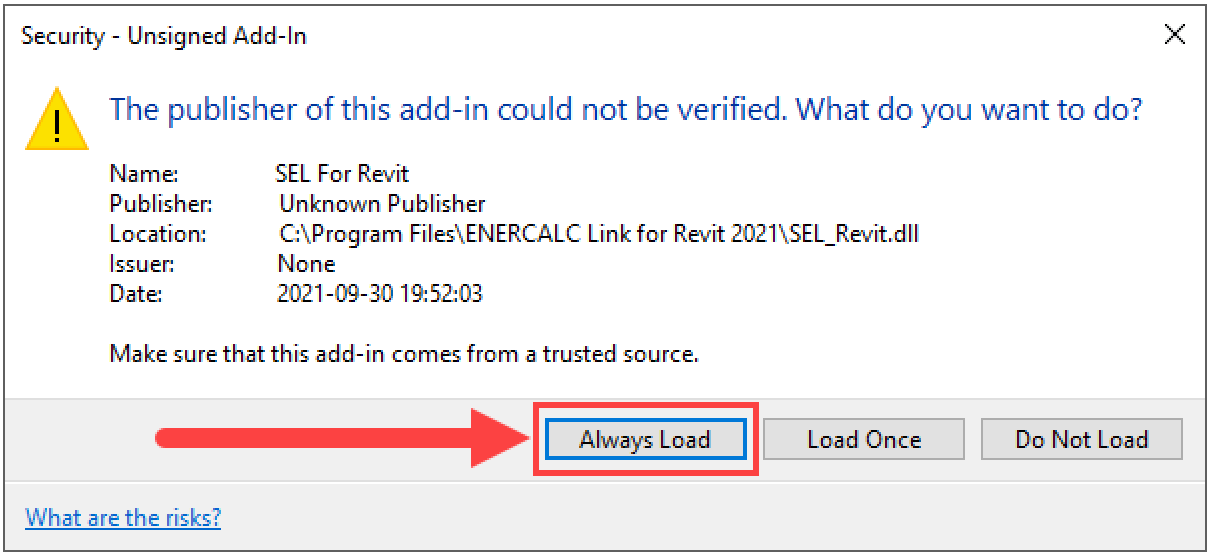
2 Installing ENERCALC for Revit

ENERCALC for Revit is powered by a Revit add-in that installs to one or more versions of Revit, as desired by the user. Upon purchasing a subscription or initiating a trial, you will receive an installation download. Prior to beginning installation, close all instances of whichever versions of Revit will be used in conjunction with ENERCALC for Revit. You may begin the installation process by double-clicking the installer .exe and following the prompts in the installer.

Upon completing installation, the first launch of Revit afterwards may result in the following message.



Select the “Always Load” option to allow Revit to trust the ENERCALC for Revit add-in in the future and avoid subsequent notifications.



2.1 Manual Build Updates

In some specific cases, the ENERCALC support team may suggest to resolve a specific bug or feature request using a provisional pre-release build of ENERCALC for Revit.

In such cases, users should be aware that the pre-release build may potentially contain both untested features and unresolved bugs not found in the current release build of ENERCALC for Revit.

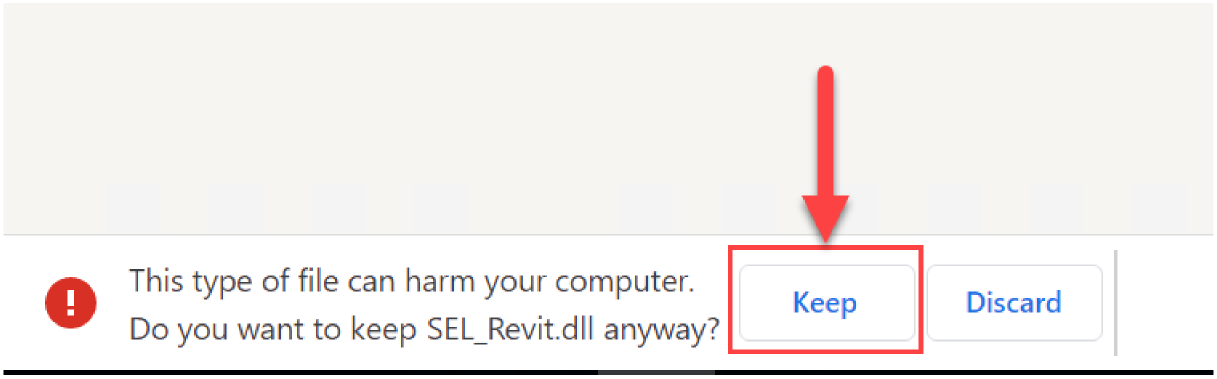
If the user is comfortable with the inherent risks of working with a pre-release build, the support team will provide a download link or attachment to access the new build. The new build will not be delivered in an executable (.EXE) updater. Instead, it will be a single assembly file (.DLL) to be placed in a specified location on the user's machine. Once the file has been downloaded, refer to these instructions to install the provisional build.

NOTE: Due to Windows protections on "Program Files" folders, you will need to have administrator permissions to perform this process. If elevation to admin is out of your control, contact your IT team in advance for assistance.

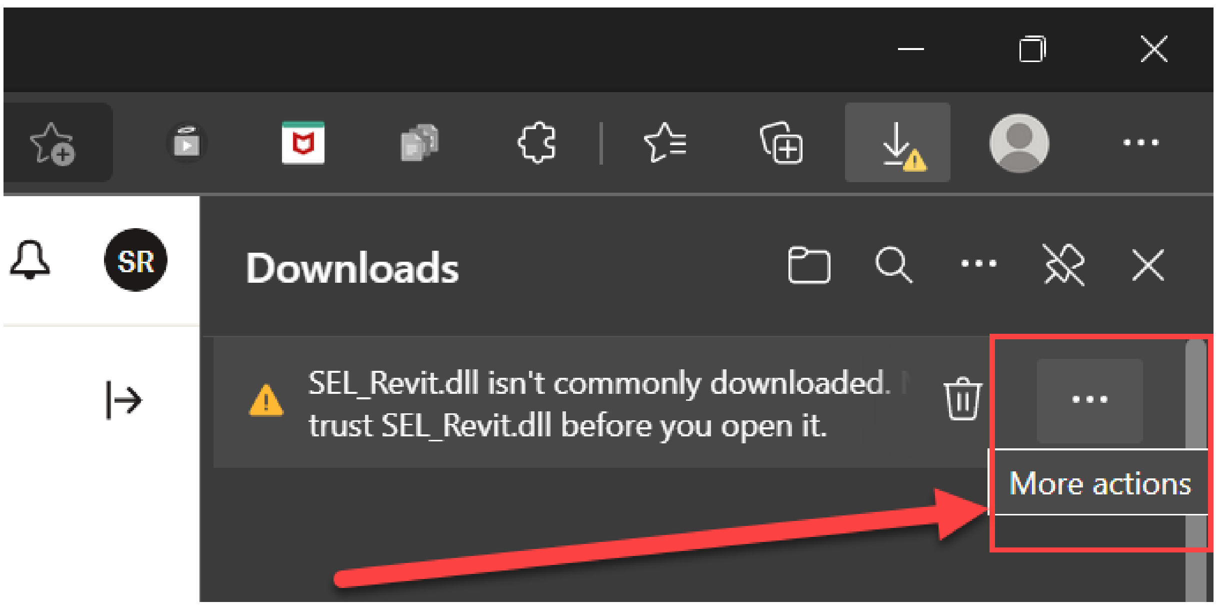
NOTE: Make sure that all instances of Revit for the applicable version year are closed prior to beginning this process.

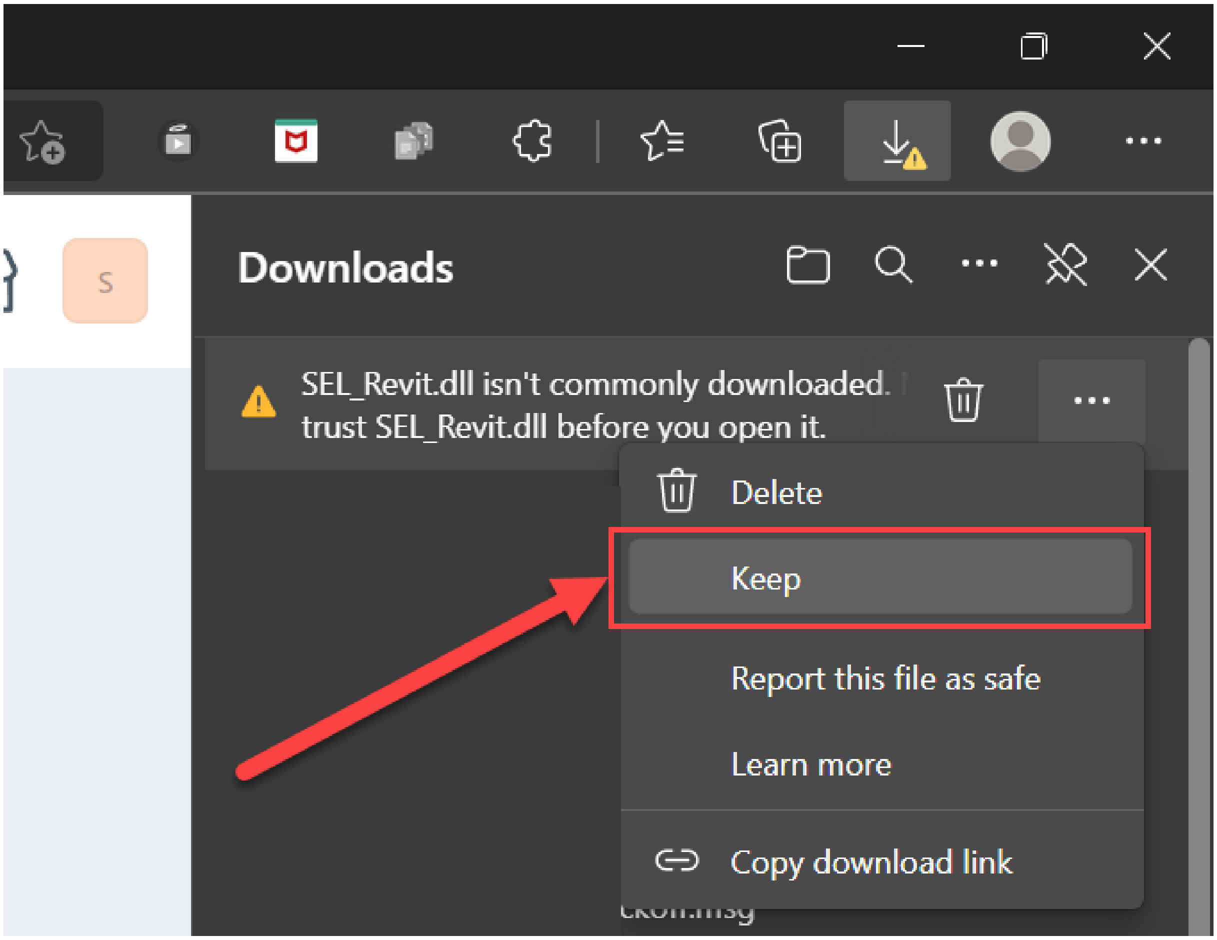
Download the attached or linked file ("SEL_Revit.dll"). Your computer will recognize that the file contains source code and will warn you. Approve the download to proceed.

In Google Chrome:

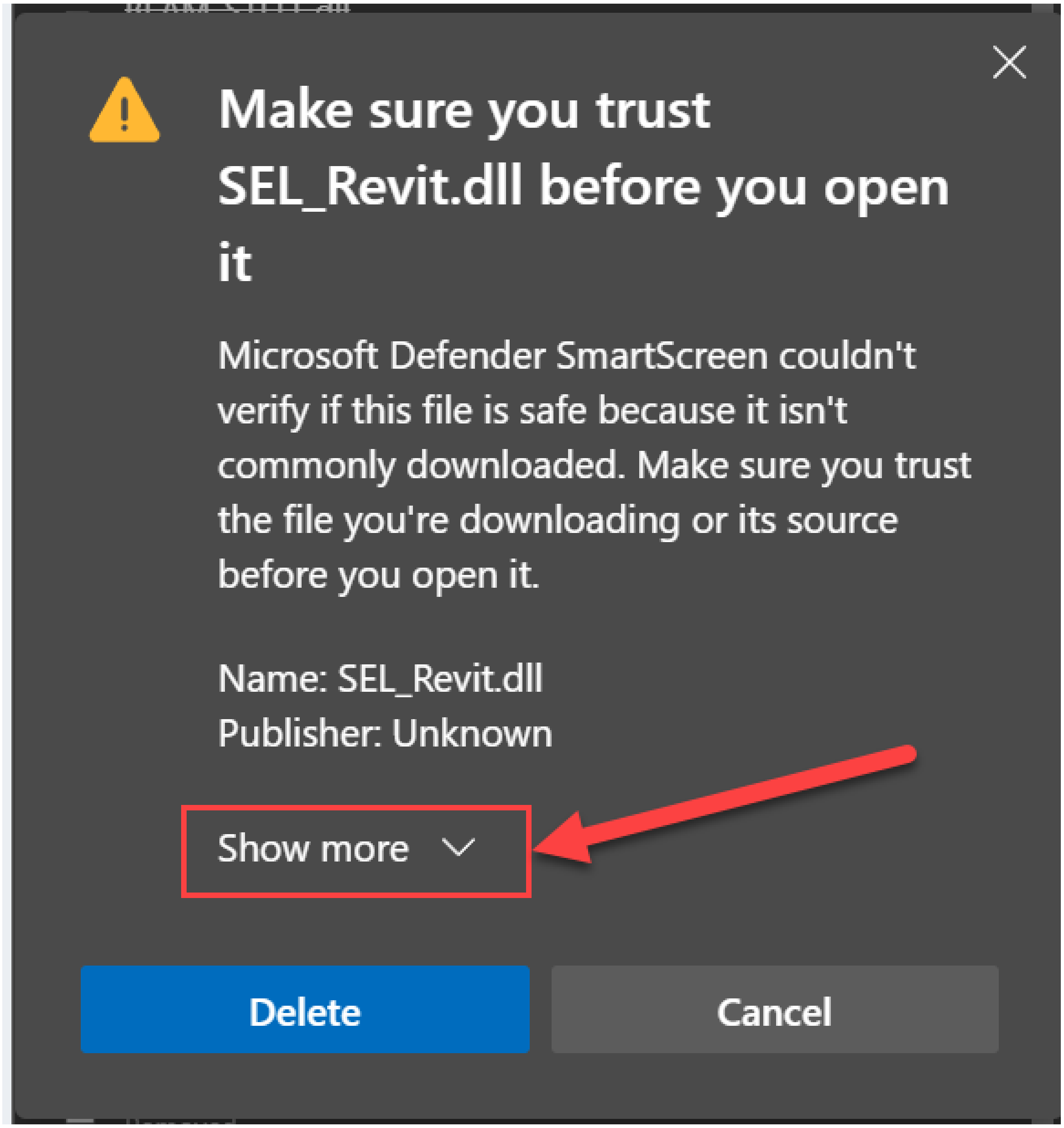

This type of file can harm your computer.
Do you want to keep SEL_Revit.dll anyway?
Keep
Discard

In MS Edge:

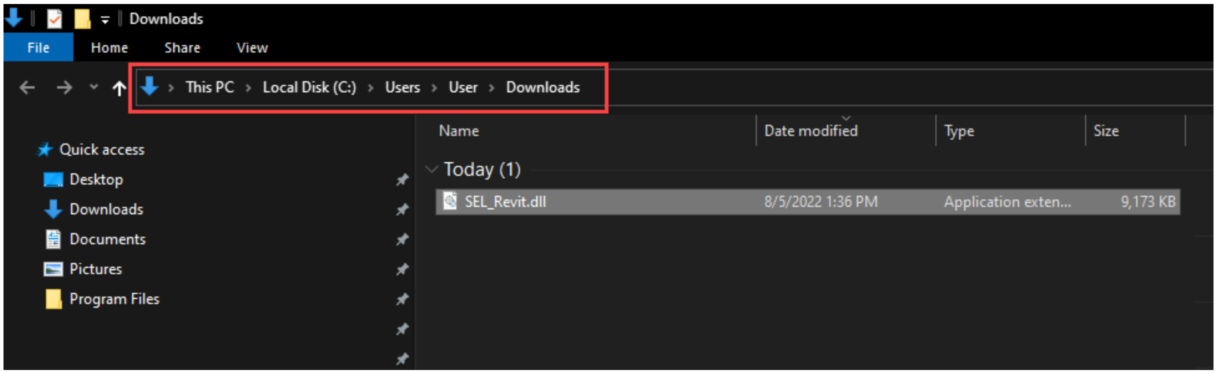

SR
Downloads
SEL_Revit.dll isn't commonly downloaded.
trust SEL_Revit.dll before you open it.
More actions



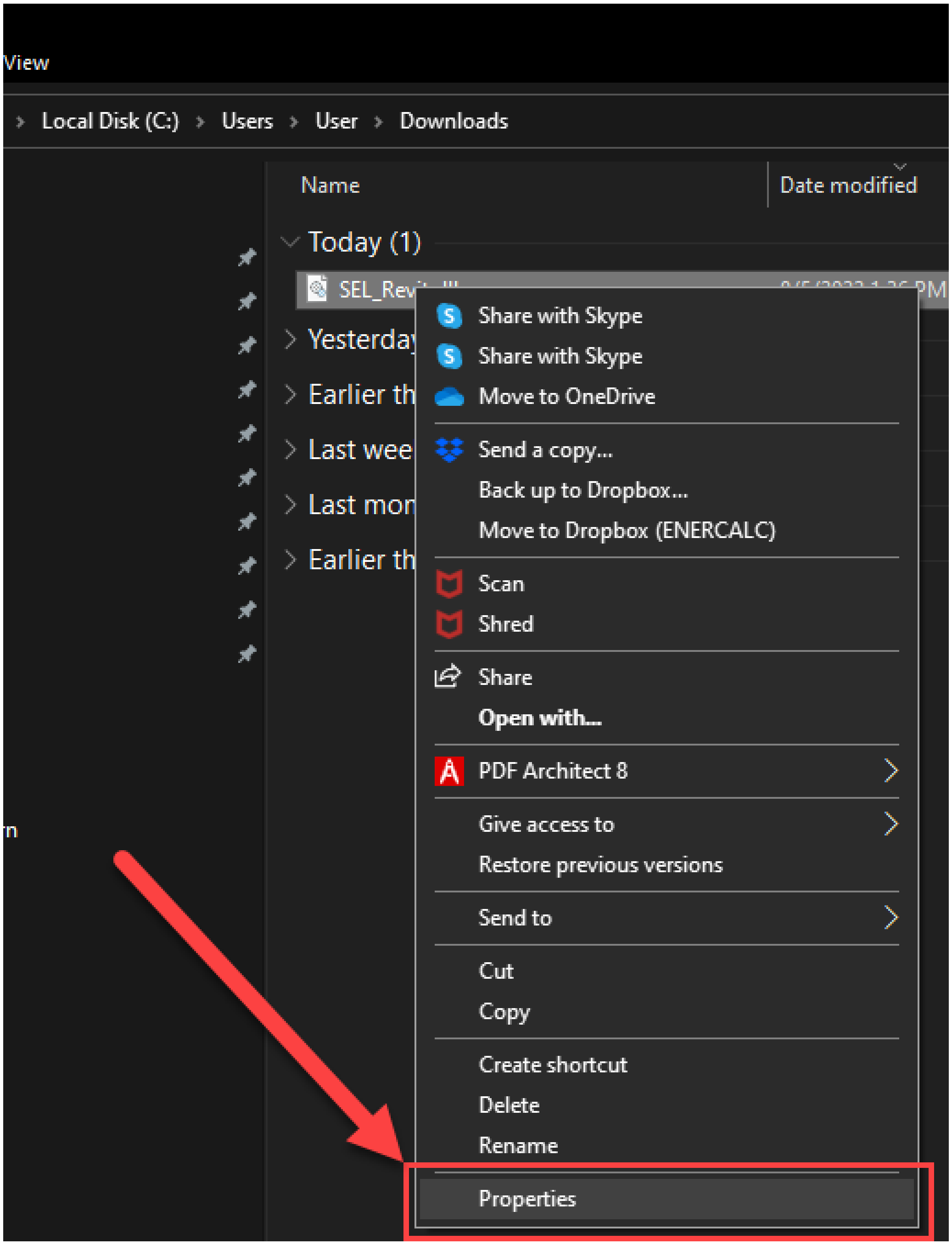
Depending on your computer settings, Windows Defender may prompt for an additional verification after you select "Keep":

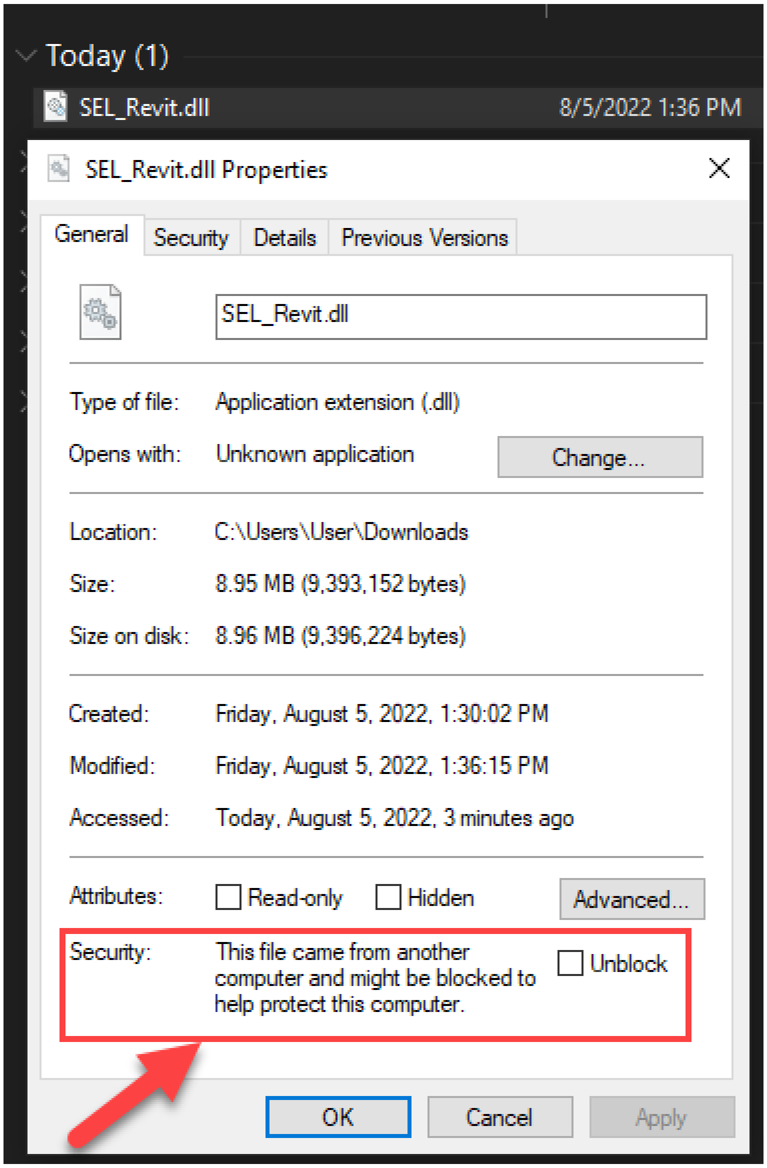


Once you finish trusting the download, the file will be placed in your Downloads folder:

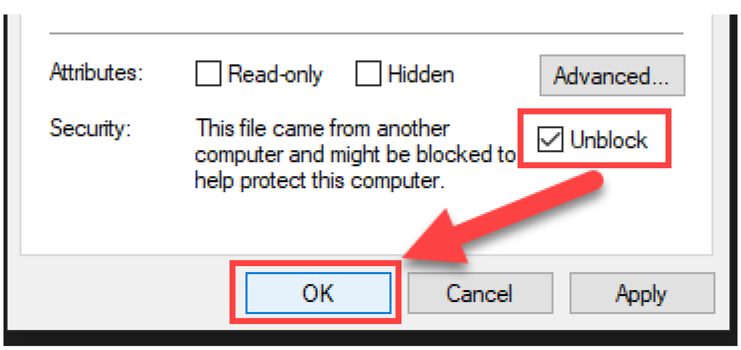


Since Windows recognizes that the download is a code file, it will be blocked by default. Unblock the file by right-clicking it and opening "Properties":


View
Local Disk (C:) > Users > User > Downloads
Name
Date modified
Today (1)
Yesterday
Last week
Last month
Share with Skype
Share with Skype
Move to OneDrive
Send a copy...
Back up to Dropbox...
Move to Dropbox (ENERCALC)
Scan
Shred
Share
Open with...
PDF Architect 8
Give access to
Restore previous versions
Send to
Cut
Copy
Create shortcut
Delete
Rename
Properties

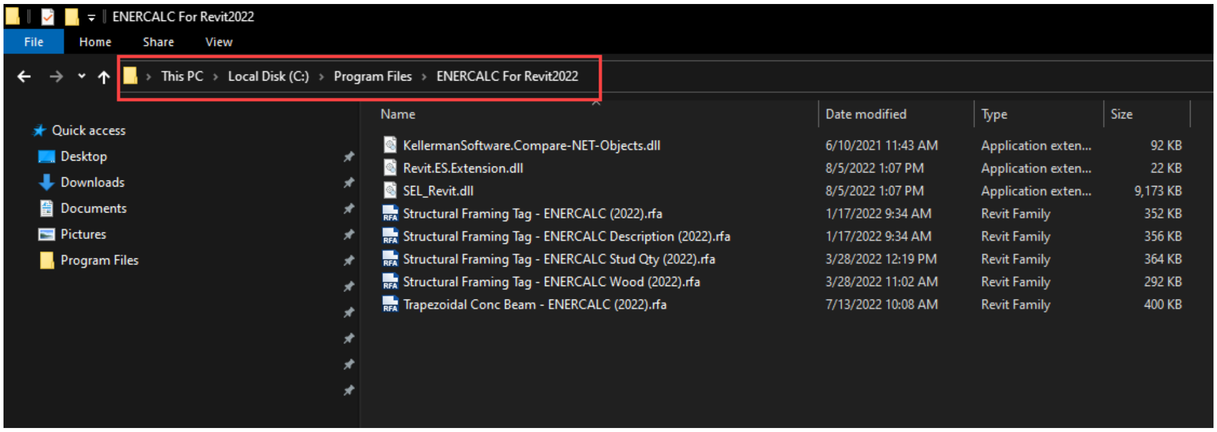


Check the "Unblock" box and click "OK":

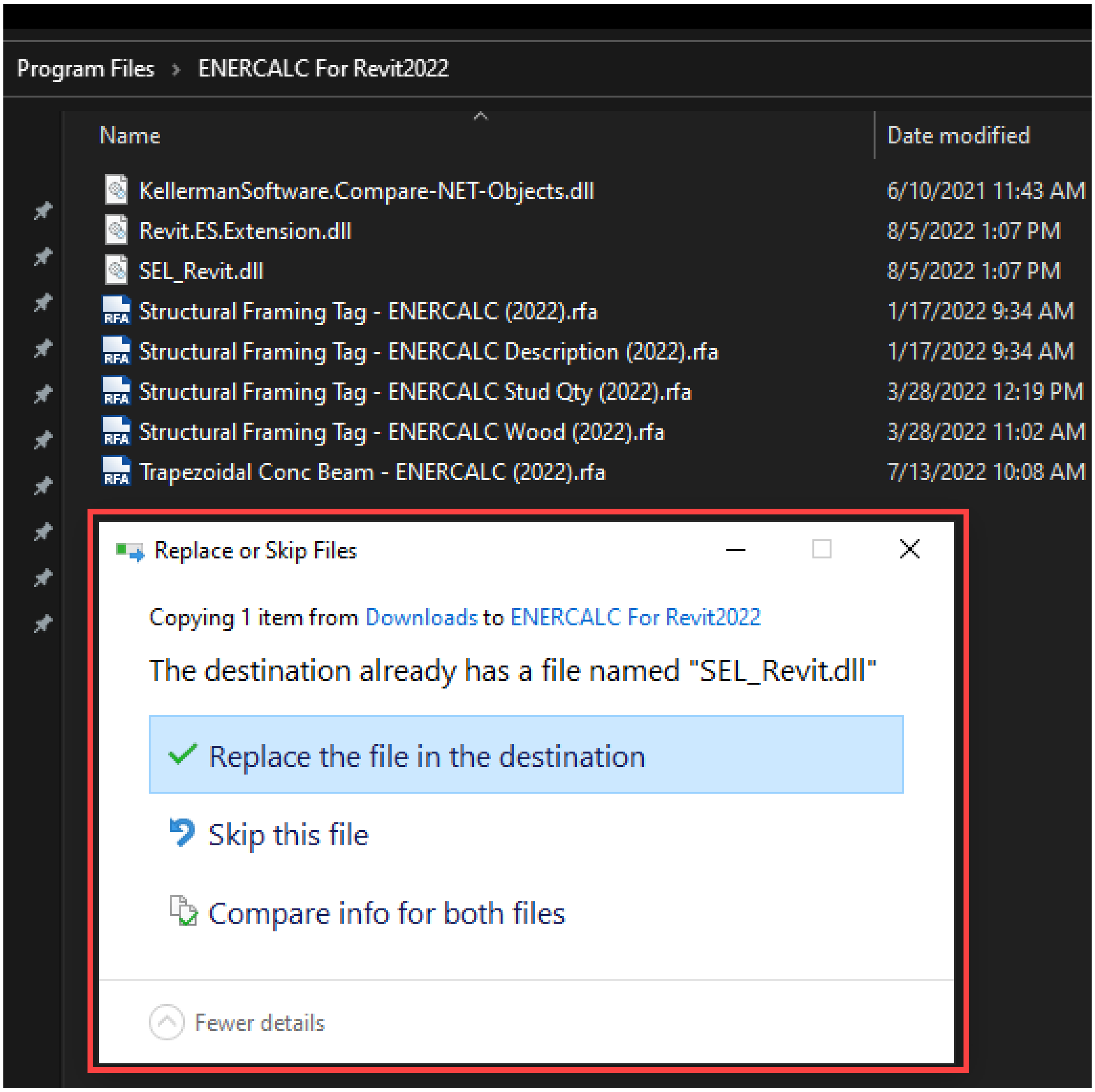


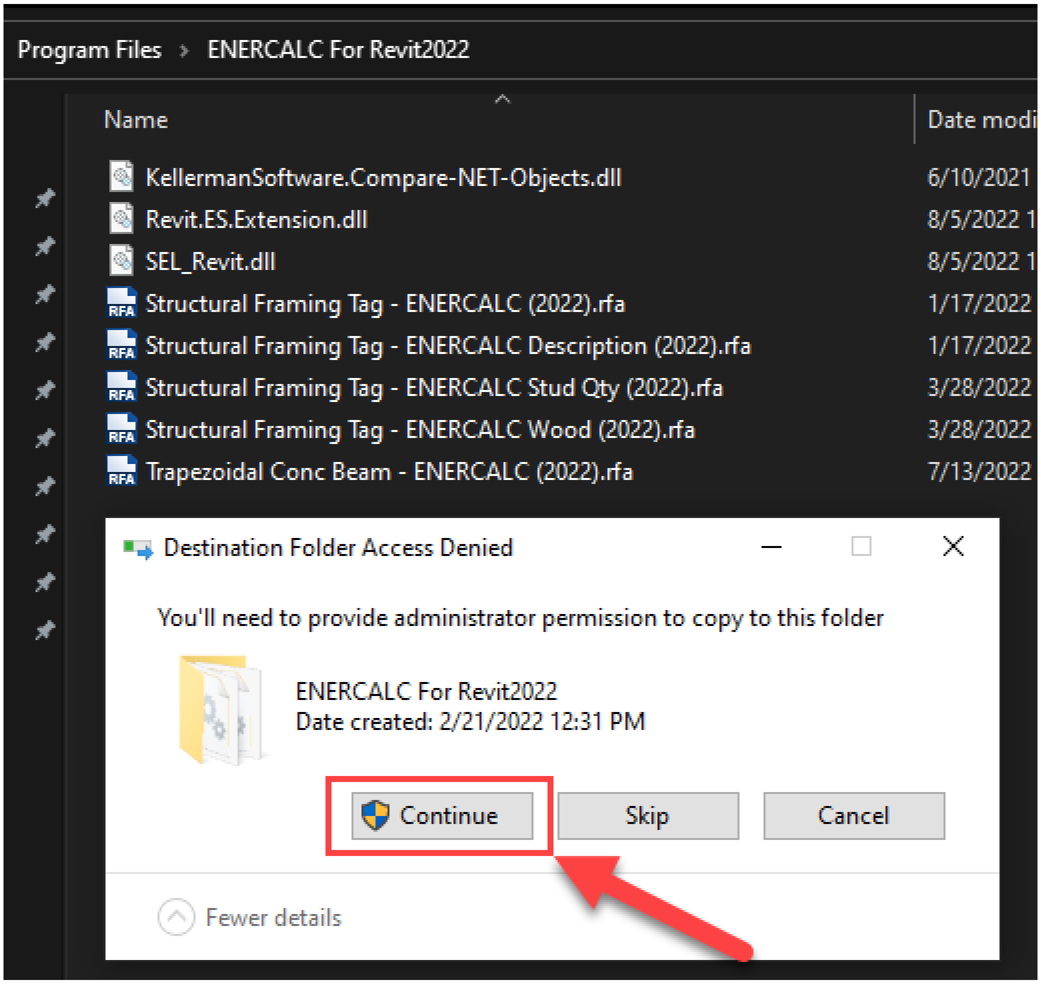
Now that the file is downloaded and unblocked, prepare to place it in the proper location by cutting (Ctrl+X) or copying it (Ctrl+C).

To manually replace the existing assembly (.DLL) with the new one, navigate to the Program Files folder for the appropriate version year: "\Program Files\ ENERCALC For Revit20XX":

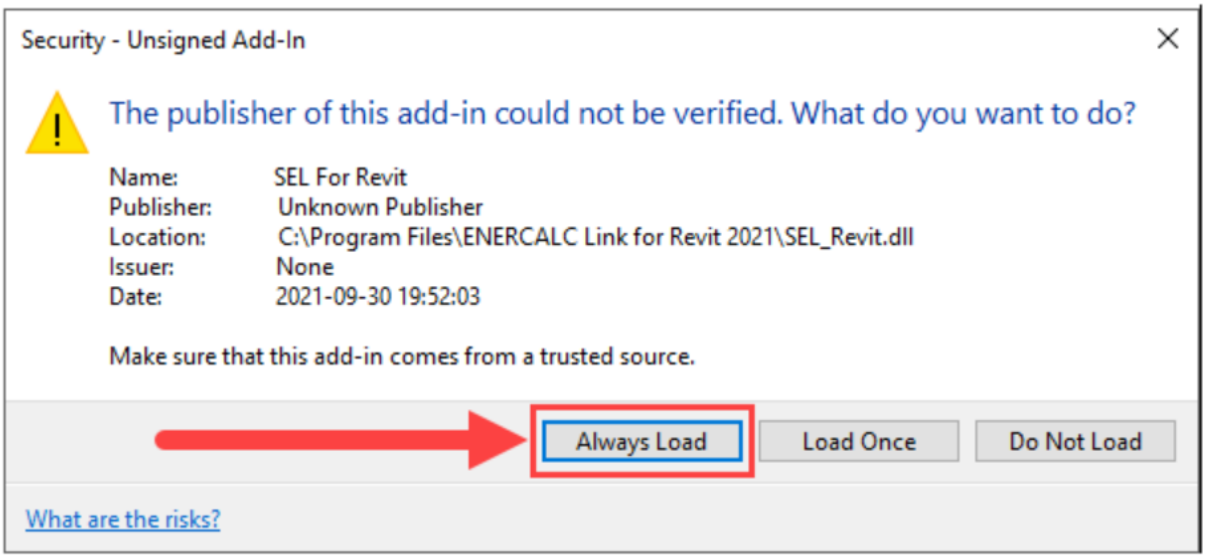


Paste the new .DLL file and select "Replace the file..." when prompted:


Program Files › ENERCALC For Revit2022
Name
Date modified
KellermanSoftware.Compare-NET-Objects.dll 6/10/2021 11:43 AM
Revit.ES.Extension.dll 8/5/2022 1:07 PM
SEL_Revit.dll 8/5/2022 1:07 PM
Structural Framing Tag - ENERCALC (2022).rfa 1/17/2022 9:34 AM
Structural Framing Tag - ENERCALC Description (2022).rfa 1/17/2022 9:34 AM
Structural Framing Tag - ENERCALC Stud Qty (2022).rfa 3/28/2022 12:19 PM
Structural Framing Tag - ENERCALC Wood (2022).rfa 3/28/2022 11:02 AM
Trapezoidal Conc Beam - ENERCALC (2022).rfa 7/13/2022 10:08 AM
Replace or Skip Files
Copying 1 item from Downloads to ENERCALC For Revit2022
The destination already has a file named "SEL_Revit.dll"
Replace the file in the destination
Skip this file
Compare info for both files
Fewer details



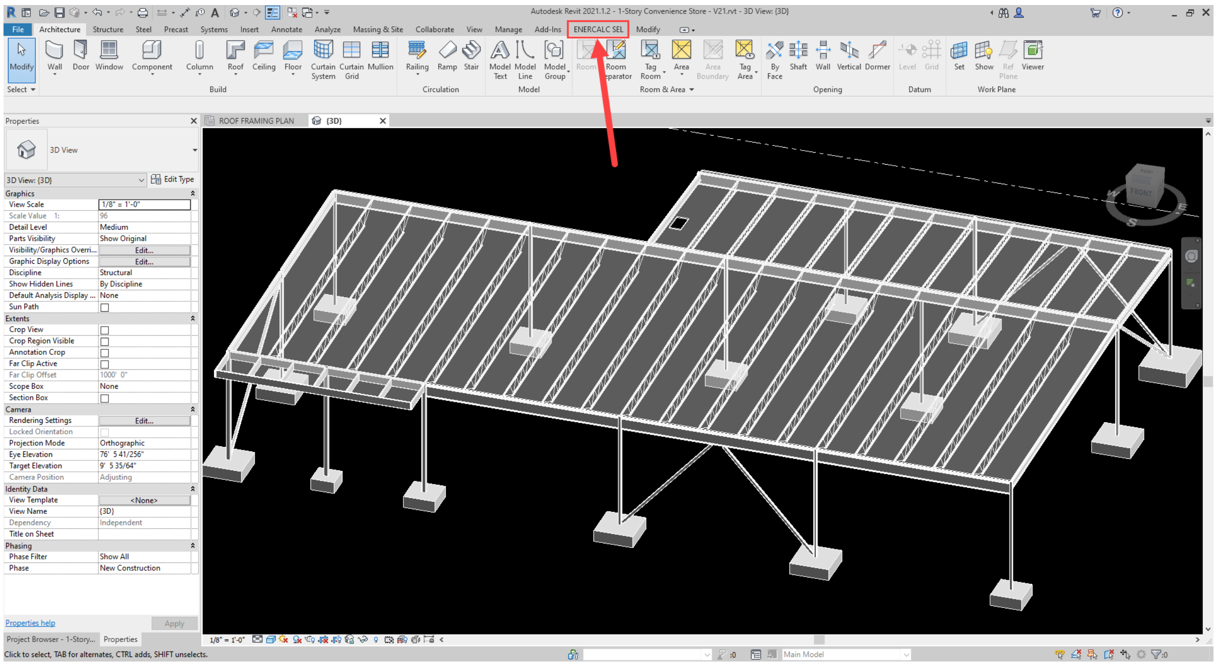
Once you have successfully placed the new file, relaunch the appropriate version of Revit. On Revit startup, you may be prompted to trust the new assembly with a pop-up similar to the following. Select "Always Load" to trust the revised EFR build and continue.


Security - Unsigned Add-In
The publisher of this add-in could not be verified. What do you want to do?
Name: SEL For Revit
Publisher: Unknown Publisher
Location: C:\Program Files\ENERCALC Link for Revit 2021\SEL_Revit.dll
Issuer: None
Date: 2021-09-30 19:52:03
Make sure that this add-in comes from a trusted source.
Always Load
Load Once
Do Not Load
What are the risks?

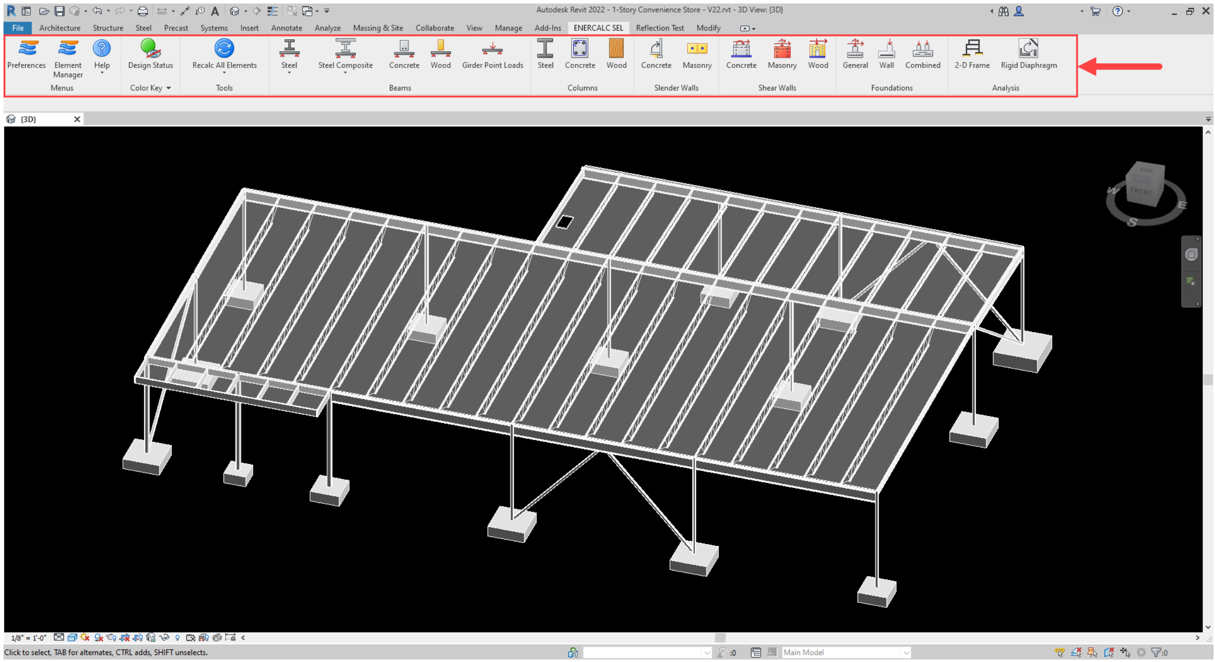
Part III

3 Navigating the Ribbon Bar

Upon opening a Revit model, all controls relating to ENERCALC SEL may be found on the native Revit ribbon bar, on a newly added tab labeled “ENERCALC SEL”.

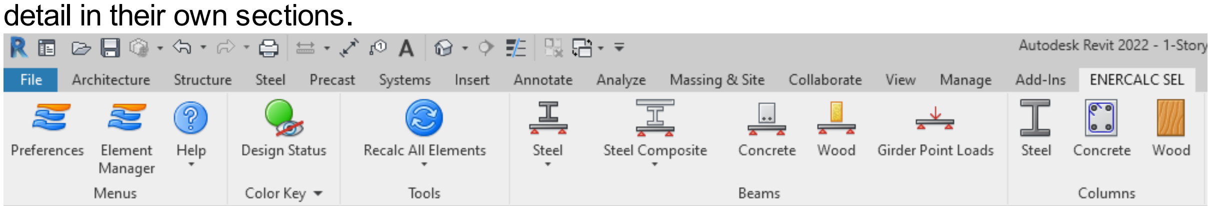


Selecting this tab displays a ribbon bar of all the tools used to manage ENERCALC structural calculations from Revit. ***The ribbon bar shown below is for illustration only. Refer to your particular installation to verify which module design tools will be displayed in your Revit interface.***



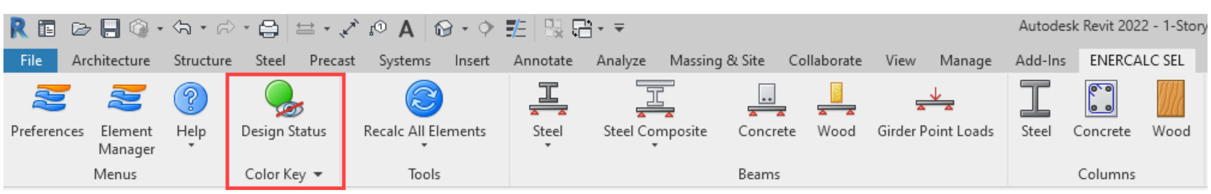
3.1 Preferences Menu and Element Manager Window

The first two buttons (both denoted by the ENERCALC “E” logo) launch the Preferences menu and the Element Manager window, respectively. Each of these menus is discussed in detail in their own sections.



3.2 Design Status Control

Immediately to the right of the Element Manager button is a button marked “Design Status”. This control is used to overview the overall status of all calculations in the Revit model.



Any individual Revit element that has been corelated to an ENERCALC calculation will have one of the following four statuses:

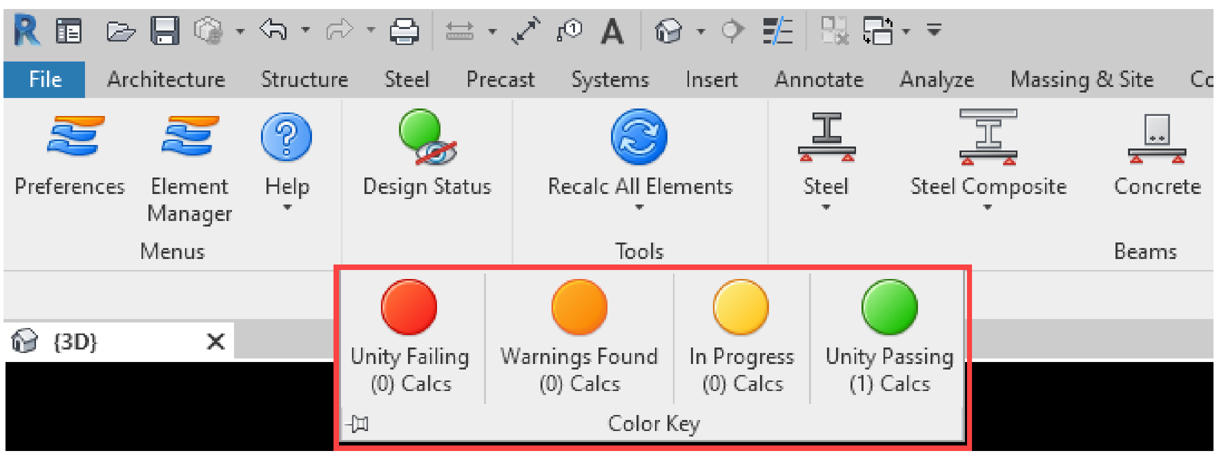
- **Green** – Calculation is passing its unity check (unity <= 1.0)
- **Yellow** – Calculation is incomplete (i.e., pending or cancelled)

- **Orange** – Calculation has a warning (i.e., Revit geometry changes have caused the element to diverge from its ENERCALC SEL calculation data)
- **Red** – Calculation is failing its unity check (unity > 1.0)

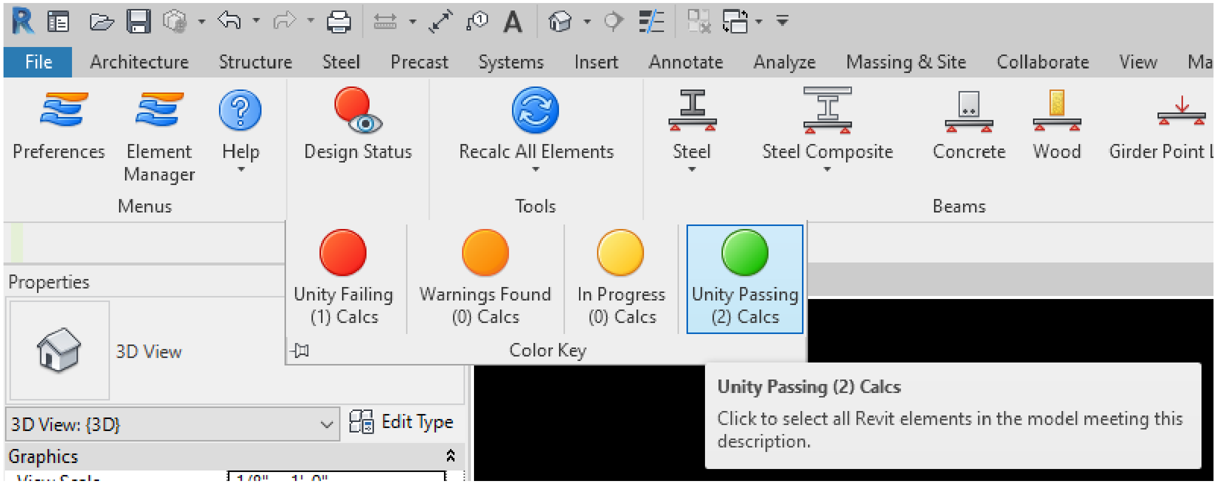
At any given time, the color of the circle indicates the worst status of any designed element found in the entire model. For display purposes, severity of status is ranked from “best” to “worst” in the same order as the above list, where, “worst” is defined as the most severe cause for concern for the design professional (i.e., failing unity). This ranking is illustrated in “Overall Design Status Ranking 28”.

3.3 Color Key and Quick Select

Expanding the “Color Key” slide-out below the Design Status button shows a key that indicates the meaning of each status color and the quantity of elements having each status found in the Revit model.



When displayed, each button in the slide-out menu acts a quick-select control to easily overview the elements having various different statuses. Clicking an individual button on the slide-out will select the corresponding elements in the Revit model.

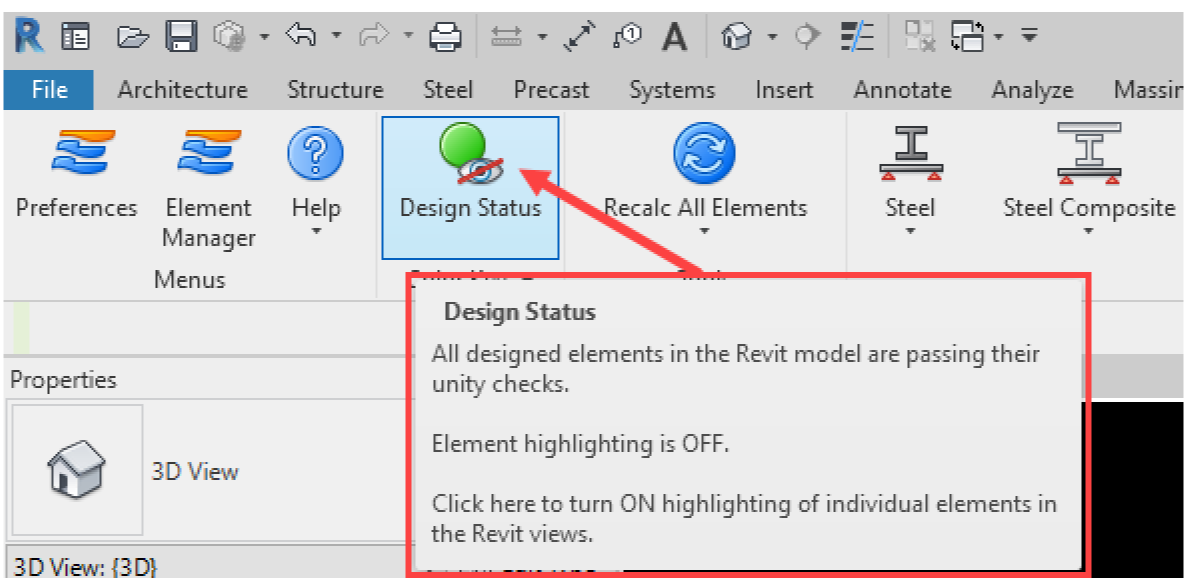


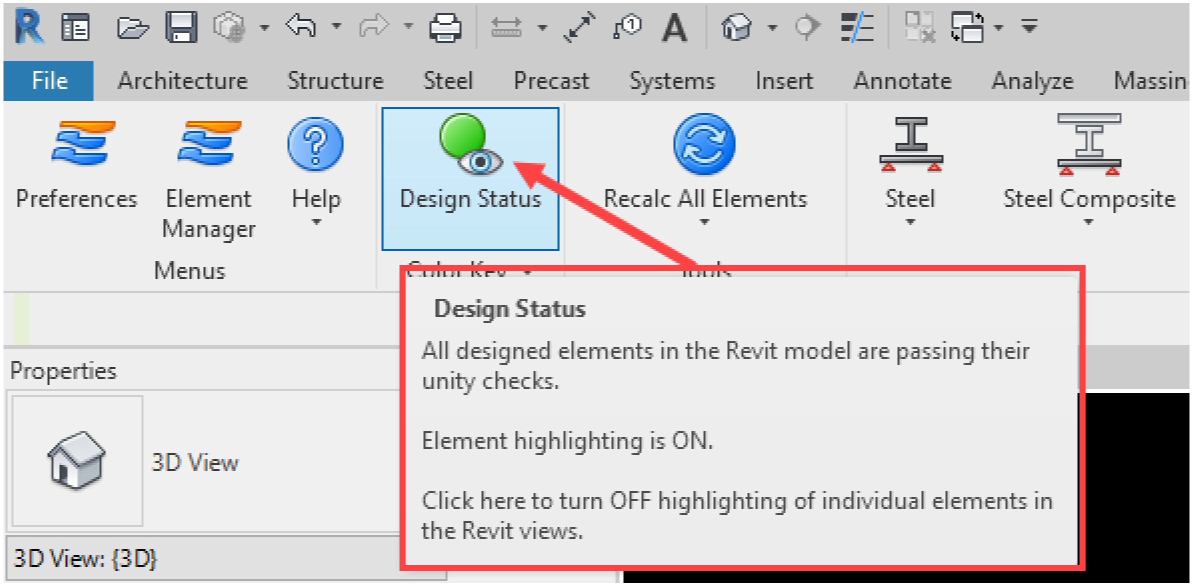
The color coding indicated here in the slide-out menu corresponds directly to the colors displayed on each individual element when highlighting is toggled to “ON”.

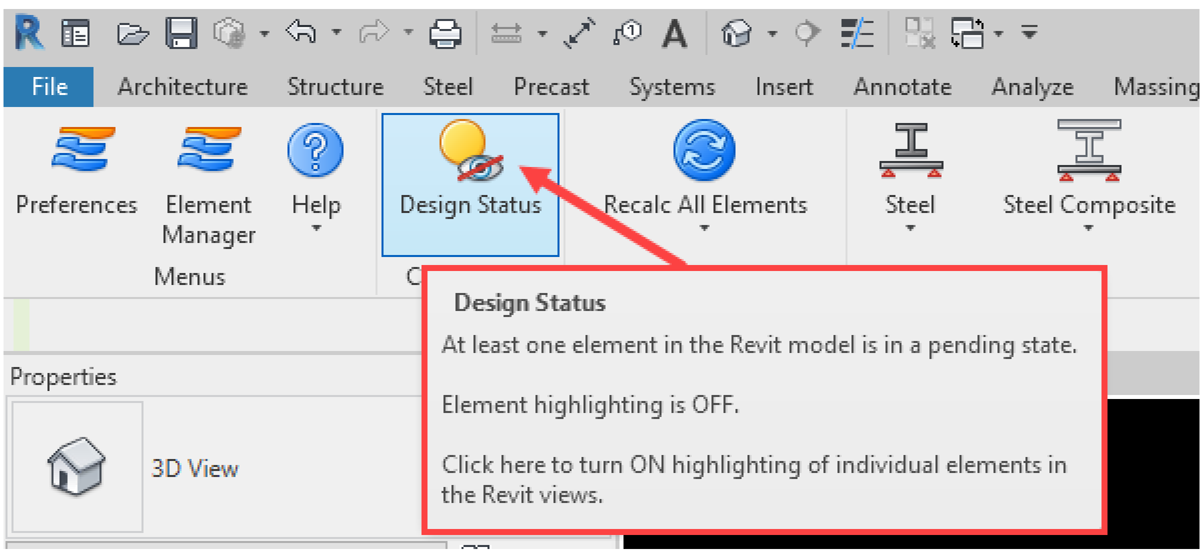
3.4 Highlighting Toggle and Tooltip Info

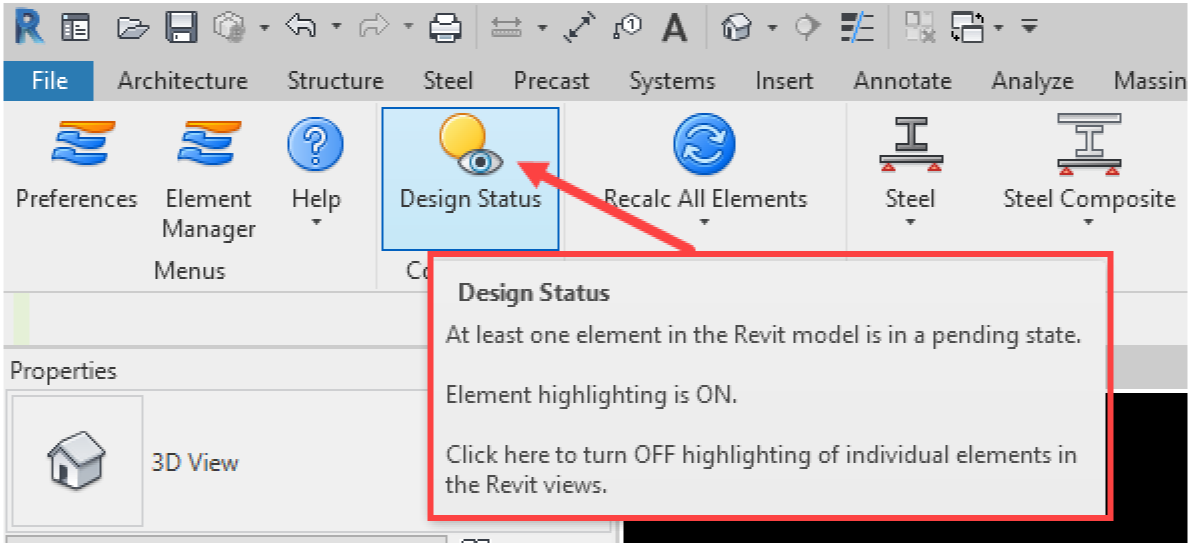
The status of the “eyeball” symbol also indicates at all times whether status highlighting of individual elements in the Revit views is “ON” or “OFF”. Clicking the “Design Status” button toggles color highlighting of elements “ON” or “OFF” for all active views in the current instance of Revit. This toggle is an application-wide setting and does not need to be performed for individual views. When the status highlighting is toggled to “OFF”, the “eyeball” symbol will have a red slash through it. The slash will disappear when status highlighting is toggled to “ON”.

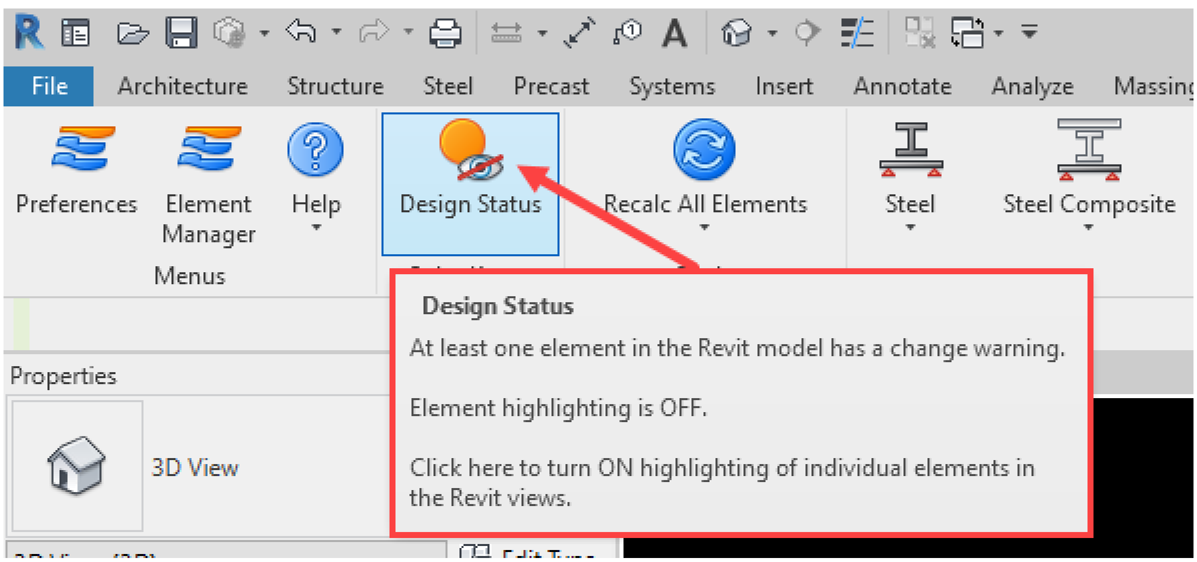
The overall design status of the model and the current highlighting setting are also explained via a dynamic tooltip balloon, visible when hovering over the “Design Status” button.

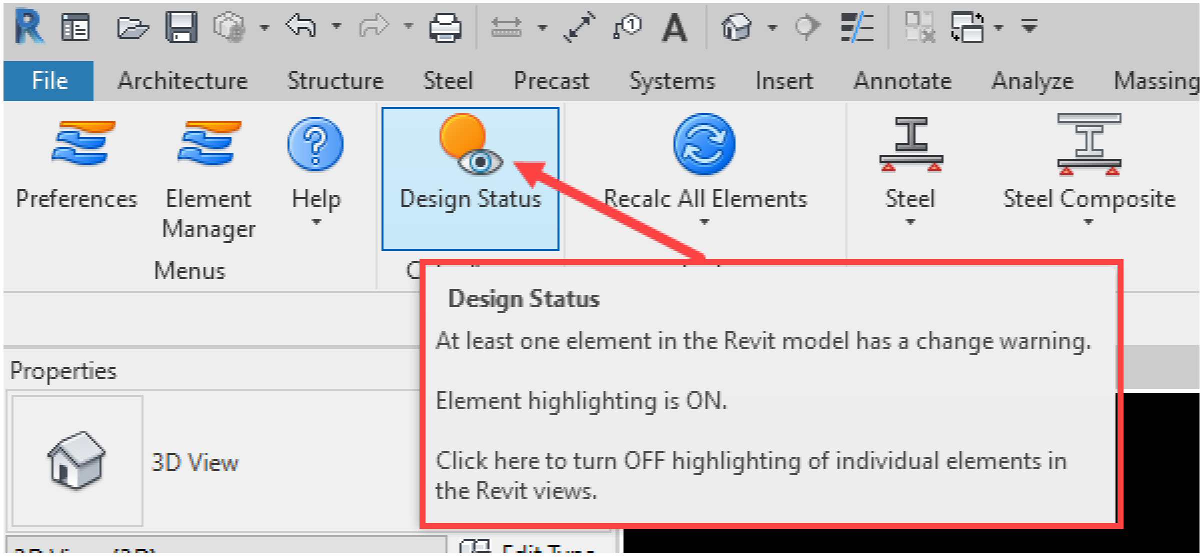


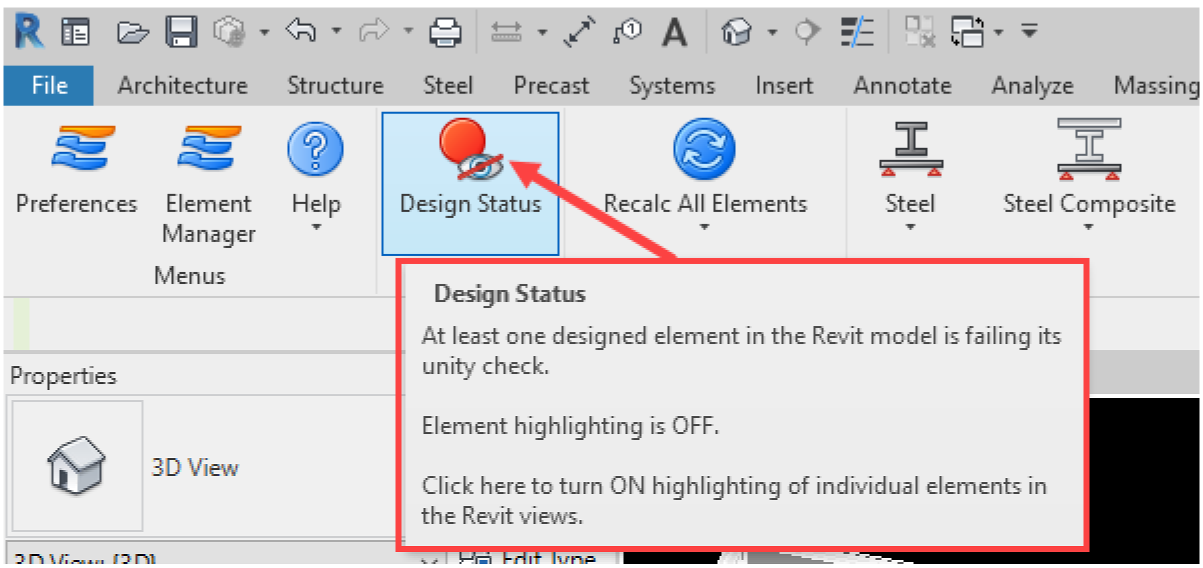

File
Architecture
Structure
Steel
Precast
Systems
Insert
Annotate
Analyze
Preferences
Element Manager
Help
Design Status
Recalc All Elements
Steel
Steel Composite
Menus
Properties
3D View
3D View: {3D}
Design Status
All designed elements in the Revit model are passing their unity checks.
Element highlighting is ON.
Click here to turn OFF highlighting of individual elements in the Revit views.

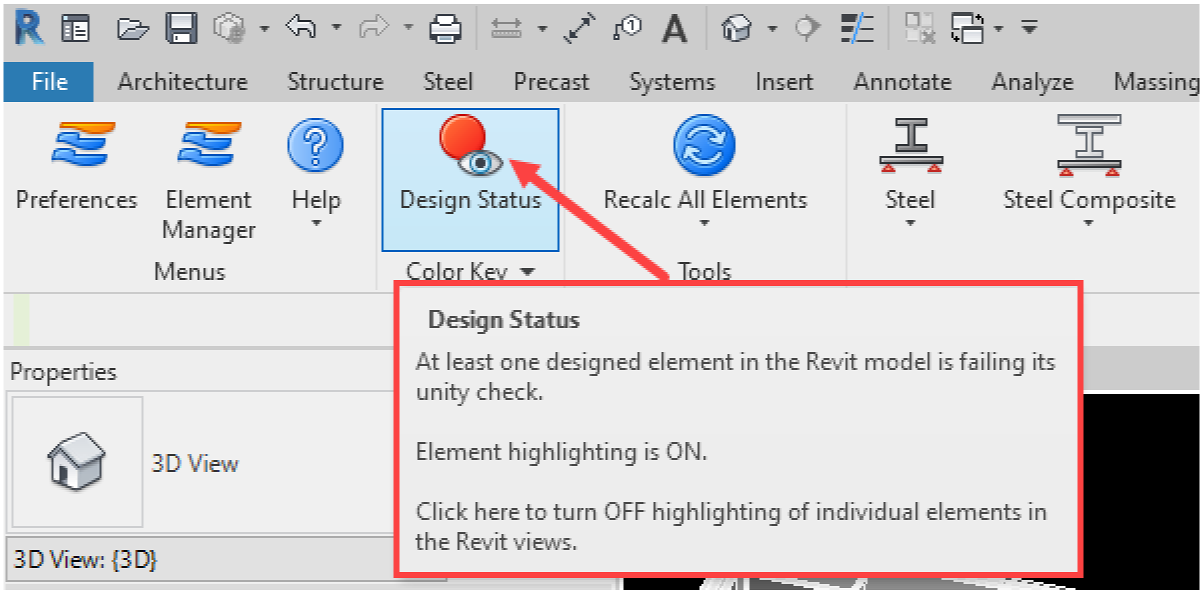

File
Architecture
Structure
Steel
Precast
Systems
Insert
Annotate
Analyze
Massing
Preferences
Element Manager
Help
Design Status
Recalc All Elements
Steel
Steel Composite
Menus
Properties
3D View
Design Status
At least one element in the Revit model is in a pending state.
Element highlighting is OFF.
Click here to turn ON highlighting of individual elements in the Revit views.


File
Architecture
Structure
Steel
Precast
Systems
Insert
Annotate
Analyze
Massin
Preferences
Element Manager
Help
Design Status
Recalc All Elements
Steel
Steel Composite
Menus
Properties
3D View
Design Status
At least one element in the Revit model is in a pending state.
Element highlighting is ON.
Click here to turn OFF highlighting of individual elements in the Revit views.


File
Architecture
Structure
Steel
Precast
Systems
Insert
Annotate
Analyze
Massing
Preferences
Element Manager
Help
Design Status
Recalc All Elements
Steel
Steel Composite
Menus
Properties
3D View
Design Status
At least one element in the Revit model has a change warning.
Element highlighting is OFF.
Click here to turn ON highlighting of individual elements in the Revit views.

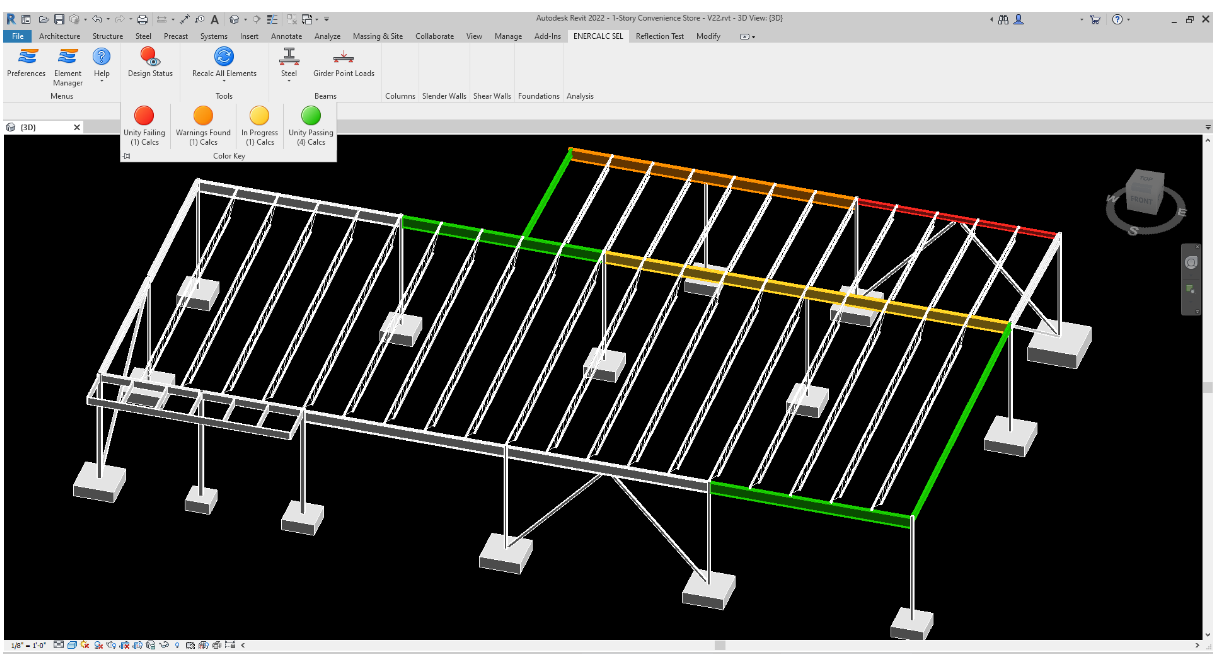

File
Architecture
Structure
Steel
Precast
Systems
Insert
Annotate
Analyze
Massing
Preferences
Element Manager
Help
Design Status
Recalc All Elements
Steel
Steel Composite
Menus
Properties
3D View
Design Status
At least one element in the Revit model has a change warning.
Element highlighting is ON.
Click here to turn OFF highlighting of individual elements in the Revit views.

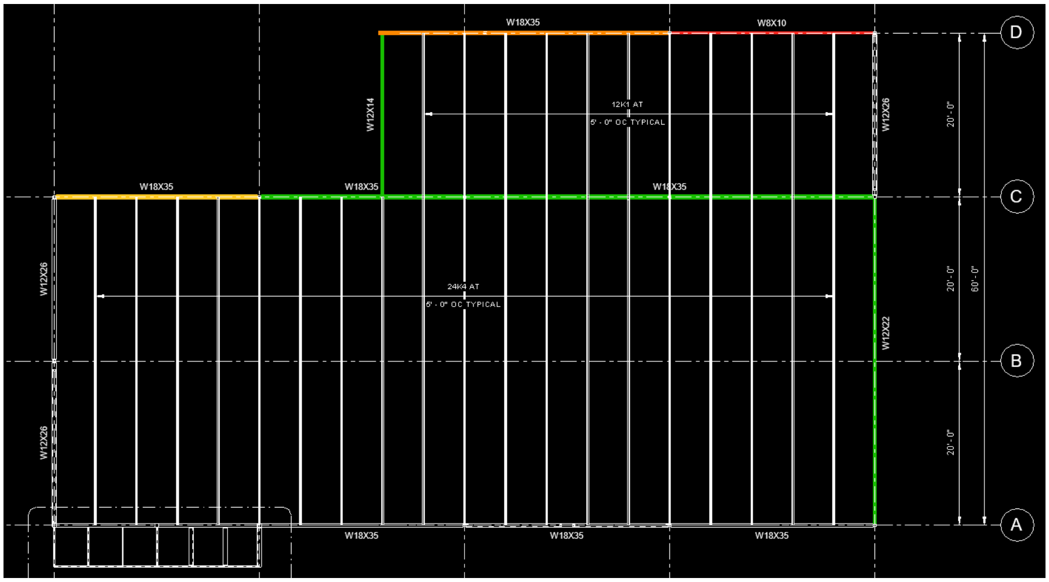

File
Architecture
Structure
Steel
Precast
Systems
Insert
Annotate
Analyze
Massing
Preferences
Element Manager
Help
Design Status
Recalc All Elements
Steel
Steel Composite
Menus
Properties
3D View
Design Status
At least one designed element in the Revit model is failing its unity check.
Element highlighting is OFF.
Click here to turn ON highlighting of individual elements in the Revit views.



3.4.1 Status Highlighting of Elements

If status highlighting is set to “ON”, then the appropriate color that reflects the design status of each individual element will be applied to each individual element that has been designed using ENERCALC for Revit. The highlighting is applied in all active views, and is also applied automatically in new views that are subsequently opened after the setting is activated. Toggling the main status button from “ON” to “OFF” will remove the color-coded highlighting from all active views.



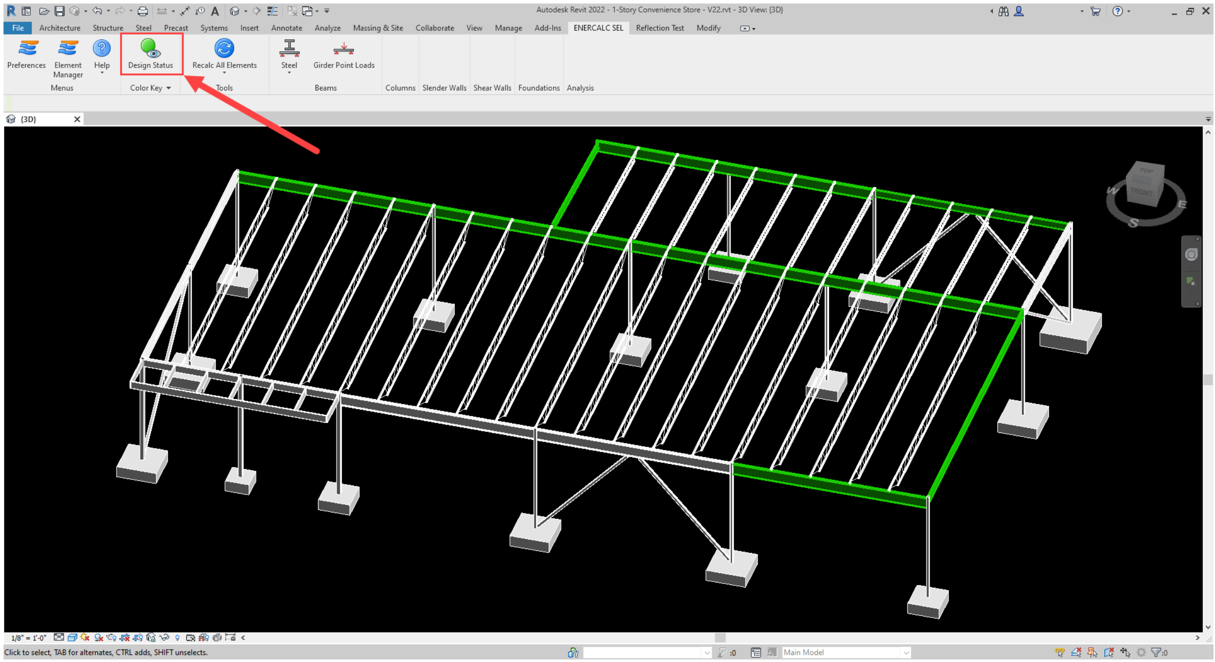


Color coding of the individual elements is applied by taking advantage of the same native graphic override controls that Revit users access manually via right click > Override Graphics in View > By Element. This means that toggling the highlighting “ON” or “OFF” creates actual graphical changes to all active Revit views. Upon synchronizing a workshared Revit model, the color highlighting setting will also be applied to other team members’ local copies of the model, just like any other graphical override would be. To avoid interfering with the desired graphics settings of other team members, it may be advisable for all users to toggle status highlighting to “OFF” ***prior to synchronizing***.

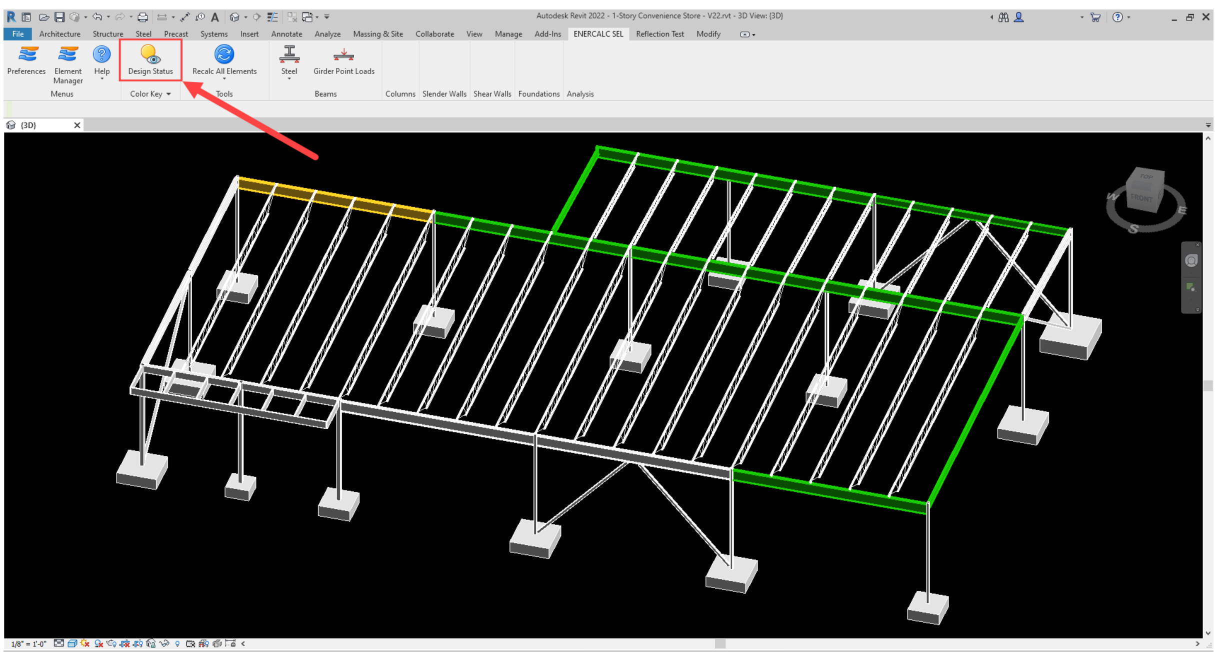
3.4.2 Overall Design Status Ranking

Since each designed element in the Revit model has its own status, the status light on the ribbon bar summarizes the conditions of the model by displaying the worst status currently found. For display purposes, severity of status is ranked from “best” to “worst”, where, “worst” is defined as the most severe cause for concern for the design professional (i.e., failing unity).

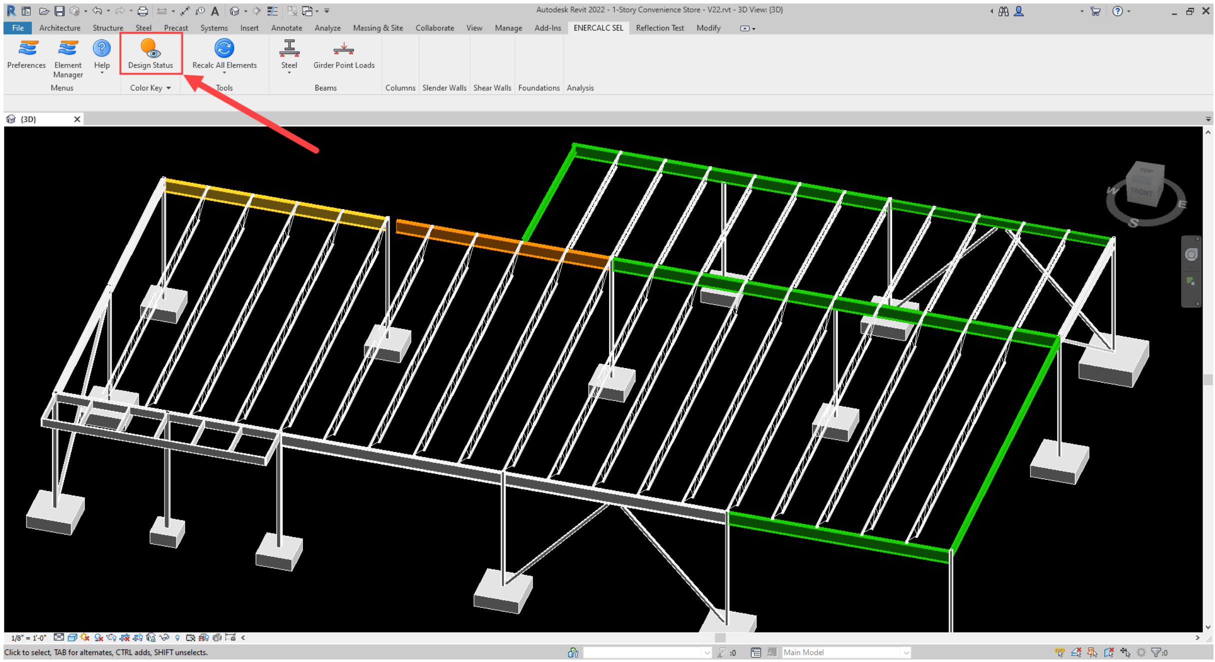
If all elements in the model are passing their unity checks, then the status light will show green.



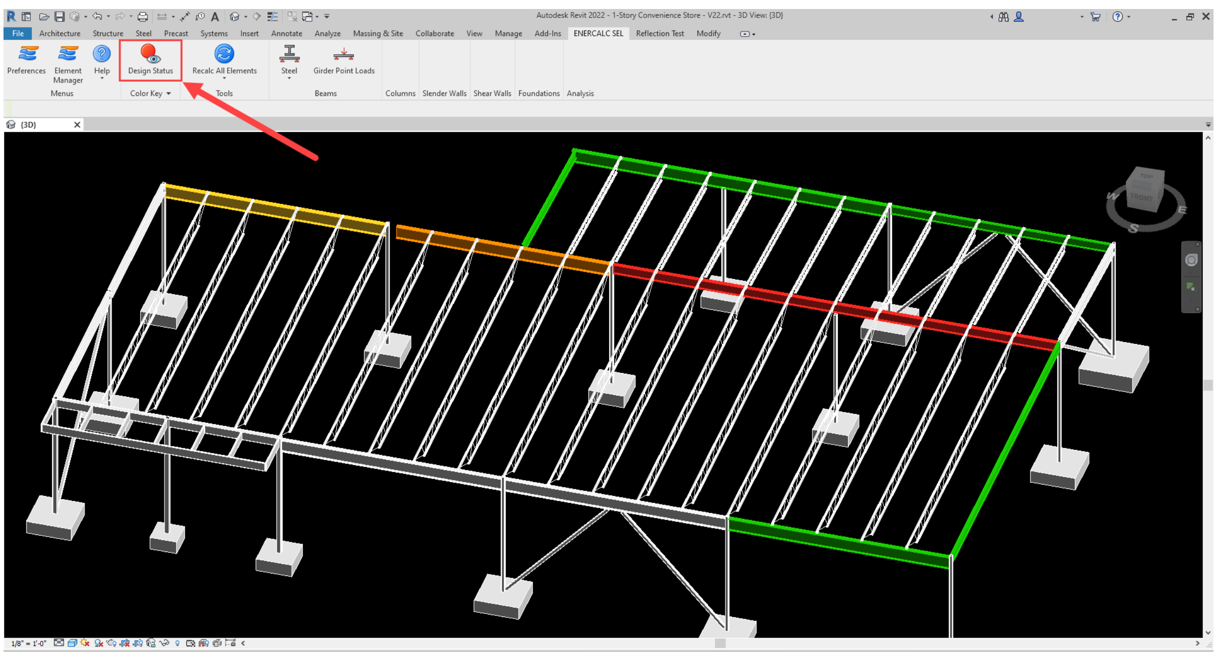
If the worst element status found in the model is a pending, cancelled, or incomplete status, then the status light will show yellow.



If worst element status found in the model is a change warning, then the status light will show orange.



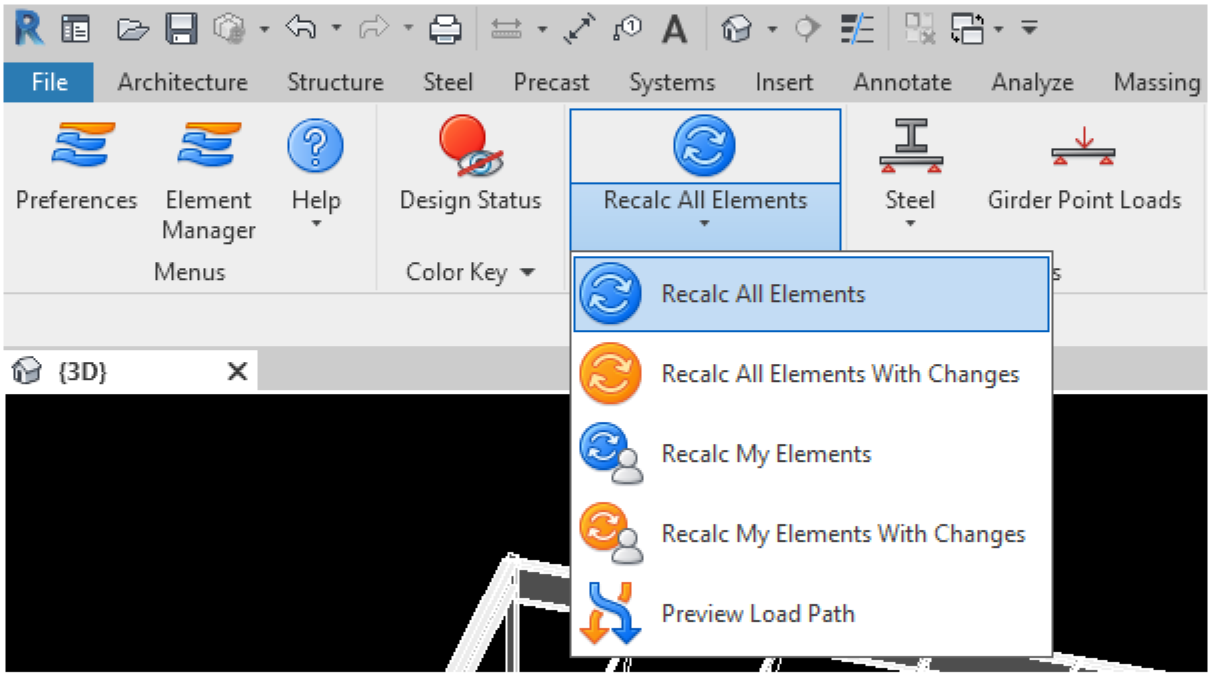
If worst element status found in the model is a unity check failure, then the status light will show red.

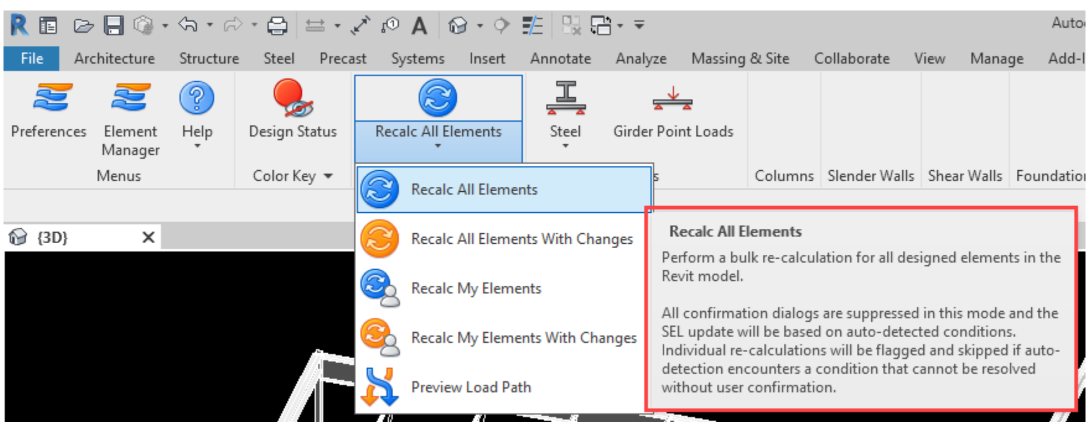


3.5 Recalc Tools

Immediately to the right of the “Design Status” button is the bulk recalculation button. This recalc drop-down menu houses a set of different options for refreshing (recalculating) each of the existing element calculations currently found in the Revit model.

NOTE: At this time, the recalc tools provided in ENERCALC for Revit are ONLY for triggering bulk updates to existing calculations that were previously created and verified via manual interaction by the design professional. No tools are provided for fully autonomous generation of component design calculations.



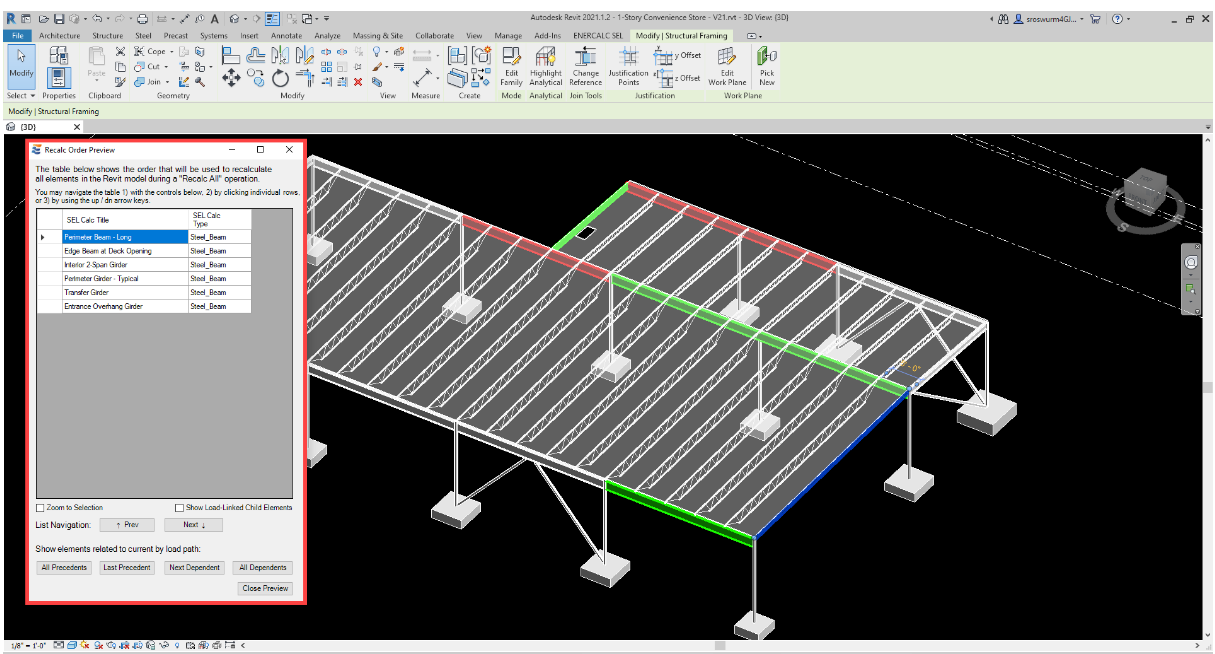


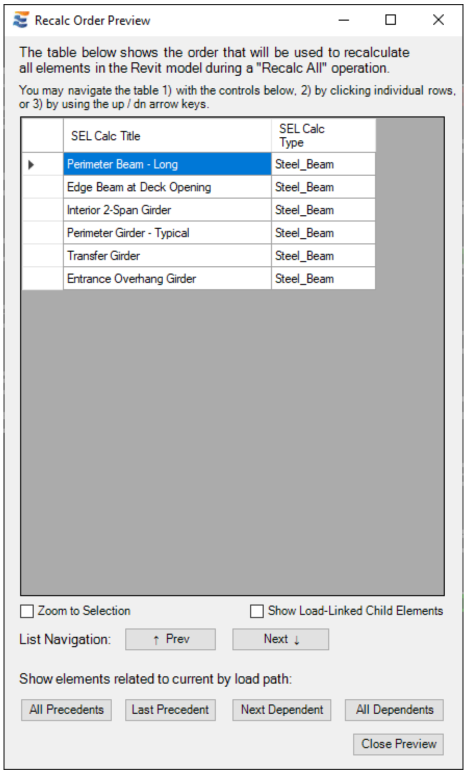
As noted in the tooltip balloon, the bulk recalculation process is fully automatic and does not require any manual intervention or approval actions by the user. Bulk recalculation is discussed in greater detail in the element-specific sections of this manual (beam, column, etc.).

- Recalc All Elements: This option will trigger recalculation of every element that has been correlated to an ENERCALC SEL calculation.

- Recalc All Elements With Changes: This option will trigger recalculation of every element that has been correlated to an ENERCALC SEL calculation and currently carries a change warning.
- Recalc My Elements: This option will trigger recalculation of every element whose ENERCALC SEL calculation was created by the current user. User identity and ownership of calculations is discussed in greater detail in “Linking a Project” and “Launching and Editing”.
- Recalc My Elements With Changes: This option will trigger recalculation of every element whose ENERCALC SEL calculation was created by the current user and currently carries a change warning. User identity and ownership of calculations is discussed in greater detail in “Linking a Project” and “Launching and Editing”.
- Preview Load Path: This option does not trigger a recalculation. Instead, it launches a tabular menu for the user to review the exact order in which the recalculation will be executed. Since ENERCALC for Revit allows users to associate reaction forces from one calculation to the next, the order of recalculation is crucial to ensure the integrity of the load path. Prior to initiating a recalc action ENERCALC for Revit automatically evaluates all support and tributary relationships approved by the user in order to establish the correct order for refreshing the calculations. The Preview Load Path tool makes this recalc order available for user review.

The preview tool does not freeze the Revit interface. Users retain full ability to zoom, pan, or change views while the form is open.



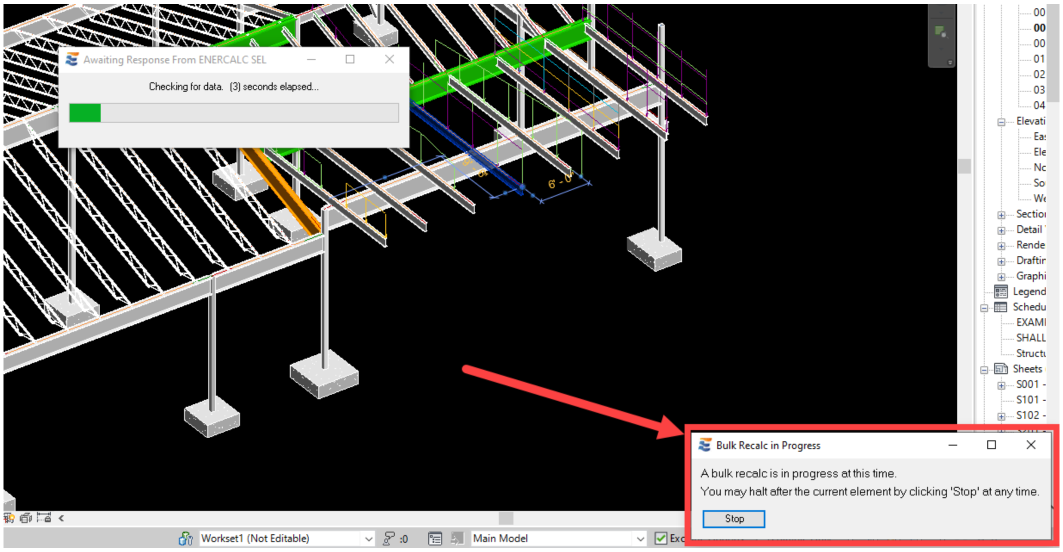


Navigating this form allows the user to observe the actual order in which the model will be recalculated. As indicated above the table in this window, the user may navigate the list via (3) different methods:

- By using the built-in navigation buttons
- By manually clicking through the table rows
- By using the up and down arrows on the keyboard

Advancing through the list automatically selects each individual element for visual review. The controls at the bottom of the form also allow the user to view all elements that have a “precedent” or “dependent” relationship with the current element.

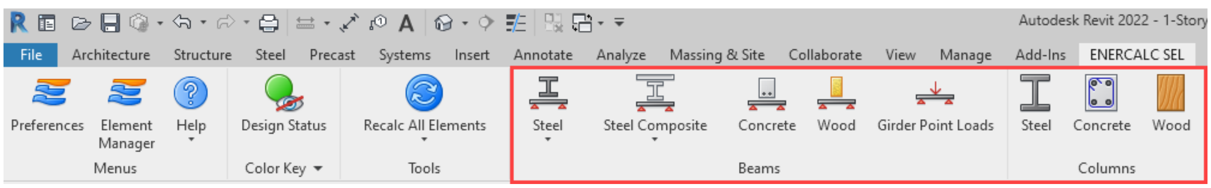
Once a recalculation is in progress, the process may be halted at any time using the window in the lower right-hand corner of the main Revit window. This button will cause the recalc to end after the current element. The element currently being recalculated will not be interrupted.



3.6 Calculation Launch Tools

To the right of the bulk recalculation controls are the buttons for launching ENERCALC SEL calculations. These controls are discussed in detail in subsequent sections.

The ribbon bar shown below is for illustration only. Refer to your particular installation to verify which module design tools will be displayed in your Revit interface.

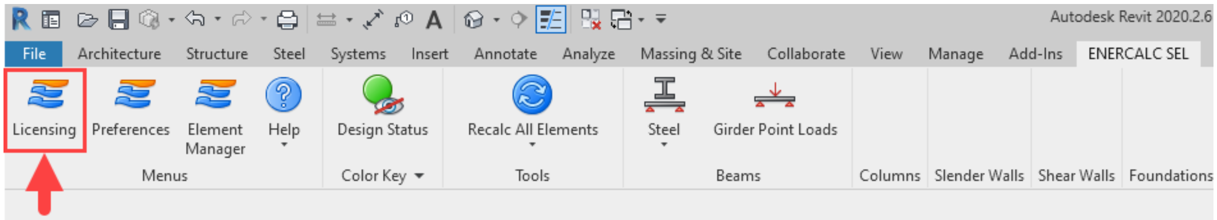


For more information about launching calculations, refer to "Launching and Editing ENERCALC Calculations 64".

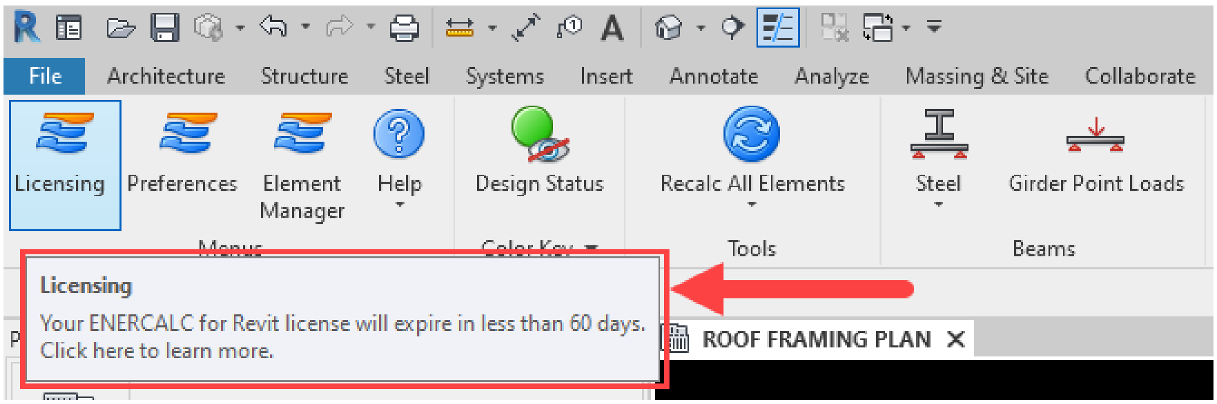
3.7 Licensing Controls

The ENERCALC for Revit tab on the Revit ribbon bar will display in one of several different conditions, depending on the state of the user's license. When the user's ENERCALC for Revit license is valid and current, the ribbon bar will display normally as depicted in the preceding sections.

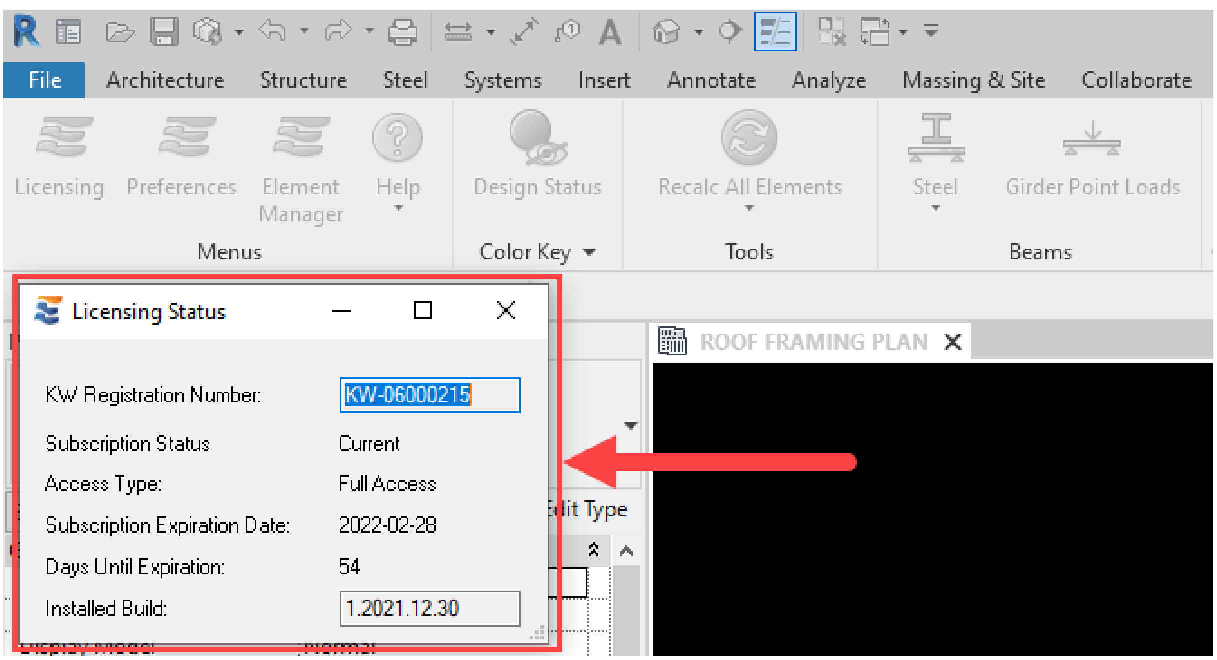
If the user's ENERCALC for Revit license is about to expire in 60 days or less, a "Licensing" button will appear on the ribbon bar as a reminder.



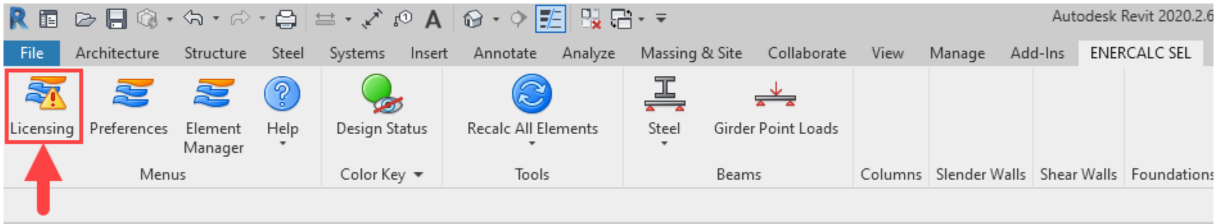
Hovering the cursor over the button displays a tooltip balloon with detailed information.



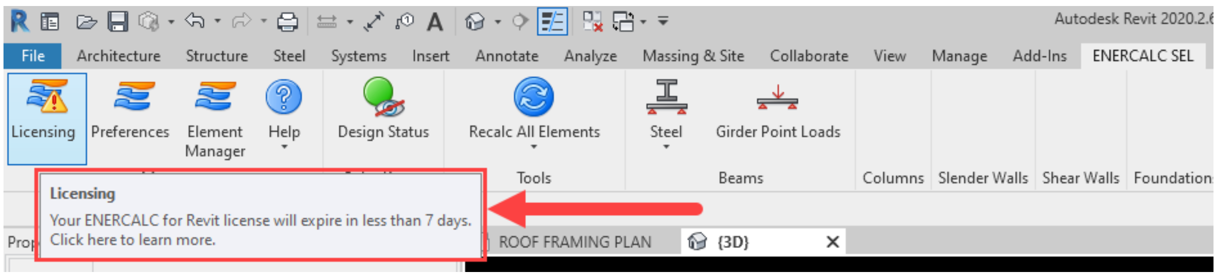
Clicking the button will display a small window with licensing status details.



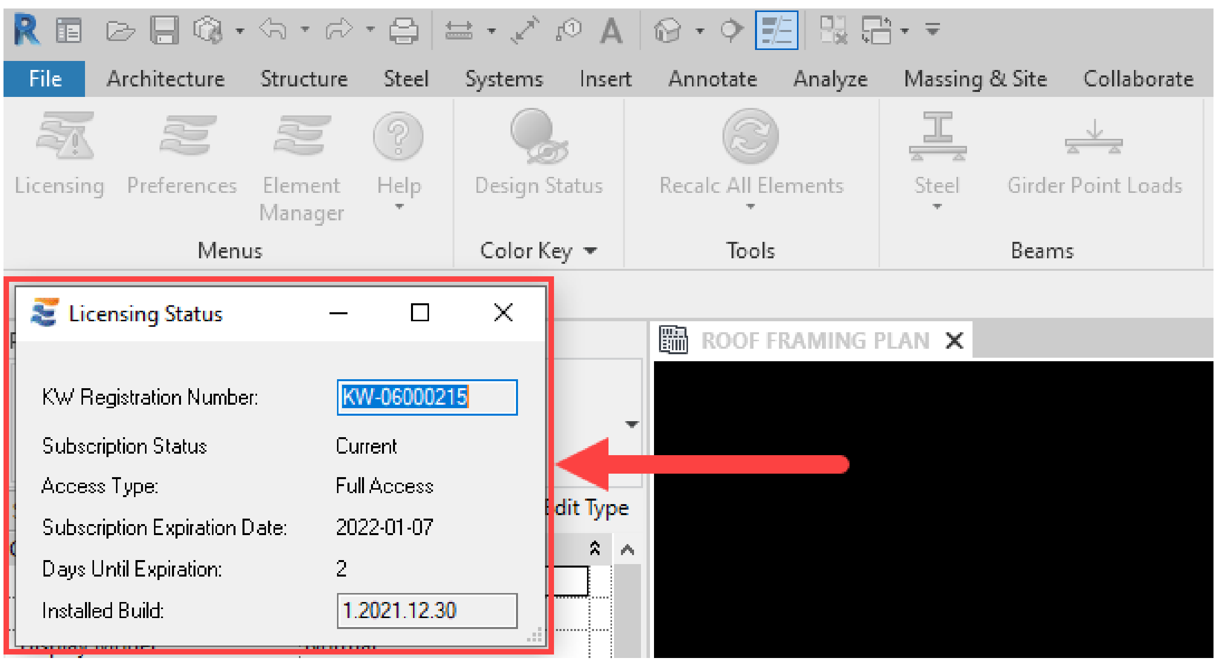
If the user's ENERCALC for Revit license is about to expire in 7 days or less, a "Licensing" button will appear on the ribbon bar as a reminder. In this case, the button icon is decorated with a "warning" symbol to remind the user that expiration is impending.



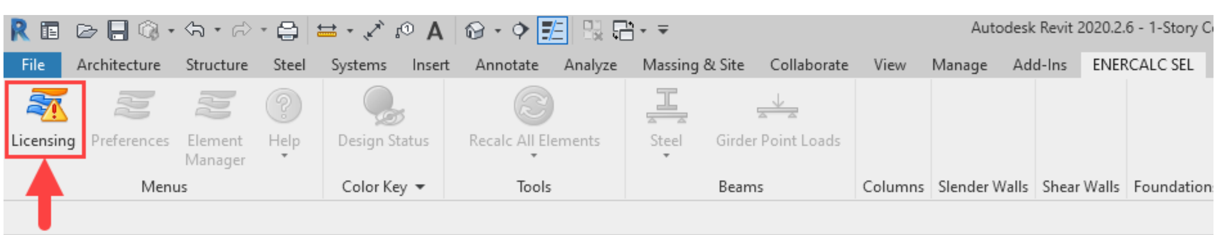
Hovering the cursor over the button displays a tooltip balloon with detailed information.



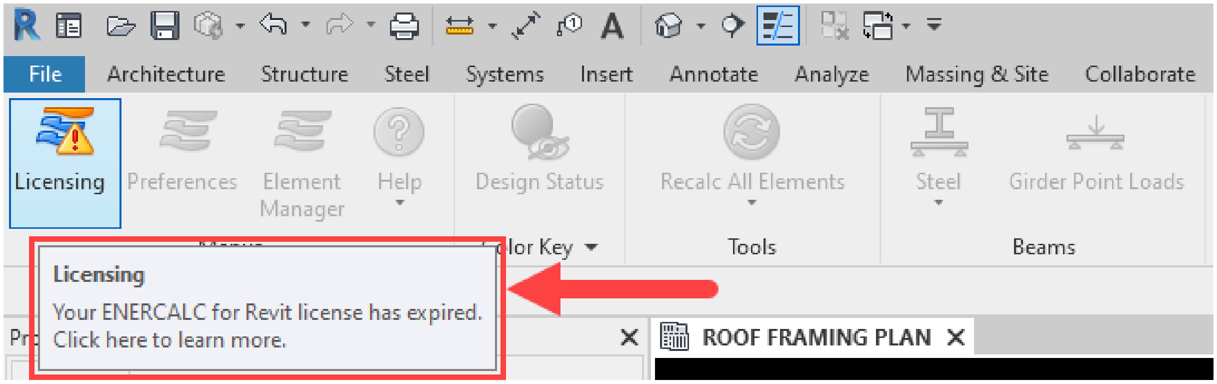
Clicking the button will display a small window with licensing status details.



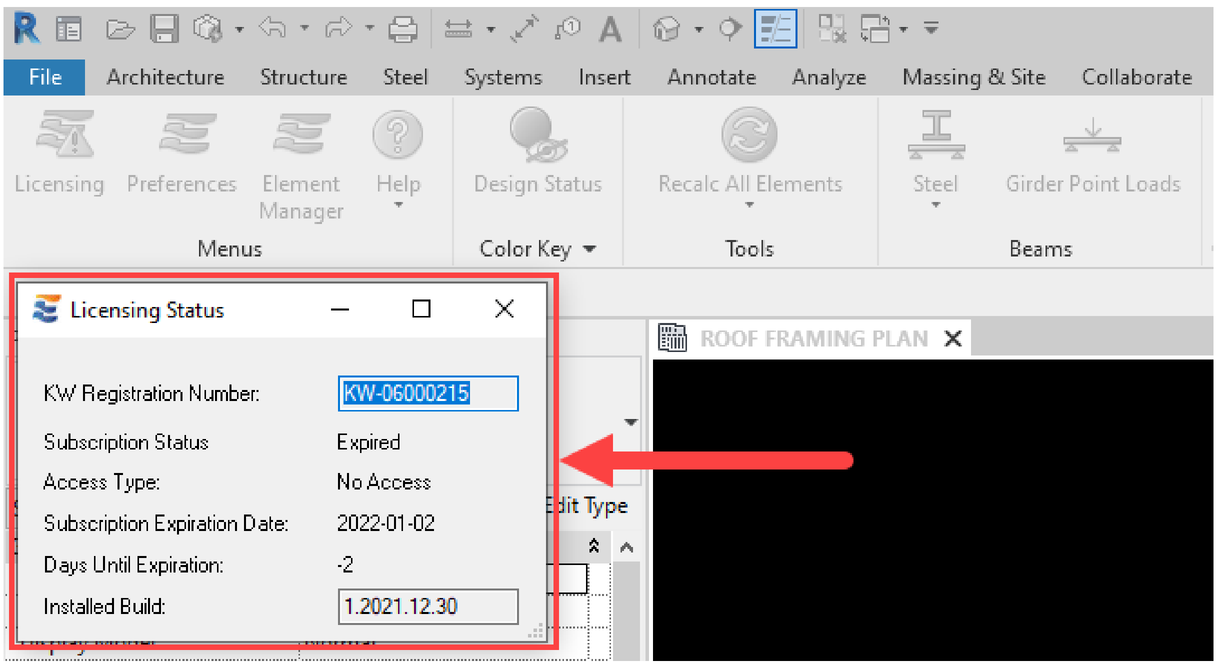
If the user's ENERCALC for Revit license has expired, a "Licensing" button will appear on the ribbon bar as a reminder. In this case, the button icon is decorated with a "warning" symbol to remind the user that the license has expired and all other ribbon bar icons will be disabled.



Hovering the cursor over the button displays a tooltip balloon with detailed information.

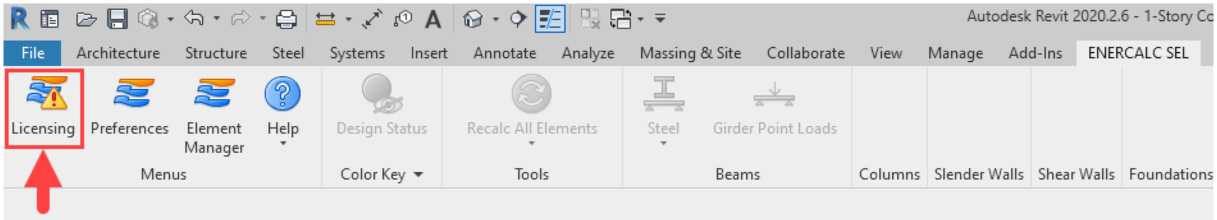


Clicking the button will display a small window with licensing status details.

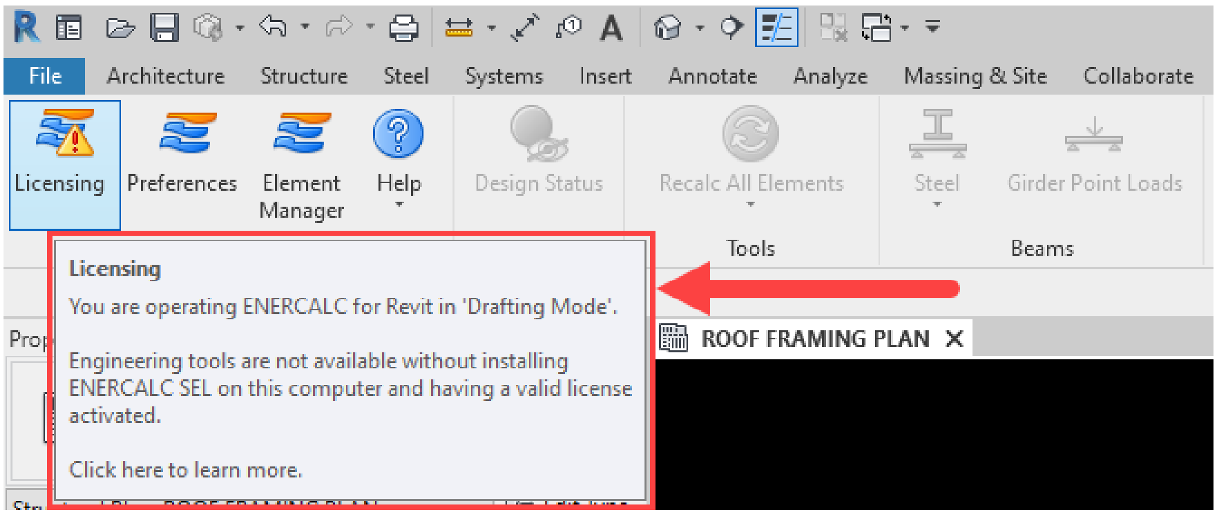


If ENERCALC SEL is not installed on the local machine, ENERCALC for Revit will default to “Drafting Mode”. This mode is specifically intended for use by BIM designers and support staff who interact with the Revit model but do not perform engineering tasks. When operating in “Drafting Mode”, monitoring systems will remain active and change warnings will still be provided when a manual modification of the Revit model may jeopardize agreement between ENERCALC calculations and Revit model geometry.

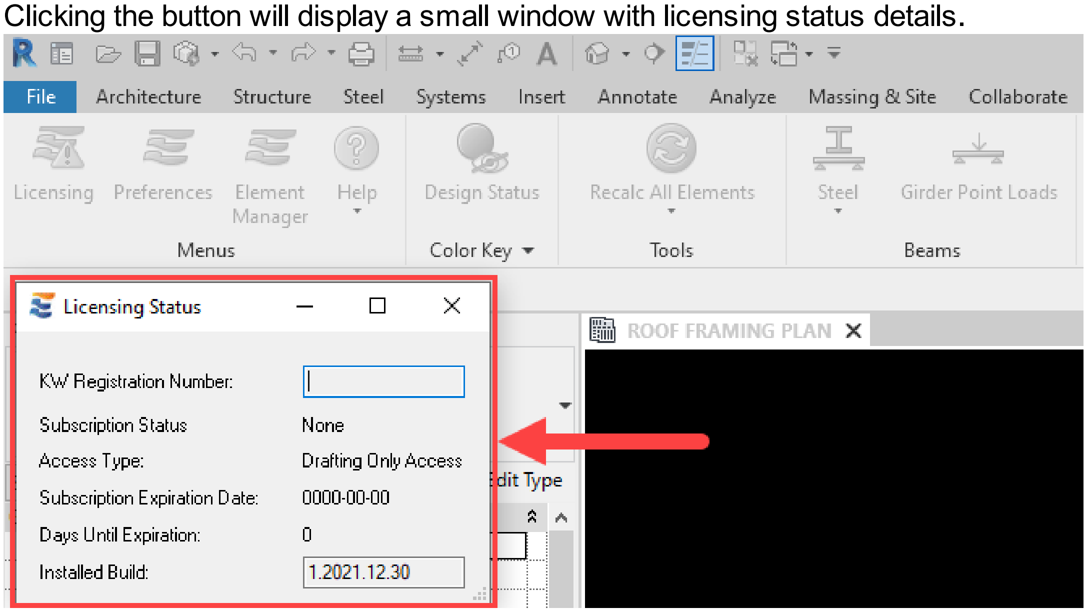
When running in “Drafting Mode”, a “Licensing” button will appear on the ribbon bar as a reminder. In this case, the button icon is decorated with a “warning” symbol to remind the user that all engineering features are disabled.



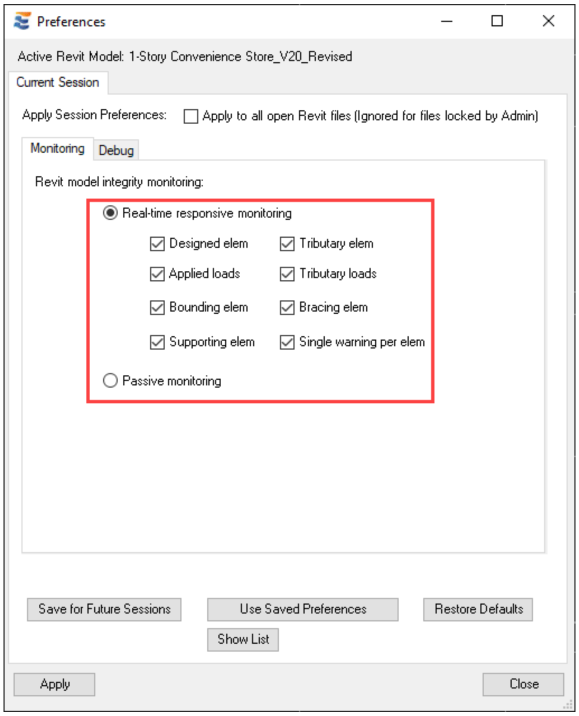
Hovering the cursor over the button displays a tooltip balloon with detailed information.



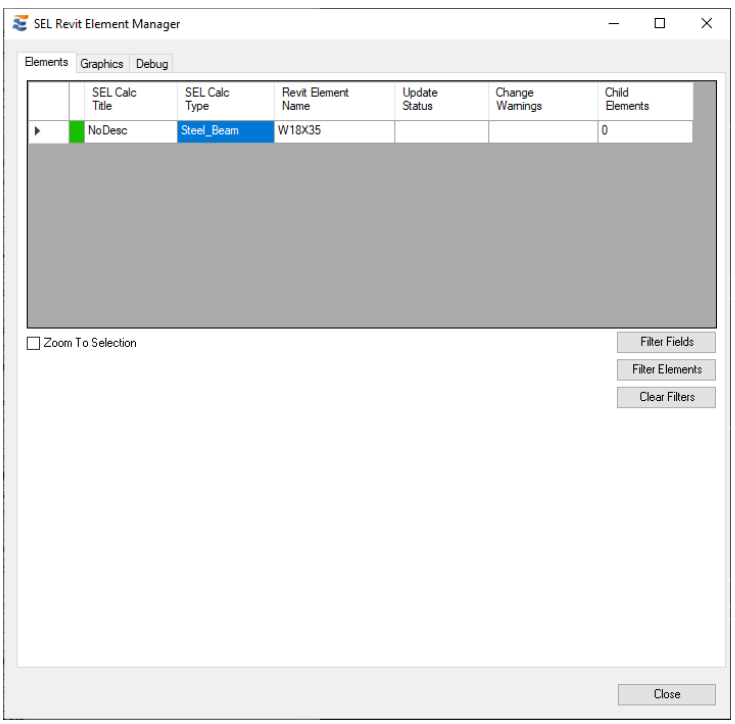
Clicking the button will display a small window with licensing status details.



When operating ENERCALC for Revit in “Drafting Mode”, the user will be presented with limited views and controls in the Preferences and Element Manager windows. In the Preferences menu, the only controls available will be the option to filter pop-up notifications or toggle to passive monitoring mode.



In the Element Manager window, the user will be allowed to view the calculations present on the Revit model, but will not be able to view details or interact with the calculation in any way.



Each of these windows is discussed in greater detail in the appropriate sections.

3.7.1 Unexpected License Status

If after launching Revit and opening a Revit project the ENERCALC for Revit ribbon bar shows an unexpected license status, use the following steps to troubleshoot:

1.) If ENERCALC for Revit shows "Drafting Mode" unexpectedly, verify that ENERCALC SEL is installed on the local machine.

2.) If ENERCALC for Revit shows an unexpected expiration status:

a.) Close the applicable version of Revit.

b.) Close and reopen ENERCALC SEL.

c.) Reopen the applicable version of Revit.

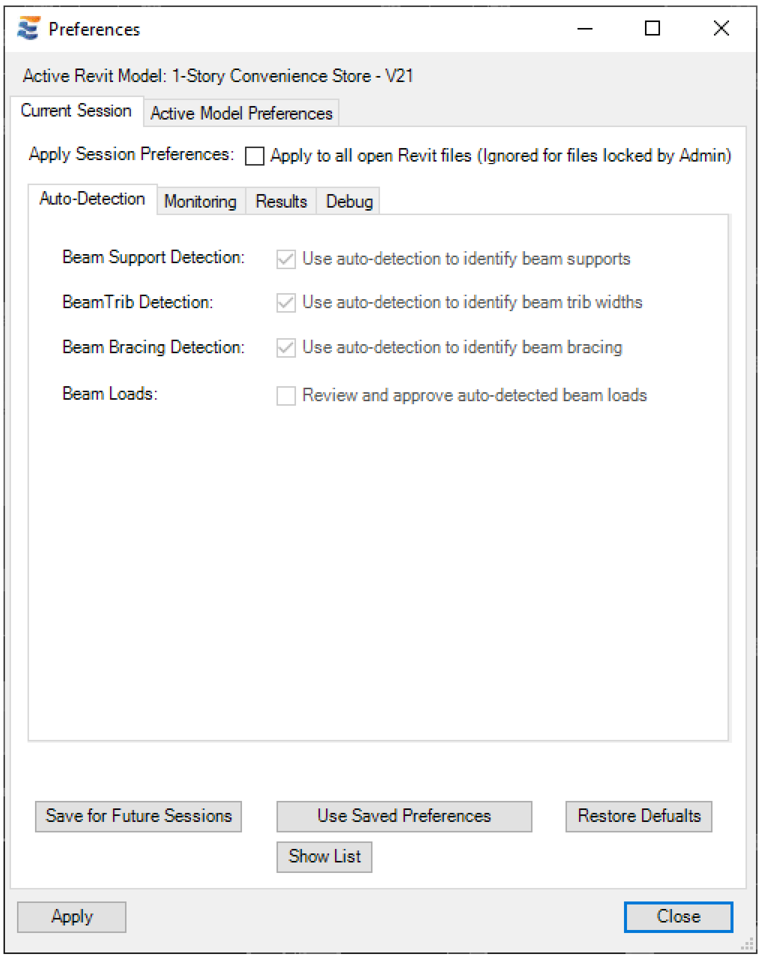
d.) If the expiration status persists, check your subscription information to verify the proper expiration date.

e.) If the expiration status displayed by ENERCALC for Revit does not agree with your ENERCALC subscription expiration date, contact ENERCALC via support@enercalc.com for assistance.

Part IV

4 Navigating the Preferences Menu

The ENERCALC for Revit preferences menu is used to control a variety of settings. Each of these settings is discussed in complete detail in the appropriate section of this manual based on the processes they manage. The preferences are broadly separated into “Current Session” and “Active Model”. For cases where the user may have more than one Revit model open at the time, the active model name is indicated at the top of the menu. This is the Revit model to which “Active Model” preference changes will be applied. “Current Session” preferences are application-level settings that will apply to the behavior of ENERCALC for Revit in all open projects within the current instance of Revit.



The “Active Model” preferences are stored within the Revit model, while the session preferences are stored in memory during the session. If the session preferences are saved, they area stored in the Revit model along with the current user’s identity and can be retrieved in future sessions. In workshared projects, each individual user may store there own unique session preferences if desired.

Part V

5 Linking a Revit Project to ENERCALC

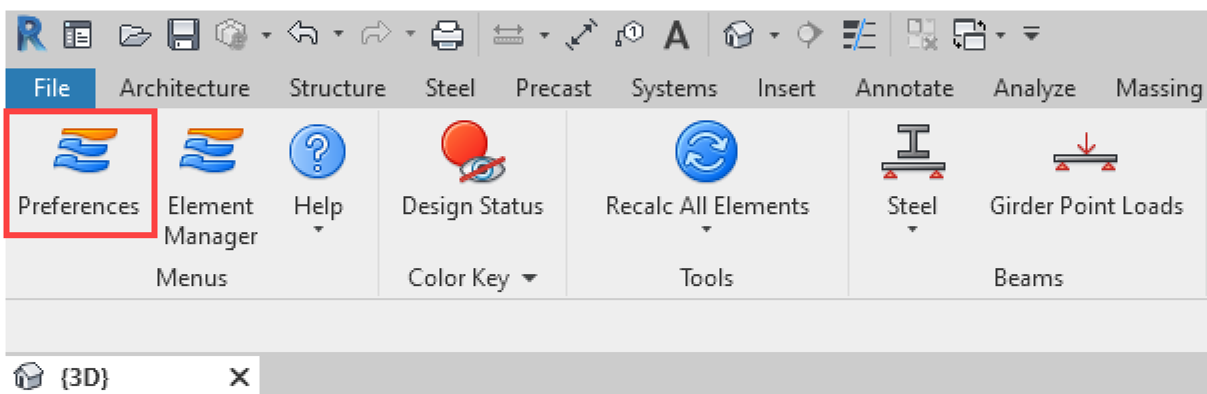
5.1 Basic File Linking

ENERCALC for Revit is driven by manually connecting a Revit project to one or more ENERCALC project files (extension ".EC6") where calculations will be stored. Within a particular Revit project, the following guidelines apply to this process:

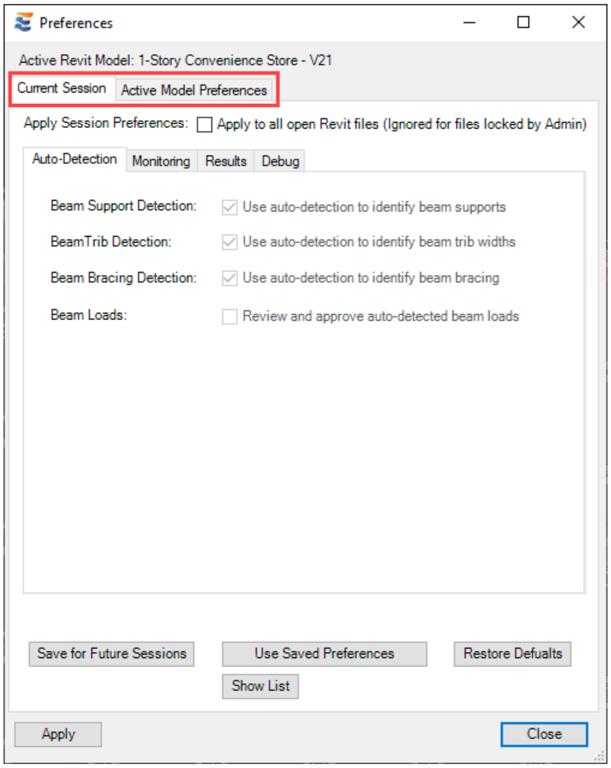
- At any given time, an individual user is allowed only **one** linked .EC6 file.
- Any number of unique individual users may specify their own linked .EC6 file.
- ENERCALC for Revit identifies individual users via the unique combination of Windows account username and computer name

Any .EC6 file that is linked to a particular Revit project will be automatically associated with the identity of the specific user who linked it. This identity is expressed in the format "username_machine name". The username used for this identity is the user's account username on their local machine, not their Autodesk Revit account username.

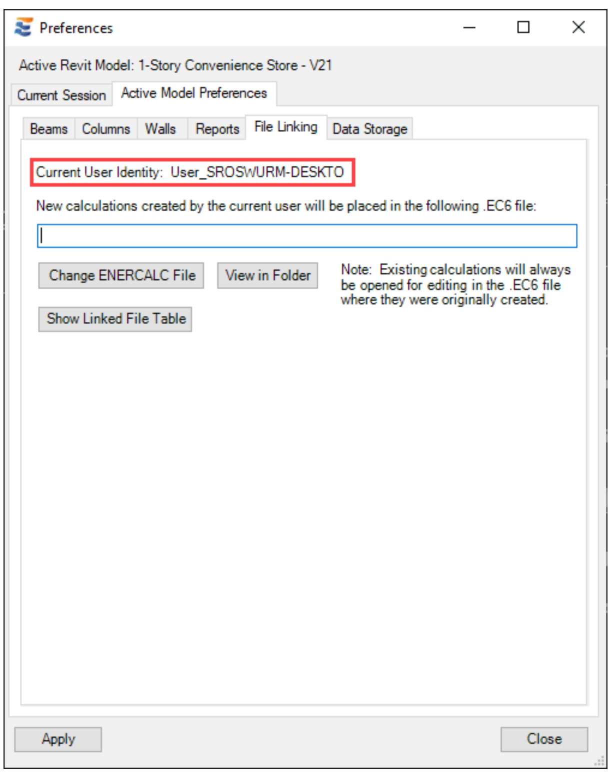
The process of specifying this .EC6 file is managed from the ENERCALC for Revit preferences menu.



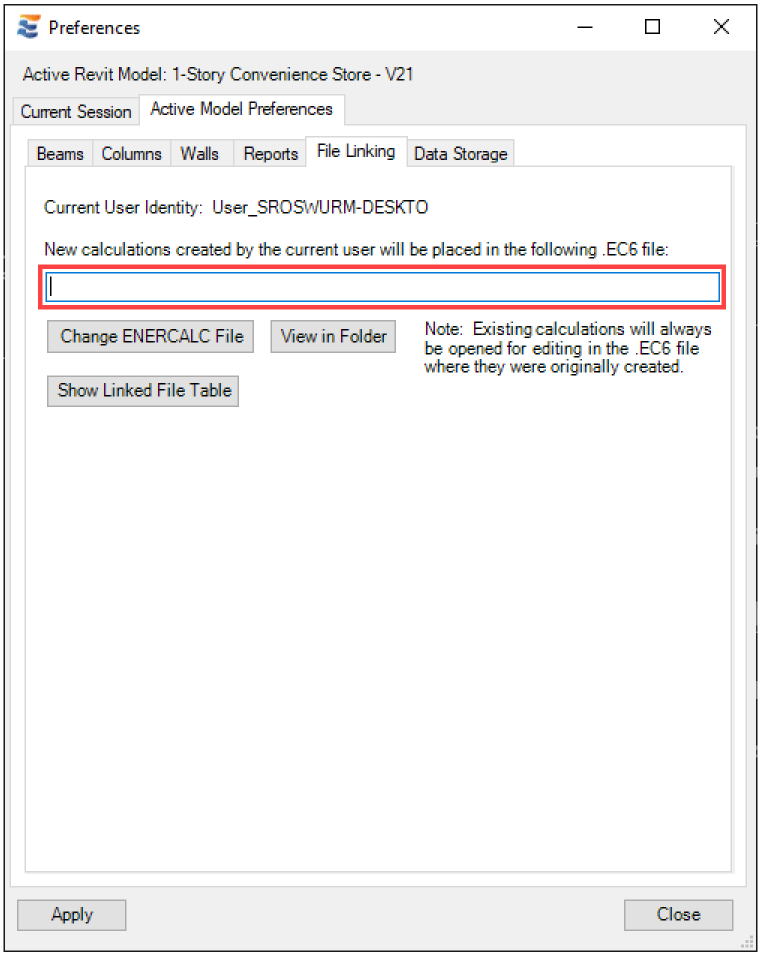
When the Preferences menu launches, the rest of the Revit interface will remain frozen. The form is organized into two main tabs: "Current Session" and "Active Model".



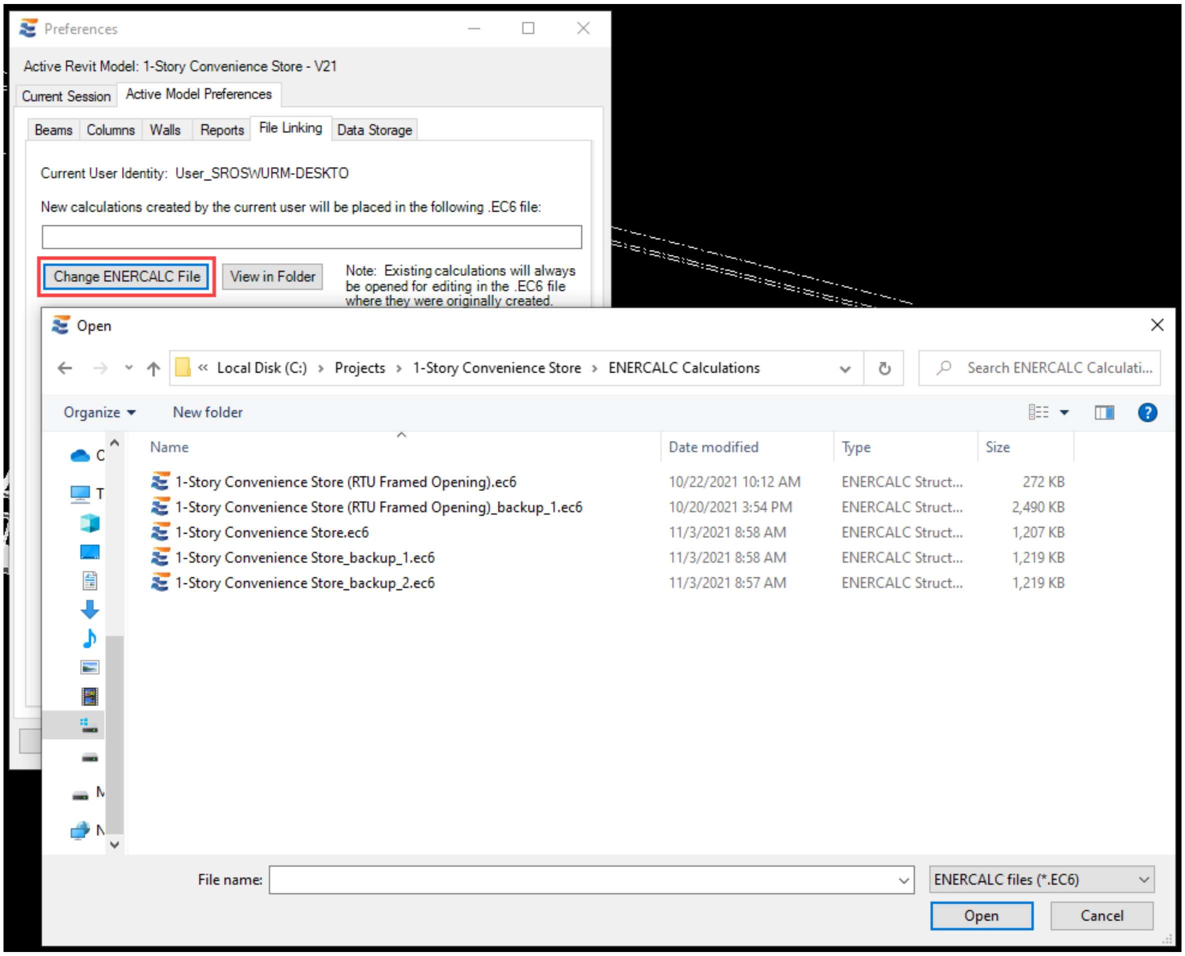
Navigate to Preferences > Active Model Preferences > File Linking. At the top of this tab page, the identity of the current user is indicated. Any file linking changes made here will only impact the current user, whose identity is noted.



Below the user identity is a textbox showing the ENERCALC Structural Engineering Library (.EC6) file that the current user has specified for storing ENERCALC SEL calculations. If no file path is shown, then the current user has not yet linked a file.

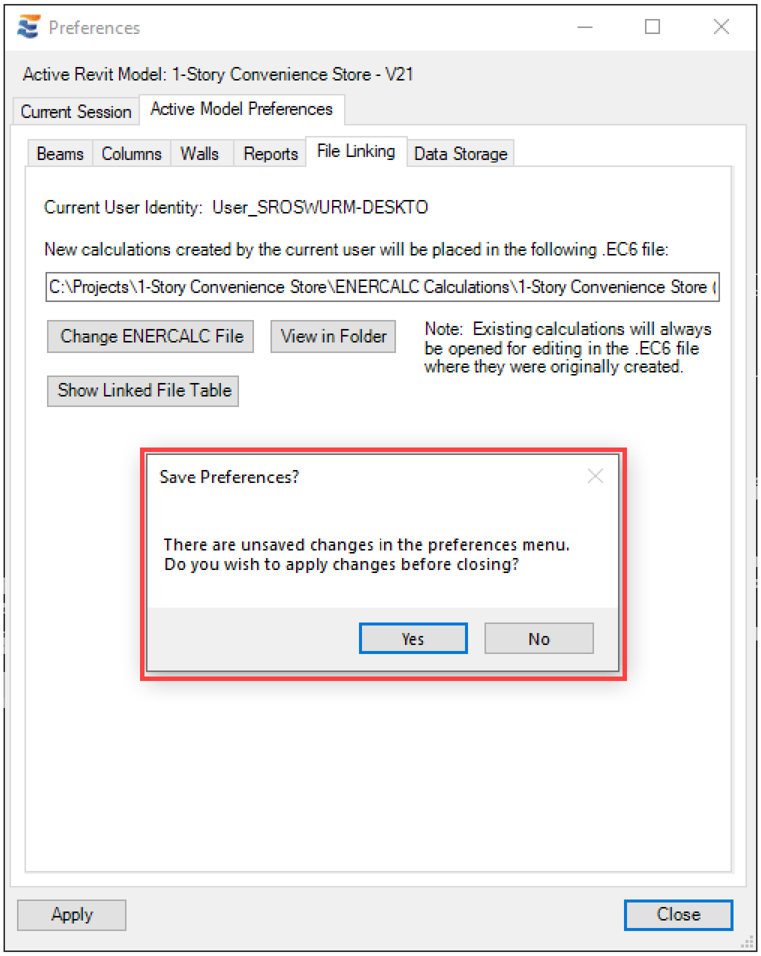


To set or change the linked .EC6 file, click the “Change ENERCALC File” button and use the file browsing dialog to choose an .EC6 file.



It is recommended that .EC6 files be selected from local drive locations if possible. Network drives and file backup services such as Dropbox or similar may not perform as desired. ENERCALC for Revit does not support cloud-stored files at this time.

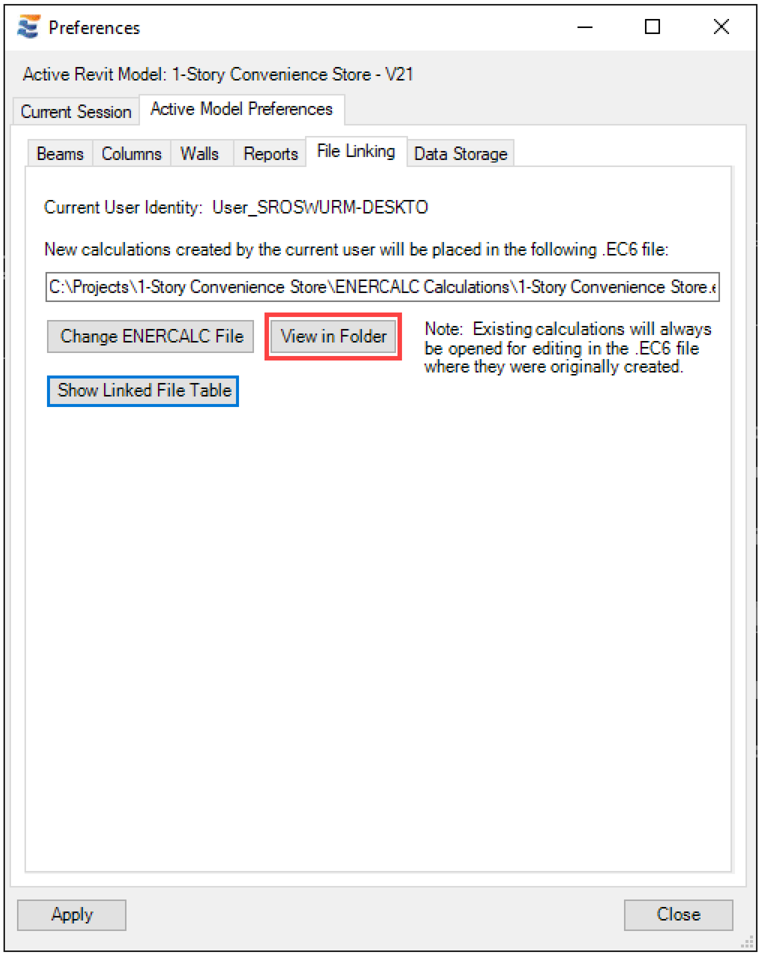
After selecting an .EC6 file to connect to, use the “Apply” button in the lower left corner of the Preferences menu to save your changes. Closing the menu without clicking “Apply” will trigger a warning if changes are found. You may click “Yes” to apply and close, or “No” to discard the Preference changes.



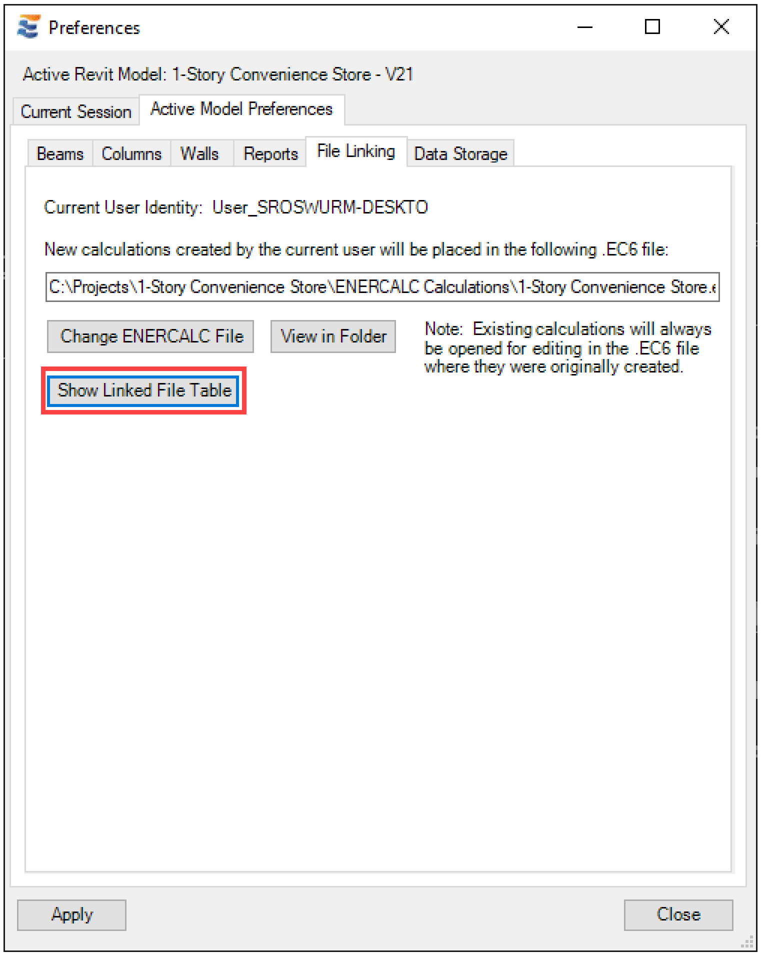
Once the Preference changes have been applied, the Revit model is ready for new calculations to be created. All new calculations created by the current user will be stored in the currently specified .EC6 file. If the .EC6 file is changed in the future, all previously created calculations will remain in the original .EC6 file, but future calculations will be created in the newly specified file.

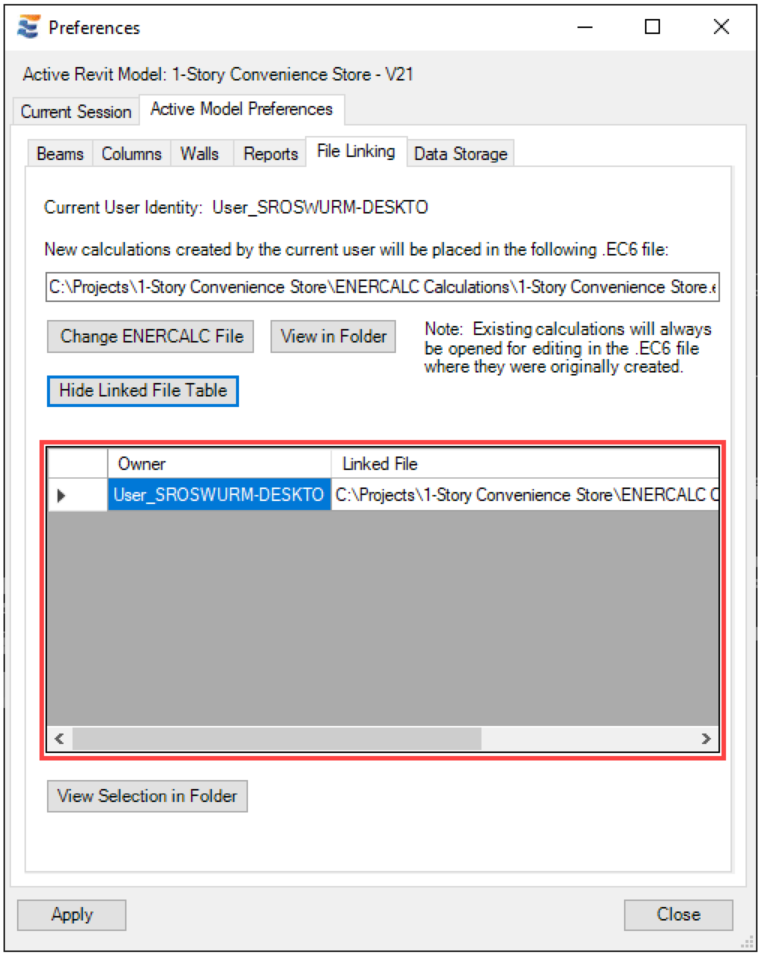
Note that it is **NOT NECESSARY** for a particular .EC6 file to be used ONLY for Revit-linked calculations. Users may choose to specify an .EC6 file that already contains conventional non-Revit-linked calculations. These existing calculations will remain in the file and will be fully functional just as they were prior to the file being linked to Revit.

The currently specified .EC6 file may be directly viewed in its actual folder location using the “View in Folder” button. Clicking this button will launch a new Windows File Explorer window with the location of the current linked file.



When a user accesses the Preferences menu and specifies an .EC6 file for linking, the file is by default associated with the identity of the specific user who selected it. A summary of this information for all users who have linked a file can be viewed using the “Show Linked File Table” button.

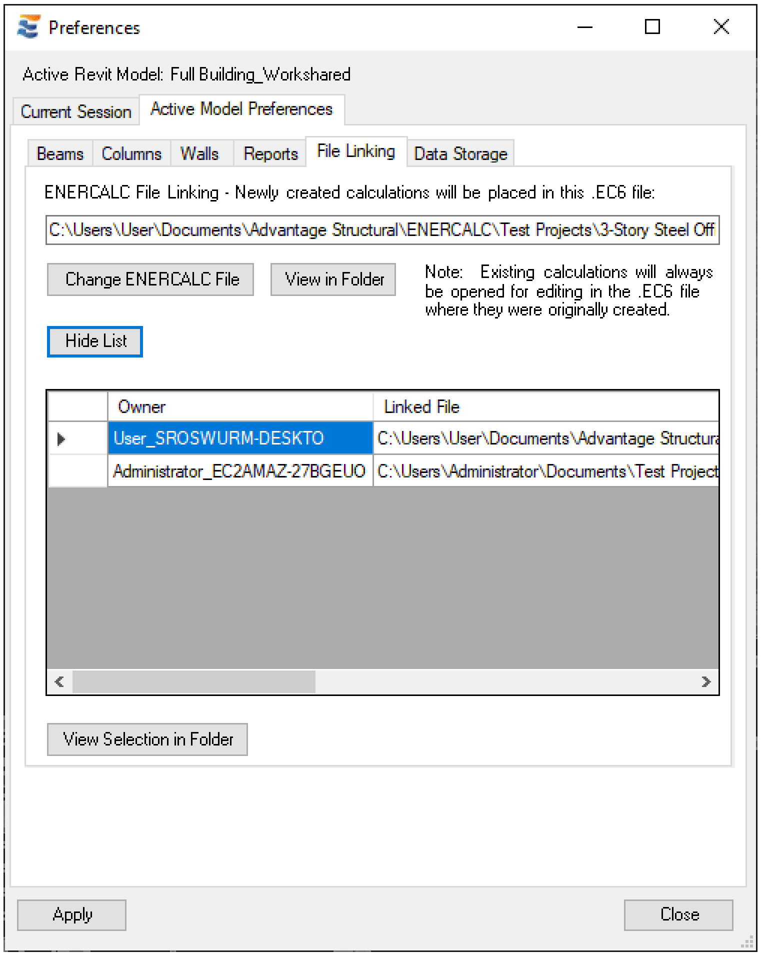

Preferences
Active Revit Model: 1-Story Convenience Store - V21
Current Session
Active Model Preferences
Beams
Columns
Walls
Reports
File Linking
Data Storage
Current User Identity: User_SROSWURM-DESKTO
New calculations created by the current user will be placed in the following .EC6 file:
C:\Projects\1-Story Convenience Store\ENERCALC Calculations\1-Story Convenience Store.e
Change ENERCALC File
View in Folder
Note: Existing calculations will always be opened for editing in the .EC6 file where they were originally created.
Show Linked File Table
Apply
Close



Once visible, the expanded list may be hidden with the “Hide Linked File Table” button.

5.2 Worksharing and Multiple File Linking

In workshared Revit models, it is expected that multiple members of the structural design team may possibly desire to each create their own respective design calculations for various components in the structure. To accommodate this condition, each team member may specify their own .EC6 file. As additional files are linked by various members of the design team, they will be visible in the linked file list.



New calculations created by each individual member of the design team will automatically be stored in the specific .EC6 file that matches their unique user identity (username + machine name).

The identity of the user who created a particular calculation may be easily viewed at any time via the Element Manager, as discussed in "Filtering Fields and Elements 78".

5.3 Re-Pointing a Calculation

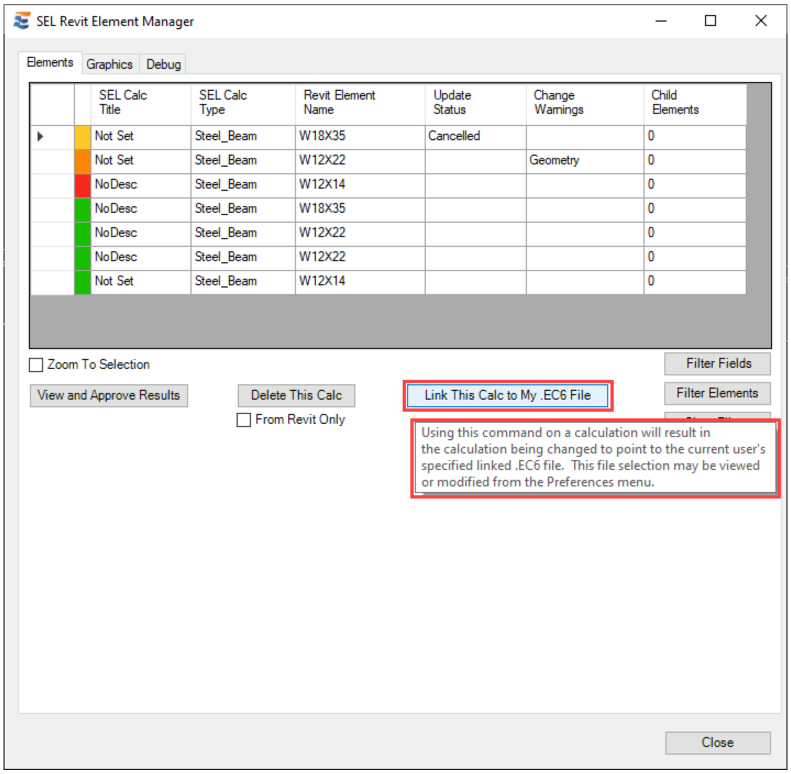
When managing the linking relationships of various calculations, users should remember that although a particular Revit project may be linked to many .EC6 files, each individual element calculation has a one-to-one relationship with a single linked file.

Throughout the natural evolution of a building design project, various factors may contribute to difficulty in accessing a particular linked .EC6 file. This could include:

- Moving of .EC6 files
- Renaming of .EC6 files or project folders

- Adding or removing members of a project team

If at any point in a design project it becomes necessary to re-point a calculation from its original linked .EC6 file to a different file, this can be done from the Element Manager window. Using the “Link This Calc to My .EC6 File” button will result in the currently selected calculation being modified to launch in the user’s current linked .EC6 file.



Note that this will ***NOT*** result in any direct transfer of design information from the prior .EC6 file to the current .EC6 file, and does ***NOT*** result in the calculation being removed from the prior .EC6 file. It will simply allow the calculation to be opened, edited, and saved in a different .EC6 file. The calculation will need to be relaunched from Revit and edited as desired in the new .EC6 file. Design parameters that are managed exclusively from the ENERCALC SEL interface (for example, steel beam flange bracing, etc.) will be set to default values the first time the calculation is launched in its new file location.

Part VI

6 Managing Data Storage

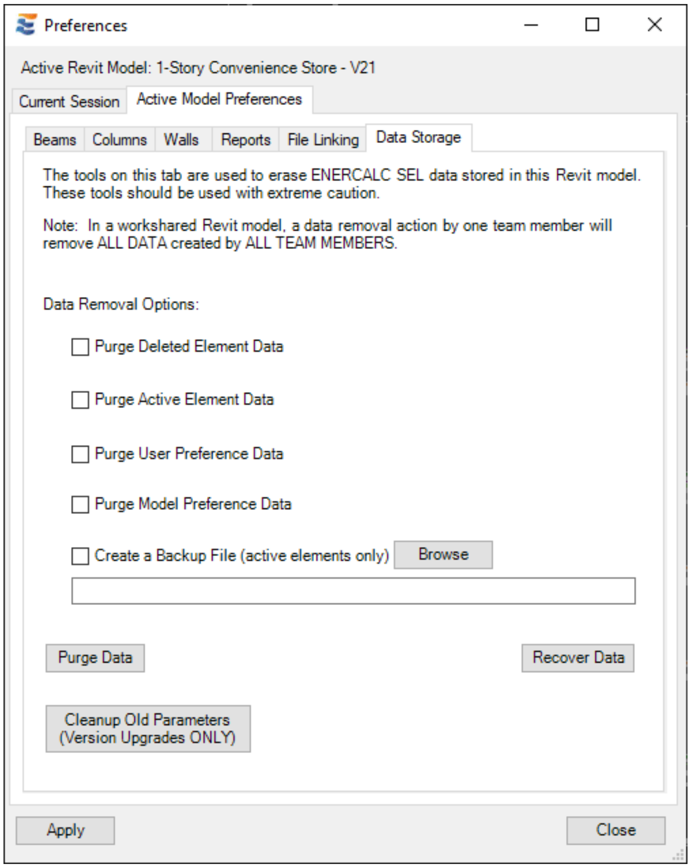
During the process of using ENERCALC for Revit to design structural elements, various data is stored in the Revit model. It could become necessary for the user to remove this data for a number of reasons:

- Restarting a Revit project
- Copying an existing Revit project to launch a new one
- Sharing the Revit file with an out-of-house entity (client or consultant)

This stored data may be removed at any time using Preferences > Active Model Preferences > Data Storage.

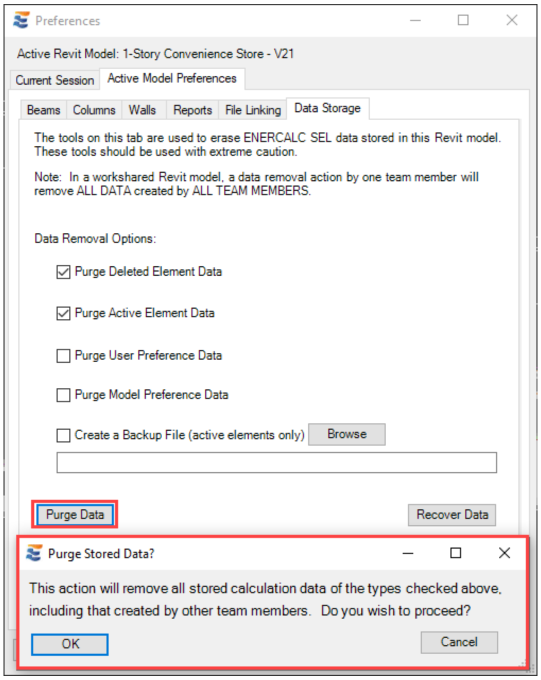
NOTE: The use of the purge "Active Element" checkbox removes ALL Revit-based records of the calculations users have created in the project. At the conclusion of the purge process, the Element Manager summary table will be empty, meaning no calculation data remains.

This purge process should NOT be used on an active working project file where the user plans to KEEP the design information. Although a "Backup" and "Recovery" workflow is provided as described below when purging data, the recommended approach for sharing the Revit model with an out-of-house entity is to save off a purge copy of the file to do the data removal.



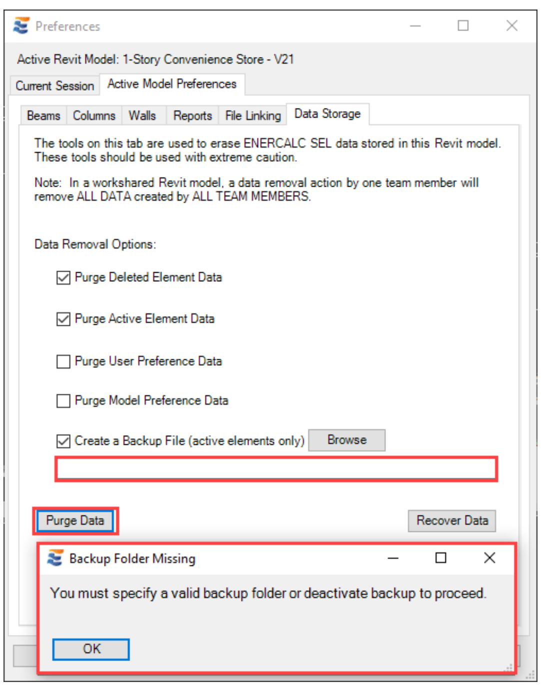
Stored data are removed from the Revit model by using the checkboxes to select which kinds of information to remove, and then clicking the “Purge Data” button.

If desired, the user may also choose to create a plain text (.TXT) backup file of the stored calculation data when purging. If a backup file is to be created, use the “Browse” button to select a folder where the backup will be stored. Triggering the “Purge Data” command will produce a warning to verify that the specified data will be removed.



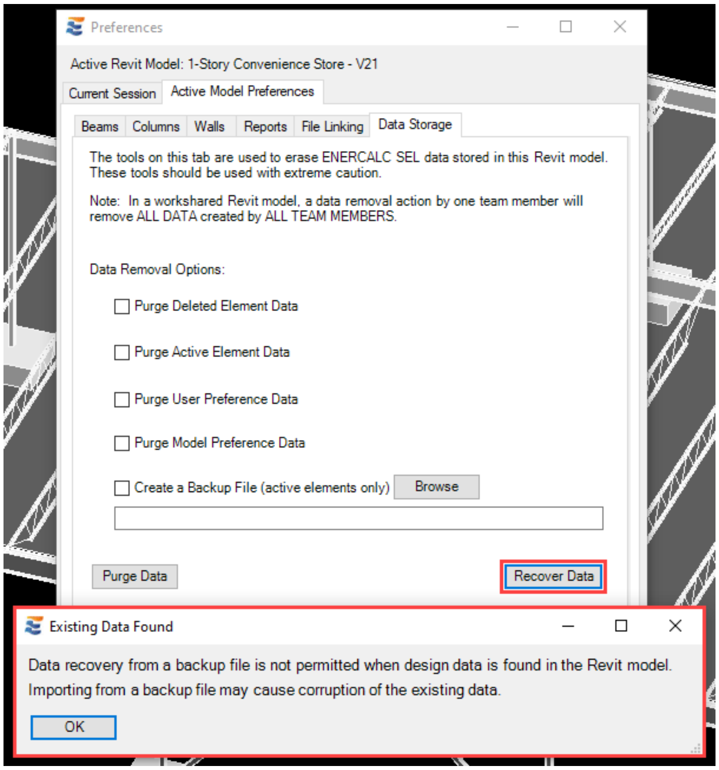
Selecting “Cancel” will abort the data purge.

If the option to “Create Backup…” is activated, then the user-specified folder will be validated before proceeding. If the folder is blank or the directory is invalid, then a warning will abort the purge.

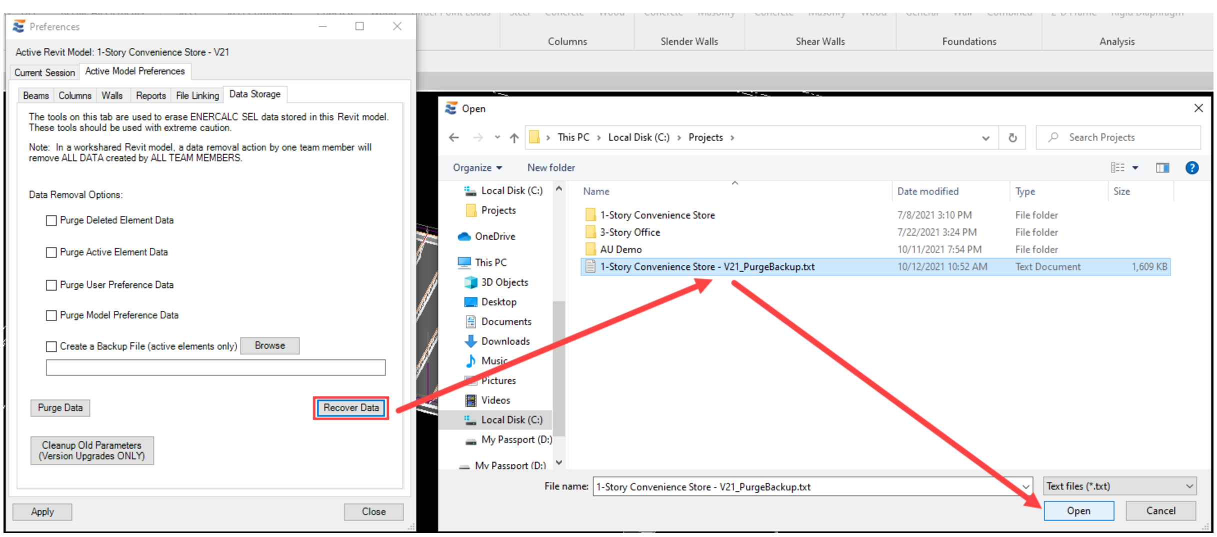


If a backup file was created, the calculation data stored in the backup may be restored to the Revit model using the “Recover Data” button.

Note that data recovery is only permitted on a Revit file that contains no calculation data. If new calculations have been created after the purge, a warning will prevent recovery.



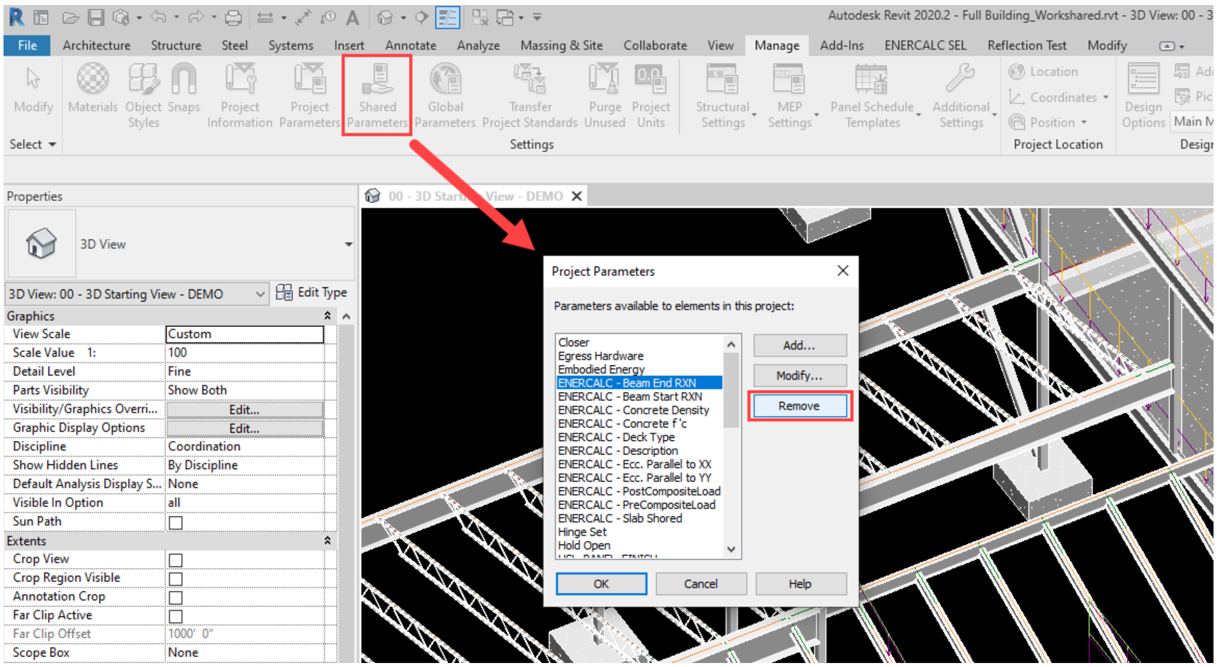
If recovery is permitted, clicking this button triggers an open file dialog for the user to browse to the backup file.



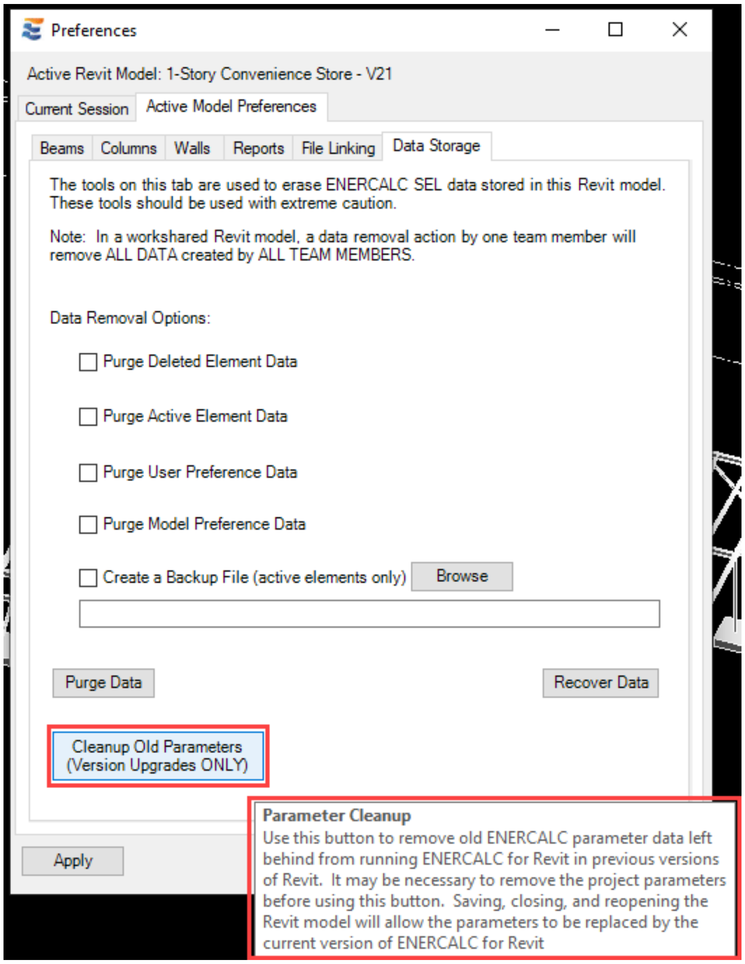
Opening the backup file will result in the data automatically being restored to the Revit model.

ENERCALC for Revit makes use of custom parameters to expose certain design information to user view for tagging, scheduling, and other design tasks. Since different versions of Revit contain different core capabilities for parameter behavior, it may sometimes be necessary to clean up the parameter data after performing a version upgrade on a Revit model.

In cases where there is a known version conflict with respect to parameter behavior (i.e., from Revit 2019 to Revit 2020), parameters with the outdated behavior may be manually removed via the native Revit manage tab on the ribbon bar.



Once the outdated parameters created in a previous version of Revit have been removed, the “cleanup” control provided in the ENERCALC for Revit Preferences menu will remove any traces of these parameters from the Revit model database to prepare for the new parameters to be created.



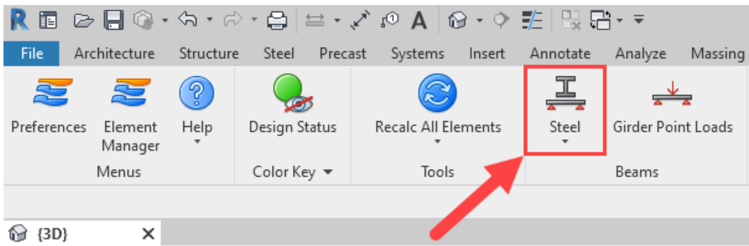
Once the parameter cleanup control has been used, simply save, close, and reopen the Revit model. Upon reopening, the appropriate parameters will be recreated in the current version of Revit.

Note that any information stored in the parameters will be erased when the parameters are removed and recreated. In the case of user-editable parameters used for design inputs, these values will need to be reset manually. For non-editable parameters populated with calculation results, the values will refresh automatically when the calculation is recalculated.

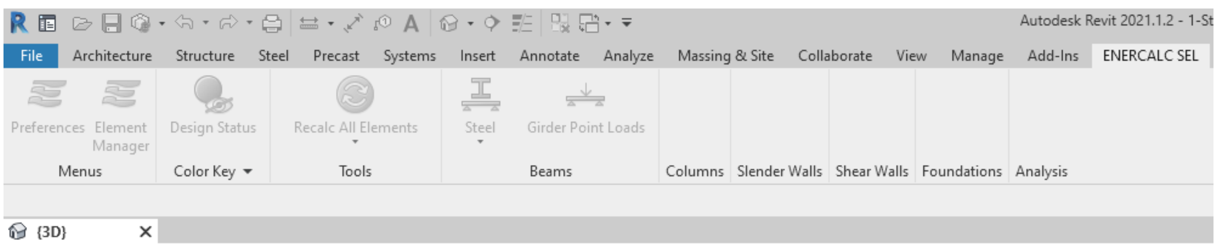
Part VII

7 Launching and Editing ENERCALC Calculations

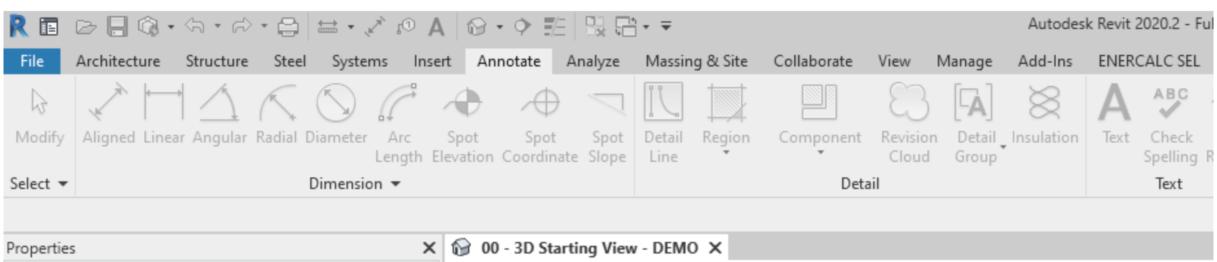
Calculations are launched using the controls in the ENERCALC ribbon bar in the Revit interface. Each individual button on the ribbon bar corresponds to launching a calculation of the type shown. Clicking one of these buttons will cause the Revit interface to enter a user selection mode.



Upon entering selection mode, the ribbon bar tools will be disabled.



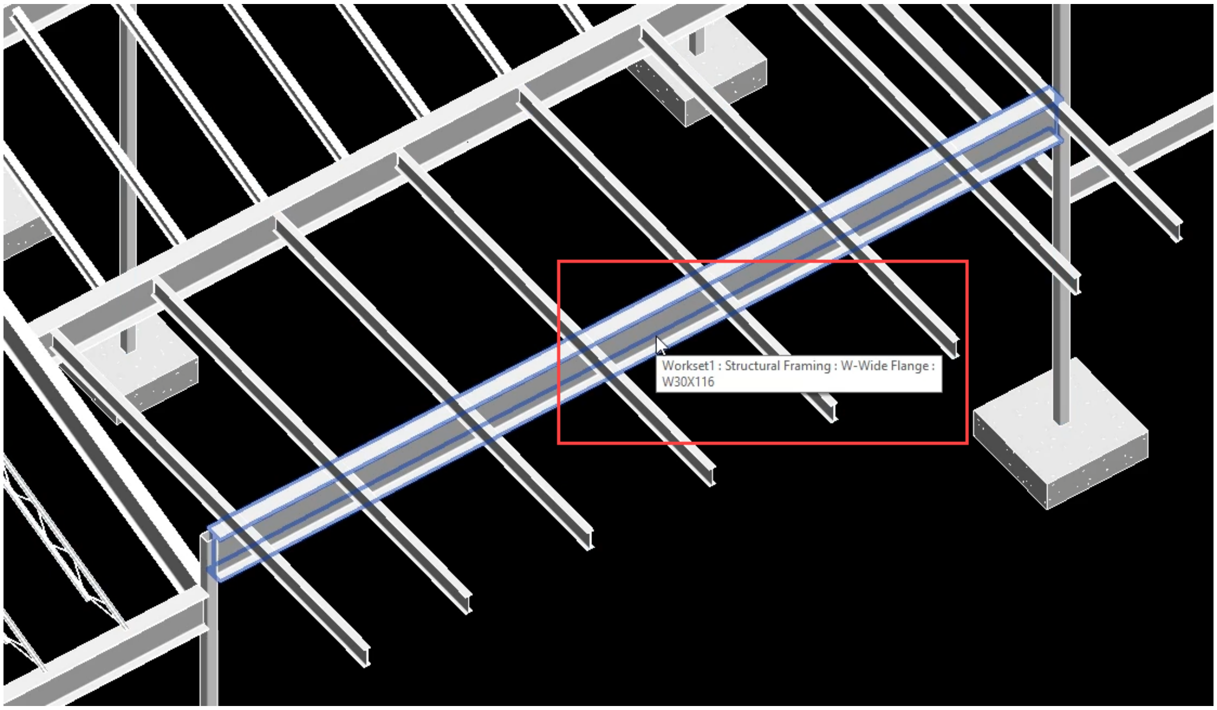
This includes all tools on the ENERCALC ribbon bar tab and on the other native Revit ribbon bar tabs.



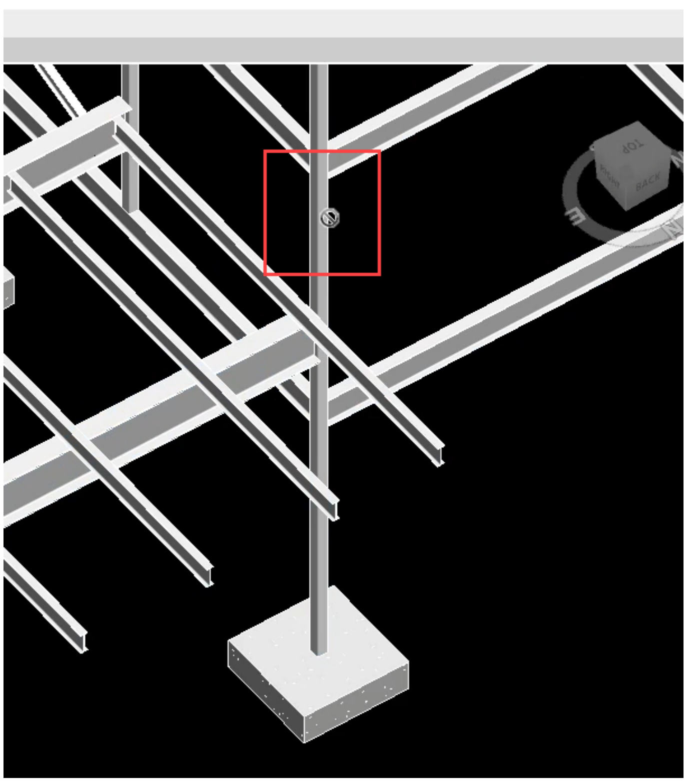
Revit will remain in selection mode until an element is selected or the user cancels the operation by using the “ESC” key on the keyboard.

The selection mode will automatically filter to only allow selection of elements that fit the criteria for the chosen calculation type. For example, a steel column or wood beam will not be eligible for selection if the user is attempting to launch a “Steel Beam” calculation.

Elements that are available for design using the selected calculation type will highlight in the Revit view when the cursor passes over.



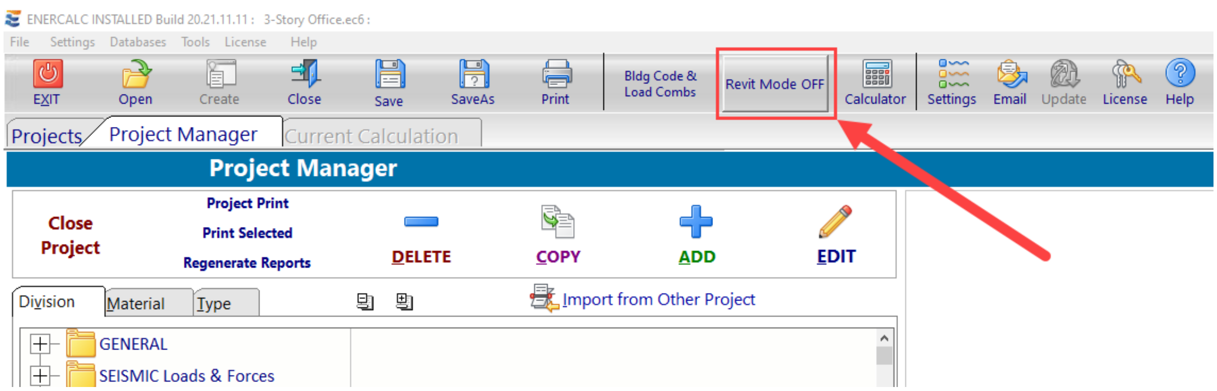
Elements that are not available for design using the selected calculation type will not highlight but will display a disabled cursor in the Revit view when the cursor passes over.



Instead of the order of operations described above, a user may alternatively choose to first select a Revit element and then proceed to the ribbon bar to click the calculation launch control. If the selected element matches the calculation type, then the calculation launch will begin. If the element is not eligible, then no launch will occur and work will not be interrupted by a popup notification that the element was ineligible.

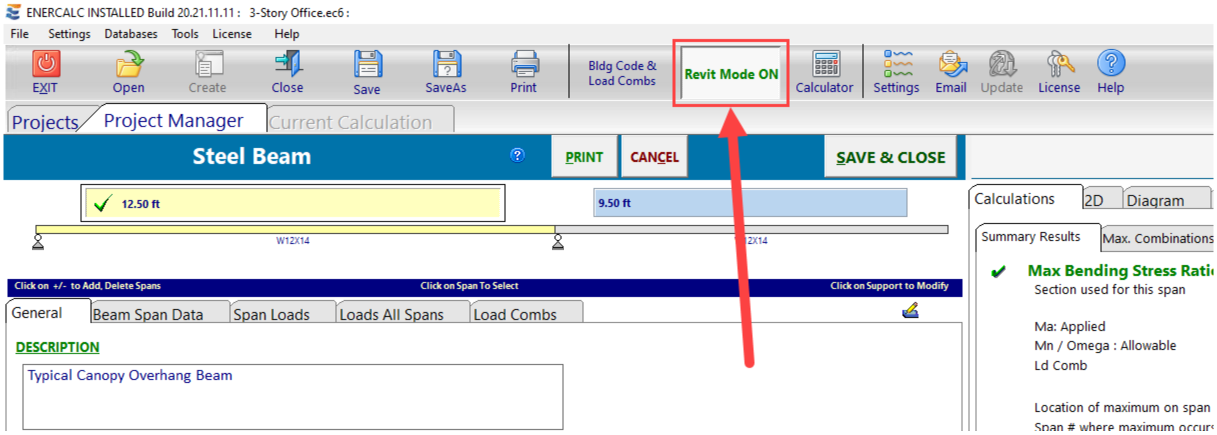
Once the calculation launch process begins, the user will be taken through a series of approval steps to verify critical design data about the element so that ENERCALC for Revit can build a complete ENERCALC SEL calculation. Once the verification process in complete, the calculation will automatically appear in the ENERCALC SEL interface. The following provisions apply:

- If ENERCALC SEL is already running with “Revit Mode OFF”, then a calculation cannot be launched. ENERCALC SEL will not respond when the Revit launch operations is finished.



If the user’s ENERCALC subscription includes Revit integration, the mode may be manually toggled to “Revit Mode ON” by simply clicking the button. Re-launching the calculation from Revit will allow it to populate in the ENERCALC SEL interface after “Revit Mode ON” is set.

- If ENERCALC SEL is already running with “Revit Mode ON”, then ENERCALC SEL will automatically retrieve the appropriate project and display the calculation.
- If ENERCALC SEL is closed at time of launch, it will be opened automatically. The appropriate linked .EC6 project will be located and opened automatically. Users will note on startup that ENERCALC SEL has automatically been toggled to “Revit Mode ON”. ENERCALC SEL will automatically retrieve the appropriate project and display the calculation.


ENERCALC INSTALLED Build 20.21.11.11 : 3-Story Office.ec6 :
File Settings Databases Tools License Help
EXIT Open Create Close Save SaveAs Print
Bldg Code & Load Combs
Revit Mode ON
Calculator Settings Email Update License Help
Projects Project Manager Current Calculation
Steel Beam
PRINT CANCEL SAVE & CLOSE
12.50 ft
9.50 ft
W12X14
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
General Beam Span Data Span Loads Loads All Spans Load Combs
DESCRIPTION
Typical Canopy Overhang Beam
Calculations 2D Diagram
Summary Results Max. Combinations
Max Bending Stress Rati
Section used for this span
Ma: Applied
Mn / Omega : Allowable
Ld Comb
Location of maximum on span

Part VIII

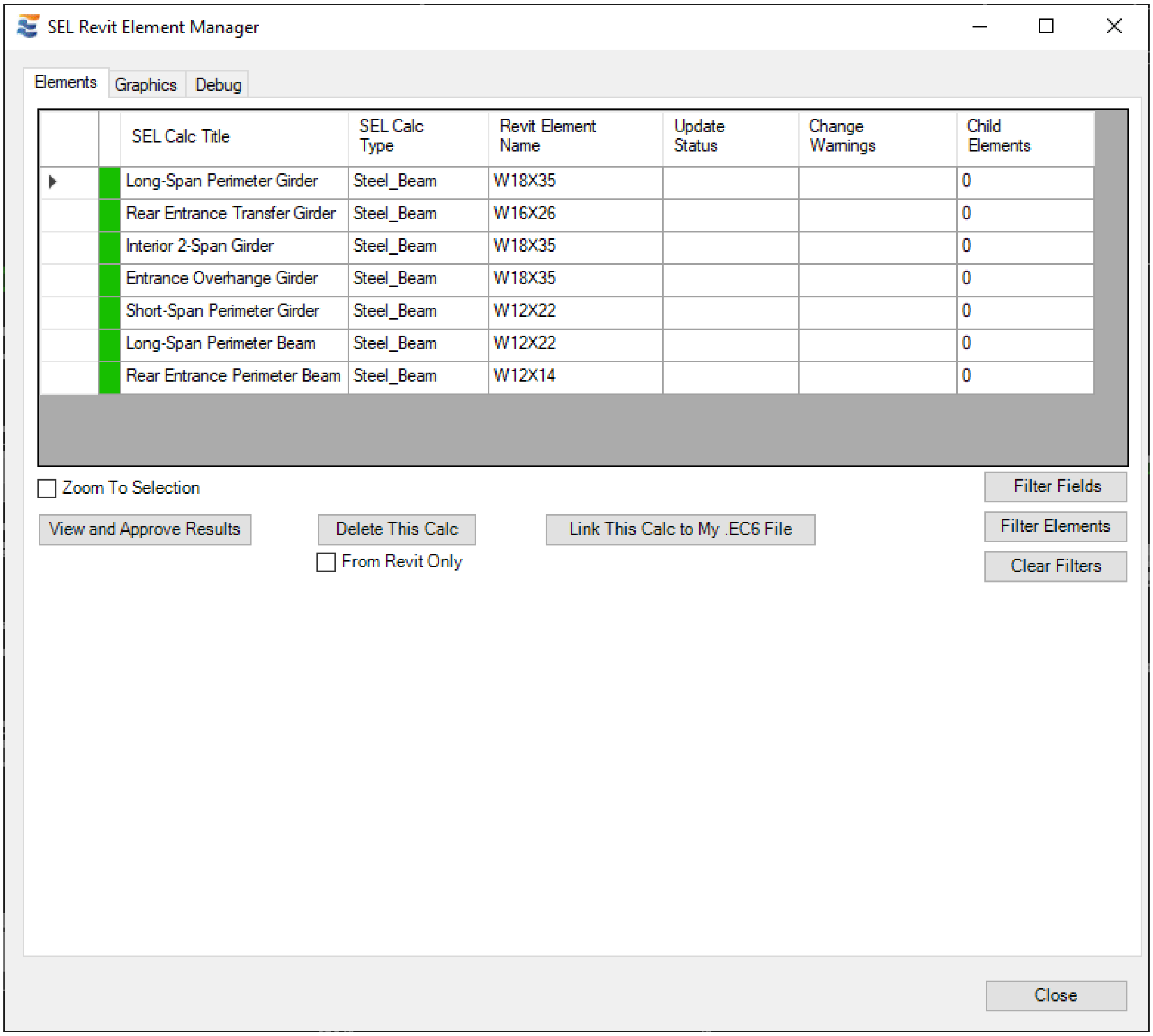
8 Using the Element Manager

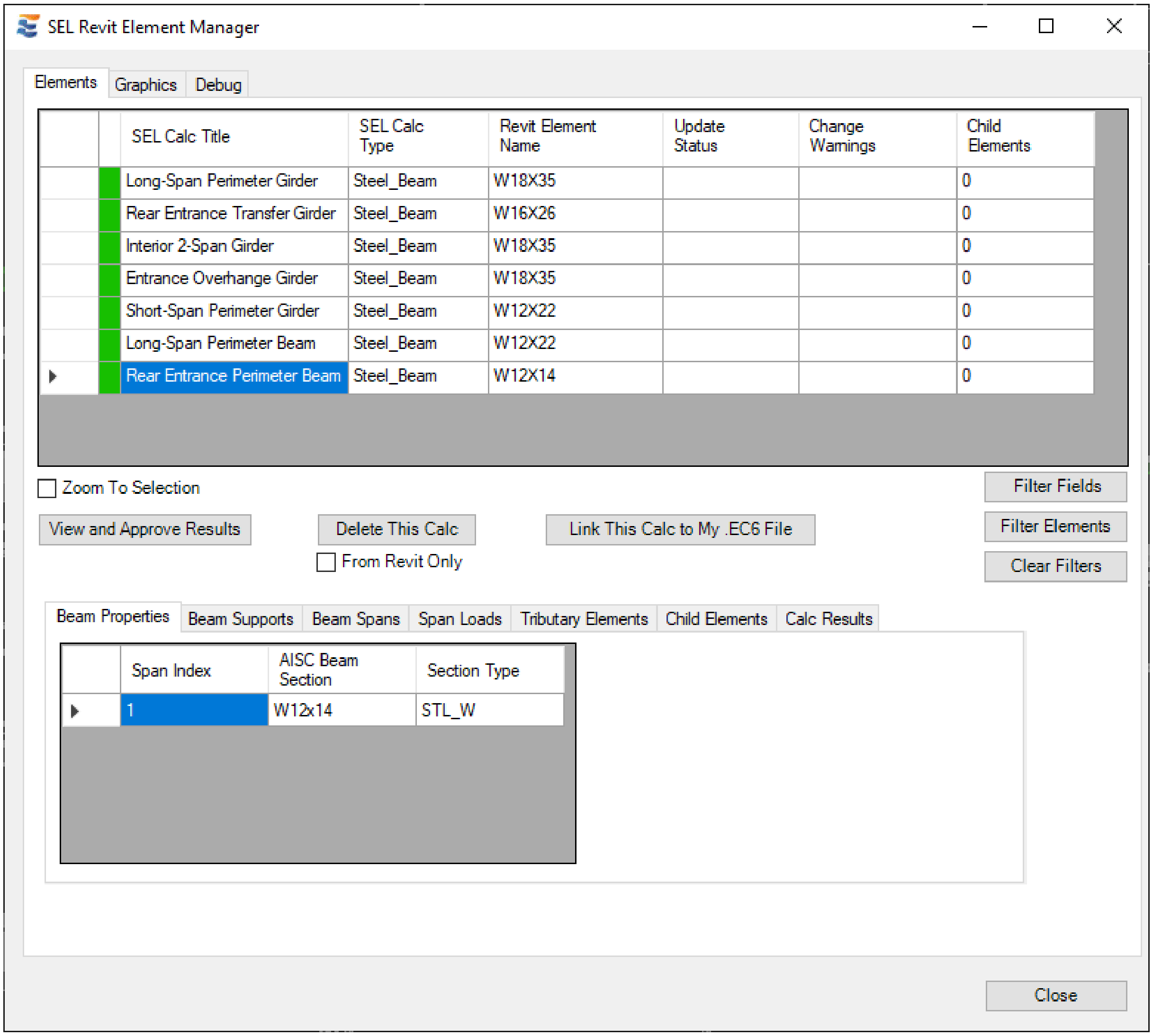
The Element Manager window is the central interface for viewing and managing Revit elements that have been designed using ENERCALC for Revit. The Element Manager does not block access to the Revit graphic interface. The user maintains full use of Revit views and controls while the Element Manager is active.

The menu is organized into three separate tabs:

- Elements
- Graphics
- Debug

The primary tab for use during daily design work is the “Elements” tab. This tab is divided into an upper half and a lower half. The upper portion contains a summary table of all designed elements, while the lower half displays details about the currently selected element.

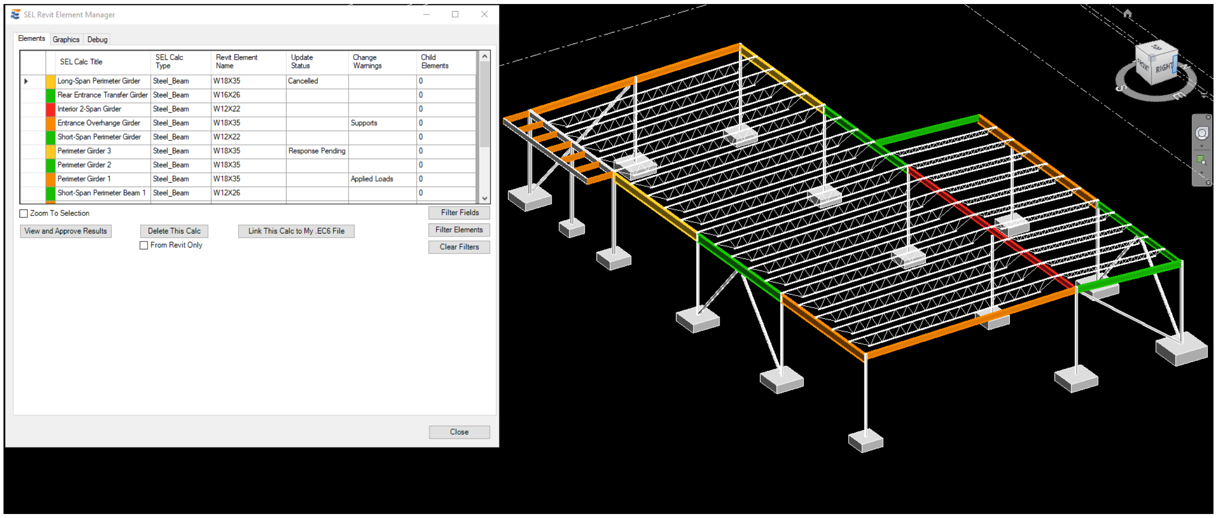

SEL Revit Element Manager
Elements
Graphics
Debug
SEL Calc Title
SEL Calc Type
Revit Element Name
Update Status
Change Warnings
Child Elements
Long-Span Perimeter Girder Steel_Beam W18X35 0
Rear Entrance Transfer Girder Steel_Beam W16X26 0
Interior 2-Span Girder Steel_Beam W18X35 0
Entrance Overhange Girder Steel_Beam W18X35 0
Short-Span Perimeter Girder Steel_Beam W12X22 0
Long-Span Perimeter Beam Steel_Beam W12X22 0
Rear Entrance Perimeter Beam Steel_Beam W12X14 0
Zoom To Selection
Filter Fields
View and Approve Results
Delete This Calc
Link This Calc to My .EC6 File
Filter Elements
From Revit Only
Clear Filters
Close



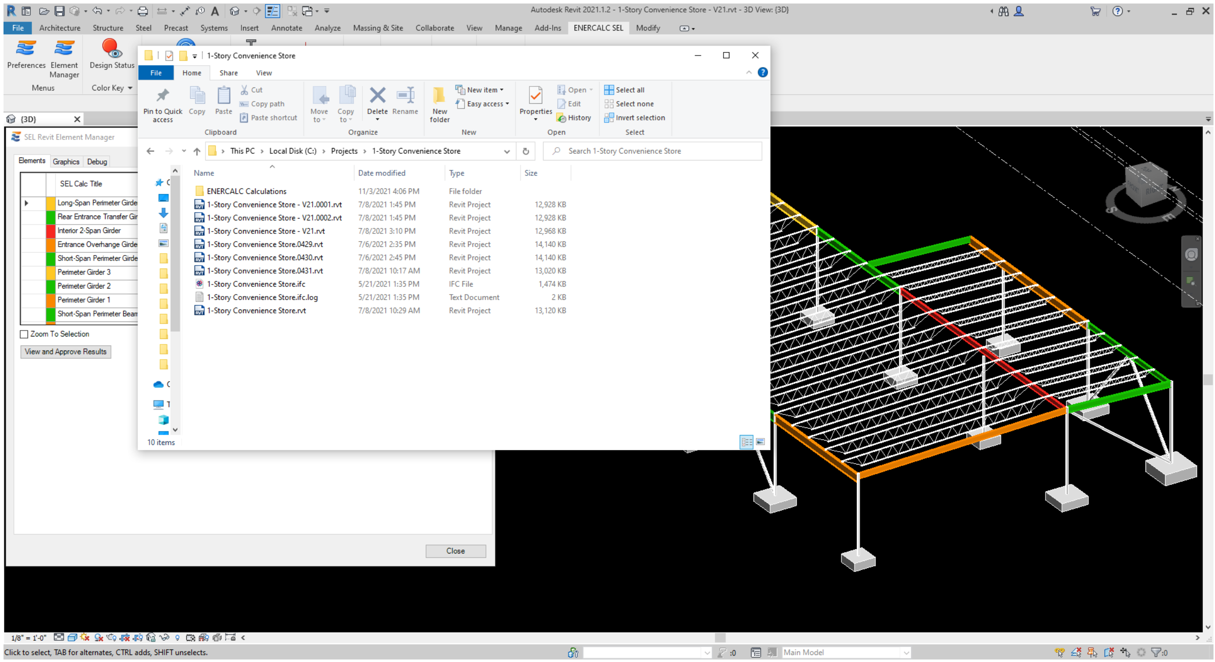
The far-left column of the summary table indicates a color-coded design status of the element's design calculation. This color coding corresponds exactly to the 4-color system used for the ribbon bar status indicator and the view-based highlighting of individual elements:

- **Green** – Calculation is passing its unity check (unity <= 1.0)
- **Yellow** – Calculation is incomplete (i.e., pending or cancelled)
- **Orange** – Calculation has a warning (i.e., Revit geometry changes have caused the element to diverge from its ENERCALC SEL calculation data)
- **Red** – Calculation is failing its unity check (unity > 1.0)

The user-assigned title for each calculation will also be visible if the user has assigned one via the ENERCALC SEL interface. The Element Manager may be closed at any time if desired, but the element information and statuses will update in real time if the window remains open.

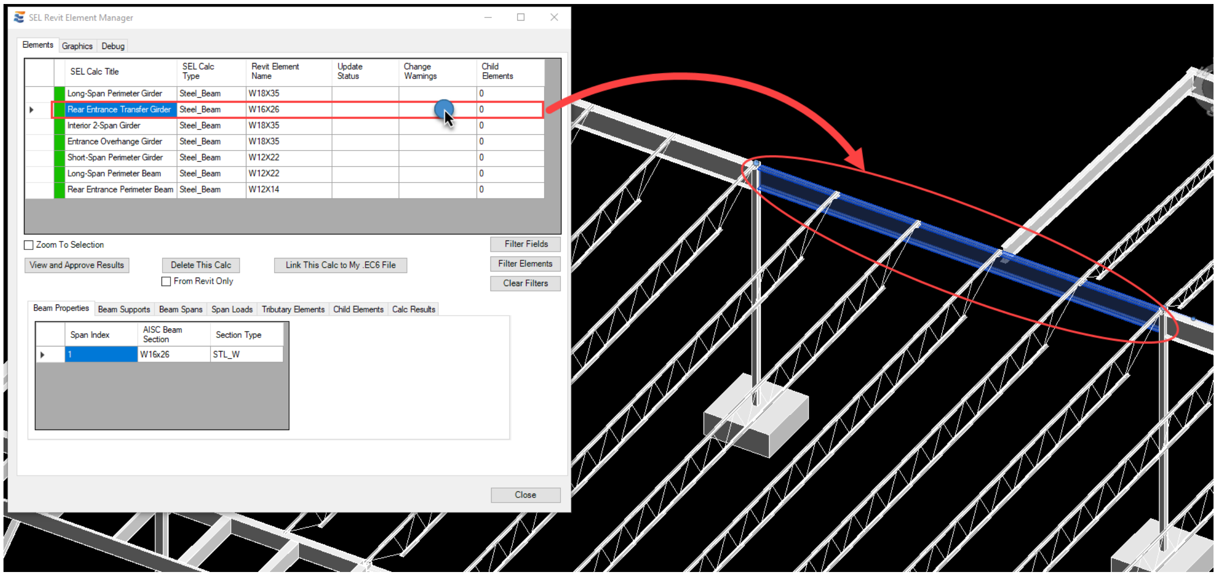


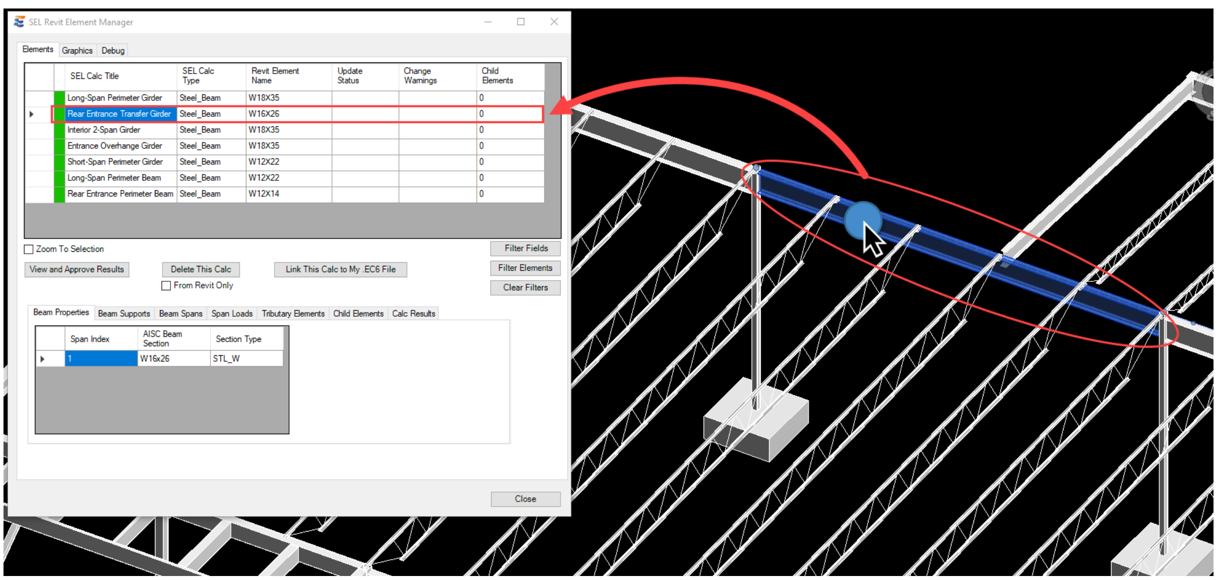
The Element Manager is not constrained to the extents of the main Revit window. It may be dragged to a separate display if desired. However, the Element Manager is a “child” of the main Revit window, meaning that it will minimize and restore along with the main Revit window and will not obscure other programs the user may choose to place in front of Revit on a display.



8.1 Element Selection

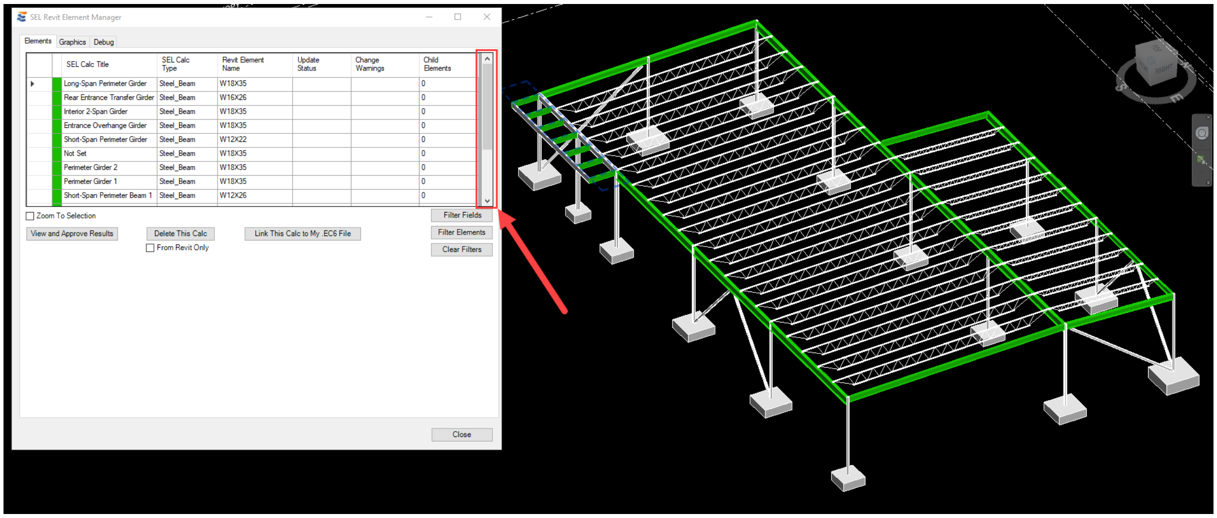
When navigating the Element Manager window, designed elements may be selected by two different methods. Selecting an element by clicking it in the summary causes it to automatically be selected in the Revit interface.



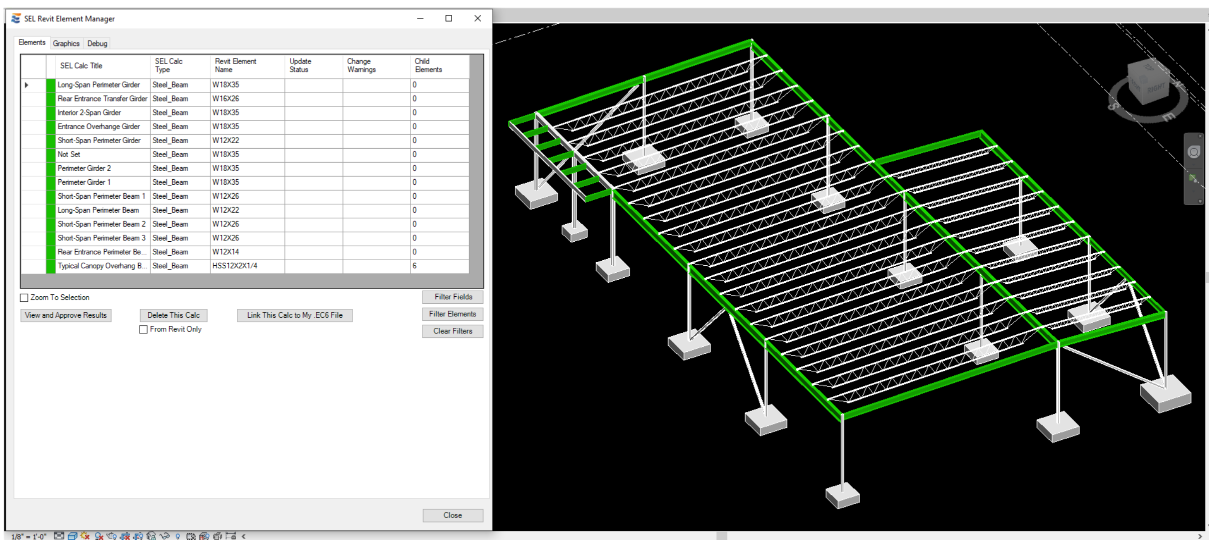


Conversely, selecting an element in the Revit UI will automatically highlight the corresponding calculation in the Element Manager and load its detail info in the lower half of the window.

In a model that contains many designed elements, the Element Manager will automatically display a scroll bar to navigate down the list of elements.



For easier navigation without a scroll bar, the summary table may be enlarged by manually stretching the Element Manager window. When the window is stretched, the lower details area remains the same size and the summary table grows to fit.

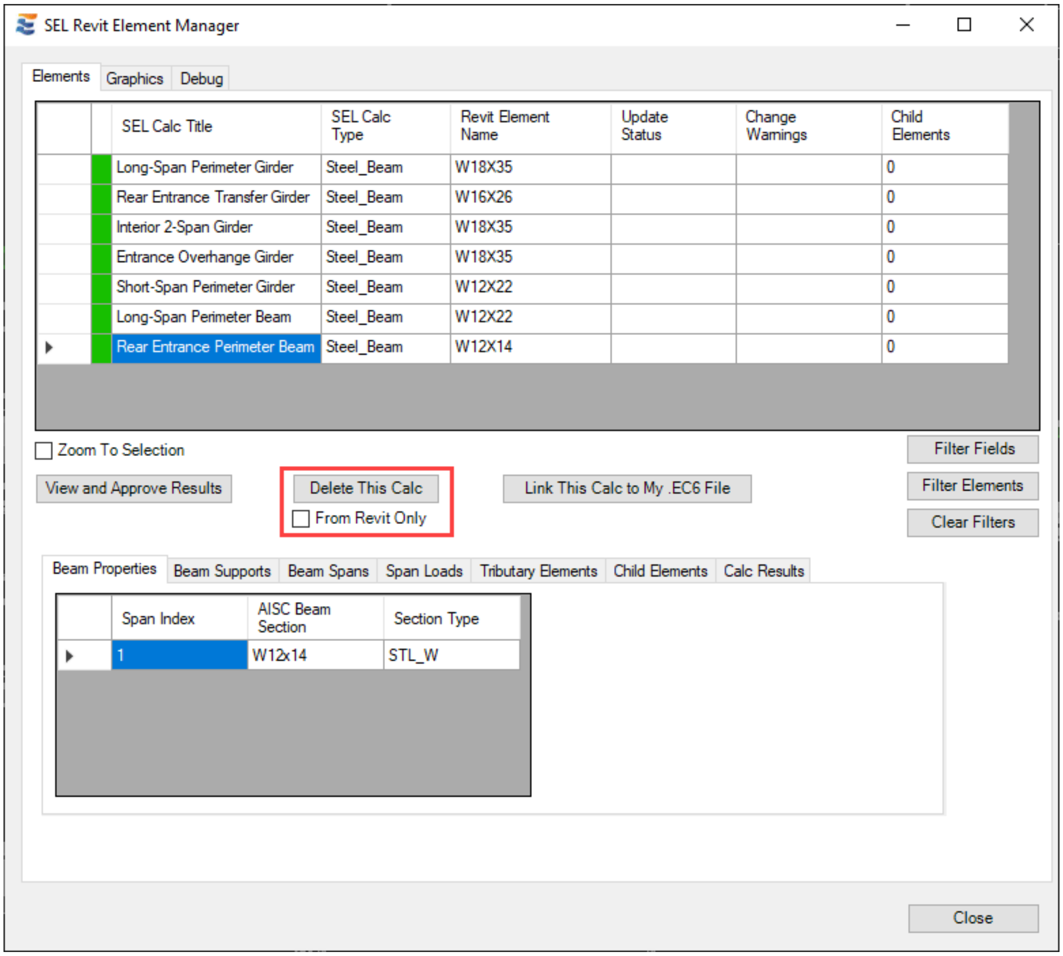


8.2 Removing a Calculation

A calculation may be removed from both the Revit model and the corresponding ENERCALC .EC6 file at any time using the “Delete This Calc” button on the Element Manager window.

NOTE: Deleting of a calculation removes the design data stored for the specified element, but load path information such as support and tributary relationships will be preserved. For example, consider a designed beam bearing on a designed girder. If the girder calculation is removed, the girder still serves as a support for the beam design. The beam is still considered to be tributary to the girder, and the beam

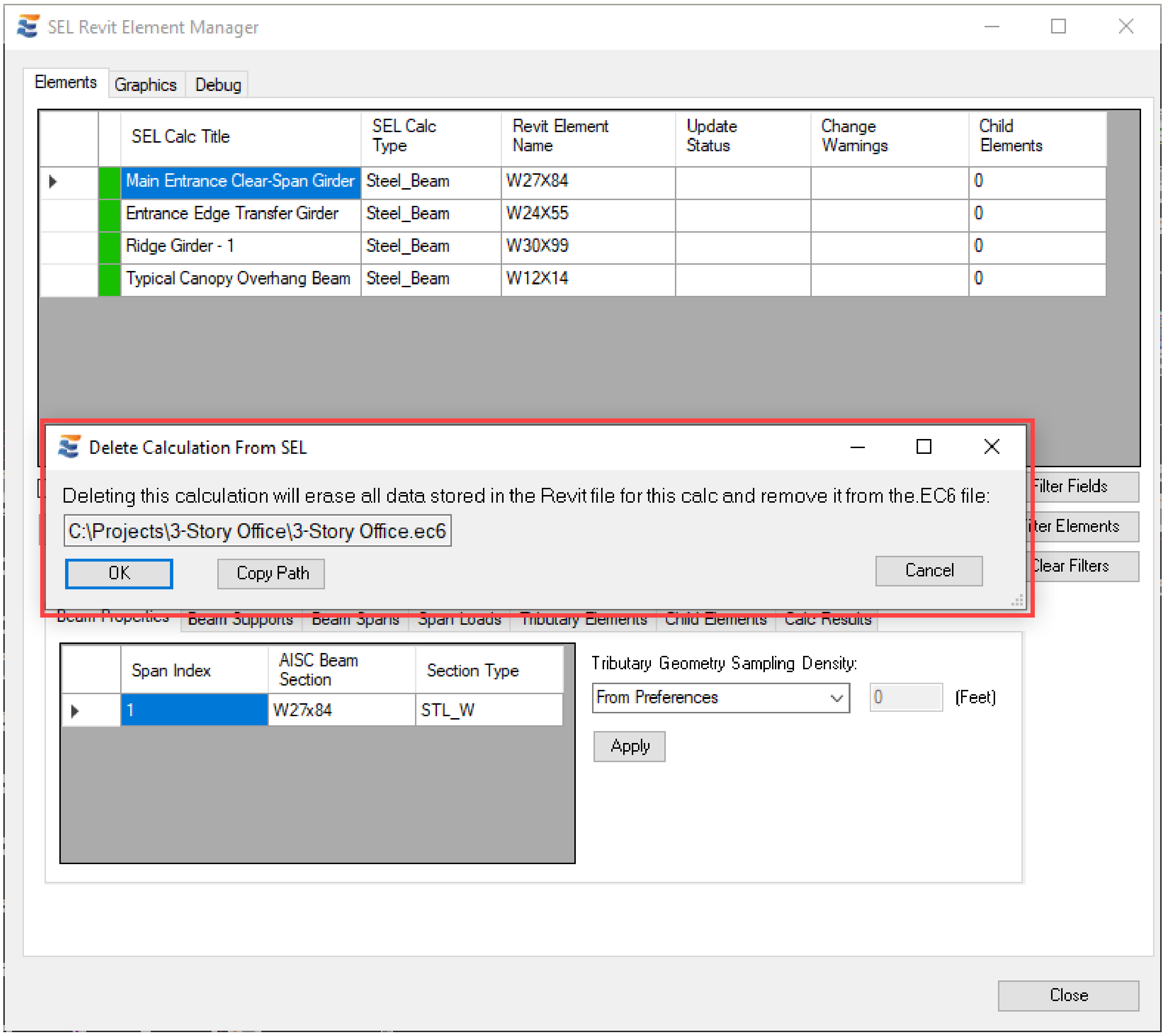
reaction force will be linked in automatically if a new girder calculation is created in the future.



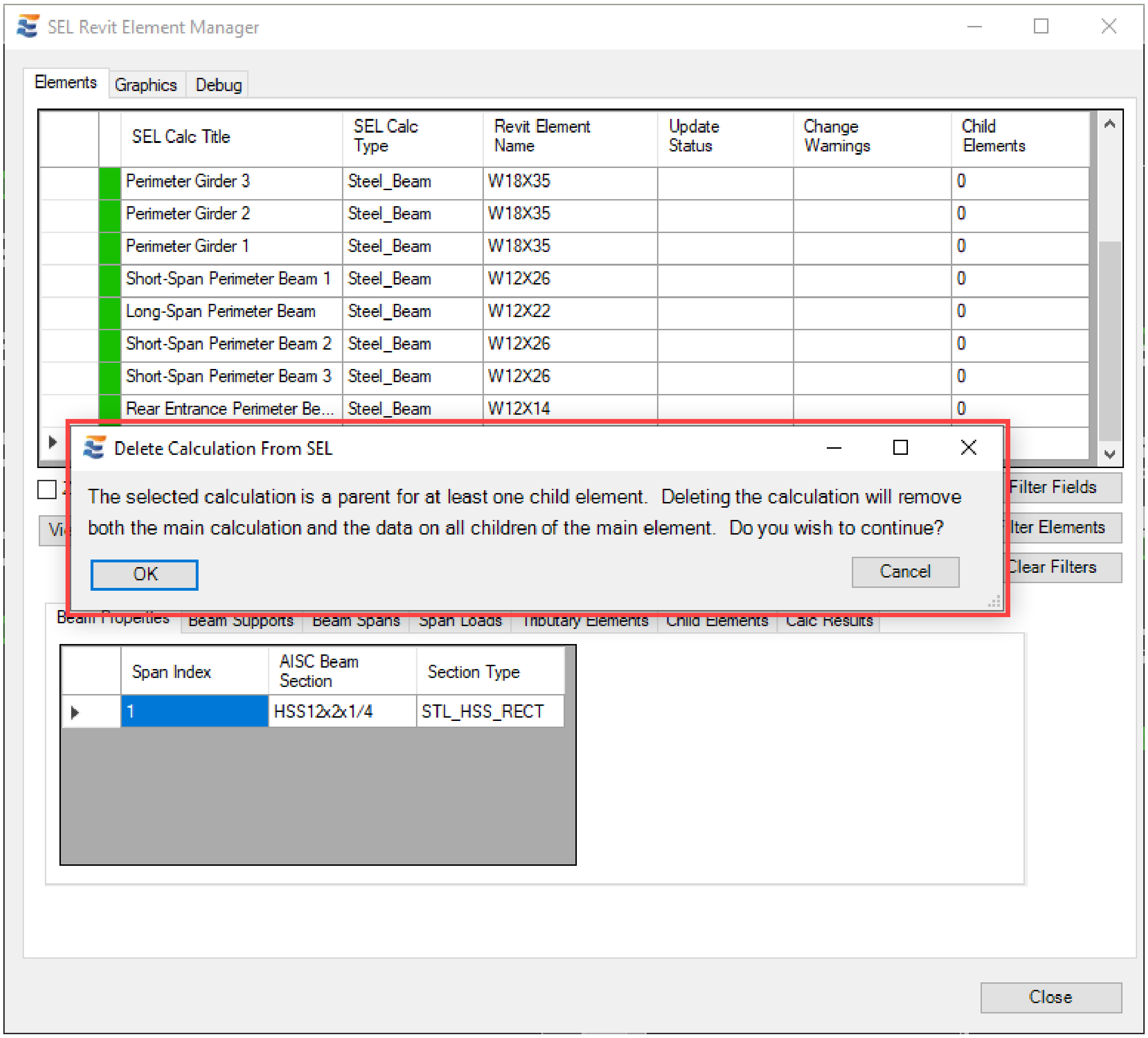
Clicking this button while a calculation is selected will cause ENERCALC SEL to launch so that the calculation data may be removed from the appropriate .EC6 file. When ENERCALC SEL finishes finding and removing the calculation, the calc data will also be erased from the Revit element in the model, and the line item will disappear from the summary table. Any calculation may be removed by any user whose local machine has access to the linked .EC6 file. If the .EC6 file cannot be accessed, a pop-up warning is provided.

Checking the “From Revit Only: checkbox will remove the calculation data stored in Revit without modifying the .EC6 file in any way. This option would be appropriate for removing a calculation whose .EC6 file has been moved, deleted, or otherwise become unreachable.

Prior to removal, the user is prompted with a warning to help prevent accidental removal.

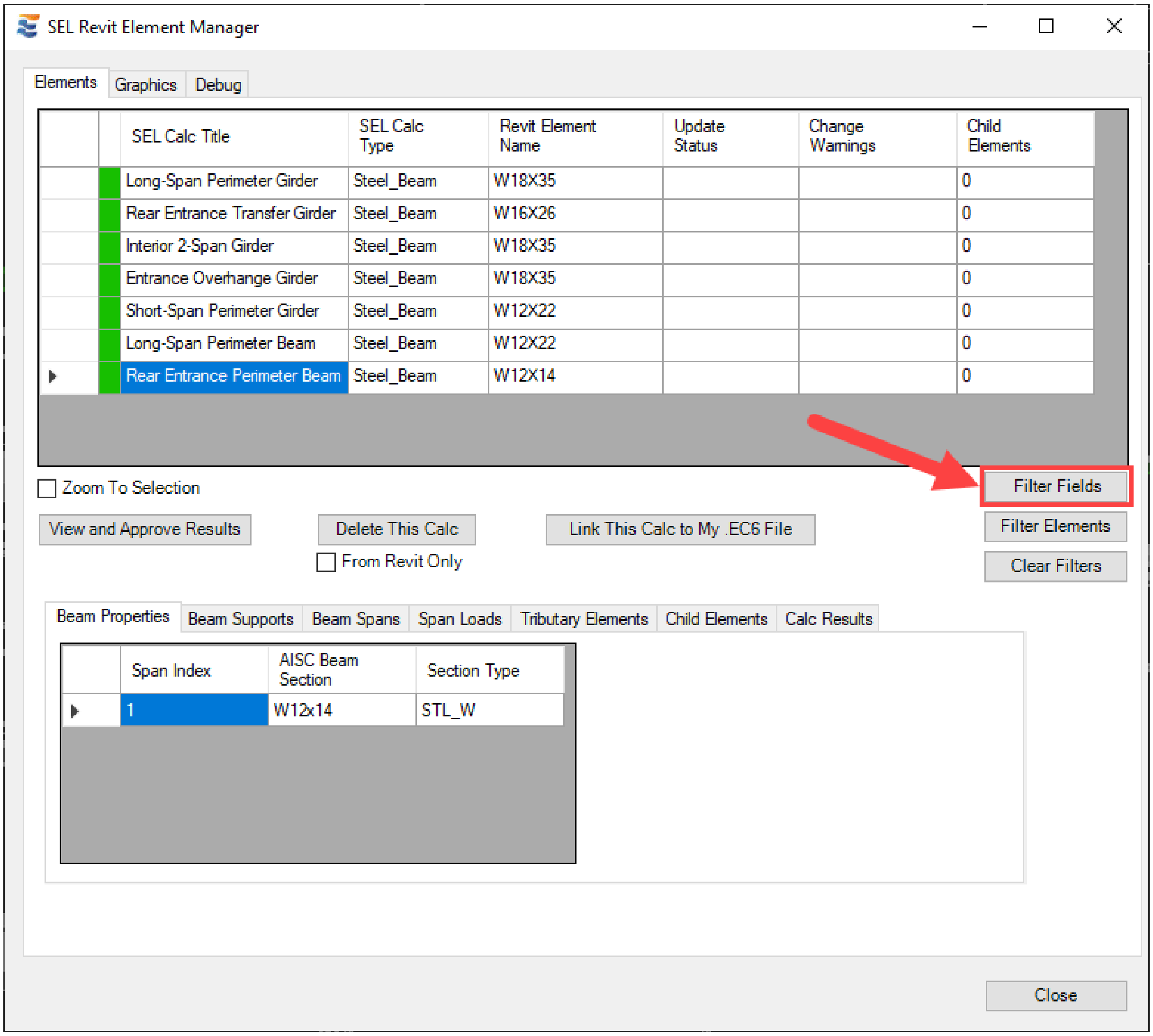


If the calculation to be removed also carries associations to child elements, then the user will be presented with an additional warning. Proceeding with the removal will erase the parent/child relationship that associates the parent element being removed to its children. The children whose parent calculation was removed will not carry any color highlighting and they will no longer be accessible via the Element Manager.

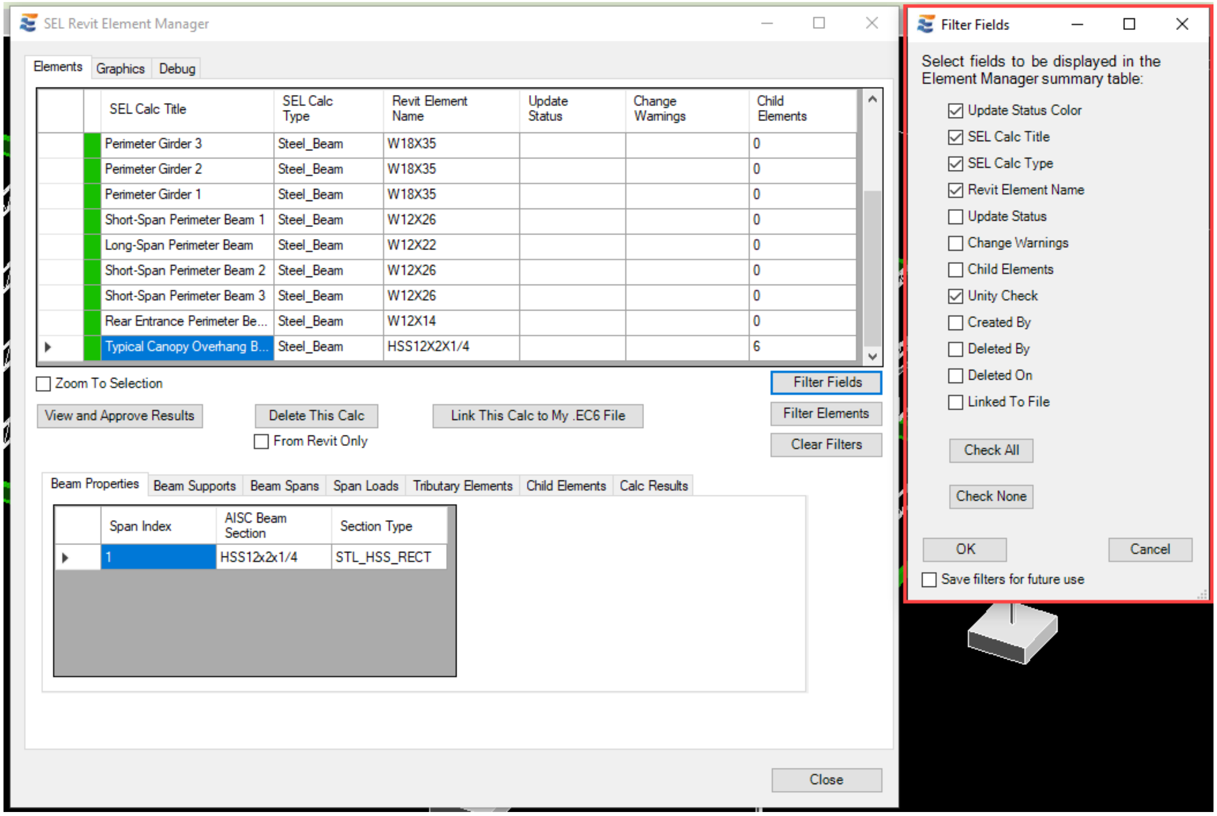


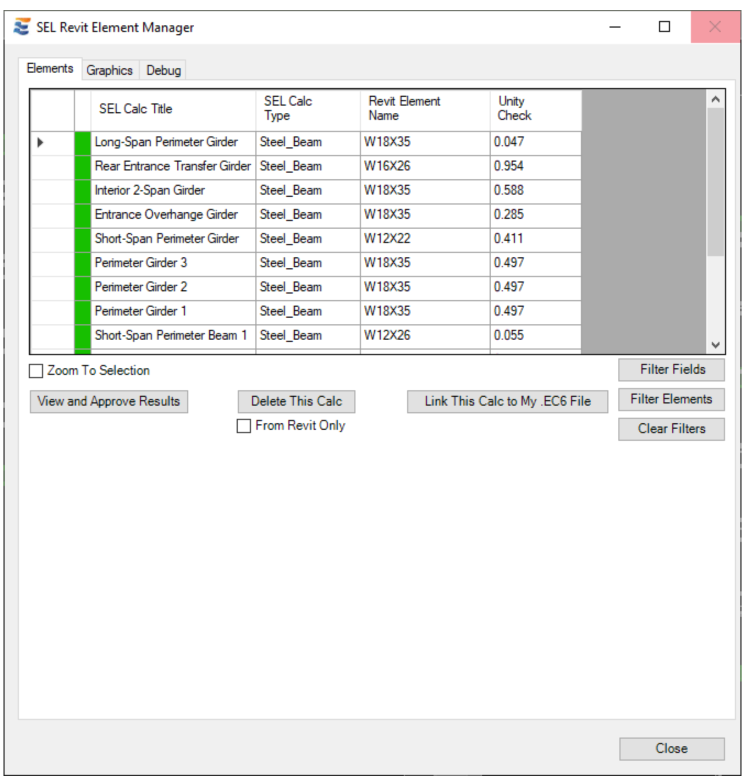
8.3 Filtering Fields and Elements

The Element Manager includes controls for filtering the amount of information presented to the current user in the summary table. Fields (columns) in the table can be easily added or removed using the “Filter Fields” button.

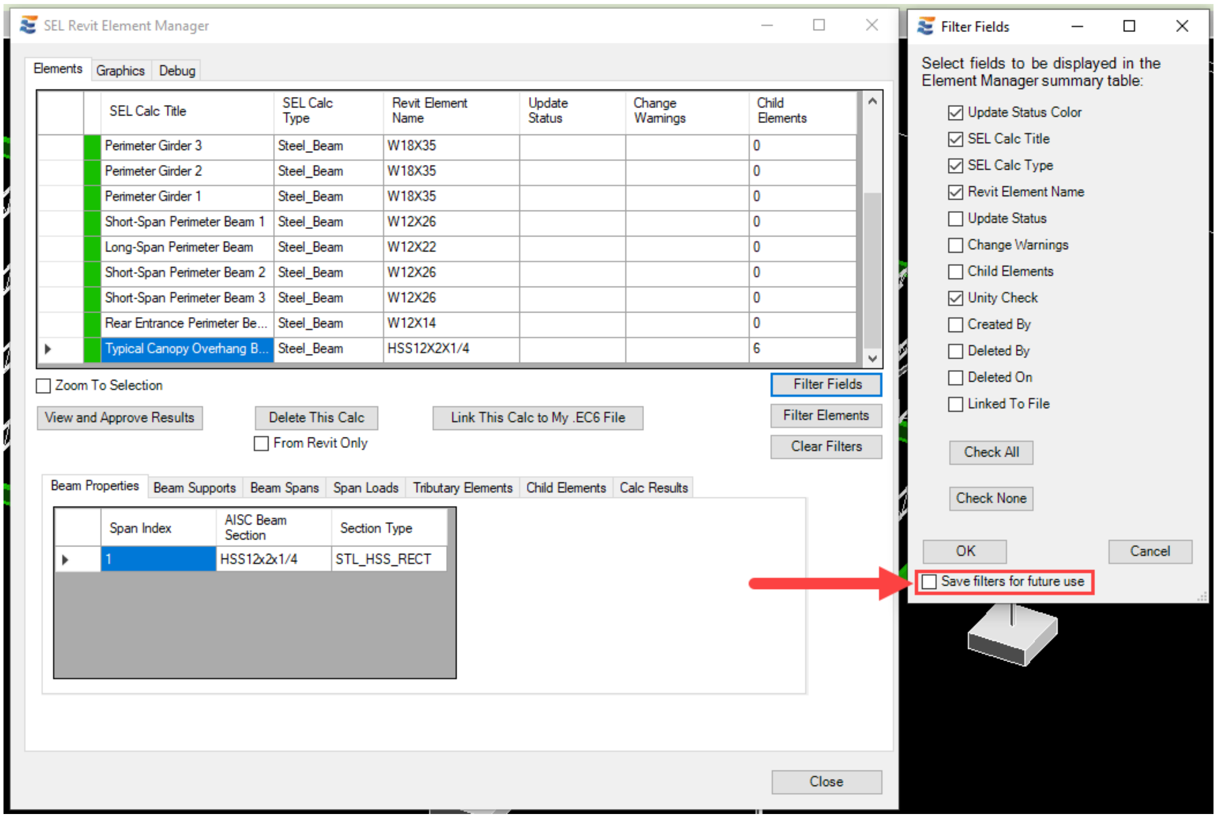


Adding or removing display fields using the checkboxes in the “Filter Fields” form causes the Element Manager window to dynamically resize to fit the contents of the summary table.



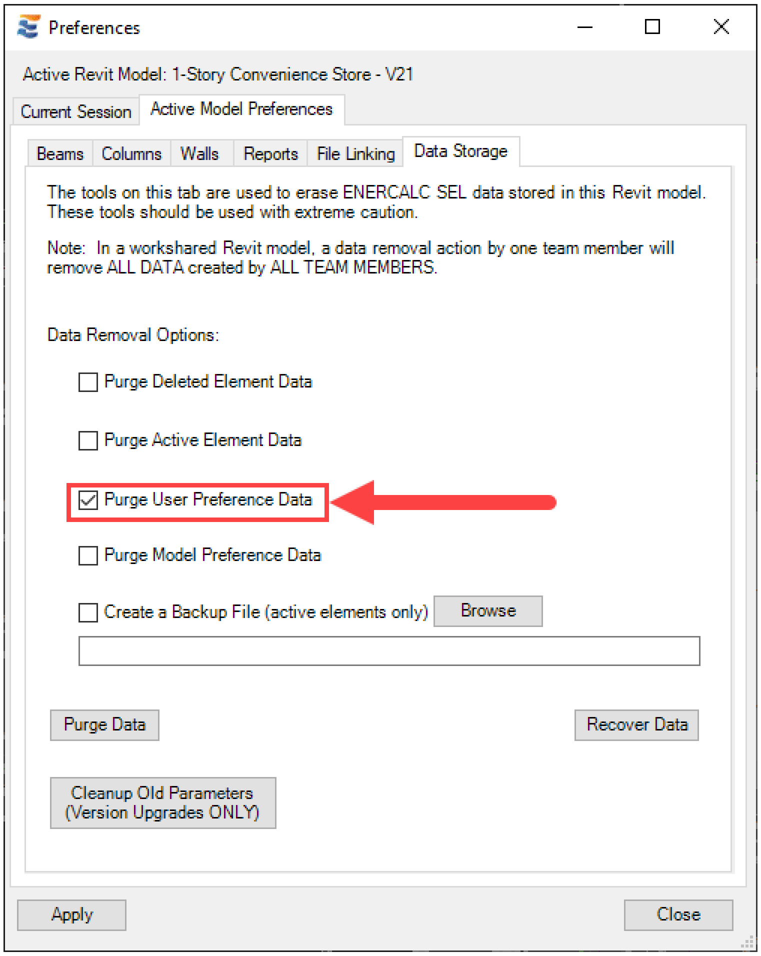


Upon closing the Element Manager, it will revert to default column display settings on the next open unless the user has specified to “Save filters for future use”. Saving of filter settings is specific to individual users and will not affect the appearance of the Element Manager for other team members.

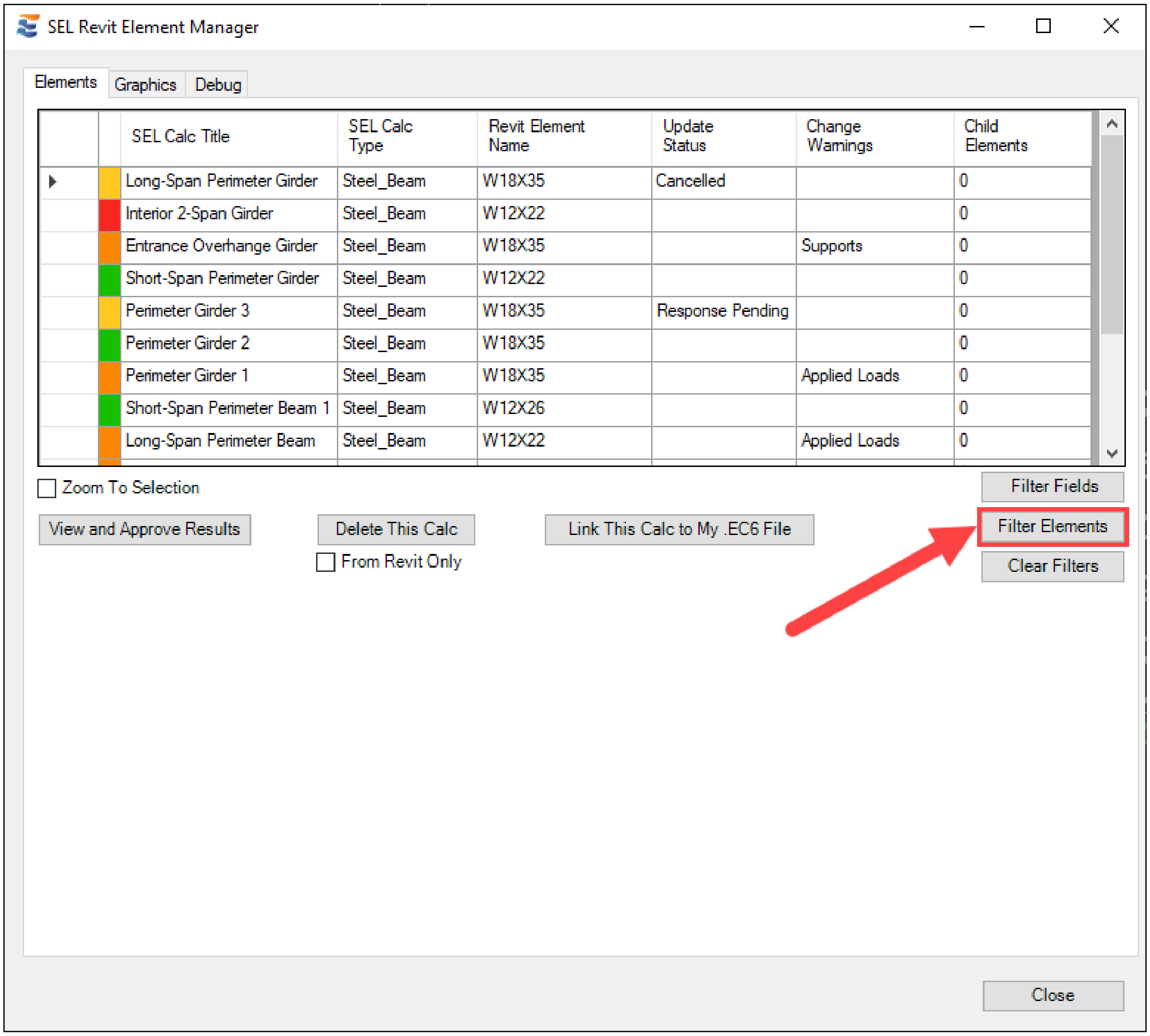


If field filters are saved, the Element Manager will retain this formatting on future opens of the Element Manager both in the current Revit session and in future sessions. Note that these field filter settings are tied to the current user only and to the specific Revit model that is active at the time they are applied. Future changes to the settings will overwrite the existing settings if the “Save” checkbox is activated.

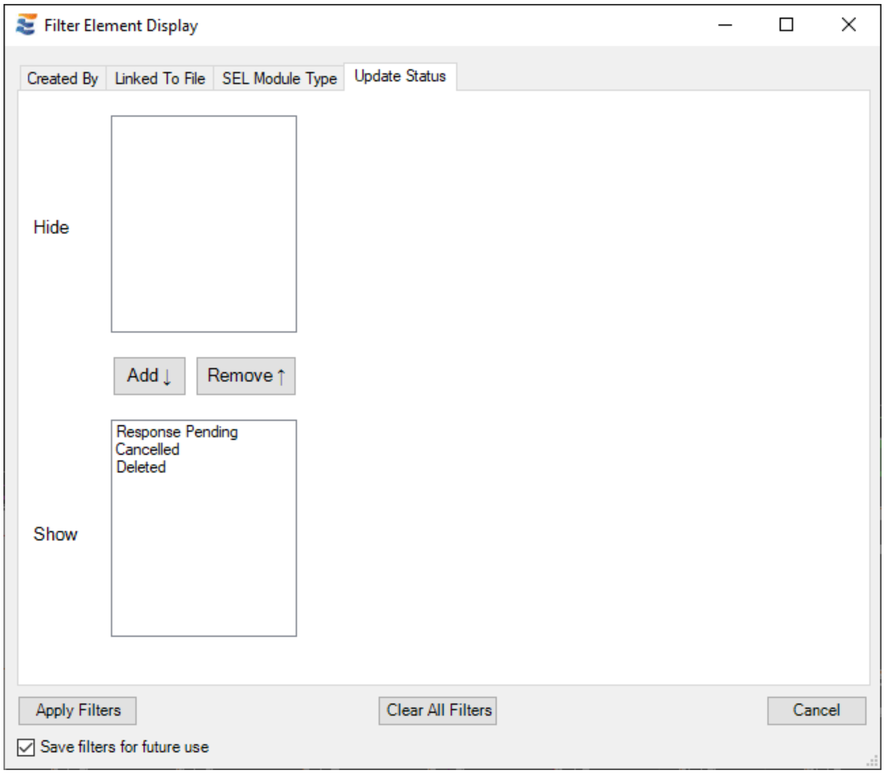
These settings will be removed and the Element Manager will revert to default field display if the “Purge User Preference Data” checkbox is activated during a data purge operation in the Revit model.



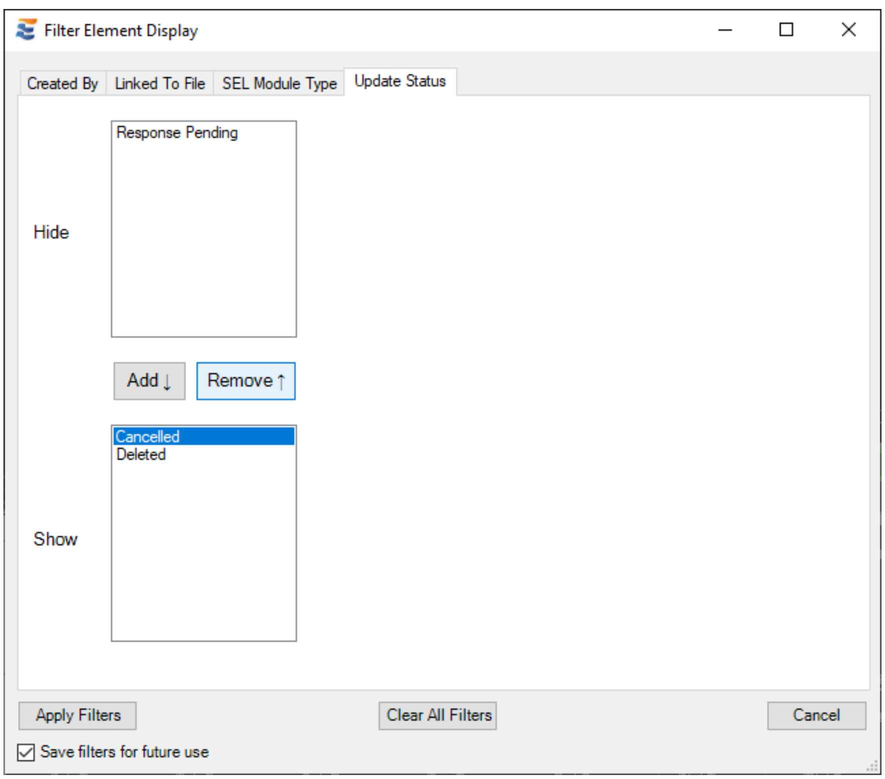
In addition to filtering the visibility of fields (columns) in the summary table, users may also select from a variety of criteria to show or hide individual element entries in the table. These options are accessed using the “Filter Elements” button on the Element Manager window.



The Filter Elements window contains several tabs for different criteria by which elements may be either shown or hidden. Each tab contains a pair of list boxes with controls to “Add” or “Remove” elements with the corresponding properties from display in the main summary table.



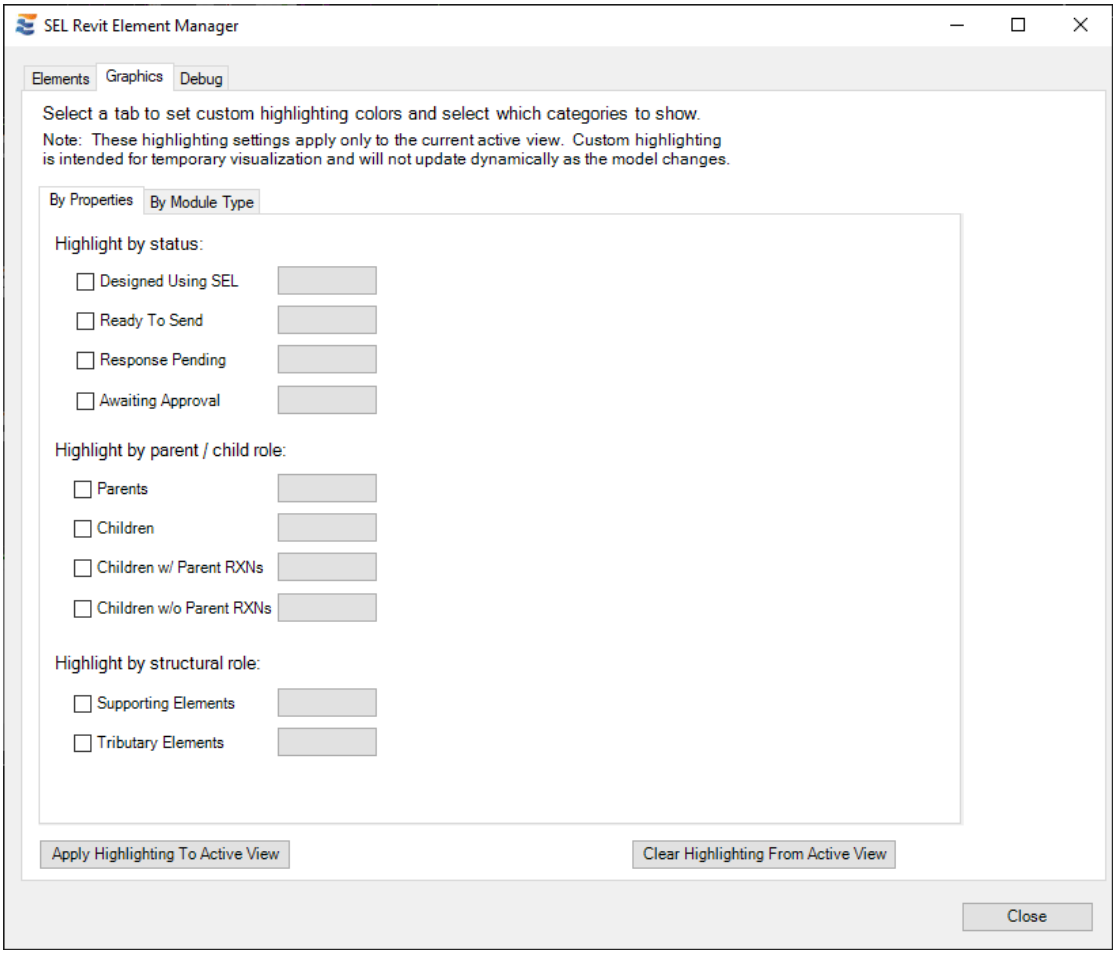
Any elements having properties moved to the "Hide" list will then be hidden from view in the main summary table.



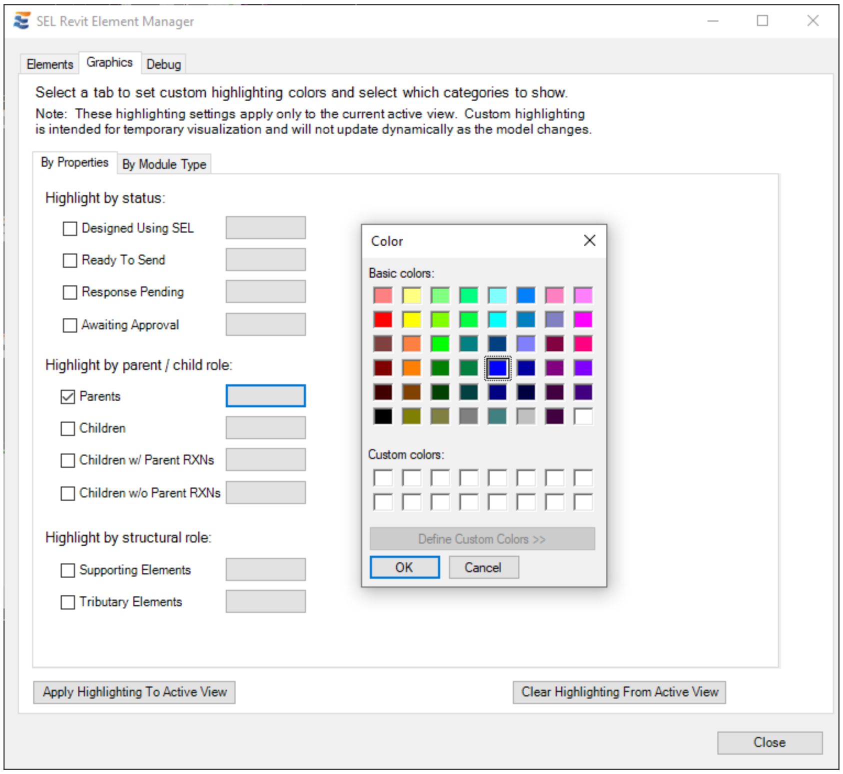
As with the field filters, these settings may be retained in future sessions using the “Save Filters” checkbox.

8.4 Advanced Graphics Controls

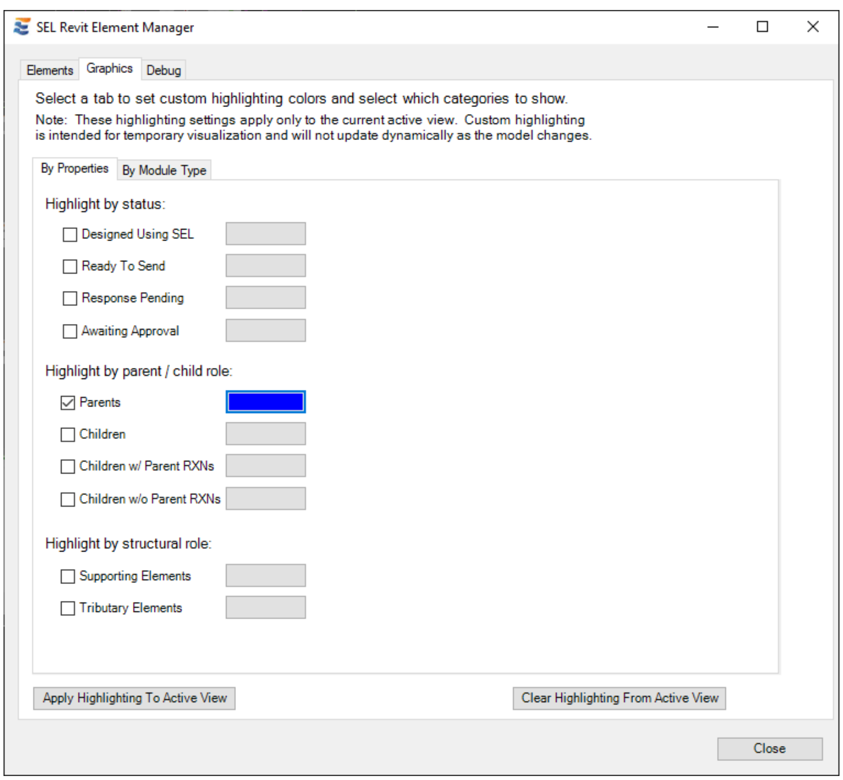
In some cases, users may want to apply additional visualization beyond the default design status highlighting tools found on the ribbon bar. For this purpose, additional highlighting controls are available on the “Graphics” tab of the Element Manager window. The controls are divided into two groups, allowing the user to apply custom highlighting either “By Properties” or “By Module Type”.



The checkboxes are used to control which highlighting is active, while the button next to each property option is used to set the custom color to display in the active view. Clicking the button shows a color palette for the user to choose from.

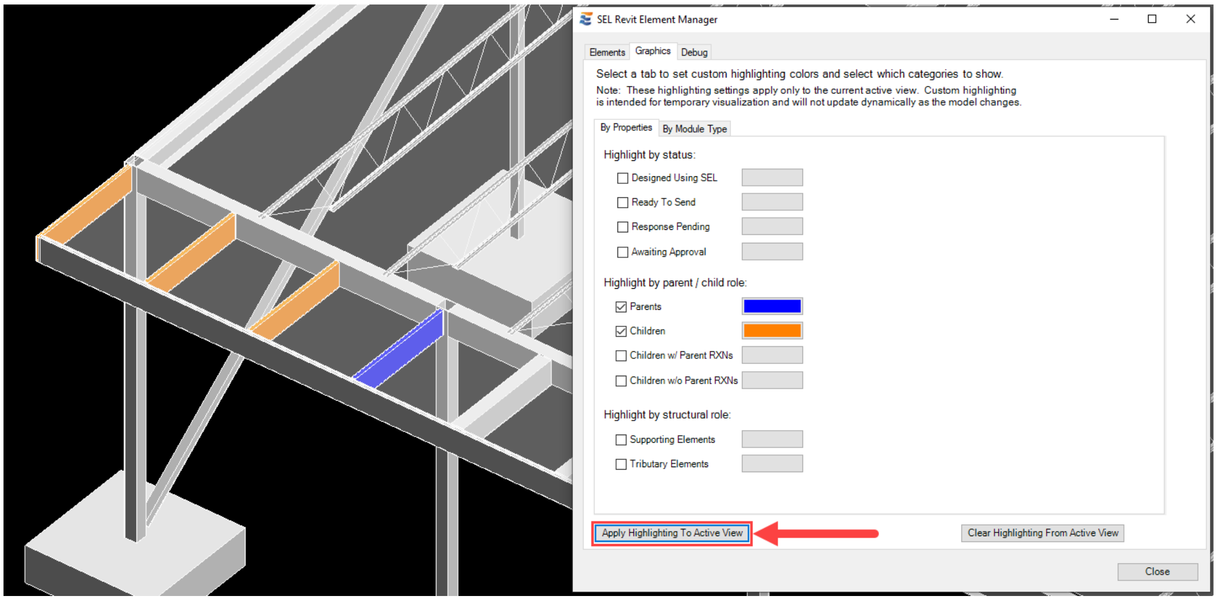


Choosing a color and clicking “OK” will update the button to show the selected color.

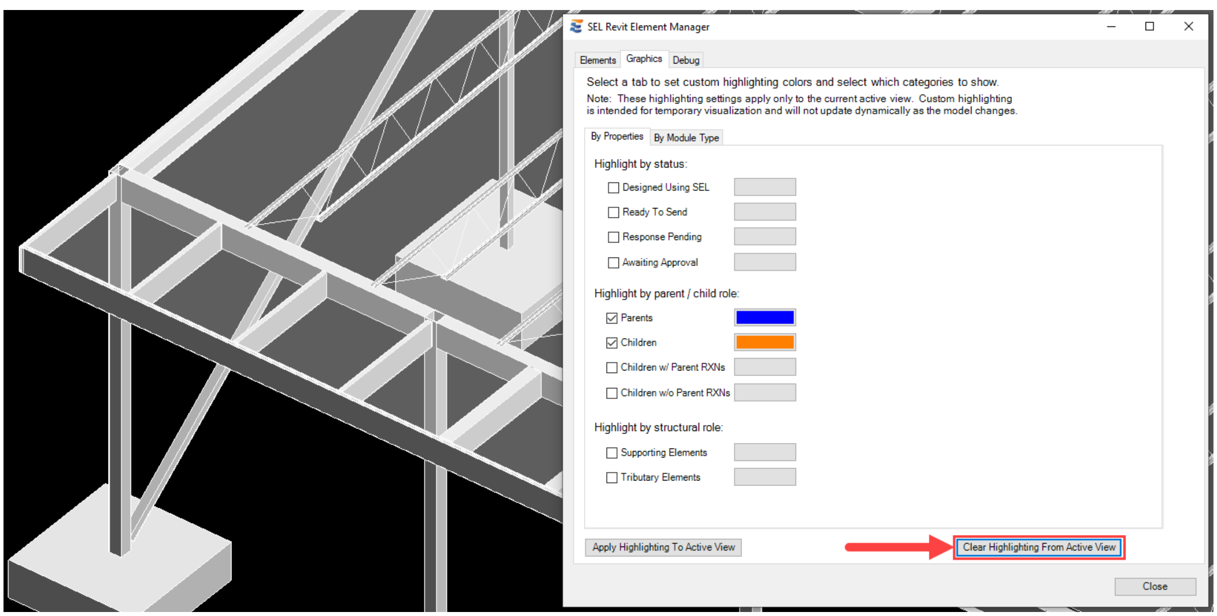


If the Revit model is saved after modifying these custom color selections, the colors will be automatically stored and will be available the next time the user accesses this menu. These custom colors are stored according to user identity and will not impact the colors used by other team members.

After selecting the desired properties and colors, the custom visualization may be applied using the “Apply Highlighting to Active View” button. This highlighting applies only to the current view. It will not be applied to any other view in the Revit model.



The custom highlighting may be removed using the “Clear Highlighting From Active View” button.



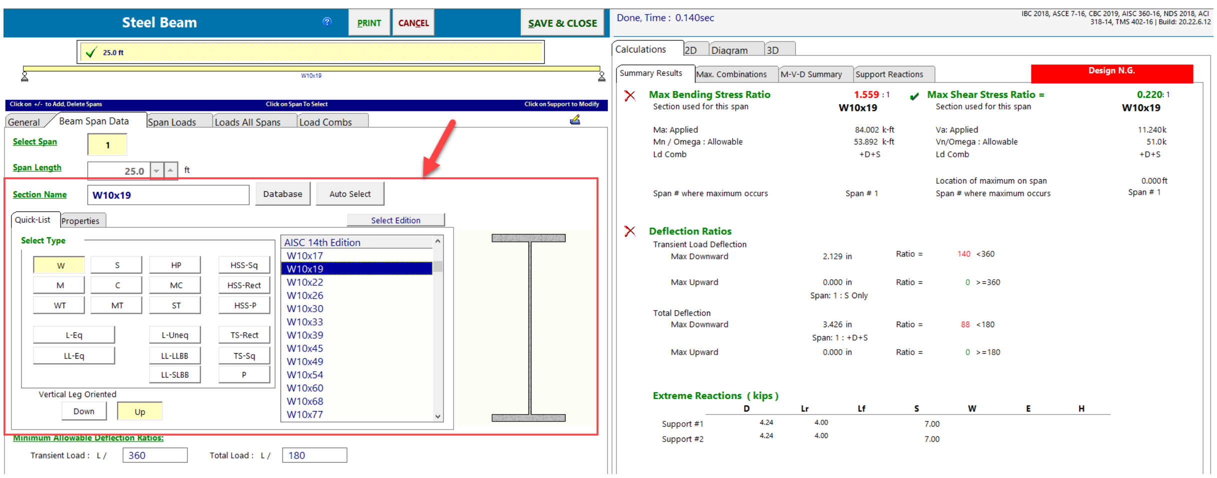
Custom graphics settings are unique to each individual user and will not affect the display settings for other team members within a given project.

Part IX

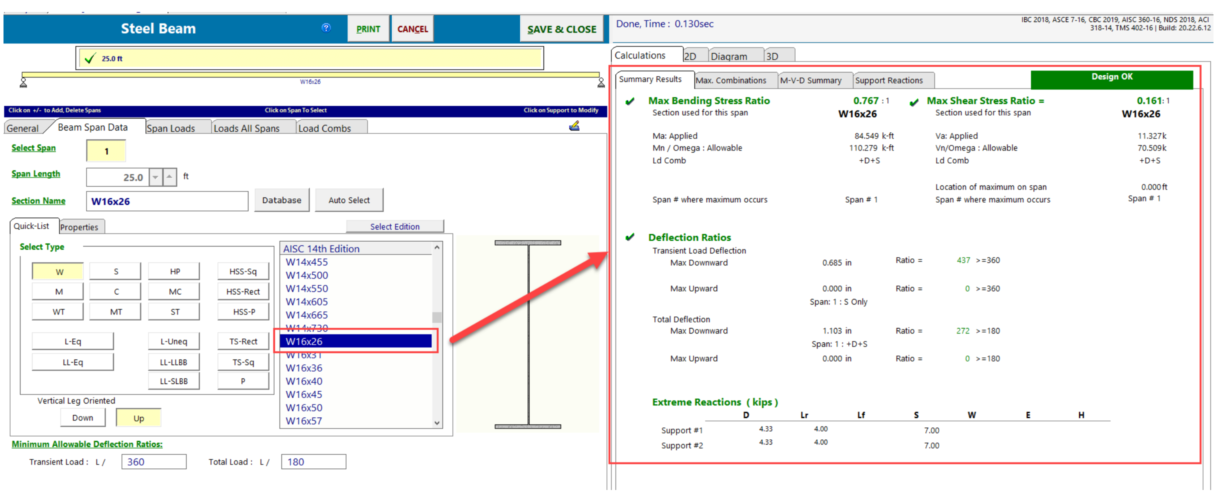
9 Changing Section Sizes

Once a calculation has been sent from Revit and has loaded in the ENERCALC interface, a number of different calculation types permit the user to select a new section size. This includes both beam and column calculations which rely on standardized section types, including AISC steel sections and NDS wood sections. Design types that use open-ended arbitrary geometry (such as concrete beams or columns) share some similarities but will not behave identically. Refer to the appropriate sections for specific calculation types for more detail.

Once a calculation has loaded in the ENERCALC interface, the section size is available for modification by the user using the same controls typically seen in a conventional manually built calculation.



As with a conventional manually-built ENERCALC calculation, the calculations and code checks for the design will update in real time when the section size is changed:



When the user completes the design and clicks "Save and Close", any changes to the section size will be automatically applied to the corresponding physical element in the Revit model. As long as the appropriate section already exists in the Revit project OR EFR can successfully locate and load the appropriate family (".rfa") file, then the element will seamlessly and instantaneously toggle to the new section size. If this process occurs without any errors or warnings, then the section size update will not be accompanied by any pop-ups or notifications.

If any issues occur while obtaining the loadable content or updating the section, the user will be notified accordingly. Typical issues associated with this process are detailed in Size Change Errors91.

9.1 Section Change Warnings

In order to apply section size changes to Revit elements, ENERCALC for Revit locates the appropriate section from the appropriate family using a multi-step logical search process. As indicated in Changing Section Sizes90, this process is designed to be a seamless automatic process. The EFR process for locating, loading, and applying section sizes contains a highly granular framework of warning messages. These warning messages provide explicit and actionable information for engineering teams and BIM staff to troubleshoot any failure. The complexity of search performed to obtain content and change sizes does not impact user experience during daily use of the software, but is examined here for informational purposes in the event that troubleshooting is required.

NOTE: In order to understand how ENERCALC for Revit obtains and utilizes Revit content for section changes, it is advisable for users to have a functional grasp of the following concepts:

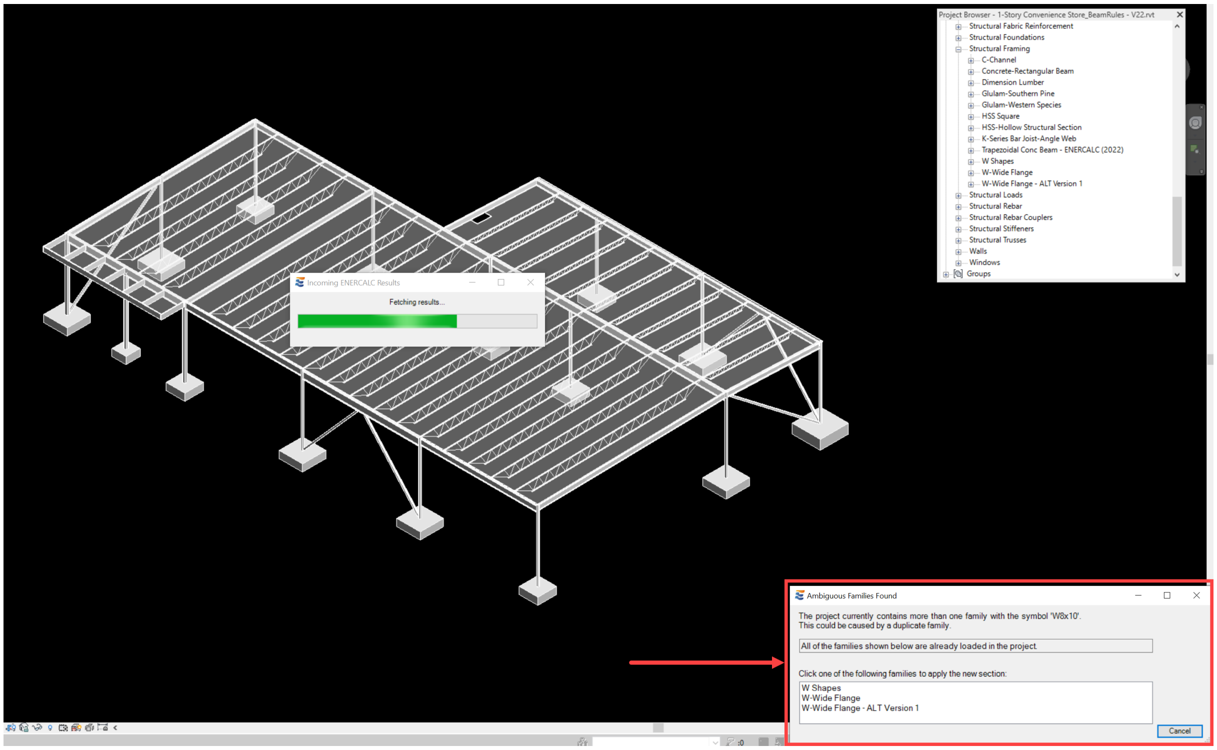
- What is a Revit family?
- How are Revit families created and used?
- What is a family type catalog?
- How do my company and/or my team store and access Revit loadable content?

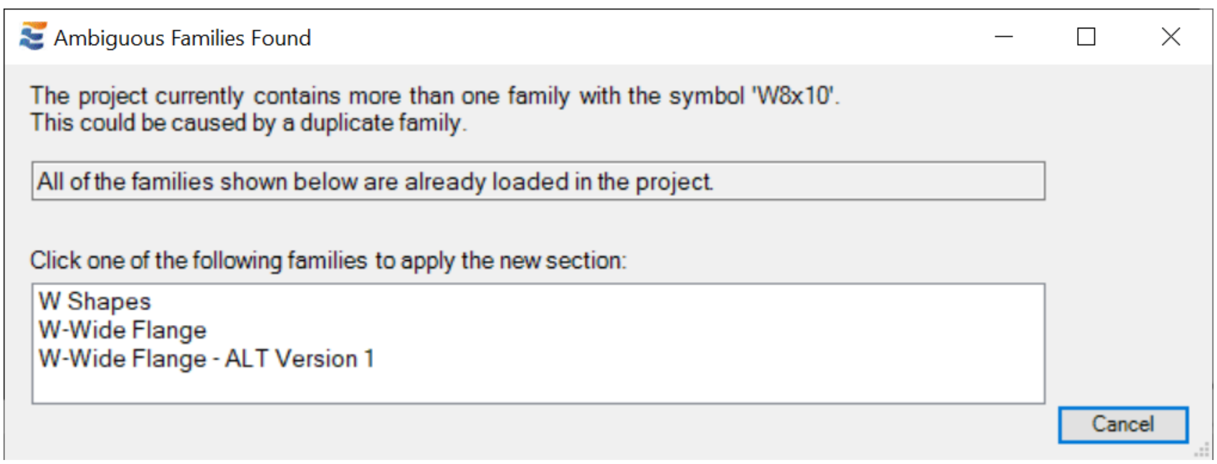
These topics are not addressed in detail here in this document. For further reading on this topic, refer to the Autodesk Knowledge Base: About Families | Revit 2022 | Autodesk Knowledge Network

NOTE: Standardized sections, including both AISC steel section and NDS wood sections are located and matched using the literal name of the section. Sections are NOT matched up by comparing actual section geometry properties. Section names found in your team's family content must exactly match the corresponding material design code naming for successful use of the content. For example, Wide Flange beams would be retrieved using the standard AISC format (i.e., "W8x10"), and will not succeed if an alternate naming is in use (i.e., "W_BM8x10_plf").

Before checking available loadable family content, EFR starts by first inspecting sections in families that already exist in the Revit model. Section matches from existing families are restricted to only the appropriate structural material, meaning that families of other materials will not be inspected. For example, if the section size change was sent by a Steel Beam calculation, Revit families of structural material "Wood" or "Concrete" will not be inspected for a section size match. Even if an ambiguous section name exists, it will not be considered if the owning family does not have the intended material. For example, a wood beam family containing a type entitled "W8x10" will not cause ambiguity when toggling steel sizes, because the family is not of the specified material. If a single section found with both the appropriate material AND the specified section name, the section will be immediately applied to the Revit element.

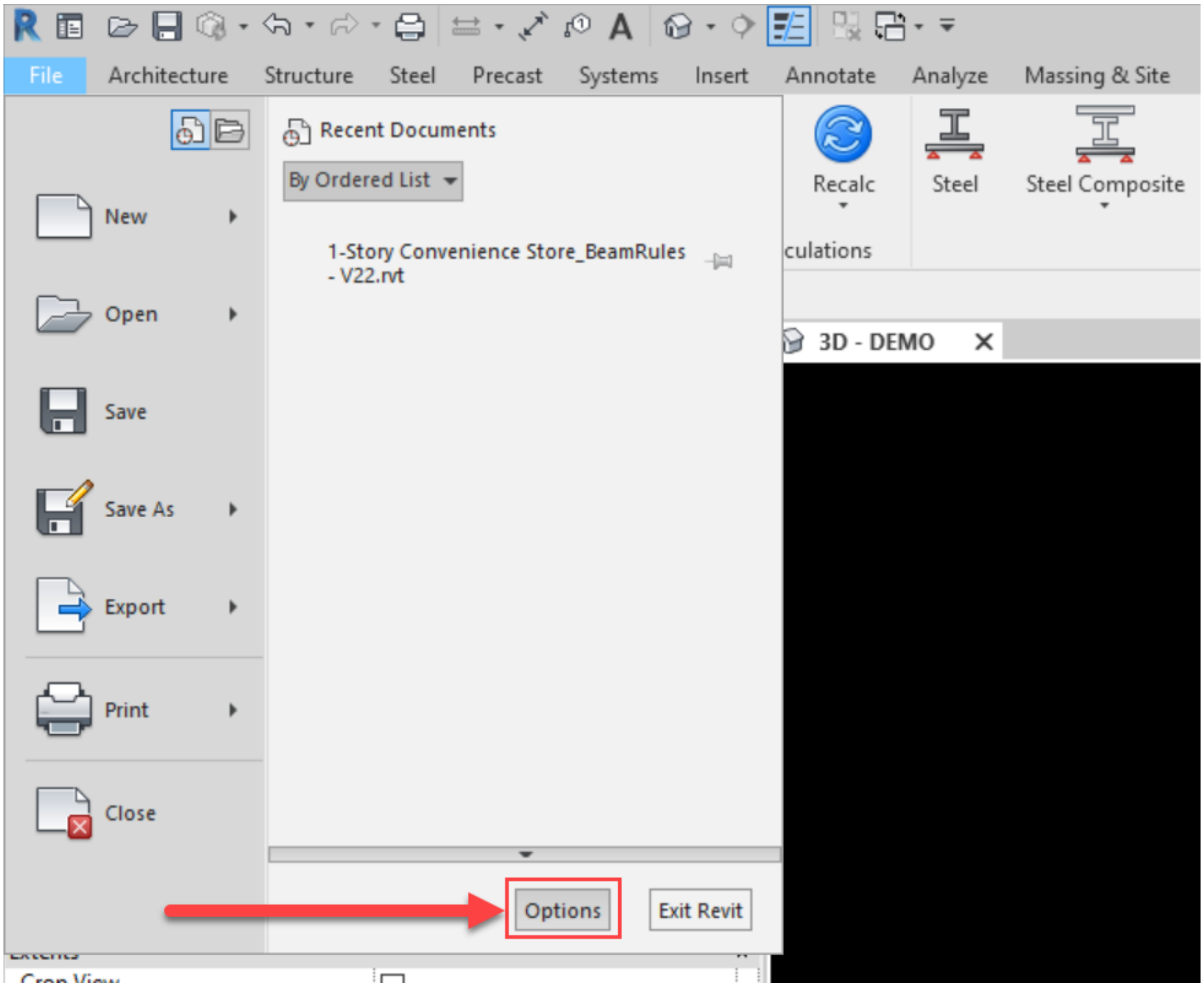
If multiple existing families in the project are found to contain a section of the specified name, then the user will be prompted to manually pick which family should be used:

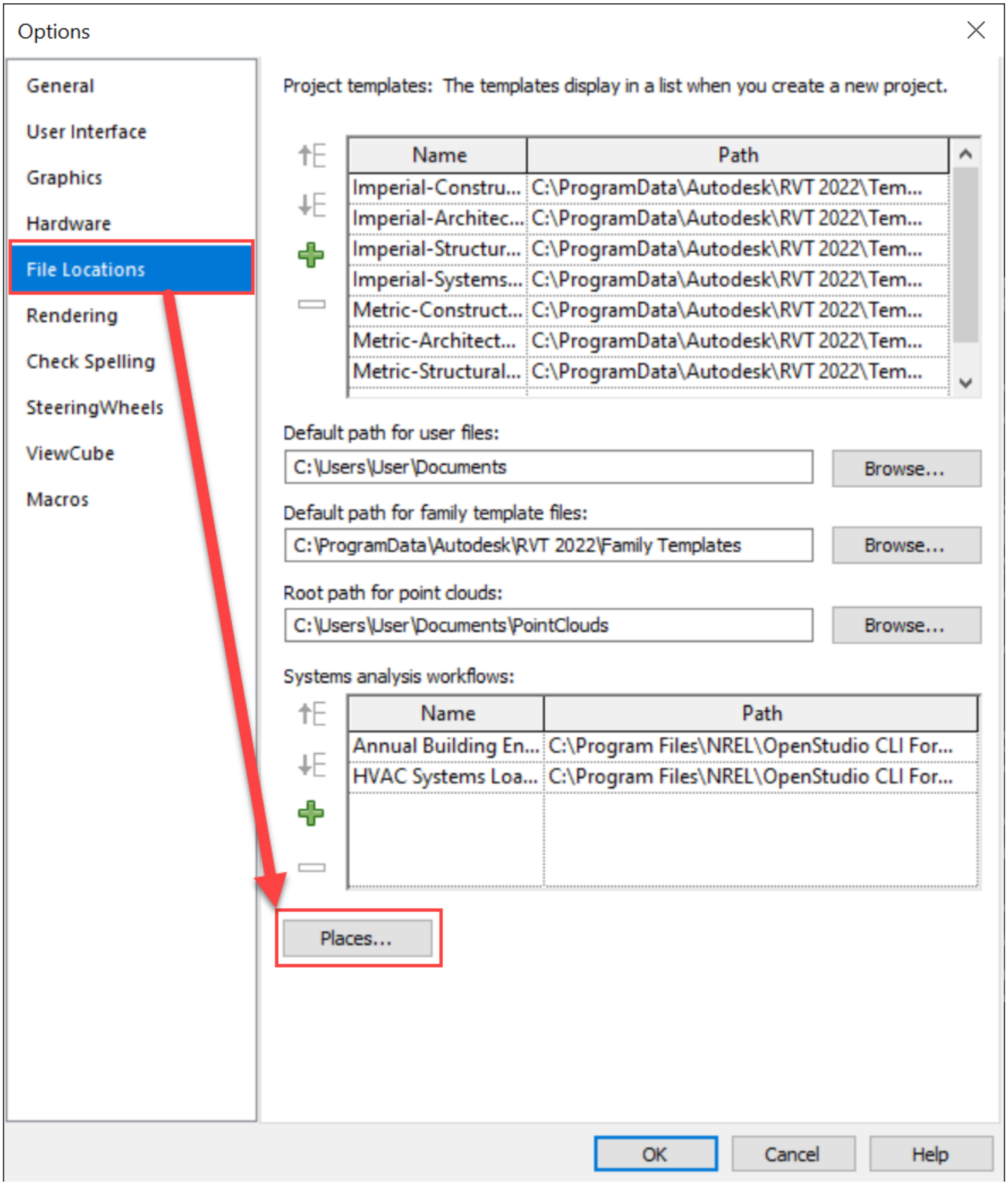


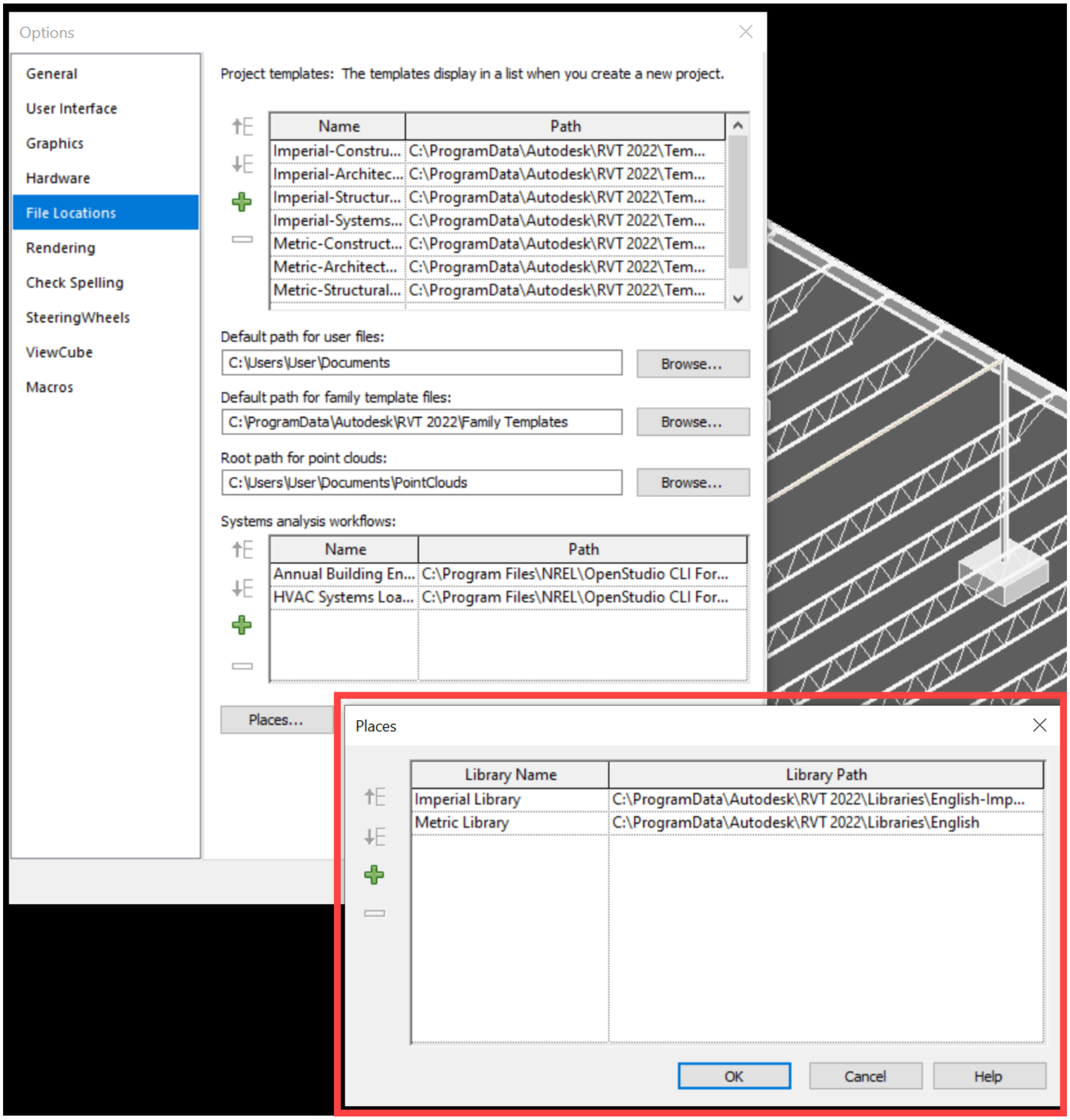


Clicking one of the options shown in the list will allow the section update process to proceed. The Revit element will then be updated to the specified section size using the family indicated by the user.

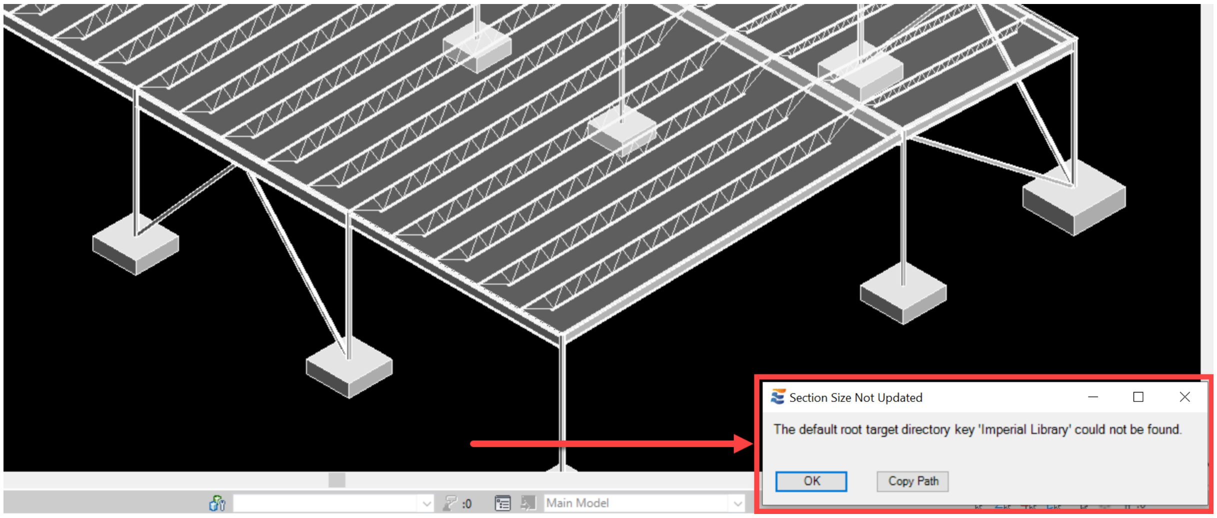
If all families currently loaded in the Revit project are inspected and none of them contain a match for the section name sent from the ENERCALC calculation, ENERCALC for Revit will automatically search for an appropriate family file (.rfa).
Family files are located by inspecting the default library directory specified in the native Revit interface at the following location: File > Options > File Locations > Places > Imperial Library.


File
Architecture
Structure
Steel
Precast
Systems
Insert
Annotate
Analyze
Massing & Site
Recent Documents
By Ordered List
1-Story Convenience Store_BeamRules - V22.rvt
New
Open
Save
Save As
Export
Print
Close
Recalc
Steel
Steel Composite
culations
3D - DEMO
Options
Exit Revit

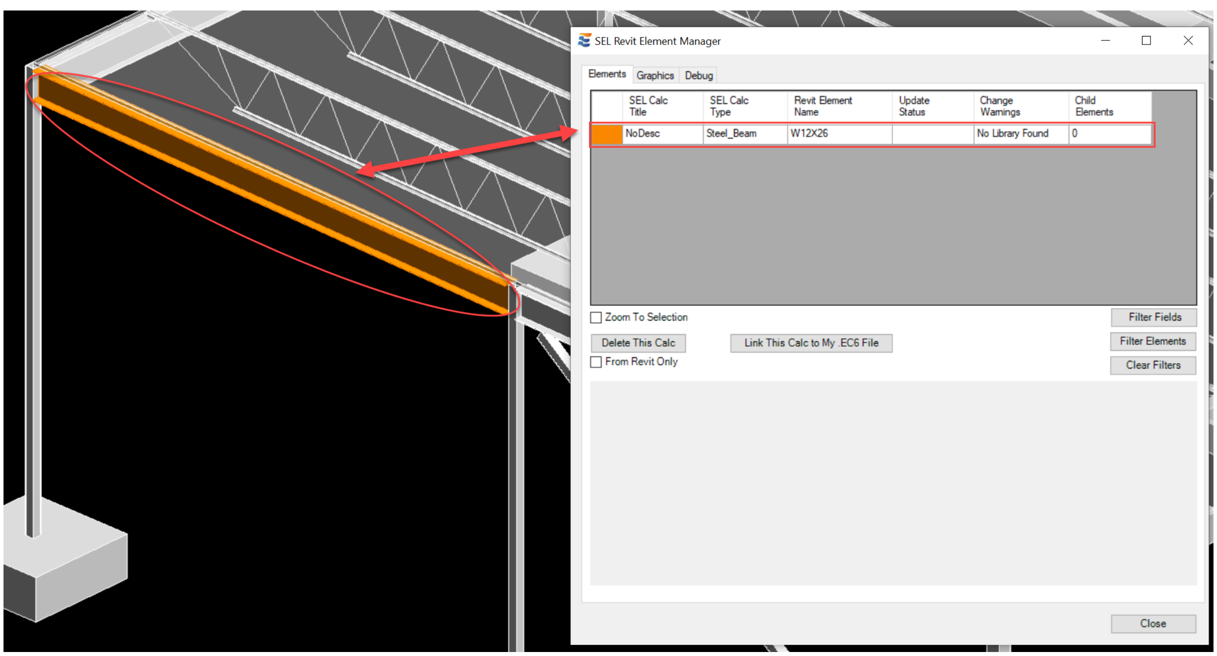

Options
General
User Interface
Graphics
Hardware
File Locations
Rendering
Check Spelling
SteeringWheels
ViewCube
Macros
Project templates: The templates display in a list when you create a new project.
Name
Path
Imperial-Constru... C:\ProgramData\Autodesk\RVT 2022\Tem...
Imperial-Architec... C:\ProgramData\Autodesk\RVT 2022\Tem...
Imperial-Structur... C:\ProgramData\Autodesk\RVT 2022\Tem...
Imperial-Systems... C:\ProgramData\Autodesk\RVT 2022\Tem...
Metric-Construct... C:\ProgramData\Autodesk\RVT 2022\Tem...
Metric-Architect... C:\ProgramData\Autodesk\RVT 2022\Tem...
Metric-Structural... C:\ProgramData\Autodesk\RVT 2022\Tem...
Default path for user files:
C:\Users\User\Documents
Browse...
Default path for family template files:
C:\ProgramData\Autodesk\RVT 2022\Family Templates
Browse...
Root path for point clouds:
C:\Users\User\Documents\PointClouds
Browse...
Systems analysis workflows:
Name
Path
Annual Building En... C:\Program Files\NREL\OpenStudio CLI For...
HVAC Systems Loa... C:\Program Files\NREL\OpenStudio CLI For...
Places...
OK
Cancel
Help



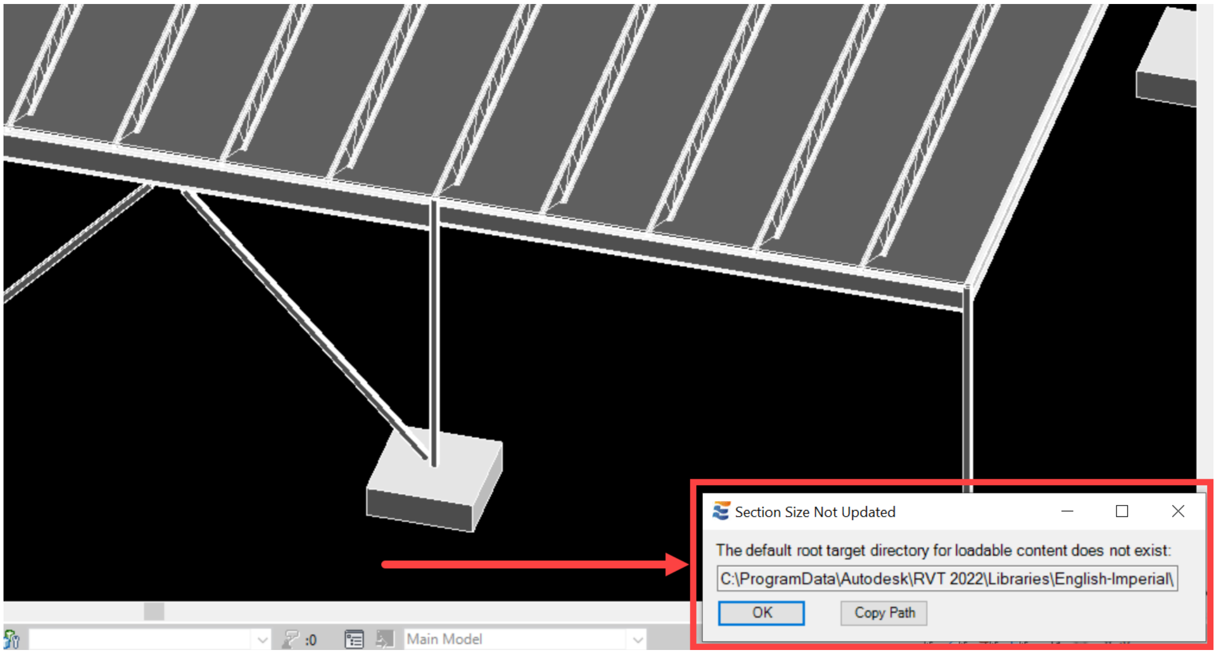
The folder path specified under "Imperial Library" will serve as the default directory referenced by ENERCALC for Revit when searching for loadable content. For successful automatic loading of families, users should ensure that this path points to the location the company or team has designated for storing .rfa files. It is not advisable to change the default Library Name field to any other value besides "Imperial Library". Doing so will result in a failure to find and load families, indicated by the following error message in Revit following a "Save and Close" operation:



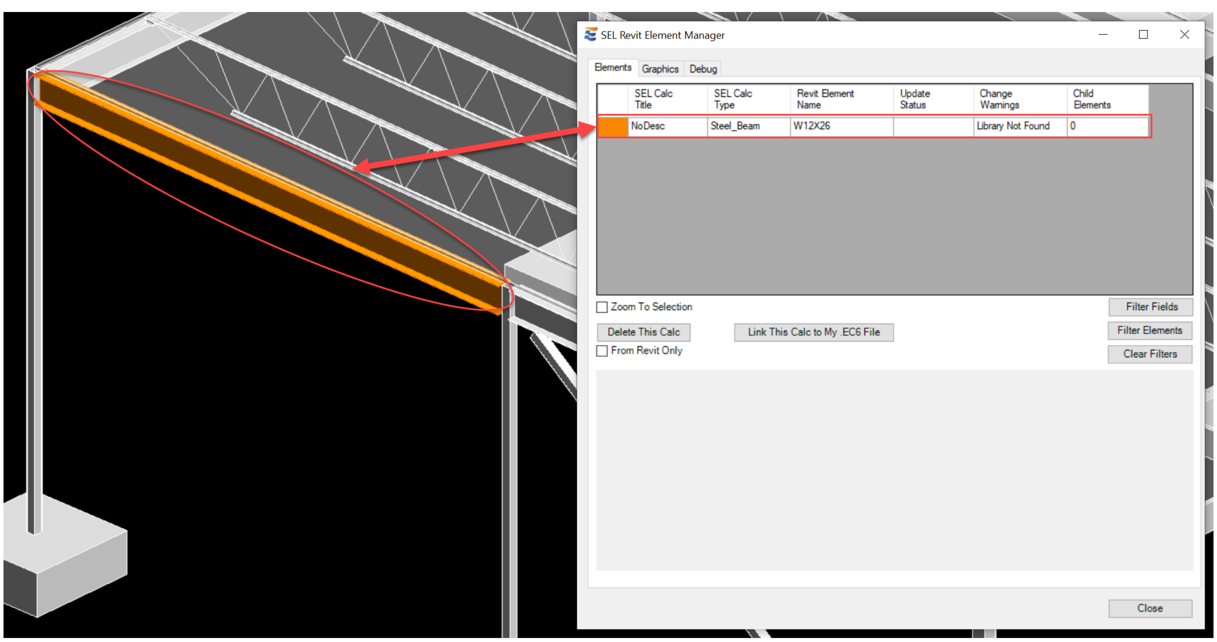
After the user dismisses this notification, the physical Revit element whose section size could not be changed will be placed in a warning state to remind the user that the ENERCALC calculation and Revit physical element are not in complete agreement. This warning state is visible via tabular summary in the Element Manager AND via color-coded highlighting in the Revit view:



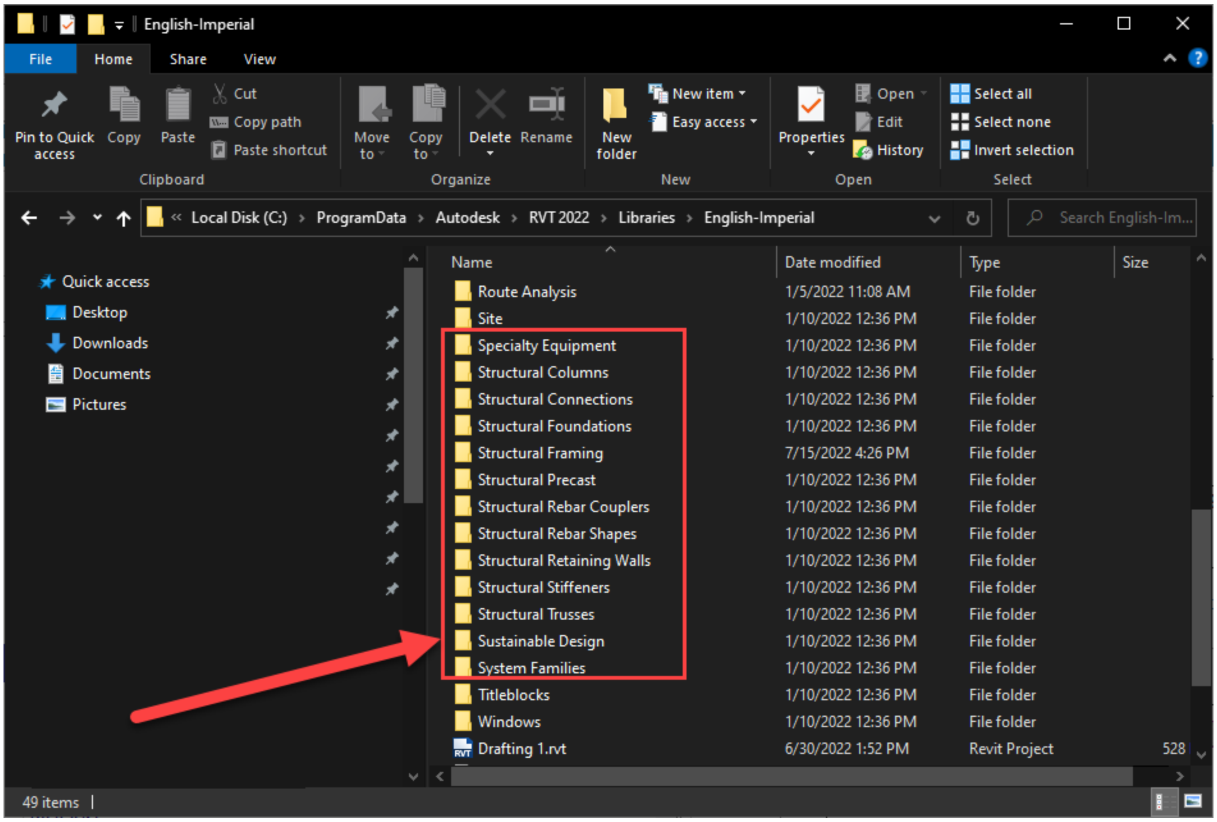
Although the native Revit dialog shown above does automatically verify that the user-provided default path is valid, it is possible that a folder may move or become unreachable AFTER the path has been set. In such cases, the following error message will be presented when EFR attempts to locate the library:



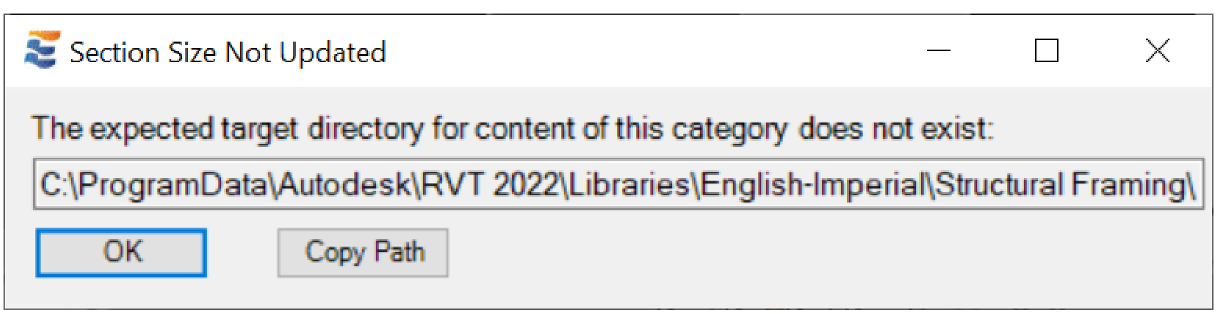
The warning message includes a courtesy option to copy the path shown so that the user may manually navigate folders via File Explorer and verify what has happened to the target directory. After the user dismisses this notification, the physical Revit element whose section size could not be changed will be placed in a warning state to remind the user that the ENERCALC calculation and Revit physical element are not in complete agreement. This warning state is visible via tabular summary in the Element Manager AND via color-coded highlighting in the Revit view:



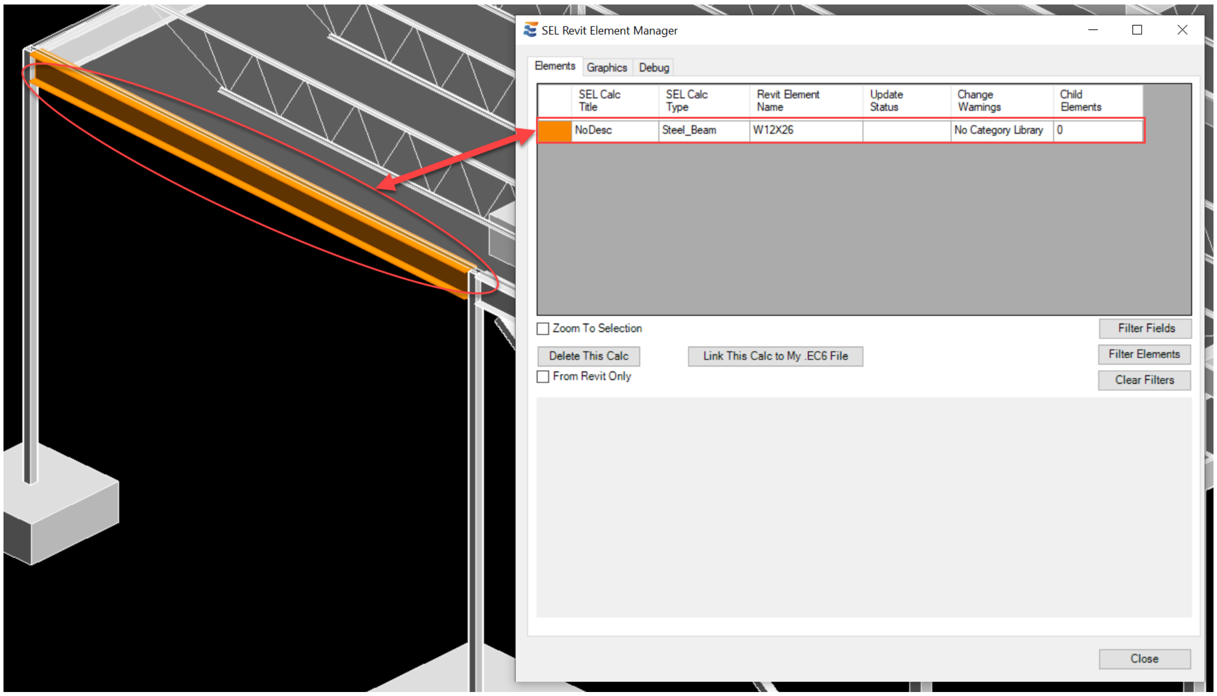
If the default library has been specified properly and the path is valid (i.e., the directory exists), then EFR will automatically inspect the target directory to obtain content of the proper category. This inspection is based on the file structure used by Autodesk's default out-of-box loadable content:



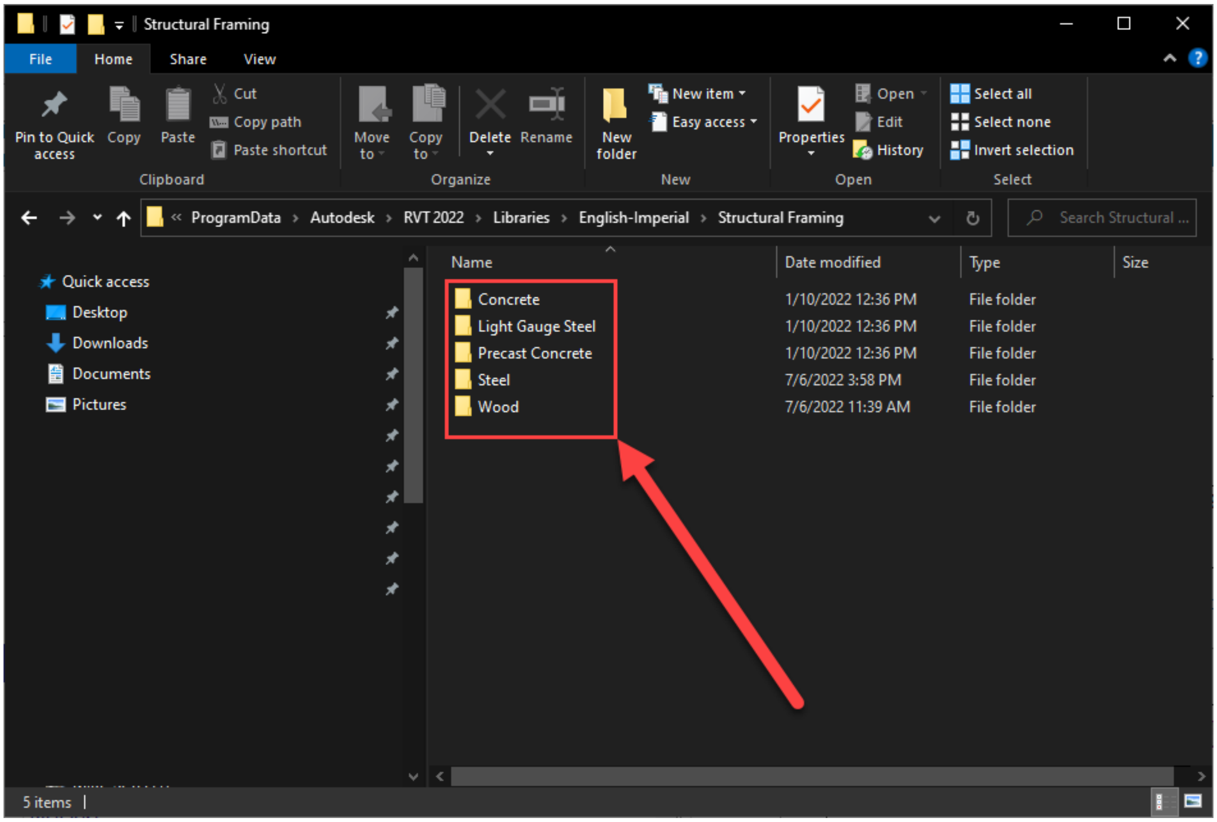
Based on this default file structure, EFR will automatically search for the sub-folder "\Structural Framing" for beam calculations, "\Structural Columns" for column calculations, etc. If the appropriate category sub-folder cannot be found, a warning is presented. As with the previous scenarios, the message is presented in the lower right corner of the main Revit window:



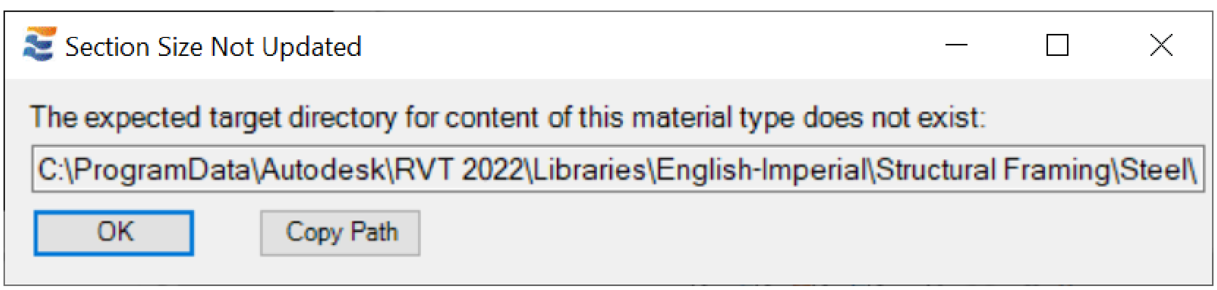
The warning message includes a courtesy option to copy the path shown so that the user may manually navigate folders via File Explorer and verify what has happened to the target directory. After the user dismisses this notification, the physical Revit element whose section size could not be changed will be placed in a warning state to remind the user that the ENERCALC calculation and Revit physical element are not in complete agreement. This warning state is visible via tabular summary in the Element Manager AND via color-coded highlighting in the Revit view:



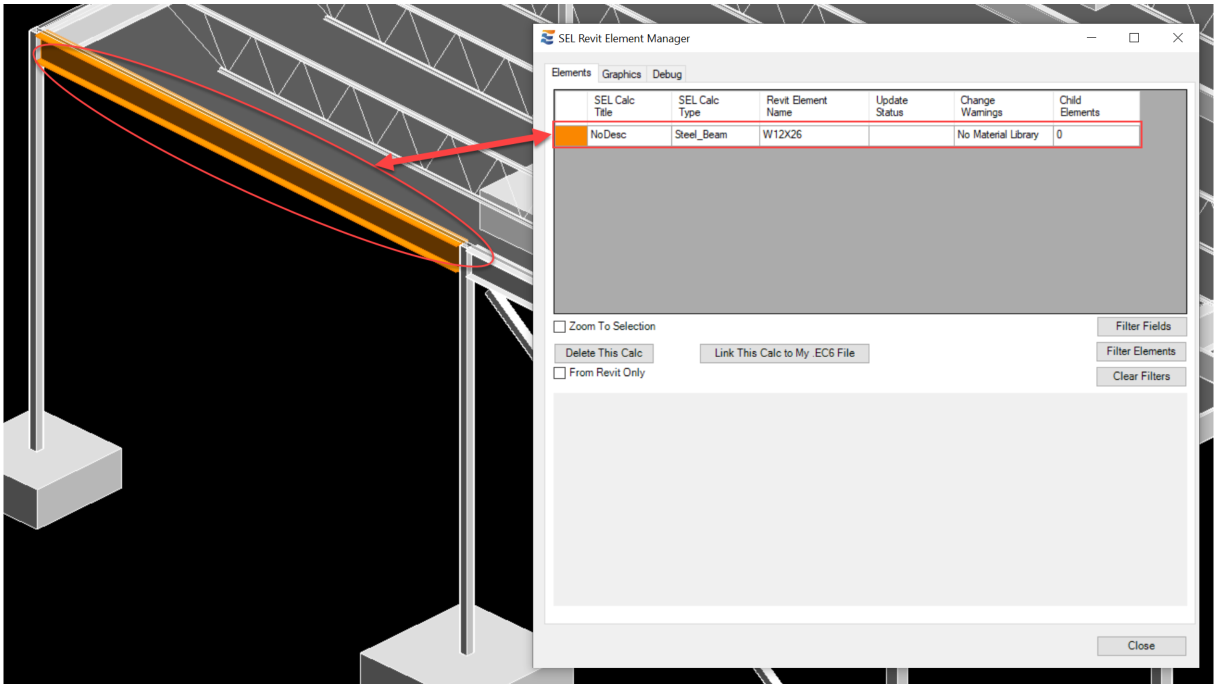
If the category library is found then EFR will automatically inspect the target directory to obtain content of the proper material. This inspection is based on the file structure used by Autodesk's default out-of-box loadable content:



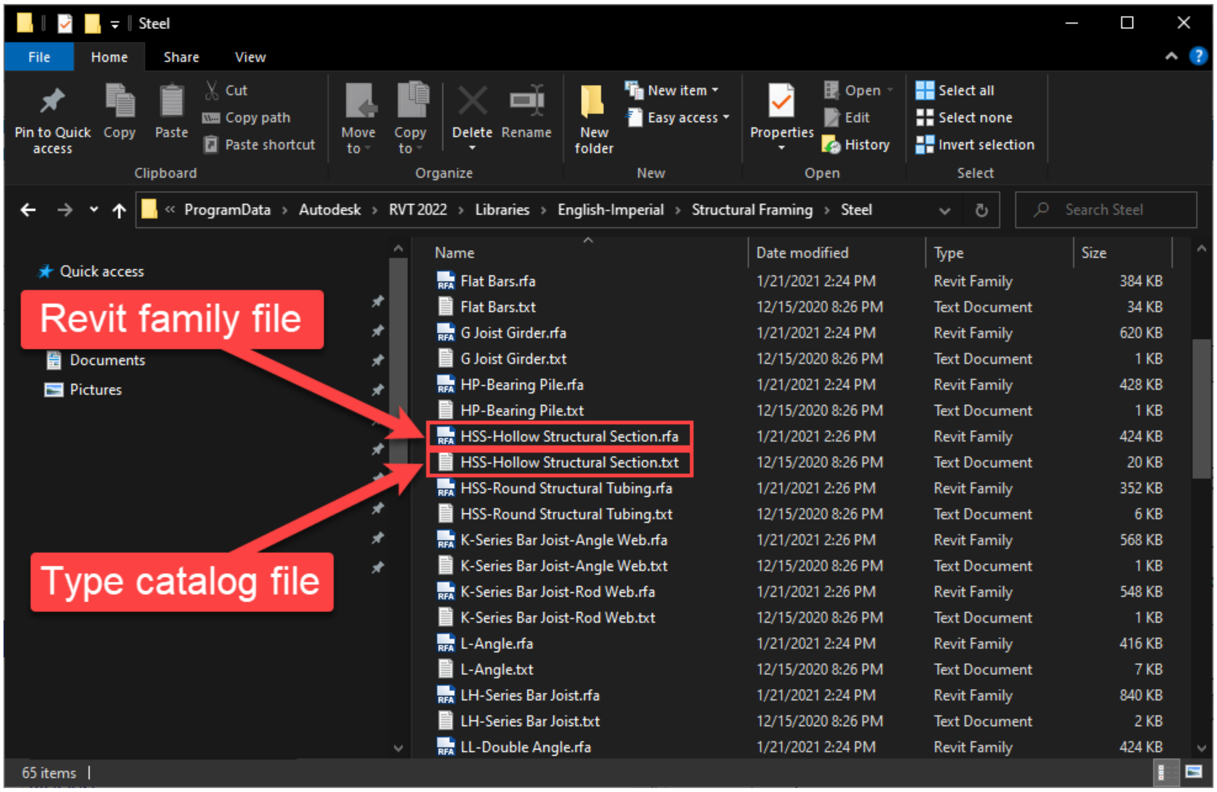
Based on this default file structure, EFR will automatically search for the sub-folder "\Steel" for steel calculations, "\Wood" for wood calculations, etc. If the appropriate category sub-folder cannot be found, a warning is presented. As with the previous scenarios, the message is presented in the lower right corner of the main Revit window:



The warning message includes a courtesy option to copy the path shown so that the user may manually navigate folders via File Explorer and verify what has happened to the target directory. After the user dismisses this notification, the physical Revit element whose section size could not be changed will be placed in a warning state to remind the user that the ENERCALC calculation and Revit physical element are not in complete agreement. This warning state is visible via tabular summary in the Element Manager AND via color-coded highlighting in the Revit view:



If the root library path and the required category and material sub-folders are found without any error messages, EFR will proceed to inspect the contents of the target folder for the required section size. The first phase of this search is to inspect the type catalog files found in the directory. The type catalog file for each family (.rfa) has the exact same file name as the family file, but ends with extension ".txt", as shown here:



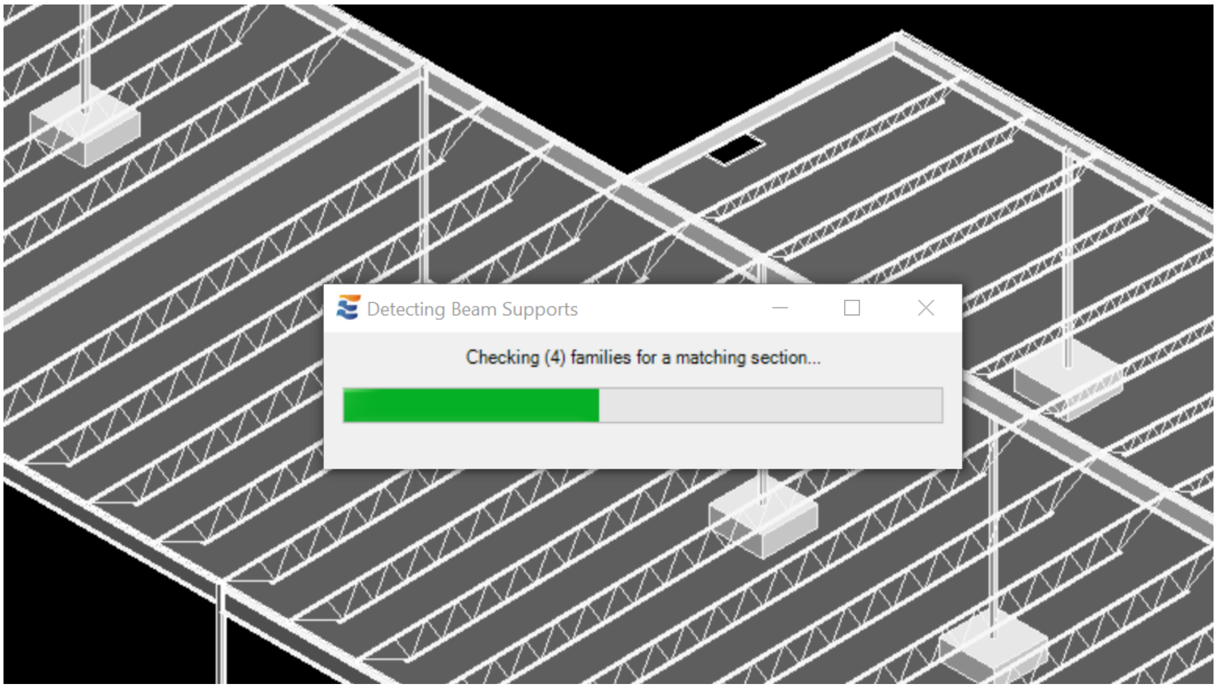
Type catalogs are typically used as a fast and lightweight approach to create a family with many types without the hassle of repetitive manual creation inside Revit's family editor GUI. The purpose of the type catalog file is to enumerate the type name and parameter values needed to construct each individual type within each family. For AISC standard steel sections and NDS wood sections, exhaustive type catalogs already exist and need not be created or edited by users. For example:

HSS-Hollow Structural Section.txt - Notepad

File Edit Format View Help

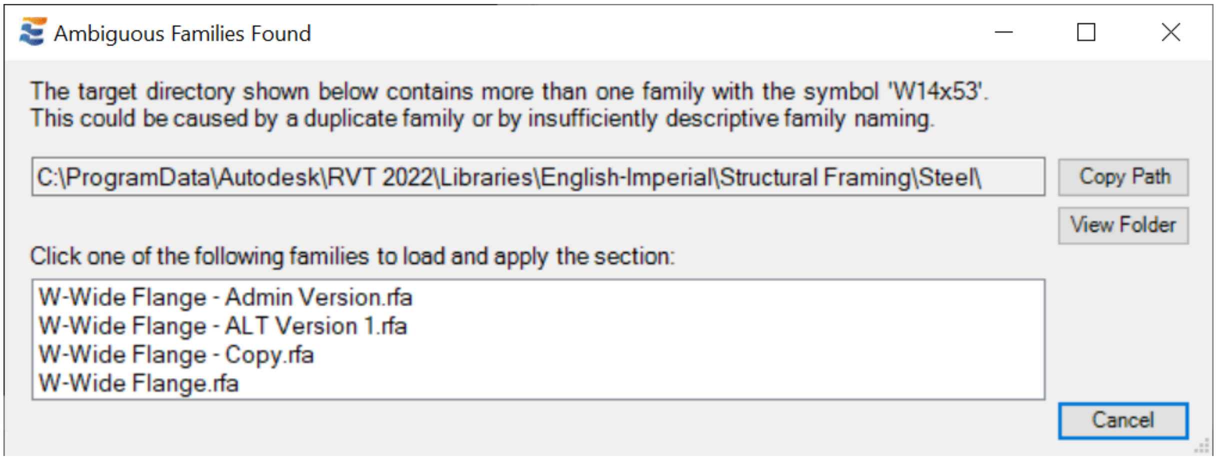
```
,Shape##other##,W##other##,A##area##inches,Ht##length##inches,b##length##inches,t##length##inches
HSS20X12X5/8,Rect.,127.000,35.000,20.000,12.000,0.581
HSS20X12X1/2,Rect.,103.000,28.300,20.000,12.000,0.465
HSS20X12X3/8,Rect.,78.400,21.500,20.000,12.000,0.349
HSS20X12X5/16,Rect.,65.800,18.100,20.000,12.000,0.291
HSS20X8X5/8,Rect.,110.000,30.300,20.000,8.000,0.581
HSS20X8X1/2,Rect.,89.600,24.600,20.000,8.000,0.465
HSS20X8X3/8,Rect.,68.300,18.700,20.000,8.000,0.349
HSS20X8X5/16,Rect.,57.300,15.700,20.000,8.000,0.291
HSS20X4X1/2,Rect.,75.900,20.900,20.000,4.000,0.465
HSS20X4X3/8,Rect.,58.100,16.000,20.000,4.000,0.349
HSS20X4X5/16,Rect.,48.900,13.400,20.000,4.000,0.291
HSS20X4X1/4,Rect.,39.500,10.800,20.000,4.000,0.233
HSS18X12X5/8 - Obsolete,Rect.,119.000,32.600,18.000,12.000,0.581
```

The existence of this file permits users to quickly and easily load any of the sections enumerated in the type catalog into the Revit project. The existence of the type catalog file also enables EFR to automatically preview and search the types available within each family to find the desired section sent by a given ENERCALC calculation. If a search of these type catalogs does not identify a matching section, EFR will automatically inspect the contents of the .rfa files themselves to verify if section types are stored within. This additional search step creates a short delay, denoted by the following progress bar:

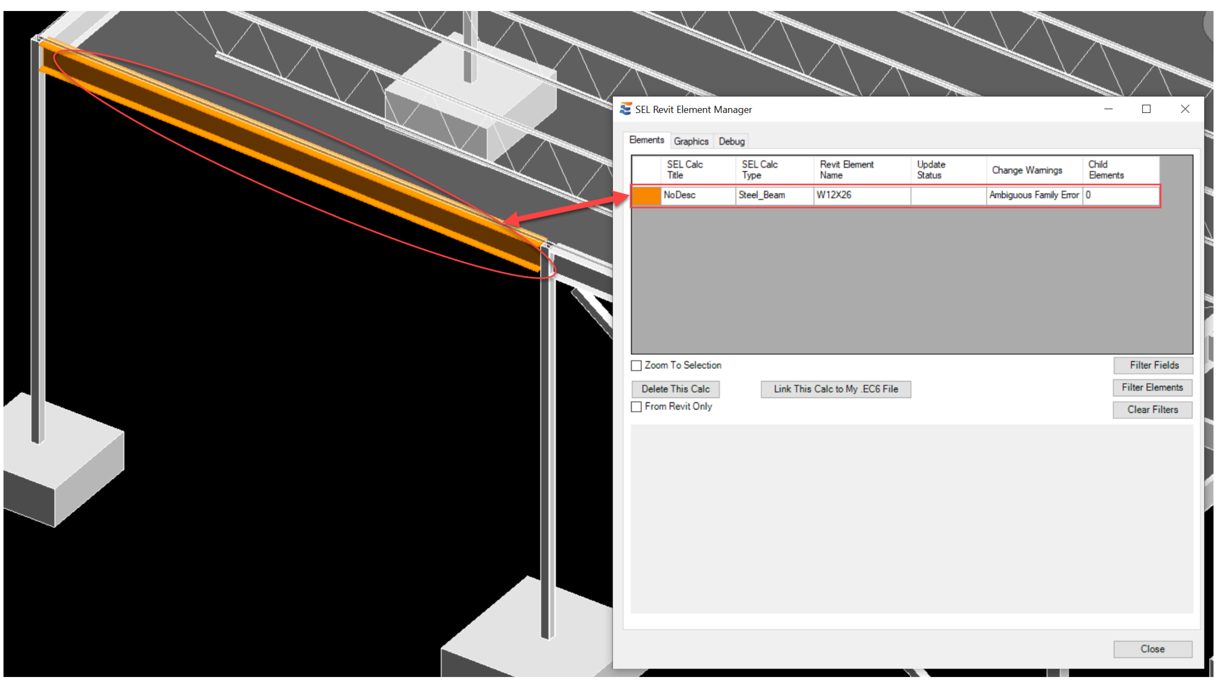


If EFR's search through these type catalogs and/or family files identifies one and only one section with the required name, then the section will be automatically loaded from the corresponding family.

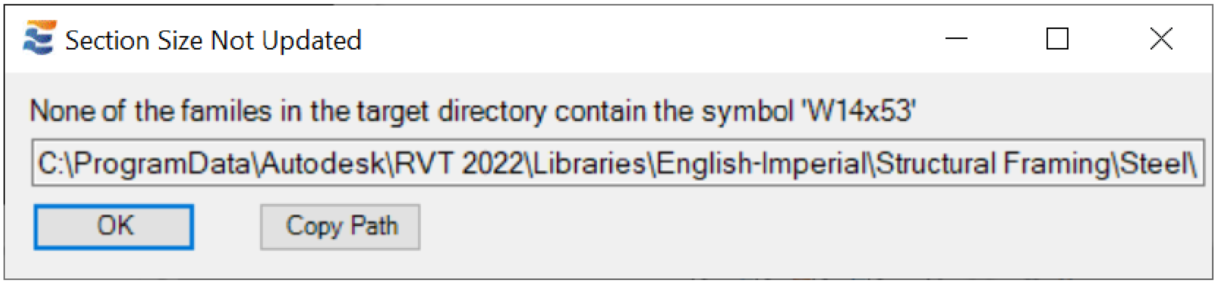
If EFR's search through these type catalogs identifies more than one family containing a section with the required name, the user will be prompted to resolve the ambiguity by specifying which family should be used. As with the other warnings discussed above, this dialog will be presented in the lower right corner of the main Revit window:



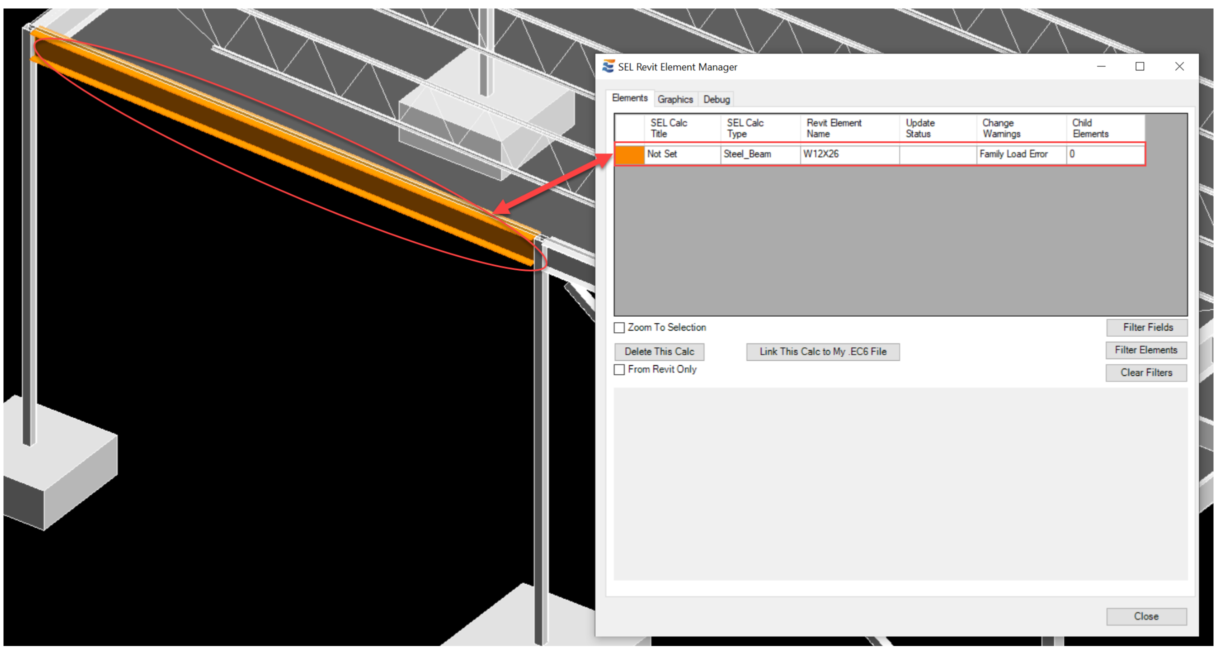
As with the previous warnings, this message includes a courtesy option to copy the path shown or view the folder to manually navigate folders via File Explorer and verify the presence of ambiguous families in the target directory. Clicking one of the family names shown in this list will allow the process to complete. Once a family is selected, the menu will be dismissed, the new section size will be loaded from the selected family, and the size of the Revit beam element will be toggled. If the user chooses to cancel or close this dialog the physical Revit element whose section size could not be changed will be placed in a warning state to remind the user that the ENERCALC calculation and Revit physical element are not in complete agreement. This warning state is visible via tabular summary in the Element Manager AND via color-coded highlighting in the Revit view:



Conversely, if EFR's search through these type catalogs identifies no family containing a section with the required name, the user will be notified that no match was found. As with the other warnings discussed above, this dialog will be presented in the lower right corner of the main Revit window:



If no section match could be found, the physical Revit element whose section size could not be changed will be placed in a warning state to remind the user that the ENERCALC calculation and Revit physical element are not in complete agreement. This warning state is visible via tabular summary in the Element Manager AND via color-coded highlighting in the Revit view:

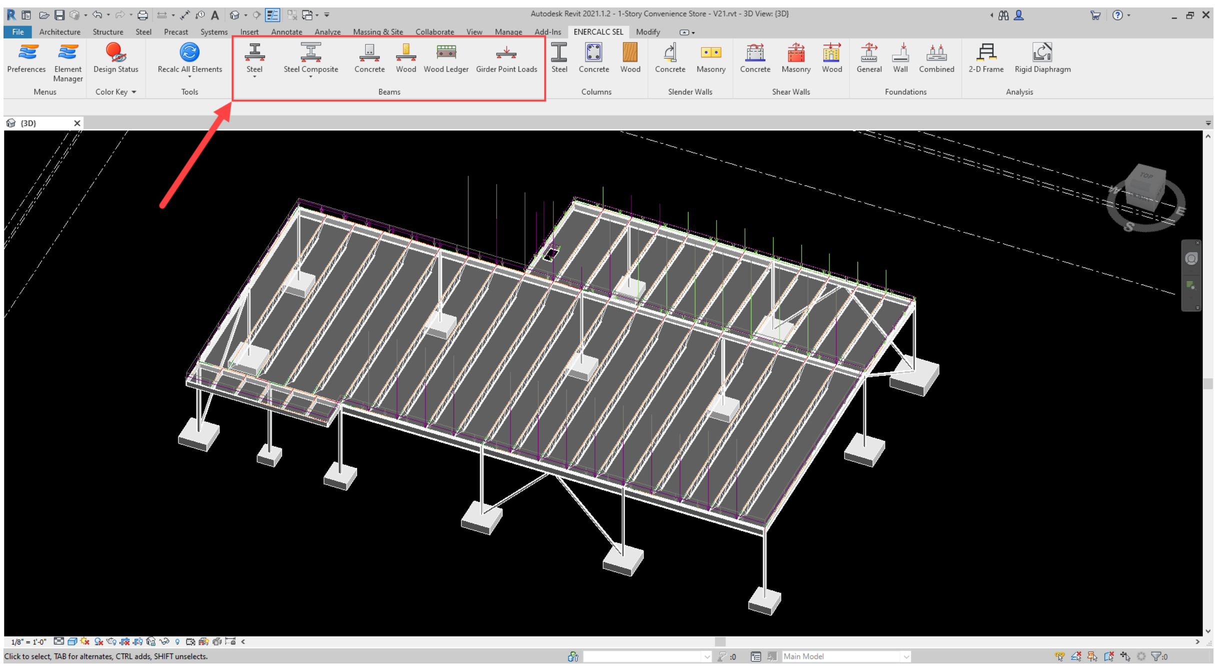


Part X

10 Working With Beam Calculations

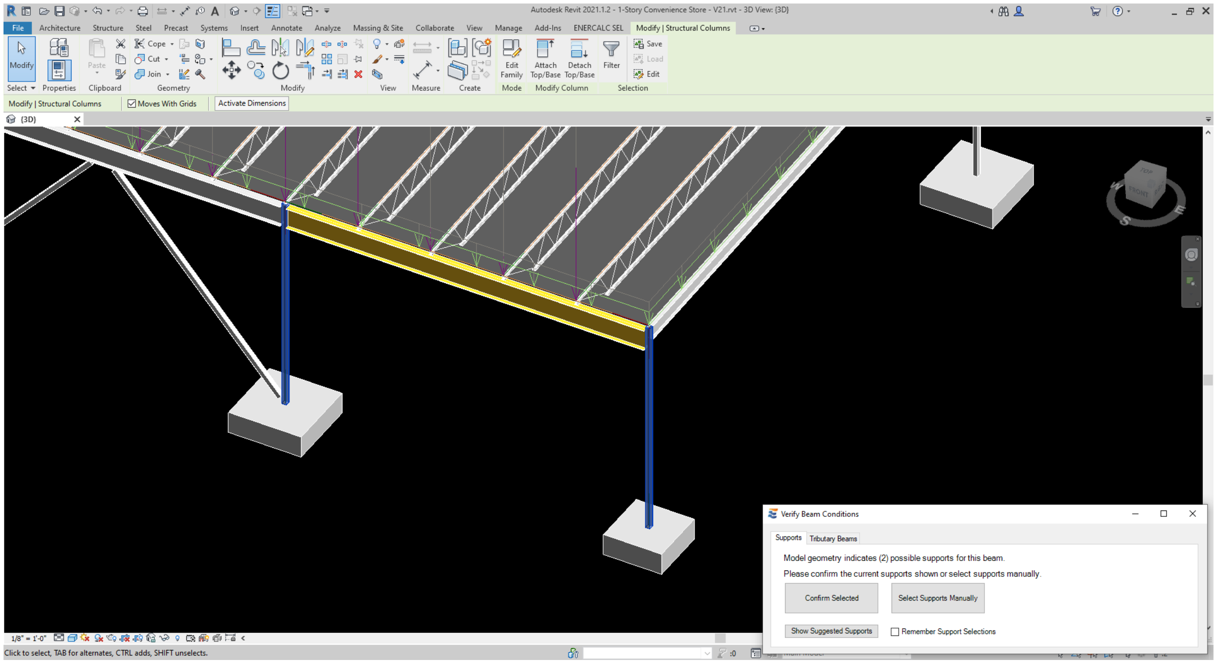
Various types of beam calculations share common behavior during the launch process. This section will discuss the specific aspects of ENERCALC for Revit that are shared by all beam calculations. All beam calculations are launched using the appropriate button from the "Beams" panel of the ENERCALC ribbon bar in Revit.

The ribbon bar shown below is for illustration only. Refer to your particular installation to verify which module design tools will be displayed in your Revit interface.

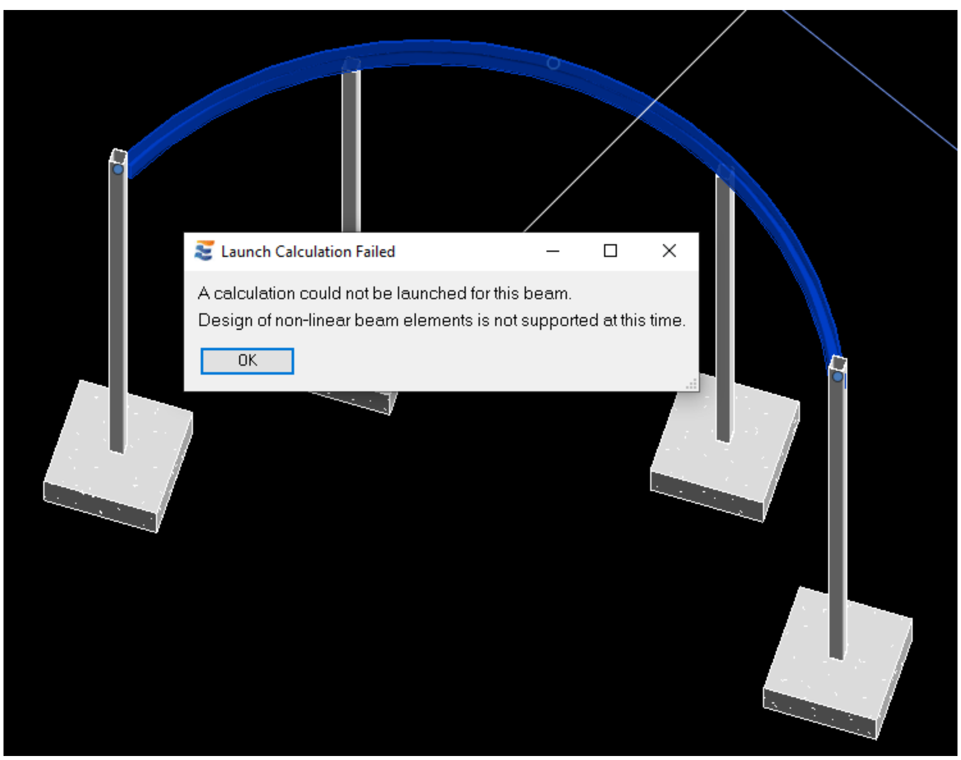


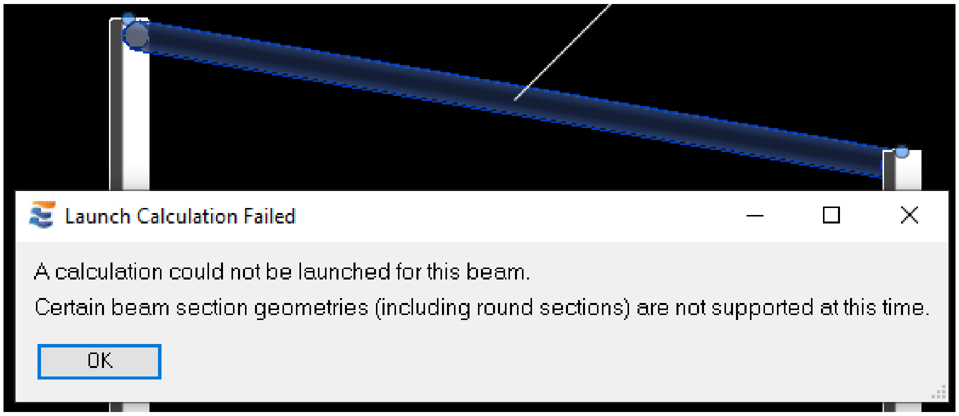
The launching of a beam calculation requires user input to confirm various information and construct an accurate and detailed structural design upon arrival in the ENERCALC interface. This process is managed from a tabular form that appears in the lower right-hand corner of the Revit interface when a calculation is launched.

The form is progressive and leads the user from tab to tab as approvals and verifications are completed. This form does not block access to the Revit UI. The user may freely navigate Revit views via zoom, pan, and rotate controls and may move between Revit views or open new views, as well as using measuring or annotation tools to verify geometry. During the course of the approval launch process, the beam that is to be designed will be highlighted in yellow (meaning "in progress"). This highlighting is only applied to the view that was active at the time of calculation launch. Closing the launch form at any time during the process will abort the calculation launch process. In the case of an aborted launch, the yellow highlighting on the beam will be removed and no calculation will be created in the linked .EC6 file.



At the present time, there are two specific cases (curved beams and round sections) that are NOT supported in ENERCALC for Revit. A beam meeting either one of these conditions will not result in a beam calculation launch.





10.1 Beam Supports

Upon launching a beam calculation, ENERCALC for Revit will start the process by identifying the elements that support the beam. This will allow span geometry to be defined in the ENERCALC calculation module, and will establish the connective relationships that determine load path, re-calculation order, and change warnings. ENERCALC for Revit leverages a variety of information available in the Revit model, including physical solid location of the beam element, line location of the beam element, and proximity to other elements in order to suggest the most probable support elements and present them to the user.

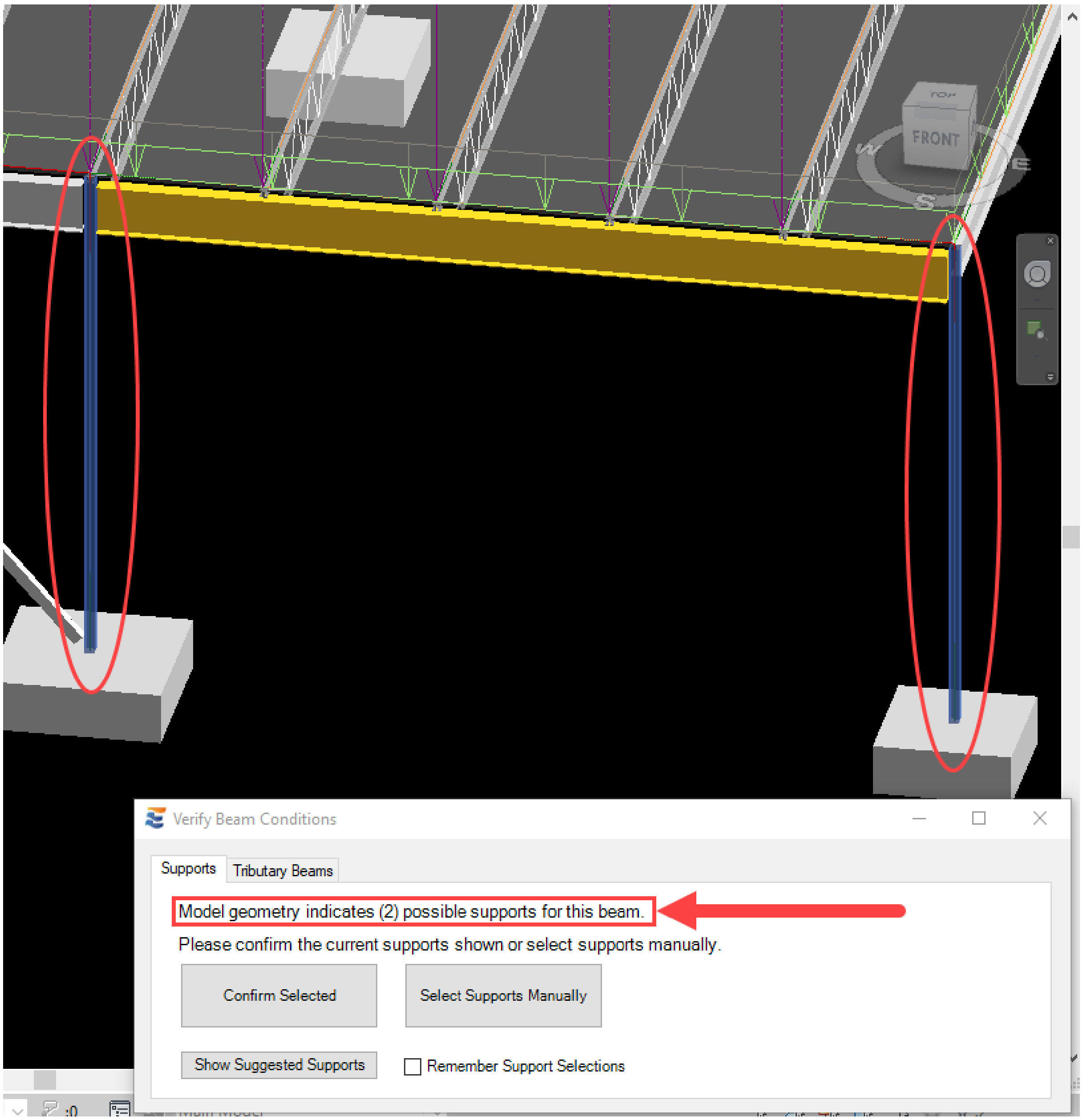
Beams may be supported by any of the following element types:

- Structural Columns (from below or above)
- Structural Framing as beams
- Structural Walls

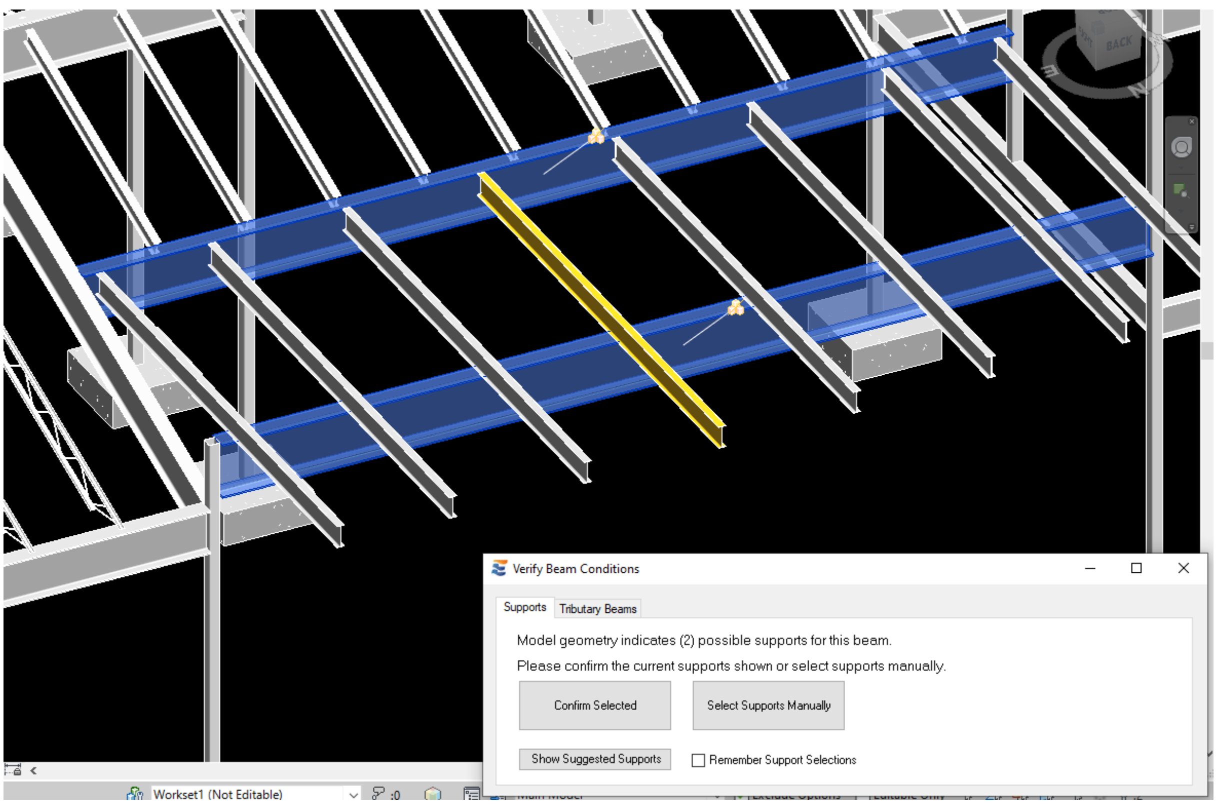
Beams may be not supported by any of the following element types:

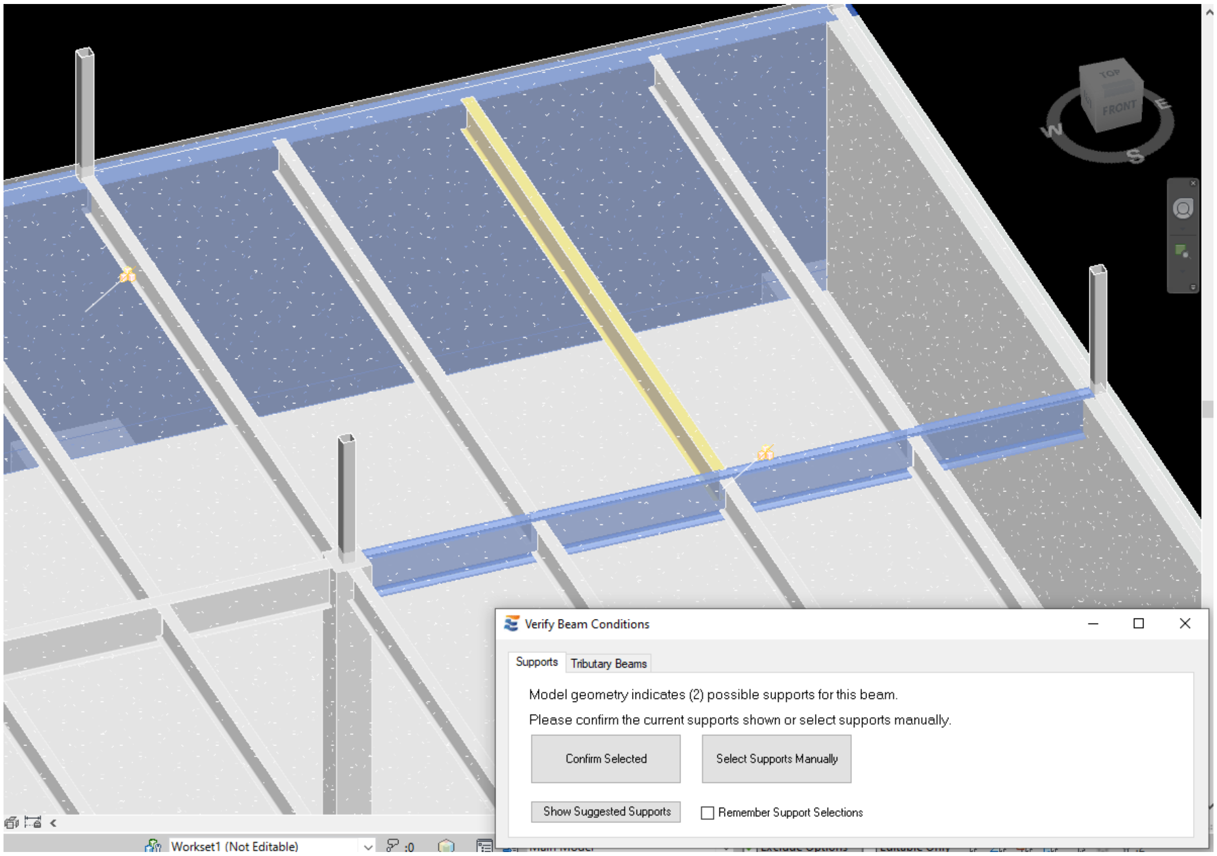
- Structural Framing as vertical bracing
- Structural Framing as horizontal bracing
- Structural Floors
- Structural Foundations
- Structural Walls with a non-bearing usage type
- Non-Structural Walls
- Any other non-structural element

The suggested supports will be automatically selected in the Revit interface for the user's attention. The prompt on the approval form will indicate the number of suggested supports found.




Verify Beam Conditions
Supports
Tributary Beams
Model geometry indicates (3) possible supports for this beam.
Please confirm the current supports shown or select supports manually.
Confirm Selected
Select Supports Manually
Show Suggested Supports
Remember Support Selections

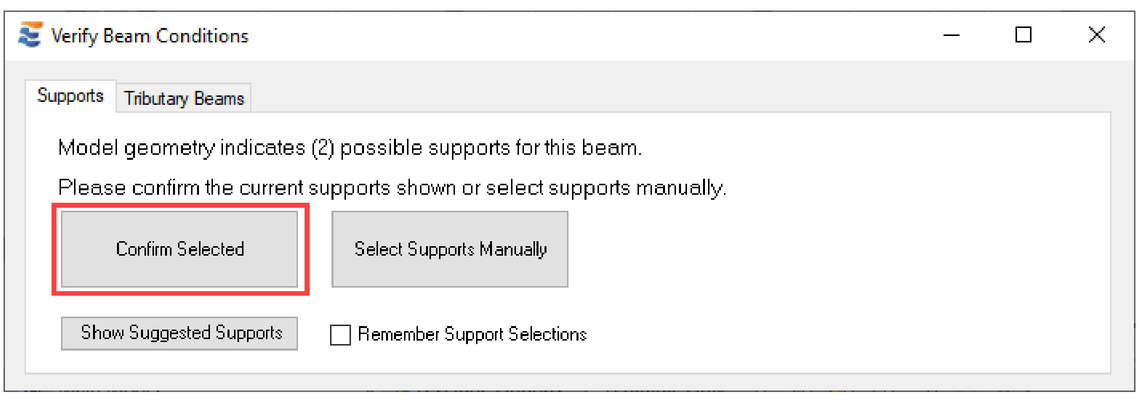

Verify Beam Conditions
Supports
Tributary Beams
Model geometry indicates (2) possible supports for this beam.
Please confirm the current supports shown or select supports manually.
Confirm Selected
Select Supports Manually
Show Suggested Supports
Remember Support Selections
Workset1 (Not Editable)



.

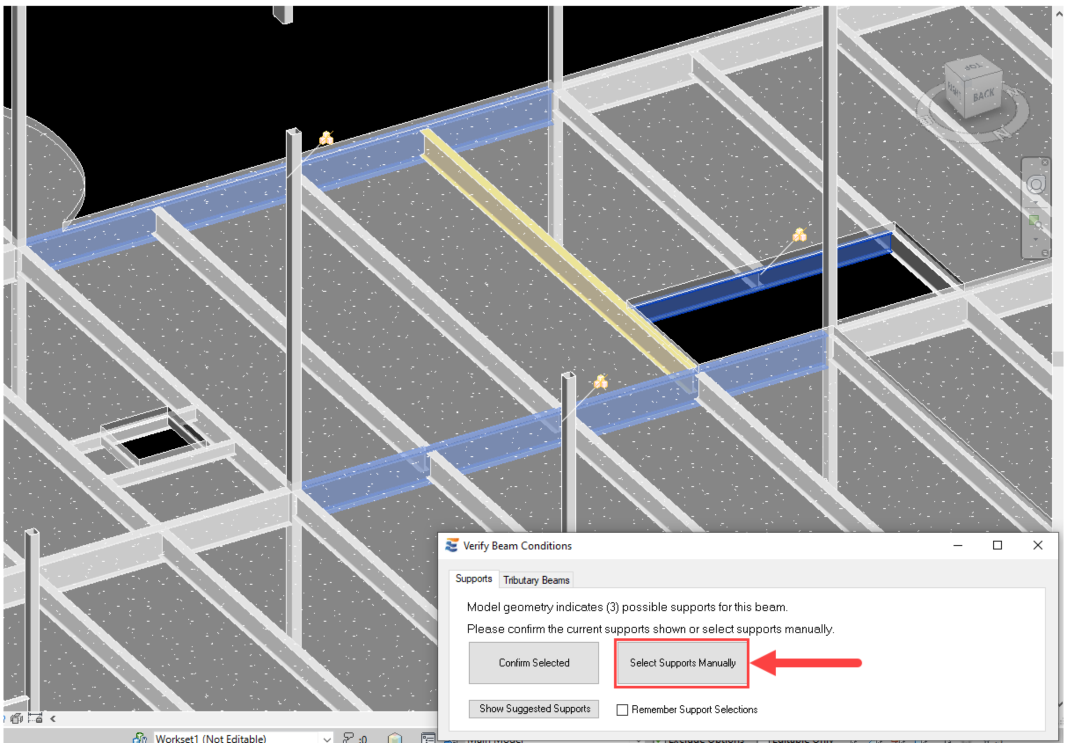
10.1.1 Approving Support Conditions

If the user is satisfied with the support conditions suggested by ENERCALC for Revit, the supports may be approved by simply clicking the “Confirm Selected” button. After confirmation, the launch form will automatically proceed to the next tab of the launch process.

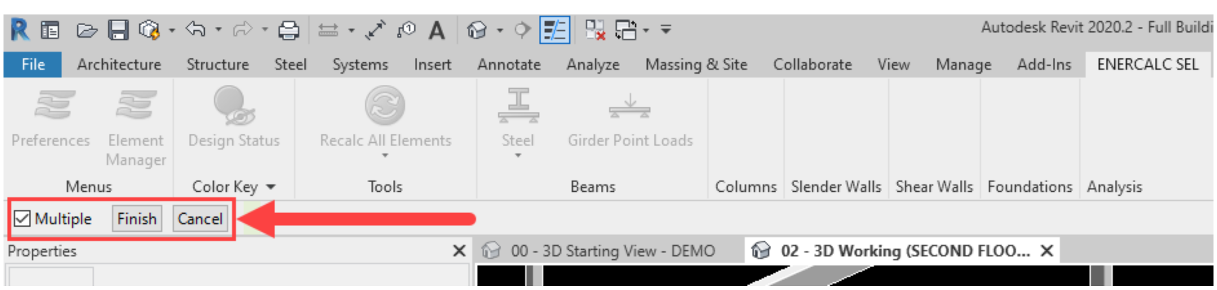


10.1.2 Manually Selecting Support Conditions

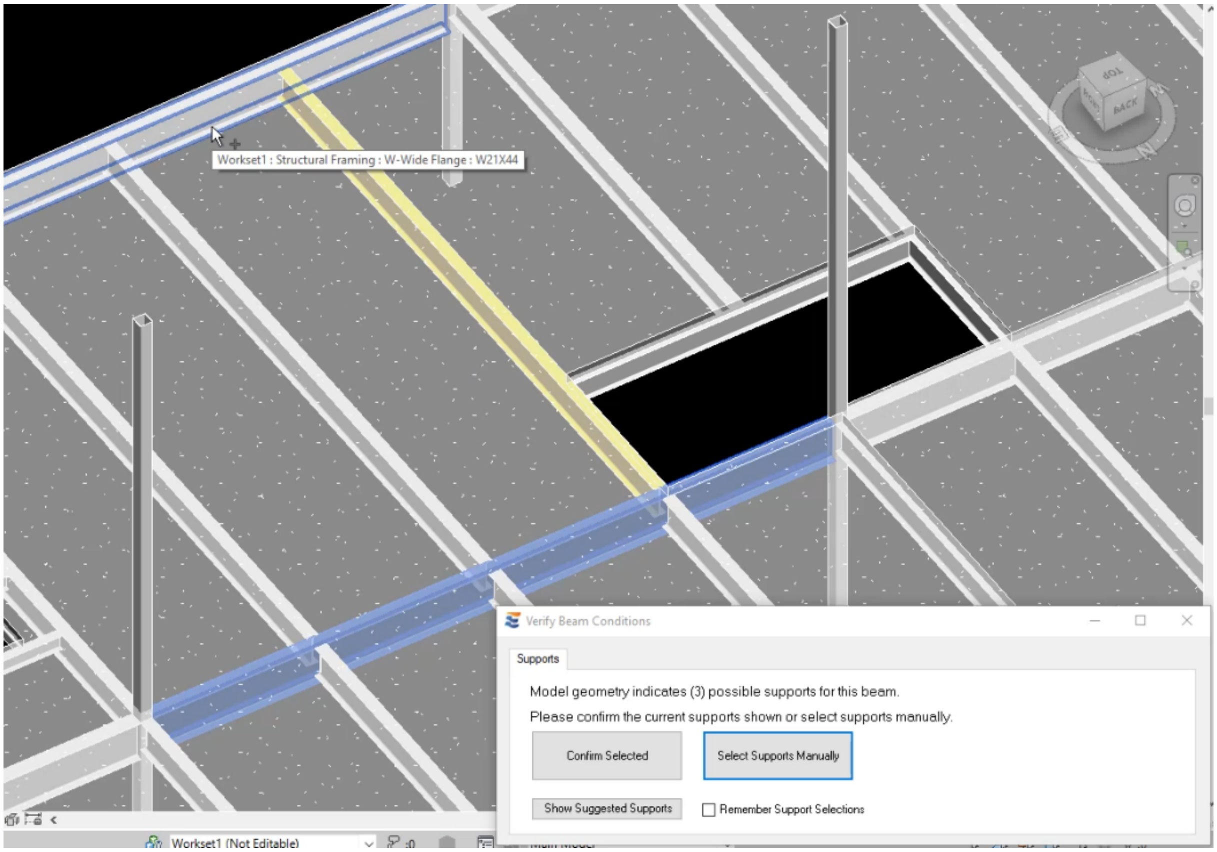
Although the automatic suggestion of support conditions is provided as a convenience, ENERCALC for Revit cannot autonomously interpret the user's design intent. It is important that the design professional diligently monitor the suggestions produced during launch and manually intervene to ensure accuracy whenever necessary. In situations where the suggested supports do not accurately describe the design intent, a user may alternatively choose to manually specify the supporting elements. Custom selection of supports is triggered by clicking the "Select Supports Manually" button.



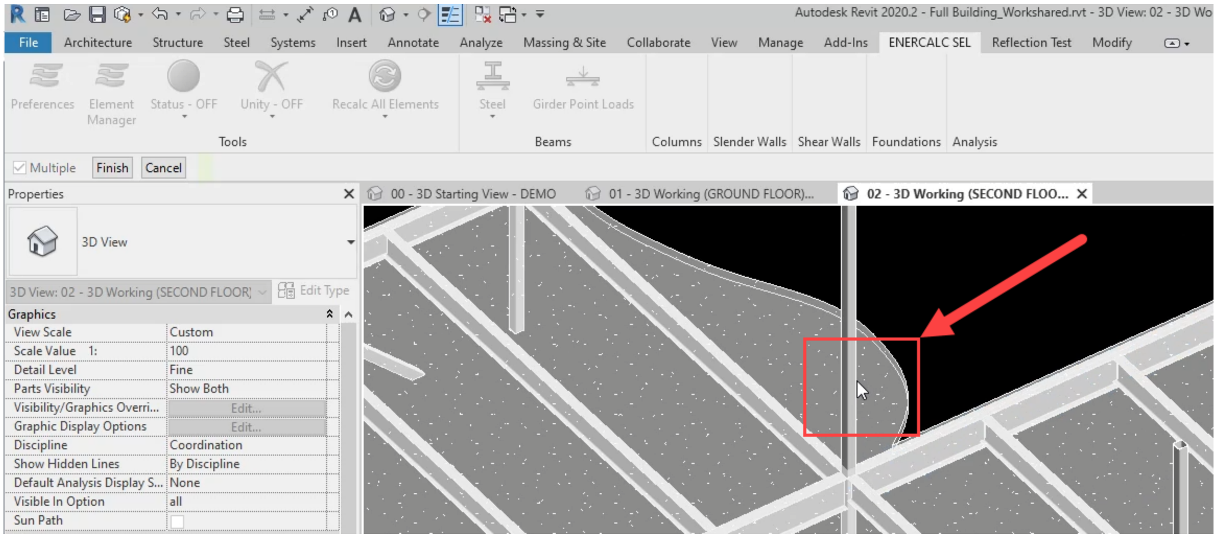
Clicking this button will initiate a Revit multi-select process similar to that used for other native Revit operations. When the multi-select is active, the Revit ribbon bar will display "Finish" and "Cancel" buttons on the upper left-hand corner of the main Revit window, and the user will have the ability to manually pick elements.



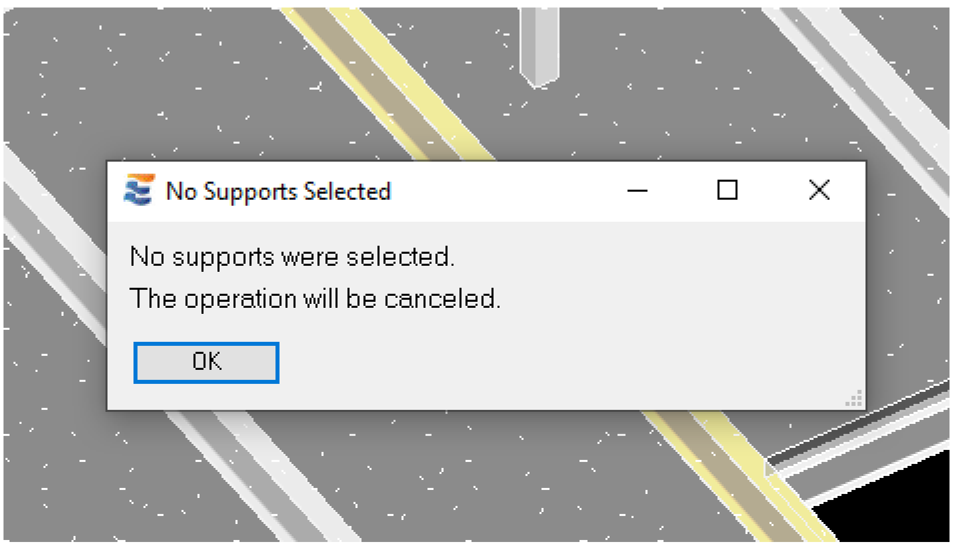
During the selection process, elements already picked will remain selected (and highlighted) in the active view. Items eligible for selection will light up with a "+" symbol when the cursor passes over.



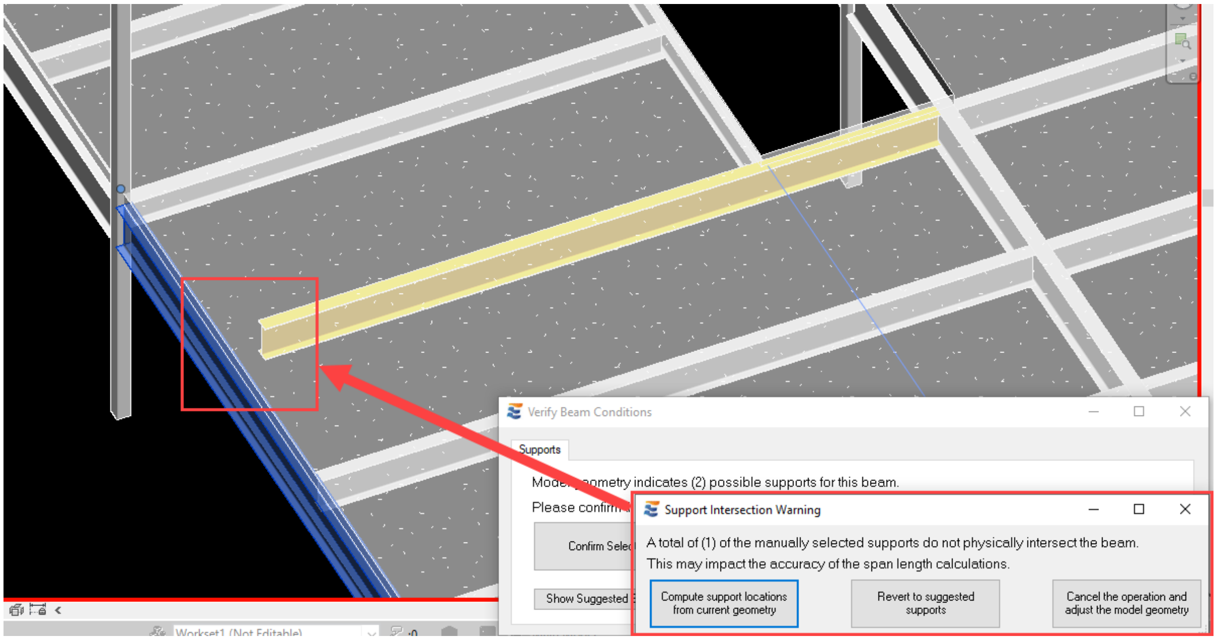
The support selection process is also subject to several restrictions. In order to limit downstream errors from difficulty in defining span geometry, the user is restricted to only selecting elements that are in actual proximity to the beam. Elements outside the immediate vicinity of the beam will not be clickable. The user is also restricted to only selecting element types that are eligible to act as beam supports. When the user points to an element that is not eligible to act as a support for the current beam, the element will not highlight and the cursor will not show the “+” symbol. For example, the column in the image below is too distant from the beam to be eligible for selection as a support.



If at any point during the multi-select process the user clicks the “Cancel” button on the Revit ribbon bar, the calculation launch process will abort with a notification. After the notification is dismissed, the launch approval form will close and the yellow highlighting on the beam will be removed.



In some cases, the user may choose to manually select a supporting element that is near enough to the beam to be eligible but does not physically intersect. When this happens, the user will be presented with a warning and three options to resolve the issue.



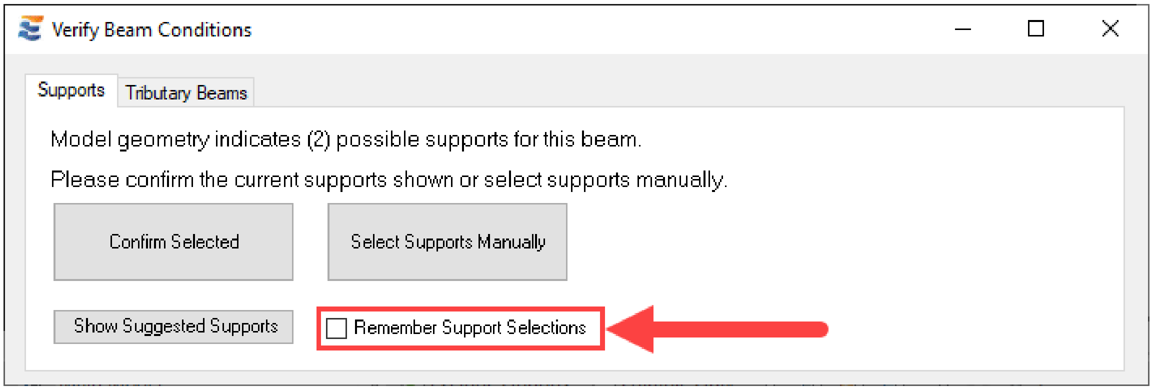
Clicking the left-most button will cause the program to proceed with the launch process using the specified support. Span geometry will be computed from the nearest extrapolated intersection of the two elements. For some geometry cases (such as the one shown above), this option is appropriate and will result in accurate beam analysis. In some cases, however, the lack of physical intersection could indicate the lack of any meaningful relationship by which spans may be established. If choosing this option results in a downstream warning when span geometry cannot be calculated (or if the calculation in ENERCALC SEL shows inaccurate geometry) then it is advisable to adjust the Revit model for a more accurate support condition.

Clicking the middle button will cause the program to proceed with the suggested supports rather than the manually specified ones. Suggested supports will typically be in direct contact with the beam to be designed, meaning that extrapolation of nearest intersection points is not required.

Clicking the right-most button will cause the launch approval form to close and the yellow highlighting on the beam will be removed.

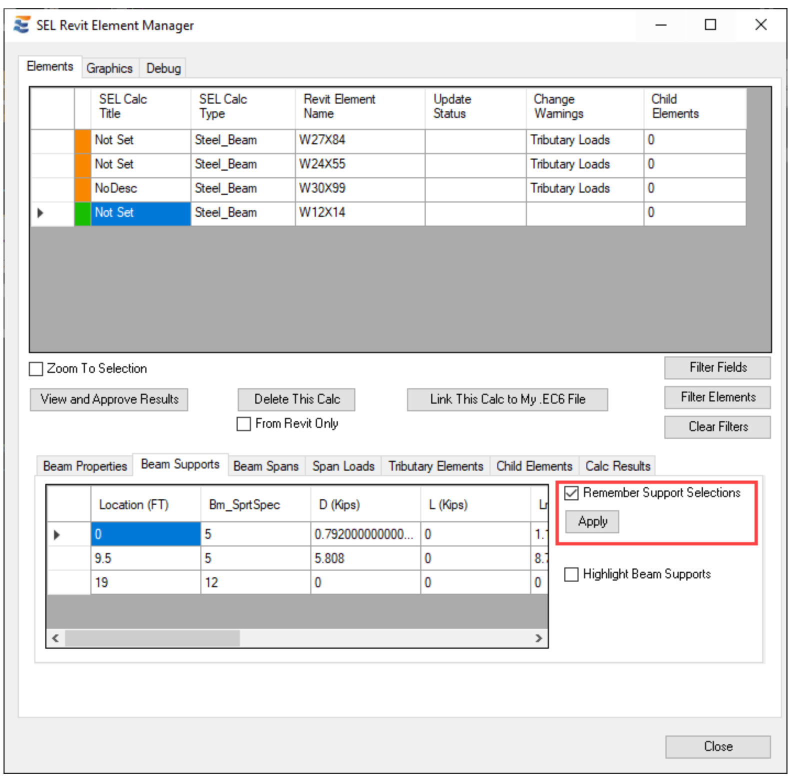
10.1.3 Remembering Support Conditions

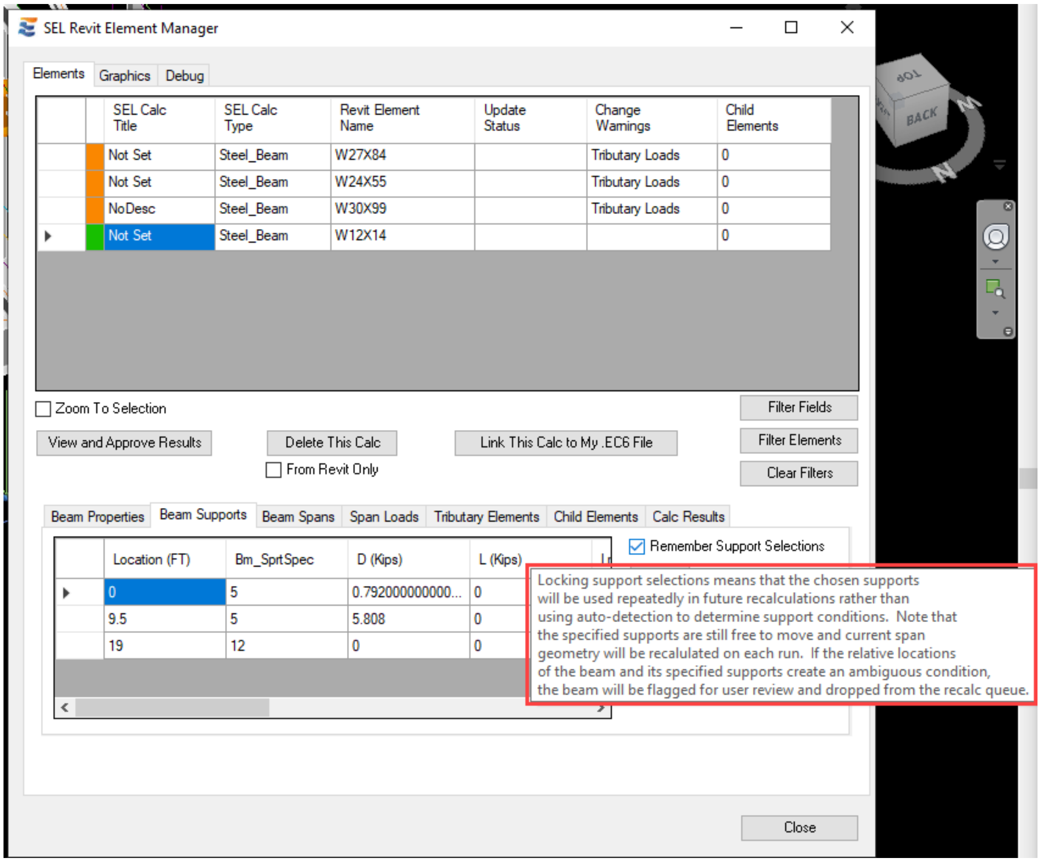
Regardless of the method used to choose the beam supports, the launch window includes an option to “Remember Support Selections”.



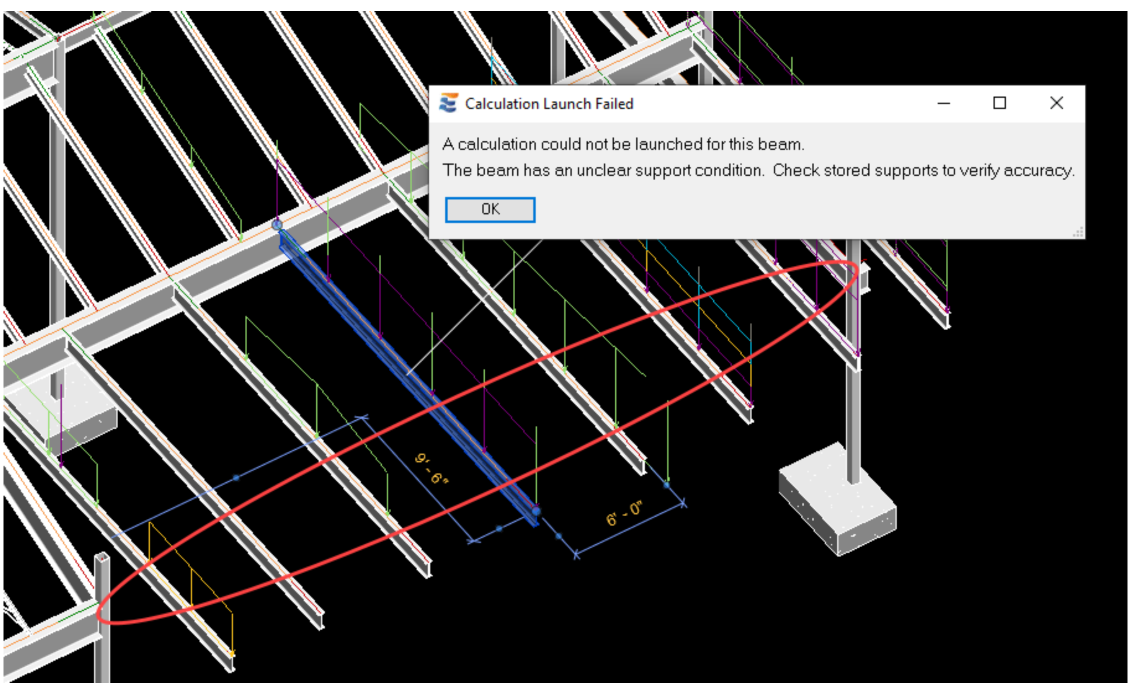
Checking this box prior to the approval of support conditions will allow ENERCALC for Revit to store the identities of the approved supports during future launches of the calculation. This will permit rapid updating of the beam's span geometry without the inconvenience of repetitive approvals. If supports have been stored, the launch window will not show a "Supports" tab for user approval.

If at any point it becomes necessary to choose new support conditions for a beam, the previously stored supports may be unlocked via the Element Manager. Conversely, if the supports were approved but not "Remembered" during the previous launch, the "Remember" option may be enabled from this same menu.





Note that even when supports have been stored, the launch still includes error checking to validate the previously stored supports. If a previously stored support no longer exists in the Revit model, then the launch will be aborted with a notification to the user.

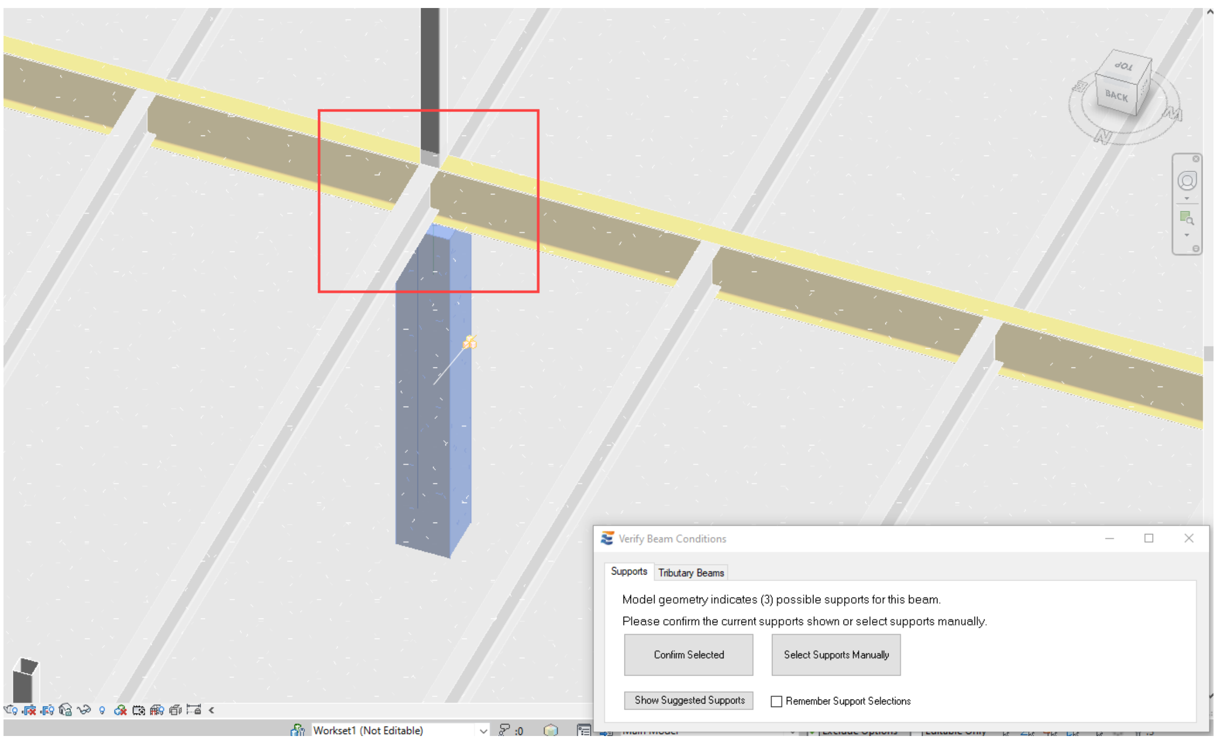


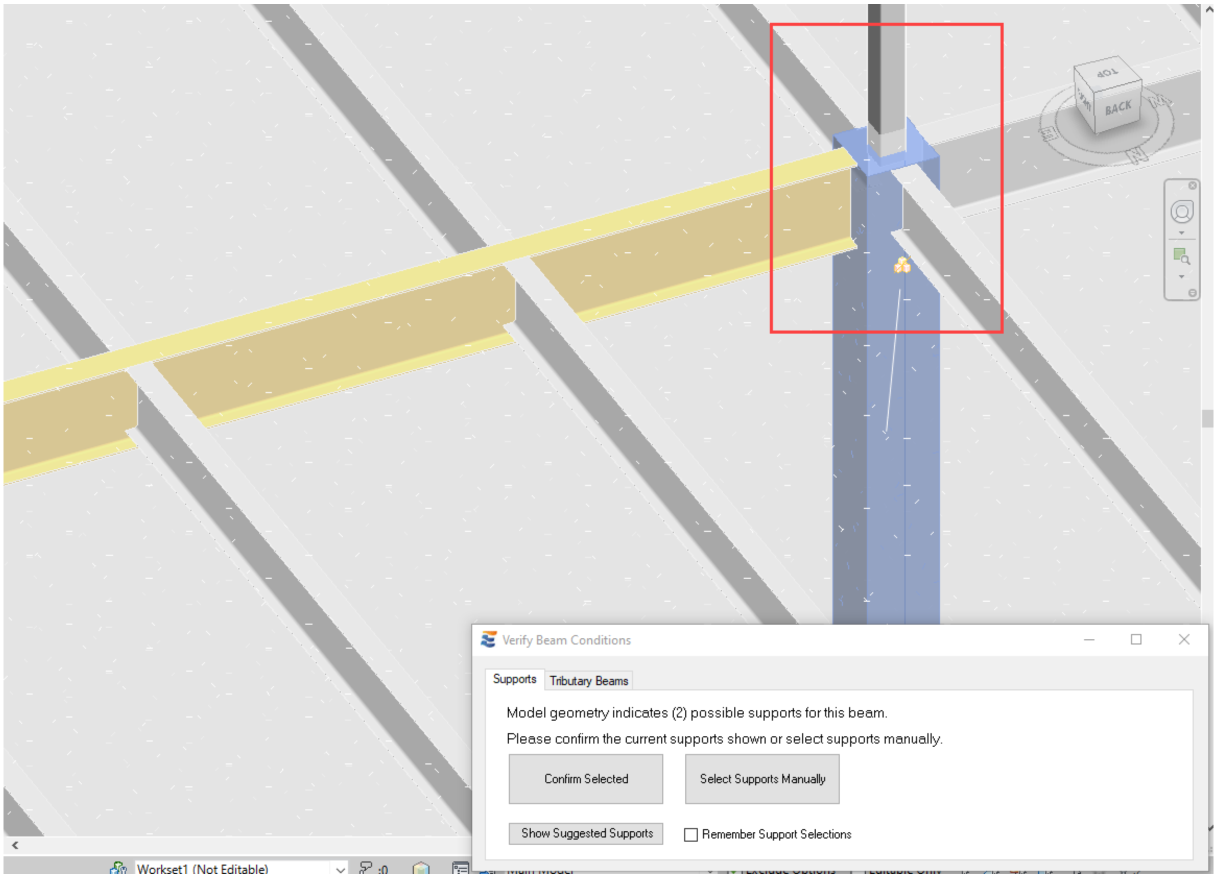
If the stored supports exists but does not physical intersect the beam, the nearest intersection will be extrapolated when span geometry is constructed.

10.1.4 Resolving Ambiguous Support Conditions

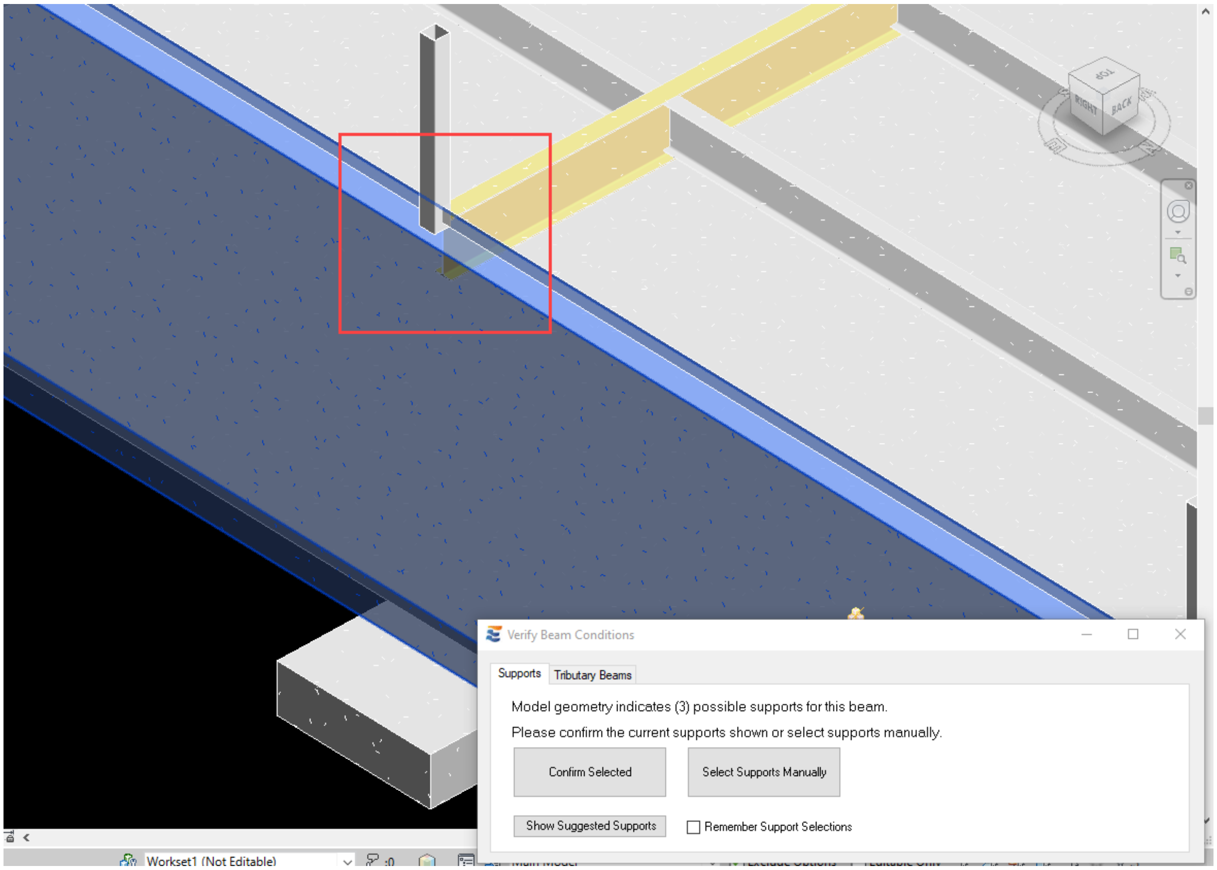
In some framing conditions, ENERCALC for Revit's detection process may identify multiple possible support elements very close to each other. A number of these ambiguities are resolved on the fly without user input by the following precedence rules:

1. When two possible supports coincide at the same location, priority is given to the element whose solid centroid is lower in 3D space. According to this logic, framing conditions with stacked columns will not result in ambiguity warnings because the lower of the two columns is suggested by default.





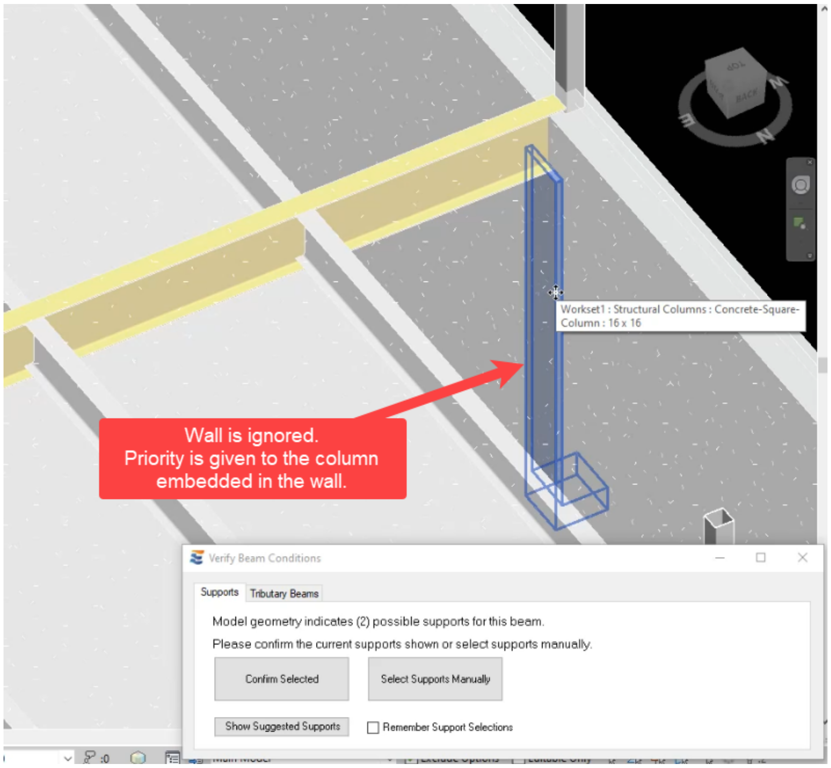
Similarly, a wall below with column above condition will not result in an ambiguity warning because the lower of the two (the wall) is suggested by default.

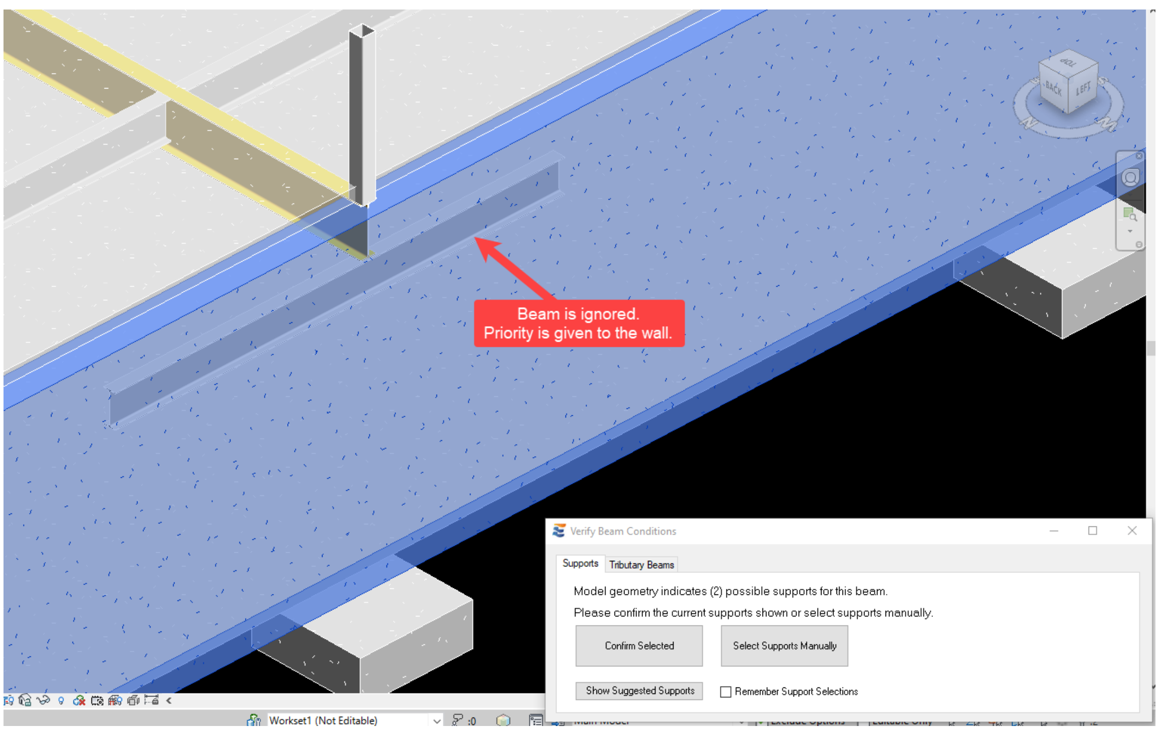


2. If the coincident supports have closely matched physical locations in space such that the “lowest support” test is inconclusive, then the following priority is applied:

   a. Columns have priority over walls and beams

   b. Walls have priority over beams

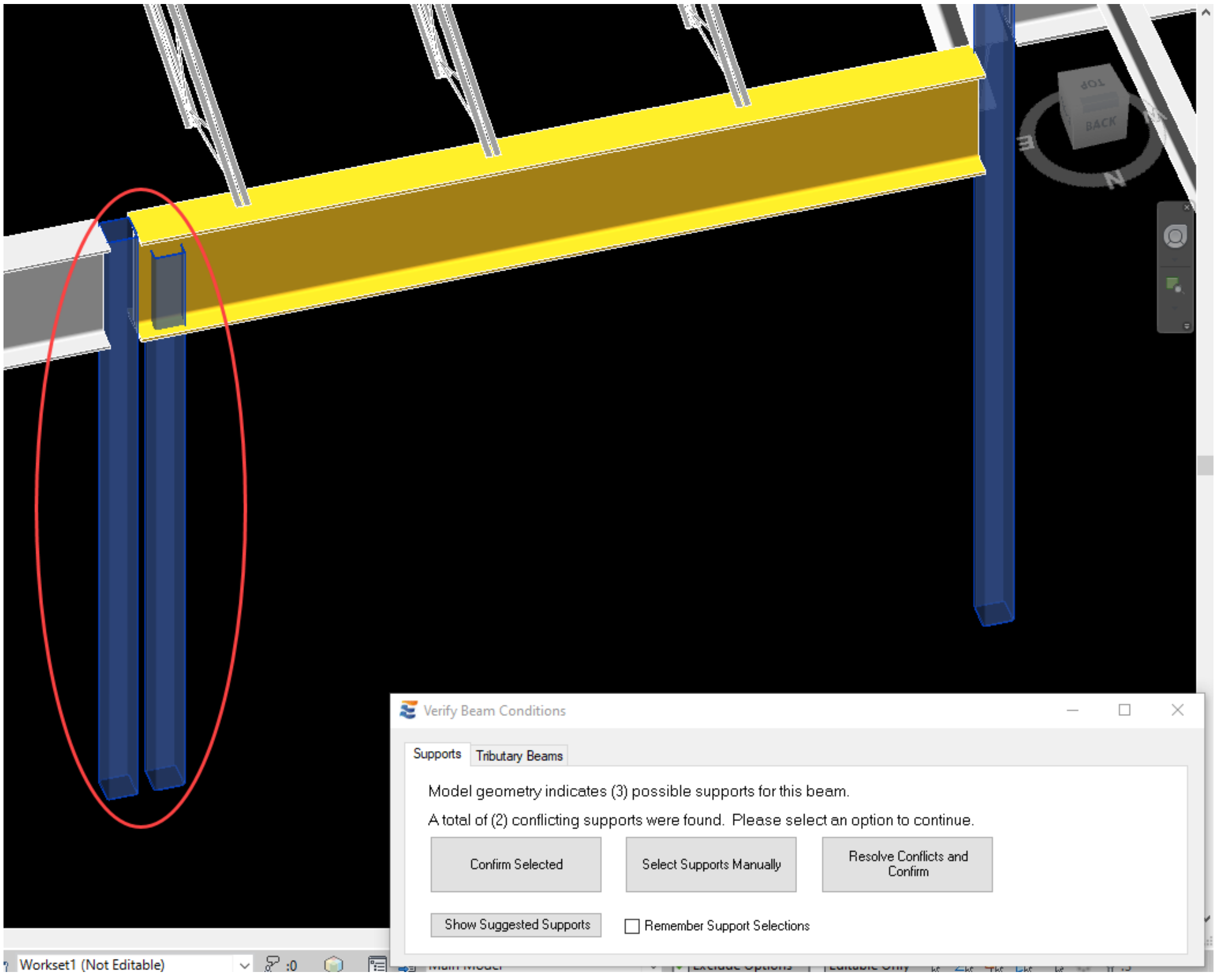




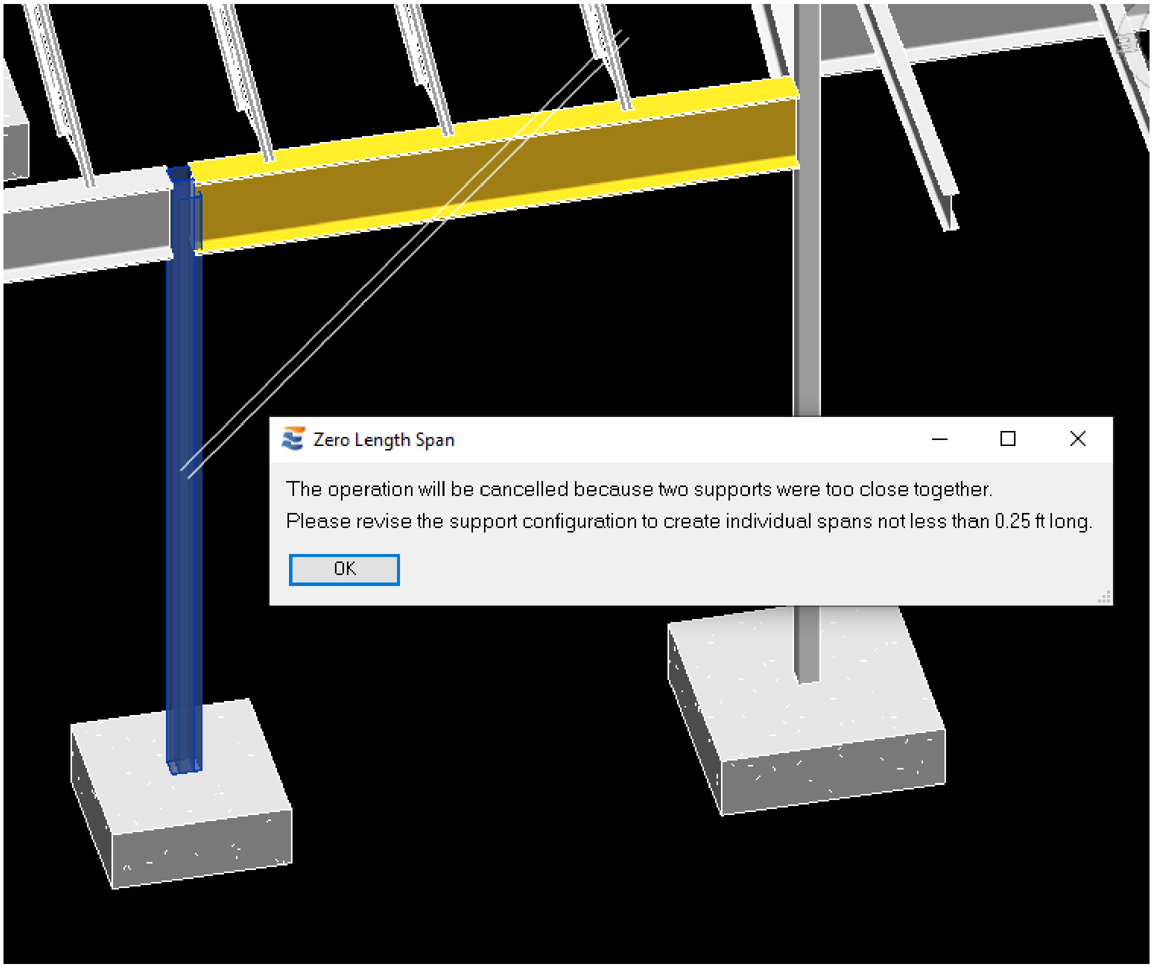
For any framing condition where the user's design intent does not align with the logic used to automatically limit ambiguous conditions, the load path may always be corrected by choosing the "Select Supports Manually" button. As discussed in "Manually Selecting Support

Conditions 114", the user may manually choose from ***ANY*** beam, column, or wall element that is found in physical proximity to the beam to be designed.

The proximity and geometry of the possible supports may not allow for a clear recommendation for one element or the other in all cases. If this occurs, the user will be notified in the approval form that a potentially conflicting support condition is found. For example, if two columns with nearly identical geometry are found very close to each other, the user will be presented with the option to "Confirm Selected", "Select Supports Manually", or "Resolve Conflicts".

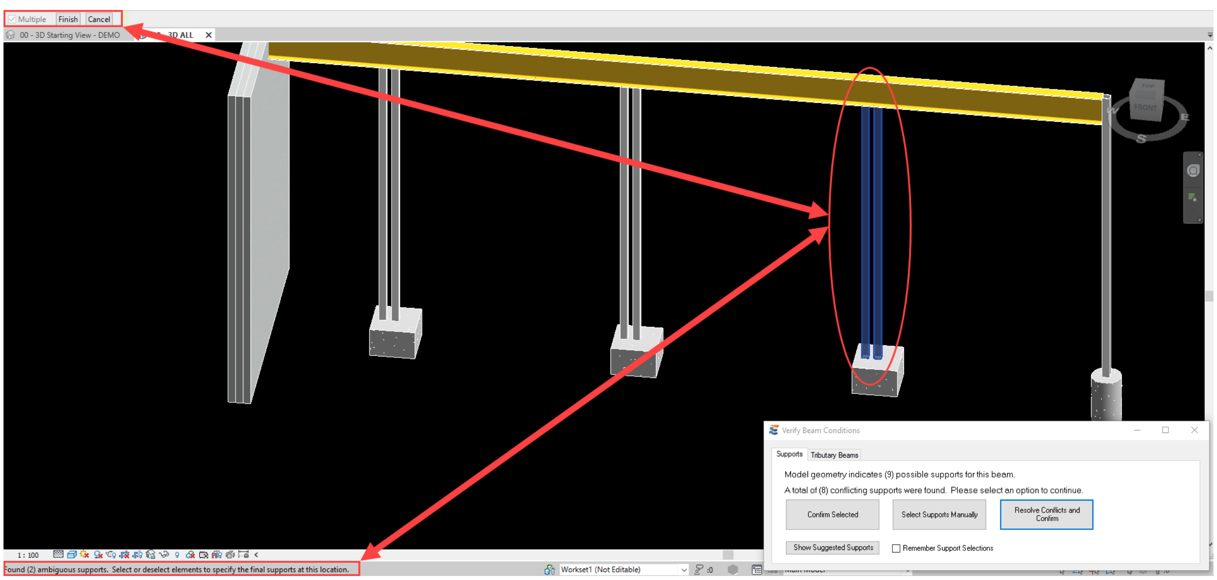


Users should be aware that proceeding via "Confirm Selected" or "Select Manually" may result in a downstream warning if the chosen supports are placed too close together. The minimum span length permitted is 0.25 ft. If any two support centers are more closely spaced, the calculation launch will be terminated with a warning, and the problematic supports will be indicated in the Revit view.

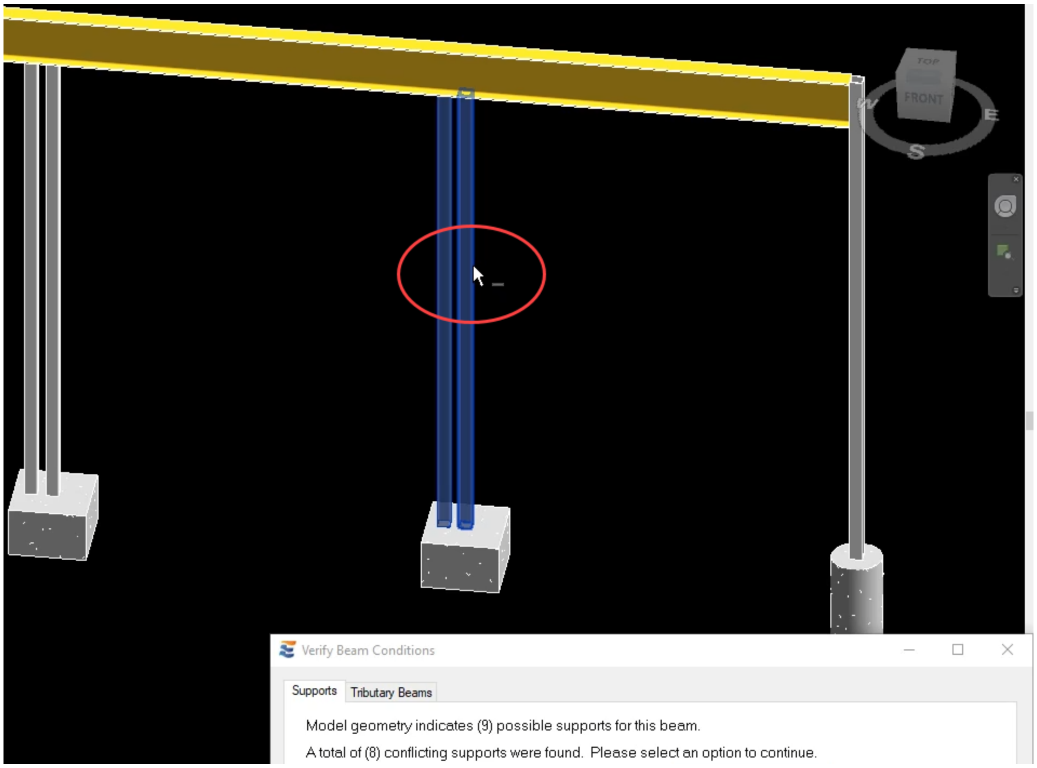


If the user opts to “Resolve…”, then the program will lead the user through a series of verifications for each location along the beam where conflicting supports are found. Clicking this button will take the user to the first location where supports conflict.

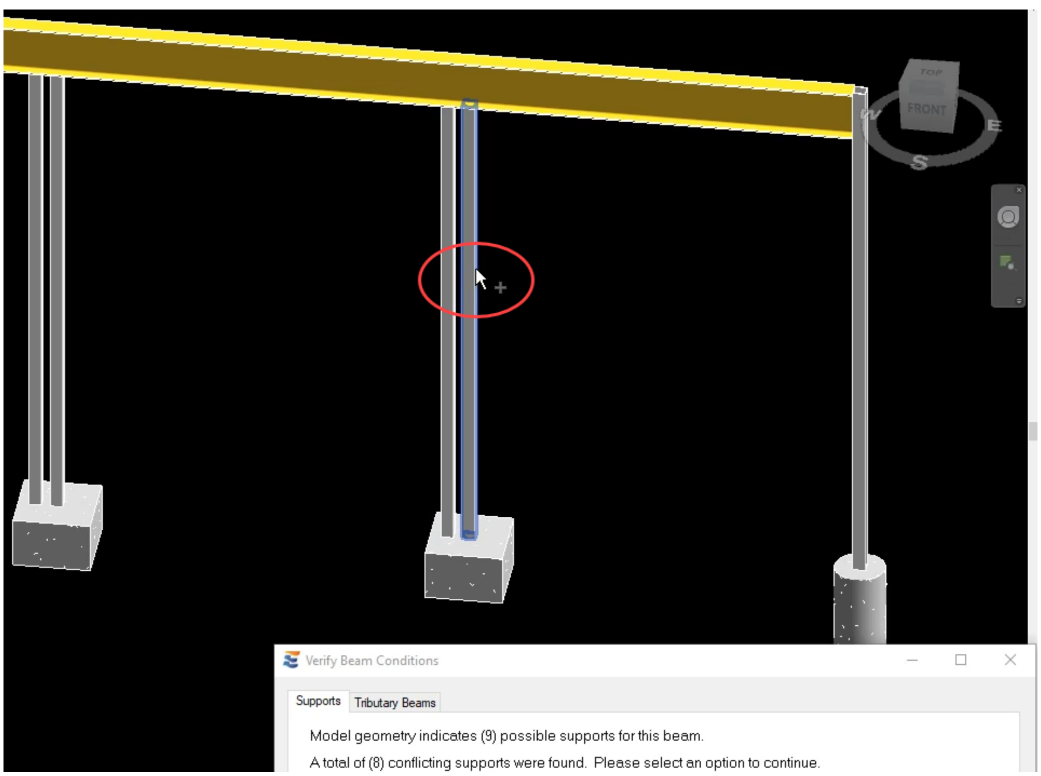
At each location, the conflicting supports will be automatically selected in the Revit view and a Revit multi-select process will begin. The Revit status bar (lower left corner of the main Revit window) indicates the number of conflicts found and gives instructions about how to proceed.



Moving the cursor over an element that is already selected will indicate the "-" option to remove it from the selection.



Moving the cursor over an element that is not selected will indicate the “+” option to add it to the selection.



After manual selections have clarified the desired condition at the current conflict, clicking ”Finish” will cause the program to proceed directly to the next conflict. When all conflicts are resolved, the program will proceed to the next step of the calculation launch process.

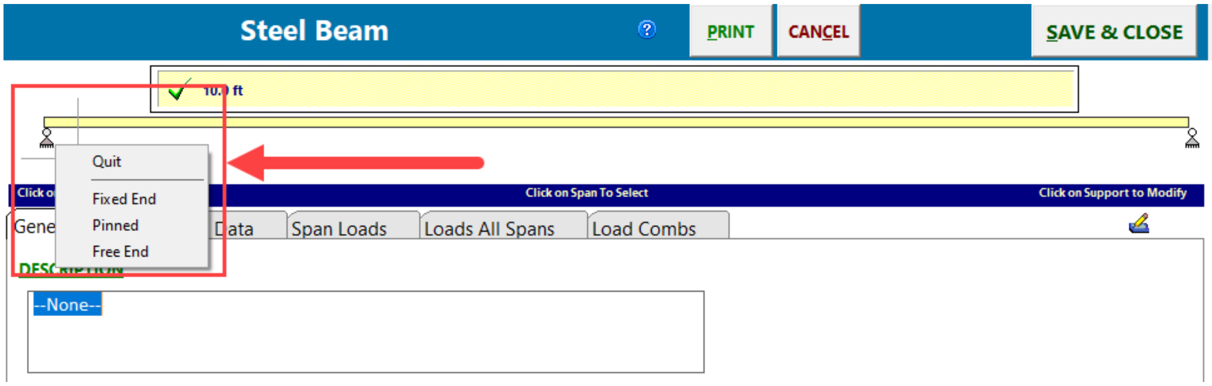
During the process of resolving conflicting supports, clicking “Cancel” on the Revit ribbon at any time to end a multi-select will terminate the calculation launch with a notification.

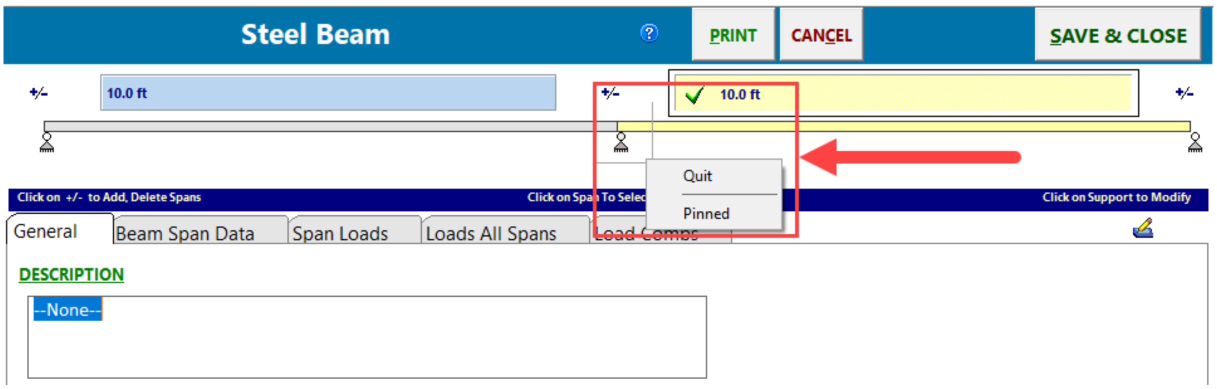
10.1.5 Support Fixity

In conventional steel beam calculations, ENERCALC SEL provides several options for users to set fixity at beam end supports:

- Fixed: translation and rotation restraint, resulting in both force and moment reactions
- Pinned: translation restraint only, resulting in force reactions
- Free: No restraint, resulting in no reactions

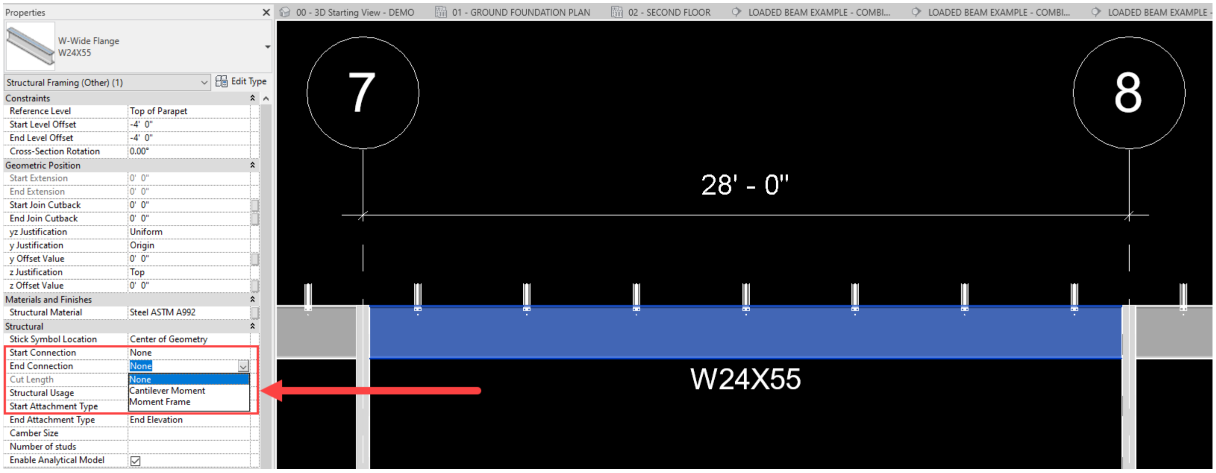
All interior supports are restricted to pinned only.



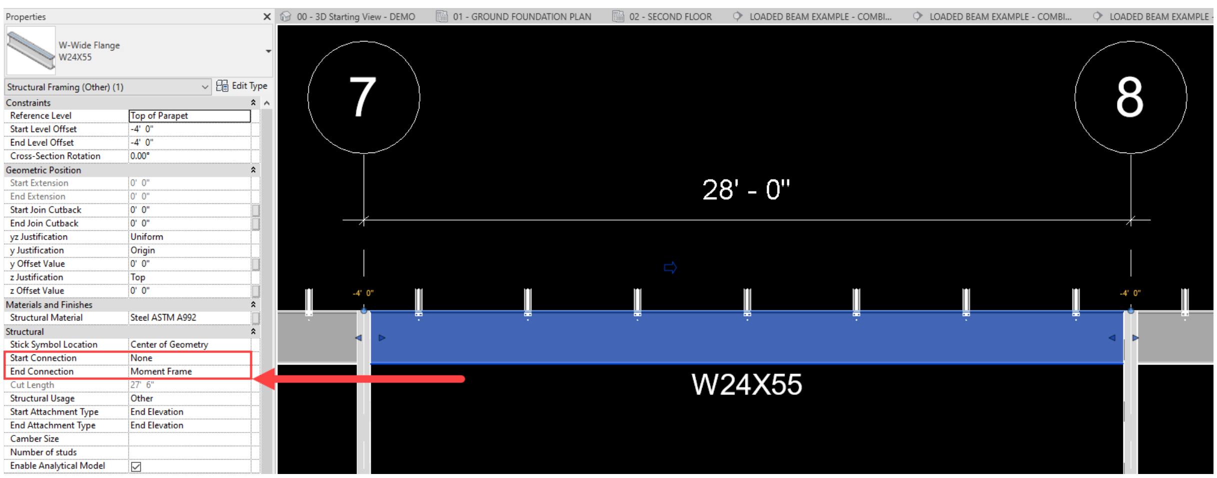


ENERCALC for Revit provides the ability to use these same fixity options when launching a beam calculation from the Revit model. The following rules are used to determine fixity during a beam calculation launch:

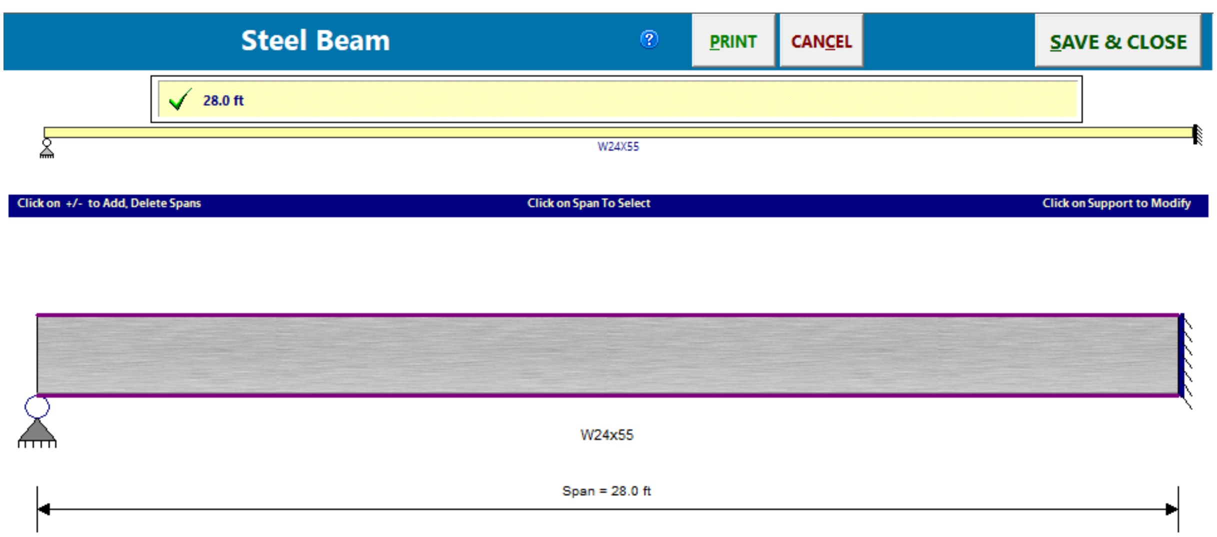
- All interior supports are set as pinned
- Free ends are inferred when a Revit beam element end location is found without a support
- Supported ends are set as pinned or fixed by reading the native Revit “Start Connection” and “End Connection” parameters.
  - “None” = pinned
  - “Moment Frame” = fixed
  - “Cantilever Moment” = fixed



Properties
W-Wide Flange
W24X55
Structural Framing (Other) (1)
Edit Type
Constraints
Reference Level
Top of Parapet
Start Connection
None
End Connection
None
Cantilever Moment
Moment Frame
7
8
28' - 0"
W24X55



Properties
W-Wide Flange
W24X55
Start Connection
None
End Connection
Moment Frame
7
8
28' - 0"
W24X55



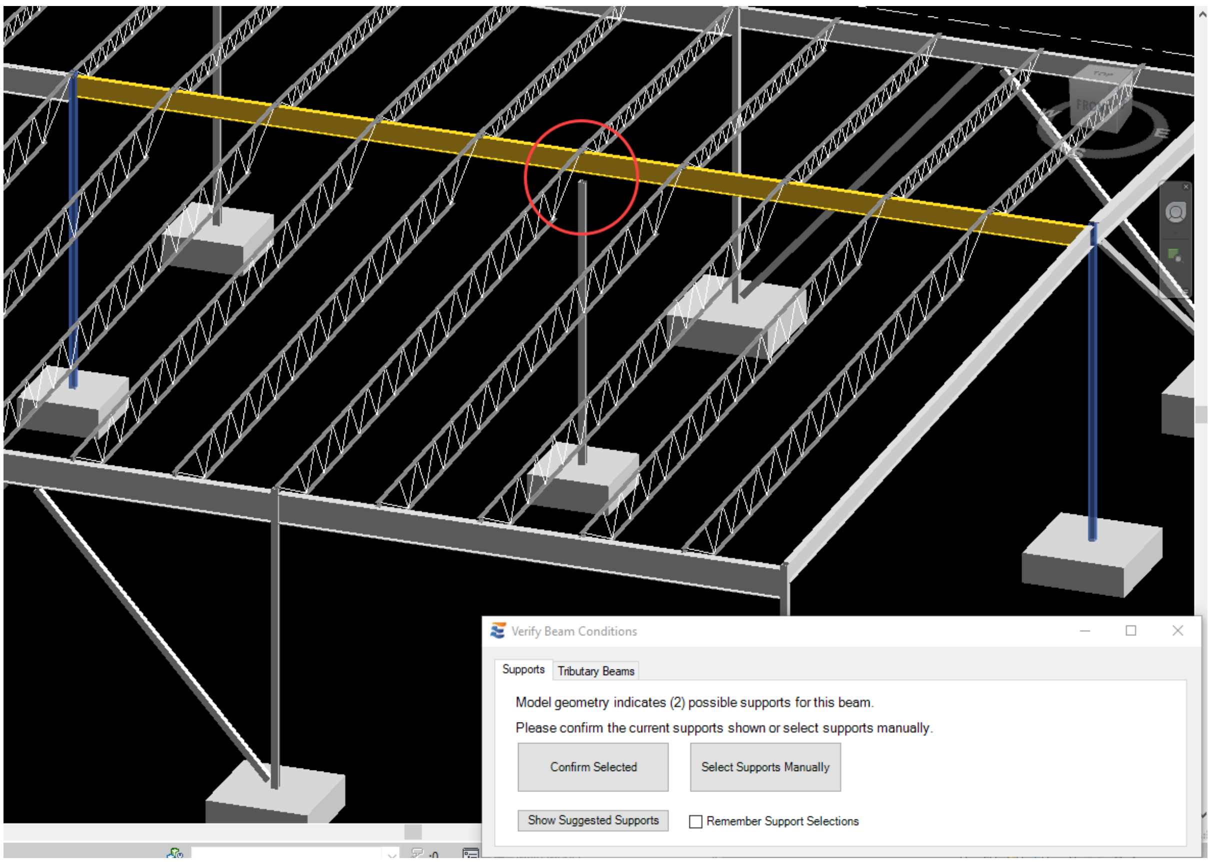
Steel Beam
PRINT
CANCEL
SAVE & CLOSE
28.0 ft
W24X55
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
W24x55
Span = 28.0 ft

10.1.6 Troubleshooting Support Detection Issues

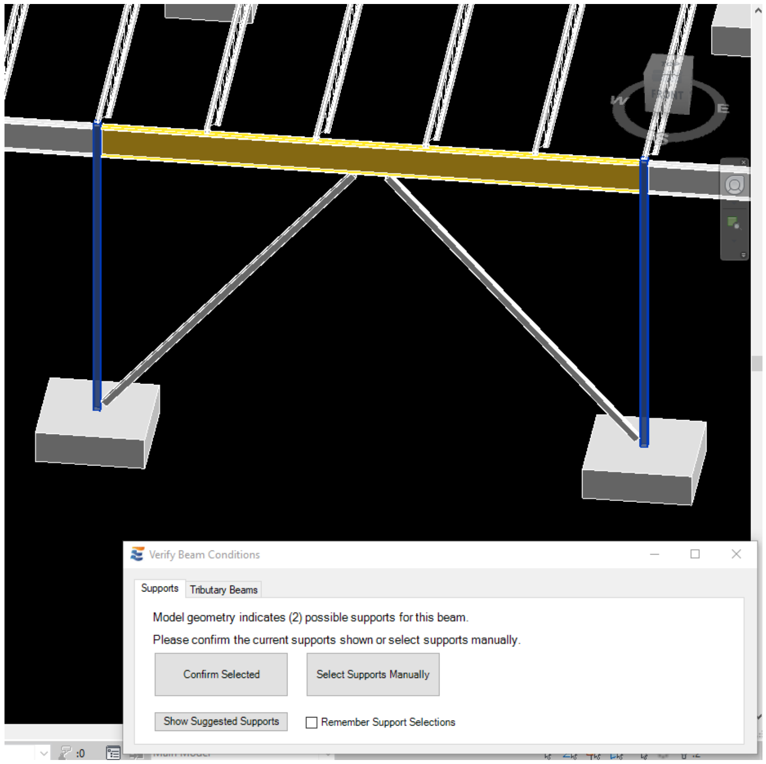
This section provides recommendations for resolving a number of common issues related to support detection.

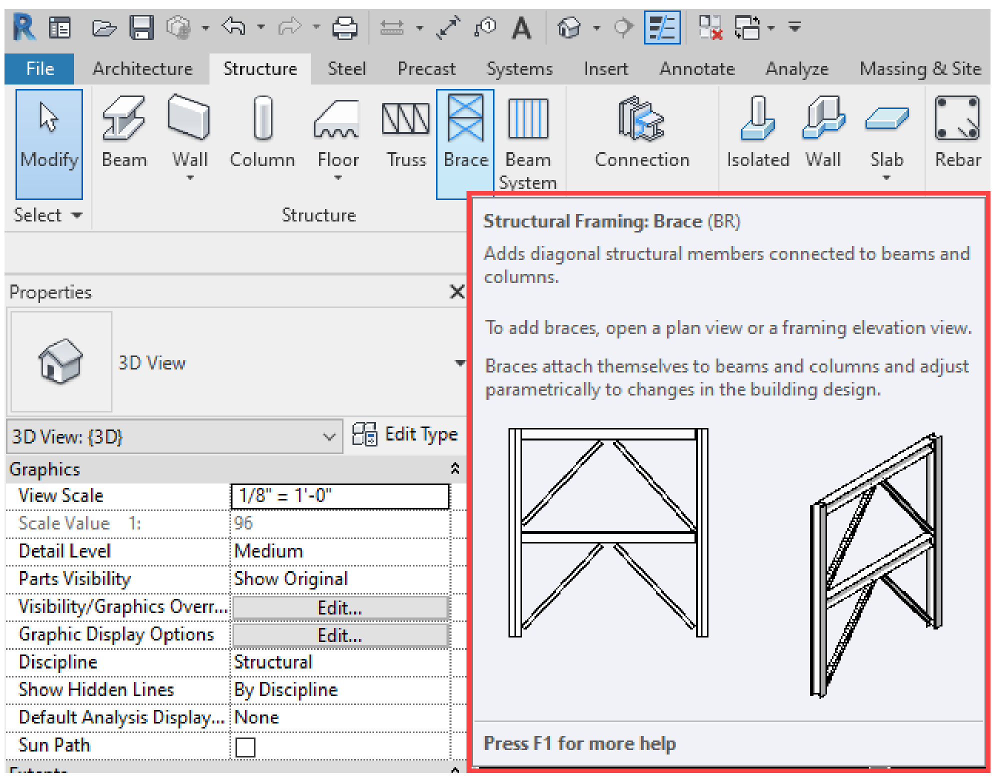
1. Problem: Desired support is not automatically suggested during calculation launch.

    a. Verify that the beam to be designed and the desired supporting element physically intersect with each other.

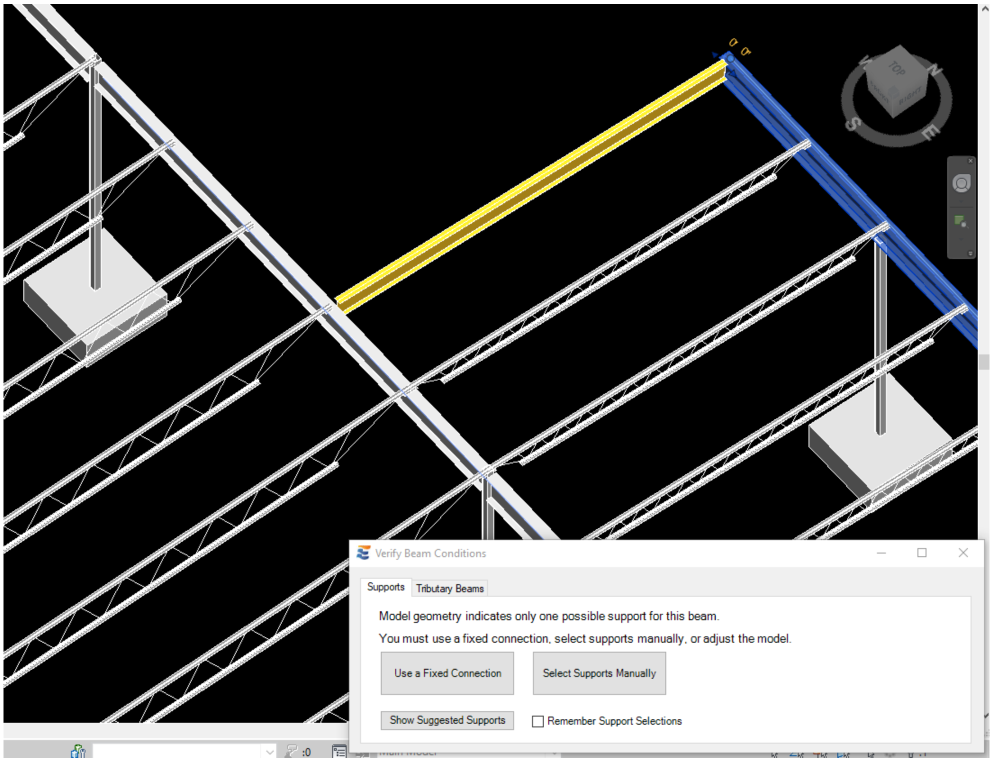


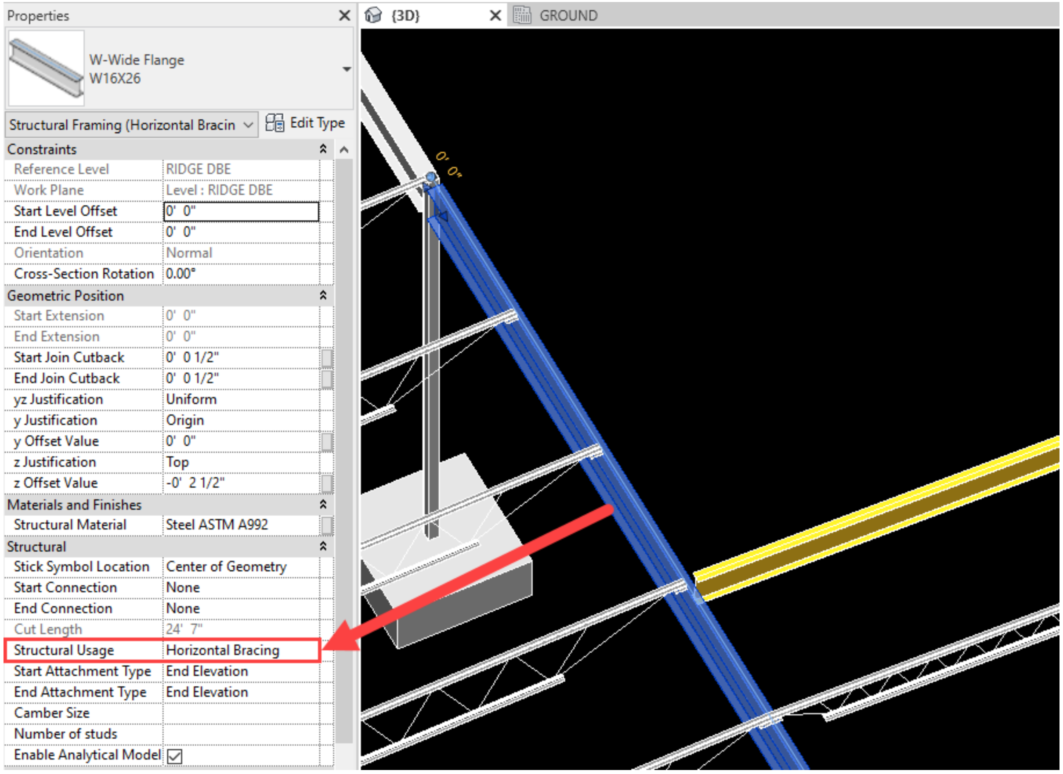
    b. If the desired support is a beam, verify that it has not been modeled as a diagonal brace. Braces are not considered as support candidates.


Verify Beam Conditions
Supports
Tributary Beams
Model geometry indicates (2) possible supports for this beam.
Please confirm the current supports shown or select supports manually.
Confirm Selected
Select Supports Manually
Show Suggested Supports
Remember Support Selections

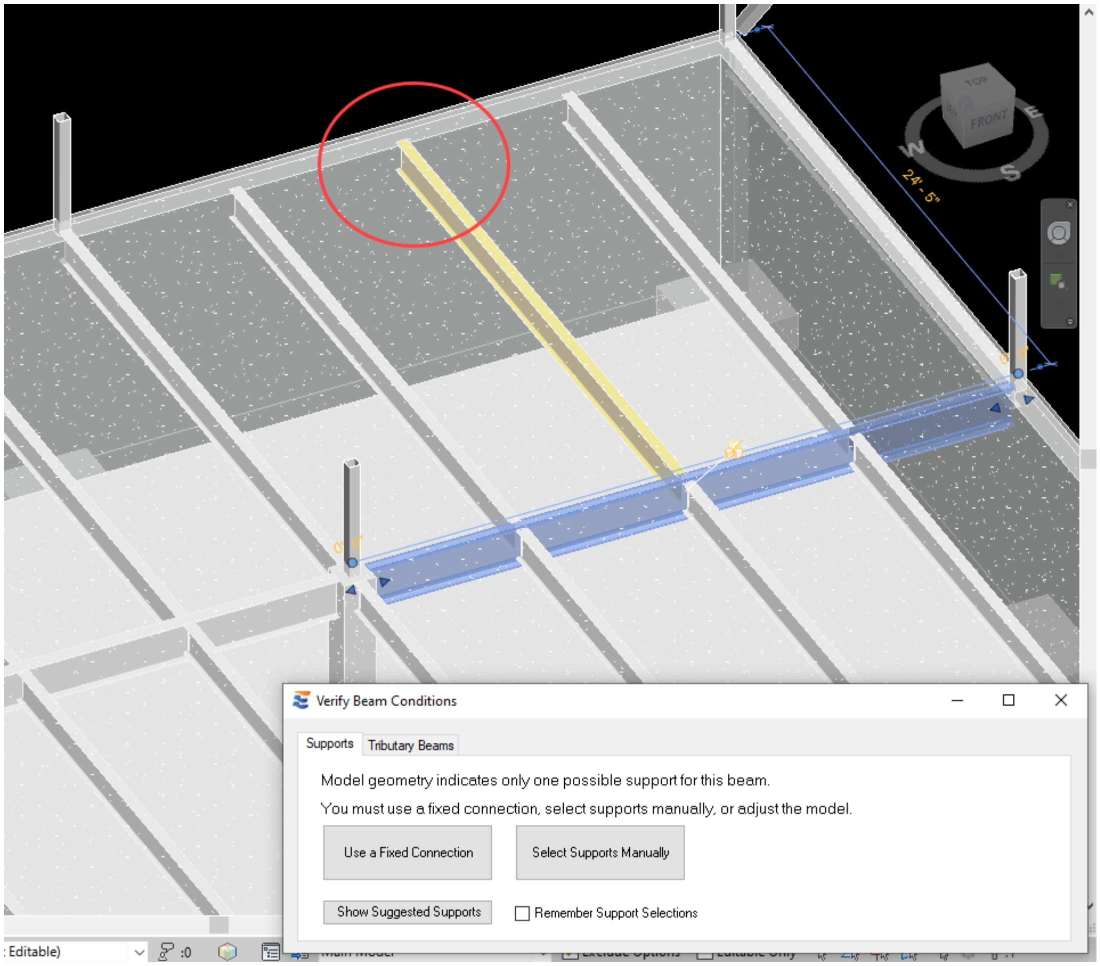

File
Architecture
Structure
Steel
Precast
Systems
Insert
Annotate
Analyze
Massing & Site
Modify
Beam
Wall
Column
Floor
Truss
Brace
Beam System
Connection
Isolated
Wall
Slab
Rebar
Select
Structure
Structural Framing: Brace (BR)
Adds diagonal structural members connected to beams and columns.
To add braces, open a plan view or a framing elevation view.
Braces attach themselves to beams and columns and adjust parametrically to changes in the building design.
Press F1 for more help
Properties
3D View
3D View: {3D}
Edit Type
Graphics
View Scale
1/8" = 1'-0"
Scale Value 1:
96
Detail Level
Medium
Parts Visibility
Show Original
Visibility/Graphics Overr...
Edit...
Graphic Display Options
Edit...
Discipline
Structural
Show Hidden Lines
By Discipline
Default Analysis Display...
None
Sun Path

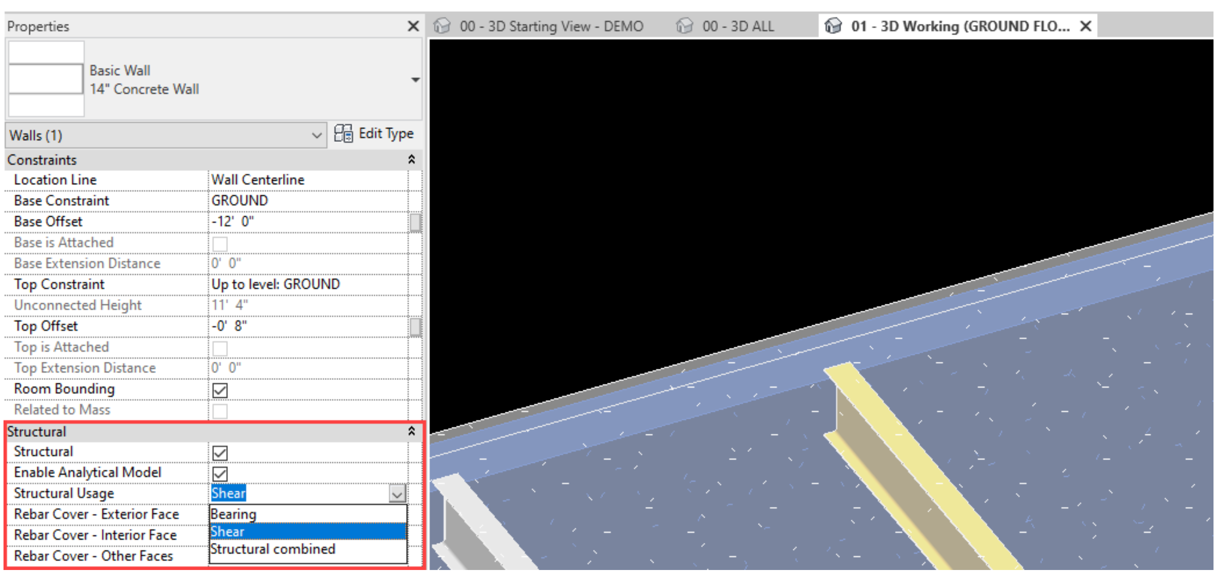
c. If the desired support is a beam, verify that the support's structural usage property is not set to "Horizontal Bracing".





d. If the desired support is a wall, verify that the wall's structural usage property is set to a mode that is capable of bearing behavior.


Verify Beam Conditions
Supports
Tributary Beams
Model geometry indicates only one possible support for this beam.
You must use a fixed connection, select supports manually, or adjust the model.
Use a Fixed Connection
Select Supports Manually
Show Suggested Supports
Remember Support Selections
24' - 5"
TOP
FRONT
W
E
S
Editable)

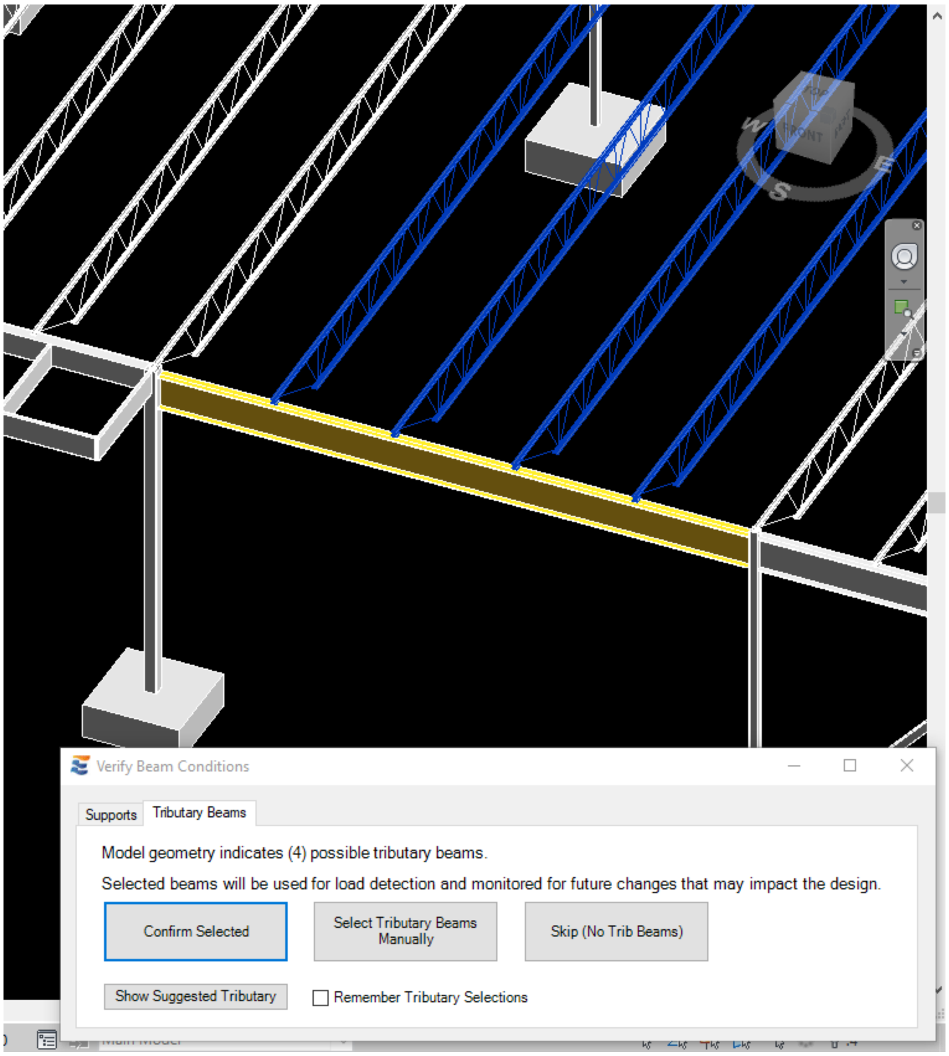

Properties
Basic Wall
14" Concrete Wall
Walls (1)
Edit Type
Constraints
Location Line
Wall Centerline
Base Constraint
GROUND
Base Offset
-12' 0"
Base is Attached
Base Extension Distance
0' 0"
Top Constraint
Up to level: GROUND
Unconnected Height
11' 4"
Top Offset
-0' 8"
Top is Attached
Top Extension Distance
0' 0"
Room Bounding
Related to Mass
Structural
Structural
Enable Analytical Model
Structural Usage
Shear
Rebar Cover - Exterior Face
Rebar Cover - Interior Face
Rebar Cover - Other Faces
Bearing
Shear
Structural combined
00 - 3D Starting View - DEMO
00 - 3D ALL
01 - 3D Working (GROUND FLO...

10.2 Tributary Beams

After the final approval of all supporting elements, the next step in the beam calculation launch process is to confirm the identity of tributary beams. Tributary beams are defined as beams supported by the main designed beam, which may contribute load that influences the design.

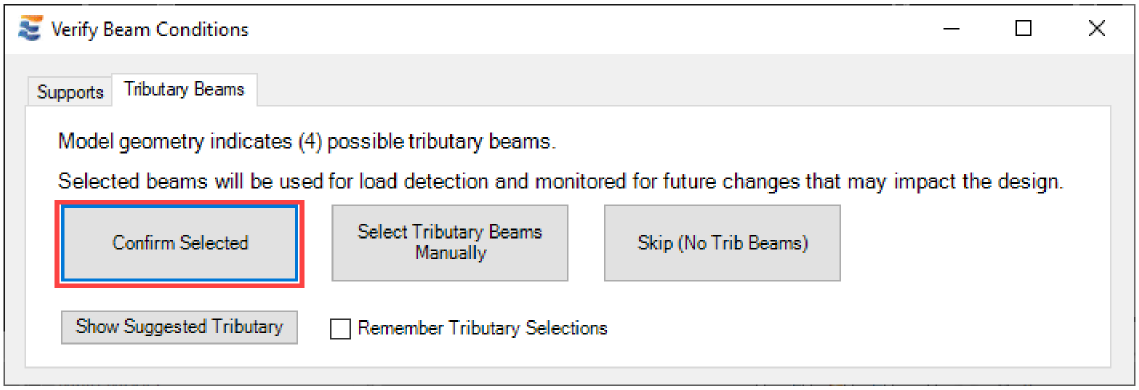
Identifying these beams will establish the connective relationships that determine load path, re-calculation order, and change warnings, as well as allowing ENERCALC for Revit to retrieve load-linked reaction forces from upstream calculations. ENERCALC for Revit leverages a variety of information available in the Revit model, including physical solid location of the beam element, line location of the beam element, and proximity to other elements in order to suggest the most probable tributary elements and present them to the user.

The calculation launch form will proceed automatically to the “Tributary Beams” tab as soon as the support confirmation is complete.



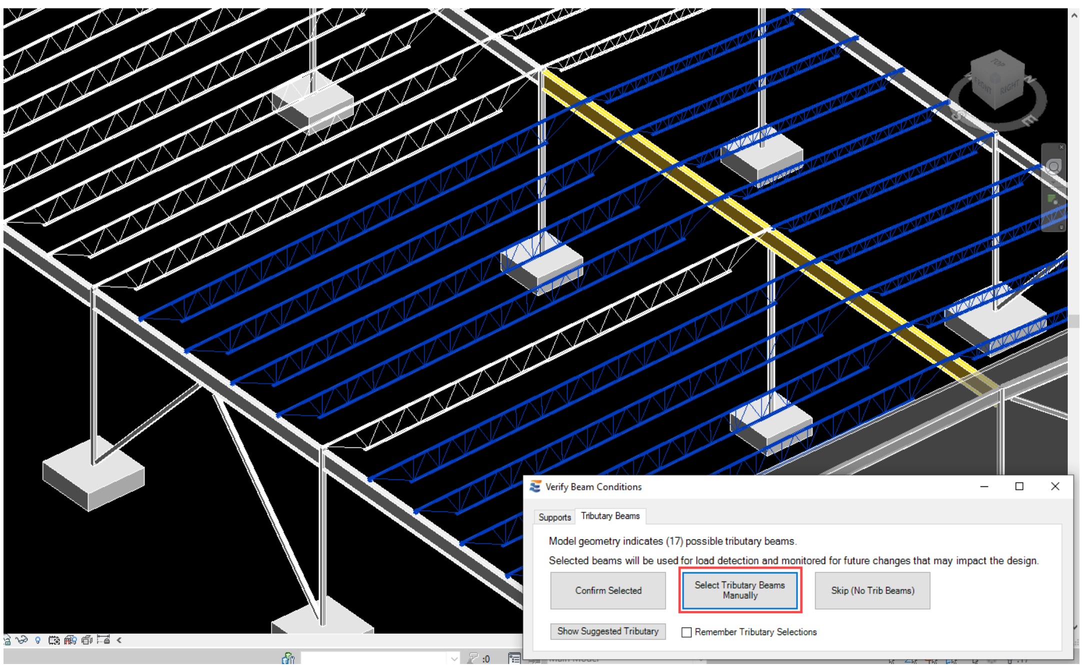
10.2.1 Approving Tributary Beams

If the user is satisfied with the tributary conditions suggested by ENERCALC for Revit, the tributaries may be approved by simply clicking the “Confirm Selected” button. After confirmation, the launch form will automatically proceed to the next tab of the launch process (if applicable). If no further approvals are required, the calculation will launch.

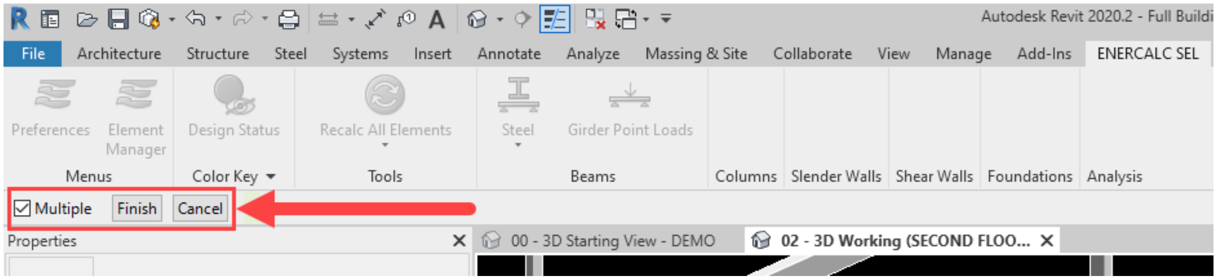


10.2.2 Manually Selecting Tributary Beams

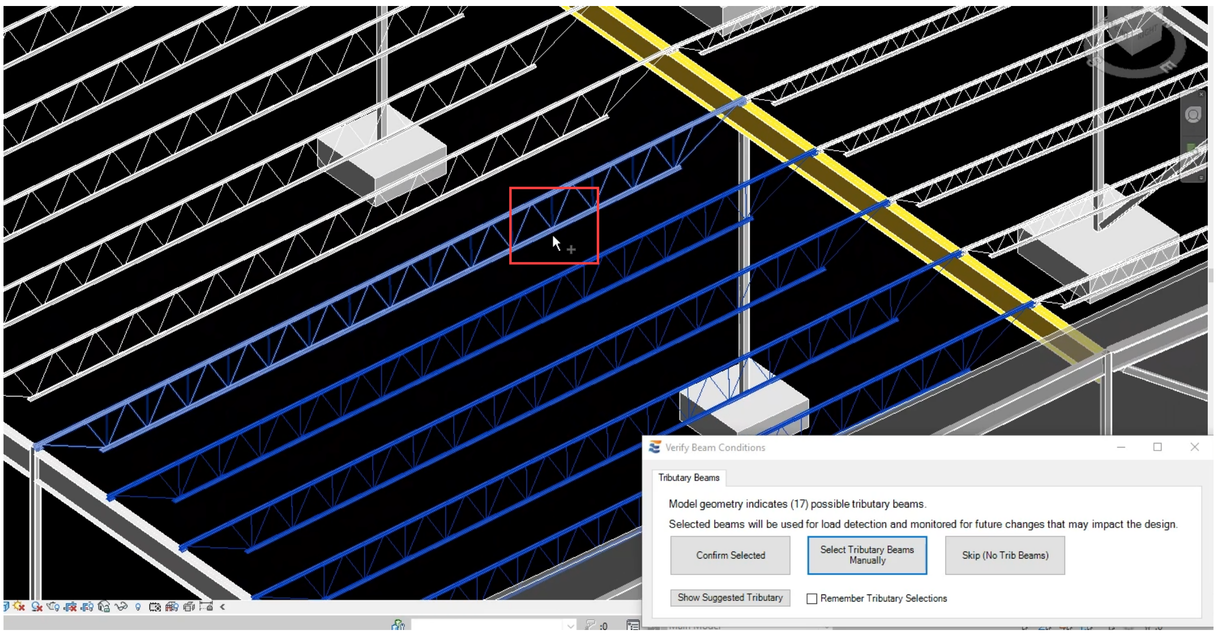
In situations where the suggested tributary beams do not accurately describe the design intent, a user may alternatively choose to manually specify the tributary elements. Custom selection of tributary beams is triggered by clicking the “Select Tributary Beams Manually” button.



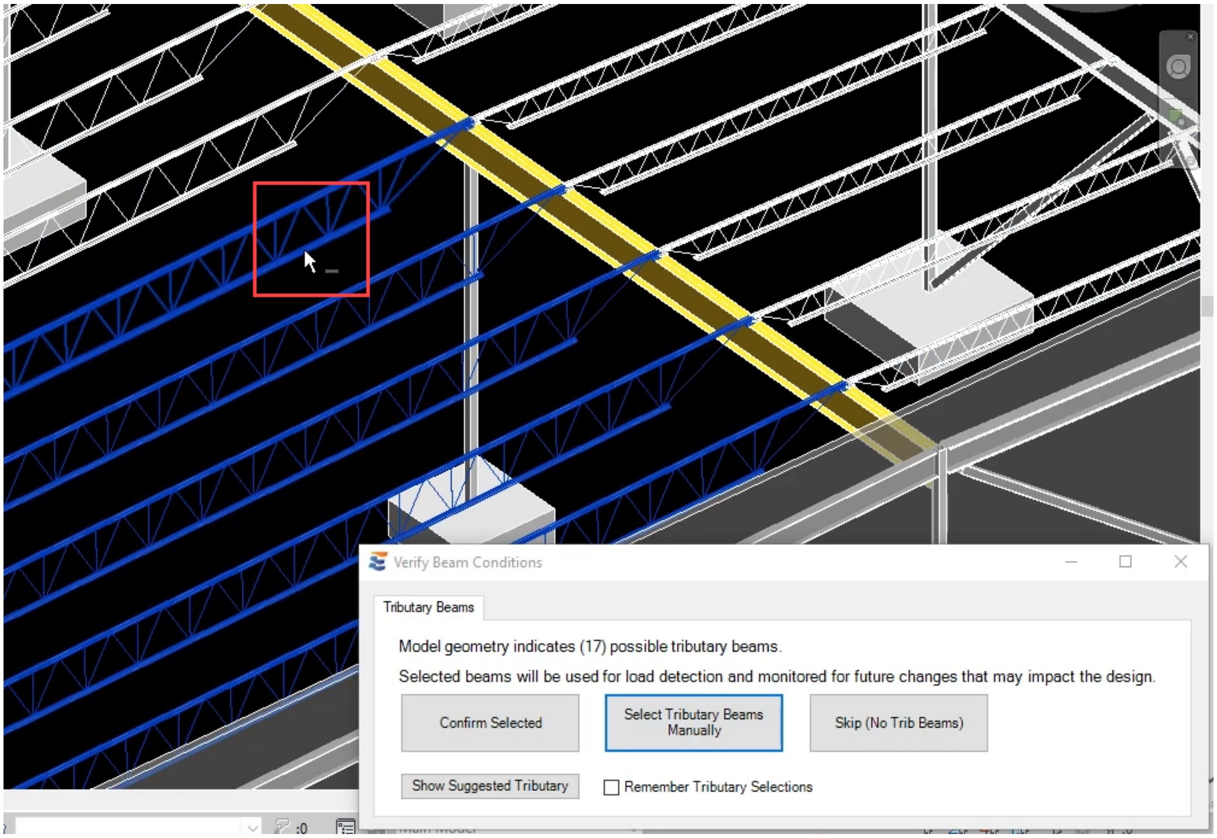
Clicking this button will initiate a Revit multi-select process similar to that used for other native Revit operations. When the multi-select is active, the Revit ribbon bar will display “Finish” and “Cancel” buttons on the upper left-hand corner of the main Revit window and the user will have the ability to manually pick elements.



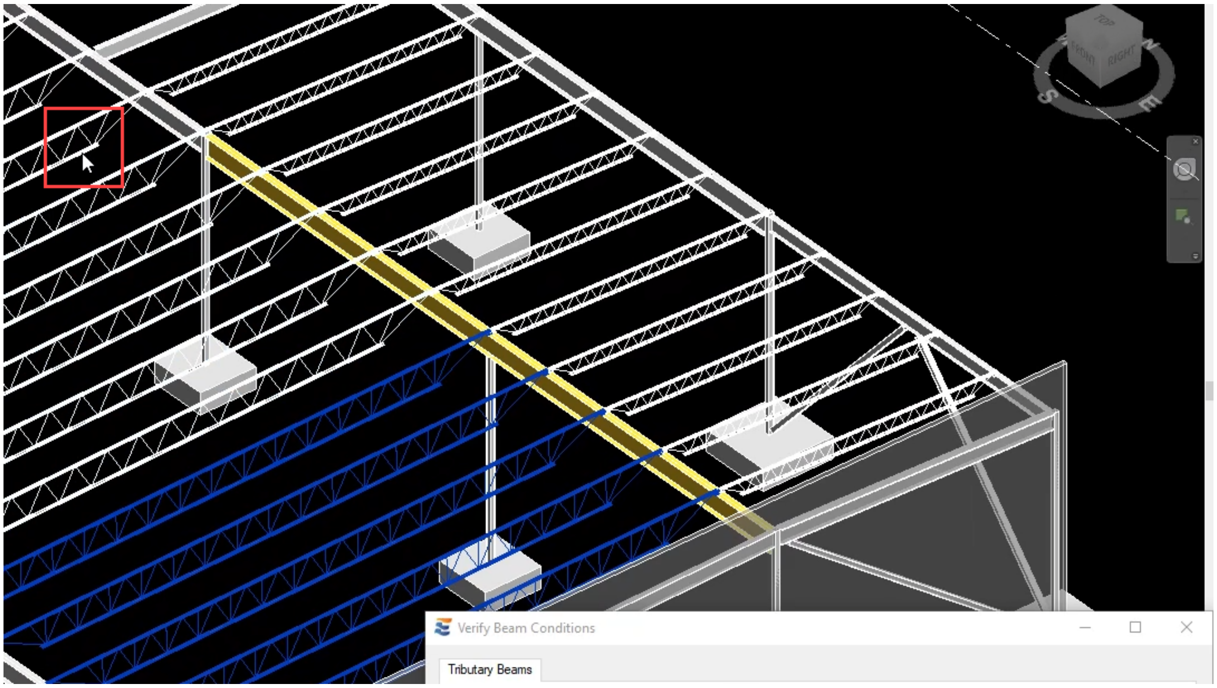
During the selection process, elements already picked will remain selected (and highlighted) in the active view. Items eligible for selection will light up with a "+" symbol when the cursor passes over.



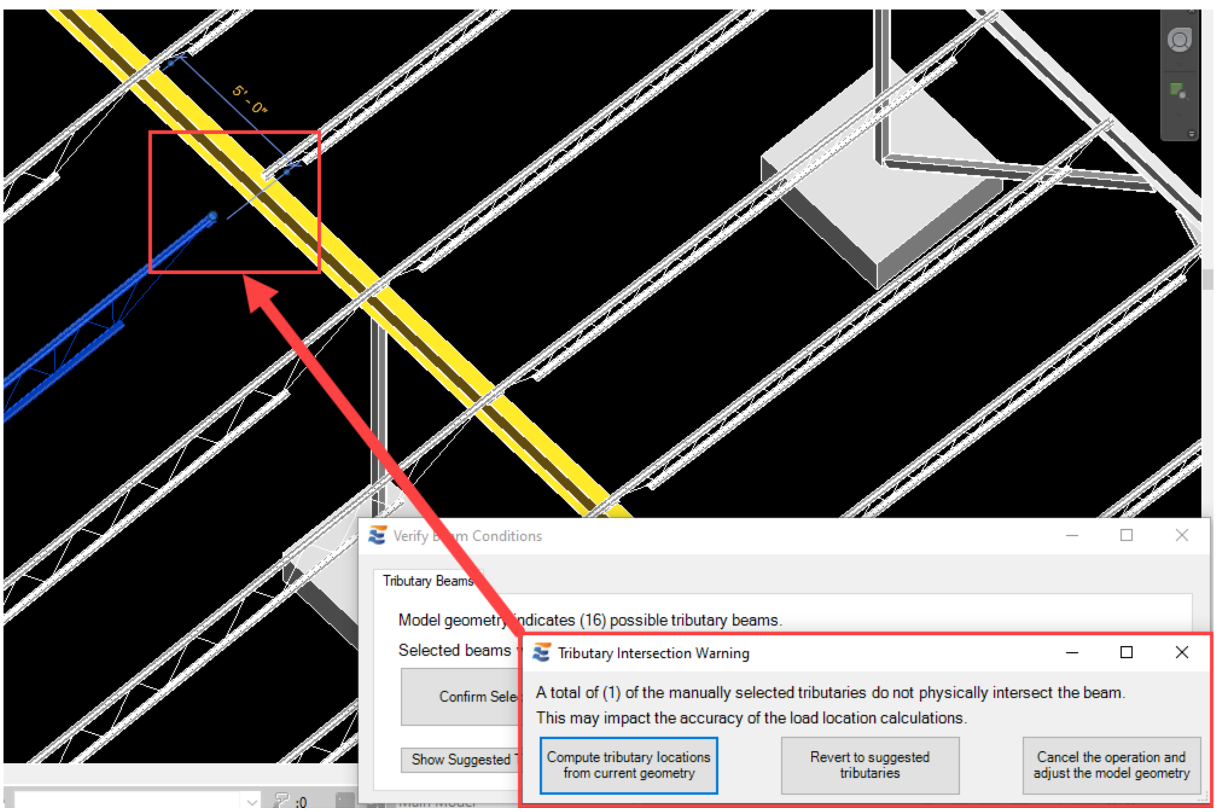
Similarly, an already-selected element eligible for deselection will light up with a "-" symbol when the cursor passes over.



The tributary beam selection process is also subject to several restrictions. In order to limit downstream errors from difficulty in defining linked reaction locations, the user is restricted to only selecting elements that are in actual proximity to the beam. Elements outside the immediate vicinity of the beam will not be clickable. The user is also restricted to only selecting element types that are eligible to act as tributary elements. When the user points to an element that is not eligible to act as a tributary for the current beam, the element will not highlight and the cursor will not show the “+” symbol.



In some cases, the user may choose to manually select a tributary beam that is near enough to the designed beam to be eligible for selection but does not physically intersect. When this happens, the user will be presented with a warning and three options to resolve the issue.

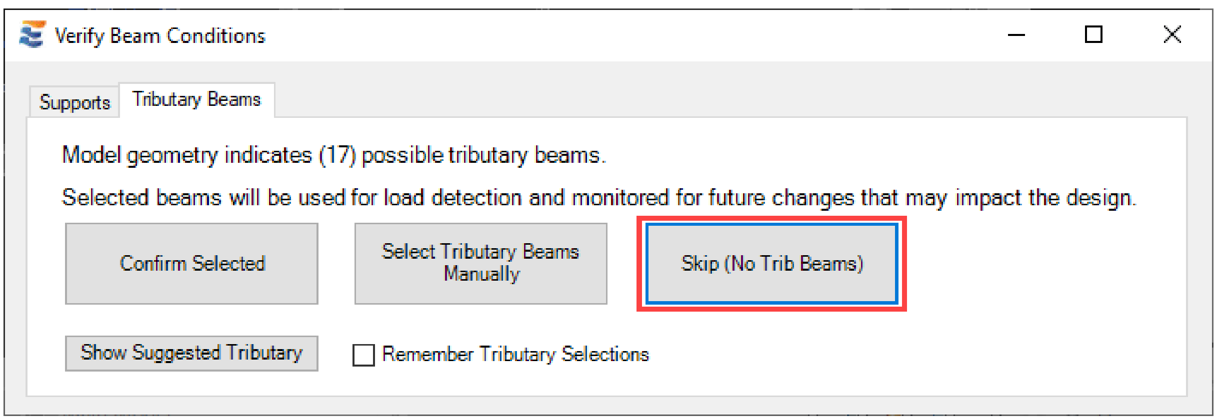


Clicking the left-most button will cause the program to proceed with the launch process using the specified tributary. Applicable linked load locations will be computed from the nearest extrapolated intersection of the two elements. For some geometry cases (such as the one shown above), this option is appropriate and will result in accurate beam analysis. In some cases, however, the lack of physical intersection could indicate the lack of any meaningful relationship by which load linking may be established. If choosing this option results in a downstream warning when span geometry cannot be calculated (or if the calculation in ENERCALC SEL shows inaccurate geometry) then it is advisable to adjust the Revit model for a more accurate tributary condition.

Clicking the middle button will cause the program to proceed with the suggested tributaries rather than the manually specified ones. Suggested tributaries will typically be in direct contact with the beam to be designed, meaning that extrapolation of nearest intersection points is not required.

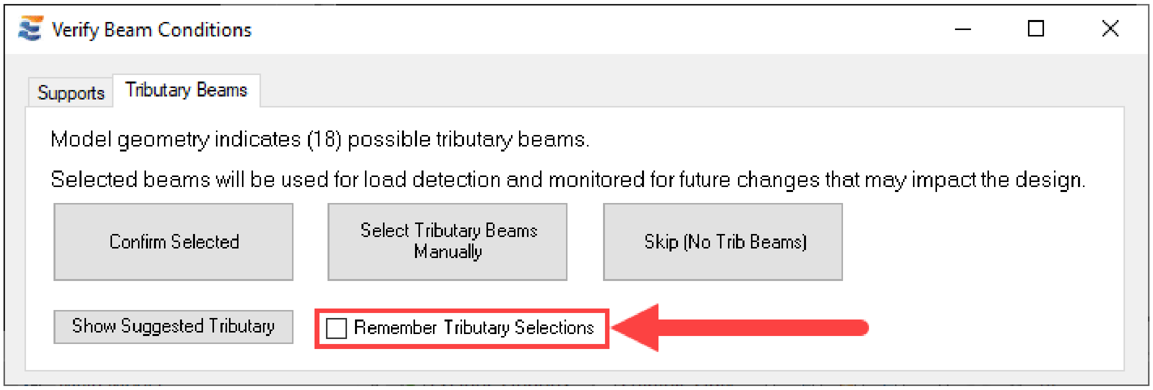
Clicking the right-most button with cause the launch approval form to close and the yellow highlighting on the beam will be removed. In the case of an aborted launch, no calculation will be created in the linked .EC6 file.

If at any point during the multi-select process the user clicks the “Cancel” button on the Revit ribbon bar, the calculation launch process will proceed. Cancelling out of a multi-select operation has the same effect as using the “Skip (No Trib Beams)” button. In either case, the calculation loaded into SEL will not include force effects associated with any tributary beams.



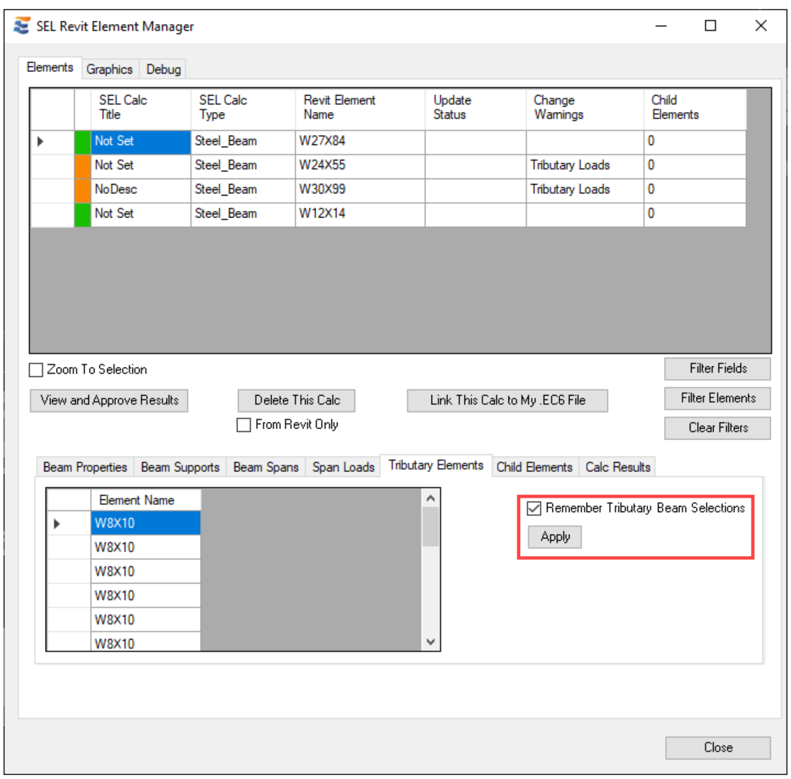
10.2.3 Remembering Tributary Conditions

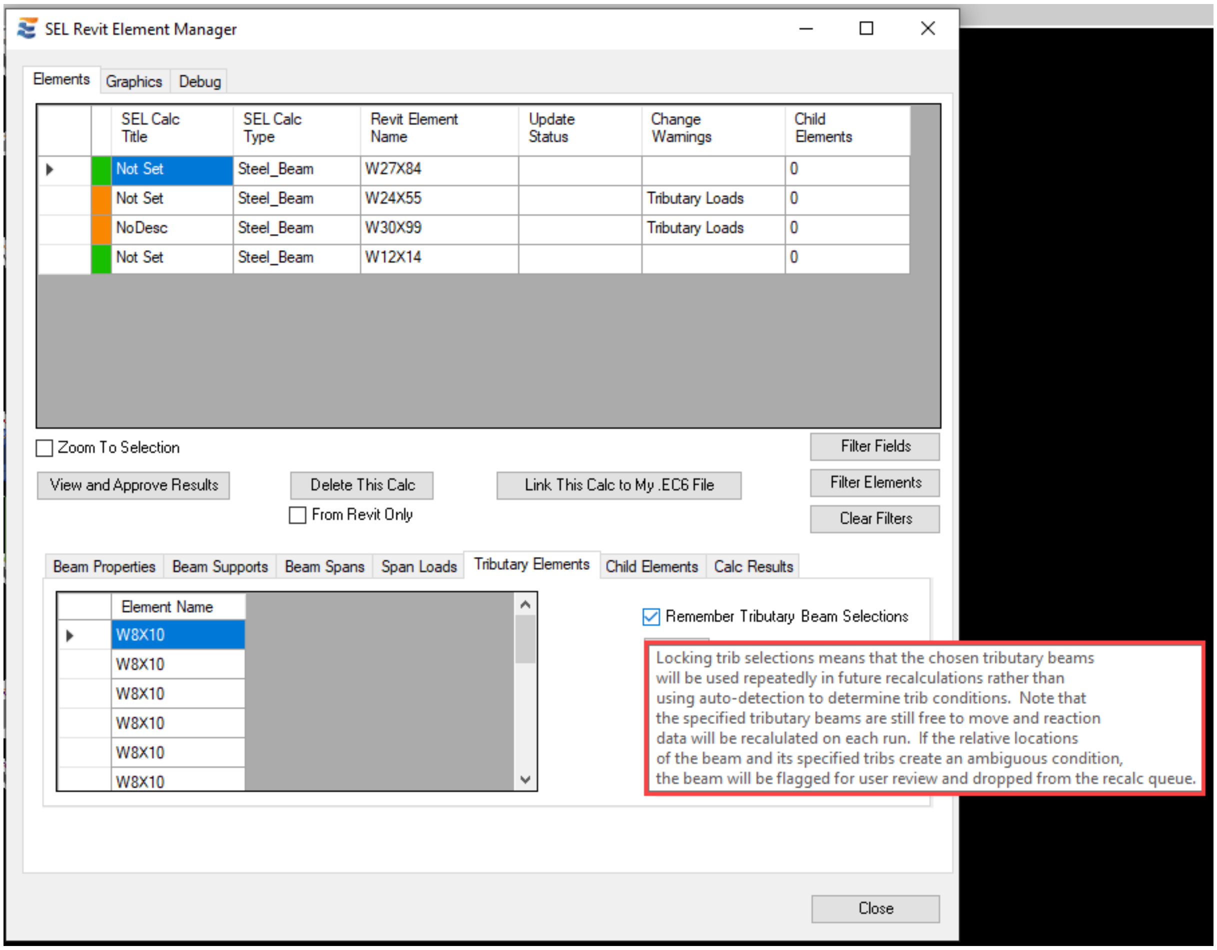
Regardless of the method used to choose the beam tributaries, the launch window includes an option to “Remember Support Selections”.



Checking this box prior to the approval of tributary conditions will allow ENERCALC for Revit to store the identities of the approved tributaries during future launches of the calculation. This will permit rapid updating of the beam's load linking conditions without the inconvenience of repetitive approvals. If tributaries have been stored, the launch window will not show a "Tributary Beams" tab for user approval.

If at any point it becomes necessary to choose new tributary conditions for a beam, the previously stored tributaries may be unlocked via the Element Manager. Conversely, if the tributaries were approved but not "Remembered" during the previous launch, the "Remember" option may be enabled from this same menu.





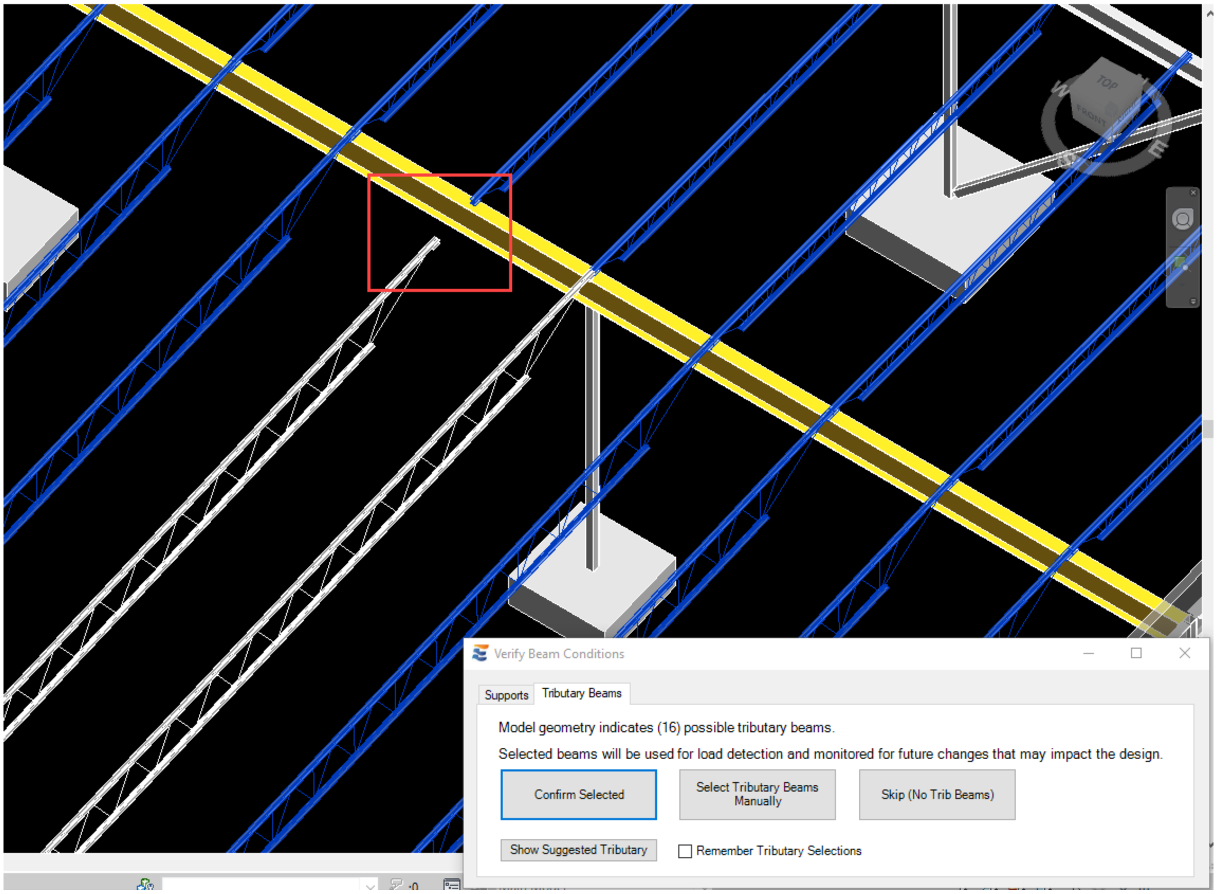
Note that even when tributary beams have been stored, the launch still includes error checking to validate the previously stored list. If a previously stored tributary no longer exists in the Revit model, then the launch will proceed without it. If a previously stored tributary does not physically intersect the beam, the launch process will revert to prompting for approval of tributary conditions.

10.2.4 Troubleshooting Tributary Detection Issues

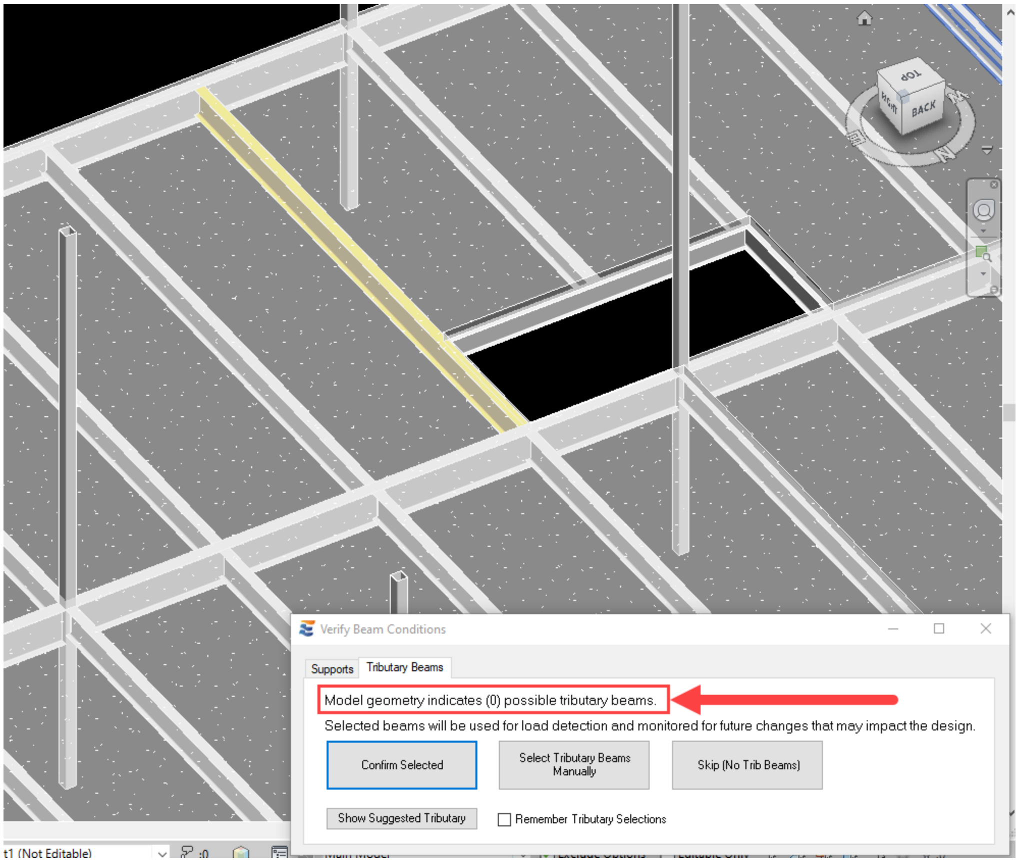
This section provides recommendations for resolving a number of common issues related to tributary beam detection.

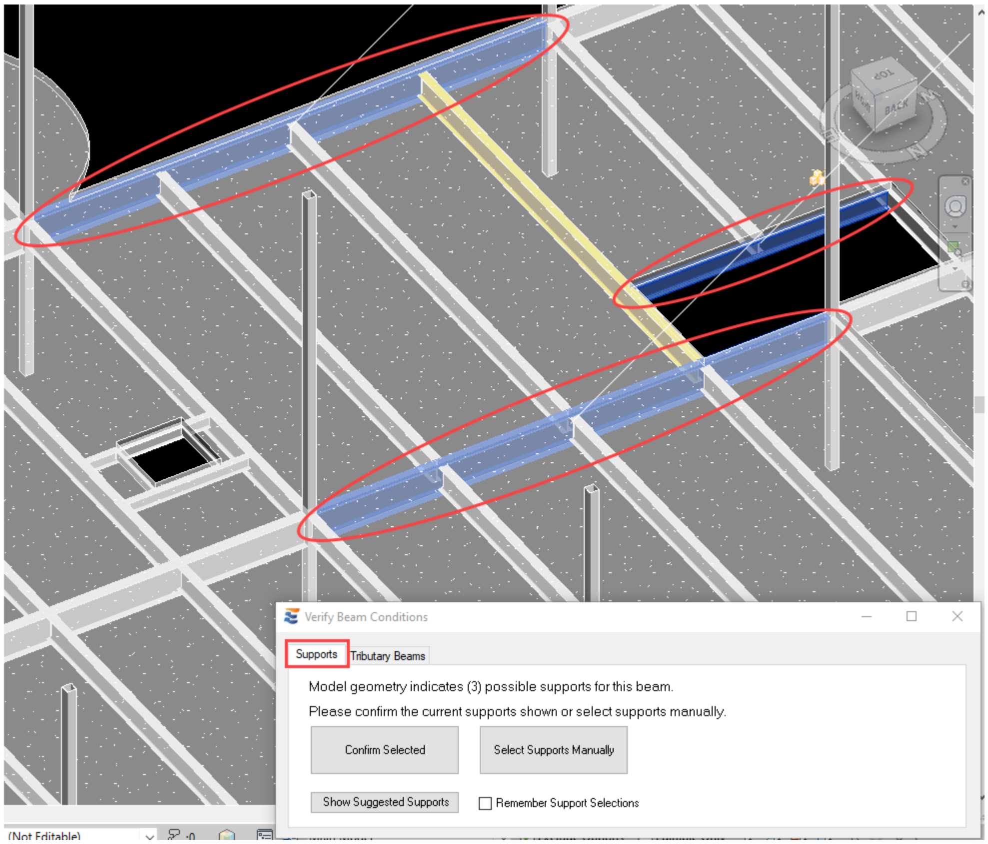
1. Problem: Desired tributary beam is not automatically suggested during calculation launch.

    a. Verify that the beam to be designed and the desired tributary element physically intersect with each other.



b. Verify whether the desired tributary element was incidentally specified as a support during the previous approval step. Any element specified to act as a support for a beam is automatically disqualified from acting as a tributary element for that same beam. You can easily review supports specified in the previous step by clicking the “Supports” tab on the approval form. This will automatically highlight the approved supports by selecting them in the Revit view.

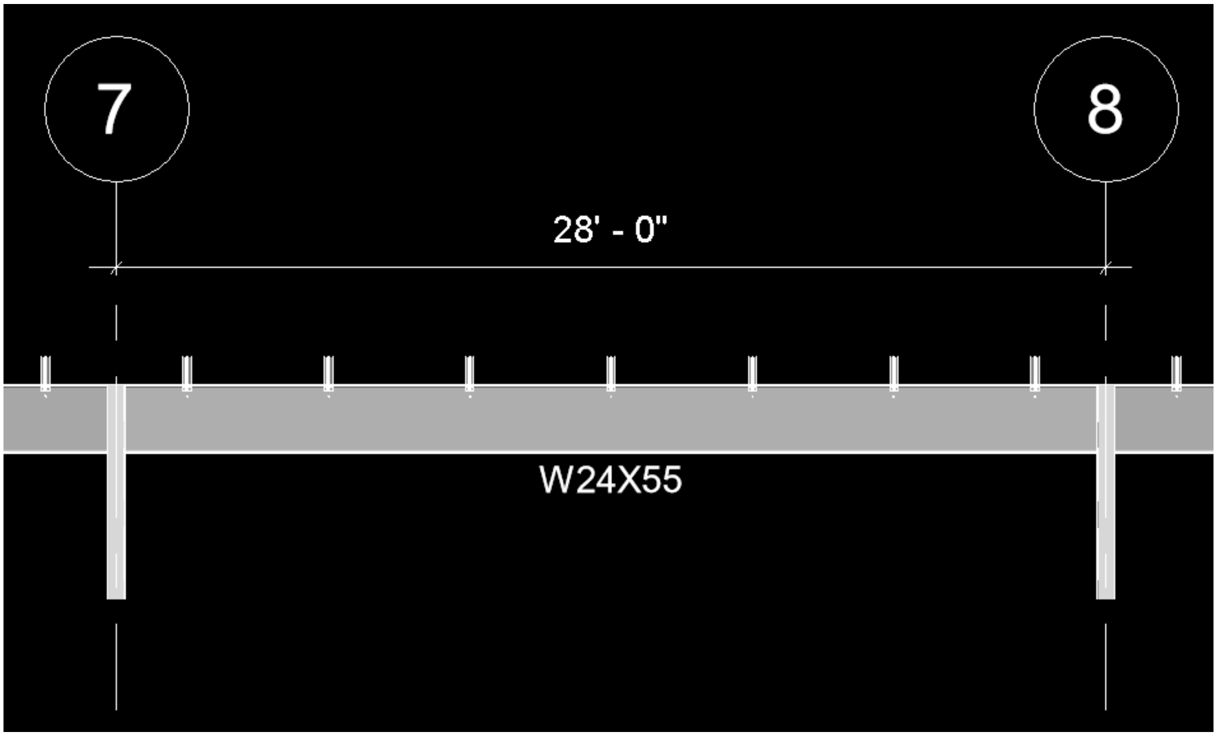

Verify Beam Conditions
Supports
Tributary Beams
Model geometry indicates (0) possible tributary beams.
Selected beams will be used for load detection and monitored for future changes that may impact the design.
Confirm Selected
Select Tributary Beams Manually
Skip (No Trib Beams)
Show Suggested Tributary
Remember Tributary Selections

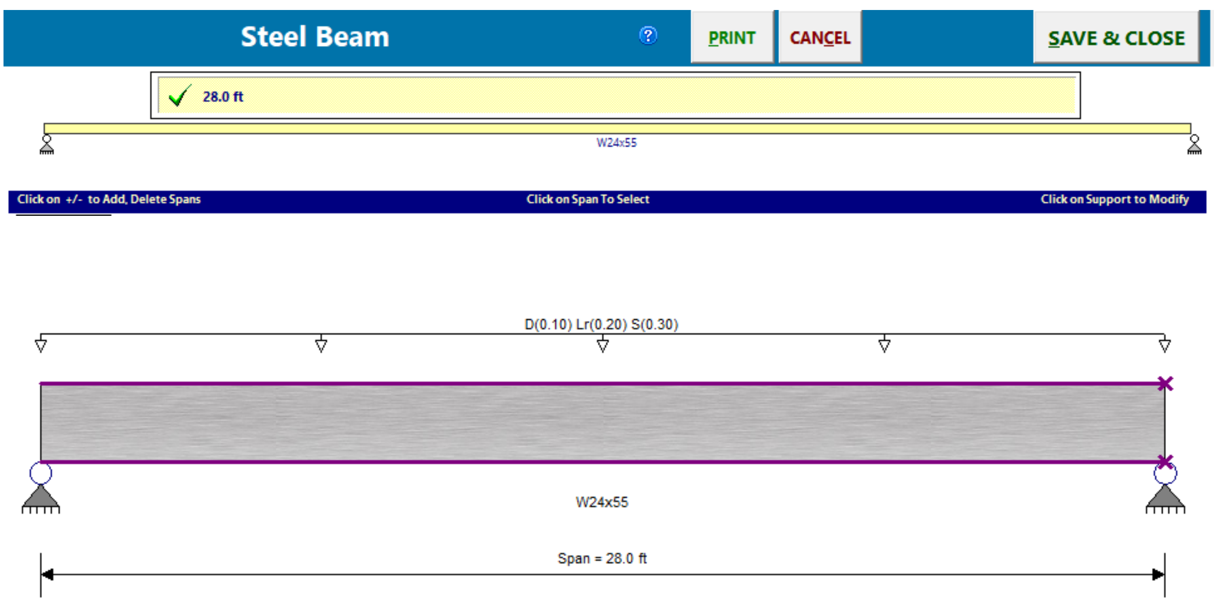


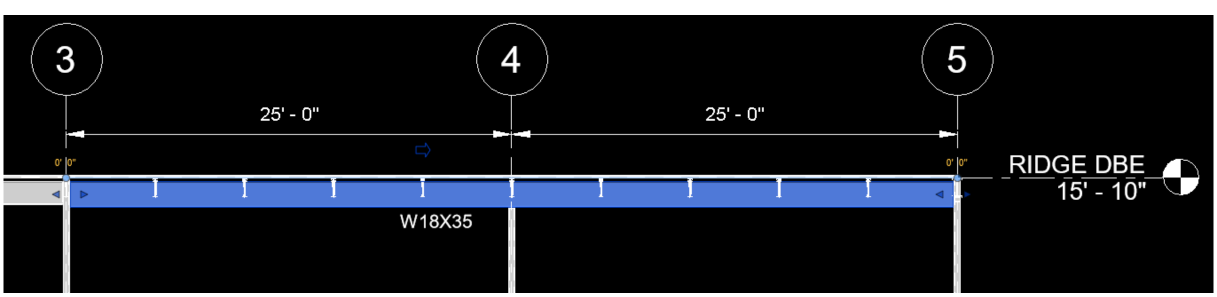
10.3 Beam Span Geometry

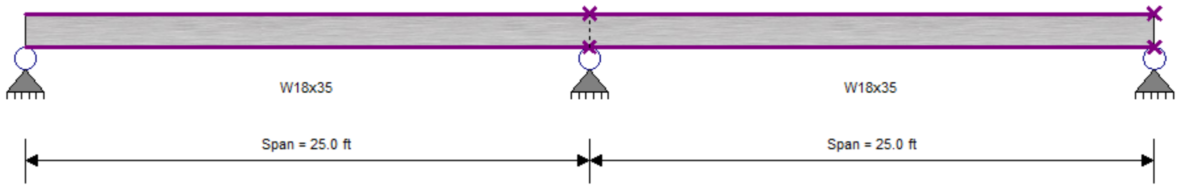
After the necessary approval steps have been completed, ENERCALC for Revit will automatically compute the span geometry of the beam without further input from the user. Once supporting locations are established via user approval the resulting geometry created in the ENERCALC SEL calculation should precisely match the framing layout found in Revit. This includes not only simple-span and multi-span beams, but also beams with free ends that create cantilever spans.

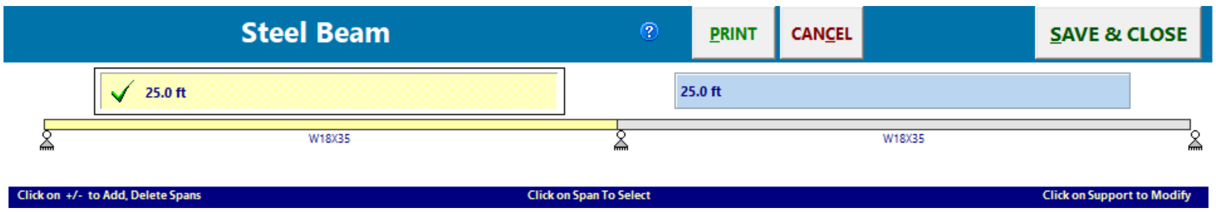
As illustrated in the examples below, the span geometry is established using the centerline location of the supporting element. There are no automatic reductions or adjustments to the span based on width of supports or assumed rigid offsets.

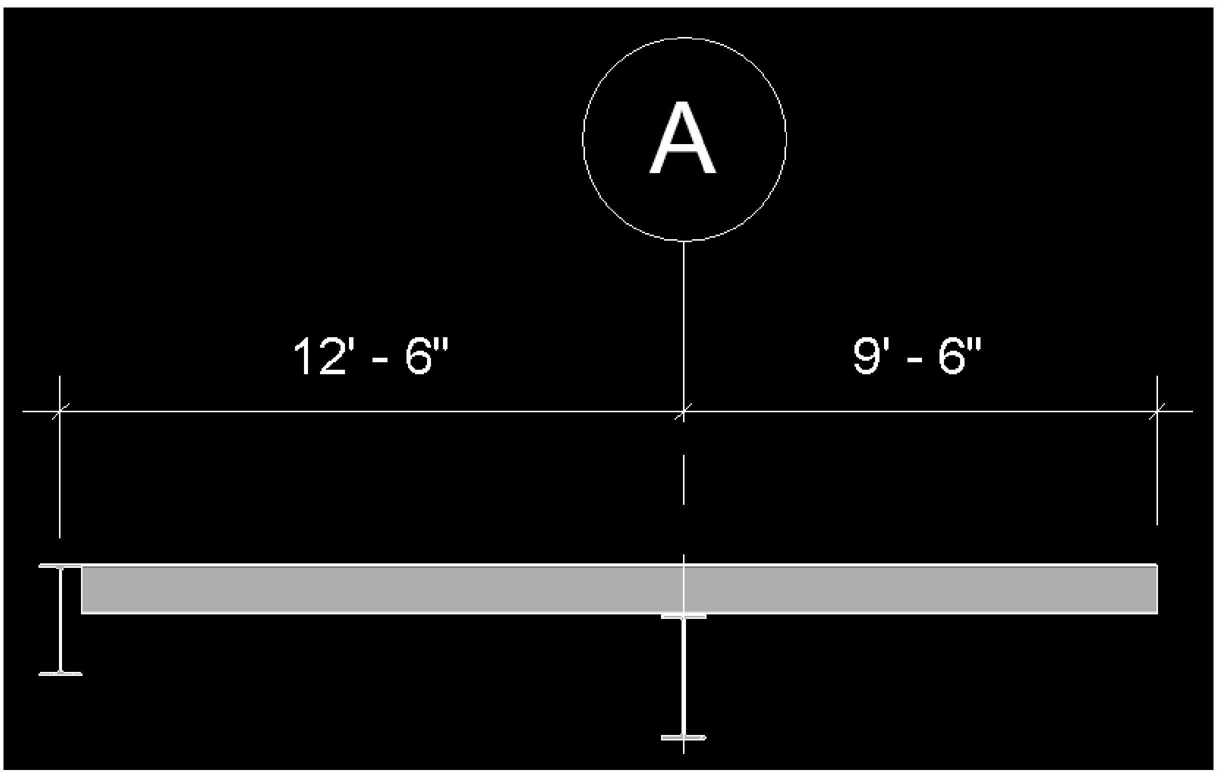

7
8
28' - 0"
W24X55

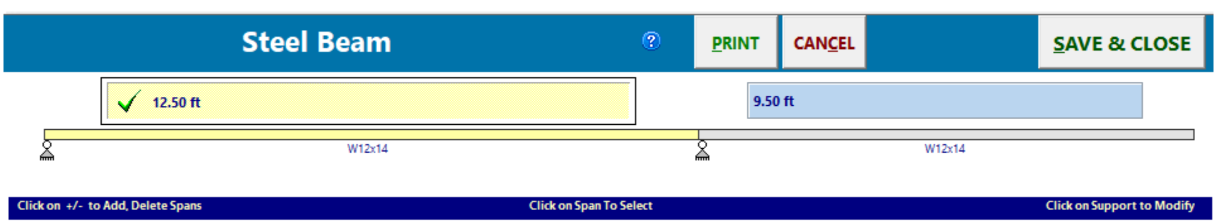

Steel Beam
PRINT
CANCEL
SAVE & CLOSE
28.0 ft
W24x55
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
D(0.10) Lr(0.20) S(0.30)
W24x55
Span = 28.0 ft

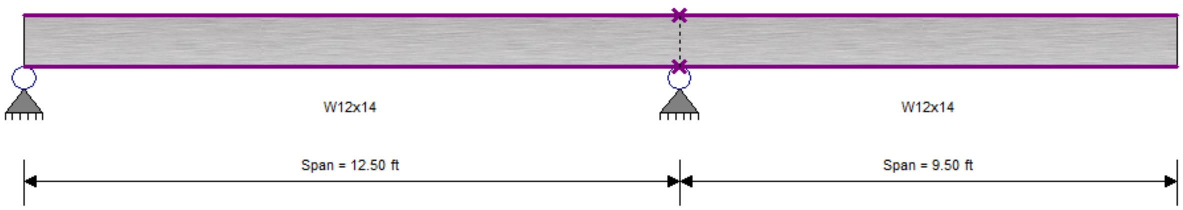

3
4
5
25' - 0"
25' - 0"
W18X35
RIDGE DBE
15' - 10"


W18x35
W18x35
Span = 25.0 ft
Span = 25.0 ft


Steel Beam
PRINT
CANCEL
SAVE & CLOSE
25.0 ft
25.0 ft
W18X35
W18X35
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify


A
12' - 6"
9' - 6"


Steel Beam
PRINT
CANCEL
SAVE & CLOSE
12.50 ft
9.50 ft
W12x14
W12x14
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify



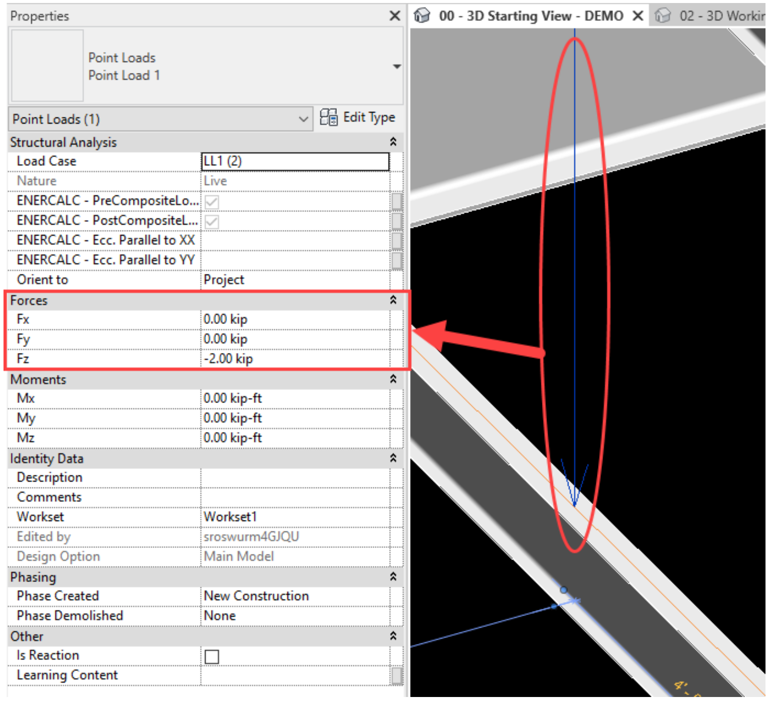
10.4 Beam Loads

Rather than being subject to review and manual approval by the user, all loads on a beam are detected automatically and incorporated automatically into the resulting SEL calculation. Beam calculations consider the presence of both hosted and non-hosted loads in the Revit model. This includes all of the following:

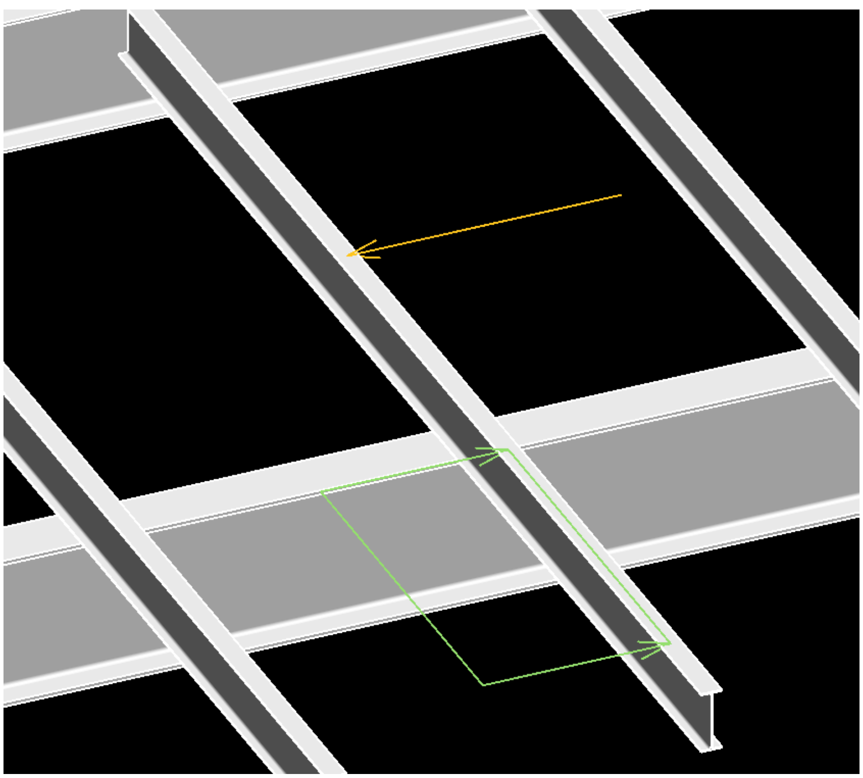
- Point loads (force or moment):
  - Hosted
  - Non-Hosted
- Line Loads:
  - Hosted
  - Non-hosted
- Area Loads:
  - Hosted

10.4.1 Load Directionality and Components

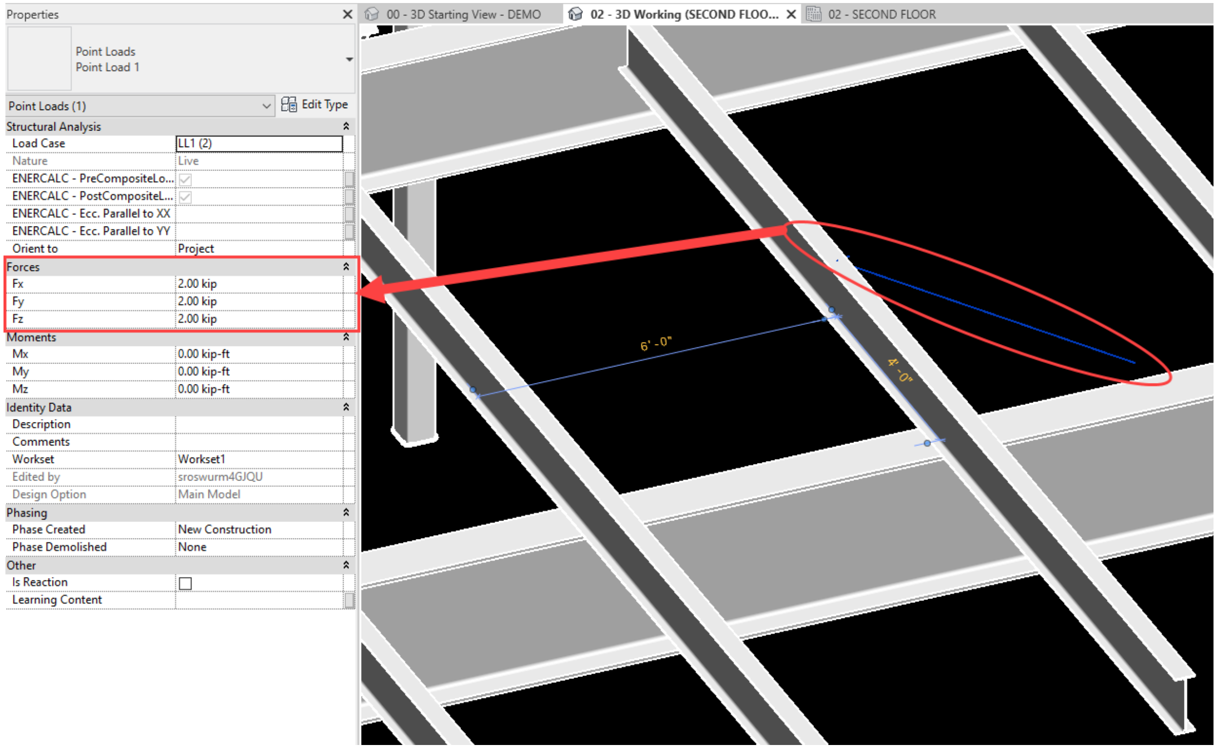
The general sign convention used in Revit is opposite to that used in ENERCALC SEL. In Revit, loads with a global downward direction use a negative magnitude. ENERCALC for Revit converts these signs automatically between conventions during calculation launches to prevent any additional management burden on the user. Users should use the normal directional convention appropriate to each individual program.

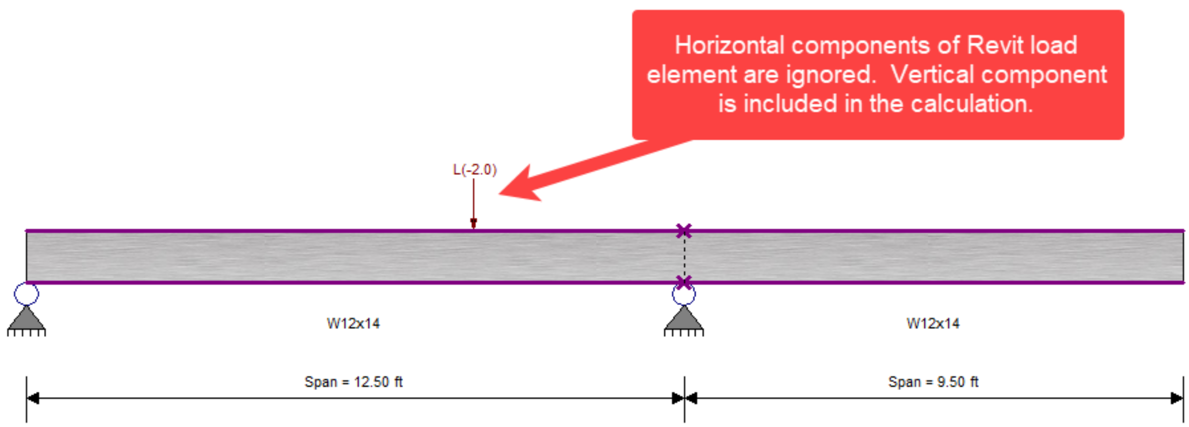


In conventional beam calculations that do not consider torsion or biaxial bending, loads that do not have a vertical component (Global Z in Revit) will be ignored during calculation launch.

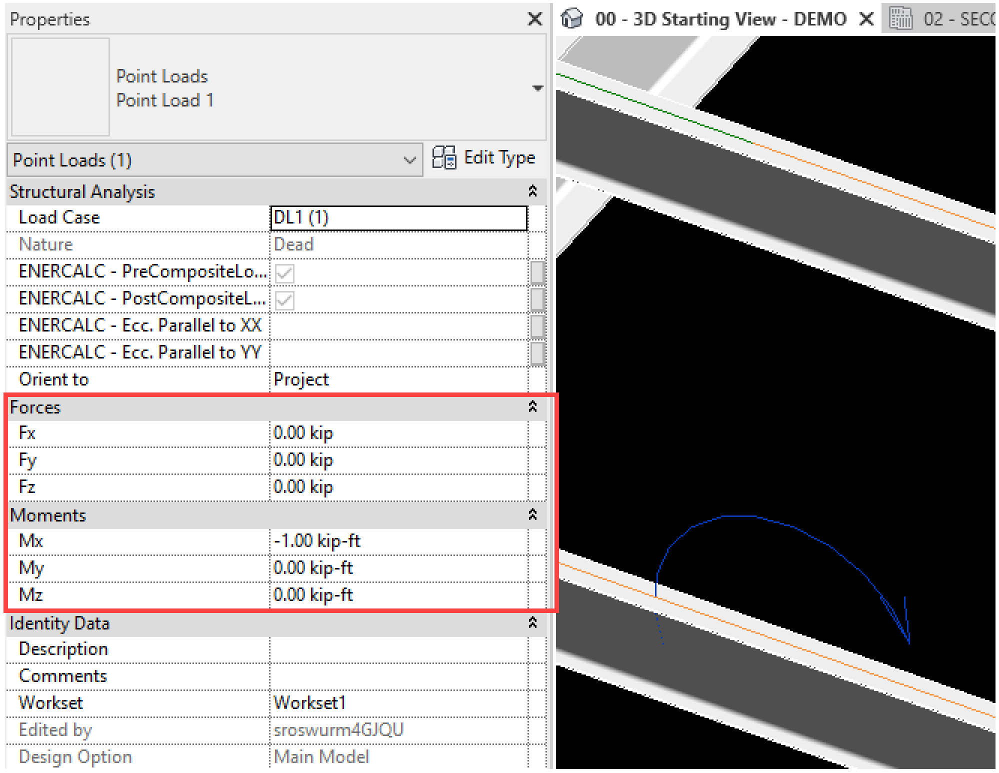


Loads that include horizontal components (i.e., Global X and Y in Revit) in addition to vertical will be included in the calculation, but only the vertical component will be used.





In addition to the cases discussed above, beam calculation launches also consider the presence of applied moments found in the Revit model. These moments are modeled in Revit as point loads and then the moment magnitudes are set using the “Forces” and “Moments” fields in the native Revit Properties pane.

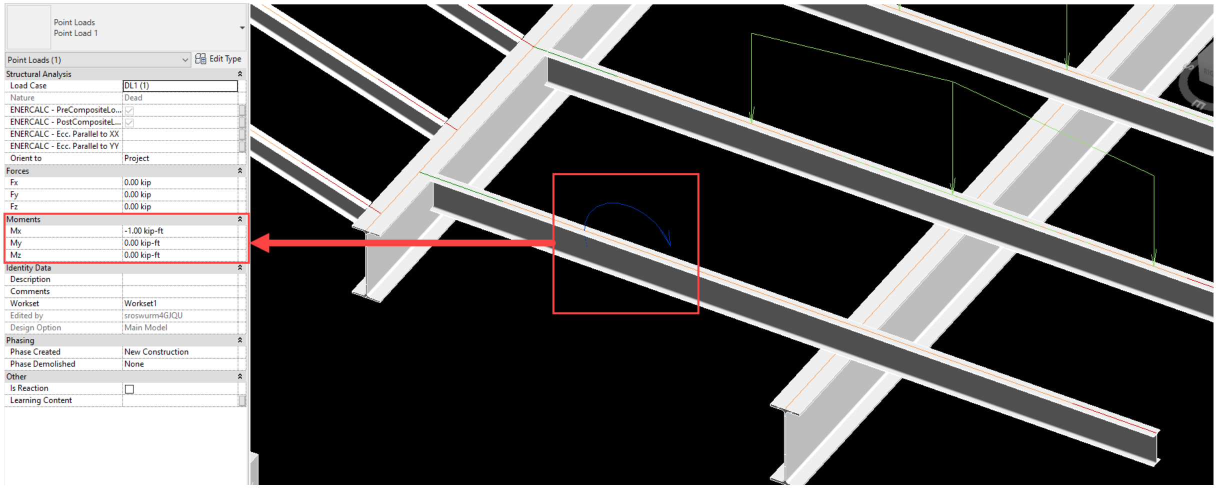


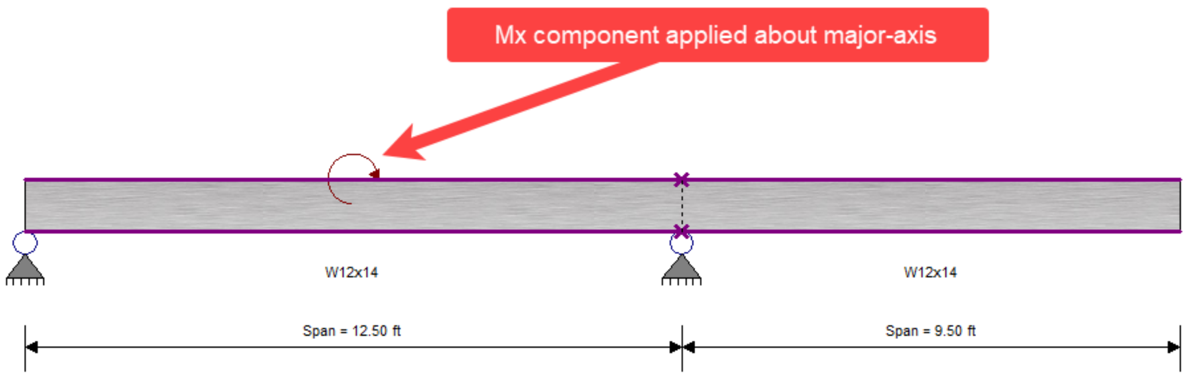
Similar to point loads with only force magnitudes, these moments are detected by proximity. In order to ease the user's workload in placing the moments and setting the magnitudes, the processing of moments into ENERCALC SEL uses a simplified approach. Although Revit point moments are defined by global axis magnitudes in the Revit interface, it is **NOT NECESSARY** for the user to set components of the moment about each axis to ensure that the vector sum resultant acts about the proper local axis of the beam. Instead, the components of the moment are taken to apply directly to the corresponding local axis of the same name:

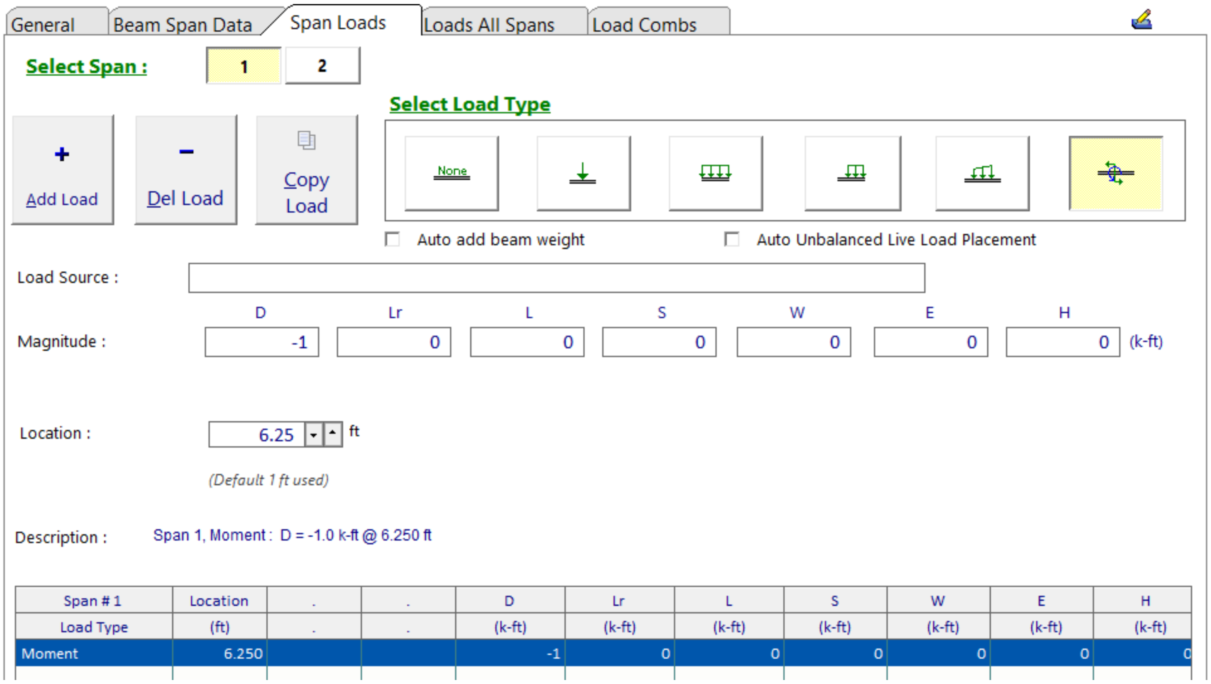
- Revit parameter Mx would cause bending about the beam's X-X axis in ENERCALC SEL
- Revit parameter My would cause bending about the beam's Y-Y axis in ENERCALC SEL
- Revit parameter Mz will cause bending about the beam's Z-Z axis in ENERCALC SEL

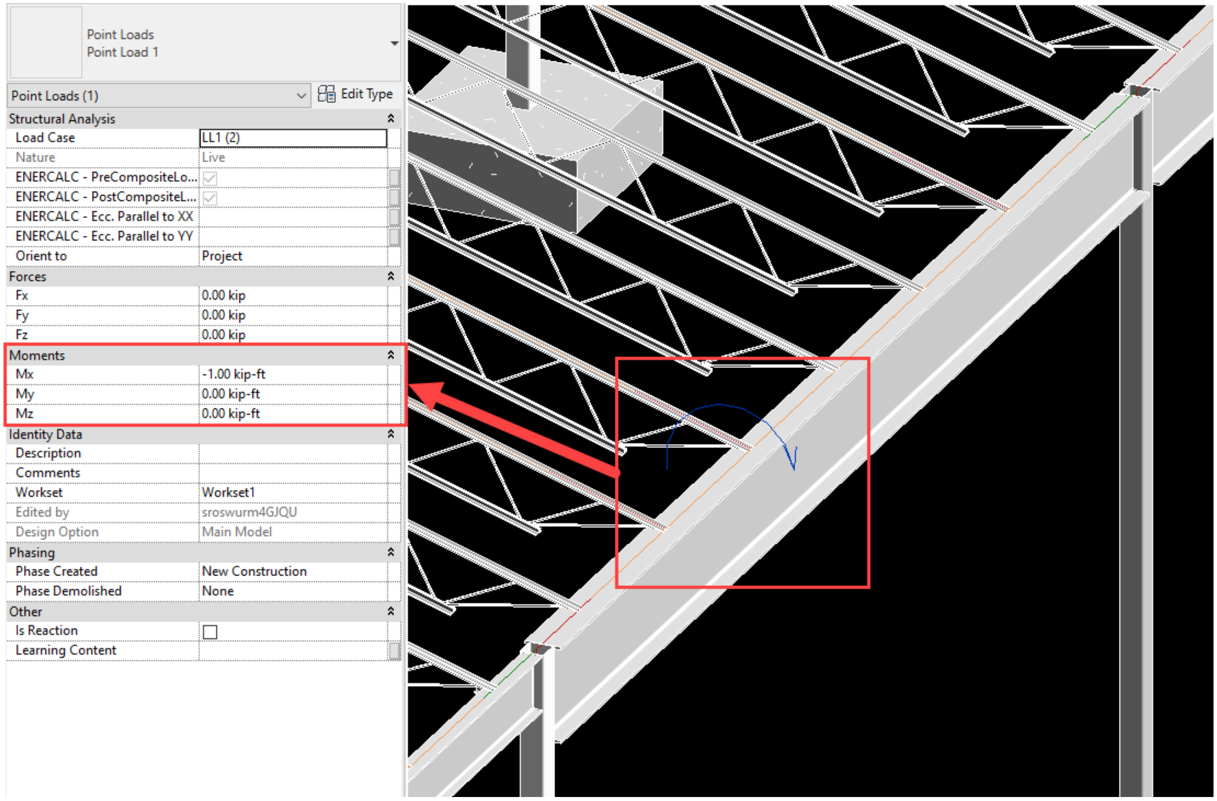
This means that in conventional beam calculations that do not consider torsion or biaxial bending, only Mx is of interest. This also means that the visual / graphical orientation of the moment symbol relative to the beam in the Revit model does not control the directionality in the ENERCALC SEL calculation.

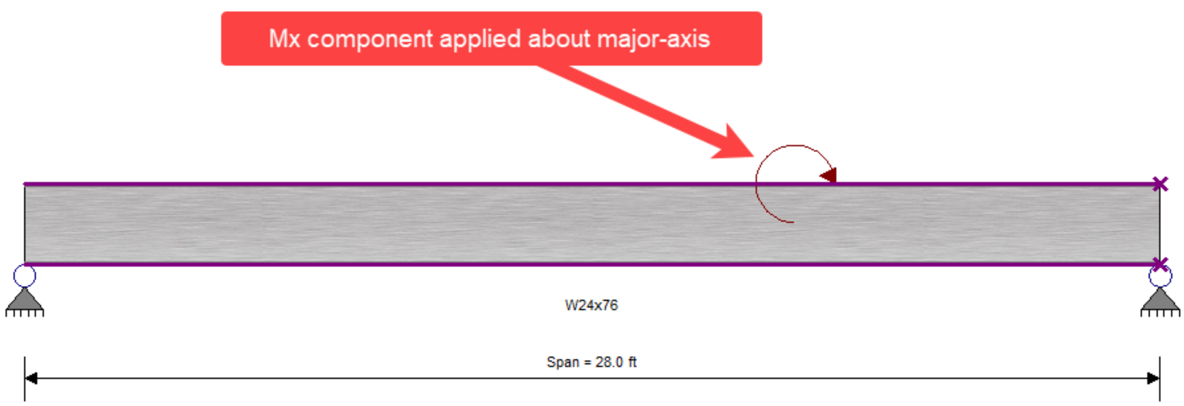
Note: Both point loads and point moments are assigned to beams on a first-come / first-served basis. Once a particular applied moment or load is automatically marked as having been associated to a particular beam calculation, it will not be eligible for inclusion in the launch of other calculations. This prevents the forces effects from being accidentally double-applied.

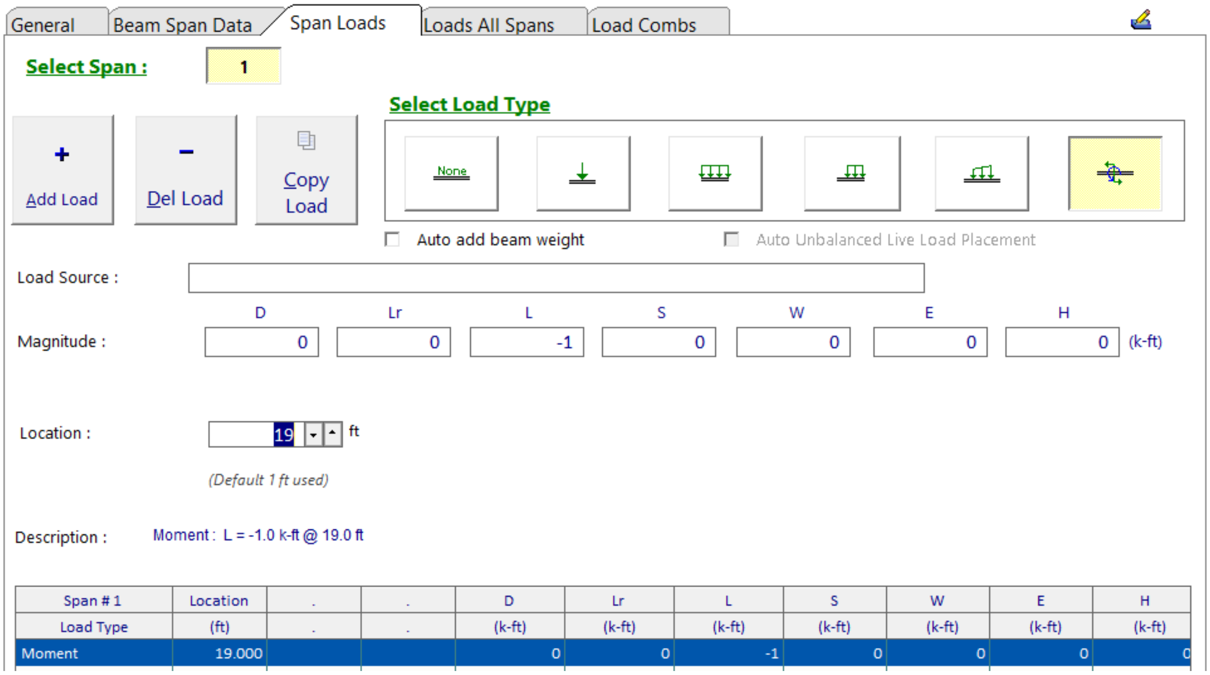





General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Select Span :
1
2
Select Load Type
Add Load
Del Load
Copy Load
None
Auto add beam weight
Auto Unbalanced Live Load Placement
Load Source :
D
Lr
L
S
W
E
H
Magnitude :
-1
0
0
0
0
0
0
(k-ft)
Location :
6.25
ft
(Default 1 ft used)
Description :
Span 1, Moment : D = -1.0 k-ft @ 6.250 ft
Span # 1 | Location | . | . | D | Lr | L | S | W | E | H
Load Type | (ft) | . | . | (k-ft) | (k-ft) | (k-ft) | (k-ft) | (k-ft) | (k-ft) | (k-ft)
Moment | 6.250 | | | -1 | 0 | 0 | 0 | 0 | 0 | 0

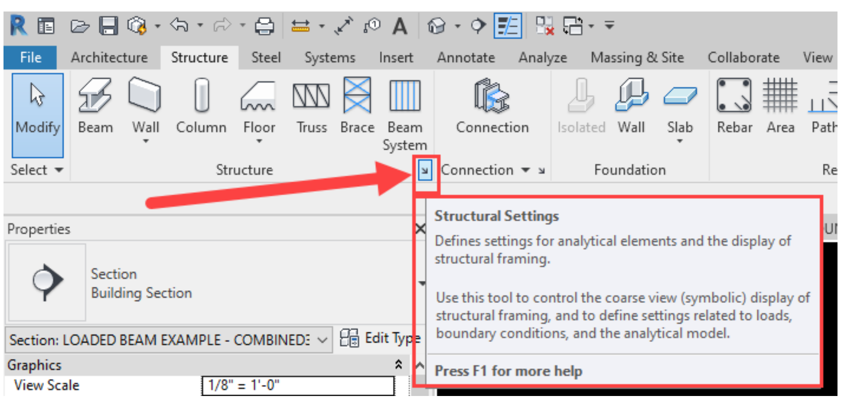

Point Loads
Point Load 1
Point Loads (1)
Edit Type
Structural Analysis
Load Case LL1 (2)
Nature Live
ENERCALC - PreCompositeLo...
ENERCALC - PostCompositeL...
ENERCALC - Ecc. Parallel to XX
ENERCALC - Ecc. Parallel to YY
Orient to Project
Forces
Fx 0.00 kip
Fy 0.00 kip
Fz 0.00 kip
Moments
Mx -1.00 kip-ft
My 0.00 kip-ft
Mz 0.00 kip-ft
Identity Data
Description
Comments
Workset Workset1
Edited by sroswurm4GJQU
Design Option Main Model
Phasing
Phase Created New Construction
Phase Demolished None
Other
Is Reaction
Learning Content


Mx component applied about major-axis
W24x76
Span = 28.0 ft



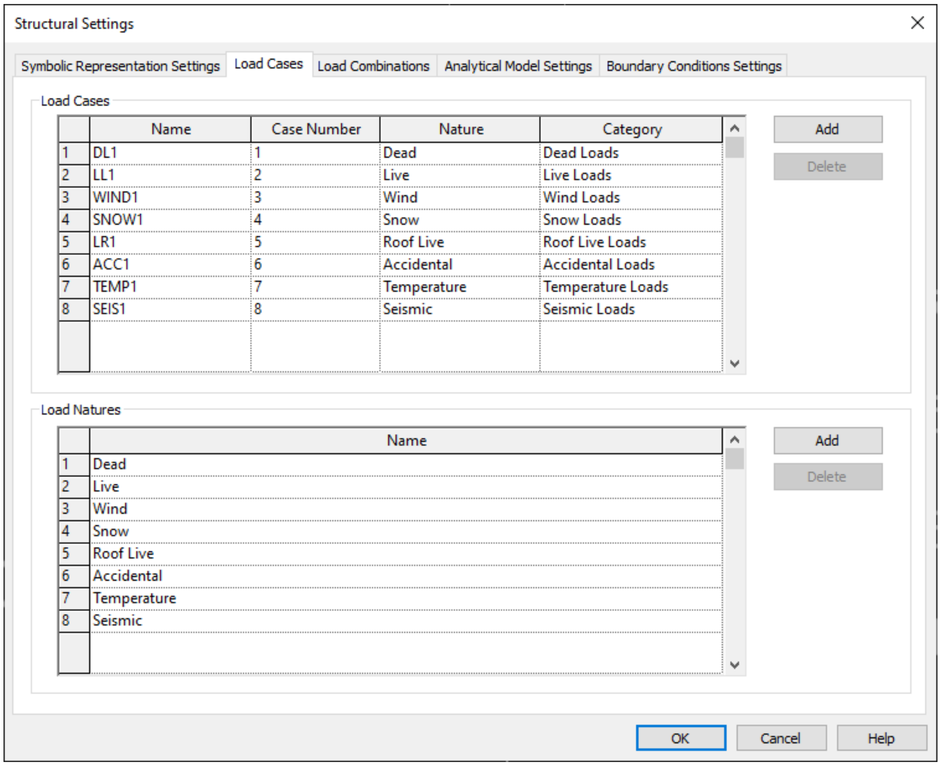
10.4.2 Load Cases and Combinations

Load are managed and mapped from Revit to ENERCALC SEL using the basic load cases that are created using the default Revit structural template. These cases may be viewed at any time using the native Revit button for Structural Settings on the Structure ribbon tab.



Load cases are found under Structure > Structural Settings > Load Cases. These are default settings and typically do not need to be manually created or modified in any way. Changing the default settings will interfere with the mapping of loads to the ENERCALC SEL calculation. In the table shown here, all load cases are mapped directly to the corresponding case in ENERCALC SEL, with two exceptions:

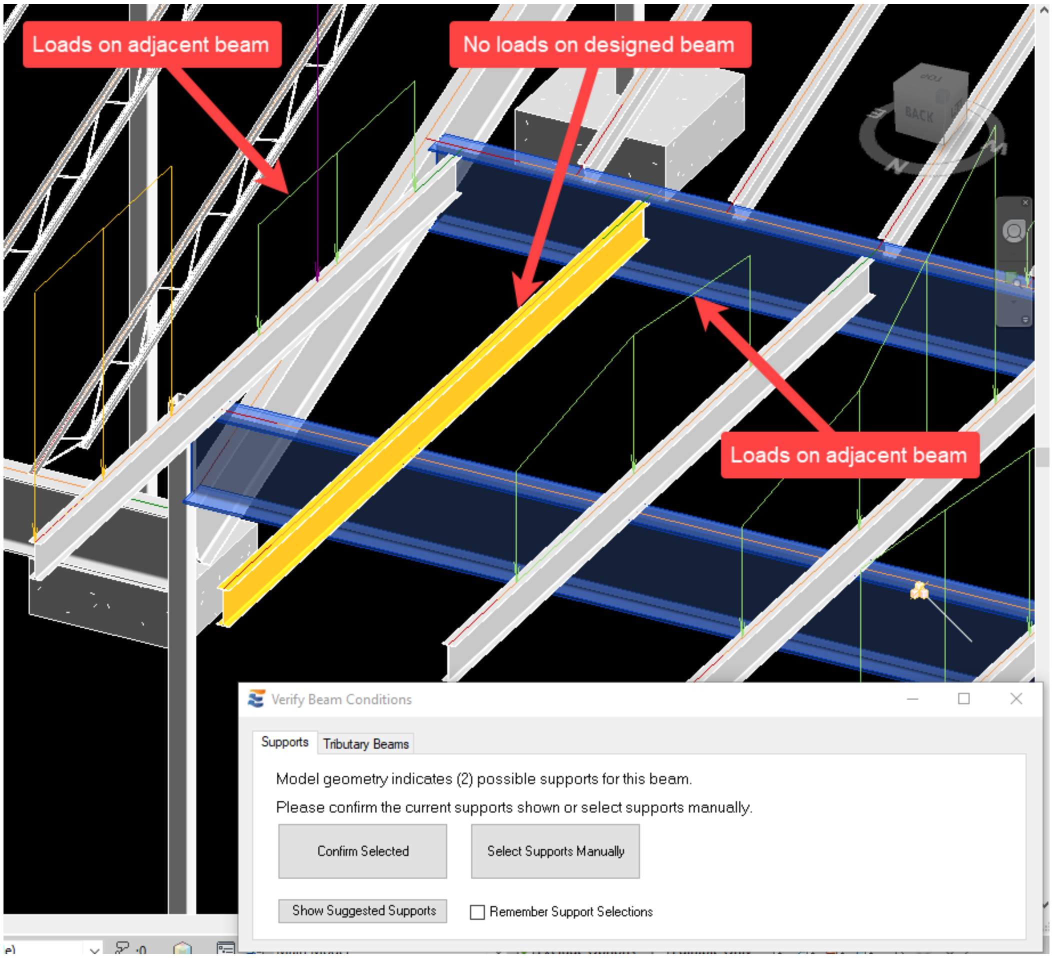
- ACC1 maps to earth load (H) in ENERCALC SEL
- TEMP1 is ignored and does not map to any case in ENERCALC SEL

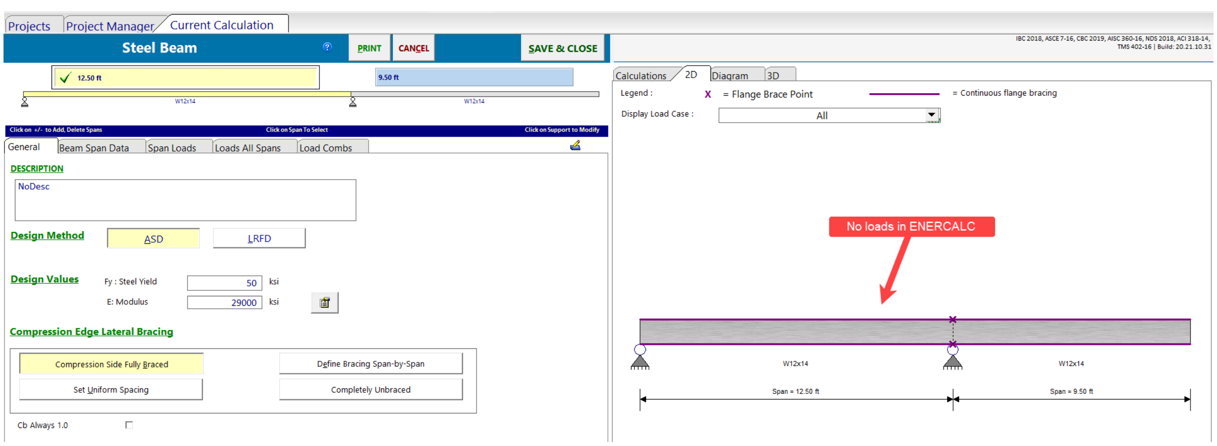


Load combinations are managed exclusively from the ENERCALC SEL interface. Load combination definitions found in the Revit model will not influence the design calculations in any way.

10.4.3 Launching Without Revit Loads

Although detection of Revit load elements is supported in ENERCALC for Revit, it is also permissible to launch a beam calculation with no existing Revit load objects. In such cases, the beam calculation arrives in the ENERCALC SEL interface with no loads applied.

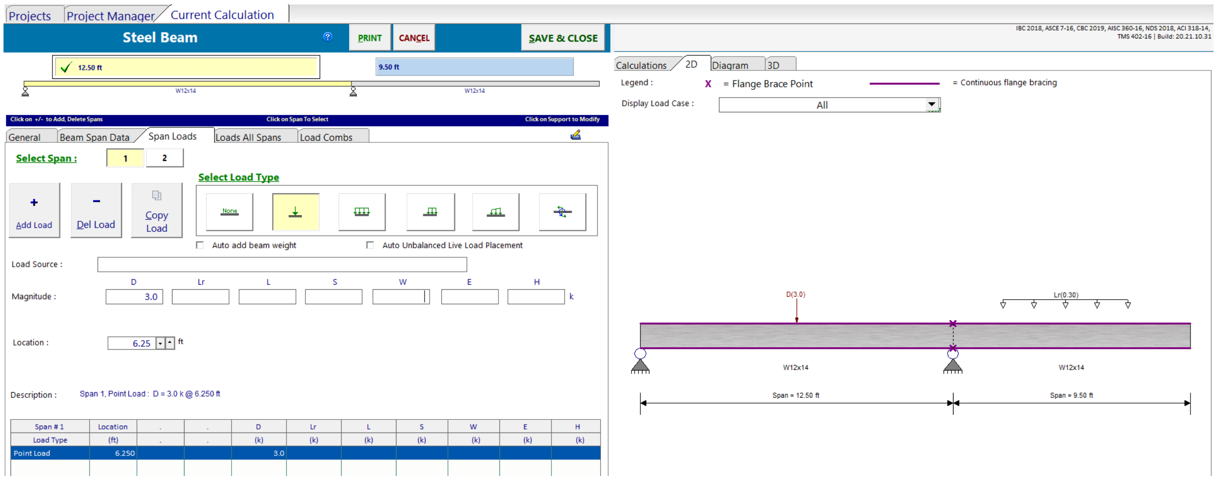


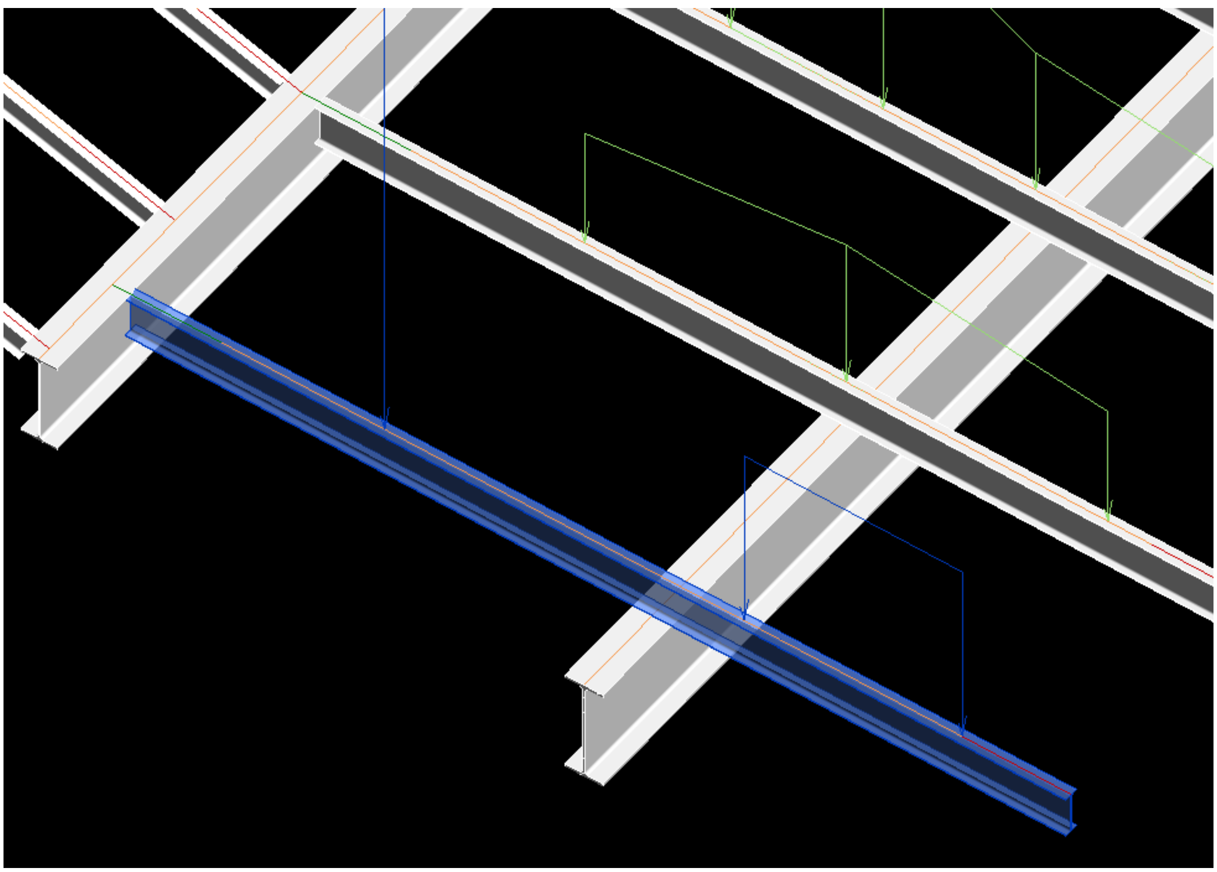


Once the calculation appears in ENERCALC SEL, loads may then be applied to the beam calculation manually via the “Span Loads” and “Loads All Spans” tabs.

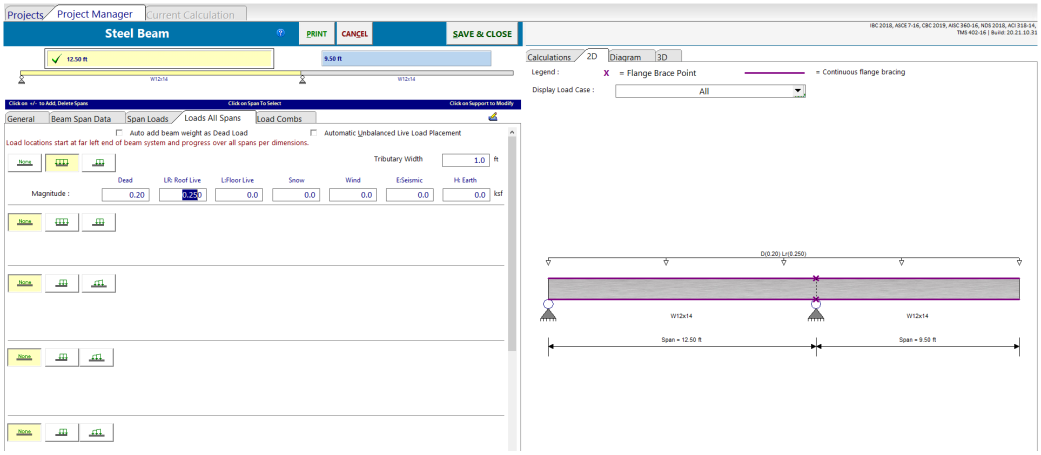
Loads created manually by the user in ENERCALC SEL will be automatically stored in the Revit model after the “Save and Close” operation is complete. The general protocol for mapping these loads from ENERCALC SEL back to Revit is as follows:

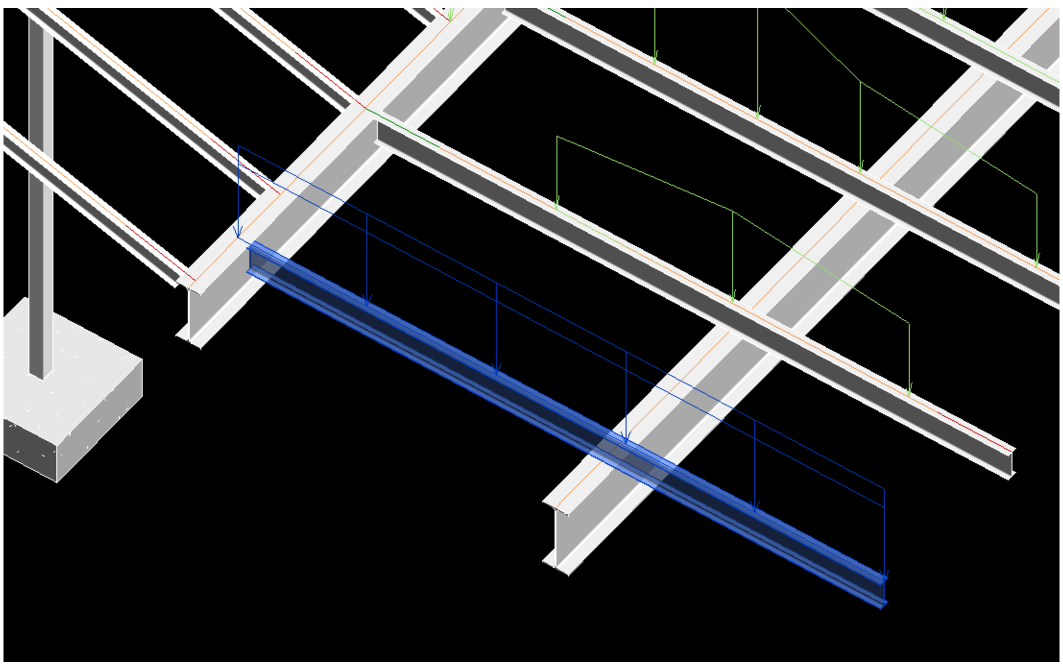
- Loads created in the “Span Loads” tab are mapped back to Revit as ***non-hosted*** load elements.



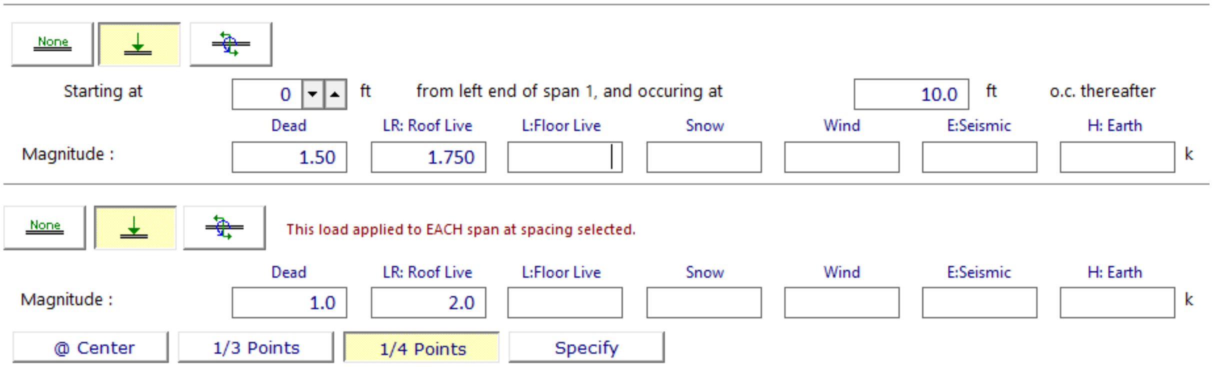


- Loads created in the “Loads All Spans” tab are mapped back to Revit as ***hosted*** load elements.





Users will note that there are certain minor differences between the conventional “Loads All Spans” tab and the tab available when the calculation is sent from Revit. The input controls for arrays of point loads extending across multiple spans are **NOT** available when working with Revit-based calculations.

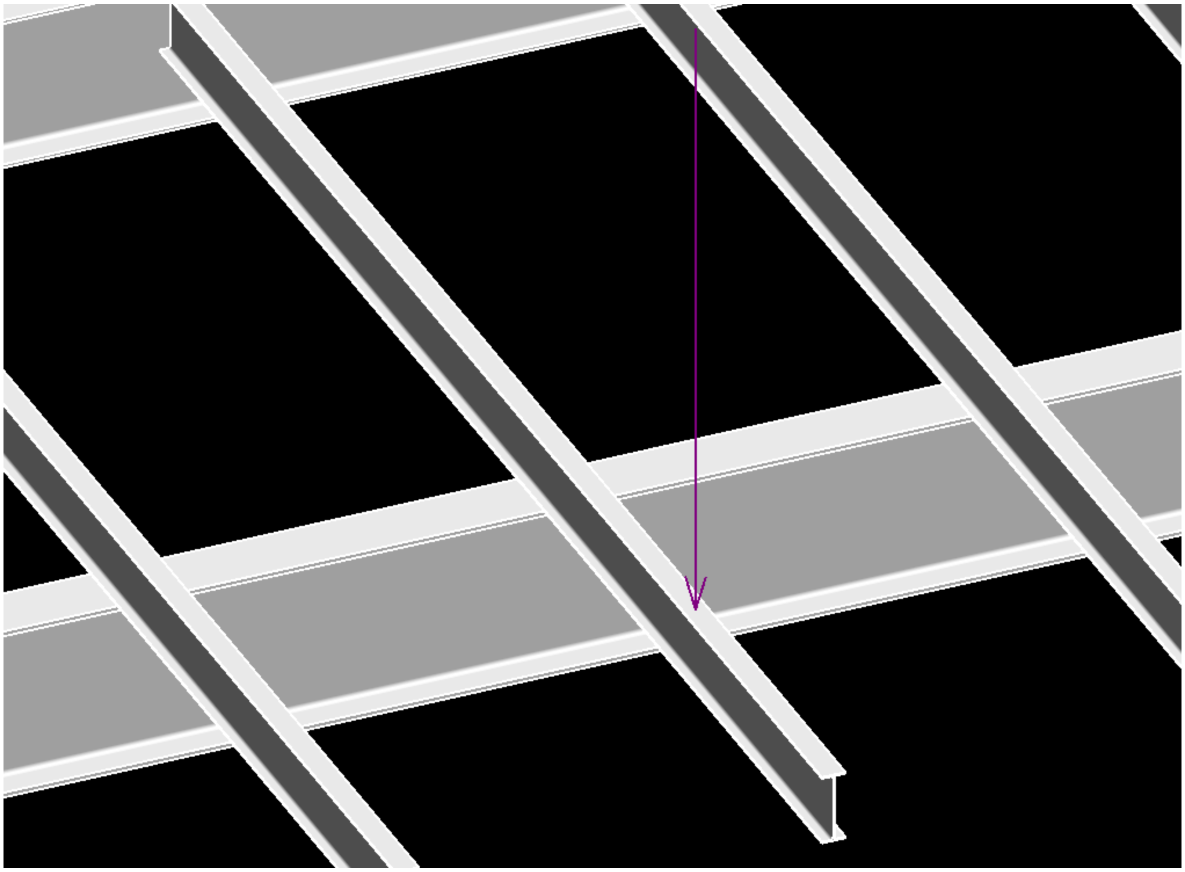


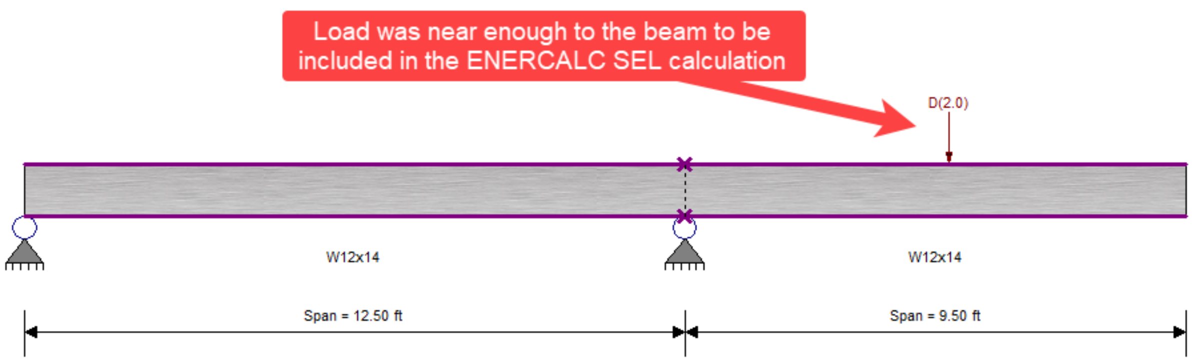
Hosted point loads may not be directly created from SEL because they are only used to represent reactions from other beams. As a result, any point load created in the SEL interface will populate the Revit model as a non-hosted point load.

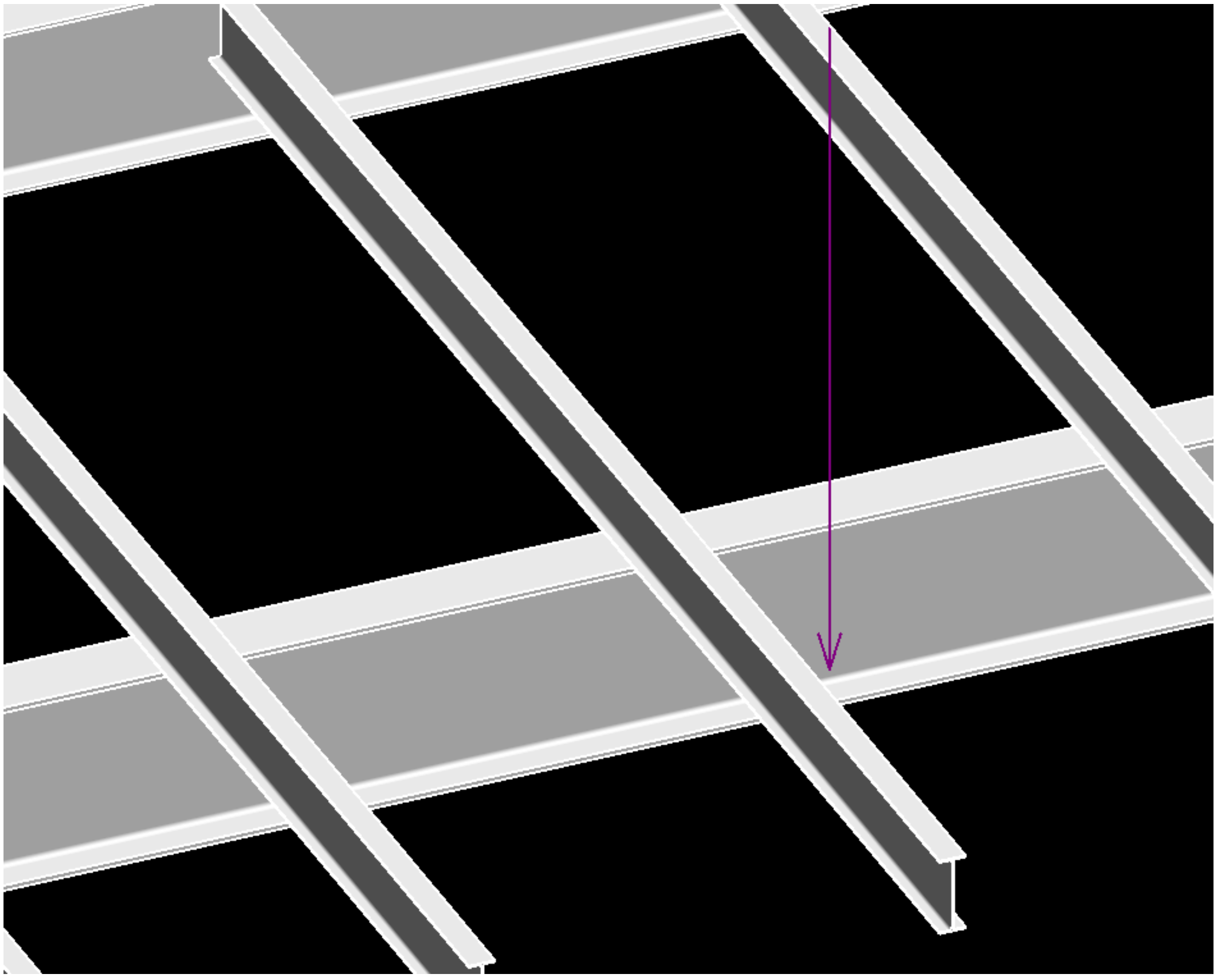
10.4.4 Launching With Revit non-Hosted loads

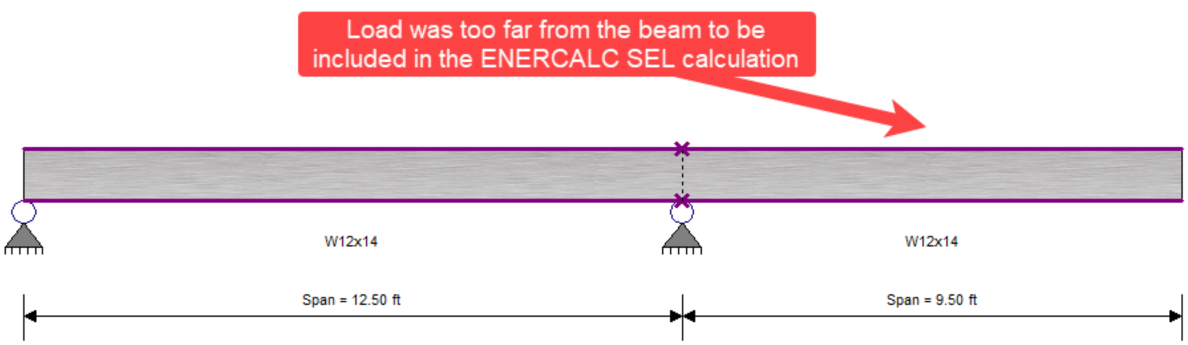
During the beam calculation launch process, non-hosted load elements are detected by physical proximity to the beam. Physical proximity is tested with a tolerance of 2 inches beyond the face of the beam in each direction (top, bottom, left side, right side, start, end). Any non-hosted load falling within this boundary will be considered for inclusion in the calculation.

Note: To be included in a calculation, non-hosted line loads and point loads must be found in direct proximity to the beam, even when a floor element is present. ENERCALC for Revit does NOT consider the transfer of force effects of miscellaneous linear loads or point loads found on a floor in between beams.

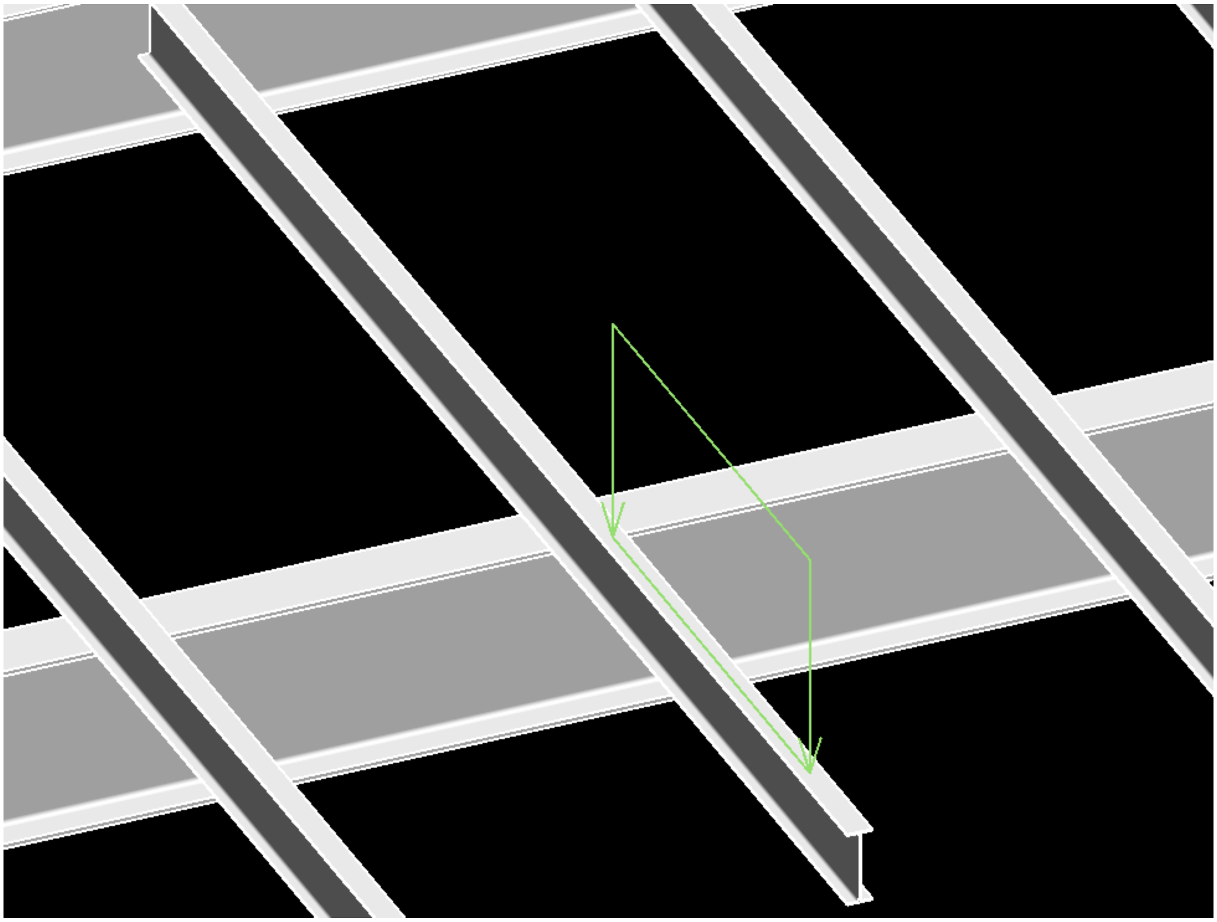


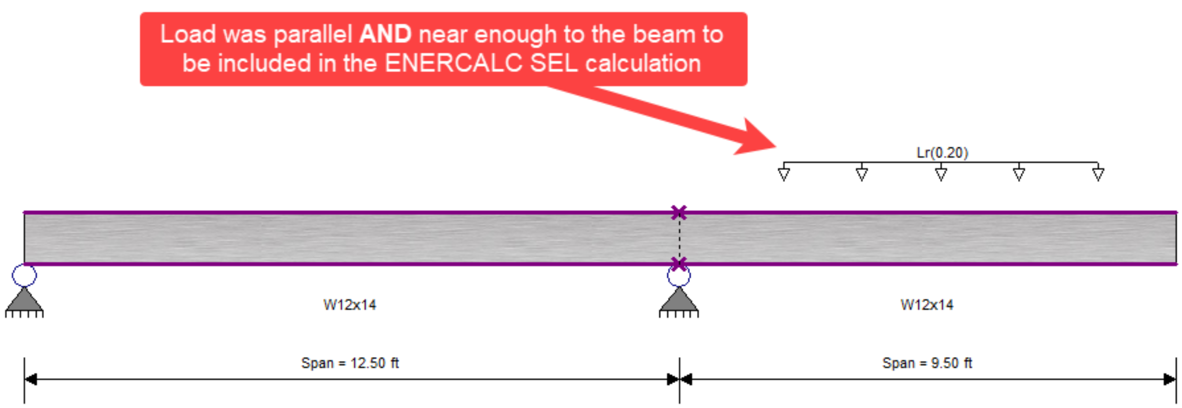

Load was near enough to the beam to be included in the ENERCALC SEL calculation
D(2.0)
W12x14
W12x14
Span = 12.50 ft
Span = 9.50 ft

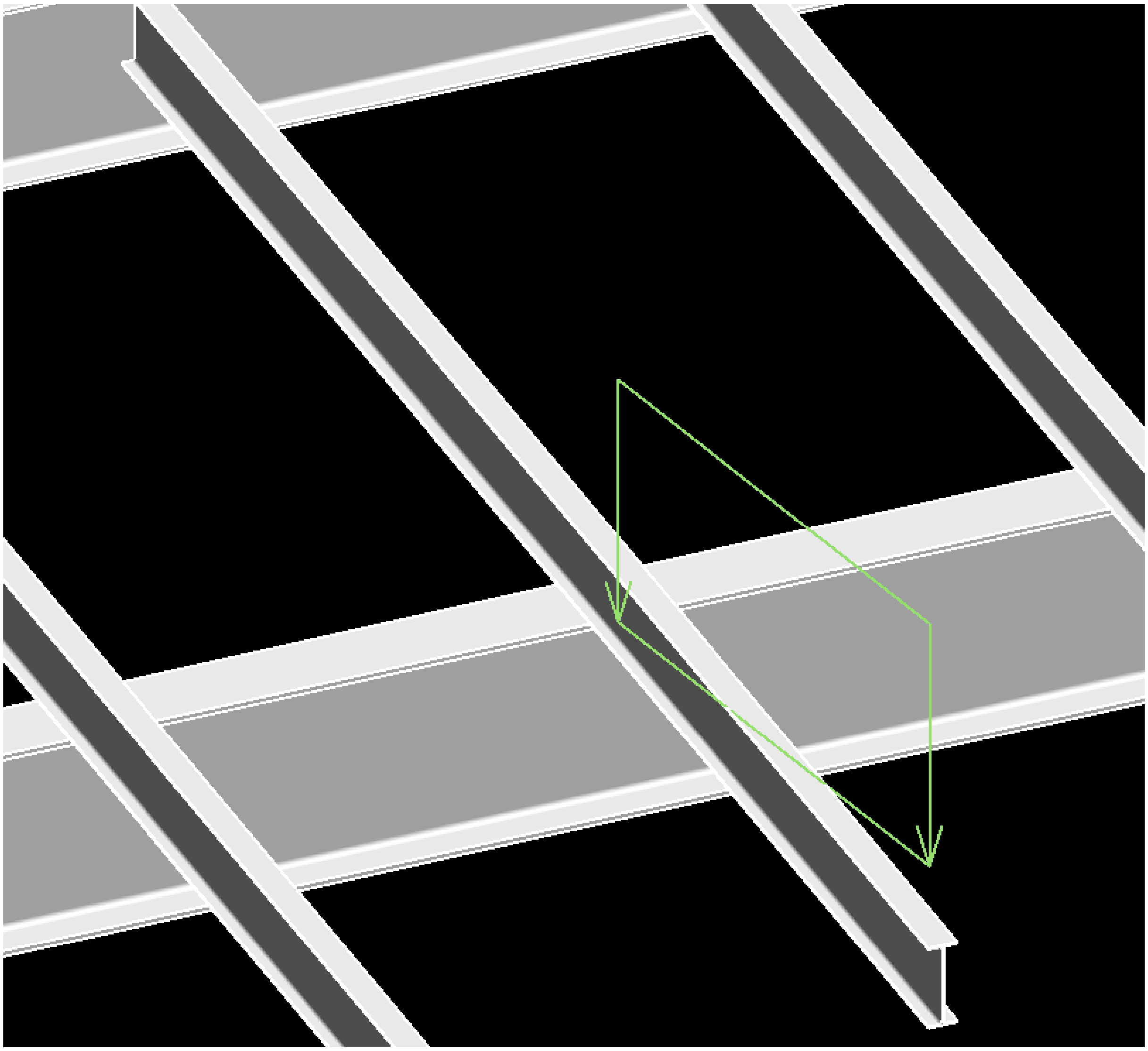


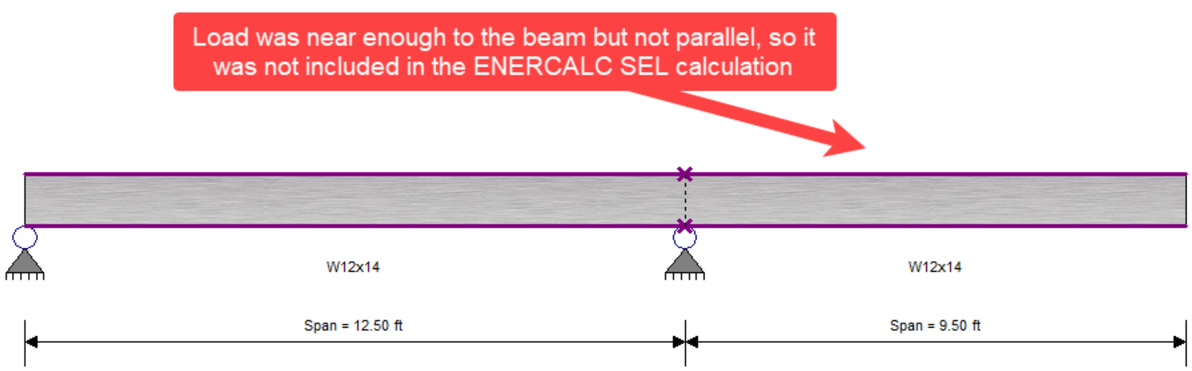


In addition to proximity, linear loads are also tested for alignment. Linear loads must be oriented parallel to the designed beam in order to be included.

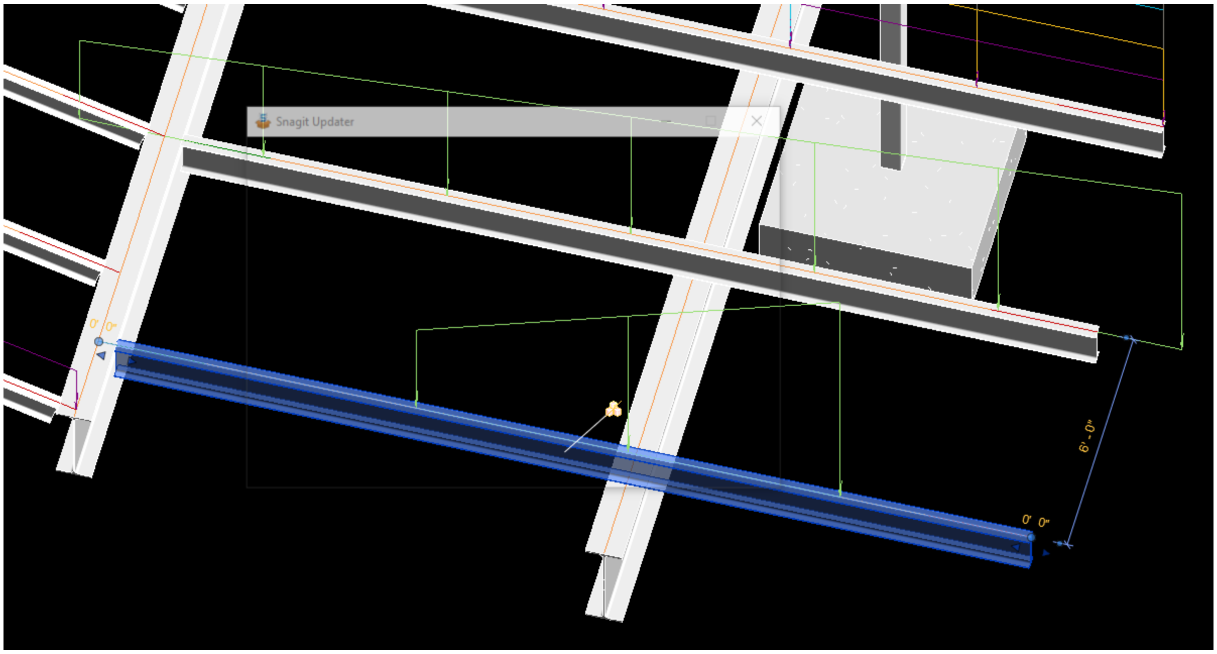


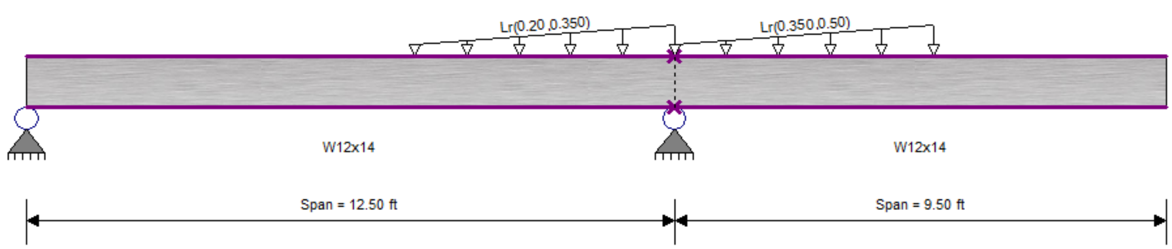

Load was parallel **AND** near enough to the beam to be included in the ENERCALC SEL calculation
Lr(0.20)
W12x14
W12x14
Span = 12.50 ft
Span = 9.50 ft



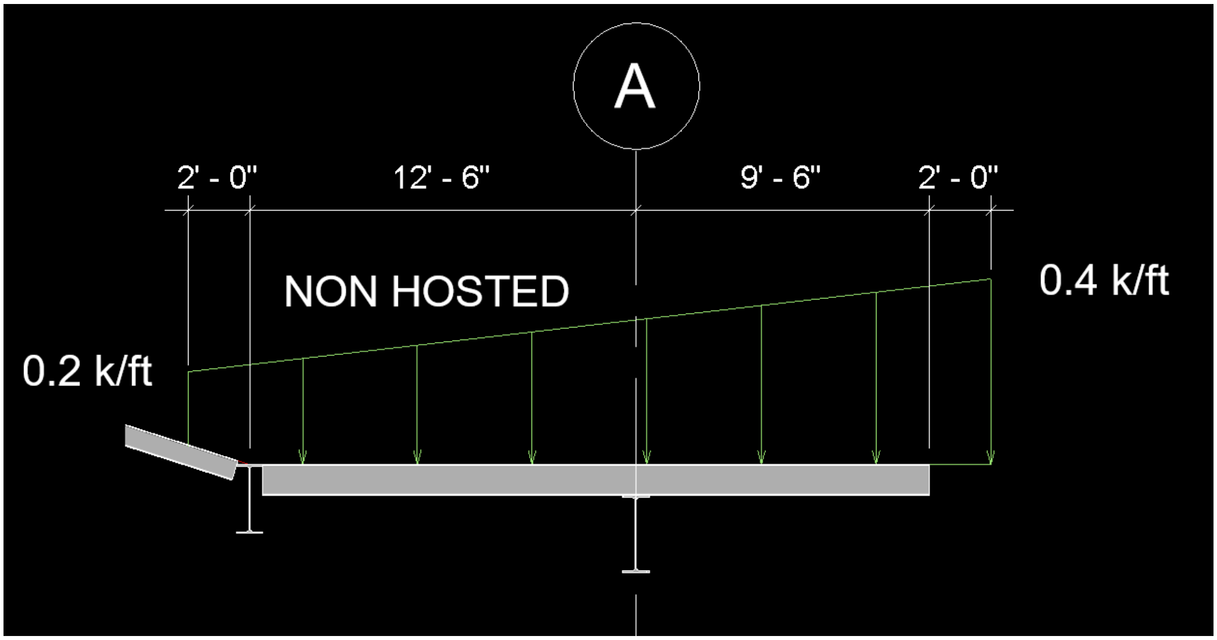

Load was near enough to the beam but not parallel, so it was not included in the ENERCALC SEL calculation
W12x14
W12x14
Span = 12.50 ft
Span = 9.50 ft

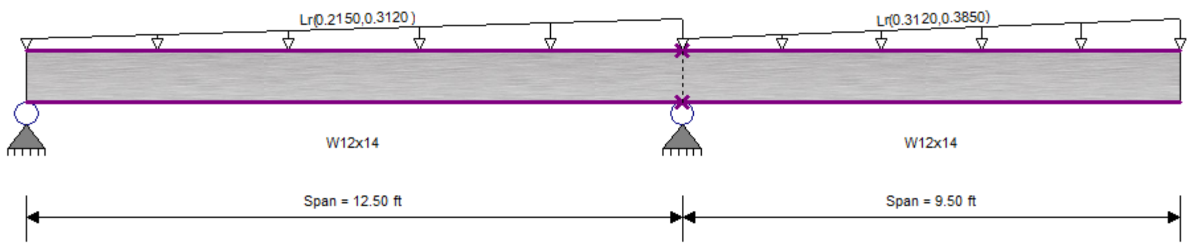
Non-hosted load elements that cross elements designated as supports for the loaded member are automatically divided and will appear in the ENERCALC SEL interface as separate loads applied to the two adjacent spans.



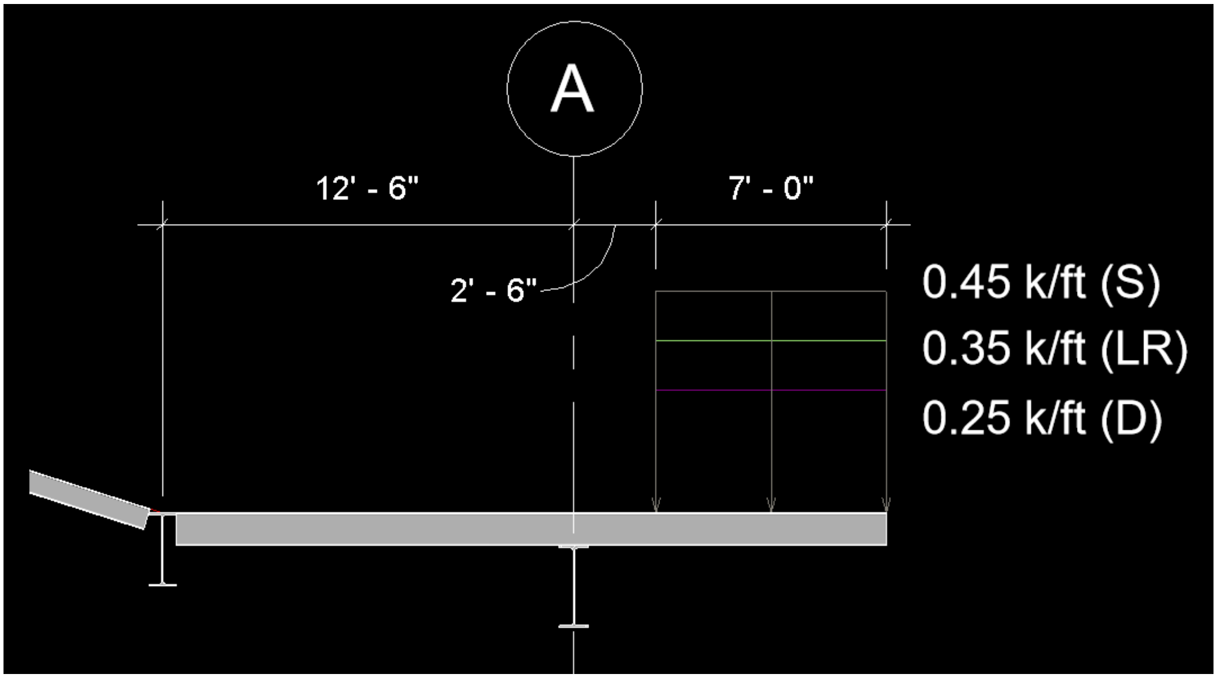


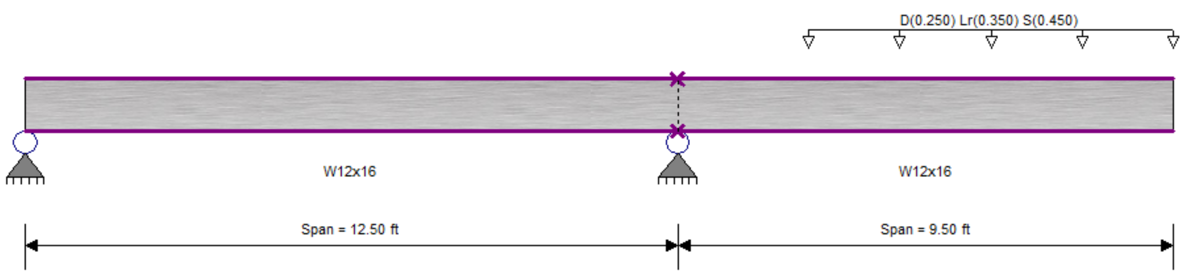
Non-hosted loads that extend beyond the ends of the beam will be interpolated as necessary to set the appropriate magnitudes at the ends of the beam.

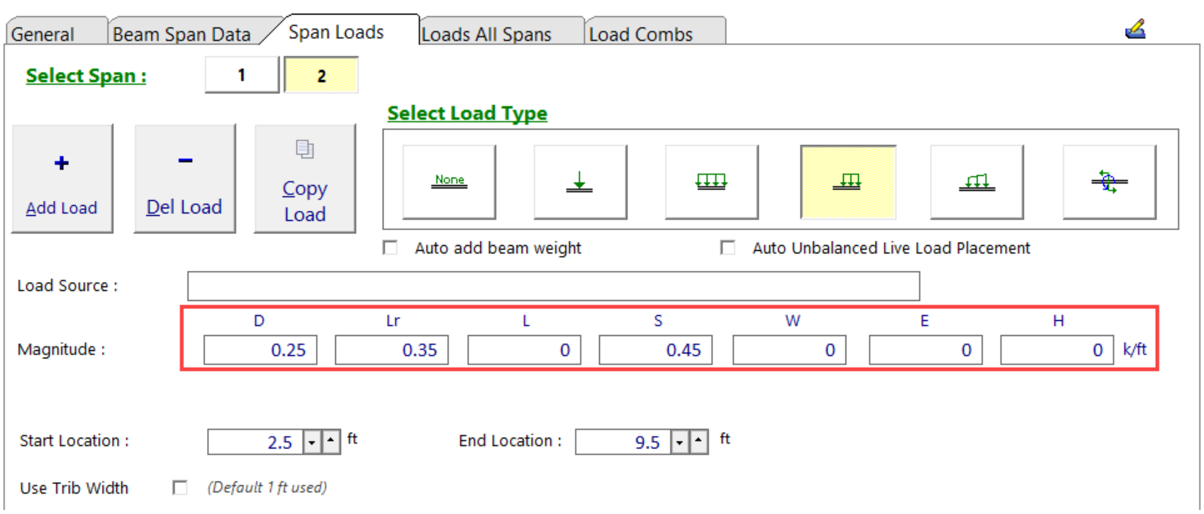




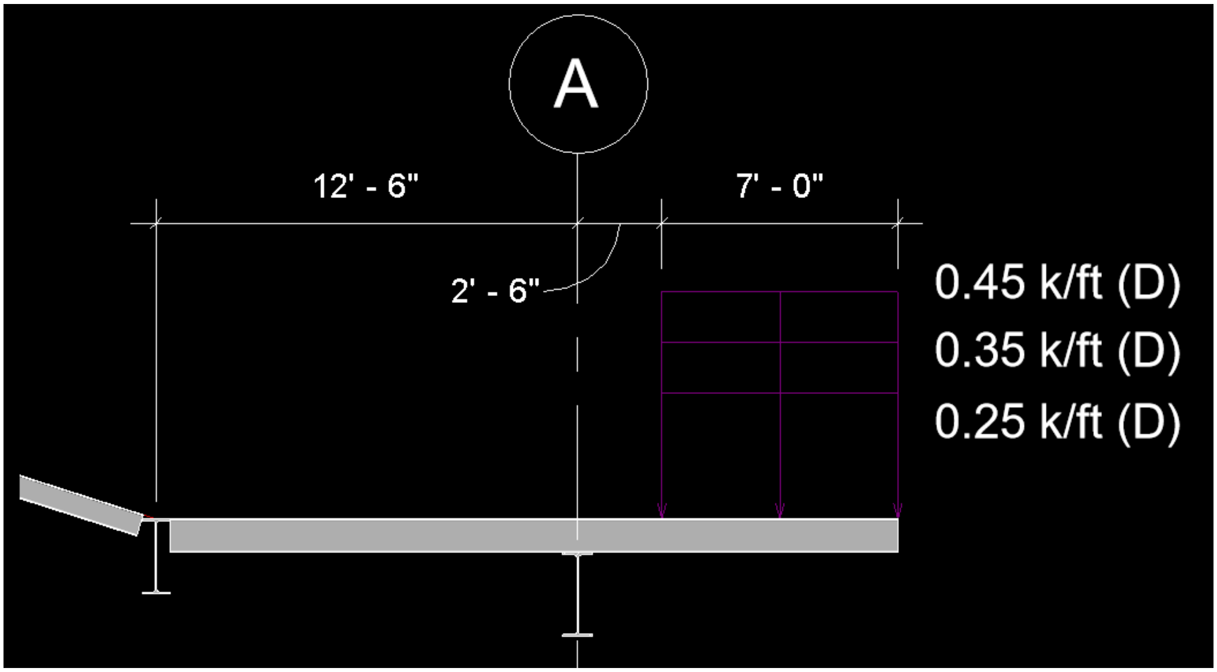
When multiple non-hosted load elements overlap with identical geometry but differing load cases, they are automatically consolidated to a single load item in the ENERCALC SEL interface, with the magnitudes corresponding to each load case preserved for individual review and modification.

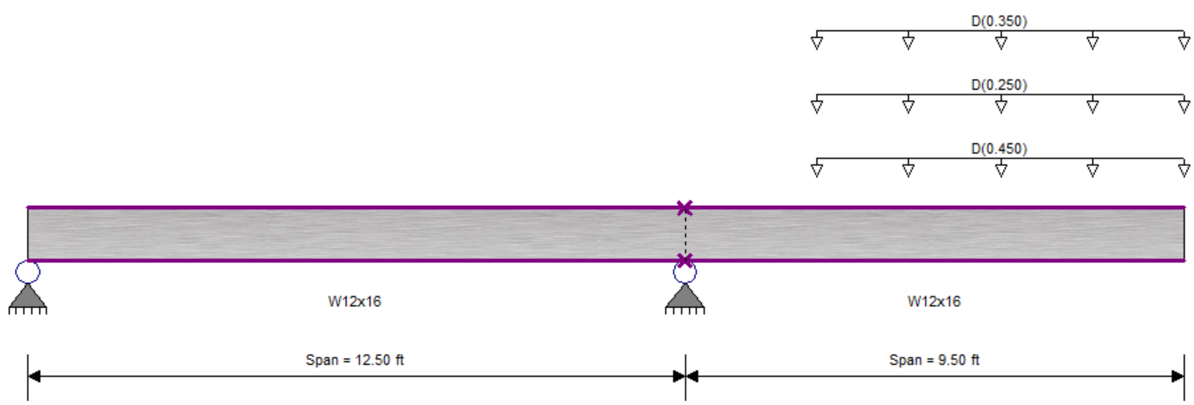

A
12' - 6"
7' - 0"
2' - 6"
0.45 k/ft (S)
0.35 k/ft (LR)
0.25 k/ft (D)

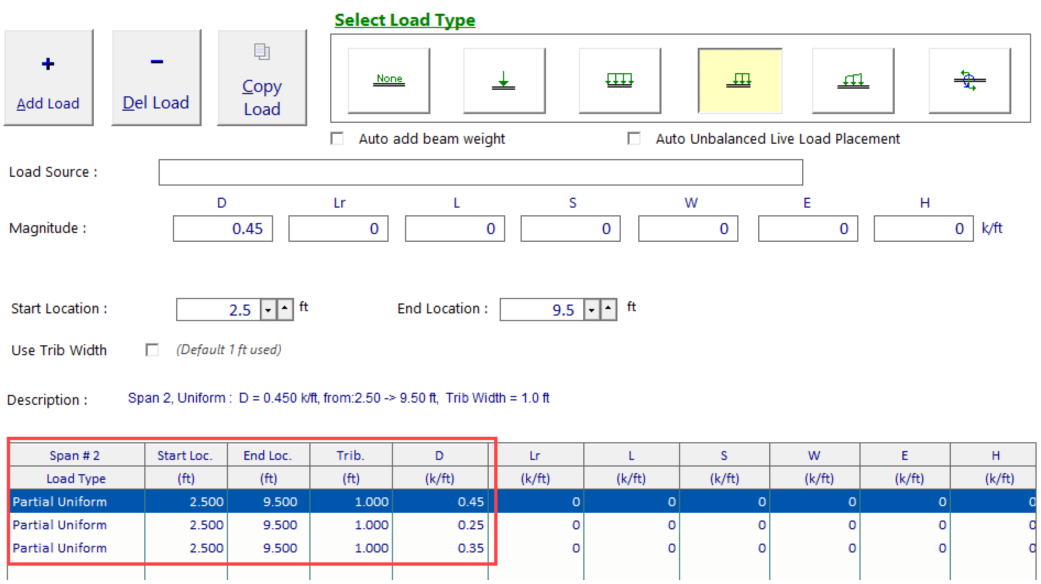

D(0.250) Lr(0.350) S(0.450)
W12x16
W12x16
Span = 12.50 ft
Span = 9.50 ft


General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Select Span :
1
2
Add Load
Del Load
Copy Load
Select Load Type
None
Auto add beam weight
Auto Unbalanced Live Load Placement
Load Source :
D
Lr
L
S
W
E
H
Magnitude :
0.25
0.35
0
0.45
0
0
0
k/ft
Start Location :
2.5
ft
End Location :
9.5
ft
Use Trib Width
(Default 1 ft used)

When multiple non-hosted load elements overlap with identical geometry and repetitive load cases, they are not consolidated but instead will appear as individual load items in the ENERCALC SEL interface.



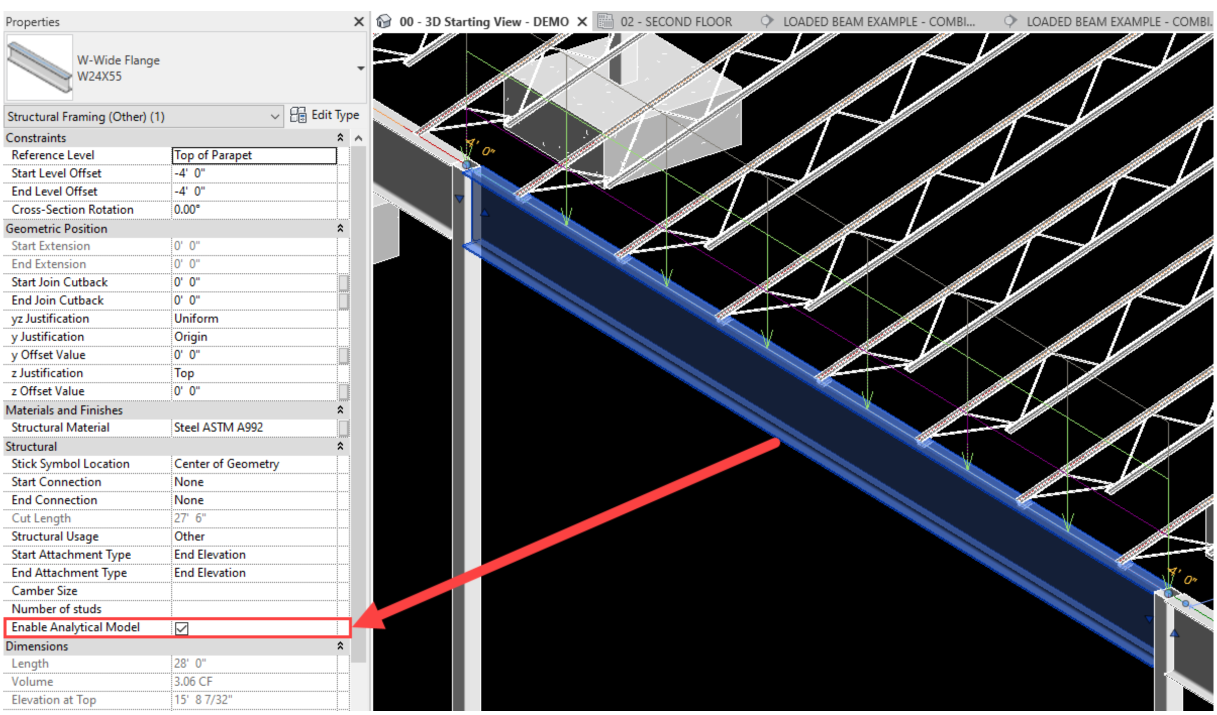




10.4.5 Launching With Revit Hosted Loads

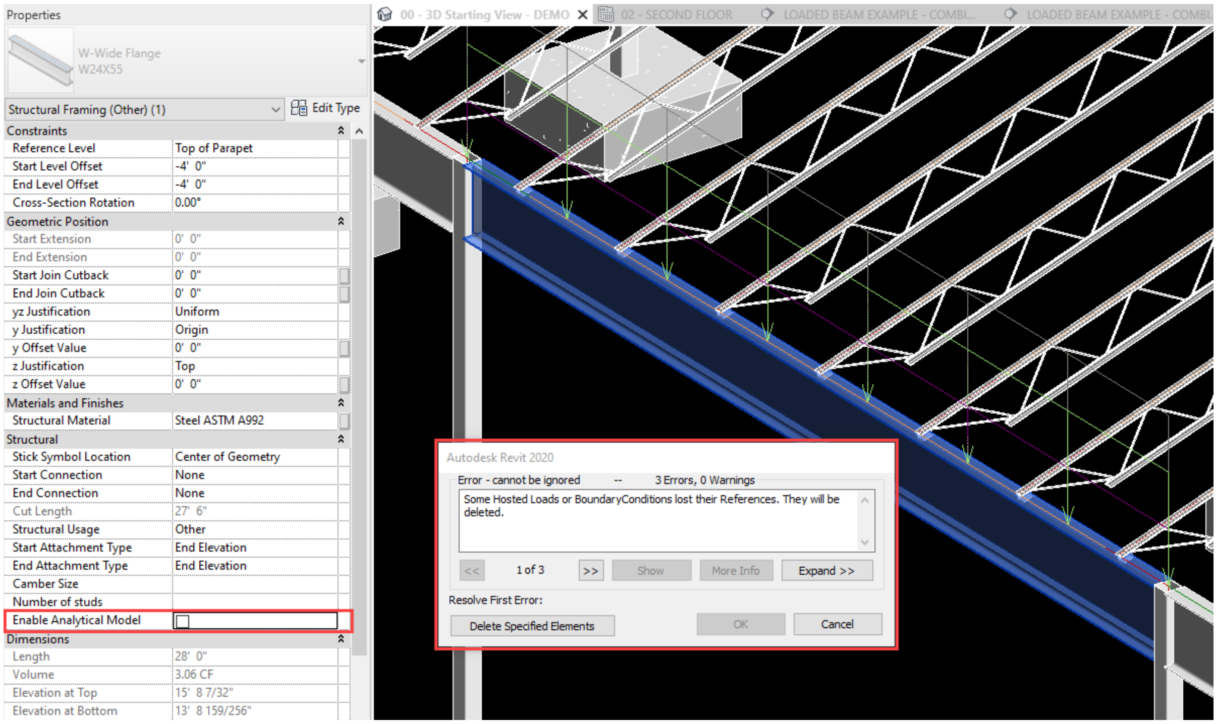
During the beam calculation launch process, hosted load elements are not detected by physical proximity to the beam. Instead, they are detected exclusively by their hosting relationship to the beam whose calculation is being launched.

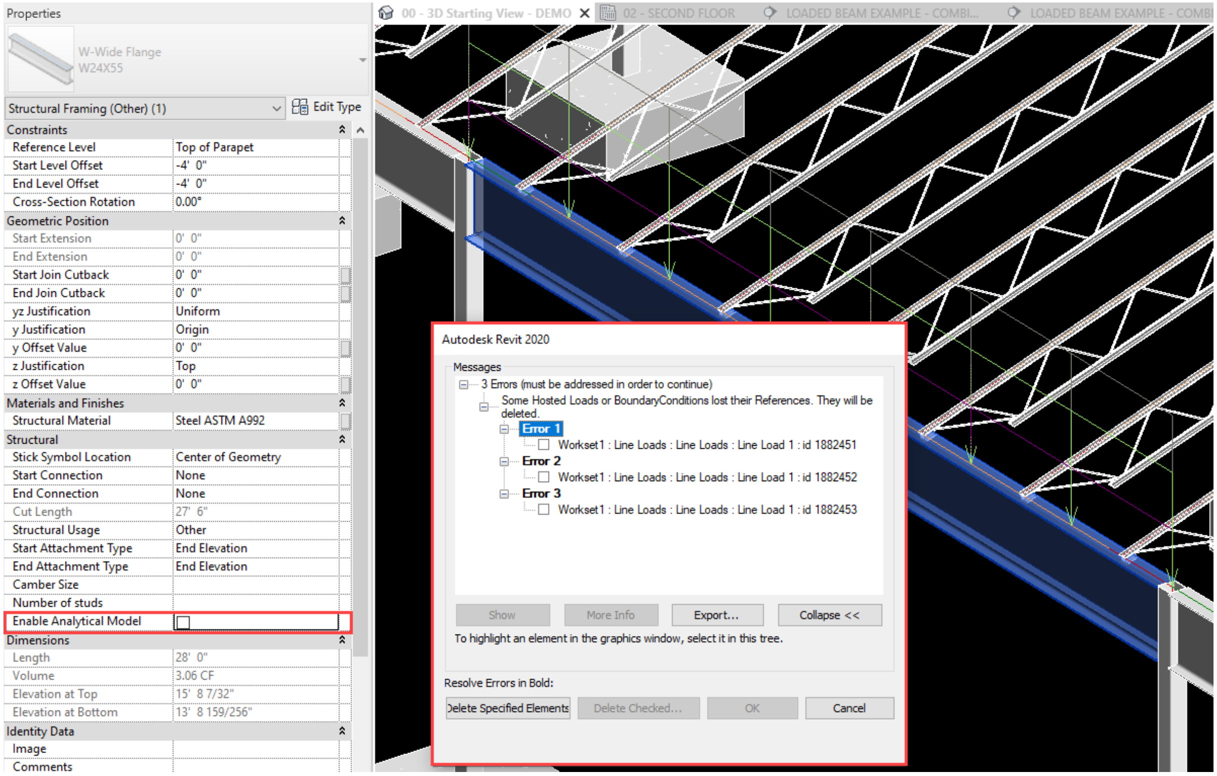
NOTE: It is **NOT** mandatory to enable the use of analytical model on a given beam element in order to launch a steel beam calculation. Steel beam calculations can be launched, edited, and saved without checking the "Enable…" control on the Properties pane.



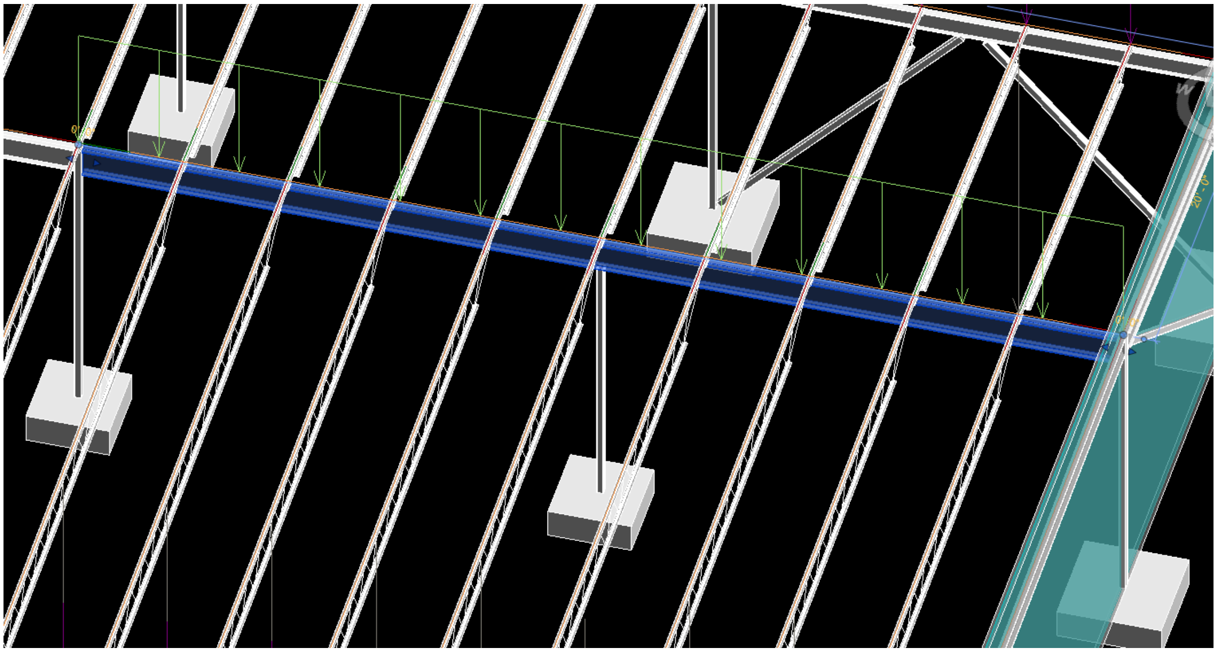
However, if the analytical model remains disabled on a given beam, the user will not be able to place hosted loads on the beam. This is because hosted loads are hosted to the underlying analytical element rather than to the physical element.

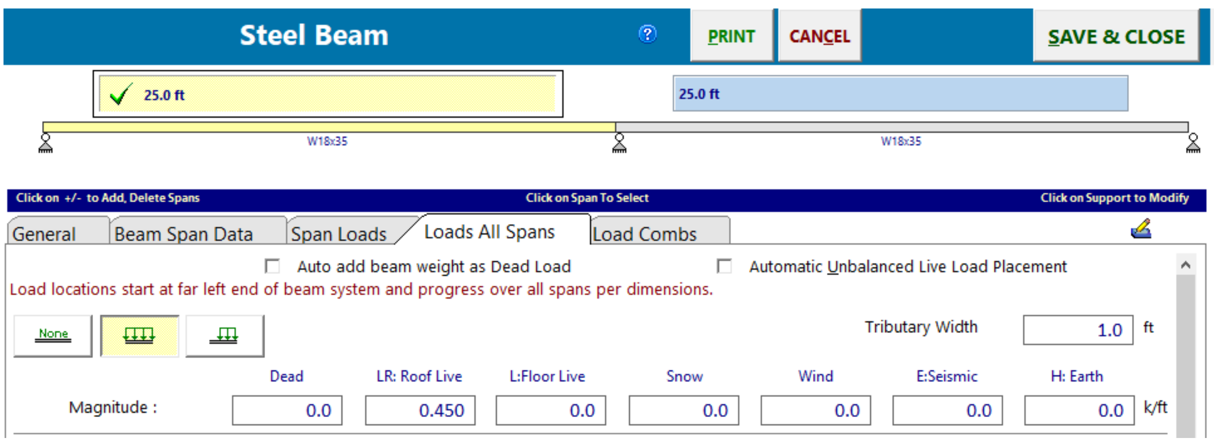
Similarly, if the analytical model is disabled on a particular beam AFTER hosted loads have been created, then the hosted loads belonging to that analytical element will be automatically removed from the Revit model.

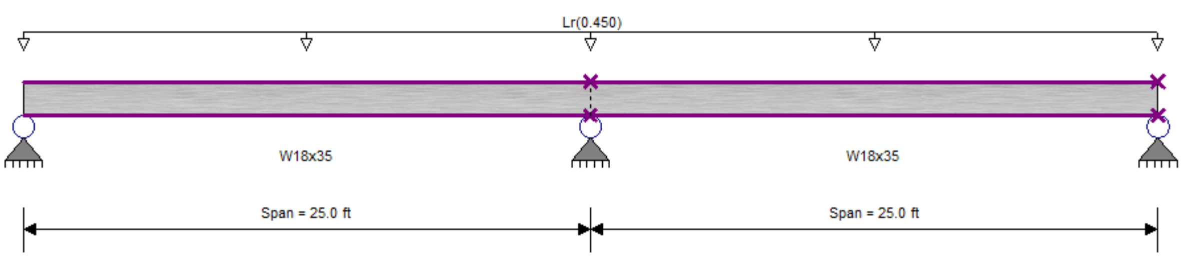




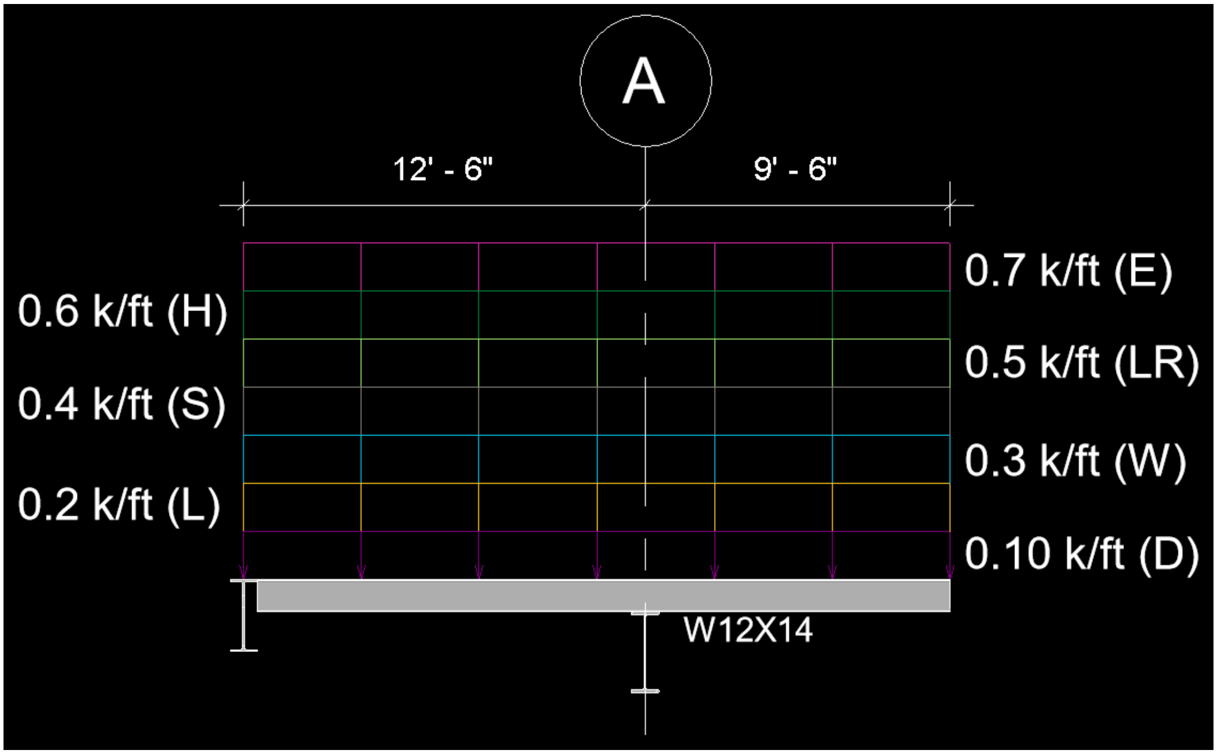
As indicated previously, hosted load elements detected during a beam calculation launch are mapped to the “Loads All Spans” tab in the ENERCALC SEL interface. Since Revit hosted load elements automatically assume the full extents of the analytical element, all hosted loads detected during calculation launch automatically extend from the start of the beam to the end of the beam when populated in the ENERCALC SEL calculation. At this time, no provision is made for scenarios where the extents of the analytical beam differ from the extents of the physical beam.

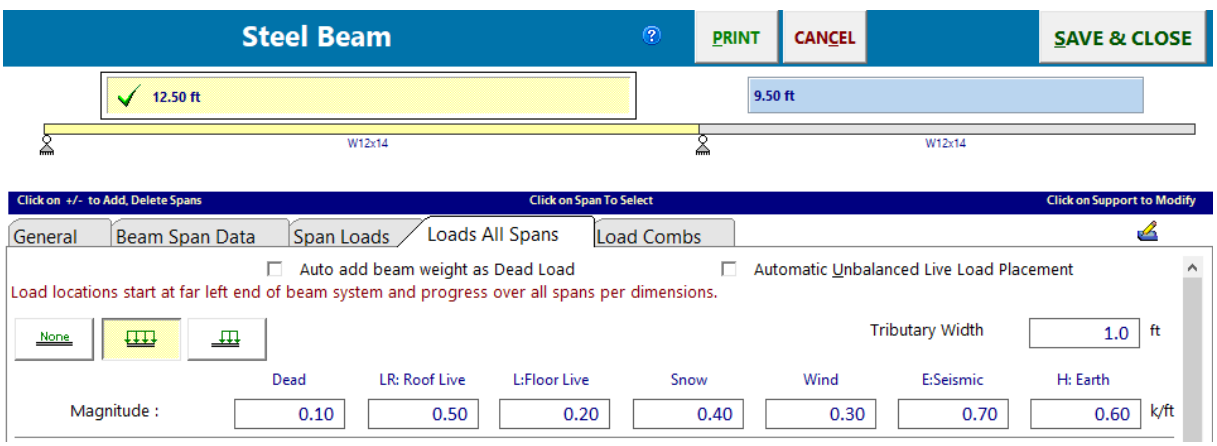


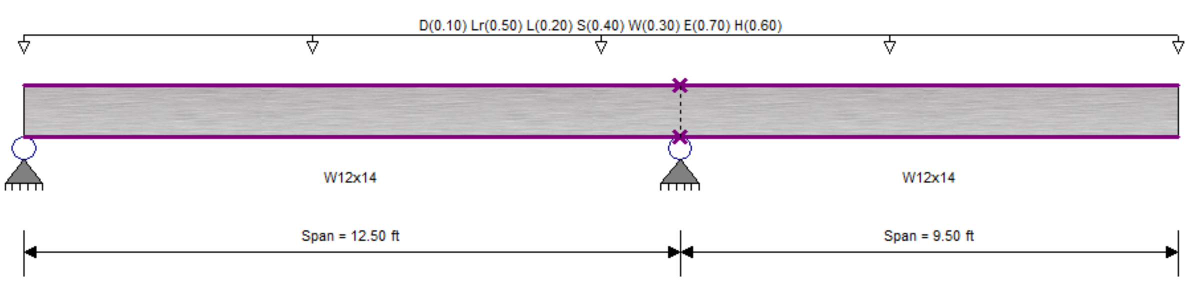




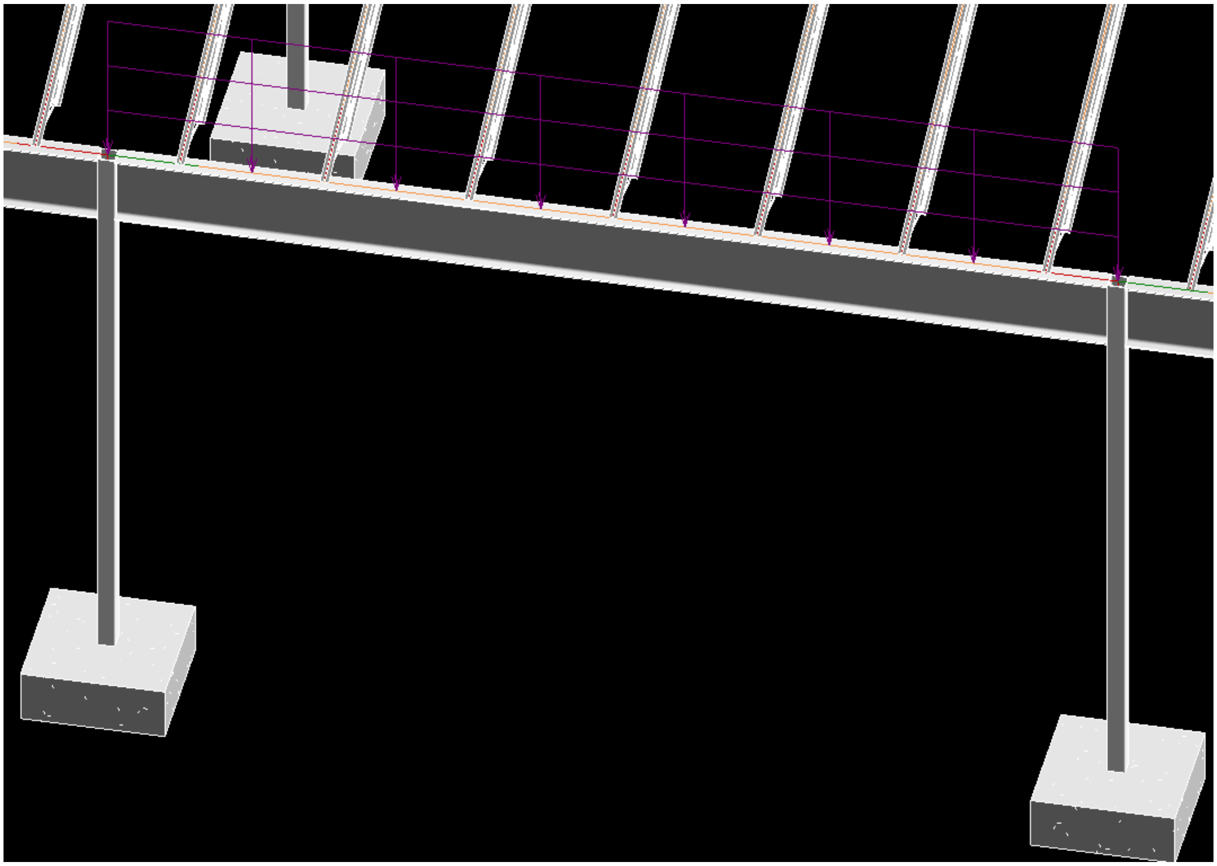
When multiple hosted load elements overlap with identical geometry but differing load cases, they are automatically consolidated to a single load item in the ENERCALC SEL interface.

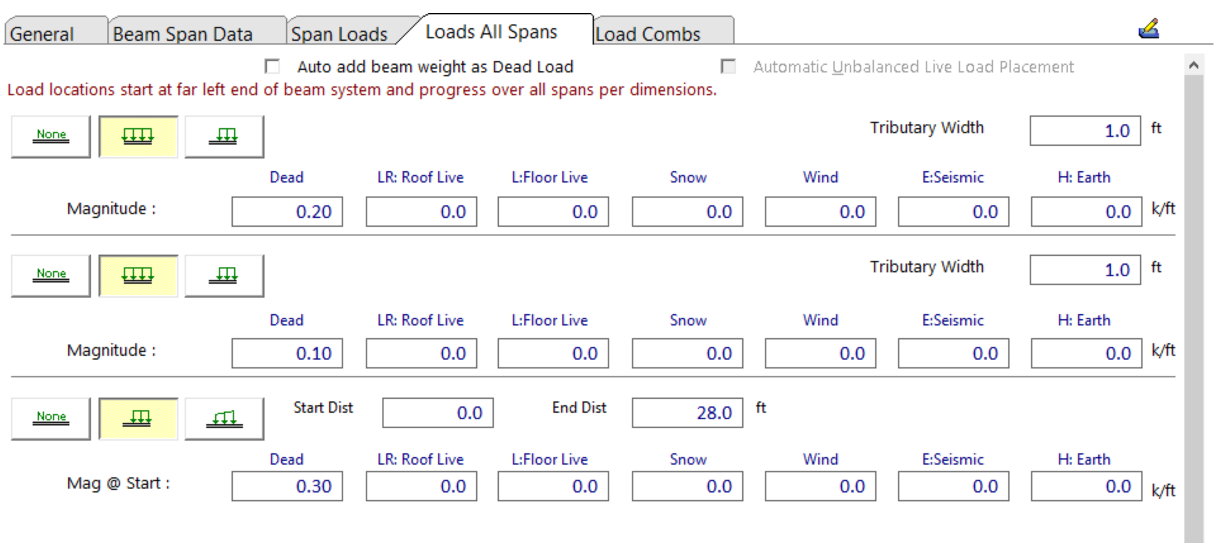

A
12' - 6"
9' - 6"
0.6 k/ft (H)
0.4 k/ft (S)
0.2 k/ft (L)
0.7 k/ft (E)
0.5 k/ft (LR)
0.3 k/ft (W)
0.10 k/ft (D)
W12X14

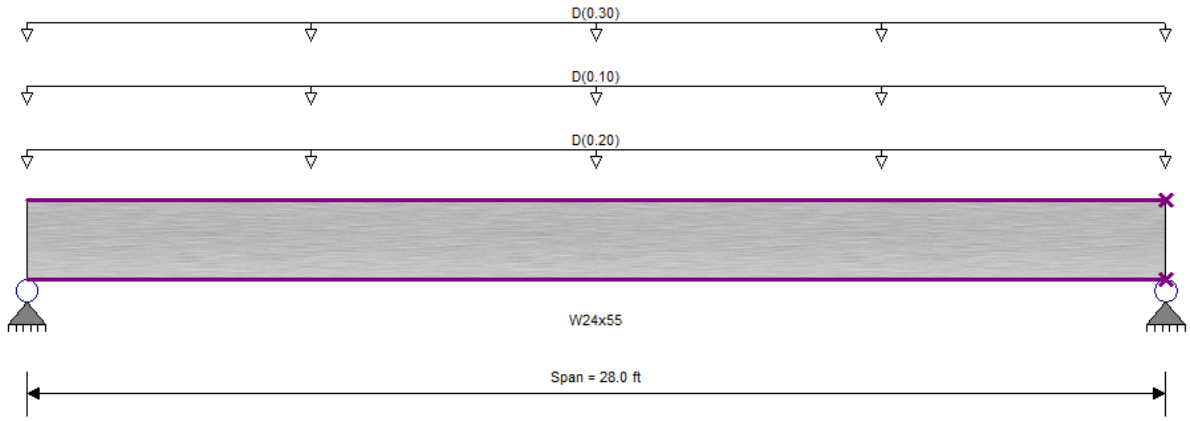

Steel Beam
PRINT
CANCEL
SAVE & CLOSE
12.50 ft
9.50 ft
W12x14
W12x14
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Auto add beam weight as Dead Load
Automatic Unbalanced Live Load Placement
Load locations start at far left end of beam system and progress over all spans per dimensions.
None
Tributary Width
1.0
ft
Dead
LR: Roof Live
L:Floor Live
Snow
Wind
E:Seismic
H: Earth
Magnitude :
0.10
0.50
0.20
0.40
0.30
0.70
0.60
k/ft


D(0.10) Lr(0.50) L(0.20) S(0.40) W(0.30) E(0.70) H(0.60)
W12x14
W12x14
Span = 12.50 ft
Span = 9.50 ft

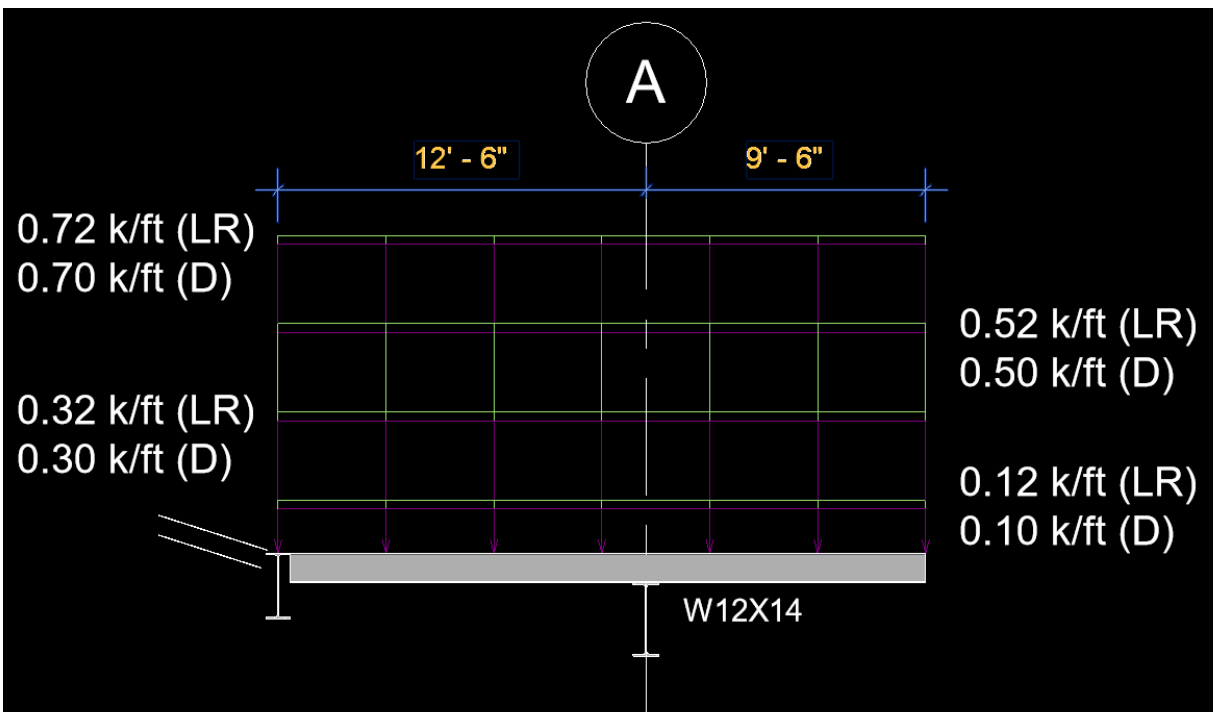
When multiple hosted load elements overlap with identical geometry and repetitive load cases, they are not consolidated but instead will appear as individual load items in the ENERCALC SEL interface.

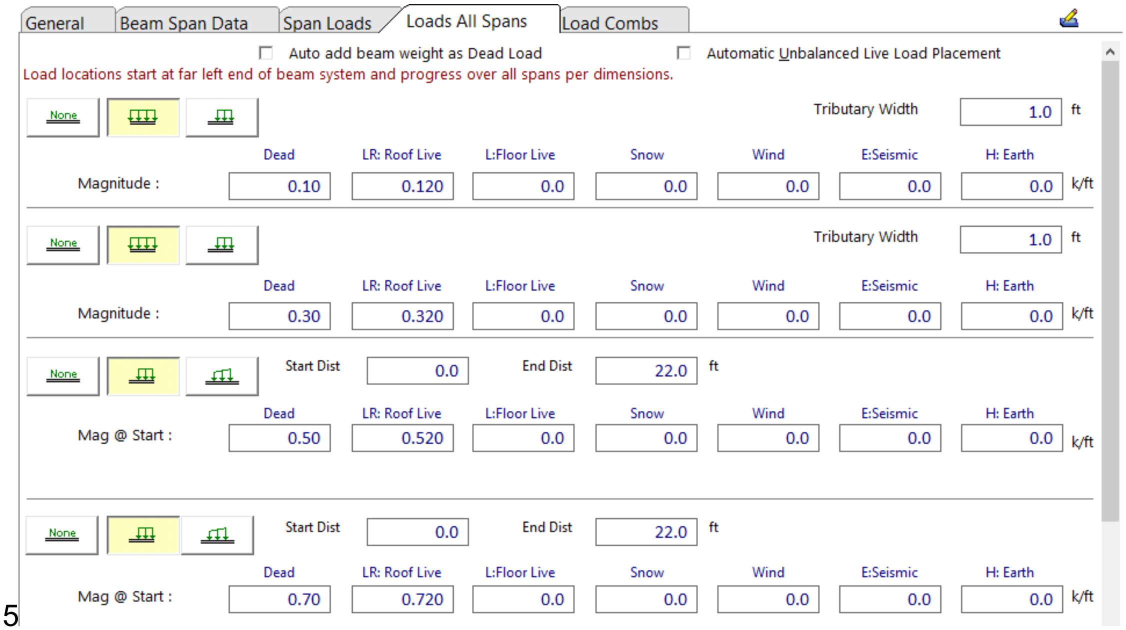




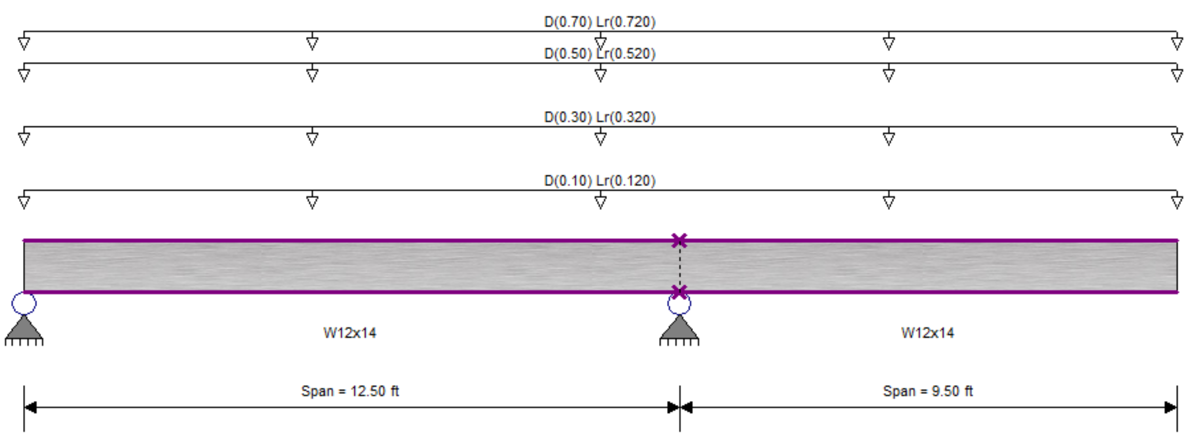


When multiple hosted load elements overlap with identical geometry and multiple repetitive load cases, the loads are consolidated into the lowest possible number of load items in the ENERCALC SEL interface.



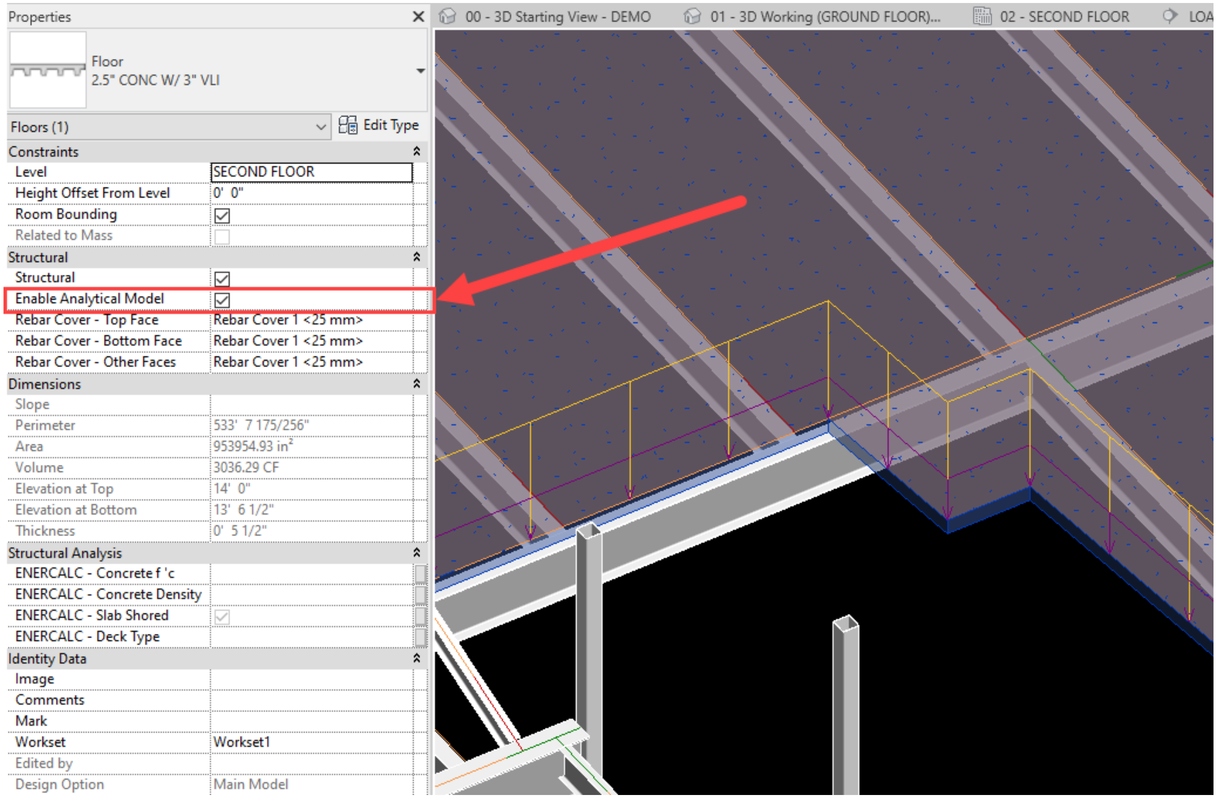


5



10.4.6 Launching With Revit Floors With Area Loads

ENERCALC for Revit supports the use of hosted area loads applied to structural floor elements when launching beam calculations. Similar to the previous discussion of hosted loads on beams, the use of hosted loads on floors requires that the "Enable Analytical Model" property be activated in the Properties pane for the floor.

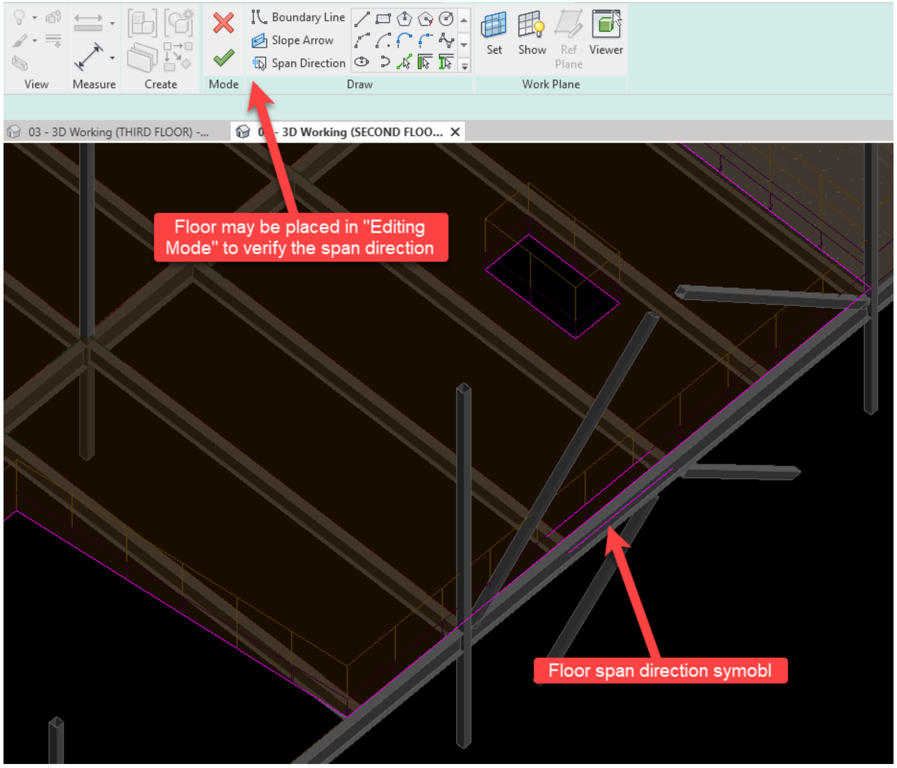


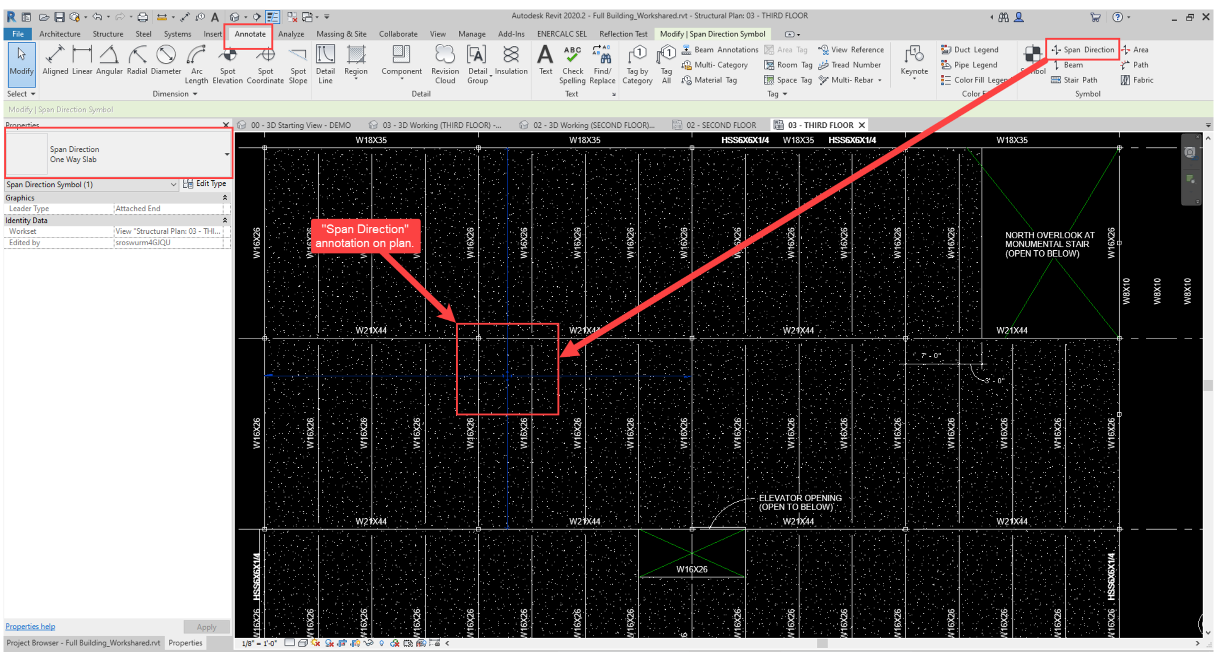
The following Revit element types are not eligible for this workflow:

- Non-structural floors
- Architectural roofs
- Structural floors loaded only with ***non-hosted*** area loads

10.4.6.1 Tributary Width Sampling

ENERCALC for Revit makes use of the floor geometry relative to the beam to be designed in order to map area loads into equivalent linear loads. This is accomplished by testing the beam at a number of locations along its length in order to compute the instantaneous tributary width at each. Tributary width testing is performed by searching on each side of the beam in a direction parallel to the deck span direction specified in Revit for the floor.





At any given point on the beam, tributary widths are defined by locating nearby constraints on each side of the beam. The following Revit element types are eligible for use in defining tributary width:

- Structural Framing as beams
- Structural Walls

- Structural Floor edges (i.e., openings or perimeters)

The following Revit element types are not eligible for use in defining tributary width:

- Structural Framing as vertical bracing
- Structural Framing as horizontal bracing
- Non-Structural Floor edges
- Structural Foundations
- Structural Walls with a non-bearing usage type
- Non-Structural Walls
- Any other non-structural element

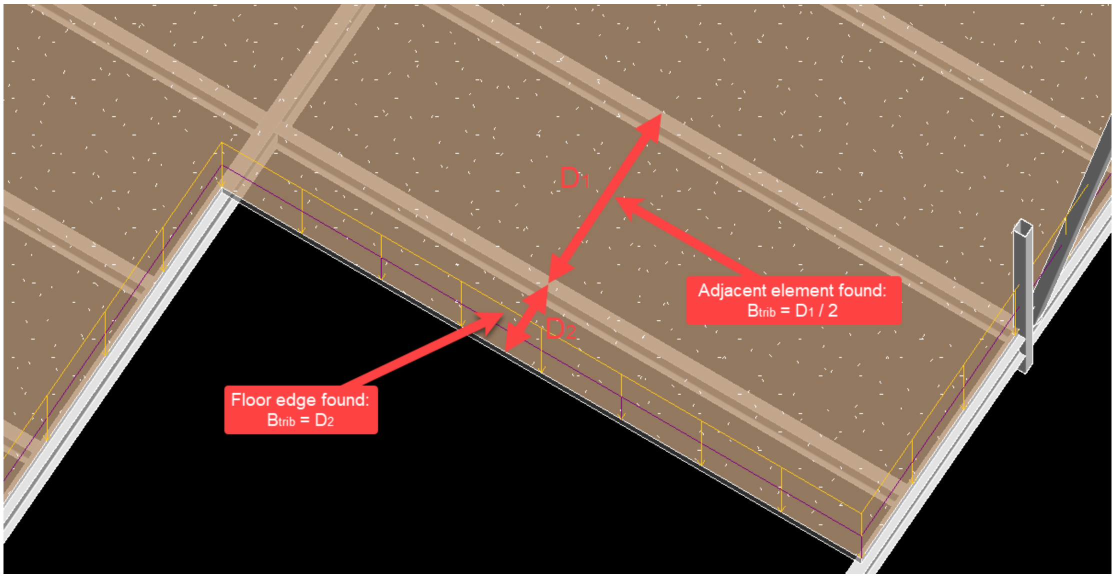
Once the nearby constraints are identified, the following rules dictate the tributary width at a given location on a beam:

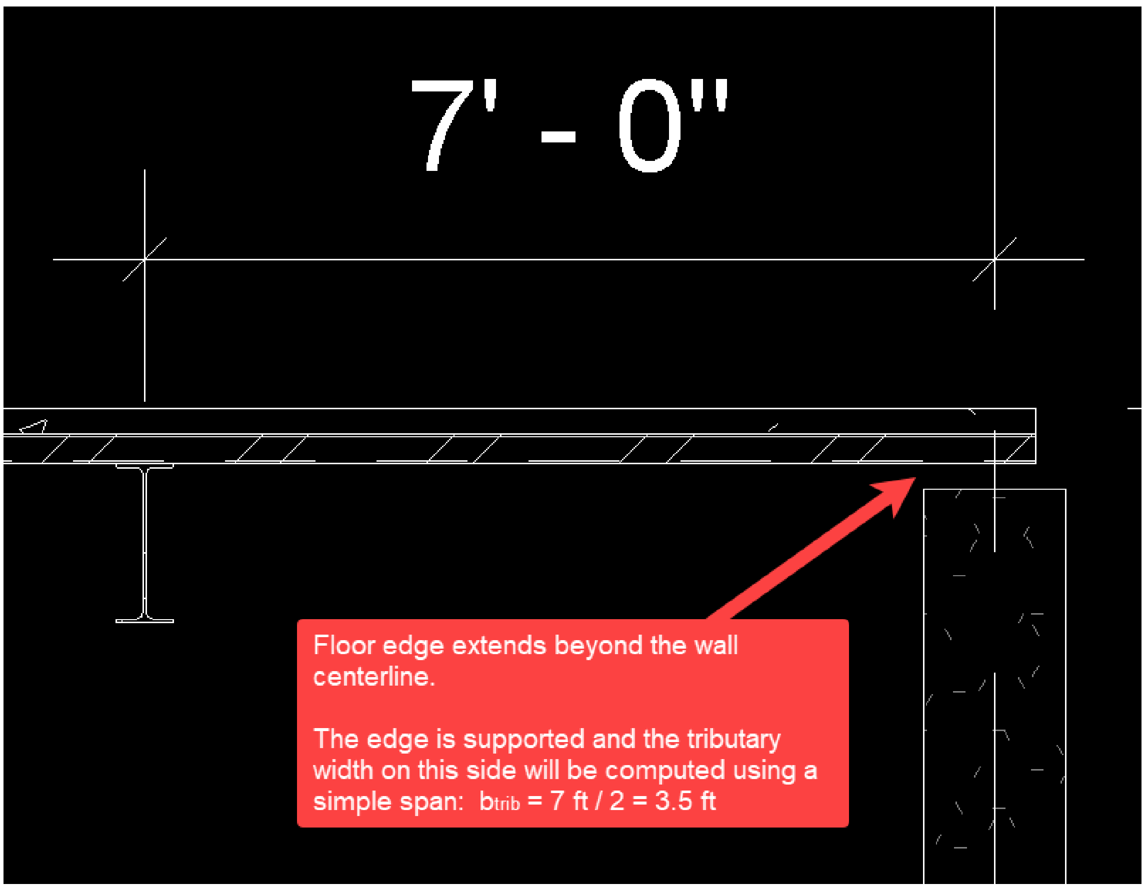
1. When an eligible adjacent element (structural beam or structural wall) is found: The tributary width in that direction is taken as half the distance from the designed beam to the nearby element (simple-span floor behavior is assumed).

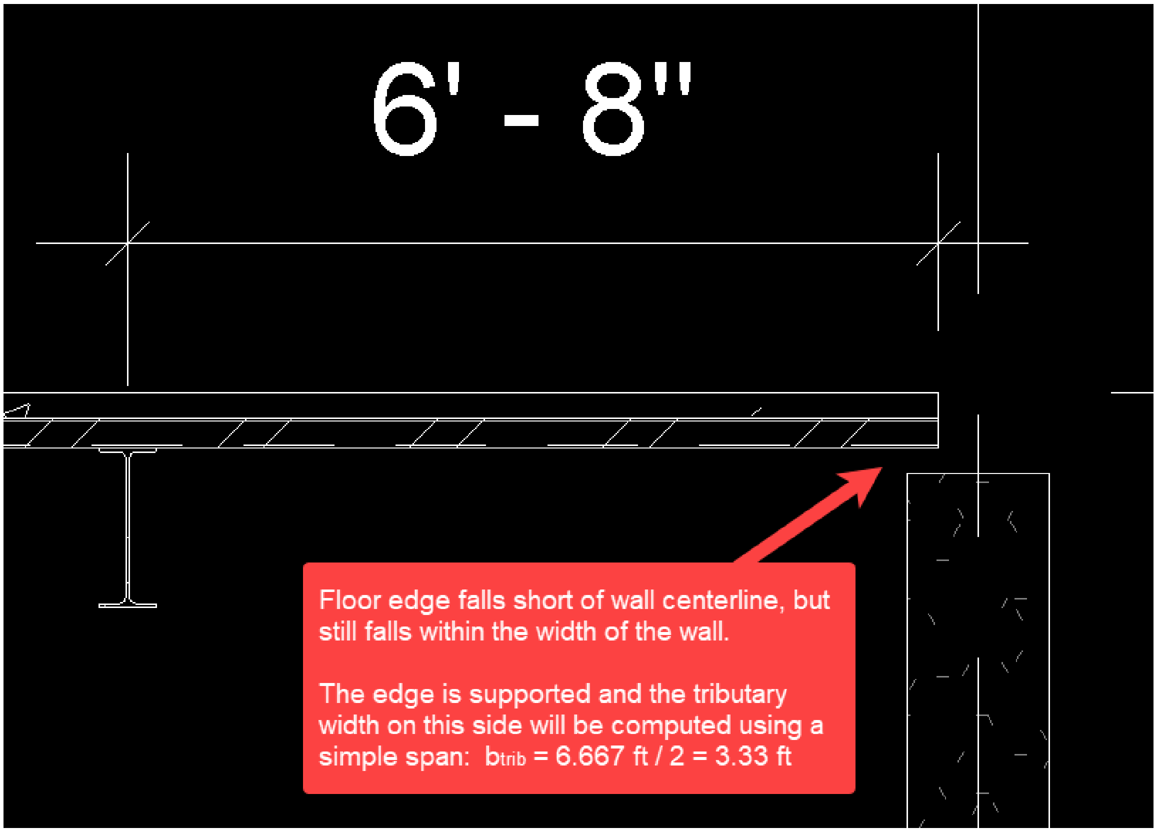
2. When a floor edge is found: The tributary width in that direction is taken as the full distance from the designed beam to the floor edge (indicating a cantilever floor).

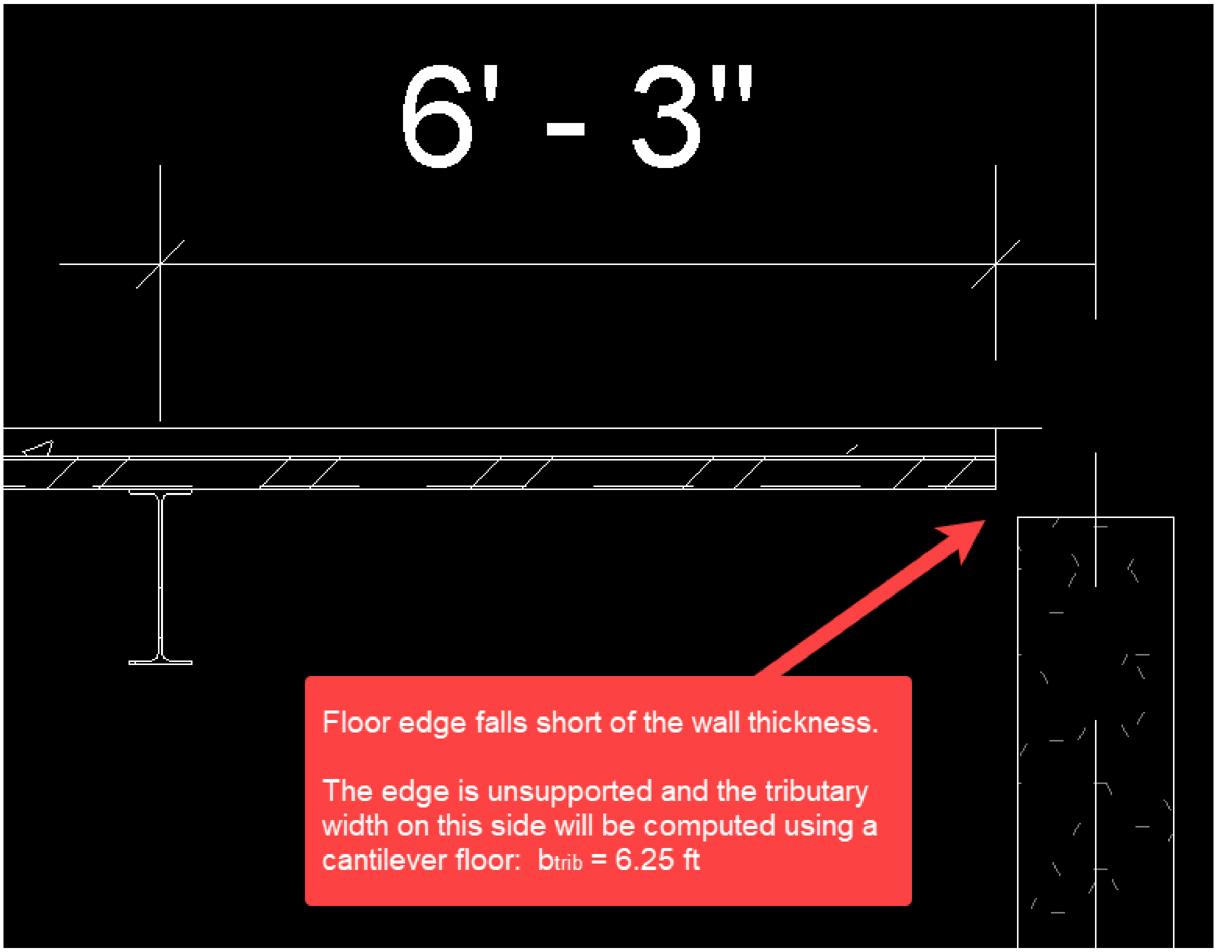
3. If both multiple edges and/or elements are found nearby, the nearest is taken as the limiting constraint for tributary width, and tributary width is computed according to 1) or 2) as applicable.

4. If a floor edge coincides closely with a beam or wall, the edge is automatically evaluated to see if it is supported or falls short, creating a cantilever condition. This check is performed using the thickness of the wall or width of the beam.


D1
D2
Adjacent element found:
Btrib = D1 / 2
Floor edge found:
Btrib = D2


7' - 0"
Floor edge extends beyond the wall centerline.
The edge is supported and the tributary width on this side will be computed using a simple span: btrib = 7 ft / 2 = 3.5 ft

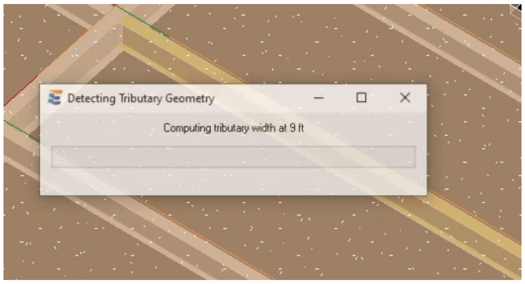

6' - 8"
Floor edge falls short of wall centerline, but still falls within the width of the wall.
The edge is supported and the tributary width on this side will be computed using a simple span: b_{trib} = 6.667 ft / 2 = 3.33 ft

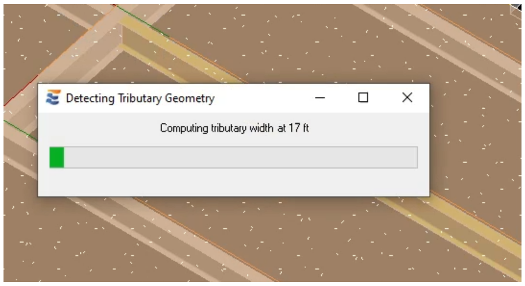


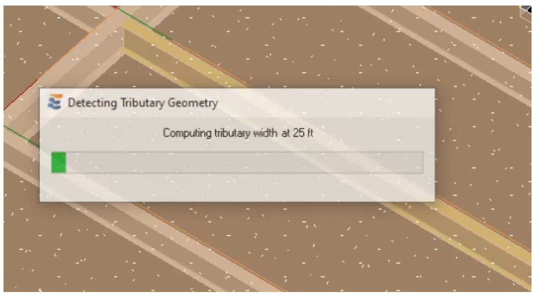
Determining both tributary width and area load is adequate to compute an instantaneous linear load magnitude at any given location. Doing so for multiple points along a beam results in a linear load with either constant or varying magnitude. The location and spacing of tributary width sampling is discussed in the next section.

There is no load contributed to a beam by a floor whose span direction is found to be parallel to the beam. Due to this provision, it is always expected that elements such as girders or miscellaneous framing (i.e., framed opening beams) oriented parallel to the floor will reflect no line loads mapped from floor area loads when the calculation in displayed in ENERCALC SEL.

When the beam calculation is launched, the user will notice that ENERCALC for Revit automatically performs tributary width sampling when a floor element is present on the beam. This short intermediate step during launch is marked by a progress bar showing the progress of sampling along the length of the beam. In most cases, the progress bar may not be visible due to very fast execution of the sampling.





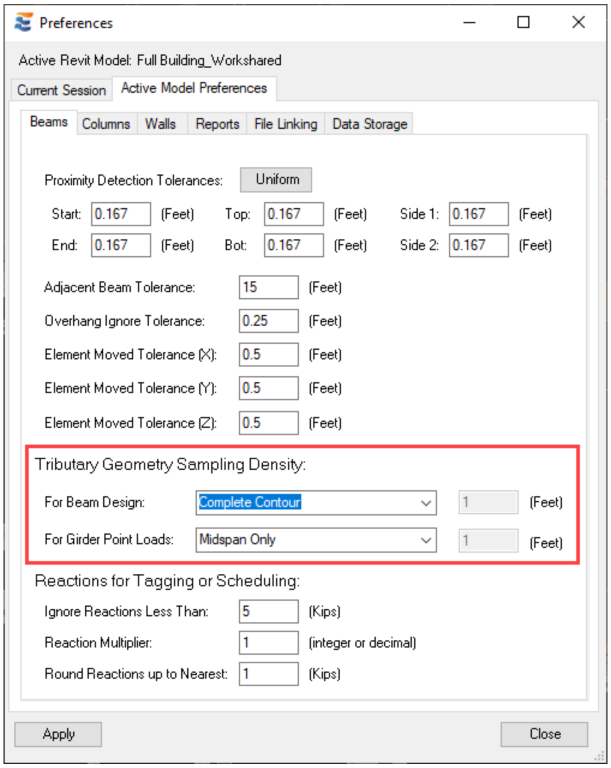


When the calculation appears in the ENERCALC SEL interface, the equivalent linear loads derived from the area loads and tributary geometry will be automatically displayed. Although some special cases may arise, the general approach is that fully uniform loads will be displayed in the “Loads All Spans” tab, while segmented loads, linear varying loads, and loads with redundant load cases will be displayed in the “Span Loads” tab. Loads generated from Revit area loads will be displayed using the area load magnitudes and tributary widths detected in the Revit model.

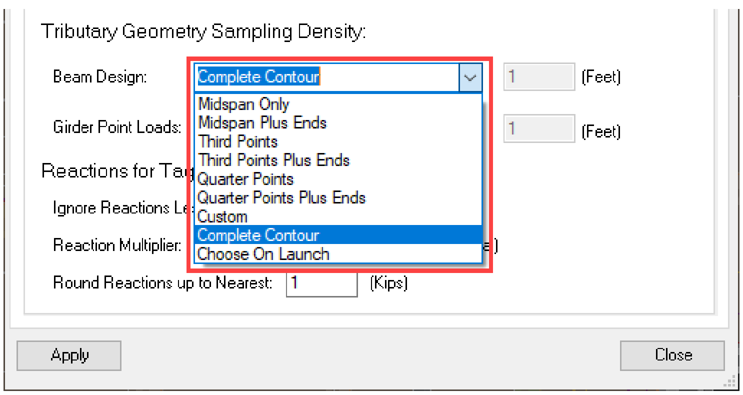
The implementation of area loads for various conditions is discussed in subsequent sections.

10.4.6.2 Sampling Density Controls

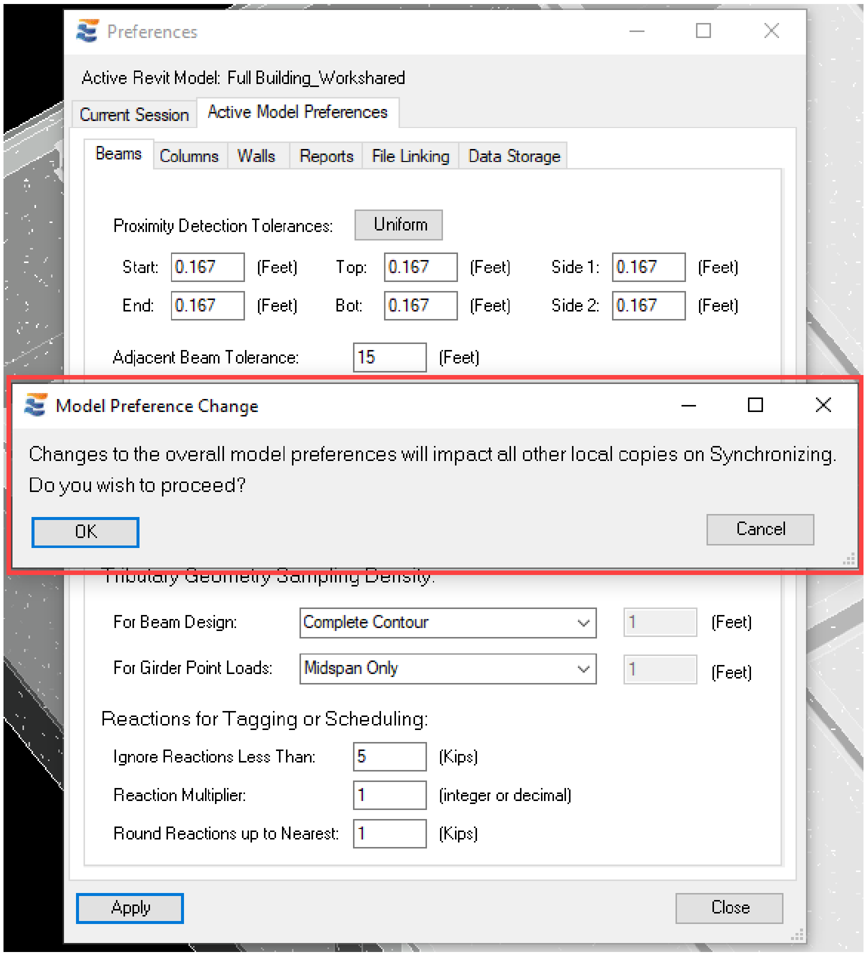
In order to give the user control over this testing and calculation process, the ENERCALC for Revit Preferences menu provides settings in Preferences > Beams > Tributary Geometry Sampling Density.



The default setting is “Complete Contour”, with additional options as follows:



Note: The beam tributary width sampling density is a project-level change that applies to all team members who create beam calculations in the project. Modifying this property in a workshared Revit project will result in a notification.

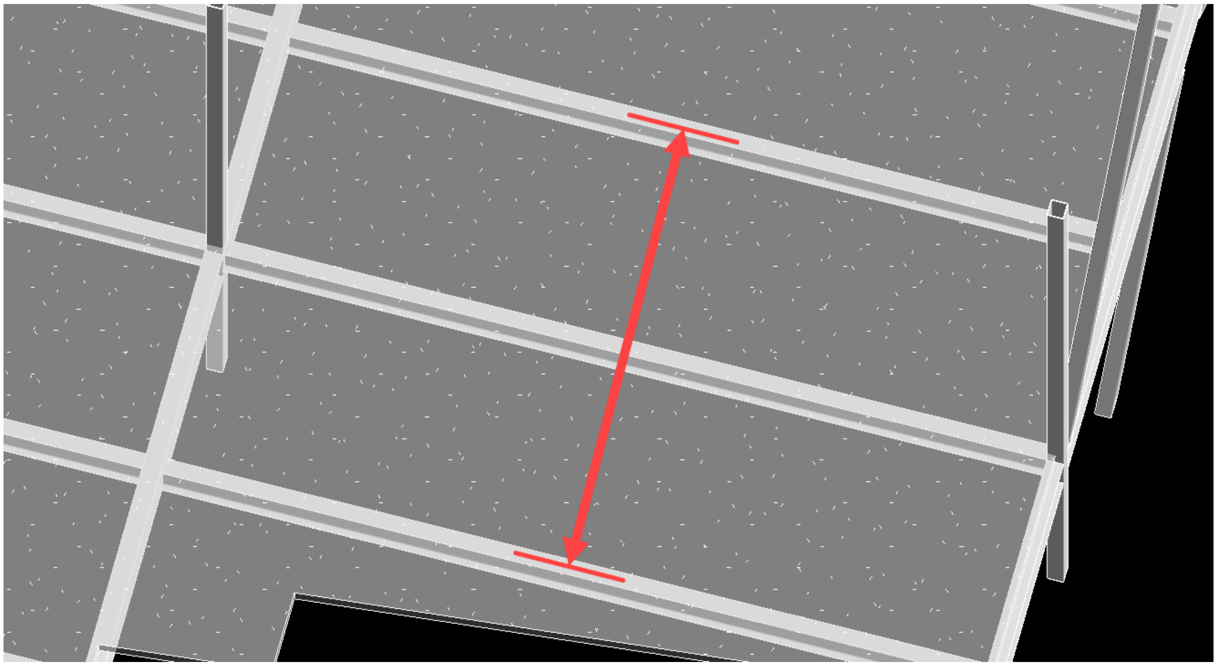


Since the setting is stored in the database of the Revit project, it is necessary for members of the design team to synchronize in order to obtain the most recent setting. If the setting is changed and team members do not synchronize their local Revit files with the central file, they will be using whatever sampling density was previously set.

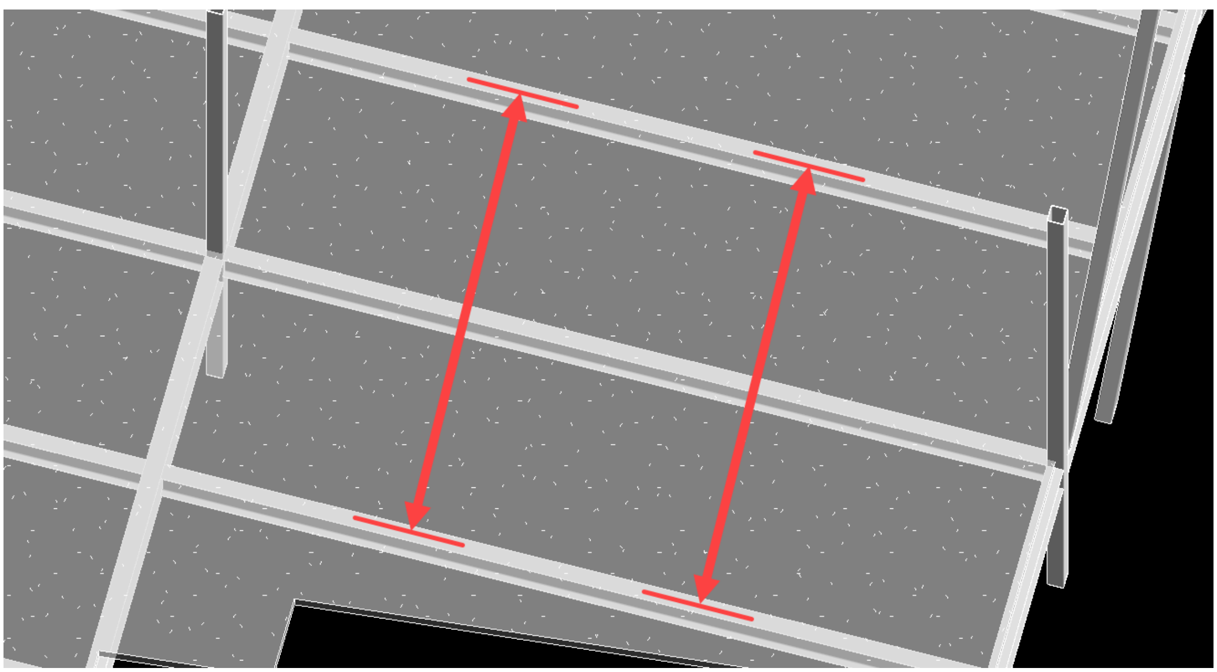
As discussed later in greater detail, the changing of this overall Preference setting does not alter the behavior of existing beam calculations. Instead, each individual calculation will automatically retain the density that was used during its previous launch. This information stored individually on each calculation takes precedence over the Preference setting. The Preference setting is used by default on all newly created calculations.

The tributary sampling performed by each option is as follows:

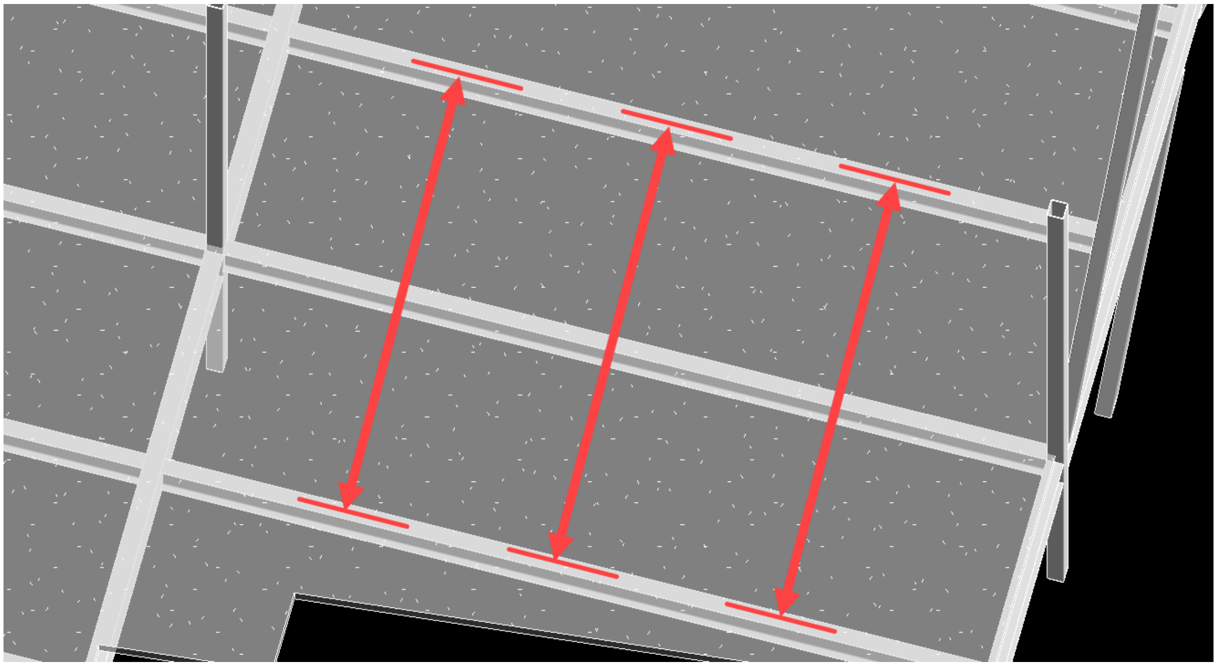
- “Midspan Only”: The beam’s tributary width is calculated at one location only, at the exact mid-length of the beam (L / 2).



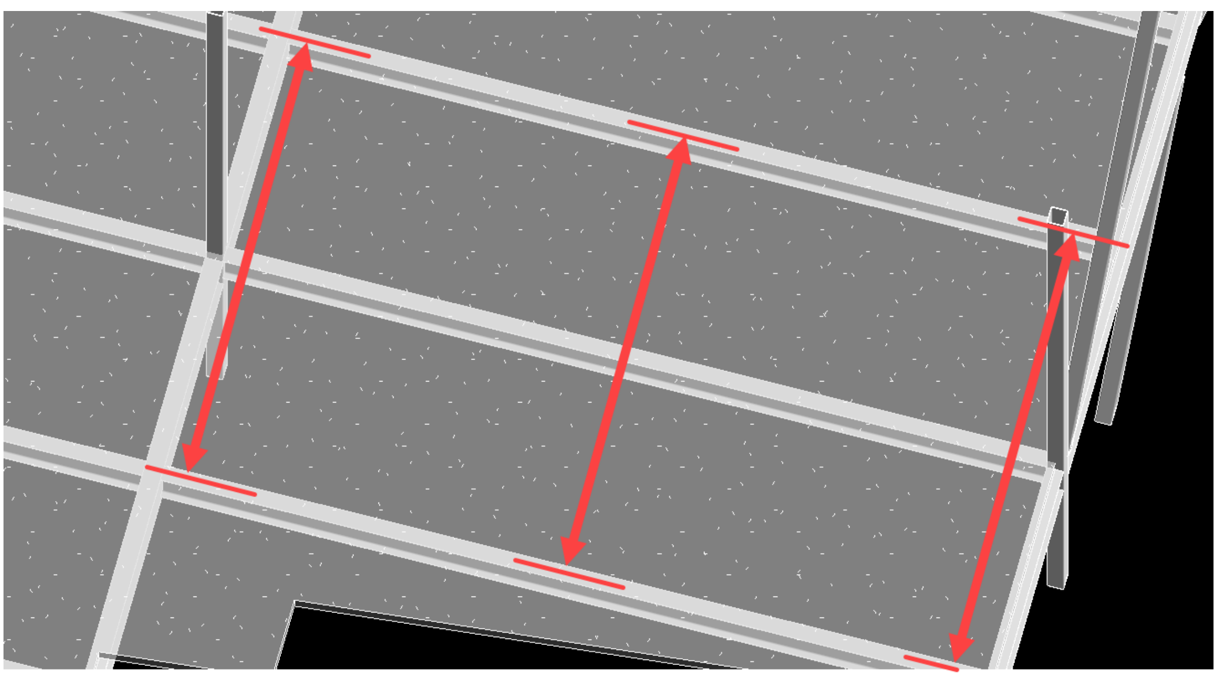
- “Third Points”: The beam’s tributary width is calculated at 2 locations, at the third points of the beam (L / 3 and 2L / 3).



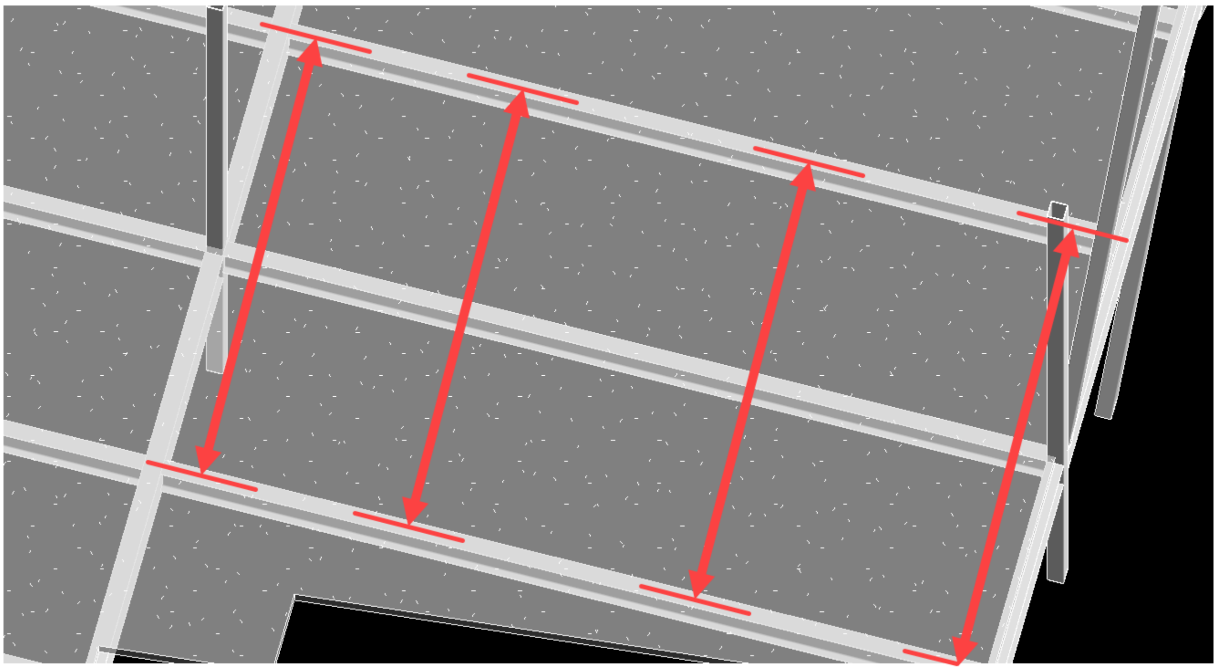
- “Quarter Points”: The beam’s tributary width is calculated at 3 locations, at the quarter points of the beam (L / 4, 2L / 4, 3L / 4).



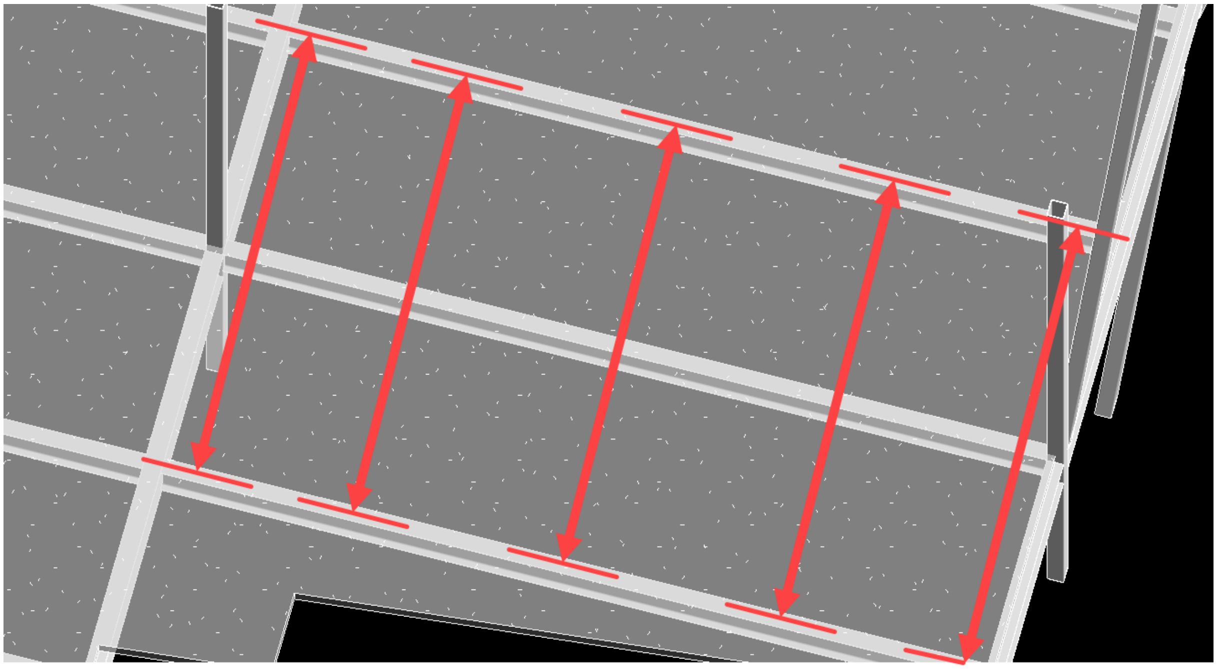
- “Midspan Plus Ends”: The beam’s tributary width is calculated at 3 locations, at the exact mid-length of the beam (L / 2) and 6” from each end point. The sampling is performed a short distance from the extreme end point to avoid irregular conditions that complicate the sampling.



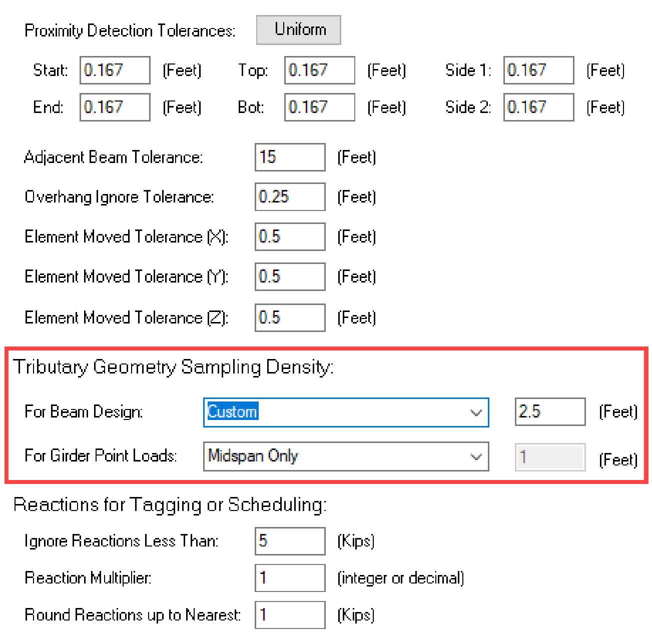
- “Third Points Plus Ends”: The beam’s tributary width is calculated at 4 locations, at the third points of the beam (L / 3 and 2L / 3) and 6” from each end point. The sampling is performed a short distance from the extreme end point to avoid irregular conditions that complicate the sampling.



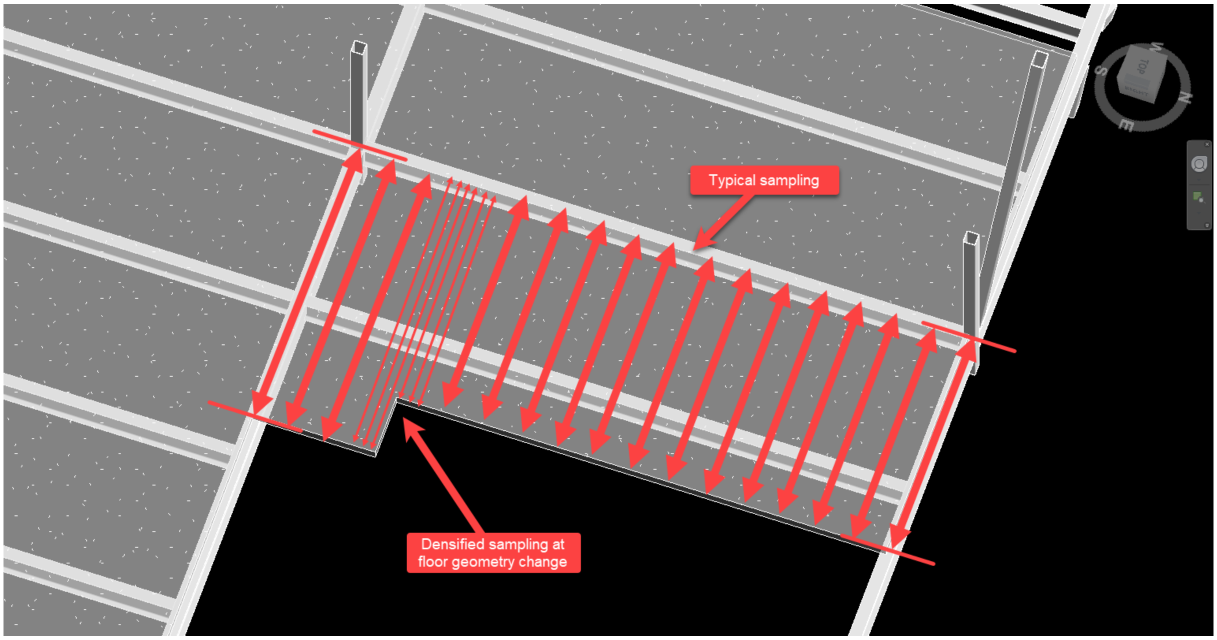
- “Quarter Points Plus Ends”: The beam’s tributary width is calculated at 5 locations, at the quarter points of the beam (L / 4, 2L / 4, 3L / 4) and 6” from each end point. The sampling is performed a short distance from the extreme end point to avoid irregular conditions that complicate the sampling.



- "Custom": When the drop-down is toggled to "Custom", the user will have access to the input box to specify an arbitrary spacing for tributary width sampling locations. This will result in sample points 6" from each end point and at the specified spacing.



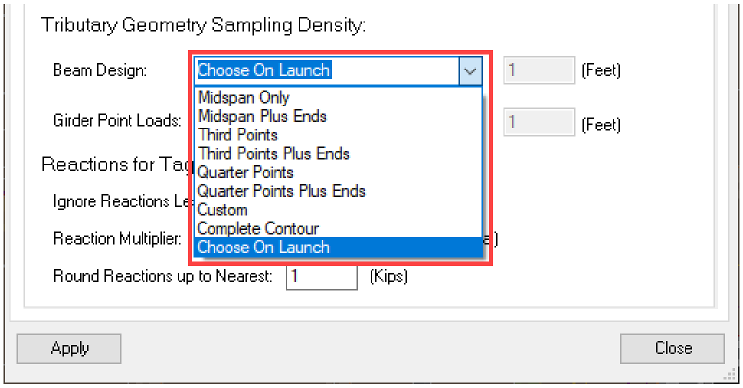
- "Complete Contour": This setting will result in an autonomous process that dynamically senses varying conditions along the beam and produces an appropriately dense sampling contour. When generating a complete contour, the beam is tested at the extreme end points for a higher degree of accuracy, rather than using a 6” standoff. The beam will be automatically tested at 1’-0” on-center, and then densified at finer spacings in any location where tributary width changes and floor discontinuities are discovered.



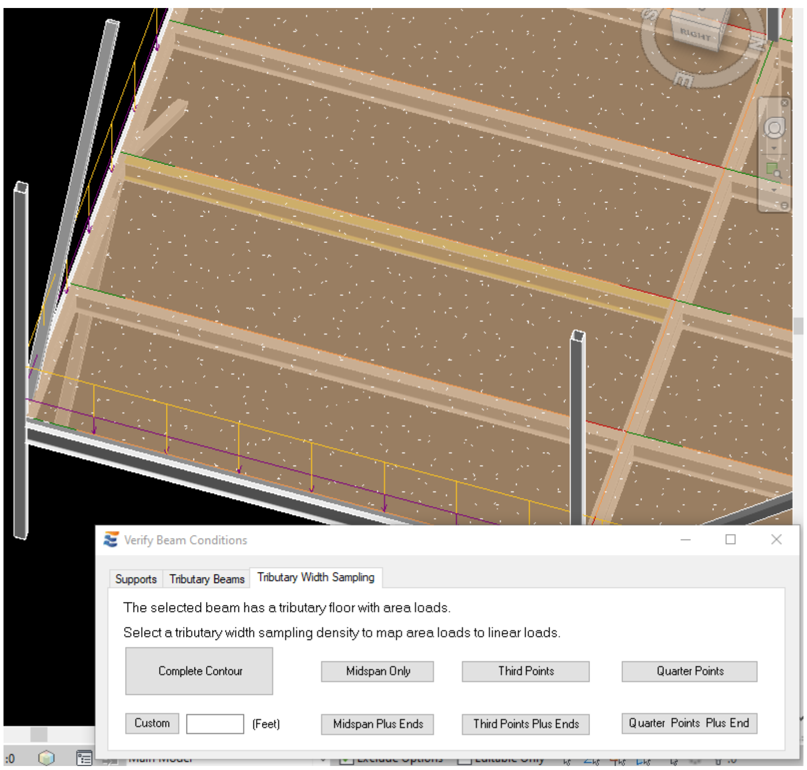
The tributary sampling options described above are arranged in order from lowest density to highest density. This rise in sampling density also corresponds to marginally slower computation times when launching a beam calculation. The lowest density options execute fastest, but will produce less detailed tributary width contours. While highly efficient for beams with uniform tributary geometry, lower density settings are problematic when a beam manifests variations along its length, including floor openings, floor edge changes, and beam framing changes.

Beams with any form of observable variation in tributary geometry to be captured in the design are best served by using the “Complete Contour” setting. Discrete sampling modes often prove useful in situations where the user desires to perform a simple calculation that does not depict the full complexity of a Revit model, or when engineering judgment dictates that a lower sampling density will yield satisfactory results. For example, sampling at midspan only allows the user to ignore miscellaneous slab variations, such as small mechanical piping openings. Specific use Revit model cases are discussed in subsequent sections.

Rather than directly selecting one of the above sampling densities from the Preferences menu, the user alternatively may opt to “Choose On Launch” for each individual calculation. This allows for generating many different beam calculations sampled at different densities without repeatedly modifying the overall Preferences setting for the project.

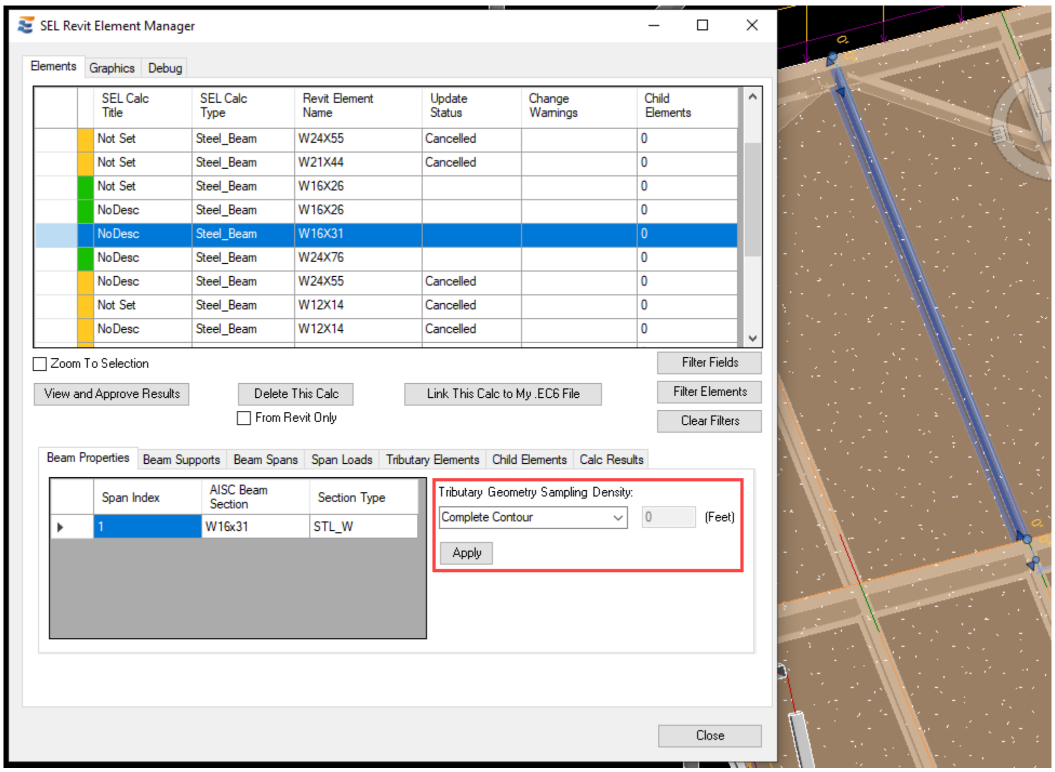


When the Preference is set to “Choose On Launch”, an additional approval tab will be added to the calculation launch window for the user to select a sampling density where applicable. This extra tab will not be included when the beam being launched does not have area-loaded floors present in the Revit model.

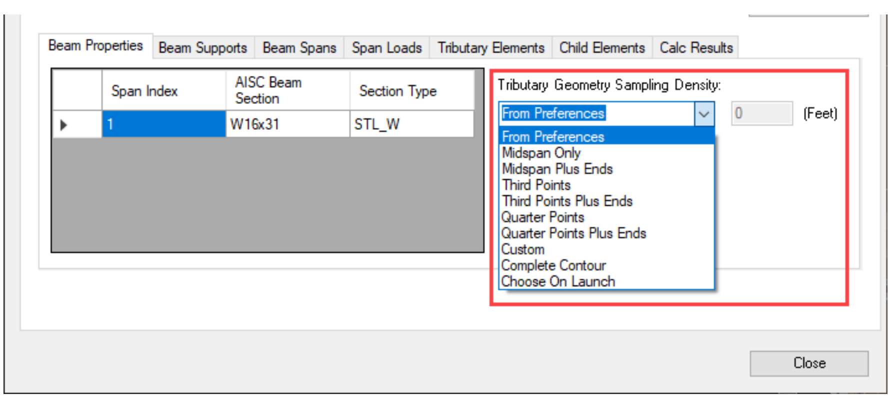


After a sampling density is selected (either via Preferences or during launch), ENERCALC for Revit will test the beam at an array of locations corresponding to the chosen density and construct a tributary width contour. Once the tributary width contour has been computed, the individual points will be evaluated to determine if they constitute a uniform load, a linear varying load, or an irregular load and will be mapped to the ENERCALC SEL calculation accordingly.

In order to ensure consistent calculation results and limit repetitive mandatory interaction with approval steps during launches, ENERCALC for Revit automatically stores the most recent sampling density selected for each individual beam. As a result, users will **NOT** be forcibly prompted to select sampling density more than once on a given beam calculation. Future launches of the calculation will use the same sampling density used on the previous launch. If at any point the user wishes to view or change the sampling density for a particular beam, this may be done via the Element Manager window. When a beam calculation is selected in the Element Manager, detailed information automatically populates in the lower half of the window.



From this drop-down menu, the user may manually select any of the alternate density modes discussed above, or may choose to revert the beam to whichever option is set in the Preferences menu.



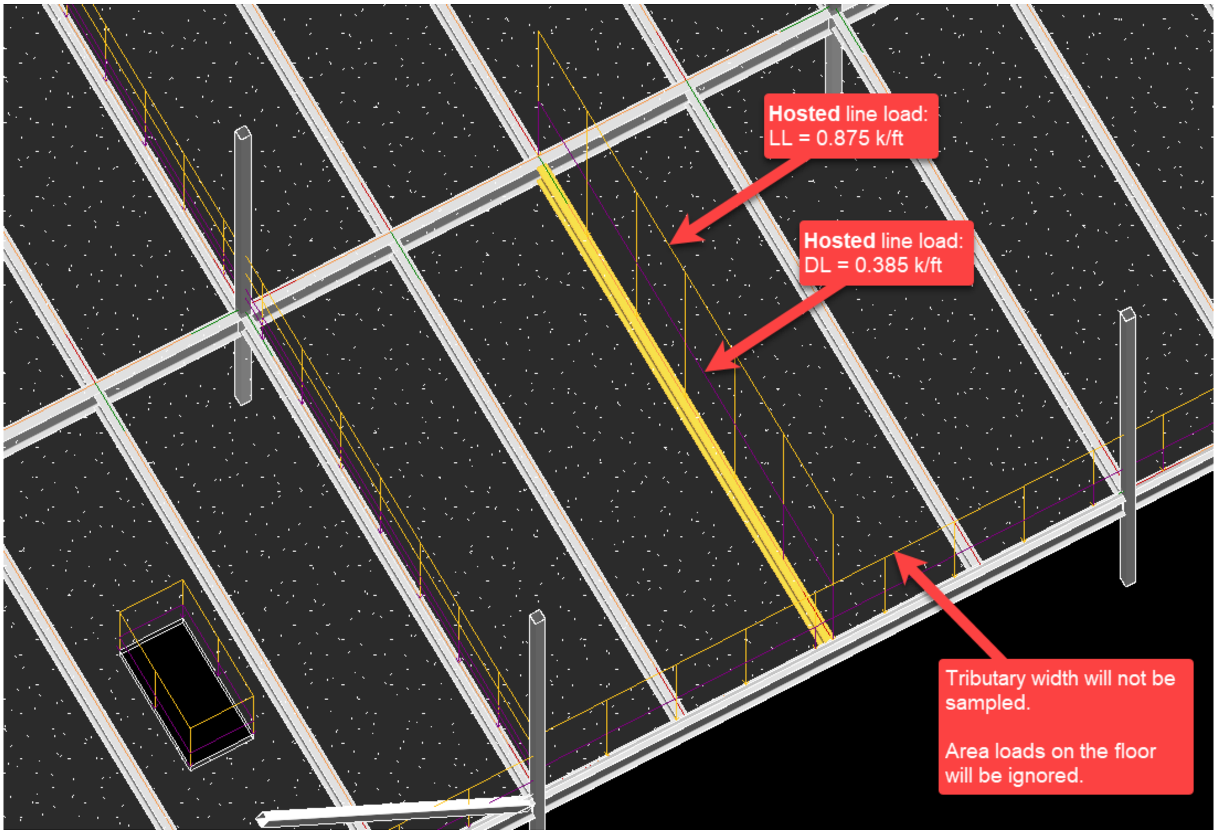
Choosing an option from this menu and clicking the “Apply” button will cause the selected sampling mode to be used on the next launch of this particular beam calculation. Setting this option to “Choose On Launch” or setting to “From Preferences” when the overall preference is set to “Choose On Launch” will cause the user to again be prompted to select manually on the next launch of this beam calculation.

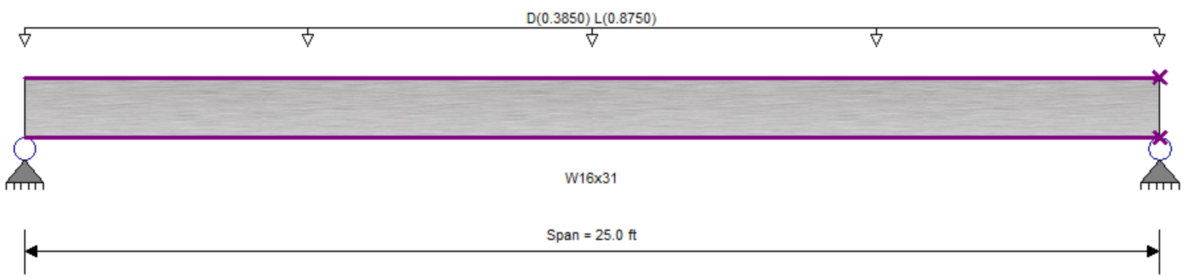
10.4.6.3 Superposition With Other Loads

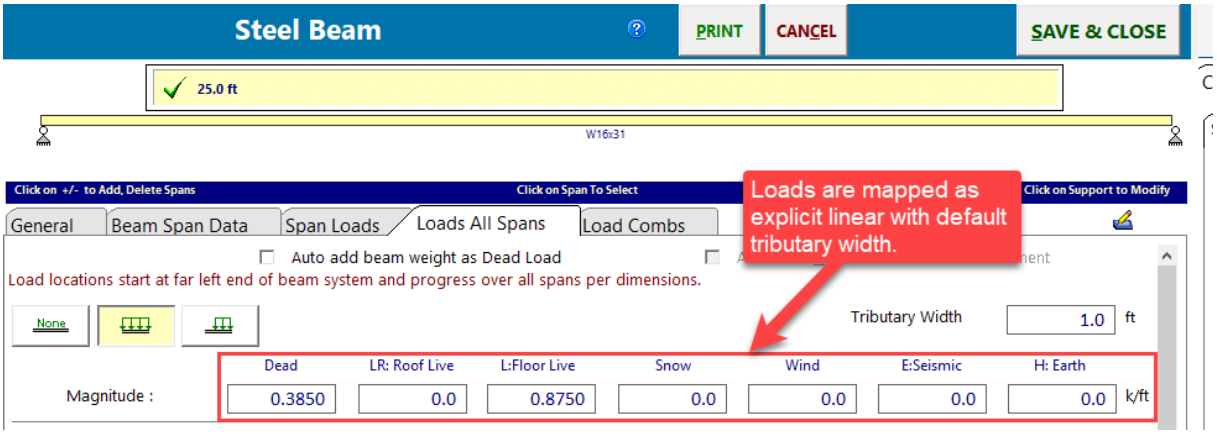
ENERCALC for Revit allows the use of linear loads and point loads when hosted area loads are present. These other loads may be additive with the area load effects, or may be used to override the area load effects if desired. The following rules govern the superposition:

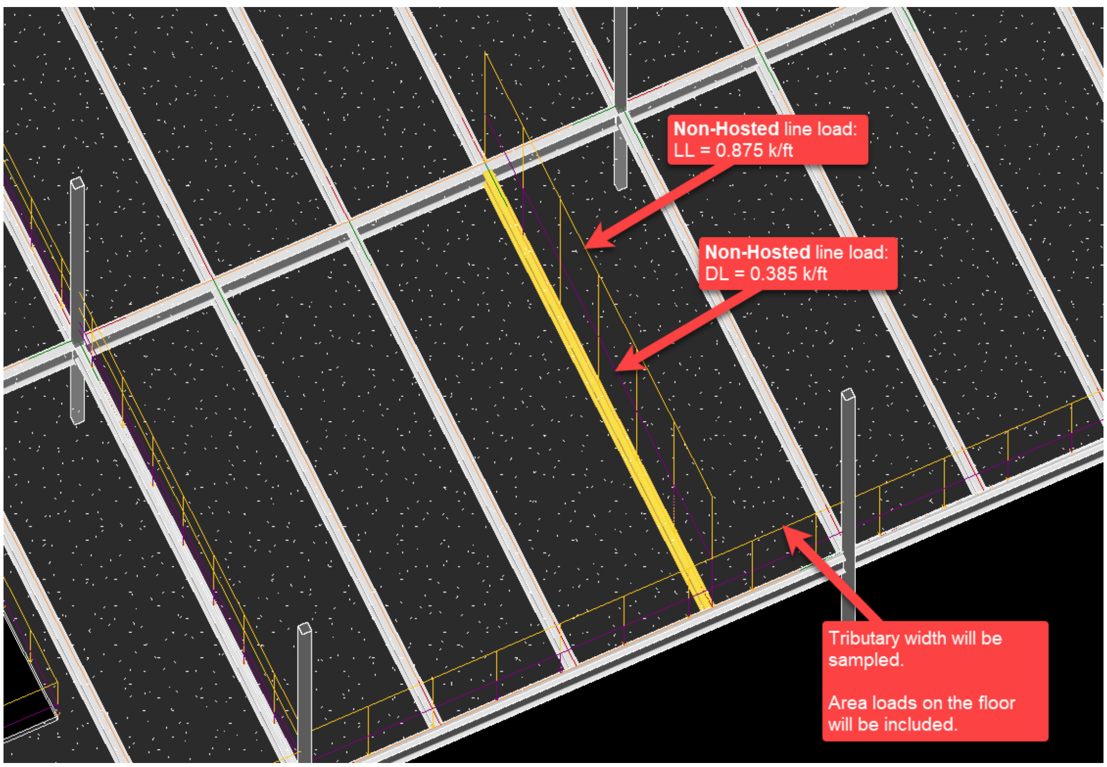
Other Load Found:		Load Mapping to SEL:	
Type:	**Hosting:**	**Tributary Width Sampled?**	**Loads Additive?**
Point Load	Non-Hosted	Yes	Yes
Point Load	Hosted	Yes	Yes
Point Load	Linked Load	Yes	Yes
Line Load	Non-Hosted	Yes	Yes
Line Load	Hosted	No	No

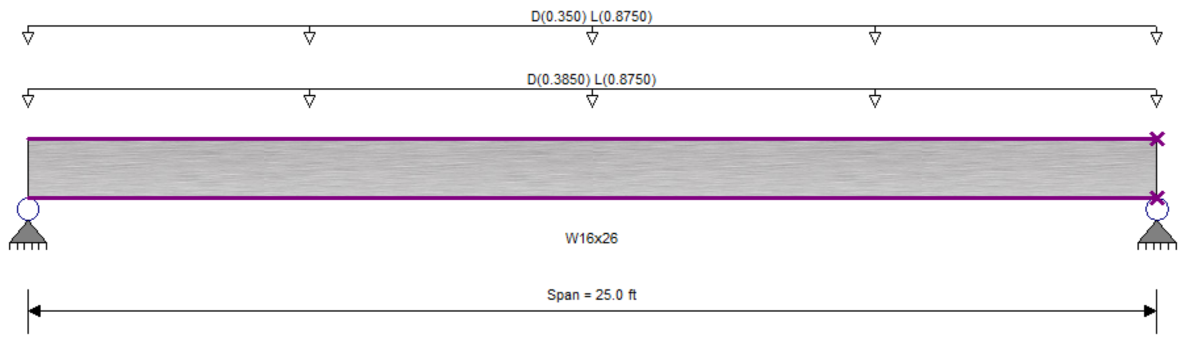
As indicated above, the presence of a Hosted line load acts as a “master override”. Placing Hosted loads with explicit magnitudes allows the user to bypass any force effects from the floor area loads. This makes it possible to achieve a very specific alternate loading for design on a single beam without being forced to modify physical geometry or the overall loading of the floor system. All loads of other types are mapped directly to the ENERCALC SEL calculation in addition to the floor area loads and are superimposed.


Hosted line load:
LL = 0.875 k/ft
Hosted line load:
DL = 0.385 k/ft
Tributary width will not be sampled.
Area loads on the floor will be ignored.

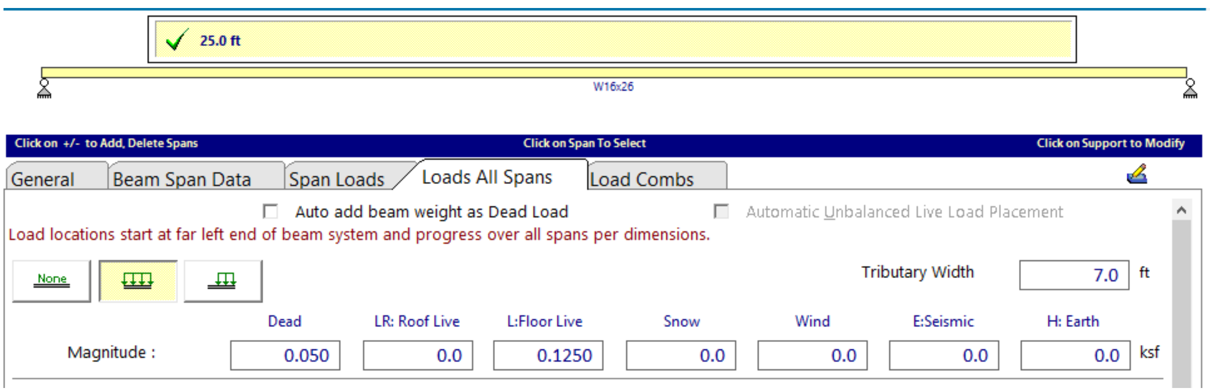

D(0.3850) L(0.8750)
W16x31
Span = 25.0 ft


Steel Beam
PRINT
CANCEL
SAVE & CLOSE
25.0 ft
W16x31
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
Loads are mapped as explicit linear with default tributary width.
General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Auto add beam weight as Dead Load
Load locations start at far left end of beam system and progress over all spans per dimensions.
None
Tributary Width
1.0
ft
Dead
LR: Roof Live
L:Floor Live
Snow
Wind
E:Seismic
H: Earth
Magnitude :
0.3850
0.0
0.8750
0.0
0.0
0.0
0.0
k/ft

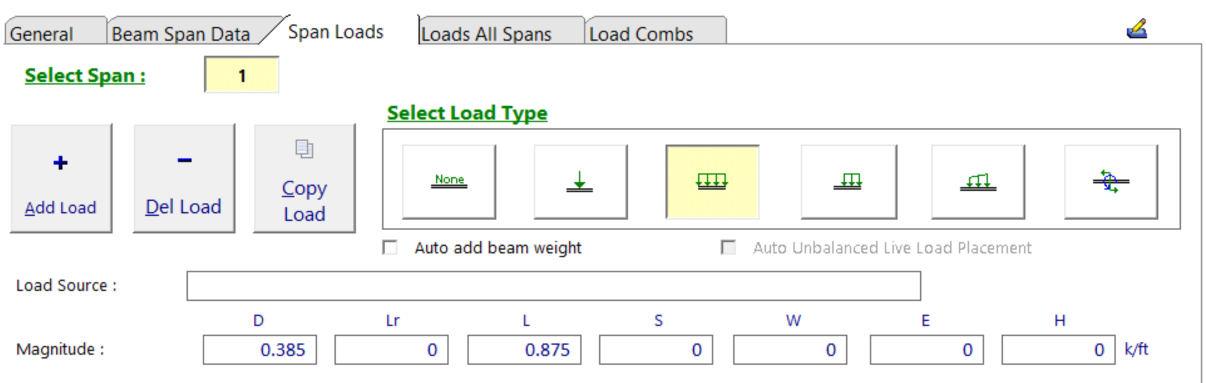

Non-Hosted line load: LL = 0.875 k/ft
Non-Hosted line load: DL = 0.385 k/ft
Tributary width will be sampled.
Area loads on the floor will be included.



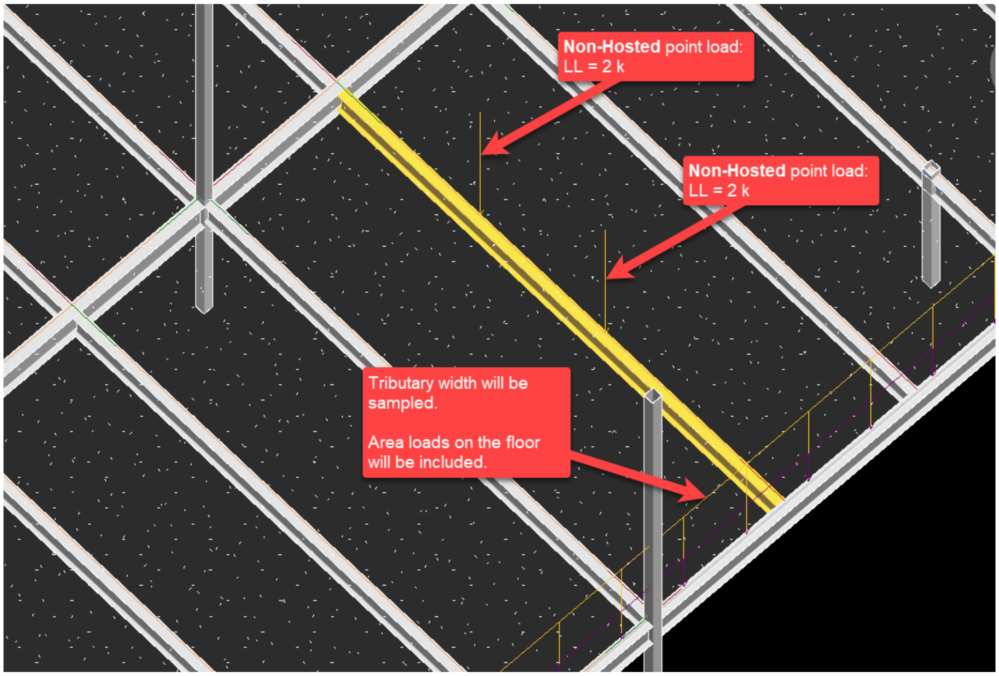
D(0.350) L(0.8750)
D(0.3850) L(0.8750)
W16x26
Span = 25.0 ft

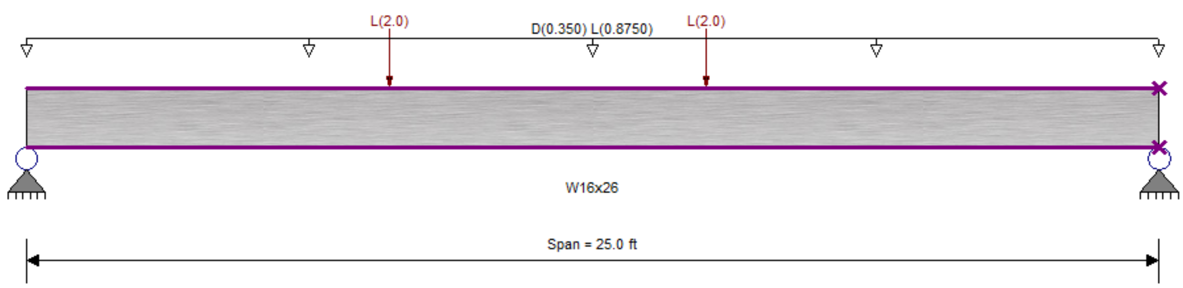


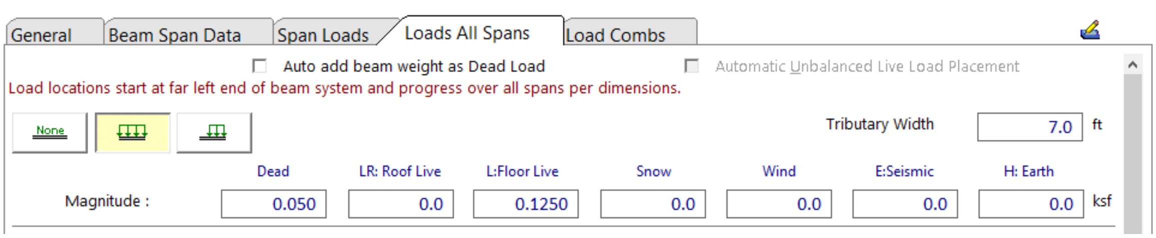
25.0 ft
W16x26
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Auto add beam weight as Dead Load
Automatic Unbalanced Live Load Placement
Load locations start at far left end of beam system and progress over all spans per dimensions.
None
Tributary Width
7.0
ft
Dead
LR: Roof Live
L:Floor Live
Snow
Wind
E:Seismic
H: Earth
Magnitude :
0.050
0.0
0.1250
0.0
0.0
0.0
0.0
ksf

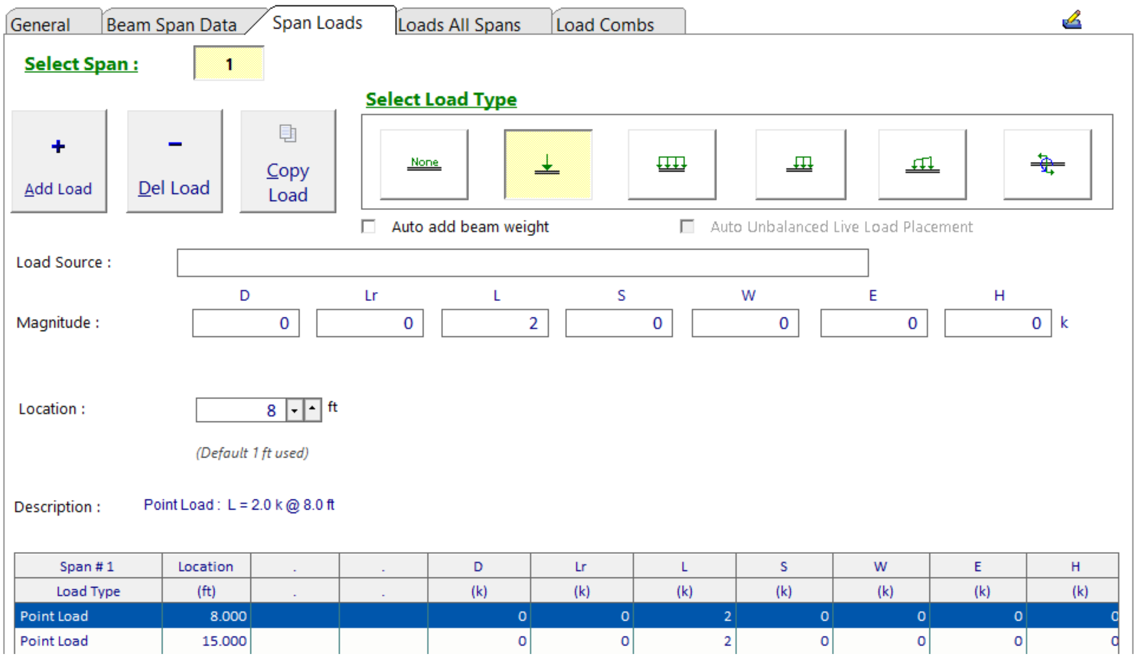


General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Select Span :
1
Select Load Type
Add Load
Del Load
Copy Load
None
Auto add beam weight
Auto Unbalanced Live Load Placement
Load Source :
D
Lr
L
S
W
E
H
Magnitude :
0.385
0
0.875
0
0
0
0
k/ft


Non-Hosted point load:
LL = 2 k
Non-Hosted point load:
LL = 2 k
Tributary width will be sampled.
Area loads on the floor will be included.


L(2.0)
D(0.350) L(0.8750)
L(2.0)
W16x26
Span = 25.0 ft


General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Auto add beam weight as Dead Load
Automatic Unbalanced Live Load Placement
Load locations start at far left end of beam system and progress over all spans per dimensions.
None
Tributary Width
7.0
ft
Dead
LR: Roof Live
L:Floor Live
Snow
Wind
E:Seismic
H: Earth
Magnitude :
0.050
0.0
0.1250
0.0
0.0
0.0
0.0
ksf

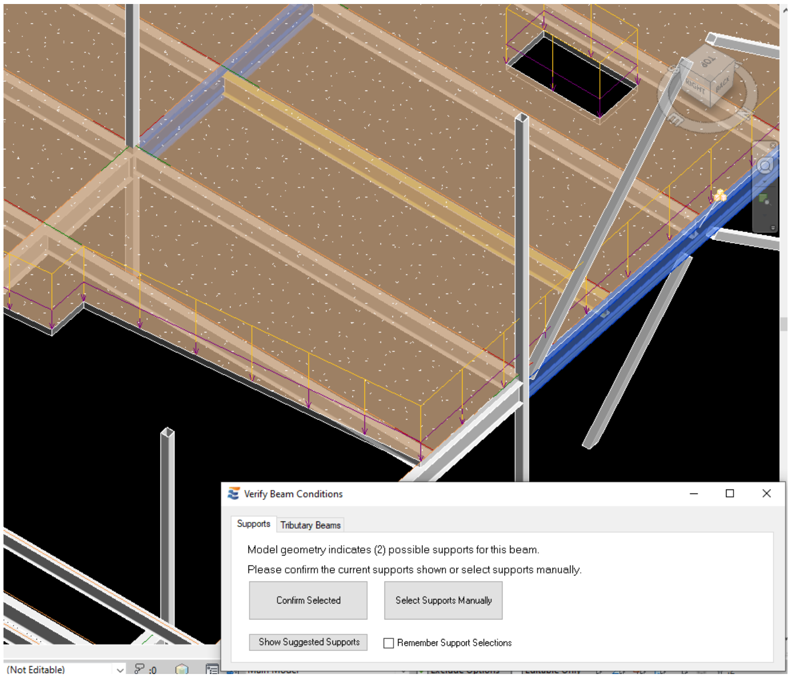


10.4.6.4 Typical Floor Conditions

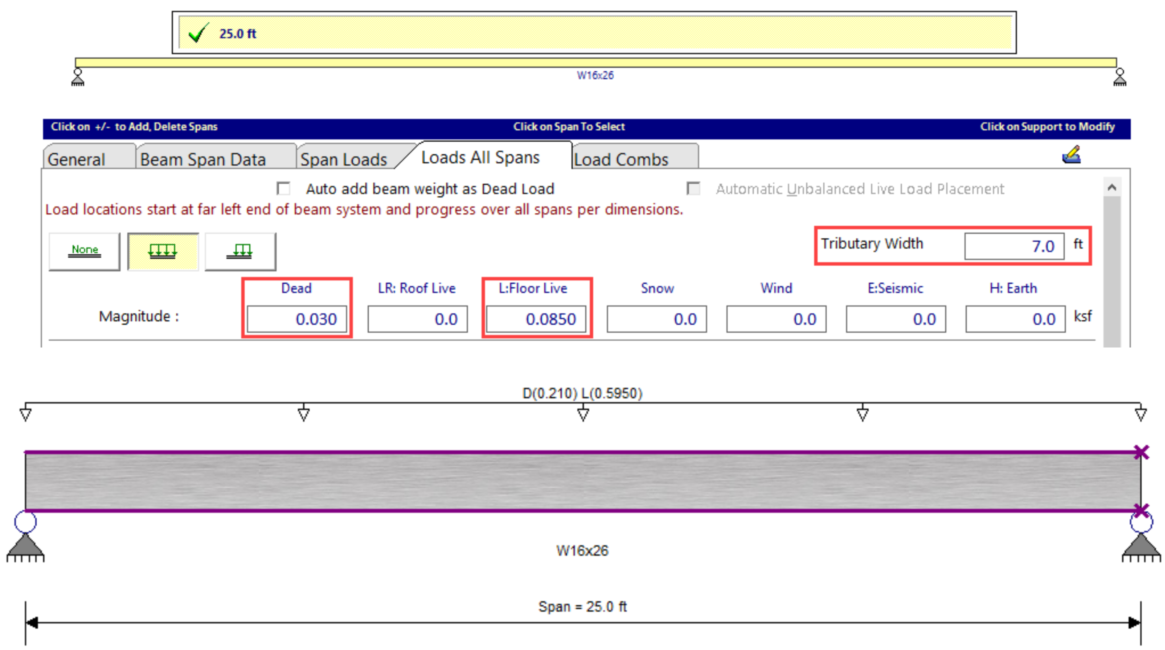
See subsections.

Typical Interior Floor Conditions

Typical interior beams with a uniform floor where the tributary width does not vary will produce similar results regardless of which sampling density has been set in the Preferences menu. Whether the beam is sampled at one point or many points, the sampling will return matching trib conditions at all points, resulting in a uniform linear load in the ENERCALC SEL interface.

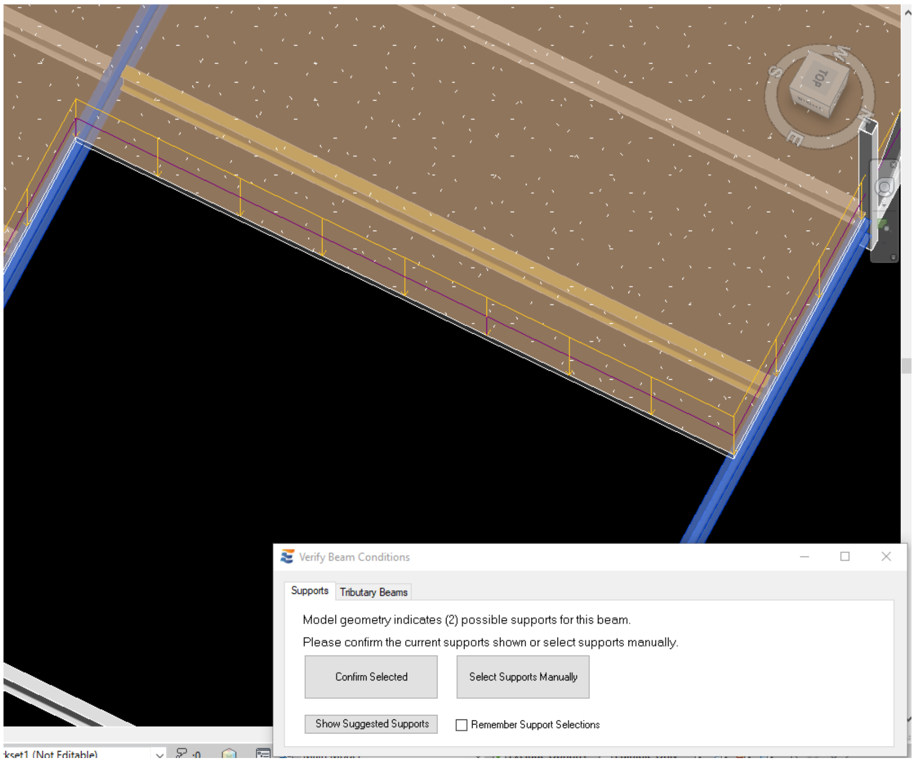


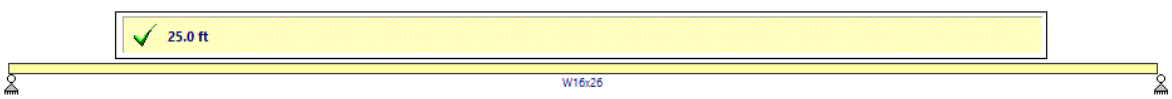
In this case, ENERCALC for Revit detects two adjacent beams and reports a tributary width of 7'-0", along with the area loads found on the floor. Since the floor geometry and loading are fully uniform and identical on both sides of the beam, the loads from the two sides of the beam are **NOT** reported individually.

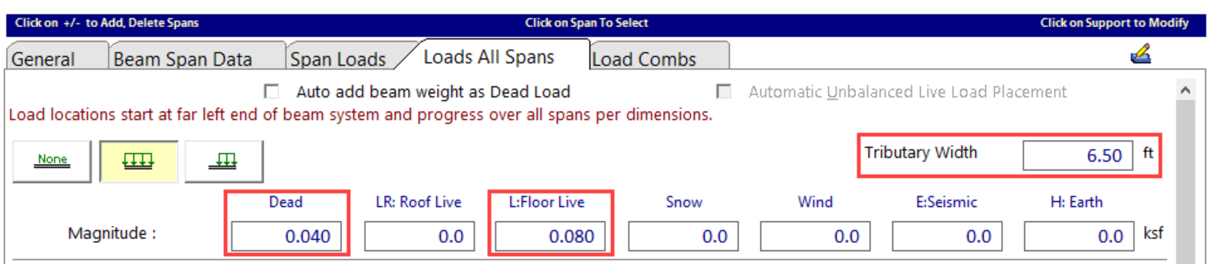


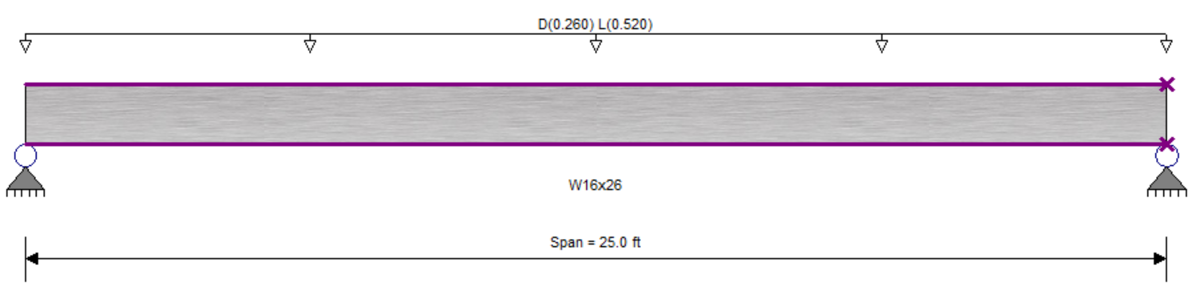
Uniform Floor Edges

Similar to fully uniform interior floor conditions, beams with a uniform floor edge will produce identical results regardless of which sampling density has been set in the Preferences menu. Whether the beam is sampled at one point or many points, the sampling will return matching trib conditions at all points, resulting in a uniform linear load in the ENERCALC SEL interface.



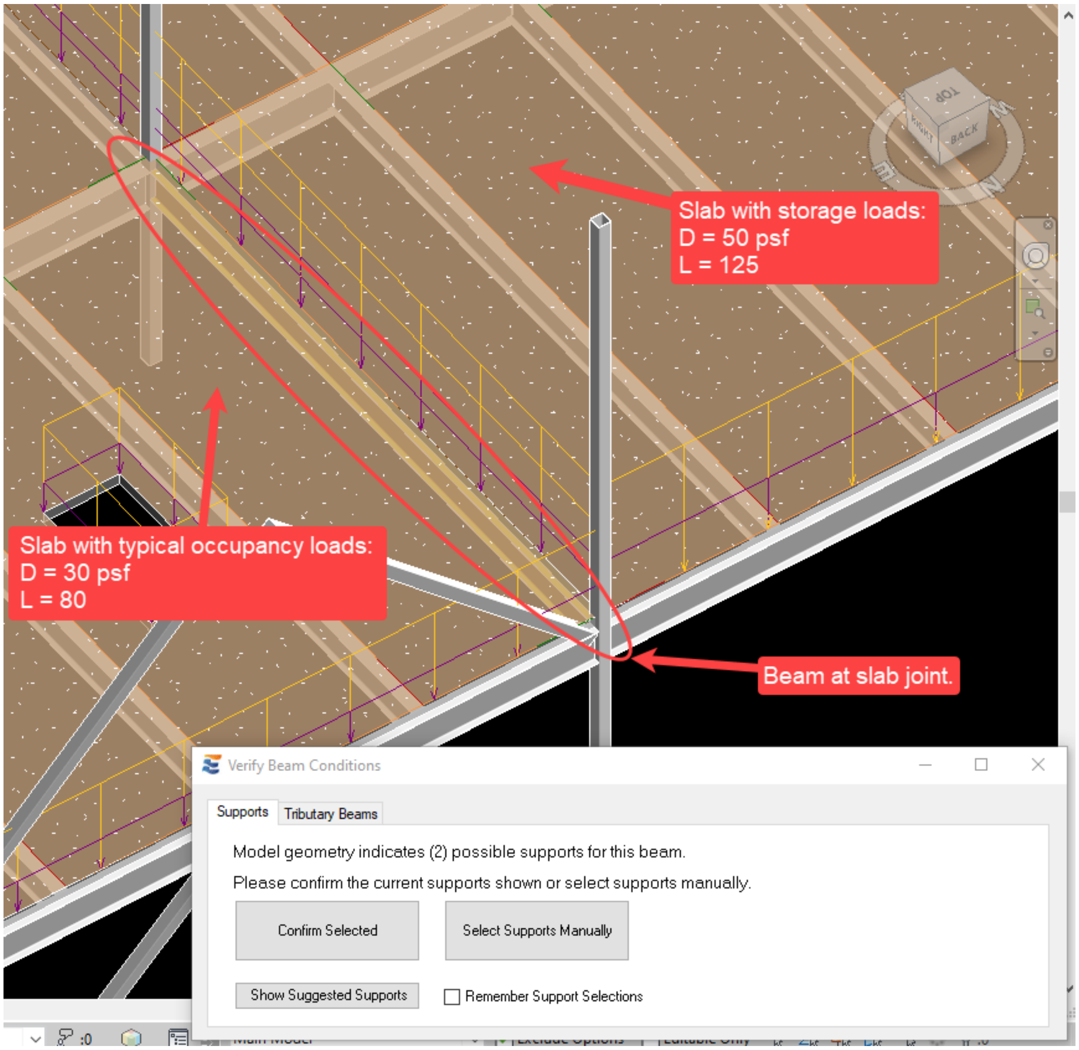




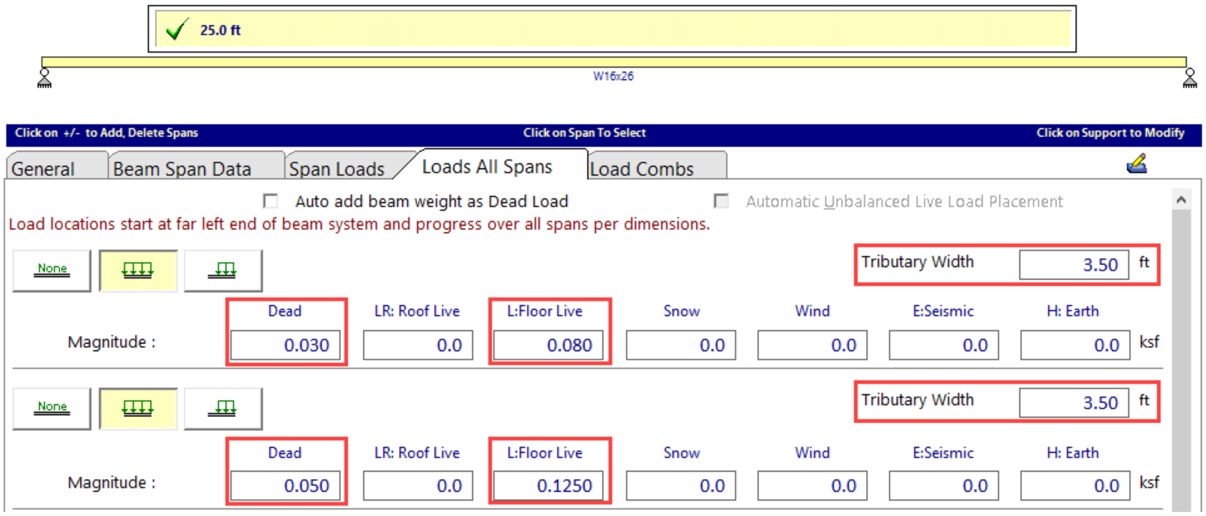


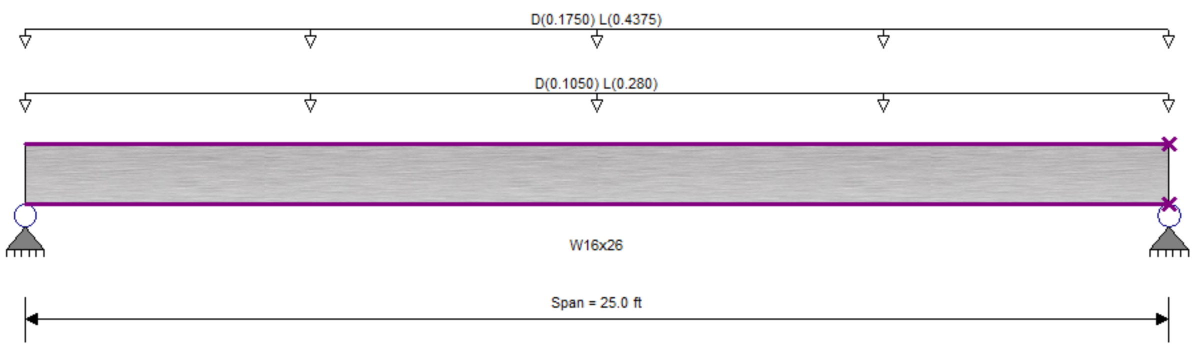
Floor Joint Conditions

Beams that fall at a location where two uniform floors meet in the center of the beam will produce similar results regardless of which sampling density has been set in the Preferences menu. Whether the beam is sampled at one point or many points, the sampling will return matching trib conditions at all points, resulting in a uniform linear load in the ENERCALC SEL interface.



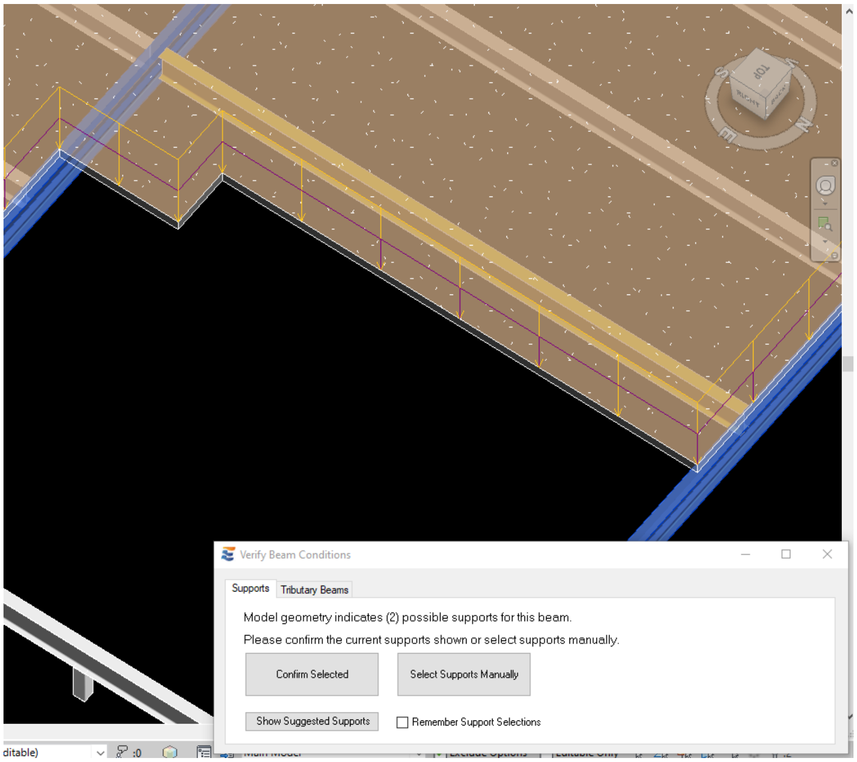
When the two sides of the beam do not exhibit identical loading, the loads will be separated in the ENERCALC SEL interface to reflect the behavior modeled in Revit.





Stepped Floor Edges

Conditions where a step or discontinuity occurs in the floor edge require more careful attention to the sampling density currently selected in the Preferences menu. Discrete sampling approaches, especially the modes with very few sampling points, are unlikely to actually capture the behavior in such cases.

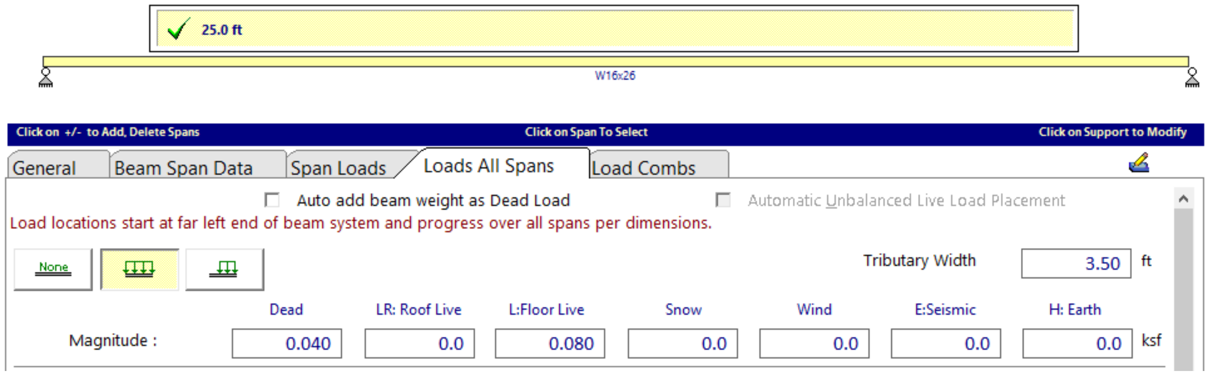


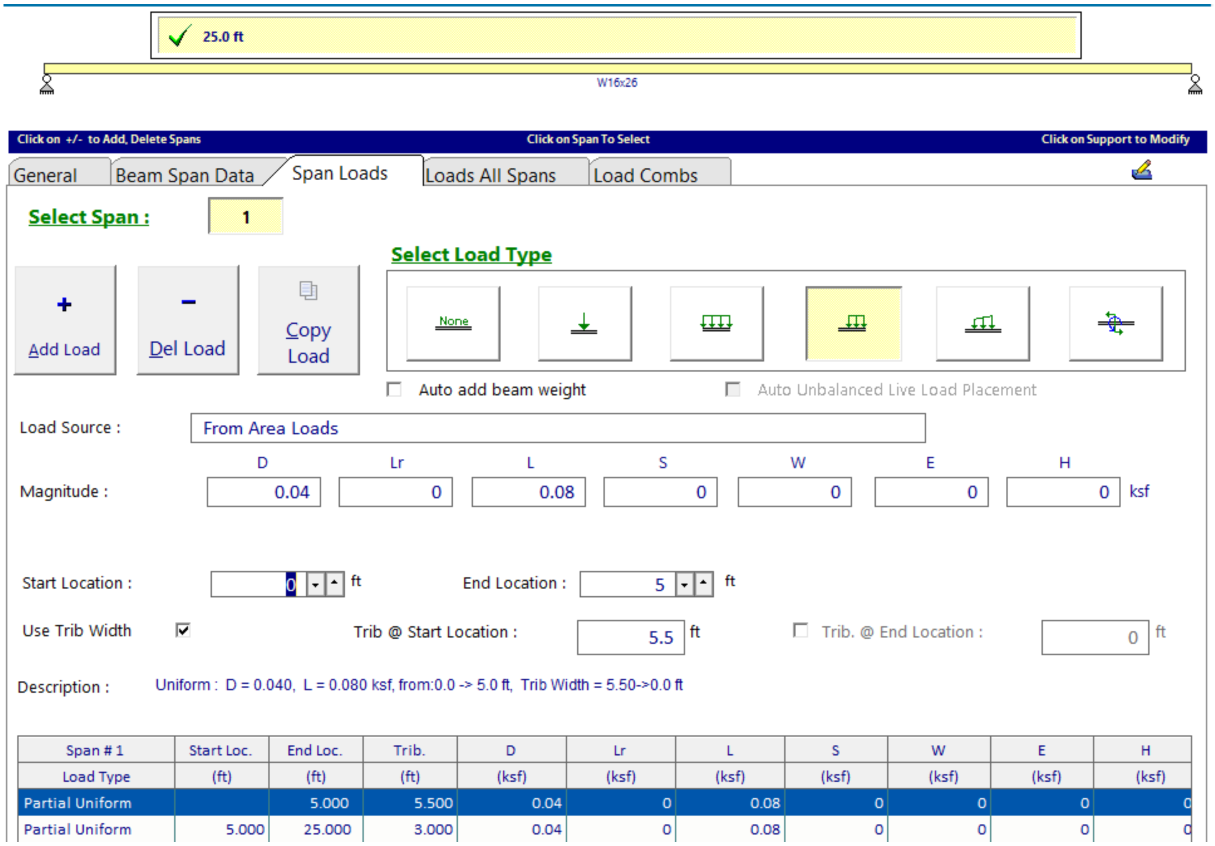
Sampling the beam shown above at “Midspan Only” would incorrectly result in a fully uniform loading diagram. Sampling the beam with a slightly denser but still discrete mode, such as “Quarter Points Plus Ends” would detect the existence of variation, but would not have adequate density of data to interpret the loads properly.

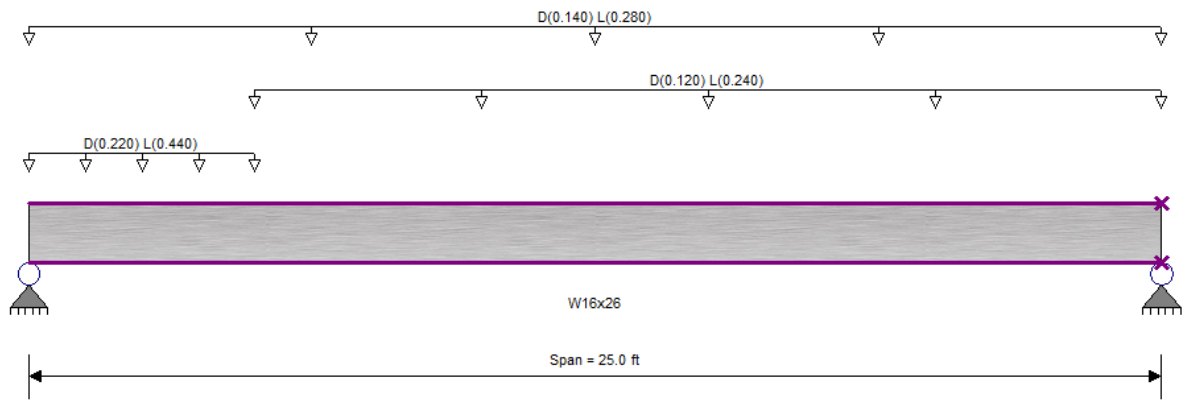
For this example beam, the recommended approach is to use the “Complete Contour” sampling mode. This results in a complete and accurate loading diagram that shows all of the following features:

1. A uniform load on one side of the beam:
    a. Tributary width = 3.5 ft
    b. Mapped to the “Loads All Spans” tab
2. The larger partial uniform load due to large slab overhang:
    a. Tributary width = 5.5 ft
    b. Mapped to the “Span Loads” tab
3. The smaller partial uniform load due to the small slab overhang
    a. Tributary width = 3.0 ft

b. Mapped to the “Span Loads” tab

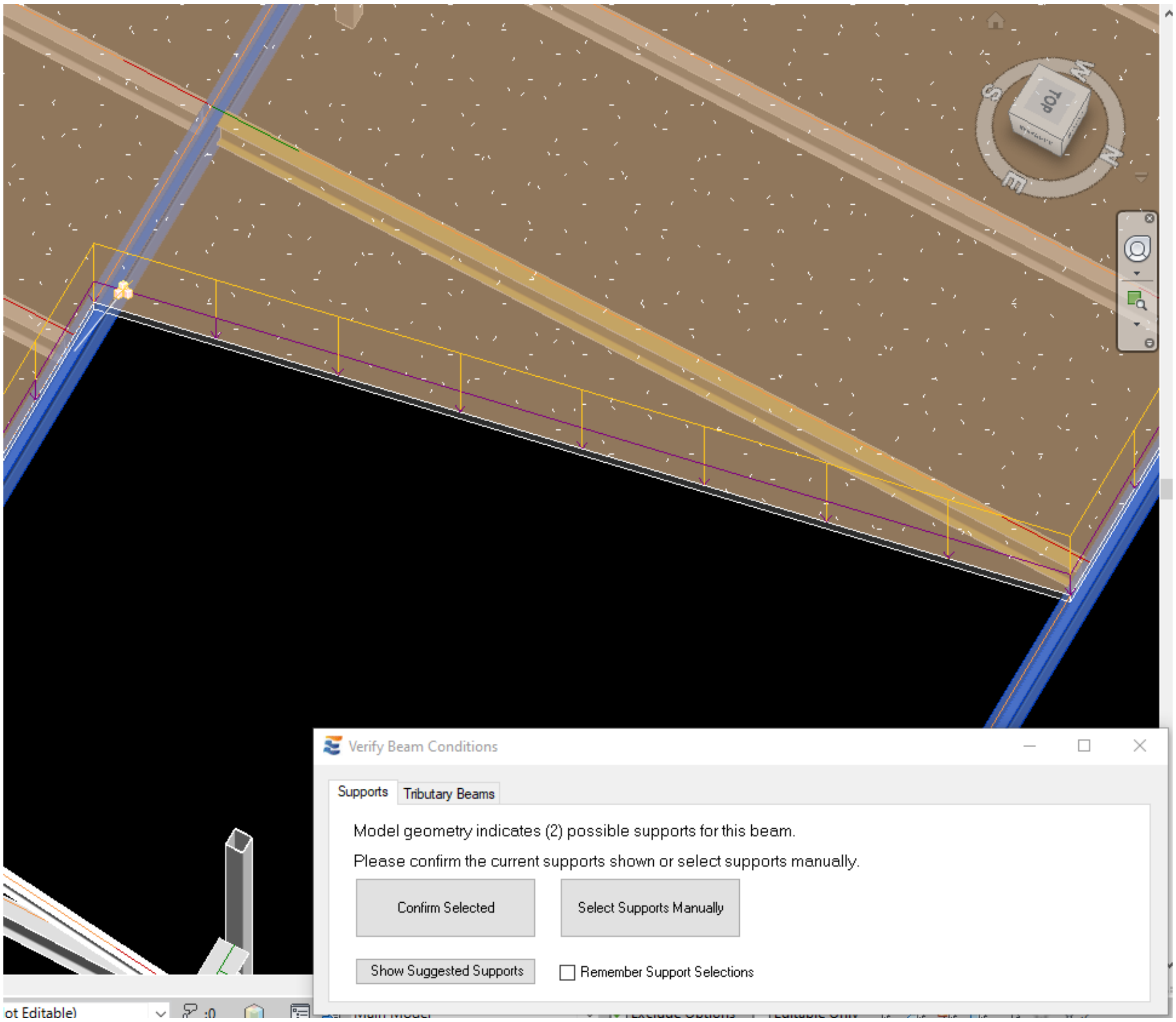






Linear Varying Floor Edges

Linear varying floor edge conditions are best suited to evaluation using the “Complete Contour” mode. This will result in a dense data set that can be automatically converted to linear varying loads for display in the ENERCALC SEL calculation.



The complete contour for this example beam results in the following load mappings:

1. A uniform load on one side of the beam:

   a. Tributary width = 3.5 ft

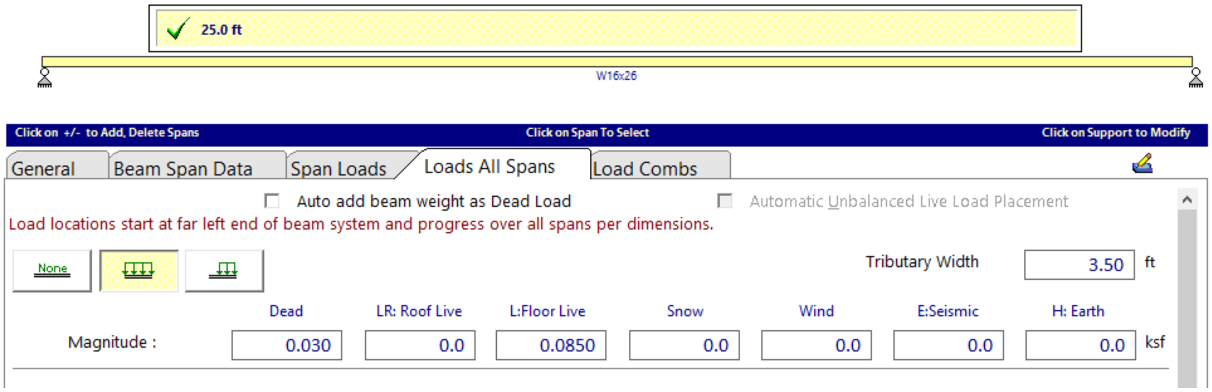
   b. Mapped to the “Loads All Spans” tab

2. The linear varying (trapezoidal) load due to the skewed slab overhang:

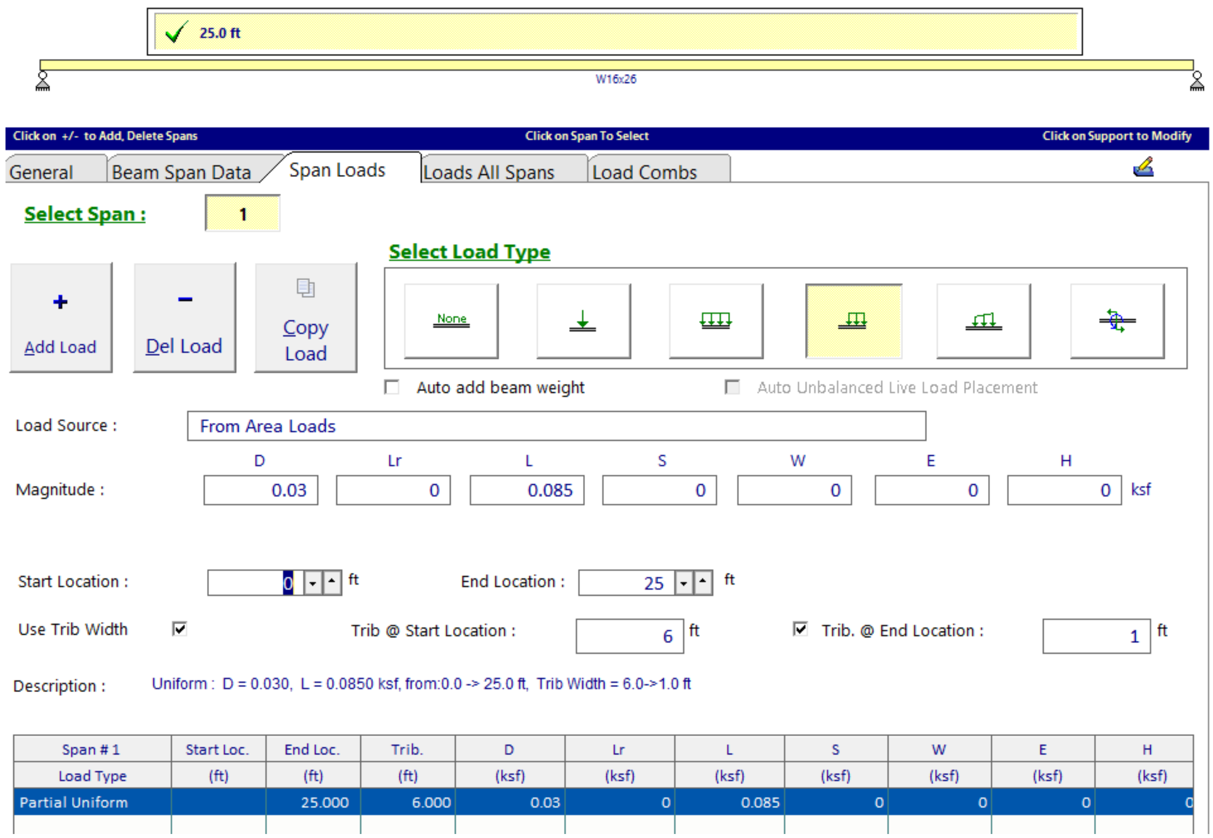
   a. Starting Tributary width = 6 ft

   b. Ending Tributary width = 6 ft

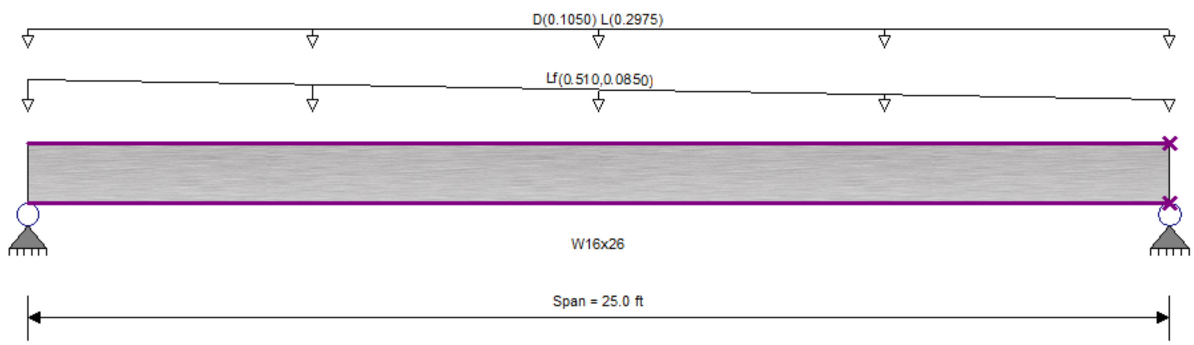
   c. Mapped to the “Span Loads” tab



25.0 ft
W16x26
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Auto add beam weight as Dead Load
Automatic Unbalanced Live Load Placement
Load locations start at far left end of beam system and progress over all spans per dimensions.
None
Tributary Width
3.50
ft
Dead
LR: Roof Live
L:Floor Live
Snow
Wind
E:Seismic
H: Earth
Magnitude :
0.030
0.0
0.0850
0.0
0.0
0.0
0.0
ksf

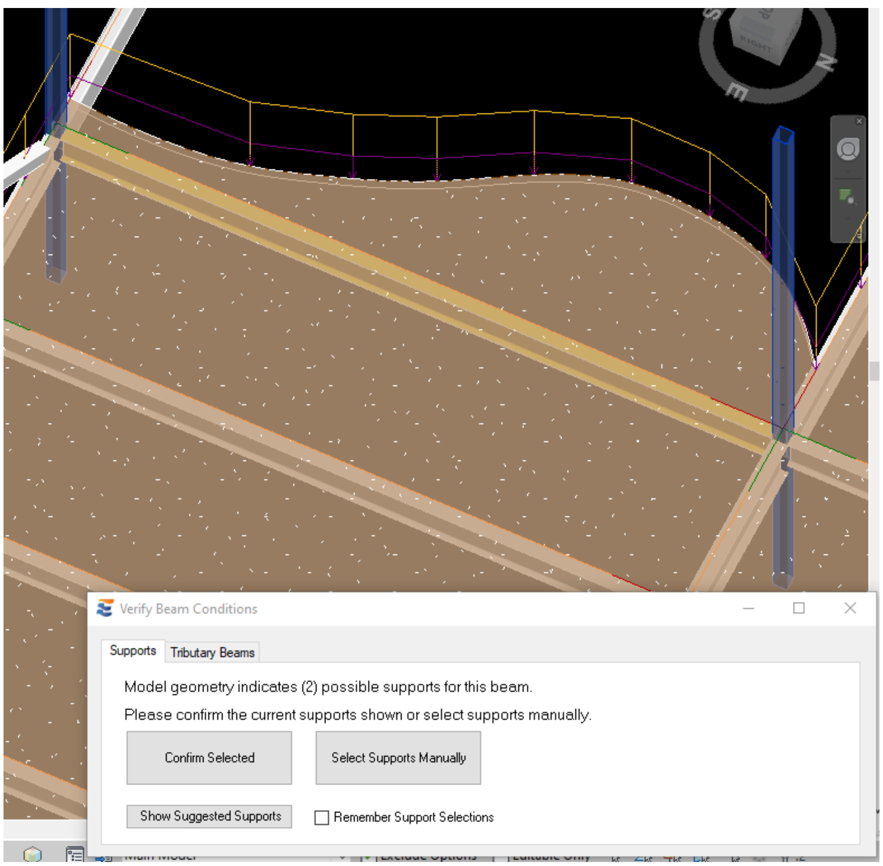


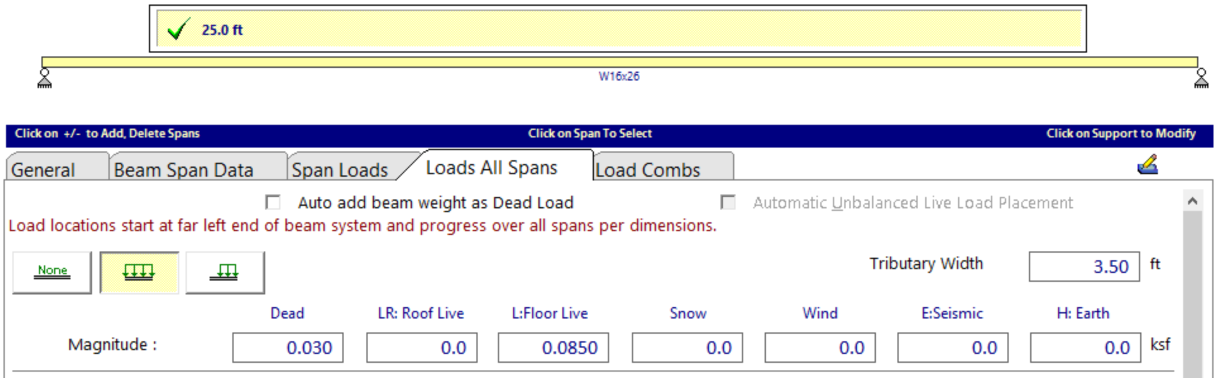
25.0 ft
W16x26
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Select Span :
1
Select Load Type
Add Load
Del Load
Copy Load
None
Auto add beam weight
Auto Unbalanced Live Load Placement
Load Source :
From Area Loads
D
Lr
L
S
W
E
H
Magnitude :
0.03
0
0.085
0
0
0
0
ksf
Start Location :
0
ft
End Location :
25
ft
Use Trib Width
Trib @ Start Location :
6
ft
Trib. @ End Location :
1
ft
Description :
Uniform : D = 0.030, L = 0.0850 ksf, from:0.0 -> 25.0 ft, Trib Width = 6.0->1.0 ft
Span # 1 Load Type
Start Loc. (ft)
End Loc. (ft)
Trib. (ft)
D (ksf)
Lr (ksf)
L (ksf)
S (ksf)
W (ksf)
E (ksf)
H (ksf)
Partial Uniform
25.000
6.000
0.03
0
0.085
0
0
0
0

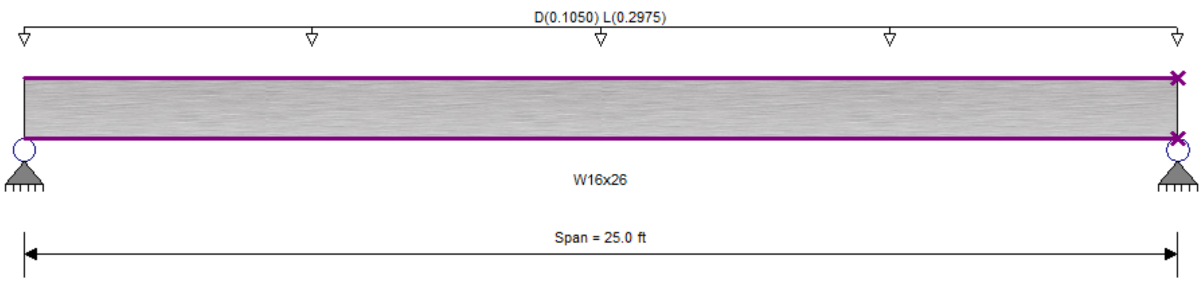


Irregular Floor Edges

ENERCALC for Revit does not support continuously varying tributary width profiles at this time. This includes the mapping of irregular floor edges. Loads that do not meet the criteria of 1.) uniform or 2.) linear varying will be ignored during calculation launch.

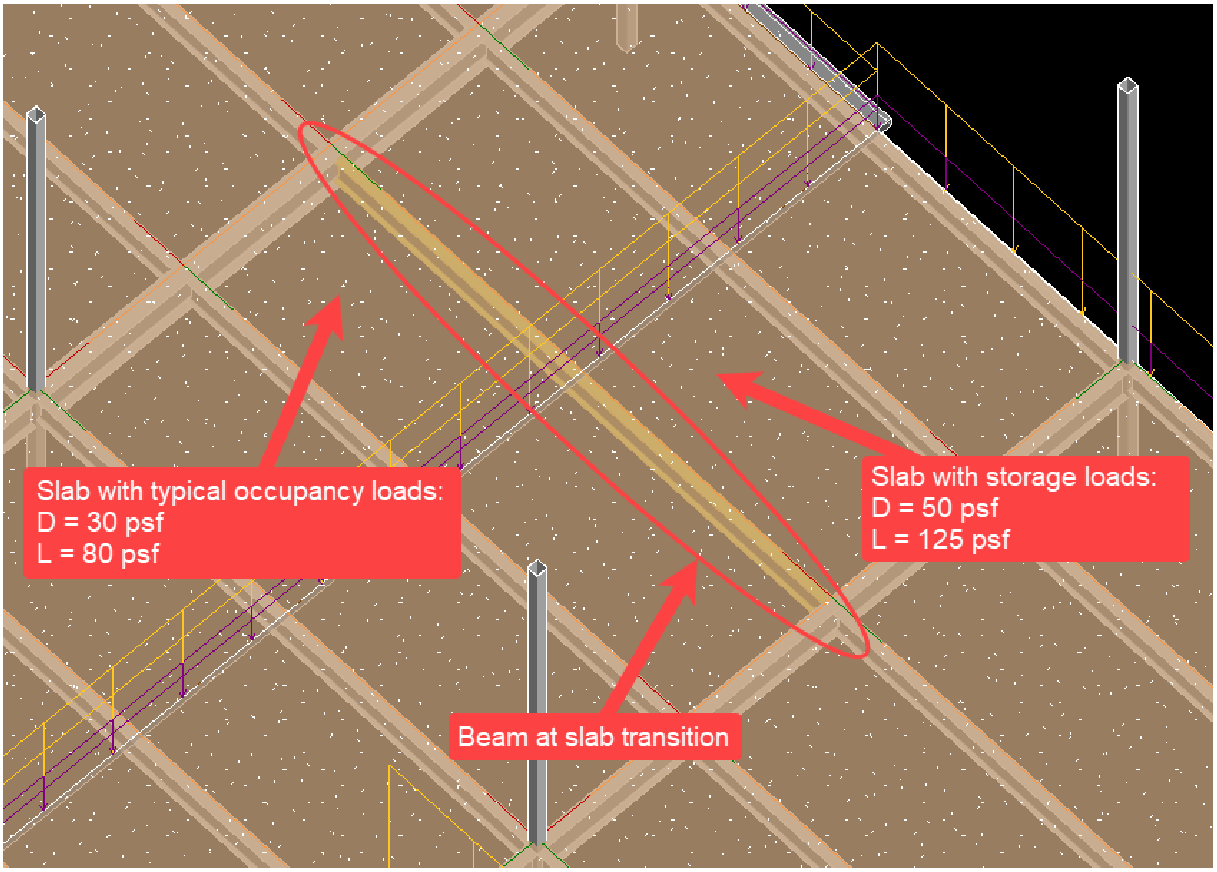




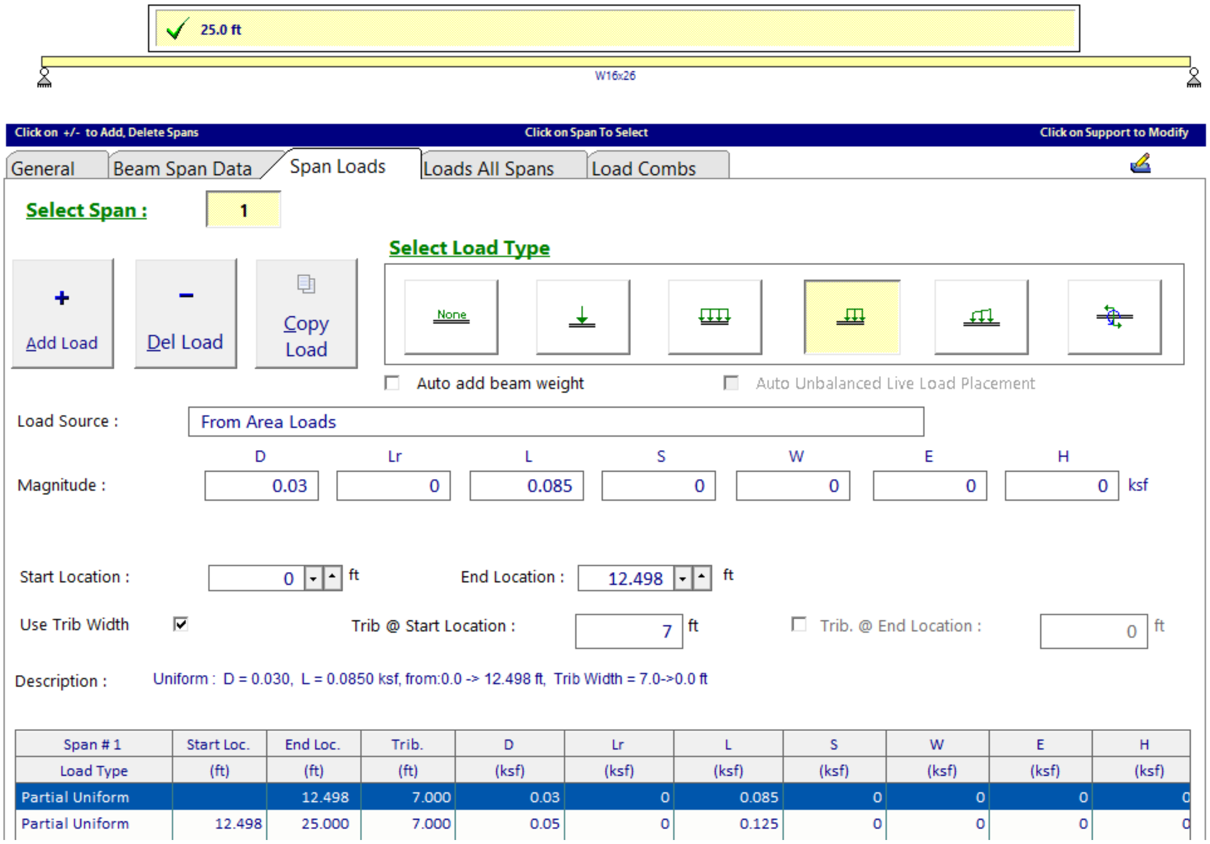


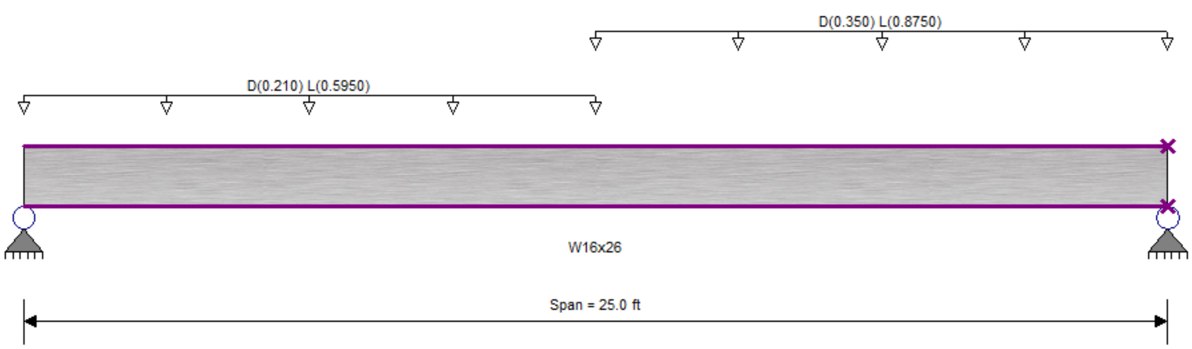
Floor Transitions

Beams that fall at a location where different floors bear on different portions of the beam are best analyzed using the “Complete Contour” mode. While a sparser sampling mode might create a similar loading diagram, it may not be dense enough to accurately locate the transition between floors.



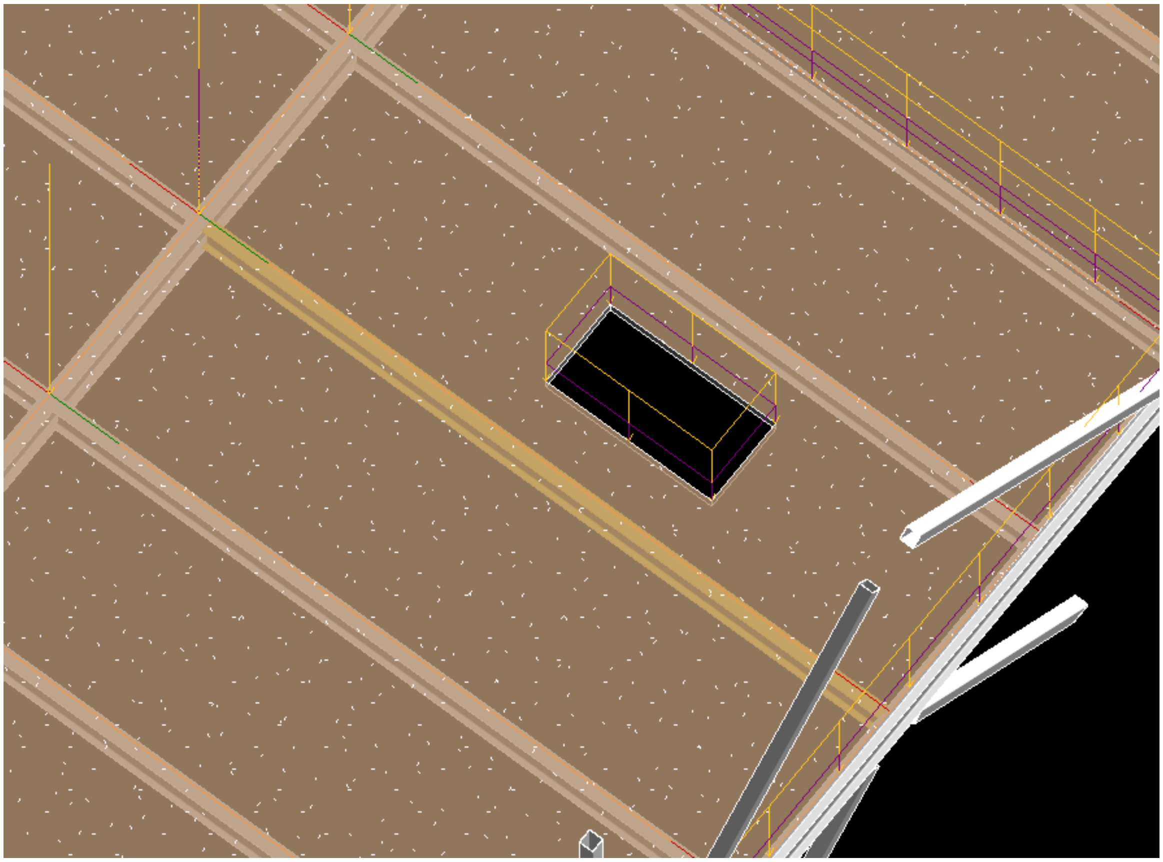
This loading scenario results in two partial uniform loads of differing magnitudes, corresponding to the portions of the beam supporting the different floors. Since they are segmented, the loads are mapped to the “Span Loads” tab.



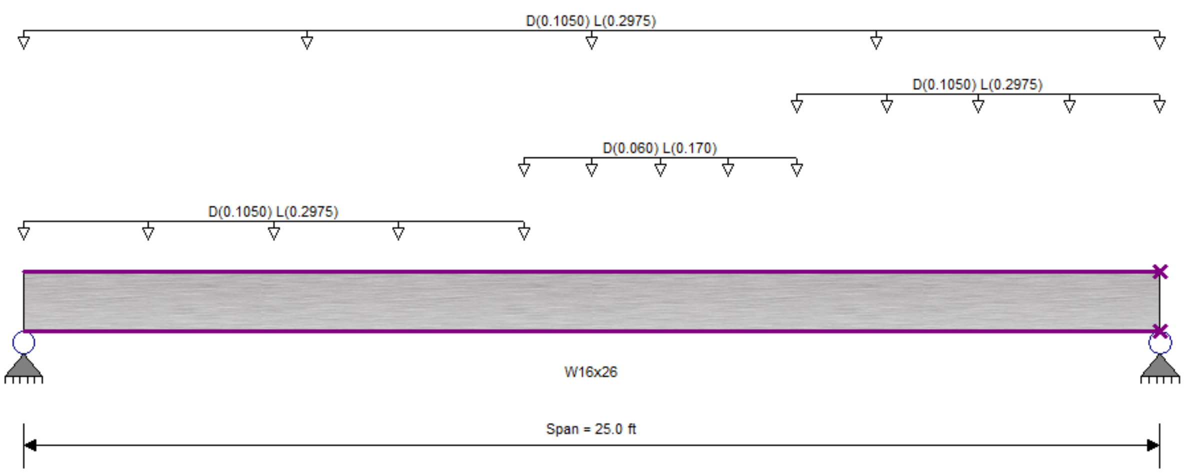


Floor Openings

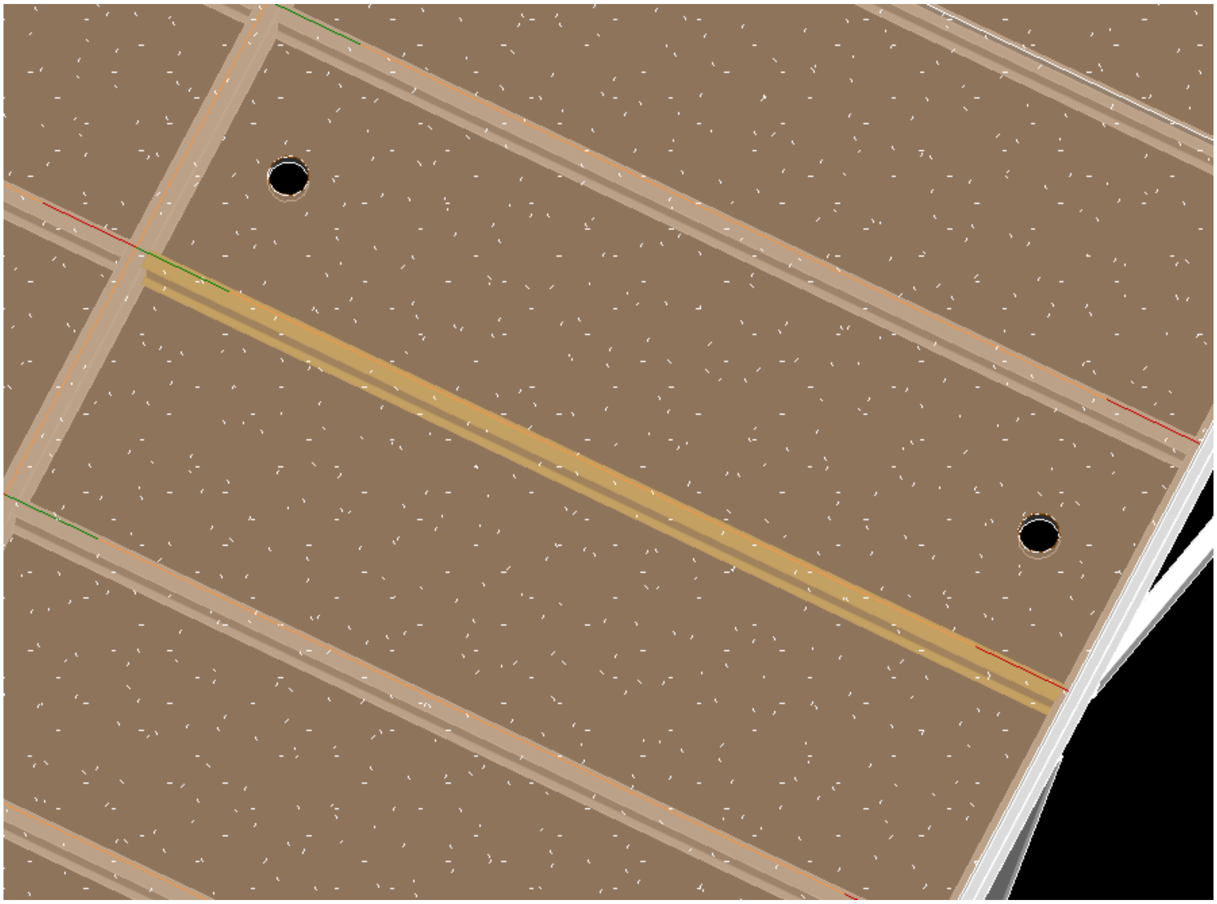
The optimal sampling mode for beams adjacent to floor openings depends on the user's desired outcome.

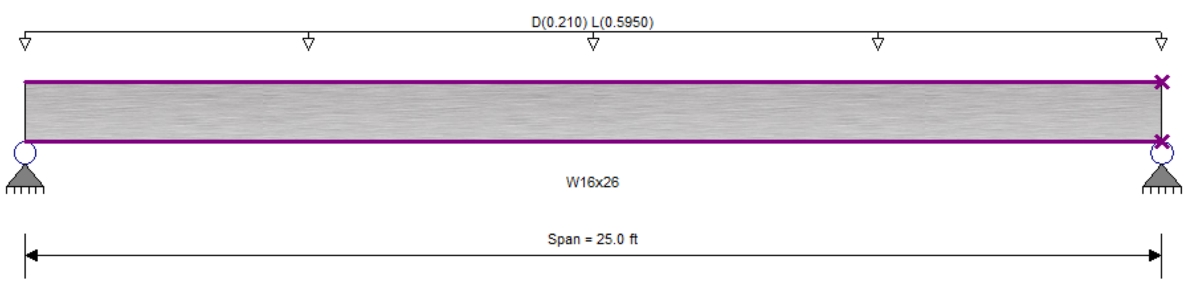


To accurately model all variations in load due to the opening, the beam should be launched using the “Complete Contour” mode. This will result in segmented loads that reflect the transition from simple span slab, to cantilever slab at the opening, and back to simple span.



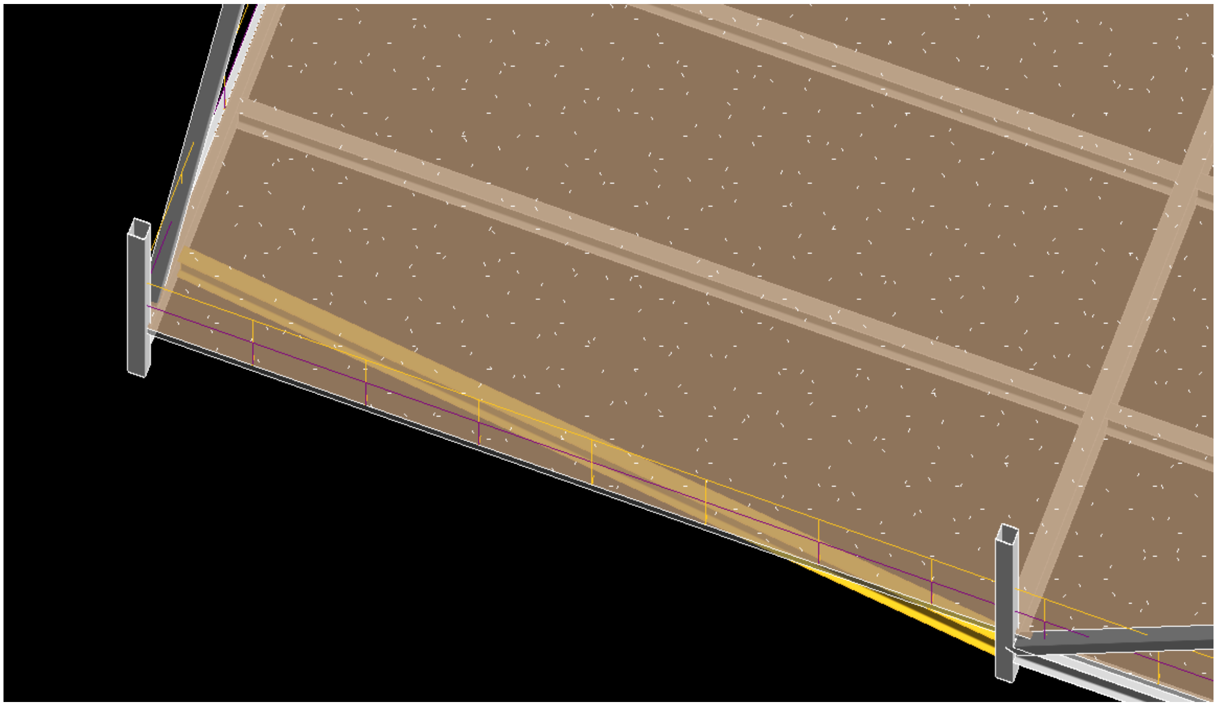
However, in some cases a user may desire to design a beam without considering small insignificant floor openings and instead design the beam as if uniformly loaded. In such cases, the user can easily select an alternate sampling density that does not capture the subtle variations the user wishes to ignore. For example, sampling the beam shown below at midspan, third points, or quarter points would result in the small piping sleeve holes being ignored.

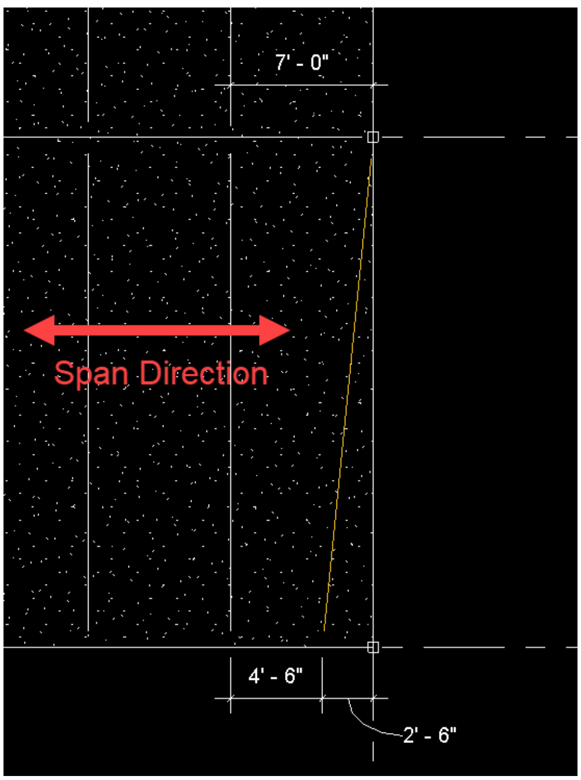




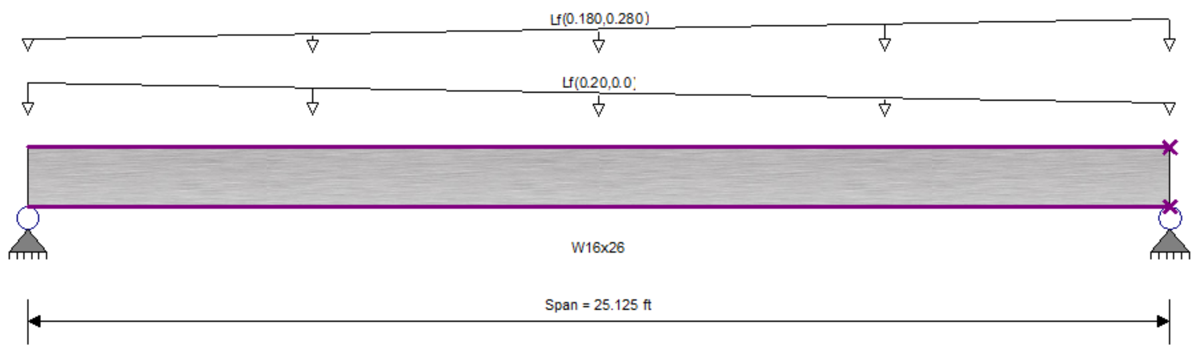
Skewed Beams

A “skewed” beam is any beam that is not perfectly perpendicular to the span direction of the floor that it supports. ENERCALC for Revit supports the design of beams skewed from the span direction by using the “Complete Contour” sampling mode to capture the continuous variation of tributary width caused by the skew.

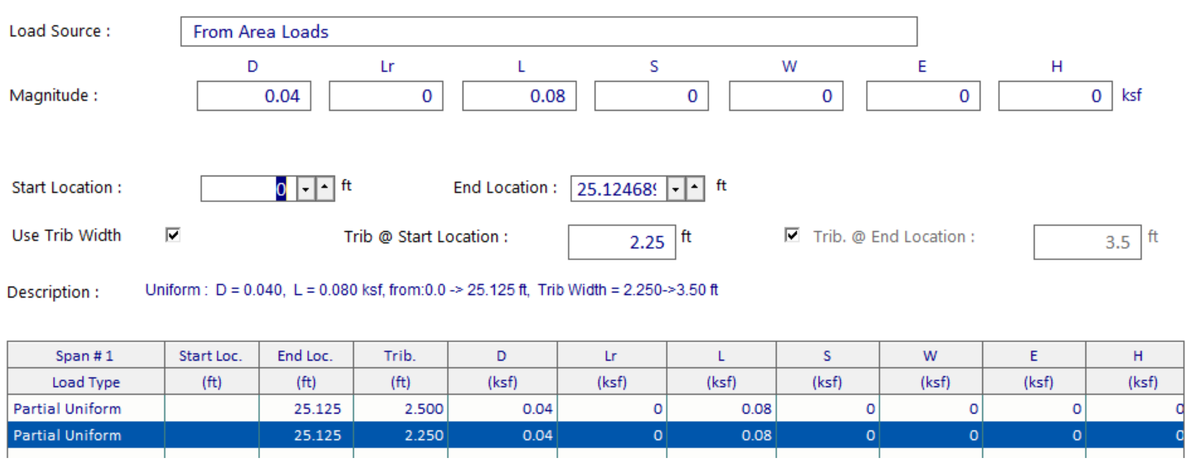




The result of this geometry is two full-length linear varying loads (one for each side of the beam).



The load from the left side of the beam tapers from a tributary width of 2.25 ft up to 3.5 ft.



Load Source : From Area Loads

	D	Lr	L	S	W	E	H	
Magnitude :	0.04	0	0.08	0	0	0	0	ksf

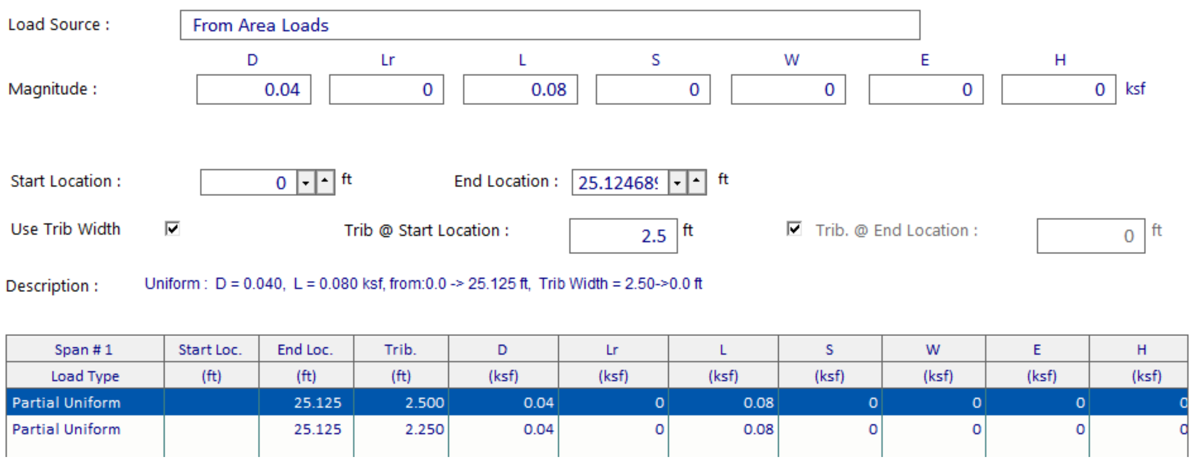
Start Location : 0 ft End Location : 25.12468 ft

Use Trib Width ☑ Trib @ Start Location : 2.25 ft ☑ Trib. @ End Location : 3.5 ft

Description : Uniform : D = 0.040, L = 0.080 ksf, from:0.0 -> 25.125 ft, Trib Width = 2.250->3.50 ft

Span # 1	Start Loc.	End Loc.	Trib.	D	Lr	L	S	W	E	H
Load Type	(ft)	(ft)	(ft)	(ksf)	(ksf)	(ksf)	(ksf)	(ksf)	(ksf)	(ksf)
Partial Uniform		25.125	2.500	0.04	0	0.08	0	0	0	0
Partial Uniform		25.125	2.250	0.04	0	0.08	0	0	0	0

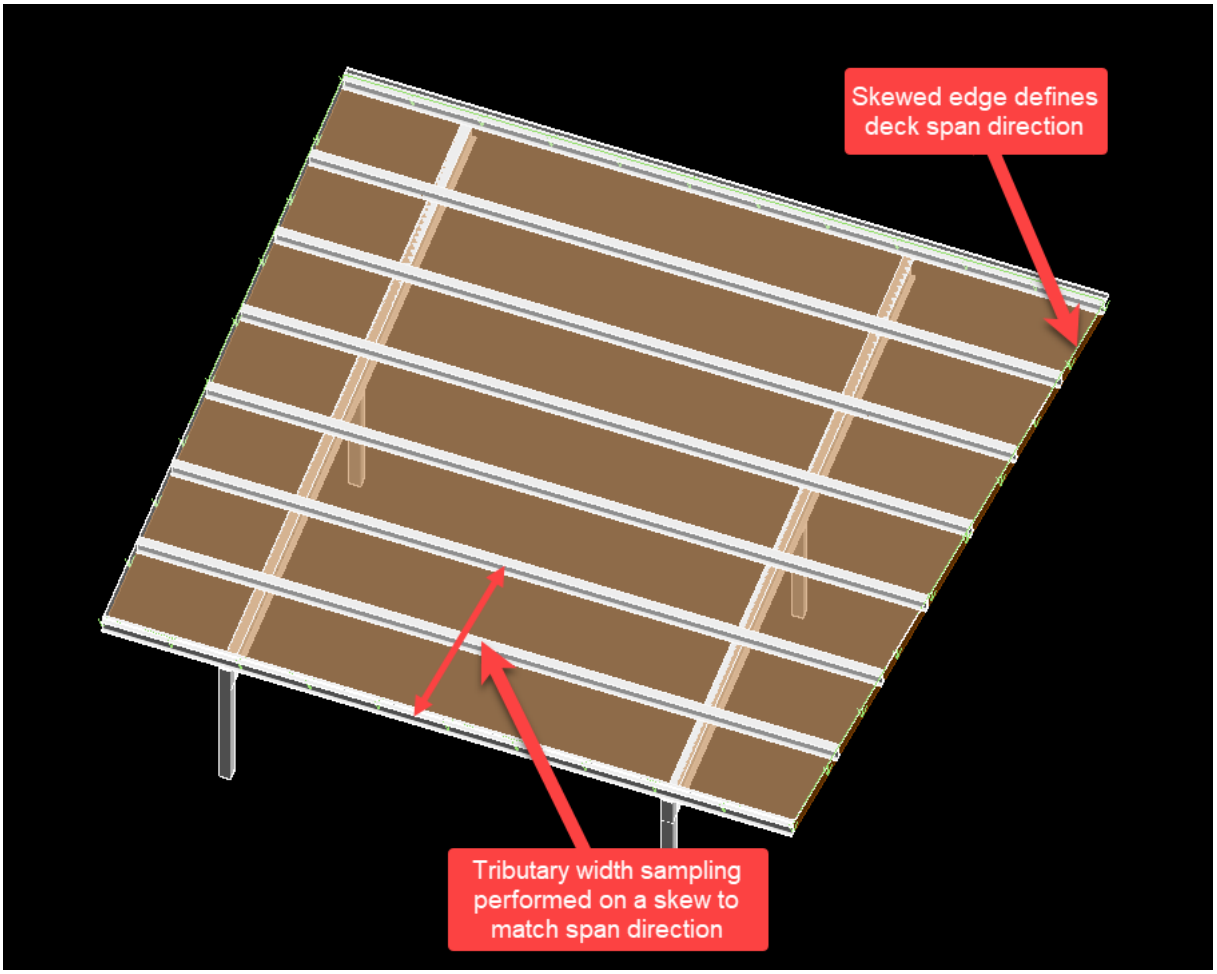
The load on the right side of the beam tapers from 2.5 ft to zero.



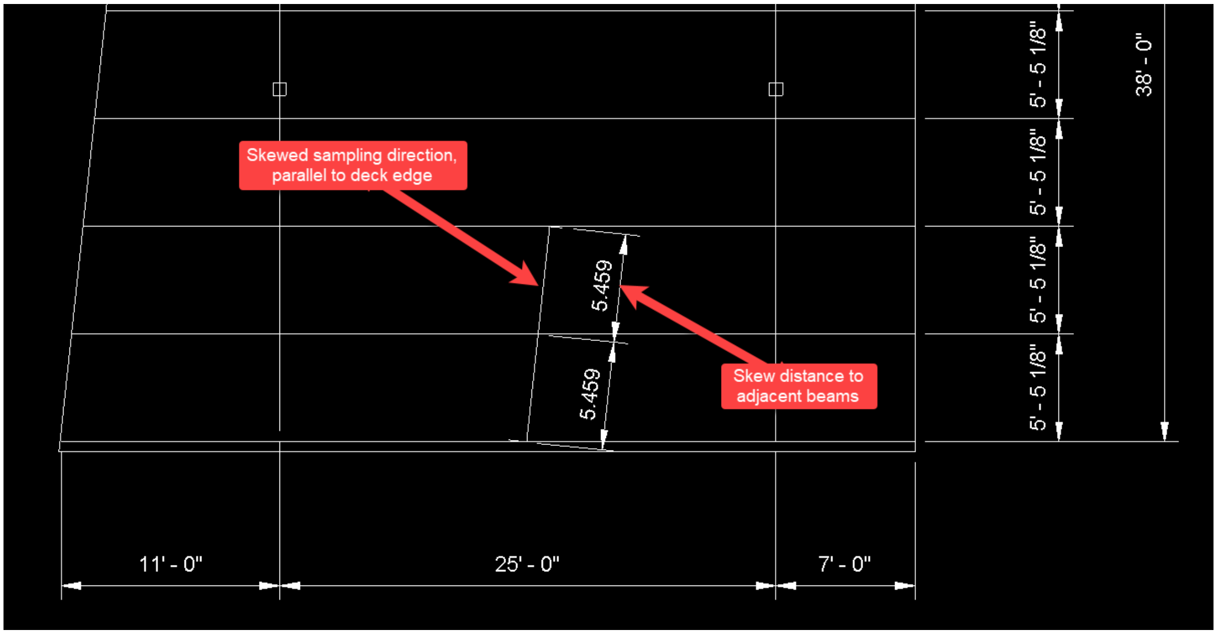
Skewed Floor Span Directions

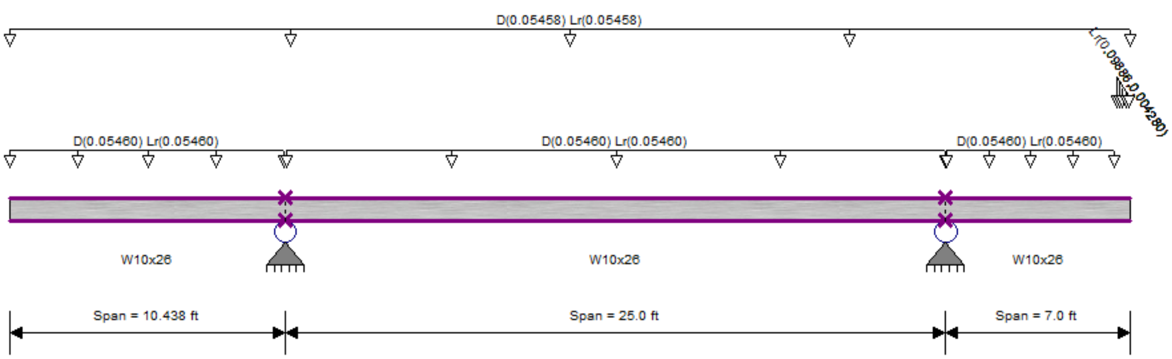
Similar to the example above, “Complete Contour” mode is also recommended for design of beams where the floor span direction is skewed from the overall framing system. This is not a common everyday challenge for most user, but may arise on occasion. In this case, the tributary width is not computed perpendicularly from beam to beam, but instead is computed parallel to the deck span.

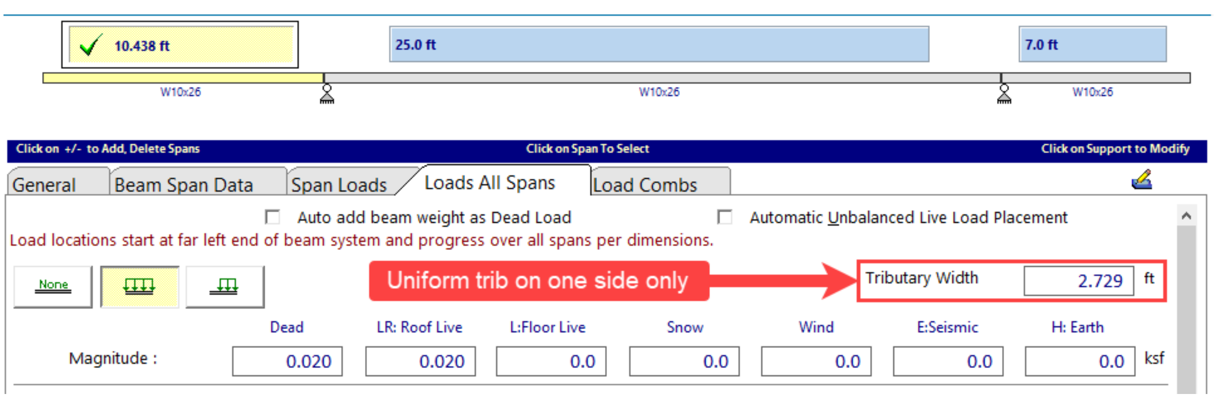
In the example canopy shown below, the deck span direction is set parallel to the skewed edge rather than the orthogonal edge of the deck. This results in the tributary width being sampled in a skewed direction to match the deck behavior.



Users will also observe from the loading diagram shown below that the trib sampling captures the variation in tributary width at the end regions of the beam where the skewed sampling direction causes the trib to be truncated by the outside edge of the deck.

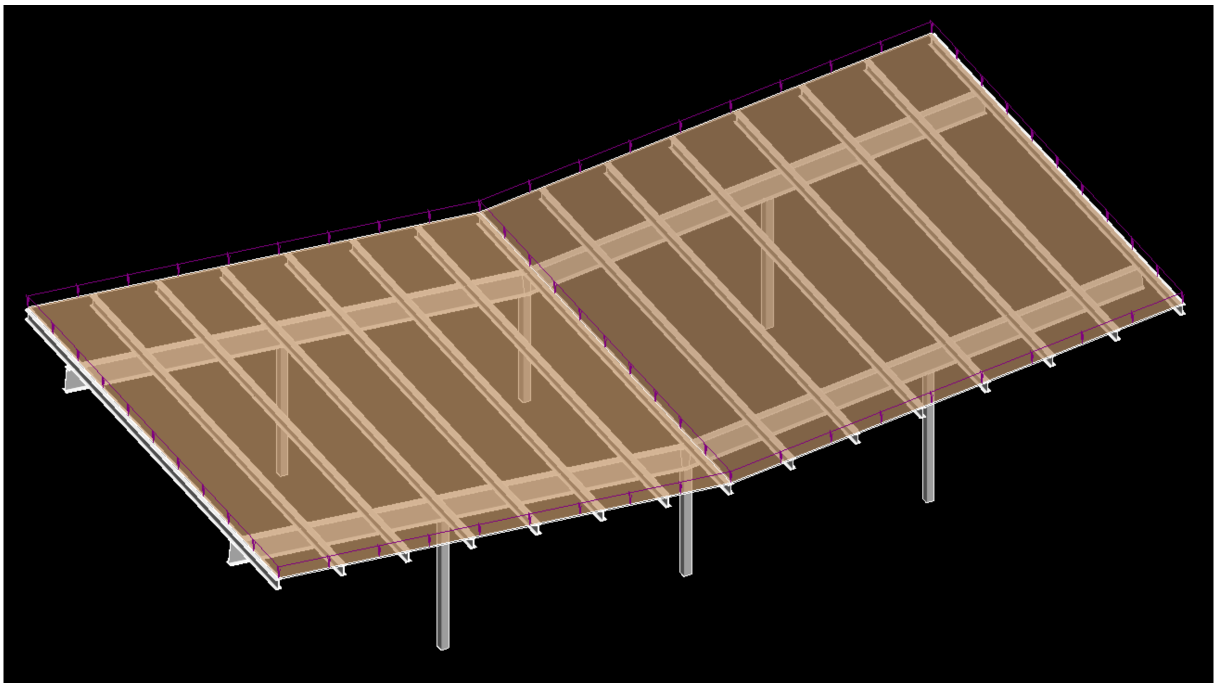

Skewed sampling direction, parallel to deck edge
Skew distance to adjacent beams
5.459
5.459
5' - 5 1/8"
5' - 5 1/8"
5' - 5 1/8"
5' - 5 1/8"
38' - 0"
11' - 0"
25' - 0"
7' - 0"


D(0.05458) Lr(0.05458)
D(0.05460) Lr(0.05460)
D(0.05460) Lr(0.05460)
D(0.05460) Lr(0.05460)
W10x26
W10x26
W10x26
Span = 10.438 ft
Span = 25.0 ft
Span = 7.0 ft

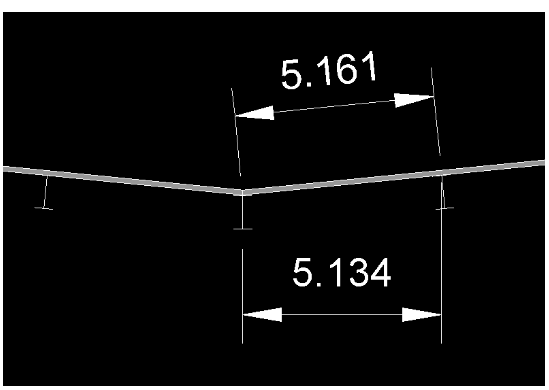

10.438 ft
25.0 ft
7.0 ft
W10x26
W10x26
W10x26
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Auto add beam weight as Dead Load
Automatic Unbalanced Live Load Placement
Load locations start at far left end of beam system and progress over all spans per dimensions.
None
Uniform trib on one side only
Tributary Width
2.729
ft
Dead
LR: Roof Live
L:Floor Live
Snow
Wind
E:Seismic
H: Earth
Magnitude :
0.020
0.020
0.0
0.0
0.0
0.0
0.0
ksf

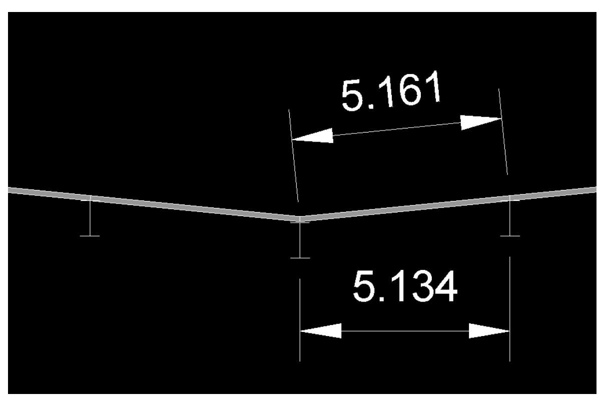
Sloped Floors

ENERCALC for Revit also provides the ability to obtain equivalent linear loads from area loads applied on a sloping deck. This can be done whether the beams that support the deck are in default orientation or have their cross-sections rotated to fit the deck slope. Regardless of the beam section orientation, however, the tributary width sampling is performed along the slope of the floor element, NOT orthogonal to the beam cross-section.

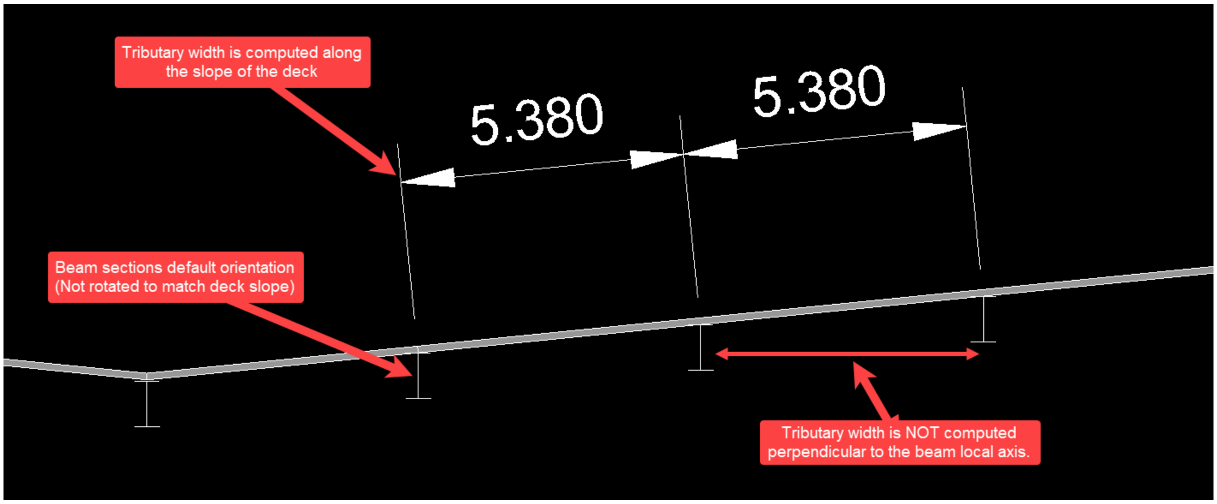


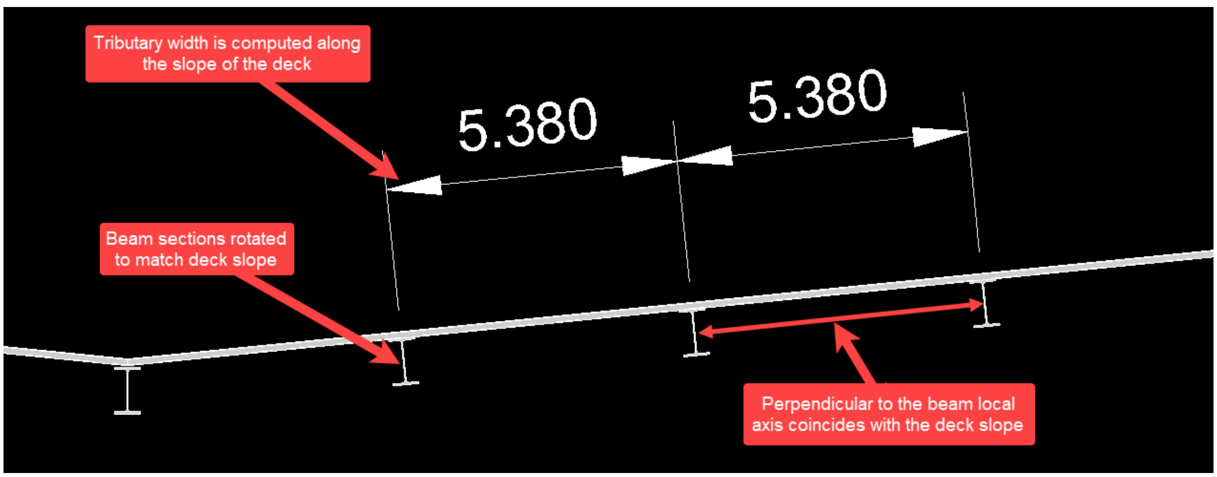
Because the cross-section rotation shifts the line location of the beams, users will observe that two sets of beams modeled with identical ***horizontal*** distance between working points will also show identical tributary width along the slope of the deck. This is also true regardless of whether the designed beam and its neighbor have the same section orientation.

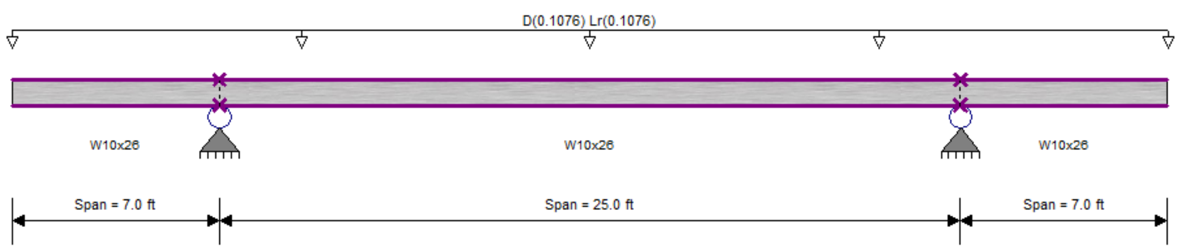


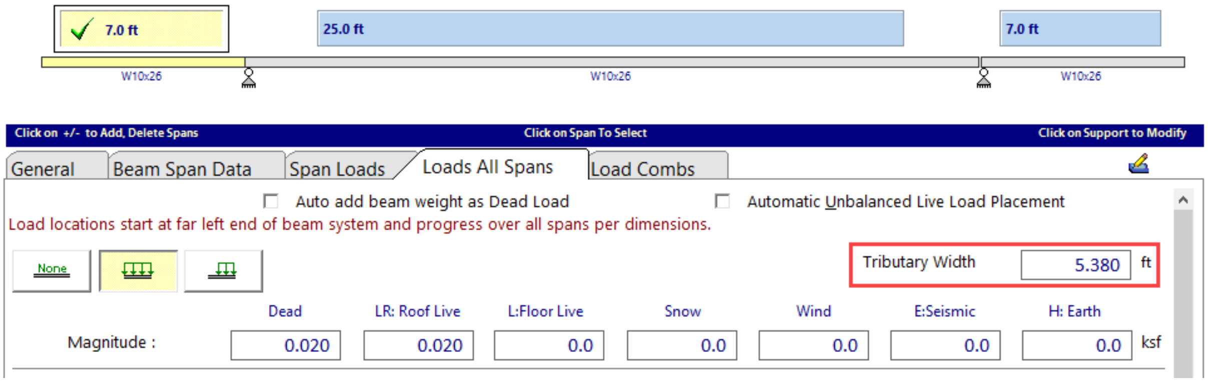


The following images show two modeling approaches to the same example, modeled first with the sections in default orientation and then rotated to fit the deck slope. The resulting load diagram and tributary width are identical.


Tributary width is computed along the slope of the deck
5.380
5.380
Beam sections default orientation (Not rotated to match deck slope)
Tributary width is NOT computed perpendicular to the beam local axis.

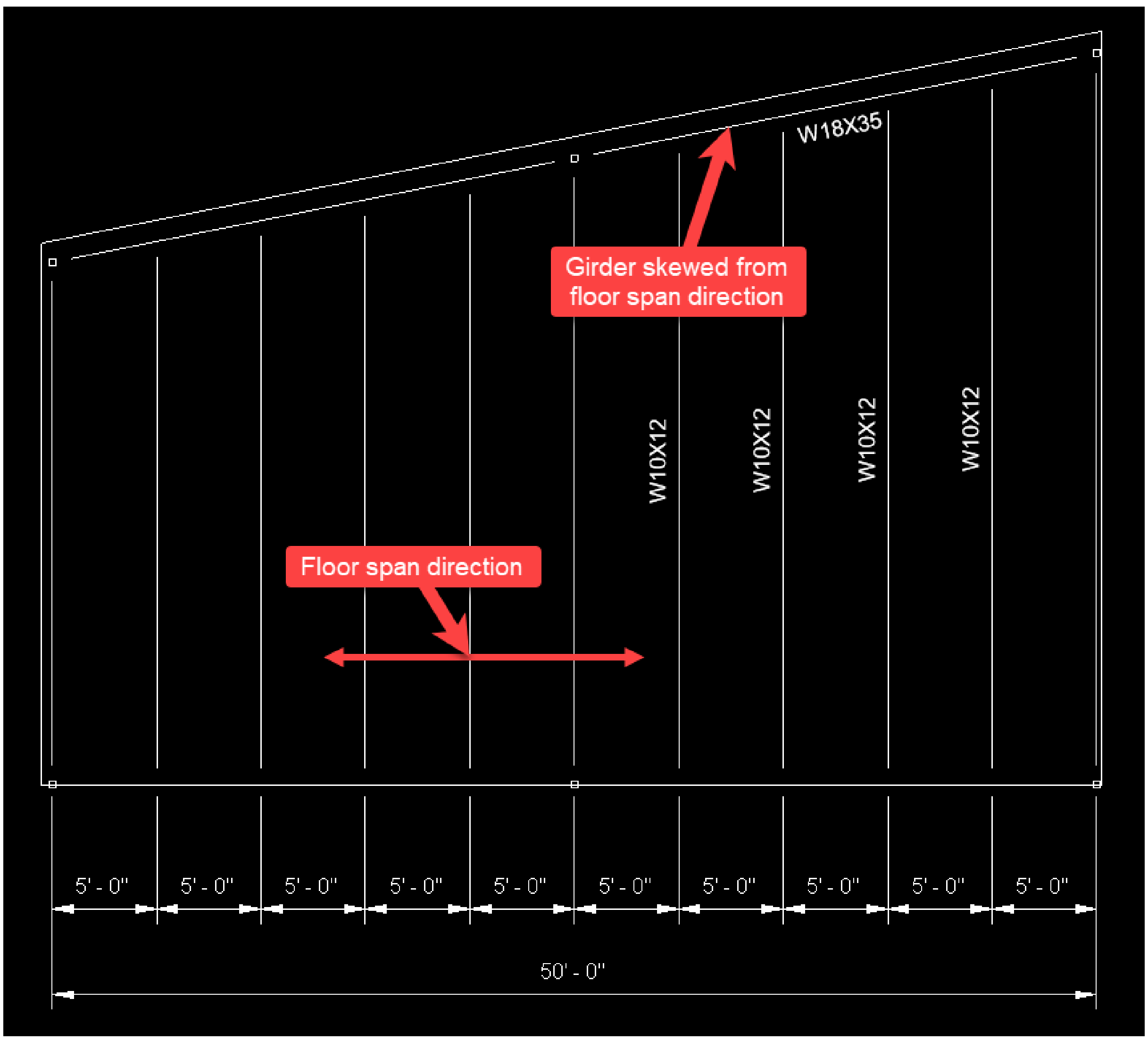

Tributary width is computed along the slope of the deck
5.380
5.380
Beam sections rotated to match deck slope
Perpendicular to the beam local axis coincides with the deck slope


D(0.1076) Lr(0.1076)
W10x26
W10x26
W10x26
Span = 7.0 ft
Span = 25.0 ft
Span = 7.0 ft

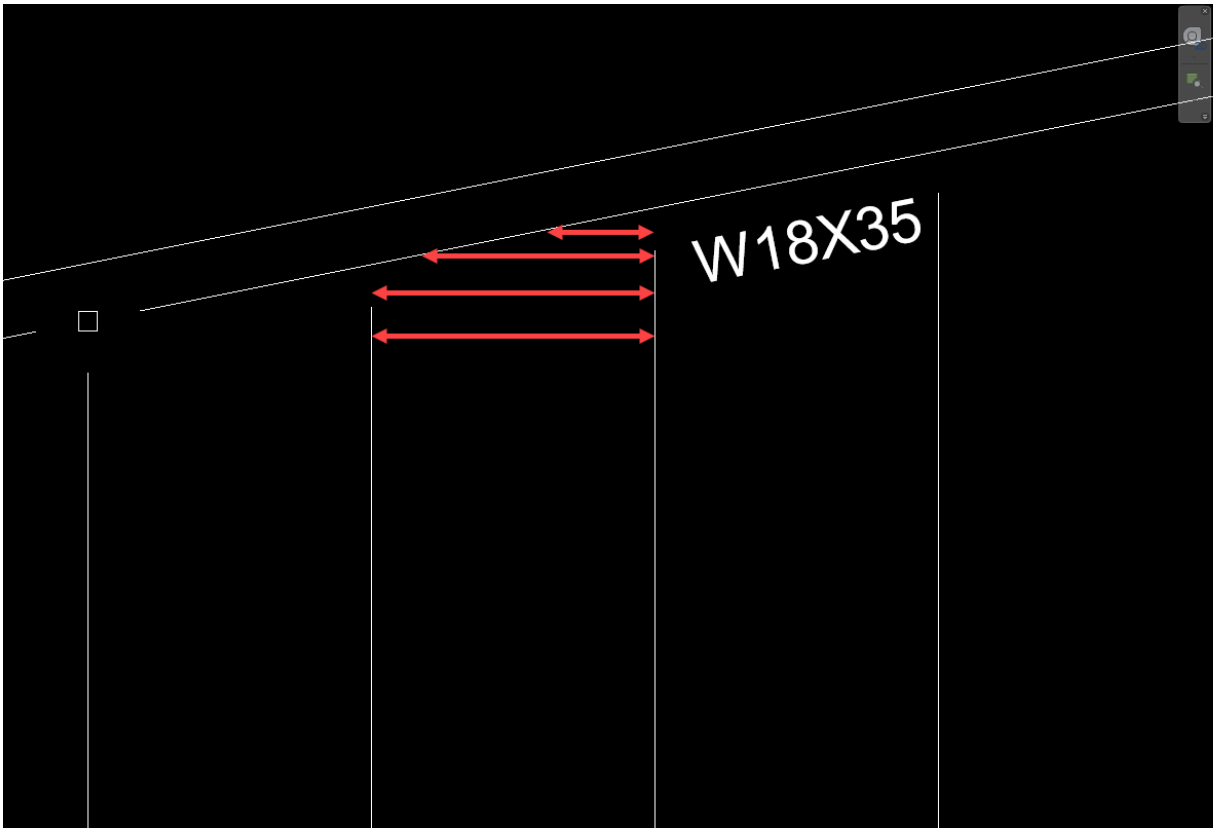


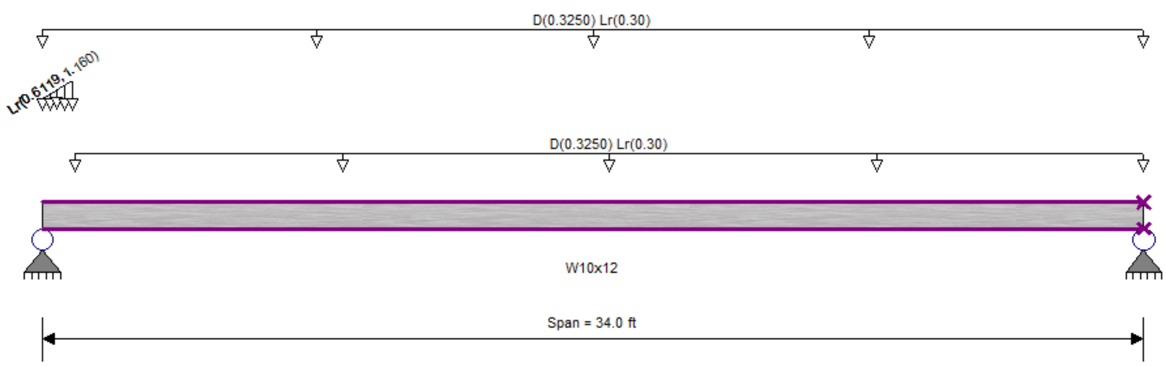
Skewed Perimeter Girders

In the case of a perimeter girder that is skewed from the floor span direction, users should be aware that the end regions of the supported floor beams will reflect the effects of this skew.

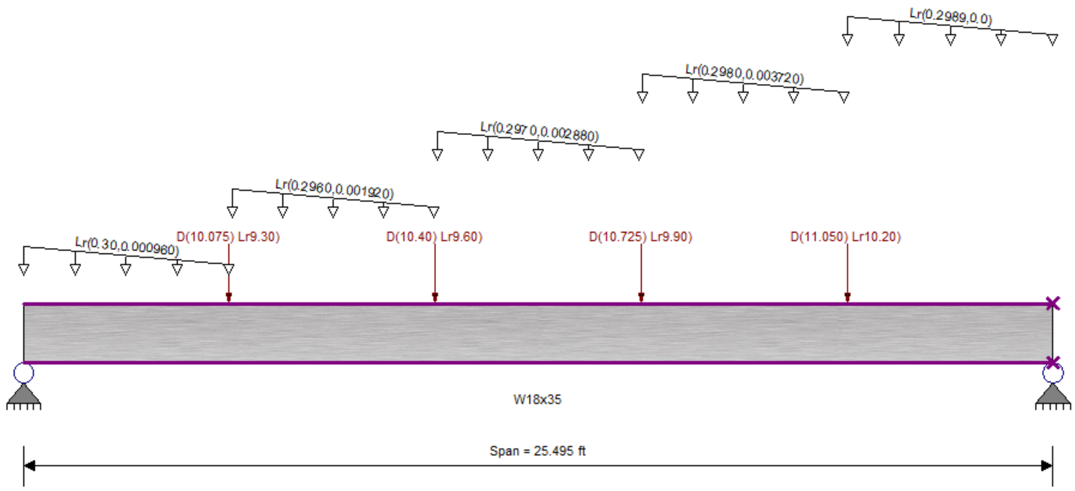


If the infill beams supported by this skewed girder are sampled with a dense tributary width contour, the resulting load diagram will reflect the fact that the skewed girder reduces the trib width near the end of the beam by acting as deck support.



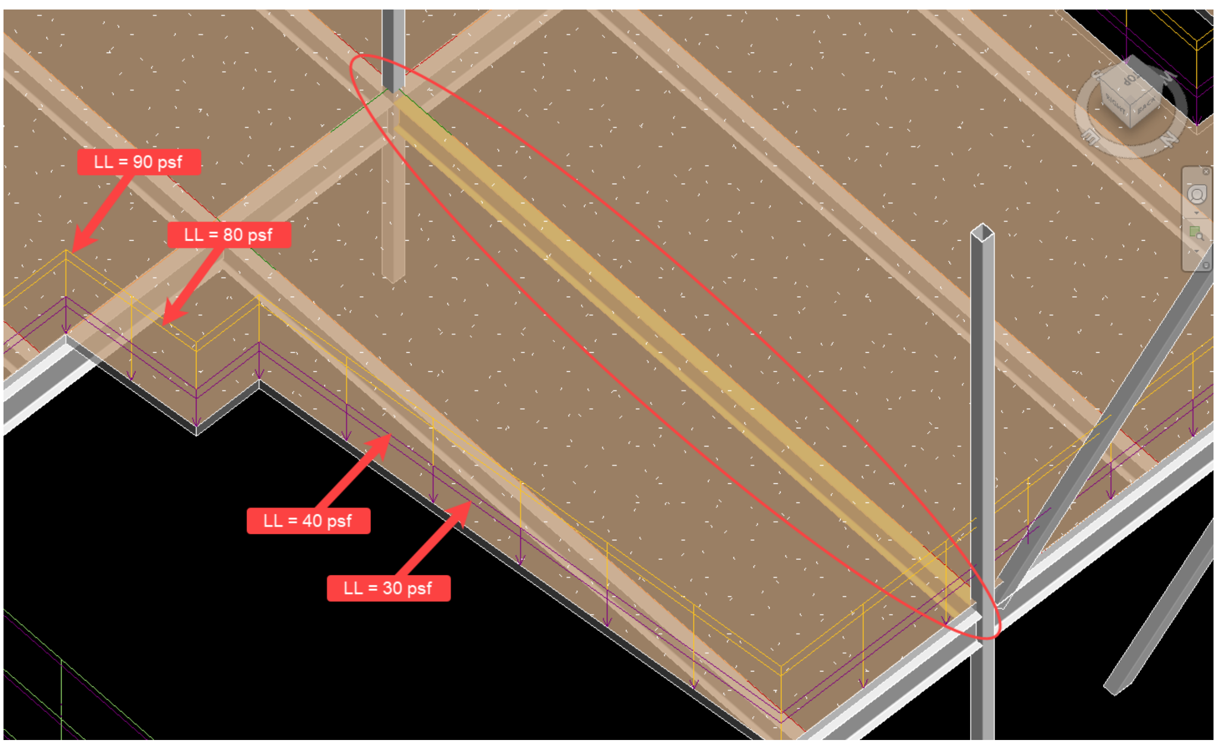


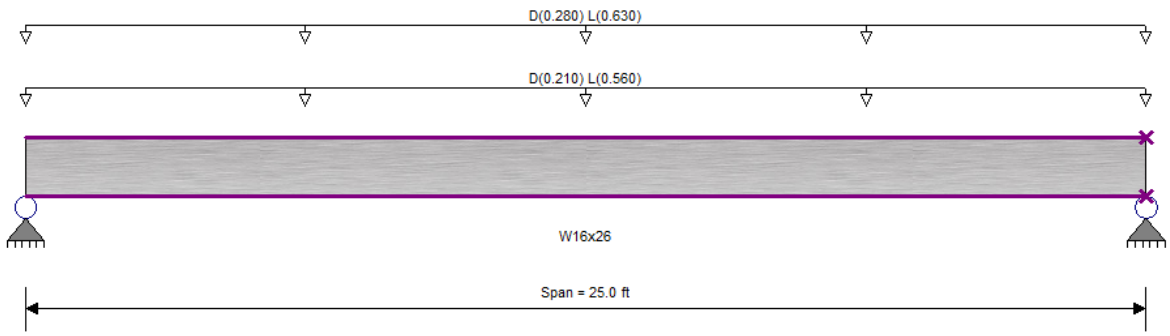
If it is not desirable to model the behavior in this manner, the beam may always be designed with fully uniform linear loads by sampling at a lower density that does not detect the reduction in tributary width near the beam end. If using the girder point load tool, sampling the infill beams at a lower density will result in reactions being estimated from full uniform reactions. Similarly, the design of the skewed girder will also accurately the small amount of trapezoidal load that the girder experiences due to supporting a short segment of deck between each infill beam.

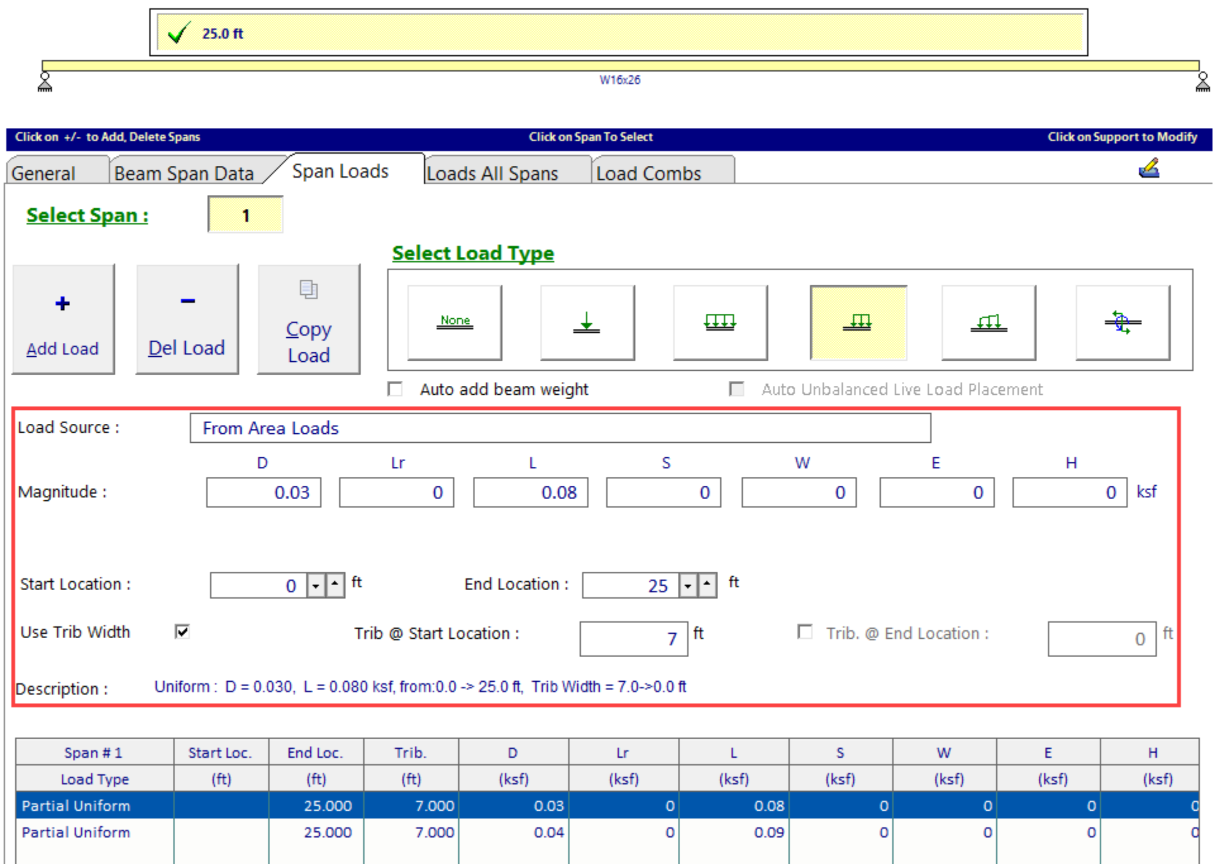


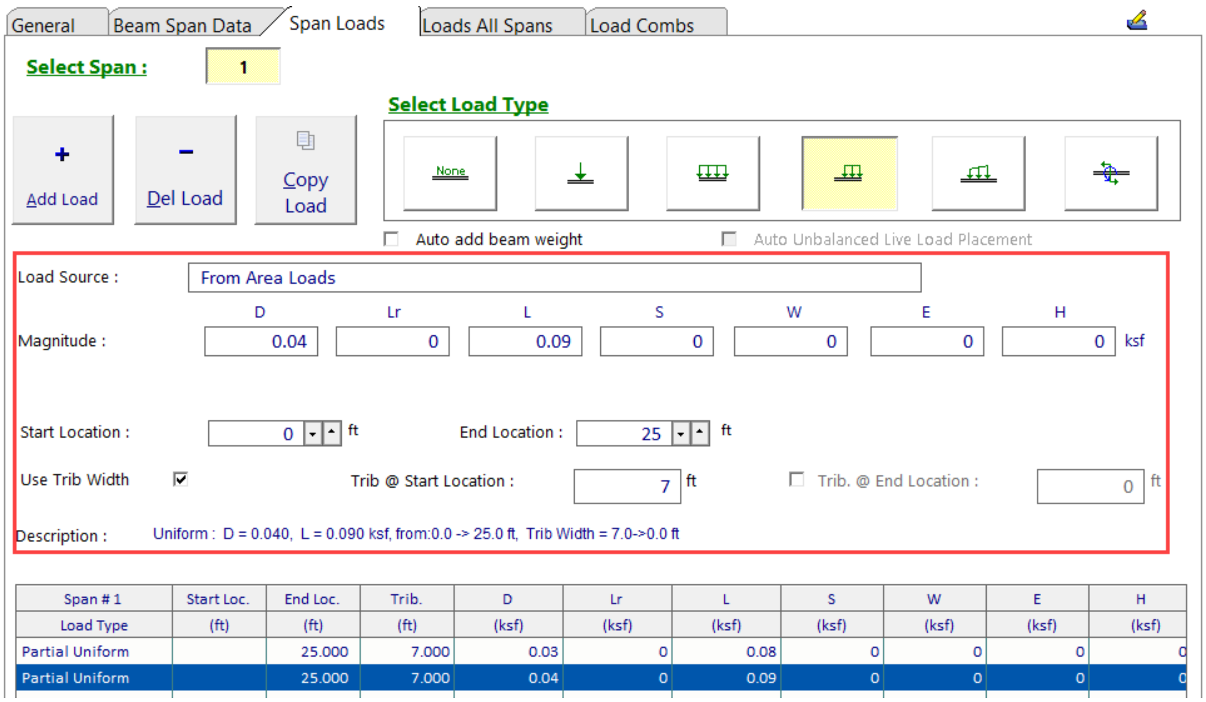
10.4.7 Redundant Area Load Cases

ENERCALC for Revit permits users to load a given floor with more than one area load of a particular case if the need arises. In such cases, the resulting linear loads in the ENERCALC SEL interface will populate in the “Span Loads” tab. Note that capturing this behavior requires that the beam be launched with a “Complete Contour” tributary width sampling mode.



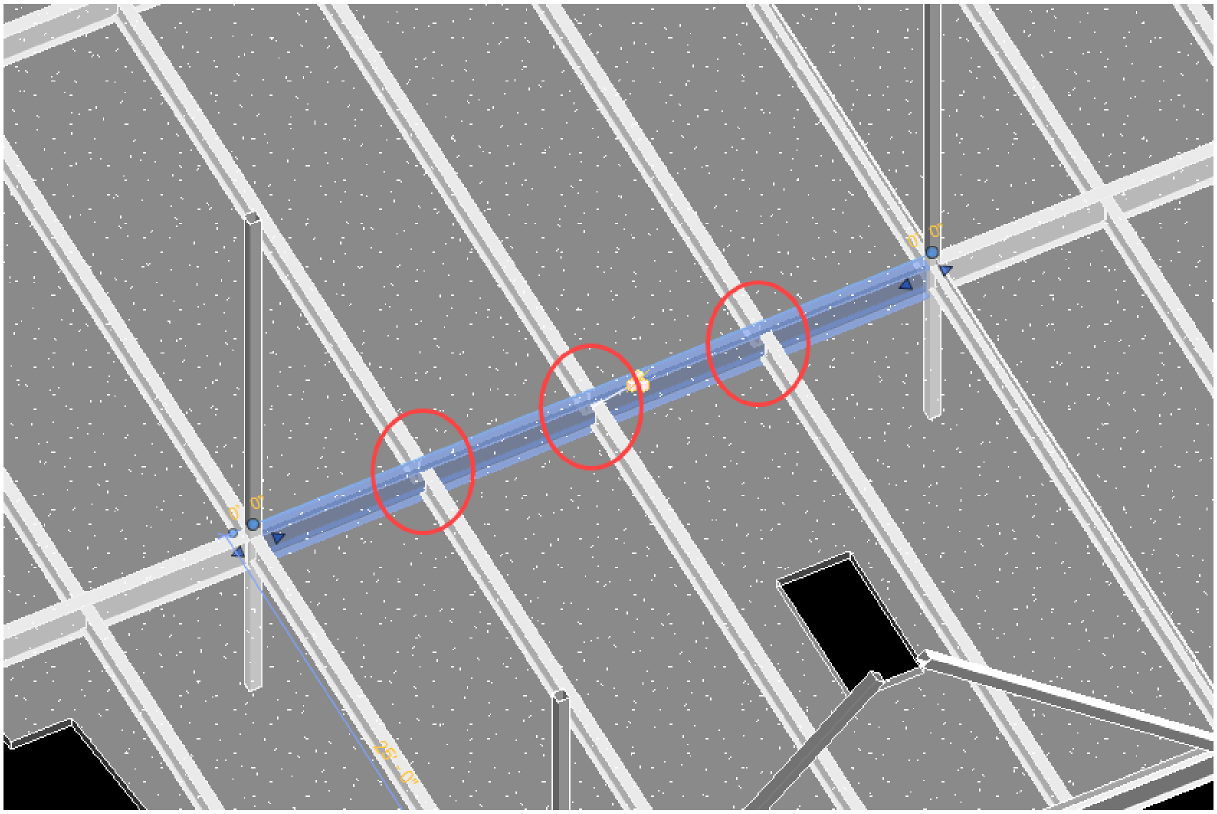

D(0.280) L(0.630)
D(0.210) L(0.560)
W16x26
Span = 25.0 ft


25.0 ft
W16x26
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Select Span :
1
Select Load Type
Add Load
Del Load
Copy Load
None
Auto add beam weight
Auto Unbalanced Live Load Placement
Load Source :
From Area Loads
D
Lr
L
S
W
E
H
Magnitude :
0.03
0
0.08
0
0
0
0
ksf
Start Location :
0
ft
End Location :
25
ft
Use Trib Width
Trib @ Start Location :
7
ft
Trib. @ End Location :
0
ft
Description :
Uniform : D = 0.030, L = 0.080 ksf, from:0.0 -> 25.0 ft, Trib Width = 7.0->0.0 ft
Span # 1 Load Type
Start Loc. (ft)
End Loc. (ft)
Trib. (ft)
D (ksf)
Lr (ksf)
L (ksf)
S (ksf)
W (ksf)
E (ksf)
H (ksf)
Partial Uniform 25.000 7.000 0.03 0 0.08 0 0 0 0
Partial Uniform 25.000 7.000 0.04 0 0.09 0 0 0 0



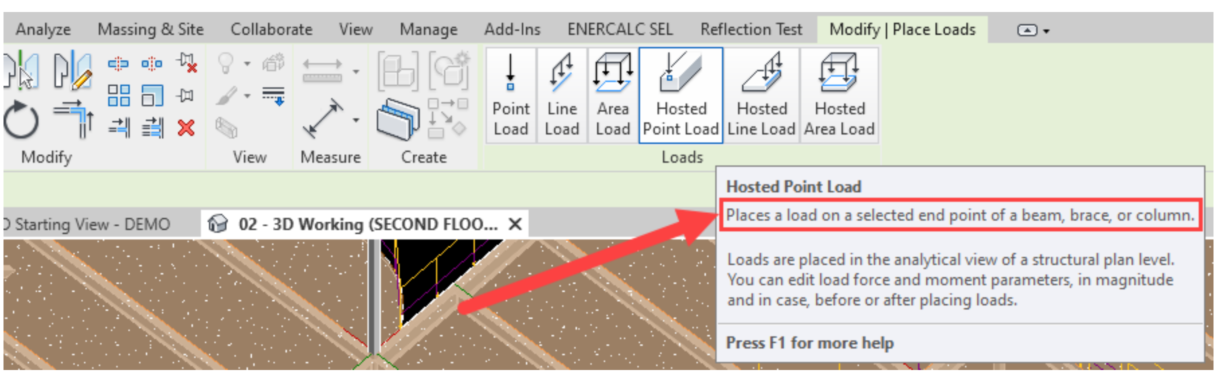
10.4.8 Girder Point Load Generator Tool

When designing a girder, users will often find it necessary to account for the reactions exerted by each of the infill beams that the girder supports. While this task is sometimes done using an equivalent line load approximation for the sake of simplicity, ENERCALC for Revit provides the user with a variety of options for applying these reactions to ensure fast and accurate design of girders.

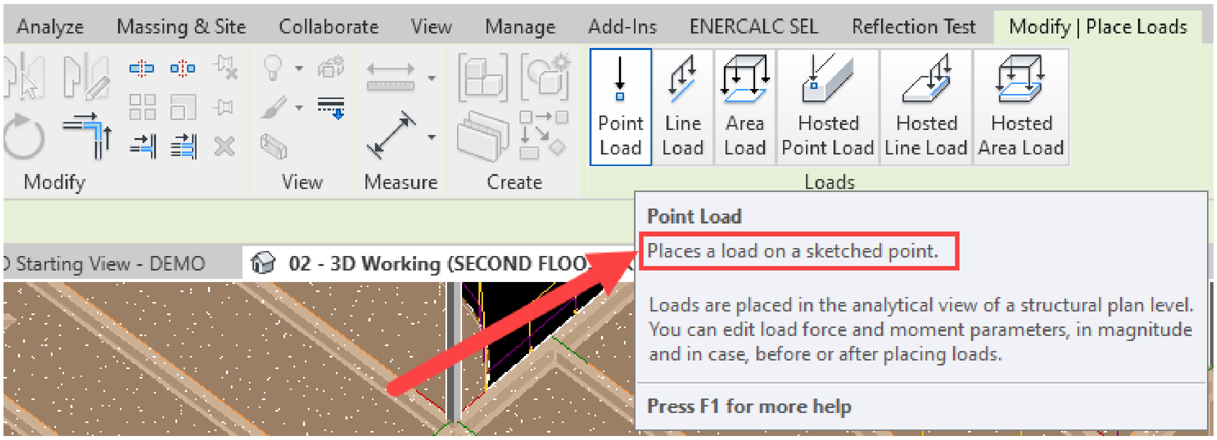


Option 1: Non-Hosted Point Loads

It's always possible to explicitly apply point loads to a girder manually using the native Revit interface (via the Revit Analyze ribbon tab). Unfortunately, per native constraints in Revit, loads hosted to the ***girder itself*** (Analyze > Loads > Hosted Point Loads) would only be permitted to be defined at the girder's two end joints. Therefore hosting loads to the girder itself is not helpful for girder design.



This means that loads at arbitrary intermediate locations along the length of a girder would need to be placed using Analyze > Loads > Point Loads. This native Revit tool permits the user to place a point load of arbitrary magnitude and direction at any location in the 3D model environment. However, the use of free-floating non-hosted loads can be tedious due to the lack of geometry control and doesn't take full advantage of the geometry and loading information already contained in a typical Revit model. The other loading options available involve varying degrees of automation to reduce this difficulty.



If users choose to model point loads on a girder using non-hosted point loads, the loads will be detected and automatically included in the calculation upon launch, as discussed in "Launching With Revit Non-Hosted Loads 162".

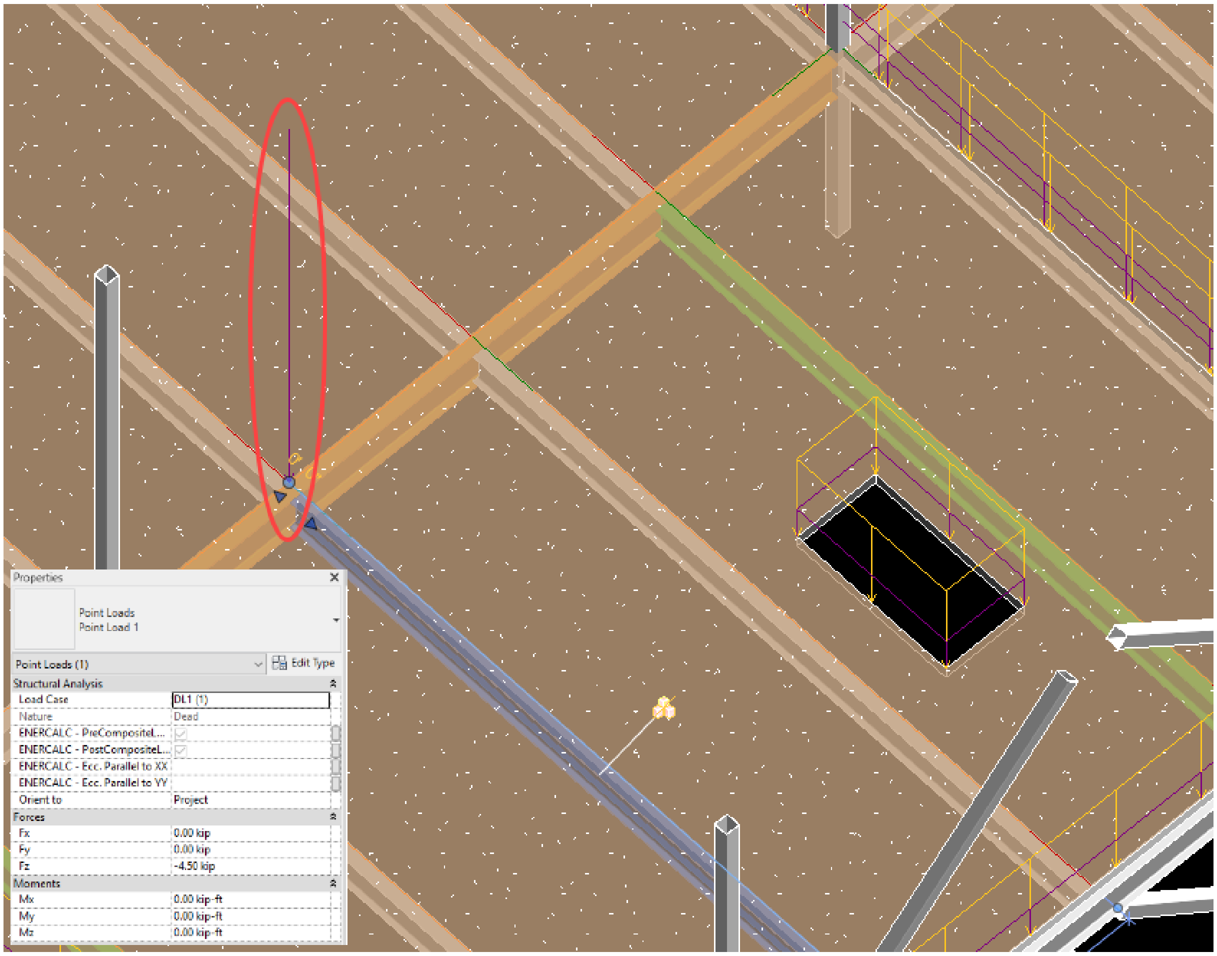
Option 2: Load-Linked Beam Reactions

The first and most efficient alternative to using manually applied free-floating point loads is to instead directly design each of the infill beams using ENERCALC for Revit. This allows users to take advantage of "Load-Linking" when designing the girder. This topic is discussed in detail under "Beam/Girder Load Linking 250".

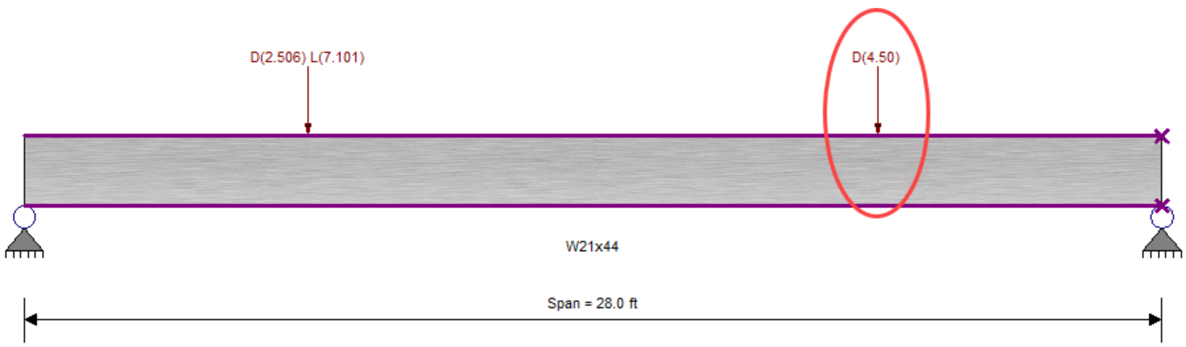
Option 3: Hosted Point Loads

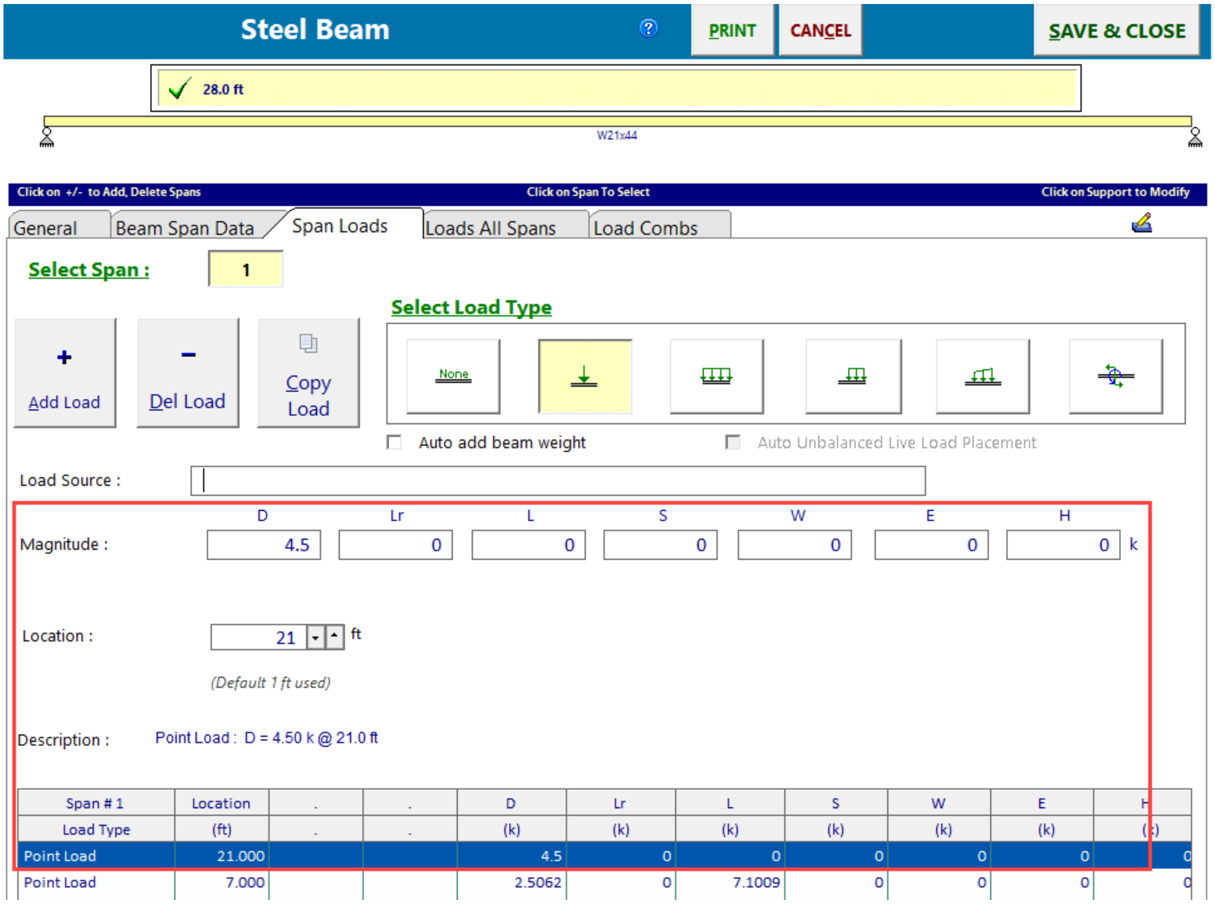
If the infill beams supported by a girder are not designed using ENERCALC for Revit, then there will not be any stored reactions available for use via automatic load linking when the girder calculation is launched. This is frequently the case in the early stages of a project before beam design has been performed, as well as in projects where the infill framing are manufactured elements not designed in ENERCALC (i.e., open-web steel joists, cold-formed steel joists, wood joists, etc.).

For girder designs where loading linking is unavailable, the next alternative is to apply point loads hosted (Analyze > Loads > Hosted Point Loads) to the bearing end joint of each of the infill beams.



If users choose to model hosted point loads on the ends of the infill beams, the loads will be detected and automatically included in the girder calculation upon launch, as discussed in "Launching With Revit Hosted Loads 171".



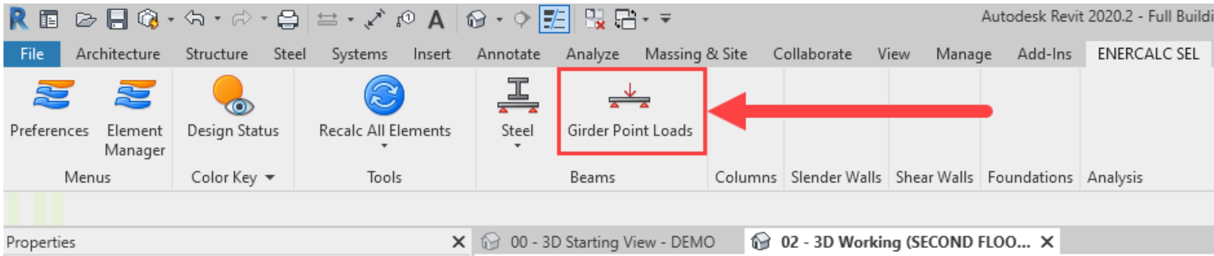


Option 4: Girder Point Load Generation

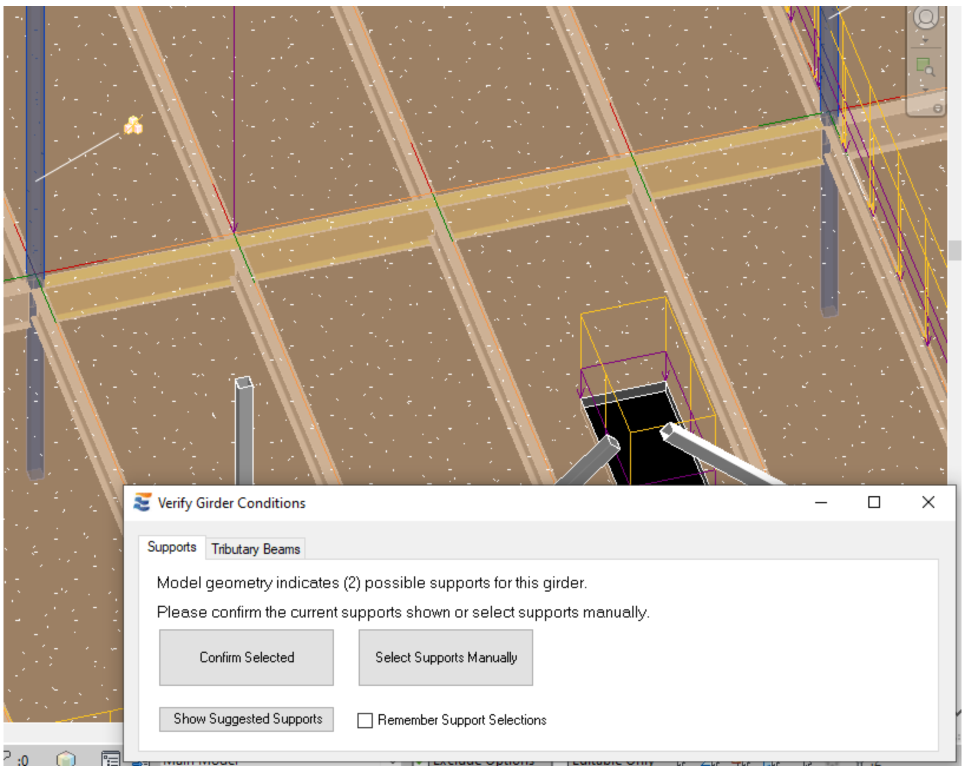
Although reactions may be modeled manually at the end joints of infill beam as described in the previous section, this process is time-consuming and necessitates hand-calculation of the appropriate reactions for each beam. This requires knowledge of a variety of information for each individual beam, including the following:

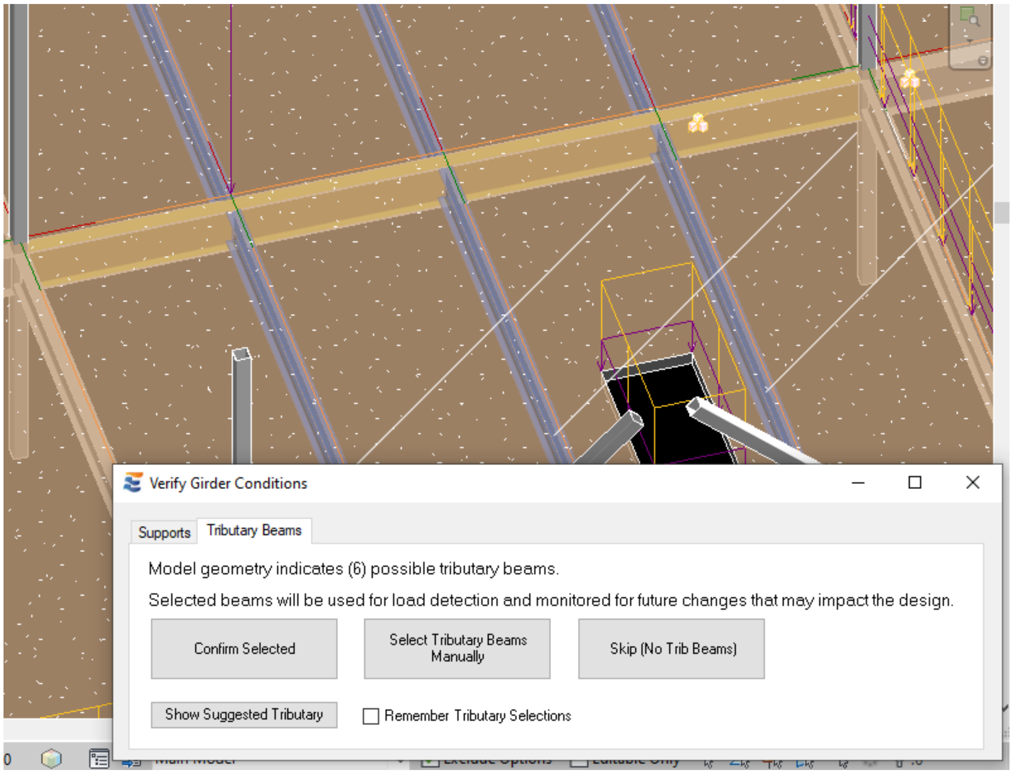
- Beam Span
- Beam Spacing / Tributary width
- Area loads on the roof or floor

Rather than forcing the user to obtain this information manually, hand-calculate, and manually apply forces, the “Girder Point Loads” tool provides the ability to perform this task automatically by detecting the critical design information from the Revit model. This tool is accessed from the “Beams” panel of the ENERCALC for Revit ribbon tab.



Similar to the launch of a typical beam calculation, the "Girder Point Loads" tool begins with an approval window for the user to verify critical information about the girder. This includes verification of both supports and tributary (infill) beams.





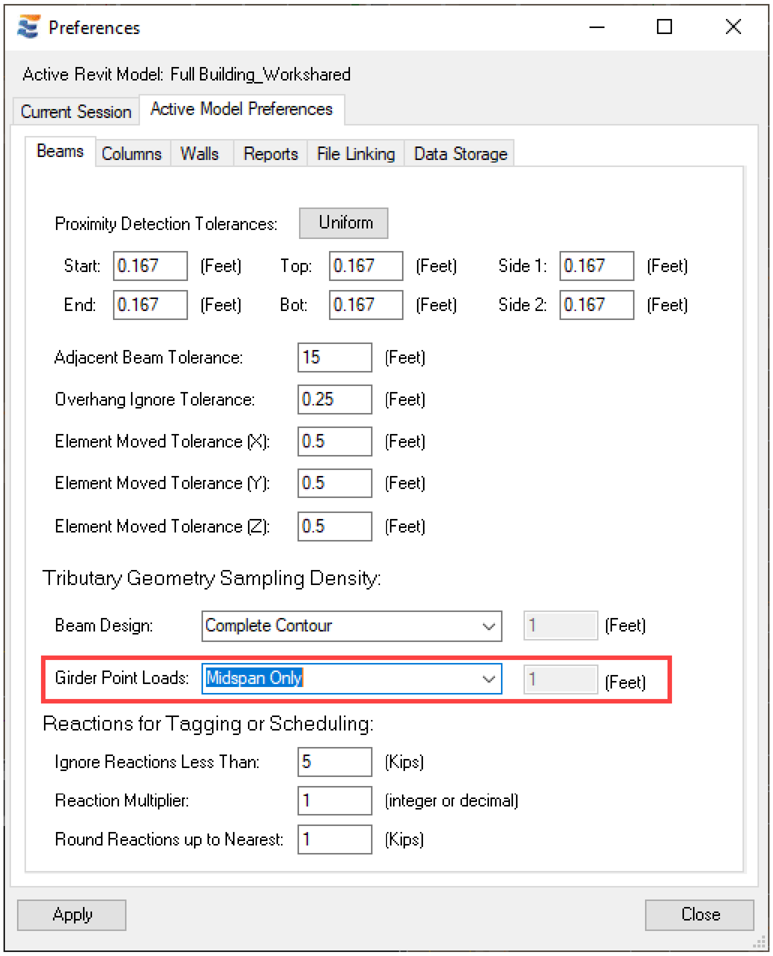
The navigation of this launch window is identical to that described in “Beam Supports 110” and “Tributary Beams 136”. When the approval process concludes, the girder point load tool will automatically test each of the tributary beams and detect the information necessary to compute the reaction each will exert on the girder:

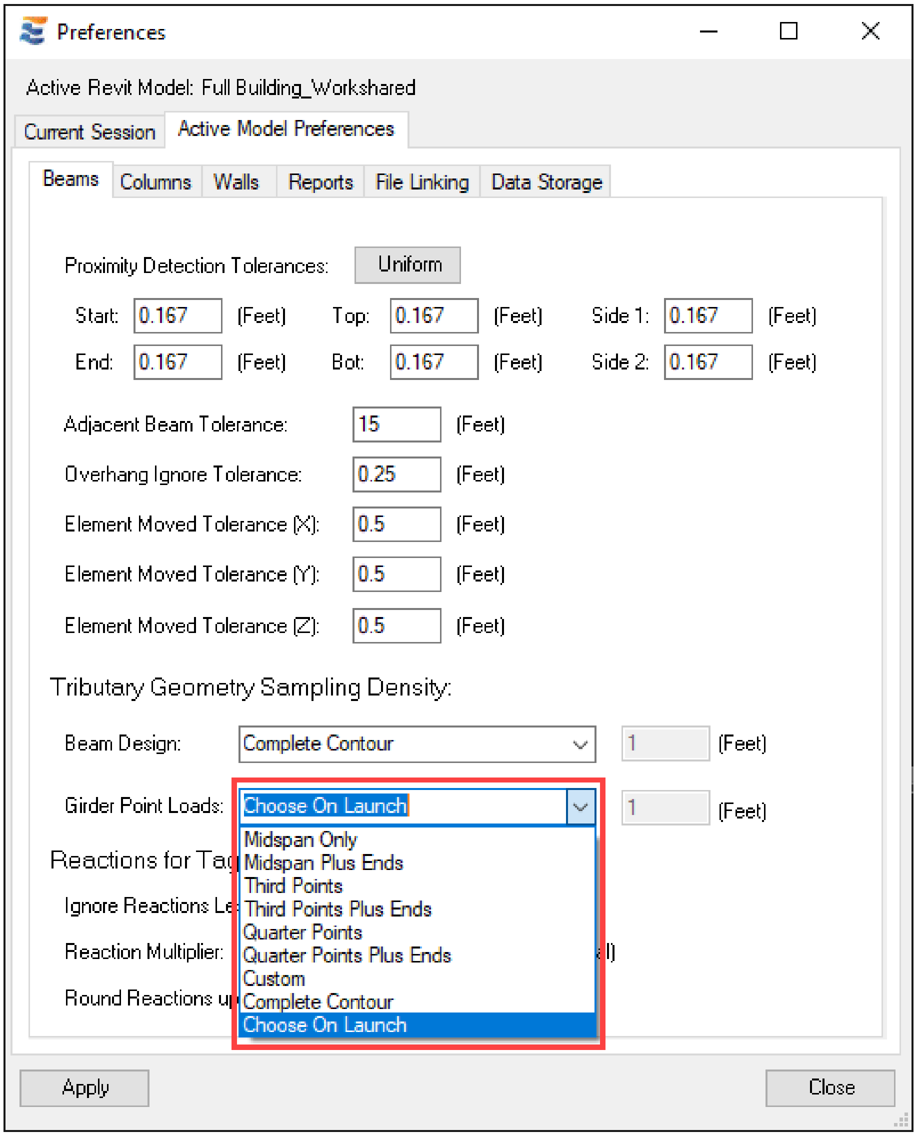
- Beam Span
- Beam Spacing / Tributary width
- Area loads on the roof or floor

This information is used to compute an estimated reaction from each infill beam by assuming uniformly loaded simple span behavior:

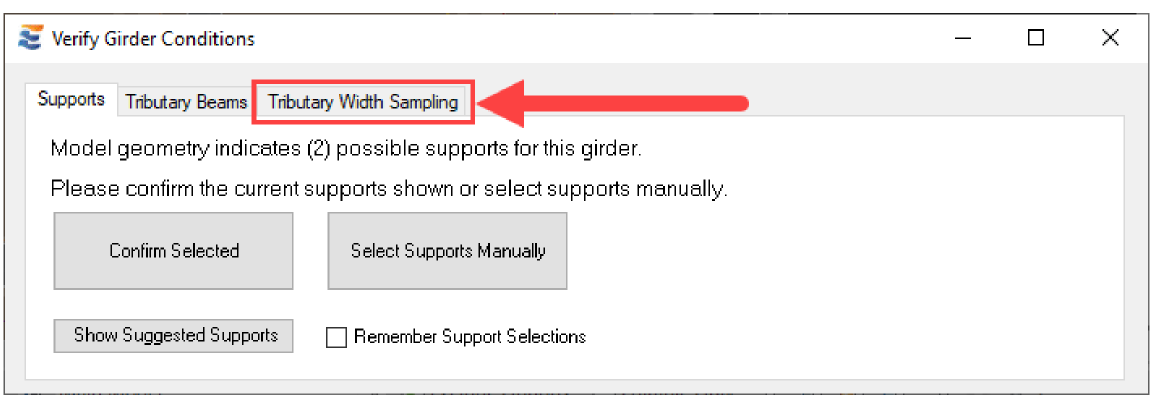
Reaction = (Area Load * Tributary Width) * Span / 2

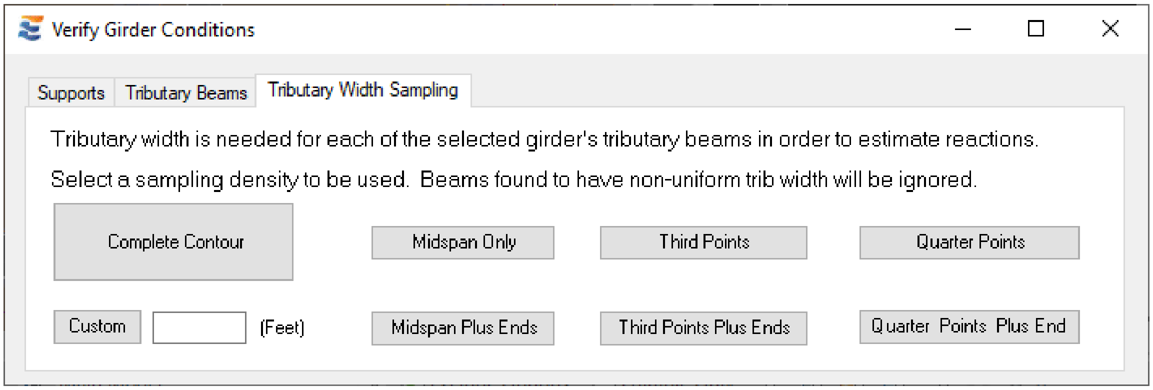
The tributary width is obtained using the same process as previously described in “Tributary Width Sampling 179” and “Sampling Density Controls 185”. The density used for sampling may be specified in the ENERCALC for Revit Preferences menu, or may be specified at time of launch.


Preferences
Active Revit Model: Full Building_Workshared
Current Session
Active Model Preferences
Beams
Columns
Walls
Reports
File Linking
Data Storage
Proximity Detection Tolerances:
Uniform
Start: 0.167 (Feet)
Top: 0.167 (Feet)
Side 1: 0.167 (Feet)
End: 0.167 (Feet)
Bot: 0.167 (Feet)
Side 2: 0.167 (Feet)
Adjacent Beam Tolerance: 15 (Feet)
Overhang Ignore Tolerance: 0.25 (Feet)
Element Moved Tolerance (X): 0.5 (Feet)
Element Moved Tolerance (Y): 0.5 (Feet)
Element Moved Tolerance (Z): 0.5 (Feet)
Tributary Geometry Sampling Density:
Beam Design: Complete Contour 1 (Feet)
Girder Point Loads: Midspan Only 1 (Feet)
Reactions for Tagging or Scheduling:
Ignore Reactions Less Than: 5 (Kips)
Reaction Multiplier: 1 (integer or decimal)
Round Reactions up to Nearest: 1 (Kips)
Apply
Close



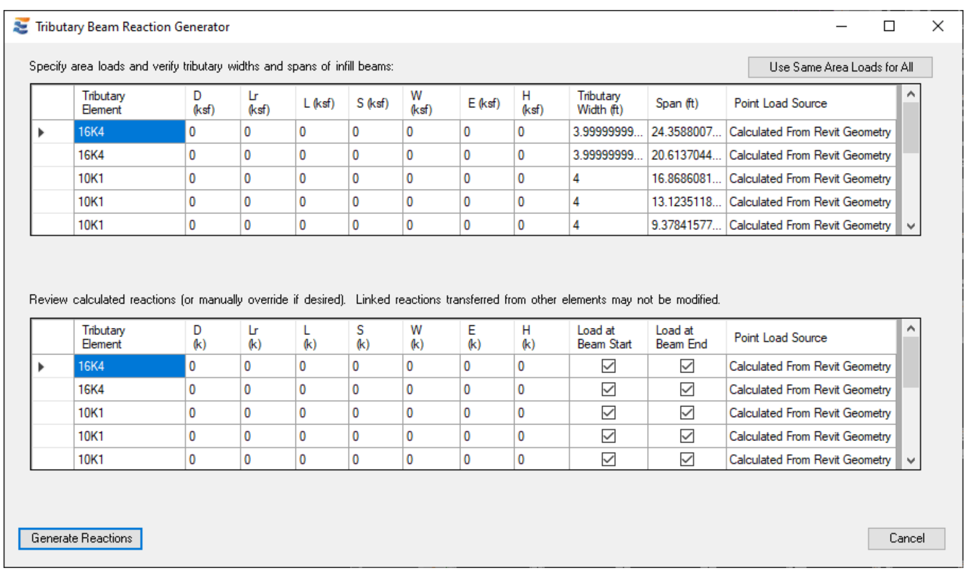
When the sampling density is set to “Choose On Launch”, an additional tab will appear on the launch window for the user to choose a sampling density.



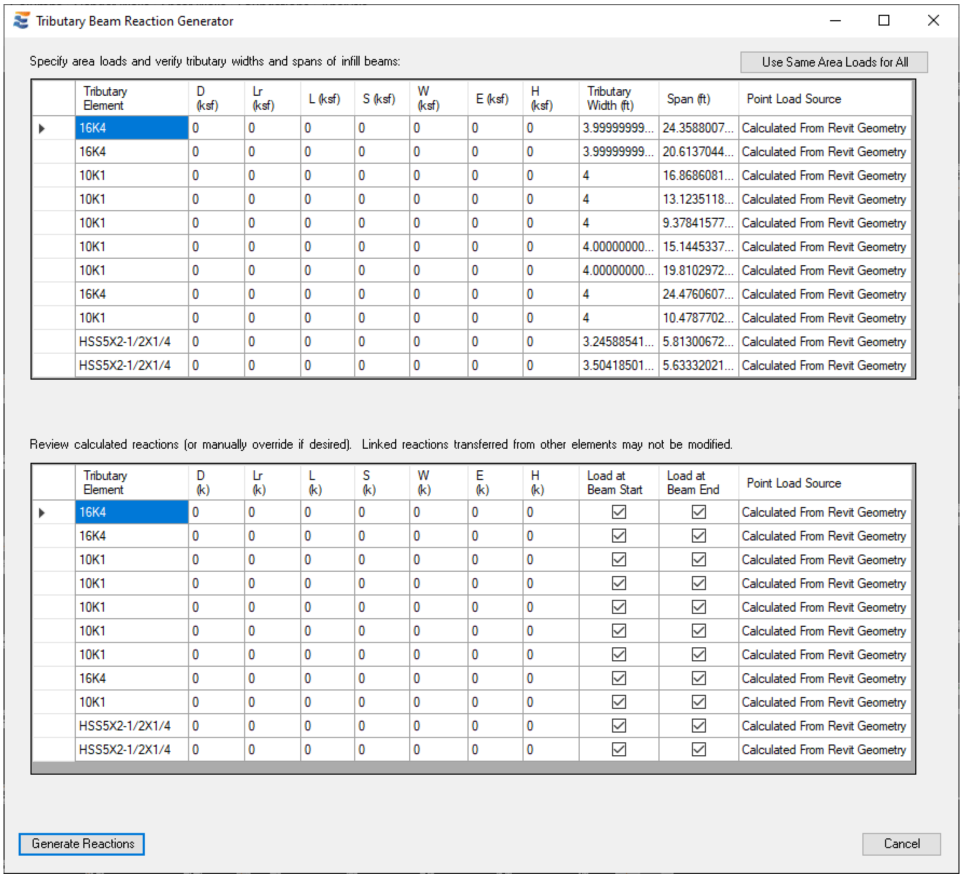


Once trib sampling is complete for a given beam, the individual tributary width measurements are compared to each other to validate the assumption of uniformly loaded behavior (excepted when sampling at “Midspan Only”, in which case there is only one value). A beam whose tributary width measurements along the beam do not match will not be used for computing reactions. As noted in the approval window, the user has the ability to specify a wide variety of different sampling conditions, ranging from sparse (Midspan Only) to very dense (Complete Contour). Users should bear in mind that denser sampling modes are more likely to detect subtle variations which render a contour non-uniform (and therefore result in a beam being disqualified for automatic reaction generation by this tool), while sampling at “Midspan Only” will never result in a beam being disqualified from point load generation.

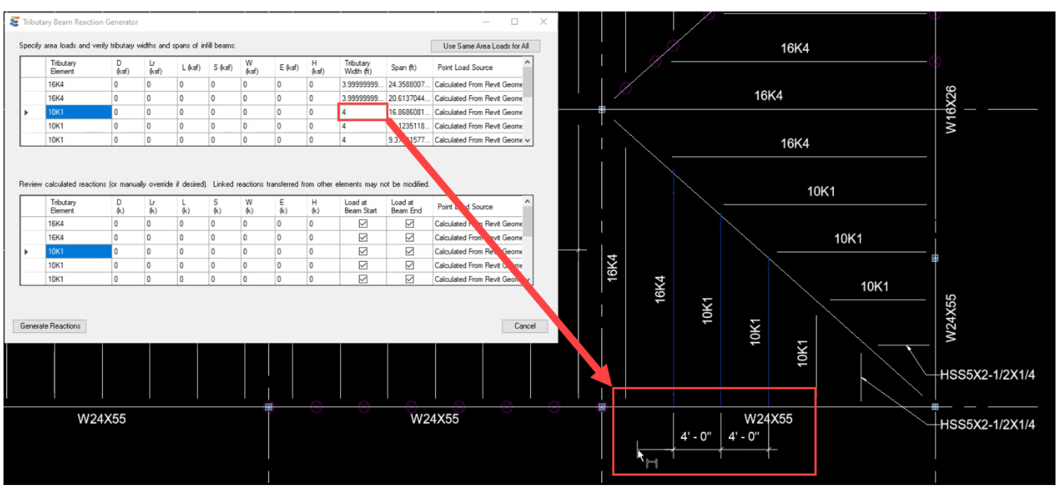
When the tributary width sampling process is complete for all tributary beams, the Reaction Generator window will load. The window is divided into two halves, where the upper summary table displays the input data detected for each infill beam and the lower half displays the reactions computed from this data.

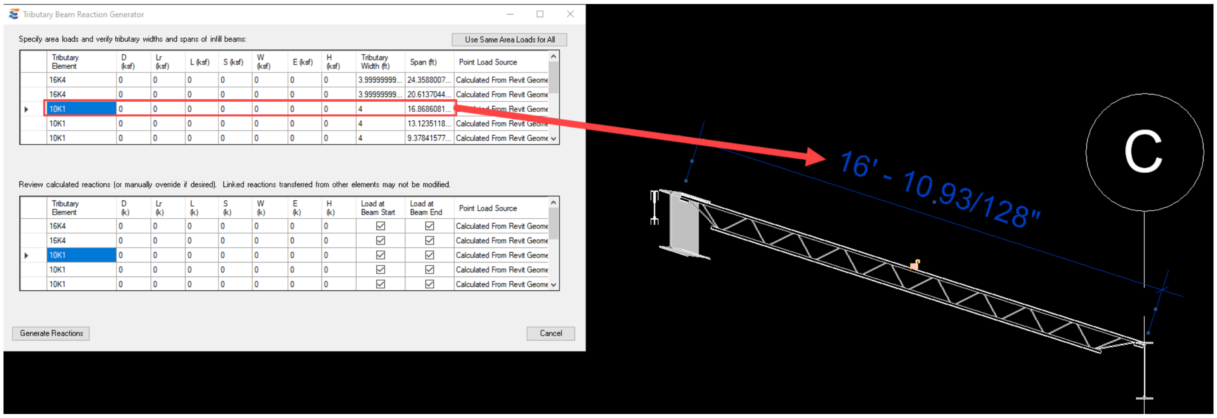


For ease of viewing, the Reaction Generator tables may be expanded by using the drag control at the lower edge of the window. Enlarging the window will cause both of the tables to expand.

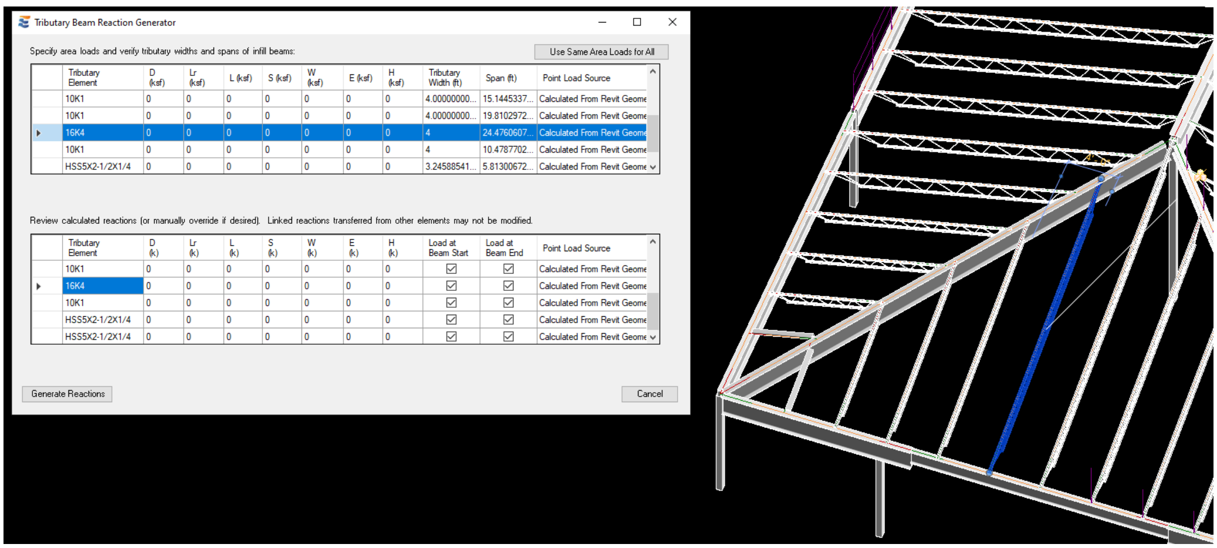


Similar to the calculation launch window, the Reaction Generator form allows the user to retain full navigation of the Revit interface to perform a variety of actions, including changing the active view, opening or closing views, and using dimension annotation or measure tools to verify the data shown in the table.

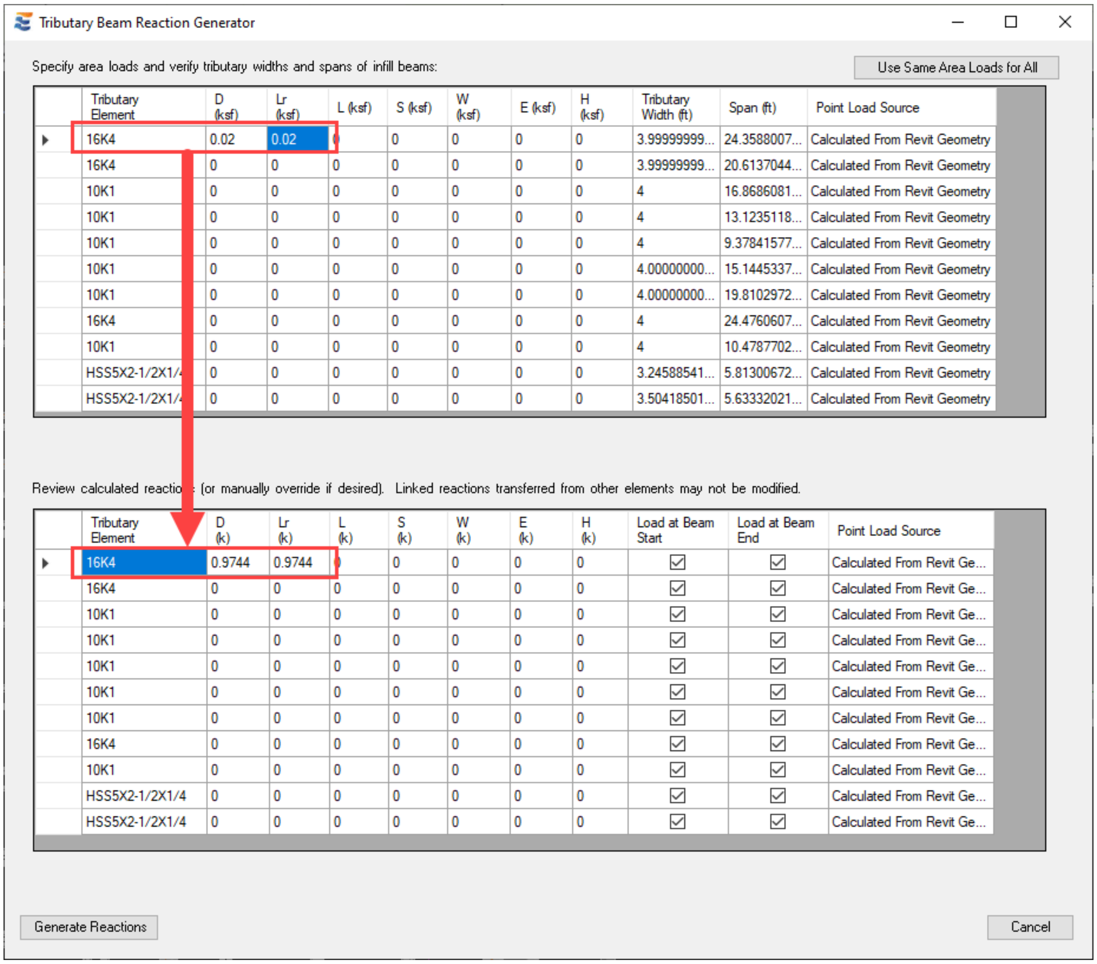




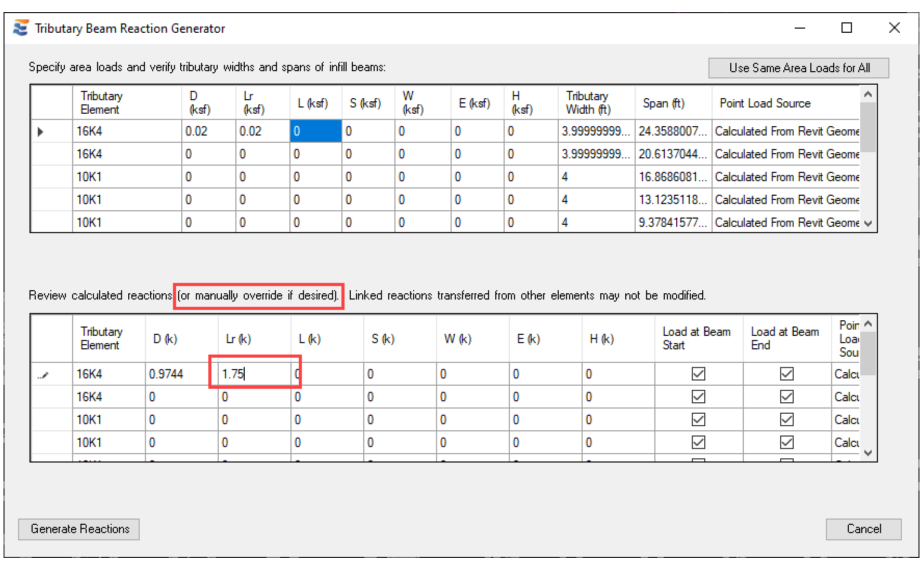
Selecting a specific row in the Reaction Generator tables automatically triggers selection of the corresponding element in the active Revit view.



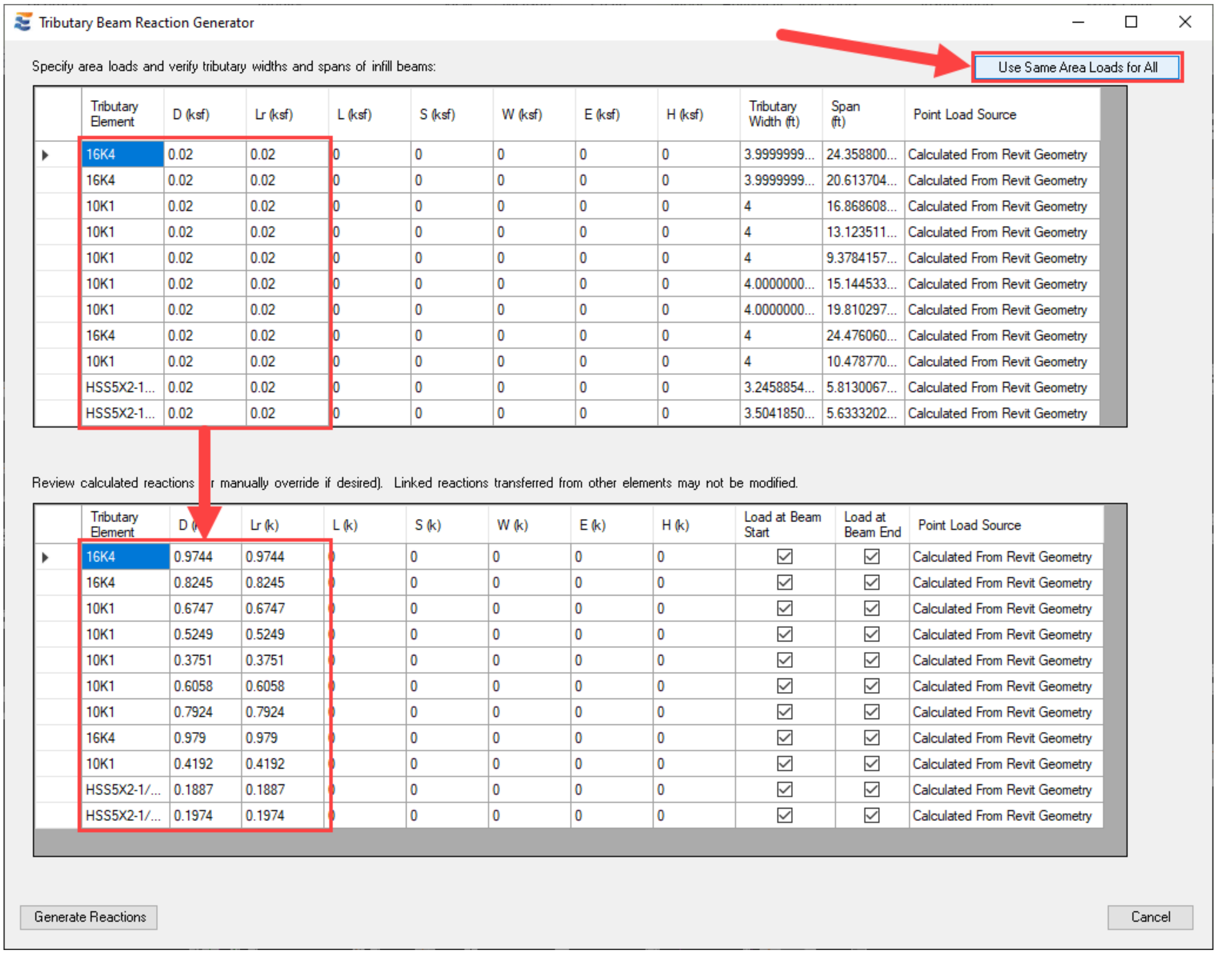
When Revit floors and area loads are not present on the tributary beams, the area loads in the input table will initially be shown with zero magnitudes. The desired area loads for reaction calculation may be supplied manually by typing in the appropriate cells. The resulting reactions will update in the lower table automatically.



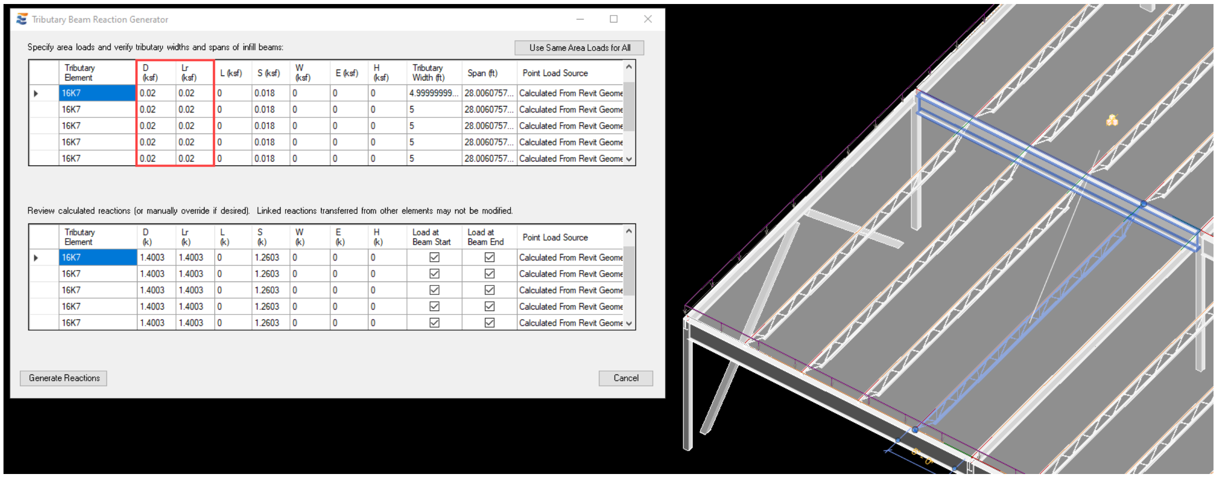
In addition to manually setting the area loads, the user may directly override the reactions in the lower table by typing in the appropriate cell.



Different area loads may be supplied in each row or, if desired, the same area loads may be applied to all reaction calculations in the table by clicking the “Use Same Area Loads for All” button in the upper right hand of the window. Using this button will populate all subsequent rows with the first user-supplied area load magnitudes found in the table.



When Revit floors and area loads are found on the tributary beams, the magnitudes of the area loads will automatically populate in the Reaction Generator input table. The load magnitudes may be manually overwritten in the table if desired and the reactions will update accordingly, but this will not cause any change to the properties of the actual area load elements in the Revit model.



Note also that the “Girder Point Loads” tool also has the ability to interpret and display reaction loads that have already been applied to the girder by other means. This includes alerting the user of point loads created via Option 2 and Option 3 discussed previously, but not Option 1. Non-Hosted point loads referenced above as Option 1 are floating in 3D space and do not have a direct relationship with the girder that can be discerned by this tool. Even though not displayed in the table, non-hosted loads may still be present in the model and will be included in the girder design calculation if they are located appropriately.

The “Girder Point Loads” tool automatically considers the presence of pre-existing reaction forces when evaluating whether a reaction should be calculated for each individual beam. The following logical framework establishes precedence for obtaining reactions on a beam-by-beam basis:

- If a girder has explicitly modeled loads as non-hosted Revit load elements (Option 1), these forces will not be presented for review in the reaction table. Any non-hosted loads will be additive with the loads generated by the “Girder Point Loads” tool. It will not remove or replace them.

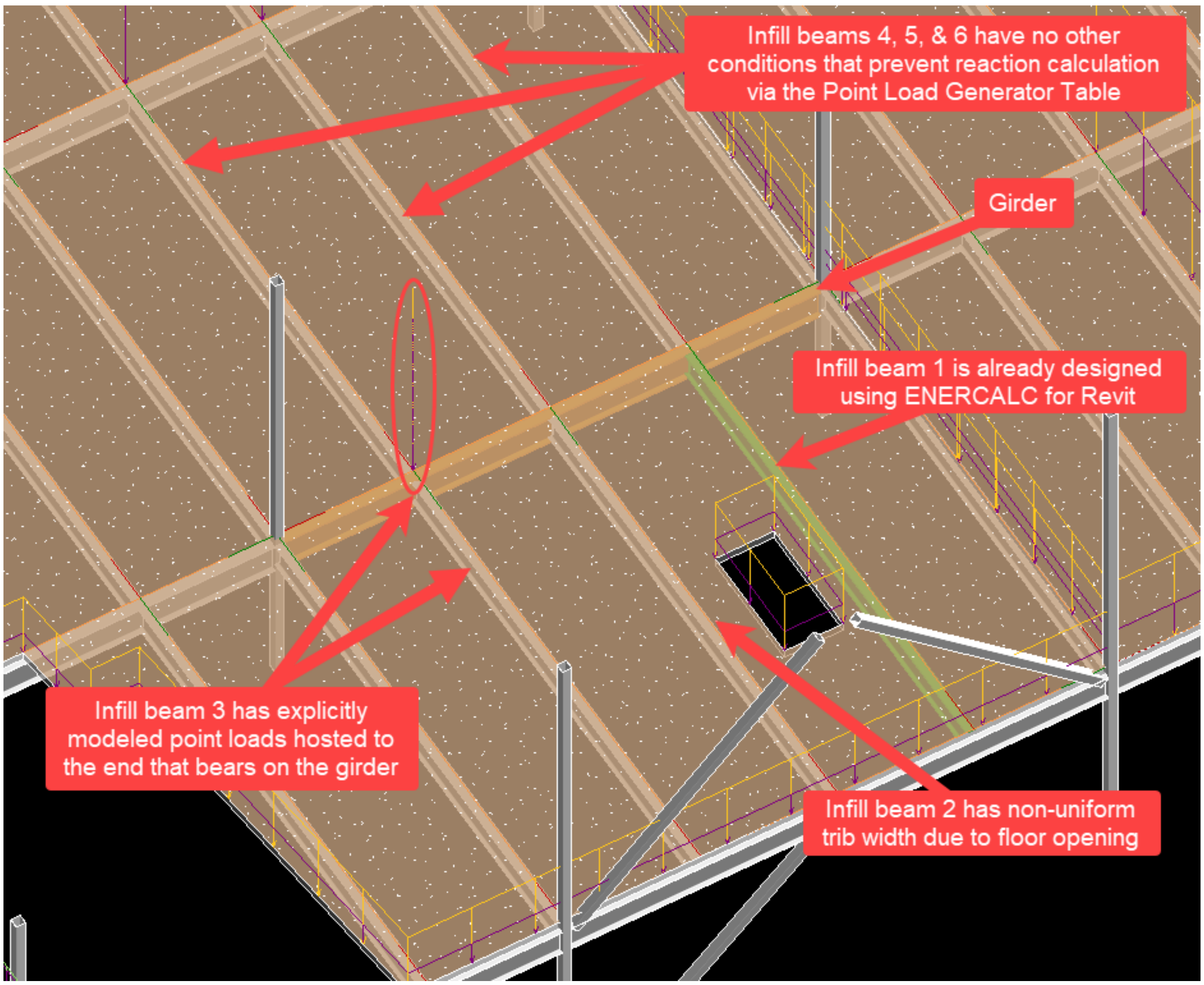
- If a particular infill beam has previously been designed using ENERCALC for Revit and already has a reaction at the connection to the girder via “Load-Linking” (Option 2), that reaction will be presented in the girder point load table.

- If a particular infill beam has NOT been previously been designed using ENERCALC for Revit, the program will look to see if the user has explicitly applied a reaction as a hosted load at the end of the infill beam (Option 3). If so, that reaction will be presented in the girder point load table.

- If a particular infill beam has NOT been designed using ENERCALC for Revit (Option 2), and if the user has NOT explicitly applied a reaction as a hosted load at the end of the infill beam (Option 3), then the girder point load tool will present the span, tributary width, and area load found for that beam, along with the resulting estimated reaction force (if the beam is not disqualified on the basis of a non-uniform tributary width). The user may then review this information and/or interact with the table values as previously described.

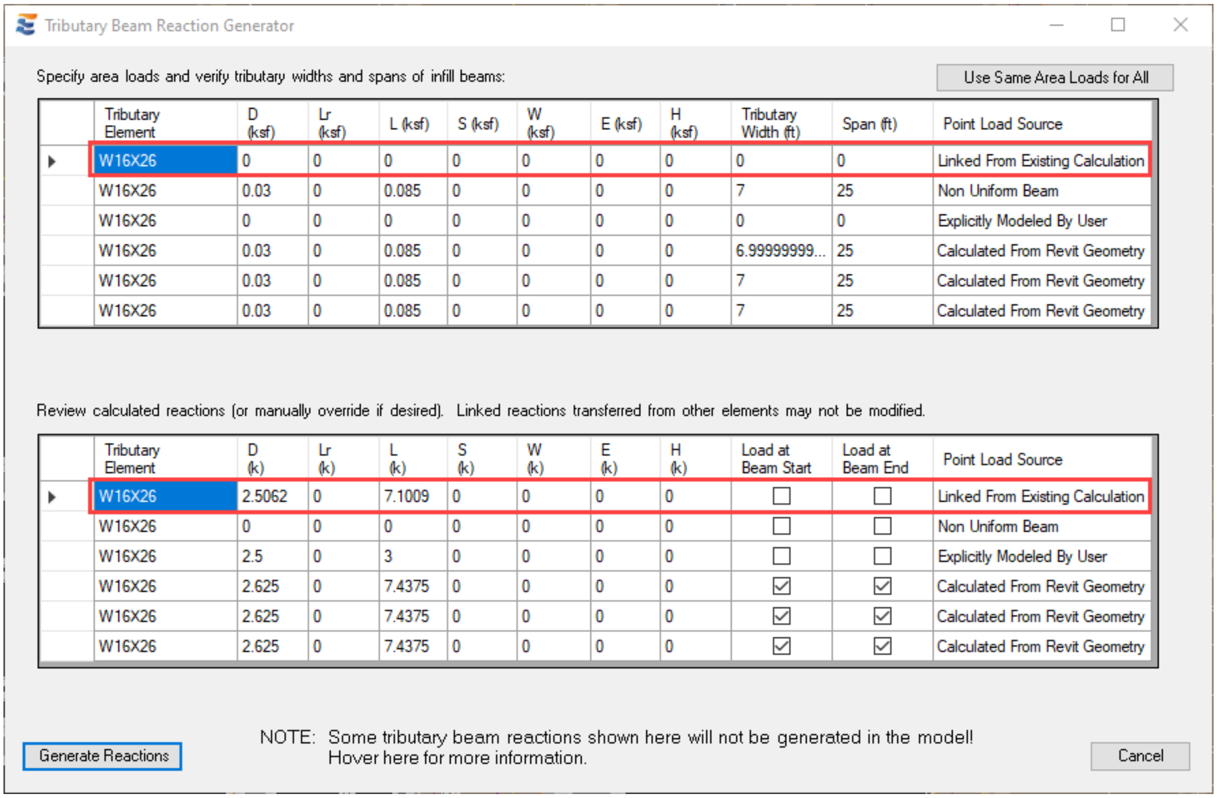
- If an infill beam is eligible for reaction generation due to not meeting Option 2 or Option 3, but is found NOT to have uniform tributary widths, the girder point load tool will place an entry in the table with no load applied. This is for the user's convenience to be alerted that the member was found, despite the fact that the program could not calculate an accurate reaction for that end of the member.

An example girder scenario is shown below, with all of the significant conditions that influence the information presented to the user in the girder load generator table.



Launching the "Girder Point Loads" tool for this girder will provide a summary of the beams whose reactions are already present in the Revit model, as well as the beams for which reactions are calculated. As discussed previously, it is necessary to sample the infill beam tributary widths at a density high enough to detect the opening in order to reproduce the results shown here.

Infill beam #1 is automatically found to have existing reactions stored from its own design calculation. The “Point Load Source” column at the right end of the table indicates “Linked From Existing Calculation”. As a result, the tributary width is not sampled and no model data are reported in the input table. The forces shown in the lower half of the window on the reaction table are the stored reactions found from the existing calculation. New reaction forces will **NOT** be generated for this beam.



Tributary Beam Reaction Generator

Specify area loads and verify tributary widths and spans of infill beams:

Use Same Area Loads for All

Tributary Element	D (ksf)	Lr (ksf)	L (ksf)	S (ksf)	W (ksf)	E (ksf)	H (ksf)	Tributary Width (ft)	Span (ft)	Point Load Source
W16X26	0	0	0	0	0	0	0	0	0	Linked From Existing Calculation
W16X26	0.03	0	0.085	0	0	0	0	7	25	Non Uniform Beam
W16X26	0	0	0	0	0	0	0	0	0	Explicitly Modeled By User
W16X26	0.03	0	0.085	0	0	0	0	6.99999999...	25	Calculated From Revit Geometry
W16X26	0.03	0	0.085	0	0	0	0	7	25	Calculated From Revit Geometry
W16X26	0.03	0	0.085	0	0	0	0	7	25	Calculated From Revit Geometry

Review calculated reactions (or manually override if desired). Linked reactions transferred from other elements may not be modified.

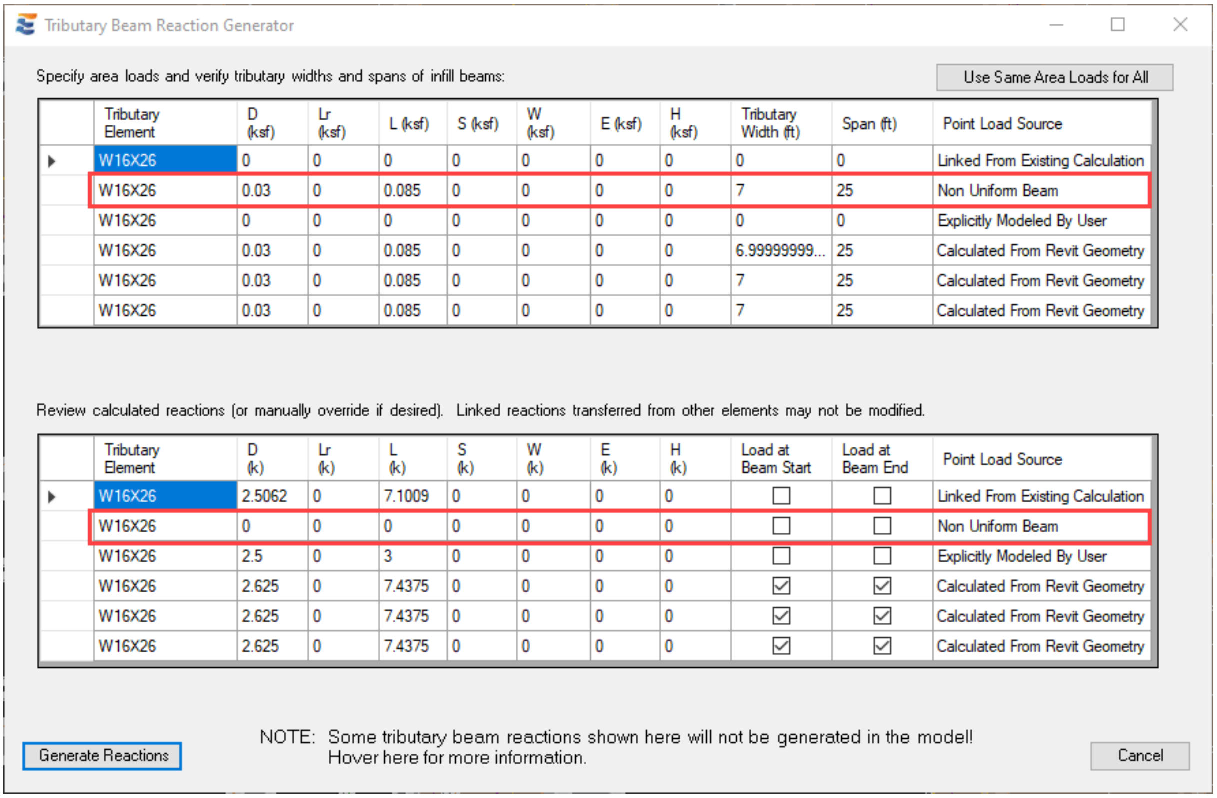
Tributary Element	D (k)	Lr (k)	L (k)	S (k)	W (k)	E (k)	H (k)	Load at Beam Start	Load at Beam End	Point Load Source
W16X26	2.5062	0	7.1009	0	0	0	0	☐	☐	Linked From Existing Calculation
W16X26	0	0	0	0	0	0	0	☐	☐	Non Uniform Beam
W16X26	2.5	0	3	0	0	0	0	☐	☐	Explicitly Modeled By User
W16X26	2.625	0	7.4375	0	0	0	0	☑	☑	Calculated From Revit Geometry
W16X26	2.625	0	7.4375	0	0	0	0	☑	☑	Calculated From Revit Geometry
W16X26	2.625	0	7.4375	0	0	0	0	☑	☑	Calculated From Revit Geometry

Generate Reactions

NOTE: Some tributary beam reactions shown here will not be generated in the model!
Hover here for more information.

Cancel

Because of the high-density sampling which detects the floor opening, infill beam #2 is automatically found to have non-uniform tributary width along its length. The “Point Load Source” column at the right end of the table indicates “Non Uniform Beam”. As a result, the tributary width, span, and area loads found in the model are reported in the input table for reference only. There is no calculated reaction shown in the lower half of the window on the reaction table. New reaction forces will **NOT** be generated for this beam.



Tributary Beam Reaction Generator

Specify area loads and verify tributary widths and spans of infill beams:

Use Same Area Loads for All

Tributary Element	D (ksf)	Lr (ksf)	L (ksf)	S (ksf)	W (ksf)	E (ksf)	H (ksf)	Tributary Width (ft)	Span (ft)	Point Load Source
W16X26	0	0	0	0	0	0	0	0	0	Linked From Existing Calculation
W16X26	0.03	0	0.085	0	0	0	0	7	25	Non Uniform Beam
W16X26	0	0	0	0	0	0	0	0	0	Explicitly Modeled By User
W16X26	0.03	0	0.085	0	0	0	0	6.99999999...	25	Calculated From Revit Geometry
W16X26	0.03	0	0.085	0	0	0	0	7	25	Calculated From Revit Geometry
W16X26	0.03	0	0.085	0	0	0	0	7	25	Calculated From Revit Geometry

Review calculated reactions (or manually override if desired). Linked reactions transferred from other elements may not be modified.

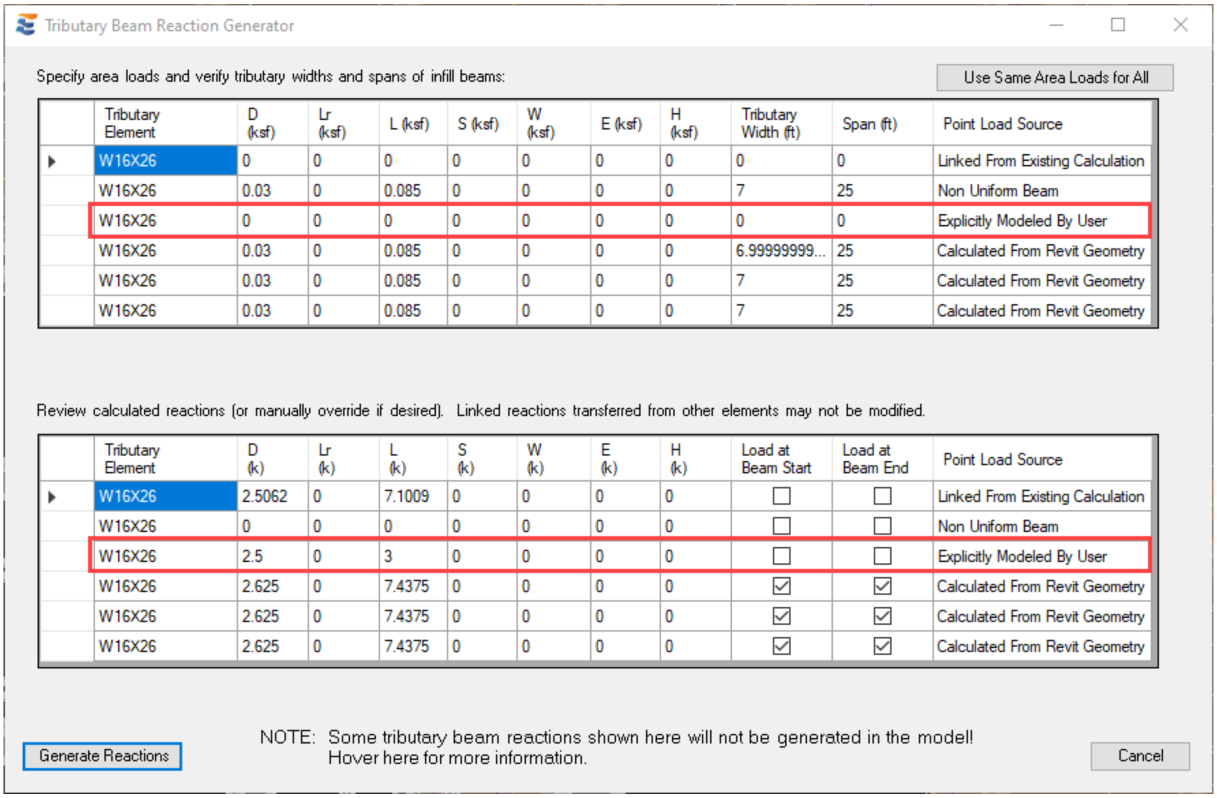
Tributary Element	D (k)	Lr (k)	L (k)	S (k)	W (k)	E (k)	H (k)	Load at Beam Start	Load at Beam End	Point Load Source
W16X26	2.5062	0	7.1009	0	0	0	0	☐	☐	Linked From Existing Calculation
W16X26	0	0	0	0	0	0	0	☐	☐	Non Uniform Beam
W16X26	2.5	0	3	0	0	0	0	☐	☐	Explicitly Modeled By User
W16X26	2.625	0	7.4375	0	0	0	0	☑	☑	Calculated From Revit Geometry
W16X26	2.625	0	7.4375	0	0	0	0	☑	☑	Calculated From Revit Geometry
W16X26	2.625	0	7.4375	0	0	0	0	☑	☑	Calculated From Revit Geometry

Generate Reactions

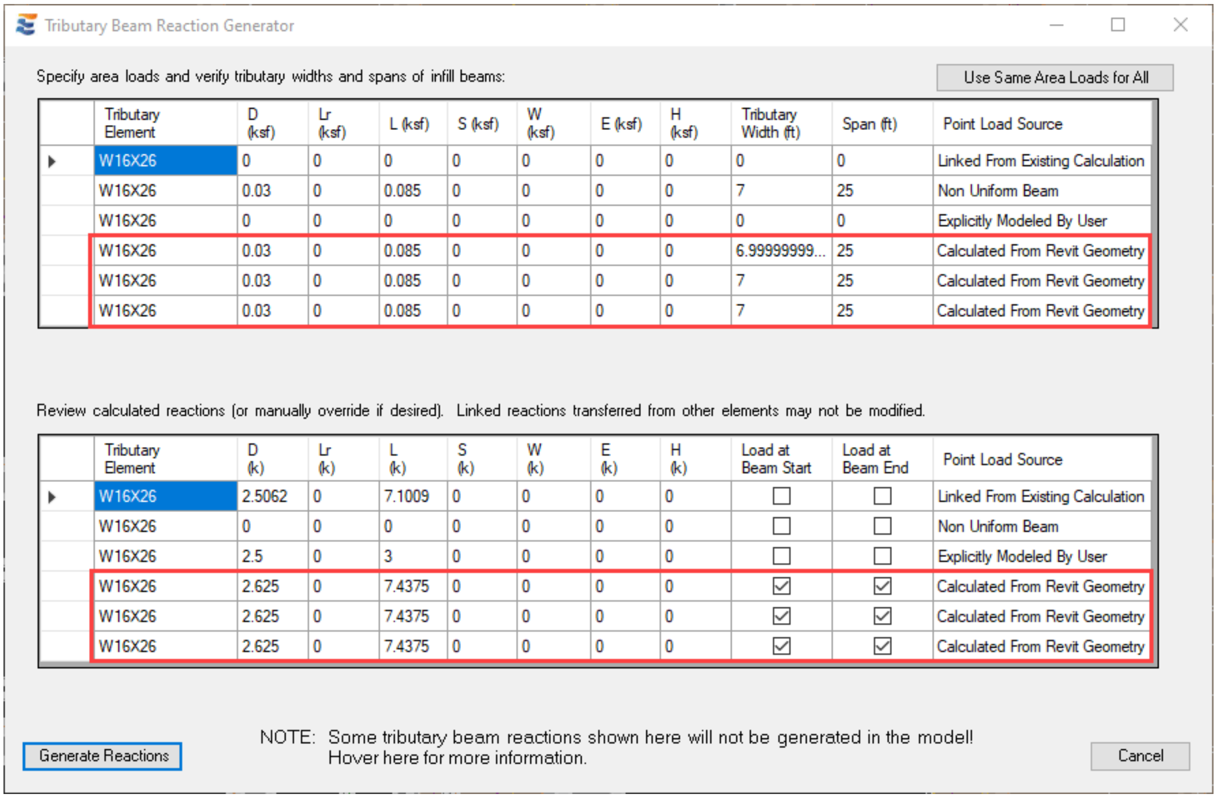
NOTE: Some tributary beam reactions shown here will not be generated in the model!
Hover here for more information.

Cancel

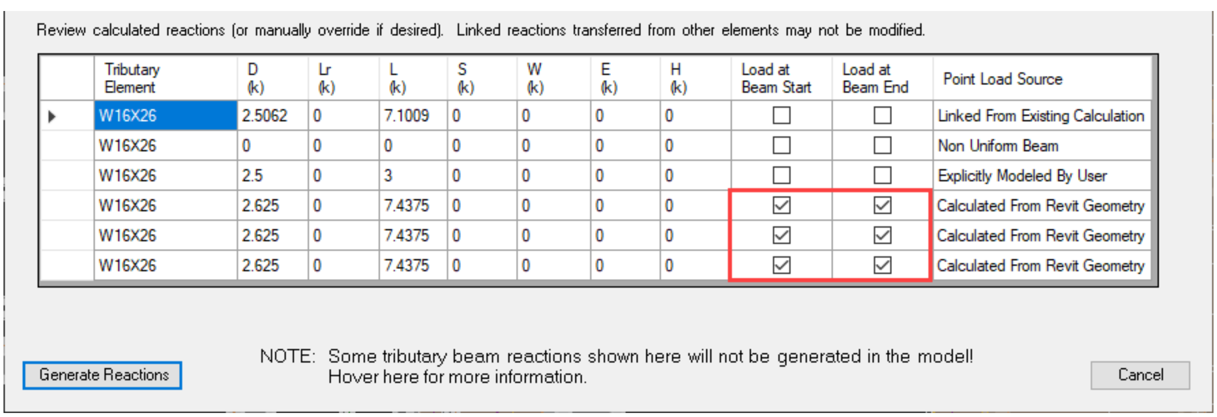
Infill beam #3 is automatically found to have pre-existing hosted point loads at the end joint which bears on the girder. The “Point Load Source” column at the right end of the table indicates “Explicitly Modeled By User”. As a result, the tributary width is not sampled and no model data are reported in the input table. The forces shown in the lower half of the window on the reaction table are the modeled reactions already found in the Revit model on the end of the beam. New reaction forces will **NOT** be generated for this beam.



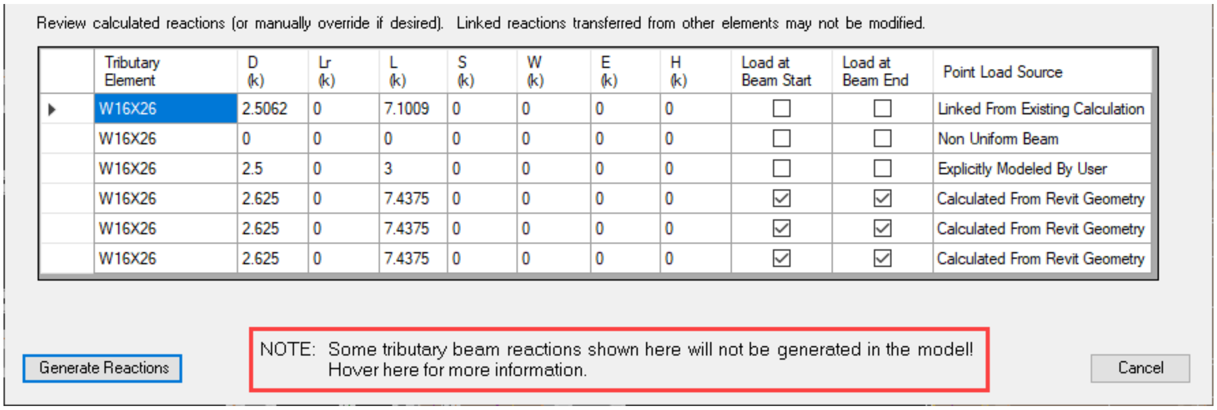
The remaining infill beams, #4, #5, and #6 are not found to have any constraints or existing loads that prevent tributary width sampling. As a result, they are sampled at the density specified and the tributary width, span, and area loads found in the model are reported in the input table. The “Point Load Source” column at the right end of the table indicates “Calculated From Revit Geometry”. The calculated reactions shown in the lower half of the window on the reaction table are the proposed reactions that will be created automatically when the user clicks the “Generate Reactions” button.



Prior to generating these reactions in the Revit model, the user has the ability to manually select whether the reactions will be created at both the start and end of the beam. This selection is made using the checkboxes in the “Load at Beam Start” and “Load at Beam End” columns. Unchecking any of these boxes will mean that no reaction is generated at the specified end of that particular beam. The “Start” and “End” definitions are based on the Revit end definitions found in the model.

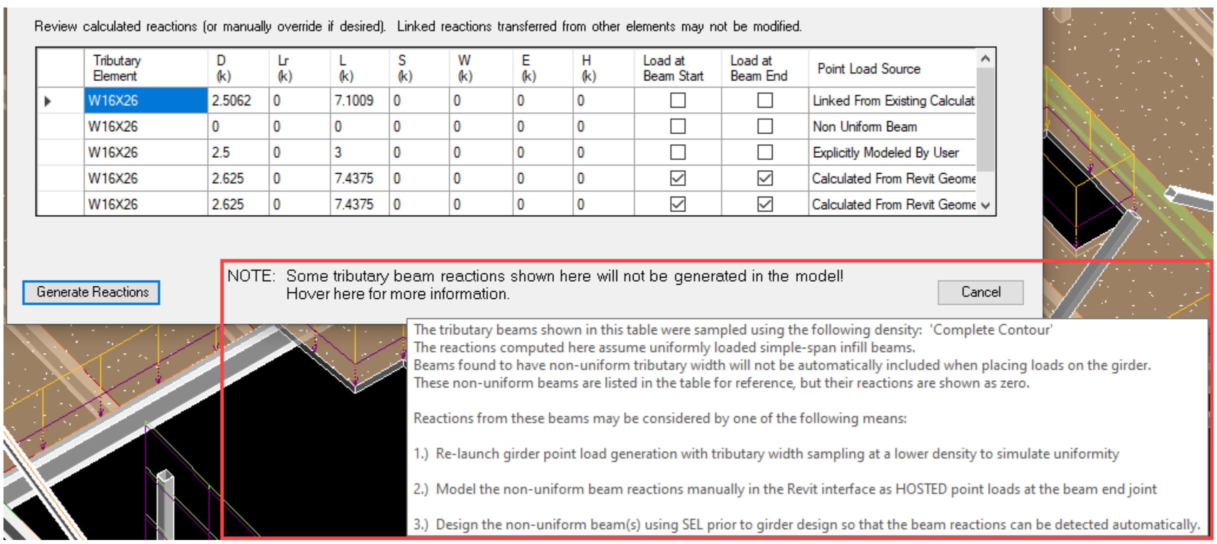


Note also that when any conditions are found that preclude the creation of new reactions, the generator window will include a warning reminding the user.



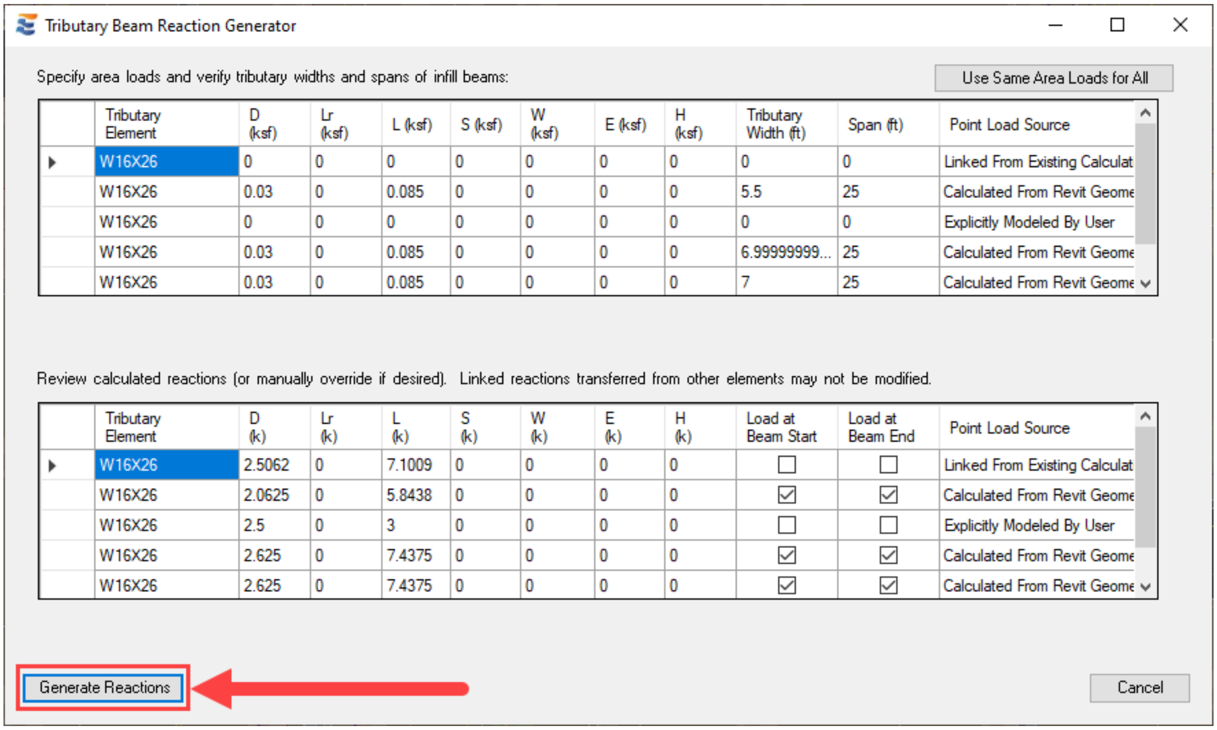
Tributary Element	D (k)	Lr (k)	L (k)	S (k)	W (k)	E (k)	H (k)	Load at Beam Start	Load at Beam End	Point Load Source
W16X26	2.5062	0	7.1009	0	0	0	0	☐	☐	Linked From Existing Calculation
W16X26	0	0	0	0	0	0	0	☐	☐	Non Uniform Beam
W16X26	2.5	0	3	0	0	0	0	☐	☐	Explicitly Modeled By User
W16X26	2.625	0	7.4375	0	0	0	0	☑	☑	Calculated From Revit Geometry
W16X26	2.625	0	7.4375	0	0	0	0	☑	☑	Calculated From Revit Geometry
W16X26	2.625	0	7.4375	0	0	0	0	☑	☑	Calculated From Revit Geometry

Hovering over this notification with the cursor displays a tooltip balloon with more detailed information, including the sampling density that was use, and steps the user can take to consider force effects from the disqualified infill beams.



Tributary Element	D (k)	Lr (k)	L (k)	S (k)	W (k)	E (k)	H (k)	Load at Beam Start	Load at Beam End	Point Load Source
W16X26	2.5062	0	7.1009	0	0	0	0	☐	☐	Linked From Existing Calculat
W16X26	0	0	0	0	0	0	0	☐	☐	Non Uniform Beam
W16X26	2.5	0	3	0	0	0	0	☐	☐	Explicitly Modeled By User
W16X26	2.625	0	7.4375	0	0	0	0	☑	☑	Calculated From Revit Geome
W16X26	2.625	0	7.4375	0	0	0	0	☑	☑	Calculated From Revit Geome

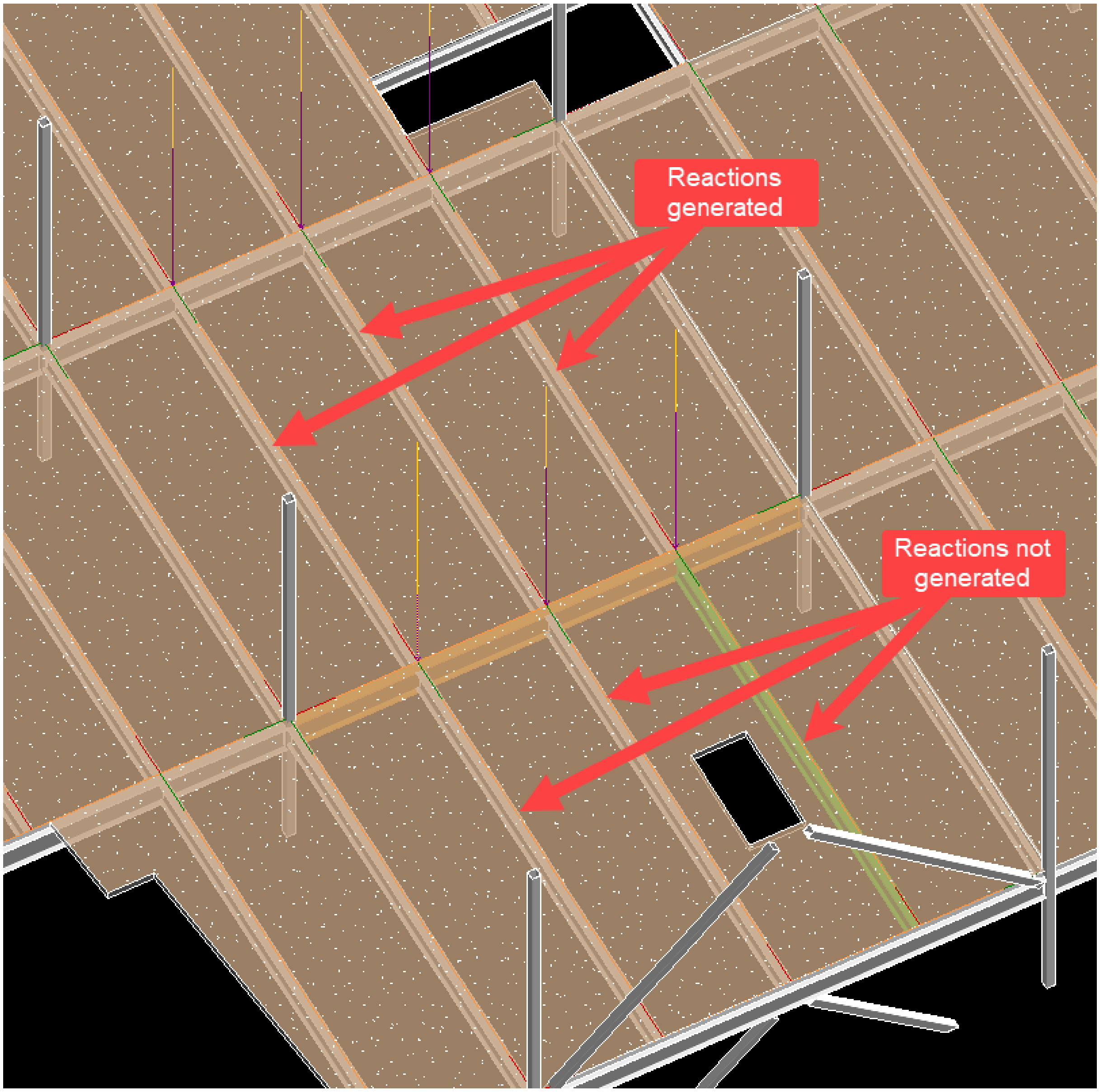
When the user has reviewed and verified (or modified as needed) all reaction information in the table, the reactions may be automatically created in the Revit model by clicking the “Generate Reactions” button.

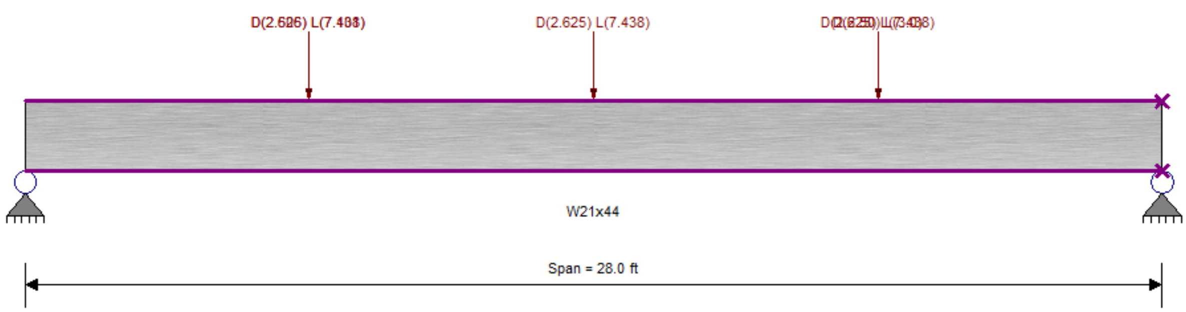


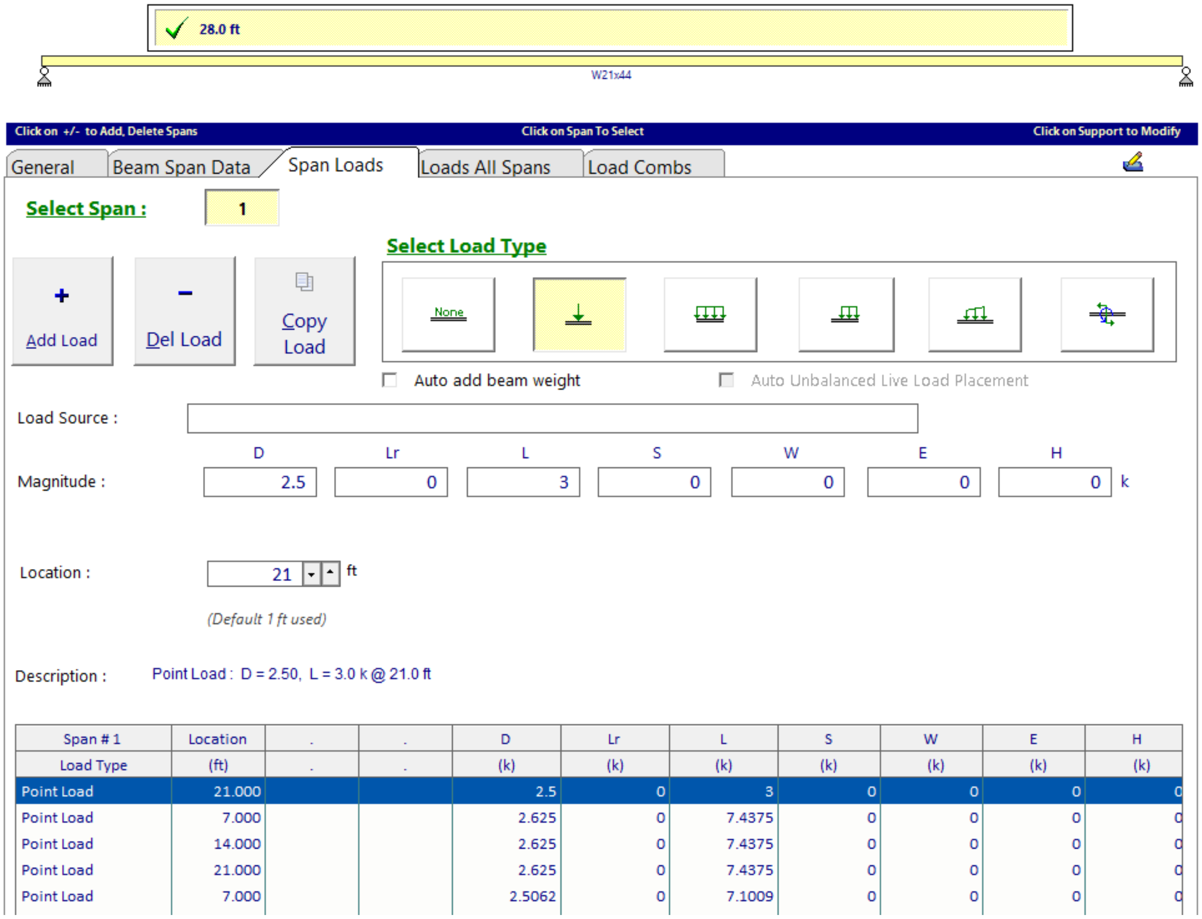
The “Generate” command results in two processes:

1. Any existing reactions that were previously created by the “Girder Point Loads” tool will be automatically removed.
   a. This prevents the accidental creation of duplicate loads.
   b. Hosted point loads created manually by the user will NOT be removed.
   c. Non-hosted point loads will not be removed.

2. Following the removal of existing reactions, the new reactions specified in the table will be generated.
   a. Note that the reactions will only be generated at beam end locations where the “Load Beam At…” box is checked on.
   b. The reactions will appear in the Revit model as Hosted point loads on the end joints of the applicable beams.
   c. These loads will then be available for automatic detection when a beam design calculation is launched for the girder.


Reactions generated
Reactions not generated

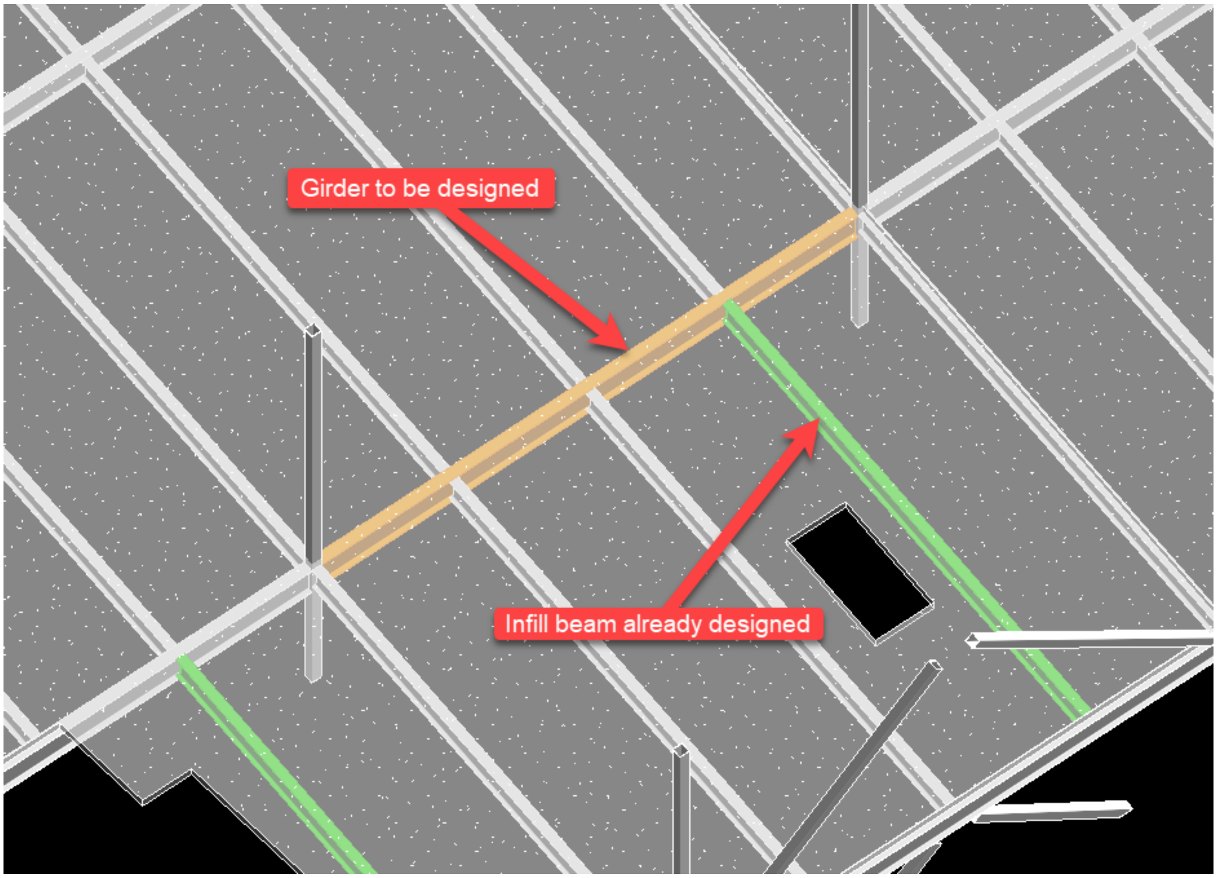

D(2.625) L(7.438)
W21x44
Span = 28.0 ft



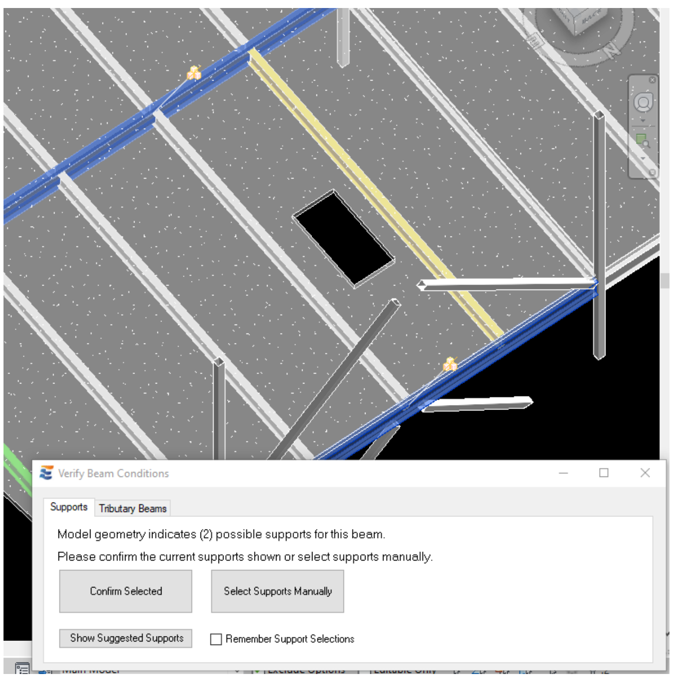
During future re-runs of the “Girder Point Loads” tool, the program will recognize these reactions as having been created by ENERCALC for Revit rather than by the user. As a result, the reactions will be eligible for removal and replacement when the tool is used again in the future. In addition, if the hosted point loads explicitly modeled by the user are manually deleted, then infill beam #3 will become eligible for reaction generation during future re-runs of the “Girder Point Loads” tool. Similarly, if the existing ENERCALC for Revit calculation on infill beam #1 were to be deleted, this beam might also become eligible for reaction generation, depending on the selected sampling density.

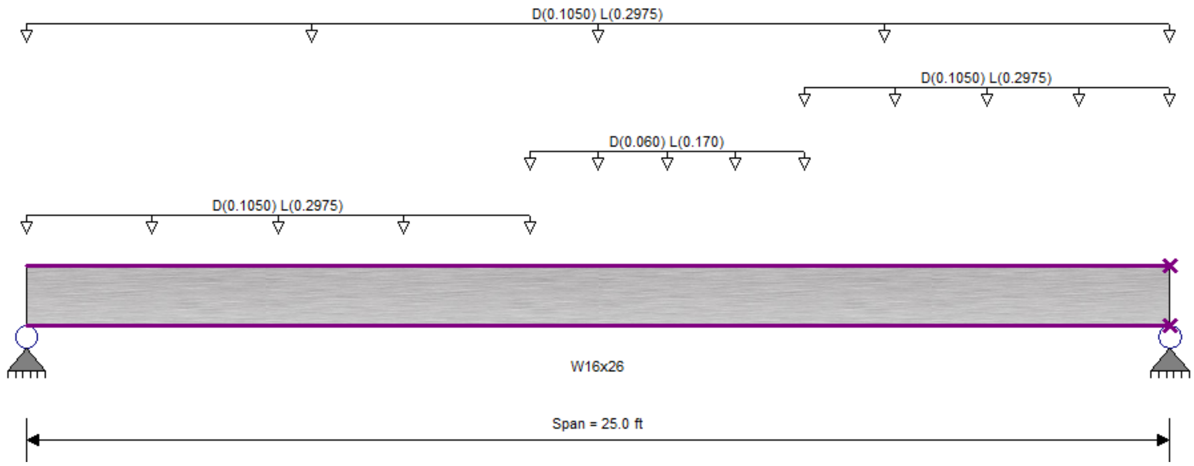
10.4.9 Beam/Girder Load Linking

When designing beams and girders using ENERCALC for Revit, reaction forces from structurally connected elements will automatically be incorporated into subsequent calculations further down in the load path.



As discussed previously in “Beam Supports 110”, the user will be prompted during launch of the infill beam calculation in order to verify its structural relationship with the girder.



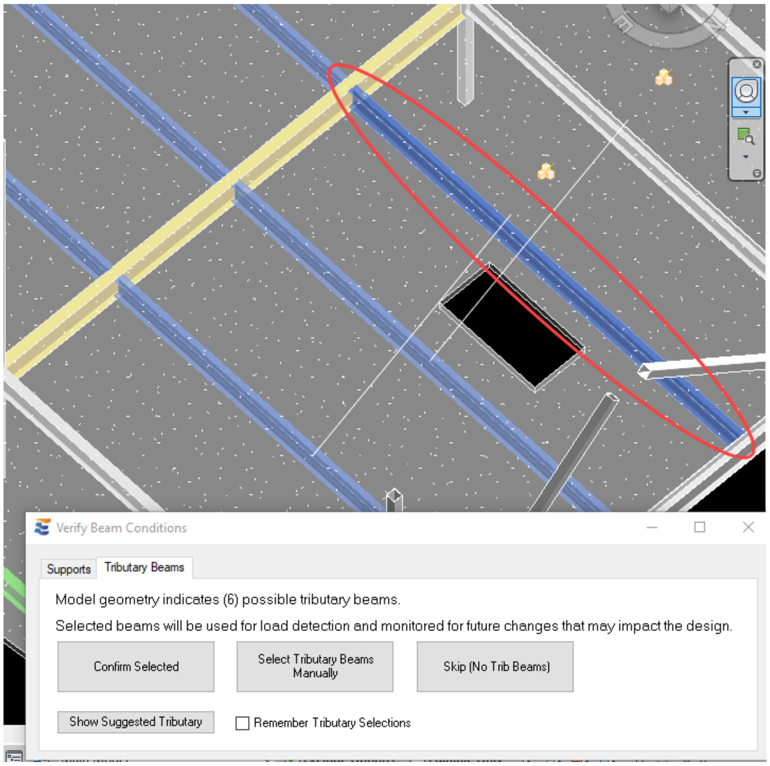


Design of each infill beam results in end reactions calculated in ENERCALC SEL, which are then stored in the Revit model for later use.

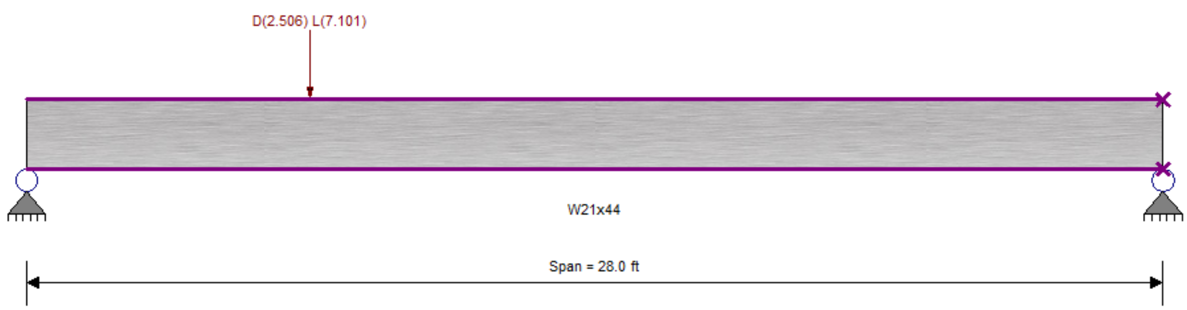
Extreme Reactions (kips)

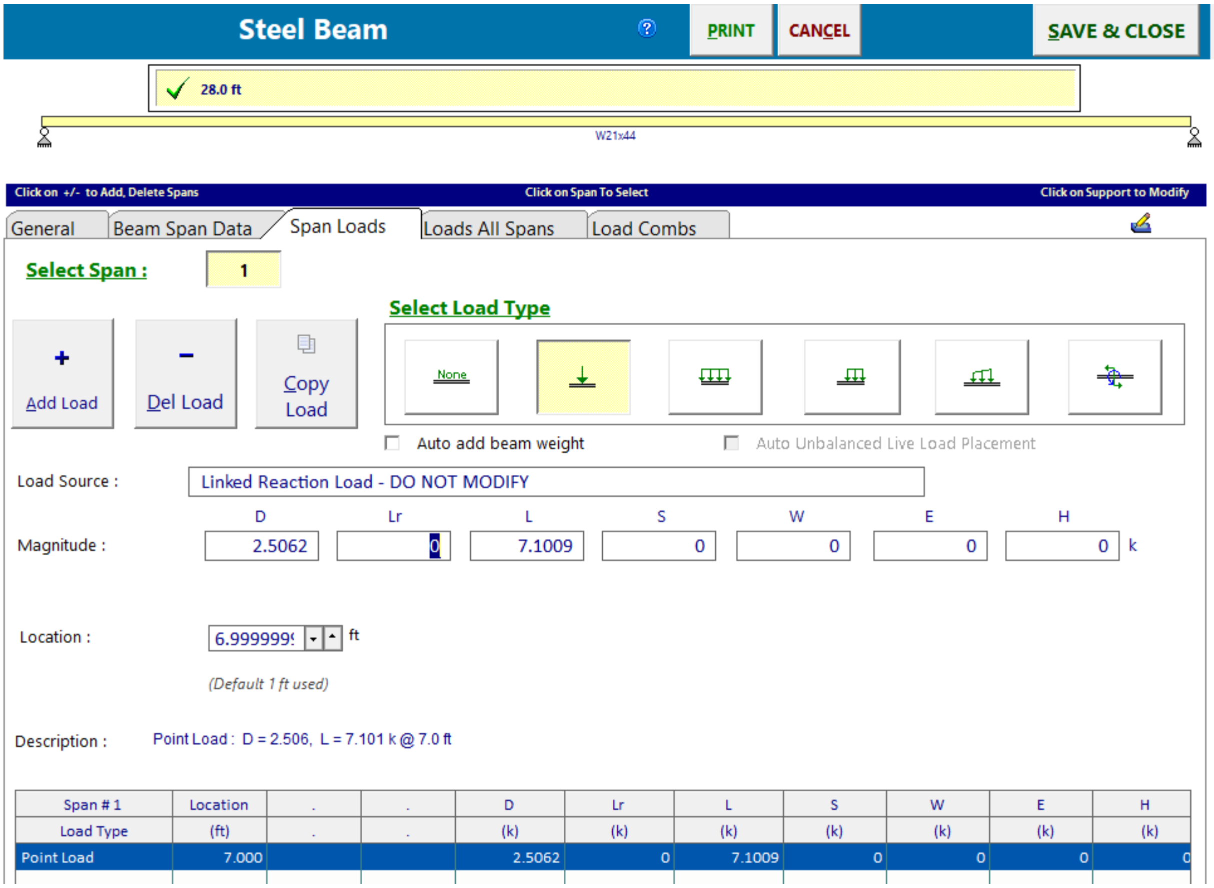
	D	Lr	Lf	S	W	E	H
Support #1	2.51		7.10				
Support #2	2.47		7.01				

When subsequently launching a calculation for the girder, the user will again be prompted to verify a structural relationship with the beam.

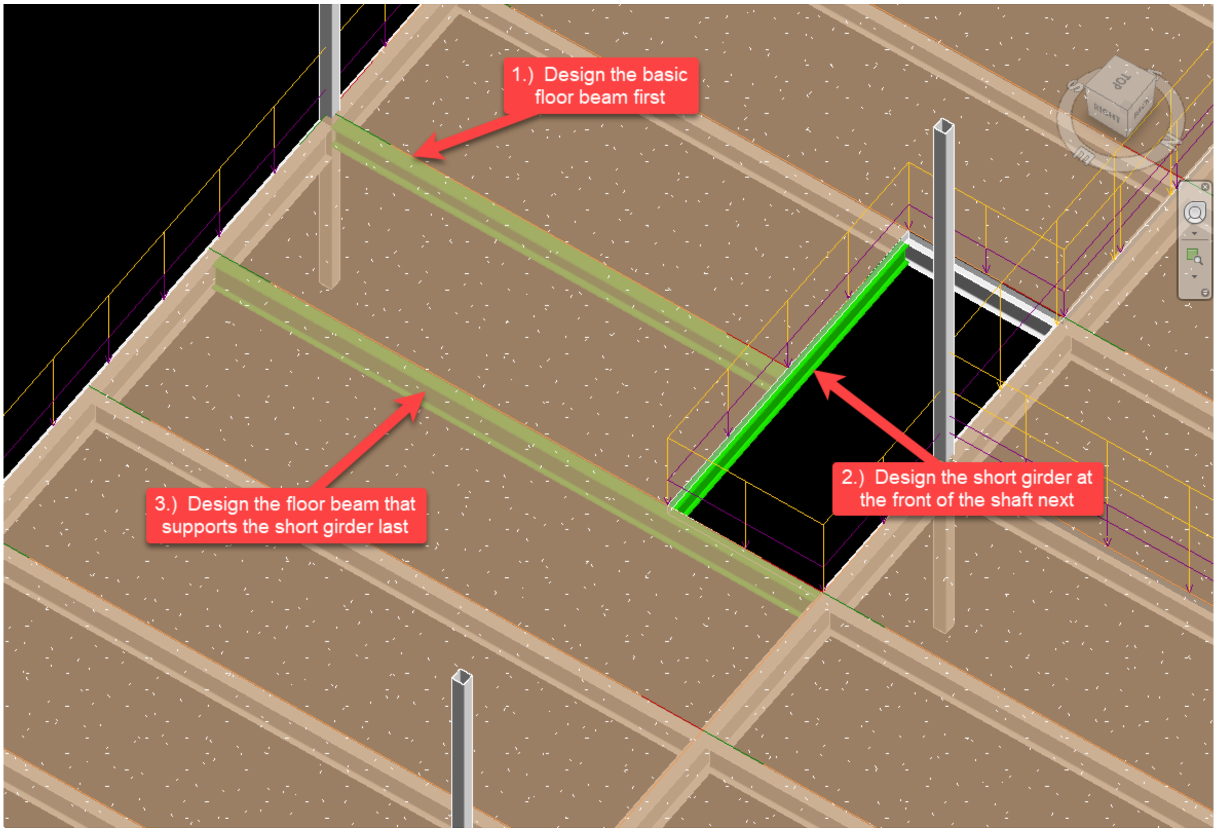


Once the structural relationship between the beam and girder has been confirmed during launch, the beam reaction is automatically included in the girder calculation with no additional effort from the user.

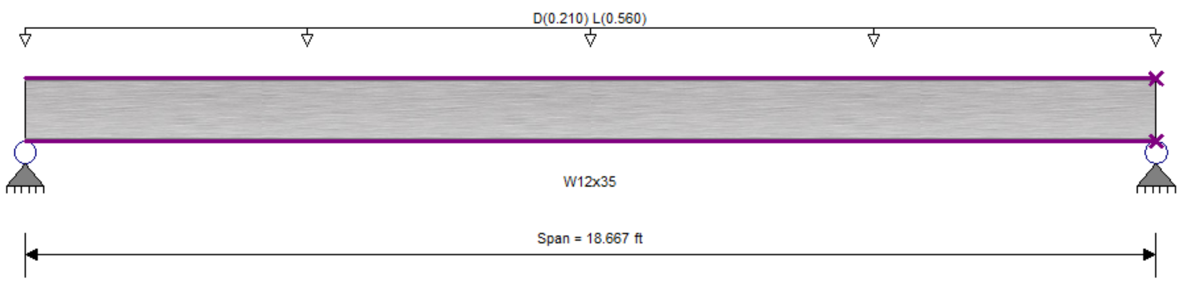




Users should be aware that "Load-Linking" of reactions forces between beam calculations is active even when the beam to be designed is not acting purely as a girder. "Load-Linking" applied even when the beam to be designed exhibits both beam and girder behavior. As an example, consider the elevator shaft opening framing shown below. The framing assembly uses several layers of "Load-Linking" transfer, which is achieved by designing the beams in order of load path.



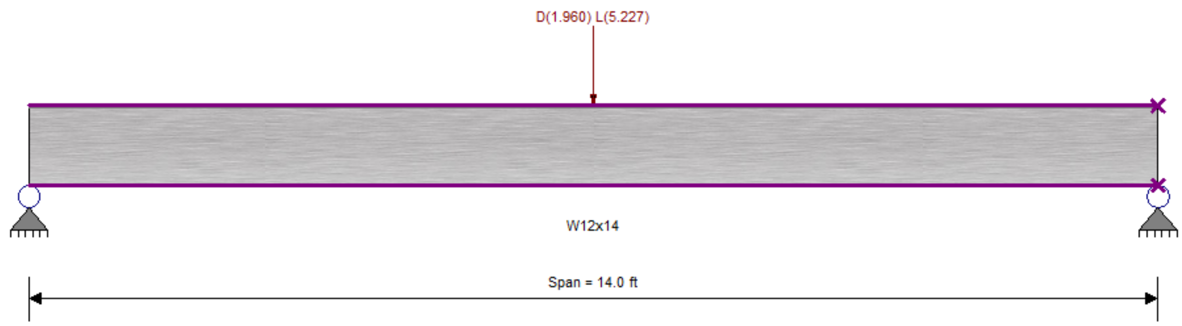
During the design of the basic floor beam, there are no linked reactions involved.

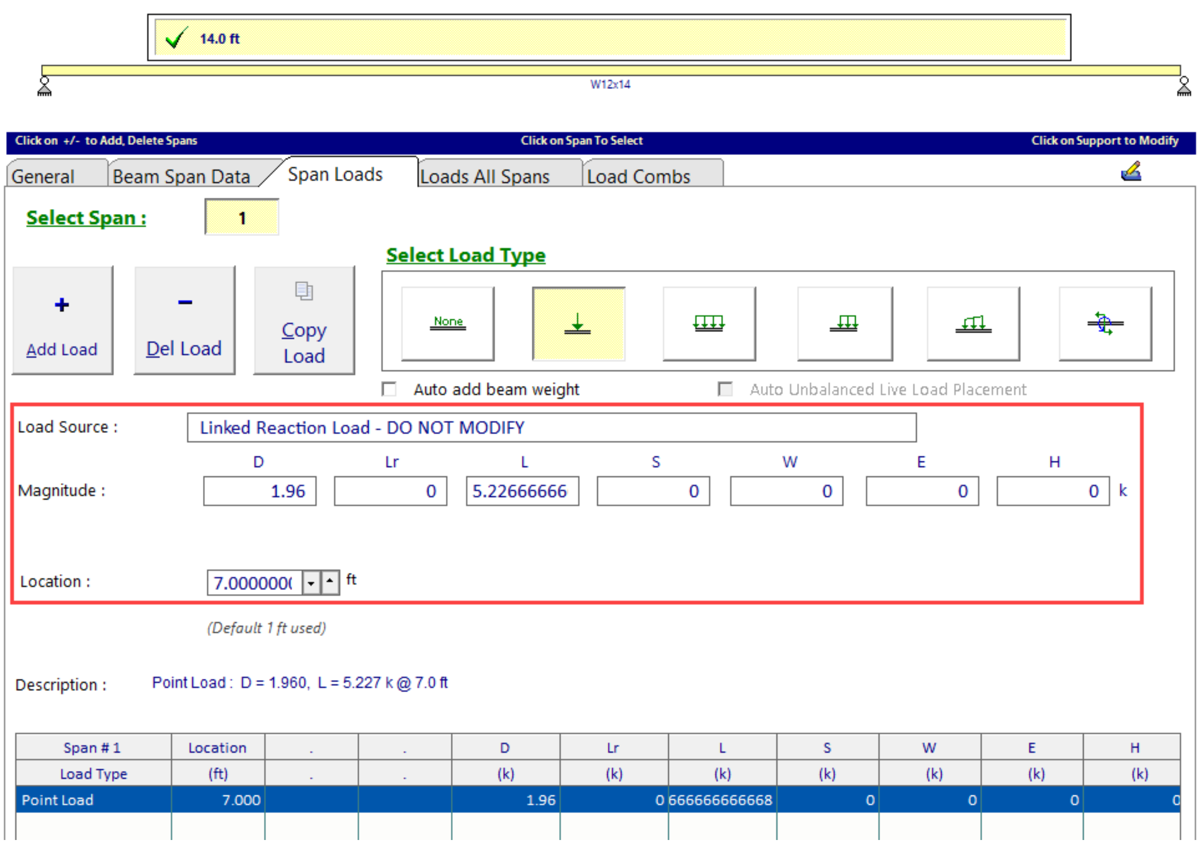


Extreme Reactions (kips)

	D	Lr	Lf	S	W	E	H
Support #1	1.96		5.23				
Support #2	1.96		5.23				

The design of the short girder at the front of the shaft then automatically considers this reaction and locates it appropriately.

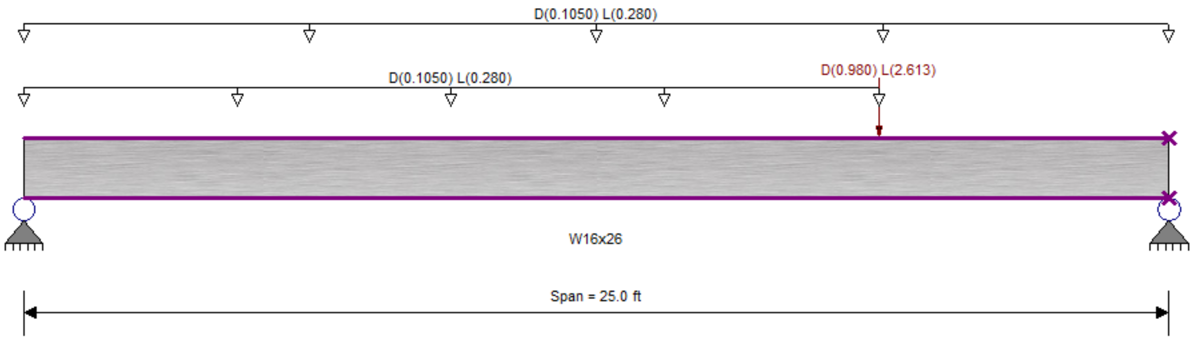


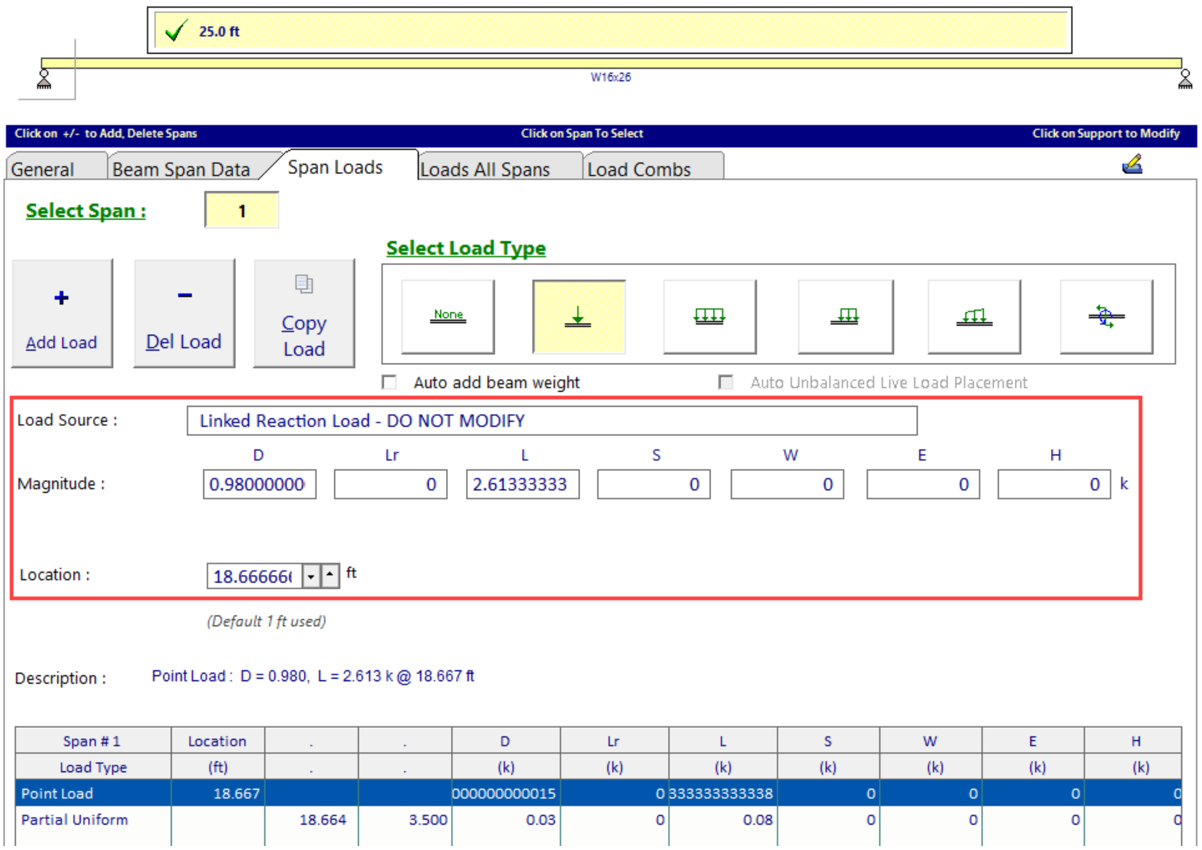


Extreme Reactions (kips)

	D	Lr	Lf	S	W	E	H
Support #1	0.98		2.61				
Support #2	0.98		2.61				

Design of the opening edge beam then automatically considers the loading associated with area loads on the floor and the reactions exerted by the short girder.





10.4.10 Troubleshooting Load Detection Issues

For any situations where loads in the Revit model are not detected or are not mapped properly, it is advisable to study each applicable section of this manual to understand how each type of load is managed. The following list may be helpful as a general reference for various load mapping issues:

Load Type :	Load Hosting :	Problem:	Recommended Action:	For More Info:
Any	Any	Load maps into ENERCALC SEL calculation with the wrong sign	Review the Revit load object to verify sign convention.	“Load Directionality and Components”
		Load does not map into ENERCALC SEL calculation	Use Revit visibility settings to verify that the desired load is still present in the Revit model	None
			Verify that the load case assigned in Revit is one of the default 7 combinations supported by ENERCALC for Revit.	“Load Cases and Combinations”
Point Load	Non-Hosted	Load does not map into ENERCALC SEL calculation	Verify that the load has a valid vertical (Z) component	“Load Directionality and Components”
			Verify that the load is close enough to the beam to be detected.	"Launching With Non-Hosted Loads"
Point Load	Hosted	Load does not map into ENERCALC SEL calculation	Verify that the load has a valid vertical (Z) component	“Load Directionality and Components”
			Verify that the element hosting the load has been specified as a tributary of the designed beam.	"Point Loads for Girder Design"
			Verify that the load has not been accidentally hosted to the beam to be designed.	"Launching With Revit Hosted Loads"
Point Load	Linked Load	Load does not map into ENERCALC SEL calculation	Verify that the element corresponding to the reaction is not in a warning state (calculation results erased).	"Monitoring and Change Warnings"
Line Load	Non-Hosted	Load does not map into ENERCALC SEL calculation	Verify that the load has a valid vertical (Z) component	"Launching With Non-Hosted Loads"
			Verify that the load is close enough to the beam to be detected.	"Launching With Non-Hosted Loads"
			Verify that the load is oriented parallel to the length of the beam.	"Launching With Non-Hosted Loads"
Line Load	Hosted	Load does not map into ENERCALC SEL calculation	Verify that the load has been hosted to the beam to be designed.	"Launching With Hosted Loads"

Load Type:	Load Hosting:	Problem:	Recommended Action:	For More Info:
Area Load	Non-Hosted	Load does not map into ENERCALC SEL calculation	Re-draw the load as a Hosted Area load (non-hosted are not supported at this time).	"Launching With Area Loads on Revit Floors"
Area Load	Hosted	Load does not map into ENERCALC SEL calculation	Verify that the load has been hosted to the appropriate floor analytical element.	"Launching With Area Loads on Revit Floors"
			Verify that the floor is in physical contact (or within tolerance of) the beam to be designed.	"Launching With Area Loads on Revit Floors"
			Verify that the floor has a span direction that is not parallel to the beam to be designed.	"Launching With Area Loads on Revit Floors"
			Verify that the beam tributary width is sampled at a density sufficient to capture the behavior of the beam	"Tributary Width Sampling"
			Verify that the tributary width behavior along the beam has a discernible pattern that can be mapped (fully irregular trib profiles are not supported).	"Irregular Floor Edges"
		Load maps into ENERCALC SEL calculation with incorrect geometry	Verify that the beam tributary width is sampled at a density sufficient to capture the behavior of the beam	"Tributary Width Sampling"
			Verify that the framing condition of the beam to be designed is permissible using ENERCALC for Revit.	"Launching With Area Loads on Revit Floors"

10.4.11 Modifying Loads from the SEL Interface

Once a beam calculation has loaded in the ENERCALC SEL interface, most loads found in the calculation may be manually modified in SEL if desired. The following table provides an overview of the user's ability to manually modify various aspects of each load type from the SEL interface:

Load:		Can Add New From SEL?	Can Modify Existing From ENERCALC SEL Interface?			
Type:	Hosting:		Magnitude?	Location / Extents?	Add/Remove Cases?	Tributary Width?
Point Load	Non-Hosted	Yes	Yes	Yes	Yes	N/A
Point Load	Hosted	No	Yes	Yes	Yes	N/A
Point Load	Linked Load	No	No	No	No	N/A
Line Load	Non-Hosted	Yes	Yes	Yes	Yes	Yes
Line Load	Hosted	Yes	Yes	Yes	Yes	Yes
Area Load	Hosted	No	Yes	No	Yes	No

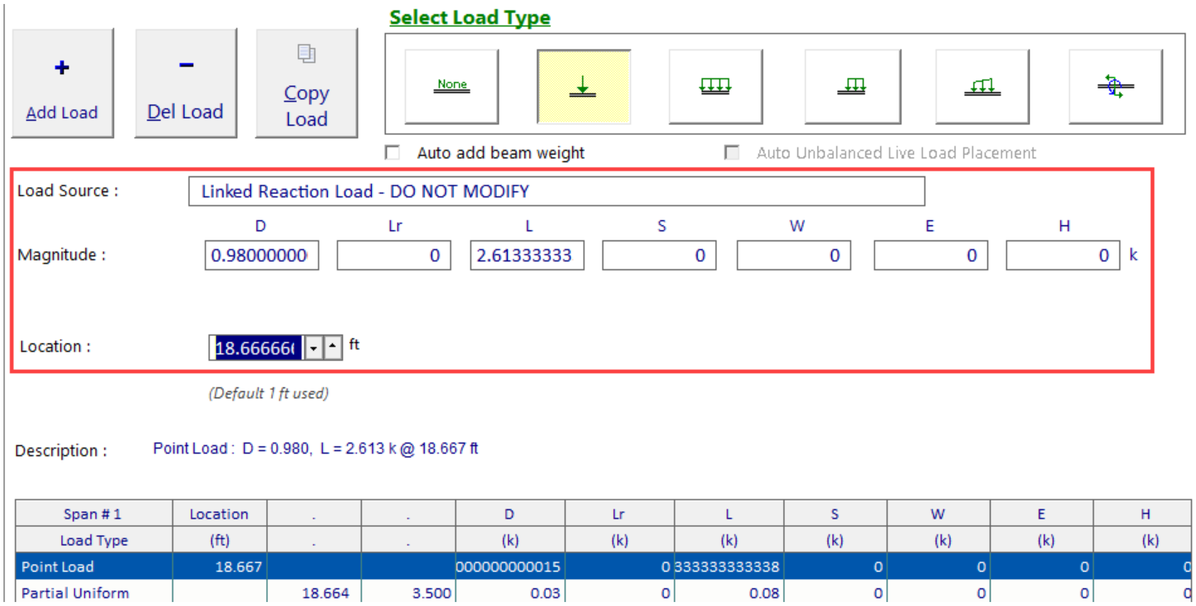
Details and any exceptions are specifically noted in the following sections.

10.4.11.1 Point Loads

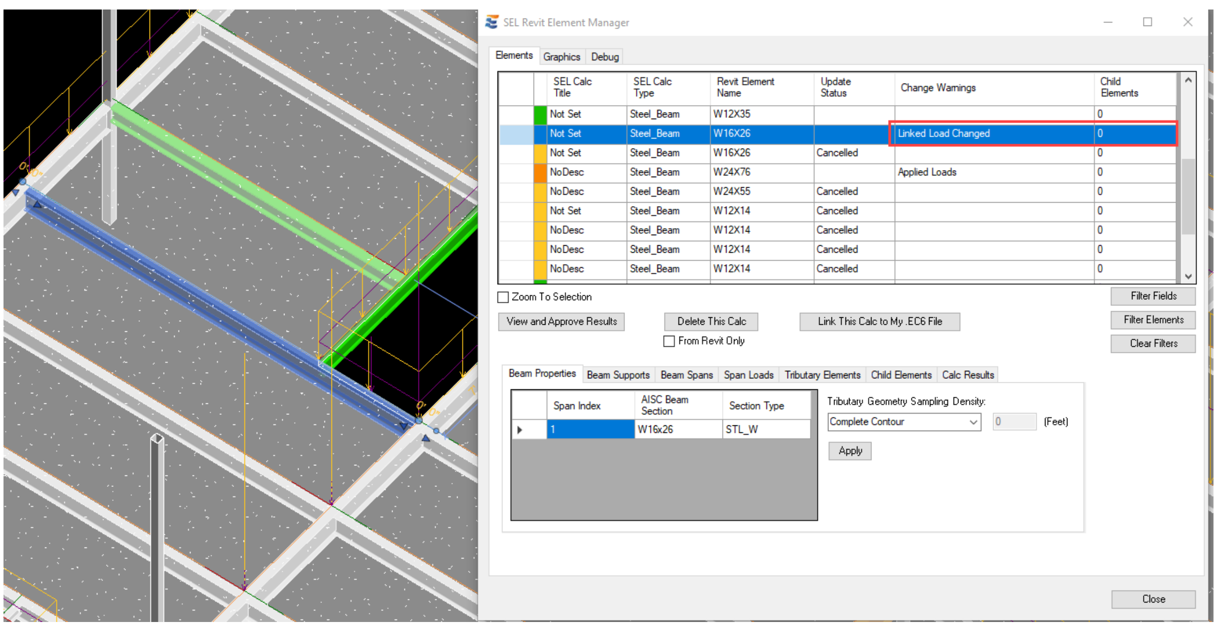
Point loads are viewed and modified from the “Span Loads” tab in the SEL interface. The user may also add new non-hosted point loads using the “Add Load” and “Point Load” controls, as discussed in “Launching Without Revit Loads 158”.

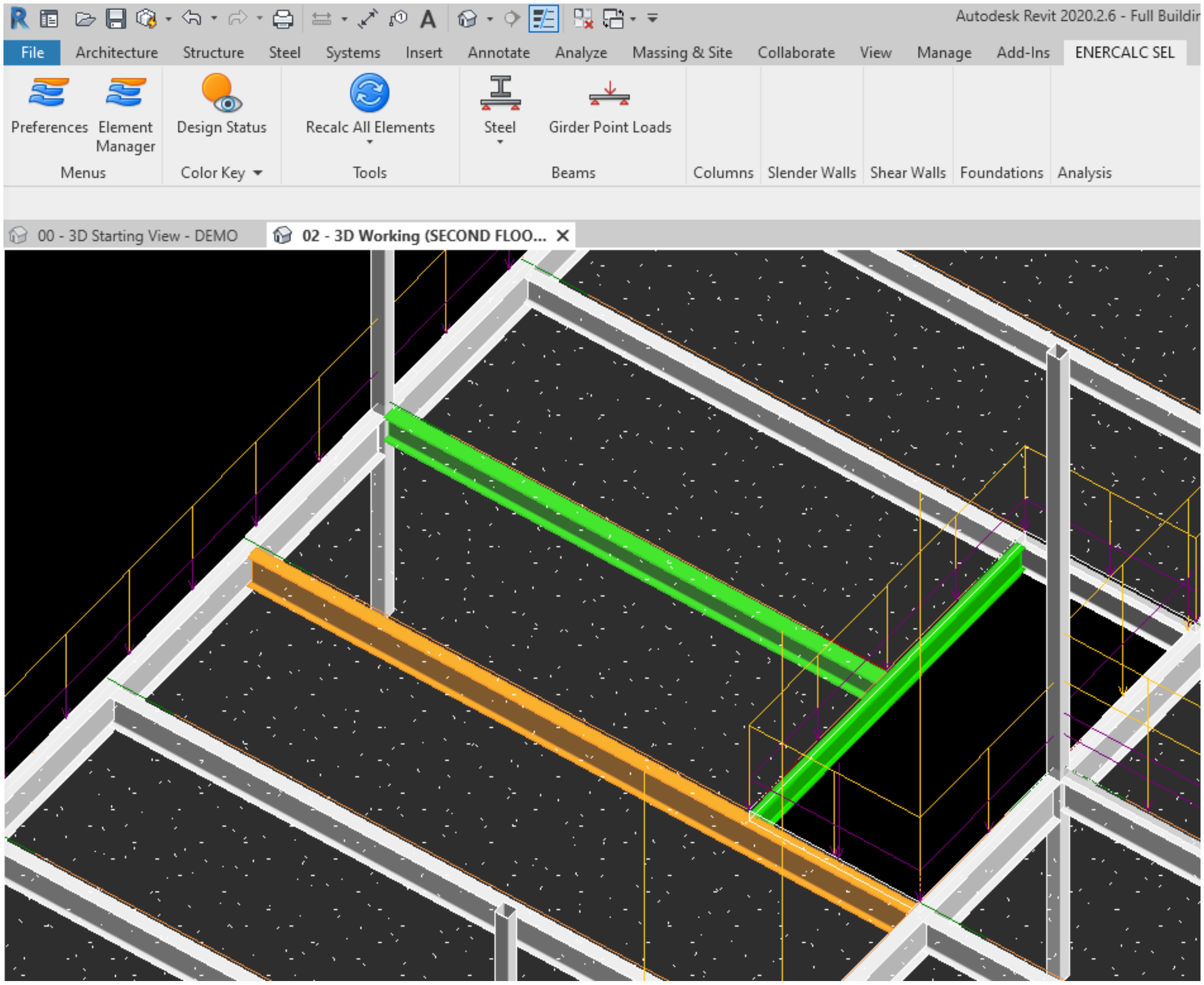
As noted in the table above, non-hosted loads may be modified from SEL without any restrictions. These modifications are made by simply editing the properties of the load item using the same SEL controls users are accustomed to. Hosted loads carry the same ability to modify, but must be used with greater caution. Any manual modification of a hosted load by the user in SEL will cause it to return to the Revit model as a non-hosted load. This means the revised load in Revit is no longer eligible for removal and replacement by the girder point load generator tool. It will become additive with auto-generated reactions unless manually removed. Adding a magnitude to a load case that originally showed “0” will result in a point load of that case being newly created in the Revit model at the specified location.

Linked loads that are automatically associated to a beam calculation from tributary elements are not locked against manual modification but will result in a warning if changed. Linked reactions are labeled with descriptive titles to help remind the user to avoid editing their properties.



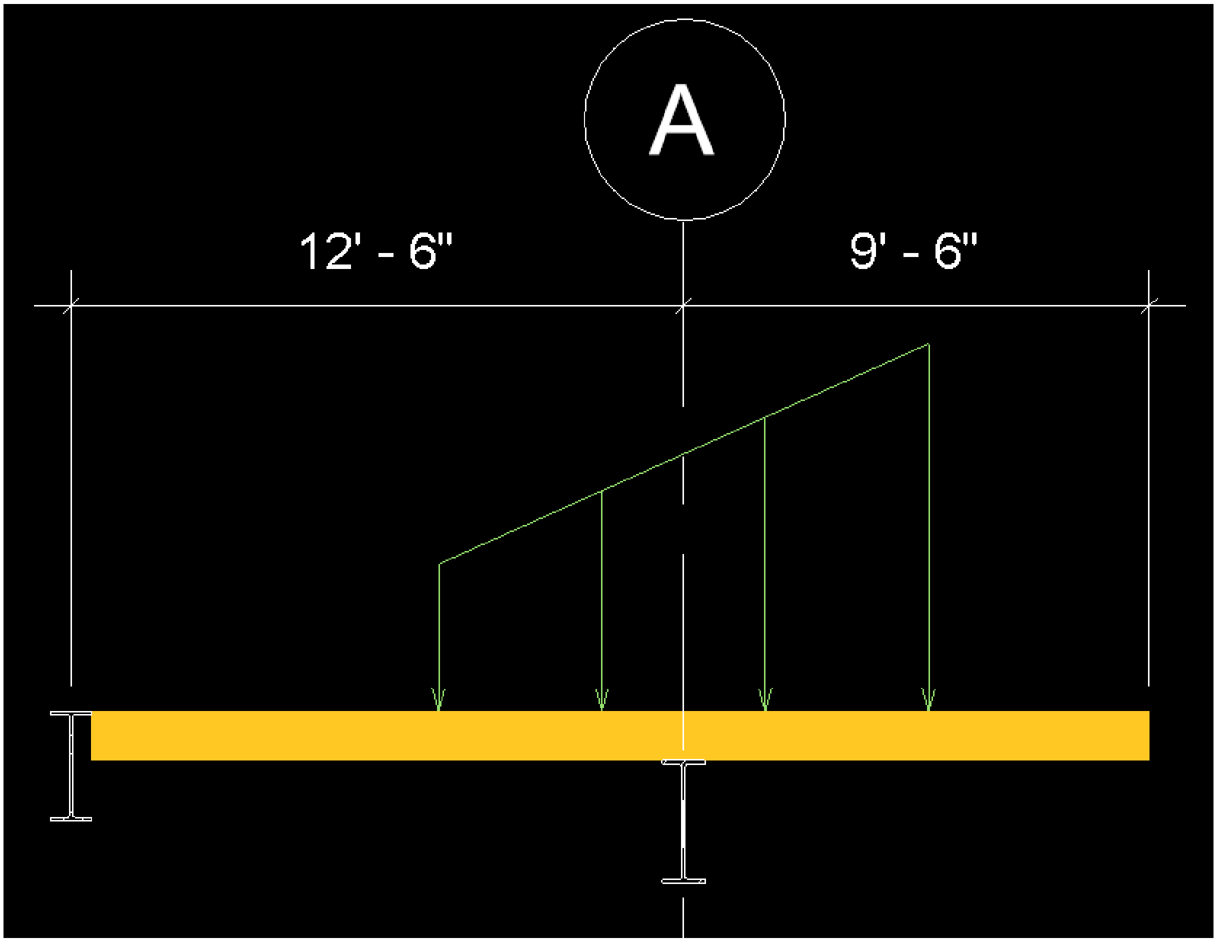
If the user proceeds with edited properties on a linked reaction, the calculation will be flagged with a warning in Revit when the “Save and Close” operation finishes, and will remain in a warning state (orange) until the calculation is re-run with accurate (unmodified) linked reactions.

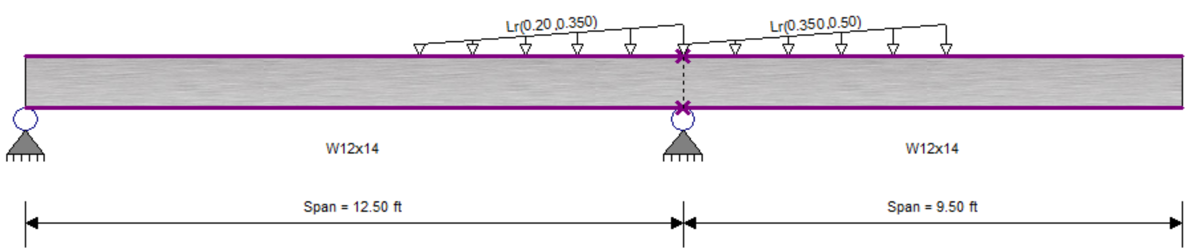


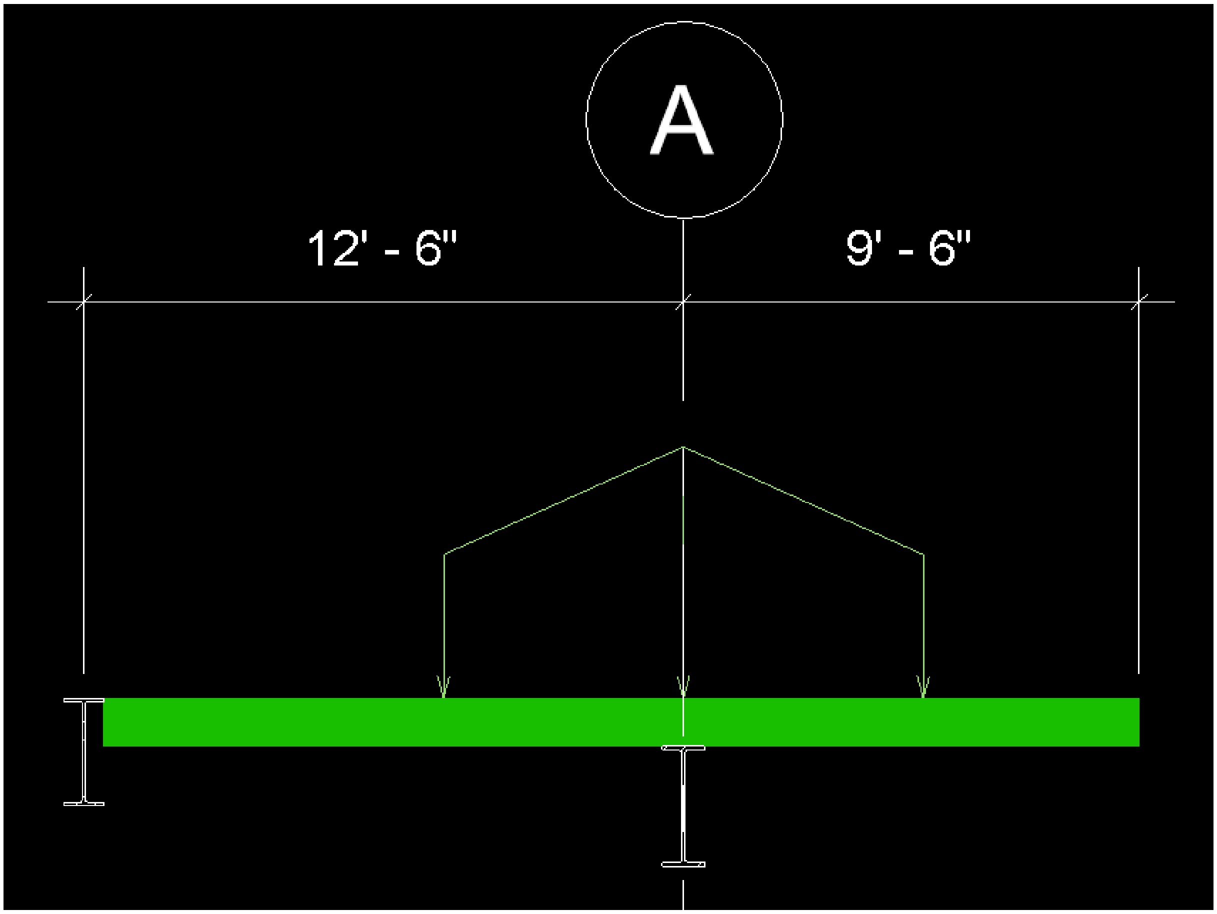


10.4.11.2 Linear Loads

Linear loads (either hosted or non-hosted) may be modified from the SEL interface without any restrictions. Any changes to magnitude, location, cases, etc. will be automatically translated directly to the Revit model when the "Save and Close" operation is completed. Note that modifying the "Tributary Width" property of the linear load will cause the calculated equivalent linear load magnitude to be stored in the Revit model. It will not result in creation of an actual area load. Note also that in situations where a multi-span load has been interpolated across a support location, modification of the load properties will necessitate the load being returned to Revit as individual loads on the respective spans.


A
12' - 6"
9' - 6"


Lr(0.20,0.350)
Lr(0.350,0.50)
W12x14
W12x14
Span = 12.50 ft
Span = 9.50 ft



10.4.11.3 Area Loads

Equivalent linear loads mapped from Revit area loads may be modified in some (but not all) of the same ways that a basic linear load may be. This includes modification of the area load magnitude and addition of new cases, but not editing of the extents and tributary width. These are geometric properties derived from the physical conditions found in the Revit model. Manual changes to these geometric properties will cause the associated Revit beam element to remain in a warning state until re-calculated with accurate loading information. Similar to other loads, populating an empty case with a new magnitude will cause a new corresponding area load to be created on the associated floor in the Revit model.

10.4.12 Live Load Reduction

In certain design scenarios, ASCE 7 and IBC code provisions allow design engineers to reduce the magnitude of live loads applied to both floors and roofs in the structure. ENERCALC for Revit provides a set of tools to implement live load reduction during the launch of a beam calculation.

NOTE: It is advisable for any user to be thoroughly familiar with all code provisions and limitations associate with Live Load Reduction prior to using these tools in ENERCALC for Revit.

10.4.12.1 Live Load Reduction Limitations

To prevent frustration from unneeded steps during calculation launches, beams and girders are pre-screened to determine if they are eligible for live load reduction. Beam and girder elements must meet ***at least one*** the following criteria to be eligible for live load reduction during calculation launch:

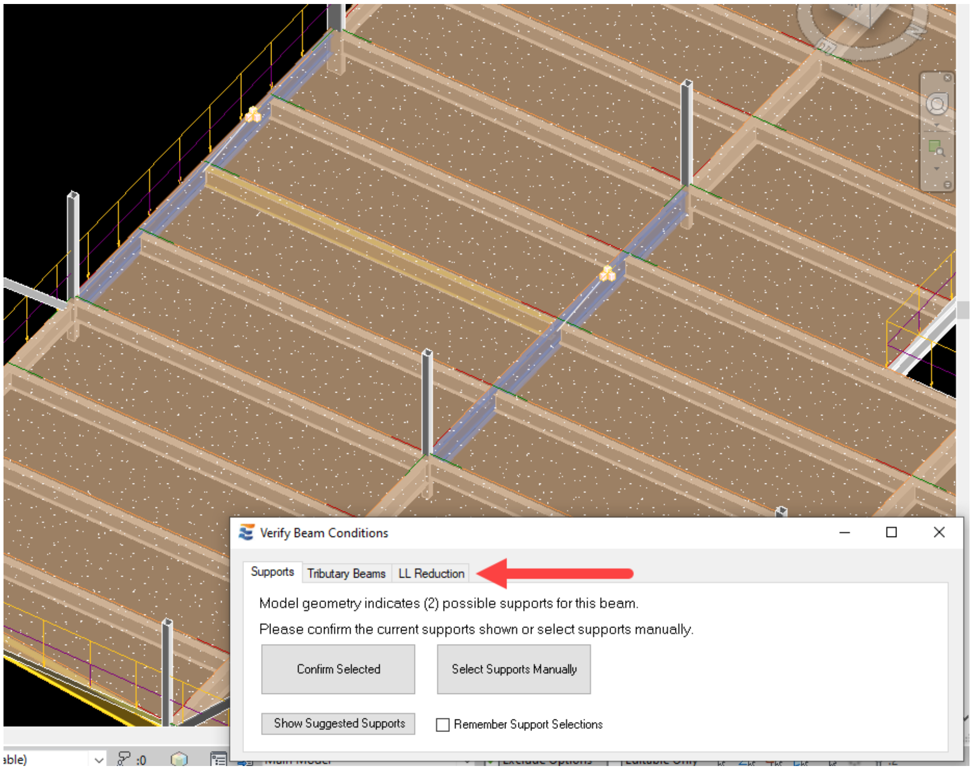
- Beams:
    - Supports a non-hosted point load of case Live (LL) or Roof Live (LR)
    - Supports a non-hosted line load of case Live (LL) or Roof Live (LR)
    - Supports a hosted line load of case Live (LL) or Roof Live (LR)
    - Supports a floor with hosted area loads of case Live (LL) or Roof Live (LR)
- Girders:
    - Meets any beam criteria listed above
    - Supports a point load reaction of case Live (LL) or Roof Live (LR) hosted to a tributary beam
    - Supports a “Load-Linking” reactions of case Live (LL) or Roof Live (LR) from a tributary beam

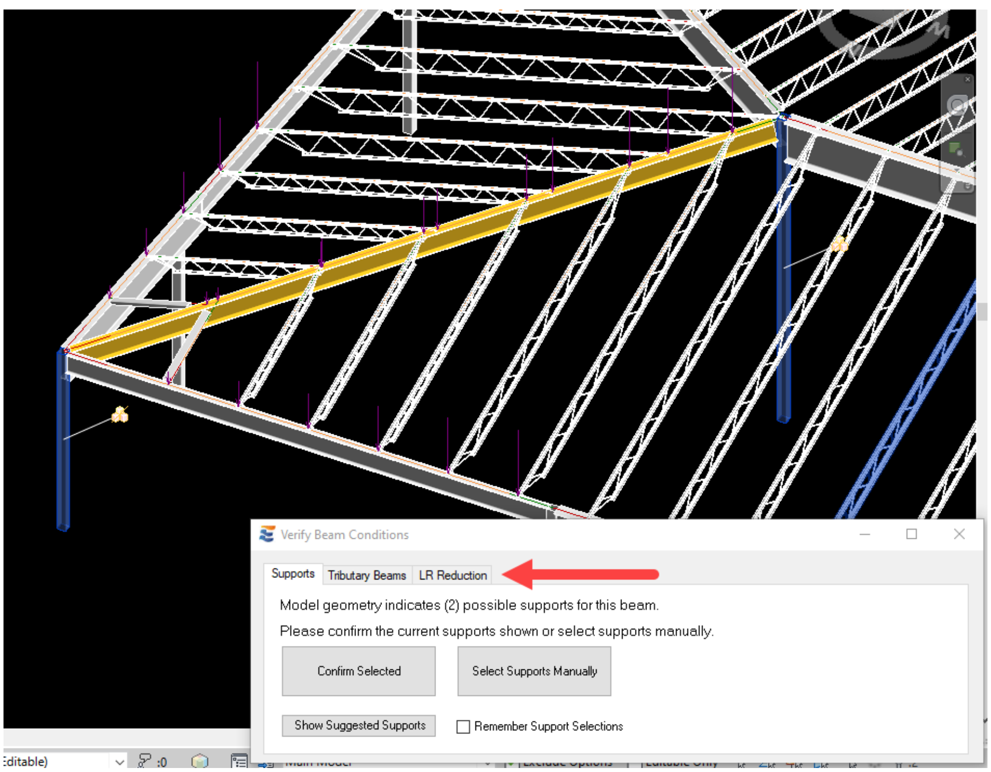
The use of the above restrictions permit ENERCALC for Revit to clearly determine whether the code provisions for reduction of live and roof live loads are applicable. If none of the above criteria are met, then options for LL reduction will not be presented during the beam calculation launch.

ENERCALC for Revit permits only one factor for floor live load (LL) reduction and one factor for roof live load (LR) reduction on a particular beam calculation. These reduction factors will be applied uniformly to all live loads (LL) or all roof live loads (LR) found on the beam. ENERCALC for Revit does not support the ability to reduce individual loads of the same case by differing factors within a single calculation. As a result, if code provisions limit or prohibit the reduction of any specific live load found on the beam, this restriction will necessarily be applied uniformly to all live loads found on the beam.

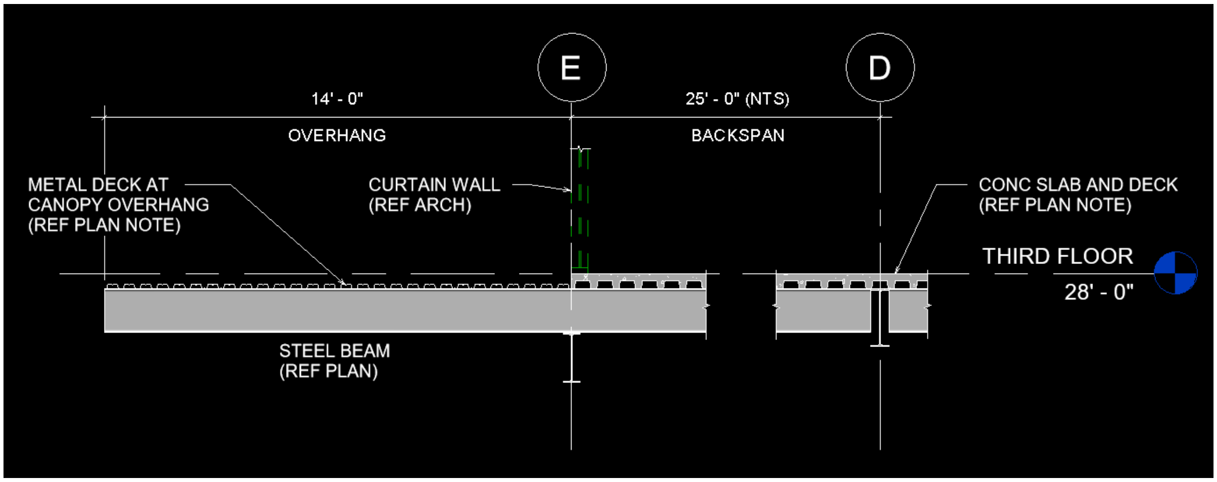
10.4.12.2 Navigation Overview for Live Load Reduction Controls

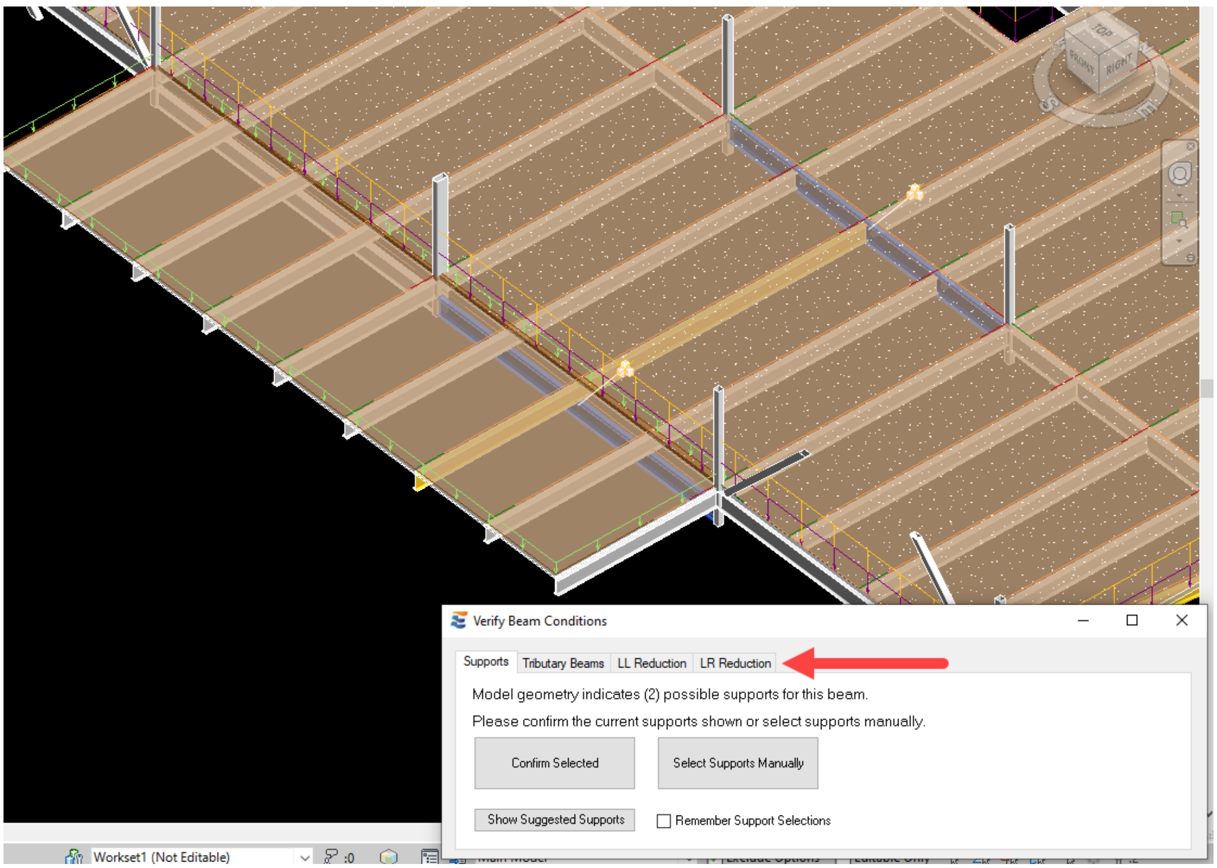
If ENERCALC for Revit detects one or more conditions from the above list that make the beam eligible for live load reduction, an additional tab for these controls will appear on the launch approval window.





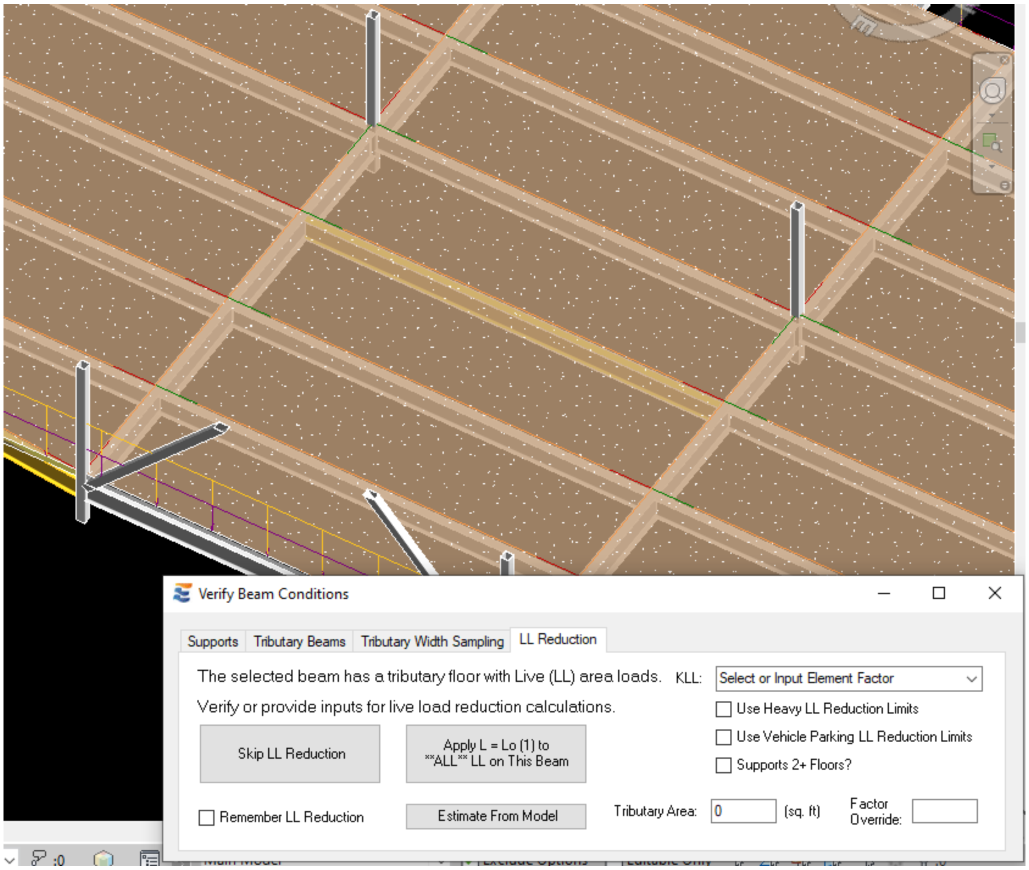
In some unique framing conditions, a particular beam or girder may be subject to both live (LL) and roof live (LR) loads. In such cases, both sets of options will be presented to the user during the calculation launch.





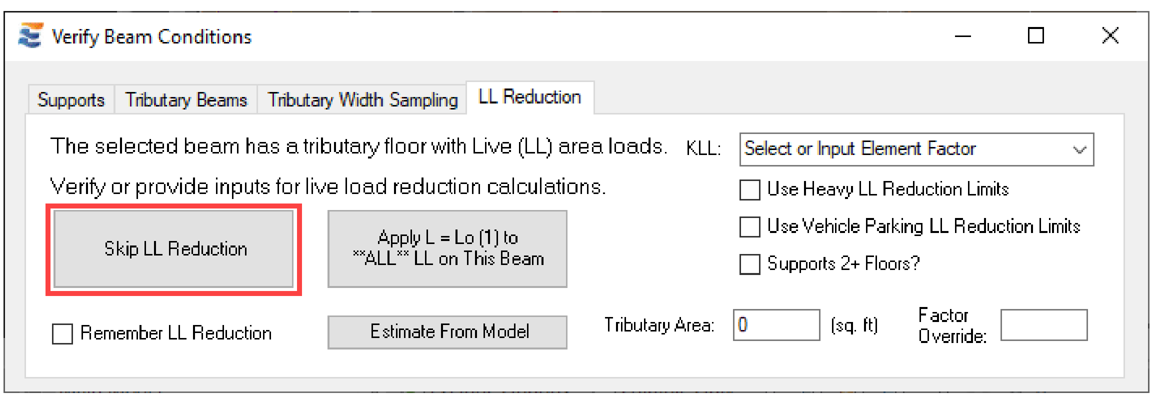
As with the other approval tabs in the launch window, looking ahead to the “LL Reduction” or “LR Reduction” tabs before the preceding approvals are complete will display the controls in a “disabled” grayed-out mode.

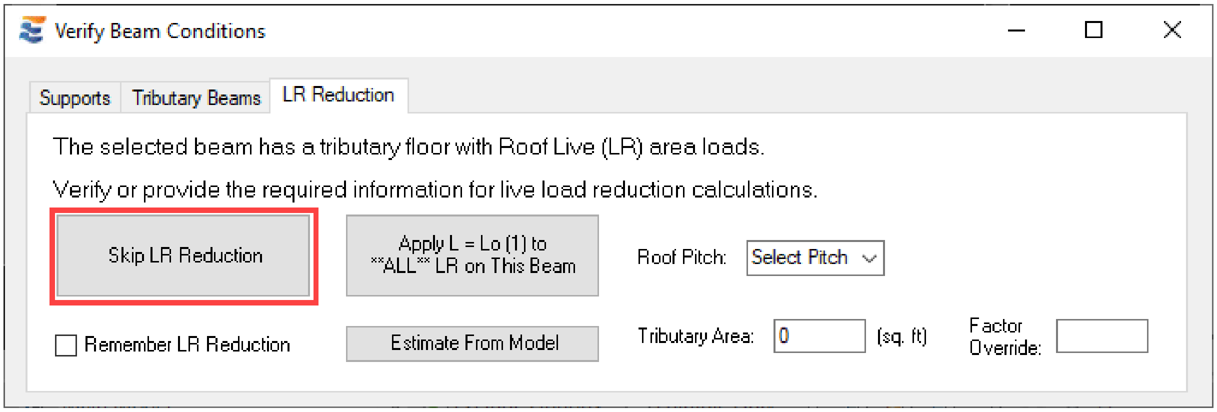
These options automatically become available for user review and interaction after approvals have been furnished in the preceding tabs.



In any situation where the beam has loads eligible for reduction but the user does not wish to use these options, reduction may be skipped using the "Skip LL Reduction" or "Skip LR Reduction" buttons, as applicable. The decision to skip reduction will be automatically applied on future launches of the calculation.

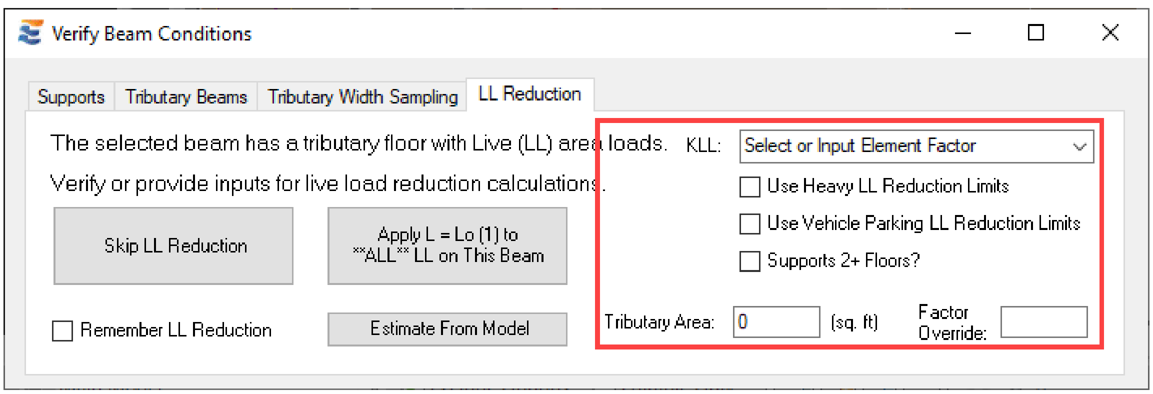
The decision to skip floor (LL) reduction does ***NOT*** automatically skip LR reduction, and vice versa.





10.4.12.3 Reduction of Live Loads (LL)

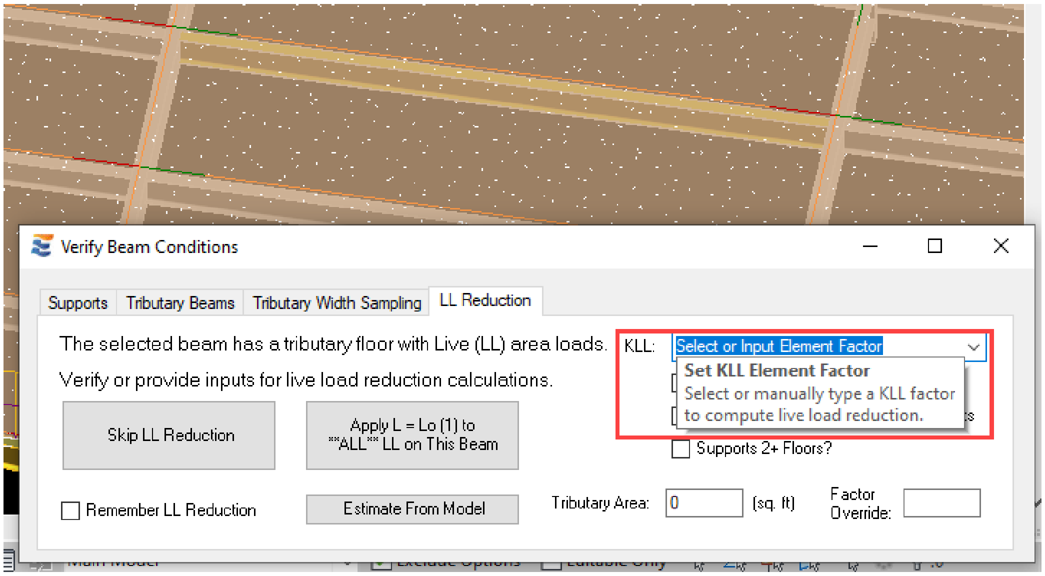
If the user does wish to take advantage of LL reduction, the appropriate reduction factor is calculated using the inputs provided in the window. These inputs correlate directly to the parameters and limitations outlined in ASCE 7 and IBC for reduction of floor live loads.

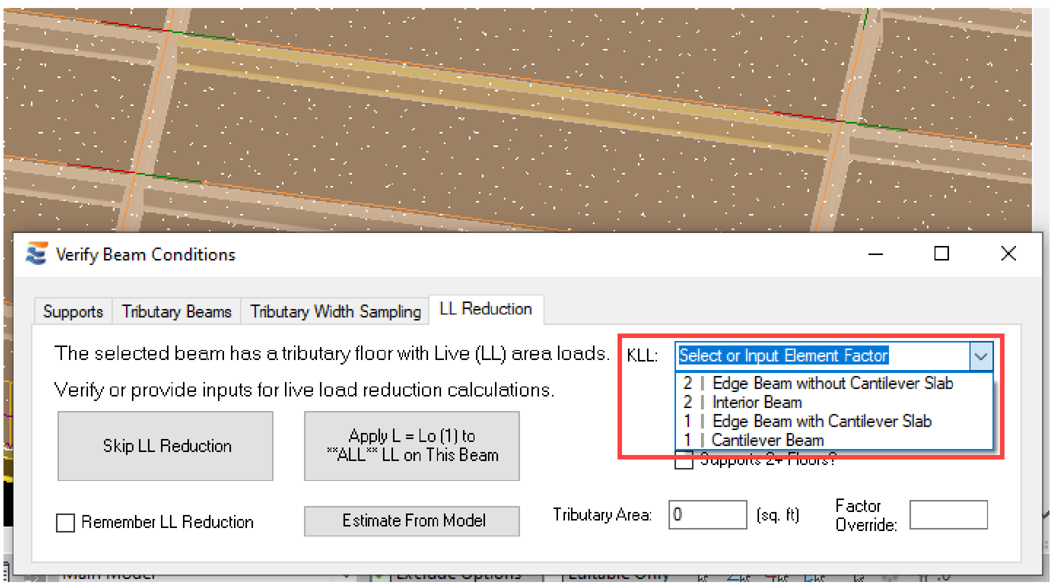


As the inputs in this window are modified, the LL reduction factor to be applied will update in real-time, as indicated on the “Apply…” button.

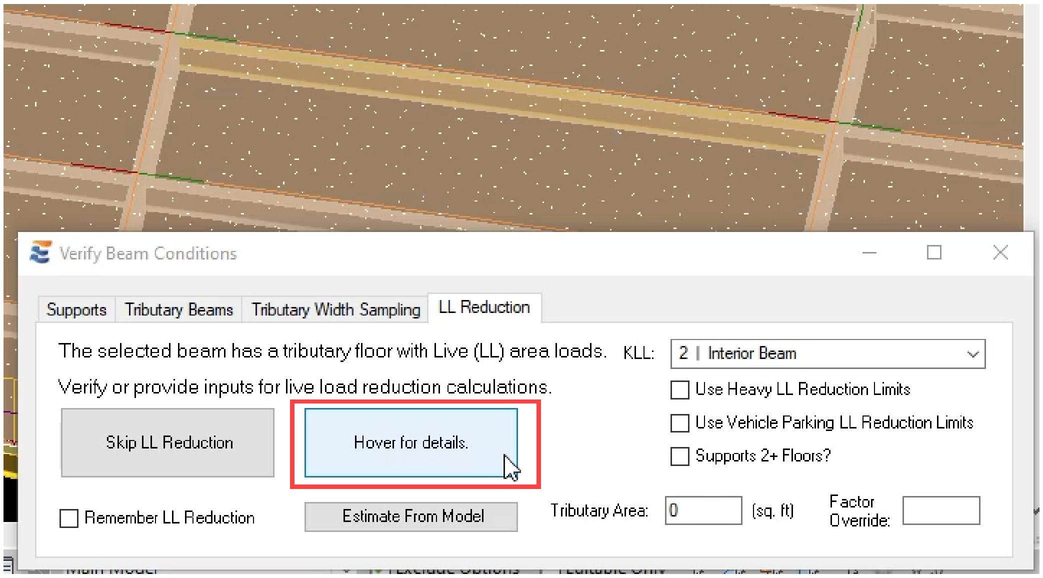
The tributary area is manually modified by typing a numerical value into the “Tributary Area:” textbox. Note that the window will not respond to tributary area modifications when the KLL factor has not been set.

The KLL element factor may be selected from preset options found in the building code, or manually entered if hand calculated by the user:

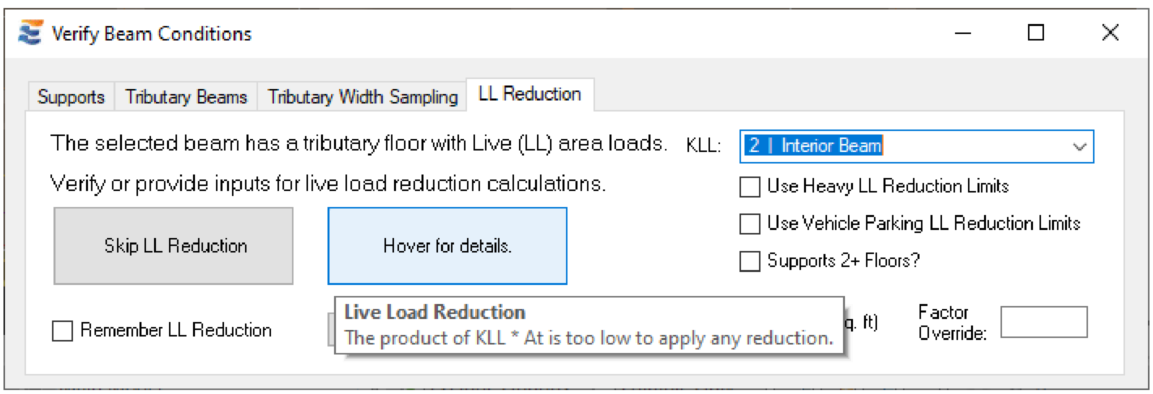




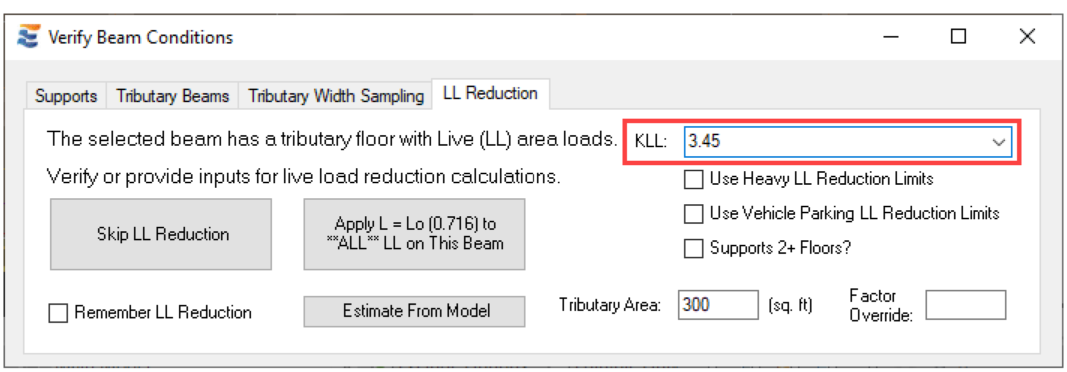
When the cursor passes over the “Apply…” button, the button will show a notification reminding the user to hover over the button for detailed information about the LL reduction factor being presented.



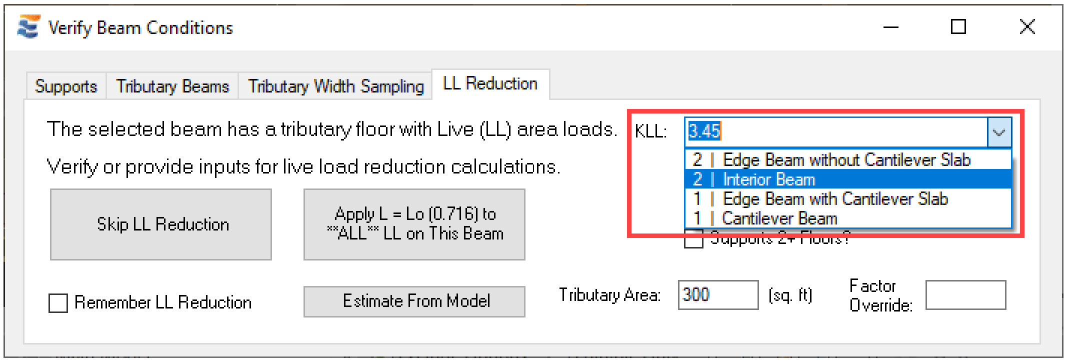
Hovering over the button will display a tooltip balloon that provides details about the calculated reduction factor.



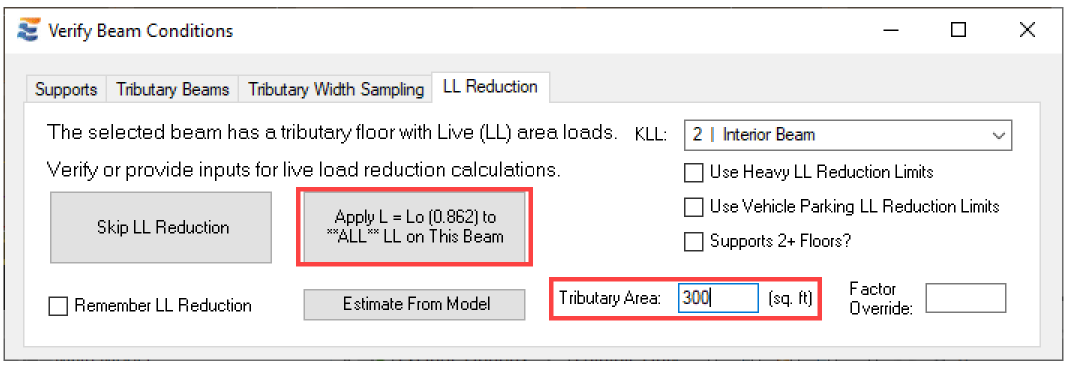
As indicated previously, the product of KLL * At used in the reduction factor calculation may be manually altered outside the preset options found in the code by manually typing an arbitrary numerical value into the KLL menu box. The form will not update if a non-numerical value is typed.

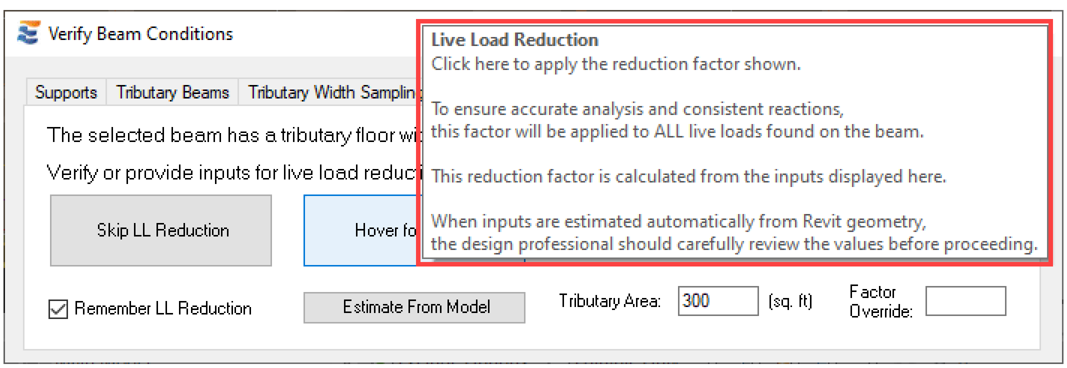


A manual entry for KLL may be reverted to default code values from the drop-down at any time.

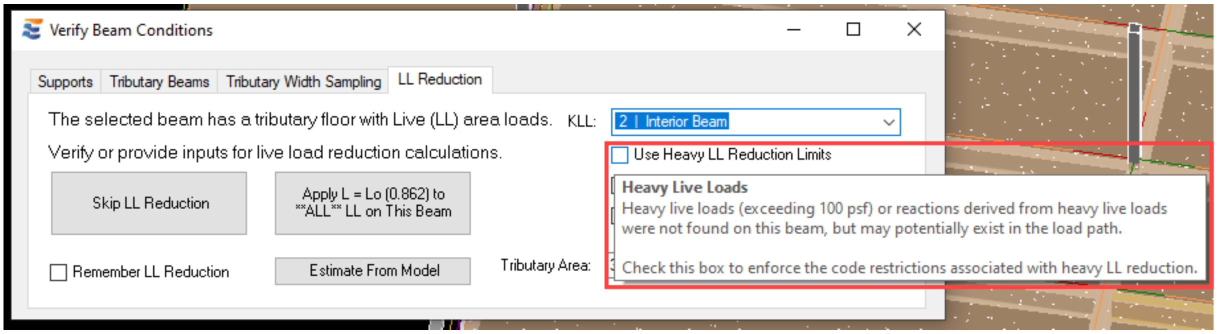


Manually changing any input in the window will cause both the calculated factor and the associated tooltip details to update.

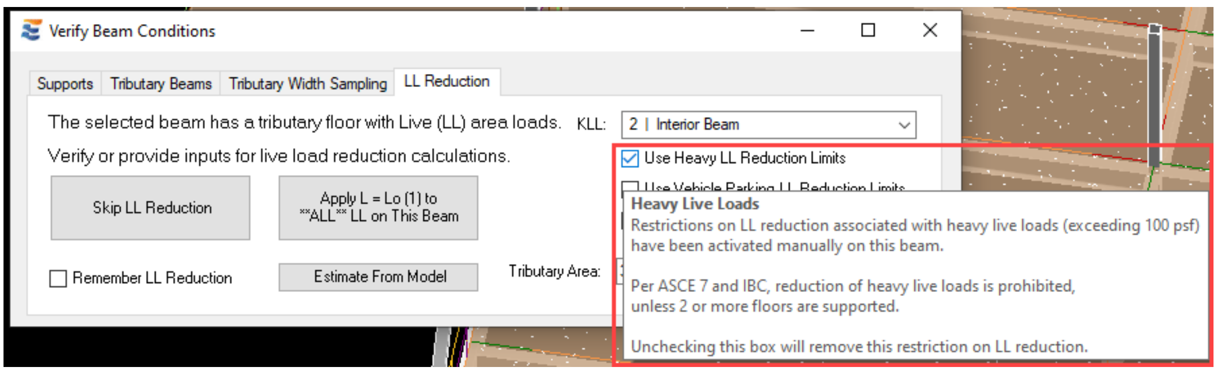




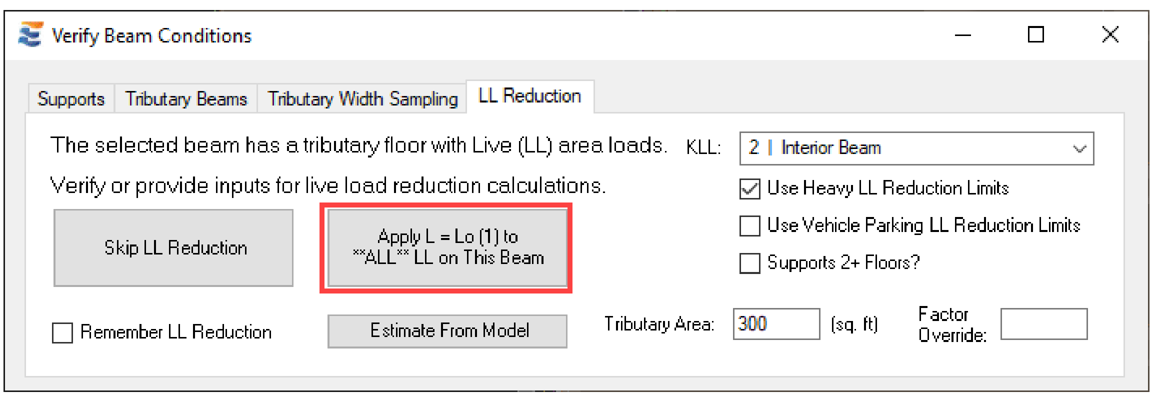
In addition to the KLL and tributary area inputs, the window also includes three checkboxes which allow the user to manually invoke specific building code provisions that restrict the reduction of live loads. The first checkbox allows the user to enforce the code limitations that would apply if the beam or girder supported heavy live loads.

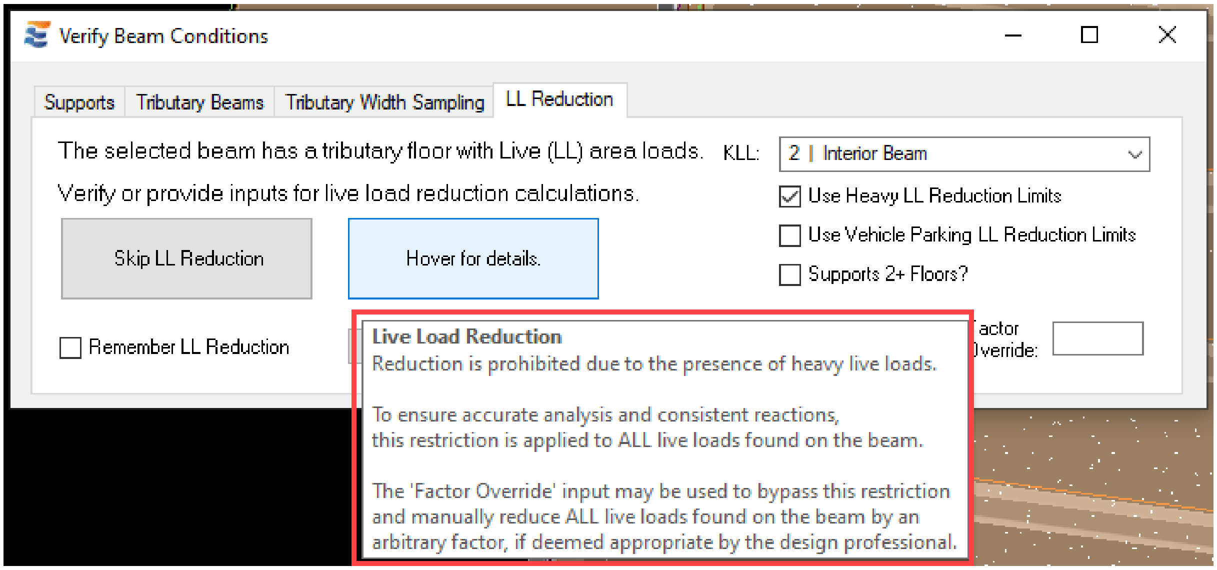


The tooltip balloon for the checkbox updates dynamically and presents the user with different information based on whether the restriction has been activated.

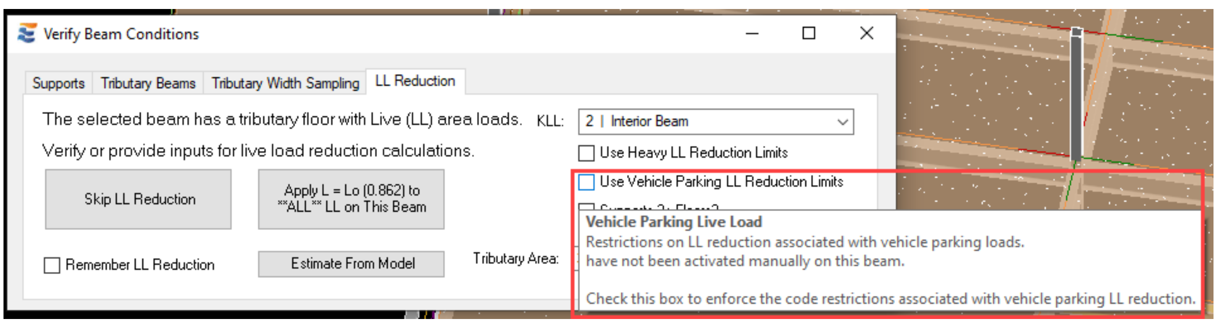


When this restriction is manually activated, the calculated reduction factor and its accompanying tooltip balloon also update in real time.

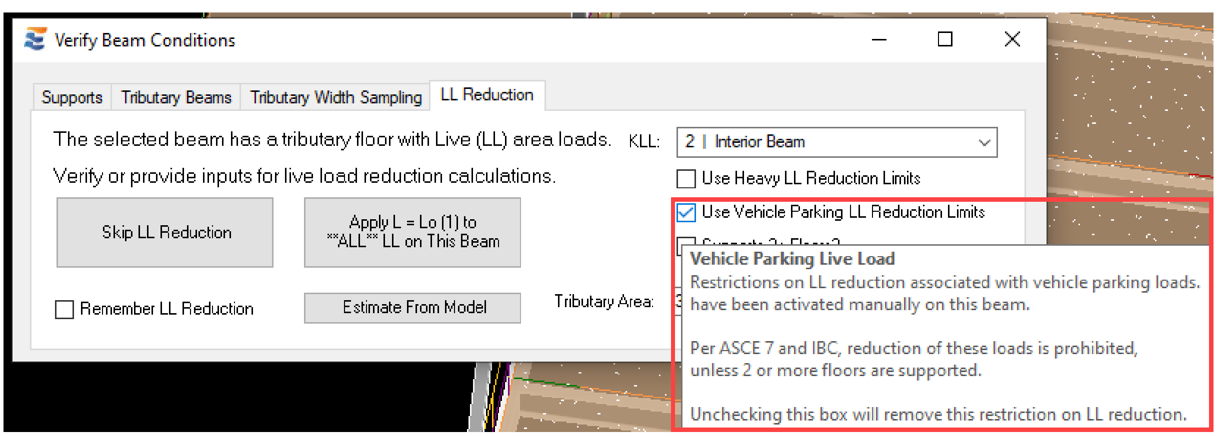




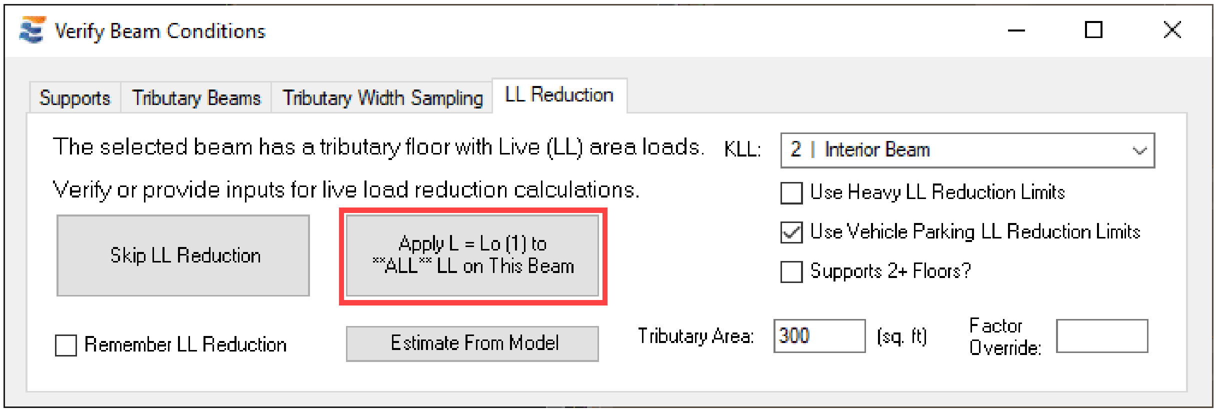
The second checkbox allows the user to enforce the code limitations that would apply if the beam or girder supported vehicle parking live loads.

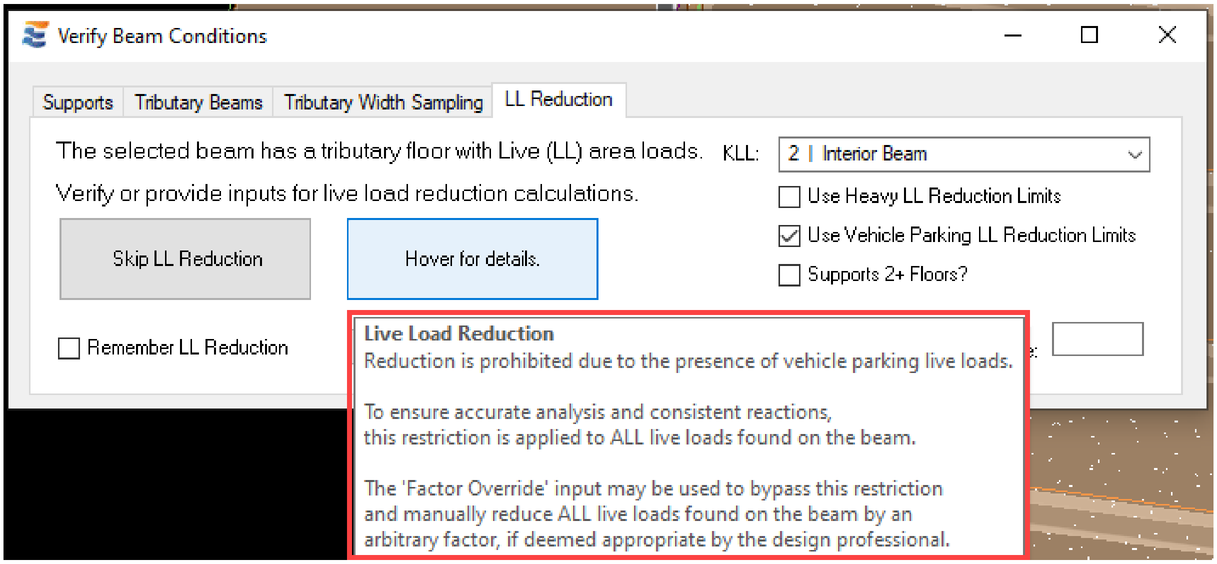


The tooltip balloon for the checkbox updates dynamically and presents the user with different information based on whether the restriction has been activated.

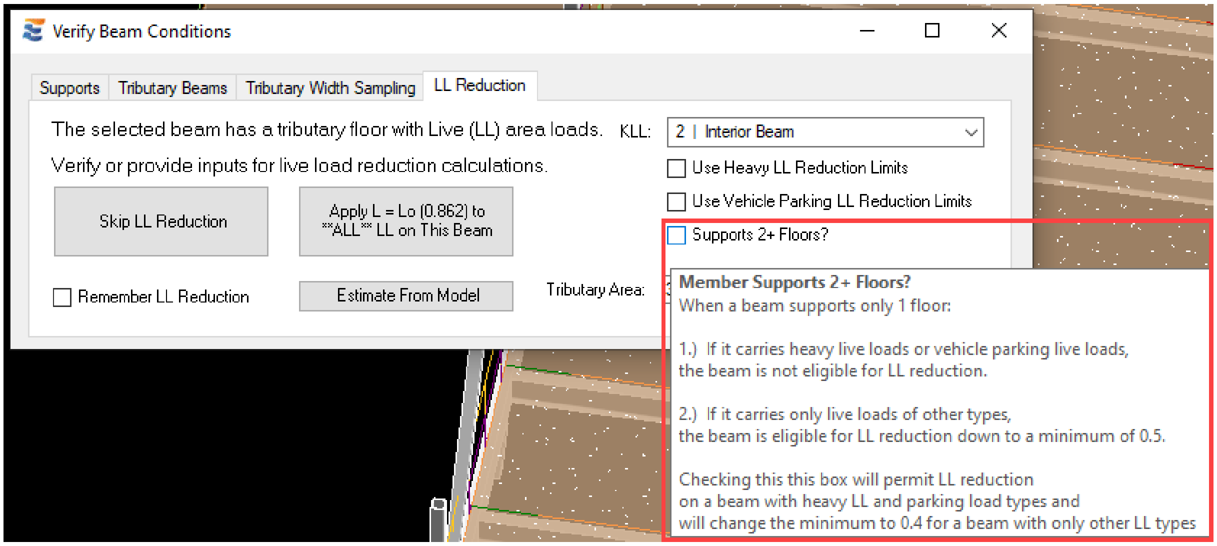


When this restriction is manually activated, the calculated reduction factor and its accompanying tooltip balloon also update in real time.

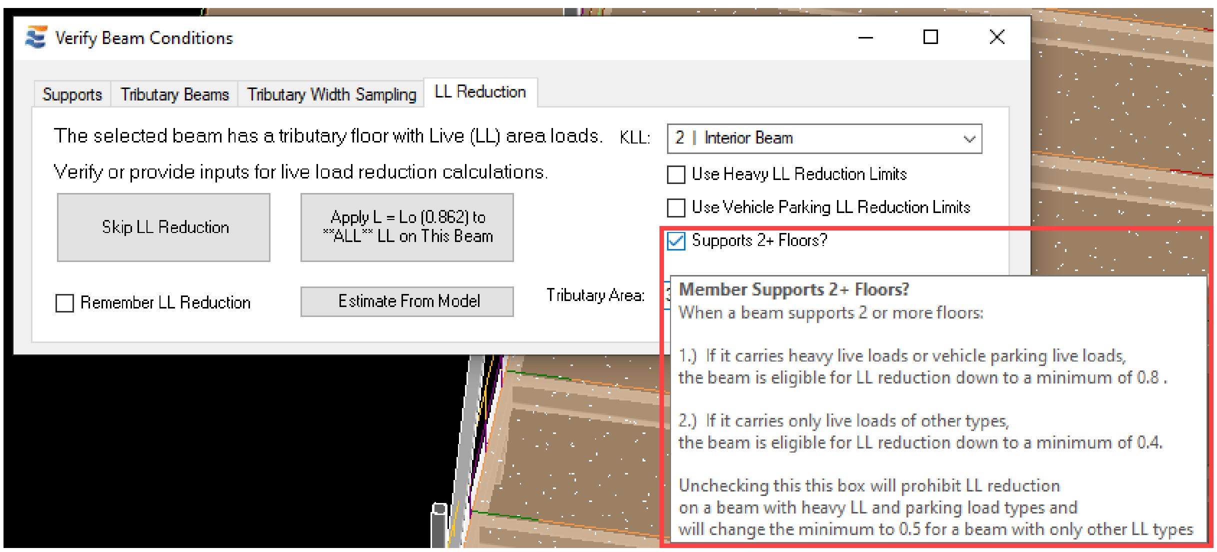




The third checkbox allows the user to manually invoke the alternate live load reduction criteria permitted in the building code if the beam or girder supports 2 or more floors. This has a potential impact on reduction calculations regardless of whether the beam carries heavy live load, parking live load, or other reducible types. The impacts are documented in the tooltip balloon for the checkbox.

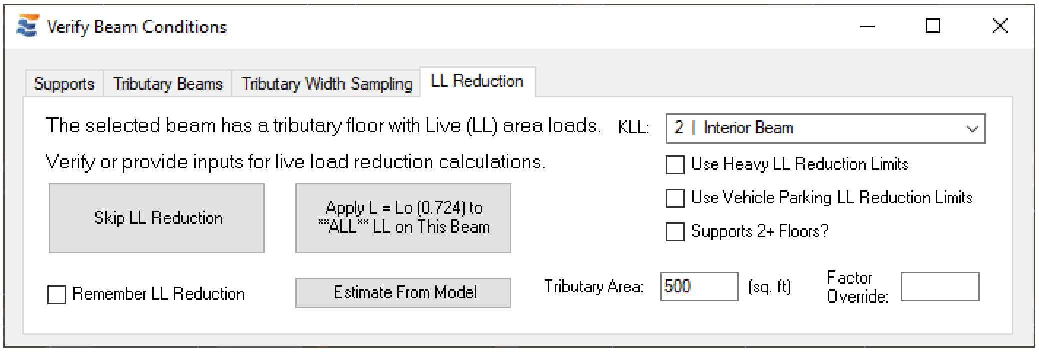


As with the other controls in the launch window, altering the state of the checkbox results in dynamically updated information in the tooltip balloon.

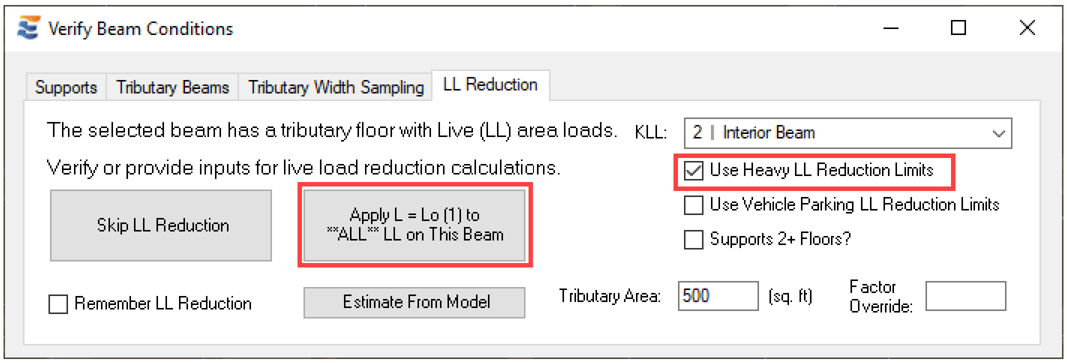


As described in the tooltip balloon, the “Supports 2+ Floors” checkbox is particularly significant for beams or girders that support heavy live loads or vehicle parking loads. As indicated in previous images, the presence of heavy or parking live loads will cause the calculated reduction factor to default to 1.0 with an indication that reduction is prohibited. However, activating the “Supports 2+ Floors” option will allow calculated reduction, down to the code minimum of 0.8.

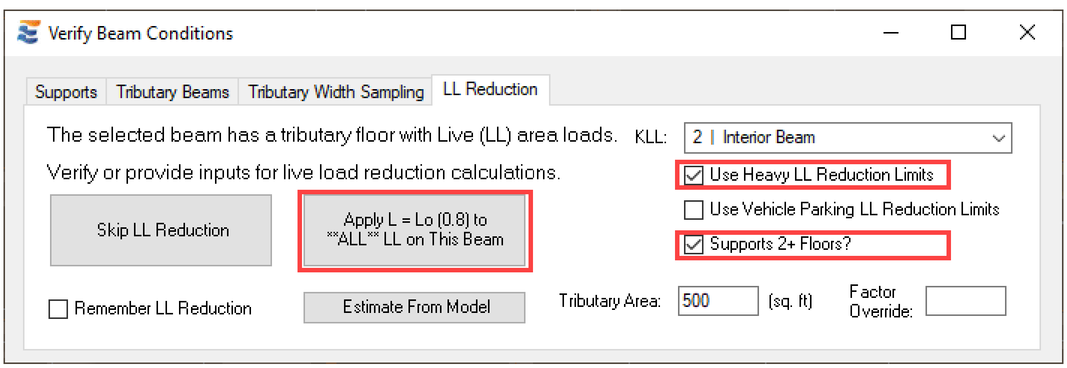
With no restrictions, where the calculated restriction falls below 0.8:

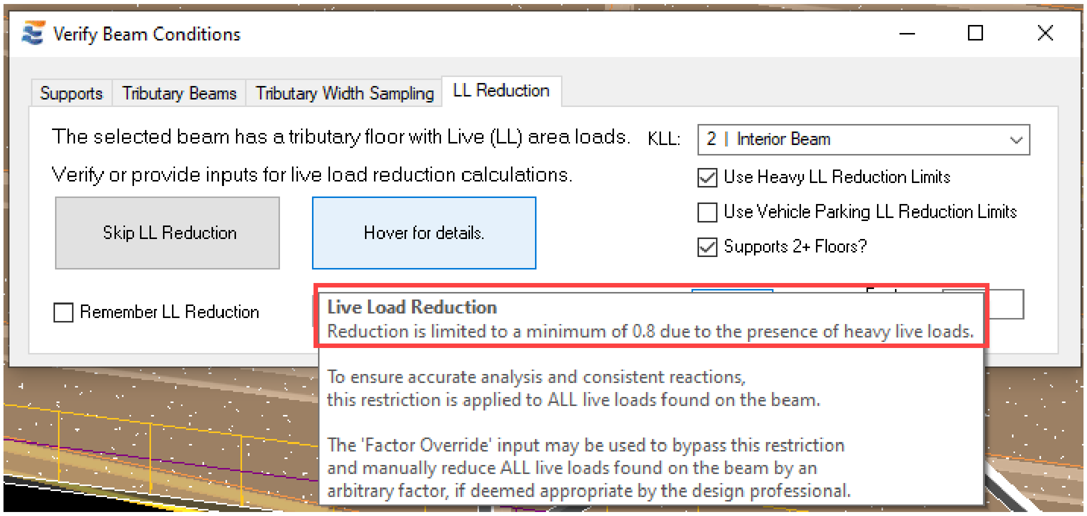


With heavy live load default restrictions:

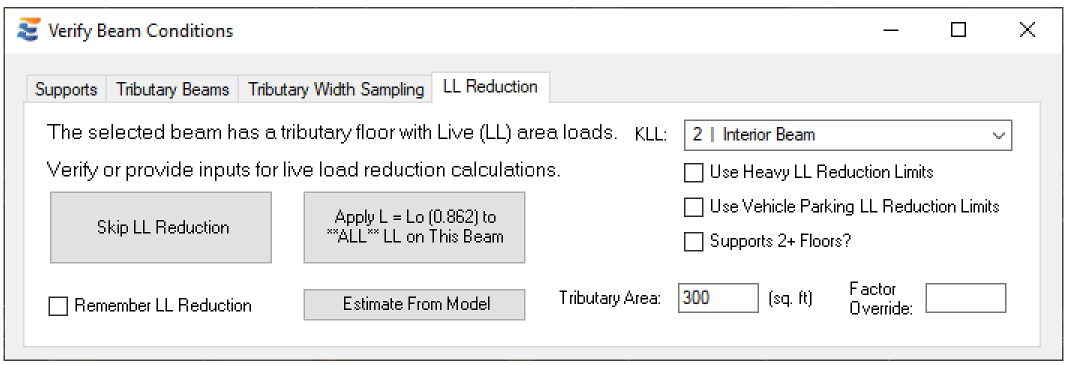


With heavy live load restrictions and supporting 2+ floors:

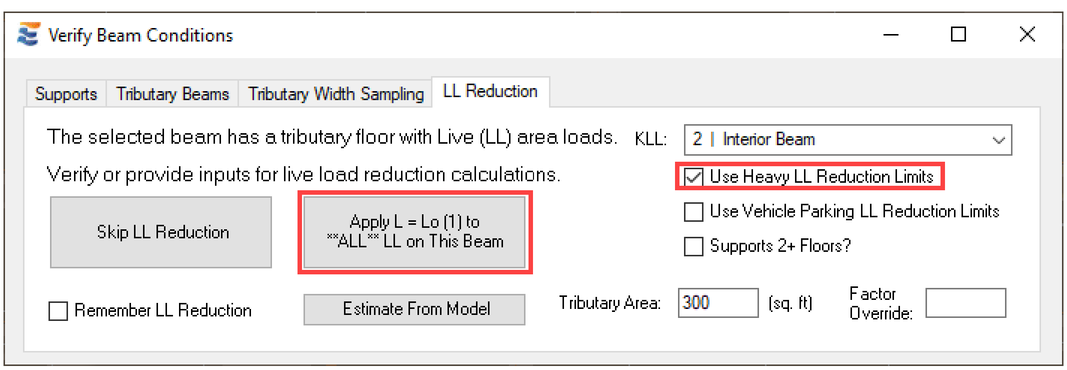




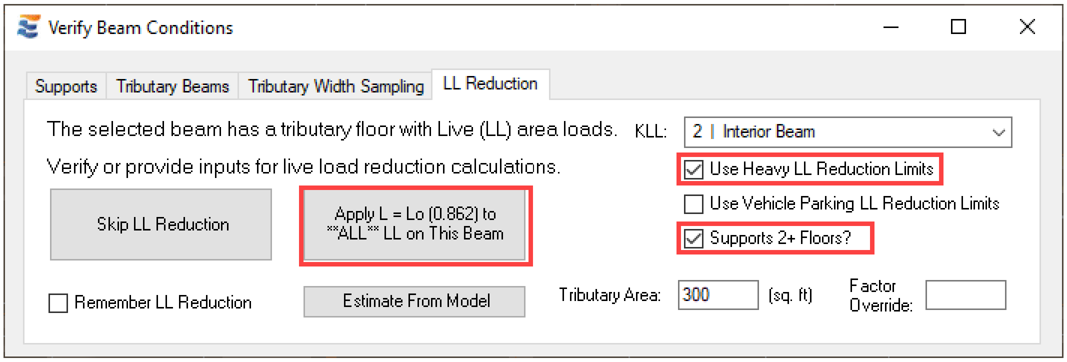
With no restrictions, where the calculated restriction falls above 0.8:



With heavy live load default restrictions:

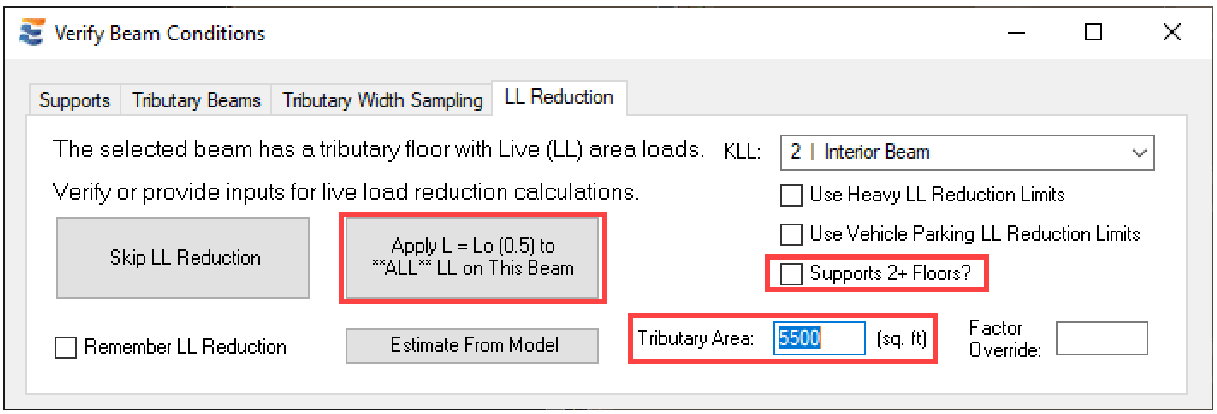


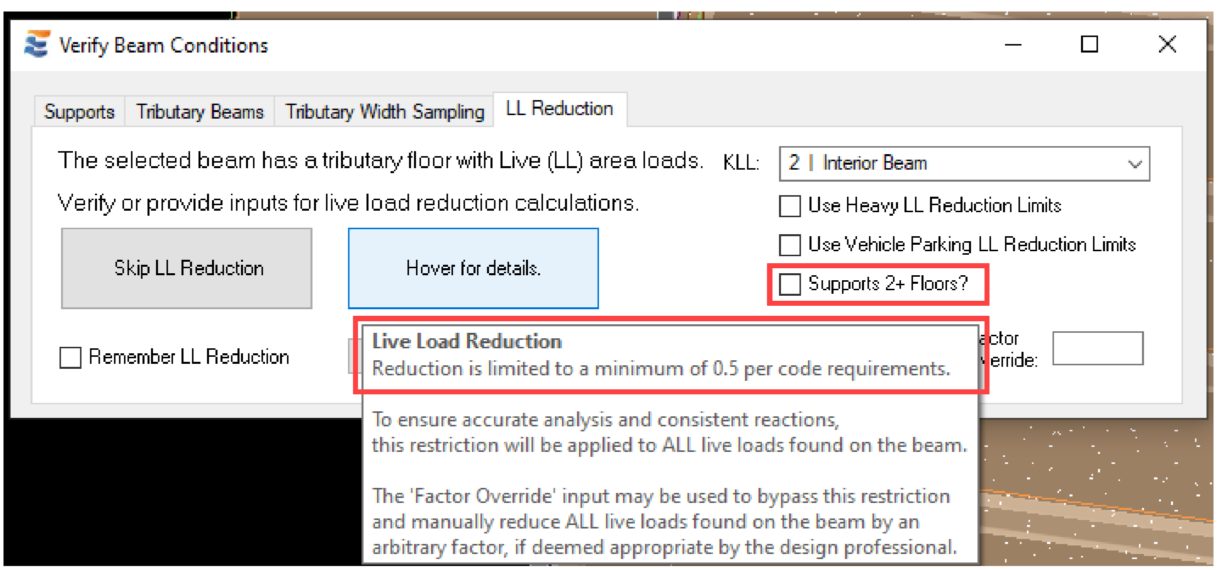
With heavy live load restrictions and supporting 2+ floors:



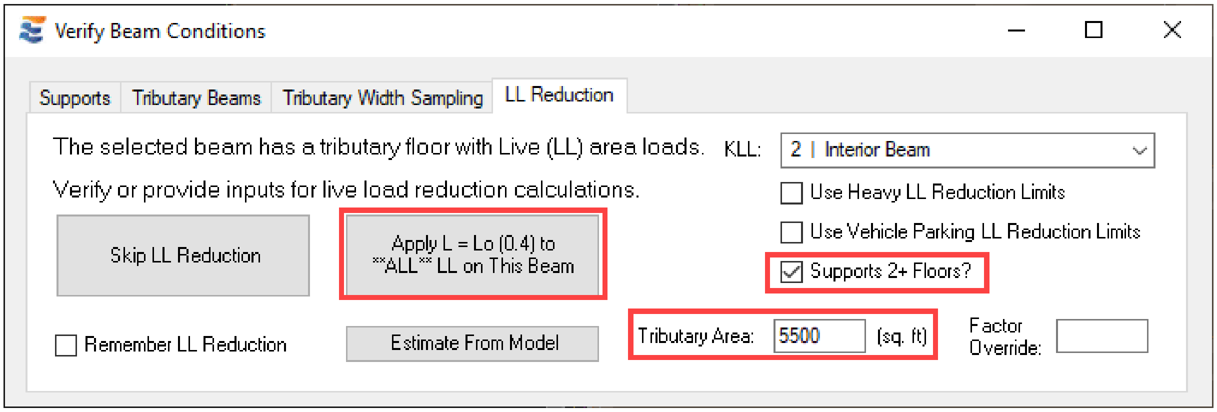
Alternatively, these same settings may be activated for vehicle parking loads, with identical results since the code provisions for minimum reduction are the same.

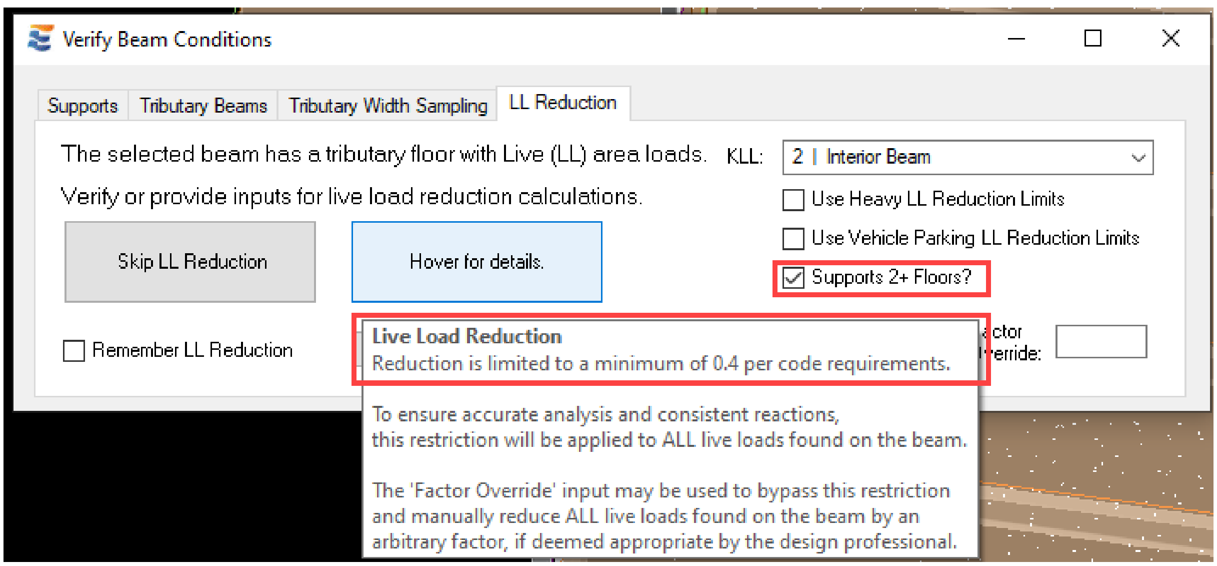
For beams or girders that do not carry heavy live loads or vehicle parking loads, the “Supports 2+ Floors” checkbox is important for setting the code-dictated lower bound on permissible reduction when KLL * At becomes very large.





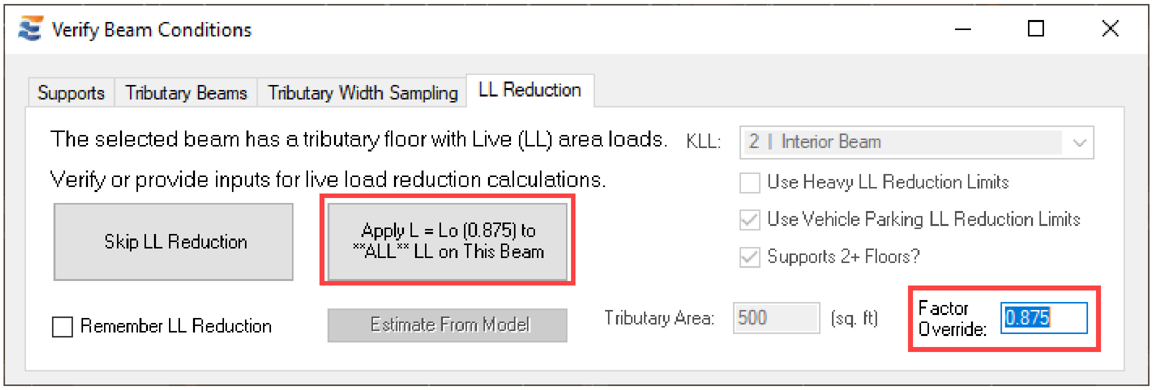
In the case of an element supporting 2 or more floors, ASCE 7 and IBC permit an abridged lower limit on reduction.



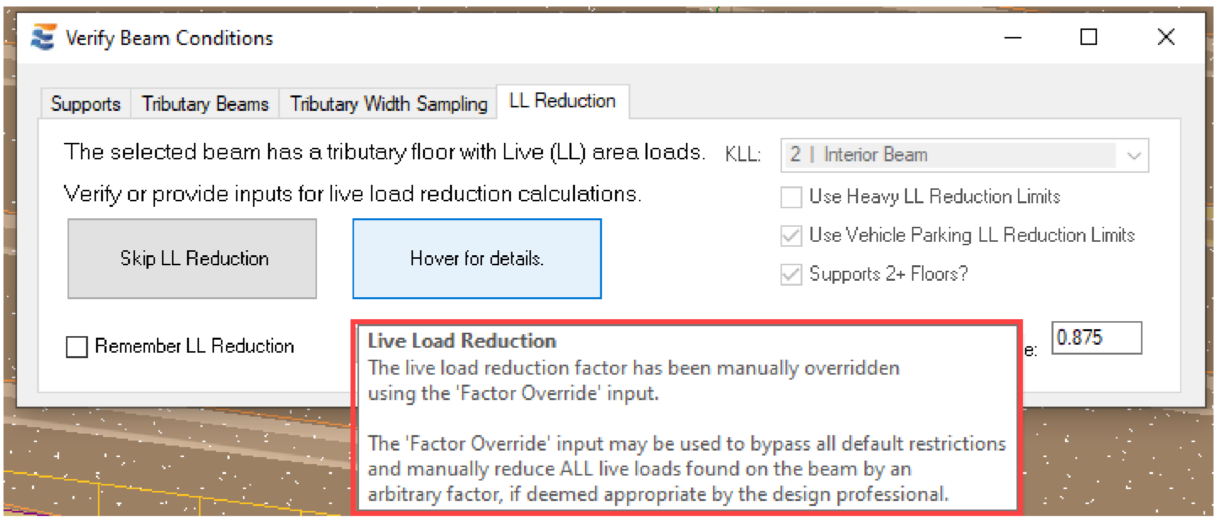


In any situation where the design professional deems appropriate, all of the restrictions discussed here may be bypassed in order to reduce all loads of type LL by an arbitrary factor. This is done using the “Factor Override” textbox. When a numerical value is entered in this field, all other inputs will be disabled. The form will not update in response to a non-numerical value.

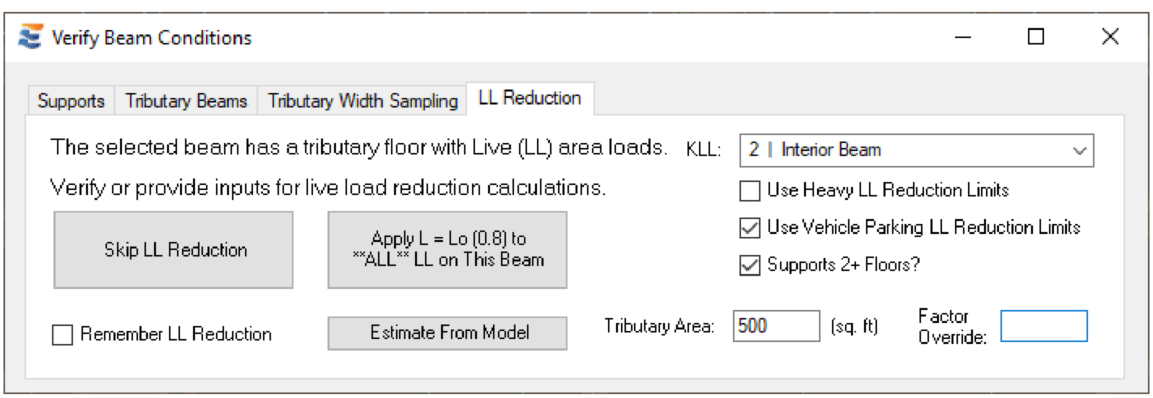
When the form is in this state, the reduction factor is not a calculated value. All live loads will be uniformly reduced using the exact factor entered by the user.



When the form is in this state, ENERCALC for Revit does not perform any checks to validate the provided reduction factor against code restrictions.

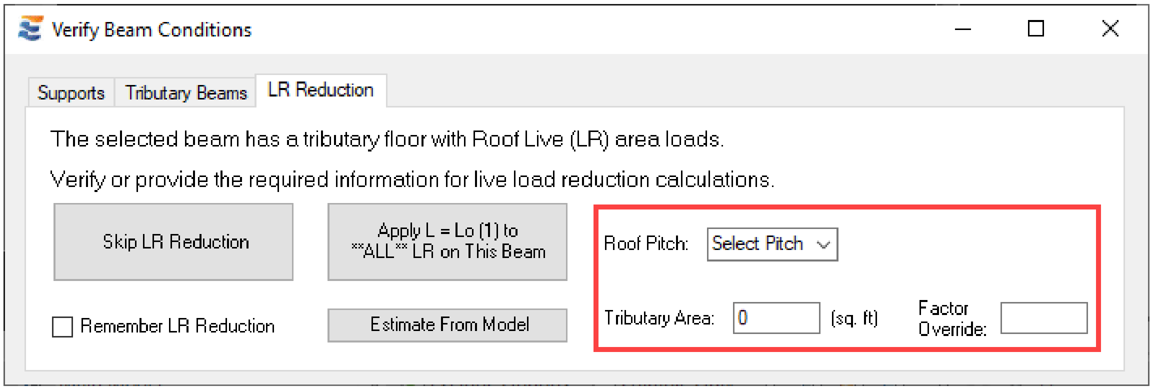


Removing the override by “backspacing” the numerical value in the textbox will return the form to a calculated state based on current inputs.



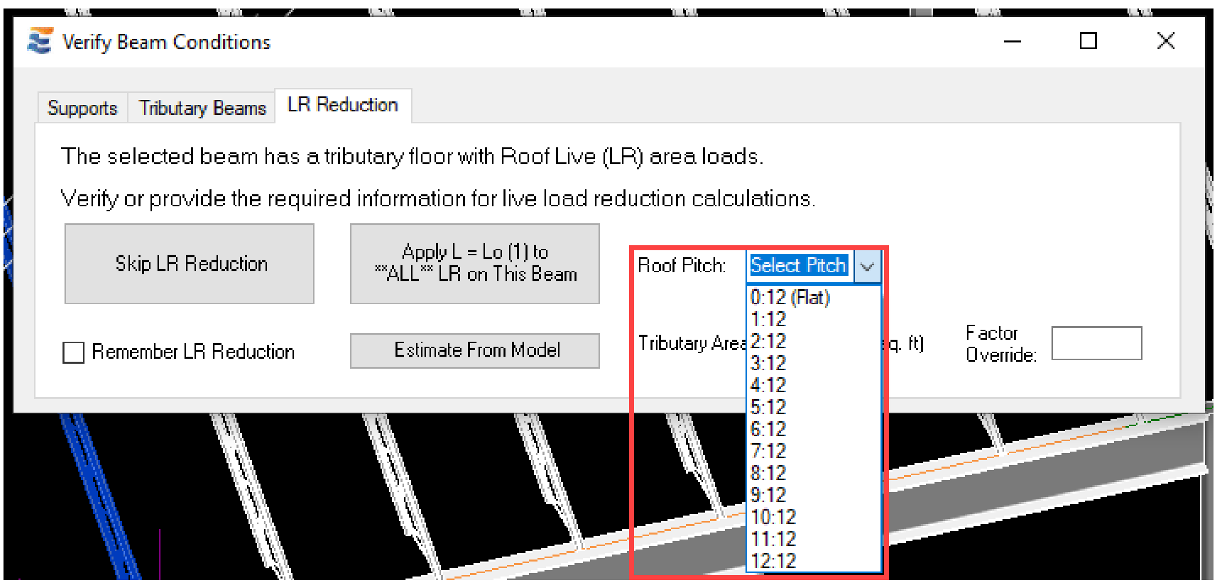
10.4.12.4 Reduction of Roof Live Loads (LR)

If the user does wish to take advantage of LR reduction, the appropriate reduction factor is calculated using the inputs provided in the window. These inputs correlate directly to the parameters and limitations outlined in ASCE 7 and IBC for reduction of roof live loads.

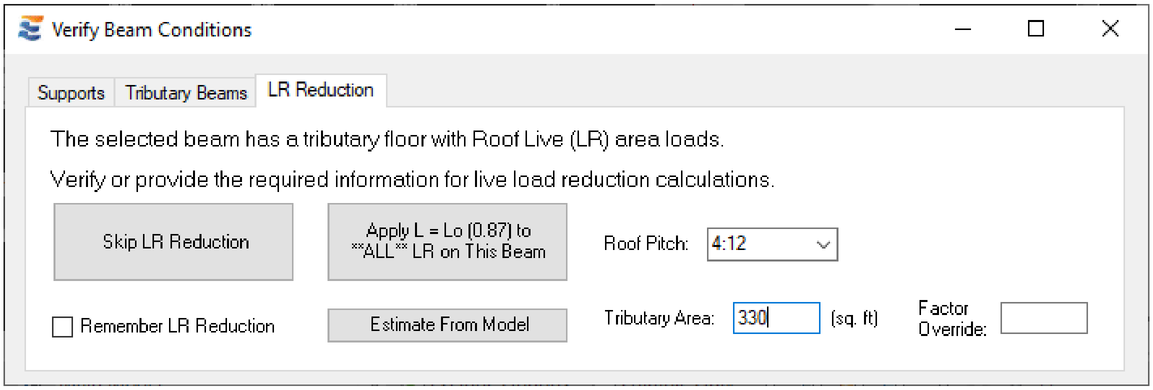


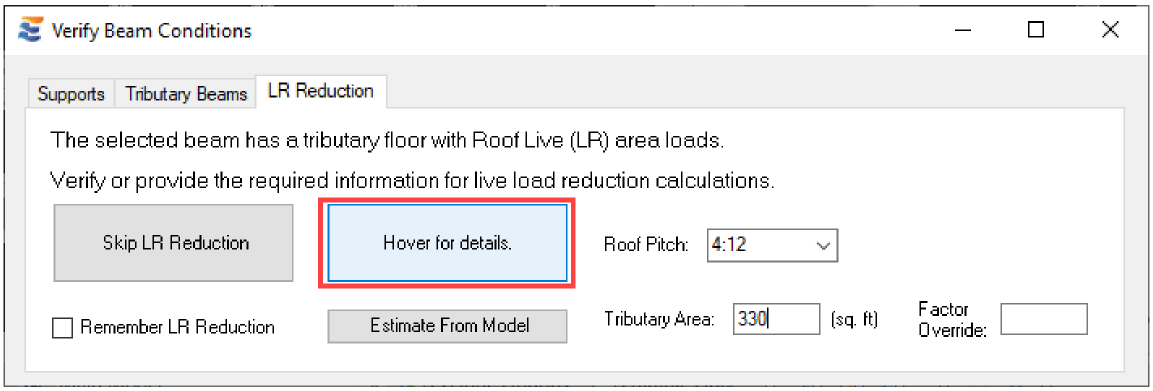
As the inputs in this window are modified, the LR reduction factor to be applied will update in real-time, as indicated on the “Apply…” button.

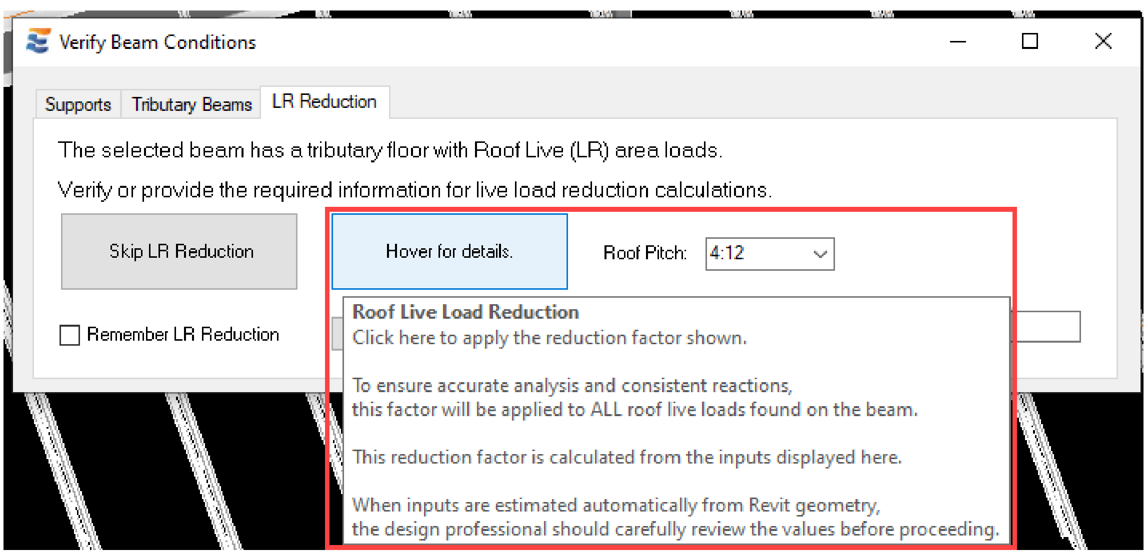
The tributary area is manually modified by typing a numerical value into the “Tributary Area:” textbox. Note that the window will not respond to tributary area modifications when the roof pitch has not been set. The roof pitch is selected from preset options.



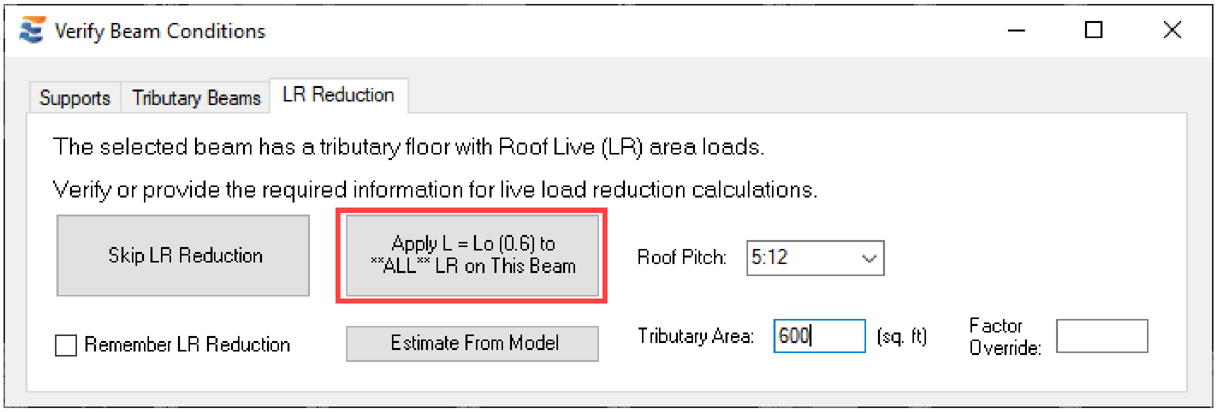
When the cursor passes over the “Apply…” button, the button will show a notification reminding the user to hover over the button for detailed information about the LR reduction factor being presented.

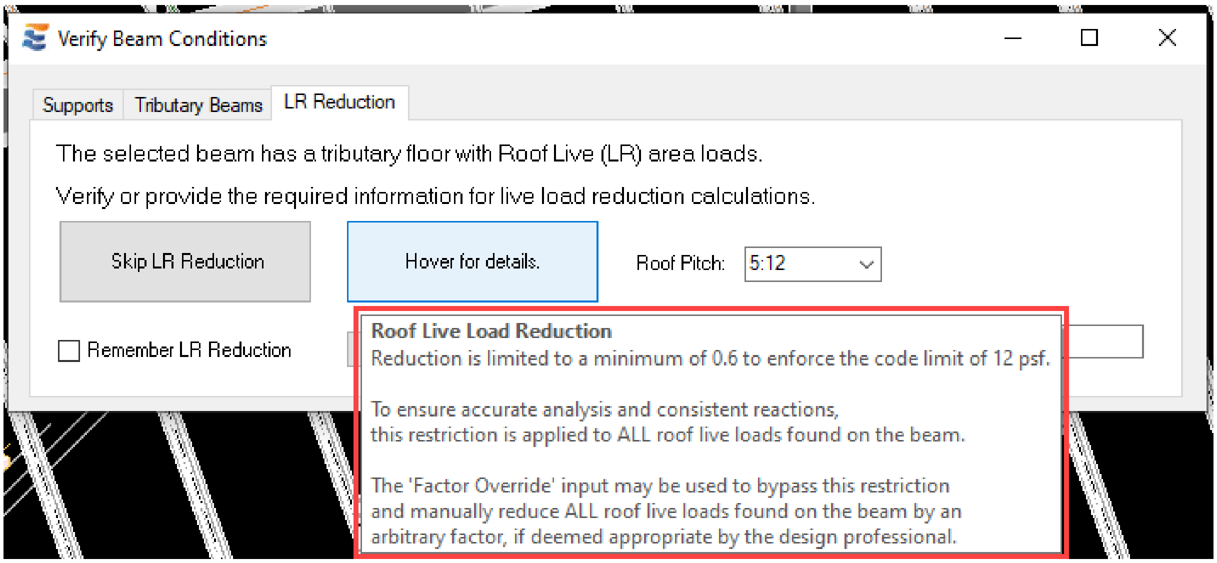






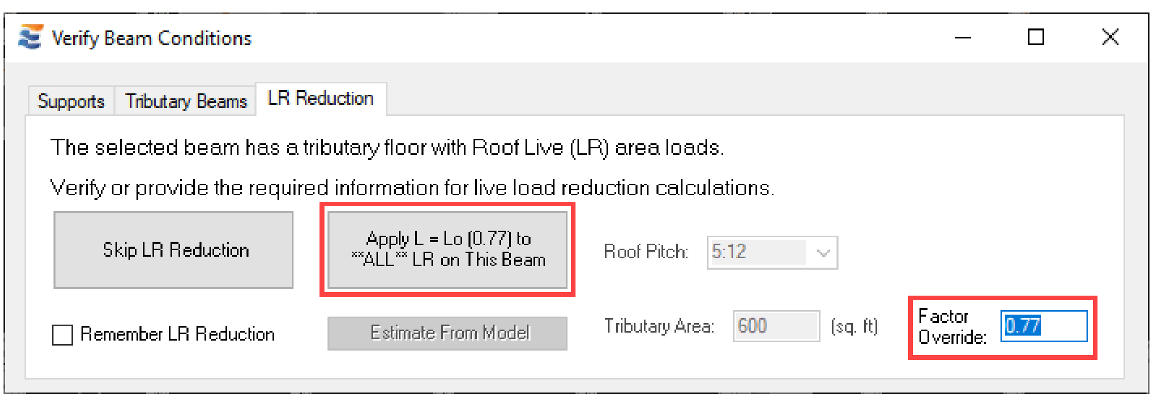
In scenarios where the R1 and R2 factors for roof live load reduction both reach minimum values simultaneously, the resulting reduction factor is forcibly limited to a minimum of 0.6. The basis for this restriction is the code requirement that roof live loads may not be reduced below a magnitude of 12 psf. Based on the presumption that nearly all uniform roof live loads will have an unreduced magnitude of 20 psf, any reduction factor less than 0.6 would violate this requirement.



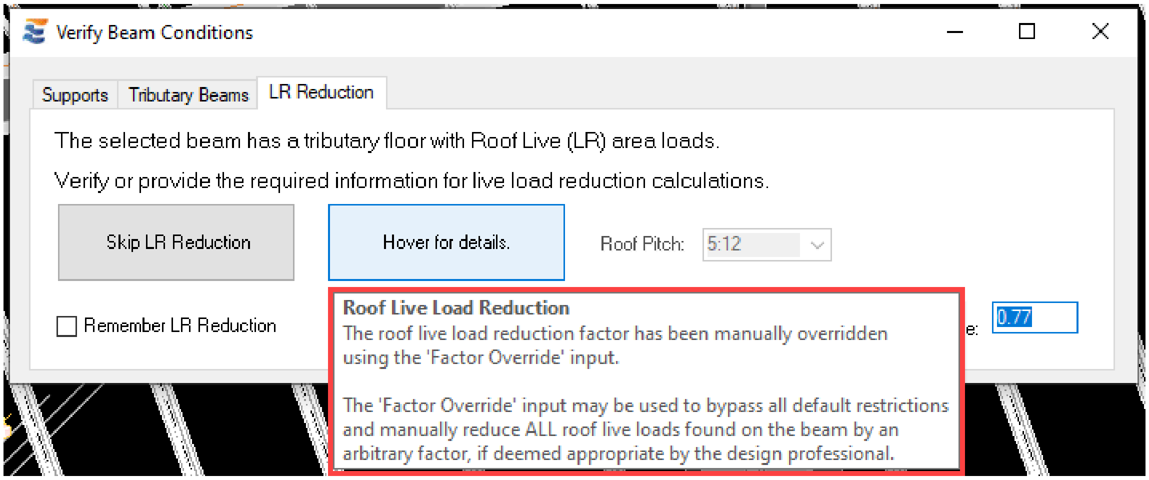


In special cases where the user deems it appropriate to reduce below this threshold or apply an arbitrary reduction factor for any reason, the “Factor Override” input may be used. This override functions identically to the corresponding field on the “LL Reduction” tab.

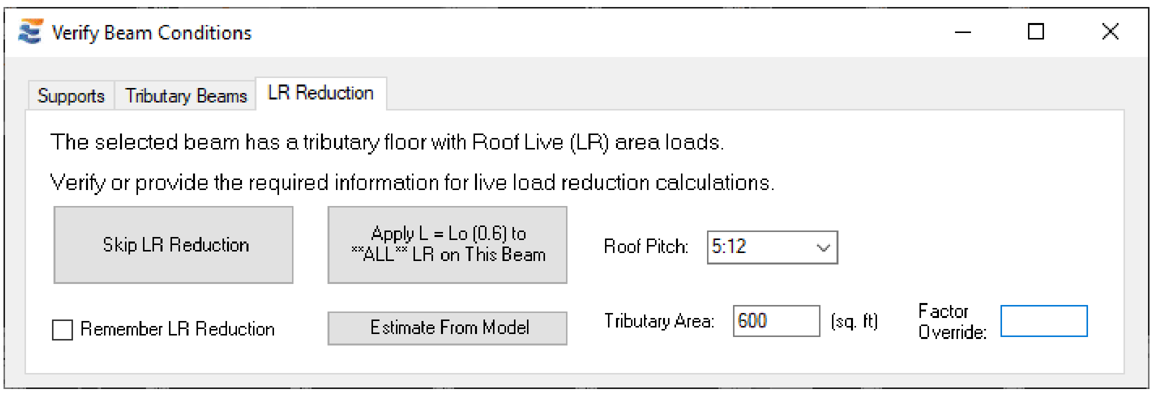
When a numerical value is entered in this field, all other inputs will be disabled. The form will not update in response to a non-numerical value. When the form is in this state, the reduction factor is not a calculated value. All roof live loads will be uniformly reduced using the exact factor entered by the user.



When the form is in this state, ENERCALC for Revit does not perform any checks to validate the provided reduction factor against code restrictions.

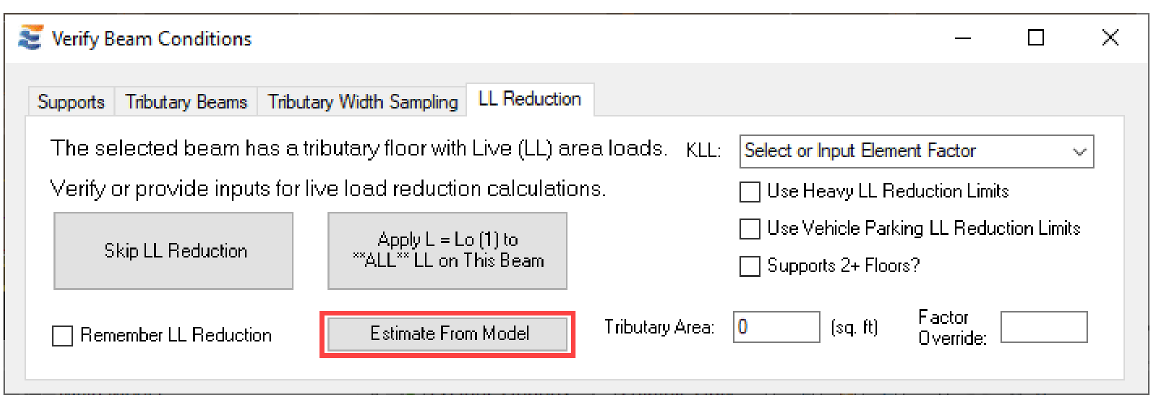


Removing the override by “backspacing” the numerical value in the textbox will return the form to a calculated state based on current inputs.



10.4.12.5 Using Revit Model Geometry

Although the user has the option to populate these inputs manually, ENERCALC for Revit provides the ability to automatically estimate the appropriate values from the geometry of the Revit model using the “Estimate From Model” button.



Clicking this button will initiate a brief geometry detection process punctuated by progress bars to indicate the completion of each step in the detection process. This process produces an estimate of the tributary area and a best guess for the most likely KLL element factor of the beam. This detection process also automatically detects the presence of heavy live loads which are subject to reduction limitations, as discussed previously. Although LL reduction inputs can be estimated for virtually any beam, the following factors impact the speed and accuracy of the detection process:

- Structural floors:
  - Beams supporting a floor will take incrementally longer to estimate than beams without
  - The geometry of a floor generally helps produce more accurate tributary area results

- Tributary beams:
  - Beams or girders supporting other beams require longer calculation times than those without
  - Longer execution times are due to recursive testing of each connected element to estimate total tributary area.

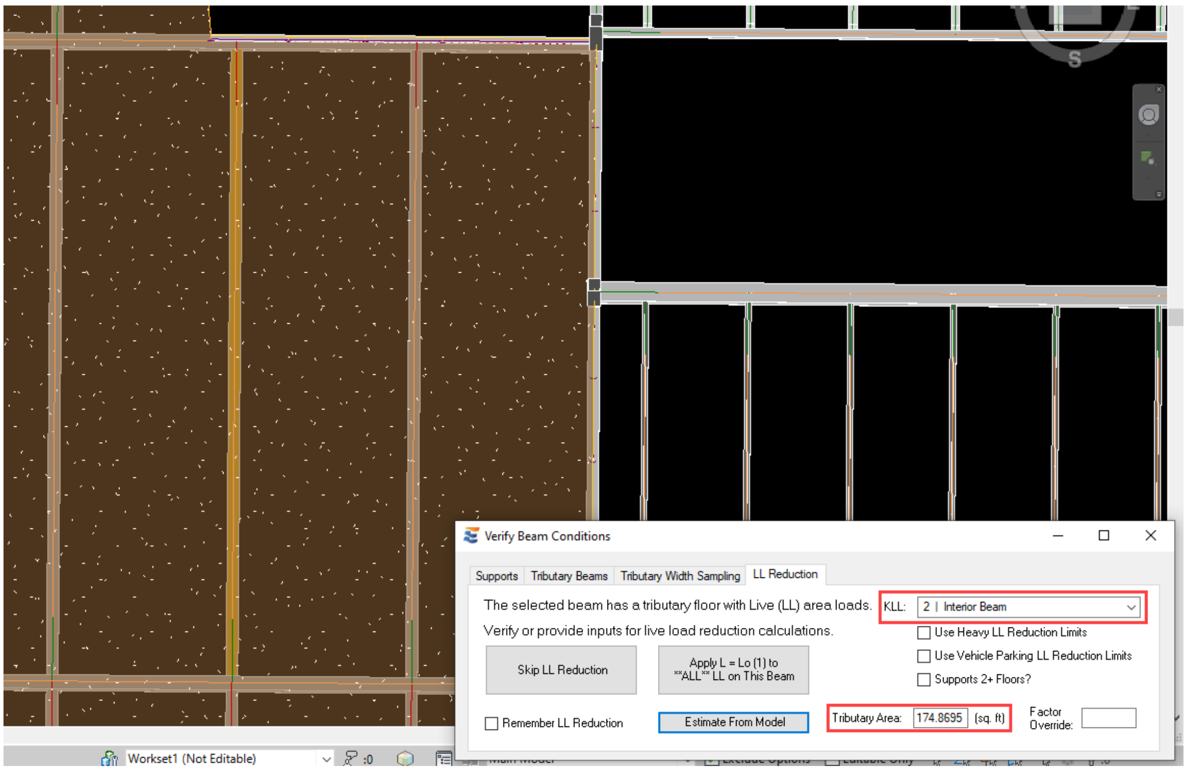
- “Load-Linking”:
  - Beams or girders with “Load-Linked” reaction forces require longer calculation times than those without
  - Longer execution times are due to recursive testing of each connected element to estimate total tributary area.

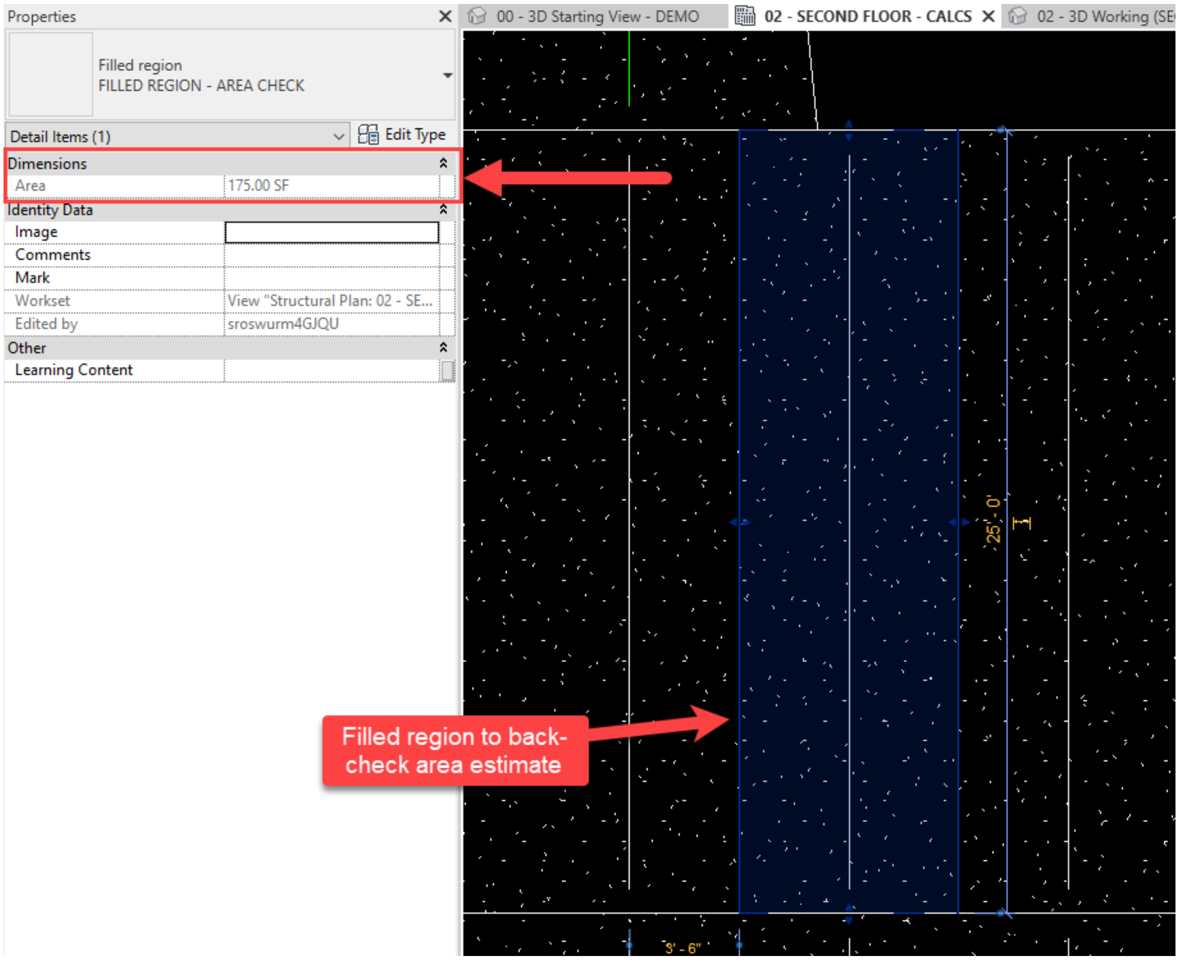
- Span Geometry:
  - Due to the limitations of the estimation process, beams or girders that fall within a broader load path that is subject to tributary area estimation are treated as simple-span
  - This means that when a girder is tested for tributary area, each of the infill beams it supports is presumed to contribute 50% of its respective total tributary area to the supporting girder
  - More complex framing systems where this assumption is not valid will require careful attention and manual adjustment of the resulting area by the design professional

The series of examples that follows will illustrate the results produced by this estimation process in a number of different beam and girder framing conditions.

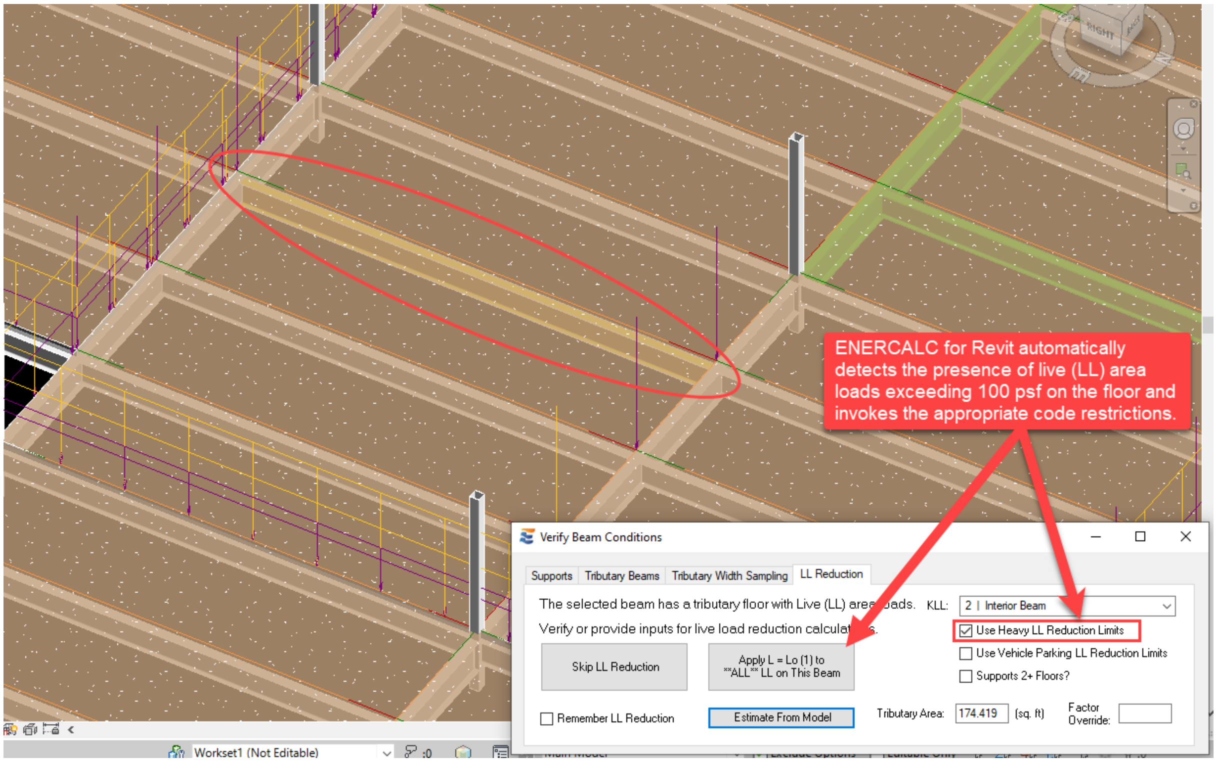
For each individual scenario, a filled region is shown as an independent verification of the tributary area that ENERCALC for Revit reports. These regions are shown for illustration only; they are ***NOT*** created by ENERCALC for Revit.

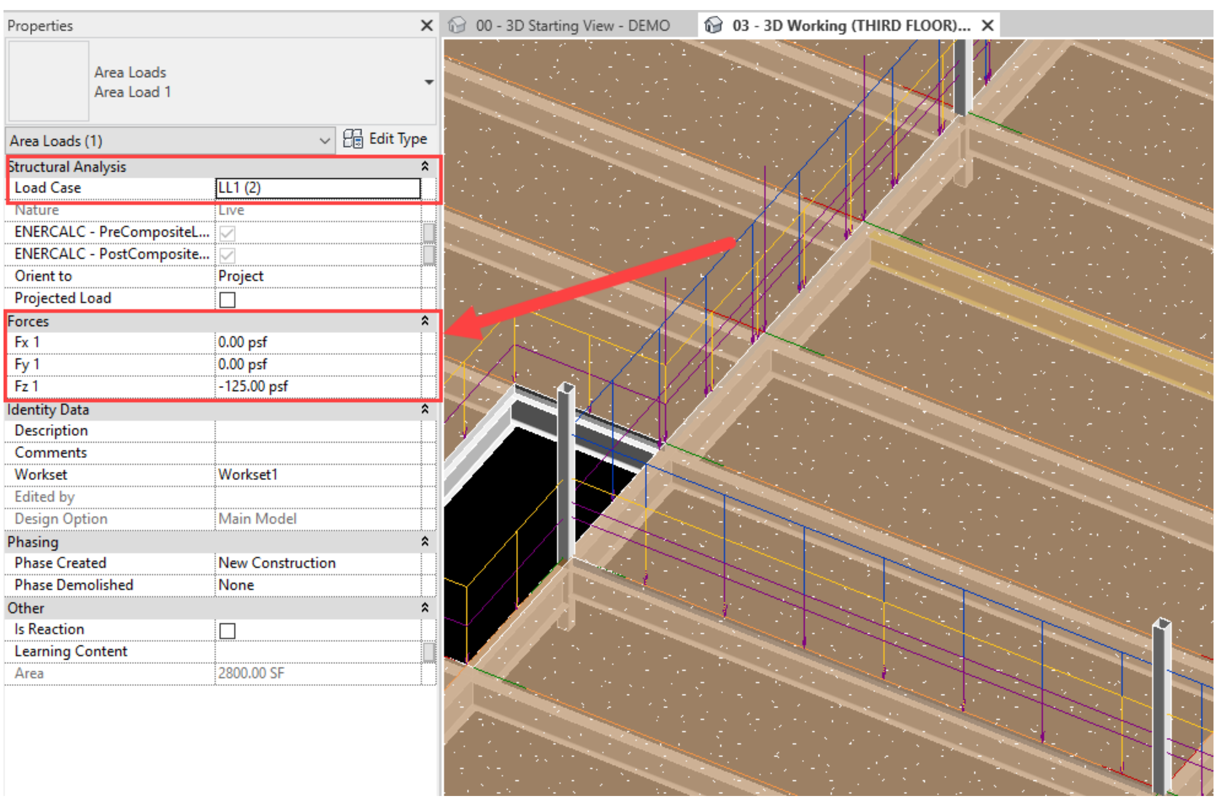
Basic interior floor beam (with slab):

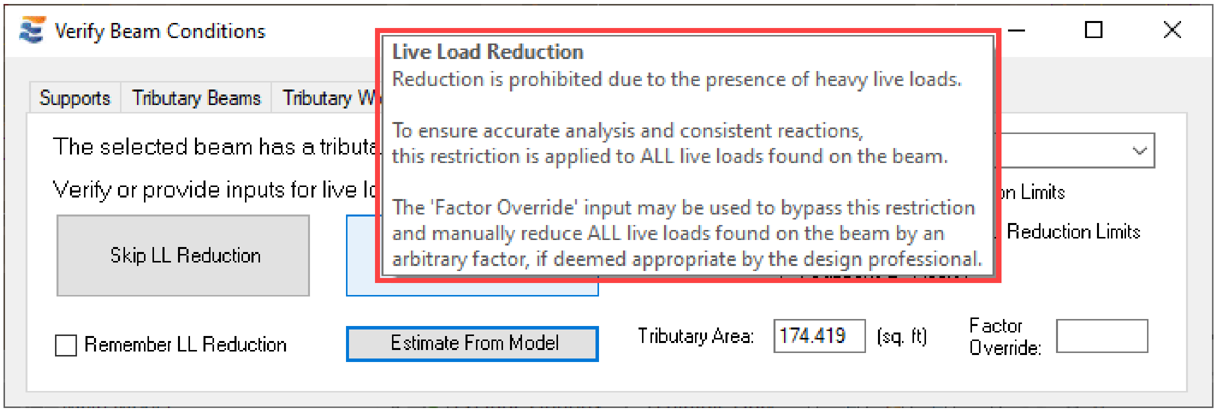




Basic interior floor beam (with slab and heavy area loads):

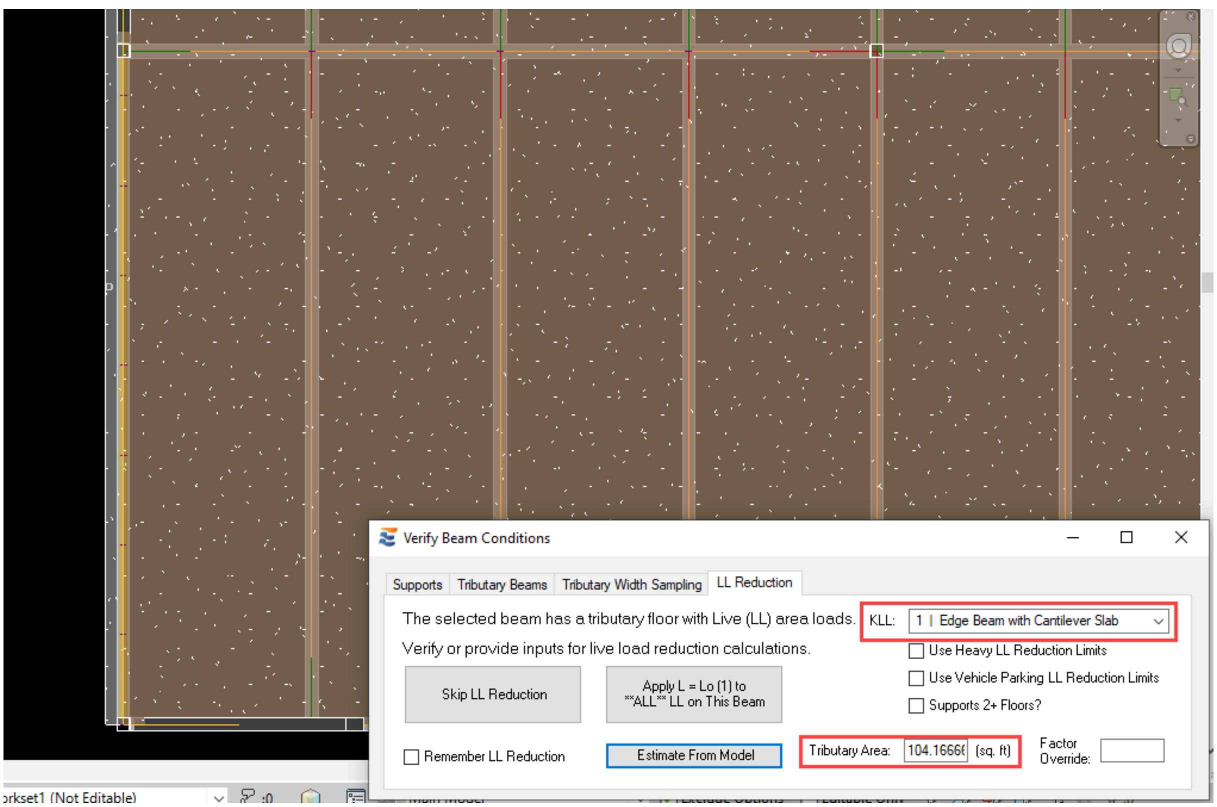

ENERCALC for Revit automatically detects the presence of live (LL) area loads exceeding 100 psf on the floor and invokes the appropriate code restrictions.
Verify Beam Conditions
Supports
Tributary Beams
Tributary Width Sampling
LL Reduction
The selected beam has a tributary floor with Live (LL) area loads.
Verify or provide inputs for live load reduction calculations.
KLL:
2 | Interior Beam
Use Heavy LL Reduction Limits
Use Vehicle Parking LL Reduction Limits
Supports 2+ Floors?
Skip LL Reduction
Apply L = Lo (1) to **ALL** LL on This Beam
Remember LL Reduction
Estimate From Model
Tributary Area:
174.419
(sq. ft)
Factor Override:
Workset1 (Not Editable)

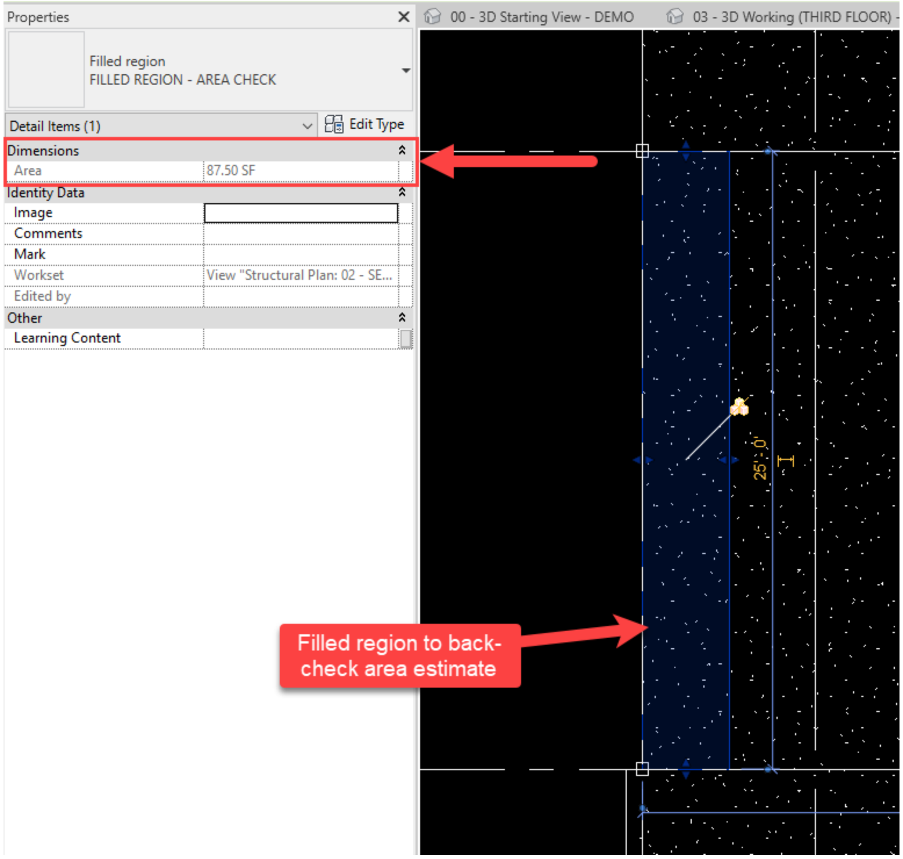

Properties
Area Loads
Area Load 1
Area Loads (1)
Edit Type
Structural Analysis
Load Case
LL1 (2)
Nature
Live
ENERCALC - PreCompositeL...
ENERCALC - PostComposite...
Orient to
Project
Projected Load
Forces
Fx 1
0.00 psf
Fy 1
0.00 psf
Fz 1
-125.00 psf
Identity Data
Description
Comments
Workset
Workset1
Edited by
Design Option
Main Model
Phasing
Phase Created
New Construction
Phase Demolished
None
Other
Is Reaction
Learning Content
Area
2800.00 SF
00 - 3D Starting View - DEMO
03 - 3D Working (THIRD FLOOR)...



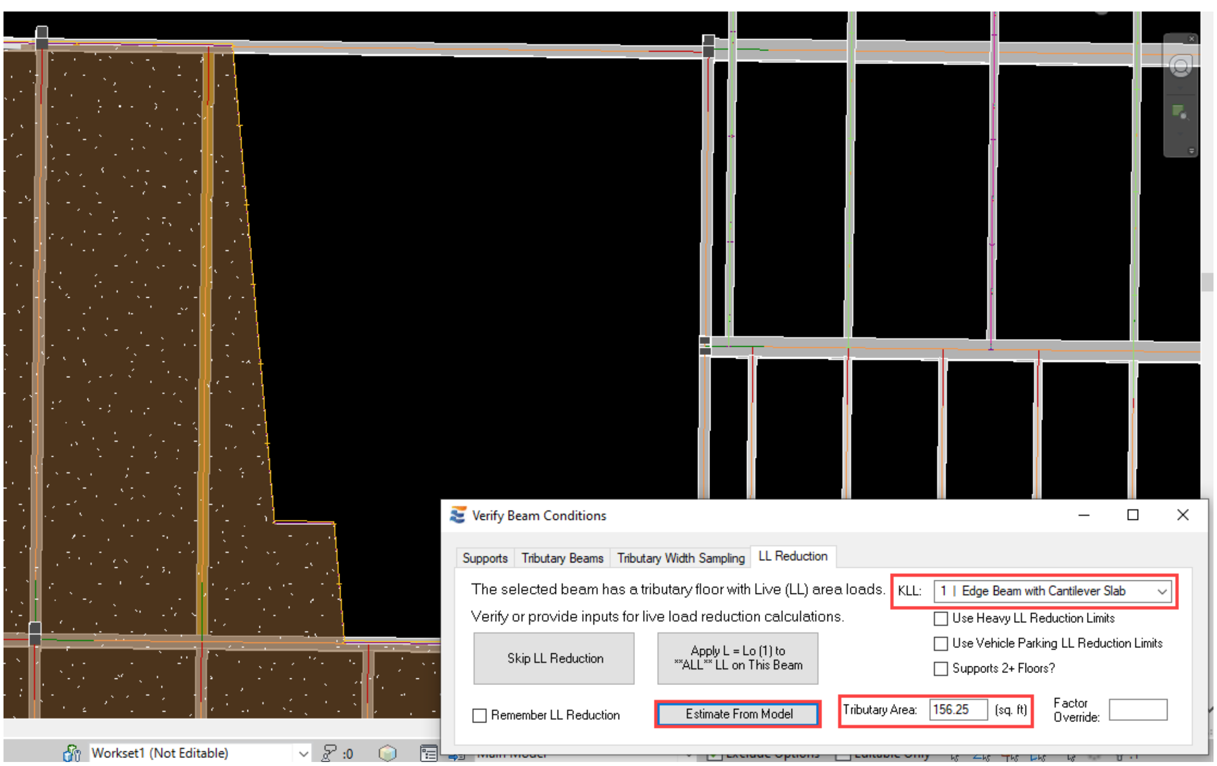
NOTE: Only hosted area loads and point loads created by the Girder Point Loads tool are usable for identifying the presence of heavy live loads. Other point loads and line loads created in the model do not carry any information concerning the magnitude of the underlying area load. The design professional is responsible for verifying in any situation that live load reduction is permissible by code and appropriate for the design.

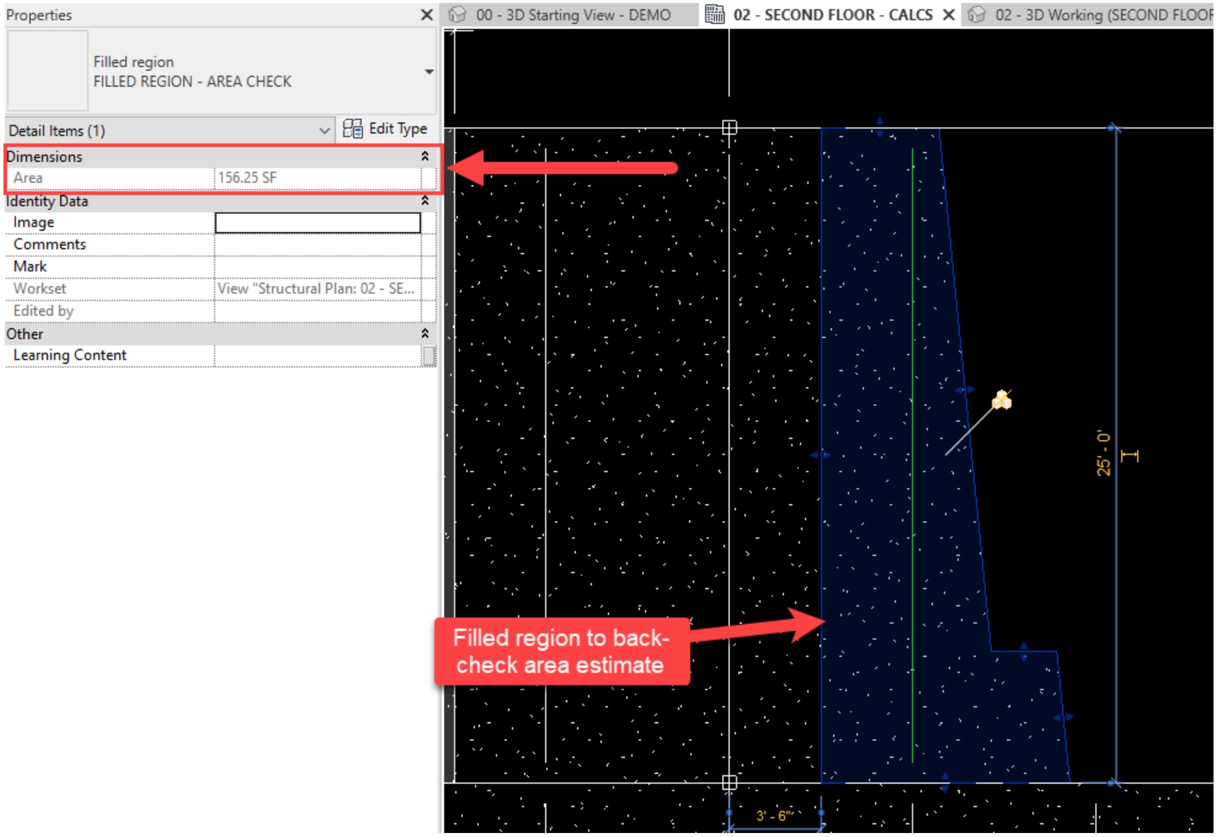
Perimeter beam with uniform slab overhang edge:



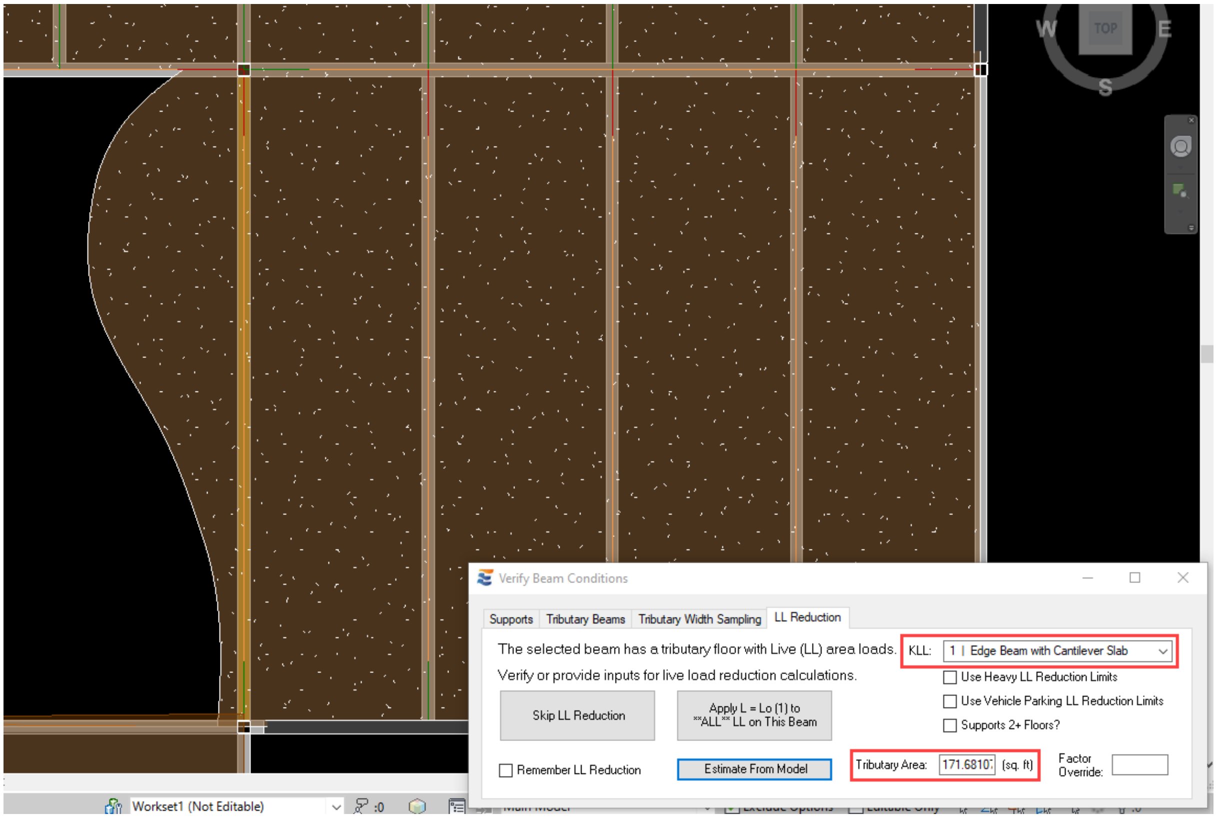


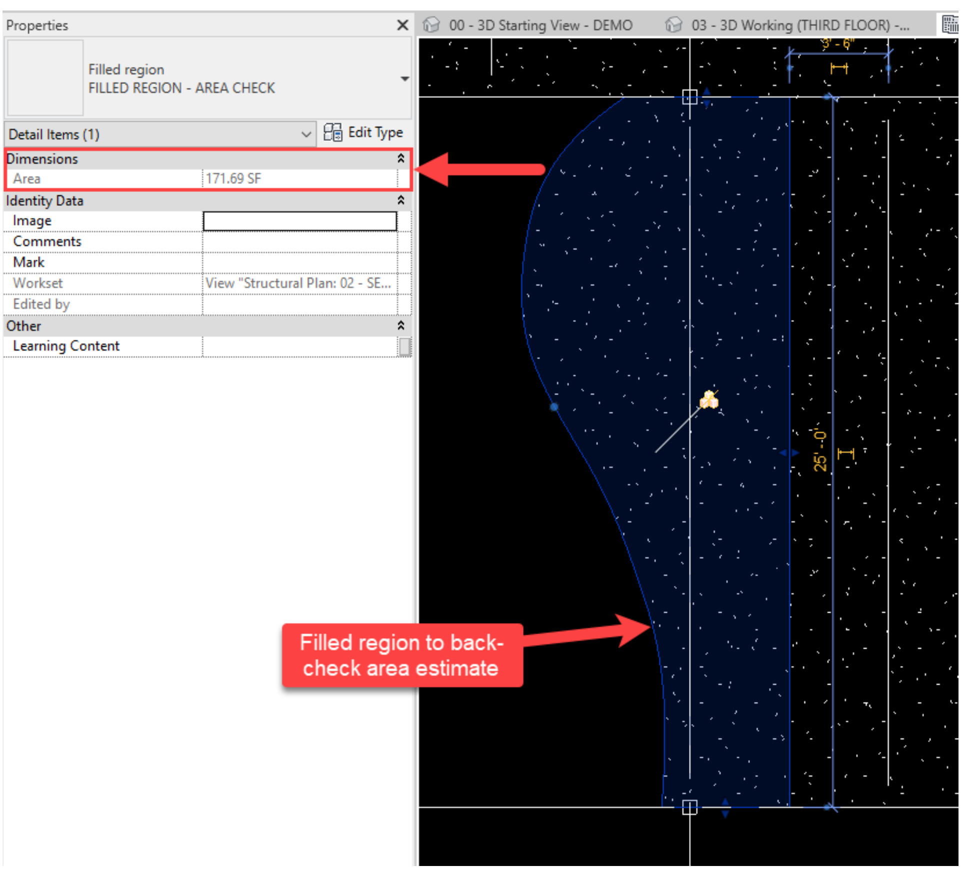
Perimeter beam with a linear varying slab overhang edge:



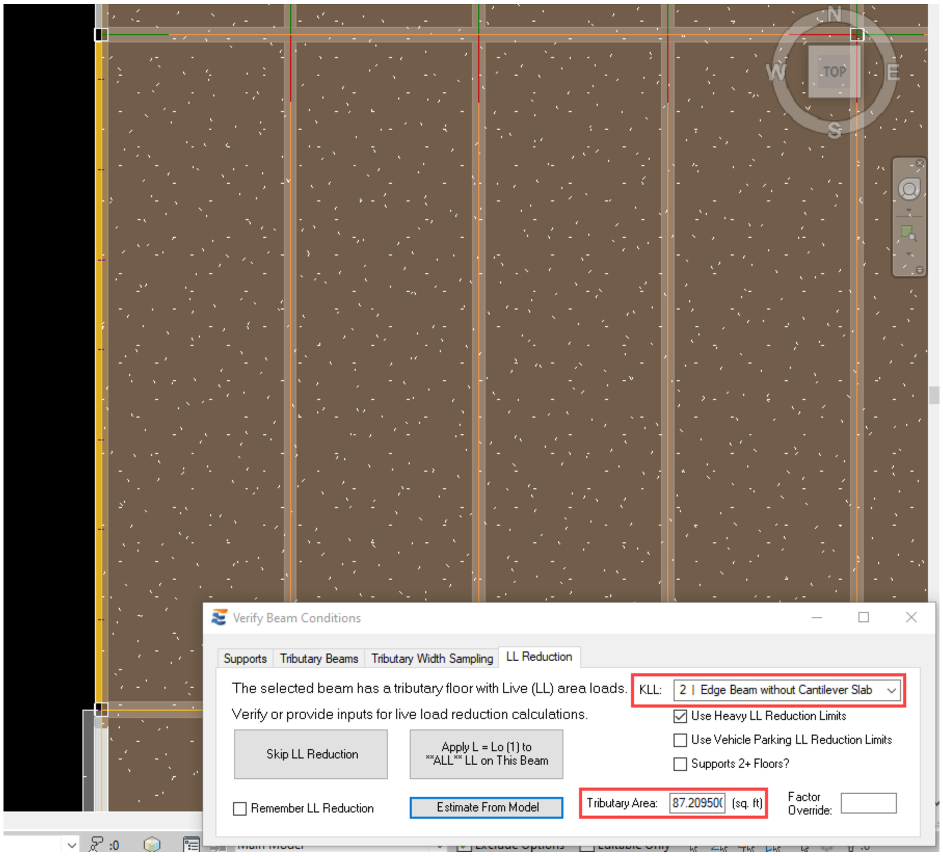


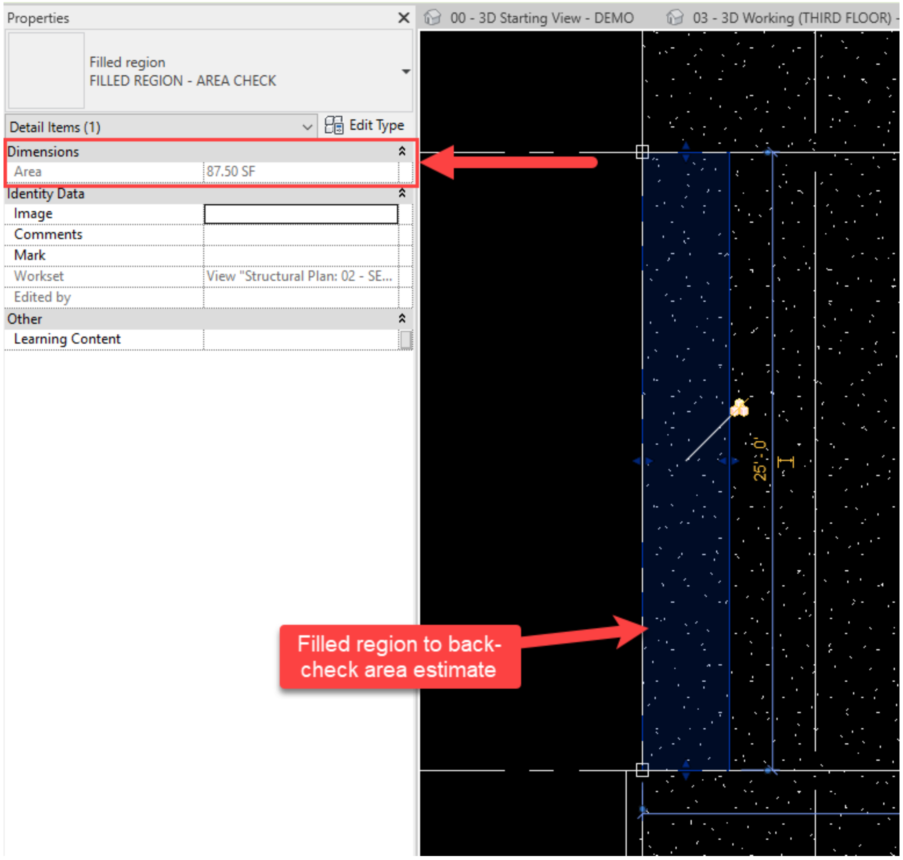
Perimeter with irregular varying overhang edge:


W
TOP
E
S
Verify Beam Conditions
Supports
Tributary Beams
Tributary Width Sampling
LL Reduction
The selected beam has a tributary floor with Live (LL) area loads.
Verify or provide inputs for live load reduction calculations.
KLL:
1 | Edge Beam with Cantilever Slab
Use Heavy LL Reduction Limits
Use Vehicle Parking LL Reduction Limits
Supports 2+ Floors?
Skip LL Reduction
Apply L = Lo (1) to **ALL** LL on This Beam
Remember LL Reduction
Estimate From Model
Tributary Area:
171.6810
(sq. ft)
Factor Override:
Workset1 (Not Editable)

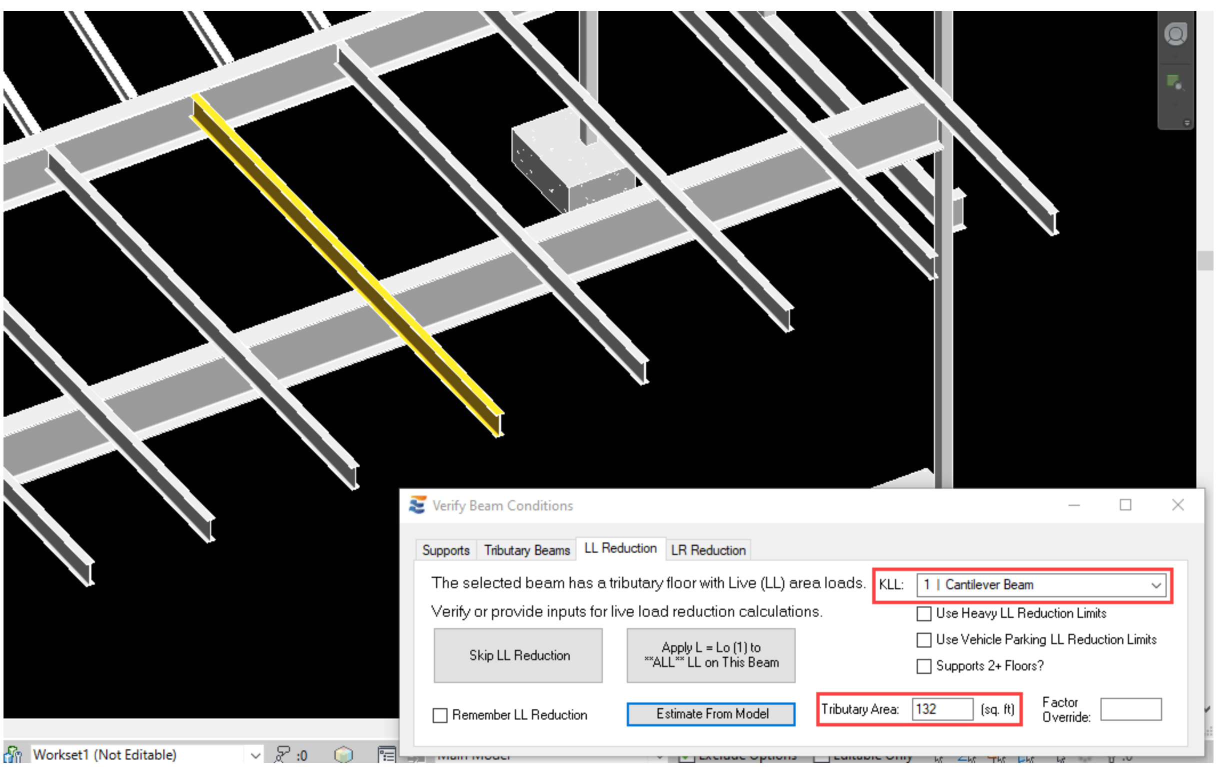

Properties
Filled region
FILLED REGION - AREA CHECK
Detail Items (1)
Edit Type
Dimensions
Area
171.69 SF
Identity Data
Image
Comments
Mark
Workset
View "Structural Plan: 02 - SE...
Edited by
Other
Learning Content
00 - 3D Starting View - DEMO
03 - 3D Working (THIRD FLOOR) -...
25' - 0"
Filled region to back-check area estimate

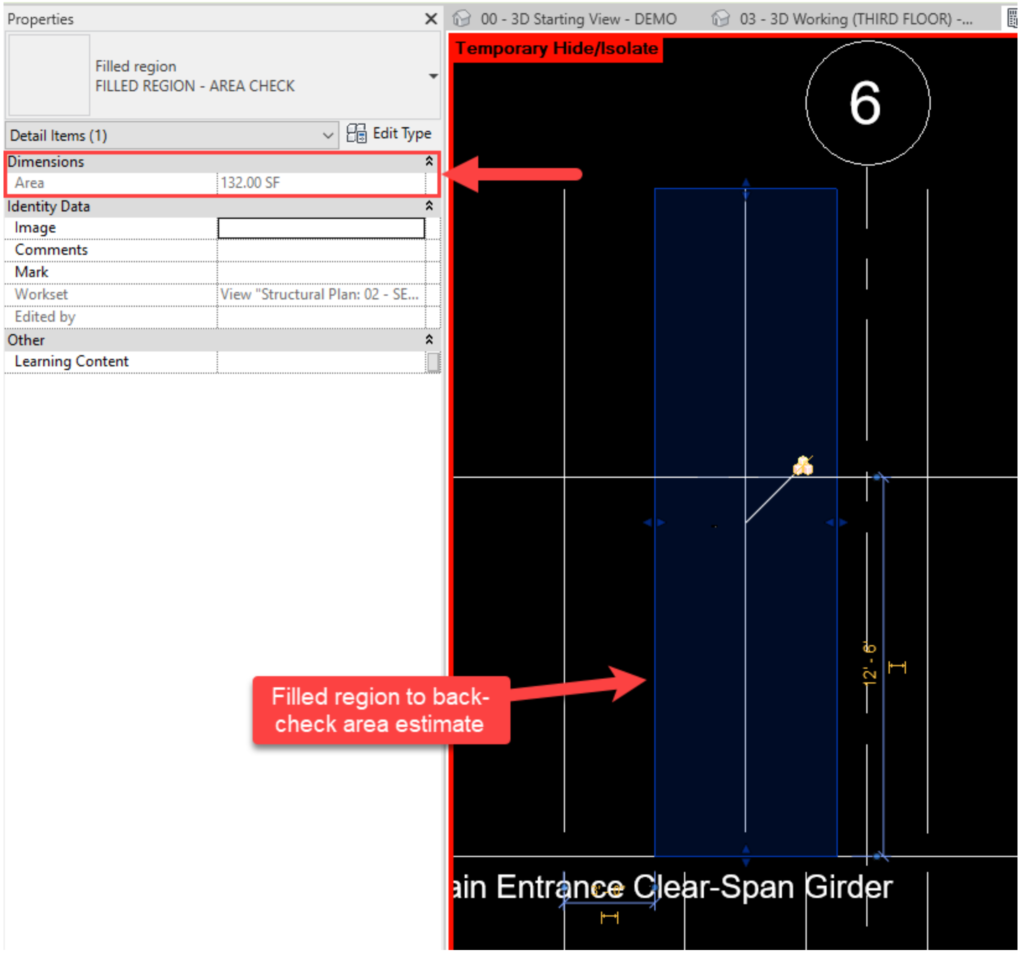
Perimeter beam without an overhang edge:



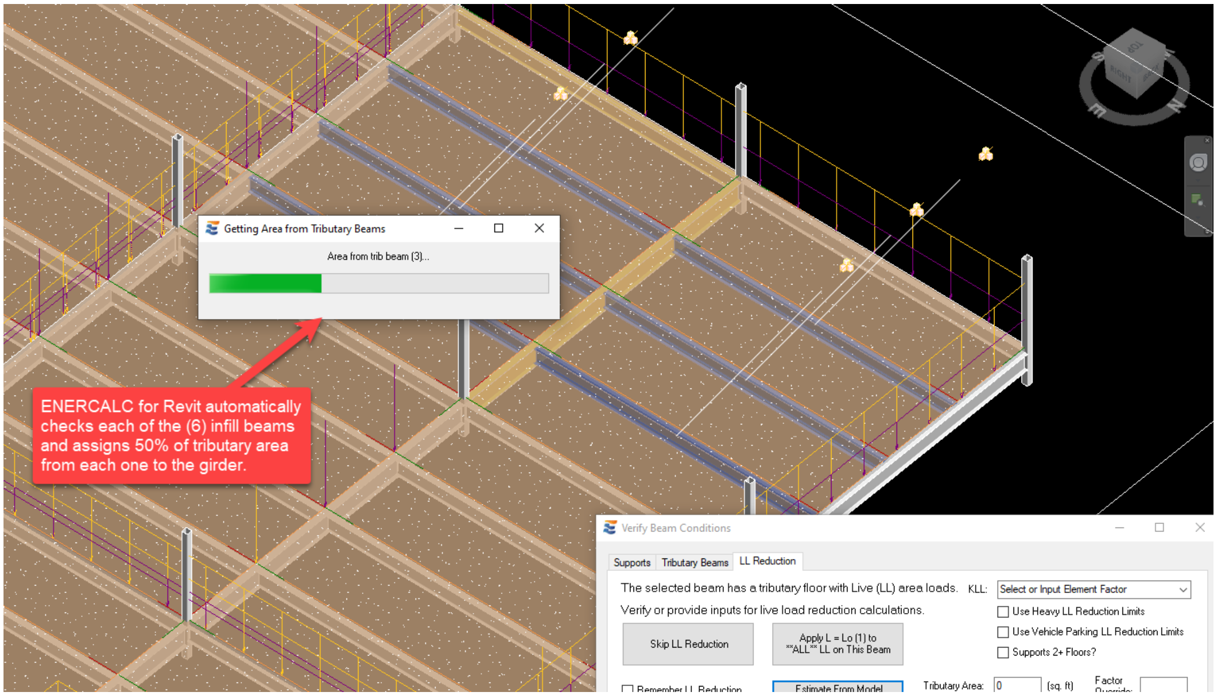


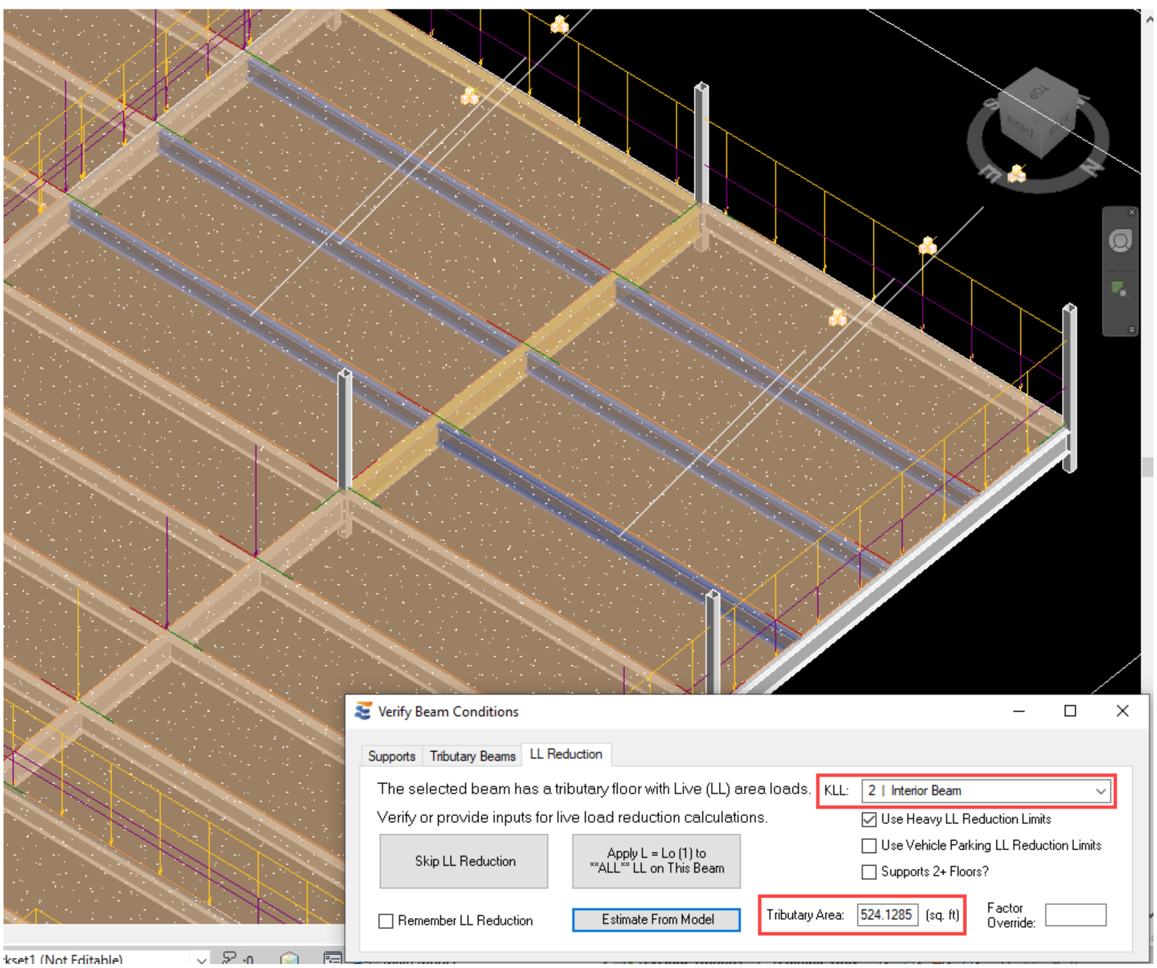
Interior beam (cantilever) without floor or deck:

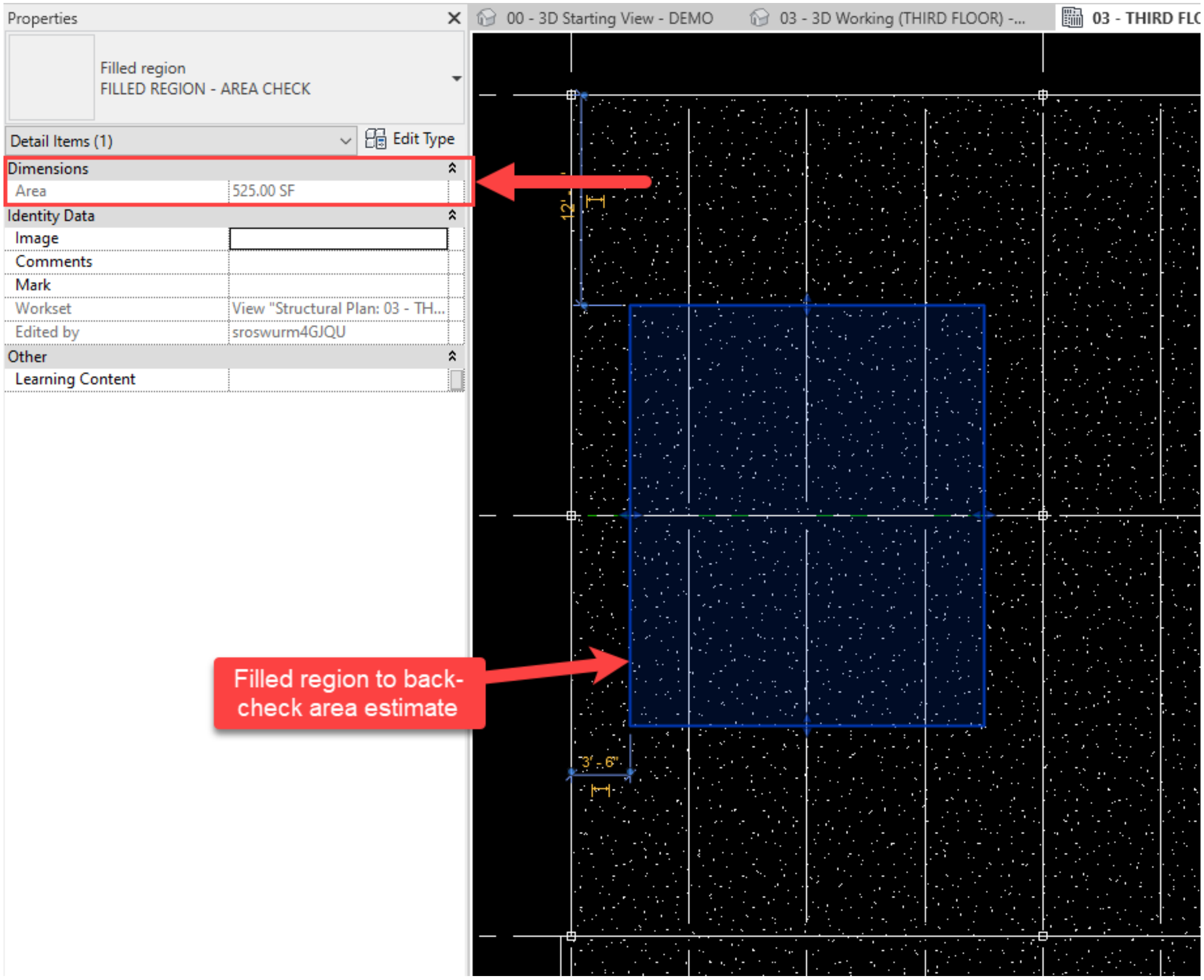




Interior girder supporting infill beams with floor slab:

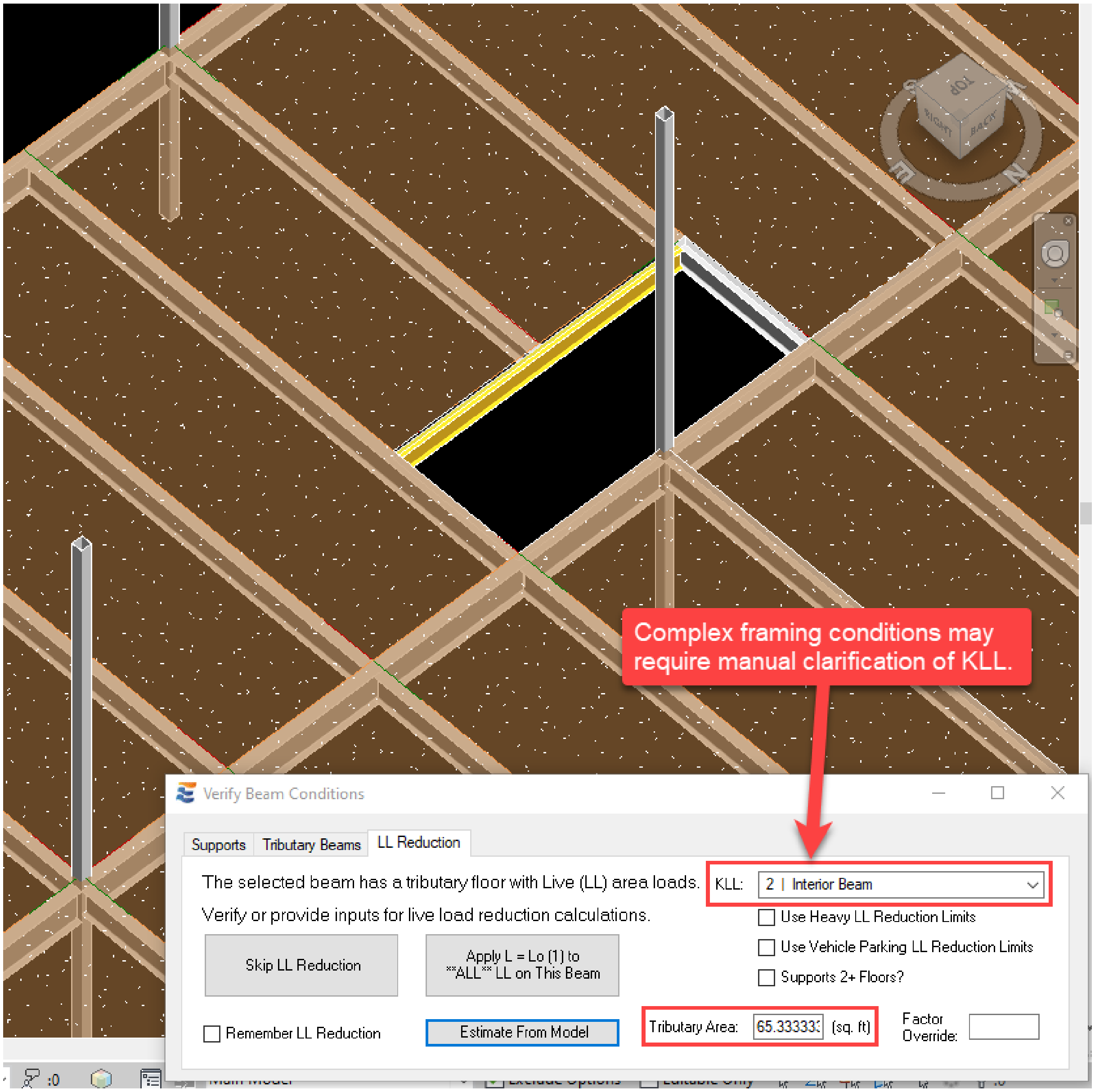

Getting Area from Tributary Beams
Area from trib beam (3)...
ENERCALC for Revit automatically checks each of the (6) infill beams and assigns 50% of tributary area from each one to the girder.
Verify Beam Conditions
Supports
Tributary Beams
LL Reduction
The selected beam has a tributary floor with Live (LL) area loads.
KLL:
Select or Input Element Factor
Verify or provide inputs for live load reduction calculations.
Use Heavy LL Reduction Limits
Use Vehicle Parking LL Reduction Limits
Supports 2+ Floors?
Skip LL Reduction
Apply L = Lo (1) to **ALL** LL on This Beam
Tributary Area: 0 (sq. ft)


Verify Beam Conditions
Supports
Tributary Beams
LL Reduction
The selected beam has a tributary floor with Live (LL) area loads.
KLL:
2 | Interior Beam
Verify or provide inputs for live load reduction calculations.
Use Heavy LL Reduction Limits
Use Vehicle Parking LL Reduction Limits
Supports 2+ Floors?
Skip LL Reduction
Apply L = Lo (1) to **ALL** LL on This Beam
Remember LL Reduction
Estimate From Model
Tributary Area: 524.1285 (sq. ft)
Factor Override:

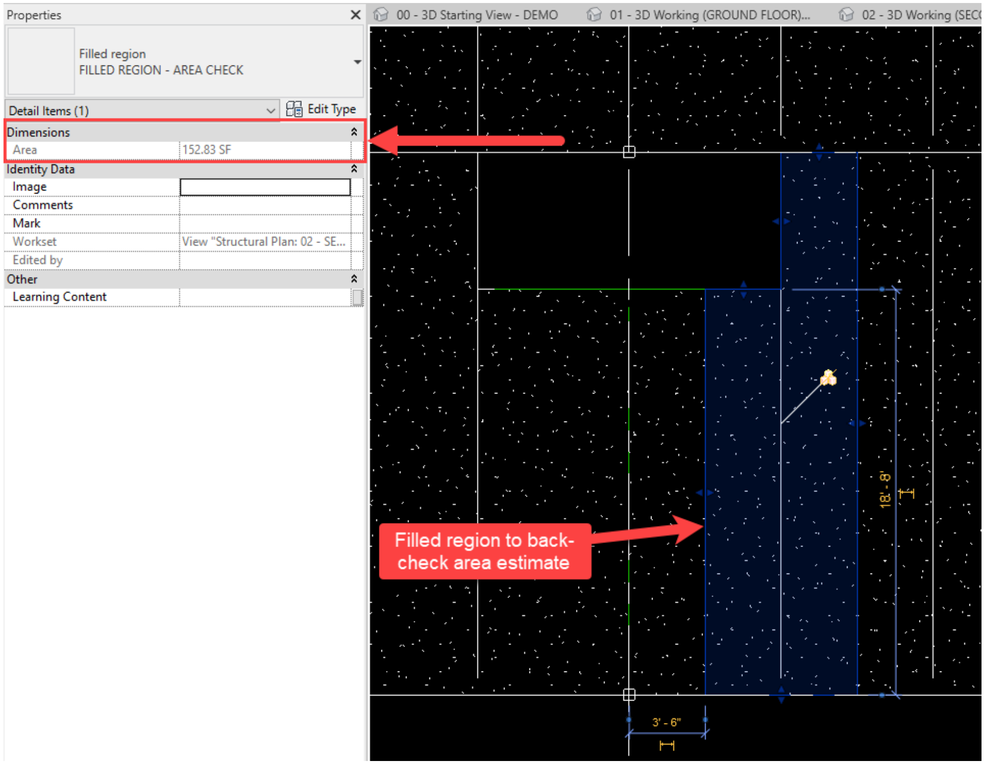


For more complex framing conditions, the example of an elevator shaft opening illustrates the manner in which tributary areas are “inherited” from one element to the next. When the members of this framing system are designed in order of load path, the tributary are from each preceding element is automatically considered when designing the next element.

When designing the girder at the front of the elevator shaft opening, then, the short floor beam is treated as a tributary beam and the girder automatically receives 50% of the tributary area from the beam.

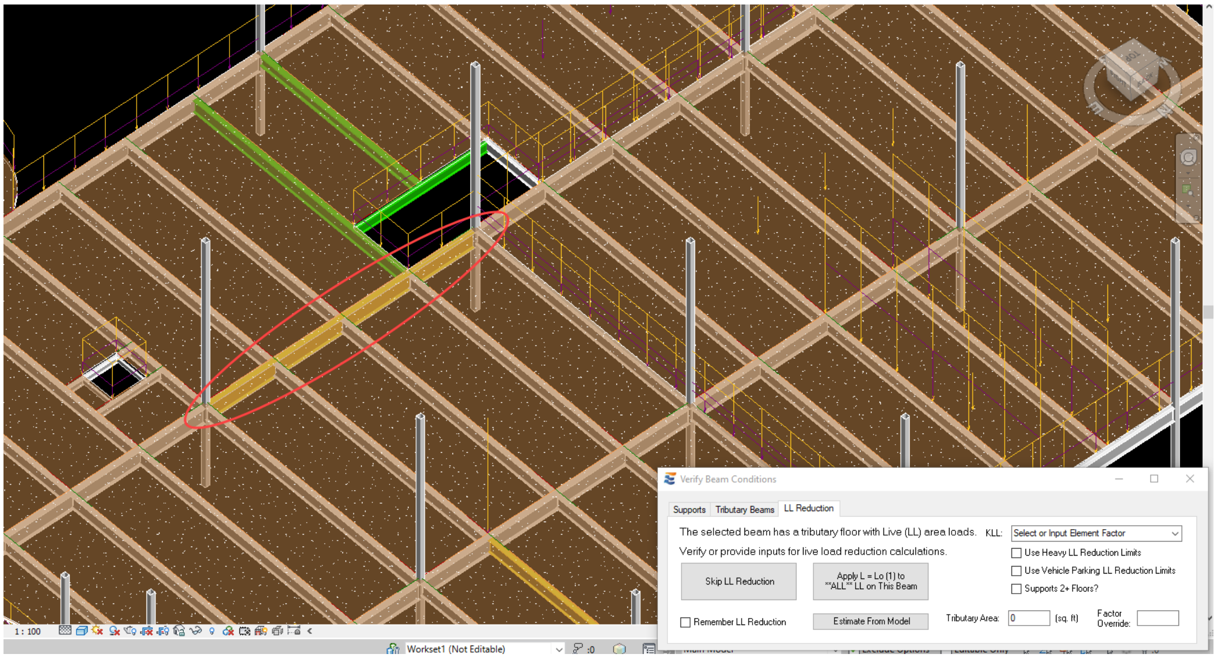


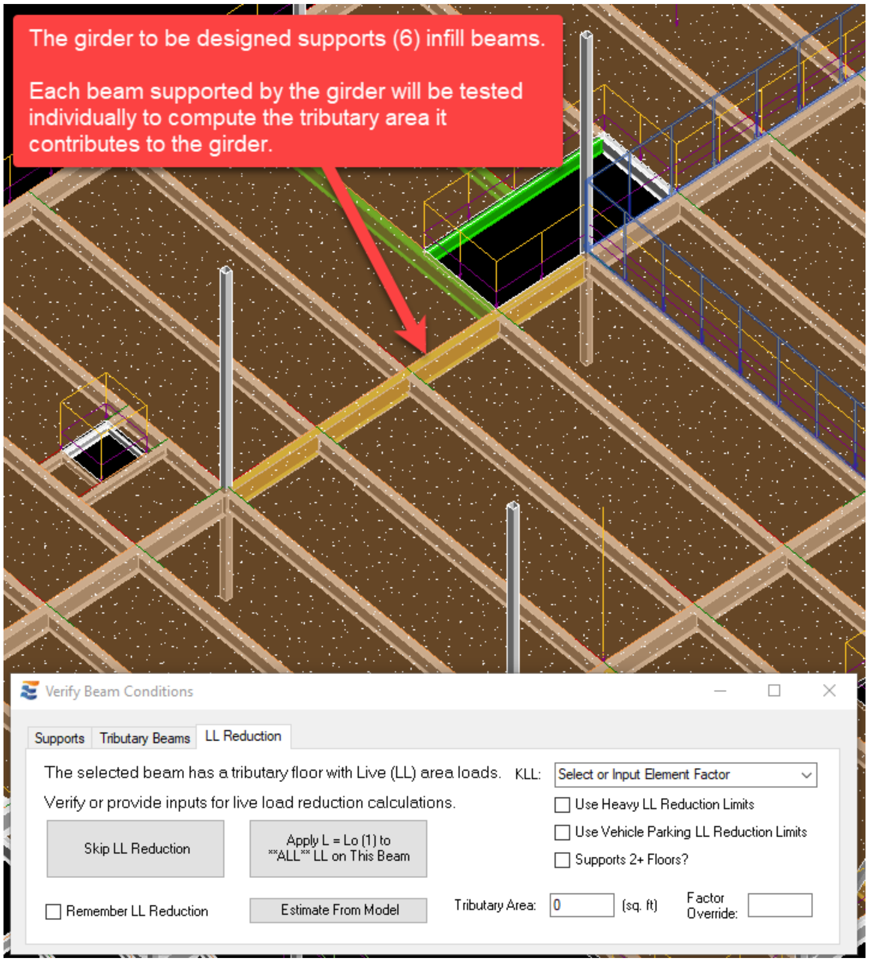
The floor beam at the edge of the opening then carries its own tributary area associated with directly supporting a portion of floor, as well as 50% of the tributary area carried by the short girder shown above. The direct tributary area may be verified with an annotation region.

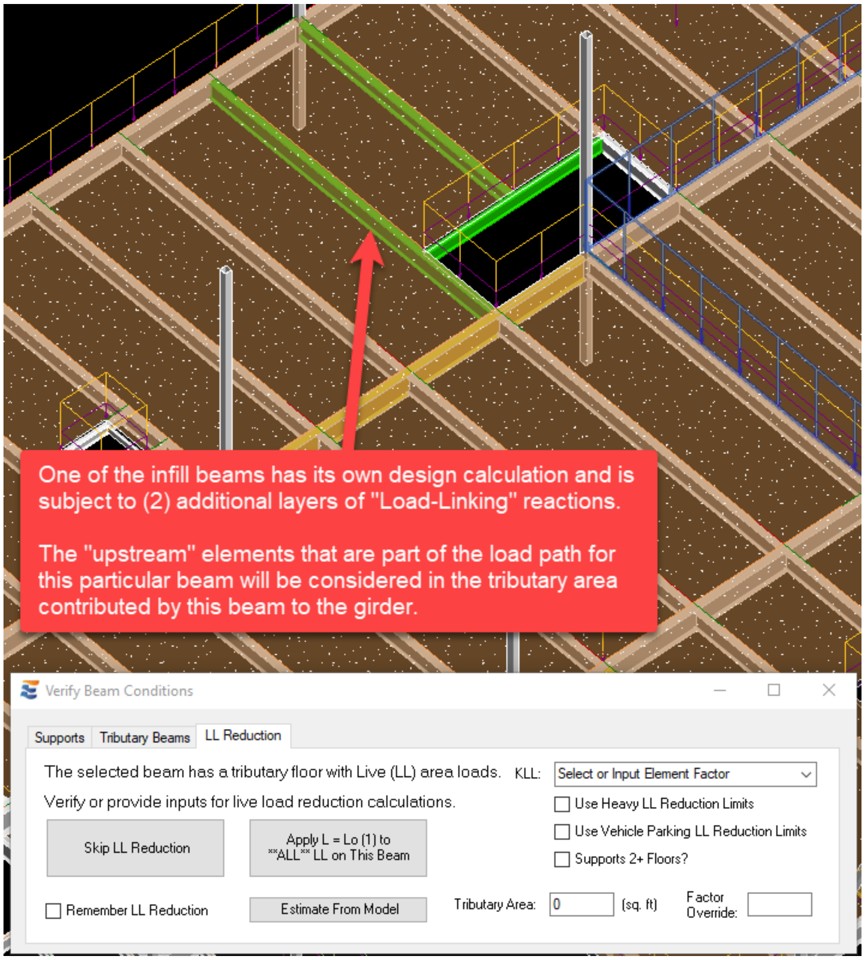


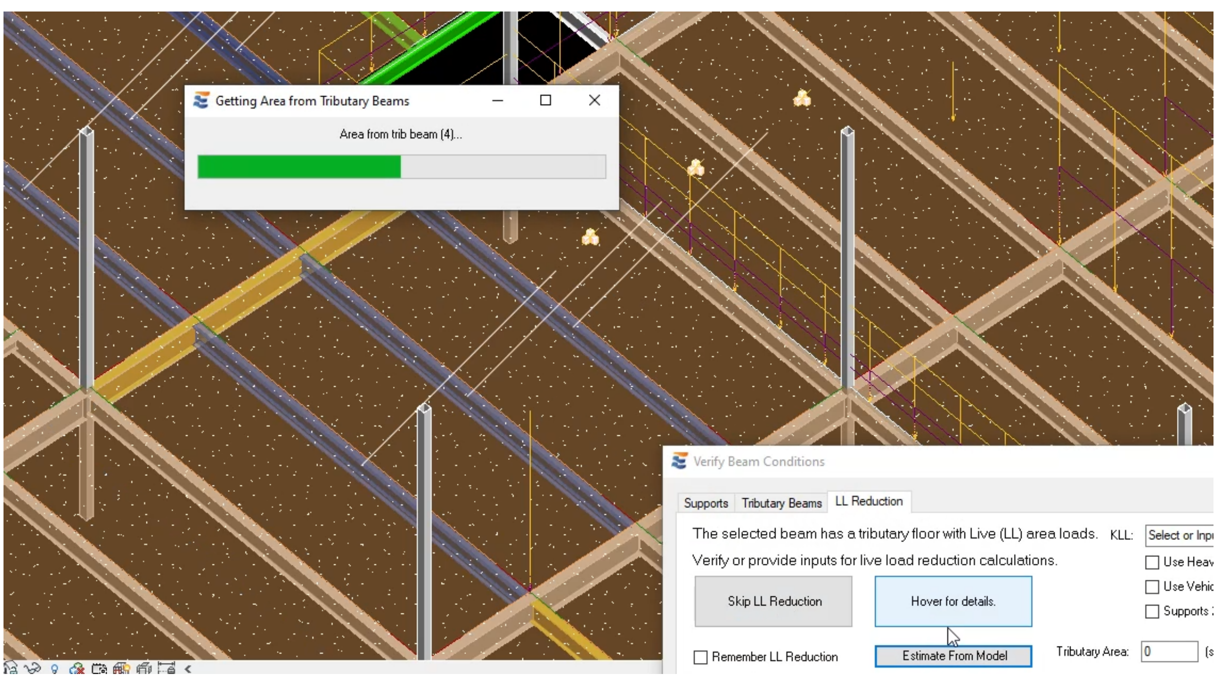
The appropriate tributary area (accurately determined by ENERCALC for Revit) is 153.83 + (65.33 / 2) = 185.5 sq ft.

Estimating the tributary area for the main girder indicated below with then automatically consider the effects of tributary area born by the elevator shaft opening edge beam, as well as the areas of the other 5 infill beams.

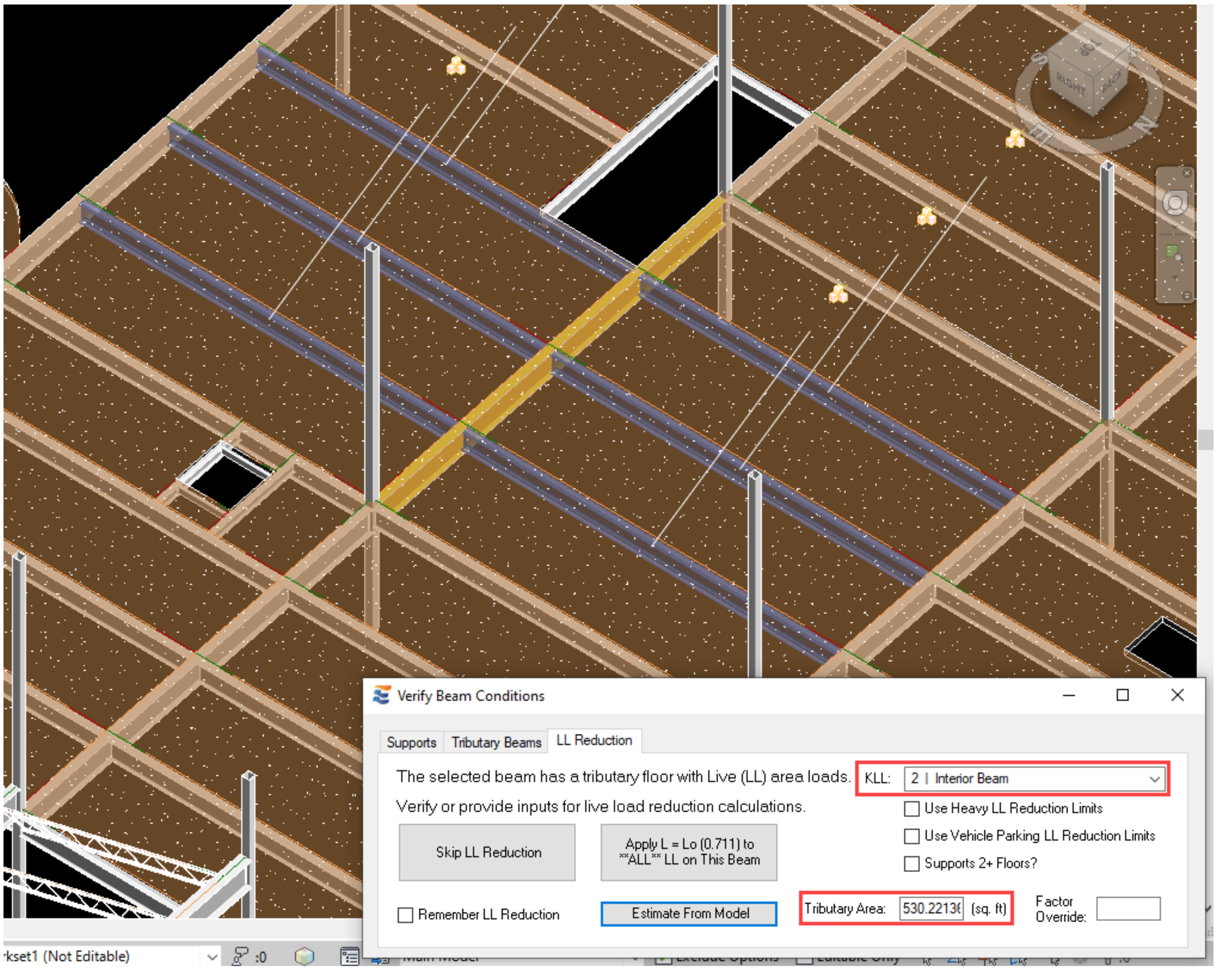



The girder to be designed supports (6) infill beams.
Each beam supported by the girder will be tested individually to compute the tributary area it contributes to the girder.
Verify Beam Conditions
Supports
Tributary Beams
LL Reduction
The selected beam has a tributary floor with Live (LL) area loads.
Verify or provide inputs for live load reduction calculations.
KLL:
Select or Input Element Factor
Use Heavy LL Reduction Limits
Use Vehicle Parking LL Reduction Limits
Supports 2+ Floors?
Skip LL Reduction
Apply L = Lo (1) to **ALL** LL on This Beam
Remember LL Reduction
Estimate From Model
Tributary Area:
0
(sq. ft)
Factor Override:

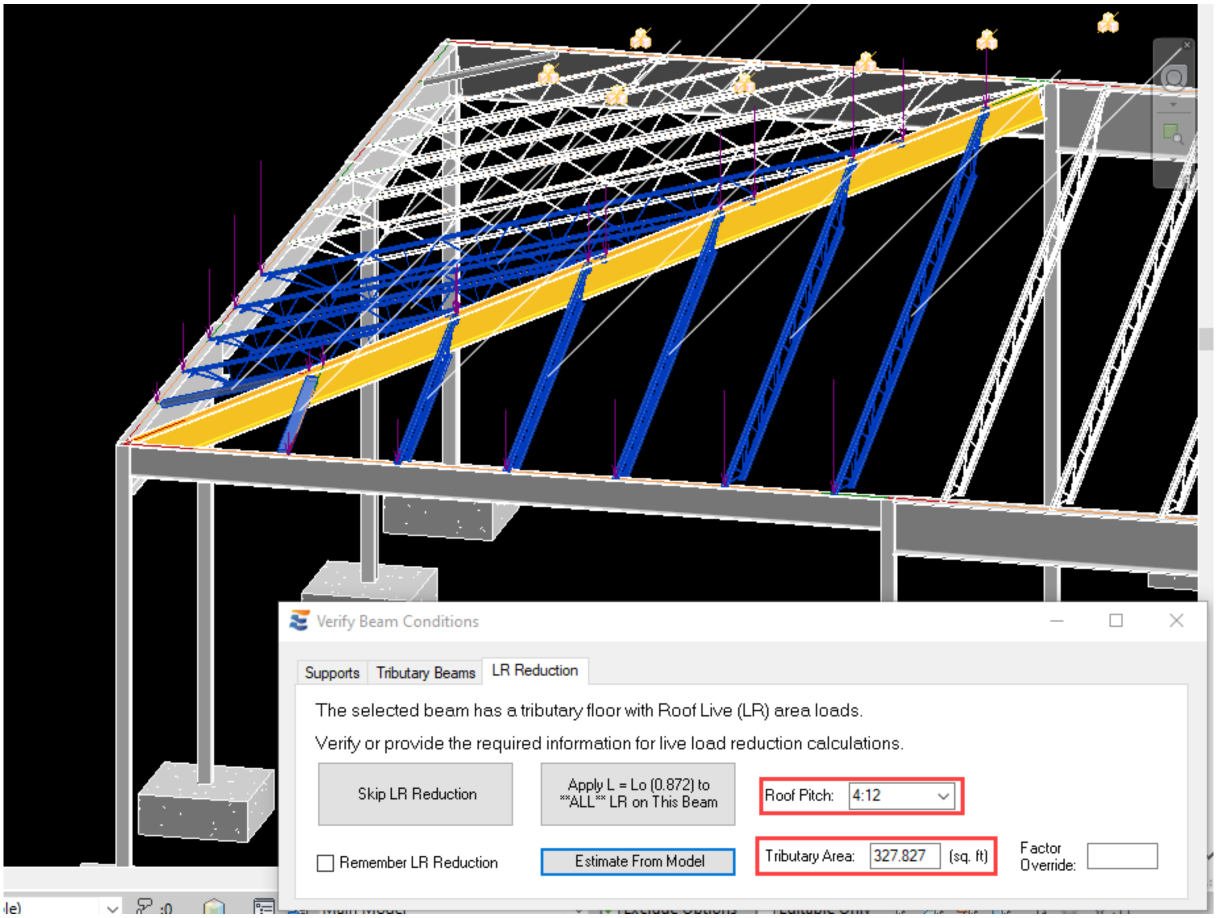

One of the infill beams has its own design calculation and is subject to (2) additional layers of "Load-Linking" reactions.
The "upstream" elements that are part of the load path for this particular beam will be considered in the tributary area contributed by this beam to the girder.
Verify Beam Conditions
Supports
Tributary Beams
LL Reduction
The selected beam has a tributary floor with Live (LL) area loads.
KLL:
Select or Input Element Factor
Verify or provide inputs for live load reduction calculations.
Use Heavy LL Reduction Limits
Use Vehicle Parking LL Reduction Limits
Supports 2+ Floors?
Skip LL Reduction
Apply L = Lo (1) to **ALL** LL on This Beam
Remember LL Reduction
Estimate From Model
Tributary Area:
0
(sq. ft)
Factor Override:

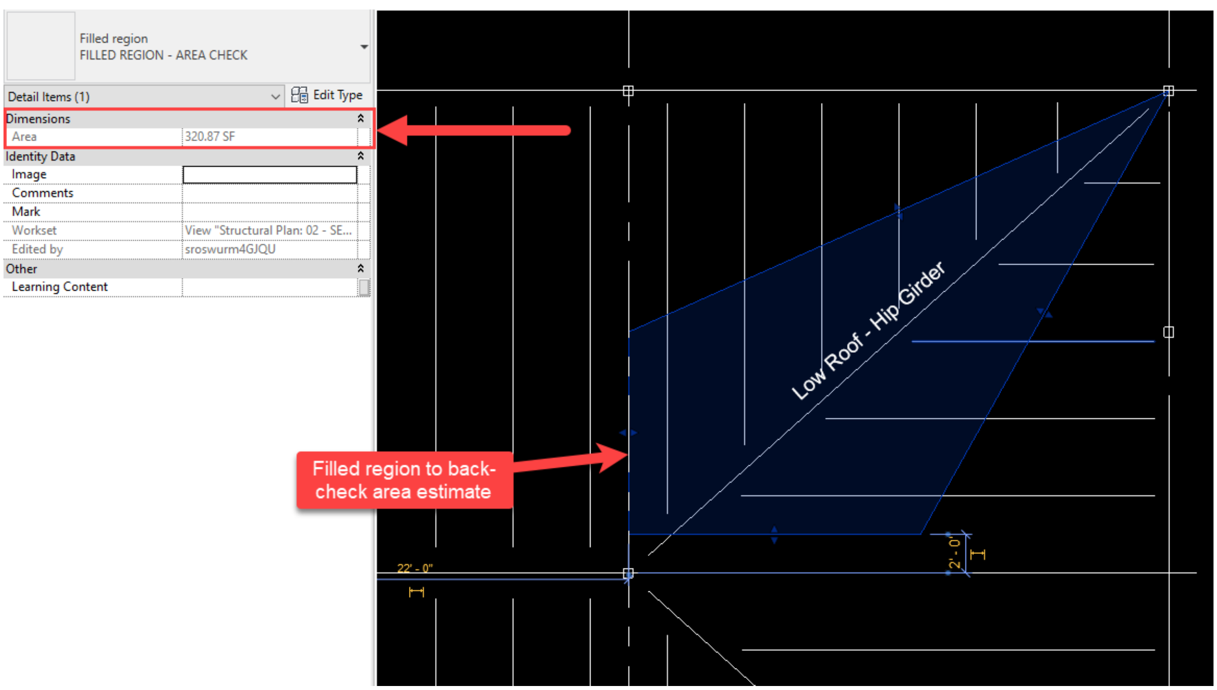

Getting Area from Tributary Beams
Area from trib beam (4)...
Verify Beam Conditions
Supports
Tributary Beams
LL Reduction
The selected beam has a tributary floor with Live (LL) area loads.
KLL:
Verify or provide inputs for live load reduction calculations.
Skip LL Reduction
Hover for details.
Remember LL Reduction
Estimate From Model
Tributary Area:
0

The anticipated total tributary area for the girder would be 5(87.5) + (185.5 / 2) = 530.25 sq ft.



Interior girder supporting roof joists with no slab or deck:

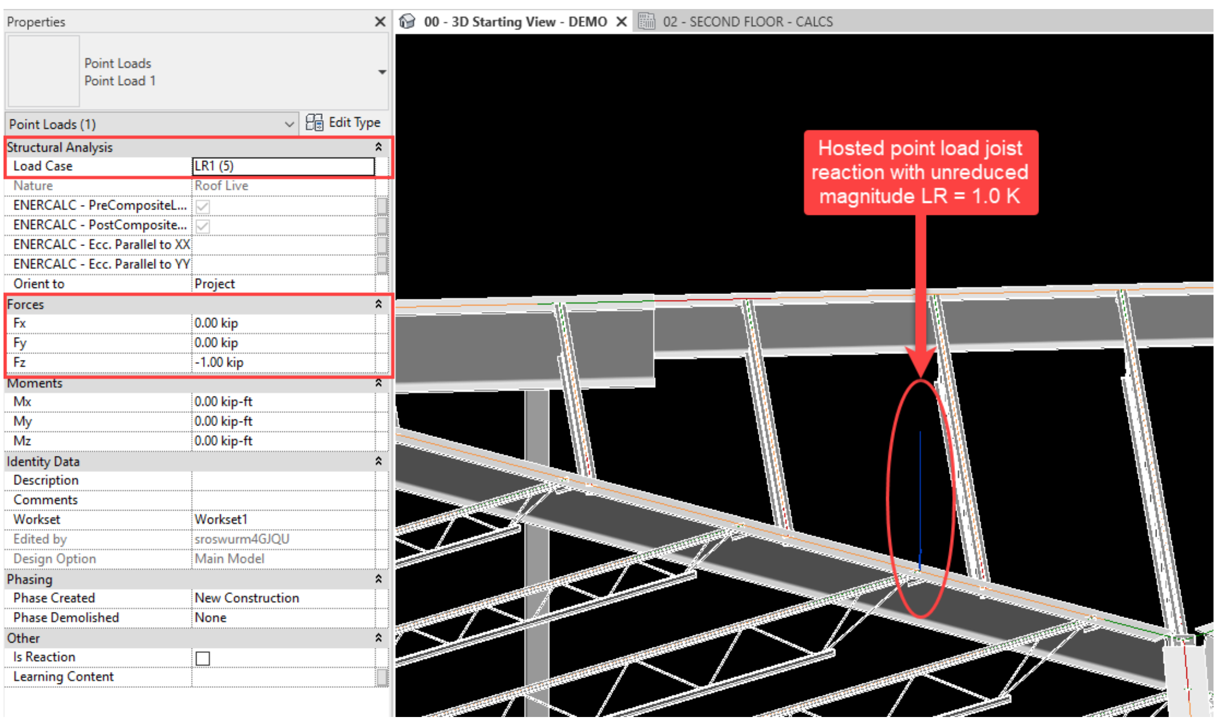

Verify Beam Conditions
Supports
Tributary Beams
LR Reduction
The selected beam has a tributary floor with Roof Live (LR) area loads.
Verify or provide the required information for live load reduction calculations.
Skip LR Reduction
Apply L = Lo (0.872) to **ALL** LR on This Beam
Roof Pitch: 4:12
Remember LR Reduction
Estimate From Model
Tributary Area: 327.827 (sq. ft)
Factor Override:

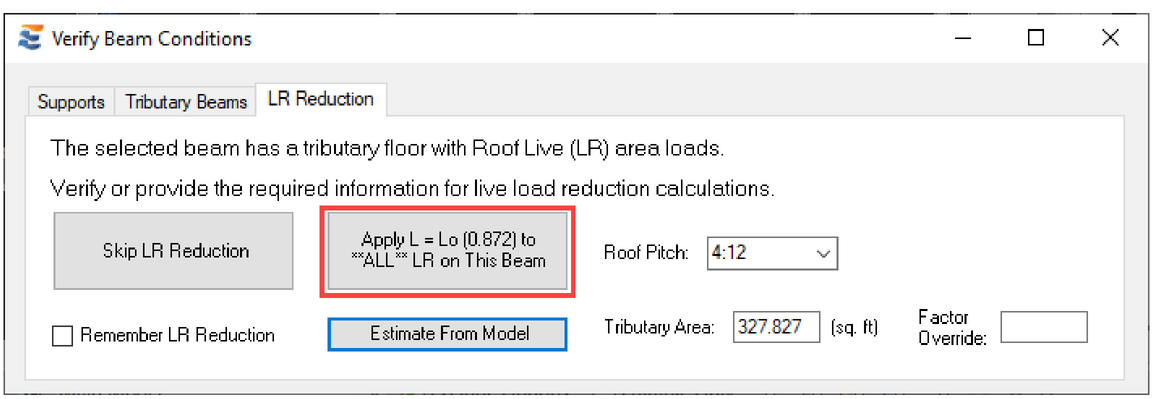

Filled region
FILLED REGION - AREA CHECK
Detail Items (1)
Edit Type
Dimensions
Area 320.87 SF
Identity Data
Image
Comments
Mark
Workset View "Structural Plan: 02 - SE...
Edited by sroswurm4GJQU
Other
Learning Content
Low Roof - Hip Girder
Filled region to back-check area estimate
22' - 0"
2' - 0"

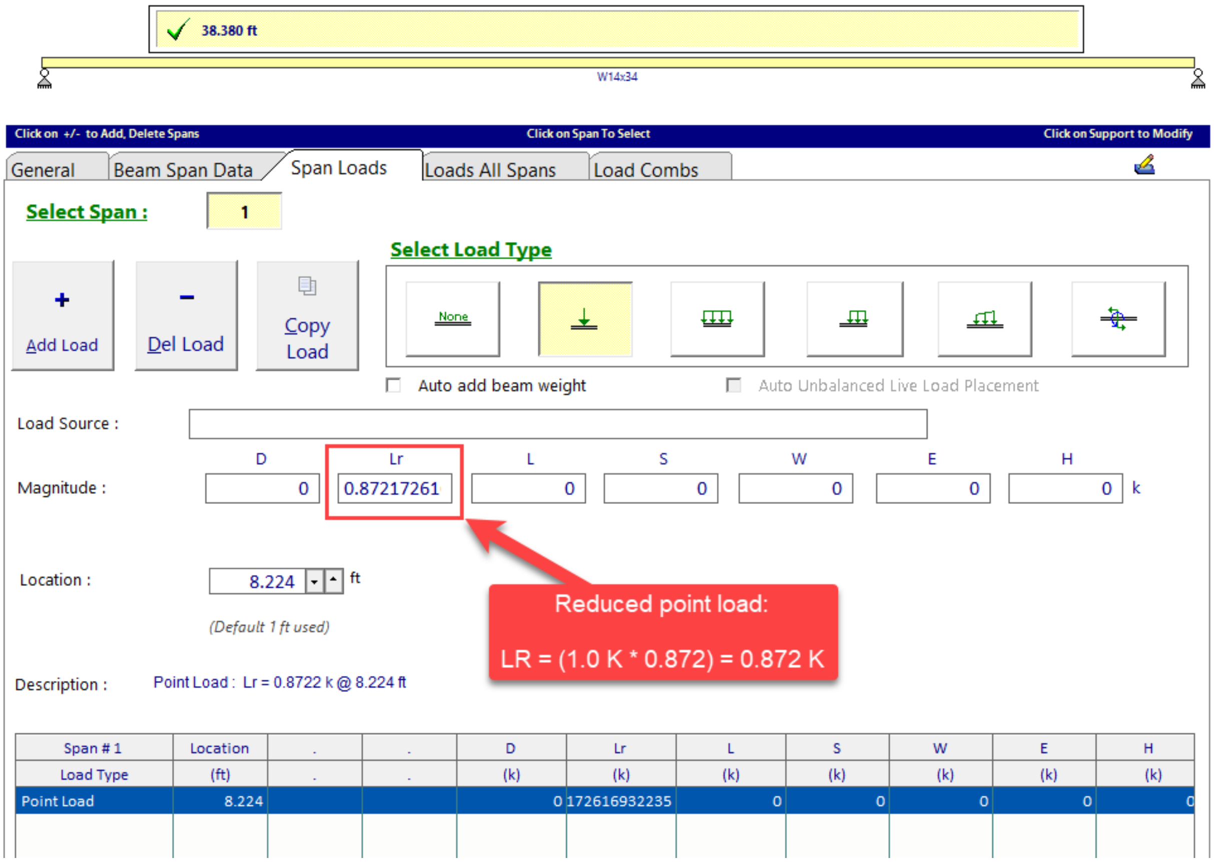
10.4.12.6 Reduced Loads in SEL – Point Loads

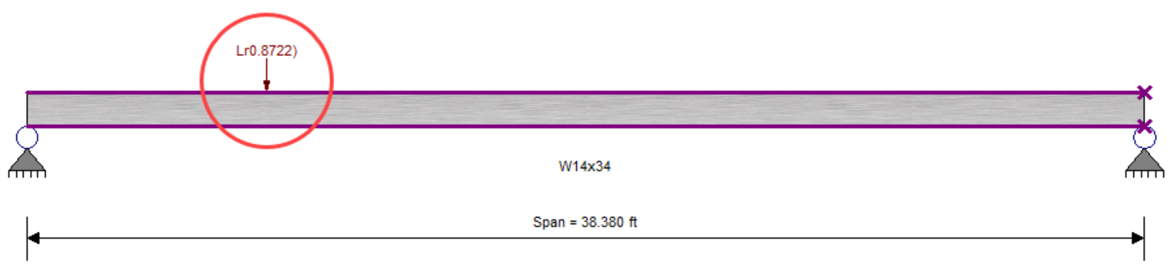
Once LL and LR reduction factors have been set and the beam calculation is launched, the factors will be applied to all applicable loads found on the beam. These loads will populate in the ENERCALC SEL interface with reduced magnitudes.

NOTE: Reduction factors will be applied to **ALL** LL or LR loads present on the beam. This reduction is applied uniformly to all three Revit load types (point, linear, and area), as well as "Load-Linked" reaction forces. Reduction is applied regardless of whether the forces are populated in the "Span Loads" or "Loads All Spans" tabs. ***The design professional is responsible for verifying that reduction of all loads is appropriate for the design.***





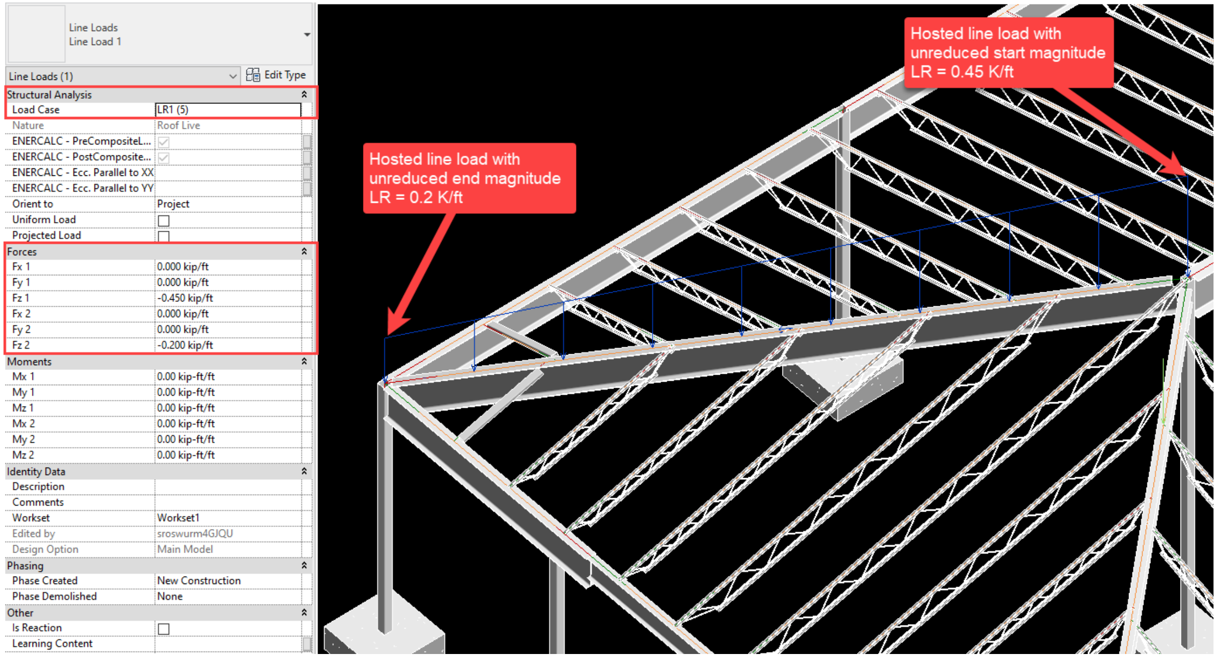


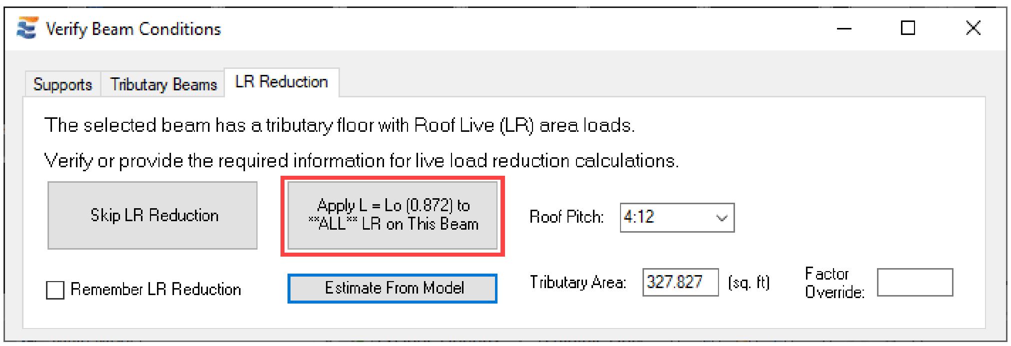


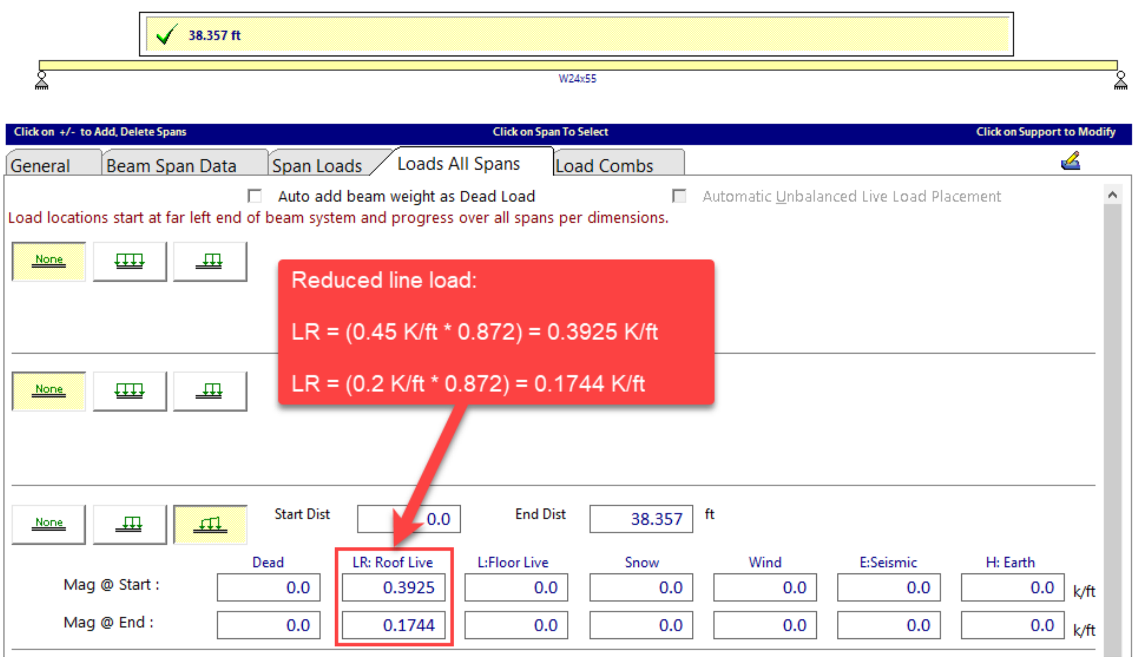
10.4.12.7 Reduced Loads in SEL – Line Loads

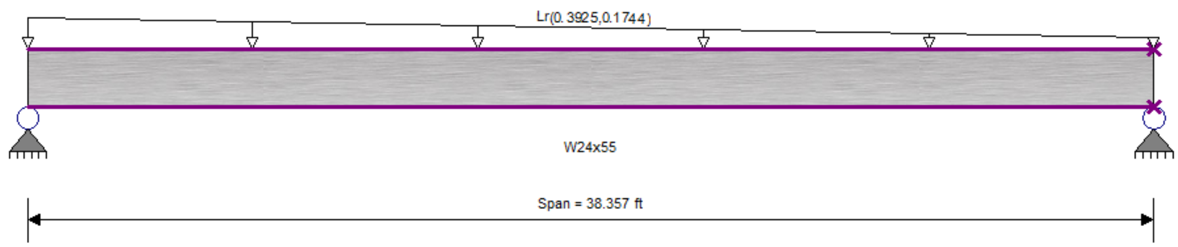
Once LL and LR reduction factors have been set and the beam calculation is launched, the factors will be applied to all applicable loads found on the beam. These loads will populate in the ENERCALC SEL interface with reduced magnitudes.

NOTE: Reduction factors will be applied to **ALL** LL or LR loads present on the beam. This reduction is applied uniformly to all three Revit load types (point, linear, and area), as well as “Load-Linked” reaction forces. Reduction is applied regardless of whether the forces are populated in the “Span Loads” or “Loads All Spans” tabs. ***The design professional is responsible for verifying that reduction of all loads is appropriate for the design.***


Line Loads
Line Load 1
Line Loads (1)
Edit Type
Structural Analysis
Load Case LR1 (5)
Nature Roof Live
ENERCALC - PreCompositeL...
ENERCALC - PostComposite...
ENERCALC - Ecc. Parallel to XX
ENERCALC - Ecc. Parallel to YY
Orient to Project
Uniform Load
Projected Load
Forces
Fx 1 0.000 kip/ft
Fy 1 0.000 kip/ft
Fz 1 -0.450 kip/ft
Fx 2 0.000 kip/ft
Fy 2 0.000 kip/ft
Fz 2 -0.200 kip/ft
Moments
Mx 1 0.00 kip-ft/ft
My 1 0.00 kip-ft/ft
Mz 1 0.00 kip-ft/ft
Mx 2 0.00 kip-ft/ft
My 2 0.00 kip-ft/ft
Mz 2 0.00 kip-ft/ft
Identity Data
Description
Comments
Workset Workset1
Edited by sroswurm4GJQU
Design Option Main Model
Phasing
Phase Created New Construction
Phase Demolished None
Other
Is Reaction
Learning Content
Hosted line load with unreduced end magnitude LR = 0.2 K/ft
Hosted line load with unreduced start magnitude LR = 0.45 K/ft


Verify Beam Conditions
Supports
Tributary Beams
LR Reduction
The selected beam has a tributary floor with Roof Live (LR) area loads.
Verify or provide the required information for live load reduction calculations.
Skip LR Reduction
Apply L = Lo (0.872) to **ALL** LR on This Beam
Roof Pitch: 4:12
Remember LR Reduction
Estimate From Model
Tributary Area: 327.827 (sq. ft)
Factor Override:

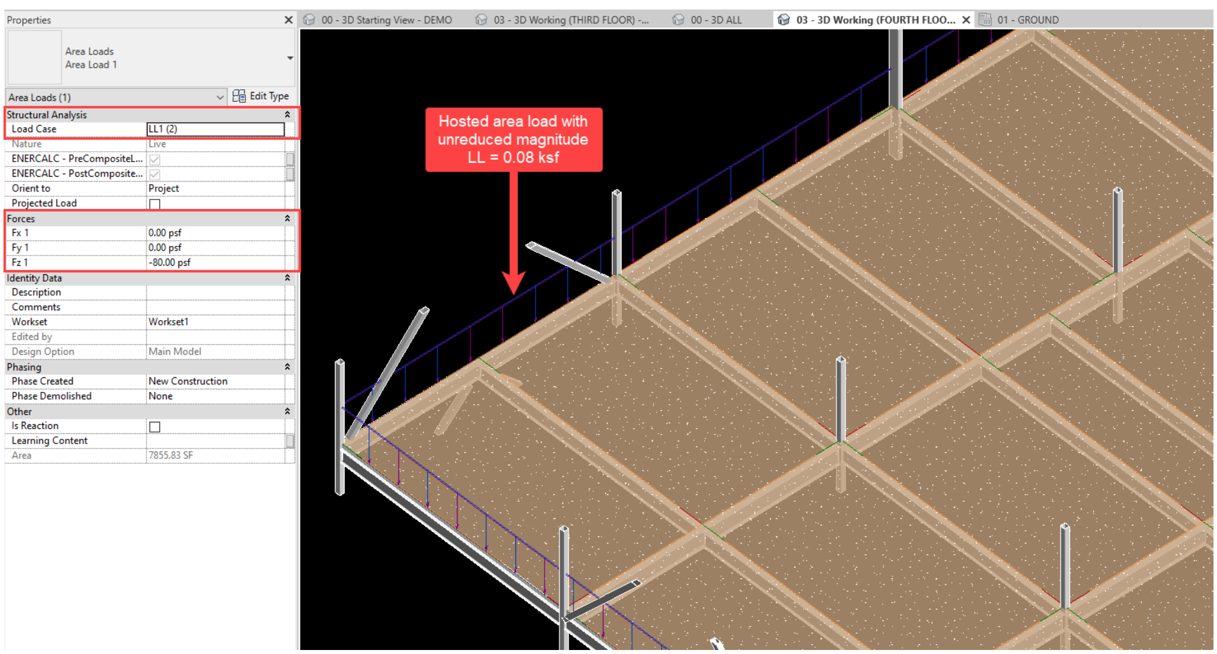

38.357 ft
W24x55
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Auto add beam weight as Dead Load
Automatic Unbalanced Live Load Placement
Load locations start at far left end of beam system and progress over all spans per dimensions.
None
Reduced line load:
LR = (0.45 K/ft * 0.872) = 0.3925 K/ft
LR = (0.2 K/ft * 0.872) = 0.1744 K/ft
Start Dist 0.0
End Dist 38.357 ft
Dead
LR: Roof Live
L:Floor Live
Snow
Wind
E:Seismic
H: Earth
Mag @ Start : 0.0 0.3925 0.0 0.0 0.0 0.0 0.0 k/ft
Mag @ End : 0.0 0.1744 0.0 0.0 0.0 0.0 0.0 k/ft

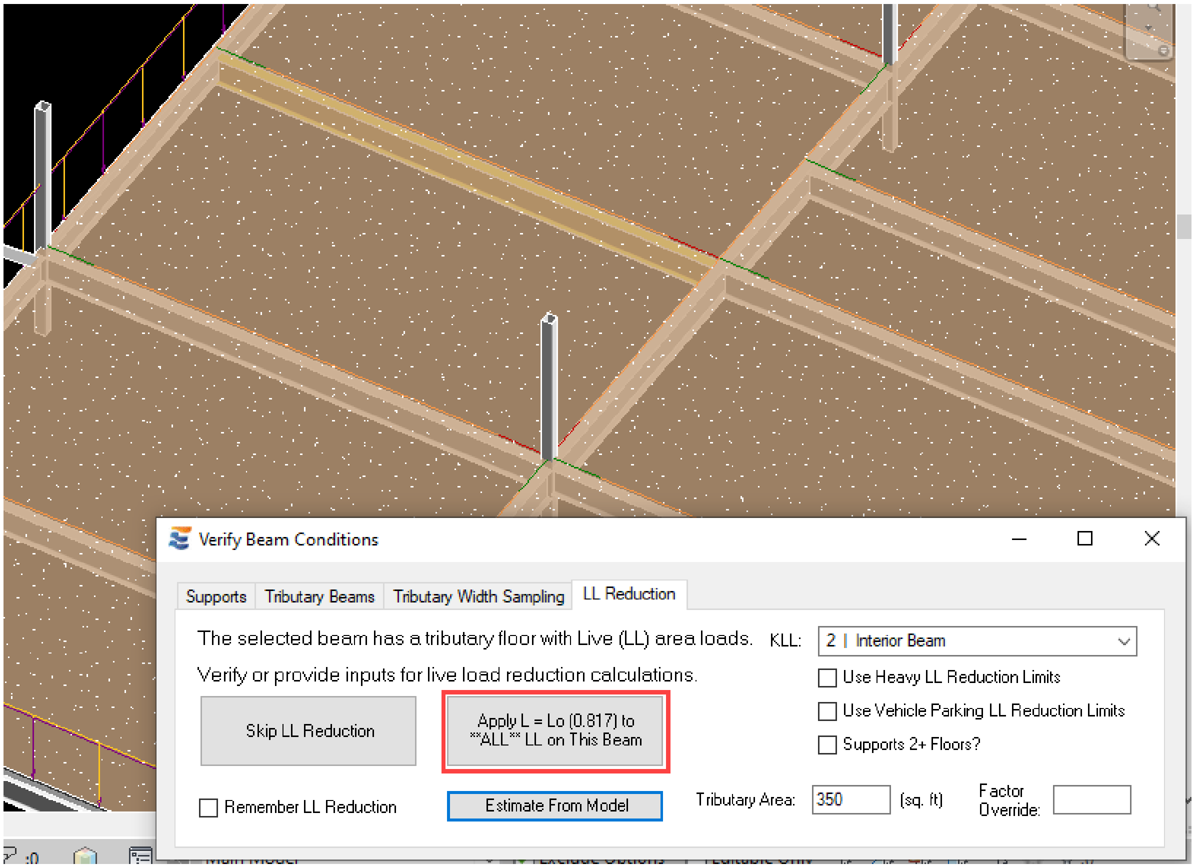


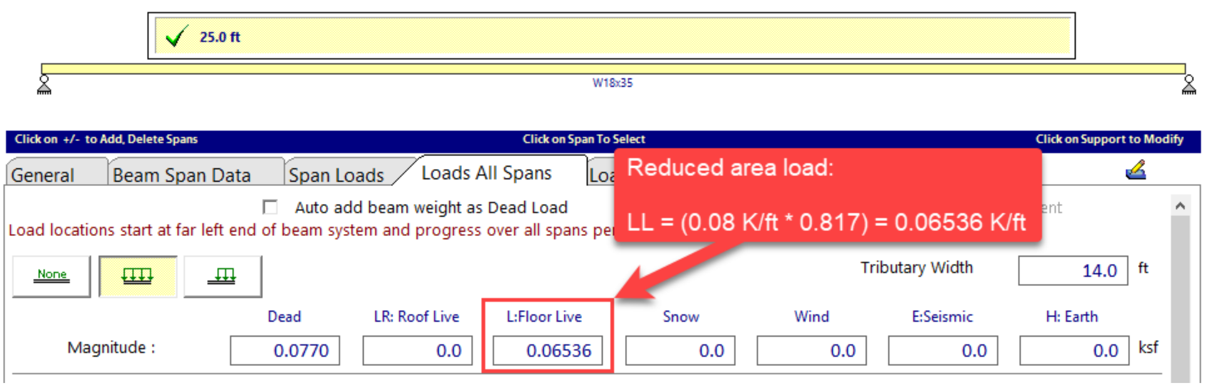
10.4.12.8 Reduced Loads in SEL – Area Loads

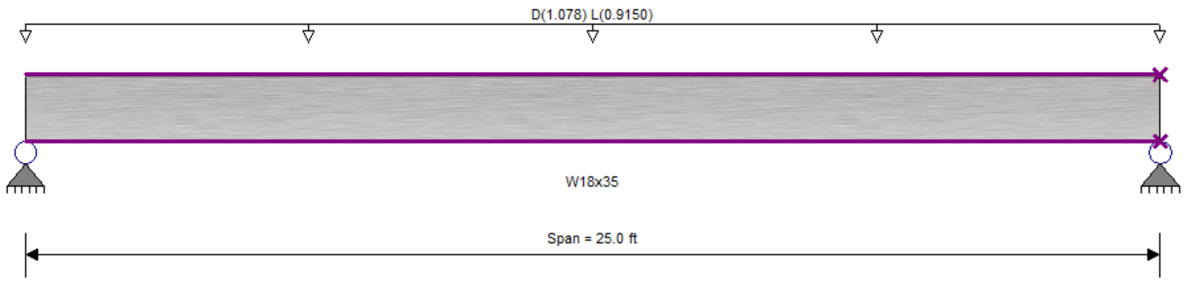
Once LL and LR reduction factors have been set and the beam calculation is launched, the factors will be applied to all applicable loads found on the beam. These loads will populate in the ENERCALC SEL interface with reduced magnitudes.

NOTE: Reduction factors will be applied to **ALL** LL or LR loads present on the beam. This reduction is applied uniformly to all three Revit load types (point, linear, and area), as well as "Load-Linked" reaction forces. Reduction is applied regardless of whether the forces are populated in the "Span Loads" or "Loads All Spans" tabs. ***The design professional is responsible for verifying that reduction of all loads is appropriate for the design.***




Verify Beam Conditions
Supports
Tributary Beams
Tributary Width Sampling
LL Reduction
The selected beam has a tributary floor with Live (LL) area loads.
Verify or provide inputs for live load reduction calculations.
KLL:
2 | Interior Beam
Use Heavy LL Reduction Limits
Use Vehicle Parking LL Reduction Limits
Supports 2+ Floors?
Skip LL Reduction
Apply L = Lo (0.817) to **ALL** LL on This Beam
Remember LL Reduction
Estimate From Model
Tributary Area:
350
(sq. ft)
Factor Override:


25.0 ft
W18x35
Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
General
Beam Span Data
Span Loads
Loads All Spans
Reduced area load:
LL = (0.08 K/ft * 0.817) = 0.06536 K/ft
Auto add beam weight as Dead Load
Load locations start at far left end of beam system and progress over all spans pe
None
Tributary Width
14.0
ft
Dead
LR: Roof Live
L:Floor Live
Snow
Wind
E:Seismic
H: Earth
Magnitude :
0.0770
0.0
0.06536
0.0
0.0
0.0
0.0
ksf



10.4.12.9 Reduced Loads in SEL – Linked Reactions

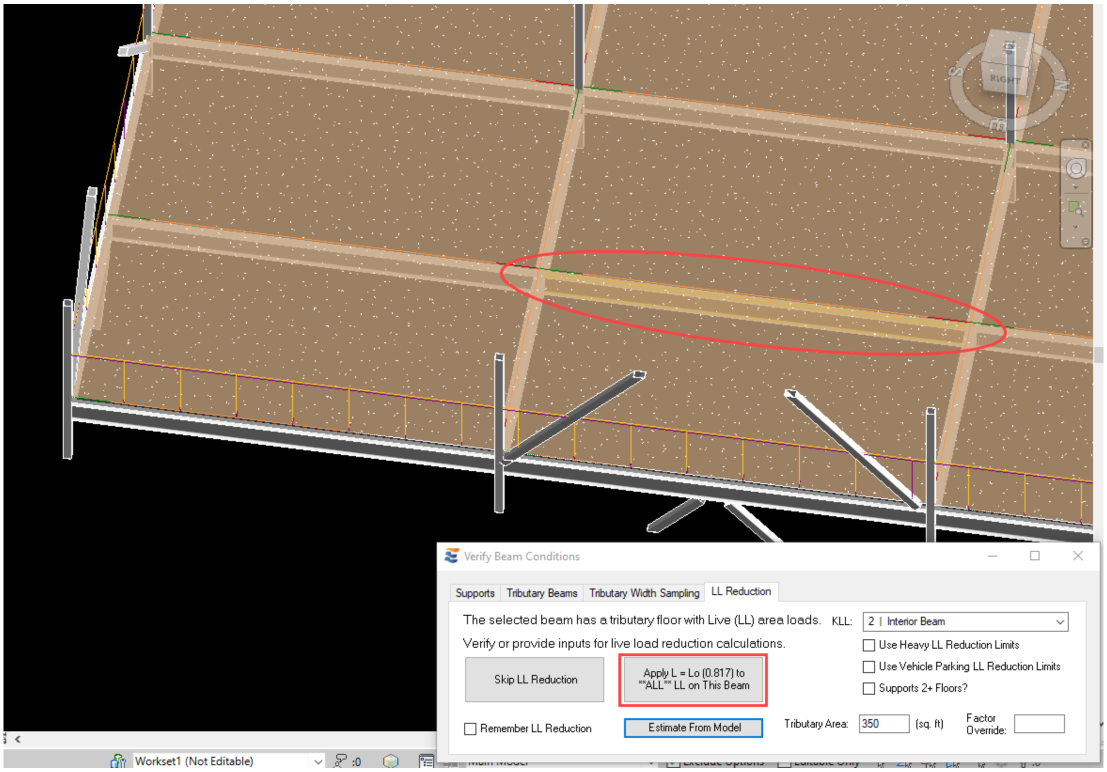
Once LL and LR reduction factors have been set and the beam calculation is launched, the factors will be applied to all applicable loads found on the beam. These loads will populate in the ENERCALC SEL interface with reduced magnitudes.

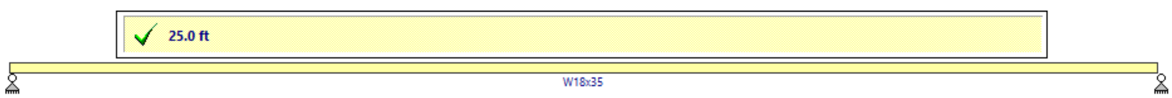
NOTE: Reduction factors will be applied to **ALL** LL or LR loads present on the beam. This reduction is applied uniformly to all three Revit load types (point, linear, and area), as well as “Load-Linked” reaction forces. Reduction is applied regardless of whether the forces are populated in the “Span Loads” or “Loads All Spans” tabs. ***The design professional is responsible for verifying that reduction of all loads is appropriate for the design.***

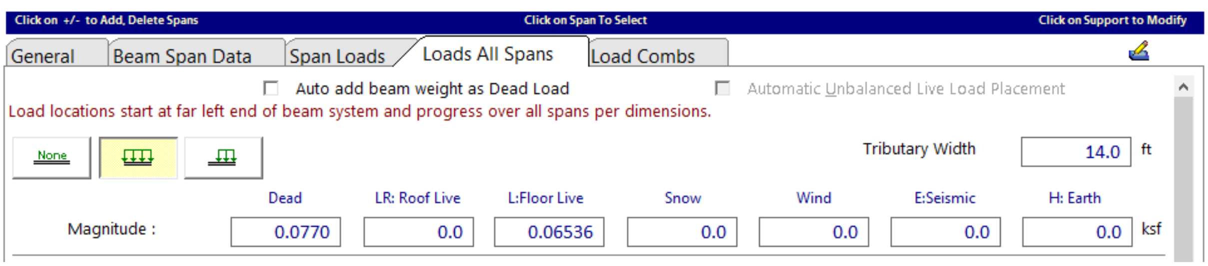
Users should be aware that “Load-Linked” reaction forces produced by a calculation where LL reduction was used will ***NOT*** be stored in the Revit model with reduced magnitudes. This is because LL reduction is an element-specific code provision, and the permissible level of reduction will vary from one structural component to the next.

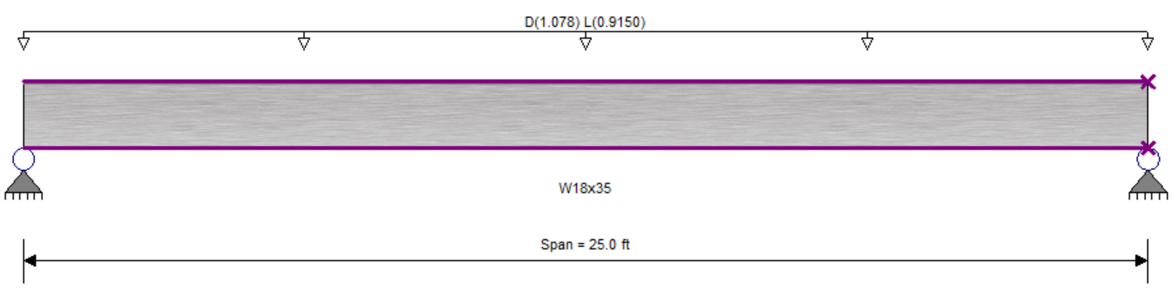
As a result, reactions are stored in the Revit model with an “unreduced” magnitude obtained by ***dividing*** the reduced LL or LR reactions by the applicable reduction factor. This process prevents “Load-Linking” reactions from being stored in the Revit model with a magnitude that reflects code provisions unique to only one specific element and allows loading in the Revit model to remain valid even when the LL reduction settings on various elements change over time.

As an example, consider the design scenario illustrated below. The floor framing layout uses long-span deck with a large spacing between floor beams, causing both the beam and its supporting girder to be individually eligible for LL reduction. The floor beam is designed first, and live load reduction is activated:


Verify Beam Conditions
Supports
Tributary Beams
Tributary Width Sampling
LL Reduction
The selected beam has a tributary floor with Live (LL) area loads.
Verify or provide inputs for live load reduction calculations.
KLL:
2 | Interior Beam
Use Heavy LL Reduction Limits
Use Vehicle Parking LL Reduction Limits
Supports 2+ Floors?
Skip LL Reduction
Apply L = Lo (0.817) to "ALL" LL on This Beam
Remember LL Reduction
Estimate From Model
Tributary Area:
350
(sq. ft)
Factor Override:
Workset1 (Not Editable)


25.0 ft
W18x35


Click on +/- to Add, Delete Spans
Click on Span To Select
Click on Support to Modify
General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Auto add beam weight as Dead Load
Automatic Unbalanced Live Load Placement
Load locations start at far left end of beam system and progress over all spans per dimensions.
None
Tributary Width
14.0
ft
Dead
LR: Roof Live
L:Floor Live
Snow
Wind
E:Seismic
H: Earth
Magnitude :
0.0770
0.0
0.06536
0.0
0.0
0.0
0.0
ksf

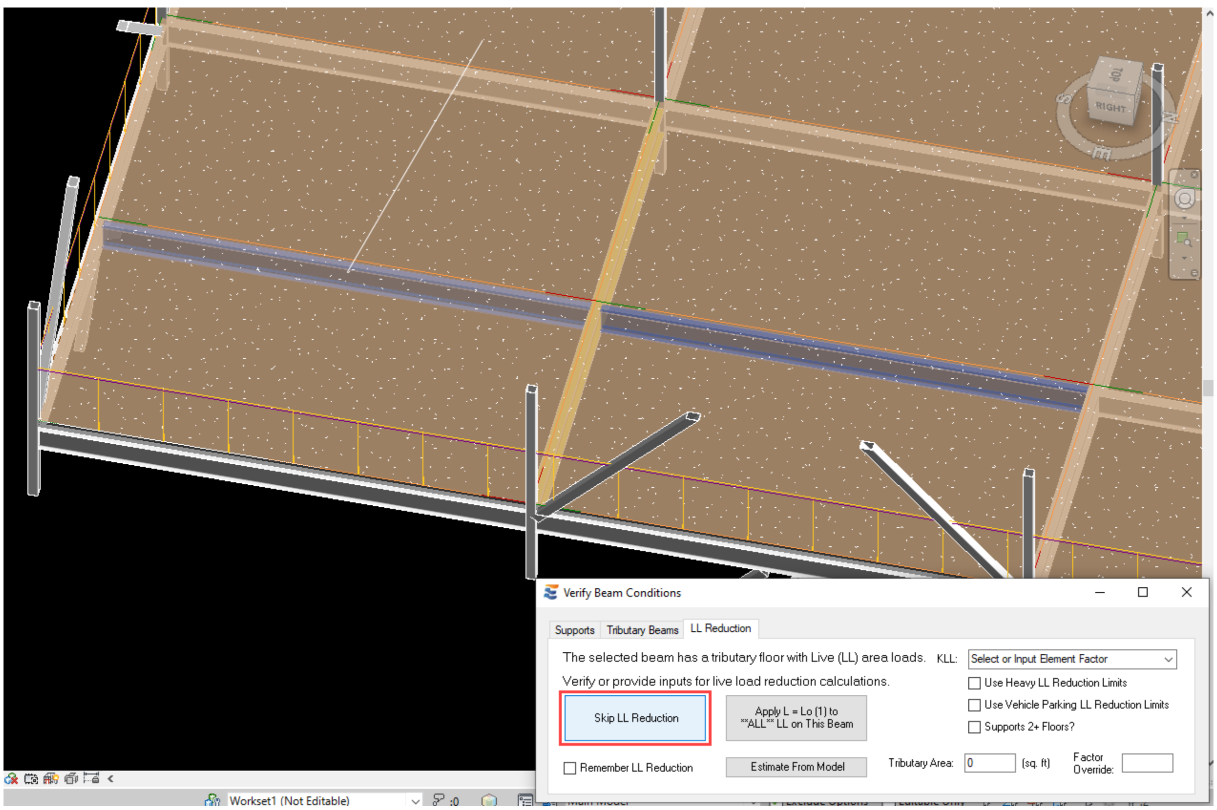

D(1.078) L(0.9150)
W18x35
Span = 25.0 ft

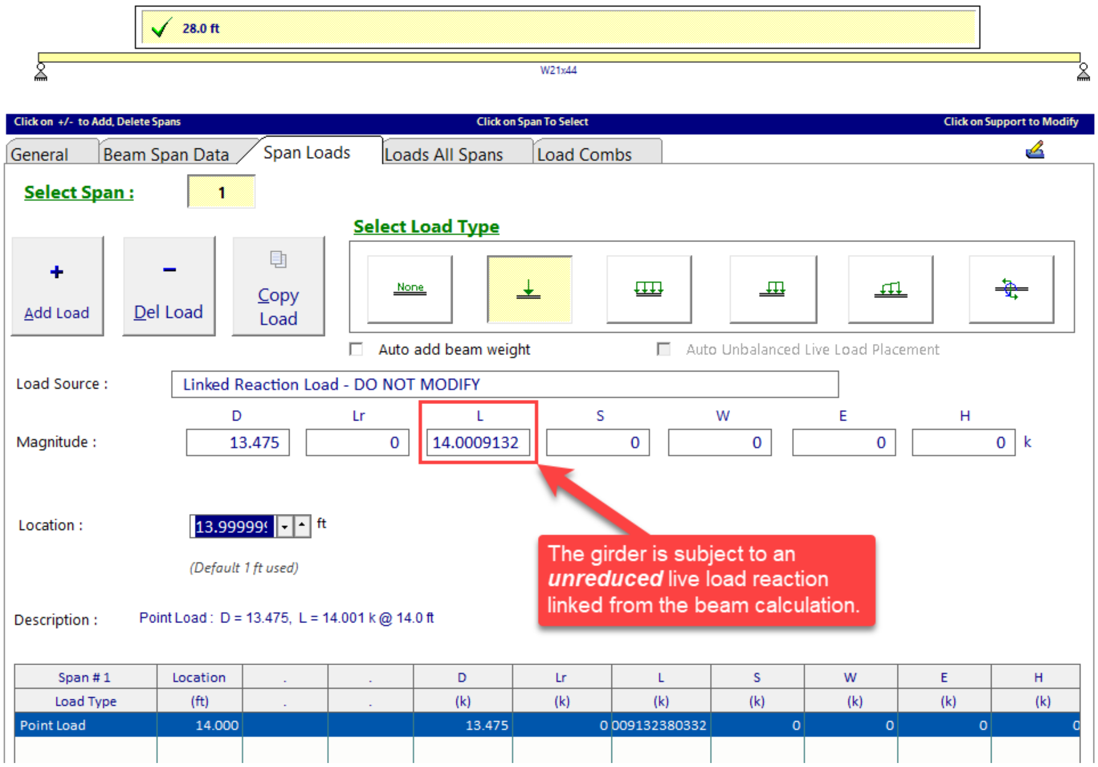
Extreme Reactions (kips)

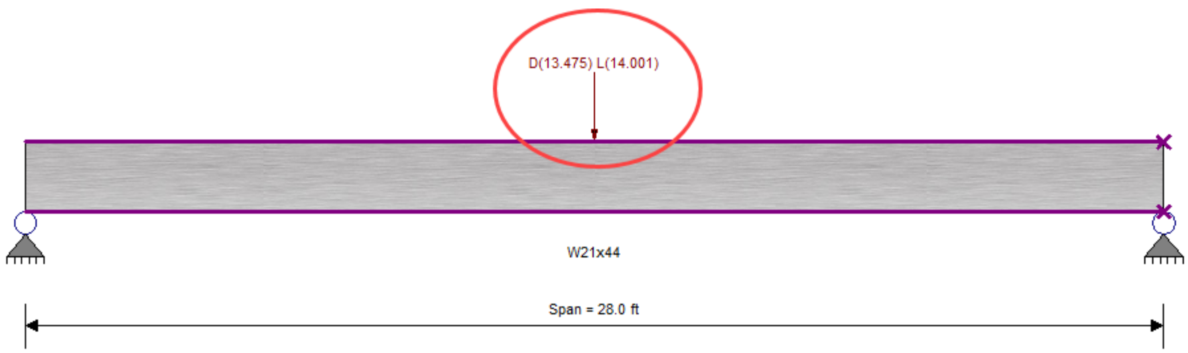
	D	Lr	Lf	S	W	E	H
Support #1	13.48		11.44				
Support #2	13.48		11.44				

Support reactions caused by ***reduced*** live load magniutude.

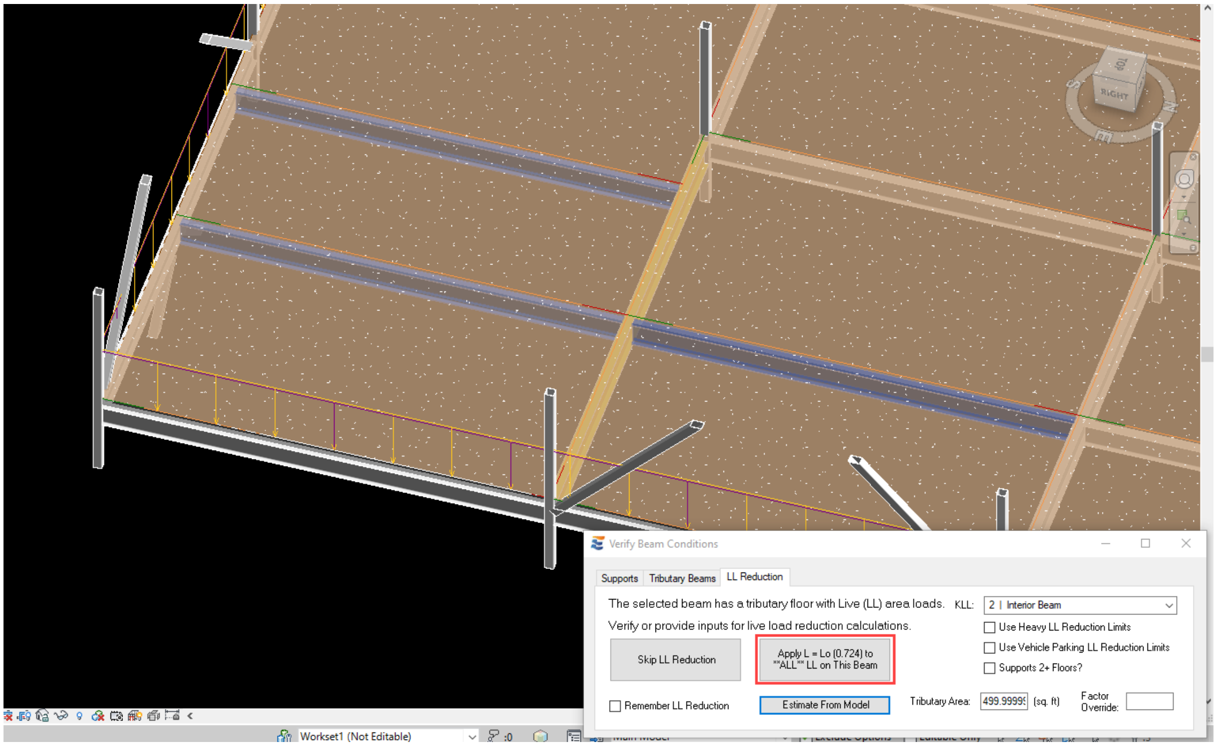
Following the design of the infill floor beam, the girder is designed next, but LL reduction on the girder is NOT activated. The girder calculation populates in ENERCALC SEL with unreduced reactions from the beam, demonstrating that the design of the girder is ***NOT*** inappropriately influenced by the reduction factor previously used to design the beam.



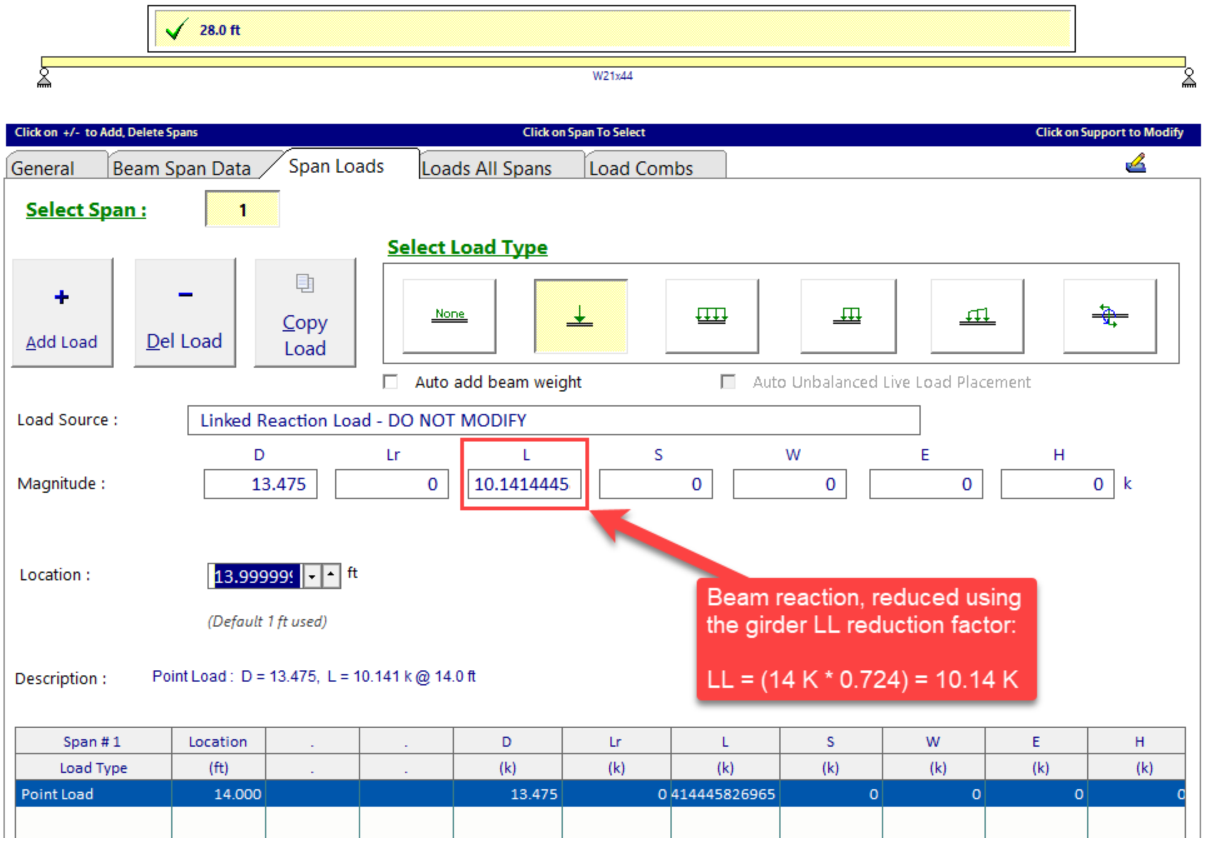


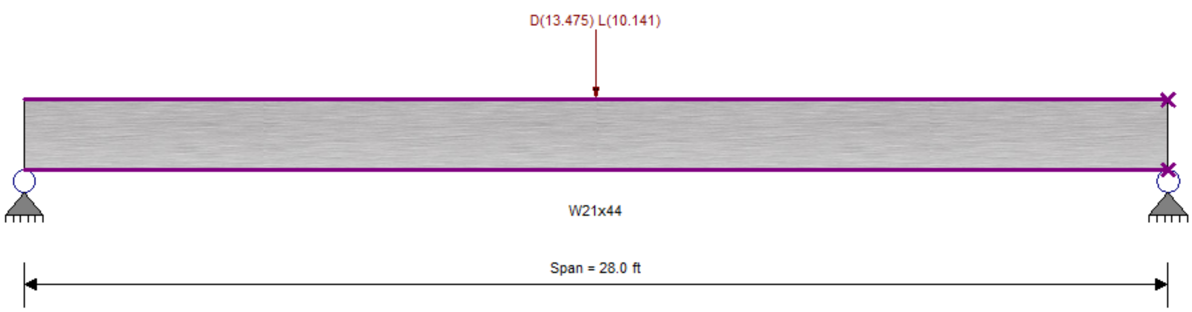


The girder design is then re-launched, this time with LL reduction activated. In this case, the girder tributary area differs significantly from the beam tributary area, corresponding to a different LL reduction factor.



When the girder calculation is launched, the unreduced beam LL reaction is detected and automatically reduced using the reduction factor computed for the girder calculation.





10.4.12.10 Adding or Modifying Loads in ENERCALC SEL

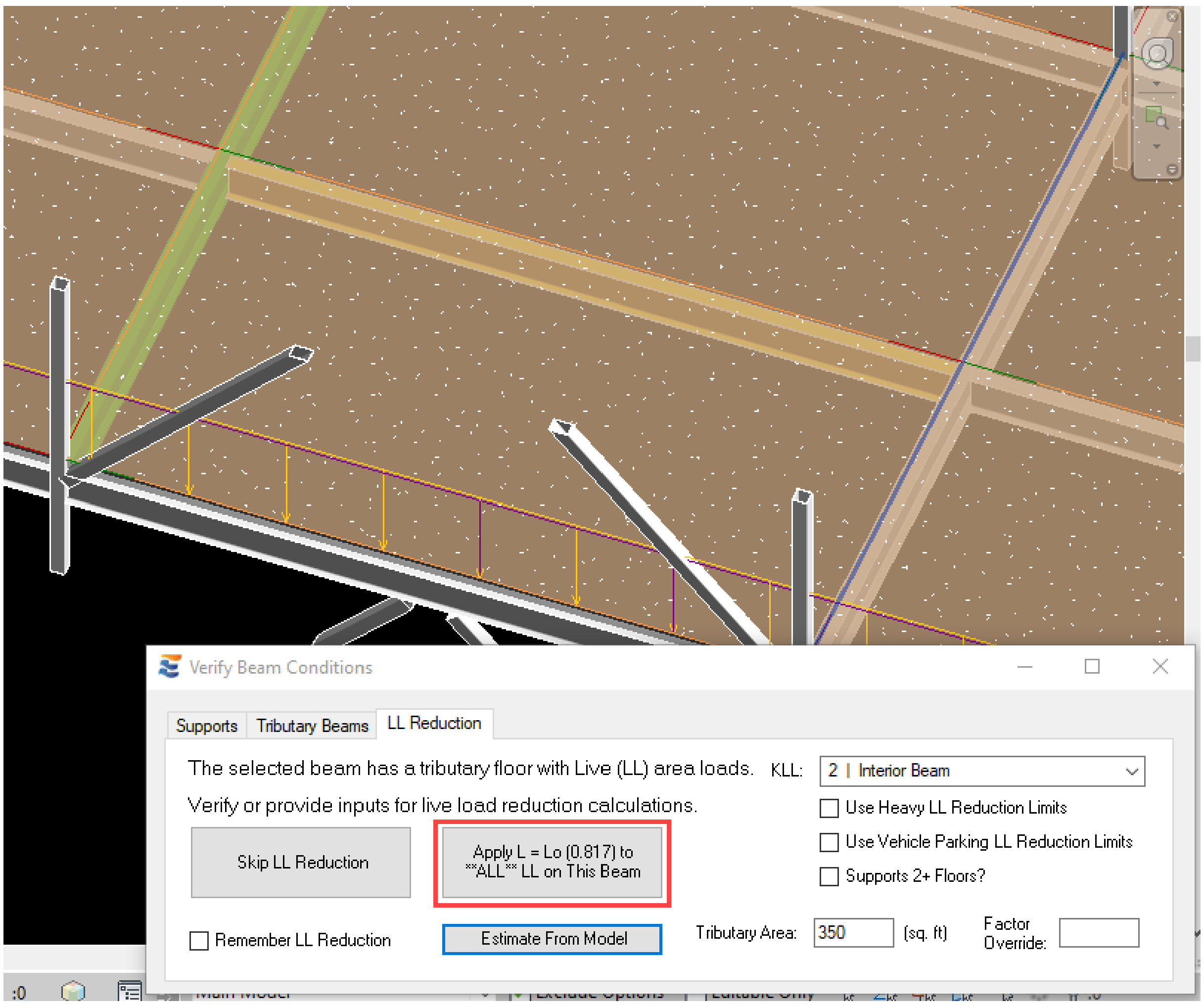
It is advisable for users to remain aware of whether live load reduction is currently active in an ENERCALC SEL calculation at any given time. Although there is not a direct indication of reduction status within the ENERCALC SEL interface, the reduction settings are managed during calculation launch from Revit as previously described.

NOTE: Any live loads (LL) or roof live loads (LR) that are added or modified from the ENERCALC SEL interface while LL or LR reduction is active in a calculation will be treated as REDUCED magnitudes. This load will then be created or updated in the Revit model with an “unreduced” magnitude obtained by ***dividing*** the reduced magnitude by the applicable reduction factor.

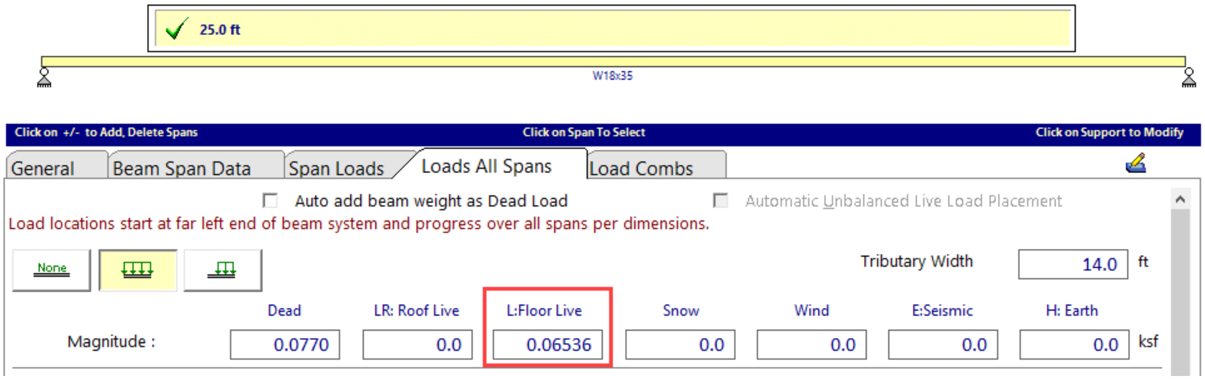
This process prevents loads from being stored in the Revit model with a magnitude that reflects code provisions unique to only one specific element and allows loading in the Revit model to remain valid even when the LL reduction settings on various elements change over time.

If this behavior is undesirable, loads may be modified directly from the Revit model, or from ENERCALC SEL calculations where LL reduction is not activated.

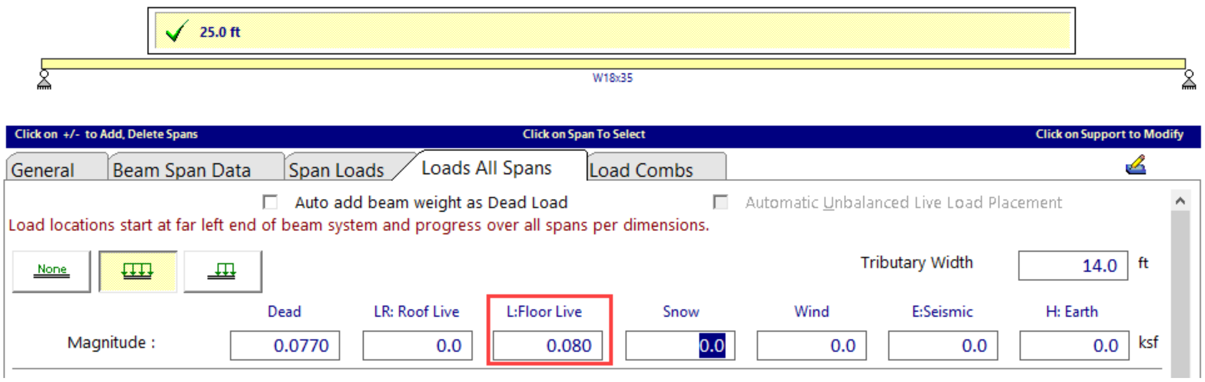
In the example below, a calculation for a floor beam supporting long-span deck is launched with LL reduction activated.



Since LL reduction was activated during launch, all LL magnitudes displayed in the ENERCALC SEL interface are reduced.

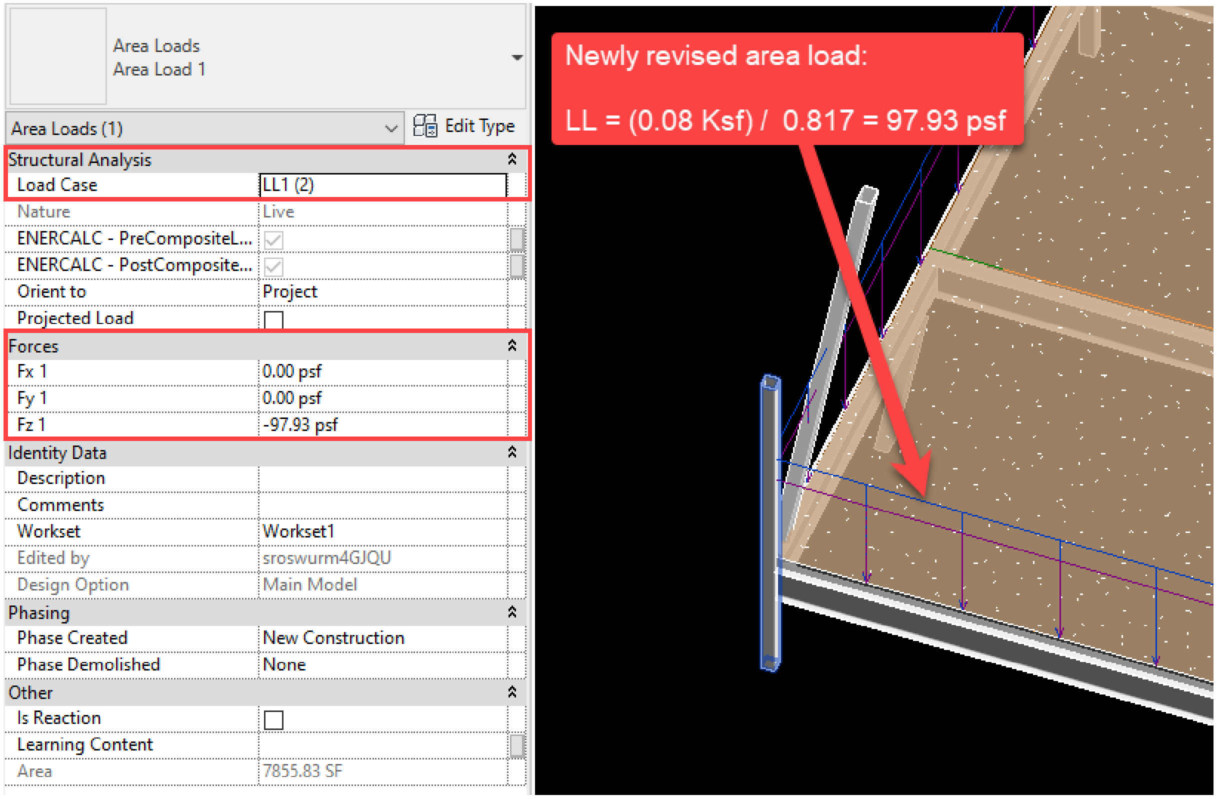


If this live load magnitude is manually modified by the user, the newly revised magnitude is treated as a reduced magnitude.



This will result in the underlying area load in the Revit model being updated to a magnitude of:

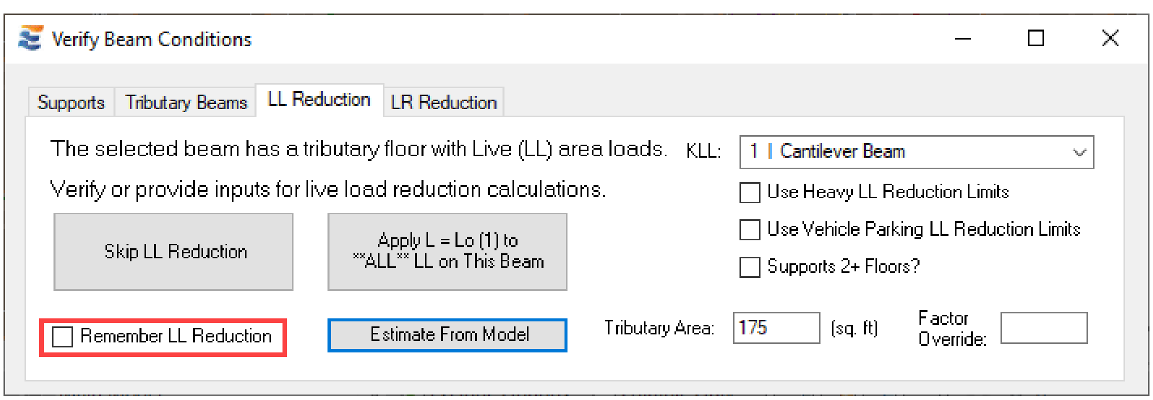
LL = (0.08 Ksf) / 0.817 = 0.09792 Ksf

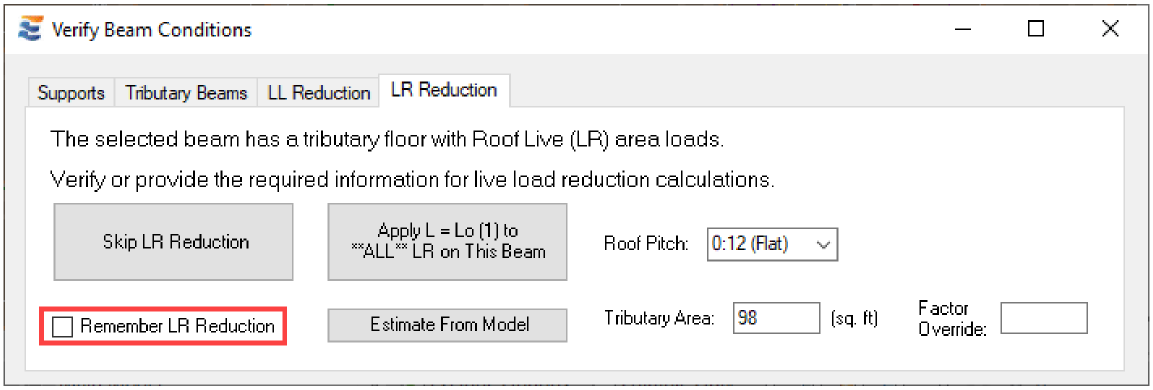


10.4.12.11 Relaunches and “Remember” Controls

In order to help ensure a fast and intuitive design process, ENERCALC for Revit provides a number of tools for retaining various information about LL reduction settings from one launch of a beam calculation to the next.

For any calculation where the user wishes to verify and approve LL reduction or LL reduction settings only once and not be prompted again, the launch window includes checkbox options marked “Remember LL Reduction” and “Remember LR Reduction”.

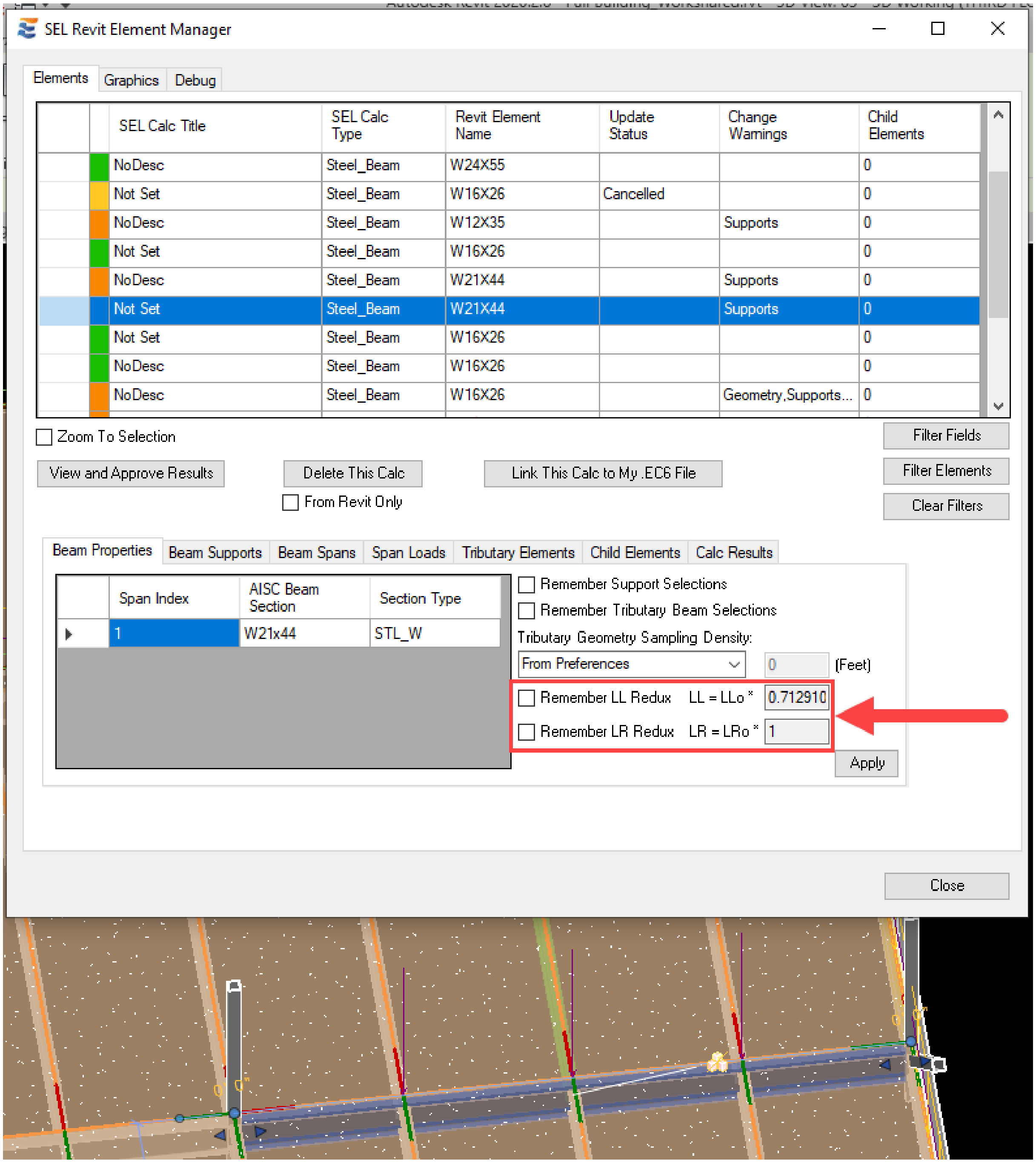




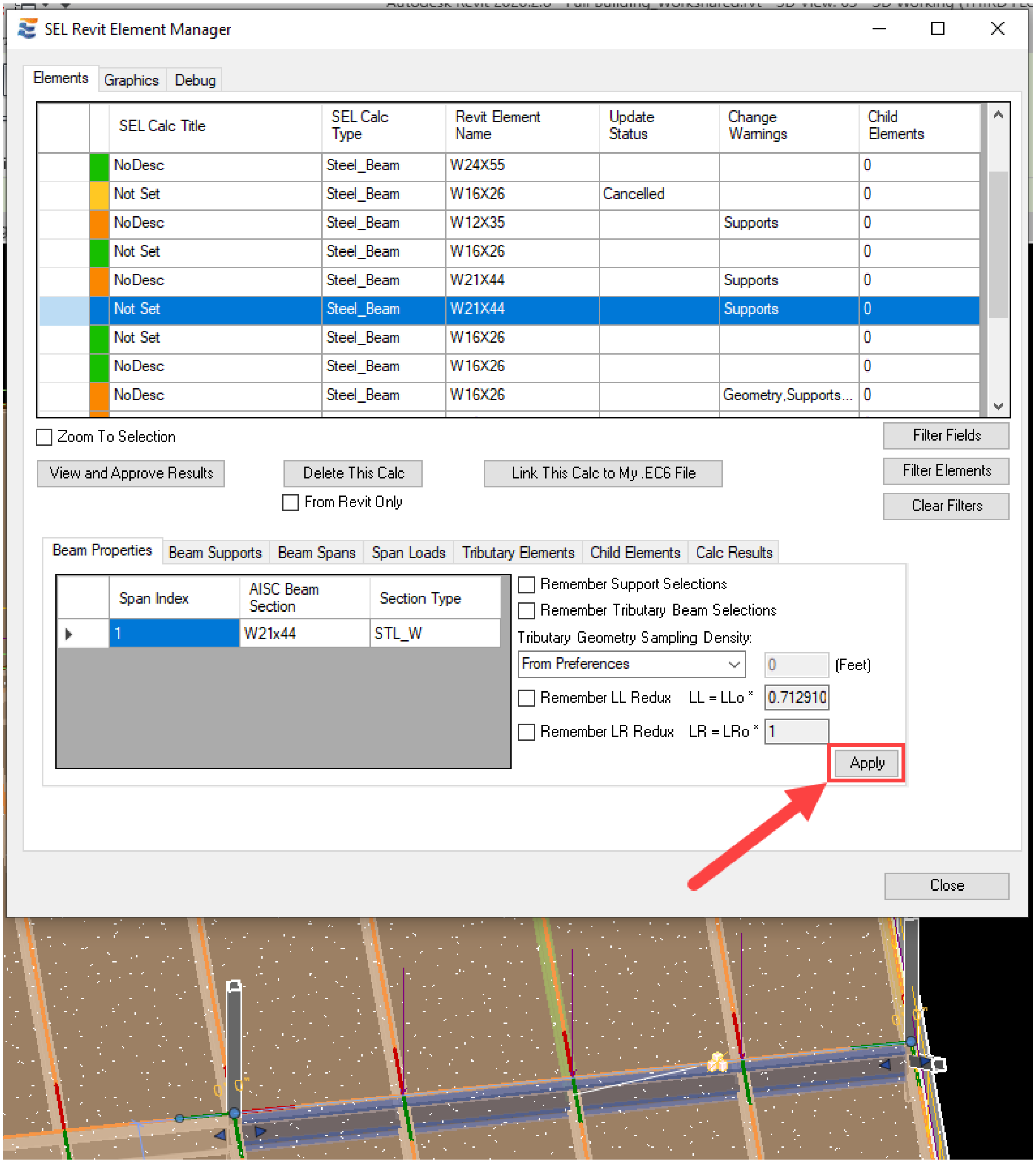
Activating the “Remember…” checkbox prior to clicking the “Apply…” button will cause ENERCALC for Revit to retain the input values and calculated factor currently displayed in the launch window on subsequent launches of the calculation. Future launches of the beam or girder calculation will automatically apply the stored reduction factors rather than calculating new factors or presenting data to the user for approval.

For any calculation where the user does not wish to use LL reduction or LR reduction at all, these options may be bypassed using the “Skip…” button, as described in “Navigation Overview 266”. Selecting the skip option will automatically cause associated reduction options to be suppressed on subsequent launches of the current beam or girder calculation. It is not necessary to activate the “Remember…” checkbox prior to choosing the “Skip” the reduction options. Future launches of the beam or girder calculation will automatically apply reduction factors of 1.0 rather than calculating new factors or presenting data to the user for approval.

For calculations where the user has previously chosen to “Skip…” or “Remember…” on the live load reduction options during an earlier launch, these options will be bypassed indefinitely on all future launches of the calculation. If at any point the user wishes to alter this setting and once again be prompted to verify or modify these values, the “Remember” setting may be altered from the Element Manager window. These settings are displayed on the first tab of “Beam Properties” when a calculation is selected from the summary table of the Element Manager window.



The respective reduction factors for LL and LR are shown for reference only and may not be modified directly from this view. If the “Remember…” checkbox is unchecked, then the corresponding reduction settings are not locked and they will be presented for review and approval on the next launch of this calculation. If the “Remember…” checkbox are checked, then the corresponding reduction settings are locked and will not be presented for review and approval on the next launch of this calculation. The reduction factor shown next to the checkbox will be replied automatically on all launches and recalcs of the calculation. Each setting may be locked or unlocked as desired by toggling the status of the checkboxes and then clicking “Apply”.



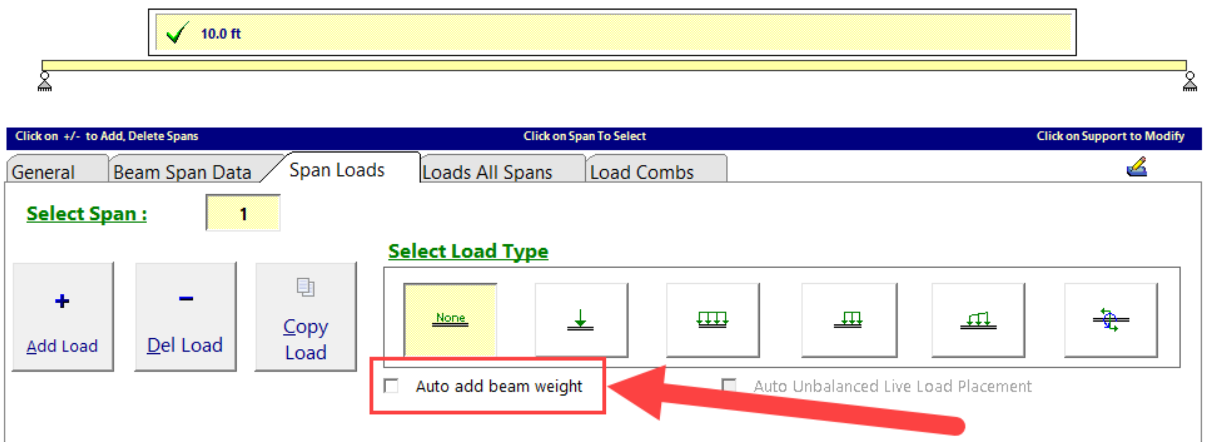
For calculations where the user has ***NOT*** previously chosen to “Skip…” or “Remember…” the live load reduction options during an earlier launch, these options will be presented for review and approval on all future launches of the calculation. By default the reduction inputs and calculated reduction factors from the previous run will automatically populate the form on subsequent runs. These values may be manually modified or overwritten by choosing the “Estimate from Model” option.

10.5 Beam Design Rules

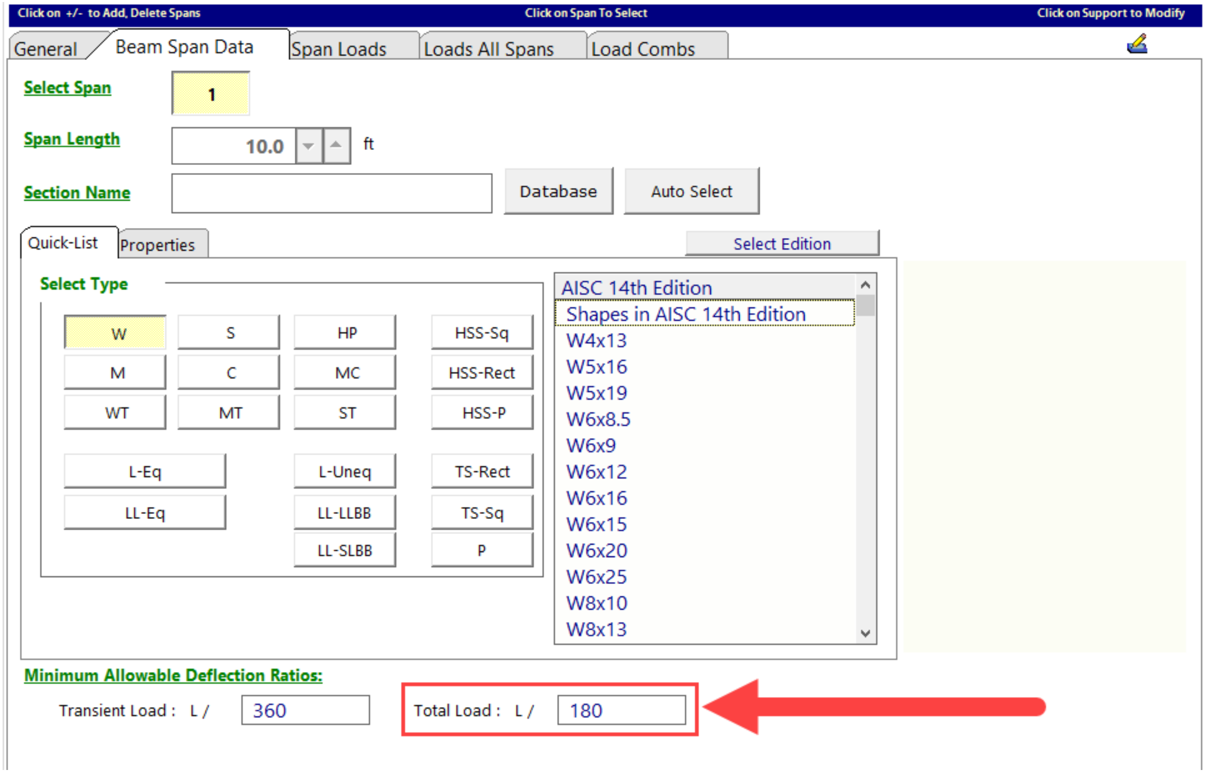
ENERCALC for Revit provides users with the ability to create and assign customized beam design rules to various beam elements throughout the Revit model. The use of beam design rules enables users to exert large-scale control over the design behavior of beams across a Revit project ***without*** needing to manually change settings repeatedly in the ENERCALC interface after each calculation launch. Beam design rules are defined and stored within individual Revit projects (.rvt files). Rule definitions are not able to be transferred from one Revit project file to another at this time.

Beam design rules can control all of the following design inputs:

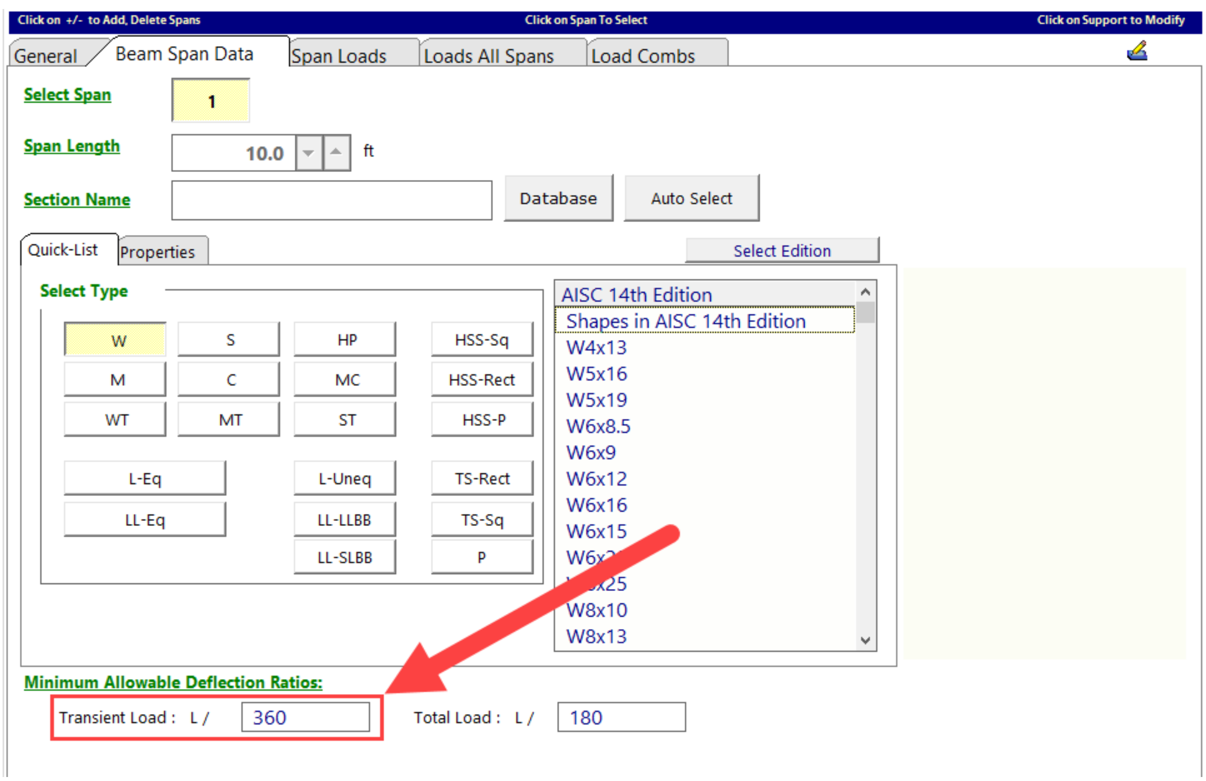
- **Auto add beam weight**



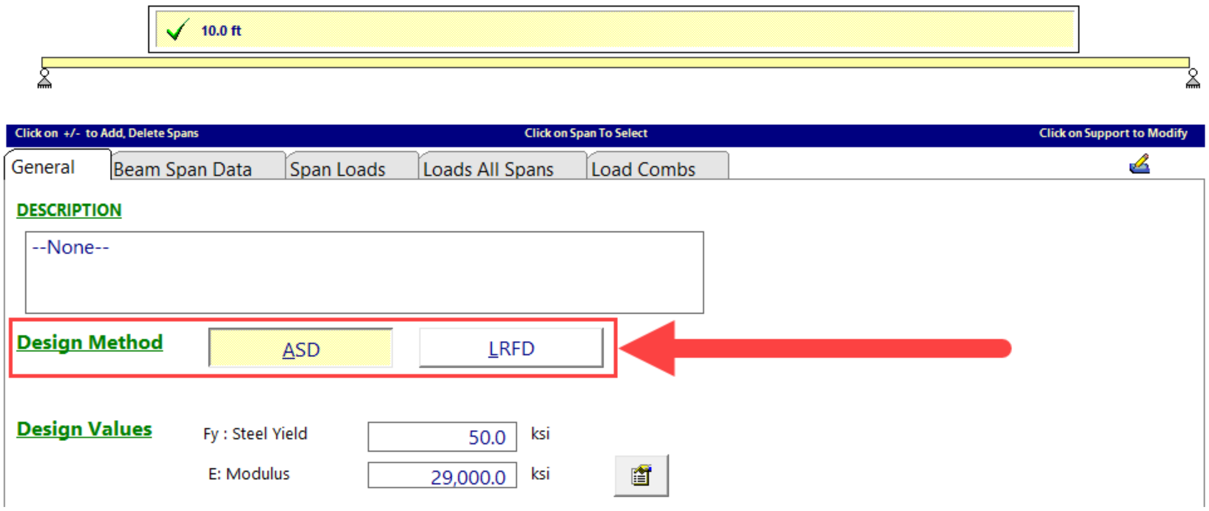
- **Deflection ratio - Total**



- **Deflection ratio - Live Load**



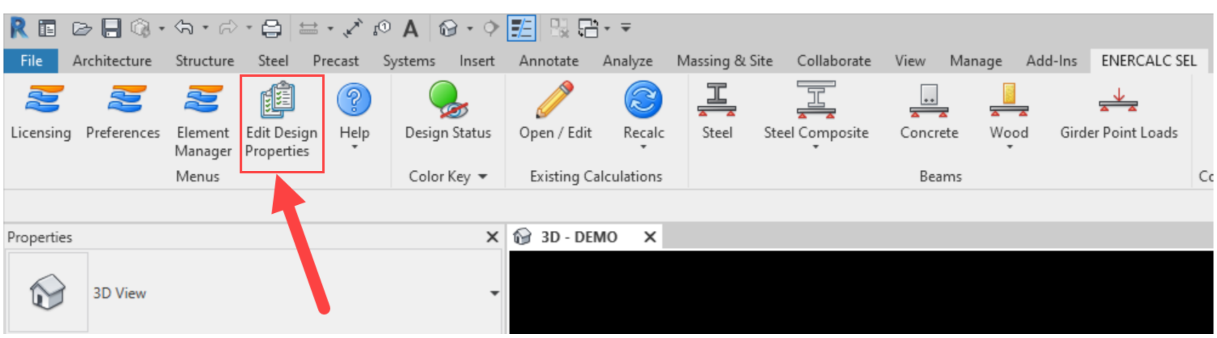
- **Design method (ASD / LRFD)**

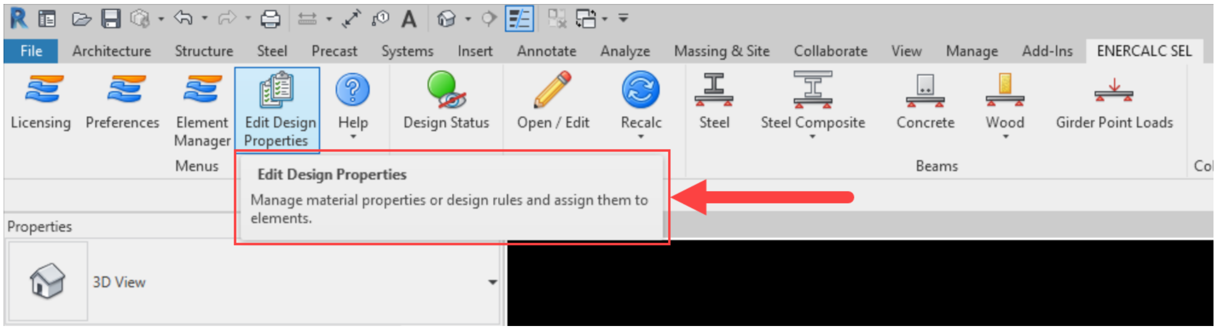


NOTE: Assignment of Beam Design Rules is **NOT** mandatory to launch a calculation. Any beam calculation launched without explicitly applied rules will be launched with the following default rules:

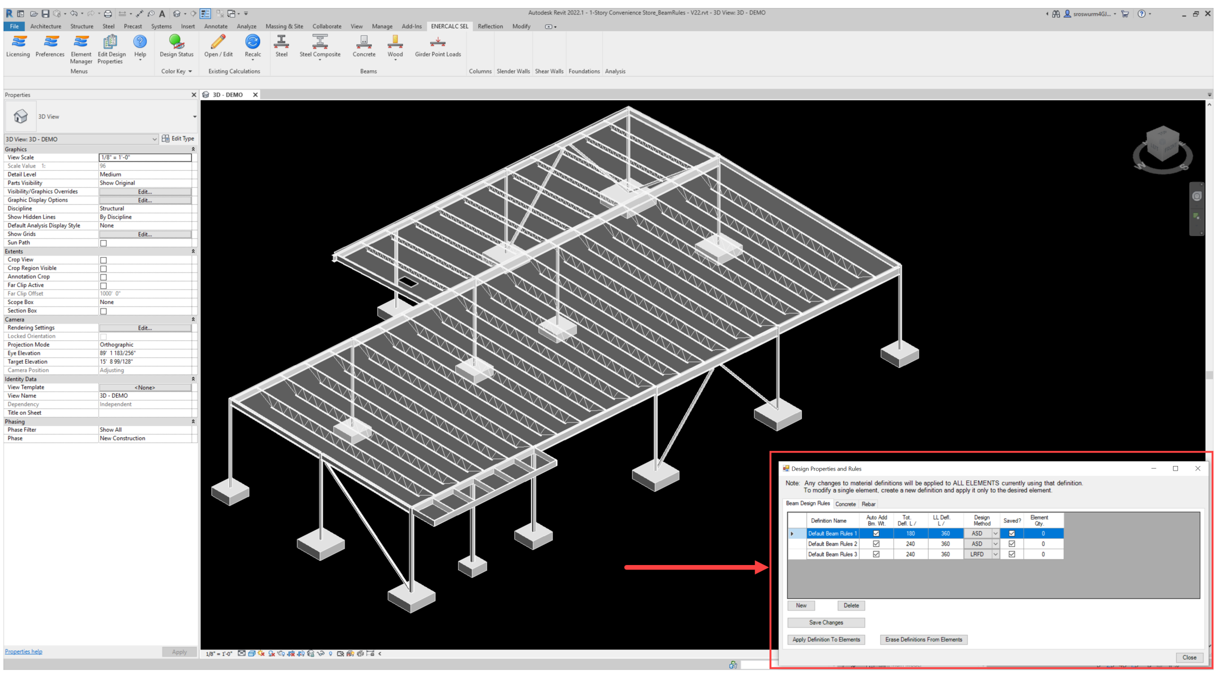
- Auto add beam weight = true
- Deflection ratio - Total = 180
- Deflection ratio - Live Load = 240
- Design method = ASD

Beam design rules are accessed from the ribbon bar, using the "Edit Design Properties" button:





When triggered from the ribbon button, the menu will load in the lower right corner of the Revit UI:



This menu also contains tabs for managing other design properties, including concrete and rebar material properties. For more detail on the use of these tabs, refer to Concrete Material Properties 480 and Rebar Material Properties 485. Users should note that the basic mechanics are the same for all design definition tabs found in this menu. The processes to "Apply", "Modify", "Add", "Delete", or "Erase from Elements" do not vary from tab to tab. Regardless of which definition tab is being used, users may reference the following sections for guidance about managing design definitions:

Applying Rules to Elements 330

Modifying Rules 333

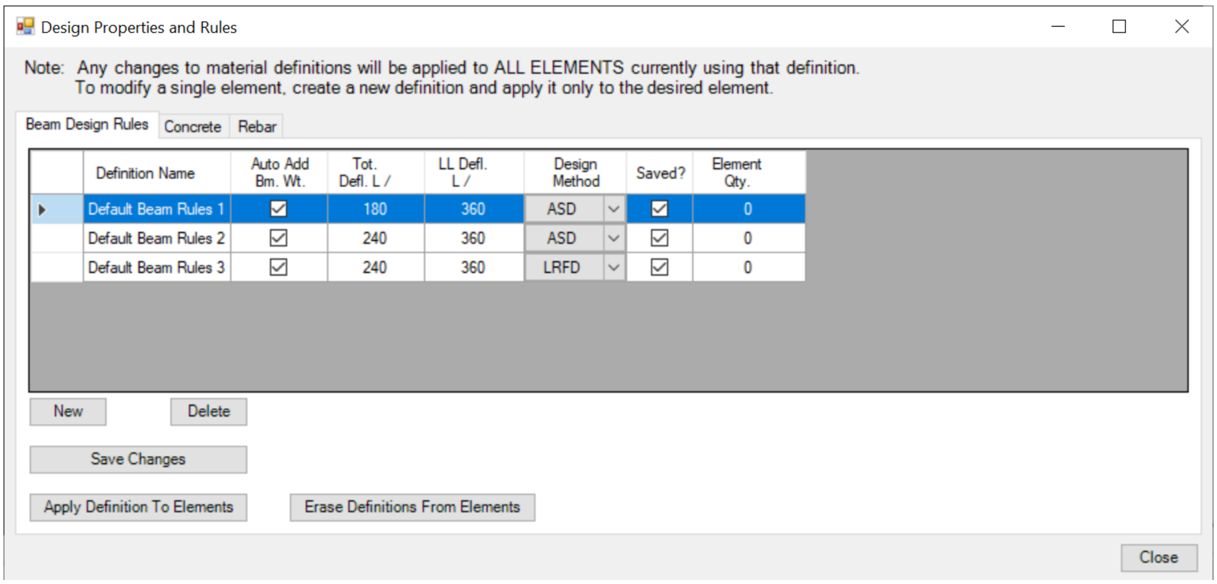
Adding Rule Definitions to a Project 337

Deleting Rule Definitions from a Project 339

Erasing Rule Definitions From Elements 341

Navigating Rule Definitions 343

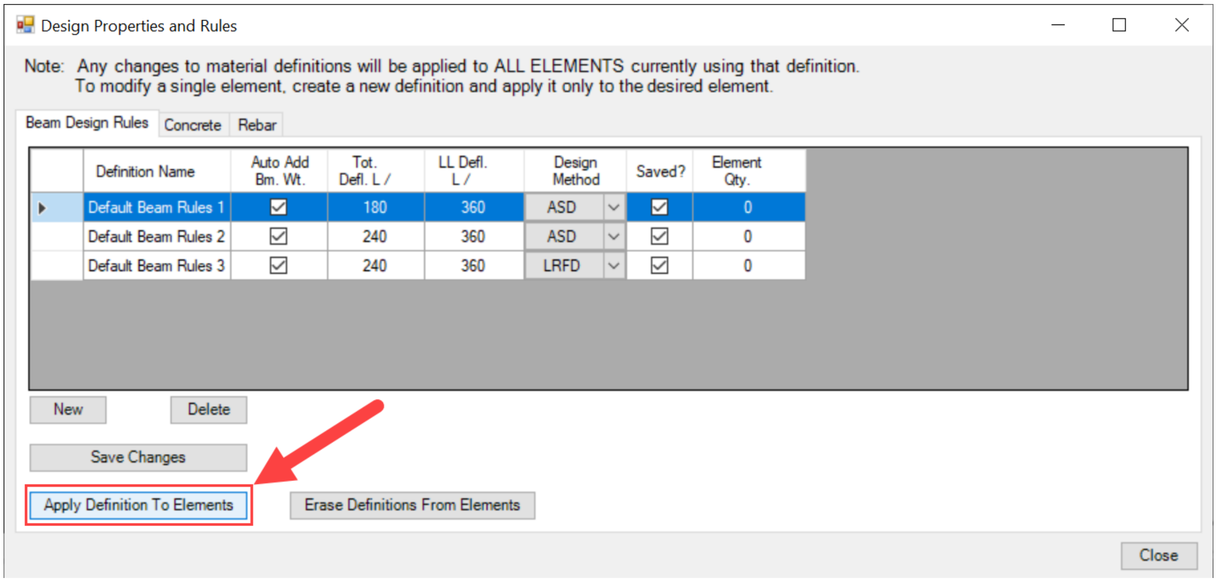
In a new project where no previous definition editing has been performed by the users, the menu will be populated automatically with a set of 3 sample beam design rules which are stored in the .rvt file by default. Use of these default rules is optional. They may be removed or edited/renamed as desired by the user (see Modifying Rules 333). Even though they exist in the Revit model, the default rules will not influence the behavior of beam calculations unless manually applied to specific beam elements by the user.



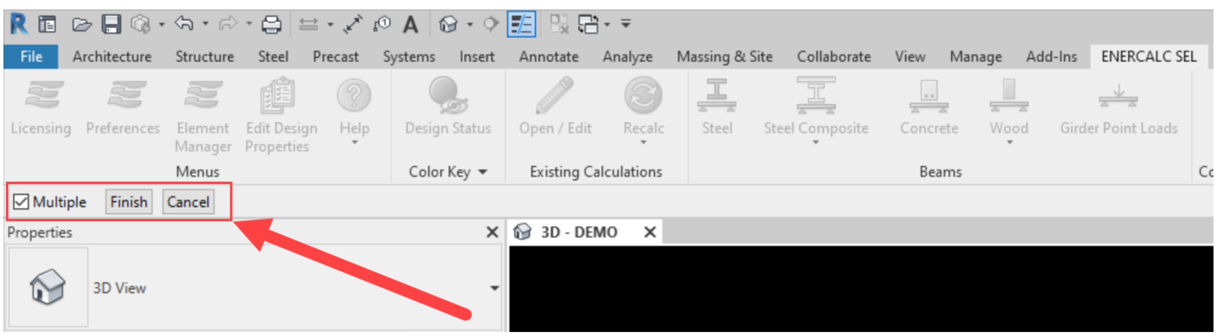
10.5.1 Applying Rules to Elements

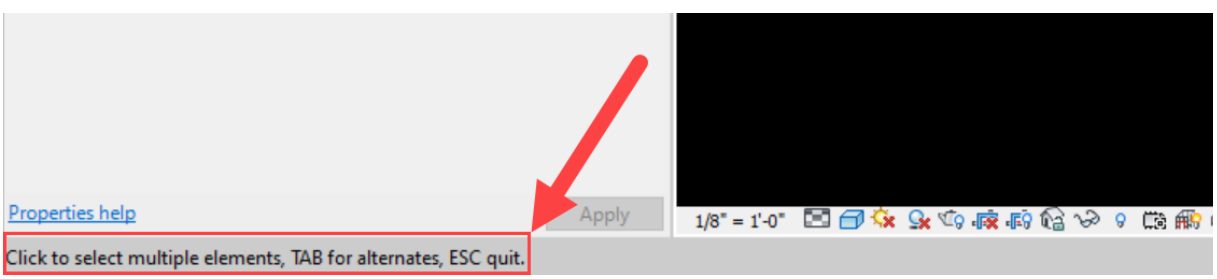
The beam design rules stored in a project and displayed in the "Design Properties and Rules" menu do not have any influence over the behavior of beam calculations unless they are manually applied to specific elements by the user. Once a design rule has been applied to a beam, the rule will be detected automatically during subsequent calculation launches for that element. The calculation will then load in the ENERCALC interface with the design properties specified by the design rule already applied.

Rules are applied using the "Apply Definition To Elements" button:

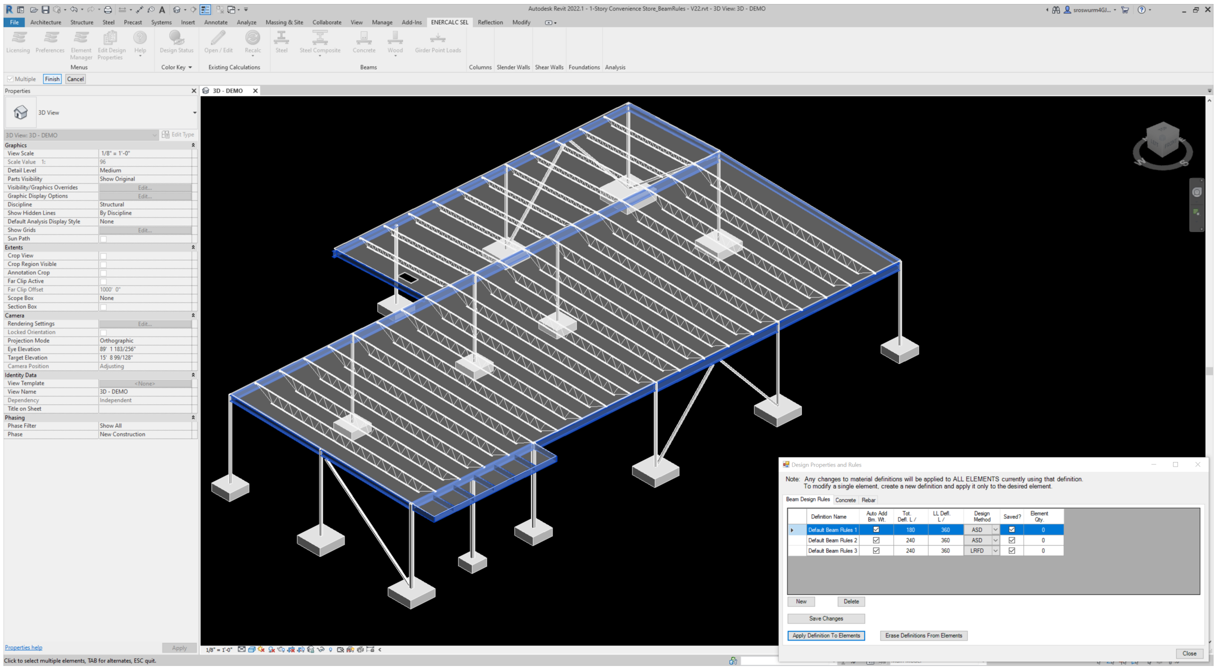


When a definition is selected in the table, clicking the "Apply" button will trigger a multi-select process in the Revit UI. EFR will await element selection by the user:

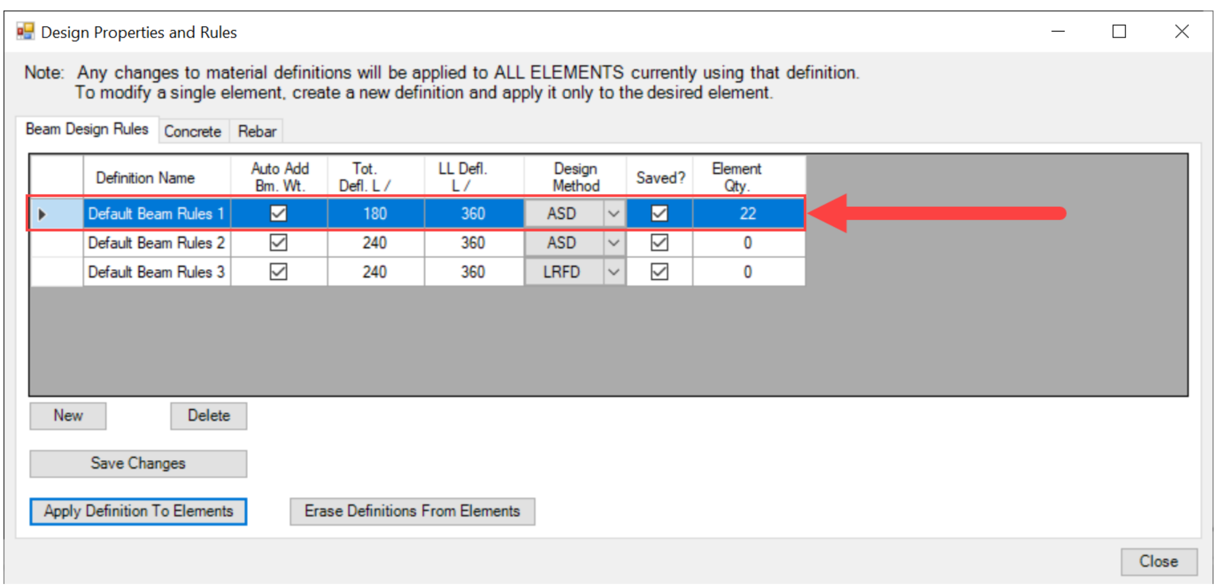




When navigating the Revit UI during the selection process, only beam and girder elements (of all materials) are eligible for selection to apply beam design rules:

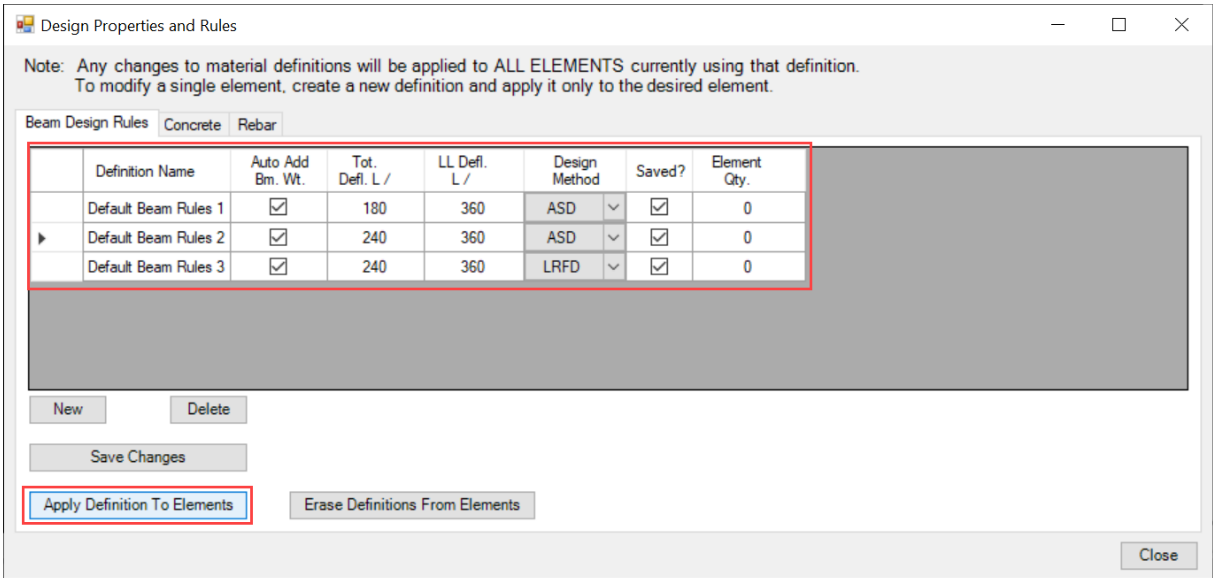


When the desired elements have been selected, clicking the "Finish" button will cause the menu table to update with the quantity of elements to which the definition has been applied:

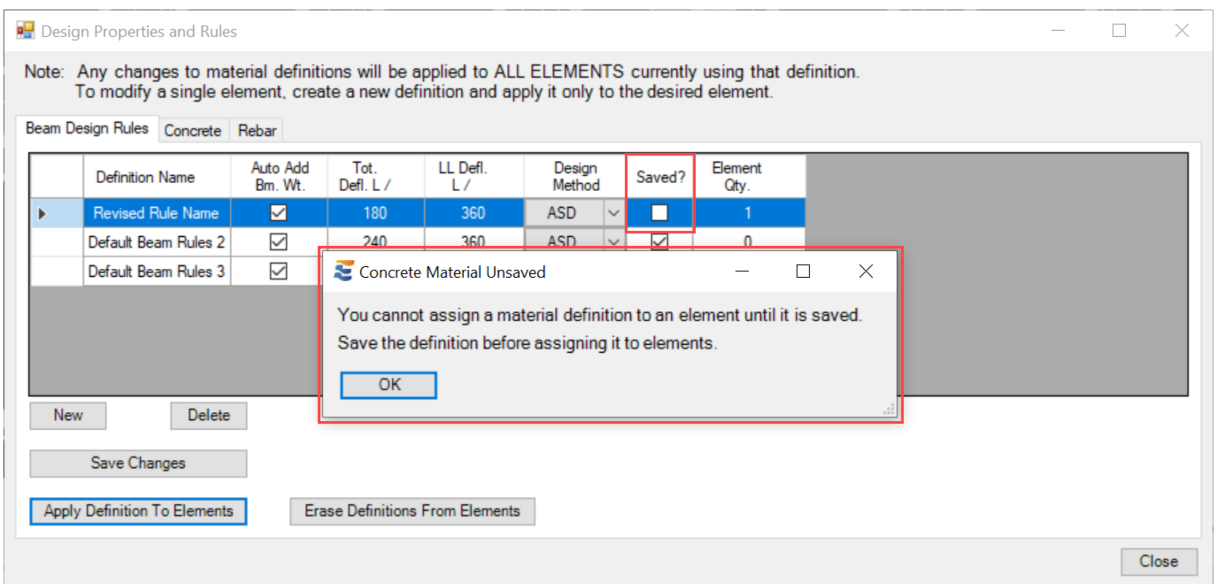


Clicking "Cancel" during the selection process will halt the operation with no impact to the Revit model or the menu table.

Clicking the "Apply" button will have no effect when there is no definition selected in the table:



Clicking the "Apply" button when the currently selected definition contains unsaved changes will result in the following warning:

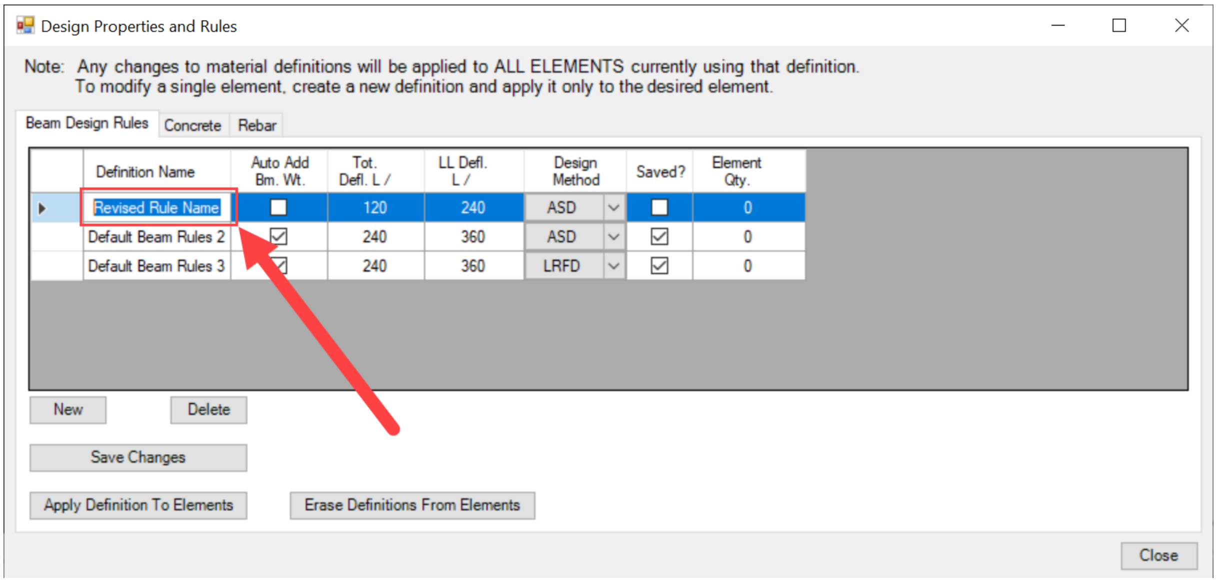


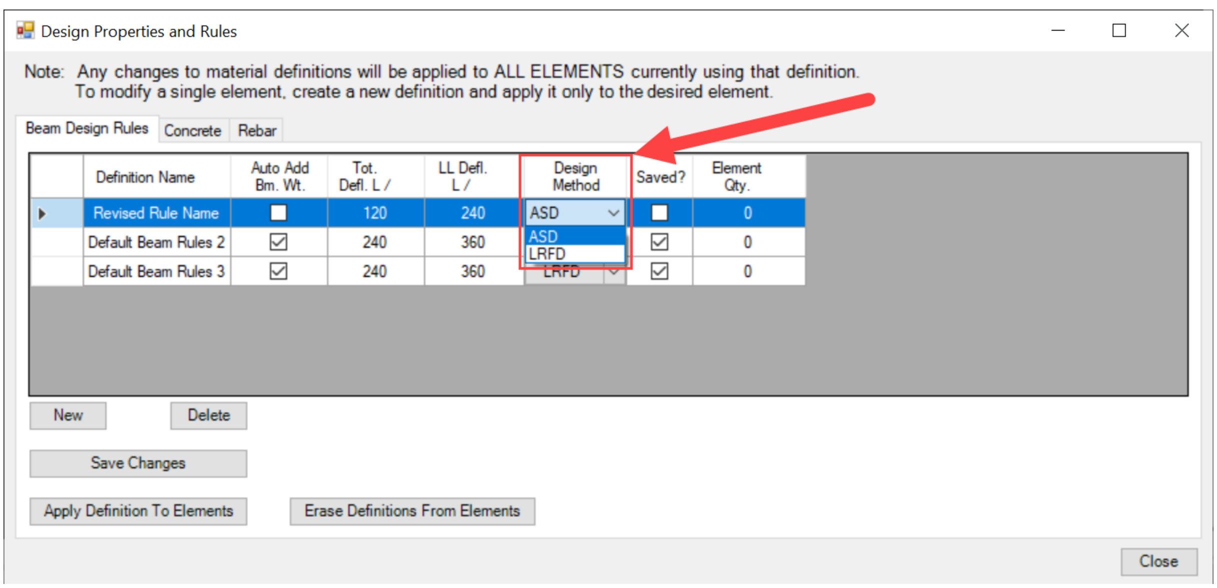
For more information about "Saved" vs "Unsaved" definitions, refer to Modifying Rules 333 and Adding New Rules 337.

10.5.2 Modifying Rules

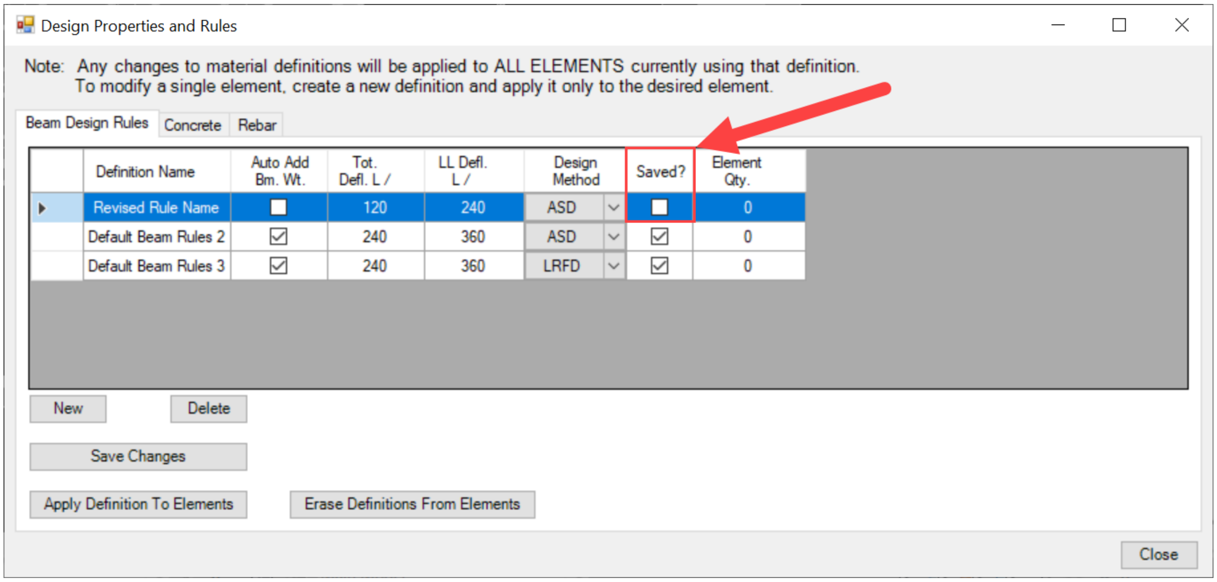
Any design rule definition found in the table may be modified as desired by the user. The final (2) columns in the table ("Saved" and "Element Qty.") are read-only and may not be modified. The "Saved?" check box indicates whether the changes to the definition have been saved in the Revit project. If the "Saved" check box is un-checked, any changes made to the definition will be lost when the menu is closed. The "Element Qty." column indicates how many elements in the Revit model have been assigned to use the corresponding definition (see Applying Rules to Elements 330).

The individual values in the first (5) columns of each definition may be manually modified by simply clicking in the table cell and changing the value:

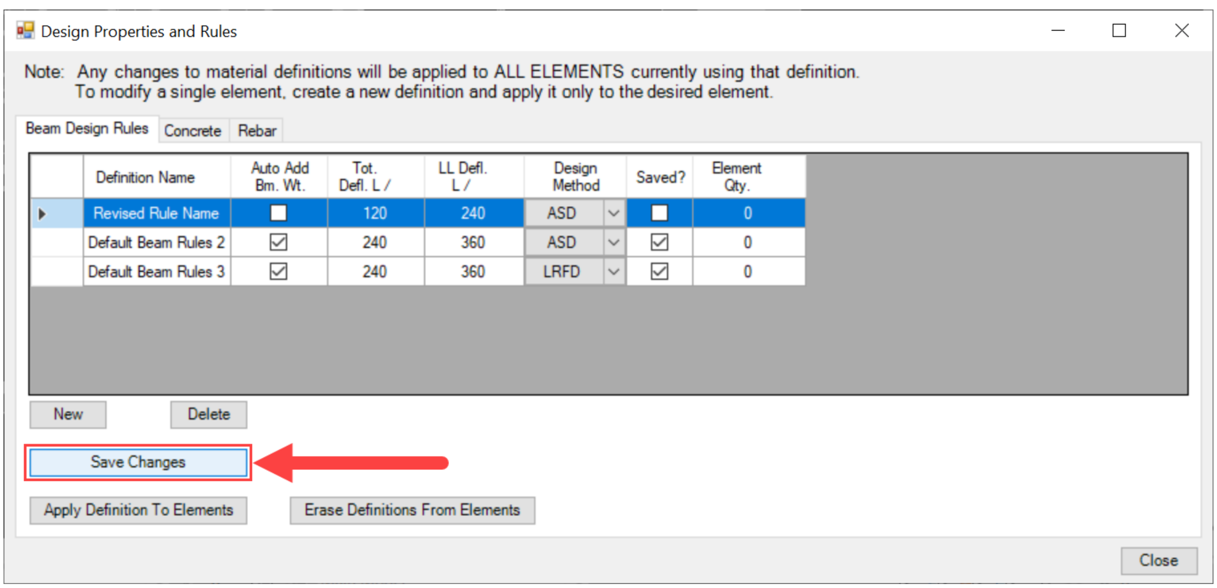




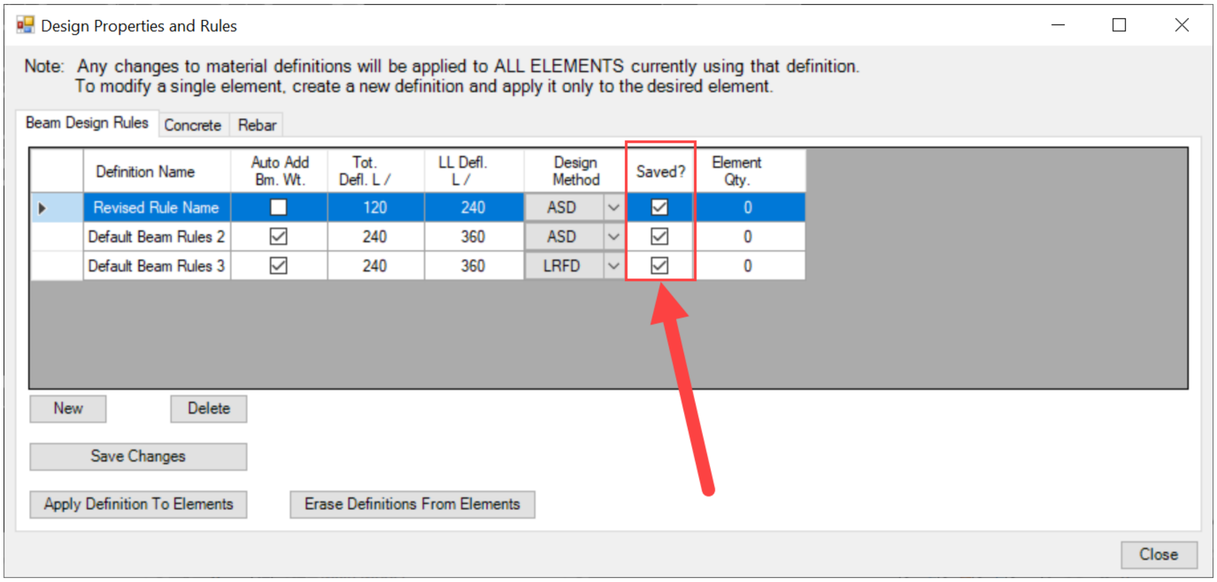
Once changes have been made to a definition, the "Saved" column check box will be cleared (un-checked), indicating that the row's definition has unsaved changes:



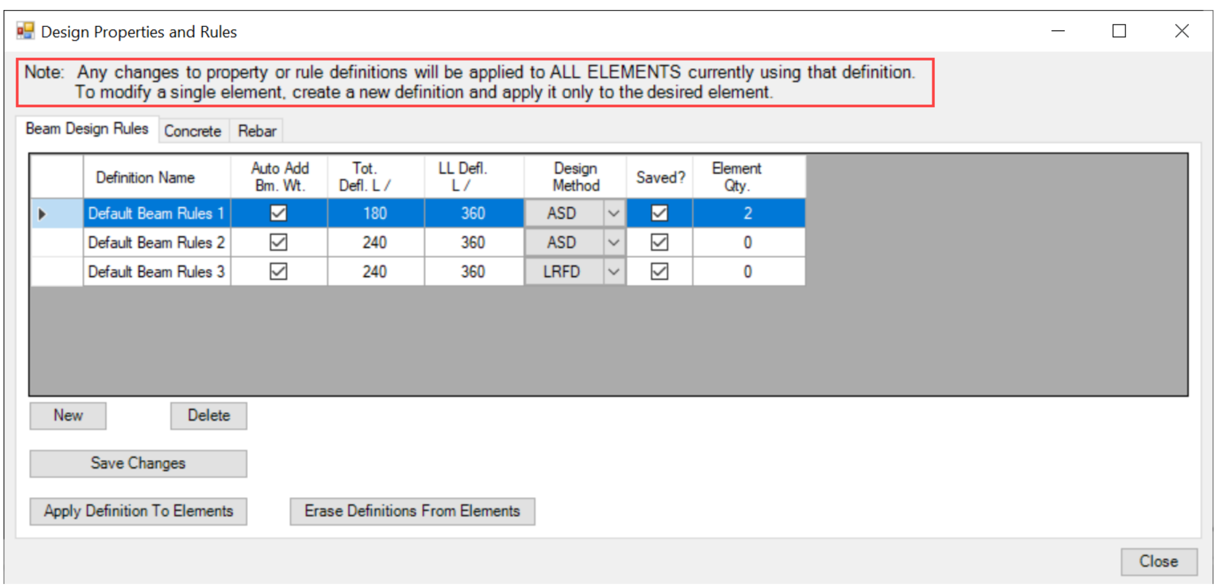
Values may be changed in any or all rows of the table, but changes will not be permanent until saved using the "Save Changes" button. The "Save Changes" command is not specific to the currently selected row. It will save all changes to all rows in the "Beam Design Rules" table.



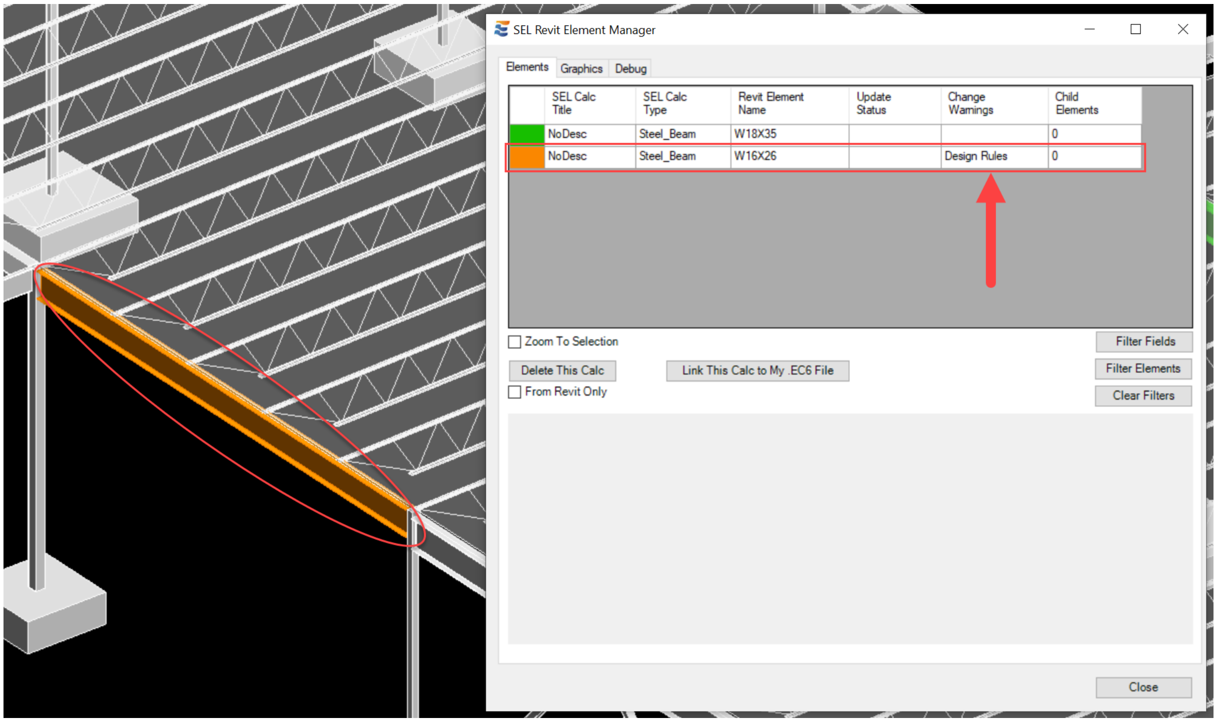
After changes have been saved, all rows in the table will again show that they are "Saved" via the check box indicator:



As noted in the heading above the table, users should be aware that changing a definition will automatically impact **ALL ELEMENTS** to which the definition has been assigned:

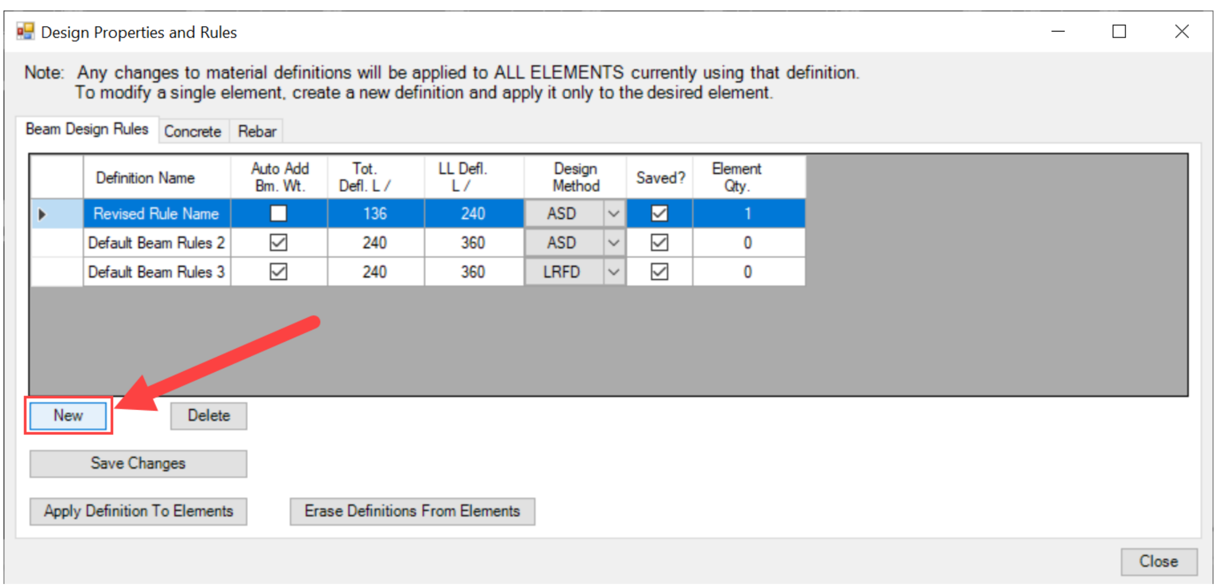


Whenever changes are made to a design rule definition, there is a risk of impact to existing calculations that were launched using a previous version of the rule. As a result, any beam calculation bearing the modified definition will automatically be placed in a warning state. This is evident via both color-coded highlighting in the Revit view and via the Element Manager:

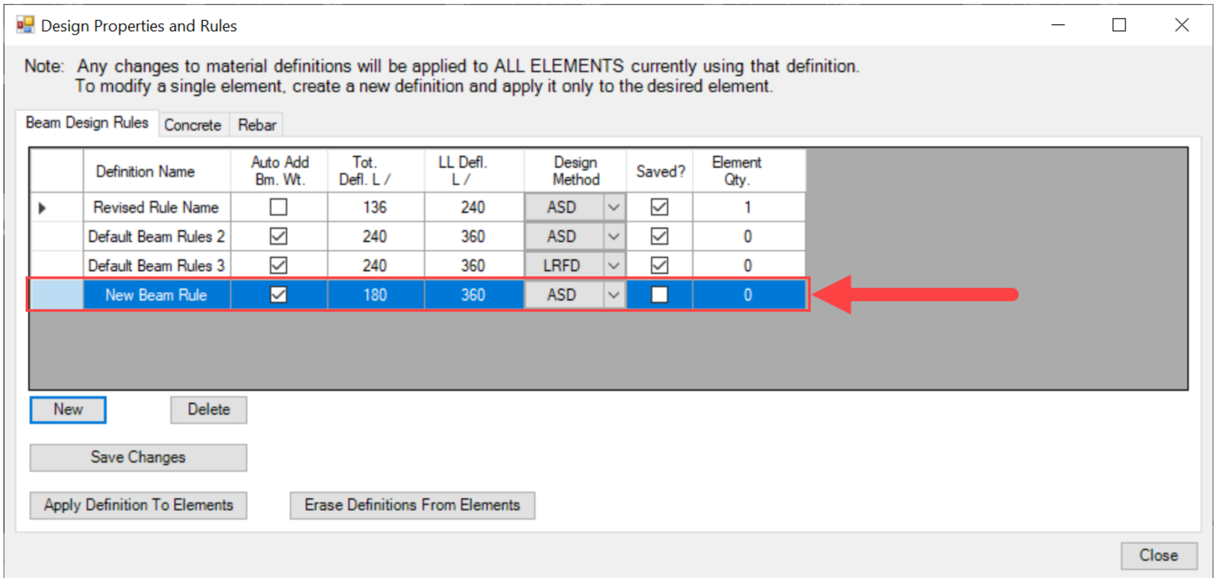


10.5.3 Adding Rule Definitions to a Project

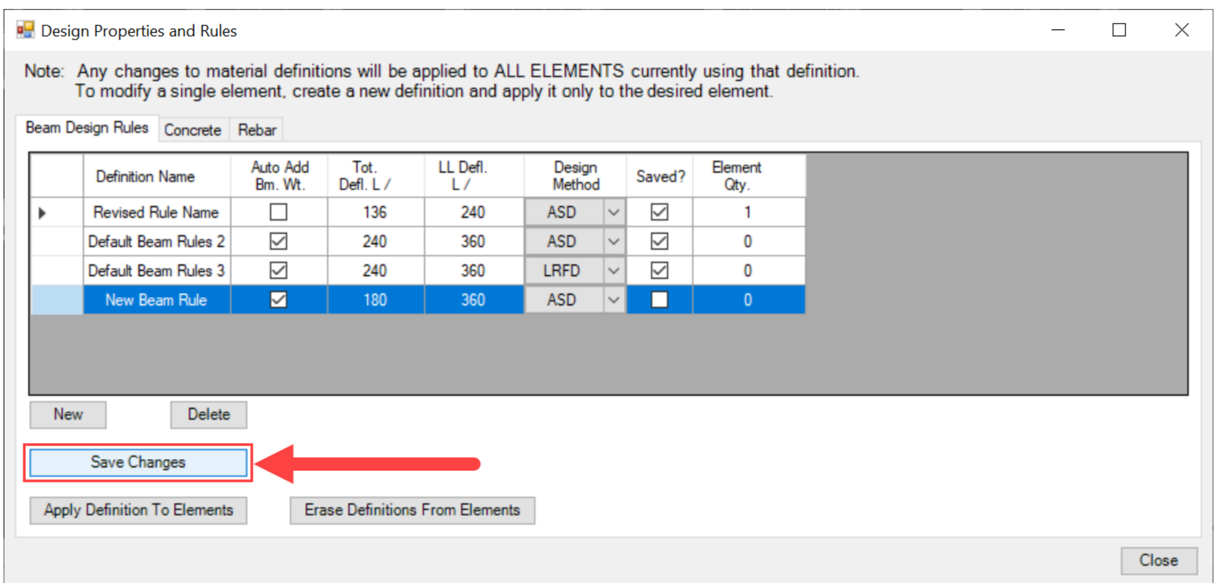
New beam design rules may be added to the table using the "New" button:

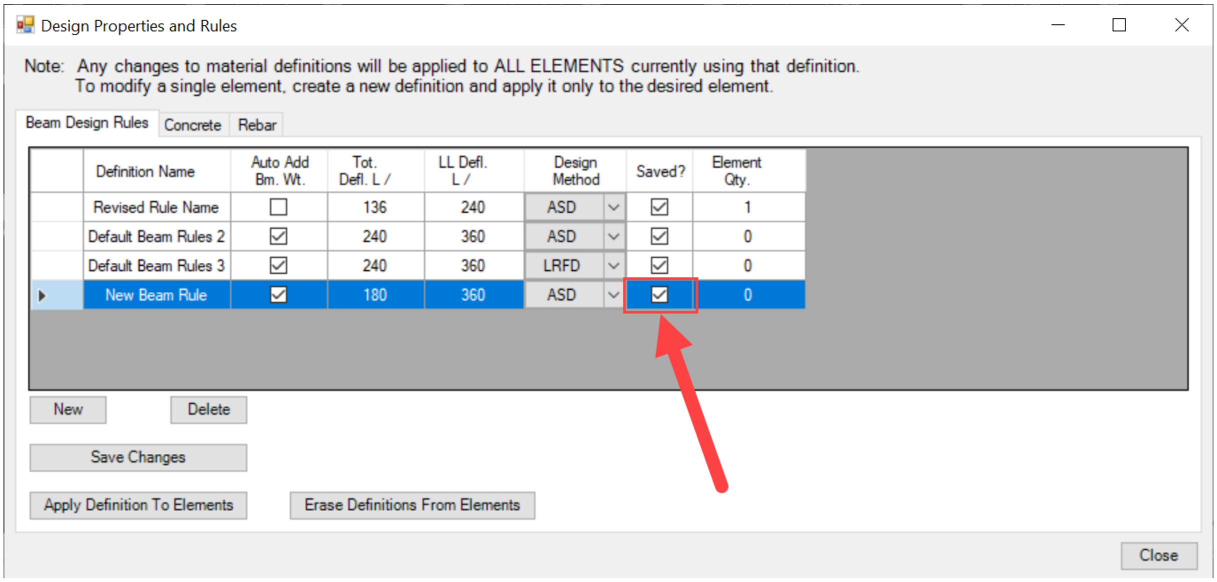


The "New" button will generate a new row in the table. The user may name and modify this new definition as described in Modifying Rules 333.



When a new rule is created and modified, it will not be permanently added to the project until saved using the "Save Changes" button. Closing the menu before saving will cause the new rule to be discarded.

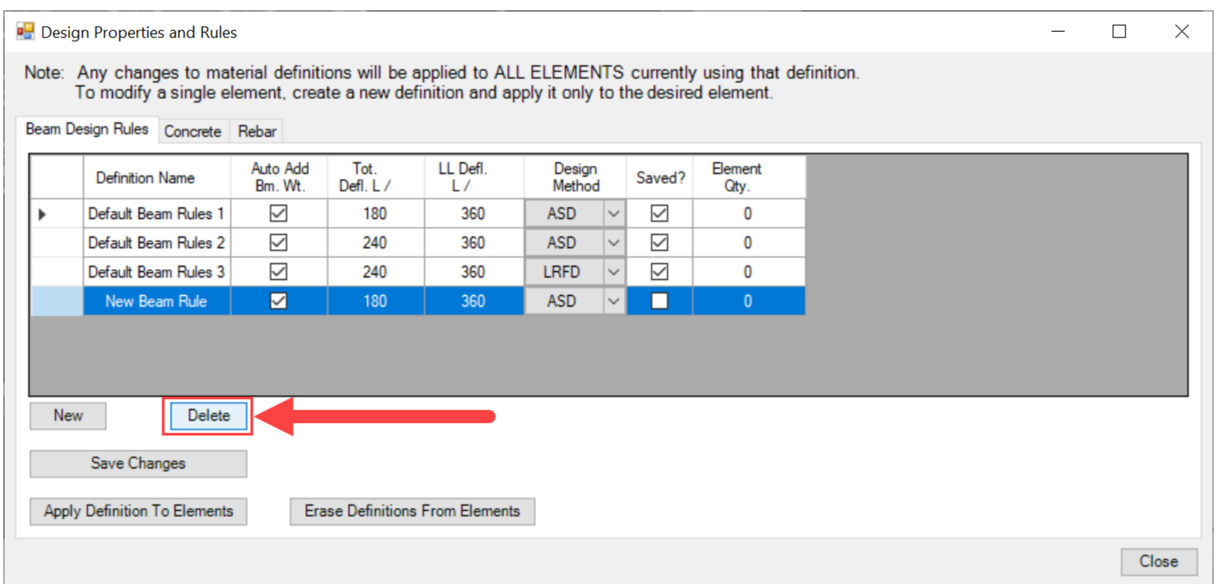




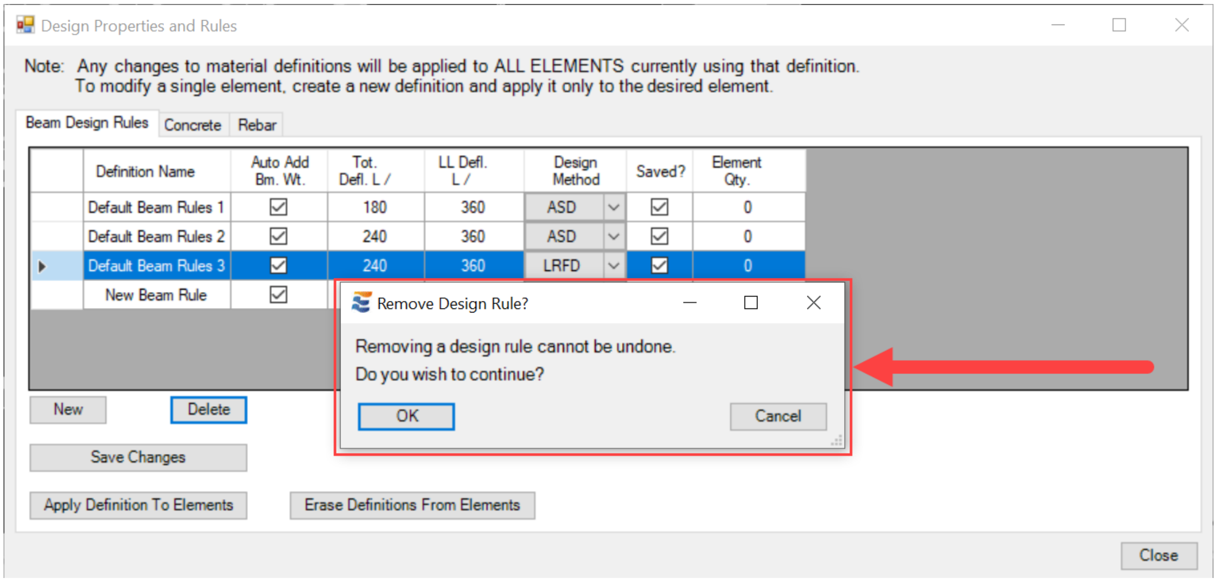
A newly added beam design rule does not have any influence over the behavior of beam calculations until it is manually applied to specific elements by the user (refer to Applying Rules to Elements 330).

10.5.4 Deleting Rule Definitions from a Project

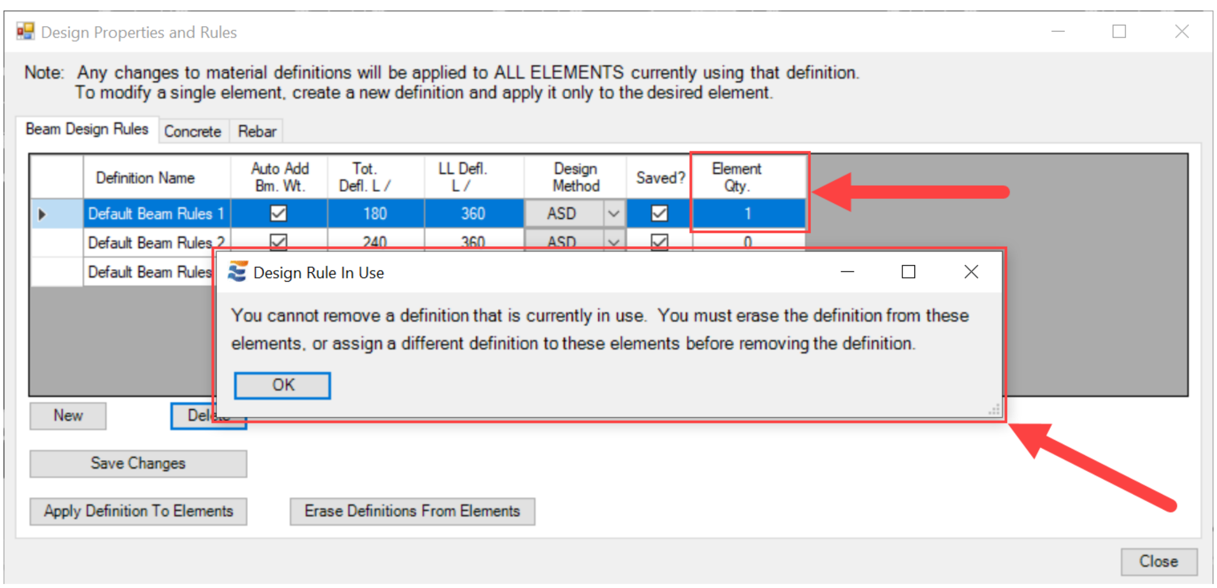
Beam design rule definitions may be removed using the "Delete" button. Using this button will remove the currently selected definition.



If the currently selected definition is newly added to the table during the current menu session and has never been saved, then the "Delete" button will remove it immediately without any additional confirmation required. However, if the definition has previously been saved, then the "Delete" button will trigger a warning requiring confirmation before removal:



If the currently selected definition has already been applied to one or more elements, the user will be warned that definitions cannot be removed while in use:

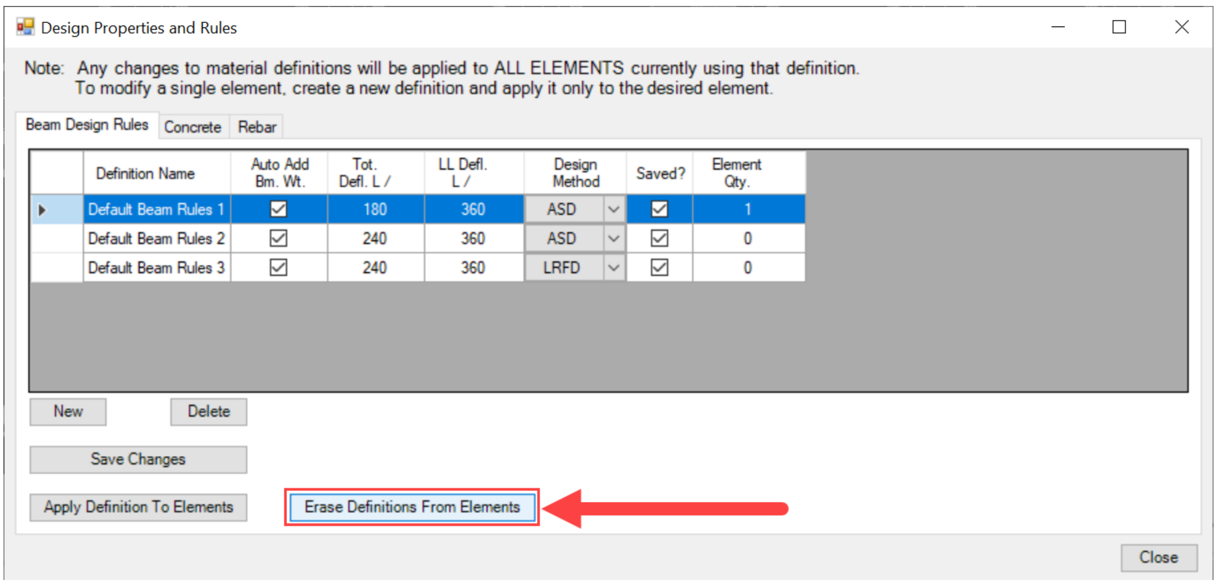


The elements in question may be assigned a different rule as described in Applying Rules to Elements 330, or the rule may be erased from the elements, as described in Erasing Rules From Elements 341. Once the definition is no longer assigned to any elements, it may be removed.

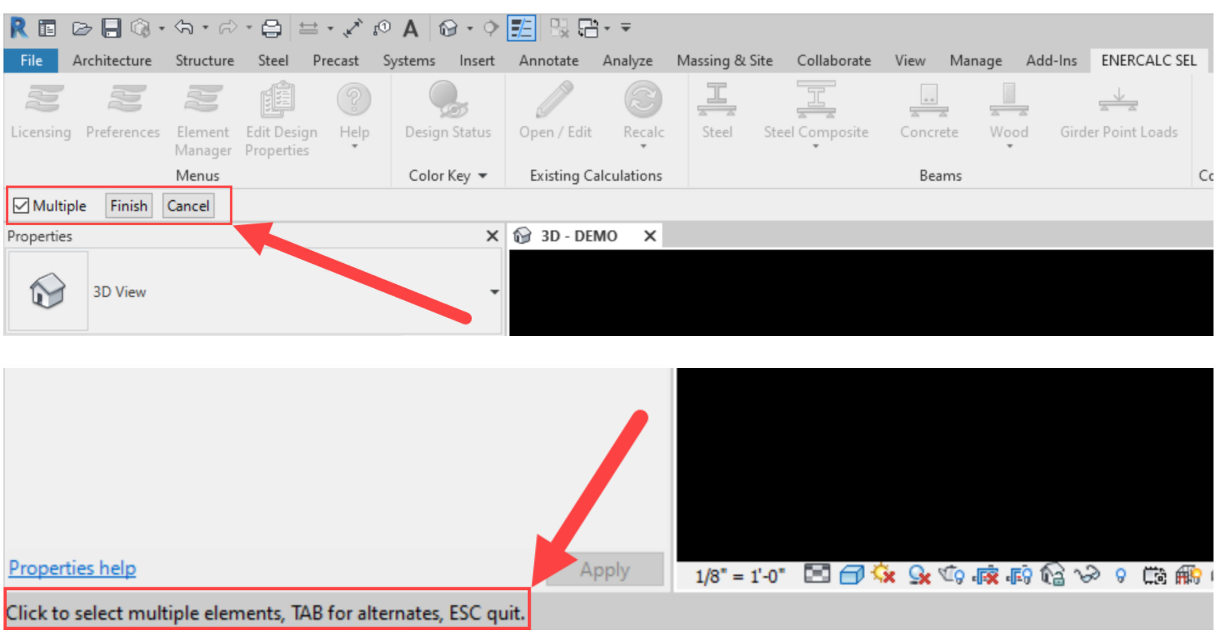
10.5.5 Erasing Rule Definitions From Elements

Rules may be selectively erased from the elements they were previously assigned to using the "Erase Definitions From Elements" button.

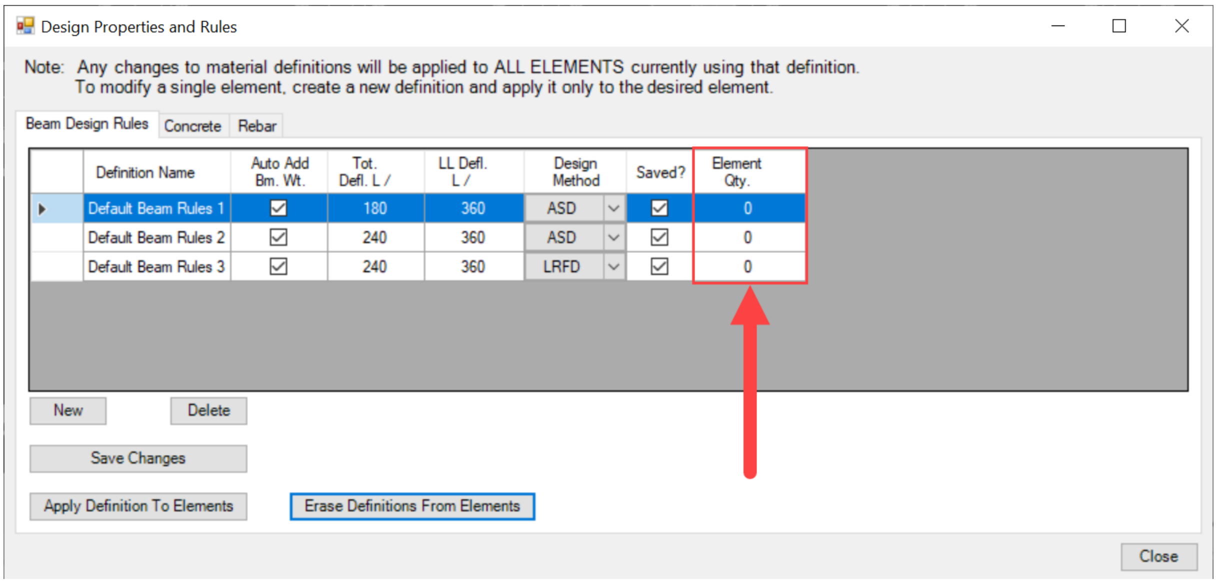
Users should take note that the "Erase Definitions" control is independent of the current table row selection. Using this command will erase **ANY** beam design rule from the elements the user picks.



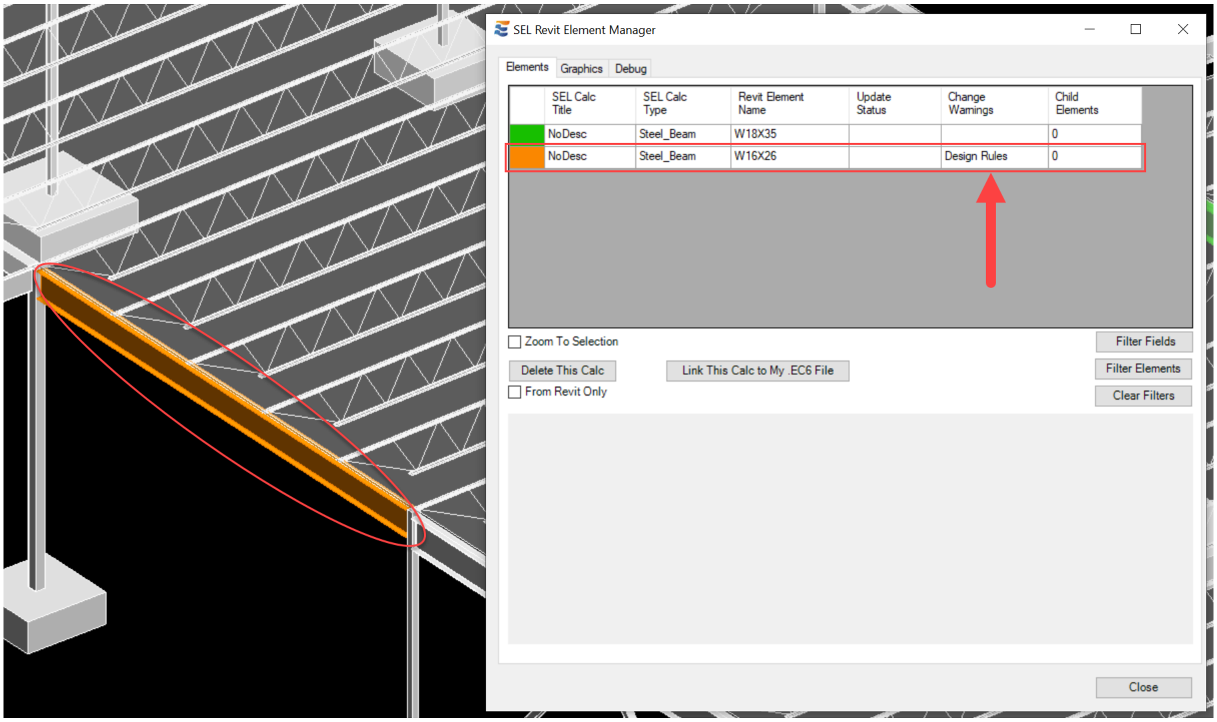
Similar to the "Apply" control, the "Erase" control will trigger a multi-select process and await element selection by the user:



When the user clicks "Finish" to complete the selection operation, all selected elements will have their beam design rule erased, regardless of which rule is highlighted in the table. The "Element Qty." column will then automatically update to reflect the fact that some elements have had their beam design rule erased:



Similar to the case of modifying rules, there is a risk of impact to existing calculations when rules are erased from elements. As a result, any existing beam calculation whose beam design rule was erased will automatically be placed in a warning state. This is evident via both color-coded highlighting in the Revit view and via the Element Manager:



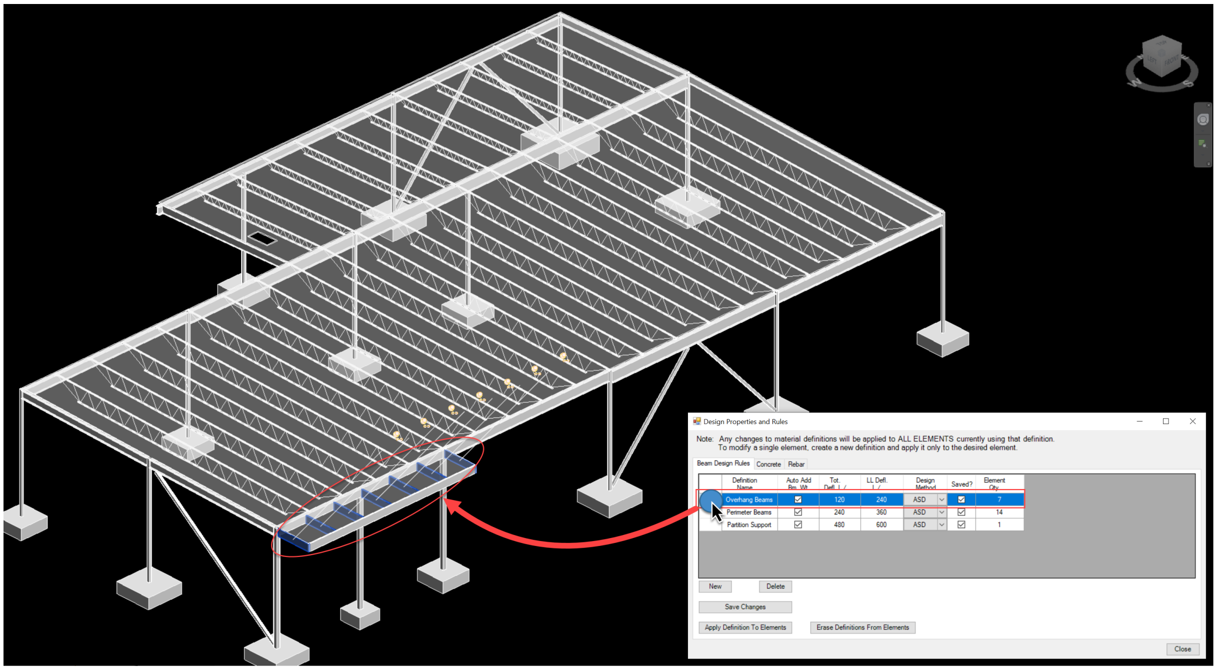
Any beam calculation that is subsequently relaunched after its design rules have been erased will be launched with the following default rules:

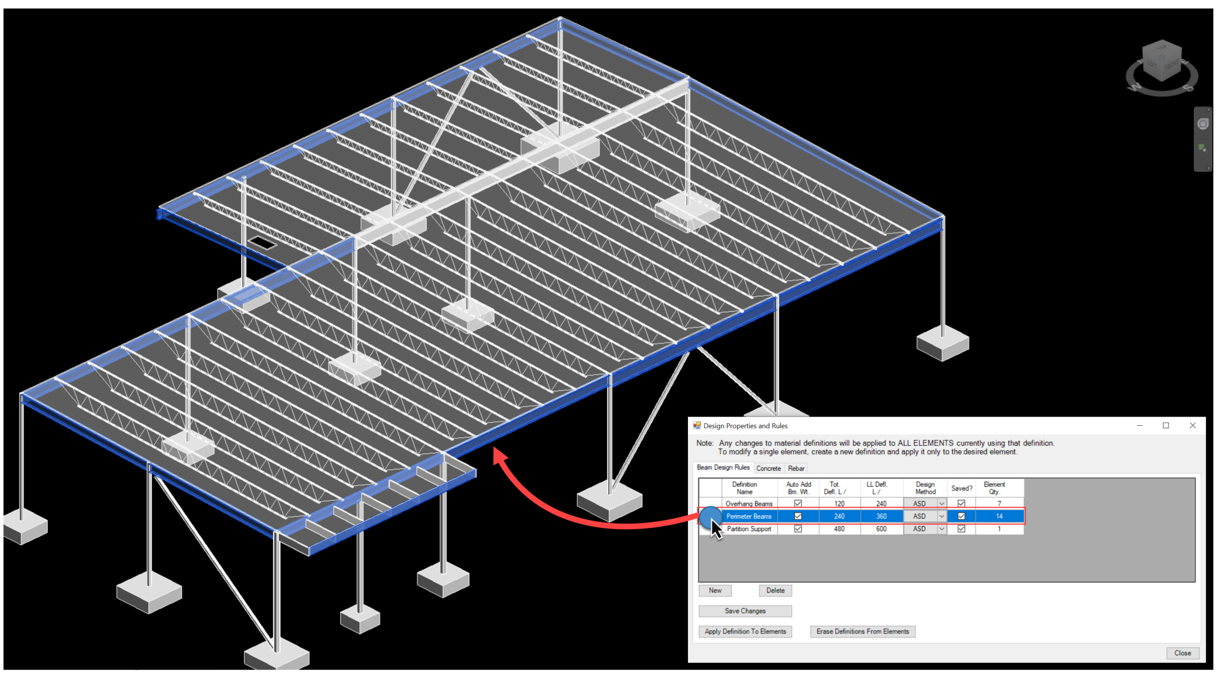
- Auto add beam weight = true
- Deflection ratio - Total = 180
- Deflection ratio - Live Load = 240
- Design method = ASD

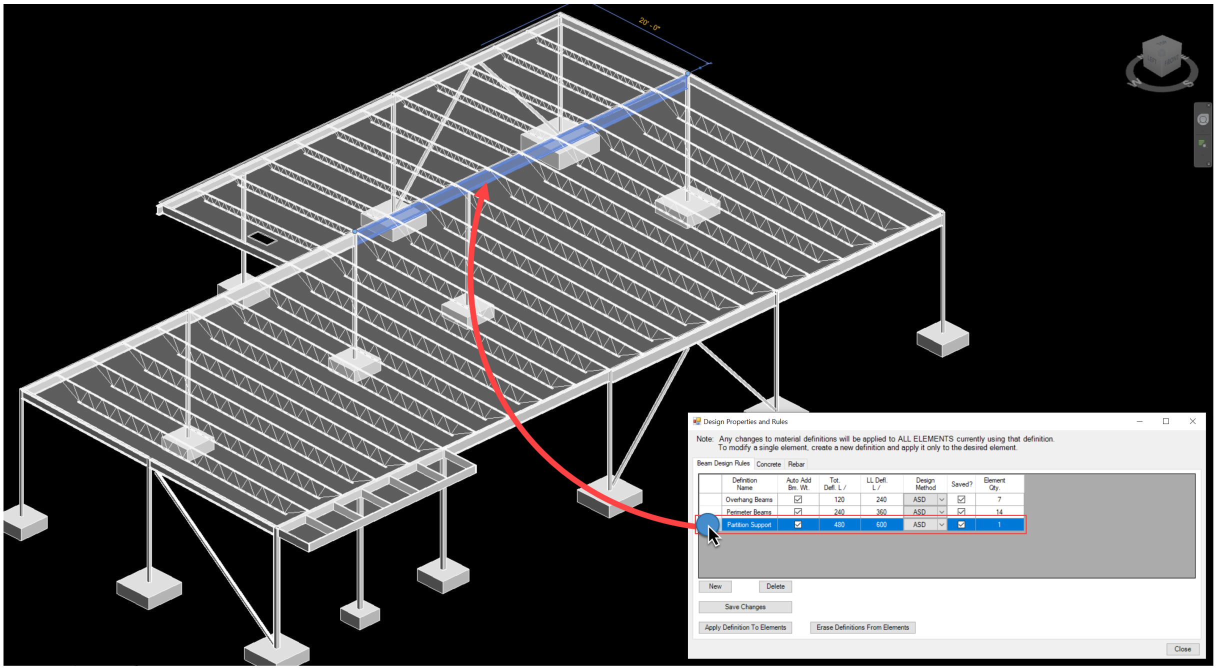
10.5.6 Navigating Rule Definitions

Once beam design rules have been defined and applied to various elements throughout the Revit model, the assignments can be conveniently reviewed from the "Design Properties and Rules" menu. Similar to the selection behavior of the Element Manager, this menu provides users with a bi-directional selection workflow.

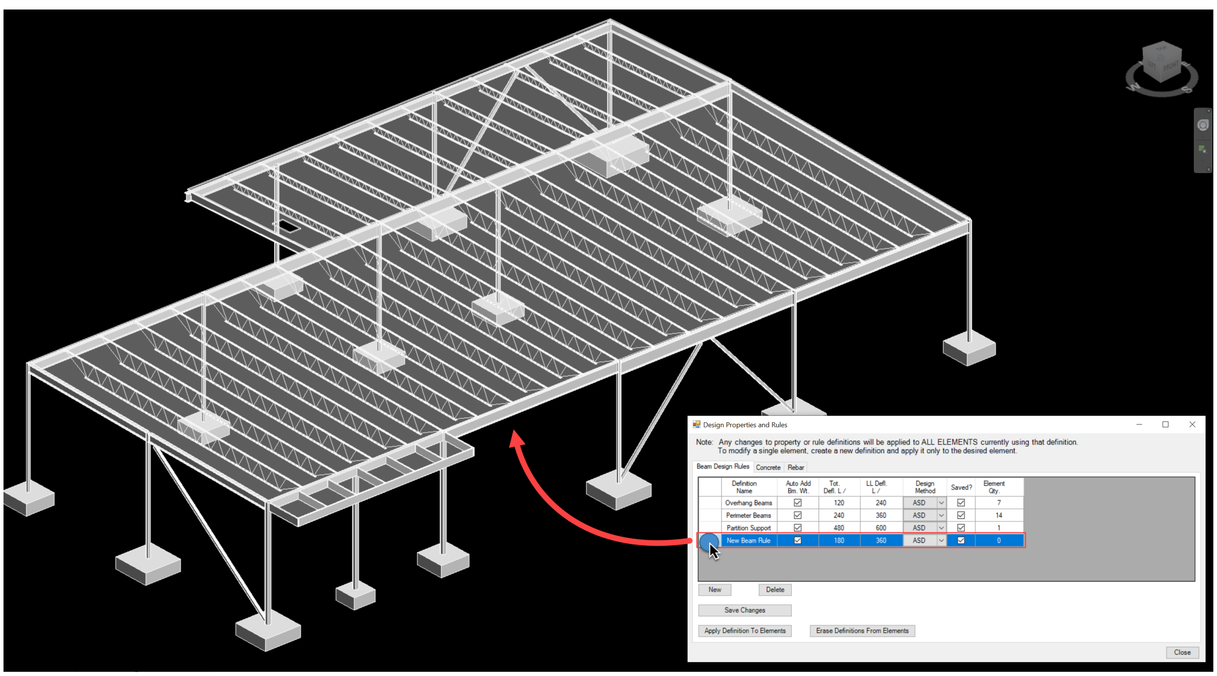
Selecting a definition in the design rule table will automatically select all elements in the Revit model that carry the selected definition:


Design Properties and Rules
Note: Any changes to material definitions will be applied to ALL ELEMENTS currently using that definition.
To modify a single element, create a new definition and apply it only to the desired element.
Beam Design Rules
Concrete
Rebar
Definition Name
Auto Add Bm. Wt.
Tot. Defl. L /
LL Defl. L /
Design Method
Saved?
Element Qty.
Overhang Beams 120 240 ASD 7
Perimeter Beams 240 360 ASD 14
Partition Support 480 600 ASD 1
New
Delete
Save Changes
Apply Definition To Elements
Erase Definitions From Elements
Close

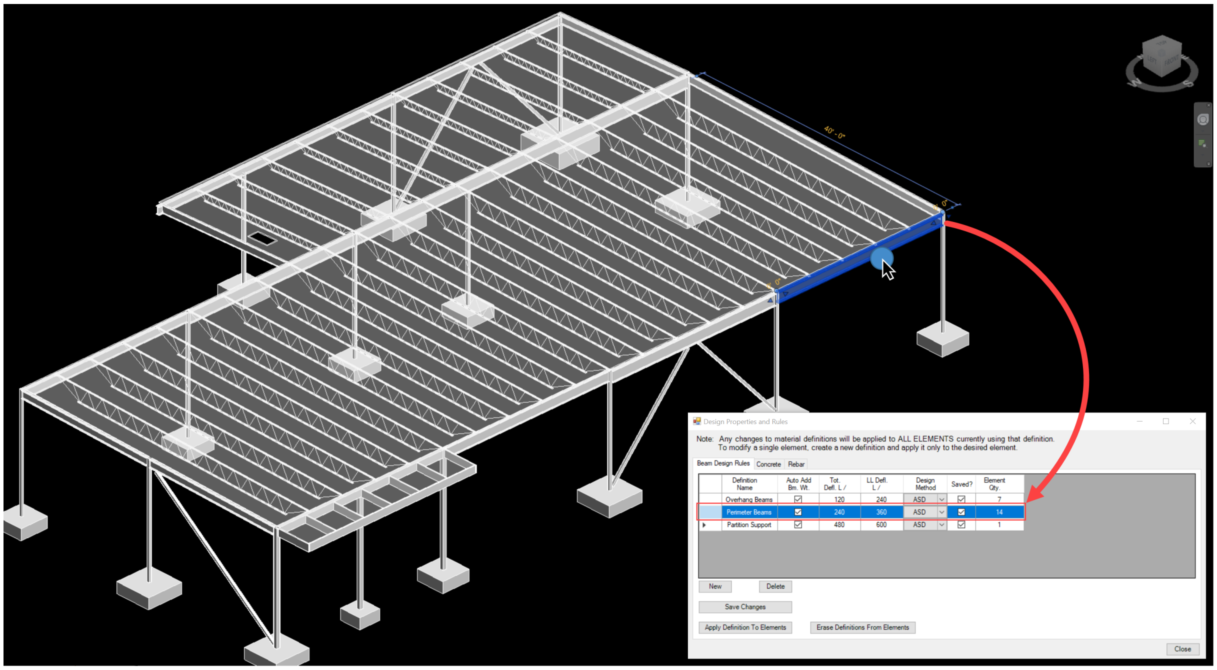

Design Properties and Rules
Note: Any changes to material definitions will be applied to ALL ELEMENTS currently using that definition.
To modify a single element, create a new definition and apply it only to the desired element.
Beam Design Rules
Concrete
Rebar
Definition Name
Auto Add Bm. Wt.
Tot. Defl. L /
LL Defl. L /
Design Method
Saved?
Element Qty.
Overhang Beams 120 240 ASD 7
Perimeter Beams 240 360 ASD 14
Partition Support 480 600 ASD 1
New
Delete
Save Changes
Apply Definition To Elements
Erase Definitions From Elements
Close

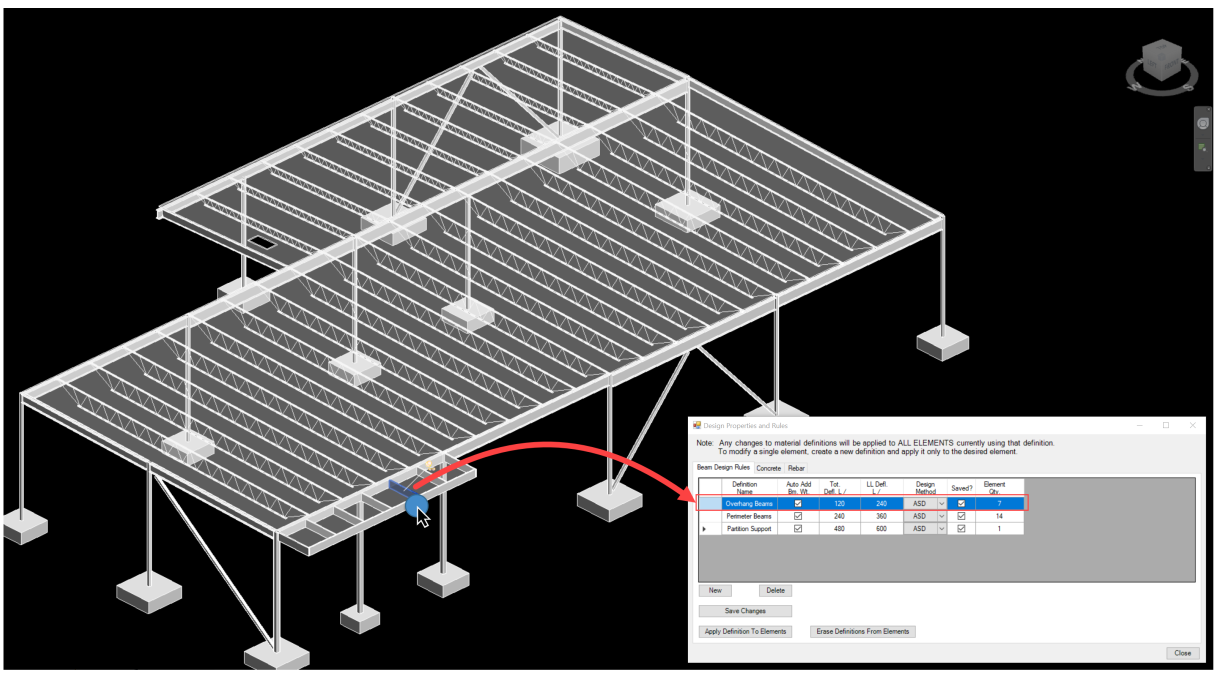


Clicking a definition row in the table that has not been assigned to any elements will not cause any selection in the Revit model:

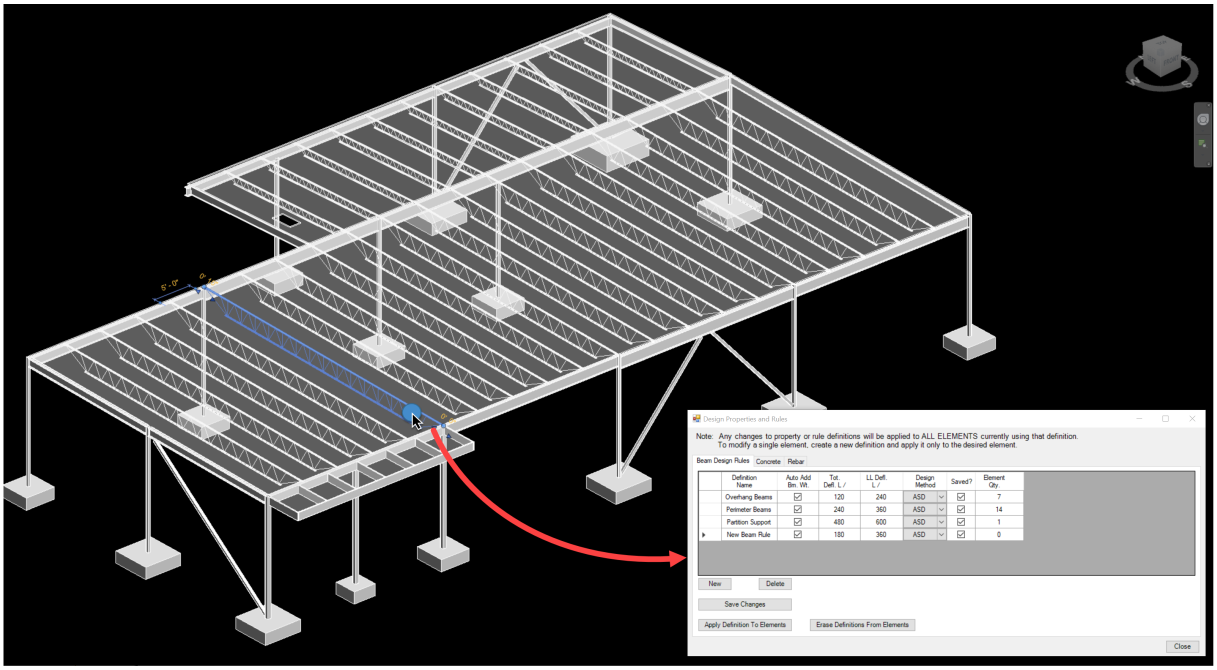


Conversely, clicking a single element in the Revit view will automatically select the table row of the definition assigned to the element:



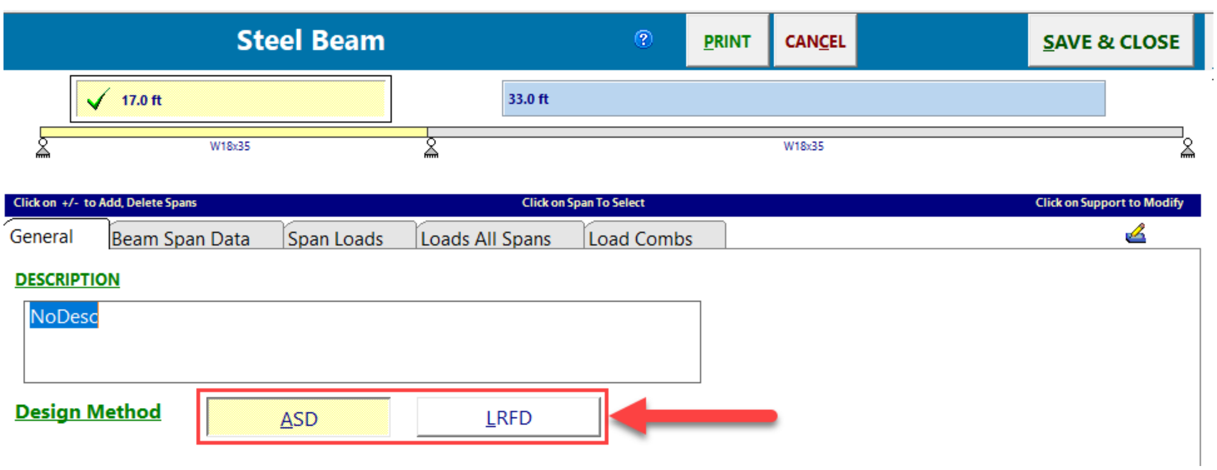


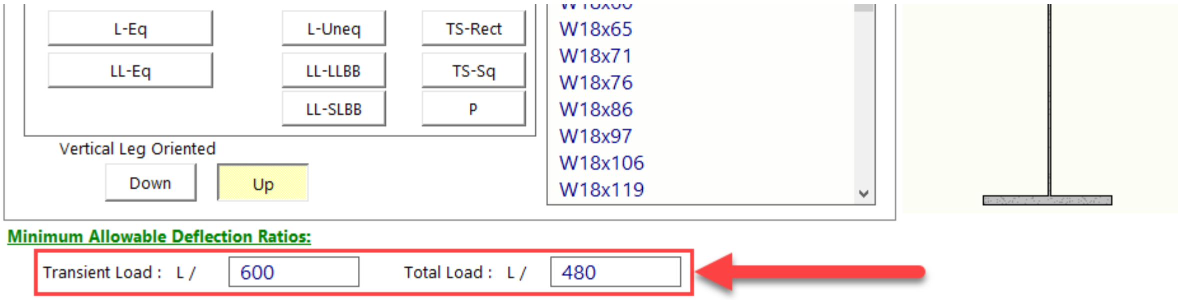
Clicking an element that has no definition assigned to it will not result in any table row selection:

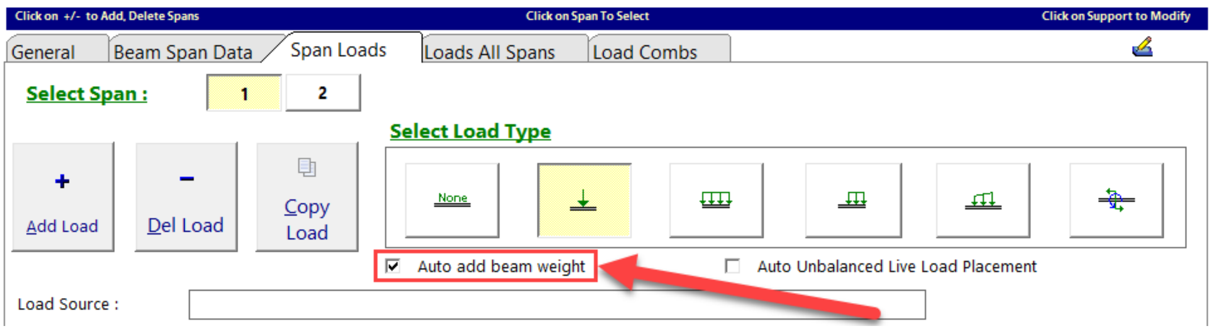


10.5.7 Changing Design Rules in ENERCALC

Once a beam calculation has loaded in the ENERCALC interface, the various design properties driven by beam design rules are exposed to view and modification by the user in various locations. The precise location of these controls may vary from one beam module to the next.







If any of these values are manually modified in the ENERCALC interface, the changes will impact ONLY the current beam element. Changes made to an individual beam's design rules within ENERCALC will NOT propagate to other beams designed using the same rule definition. If the user does wish to modify many beams at once, refer instead to Modifying Rules 333. Once beam design rule values have been changed in a particular calculation, the new values will be applied to the Revit model in one of two ways:

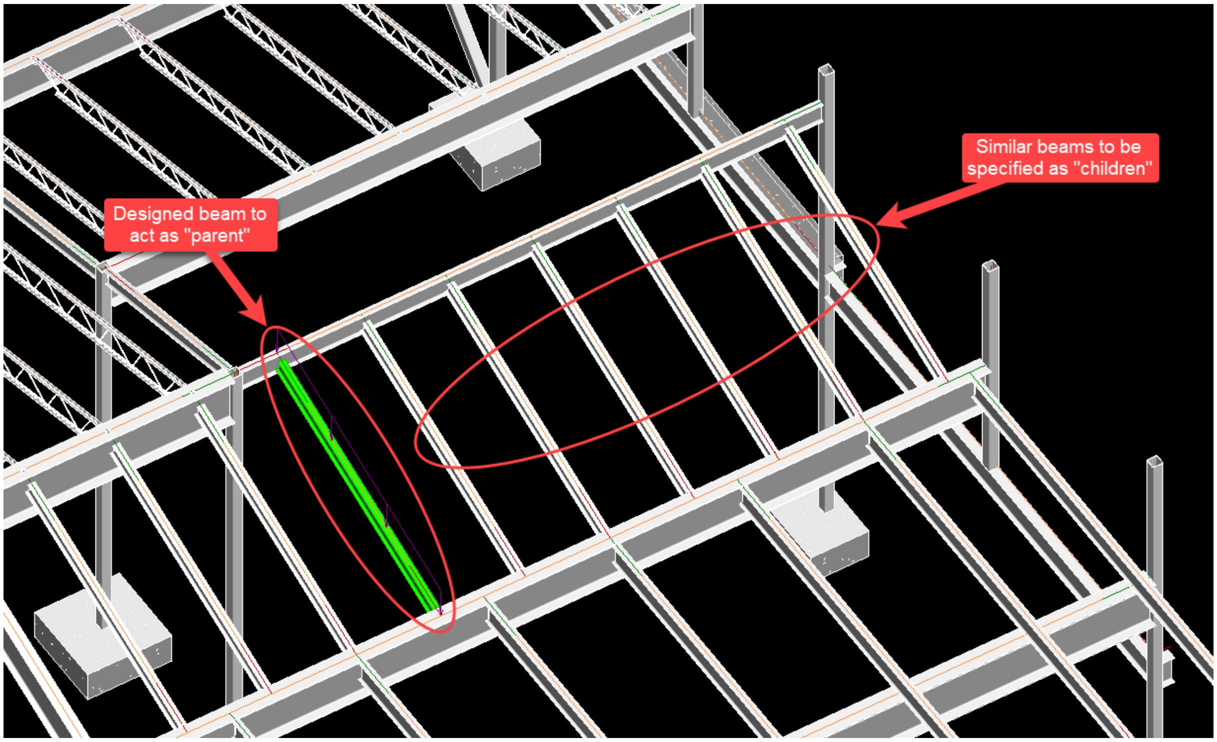
1.) If any existing rule definition found in the Revit project has properties that match the revised settings on the beam, then the matching definition will be found and applied to the Revit beam element on "Save and Close". The rule definition previously assigned to the beam will NOT be modified, and no other Revit beam elements will be affected by the rule change.

2.) If there is no existing rule definition found in the Revit project with properties that match the values assigned in ENERCALC, then a new rule definition will be created and applied to the Revit beam element on "Save and Close". The rule definition previously assigned to the beam will NOT be modified, and no other Revit beam elements will be affected by the newly created rule.

10.6 Beam Parent / Child Relationships

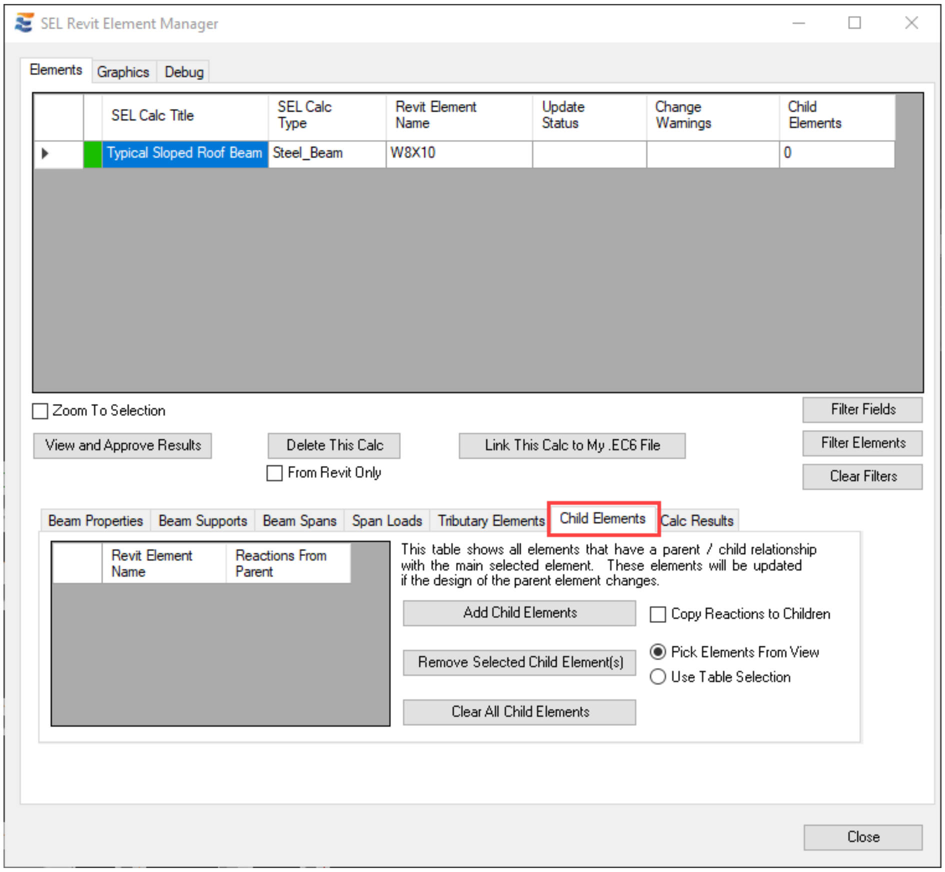
Parent / Child relationship describes the ability for the user to manage the design of many Revit elements with a single ENERCALC SEL calculation. This is most commonly useful in situations where a framing system contains multiple elements with similar geometry and similar structural function leading to similar expected loads. Parent / child allows multiple elements (i.e., children) to be manually associated to a particular controlling calculation (i.e., the parent). Once the association is made, the child beams will automatically update to imitate design updates performed on the parent beam.

In the example below, a single typical beam has been loaded and designed. In the adjacent framing system, 6 additional beams have identical geometry and based on layout will experience similar loading.

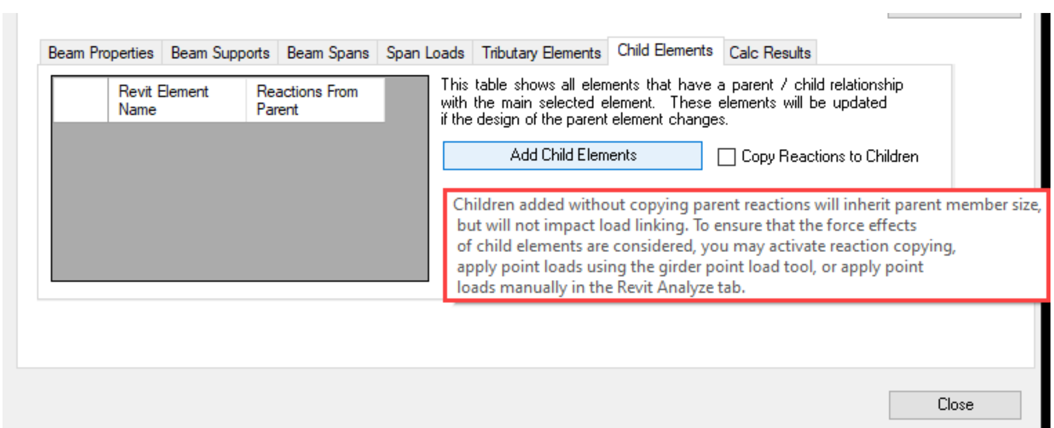


Parent/child relationships are created and managed from the Element Manager by accessing the calculation of the parent (or intended parent) element. When the Element Manager window is open, selecting the parent beam from the Revit UI or from the Element Manager summary table will open the details of the calculation in the lower half of the window. Parent/child controls are found on the "Child Elements" tab.

The identification of parent/child relationships is entirely at the discretion of the user. There is no attempt made by the program to automate the identification of potential parent members or children, except in the case of a Revit beam system. For more information on this condition, refer to "Working With Beam Systems 364".



If the selected beam already has child elements, they will be displayed in the table. If not, the table will be empty. From this interface, the user has the option to add new children, remove specific children, or clear all children. When adding new children, there is also an option to store reactions on the child elements. This option is enabled via the “Copy Reactions to Children” checkbox and will allow the child elements to exert “Load-Linking” reaction forces on their supporting element without the need for explicit analysis of each individual child beam by imitating the reactions of the parent element. More detail is available in a tooltip balloon that appears when the cursor hovers over the “Add Child Elements” button.

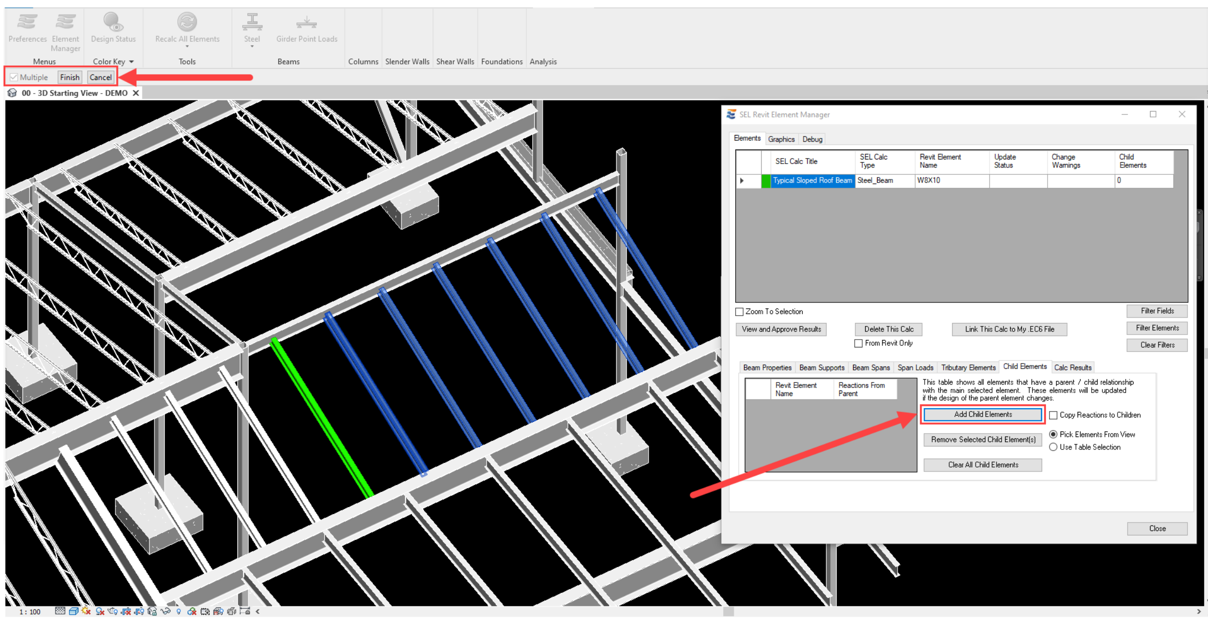


10.6.1 Adding Child Elements

NOTE: For parent/child management, the terms "Add" and "Remove" describe the creation or removal of a conceptual relationship between the parent element and another existing element in the Revit model. These controls do not create or delete actual elements in the Revit model or calculations in the .EC6 file.

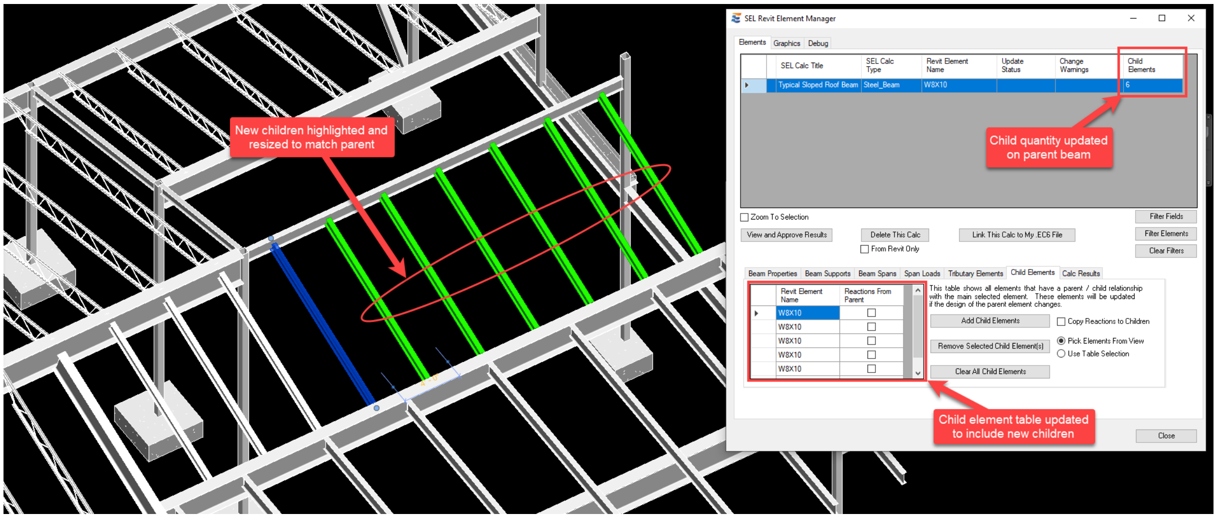
If the user intends to take advantage of "Load-Linking" from child elements to supports, the "Copy Reactions to Children" checkbox must be activated before starting the "Add" operation. Otherwise, the children will be added without any reaction force linking behavior.

Clicking "Add Child Elements" will activate a Revit multi-select operation. The program will await user selection with "Finish" and "Cancel" buttons on the ribbon bar. As with all other multi-select operations, the cursor will display a "+" symbol to add new elements to the selection and a "-" to remove elements from the selection. When all desired child elements are selected, clicking the "Finish" button on the ribbon will complete the operation, while "Cancel" will halt the operation and no children will be added.



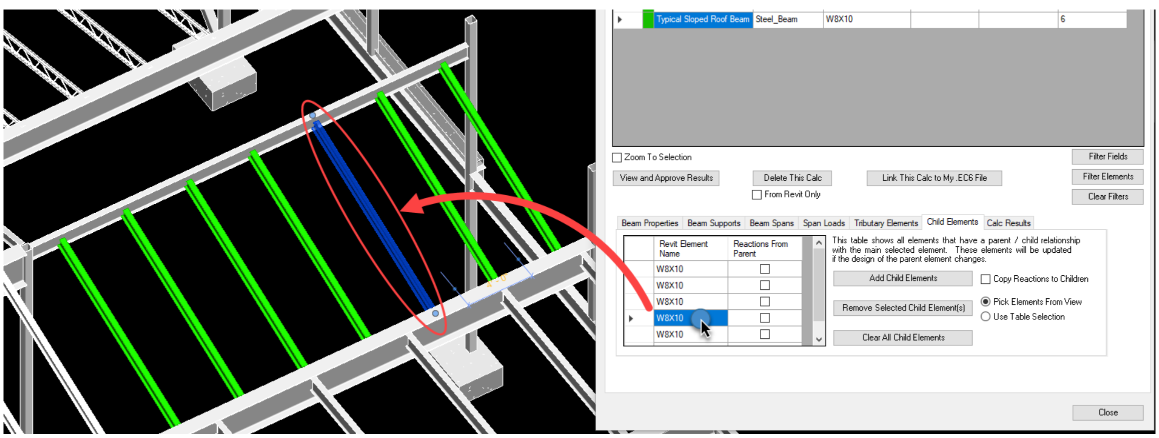
When the "Finish" button is clicked, the following changes will occur immediately:

1. The section size of all children will be updated to match the parent size
2. The table of child elements will be updated to show all specified children
3. Parent calculation's line item in the Element Manager summary table will be updated to show the current quantity of child elements.
4. If status highlighting is enabled, all newly added child elements will be color highlighted to match the parent element, indicating that they are now associated with a design calculation.

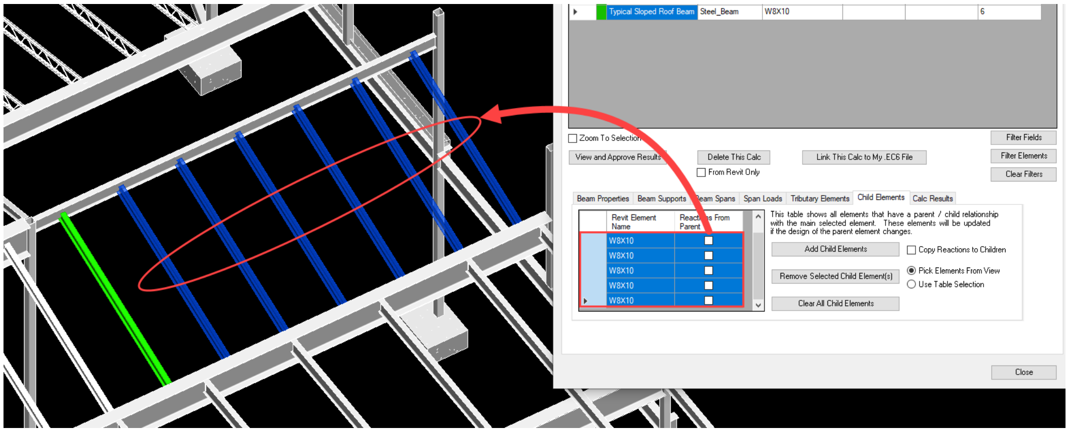


10.6.2 Navigating Child Elements

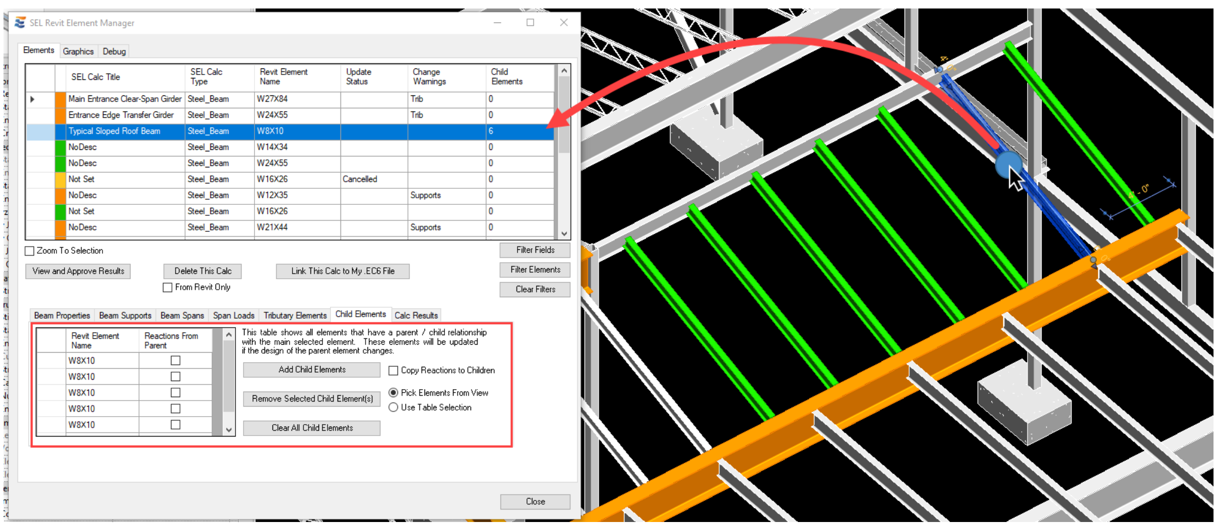
Once the selected children have been added, they may be viewed or navigated at any time by clicking row by row in the child element summary table. Clicking a particular child element in the table will automatically select it in the Revit model.



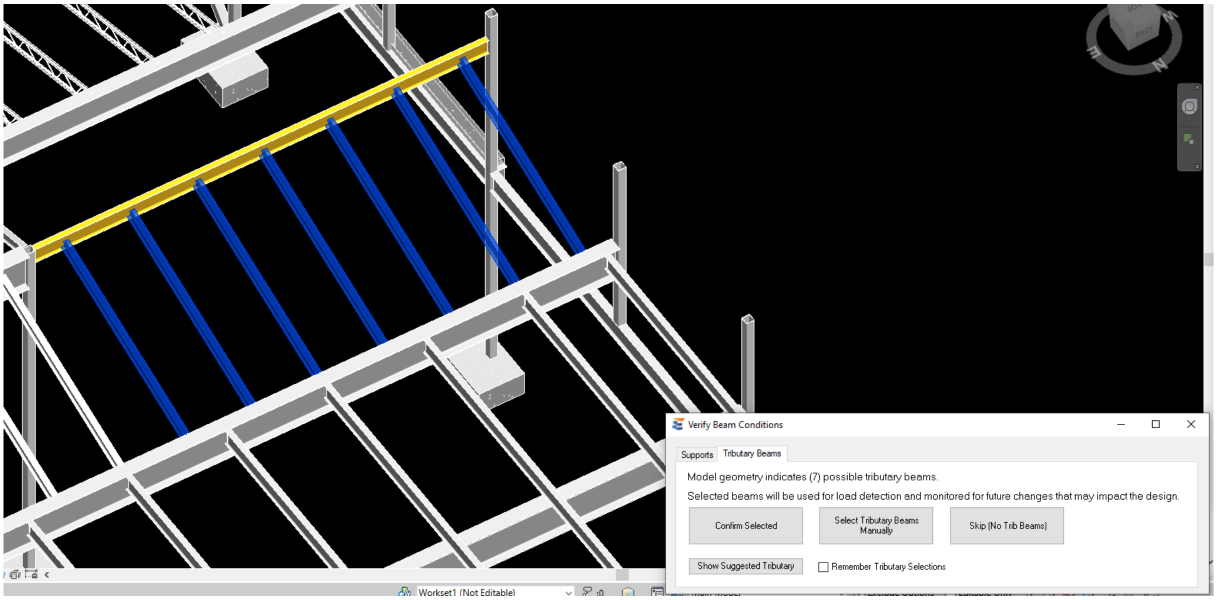
Selecting either multiple rows or all rows via Shift+Click in the table will select all applicable elements in the Revit model.



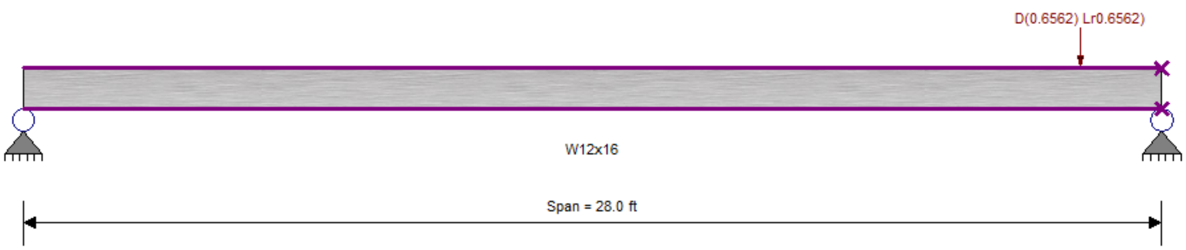
In addition to navigating children directly from the child elements table, picking any child element in the Revit view while the Element Manager is open will automatically select the parent element in the summary table and redirect to the “Child Elements” tab.

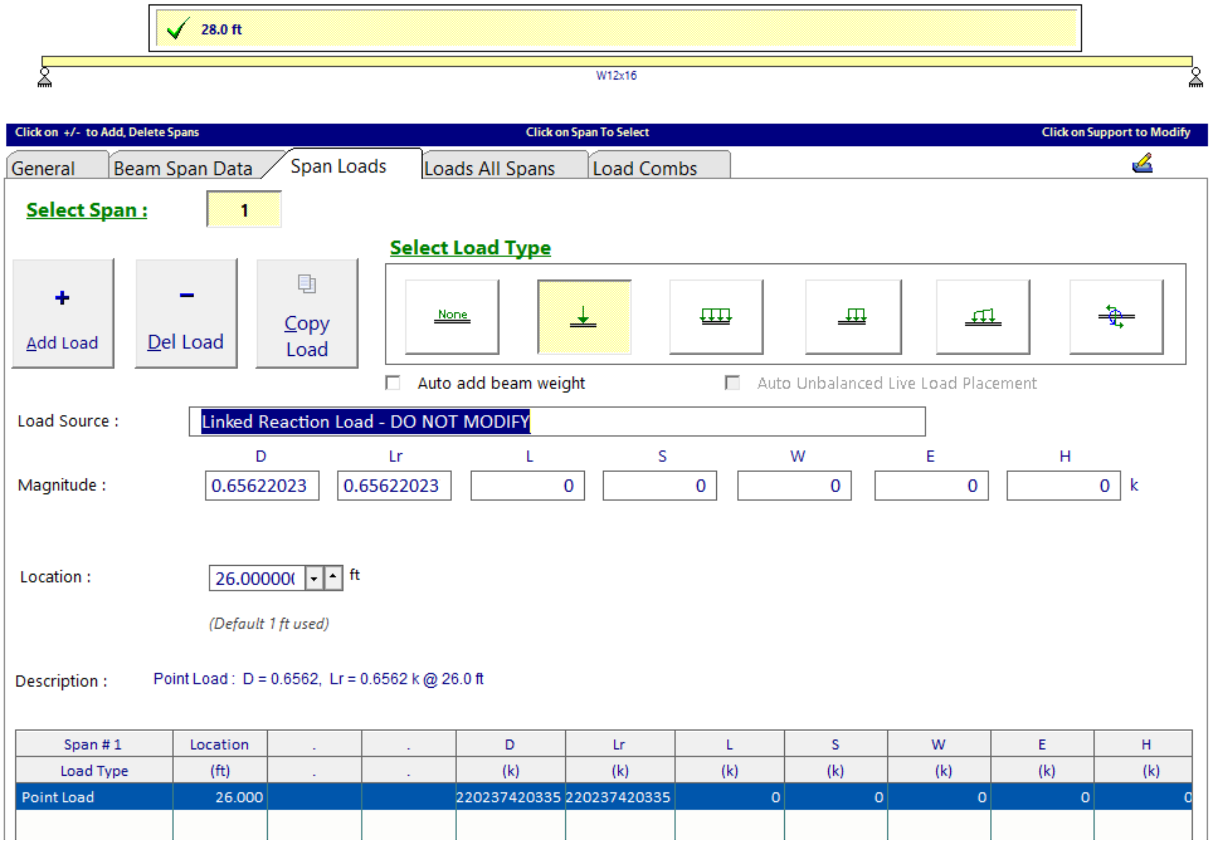


Note also that once the child elements table updates, it indicates in the right-hand column (“Reactions From Parent”) whether the parent’s reactions are copied to each individual child. If the reactions are not copied, then the child elements will not exert any “Load-Linking” forces on other elements in the load path. This may be observed by launching a beam calculation for the supporting girder.



The girder calculation will arrive in the ENERCALC SEL interface with “Load-Linked” reactions from the parent beam, but not from the children.

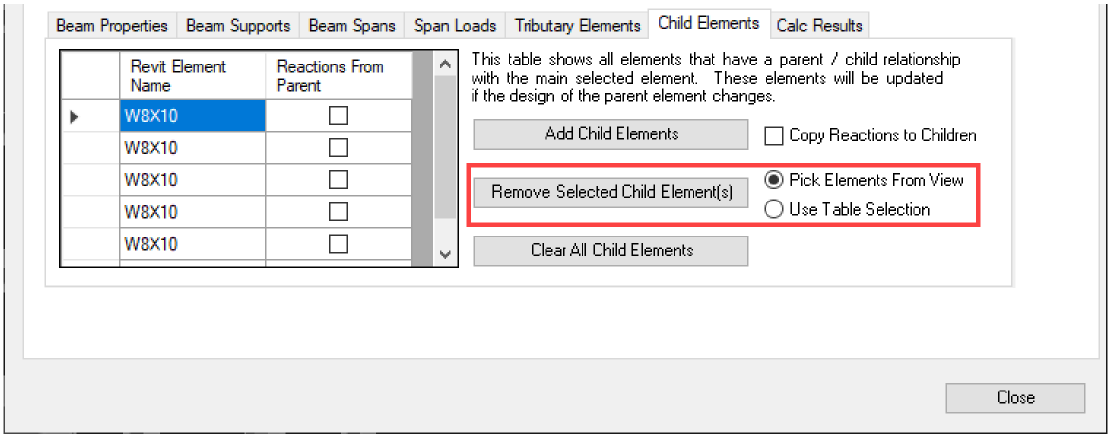


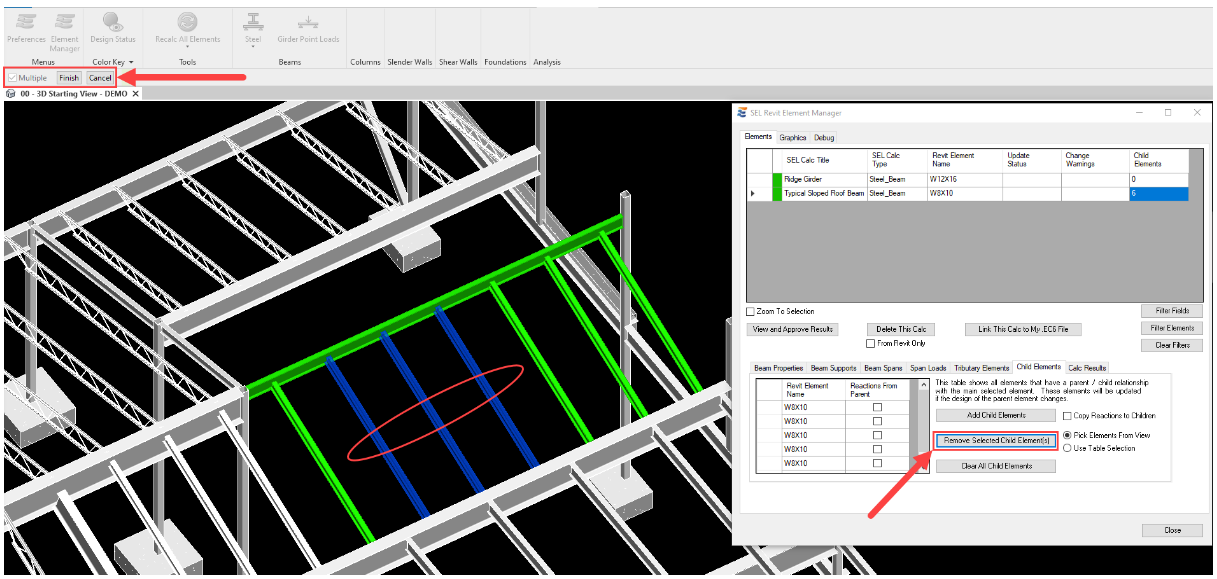


10.6.3 Removing Child Elements

NOTE: For parent/child management, the terms “Add” and “Remove” describe the creation or removal of a conceptual relationship between the parent element and another existing element in the Revit model. These controls do not create or delete actual elements in the Revit model or calculations in the .EC6 file.

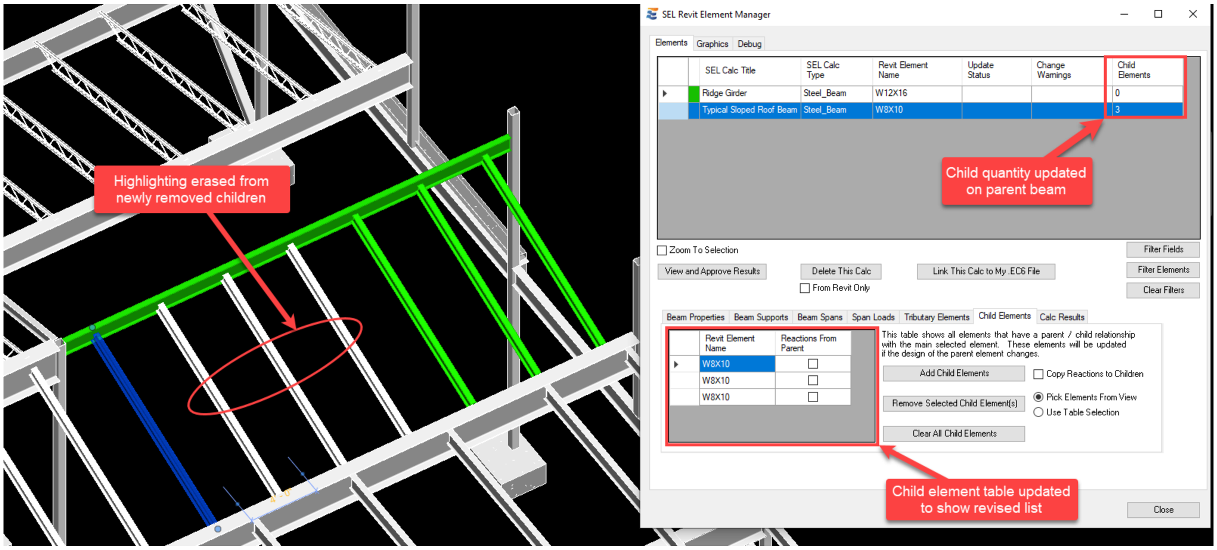
Child elements may be removed in a manner similar to the “Add” operation. The “Remove Selected” button has two different options next to it. The default option labeled “Pick Elements From View” will prompt the user to manually pick which children to remove via Revit UI just like the “Add” process. Checking the “Use Table Selection” option instead will directly remove any child elements currently selected in the child elements table.





If the operation is cancelled, no children will be removed. If it is finished, the following changes will occur immediately:

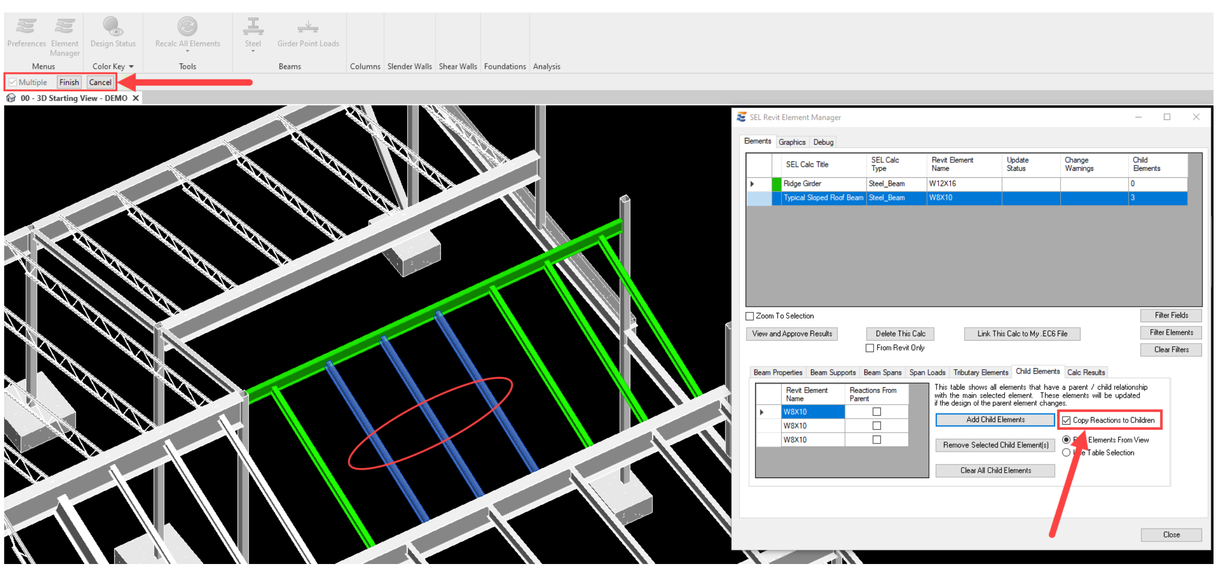
1. The table of child elements will be updated to show the revised list of children
2. Parent calculation's line item in the Element Manager summary table will be updated to show the revised quantity of child elements.
3. If status highlighting is enabled, highlighting will be erased from all removed child elements to show that they are no longer associated with a design calculation.

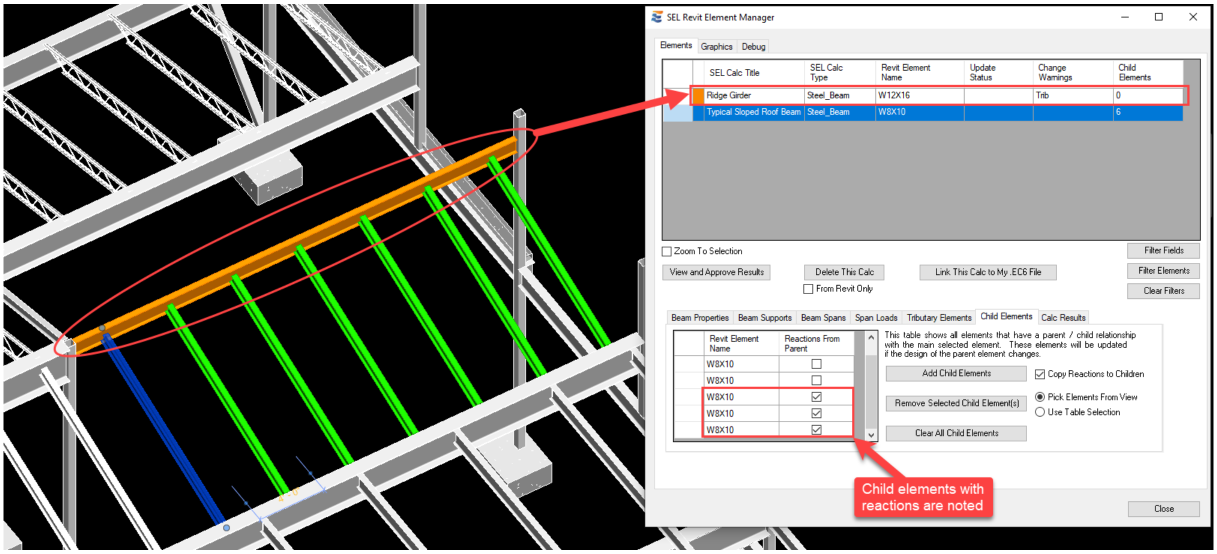


10.6.4 Change Warnings

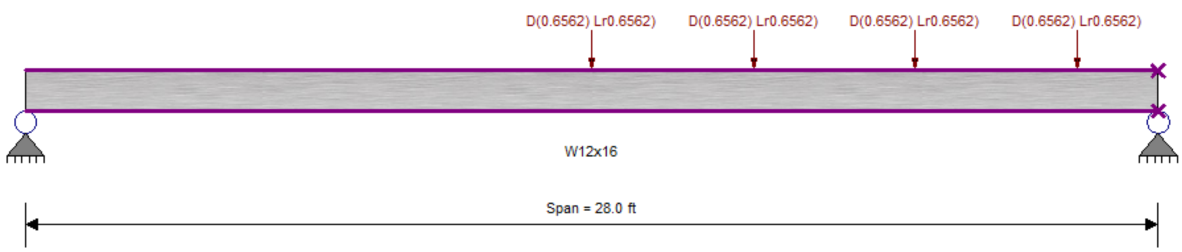
NOTE: For parent/child management, the terms “Add” and “Remove” describe the creation or removal of a conceptual relationship between the parent element and another existing element in the Revit model. These controls do not create or delete actual elements in the Revit model or calculations in the .EC6 file.

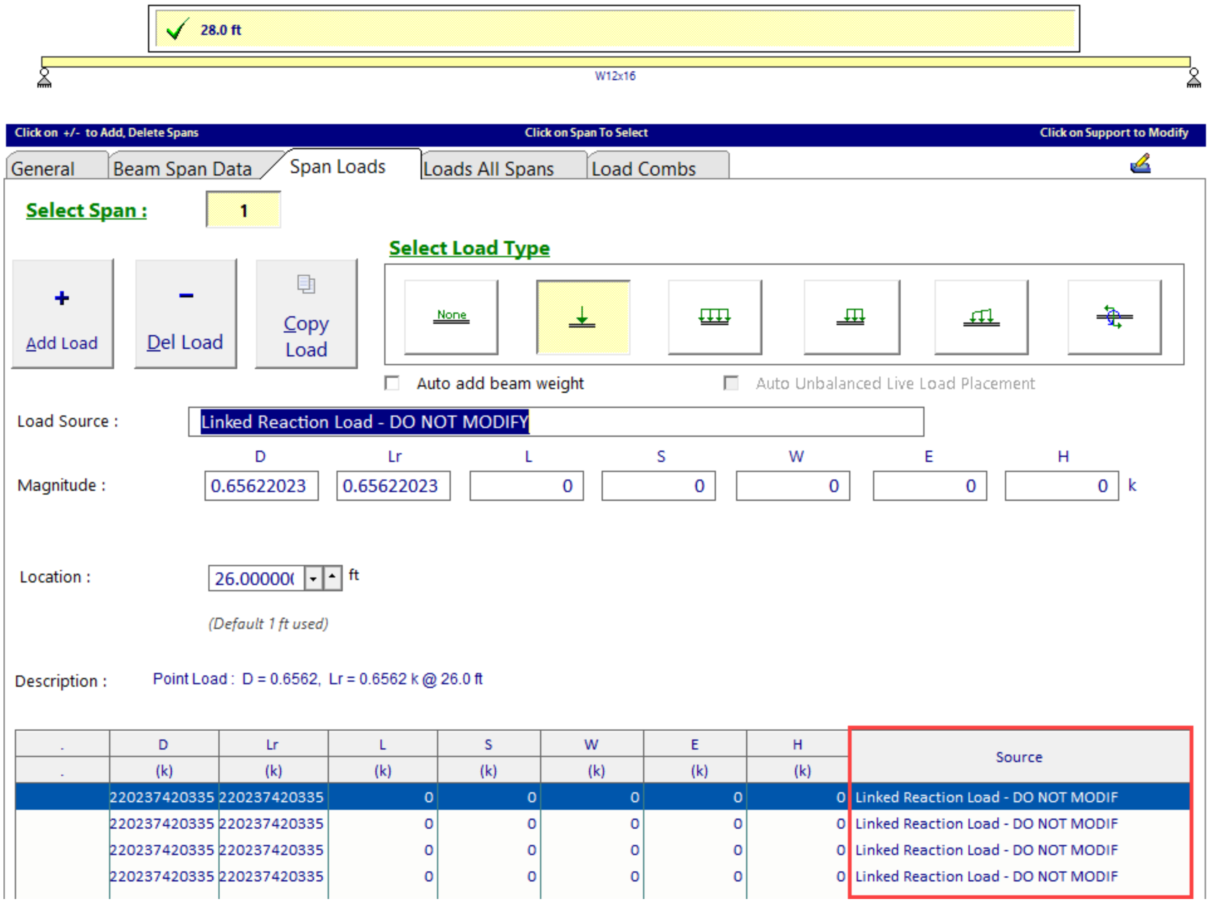
As an added benefit for the design engineer, load path warnings are produced automatically when the addition of child elements WITH copied reactions impacts the behavior of an existing calculation. The supporting girder designed above is an example of this. If the three child beams removed above are added back with copied reactions from the parent, the supporting girder will toggle to a warning state (orange) because new load has been added that was not reflected in the previous run of the SEL calculation.





After the addition of new child beams that include reactions copied from the parent, re-launching the girder calculation will allow the user to directly view the reactions.

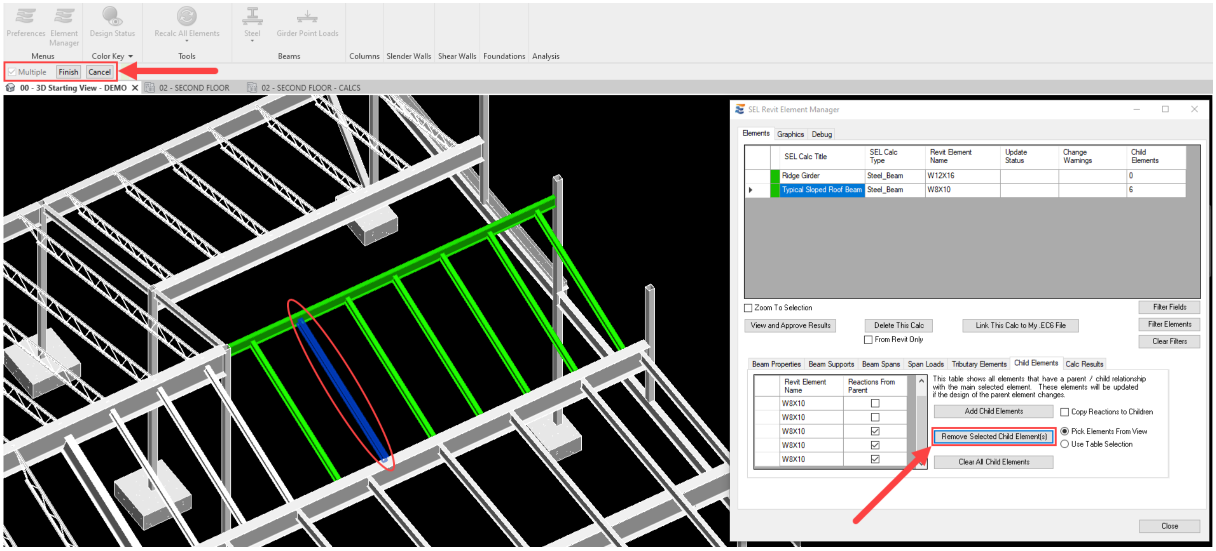


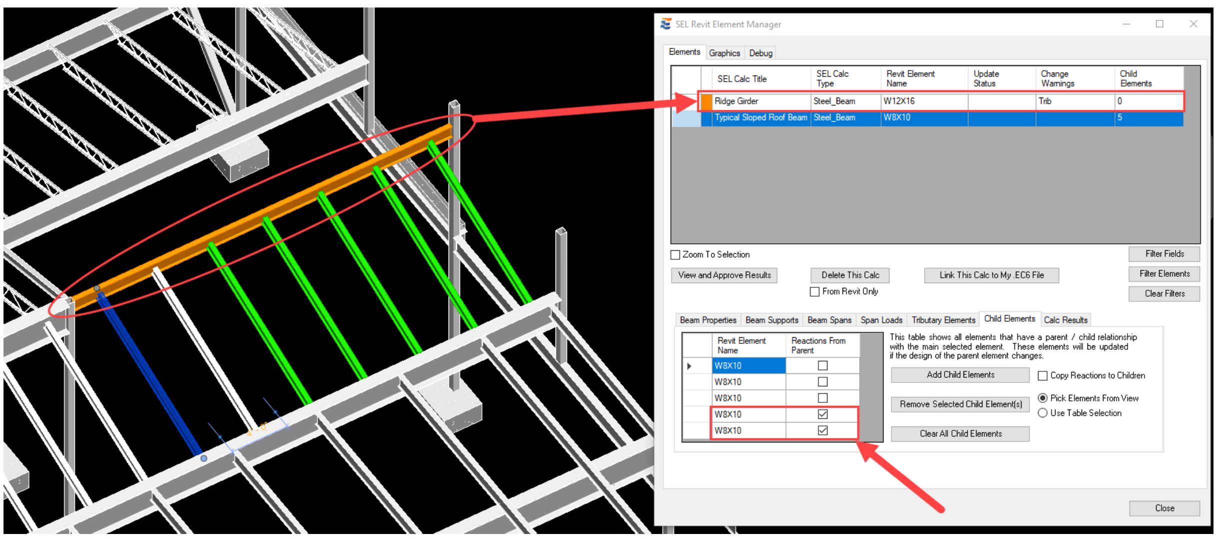


10.6.4.1 Change Warnings from Removal of Child Elements

NOTE: For parent/child management, the terms "Add" and "Remove" describe the creation or removal of a conceptual relationship between the parent element and another existing element in the Revit model. These controls do not create or delete actual elements in the Revit model or calculations in the .EC6 file.

Once a girder has been designed using the reactions exerted by "Load-Linked" child elements, it will also be subject to warnings in the future if one of the parent/child relationships is removed. Removal of a child that does not carry reactions has no effect, but removal of a "Load-Linked" child will trigger a warning state (orange) on the girder similar to that caused by adding a new child, as shown previously.

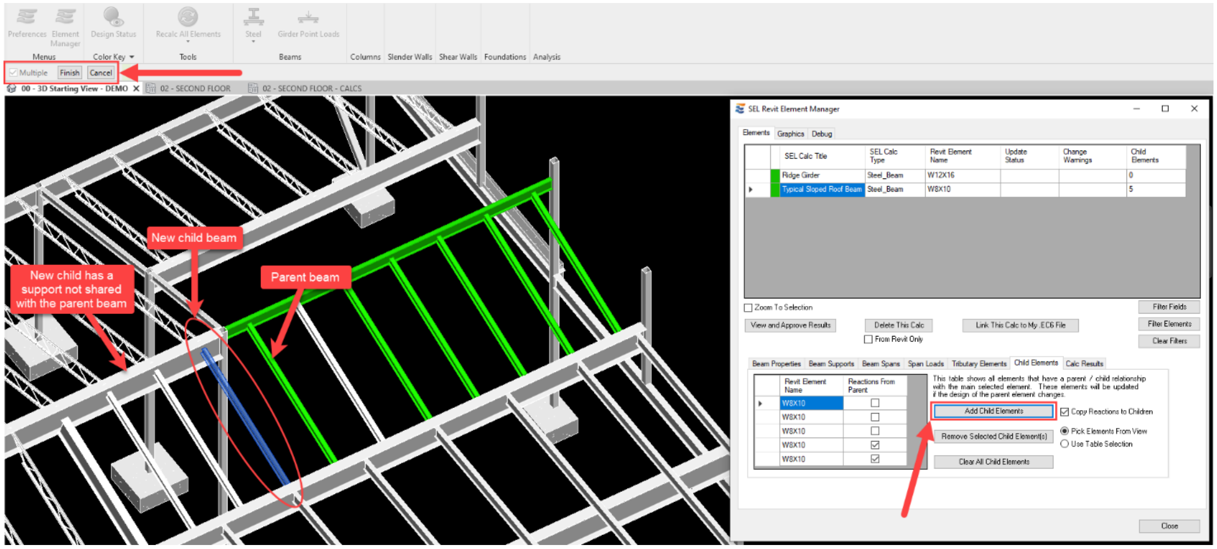




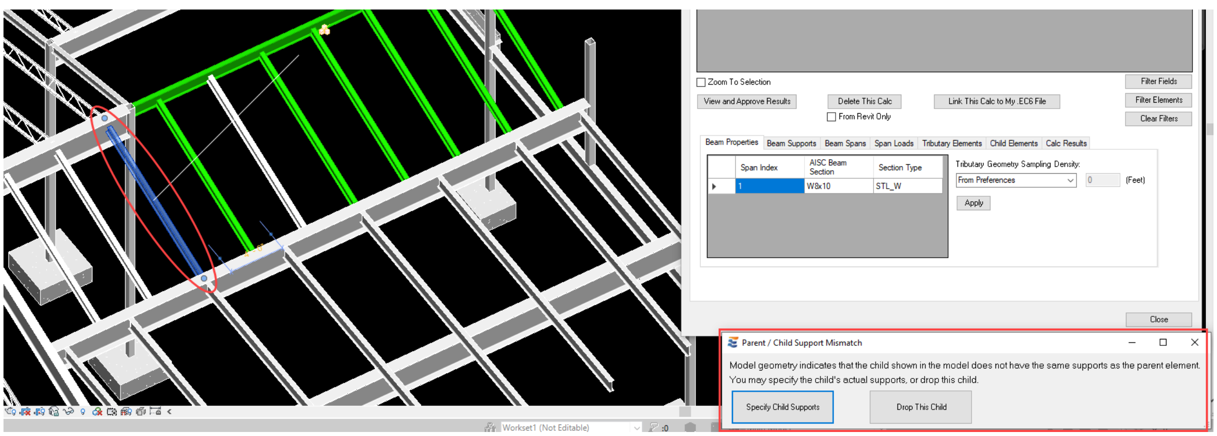
10.6.5 Parent/Child Similarity

When assigning new children to a parent beam, it is advisable for the new children to be as similar to the parent beam as possible. Child beams are not analyzed separately, nor will the user be alerted if a child beam is significantly different than the parent in critical ways that could result in an unsafe design. It is the responsibility of the design professional to ensure that any parent/child relationships are appropriate and will result in a safe design.

When adding children with reactions, certain error checking is performed to ensure the integrity of the load linking load path. These checks will result in a warning if the user specifies a child beam that does not share common supports with the parent beam. This discrepancy will harm the accuracy of the load path unless corrected. When such a disagreement is detected, the user is notified with an option to either correct the support conditions or drop the child from the list.

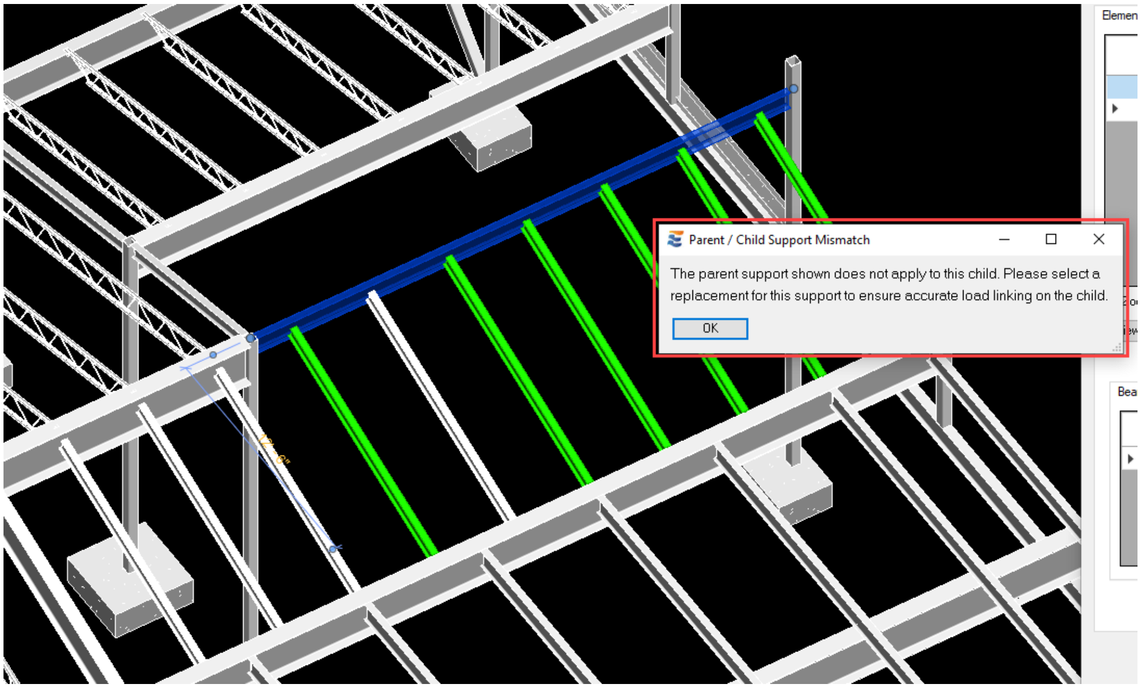


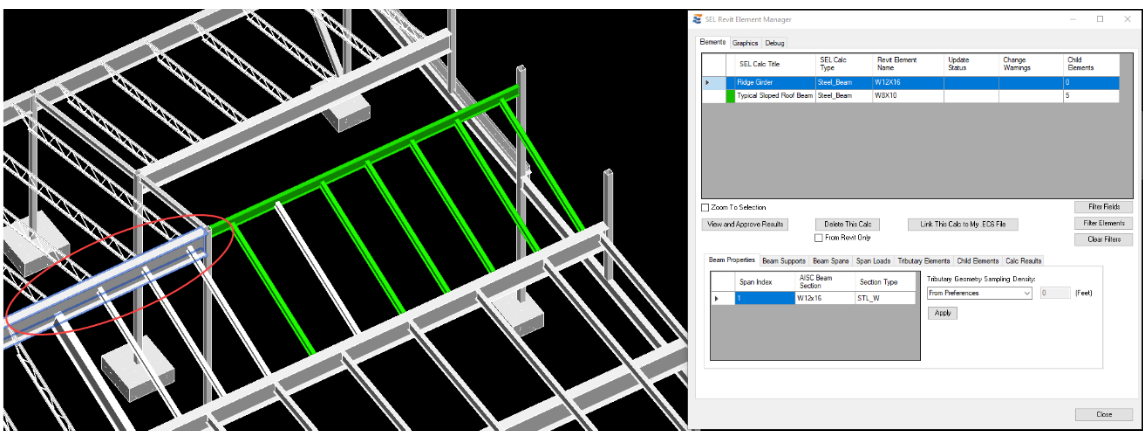
When the warning is presented, the corresponding child element is automatically selected for reference.



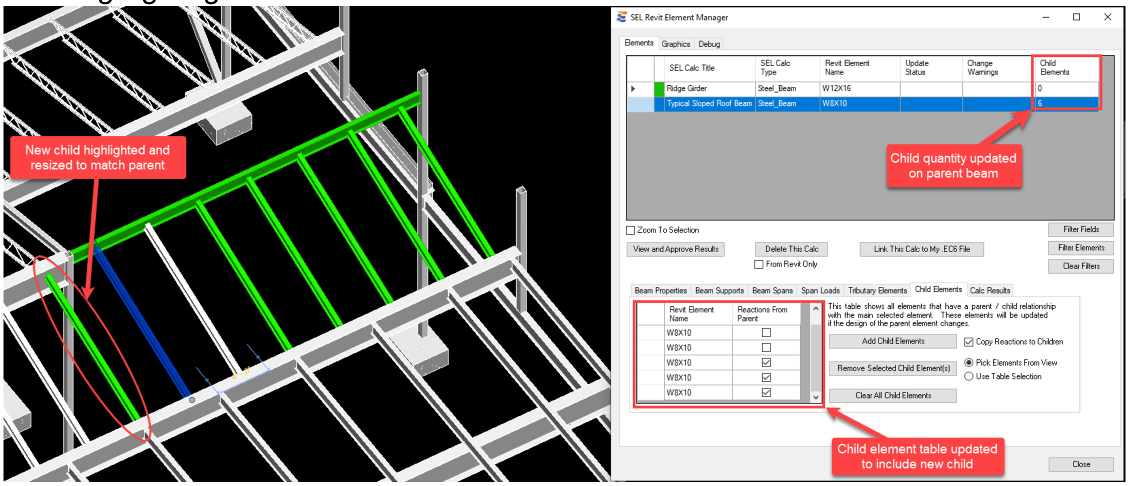
Selecting “Drop This Child” will result in this specific child element being skipped while the remaining children without warnings are added. Selecting “Specify Child Supports” will lead the user through a process of clarifying the support conditions of the child beam in order to preserve load path integrity.

The support that belongs to the parent but does not apply to the child is presented for user reference. Clicking “OK” on this notification will prompt the user to select a replacement support for the ambiguous end of the new child element.

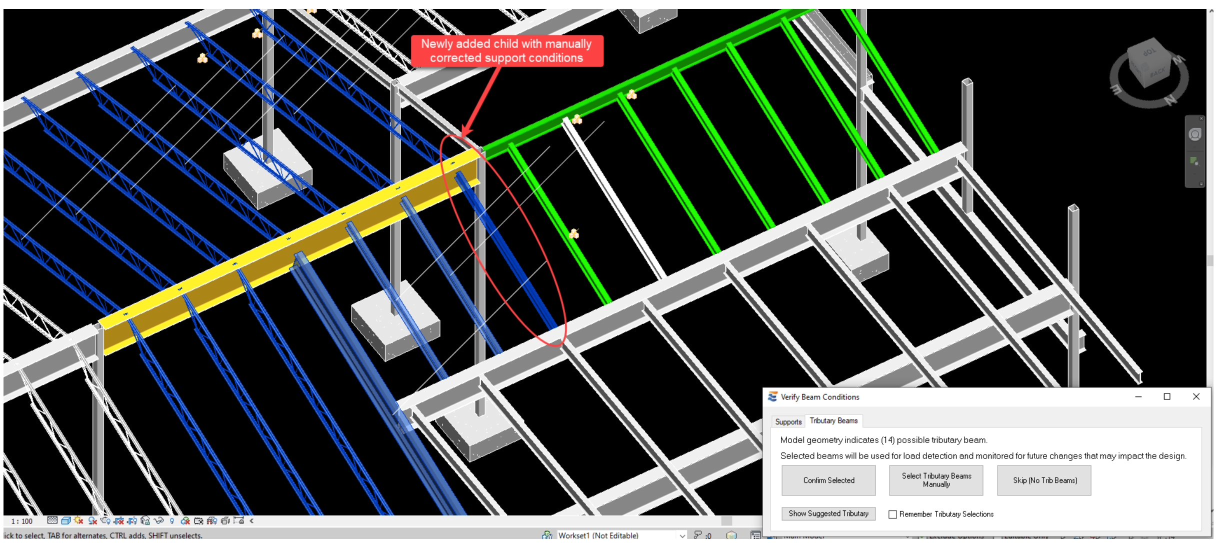

Parent / Child Support Mismatch
The parent support shown does not apply to this child. Please select a replacement for this support to ensure accurate load linking on the child.
OK

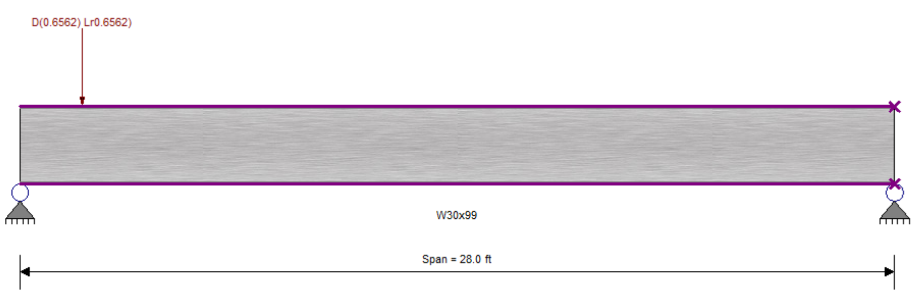

SEL Revit Element Manager
Elements
Graphics
Debug
SEL Calc Title
SEL Calc Type
Revit Element Name
Update Status
Change Warnings
Child Elements
Ridge Girder
Steel_Beam
W12X16
0
Typical Sloped Roof Beam
Steel_Beam
W8X10
5
Zoom To Selection
Filter Fields
View and Approve Results
Delete This Calc
Link This Calc to My .ECS File
Filter Elements
From Revit Only
Clear Filters
Beam Properties
Beam Supports
Beam Spans
Span Loads
Tributary Elements
Child Elements
Calc Results
Span Index
AISC Beam Section
Section Type
1
W12x16
STL_W
Tributary Geometry Sampling Density:
From Preferences
0
(Feet)
Apply
Close

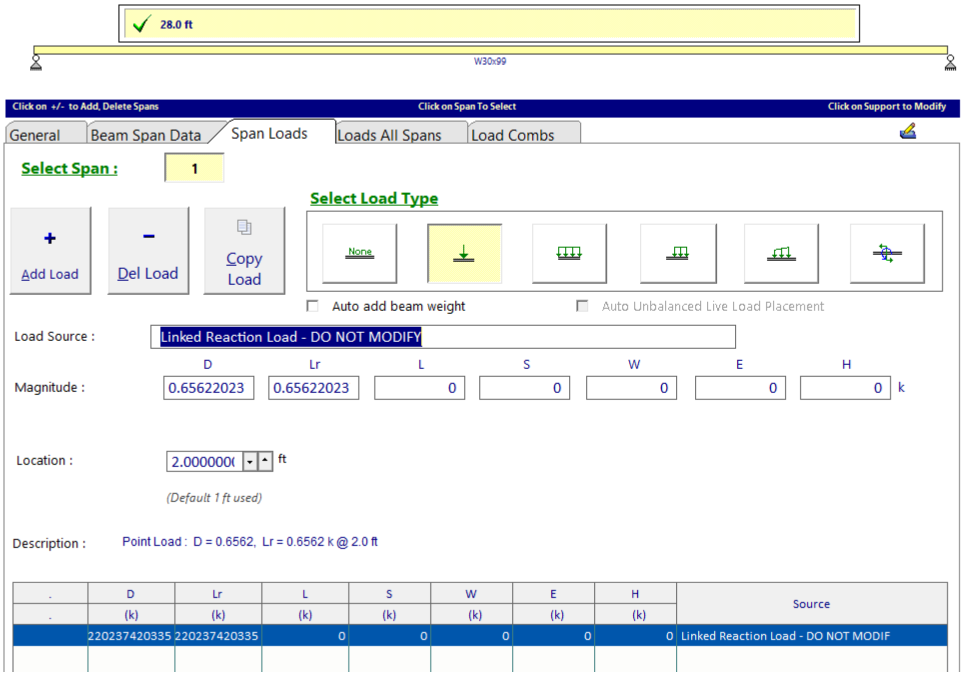
Picking the replacement support will complete the operation and the Element Manager and color highlighting will refresh.



Since the supporting relationships have been manually corrected, the stored reaction on the newly created child beam will automatically be detected and applied when launching a design calculation for the neighboring girder.

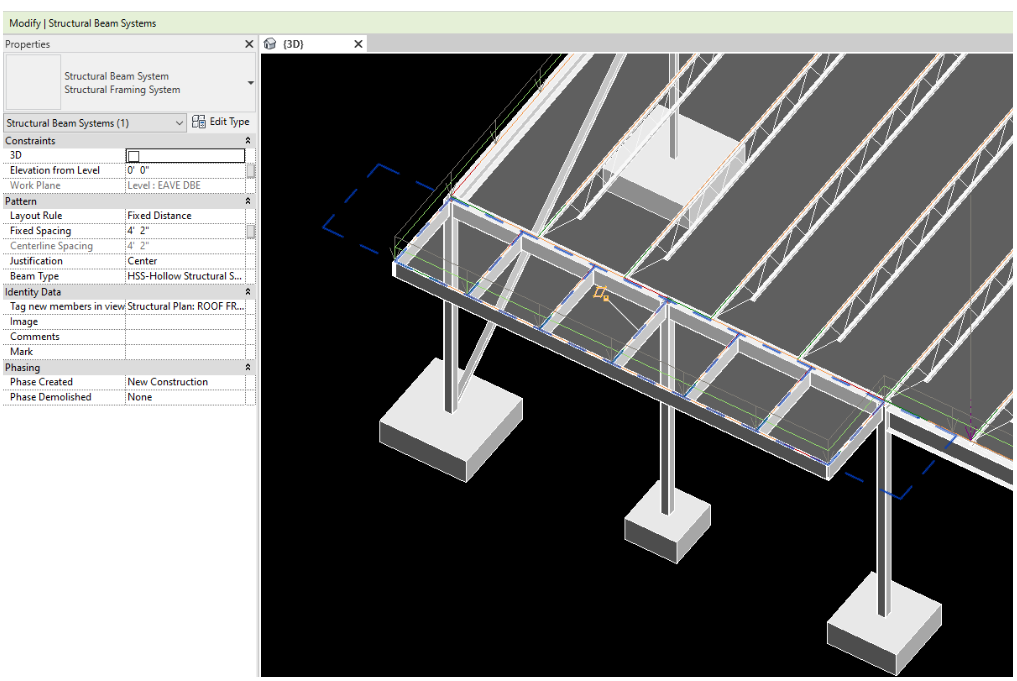






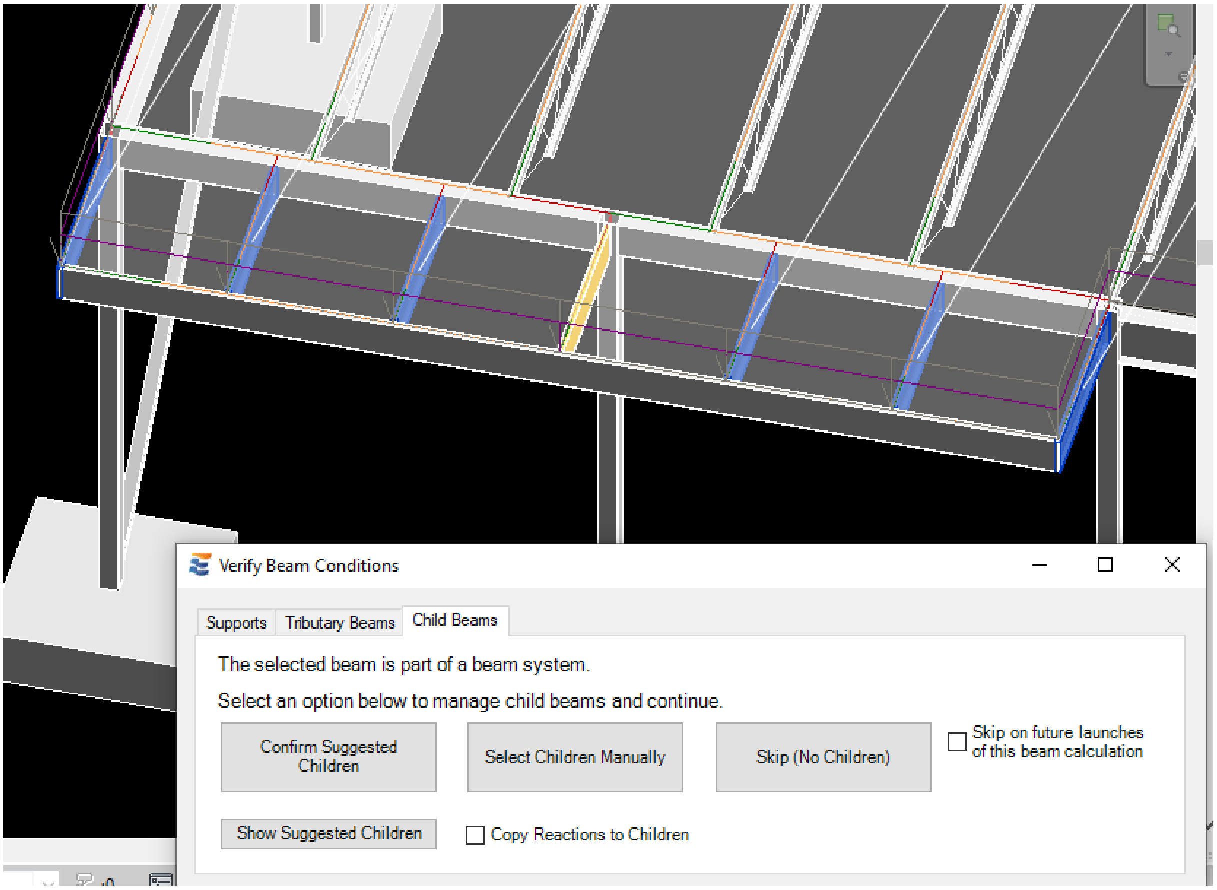
10.6.6 Working With Beam Systems

A beam system is a native Revit method to create a group of framing elements that have a common parametric relationship with each other. Beam systems are often used to accelerate work and enhance collective geometry control when modeling large repetitive beam framing systems. Since beam systems are often used to create many beams with similar geometry and spacing, ENERCALC for Revit detects the presence of beam systems during calculation launch to assist the user with bulk design of many similar elements.



When the beam to be designed is found to be a member of a beam system, the calculation launch window will prompt the user to decide if other members of the beam system should be made children of the main beam. The presence of a beam system is noted at the beginning of calculation when an additional approval tab is added to the launch window.

When the other required approvals have been completed, the user will be presented with several options for working with the beam system members as potential children. Any member of the beam system ***not already designed or associated to another calculation*** will be proposed to the user as a potential child element.



Clicking the "Confirm Suggested Children" button will accept all available members of the beam system as children of the main beam to be designed. Alternatively, the user may choose to click "Select Children Manually", which triggers a Revit multi-select operation similar to that used for the other manual select options during calculation launch. Choosing either of the first two options will result in the specified beams taking on a parent/child relationship with the beam being launched. If the "Copy Reactions" option is checked on during calculation launch, the user will be prompted to resolve any support discrepancies found while creating the parent/child associations (as described in "Parent/Child Similarity 360").

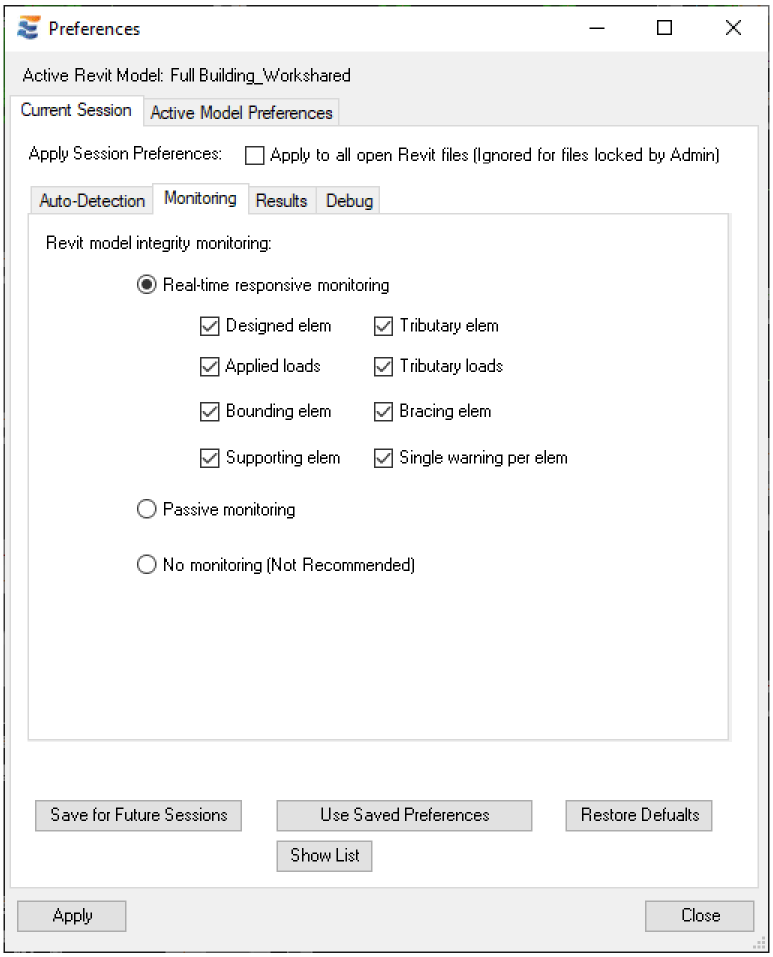
Cancelling during a manual select or choosing the "Skip" option will launch the calculation without any associated children. In this case, the remainder of the beam system will not be affected by the design of the beam currently being launched.

When the "Save and Close" operation in ENERCALC SEL completes, all specified children will have the same section size as the designed beam (as well as reactions if checked on).

10.7 Monitoring and Change Warnings

ENERCALC for Revit provides an automated monitoring and warning framework to keep users informed of various changes in the Revit model which may jeopardize the accuracy of calculations previously completed. When monitoring is enabled, any change sufficient to cause discrepancy between the Revit-modeled condition of an element and the last ENERCALC SEL calculation used to design it results in a warning on the element. Any element placed in a warning state by one of these changes will automatically have its calculation results erased. This includes the removal of unity check information, PDF reports, and “Load-Linked” reaction forces. Because of the chain reaction effect that “Load-Linking” creates, changes to one member can result in warnings on many members, if they are “downstream” of the change with respect to load path.

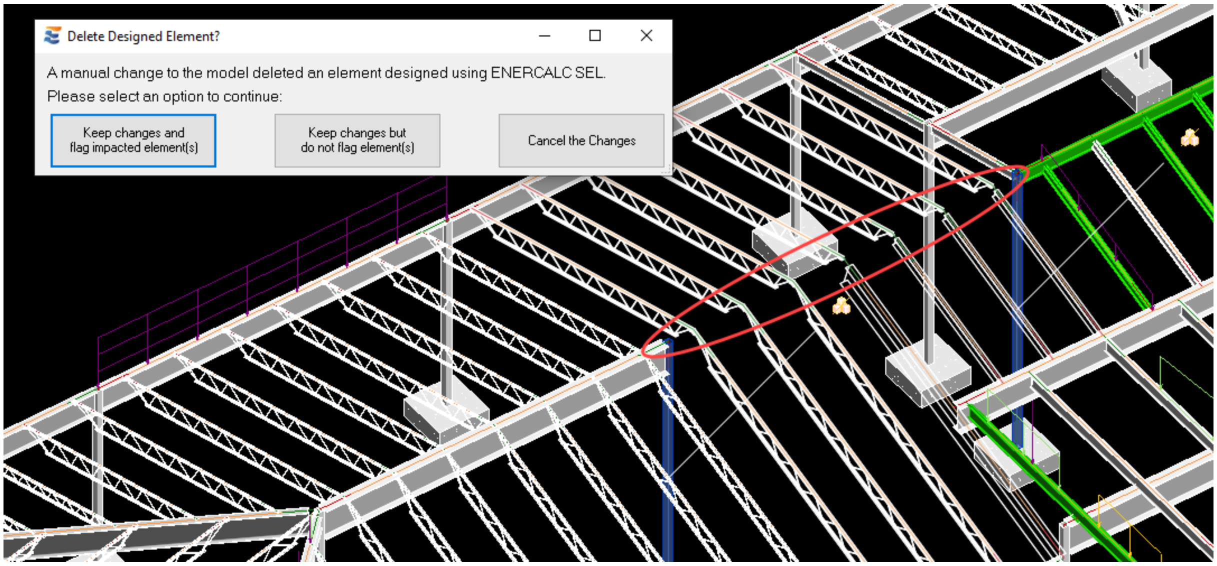
Monitoring settings are managed from Preferences > Current Session > Monitoring. Since these settings are found under “Current Session”, they are unique to the current instance of Revit on the user’s local machine and will not influence other users.



The three basic modes for monitoring are “Real-time”, “Passive”, and “No Monitoring”. The default setting is “Real-time” with all checkboxes activated. This means that a visible warning is produced for a change of any type listed in the menu. Within “Real-time” mode, these checkboxes may be selectively deactivated if the user does not wish to be interrupted for warnings about specific types of change in the Revit model. The default setting is to provide only a single warning for any given element in order to limit interruptions. Specific types of Revit model changes and the resulting warnings are discussed in the following sections.

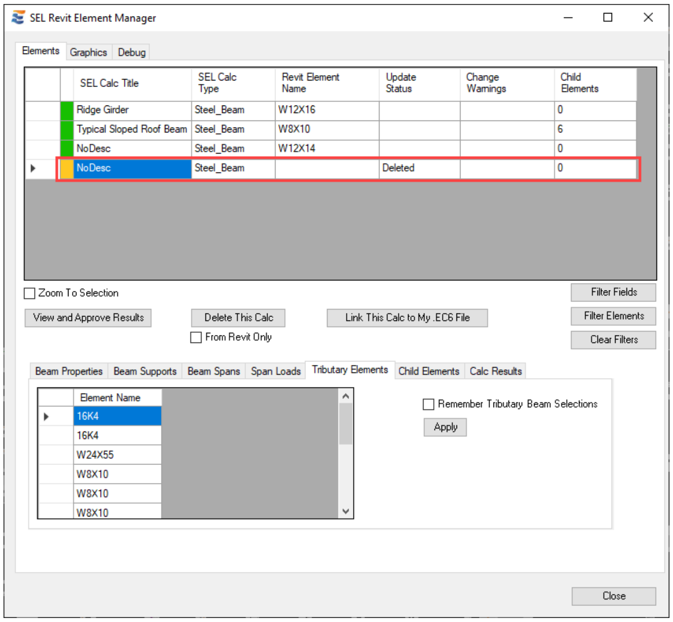
10.7.1 Deleting Designed Elements

Deletion of a beam designed using ENERCALC for Revit will result in a change warning. If working in “Real-time” monitoring mode, this will result in a pop-up notification to the user. When the pop-up notification is presented, the user has 3 choices.



The first button will proceed with removing the element. This will cause all other impacted element(s) (i.e., supported elements, supporting elements with “Load-Linking” relationships, etc.) to be placed in a warning state (orange) and a description of the change will appear in the Element Manager. The second button will preserve the change committed by the user, but will not create any warnings on the impacted element(s). The third button will undo the preceding change that triggered the notification.

If the change is kept, the Element Manager summary table will contain a line item showing the deleted element, even though it no longer exists in the model. Deleted element line items can still be selected, and limited information may be viewed after deletion.

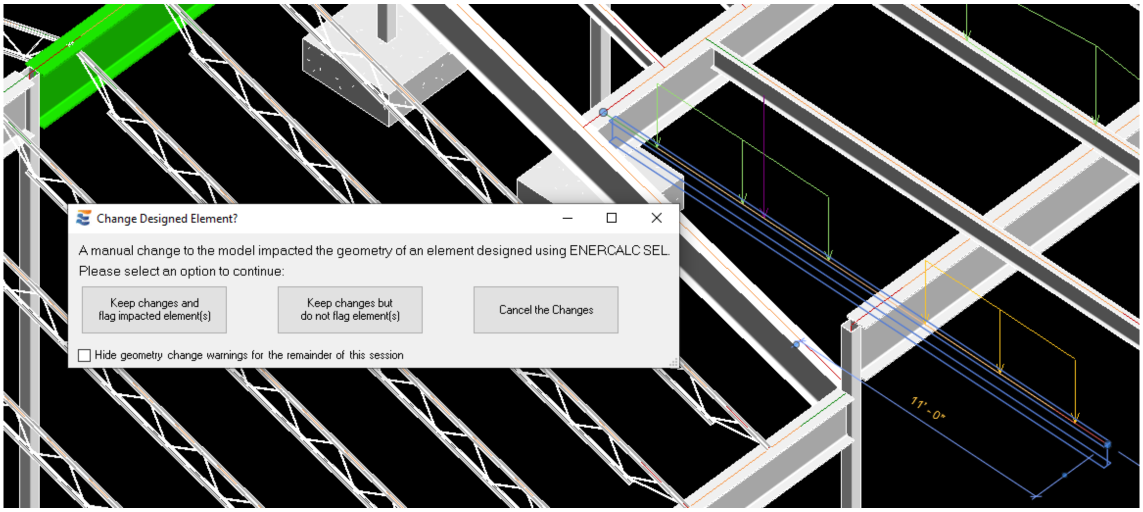


10.7.2 Geometry Changes

Direct changes to the geometry of a beam designed using ENERCALC for Revit will result in a change warning. This includes changes such as the following:

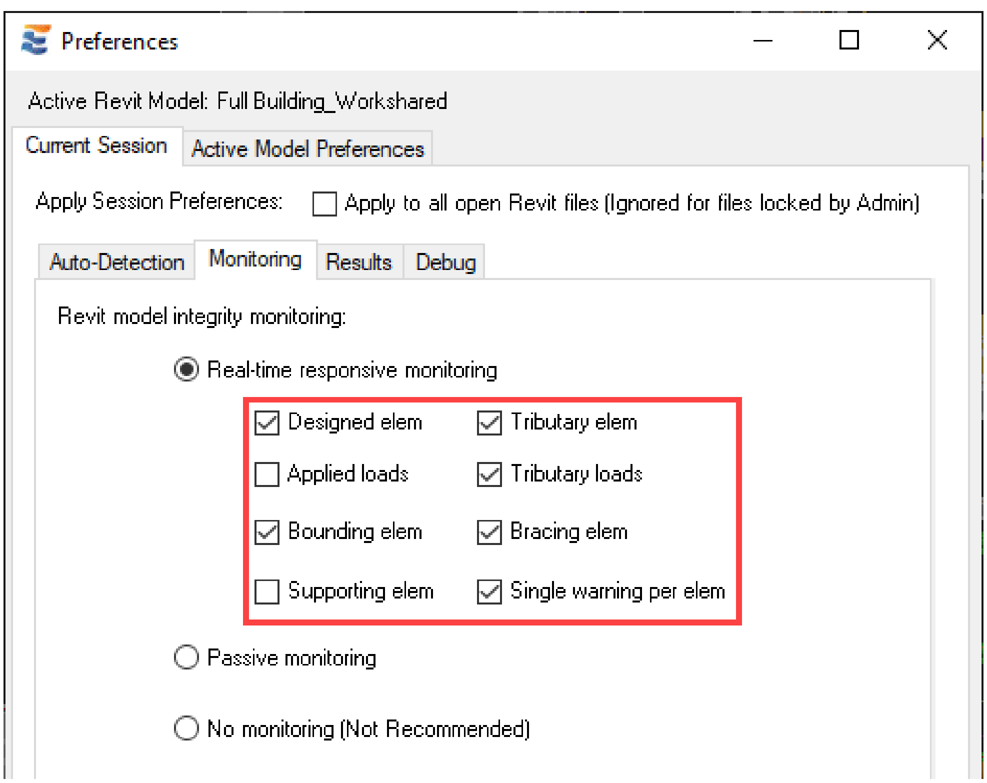
- Section size change
- Start Extension or End Extension change
- Start or End joint move
- Shift or move entire beam

If working in “Real-time” monitoring mode, these changes will result in a pop-up notification to the user. When the pop-up notification is presented, the user has 3 choices.

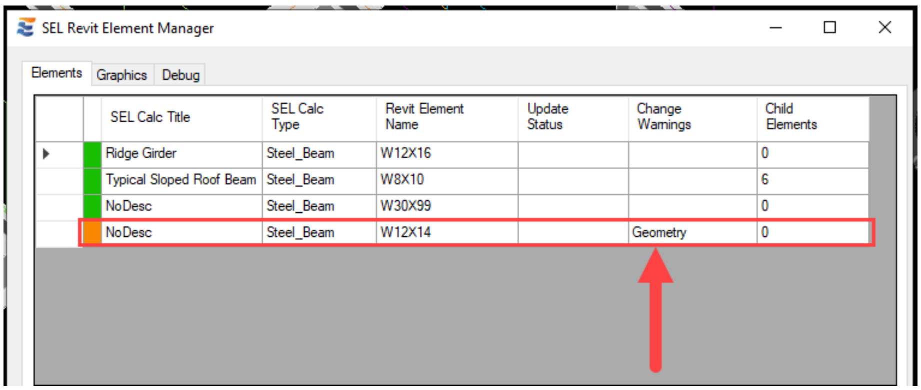


The first button will cause the impacted element(s) to be placed in a warning state (orange) and a description of the change will appear in the Element Manager. The second button will preserve the change committed by the user, but will not create any warnings on the impacted element(s). The third button will undo the preceding change that triggered the notification.

The pop-up also bears a checkbox which allows the user to “hush” future warnings about geometry changes. If this box is checked, a visit to the Preferences menu will confirm that notifications of this specific type have been deactivated. The specific warning type may be reactivated from the Preferences menu at any time if it has been previously “hushed” from the pop-up notification window.



Warning states will still be reflected in highlighting and in the Element Manager, but pop-ups will not interrupt the work session. Regardless of whether a change was accompanied by a pop-up, all changes are noted in the Element Manager summary table and via element highlighting.



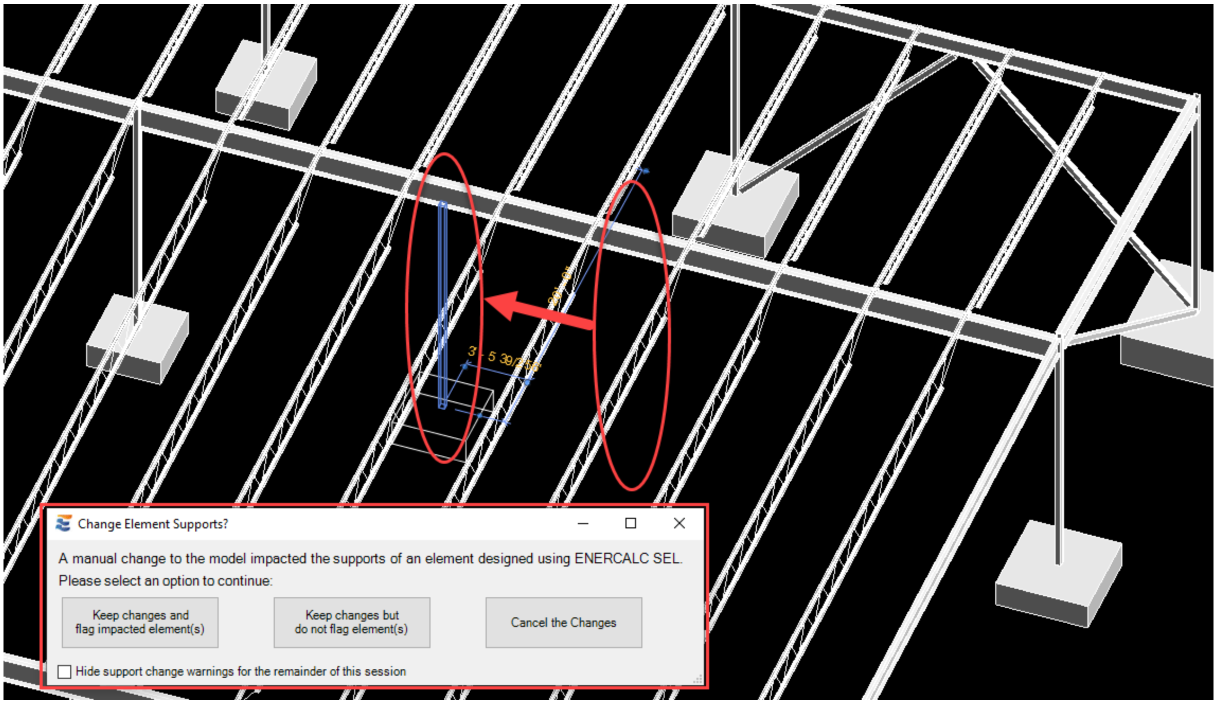
10.7.3 Changes to Supports

Changes to the supports of a beam designed using ENERCALC for Revit will result in a change warning. This includes changes such as the following:

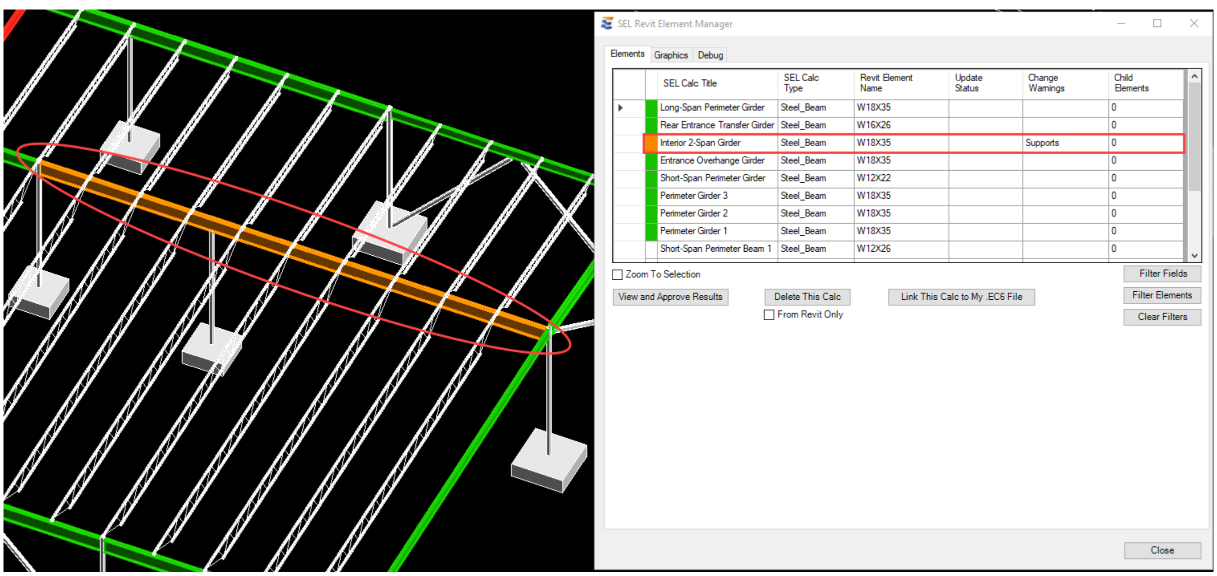
- Shifting a beam's support

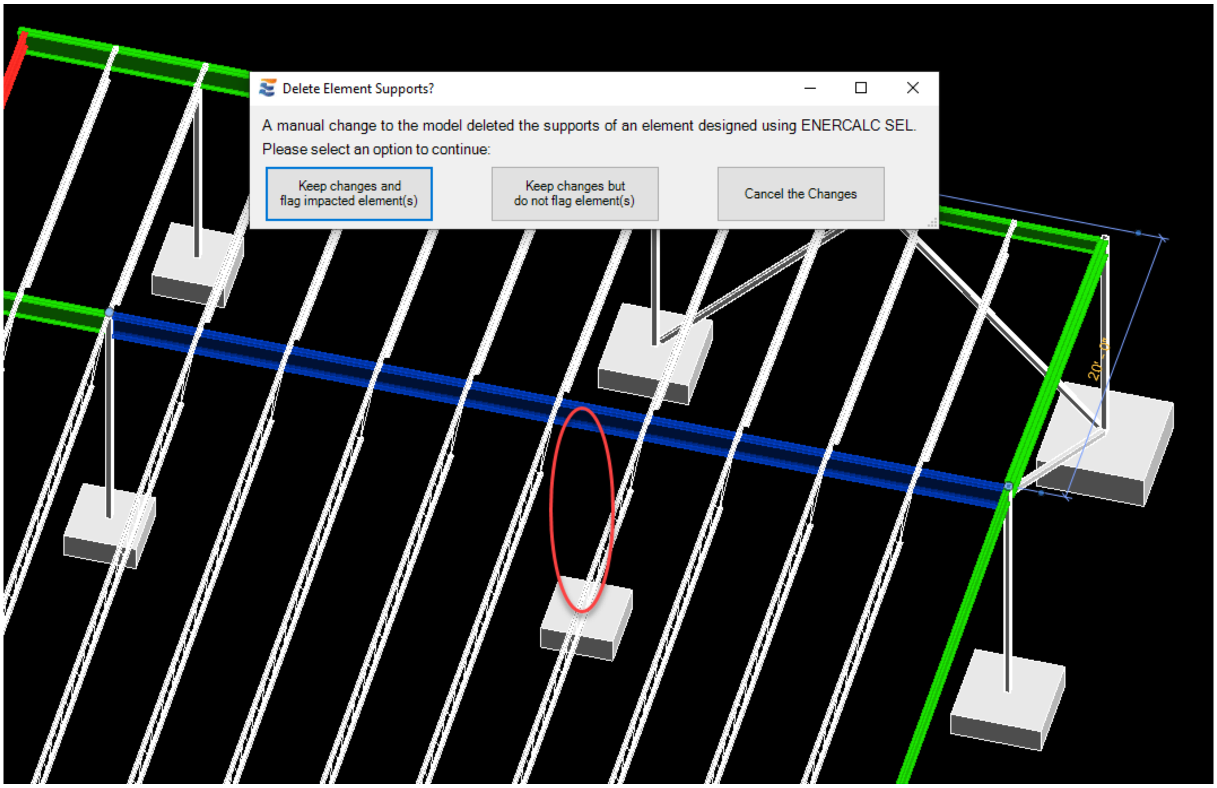
- Deleting a beam's support

If working in "Real-time" monitoring mode, these changes will result in a pop-up notification to the user.   As with other types of pop-up notifications, the user is presented with 3 choices.

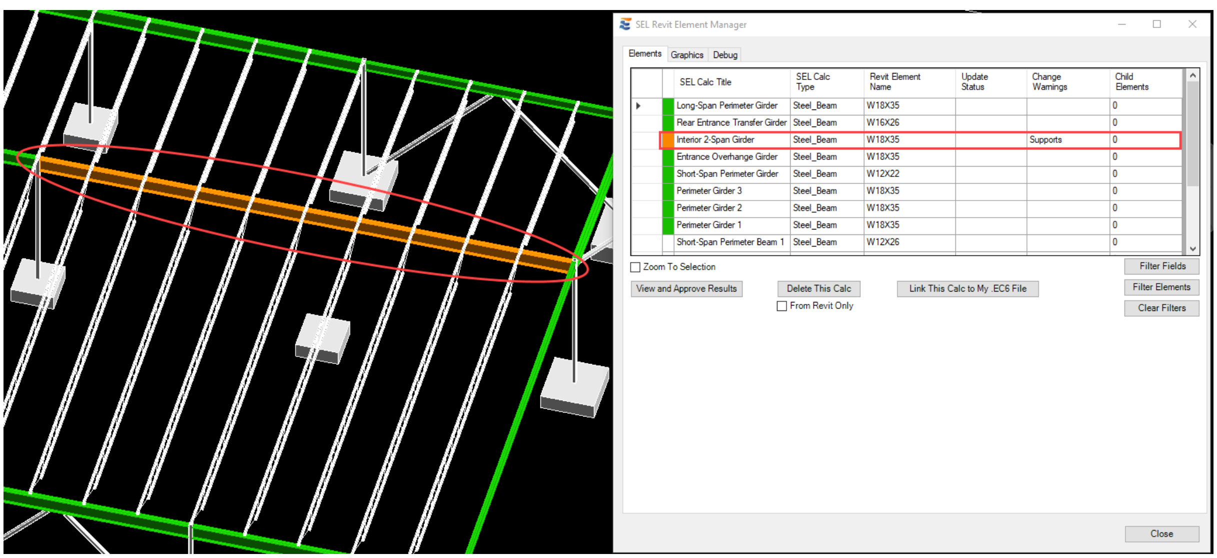


Regardless of whether a change was accompanied by a pop-up, all changes are noted in the Element Manager summary table and via element highlighting.





Regardless of whether a change was accompanied by a pop-up, all changes are noted in the Element Manager summary table and via element highlighting.



10.7.4 Changes to Applied Loads

Changes to the applied loads on a beam designed using ENERCALC for Revit will result in a change warning. This includes changes such as the following:

- Adding new applied loads

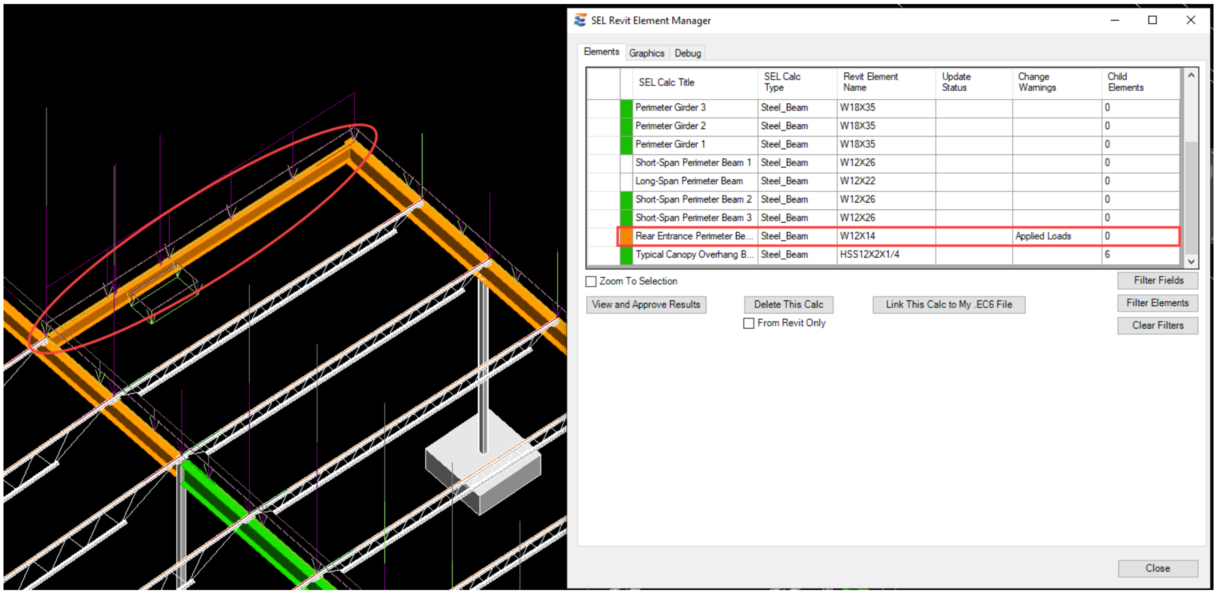
- Deleting existing applied loads

- Changing the magnitude of existing applied loads

- Changing the location of existing applied loads

If working in “Real-time” monitoring mode, these changes will result in a pop-up notification to the user. As with other types of pop-up notifications, the user is presented with 3 choices.

Regardless of whether a change was accompanied by a pop-up, all changes are noted in the Element Manager summary table and via element highlighting.



10.7.5 Changes to Tributary Loads

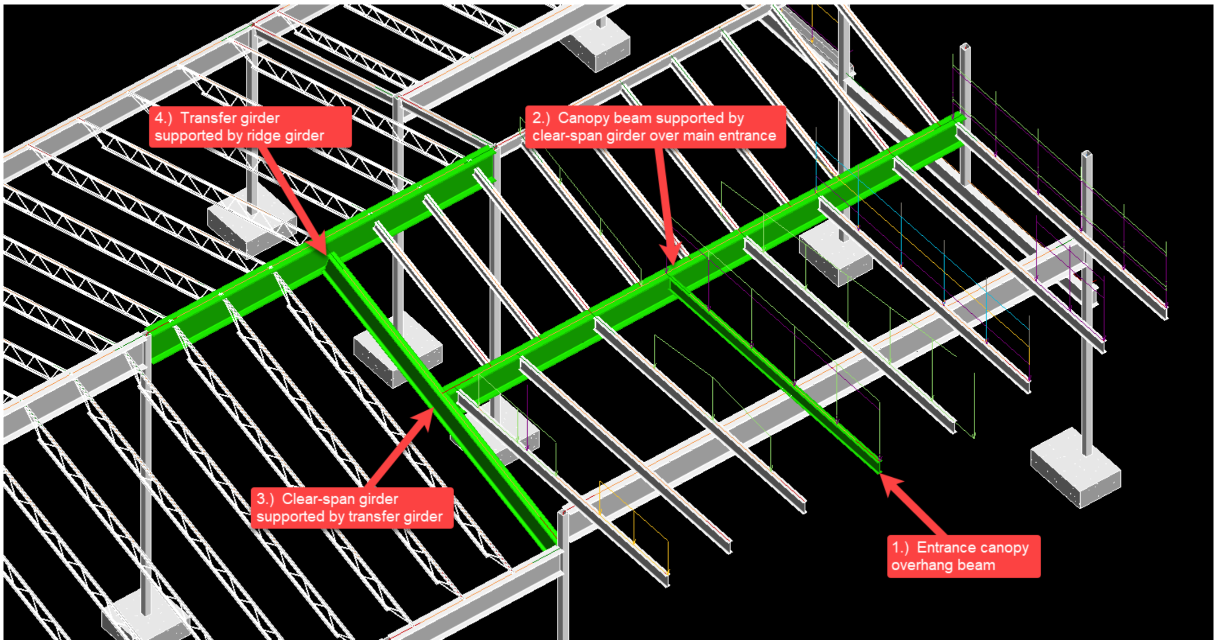
In addition to changes directly to applied loads, ENERCALC for Revit also monitors changes that have “cascading” effects down through the load path on multiple designed elements that are connected via “Load-Linking”.

This includes changes such as the following:

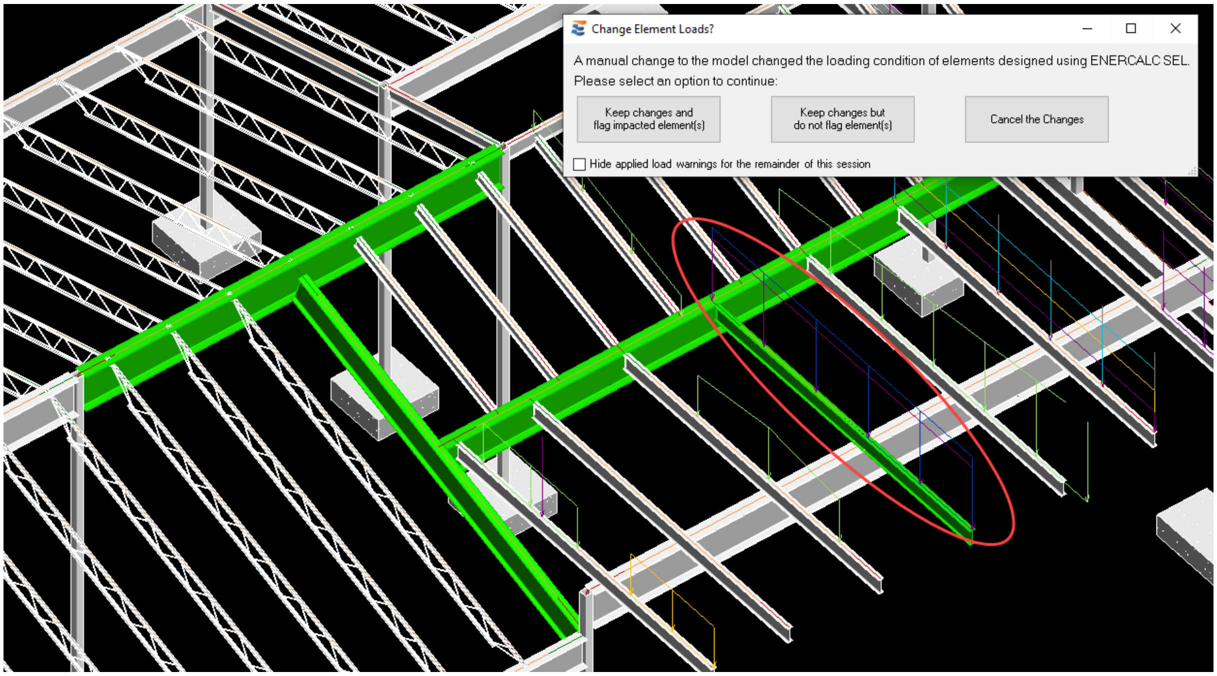
- Adding new applied loads to a designed beam supported by a designed girder
- Deleting existing applied loads from a designed beam supported by a designed girder
- Changing the magnitude of existing applied loads on a designed beam supported by a designed girder
- Changing the location of existing applied loads on a designed beam supported by a designed girder
- Changing the section size of a designed beam supported by a designed girder
- Changing the location of a designed beam supported by a designed girder

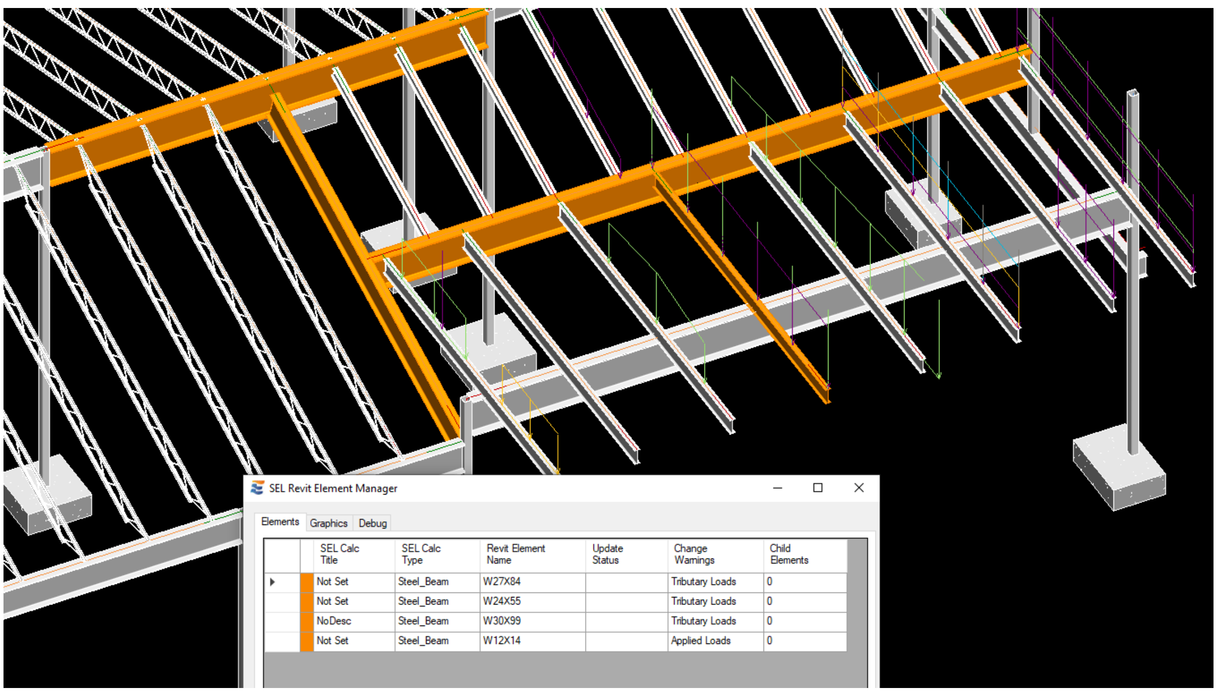
- Changing the support conditions of a designed beam supported by a designed girder

If working in “Real-time” monitoring mode, these changes will result in a pop-up notification to the user.   As with other types of pop-up notifications, the user is presented with 3 choices. The framing system shown below is an example of “cascading” load path warnings due to multiple levels of “Load-Linking”.



Any change to the applied loads on the entrance canopy beam produces warnings on all 3 girders.

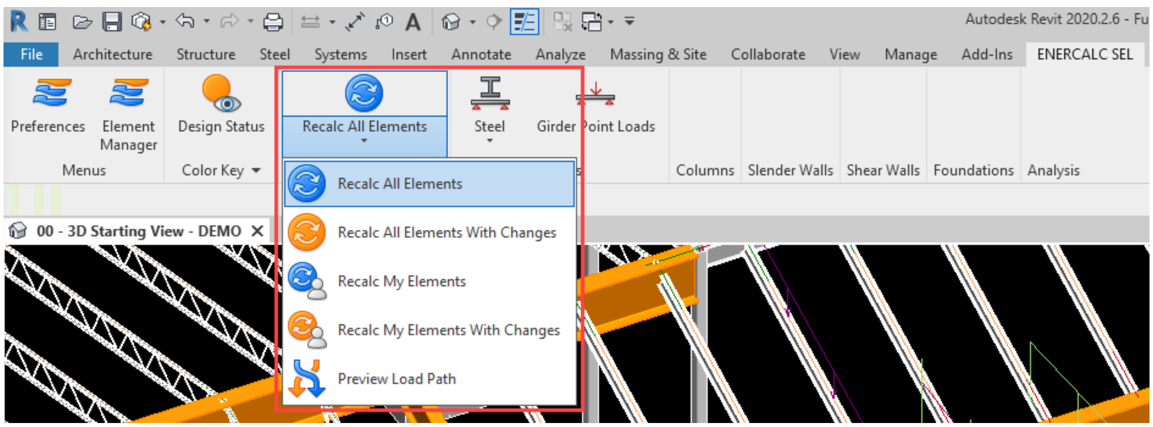


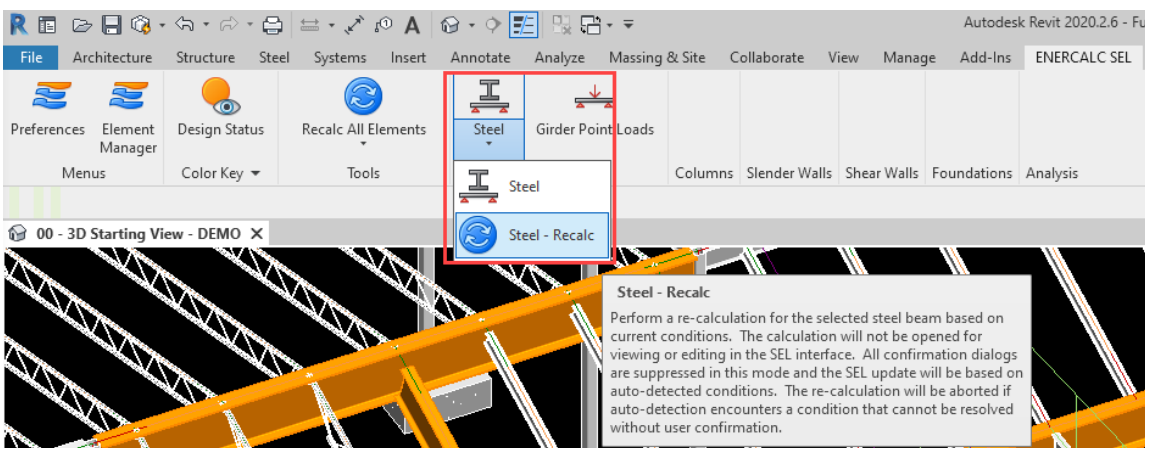


10.8 Recalculation Tools

NOTE: At this time, the recalc tools provided in ENERCALC for Revit are ONLY for triggering bulk updates to existing calculations that were previously created and verified via manual interaction by the design professional. No tools are provided for fully autonomous generation of component design calculations.

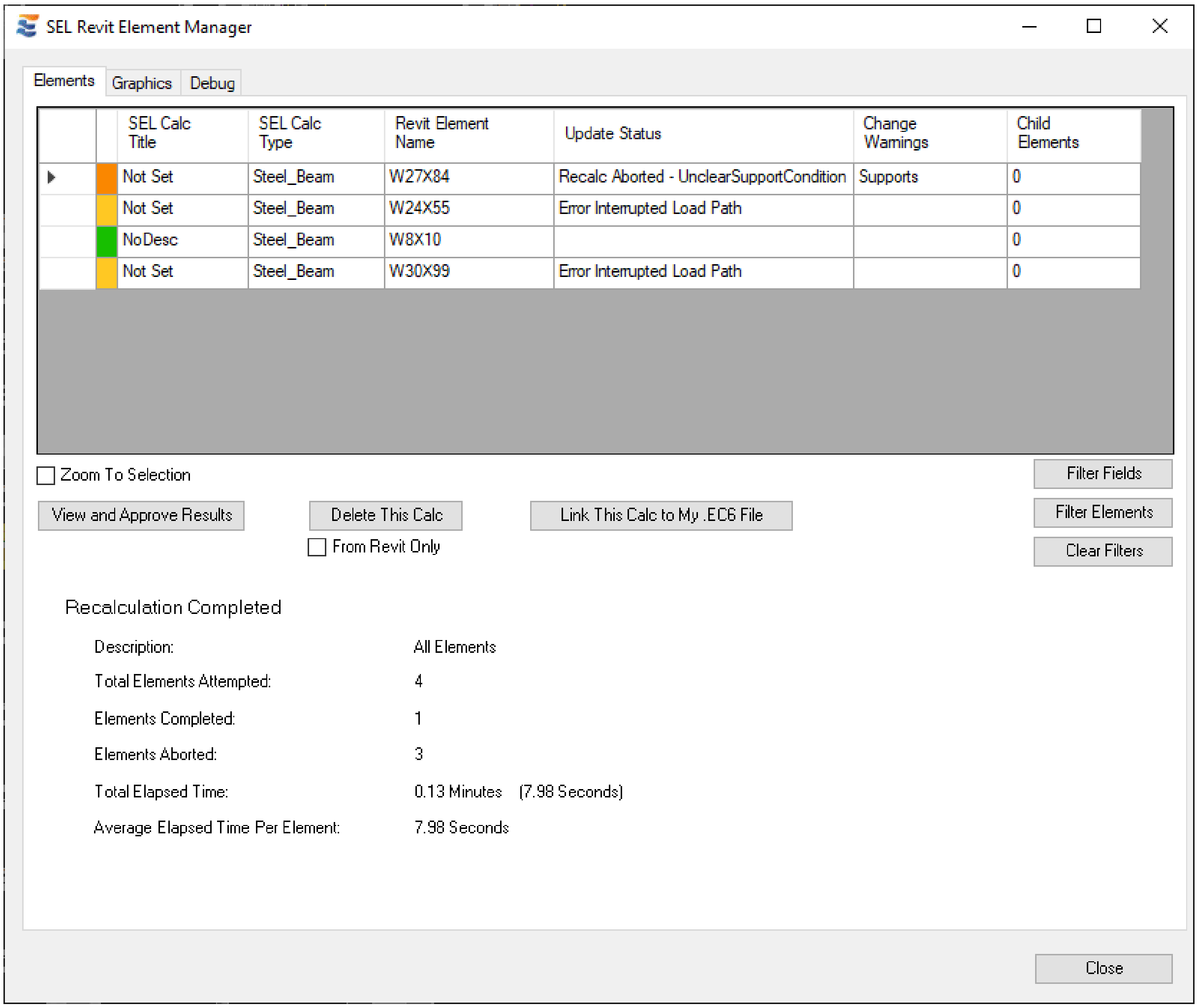
The bulk recalculation tools on the ribbon bar were introduced and briefly discussed in "Recalc Tools 30". This section will discuss recalculation in more detail specifically as it relates to beam calculations. Beams may be recalculated either by the bulk tools, or by the single-element recalc found under the primary launch control for the beam type of interest. A recalculation refreshes the analysis of a beam based on the current geometry information sent from the Revit model, without opening the complete calculation for viewing in the ENERCALC SEL interface.





The defining feature of a recalculation operation is that it does not prompt the user for any form of approval or other decisions while in progress. This means that the data used to construct the ENERCALC SEL calculation are obtained only via auto-detection or information stored during previous launches of the calculation.

When launching a beam recalculation, stored information is given first priority. This means that if supports or tributary beams were stored using the "Remember" checkbox during a previous launch, then these will be used during the calculation launch. If stored supports or tributaries fail the error checking steps, the recalc will be halted, or in the case of a bulk recalc, the specific element with errors will be skipped and the recalc will proceed to the next element in the queue. If stored data is not available, then the recalc will be performed using auto-detected supports and auto-detected tributaries. If the results of this detection process diverge from the data used during the previous launch, then the recalc will be halted, or in the case of a bulk recalc, the specific element with errors will be skipped and the recalc will proceed to the next element in the queue. If any element in the bulk recalculation queue has been skipped due to an error or ambiguous condition, all “downstream” elements in the load path found to be dependent on the skipped element will also be skipped to prevent erroneous result from “Load-Linked” elements. These elements will be flagged with the status “Error Interrupted Load Path”.



10.9 Beam Reaction Forces

When producing a set of drawings, engineering teams often find it necessary to document the reaction forces at the ends of beams for the connection designer to reference. ENERCALC for Revit provides a framework of simple tools to easily annotate beam end reactions on Revit views using a tag that displays the force results from the ENERCALC SEL calculation for each beam element. The tag family used for this annotation is included in the ENERCALC for Revit program installation and is loaded into Revit projects automatically.

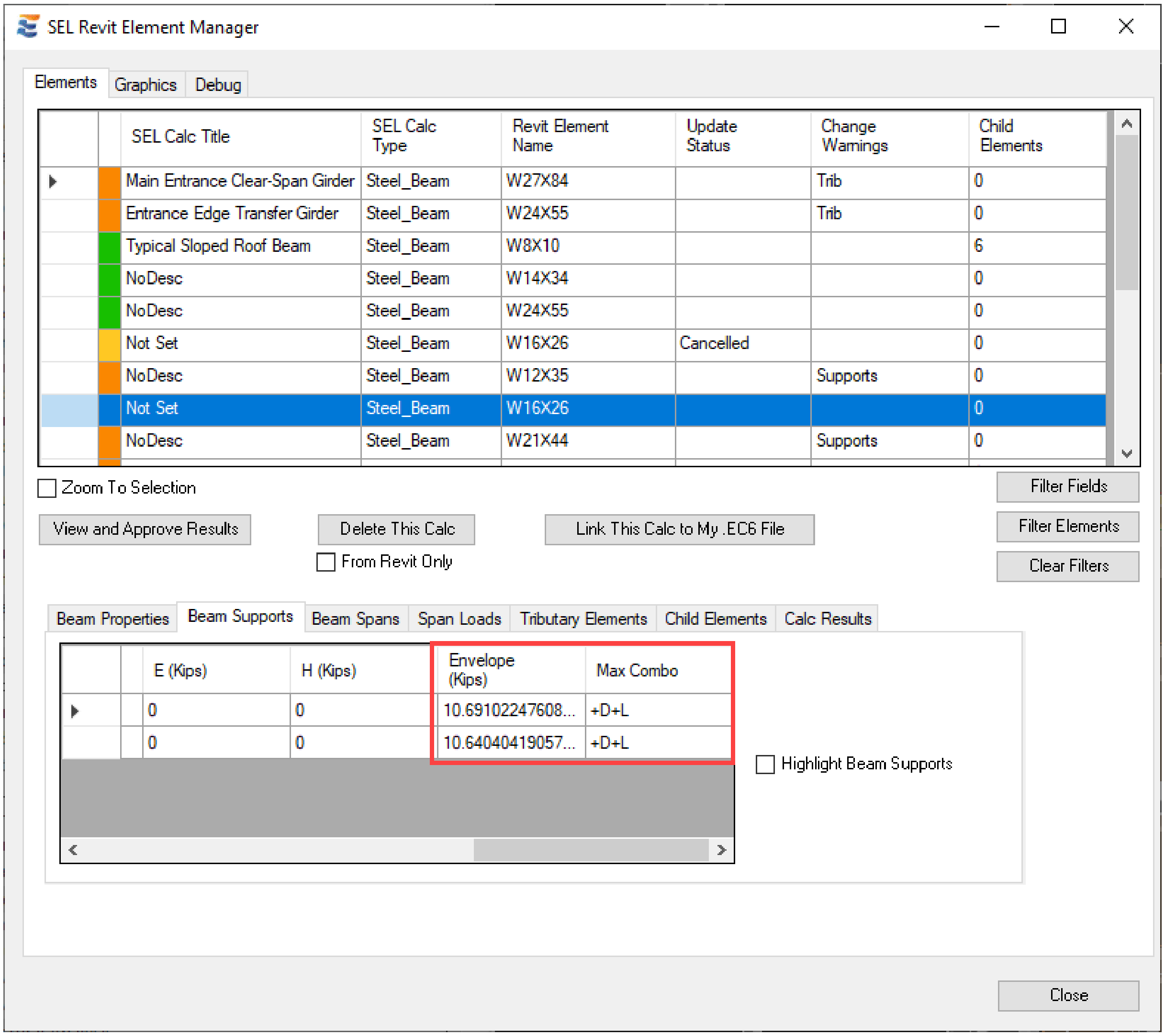
The reaction forces used for display in Revit are drawn directly from the beam or girder calculation for the element. The following guidelines outline how these reactions are obtained:

- Maximum end reactions are obtained from the reactions specified on the "Reaction Combinations" tab in ENERCALC SEL

- Prior to any adjustment or rounding, the maximum end reaction reported in Revit will exactly match the reaction reported in ENERCALC SEL under Calculations > Support Reactions > Overall Maximum

- If reduction of LL or LR loads is in use, the reported reaction will reflect the reduced reaction (if the controlling load combination includes LL or LR)

The user may view the envelope reaction and the controlling combination at any time via the Element Manager. When a calculation is selected in the summary table, this information is found on the "Beam Supports" tab.

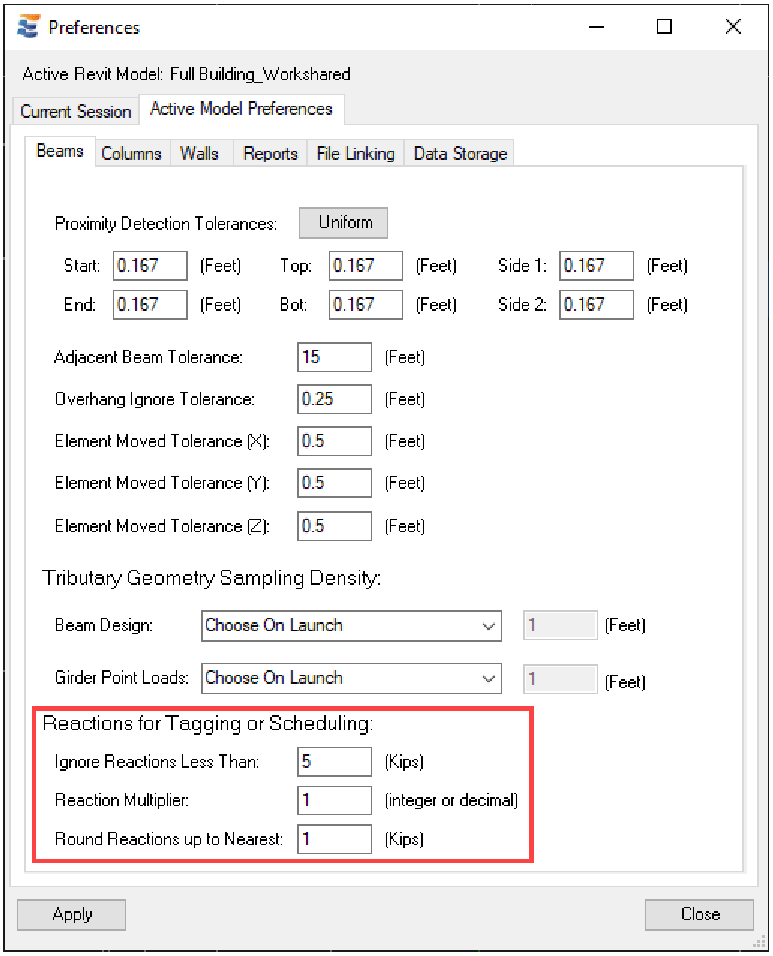


In order to provide ease of customization for various engineering teams to implement their own office standards, a set of controls are provided under Preferences > Active Model Preferences > Beams. These options permit the following customizations:

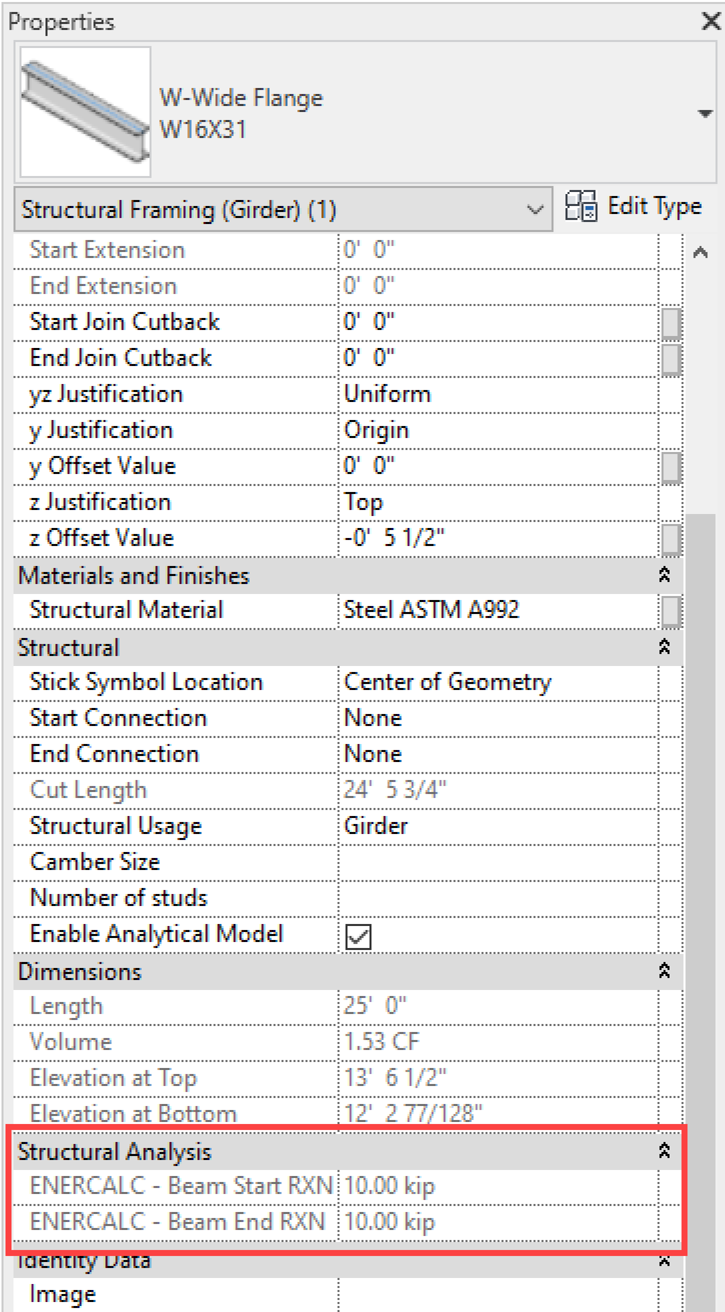
- A threshold (Kips) for hiding reactions less than a specified limit

- A custom multiplier to apply to all reactions from analysis if desired

- A custom rounding tolerance (Kips) for adjusting all reactions from analysis

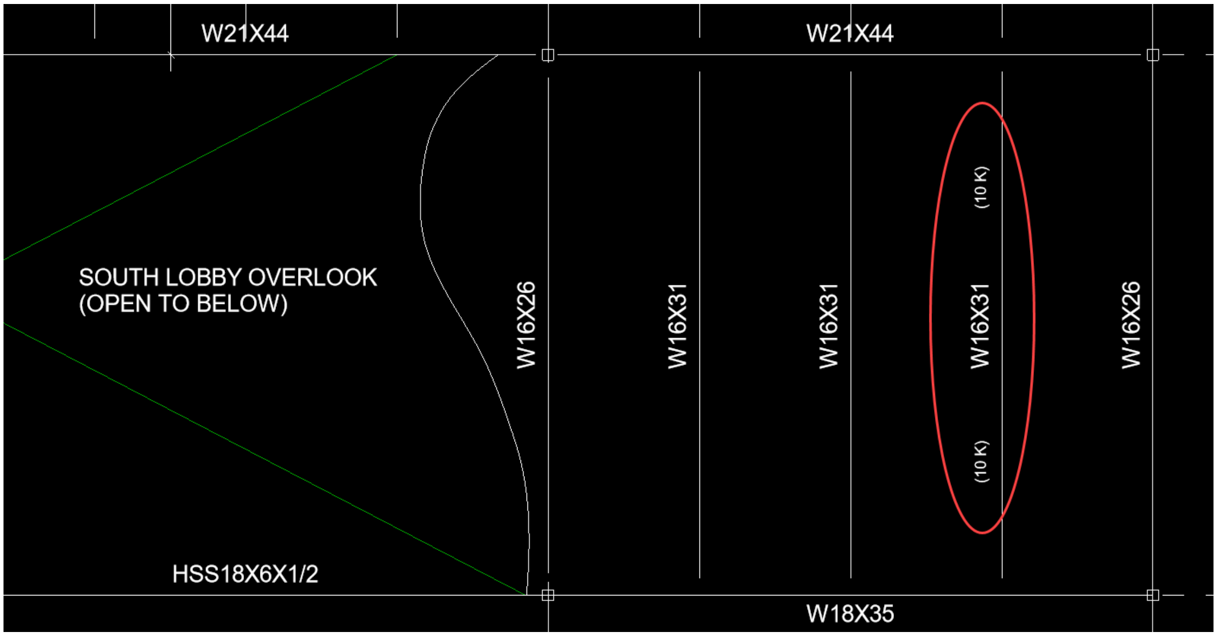


When a beam calculation "Save and Close" operation is completed, the envelope end reactions from the calculation will automatically populate to a pair of Revit shared parameters on the beam, which are visible in the native Revit Properties pane when the beam is selected. The magnitudes of the reactions will be automatically adjusted according to the rules set in the menu above prior to being displayed in the parameter fields. Note that the parameters are by default locked against user modification. Changes to these values should derive directly from the calculation and rounding preferences rather than from manual editing.



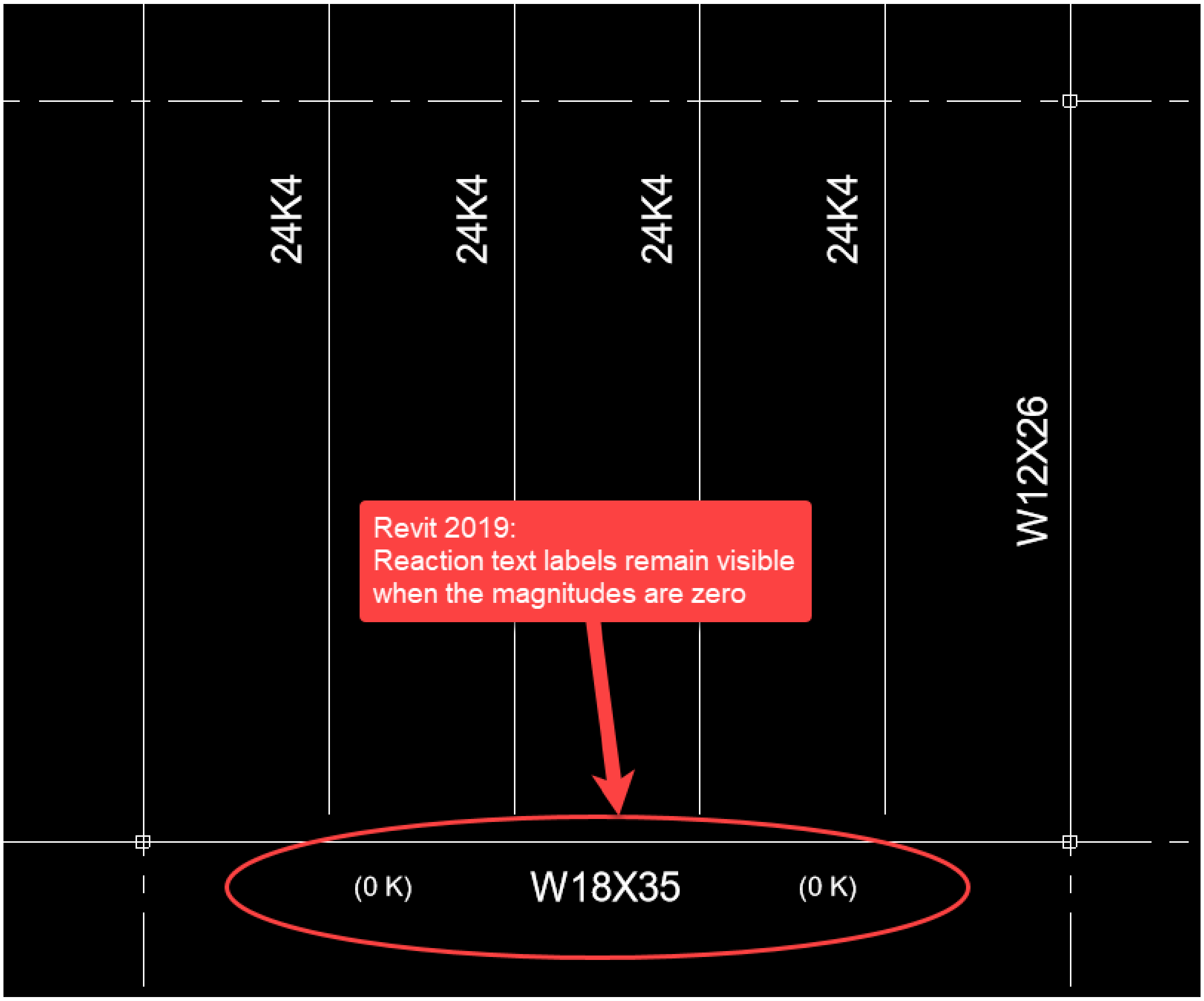
The user does not need to create or manage these parameters; they are created and managed automatically by ENERCALC for Revit. However, users should be aware that removing the associated shared parameter from the project (via the native Revit controls under Manage > Project Parameters > Remove) will cause the loss of all reaction data stored on individual beam instances.

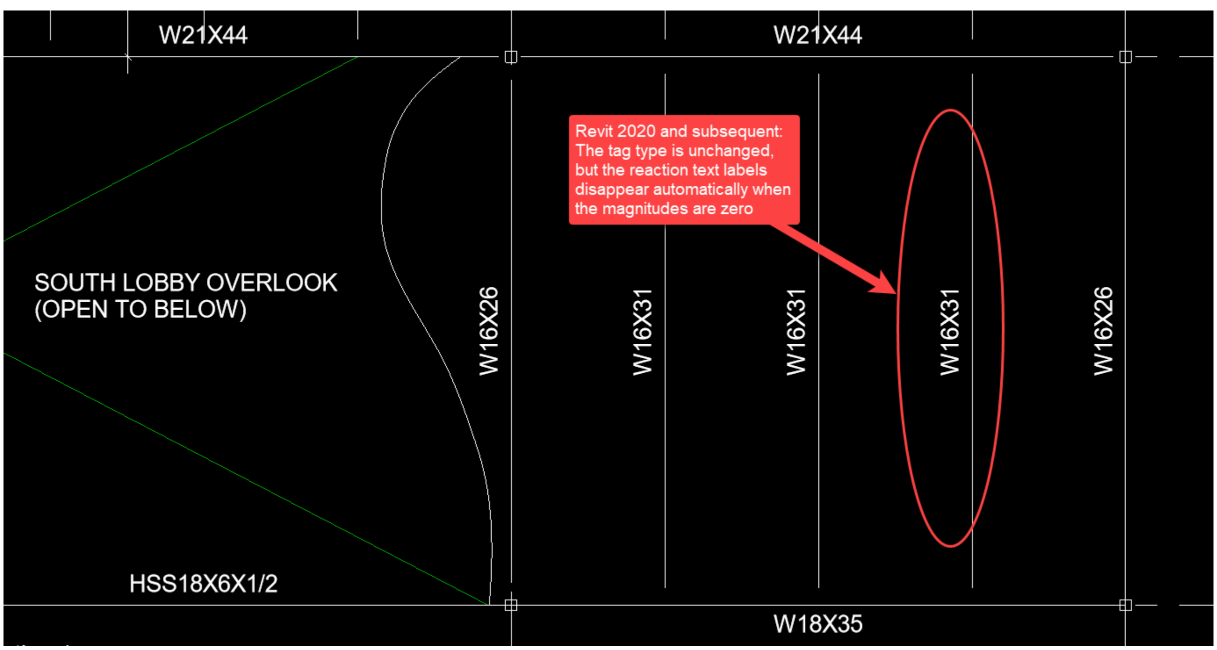
Once a calculation has been finished and the reactions are automatically updated, they may be annotated on plans by either placing new framing tags or toggling existing framing tags to the type named “Structural Framing Tag – ENERCALC (20XX)”. The version designation “20XX” corresponds to the version year of the Revit project. As indicated previously, the user does not need to manually load this family. Changing the tag type will make the reaction magnitudes visible.



Once a beam calculation has been finished with a specific set of rounding controls, the reaction parameter values will ***NOT*** automatically update in real-time if the rounding settings are subsequently changed in the Preferences menu. Instead, the reactions displayed in the parameter values will be updated the next time the calculation is either recalculated or edited and saved.

Similar to the handling of unity checks and other calculation results found in the Element Manager, beam reactions are erased automatically if a change warning invalidates the design calculation. Due to differences in the parameter capabilities between Revit 2019 and Revit 2020 - present, users will observe that the tag display behavior varies. In Revit 2019, reactions are displayed as “(0 K)” when the reactions are zero or have been erased. In Revit 2020 and subsequent versions, zero or blank reactions will not be displayed.

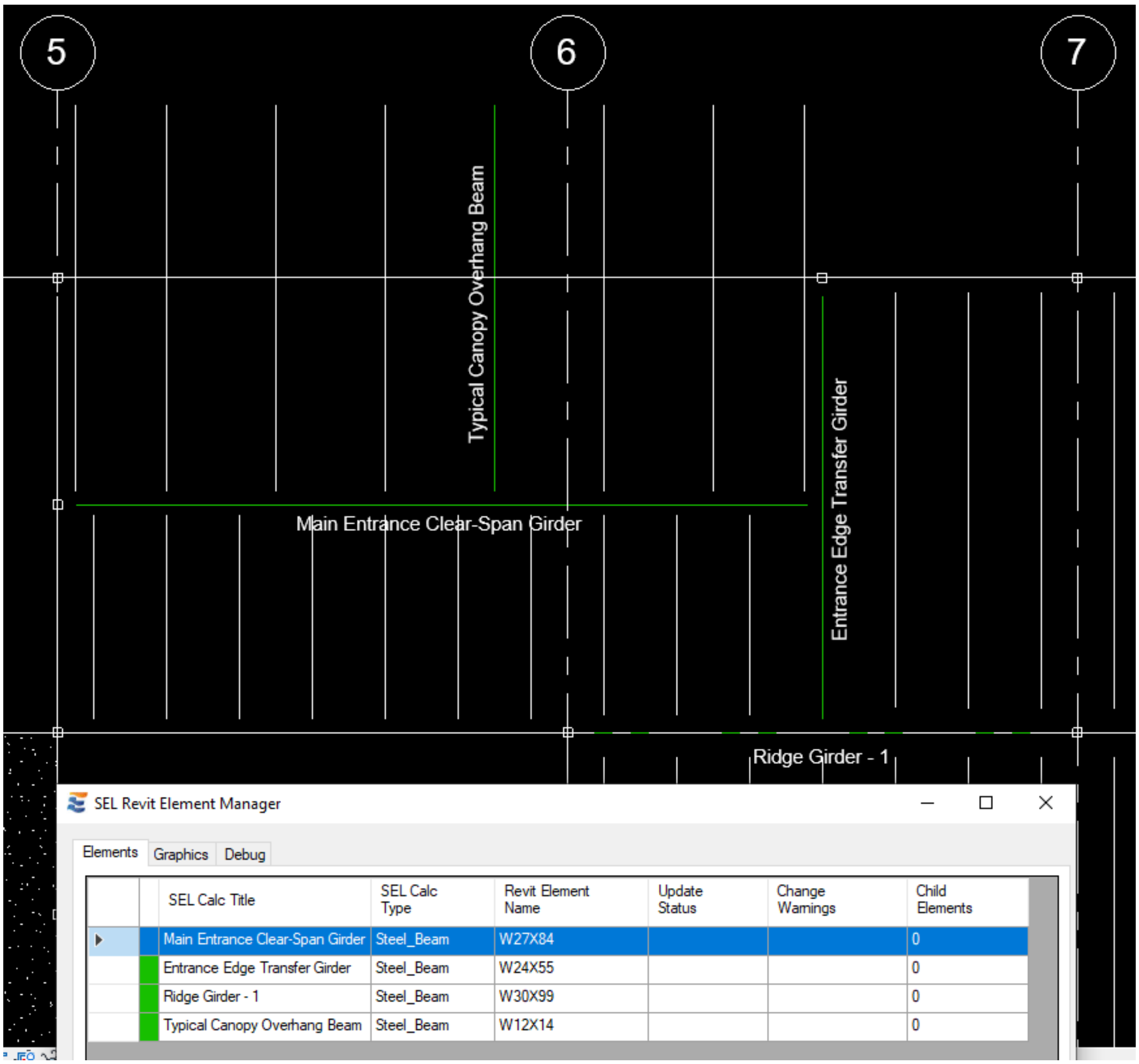

24K4
24K4
24K4
24K4
W12X26
Revit 2019:
Reaction text labels remain visible
when the magnitudes are zero
(0 K)
W18X35
(0 K)


W21X44
W21X44
Revit 2020 and subsequent:
The tag type is unchanged,
but the reaction text labels
disappear automatically when
the magnitudes are zero
SOUTH LOBBY OVERLOOK
(OPEN TO BELOW)
W16X26
W16X31
W16X31
W16X31
W16X26
HSS18X6X1/2
W18X35

10.10 Beam Calculation Labels

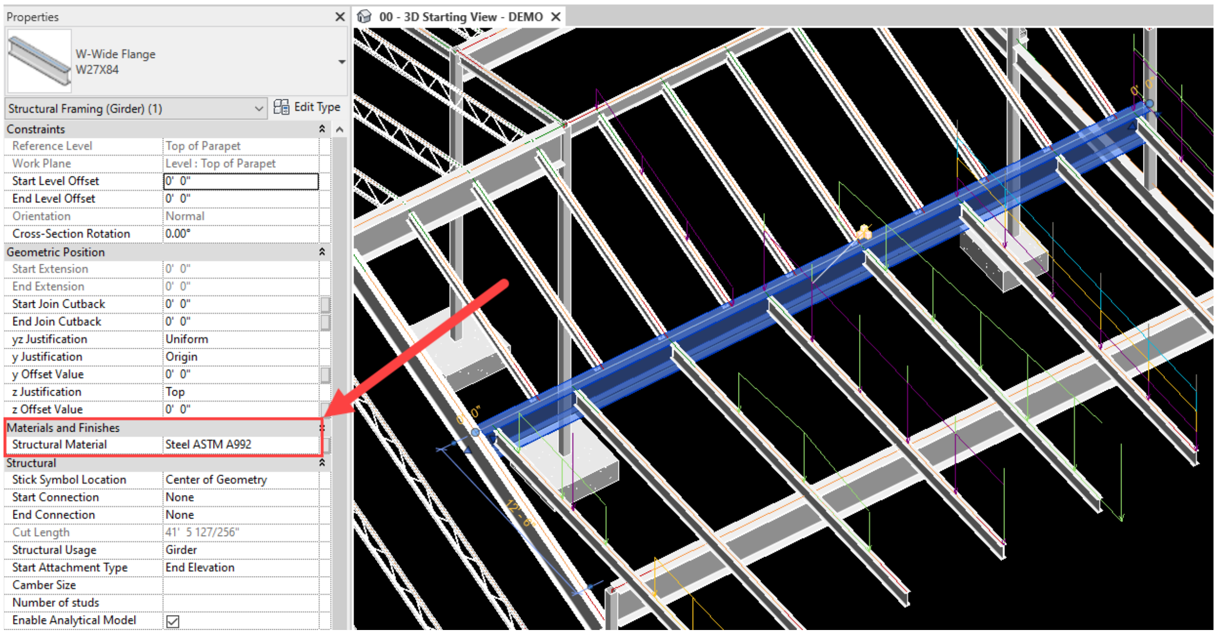
Beam Calculation Labels

In addition to the tools available inside Revit's 3D environment to manage and visualize the structural design, engineering teams may sometimes find it necessary to correlate individual element calculations to their plan locations on a 2D drawing. This could be useful for compiling a calculation package, or any number of other purposes. In order to assist with this process, ENERCALC for Revit provides a beam tag that displays the calculation name assigned via the "Description" field in the ENERCALC SEL interface. This is the same title displayed in the summary table of the Element Manager window. Similar to the reaction tag, this description tag does not need to be manually loaded.

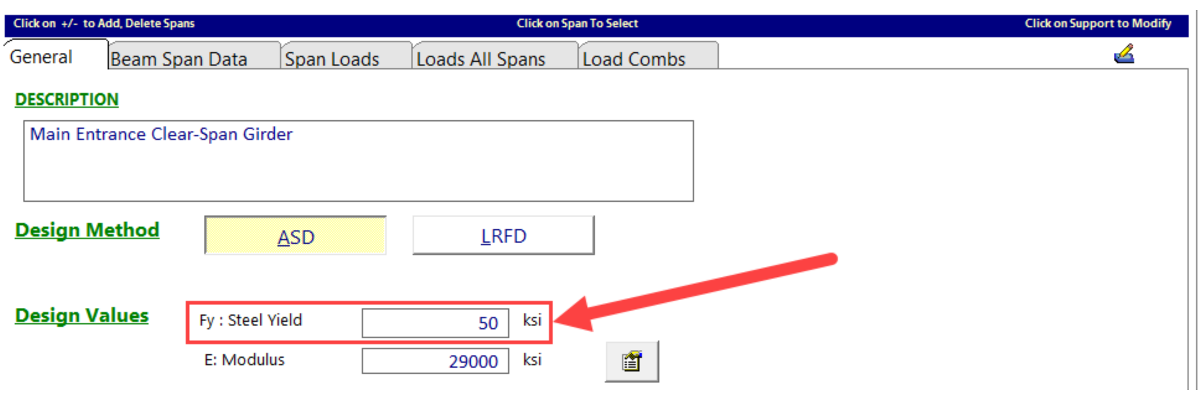


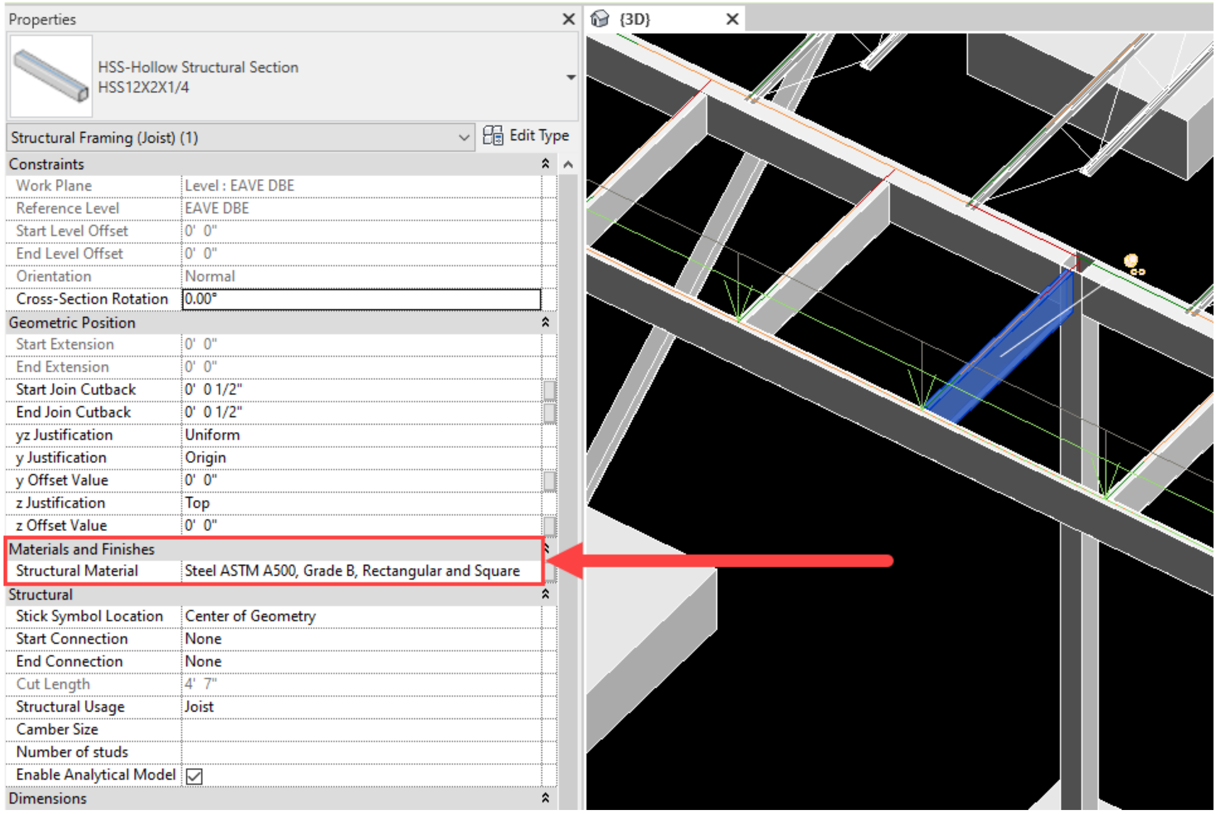
10.11 Steel Beam Calculations

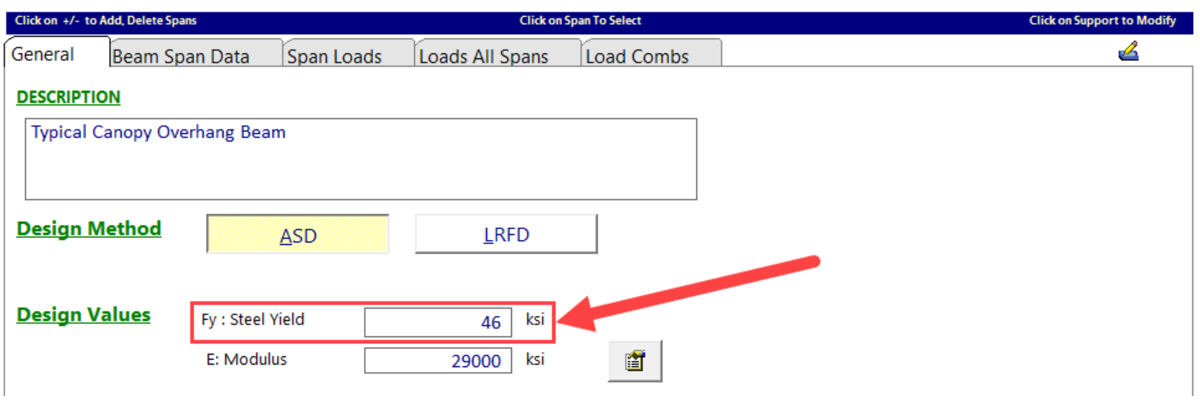
There are very few special considerations that distinguish steel beam calculation launches from the basic beam behavior described in the preceding sections. The only significant additional property of a steel beam is the steel material grade. During launch of a steel beam calculation, the material grade is obtained from the name value of the native Revit parameter “Structural Material”.



Then the steel grade name is extracted from this parameter, it is automatically converted to a corresponding yield strength (in KSI) to be used for the calculation.







Similar to other design parameters, ENERCALC for Revit provides 2-way control of the steel grade. Modifying the numerical value of the Fy yield strength to a standard steel grade value will cause the material name assigned in Revit to update automatically when the “Save and Close” operation finishes. Note that this automatic mapping of names and yield strengths only applies if the default Revit materials (or similarly named materials) are present in the project. Removal of default materials may inhibit the ability to set and manage the steel grade in steel beam calculations. The mapping of these material names and grades is as follows:

Revit Material Name Contains:	ENERCALC SEL Fy:
"A36"	36 ksi
"A500" & "B"	46 ksi
"A572"	50 ksi
"A992"	50 ksi
"Steel" **	50 ksi

** Applies only if no other specific name matches are found

When a steel beam Fy is manually changed in ENERCALC SEL, the material name parameter value will be chosen by using the same matching process in reverse.

10.11.1 Changing Steel Section Size

Steel beam section sizes may be changed from within the ENERCALC interface for any calculation. This is done using the same ENERCALC controls used to modify a conventional manually-built calculation. For more information on this topic, refer to Changing Section Sizes 90.

10.12 Steel Composite Beam Calculations

Similar to Steel Beam calculations, Steel Composite beams share all of the common beam behavior described in the preceding sections.

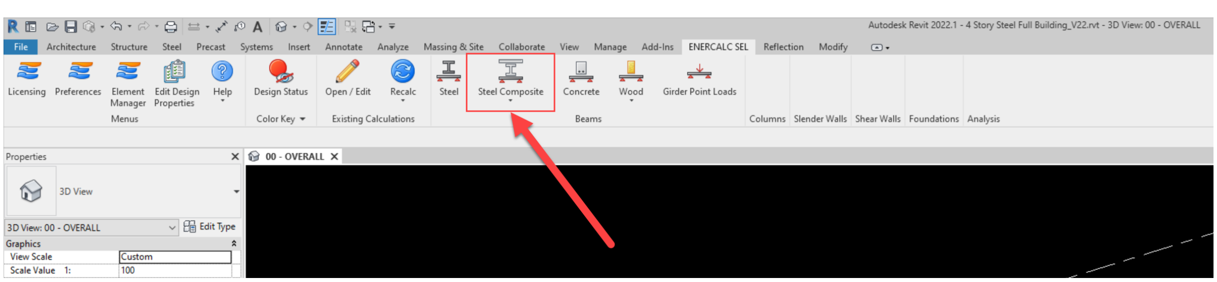
The primary characteristics that distinguish Steel Composite from conventional Steel beam calculations are:

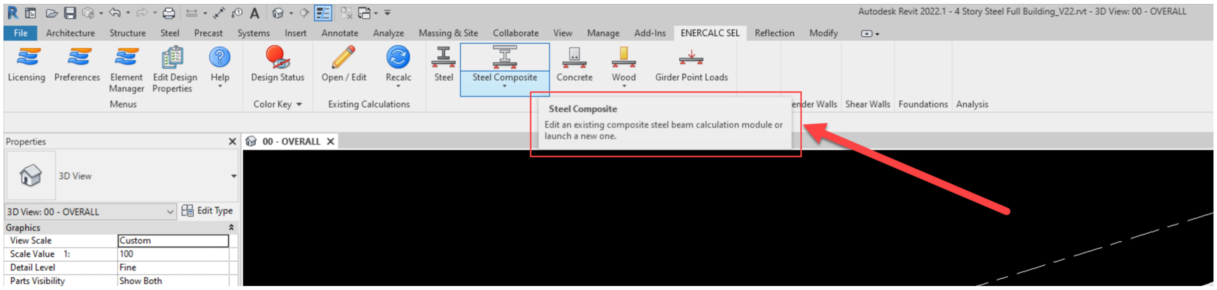
1.) Specifying the pre/post composite application state of loads on the beam

2.) Specifying material properties for the concrete slab

3.) Confirming effective width geometry of the concrete slab

As with other calculations, Steel Composite beam calculations are launched from the ENERCALC for Revit ribbon tab in the Revit interface.

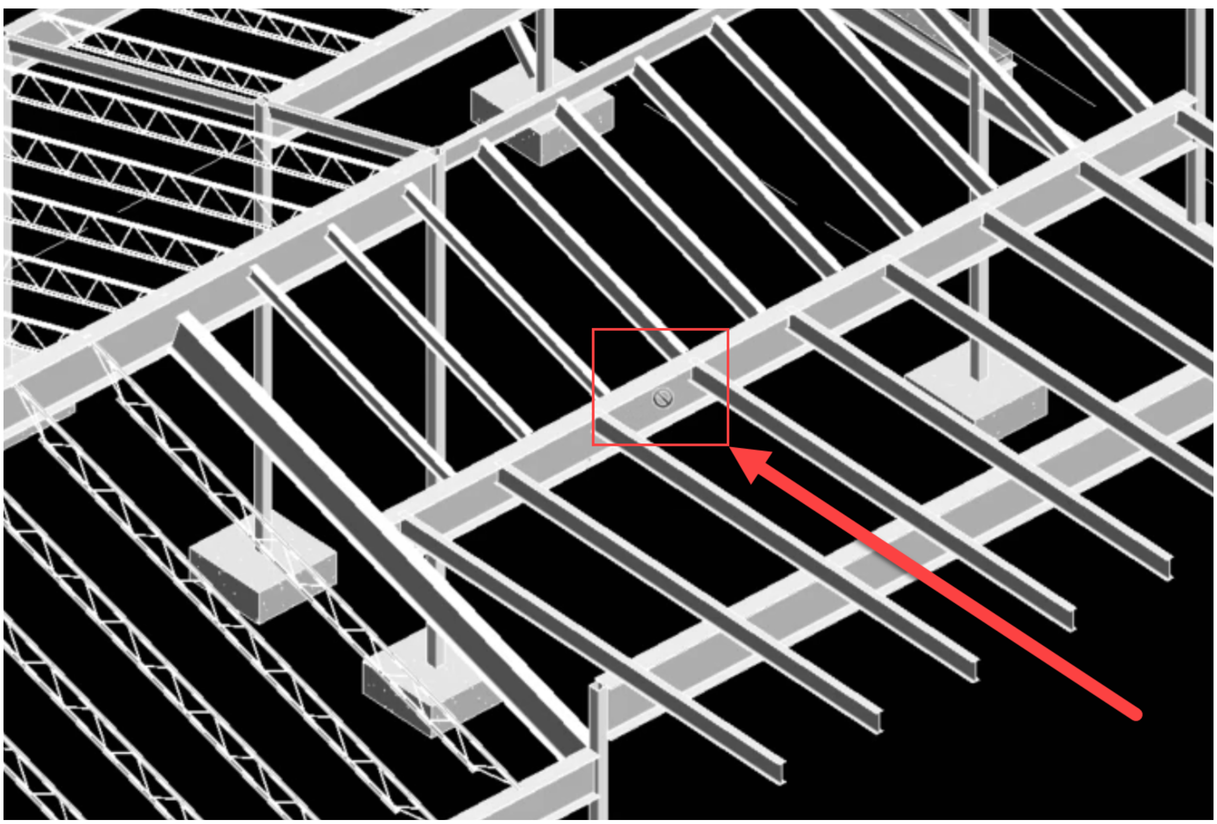




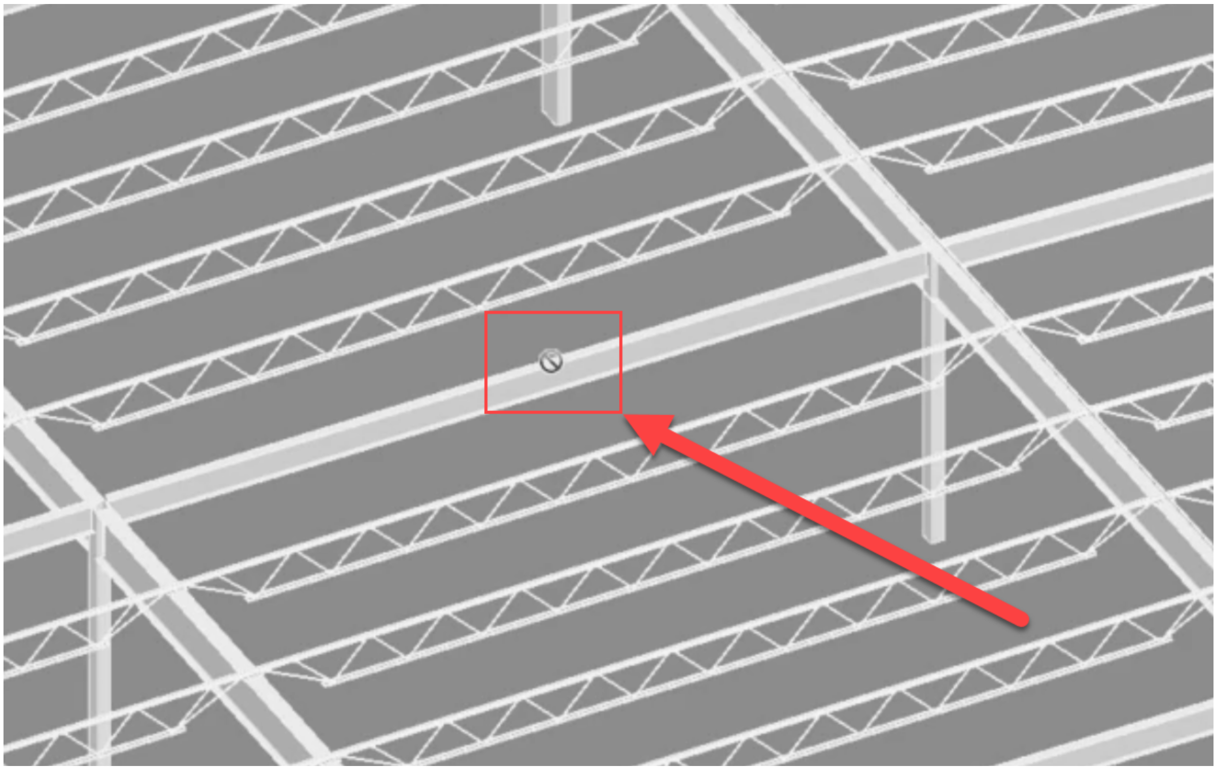
As with other calculation types, clicking the calculation launch button will trigger a selection process and will await an element pick by the user.
During this process, only steel beams that are eligible for Steel Composite design will be available for selection. Ineligible elements are denoted with a "not allowed" circle and slit icon on the cursor when mousing over.

For example:

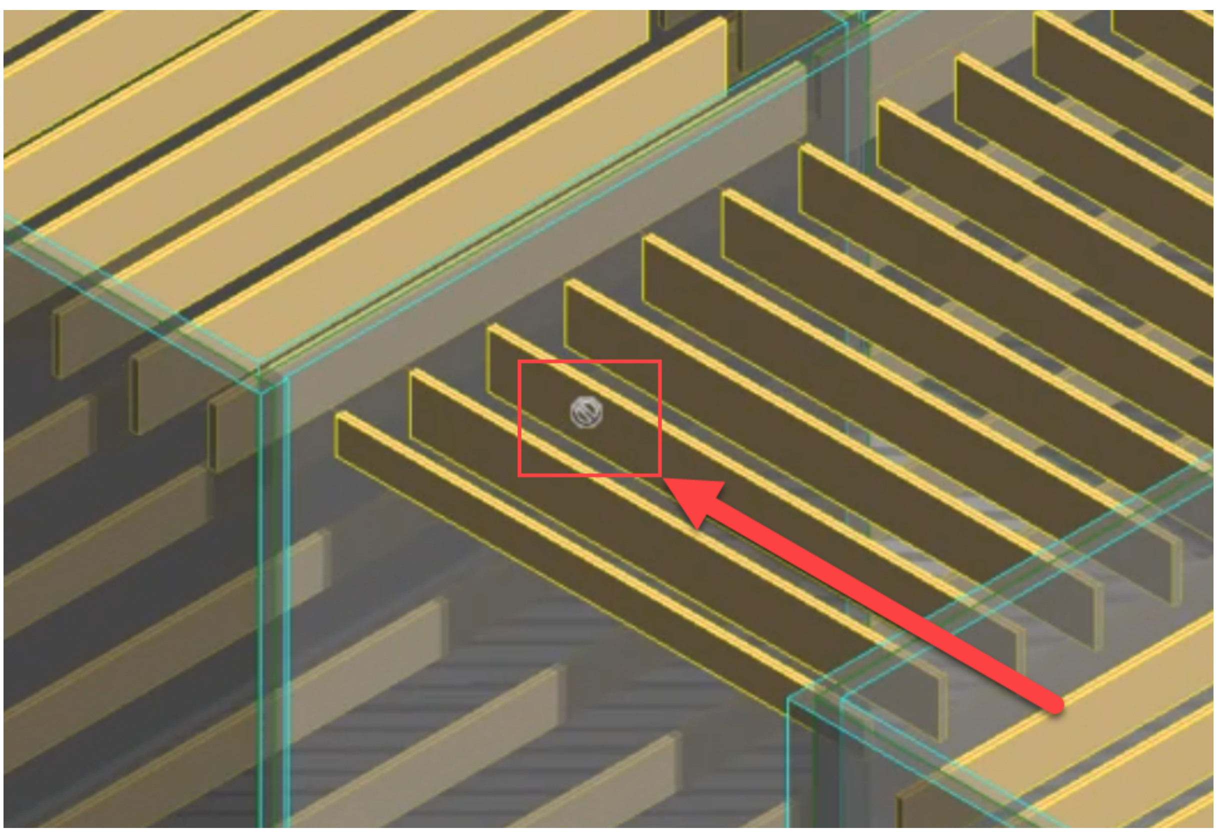
Steel beam without any deck is ineligible:



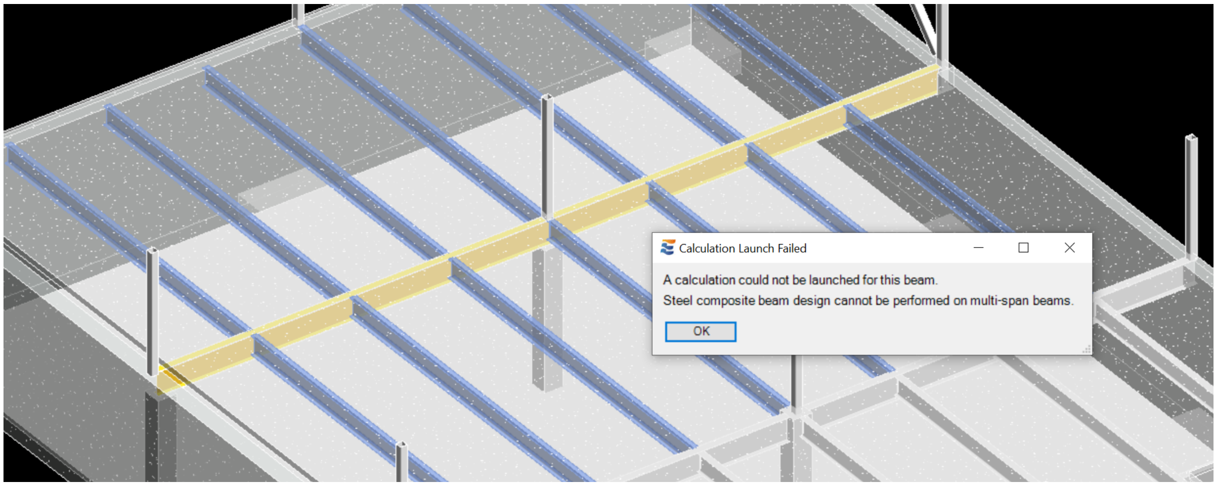
Steel beam with only metal deck (no concrete) is ineligible:



Wood beams (and concrete beam) are ineligible:



Users will also note that Steel Composite beam design is not permitted for multi-span beams. Launch of such beams will result in the following warning:



10.12.1 Load Application States

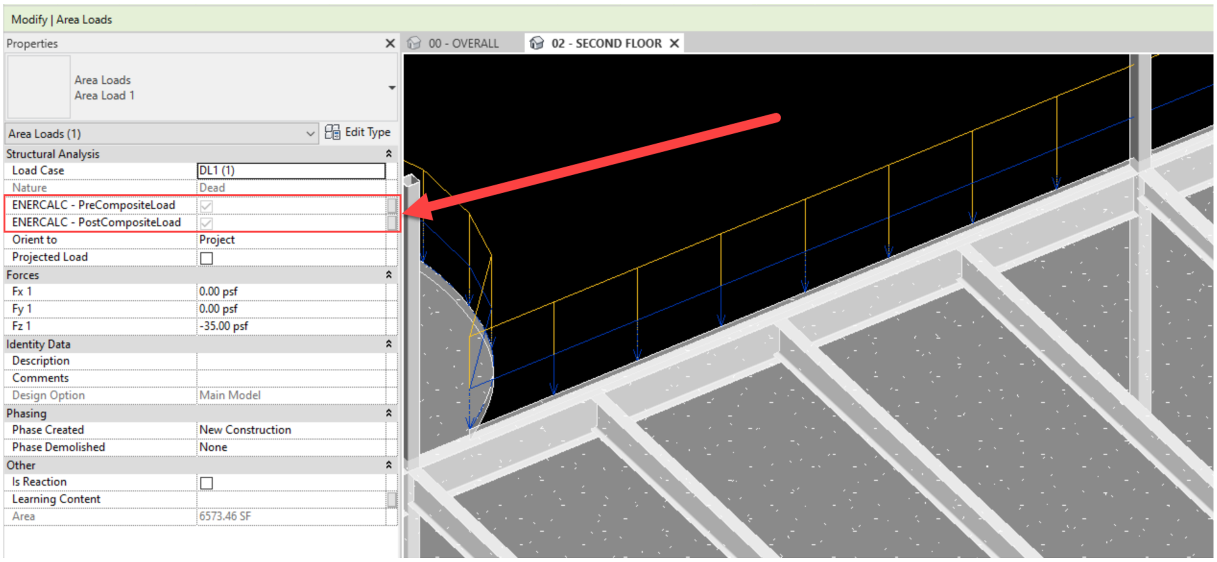
The engineering behavior of a Steel Composite beam design hinges on the timing of load application. In ENERCALC SEL, Loads applied to a composite beam may occur in any of the following (3) conditions:

1. Applied to NonComposite Section - REMOVED before curing

2. Applied to NonComposite Section - REMAINS after curing

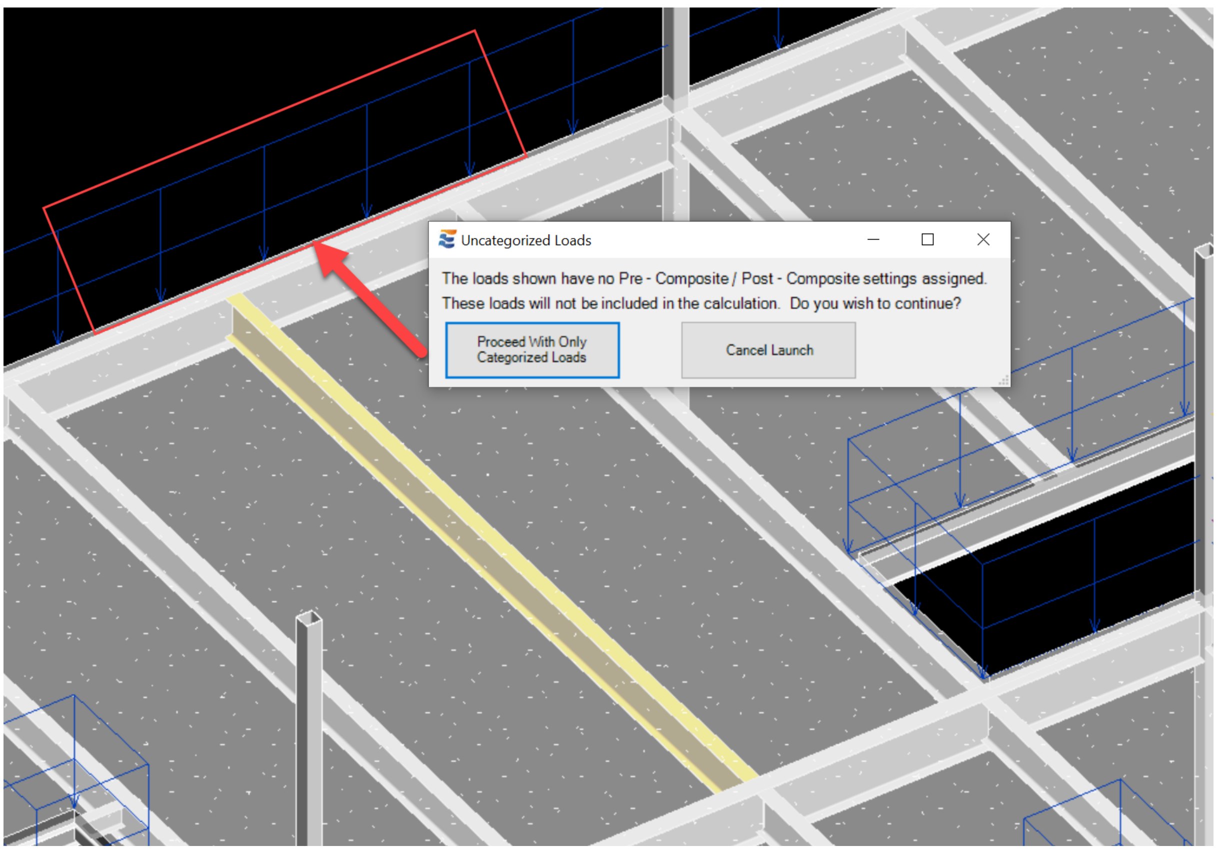
3. Applied to Composite Section ONLY

The user specifies the application state of each applied load found in the Revit model. The application state of Revit loads are set using the parameters found on the "Properties" pane when the load is selected. The default condition for an un-categorized load is for both check-boxes to be grayed out:

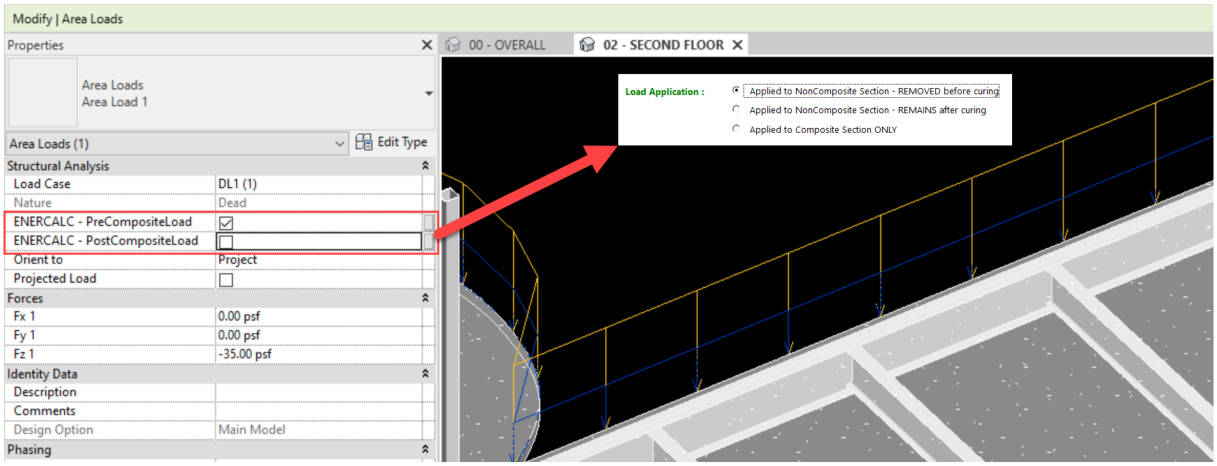


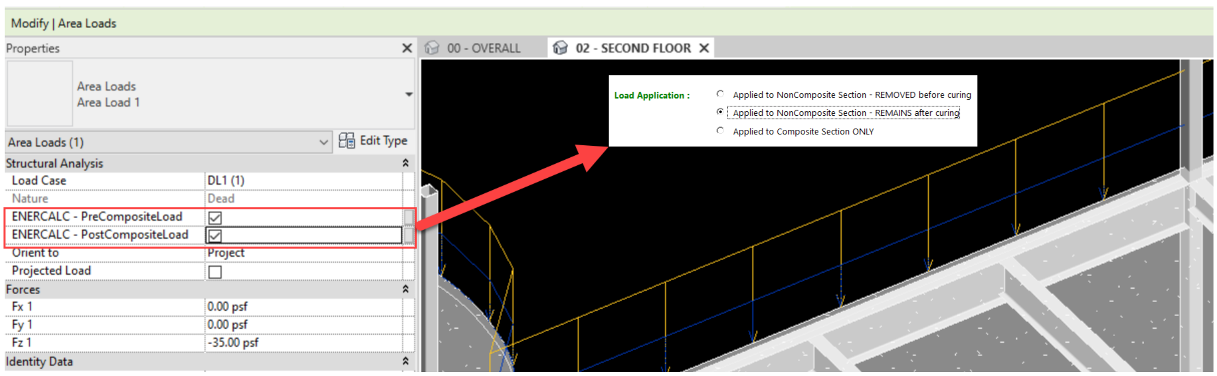
When launching a calculation, the user will be notified with a pop-up if any loads found on the beam do not yet have a load application state applied.

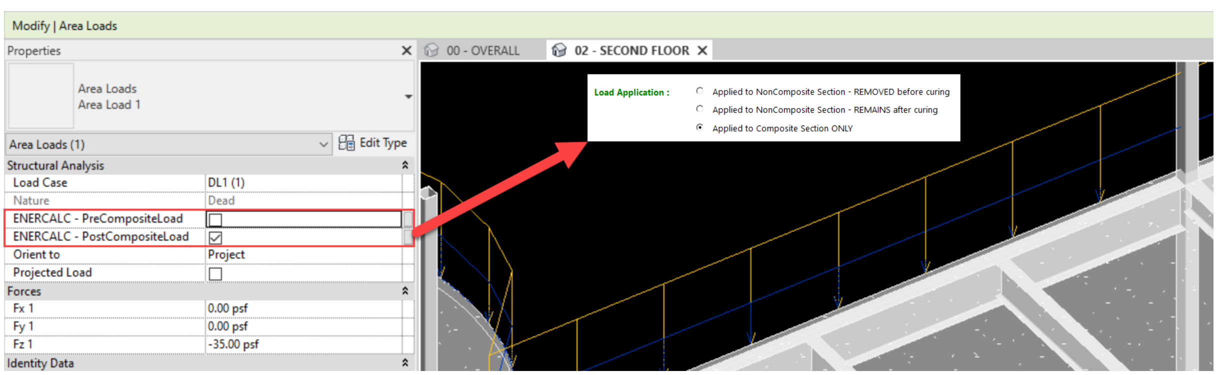
If the loads are visible (per the Visibility/Graphics settings), then the referenced loads will be selected for user attention:



The mapping of these two check-box parameters to the ENERCALC application states is as follows:





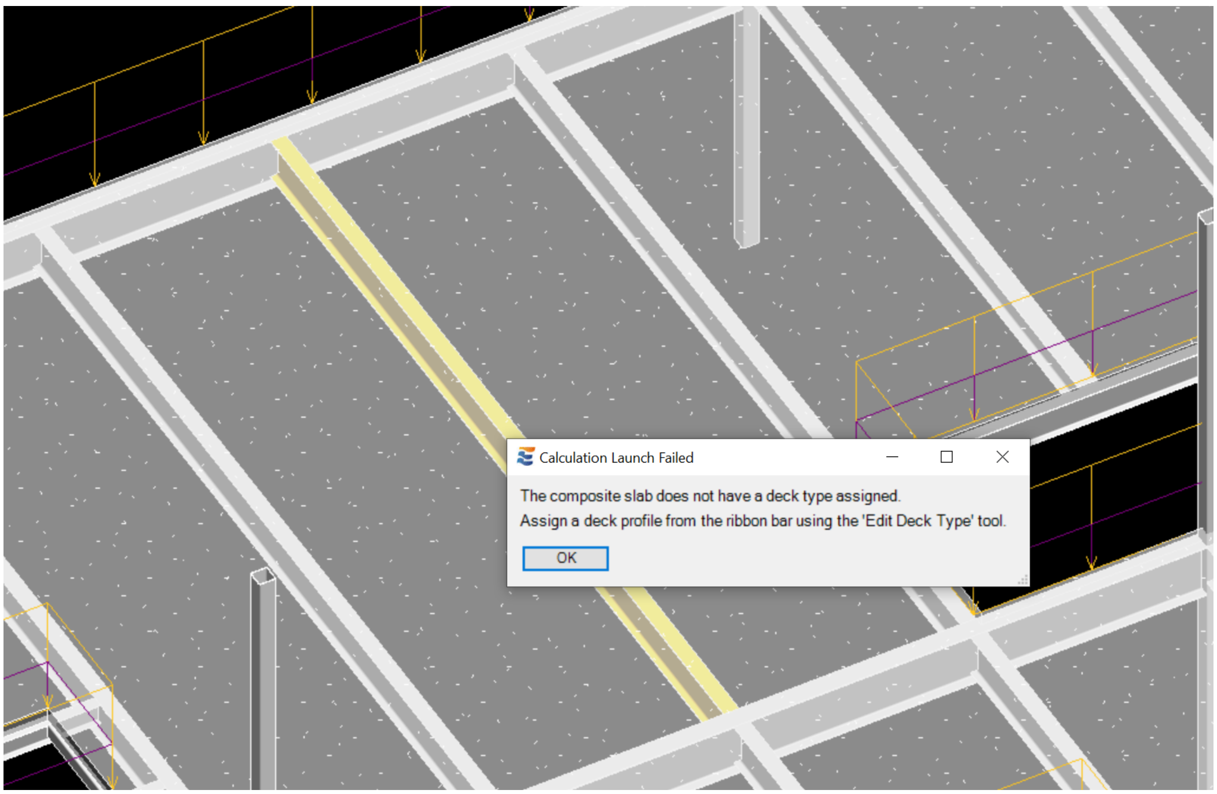


When setting these check boxes, users should keep in mind that each load case may only be eligible for specific application states. For example, wind and snow load are not eligible to be applied as "Applied to NonComposite Section - Removed Before Curing" in an ENERCALC Composite Steel Beam calculation. If a Revit load's parameters are manually set to an application state not supported by ENERCALC, they will be automatically set to a supported state.

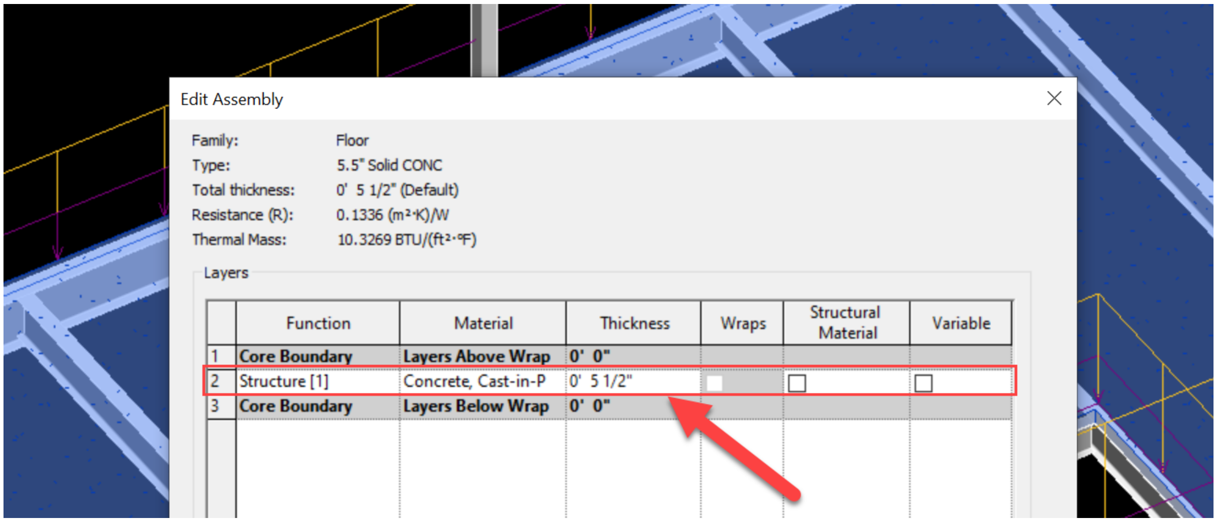
NOTE: It is only necessary for users to specify the application state of load cases that can be applied at different stages (D and L). All other load cases (Lr, S, W, E, & H), will be automatically applied as "Applied to Composite Section ONLY" regardless of whether the user has explicitly toggled the parameters. There is no ambiguity for these cases because ENERCALC only permits them to be applied on the composite section.

10.12.2 Slab Properties

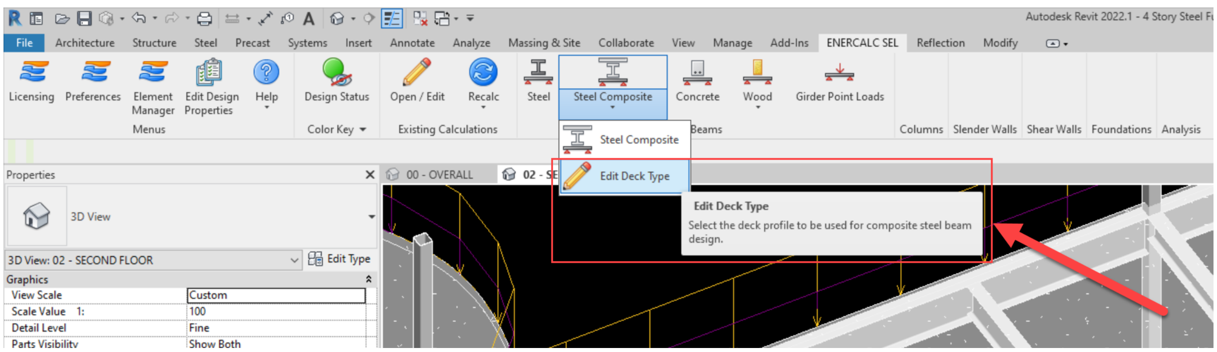
The structural design of a Steel Composite beam is strongly influenced by the properties of the concrete slab. When launching a calculation, the user will be notified with a pop-up if the properties of the slab have not yet been set:



NOTE: This warning will not be presented if the floor element is a solid concrete slab with no "structural deck" layer. When a floor element has concrete but no deck, then a profile assignment is not required:

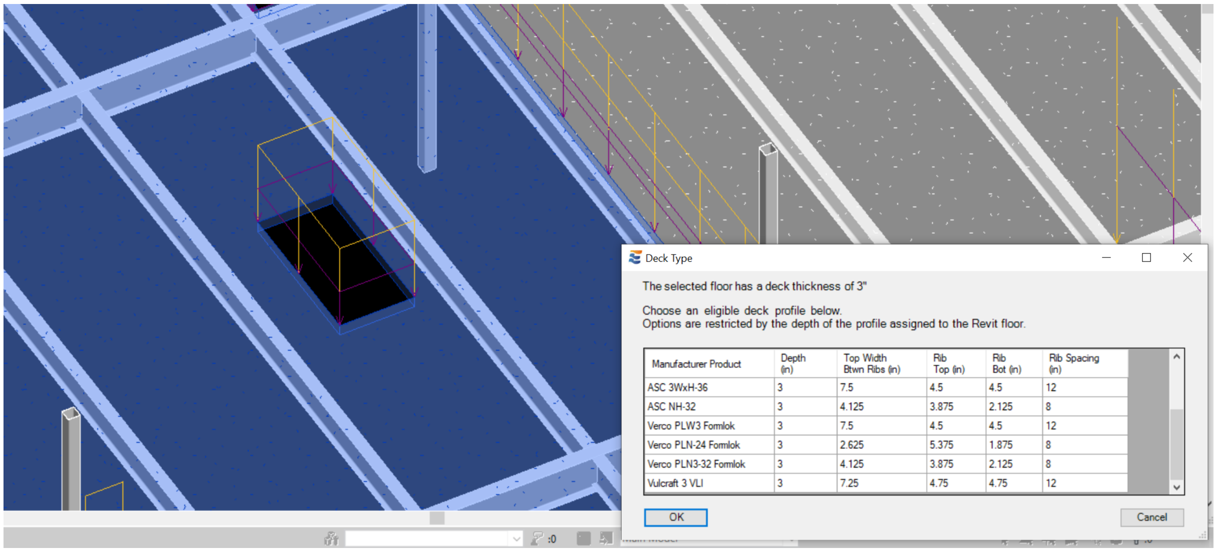


Where required, the deck type is assigned or modified using the "Edit Deck Type" tool on the ribbon bar:

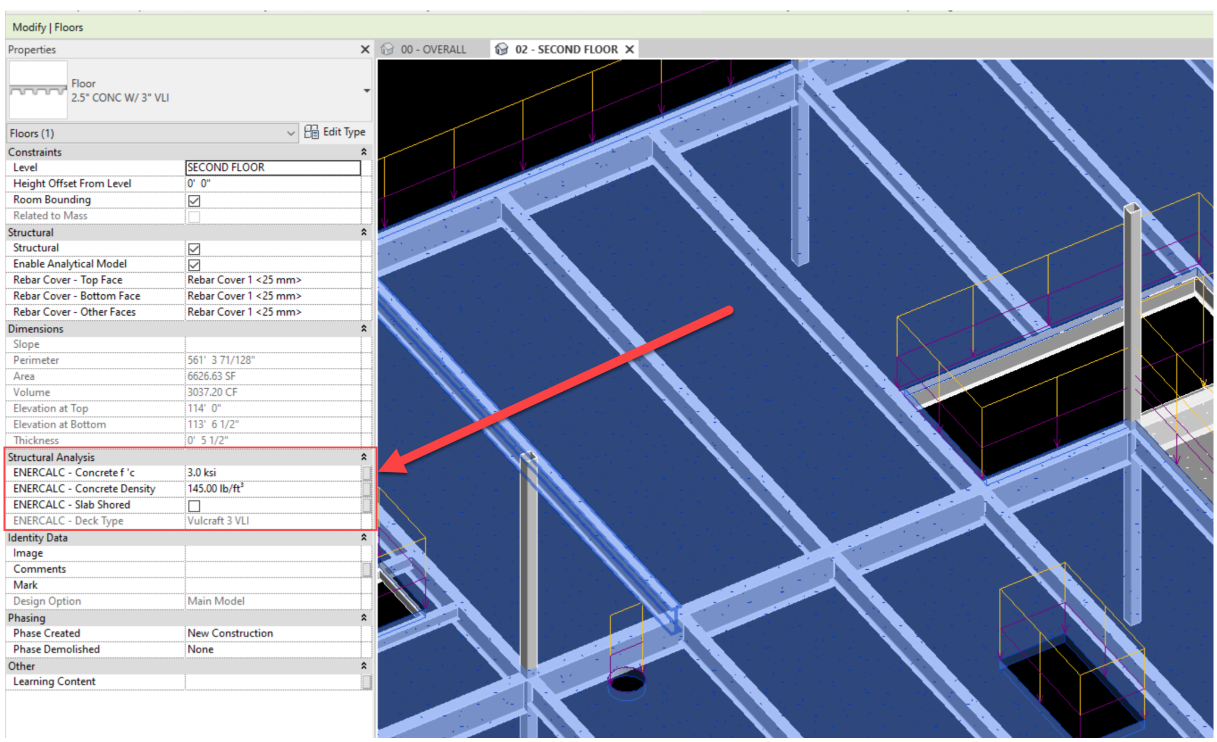


Clicking the "Edit Deck Type" button will trigger a selection process and EFR will await a floor selection by the user.

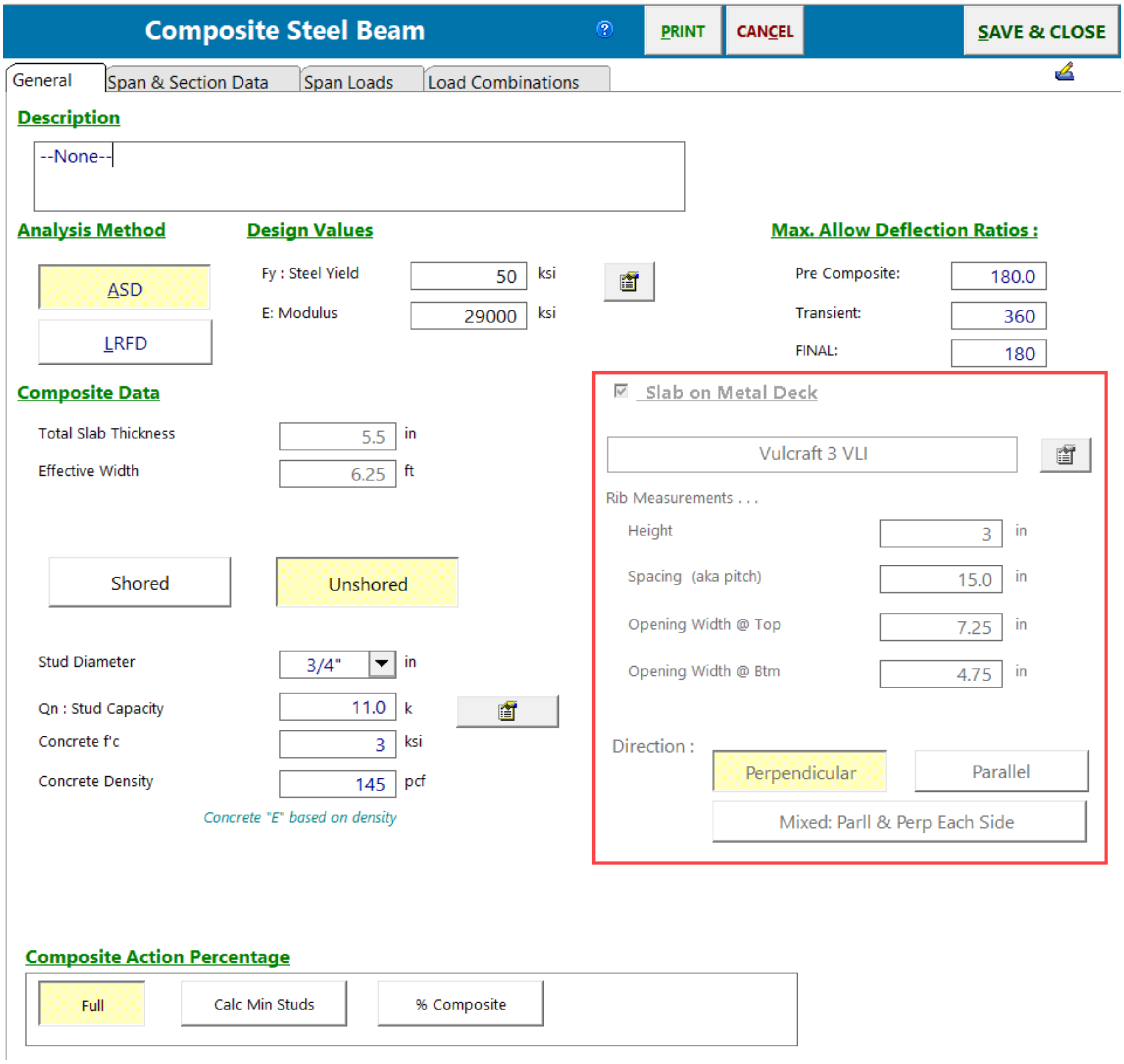
When a floor element has been selected, the deck selection menu will load. The deck profile options populated in the menu are automatically restricted to match the depth of the deck profile assigned to the floor element in Revit:



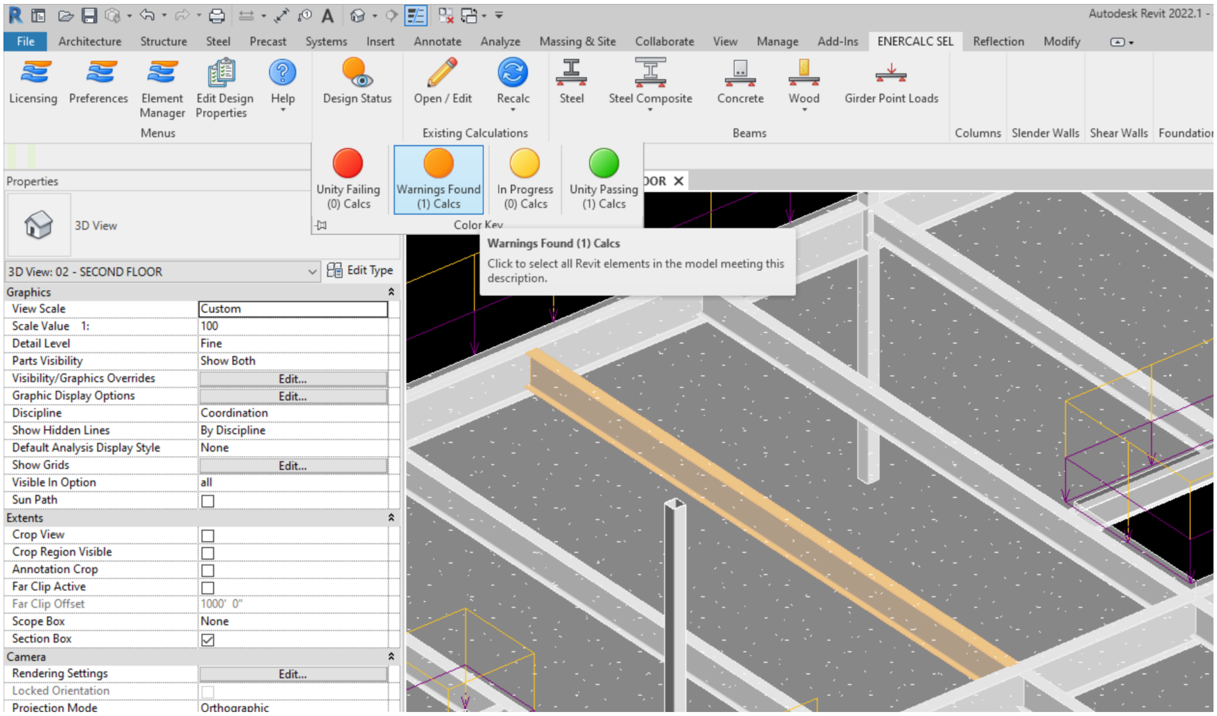
Once selected, the deck profile name is indicated in the Revit "Properties" pane for the floor element. The other properties (compressive strength, unit wt. and shored / unshored) are set directly in the "Properties" pane:

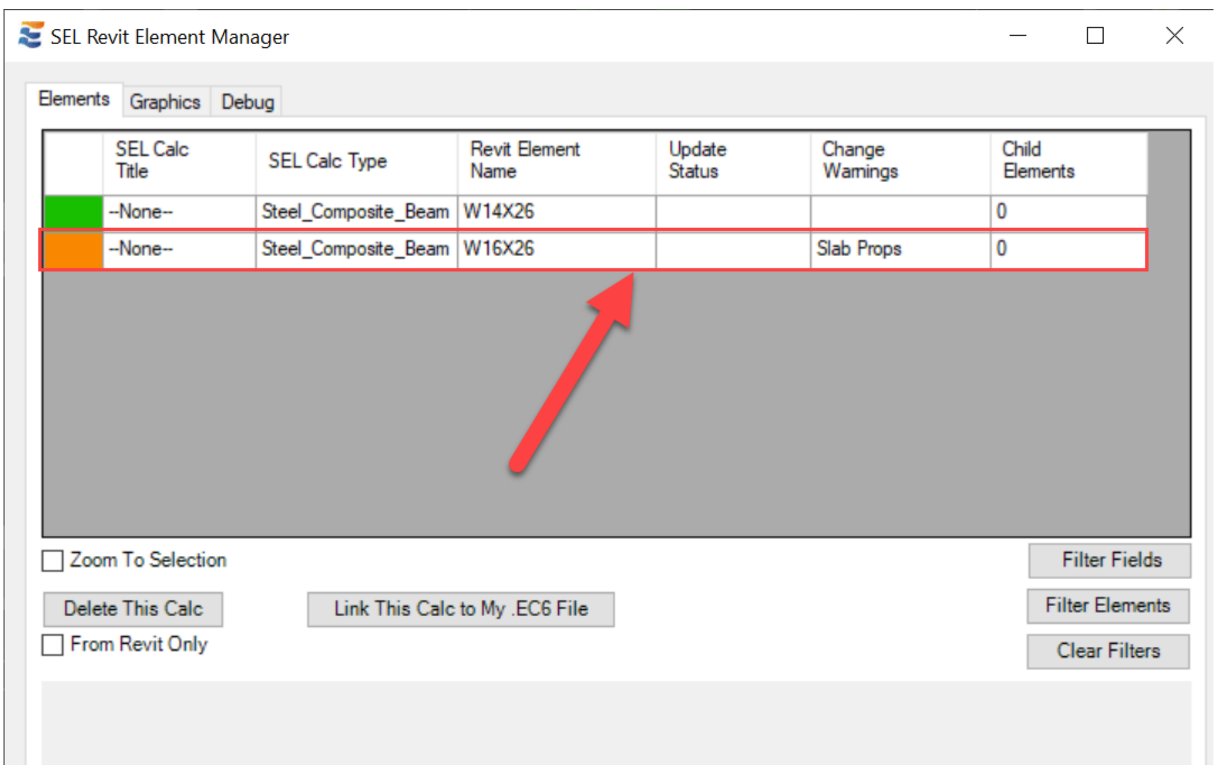


Users should note that the slab section geometry properties cannot be modified from the ENERCALC interface:



The mechanical properties of the slab (f'c and density) may be changed in the ENERCALC interface. Any changes to these properties in ENERCALC will be applied to the slab element in Revit when the "Save and Close" operation completes. In some cases, design parameter changes made in ENERCALC may impact a slab that also influences other Steel Composite beam calculations already present in the Revit model. In such cases, the affected existing calculations will automatically be flagged with the change warning "Slab Props":

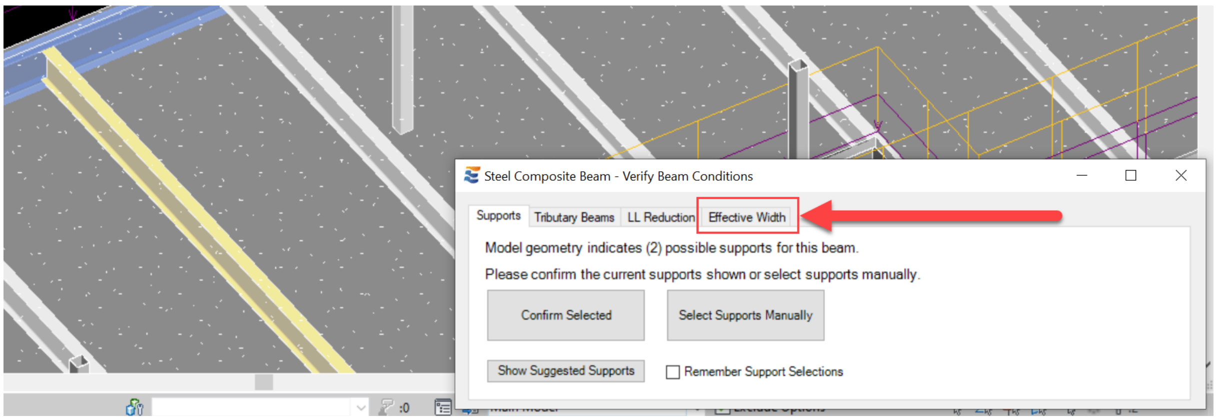

Autodesk Revit 2022.1
File
Architecture
Structure
Steel
Precast
Systems
Insert
Annotate
Analyze
Massing & Site
Collaborate
View
Manage
Add-Ins
ENERCALC SEL
Reflection
Modify
Licensing
Preferences
Element Manager
Edit Design Properties
Help
Menus
Design Status
Open / Edit
Recalc
Existing Calculations
Steel
Steel Composite
Concrete
Wood
Girder Point Loads
Beams
Columns
Slender Walls
Shear Walls
Unity Failing (0) Calcs
Warnings Found (1) Calcs
In Progress (0) Calcs
Unity Passing (1) Calcs
Color Key
Warnings Found (1) Calcs
Click to select all Revit elements in the model meeting this description.
Properties
3D View
3D View: 02 - SECOND FLOOR
Edit Type
Graphics
View Scale
Custom
Scale Value 1:
100
Detail Level
Fine
Parts Visibility
Show Both
Visibility/Graphics Overrides
Edit...
Graphic Display Options
Edit...
Discipline
Coordination
Show Hidden Lines
By Discipline
Default Analysis Display Style
None
Show Grids
Edit...
Visible In Option
all
Sun Path
Extents
Crop View
Crop Region Visible
Annotation Crop
Far Clip Active
Far Clip Offset
1000' 0"
Scope Box
None
Section Box
Camera
Rendering Settings
Edit...
Locked Orientation
Projection Mode
Orthographic


SEL Revit Element Manager
Elements
Graphics
Debug
SEL Calc Title
SEL Calc Type
Revit Element Name
Update Status
Change Warnings
Child Elements
--None--
Steel_Composite_Beam
W14X26
0
--None--
Steel_Composite_Beam
W16X26
Slab Props
0
Zoom To Selection
Filter Fields
Delete This Calc
Link This Calc to My .EC6 File
Filter Elements
From Revit Only
Clear Filters

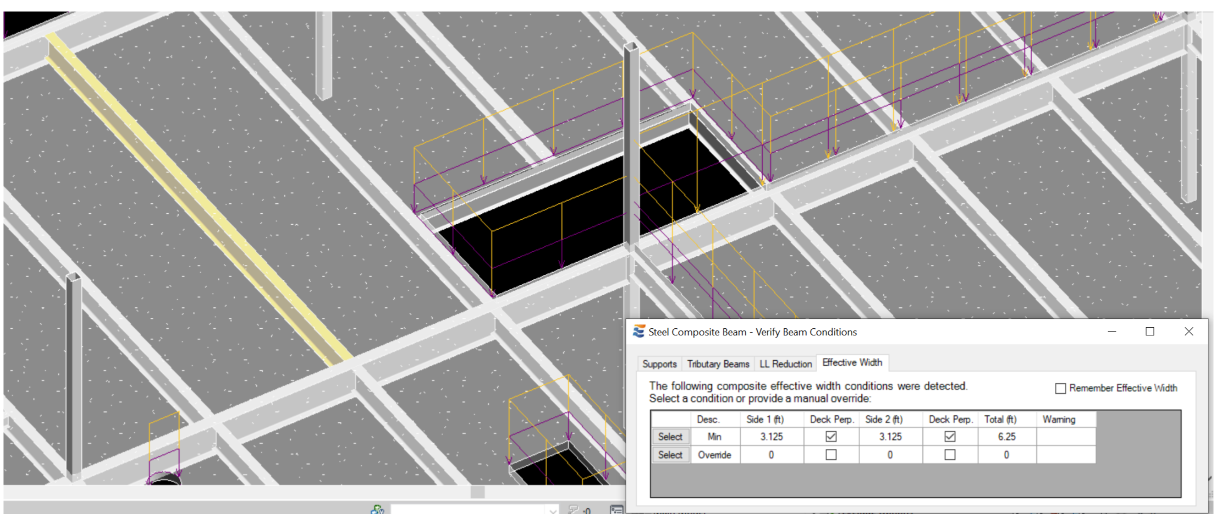
10.12.3 Effective Width

The effective width of a floor slab is critical to the design of a Steel Composite beam. ENERCALC for Revit automatically detects effective width using the physical geometry of the Revit model.

During launch of the calculation, the launch window will show an additional tab for managing the effective width conditions:



After the preceding tabs' approvals are completed, the beam effective width will be computed and presented to the user in a tabular format. The calculation will launch in ENERCALC once the user chooses an effective width condition by clicking the "Select" button on the table row that describes the desired geometry. Users should note that once the calculation has launched the effective width inputs cannot be modified from the SEL interface, since they are driven by Revit geometry.

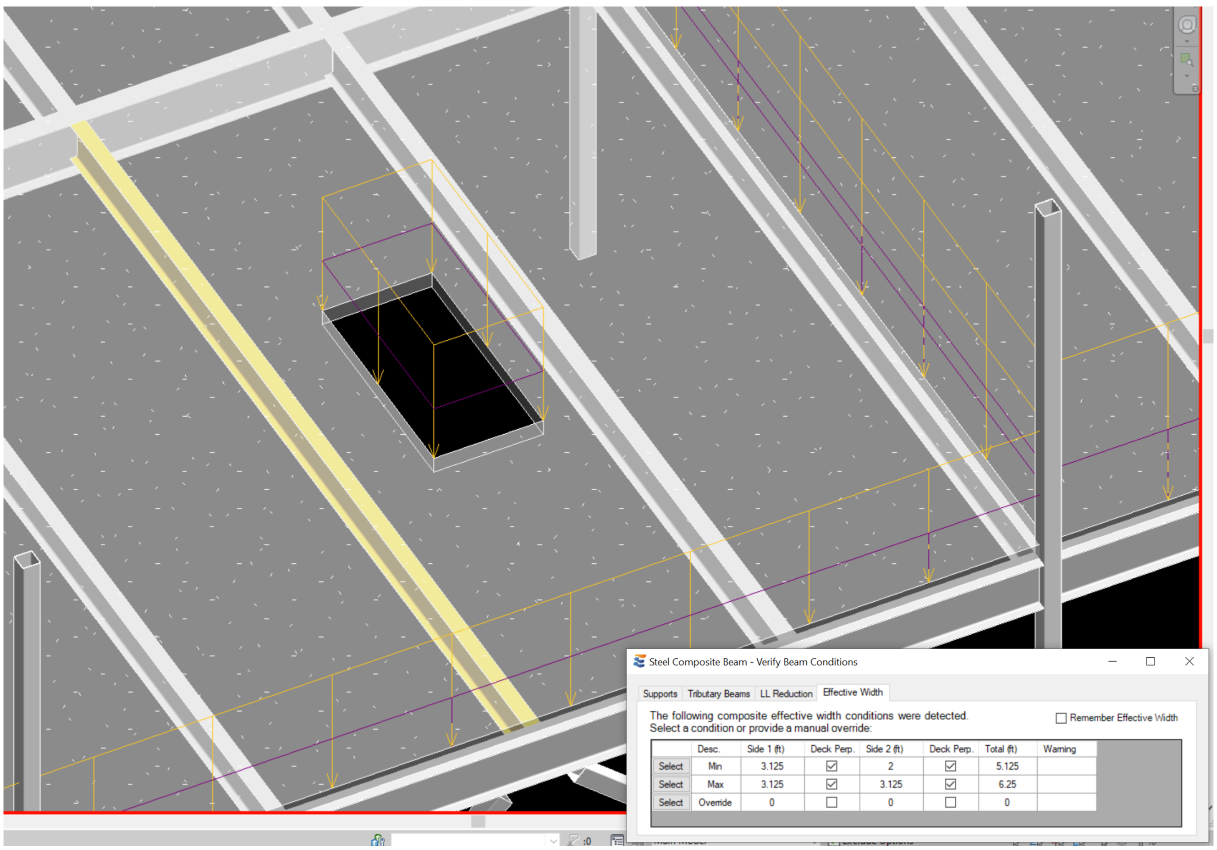


The table presents a set of deck orientation and effective width conditions found on each side of the beam. The table rows showing auto-detected geometry are read-only and cannot be manually modified. If custom geometry is needed, refer to the procedure for using the "Override" row outlined near the end of this section.

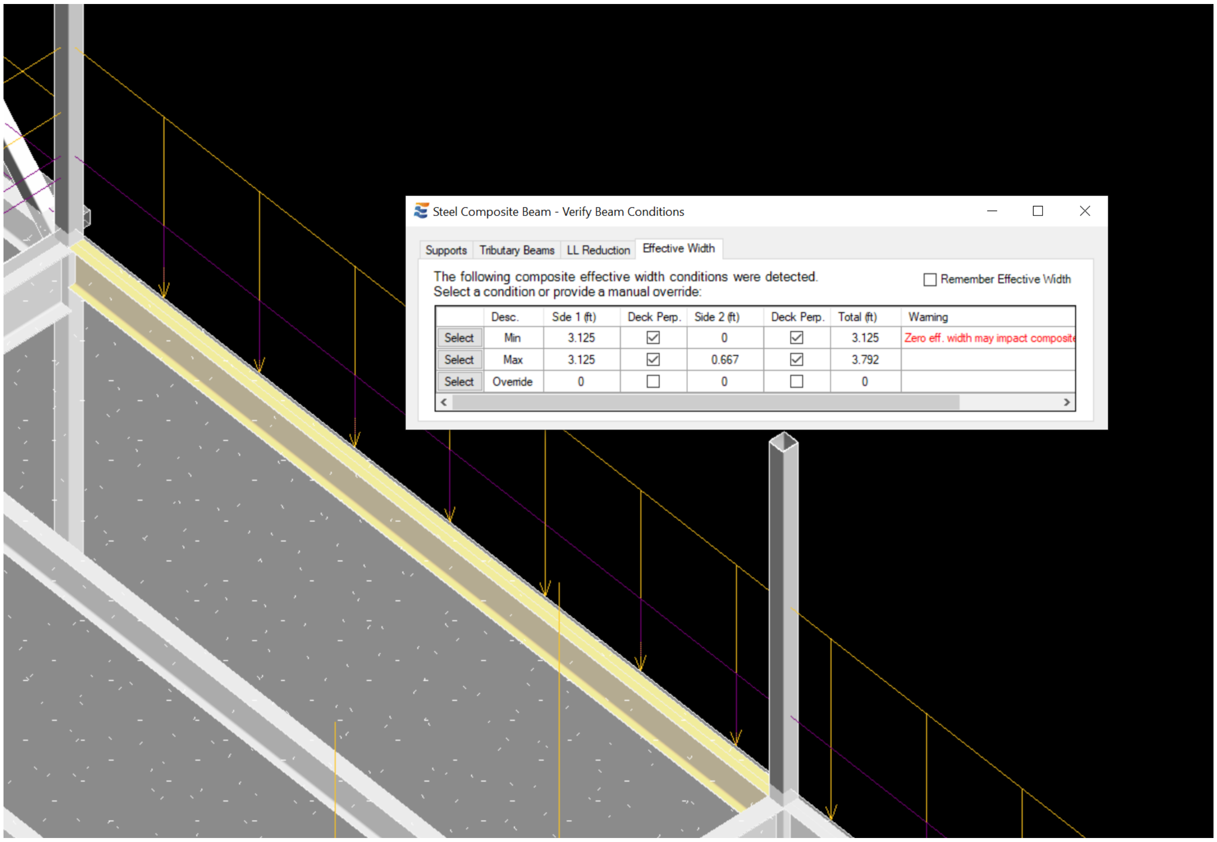
The effective width values shown are computed from the physical geometry of the Revit model, using the minimum of the 3 criteria specified in AISC for composite beam behavior:

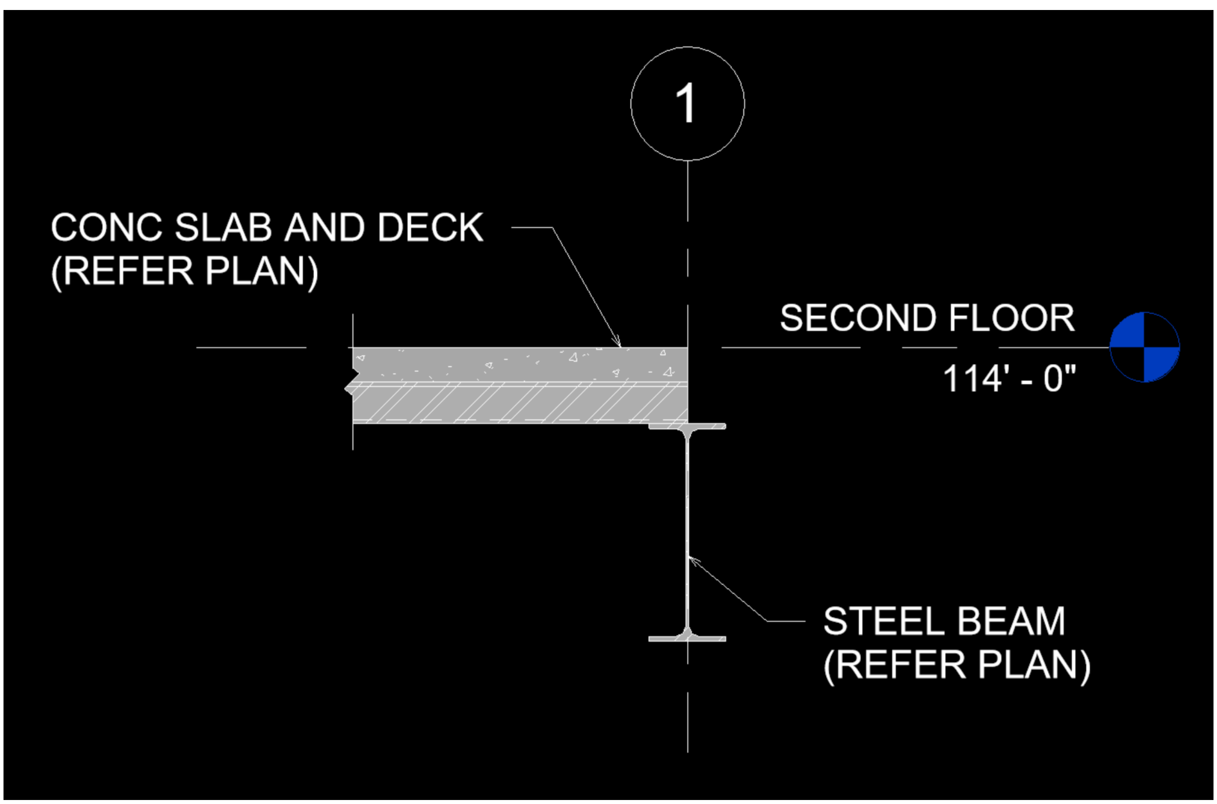
1. One-eighth of the beam span, center-to-center of supports
2. One-half the distance to the centerline of the adjacent beam
3. The distance to the edge of the slab

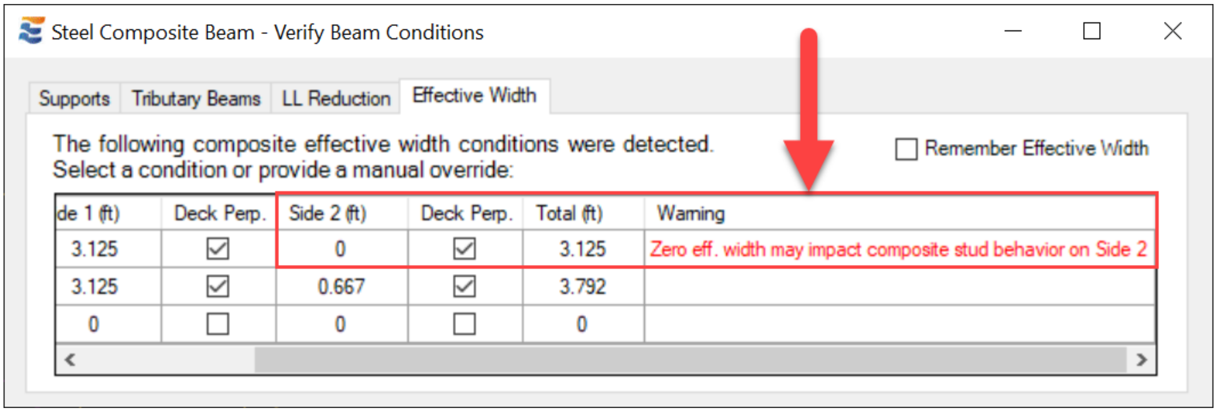
When the effective width is found to vary along the length of the beam, multiple table entries will be presented to the user:



When an effective width condition is encountered which may invalidate the design, a warning is presented in the table. In the case shown here, the user is alerted that launching an edge beam calculation where 100% of the effective width falls only on one side of the beam could jeopardize the shear transfer behavior of the headed studs:

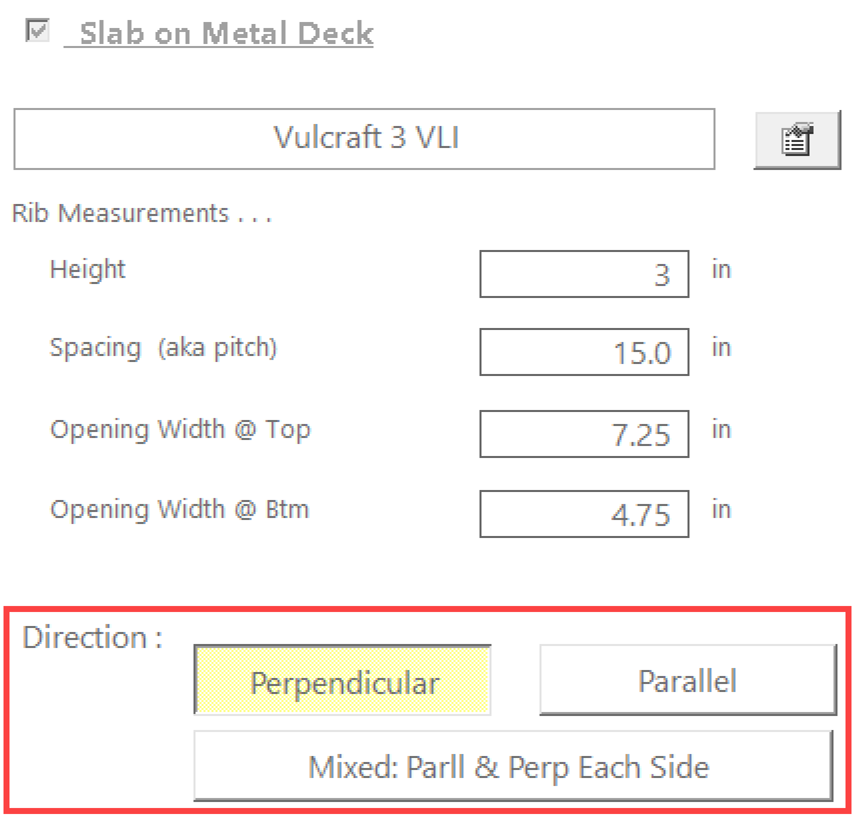

Steel Composite Beam - Verify Beam Conditions
Supports
Tributary Beams
LL Reduction
Effective Width
The following composite effective width conditions were detected.
Select a condition or provide a manual override:
Remember Effective Width
Desc. Sde 1 (ft) Deck Perp. Side 2 (ft) Deck Perp. Total (ft) Warning
Select Min 3.125 0 3.125 Zero eff. width may impact composite
Select Max 3.125 0.667 3.792
Select Override 0 0 0


1
CONC SLAB AND DECK
(REFER PLAN)
SECOND FLOOR
114' - 0"
STEEL BEAM
(REFER PLAN)

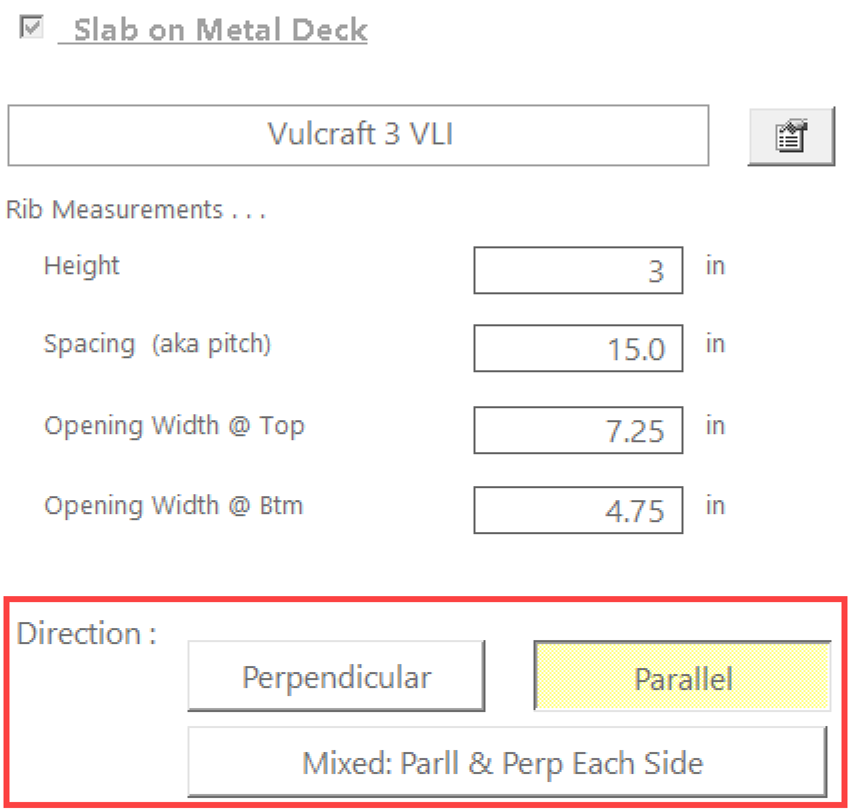


The check-boxes for "Deck Perp" on each side of the beam indicate the observed span direction of the deck. When the check-box is "checked", the deck is observed to span in any orientation NOT PARALLEL to the beam. When the check-box is "unchecked", the deck is observed to span PARALLEL to the beam. The observed span direction does not influence the design in the case of solid concrete slab without metal deck.

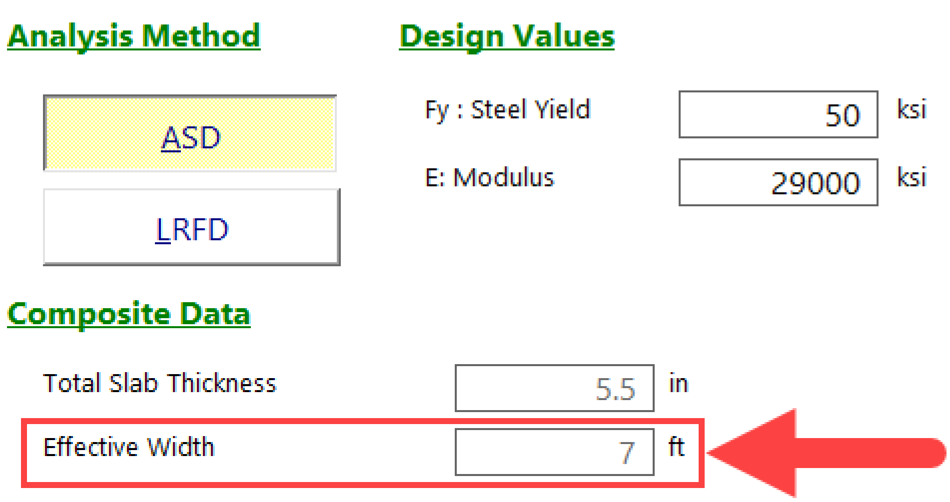
If the check-boxes for "Deck Perp" on each side of the beam are both in agreement as "checked", then the beam calculation will map to SEL as follows:



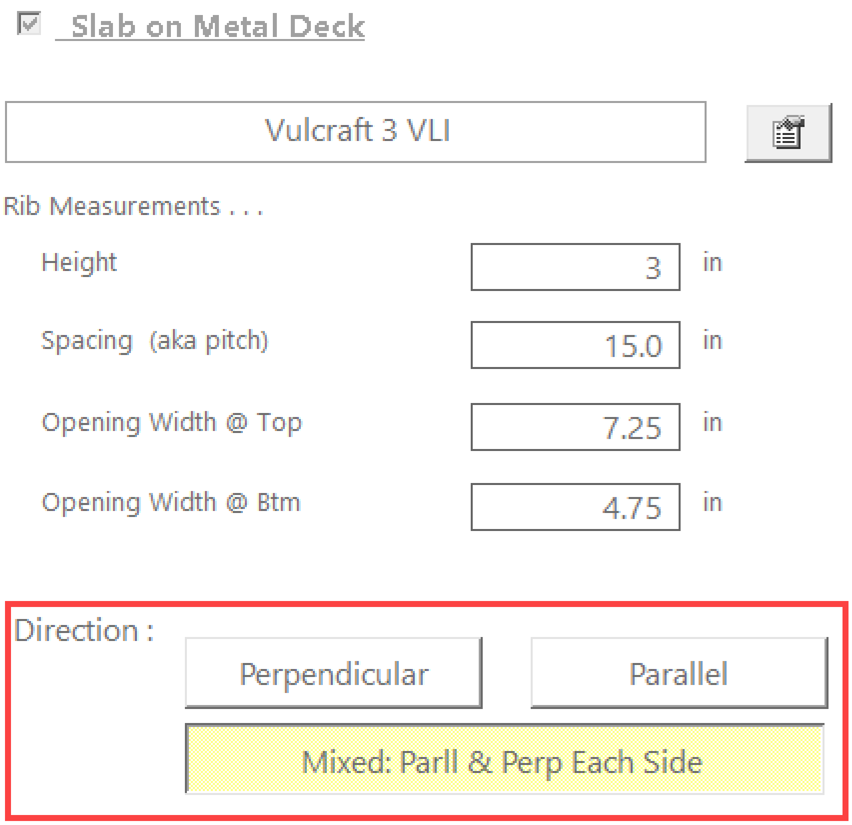
If the check-boxes for "Deck Perp" on each side of the beam are both in agreement as "unchecked", then the beam calculation will map to SEL as follows:



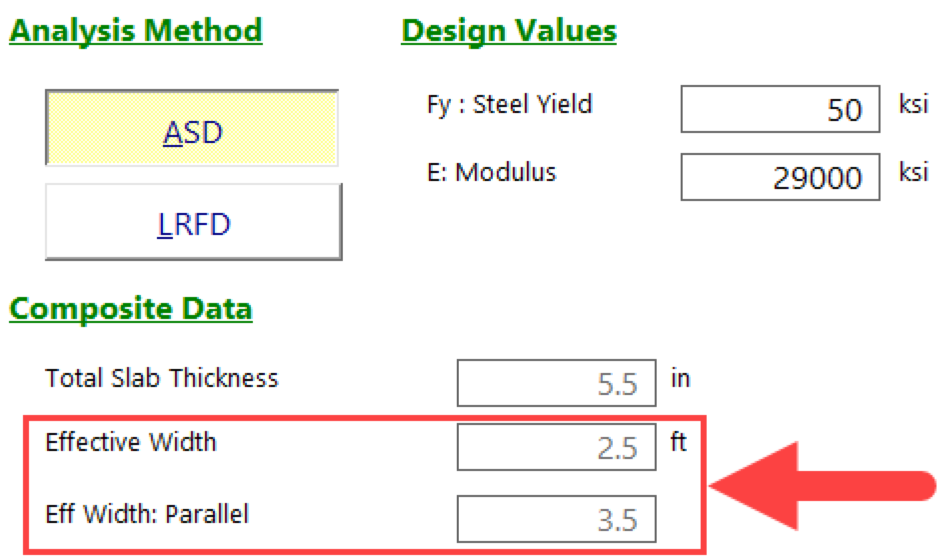
In either of these cases, the "Effective Width" field in SEL will indicate the "Total" value shown in the launch window table:



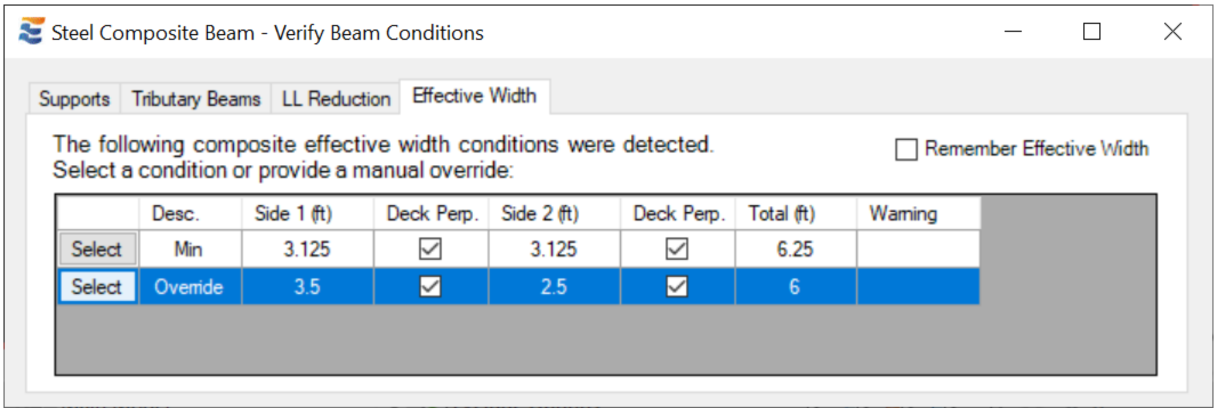
If the check-boxes for "Deck Perp" on each side of the beam are NOT in agreement, then the beam calculation will map to SEL as follows:



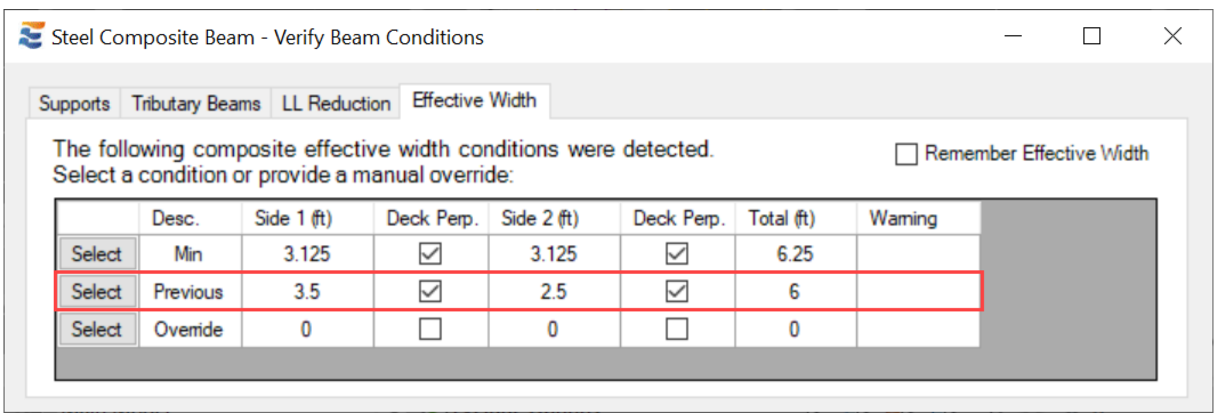
Under this condition, the effective width will display with the two sides individually:



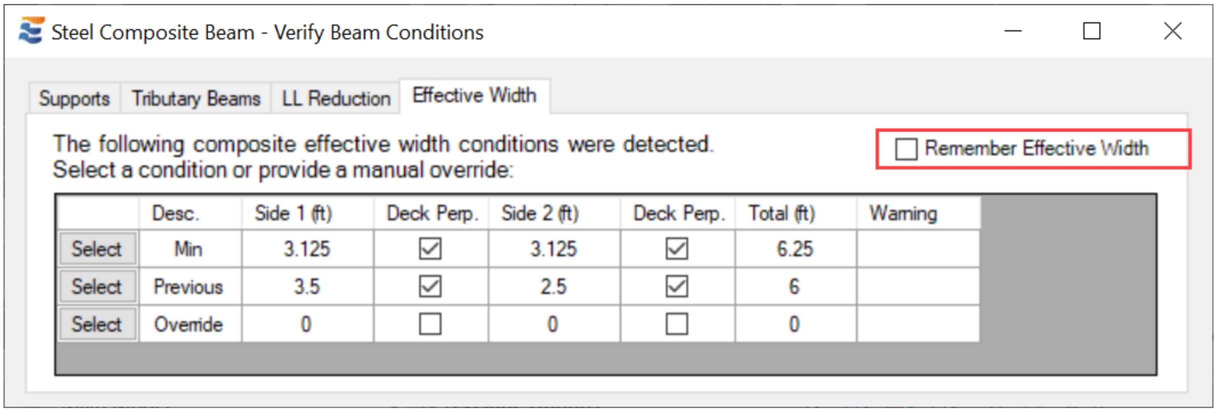
Rather than choosing a calculated value from the table, the user may alternatively choose to create a custom effective width geometry using the table row labeled "Override". The input values for effective width and span direction on each side are filled manually by the user, and the total effective width updates automatically. The calculation will not launch if the "Override" row "Select" button is clicked when the total effective width is zero.



When an existing Steel Composite beam calculation is launched from Revit, the user will by default be presented with an option in the effective width table showing the effective width geometry used during the previous launch:

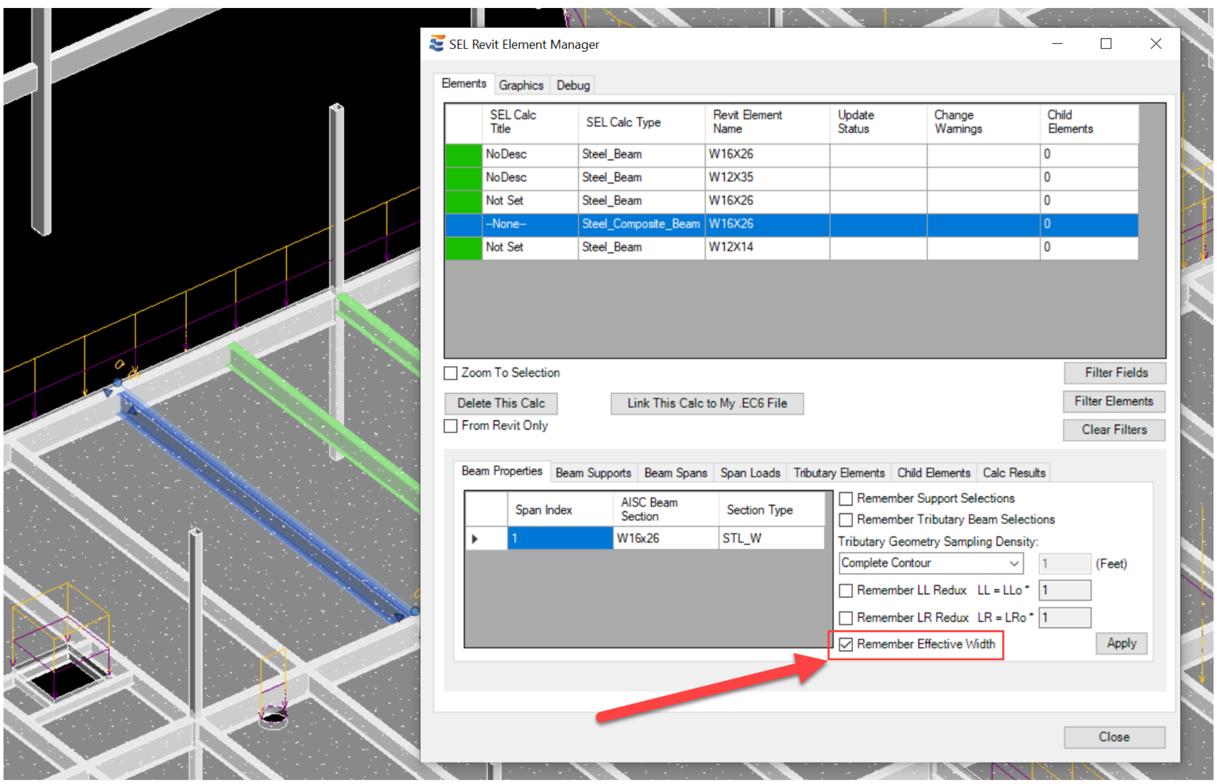


As with other aspects of the calculation launch process, the user may elect to "Remember" the specified tributary width conditions.
This will cause this approval step to be skipped during subsequent launches of the calculation:



For a calculation set to "Remember Effective Width", this approval step will be skipped indefinitely on future launches until the setting is changed.

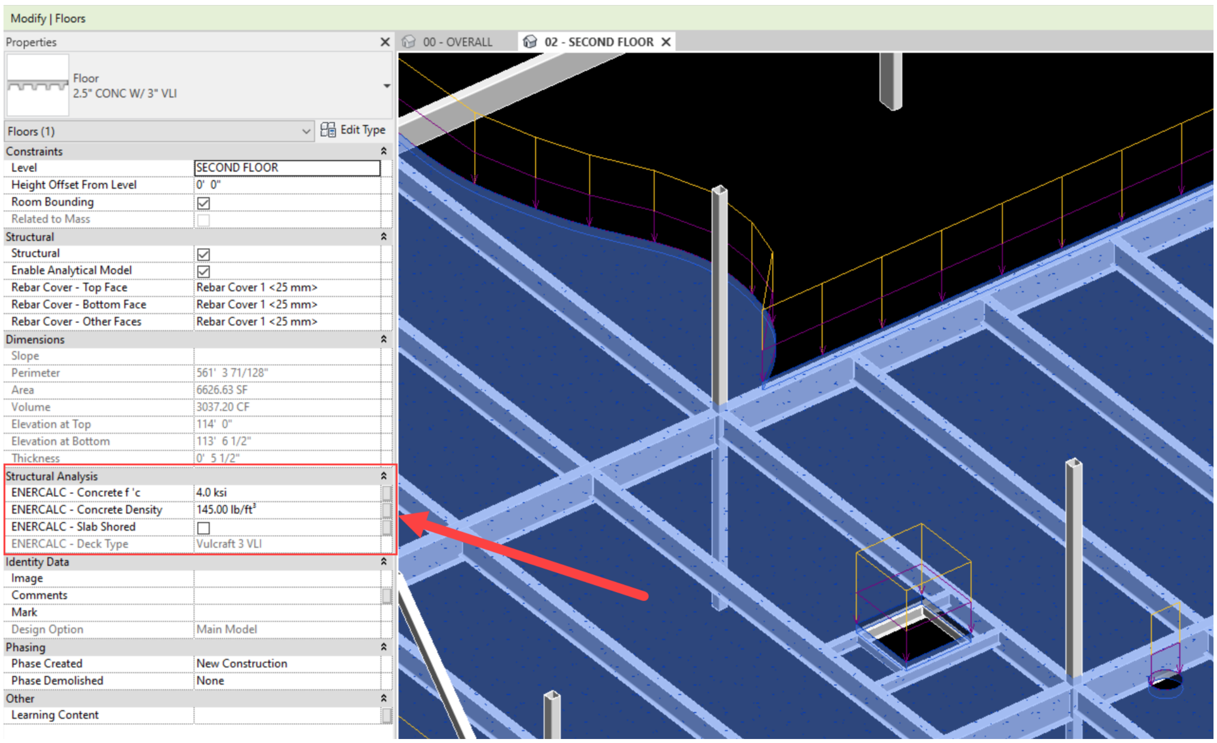
This setting may be viewed and changed from the Element Manager window. Once the "Remember" setting is toggled off and applied, the user will once again be prompted to approve the effective width during the next launch of the calculation.

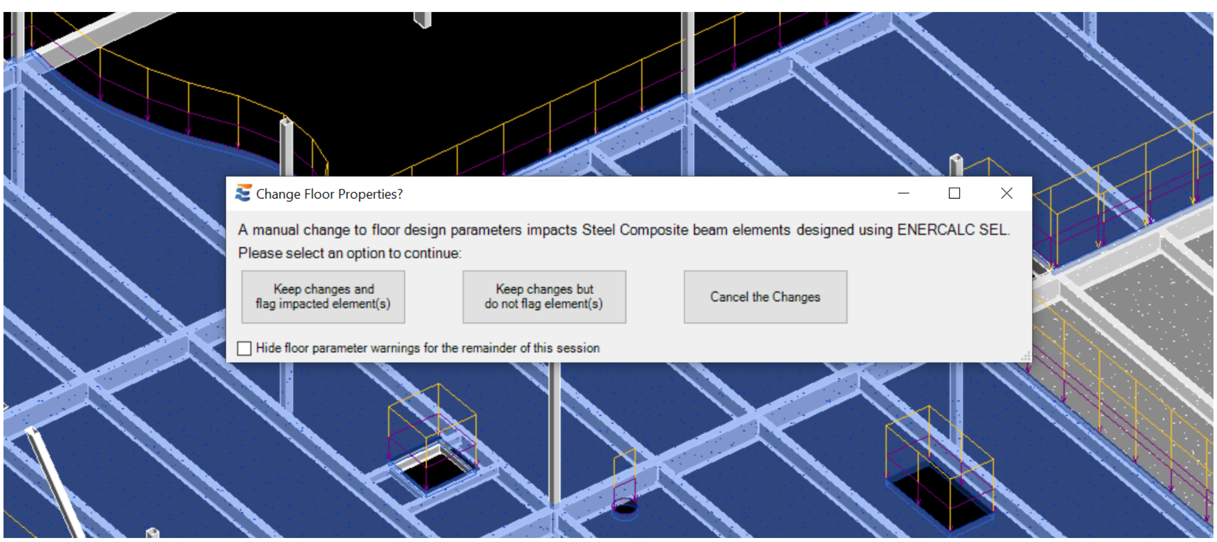


10.12.4 Composite Change Monitoring

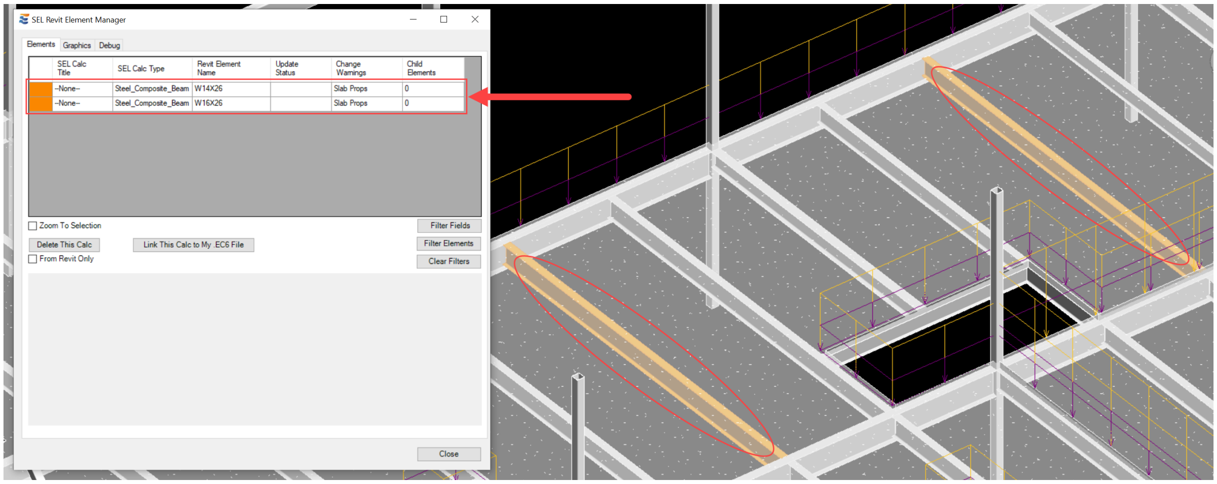
Steel Composite beam calculations are subject to all of the same change monitoring and warning considerations used for all other beam types (see Monitoring and Change Warnings 366).

In addition to the standard beam change warnings, Steel Composite beams are also subject to warnings if slab parameters in the Revit model are modified. If any existing Steel Composite beam calculations in the model were influenced by the affected slab, the user will be presented with a warning.





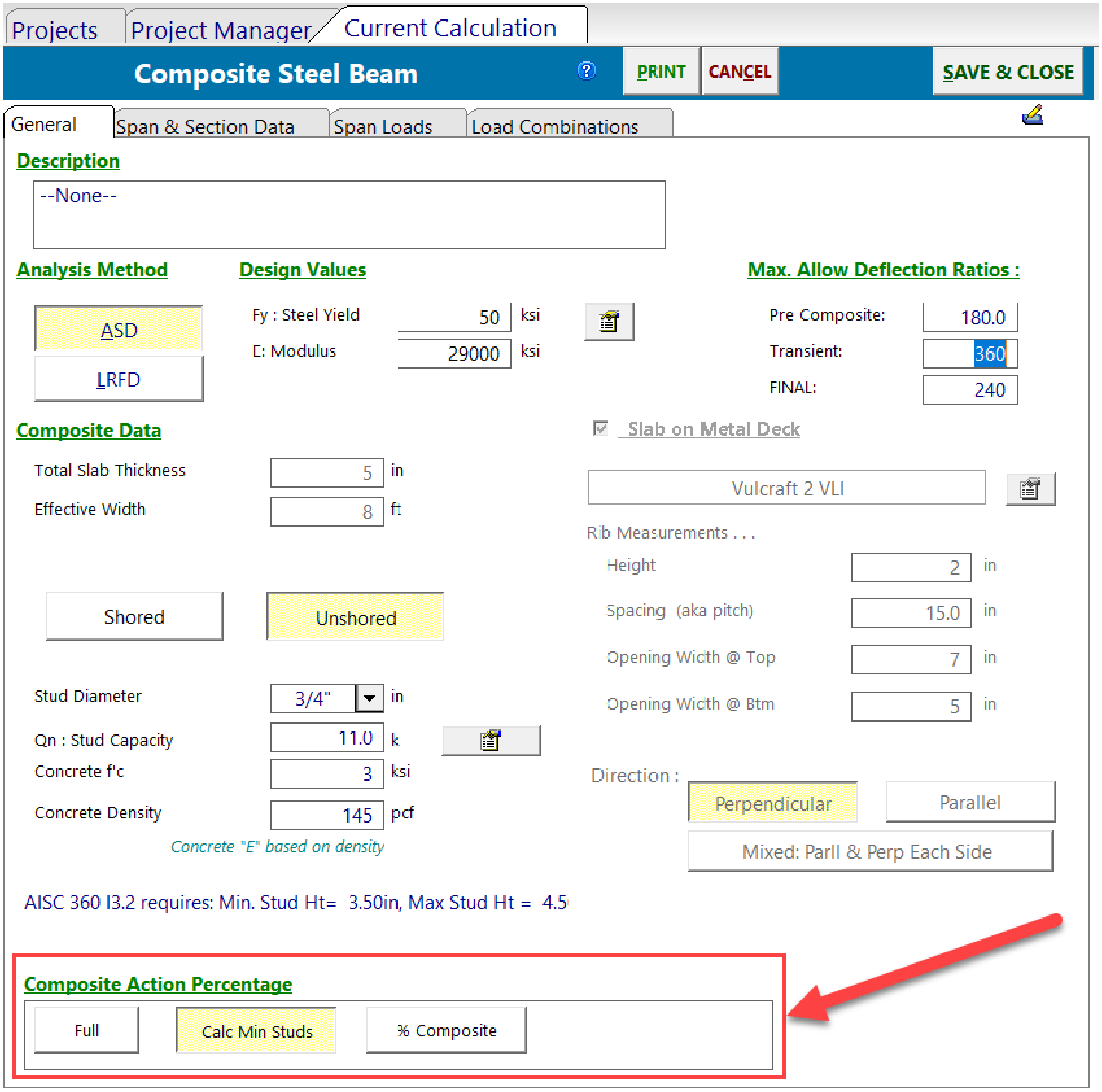
If the user chooses to keep the changes and flag the impacted elements, then each Steel Composite beam calculation influenced by the slab will be marked with a change warning.



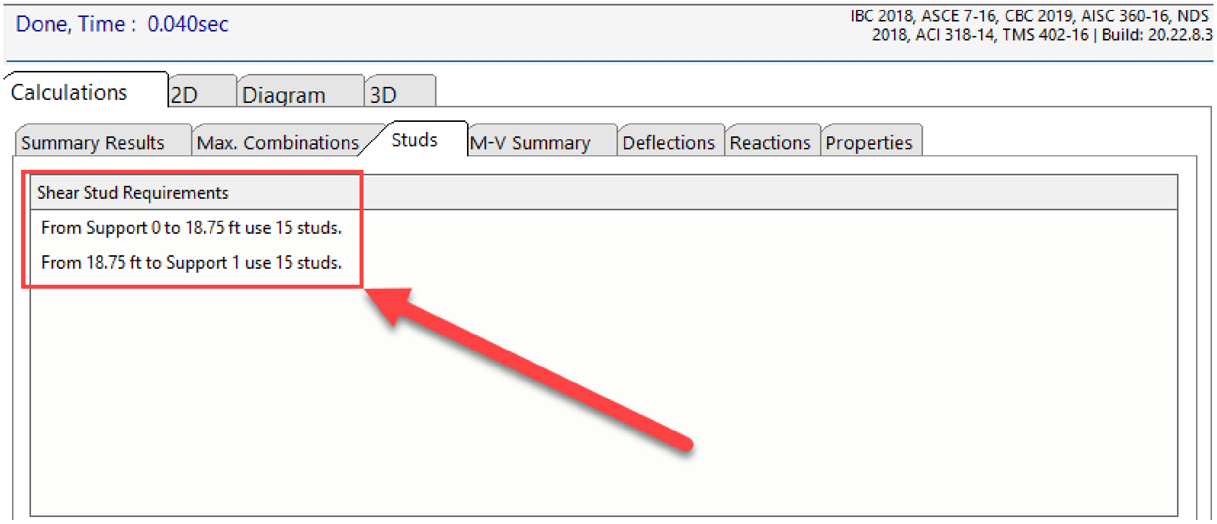
10.12.5 Composite Beam Tagging

ENERCALC for Revit provides convenient tagging capabilities to facilitate documentation during design of Steel Composite beams. Similarly to Steel Beam design (Beam Reaction Forces 379), envelope reactions will be populated on the Revit beam element when the Steel Composite "Save and Close" operation completes.

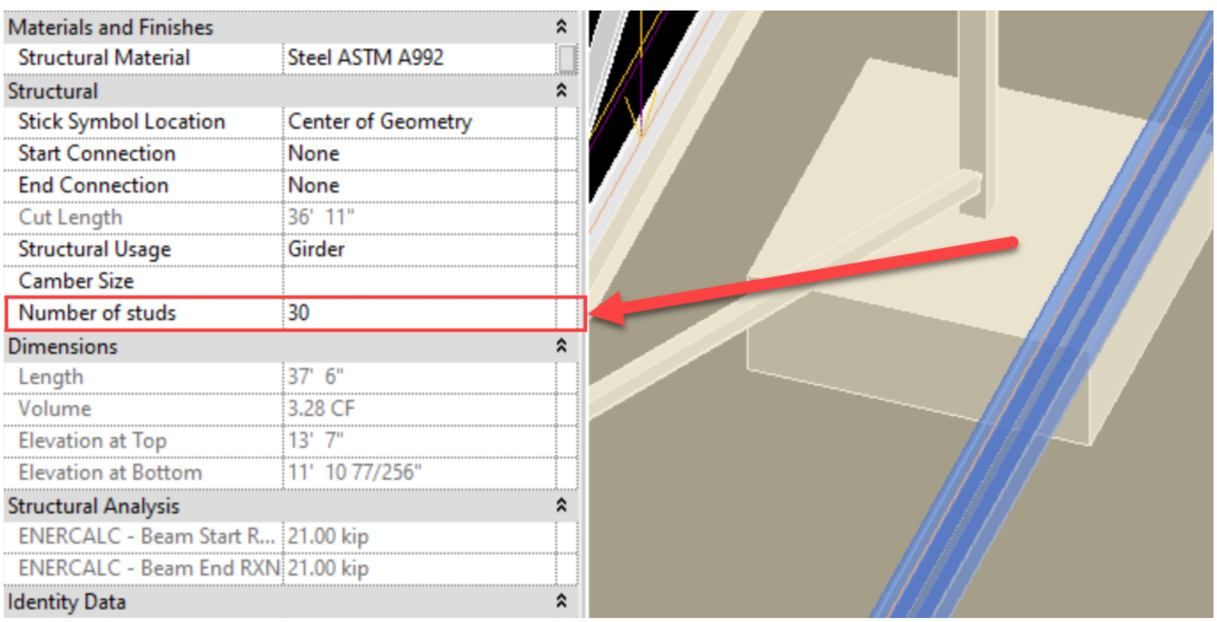
In addition to the reaction, the stud quantity specified in the ENERCALC calculation will also be stored on the Revit element. Settings for the control of the stud quantity design are found on the "General" tab:



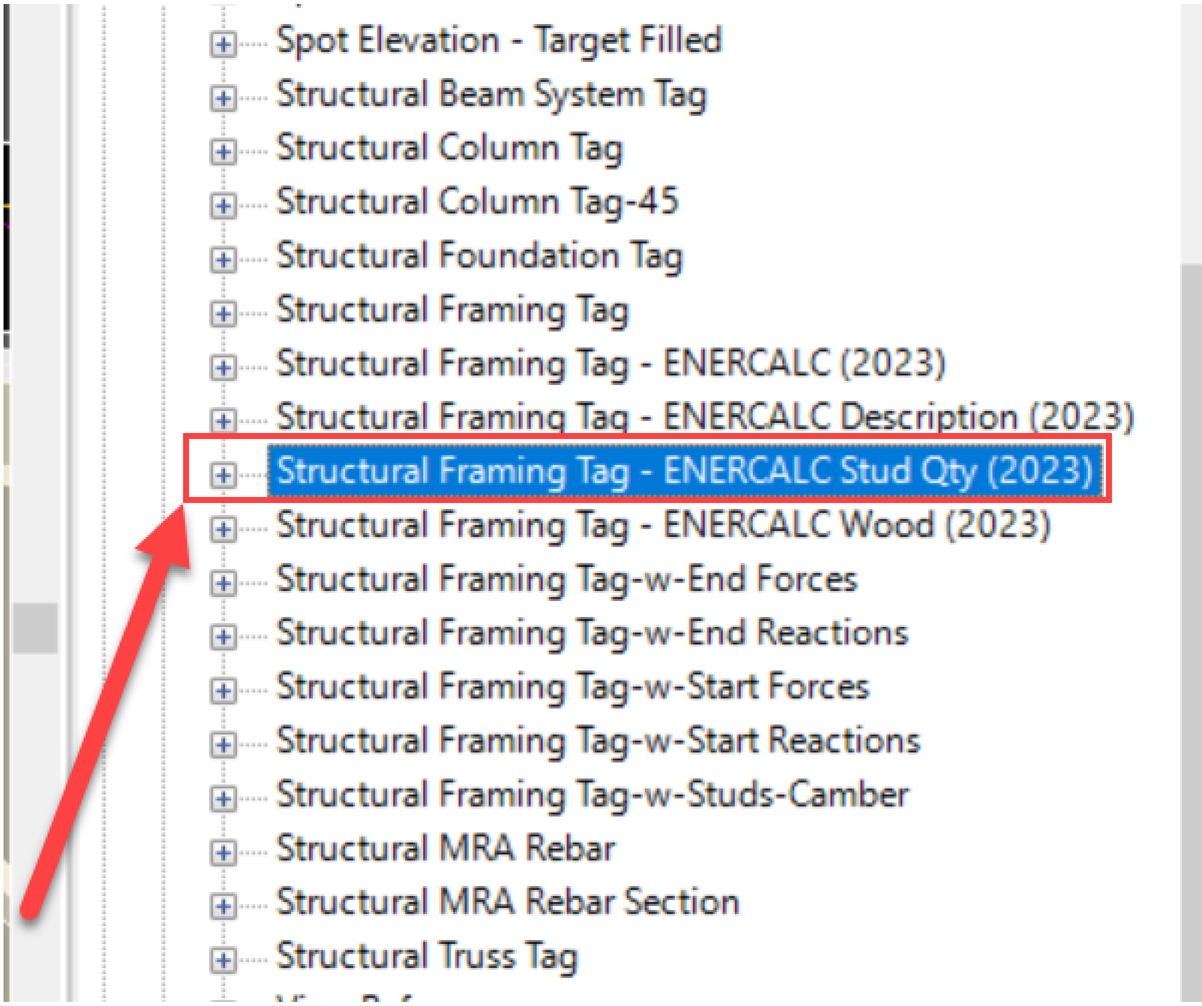
The results of this design setting are visible on the "Studs" tab of the "Calculations" display:

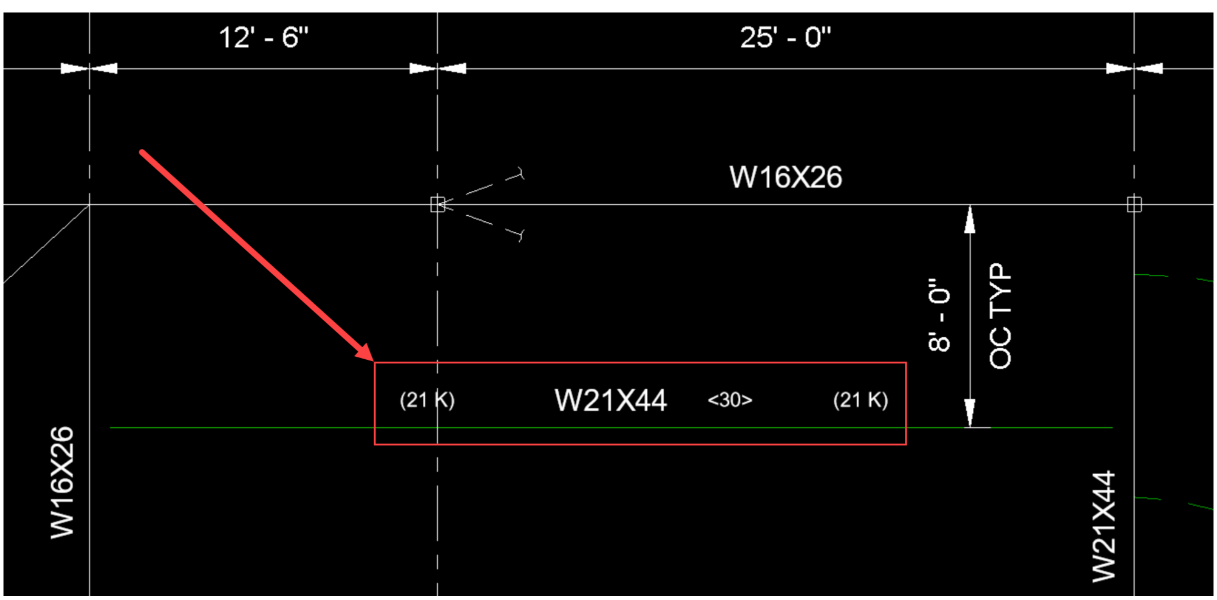


Upon "Save and Close", this information is automatically stored in the Revit-default parameter "Number of Studs":



If desired, this information is easily displayed on plan sheets or other views using the version-appropriate tag "Structural Framing Tag - ENERCALC Stud Qty (20XX)". These tags are loaded into the project automatically by ENERCALC for Revit:


Spot Elevation - Target Filled
Structural Beam System Tag
Structural Column Tag
Structural Column Tag-45
Structural Foundation Tag
Structural Framing Tag
Structural Framing Tag - ENERCALC (2023)
Structural Framing Tag - ENERCALC Description (2023)
Structural Framing Tag - ENERCALC Stud Qty (2023)
Structural Framing Tag - ENERCALC Wood (2023)
Structural Framing Tag-w-End Forces
Structural Framing Tag-w-End Reactions
Structural Framing Tag-w-Start Forces
Structural Framing Tag-w-Start Reactions
Structural Framing Tag-w-Studs-Camber
Structural MRA Rebar
Structural MRA Rebar Section
Structural Truss Tag


12' - 6"
25' - 0"
W16X26
8' - 0"
OC TYP
(21 K)
W21X44
<30>
(21 K)
W16X26
W21X44

10.13 Wood Beam Calculations

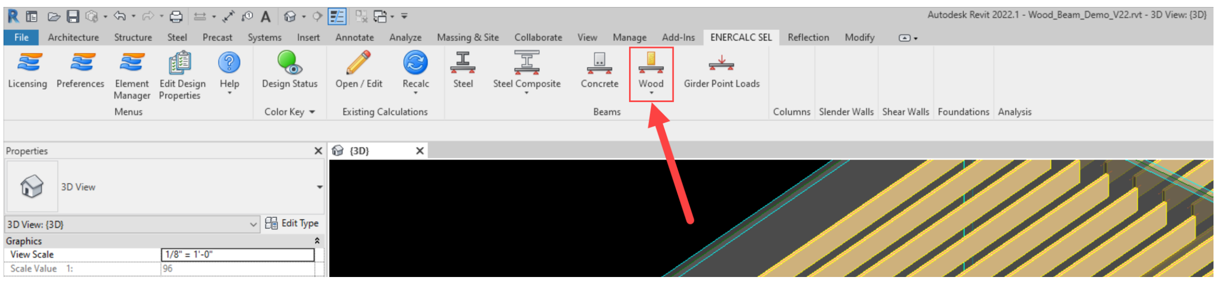
Wood beams share all of the common beam behavior described in the preceding sections.

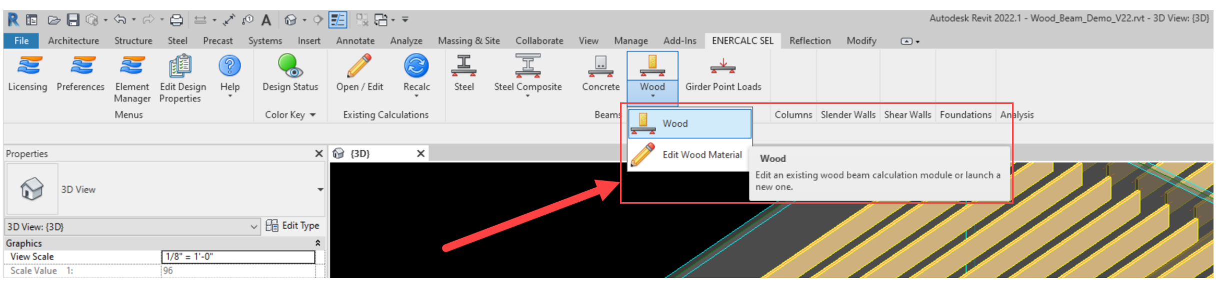
The primary characteristics that distinguish Wood beam calculations from conventional Steel beam and Steel Composite calculations are:

1.) Specifying the NDS wood material properties of the beam

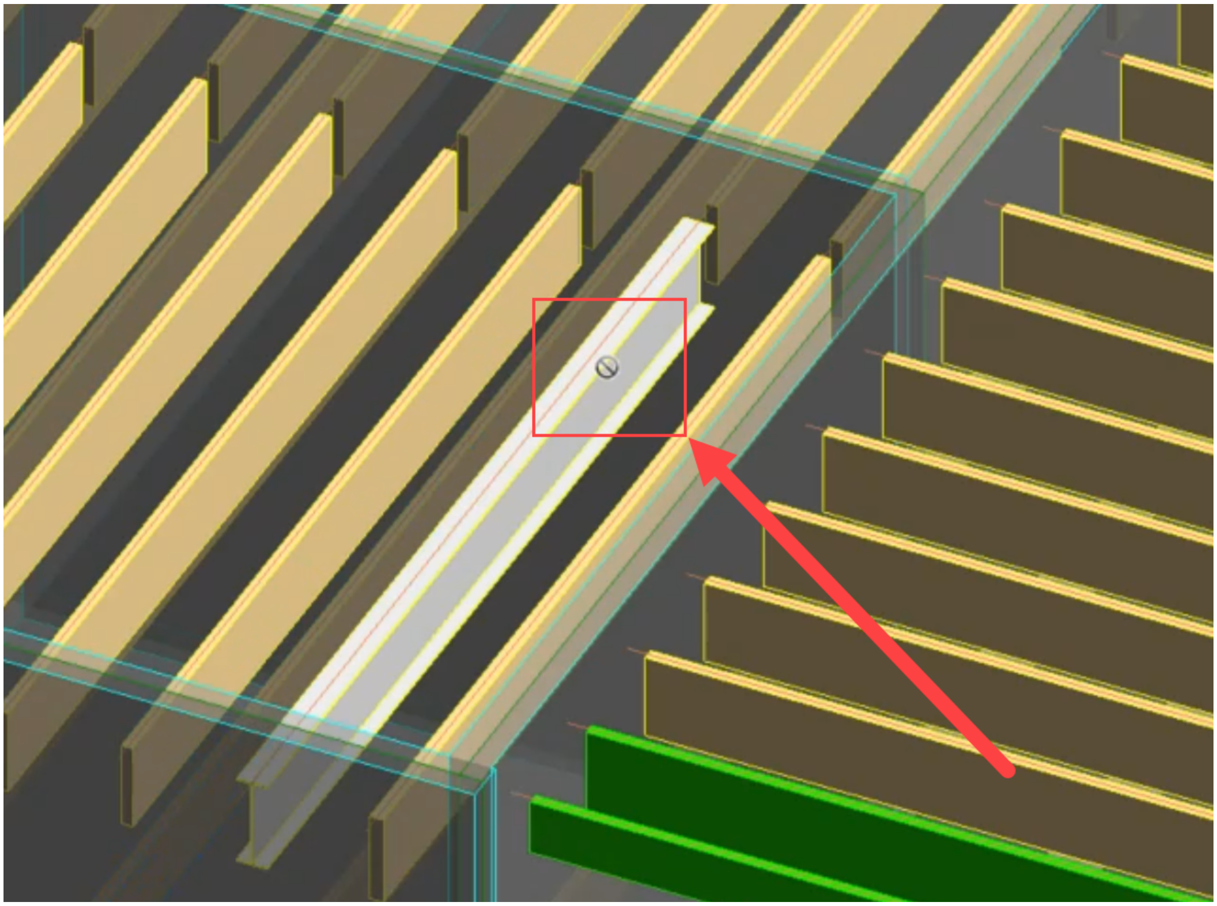
2.) Specifying the ply quantity of the beam (where applicable)

As with other calculations, Wood beam calculations are launched from the ENERCALC for Revit ribbon tab in the Revit interface.





As with other calculation types, clicking the calculation launch button will trigger a selection process and will await an element pick by the user.
During this process, only beams that are eligible for Wood Beam design will be available for selection. Ineligible elements are denoted with a "not allowed" circle and slit icon on the cursor when mousing over:



Design of wood beams using ENERCALC for Revit supports the use of the following NDS section types, corresponding to the primary default families found in Revit's out-of-box wood structural framing content:

- Dimension Lumber
- Timber
- Glulam - Southern Pine
- Glulam - Western

ENERCALC for Revit does not support the design of LVLs or any other proprietary engineered sections at this time.

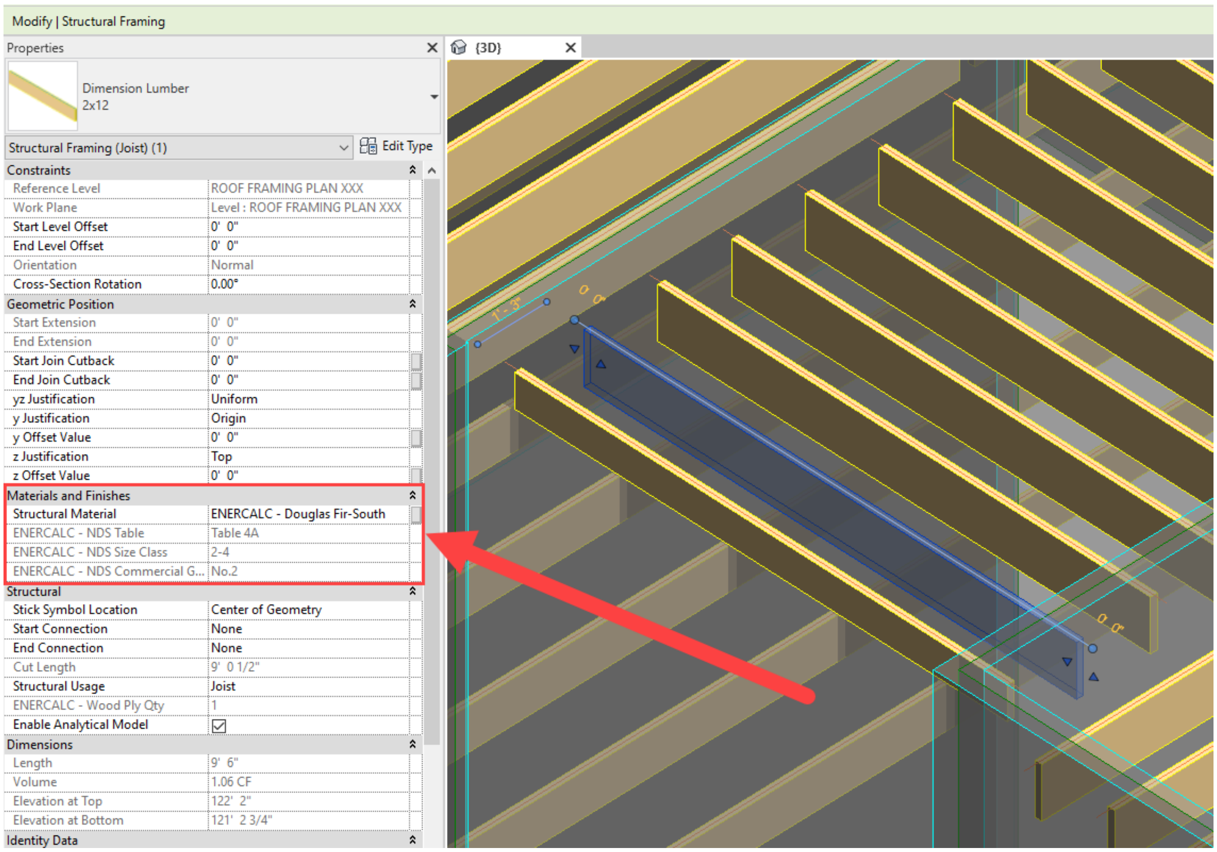
10.13.1 About Wood Beam Materials

NDS wood material properties must be assigned to a beam before a Wood Beam calculation can be launched. The core properties needed for design are as follows:

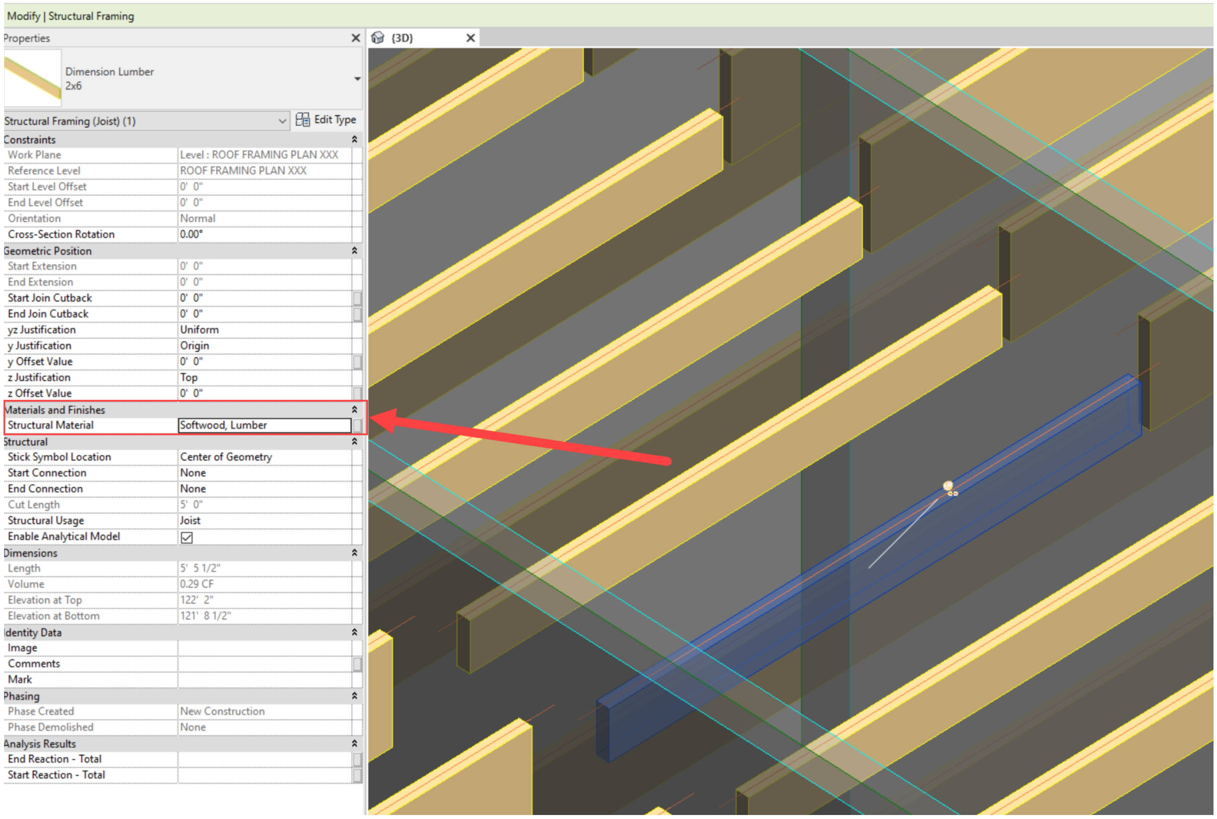
- NDS Table

- NDS Wood Species
- NDS Size Classification
- NDS Commercial Grade

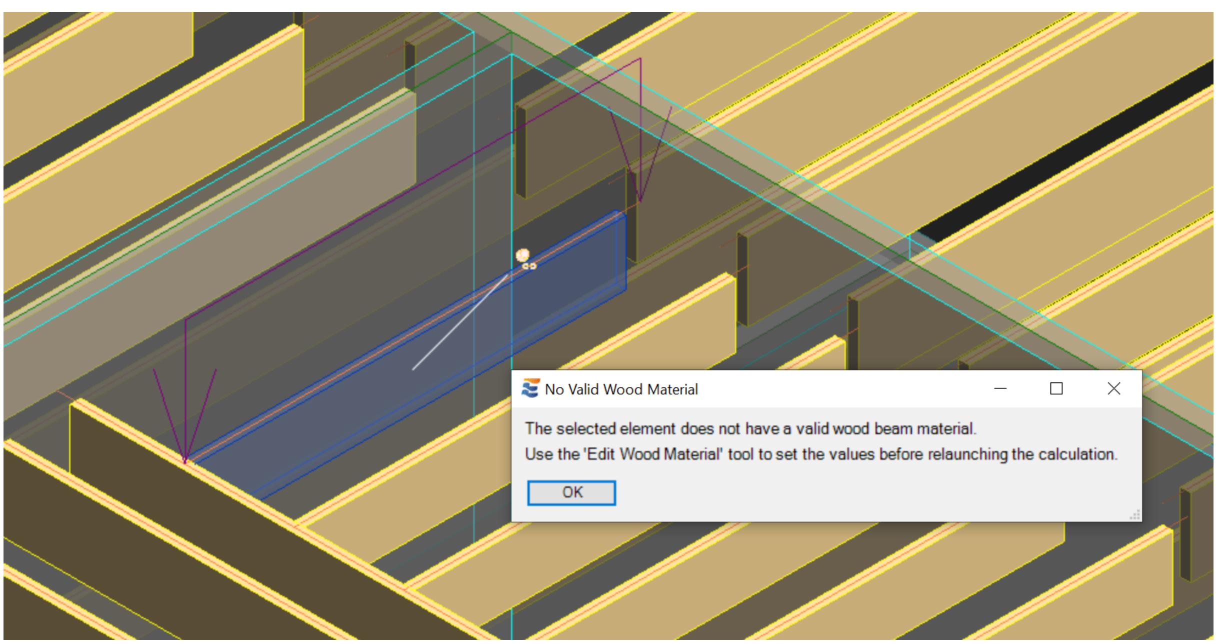
Once assigned to a particular beam, these material properties are visible in the Revit "Properties" pane when the beam is selected:



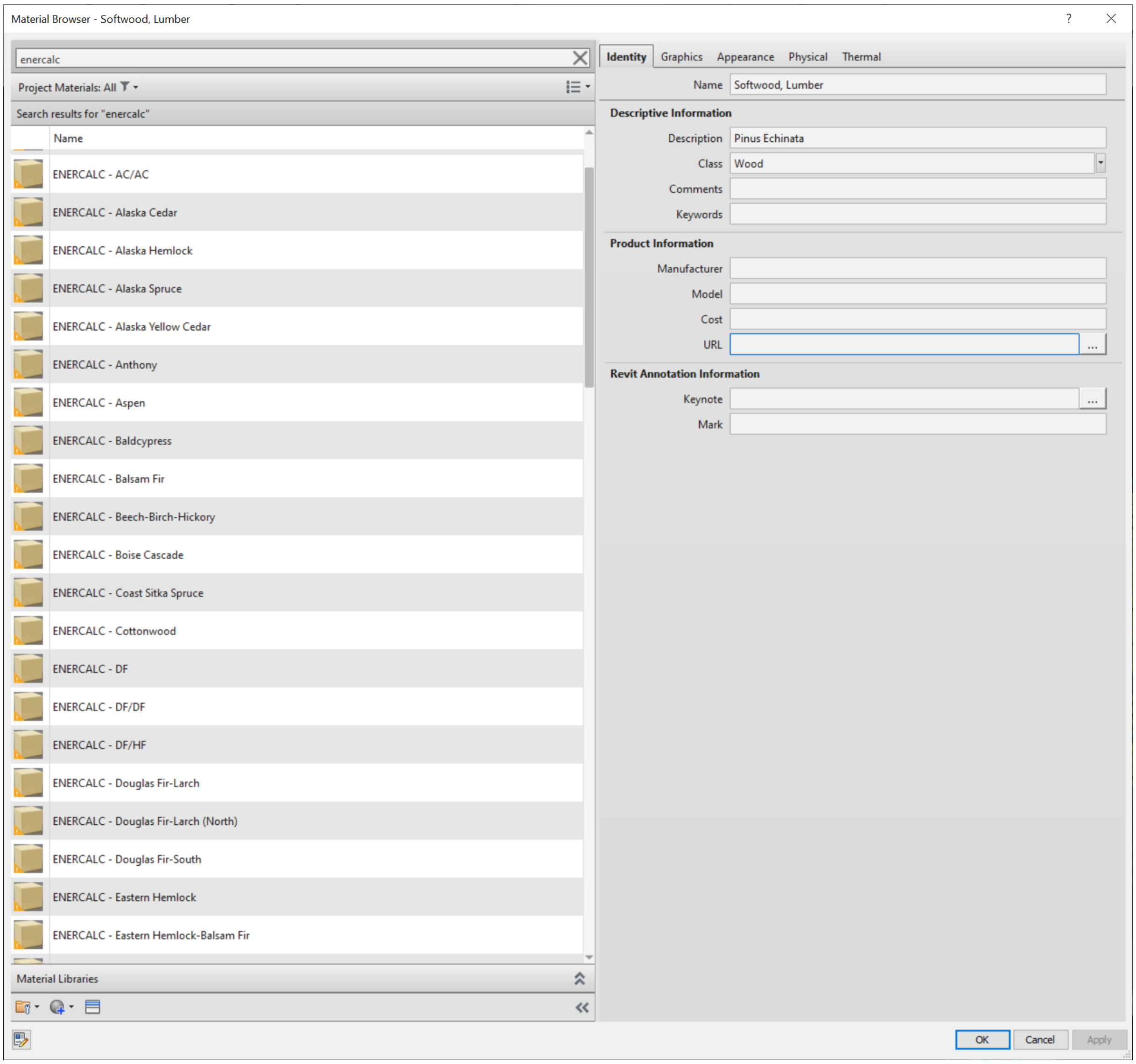
If design properties have not yet been assigned to a beam, then these parameters will not be visible in the "Properties" pane when the beam is selected:



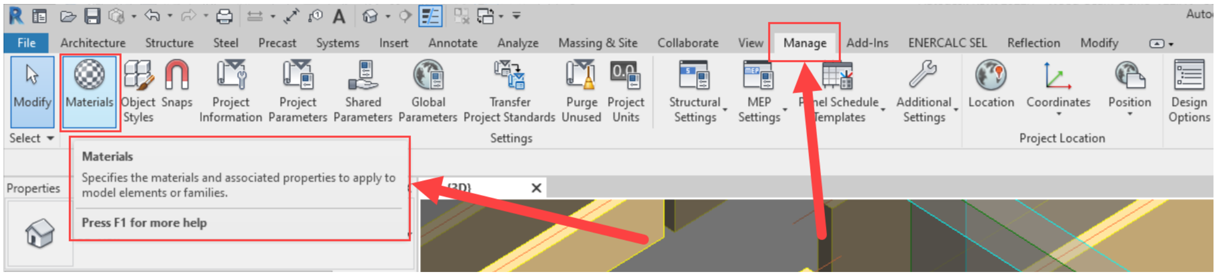
If a wood beam calculation is launched for an element that does not yet have material properties assigned, the launch will be canceled and the following warning will be presented:

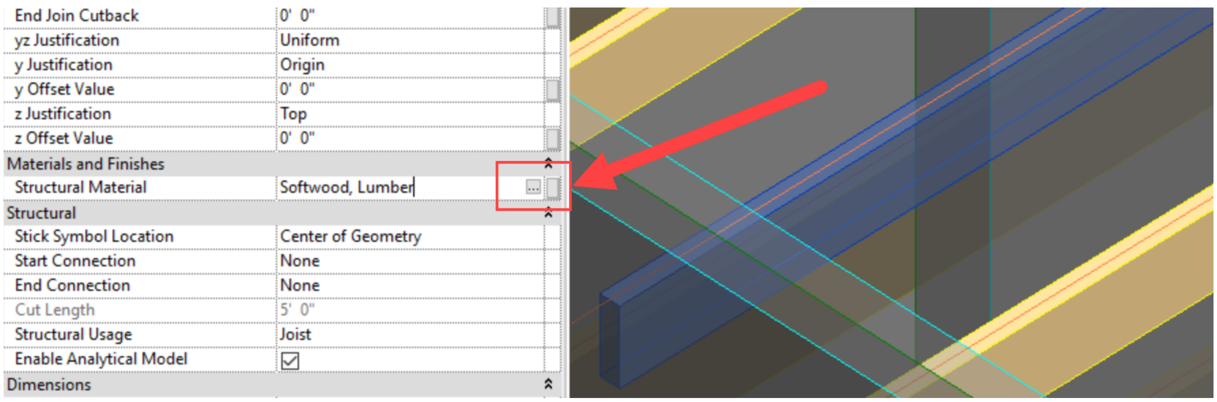


When a Revit model (.rvt file) is opened, ENERCALC for Revit automatically creates a set of wood material definitions based on NDS species names. These materials may be viewed in Revit's "Material Browser" window. All material definitions created by ENERCALC for Revit are prefixed with "ENERCALC - " in the title:



The "Material Browser" may be accessed via "Manage" > "Materials", or directly from the "Properties" pane of a beam element:

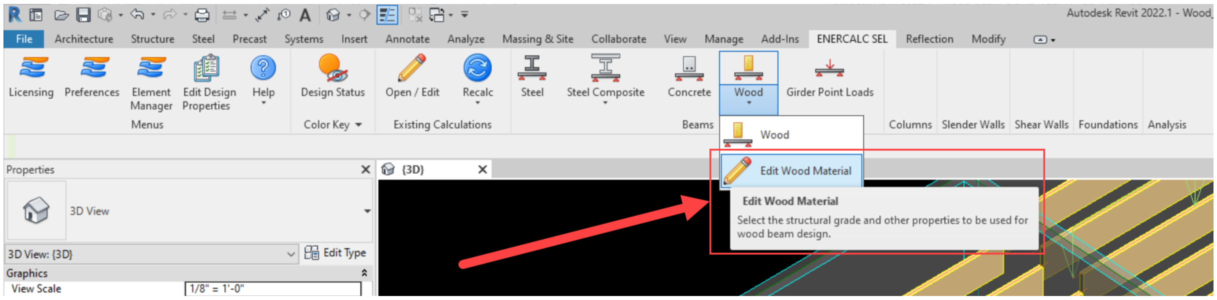




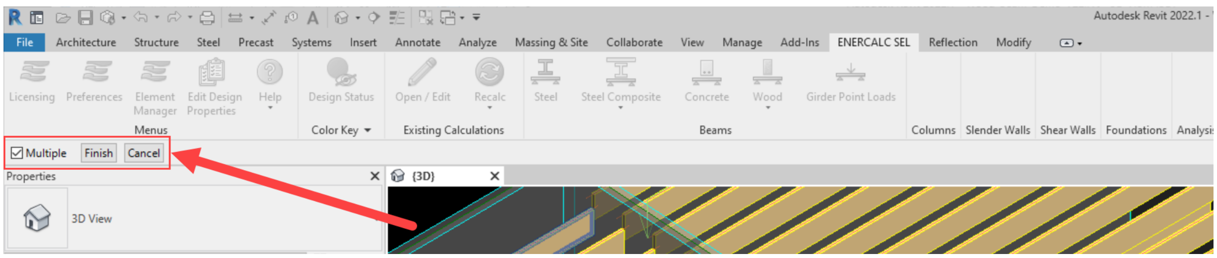
Although these material definitions may be viewed in the native Revit "Material Browser", they do not need to be manually assigned to Revit elements using this menu. Instead, they are to be assigned to Revit elements using the "Edit Wood Materials" menu (see Assigning Material Properties 418). It is not advisable to edit, rename, or remove these materials. Doing so may prevent ENERCALC for Revit from being able to assign design properties and launch calculations. In the event that any materials are removed from a project, the materials themselves will be re-generated automatically by ENERCALC for Revit the next time the .rvt file is opened. Any elements to which the removed and re-generated materials were previously assigned will need to be manually reassigned using the "Edit Wood Material" menu.

10.13.2 Assigning Material Properties

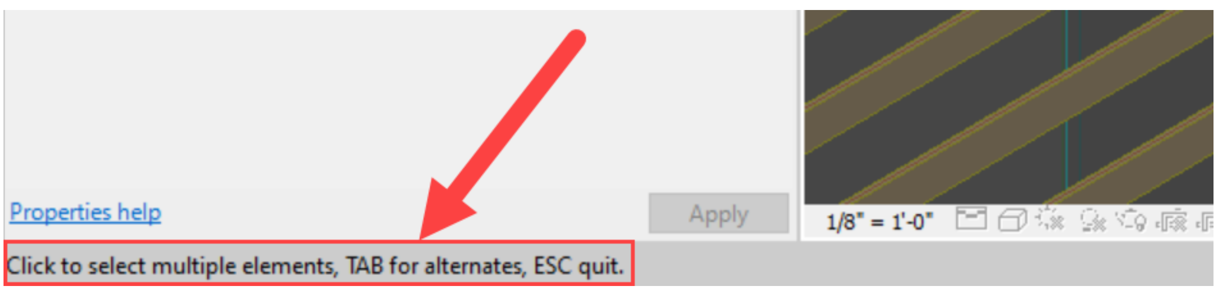
Wood Beam elements in the Revit model are prepared for calculation launch by using the "Edit Wood Material" button to assign design properties. This button is found on the ribbon bar, located in the drop-down of the "Wood" icon in the "Beams: panel:



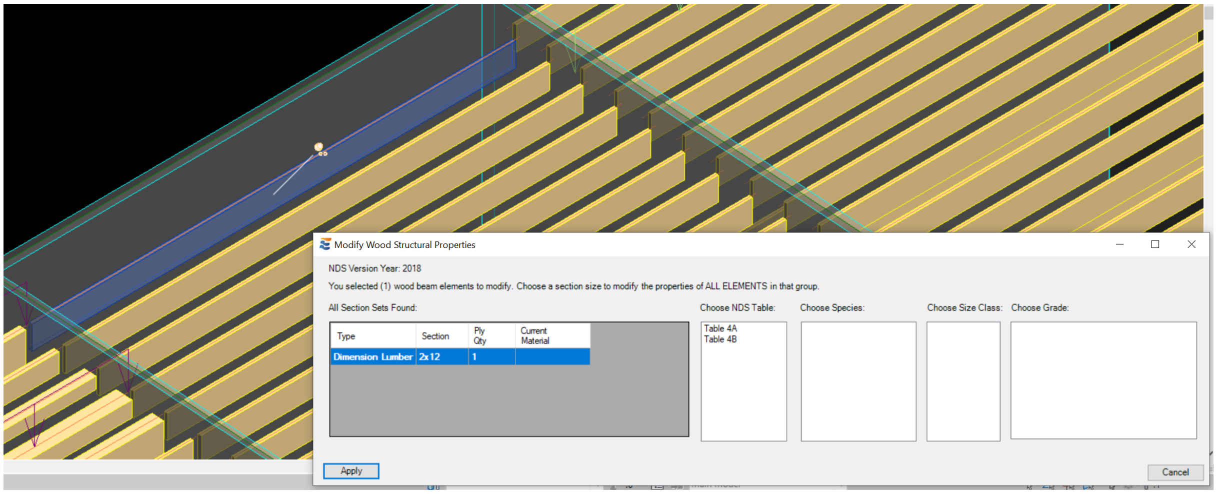
Clicking this button will launch a selection process, and EFR will await selection of one or more elements by the user:



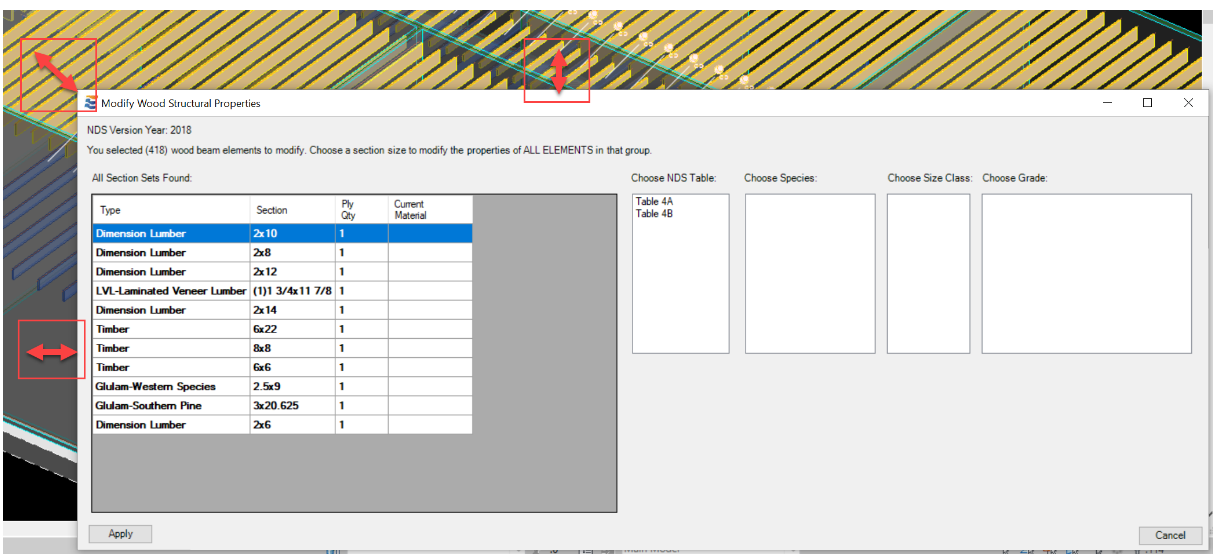
The status bar at the lower left corner of the Revit UI gives details:



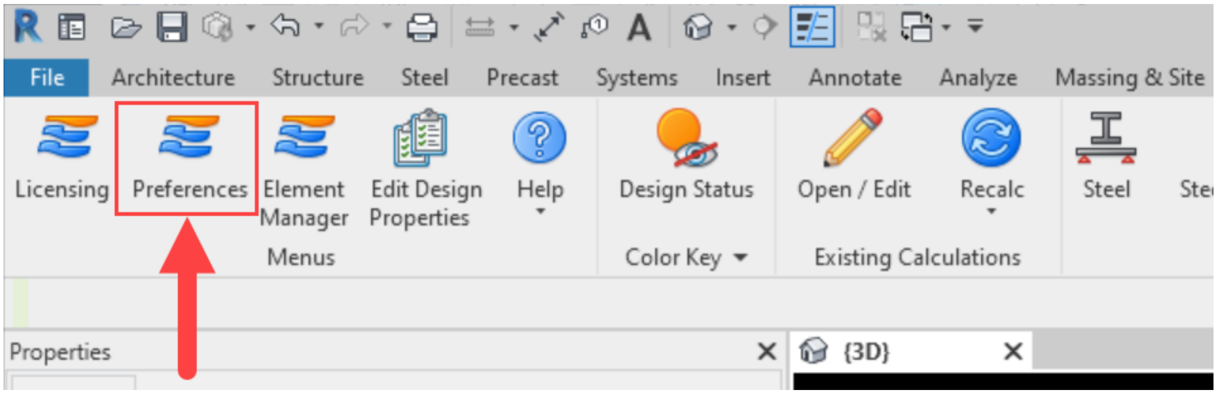
When the user finishes the element selection process, the properties form will load in the lower right hand corner of the Revit UI. The menu will indicate the current NDS year being used to populate design properties, as well as the number of elements selected by the user. The content in the "All Section Sets Found" table will vary based on the selection made by the user. For more detail on this, refer to Assigning To Single Element 424 or Assigning To Multiple Elements 432. When this window is open, the user retains full ability to interact with the Revit interface, including zoom, pan, select, changing active views, etc. Once opened, the window may be closed at any time without applying changes. The window is not docked in the corner; it may be moved to any location either on or off of the main Revit window as desired by the user. Even if the edit menu is moved off the main Revit window (or even onto another display), it will minimize or move to background along with the main Revit window.



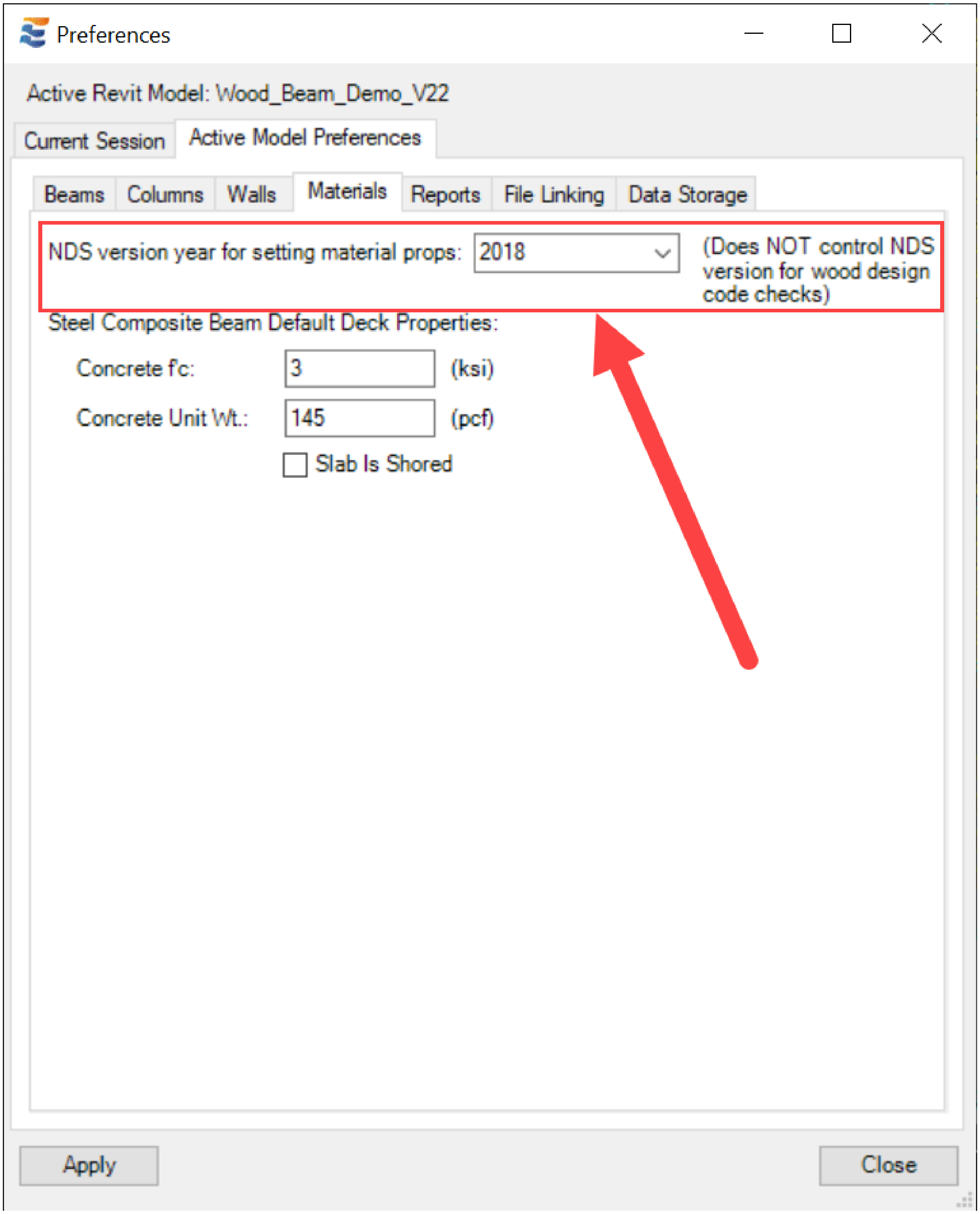
For more convenient viewing of large selection sets, the window may be manually re-sized by mousing over the edge or corner of the window and dragging to the desired size:

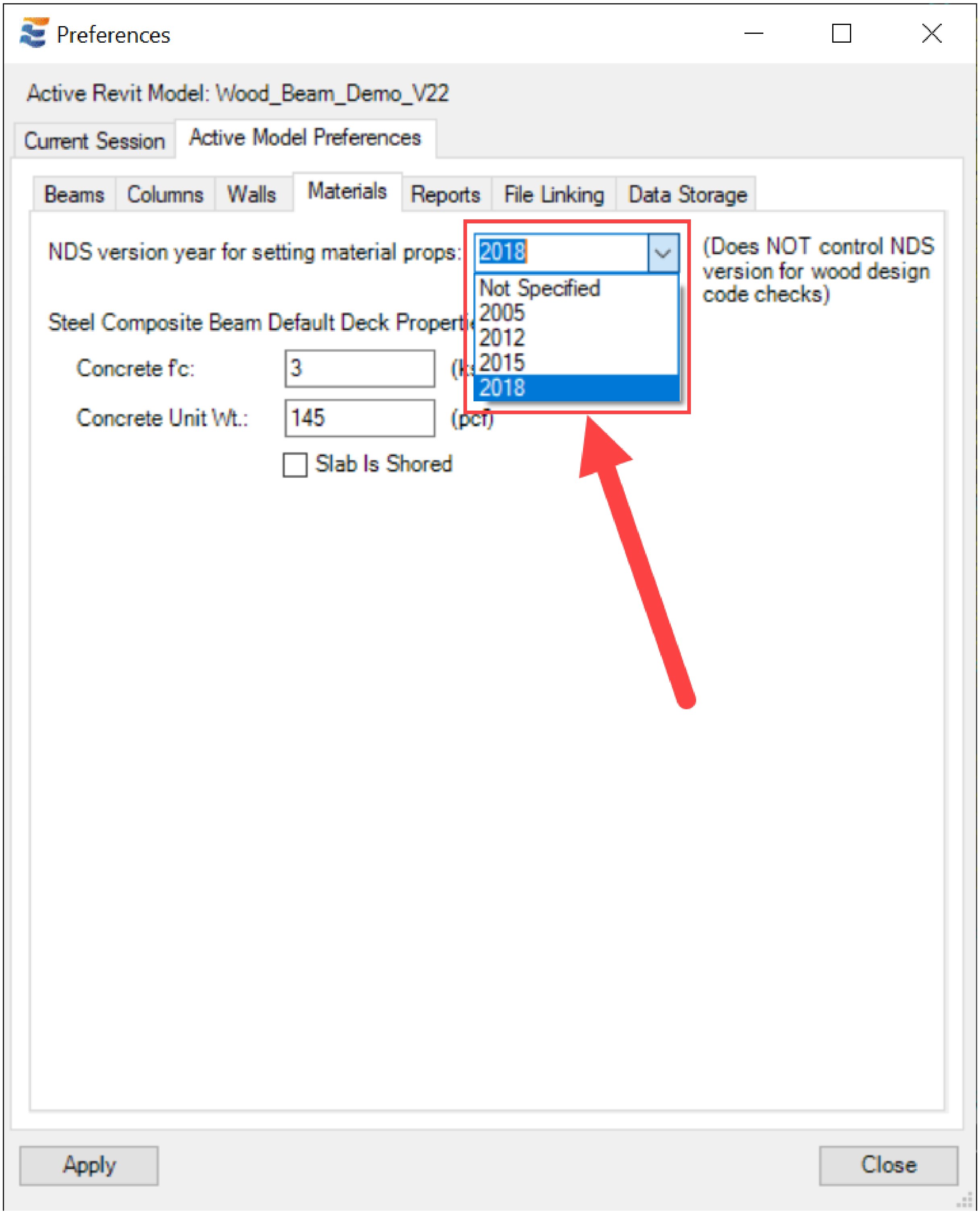


The version year of NDS being used to populate design property options in the "Modify" window may be viewed or changed from the EFR "Preferences" menu. This version year input does NOT control the NDS version being used to perform code checks in the ENERCALC calculation. It only alters which version-year's material properties are presented to the user in the "Edit Wood Materials" table.

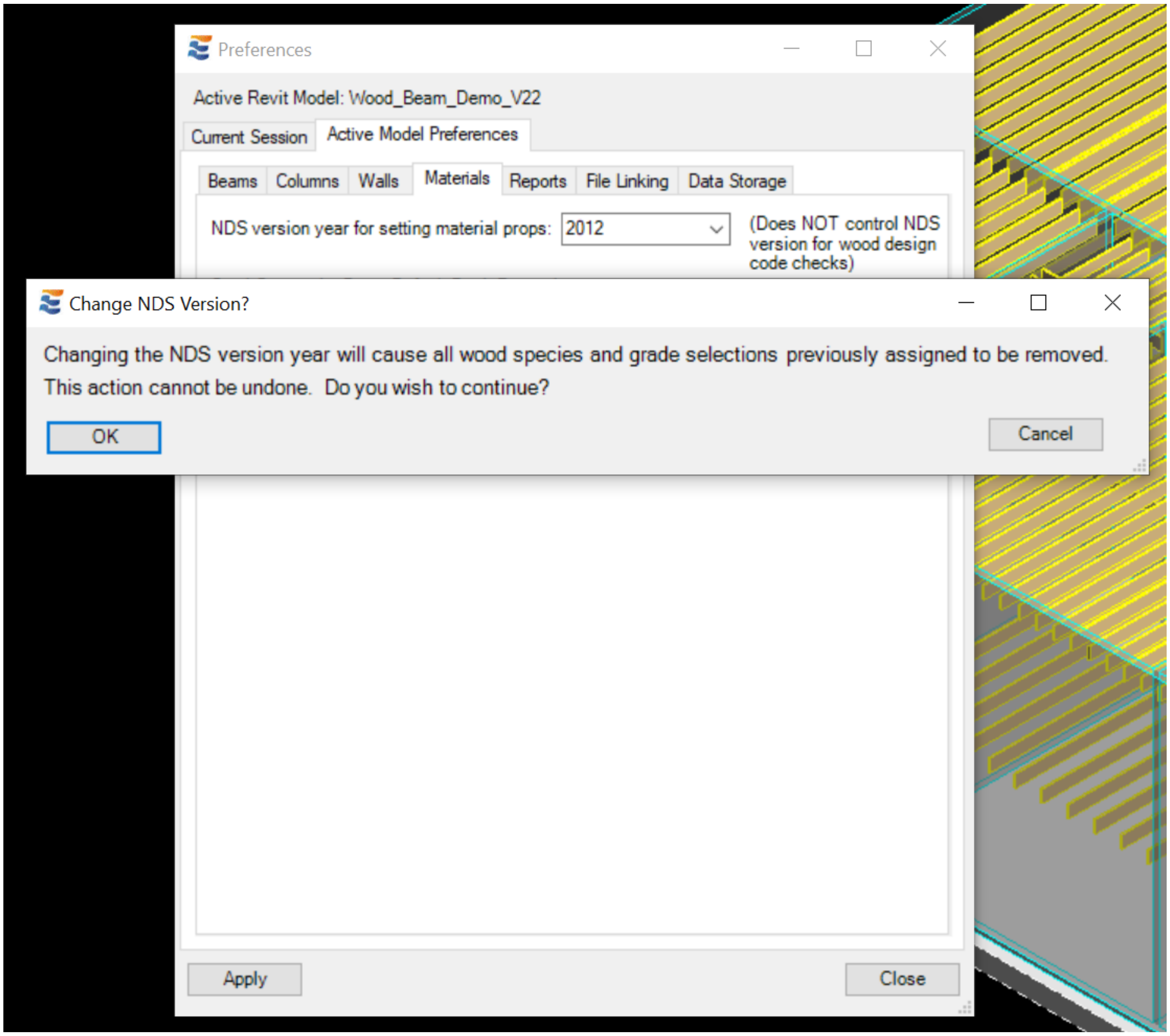


Navigate to "Preferences" > "Active Model Preferences" > "Materials":


Preferences
Active Revit Model: Wood_Beam_Demo_V22
Current Session
Active Model Preferences
Beams
Columns
Walls
Materials
Reports
File Linking
Data Storage
NDS version year for setting material props:
2018
(Does NOT control NDS version for wood design code checks)
Steel Composite Beam Default Deck Properties:
Concrete f'c:
3
(ksi)
Concrete Unit Wt.:
145
(pcf)
Slab Is Shored
Apply
Close



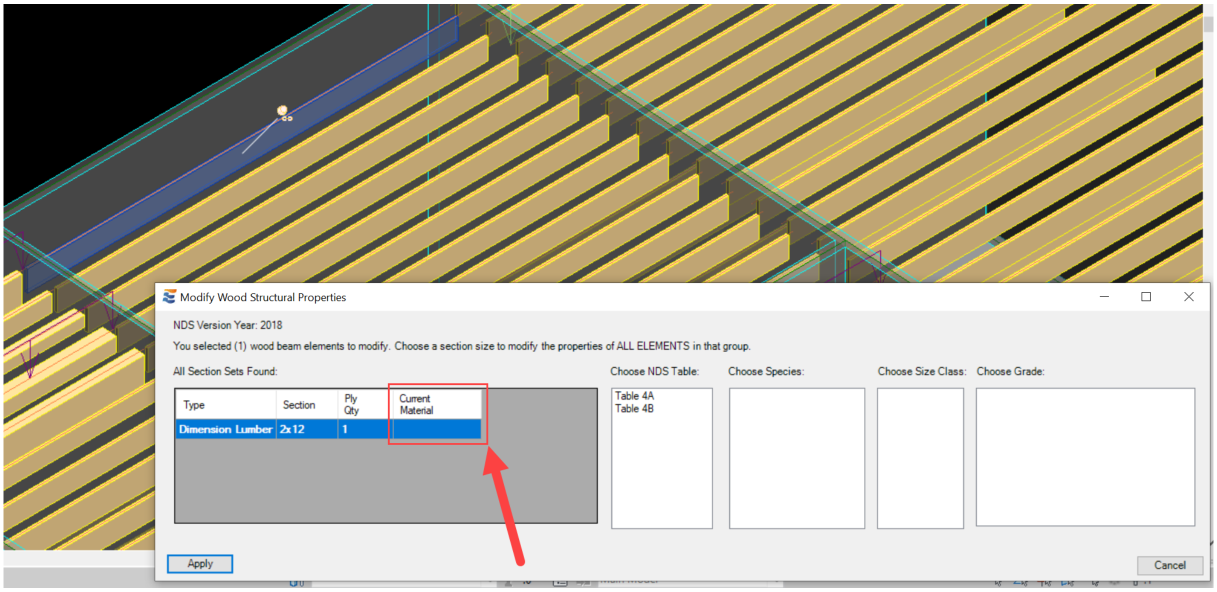
Users should be aware that any change to the specified NDS version year will impact calculations already created in the model using a different version. Changing the specified value in the "Preferences" menu will result in a warning cautioning the user that design data will be erased:



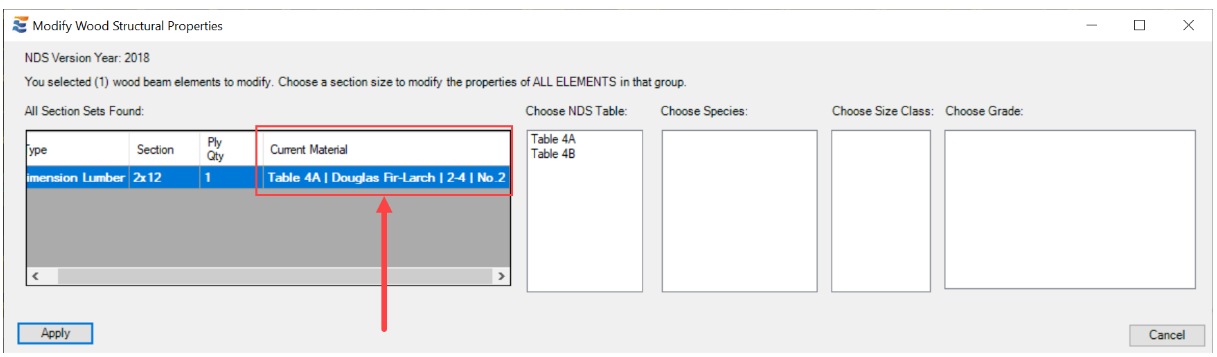
Clicking "OK" on this notification will finalize the version change and will erase the NDS design values currently assigned to existing calculations. Clicking "Cancel" will revert the drop-down selection to the pre-existing value.

10.13.2.1 Assigning To Single Element

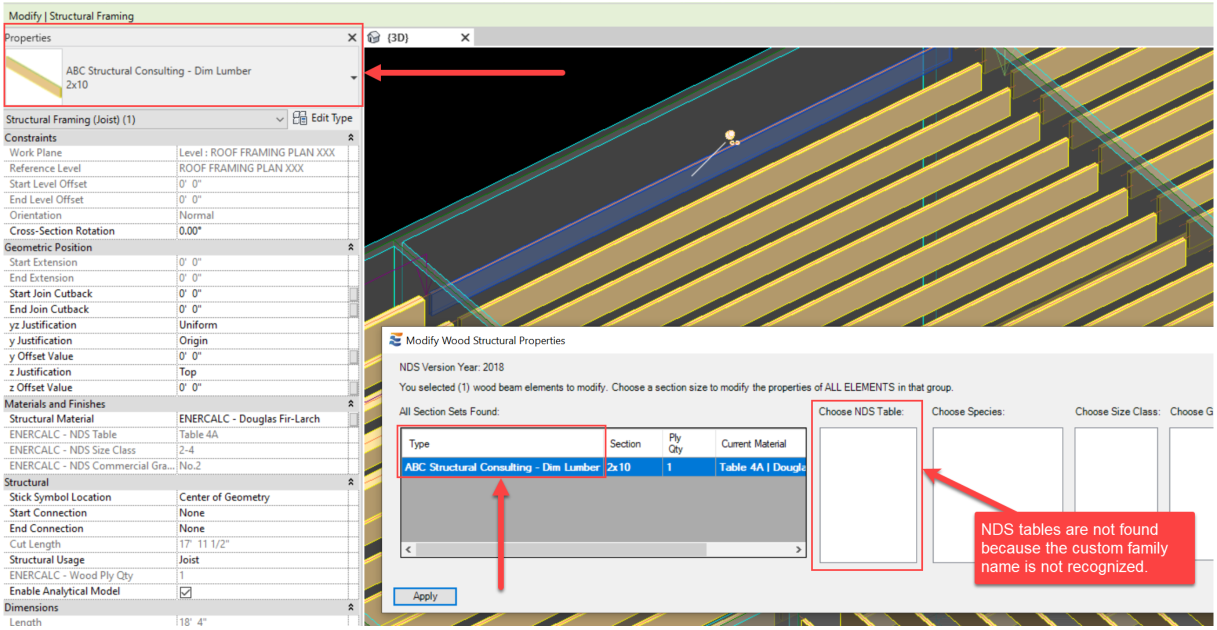
When the user chooses to select only a single element during the "Edit Wood Materials" launch, the menu will populate with one section set. If there are no material properties currently assigned to the element, then the "Current Material" display in the section set table will be blank:



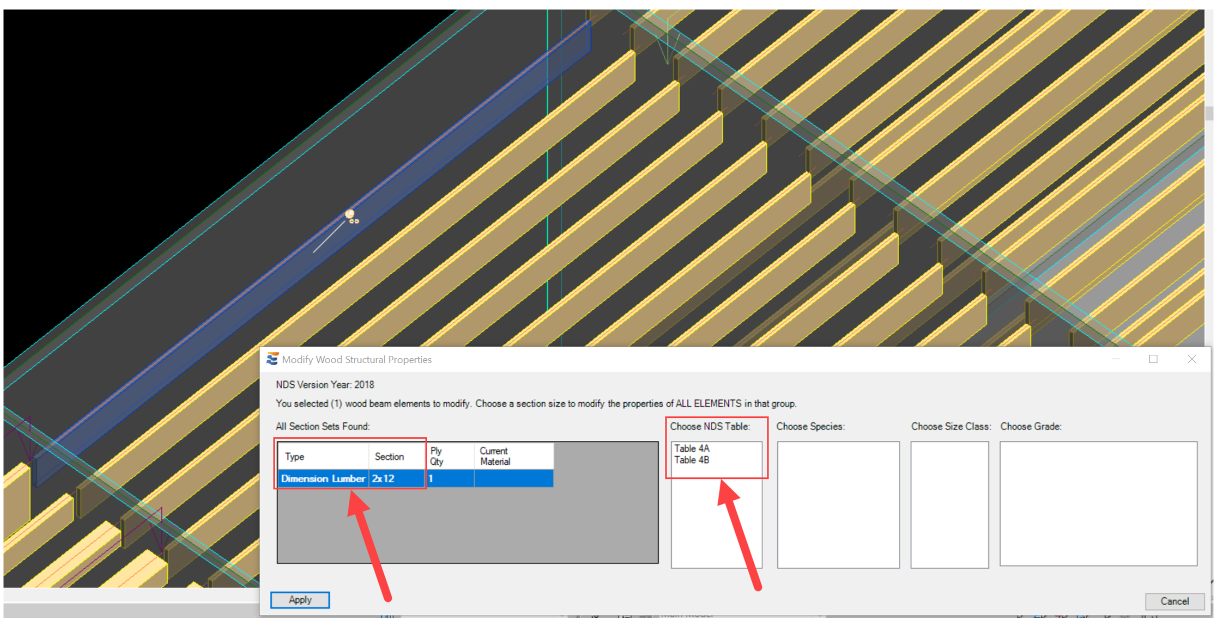
If the selected element already has wood properties assigned to it, then the existing properties will be displayed when the menu loads:

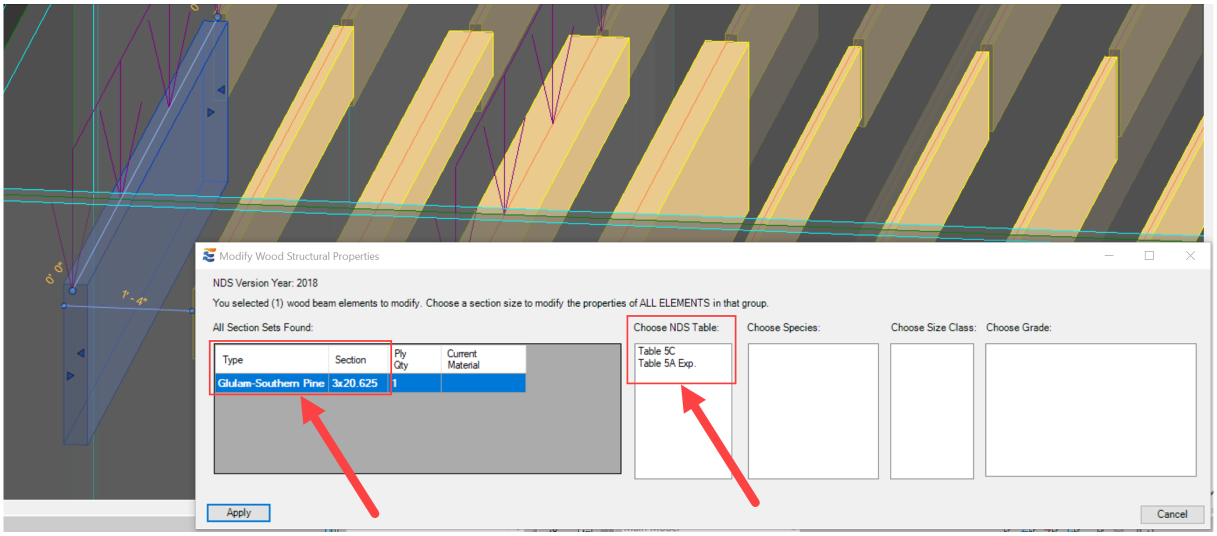


The table also displays the section type and section name. The section type and name displayed in the table are based on the family name and section name of the Revit element. It is advisable to use default Revit families for wood structural framing or ensure that family and section naming is identical to the default naming. Inability to match family and section names to NDS standard types and sections will prevent the material edit menu from functioning properly:

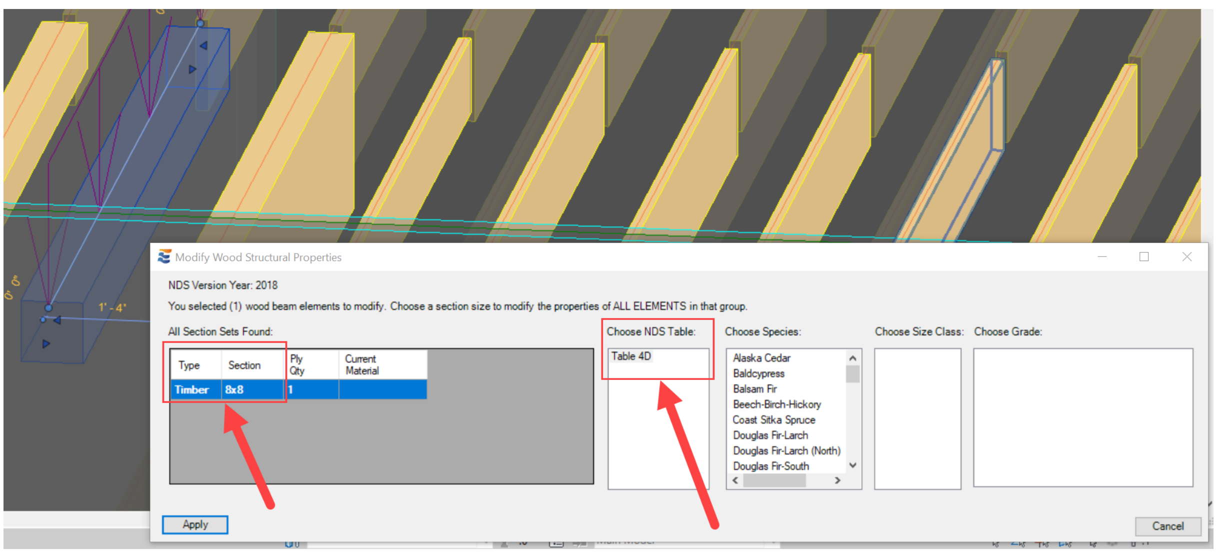


If the family name is recognized, the table automatically displays a list of eligible NDS tables that are compatible with the selected section.

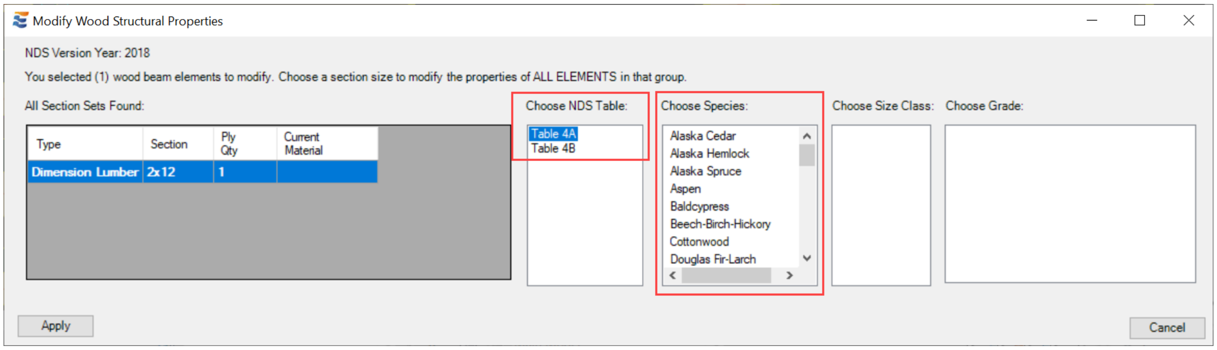




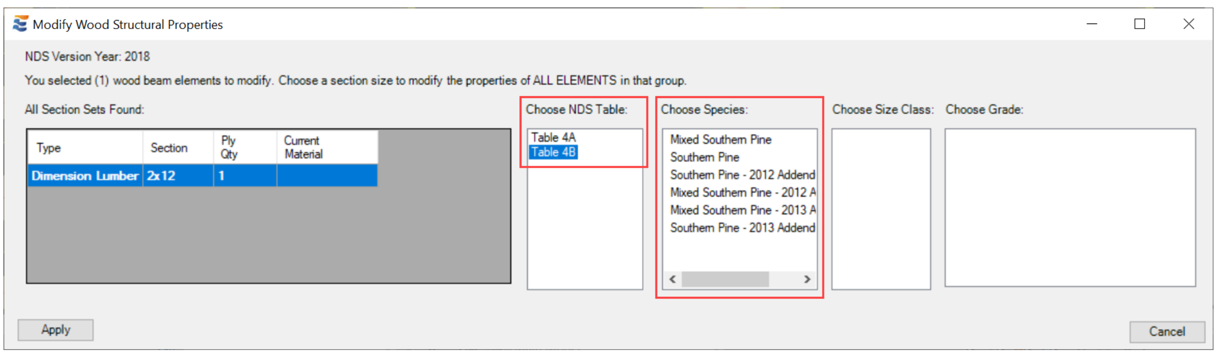
In cases where there is only one eligible NDS table, the only available option will be selected automatically, and the species options found in that NDS table will be displayed:



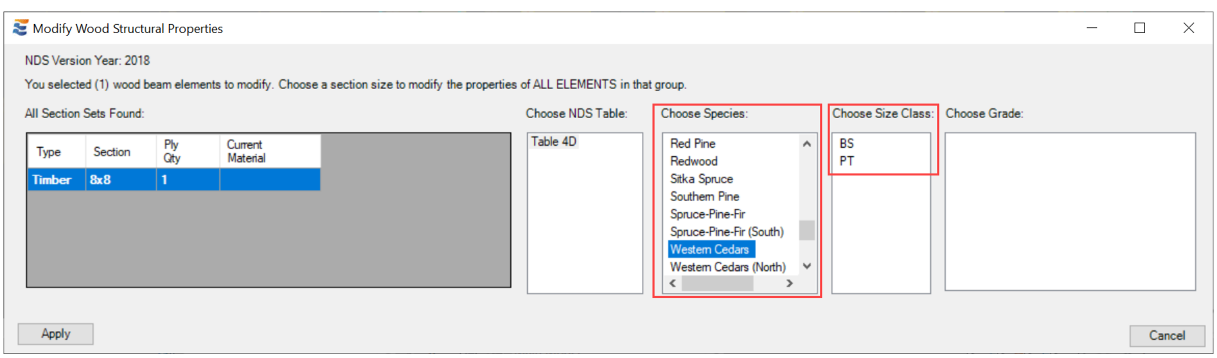
In cases where more than one table is available, the species table will populate AFTER an NDS table is selected:



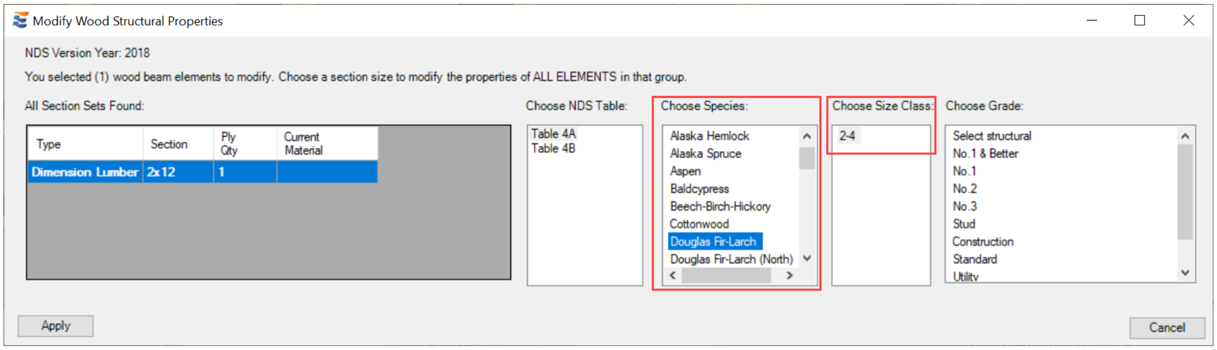
If a different NDS table is selected, the species options will update accordingly:



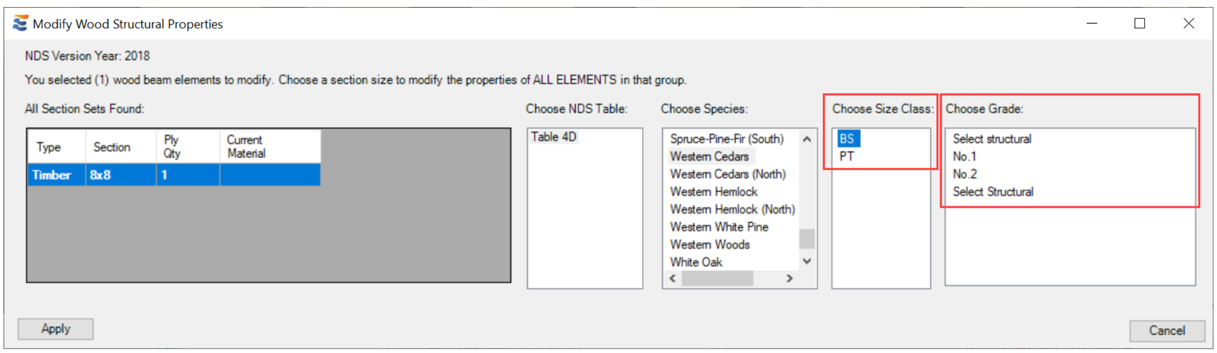
Once a wood species has been selected, the Size Class table will automatically populate with the associated NDS size classifications:



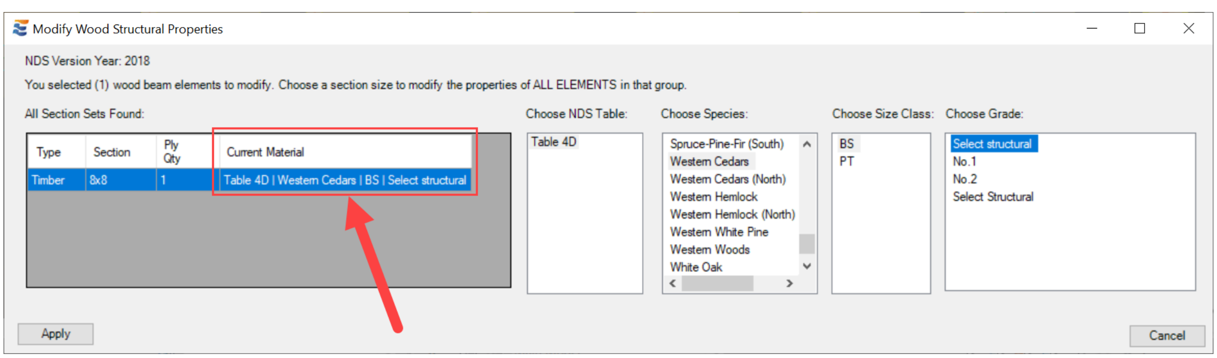
In cases where there is only one eligible NDS size classification, the only available option will be selected automatically, and the commercial grades found in that NDS table will be displayed:



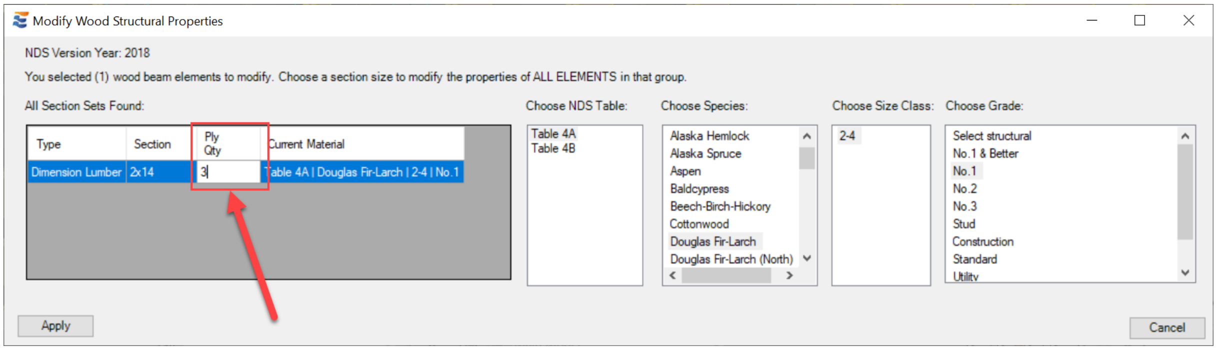
In cases where more than one size classification is available, the grade table will populate AFTER an NDS size class is selected:



Once a commercial grade option has been selected, the "Current Material" field in the section set table will show a summary of the complete material definition the user has selected:



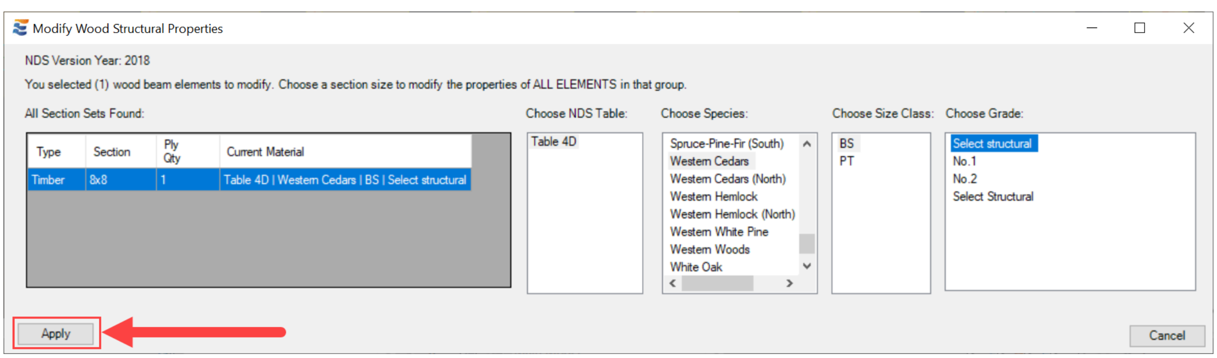
Where applicable, the user may also choose to modify the "Ply Qty" value to create a design where multiple sections are used together to achieve structural performance. In structural design practice, this is predominately done with dimensional lumber and occasionally with certain engineered sections (i.e., LVLs). The ply quantity may be directly modified using the corresponding cell in the table. For more detail on this topic, refer to Multiple Ply Beams 438.



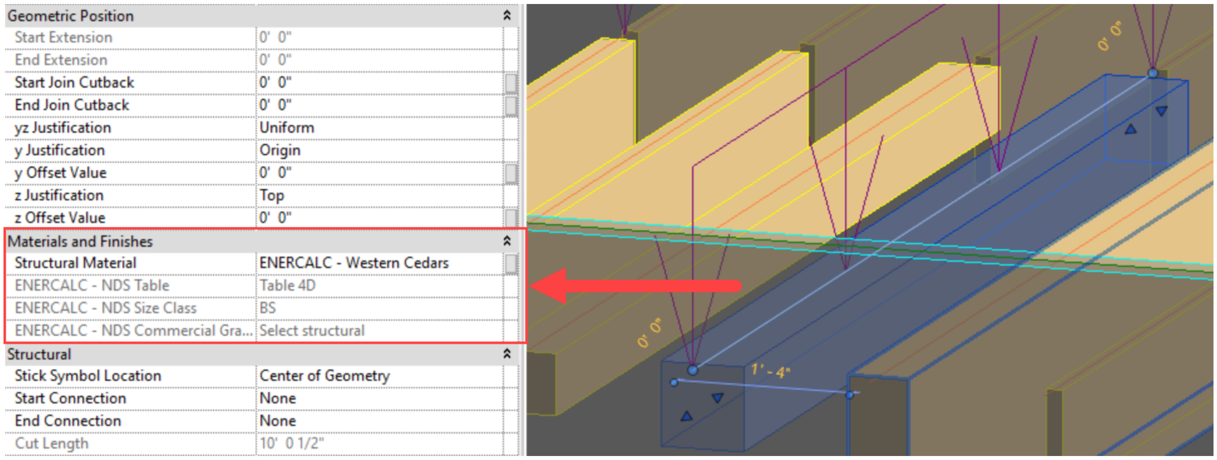
Changing the ply quantity does not have any influence over the physically modeled beam instance. It does not change the 2D or 3D graphics of the beam, nor does it alter the total number of actual beams modeled in the Revit project. It is purely a numerical parameter that allows ENERCALC for Revit to communicate the gross section geometry to ENERCALC when building the calculation. If desired, the parameter may also be referenced in schedules or quantity take-offs, since it is a Revit "Shared Parameter". For further reading on this topic, refer to "Wood Beam Tagging". (ed note: link removed temporarily). Users should also be aware that modifying the ply quantity on a section type for which ENERCALC does not support multi-ply configurations (i.e., glulam beams) will result in an error when attempting to launch the calculation.

Alternatively, a design team may choose to use custom beam sections which represent accurate geometry and graphical appearance of a multiple-ply beam. In such cases, the multiple-ply beam sections should be named using the exact convention as that used in ENERCALC in order to ensure interoperability.

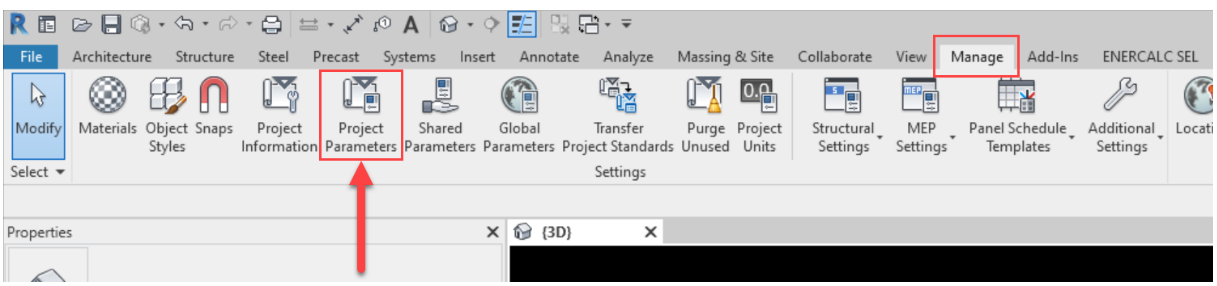
Once the definition is complete, clicking the "Apply" button will apply the chosen material properties to the Revit beam element. Clicking "Apply" when the definition is incomplete will have no effect on the element.

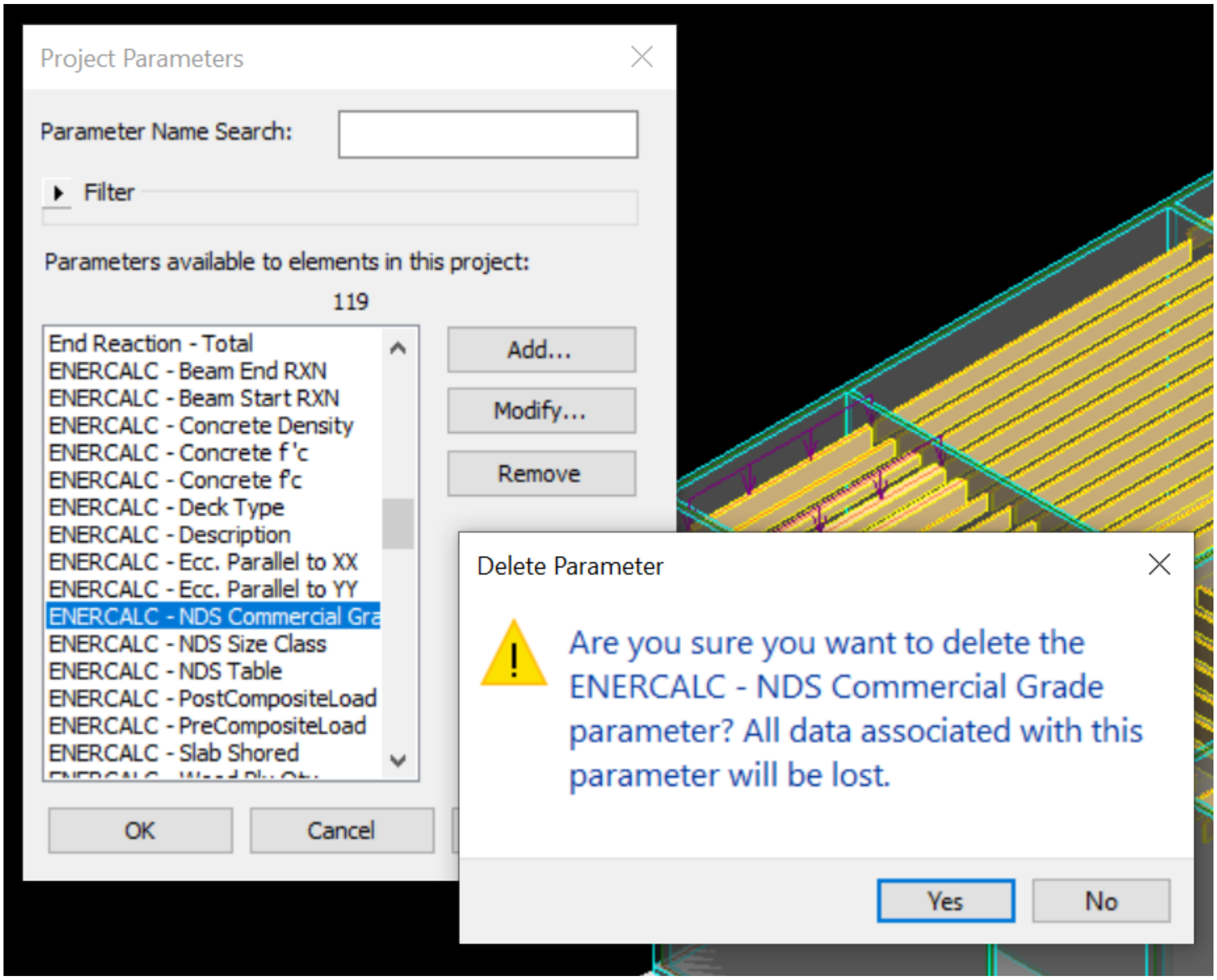


After the "Apply" button is clicked, the window will close and the material properties will be visible in the Revit "Properties" pane for the beam:



Users should also take note that since wood material property information is stored using Revit's native "Shared Parameter" framework, the parameters are exposed to view and possible removal by any team member. Removal of the underlying parameter that stores this information will result in loss of design data and will render ENERCALC for Revit unable to launch Wood Beam calculations in the project. Revit's native management tools for parameters include a warning on this topic:





In the event that these parameters are removed from a project, the parameters themselves will be re-generated automatically by ENERCALC for Revit the next time the .rvt file is opened. Any material properties formerly stored in the parameters will need to be manually repopulated using the "Edit Wood Material" menu.

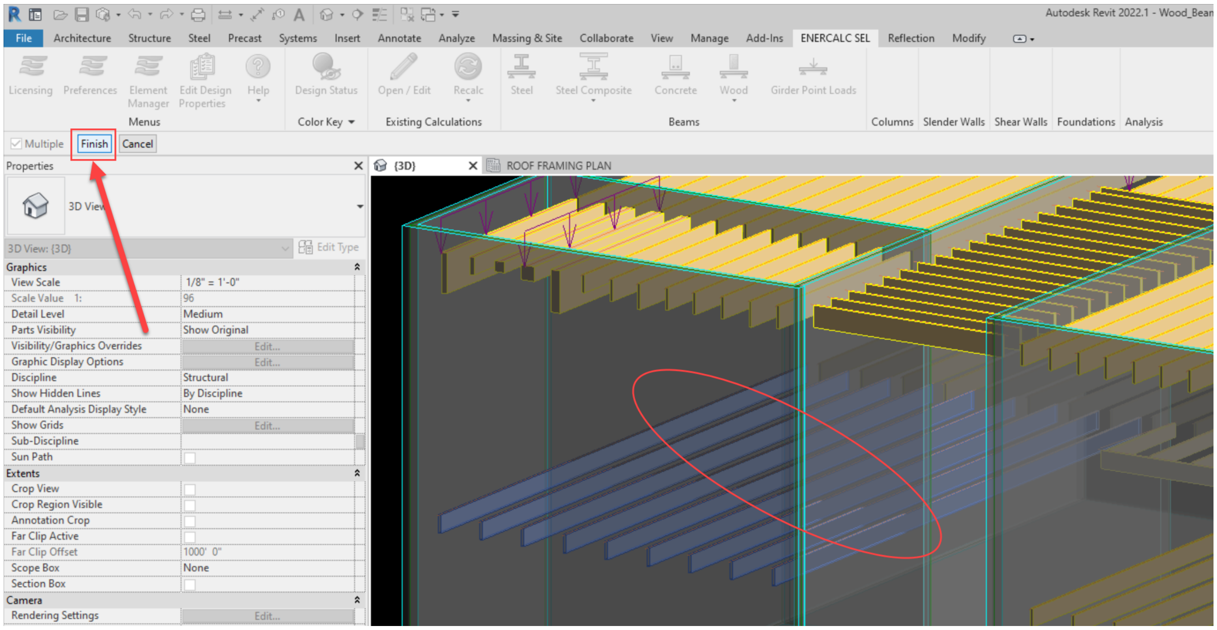
10.13.2.2 Assigning To Multiple Elements

There are two different approaches for selecting multiple elements to display in the "Edit Wood Material" menu:

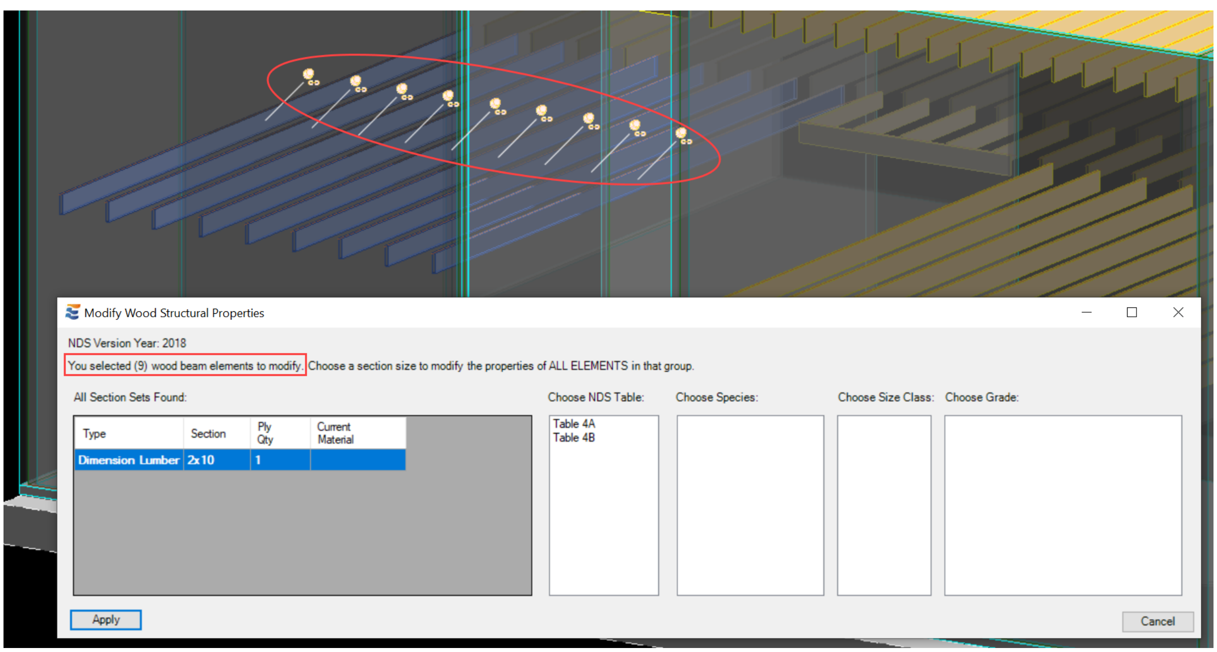
1. Multi-element pick + "Finish": Only the selected elements will be displayed in the edit menu.

2. "Cancel": All wood beam elements found in the entire model will be displayed in the edit menu.

Regardless of whether the user opts to select all via "Cancel" or only select certain elements via picking / looping, selected elements will be displayed in the table, broken down by section sets.

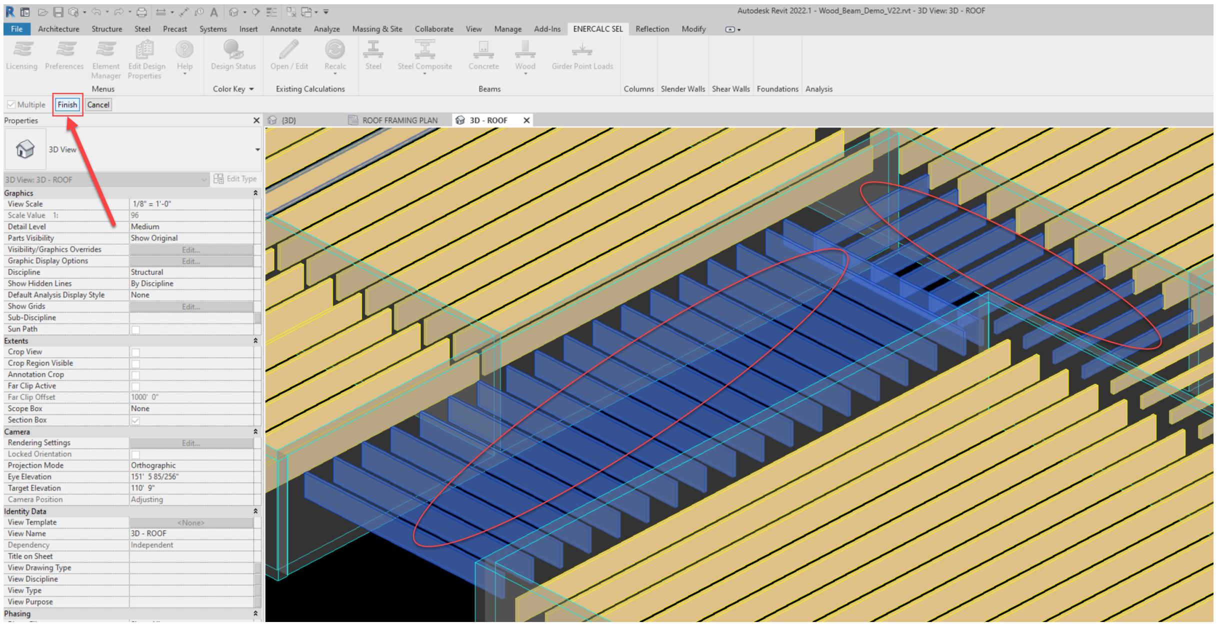


If all of the selected elements share a common section size and design properties, then they will appear as a single entry in the table:

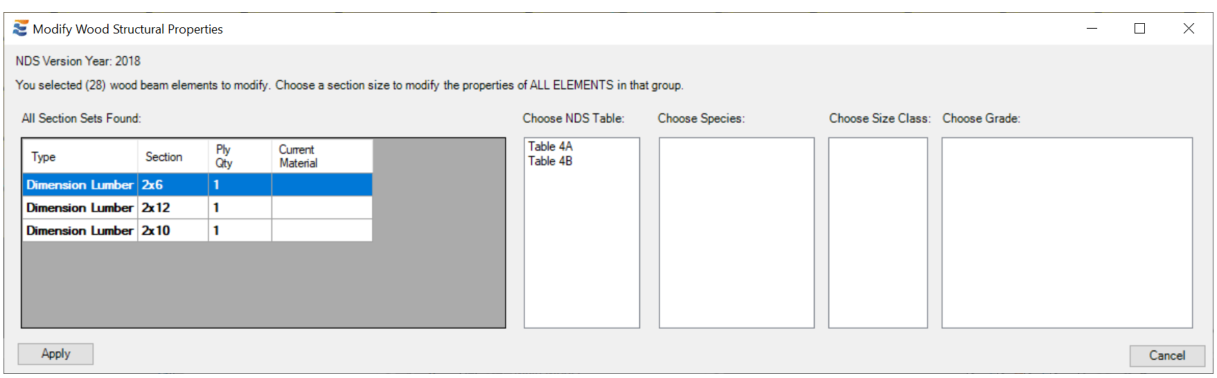


Any design properties specified in the menu will then be applied to all of the selected elements when the "Apply" button is clicked.

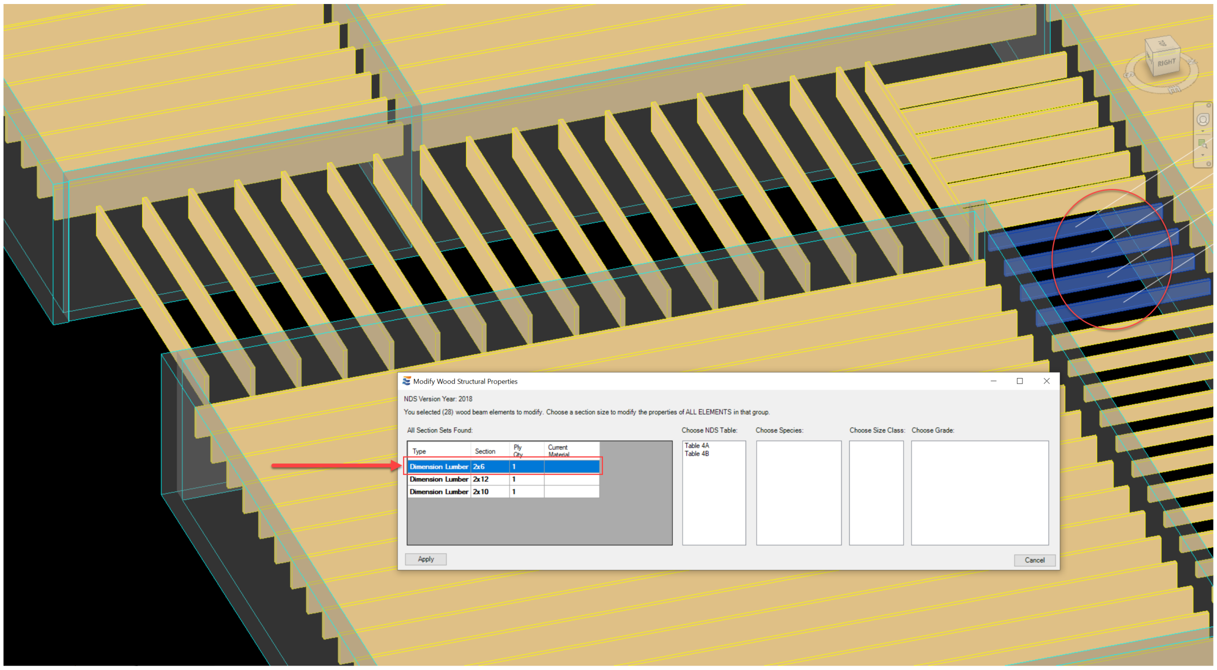
If the bulk selection encompasses elements of differing section size, then the differing sections will be grouped into individual line items in the table:

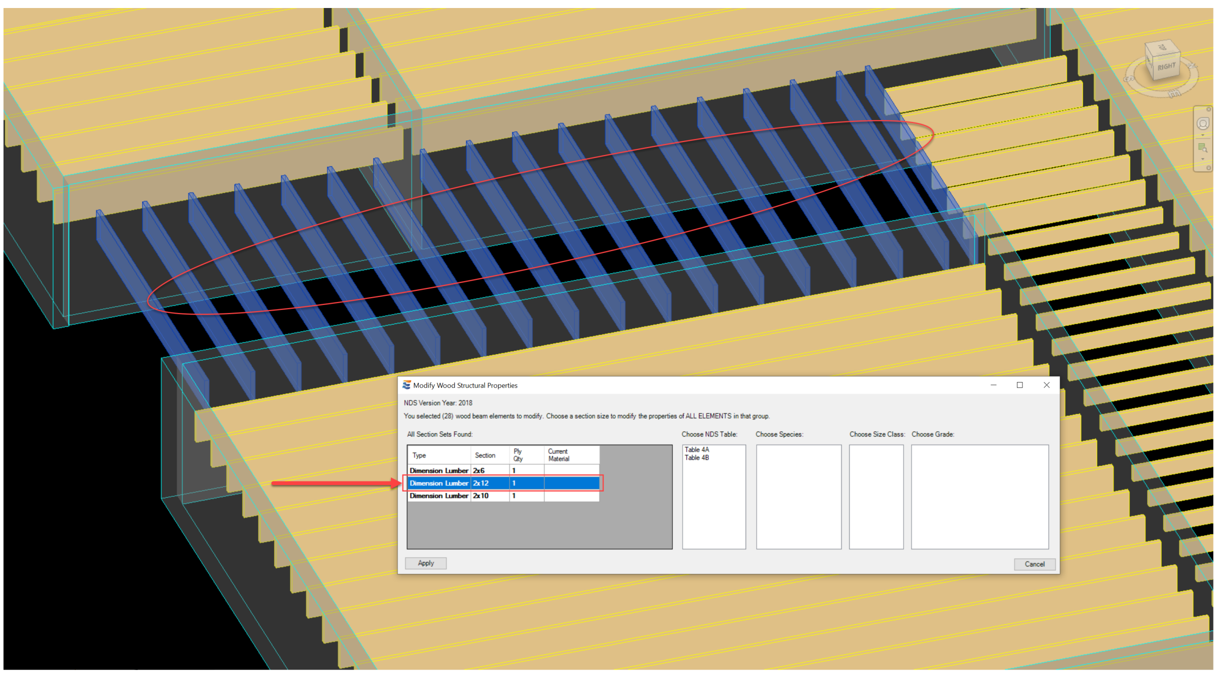


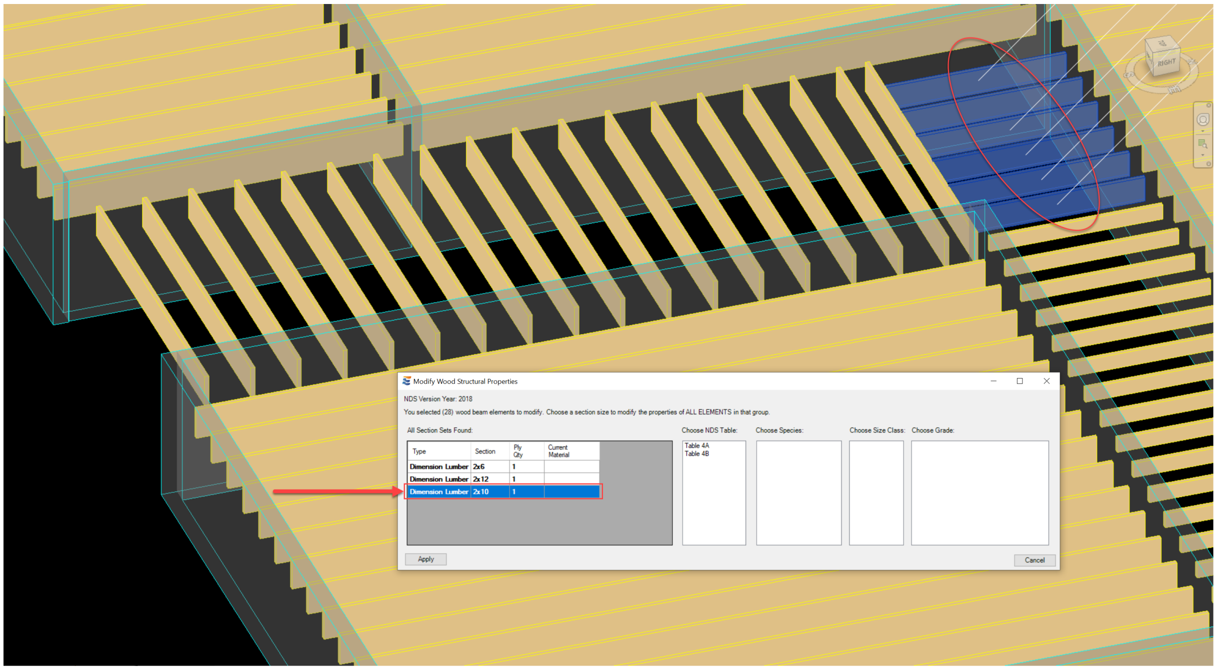
When the edit window opens, the display table will include a separate line item for each individual section type found:



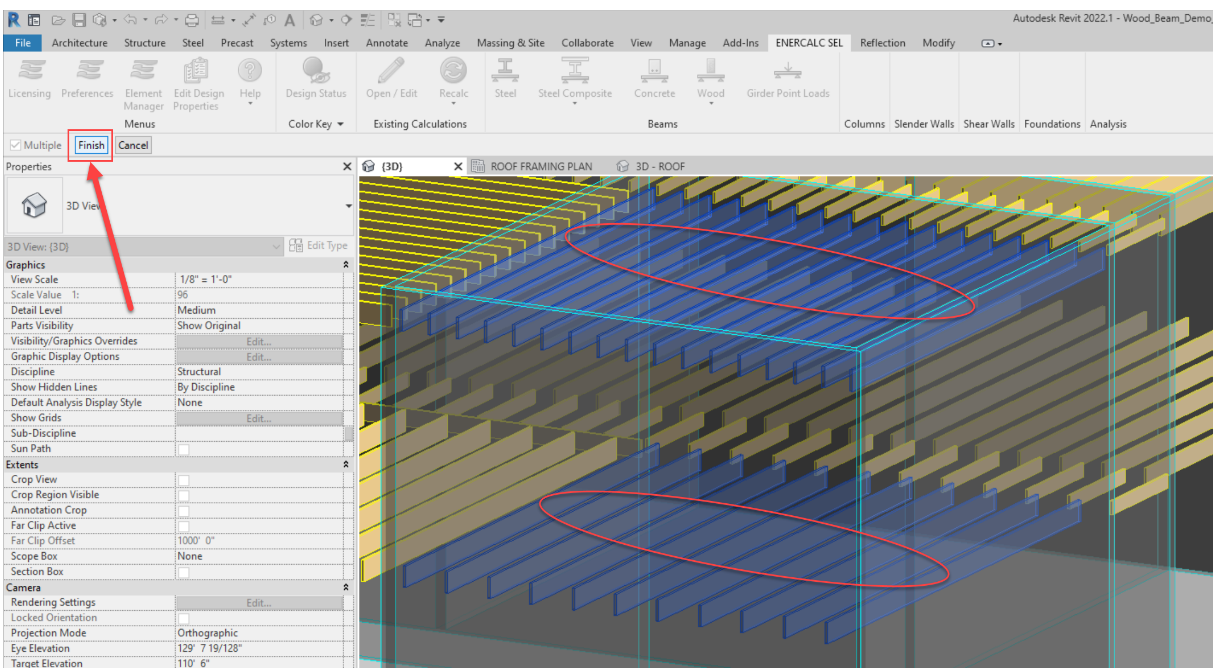
Selecting a line item in the table will automatically select the corresponding beam elements in the active Revit view for visual reference:

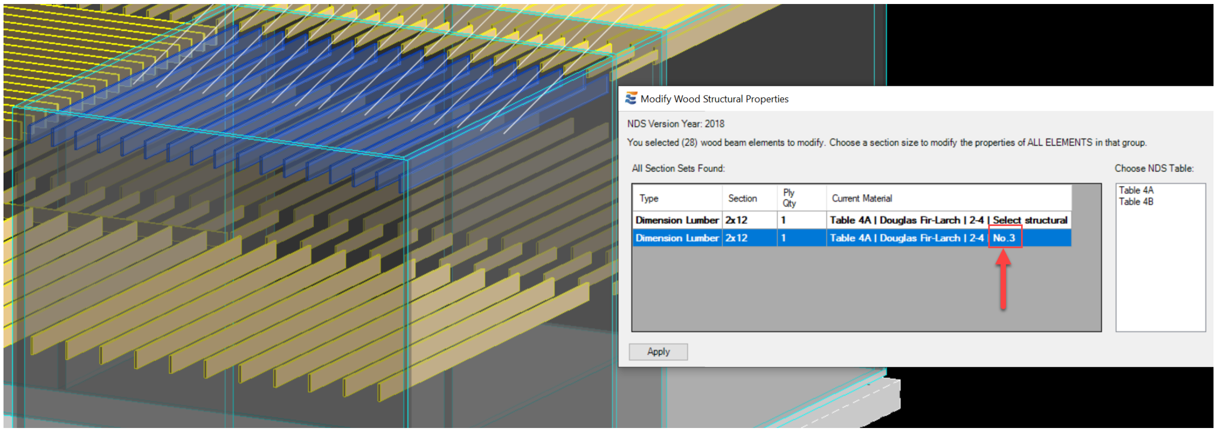

Modify Wood Structural Properties
NDS Version Year: 2018
You selected (28) wood beam elements to modify. Choose a section size to modify the properties of ALL ELEMENTS in that group.
All Section Sets Found:
Type
Section
Ply Qty
Current Material
Dimension Lumber 2x6 1
Dimension Lumber 2x12 1
Dimension Lumber 2x10 1
Choose NDS Table:
Table 4A
Table 4B
Choose Species:
Choose Size Class:
Choose Grade:
Apply
Cancel

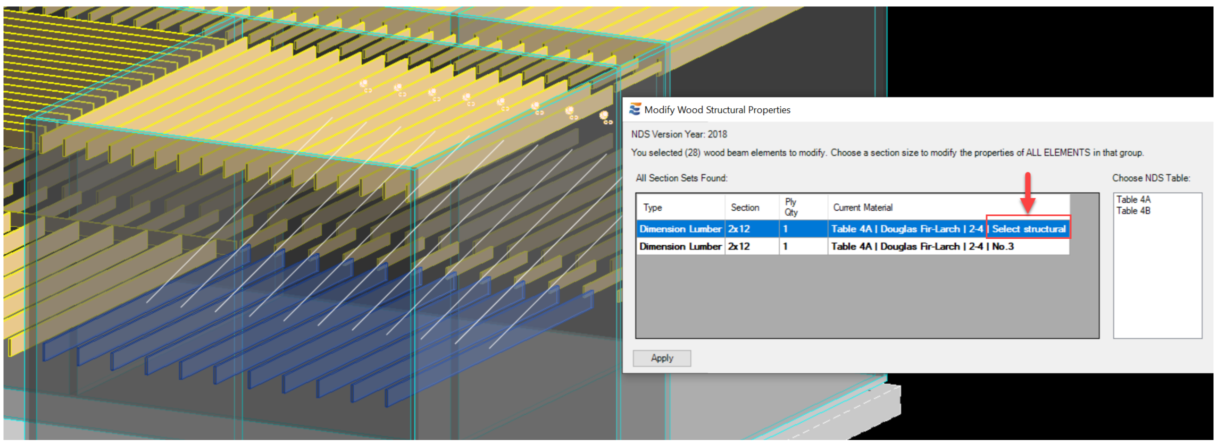

Modify Wood Structural Properties
NDS Version Year: 2018
You selected (28) wood beam elements to modify. Choose a section size to modify the properties of ALL ELEMENTS in that group.
All Section Sets Found:
Type
Section
Ply Qty
Current Material
Dimension Lumber 2x6 1
Dimension Lumber 2x12 1
Dimension Lumber 2x10 1
Choose NDS Table:
Table 4A
Table 4B
Choose Species:
Choose Size Class:
Choose Grade:
Apply
Cancel



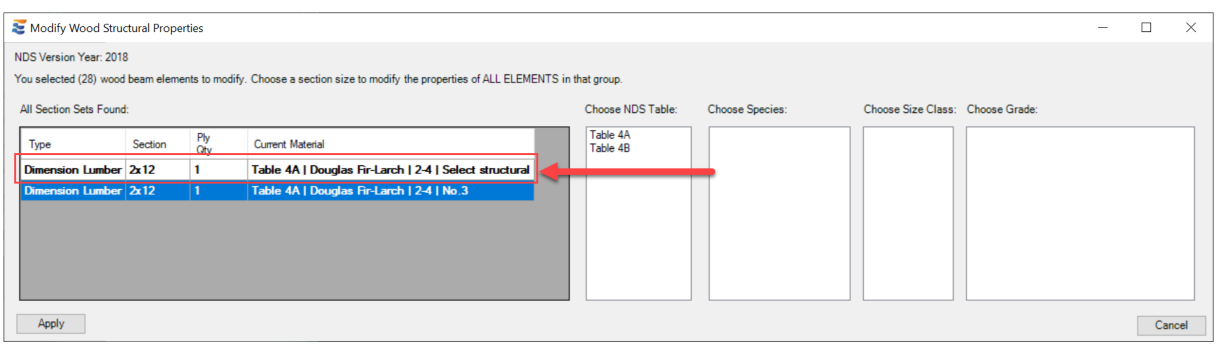
If the selection for launching the edit window contains beams with the same section size but differing design properties, then the groups will be given separate line items in the table:



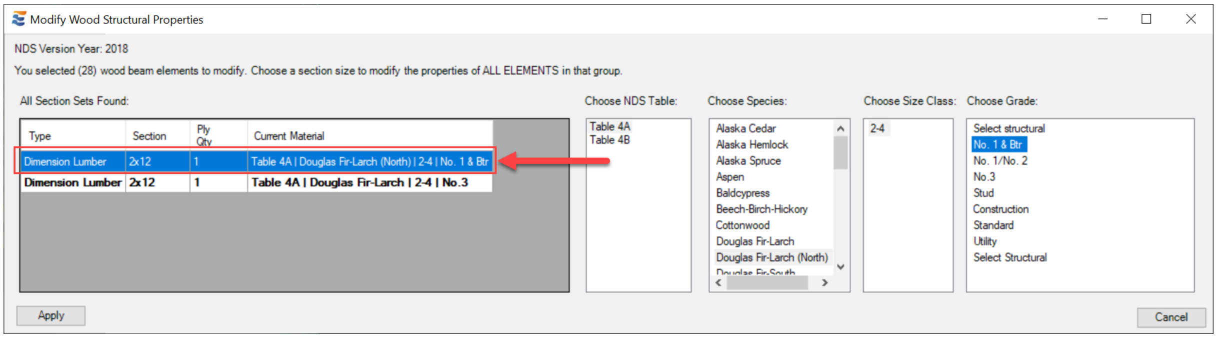




When a line item in the display table has not yet been modified or had any properties assigned to it, the text will appear in **bold** font:



Once a complete set of material properties has been assigned, the **bold** font will change to standard font:

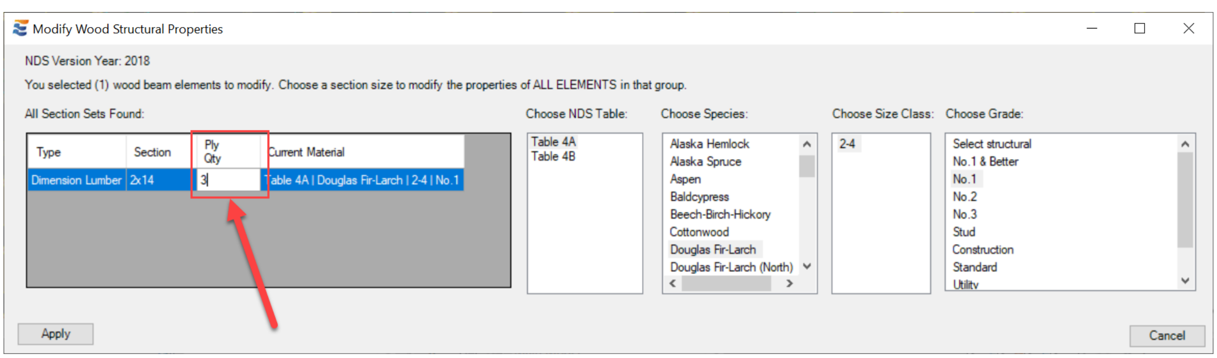


When multiple section sets are present in the table, clicking the "Apply" button will apply each individual line item's design properties to ALL of the elements associated with that particular line item (i.e., only the beams highlighted when that row is picked). Any line item lacking a complete definition will not be applied.

10.13.3 Multiple Ply Beams

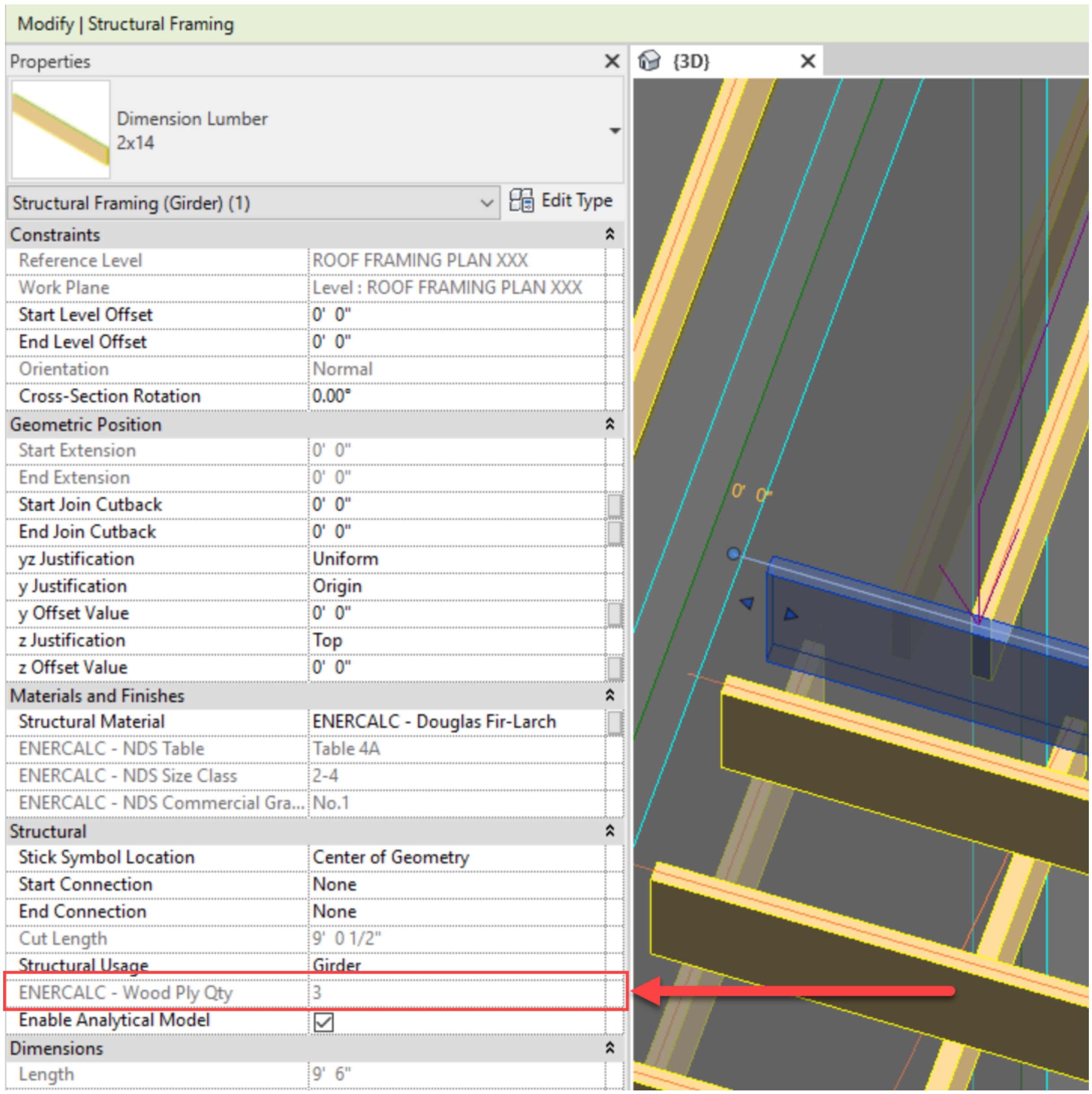
Where applicable, users may choose to create a design where multiple plys of an NDS standard are used together to achieve structural performance. In structural design practice, this is predominately done with dimensional lumber and occasionally with certain engineered sections (i.e., LVLs). ENERCALC for Revit supports interoperability with the specific sections in the ENERCALC database that support multiple-ply design.

The design properties for launching a wood beam calculation include a ply quantity parameter to assist the user with constructing designs of this nature. This ply quantity may be directly modified using the corresponding cell in the "Modify Wood Structural Properties" table.

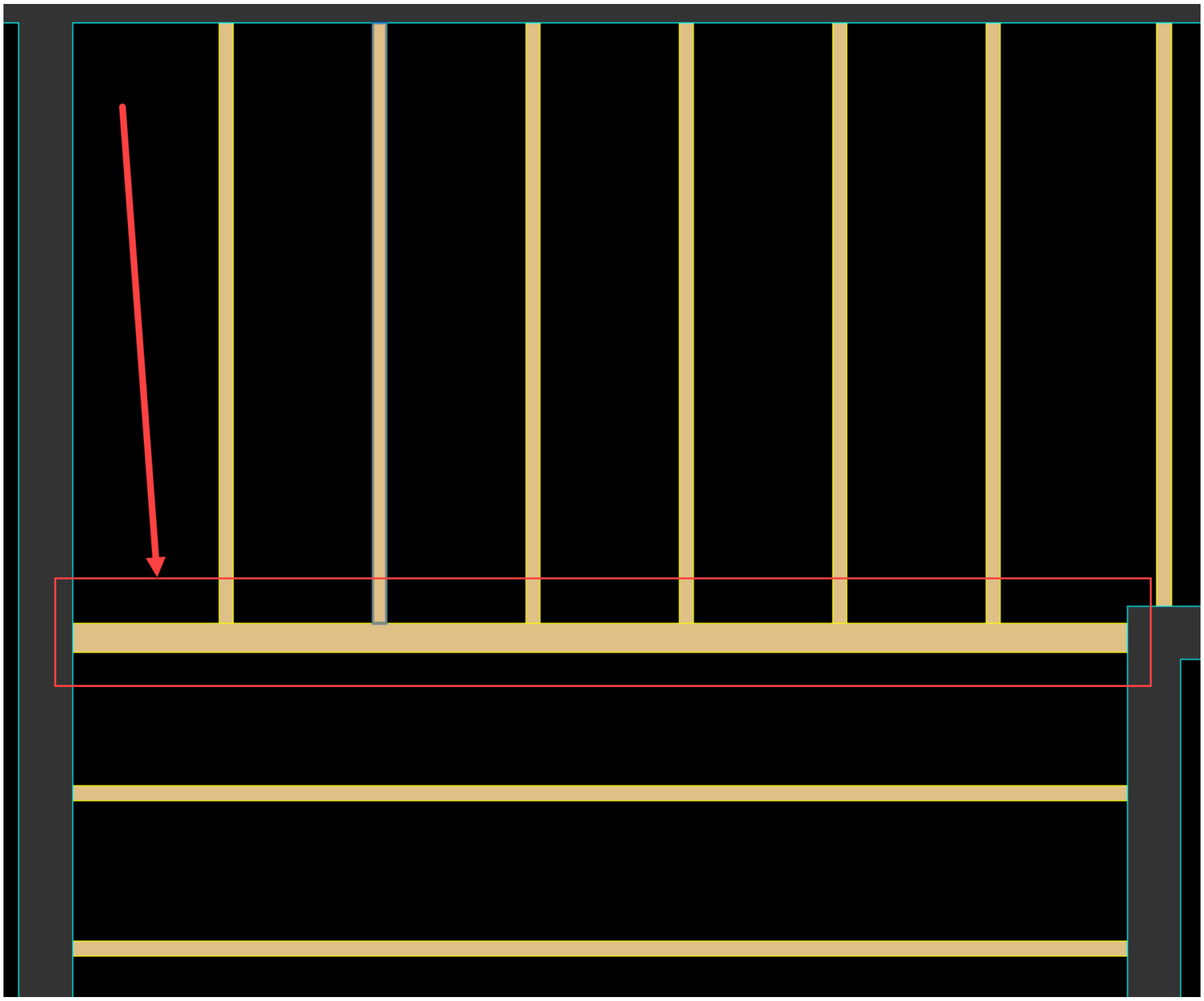


Changing the ply quantity does not have any influence over the physically modeled Revit beam instance. It does not change the 2D or 3D graphics of the physical beam element in the Revit model, nor does it alter the total number of actual beams modeled in the Revit project. It is purely a numerical parameter that allows ENERCALC for Revit to communicate the appropriate section geometry to ENERCALC when building the calculation. If desired, the parameter may also be referenced in schedules or quantity take-offs, since it is a Revit "Shared Parameter". Further reading on this topic is available at the conclusion of this section. Users should also be aware that modifying the ply quantity on a section type for which ENERCALC does not support multi-ply configurations (i.e., glulam beams) will result in an error when attempting to launch the calculation.

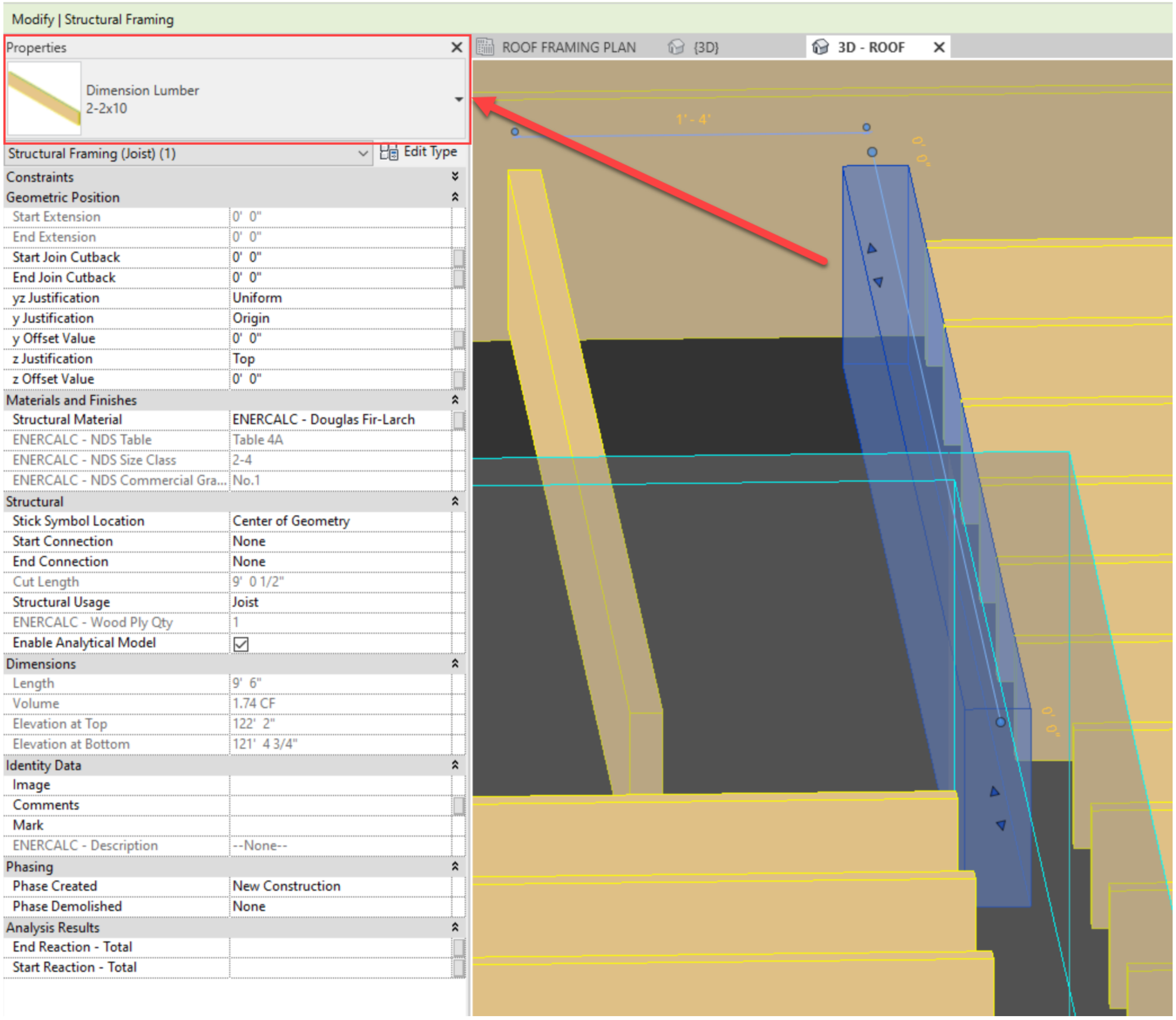
If the lumber ply quantity was changed using the "Modify Wood Properties" menu, the new value will be displayed in a parameter in the beam's "Properties" pane:



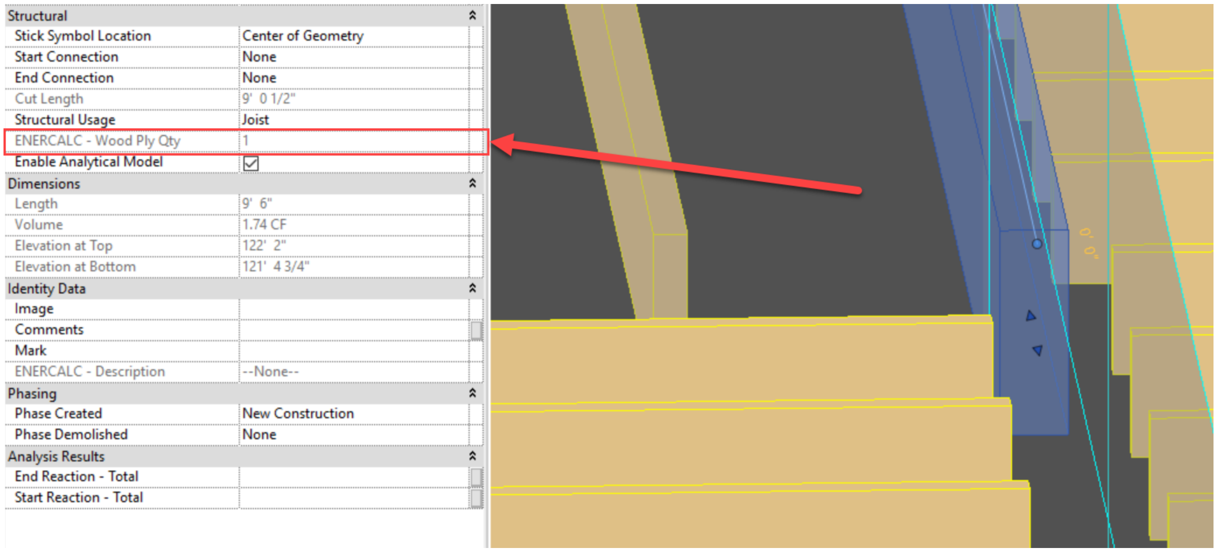
Rather than using default sections with parametric ply indicators, a design team may alternatively choose to use custom beam sections which represent accurate geometry and graphical appearance of a multiple-ply beam:



In such cases, the multiple-ply beam sections should be named using the exact same naming convention as that used for the corresponding multiple-ply beam option in the ENERCALC database in order to ensure interoperability. Any divergence in naming convention or use of a multiple-ply section not found in the ENERCALC database will result in an error when attempting to build the calculation.

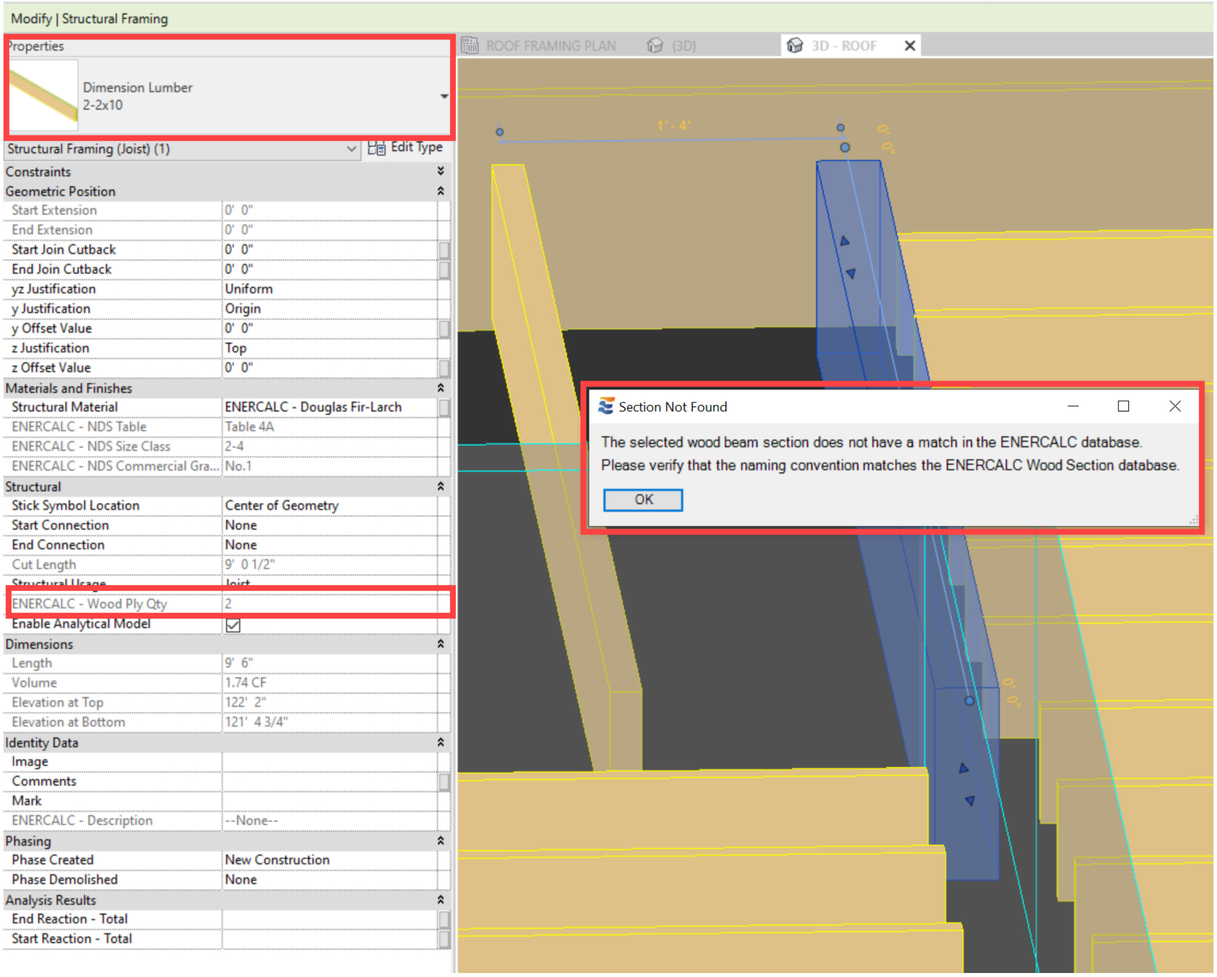


When the ply quantity is written directly into the name of the custom multiple-ply section, the parameter "ENERCALC - Wood Ply Qty" should always be set to a value of 1 (see below).

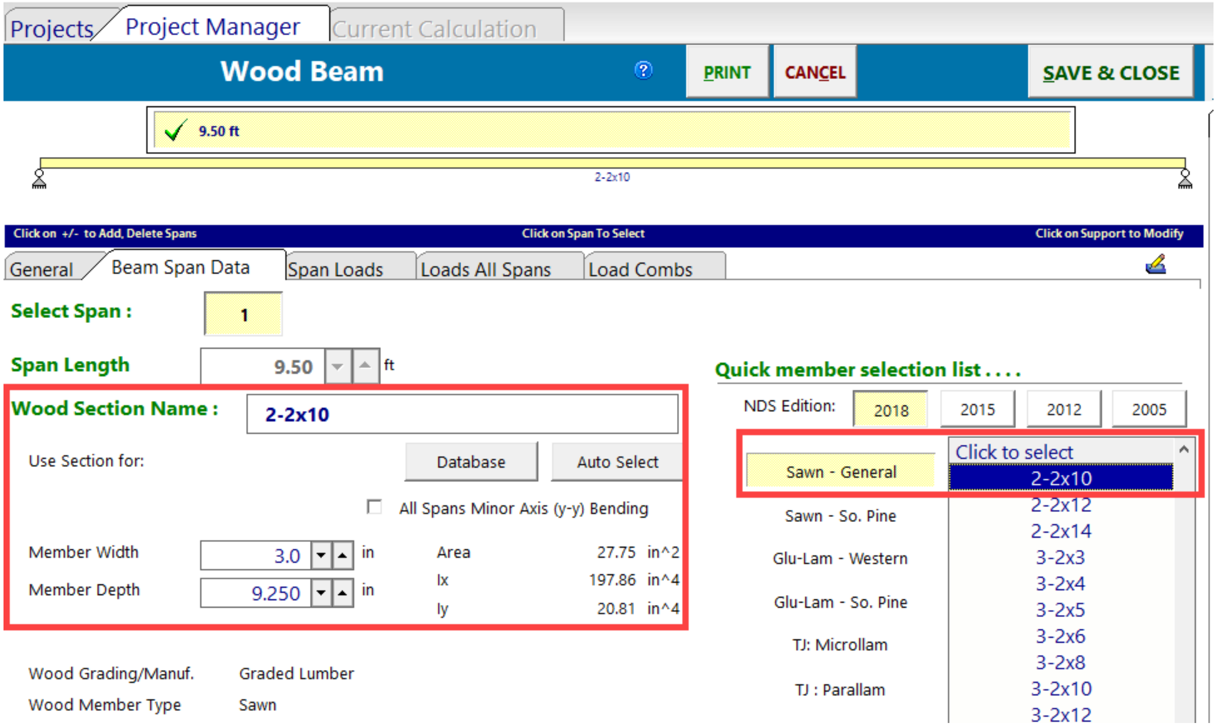


In this case, the "1" will be ignored and the appropriate ply quantity will instead be inferred directly from the section name. For example, in the case above, EFR will automatically recognize that the above section label "2-2x10" is a 2-ply beam made of 2x10 dimension lumber.

Use of any value other than 1 in such cases will result in EFR attempting to use a section name obtained by using the literal name of the beam instance prefixed with the ply quantity specified in the parameter. For example: 2-2-2x10:

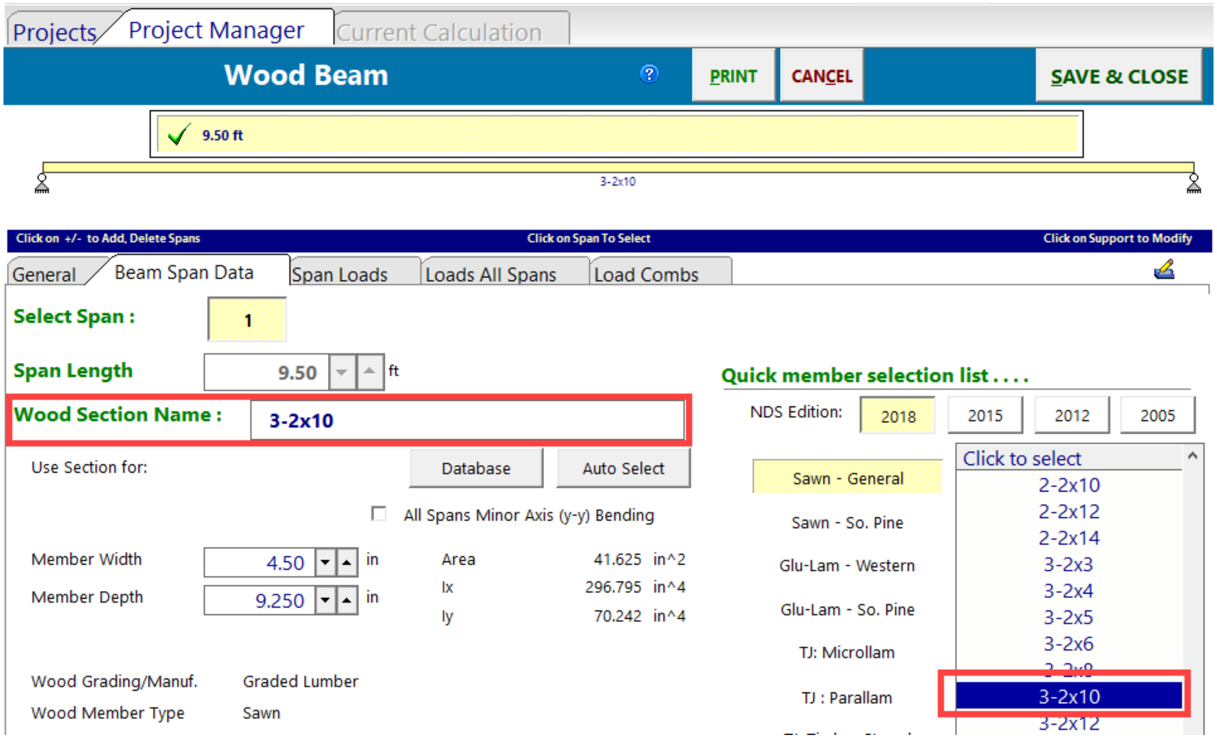


When a calculation is launched using a valid multiple-ply beam section name, the calculation will populate in the ENERCALC interface the same way, regardless of whether it was generated from a custom section or from a default section with a quantity parameter:



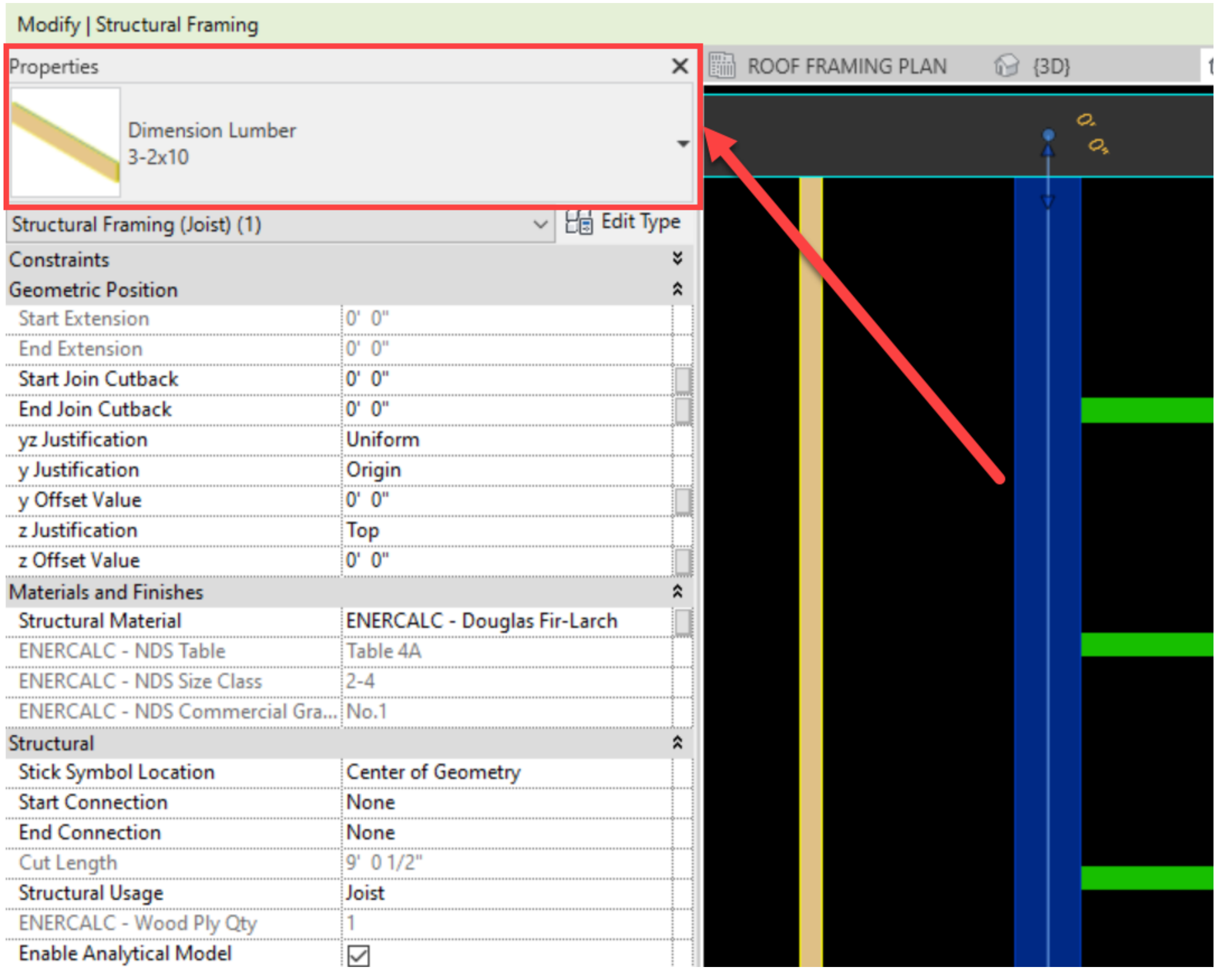
If the user then selects a single-ply section type in the ENERCALC interface, the ply qty parameter on the Revit element will automatically revert to a value of "1" when the "Save and Close" operation concludes.

Conversely, the user may choose to select a new multiple-ply section size:

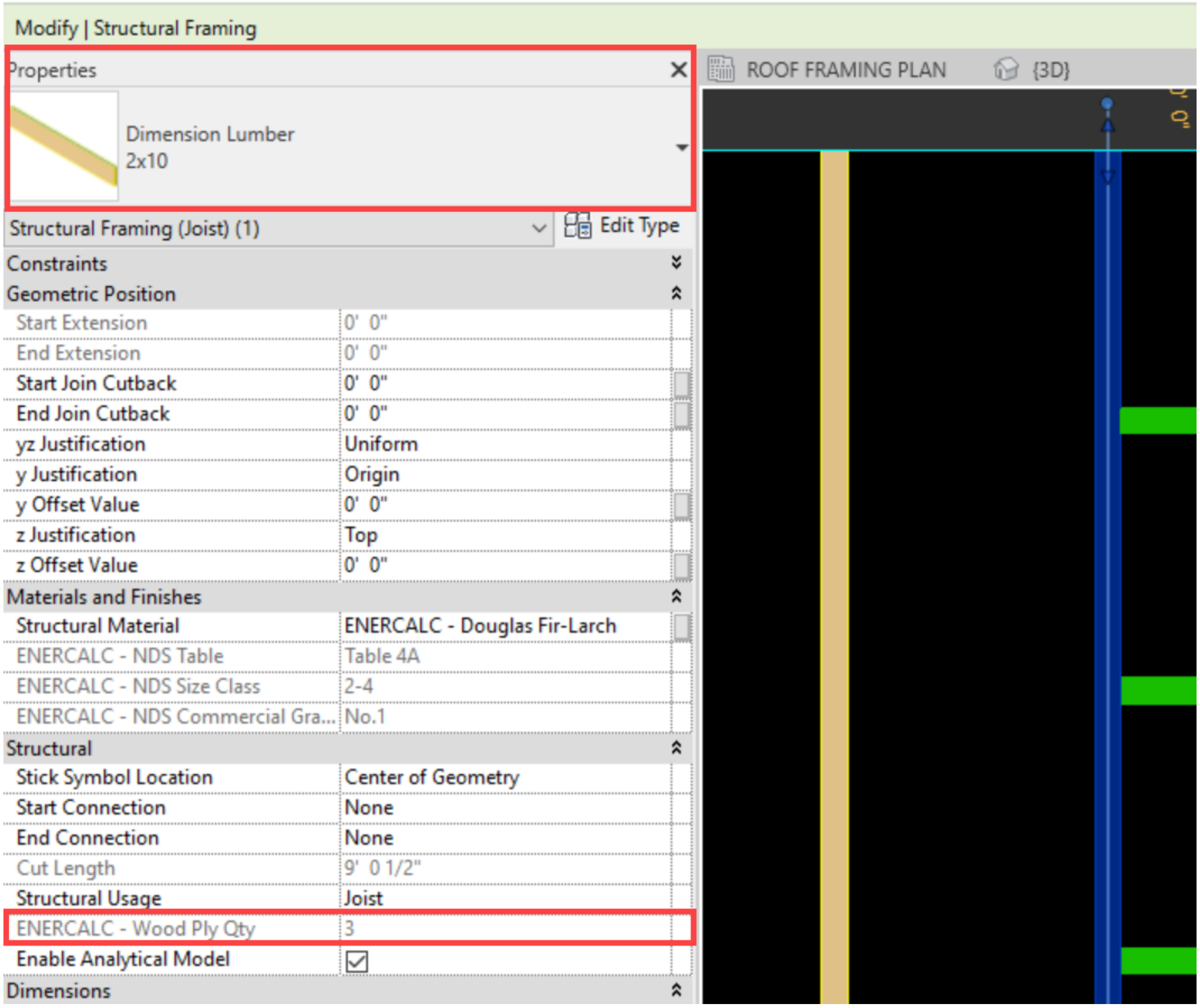


In this case, the following steps will occur during the "Save and Close" operation:

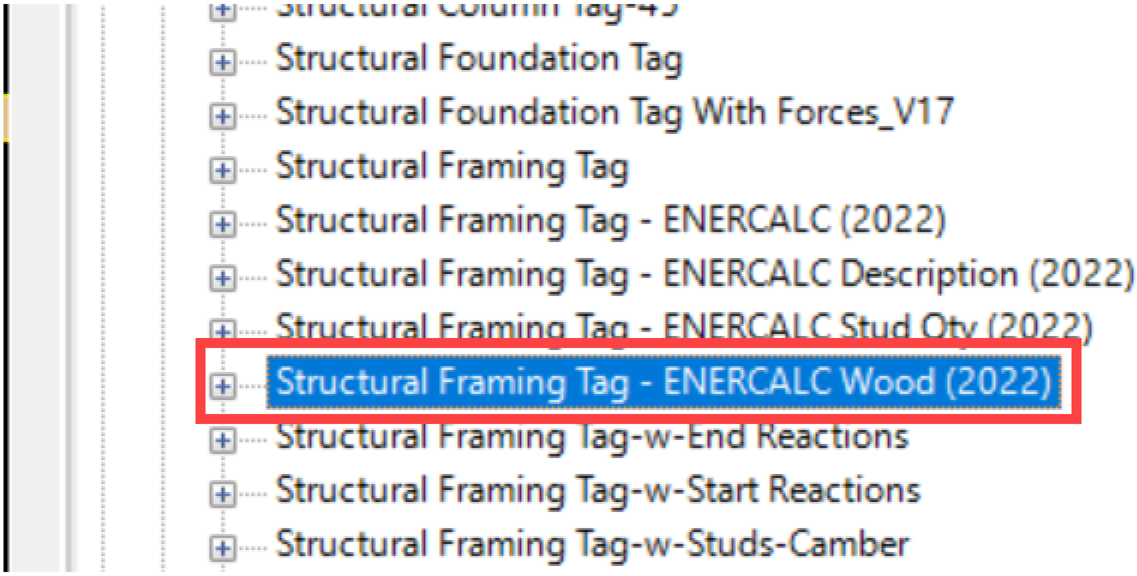
1. EFR will search both the current .rvt project AND the appropriate loadable content directories for a section whose type name is a perfect match for the literal name sent from ENERCALC. If a custom section having the desired name is found, EFR will apply it automatically:

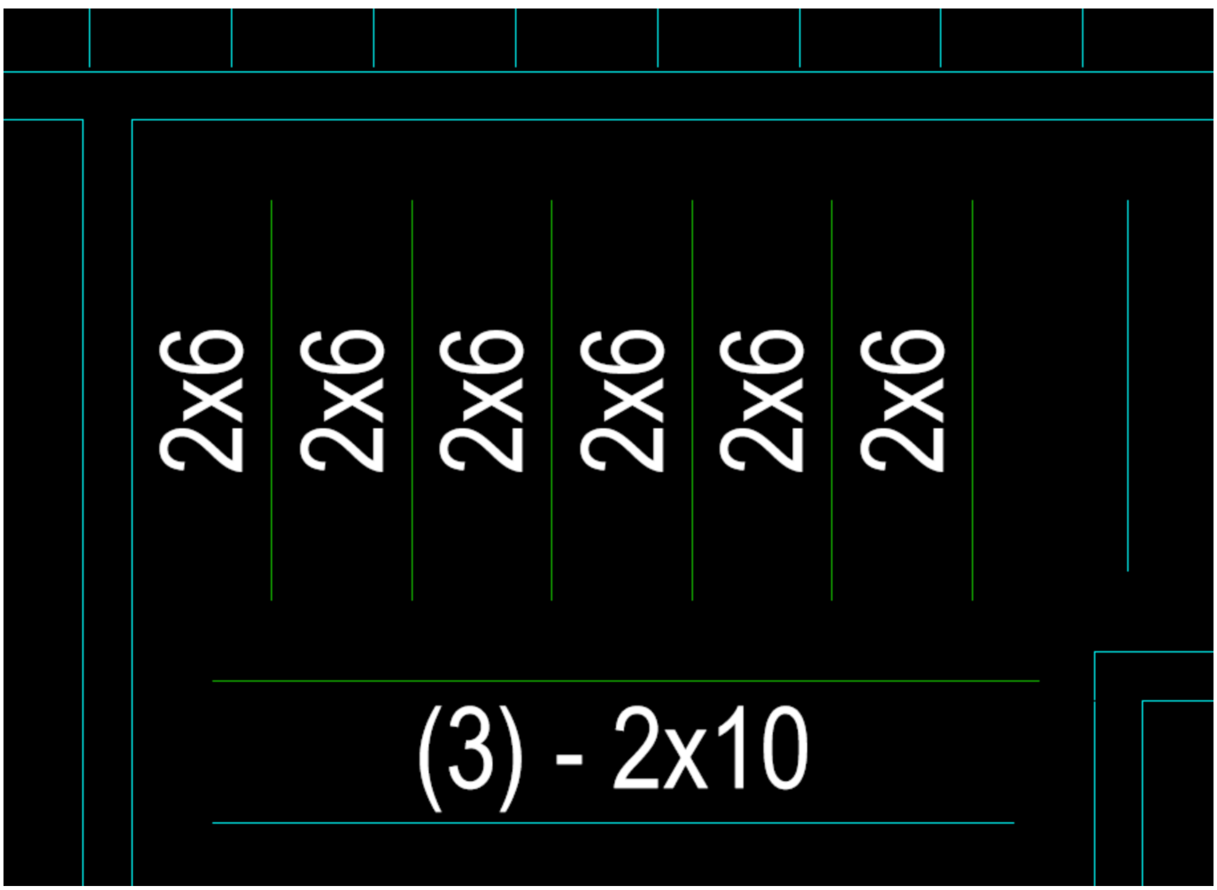


2. If a section of the desired multiple-ply name is not found either in the project OR in the appropriate loadable content directories, then EFR will apply updates to the Revit model using the default single-ply section, coupled with an appropriate ply quantity parameter:



ENERCALC for Revit provides a structural tag family for easy documentation on plans and other views for situations where a multiple-ply beam has been designed using the ply quantity parameter:


Structural Foundation Tag
Structural Foundation Tag With Forces_V17
Structural Framing Tag
Structural Framing Tag - ENERCALC (2022)
Structural Framing Tag - ENERCALC Description (2022)
Structural Framing Tag - ENERCALC Stud Qty (2022)
Structural Framing Tag - ENERCALC Wood (2022)
Structural Framing Tag-w-End Reactions
Structural Framing Tag-w-Start Reactions
Structural Framing Tag-w-Studs-Camber


2x6
2x6
2x6
2x6
2x6
2x6
(3) - 2x10

10.14 Concrete Beam Calculations

Concrete beams share all of the common beam behavior described in the preceding sections.

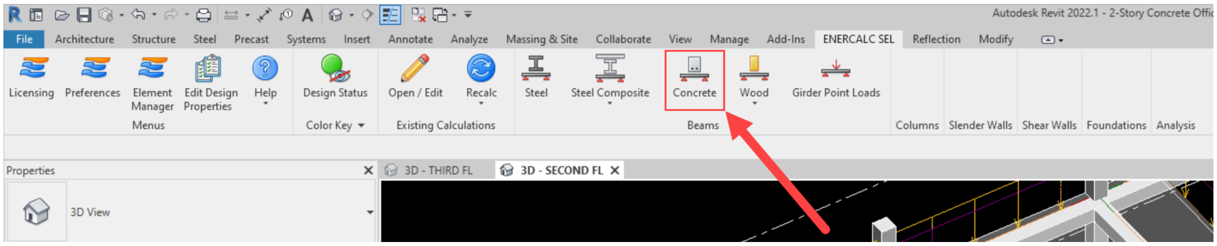
The primary characteristics that distinguish concrete beam calculations from conventional Steel beam, Steel Composite, and Wood calculations are:

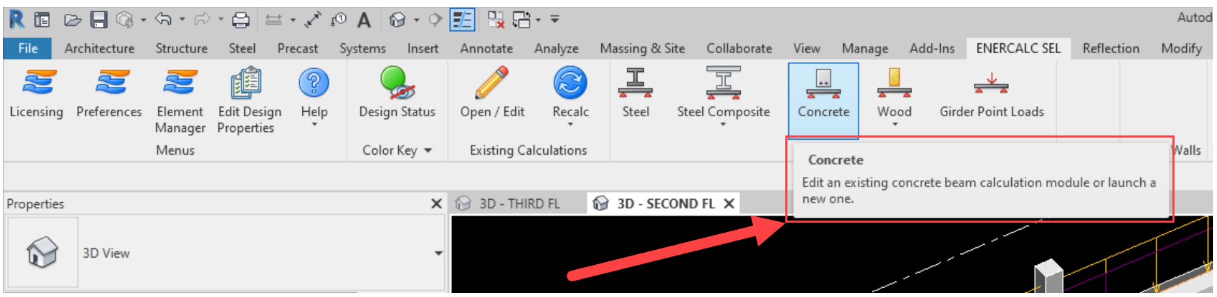
1. Specifying the concrete material properties of the beam
2. Specifying the rebar materials of the beam reinforcement
3. Verifying T-Beam effective width where applicable
4. Open-ended modification of section geometry

Before using ENERCALC for Revit for the design of Concrete Beams, users should take special note of the following limitations:

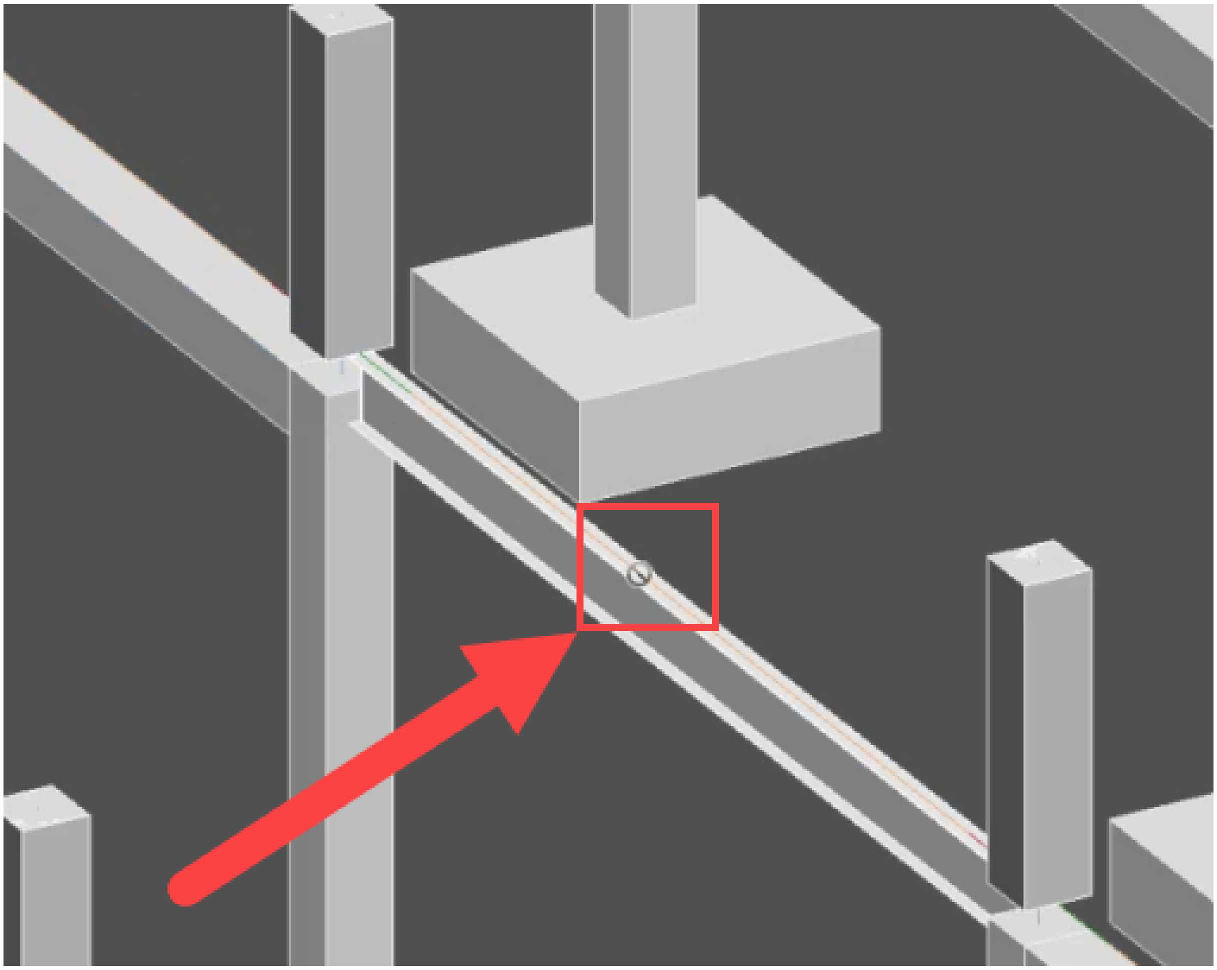
1. **Regardless of how the Revit model is configured, any Concrete Beam sent from Revit to ENERCALC remains subject to the same analysis and design limitations as ENERCALC itself.**
   a. **Members are analyzed individually, not as an integral frame.**
   b. **No P-Delta or P-delta analysis is performed.**
   c. **No torsional analysis is performed.**
   d. **Additional net compression in concrete beams due to axial load is disregarded.**
   e. **Supports are treated as “knife-edge” (zero length).**
   f. **Stiffness is calculated at each analytical location using Bischoff’s method, not a percentage of gross properties.**
   g. **Creep is not considered.**
2. **Concrete Beam support locations may ONLY be analyzed as either fully fixed, pinned, or free.**
3. **Concrete Beam support locations DO NOT consider the stiffness of surrounding connected elements.**
4. **As described under general beam behavior (see Beam/Girder Load Linking 250), Concrete Beams with load-linking relationships consider only reaction FORCES. Fixed-end moments are NOT transferred between connected elements via load-linking.**
5. **ENERCALC for Revit does NOT provide any integration with or control over physically modeled rebar elements in the Revit model at this time.**
6. **As with manually-built Concrete Beam calculations, ENERCALC for Revit does not permit differing section geometries on different spans. All spans of a multi-span beam have the same section geometry.**

As with other calculations, Concrete Beam calculations are launched from the ENERCALC for Revit ribbon tab in the Revit interface.





As with other calculation types, clicking the calculation launch button will trigger a selection process and will await an element pick by the user.
During this process, only beams that are eligible for Concrete Beam design will be available for selection. Ineligible elements are denoted with a "not allowed" circle and slit icon on the cursor when mousing over:



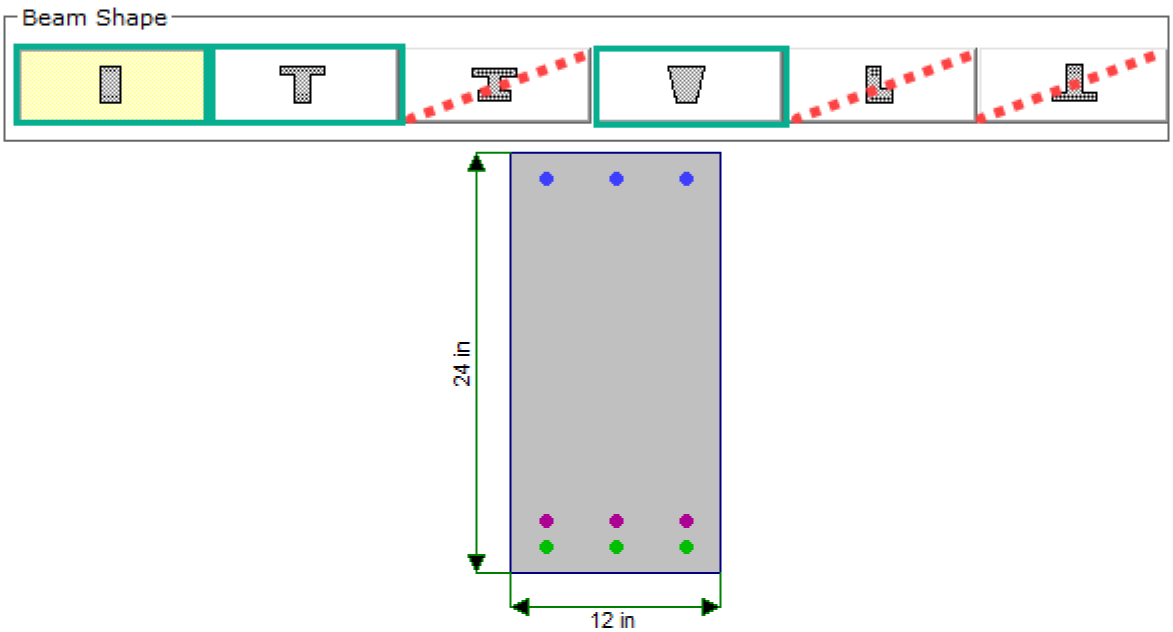
10.14.1 Beam Section Limitations

Design of concrete beams using ENERCALC for Revit is limited to the following (3) section types:

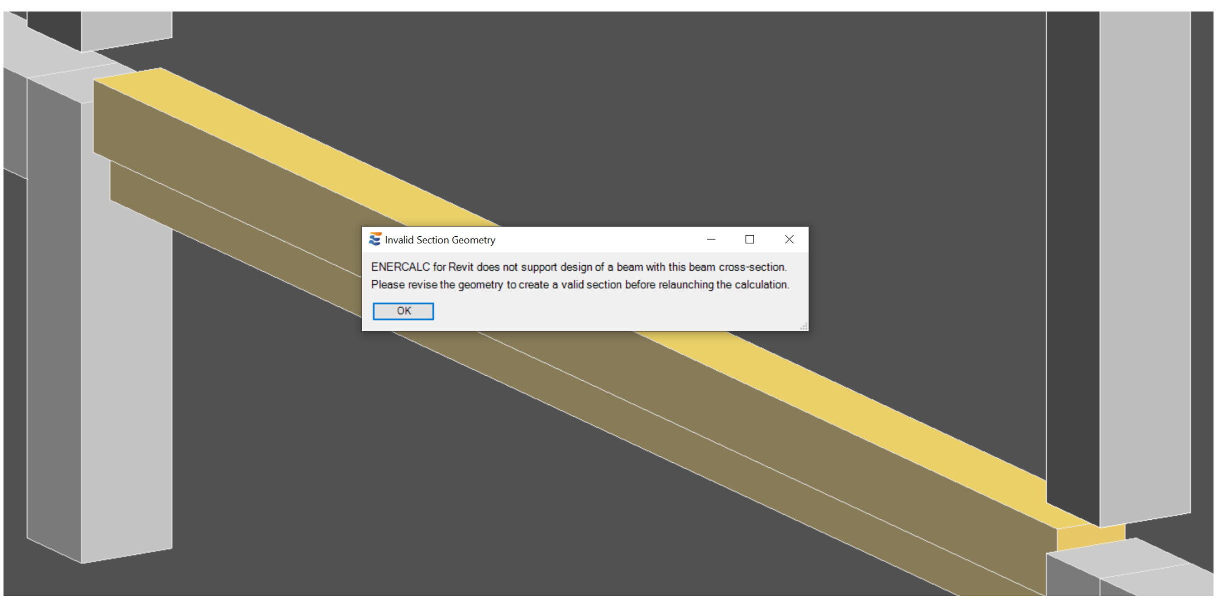
- Rectangular Beams: Rectangular Beam Sections - Unjoined 454 OR Rectangular Beam Sections - Joined 461
- T Beams: T-Beam Sections 468
- Trapezoidal Beams: Trapezoidal Beam Sections 476

The (3) additional types supported in ENERCALC but not in ENERCALC for Revit do not have physically modeled counterparts in Revit's default out-of-box family content:

- I (or H) Beams
- L Beams
- Inverted T Beams

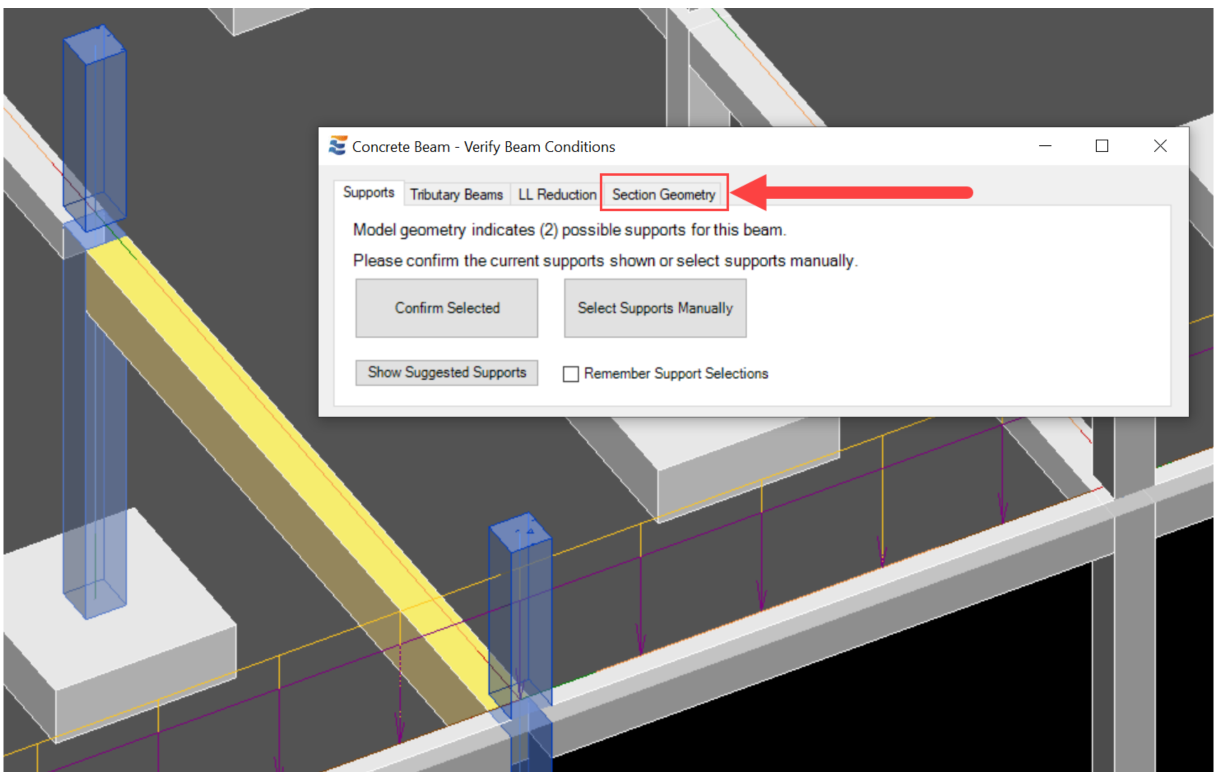


Any attempt to launch a calculation with an unsupported cross-section geometry will be halted with the following warning:



In cases where the concrete beam element is found to be joined to a concrete floor slab, calculation launch will be halted with a warning if the beam is found to have a slope that differs from that of the slab:

Any verifications, approvals, or manual changes related to the geometry of the beam cross-section will be addressed on the tab labeled "Section Geometry":



10.14.2 Rectangular Beam Sections - Unjoined

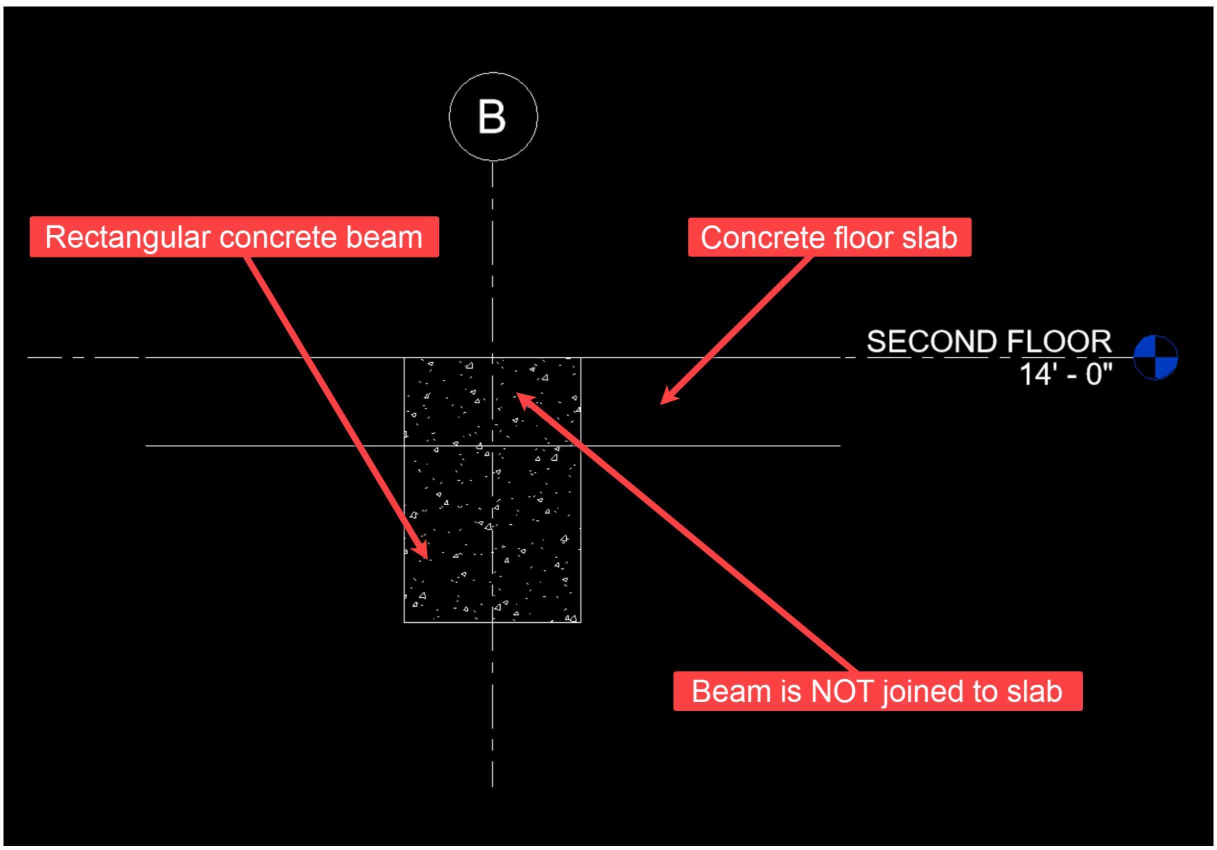
Revit provides the ability for users to control whether or not adjacent physical elements are joined to each other. When they are joined, their solid geometries will be combined, rather than having two distinct geometries which overlap in 3D space and 2D views. Since the design behavior differs in joined vs. unjoined conditions, ENERCALC for Revit users should remain aware of the join relationships between concrete beams and concrete floors in a Revit model.

The simplest approach to launch a Concrete Beam calculation as a basic rectangular beam is to select a beam in the Revit model that meets to following criteria:

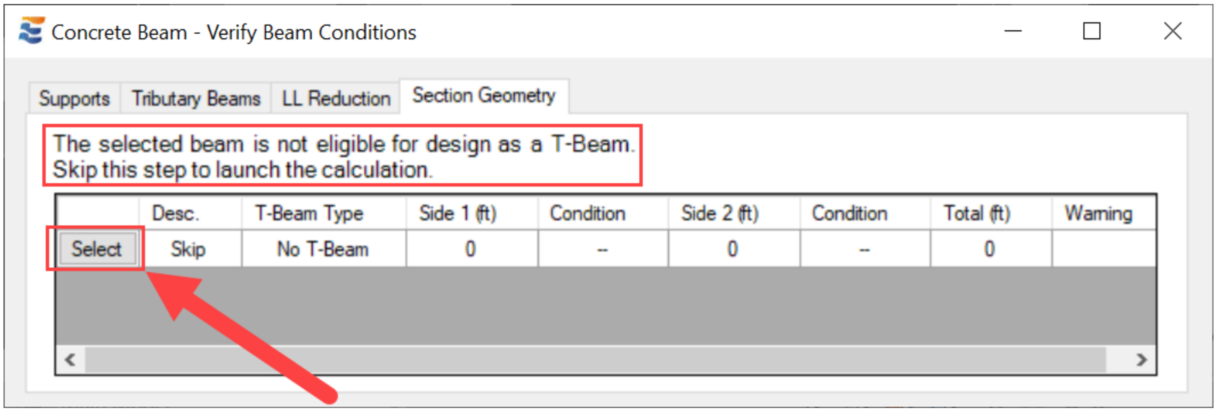
1. Has a rectangular cross-section

AND

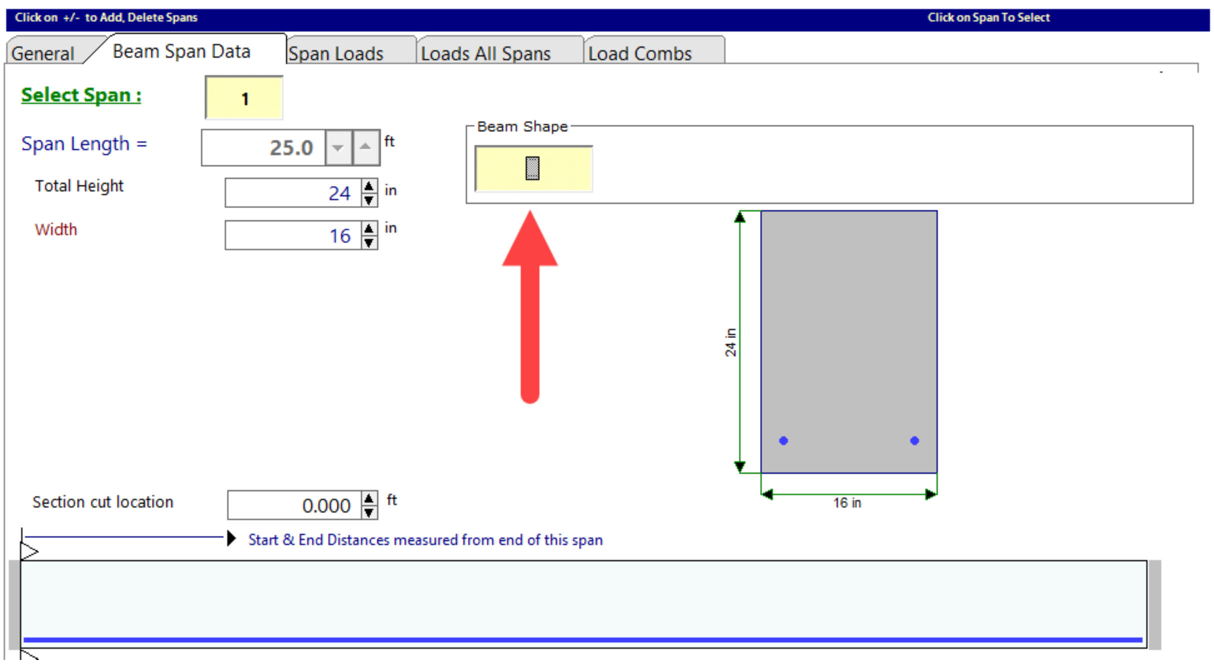
2. Is not joined to any slab



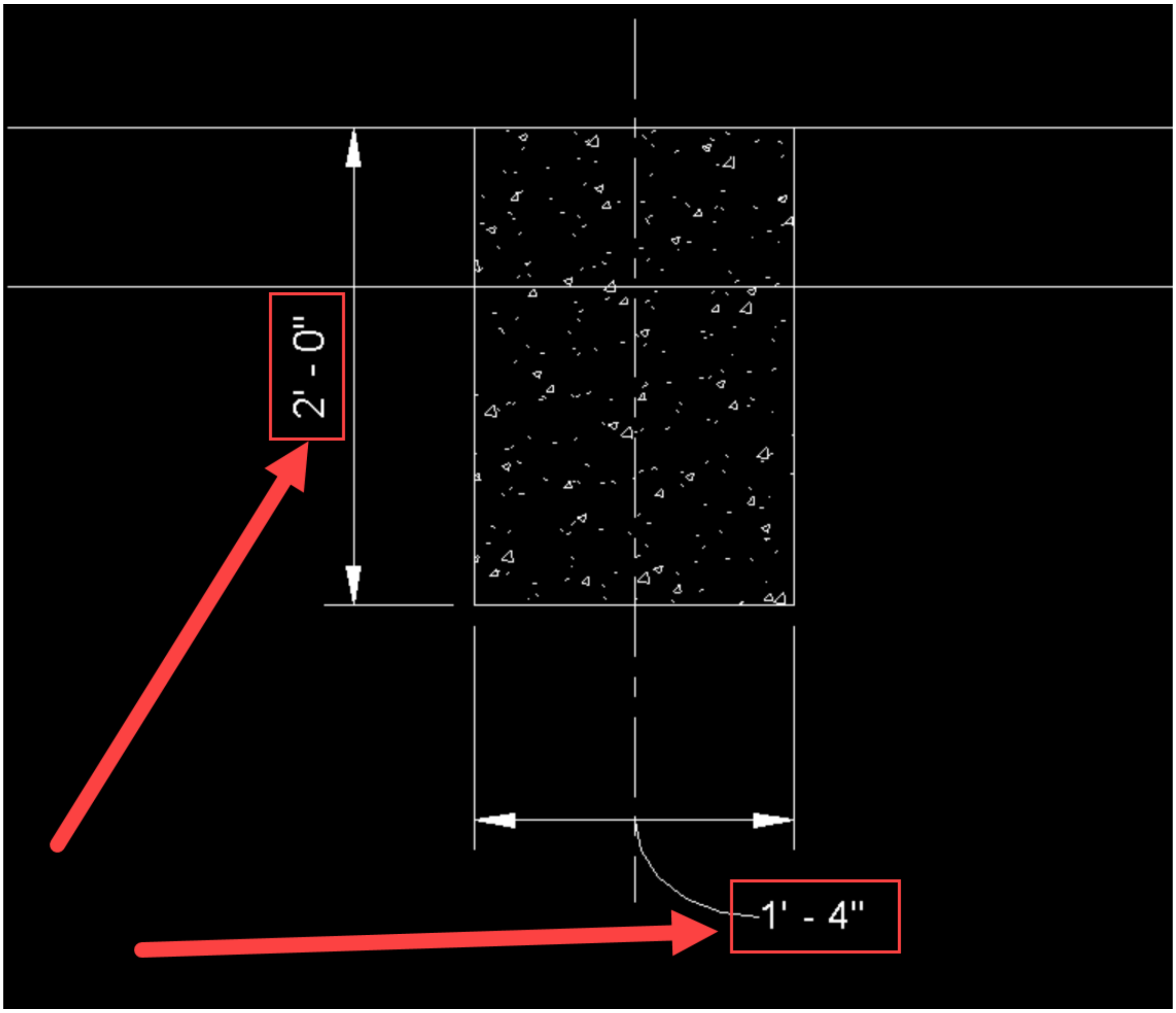
When the calculation launch process reaches the "Section Geometry" tab, the user will be informed that the beam is not eligible for design as a T-Beam. Using the "Select" button on the row marked "Skip" and "No T-Beam" will launch the calculation:

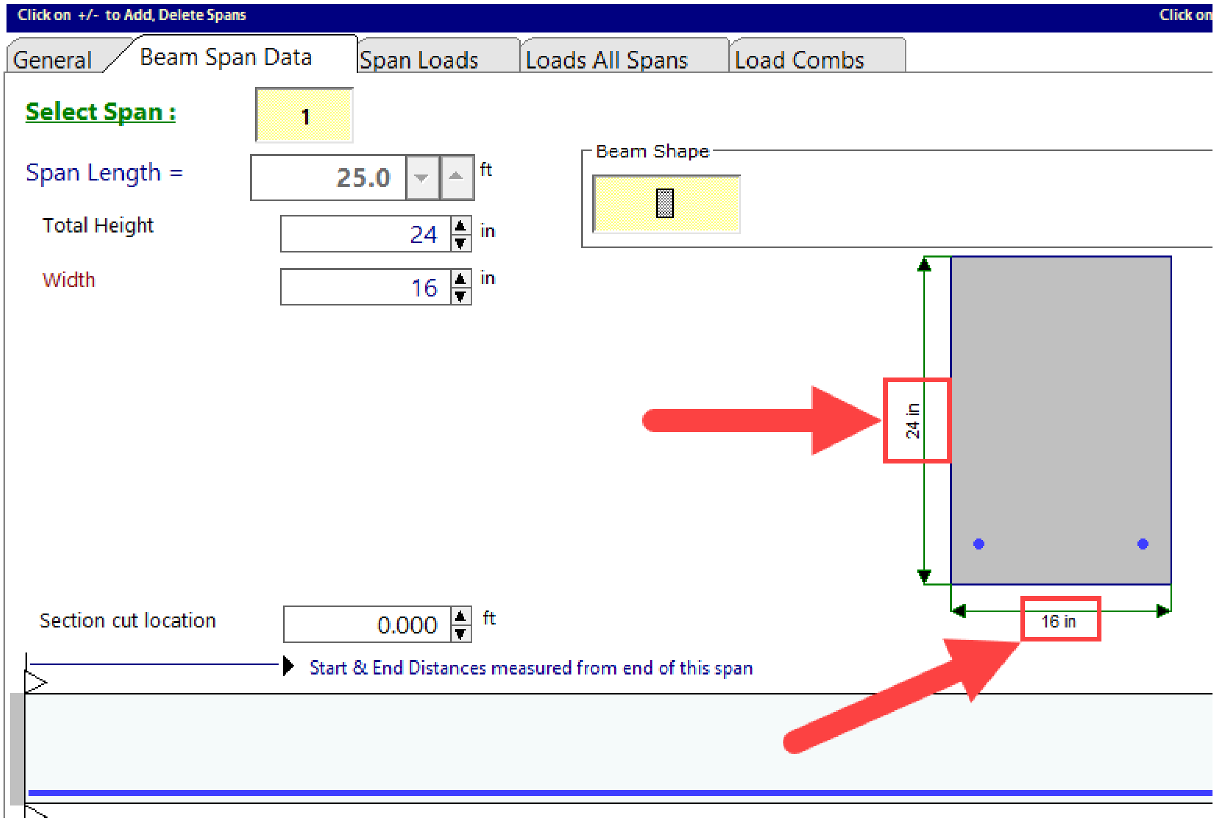


When a concrete beam calculation loads in the ENERCALC interface, the user will not find options to toggle the beam section type (unlike conventional non-Revit-linked calculations). If a concrete beam is sent to ENERCALC as a rectangular beam, then the calculation will load with only this option displayed:

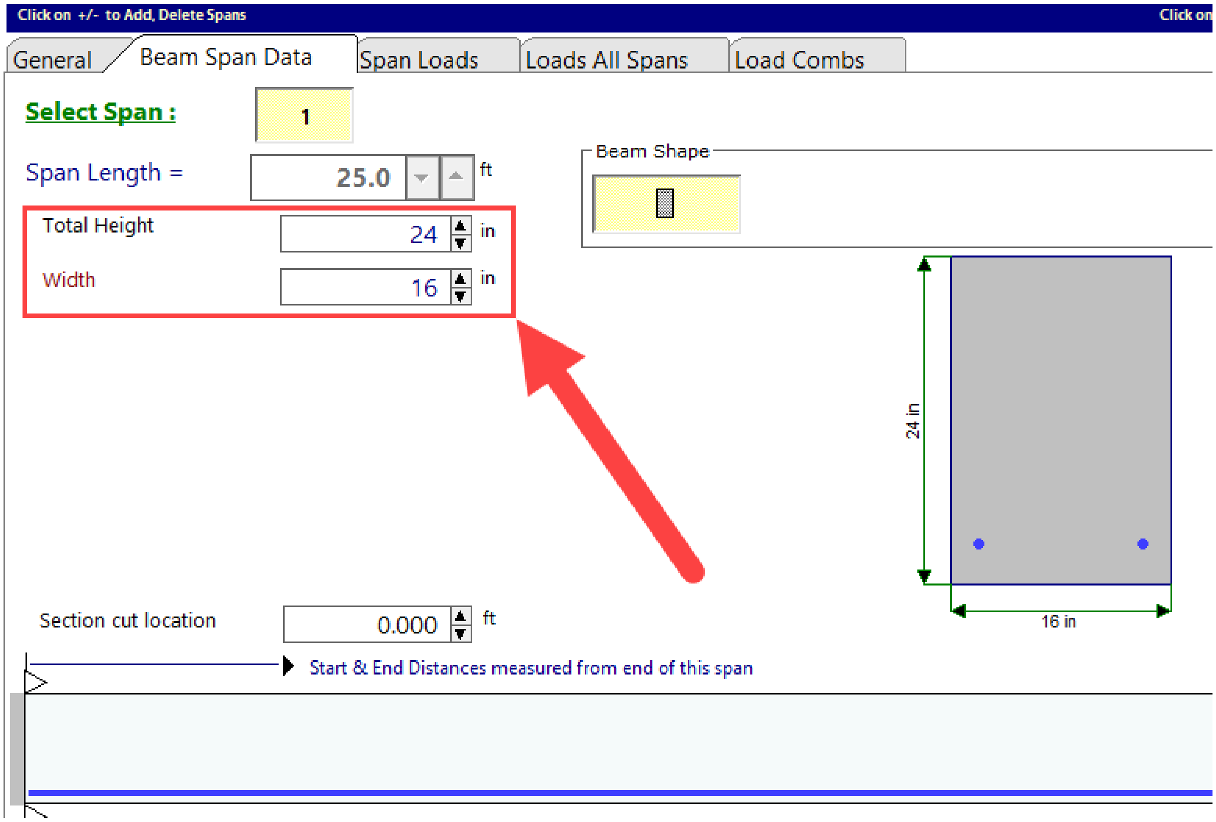


In cases where the rectangular beam was NOT joined to a slab at time of launch, the slab will have no influence over the design geometry of the beam. The design dimensions of the cross-section displayed in ENERCALC will exactly match the gross dimensions of the Revit beam element itself. The slab thickness and spatial relationship of the slab to the top of the concrete beam will not influence the design geometry displayed in the ENERCALC calculation. Only the dimensions of the physical beam will be used.

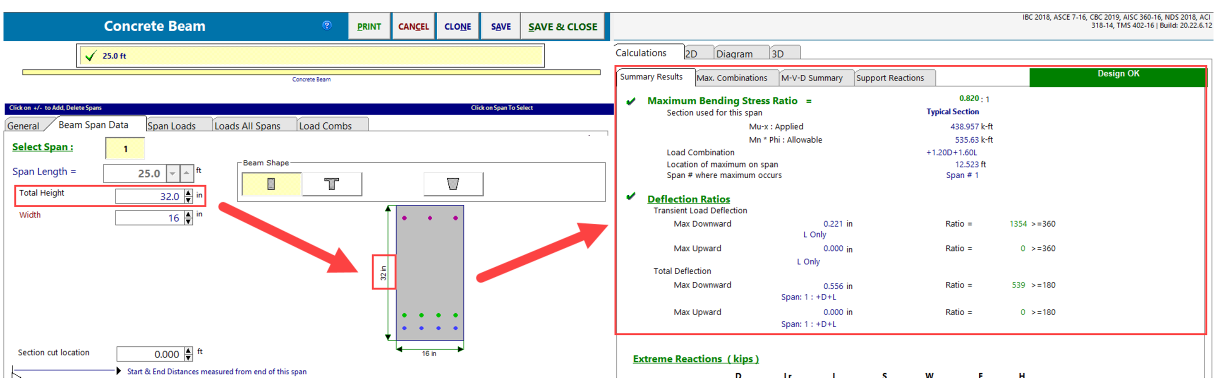

2' - 0"
1' - 4"



The dimension controls for the concrete beam section are not locked, and may be edited by the user in the ENERCALC interface:

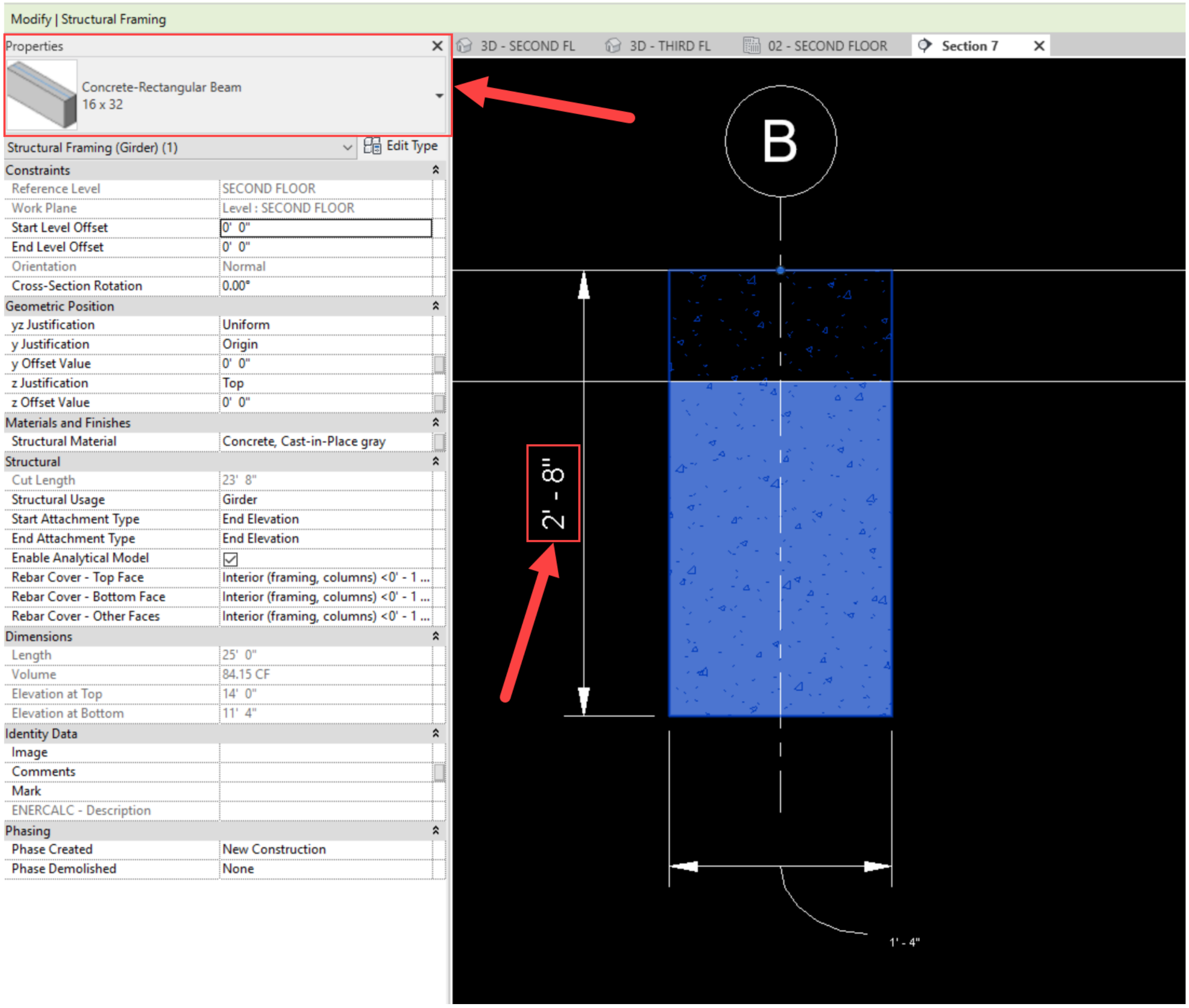


As with traditional ENERCALC Concrete Beam calculations, changing the section dimensions will cause both the beam section schematic and the concrete design code checks to update automatically in real-time.

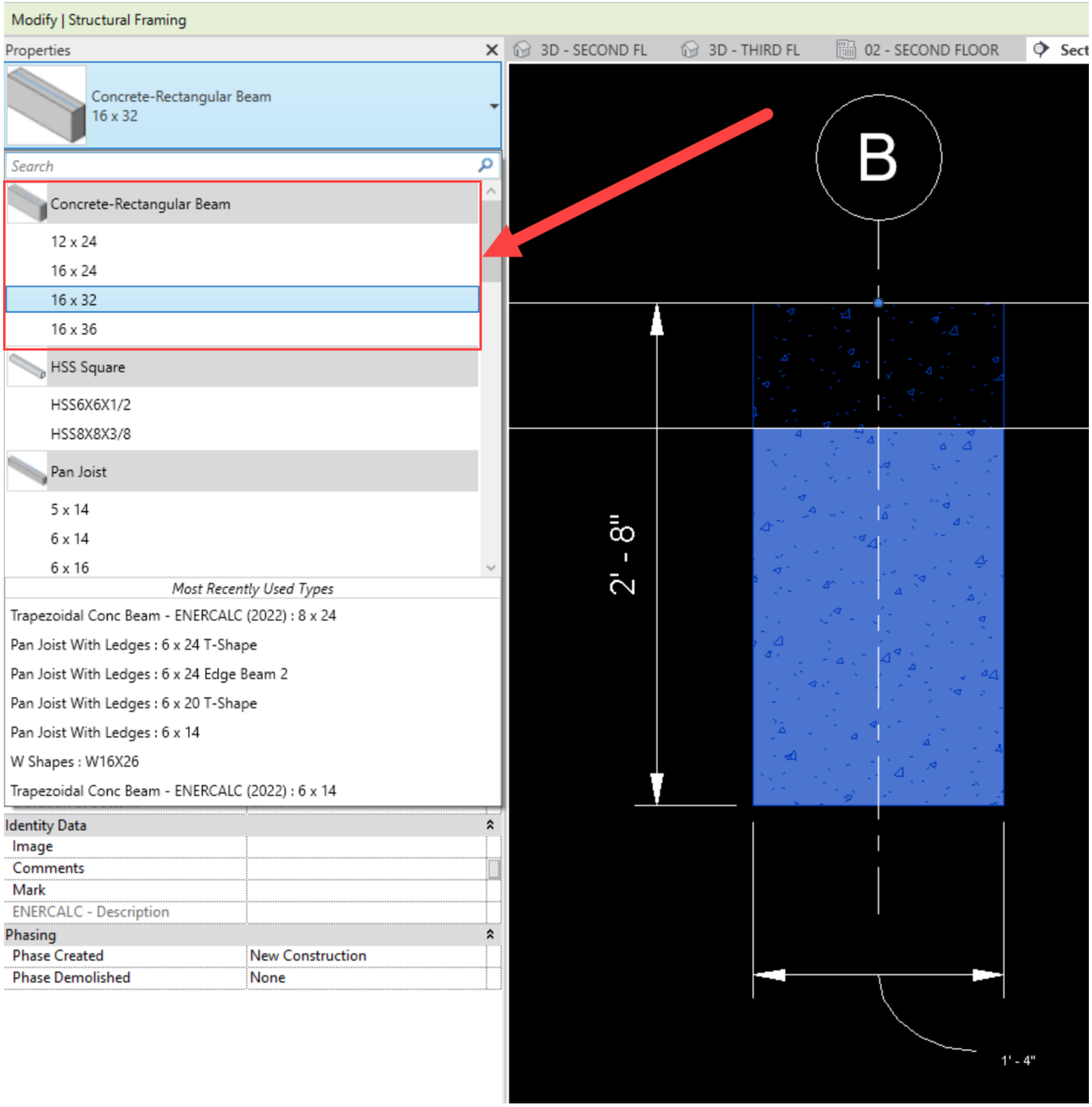


Once the design is satisfactory and the user clicks "Save and Close", the changes to the beam's design section size in ENCERCALC will be automatically applied to the physical beam in the Revit model.

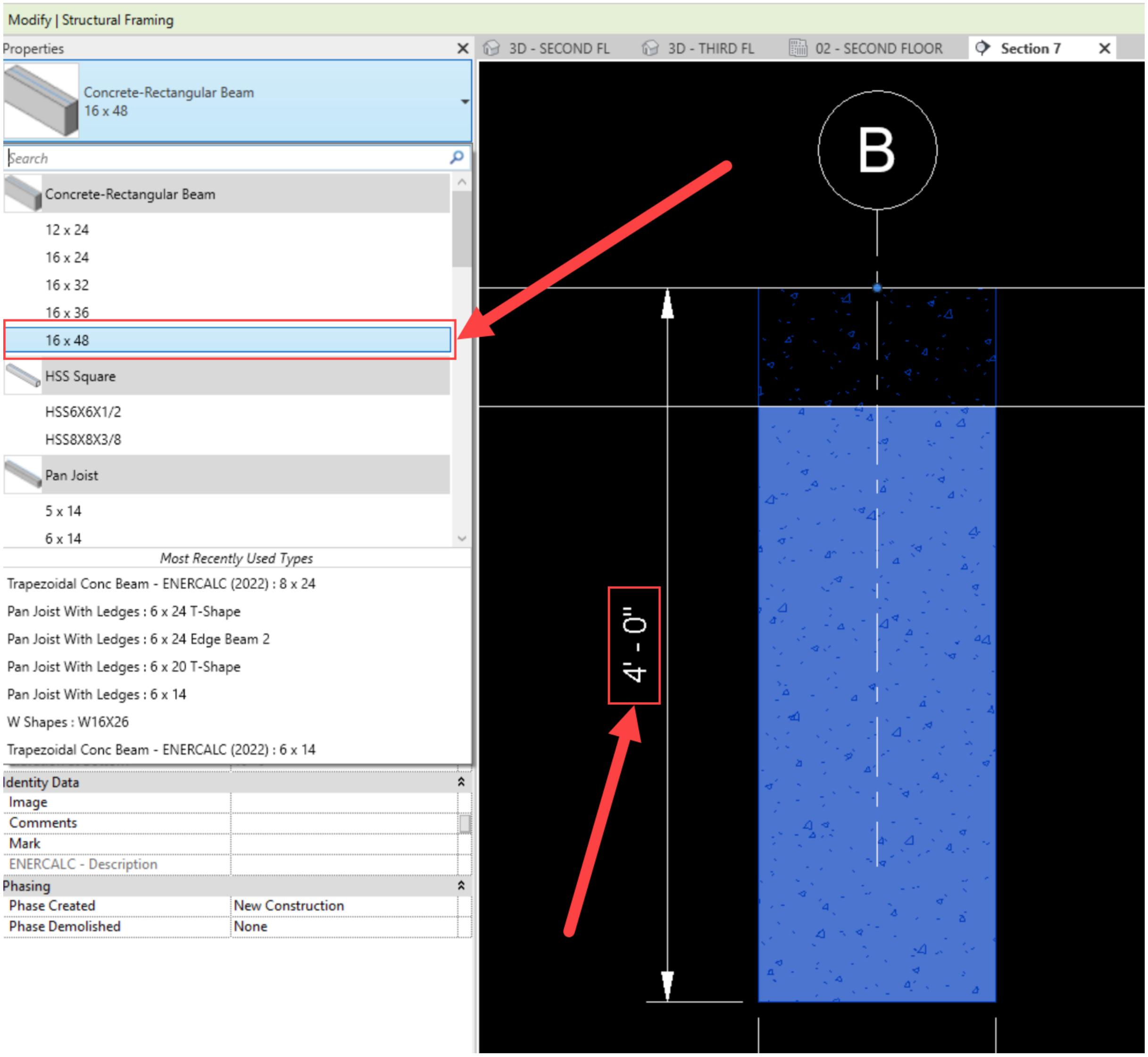
Note that the new dimensions applied to the Revit beam element are the EXACT dimensions specified in the ENERCALC calculation. This is because the un-joined slab is completely ignored when updating the size of the beam section.



If the rectangular beam family in the Revit model already contains a type with the required dimensions, then the beam instance is updated by automatically toggling to the appropriate type:



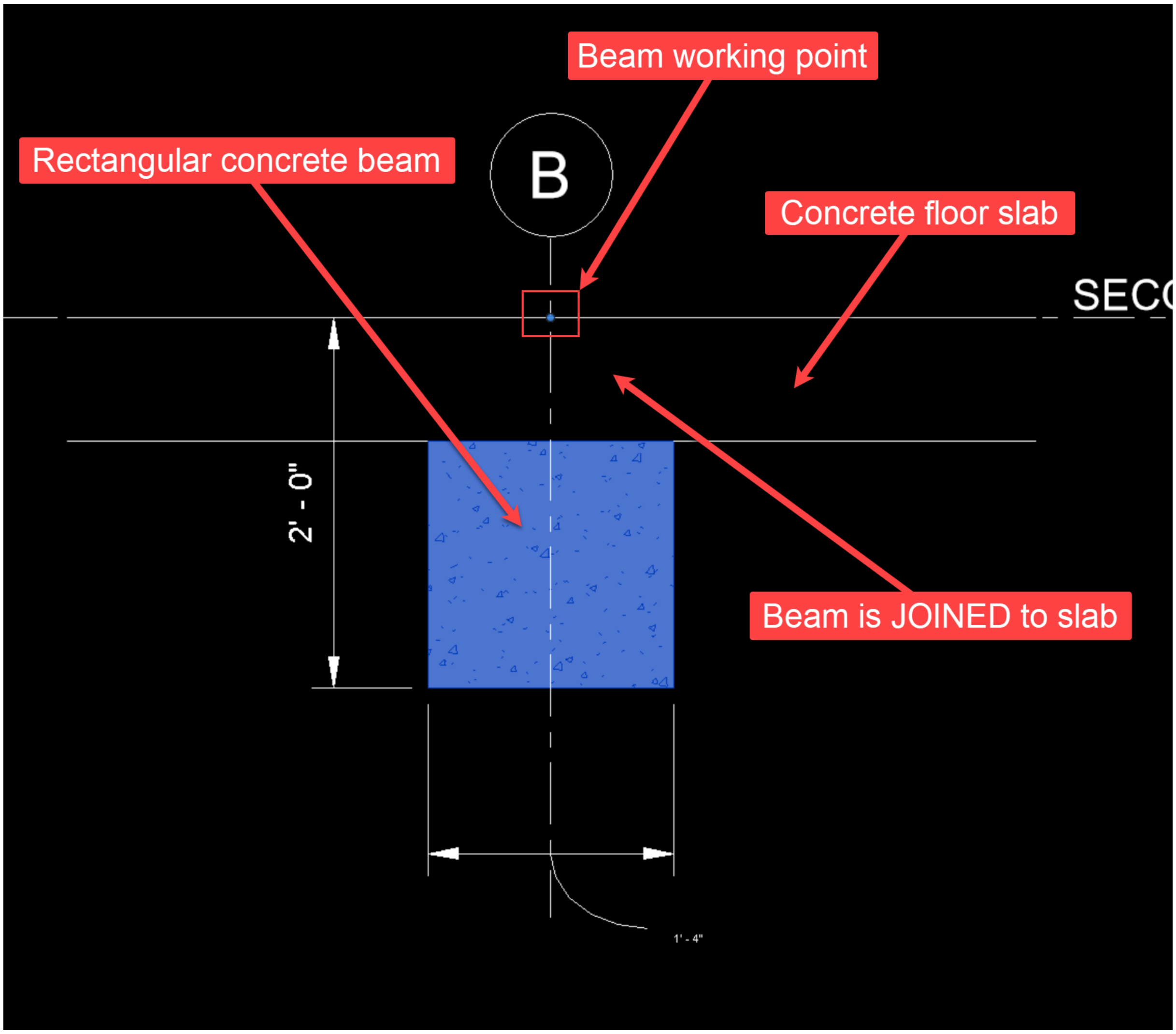
If there is no existing type with the appropriate dimensions, then a new type is automatically created in the family and applied to the physical beam instance:



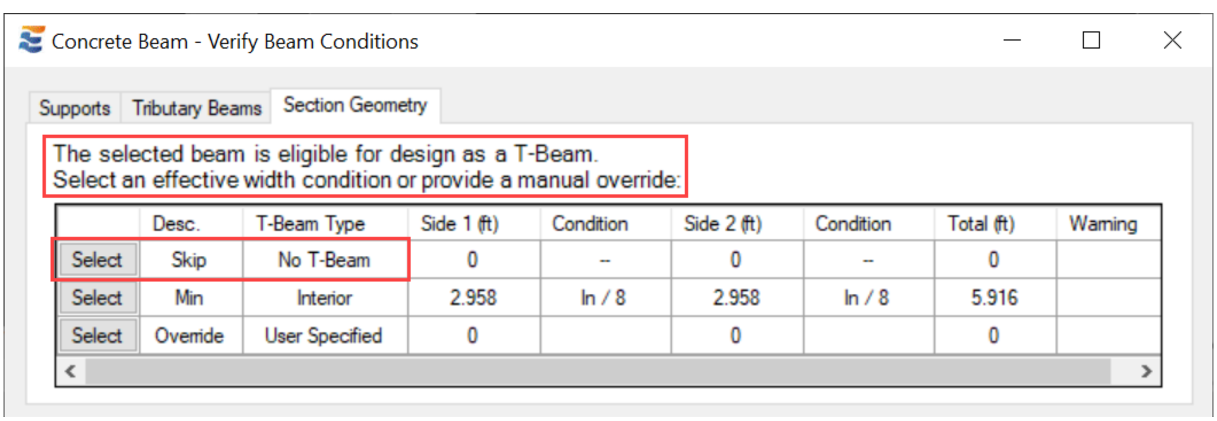
10.14.3 Rectangular Beam Sections - Joined

Revit provides the ability for users to control whether or not adjacent physical elements are joined to each other. When they are joined, their solid geometries will be combined, rather than having two distinct geometries which overlap in 3D space and 2D views. Since the design behavior differs in joined vs. unjoined conditions, ENERCALC for Revit users should remain aware of the join relationships between concrete beams and concrete floors in a Revit model.

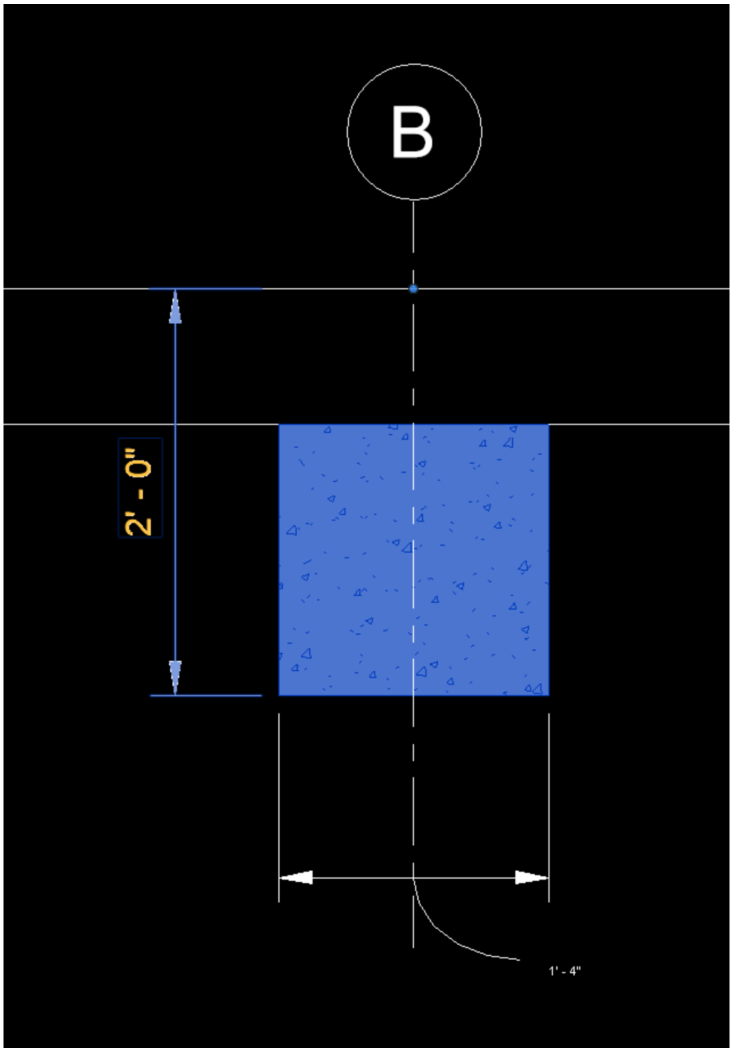
Instead of the approach described in the previous section (Rectangular Beam Sections - Unjoined 454), the user may alternatively choose to launch a rectangular beam calculation for a beam that IS JOINED to a slab in the Revit model.

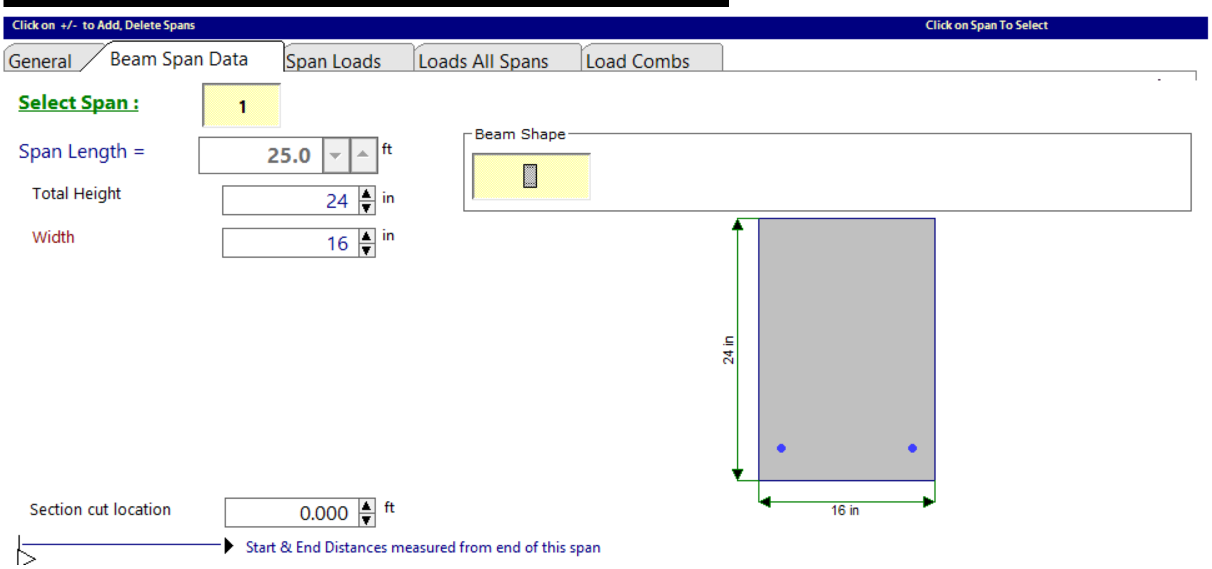


When the calculation launch process reaches the "Section Geometry" tab, the user will be informed that the beam IS eligible for design as a T-Beam. Using the "Select" button on the row marked "Skip" and "No T-Beam" will launch the calculation as a rectangular beam without any Tee geometry:

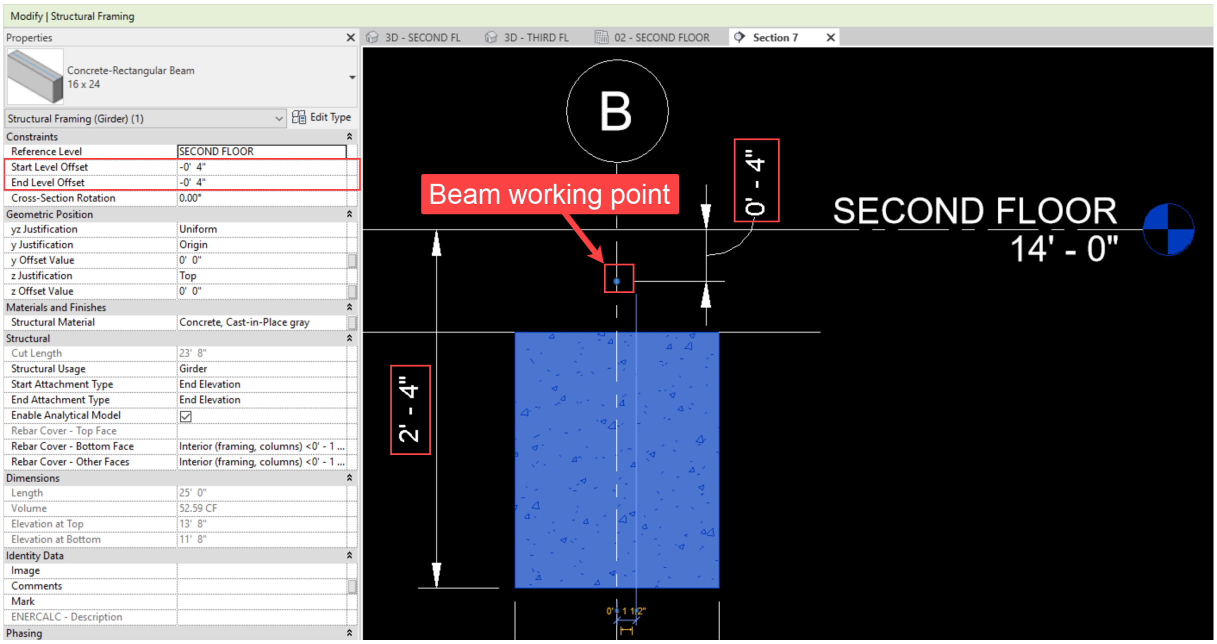


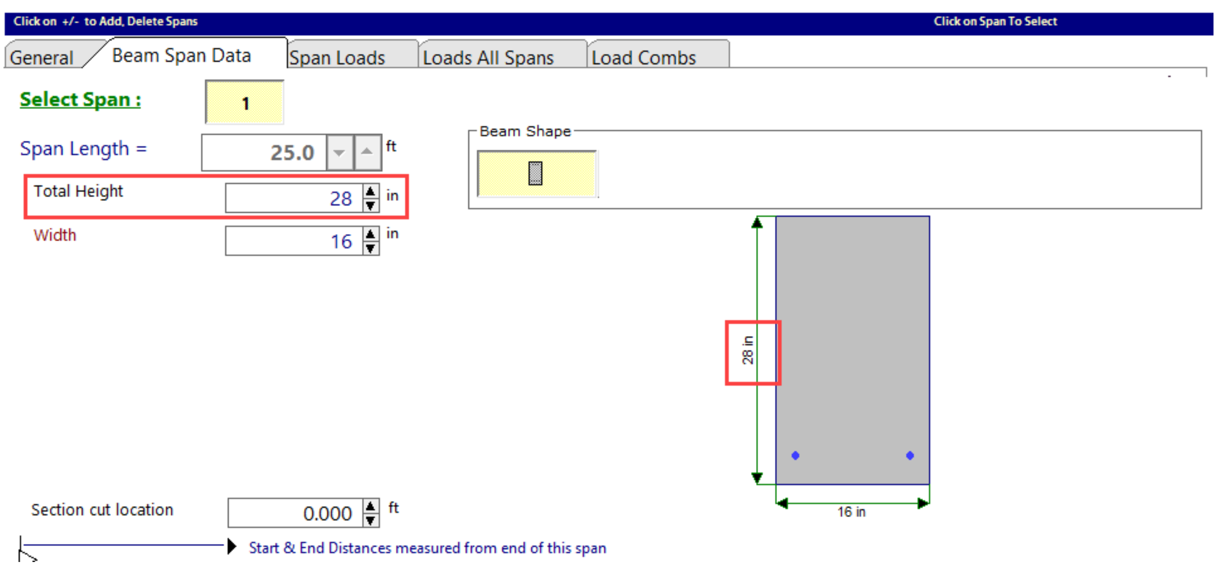
In doing so, the user will observe that the design calculation constructed in ENERCALC represents the gross combined depth of the beam AND the floor slab. No T-Beam flanges will be used.


B
2' - 0"
1' - 4"

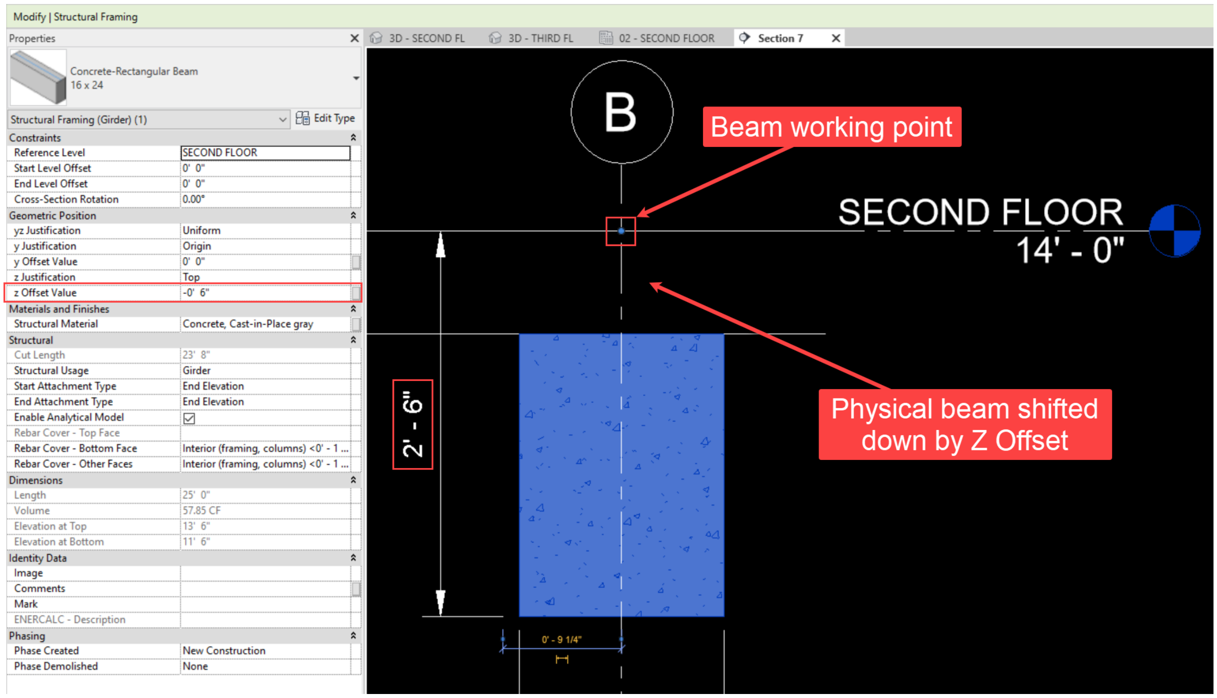

Click on +/- to Add, Delete Spans
Click on Span To Select
General
Beam Span Data
Span Loads
Loads All Spans
Load Combs
Select Span :
1
Span Length =
25.0
ft
Total Height
24
in
Width
16
in
Beam Shape
24 in
16 in
Section cut location
0.000
ft
Start & End Distances measured from end of this span

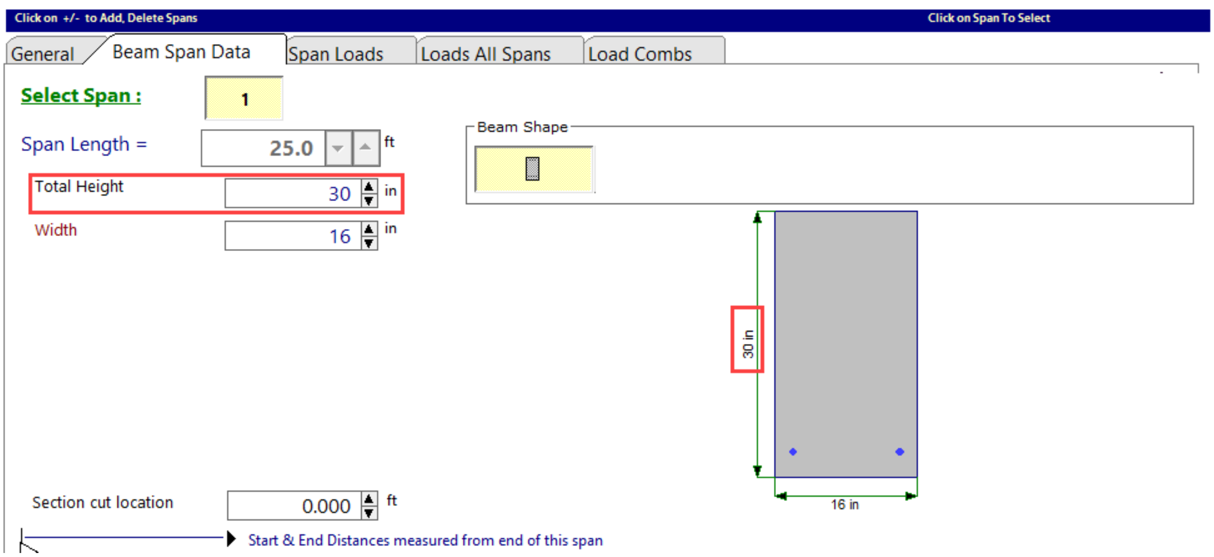
This relationship also holds true if the working point location of the beam is altered by physical movement or by adjustment of the "Start Level Offset" and "End Level Offset" parameters. Rather than matching the basic gross dimensions of the Revit beam element, the design dimensions displayed in ENERCALC will represent the combined geometry of the beam and the floor slab:





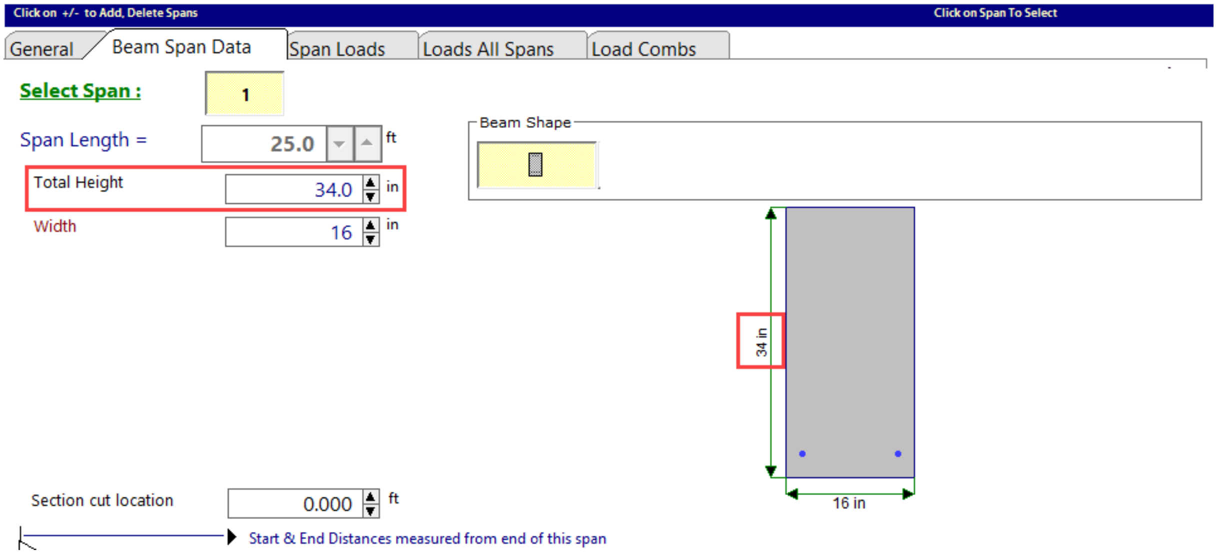
This relationship also holds true if the physical geometry of the beam is offset from its working point location by adjustment of the "Z Offset Value" parameter. Rather than matching the basic gross dimensions of the Revit beam element, the design dimensions displayed in ENERCALC will represent the combined geometry of the beam and the floor slab:

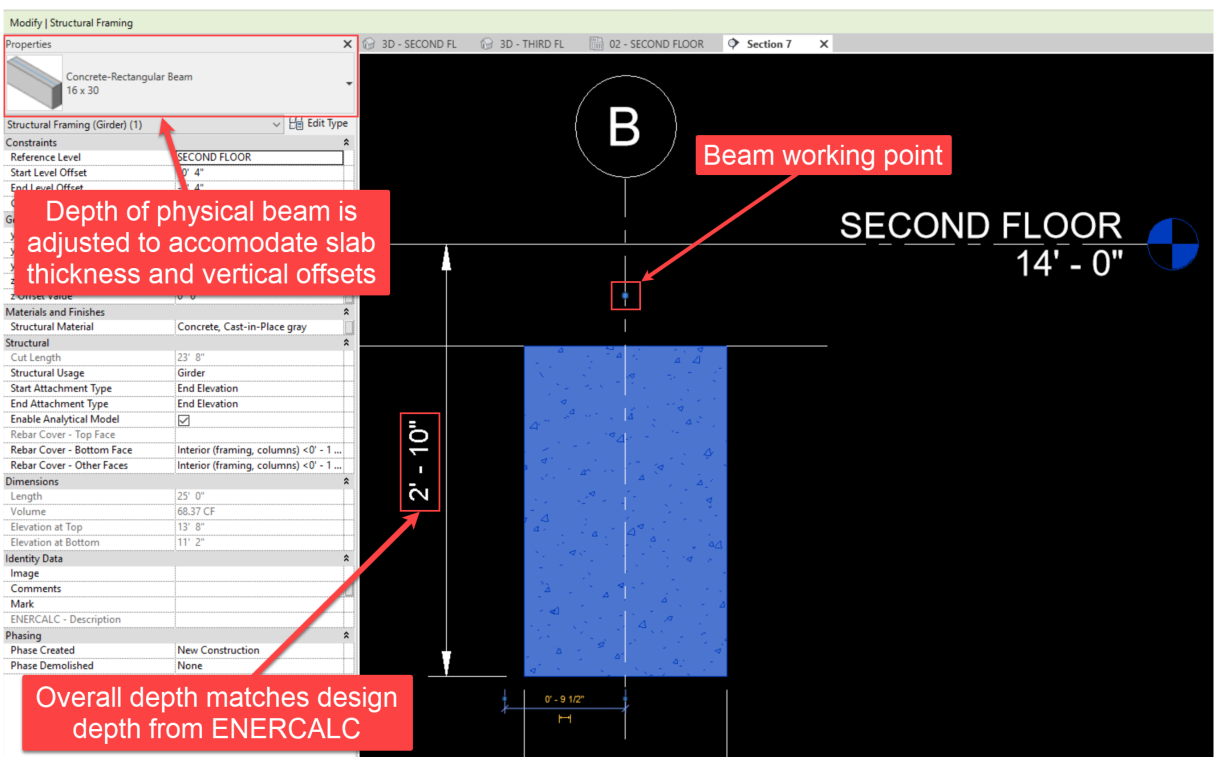




Once the design is satisfactory and the user clicks "Save and Close", the changes to the beam's design section size in ENERCALC will be automatically applied to the physical beam in the Revit model.

Note that the new dimensions applied to the Revit beam element are **NOT the EXACT** dimensions specified in the ENERCALC calculation. This is because the geometry of the joined slab is automatically considered when updating the size of the beam section. Instead, ENERCALC for Revit will assign a section geometry whose location and gross dimensions will combine with the thickness of the joined slab to create the overall geometry specified in the ENERCALC calculation:

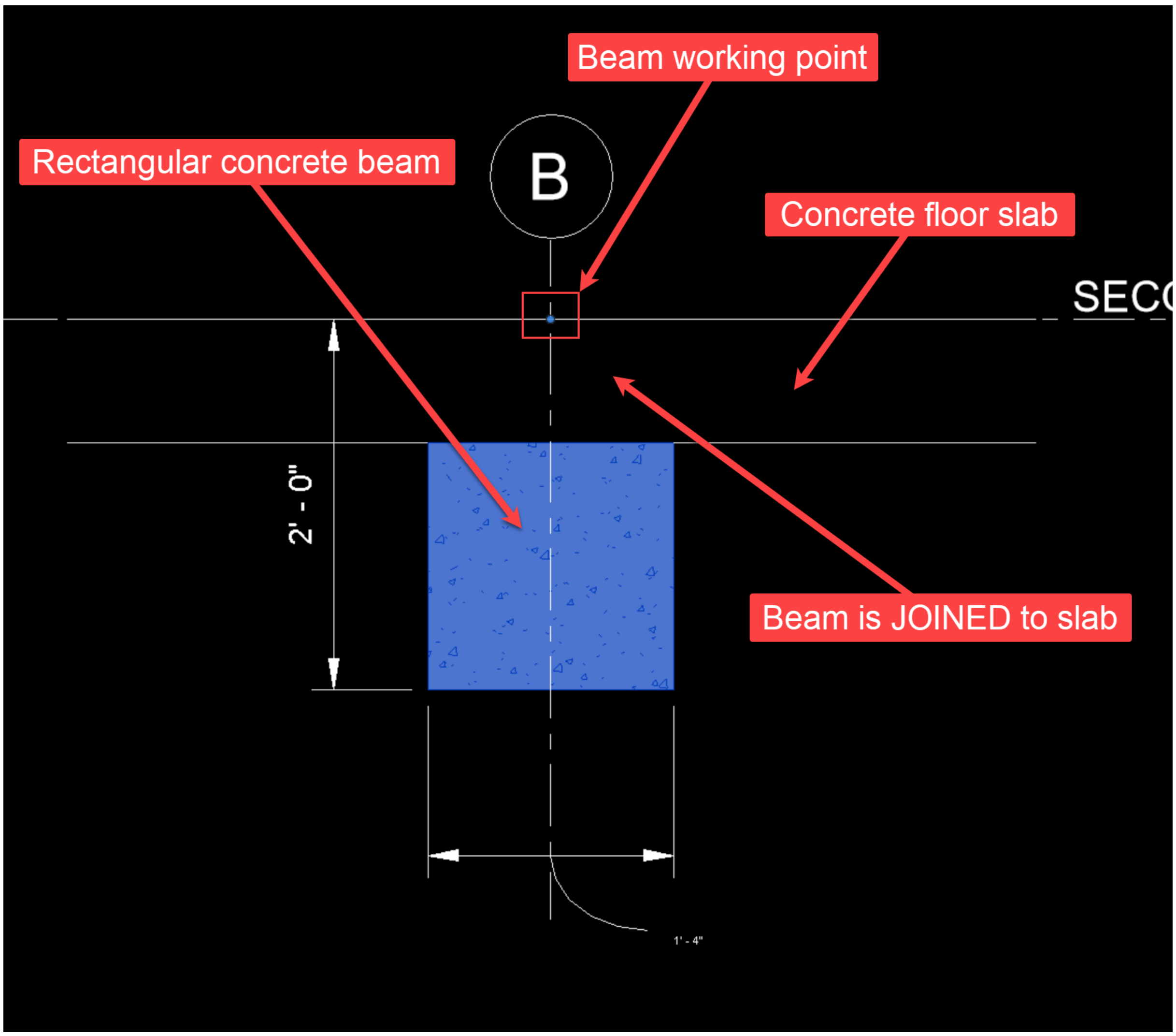




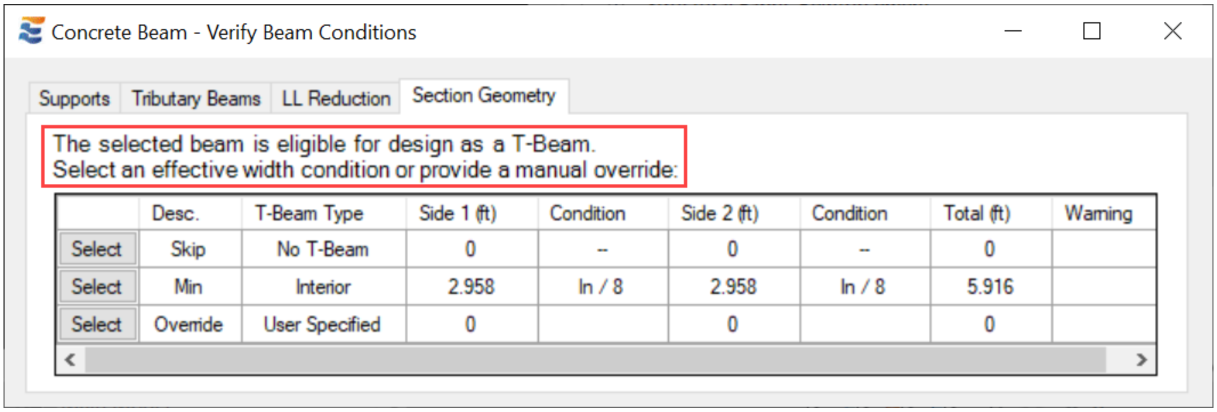
10.14.4 T-Beam Sections

Concrete beam calculations may be launched with T-Beam geometry IF AND ONLY IF the beam and slab have been joined to each other. If the beam and slab are not joined, the launched process will behave as described in Rectangular Beam Sections - Unjoined 454.

An example of a concrete beam joined to a slab was previously illustrated in Rectangular Beam Sections - Joined 461:



When the calculation launch process reaches the "Section Geometry" tab, the user will be informed that the beam IS eligible for design as a T-Beam. Using the "Select" button on the row marked "Skip" and "No T-Beam" will launch the calculation as a rectangular beam without any Tee geometry. Clicking the "Select" button of any other row in the table will launch the concrete beam calculation with the T-Beam geometry shown in that row.



As discussed in Rectangular Beam Sections - Joined 461, the table always includes an option to skip T-Beam behavior. In addition, the table also displays any T-Beam section geometry detected from the model, as well as an option for the user to manually override the effective width conditions. The "Side 1" and "Side 2" conditions displayed in the table are computed from physical conditions in the Revit model using the ACI 318 criteria for concrete T-Beam effective width. In ACI 318-14 and -19, these provisions are displayed in Table 6.3.2.1 "Dimensional limits for effective overhanging flange width for T-beams".

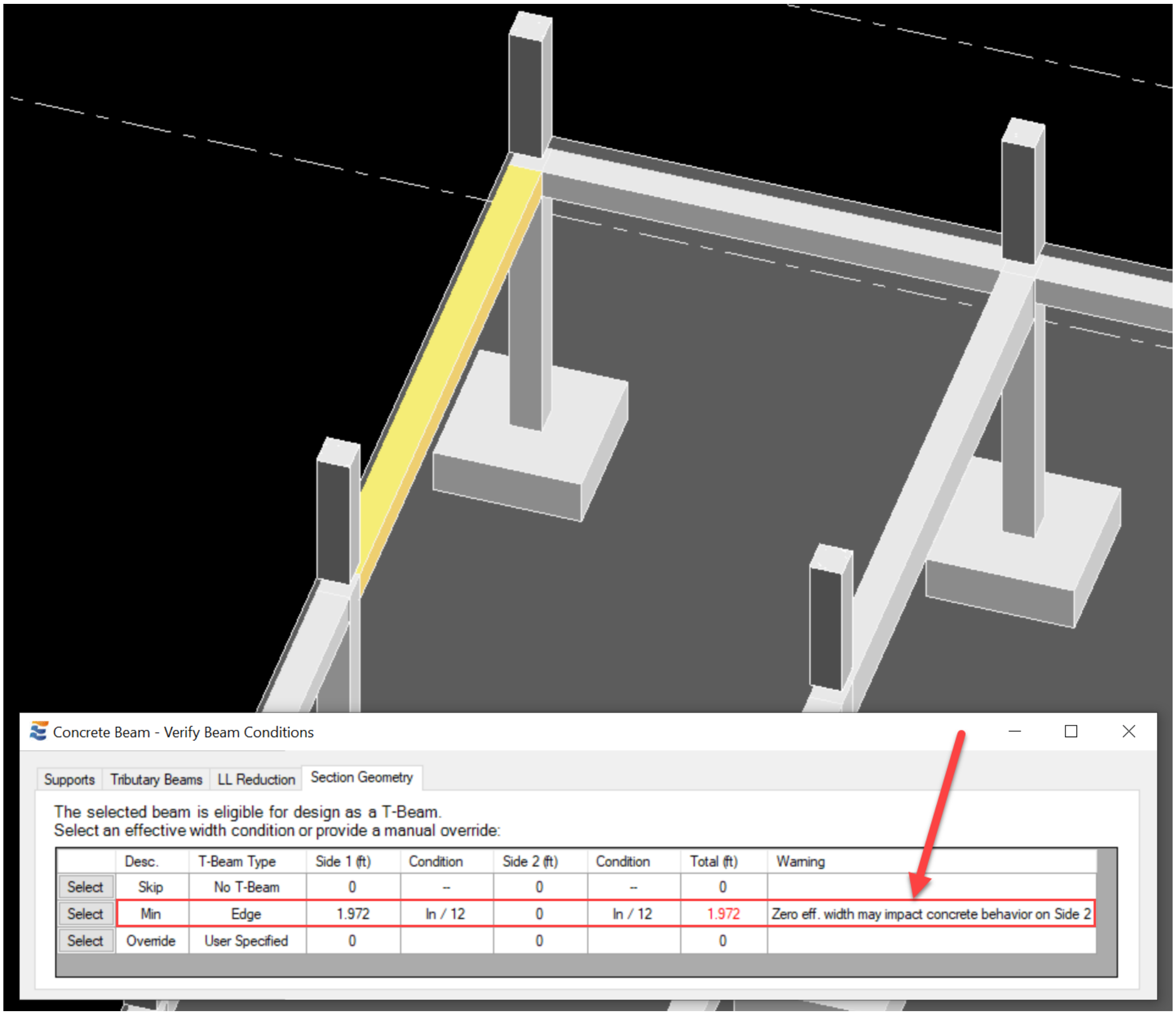
In the case of an interior (non edge) beam, the effective width on EACH SIDE is taken as the least of:

- 8h 8 times the slab thickness (h)
- sw / 2 Half the clear distance (sw) to the nearest adjacent beam web
- ln / 8 One-eighth of the current clear span (ln)

In the case of an edge beam, the effective width on ONE SIDE is taken as the least of:

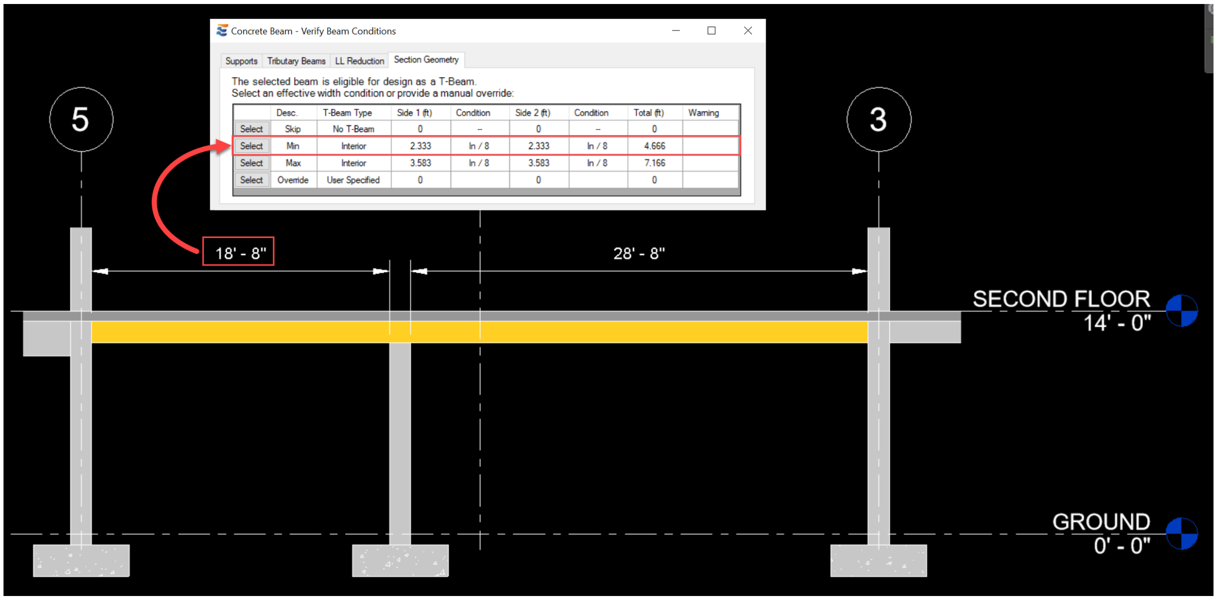
- 6h 6 times the slab thickness (h)
- sw / 2 Half the clear distance (sw) to the nearest adjacent beam web
- ln / 12 One-twelfth of the current clear span (n)

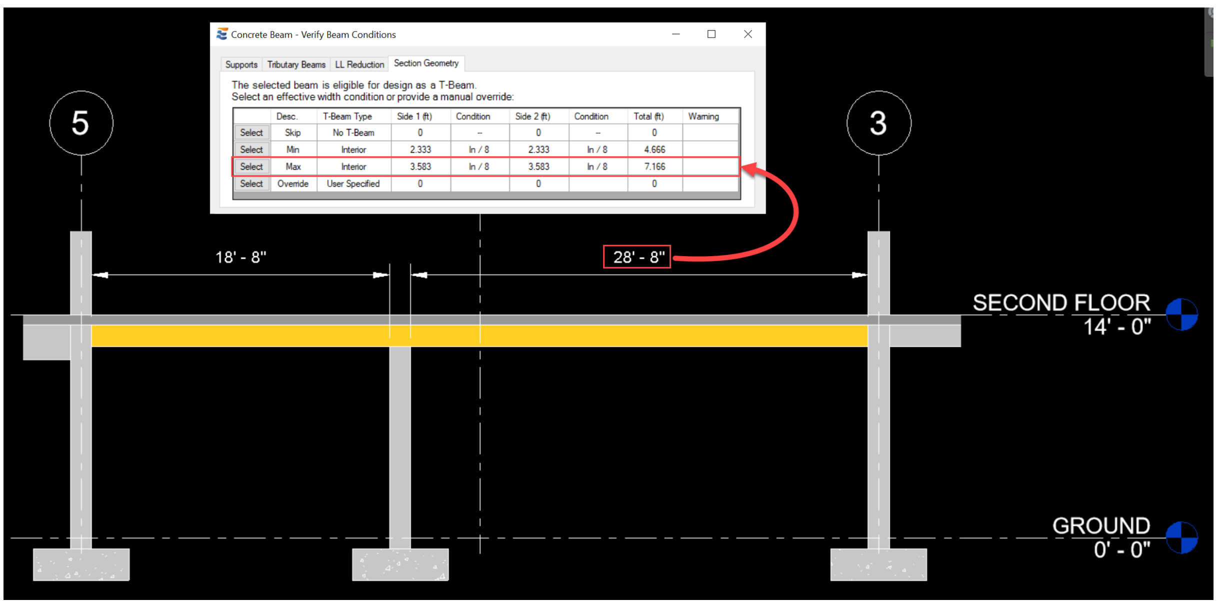
ENERCALC for Revit checks each of these criteria automatically, and the criterion found to control effective width for each respective side is indicated in the "Condition" column. This includes detecting whether the beam falls at an edge condition, and invoking the applicable ACI criteria accordingly:



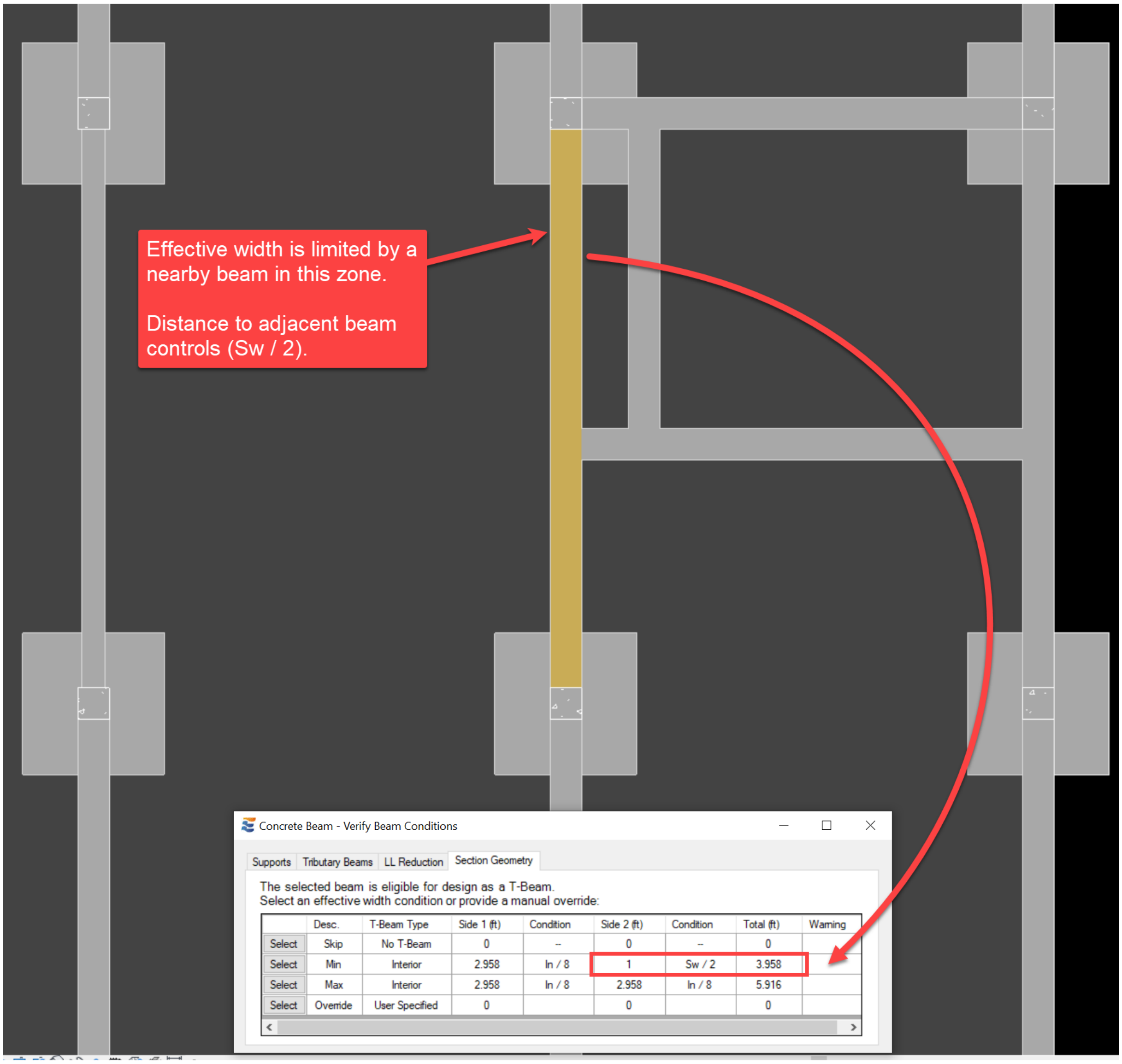
When varying conditions are discovered, the user will be presented with multiple geometry conditions to choose from. The user must assign a single controlling section geometry which will apply to all spans (special note #6 in Concrete Beam Sections) 449.

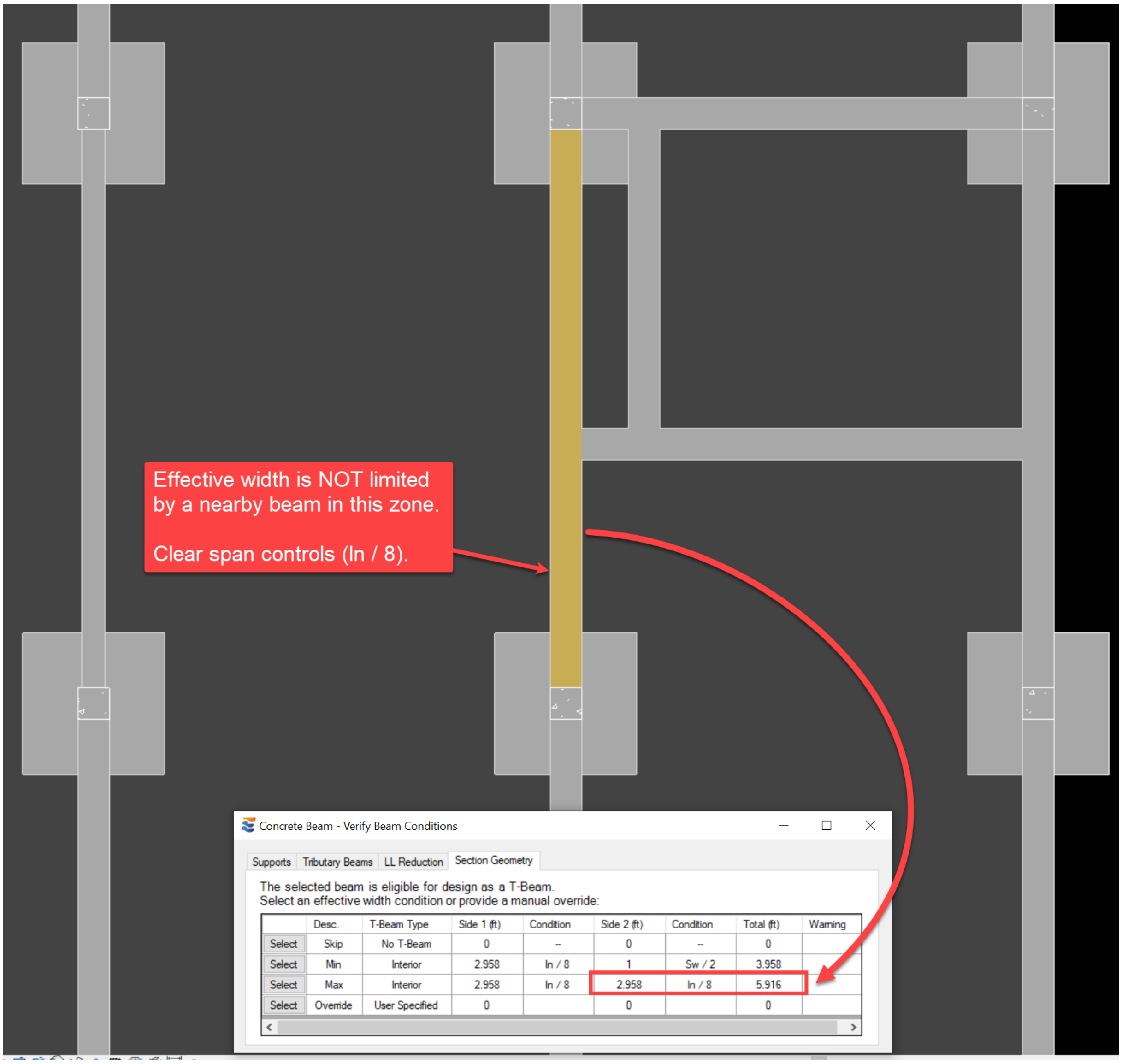
For example, when a multi-span beam has differing clear spans, the table displays a T-Beam geometry line item for each condition found:



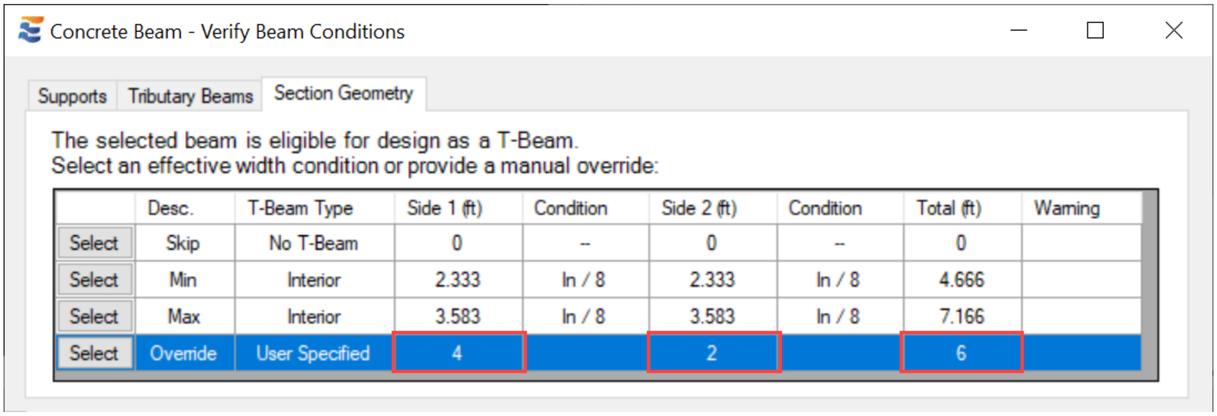


Similarly, when a beam has varying adjacency conditions, different geometry options will be presented to the user accordingly. Of the conditions automatically discovered by EFR, only one may be selected to control the design. The user should select whichever conditions is deemed to be the most appropriate limitation for a safe design calculation.

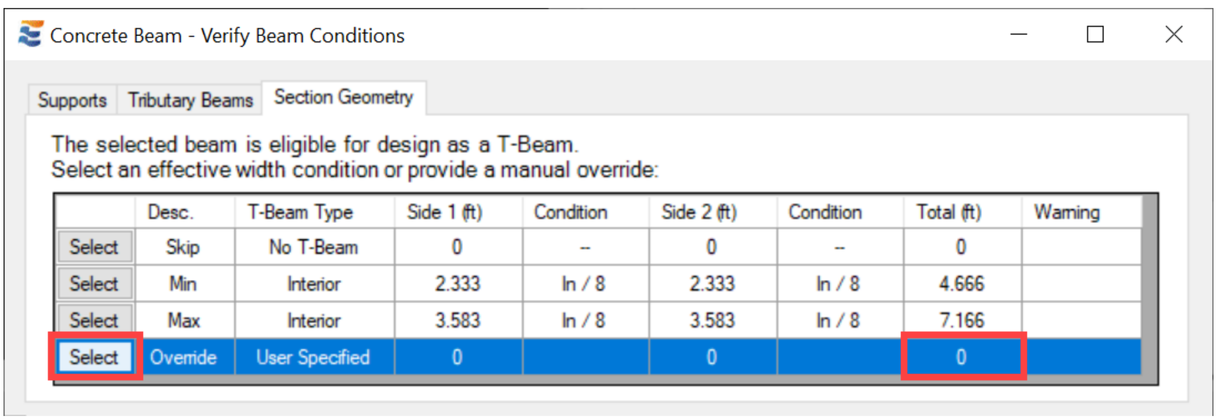

Effective width is limited by a nearby beam in this zone.
Distance to adjacent beam controls (Sw / 2).
Concrete Beam - Verify Beam Conditions
Supports
Tributary Beams
LL Reduction
Section Geometry
The selected beam is eligible for design as a T-Beam.
Select an effective width condition or provide a manual override:
Desc. | T-Beam Type | Side 1 (ft) | Condition | Side 2 (ft) | Condition | Total (ft) | Warning
Select | Skip | No T-Beam | 0 | -- | 0 | -- | 0
Select | Min | Interior | 2.958 | ln / 8 | 1 | Sw / 2 | 3.958
Select | Max | Interior | 2.958 | ln / 8 | 2.958 | ln / 8 | 5.916
Select | Override | User Specified | 0 | | 0 | | 0



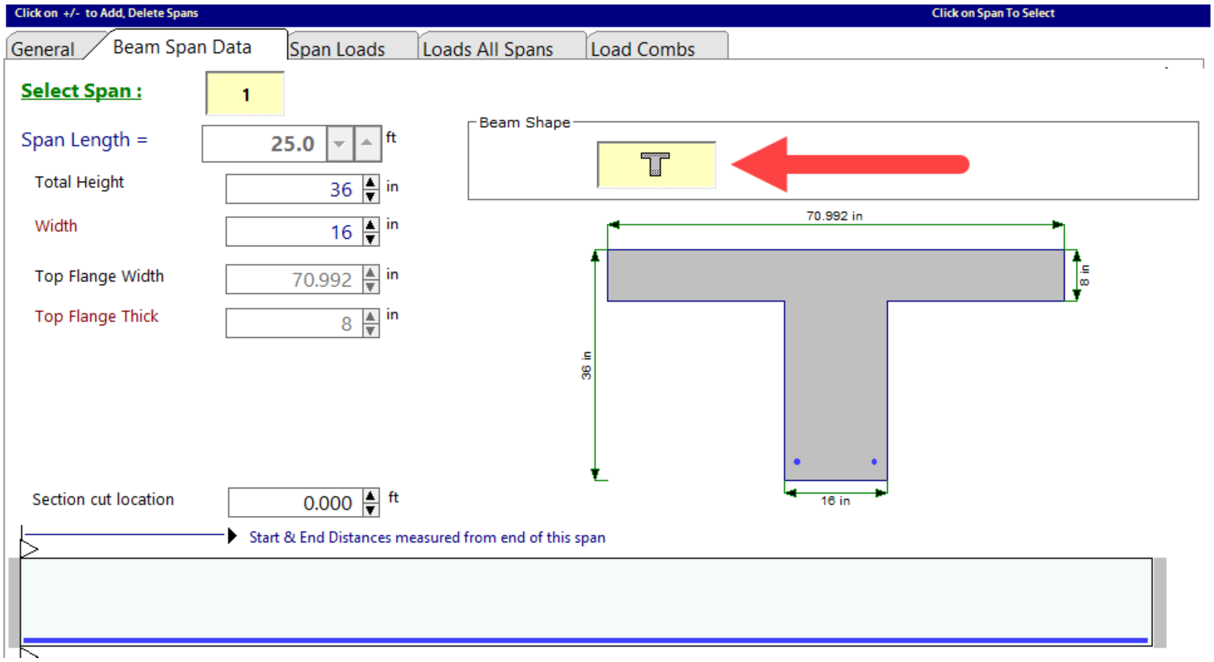
In addition to the auto-detected geometry options, the user also has the option to specify an arbitrary custom T-Beam effective width geometry using the table row marked "Override". Override geometry is set by manually editing the "Side 1" and "Side 2" cells to the desired effective widths. The "Total" cell value will update automatically as each side is modified:



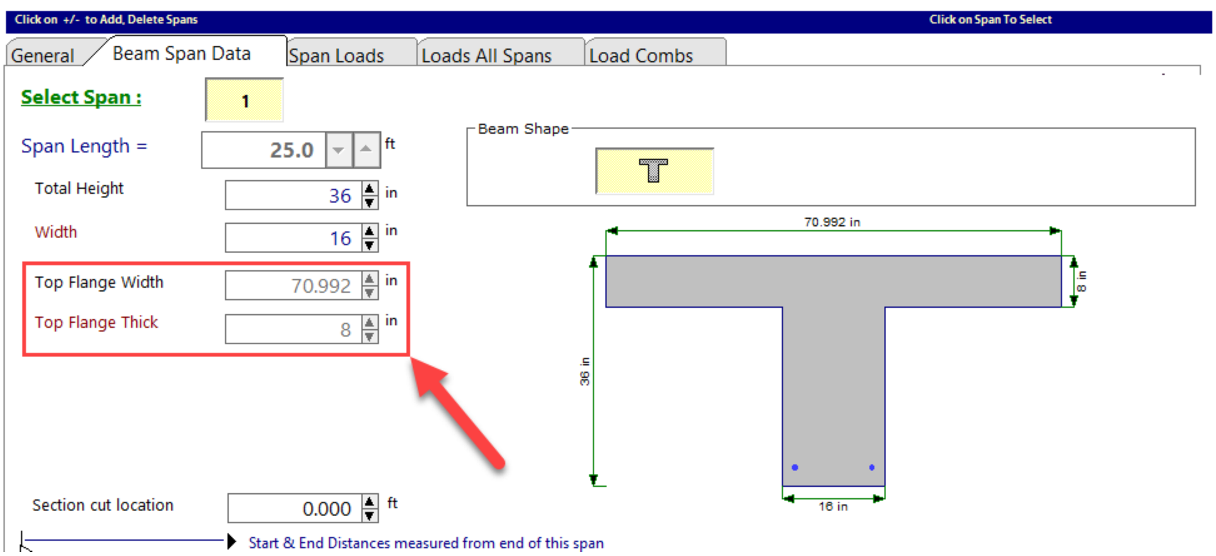
Note that if the total effective width cell value is zero, then clicking the "Select" button on the "Override" table row will NOT cause the calculation to launch. This button will not respond until one or both side width override values are set to non-zero amounts:



When a concrete beam calculation loads in the ENERCALC interface, the user will not find options to toggle the beam section type (unlike conventional non-Revit-linked calculations). If a concrete beam is sent to ENERCALC as a T Beam, then the calculation will load with only this option displayed:



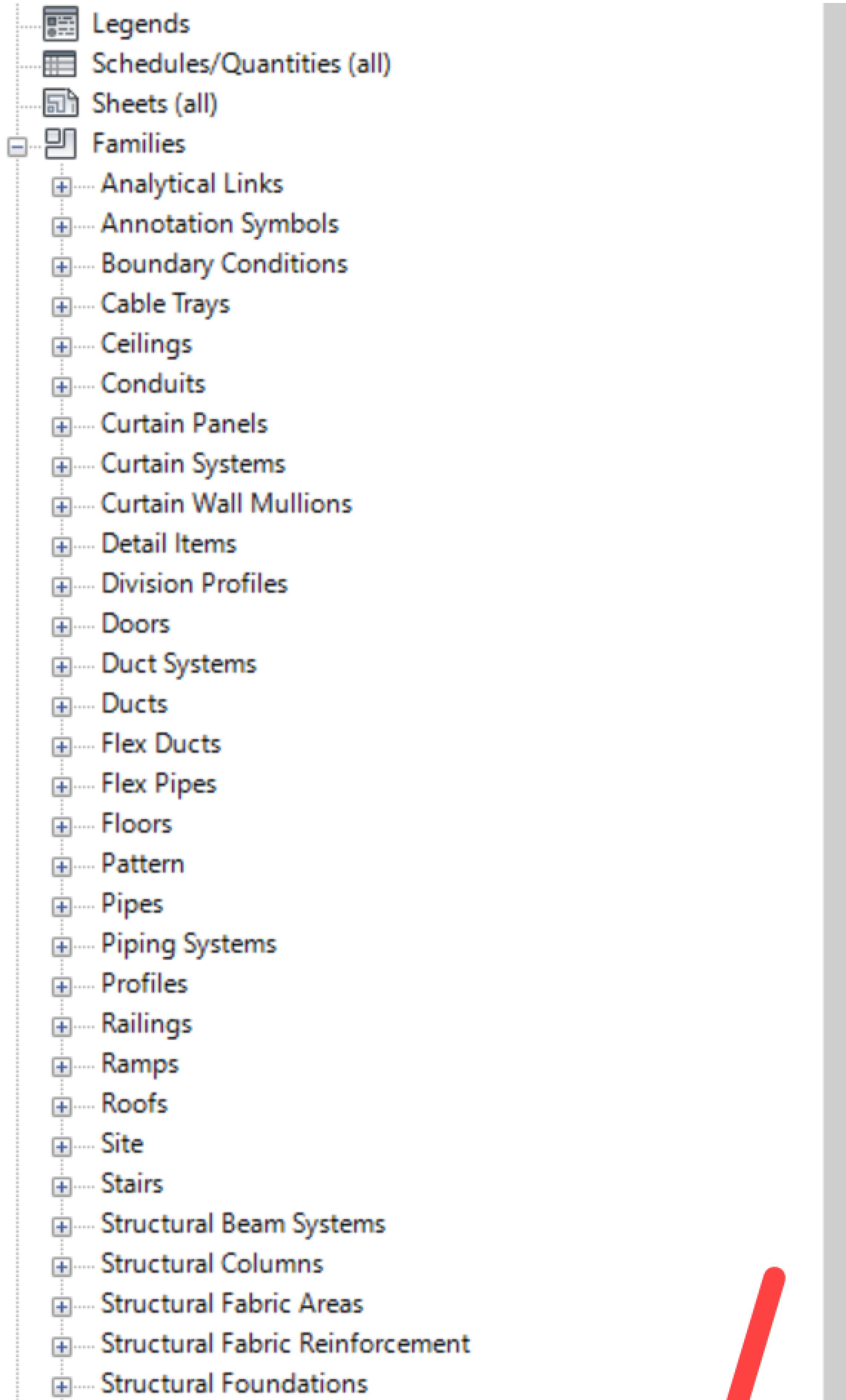
Users will also note that the "Top Flange Width" and "Top Flange Thick" input values that define T-Beam geometry are NOT available for modification in the ENERCALC interface. These values are driven exclusively by the physical geometry of the Revit model, or by user overrides applied during calculation launch:

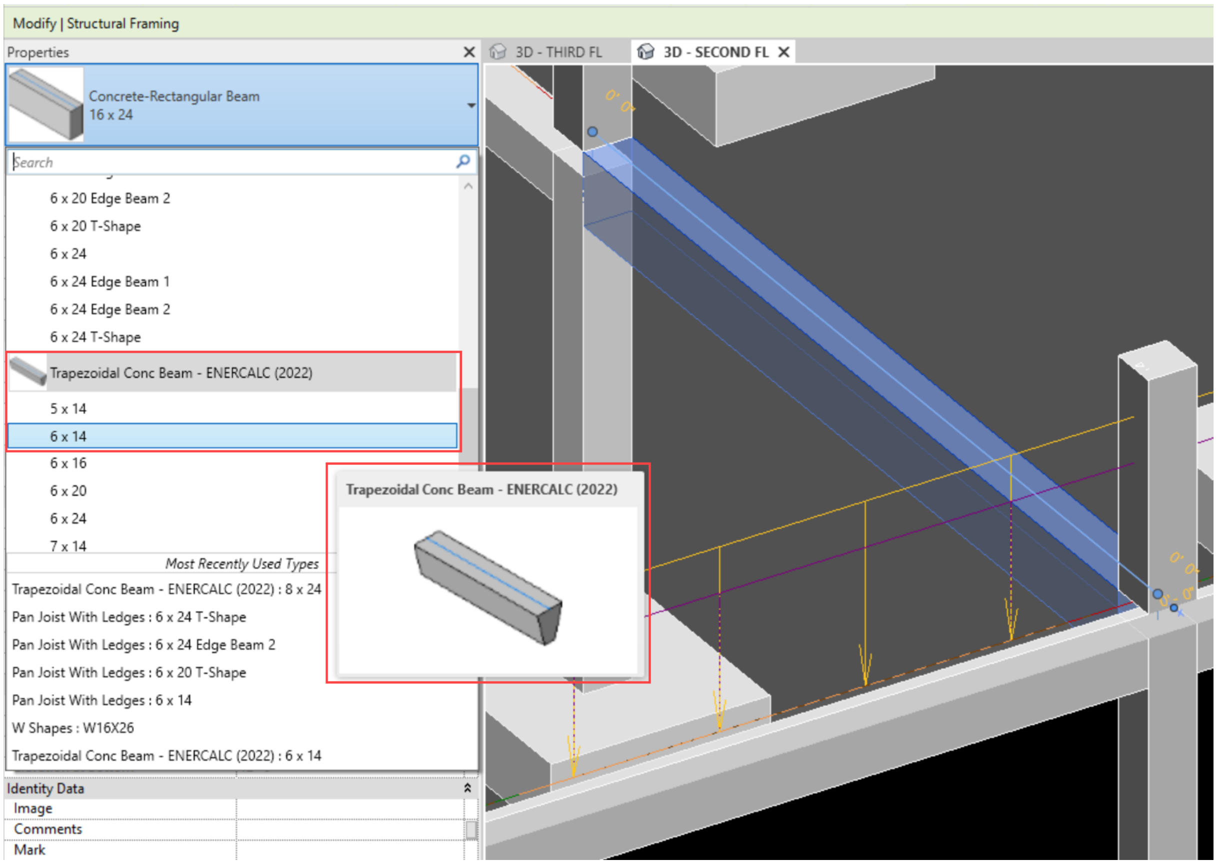


The value displayed for "Total Height" of the beam uses the same behavior described in Rectangular Beam Sections - Joined 461. The total height assigned to the T-Beam will represent the combined effect of 1.) slab thickness, 2.) beam depth, and 3.) slab and beam relative locations (i.e., vertical offsets). Similarly, when the calculation is saved, the Revit model will be updated with a new beam section geometry which creates the proper overall design depth in conjunction with the slab. As a result, the gross dimensions of the beam element itself may not necessarily match the design depth shown in ENERCALC, but the overall combined depth will match.

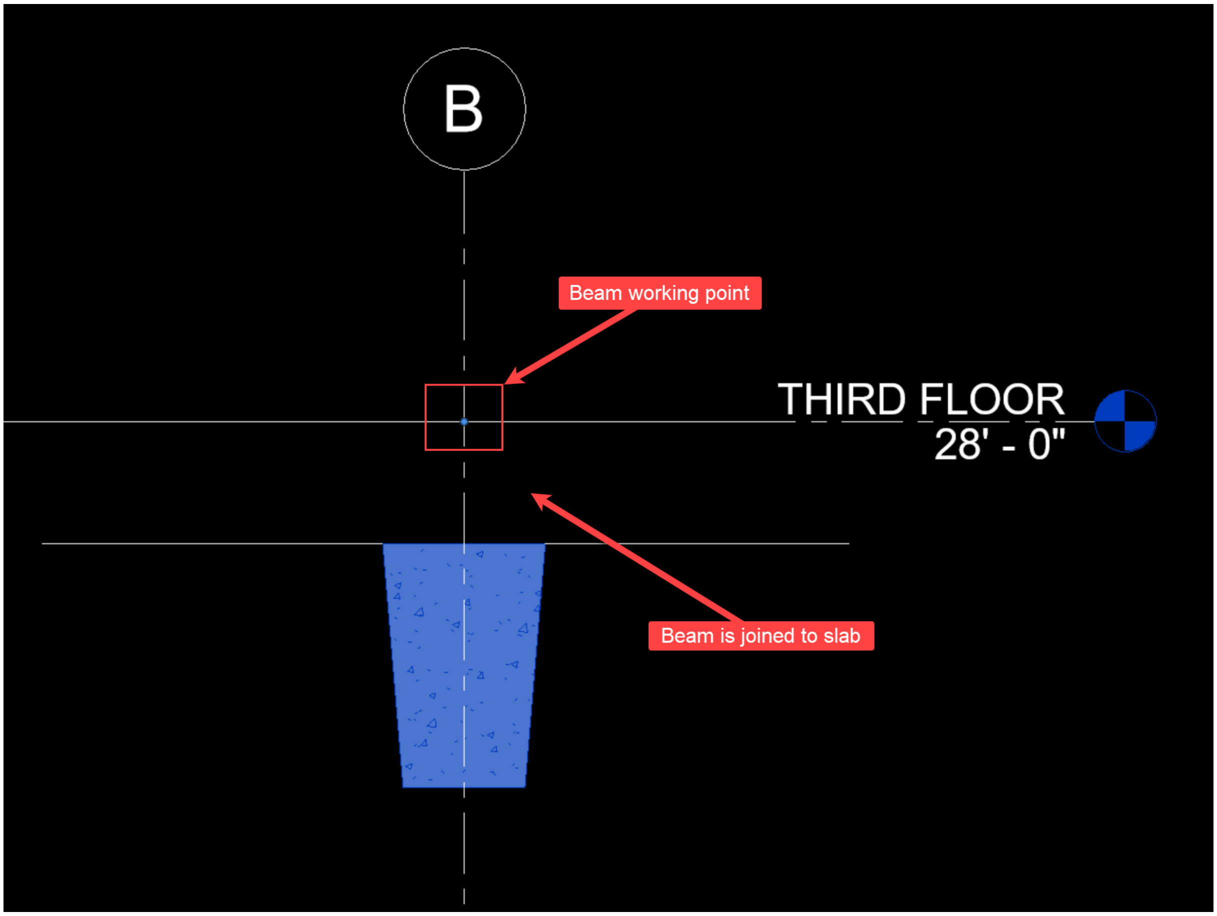
10.14.5 Trapezoidal Beam Sections

Because the default out-of-box family content provided by Revit does not include a trapezoidal beam family adequate for ENERCALC for Revit data exchange, ENERCALC for Revit will automatically load a version-appropriate trapezoidal beam family when a new Revit project (.rvt file) is opened. This beam family is visible in the Revit "Project Browser" and may be manually applied to any beam instance using the "Properties" pane:

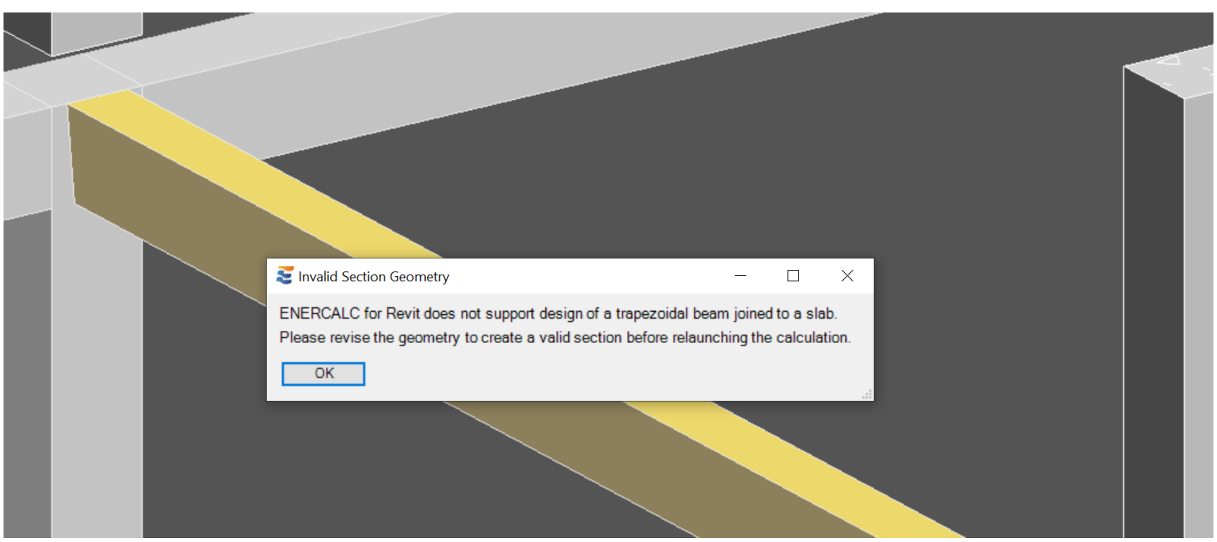

Legends
Schedules/Quantities (all)
Sheets (all)
Families
Analytical Links
Annotation Symbols
Boundary Conditions
Cable Trays
Ceilings
Conduits
Curtain Panels
Curtain Systems
Curtain Wall Mullions
Detail Items
Division Profiles
Doors
Duct Systems
Ducts
Flex Ducts
Flex Pipes
Floors
Pattern
Pipes
Piping Systems
Profiles
Railings
Ramps
Roofs
Site
Stairs
Structural Beam Systems
Structural Columns
Structural Fabric Areas
Structural Fabric Reinforcement
Structural Foundations



ENERCALC for Revit supports the design of trapezoidal concrete beams, but does not permit a concrete beam to be joined to the concrete slab:

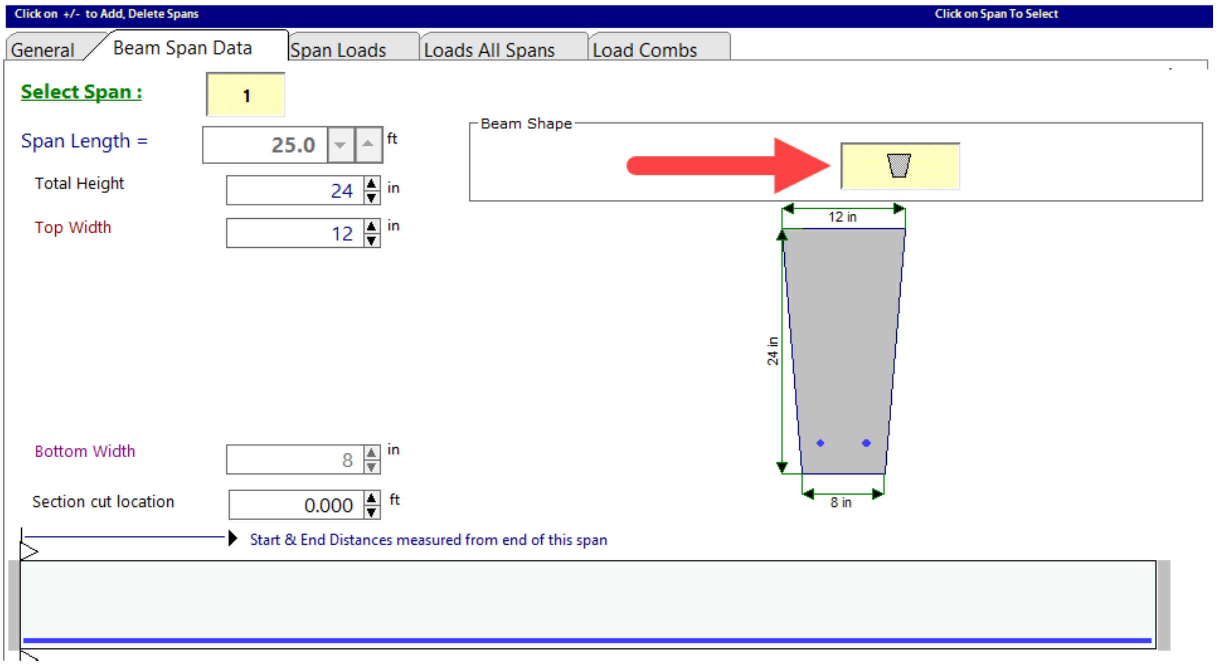


Since there is no design counterpart in ENERCALC for a trapezoidal T-Beam or a trapezoidal beam connected to a slab, this configuration will cause the calculation launch to halt with a warning:



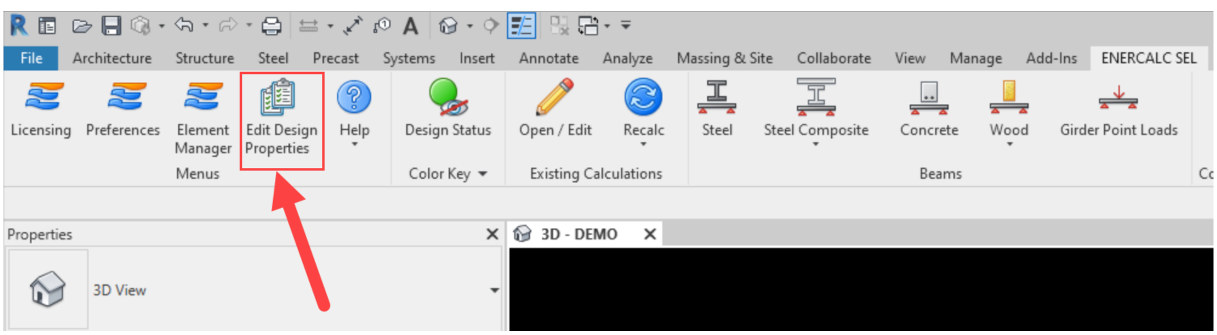
When a concrete beam calculation loads in the ENERCALC interface, the user will not find options to toggle the beam section type (unlike conventional non-Revit-linked calculations).

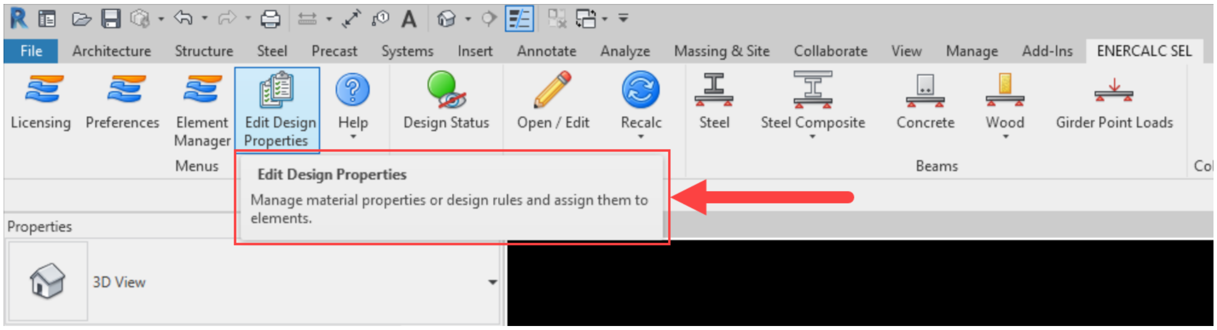
If a concrete beam is sent to ENERCALC as a trapezoidal beam, then the calculation will load with only this option displayed:



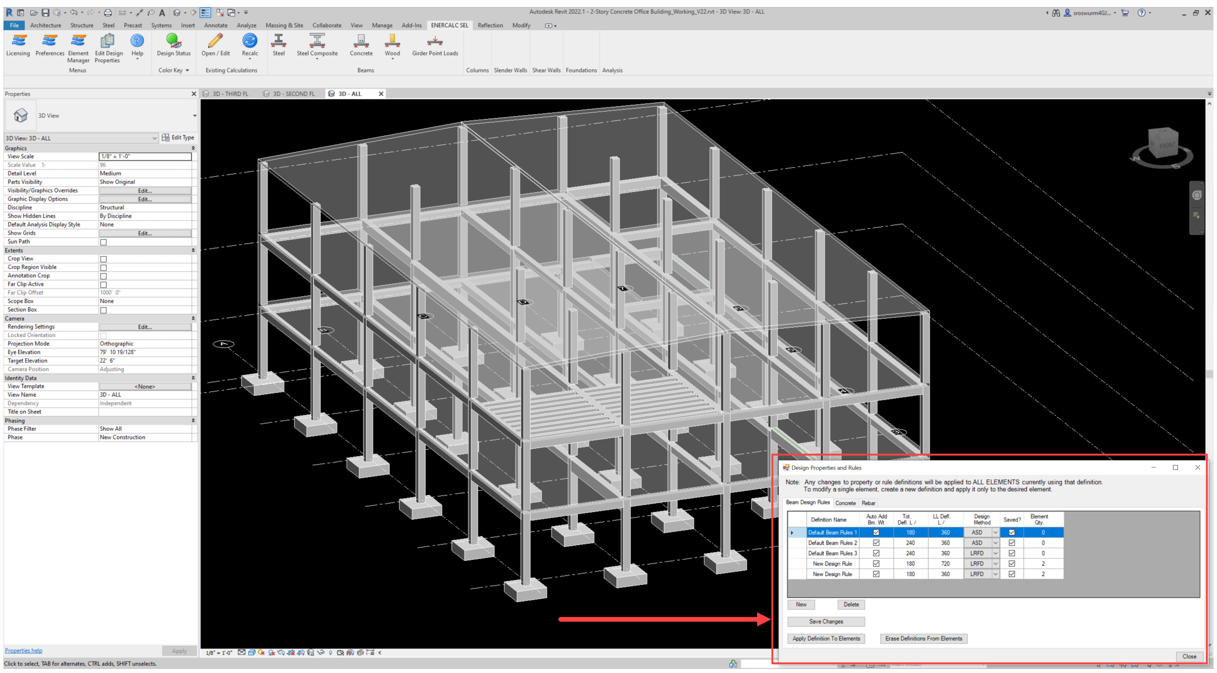
10.14.6 Concrete Material Properties

Concrete beam material properties are created and managed using the "Edit Design Properties" button on the ribbon bar:



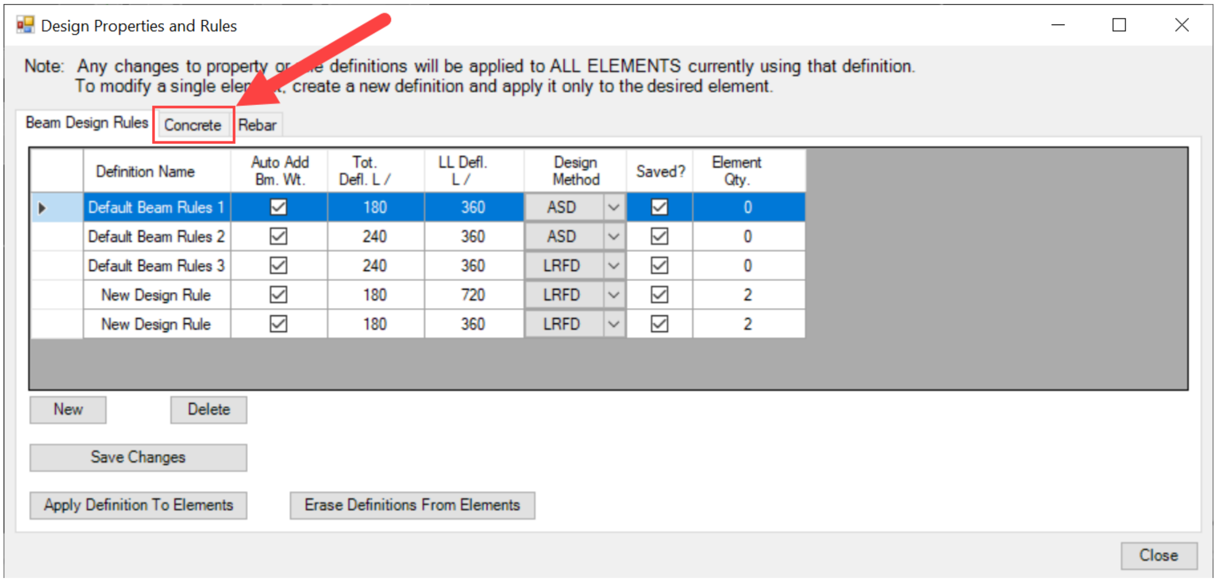


When triggered from the ribbon button, the menu will load in the lower right corner of the Revit UI:

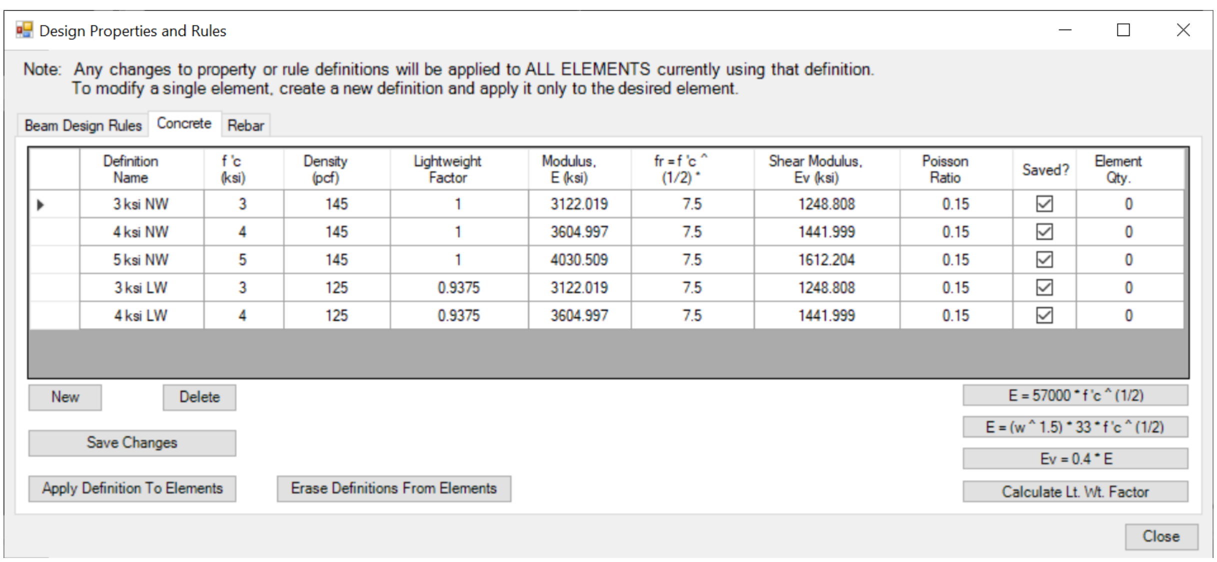


This menu also contains tabs for managing other design properties, including beam design rules and rebar material properties. For more detail on the use of these tabs, refer to [Beam Design Rules](326) and [Rebar Material Properties](485).

Concrete material definitions are found on the second tab of the menu:



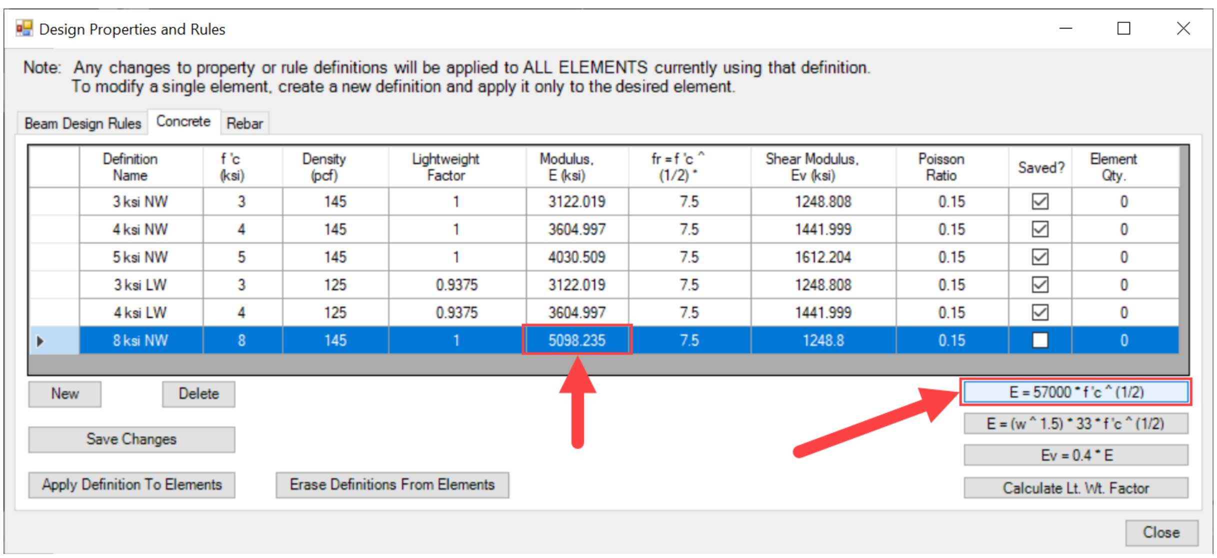
In a new project where no previous definition editing has been performed by the users, the menu will be populated automatically with a set of 5 sample concrete materials (3 normal weight and 2 lightweight) which are stored in the .rvt file by default. Use of these default materials is optional. They may be removed or edited/renamed as desired by the user. Even though they exist in the Revit model, the default materials will not influence the behavior of concrete beam calculations unless manually applied to specific beam elements by the user.

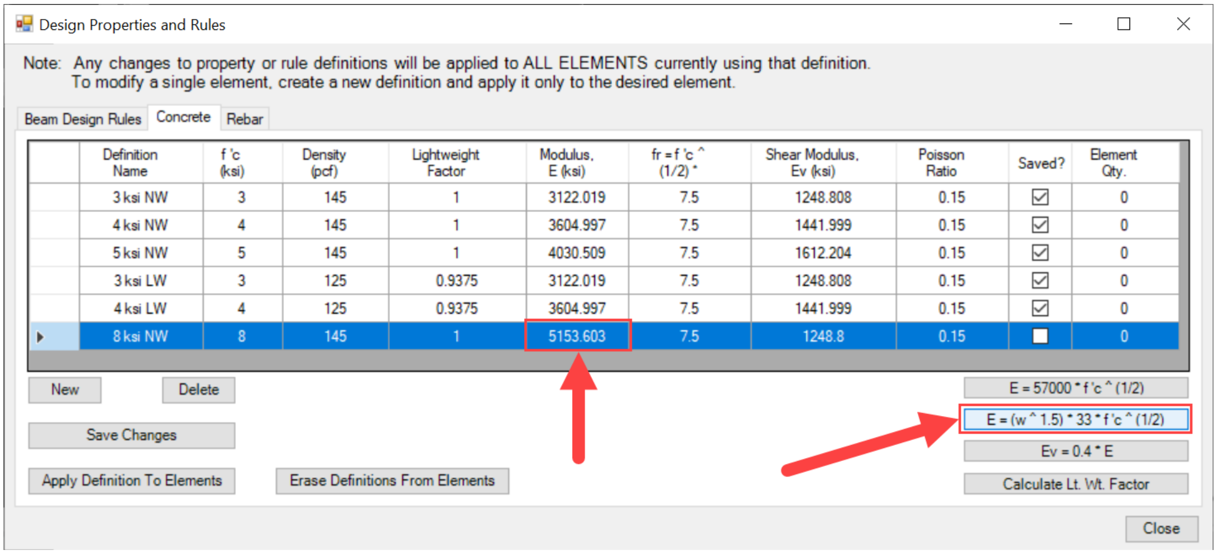


Users should note that the basic mechanics are the same for all design definition tabs found in this menu. The processes to "Apply", "Modify", "Add", "Delete", or "Erase from Elements" do not vary from tab to tab. Regardless of which definition tab is being used, users may reference the following sections for guidance about managing design definitions:

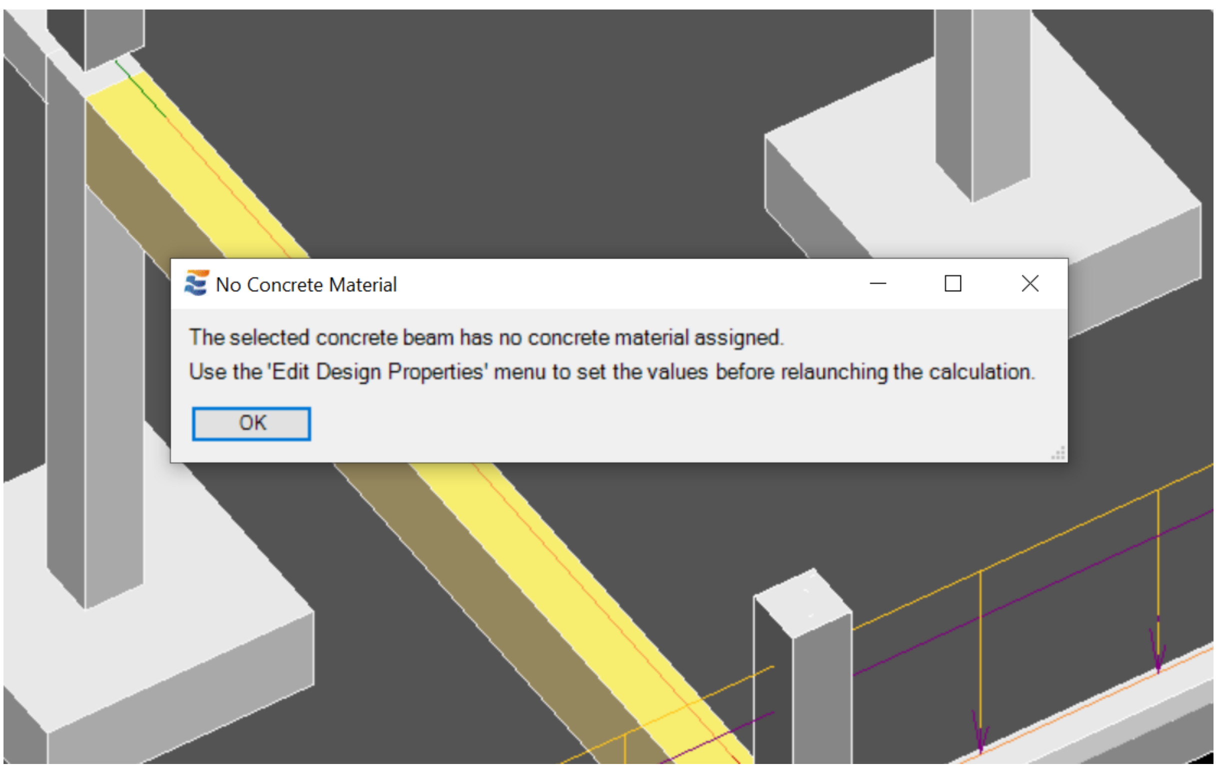
Applying Rules to Elements 330

In addition to the ability to edit each cell of each definition manually, the buttons in the lower right corner of the menu also allow the user to auto-update certain aspects of the material definition per industry-standard methods. This prevents any need for time consuming manual re-calculation of material properties in the event that critical properties such as compressive strength or unit weight are altered. Using any one of these buttons while a definition row is selected in the table will cause the corresponding value in the selected row to update automatically. The "Saved?" check box will then be cleared to indicate that the row has unsaved changes:



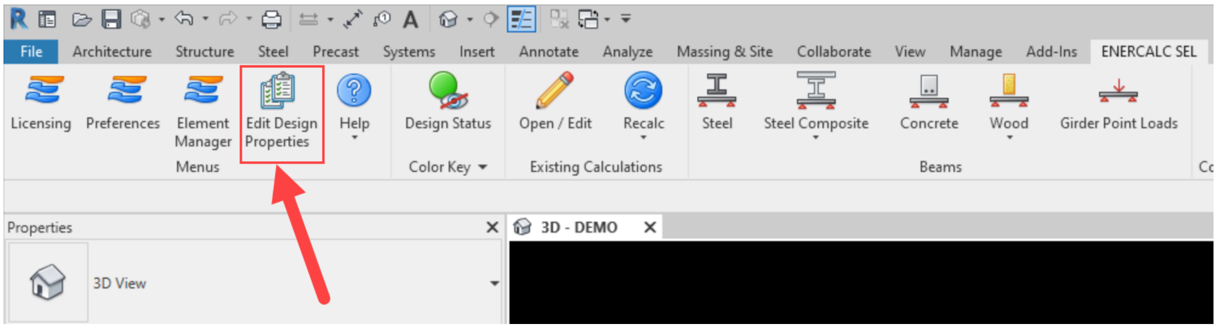


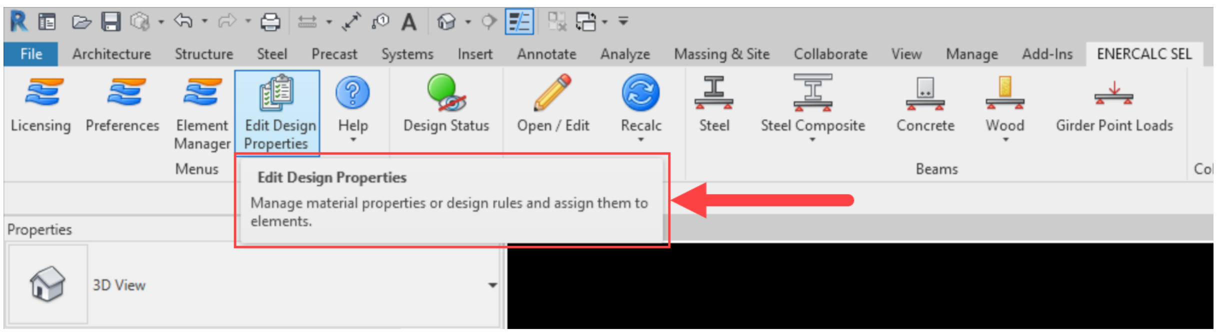
There are no default concrete and rebar material properties applied to elements automatically. They must be assigned to elements manually via the process outlined above. Any attempt to launch a concrete beam calculation when no concrete material has been applied will be halted with a warning:



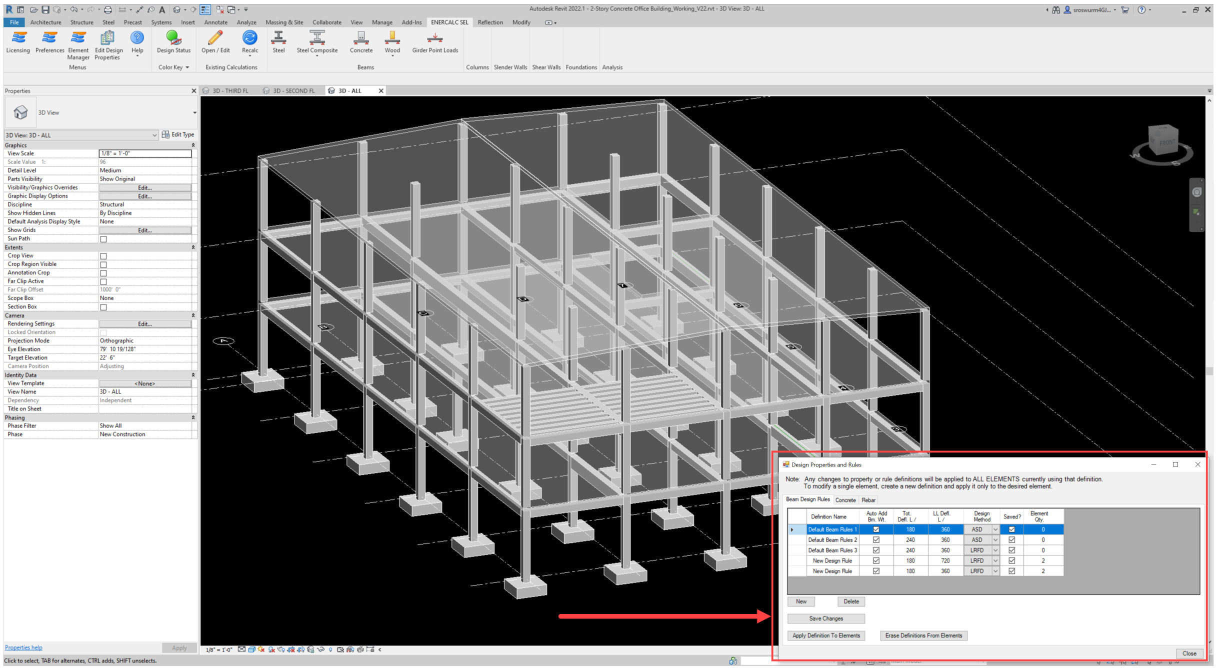
10.14.7 Rebar Material Properties

Concrete beam rebar material properties are created and managed using the "Edit Design Properties" button on the ribbon bar:



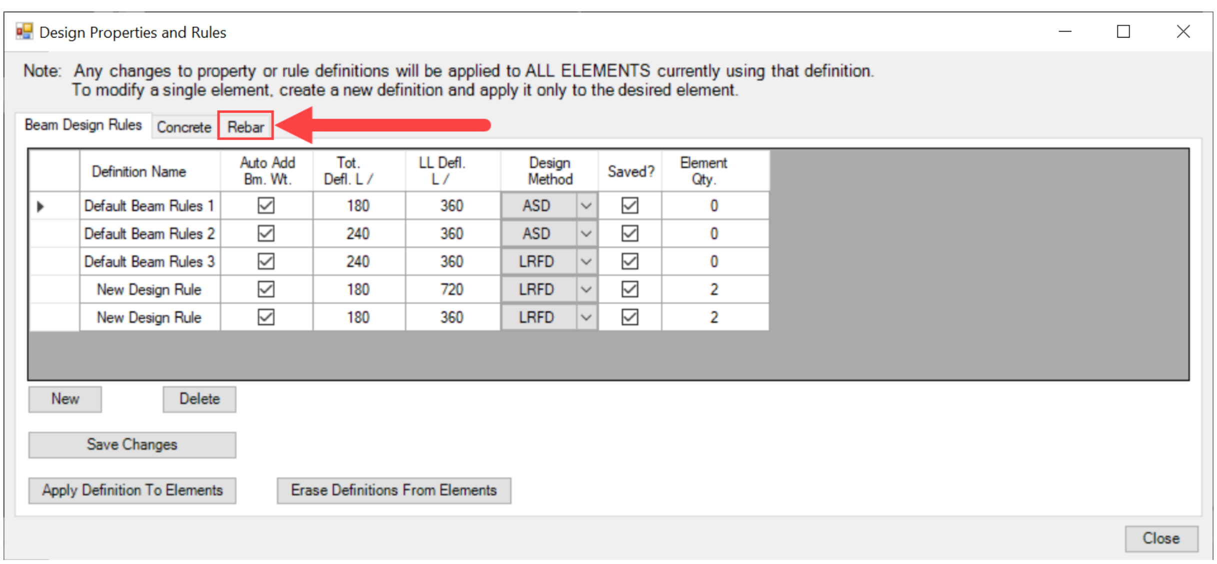


When triggered from the ribbon button, the menu will load in the lower right corner of the Revit UI:

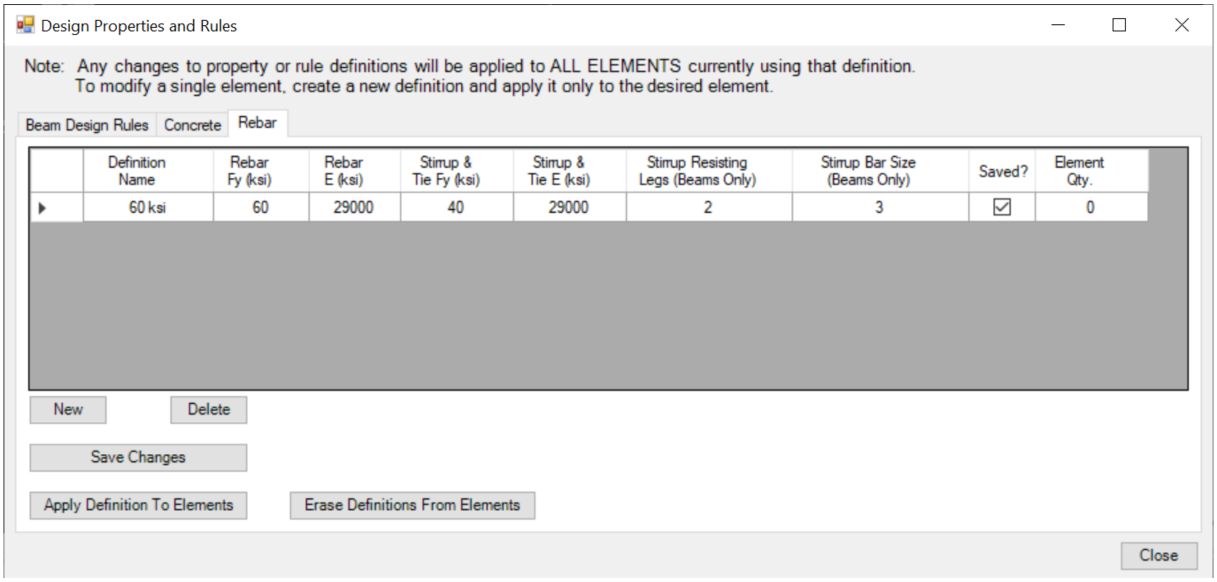


This menu also contains tabs for managing other design properties, including beam design rules and concrete material properties. For more detail on the use of these tabs, refer to Beam Design Rules 326 and Concrete Material Properties 480.

Rebar material definitions are found on the third tab of the menu:



In a new project where no previous definition editing has been performed by the users, the menu will be populated automatically with one sample rebar material which is stored in the .rvt file by default. Use of this default materials is optional. It may be removed or edited/renamed as desired by the user. Even though it exists in the Revit model, the default material will not influence the behavior of concrete beam calculations unless manually applied to specific beam elements by the user.



Users should note that the basic mechanics are the same for all design definition tabs found in this menu. The processes to "Apply", "Modify", "Add", "Delete", or "Erase from Elements" do not vary from tab to tab. Regardless of which definition tab is being used, users may reference the following sections for guidance about managing design definitions:

Applying Rules to Elements 330

Modifying Rules 333

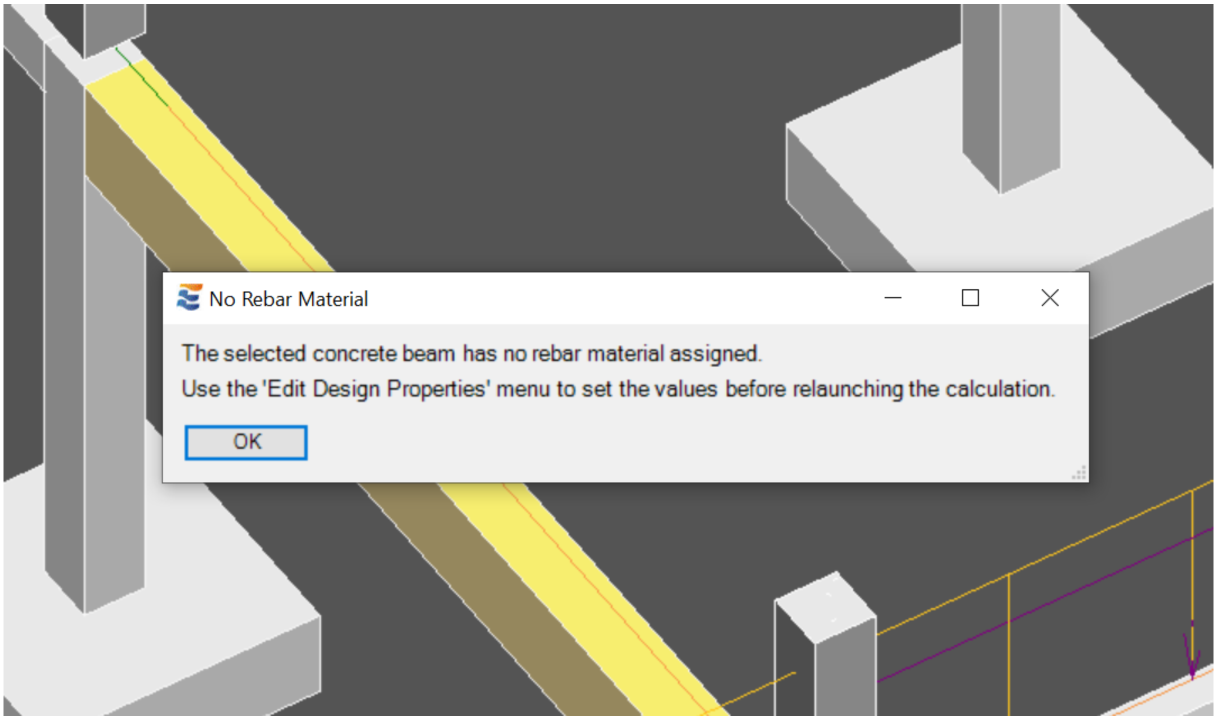
Adding Rule Definitions to a Project 337

Deleting Rule Definitions from a Project 339

Erasing Rule Definitions From Elements 341

Navigating Rule Definitions 343

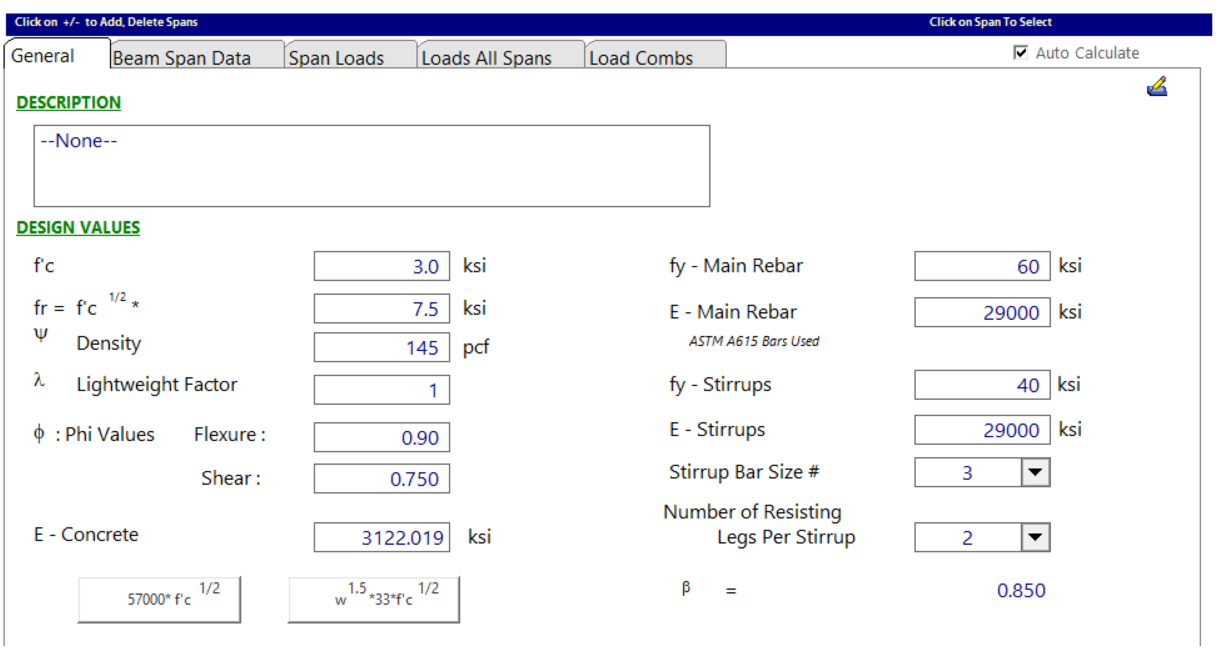
Any attempt to launch a concrete beam calculation when no rebar material has been applied will be halted with a warning:



10.14.8 Changing Material Props in ENERCALC

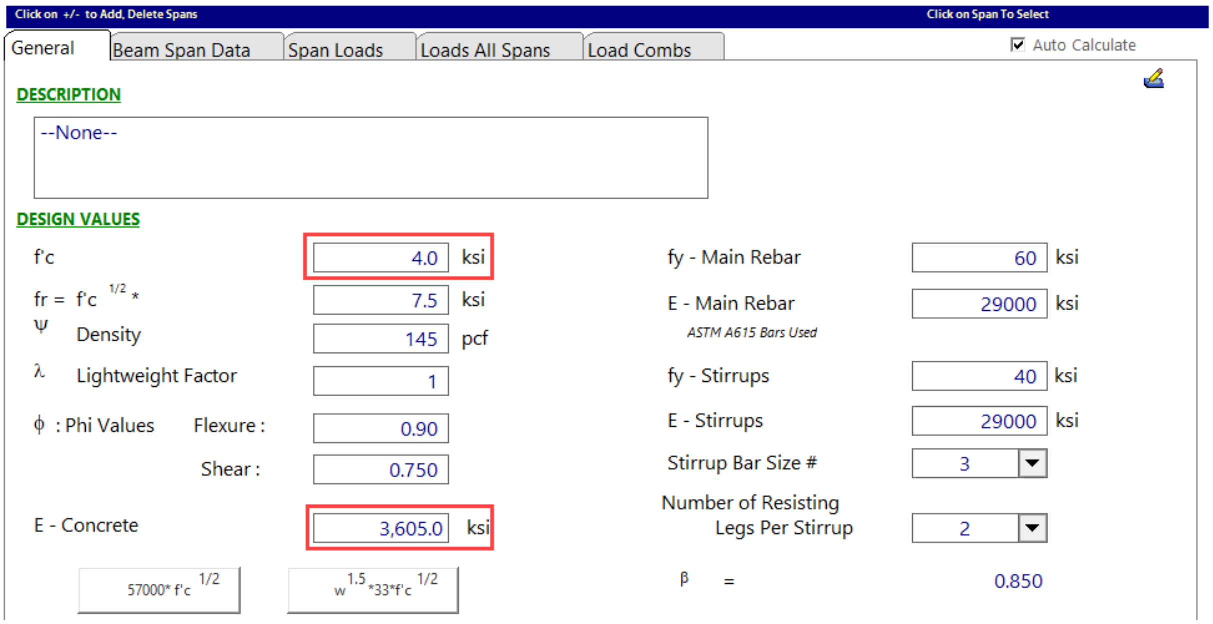
Prior to launching a concrete beam calculation, the user must assign material properties to the Revit beam element. This process is described in Concrete Material Properties 480 and Rebar Material Properties 485.

Once a concrete beam calculation has loaded in the ENERCALC interface, the concrete and rebar material properties will be exposed to view and modification by the user.

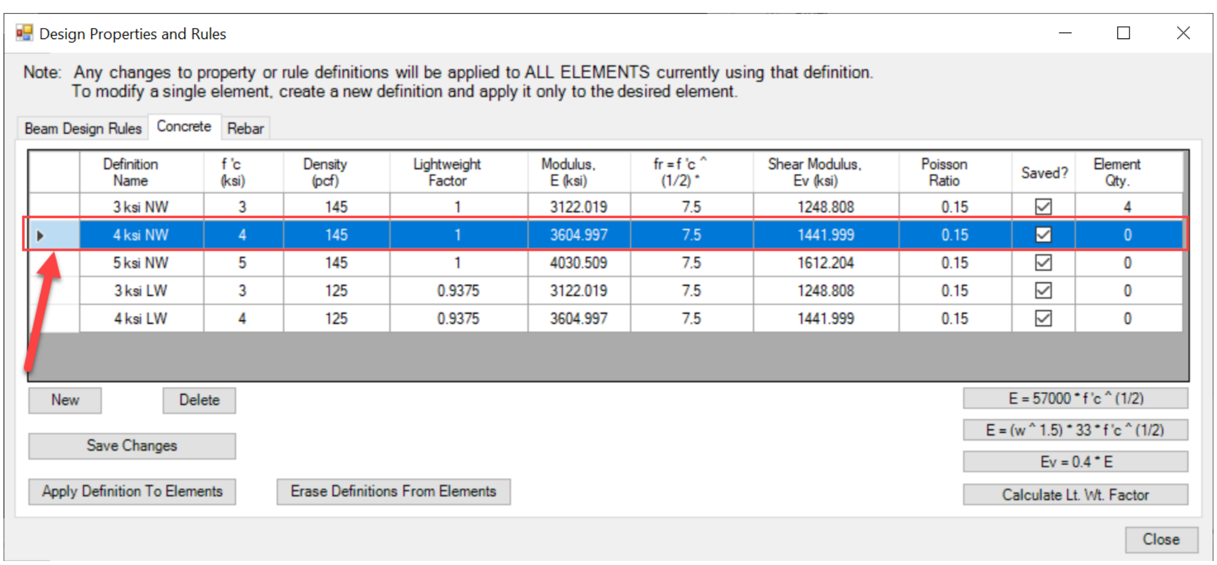


If the user manually modifies the material properties, then the new values will be applied to the Revit model in one of two ways:

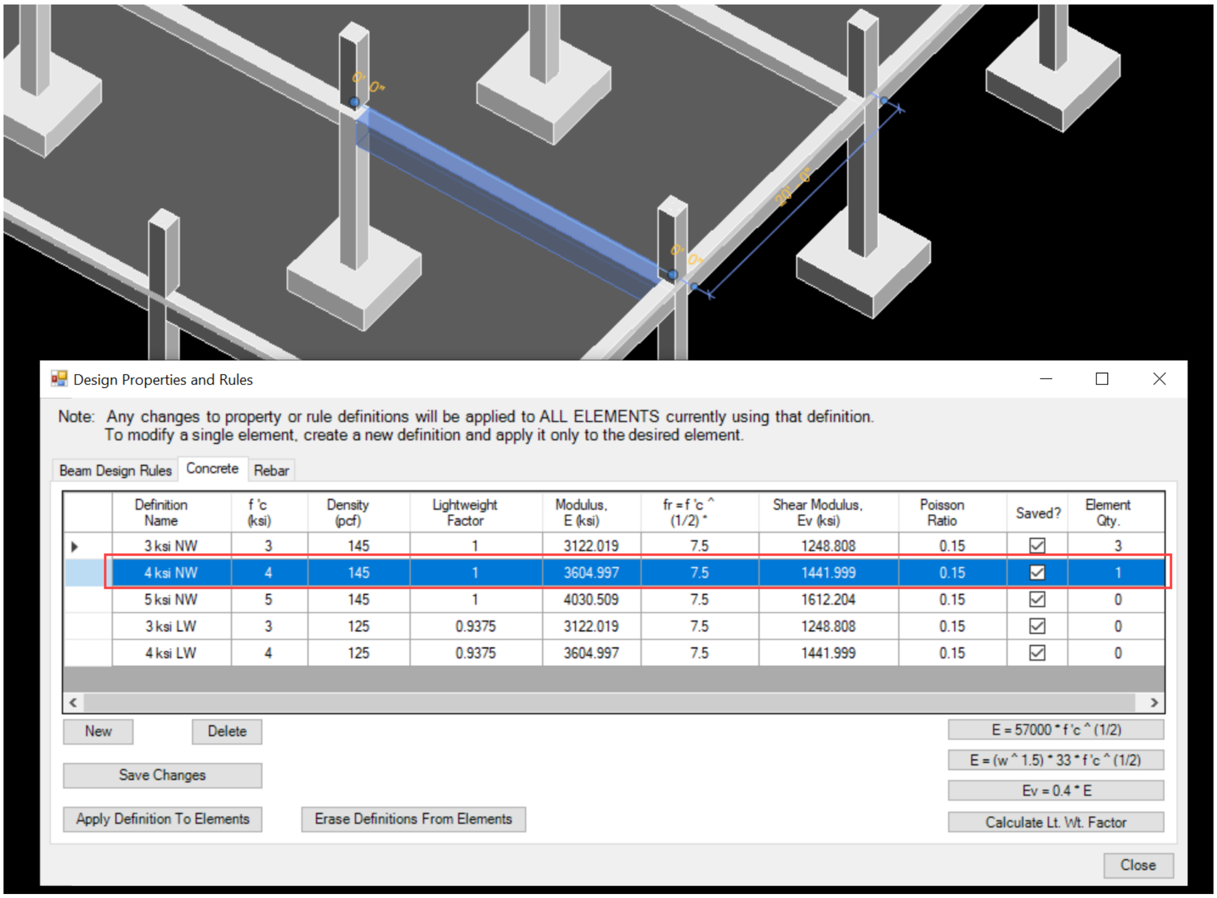
1.) If an existing material definition found in the Revit project has matching properties, then the matching definition will be applied to the concrete beam on "Save and Close". In the example shown here, the compressive strength is changed to 4.0 ksi and the modulus of elasticity is given a corresponding calculated value:



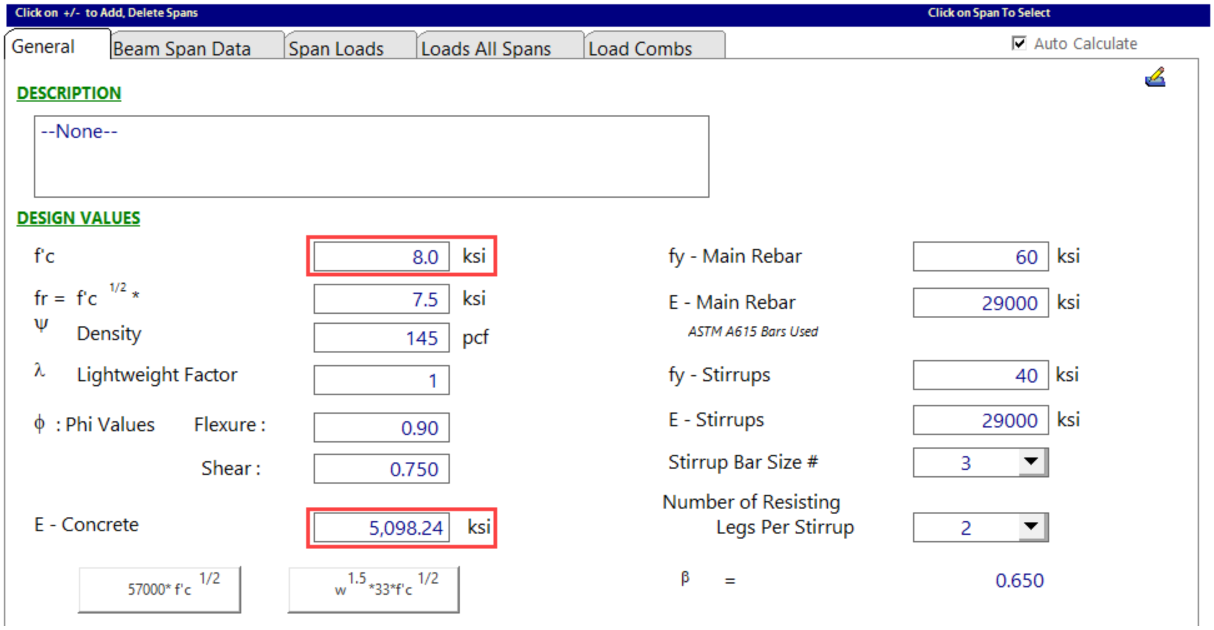
Viewing the "Design Properties and Rules" menu, it can be seen that a matching definition exists in the Revit model:



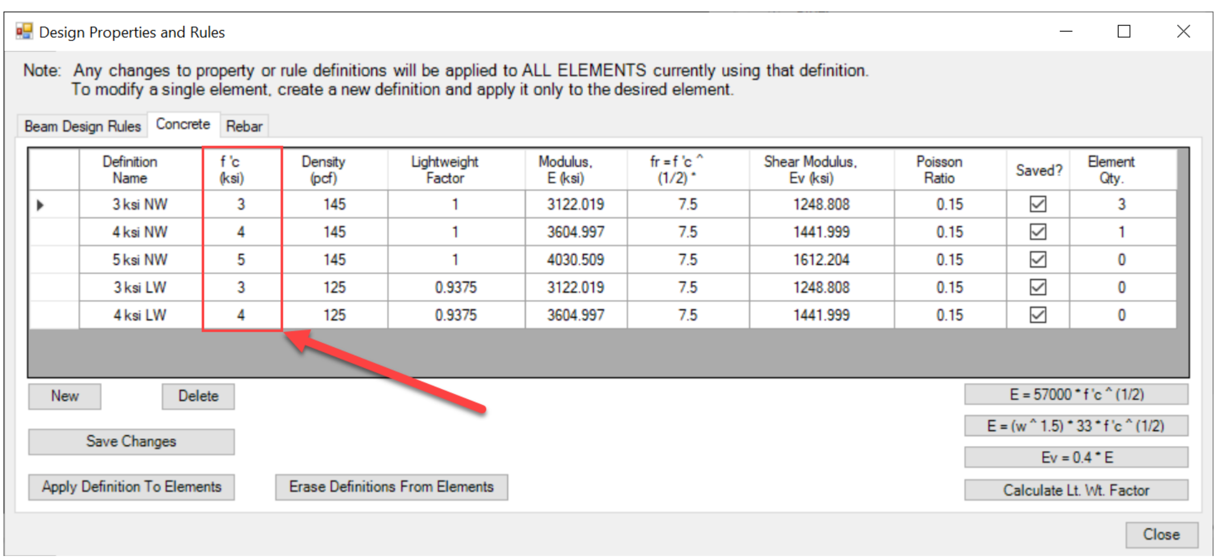
When the "Save and Close" operation concludes, the "Design Properties and Rules" menu indicates that the beam has been automatically reassigned to have the existing definition that matches the properties set in the ENERCALC interface:



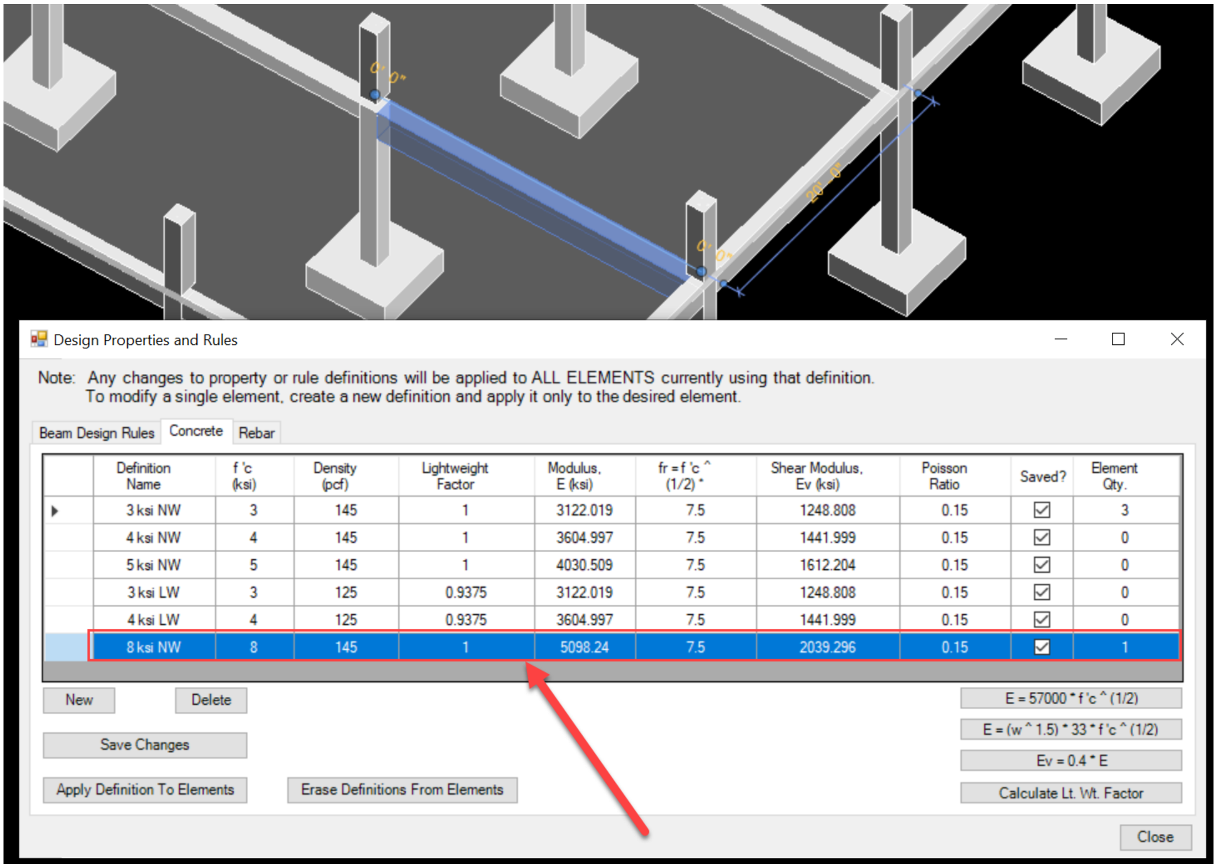
2.) If there is no existing material definition found in the Revit project with properties that match the value assigned in ENERCALC, then a new definition will be created and applied to the concrete beam on "Save and Close". In the example shown here, the compressive strength is changed to 8.0 ksi and the modulus of elasticity is given a corresponding calculated value:



Viewing the "Design Properties and Rules" menu, it can be seen that no matching definition exists in the Revit model:

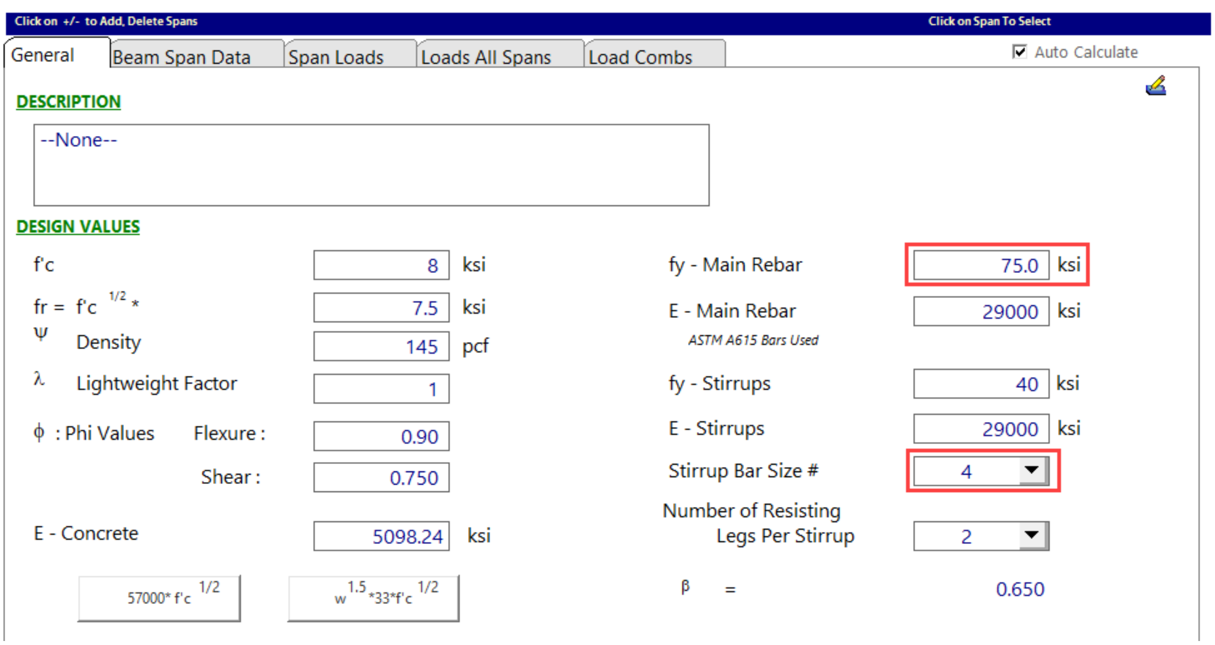


When the "Save and Close" operation concludes, the "Design Properties and Rules" menu indicates that a completely new definition has been created in the Revit project and the Revit beam element has been automatically reassigned to have the new definition with the properties set in the ENERCALC interface:

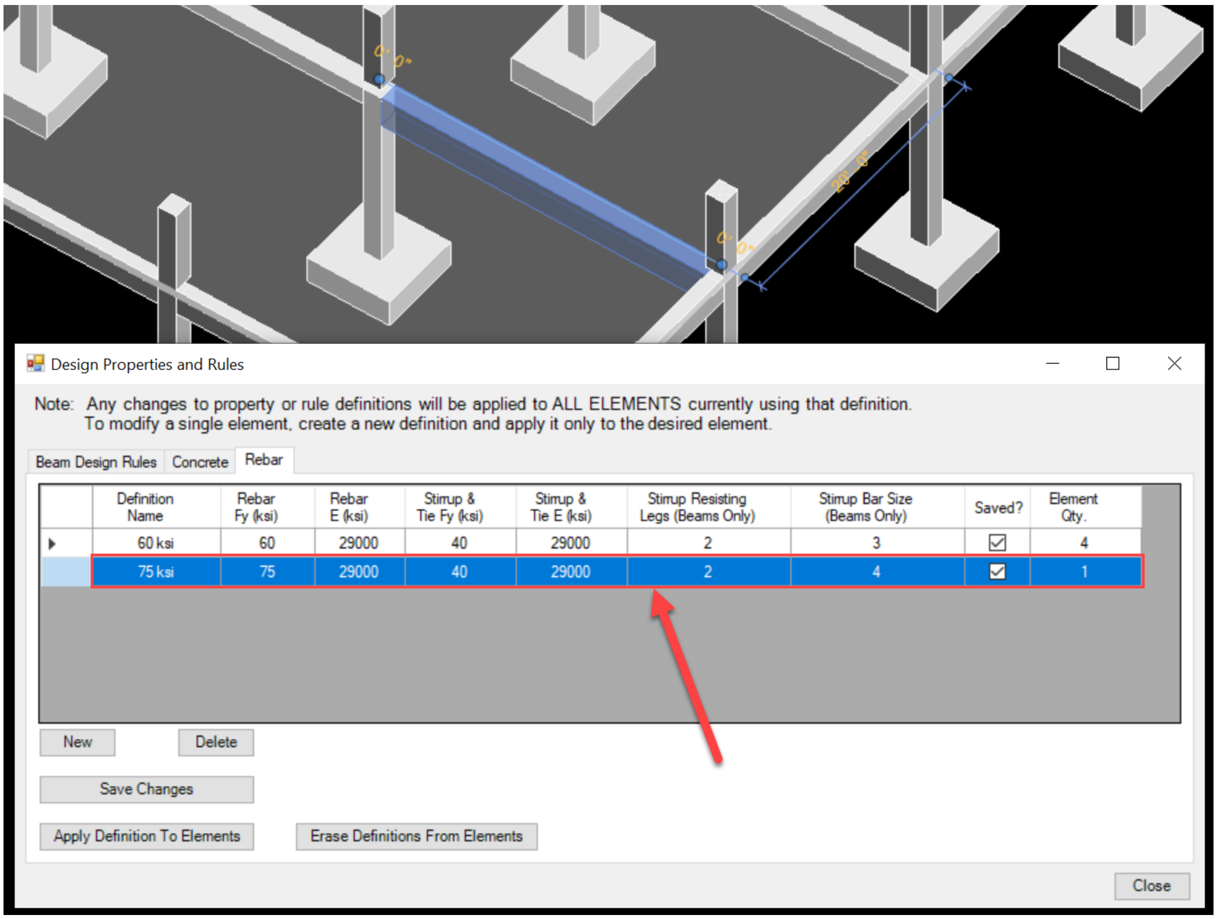


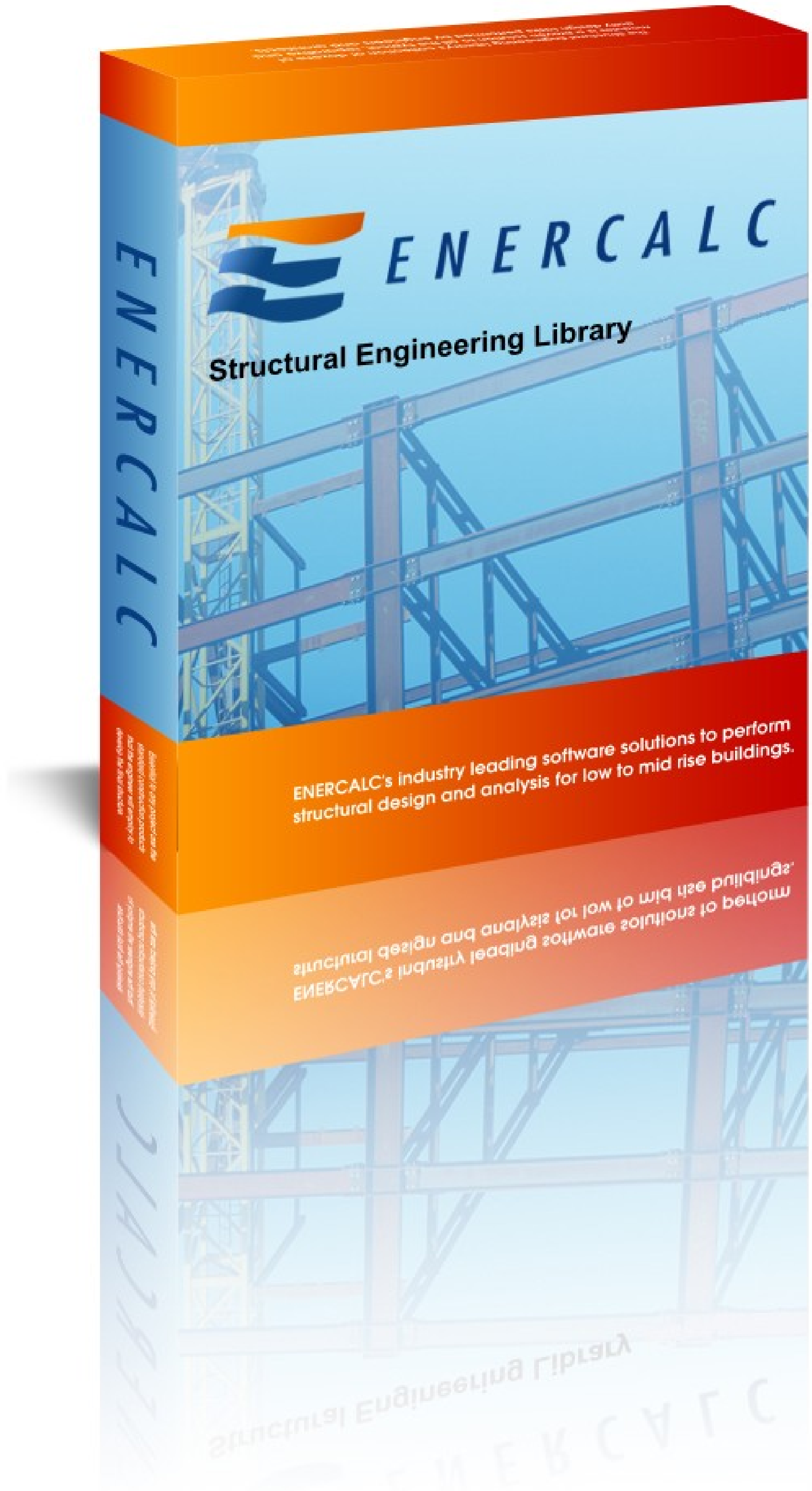
Definition Name	f 'c (ksi)	Density (pcf)	Lightweight Factor	Modulus, E (ksi)	fr = f 'c ^ (1/2) *	Shear Modulus, Ev (ksi)	Poisson Ratio	Saved?	Element Qty.
3 ksi NW	3	145	1	3122.019	7.5	1248.808	0.15	☑	3
4 ksi NW	4	145	1	3604.997	7.5	1441.999	0.15	☑	0
5 ksi NW	5	145	1	4030.509	7.5	1612.204	0.15	☑	0
3 ksi LW	3	125	0.9375	3122.019	7.5	1248.808	0.15	☑	0
4 ksi LW	4	125	0.9375	3604.997	7.5	1441.999	0.15	☑	0
8 ksi NW	8	145	1	5098.24	7.5	2039.296	0.15	☑	1

The provisions and behaviors outlined above also apply to rebar material definitions. An example of revised rebar properties is shown below:



When the "Save and Close" operation for this calculation concludes, the user will observe that a new rebar material definition has been automatically created in the Revit project and assigned to the Revit beam element:




ENERCALC
Structural Engineering Library
ENERCALC's industry leading software solutions to perform structural design and analysis for low to mid rise buildings.
ENERCALC