RISA-3D

Tutorial – Version 22

RISA Tech, Inc.
27442 Portola Parkway, Suite 200
Foothill Ranch, California 92610
Phone: (949) 951-5815
Email: info@risa.com
risa.com

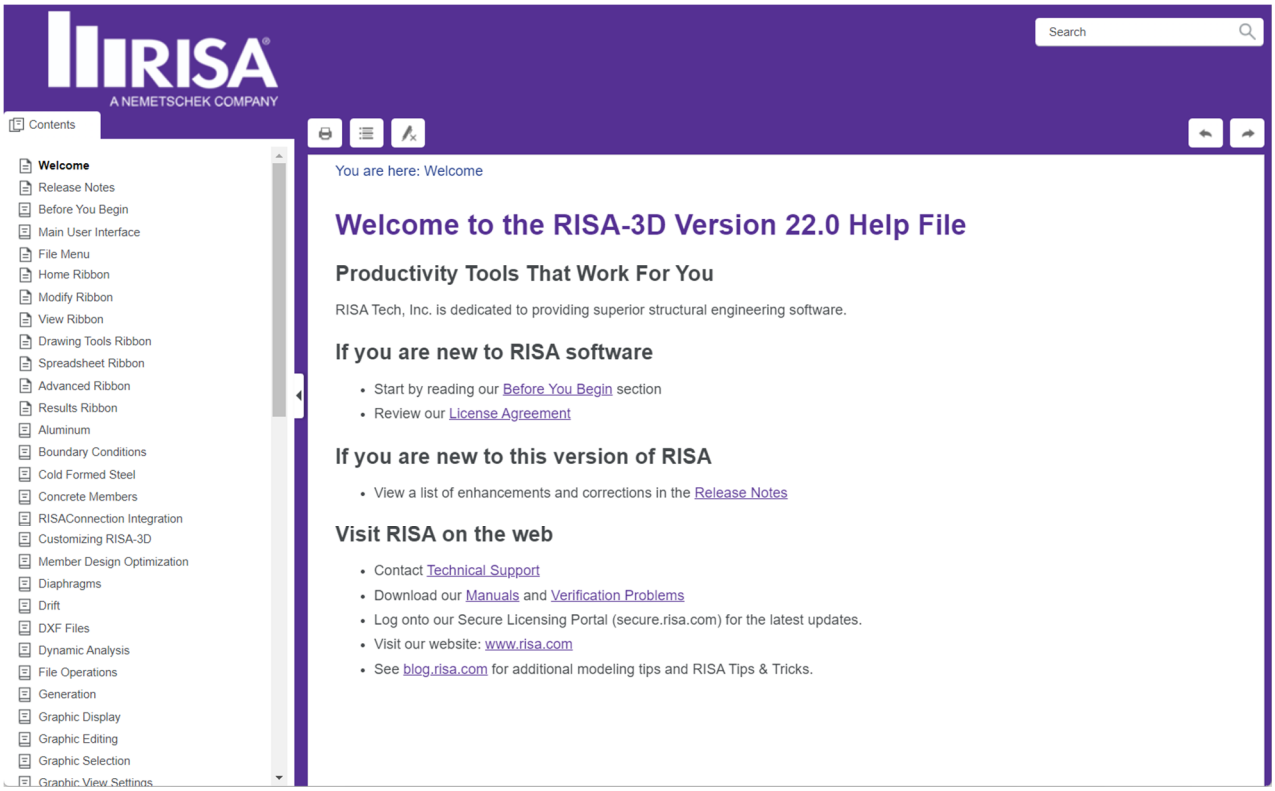
Table of Contents

Online Help

Whether you need help on general topics, specific features, the interface or licensing the RISA-3D online Help system has the answers you are looking for. The Help menu was designed to enable you to pinpoint the information you need quickly, by offering various ways for you to access and locate the Help topics you are interested in.

If you are interested in Help on **General Topics**, click the **RISA Help** button found in the top right corner of the **Quick Access** toolbar. When working inside the software, this is the fastest way to get help on General Topics as well as specific features. It is also possible to go to **File > Help** where the **RISA-3D Tutorials, General Reference Guide** and **Licensing Guide** can be found.

When you enter the online help system, you will notice two tabs on the left: **Contents** and **Index.** From here you can explore the general help topics by selecting specific sections or by selecting **Index** and searching the list of indexed terms. Additionally, the **Search** bar in the top right corner of the window allows for a keyword search of the entire online help database for easy access to more specific topics.



Technical Support

Technical support is an integral part of the software packages offered by RISA and is available to all customers with an active maintenance agreement or subscription license at no additional charge. Our knowledgeable support team, comprised of registered engineers, is ready to answer your most challenging questions.

Before contacting technical support, you may want to take a few minutes to do the following:

- Search the **Online Help** and all user documentation available for the product.
- Search the **Support** and **Learn** articles posted on our website by visiting RISA Support Articles

When you are ready to make a support request, please be prepared to send us your model, and include the following information:

- Your name, company name, and phone number
- Product name and serial number or Key ID
- A detailed description of your problem
- Your model (filename.r3d) as an e-mail attachment. If your model contains multiple members, plates, or load combinations, please specify which ones we should look at.

When you are ready, you can reach out to Technical Support using the following contact methods:

- **E-mail:** support@risa.com
  E-mail is usually the best way to communicate with us when sending a model. When doing so, please include all the information listed above.

- **Phone:** (800) 332-RISA (7472)
  Technical support personnel are available from 6:00 A.M. to 5:00 P.M. Pacific Standard Time, Monday through Friday.

Online Resources

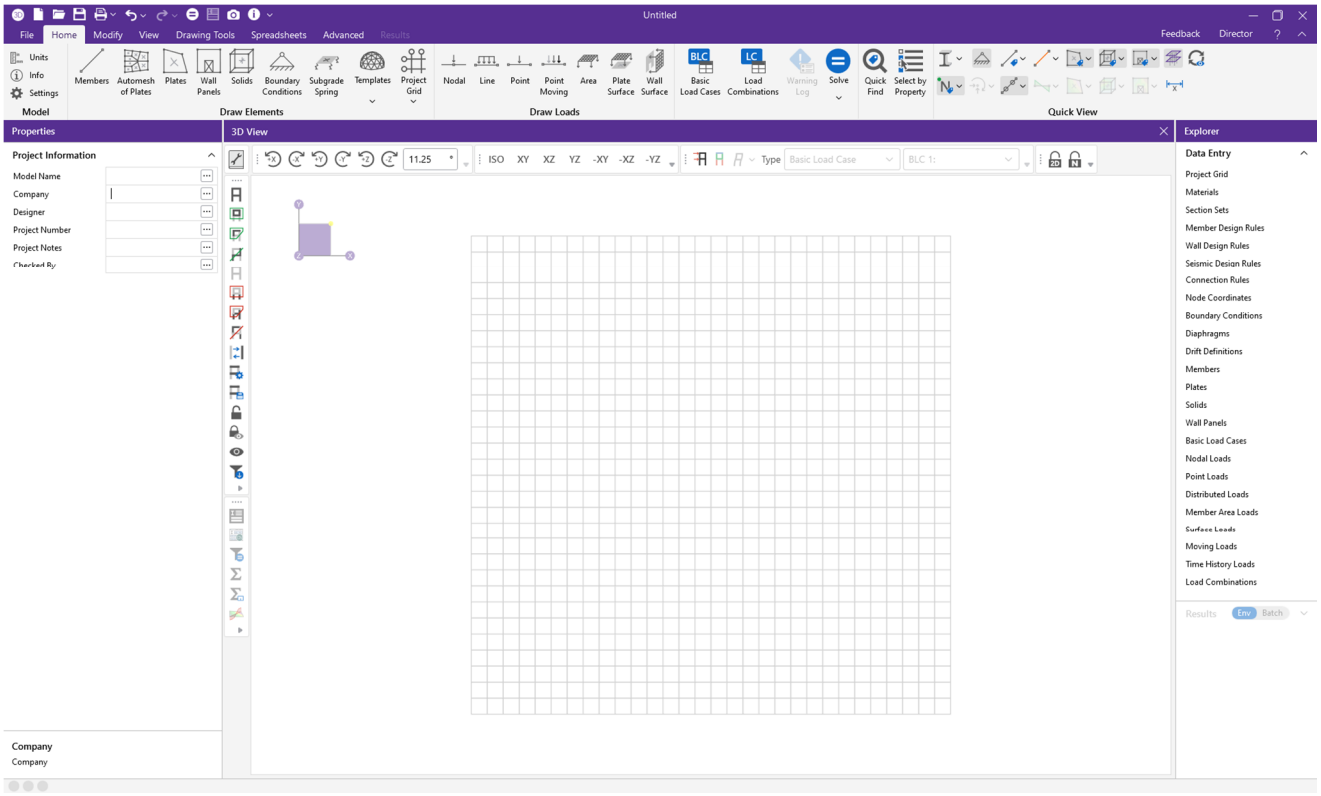
Additional information is available 24/7 by visiting our website, risa.com. There you will find:

- Answers to FAQ's and technical support articles
- Downloads of general reference guides, tutorials and verification problems
- Videos and upcoming live webinars
- Information on upcoming training classes and events
- Product specific case studies
- Software updates – Any known problems are posted on the website, along with possible workaround procedures and/or service releases to update your software.

First Look at RISA-3D

This section includes a quick look at the user interface of RISA-3D and its various components. If you are new to RISA-3D, review this section before beginning the tutorial to ensure that you are able to navigate all aspects of the user interface. You can also visit our YouTube channel to view a video about the user interface, called Experience the Updated User Interface in RISA-3D. Additionally, if you are already familiar with the user interface, proceed to the **Modeling** section of the tutorial on **page 10.**

The user interface of RISA-3D is set up to provide a streamlined workflow, allowing engineers to get the most out of the software. Additionally, each area of the interface serves a specific function which allows engineers to work more efficiently when modeling, analyzing or documenting their projects. Let's take a look at the various areas of the interface in order to better understand their function.

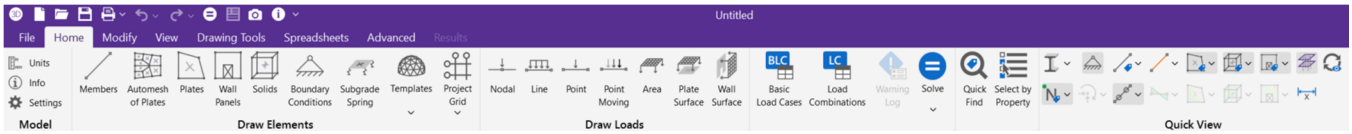


Quick Access Toolbar ①

The **Quick Access Toolbar** provides easy access to the following features:

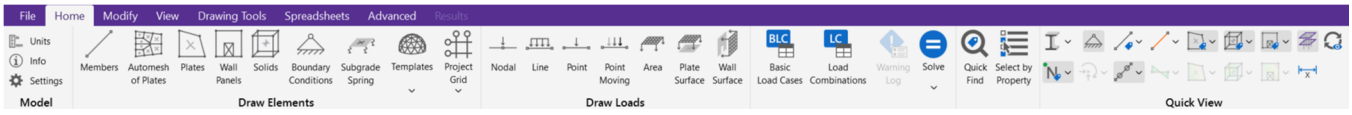
- New, Open & Save Project
- Print Report or Graphic
- Undo & Redo
- Solve
- Detailed Report
- Camera Snapshot
- Help/Info

Ribbon Toolbar ②

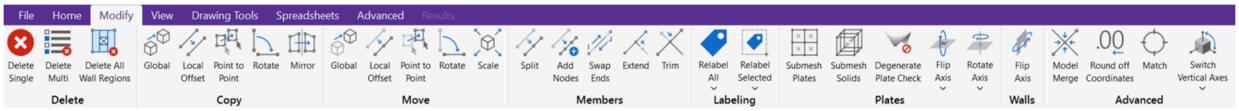


The ribbon toolbar is the hub for all features and functions within RISA-3D. With a logical left-to-right, tab-based approach, the ribbon makes it effortless to progress through a model from its conception to its final optimization by simply following the icon sequencing. Within the toolbar, the following tabs are included:

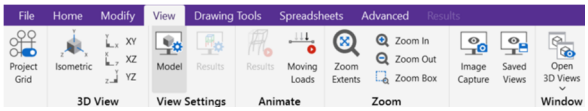
Home



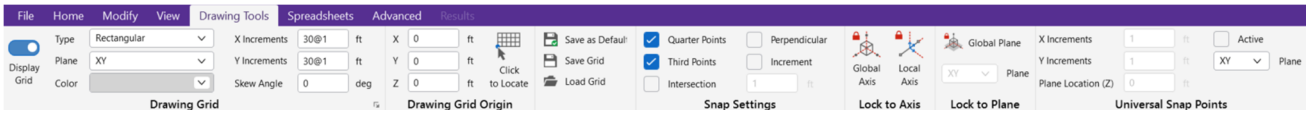
Modify



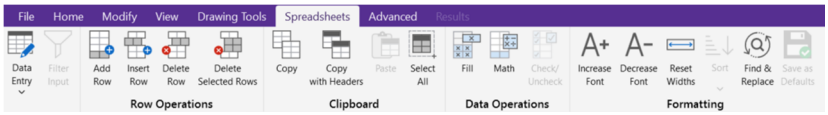
View



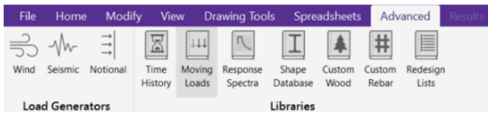
Drawing Tools



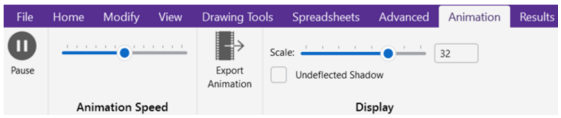
Spreadsheets



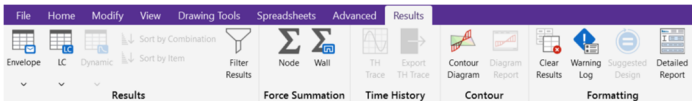
Advanced



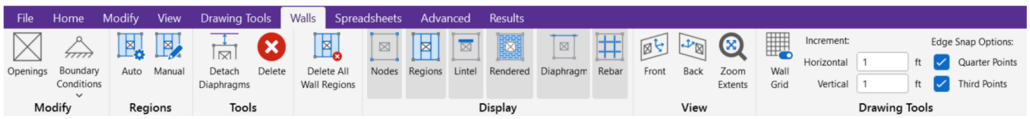
Animation



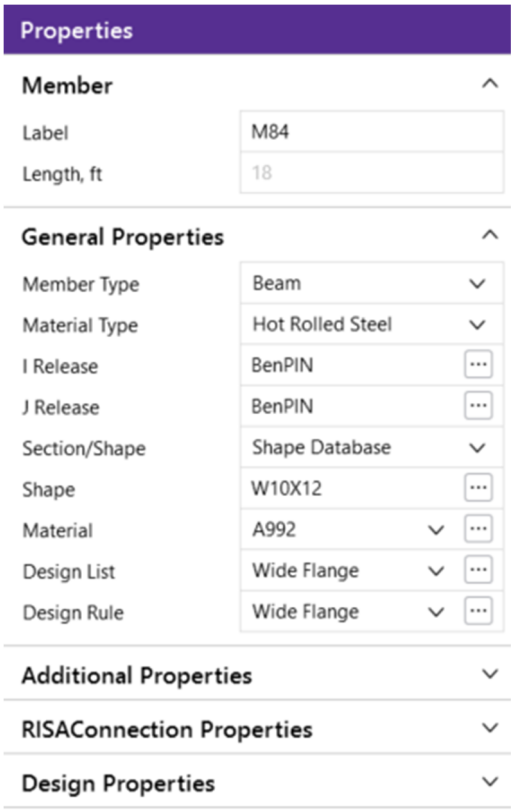
Results



Walls

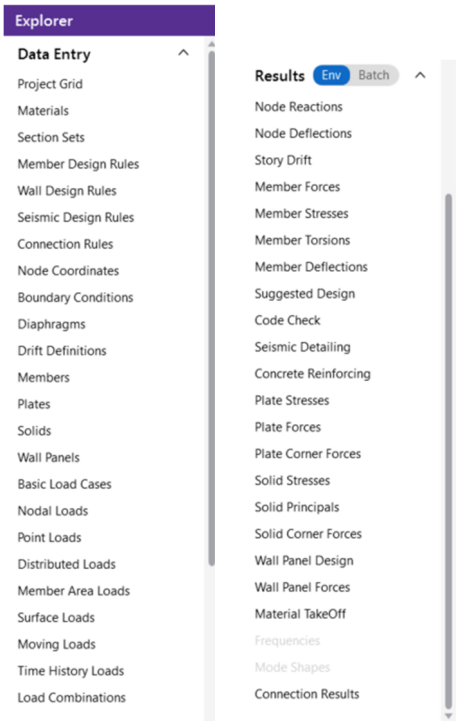


Properties Panel ③



The **Properties Panel** allows you to quickly view every property associated with a given element. Additionally, the modification of properties associated with a given node, boundary condition, member, plate, wall panel or load can be completed within the same interface.

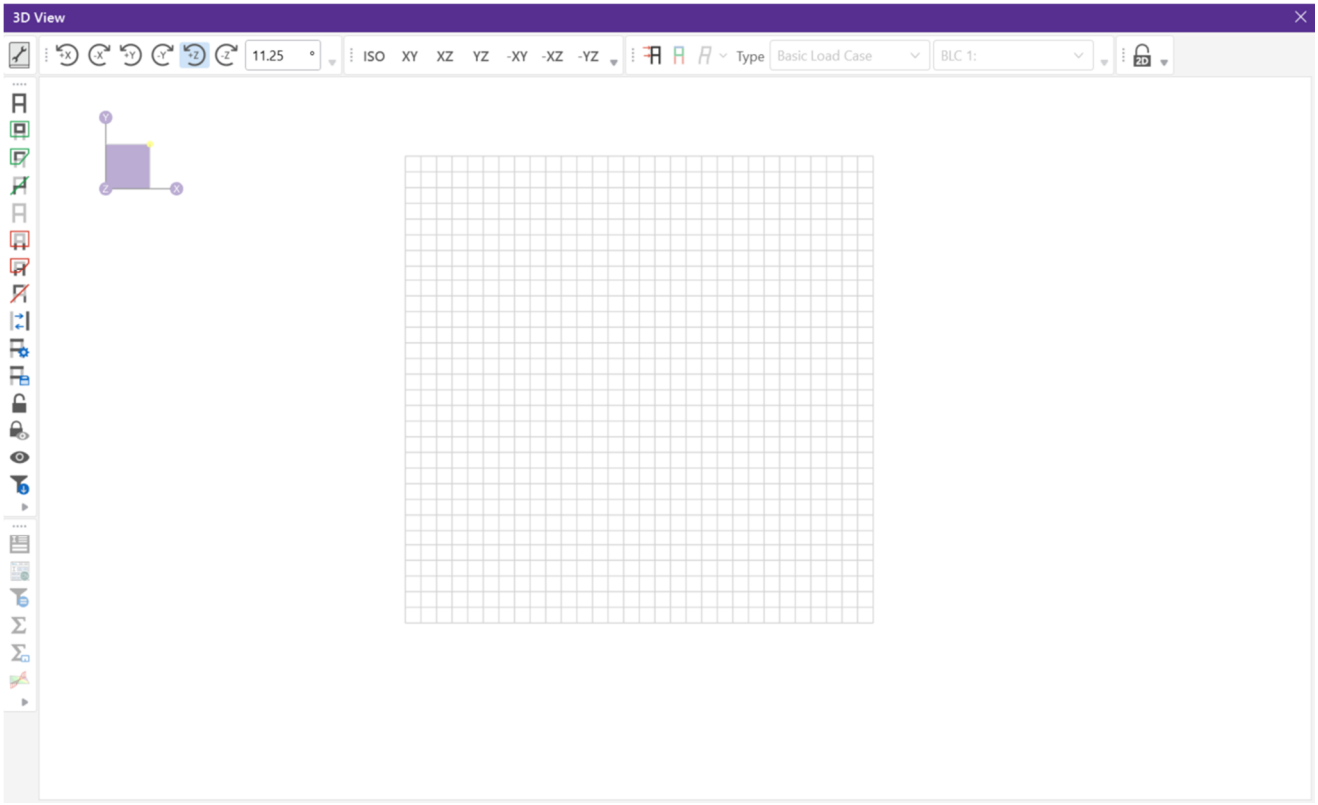
Explorer Panel ④



The **Explorer Panel** is the easiest way to access the data entry and results spreadsheets. These toolbars are used to access input spreadsheets such as Section Sets, Design Rules and Basic Loads Cases, as well as results spreadsheets such as Node Reactions, Member Forces and Design Results.

3D View Window ⑤

The **3D View** window is where the model will be created. When starting a new project, the workspace is blank and includes a default **Drawing Grid.** Additional 3D view windows and spreadsheets can be added or enabled and docked within the interface in order to provide better model visibility.



The 3D View window also includes the following toolbars:

Selection Toolbar ⑥

The selection toolbar is located on the left side of the 3D View window and provides model selection tools to help you work with and make changes to smaller portions of the model.

Design Results Toolbar ⑦

The design results toolbar is available when a solution is present. Tools such as detailed report, filtering and internal force summation help to fine tune the presentation of results.

View Toolbar ⑧

The view toolbar provides preset view options (ISO, XY, XZ & YZ) as well as the ability to enable and toggle the visibility of loads, results and deflections.

Quick View ⑨

The quick view section of the **Home** tab allows users to easily change the model view properties of the 3D View Window. View properties for inputs (boundary conditions, members, diaphragms, etc.) exists as well as view properties specifically used to change the way a specific result (members, plates, wall panels, etc.) are viewed. If greater control of the view properties is required, open either the **Model** or **Results** view settings found on the **View** tab.

Keyboard Shortcuts & Hot Keys

Shortcut Keys and **Hot Keys** allow you to use the keyboard to quickly access features. The difference between the two is simply that the shortcut keys are related to a specific window and will only work in that window while the hot keys will perform in any situation.

General Hot Keys

Key Combination	Function
F1	Help on the active window
F5	Activates the Distance Tool
Ctrl-F1	Main Help topics
F7	Opens solution choices
Ctrl-F7	Solve (Batch + Envelope Solution)
Ctrl-Alt-F7	Replace shapes with suggested shapes and resolve the model
Ctrl-C	Copy to the clipboard
Ctrl-V	Paste from clipboard
Ctrl-N	Start a new file
Ctrl-O	Open an existing file
Ctrl-S	Save the current file
Ctrl-P	Print
Ctrl-Z	Undo

Shortcut Keys available for Specific Windows

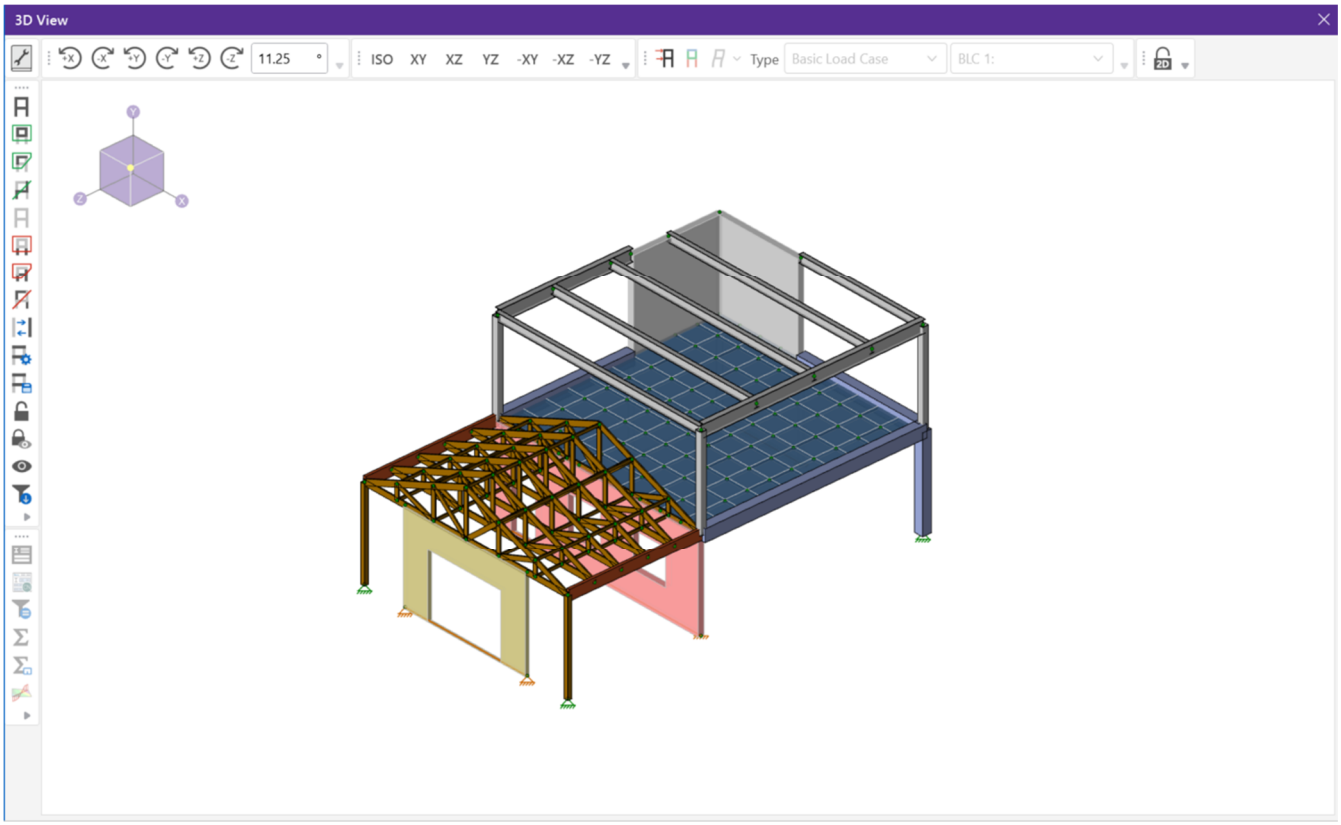
Key Combination	Model View Window	Spreadsheet
Ctrl-A	Select All	Select All
Ctrl-U	Unselect All	Unselect All
Ctrl-F	-	Block Fill
Ctrl-M	-	Block Math
Ctrl-I	Invert Selection	-
Ctrl-L	Toggle Lock unselected	-
F2	Open Model View Settings	Start/Stop Cell Edit
F3	-	Insert line
F4	-	Delete Line
F5	Initiates the distance tool	Initiates the Find and Replace Tool
F8	-	Repeat Current Line
F9	-	Sort Spreadsheet

Shortcut Keys that open Spreadsheets

Key Combination	Unsolved Model	Solved Model
Ctrl-Alt-B	Basic Load	-
Ctrl-Alt-C	Node Coordinates	Corner Forces
Ctrl-Alt-D	Member Distributed Loads	Node Deflections
Ctrl-Alt-E	Member Primary Data	Deflection Results
Ctrl-Alt-F		Member Forces
Ctrl-Alt-G	Model Settings	Model Settings
Ctrl-Alt-H		Suggested Shapes
Ctrl-Alt-I	Diaphragms	Member Torsions
Ctrl-Alt-J	Wall Panel Rules	Solid Principal Stress
Ctrl-Alt-K		Solid Stresses
Ctrl-Alt-L	Load Combinations	Plate Forces
Ctrl-Alt-M	Materials	Material Take Off
Ctrl-Alt-N	Nodal Loads	Concrete Reinforcing
Ctrl-Alt-O	Boundary Conditions	Boundary Conditions
Ctrl-Alt-P	Member Point Loads	Plate Stresses
Ctrl-Alt-R	Design Code Check	Node Reactions
Ctrl-Alt-S	Section Sets	Member Stresses
Ctrl-Alt-T	Time History Loads	Story Drift
Ctrl-Alt-U	Seismic Design Rules	Code Check
Ctrl-Alt-V	Moving Loads	
Ctrl-Alt-X	Surface Loads	
Ctrl-Alt-Y	Dynamics Settings	
Ctrl-Alt-Z	Member Area Loads	
Ctrl-Alt-4	Plates	

Before You Start

This tutorial is designed to help guide you through the process of modeling, loading, analyzing, designing and documenting a simple structure in RISA-3D. The intent is to provide a straightforward workflow that simulates the “real world” design process and allows you to seamlessly make changes to the model, as would be required in any real project. In the sections that follow, the model shown below will be created:



As you work through the sections of the tutorial pay attention to the provided steps and images which aim to explain how to perform a given task, specifically the **bold** text highlights features or interface items that are important to the overall design process.

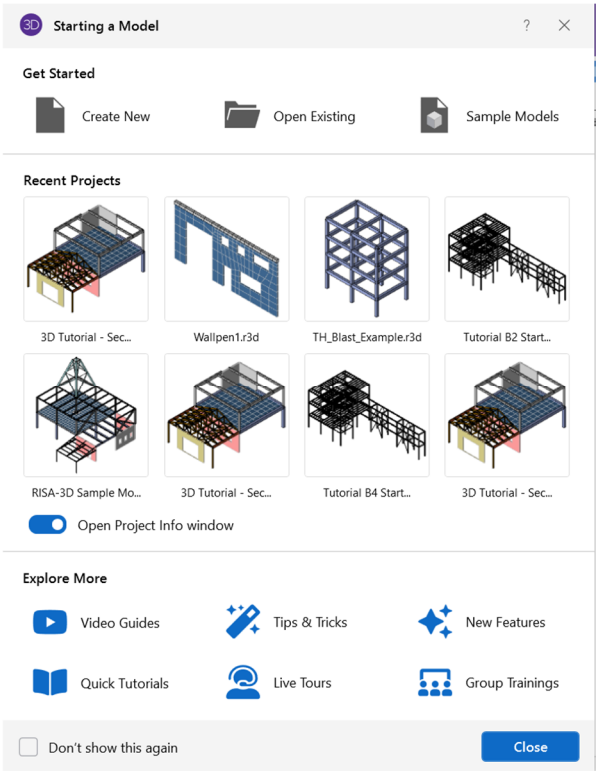
The model being created consists of a wide range of element types (members, wall panels, plates) as well as a variety of materials (wood, glulam, masonry, steel and concrete) and the majority of the elements will be created within the graphical interface using the drawing tools provided. However, it is also possible to create many of the same elements using the corresponding spreadsheet. Overall, this tutorial should take only a few hours and should help users get up to speed in the software.

Section 1: Modeling

Model Settings

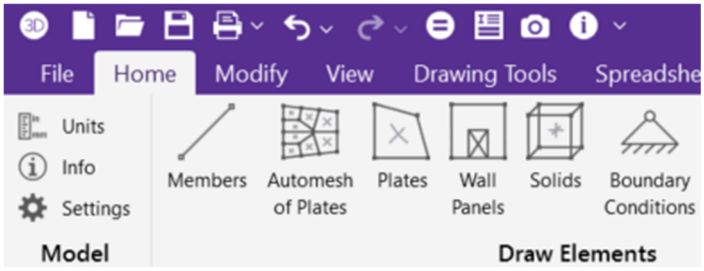
If you haven't already, launch RISA-3D by clicking on the shortcut icon on your desktop or by accessing the software from the **Windows Start Menu.**

When launching RISA-3D, the program will open the **Starting a Model** dialog which allows users to choose to start a **new** project, **open** a previous project or select a project from the **recent projects** list.



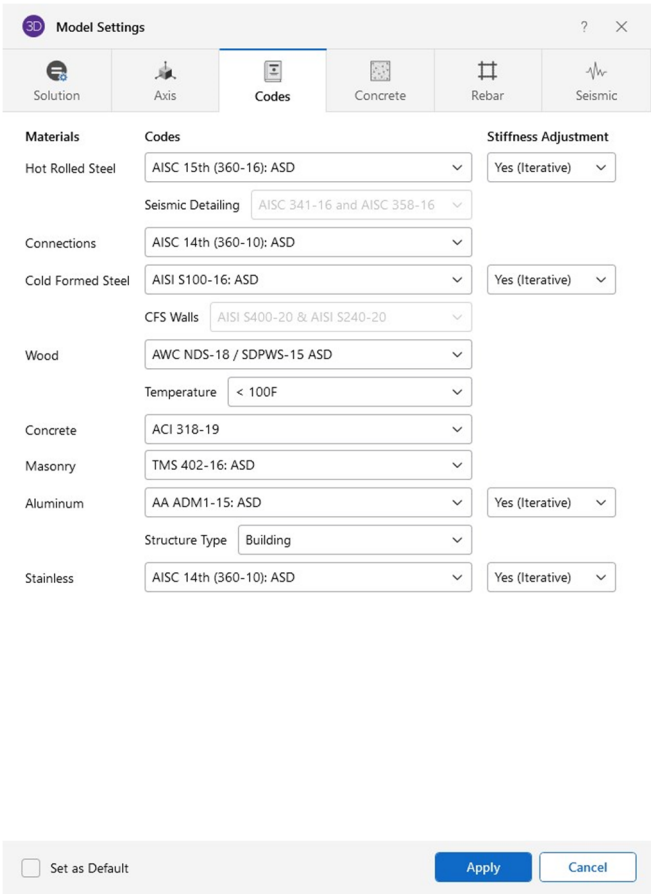
In this tutorial, we will choose the **Create New** model option which will open a blank workspace as well as the **Project Information** dialog box.

Before creating any model information, it is important to review the **Model Settings** of the project which include the applicable material design codes. To access the dialog, click the **Settings** button located on the **Home** tab.



The first tab within the model settings shows various setup options for the solution including approximate mesh size, number of reported sections and **Processor Core Utilization,** which allows users to select how the multiple cores of their computer will be utilized during the solution. RISA recommends using **Balanced** as this will allow load combinations to be solved in parallel while still reserving processing power for other applications on your computer.

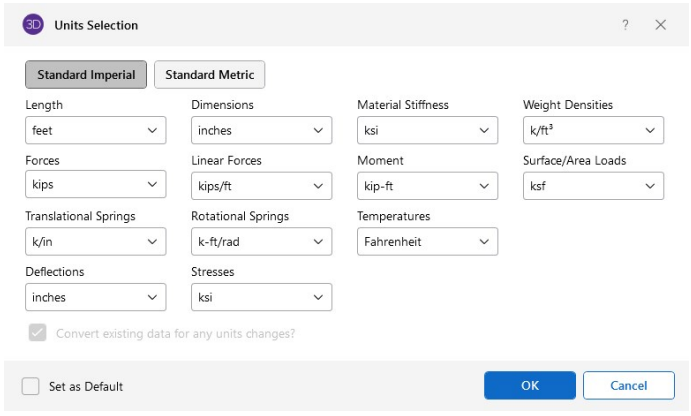
We can now enter the **Codes** tab and choose the design codes that are going to be used for the model. In this tutorial, we will utilize the ASD codes for every material except for concrete, where LRFD (strength based) design is required.



When the codes for each material are set as shown above, click **Apply** to exit the **Model Settings** dialog.

Units

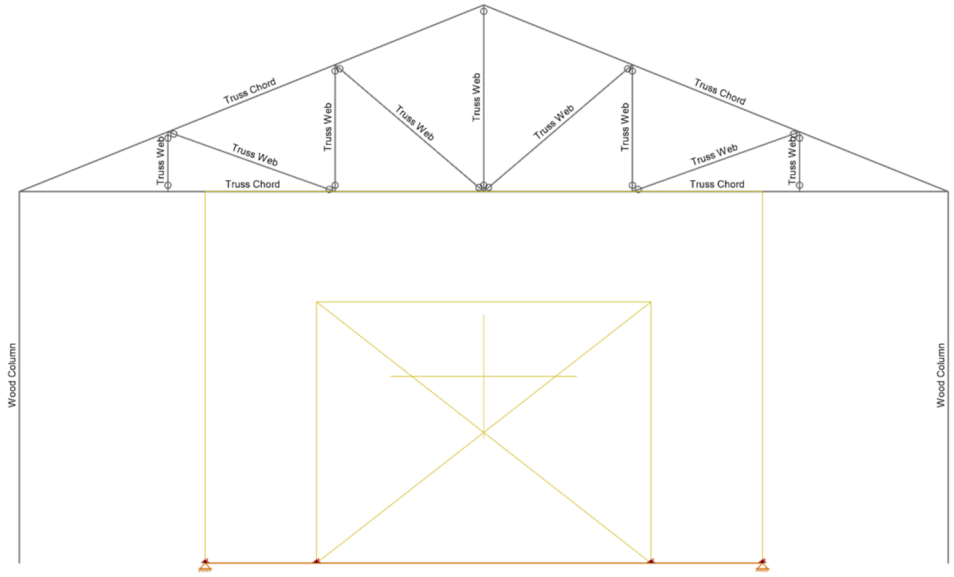
RISA-3D allows users to work in Imperial units (kips, inches, etc.), Metric units (kN, meters, etc.) or a combination of both. To access the current units setup, click the Units button found in the **Model** section of the **Home** tab. The **Units** dialog not only allows for the definition of units for specific measurements but also gives users the ability to change between the **Standard Imperial** setup and the **Standard Metric.**



Finally, it is also possible to **Save the Current Settings as Default** so that your specific units settings are remembered each time you start a new project.

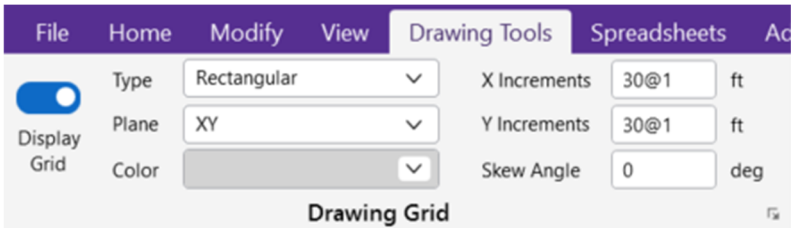
Creating the Model

With the code information and units set, let's begin modeling the structure by creating the frame, wood truss and wood wall panel at the far-left side of the model.

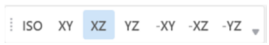


The easiest way to do this is by using the drawing grid.

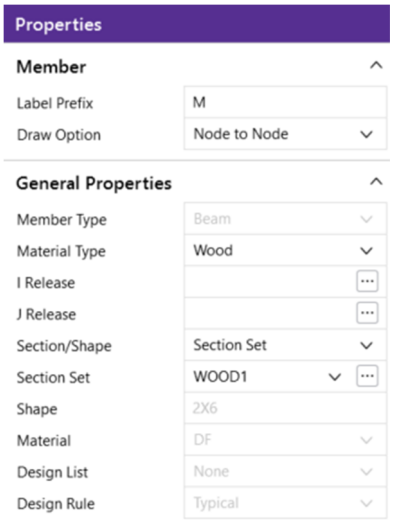
- Click on the **Drawing Tools** tab in the ribbon toolbar. In the section labeled, **Drawing Grid**, set up the grid as shown in the image below:



- Select the **XY** view by clicking on the button located on the **View Toolbar** found in the **3D View Window.**
- Select **Members** from the **Draw Elements** section of the **Home** tab to begin drawing the wood columns.

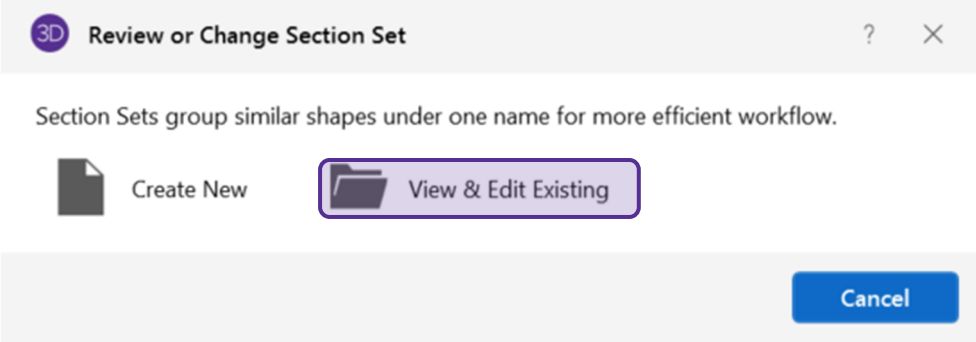


When the draw members tools is activated the **Properties Panel** allows the user to select all of the properties of the member to be drawn as well as the **Draw Option.** Before drawing the columns:



- Set the **Material Type** for the member to **Wood.**
- Select **Section Set** from the **Section/Shape** dropdown and then click on the ellipsis button ... to open the Section Set properties.

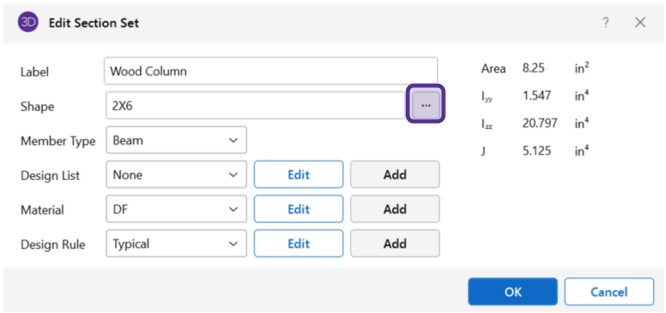
The next dialog will allow for the modification of an existing section set or the creation of a new section set. For this tutorial, let's click **View & Edit the Existing Section Set.**

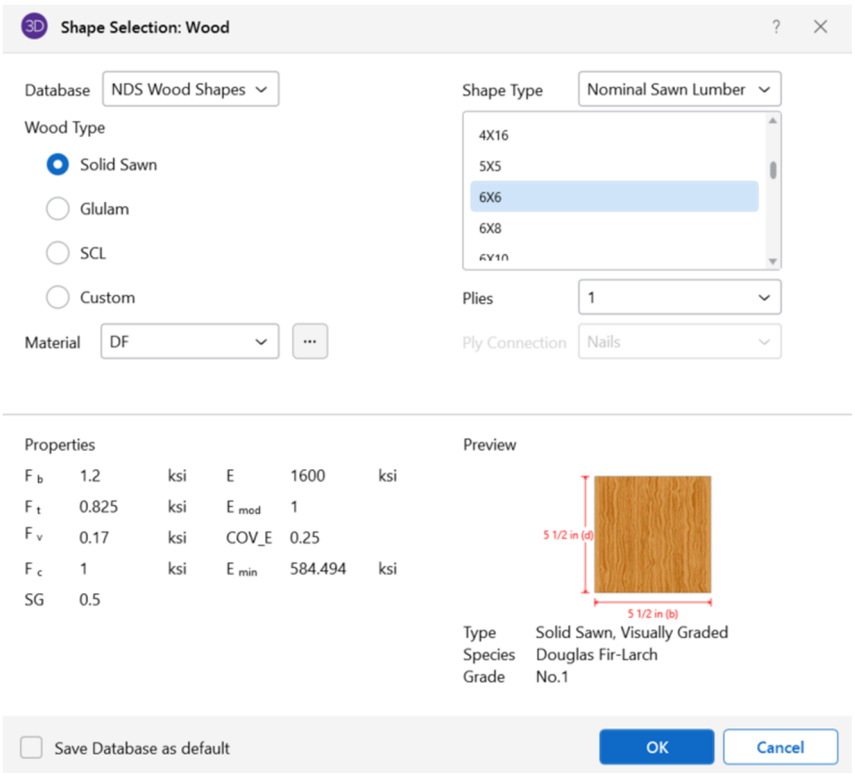


- Change the label to **Wood Column** and click the ellipsis button ··· next to **Shape** input to open the **Shape Selection** dialog.

Next, define the wood column with the following properties:

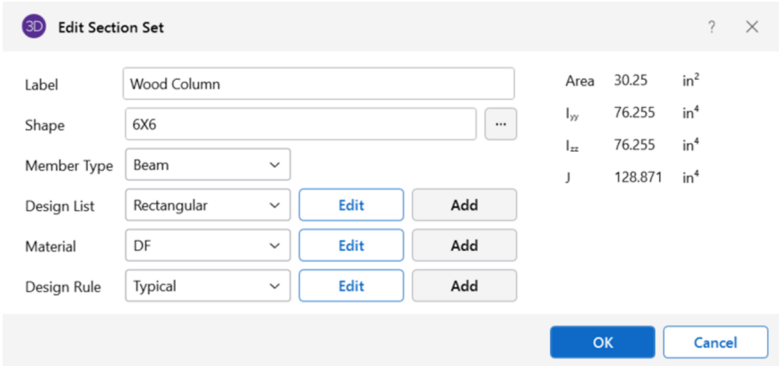
- Select the **NDS Wood Shapes** database
- Set the **Wood Type** as **Solid Sawn**
- Set the material to **DF**
- Set the **Shape Type** to **Nominal Sawn Lumber**
- Choose **6X6** and click **OK** to exit the dialog.





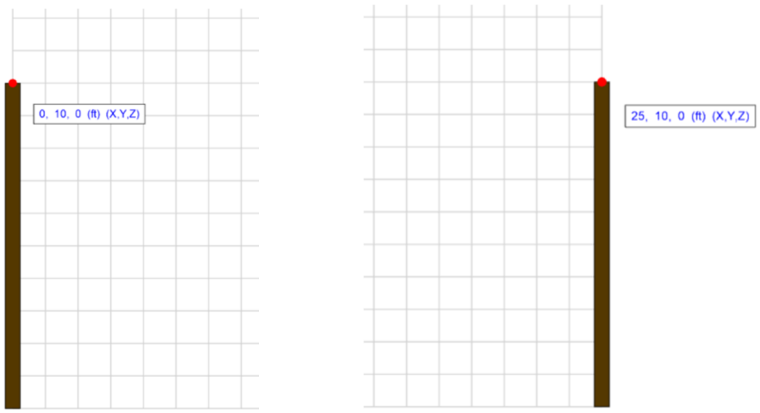
Note: The shape selection dialogue allows users to choose the **Database** (NDS or CSA), **Wood Type** (Solid Sawn, Glulam, SCL or Custom) as well as the **Shape Type** (Nominal Sawn Lumber, Net Rectangle or Net Round).

- Match the **Section Set** properties as shown in the image below and then click **OK** to return to the modeling of the column.



With the member properties set begin drawing the wood columns.

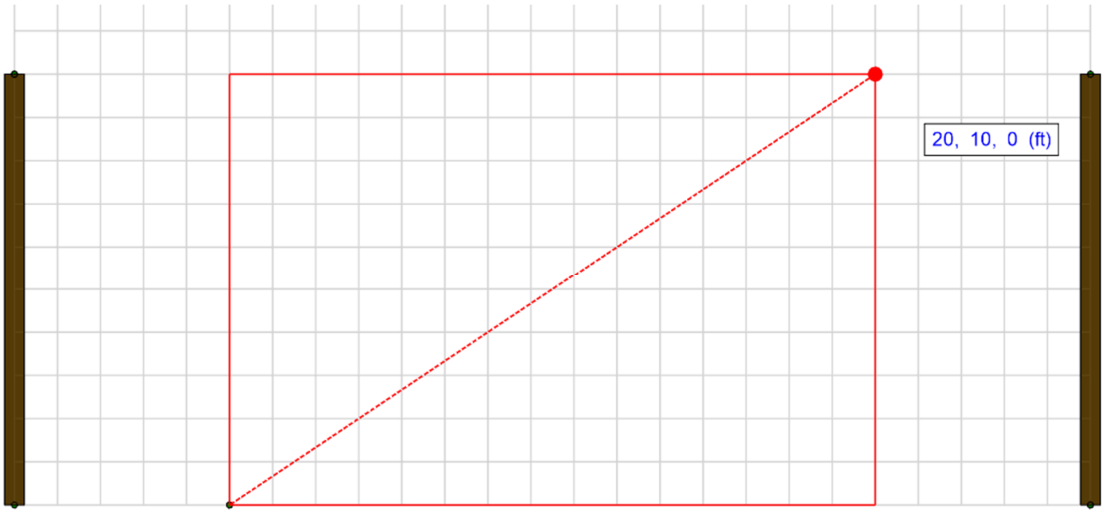
- Draw a line **10ft** by first clicking on the start point **(0,0,0)** and then clicking the end point **(0,10,0)**
- Right-click to "release" the mouse from the end point
- Draw a line **10ft** by first clicking on the start point **(25,0,0)** and then clicking the end point **(25,10,0)**
- Right-click or press **ESC** to end the draw members command.



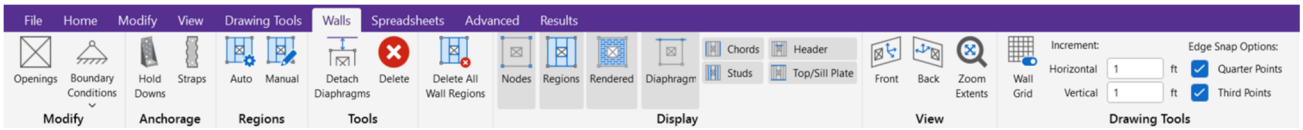
In the same view and using the same drawing grid, the wood wall panel can be created. Select **Wall Panels** from the **Draw Elements** section of the **Home** tab. Again, the **Properties Panel** changes to allow the user to input the required information for the modeling of a wall panel including the panel thickness, material and design rule.

To begin drawing the wall panel, set the **Wall Material** as **Wood** and make sure the **Material Set** is **DF.**

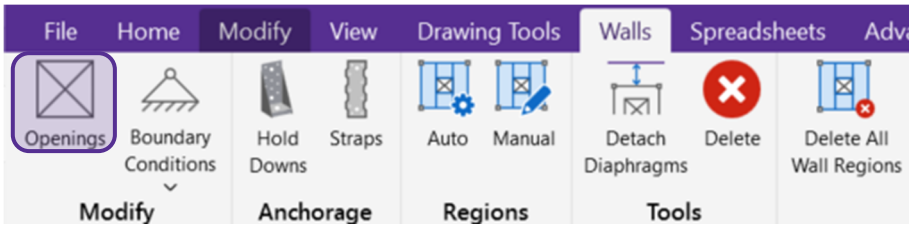
- Click the point that defines the bottom left corner of the wall panel **(5,0,0)**
- Drag your mouse to the point that defines the top right corner of the wall panel **(20,10,0)** and click again to complete drawing the wall panel



With the wall panel added, it is also possible to add openings to the panel. To do so, first click “Escape” on your keyboard and then double click on the wall panel to open the **Wall Panel Editor.** When the editor is open, a new tab is added to the ribbon toolbar for the modification of the wall.

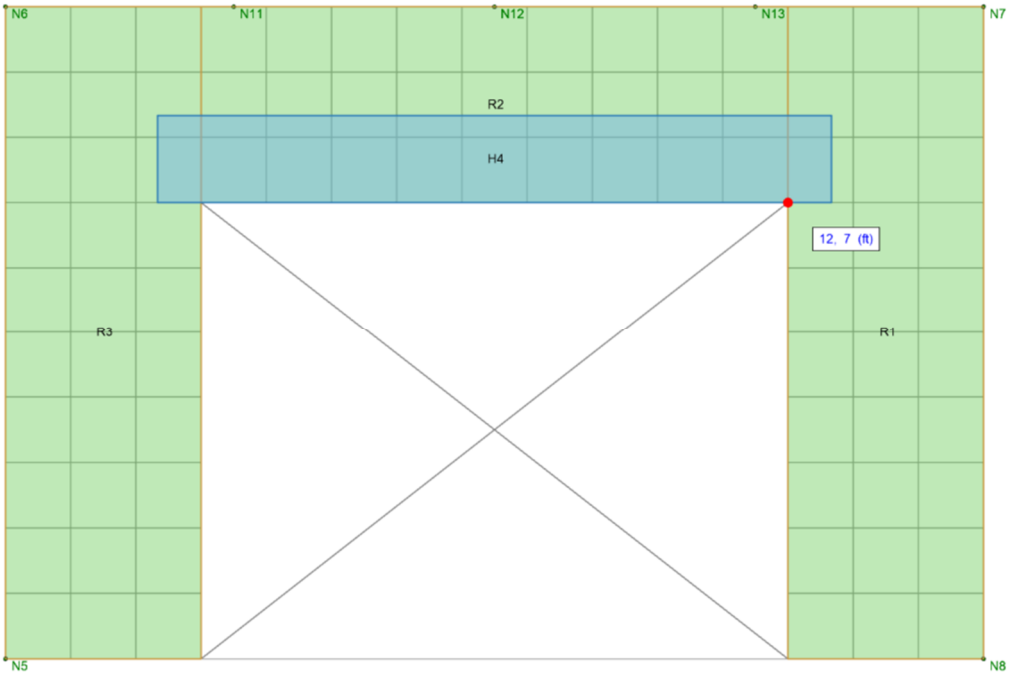


The toolbar includes sections for adding openings, anchorage and regions as well as display properties and wall panel specific drawing tools. For this tutorial, we are going to add a simple door opening by clicking on the **Opening** button.



- Click the point that defines the bottom left corner of the opening **(3,0)**
- Drag your mouse to the point that defines the top right corner of the wall panel **(12,7)** and click again to complete drawing the opening

When the opening is added, the lintel above the opening is automatically added. After adding the opening, select the **Auto** button from the **Regions** section to create the wall panel regions.



Next, let's add a boundary condition at the base of the wall.

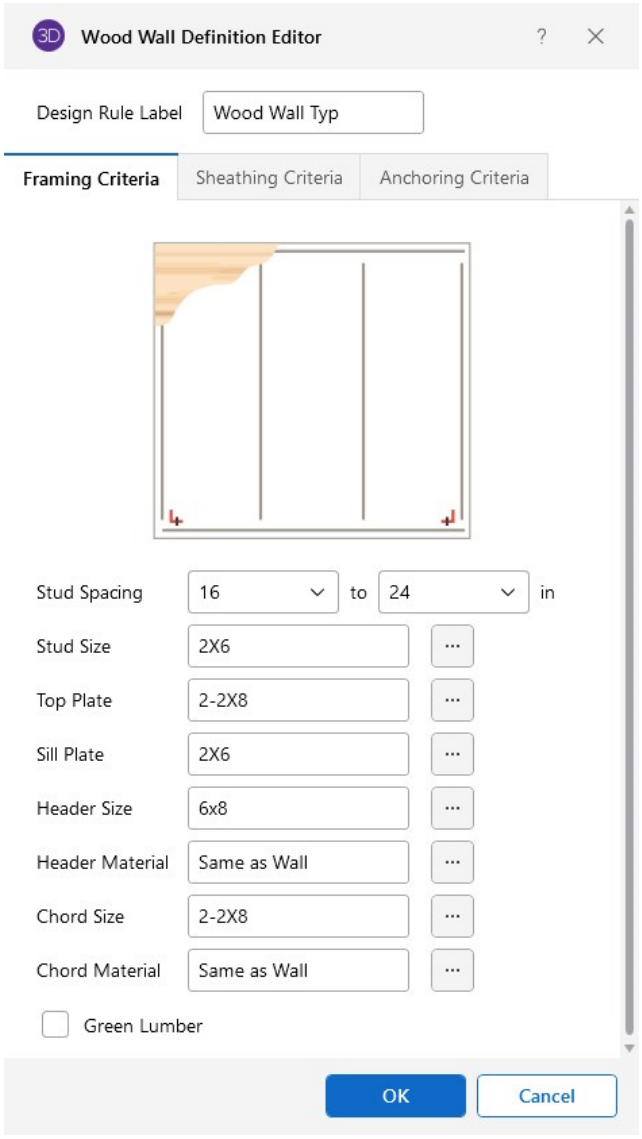
- Select the **Boundary Conditions** button from the **Modify** section.
- Select **Pinned**
- Click on one of the nodes at the base of the wall to add the pinned boundary condition

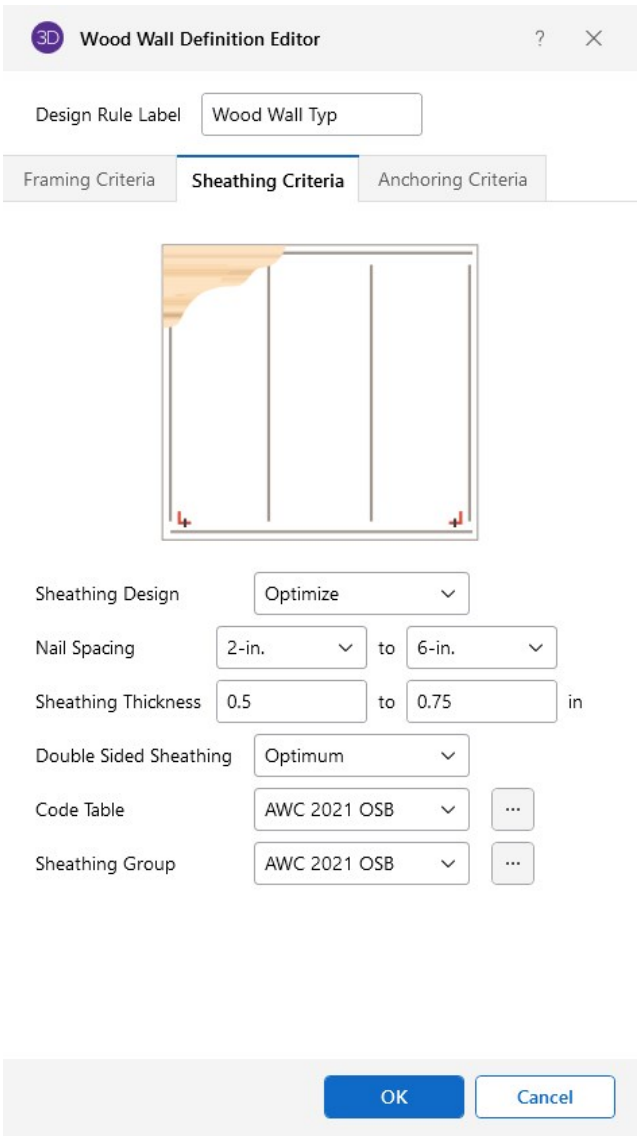
When the opening has been added and the regions are created, click the "x" to close the **Wall Panel Editor.**

Note: RISA-3D includes three different design methods for wood wall panels; Segmented, Force Transfer Around Opening and Perforated. For more information about these design methods and to determine which is best for a specific situation, visit the Wood Walls – Design section of the RISA-3D Web Help.

With the wood panel modeled, the **Design Rule** for the element can also be evaluated. To access the **Wood Panel** design rule, click on the ellipsis button ... next to **Design Rule** in the properties of the wood wall panel.

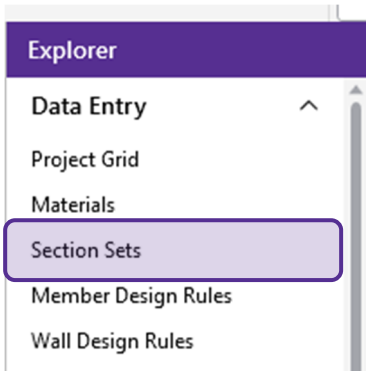
- Click **Create New** to create a new wall design rule.
- Input **Wood Wall Typ** as the **Design Rule Label**
- Set the Wood Wall Panel design rule as shown in the images on the next page.
- Once the properties are set, click **OK** to exit the dialog.



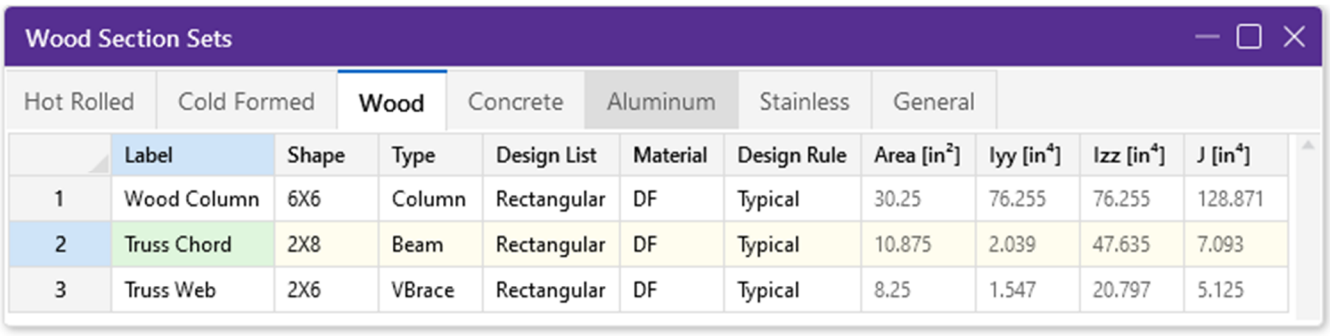


In RISA-3D, design rules are used to set the limitations of the design for a specific type of element. For wood wall panels, the **Framing Criteria, Sheathing Criteria,** and **Anchoring Criteria** can all be modified to fit the project/design needs.

With the columns and wall panel modeled, it is possible to now add the wood truss. Before doing this, the **Section Sets** for both the truss chords and webs can be created by accessing the **Section Sets** spreadsheet directly from the **Explorer Panel.**

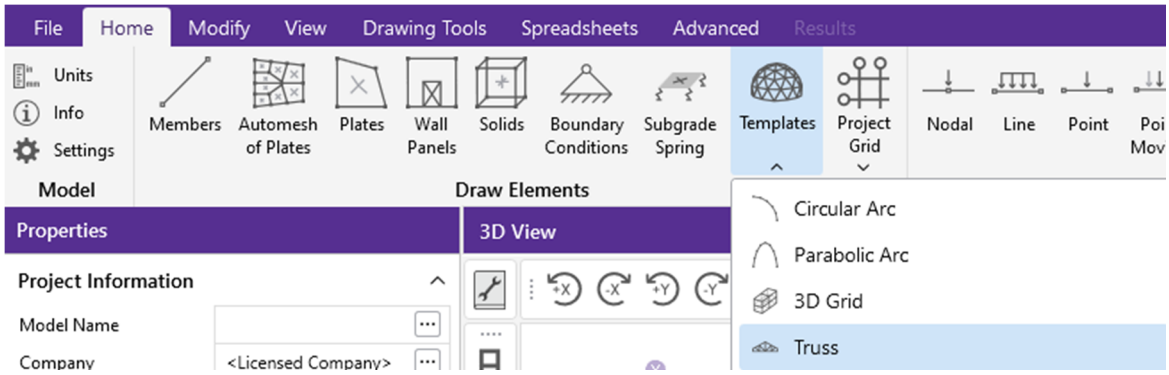


Within the **Wood** tab of the spreadsheet, add two new rows for the **Truss Chord** and **Truss Web**, with each Section Set having the properties shown in the image below:



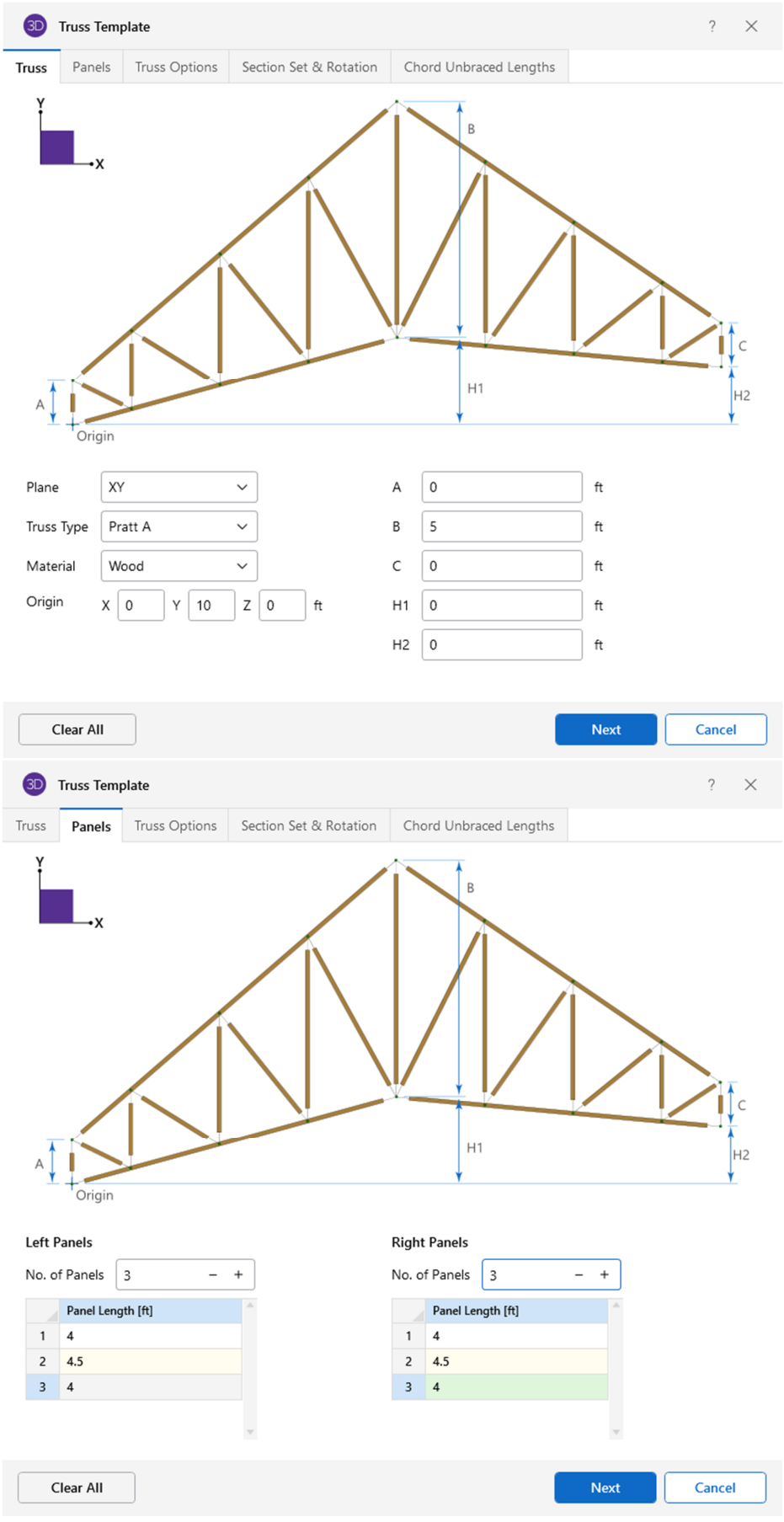
	Label	Shape	Type	Design List	Material	Design Rule	Area [in²]	Iyy [in⁴]	Izz [in⁴]	J [in⁴]
1	Wood Column	6X6	Column	Rectangular	DF	Typical	30.25	76.255	76.255	128.871
2	Truss Chord	2X8	Beam	Rectangular	DF	Typical	10.875	2.039	47.635	7.093
3	Truss Web	2X6	VBrace	Rectangular	DF	Typical	8.25	1.547	20.797	5.125

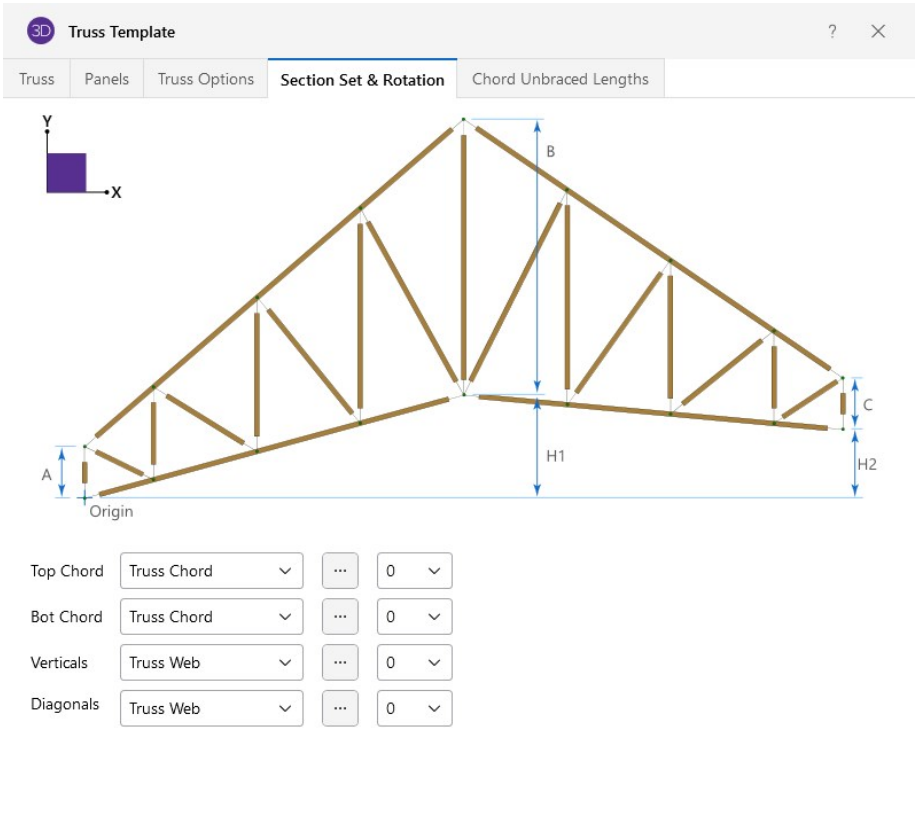
Now that the section sets for the truss have been created, the truss can be modeled manually like any other element in RISA-3D. However, RISA-3D gives you a quick and easy way to create "systems" based on predefined templates. To access the template, click the **Templates** button found in the **Draw Elements** section of the **Home** tab and select **Truss.**

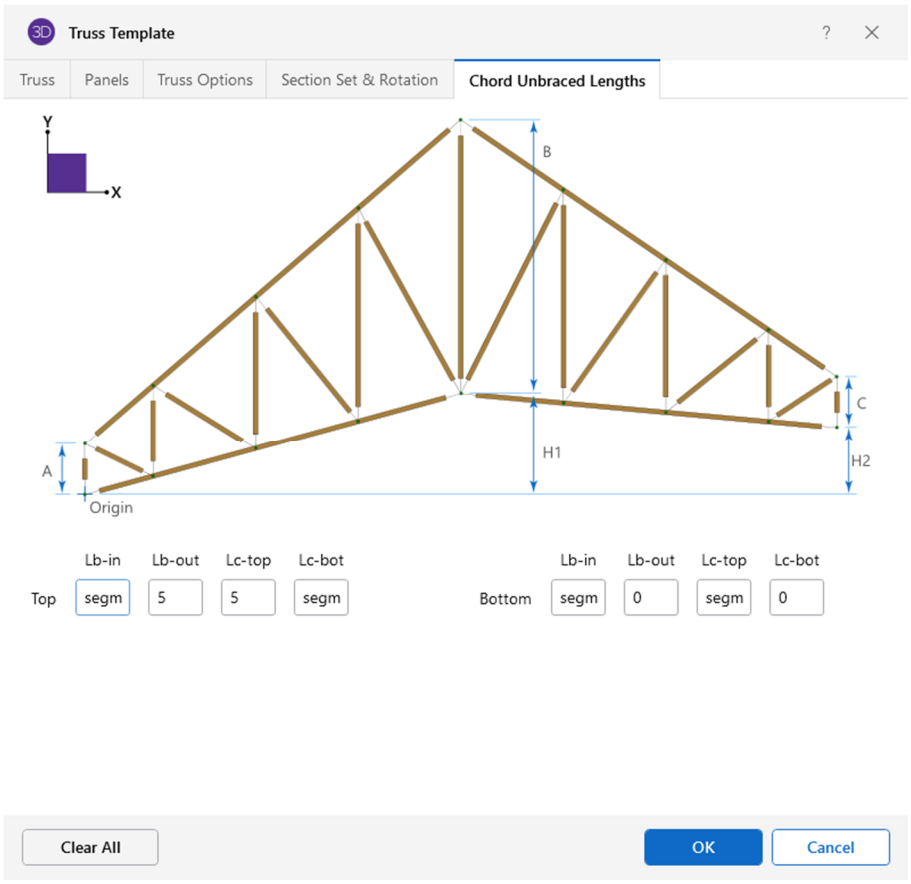


The **Truss Template** dialog consists of all the inputs necessary to build, position and design the truss system. Inputs include the types of truss, number of panels, members and unbraced length properties.

For the tutorial, a **Wood** truss with type **Pratt A** will be created in the **XY Plane**. The truss properties can be input as shown in the images on the following pages:


Truss Template
Truss
Panels
Truss Options
Section Set & Rotation
Chord Unbraced Lengths
Y
X
B
C
H1
H2
A
Origin
Plane
XY
Truss Type
Pratt A
Material
Wood
Origin
X 0 Y 10 Z 0 ft
A 0 ft
B 5 ft
C 0 ft
H1 0 ft
H2 0 ft
Clear All
Next
Cancel
Truss Template
Truss
Panels
Truss Options
Section Set & Rotation
Chord Unbraced Lengths
Y
X
B
C
H1
H2
A
Origin
Left Panels
No. of Panels 3
Panel Length [ft]
1 4
2 4.5
3 4
Right Panels
No. of Panels 3
Panel Length [ft]
1 4
2 4.5
3 4
Clear All
Next
Cancel


Truss Template
Truss
Panels
Truss Options
Section Set & Rotation
Chord Unbraced Lengths
Y
X
B
C
H1
H2
A
Origin
Top Chord
Truss Chord
0
Bot Chord
Truss Chord
0
Verticals
Truss Web
0
Diagonals
Truss Web
0
Clear All
Next
Cancel


Truss Template
Truss
Panels
Truss Options
Section Set & Rotation
Chord Unbraced Lengths
Y
X
B
C
H1
H2
A
Origin
Lb-in
Lb-out
Lc-top
Lc-bot
Top
segm
5
5
segm
Bottom
segm
0
segm
0
Clear All
OK
Cancel

Note: The input of the **unbraced lengths** for the chord members is important for the overall design of the members. Within the truss template, various unbraced lengths can be defined. These include:

- Lb-in (Top/Bot): Unbraced length for in-plane bending
- Lb-out (Top/Bot): Unbraced length for out-of-plane bending
- Lb-top (Top/Bot): Unbraced length for bracing against Lateral Torsional Buckling @ top or bottom flange of member
- Lb-bot (Top/Bot): Unbraced length for bracing against Lateral Torsional Buckling @ top or bottom flange of member

Additionally, the **Segment** command can be used when the unbraced length is defined by the location of brace points which are created when nodes are created along the length of a member. In the case of a truss, it's where the web members frame into the chord members. For more information about unbraced lengths in RISA-3D, visit the Unbraced Lengths section of the RISA-3D Web Help.

When the properties are input, click **OK** to create the truss. A separate window will open showing the location of the created truss. Close this window.

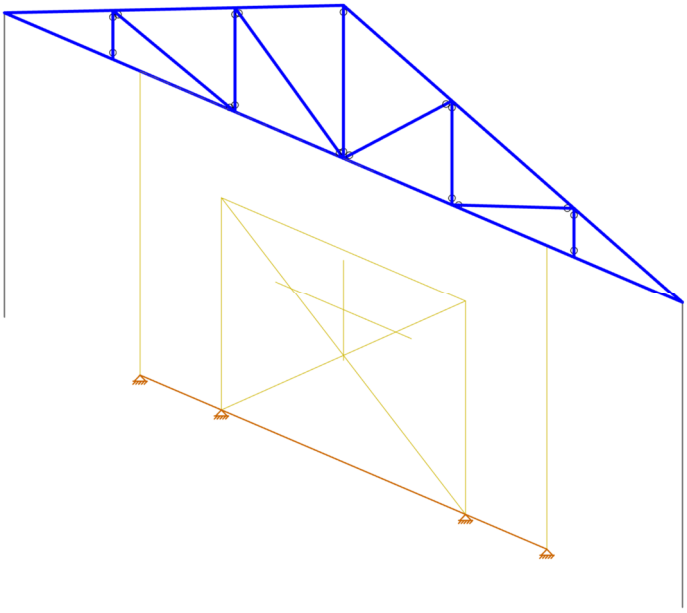
With the truss created, the rest of the gable roof structure can be created by making multiple copies of the truss. We first need to select only the truss members (chords and webs) to be copied.

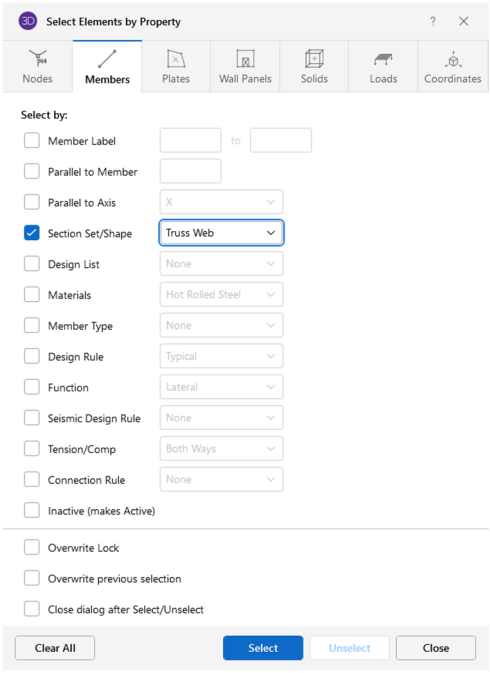
To do this, we can use the **Select by Property** 

button found on the **Home** tab.

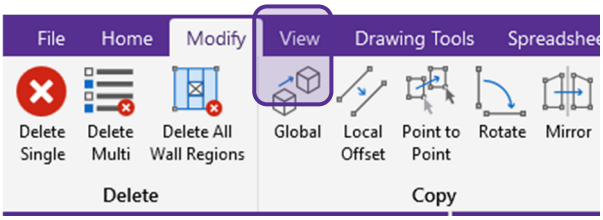
The **Select Elements by Property** tool allows users to specify properties specific to nodes, members, plates, wall panels, solids or loads in order to create a more "refined" selection. This makes it easier to modify or review a specific part of the model.

- With the dialog open, select the **Members** tab.
- Click on the checkbox for **Section Set/Shape**
- Choose **Truss Chord** from the dropdown and click **Select**
- Choose **Truss Web** from the dropdown, click **Select** and then click **Close.**





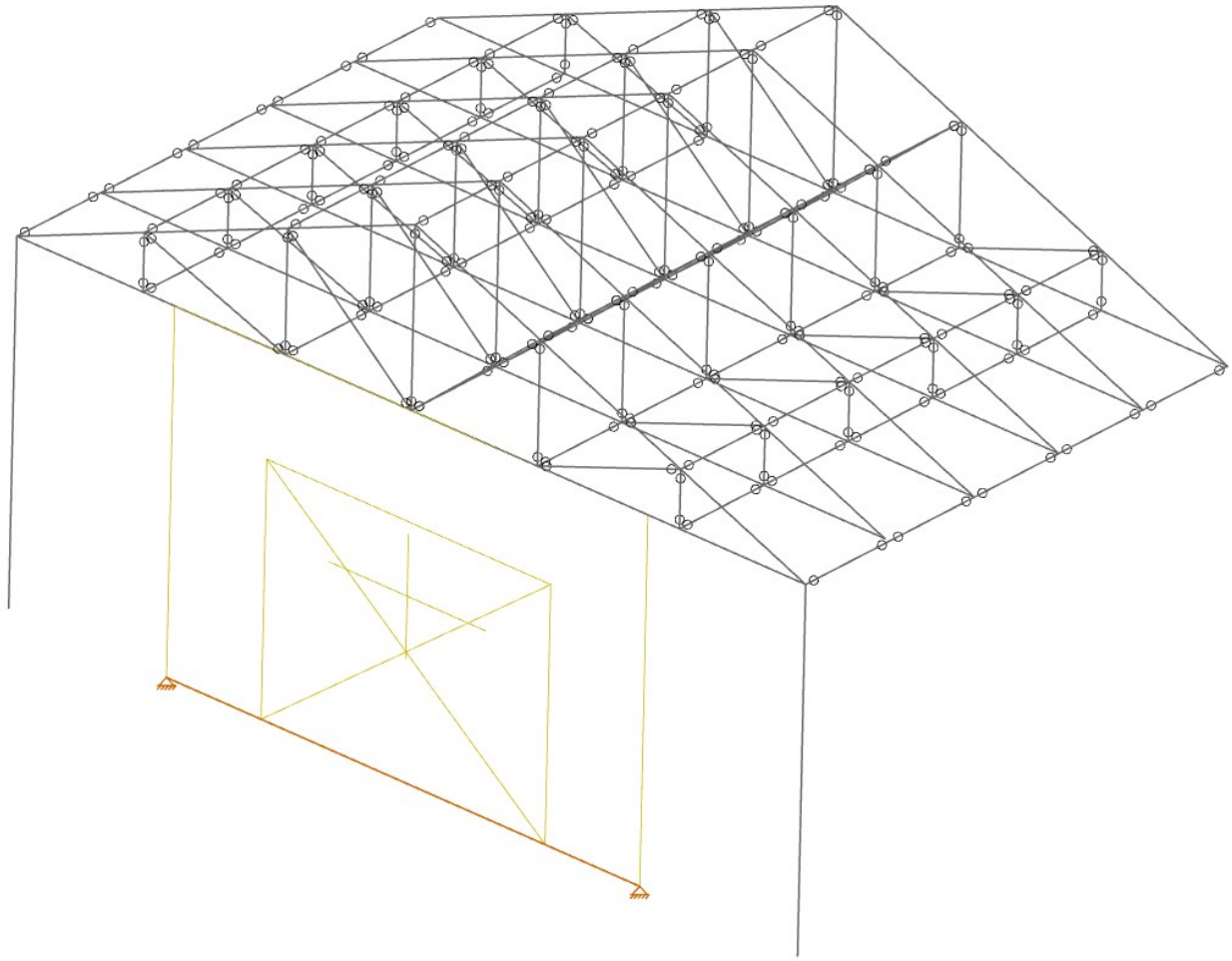
With the truss selected, click the **Modify** tab on the ribbon toolbar and in the **Copy** section, select **Global.**



The **Properties Panel** changes to show the options for the global copy. In this model, we want to create multiple copies in a specified direction. To do this:

- Input **5@-3** in the **Z Increment** box.
- Click the checkbox for **Connect Bays**
- Set the **Bay Fixity** property to **Pinned End Struts**
- Click **Apply to Selected**
- Right click or press **ESC** to end the command

Your model should now look like this:



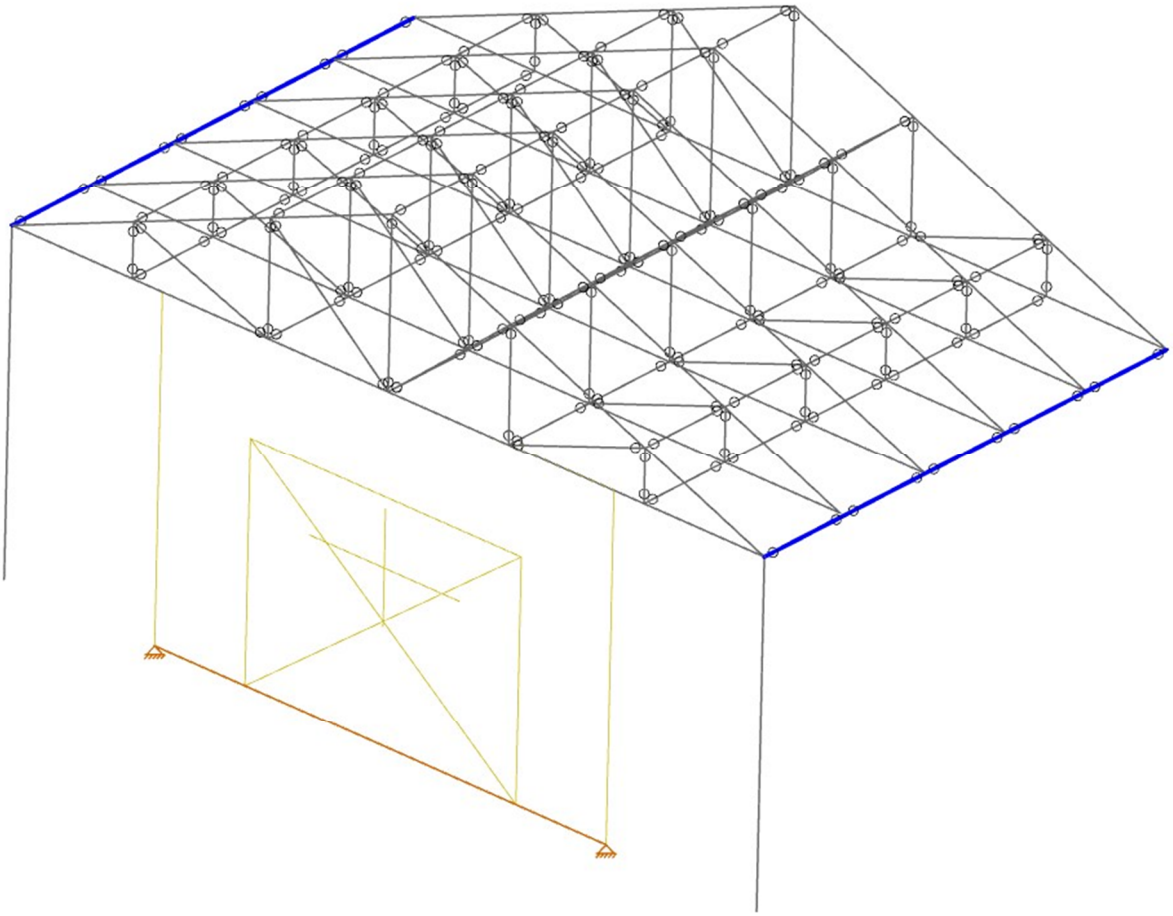
Before moving on, we need to make some changes to how the trusses are connected. Let's first change all the struts that connect the trusses together:

- Click the **Select by Property** button found on the **Home** tab.
- Select the **Members** tab
- Enable the checkbox for **Section Set/Shape** and select **W8x10** from the dropdown
- Click **Select** and then click **Close**

With all the struts selected, we can change the properties of the members in the **Properties Panel:**

- Click on the ellipsis button ... in the **Shape** cell to open the **Shape Selection** dialog.
- Click on the **Wood** tab and under **Shape Type** select **Nominal Sawn Lumber**
- Choose **2x4** from the list
- Click **OK** to exit the dialog

Next, we will delete the struts at the extreme ends of the trusses. Select the members shown in the following image and hit the delete key to remove the end struts.

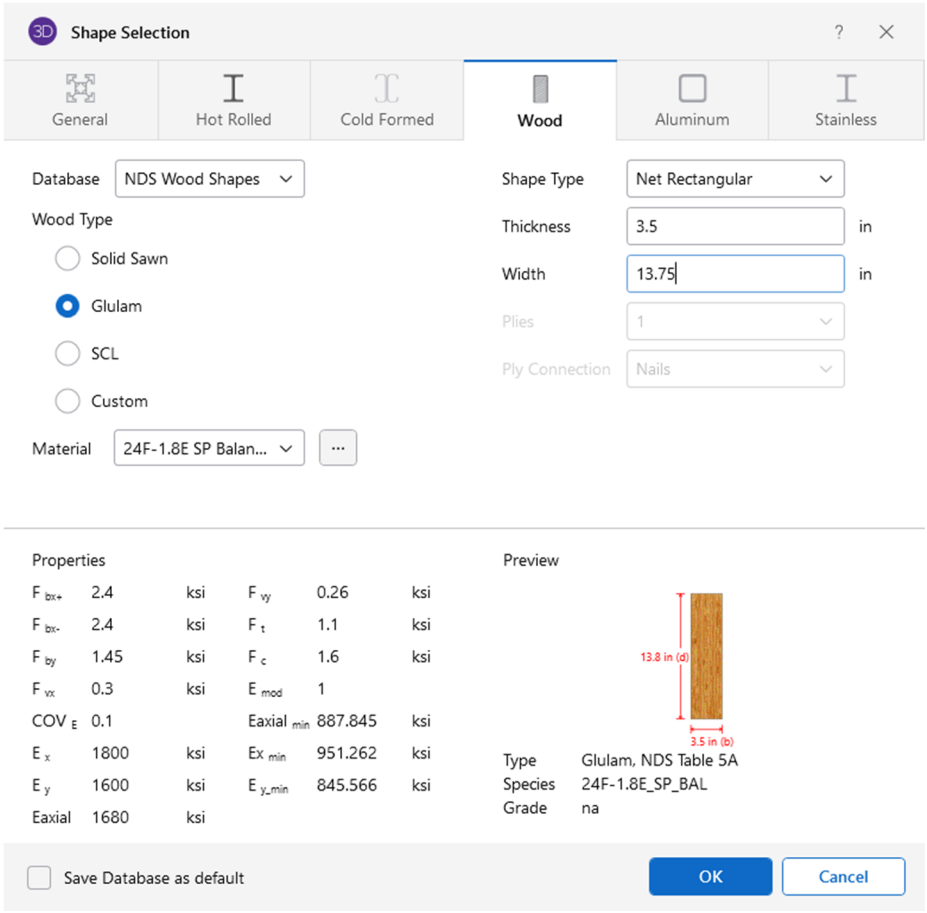


Now we can model the glulam beams at each roof eave. Before doing this, the **Glulam Beam** section set needs to be created:

- Click on the **Members** button on the **Home** tab
- Set the **Material Type** to **Wood**
- Select **Section Set** from the **Section/Shape** dropdown and click on the ellipsis button ... next to the **Section Set** input field

The next dialog will allow for the modification of an existing section set or the creation of a new section set. For this tutorial, let's click **Create New** to create a new section set with the following properties:

- Set the **Label** to **Glulam Beam**
- Click on the ellipsis button ... in the **Shape** cell to open the **Shape Selection** dialog.



- Select the **NDS Wood Shapes** database
- Set the **Wood Type** as **Glulam**
- Set the material to **24F-1.8E SP Balanced**
- Set the **Shape Type** to **Net Rectangular**
- Input **3.5"** as the thickness and **13.75"** as the width, then click **OK** to exit the dialog.
- Set the **Member Type** to **Beam**
- Set the **Design List** to **Glulam_SouthernPine**
- Click **OK** to finish the creation of the Section Set

The glulam beams can now be modeled at each of the roof eaves.

- Draw a member from left eave of first truss **(0,10,0)** to the left eave of the last truss **(0,10,-15)**
- Right click to release the cursor
- Draw a member from right eave of first truss **(25,10,0)** to the right eave of the last truss **(25,10,-15)**
- Right click or press **ESC** to end the command and press **ESC** again to exit the **Drawing Tools**

Before finishing the modeling of the truss, let's make some changes to the properties of the eave members that were just added.

- Hold the **CTRL** key and click the two eve members to “multi-select”.
- Expand the section for **Design Properties**
- Type **Segment** for the value of **le2**
- Press **ESC** to clear the selection

With the entire truss system created, it may be valuable to save a **Selection State.** Selection states make it easy to recall a group of selected items (nodes, members, plates, etc.). In this example, let’s create a selection state for the entire truss system.

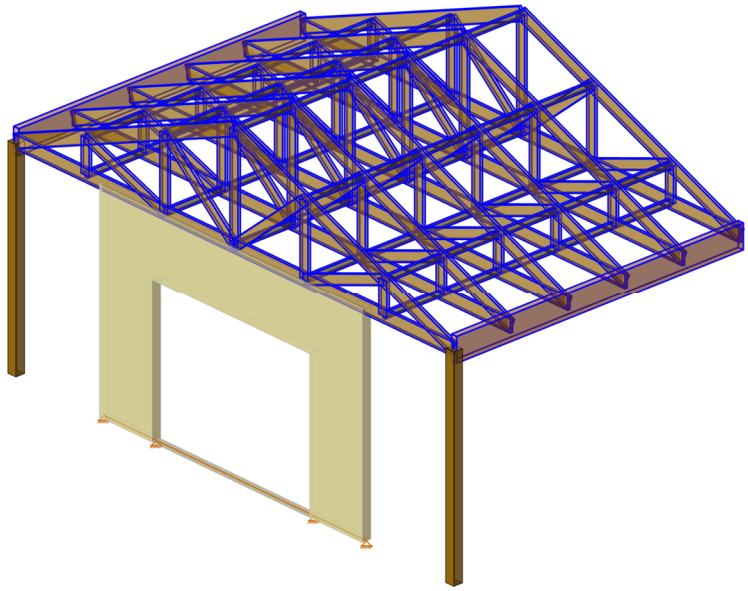
- To do this, we can use the **Select Elements by Property** 

button found on the **Home** tab.
- Select the **Members** tab
- Enable the checkbox for **Section Set/Shape**
    - Choose **2x4** and click **Select**
    - Choose **Truss Chord** and click **Select**
    - Choose **Truss Web** and click **Select**
    - Choose **Glulam Beam** and click **Select**
- Click **Close** to exit the dialog

With the members selected, click on the **Saved Selections** button  found on the **Selection Toolbar.**

- Click **Save Current** and enter the description for the selection state as **Wood Truss** and click **Save.**
- Click the **Close** button to exit the **Saved Selections** dialog.

With the selection state now saved, the selection can be recalled by first clicking the **Save and Recall Selection States** button and then selecting **Wood Truss** and clicking **Retrieve.**



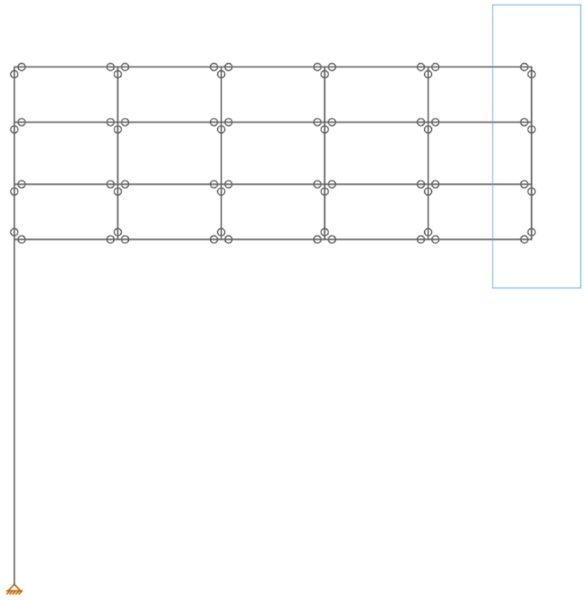
It is now time to model the masonry wall that supports the last truss as well as the end of the glulam beams. Before creating the wall, click the **YZ** button on the **View Toolbar** in the **3D View Window** so that only the last truss can be easily selected.

With only the last truss selected click the **Lock** button from the **Selection Toolbar** found in the **3D View Window.** Doing this will "hide" the parts of the model that were not selected.

After selecting the truss, click the **ISO** button on the **View Toolbar** in the **3D View Window.**

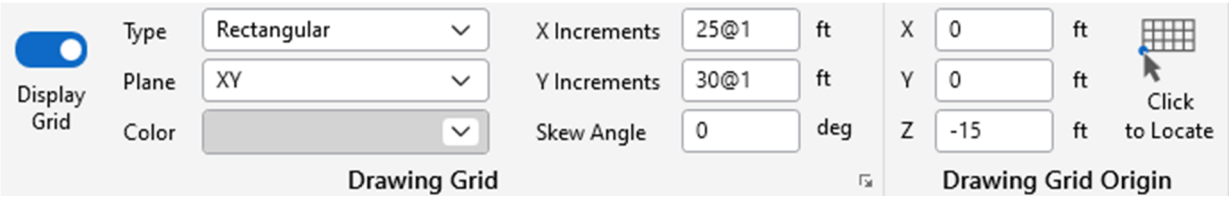
To start drawing the masonry wall panel:

- Click on the **Wall Panel** button found in the **Draw Elements** section of the **Home** tab.



Note: When selecting members, drag from **left to right** to select all objects that are entirely enclosed in the selection rectangle (**window selection**). Drag from **right to left** to select all objects that are crossed by the selection rectangle (**crossing selection**).

Let's enable the **Drawing Grid** and set the grid as shown in the image below:



With the drawing grid visible, let's first evaluate the **Design Rule** for the masonry wall.

- Set the **Wall Material** as **Masonry**
- Click on the ellipsis button ... located in the **Design Rule** row.
- When the dialog opens, click **Create New** to create a new Wall Design Rule

The **Masonry Wall Definition Editor** will open, and the **Design Rule Label** can be set to **Masonry Wall Typ**. Within this interface the wall type (reinforced or unreinforced), block thickness, rebar size and spacing as well as details for the lintel can be modified. For this example, let's review one of the properties:

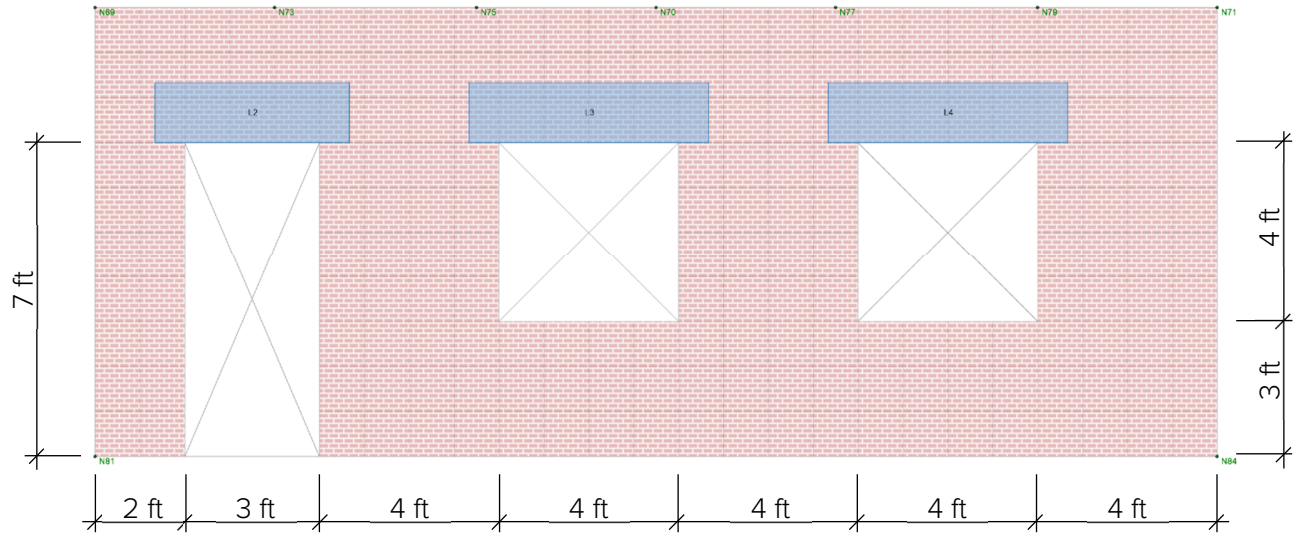
- Verify the **Block Thickness** is set to **8"** and click **OK.**

Now the rest of the wall properties can be set, and the wall can be modeled:

- Set the **Material Set** to **Concrete Matl**
- Click the point that defines the bottom left corner of the wall panel **(0,0,-15)**
- Drag your mouse to the point that defines the top right corner of the wall panel and click again **(25,10,-15)**
- Press **ESC** to end the command

With the wall created, let's now add a door opening and two window openings to the masonry wall:

- Double click on the masonry wall panel to open the **Wall Panel Editor**
- Click on the **Openings** button to add the door and window openings as shown in the image below:

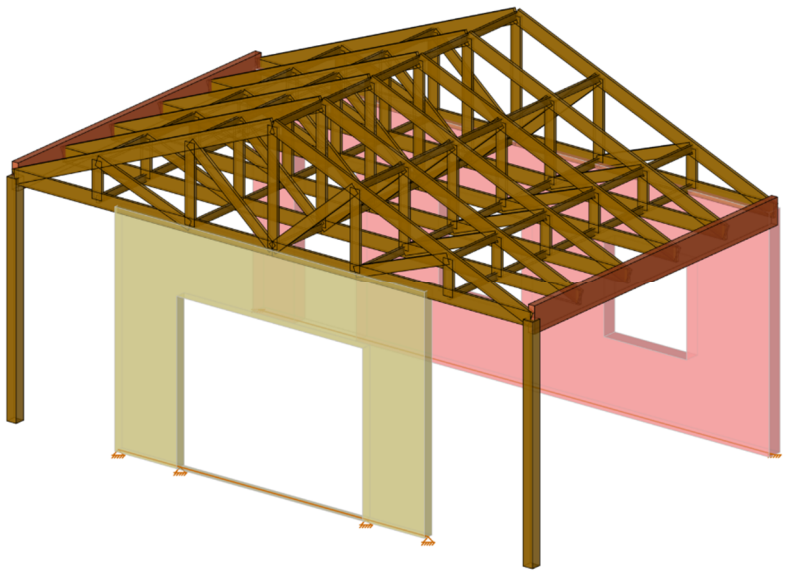


- Click on the **Boundary Conditions** button and select **Fixed**
- Click on any of the nodes at the base of the wall to apply the fixed boundary condition
- Click the **Auto** button in the **Regions** section to create the wall panel regions
- Close the wall panel editor

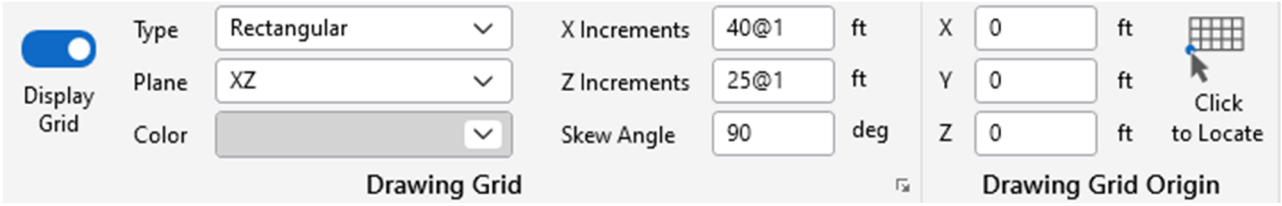
With only the masonry wall panel selected, click the **Turn OFF the Unselection Lock** button from the **Selection Toolbar** found in the **3D View Window.** Doing this will show the entire model.

In the **Quick View** section of the **Home** tab, click on the rendering button to show the rendered view of the model.

With the wood and masonry portion of the model complete, it is time to move on to modeling the steel and concrete two-story building that completes the structure.

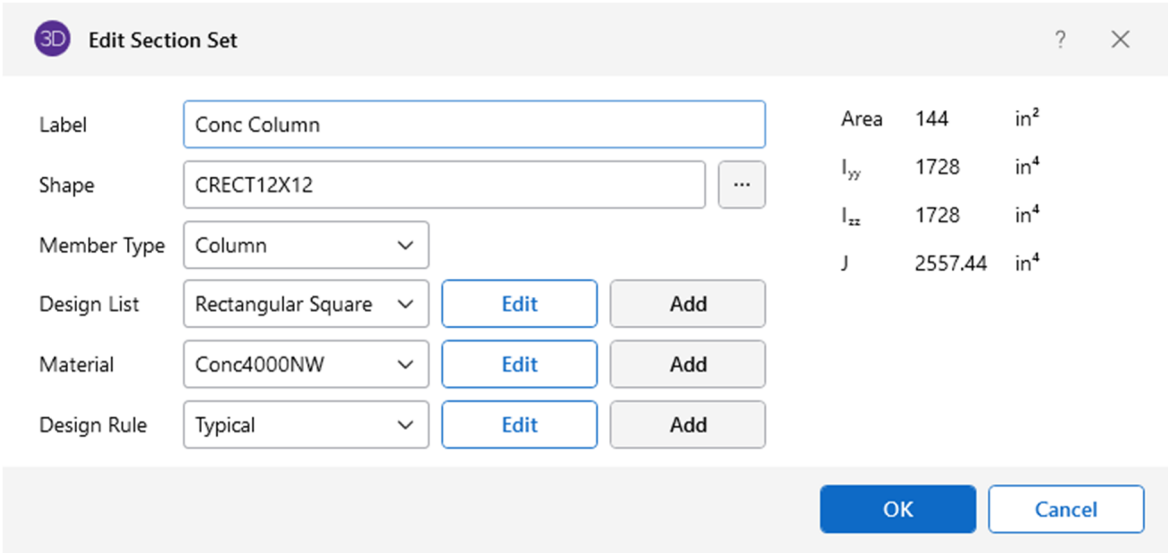


Let's start by creating the **Drawing Grid** by clicking on the **Drawing Tools** tab. Set up the **Drawing Grid** as shown in the image below:



With the grid set, let's specify the properties of the concrete column.

- On the **Home** tab, select **Members** to open the draw members tool.
- Change the **Material Type** to **Concrete**
- Select **Section Set** from the **Section/Shape** dropdown
- Click on the ellipsis button ... located in the **Section Set** row and click to **Edit/View the Existing Section Set**
- Create the **Conc Column** section set with the properties shown in the image below:

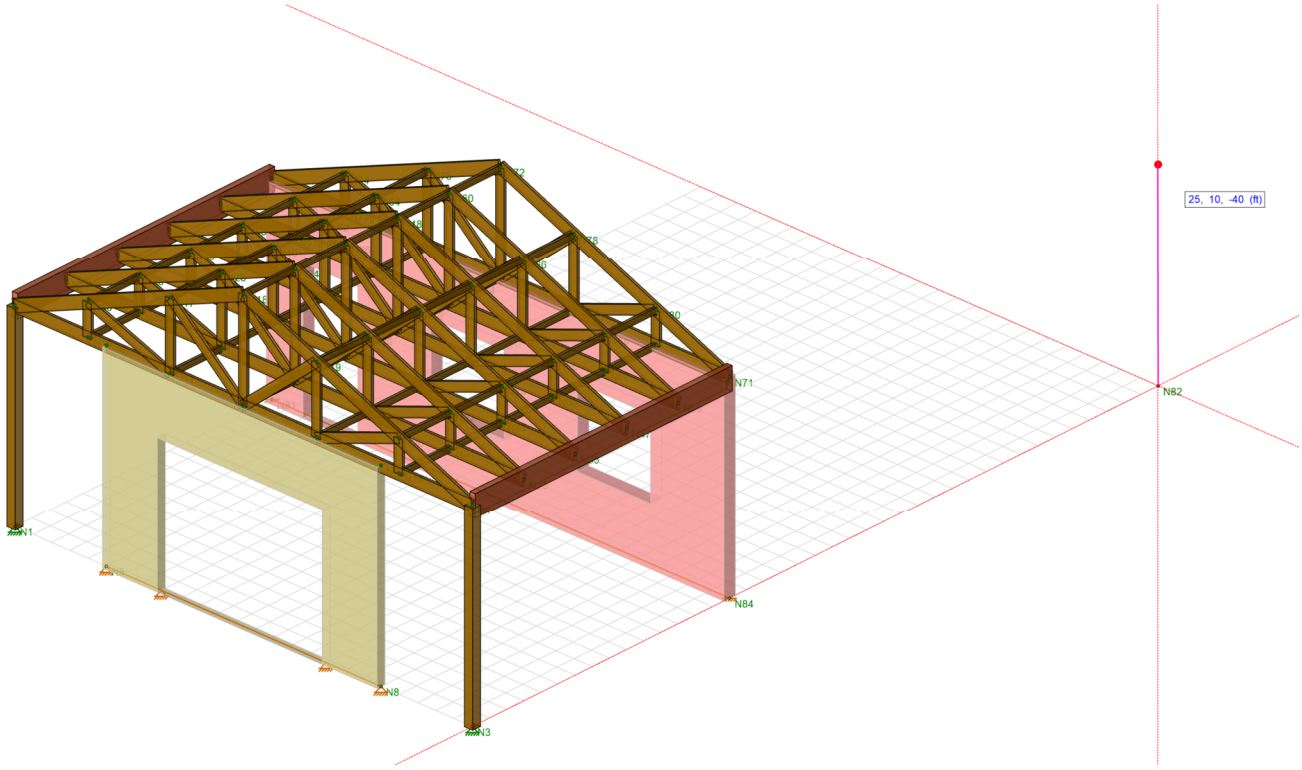


- Click **OK** to finish the creation of the section set.

Now to draw the column:

- Click the column start point **(25,0,-40)**
- On the **Drawing Tools** tab, click the **Lock to Global Axis** button 

to "lock" the drawing to the global axes (X,Y,Z) similar to the "Ortho" command in AutoCAD.
- Move the cursor **10ft** vertically (along the Y axis) to the column end point **(25,10,-40)**
- Right click or press **ESC** to end the command



Let's now use the **Data Entry** spreadsheets to help create the concrete walls that support the floor and roof framing.

- Click **Node Coordinates** on the **Explorer Panel** to open the node coordinates spreadsheet
- Scroll to the bottom of the spreadsheet and press **Enter** to add a new row
- Add two nodes with the coordinate information shown in the image below (your labels may be slightly different):

Node Coordinates

	Label	X [ft]	Y [ft]	Z [ft]	Temp [F]	Detach From Diaphragm
83	N86	0	0	-40		☐
84	N87	10	20	-40		☐

With the node coordinates entered, click on the **Wall Panel** button found in the **Draw Elements** section of the **Home** tab to create concrete walls with the following properties:

- Set the **Wall Material** to **Concrete**
- Set the **Thickness** to **8".**
- Set the **Material Set** to **Conc4000NW**
- Click on the ellipsis button ... located in the **Design Rule** row.
- When the dialog opens, click **Create a New Wall Design Rule**

The **Concrete Wall Definition Editor** will open, and the **Design Rule Label** can be set to **Concrete Wall Typ.** Within this interface the rebar placement, rebar cover, as well as rebar size and spacing can be modified. For this example, let's make one simple change:

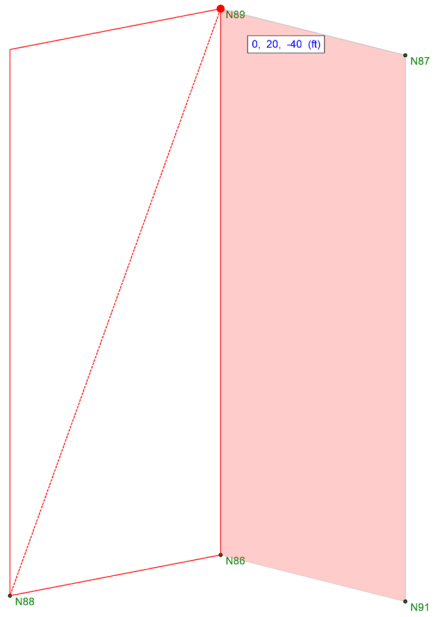
- Change the **Vertical Reinforcing** to **#7** and click **OK.**

Now the rest of the wall properties can be set, and the wall can be modeled:

- Click the point that defines the bottom left corner of the first wall panel **(0,0,-40)**
- Drag your mouse to the point that defines the top right corner of the wall panel **(10,20,-40)**
- Double click on wall to add a **Fixed** boundary condition

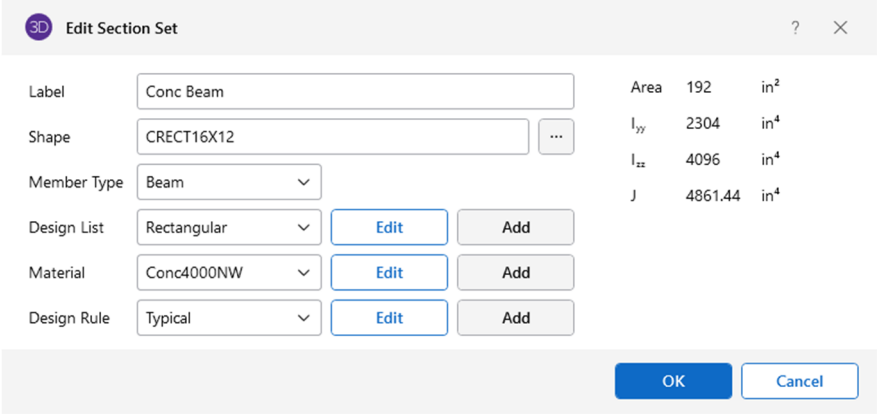
Once the first wall panel is created, model a 2nd panel with the same properties:

- Click the point that defines the bottom left corner of the first wall panel **(0,0,-30)**
- Drag your mouse to the point that was created when the 1st wall panel was defined **(0,20,-40)**
- Double click on wall to add **Fixed** boundary condition



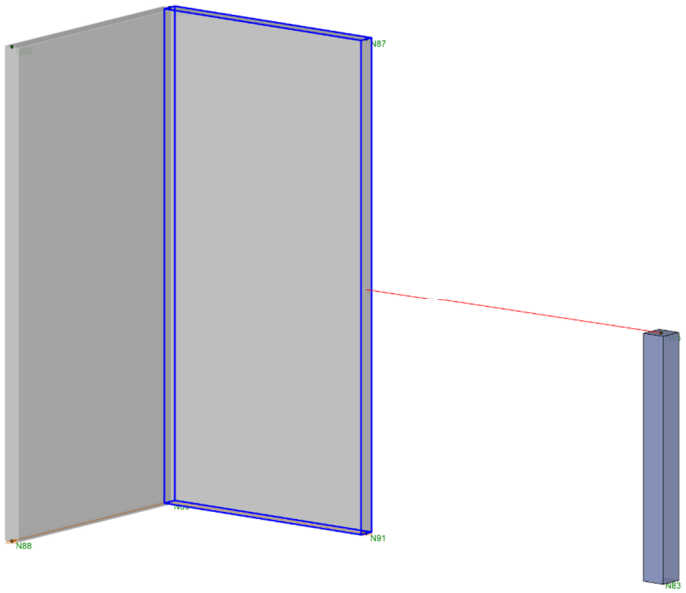
With the concrete column and concrete walls modeled, the concrete beams that support the floor can be modeled as well. To begin, click the **Members** button found in the **Draw Elements** section of the **Home** tab:

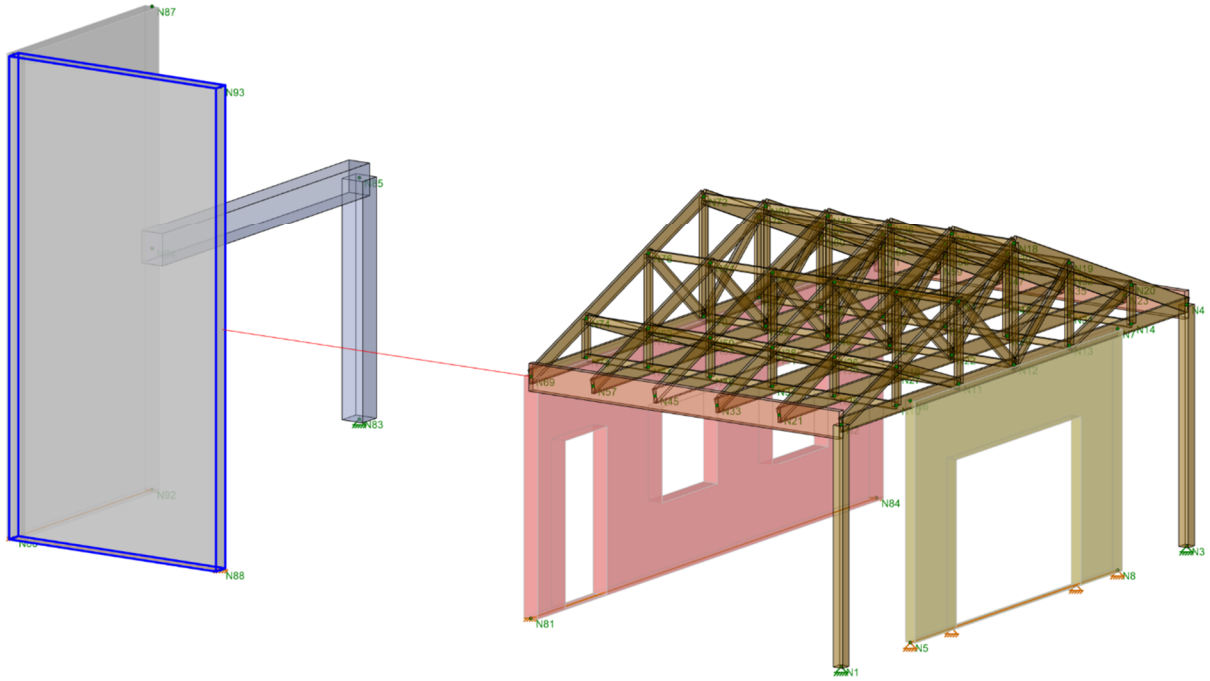
- In the beam properties, select the **Draw Option, Node to Member/Wall**
- Set the **Offset** to **10**
- Click on the ellipsis button ... located in the **Section Set** row and click to **Create a New Section Set**
- Create the **Conc Beam** section set with the properties shown in the following image.
- Click **OK** to finish the creation of the section set.



To begin modeling the concrete beams:

- Click the top node of the concrete column as the start point and then click the concrete wall in the same plane as the end point (as shown in the image to the right).
- Repeat this same process, by first selecting the node at the top of the masonry wall **(0,10,-15)** and then the concrete wall in the same plane as the node (as shown below).

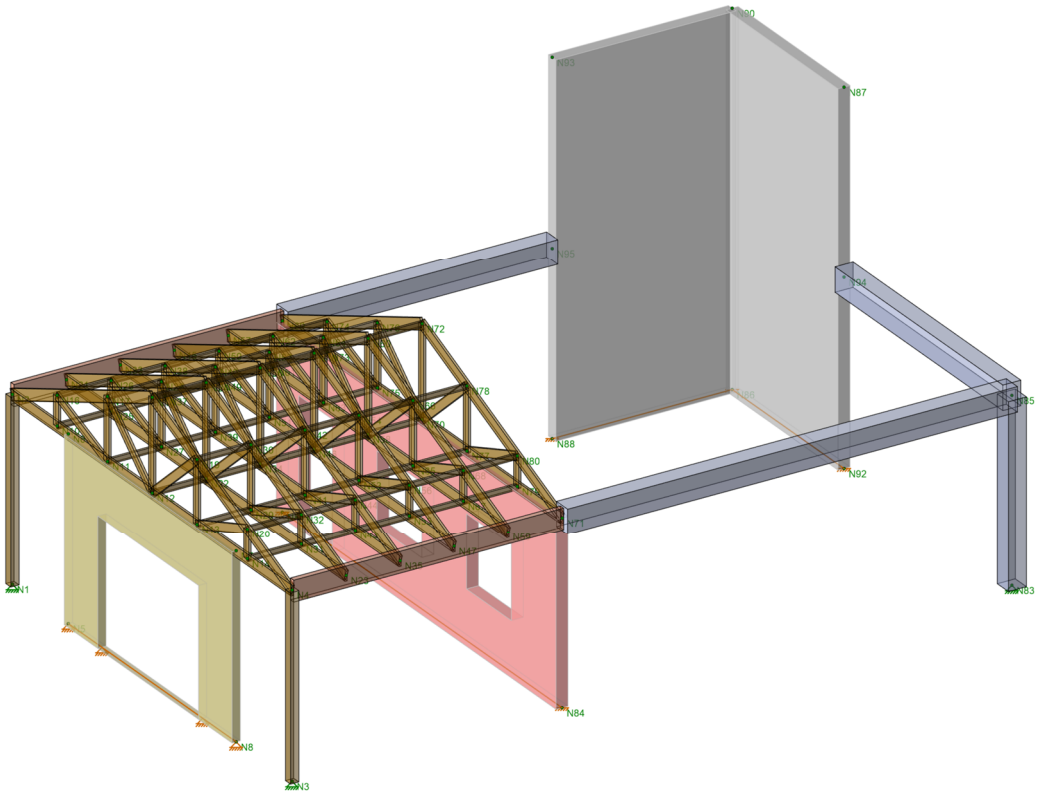




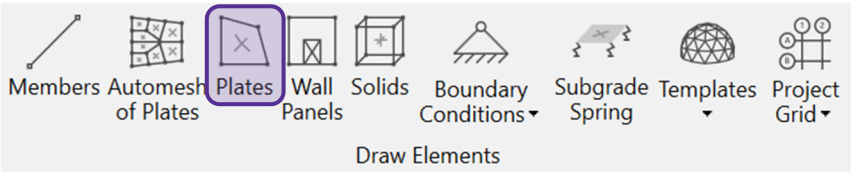
We can now draw the 3rd and final concrete beam:

- First, change the **Draw Option** property to **Node to Node**
- Draw the beam by selecting the top node of the masonry wall **(25,10,-15)** as the first start point and the top of the concrete column **(25,10,-40)** as the end point.
- Press **ESC** to end the command

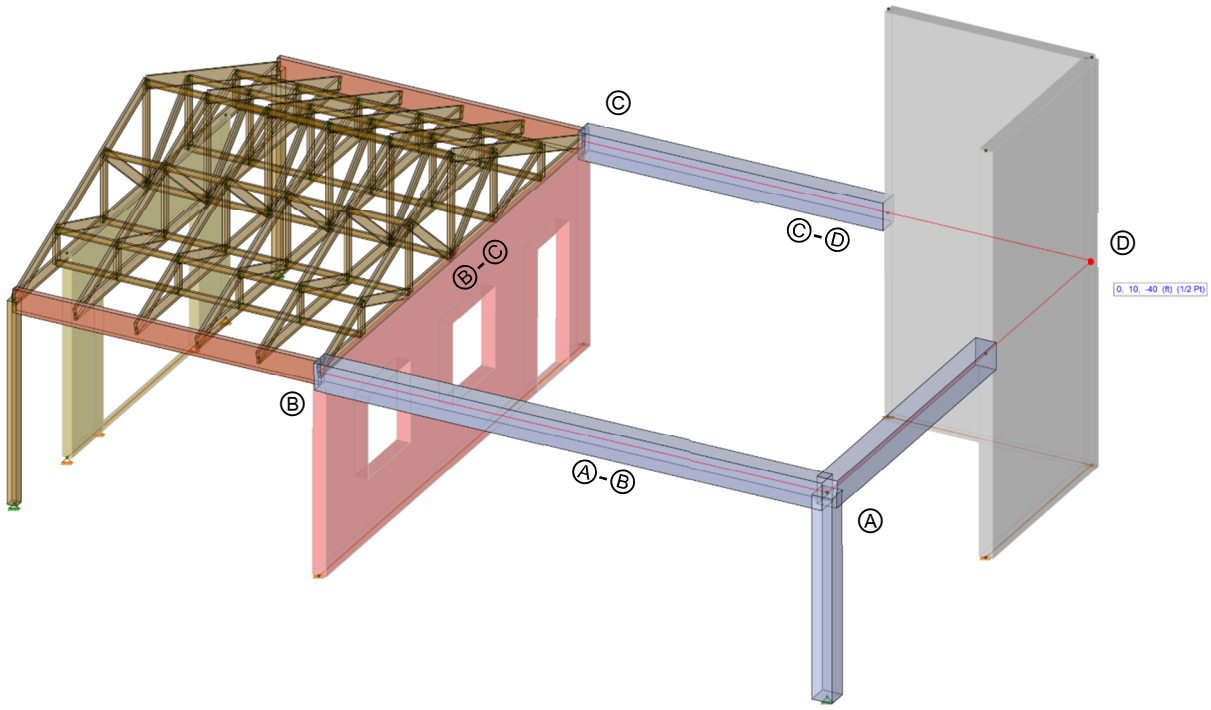
When you are finished the structure should currently look like the image below:



To finish the floor, let's draw in a 2-way concrete plate. To begin, click the **Plates** button in the **Draw Elements** section of the **Home** tab.



- Set the **Material** property to **gen_Conc4NW**
- Set the **Thickness** to **4"**
- Select **Quad Submesh** as the **Meshing** property
    - Set the **Number of Pieces** along both the **A-B** and **B-C** side to **10**
- Select the node at the top of the concrete column as the starting point and define the plate as shown on the next page following the edges shown. The end node of the plate is located at the mid-height of the concrete wall **(0,10,-40).**

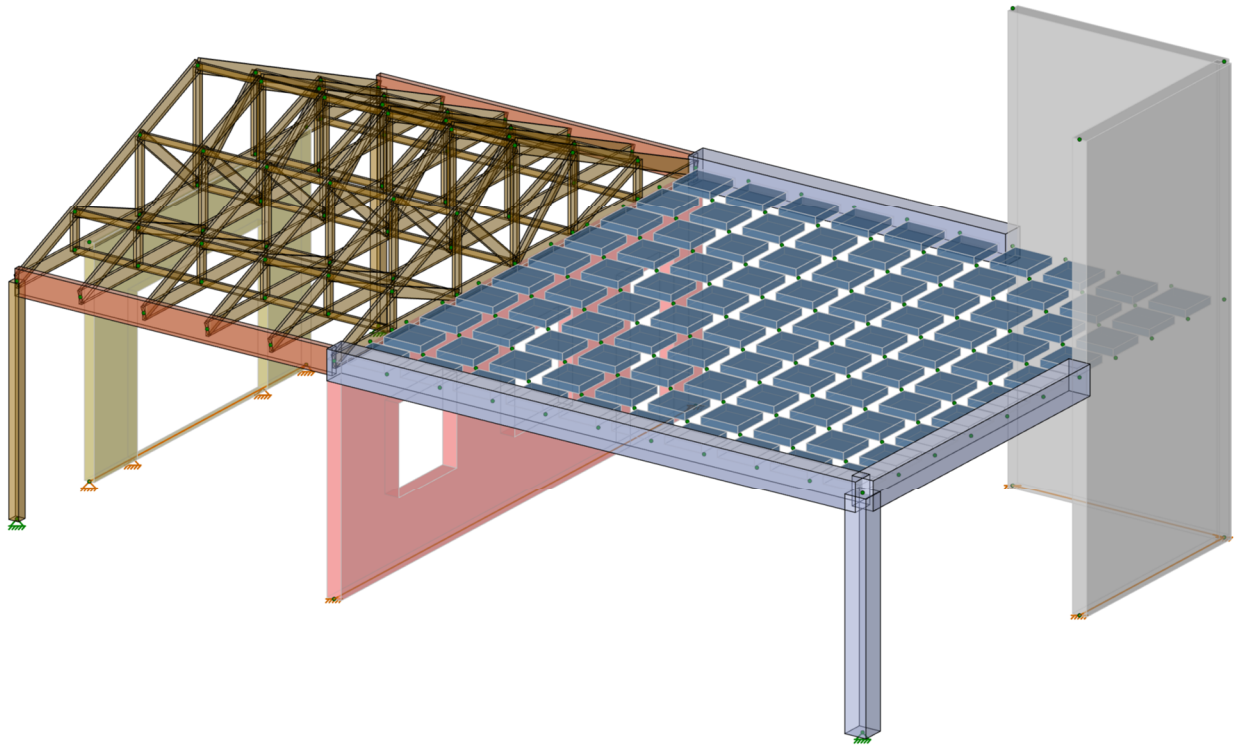


Note: When modeling **Plates**, there are four mesh options available:

- **None:** The plate that is drawn will not be "sub-divided" into smaller segments and therefore the polygon that is drawn will remain the same.
- **Auto Submeshing:** The drawn polygon will automatically be submeshed into smaller quadrilateral plate elements. The Auto Submeshing feature will also mesh to any nodes previously defined in the plane of the polygon. Thus, if you have columns supporting a slab already defined in your model the Automesh will mesh to the ends of these columns. Additionally, if you create two auto-meshed regions side by side the joints shared between the two regions will be aligned automatically.
- **Quad Submeshing:** This technique is used to submesh quadrilateral (4 sided) plate elements into a mesh of smaller elements. The new mesh can be any size up to the program limits for nodes and/or plates. The tool also allows for submesh increments to be defined differently in both directions. The A-B side is always the first side of the plate drawn.
- **Tri Submeshing:** This technique is used to submesh triangular (3 sided) elements into a mesh of 3 quadrilaterals. This is done by first creating a new node at the center of each selected triangular element and at the center point along each edge of the triangular element. These nodes are then used to create three quadrilateral elements that replace the triangular element.

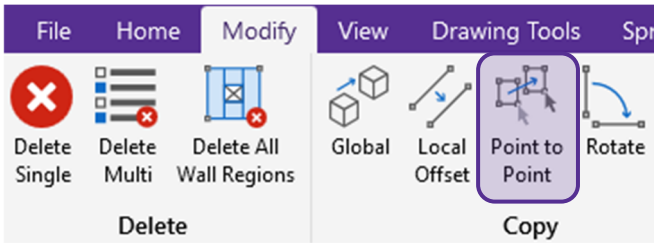
For more information about plates and meshing, visit the Plates/Shells section of the RISA-3D Web Help.

The model with submeshed floor should now look like this:



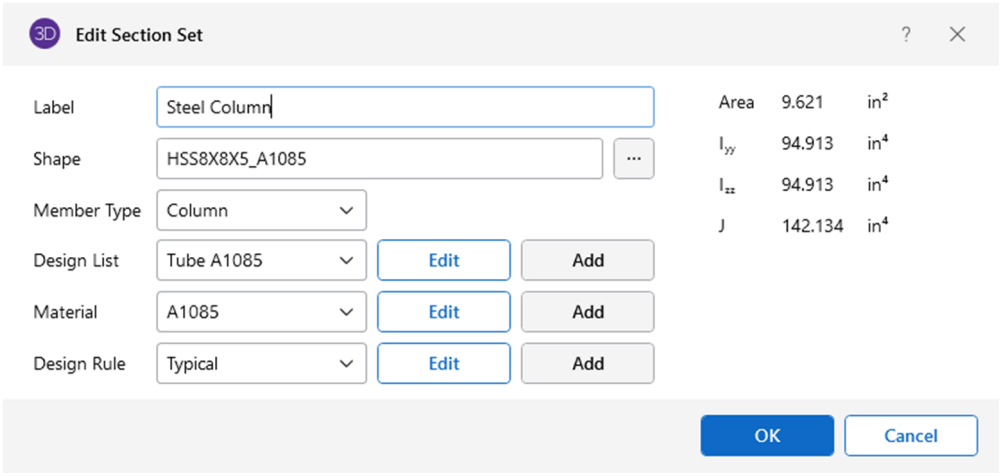
With the floor framing complete, we can move on to modeling the steel columns and beams which make up the roof framing. To create the columns, let's copy the concrete column from below and modify its properties:

- Select the concrete column
- Select **Point to Point Copy** from the **Copy** section of the **Modify** tab



- Click **Keep Tool Active**
- Click **Apply to Selected**
- Define the "reference point" by selecting the base node of the column **(25,0,-40)**
- Click on the following three node coordinate locations to define the new columns. Note that you must click the base node of the column to define the reference point between each copy command.
    - **(25,10-40)**
    - **(25,10,-15)**
    - **(0,10,-15)**
- Right click or press **ESC** to end the command

- Select all three newly modeled columns (holding in **CTRL** to multi-select)
- Change the **Material Type** to **Hot Rolled Steel**
- Click on the ellipsis button ... in the **Section Set** row and click **Edit/View the Existing Section Set**
- Create the **Steel Column** section set with the properties shown in the image below:

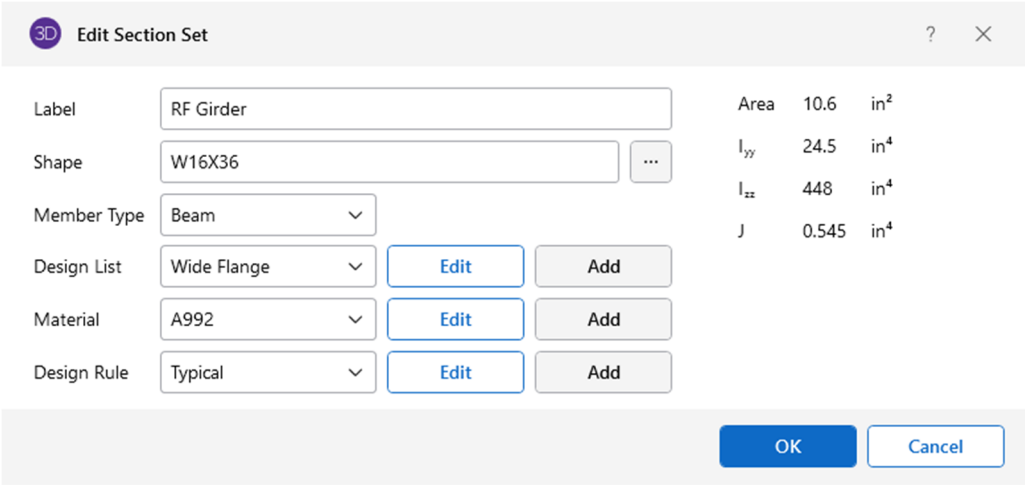


Note: The shape selection dialogue allows users to choose the **Database** (AISC, Canadian, Euro) as well as the **Shape Type.** It is also possible to add a custom shape to any library.

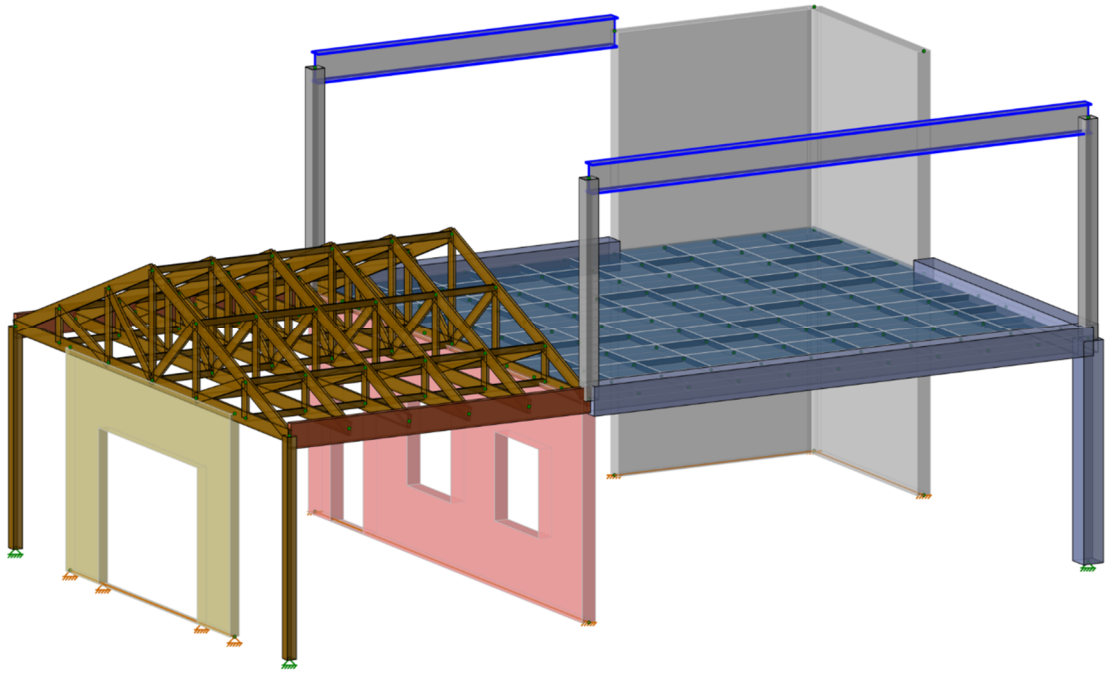
- Click **OK** to finish the creation of the section set.

With the steel columns now in the model, we can finish the roof framing by adding the roof beams. Let's start by drawing the exterior members (girders):

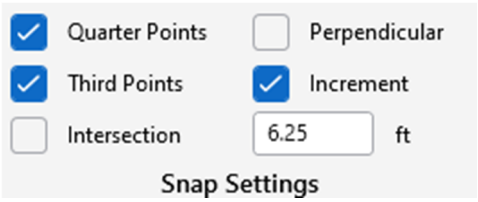
- Click the **Members** button found in the **Draw Elements** section of the **Home** tab
- Change the **Draw Option** property to **Node to Node**
- Change the **Material Type** to **Hot Rolled Steel**
- Click on the ellipsis button ... in the **Section Set** row and click to **Create a New Section Set**
- Create the **RF Girder** section set with the properties shown in the image below:



- Click **OK** to finish the creation of the section set.
- Draw the two girders as shown in the image on the next page:

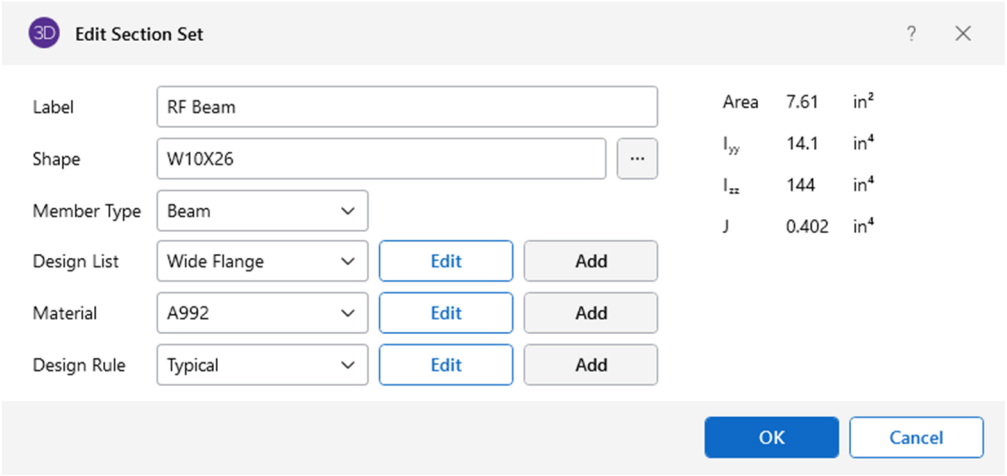


With the roof girders created, the roof structure can be completed by modeling the roof beams that are framed perpendicular to the girders. Before modeling the beams, let's modify the **Snap Settings** found on the **Drawing Tools** toolbar. For this input, change the **Increment** spacing to **6.25ft.**

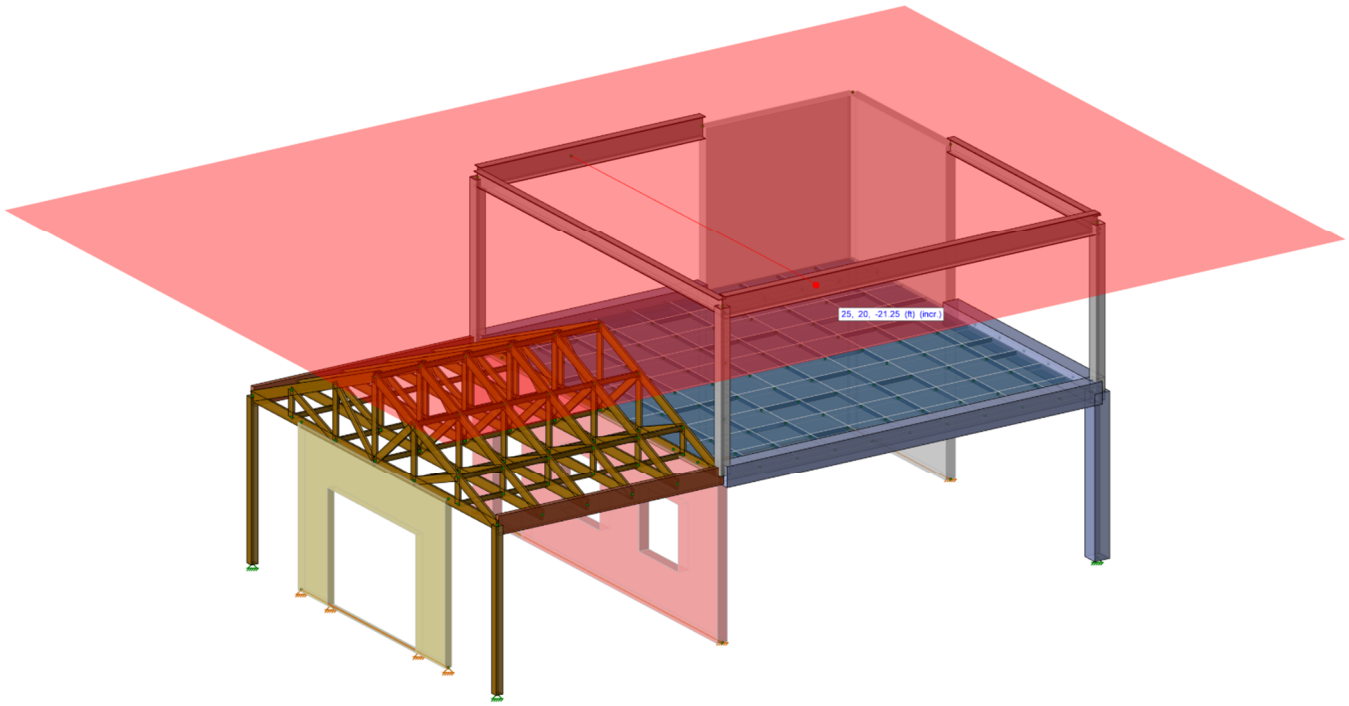


Next, let's lock the drawing plane global **XZ** plane of the girders by clicking the **Lock to Plane** button and choosing **XZ** from the dropdown. With the snap settings and the drawing plane set, model the roof beams as follows:

- Click the **Members** button found in the **Draw Elements** section of the **Home** tab
- Change the **Draw Option** property to **Node to Node**
- Change the **Material Type** to **Hot Rolled Steel**
- Click on the ellipsis button ... in the **Section Set** row and click to **Create a New Section Set**
- Create the **RF Beam** section set with the properties shown in the image below and click **OK.**

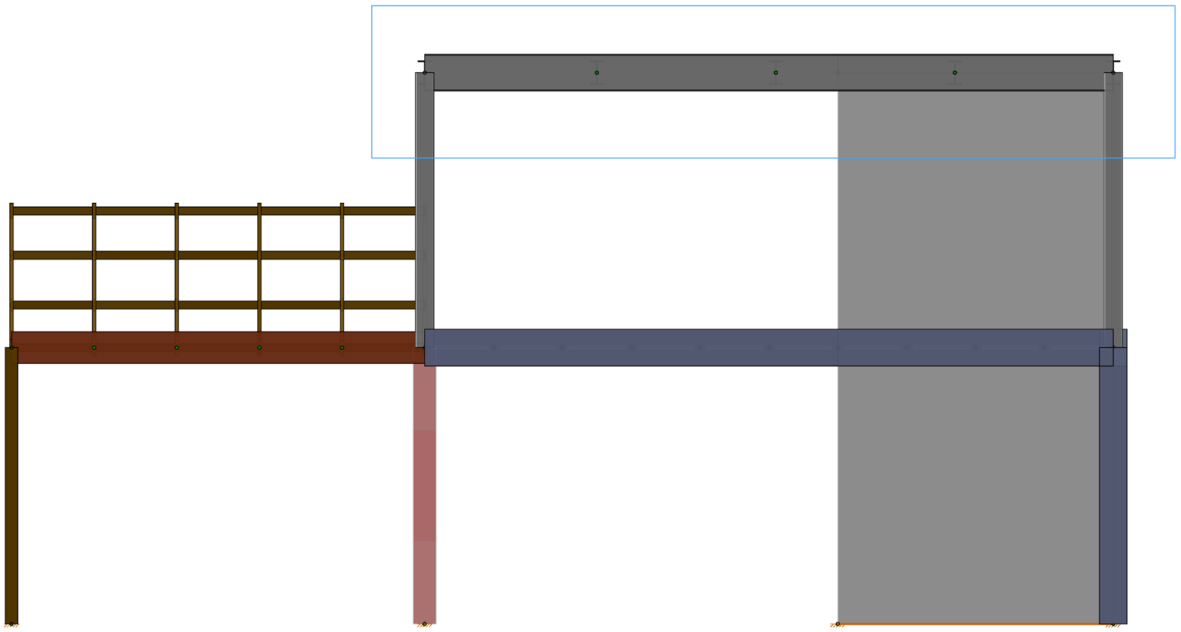


- Model the beam from the top of the steel column at **(0,20,-15)** to the top of the steel column at **(25,20,-15)**
- Model the beam from the top of the steel column at **(25,20,-40)** to the top corner of the concrete wall at **(10,20,-40)**
- Next, use the increment snap distance, to define the start point as the point along the steel girder that is **6.25ft** from the end of the member at the node coordinate **(0,20,-21.25).**
- Use the increment snap distance, to define the end point as the point along the opposite steel girder that is **6.25ft** from the end of the member at the node coordinate **(25,20,-21.25).**
- On the **Drawing Tools** tab, unselect the **Lock to Plane** button.
- Press **ESC** to end the command.

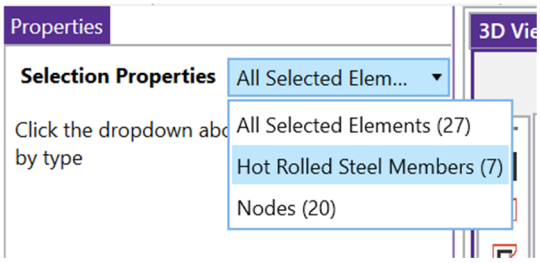


With the first beam drawn, we can utilize the **Global** copy feature found on the **Modify** tab to create the other two roof beams.

- After clicking on the **Global** copy button, input **2@-6.5** as the **Z increment**
- Uncheck the option for **Connect Bays**
- Make sure the roof beam is selected and click **Apply to Selected**
- Click the **YZ** button on the **View Toolbar** in the **3D View Window** and use a rectangular selection area to select all the roof members



- In the **Properties Panel**, change the **Selection Properties** dropdown to select and show properties for only the **Hot Rolled Steel Members**

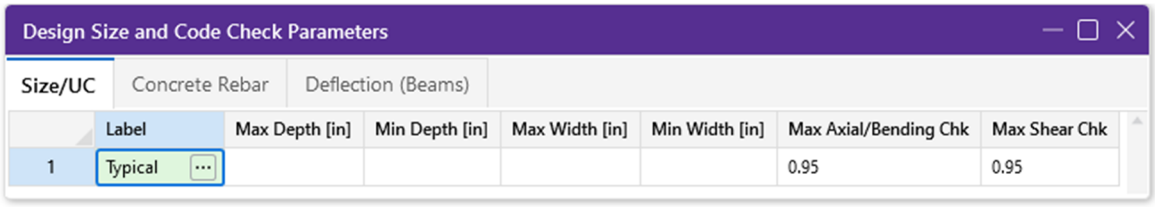


- Click on the ellipsis button ... next to the property **I Release** and select **Pinned (Torsion Fixed)** for both the **I and J End Release Codes.**
- Click **OK** to exit the dialog and press **ESC** to de-select the roof members.

Note: To see the release graphically, click on the **Member Ends** button in the **Quick View** section of the **Home** tab.

At this point, it makes sense to review the **Member Design Rules** for all members. To do this, click on **Member Design Rules** found in the **Explorer Panel** to open the spreadsheet. This spreadsheet allows users to control member depth limitations, unity check limitations, deflection criteria as well as concrete rebar standards for the design of concrete members. For this example, let's make two simple changes:

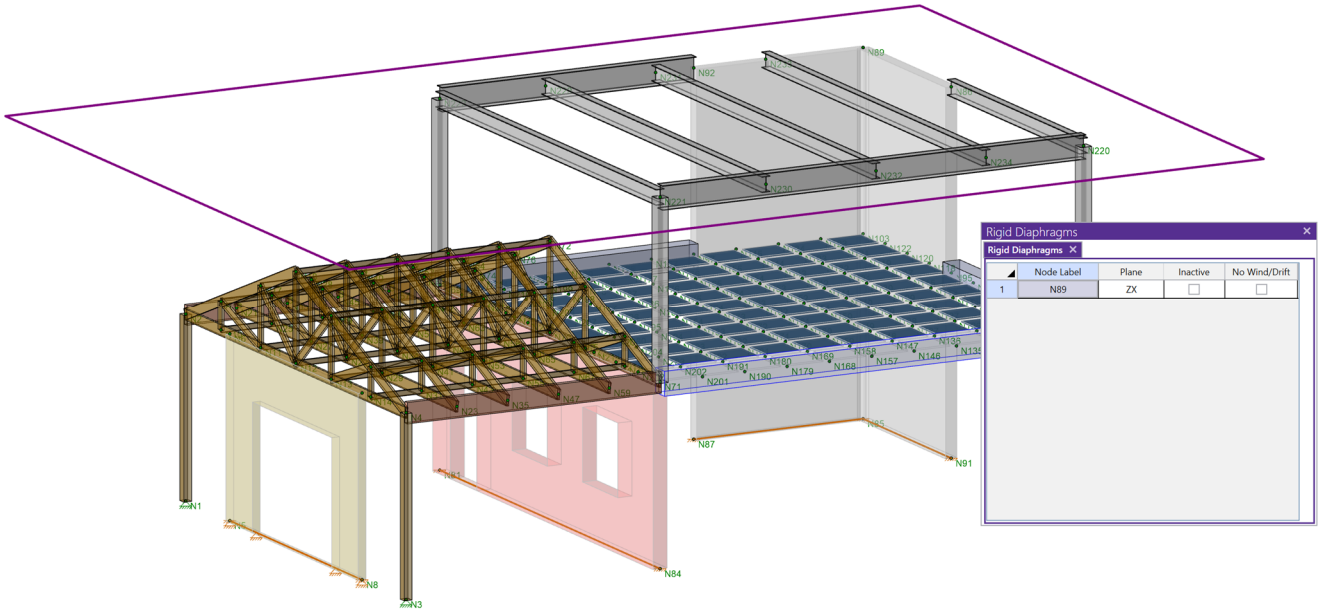
- Change the **Max Axial/Bending Chk** and **Max Shear Chk** values to **0.95** and close the spreadsheet**.**



After the steel roof beams have been created, it make sense to create a **diaphragm** so that any lateral load in the system can be distributed to the lateral load resisting elements (in this example, the concrete shear walls). In RISA-3D, it is possible to add **rigid diaphragms** such that all the nodes on a given plane are tied together. To do this, click on the **Diaphragms** spreadsheet in the **Data Entry** section of the **Explorer Panel.**

- Click or press Enter to add a new row to the spreadsheet
- Input a valid **Node Label** (ex. N89) for any node that is in the roof plane of the model **(Y = 20ft)**
- Set the **Plane** to **ZX** and then close the spreadsheet

To view the location of the diaphragm plane visually, click **the Toggle Display of Diaphragms**  button in the **Quick View** toolbar on the **Home** tab.



Note: **Rigid diaphragms** available in RISA-3D, distribute load to elements which connect to them solely based on the stiffness of the elements. They achieve this by tying all of the joints within the diaphragm plane together for both translation and rotation, but only within the plane of the diaphragm. This is typical behavior for most slabs and decks, which attribute vertical loads based on the tributary area of their supporting members.

By default, all nodes which fall within the plane of the diaphragm automatically become connected to it. Joints may be intentionally disconnected from the diaphragm by enabling the **Detach From Diaphragm** box in the **Node Coordinates** spreadsheet.

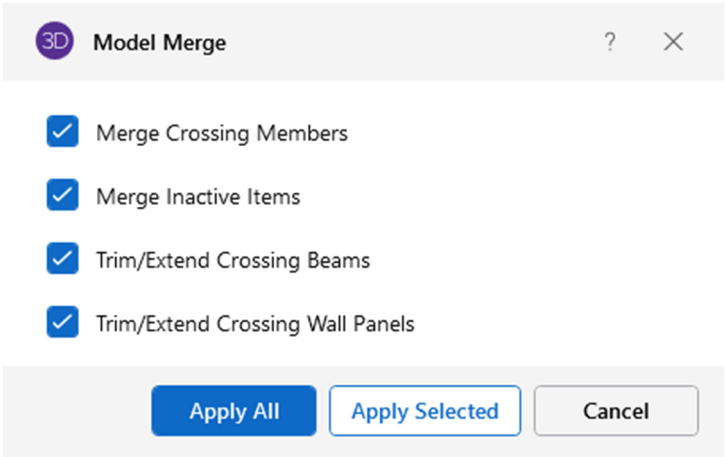
It is also possible to utilize diaphragms (rigid, flexible, semi-rigid) when using a combined **RISAFloor** and **RISA-3D** model. For more about how these diaphragms work, visit the Diaphragms section of the RISA-3D Web Help.

Before the model is finished, we need to apply boundary conditions to the base of the wood and concrete columns.

- On the **Home** tab, click the **Boundary Conditions** button and select Pinned
- In the **Properties Panel**, select **Click to Apply** and click on the nodes at the base of the wood columns and the concrete column
- Press **ESC** to end the command

Before moving on to apply loads and running the analysis it is important to run a **Model Merge**. The **Model Merge** button is in the **Advanced** section of the **Modify** tab and is a feature that scans through your model and automatically merges elements in the model. It also detects unconnected joints along member spans, unconnected crossing members and duplicate joints, members and plates.

- Click **Model Merge** and enable all checkboxes before clicking **Apply All.**



Once the **Model Merge** is complete, you will be presented with the results including “duplicate nodes found and merged” as well as “beams trimmed or extended”. Click **OK** to close the dialog.

At this point, the modeling of the structure is now complete, and you can move on to **Section 2: Loading.**

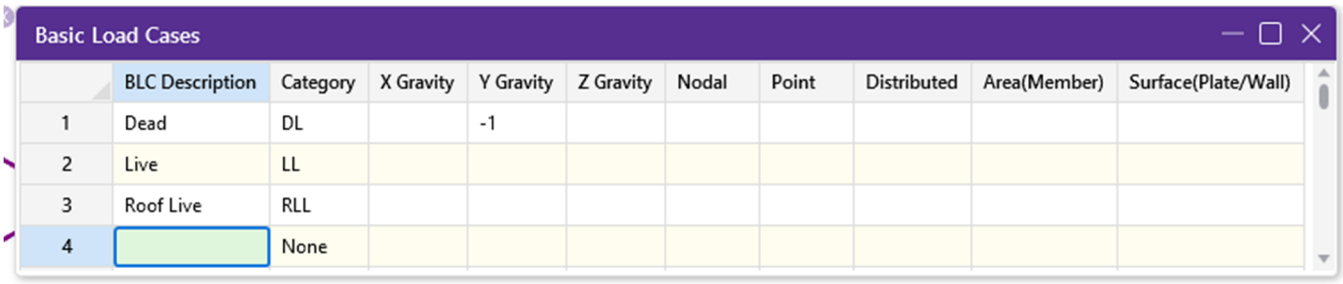
Section 2: Loading

Basic Load Cases

Before being able to add loads, **Basic Load Cases** need to be created. **Basic Load Cases** are used to group loads together so that they can be included in the load combinations applied to the structure. Additionally, **Basic Load Cases** can be assigned to a category such as Dead, Live or Wind when code-based load combinations are desired.

- To create **Basic Load Cases**, click the button 

found on the **Home** tab or select **Basic Load Cases** from the **Explorer Panel.**
- Create the **Basic Load Cases** for this structure based on the image shown below and then click the **X** in the top right corner to close the spreadsheet.



	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Nodal	Point	Distributed	Area(Member)	Surface(Plate/Wall)
1	Dead	DL		-1						
2	Live	LL								
3	Roof Live	RLL								
4		None								

Note: The self-weight of a structure can be applied automatically by including a value of **"-1"** in the **Y Gravity** column assuming that the Y axis is the vertical axis. Typically, this self-weight is associated with a Dead Load category.

Area Loads

The roof loads on both the gable roof and the flat roof will be applied as area loads. This means you will specify a load over an area and RISA-3D will attribute that load to the appropriate members. RISA-3D gives the option of one or two-way load attribution for area loads. One-way loads are attributed to the closest member in a certain direction; and two-way loads are attributed to the closest member, regardless of direction.

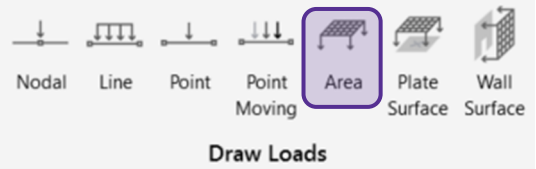
Before applying the area load, let's select just the roof plane:

- Click the **Select Elements by Property** 

button found on the **Home** tab.
- Select the tab for **Coordinates**
- Set the **Minimum** value for **Y** to be **20ft**
- From the **Nodes** dropdown choose **Select/Unselect**
- From the **Members** dropdown choose **Both Ends Must Be in Range**
- Click **Select** and then **Close** the dialog

With only the roof members selected click the **Lock Selected Parts of the Model and Dim Unselected** button from the **Selection Toolbar** found in the **3D View Window.** Doing this will "hide" the parts of the model that were not selected.

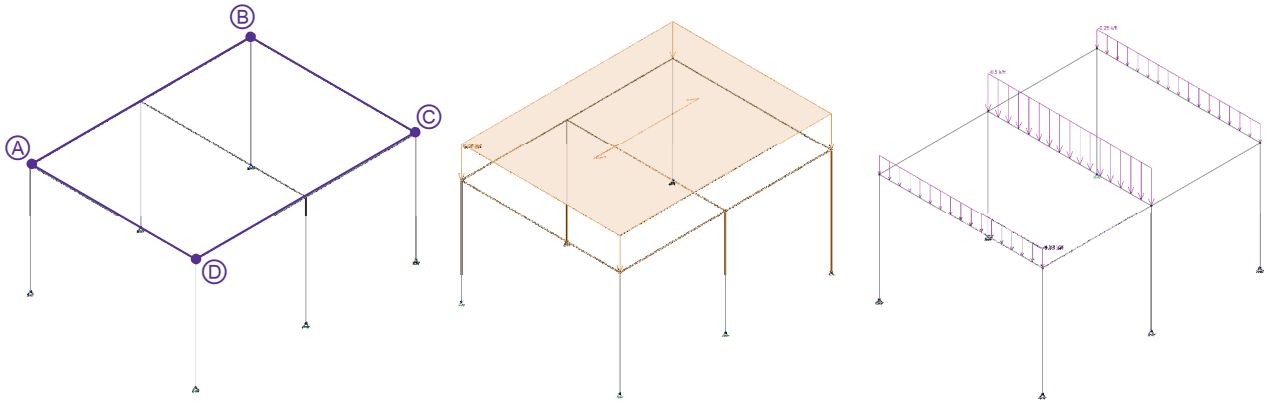
- To begin applying the area loads, click on the **Area Loads** button found in the **Draw Loads** section of the **Home** tab



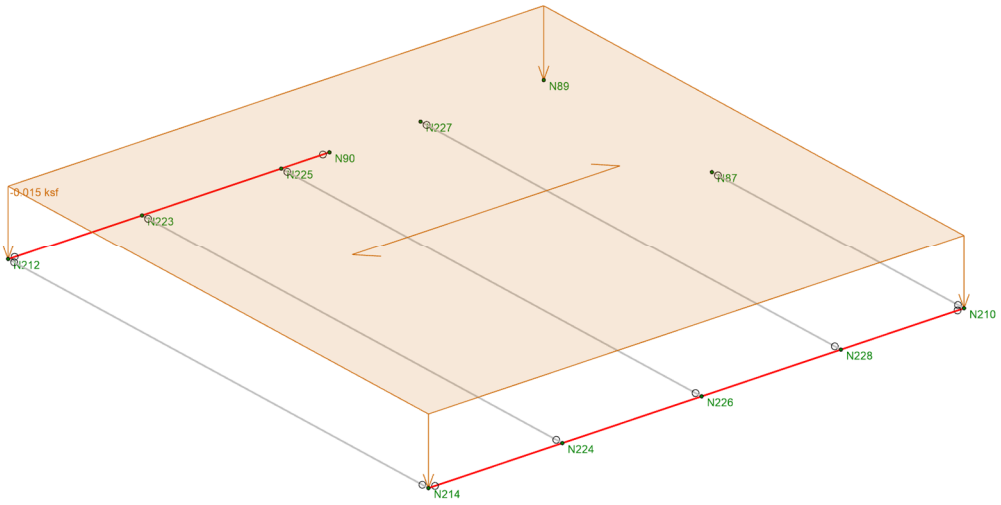
The **Properties Panel** will change so that the properties of the area load can be input, and the load can then be applied.

- Set the **Direction** to **Y**
- Choose **Dead** as the **BLC**
- Set the **Load Direction** to **A-B**
- Input **-0.015** as the **Magnitude**

Note: When choosing to use a one-way distribution for the area load, it is important to know that the distribution of each one-way option depend on the order that you click to create the area load. For example, the **A-B** option means that the load will span one-way in a direction parallel to the first two corners (nodes) of the area load. The image below shows what a one-way area load distribution would load like:



- Click on the node at **(0,20,-15)** to start the modeling the area load and continue along the beam to the node at **(0,20,-40)**
- Finish the polygon for the area load (based on the extents of the roof framing to generate the area load. When you are finished, your model should look like this:

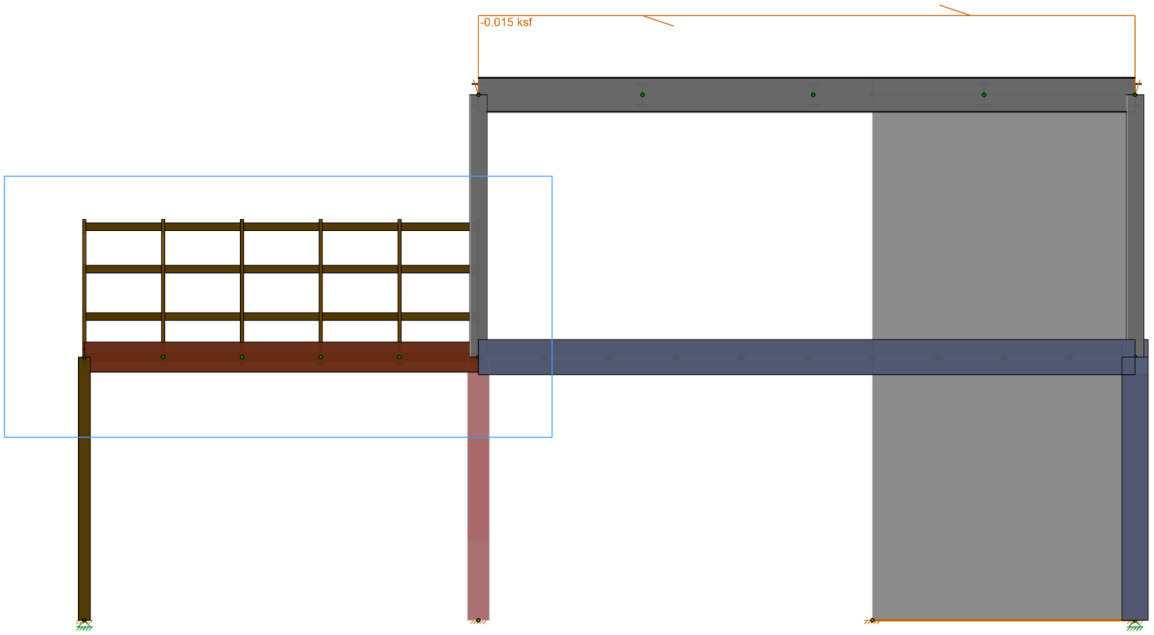


To add the roof live load, keep the **Area Load** command active:

- Switch the **BLC** property of the area load to **Roof Live**
- Change the **Magnitude** to **-0.025**
- Apply the **Roof Live** load in the same manner as the **Dead** load
- Press **ESC** to end the command

We can now move on to adding the Dead Load and Roof Live Load on the gable roof.

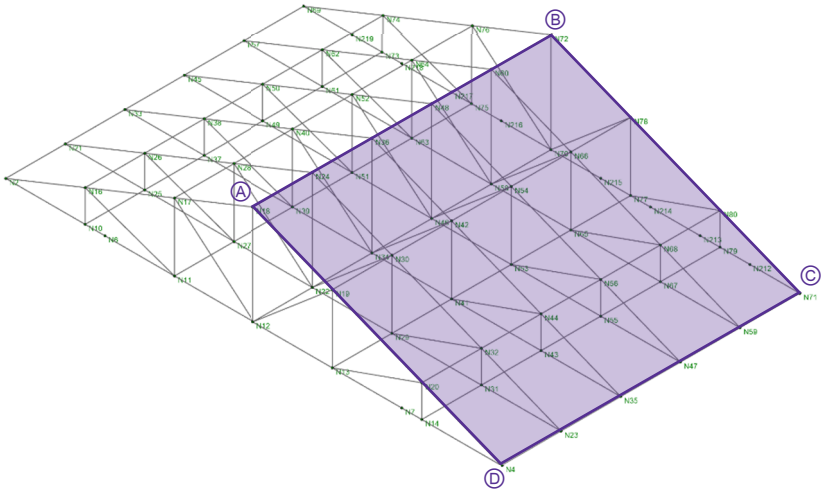
- Start by clicking the **Turn OFF the Lock Selected Parts of the Model and Dim Unselected** button from the **Selection Toolbar** found in the **3D View Window.**
- Click the **YZ** button on the **View Toolbar** in the **3D View Window** and using the left to right rectangular section (window selection) select only the gable roof members



- Click the **Lock Selected Parts of the Model** button from the **Selection Toolbar** found in the **3D View Window.**
- Click the **ISO** button on the **View Toolbar** in the **3D View Window**.

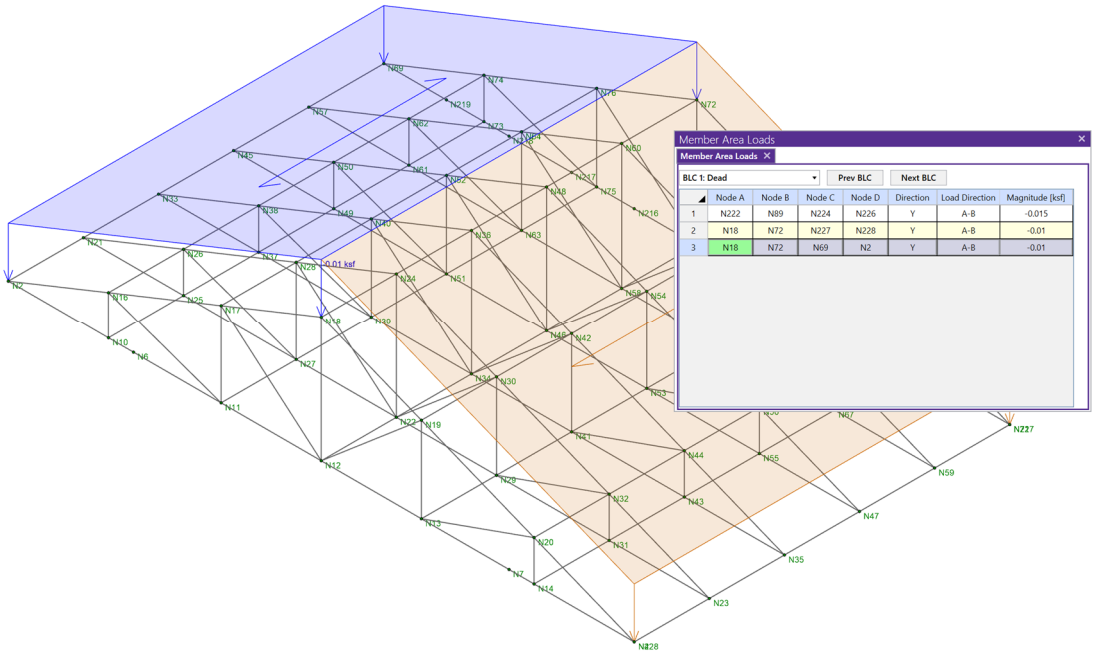
Create the **Dead Load** with the following properties making sure that the first edge of the load is drawn along the roof ridge (see image to the right).

- Set the **Direction** to **Y**
- Choose **Dead** as the **BLC**
- Set the **Load Direction** to **A-B**
- Input **-0.010** as the **Magnitude**

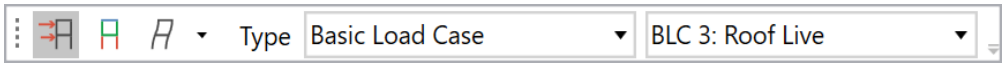


Now that the load has been added to one side of the roof, let's use the spreadsheets to add the additional loads on the gable roof.

- From the **Explorer Panel** on the right side of the screen, click on **Member Area Loads** to open the spreadsheet.
- Press **Enter** in the spreadsheet to add a new row
- Enter the area load information as shown in the image below. One thing to note is that it's possible that the node coordinate labels in your model are different. If this is the case, make sure to enter the nodes that define the start and end points of the roof ridge beam first.



Note: The loads, results and deflections can be easily viewed on the model through the buttons and dropdown boxes found on the **View Toolbar** in the **3D Window.**



From the dropdowns it is possible to select and view loads and results for **Basic Load Cases, Load Categories, Load Combinations** and **Envelopes.**

The various spreadsheets in RISA-3D allow us to utilize table operations to create a Roof Live load that uses the same node coordinates.

- Click the row labels **"2"** and **"3"** while holding **CTRL** to select all cells in both rows
- Right click and select **Copy** from the dropdown dialog
- Click the **Next BLC** button twice to open the **BLC: Roof Live** member area load spreadsheet
- Press **Enter** twice in the spreadsheet to add two new rows
- Right click and select **Paste** to paste the contents of the previous rows to the spreadsheet
- Change the **Magnitude** of the newly created loads to **-0.020 ksf**

Member Area Loads

BLC 3: Roof Live | Prev BLC | Next BLC

	Node A	Node B	Node C	Node D	Direction	Load Direction	Magnitude [ksf]	Inactive
1	N212	N90	N210	N211	Y	A-B	-0.025	
2	N18	N72	N71	N4	Y	A-B	-0.02	
3	N72	N18	N2	N69	Y	A-B	-0.02	

- Close the spreadsheet and click the **Turn OFF the Unselection Lock** button to show the entire model.

The last member area load to be created is the wind load in the X direction. To be able to apply the load properly, first click the **ISO** button on the **View Toolbar** in the **3D View Window**. Additionally, on the **Drawing Tools** tab, click on **Lock to Global Plane** button and set the plane to **YZ**

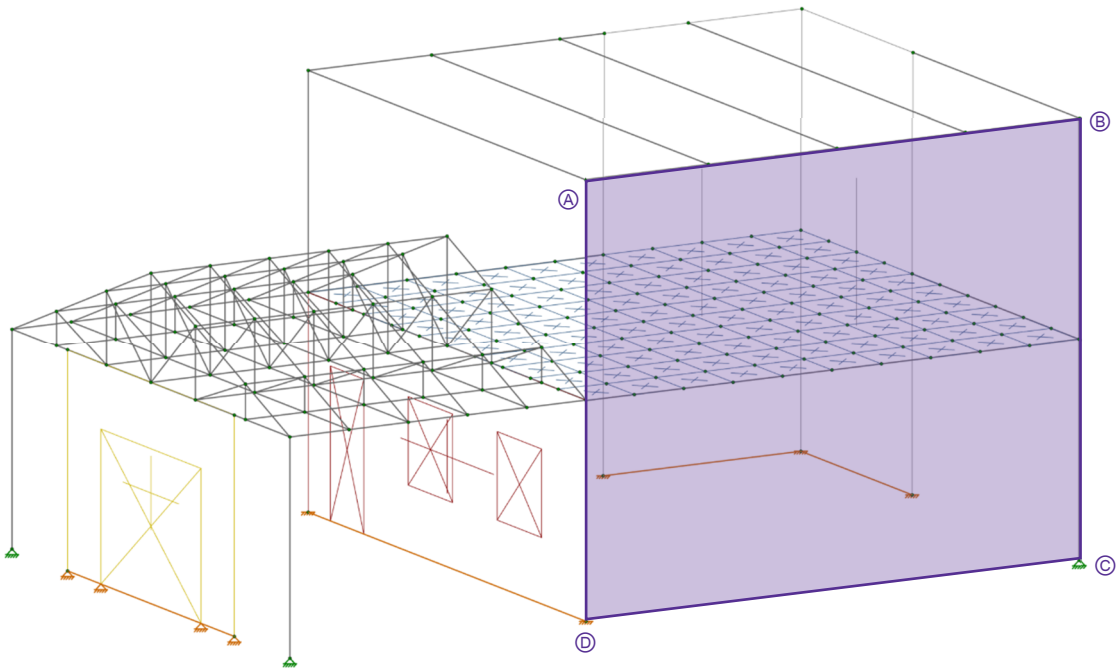
Since we don't yet have a Basic Load Case established for the Wind Load in the X Direction, we will need to create one. To begin, click on the **Area Loads** button found in the **Draw Loads** section of the **Home** tab and create the area load with the following properties:

- Click on the ellipsis button ... in the **BLC** row to **Edit Basic Load Cases**
- Choose basic load case **"4:"** as the **BLC**
- Input **Wind X** as the **BLC Description**
- Change the **Category** to **WL – Wind Load** and click **OK.**

With the BLC information for **Wind X** created, let's model the area load.

- Set the **Direction** to **X**
- Set the **Load Direction** to **Two Way**
- Set the **Magnitude** at **-0.025 ksf**

Draw the load on the frame between the edge of the masonry wall and the concrete/steel column as shown in the image below:



Surface Loads

Now that all the member area loads, both for gravity and lateral loading, have been modeled, let's model gravity load on the concrete plates using **Plate Surface Loads.**

Before drawing the load, we want to select only the concrete plates. To do this, click the **Select Elements by Property** button on the **Home** tab.

- Click on the **Plates** tab
- Enable the checkbox for **Materials** and select **gen_Conc4NW**
- Press **Select** and the click **Close** to exit the dialog

With the plate elements still selected, click on the **Plate Surface** button 

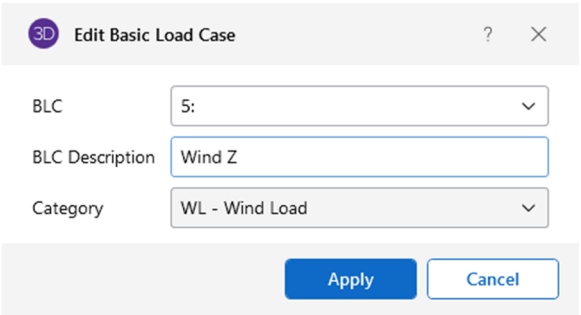
found in the **Draw Loads** section of the **Home** tab and set the load properties as follows:

- Set the **Direction** to **z**
- Choose **Dead** as the **BLC**
- Input **-0.020** as the **Magnitude**
- Click **Apply to Selected** to apply the dead load to the plates
- Switch the **BLC** to **Live**
- Change the **Magnitude** to **-0.040**
- Click **Apply to Selected** to apply the live load to the plates
- Press **ESC** to end the command and de-select the concrete plates

Note: When applying **Plate** or **Wall** surface loads RISA-3D allows the loads to be applied in the global axes (X,Y,Z), the local axes (x,y,z), or projected (PX,PY,PZ) in the direction of the global axes. When applying loads to plates or walls in the local axis (x,y,z) it may be necessary to flip the axis in order for the load to be applied properly. To do this, click on the **Flip Axis** button found in the **Plates** section of the **Modify** tab and choose either **Single** or **Selected**. Additionally, the direction of the surface load can also be flipped by removing, the "-" sign given to a specific load.

Let's now add a **Wall Surface** load on the wood wall panel. To begin drawing the load, click on the **Wall Surface** button found in the **Draw Loads** section of the **Home** tab.

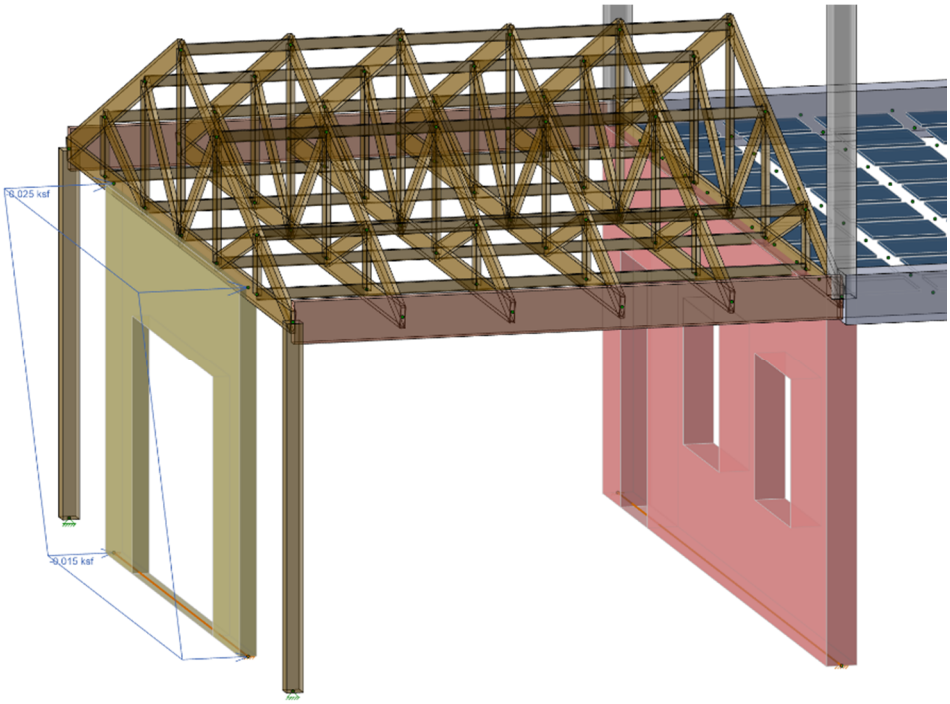
- Set the **Direction** to **z** (local axis – perpendicular to the wall)
- Click on the ellipsis button ... in the **BLC** row to invoke the **Edit Basic Load Cases** dialog
- Choose basic load case **"5:"** as the **BLC**
- Input **Wind Z** as the **BLC Description**
- Change the **Category** to **WL – Wind Load** and click **OK.**



With the BLC information for **Wind Z** created, let's model the wall surface load.

- Change the **Load Distribution** to **Tapered**
- Input **-0.025** as the **Top Magnitude** and **-0.015** as the **Bottom Magnitude**
- Select **Entire Wall** as the **Load Height**
- **Click to Apply** and select the wood wall panel and then click **ESC** to end the command

The applied load should look as it does in the image on the next page after it is applied.



Distributed Loads

The last type of load we will apply is a **distributed load** that can be used for the cladding on the exterior of the building.

Before applying the distributed load, let's select the concrete and glulam beams that we want to apply the distributed load to.

- Click the **Select Elements by Property** 

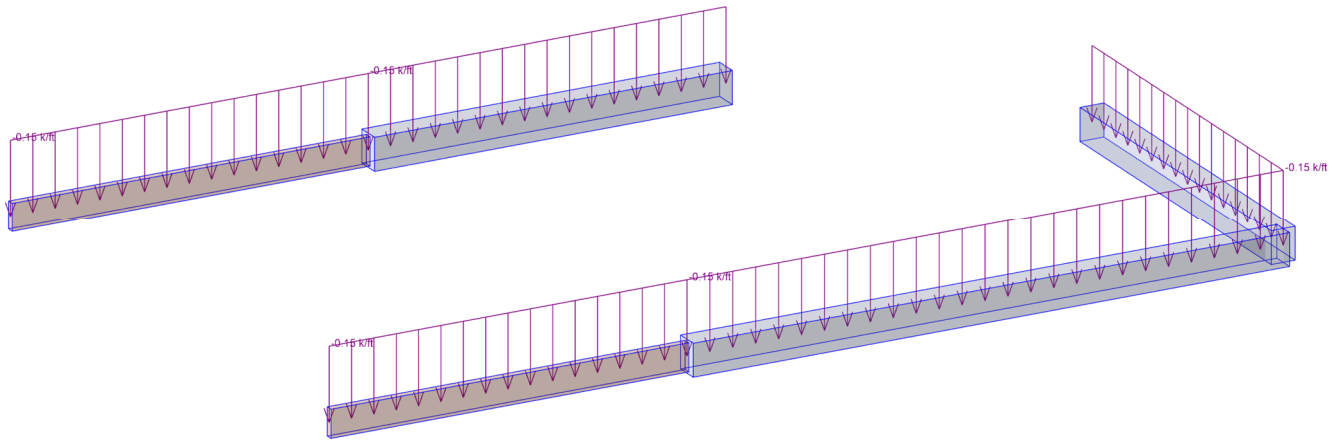
button found on the **Home** tab.
- Select the tab for **Members**
- Enable the checkbox for **Section Set/Shape**
- Select **Conc Beam** from the dropdown and click **Select**
- Select **Glulam Beam** from the dropdown and click **Select**
- Click **Close** the dialog

With the five beams selected, click on the **Line Loads** button 

in the **Draw Loads** section of the **Home** tab and set the load properties as follows:

- Set the **Direction** to **Y**
- Choose **Dead** as the **BLC**
- Input the **Start Magnitude** as **-0.15**
- Input the **End Magnitude** as **-0.15**
- Select **Full Length** as the **Load Distribution**
- Click **Apply to Selected** to apply the loads

Once the loads are added, click the **Lock Selected Parts of the Model** button and enable the **Rendered View** by clicking the button found in the **Quick View** section of the **Home** tab. Window. The loads should look as they do in the image on the next page after it is applied. When you are finished reviewing the loads you can click the **Turn OFF the Unselection Lock** button to show the entire model and press **ESC** to clear the selection.



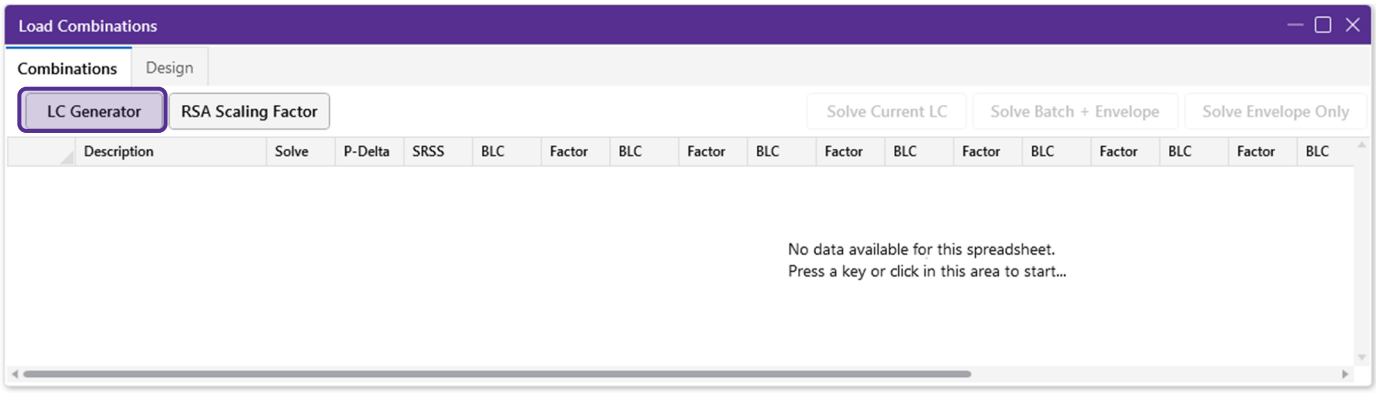
Load Combinations

With the basic load cases created and the loads applied to the structure, we now can create the **Load Combinations** that will be used for both strength and serviceability level design.

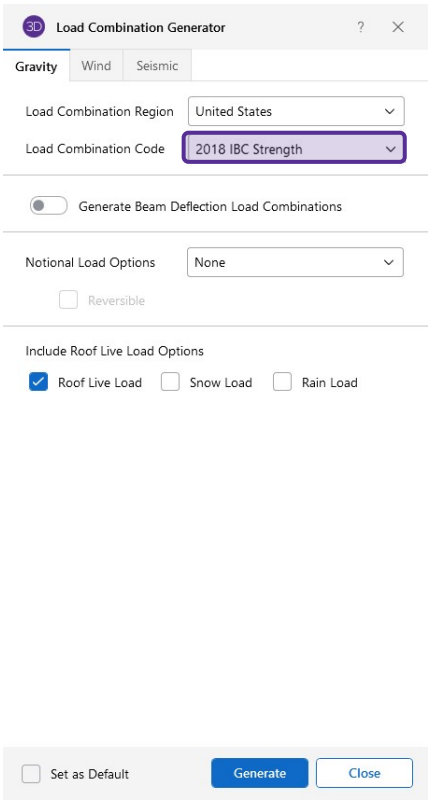
- To create **Load Combinations**, click the button 

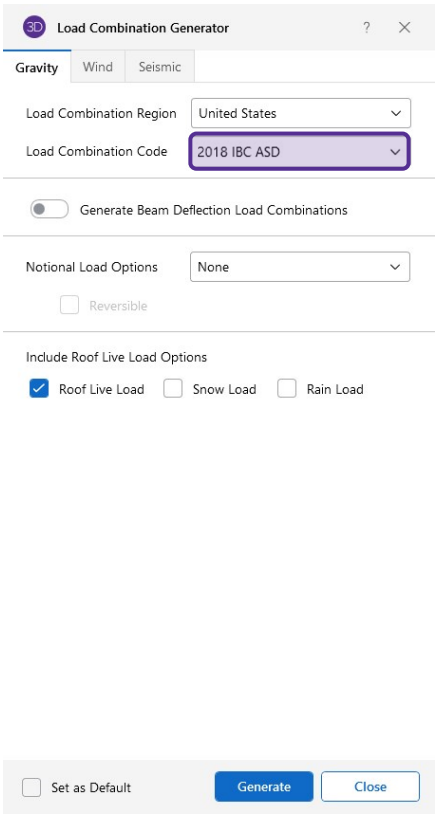
found on the **Home** tab or select **Load Combinations** from the **Explorer Panel.**

When the spreadsheet opens the first time, no load combinations will have been created yet. At this point it is possible to click in the spreadsheet to begin to create load combinations manually or you can use the **LC Generator** to create code-based combinations for gravity, wind and seismic for you. Let's create the combinations for this model, by clicking on the **LC Generator** button.

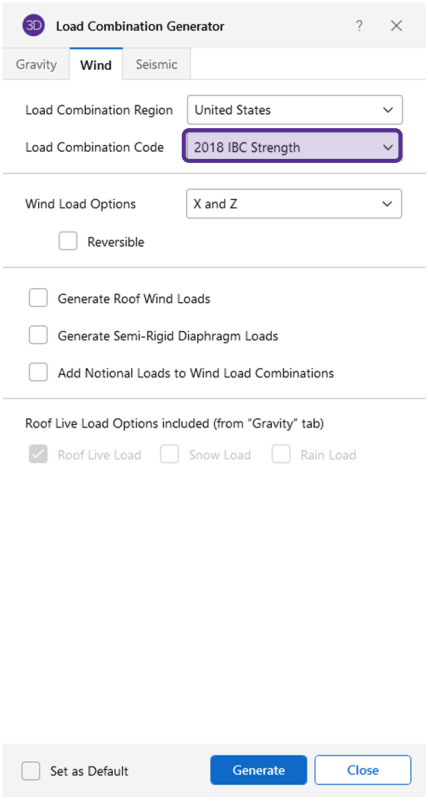


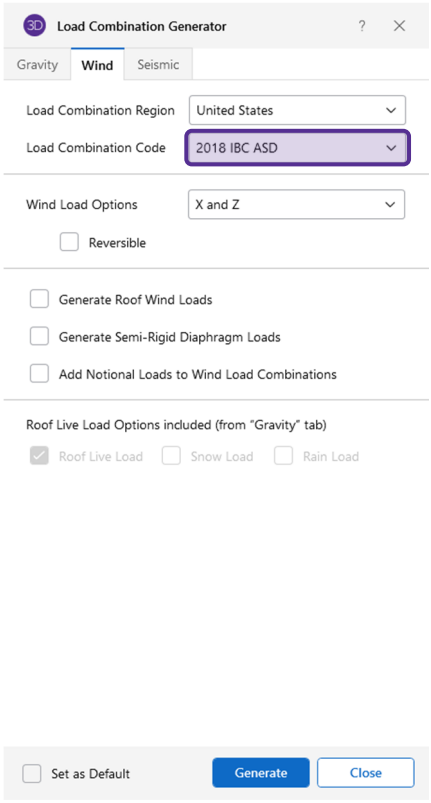
When the **LC Generator** button is clicked, the load combination generator dialog will open and show three different tabs: Gravity, Wind and Seismic. On each tab, a **LC Region** and **LC Code** can be selected in addition to various options for the set up and generation of code-based load combinations. Let's set up the load combinations for **Gravity** so they match the images shown below. Both **2018 IBC Strength** (LRFD) and **2018 IBC ASD** combinations can be created as shown in the images below. Make sure to click the **Generate** button to create the load combinations in the load combinations spreadsheet.





With the gravity combinations created, click on the **Wind** tab and create both the **2018 IBC Strength** and **2018 IBC ASD** combination sets as shown in the images on the next page.





After all the combinations are generated, click **Close** to close the **LC Generator.** The **Load Combinations** spreadsheet should now include **26** different load combinations.

Load Combinations

Combinations | Design

LC Generator | RSA Scaling Factor | Solve Current LC | Solve Batch + Envelope | Solve Envelope Only

	Description	Solve	P-Delta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	IBC 16-1	✓	Y		DL	1.4								
2	IBC 16-2 (a)	✓	Y		DL	1.2	LL	1.6	LLS	1.6	RLL	0.5		
3	IBC 16-2 (b)	✓	Y		DL	1.2	LL	1.6	LLS	1.6				
4	IBC 16-3 (a)	✓	Y		DL	1.2	RLL	1.6	LL	0.5	LLS	1		
5	IBC 16-8	✓	Y		DL	1								
6	IBC 16-9	✓	Y		DL	1	LL	1	LLS	1				
7	IBC 16-10 (a)	✓	Y		DL	1	RLL	1						
8	IBC 16-11 (a)	✓	Y		DL	1	LL	0.75	LLS	0.75	RLL	0.75		
9	IBC 16-3 (b) (a)	✓	Y		DL	1.2	RLL	1.6	WLX	0.5				
10	IBC 16-3 (b) (b)	✓	Y		DL	1.2	RLL	1.6	WLZ	0.5				
11	IBC 16-3 (d) (a)	✓	Y		DL	1.2	WLX	0.5						
12	IBC 16-3 (d) (b)	✓	Y		DL	1.2	WLZ	0.5						
13	IBC 16-4 (a) (a)	✓	Y		DL	1.2	WLX	1	LL	0.5	LLS	1	RLL	0.5
14	IBC 16-4 (a) (b)	✓	Y		DL	1.2	WLZ	1	LL	0.5	LLS	1	RLL	0.5
15	IBC 16-4 (b) (a)	✓	Y		DL	1.2	WLX	1	LL	0.5	LLS	1		
16	IBC 16-4 (b) (b)	✓	Y		DL	1.2	WLZ	1	LL	0.5	LLS	1		
17	IBC 16-6 (a)	✓	Y		DL	0.9	WLX	1						
18	IBC 16-6 (b)	✓	Y		DL	0.9	WLZ	1						
19	IBC 16-12 (a) (a)	✓	Y		DL	1	WLX	0.6						
20	IBC 16-12 (a) (b)	✓	Y		DL	1	WLZ	0.6						
21	IBC 16-13 (a) (a)	✓	Y		DL	1	WLX	0.45	LL	0.75	LLS	0.75	RLL	0.75
22	IBC 16-13 (a) (b)	✓	Y		DL	1	WLZ	0.45	LL	0.75	LLS	0.75	RLL	0.75
23	IBC 16-13 (b) (a)	✓	Y		DL	1	WLX	0.45	LL	0.75	LLS	0.75		
24	IBC 16-13 (b) (b)	✓	Y		DL	1	WLZ	0.45	LL	0.75	LLS	0.75		
25	IBC 16-15 (a)	✓	Y		DL	0.6	WLX	0.6						
26	IBC 16-15 (b)	✓	Y		DL	0.6	WLZ	0.6						

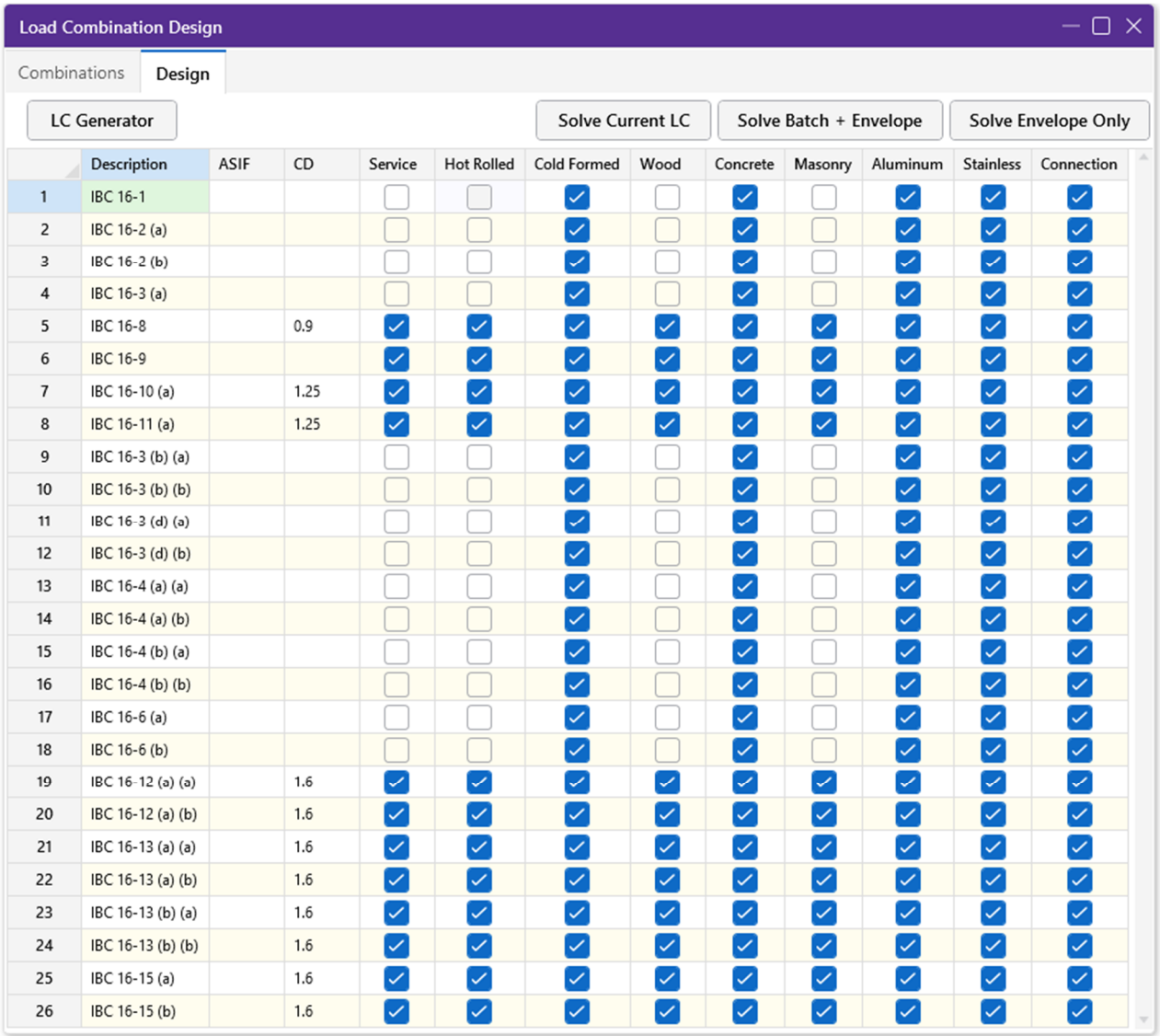
Note: The load combinations spreadsheet includes options that enable users to select specific load combinations to include or exclude during batch solution **(Solve)**, analyze member secondary effects **(P-Delta)**, and combine response spectra results for different directions using the Square Root of Sum of Squares **(SRSS)** method. Additionally, load combinations can be created manually by entering the desired **Load Category** or **Basic Load Case (BLC)** as well as the corresponding **Factor.**

Other options include nesting one combination within another and including response spectra results. For more information, visit the Load Combinations section of the RISA-3D Web Help.

Another feature of the load combinations spreadsheet is the **Design** tab. This tab differs from the **Combinations** tab because it allows a user to select which combinations will be used for the design of specific materials as well as selecting which combinations will be used for serviceability checks. For this model, we want to disable all strength level (LRFD) load combinations for all materials except concrete. To do this, first click on the **Design** tab in the **Load Combinations** spreadsheet.

Individual load combinations can be enabled or disabled by selecting the checkbox that corresponds to the desired code-based combination and material. Additionally, spreadsheet tools can be used to quickly modify the combinations.

- Highlight the first **4** cells in the **Hot Rolled** column of the spreadsheet. These cells represent strength level (LRFD) combinations that we don't want to use in the design of **Hot Rolled** steel.
- Right click somewhere within the highlighted cells and click **Check/Uncheck Selected Cells** from the dropdown
- Repeat the process to enable/disable design combinations so that the spreadsheet matches the image shown on the next page:

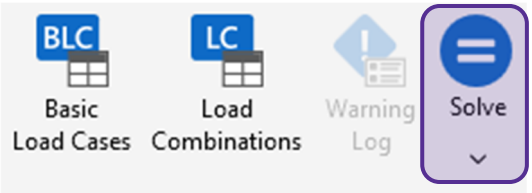


	Description	ASIF	CD	Service	Hot Rolled	Cold Formed	Wood	Concrete	Masonry	Aluminum	Stainless	Connection
1	IBC 16-1			☐	☐	☑	☐	☑	☐	☑	☑	☑
2	IBC 16-2 (a)			☐	☐	☑	☐	☑	☐	☑	☑	☑
3	IBC 16-2 (b)			☐	☐	☑	☐	☑	☐	☑	☑	☑
4	IBC 16-3 (a)			☐	☐	☑	☐	☑	☐	☑	☑	☑
5	IBC 16-8		0.9	☑	☑	☑	☑	☑	☑	☑	☑	☑
6	IBC 16-9			☑	☑	☑	☑	☑	☑	☑	☑	☑
7	IBC 16-10 (a)		1.25	☑	☑	☑	☑	☑	☑	☑	☑	☑
8	IBC 16-11 (a)		1.25	☑	☑	☑	☑	☑	☑	☑	☑	☑
9	IBC 16-3 (b) (a)			☐	☐	☑	☐	☑	☐	☑	☑	☑
10	IBC 16-3 (b) (b)			☐	☐	☑	☐	☑	☐	☑	☑	☑
11	IBC 16-3 (d) (a)			☐	☐	☑	☐	☑	☐	☑	☑	☑
12	IBC 16-3 (d) (b)			☐	☐	☑	☐	☑	☐	☑	☑	☑
13	IBC 16-4 (a) (a)			☐	☐	☑	☐	☑	☐	☑	☑	☑
14	IBC 16-4 (a) (b)			☐	☐	☑	☐	☑	☐	☑	☑	☑
15	IBC 16-4 (b) (a)			☐	☐	☑	☐	☑	☐	☑	☑	☑
16	IBC 16-4 (b) (b)			☐	☐	☑	☐	☑	☐	☑	☑	☑
17	IBC 16-6 (a)			☐	☐	☑	☐	☑	☐	☑	☑	☑
18	IBC 16-6 (b)			☐	☐	☑	☐	☑	☐	☑	☑	☑
19	IBC 16-12 (a) (a)		1.6	☑	☑	☑	☑	☑	☑	☑	☑	☑
20	IBC 16-12 (a) (b)		1.6	☑	☑	☑	☑	☑	☑	☑	☑	☑
21	IBC 16-13 (a) (a)		1.6	☑	☑	☑	☑	☑	☑	☑	☑	☑
22	IBC 16-13 (a) (b)		1.6	☑	☑	☑	☑	☑	☑	☑	☑	☑
23	IBC 16-13 (b) (a)		1.6	☑	☑	☑	☑	☑	☑	☑	☑	☑
24	IBC 16-13 (b) (b)		1.6	☑	☑	☑	☑	☑	☑	☑	☑	☑
25	IBC 16-15 (a)		1.6	☑	☑	☑	☑	☑	☑	☑	☑	☑
26	IBC 16-15 (b)		1.6	☑	☑	☑	☑	☑	☑	☑	☑	☑

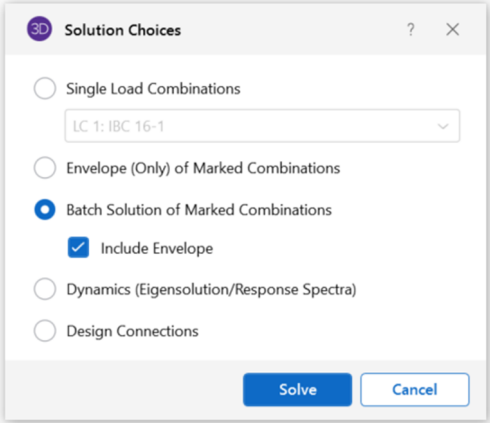
Section 3: Solution and Results

Solving the Model

When the basic load cases, corresponding loads and all load combinations have been created the model is ready to be solved so that analysis results and design can be reviewed. To solve the model, click on the **Solve** button found on the **Home** tab.



There are multiple solution choices that can be utilized depending on what type of results are required.



Note: Prior to executing the solution, users can run the **Model Merge** tool (found in the **Advanced** section of the **Modify** tab) in order to ensure that the solution is not interrupted by unconnected joints along member spans, unconnected crossing members and/or duplicate joints, members and plates.

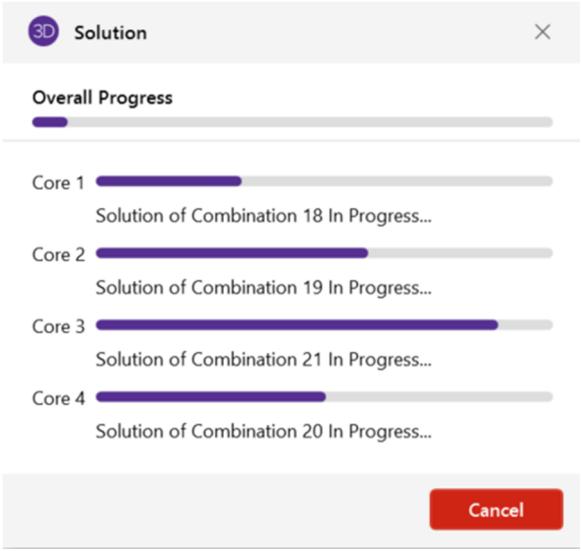
Single Load Combinations: Solves one load combination by itself.

Envelope (Only) of Marked Combinations: Static solutions may be performed on multiple combinations (based on those enabled in the load combinations spreadsheet) and the results enveloped to show only the minimum and maximum results. Each of the results spreadsheets will contain minimum and maximum values for each result and also the corresponding load combination.

Batch Solution of Marked Combinations: Static solutions may be performed on multiple combinations and the results retained for each solution. When performing a batch solution, you have the option to also include a set of envelope results. This is useful when an envelope result is desired to quickly determine a controlling load combination, but when the investigation of that load combination required the greater details given with batch solution results.

Dynamics (Eigensolution/Response Spectra): Solves according to the dynamic properties of the structure including assembling the mass matrix, solving for the eigen values (natural periods/frequencies), as well as determining the mode shapes of vibration associated with each period/frequency.

For this model, let's use a **Batch Solution** and include the **Envelope** with the batch results. To do this, you can either press **Solve,** or it is also possible to execute various solution types from the **Load Combinations** spreadsheet. Once the solution begins various progress bars for the solution of different load combinations (per core if applicable) will be visible.



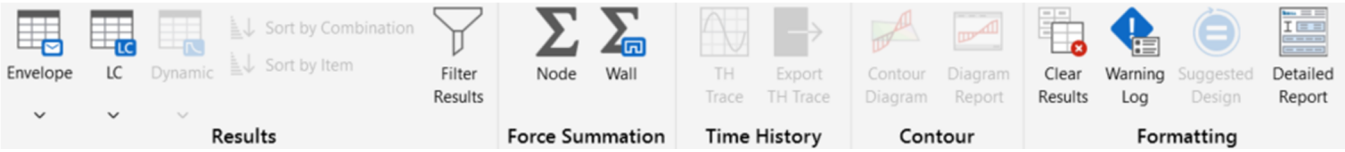
Once the solution is complete, the **Envelope Node Reactions** spreadsheet will open automatically along with the **Warning Log.** The warning log includes errors and warnings within the model that need to be reviewed or addressed by the engineer. Let's close out of these two windows to begin evaluating the analysis results as well as the design results for the model.

Note: It is possible that a user will obtain analysis and **design results that differ** from those found in the following pages. This could be due to a variety of factors including the way the model was created as well as the version of the software. Use these results as **representative only** and as a way to convey the general features and functionality associated with results in RISA-3D.

Analysis Results

Analysis and Design results are easily available both graphically and in table format. The results spreadsheets can be accessed from the **Explorer Panel** or from the **Results** tab on the ribbon toolbar. Since the **Batch + Envelope** solution was executed, results for each individual load combination as well as the envelope exists.

Let's take a look at the **Results** tab to see all the available options and featured for managing and reviewing analysis and design results.



- On the **Results** tab, click the **Envelope** spreadsheet button and select **Node Reactions.**

The spreadsheet will open to show all of the node reactions (both forces and moments) and the corresponding controlling load combinations. With the spreadsheet open, let's find the controlling vertical (Y) reaction in the model:

- Select the column with header **Y [k]**

- Right click anywhere in the column, select **Sort** from the dropdown and choose **Sort Max to Min**

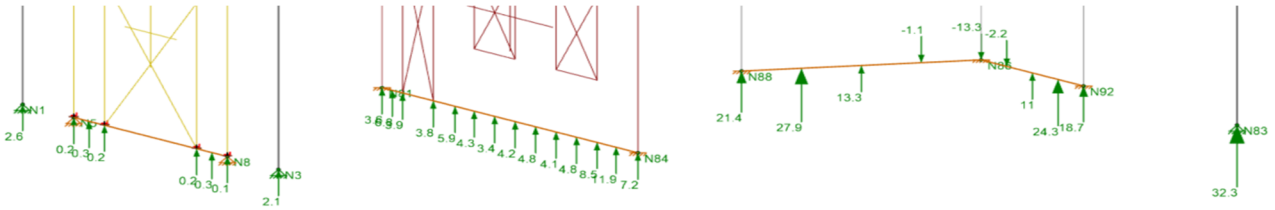
The spreadsheet should sort to identify the maximum vertical reaction found at node coordinate **(25,0,-40)** (base of concrete column, N83 in this example) in **LC2.**

Envelope Node Reactions

	Node La...		X [k]	LC	Y [k]	LC	Z [k]	LC	MX [k...	LC	MY [k...	LC	MZ [k-...
1	N83	max	-0.709	26	32.286	2	4.895	3	0	26	0	26	0
2		min	-1.971	3	11.894	25	1.81	25	0	1	0	1	0
3	N1	max	0.015	10	4.156	10	-0.033	26	0	26	0	26	0
4		min	0.002	25	1.048	25	-0.147	9	0	1	0	1	0

If we want to see this result graphically, we can use the **Quick View** buttons found on the **Home** tab.

- Click the **Reactions** button and select **Y Direction**
- On the **View Toolbar**, enable the visibility of the results by clicking on the **Results** button
- Select **Load Combination** and **LC2: IBC 16-2 (a)**



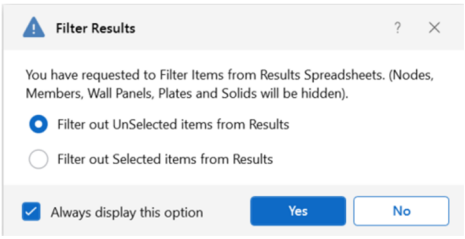
Note: At any time if you wish to clear the graphical window of all visible labels, loads, results, etc. you can click on the **Reset View** button found on the **Quick View** toolbar.

The same process can also be used to review member results. Let's utilize the wood truss **Selection State** that we created previously to quickly look at the forces in the wood truss members.

- Click on the **Save or Recall Selection States** button found on the **Selection Toolbar.**
- Select **Wood Truss** and click **Retrieve.**
- Click on the **Lock Selected Parts of the Model and Dim Unselected** button from the **Selection Toolbar** in order to only show the wood truss members as active.

Next, let's show the spreadsheet that includes the forces in only the wood truss members.

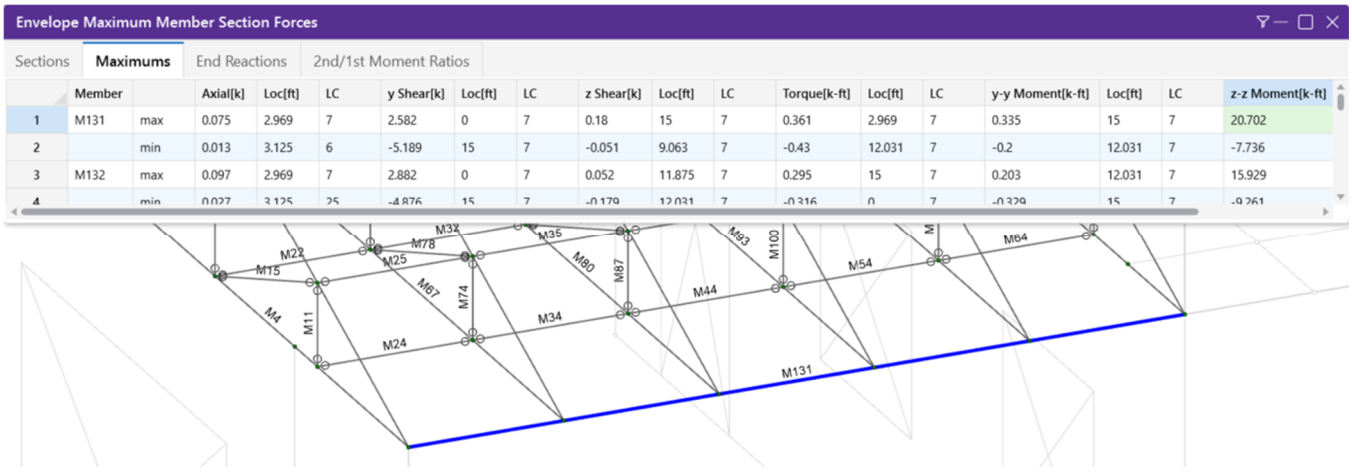
- On the **Results** tab, click on the **Filter Results** button.
- Select the option to **Filter Out UnSelected items from the Results** and then click **Yes.**



Now, click on the **Envelope** spreadsheet button and select **Member Forces.**

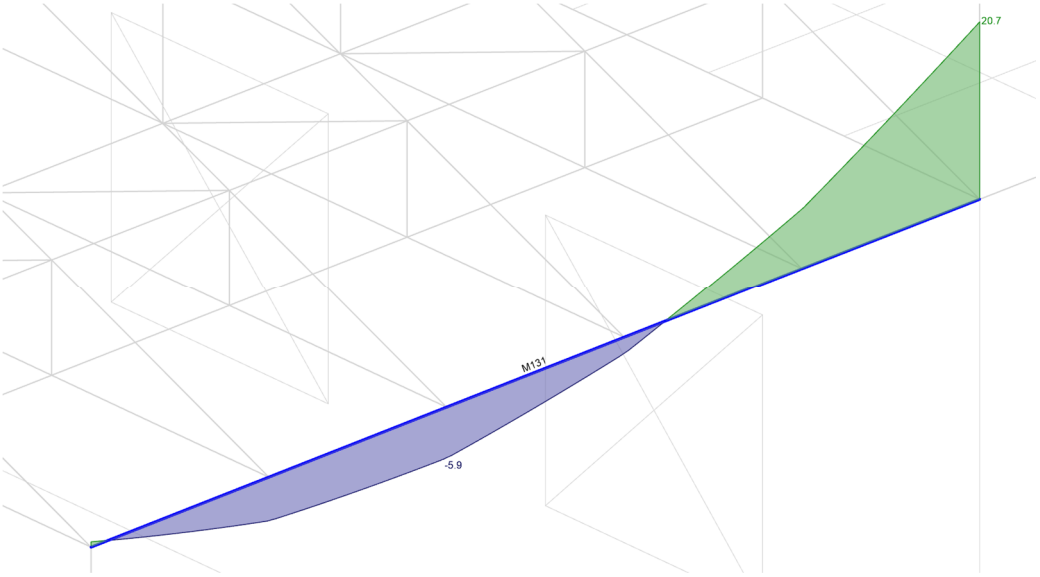
- Click on the **Maximums** tab
- Select the column with header **z-z Moment [k-ft]**
- Right click anywhere in the column, select **Sort** from the dropdown and choose **Sort Abs Max to Abs Min**

Since the result filter was enabled, the spreadsheet only shows member forces for the selected elements (wood truss members). This makes it much easier to review the data in the spreadsheet and identify the worst-case loading conditions, in this case the glulam beam has the largest positive moment in **LC22.**



Additionally, it is possible to review the forces graphically, using the **Quick View** buttons found on the **Home** tab.

- Click the **Member Forces** button and select **Mzz**
- Select **Load Combination** and choose **LC22: IBC 16-13** to see the matching result shown in the table above



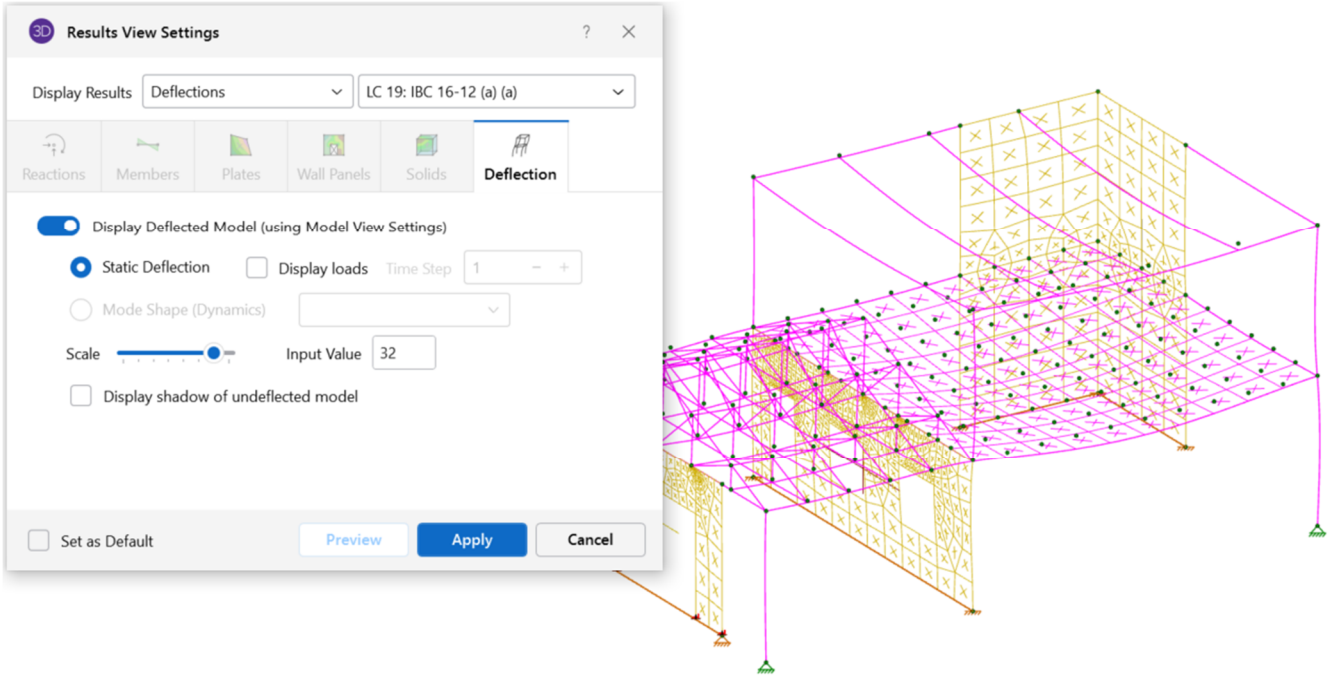
When your finished reviewing the member force results, click the **Turn OFF the Lock Selected Parts of the Model and Dim Unselected** button from the **Selection Toolbar** to toggle on the full model.

You can also disable the results filter by clicking the **Filter Results** button found on the **Results** tab.

Another important aspect in the evaluation of the model results, is the review of the deflected shape of the structure. To quickly access the deflected shape:

- Click the **Deflection** button on the **View Toolbar** to enable the visibility of the deflected shape.
- Select the **Load Combination** and **LC19: IBC 16-12 (a)(a)**

The deflected shape can be shown either with or without the undeflected shadow. Additionally, all properties of viewing the deflection graphically can be found clicking on the **Results** button in the **View Settings** section of the **View** tab and then clicking on the **Deflection** tab. There you can choose to enable the deflected shape, choose a load type and set the deflection scale. For this tutorial, the **Result View Settings** are set as shown below.



Note: There are two types of deflection output available: **With Undeflected Shadow** shows the deflected shape of the structure in addition to the "undeflected" shape shown in greyscale. **Without Undeflected Shadow** shows only the deflected shape of the structure.

Before disabling the deflected shape, let's create a **Snapshot** that we can incorporate into our report later. To do this, click on the **Camera Snapshot** button found on the **Quick Access Toolbar.**

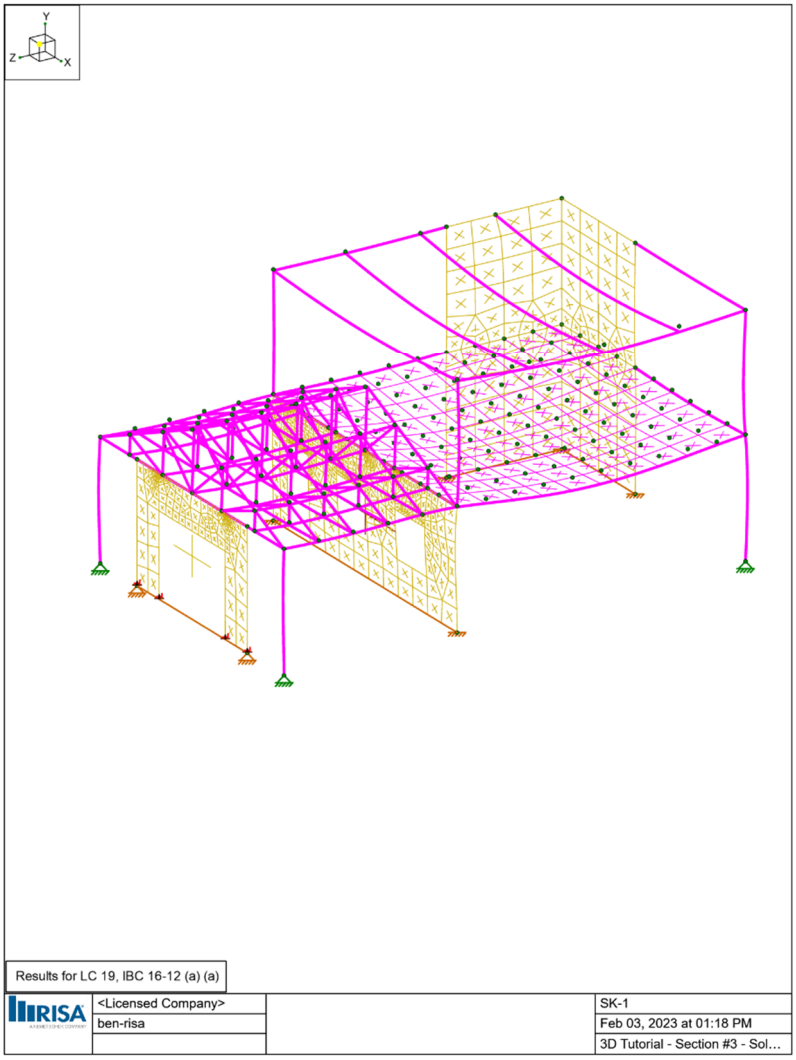
The **Snapshot Preview** window will open which includes the live preview of the image. This live preview allows you to pan, rotate and zoom in/out in order to properly position the image within the title block. Additionally, it is possible to change the visibility of loads, results and a variety of other input or output items using the **View Toolbar,** or the shortcuts found in the **Quick View** section of the **Home** tab.

The **Snapshot Preview** also includes page setup options including:

- Image file type: **PNG**
- Page Orientation: **Portrait** or **Landscape**
- Scale Factor: **Text, Symbols** and **Navigation Panel**
- Title Block Info: **Sketch #, Sketch Prefix, Font Size and Date/Time**

For this example, let's keep all of the page setup options as default and create the snapshot:

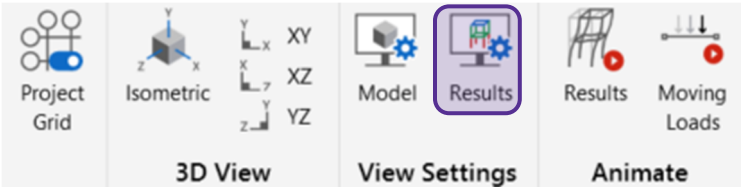
- Set the **Image File Name** to be **Deflected Shape – LC19**
- Position the image in the title block as shown on the next page
- Click the **Snapshot** button to create the snapshot



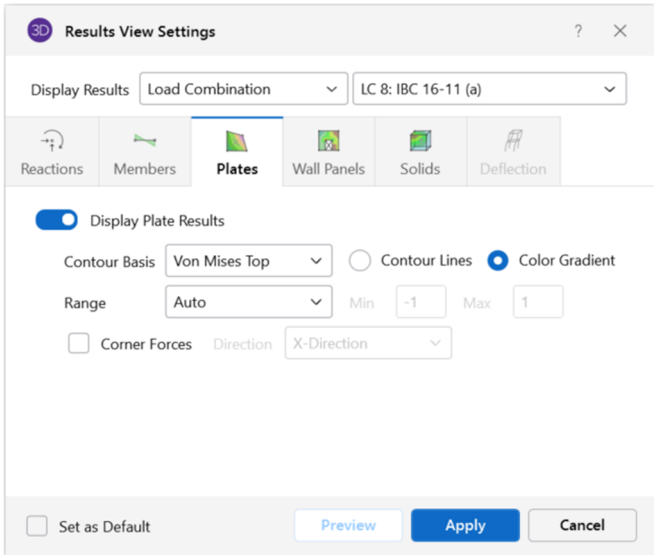
Note: If the model has yet to be saved when a Snapshot is created, the image will be saved in the **Model Data Files** location listed in **File > Application Settings > File Locations.** If the file has already been saved, then the image files will be saved in the same location as the .r3d file.

Once you have finished creating the **Snapshot** of the deflected shape, click on the **Deflection** button and select **None** to disable the deflection visibility.

Before evaluating design results, let's review the analysis results on the concrete plate elements. In this case, let's access the full view settings for plate results by clicking on the **Results** button in the **View Settings** section of the **View** tab.



Once the dialog opens, click on the **Plates** tab to see all the options for how the plate results on the model can be viewed.

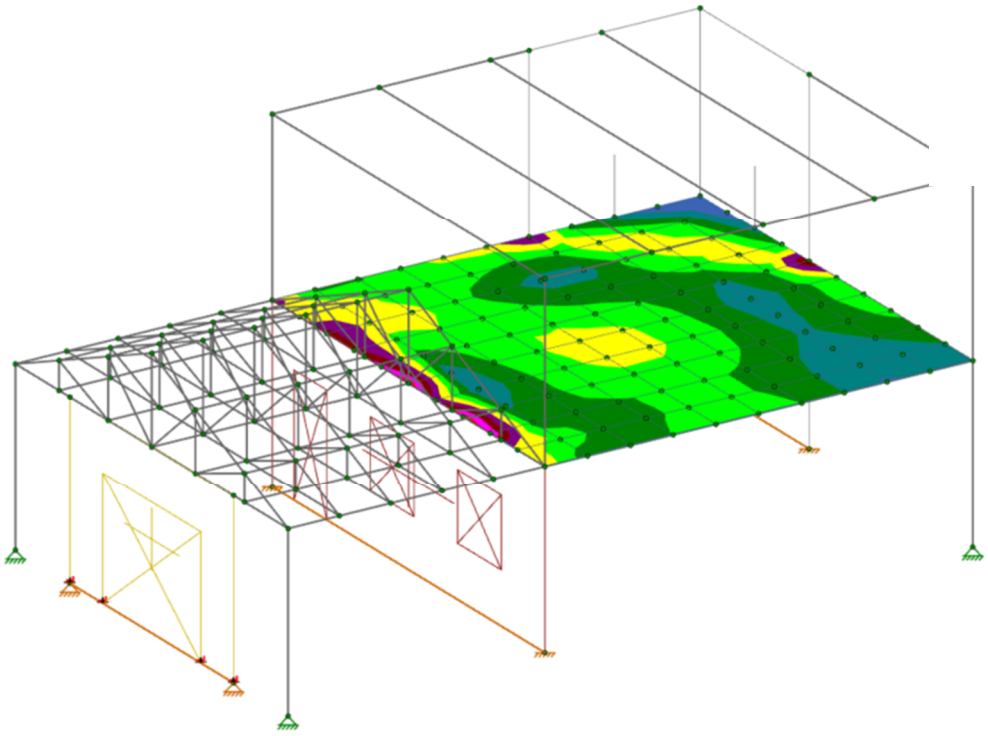


These options include the load type of result (load case, load category, load combination or envelope), the contour basis (Qx, My, Von Mises, etc.) and the plate corner forces. To see the results graphically, set up the dialog as shown in the image to the right and click **Apply.**

The color gradient contour plot for the result chosen will be shown graphically on the model with a corresponding key (see the image below).

The graphical results can also be viewed by using the **Quick View** button for **Plates** found on the **Home** tab.

The corresponding spreadsheet results can be found by clicking the **Envelope** or **Load Combination** results button and then selecting **Plate Stresses, Plate Forces** or **Plate Corner Forces.** These same spreadsheets can also be accessed by clicking on the **Plate Stresses, Plate Forces** or **Plate Corner Forces** buttons in the **Results** section of the **Explorer** toolbar.

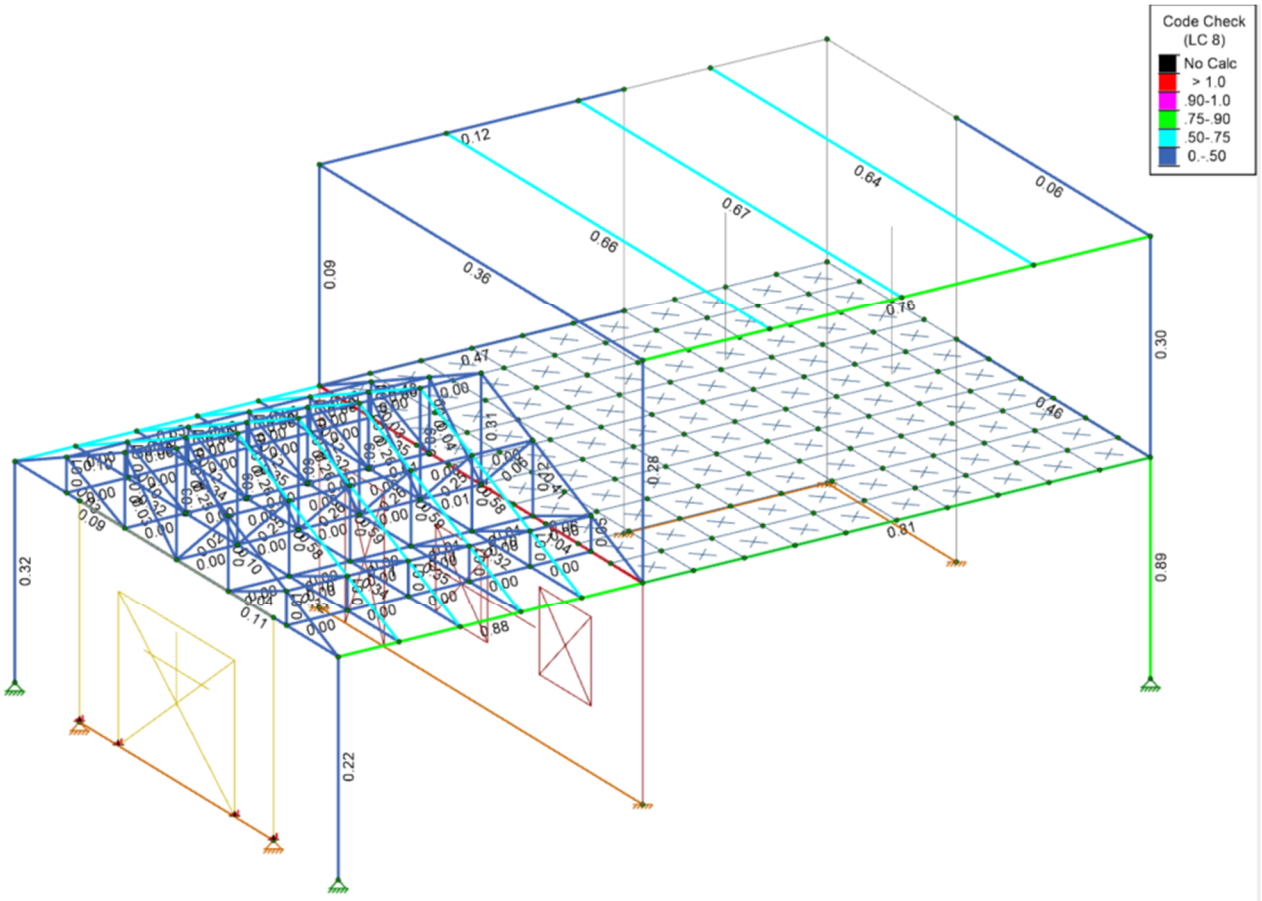


Design Results

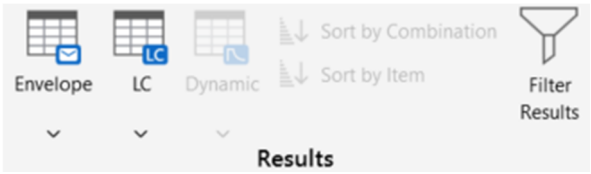
In addition to code independent results, RISA-3D allows users to easily evaluate design results for various materials graphically and in detailed reports as well as spreadsheets. Let's first look at how to quickly review the unity check values of members graphically:

- Open the **Home** tab and find the **Quick View** toolbar
- Click on the **Member Label** button and select **Unity Check**
- To enable the color coding, click on the **Member Color** button and select **Unity Check**

The 3D view will show the unity check values as well as the color coding, along with the color-coding key. The type of result can also be changed between **Load Case, Category, Load Combination** and **Envelope** by using the dropdown in the **View** toolbar.



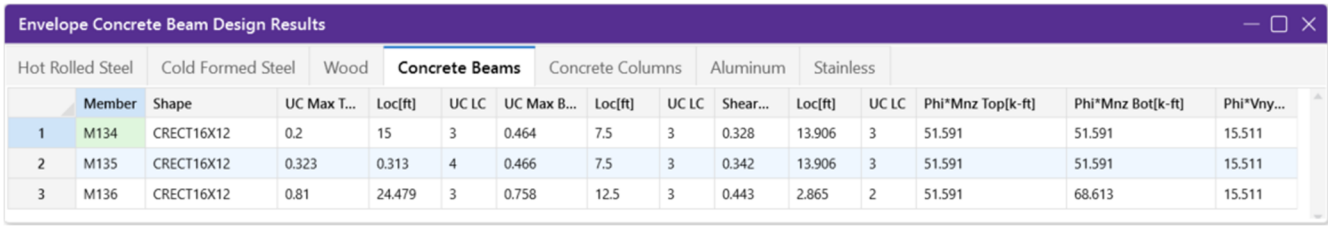
From the **Results** tab, you can also open the spreadsheets for **Results** for either a specific **Load Combination** or the **Envelope.**



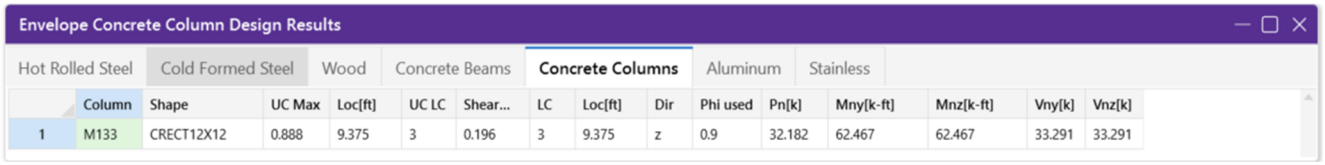
For this example, let's review the enveloped design results.

- Click on the **Envelope** spreadsheet results button and select **Code Check.**

The spreadsheet will open and includes tab for the various materials that can be designed in RISA-3D. Let's first review the results for the **Concrete Beams** and **Concrete Columns.**



	Member	Shape	UC Max T...	Loc[ft]	UC LC	UC Max B...	Loc[ft]	UC LC	Shear...	Loc[ft]	UC LC	Phi*Mnz Top[k-ft]	Phi*Mnz Bot[k-ft]	Phi*Vny...
1	M134	CRECT16X12	0.2	15	3	0.464	7.5	3	0.328	13.906	3	51.591	51.591	15.511
2	M135	CRECT16X12	0.323	0.313	4	0.466	7.5	3	0.342	13.906	3	51.591	51.591	15.511
3	M136	CRECT16X12	0.81	24.479	3	0.758	12.5	3	0.443	2.865	2	51.591	68.613	15.511



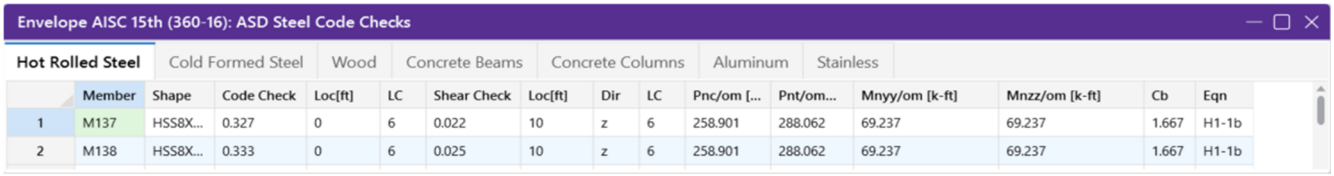
	Column	Shape	UC Max	Loc[ft]	UC LC	Shear...	LC	Loc[ft]	Dir	Phi used	Pn[k]	Mny[k-ft]	Mnz[k-ft]	Vny[k]	Vnz[k]
1	M133	CRECT12X12	0.888	9.375	3	0.196	3	9.375	z	0.9	32.182	62.467	62.467	33.291	33.291

These spreadsheets show the basic information for how the unity checks for bending and shear are determined.

Next, switch to **Hot Rolled Steel** tab to begin to review the design results for the steel beams and columns.

- Right click on the column heading **Code Check** and select **Sort Max to Min** to see the unity checks for each steel member in descending order.

Note: Spreadsheets are also available in the **Code Check** section of the **Explorer Panel.**



	Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Pnc/om [...	Pnt/om...	Mnyy/om [k-ft]	Mnzz/om [k-ft]	Cb	Eqn
1	M137	HSS8X...	0.327	0	6	0.022	10	z	6	258.901	288.062	69.237	69.237	1.667	H1-1b
2	M138	HSS8X...	0.333	0	6	0.025	10	z	6	258.901	288.062	69.237	69.237	1.667	H1-1b

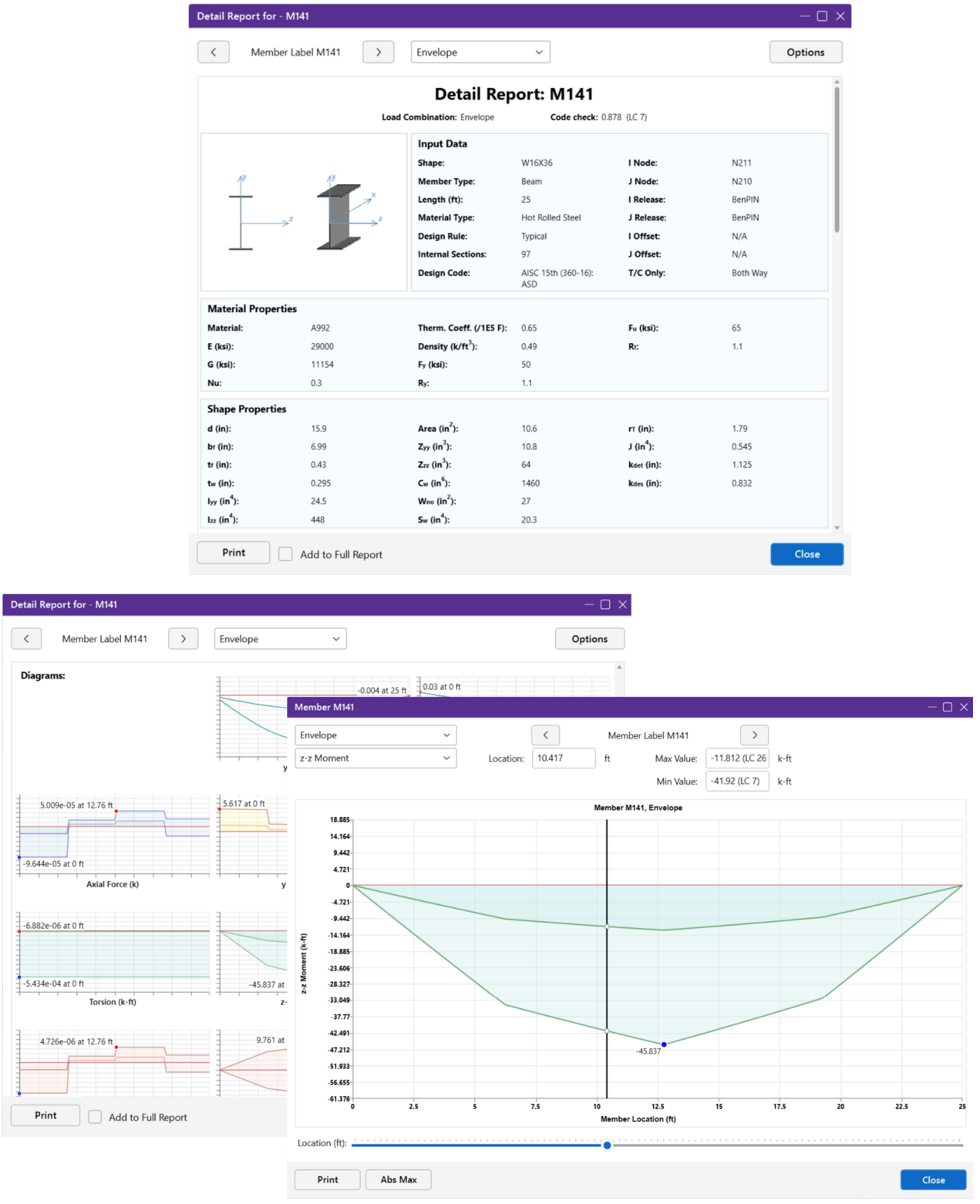
While the spreadsheet for the results gives you a significant amount of data regardless of which material you are looking at, it may be necessary to obtain a deeper understanding of how the calculated values were determined. To do this, we can view the **Detailed Report** for a given member.

The **Detailed Report** is accessible in a variety of different ways including:

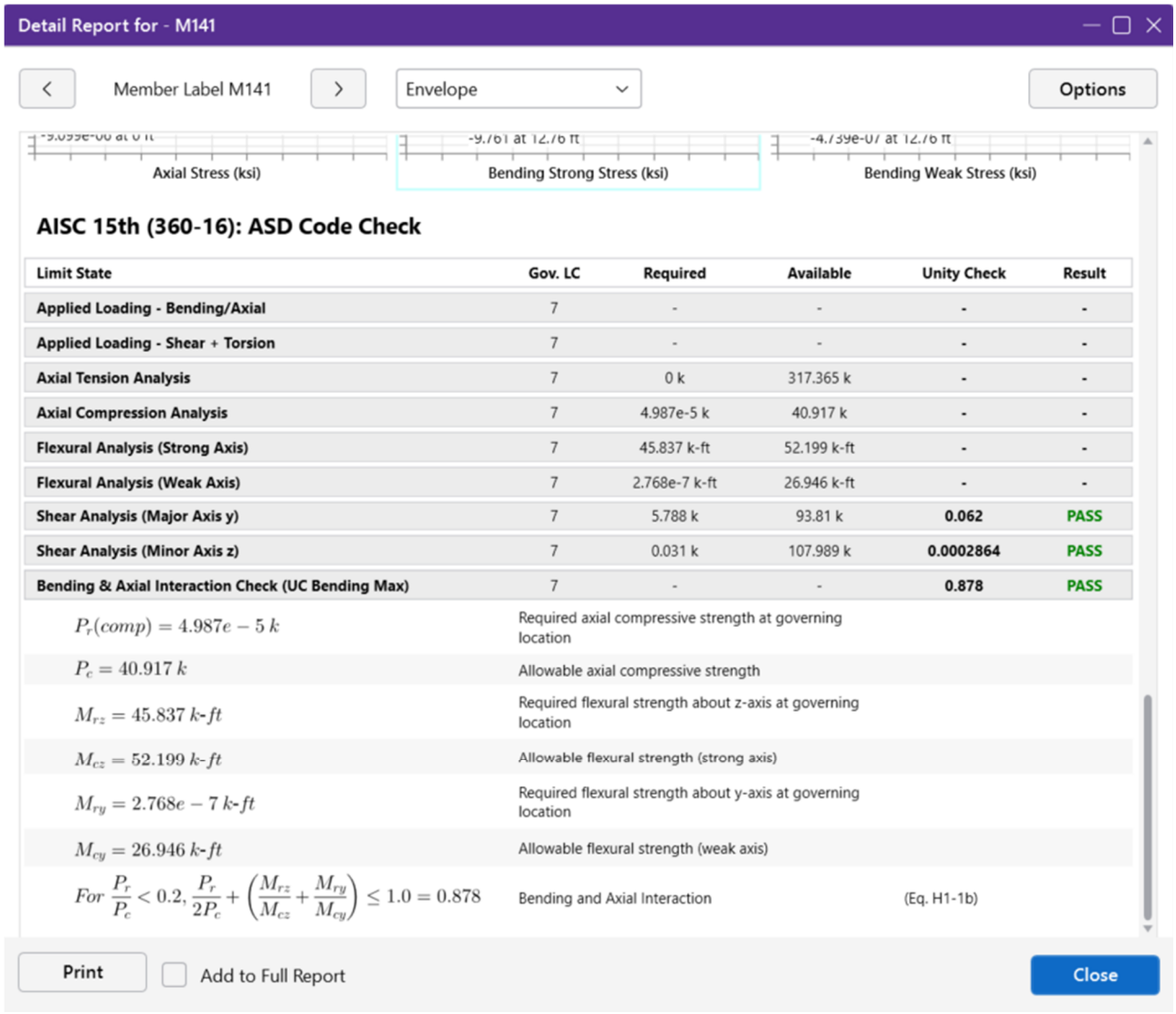
- Right clicking in on a specific member row in the spreadsheet and selecting **Detailed Report**
- Clicking and selecting a specific member row in the spreadsheet and clicking the **Detailed Report** button found on the **Results** tab.
- Clicking on the **Detailed Report** button found on the **Design Results Toolbar.**

For this example, open the **Detailed Report** for the steel member with the worst-case unity check.

When the detailed report opens you can see sections for **Input Data, Force Diagrams** and **Limit State Checks** (see the images below). The force diagrams can be expanded by clicking on them individually allowing users to evaluate results at any location along the length of a member.

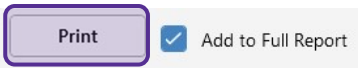


The expandable limit states provide users with all governing equations and properties, allowing users to easily verify member designs.



Additionally, users can click the **Options** button to modify the report sections that are included. In this example, let's click **Options** and enable the **Code Check** option **Expanded.** This will automatically expand all limit states shown in the detailed report.

Before closing the **Detailed Report** for the steel member, let's add it to our full report. To do this, check the **Add to Full Report** checkbox at the bottom of the **Detailed Report.**



Detailed reports are also available for all other materials and member types in RISA-3D. Let's open a **Detailed Report** for a wood shear wall.

- Click on the **Detailed Report** button found on the **Design Results Toolbar.**
- Click on the wood wall panel, **WP1.**

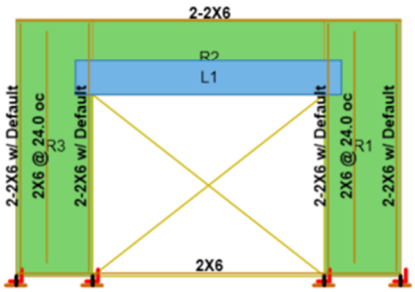
The detailed report for the wood wall panel, **WP1** will open and will include input data, material properties, enveloped wall results and individual design results for regions.

Detail Report for - WP1

Wall Panel Label WP1 | Wall

Detail Report: WP1

Wood Wall



GENERAL

Code:	AWC NDS-18 / SDPWS-15:ASD
Design Method:	Segmented
Wall Material:	DF
Panel Schedule:	AWC 2015 OSB
Sel. Shear Panel:	RS_19/32_10d@6

GEOMETRY

Total Height (ft):	10
Total Length (ft):	15
Max H/W Ratio:	3.33

MATERIALS

Top Pl.	DF	2-2X6
Sill	DF	2X6
Wall Stud	DF	2X6
Chord	DF	2-2X6

Print | Add to Full Report | Close

ENVELOPED RESULTS

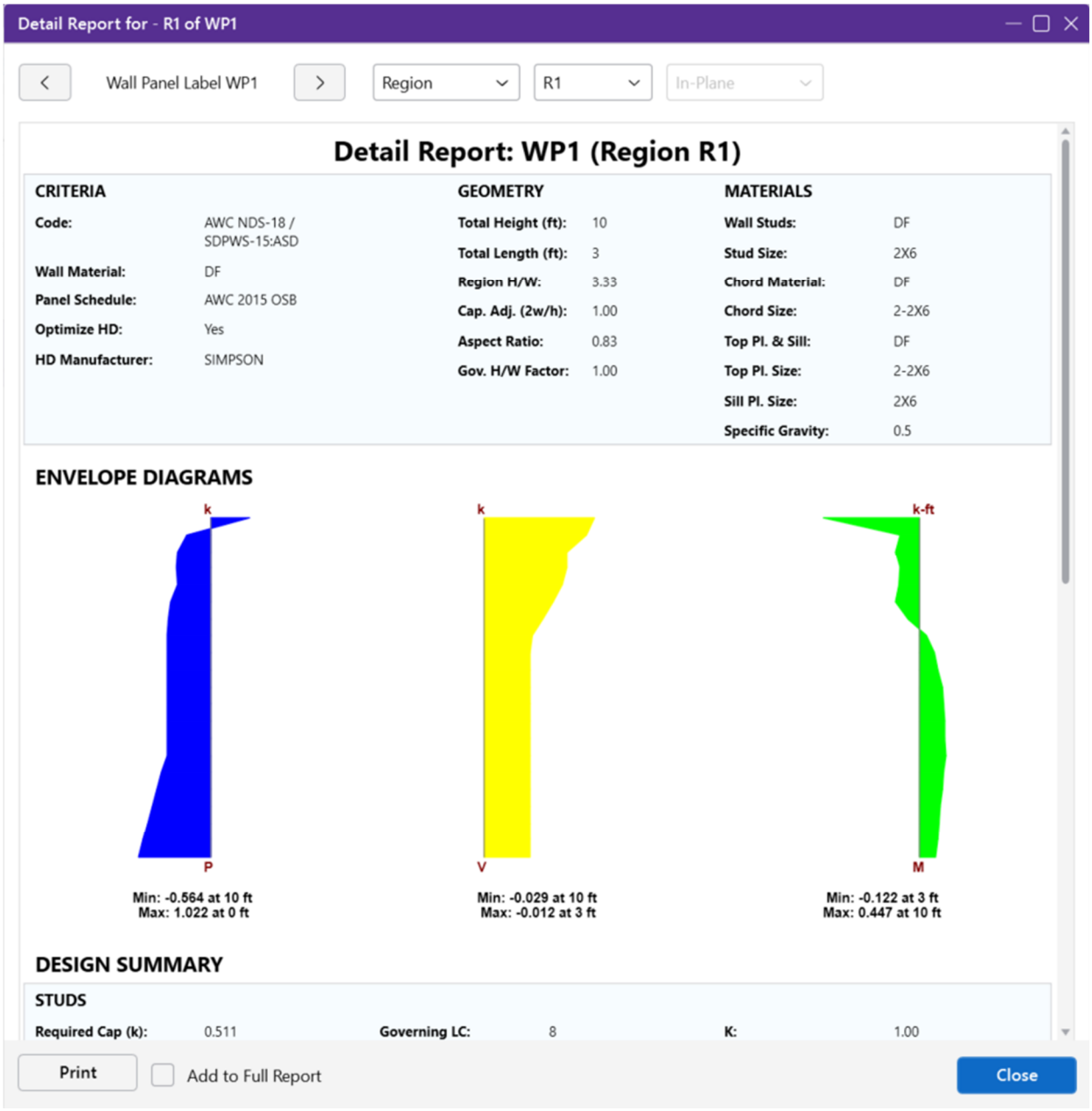
Controlling Shear Region	Shear Panel	Shear UC	Shear LC	Controlling Hold Down	Hold Down UC	Hold Down LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	RS_19/32_10d@6	0.028	7	Not Req'd	NC	NC	0.042	6	0.064	8

REGION INFORMATION

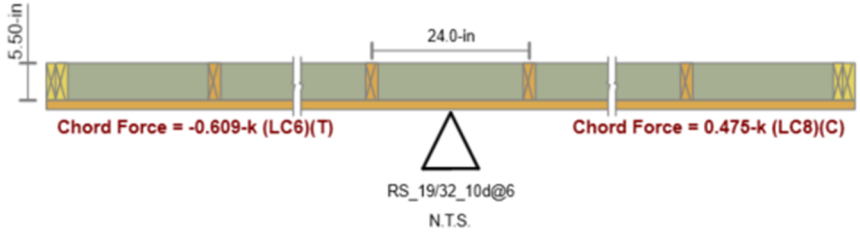
Region Label	H/W Ratio	Shear UC	Shear LC	Hold-Down	Hold-Down UC	Hold-Down LC	Chord UC	Chord LC	Stud UC	Stud LC
R1	3.33	0.028	7	Not Req'd	NC	NC	0.042	6	0.063	8
R3	3.33	0.027	7	Not Req'd	NC	NC	0.042	6	0.064	8

DEFLECTION RESULTS

Maximum Region Deflection (in)	Deflection LC	Finite Element Deflection (in)	Shear Stiffness Adjustment Factor
0.006 (R1)	7	0.0003374	1



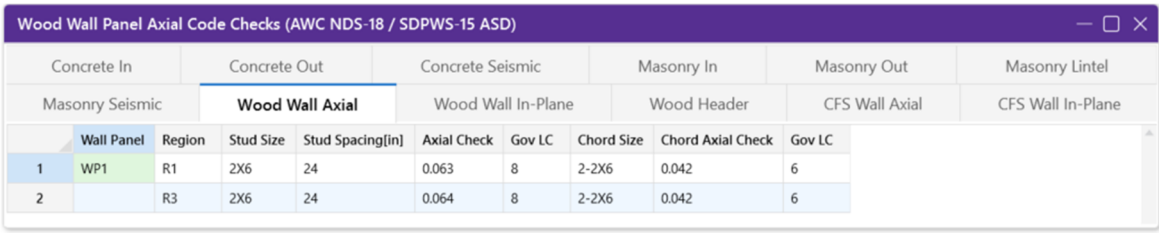
The last section of the detail report consists of the wall detailing information. This information is provided as a visual confirmation of the wall design with the wall thickness and stud spacing shown as dimensions. The triangle shows sheathing on one side of the wall, with the abbreviated panel designation. Additionally, the chord sizes/forces and hold down or strap designations/forces are shown at either end. If either chord is only experiencing a compression force, the hold down or strap will not be drawn.



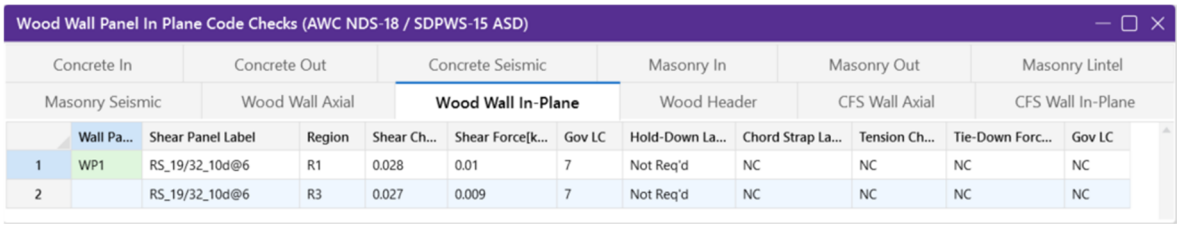
In addition to the detailed output for wall panels, spreadsheet results are also available for all wall panels. These spreadsheet results can be accessed from the **Results** tab using either the **Envelope** or **LC**

dropdowns and selecting **Wall Panel Design.** The same spreadsheet can also be found in the **Explorer Panel** by clicking on **Wall Panel Design.**

In this example, we can review the results for **Wood Wall Panel, WP1.** The spreadsheet includes separate tabs for **Wood Wall Axial, Wood Wall In-Plane** and **Wood Header** (in addition to tabs for the other wall panel design types available in RISA-3D).



	Wall Panel	Region	Stud Size	Stud Spacing[in]	Axial Check	Gov LC	Chord Size	Chord Axial Check	Gov LC
1	WP1	R1	2X6	24	0.063	8	2-2X6	0.042	6
2		R3	2X6	24	0.064	8	2-2X6	0.042	6



	Wall Pa...	Shear Panel Label	Region	Shear Ch...	Shear Force[k...	Gov LC	Hold-Down La...	Chord Strap La...	Tension Ch...	Tie-Down Forc...	Gov LC
1	WP1	RS_19/32_10d@6	R1	0.028	0.01	7	Not Req'd	NC	NC	NC	NC
2		RS_19/32_10d@6	R3	0.027	0.009	7	Not Req'd	NC	NC	NC	NC

Section 4: Reports

When the analysis and design for a project is complete, a report can be assembled in order to document the design. This report can include all input and output data as well as additional items such as images of the model. Before creating a report, let's create an image of the model to include in our report.

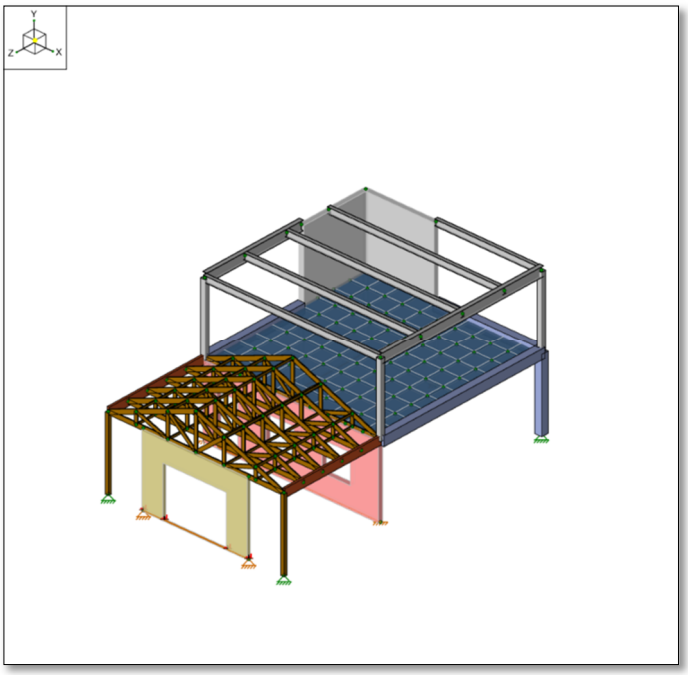
- Click on the **Reset View** button found on the **Quick View** toolbar to reset the 3D view and show only the rendered model.
- On the **Quick Access Toolbar**, click the **Camera Snapshot** button

The **Snapshot Preview** window will open and show a dynamic preview of the model. This preview allows you to position the model exactly as you want while also enabling/disabling various visibility items include loads, results, model rendering, labels, etc.

Additionally, the options for the snapshot can also be set including the orientation, paper size, scaling and image file type. For this tutorial, let's create an image with the following properties:

- Input an **Image File Name** of **3D Model View**
- Set the **Image File Type** to **PNG**
- Set the **Orientation** to **Portrait**
- Set the **Text Scale** to 1, the **Symbols, Diagrams Scale** to **2** and the **Navigation Panel Scale** to **5**
- Click on the **Snapshot** button in the top left corner of the **Properties Panel** to save the snapshot shown in the image below.

The snapshot preview should now close and your snapshot should be saved either in the default directory or in the same location as the project file.

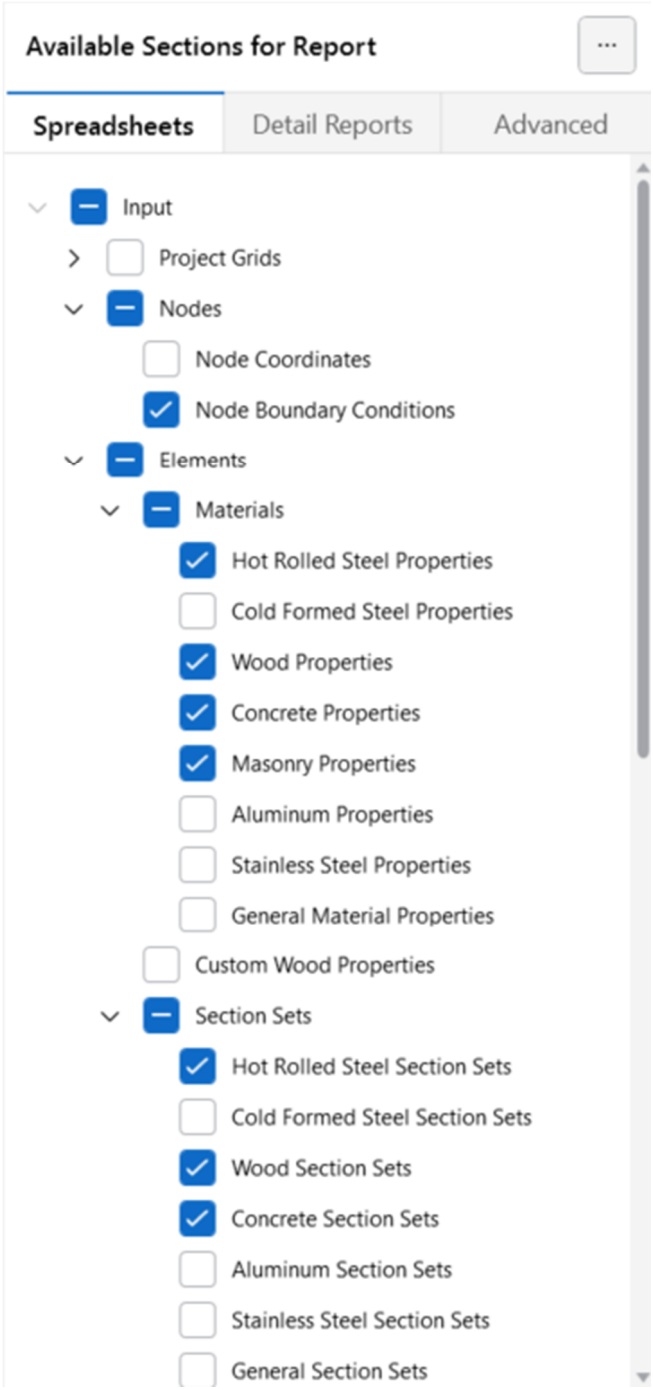


With the snapshot created, let's create a **Report.** To do this, click the **Print Report/Graphic** button on the **Quick Access Toolbar** and select **Print Reports.**

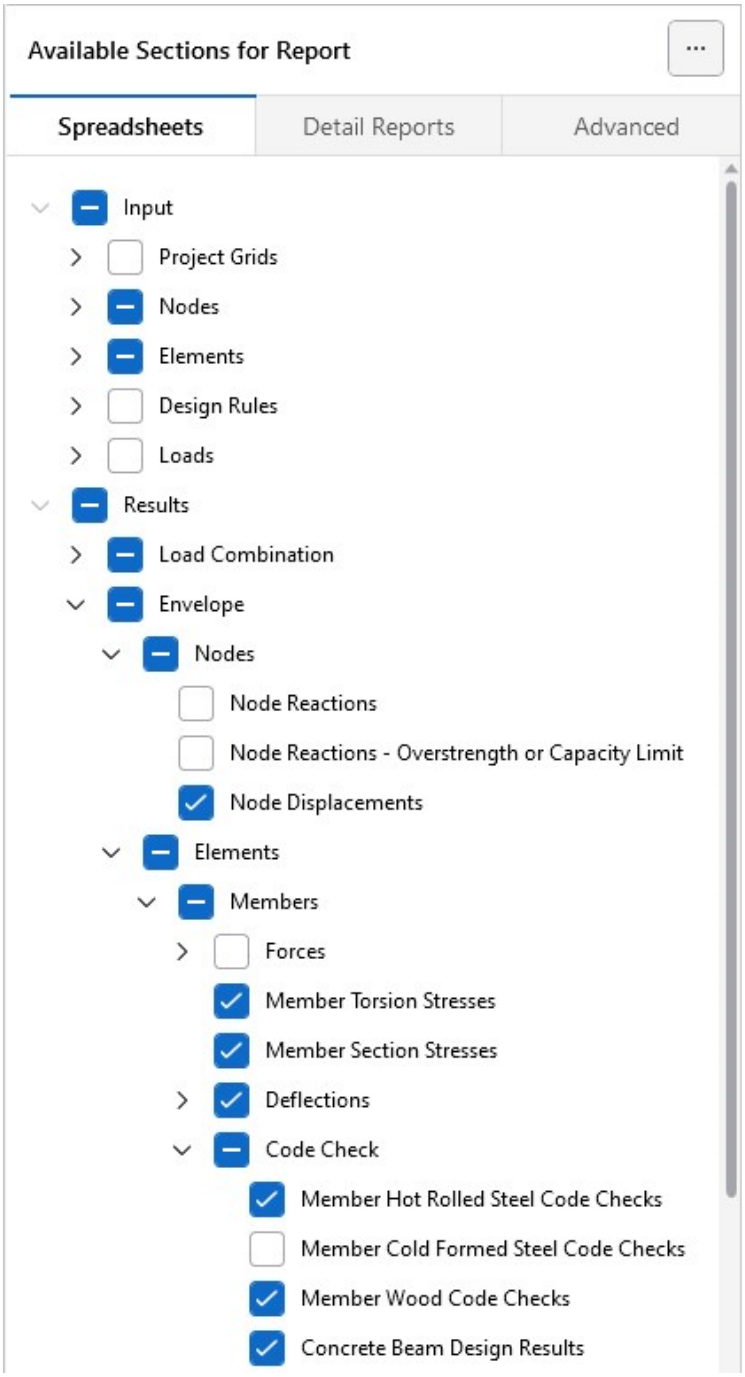
The **Report Printing** interface will open, and the user can either select a pre-generated report from the list of **Report Templates** or build a report from scratch. For this tutorial, let's build a report from scratch.

The **Report Printing** interface has three main sections: the list of **Available Sections for Report**, the **Current Sections in Report** and the **Report Preview.**

To begin creating a report, click on the **Spreadsheets** button in the **Available Sections for Report** with expandable lists for both input and results available. Let's first add our **Input** spreadsheets as shown in the image below:



When each spreadsheet is added, the corresponding item will be added in the **Current Sections in Report** section. Next, collapse the **Input** spreadsheets and expand the **Results** spreadsheets to be added as shown in the image below:



In addition to each spreadsheet being added to the **Current Sections in Report** section, the **Report Preview** also expands to include the added spreadsheets. This allows users to review the data in each spreadsheet prior to printing to ensure that the correct spreadsheet and corresponding data is included.

With both the Input and Results spreadsheets added to the current report, let's create a **Report Filter.** The **Report Filter** works in a similar fashion to the **Filter Results** feature that was used previously. For this example, let's create a filter for the **Results – Envelope – Elements – Members – Code Check – Wood Code Checks** spreadsheet so that we show only design results for the glulam beam members.

- Select the spreadsheet mentioned above from the **Current Sections in Report** list
- Right-click on the selection and choose **Edit Filter**

- Choose the **Members** tab and enable the **Section Set/Shape** filter
- Choose **Glulam Beam** from the dropdown and click **Filter**

You can see how the filter has modified the spreadsheet result using the **Report Preview** (page 31 – may be slightly different based on inputs and results added to report).

Envelope AISC 15TH (360-16): ASD Member Steel Code Checks

	Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Pnc/om [k]	Pnt/om [k]	Mnyy/om [k-ft]	Mnzz/om [k-ft]	Cb	Eqn
1	M137	HSS8X8X5_A1085	0.327	0	6	0.022	10	z	6	258.901	288.062	69.237	69.237	1.667	H1-1b
2	M138	HSS8X8X5_A1085	0.333	0	6	0.025	10	z	6	258.901	288.062	69.237	69.237	1.667	H1-1b
3	M139	HSS8X8X5_A1085	0.129	0	6	0.008	10	z	6	258.901	288.062	69.237	69.237	1.666	H1-1b
4	M140	W16X36	0.14	6.25	7	0.051	15	y	7	113.659	317.365	26.946	122.369	1.224	H1-1b
5	M141	W16X36	0.878	12.76	7	0.062	25	y	7	40.917	317.365	26.946	52.199	1.145	H1-1b
6	M142	W10X26	0.415	12.5	7	0.036	25	y	7	23.548	227.844	18.713	28.414	1.136	H1-1b
7	M143	W10X26	0.072	7.5	7	0.021	15	y	7	65.412	227.844	18.713	54.995	1.136	H1-1b
8	M144	W10X26	0.772	12.5	7	0.068	25	y	7	23.548	227.844	18.713	28.414	1.136	H1-1b
9	M145	W10X26	0.787	12.5	7	0.068	25	y	7	23.548	227.844	18.713	28.414	1.136	H1-1b
10	M146	W10X26	0.745	12.5	7	0.065	25	y	7	23.548	227.844	18.713	28.417	1.136	H1-1b

Envelope AWC NDS-18 / SDPWS-15 ASD Member Wood Code Checks

	Member	Shape	Code Check	Loc[ft]	LC	Shear Check	Loc[ft]	Dir	LC	Fc' [ksi]	Ft' [ksi]	Fb1' [ksi]	Fb2' [ksi]	Fv' [ksi]	RB	CL	CP	Eqn
131	M131	3.5X13.75FS	0.884	15	8	0.722	15	y	7	1.87	1.375	2.82	1.813	0.375	14.214	0.94	0.935	3.9-3
132	M132	3.5X13.75FS	0.703	15	7	0.605	15	y	7	1.87	1.375	2.82	1.813	0.375	14.214	0.94	0.935	3.9-3

Envelope Concrete Beam Design Results

	Member	Shape	UC Max Top	Loc[ft]	UC LC	UC Max Bot	Loc[ft]	UC LC	Shear UC	Loc[ft]	UC LC	Phi*Mnz Top[k-ft]	Phi*Mnz Bot[k-ft]	Phi*Vny[k]
1	M134	CRECT16X12	0.2	15	3	0.464	7.5	3	0.328	13.906	3	51.591	51.591	15.511
2	M135	CRECT16X12	0.323	0.313	4	0.466	7.5	3	0.342	13.906	3	51.591	51.591	15.511
3	M136	CRECT16X12	0.81	24.479	3	0.758	12.5	3	0.443	2.865	2	51.591	68.613	15.511

Next, let's add detailed reports to the overall report. Before adding new detailed reports, let's move the detailed report that we added earlier so that it is after all the Input and Results spreadsheet.

- Click and hold on the only detailed report item **Detail – Beam – Envelope – Custom – Member #** found in **Current Sections in Report** (the member number is dependent on the label of the individual members in the user's model)
- Drag the item to the end of the report (it is possible to drop the item anywhere in the report)

Now, to add new detailed reports, click on the **Detailed Reports** tab. In the **Detailed Reports** tab, there are various options for how the reports that are available will be grouped or what level of detail will be shown in each report. These options are as follows:

- **Element Type:** Select either Member or Wall Panel
- **Result Type:** Select Load Combination, Batch, Envelope or Batch + Envelope
- **Detailed Report Type:** Select Summary or Custom (for Custom, the specific sections you wish to include in your report can be managed by clicking the ellipsis ... button.

For this tutorial, let's first add a glulam beam **Member** detailed report for the controlling load combination.

- Set the **Element Type** to **Member**
- Set the **Result Type** to **Load Combination**
- Set the **Load Combination** to **LC8: IBC 16-11 (a)**
- Set the **Detailed Report Type** to **Summary**
- Choose **Detailed – Beam – LC 8: IBC 16-11 (a) – Summary – M131**

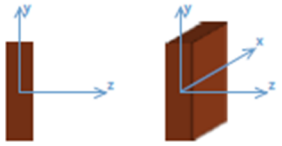
Following the above steps should add the following report to the active report:

Company : <Licensed Company>
Designer : ben-risa
Job Number :
Model Name :

2/3/2023
3:05:24 PM
Checked By :

Detail Report: M131

Load Combination: LC 8: IBC 16-11 (a) **Code check:** 0.884 (axial/bending)

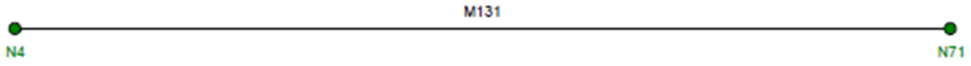


Input Data

Shape:	3.5X13.75FS (nominal)	**I Node:**	N4
Member Type:	Beam	**J Node:**	N71
Length (ft):	15	**I Release:**	Fixed
Material Type:	Wood	**J Release:**	Fixed
Design Rule:	Typical	**I Offset:**	N/A
Internal Sections:	97	**J Offset:**	N/A
Design Code:	AWC NDS-18 / SDPWS-15 ASD	**T/C Only:**	Both Way

Material Properties

Material:	24F-1.8E SP Balanced	**Grade:**	na	**Nu:**	0.3
Type:	Glulam	**Cm:**	No	**Alpha ($1e^{-6}$°F^{-1}):**	0.3
Database:	NDS Table 5A	**Ci:**	No	**Density (k/ft^3):**	0.035
Species:	24F-1.8E_SP_BAL	**Emod:**	1		



AWC NDS-18 / SDPWS-15 ASD Code Check

Limit State	Required	Available	Unity Check	Result
Applied Loading - Bending/Axial	-	-	-	-
Applied Loading - Shear + Torsion	-	-	-	-
Axial Compression Analysis	0.001 ksi	1.87 ksi	-	-
Axial Tension Analysis	0 ksi	1.375 ksi	-	-
Flexural Analysis, Fb1'	2.253 ksi	2.82 ksi	-	-
Flexural Analysis, Fb2'	0.123 ksi	1.812 ksi	-	-
Bending & Axial Compression Analysis	-	-	**0.884**	PASS
Bending & Axial Tension Analysis	-	-	**0.867**	PASS
Shear Analysis	0.231 ksi	0.375 ksi	**0.616**	PASS

RISA-3D Version 20 [3D Tutorial - Section #3 - Solution & Res... Page 1

Next, let's first add a concrete **Wall Panel** detailed report for the controlling load combination.

- Set the **Element Type** to **Wall Panel**
- Set the **Detailed Report Type** to **Summary**
- Select **Wall – Summary – WP3**

Following the above steps should add the following report to the active report:

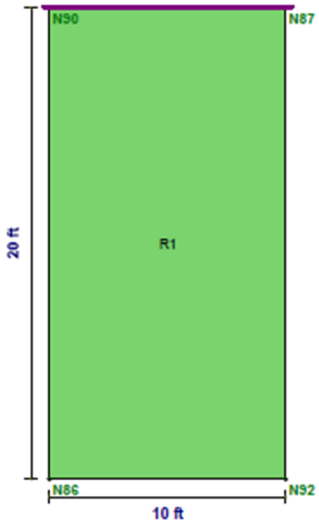
RISA
A NEMETSCHEK COMPANY

Company : <Licensed Company>
Designer : ben-risa
Job Number :
Model Name :

2/3/2023
3:07:27 PM
Checked By :

Detail Report: WP3

Concrete Wall



CRITERIA		GEOMETRY		MATERIALS	
Code:	ACI 318-14	**Total Height (ft):**	20	**Material Set:**	Conc4000NW
Design Rule:	Concrete Wall Typ	**Total Length (ft):**	10	**Concrete f'c (ksi):**	4
Seismic Rule:	None	**Thickness (in):**	8	**Concrete E (ksi):**	3644
Loc of r/f:	Each Face	**Int Cover (-z) (in):**	1	**Concrete G (ksi):**	1584
Outer Bars:	Vertical	**Ext Cover (+z) (in):**	1	**Conc Density (k/ft³):**	0.145
Vert Bar Size:	#7	**Cover Open/Edge (in):**	2	**Lambda:**	1
Horz Bar Size:	#4	**K:**	0	**Conc Str Blk:**	Rectangular
Transfer In?:	No	**Use Cracked?:**	Yes	**Vert Bar Fy (ksi):**	60
Transfer Out?:	No	**In Icr Factor:**	0.7	**Horz Bar Fy (ksi):**	60
Group Wall?:	No	**Out Icr Factor:**	0.35	**Steel E (ksi):**	29000

REGION RESULTS

Region	UC Max In Plane	LC	UC Shear In Plane	LC	Delta Max In Plane(in)	LC	UC Max Out Plane	LC	UC Shear Out Plane	LC	Delta Max Out Plane(in)	LC
R1	0.076	2	0.058	2	0.016	2	0.183	2	0.115	3	0.044	2

REINFORCEMENT RESULTS

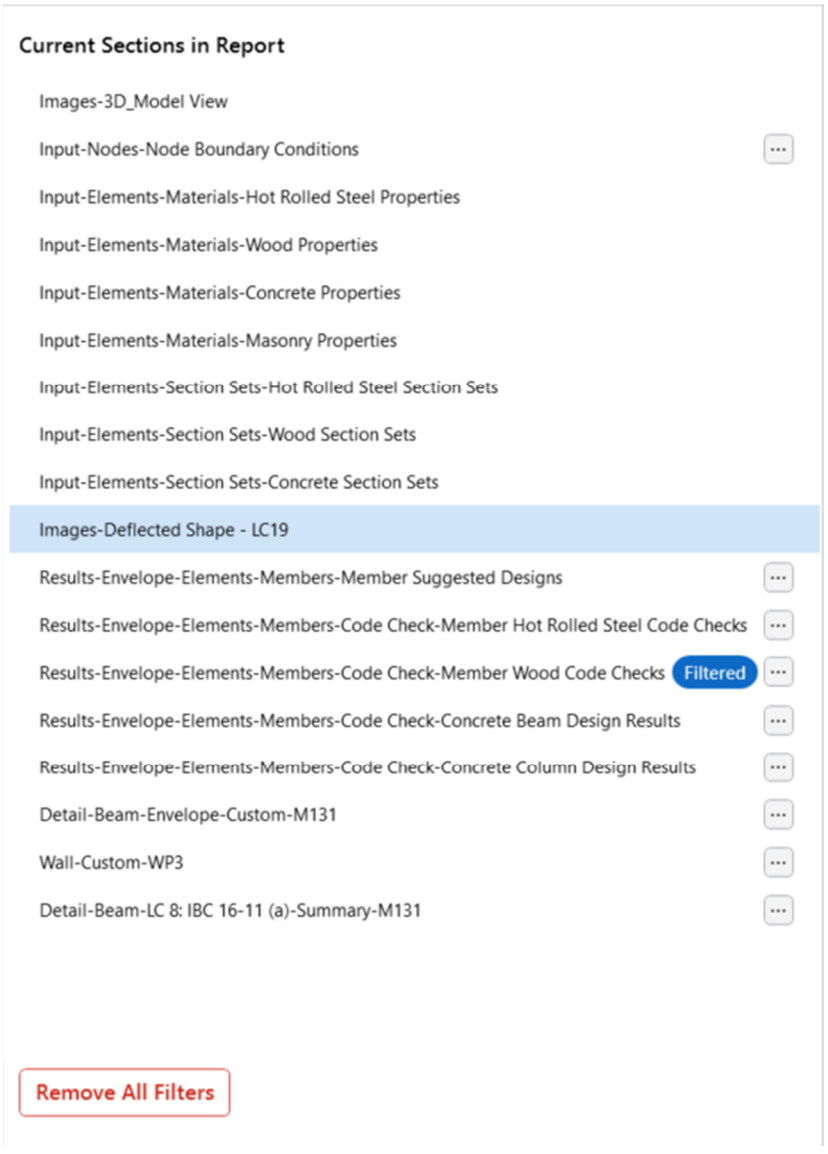
Region	Vertical Reinforcement	Horizontal Reinforcement	Diagonal Reinforcement
R1	#7@18in oc e.f.	#4@18in oc e.f.	N/A

RISA-3D Version 20 [3D Tutorial - Section #3 - Solution & Res... Page 1

With the detailed reports for various member and material types added, we can now add other items found in the **Advanced** section. This section includes items such as seismic, wind and notional load generation results, model settings, dynamic information and images. In this example, let's add the two **Snapshot** images that we created previously:

- Click the arrow next to **Images** to expand the list
- Select both images by clicking them
- Click and hold on the **3D Model View.png** and drag the item to the beginning of the report
- Click and hold on the **Deflected Shape – LC19.png** and drag the item so that it is following the **Results – Envelope – Nodes – Node Displacements** spreadsheet in the report.

When you are finished adding the images, your **Current Sections in Report** list should look like this:

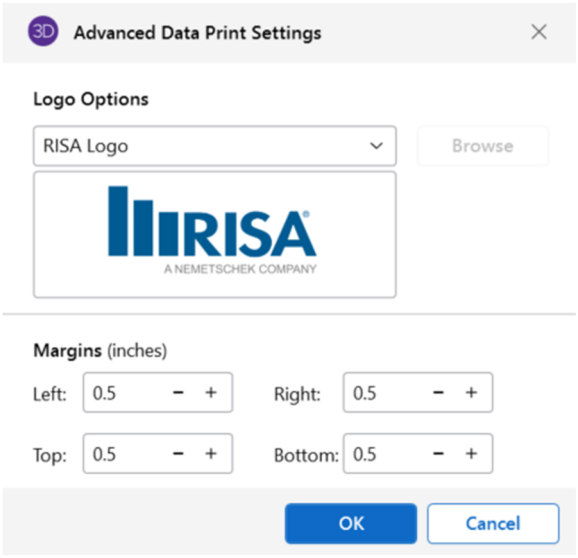


Note: It is possible that the member labels shown above are different from the ones found in a specific users model. This is most likely due to the order in which the model was created.

With all the required items now added to the report, the report can now be printed (or saved). On the far-left hand side of the **Report Printing** interface, the **Print** section gives the user the option to modify the page layout settings (orientation, paper type and pixel density) as well as the whether to include the header.

Additional **Advanced Settings** exist which allow for the use of a **Custom Logo** and the modification of the **margins.**

At this point, utilize your default **PDF** printer to print the custom report as a PDF by clicking the **Print** button.



Conclusion

Congratulations on completing the RISA-3D tutorial. We appreciate your desire to take the time to learn the software and are confident that the knowledge gained during this tutorial will help increase your productivity and allow you to complete future projects more efficiently.

Finally, while the tutorial is an in-depth look at how to model, analyze, design and document a simple structure in RISA-3D, we understand that there may be topics that were not covered. For any feature specific or technical software questions please reach out to our support team (support@risa.com) directly.