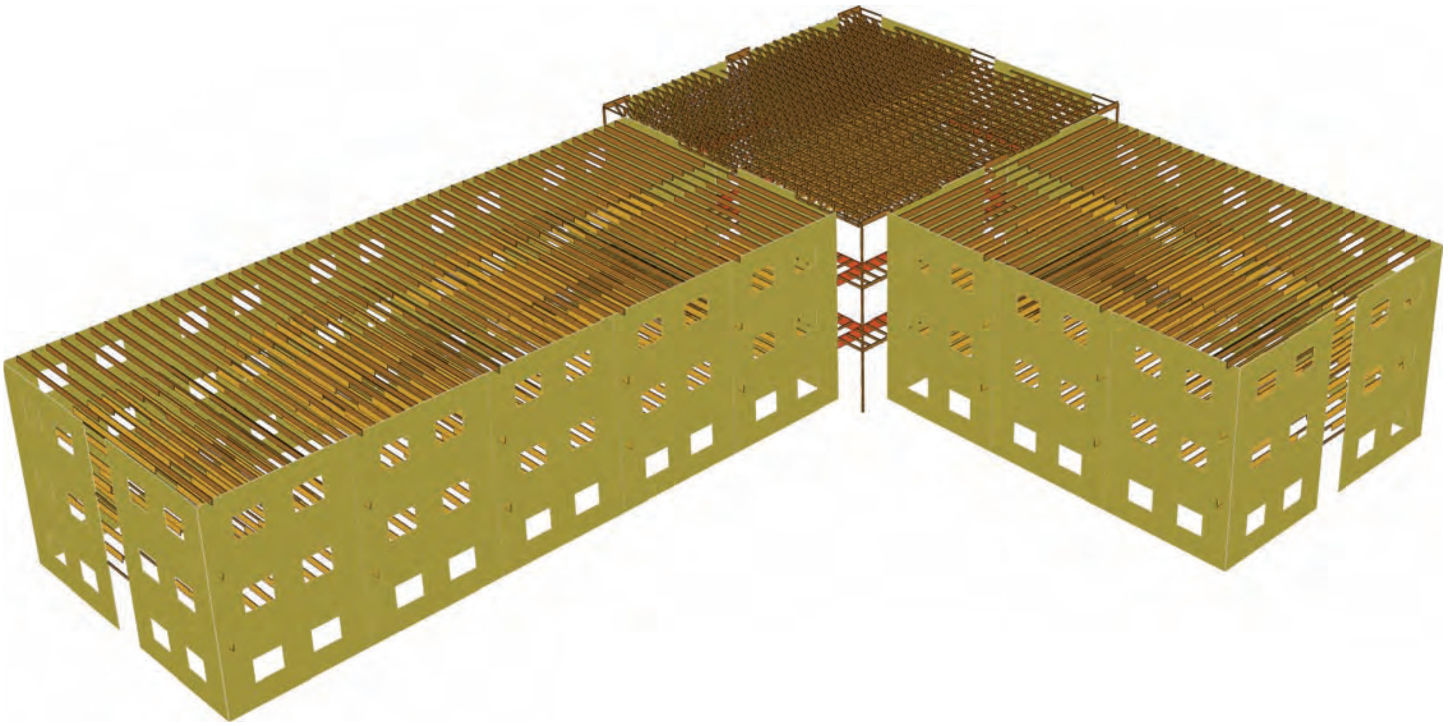
RISA Wood Design Training Manual



RISA TECHNOLOGIES

26632 Towne Centre Dr., Suite 210, Foothill Ranch, CA 92610 • 800.332.RISA

RISA Wood Design Training Manual

Rapid Interactive Structural Analysis – Building Systems

Training Manual for Wood Products Council

RISA TECHNOLOGIES

26632 Towne Centre Drive, Suite 210
Foothill Ranch, California 92610

(949) 951-5815
(949) 951-5848 (FAX)

www.risatech.com

Table of Contents

Introduction

RISA Technologies, LLC has been developing cutting-edge structural design and optimization software since 1987. With a well-trained team of engineers and software developers, we are working to meet the needs of our growing client base by implementing new design features and expanding the suite of software tools that we offer.

We have developed a comprehensive suite of products to meet your needs, whatever the application may be. Whether you need rapid results for everyday designs, or comprehensive designs to solve your most complex problems, RISA has your solution.

Providing exceptional customer service is a priority to us and is something we continue to strive for in our day-to-day operations. We provide fast, dependable technical support to our users and want them to know that we are proactively working for them. We have a wide variety of engineering design experience within the office, which gives us a great perspective for future development goals. This is also essential in providing quality technical support, as we have experienced many of the same engineering design challenges ourselves. Our focus is squarely on our clients

The RISA Building System

The RISA Building System is the integrated collection of finite element analysis programs for structural engineering offered by RISA Technologies. The main programs in this collection include RISAFloor, RISA-3D, and RISAFoundation.

This training session will focus on RISAFloor and RISA-3D and the interaction between the two programs.

RISAFloor is used in the layout, loading, and design of multi-story, multi-material structures for gravity loads and is the initial program for most building-type structures.

RISA-3D, which is integrated with RISAFloor, is used for the generation of building lateral loads, the modeling of lateral force resisting system and non-floor system elements, as well as the design and optimization of such elements for the combination of gravity and lateral loads. RISA-3D is a more general purpose program than RISAFloor and can be the initial program for more unique or non-building structures.

Training Course & Manual

This training course has been designed to provide class attendees with the necessary modeling skills and understanding required to fully utilize the RISAFloor and RISA-3D programs in tandem. The intent of this manual is to provide procedural instruction for the building of the training models and is NOT a ‘click by click’ tutorial. The course instructor is essential to the training course and will provide attendees with the necessary guidance and ‘know how’ to utilize the program. This includes navigation of the program features as well as tips and tricks for efficient and accurate use.

How to Use this Manual

Different nomenclature is used in this manual as we go through:

Normal text is used to explain what is happening in the course of modeling or analyzing a model.

- A bullet is an action item

    An indented line below a bullet is also an action item.

Note: A note gives more information about a specific topic or concept that may be glossed over in the training but is there for your information.

Course Instructors

Amber K. Freund, P.E.

Director of Marketing

Education
B.S. Architectural Engineering, California Polytechnic State University, San Luis Obispo
M.S. Civil Engineering, University of California, Irvine

Background
Amber joined RISA Technologies in 2005 and is responsible for technical support, training and sales. She also works on the testing and development of new programs and features.

Prior to joining RISA Technologies, Amber was a design engineer for Hope Engineering, a mid-size engineering firm in San Diego. Amber has designed a number of pharmaceutical buildings and manufacturing facilities. Her design experience also includes a residential and commercial high-rise building in downtown San Diego.

Deborah Brisbin, P.E.

Technical Marketing Engineer

Education
B.S. Civil Engineering, University of Colorado, Boulder, Colorado

Background
Deborah joined RISA Technologies in 2007 and has been involved in technical support, marketing, and sales, as well as testing and development of the new features.

Deborah joins RISA after working in New York for Wexler and Associates designing commercial concrete Hi-Rise buildings. She has also specialized in Cold Formed Steel and trusses working for a software company called Keymark Enterprises. In both Colorado and New York, she has worked in the telecom industry, designing and analyzing cellular towers.

Michael Olson, P.E.

Applications Engineer

Education

B.S. Civil Engineering, University of Wisconsin, Madison
M.C.E Civil Engineering (Structural Emphasis), University of Minnesota, Minneapolis

Background

Michael joined RISA Technologies in July 2008 and works as an engineer involved with technical support and training, as well as general testing and development of our software suite.

Prior to joining RISA Technologies, Michael's primary work experience was as a project engineer for Van-Sickle Allen and Associates, LLC (VAA, LLC) in Minneapolis, a company that works in the commercial, agricultural, and industrial sectors. Michael has designed office buildings for financial institutions, flour and feed mills for agricultural clients and worked on biodiesel structures for industrial clients. Many of these projects utilized RISA-3D.

Matthew G. Brown, P.E.

Applications Engineer

Education

B.S. Civil Engineering, Lawrence Technological University, Southfield, Michigan
B.S. Architecture, Lawrence Technological University
M.S. Civil Engineering (Structural), Lawrence Technological University

Background

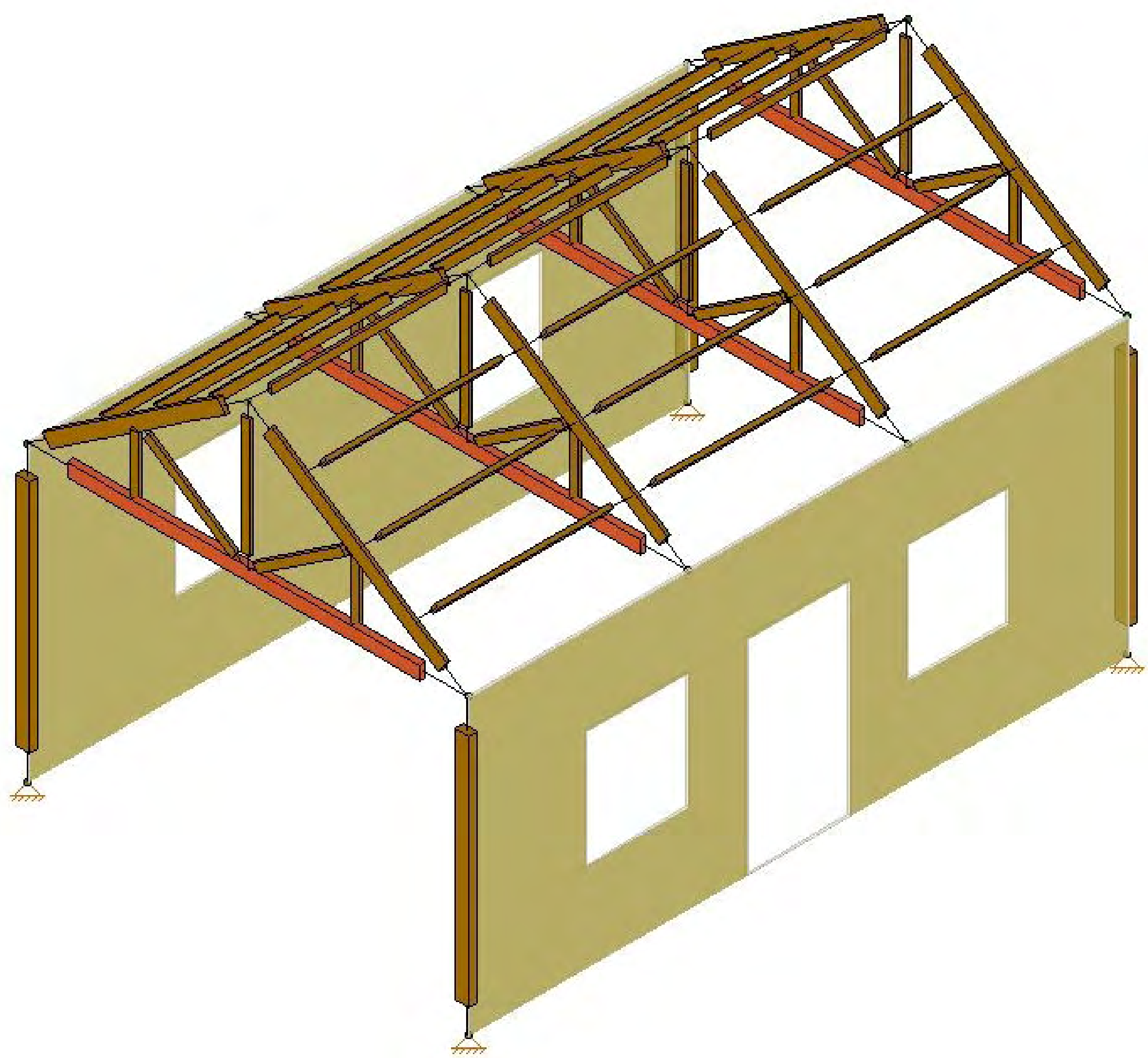
Matt joined RISA Technologies in 2008 and is responsible for technical support, training and testing of new products.

Prior to joining RISA Technologies, Matt was a project engineer for Harley Ellis Devereaux, a jumbo A/E firm based in Detroit. Matt has designed and modeled a number of structures ranging from steel framed hospitals to a concrete flat slab high-rise. Prior to working at HED, Matt worked for the U.S. Army Corps of Engineers.

Notes:

RISA-3D Basics

This model is intended to illustrate a basic model within RISA-3D. We will use this model to navigate through the RISA-3D interface, and become familiar with the different ways to view the model.



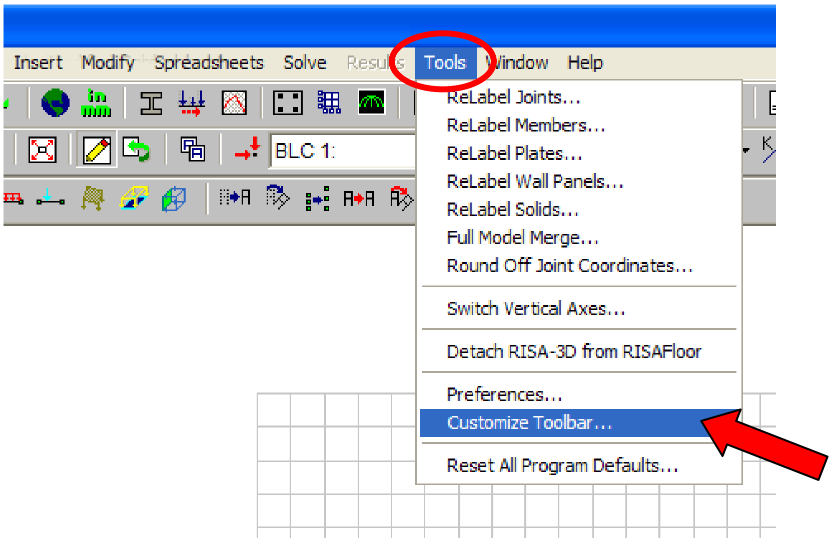
Project Definition: RISA3D Basics.r3d

Wood Trusses
Wood Columns: 4x6
Materials: Douglas Fir/Southern Pine, Glulam 24F-1.8E DF Balanced

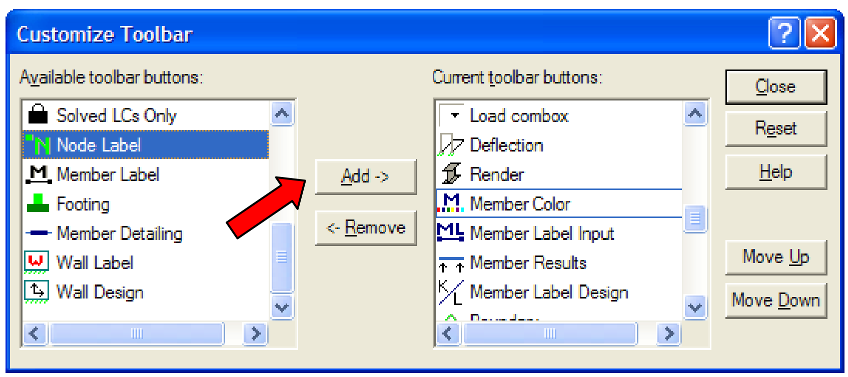
Let's customize your RISA toolbar.

By creating your personalized toolbar, you can quickly access your most frequently used buttons and optimize the power of RISA.

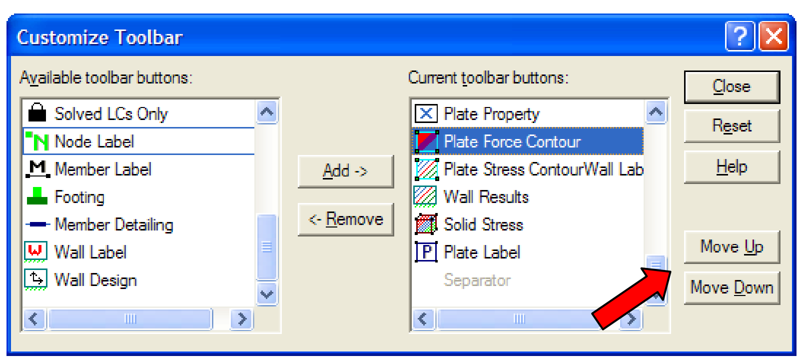
1. Go to **Tools** menu and select **Customize Toolbar**



2. Select one of the toolbars by clicking in the box **Available toolbar buttons**, and click on **Add** to place them on the current toolbar.

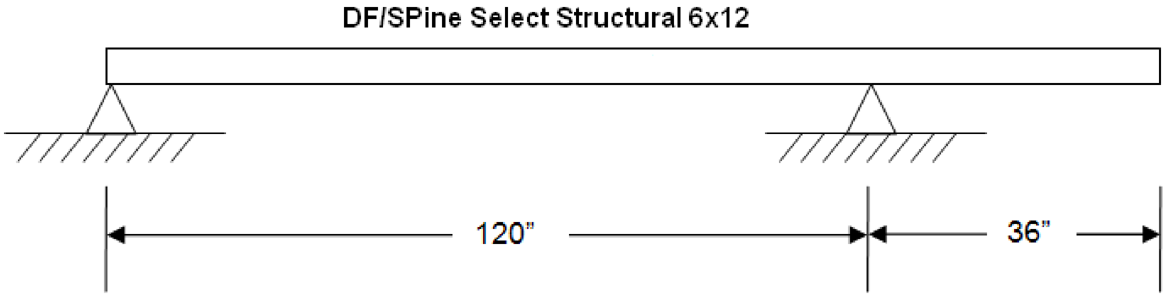


3. Once you've moved the buttons to the Current Toolbar, you can rearrange them by clicking on **Move Up** or **Move Down**.



Cantilever Beam Model

This simple cantilever model is intended to illustrate basic modeling within RISA-3D. It will also highlight some of the graphical results.

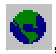


Project Definition: **Cantilever.r3d**

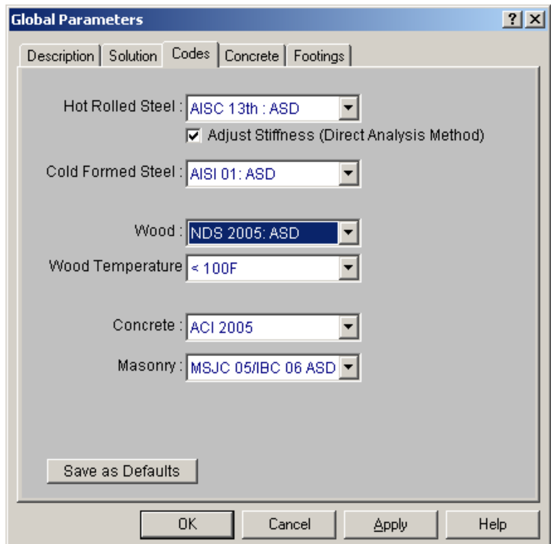
Wood Beam: 6x12
Material: DF/Spine
Species: Com Species Group I DF, SP,
Grade: Select Structural

Modeling Procedure

We'll start by setting up the design parameters for the wood wall panel.

- Click the Global icon ,

  On the "Codes" tab, select "NDS 2005: ASD" for the wood code.



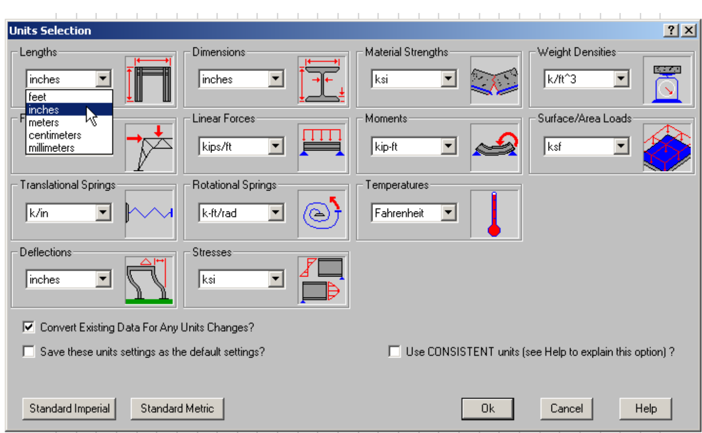
Here we can also add the project information.

Note: This information will also show up in the header and footer for your printed output.

Change length units to inches.

- Click on the Units button from the RISA Toolbar.

  Select "inches" from the Lengths pull-down menu:



Notice the other options available:

☑ Convert Existing Data For Any Units Changes?

This option allows you to convert the existing model to your Unit changes. For example, if you started off in units of feet for length and wanted to change to inches, keeping this checked will convert your model in feet to equivalent lengths in inches.

☐ Save these units settings as the default settings?

This option allows you to save your Units settings for the next time you start a new model.

☐ Use CONSISTENT units (see Help to explain this option) ?

This option creates a unit-less model and requires the user to keep consistent units. Steel and wood design results are not available when consistent units are used.

Now let's model this beam using the spreadsheets.

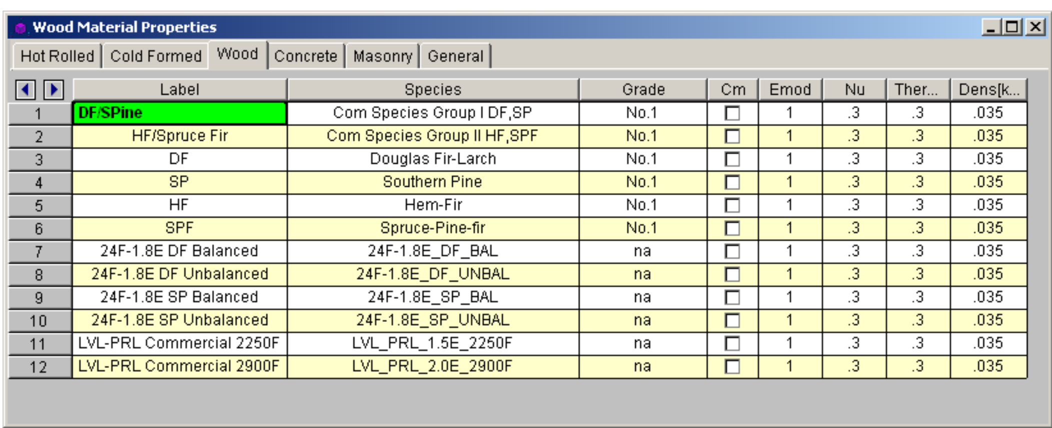
- Open the Material spreadsheet by clicking on the Material button [Materials] on the Data Entry Toolbar.

  Click the "Wood" tab.

  View the DF/SPine material.

  Change the Grade is set to "Select Structural".

Note: There are default materials for every type: Hot Rolled, Cold Formed, Wood, Concrete, and General Materials. You can add to these by pressing enter and choosing your own species/grade information.



	Label	Species	Grade	Cm	Emod	Nu	Ther...	Dens[k...
1	DF/SPine	Com Species Group I DF,SP	No.1	☐	1	.3	.3	.035
2	HF/Spruce Fir	Com Species Group II HF,SPF	No.1	☐	1	.3	.3	.035
3	DF	Douglas Fir-Larch	No.1	☐	1	.3	.3	.035
4	SP	Southern Pine	No.1	☐	1	.3	.3	.035
5	HF	Hem-Fir	No.1	☐	1	.3	.3	.035
6	SPF	Spruce-Pine-fir	No.1	☐	1	.3	.3	.035
7	24F-1.8E DF Balanced	24F-1.8E_DF_BAL	na	☐	1	.3	.3	.035
8	24F-1.8E DF Unbalanced	24F-1.8E_DF_UNBAL	na	☐	1	.3	.3	.035
9	24F-1.8E SP Balanced	24F-1.8E_SP_BAL	na	☐	1	.3	.3	.035
10	24F-1.8E SP Unbalanced	24F-1.8E_SP_UNBAL	na	☐	1	.3	.3	.035
11	LVL-PRL Commercial 2250F	LVL_PRL_1.5E_2250F	na	☐	1	.3	.3	.035
12	LVL-PRL Commercial 2900F	LVL_PRL_2.0E_2900F	na	☐	1	.3	.3	.035

- Open the Section Set spreadsheet [Section Sets],

  Select the "Wood" tab.

- Enter "Beam" as the label for the first row,

  Type "6x12" into the Shape column.

  Choose "DF/SPine" for the Material.

Wood Section Sets

Hot Rolled | Cold Formed | Wood | Concrete | General

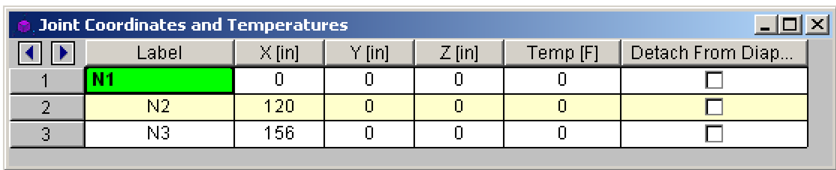
	Label	Shape	Type	Design List	Material	Design Ru...	A [in2]	Iyy [in4]	Izz [in4]	J [in4]
1	Beam	6x12	Beam	Rectangular	DF/SPine	Typical	63.25	159.443	697.068	446.446

- Click on the Joint Coordinates spreadsheet button on the Data Entry Toolbar. Enter the coordinates for the 3 nodes as:

  N1 – 0,0,0

  N2 – 120,0,0

  N3 – 156,0,0



Joint Coordinates and Temperatures

	Label	X [in]	Y [in]	Z [in]	Temp [F]	Detach From Diap...
1	N1	0	0	0	0	☐
2	N2	120	0	0	0	☐
3	N3	156	0	0	0	☐

- Click on the Members spreadsheet button on the Data Entry Toolbar.

Note: All RISA-3D spreadsheets default to a blank spreadsheet if empty. Simply click inside the spreadsheet to add the first line.

- Click the mouse into the spreadsheet and enter the following:

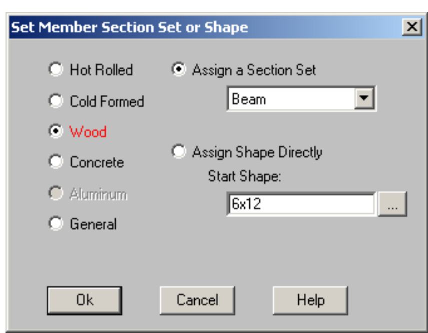
  Type “M1” as the label,

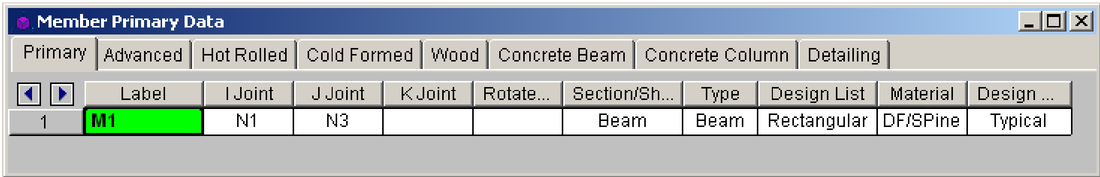
  Type “N1” as the **I** Joint and “N3” as the J Joint.

- Click the red arrow in the Section/Shape cell to open the Set Member Shape dialog.

  Select Wood as your material,

  Select “Beam” from the “Assign a Section Set” dropdown menu.





Accept the default entries for Design List, Type, and Design Rules as we will touch on those later.

Let's create pinned boundary conditions for the cantilever.

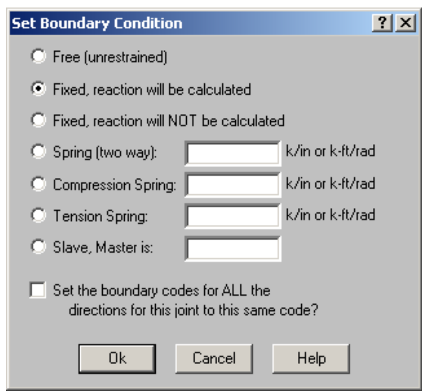
- Open the Boundary Conditions spreadsheet Boundary Conditions.

  Clicking in the spreadsheet will create the first node N1.

- Click in the X Translation column and click on the red arrow at the corner of the cell.

  Select "Fixed, reaction will be calculated,"

  Click "OK."



In the spreadsheet you'll see "Reaction" as the boundary condition; this indicates that the program will report back to us the reaction value at joint N1 in the X direction.

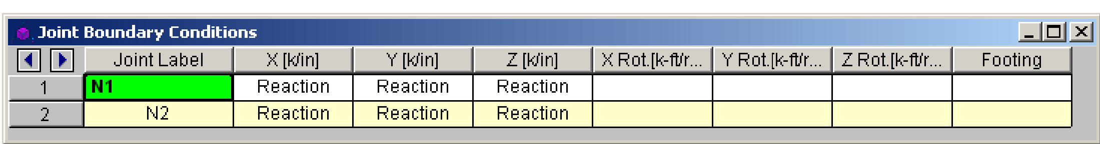
- Click in the Y Translation column and type "Reaction" or "R".

  Click in the Z Translation column and type "Reaction" or "R".

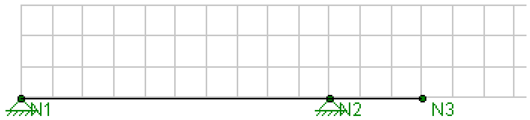
  Repeat the above steps for N2 by pressing enter on the spreadsheet to create another boundary condition.

Note: The difference between "Reaction" and "Fixed" is that they both act as fixed in their degree of freedom, but the "Reaction" option will report the value of the reaction, while "Fixed" will not give a value in the output. Therefore, "Reaction" should almost always be used. "Fixed" would only be used when you have a need to suppress output and are very sure of your boundary conditions.

The Joint Boundary Conditions spreadsheet should now look like this:



The model should now look like this:



Now let's recreate the same model by using the graphical drawing tools.

- Click on the Draw New Members button .

Note: If you don't see the Graphical Editing toolbar press the Icon or Ctrl-G.

- On the "Draw Members" tab,

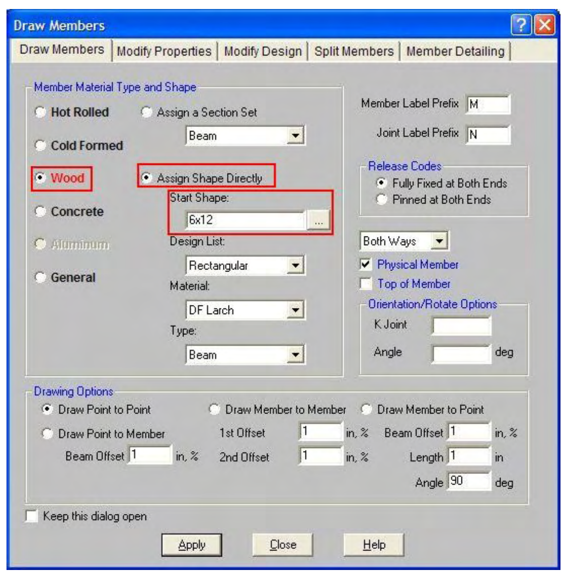
  Select the "Wood" radio button,

  Select the "Assign Shape Directly" button,

  Click the button under "Start Shape."

  Select "6x12" into the Shape Type.

  The dialog box should look like this:



- Click "Apply."
- Using the default drawing grid (if you do not see it, press the icon),

  Click on coordinate (0,36,0),

  Click on coordinate (156,36,0).

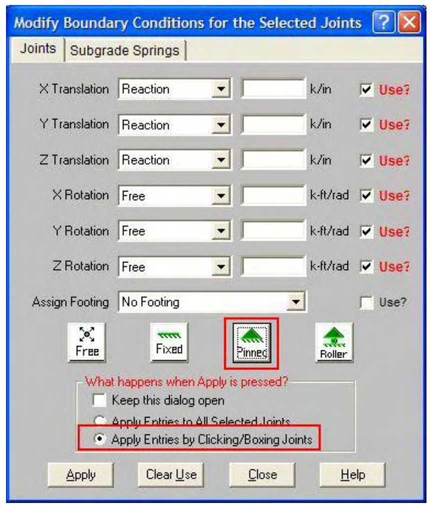
Note: The Status Bar displays the coordinates of your cursor in the lower right-hand corner of the screen.

Let's create pinned boundary conditions for the cantilever beam graphically.

- Click on the Boundary Condition icon

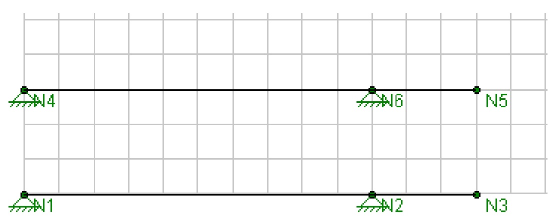
  Click on the "Pinned" button,

  Select "Apply Entries by Clicking/Boxing Joints."



- Click "Apply."
- Click on the drawing grid at (0,36,0) and again at (120,36,0).

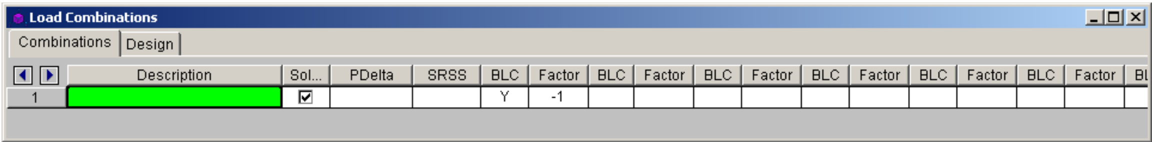
You should now see two identical beams drawn next to one another:



Let's load both cantilever beams with gravity loads.

- Open the Load Combination spreadsheet from the Data Entry Toolbar.
- Type a "Y" in the first BLC column and a "-1" in the first Factor column.

Note: By entering in "Y" as your BLC with a factor of "-1" in the Load Combinations spreadsheet, you are telling the program to include the structure's self weight in the solution. RISA-3D will then multiply the weight of our entire structure by a -1 factor (thus in the negative direction) in the Global Y direction, which is our vertical axis. This will create an additional downward load equal to our self weight. This load will not show up graphically.



Load Combinations

Combinations | Design

	Description	Sol...	PDelta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	Bl
1		☑			Y	-1											

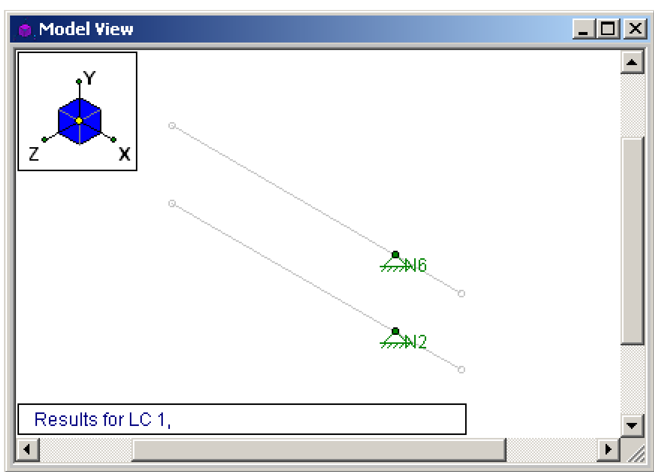
- Solve the model by clicking on the button at the top of the screen. Solve Current.

Note: The program will report instabilities in this model. Whenever there is instability in your model, RISA will typically lock that joint and continue with the analysis. This allows the program to still provide results. If this occurs, you will receive a warning which gives you the option to view a new model view which will highlight the instabilities.

Rules of Thumb for Instabilities

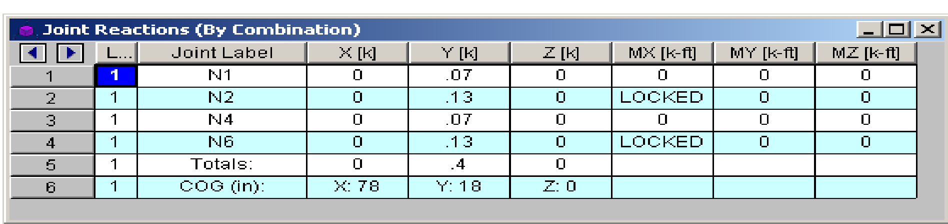
- Be aware of members spinning on own axis.
- Always have one member framing into a joint as fixed.
- With a 2D model make sure that you are braced out of plane.
- With tension only members, understand that members will be removed if they see any tension.

- Select "Yes" from the warning message to see the new model view and review the joints which are reporting instabilities:



- Click on the Joint Reactions spreadsheet on the Results Toolbar.

The program will show which joints were locked and for which direction/rotation in the Joint Reactions spreadsheet:



Joint Reactions (By Combination)

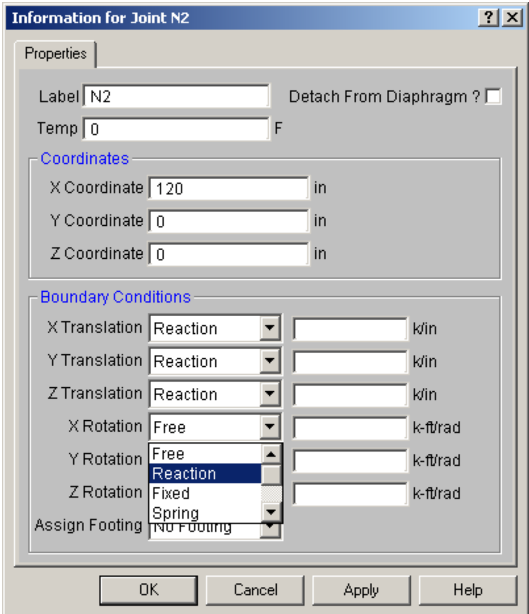
	L...	Joint Label	X [k]	Y [k]	Z [k]	MX [k-ft]	MY [k-ft]	MZ [k-ft]
1	1	N1	0	.07	0	0	0	0
2	1	N2	0	.13	0	LOCKED	0	0
3	1	N4	0	.07	0	0	0	0
4	1	N6	0	.13	0	LOCKED	0	0
5	1	Totals:	0	.4	0			
6	1	COG (in):	X: 78	Y: 18	Z: 0			

Note: In this model we are getting a torsional instability. You can think about it as if we were to apply a torsional force to this member. There is nothing in the model to keep this beam from spinning on its own axis. Because of this, RISA has locked a rotational degree of freedom at one of the joints in the beam. A simple way to fix this is to manually add a rotational boundary condition to your beam and see if any load comes out of that boundary condition.

The best way to test whether an instability is inconsequential or not is to apply a reaction to the joint in the unstable degree of freedom. Then re-run the model and examine the reactions. If the reaction that is restraining the instability is showing a non-zero force or moment, then you have a problem with the model that must be corrected for you to get valid results. If the reaction that is restraining the instability is showing a *ZERO* force or moment, then the instability is inconsequential to the results.

Let's test the instability by restraining X-rotation for the locked nodes.

- Double-click on the node N2 and change the X Rotation to "Reaction".

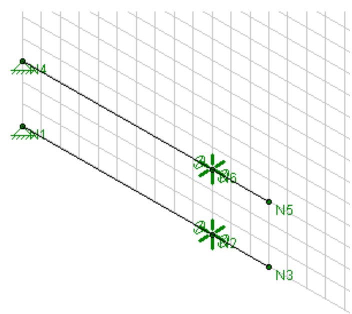


Do the same with N6.

- Open the Load Combination spreadsheet on the Data Entry Toolbar,

Click on Solve Current at the top of the screen to solve the model again.

The program should no longer report instabilities. Your model will now look like this:



If we open the Joint Reactions spreadsheet we will see that no torsional moment is being resisted, thus we are okay.

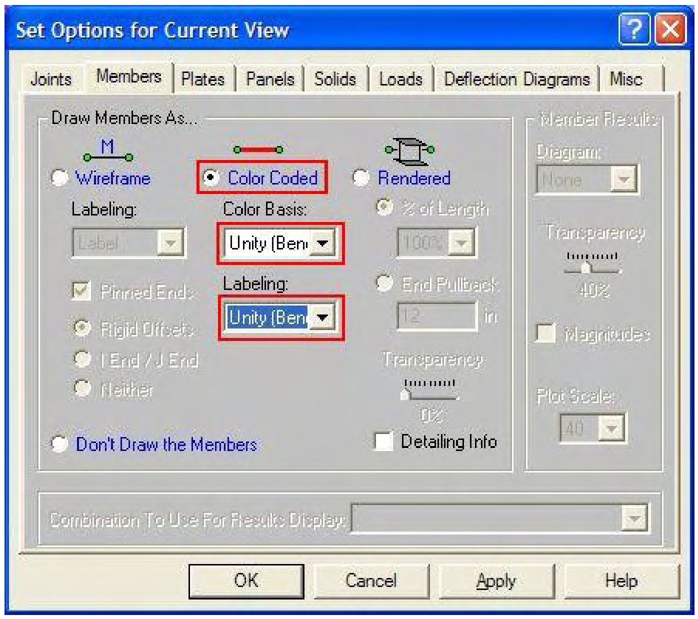
Let's take a quick look at the results.

- Click on the Plot Options button on the RISA Toolbar at the top left side of the screen.

  Click on the "Members" tab,

  Choose the "Color Coded" radio button,

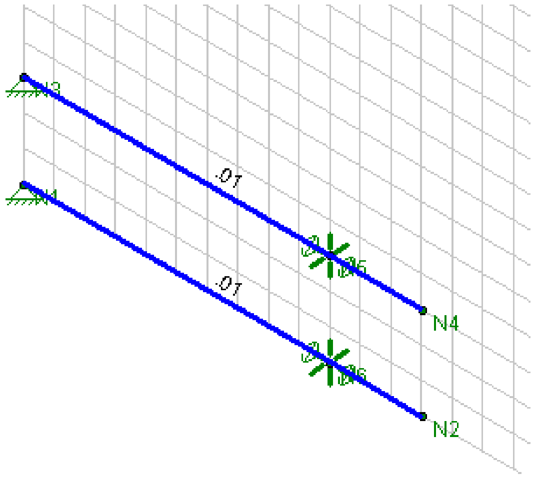
  Choose "Unity (Bending)" from the Color Basis dropdown and Labeling.



- Click "OK."

Now you can see your beams color coded by their bending check value. This value is also reported over each beam as a label.

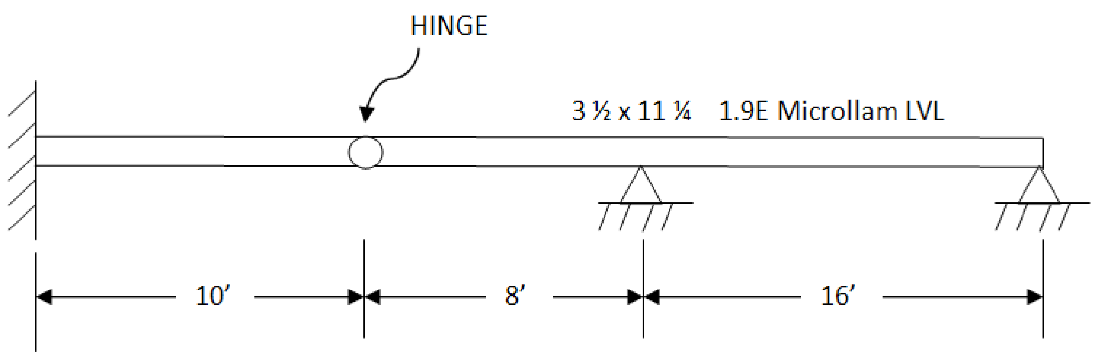
> **Note:** In the upper right hand corner, you can also see the legend to the color coding. This is a handy tool in a larger model as it allows you to simply look at the model and immediately see which members are failing (red) and which members can't be designed (black).



Notes:

Beam Hinge

This model will be used as a simple example to highlight the use of end releases, custom wood species, and more graphical results display options.



Project Definition: **BeamHinge.r3d**

Single beam: LVL_Microllam_1.9E_2600F
Size: 3 ½ x 11 ¼

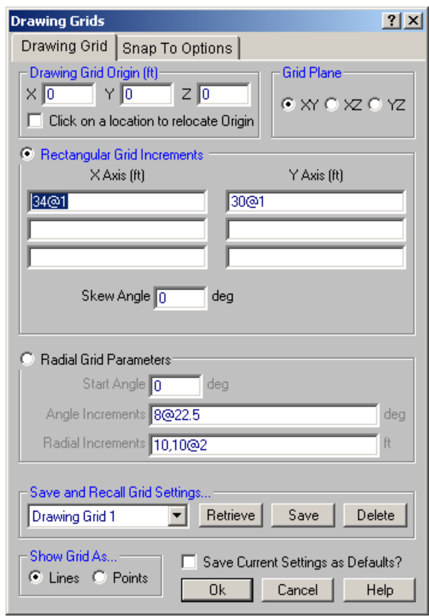
Modeling Procedure

First, modify the grid so that it is 34' wide. This will give us a snap point for the end of our member.

- Start a new model by clicking on the Start a New Model button .
- Click the Modify Drawing Grid button ,

  Change the X Axis Grid Increments to "34@1."

Note: If you don't see the Graphical Editing toolbar press the Icon or Ctrl-G.



Click OK.

Note: Here we are working with the Drawing Grid. This is used as a portable grid that you can move around your model to model specific portions of your structure much easier.

There is also the Project Grid, which is a global grid that works like a typical project grid. This may define column locations in your model, for example. If you modify your Project Grid, any items that are attached to this grid will move as well. This is great when an architect moves a grid 6". With one click you can modify your model to match.

Now let's take a look at our Materials.

Because this is an engineered wood material, the properties are not in the default wood databases, as they only have the dimension lumber values. Therefore, we must enter the Custom Wood Species spreadsheet to define our material.

- Click on "Modify" from the Windows Toolbar and select **"**Custom Wood Species Database."

Here we will view the design values for LVL_Microllam_1.9E_2600F.

Custom Wood Properties

	Label	Fb [ksi]	Ft [ksi]	Fv [ksi]	Fc [ksi]	E [ksi]	SCL
1	LVL_PRL_1.5E_2250F	2.25	1.5	.22	1.95	1500	☑
2	LVL_PRL_2.0E_2900F	2.9	1.9	.285	2.75	2000	☑
3	LVL_Microllam_1.9E_2600F	2.6	1.555	.285	2.51	1900	☑
4	PSL_Parallam_2.0E_2900F	2.9	2.025	.29	2.9	2000	☑
5	PSL_Parallam_1.8E	2.4	1.755	.18	2.5	1800	☑
6	LSL_TimberStrand_1.55E_2325F	2.325	1.07	.31	2.05	1550	☑
7	LSL_TimberStrand_1.3E_1700F	1.7	1.075	.4	1.4	1300	☑

The third item in our list is the material that we wish to use. For materials that are not explicitly defined in the wood database, you can add custom wood materials here.

Note: By checking the box for "SCL" RISA will use the structural composite lumber section of the NDS. Also, note that there is no entry for Fc_{perp}, as we do not consider crushing of sill plates, etc. in the program.

Once we have defined our material properties, we must go to the Materials spreadsheet to actually select this material.

- Open the Materials spreadsheet and go to the "Wood" tab.

  Add a line to the bottom of the spreadsheet by pressing "Enter" in the bottom line.

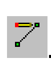
  Type "LVL" as your label and choose "LVL_Microllam_1.9E_2600F" for your species from the drop down list (you'll find this directly below the NDS species).

Wood Material Properties

Hot Rolled | Cold Formed | Wood | Concrete | Masonry | General

	Label	Species	Grade	Cm	Emod	Nu	Ther...	Dens[k...
1	DF/SPine	Com Species Group I DF,SP	No.1	☐	1	.3	.3	.035
2	HF/Spruce Fir	Com Species Group II HF,SPF	No.1	☐	1	.3	.3	.035
3	DF	Douglas Fir-Larch	No.1	☐	1	.3	.3	.035
4	SP	Southern Pine	No.1	☐	1	.3	.3	.035
5	HF	Hem-Fir	No.1	☐	1	.3	.3	.035
6	SPF	Spruce-Pine-fir	No.1	☐	1	.3	.3	.035
7	24F-1.8E DF Balanced	24F-1.8E_DF_BAL	na	☐	1	.3	.3	.035
8	24F-1.8E DF Unbalanced	24F-1.8E_DF_UNBAL	na	☐	1	.3	.3	.035
9	24F-1.8E SP Balanced	24F-1.8E_SP_BAL	na	☐	1	.3	.3	.035
10	24F-1.8E SP Unbalanced	24F-1.8E_SP_UNBAL	na	☐	1	.3	.3	.035
11	LVL-PRL Commercial 2250F	LVL_PRL_1.5E_2250F	na	☐	1	.3	.3	.035
12	LVL-PRL Commercial 2900F	LVL_PRL_2.0E_2900F	na	☐	1	.3	.3	.035
13	LVL	LVL_Microllam_1.9E_2600F	na	☐	1	.3	.3	.035

Use the graphical drawing tools to enter the model as two physical members.

- Select the Draw New Members icon .

  Select the "Wood" radio button,

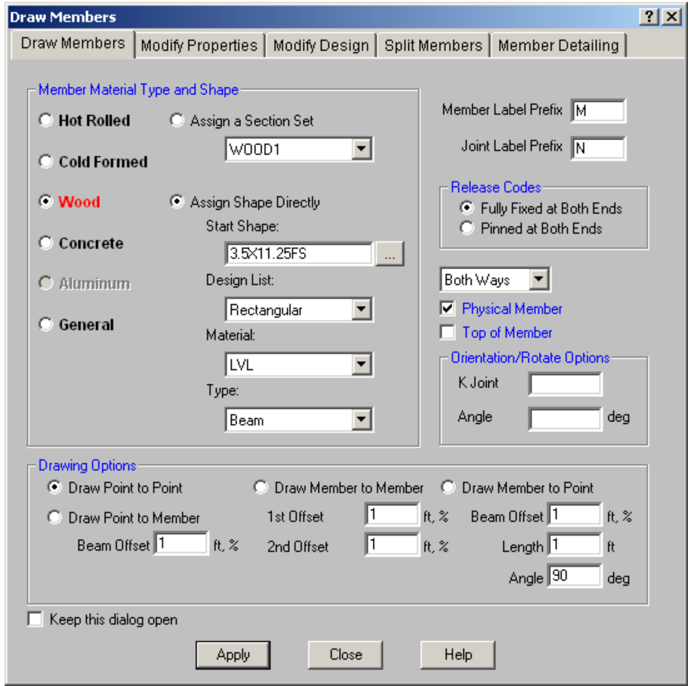
  Select "Assign Shape Directly,"

  Click the button under "Start Shape."

- Click the "Use Full Sawn Size" checkbox and input "3.5 x 11.25."

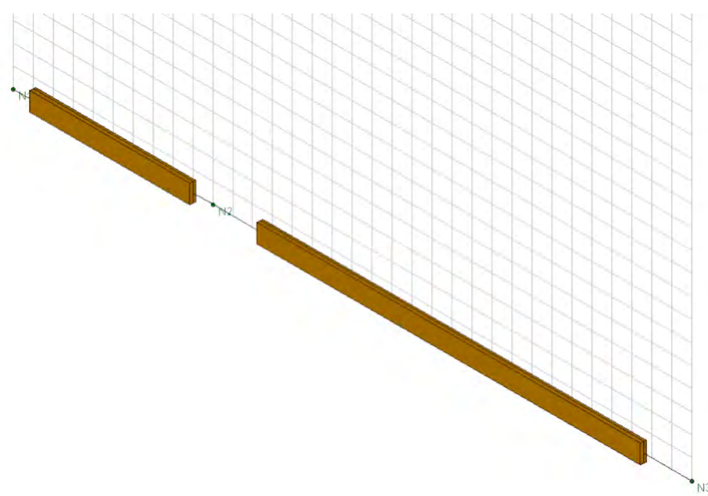
  Press "OK."

- For Material choose “LVL”, click “Apply,” and you’re ready to draw the members on the drawing grid.



- Draw two physical segments by clicking on (0,0,0), then (10,0,0), and (34,0,0).

Click the isometric view icon , then click the render icon twice and you should see this view:



Let’s create boundary conditions for the beams graphically.

- Click the Boundary Condition icon ,

Click on the Fixed button .

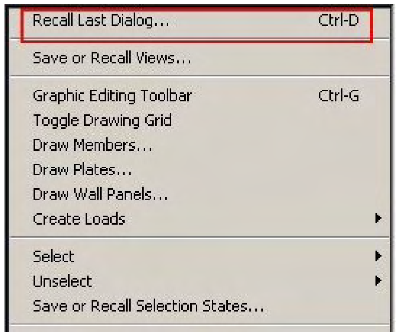
The degrees of freedom will automatically select "Reaction" for all translations and rotations.

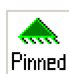
- Select "Apply Entries by clicking/Boxing Joints,"

  Click "Apply,"

  Select the first node N1.

- Add another boundary condition by right-clicking your mouse and selecting "Recall Last Dialog."



- Click the Pinned button 
.

  Select the grid locations (18,0,0) and (34,0,0).

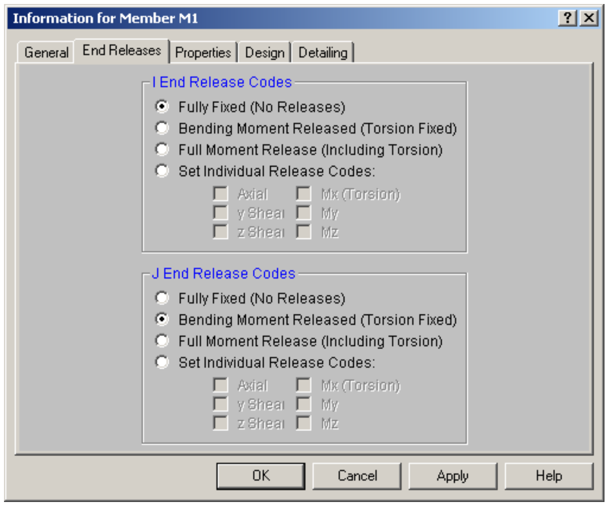
Once you've drawn the members and boundary conditions, we'll need to add the hinge at the joint.

- Click the icon to show the I and J ends of members.

> **Note:** The I end is the start of the member and is represented in red. The J end is the end of the member and is represented in blue.

We need to add a member end release to one of the two members at the hinge location, which will release the moment restraint across this joint.

- Change the member end-release by double clicking on the member between N1 and N2 and then go to the "End Releases" tab.



- Click the "Bending Moment Released (Torsion Fixed)" option on the J End Release Codes section.

Click "OK."

Click the Render toggle and you'll now see an open circle defining the end release in the graphic view.

Boundary Conditions vs. End Releases

Boundary conditions define the support conditions for things that are external to the model. A foundation for a steel frame would be an example. We are not modeling the foundation. We just want to use the reactions from 3D to design the foundation, so we would use boundary conditions to define the foundation.

End releases, on the other hand, are used to define how elements in a model are connected to one another (i.e. whether you have a fixed connection, pinned, or some other situation).

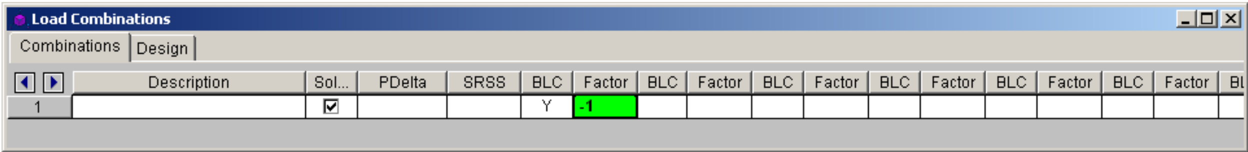
The I Release and J Release fields are used to designate whether the forces and moments at the ends of the member are considered fixed to or released from the member's points of attachment (the I and J joints). Each member has six force components at each end (axial, y-y & z-z shear, torque, and y-y & z-z bending). Any or all of these force components can be released from the member's point of attachment. If a force component is released, that force is not transferred between the joint and the member.

Note: At least one member or boundary needs to be fixed to each joint to prevent instability at the joint. If every member framing into a joint is pinned, then the joint has no ability to resist an applied joint moment. If you do this, RISA will find the instability, lock it in that direction, and report an instability message. You must then go back and check if that instability requires further investigation, similar to our previous model.

Let's load both of the beams with gravity loads.

- Open the Load Combination spreadsheet from the Data Entry Toolbar.

Type a "Y" in the first BLC column and a "-1" in the first Factor column.



- Solve the model by clicking on the button at the top of the screen. 
.

Now let's take a look at our results to verify our hinge is working properly.

- Click in the Graphic View.

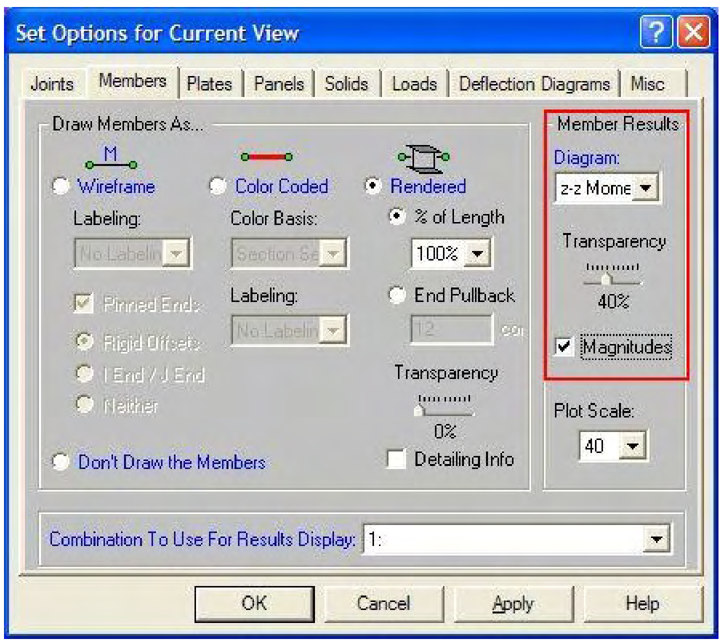
  Press F2 to go the Plot Options dialog.

Note: F2 is just one example of our hot keys within the program. For a full listing of these, go to our Help file. Under "Contents," look in Application Interface>>Shortcut and Hot Keys.

- Click the "Members" tab, in the "Member Results" section on the right side,

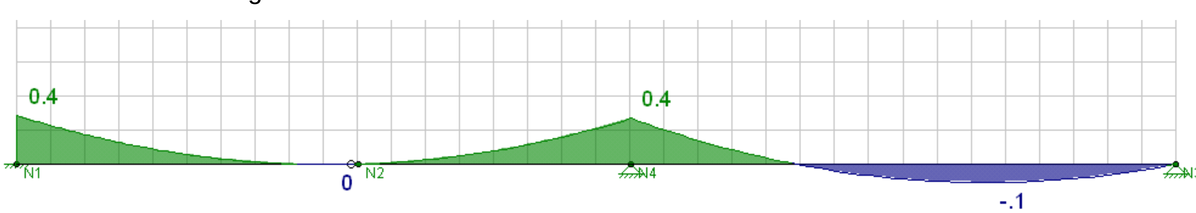
  Select "z-z Moment" from the Diagram dropdown list,

  Check the "Magnitudes" box.



- Press "OK."

  Click to get the elevation view below:

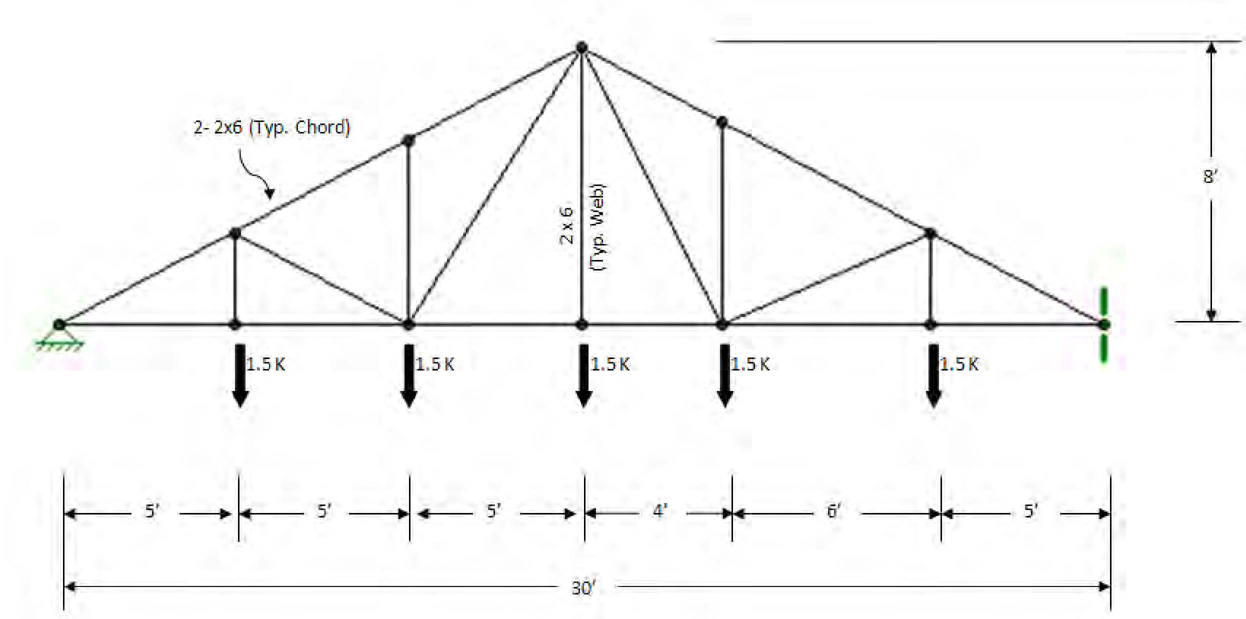


Now we can see the bending moment diagram drawn right over our members, along with their magnitudes, and we can confirm that there is no moment transfer across the hinge end release.

Notes:

2D-Truss

This is a model that will introduce us to multi-member modeling of a truss using a drawing grid. We will also talk about the design parameters, results and use our drawing tools to create a roof system.



Project Definition: 2D_Truss.r3d

Chord Members: 2-2x6
Web Members: 2x6
Material: DF Larch Grade No. 1

Modeling Procedure

Let's start by taking a look at our Materials spreadsheet.

- Open the Material spreadsheet by clicking on the Material button Materials on the Data Entry Toolbar.

  Click the "Wood" tab.

Here we can see that our first material already matches the material required for this model, so no modification is required.

Let's start by creating a drawing grid that will help us to make our modeling much easier.

- Click the Drawing Grid icon .

  Under "Rectangular Grid Increments," enter:

  X Axis: 3@5,4,6,5

  Y Axis: 8

  Click "OK."

You should see a grid like below:

Now we will start drawing members.

- Click the Draw New Members icon .

  Select "Wood" and "Assign Shape Directly,"

  Click the button next to "Start Shape",

  Select the "2x6" shape, and make sure to press the "Multiple" button with 2 plies.

  Click "OK."

- Make sure that the "Physical Member" checkbox is checked

  Change the Material to "DF".

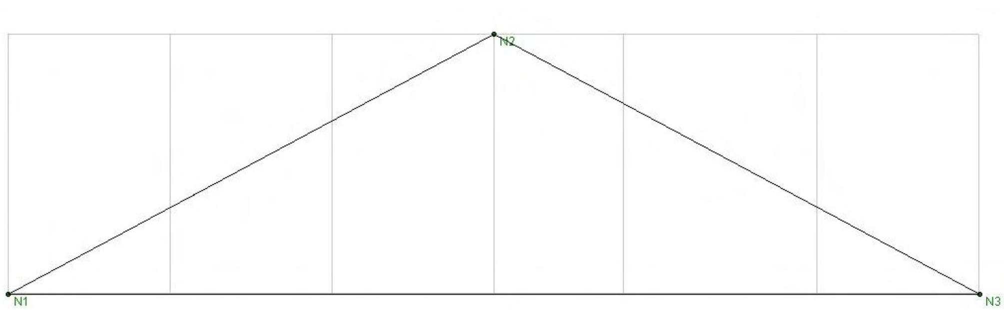
  Click "Apply."

- Draw the two diagonal top chords and the bottom chord by clicking on the grid intersections that define the ends of the members:

  N1: (0,0,0)

  N2: (15,8,0)

  N3: (30,0,0)

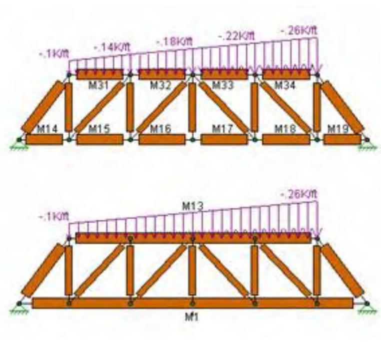


What are Physical Members?

Physical Members provide fixity to all joints that occur along the length of the member, without breaking that member into multiple smaller members. You may use the physical member feature to avoid defining one continuous member as multiple members in your model. This saves time in building and editing your model and in understanding your results.

To understand the benefits of Physical Members, let's look at the Help file topic on Physical Members and review the trusses in the figure. The first truss shows the chords modeled without physical members and thus with multiple members modeling each chord. The second truss models each chord with one Physical Member. Both models will yield the same results. However, the Physical Member model is more intuitive and easier to work with because you don't have to work with multiple members when you draw the chords, load them, edit them, or evaluate their results.

Note: Be careful with physical members is when considering unbraced lengths. We will touch on this shortly.



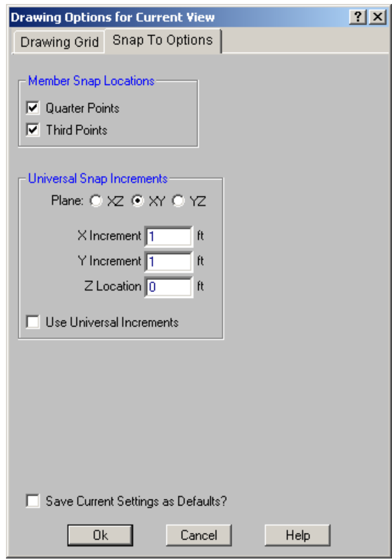
An example of why you might not use physical member is a braced frame, where the braces intersect but do not transfer loads.

- Click the Drawing Grid icon and select the “Snap To Options” tab,

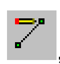
  Make sure to check “Quarter Points” and “Third Points,”

  Also make sure to turn off the “Use Universal Increments” option,

  Click “OK.”



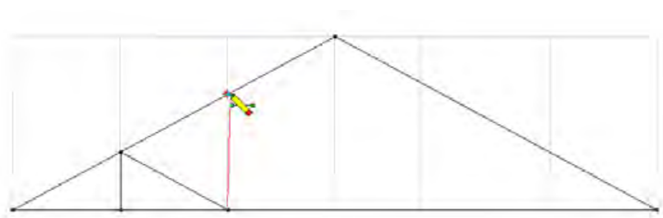
Now we can add our web members.

- Click the Draw New Members icon ,

  Enter the shape name directly into “Start Shape” by typing “2x6,”

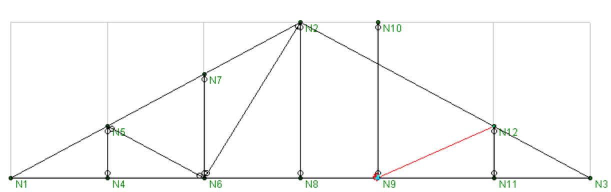
  Click “Apply.”

- Draw in the left side web members using the grid intersections and snap points:

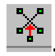


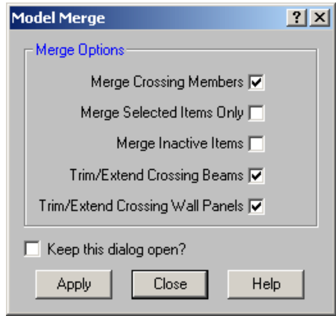
- Right-click once to disconnect your cursor from the previous joint.

- Draw in the remaining web members between the following points:
  There is one position where we do not have an intersection where it is required, thus we need to simply draw across our chord member to the grid above.



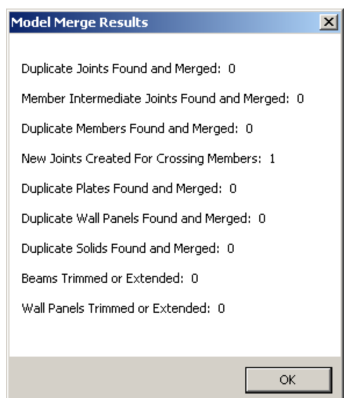
Now we can use the Model Merge tool to merge crossing members and create a joint at that intersection.

- Click the Model Merge icon . You will see this dialog:

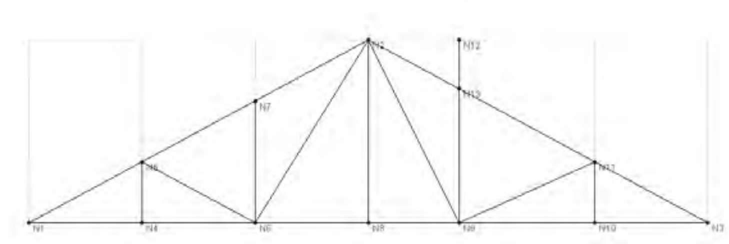


Note: The model merge feature is a helpful way for the program to "clean up" a model. Model merging will combine duplicate nodes and members, delete inactive items, and merge crossing members.

- Click "Apply" and review the results:

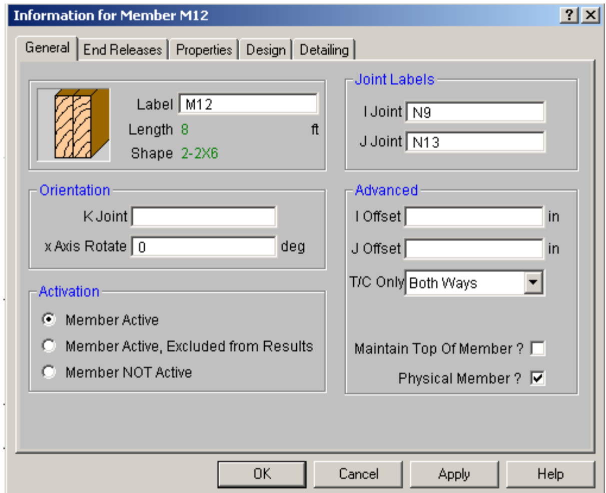


Here we see that we have one new joint created for crossing members. Looking at our model we see that we now have N13 created at our intersecting members.



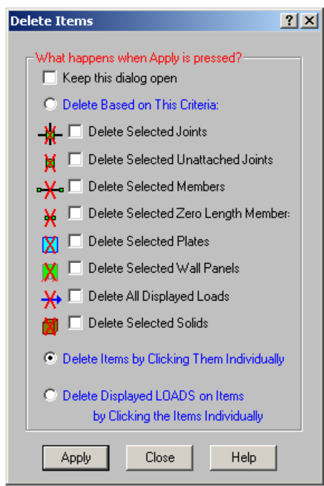
Now we can modify the joint coordinates that define our web member.

- Double-click on the web member and go to the "General" tab.
- Under the "Joint Labels" section, modify the J joint to be the new joint created.



Now we just need to delete the extra node.

- Click the Delete icon and choose to “Delete Items by Clicking Them Individually.”



- Click the extra node to delete it.

Now we can add the boundary conditions.

- Draw in the boundary conditions by clicking the Boundary Conditions icon .

  Assign the left side of the truss to be a “Pinned” boundary condition and the right side to be a “Roller”.

- Open the Boundary Conditions spreadsheet Boundary Conditions,

  Add an additional line by pressing enter at the bottom of the spreadsheet,

  Re-label the joint label to “ALL” and type “Reaction” in for the Z translation direction to keep it from falling over out of plane.

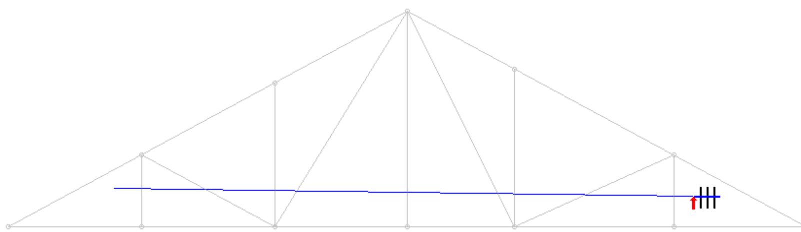
Note: The “ALL” command is used instead of applying independent boundary conditions in the Z direction. It is a good way to take care of instabilities related to 2D structures. ALL boundary conditions will not show up graphically.

Joint Boundary Conditions

	Joint Label	X [k/in]	Y [k/in]	Z [k/in]	X Rot.[k-ft/r...	Y Rot.[k-ft/r...	Z Rot.[k-ft/r...	Footing
1	N1	Reaction	Reaction	Reaction				
2	N3		Reaction					
3	ALL			Reaction				

Now let’s model the pinned end conditions of the web members.

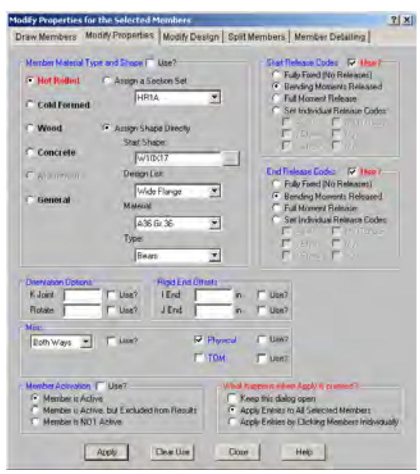
- Using the Selection tools on the left side of your screen, unselect the entire model by clicking on the Unselect Entire Model button.
- Now use the Line Select Tool to select just the web members:



- Click on “Modify” from the Windows Toolbar – then “Members.”
- Check the “Use” button on the “Start” and “End” Release Codes,

  Change the fixity to “Bending Moments Released” for both ends.

- At the bottom right, under “What happens when Apply is pressed?” make sure that the “Apply Entries to All Selected Members” option is selected.



- Click “Apply.”

  Click on the Select Entire Model button to re-select the whole truss.

Let's now give the chord members pinned end releases.

- Double-click the left top chord member and change member end releases to pins at both ends.

  Double-click the right top chord member and change the lower end to a pin release.

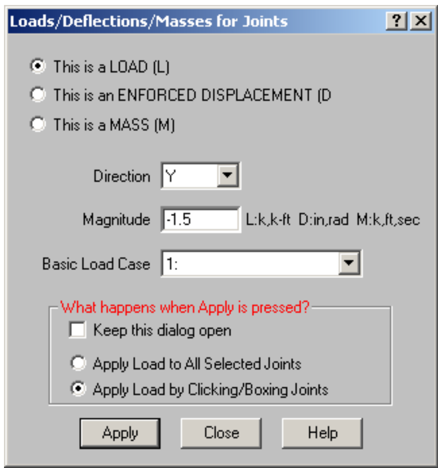
Now let's load our truss.

- Click on the Joint Load tool and load the truss as shown in the original graphic:

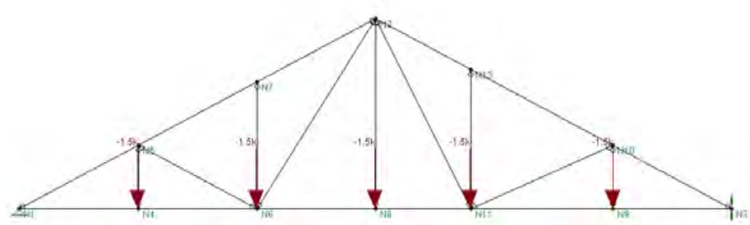
  Select "Y" Direction,

  Set Magnitude to "-1.5 k,"

  Click "Apply."



- Click on each of the five joints on the bottom chord of the truss to add the loads.



Note: The load magnitudes are negative because they are in reference to the global axes. If we had entered them as positive, they would point up in our graphic.

- Open the Load Combination spreadsheet ,

  Enter "1" for both the BLC and the Factor.

  Solve.

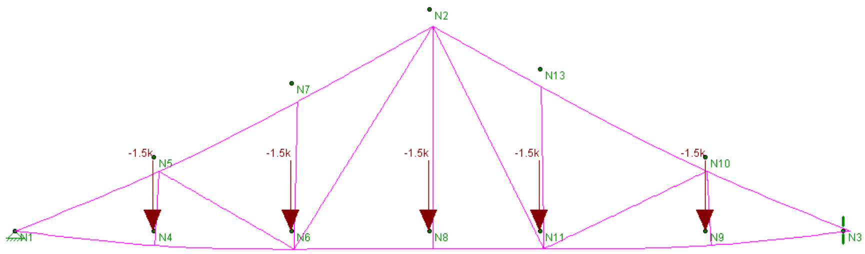
Now let's look at some results. First let's review the deflected shape to confirm that our truss is behaving as we would expect.

- Close the Load Combinations spreadsheet or click in the Graphical View.

  Press F2 and select the "Deflection Diagrams" tab,

Select “Load Combination (pick from list at bottom),”

Click “OK.”



> **Note:** Also in Plot Options- Deflection Diagrams is an option to animate your structure. This is useful for seeing how your structure got to its final deflected shape.

Now let’s review the code checks for our members.

- Click on the Design Results spreadsheet Design Results,

  Click on the “Wood” tab.

Member Wood Code Checks (By Combination)

Hot Rolled Steel | Cold Formed Steel | Wood | Concrete Beams | Concrete Columns

	L...	Member	Shape	UC Max	Loc[ft]	Shear ...	Loc[ft]	Dir	Fc'[ksi]	Ft'[ksi]	Fb1'[ksi]	Fb2'[ksi]	Fv'[ksi]	RB	CL	CP	Eqn
1	1	M1	2-2X6	- le/d is greater than 50. -													
2	1	M2	2-2X6	- le/d is greater than 50. -													
3	1	M3	2-2X6	.644	25	.052	25	y	.035	.878	1.3	1.495	.095	14.832	1	.021	3.9-1
4	1	M4	2X6	.194	0	.000	0	z	.903	.878	1.3	1.495	.095	8.844	1	.547	3.9-1
5	1	M5	2X6	.717	0	.000	0	z	.24	.878	1.3	1.495	.095	12.893	1	.145	3.6.3
6	1	M6	2X6	.005	0	.000	0	z	.27	.878	1.3	1.495	.095	12.508	1	.163	3.6.3
7	1	M7	2X6	.357	0	.000	0	z	.089	.878	1.246	1.495	.095	16.635	.959	.054	3.9-1
8	1	M8	2X6	.215	0	.000	0	z	.123	.878	1.3	1.495	.095	15.319	1	.074	3.9-1
9	1	M9	2X6	.195	0	.000	0	z	.903	.878	1.3	1.495	.095	8.844	1	.547	3.9-1
10	1	M10	2X6	- le/d is greater than 50. -													
11	1	M11	2X6	.330	0	.000	0	z	.098	.878	1.251	1.495	.095	16.198	.962	.06	3.9-1
12	1	M12	2X6	.006	0	.000	0	z	.224	.878	1.3	1.495	.095	13.118	1	.136	3.6.3

This spreadsheet reports a number of design values for each wood member in your model, along with a combined axial and bending check (UC Max) and a shear check (UC). Looking at the UC Max column we see that we have three members that are giving the “le/d is greater than 50” message, which is a reference Section 3.7.1.4 of NDS 2005 that limits column slenderness ratio.

This brings us to some design parameters that also need to be considered.

- Click on the Members spreadsheet and click on the “Wood” tab.

Wood Design Parameters

Primary | Advanced | Hot Rolled | Cold Formed | Wood | Concrete Beam | Concrete Column | Detailing

	Label	Shape	Length...	le2[ft]	le1[ft]	le-bend to...	le-bend bo...	Kyy	Kzz	CH	Cr	y sway	z sway
1	M1	2-2X6	17								☐	☐	☐
2	M2	2-2X6	17								☐	☐	☐
3	M3	2-2X6	30								☐	☐	☐
4	M4	2x6	2.667								☐	☐	☐
5	M5	2x6	5.667								☐	☐	☐
6	M6	2x6	5.333								☐	☐	☐
7	M7	2x6	9.434								☐	☐	☐
8	M8	2x6	8								☐	☐	☐
9	M9	2x6	5.867								☐	☐	☐
10	M10	2x6	2.667								☐	☐	☐
11	M11	2x6	6.566								☐	☐	☐
12	M12	2x6	8.944								☐	☐	☐

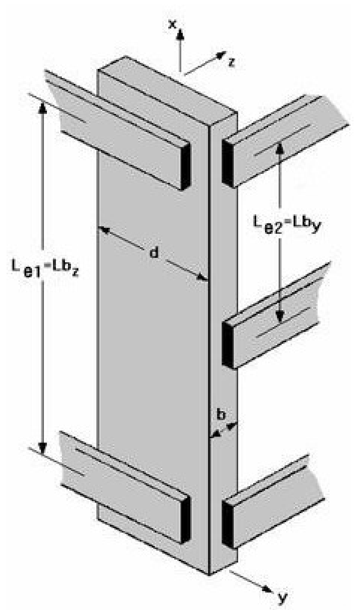
Here we are presented with some design parameters that have not yet been addressed. The first three columns are self explanatory, the next four are all unbraced length values, then we have the K-factors. These are effective length factors for axial compression.

Note: RISA will calculate K-factors for you if you click the icon. This is a simple approximation based on boundary conditions in your model. This calculation is affected by the sway flag columns.

You can also manually input any K-factors you wish and they will be used in the code checks. If nothing is input a default value of one will be used.

Let's revise the unbraced lengths for our problematic members.

Unbraced Lengths



The values **Le1** and **Le2** represent the unbraced length for the member with respect to column type buckling about the member's local z and y axes, respectively. These Le values are used to calculate Le1/d and Le2/b, which in turn impact the calculation of Cp, the column stability factor. These length to thickness ratios gauge the vulnerability of the member to buckling. Refer to Section 3.7 of the NDS for more information on this. This section also lists the limiting values of the length to thickness ratios.

The **Le-bend** values, **Le-bend-top** and **Le-bend-bot**, are the unbraced lengths of the member for bending. This unbraced length is the length of the face of the member that is in compression from any bending moments. This value should be obtained from Table 3.3.3 in the NDS code. The **Le-bend** value is used in the calculation of the slenderness ratio, RB, which is used in the calculation of CL, the beam stability factor. CL is then used to calculate the allowable bending stress. Refer to Section 3.3.3.6 in the NDS for more information on this and note that the value of RB is limited to 50.

For continuous beams the moment will reverse such that the top and bottom faces will be in compression for different portions of the beam span. **Le-bend-top** is the unbraced length of the top face and **Le-bend-bot** is the unbraced length of the bottom face.

- For the Top Chords, we will consider our rafters to be sheathed at the top,

  Therefore, set le2 and le-bend top to 2 ft.

We will also consider each of the web members to brace the chords; therefore enter "Segment" for le1.

- For the slender web member, we will assume that the member is braced at mid-span.

  Therefore, set le2 to 3.25 ft.

Wood Design Parameters

Primary | Advanced | Hot Rolled | Cold Formed | Wood | Concrete Beam | Concrete Column | Detailing

	Label	Shape	Length...	le2[ft]	le1[ft]	le-bend to...	le-bend bo...	Kyy	Kzz	CH	Cr	y sway	z sway
1	M1	2-2X6	17	2	Segment	2					☐	☐	☐
2	M2	2-2X6	17	2	Segment	2					☐	☐	☐
3	M3	2-2X6	30								☐	☐	☐
4	M4	2x6	2.667								☐	☐	☐
5	M5	2x6	5.667								☐	☐	☐
6	M6	2x6	5.333								☐	☐	☐
7	M7	2x6	9.434								☐	☐	☐
8	M8	2x6	8								☐	☐	☐
9	M9	2x6	5.867								☐	☐	☐
10	M10	2x6	2.667								☐	☐	☐
11	M11	2x6	6.566	3.25							☐	☐	☐
12	M12	2x6	8.944								☐	☐	☐

- Re-solve the model.

Reviewing the Design Results spreadsheet verifies that our changes have reconciled the code check problems from before:

Member Wood Code Checks (By Combination)

Hot Rolled Steel | Cold Formed Steel | Wood | Concrete Beams | Concrete Columns

	L...	Member	Shape	UC Max	Loc[ft]	Shear ...	Loc[ft]	Dir	Fc'[ksi]	Ft'[ksi]	Fb1'[ksi]	Fb2'[ksi]	Fv'[ksi]	RB	CL
1	1	M1	2-2X6	.333	0	.036	0	y	1.434	.878	1.3	1.495	.095	11.165	1
2	1	M2	2-2X6	.325	11.333	.035	11.333	y	1.434	.878	1.3	1.495	.095	3.83	1
3	1	M3	2-2X6	.644	5	.052	0	y	.035	.878	1.3	1.495	.095	14.832	1
4	1	M4	2x6	.194	0	.000	0	z	.903	.878	1.3	1.495	.095	8.844	1
5	1	M5	2x6	.717	0	.000	0	z	.24	.878	1.3	1.495	.095	12.893	1
6	1	M6	2x6	.005	0	.000	0	z	.27	.878	1.3	1.495	.095	12.508	1
7	1	M7	2x6	.357	0	.000	0	z	.089	.878	1.246	1.495	.095	16.635	.959
8	1	M8	2x6	.215	0	.000	0	z	.123	.878	1.3	1.495	.095	15.319	1
9	1	M9	2x6	.006	0	.000	0	z	.224	.878	1.3	1.495	.095	13.118	1
10	1	M10	2x6	.195	0	.000	0	z	.903	.878	1.3	1.495	.095	8.844	1
11	1	M11	2x6	.274	0	.000	0	z	.665	.878	1.3	1.495	.095	13.878	1
12	1	M12	2x6	.330	0	.000	0	z	.098	.878	1.251	1.495	.095	16.198	.962

Let's take a look at a detail report for the model.

Detail reports are element specific reports that give detailed information regarding the design of members and wall panels. There are two ways to view detail reports, by clicking a member graphically or by clicking a button through the Results browsers.

- While still in the "Design Results" spreadsheet, click the at the top of the screen to open the detail report for that member in the spreadsheet. This will show us our input, the forces in the member and design values.
- Click the button on the left side of the screen in a graphical view. Then click a member in the model.

Centerline to Centerline Framing

RISA-3D and RISAFloor follow a centerline to centerline framing convention. That is, when two joints are defined as the end of the member, the centroid of the section is set exactly on the straight line between the two joints. The implications here must be considered, but in most cases this assumption can be considered ok.

It is true that with most connections of members that there are some inherent eccentricities. Typically, the connections themselves help to reduce these eccentricities down to a point where a centerline model will actually be very accurate. RISA-3D does not evaluate connections. It is simply a member design program.

There is also the consideration of member length in this discussion. A beam framing into two columns will calculate the length of the beam as the center to center distance of the column. In reality, the length of the beam member is less than this value. Thus, there is a little bit of conservativeness built in.

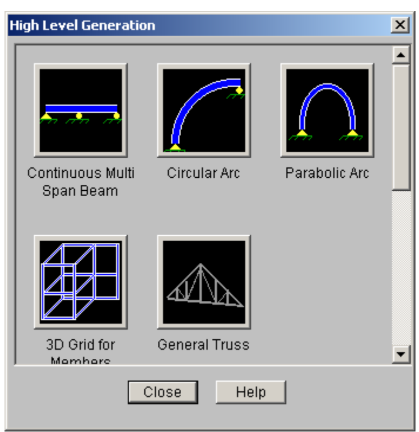
Thus, the one centerline assumption may be a little unconservative, but is balanced out by a conservative assumption.

In certain cases, if you want to consider the eccentricity of a connection, you can offset the member from the centerline. This will then induce a secondary moment or shear that you want to consider. Doing this may require the use of what RISA calls “Rigid Links”. A search of this in our help file will give a great explanation of what a rigid link is and some typical situations where they may be used.

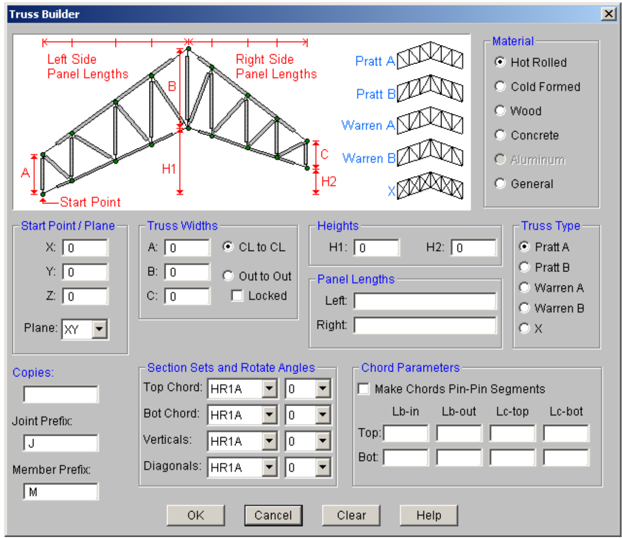
Now let’s explore the High Level Generation Tools by creating the same truss directly above the current one.

In RISA-3D we have tried to make modeling as quick and as easy as possible. In doing this we created some model generators that can either build the framework for your model, or build the model for you entirely.

- Select “Insert” >> “Structure (Generate)…” and this dialog will appear:



- Click on “General Truss” and a dialog will open.



- For Material, select "Wood."

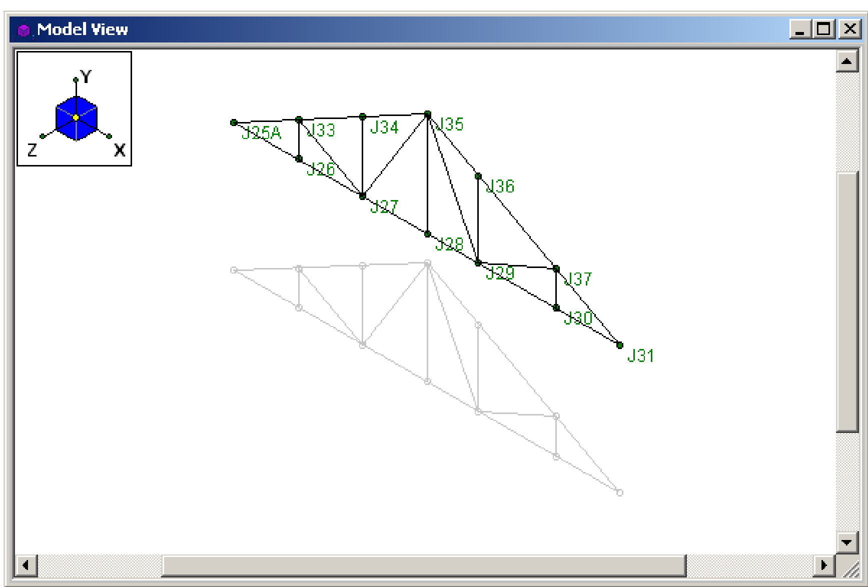
  For Start Point/Plane select (0,10,0),

  For Truss Widths change the B dimension to "8",

  For Truss Type choose "Warren B,"

  For Panel Lengths input "3@5" for the Left and "4,6,5" for the Right.
- Click "OK."

You should see a new window open up that looks like this:



Note: When you use the generator and click "OK," a new Model View is opened where the generated structure is selected and everything else is grayed out. This allows you to Move, Rotate, Copy, etc. the generated portion of your model and orient it appropriately to the existing model.

Let's quickly use our graphical editing options to create a roof system.

Note: The graphical editing tools will are only applicable to portions of your model that are selected. You can use the selection toolbar on the left side of the screen to select specific portions of your model that you wish to modify.

- Click the "Move via Linear Translation" button .

  Input "-10" in the Z direction.

  Press Apply.

- Click the "Move" button again.

  Let's move the same frame down 10' in the Y.

- Click the "Copy Linear" button .

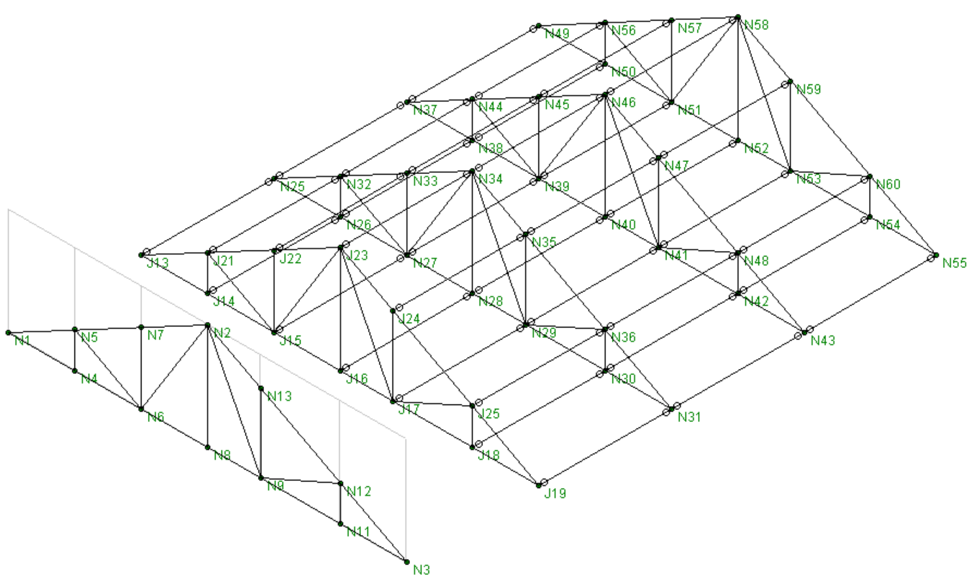
  Input "3@-10" in the top Z-inc. field.

  Check the "Inter-connect Bays".

  Choose "Pinned End Struts".

  Click Apply.

Our model should look like this:



Let's delete the front frame and rotate the entire model 90 degrees.

- Click the “Unselect Entire Model” button .

  Click the “XZ” view button .

  Click the “Box Select” button .

  Select the frame that is not connected to the rest of the model by boxing.

  Click the “Delete Items” button .

  Choose “Delete Based on this Criteria” and “Delete Selected Joints”.

  Click Apply.

The frame has been deleted.

Note: In RISA, any time that items are deleted from the model, associated items will also be deleted. For example, in this model when we deleted these joints, the members that had their end points defined by these joints were also deleted. The same thing happens with loading applied to members that are deleted.

- Click the “Toggle Drawing Grid” button , thus shutting off the drawing grid.

  Click the “Select Entire Model” button .

  Click the “Rotate Selected” button .

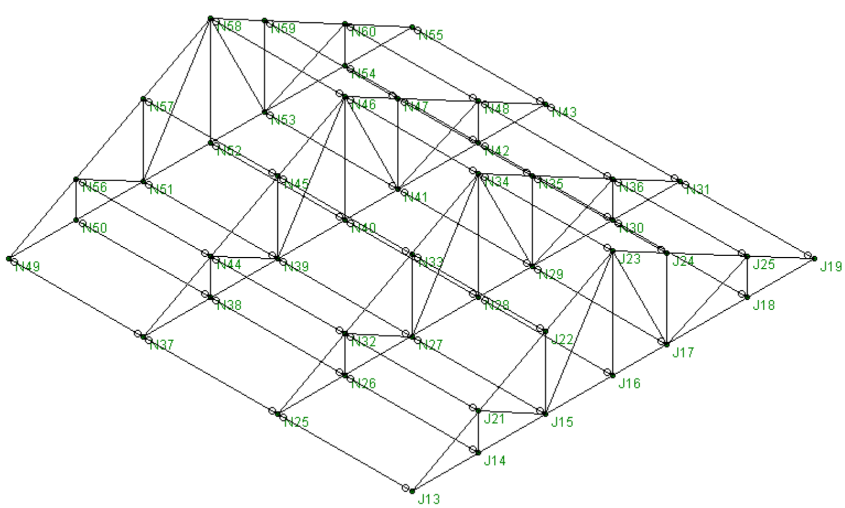
  Choose “Y” as the Rotation Axis.

  Type “90” as the Rotation Angle.

  Click Apply.

  Click the Iso button.

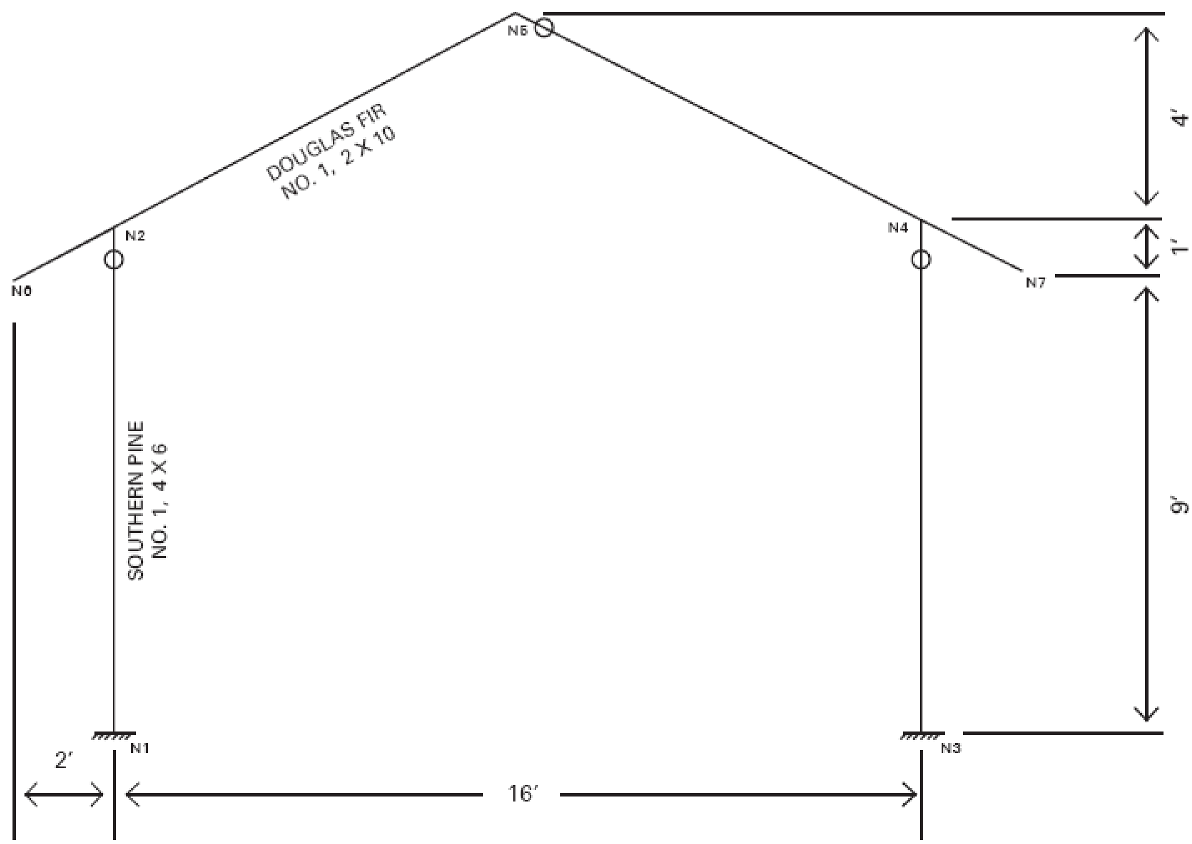
Our final model should look like this:



Notes:

Sloping Frame

This model represents a gabled roof where we don't want to transfer any moment across the peak. We are going to first model this introducing the idea of section sets, and then we will add some loads to it and look at load combinations.



Project Definition: **Sloping_Frame.r3d**

Modeling Procedure

Let's start by looking at our Materials.

- Open the Material spreadsheet by clicking on the Material button [Materials] on the Data Entry Toolbar.

  Click the "Wood" tab.

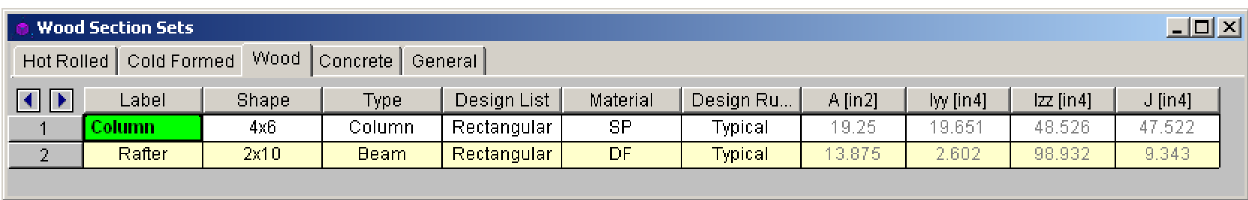
Here we see our material is already explicitly called out, so we can continue.

Let's look at Section Sets.

- Click [Section Sets].

> **Note:** Section sets are a way of grouping like members. This grouping allows you to change the properties for a large group of members easily. You would use section sets for members in your model that you know you want to end up being the exact same size. This grouping also provides good utility when looking at the optimization side of the program, which we will touch on shortly.

- Click the "Wood" tab. Enter the information as shown:

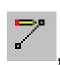


Wood Section Sets

Hot Rolled | Cold Formed | Wood | Concrete | General

	Label	Shape	Type	Design List	Material	Design Ru...	A [in2]	Iyy [in4]	Izz [in4]	J [in4]
1	Column	4x6	Column	Rectangular	SP	Typical	19.25	19.651	48.526	47.522
2	Rafter	2x10	Beam	Rectangular	DF	Typical	13.875	2.602	98.932	9.343

Now we can begin drawing using these section sets.

- Select the Draw Members icon ,

  Click the "Wood" radio button,

  Click "Assign a Section Set" and choose "Column."

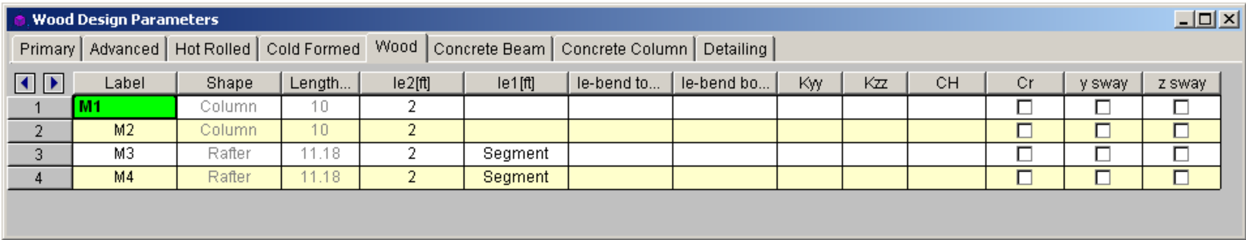
- Click "Apply."
- Draw in the columns, starting with the left-hand column at (2,0) and the right column at (18,0).
- Go back to Draw Members dialog, switch to the "Rafter" section set and draw in the rafters.
- Assign member end-releases by double-clicking on the members and going to the "End Releases" tab.
- Click the Boundary Conditions icon and set the bottom of the columns to be "Fixed."

  Set the "ALL" command in the Z direction in the spreadsheet.

Input Member Design Parameters.

We need to set our unbraced lengths in order to get accurate code checks. We will leave our K-factors as the default of 1.

- Match the spreadsheet shown below:

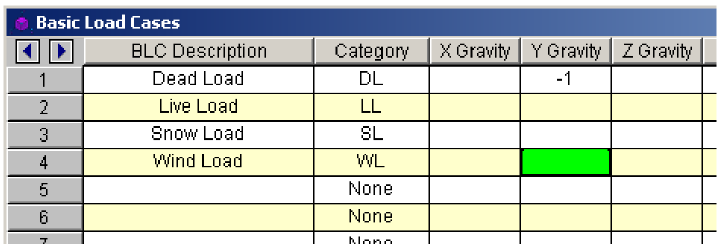

Wood Design Parameters

Primary | Advanced | Hot Rolled | Cold Formed | Wood | Concrete Beam | Concrete Column | Detailing

	Label	Shape	Length...	le2[ft]	le1[ft]	le-bend to...	le-bend bo...	Kyy	Kzz	CH	Cr	y sway	z sway
1	M1	Column	10	2							☐	☐	☐
2	M2	Column	10	2							☐	☐	☐
3	M3	Rafter	11.18	2	Segment						☐	☐	☐
4	M4	Rafter	11.18	2	Segment						☐	☐	☐

Now let's consider loading.

- Open the Basic Load Cases spreadsheet from the Data Toolbar.
- Type in the following to create our Basic Load Cases:


Basic Load Cases

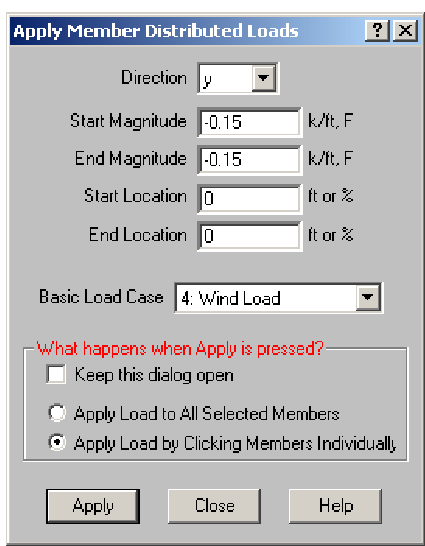
	BLC Description	Category	X Gravity	Y Gravity	Z Gravity
1	Dead Load	DL		-1	
2	Live Load	LL			
3	Snow Load	SL			
4	Wind Load	WL			
5		None			
6		None			
7		None			

- Use the pull down menu in the Category column to assign load categories to your Basic Load Cases.
- Enter "-1" in the "Y Gravity" column for row 1.

> **Note:** The "-1" in the "Y Gravity" cell tells the program to include the structure's self weight into this Basic Load Case. RISA-3D will then multiply the weight of our entire structure by a -1 factor (thus in the negative direction) in the Global Y direction, which is our vertical axis. This will create an additional downward load equal to our self weight.

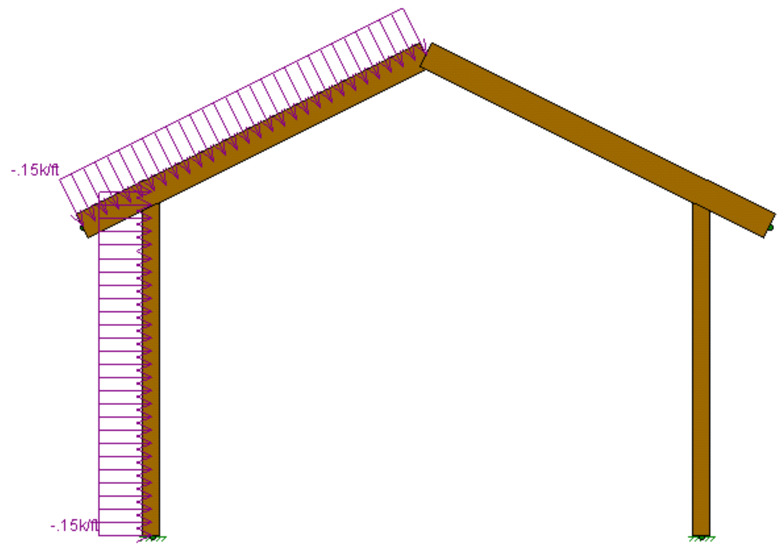
Now we will apply loads to our structure.

- Click on the Distributed Loads icon to add distributed wind loads as shown:



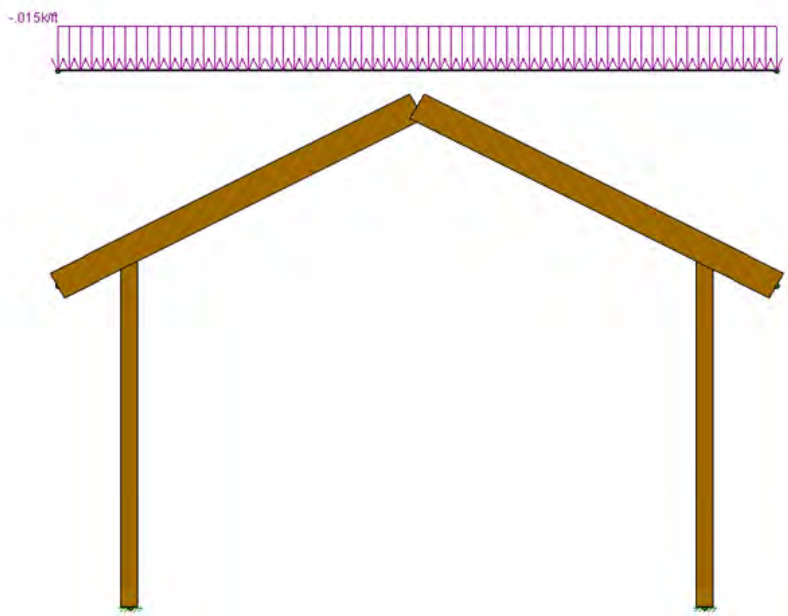
> **Note:** A lower case letter in the "Direction" field denotes the local axes of the member. The local "y" direction is what we want since this load is perpendicular to the rafter.

Wind Load:



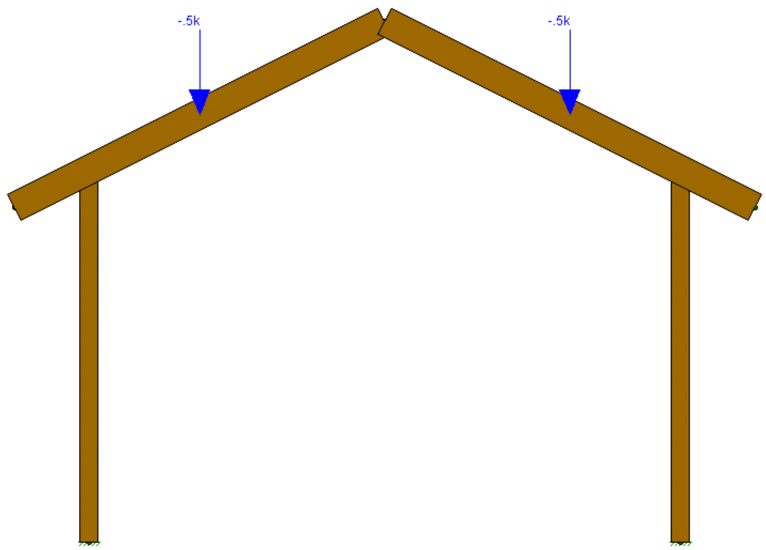
- Now add snow load to the rafters by right-clicking and selecting "Recall Last Dialog…"
- Fill in the following information:
    - Direction: PY
    - Start/End Magnitude: -0.015k/ft
    - BLC: Snow Load

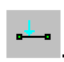
Snow Load:



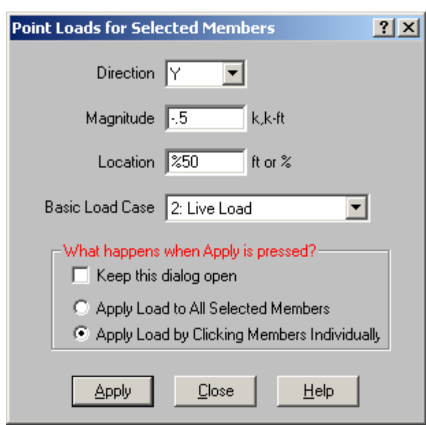
> **Note:** Because we assigned "PY" as the load direction, we are creating a projected load in the global Y direction. When a distributed load is designated as a projected load, RISA-3D will reduce it per the projected length. The reduced load will be displayed graphically on the model.

Live Load:



- Click on the Point Load icon .

  Enter the information as shown below:



- Click the two rafters to apply this load.

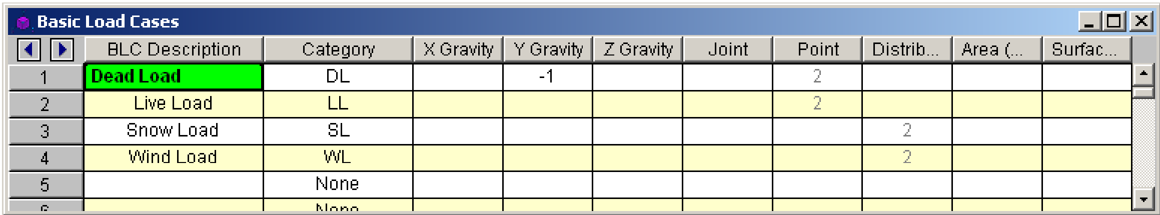
Dead Load: We will also add dead load point loads identical to the live loads.

- Open the Basic Load Cases spreadsheet .

  Click the row designation button for the "Live Load" to highlight the row in yellow.

Right-click and “Copy Basic Load Case”.

Copy the loads from “Live Load” into “Dead Load”.


Basic Load Cases

	BLC Description	Category	X Gravity	Y Gravity	Z Gravity	Joint	Point	Distrib...	Area (...	Surfac...
1	Dead Load	DL		-1			2			
2	Live Load	LL					2			
3	Snow Load	SL						2		
4	Wind Load	WL						2		
5		None								

Note: In the Basic Load Cases spreadsheet, the columns Joint, Point, Distributed, Area and Surface are shown to quickly tell you if there have been any loads defined as these for any of the BLCs in your model. You are able to click on these values and that load spreadsheet will open and allow you to view those loads.

Now that we have applied our loading, we need to create our load combinations.

- Open the Load Combinations spreadsheet .

- Click inside the window to create a line in the spreadsheet.

Under “Description” type “DL + LL”,

Under “BLC” type “DL” with a Factor of 1.

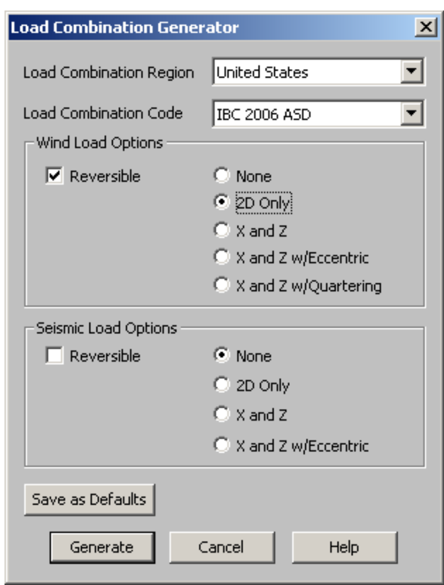
Under the second “BLC” entry, type “LL” with a factor of 1.

Note: You can define your load combinations based either on Load Category (DL, LL, WL, etc.) or based on the line in the Basic Load Cases spreadsheet (1,2,3, etc.).

We can also create load combinations using the automated Load Combination Generator.

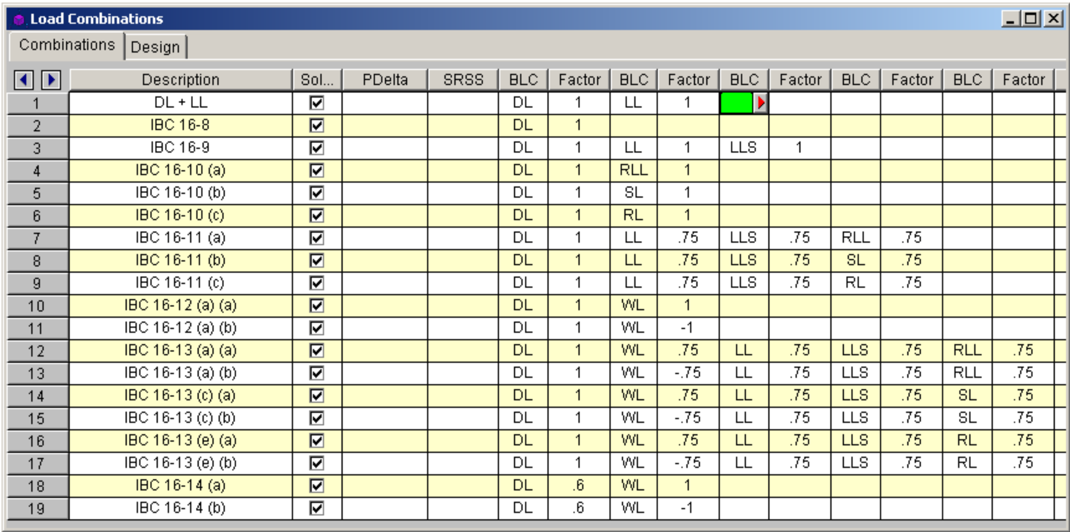
- Click the LC Generator button .

- Input the information as shown below:



- Click “Generate.”

You should see something similar to below:

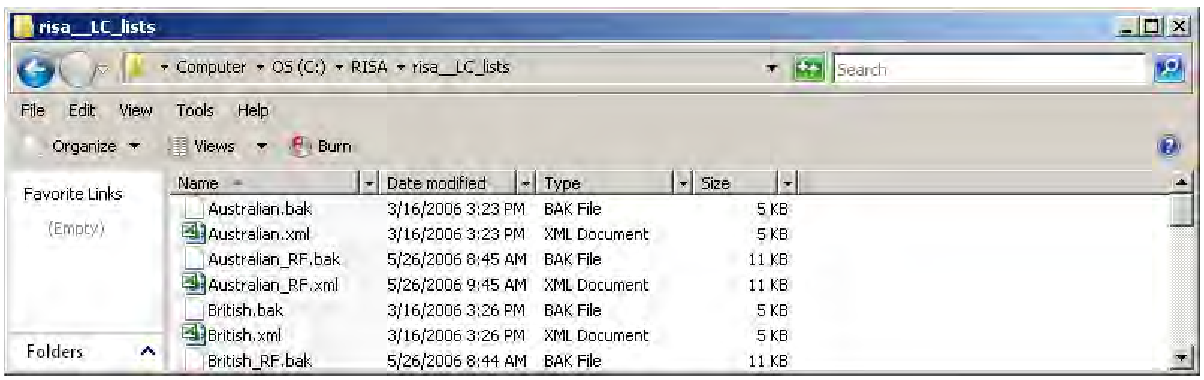


Load Combinations

Combinations | Design

	Description	Sol...	PDelta	SRSS	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor	BLC	Factor
1	DL + LL	☑			DL	1	LL	1						
2	IBC 16-8	☑			DL	1								
3	IBC 16-9	☑			DL	1	LL	1	LLS	1				
4	IBC 16-10 (a)	☑			DL	1	RLL	1						
5	IBC 16-10 (b)	☑			DL	1	SL	1						
6	IBC 16-10 (c)	☑			DL	1	RL	1						
7	IBC 16-11 (a)	☑			DL	1	LL	.75	LLS	.75	RLL	.75		
8	IBC 16-11 (b)	☑			DL	1	LL	.75	LLS	.75	SL	.75		
9	IBC 16-11 (c)	☑			DL	1	LL	.75	LLS	.75	RL	.75		
10	IBC 16-12 (a) (a)	☑			DL	1	WL	1						
11	IBC 16-12 (a) (b)	☑			DL	1	WL	-1						
12	IBC 16-13 (a) (a)	☑			DL	1	WL	.75	LL	.75	LLS	.75	RLL	.75
13	IBC 16-13 (a) (b)	☑			DL	1	WL	-.75	LL	.75	LLS	.75	RLL	.75
14	IBC 16-13 (c) (a)	☑			DL	1	WL	.75	LL	.75	LLS	.75	SL	.75
15	IBC 16-13 (c) (b)	☑			DL	1	WL	-.75	LL	.75	LLS	.75	SL	.75
16	IBC 16-13 (e) (a)	☑			DL	1	WL	.75	LL	.75	LLS	.75	RL	.75
17	IBC 16-13 (e) (b)	☑			DL	1	WL	-.75	LL	.75	LLS	.75	RL	.75
18	IBC 16-14 (a)	☑			DL	.6	WL	1						
19	IBC 16-14 (b)	☑			DL	.6	WL	-1						

The LC Generator creates these load combinations from a spreadsheet that is external to the program. These external spreadsheets are located in the RISA directory. Let’s take a look there.



If we follow the path from the image above you will see the different files for the different codes.

We can open up the United States load combination list and double-click the “United States.xml” file.

Here we can see the different tabs across the bottom of the screen for each code in the United States section of the LC Generator. These spreadsheets are fully customizable and you can even create your own tab and add any customized load combinations you wish.

Note: In these spreadsheets you must use the Load Category (i.e. DL, LL, etc.) as your call-out. Specifying a number for the BLC in reference to the line in the spreadsheet will cause an error.

Also, remember that Basic Load Combinations and Load Combinations must be synchronized for the loads to actually be considered. Load described in the WLX basic load case must be called out exactly as WLX in the load combinations spreadsheet to be considered.

Other things to consider in the Load Combinations spreadsheet:

Solve: This checkbox allows you to choose which load combinations you wish to include when running a Batch or Envelope solution.

P-Delta: This describes when P-Delta effects will be taken into account. When a model is loaded, it deflects. The deflections in the members of the model may induce secondary moments due to the fact that the ends of the member may no longer be co-linear in the deflected position. These secondary effects, for members (not plates), can be accurately approximated through the use of P-Delta analysis. You can read more about this in the RISA-3D Help File by searching “P-Delta.”

SRSS: This lists options for combining Response Spectra results for a dynamic analysis.

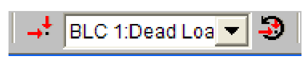
Note: There is also another tab to the load combinations spreadsheet, the “Design” tab. Here you can choose what materials should be included for code checks each load combination. If you are modeling multiple materials in the same model you may only want to check some materials for some LCs.

ASIF: This is the Allowable Stress Increase Factor applied to increase the allowable stresses for code checks.

CD: This is the Load Duration factor for NDS timber design.

ABIF: This is the Allowable Bearing Increase Factor, only used in tandem with the RISAFoot program.

Now that we have entered our load combinations, we can view the loads on our model.

Notice the Load Display dropdown list on the Window Toolbar: 

The button toggles the graphical load display on and off.

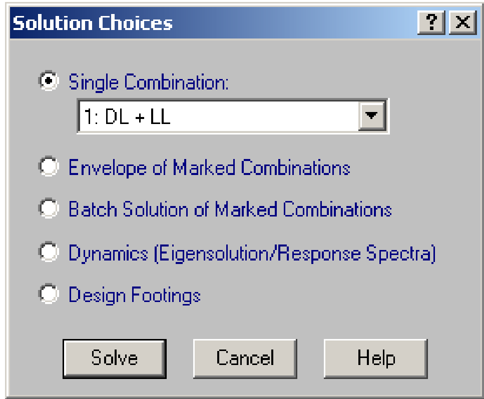
The button toggles between Load Category, Basic Load Case and Load Combination.

The dropdown list allows you to select between load categories, BLCs, and LCs.

Let’s solve the model.

- Choose Solve from the main toolbar.

Here you see that we are given four (or five) options:



- Click on the "Envelope of Marked Combinations" button,

  Click "Solve."

  Review the results.

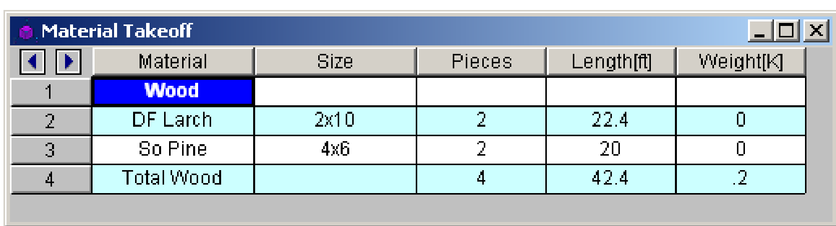
- Now, go back to the Solution Choices dialog box;

  This time select "Batch Solution" and "Solve."

  Review the difference in output.

Let's also take a look at the Material Takeoff.

- Open the Material Takeoff spreadsheet Material TakeOff.



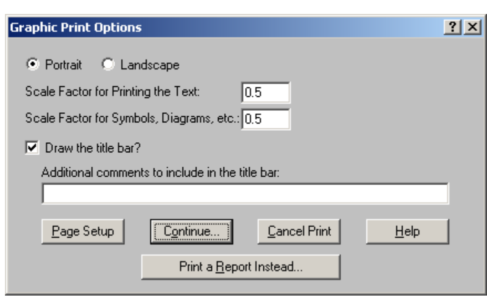
	Material	Size	Pieces	Length[ft]	Weight[K]
1	Wood				
2	DF Larch	2x10	2	22.4	0
3	So Pine	4x6	2	20	0
4	Total Wood		4	42.4	.2

Here we can see the total number of pieces of lumber required, the total length and the total weight of material required.

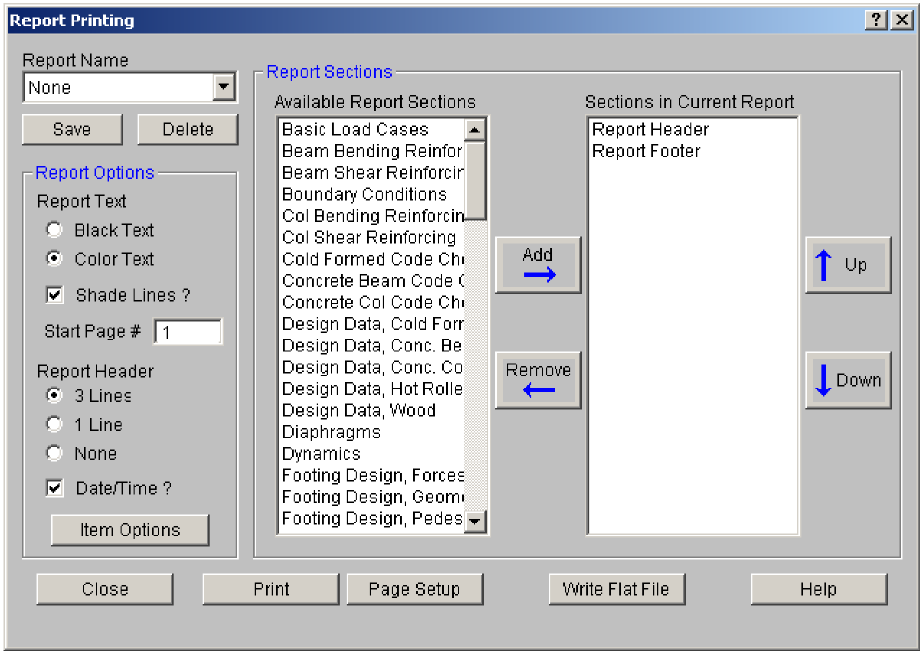
Printing Graphics and Spreadsheets

There are multiple ways to print output in the program.

- File>>Print will print up the print dialog:



- Pressing “Continue” will just print the currently displayed graphical view.
- Pressing “Print a Report Instead” will bring you into a “Report Printing Options” dialog



The “Available Report Sections” contain all of the input and output spreadsheets in the program. Here you can create a custom report by adding the specific spreadsheets you want to see in your printout.

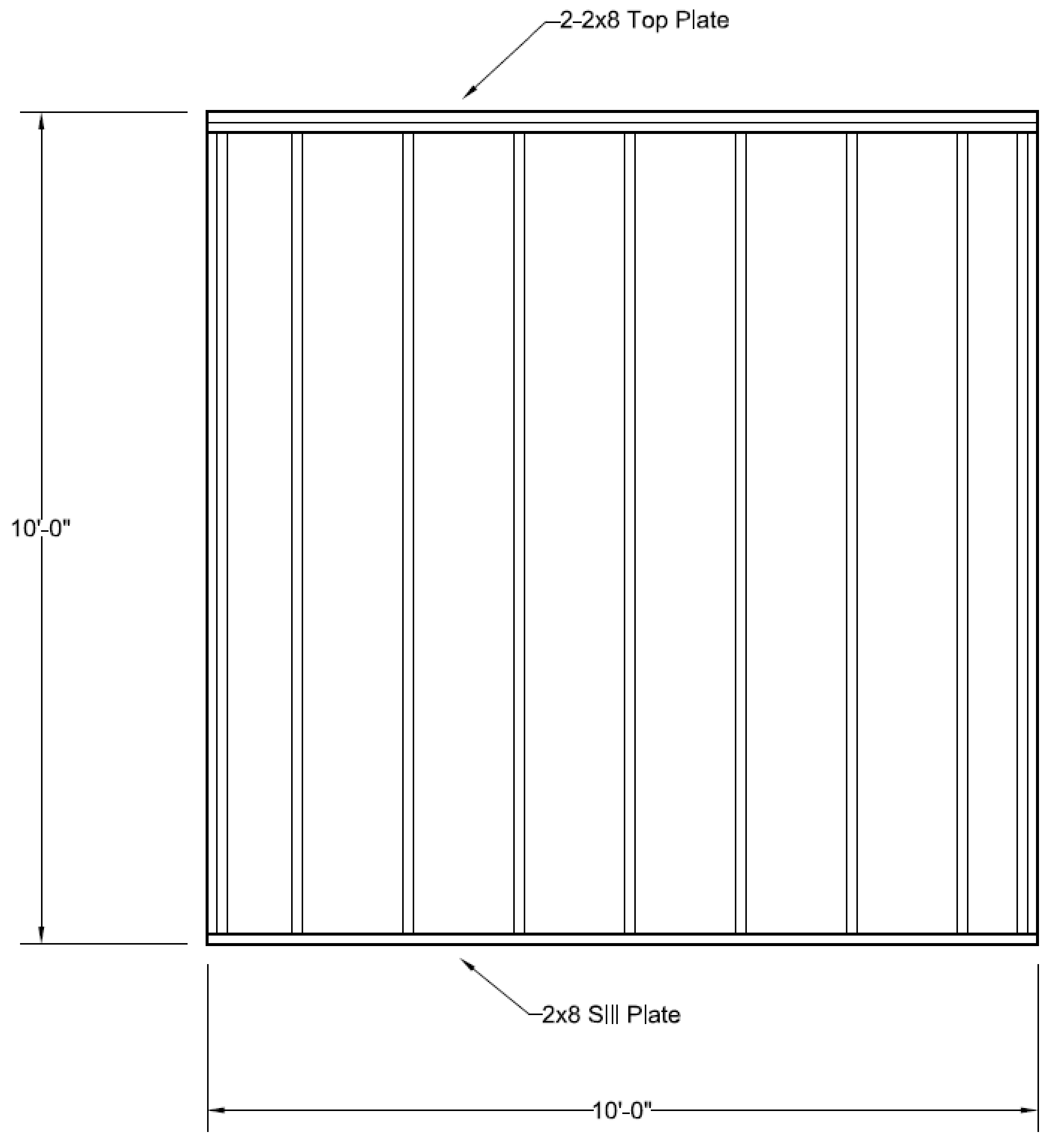
You also have the ability to save your custom reports by pressing “Save”. You can then choose your report from the “Report Name” drop down list. If you simply want to print All Input, All Output or All Input and Output those options are available by default.

This provides you with a quick way to get most of your printing needs taken care of within one print screen.

Notes:

Wood Wall Panel

Here we introduce the wood wall panel element. We will explain all of the input options, our external hold down and sheathing databases, and the results output.



Project Definition: **Wood_Wall_Panel.r3d**

Chord Members: 2-2x8
Stud Members: 2x8 @ 16" O.C.
Material: DF/Spine
Species: Com Species Group I DF,SP, Select Structural

Modeling Procedure

We'll start by setting up the design parameters for the wood wall panel.

- Click the Global icon ,

  On the "Codes" tab, select "NDS 2005: ASD" for the wood code.

  On the "Solution" tab, choose "6" for the Mesh Size.

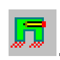
> **Note:** A wall panel analysis is a finite element analysis which requires a mesh of plate elements. This input allows you to adjust your mesh size. The finer the mesh, the more accurate your solution, however a finer mesh also can result in a longer solution run time.

- Click OK.

Now, let's set up our material properties.

- Click on Materials, and select the "Wood" tab.
- Change "DF/SPine" to Grade "Select Structural".
- Close the spreadsheet,

Let' start drawing in the geometry of the panel.

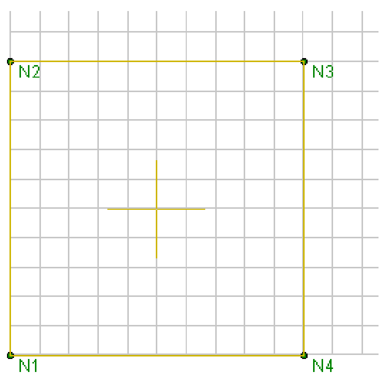
- Click the Draw Wall Panels icon .

  Under Material click on "Wood."

  Click "Apply."

- First click (0,0,0), then (0,10,0), then (10,10,0) and lastly (10,0,0).

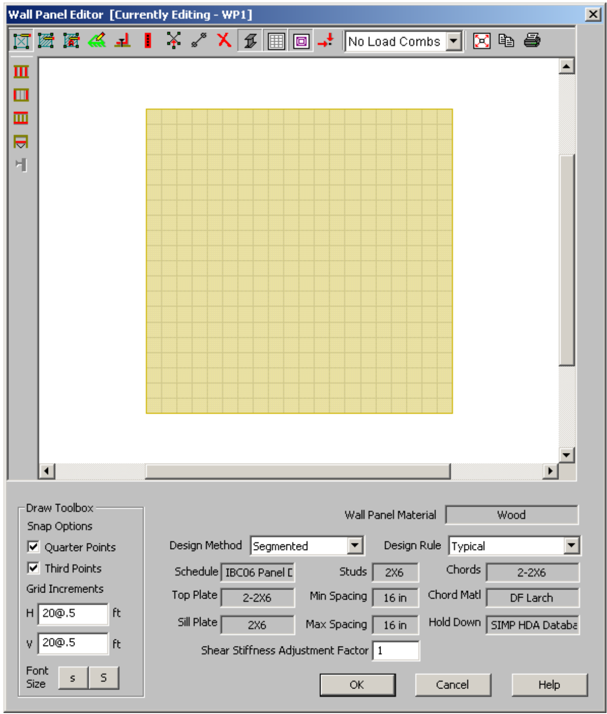
  After the fourth click a brown wall panel will appear:



- Right-click your mouse to release the drawing tool.

Now, let's look at wall panel boundary conditions and some of the other Wall Panel Editor options.

- Double-click on the wall panel to open the Wall Panel Editor:



Note: The Wall Panel Editor allows you to draw regions and openings, apply linear **boundary** conditions and display the wall panel layout after the model is solved.

It is important to note that with wall panels the key to getting results is defining regions. If there are no openings defined in a panel, then the program will automatically create one region that is defined by the entire panel. Thus, your results will be based on the entire wall.

- Click the icon within the editor.

Choose "Pinned" and then select a grid intersection at the base of the wall.

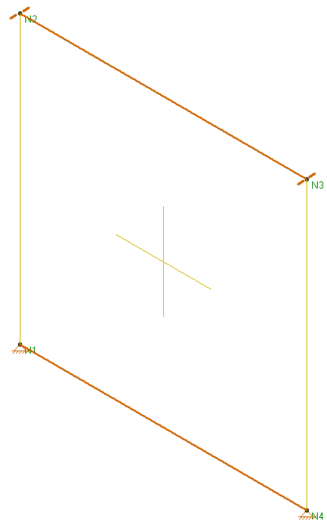
You'll now see a pinned graphic with an orange line that goes the full length of the base, thus signifying that the pin is continuous along the bottom of the wall.

- Once again click , but this time, we are only going to have a reaction in the Z direction.

Then click on a grid intersection at the top of the wall.

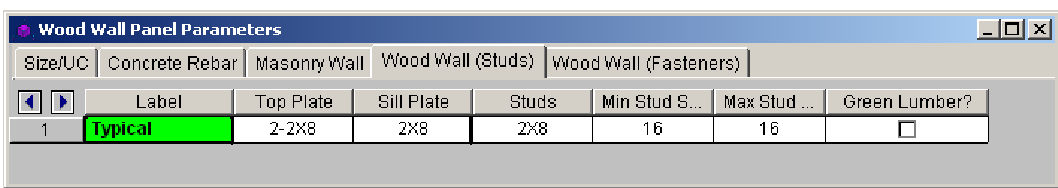
Click "OK" to exit the wall panel editor.

- Click the icon and you should see a view like this:



Let's review the framing design rules for this panel.

- Go to the Design Rules spreadsheet Design Rules.
- Click on the "Wood Wall (Studs)" tab.
  Change of all of the dimension lumber to 8" nominal depths.


Wood Wall Panel Parameters

Size/UC | Concrete Rebar | Masonry Wall | Wood Wall (Studs) | Wood Wall (Fasteners)

	Label	Top Plate	Sill Plate	Studs	Min Stud S...	Max Stud ...	Green Lumber?
1	Typical	2-2X8	2X8	2X8	16	16	☐

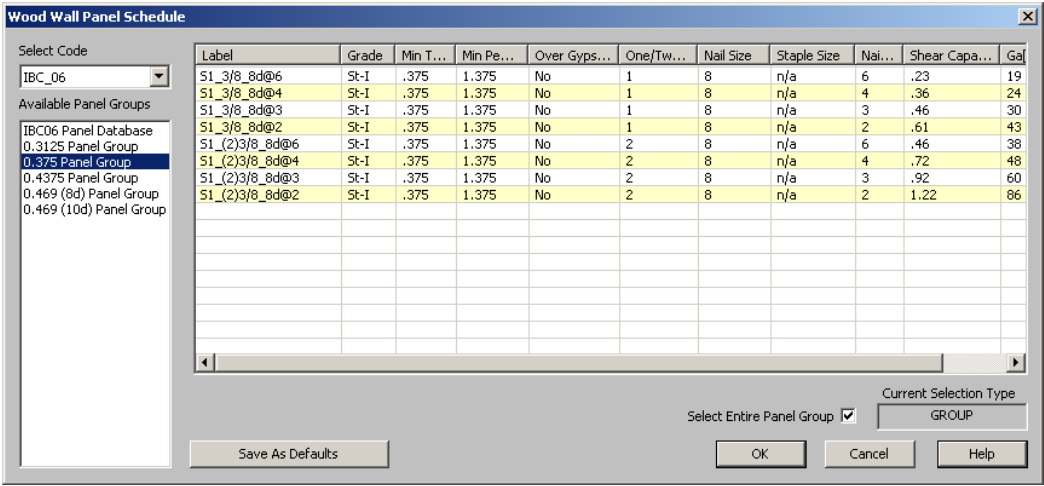
Note: If we had multiple wall panels with multiple stud spacing, stud sizing, etc., we could set up multiple Design Rules and define them per wall panel.

Now let's define our panel design.

- Click on the "Wood Wall (Fasteners)" tab.
- Click on the "Schedule" column, click the red arrow

This will open a "Wood Wall Panel Schedule" dialog that allows you to choose from groups of nailing or allows you to pick out a single nailing/sheathing thickness to use for the shear resistance of your wall panel.

- Under "IBC_06", choose "0.375 Panel Group".
  Click OK.

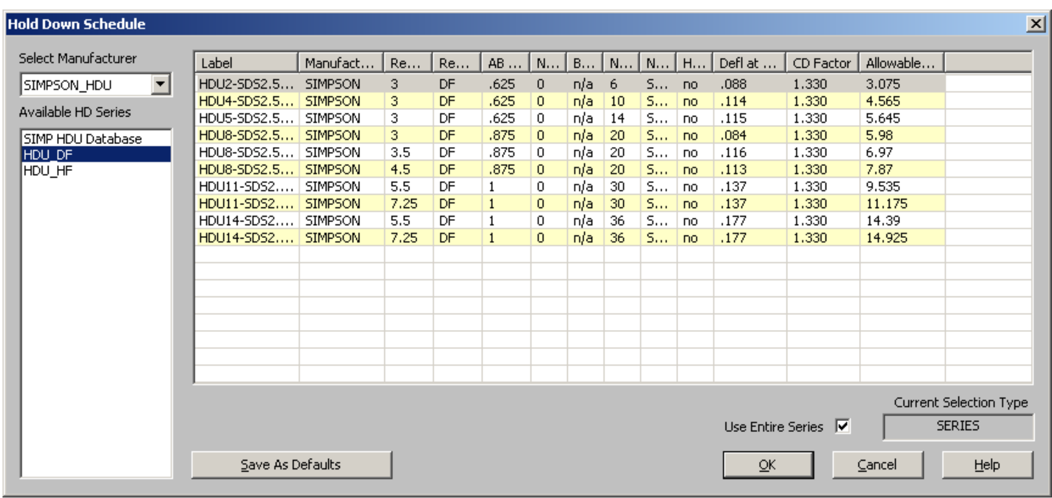


Note: If you choose a group of schedules, the program will optimize your wall panel for the most efficient schedule. If you choose a single schedule we will do a design check and give you a unity ratio to tell you if your schedule will work. You can input your own databases if these do not work for you.

- For "Double-Sided Panel?" choose "Optimum."

  For "HD Chords" choose "2-2x8" and leave "HD Chord Matl" as "Same as Wall."

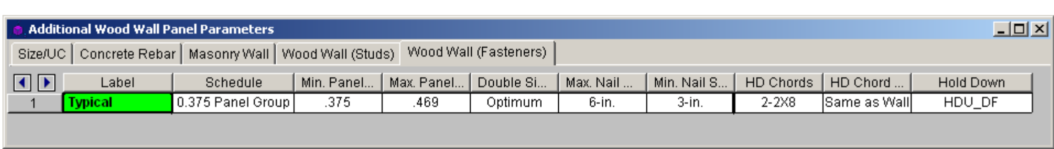
  Click on the red arrow in the "Hold Down" cell to open the Hold Down Schedule:



- Under "Simpson_HDU," choose "HDU_DF,"

  Click "OK."

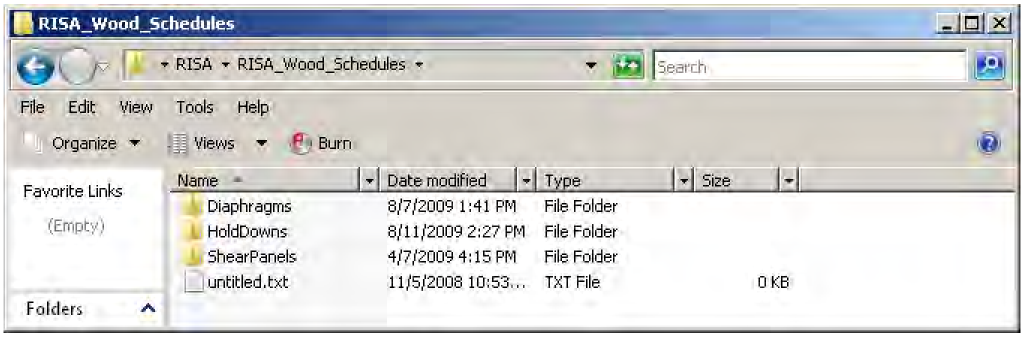
  Our spreadsheet should look like this:



Let's review the xml files that make up these databases.

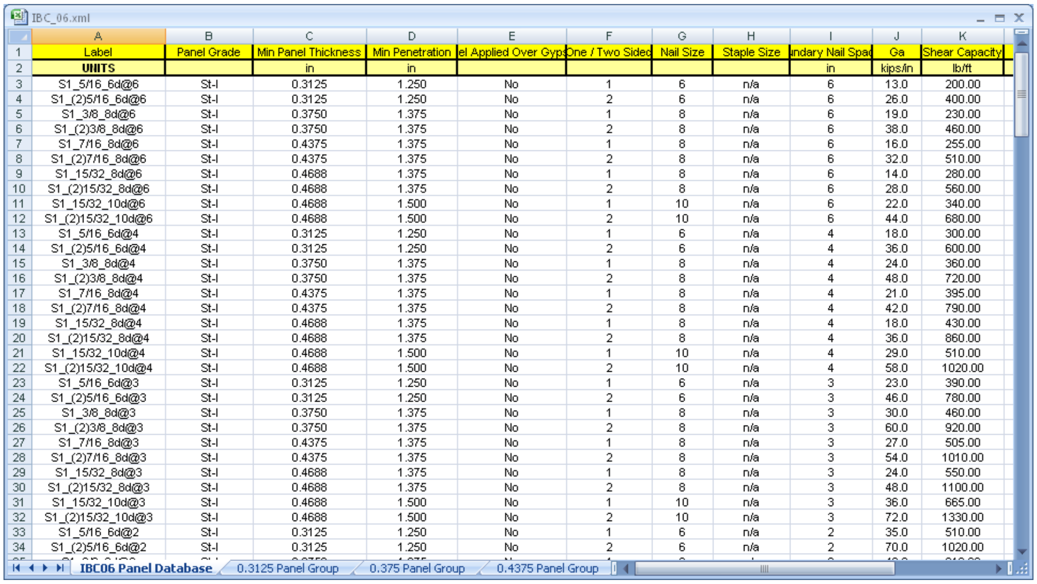
- Minimize the RISA-3D window and open a Windows Explorer window.
- Navigate to the C:\RISA\RISA_Wood_Schedules directory

  Here there will be three folders: Diaphragms, Hold Downs and Shear Panels:



- Open the shear panels folder,

  Open IBC_06.xml:



IBC_06.xml

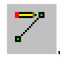
	A	B	C	D	E	F	G	H	I	J	K
1	Label	Panel Grade	Min Panel Thickness	Min Penetration	el Applied Over Gyps	One / Two Sided	Nail Size	Staple Size	undary Nail Spac	Ga	Shear Capacity
2	UNITS		in	in					in	kips/in	lb/ft
3	S1_5/16_6d@6	St-I	0.3125	1.250	No	1	6	n/a	6	13.0	200.00
4	S1_(2)5/16_6d@6	St-I	0.3125	1.250	No	2	6	n/a	6	26.0	400.00
5	S1_3/8_8d@6	St-I	0.3750	1.375	No	1	8	n/a	6	19.0	230.00
6	S1_(2)3/8_8d@6	St-I	0.3750	1.375	No	2	8	n/a	6	38.0	460.00
7	S1_7/16_8d@6	St-I	0.4375	1.375	No	1	8	n/a	6	16.0	255.00
8	S1_(2)7/16_8d@6	St-I	0.4375	1.375	No	2	8	n/a	6	32.0	510.00
9	S1_15/32_8d@6	St-I	0.4688	1.375	No	1	8	n/a	6	14.0	280.00
10	S1_(2)15/32_8d@6	St-I	0.4688	1.375	No	2	8	n/a	6	28.0	560.00
11	S1_15/32_10d@6	St-I	0.4688	1.500	No	1	10	n/a	6	22.0	340.00
12	S1_(2)15/32_10d@6	St-I	0.4688	1.500	No	2	10	n/a	6	44.0	680.00
13	S1_5/16_6d@4	St-I	0.3125	1.250	No	1	6	n/a	4	18.0	300.00
14	S1_(2)5/16_6d@4	St-I	0.3125	1.250	No	2	6	n/a	4	36.0	600.00
15	S1_3/8_8d@4	St-I	0.3750	1.375	No	1	8	n/a	4	24.0	360.00
16	S1_(2)3/8_8d@4	St-I	0.3750	1.375	No	2	8	n/a	4	48.0	720.00
17	S1_7/16_8d@4	St-I	0.4375	1.375	No	1	8	n/a	4	21.0	395.00
18	S1_(2)7/16_8d@4	St-I	0.4375	1.375	No	2	8	n/a	4	42.0	790.00
19	S1_15/32_8d@4	St-I	0.4688	1.375	No	1	8	n/a	4	18.0	430.00
20	S1_(2)15/32_8d@4	St-I	0.4688	1.375	No	2	8	n/a	4	36.0	860.00
21	S1_15/32_10d@4	St-I	0.4688	1.500	No	1	10	n/a	4	29.0	510.00
22	S1_(2)15/32_10d@4	St-I	0.4688	1.500	No	2	10	n/a	4	58.0	1020.00
23	S1_5/16_6d@3	St-I	0.3125	1.250	No	1	6	n/a	3	23.0	390.00
24	S1_(2)5/16_6d@3	St-I	0.3125	1.250	No	2	6	n/a	3	46.0	780.00
25	S1_3/8_8d@3	St-I	0.3750	1.375	No	1	8	n/a	3	30.0	460.00
26	S1_(2)3/8_8d@3	St-I	0.3750	1.375	No	2	8	n/a	3	60.0	920.00
27	S1_7/16_8d@3	St-I	0.4375	1.375	No	1	8	n/a	3	27.0	505.00
28	S1_(2)7/16_8d@3	St-I	0.4375	1.375	No	2	8	n/a	3	54.0	1010.00
29	S1_15/32_8d@3	St-I	0.4688	1.375	No	1	8	n/a	3	24.0	550.00
30	S1_(2)15/32_8d@3	St-I	0.4688	1.375	No	2	8	n/a	3	48.0	1100.00
31	S1_15/32_10d@3	St-I	0.4688	1.500	No	1	10	n/a	3	36.0	665.00
32	S1_(2)15/32_10d@3	St-I	0.4688	1.500	No	2	10	n/a	3	72.0	1330.00
33	S1_5/16_6d@2	St-I	0.3125	1.250	No	1	6	n/a	2	35.0	510.00
34	S1_(2)5/16_6d@2	St-I	0.3125	1.250	No	2	6	n/a	2	70.0	1020.00

IBC06 Panel Database | 0.3125 Panel Group | 0.375 Panel Group | 0.4375 Panel Group

Note: These are the capacity charts directly out of their respective building codes. These are what RISA-3D uses to optimize your shear panels. These spreadsheets are fully customizable. You can add new lines to the spreadsheet and even add your own database to this folder as long as you follow the existing format. Appendix F in our help file goes in depth about how to do this and the file structure required.

Let's load the structure.

Because distributed loads need to be added to a member, we need to add a "dummy" member to the top of our wall.

- Click the Draw New Members icon .

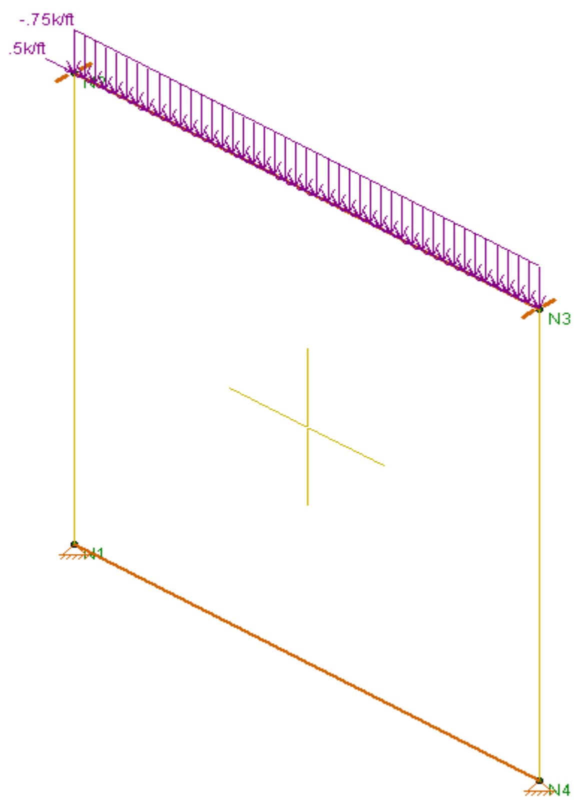
    Go to Wood >> Assign Shape Directly >> under "Start Shape," choose "2x8."

    Change "Angle" to "90". This will rotate the member 90 degrees.

    Click "Apply."

    Draw a member across the top of our wall.

- Now add a 0.5 k/ft distributed shear load (X-direction) across the top of the wall to BLC 1.
- Also add a -0.75 k/ft distributed axial load (Y-direction) across the top of the wall to BLC 1.



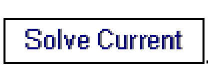
Let's solve and take a look at the results.

- In the Basic Load Cases spreadsheet 
,

    Place a "-1" in the Y Gravity column on the first line.

- In the Load Combinations spreadsheet 
,

    Place a "1" under BLC and a "1" under Factor.

- Click 
.

Now we have a solved model. Let's review the results.

In the Wall Panel Design spreadsheet Wall Panel Design, the last two spreadsheets are used to give a summary of the wood wall panel checks.

- Click the "Wood Wall Axial" tab.

  Here we get our axial code check results for the studs and chords.

- Click the "Wood Wall In-Plane" tab.

  Here we get to see which shear panel and nailing were chosen, as well as which hold down, and the corresponding code checks.

We can also review this information in a Detail Report.

- When in the Wall Panel Design spreadsheet, click Detail Report for Current Item.

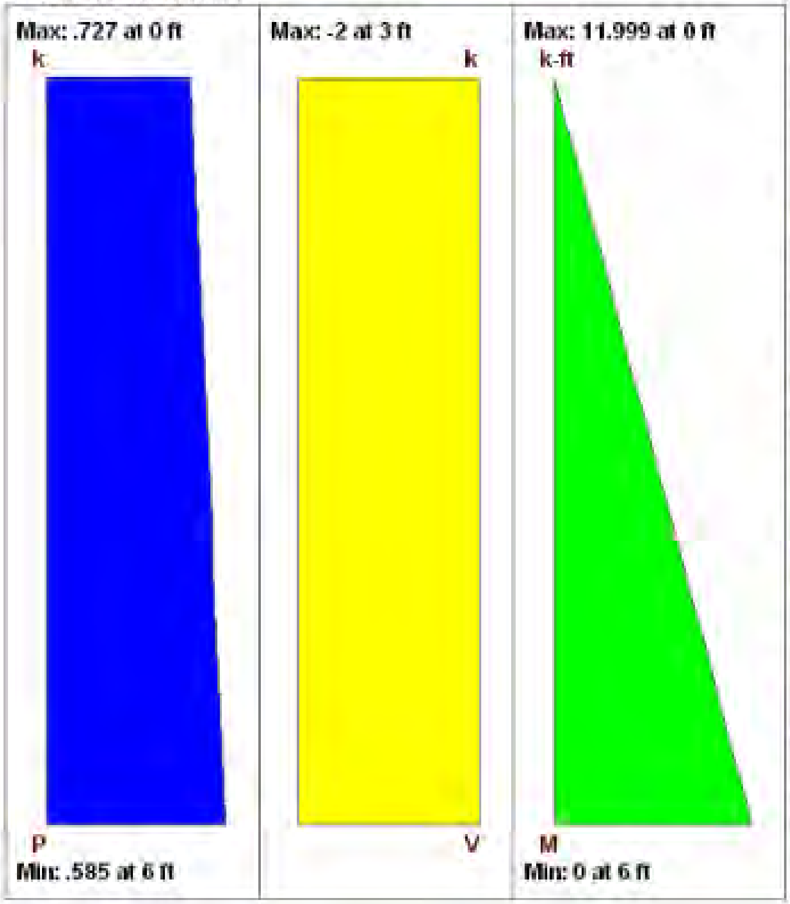
This opens a detail report for our wall panel. The wall panel detail report is divided into four sections: input echo, design results, design details and a graphical view.

The **Input Echo** gives a summary of geometry, codes and materials for this wall. If we had been optimizing the "Panel Schedule" or the "Stud Spacing", we would see our final selection here as well:

CRITERIA		MATERIALS		GEOMETRY	
Code	: NDS 2005	Wall Studs	: SPF	Total Height	: 10 ft
Type of Design	: ASD	Stud Size	: 2X8	Total Length	: 10 ft
Wall Material	: SPF	Chord Material	: SPF	Wall H/W Ratio	: 1.00
Panel Schedule	: 0.375 Panel Group	Chord Size	: 2-2X8		
Optimize HD	: Yes	Top Pl & Sill	: SPF	Stud Spacing	: 16 in
HD Manufacturer:	SIMPSON	Top Pl Size	: 2-2X8		
		Sill Pl Size	: 2X8		

The **Design Results** reports the axial, shear and moment diagrams. We also get our capacities along with a code check ratio, as well as deflection information (from help file below):

ENVELOPE DIAGRAMS



DESIGN SUMMARY

SHEAR PANEL		
Required Cap	.4	k/ft
Provided Cap	: .41	k/ft
Ratio	: .976	
Governing LC	: 1	
CHORDS		
Required Cap	: -2.036	k
Provided Cap	12.833	k
Ratio	: .159	
Governing LC	: 1	
STUDS		
Required Cap	: .145	k
Provided Cap	11.86	k
Ratio	.012	
Governing LC	: 1	
HOLD-DOWNS		
Required Cap	: 2.036	k
Provided Cap	: 2.055	k
Ratio	: .991	
Governing LC	: 1	
DEFLECTIONS		
Elastic Comp	: .005	in
Shear Comp	: .077	in
HD Elong	: .069	in
Tot Deflection	.151	in
Governing LC	: 1	

The **Design Details** gives the specifications for the shear panel and hold down selection (from help file below):

DESIGN DETAILS

SELECTED SHEAR PANEL : S1_(2)15/32_10d@3_W

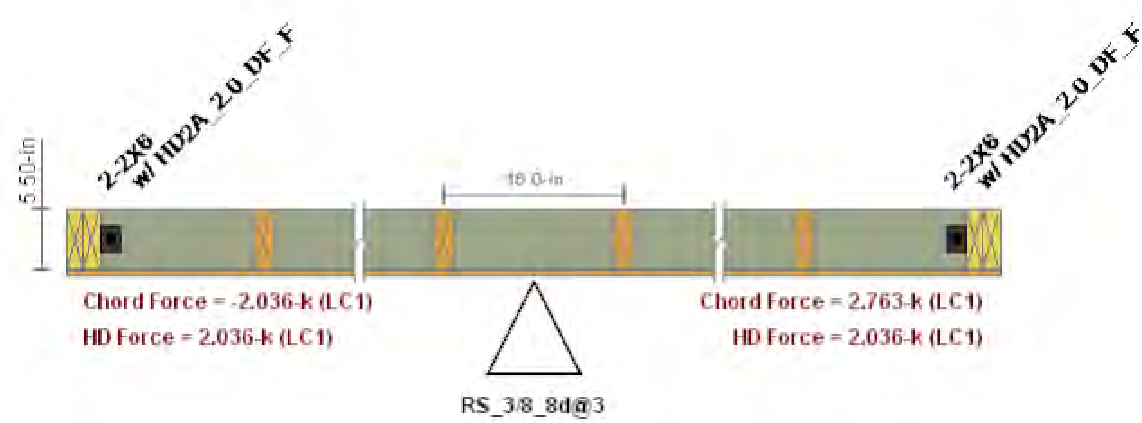
Panel Grade	: St-I		Nail Size	: 10d		Num Sides	: Two	
Panel Thick	: 0.469	in	Reqd Pen	: 1.500	in	Over Gyp Brd.	: No	
			Reqd. Spacing	: 3	in	Shear Capacity	: 1.860	k/ft

NOTE: defines a 10d nail as being 3.0" x 0.1480" common, or
3.0" x 0.122" galvanized box

SELECTED HOLD-DOWN : HD2A_1.5_HF_F

Raised	: No		Bolt Size	: 0.625	in	Reqd Chord Thk	: 1.50	in
AB Diameter	: 0.625	in	Num Bolts	: 2		Reqd Chord Mat	: Hem Fir	
						Allow Tension	: 1.320	k

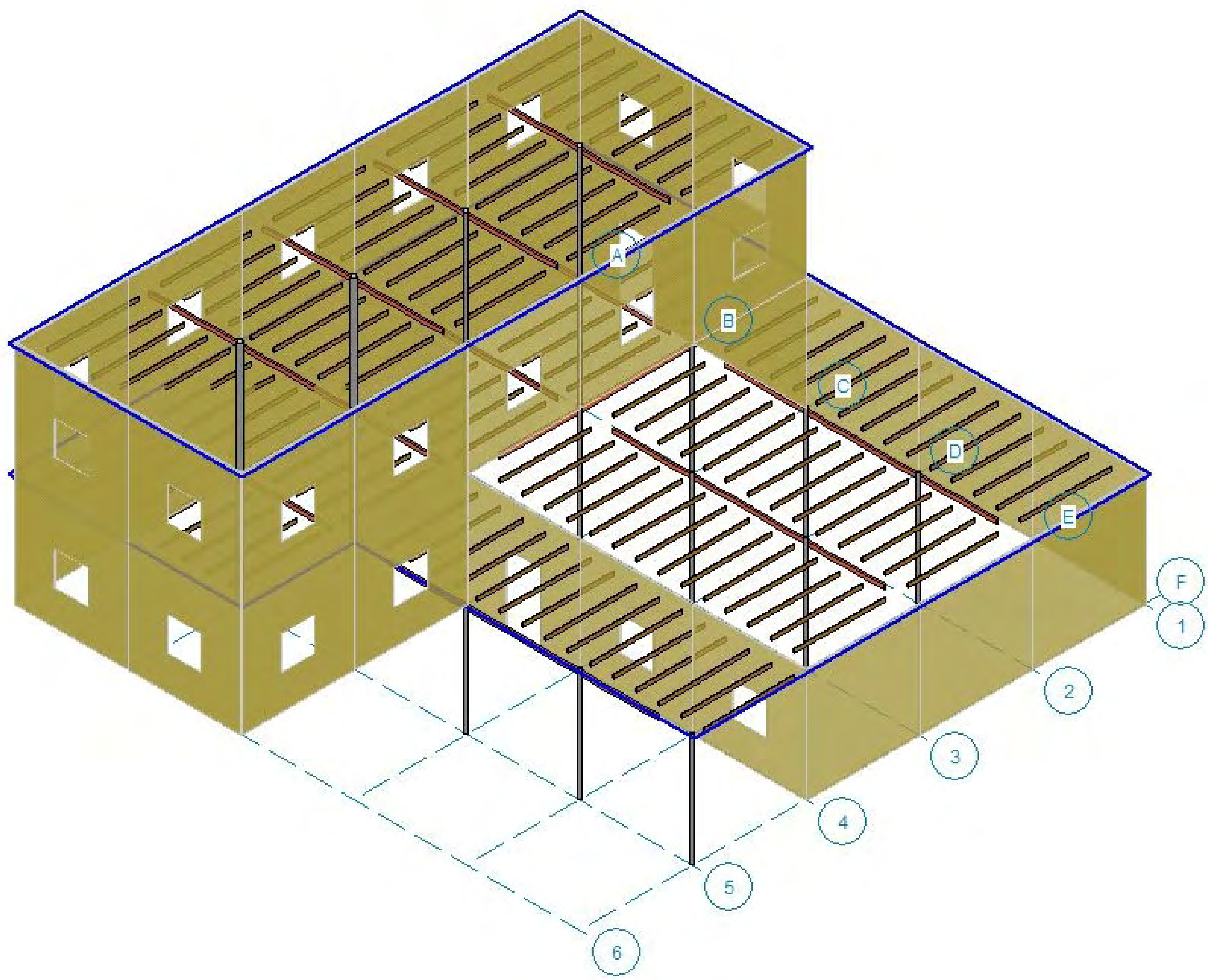
The **Graphical View** gives a graphical view of your wall, along with all of the specifications for the wall and the forces in the chords and hold downs (from help file below):



Notes:

RISAFloor Basics

Now we will consider models that focus on specific wood concepts/features in the program. This model is intended to illustrate a basic model within RISAFloor. We will use this model to navigate thru the RISAFloor interface, and become familiar with the different ways to view the model.

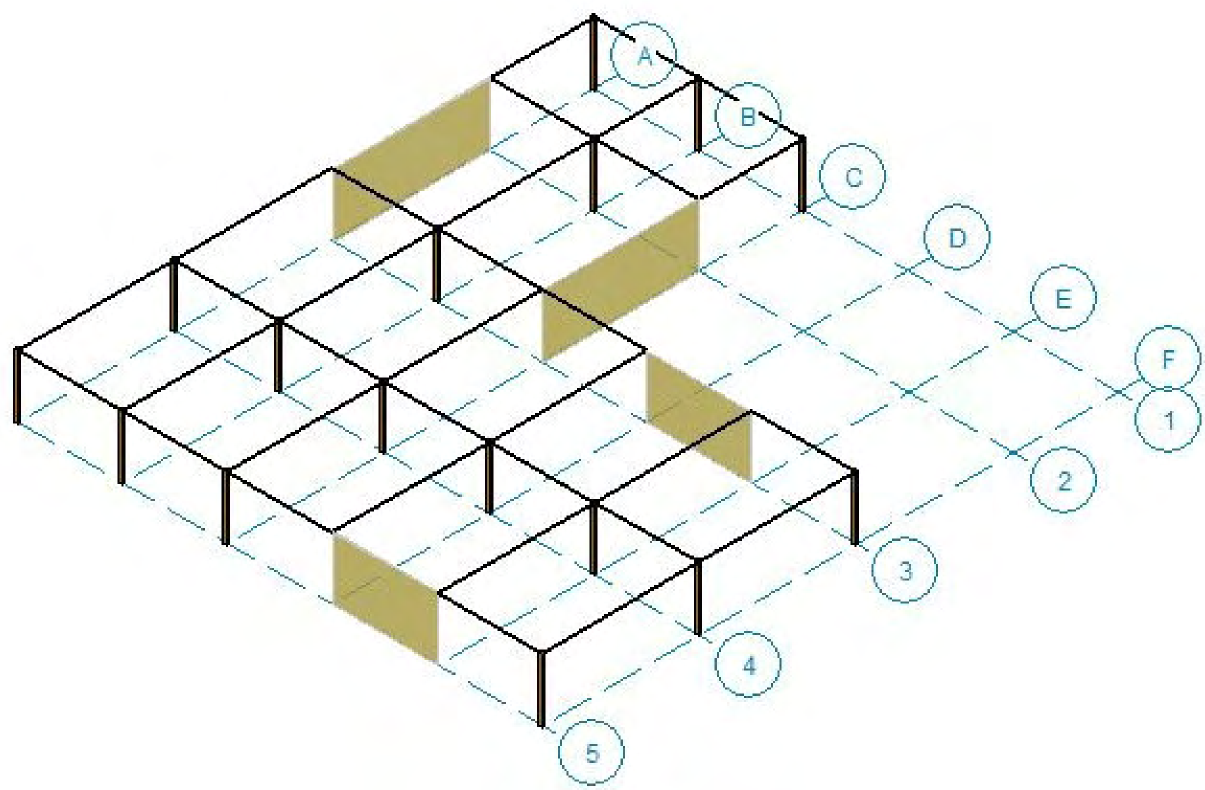


Project Definition: RISAFloor Basics.rfl

Two Story – 10, 20 feet
Wood Beams –DF/SPine, Rectangular
24F-1.8E SP Balanced- Glulam_SouthernPine
Steel Columns – Pipe
Walls – 2x6 Wood, DF/SPine, Design Rule: Typical

Graphical Modeling Basics

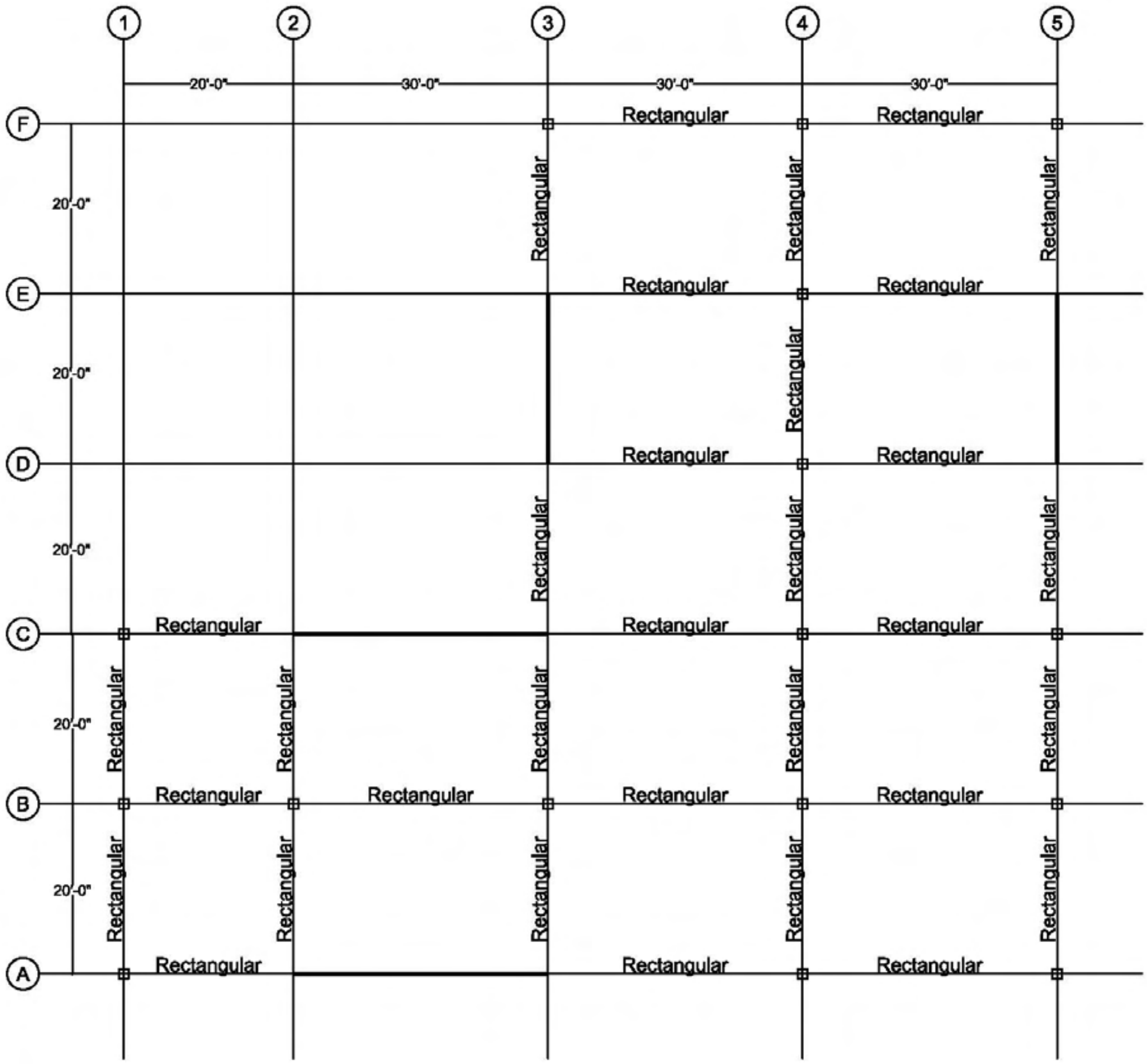
This model is intended to demonstrate various procedures and techniques associated with basic graphical modeling of walls, columns, and beams in RISAFloor. The development and use of Project Grids is also included.



Project Definition: Basics.rfl

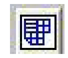
Single Story – 12 feet
Wood Beams –DF/SPine, Rectangular
Wood Columns – Gravity- Rectangular
Lateral- Double Rectangular
Walls – 2x6 Wood, DF/SPine, Design Rule: Typical

Framing Plan: Basics.rfl



Modeling Procedure

We'll begin by creating a new Floor and generating a Project Grid.

- Start a new model by clicking on the Create New Floor Plan button 
- Create an Original Floor with an elevation of 12 feet.

Ignore the Default Area Load, Deck, and Angle entries as we will revise these later.

Let's setup the design parameters for the wood wall panel.

- Click the Global icon ,

  On the "Codes" tab, select "NDS 2005: ASD" for the wood code.

  Click OK.

Define a Project Grid as follows:

- On the Horizontal Project Grid- Z Axis Tab

  Start Label of '1' and Start Location of '0'

  Enter '20,3@30' for the Increments

  Click 'Generate'.

- On the Vertical Project Grid- X Axis Tab

  Set the Start Label to 'A' and Start location to '0',

  Enter '5@20' for the Increments

  Click 'Generate'.

Let's take a look at our Materials.

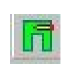
- Open the Material spreadsheet by clicking on the Materials button 

on the Data Entry Toolbar.

  Click the "Wood" tab.

  Change the DF/SPine to "Select Structural".

We will now draw the Walls within the structure.

- Draw 'Lateral' Walls by clicking on the Draw Wall button 

  Select: Wood, DF/SPine, Design Rule: Typical, Lateral

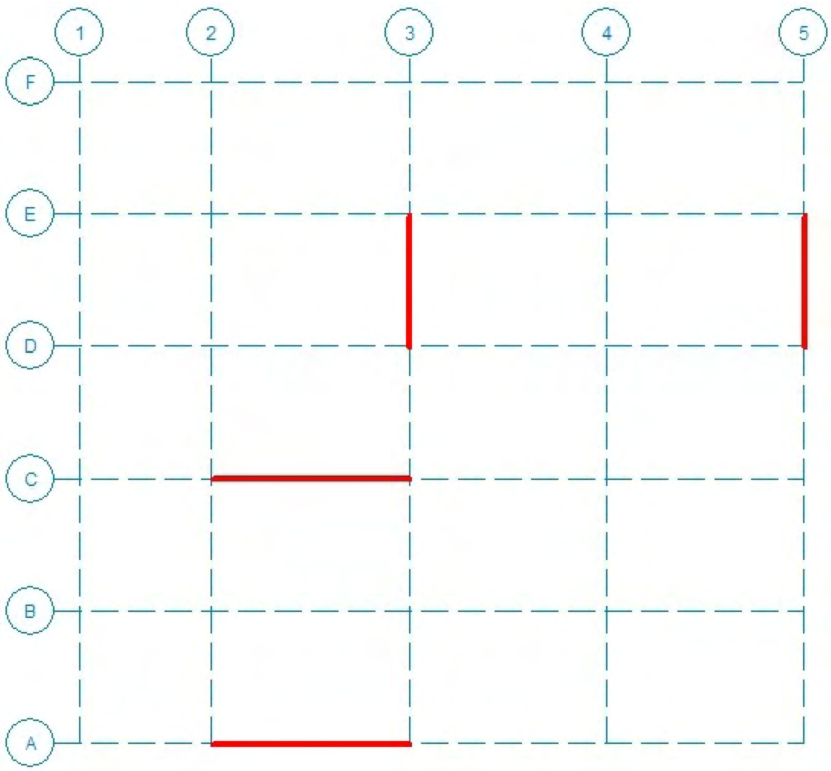
  Draw between the following Grid Locations:

  A-2 & A-3
  C-2 & C-3
  D-3 & E-3
  D-5 & E-5

Note: In RISA, you can design Wood walls, Masonry walls, or General wall types such as concrete walls. You just need to specify a different Material when you draw the wall.

Note: RISAFloor allows you to make the distinction between gravity and lateral elements. The main use of this functionality is to define which members you want to transfer to RISA-3D for the lateral analysis/design. Any elements that will participate in the lateral force resisting system should be labeled “Lateral”. Elements only to be used in RISAFloor for gravity force design should be labeled “Gravity”. This distinction also becomes important when defining diaphragm regions. RISAFloor requires that all diaphragm region boundaries are made up of lateral members.

Your model should now look like the image below:



We will now draw the Columns within the structure.

- Draw ‘Lateral’ Columns by clicking on the Draw Column button 

  Select: Wood, DF/SPine, Rectangular Double, Lateral

  Box or click the following grid Intersections:

A-1	C-1
A-4	C-5
A-5	F-3
B-1	F-4
B-5	F-5
B-3	

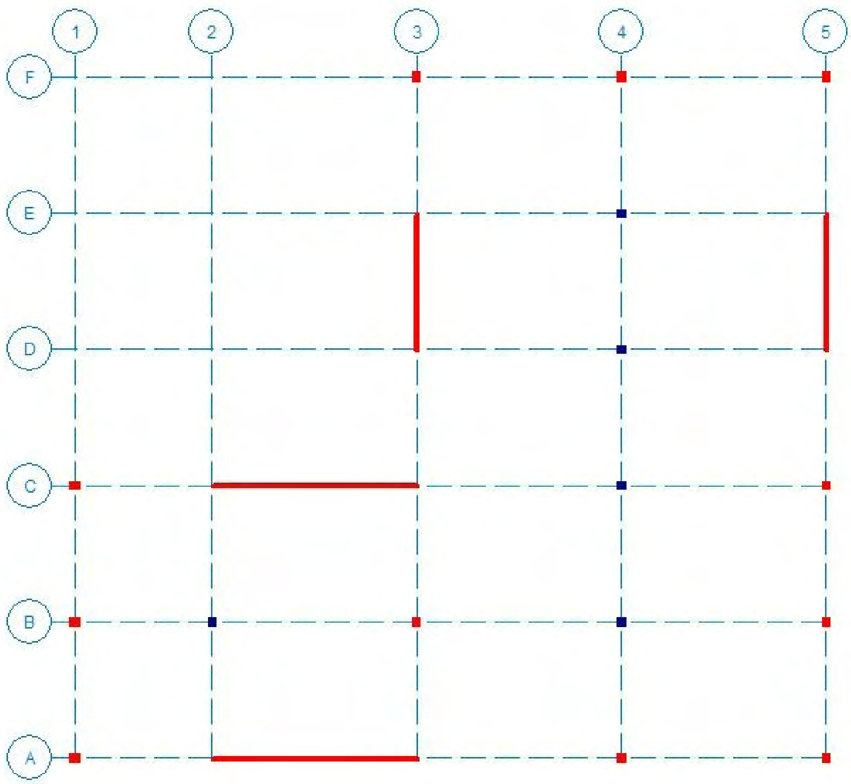
- Draw 'Gravity' Columns by clicking on the Draw Column button 

  Select: Wood, DF/SPine, Rectangular, Gravity

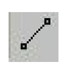
  Box or click the following Grid Intersections:

B-2	C-4
D-4	B-4
E-4	

Your model should now look like the image below:



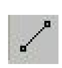
We will now draw the main Beams in the structure.

- Draw 'Lateral' Beams by clicking on the Draw Beam button 

  Select: Wood, DF/SPine, Rectangular, Pinned-Pinned, Lateral

  Draw along the perimeter of the building, and A-3 to C-3.

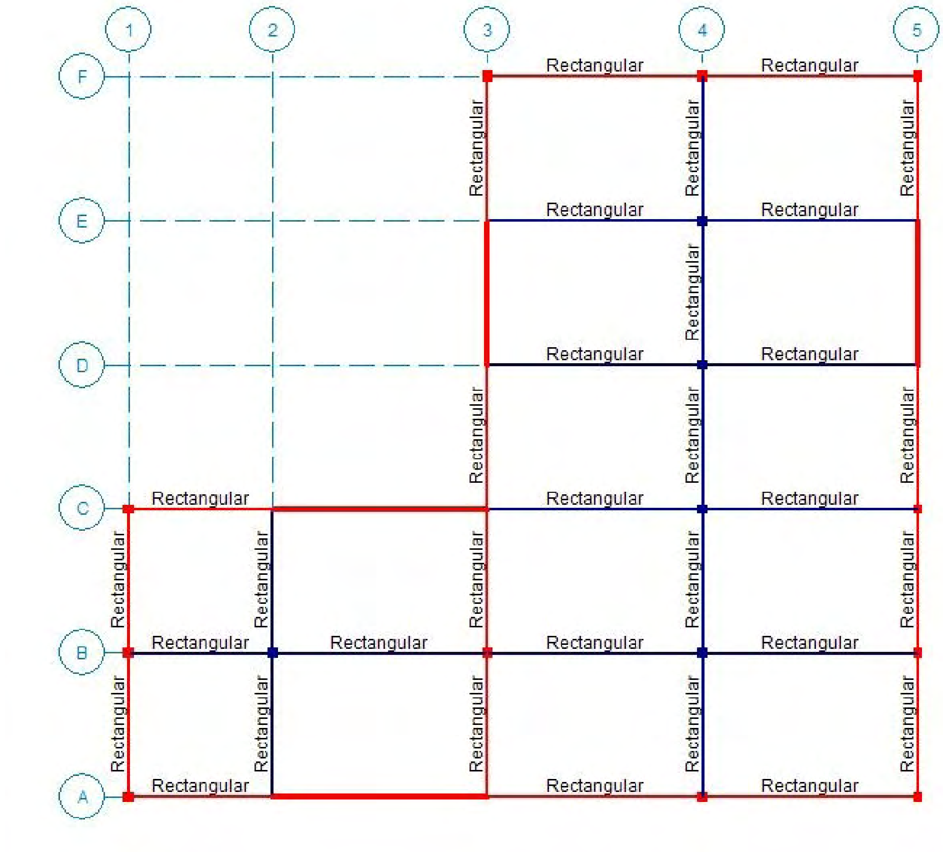
> **Note:** The Flexible diaphragm requires that we have entire the perimeter of the diaphragm region must consist of lateral members. These are the members that the diaphragm will transfer the chord and collector forces into. We'll discuss this further in the diaphragm models.

Draw 'Gravity' Beams by clicking on the Draw Beam button 

Select: Wood, DF/SPine, Rectangular, Pinned-Pinned, Gravity

Draw along all grid lines in all open bays:

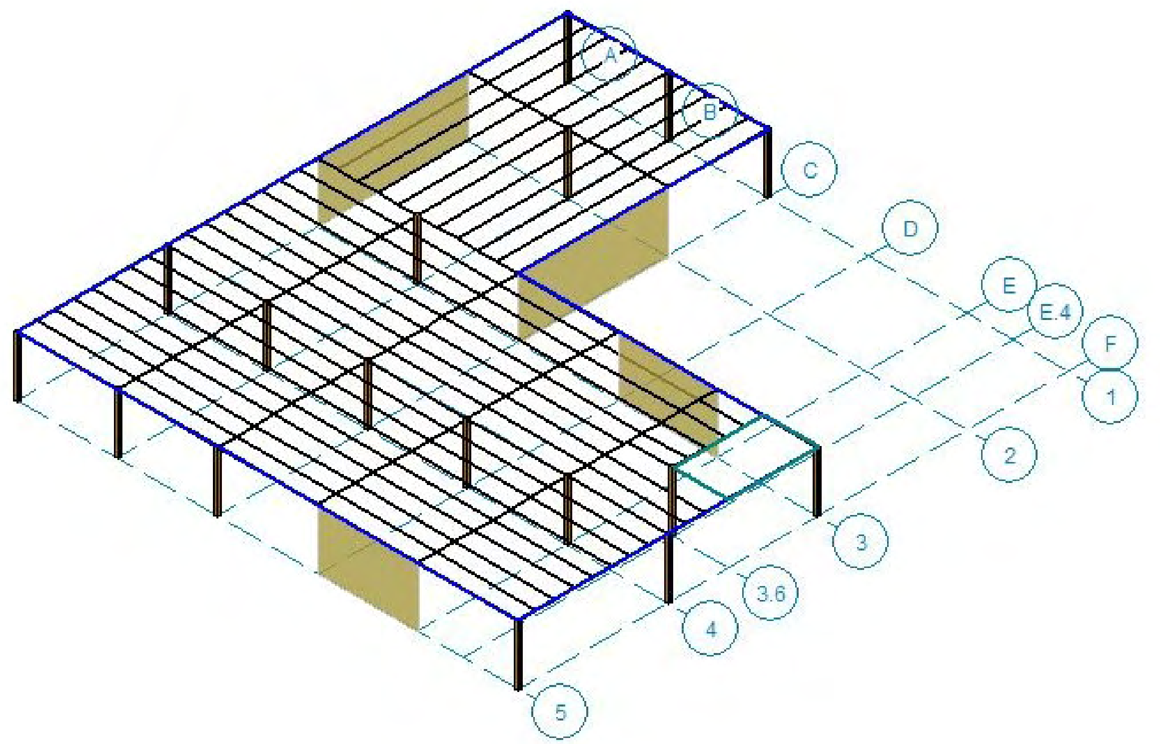
Your model should now look like the image below:



Notes:

Floor Systems Basics

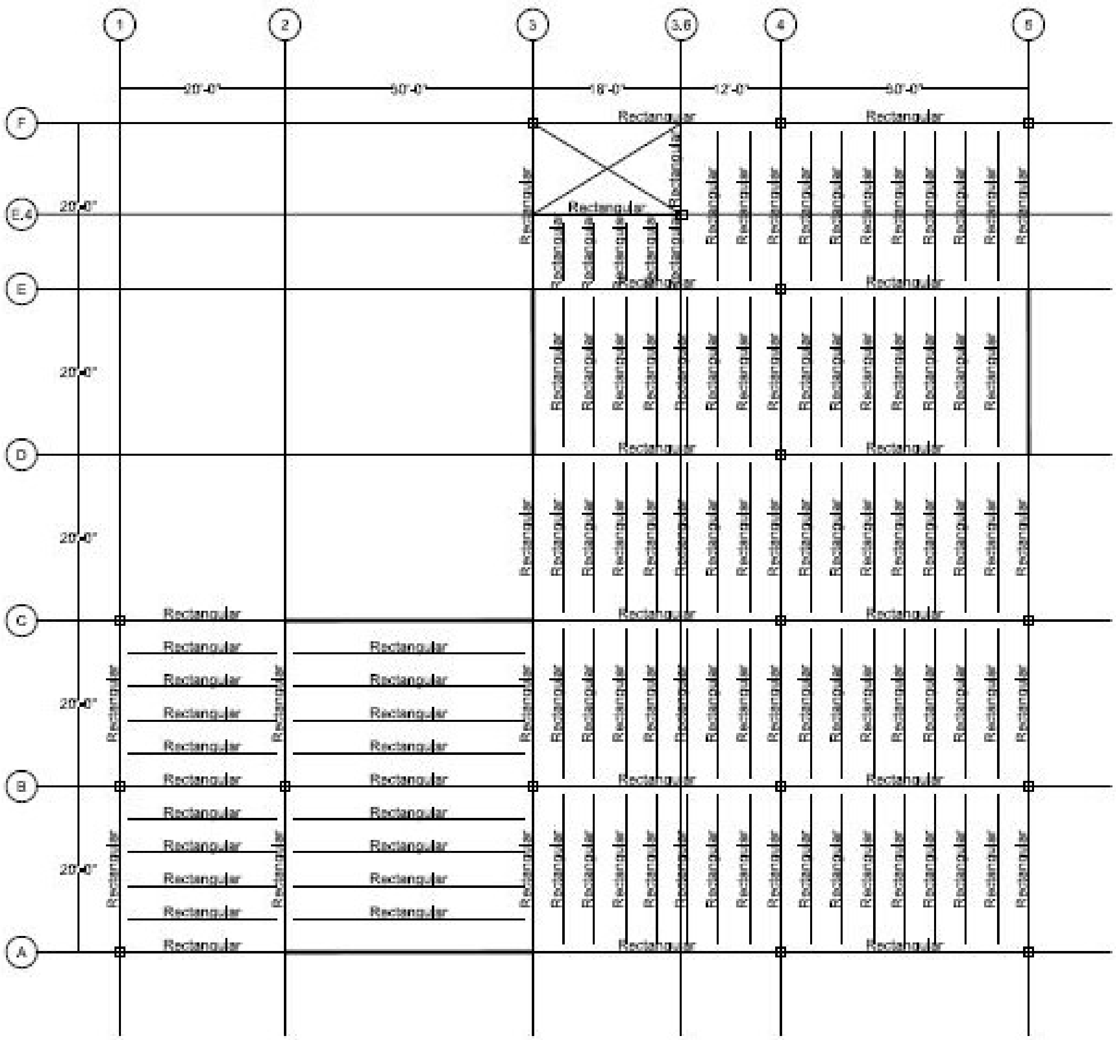
This model is intended to demonstrate various procedures and techniques associated with modeling Infill Beams (interior bay framing), Decks/Slabs, and Deck/Slab Openings and Perimeters in RISAFloor. This model will also be used to begin the discussion of Plot Options.



Project Definition: Floor_Systems.rfl

Single Story – 12 feet
Wood Beams -DF/SPine, Rectangular
Wood Columns – Gravity- Rectangular
Lateral- Double Rectangular
Walls – 2x6 Wood, DF/SPine, Design Rule: Typical

Framing Plan: Floor_System.rfl

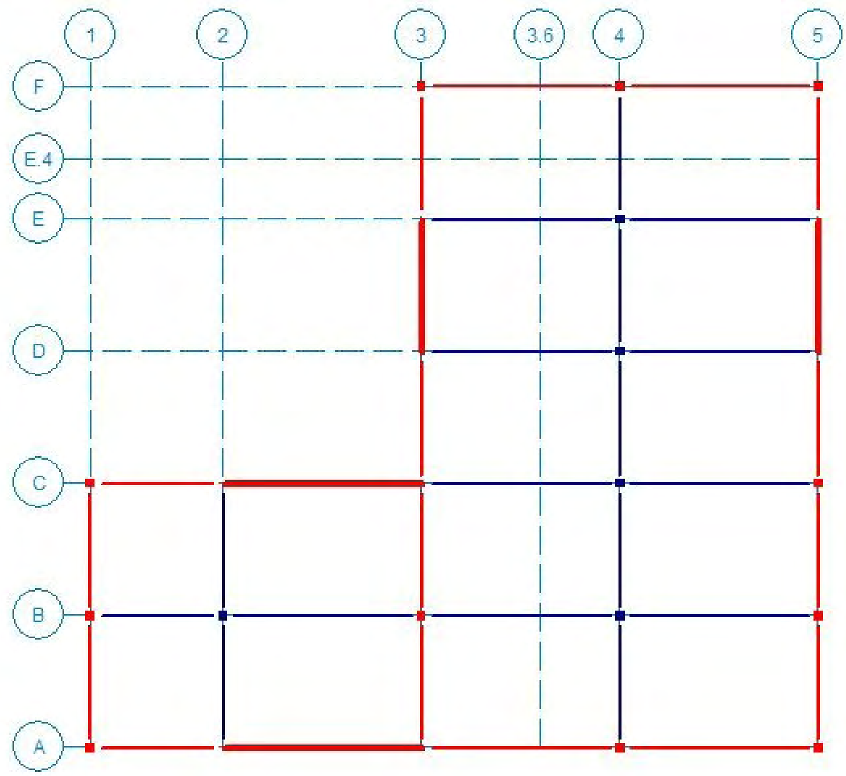


Modeling Procedure

We'll begin with the previously created model, Basics.rfl.

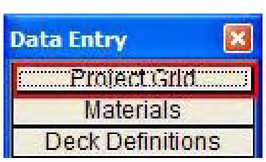
- Open the file 'Basics.rfl' from the Training CD and save the model using the new file name 'Floor_Systems.rfl'.

Your model should now look like the image below:

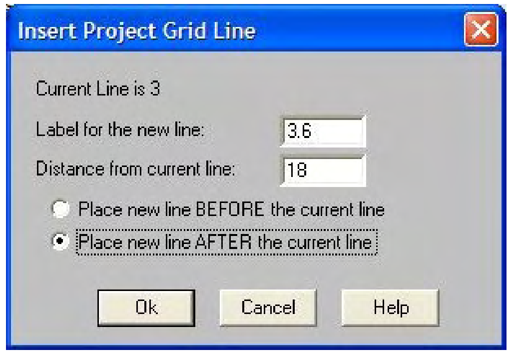


We will now modify the Project Grid and frame the Stairwell by inserting new Grid Lines and a column.

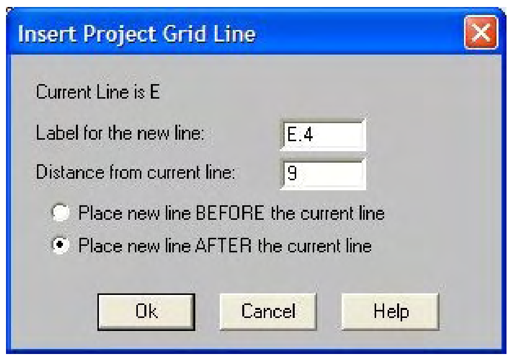
- Open the Project Grid Spreadsheet from the Data Entry toolbar



- On the Horizontal Project Grid- Z Axis Tab, move the cursor onto Label- 3 and click on the Insert New line icon or press F3
- Insert line labeled '3.6', with an increment of 18 feet AFTER Grid Line '3'



- Switch to the Vertical Project Grid- X Axis Tab
- Repeat the above steps to insert new Grid Line, labeled ‘E.4’, with an increment of 9 feet AFTER Grid Line ‘E’.

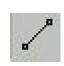


- Draw ‘Gravity’ column by clicking on the Draw column button 

  Select: Wood, DF/SPine, Rectangular, Gravity

  Click on the intersection of Grid Lines E.4 & 3.6.

> **Note:** RISA programs recall the last entries made in each dialog during the same session, even if different files are opened. This is something to pay attention to, especially when utilizing the graphical modification commands.

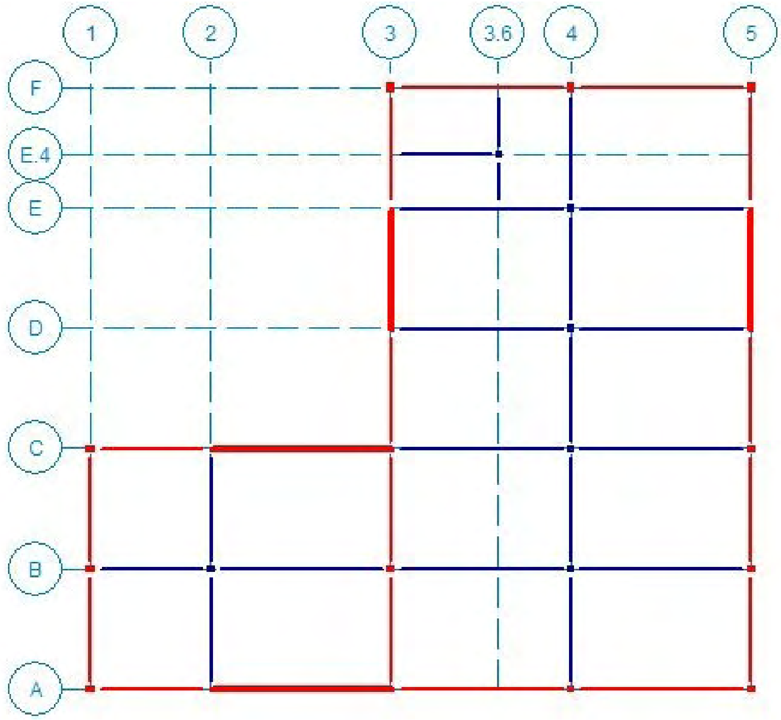
- Draw ‘Gravity’ Beams by clicking on the Draw Beam button 

  Select: Wood, DF/SPine, Rectangular, Pinned-Pinned, Gravity

  Draw in the open bays between the following column locations:

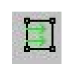
  E.4-3 & E.4-3.6
  E.4-3.6 & F-3.6
  E.4-3.6 & E-3.6

Your model should now look like the image below:



> **Note:** RISAFloor requires that all framing be a closed circuit with beams, columns or walls. The automatic load attribution and the Infill Framing require that framing be on all sides. Refer to the Stranded Column section in the Reference section for an example.

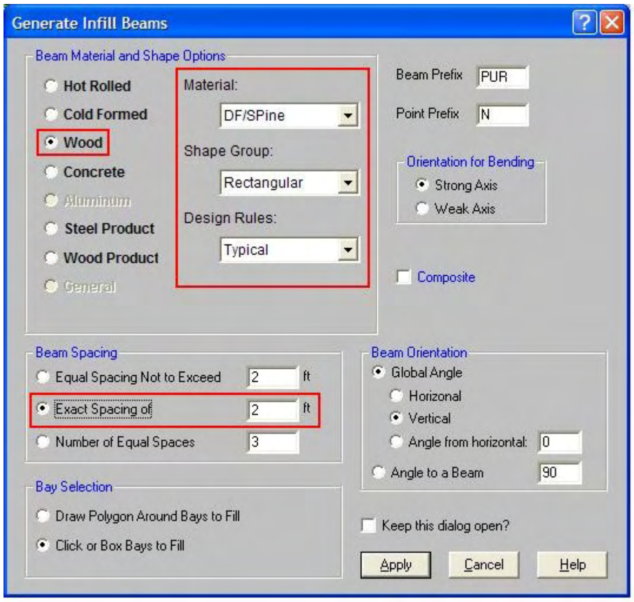
Now we will model the interior bay framing within the structure.

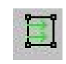
- Draw Vertical Infill Beams by clicking on the Generate Beams within a Bay button 

  Select: Wood, DF/SPine, Rectangular, Vertical

  Beam spacing exact spacing of 2’ o.c. oriented

  Draw in the open bays within ALL bays between Grid Lines ‘3’ and ‘5’ (except for the Stairwell)



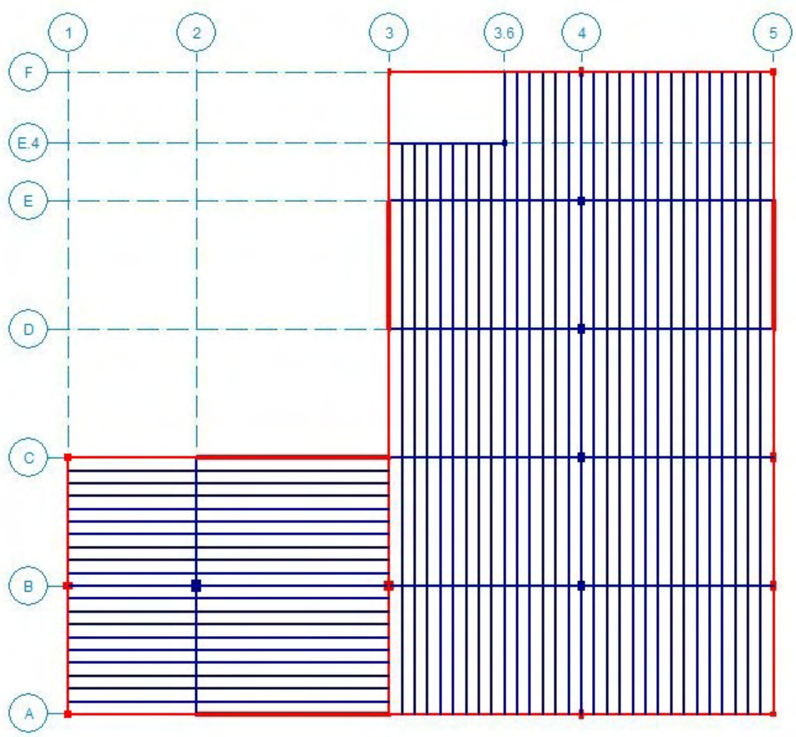
- Draw Horizontal Infill Beams by clicking on the Generate Beams within a Bay button 

  Select: Wood, DF/SPine, Rectangular, Pinned-Pinned, Horizontal

  Beam spacing exact spacing of 2' o.c. oriented

  Draw in the open bays within ALL bays between Grid Lines '1' and '3'

Your model should now look like the image below:



Now we will generate the deck/slab perimeter, as well as an opening within the stairwell.

- Draw Slab by clicking on Create Slab Edges button 

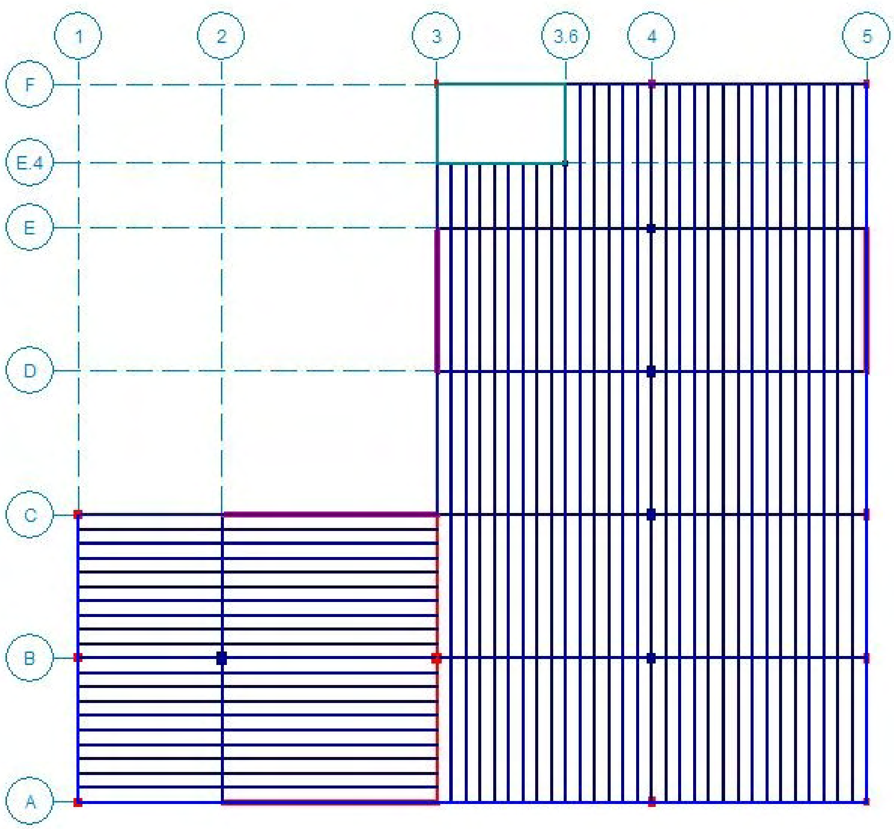
> **Note:** RISAFloor uses the terminology ‘slab edge’ to denote the extents of a diaphragm system. This applies not only to conventional slabs but also metal and wood decks.

- Enter an Edge Overhang Distance of 0 inch along the entire perimeter of the structure- Rigid.
- Click on Create a Slab perimeter- Consider ALL selected Beams/Walls for the SLAB perimeter- Press Apply
- Draw an opening in the Slab by clicking on Create Slab Edges button 

You’ll notice that the Edge Overhang Distance of 0 inch will be remembered.

- Click on Create an Opening Perimeter- Click within or Box the Beam/Wall perimeter for the Opening and click inside the stairwell.

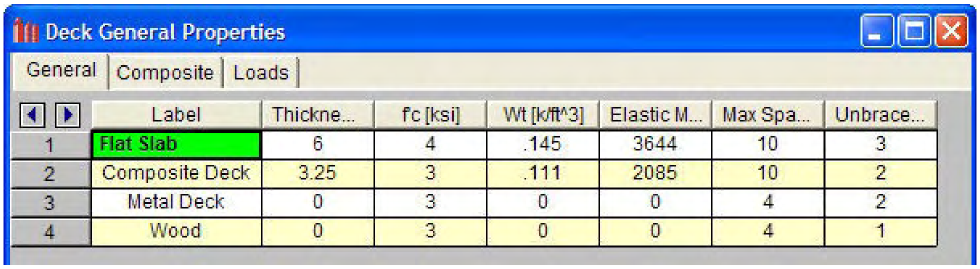
Your model should look like the image below (the dark blue outlines the slab edge, and the light blue outlines the opening):



Now we will model the Deck/Slab.

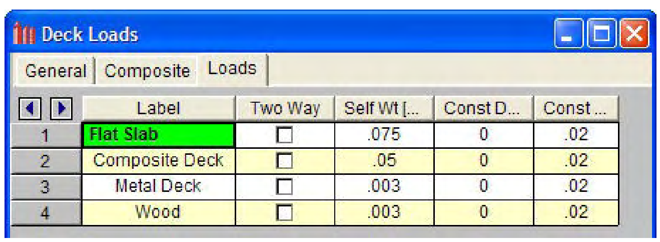
Let's review the default Deck properties:

- Click on the "Deck Definitions" spreadsheet on the Data Entry Toolbar.



	Label	Thickne...	f'c [ksi]	Wt [k/ft^3]	Elastic M...	Max Spa...	Unbrace...
1	Flat Slab	6	4	.145	3644	10	3
2	Composite Deck	3.25	3	.111	2085	10	2
3	Metal Deck	0	3	0	0	4	2
4	Wood	0	3	0	0	4	1

Note: The unbraced length that the deck provides to the top face of the beams may be entered here. If left blank the deck is assumed not to brace the top face and the unbraced length is determined from the spacing of the framing, the full length of beam or any value specified in the Beams Spreadsheet.



	Label	Two Way	Self Wt [...	Const D...	Const ...
1	Flat Slab	☐	.075	0	.02
2	Composite Deck	☐	.05	0	.02
3	Metal Deck	☐	.003	0	.02
4	Wood	☐	.003	0	.02

Note: The Self Weight is the weight of the deck system. This weight will be used as part of the dead load (Pre-DL or Post-DL). This weight will automatically calculate and be applied based on the Slab Edge.

- Click on the Floors Spreadsheet on the Data Entry Toolbar

  Change the Deck Default for 'Floor Plan 1' to 'Wood' and the Deck Angle is set to '0' degrees.

  ('0' degrees indicates the deck runs Horizontal.)

	Label	Elevatio...	Area Load ...	Deck Defa...	Deck Angl...	Parent	Inacti...	Splic...	Splic...
1	Floor Plan 1	12	Office	Wood	0	None	☐	12	Moment

- Draw a 'Local' Deck/Slab by clicking on the Assign Slab or Deck Properties button

  Select: Wood, oriented 'Parallel to X Axis'

  Select Click Within or Box the Deck Area Perimeter- Press Apply

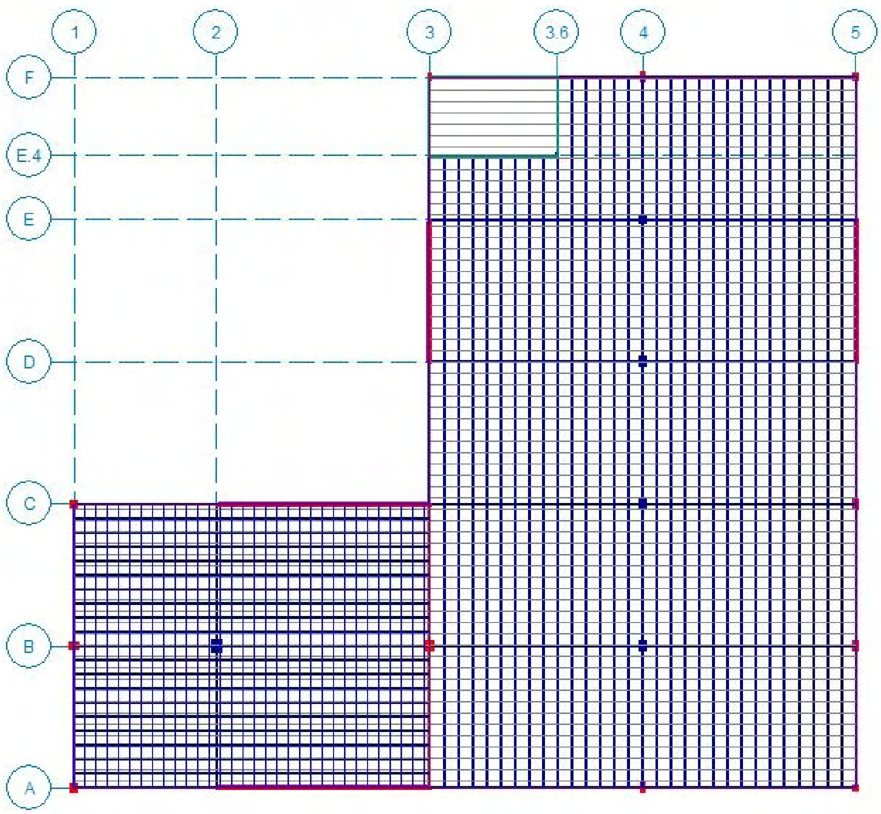
  Click and drag cursor over all framing between Grid Lines '1' and '3'.

You can change the display of the decks by clicking on the Plot options button, select the tab Points/Decks/Slabs.

Note: The Deck Default entry for each floor will be used on that floor in all areas where a 'local' deck is not defined. A 'local' deck/slab is and deck/slab area that is physically drawn in the model. 'Local' decks/slabs override the default deck in those areas.

If there is only one type of deck/slab on a floor level, setting the Deck Default to this deck/slab type is all that is required.

When Deck Assignments are displayed 'As Input' also "Include Default", your model should look like the image below:

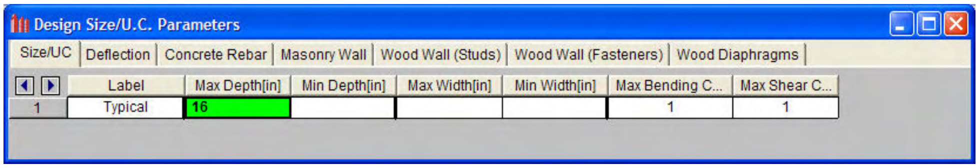


Note: The default deck will be replaced by any local deck that you add. If you have two decks on top of each other, the last deck added (or the deck on top) will govern.

Let's work with the member design.

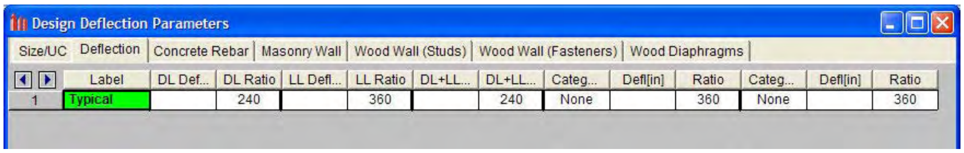
- Click on the "Design Rules" spreadsheet on the Data Entry Toolbar.

  Enter 16 inches as the Maximum Depth



	Label	Max Depth[in]	Min Depth[in]	Max Width[in]	Min Width[in]	Max Bending C...	Max Shear C...
1	Typical	16				1	1

We can also review the Design Rules for Deflection:



	Label	DL Def...	DL Ratio	LL Defl...	LL Ratio	DL+LL...	DL+LL...	Categ...	Defl[in]	Ratio	Categ...	Defl[in]	Ratio
1	Typical		240		360		240	None		360	None		360

Let's solve the model.

- Click on the Load Combination Spreadsheet on the Data Entry Toolbar to review the defaults.
- Solve the model and review the results.

  Why are some members not designed?

Let's modify some of the Design Parameters and members materials.

- Open the Design Rules Spreadsheet and delete the 16" Maximum Depth.
- Click on the Modify- Beams and select Wood, 24F-1.8E DF Balanced, Glulam _Western and makes sure to check the "Use" button.

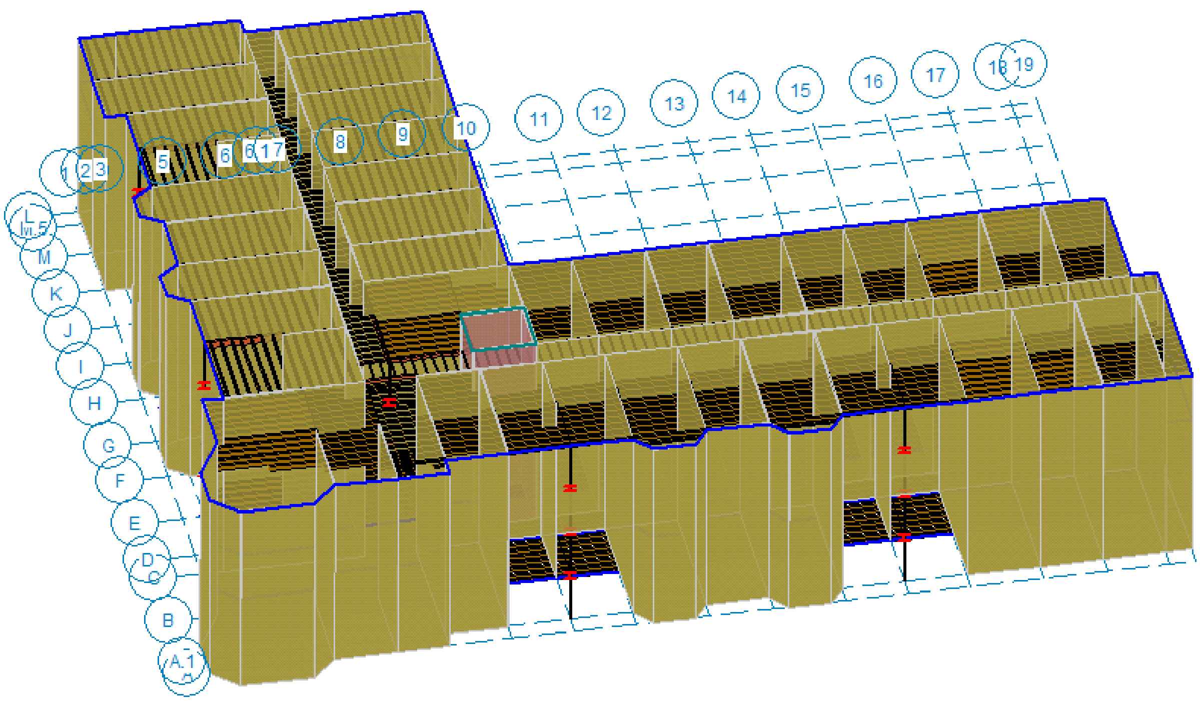
  Select all the Girders to change them to Glulam.
- Solve the model and review the results.

Notes:

What's the difference between Slab, Deck or Diaphragm edges?

Floor Loading & Attribution

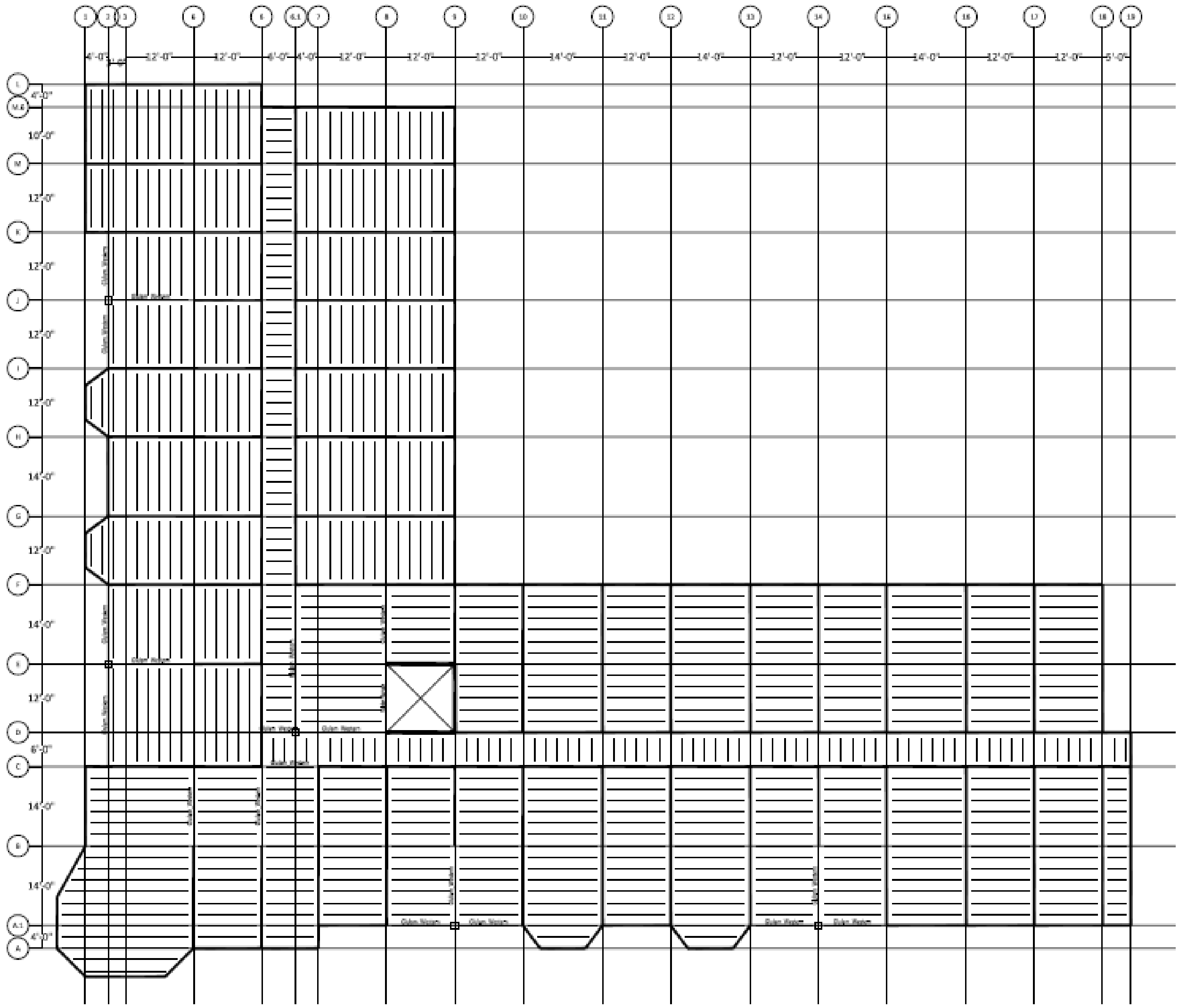
This model is intended to demonstrate the application of loads to your structure. This includes uniform and tapered Area Loads, Point Loads, and Line Loads. Other loading options such as Live Load Reduction and Parent/Child Floor loading will also be discussed, as well as attribution of loads to framing members.



Project Definition: Loading_Start.rfl

Four Stories – 40 feet (10,20,30,40)
2x wood Framing – Glulams
Plywood Deck
Floor Loading: Dead Load (Superimposed) – 25 psf
Live Load (Reducible) – 40 psf
Roof Loading: Dead Load (Superimposed) – 25 psf
Live Load (Reducible) – 20 psf

Framing Plan: Typical Floor



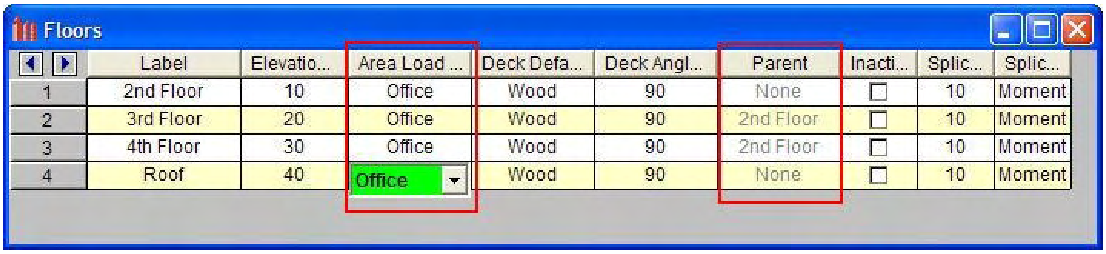
Modeling Procedure

We'll begin with a previously created model, Loading_Start.rfl.

Open the file 'Loading_Start.rfl' from the Training CD and save the model using the new file name 'Loading_Finished.rfl'.

We'll begin by reviewing the Default Area Loads on this model.

- Open the Floors spreadsheet on the Data Entry Toolbar.

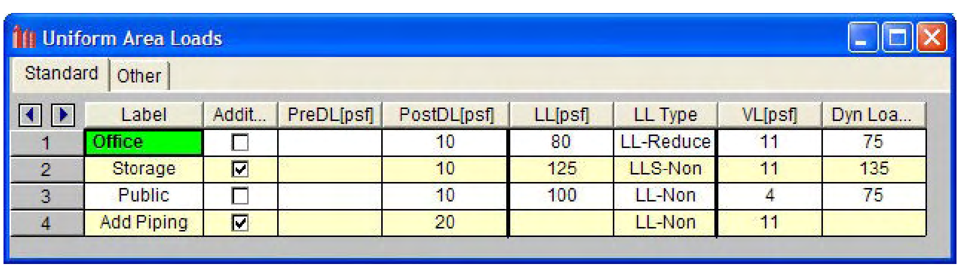

Floors

	Label	Elevatio...	Area Load ...	Deck Defa...	Deck Angl...	Parent	Inacti...	Splic...	Splic...
1	2nd Floor	10	Office	Wood	90	None	☐	10	Moment
2	3rd Floor	20	Office	Wood	90	2nd Floor	☐	10	Moment
3	4th Floor	30	Office	Wood	90	2nd Floor	☐	10	Moment
4	Roof	40	Office	Wood	90	None	☐	10	Moment

Each floor has an Area Load Default which is applied to the entire floor except where local Area loads are applied

We can also see that there is Parent/Child relationships defined in this model. When a floor is a child of another floor, it means that everything on the parent floor will be mimicked automatically on the Child floor. However, changes made to the child floor are not applied to the parent floor. You can create this relationship when you insert a floor:

- Open the Area Load Definitions spreadsheet on the Data Entry Toolbar.


Uniform Area Loads

Standard | Other

	Label	Addit...	PreDL[psf]	PostDL[psf]	LL[psf]	LL Type	VL[psf]	Dyn Loa...
1	Office	☐		10	80	LL-Reduce	11	75
2	Storage	☑		10	125	LLS-Non	11	135
3	Public	☐		10	100	LL-Non	4	75
4	Add Piping	☑		20		LL-Non	11	

The Area loads are defined within this spreadsheet and applied to the entire floor or applied to sections of the floor. Below is a list of the pre-defined Load Cases:

Load Case	Abbreviation	Description
Self Weight	SW	Column / Beam / Wall Self Weight - see Self Weight (Gravity Loads)
Dead Loads	PreDL	Pre-Composite Dead Load
	PostDL	Post-Composite Dead Load
Live Loads	LL-Non	Non-Reducible Live Load
	LL-Reduce	Reducible Live Load
	LLS-Non	Non-Reducible Live Load for garages, public assembly areas, live load magnitudes > 100psf
	LLS-Reduce	Reducible Live Load for garages, public assembly areas, live load magnitudes > 100psf
	RLL-Non	Non-Reducible Roof Live Load
	RLL-Reduce	Reducible Roof Live Load
	SL	Snow Load
	SLN	Non-Shedding Snow Load
	RL	Rain Load
Other Loads	OL1	Other Load 1
	OL2	Other Load 2
	OL3	Other Load 3
	OL4	Other Load 4
Dynamic	Dyn Mass	Dynamic Mass (for RISA-3D analysis)
Vibration	VL	Vibration Load

What is Dyn Mass? This represents the additional loads (above and beyond self weight) that must be included in their calculation of Seismic Weight.

The program does not include all applied loads as part of the seismic weight. The only applied loads that are included are the ones assigned to the Load Case called “Dyn Mass”. The super-imposed dead loads (DLPre or DLPost) are not included. The reason for this exclusion is due to the fact that the super-imposed dead load is often a very conservative number (accounting for unknowns in the construction process).

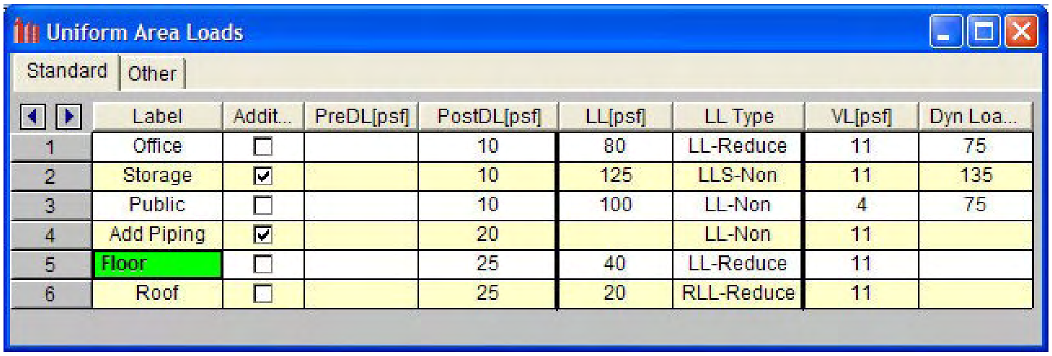
- In the “Area Load Definitions” spreadsheet- add the following Area Loads to the spreadsheet: (VL=11 psf, ignore Dyn Load)

  Floor DL = 25 psf

  LL= 40 psf (Reducible)

  Roof DL = 25 psf

  LL = 20 psf (Roof Live- Reducible)



	Label	Addit...	PreDL[psf]	PostDL[psf]	LL[psf]	LL Type	VL[psf]	Dyn Loa...
1	Office	☐		10	80	LL-Reduce	11	75
2	Storage	☑		10	125	LLS-Non	11	135
3	Public	☐		10	100	LL-Non	4	75
4	Add Piping	☑		20		LL-Non	11	
5	Floor	☐		25	40	LL-Reduce	11	
6	Roof	☐		25	20	RLL-Reduce	11	

- In the Floors spreadsheet- assign the Area Load to the floors

  2nd Floor = Floor

  3rd Floor = Floor

  4th Floor = Floor

  Roof = Roof

Let's start adding local area and point loads around the elevator and stair openings.

- Unselect the entire model using the Selection tools on the left side of the screen-

  Click on the Unselect the entire model button 

- Select the Public areas of the building using the Box and Line selection tools 

  Hallway areas:

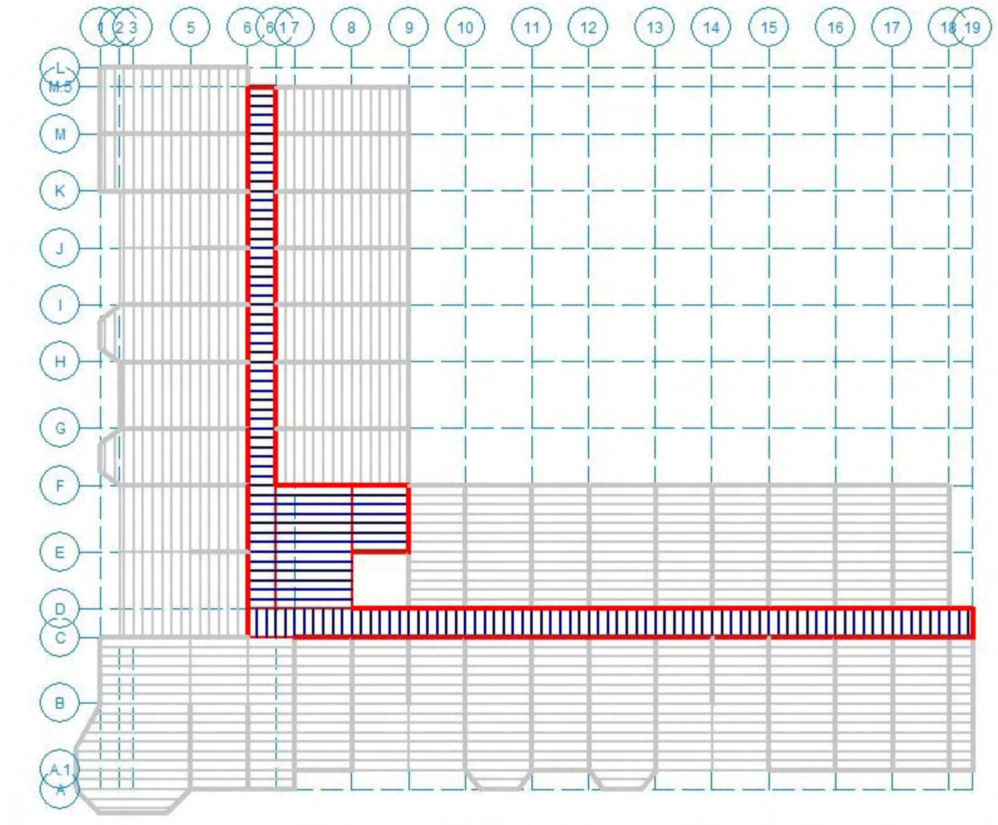
  '6' & '6.1'

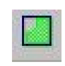
  'C' & 'D'

  Elevator and Stairwell areas:

  'F-6' & 'F-9'

  'C-6' & 'C-8'



- Draw a 'Public' Area Load on the '2nd Floor' by clicking on the Draw Area Load button 

  Select "Public" from the dropdown list of Uniform Area Loads

  Click within or box the Perimeter- Press Apply

  Click the mouse and drag a box around the entire selected area

Let's add Elevator point loads.

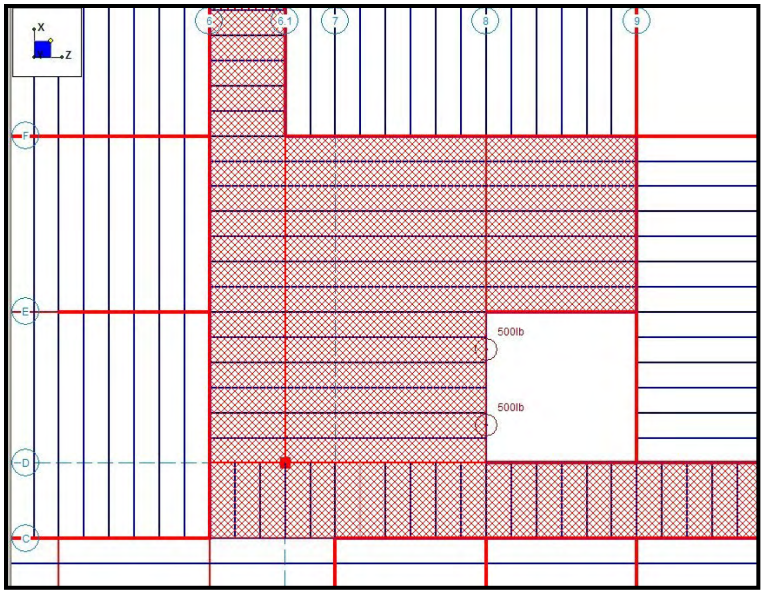
- Draw a Point Load on the '2nd Floor' in the Elevator shaft by clicking on the Assign Point Loads button 

  Enter 500 lb DL Point Load- Apply Load by Clicking/Boxing Points at the ¼ point of the Elevator Beam (53,41) & (53,47).

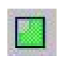
> **Note:** The '3rd Floor' and "'4th Floor' are a 'Child' of '2nd Floor.' Therefore, the load applied at the '2nd Floor' will automatically be applied to the '3rd Floor' and '4th Floor' as well.

When Uniform Area Loads are displayed 'As Input' for the Load Category 'DL Post Comp', the '2nd Floor', '3rd Floor', and '4th Floor' should look like the image below:

(Click on the Plot Options button on the RISA Toolbar at the top left side of the screen- Loads Tab to setup the display).

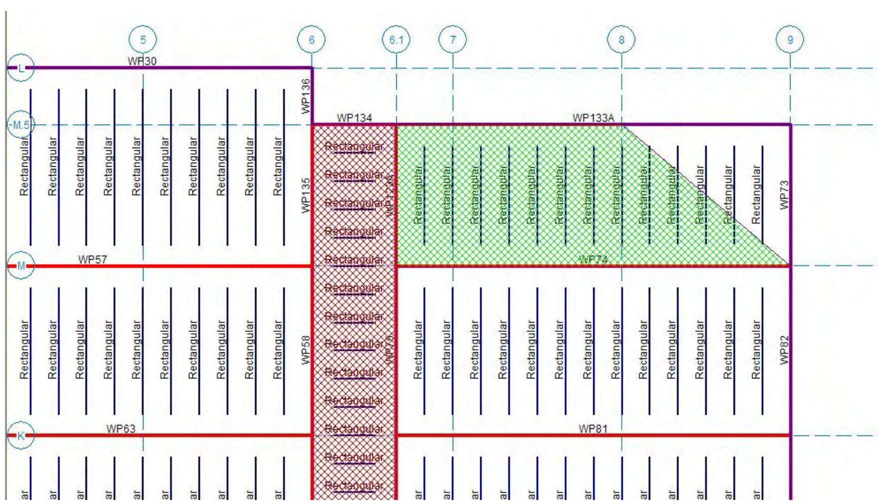


Now we will add Storage Areas to the 3rd Floor.

- Click on the Draw Area Load Icon 
- Draw a Uniform 'Storage' Area Load on the '3rd Floor'- select Point to Point Draw of the Area with perimeter joints at the following Grid Intersections:

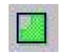
M – 6.1	M.5 – 8
M.5- 6.1	M -9

When Uniform Area Loads are displayed 'As Input' and 'Include Default' for the Load Category 'DL PostComp', the '3rd Floor' should look like the image below:



Note: The '3rd Floor' is a 'Child' of '2nd Floor.' Loads applied to the 'Child' floor are not automatically applied to the 'Parent' floor. Therefore, the load applied at the '3rd Floor' will only be applied at that level.

Now we will add Storage Areas to the Roof Floor.

- Click on the Draw Area Load Icon 
- Draw a Point to Point Draw of the Area 'Storage' on the 'Roof' with between the following points:

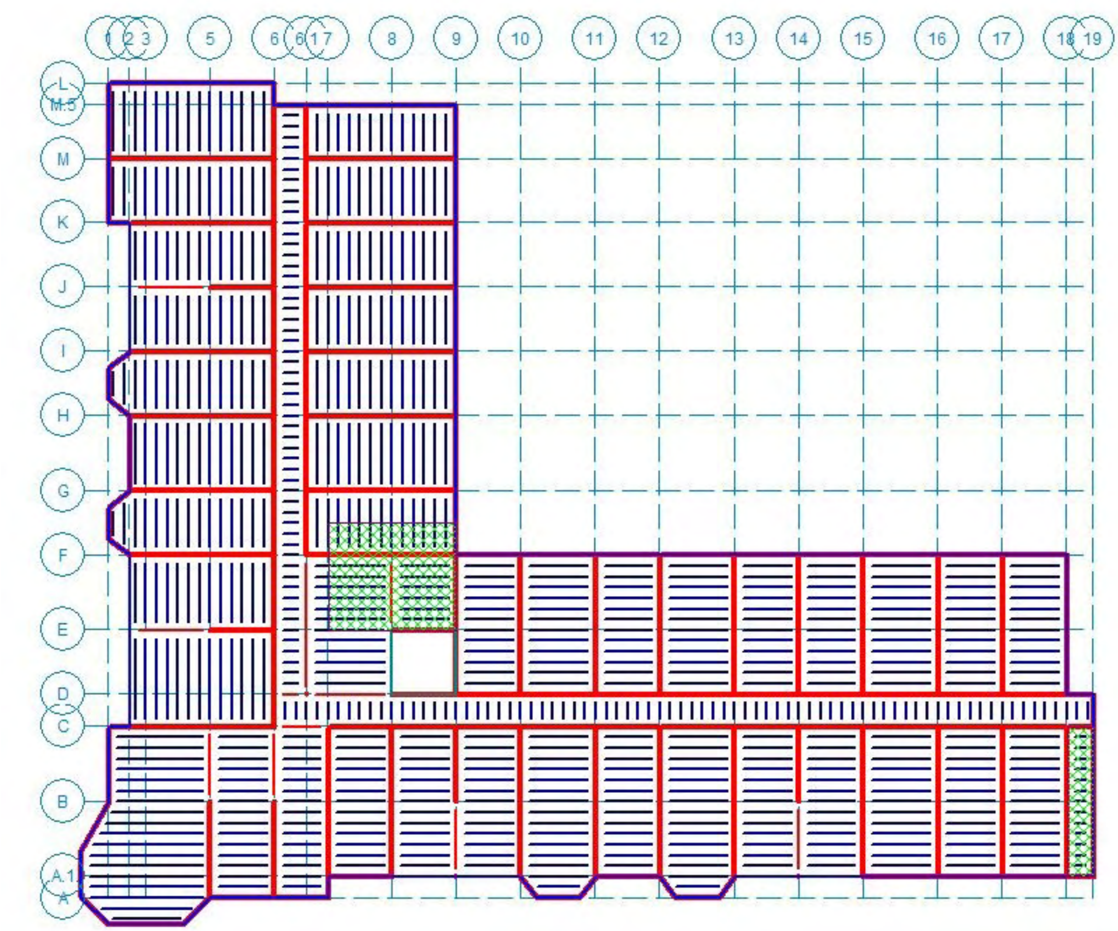
  (Be sure the 'Snap To' options are enabled in order to snap to the ½ points on the members.)

41, 70	65, 50
65, 70	41, 50

- Draw a Polygon Around the Perimeter 'Storage' Area Load on the 'Roof' with between the following grids:

C-18	C-19
A-18	A-19

When Uniform Area Loads are displayed 'As Input' and 'Include Default' for the Load Category 'DL PostComp', the 'Roof should look like the image below:



Now we will add Piping to the Roof.

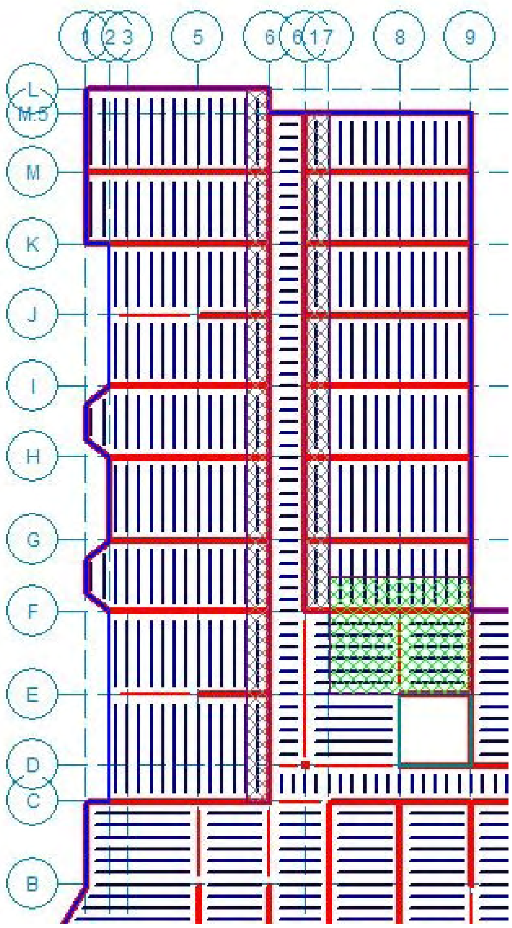
- Draw rectangular 'Add Piping' Area Loads on the 'Roof' with perimeter joints at the following coordinates.

NOTE: Piping Loads are additive. See the “Additive” column in the Area Load Definitions spreadsheet.

Be sure the ‘Snap To’ options are enabled in order to snap to the ¼ points on the members

1st Area Load	2nd Area Load
27,152	37,148
31,152	41,148
31,32	37,64
27,32	41,64

When Uniform Area Loads are displayed ‘As Input’ and ‘Include Default’ for the Load Category ‘DL Post Comp’, the ‘Roof’ should look like the image below:



Now we will add Snow Loads to the Roof.

- Draw Tapered Area Loads (‘SL’ Category) the Peak Magnitude should be at the exterior of the building and the snow drift magnitudes are as follows:

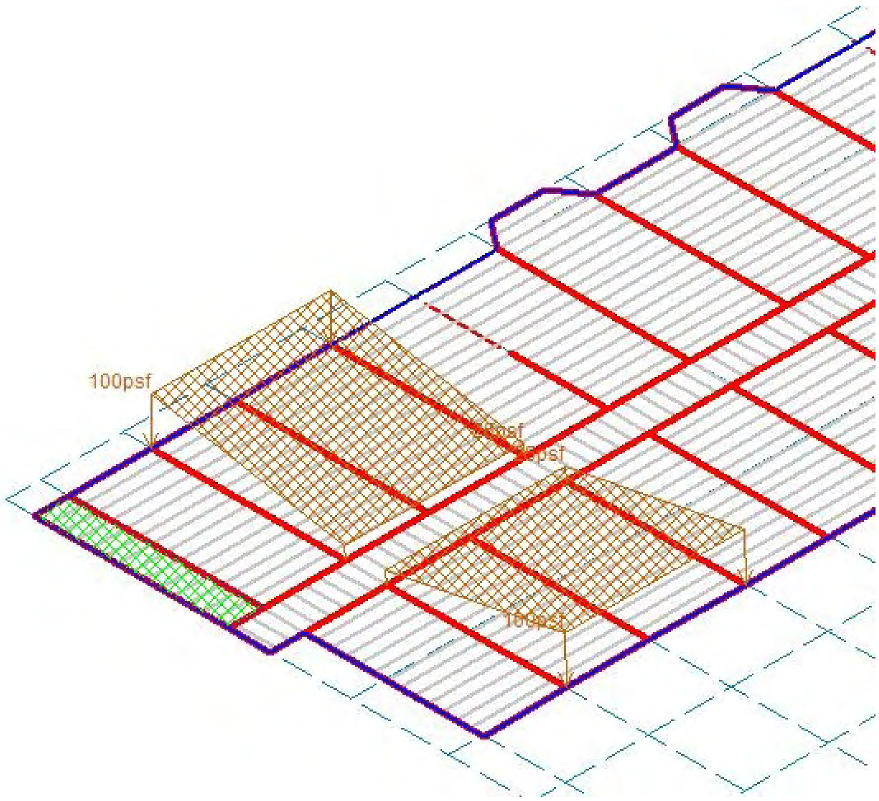
  Base Magnitude – 25 psf
  Peak Magnitude – 100 psf

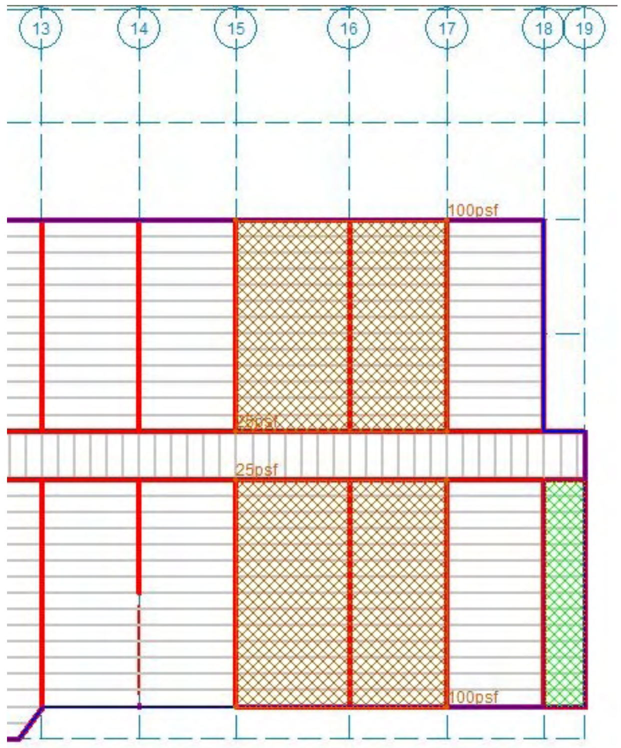
  In the area between Grid Intersections:

  ‘D-15’ – ‘D- 17’ and ‘F-17’ – ‘F- 15’

‘C-15’ – ‘C-17’’ and ‘A.1-17’ – ‘A.1-17 –‘A.1-15’

When loads are displayed ‘By Category’ using ‘SL Snow Load’ as the criteria the ‘Roof’ should look like the image below:





Note: When drawing tapered area loads the first two clicks define the base magnitude and second two clicks define the peak magnitude. Since it is easier to first click the edge beams, the base and peak magnitudes have been flipped. For triangular tapered snow loads, you cannot define the first two points at the same location. The points need to be slightly off. Any other points can be the same with no problems. This is because the program doesn't yet allow for the first two clicks, defining the base magnitude, to use the same point.

Now we will add loading from a Roof Top Unit to the model.

(Make sure you have Universal Grids turned on, by clicking on the Icon.)

- Click on the Point load button 
- Place (3) 200 lb DL Point Loads in the bay bounded by Grid Intersections 'M-3' and 'L-5', at the following locations:

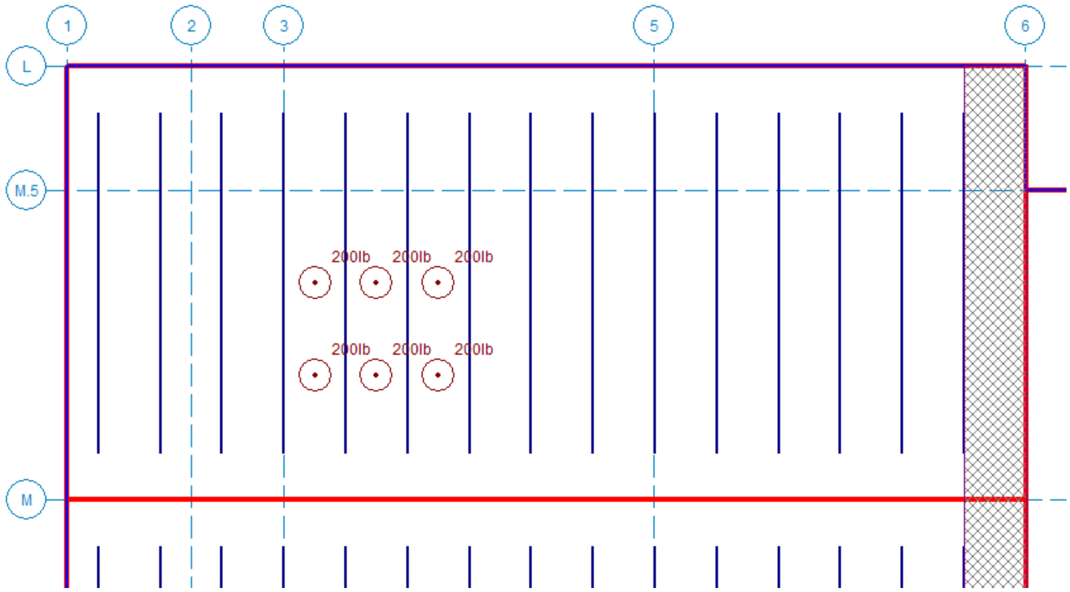
  (8, 145, 0)

  (10, 145, 0)

  (12, 145, 0)

- Now, in the same bay, place (3) 200 lbs DL Point Loads 3 feet 'below' the point loads that were just added.

When loads are displayed 'By Category' using 'DL PostComp' as the criteria, your model should look like the image below:



> **Note:** If a load is to be taken into account when the Seismic Mass of the structure is calculated, that load must have a 'Dyn Mass' component equal to the mass to be included in the calculation. Superimposed Dead Load (DLPre and DLPost) are not automatically included in the Seismic Mass of the structure.
>
> For example, if the Roof Top Unit applied above is to be considered in the Seismic Mass calculation, a 200 lbs 'Dyn Mass' component must be applied in addition to the 200 lbs 'DL Post' loading.

Now we will add loading to simulate the weight of the Exterior (Cladding) Wall on the building.

The curtain wall has a weight of 5 psf and the floor to floor height is 10 feet. Therefore, the equivalent load at each floor level based on tributary area is 50 plf at the center floors and 25 plf at the 'Roof'.

- Viewing the 2nd Floor, unselect the entire model
- Select just the perimeter beams at each floor level using the Select based on Other Criteria button Box,

    Select the "Walls" tab, and check Perimeter Walls Only- Click Apply

    Select the "Beams" tab, and check Perimeter Beams Only- Click Apply

- Assign 'DL Post' Line Loads by clicking on the Distributed Loads button , apply following magnitude making sure to Apply to All Selected Members/Walls:

    2nd Floor = 50 plf

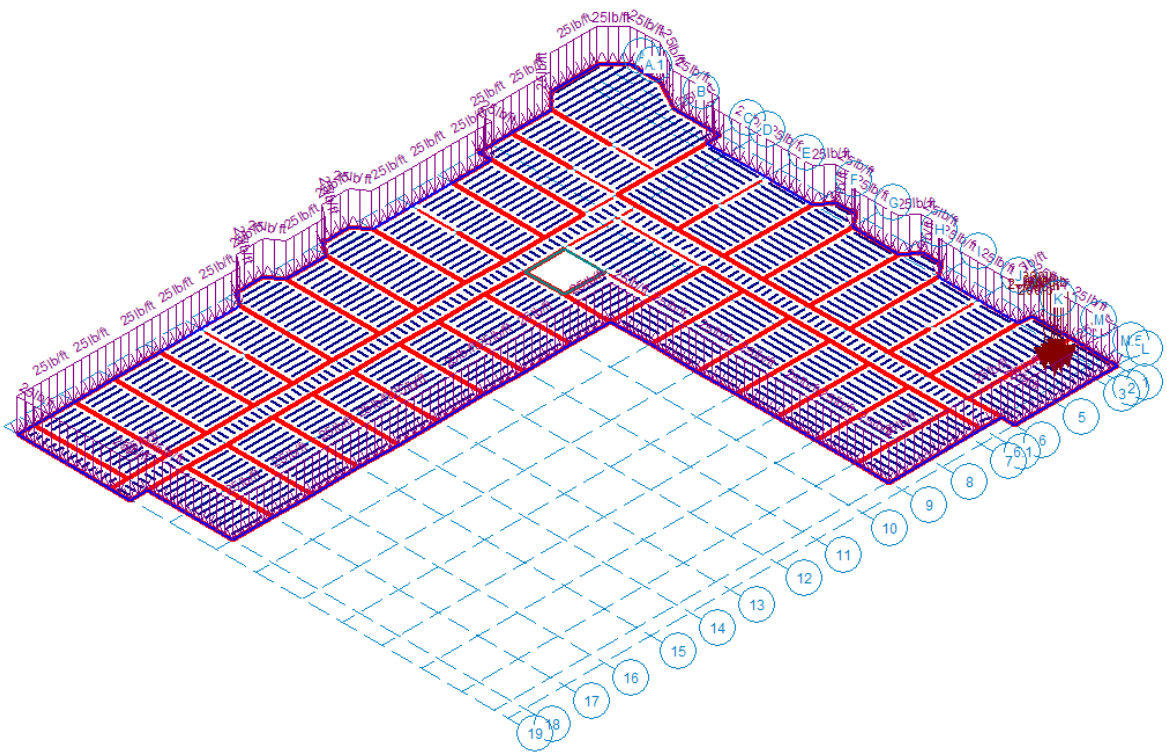
View the 3rd Floor and 4th Floor, and you'll see that the loads applied to the 2nd Floor is also applied to the 3rd Floor because of the Parent Child relationship.

- Following the steps above, add distributed loads to the next two floors:

  Roof = 25 plf

> **Note:** See the note above concerning Seismic Mass. A 'Dyn Mass' equal to the loadings listed above should be included in these line loads to account for the weight of the curtain wall in the calculation of the building's seismic mass.

When loads are displayed 'By Category' using 'DL PostComp' as the criteria, the 'Roof' should look like the image below:



Now we will build load combinations and solve the model.

- Open the Load Combinations Spreadsheet on the Data Entry Toolbar.
- Click on the Load Combination Generator button

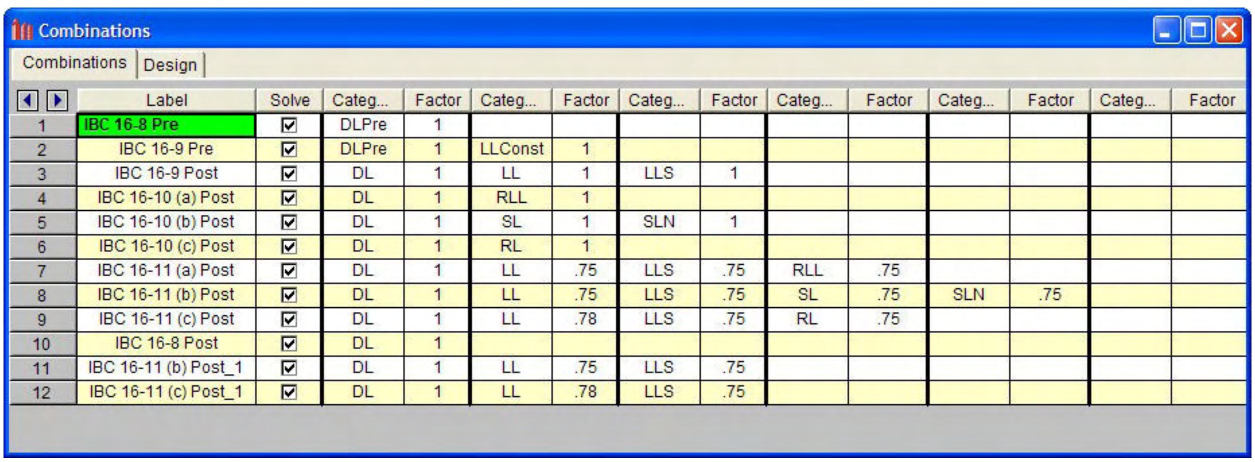
  Select the IBC 2006 ASD building code include Roof Live Load (RLL) and Snow Load (SL).

  Pre-Composite combinations are not required in this model- uncheck that option.

The Load Combination spreadsheet uses Load Categories which are described below:

Load Category	Includes Addition of Load Cases	Sloped Members Load Direction
DLPre	SW + Deck SW + Const DL + PreDL	Y
LLConst	Const LL	PY
DLConst	Const DL	Y
DL	SW + Deck SW + PreDL + PostDL	Y
LL	LL-Non + LL-Reduce	PY
LLS	LLS-Non + LLS-Reduce	PY
RLL	RLL-Non + RLL-Reduce	PY
SL	SL	PY
SLN	SLN	PY
RL	RL	PY
OL1	OL1	Y,PY,y
OL2	OL2	Y,PY,y
OL3	OL3	Y,PY,y
OL4	OL4	Y,PY,y

The Load Combination spreadsheet should now look like this:



	Label	Solve	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor	Categ...	Factor
1	IBC 16-8 Pre	☑	DLPre	1										
2	IBC 16-9 Pre	☑	DLPre	1	LLConst	1								
3	IBC 16-9 Post	☑	DL	1	LL	1	LLS	1						
4	IBC 16-10 (a) Post	☑	DL	1	RLL	1								
5	IBC 16-10 (b) Post	☑	DL	1	SL	1	SLN	1						
6	IBC 16-10 (c) Post	☑	DL	1	RL	1								
7	IBC 16-11 (a) Post	☑	DL	1	LL	.75	LLS	.75	RLL	.75				
8	IBC 16-11 (b) Post	☑	DL	1	LL	.75	LLS	.75	SL	.75	SLN	.75		
9	IBC 16-11 (c) Post	☑	DL	1	LL	.78	LLS	.75	RL	.75				
10	IBC 16-8 Post	☑	DL	1										
11	IBC 16-11 (b) Post_1	☑	DL	1	LL	.75	LLS	.75						
12	IBC 16-11 (c) Post_1	☑	DL	1	LL	.78	LLS	.75						

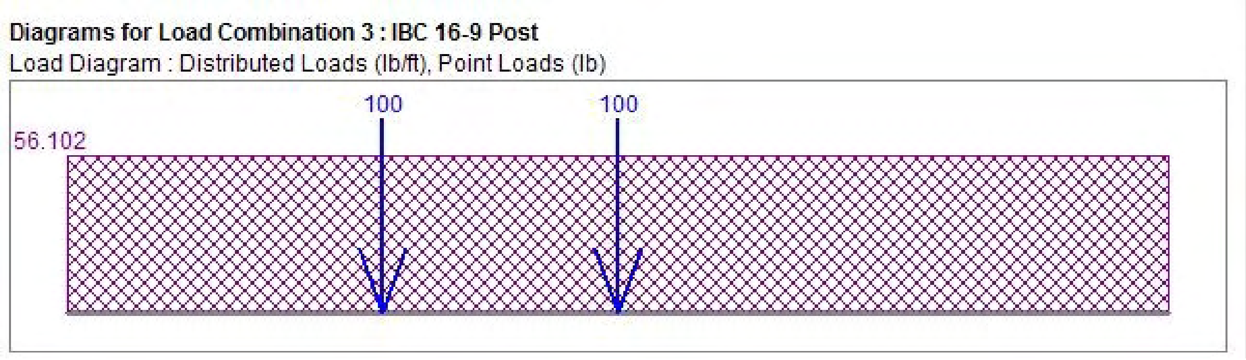
- Solve the model and review the results.

Now we will take a closer look at the load attribution results.

There are a few ways to review results: spreadsheet results, graphical display of results and member detail reports. For load attribution results we will look at the member detail reports so that we can examine the force diagrams in more detail.

- Open the member detail report for the vertical joist along Grid Line ‘3’ between Grid Lines ‘M’ & ‘M.5’ on the ‘Roof’. Choose ‘IBC 16-8 Post’ from the LC/Cat drop down list at the top of the dialog box.

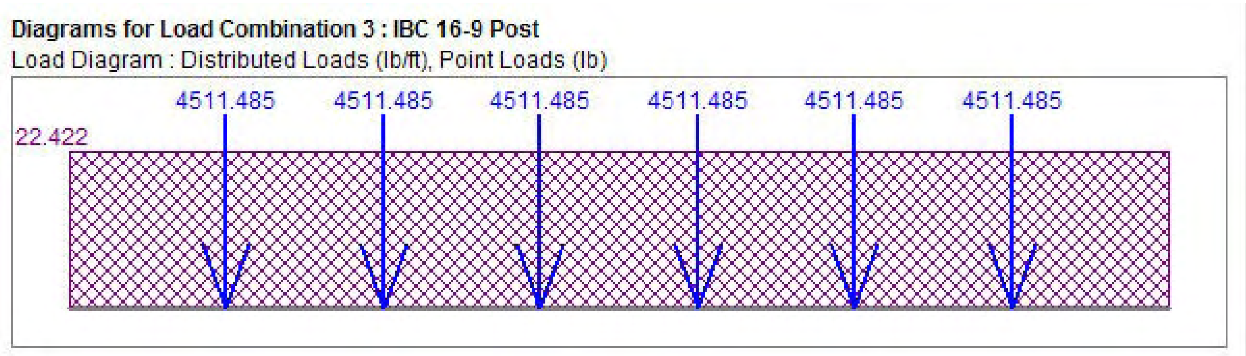
Your screen should look like the image below:



Note: Notice how the loading diagram shows two point loads. The program automatically distributed the point loads we applied for the roof top unit to the framing members (even though the point loads were applied on the deck – not the member directly).

- Open the member detail report for the horizontal girder along Grid Line '8' between Grid Lines 'E' & 'F' on the '2nd Floor'. Choose 'IBC 16-8 Post' from the LC/Cat drop down list at the top of the dialog box.

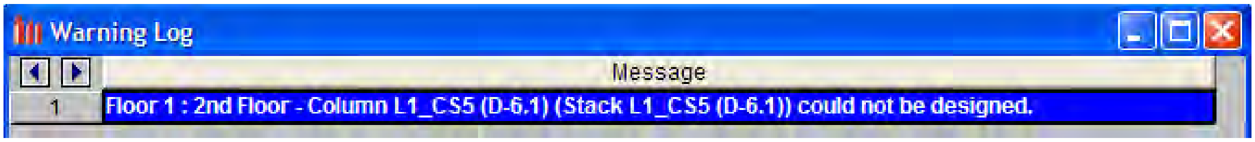
Your screen should look like the image below:



Note: Notice how the loading diagram shows the six point loads from the joists. The program automatically distributes the end forces from the joists to the girder and also displays their magnitude in this loading diagram.

Let's address the failing column in this model.

There is a Warning message that is showing the Column D- 6.1 on the '2nd Floor' cannot be designed.



- Open the Column Results spreadsheet on the Results toolbar. This shows us that only the 2nd Floor could not be designed.
- Open the Column Forces Spreadsheet to review the loads on this column.
- Double Click on this column to open the Column Editor.

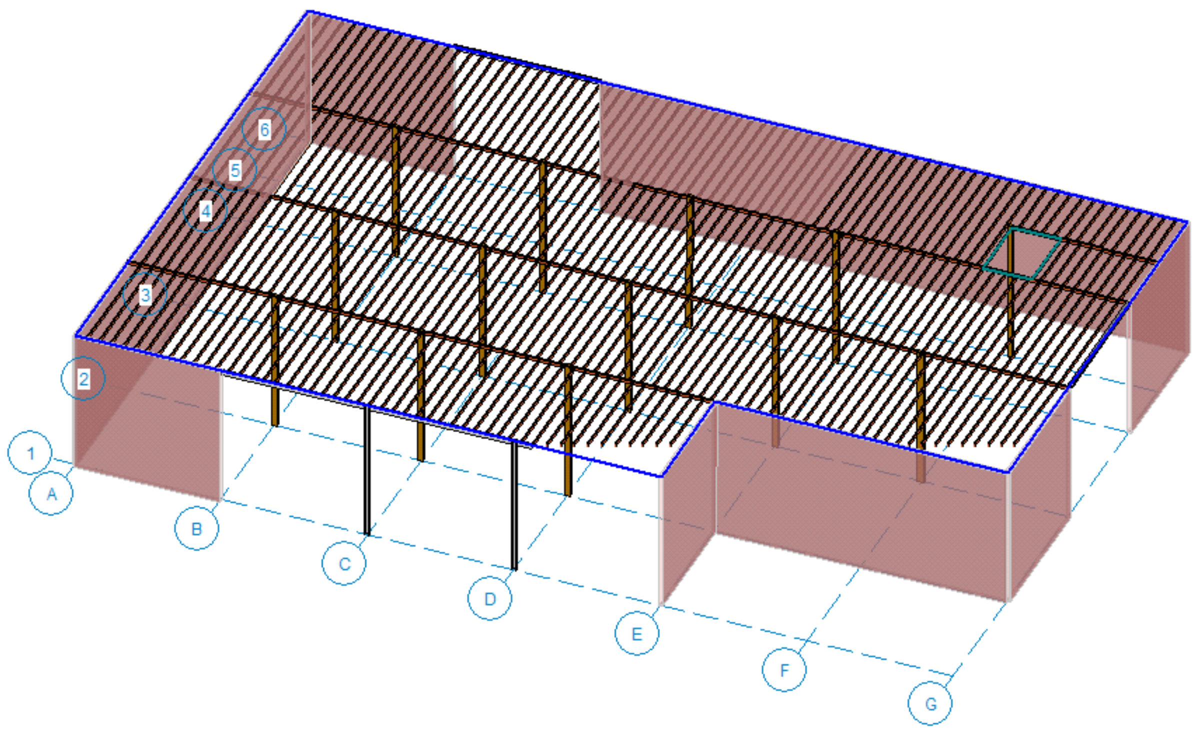
  Select Shape -Rectangular Triple- Check the “Use” button.

  Select “Apply Entries to All Lifts” -Click Apply & Close.
- Solve the model and review the results.

Notes:

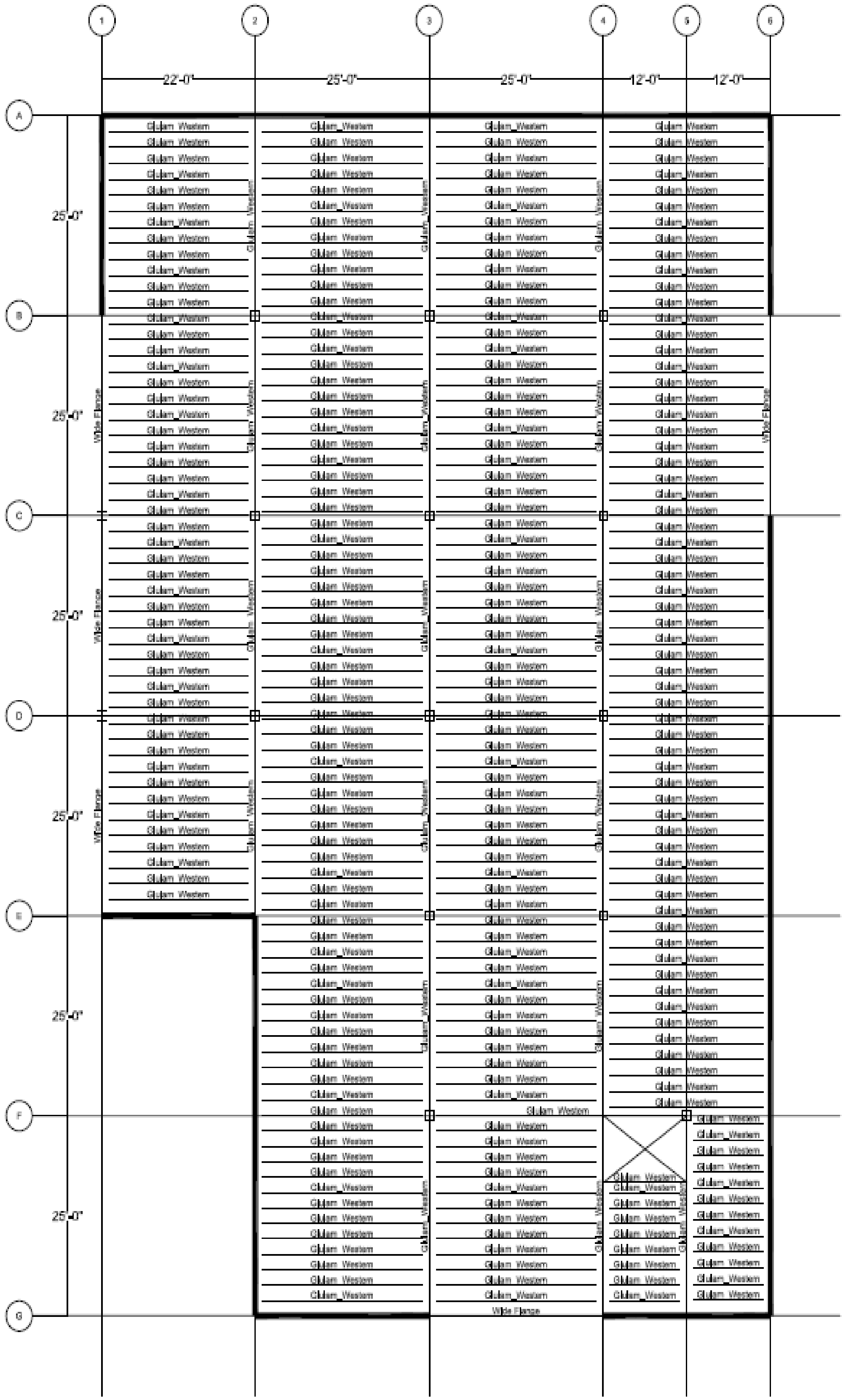
Masonry Warehouse

This model is intended to demonstrate modeling with multiple materials, masonry, wood framing and steel beams.



Project Definition: Masonry_Warehouse.rfl

Single Story (Roof) – 25 feet
Plywood Deck
Dead Load (Superimposed) – 10 psf
Base Snow Load (Controls) – 40 psf
Glulam Wood framing
Masonry Exterior walls with Steel columns and beams
Design Codes: Wood -NDS 2005
Masonry- MSJC 05/IBC 06 ASD
Steel- AISC 13th ASD


1
2
3
4
5
6
A
B
C
D
E
F
G
22'-0"
25'-0"
25'-0"
12'-0"
12'-0"
25'-0"
Glulam_Western
Wide Flange

Modeling Procedure

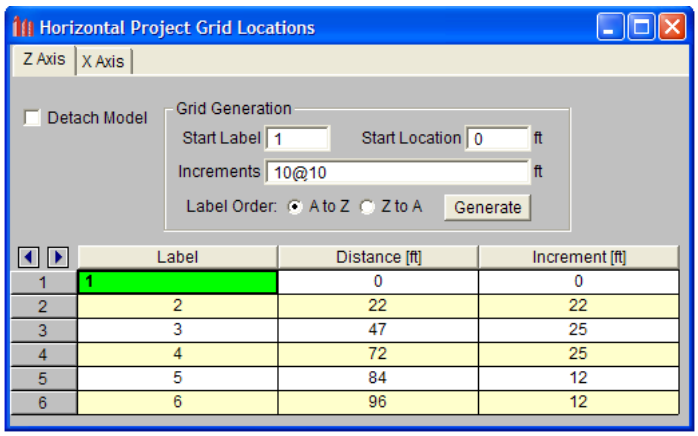
We'll begin by creating a new Floor and generating a Project Grid.

- Start a new model by clicking on the Create New Floor Plan button 
- Create an Original Floor with an elevation of 25 feet, Area Load- Office, Deck Default- Wood, Deck Angle- 90

Now we will define the Project Grid.

- On the Z Axis – Horizontal Tab of the Project Grid Dialog, enter '5@25' for the Increments, and click 'Generate'.

  Modify the Increments to match below:



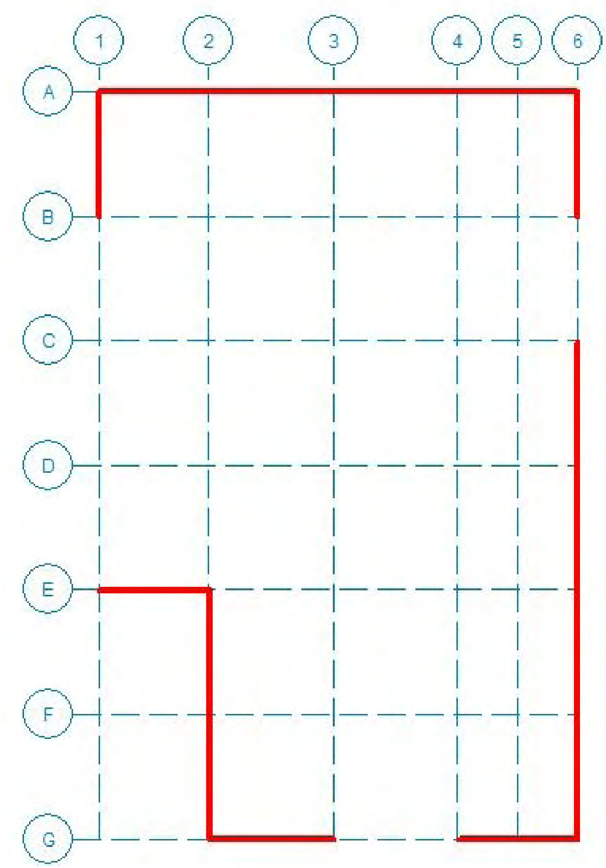
- On the X Axis- Vertical Tab, set the Start Label to 'G' and enter '6@25' for the Increments, select 'Z to A' and click 'Generate'.

We will now begin modeling the exterior walls.

- Draw a 'Lateral' Wall Masonry along the following Grid Intersections:

A-1 & B-1	E-1 & E-2	G-4 & G-6
A-1 & A-6	E-2 & G-2	C-6 & G-6
A-6 & B-6	G-2 & G-3	

Your model should now look like the image below:



- Draw 'Lateral' Columns (Hot Rolled, A500 Gr.46, Tube) between the following column location

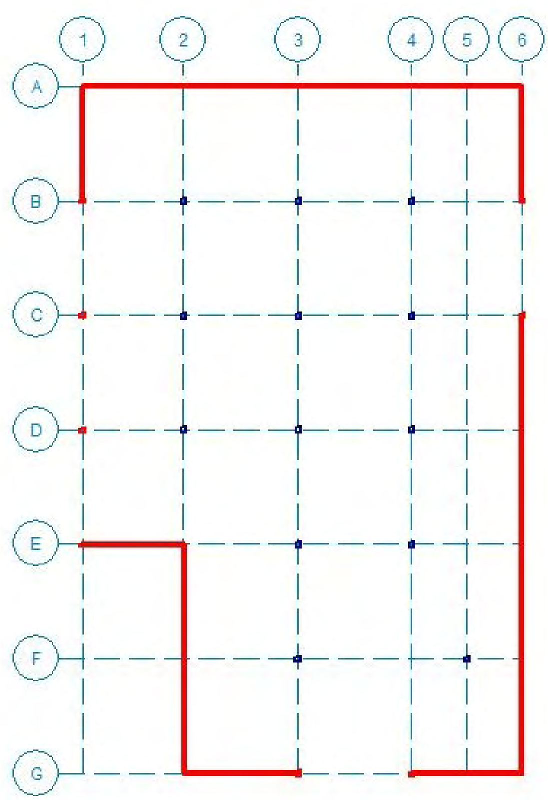
  D-1

  C-1

- Draw 'Gravity' Columns (Hot Rolled, A500 Gr.46, Tube) between the following column locations

B-2	D-3	D-4
C-2	E-3	E-4
D-2	F-3	F-5
B-3	B-4	
C-3	C-4	

Your model should now look like the image below:



We will now model the exterior Beams in the main structure, as well as the internal wood framing.

- Draw 'Lateral' Beams along the perimeter of the building.

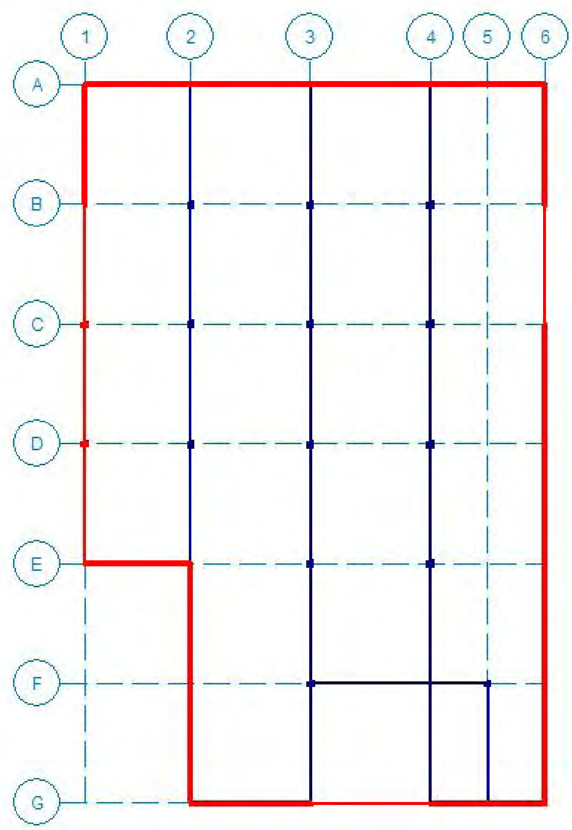
  Select Hot Rolled Steel, A992, Wide Flange, Pinned-Pinned and Non-Composite at the following Grid Intersections:

E-1 & D-1	G-3 & G-4
D-1 & C-1	C-6 & B-6
C-1 & B-1	

- Draw Gravity Beams along the following grid intersections:

  Select Wood beam, Material 24F-1.8E DF Balanced, shape Glulam_Western, Gravity and Pinned-Pinned at the following Grid Intersections:

F-3 & F-5	All Along Grid Line 2
F-5 & G-5	All Along Grid Line 3
F-3 & F-5	All Along Grid Line 4



We'll frame one opening in the building just below grid line 'F':

- Draw one additional 'Gravity' beam with the same properties as above from:

  (Be sure the 'Snap To' options are enabled in order to snap to the 1/3 points on the members.)

  (72, 16.6667, 0) to (84, 16.6667,0)

We'll finish framing this building with Infill framing and add the Slab:

- Draw the 'Gravity' infill beams over the entire building Horizontally except in the opening.

  Select Wood beam, Material 24F-1.8EDF Balanced, shape Glulam_Western,

  Horizontal, 2' O.C.

  (You may need to use the "Draw Polygon" feature to draw around the opening).

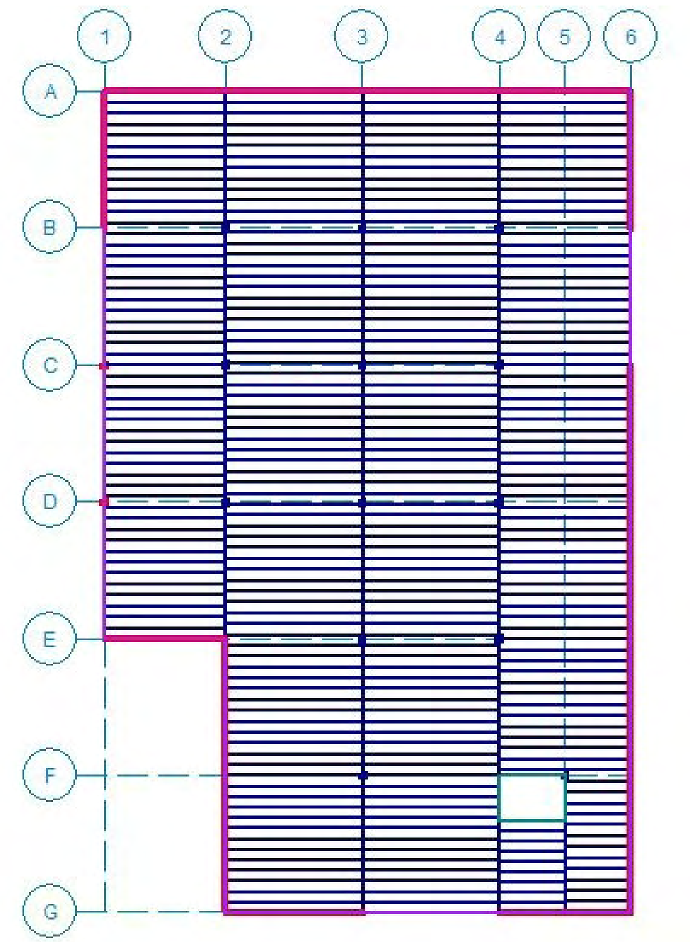
- Create a Slab around the entire structure by clicking on the "Create or modify edge perimeters" button.

Select Create a Slab perimeter with Edge Overhang Distance of 0”- Rigid.

- Create an opening for the shaft by clicking on the “Create or modify edge perimeters” button.

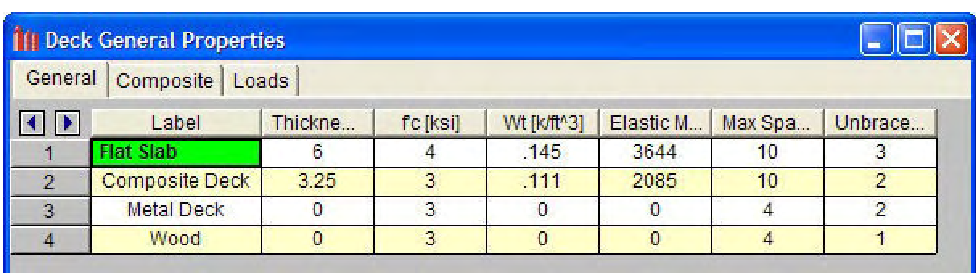
  Select Create an Opening with Edge Overhang Distance of 0”

Your model should now look like the image below:



Let’s assign the Deck Properties and Area Loads:

- Open the Deck Definitions spreadsheet on the Data Entry Toolbar- and review the Wood deck:


Deck General Properties

General | Composite | Loads

	Label	Thickne...	f'c [ksi]	Wt [k/ft^3]	Elastic M...	Max Spa...	Unbrace...
1	Flat Slab	6	4	.145	3644	10	3
2	Composite Deck	3.25	3	.111	2085	10	2
3	Metal Deck	0	3	0	0	4	2
4	Wood	0	3	0	0	4	1

Deck Loads

General | Composite | Loads

	Label	Two Way	Self Wt [...	Const D...	Const ...
1	Flat Slab	☐	.075	0	.02
2	Composite Deck	☐	.05	0	.02
3	Metal Deck	☐	.003	0	.02
4	Wood	☐	.003	0	.02

- Open the Area Load Definitions spreadsheet on the Data Entry Toolbar.

  Create a new area load called ‘Roof’

  DL = .010 ksf

  LL = .040 ksf (Reducible)

Uniform Area Loads

Standard | Other

	Label	Addit...	PreDL[ksf]	PostDL[ksf]	LL[ksf]	LL Type	VL[ksf]	Dyn Loa...
1	Office	☐		.01	.08	LL-Reduce	.011	.075
2	Storage	☐		.01	.125	LLS-Non	.011	.175
3	Public	☐		.01	.1	LL-Non	.004	.075
4	Add Piping	☑		.02		LL-Non	.011	
5	Roof	☐		.01	.04	LL-Non	.011	.03

- Open the Floors spreadsheet on the Data Entry Toolbar.

  Select the Area Load ‘Roof’, Deck Default ‘Plywood’ and Deck Angle ‘90’

Floors

	Label	Elevatio...	Area Load ...	Deck Defa...	Deck Angl...	Parent	Inacti...	Splic...	Splic...
1	Floor Plan 1	25	Office	Wood	90	None	☐	25	Moment

Now we will build load combinations and solve the model.

- Open the Load Combination Spreadsheet

  Generate Load Combinations based on the IBC 2006 ASD building code which include Roof Live Load (RLL).

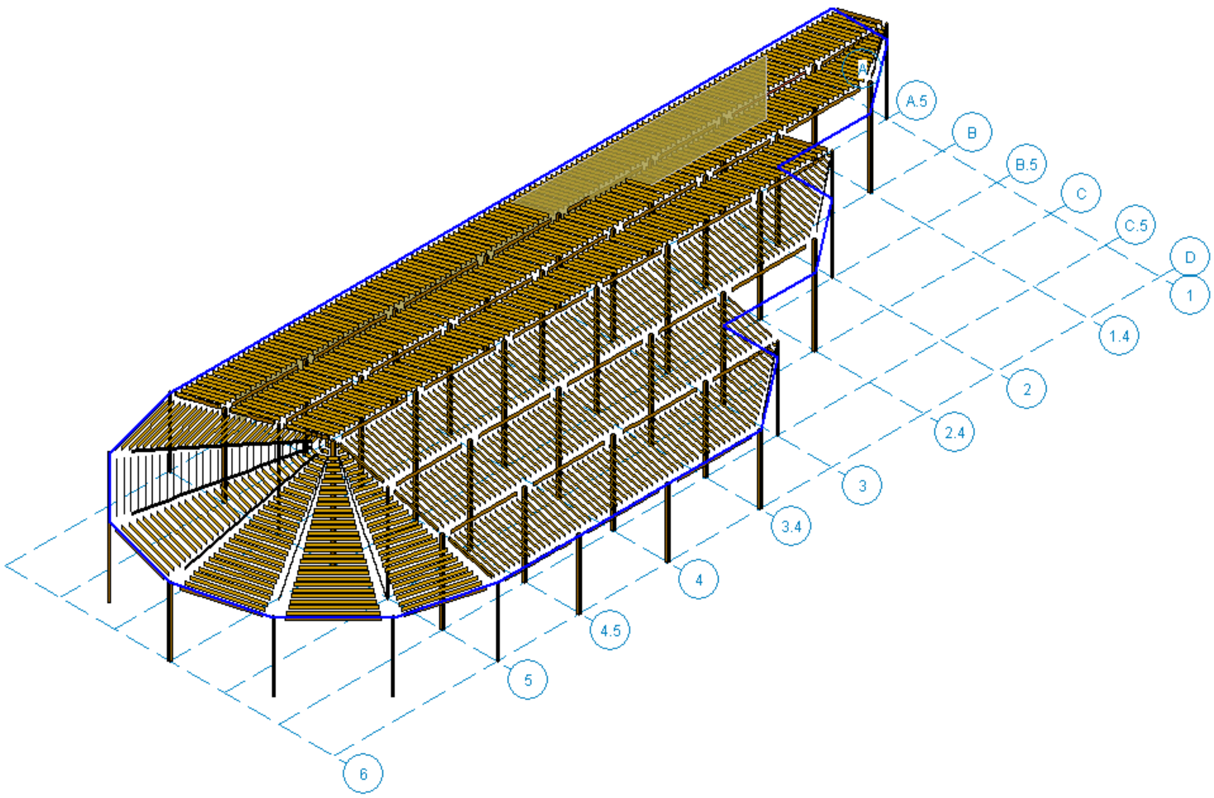
  Pre-Composite combinations are not required in this model.

Solve the model and review the results.

Notes:

Wood Sloping Roof

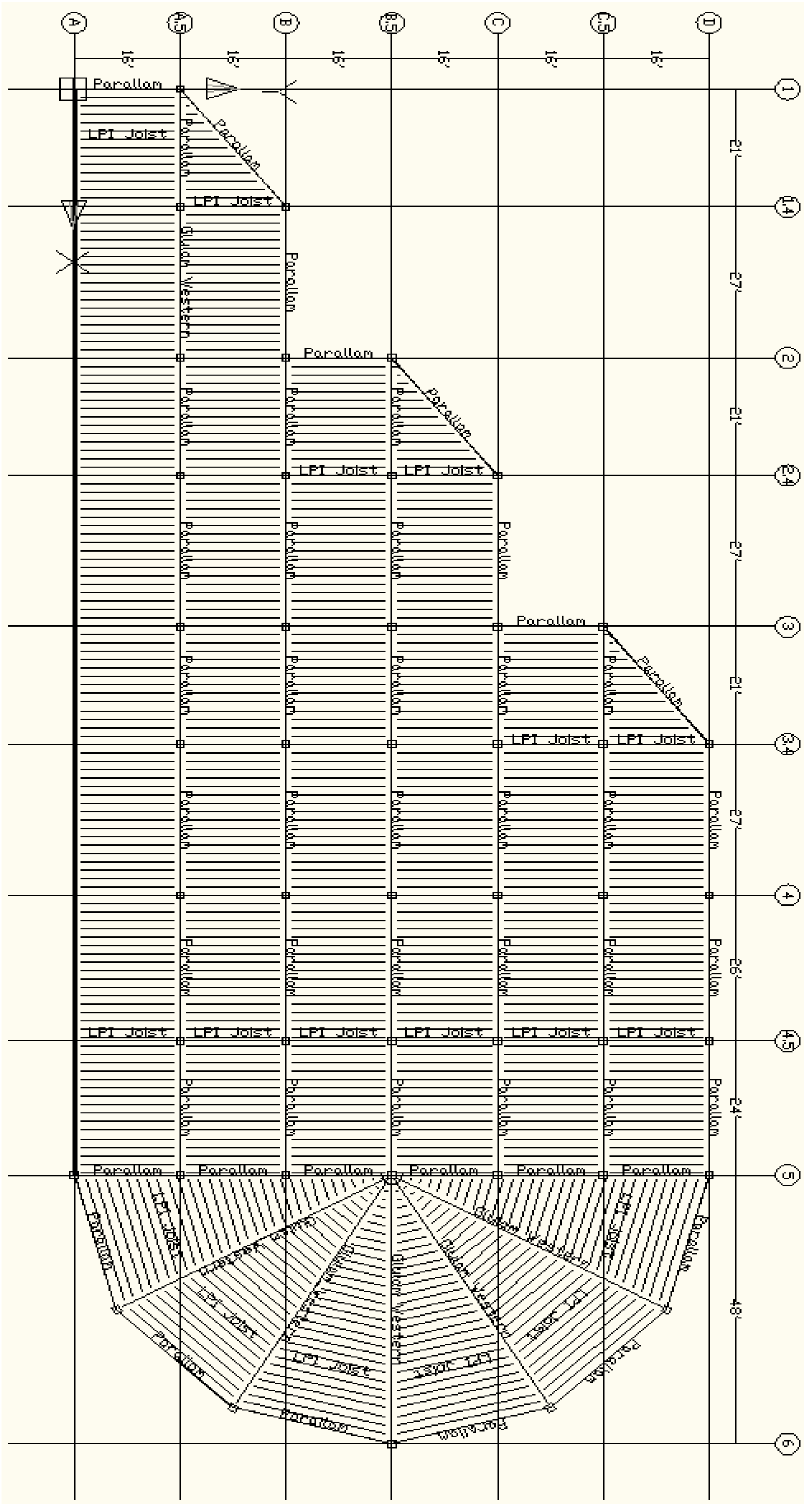
This model is intended to demonstrate various procedures and techniques associated with modeling framing systems utilizing Wood Joists, Glulams and Parallams. Other topics pertinent to roof systems such as sloping roofs, parapets and snow drifts will also be discussed.



Project Definition: Sloping_Roof.rfl

Single Story – 12 feet
Wood Beams - PSL_Parallam_2.0E_2900F
Glulam 16F-V2_SF/SP
LPI Joists
Wood Columns – Rectangular Doubles/Triples
Walls – 2x6 Wood, DF Larch, Design Rule: Typical

Framing Plan:



Modeling Procedure

Let's create a floor and generate a project grid.

- Create a new floor, labeled "Roof," with an elevation of 12 feet.

  Set the Deck Default to Wood, set the angle to 0, ignore the Default Area Load.

Now we will define the Project Grid.

- On the "Z Axis" tab of the Project Grid dialog [Project Grid],

  Enter "5@48" for the increments,

  Click [Generate].

  Insert a new Project Grid Line, labeled "1.4," with an increment of 21 after Grid Line "1."

  Insert new Project Grid Lines, labeled "2.4" and "3.4," with increments of 21 feet after Grid Lines "2" and "3," respectively.

  Insert a new Project Grid Line, labeled "4.5," with an increment of "26" after Grid Line "4".

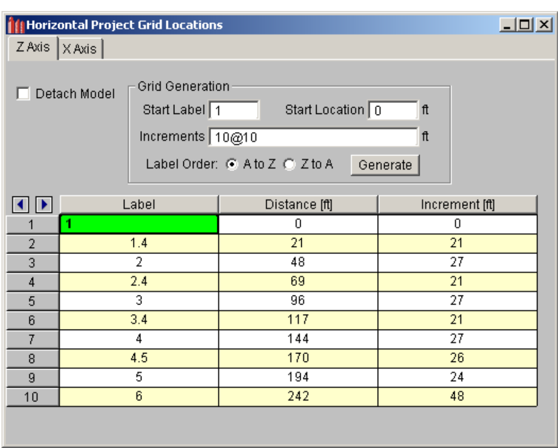
  Change the increment for Grid Line "5" to 24 feet.

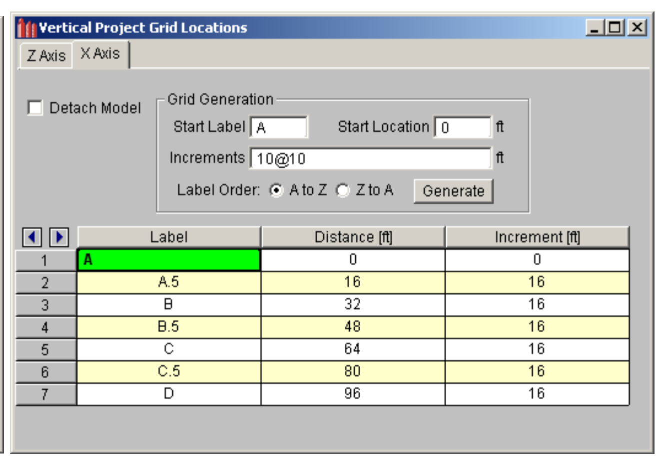
- On the "X Axis" tab, set the Start Label to "A" and enter "6@16" for the increments,

  Click [Generate].

- Rename the grid lines beginning with Grid Line "B" as follows:

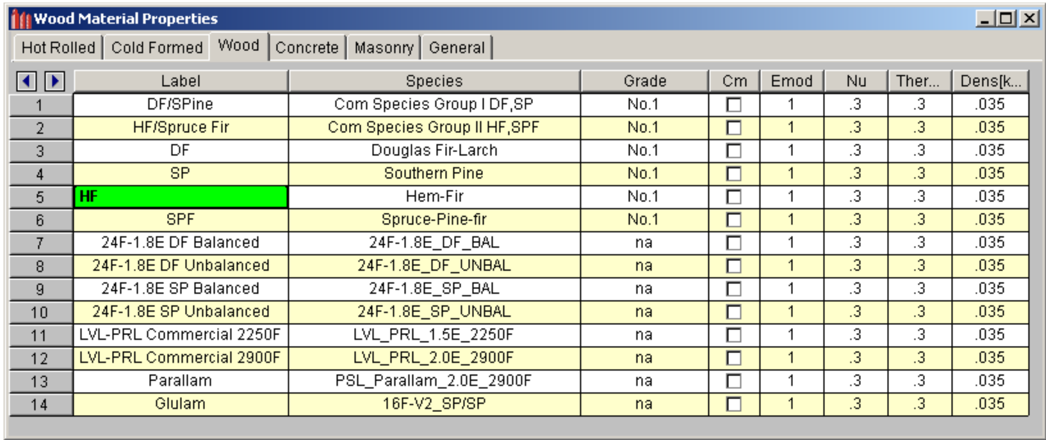
A.5	C
B	C.5
B.5	D





Let's verify our code and look at materials.

- Go to Global Parameters >> Codes and choose "NDS 2005" from the Wood drop down list.
- In the Materials spreadsheet under the Wood tab, we will add a "Parallam" and a "Glulam" material. See below:


Wood Material Properties

Hot Rolled | Cold Formed | Wood | Concrete | Masonry | General

	Label	Species	Grade	Cm	Emod	Nu	Ther...	Dens[k...
1	DF/SPine	Com Species Group I DF,SP	No.1	☐	1	.3	.3	.035
2	HF/Spruce Fir	Com Species Group II HF,SPF	No.1	☐	1	.3	.3	.035
3	DF	Douglas Fir-Larch	No.1	☐	1	.3	.3	.035
4	SP	Southern Pine	No.1	☐	1	.3	.3	.035
5	HF	Hem-Fir	No.1	☐	1	.3	.3	.035
6	SPF	Spruce-Pine-fir	No.1	☐	1	.3	.3	.035
7	24F-1.8E DF Balanced	24F-1.8E_DF_BAL	na	☐	1	.3	.3	.035
8	24F-1.8E DF Unbalanced	24F-1.8E_DF_UNBAL	na	☐	1	.3	.3	.035
9	24F-1.8E SP Balanced	24F-1.8E_SP_BAL	na	☐	1	.3	.3	.035
10	24F-1.8E SP Unbalanced	24F-1.8E_SP_UNBAL	na	☐	1	.3	.3	.035
11	LVL-PRL Commercial 2250F	LVL_PRL_1.5E_2250F	na	☐	1	.3	.3	.035
12	LVL-PRL Commercial 2900F	LVL_PRL_2.0E_2900F	na	☐	1	.3	.3	.035
13	Parallam	PSL_Parallam_2.0E_2900F	na	☐	1	.3	.3	.035
14	Glulam	16F-V2_SP/SP	na	☐	1	.3	.3	.035

We will now begin modeling the lateral system.

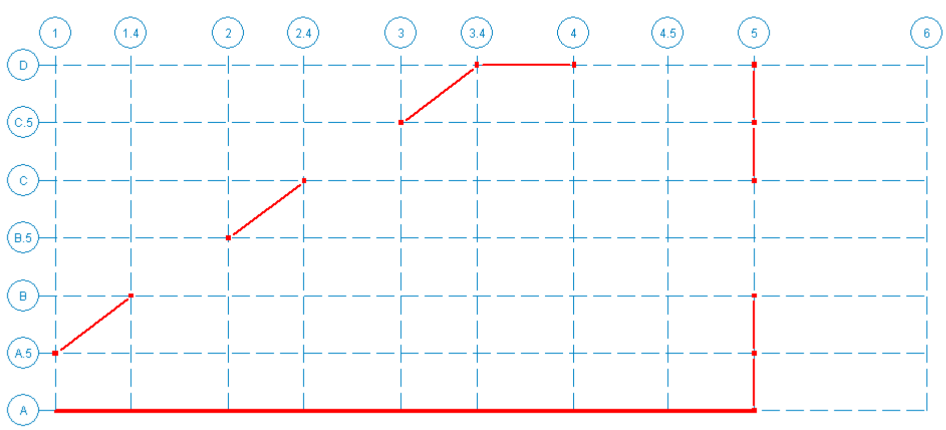
- Draw a lateral wall panel (Wood Material, DF) along Grid Line "A" between Grid Lines "1" and "5."
- Now place lateral columns (Wood, DF, Rectangular Double) at the following grid intersections:

A.5-1	D-3.4	C-5
B-1.4	D-4	C.5-5
B.5-2	A-5	D-5
C-2.4	A.5-5	
C.5-3	B-5	

- Draw lateral beams (Wood, Parallam Material, Parallam_2.0E Shape, Fixed-Fixed) between the following column locations:

A.5-1 & B-1.4	A-5 & A.5-5
B.5-2 & C-2.4	A.5-5 & B-5
C.5-3 & D-3.4	C-5 & C.5-5
D-3.4 & D-4	C.5-5 & D-5

Your model should now look like the image below:



We will now model the remaining columns, as well as the main glulam members and wood joists.

- Place gravity columns (Wood, DF, Rectangular Double) at the following grid intersections:

A.5-1.4	B-2	B.5-3	C-4.5
A.5-2	B-2.4	B.5-3.4	C.5-3.4
A.5-2.4	B-3	B.5-4	C.5-4
A.5-3	B-3.4	B.5-5	C.5-4.5
A.5-3.4	B-4	C-3	D-4.5
A.5-4	B-4.5	C-3.4	
A.5-4.5	B.5-2.4	C-4	

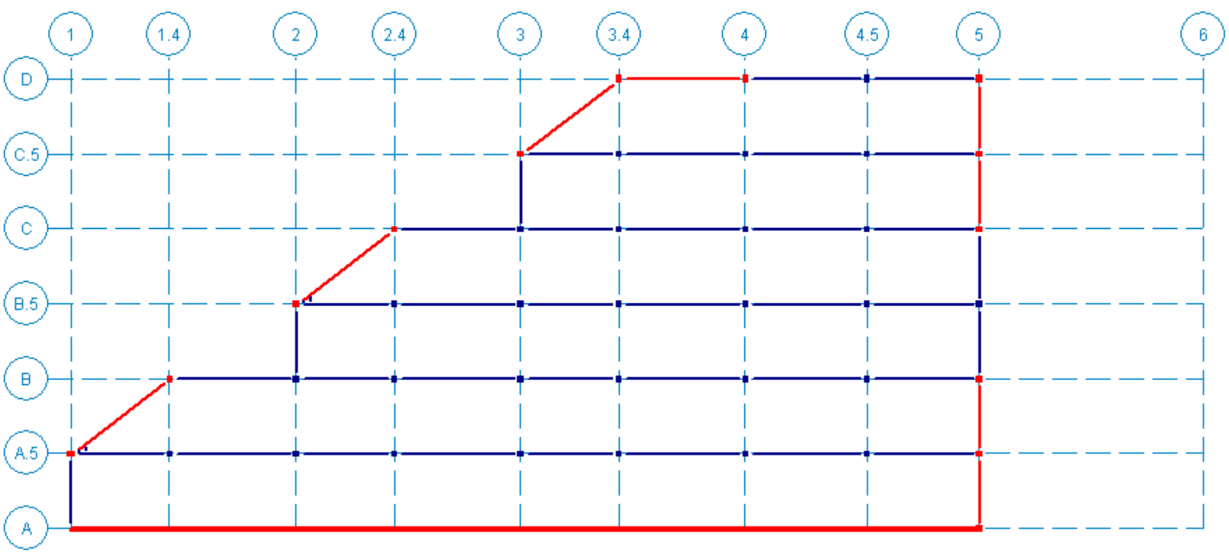
- Draw a gravity beam (Wood, Parallam Material, Parallam_2.0E Shape, Pinned-Pinned) in the two open bays along Grid Line "5."
- Draw gravity beams (Wood, Parallam Material, Parallam_2.0E Shape, Pinned-Pinned) along each grid Horizontally:

  Grid A.5, Grid B, Grid B.5, Grid C, Grid C.5, Grid D

- Draw gravity beams (Wood, Parallam Material, Parallam_2.0E Shape, Pinned-Pinned) between the columns at grid intersections:

  A-1 & A.5-1
  B-2 & B.5-2
  C-3 & C.5-3

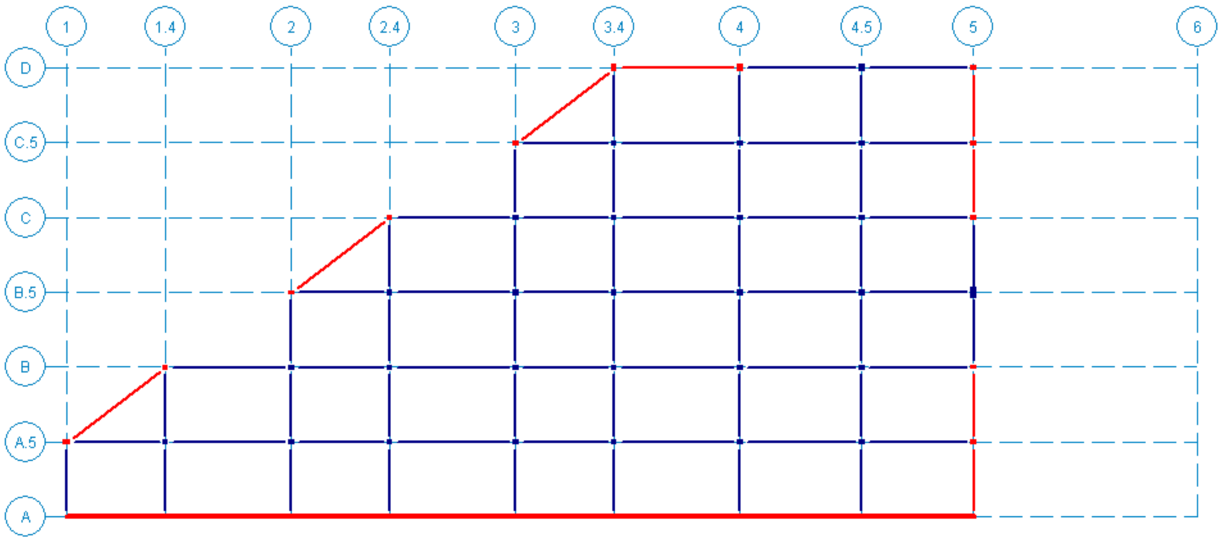
Your model should now look like the image below:



- Draw gravity beams (Wood, Wood Product, LPI Joist Shape Group, Pinned-Pinned) along each grid vertically:

  Grid 1.4, Grid 2, Grid 2.4, Grid 3, Grid 4, Grid 4.5

Your model should now look like the image below:

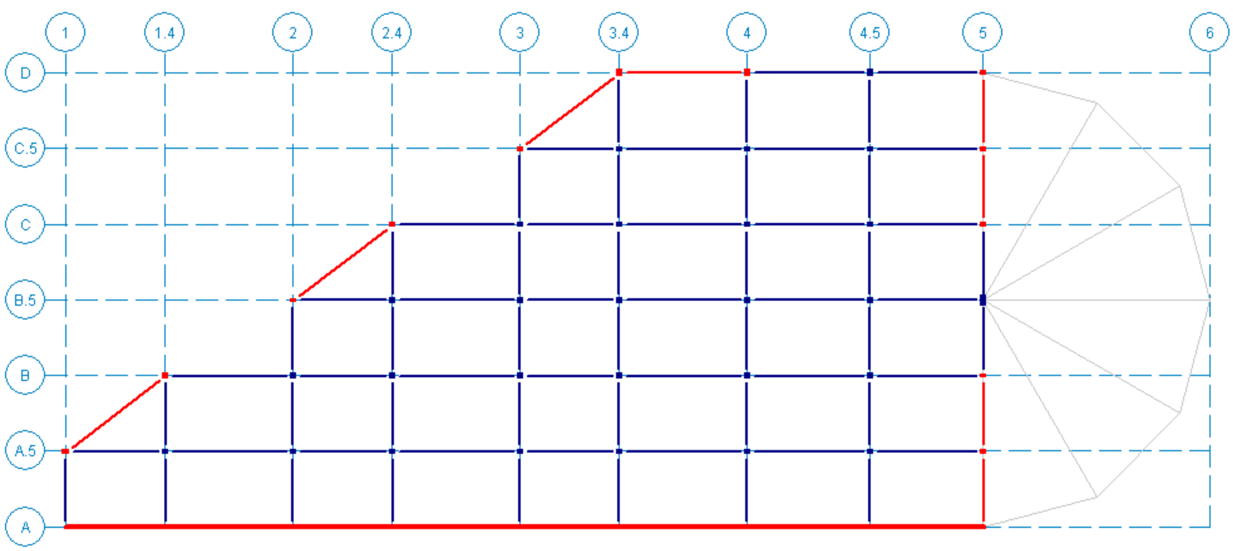


We will now model the radial portion of the structure.

Define a Radial Drawing Grid as follows:

- Set the Drawing Grid Origin to Z Axis to ”194,” X Axis to “48,” and the Start Angle to “-90” degrees.
- Enter “6@30” for the Angle Increments and “48” for the Radial Increments.
- Click the “Save” button, enter “Radial” as the grid description, and click “OK.”

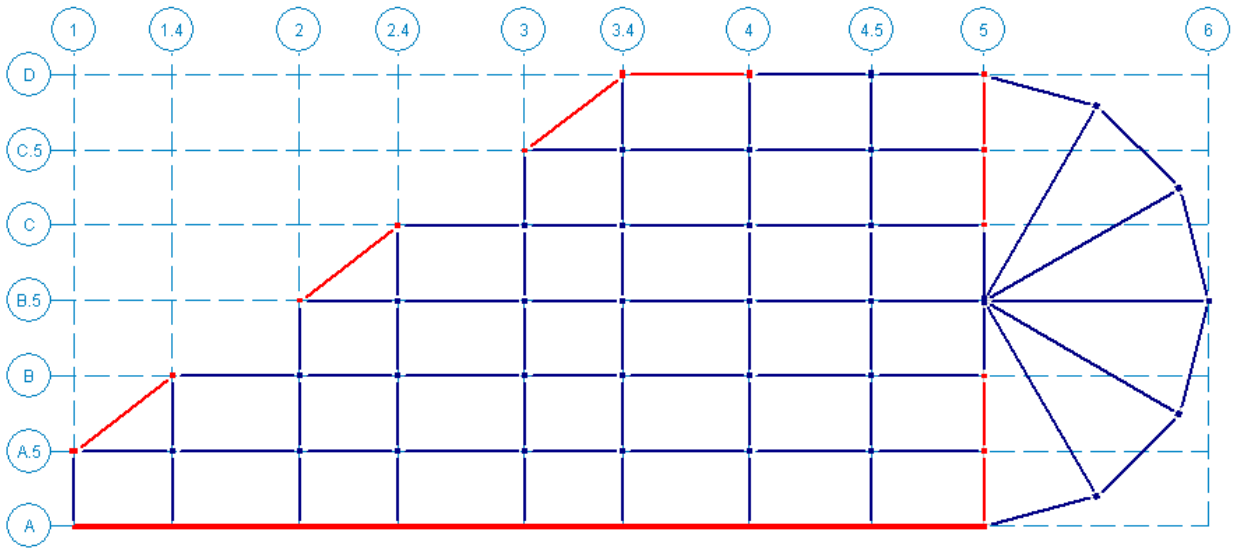
The Radial Drawing Grid should look like the following image:



- Place gravity columns (Wood, DF, Rectangular Double) along the perimeter of the radial grid at every spoke. Also, be sure to set the Orient to the center of the radius.
- Connect the columns with gravity beams (Wood, Parallam Material, Parallam_2.0E Shape, Pinned- Pinned) around the circumference.

- Connect the columns to the center column with gravity beams (Wood, Glulam Material, Glulam_Western Shape, Pinned-Pinned).

Your model should now look like the image below:



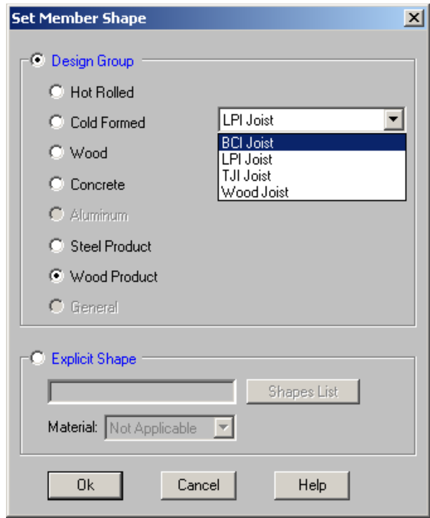
Now we will model the rest of the roof joists in the structure.

- Draw gravity beams (Wood Product, LPI Joist Shape Group) at 18" O.C., oriented vertically, within the six bays of the main structure.

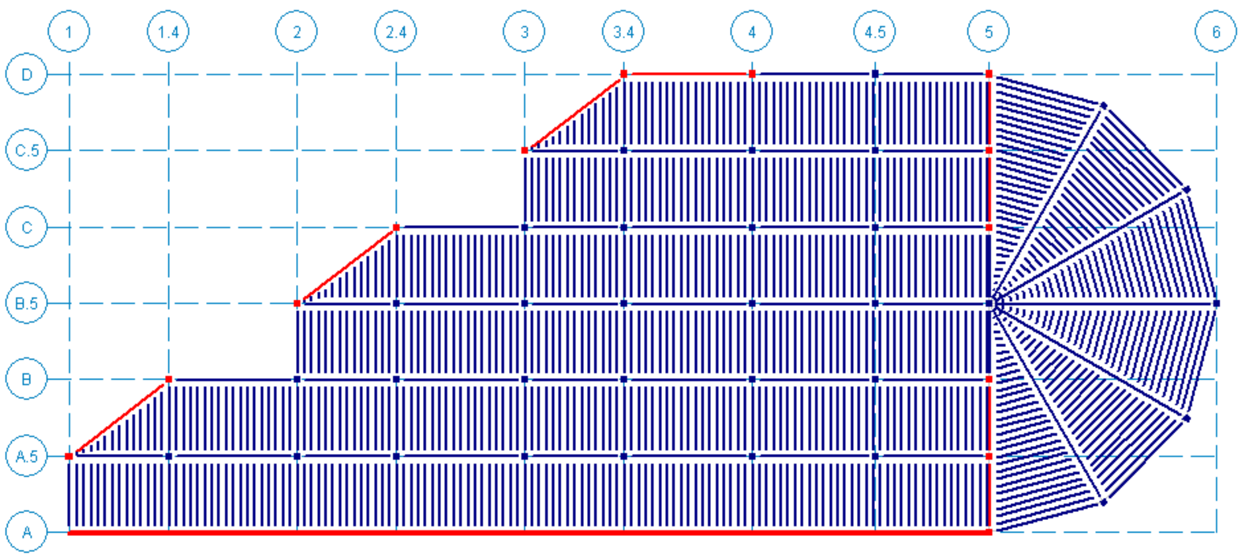
Note: When given an option, pick the green layout vs. the blue layout. The options exist because some of the infill framing dimensions are not exactly split into equal 18" O.C. segments. Thus, you must tell the program which end the spacing starts from.

- Delete the joists created that are 6" off of Grid 4.5 using the Delete tool.
- Draw gravity beams (Wood Product, LPI Joist Shape Group) at a spacing not to exceed 18" O.C., oriented parallel to the perimeter joists, within the six bays of the radial portion of the structure.

Note: The options for Wood Products consist of BCI Joists (Boise-Cascade), LPI Joists (Louisiana Pacific), TJI Joist (Trus Joist) and a generic "Wood Joist" that will choose from any of the manufacturers. The design values for these joists were pulled directly from the ICC (International Code Council) reports. Unfortunately, the wood joist industry is evolving rapidly and coming up with new types of joists that vary by manufacturer. Some of the new joists, LP SolidStart I-Joists for example, have not been considered in the program. There has been discussion in the wood industry that there are going to be some unified joist sizing/specification coming soon that RISA will quickly implement.



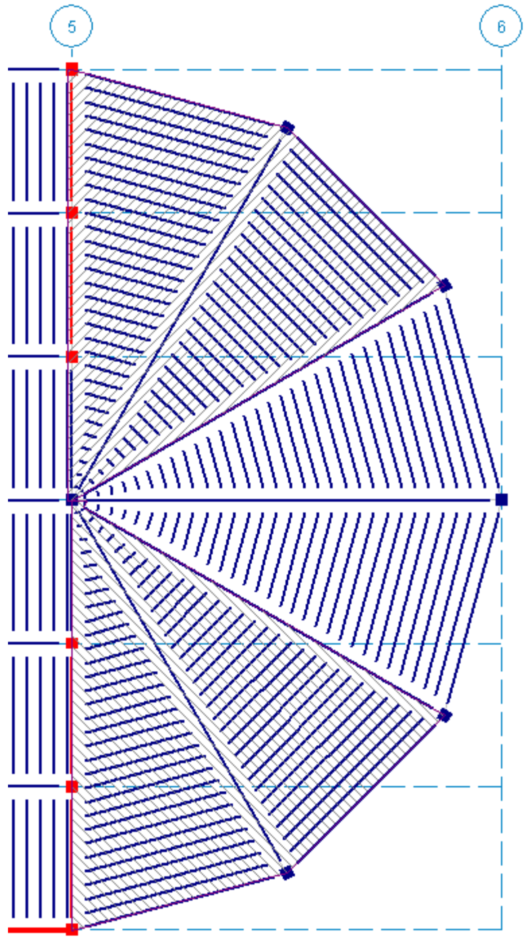
Your model should now look like the image below:



Now we will configure the deck.

- Create an Edge Perimeter with a 0 inch Overhang Distance along the entire perimeter of the structure with a Rigid Diaphragm.
- Draw a local roof deck (Wood) with an angle from horizontal of 60 degrees in the upper one-third of the radial portion of the structure.
- Draw a local roof deck (Wood) with an angle from horizontal of 120 degrees in the lower one-third of the radial portion of the structure.

When Deck Assignments are displayed "As Input" without the 'Include Default', your model should look like the image below:



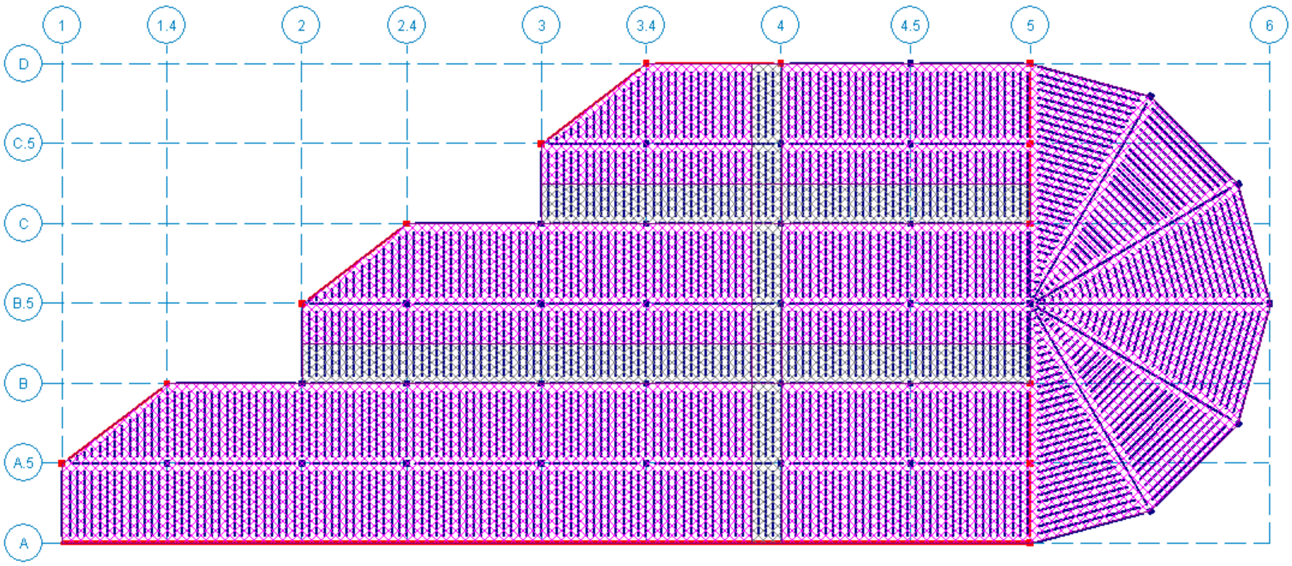
We will now begin loading the model.

- Create a new Area Load Definition, labeled "Roof," with 0.01 ksf PostDL, 0.02 ksf SL, and 0.01 Dyn Load.
- Set the Default Area Load for the structure to "Roof."
- Draw an "Add Piping Area Load" strip **8** feet wide along Grid Line "B" from Grid Line "2" to Grid Line 5.

  Similarly, draw an "Add Piping Area Load" strip **8** feet wide along Grid Line "C" from Grid Line "2" to Grid Line 5."

  In the opposite direction, draw an "Add Piping Area Load" strip **6** feet wide along Grid Line "4" from Grid Lines "A" to "D."

When Uniform Area Loads are displayed “As Input” and “Include Default” (make sure the decks are not showing), your model should look like the image below:



Note: When drawing tapered area loads the first two clicks define the base magnitude and second two clicks define the peak magnitude. Since it is easier to first click the edge beams, the base and peak magnitudes have been flipped. For triangular tapered snow loads, you cannot define the first two points at the same location. The points need to be slightly off. Any other points can be the same with no problems. This is because the program doesn’t yet allow for the first two clicks, defining the base magnitude, to use the same point.

Now we will build load combinations and solve the model.

- Generate Load Combinations based on the IBC 2006 ASD building code including Snow Load (SL). Pre-Composite combinations are not required in this model.
- Solve the model.

As soon as the model is solved the Warning Log opens up, stating that a few of the columns could not be designed. This most likely means that there was not a shape that worked for this design list. If we look at our model we can see that these column locations appear to be the most heavily loaded columns in our structure. Let’s try a rectangular triple for these members:

- Open the “Columns” spreadsheet.

  Choose “Rectangular Triple” for these members.
- Solve the model again.

Note: After solving this time the warning log does not show up, thus it appears that everything was able to be designed.

Take a look at the results.

The three main spreadsheets that will summarize our results are the Design Results, Code Checks, and the Column Results.

Here we can see all of the code check ratios and member sizes as well.

> **Note:** Under the Wood Products tab is where the information for wood joists is located. There is no explicit code check for wood joists displayed in this spreadsheet. The only way to know if there are any problems with these members would be shown in the warning log or looking at the detail report for these members.

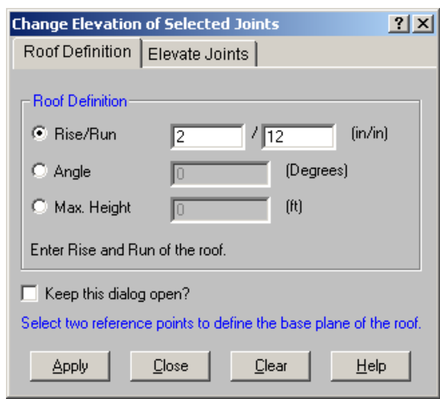
Sloping the Roof

A newer feature implemented in RISAFloor is the ability to slope the highest floor level in your structure. Here we will run through a quick demonstration of the procedure.

- Select all of members that are on or between Grids “1” &” 5” and Grids “A” & “B.5.”

  Click the Sloping Member icon .

  Input 3:12 for “Rise/Run.”

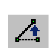


  Click the grid intersections at A-1 and A-5.

This will slope this area at 3:12. We now need to do the same to the other side of the structure.

- Unselect the entire model.

  Select all of members that are on or between Grids 1 & 5 and Grids B.5 & D.

  Click the Sloping Member icon .

  Input 3:12 for “Rise/Run.”

- Click the grid intersections at D-1 and D-5.

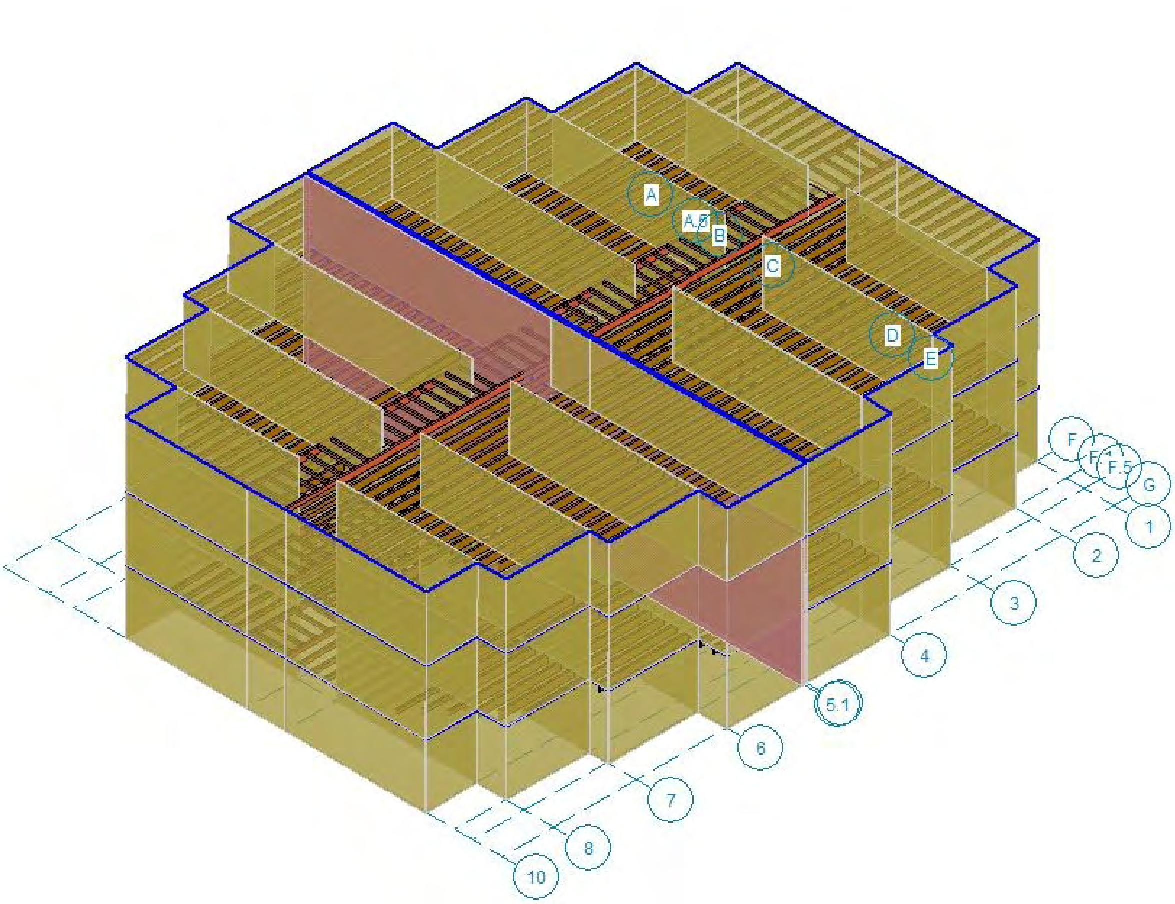
This model should now look like the initial picture. We can also open up the “Point Locations” spreadsheet and see the new elevations of our joints.

Note: There are many different options for sloping your roofs, depending on geometry. See our help file for much more information on this. We do consider axial forces induced by the sloped members for wood and steel. There are some limitations as to how your roof must be framed to use this feature. For example, saw tooth roofs are not allowed because any point in plan on a roof level can only exist at a single elevation. For specialized roof framing that does not work with RISAFloor, the model can always be taken into RISA-3D.

Notes:

Multi-Story Hotel

This model is used to introduce the Floor-3D interaction as well as describe wood wall modeling and show advanced modeling techniques.



Project Definition: Design_Methods.rfl

Wood Walls with a Masonry wall at the expansion joint
Stud Members: 2x @ 24" O.C.
Material: DF/SPine Select Structural

RISAFloor and RISA-3D Interaction

RISAFloor

The primary function of RISAFloor is to optimize the floor and roof system for gravity loads. The lateral system is defined in RISAFloor in order to utilize the interaction between RISAFloor and RISA-3D.

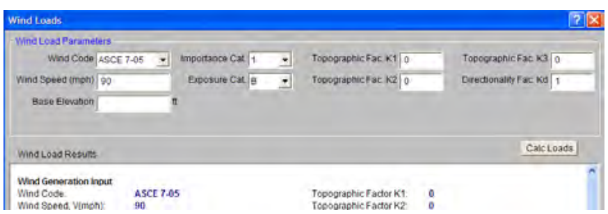
Beams, columns, and walls whose function is set to 'Lateral' on the will automatically be generated in the RISA-3D model when accessed via the Director Menu.

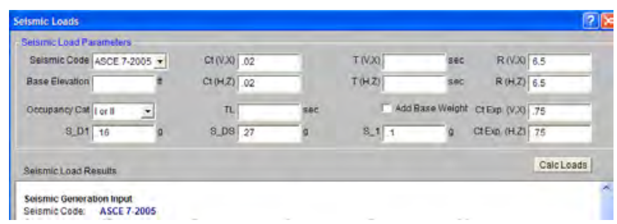
Click the Director Menu on the far right end of the Main Menu and choose 'RISA-3D'.

The RISA Application Interface will then switch from RISAFloor to that of RISA-3D.

Integration

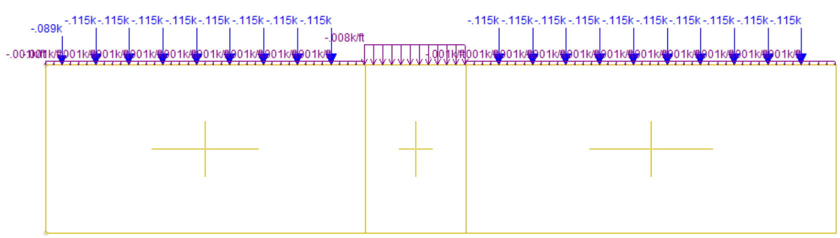
As the model is sent to RISA-3D, the Automated Wind Load generator and Automated Seismic Load Generator will appear. (You can set-up your lateral load options in RISAFloor Global Parameters before making the interaction). Shown below is the data entry for the lateral loads:





RISA-3D

All the gravity loads for the lateral members are also brought into RISA-3D as point or distributed loads:



The generated wind and seismic loads will create a point load which will be applied to each floor level and placed at the center of mass. Additional point loads will be placed at 5% eccentricity for Seismic loads, and partial wind loads. New joints will be created to define these locations. These joints will be attached to the diaphragm.

In RISA-3D, you will notice that you can use the RISA-3D features to edit and solve the model. You can add braces, beams, columns, walls, and additional loads just as you would in a regular RISA-3D model.

Return to RISAFloor

The "gravity" model in RISAFloor and the "lateral" model in RISA-3D are fully linked. Subsequently, any changes made to RISAFloor generated members in the RISA-3D model will automatically update those same members in RISAFloor model.

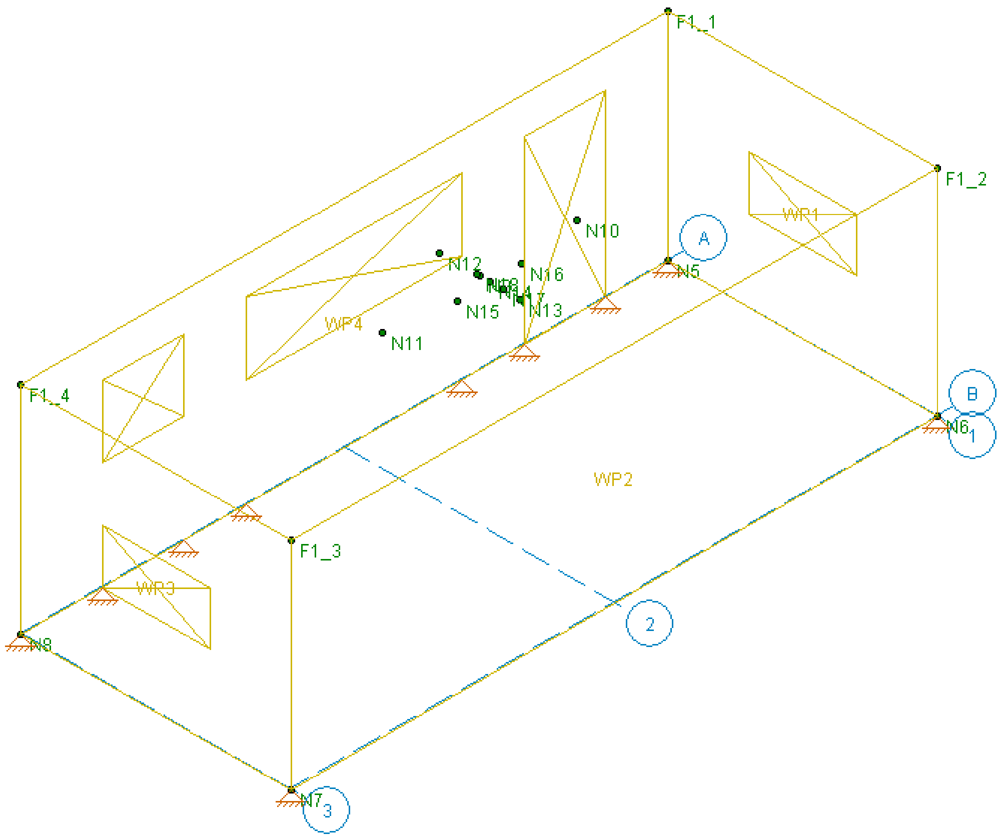
Once your model RISA-3D model is designed, you can use the Director button to return back to RISAFloor and any changes to the model will be remembered.

Note: RISAFloor 'Gravity' members may be viewed in RISA-3D via the Misc Tab of the Plot Options Dialog. These members will be displayed for visual effect only in the model view but will not contribute to the stiffness of the RISA-3D model.

Notes:

Wood Walls with Openings

This model is used to further introduce the Floor-3D interaction as well as describe wood wall modeling. We will discuss the three design methods: Segmented, Perforated and FTAO and give some insight as to design parameters, etc.

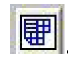


Project Definition: Wood_Wall_Design_Methods.r3d

Chord Members: 2-2x6
Stud Members: 2x6 @ 16" O.C.
Material: HF/Spruce Fir Species: Com Species Group II HF,SPF Select Structural

Modeling Procedure

We'll begin by creating a new Floor and generating a Project Grid.

- Start a new model by clicking on the Create New Floor Plan button .
- Create an Original Floor with an elevation of 8 feet.

    Ignore the Default Area Load, Deck, and Angle entries as we will revise these later.

Define a Project Grid as follows:

- On the Horizontal Project Grid – “Z Axis” Tab

    Start Label of “1” and Start Location of “0”.

    Enter “2@12” for the Increments.

    Click “Generate.”

- On the Vertical Project Grid- “X Axis” Tab

    Set the Start Label to “A” and Start location to “0”,

    Enter “10” for the Increments.

    Click “Generate.”

Define your Materials.

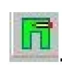
- Click on 

, and select the “Wood” tab.

    Change “HF/Spruce Fir” to Grade “Select Structural”.

    Close the spreadsheet.

We will now draw the walls of the structure.

- Draw lateral walls by clicking on the Draw Wall button .

    Select: Wood, HF/Spruce Fir, Design Rule: Typical, Lateral.

    Draw between the following Grid Locations:

        B-1 and A-1

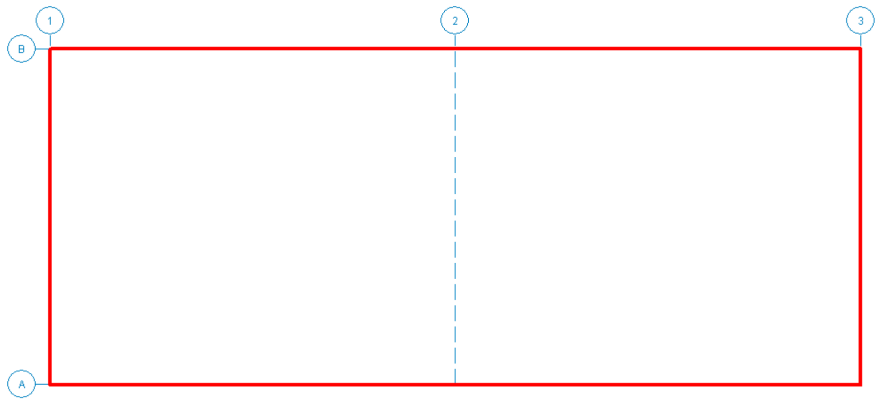
        B-3 and B-1

        A-3 and B-3

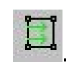
        A-1 and A-3

> **Note:** The direction that a wall panel was drawn affects which way the wall panel will be seen in the Wall Panel Editor. Be sure to draw your wall panel from left to right in the direction you want to view it to add openings, etc. In this model we are going to set it up so we are looking at it from the outside.

Your model should now look like the image below:

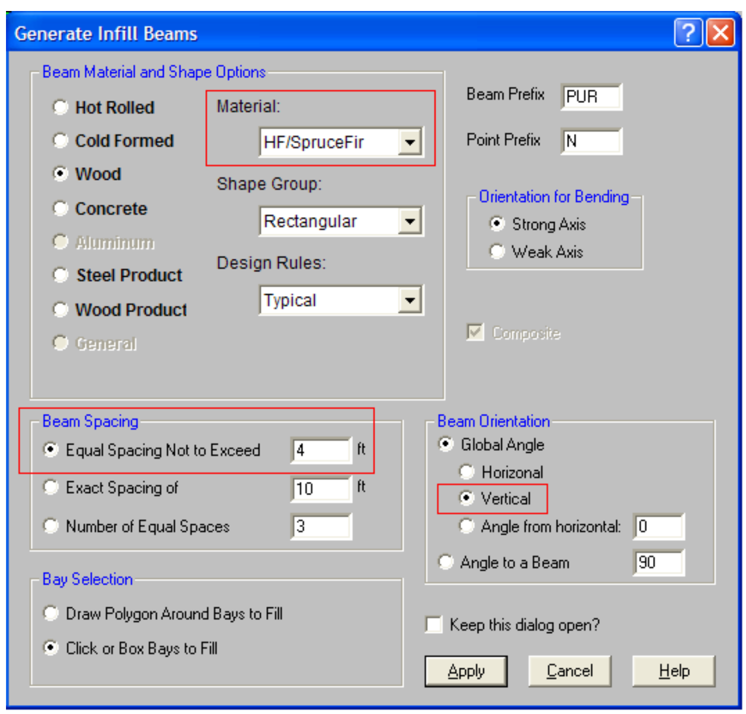


Now we will model the interior bay framing within the structure.

- Draw vertical Infill Beams by clicking on the “Generate Beams within a Bay” button .

  Select: Wood, HFir/Spruce Fir, Rectangular, Vertical.

  Select Beam Spacing Not to Exceed 4’ O.C.



- Click inside of the project grid to create our framing.

Now we will generate the deck/slab perimeter.

- Draw Slab by clicking on Create Slab Edges button .

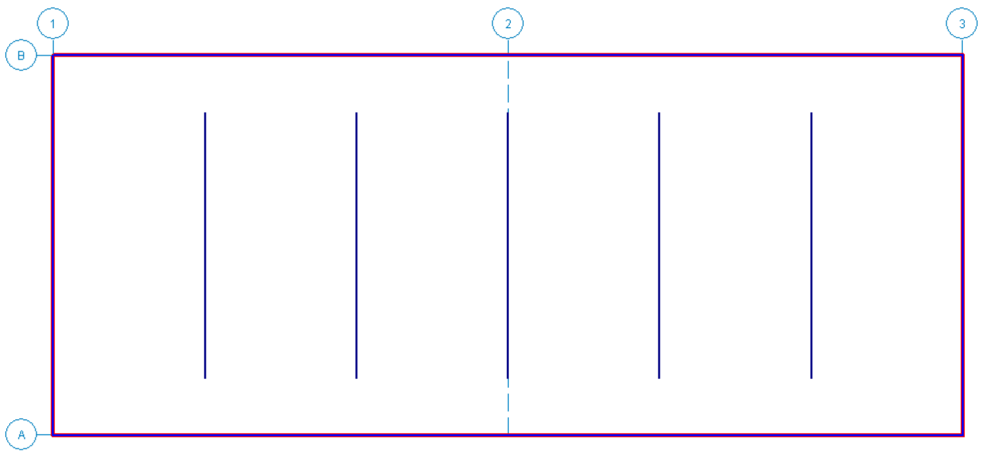
> **Note:** RISAFloor uses the terminology "slab edge" to denote the extents of a slab and diaphragm system. This applies not only to conventional slabs but also metal and wood decks.

- Enter an Edge Overhang Distance of "0" inches along the entire perimeter of the structure.
- Click on "Create a Slab Perimeter," choose "Rigid"

  Click on "Consider ALL selected Beams/Walls" for the SLAB perimeter,

  Press Apply.

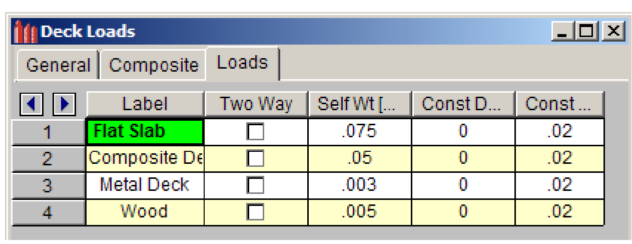
Your model should look like the image below (the dark blue outlines the slab edge; because of the zero inch slab edge, the graphic for the edge and the wall panels overlap):



Now we will model the Deck/Slab.

- Using the "Deck Definitions" spreadsheet we will modify the "Wood" decking.

  Under the "Loads" tab input a self-weight of 0.005.



Deck Loads

General | Composite | Loads

	Label	Two Way	Self Wt [...	Const D...	Const ...
1	Flat Slab	☐	.075	0	.02
2	Composite De	☐	.05	0	.02
3	Metal Deck	☐	.003	0	.02
4	Wood	☐	.005	0	.02

Now we will model the gravity loading.

- Click the Area Load Definitions button 

.

  Add a line to the bottom of the spreadsheet by pressing "Enter" on the last line.

  Input values for roof loading as seen below:

Uniform Area Loads

Standard | Other

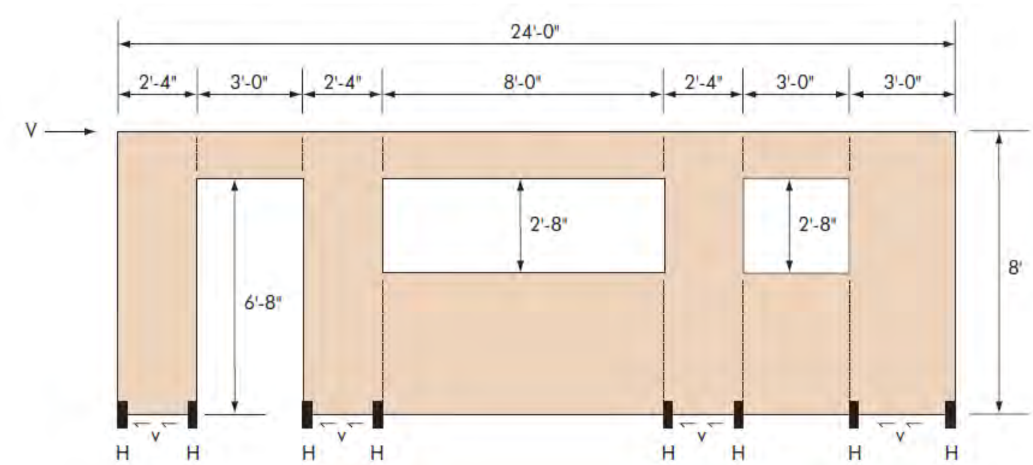
	Label	Addit...	PreDL[ksf]	PostDL[...	LL[ksf]	LL Type	VL[ksf]	Dyn Loa...
1	Office	☐		.01	.08	LL-Reduce	.011	.075
2	Storage	☐		.01	.125	LLS-Non	.011	.175
3	Public	☐		.01	.1	LL-Non	.004	.075
4	Add Piping	☑		.02		LL-Non	.011	
5	Roof	☐		.01	.02	LL-Non	.011	.01

- Click on the Floors Spreadsheet on the Data Entry Toolbar.
- Verify that for “Floor Plan” the Area Load Default is set to “Roof,” the Deck Default is set to “Wood,” and the Deck Angle is set to “0” degrees.

Floors

	Label	Elevatio...	Area Load ...	Deck Defa...	Deck Angl...	Parent	Inacti...	Splic...	Splic...
1	Floor Plan 1	8	Roof	Plywood	0	None	☐	8	Moment

Let’s add openings to a wall to match this diagram:

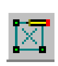


- Turn on the wall panel labels through “Plot Options”,

  Double-click on WP4.
- Change the Units of length to inches.
- Double-click WP4 to open the Wall Panel Editor.
- For the “Grid Increments,” set H to “28,36,28,96,28,36,36” and V to “48,32,16”.

Now we can use the coordinates in the lower right hand corner to locate where we are at the wall. Each grid intersection will snap to a specific coordinate.

	Deck & Uni. Area Load Display Off	0, 0, 2.33333 (ft)

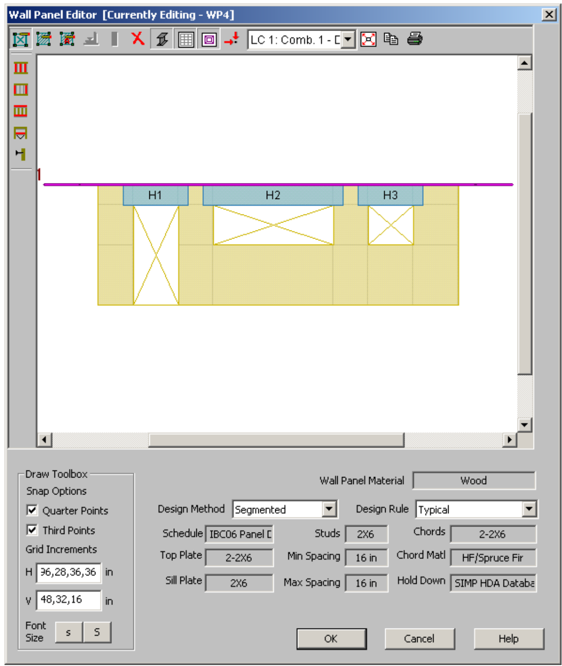
- Click the “Draw New Openings” icon  in the Wall Panel Editor.

  For the doorway, click at (0,0,28) and at (0,80,64).

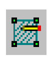
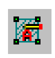
  Input the two window openings in a similar fashion: (0,48,92) and (0, 80,188);

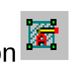
(0,48,216) and (0,80,252).

- Notice that header beams are shown graphically. The wall panel editor should look like this:



In order to get results in from the program we know need to define regions for our wall panel.

There are two icons for laying out regions  and . The first allows the user to manually define regions, while the second creates regions automatically.

- Click the “Generate Wall Regions Automatically” icon  .

Now you will see that within the wall panel editor regions now make up the entire wall. Also, the potential framing is laid out for you based on your default design rules. There are toggles on the left side of the editor to shut off the graphical view of this framing.

Now let’s define our design method.

RISA has included three options for wood wall panel design: Segmented, Perforated, and Force Transfer Around Openings. For this wall we will use the Force Transfer method.

- Choose “Force Transfer” in the drop down list for Design Method.

  Press “OK”.

> **Note:** In RISAFloor, the different design method options are not considered because they are shear wall design methods. Because RISAFloor only considers gravity loads, then the wall studs are simply checked for axial compression. All other design considerations are taken into account in RISA-3D.

Now let's consider the end walls.

We will define WP1 and WP3 as Perforated and Force Transfer Around Openings respectively.

- Change the units of length back to "Feet".
- Double-click WP1.

  Add an opening by clicking (3,3,0) and (7,5,0).

  Auto-generate regions.

  Change "Design Method" to 'Force Transfer'.

  Press OK.
- Double-click WP3.

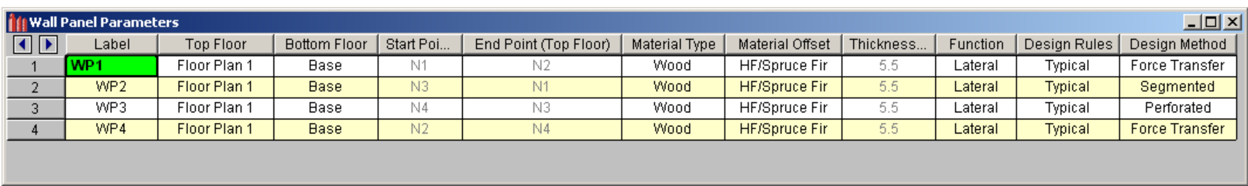
  Add an opening by clicking (3,3,24) and (7,5,24)

  Auto-generate regions.

  Change "Design Method to "Perforated".

  Press OK.

The "Wall Panels" spreadsheet should now look like this:



Wall Panel Parameters

	Label	Top Floor	Bottom Floor	Start Poi...	End Point (Top Floor)	Material Type	Material Offset	Thickness...	Function	Design Rules	Design Method
1	WP1	Floor Plan 1	Base	N1	N2	Wood	HF/Spruce Fir	5.5	Lateral	Typical	Force Transfer
2	WP2	Floor Plan 1	Base	N3	N1	Wood	HF/Spruce Fir	5.5	Lateral	Typical	Segmented
3	WP3	Floor Plan 1	Base	N4	N3	Wood	HF/Spruce Fir	5.5	Lateral	Typical	Perforated
4	WP4	Floor Plan 1	Base	N2	N4	Wood	HF/Spruce Fir	5.5	Lateral	Typical	Force Transfer

Let's generate load combinations and solve the model.

- Open the "Load Combinations" spreadsheet.

  Delete the two default entries.

  Click the LC Generator button LC Generator.

  Choose "ASCE 2005 ASD".

  Uncheck all of the options except "Post Composite" and Generate.

  Press Solve.

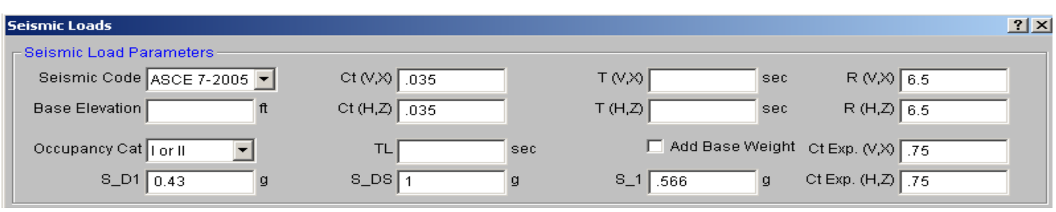
Now let's move to RISA-3D.

- Click the Director button in the upper left-hand corner and choose "RISA-3D".

Moving from RISAFloor to RISA-3D will trigger the Wind and Seismic Load Generators.

- For "Wind Loads" leave the default settings.

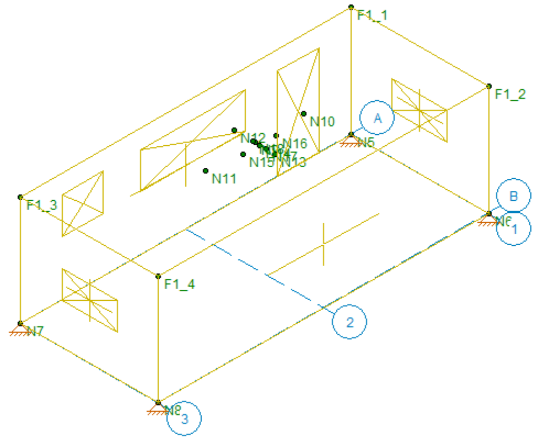
  Press OK.
- For "Seismic Loads" , enter the values as shown below:



- Press Calc Loads. We will see the seismic values change per the ELF method.

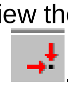
  Press OK.

Now the model is shown in ISO view in RISA-3D. Also, note the presence of hold-downs.



> Hold-downs are just boundary conditions used to model the anchorage of your walls. They work similarly to boundary conditions, except that they can only be applied in the Wall Panel Editor. There are also hold-down requirements for wall design.
>
> - For Segmented design, a hold-down is required at the bottom corners of each full-height segment of wall.
>
> - For Perforated or FTAO design, a hold-down is required at the bottom corners of each full wall panel.
>
> If these have not been defined, the program will not design your wall panels.
>
> Also, note that automatic hold-downs are created when you come into RISA-3D, so your Perforated, FTAO, and Segmented walls with no openings are already taken care of.

- Open the Basic Load Cases spreadsheet and note that all of our BLCs are already populated.

  We can also view the loads graphically using the dropdown list and the Display Load Toggle button. .

  Let's open the Load Combinations and use the LC Generator to create "ASCE 2005 ASD" load combinations.

  For wind, click the "X and Z w/Eccentric".

  For seismic, click the "X and Z w/Eccentric".

  Click Generate.

> **Note**: Per Figure 6-9 of the ASCE 7-05, the "X and Z w/Quartering" covers every possible wind combination. Since we want to decrease the total load combinations, we will not run all of these.

We now see these combinations created with their ASCE load combination number. Because of the large amount of load combinations, we will not solve all of them.

- Click the Solve header in the LC's spreadsheet, which will highlight the entire column.

  Right-click and select "Fill Block".

Leaving the entry blank, click OK.

Now, highlight the first 20 lines in the Solve column.

Right-click again and “Fill Block”.

Type any value in the space provided and click OK.

Here we simply selected the first 20 load combinations as the ones that we wish to solve.

- Click the Solve Batch button.

Here we see the program run through each load combination. When the program stops, we see a Warning Log message.

Notes on the Warning Logs for Wall Panels:

-For the FTAO method a door cannot be present. There must be space above and below an opening, otherwise the method is not valid and the program will not design it.

-Also, note that there are aspect ratio limitations the program considers. If a wall panel or region does not meet the aspect ratio requirements of the NDS Special Design Provisions for Wind & Seismic, then we will not design this wall either.

-Also, if you have defined a wall panel as FTAO or Perforated but there is no opening in the wall, then we will do a Segmented design for those wall panels.

If we take a look at the detail report we will see no design present. Also, in the “Wall Panel Design” spreadsheet, we will NC (no calculation values for these wall panels). Thus, we must change our design method for these to get a valid design.

- Open the “Wall Panel” spreadsheet.

Change WP4 from the “Force Transfer” method to “Segmented”.

Wall Panel Data

	Label	A Joint	B Joint	C Joint	D Joint	Material Type	Material Set	Thickness...	Design Rule	Design Method	SSRF
1	WP1	F1_1	F1_2	N6	N5	Wood	HF/SpruceFir	5.5	Typical	Perforated	1
2	WP2	F1_3	F1_4	N8	N7	Wood	HF/SpruceFir	5.5	Typical	Force Transfer	1
3	WP3	F1_2	F1_4	N8	N6	Wood	HF/SpruceFir	5.5	Typical	Segmented	1
4	WP4	F1_1	F1_3	N7	N5	Wood	HF/SpruceFir	5.5	Typical	Segmented	1

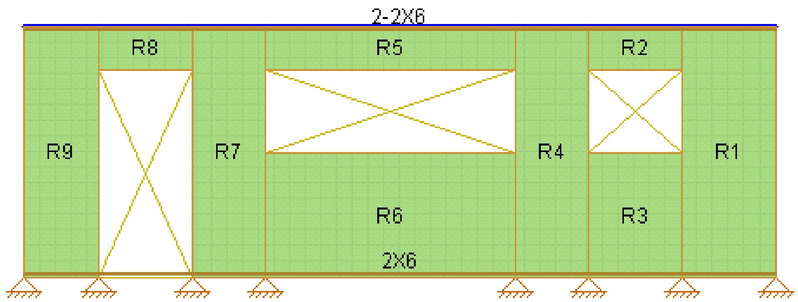
- Now solve a batch solution again.

This time we get a new error log that points us to specific locations where hold-downs must be added to get a design for our Segmented wall. Let’s add those hold-downs.

- Double-click WP4, opening the Wall Panel Editor.

Click the Add New Hold-downs button .

Click the lower corners of each full-height segment.



- Once we have applied all of these, click OK to exit the Wall Panel Editor and solve one final time.

Let's take a look at our results.

This time we get no warning logs, thus we can now take a look at our design results.

- Click the "Wall Panel Design" spreadsheet and look at the "Wood Wall Axial" tab.

Note: We see that WP2 and WP4 have region call-outs and WP1 and WP3 do not. This is because Perforated and FTAO design methods do give results on a per region basis. They give results on a full wall basis. Segmented design simply considers each full-height region as its own separate shear wall. Thus, the results for these are based solely on the individual regions we are looking at. We will see in the spreadsheet that regions above and below the opening are not considered in design, thus they have "NC" shown for their design results.

Here we can see our stud size, spacing and code check, as well as our chord size and code check.

- Click the Wood Wall In-Plane tab.

Here we see the shear panel and hold-downs chosen from the databases, as well as the code checks for each.

The spreadsheet results give a great overview to what is happening with our wall panels.

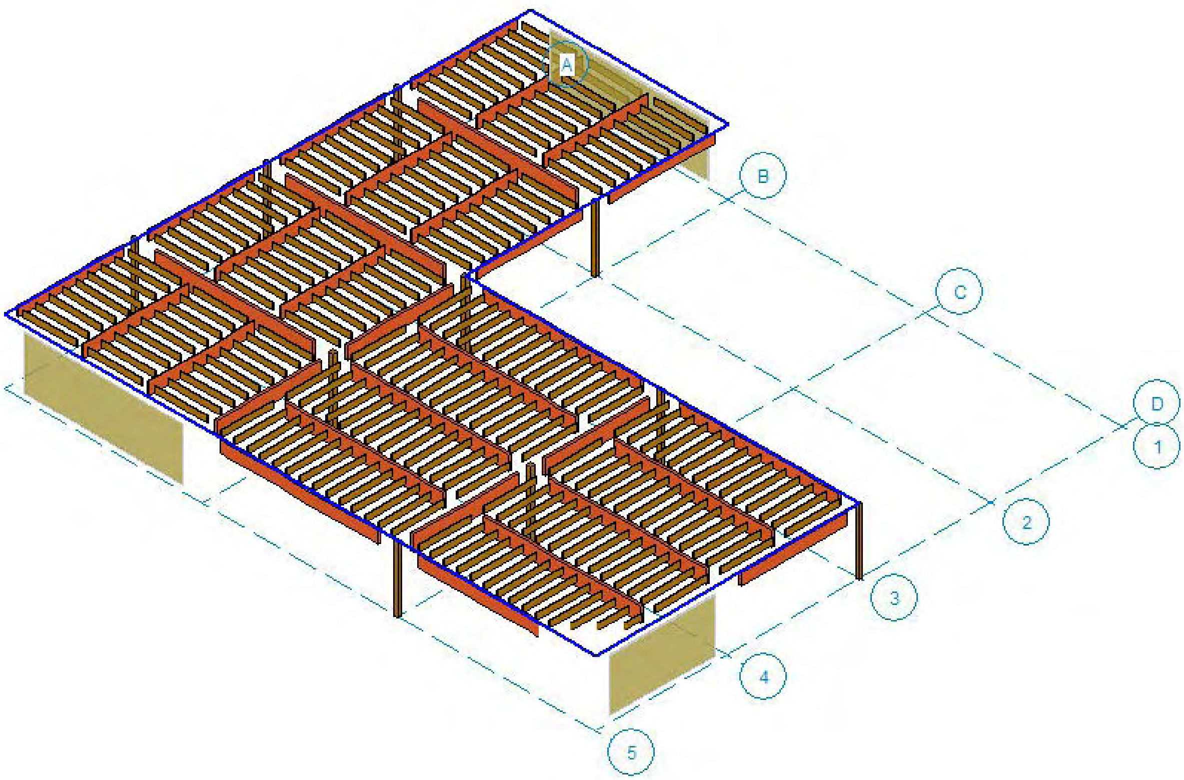
We can also take a closer look at our wall design parameters and forces. Here we will step through the detail reports for each design method focusing on Perforated and Force Transfer, as the Segmented method is identical to the design of a wall with no openings.

- Click the button on the left side of the screen and click on WP3 graphically to go through the Perforated results.
- Now, within the "Wall Panel Design" spreadsheet, click on the WP1 label and then click to go through the FTAO results for this wall.

Notes:

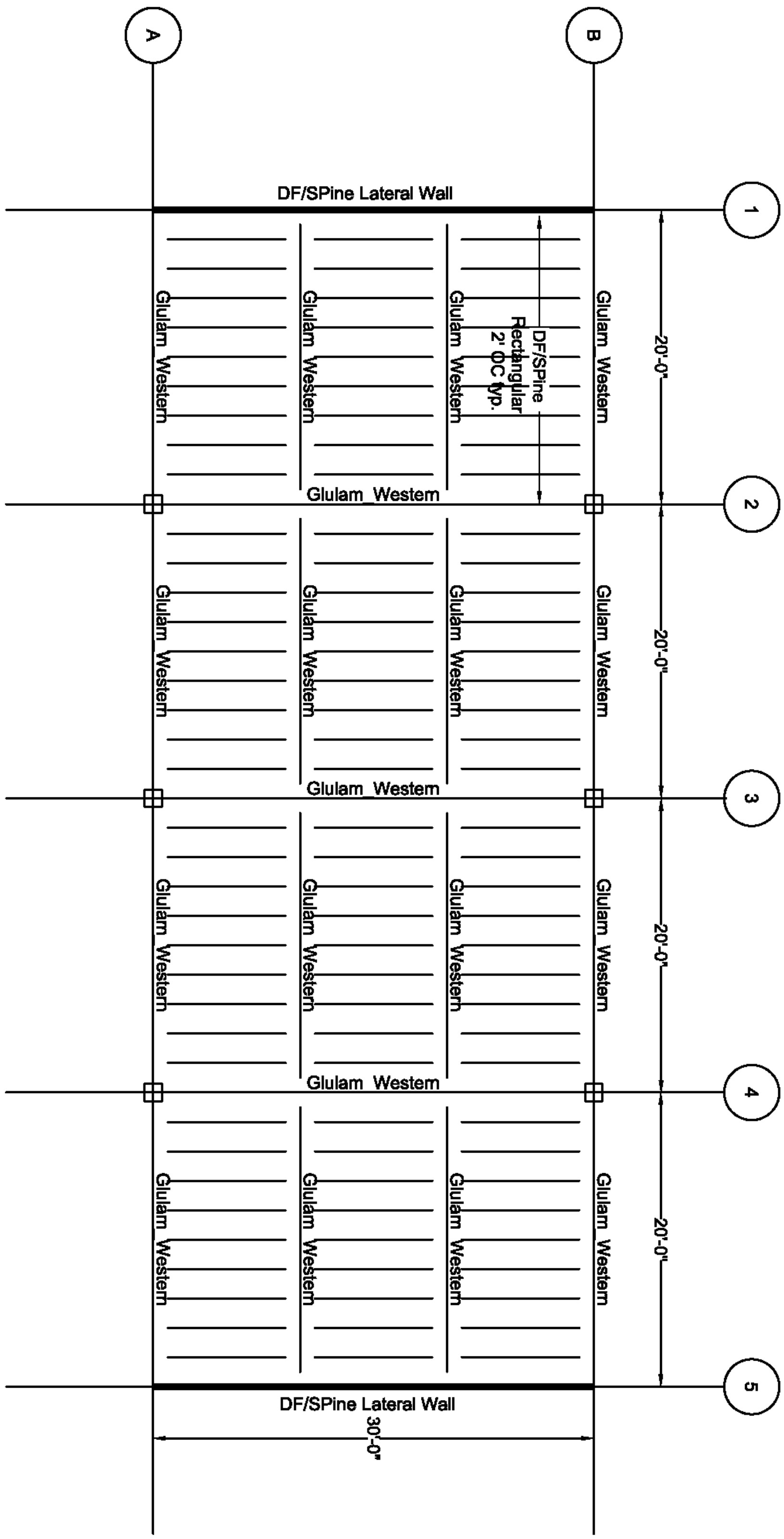
Flexible Diaphragm

This model is intended to demonstrate modeling with flexible diaphragm analysis and design.

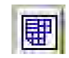


Project Definition: Diaphragm.rfl

Single Story – 10 feet
Wood Beams – DF/ SPine- Rectangular, Glulam_Western- 24F-1.8E DF Balanced
Wood Columns – Lateral- Rectangular
Walls – 2x6 Wood, DF/SPine, Design Rule: Typical


A
B
1
2
3
4
5
DF/SPine Lateral Wall
DF/SPine
Rectangular
2' OC typ.
Glulam_Western
20'-0"
20'-0"
20'-0"
20'-0"
DF/SPine Lateral Wall
30'-0"

We'll begin by creating a new Floor and generating a Project Grid.

- Start a new model by clicking on the Create New Floor Plan button 
- Create an Original Floor with an elevation of 10 feet, Area Load- Office, Deck Default- Wood, Deck Angle- 0

Now let's change the default loading.

- Open the Area Load Definitions spreadsheet, and set the Office loads (Ignore VL):

  Dead Load- 0.01ksf

  Live Load (Reducible)- 0.08 ksf

  Dyn Load- 0.01ksf

Define a Project Grid as follows:

- On the Horizontal Project Grid- Z Axis Tab

  Start Label of '1' and Start Location of '0'

  Enter '4@20' for the Increments

  Click 'Generate'.

- On the Vertical Project Grid- X Axis Tab

  Set the Start Label to 'A' and Start location to '0',

  Enter '1@30' for the Increments

  Click 'Generate'.

We will now draw the Walls of the structure.

- Draw 'Lateral' Walls by clicking on the Draw Wall button 

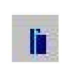
  Select: Wood, DF/S Pine, Design Rule: Typical, Lateral

  Draw between the following Grid Locations:

  A-1 & B-1

  A-4 & B-4

We will now draw the Columns within the structure.

- Draw 'Lateral' Columns by clicking on the Draw Column button 

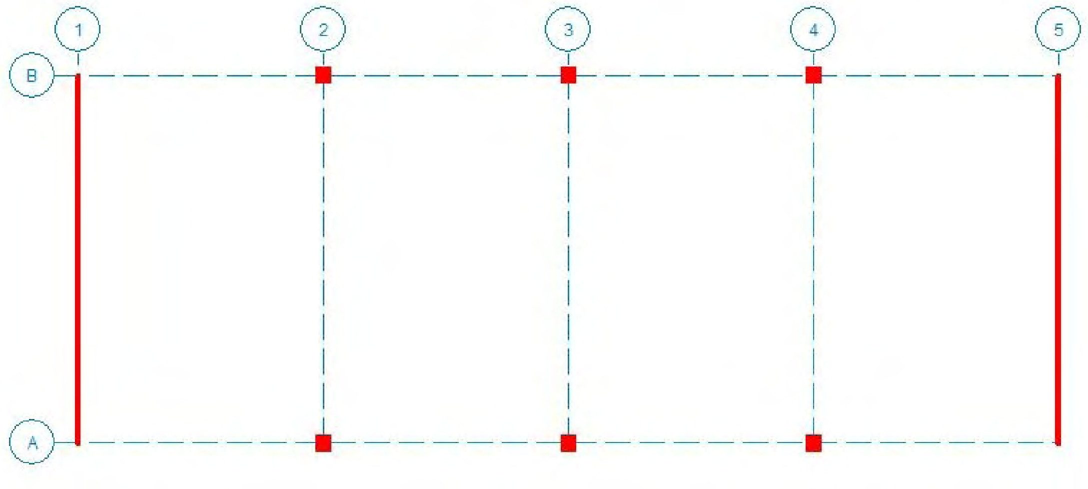
  Select: Wood, DF/S Pine, Rectangular, Lateral

  Box or click the following Grid Intersections:

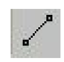
  A-2 B-2

  A-3 B-3

Your model should now look like the image below:



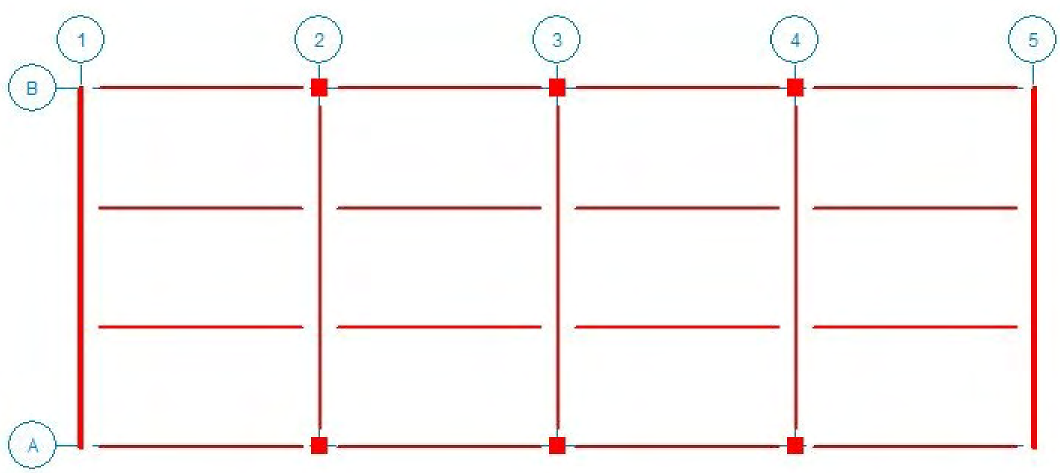
We will now draw the main Beams in the structure.

- Draw 'Lateral' Beams by clicking on the Draw Beam button 

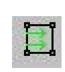
  Select: Wood, 24F-1.8E DF Balanced, Glulam_Western, Pinned-Pinned, Lateral

  Draw in the beams to match the picture below:

  (Note: Beams frame into 1/3 Pts)

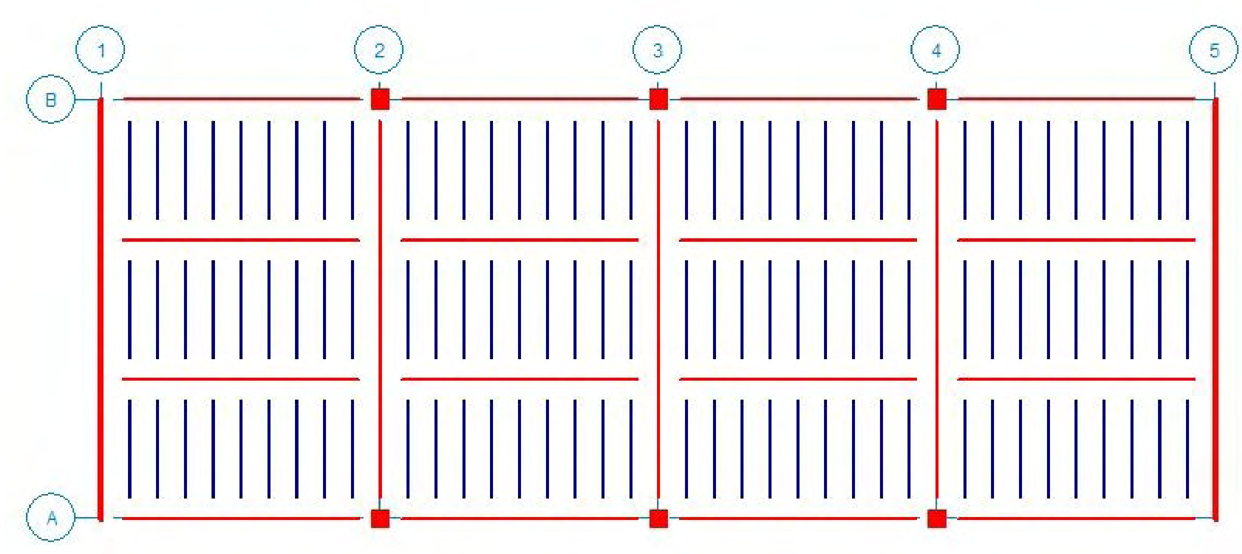


We will now draw the infill framing in the structure.

- Draw Vertical Infill Beams by clicking on the Generate Beams within a Bay button 

  Select: Wood, DF/SPine, Rectangular, Pinned-Pinned, Vertical

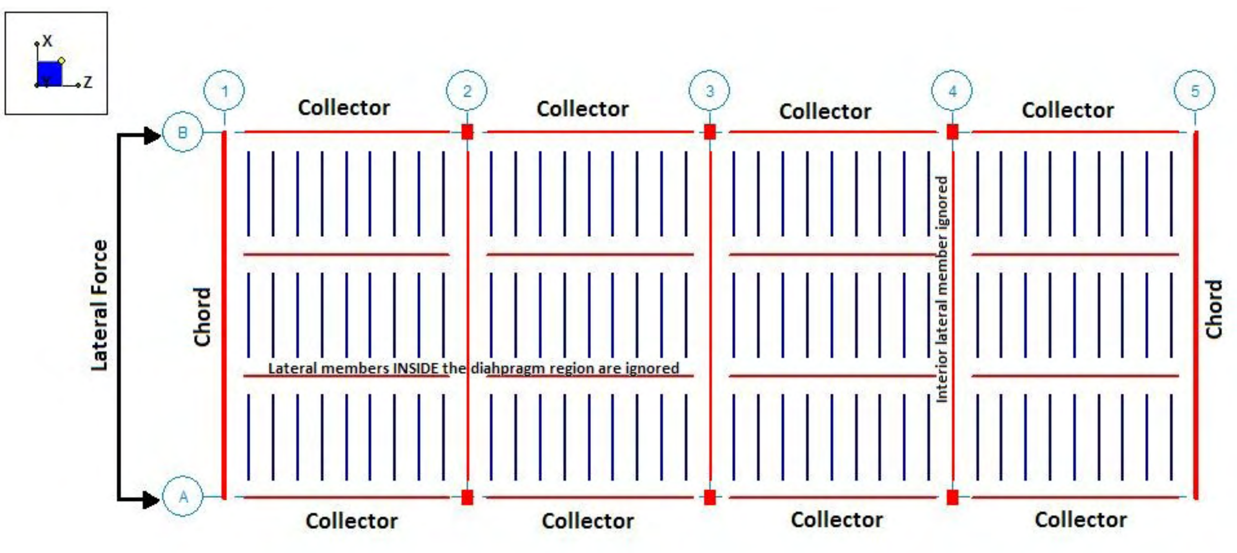
  Beam spacing not to exceed 2' o.c. oriented vertically



We'll keep this building simple with only one diaphragm for now. There are some rules for Flexible Diaphragms in RISAFloor:

1. The entire perimeter of the diaphragm region must consist of lateral members. All Chord and Collectors must be defined as lateral members.
2. Diaphragm regions are limited to only rectangular shapes, and must be oriented along the principal X and Z axes.
3. Lateral members that fall within a diaphragm region (as opposed to along its perimeter) will be ignored for diaphragm force distribution.

For Example:



Note: Rigid versus Flexible Diaphragms

- In our training we are focusing on flexible diaphragms, as this is the most used application for wood. Our wood diaphragm design is based on the assumption that the diaphragm is considered flexible, thus load attribution is done considering tributary area of lateral force resisting elements only.
- You can also define your diaphragms as rigid, thus your lateral loading in 3D will be defined only by the stiffness of the lateral force resisting elements. Defining a rigid diaphragm means that all of the joints at that diaphragm level will rotate and translate together. There is currently no design for rigid diaphragms in the program.

We will generate the slab perimeter, deck, and diaphragm region:

- Draw Slab by clicking on Create Slab Edges button

  Select 0 in Edge Overhang Distance, & Flexible

  Select Create a SLAB Perimeter- Consider ALL selected beams/Walls for the SLAB perimeter.

- Open the Design Rules spreadsheet- Diaphragm tab

  Rename the design rule to "Diaphragm 1"

  Define the following design rules:

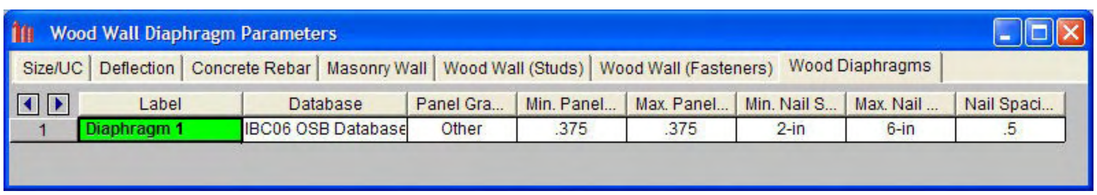
  IBC OSB Case 1_3 Blocked – Use Entire Case

  Panel Grade: Other

  Min Thickness: 0.375 Max Thickness: 0.375

  Min Nail Spacing: 2in Max Nail Spacing: 6in

  Nail Spacing Increment: .5in


Wood Wall Diaphragm Parameters

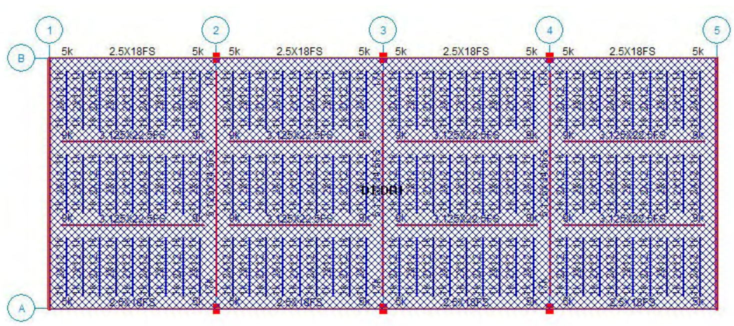
Size/UC | Deflection | Concrete Rebar | Masonry Wall | Wood Wall (Studs) | Wood Wall (Fasteners) | Wood Diaphragms

	Label	Database	Panel Gra...	Min. Panel...	Max. Panel...	Min. Nail S...	Max. Nail ...	Nail Spaci...
1	Diaphragm 1	IBC06 OSB Database	Other	.375	.375	2-in	6-in	.5

- Draw Diaphragm Region by clicking on Add New Diaphragm button, select Diaphragm1

  Click on the lower left corner grid intersection A-1 and then click the upper right corner grid intersection B-5.

- Now solve the model.



Now we will take this model into RISA-3D to complete the Lateral design:

- Click the Director button at the top right corner of the screen.
- Define the Wind Loads:

  Wind Code: ASCE 7-05
  Wind Speed: 110mph
  Importance Category: 2
  Exposure Category: B

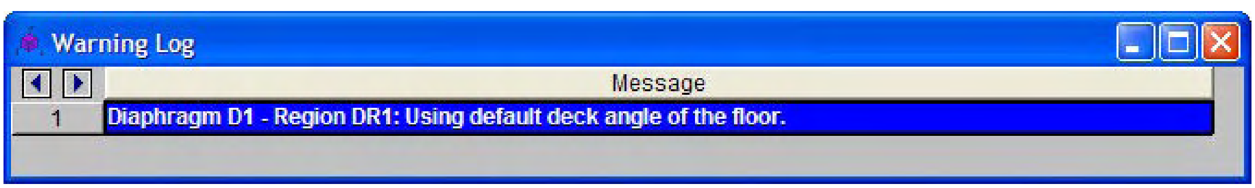
- Define the Seismic Loads:

  Seismic Code: ASCE 7 -05
  Ct: 0.02
  R: 6.5
  Ct Exp: 0.75
  S_D1: 0.43
  S_DS: 1
  S_1: 0.566

Now we will build load combinations and solve the model.

- Open the Load Combination Spreadsheet and generate Load Combinations based on the IBC 2006 ASD building code which include Wind Load X and Z, and Seismic X and Z.
- Solve the Envelope solution.

Note: A warning message will appear which indicates RISA-3D automatically uses the default deck if no deck was defined. The diaphragm strong direction is perpendicular to the deck angle defined in RISAFloor.

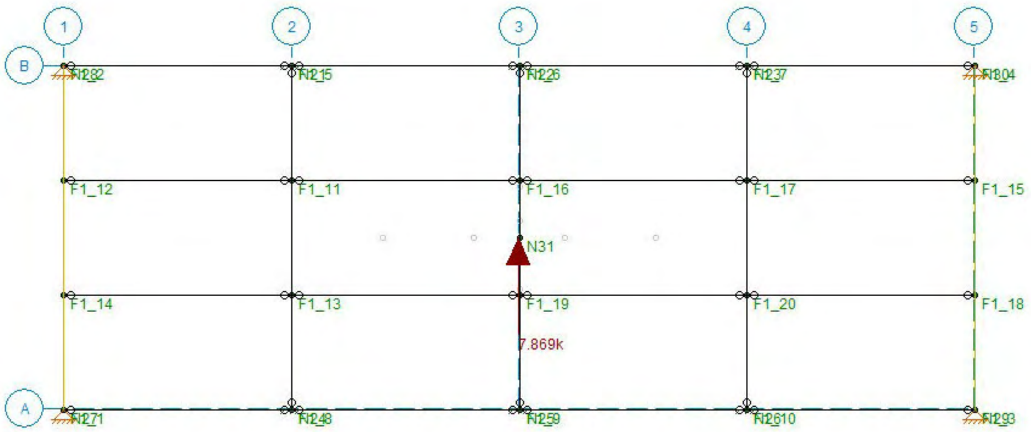


Let's review how lateral loads are distributed onto the model:

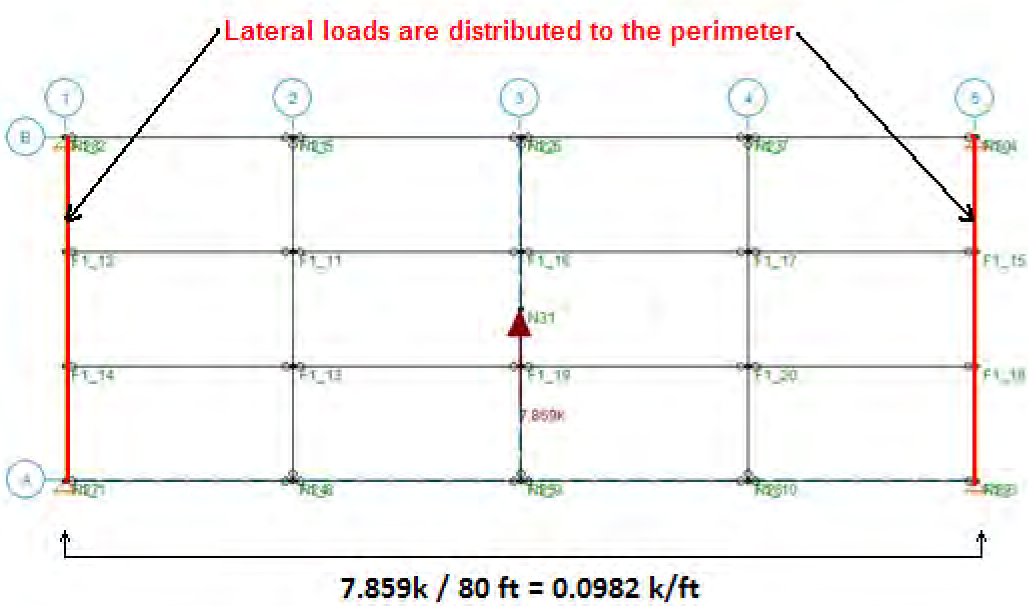
1. The Wind Load and Seismic Load Generators create a Point load in the X and Z directions which is applied to the each diaphragm.

   (In Rigid Diaphragm design, partial wind loads and eccentric seismic loads are also created as point loads).

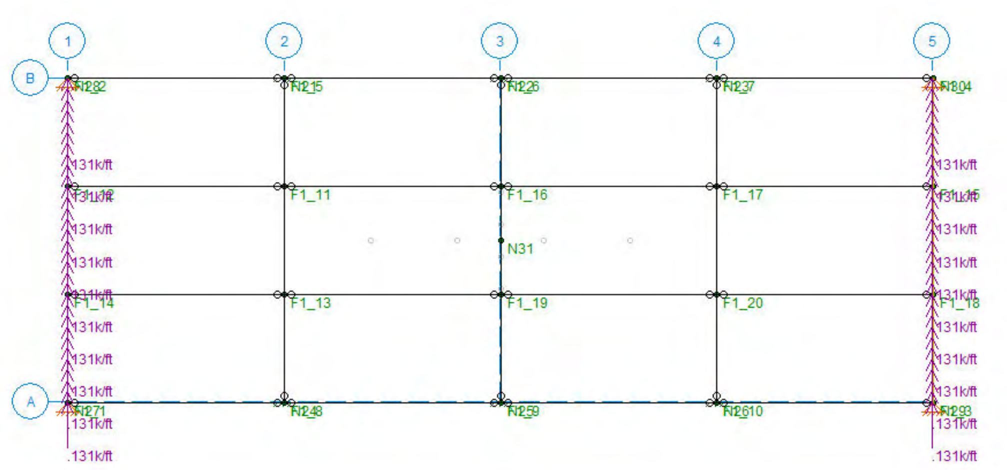
   The automated Wind Load X direction is a point load, shown below is BLC 8: WLX:



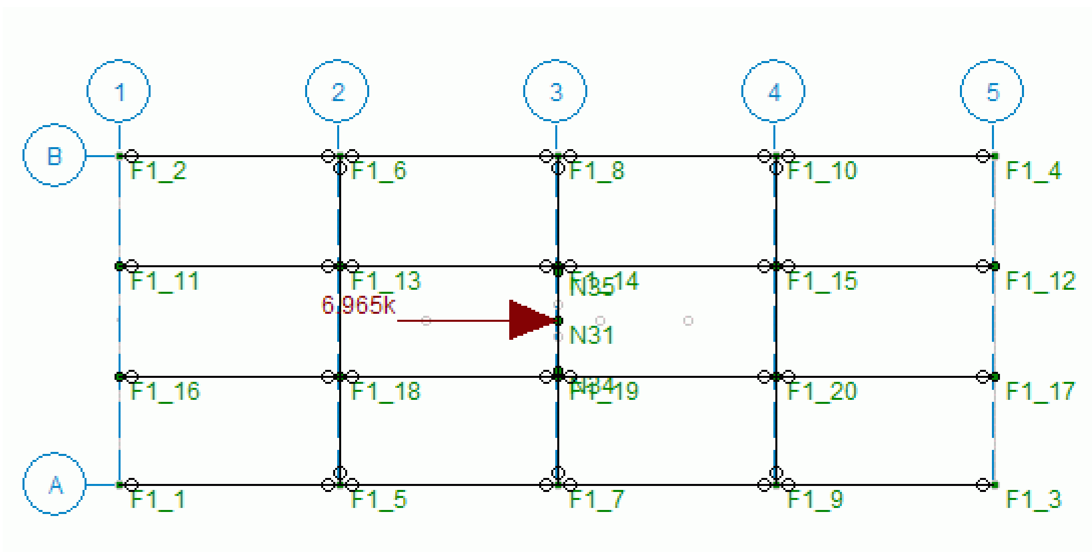
2. Internally the load is split up into a line load for wind and a surface load for seismic.
3. These loads are then attributed to the collector members that are tributary to those loads.



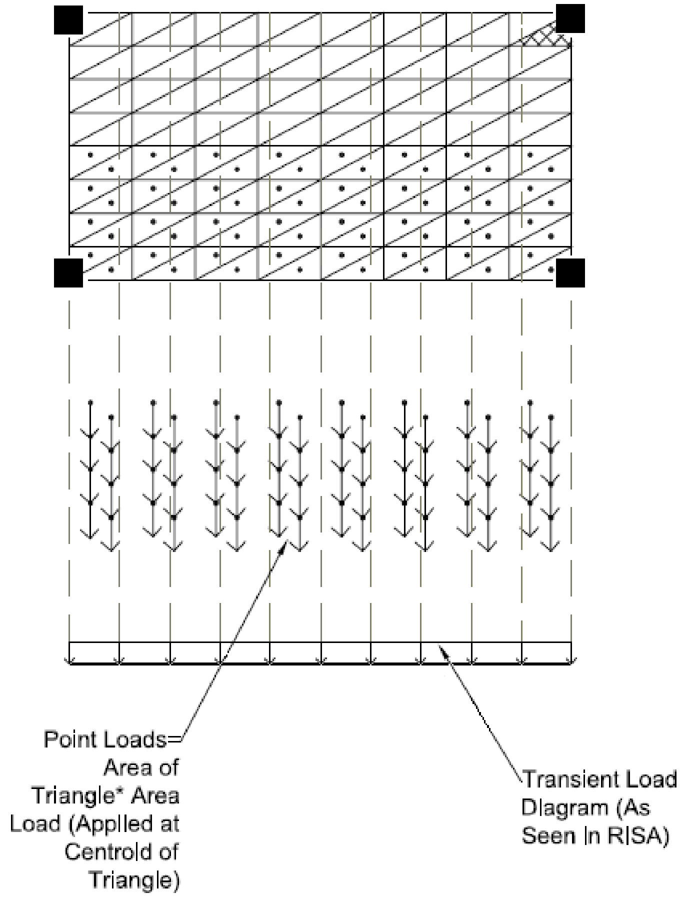
The Flexible Diaphragm converts the point load to a distributed load to the perimeter of the diaphragm region after solution shown below is BLC 24: BLC8 Transient Area Load:



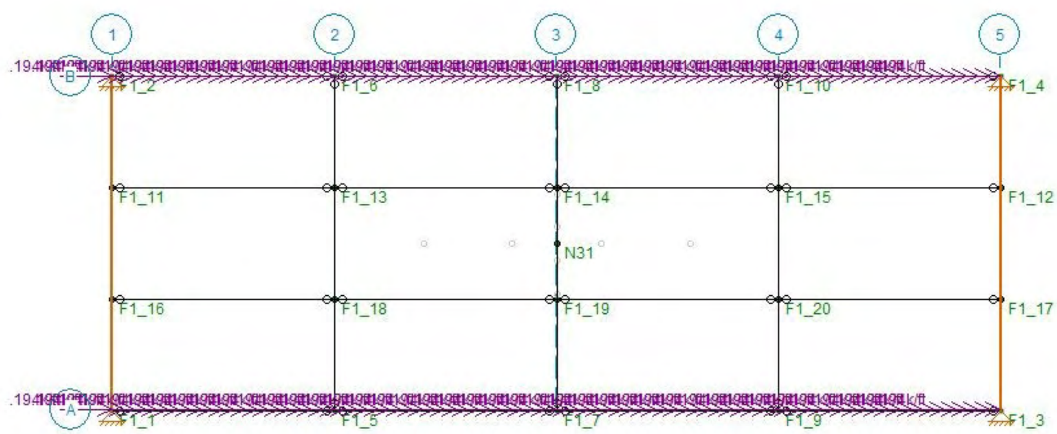
The automated Earthquake load Z direction is a point load, shown below is BLC 17: ELZ:



For the seismic load, the load is converted into an area load and then distributed to the collectors of the diaphragm according to tributary area. The diagrams below describe how the area is broken into small triangles. RISA then calculates point loads based on the magnitude of load multiplied by the area of the individual triangles. These point loads are then converted into a distributed load and displayed in the Transient load diagram.



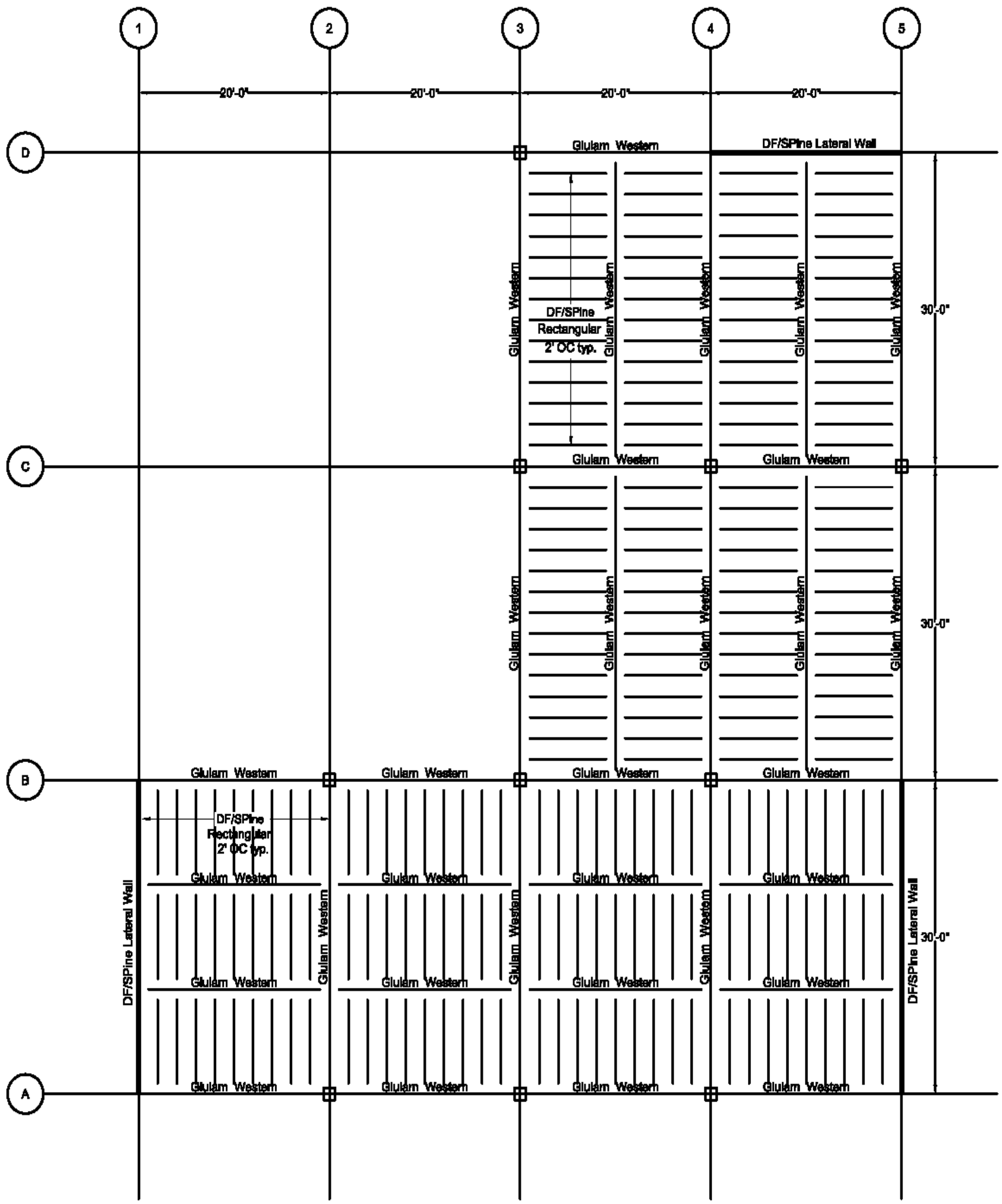
Shown below is BLC 27:BLC17 Transient Area Load ELZ:0



Let's review the lateral design:

- Select Results- Diaphragms

Floor Plan (Expanded)



We'll expand this simple model to see how multiple regions work.

Add to the Project Grid as follows:

- On the Vertical Project Grid- X Axis Tab

  Add the Start Label to 'C' and 'D' – 30 ft AFTER the current line,

We'll delete the existing slab edge before we start drawing the new section of the building:

- Click the Delete icon and choose to "Delete Slab/Opening Perimeters by Clicking Them"

  Click on the slab edge.

We will now draw the walls of the structure.

- Draw 'Lateral' Walls by clicking on the Draw Wall button 

  Select: Wood, DF/S Pine, Design Rule: Diaphragm 1, Lateral

  Draw between the following Grid Locations:

  D-4 & D-5

We will now draw the columns of the structure.

- Draw 'Lateral' Columns by clicking on the Draw Column button 

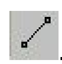
  Select: Wood, DF/S Pine, Rectangular, Lateral

  Box or click the following Grid Intersections:

  C-3 C-4

  C-5 D-3

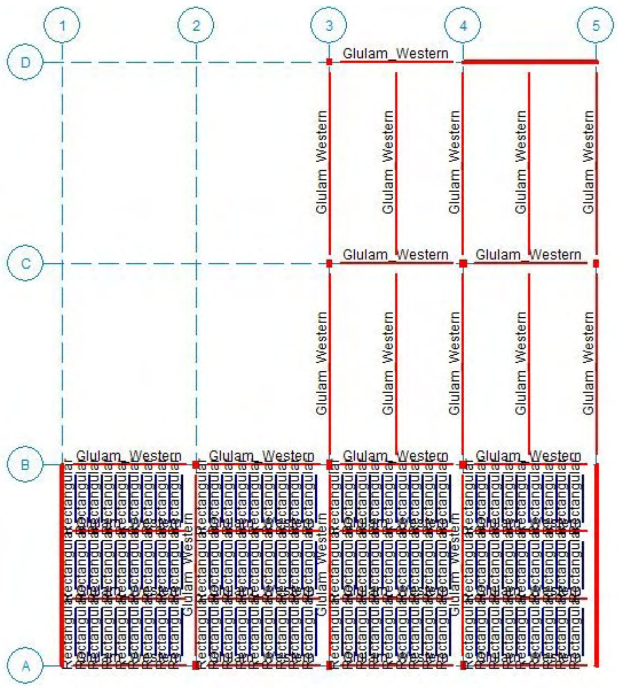
We will now draw the main beams in the structure.

- Draw 'Lateral' Beams by clicking on the Draw Beam button .

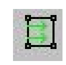
  Select: Wood, 24F-1.8E DF Balanced, Glulam_Western, Pinned-Pinned, Lateral

  Draw in the beams to match the picture below:

  (Note: Beams frame into 1/2 Pts)

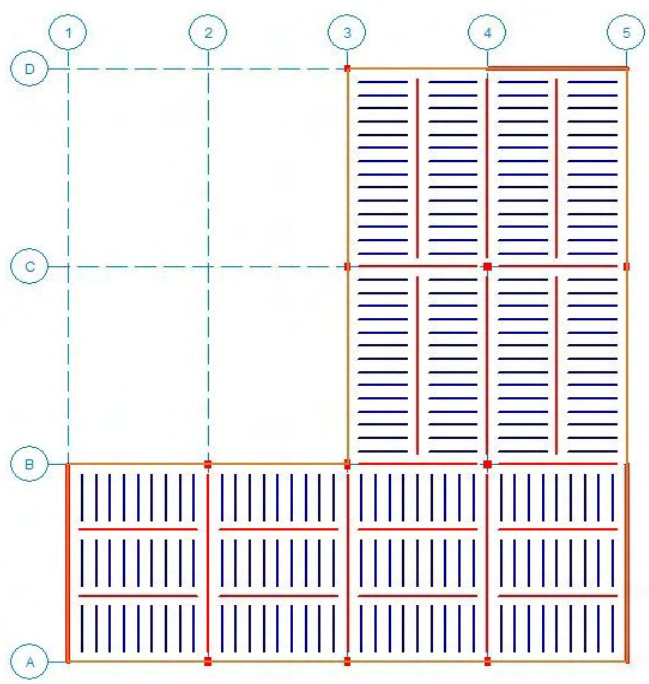


We will now draw the infill framing in the structure.

- Draw Horizontal Infill Beams by clicking on the Generate Beams within a Bay button 

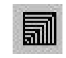
  Select: Wood, DF/SPine, Rectangular, Horizontal

  Beam spacing not to exceed 2’ o.c. oriented horizontally.



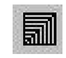
Now we will configure the slab and deck.

- Create an Edge Perimeter with a 0 inch Overhang Distance along the entire perimeter of the structure.

- Draw a 'Local' Deck/Slab by clicking on the "Assign Slab or Deck Properties" button 

  Select: Wood, oriented 'Parallel to Z Axis'

  Click grid intersection A-1- to B-5

- Draw a 'Local' Deck/Slab by clicking on the "Assign Slab or Deck Properties" button 

  Select: Wood, oriented 'Parallel to X Axis'

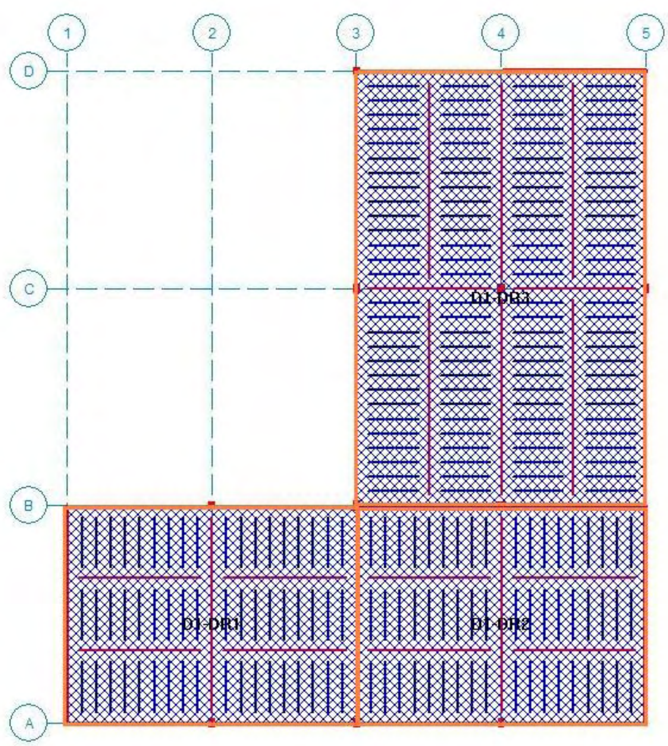
  Click grid intersection B-3- to D-3

- Change the Design Rules to IBC_06_OSB Case_5_6_Blocked, use Entire Case.
- Draw Diaphragm Region by clicking on Add New Diaphragm button – select Diaphragm 1

  Click grid intersection A-1- to B-3

  Click grid intersection A-3- to B-5

  Click grid intersection B-3- to D-5

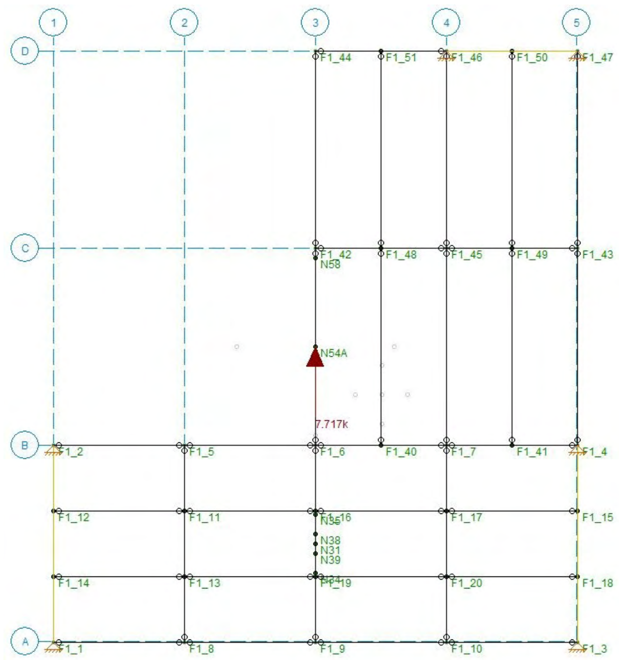


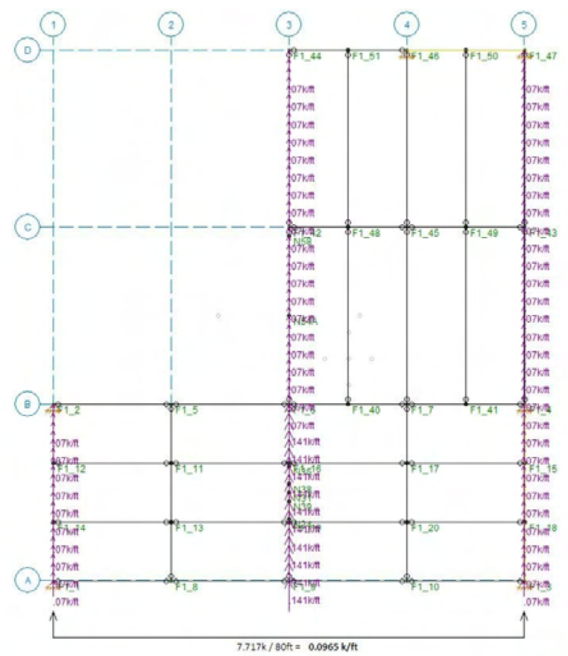
Now we will take this model into RISA-3D to complete the Lateral design:

- Now solve the model.
- Click the Director button at the top right corner of the screen.

Let's review the lateral loads:

The automated Wind Load X direction is a point load. The Point load is then converted into a Distributed load and applied to the lateral members on that perimeter of the diaphragm regions.



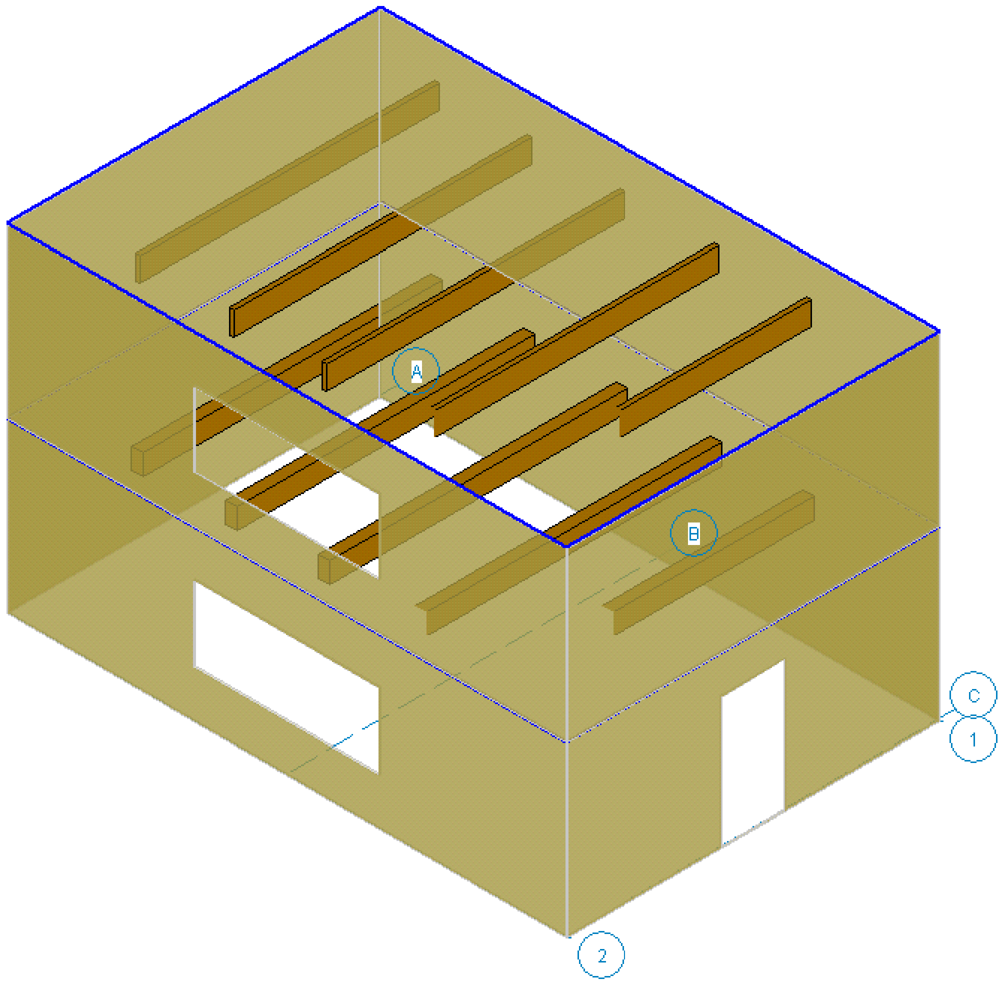


Let's review the lateral design:

- Select Results- Diaphragms

Stacked Walls

This model is intended to demonstrate modeling of stacked wall systems.

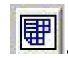


Project Definition: Stack.rfl

2- Story – 18 feet total height
Wood Beams –DF/SPine No.1, Rectangular
Walls – 2x6 Wood, DF/SPine , Design Rule: Typical

Modeling Procedure

We'll begin by creating a new Floor and generating a Project Grid.

- Start a new model by clicking on the Create New Floor Plan button .
- Create an Original Floor with an elevation of 9 feet.

  Ignore the Default Area Load, Deck, and Angle entries as we will revise these later.

Define a Project Grid as follows:

- On the Horizontal Project Grid – "Z Axis" Tab

  Start Label of "1" and Start Location of "0".

  Enter "20" for the Increments.

  Click "Generate."
- On the Vertical Project Grid- "X Axis" Tab

  Set the Start Label to "A" and Start location to "0",

  Enter "2@15" for the Increments.

  Click "Generate."

We will now draw the Walls of the structure.

- Draw lateral walls by clicking on the Draw Wall button .

  Select: Wood, DF/SPine, Design Rule: Typical, Lateral.

  Draw between the following Grid Locations:

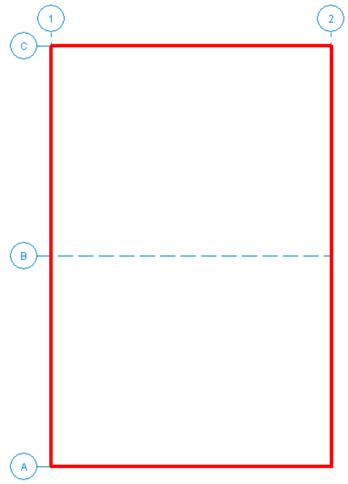
  A-1 and A-2

  A-2 and C-2

  C-2 and C-1

  C-1 and A-1

Your model should now look like the image below:



Now we will add openings to the walls.

- Double-click on the wall panel along grid line C.

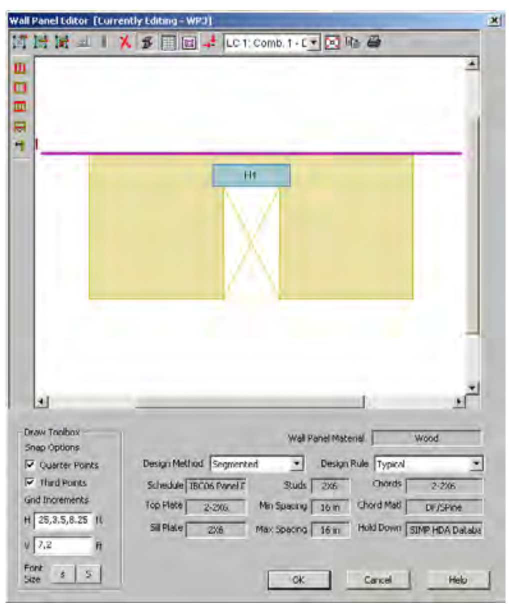
  For the "Grid Increments", set H to "8.25,3.5,8.25" and V to "7,2".

Now we can use the coordinates in the lower right hand corner to locate where we are at the wall. Each grid intersection will snap to a specific coordinate.

- Click the "Draw New Openings" icon in the Wall Panel Editor.

  For the doorway, click at (30,0,11.75) and (30,7,8.25).

Notice that header beams are shown graphically. The wall panel editor should look like this:



In order to get results in from the program we now need to define regions for our wall panel.

There are two icons for laying out regions and . The first allows the user to manually define regions, while the second creates regions automatically.

Now you will see that within the wall panel editor regions now make up the entire wall. Also, the potential framing is laid out for you based on your default design rules. There are toggles on the left side of the editor to shut off the graphical view of this framing.

Now let's define our design method.

RISA has included three options for wood wall panel design: Segmented, Perforated, and Force Transfer Around Openings.

- Choose "Segmented" in the drop down list for Design Method.

  Press "OK".

> **Note:** In RISAFloor, the different design method options are not considered because they are shear wall design methods. Because RISAFloor only considers gravity loads, then the wall studs are simply checked for axial compression. All other design considerations are taken into account in RISA-3D.

- Double click on the wall panel along grid line 2 which will open the Wall Panel Editor.

  For the "Grid Increments," set H to "3@10" and V to "2.5,4,2.5"

  Draw a window, clicking at (10,2.5,20) and at (20,6.5,20).

  Click the "Generate Wall Regions Automatically" icon .

  Choose "Segmented" in the drop down list for Design Method.

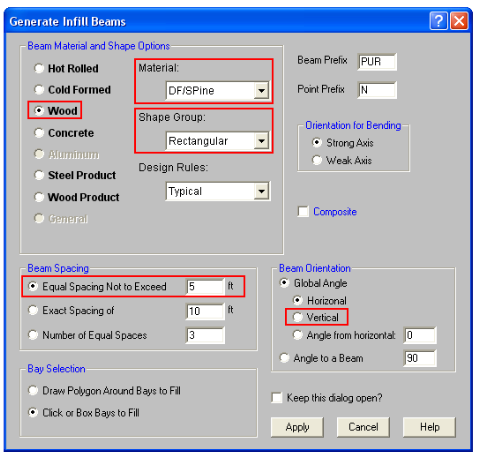
  Press "OK".

Now we will model the interior bay framing within the structure.

- Draw horizontal Infill Beams by clicking on the "Generate Beams within a Bay" button .

  Select: Wood, DF/SPine, Rectangular, Horizontal.

  Select Beam Spacing Not to Exceed 5' O.C.



- Click inside of the two grid squares to create our framing.

Now we will generate the deck/slab perimeter.

- Draw Slab by clicking on Create Slab Edges button .

  Enter an Edge Overhang Distance of "0" inches.

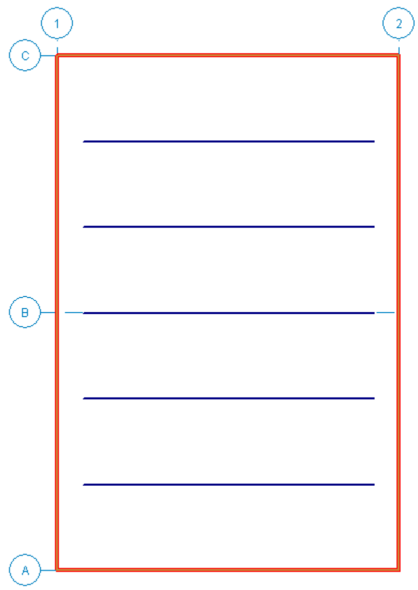
  Choose "Flexible" as the Diaphragm Type.

  Click on "Create a Slab Perimeter,"

  Click on "Consider ALL selected Beams/Walls" for the SLAB perimeter,

  Press Apply.

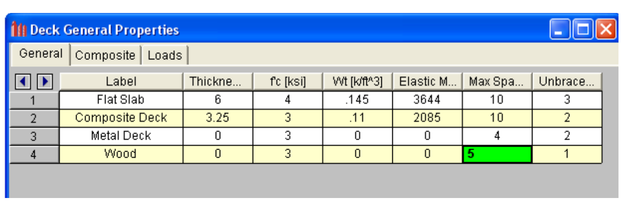
Your model should look like the image below (the dark blue outlines the slab edge; because of the zero inch slab edge, the graphic for the edge and the wall panels overlap):



Now we will model the deck properties.

- Click the Deck Definitions button .

- Set the Max Span for "Wood" to 5 ft.



Deck General Properties

General | Composite | Loads

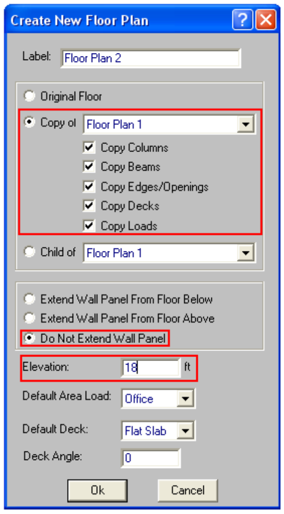
	Label	Thickne...	f'c [ksi]	Wt [k/ft^3]	Elastic M...	Max Spa...	Unbrace...
1	Flat Slab	6	4	.145	3644	10	3
2	Composite Deck	3.25	3	.11	2085	10	2
3	Metal Deck	0	3	0	0	4	2
4	Wood	0	3	0	0	5	1

- Click on the Floors Spreadsheet on the Data Entry Toolbar.

  For "Floor Plan 1" verify the Area Load Default is set to "Office," and modify the Deck Default "Wood," and the Deck Angle to "90" degrees.

Let's add the second floor:

- From the Insert pull down menu, select "Floor"
- Create a Copy of Floor Plan 1 with an elevation of 18 feet and Do Not Extend the Wall Panel.
- Ignore the Default Area Load, Deck, and Angle entries as we will revise these later.



We will now draw the Walls on the Second Floor.

- Draw lateral walls by clicking on the Draw Wall button .

Select: Wood, DF/SPine, Design Rule: Typical, Lateral.

Draw between the following Grid Locations:

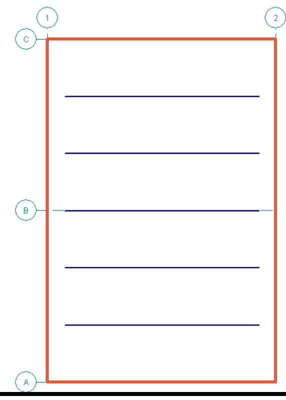
A-1 and A-2

A-2 and C-2

C-2 and C-1

C-1 and A-1

Your model should now look like the image below:



Note: When you draw the walls in separately for each level, the program treats the walls as two separate panels. This means you must connect these using wall straps at each level. We must wait to define the straps once we take the model into RISA-3D, as we are not allowed to model these in RISAFloor. The straps are the means by which load is transferred between two stacked walls.

Now we will add openings to one of the walls.

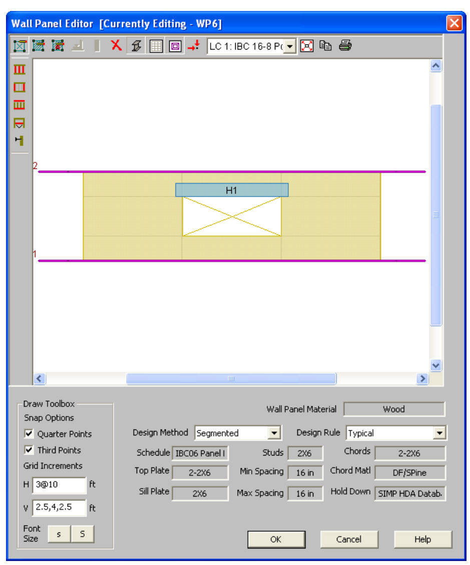
- Double click on the wall panel along grid line 2 which will open the Wall Panel Editor.
- For the "Grid Increments," set H to "3@10" and V to "2.5,4,2.5"

Now we can use the coordinates in the lower right hand corner to locate where we are at the wall. Each grid intersection will snap to a specific coordinate.

- Click the "Draw New Openings" icon in the Wall Panel Editor.

  For the window, click at (10,11.5,20) and at (20,15.5,20).

- Notice that header beams are shown graphically. The wall panel editor should look like this:



In order to get results in from the program we now need to define regions for our wall panel.

- Click the "Generate Wall Regions Automatically" icon .

Click OK to close the Wall Panel Editor.

Now we will modify the gravity loading.

- Click the Area Load Definitions button Area Load Definitions.

Add a line to the bottom of the spreadsheet by pressing "Enter" on the last line.

Input values for roof loading as seen below:

Uniform Area Loads

Standard | Other

	Label	Addit...	PreDL[ksf]	PostDL[...	LL[ksf]	LL Type	VL[ksf]	Dyn Loa...
1	Office	☐		.01	.08	LL-Reduce	.011	.075
2	Storage	☐		.01	.125	LLS-Non	.011	.175
3	Public	☐		.01	.1	LL-Non	.004	.075
4	Add Piping	☑		.02		LL-Non	.011	
5	Roof	☐		.01	.02	RLL-Non		

- Click on the Floors Spreadsheet on the Data Entry Toolbar.

- For "Floor Plan 2" change the Area Load Default to "Roof," the Deck Default to "Wood," and the Deck Angle to "90" degrees.

Floors

	Label	Elevatio...	Area Load ...	Deck Defa...	Deck Angl...	Parent	Inacti...	Splic...	Splic...
1	Floor Plan 1	9	Roof	Wood	0	None	☐	9	Moment

Now we will build load combinations and solve the model.

- Generate Load Combinations based on the IBC 2006 ASD building code which include Roof Live Load (RLL) but does not include Pre-Composite combinations.

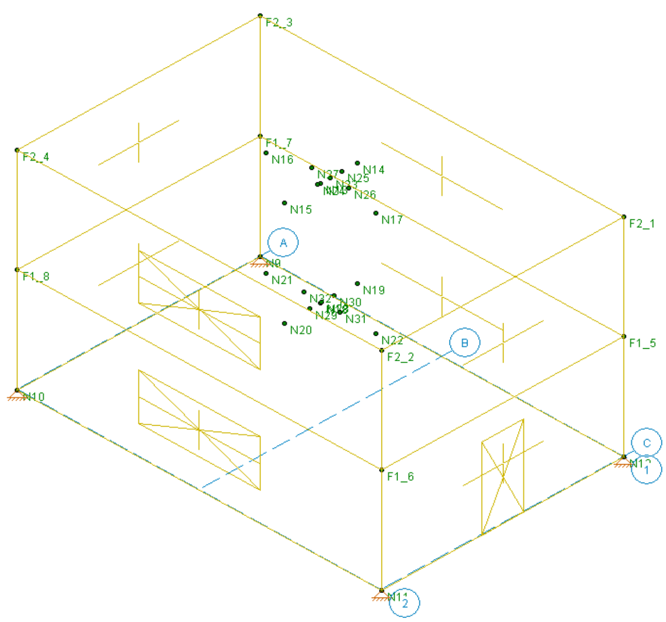
- Solve the model and review the results.

Now let's take the model into RISA-3D to perform the Lateral Design.

- Click on the Director button and select RISA-3D.

- Click OK for both the Wind and Seismic Load dialogue boxes.

Your model should look like this in RISA-3D:



Now let's add Straps to the Shear Walls.

- Double click on the second story wall panel along Gridline A to open the Wall Panel Editor.

  Click on the Add New Wood Straps option and add a strap to the bottom corners of the wall.

  Repeat for the second story wall panel along Gridlines 1 and C.

- Double click on the second story wall panel along Gridline 2 to open the Wall Panel Editor.

  Click on the Add New Wood Straps option and add a strap to the bottom corners of each region.

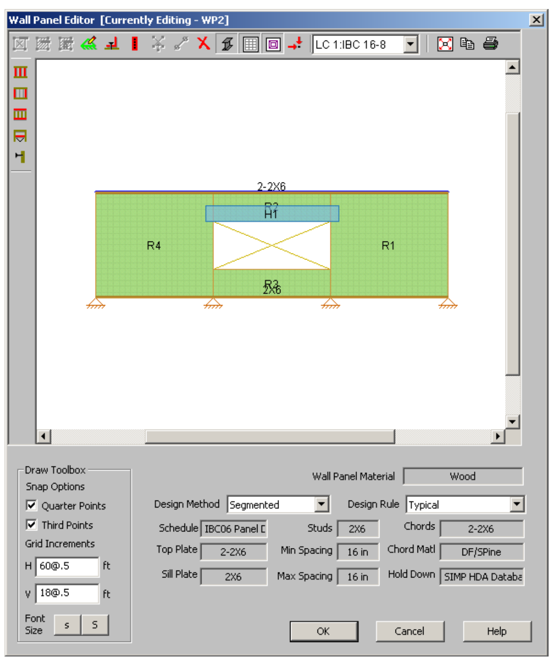
> **Note:** For Segmented walls that are stacked with openings, it is necessary to add Straps at the locations where you'd typically apply Hold-downs; at the bottom of the wall on each region around the opening. Stacked walls panels using the Segmented design require the openings in the wall panels to be aligned on both sides of the opening. If this criteria is not satisfied, design will not be done on these walls. This is not a requirement for Perforated or FTAO walls.

Now let's add Hold downs to the Shear Walls.

> **Note:** RISA-3D auto-create hold-downs at the corners of wall panels at the lowest level in the structure. Thus, if running Perforated or FTAO walls, no hold-downs would need to be added. For Segmented design the only hold-downs that need to be added are the ones that align with the openings inside the wall panels.

- Double click on the first story wall panel along Gridline 2 to open the Wall Panel Editor.

  Click on the Add New Wood Hold Downs option and add a hold down to the bottom corners of each region.



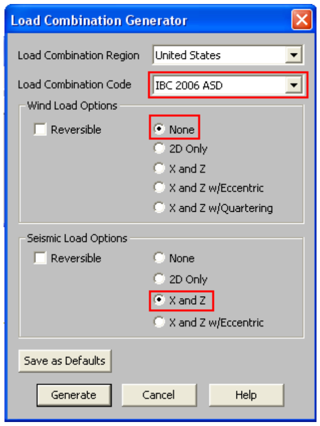
- Double click on the first story wall panel along Gridline C to open the Wall Panel Editor.

  Click on the Add New Wood Hold Downs option and add a hold down to the bottom corners of each region.

Now we will build load combinations and solve the model.

- Open the Load Combinations spreadsheet and click on the Generate option.

    Generate Load Combinations based on the IBC 2006 ASD building code. Include X and Z Seismic Loads and No Wind Loads.



Now let's solve the model and look at the results

- Run a Batch Solution of load combinations 17 and 18

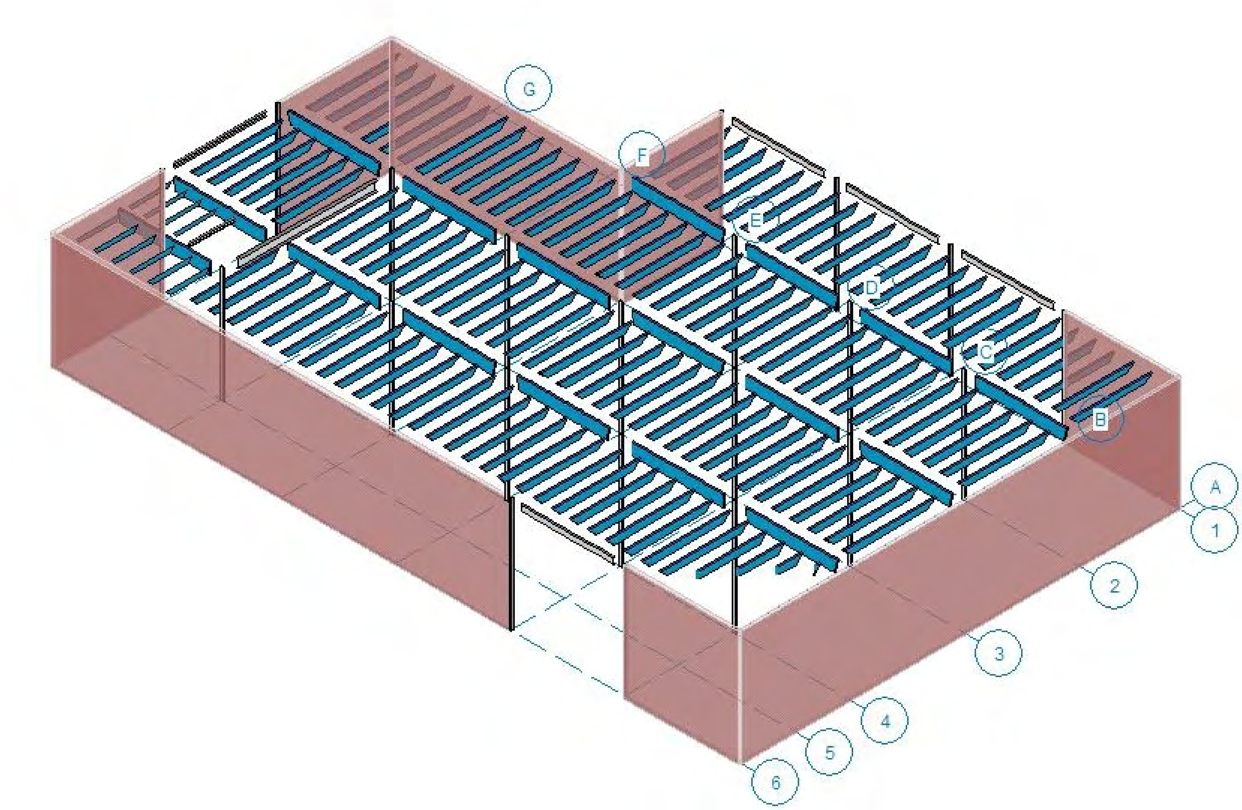
    Look at the "Wall" portion of the detail report for the 2nd story wall panel on Grid 2.

Here we see the forces for each of the straps and which load combination caused those forces.

Notes:

Warehouse – Steel Conversion

This model is intended to demonstrate how to convert a Steel model into a Wood model.



Project Definition: **Masonry Warehouse-Steel.rfl**

Single Story (Roof) – 25 feet

Dead Load (Superimposed) – 10 psf
Base Snow Load – 40 psf

Masonry Exterior walls.

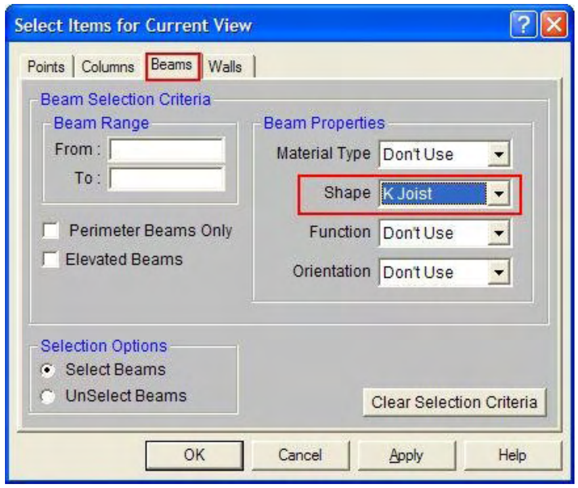
Modifications:
Metal Deck --> Wood Decking

Steel Joists at 4' o.c. with Joist Girders --> Glulam Wood framing 2' o.c.

Steel columns and beams --> Glulam Beams and Posts

We'll follow these easy steps to convert this steel framed building to a wood building:

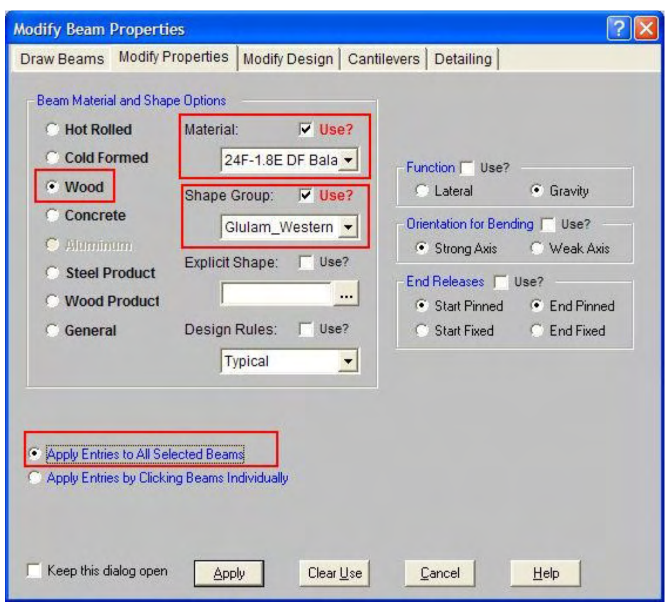
- Open the file 'Masonry_Warehouse-Steel.rfl' from the Training CD.
- Unselect the entire model and select only the K Joists using the Select based on Criteria.



- Delete all the K-joists using the "Delete Selected Beams".

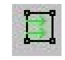
- Modify all remaining beams using the Modify Beams tool.

  Select: Wood ,24F-1.8 DF Balanced, Glulam_Western.



- Convert all Steel columns using the Modify Columns tool to Wood Posts

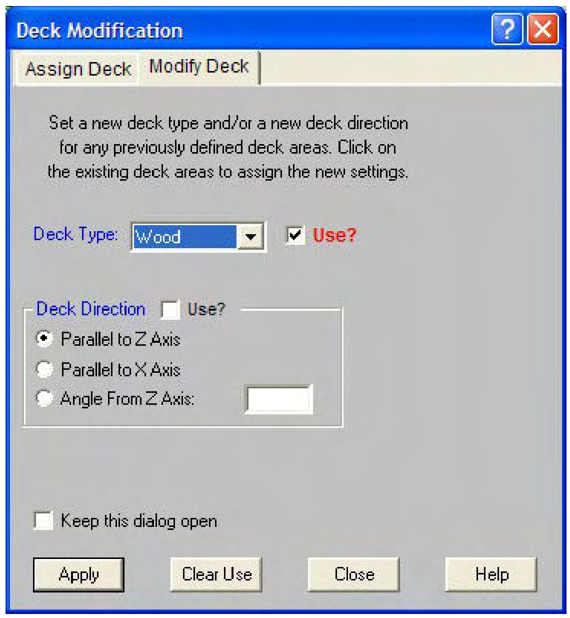
  Select: Wood, DF/SPine, Posts

- Draw Horizontal Infill Beams by clicking on the Generate Beams within a Bay button 

  Select: Wood, 24F-1.8 DF Balanced, Glulam_Western, Pinned-Pinned, Horizontal

  Beam spacing not to exceed 2' o.c. oriented

- Modify the Deck from "Metal Deck" to "Wood" :



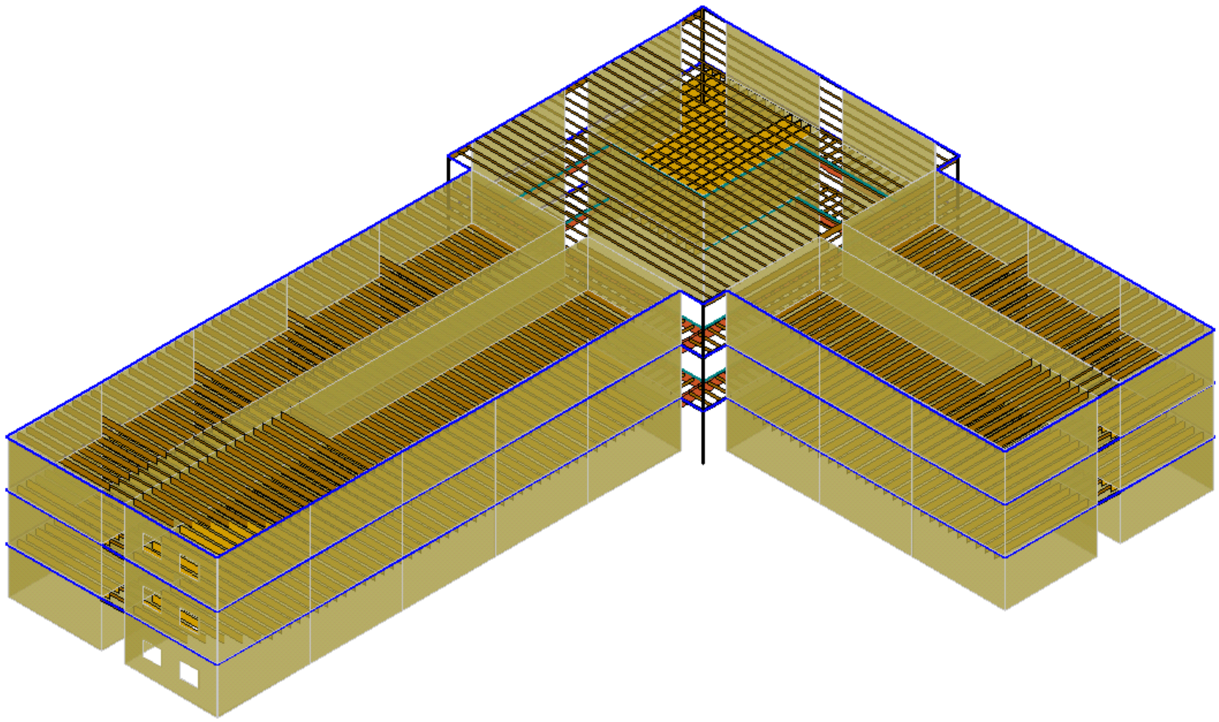
- Now solve the model.

Notes:

Comprehensive Review

This model is intended as a review of most of the topics that have been covered. It should not be difficult for users who have completed the preceding portions of this manual. This model is meant to represent a real-life wood framed building.

Project Definition: Hotel_9.rfl

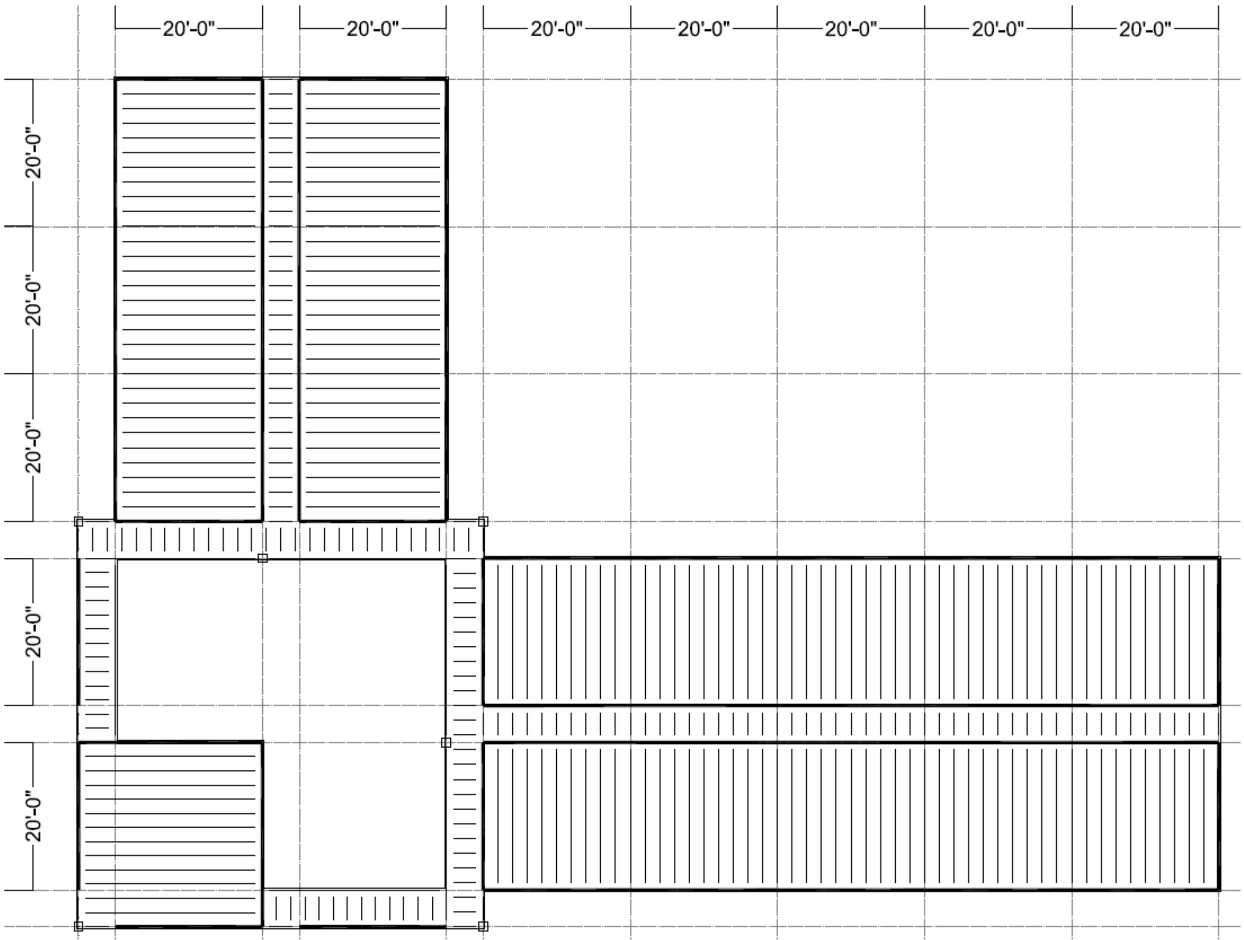


Three Stories – 30 feet (3 @ 10'-0")

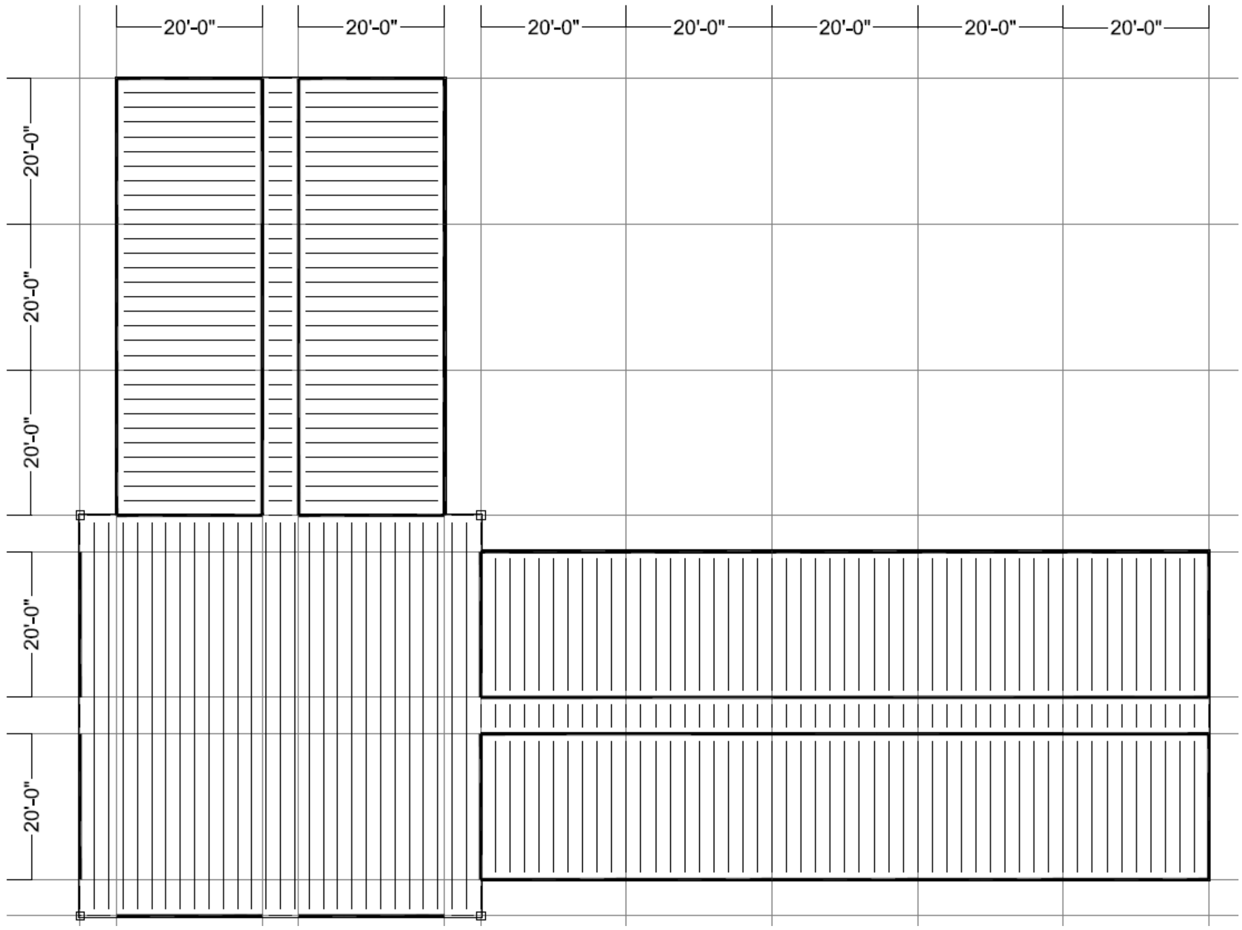
2nd/3rd Floor:
Dimension Lumber Framing (Southern Pine)
Wood Structural Panel Deck (One Way Span)
Dead Load (Superimposed) – 10 psf
Live Load (Reducible) – 40 psf / 100 psf

Roof:
Manufactured Wood Joists
Wood Structural Panel Deck (One Way Span)
Dead Load (Superimposed) – 10 psf
Roof Live Load (Non-Reducible) – 25 psf

Framing Plan: 2nd / 3rd Floors



Framing Plan: Roof



Modeling Procedure

Start by reviewing the Global Parameters

It is always important to review the global parameters whenever starting a model.

- Click on the Global Parameters button on the RISA Toolbar.
- Click on the "Solutions" tab.

  Select "Shear Deformation" On

  Select "Use Column Stiffness" Off

  Select "Show Factored End Reactions" Off

  Select "Sparse Accelerated" Solver
- Click on the "Codes" tab and verify we have the correct default codes selected.
- Click on the "Wind" tab

  Select Wind Code "ASCE 7-05"

  For Wind Speed enter 90 mph

  Select Occupancy Category 2

Select Exposure Category C

For the Directionality Factor (Kd) enter 1

- Click on the Seismic Tab

Select Seismic Code “ASCE 7-2005”

For Ct(X) enter 0.02

For Ct(Z) enter 0.02

For R(X) enter 6.5

For R(Z) enter 6.5

For S_D1 enter 0.505

For S_DS enter 0.937

For S_1 enter 0.505

Next define your area loads

We will set up our pre-defined area loads ahead of time to save time later when creating the model. The PreDL load only applies to composite steel, and the VL (vibration load) also only applies to steel.

- Open the “Area Load Definitions” spreadsheet and create your corridor, room, and roof loads as shown below

Uniform Area Loads

Standard | Other

	Label	Addit...	PreDL[ksf]	PostDL[...	LL[ksf]	LL Type	VL[ksf]	Dyn Load[ksf]
1	Corridor	☐		.01	.1	LL-Reduce		.025
2	Room	☐		.01	.04	LL-Reduce		.025
3	Roof	☐		.01	.025	RLL-Non		.01

- Close the Area Loads spreadsheet

Create the 2nd Floor

- Click on the Insert Menu and select “Floor”

For Label enter ‘2nd Floor’

For Elevation enter 10 ft

Select Default Area Load “Room”

Select Default Deck “Wood”

Click OK to create the floor.

Create the Project Grids

The Project Grid dialog should automatically appear after creating the 2nd Floor.

- On the Horizontal Project Grid- Z Axis Tab

Start Label of ‘1’ and Start Location of ‘0’

Enter ‘5, 20, 5, 20, 5, 5@20’ for the Increments

Click ‘Generate’.

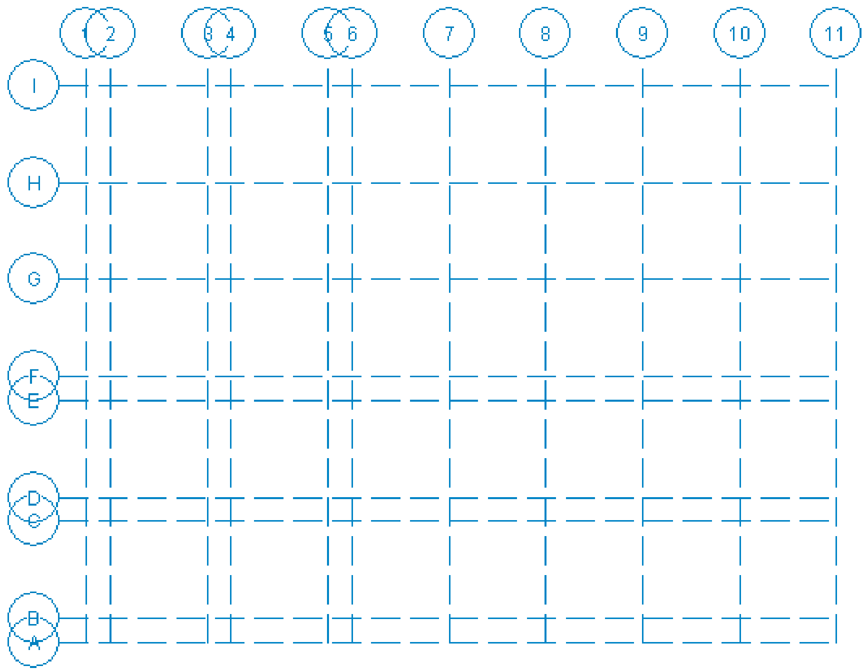
- On the Vertical Project Grid- X Axis Tab

Start Label of ‘A’ and Start Location of ‘0’

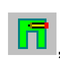
Enter ‘5, 20, 5, 20, 5, 3@20‘for the Increments

Click Generate

Your grids should look like this:



Draw the Walls and Columns

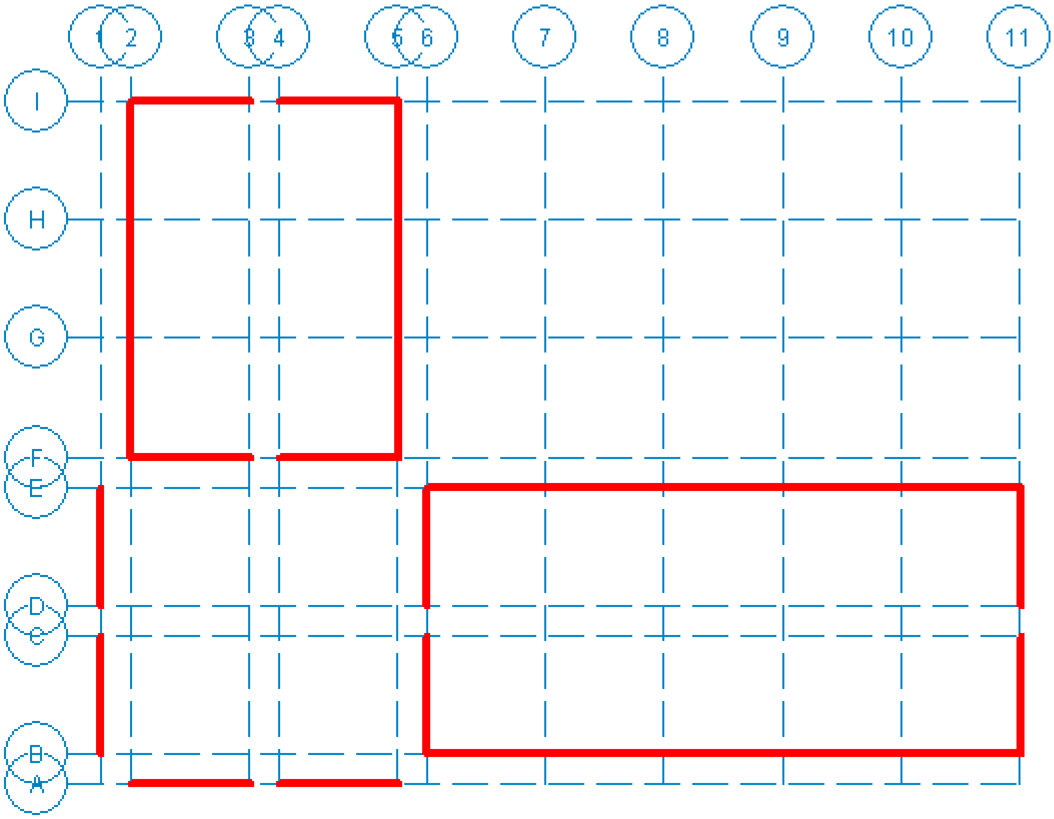
- Click on the “Draw Wall Panels” button , from the ‘Base’ level to the ‘2nd’ Floor’

Select Material: Wood, DF/SPine, Lateral

- Click “Apply” to start drawing wall panels

Drawing the walls as individual segments can make it easier to model openings, hold-downs, etc.

- Draw the wall panels as shown below. Draw the long walls as individual segments between gridlines.

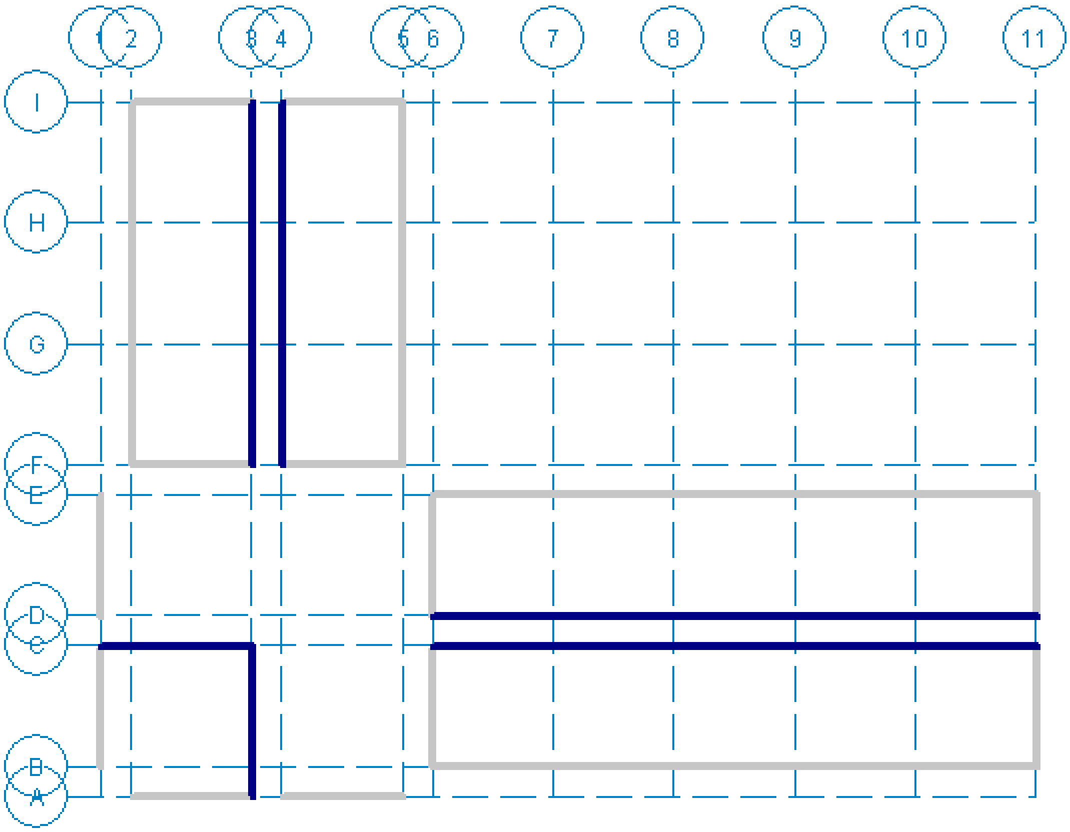


- Click on the "Draw Wall Panels" button from the 'Base' level to the '2nd Floor'

  Select Material: Wood, DF/SPine, Gravity

- Click "Apply" to start drawing wall panels

- Draw the wall panels as shown in bold below.



- Draw 'Lateral' Columns by clicking on the Draw Column button 

    Select: Wood, DF/SPine, Rectangular, Lateral

    Box or click the following grid Intersections:

        A-1, A-6, F-1, F-6

- Draw 'Lateral' Columns by clicking on the Draw Column button 

    Select: Wood, DF/SPine, Rectangular Double, Gravity

    Box or click the following grid Intersections:

        C-5, E-3

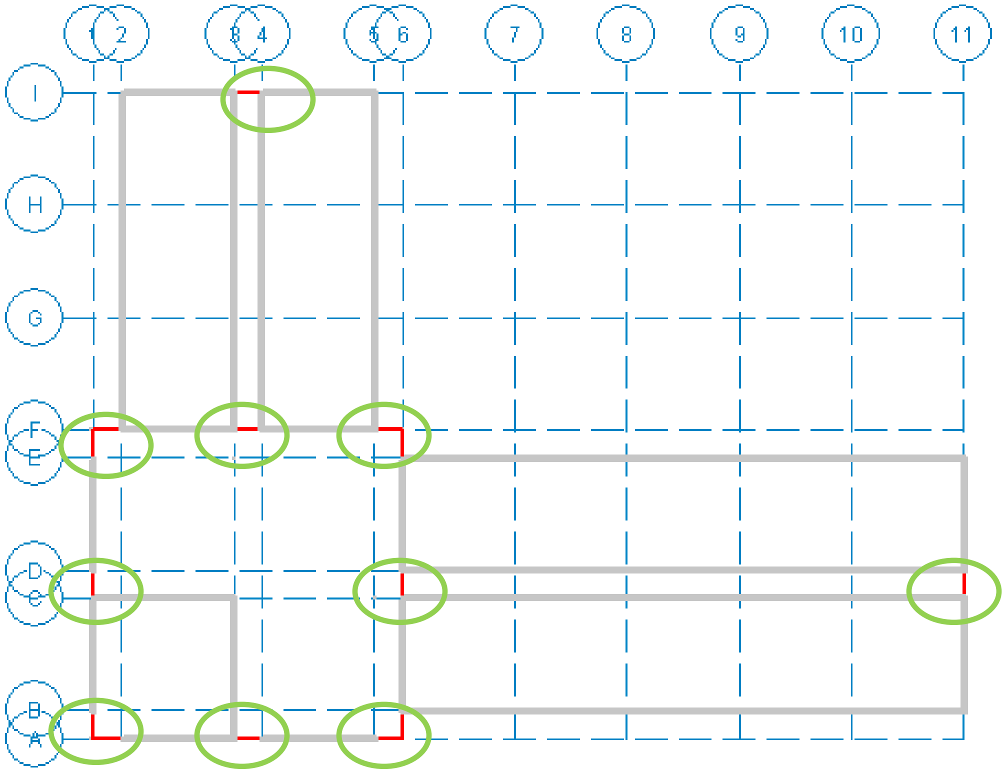
Draw the Beams

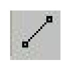
We will start by drawing some lateral beams that will act as chords/collectors to form our closed-circuit around the diaphragms that we will define. Then we will add the gravity framing.

- Draw 'Lateral' Beams by clicking on the Draw Beam button 

    Select: Wood, DF/SPine, Rectangular, Pinned-Pinned, Lateral

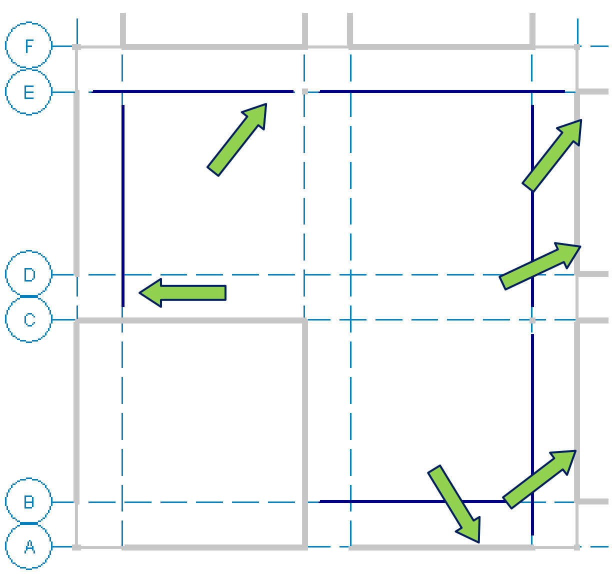
- Draw the beams in the locations circled below:



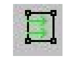
- Draw ‘Gravity’ Beams by clicking on the Draw Beam button 

    Select: Wood, 24F-1.8E SP Balanced, Glulam_Southern Pine, Pinned-Pinned, Gravity

    Draw the beams in the locations indicated by arrows below:



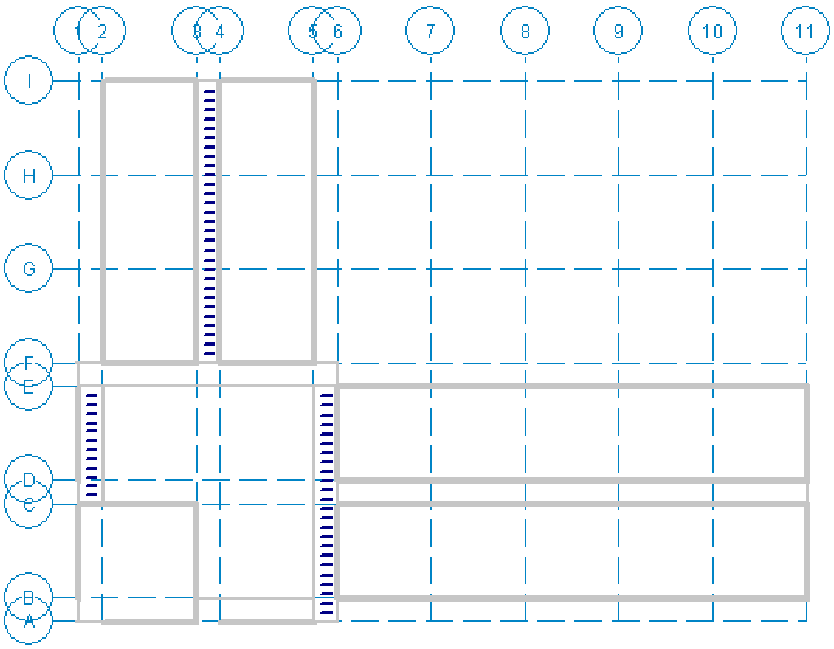
Next we will generate all of the infill framing.

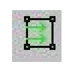
- Draw Horizontal Infill Beams by clicking on the Generate Beams within a Bay button 

  Select: Wood, DF/SPine, Rectangular, Pinned-Pinned, Horizontal

  Beam spacing not to exceed 2’ o.c. oriented horizontally

  Click within the regions to generate the framing   as shown in bold below:

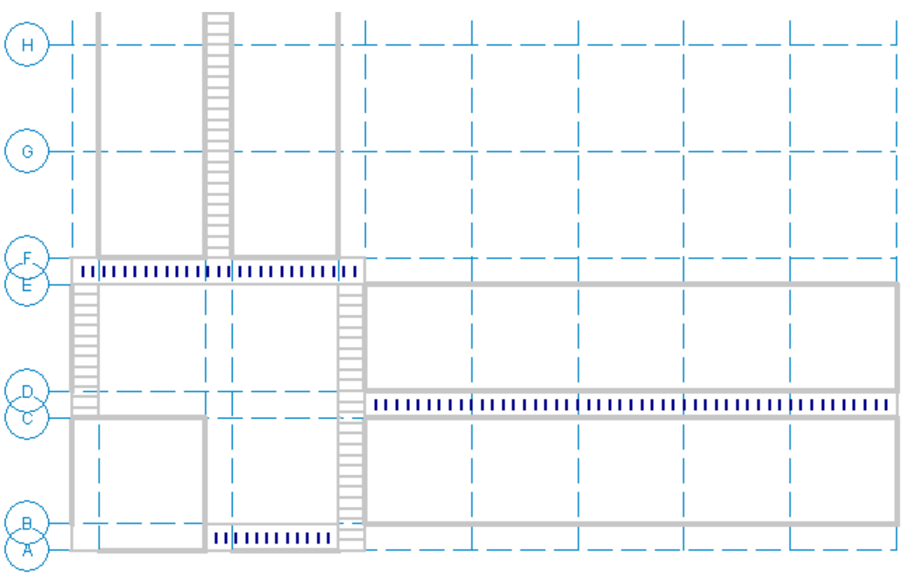


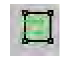
- Draw Vertical Infill Beams by clicking on the Generate Beams within a Bay button 

  Select: Wood, DF/SPine, Rectangular, Pinned-Pinned, Vertical

  Beam spacing not to exceed 2’ o.c. oriented vertically

  Click within the regions to generate the framing as shown in bold below:

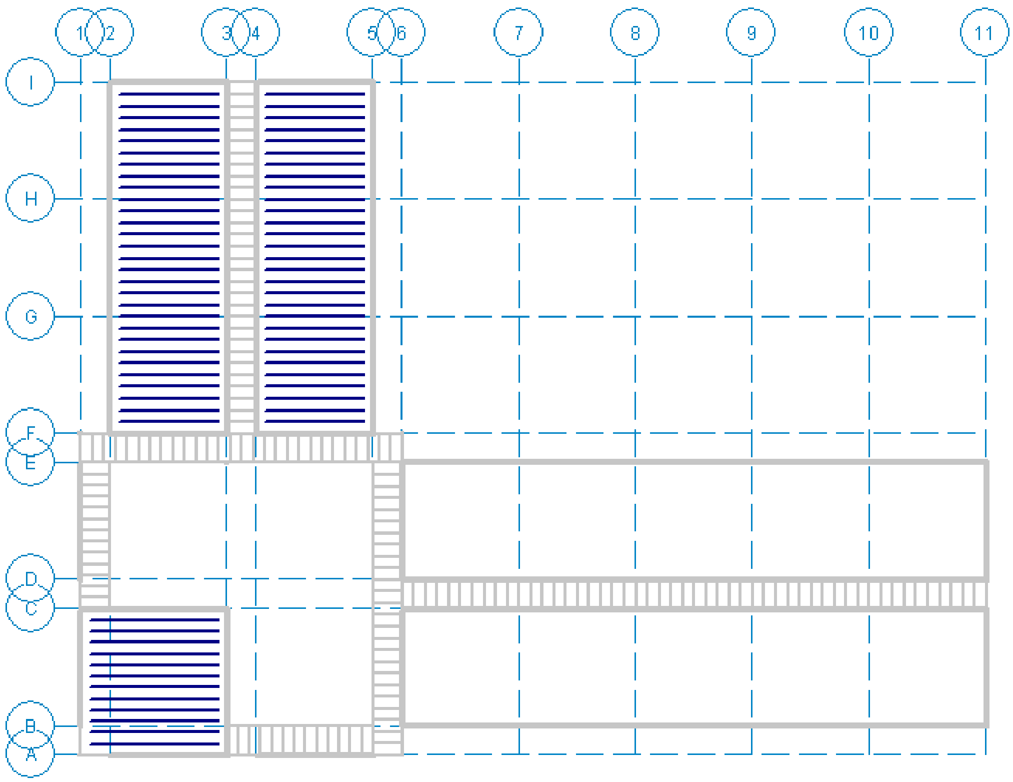


- Draw Infill Beams by clicking on the Generate Beams within a Bay button 

  Select: Wood Product, TJI Joist

  Beam spacing not to exceed 2’ o.c. oriented horizontally

  Click within the regions to generate the framing as shown in bold below:

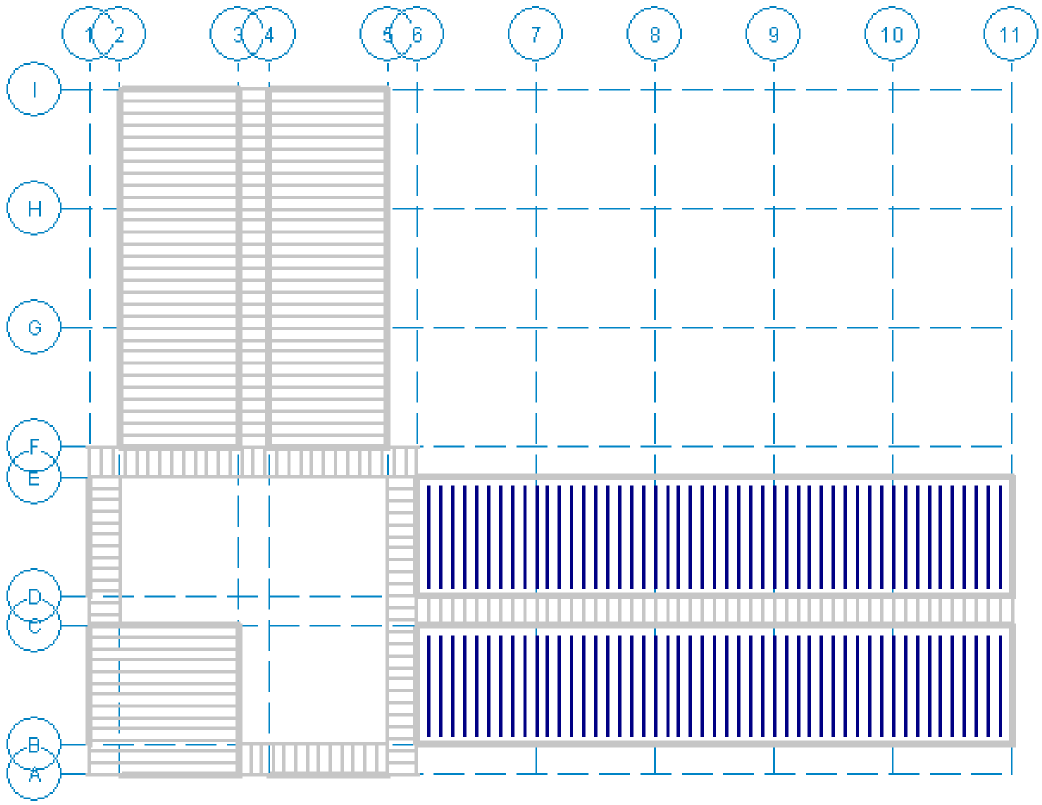


- Draw Infill Beams by clicking on the Generate Beams within a Bay button 

  Select: Wood Product, TJI Joist

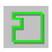
  Beam spacing not to exceed 2’ o.c. oriented vertically

- Click within the regions to generate the framing as shown in bold below:



Draw the Slab Edge

Note: We must define a slab edge to specify the extents of the area load. The slab edge also specifies whether the diaphragm in this building is to be treated as rigid or flexible. There are various schools of thought when it comes to slab edge overhangs with wood. Obviously wood decks do not cantilever out like concrete slabs, however one could argue that they do extend a distance equal to half of the wall thickness, since the wall is modeled as a center-line element. One advantage of using this distance (half of a 2x6 = 2.75 inches) is that the slab edge is rendered slightly offset from the beams and walls, making it much easier to see the perimeter beams and walls within the model.

- Click on the "Create Edge Perimeters" button 

  Select Type "Flexible", Edge Overhang Distance enter 2.75 inches

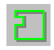
  Select "Create a Slab perimeter", "Consider ALL selected Beams/Walls for the Slab Perimeter"

  Click "Apply" to create the edge perimeter

You will see an orange line drawn around the perimeter of the building. This is the extent of the deck.

Draw the Slab Opening

We want the central portion of the building to be an open atrium area, so we do not want any area load attributed from this area. We can accomplish this by defining an opening in the "slab"

- Click on the "Create Edge Perimeters" button 

  For Edge Overhang Distance enter 2.75 inches

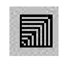
  Select "Create an Opening perimeter", "Click Within or Box the Beam/Wall perimeter for the Opening"

  Click "Apply" and Click within the central atrium area to draw the opening

You will see a bluish-gray line drawn around the perimeter of the atrium. This is the boundary of the opening.

Draw the Deck

We specified a default deck angle of zero degrees, which means that currently the deck spans in the plan-horizontal direction. We must manually assign a plan-vertical deck so that the floor load is properly attributed to the framing.

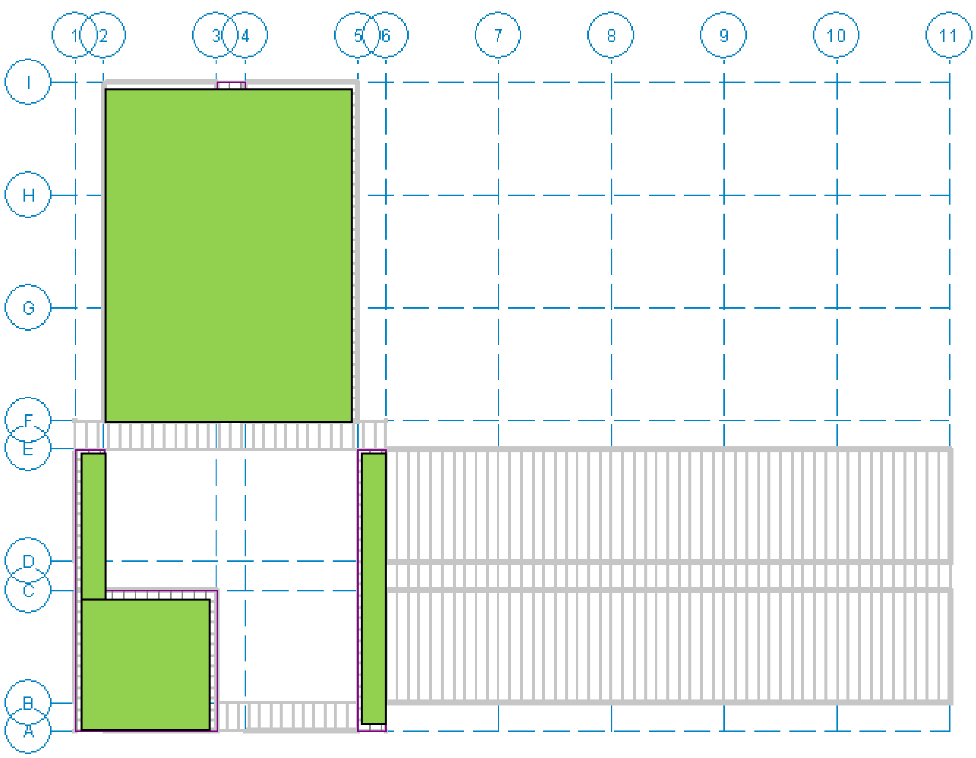
- Draw a 'Local' Deck/Slab by clicking on the Assign Slab or Deck Properties button 

  Select: Wood, oriented 'Parallel to X Axis'

  Select "Point to Point Draw of the Deck Area"

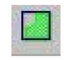
  Click the "Apply" button to start drawing the deck

  Draw the deck as shown in the shaded regions below



Draw the Area Load

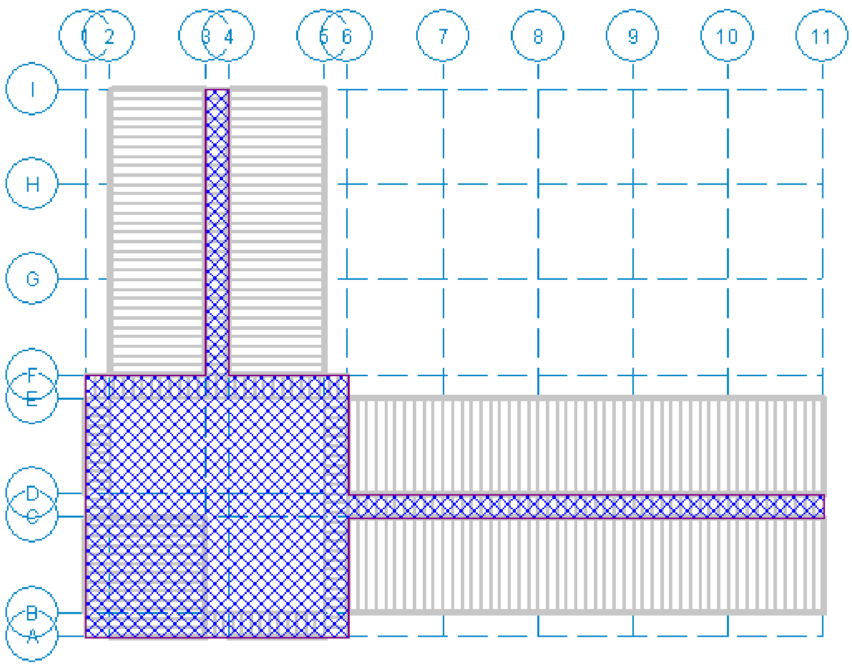
We have specified a default area load of “Room”, however there is a Corridor load that must be applied to the corridors, as well as the corner area which is designated for public assembly. We must specify this higher load within the model.

- Draw a ‘Corridor’ Area Load by clicking on the Draw Area Load button 

    Select “Corridor” from the dropdown list of Uniform Area Loads

    Point to Point Draw of the Area - Press Apply

    Draw the area load as shown in the shaded region below:



Note: that even though we have applied the area load over the central atrium area it will only be applied to the corridors since we have defined an opening in that area.

Apply the load from Stairs

For this model we will not model or design the stairs, however we still must account for the load from the stringers. It will be hard to see where to apply these point loads though, so first we must turn off the visibility of the area load.

- Click on the “Plot Options” button (upper left corner of screen)
- Click on the “Loads” tab

    Unselect “Uniform Area Loads”

    Click “OK”

Now it will be easier to see where we will be assigning the point loads.

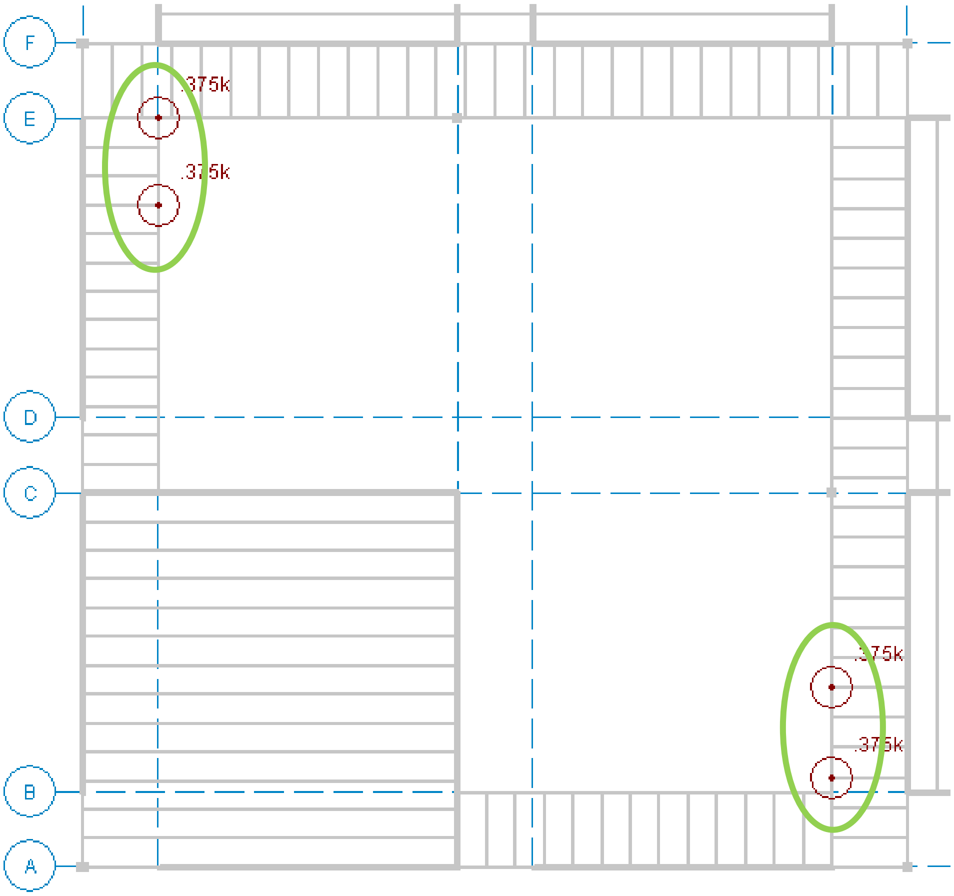
- Click on the “Assign Point Loads” button 

    Enter 0.375 k for DL Post

Enter 2.50 k for LL-Non

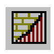
Select "Apply Load by Clicking/Boxing Points"- Click Apply

Draw the Point Loads as shown circled below:



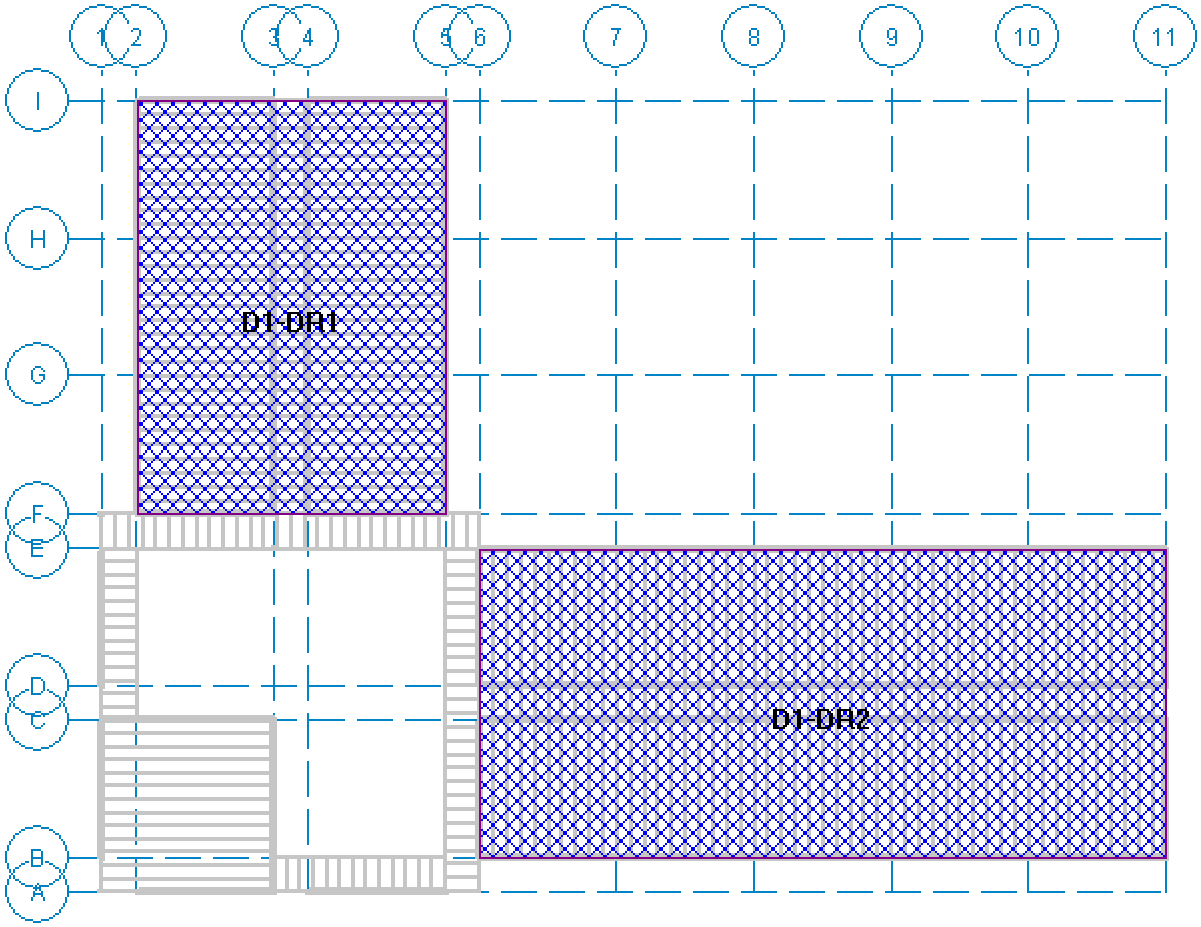
Define Diaphragm Regions

We must define where our flexible diaphragms exist, for design purposes. There is no diaphragm in the central portion of the building so we will not draw one there. The wind and seismic loads that would be attributed to that diaphragm will be automatically dumped into the diaphragms in each wing.

- Draw a Diaphragm Region by clicking on the "Draw Diaphragm Regions" button 

  Select Diaphragm Design Rule "Typical"- Click Apply

- Draw a diaphragm region in each wing of the building as shown below:



We are now finished defining the Second Floor.

Create the 3rd Floor

The third floor will be an exact copy of the second floor. The program is capable of automatically extending wall panels vertically, however we must redraw the wall panels because we want them stacked, not extended. It is likely that the program will automatically do this task in an upcoming version.

- Create an Copy of the '2nd Floor' by clicking on the Insert Menu and select 'Floor'

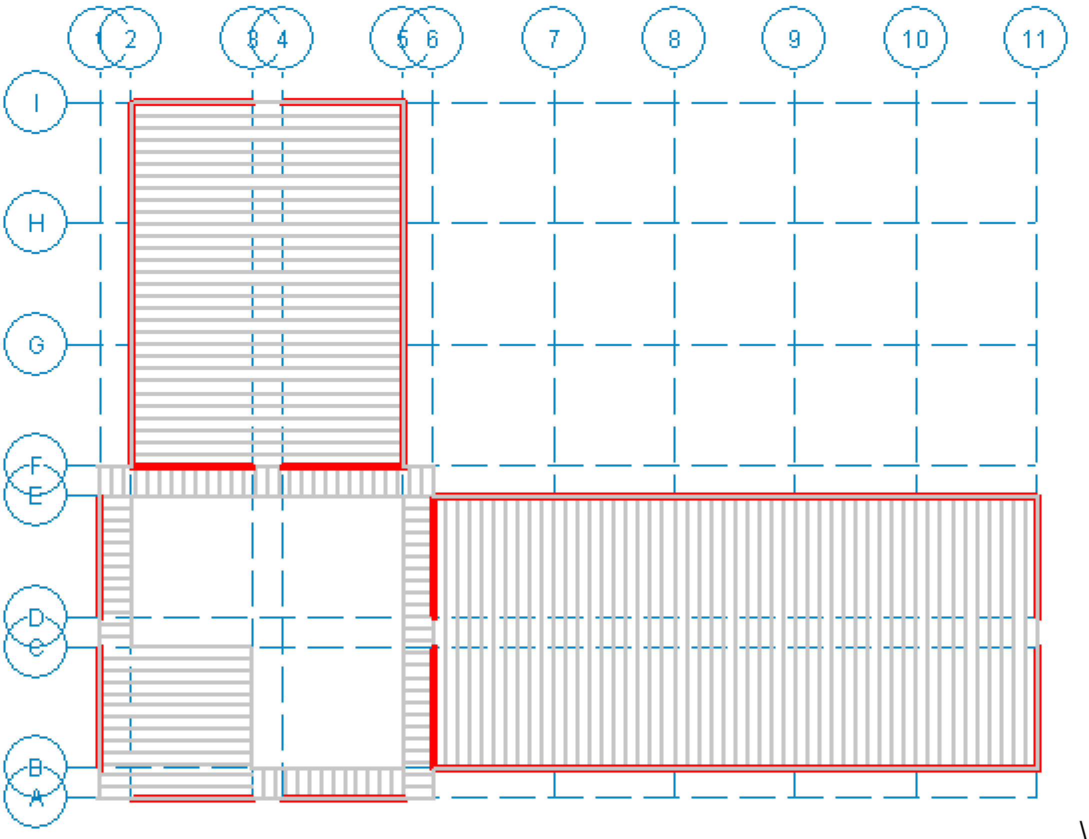
  Label: '3rd Floor' with an elevation of 20 feet, Area Load- Room, Deck Default- Wood, Deck Angle- 0, "Do Not Extend Wall Panel"

Draw the 3rd Floor Walls

- Draw 'Lateral' Walls by clicking on the Draw Wall button

  Select: Levels '2nd Floor' to '3rd Floor', Wood, DF/S Pine, Design Rule: Typical, Lateral

  Draw the wall panels as shown below. (Make sure to draw the long walls as individual segments between gridlines.)

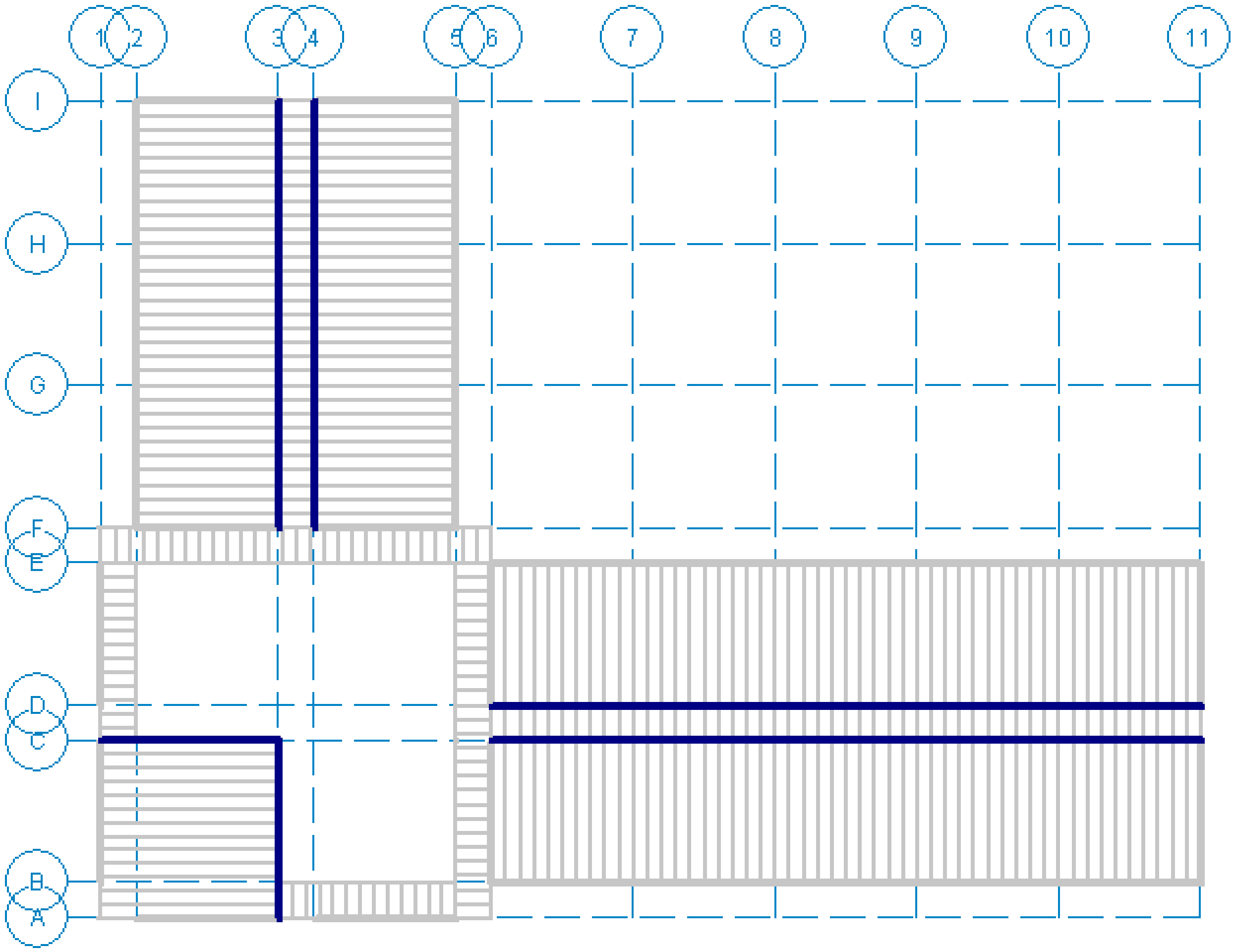


Note: Drawing the walls as individual segments can make it easier to model openings, hold-downs, etc. Be very careful to click on project grid intersections when drawing the walls, as opposed to beam intersections which may be nearby.

- Draw 'Gravity' Walls by clicking on the Draw Wall button 

  Select: Levels '2nd Floor' to '3rd Floor', Wood, DF/S Pine, Design Rule: Typical, Gravity

  Draw the wall panels as shown below.



Create the Roof

The roof will copy many of the second floor elements. The program is capable of automatically extending wall panels vertically, however we must redraw the wall panels as we want them stacked, not extended.

- Create an Copy of the '2nd Floor' by clicking on the Insert Menu and select 'Floor'

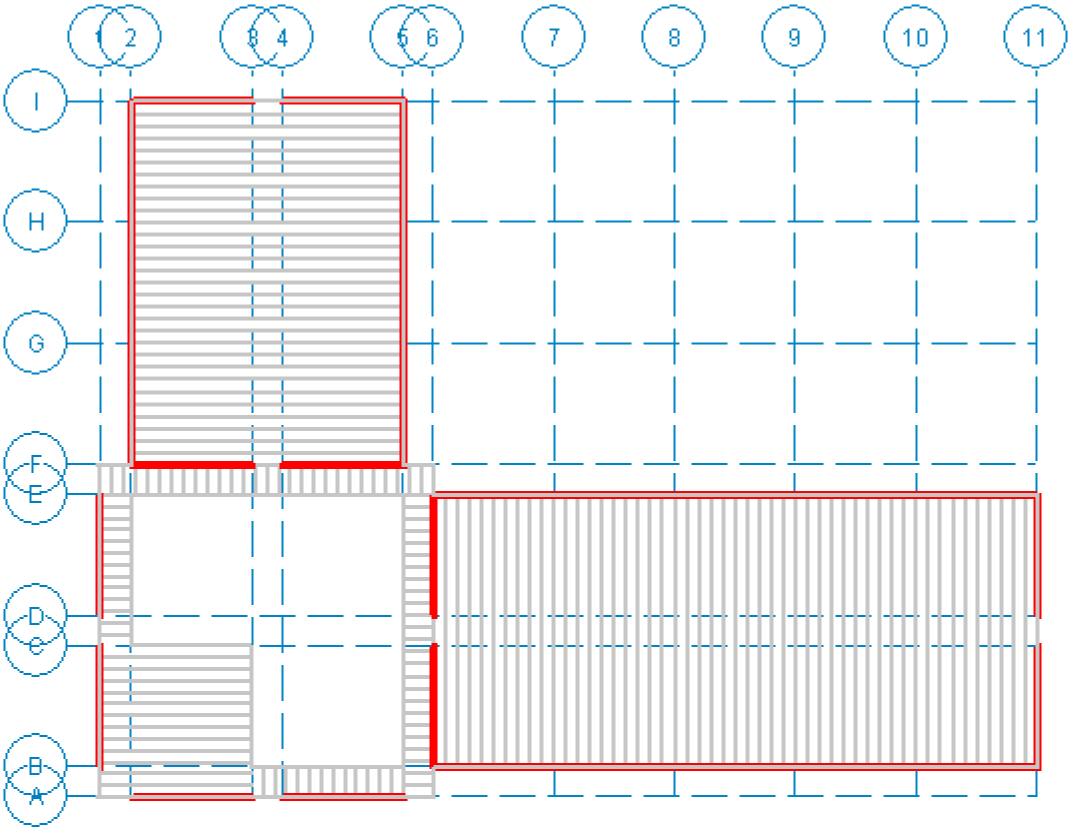
  Label: 'Roof' with an elevation of 30 feet, Area Load- Roof, Deck Default- Wood, Deck Angle- 0, "Do Not Extend Wall Panel"

Draw the Roof Walls

- Draw 'Lateral' Walls by clicking on the Draw Wall button 

  Select: Levels '3rd Floor' to 'Roof', Wood, DF/S Pine, Design Rule: Typical, Lateral

  Draw the wall panels as shown below. Draw the long walls as individual segments between gridlines.

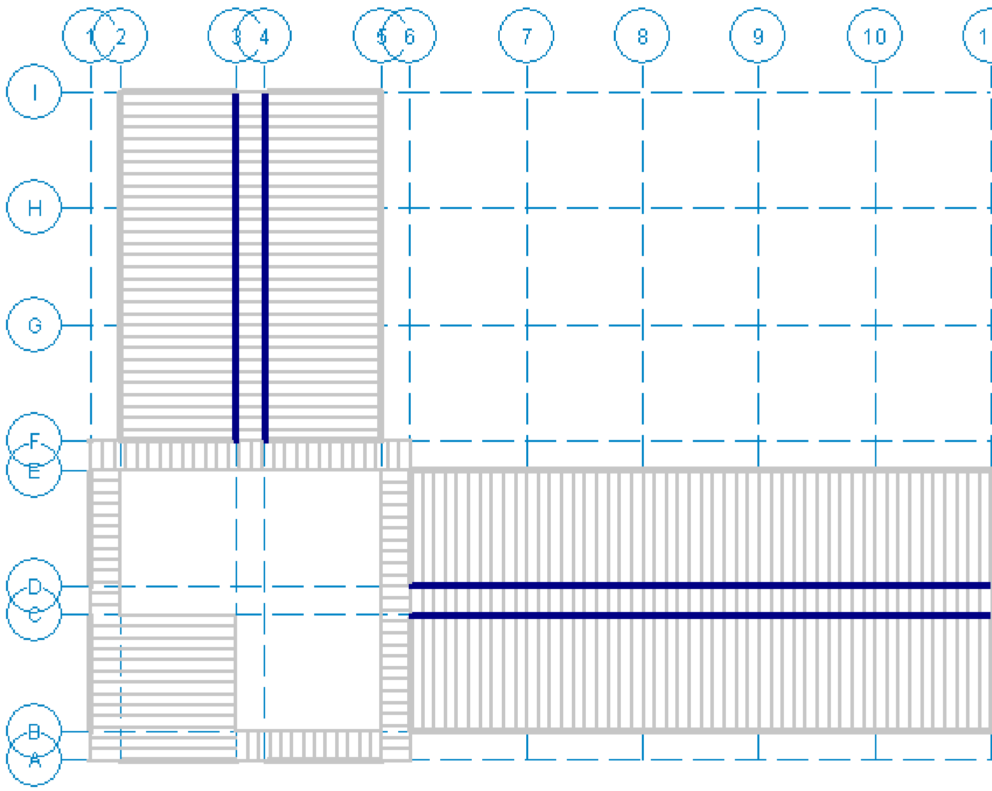


Note: Drawing the walls as individual segments can make it easier to model openings, hold-downs, etc. Be very careful to click on project grid intersections when drawing the walls, as opposed to beam intersections which may be nearby.

- Draw 'Gravity' Walls by clicking on the Draw Wall button 

  Select: Levels '3rd Floor' to 'Roof', Wood, DF/S Pine, Design Rule: Typical, Gravity

  Draw the wall panels as shown below.



Delete the Atrium Decks, Framing, Openings on the Roof Level.

Earlier we assigned a deck to the atrium area, and it was automatically copied up when we created the Roof. We must remove this.

- Delete the applied decks in the central atrium portion of the building by clicking on the "Delete" button 

  Select "Delete Decks by Clicking Them", Click Apply and click on the deck.

Note: that the visibility of decks has been turned on automatically. If you can't see the deck, you must turn it on in the Plot Options, Points/Decks/Slabs, Show Deck Assignments.

Next we must delete the opening in the central atrium area.

- Delete all opening perimeter in the central atrium area by clicking on the "Delete" button 

  Select "Delete Slab/Open Perimeters by Clicking Them", Click on the openings.

A more sophisticated approach to this would be to unselect the entire floor except for the beams in question, then use the “Delete Selected Beams” tool to delete them all at once. We will use such a method to delete the columns next.

Now we will delete the framing in the central atrium area

- Click “Apply” to begin deleting items
- Click on all of the beams inside of the central atrium area to delete them

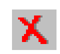
Lastly, we must delete the two gravity columns in the central area.

- Click on the “Select Based on Criteria” button (on the left side of the screen)

  Select the “Columns” tab

  Select “Function” Lateral, “UnSelect Columns”, Click “OK”

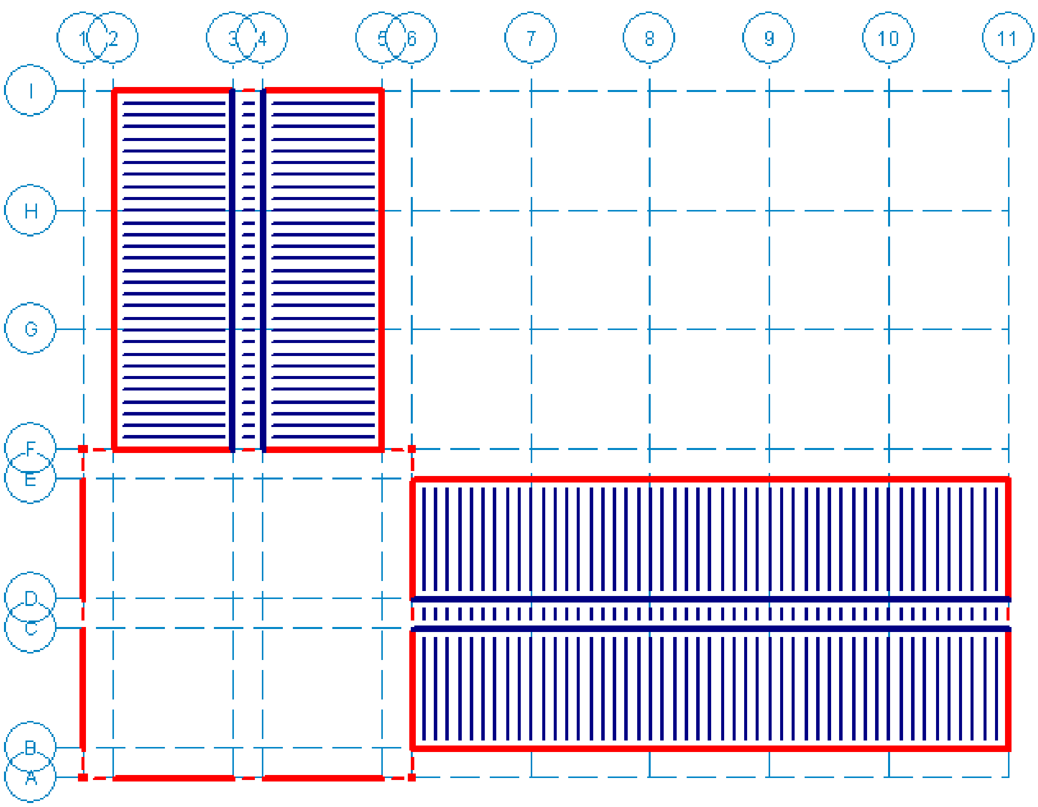
Now all of the lateral columns have been unselected.

- Delete the columns based on Criteria by clicking on the “Delete” button 

  Select “Delete Based on This Criteria”, “Delete Selected Columns”, Click “Apply”

  In the Column Removal Options dialog select “Remove a single Floor Column”, “Remember this answer”

After deleting all of these your layout should look like below:



Modify/Add Framing

We have specified wood joists for the floors of the rooms; however we want to use regular dimension lumber for the roof. Instead of deleting and redrawing these members we can just modify their properties.

- Click on the “Unselect Entire Model” button
- Click on the “Select Based on Criteria” button (on the left side of the screen)
- Click on the “Beams” tab

  Select “Shape” TJI Joist, Select “Select Beams”, Click “OK”

Now all of the wood joists have been selected.

- Modify the beams by clicking on the “Draw/Modify Beams” button , “Modify Properties” tab
- Click on the “Clear Use” button. This ensures that no previous use settings will apply.

  Select “Wood”, “DF/SPine”, “Rectangular”, Select “Apply Entries to All Selected Beams”

Note: Make sure to check the Use box above the options selected.

Now all of the wood joists have been changed to dimension lumber.

- Click the “Select Entire Model” button (on the left side of the screen)

The next thing we must do is add framing over the central portion of the building. We want to add trusses. However, you cannot define a truss in RISAFloor, so for now we will just model the top chords. We will model them as lateral so that they transfer into RISA-3D where we can add the bottom chords and webs. They will be pin-pin members, so they will not actually participate in lateral force resistance.

- Draw Vertical Infill Beams by clicking on the Generate Beams within a Bay button

  Select: Wood, DF/SPine, Rectangular, Typical, Vertical

  Beam spacing not to exceed 2’ o.c. oriented vertically

  Click within the central atrium area to generate the beams

Note: The infill framing is always automatically created as Gravity. It is rare that you would use infill framing for lateral force resistance. However in our case, the infill framing is going to be lateral as explained above. Hence, we must convert this framing to be lateral.

- Click on the “Unselect Entire Model” button (left side of the screen)
- Select only the top chord members that we just generated by click on the “Line Select” button

Now all of the top chord members have been selected.

- Modify the beams by clicking on the “Draw/Modify Beams” button , “Modify Properties” tab
- Click on the “Clear Use” button. This ensures that no previous use settings will apply.

  Select “Lateral”, Select “Apply Entries to All Selected Beams”

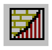
Note: Make sure to check the Use box above the options selected.

- Select the entire model by clicking on the "Select Entire Model" button

Now the top chords will be able to be transferred into RISA-3D

Roof Diaphragm

We now need to define a roof diaphragm over the central portion of the building, which has not had diaphragms at the lower floors.

- Draw a Diaphragm Region by clicking on the "Draw Diaphragm Regions" button 

  Select Diaphragm Design Rule "Typical"- Click Apply

The diaphragm regions will make it difficult to see the beams once we solve them, so let's turn them off from visibility.

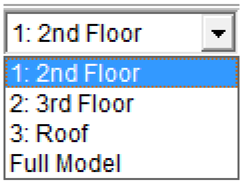
- Click on the "Plot Options" button , "Points/Decks/Slabs" tab

  Unselect "Display Diaphragm Regions"

Add Wall Openings

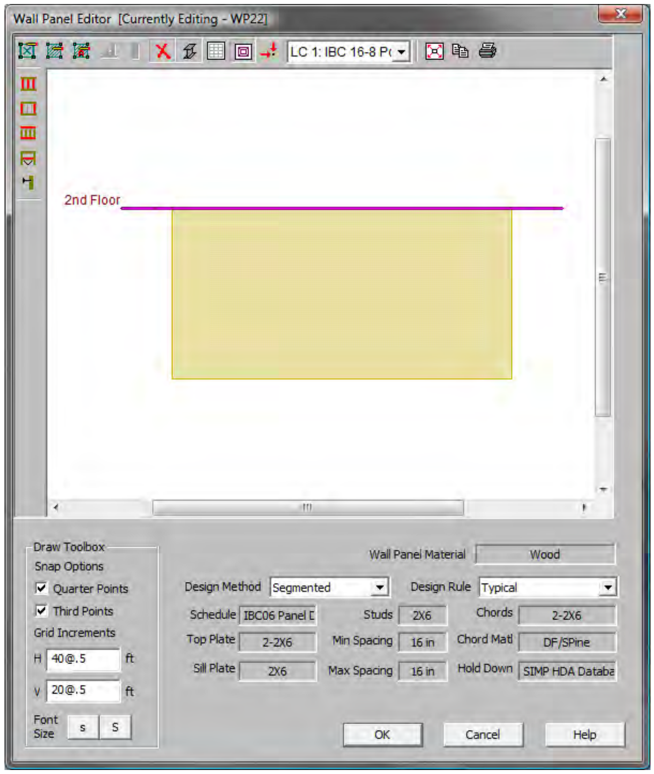
Wall openings must be defined prior to transferring the model over to RISA-3D. Obviously there would be quite a few openings in a building like this, and it would certainly take some time to get them all modeled. In the interest of saving time for this model we will model the openings on only one wall stack, but the methods would apply to doing so for all other wall stacks.

- Click on the Floor Selection Dropdown and select "2nd Floor"



Double-click on the wall that runs along Grid Line 11 between Grid Lines D and E

The wall panel editor should appear as shown below:



- In the "Grid Increments" boxes (lower left corner of the wall panel editor) enter the following values:

  H: 5 @ 4

  V: 3 @ 3.333

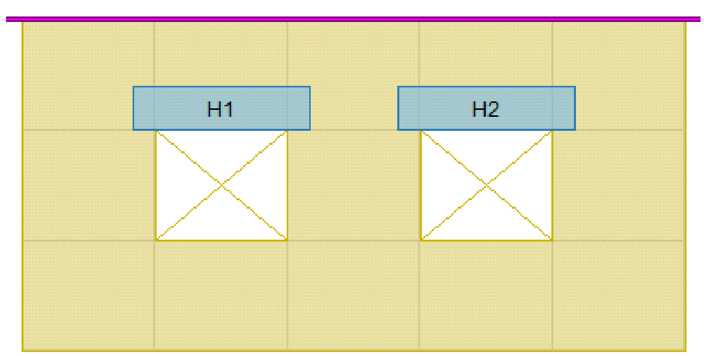
- Click anywhere in the white space of the wall panel editor for it to accept the "V" increments

  (If a grid does not become visible on your screen, click the "Toggle Grid Display" button at the top of the wall panel editor)

A grid should now be visible with which we can draw window openings.

- Draw an opening by clicking on the "Create New Openings" button (upper left corner of wall panel editor)

  Click point-to-point to define the two openings shown below:



Next we must generate regions within which the wall will be designed.

- Click on the “Automatic Region Generation” button (top of wall panel editor)

  Click “OK” to accept the changes and close the wall panel editor.

Lastly, we will add the openings to the walls stacked above this one on the “3rd Floor” and “Roof”.

- Switch to view the 3rd Floor plan.

  Follow the instructions above to add the same two openings and generate regions for the 3rd Floor Wall.

- Switch to view the Roof Floor plan.

  Follow the instructions above to add two openings and generate regions for the Roof Wall.

Model Clean-up

All of this adding and deleting of members, decks, etc. can cause the model to become cluttered with joints that were created but are no longer used. It is always good practice to delete these joints, and thereby cleanup the model.

- Click on the Tools menu and select “Delete All Unattached Points” , click “Yes”.

A report is generated, informing you of how many unattached points were in your model. These are points that are not associated with any beam, column, wall, deck, load, etc.

Create Load Combinations

RISAFloor comes with two basic load combinations by default, but we want to be sure to capture the entire load, according to the IBC requirements. Therefore we need to generate the relevant combinations.

- Open the Load Combinations Spreadsheet on the Data Entry Toolbar.
- Delete both of the existing lines by pressing F4
- Click on the Load Combination Generator button

  Select the IBC 2006 ASD building code including Roof Live Load (RLL) and Post Composite.

  Pre-Composite combinations are not required in this model- uncheck that option.

As you can see, there are more than two relevant load combinations for this model.

Confirm Columns

We should make sure that the column design parameters are correct. We do not want to specify column splices in this building, only because that will cause the program to “step-down” the column sizes at every floor, which will be difficult to detail.

- Open the “Columns” spreadsheet
- Using the Floor dropdown (top of the spreadsheet) select the 3rd Floor

  Uncheck the “Splice” box for all columns

- Using the Floor dropdown (top of the spreadsheet) select the Roof

  Uncheck the “Splice” box for all columns

- Close the “Columns” spreadsheet

Lastly, we must confirm that the columns have a shear connection (pinned) at the foundation (as opposed to a moment connection).

- Open the “Floors” spreadsheet

  For the “2nd Floor”, specify “Splice” as “Shear”

- Close the “Floors” spreadsheet

Solve and Review

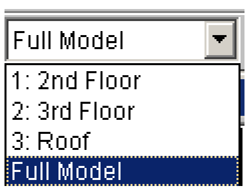
We are now ready to solve the model in RISAFloor, which will get us the gravity results for the entire model.

- Click the “Solve” button on the RISA toolbar

You should receive warnings that a number of Roof beams could not be designed. These are the truss top chords, and since we have not yet modeled them as trusses the program cannot find any lumber size that works on its own.

Take some time to review the results, including beam and column sizes.

- Go to a Model View and click on the Floor Selection Dropdown and select “Full Model”



You can now see a 3D rendering of the entire building. The wall panels have been shrunk by default, but we can correct that through the Plot Options.

- Click on the “Plot Options” button (upper left corner of screen), “Beams/Columns/Walls” tab

  In the “Show Walls” portion of the dialog, select a “Render” size set to 100%

  In the “Rendered” portion of the dialog, you can select a “Percent of Length” value. Set this to 100%

Now all of the walls and beams are rendered full size. Rotate the model and explore it. Choose individual floors from the Floor Selection dropdown (upper left corner)

RISA-3D Integration

We are now ready to take the model into RISA-3D for the design of the roof trusses and the lateral systems.

- Use the Director tool to transfer your model into RISA-3D.

  Click “OK” at the Wind Loads Dialog and Seismic Loads dialog (we have already defined these parameters)

- When the Global Parameters window pops up, click on the “Solution” tab

  Set Number of Sections: 5, Internal Sections: 100

Turn on Shear Deformation and Torsional Warping, Turn on Load Transfer between intersecting walls

Set Area Load Mesh:144 in^2, Merge Tolerance: 0.12 in, Mesh Size: 24 inches

Click “OK” to accept the values.

Model the Trusses

We will use the automated truss generator. In order to do this we must set up some Section Sets first. These will ensure that the trusses will all be made of the same members.

- Open the Section Sets spreadsheet and click on the “Wood” tab
- Rename any existing entries in the spreadsheet and ultimately create three section sets as shown below:

Wood Section Sets

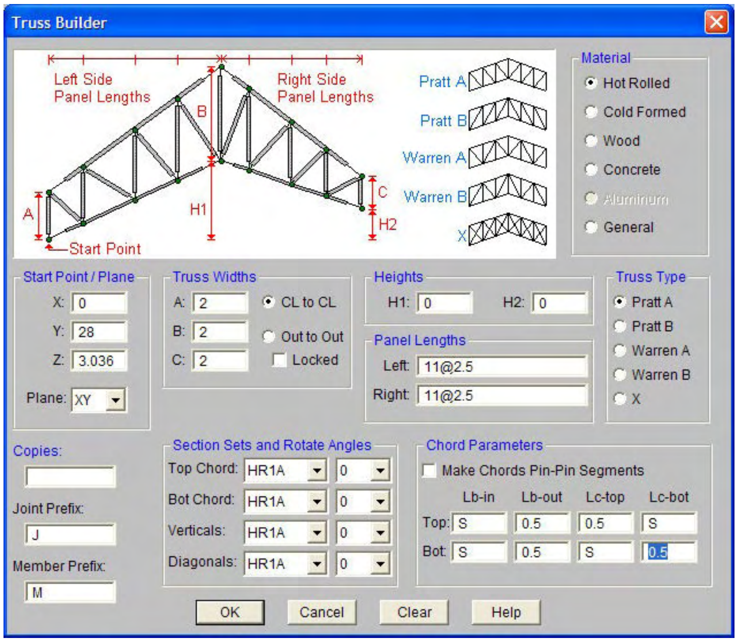
Hot Rolled | Cold Formed | Wood | Concrete | General

	Label	Shape	Type	Design List	Material	Design Ru...
1	Web	2X6	Beam	Rectangular	DF/SPine	Typical
2	Bott Chord	2X6	Beam	Rectangular	DF/SPine	Typical
3	Top Chord	2X6	Beam	Rectangular	DF/SPine	Typical

- Click on the “Unselect Entire Model” button 
- Click on the Insert Menu, and select “Structure Generate”

Click on the “General Truss” button.

If you view the appropriate joint coordinates ahead of time you can create the roof trusses very quickly and easily. For the sake of convenience in creating this model the appropriate values have already been determined. Enter them as shown below:



- For "Material" select Wood

  For "Start Point/Plane": X: 0 ft, Y: 28 ft, Z: 53.036 ft

  Plane: XY

  For "Truss Widths" enter the following values

  A: 2 ft, B: 2 ft, C: 2 ft

  Select CL to CL

  For "Heights" enter 0 for both values

  For "Panel Lengths" enter the following values

  Left: 11@2.5 ft

  Right: 11@2.5 ft

  Select Pratt A for "Truss Type"

  For "Section Sets " (as we defined these earlier) select the following:

  Top Chord: "Top Chord"

  Bot Chord: "Bott Chord"

  Verticals: "Web"

  Diagonals: "Web"

  For "Chord Parameters" leave the "Pin-Pin" box empty and enter the following values

	Lb-in	Lb-out	Lc-top	Lc-bot
Top:	S	0.5	0.5	S
Bot:	S	0.5	S	0.5

> **Note:** What do these unbraced values mean?
>
> They are the respective unbraced lengths for the top and bottom chord. “S” is short for “Segment”. It specifies that a brace point exists wherever a joint falls along that member, which in this case is where a diagonal frames in.
>
> For the top chord we have specified that its compression flange is braced every 6 inches (the assumed distance between nails) and its bottom flange is braced wherever a diagonal frames into it. Also for the top chord we have specified that it is braced against buckling in the same manner. For the bottom chord we are specifying 6” bracing on the bottom side (we assume that ceiling framing from below is attached every 6 inches and that the webs brace the top.

- Click “OK” to create the truss- close the model view that pops up immediately after creating the truss, as it has the wall panels selected.

This truss is almost what we are looking for, but it must be modified slightly to accommodate this application.

- Click on the “XY view” button on the Window Toolbar
- Click on the “Lock Unselected portion of Model” button (on the left side of the screen)
- Zoom in/out accordingly so that only the truss is shown on screen

You should now be able to see the truss in profile.

- Click the “Activate Graphic Editing Toolbar” button on the Window Toolbar
- Click the “Delete” button on the Graphic Editing Toolbar

    Select “Delete Items by Clicking Them Individually”

    Click “Apply” and delete the vertical web members at each far end, as well as the top chords.

The truss generator automatically creates top chords; however we have already modeled them in RISAFloor so these would be duplicates.

We do not want the bottom chord to extend all the way to the walls, so we must pull it back on each side

- Double-click on one of the bottom chords

The “Member Information” dialog box should now appear. Take note of the “Joint Labels” (upper right corner) and figure out based on the image shown in the model view which joint is on the outside end.

- Redefine the outside end joint to instead be the joint that is one panel point ‘in’ from that location. You can do this by typing the other joint label into the dialog box.
- Click “OK” to set the new end joint definitions

The bottom chord should now be pulled back so that it only extends to the diagonal.

- Repeat the process above for the other bottom chord.

We now have two extra joints which fall in the plane of the supporting wall panels. The wall panels will be forced to mesh around those joints, which will slow down the solution. It would be a good idea to remove them.

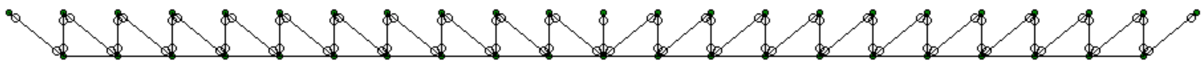
- Click the “Delete” button on the Graphic Editing Toolbar

    Select “Delete Items by Clicking Them Individually”

    Click “Apply” and click on the two unattached joints to delete them

- Right-click to drop the “Delete” tool

Your truss should now look like this:



- Click on the “Unlock Unselected portion of Model” button (on the left side of the screen)
- Click on the Iso view button on the Window Toolbar
- Zoom out so that you can see the entire central portion of the building

Now that we have successfully created one truss we can make copies of it to populate the rest of the model.

- Click on the “Copy Offset” button on the Graphic Editing Toolbar

  Leave all of the X and Y “Increment” fields blank

  Enter a “Z increment” of 26@-1.96429 ft

  Click “Apply”

Note: The (-1.96429 ft) increment is not exactly 2 feet. Earlier in RISAFloor when we generated the top chords we specified that they should be at an equal spacing not to exceed 2 ft. Because the central area has an odd dimension for length the remainder was cut out equally from each truss spacing. Alternatively, we could have defined the spacing to be exactly 2 feet, and we would have had an odd remainder on one side or the other. The negative sign is present because we want to copy this truss in the negative global Z direction.

You will see that the truss has been copied 26 times. Now all of the trusses have been created. They will automatically connect to the top chords (which are physical members, which means that they automatically connect to any member framing into them) and the RISAFloor gravity loads are already attributed to these top chords. We must still specify the unbraced lengths for the top chords however. It could be challenging to select them in a 3D environment, however there are certain selection tools to make this task easier.

- Click on the “Unselect Entire Model” button  (left side of the screen)
- Click on the “Select Based on Criteria” button (on the left side of the screen)
- Click on the “Members” tab

  Click the “Clear Selection Criteria” button

  For Parallel to Axis select “X”

- Click on the “Coordinates” tab

  Enter 30 Min and 30 Max for “Y Coordinate”

  Enter 1 Min and 54 Max for “Z Coordinate”

  For “Members” select “Both Joints Only”

  For “Plates/Panels” select “No Plates”

  For “Solids” select “No Solids”

  Uncheck “Select/Unselect Joints”

Check “Include Criteria on Other Pages”

Check “Select Items”

Click “OK” to select the members based on the specified criteria

- Modify the members by clicking on the “Draw/Modify Members” button on the Graphic Editing Toolbar, ‘Modify Design’ tab
- Click the “Clear Use” button

For “Lbyy/le2” enter 0.5 ft and check the “Use” box

For “Lbzz/le1” enter S and check the “Use” box

For “Lb-comp/le-bend(top)” enter 0.5 ft and check the “Use” box

For “Lb-comp/le-bend(bot)” enter S and check the “Use” box

Select “Apply Entries to All Selected Members”

- Click “Apply”

Now all of the top chords have the appropriate unbraced lengths specified.

- Click on the “Select Entire Model” button (left side of screen)

Add Straps, Hold-downs

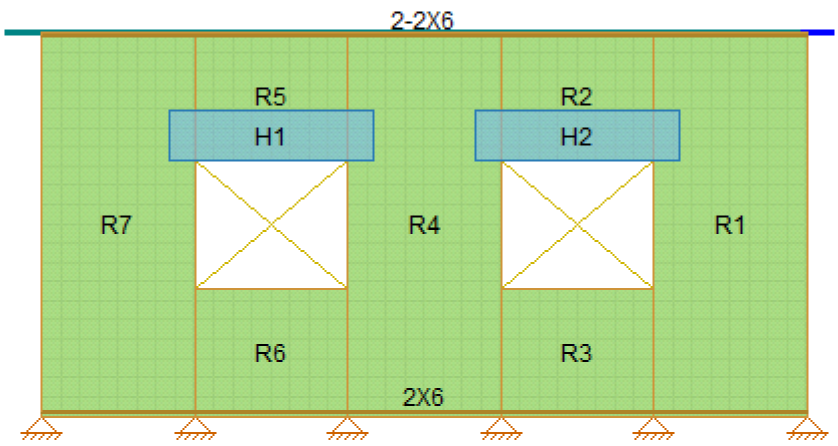
We must add the straps and hold-downs for the walls that we defined openings for earlier. You will see graphically that hold-downs have already automatically been generated for all of the solid walls.

- Double-click on the bottom wall with openings (Grid Line 11 between Lines D and E)

The wall panel editor should appear. Click on the “Add Hold-Downs” button (top of wall panel editor window)

We have defined this wall as having Segmented design, therefore we must add a hold-down at each “pier”

- Click on the corners of the regions to add the hold downs shown in the image below:



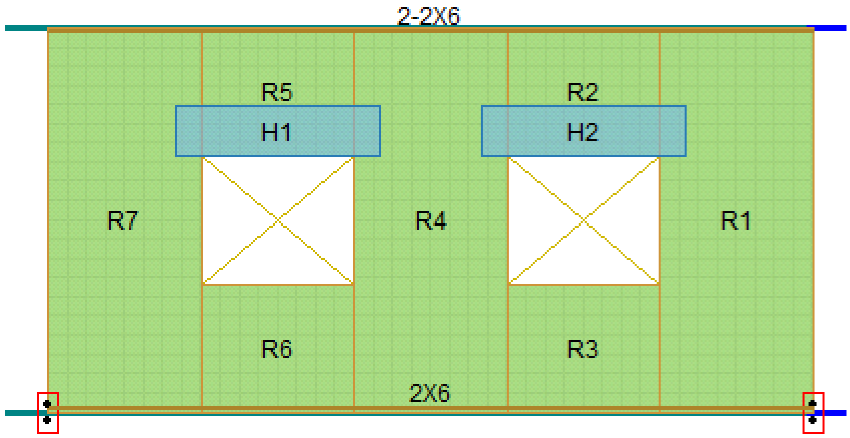
- Click “OK” to accept the changes and close the wall panel editor.

Double-click on the middle wall with openings (Grid Line 11 between Lines D and E)

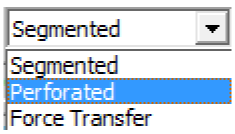
We will design this wall using the Perforated method, which means that we only need straps at each end to connect it to the wall below.

- Click on the "Add Straps" button (top of wall panel editor window)

  Click on the bottom corners of the wall to add straps as shown in the image below



- Click on the "Design Method" dropdown within the Wall Panel Editor and select "Perforated"



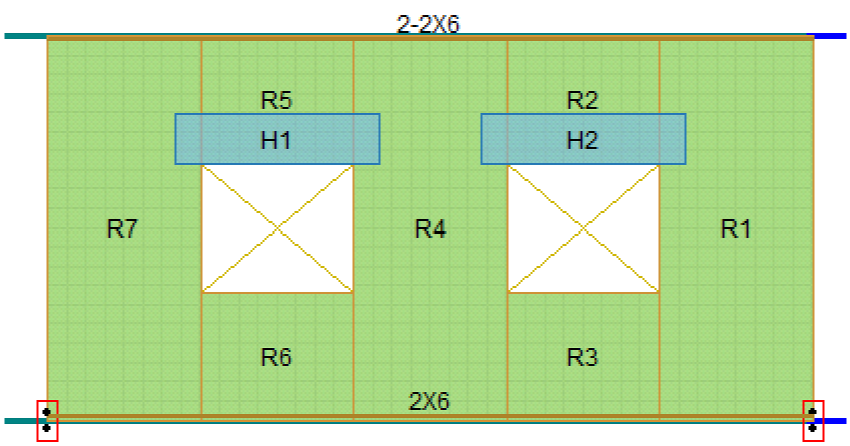
  Click "OK" to accept the changes and close the wall panel editor.

- Double-click on the top wall with openings (Grid Line 11 between Lines D and E)

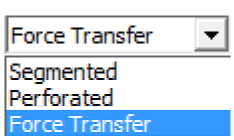
We will design this wall using the Force Transfer Around Openings method, which means that we only need straps at each end to connect it to the wall below.

- Click on the "Add Straps" button (top of wall panel editor window)

  Click on the bottom corners of the wall to add straps as shown in the image below



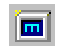
- Click on the "Design Method" dropdown within the Wall Panel Editor and select "Force Transfer"



  Click "OK" to accept the changes and close the wall panel editor.

Review and Solve

Now we will switch to a more thorough model view

- Deselect the joints by opening the “Plot Options” button , “Joints” tab

  Uncheck the “Show Joints” box, Uncheck the “Show Boundary Conditions” box
- View the members fully rendered by clicking on the “Members” tab

  Select “Draw Member As” Rendered, select 100% of length
- View the walls as fully rendered by clicking on the “Panels” tab

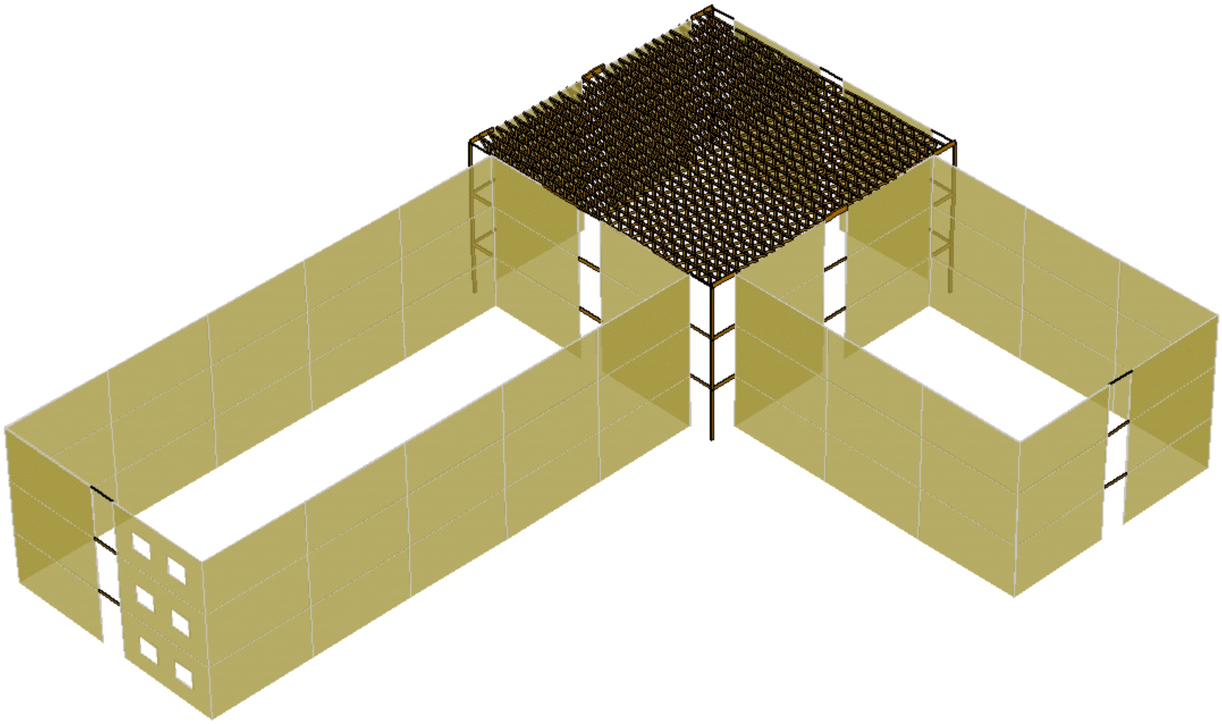
  Select “Draw Wall Panels as” Rendered, select “Size” 100%
- Click on the “Misc” tab

  Uncheck the “Display Project Grid Lines” box

You should now see a 3D rendered view of the structure.

- Right-click in the model view and select “Depth Effect”, “Increase Depth Effect A Lot”

Now your model is rendered in perspective rather than isometrically. It should look like the image below:



Now let’s set up the load combinations.

- Open the “Load Combinations” spreadsheet
- Click on the Load Combination Generator button

  Select the United States, IBC 2006 ASD building code

  Select Wind Load “X and Z”, Reversible

  Select Seismic Load “X and Z”, “Reversible”

- Click on the “Solve” button  on the RISA toolbar
- Select “Envelope of Marked Combinations”
- Click on the “Solve” button

This solution might take awhile depending on the speed of your computer. A warning log will appear once it is finished, warning you that the program used the default deck angle (direction) when considering the span of the wood panels in your deck (diaphragm).

Review the results.

Notes:

RISAFloor Wood Design - Reference

TECHNOLOGIES

26632 Towne Centre Drive, Suite 210
Foothill Ranch, California 92610

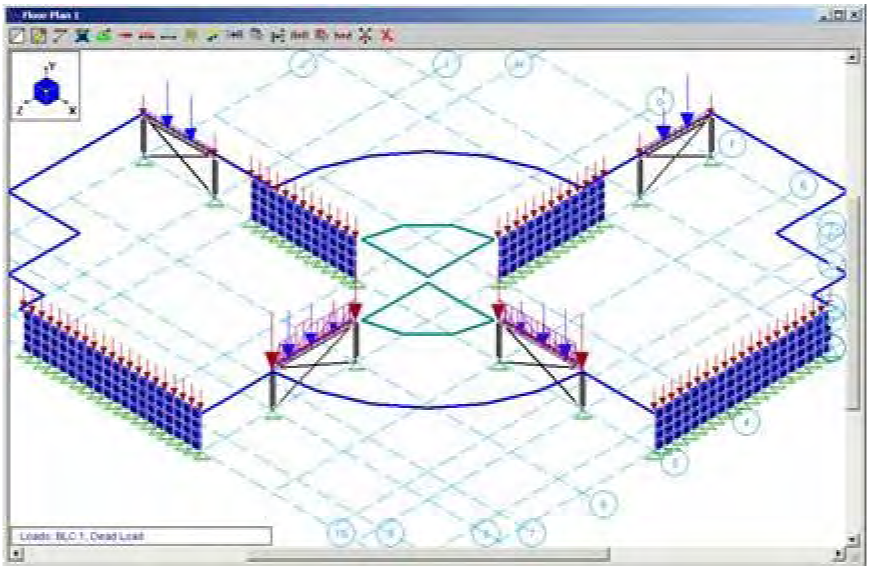
(949) 951-5815
(949) 951-5848 (FAX)

www.risatech.com

Table of Contents

RISAFloor and RISA-3D Integration

While the primary function of RISAFloor is to create and optimize floor systems, another strength is that it can be used to automatically generate a model of the lateral force resisting system in RISA-3D.



Note

- The features described in this section of the manual are only available to users who are running both RISAFloor and RISA-3D.

Lateral System Model Generation

Beams, columns, and walls whose function is set to 'Lateral' on the Primary Data Tab of their respective spreadsheets will automatically be generated in the RISA-3D model when accessed via the Director Menu. To access this model click the Director Menu on the far right end of the Main Menu and choose 'RISA-3D'. The RISA Application Interface will then switch from RISAFloor to that of RISA-3D.

Once in RISA-3D you will notice that you can use the RISA-3D features to edit and solve the model. You can add braces, beams, columns, walls, and additional loads just as you would in a regular RISA-3D model. Refer to the RISA-3D General Reference Manual and User’s Guide for documentation of RISA-3D's features.

The "gravity" model in RISAFloor and the "lateral" model in RISA-3D are fully linked. Subsequently, any changes made to RISAFloor generated members in the RISA-3D model will automatically update those same members in RISAFloor model.

Note

- Beams, columns, and walls whose function is set to 'Gravity' in RISAFloor will NOT be generated in RISA-3D for optimization. These members have been indicated as "gravity-only" members and should not collect any lateral load in the RISA-3D model. Therefore, these members would only "clutter" the lateral system model in RISA-3D and are subsequently left out.
- RISAFloor 'Gravity' members may be viewed in RISA-3D via the Misc Tab of the Plot Options Dialog. These members will be displayed for visual effect only in the model view but will not contribute to the stiffness of the RISA-3D model.

Diaphragms

Diaphragms are created in RISA-3D for every floor slab in RISAFloor. Although you cannot delete these diaphragms, you can make them **inactive** in RISA-3D. The **Mass**, **Mass Moment of Inertia** and **Center of Mass** are automatically calculated based on the RISAFloor loads and on the settings in Global Parameters.

Floor Diaphragms

	Elevati...	Mass[k]	Mass ...	Center of Mas...	Plus X Ecc...	Minus X E...	Plus Z Ecc...	Minus Z E...	Inactive	Diaphragm	Type	Region	Design Rule
1	12	62.399	4199.72	10, 10	%5	%5	%5	%5		D2	Flexible	DR1	Lower Level
2												DR2	Lower Level
3												DR3	Roof Level
4												DR4	Typical
5	12	247.759	116858	89.997, 34.453	%5	%5	%5	%5		D3	Flexible	DR1	Typical
6												DR2	Lower Level
7												DR3	Roof Level
8	12	46.582	3964.757	45, 15	%5	%5	%5	%5		D4	Rigid		

The **X and Z eccentricities** are used in the equivalent lateral force method for calculation of seismic loads. This allows you to quickly and easily account for the effects of accidental torsion when calculating your seismic response.

Note:

- Eccentricity does not apply to flexible diaphragms.

The **Diaphragm** and **Region** columns are the names of diaphragms and regions in the model.

Note:

- Diaphragms defined as rigid are not designed, thus the Region column is blank.

The **Type** allows you to toggle between flexible and rigid.

Note:

- The program does not have a semi-rigid option at this time. To consider a semi-rigid diaphragm, modeling the diaphragm as a plate model would be the route to go. Here is more information on Plates.

The **Design Rule** allows you to switch between design rules for the diaphragm regions. The Design Rules spreadsheet is where many parameters are defined for your diaphragm. For more information on diaphragm modeling and interaction, see the Diaphragms topic.

Gravity Loads

The gravity loads on the lateral members, including the beam reactions from the gravity only members, become part of the RISA-3D model. The Load Categories in RISAFloor are automatically converted into Basic Load Cases in RISA-3D. The exceptions to this are the Load Categories that deal specifically with construction loads (DL Const, and LL Const), which are not converted.

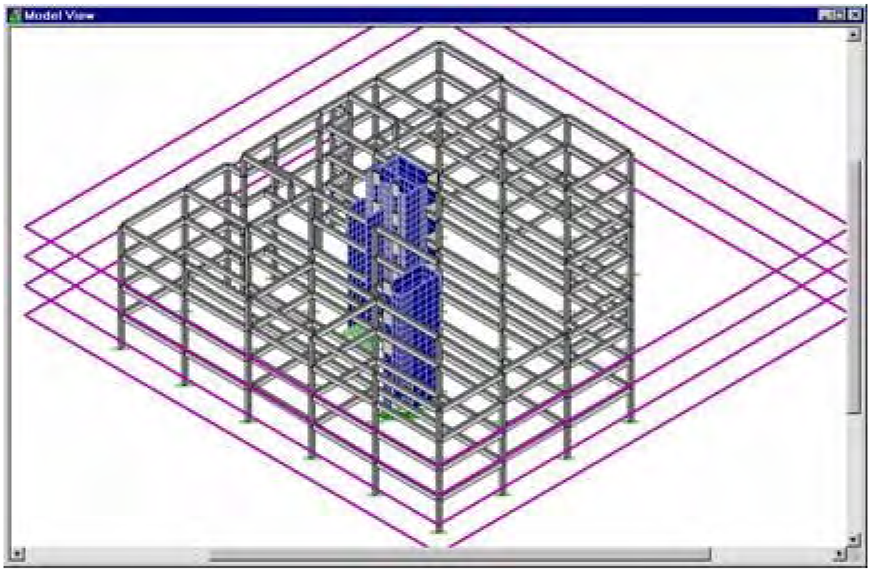
Wind Loads

Wind loads can be automatically generated for the 1995 – 2005 ASCE 7, the IS 875:87 (Indian) code, the 2005 Canadian code, and the Mexican code.

Seismic Loads

Seismic loads can be automatically generated according to the equivalent static methods of the 1997 UBC, 2000 IBC, 2001 CBC (i.e. California Amended UBC), the ASCE 7-2002 and 2005 (which is referenced directly by the 2003 and 2006 IBCs respectively), the 2002 Indian code (IS 1893), the 2005 Candian code, and the 2004 NTC (Mexican) code.

Diaphragms - General



Diaphragms can be defined in both RISAFloor and RISA-3D. The RISAFloor diaphragms are used in RISA-3D when using the Director tool. Here we will first have a general diaphragm discussion. Then we will get into the RISAFloor-RISA-3D diaphragm interaction. Finally, we will discuss some other concepts/ideas to keep in mind.

RISA-3D has two types of rigid diaphragms and a flexible diaphragm.. The two types of rigid diaphragms are a rigid membrane and a rigid planar diaphragm. The flexible diaphragm options are described in more detail in the Diaphragms - Flexible topic.

The two different types of diaphragms are provided to handle different modeling situations. The planar diaphragm option is rigid in all 6 degrees of freedom. It has a very large stiffness in plane and out of plane. The membrane diaphragm option is only rigid in the plane of the diaphragm. It has NO stiffness out of plane.

Note

- You should not apply a boundary condition to a joint that is part of a Rigid Diaphragm. See Defining Diaphragms.

Rigid Diaphragm Applications

The rigid diaphragm feature can be used to aid the engineer in quickly modeling the transfer of lateral forces into resisting elements such as shear walls, braced frames, and columns. These loads can be of a static or dynamic nature. Thus, lateral loads can be applied where they really occur on a structure, and the effects of the center of applied force being different from the center of rigidity will be accounted for automatically in the solution. Dynamic mass can also be applied where it actually occurs on the structure, and the differences between the center of mass and the center of stiffness will be accounted for automatically as part of the dynamic solution.

RISAFloor and RISA-3D Diaphragm Interaction

RISAFloor Diaphragms

When working with an integrated RISA-3D / RISAFloor model the diaphragms (both flexible and rigid) defined in RISAFloor will automatically be created within RISA-3D. The extents of these diaphragms are determined based on the defined slab edges and openings in the RISAFloor model. In the case of rigid diaphragms, they are created as rigid membrane diaphragms.

Interaction

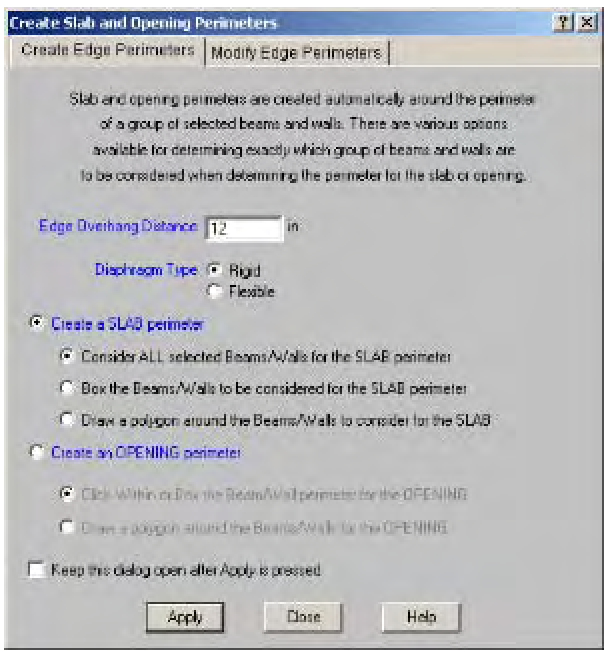
Input Interface

The layout (modeling) of the diaphragms is done using the RISAFloor interface, while the actual analysis/design is done within RISA-3D. Below are some definitions within the context of this topic that will be helpful to know during this process.

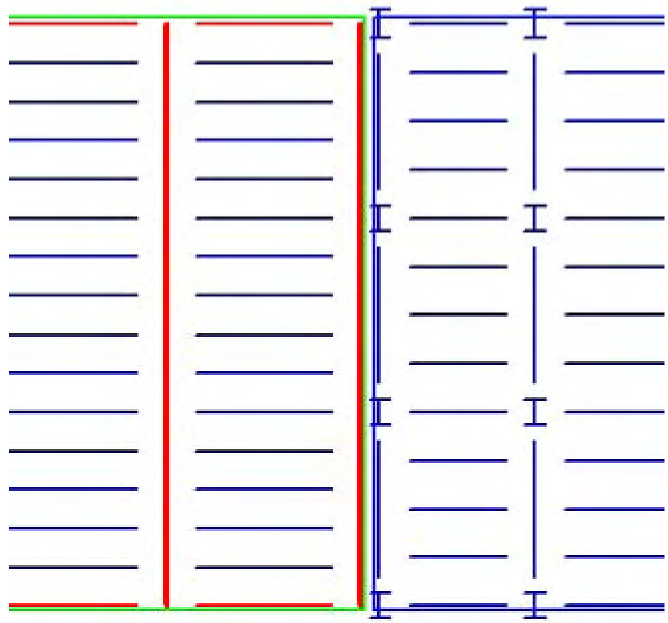
- Slab Edge – Defines the extents of a diaphragm, whether it is flexible or rigid.
- Chords/Collectors – Lateral members (wall or beams) modeled in RISAFloor to form a closed rectangular circuit around a diaphragm region.
- Diaphragm Region – A floor area which is assumed to distribute lateral loads to chords/collectors by acting similar to a simply-supported beam in the horizontal plane.

The best way to model diaphragms within RISAFloor is to follow the steps below:

1. Model the entire building (gravity and lateral members) within RISAFloor. If a given floor contains both rigid and flexible diaphragms, model a 'gap' between these two portions of the building such that slab edges can be drawn on either side of the gap. Use the Create Edge Perimeters tool to create a slab edge.



Here is a graphic of the gap:



2. Go to the Design Rules spreadsheet and set up your wood diaphragm design rules. These will control what panels and nailings are chosen for each diaphragm region.
3. Click the Draw Diaphragm Regions button and select the applicable Design Rule. Click Apply.

4. Click on two opposite corners of the diaphragm region.

Limitations:

- The entire perimeter of the diaphragm region must consist of a closed-circuit of lateral members. These are the members that the diaphragm will dump the chord and collector forces into, and it is critical that there is a complete load path from the diaphragm to the main lateral force resisting system.
- Diaphragms are limited to only rectangular shapes, and must be oriented along the principal X and Z axes.
- Lateral members that fall within a diaphragm region (as opposed to along its perimeter) will be ignored for diaphragm force distribution.

- Flexible diaphragms cannot be analyzed or designed for sloped roofs due to load attribution issues.

- Openings that fall within a diaphragm region are ignored for design, but considered for load attribution.

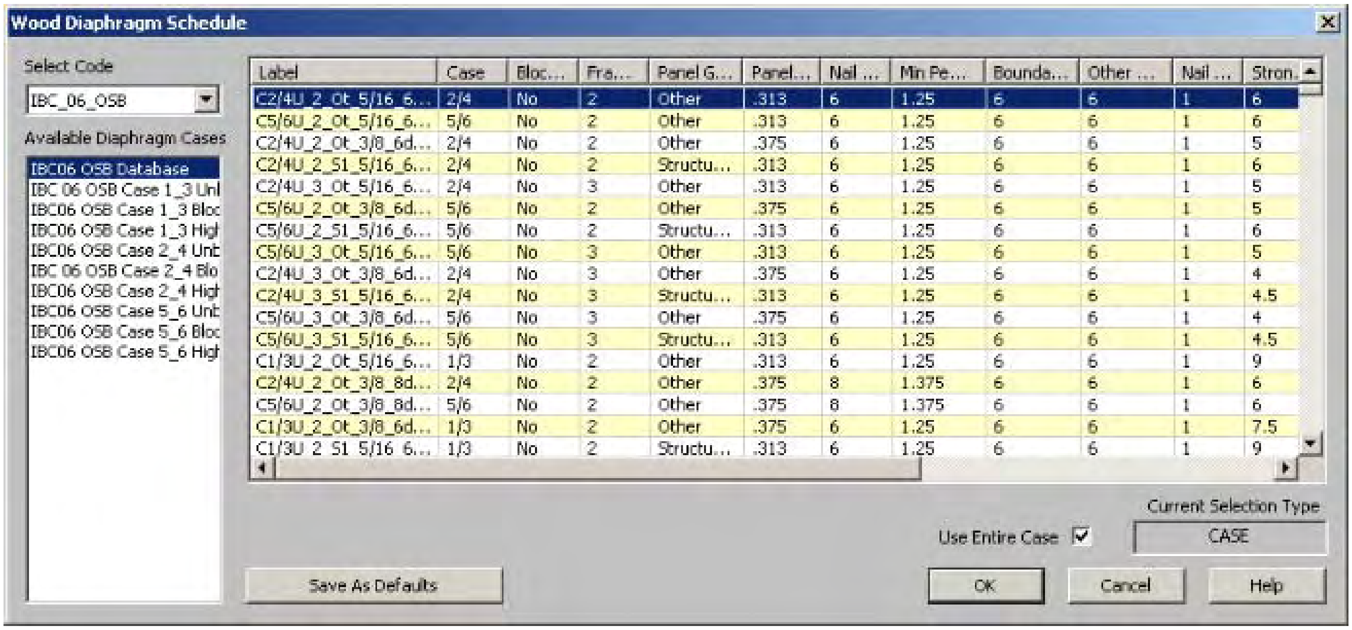
Results Viewing

Below are some general guidelines when reviewing results in a combined RISAFloor/RISA-3D model.

- The diaphragm results can only be viewed in RISA-3D under Results>>Diaphragms. See the Diaphragms - Analysis and Results topic for more information.

General Diaphragm Functionality and Discussion

Diaphragm Optimization



The procedure that RISA uses for diaphragm optimization is fundamentally based upon the assumption that there is a 'cost' to allowable shear in a product, and therefore the ideal diaphragm would have as little shear capacity as possible to meet code requirements. Once the program has determined the shear force required it will choose the most economical diaphragm thickness and nailing based on that which most closely matches (but does not exceed) the shear demand. The program looks to the **Shear Capacity** field of the diaphragm schedule to choose the design.

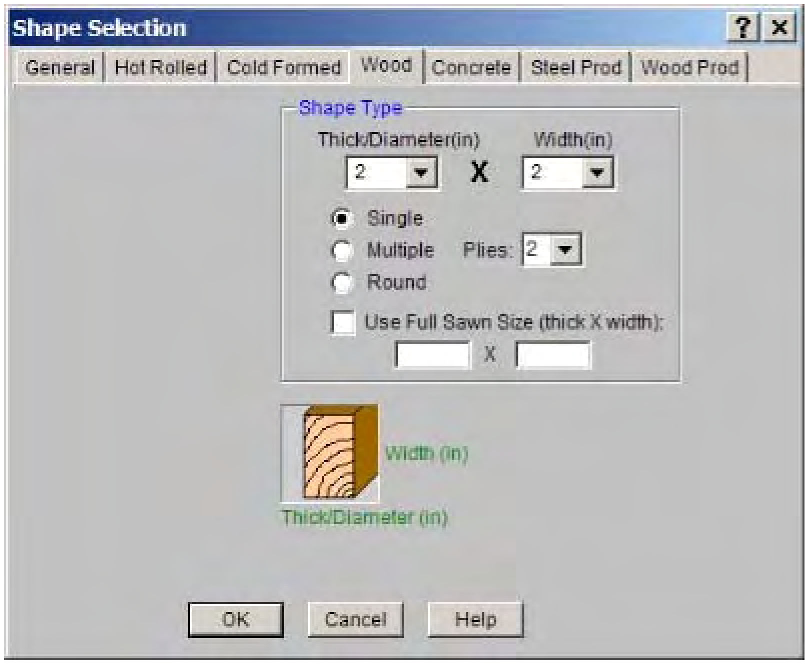
Optimization Procedure

For users who are new to diaphragm design within RISA, the best procedure is to utilize the full databases, and to limit the potential designs by utilizing the design rules spreadsheets. This results in a design based on the maximum number of options, which is often the most efficient design.

For experienced users who have more specific limitations in terms of the designs they would like to see, user-defined **Groups** (or families) are the solution. For example, an engineer who prefers to use only one panel layout or to force blocking can create a custom Group that contains only the arrangements they want. For more information on creating these custom groups see Appendix F-Wood Database Files.

Wood - Database

The Wood Database may be accessed by clicking the **Shape Database** button on the **RISA** toolbar and clicking the **Wood** tab. See Wood Products for information on the Wood Products Database.



To Select a Wood Database Shape

1. On the **Wood** tab of the **Section Sets Spreadsheet**, move the cursor to the **Shape** field and click .
2. Specify the shape type you wish to use (single, multiple, or round), then select from the lists of thicknesses and widths by clicking on .

Note

- If the value that you need is not given on the drop down list, you may directly enter any whole number for the thickness or width.
- Enter the nominal dimensions (or round diameter) in inches, regardless of what units system you are using. These will be automatically adjusted to the actual (dressed) dimensions for stiffness and stress calculations. For example, if you enter “2X4” as the size, the calculated properties are based on an actual size of “1.5 in. X 3.5 in.”

Custom Wood Sizes

If you would like to enter explicit dimensions of a member or if the member is "Not Dressed", the member must be designated as a "Full Sawn" member by checking the **Use Full Sawn Size** check box and entering the exact dimensions of the member in the boxes below. This applies to regular wood species, custom wood species, Structural Composite Lumber (SCL), and Glu-Lam members. The member thickness should be entered in the box on the left and the member width should be entered in the box on the right.

Wood Design Values

The program allows you to define members and walls as any of the wood species laid out in the NDS. The program does not explicitly call out the design values for these species, but they are taken directly from the NDS manuals. You can also get some verification of these design values by looking at the detail report for individual members.

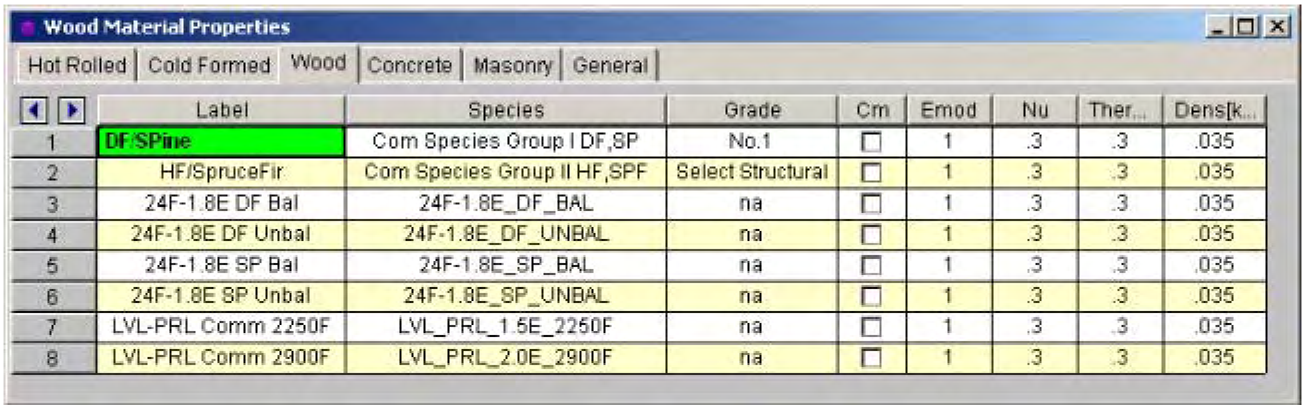
Also, the NDS has created some wood species groupings that will be implemented into the next version of the code. These new material call-outs have been implemented into the program. Here are some new materials defined for both RISAFloor and RISA-3D.

New NDS Wood Material Combinations

The **Com Species Group I DF,SP** and **Com Species Group II HF,SPF** come from the "Commercial Lumber Design Values" tables below.

The **Glulam** species groups come directly from the NDS 2005.

The **LVL** groupings come from the "Proposed PRL Commercial LVL Design Properties" below.



	Label	Species	Grade	Cm	Emod	Nu	Ther...	Dens[k...
1	DF/SPine	Com Species Group I DF,SP	No.1	☐	1	.3	.3	.035
2	HF/SpruceFir	Com Species Group II HF,SPF	Select Structural	☐	1	.3	.3	.035
3	24F-1.8E DF Bal	24F-1.8E_DF_BAL	na	☐	1	.3	.3	.035
4	24F-1.8E DF Unbal	24F-1.8E_DF_UNBAL	na	☐	1	.3	.3	.035
5	24F-1.8E SP Bal	24F-1.8E_SP_BAL	na	☐	1	.3	.3	.035
6	24F-1.8E SP Unbal	24F-1.8E_SP_UNBAL	na	☐	1	.3	.3	.035
7	LVL-PRL Comm 2250F	LVL_PRL_1.5E_2250F	na	☐	1	.3	.3	.035
8	LVL-PRL Comm 2900F	LVL_PRL_2.0E_2900F	na	☐	1	.3	.3	.035

Commercial Lumber Design Values, January 14, 2009

Joists and Rafters

dimension lumber 2" to 4" thick 2" and wider, Use with appropriate Adjustments for size and service conditions

Commercial Lumber Species Group	Grade	Fb (psi)	Ft (psi)	Fv (psi)	Fc-perp (psi)	Fc-para (psi)	E (psi)	Specific Gravity (OD Wt. & OD Vol.)
I	Select Structural	1500	1000	175	565	1700	1,800,000	0.50
	No. 1	1000	675			1500	1,700,000	
	No. 2	900	550			1350	1,600,000	
II	Select Structural	1250	700	135	405	1400	1,500,000	0.42
	No. 1	875	450			1150	1,400,000	
	No. 2	850	450			1150	1,300,000	

Wall Studs

dimension lumber 2" to 4" thick 2" and wider, Use with appropriate Adjustments for size and service conditions

Commercial Lumber Species Group	Grade	Fb (psi)	Ft (psi)	Fv (psi)	Fc-perp (psi)	Fc-para (psi)	E (psi)	Specific Gravity (OD Wt. & OD Vol.)
I	No. 1	1000	675	175	565	1500	1,700,000	0.50
	No. 2	900	550			1350	1,600,000	
	STUD	700	450			850	1,400,000	
II	No. 1	875	450	135	405	1150	1,400,000	0.42
	No. 2	850	450			1150	1,300,000	
	STUD	675	350			725	1,200,000	

Wall Plates

dimension lumber 2" to 4" thick 2" and wider, Use with appropriate Adjustments for size and service conditions

Commercial Lumber Species Group	Grade	Fb (psi)	Ft (psi)	Fv (psi)	Fc-perp (psi)	Fc-para (psi)	E (psi)	Specific Gravity (OD Wt. & OD Vol.)
I	No. 2	900	550	175	565	1350	1,600,000	0.50
	No. 3	525	325			775	1,400,000	
	Standard	575	375			1400	1,400,000	
II	No. 2	850	450	135	405	1150	1,300,000	0.42
	No. 3	500	250			650	1,200,000	
	Standard	550	275			1150	1,200,000	

Timbers - Beams and Columns

Beams and Stringers - 5" and Thicker, Width more than 2" greater than thickness, Use with appropriate Adjustments

Commercial Lumber Species Group	Grade	Fb (psi)	Ft (psi)	Fv (psi)	Fc-perp (psi)	Fc-para (psi)	E (psi)	Specific Gravity (OD Wt. & OD Vol.)
I	Select Structural	1500	950	165	375	950	1,500,000	0.50
	No. 1	1350	675			825	1,500,000	
	No. 2	850	425			525	1,200,000	
II	Select Structural	1300	750	140	405	925	1,300,000	0.43
	No. 1	1050	525			750	1,300,000	
	No. 2	675	350			500	1,000,000	

Posts and Timbers - 5"x5" and Larger, Width not more than 2" greater than thickness, Use with appropriate Adjustments

Commercial Lumber Species Group	Grade	Fb (psi)	Ft (psi)	Fv (psi)	Fc-perp (psi)	Fc-para (psi)	E (psi)	Specific Gravity (OD Wt. & OD Vol.)
I	Select Structural	1500	1000	165	375	950	1,500,000	0.50
	No. 1	1200	825			825	1,500,000	
	No. 2	750	475			525	1,200,000	
II	Select Structural	1200	800	140	405	975	1,300,000	0.43
	No. 1	975	650			850	1,300,000	
	No. 2	575	375			575	1,100,000	

Proposed PRL Commercial LVL Design Properties[a]

Grade/ Species	E (Joist)[b], 10^6 psi	F_b (Joist)[c], psi	F_t [d], psi	$F_{c//}$[e], psi	F_v (Joist)[f], psi	$F_{c\perp}$ (Joist)[g], psi	SG[h]				
							Nails and Wood Screws				Bolts and Lag Screws
							Withdrawal		Lateral		Lateral
							Face	Edge	Face	Edge	Face
1.5E-2250F	1.50	2,250	1,500	1,950	220	575	0.42	0.42	0.42	0.42	0.42
2.0E-2900F	2.00	2,900	1,900	2,750	285	750	0.50	0.47	0.50	0.50	0.50

(a) The tabulated values are design values for normal duration of load. All values, except E and $F_{c\perp}$, are permitted to be adjusted for other load durations as permitted by the code. The design stresses are limited to conditions in which the maximum moisture content is less than 16 percent.

(b) Bending modulus of elasticity (E), which is applicable in either edgewise or flatwise applications, includes shear deflections. For shear-free E, multiply the tabulated value by 1.03.

(c) Allowable bending stress (F_b) is applicable in either edgewise or flatwise applications. For depths of 3-1/2 inches or deeper when loaded edgewise, the tabulated F_b value shall be modified by $(12/d)^{1/6}$, where d is the actual depth in inches. For depths less than 3-1/2 inches when loaded edgewise, use the adjusted F_b for 3-1/2 inches. No adjustment on F_b is required for flatwise applications.

(d) Axial tension (F_t) of the LVL is based on a gauge length of 4 feet. For specimens longer than 4 feet, the tabulated F_t value shall be adjusted by $(4/L)^{1/8}$, where L is the actual length in feet.

(e) Compression parallel to grain ($F_{c//}$) of the LVL.

(f) Allowable shear stress (F_v) of the LVL when loaded edgewise.

(g) Allowable compressive stress perpendicular to grain ($F_{c\perp}$) of the LVL when loaded edgewise.

(h) Equivalent specific gravity for connection design.

Custom Wood Species

Within the program you are also able to define new materials in the Custom Wood Species spreadsheet. You can find this by going to Modify>>Custom Wood Species Database. This spreadsheet exists for those materials not found directly in the NDS. Many standard materials have been added here for your convenience. Any species defined here will appear in the "Species" dropdown in the Wood tab of the Materials spreadsheet.

Custom Wood Properties

	Label	Fb [ksi]	Ft [ksi]	Fv [ksi]	Fc [ksi]	E [ksi]	SCL
1	LVL_PRL_1.5E_2250F	2.25	1.5	.22	1.95	1500	☑
2	LVL_PRL_2.0E_2900F	2.9	1.9	.285	2.75	2000	☑
3	LVL_Microllam_1.9E_2600F	2.6	1.555	.285	2.51	1900	☑
4	PSL_Parallam_2.0E_2900F	2.9	2.025	.29	2.9	2000	☑
5	PSL_Parallam_1.8E	2.4	1.755	.18	2.5	1800	☑
6	LSL_TimberStrand_1.55E_2325F	2.325	1.07	.31	2.05	1550	☑
7	LSL_TimberStrand_1.3E_1700F	1.7	1.075	.4	1.4	1300	☑

Wood - Design

Full code checking can be performed on Dimension Lumber and Post and Timber size wood shapes based on the following codes:

- The 2005 edition of the NDS (National Design Specification)
- The 2001 edition of the NDS
- The 1991 / 1997 editions of the NDS

Note

- When the 1991 / 1997 NDS is selected, the 1991 NDS specification will be used with the 1997 stress tables. This is consistent with the requirements of the 1997 UBC.

Information on the design of "Wood Products", such as wood I-joists, may be found in the Wood Products topic.

Glu-Lams

Glu-Lams are treated as any other wood species and may be selected from the list of species on the Wood Tab of the Materials spreadsheet. When a Glu-Lam is selected, the grade will be listed as "na" or not applicable.

All Glu-Lam members should be dimensioned as "Full Sawn" using the format *wXdFS*, where "w" and "d" are the actual width and depth dimensions. If the size is entered as *wXd* without the *FS* designation, then the size will be dressed down as if the member were regular dimensional lumber.

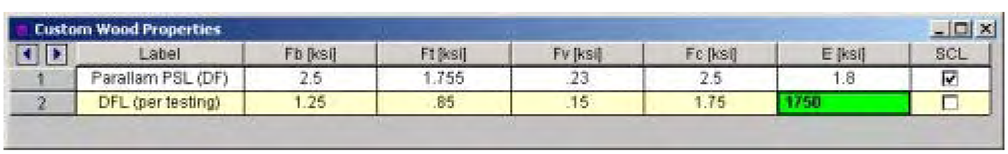
RISA includes two redesign lists for Glu-Lams: Glu-Lam_Western for Western Species and Hardwoods (HW), and Glu-Lam_SouthernPine for Southern Pine (SP/SP).

Note

- Glu-Lams from Table5A are always assumed to have the special tension laminations. Therefore, the Fbx value is not reduced.
- RISA is NOT applying any of the footnotes to Table 5A and 5C at this time except for the values for Fvx and Fvy in Table 5A.

Custom Wood Materials & Structural Composite Lumber

To use a custom wood material that is not part of the standard NDS database, you will need to define the design properties of a new / custom wood species.


Custom Wood Properties

	Label	Fb [ksi]	Ft [ksi]	Fv [ksi]	Fc [ksi]	E [ksi]	SCL
1	Parallam PSL (DF)	2.5	1.755	.23	2.5	1.8	☑
2	DFL (per testing)	1.25	.85	.15	1.75	1750	☐

To do this, select **Spreadsheets** ▸ **Custom Wood Species Spreadsheet** from the main menu toolbar. Enter a label for your new custom species, then enter the wood properties (Fb, Fc, etc.) in the columns to the right. This new material will now be included in between the NDS wood species and the glulam types in the **Species** drop down list on the wood materials spreadsheet.

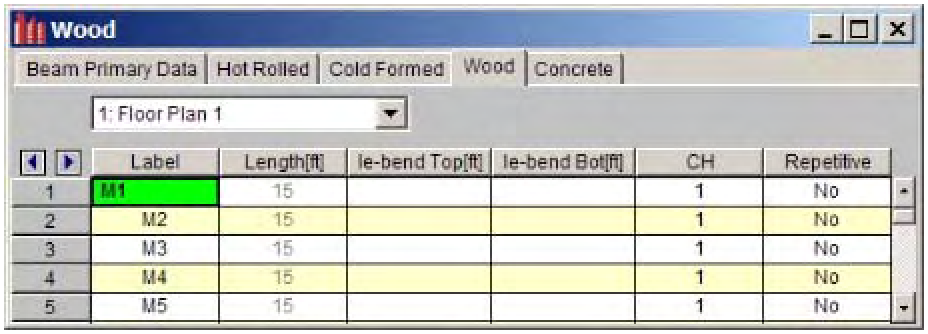
Note

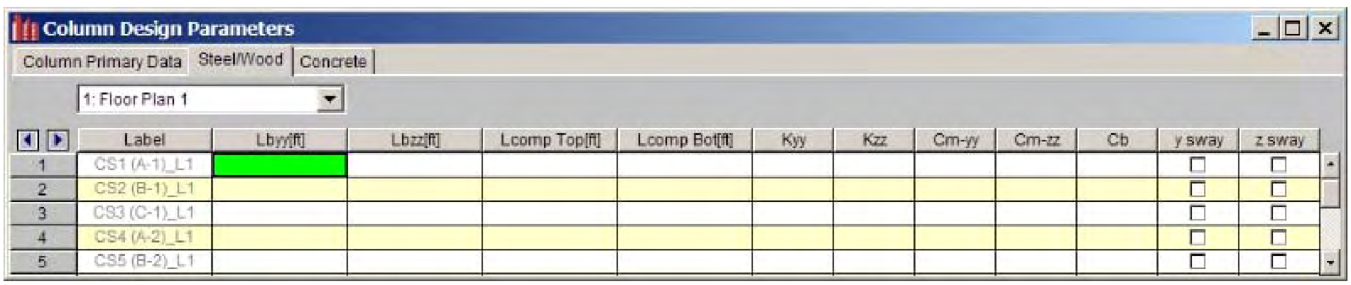
- Fc is the base value for the compressive stress parallel to grain and will be used to calculate the member's ability to resist axial compression.

- The **SCL** checkbox is used to designate whether this new species is Structural Composite Lumber as defined in the 2001/05 NDS. The 2001/05 NDS has a code checking procedure for composite lumber that is slightly different from the procedure used for standard, dimensional lumber.

Wood Member Design Parameters

The **Member Design Parameters** spreadsheet records the design parameters for the material-specific code checks and may be accessed by selecting **Design Parameters** on the **Spreadsheets** menu. These parameters may also be assigned graphically.





These parameters are defined for each member. The pull down list at the top of the spreadsheet allows you to toggle between floors.

Label

You may assign a unique **Label** to all of the members. Each label must be unique, so if you try to enter the same label more than once you will get an error message. The Label field for column members is dictated by the label entry on the **Column Stacks Spreadsheet** and may not be edited here. If you would like to edit this entry, you must do so on the Column Stacks Spreadsheet.

Length

The beam **Length** is reported in the second column. This value may not be edited as it is dependent on the beam start and end points listed on the **Primary Data** tab. It is listed here as a reference for unbraced lengths which are discussed in the next section.

Unbraced Length

You may specify unbraced lengths or have RISAFloor calculate them for you. The unbraced lengths are **Lb**zz, **Lb**yy, **Lcomp-top**, **Lcomp-bot**, **Le-bend-top**, and **Le-bend-bot**.

The values **Lbzz**, and **Lbyy** represent the unbraced length of column members with respect to column type buckling about the member's local z and y axes, respectively. They are listed on the **Steel/Wood** tab of the **Columns Spreadsheet**. These **Lb** values are also used to calculate Le1/d and Le2/b, which in turn impact the calculation of Cp, the column stability factor. These length to thickness ratios gauge the vulnerability of the member to buckling. Refer to Section 3.7 of the NDS for more information on this. This section also lists the limiting values of the length to thickness ratios.

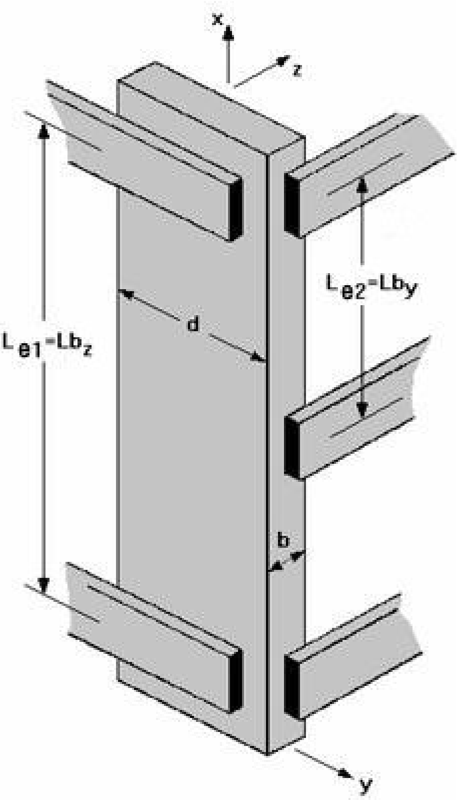
The **Lcomp** values, **Lcomp-top** and **Lcomp-bot**, are the unbraced lengths of the compression flanges of column members for flange buckling due to flexure. They are listed on the **Steel/Wood** tab of the **Columns Spreadsheet**. These may be the same as the **Lbyy** value, but not necessarily. The **Lcomp** value is also used in the calculation of the slenderness ratio, RB, which is used in the calculation of CL, the beam stability factor. CL is then used to calculate the allowable bending stress. Refer to Section 3.3.3.6 in the NDS for more information on this and note that the value of RB is limited to 50.

The **Le-bend** values, **Le-bend-top** and **Le-bend-bot**, are the effective unbraced lengths of beam members for bending. This unbraced length is the length of the face of the member that is in compression from any bending moments. **This value should be obtained from Table 3.3.3 in the NDS code**. The **Le-bend** value is used in the calculation of the slenderness ratio, RB, which is used in the calculation of CL, the beam stability factor. CL is then used to calculate the allowable bending stress. Refer to Section 3.3.3.6 in the NDS for more information on this and note that the value of RB is limited to 50.

For continuous beams the moment will reverse such that the top and bottom faces will be in compression for different portions of the beam span. **Le-bend-top** is the unbraced length of the top face and **Le-bend-bot** is the unbraced length of the bottom face.

If the **Lb** and/or **Lcomp** values are not entered (left blank) for **Column Members**, the unbraced lengths for each segment of a physical column or lift will be automatically calculated by RISAFloor based on the distances between floor levels and/or splices. This means that each physical column can have multiple unbraced lengths if the entry is left blank. However, if a number is entered, RISAFloor will use that value for ALL segments of the physical column. Unbraced lengths that have been set or modified in RISA-3D will also be used in RISAFloor.

When the **Lcomp** values are left blank for **Beam Members**, the deck properties and the members framing into the beam will determine the unbraced length used by the program and will vary along the length of the beam. This means that each beam can have multiple unbraced lengths if the entry is left blank. For the top flange, **Lcomp-top**, the smaller of the deck bracing or member framing distances will be used at each location. For the bottom flange, **Lcomp-bot**, the full length of the beam will be used. If a number is entered, RISAFloor will use that value for ALL segments of the beam.



For beam members, you can enter the code "**Segment**" in the unbraced length fields and the length of the longest segment will be used. A "segment" is the distance between the joints that are on the beam. For example, suppose you have a beam that

is 20 feet in length, and there are two joints along the beam, one 5 feet from the end and one at 15 feet. An unbraced length of 10 feet will be used for the design of the beam. The "segment" code cannot be entered for column members.

Note

- A beam is considered braced at the top wherever another beam frames into it.
- If the intermediate framing members are considered to brace the bottom flange, then you can enter “segment” for Le-bend-bot. When the “segment” command is used ALL intermediate points along the beam are viewed as brace points. Therefore, you may have to delete unused or extraneous points.
- The Top Flange is defined as the flange corresponding to the positive local y axis for the member.
- The calculated unbraced lengths are listed on the **Member Detail** report.

K Factors (Effective Length Factors)

The **K Factors** are also referred to as effective length factors. **Kyy** is for column type buckling about the member's local y-y axis and **Kzz** is for buckling about the local z-z axis.

If a value is entered for a **K Factor**, that value will be used for the segment of the physical column between the current floor level or splice above and/or the floor level or splice below. When in RISA-3D via RISAFloor, the largest K factor entered for any segment of a physical column will be used for the entire physical column. If an entry is not made (left blank), the value will internally default to '1' for that column segment. See the NDS Appendix G for an explanation of how to calculate K Factors. For wood, the **K Factors** are applied to **Lbzz** and **Lby**y to obtain the effective column length. See section 3.7 in the NDS for more on this.

Sway Flags

The **Sway Flags** indicate whether the member is to be considered subject to sidesway for bending about its local y and z axes. The **y sway** field is for y-y axis bending and the **z sway** field is for z-z axis bending. Click on the field to check the box and indicate that the member is subject to sway for that particular direction, or leave the entry blank if the member is braced against sway. These sway flags influence the calculation of the K factors.

Sway flags are reserved for column members in RISAFloor and may be applied to any column segment at any floor level. However, when in RISA-3D via RISAFloor, if a sway flag is checked for any segment of a physical column, the entire physical column will be assumed subject to sway. Beam members in RISAFloor are assumed to be braced against sway.

Timber Design Adjustment Factors

The NDS code has a lot of adjustment factors that you apply to the various allowable stresses, and in some cases, to the Young's modulus (E). All of the adjustment factors are summarized in section 2.3 of the NDS-2005 specification. The following topics help to summarize how adjustment factors are obtained and used. **The CT, Ci, and Cb factors are NOT used.**

Timber Design CH (Shear Stress Factor)

The Shear Stress Factor entry, **CH**, is the shear stress adjustment factor. This design parameter can be set on the Wood tab of the **Beams Spreadsheet**. If left blank the program will use a default value of 1.0. See the tables in the NDS supplement for information on other CH factors. Note that only tables 4A, 4B, and 4D are used. The CH factor is only available for the 1991/1997 NDS codes. For other codes, this entry will be ignored.

Timber Design Cr (Repetitive Factor)

The Repetitive Factor field, **Cr**, specifies if the beam is one of a group of repetitive members. This design parameter can be set on the **Wood** tab of the **Beams Spreadsheet**. If you enter 'Yes' in the Repetitive field, a factor of 1.15 will be applied to beam members that are 2" to 4" thick. This flag will be ignored for a NDS shape that is thicker than 4". A value of '1.0' will

be used for Wood Products. Different restrictions apply to the use of the Cr factor for Structural Composite Lumber and Glu-Lams.

Timber Design Ct (Temperature Factor)

The Temperature Factor, **Ct**, is calculated internally from the wood temperatures set in the Global Parameters. See section 2.3.3 of the NDS-2005 specification for more information on the temperature factor.

Timber Design Cfu (Flat Use Factor)

The Flat Use Factor, **Cfu**, is automatically applied to the weak axis allowable bending stress of a wood member whenever weak axis moments are present. The flat use factor will only be applied to members that are 2" to 4" thick. See the 2005 NDS, Tables 4A, 4B, 4C, 4F, 5A, 5B, 5C, 5D and the footnotes.

Timber Design CF (Size Factor)

The Size Factor, **CF**, is applied automatically when you assign a wood shape from the NDS shape database. See Tables 4A, 4B, 4D, and 4E in the NDS supplement for information on the CF factor.

Timber Design CV (Volume Factor)

The Volume Factor, **CV**, is applied automatically when you assign a Glu-Lam member from the NDS shape database. The user can override the calculated value by inputting the factor on the **Wood** tab of the **Beams Spreadsheet**. This entry is only available for beam members and when using the 2001 or 2005 NDS code.

Note:

- In the calculation of Cv, RISA takes L conservatively as the full length of the member.

Timber Design Cf (Form Factor)

The Form Factor, **Cf**, is applied automatically when designing by the NDS 91/97 or 2001 Specification and a 'Round' shape is selected from the NDS shape database. See section 2.3.8 in the NDS (91/97, 2001) for information on the Cf factor.

Note

- This factor is not applied when design by the NDS 2005 Specification
- This factor is not applied to "diamond" shaped members, which are just rectangular members on edge. This factor is not applied to diamond shapes because any applied moments are transformed internally to the local member axes for the code check calculations, which is the same as applying the "diamond" form factor and NOT transforming the moments.

Timber Design Cm (Wet Service Factor)

The Wet Service Factor, **Cm**, is applied when you check the **Cm** checkbox in the **Materials Spreadsheet**.

Timber Design CP and CL (Column/Beam Stability Factors)

The Column Stability Factor, **CP**, and the Beam Stability Factor, **CL**, are calculated internally. These calculated values are shown on the **Wood** tab of the **Bending Results Spreadsheet**, as well as in the **Member Detail Reports**. See NDS 2005 section 3.3.3 for information on the CL factor and NDS 2005 section 3.7.1 for information on the CP factor.

Timber Design CD (Load Duration Factor)

The Load Duration Factor, **CD**, is entered on the **Load Combination Spreadsheet** for each load combination for which you want wood code check results. The CD factor must be entered for each individual load combination because the CD factor is

dependent on the types of loads that are applied in each load combination. Therefore, different load combinations could have different CD factors. For example, per the NDS 2005 specification, a load combination that had only dead load, would have a CD factor of “0.9”, while another combination that was comprised of dead load plus wind load would have a CD factor of “1.6”.

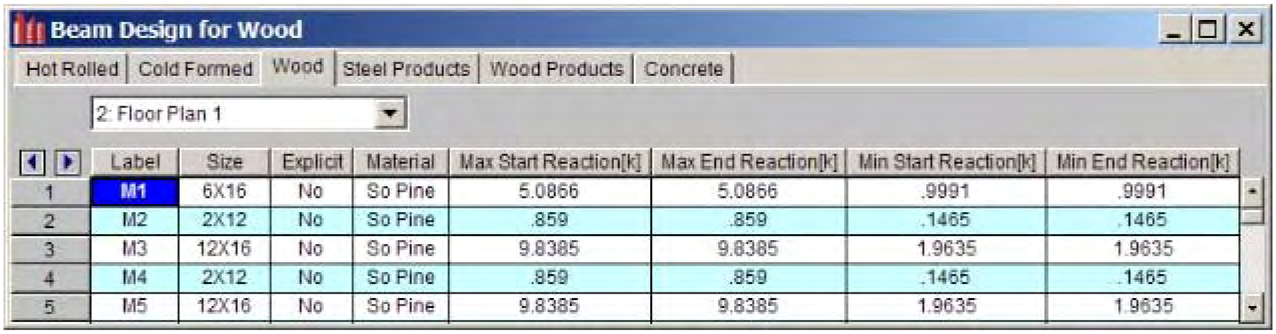
The CD factor will only be applied to wood code checks on wood members. See Table 2.3.2 in the NDS 2005 specification for the CD factors to be applied for typical loads. Appendix B (of the NDS) has additional information about the Load Duration Factor.

Note

- The CD factor used for a load combination should be for the load with the shortest load duration in that load combination.

Beam Design Results

The **Design Results Spreadsheet** displays the optimized design results for the beam elements and may be accessed by selecting **Designs** on the **Results** menu. The spreadsheet has six tabs: Hot Rolled, Cold Formed, Wood, Steel Products, Wood Products, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.



	Label	Size	Explicit	Material	Max Start Reaction[k]	Max End Reaction[k]	Min Start Reaction[k]	Min End Reaction[k]
1	M1	6X16	No	So Pine	5.0866	5.0866	.9991	.9991
2	M2	2X12	No	So Pine	.859	.859	.1465	.1465
3	M3	12X16	No	So Pine	9.8385	9.8385	1.9635	1.9635
4	M4	2X12	No	So Pine	.859	.859	.1465	.1465
5	M5	12X16	No	So Pine	9.8385	9.8385	1.9635	1.9635

The **Label** column lists the beam label.

The **Size** column displays the beam size. When no adequate member could be found from the available shapes list, this field will display the text “not designed”. Consider re-framing, relaxing the design or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Explicit** column displays “Yes” if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes “locked” and will not be automatically redesigned by the program.

Note

- To “unlock” a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.

The **Material** Column displays the material label assigned to the beam.

End Reactions

The **Max Start & End Reactions** column displays the maximum start and end reactions of the beam for ALL load combinations. If “Show Factored End reactions” in **Global Parameters** is left unchecked, these displayed forces are not factored. If it is checked, then the displayed forces will have been multiplied by the factors in the load combinations. The sign convention assigns positive reactions to downward forces. Negative reactions, if they occur, would indicate uplift.

The **Min Start & End Reactions** column displays the minimum start and end reaction of the beam.

Beam Code Checks

The **Code Checks Spreadsheet** summarizes the code check results for the beams and may be accessed by selecting **Code Checks** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.

Beam Code Summary for Wood

Hot Rolled | Cold Formed | Wood | Concrete

2: Floor Plan 1

	Label	Size	Explicit	Material	Bending Check	Loc[ft]	LC	Defl Check	Loc[ft]	Cat	Shear Check	Loc[ft]	LC
1	M1	6X16	No	So Pine	.9011	7.5	2	.555	7.5	LL	.8136	0	2
2	M2	2X12	No	So Pine	.9895	7.5	2	.7153	7.5	LL	.8484	0	2
3	M3	12X16	No	So Pine	.8369	7.5	2	.5136	7.5	LL	.7527	15	2
4	M4	2X12	No	So Pine	.9895	7.5	2	.7153	7.5	LL	.8484	0	2
5	M5	12X16	No	So Pine	.8369	7.5	2	.5136	7.5	LL	.7527	15	2

The **Label** column displays the beam label.

The **Size** column displays the beam size. When no adequate member could be found from the available shapes, this field will display the text “not designed”. Consider re-framing, relaxing the design or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

The **Explicit** column displays “Yes” if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes “locked” and will not be automatically redesigned by the program.

Note

- To “unlock” a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.

The **Material** Column displays the material label assigned to the beam.

Bending, Shear, and Deflection Checks

The **Bending Check** and **Shear Check** columns display the maximum bending check and shear check calculated by the program. This value is equal to the actual bending or shear demand (stress or force) divided by the actually beam resistance (allowable stress or ultimate capacity). You can see the details of these values in the **Bending Results** or **Shear Results** spreadsheet. This check is calculated at 100 locations along each beam for each load combination and the maximum check is reported. See Results Spreadsheets for more information.

The **Deflection Check** displays the maximum deflection check. This value is equal to the ratio of actual deflection to allowable deflection. You can see the details of these values in the **Deflection Results** spreadsheet. This check is calculated at 100 locations along each beam and the maximum check is reported. See Beam Results - Deflection for more information.

The **Location** columns display the location along the member where the maximum bending, shear, or deflection check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum bending or shear check.

Deflection checks are based on Load Categories (Dead, Live or DL+LL), not Load Combinations. Therefore, the controlling Load Category for deflections is reported in the **Cat** column.

Beam Shear Results

The **Shear Results Spreadsheet** records the shear results for the beam elements and may be accessed by selecting **Shear** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.

Beam Shear Results for Wood

Hot Rolled | Cold Formed | Wood | Concrete

2: Floor Plan 1

	Label	Size	F'v[ksi]	fv[ksi]	Shear Check	Loc[ft]	LC
1	M1	6X16	.11	.0895	.8136	0	2
2	M2	2X12	.09	.0764	.8484	0	2
3	M3	12X16	.11	.0828	.7527	15	2
4	M4	2X12	.09	.0764	.8484	0	2
5	M5	12X16	.11	.0828	.7527	15	2

The **Label** column displays the beam label.

The **Size** column displays the beam size. When no adequate member could be found from the available shapes, this field will display the text "not designed". When this occurs, consider re-framing, relaxing the design or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

Shear Stress and Capacity

The **F'v** column displays the calculated allowable shear stress based on Chapter 3 of the applicable NDS code. The **fv** column displays the maximum actual shear stress that the member experiences.

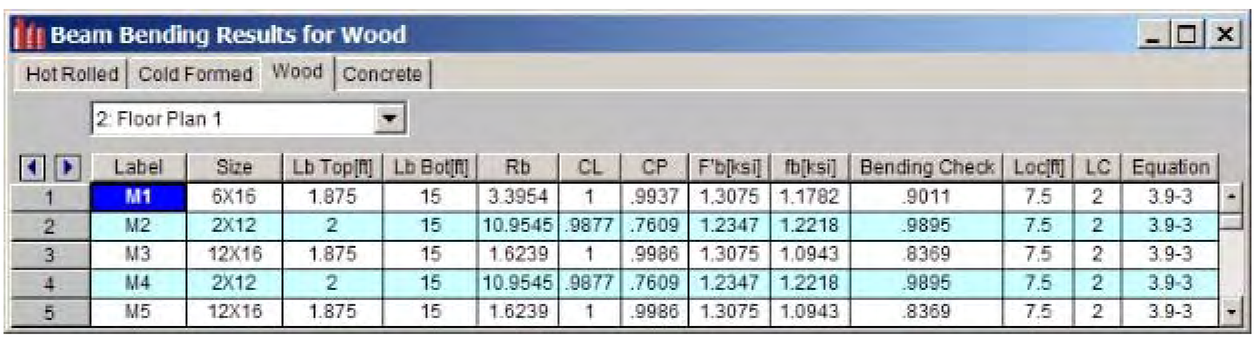
The **Shear Check** column displays the maximum shear check. This value is equal to the actual shear demand (stress) divided by the actual beam resistance (allowable stress). This shear check is calculated at 100 locations along each beam for each load combination. The maximum shear stress, its location, and the controlling load combination are reported. See Results Spreadsheets for more information.

The **Location** column displays the location along the member where the maximum shear check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum shear check.

Beam Bending Results

The **Bending Results Spreadsheet** records the bending results for the beams and may be accessed by selecting **Bending** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete. The pull down list at the top of the spreadsheet allows you to toggle between floors.



Beam Bending Results for Wood

Hot Rolled | Cold Formed | Wood | Concrete

2: Floor Plan 1

	Label	Size	Lb Top[ft]	Lb Bot[ft]	Rb	CL	CP	F'b[ksi]	fb[ksi]	Bending Check	Loc[ft]	LC	Equation
1	M1	6X16	1.875	15	3.3954	1	.9937	1.3075	1.1782	.9011	7.5	2	3.9-3
2	M2	2X12	2	15	10.9545	.9877	.7609	1.2347	1.2218	.9895	7.5	2	3.9-3
3	M3	12X16	1.875	15	1.6239	1	.9986	1.3075	1.0943	.8369	7.5	2	3.9-3
4	M4	2X12	2	15	10.9545	.9877	.7609	1.2347	1.2218	.9895	7.5	2	3.9-3
5	M5	12X16	1.875	15	1.6239	1	.9986	1.3075	1.0943	.8369	7.5	2	3.9-3

The **Label** column displays the beam label.

The **Size** column displays the beam size. When no adequate member could be found from the available shapes, this field will display the text "not designed". When this occurs, consider re-framing, relaxing the design or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

Unbraced Lengths

The **Lb Top** and **Lb Bottom** columns display the unbraced lengths associated with the controlling bending check. See Unbraced Lengths for more information on how these values are calculated.

Wood Slenderness and Stability Factors

The **Rb** column displays the slenderness ratio of the beam per equation 3.3-5 of the NDS code. The **CL** column displays the beam stability factor per equation 3.3-6 of the NDS specification. The **CP** column displays the column stability factor per equation 3.7-1 of the NDS specification. The column stability factor, CP, is not used for beam members in RISAFloor, only reported.

Bending Stress and Capacity

The **F'b** column displays the calculated allowable bending stress based on Chapter 3 of the applicable NDS code. The **fb** column displays the maximum actual bending stress that the member experiences. The sign convention is defined so that positive bending will result in tension in the bottom fiber.

The **Bending Check** column displays the maximum bending check calculated by the program. This check is equal to the actual demand (bending stress) divided by the beam resistance (allowable stress). This bending check is calculated at 100 locations along each beam for each load combination and the maximum demand (stress), the location, and the controlling load combination are reported. See Results Spreadsheets for more information.

The **Location** column displays the location along the member where the maximum bending check occurs.

The **LC** column displays the controlling load combination which resulted in the maximum bending check.

The **Equation** column displays the code equation that controlled in the calculation of the bending check.

Column Results

The **Column Results Spreadsheet** summarizes the code check results and records the design results for columns and may be accessed by selecting **Column Results** on the **Results** menu. The spreadsheet has four tabs: Hot Rolled, Cold Formed, Wood, and Concrete.

Wood Column Code Checks

Hot Rolled | Cold Formed | Wood | Concrete

	Stack	Lift	Shape	Code Check	Elev[ft]	lc	Shear Check	Elev[ft]	Dir	lc	Fc' [ksi]	Ft' [ksi]	Fb1' [ksi]	Fb2' [ksi]	Fv' [ksi]	Eqn
1	C1 (A-1)	1	6X6	.875	0	2	.000	0	z	2	.501	.9	1.35	1.35	.11	3.6.3
2		2	4X4	.979	15	2	.000	15	z	2	.499	1.05	1.85	1.85	.1	3.6.3
3	C2 (B-1)	1	8X8	.790	-3	2	.000	-3	z	2	.554	.9	1.35	1.35	.11	3.6.3
4		2	5X6	.811	15	2	.000	15	z	2	.554	.9	1.35	1.35	.11	3.6.3
5	C3 (C-1)	1	8X8	.790	-3	2	.000	-3	z	2	.554	.9	1.35	1.35	.11	3.6.3

The **Stack** column displays the column stack label.

The **Lift** column displays the lift number for the physical column. Lift No. 1 is the lowermost physical column in a stack and the lifts are numbered sequentially moving up the column stack.

The **Shape** column displays the physical column size. When no adequate member could be found from the available shapes, this field will display the text “not designed”. Consider re-framing, relaxing the design or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists).

Code Checks

The **Code Check** column displays the maximum combined axial and bending check calculated by the program. This value is equal to the actual combined axial and bending demand (stress or force) divided by the actual column resistance (allowable stress or ultimate capacity). You can see the details of this value in the subsequent **Axial Resistance (Fc'** or **Ft')** and **Flexural Resistance (Fb1' & Fb2')** columns of this spreadsheet. This check is calculated at 100 locations along each physical column for each load combination and the maximum check is reported. See Results Spreadsheets for more information.

The **Shear Check** column displays the maximum shear check calculated by the program. This value is equal to the actual shear demand (stress or force) divided by the actual column resistance (allowable stress or ultimate capacity). You can see details of this value in the subsequent **Shear Resistance (Fv')** column of this spreadsheet. This check is calculated at 100

locations along each column for each load combination and the maximum check is reported. See Results Spreadsheets for more information.

The **Elev** columns displays the absolute elevation along the column stack where the maximum code check occurs.

The **LC** column displays the controlling load combination which produced the maximum code check and/or shear check.

The **Dir** column displays the column local axis along which the maximum shear check occurs.

Column Stress and Capacity

The **Fc'** column displays the calculated factored allowable axial compressive stress based on Chapter 3 of the applicable NDS specification.

The **Ft'** column displays the calculated factored allowable axial tensile stress based on Chapter 3 of the applicable NDS specification.

The **Fb1'** and **Fb2'** columns display the calculated factored allowable bending stress based on Chapter 3 of the applicable NDS specification.

The **Fv'** column displays the calculated factored allowable shear stress based on Chapter 3 of the applicable NDS specification.

The **Eqn** column displays the code equation used in the calculation of the controlling code check.

Special Messages - Wood Design

In some instances code checks are not performed for a particular member. A message explaining why a code check is not possible will be listed instead of the code check value. You may click the cell that contains the message and look to the status bar to view the full message. Following are the messages that may be listed:

NDS Code Check Not Calculated

This is the general message displayed when code checks were not performed for a member.

RB value is greater than 50

Section 3.3.3.7 of the NDS 1991/1997, 2001 and 2005 codes limits the slenderness ratio RB to a maximum of 50. You need to reduce the effective span length, increase the thickness of the shape, or reduce the depth of the shape.

le/d is greater than 50

Section 3.7.1.4 of the NDS 1991/1997, 2001 and 2005 codes limits the column slenderness ratio of Le1/b or Le2/d to a maximum of 50. You need to reduce your effective length by reducing the actual length between supports or changing the effective length factor “K”. You can also use a thicker shape.

fc is greater than FcE1

Section 3.9.3 of the NDS 1991/1997, 2001 and 2005 codes limits the actual axial compressive stress to be less than the term FcE1. This term is approximately the Euler buckling stress for buckling about the strong axis of the member. (Buckling is in the plane of bending)

fc is greater than FcE2

Section 3.9.3 of the NDS 1991/1997, 2001 and 2005 codes limits the actual axial compressive stress to be less than the term FcE2. This term is approximately the Euler buckling stress for buckling about the weak axis of the member. (Buckling is in the plane of bending)

fb1 is greater than FbE

Section 3.9.3 of the NDS 1991/1997, 2001 and 2005 codes limits the actual strong axis bending compressive stress to be less than the term FbE. This term is approximately the lateral buckling stress.

Limitations - Wood Design

It is assumed that the axial load on the member is occurring through the member's shear center. This means local secondary moments that may occur if the axial load is not applied through the shear center are not considered.

Buckling Stiffness Factor - The buckling stiffness factor, CT, is not currently accounted for.

Incising Factor - The incising factor, Ci, is not currently accounted for.

Bearing Area Factor - The bearing area factor, Cb, is not currently accounted for.

Column Stability Coefficient for bolted and nailed built-up columns - This factor (K_f) is always taken as 1.0 (See NDS 15.3)

Wall Panels

The wall panel element allows you to easily model walls for in plane and out of plane loads. Wall panel data may be viewed and edited in two ways: graphically in the Wall Panel Editor or in the **Wall Panels Spreadsheet**.

Drawing Wall Panels

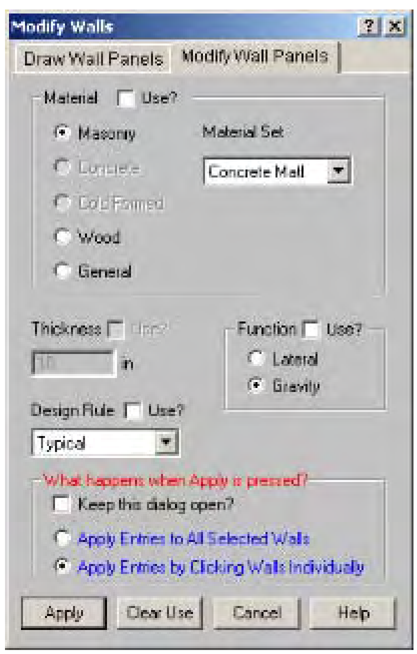
The **Draw Wall Panels**  button lets you graphically draw wall panels in your model. Enter the appropriate wall panel parameters, click **OK** and draw wall panels between along the project grid, or the drawing grid. You will also notice that the coordinates of the joint or grid point that is closest to your cursor are displayed in the lower right hand corner of the model view. The new wall panels will be shown on screen and will be recorded in the **Wall Panels Spreadsheet**.

Modifying Wall Panels

There are a number of ways to modify wall panels. You may view and edit the member data in the **Wall Panel Spreadsheet**, you may double-click a wall panel to view and edit its properties, or you can use the **Modify Wall Panels** tool to graphically modify a possibly large selection of panels.

The graphical **Wall Panel Modify** tool discussed here lets you modify the properties of wall panels that already exist in your model. To use this, you will typically specify the properties you want to change, then select the wall panels that you want to modify. You can modify wall panels one at a time by selecting the **Click to Apply** option and then click on the wall panels you wish to modify. You may also modify entire selections of wall panels by selecting the wall panels first and then use the **Apply to Selected** option. See the Graphic Selection topic for more on selecting.

The parameters shown are the same as those used to define new wall panels.



The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected wall panels. If the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on members without affecting all the rest of the properties. Note that if a no value is entered in a field

(i.e. the field is blank) and the corresponding check box is checked, clicking "Apply" will have the effect of clearing the data for these fields.

To Modify Wall Panels.

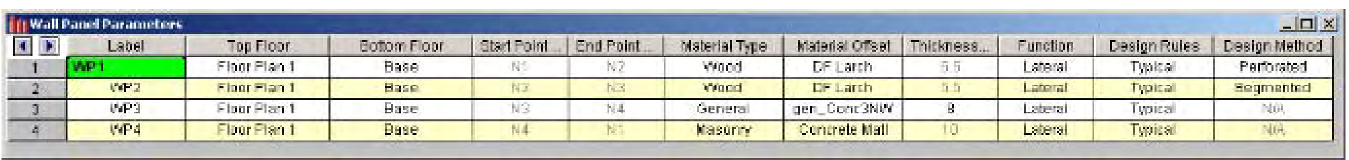
1. Click the **Draw / Modify Wall Panels** button and select the **Modify Wall Panels** tab. Then set the parameters for the new wall panels. Check the **Use?** Box for the items to apply.
2. You may choose to modify a single wall panel at a time or to an entire selection of wall panels.
    1. To modify a few wall panels choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the wall panels with the left mouse button.
    2. To modify a selection of wall panels, choose **Apply Entries to All Selected Items** and click **Apply.**

Note

- To modify more wall panels with different parameters, press CTRL-D to recall the **Modify Wall Panels** settings.
- You may also modify wall panels in the **Wall Panels Spreadsheet**.
- You may undo any mistakes by clicking the **Undo** button.
- The thickness option is only available if you are choosing a General material. Wood and masonry require you to change their thickness in the Design Rules spreadsheet.

Wall Panel Spreadsheets

Another way of adding or editing wall panels is through the **Wall Panel Spreadsheet**. This spreadsheet is accessible through the **Data Entry Toolbar** and includes primary joint, material, and thickness data.



Wall Panel Parameters

	Label	Top Floor	Bottom Floor	Start Point ...	End Point ...	Material Type	Material Offset	Thickness...	Function	Design Rules	Design Method
1	WP1	Floor Plan 1	Base	N1	N2	Wood	DF Larch	5.5	Lateral	Typical	Perforated
2	WP2	Floor Plan 1	Base	N2	N3	Wood	DF Larch	5.5	Lateral	Typical	Segmented
3	WP3	Floor Plan 1	Base	N3	N4	General	gen_Conc3NW	8	Lateral	Typical	N/A
4	WP4	Floor Plan 1	Base	N4	N1	Masonry	Concrete Mall	10	Lateral	Typical	N/A

The following data columns hold the primary data for the wall panels:

Wall Panel Labels

You may assign a unique label to any or all of the wall panels. You can then refer to the wall panel by its label. Each label has to be unique, so if you try to enter the same label more than once you will get an error message. You may relabel wall panels at any time with the Relabel Wall Panels option on the Tools menu.

Top/Bottom Floor

This defines what level in the structure the wall panel starts and stops. This can be edited in this window.

Start/End Point

This shows the start and end joint that defines the extreme ends of the wall. This is for display only. You must delete and redraw wall panels to modify these values.

Wall Panel Material Type and Material Set

The material set label links the wall panel with the desired material defined on the Material Spreadsheet.

Note

- Currently wall panels can only be made up of masonry, wood, or general materials.

Wall Panel Thickness

The thickness field on the Wall Panels Spreadsheet is the thickness of the element. This thickness is constant over the entire element. Note that the thickness for Masonry and Wood wall panels are set in the Design Rules spreadsheet.

Wall Panel Function

If the function is defined as Lateral, then this member will be brought into RISA-3D for the lateral design.

Design Rule

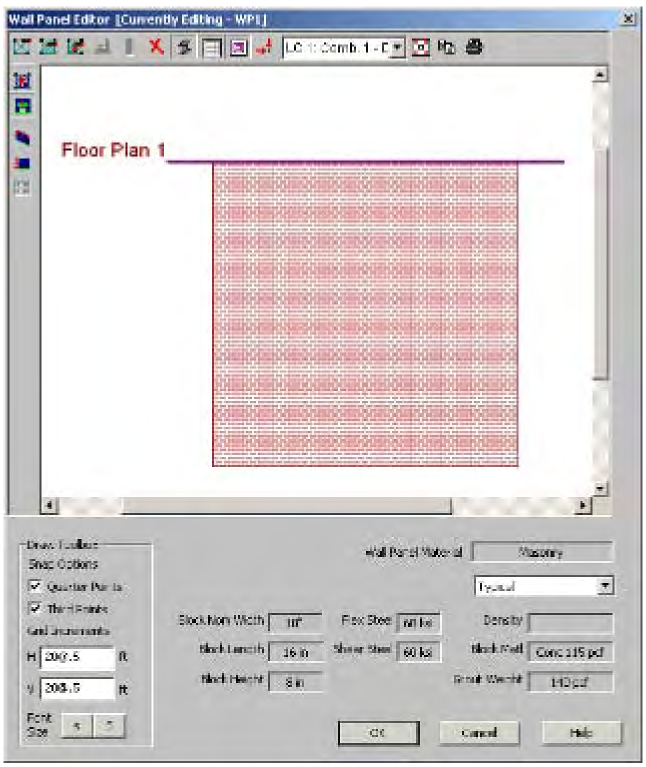
This allows you to choose a specific design from the Design Rules spreadsheet.

Design Method

This is a column specific to wood wall panels and allows you to choose which design method you choose to work with: Segmented, Perforated or Force Transfer. See the Wood Wall Panels topic for more information. These design methods are not applicable for masonry or general wall panels.

Wall Panel Editor

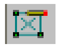
The **Wall Panel Editor** allows the user to edit the detailed properties of a wall panel including openings, regions, boundary conditions, end releases, and hold downs straps. This also gives design options and details for the specific panel. This application is accessible by double clicking on an existing wall panel.



Note:

- There are many icons, dropdown lists and information shown depending on the type of wall panel you are working with. See the Masonry Wall - Design and Wood Wall - Design topics for more information.

Creating Openings

Within the **Wall Panel Editor**, you have the option of adding rectangular openings to the wall panel. To draw an opening, select the **Create New Openings**  button and then select two nodes or grid intersections which make up the two diagonal corners of your opening. Notice that you can view your cursor coordinates in the lower right portion of your screen. To exit this tool right-click your mouse.

Note:

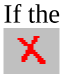
- Drawing an opening in a masonry wall will create a lintel above the opening. For more information on defining lintel geometry and design properties, see the **Masonry Wall - Design** topic on lintels.
- Drawing an opening in wood wall will create a header above the opening. For more information on defining the header properties, see the **Wood Wall - Design** topic on headers.
- When drawing an opening in a general wall panel, there is no header/lintel automatically created. This is because the program does not currently support concrete wall rebar design. The general wall panel is given as an option for analysis only. In a future release wall panel reinforcing design will be implemented.
- Openings can not overlap a region. Regions must be deleted before you draw an opening in an area. After the opening is created you can go back and redraw the regions.

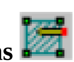
Creating Regions

Within the Wall Panel Editor, you also have the option of creating different rectangular regions within your wall panel. Regions are used to further define areas of your wall panel for use in analysis/design. If you do not specify a region in a wall panel without openings, then the entire wall panel will be considered a region.

To automatically draw regions you must first have your openings input. Once you have that you can click the **Generate Wall Regions Automatically**  button and the program will define regions as we would expect a user to want them.

Note:

- If the regions defined are not located correctly by the generator, you can delete the generated regions with the Delete  button and redraw them manually. See below for more information on this.

To manually draw a region, select the **Create New Regions**  button and use your cursor to select two nodes or grid intersections which make up the diagonal corners of the region. To exit this tool right-click your mouse.

Note:

- For masonry wall panels, there is a region editor that allows you to define design properties for the region. See the **Masonry Wall - Design** topic in the General Reference Manual for region information. Note that design and analysis results are displayed by region.
- For wood wall panels using the Segmented method of design, the design and analysis results are displayed by region. The other options, perforated and force transfer around openings, use regions but don't use them for display of results.
- For general wall panels we will not do any design for you. However, you can lay out your regions so that your analysis results will allow you to design your general wall panels much easier.

Draw Toolbox

The draw toolbox, which appears in the lower left corner of the Wall Panel Editor screen gives the user options for drawing within the Wall Panel Editor window. The options include:

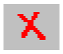
Snap Options allows you to provide snap points at the edges of the wall panel at quarter and third points.

Grid Increments allow you to set a drawing grid within the Wall Panel Editor separate from that in the main model view that you can snap to when drawing openings and regions.

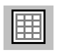
Font Size allows you to increase or decrease the font size associated with the region and opening titles and information shown in the Wall Panel Editor window.

View Controls

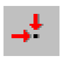
In addition to the wall panel editing tools, the **Wall Panel Editor** window includes the following view controls:

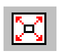
 **Delete** allows you to delete openings or regions from the wall panel.

 **Render** will turn rendering of the current model view on or off, depending on the current setting.

 **Drawing Grid** will turn the display of the Drawing Grid on or off, depending on the current setting.

 **Diaphragm Display** will turn the display of the Diaphragms on or off, depending on the current setting.

 **Loads** will turn the display of the wall panel loads on or off, depending on the current setting.

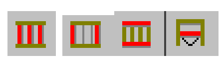
 **Redraw Full Wall View** redraws the wall panel to fit within the **Wall Panel Editor** window.

 **Copy Current View** makes a copy of the current view and saves it to the clipboard.

 **Print Current View** prints your current wall panel view.

Note:

- There are also view controls specific to wood. For more information see the Wood Wall - Design topic.



Meshing the Wall Panels

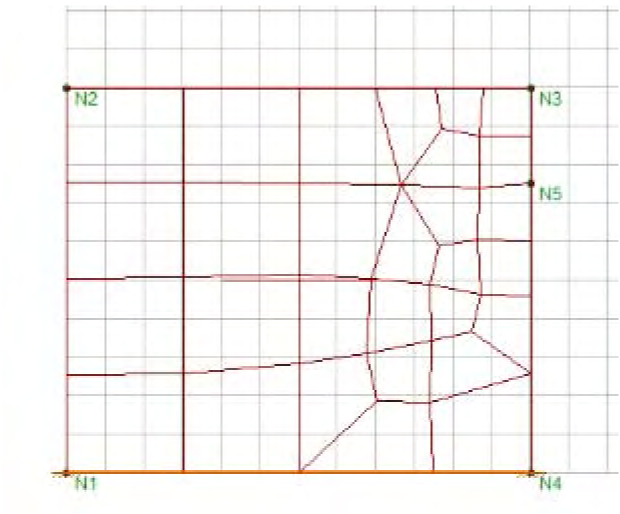
At solution time, the wall panels will be automatically meshed into quadrilateral plate elements. Unlike the plate elements created directly by the user, the automatically generated plate elements are transient in the program and will not be saved in the input file.

The wall panel meshing is treated similar to analysis results. When the results of an analysis are deleted, the wall panel mesh is cleared to be re-built during the next solution. When a solution results file is saved, the meshed elements will be included in that file.

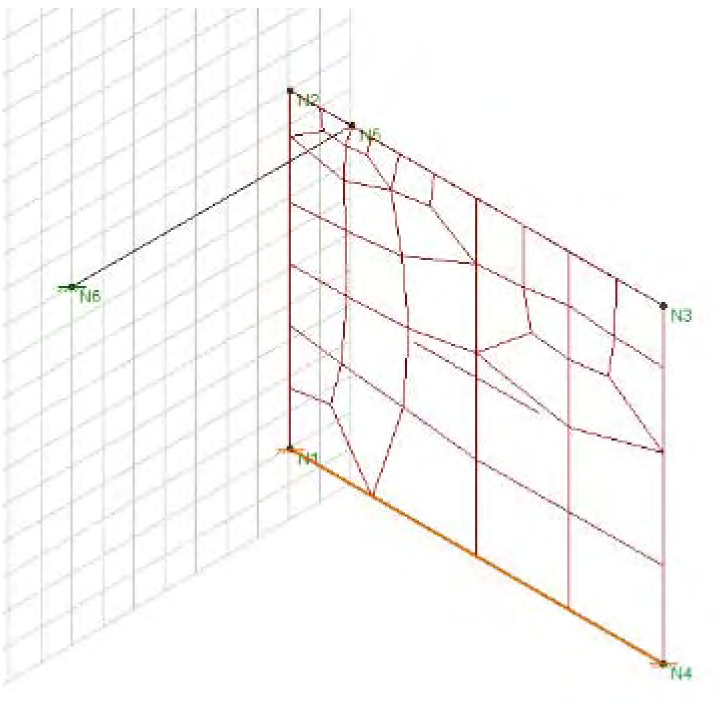
Point Constraints for the Panel Mesh

Point constraints are the locations within the wall panel that require connectivity to the meshed plate elements. The program will automatically generate point constraints at the following locations:

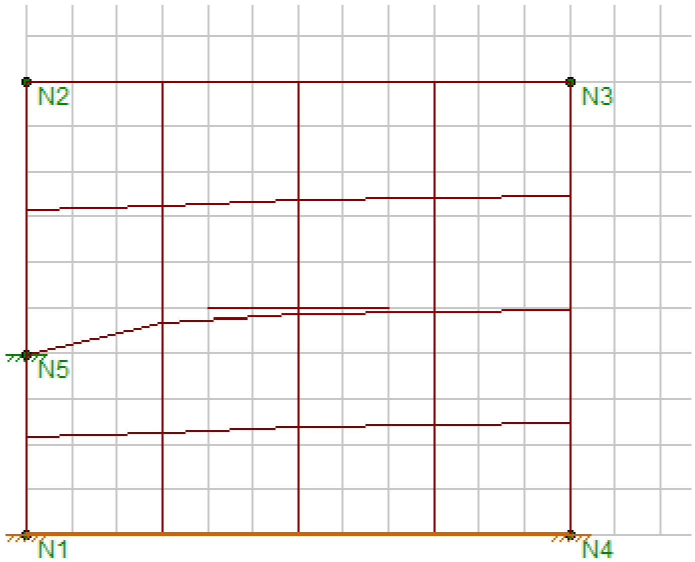
1. Location of an existing joint on the wall panel edges or inside of the wall, region boundaries and opening boundaries.



2. Where beams intersects the wall panels (out-of-plane) on the wall panel edges, region boundaries and opening edges.



3. Location of an external boundary condition.



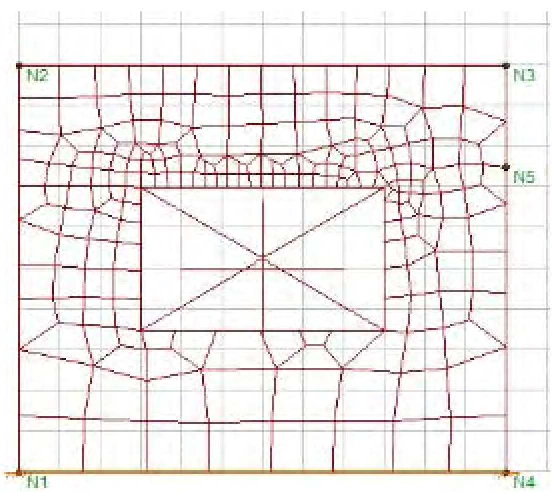
Note:

- Unattached joints that are located on the wall panels can be considered as point constraints and prolong the meshing time. It is highly recommended that the user delete any unattached joints before solving.

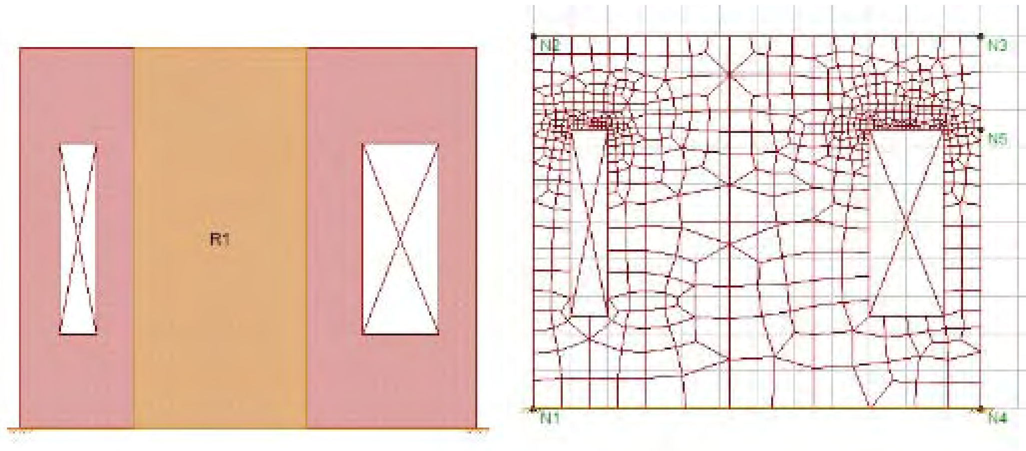
Line Constraints for the Panel Mesh

Line constraints are the locations within the wall panel that require continuous connectivity to plate edges rather than a single point. The program will automatically generate line constraints at the following locations:

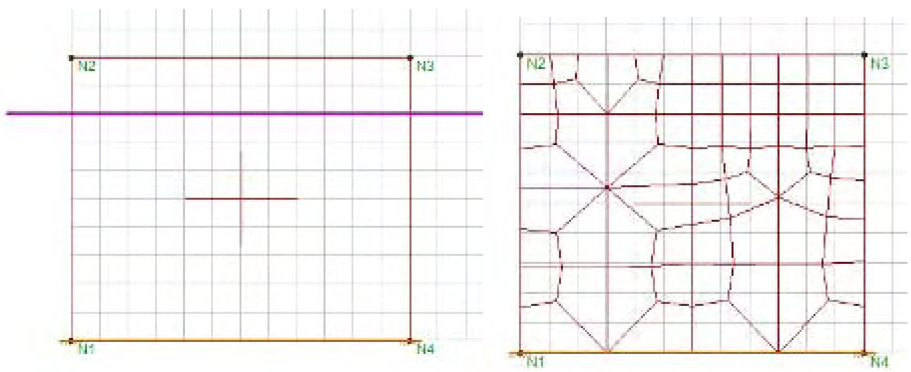
1. Opening edges.



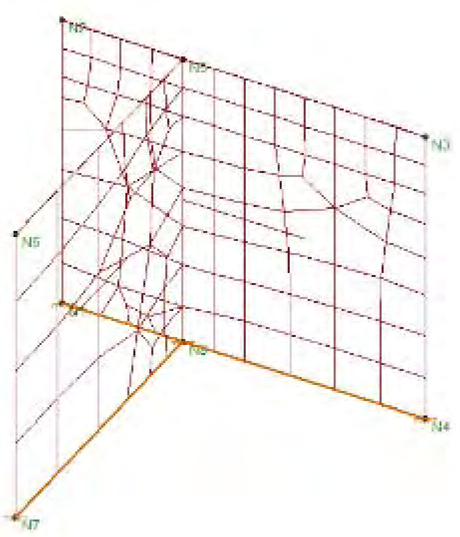
2. The edge and vertical centerline of a defined region.



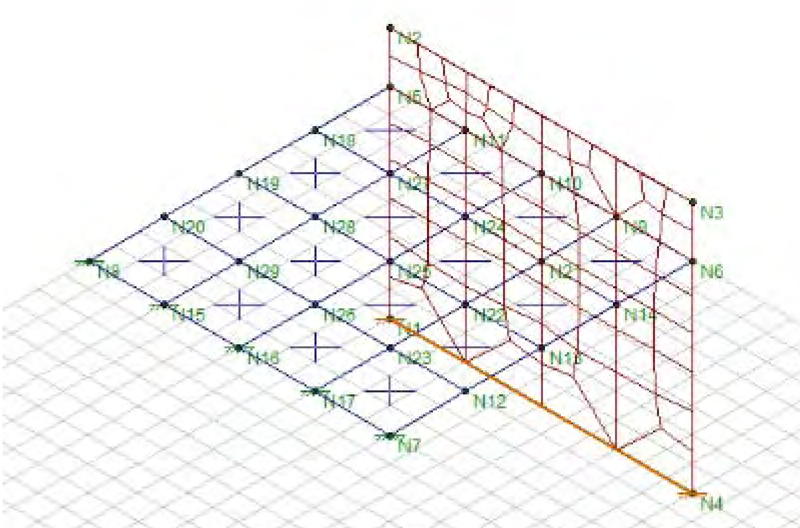
3. Where a diaphragm intersects the wall panel.



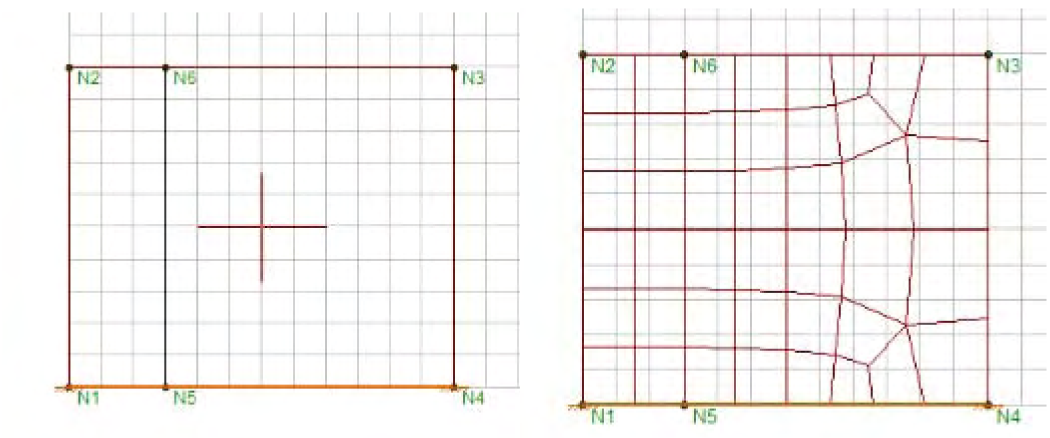
4. At the intersection of multiple wall panels.



5. Where a plate element mesh intersects the wall panel.



6. Where a beam or column element intersects within the plane of the wall panel.



Tips for Ensuring a Healthy Mesh

In order to generate an efficient mesh that gives accurate results, it is critical to place the line constraints and point constraints correctly. If line constraints or point constraints are very close to each other, the auto mesher will be forced to generate small sized elements in order to satisfy the constraints. Therefore, a large number of plate elements will be generated and the solution will be slowed down significantly.

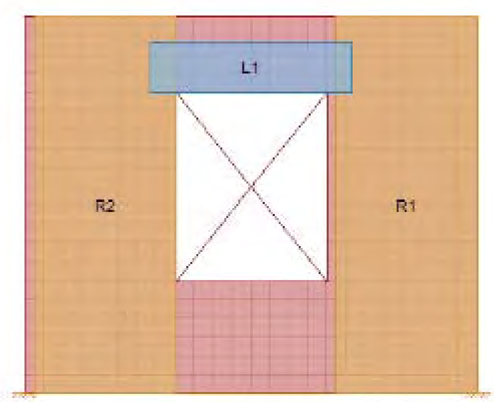
The following guidelines should be followed to ensure a quality mesh:

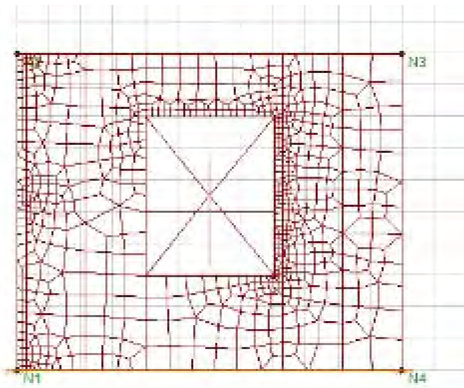
- Avoid generating very narrow regions and openings.
- Avoid small offsets between the external boundary conditions with the location of the region boundaries and wall boundaries.
- Avoid small offsets between opening edges with the region boundaries .
- When a wall panel is intersected by another wall panel, diaphragms, beams or plate elements, keep in mind that the intersection is a line/point constraint. Avoid the small offsets between intersections with the region boundaries or opening edges inside the wall panel.

Example #1: Region Boundaries

1. The left boundary region R1 is placed very close to the opening but not on the opening.
2. The left boundary of region R2 is placed very closed to the left edge of the wall but not exactly on the wall boundary.

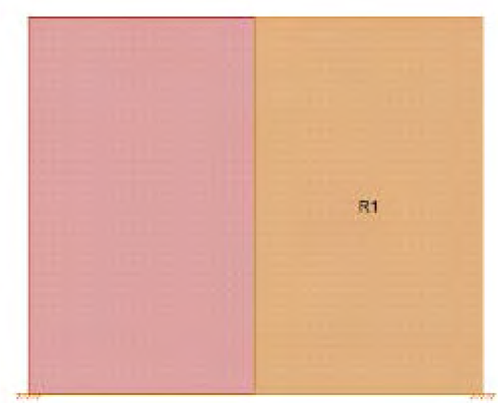
In order to satisfy the line constraints required by the opening edge, region boundaries, and wall boundaries, the program is forced to generate very small meshes in the portion of the wall adjacent to these constraints.

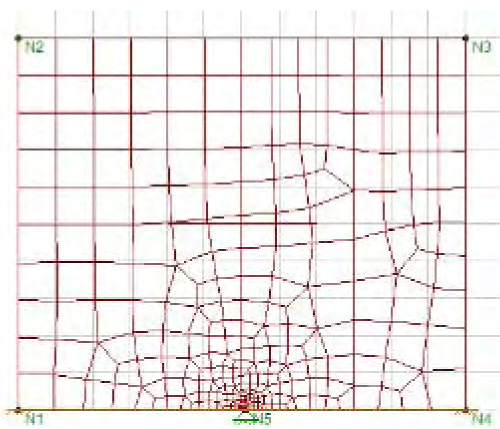




Example #2: Poorly Located Boundary Conditions:

The left boundary of region R1 is at the vertical center line of a wall panel. At the same time, the user placed an external boundary condition at the bottom of the wall panel, which is slightly offset from the center line. In order to accommodate the line constraint of the region boundary and the point constraint of the external boundary condition, the automesher is forced to generate a very small mesh adjacent to these constraints.





Merge Tolerance for Auto-Correction of Mesh

If the distance between the line constraints and point constraints are smaller than the merge tolerance specified on the Global parameters (which defaults to 0.12 inches) then the automesher will automatically snap the constraints together during the meshing. This can eliminate some of the meshing issues that occur in the examples above.

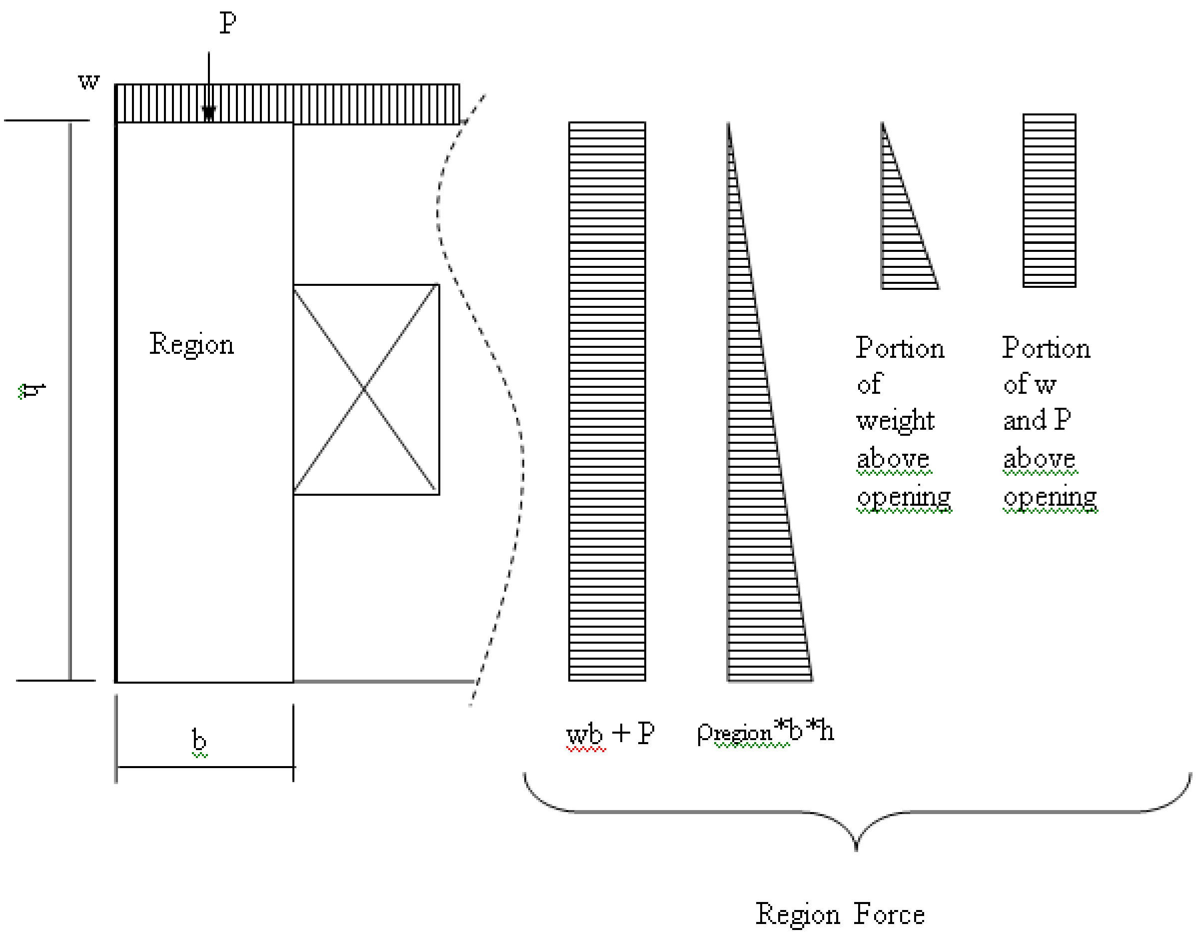
Wall Panel Load Attribution and Load Transfer

Wall Panel Load Attribution

Region Force Calculation

The region force will only be calculated when a region is defined. The region forces were calculated from three types of loads:

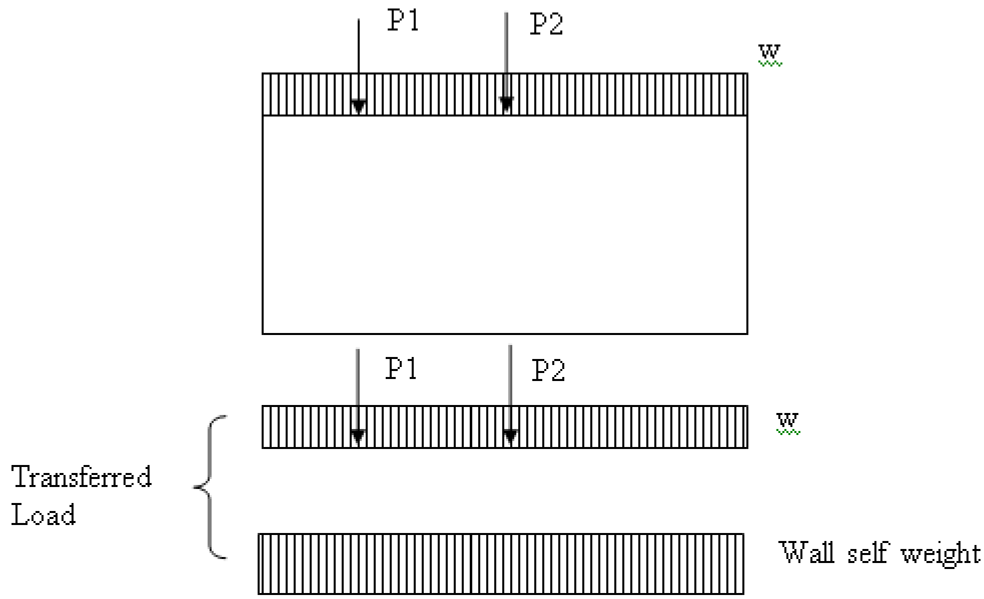
- The line load and point load at floor level that is right above the region.
- The transferred load from the lintel/header which includes part of the self weight above the lintel and part of the load above the opening.
- The weight of the region itself.



Wall Panel Load Transfer

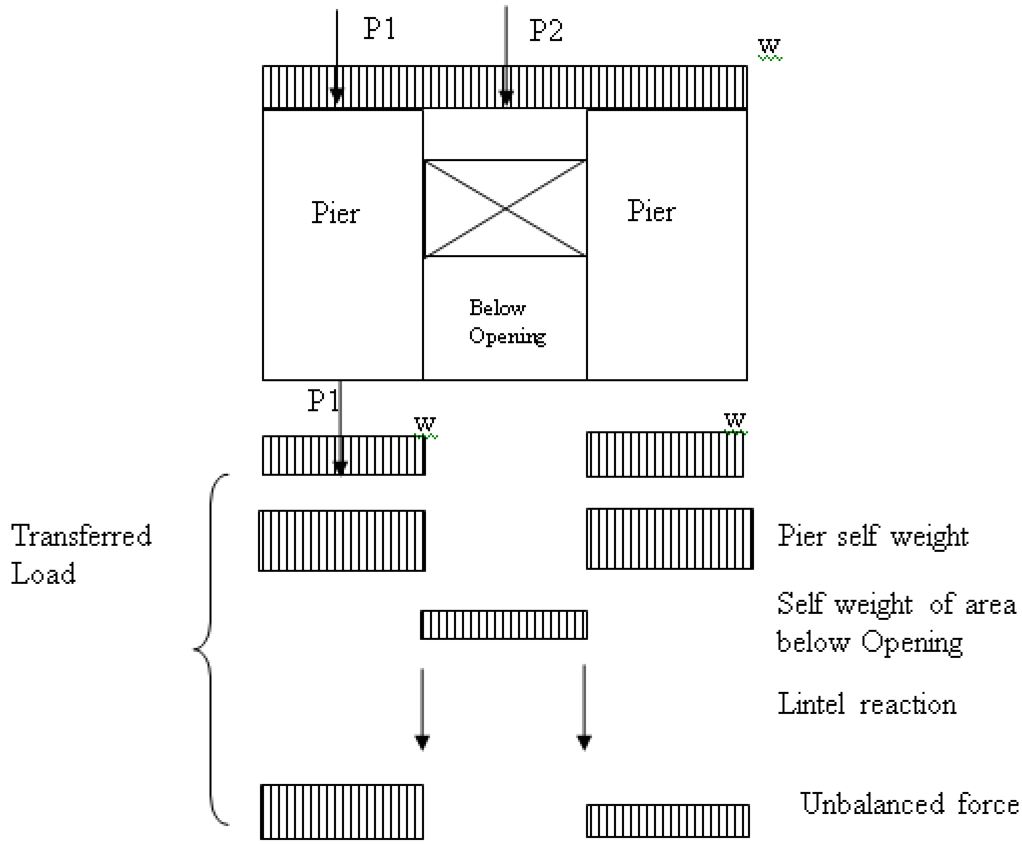
No opening

If there is no opening on the wall, the load on the floor level will be transferred intact to the floor below. The wall self weight will also be added to the load.



With Opening

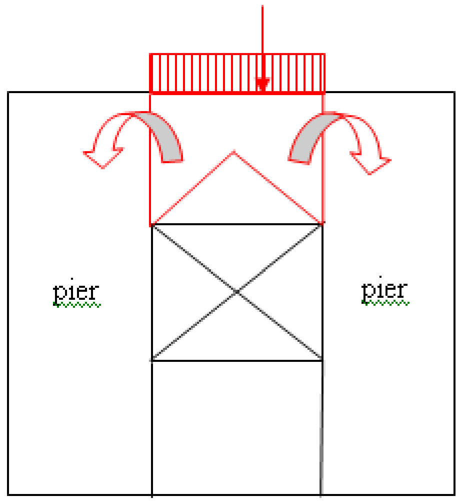
If there are openings on the wall, the load transfer to the floor below is calculated as in the following picture.



The load above the opening will be truncated. The weight above the opening will not be transferred. The lintel reactions will be calculated from the lintel/header forces attribution the opening/lintel/header and will be transferred to the floor below.

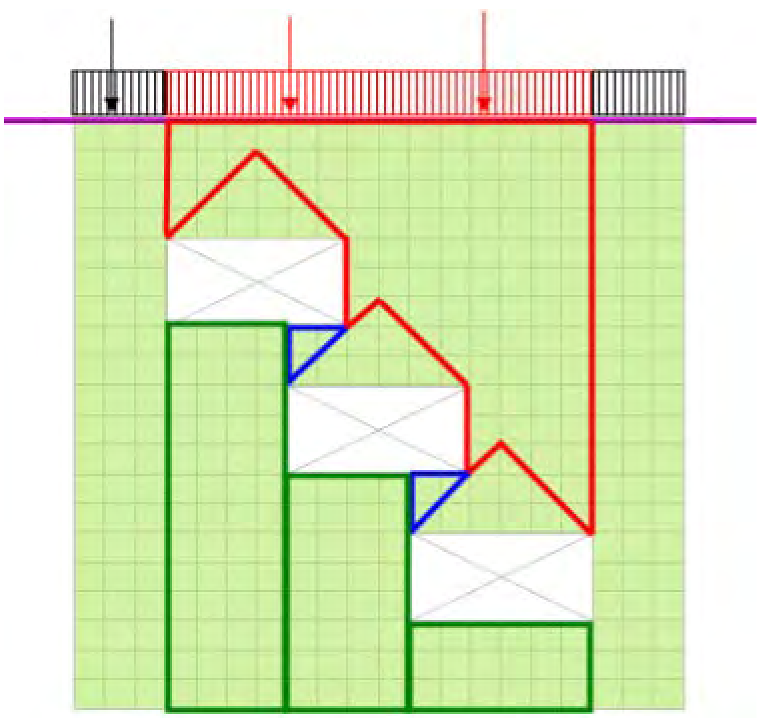
If the 45 degree triangle above the lintel crosses the floor level, there will be no arching effect, the load and self weight right above the opening will be equal to the lintel reaction. In this case the truncated loads and the self weight will be equal to the lintel reaction and there will be no unbalanced forces.

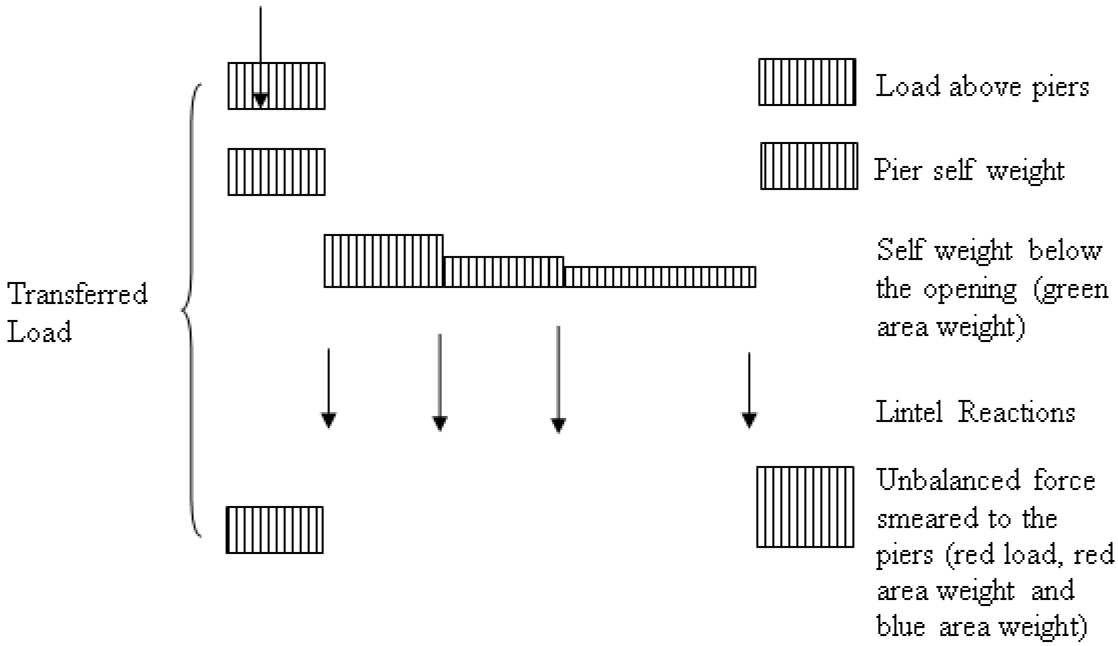
However, if the 45 triangle above the lintel/header does not cross floor level and arching effect is present, the lintel reaction will be smaller than the total wall self weight and loading. There will be an unbalanced force which equals to the weight of the red area plus the red loads in the following picture. The unbalanced force is smeared into the adjacent piers as additional uniformly distributed load.



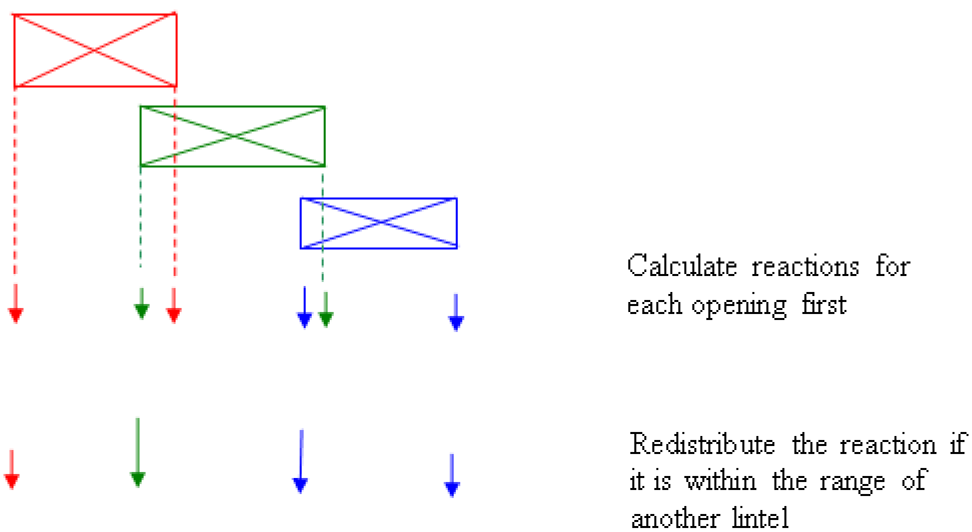
Cascading Opening

More complicated load transfer can happen when openings overlap each other in the horizontal direction as shown in the following picture. Like the wall with a single opening, the self weights of piers and the area below the opening (green box on the picture) will still be transferred as uniformly distributed load. The loads that are right above the pier (load displayed black in the following picture) will still be transferred without any change. If the 45° triangle does not cross the floor level, the wall self weight within the 45° triangle will be transferred as lintel reactions. The wall self weight above the 45° triangle and the load above (the load displayed in red and self weight enclosed in the red and blue lines) will be smeared into the bounding piers. If the 45° triangle crosses the floor level, the self weight above the opening and load above the opening will all be included as lintel reactions.





In the case of cascading opening, the interaction of the lintel/header reaction is also considered during the calculation. The individual lintel/header reactions for each opening were first calculated. Then, starting from the top opening, the lintel/header reactions are recalculated if the lintel/header support is within the range of other lintels/headers.

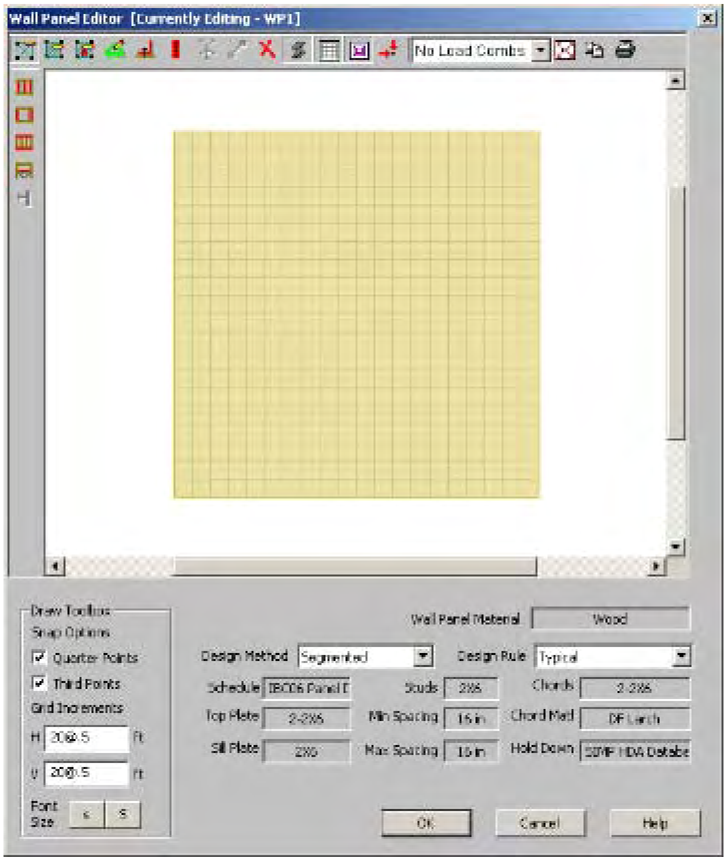


Wood Wall - Design

The wood wall panel element allows you to easily model, analyze and design wood walls for in plane loads. Here we will explain the wood specific inputs and design considerations. For general wall panel information, see the Wall Panels topic. For wood wall results interpretation, see the Wood Wall Results topic.

Wood Wall Input

The **Wall Panel Editor** gives some specific information and options for modeling/analysis of wood walls.



Wood View Controls

Toggle Wall Studs Display allows you to turn the display of the studs on and off.

Toggle Wall Chords Display allows you to turn the display of wall panel region chords on and off.

Toggle Top/Sill Plate Display allows you to turn the display of the top/sill plates on and off.

Toggle Opening Headers Display allows you to turn the display of headers on and off.

Creating Openings in Wood Walls with Headers

Within the **Wall Panel Editor**, you have the option of adding rectangular openings to wood wall panels. To draw an opening, select the **Create New Openings** button and then select two nodes or grid intersections which make up the two diagonal corners of your opening. When an opening is drawn a header beam is automatically created above the opening. To view or

edit the properties of a header beam, double-click inside the boundary of the drawn opening. This will bring up the Editing Properties window for that particular header beam.

Label - This defines the name of this header and shows up in the results output for this header.

Same as Opening - This allows you to define this header with the same properties as another header already defined in this wall panel.

Header Size - This allows you to define the size of the header member. The program will do design checks based on this size.

Header Material - This allows you to define the header material.

Trimmer Size - This allows you to define the trimmer size for this opening. This value is only used for the material take off for the wall.

Trimmer Material - This allows you to define the material for the trimmers.

General Requirements for Shear Walls

The design of wood shear walls within the framework of the NDS requires that many criteria are satisfied before a wall can be considered adequate. For RISA to work within this framework we require that certain modeling practices be followed. Outlined below are many general wall modeling practices and limitations. Also included are specific requirements for each of the three design procedures for wood wall design with openings: segmented, force transfer around openings and perforated.

The three different types of shear walls are defined in Section 4.3 of the NDS Special Design Provisions for Wind and Seismic.

Note:

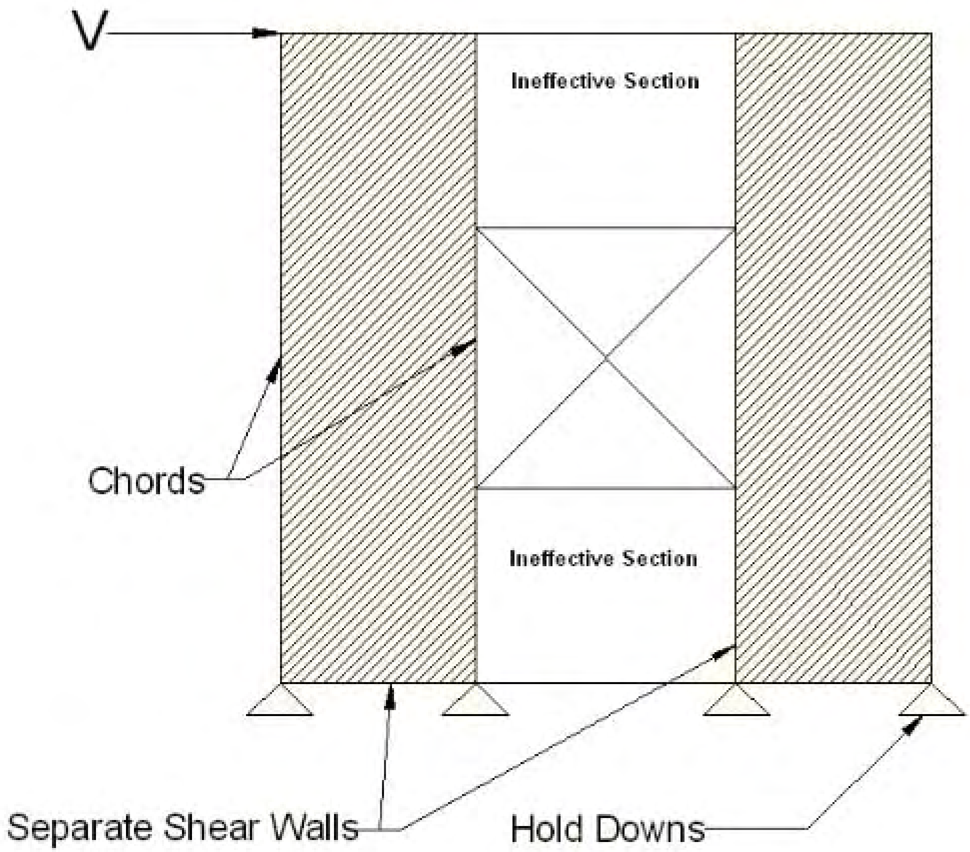
- RISAFloor only considers the Segmented design method. If using RISAFloor together with RISA-3D, simply taking your model into RISA-3D will open up the Perforated and Force Transfer Around Openings design methods.

Segmented Method

Where there is a wall panel with openings, the area above and below the openings is disregarded and the wall is designed as being made up of separate, smaller shear walls.

Like all wall panels, the segmented wood wall is broken into a series of meshed plate elements to represent the overall wall. The portions of the segmented shear wall that are considered "ineffective" in resisting shear are modeled with a plate elements that have a significantly reduced shear stiffness so that they will not receive any significant moment or shear from the FEM analysis.

See the diagram below for more information:



In addition, the out of plane stiffness and in plane stiffnesses of the segmented wood wall are modeled separately based on different assumed plate thicknesses. This is done to insure that the shear stiffness is based entirely on the properties of the sheathing and is not influenced by the out-of-plane stiffness of the wall studs.

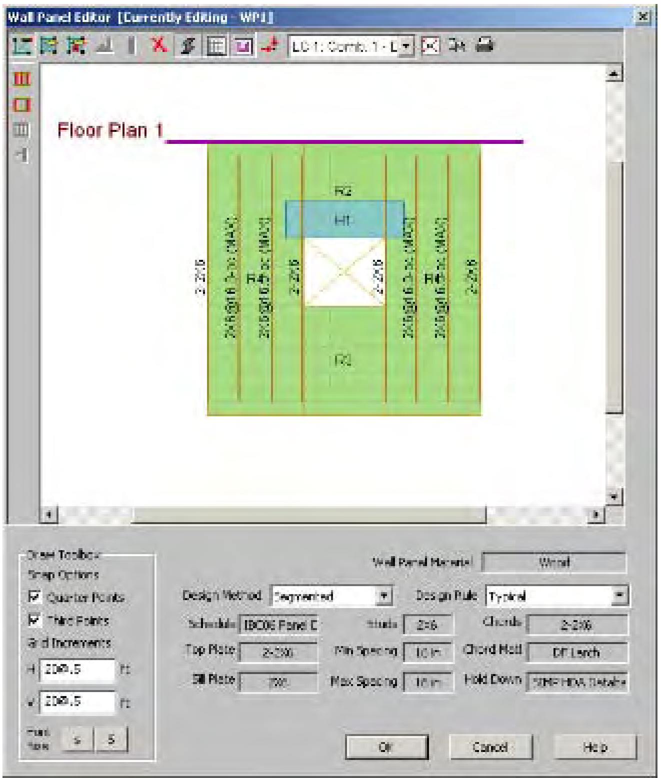
General Program Functionality and Limitations

RISAFloor and RISA-3D Interaction

When using RISAFloor and RISA-3D in combination, the interface transitions nicely between the two programs. Here is a quick walk through of this interaction.

Input Interface

1. Model the entire building (gravity and lateral members) within RISAFloor. Be sure to model all openings and regions for all of the wall panels in the model.



Note:

- You can not modify your openings or regions in RISA-3D. All region and opening modifications must be taken back to RISAFloor to be done.

2. Add loading and solve the model.
3. Take the model into RISA-3D via the Director tool.
4. Once in RISA-3D you must add your hold downs and straps to your wall panels.

Note:

- Hold downs and straps can not be added to wall panels in RISAFloor.
- Hold downs are only allowed to be added to the lower corners of the wall panel for the Perforated or Force Transfer Around Openings design methods.
- Hold downs are required at the corners of all full height regions in the wall panel for the Segmented design method.
- The Design Rule, Design Method and SSRF can be changed in either program at any time.

Results Viewing

Below are some general guidelines when reviewing results in a combined RISAFloor-RISA-3D model.

- Only wall panels designed as Segmented will be designed in RISAFloor. Perforated and Force Transfer Around Openings will give no design values in RISAFloor and must be taken to RISA-3D for design.

Stud Design

- Stud design is accomplished by taking the entire axial load in the wall panel and dividing it by the number of studs (based on spacing). This force is then checked against the allowable to give a code check. If you are optimizing for spacing, the program will start with the largest spacing and work its way down until finding a spacing that works.

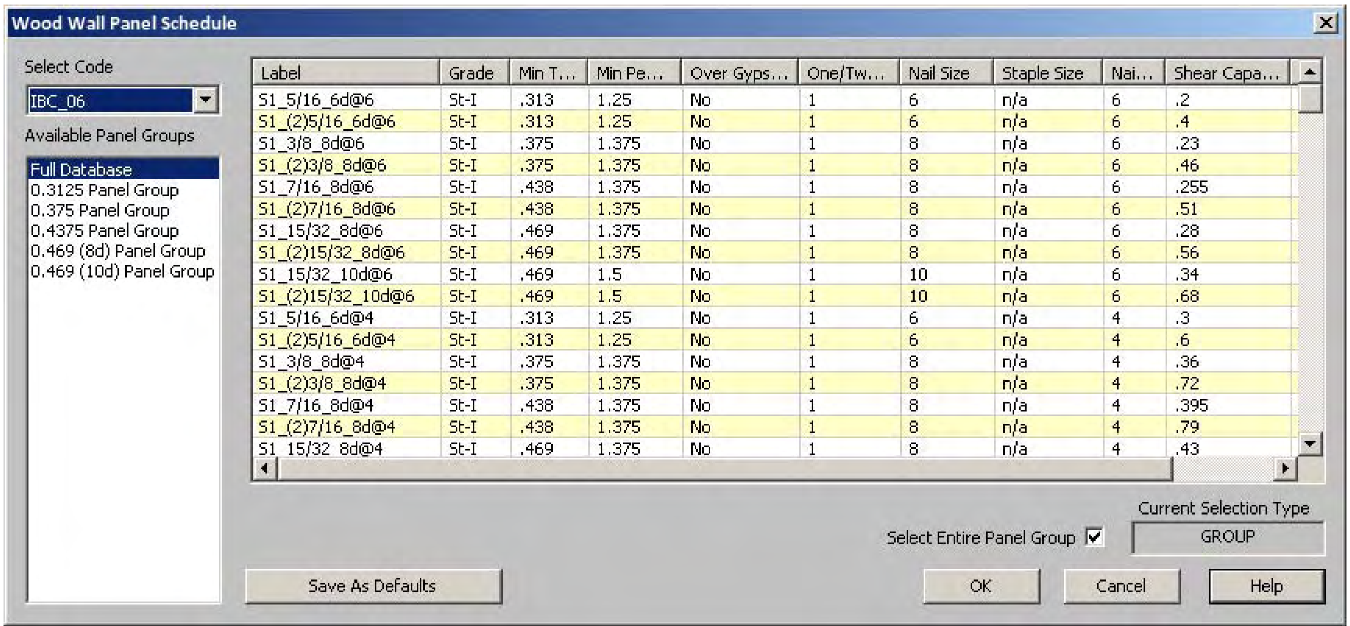
Wood Wall Self Weight

The program will calculate wood wall self weight as a sum of all the weights of the components. The material density is used to calculate the self weight of the studs, chords, top plates, sill plate, and sheathing. These are all then summed together to give the self weight of the entire wall.

Note:

- For this calculation, stud height equals wall height minus the thickness of the sill plate and the top plate.
- The number of studs is calculated using the stud spacing specified in Design Rules.

Panel Optimization



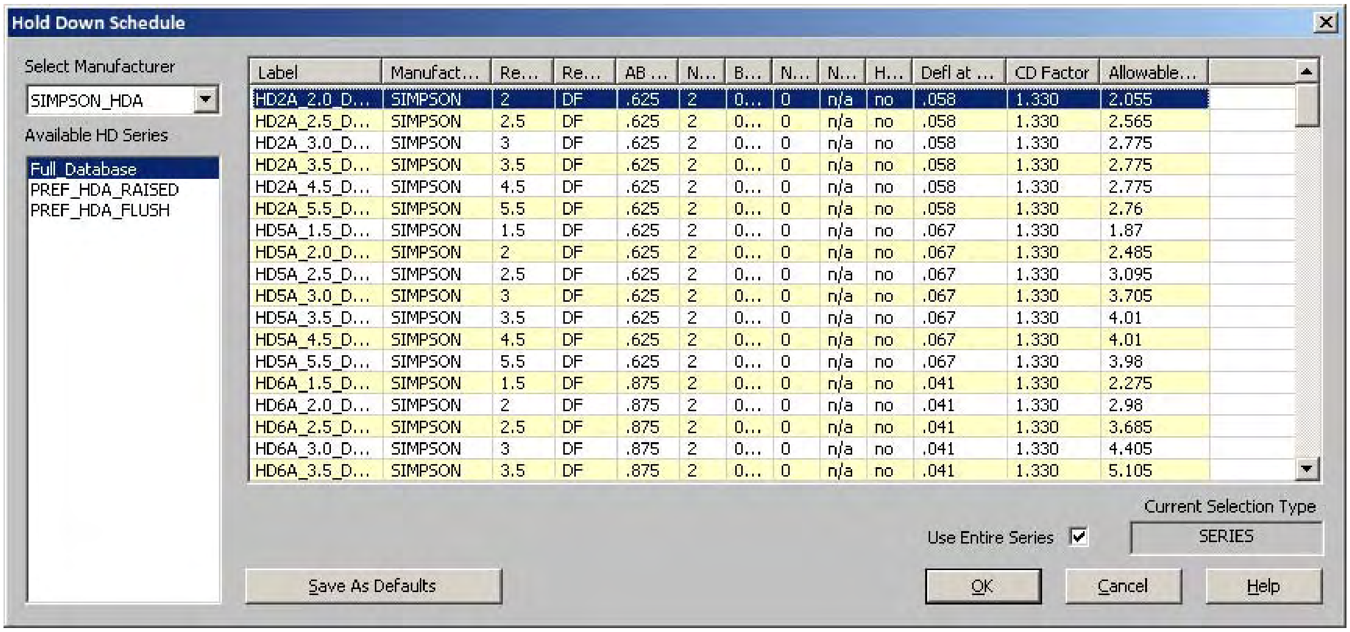
Label	Grade	Min T...	Min Pe...	Over Gyps...	One/Tw...	Nail Size	Staple Size	Nai...	Shear Capa...
S1_5/16_6d@6	St-I	.313	1.25	No	1	6	n/a	6	.2
S1_(2)5/16_6d@6	St-I	.313	1.25	No	1	6	n/a	6	.4
S1_3/8_8d@6	St-I	.375	1.375	No	1	8	n/a	6	.23
S1_(2)3/8_8d@6	St-I	.375	1.375	No	1	8	n/a	6	.46
S1_7/16_8d@6	St-I	.438	1.375	No	1	8	n/a	6	.255
S1_(2)7/16_8d@6	St-I	.438	1.375	No	1	8	n/a	6	.51
S1_15/32_8d@6	St-I	.469	1.375	No	1	8	n/a	6	.28
S1_(2)15/32_8d@6	St-I	.469	1.375	No	1	8	n/a	6	.56
S1_15/32_10d@6	St-I	.469	1.5	No	1	10	n/a	6	.34
S1_(2)15/32_10d@6	St-I	.469	1.5	No	1	10	n/a	6	.68
S1_5/16_6d@4	St-I	.313	1.25	No	1	6	n/a	4	.3
S1_(2)5/16_6d@4	St-I	.313	1.25	No	1	6	n/a	4	.6
S1_3/8_8d@4	St-I	.375	1.375	No	1	8	n/a	4	.36
S1_(2)3/8_8d@4	St-I	.375	1.375	No	1	8	n/a	4	.72
S1_7/16_8d@4	St-I	.438	1.375	No	1	8	n/a	4	.395
S1_(2)7/16_8d@4	St-I	.438	1.375	No	1	8	n/a	4	.79
S1_15/32_8d@4	St-I	.469	1.375	No	1	8	n/a	4	.43

The procedure that RISA uses for design optimization is fundamentally based upon the assumption that there is a 'cost' to shear capacity, and therefore the ideal panel design would have as little shear capacity as possible to meet code requirements. Once the program has determined the shear demand on the wall it will choose the most economical panel configuration based on that which has a **Shear Capacity** closest to, but not exceeding the shear demand.

Note:

- The shear force listed in the XML spreadsheet is currently only tabulated for the seismic values. A future update will use the “wind increase factor” in the database to increase the allowable values whenever a wind load combination is being solved.

Hold Down Optimization



The procedure that RISA uses for hold-down optimization is fundamentally based upon the assumption that there is a 'cost' to allowable tension in a product, and therefore the ideal hold-down would have as little tensile capacity as possible to meet code requirements. Once the program has determined the tensile force required to hold-down the wall it will choose the most economical hold-down product based on that which most closely matches (but does not exceed) the tension demand. The program looks to the **Allowable Tension** field of the hold-down schedule to choose the design.

Optimization Procedure

For users who are new to wood wall design within RISA, the best procedure is to utilize the full databases, and to limit the potential designs by utilizing the design rules spreadsheets. This results in a design based on the maximum number of options, which is often the most efficient design.

For experienced users who have more specific limitations in terms of the designs they would like to see, user-defined **Groups** (or families) are the solution. For example, an engineer who prefers to use only one sheathing thickness, or one nail type can create a custom Group that contains only the arrangements they want. For more information on creating these custom groups see Appendix F-Wood Shear Wall Files.

For more information on Wood Walls see Wood Wall Results.

Wall Panels - Results

When the model is solved there is a results spreadsheet for Wall Forces and Wall results. Each tab gives code checks based on the relevant code depending on the material type. These spreadsheets can be used as a summary of all of the panels in your model. To get detailed information about each panel, you can see the Wall Panel Detail Reports.

Wood Wall Spreadsheet Results

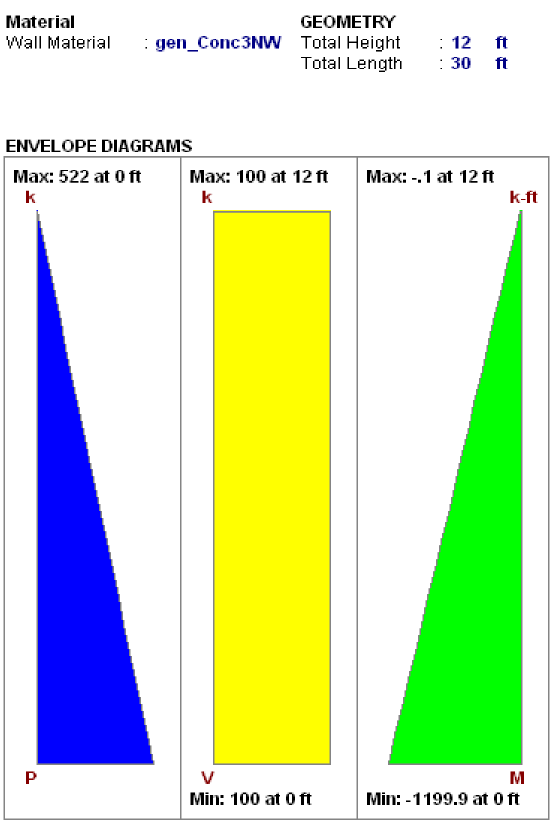
- See below

Wood Wall Detail Reports

- In-Plane or Shear Wall Detail Report

General Wall Detail Report

The general wall detail report shows the material type, height, length, and envelope forces for general walls. There is currently no concrete reinforcement design, thus general walls provide an easy way to model concrete walls for analysis.



Note

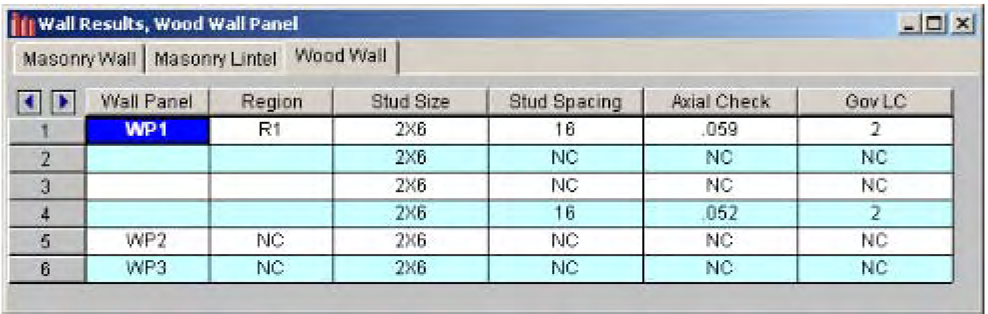
- No detail report is generated for general walls with openings when there is no region defined.

Wall Panel Results

Wall Results Spreadsheet

The **Wall Results Spreadsheet** displays the calculated results for wall elements and may be accessed by selecting **Wall Results** on the **Results** menu. The spreadsheet has three tabs: Masonry Wall, Masonry Lintel, and Wood Wall.

The **Wood Wall** tab displays the various code checks and their associated load cases as well for the studs and the chords. The results displayed on this tab are explained below.


Wall Results, Wood Wall Panel
Masonry Wall | Masonry Lintel | Wood Wall

	Wall Panel	Region	Stud Size	Stud Spacing	Axial Check	Gov LC
1	WP1	R1	2X6	16	.059	2
2			2X6	NC	NC	NC
3			2X6	NC	NC	NC
4			2X6	16	.052	2
5	WP2	NC	2X6	NC	NC	NC
6	WP3	NC	2X6	NC	NC	NC

The **Region** column reports which region the results are being shown for. See the notes below for more information.

The **Stud Size** reports the studs for the designed wall.

The **Stud Spacing** displays the spacing for the designed studs.

The **Axial Check** reports the code checks for the wood wall.

The **Gov LC** reports the governing load combinations for the code checks.

Note:

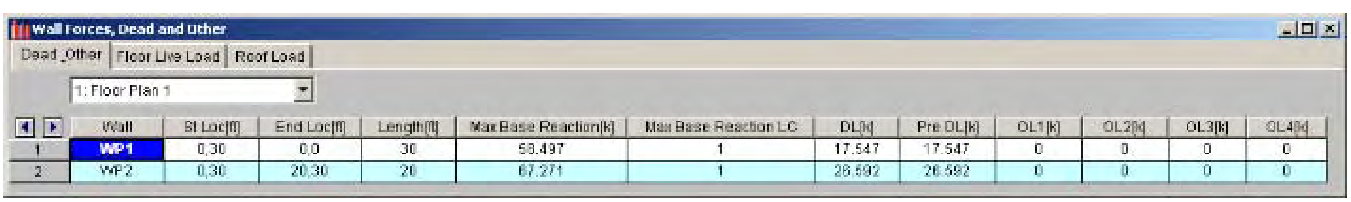
- For wood wall panels designed as Perforated or Force Transfer Around Opening there is no design and an NC (no calculation) will be shown. If you intend to transfer to RISA-3D this is not a problem, as the design will be done there.
- For wood wall panels designed as Segmented, the regions above and below openings will not be designed and an NC (no calculation) will be shown. The segmented method essentially disregards these portions of the wall in design.

Wall Forces Spreadsheet

The **Wall Forces Spreadsheet** displays the calculated results for wall panel elements and may be accessed by selecting **Wall Results** on the **Results** menu. The spreadsheet has three tabs: **Dead_Other, Floor Live Load,** and **Roof Load**.

Dead_Other Tab

The **Dead_Other** tab displays the axial forces in the wall due to dead and other load categories. The results displayed on this tab are explained below.


Wall Forces, Dead and Other
Dead_Other | Floor Live Load | Roof Load
1: Floor Plan 1

	Wall	St Loc[ft]	End Loc[ft]	Length[ft]	Max Base Reaction[k]	Max Base Reaction LC	DL[k]	Pre DL[k]	OL1[k]	OL2[k]	OL3[k]	OL4[k]
1	WP1	0,30	0,0	30	58.497	1	17.547	17.547	0	0	0	0
2	WP2	0,30	20,30	20	67.271	1	26.592	26.592	0	0	0	0

The **Wall** column displays the wall label.

The **St Loc** and **End Loc** columns display the plan coordinates for the starting and ending points of each wall.

The **Length** column displays the length of the wall.

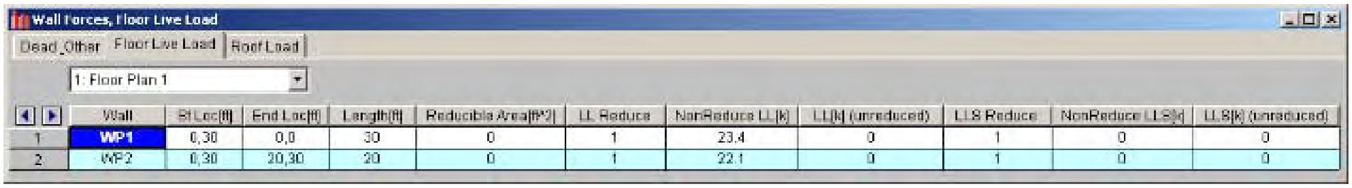
The **Max Base Reaction** and **Max Base Reaction LC** columns display the maximum base reaction and its governing load combination.

PreDL and **DL** give the *total* axial forces in the wall due to the dead load categories.

The **OL1**, **OL2**, **OL3**, and **OL4** columns give the total axial force in the wall due to the 'Other' load categories.

Floor Live Load Tab

The **Floor Live Load** tab displays the location and live load reduction information for each wall panel. The results displayed on this tab are explained below.


Wall Forces, Floor Live Load

Dead_Other | Floor Live Load | Roof Load

1: Floor Plan 1

	Wall	St Loc[ft]	End Loc[ft]	Length[ft]	Reducible Area[ft^2]	LL Reduce	NonReduce LL[k]	LL[k] (unreduced)	LLS Reduce	NonReduce LLS[k]	LLS[k] (unreduced)
1	WP1	0,30	0,0	30	0	1	23.4	0	1	0	0
2	WP2	0,30	20,30	20	0	1	22.1	0	1	0	0

The **St Loc** and **End Loc** columns display the plan coordinates for the starting and ending points of each wall.

The **Length** column displays the length of the wall.

The **Reducible Area** column lists the total tributary area to be considered for live load reduction for each wall.

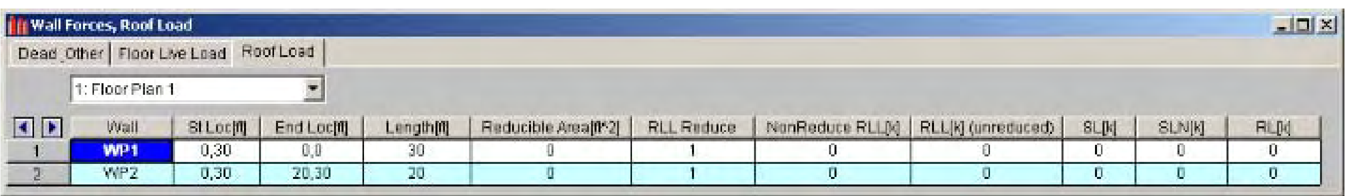
The **LL Reduce** and **LLS Reduce** columns display the live load reduction factors for the various live load categories.

The **NonReduce LL** columns give the total axial force in the wall due to the non-reducible live load cases. The included load cases are LL and LLS.

The **LL Unreduced** columns give the *total* unreduced axial forces in the wall due to the reducible live load cases. The included load cases are LL-Reduce and LLS-Reduce. It is important to note that the forces listed in this column have not been reduced by the load reduction factors.

Roof Load Tab

The **Roof Load** tab displays the location and roof live load reduction information for each wall panel. The results displayed on this tab are explained below.


Wall Forces, Roof Load

Dead_Other | Floor Live Load | Roof Load

1: Floor Plan 1

	Wall	St Loc[ft]	End Loc[ft]	Length[ft]	Reducible Area[ft^2]	RLL Reduce	NonReduce RLL[k]	RLL[k] (unreduced)	SL[k]	SLN[k]	RL[k]
1	WP1	0,30	0,0	30	0	1	0	0	0	0	0
2	WP2	0,30	20,30	20	0	1	0	0	0	0	0

The **St Loc** and **End Loc** columns display the plan coordinates for the starting and ending points of each wall.

The **Length** column displays the length of the wall.

The **Reducible Area** column lists the total tributary area to be considered for live load reduction for each wall.

The **RLL Reduce** column displays the live load reduction factor.

The **NonReduce RLL** column gives the total axial force in the wall due to the non-reducible roof live load case.

The **RLL Unreduced** column gives the *total* unreduced axial forces in the wall due to the reducible roof live load cases. It is important to note that the forces listed in this column have not been reduced by the load reduction factors

The **SL, SLN,** and **RL** columns display the total axial force in the wall due to these load cases.

Wood Wall Results

Wood Wall results are presented in the Wall Panel Design Spreadsheet and the detail reports.

Wood Wall Results Spreadsheets

See Wall Panels - Results for this topic.

Wood Wall Self Weight

The program will calculate the self weight of a wood wall based on the weights of the individual components. Using the material density, the self weight is calculated for the studs, chords, top plates, sill plates, and sheathing. These are all then summed for the total self weight of the wall.

Wood Wall Detail Reports

The detail report gives detailed information about the wall design. The detail reports are specifically molded to the type of design specified. Here we will walk through how to access the different information for each of the types of design: Segmented, Perforated and Force Transfer Around Openings.

Note:

- RISAFloor only considers the Segmented design method. If using RISAFloor together with RISA-3D, simply taking your model into RISA-3D will open up the Perforated and Force Transfer Around Openings design methods.
- Many of the values for design checks seen below are not performed in RISAFloor as it is strictly a gravity design program.

Accessing the Detail Reports and the Specific Windows

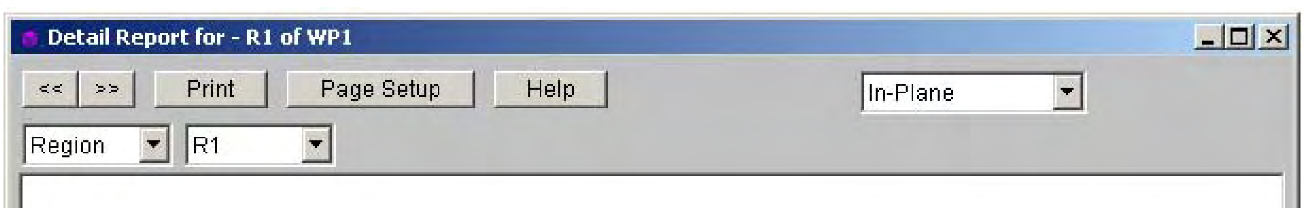
Once you have a solved model, the detail reports become available. They are accessible in two ways:

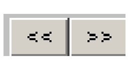
1. If you have the Wall Results spreadsheet open, there will be a button at the top of the screen: Detail Report for Current Item. This will open up the detail report window.
2. If you are in a graphic view of your model, there is a Detail button that will open up the detail report window.

Note:

- Detail report information is not available for an envelope solution.

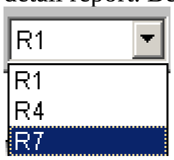
Once the detail report window is open, you will see a dialog area at the top.



- 

 - Allows you to click between the different wall panels in your model.

- 

 - This dropdown list allows you to select between the three different parts of the wood wall panel detail report. Below we will explain the importance of each of these sections.
- 

 - This dropdown list allows you to select between different regions or headers defined within the individual wall panel.

The importance of the Opening, Region and Wall detail report sections depends on the type of design you are doing: Segmented, Perforated and Force Transfer Around Openings.

Segmented Method

The segmented design method uses each of the three detail report windows to give design information.

Opening Window

This window defines the design of the header beams across the top of the openings in the wall. The different openings can be chosen from the header drop down list. This report is very similar to a wood member detail report. At the top of the detail report the Criteria, Geometry and Materials section give the input parameters defined for the opening. The Envelope Diagrams and the Design Details are given, which provide code check information and give the information required to verify the program values.

Region

The window gives information for your wall on a region basis. Note that only full-height regions of the wall panel will have a region detail report. The segmented method only considers these full height segments in the design of the wall.

The region detail report is split into four portions: input echo, diagrams and design, design details, and cross section detailing. Note that in RISAFloor the detail reports are less detailed because RISAFloor does not consider lateral forces which RISA-3D does.

Input Echo

Below is the input echo portion of the detail report.

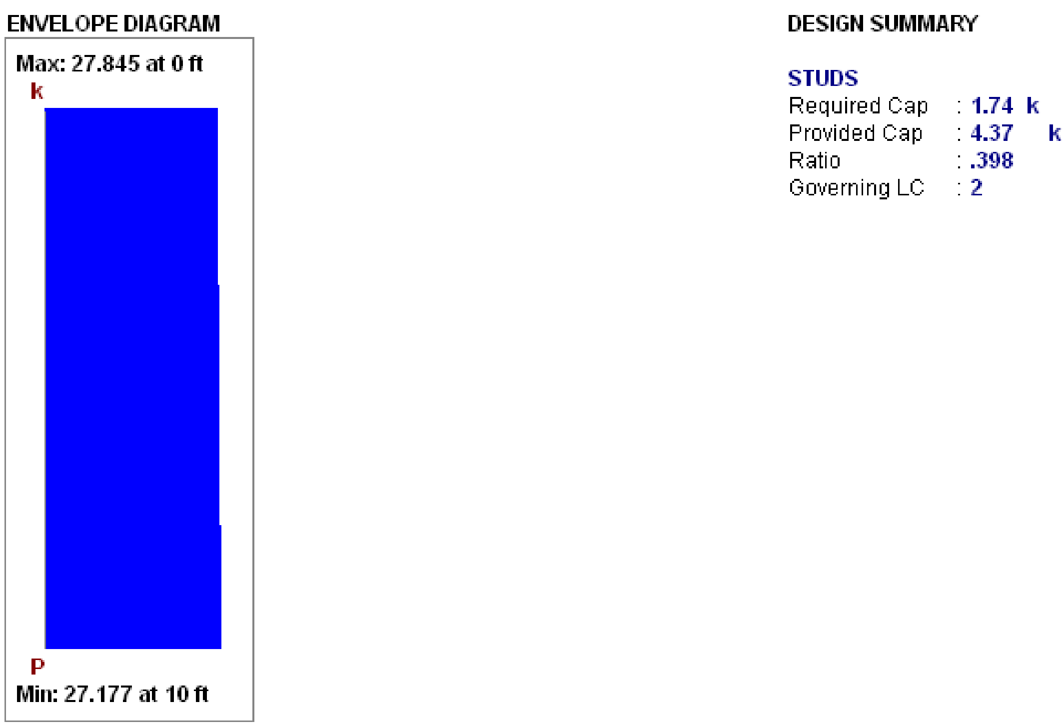
CRITERIA		**MATERIALS**		**GEOMETRY**	
Code	: **NDS 2005**	Wall Studs	: **Spruce Pine Fir**	Total Height	: **10 ft**
Type of Design	: **ASD**	Stud Size	: **2X6**	Total Length	: **20.5 ft**
Wall Material	: **Spruce Pine Fir**	Chord Material	: **Spruce Pine Fir**	Wall H/W Ratio	: **0.49**
		Chord Size	: **2-2X6**		
		Top Pl & Sill	: **Spruce Pine Fir**	Stud Spacing	: **16 in**
		Top Pl Size	: **2-2X6**		
		Sill Pl Size	: **2X6**		

Criteria	Description
Code	Gives the code used to design your wall panel.
Type of Design	Specifies which design method was used (ASD, Strength, etc)
Wall Material	Specifies the wood type assigned to the entire wall

Materials	Description
Wall Studs	Specifies the wood material type assigned to the wall studs
Stud Size	Specifies the member size used for the wall studs
Chord Material	Specifies the wood material type assigned to the chords (vertical members at both ends of the wall)
Chord Size	Specifies the member size used for the chords (vertical members at both ends of the wall)
Top Plate & Sill	Specifies the wood material type assigned to the top and sill plates
Top Plate Size	Specifies the member size used for the top plate
Sill Plate Size	Specifies the member size used for the sill plate

Geometry	Description
Total Height	This is the height of the wall panel region
Total Length	This is the length of the wall panel region
Wall H/W Ratio	This is the ratio of wall height to length, using the minimum wall height
Stud Spacing	This is the optimized stud spacing based on your Design Rules

Diagrams and Design



Envelope Diagram

This diagram shows the axial force in the wall.

Design Summary

This portion gives you the capacity and strength values at the section in the wall where the combined check is maximum, as well as the governing load combination.

Chords, Studs

The provided capacities of these members are calculated using the standard provisions for tension/compression members. These members are assumed to be fully braced in the weak axis, and unbraced in the strong axis.

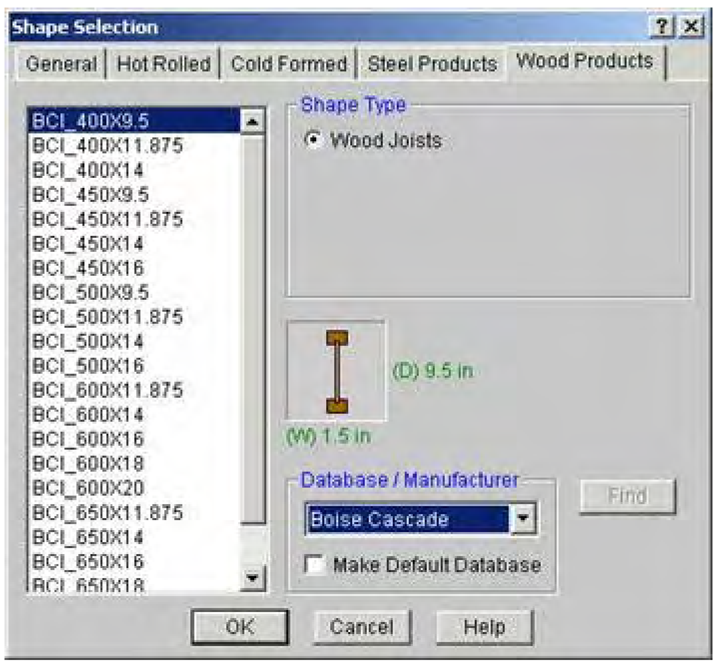
Wood Products

Full beam wood design can be performed on Wood Joist Products based on the NDS Guidelines for Engineered Wood Construction. The allowable moment and shear values are obtained from the following ICBO product reports:

Manufacturer	ICBO Report
Trus Joist	PFC - 4354
Boise Cascade	PFC - 4665
Louisiana - Pacific	PFC - 3754

Wood Products Database

The various Wood Product databases may be accessed from the **Beams** spreadsheet by clicking in the **Shape** field and then clicking .



The Wood Product database is based on data contained in the product's ICBO evaluation report. You may choose the Manufacturer and product from the drop down lists.

When you return to the **Beams** spreadsheet you will notice that a special label has been used to represent the choices that you made in the database. You may also type in this designation directly.

Wood Products Adjustment Factors

The NDS code applies adjustment factors to the allowable moments and shears of a wood joist. Currently RISAFloor only accounts for the Load Duration Factor CD. The other adjustment factors (CL, Cm, Cr, Ct, and CH) are all applied as 1.0.

Wood Design Temperature Factor

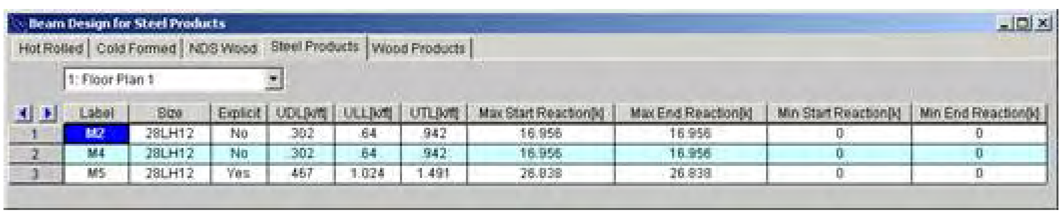
The temperature factor (Ct) is calculated internally based on the information entered in the Global Parameters dialog. See Section 2.3.4 of the NDS for more information on the temperature factor.

Load Duration Factor

The CD factor is entered on the **Load Combination** spreadsheet for each load combination that you want to use for wood code checks. If this field is left blank, it will be automatically calculated by the program. Different load combinations will have different CD factors. The CD factor will only be applied to wood code checks on wood members. See the Table below for the CD factors that are assumed for the various load categories. Note that the CD factor used for a load combination will be for the load category with the shortest load duration in that load combination.

Load Category	CD
DLPre	0.9
LLConst	1.25
DLConst	1.25
DL	0.9
LL	1.0
LLS	1.0
RLL	1.0
SL	1.15
SLN	1.15
RL	1.15
OL1, OL2, OL3, OL4	1.0

Design Results - Wood Products



The results for the **Steel and Wood Products** tab are explained below. The steel product results give the uniform loads applied to the joists. The results for the wood products give the maximum bending moments and shears as well as the allowable moments and shears for the joist. The pull down list at the top of the spreadsheet allows you to toggle between floors.

The **Label** column lists the beam label.

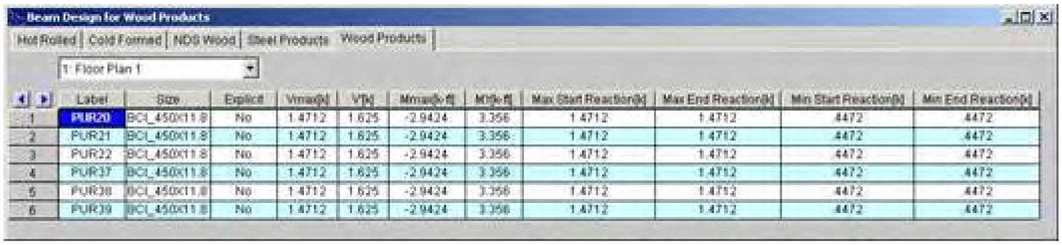
The **Size** column displays the designation of the steel or wood product. When no adequate member could be found from the redesign list, this field will display the text “not designed”. Consider reframing, relaxing the design or deflection requirements (see Design Optimization), or adding more shapes to the available Redesign List (see Appendix A – Redesign Lists of the General Reference Manual).

The **Explicit** column displays “Yes” if the beam has been locked to an explicit beam size by the user. When you have chosen a specific shape to override the programs automatic redesign, that beam becomes “locked” and will not be automatically redesigned by the program.

Note

- To "unlock" a beam, you can use the beam – modify tool to assign a shape group. If the model has already been solved, you may optimize a beam by using the Member Redesign dialog. See Member Redesign for more details.

Shear and Moment Check for Wood Products



	Label	Size	Explicit	Vmax[k]	V'[k]	Mmax[k-ft]	M'[k-ft]	Max Start Reaction[k]	Max End Reaction[k]	Min Start Reaction[k]	Min End Reaction[k]
1	PUR20	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472
2	PUR21	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472
3	PUR22	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472
4	PUR37	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472
5	PUR38	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472
6	PUR39	BCI_450X11.8	No	1.4712	1.625	-2.9424	3.356	1.4712	1.4712	.4472	.4472

The **V max** column for Wood Products displays the maximum applied shear that resulted from the load combinations that were selected for Wood Product design.

The **V`** column for Wood Products displays the allowable shear for the wood product under consideration. The program obtains this value from the product's ICBO Evaluation Report.

The **M max** column for Wood Products displays the maximum applied moment that resulted from the load combinations that were selected for Wood Product design.

The **M`** column for Wood Products displays the allowable moment for the wood product under consideration. The program obtains this value from the product's ICBO Evaluation Report.

End Reactions

The **Max Start & End Reactions** column displays the maximum start and end reactions of the beam for ALL load combinations. If "Show Factored End Reactions" in **Global Parameters** is left unchecked, these displayed loads are not factored. If it is checked, then the displayed loads will have been multiplied by the factors in the load combinations. The sign convention assigns positive reactions to downward forces. Negative reactions, if they occur, would indicate uplift.

The **Min Start & End Reactions** column displays the minimum start and end reaction of the beam.

Wood Product Limitations

Shear deflection is only calculated for wood products that are pinned at both ends and have a uniform distributed load. Shear deflection may have significant effects for wood products, especially on beams with shorter spans.

Appendix F – Wood Database Files

RISAFloor has design databases for wood shear walls and diaphragms which are used to optimize nailing, hold downs, and panels. The criteria used for this optimization is specified on the Wood Wall (Fasteners) and Wood Diaphragms tabs of the **Design Rules** spreadsheet. In addition to this basic criteria, the user may specify a subset of the overall database from which the design / optimization must be performed.

Hold Downs

Each database of hold downs is specified by an XML file in the "Hold Downs" sub-directory of the Wood Wall panels directory. The location of this directory is based on the information in the File Locations tab of the Tools - Preferences dialog.

The program comes pre-loaded with two XML files each of which contain a database of commonly used hold downs: the Simpson HDA/HD hold downs, and the Simpson LTT/MTT/HTT hold downs. The name of the XML file itself will be used in the list of databases in the Hold Down Schedule Dialog.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **SIMP HTT Database**). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, design and code check information used for each hold down. These entries are described below.

Full Database - Required Fields

The following fields are *required* information. If they are not provided or are left blank, then that hold down will not be available for use in that database.

The **Label** field is used to identify the hold down. This field must be referenced on the sheets that identify families or groups of hold downs.

The **Deflection at Peak Load** entry is used to calculate the deflection of the shear wall per APA / NDS formulas. This deflection is then reported on the shear wall detail report for each wall panel.

The **CD Factor** is the assumed load duration factor that was used as the basis for specifying the listed **Allowable Tension** value for that hold down.

A load combination may be solved with a load duration factor different from the CD Factor described above. When this is the case, the **Allowable Tension** for that hold down will be adjusted based on the difference between the assumed and actual load duration factors.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design or capacity calculations, but are reported on the detail reports for reference purposes only.

The **Manufacturer** field is an identifier for the hold down. It is provided so that the engineer can more easily identify the callouts for their final design drawings.

The allowable capacity of the hold down will vary based on the Chord Thickness. Therefore, the **Required Chord Thickness** gives the minimum chord thickness that will yield the listed allowable tension load. However, this field is NOT currently used in the calculations. A future revision may provide a warning message if the actual chord thickness provided is less than required.

The allowable capacity of the hold down will vary based on the density of the wood species being used. Therefore, the **Required Chord Density** lists the density assumed for the entered allowable tension. However, this field is NOT currently used in the calculations. A future revision may provide a warning message if the actual chord density provided is less than required.

The **AB Diameter** is not currently used in the design calculations and is reported for display purposes only.

The **Bolt Size** when specified is used to reduce the axial capacity of the hold down chord itself. The only change to the calculation is that the program will perform the allowable tension check on the net area of the chord member rather than the gross area. The **Nail Size**, on the other hand, is NOT assumed to affect the tension capacity of the hold down chord.

The **Number of Bolts** and **Number of Nails** are not used in the design calculation and are reported for reference purposes only.

The **Is Hold Down Raised?** field is not used in the design calculations and is reported for display purposed only. It is assumed that if the hold down deformation is significantly affected by the connection being flushed or raised, then the **Deflection at Peak Load** entry described in the previous section will adjusted instead.

Grouping Hold Down Schedules for Design Optimization

The other sheets allow the user to group hold downs together into families for optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The hold down labels specified on these sheets refer only to hold downs that have already been defined on the full database sheet. The information in this sheet need not be organized in a specific order. Instead, they will always be optimized based on the assumption that the hold down cost is directly related to the tension capacity. Therefore, when this group is selected, then the hold down within the group with the code check closest to unity, but still less than 1.0 will get selected during the optimization process.

Panel Nailing Schedules

Each database of wall panels is specified by an XML file in the "Shear Panels" sub-directory of the Wood Wall panels directory. This directory is located based on the information File Locations tab of the Tools - Preferences dialog.

The program comes loaded with two XML files one for the tabularized nailing schedules listed in the 1997 UBC and one for the ones listed in the 2006 IBC.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **IBC06 Panel Database**). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, nailing, design and code check information for each nailing schedule. These entries are described below.

Full Database - Required Fields

The following fields are *required* information. If they are not provided or are left blank, then that nailing schedule will not be available for use in the database.

The **Label** field is a used to identify the panel schedule and its nailing requirements. This field must be referenced on the sheets that identify families or groups of panels.

The **Min Panel Thickness** is used during the design optimization to limit the selected panels based on the Design Rules chosen by the user. It is also used to help set the elastic stiffness of the wall panel used during the FEM solution.

The **Ga** value is the **Apparent Shear Stiffness** from nail slip and panel deformation as defined in equation 4.3-1 of the NDS' Special Design Provisions for Wind and Seismic. This value (in combination with the Min Panel Thickness defined above) is used to set the elastic stiffness of the wall panel that will be used during the FEM solution.

Note:

- When a family or group of panels / nailing schedules are assigned to a shear wall, the lowest value of Ga and Min Panel thickness will be used to determine the elastic stiffness of the plate elements in the FEM solution.

The **One/Two Sided** field is used during the design optimization to limit the available panels based on the Design Rules specified by the user.

The **Boundary Nail Spacing** field is used during the design optimization to limit the available panels based on the Design Rules specified by the user.

Note:

- The maximum field spacing is never entered in the program but is generally equal to 12 inches for the nailing schedules defined in the 1997 UBC and 2006 IBC databases. If a different nail spacing is present, then the user should add in a new nailing schedule to the existing database with a user defined shear capacity.

The **Shear Capacity** listed in the spreadsheet is the primary value that controls the code checking of the shear wall.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design or capacity calculations, but are reported on the detail reports for informational purposes only.

The **Panel Grade** and **Min Penetration** fields are identifiers for the engineer, but are not used in the design calculations. They are provided so that the engineer can most easily identify the panels in their design results and drawings.

The **Panel Applied Over Gypsum** field is also a identifier for the engineer that will not be used in the design calculations.

The **Nail Size** listed in the spreadsheet is intended to refer to the Common nail size, but is reported only for reference purposes and are NOT used in the capacity calculations. If the nail size is changed by the user, then the user should also change the Shear Capacity entry accordingly. Below is a reference table for common, box, and sinker nails.

	Diameter				Length	
Penny Weight	**Common (in)**	**Common (Wired Gage)**	**Box (in)**	**Sinker (in)**	**Box and Common (in)**	**Sinker (in)**
6d	0.113	11.5	0.099	0.092	2	1.875
7d	0.113	11.5	0.099	0.099	2.25	2.125
8d	0.131	10	0.113	0.113	2.5	2.375
10d	0.148	9	0.128	0.120	3	2.875
12d	0.148	9	0.128	0.135	3.25	3.125
16d	0.162	8	0.135	0.148	3.5	3.25

The Staple size listed in the database is reported for reference purposes only. If the staple size is entered or changed by the user, then the user should also change the shear capacity entry to the appropriate value.

The **Wind ASIF** field is not currently used within the program at all. It is reserved for a future feature.

Grouping Panel / Nailing Schedules for Design Optimization

The other sheets in the database allow the user to organize multiple nailing schedules into groups or families for design optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The panel labels specified on these sheets refer only to panel / nailing schedules that have already been defined on the full database sheet. The information in this sheet need not be organized in a specific order. Instead, they will always be optimized based on the assumption that the installed cost is directly related to the shear capacity. Therefore, when a group or family is selected, then the nailing schedule within the group with the code check closest to unity, but still less than 1.0 will get selected during the optimization process.

Diaphragm Nailing Schedules

Each database of diaphragms is specified by an XML file in the "Diaphragms" sub-directory of the Wood Wall schedules directory. This directory is located based on the information on the File Locations tab of the Tools - Preferences dialog.

The program comes loaded with two XML files, both based on the shear values of the 2006 IBC and the stiffness values of the 2005 NDS Special Design Provisions for Wind and Seismic. The WSP (Wood Structural Panel) is intended to be a

generic database that could be used used for plywood or OSB panels. However, it uses the Ga (apparent stiffness) values for plywood because they are genearally lower resulting in a more conservative deflection.

Note:

- Diaprhagm design is currently only available for flexible diaphragms that were created in RISAFloor and brought into RISA-3D.

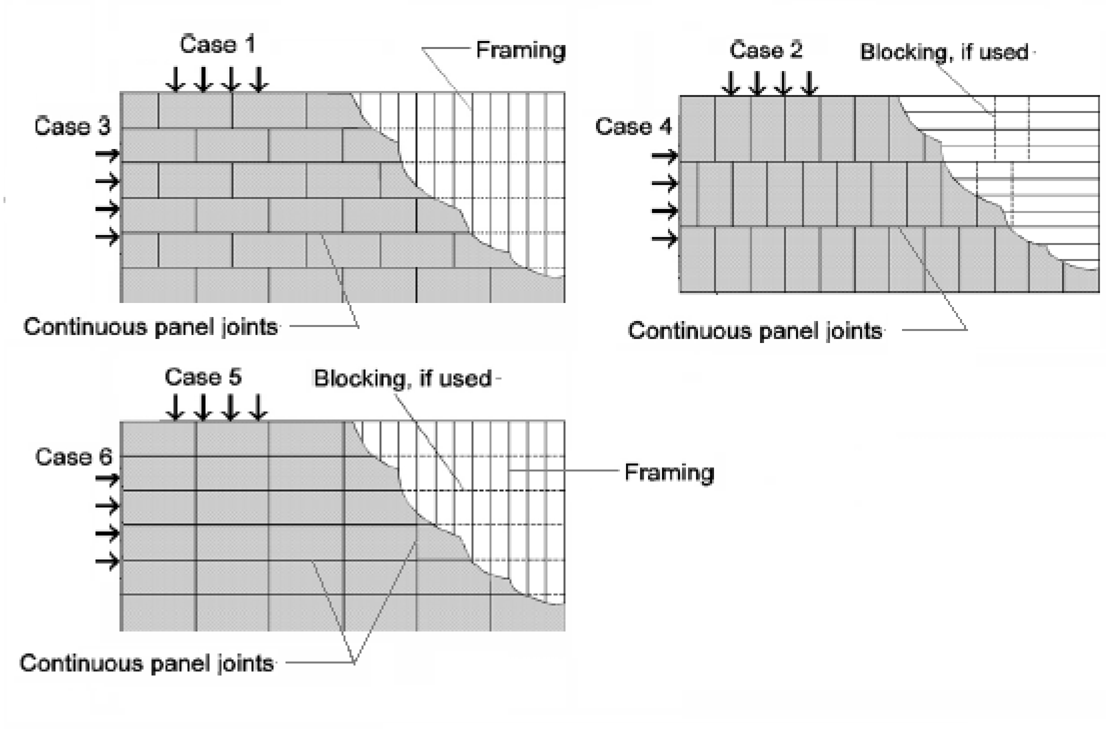
The first sheet of the XML file should always be descriptive of the contents of the database (such as IBC06 OSB Database). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, nailing, design and code check information for each nailing schedule. These entries are described below:

Full Database - Required Fields

The following fields are required information. If they are not provided or are left blank, then that nailing schedule will not be available for use in the database.

The **Label** field is a used to identify the diaphragm nailing. This field must be referenced on the sheets that identify families or groups of panels.

The **Case** field is used to specify the layout of the shear panels as shown below. Any diaphragm that has a Case 1 layout also has a Case 3 layout, and the same goes for 2/4 and 5/6.



The program considers the deck span defined in RISAFloor to coincide with the long direction of the plywood. Therefore, Cases 2, 3 and 6 are considered parallel to the to the RISAFloor deck span. Whereas, Cases 1, 2 and 5 would be considered perpendicular to the same deck span.

The **Blocked** field specifies whether blocking is used to achieve the associated design strength. It also determines the method by which the diaphragm deflection will be calculated. For more information see Diaphragm Deflection

The **Panel Grade** field specifies what grade of structural panel is used in the diaphragm. This may be set to either "Structural-I" or "Other" and is used as a criteria in the Design Rules.

The **Panel Thickness** field specifies the thickness of the structural panel used for the diaphragm. This is a decimal value that is rounded to four places for reporting convenience. For example, a 15/32” panel is listed as 0.4688

The **Boundary/Cont Edge Spacing** field specifies the nail spacing at the boundary and along any continuous edges. These must be specified as the same value.

The **Other Edge Spacing** field specifies the nail spacing at non-continuous edges.

The **Nail Lines** field specifies the number of lines of nails along each panel edge. This value is greater than (1) for High Load diaphragms. This value is not currently used in design optimization, but is reported on the output for reference purposes only.

The **Strong Shear Capacity** field specifies the shear strength of the diaphragm (lbs/ft) based on its stronger case. For example, while Case 1/3 represents the same panel layout, Case 1 has greater strength than Case 3 (based on load direction).

The **Weak Shear Capacity** field specifies the shear strength of the diaphragm (lbs/ft) based on its weaker case. There are many situations where strong and weak capacities are identical. In these cases the same value must be specified for both fields.

The **Strong Ga** field specifies the apparent shear stiffness (kips/in) of the diaphragm as specified in the NDS document *Special Design Provisions for Wind and Seismic.* Since this is the strong direction it will be based on the stronger direction / case for loading. For example, while Case 1/3 represents the same panel layout, Case 1 has greater stiffness than Case 3 (based on load direction). For more information see Diaphragm Deflection

The **Weak Ga** field specifies the apparent shear stiffness (kips/in) of the diaphragm as specified in the NDS document Special Design Provisions for Wind and Seismic.

The **Gt** field specifies the shear stiffness of panel depth. It is always the same for both strong weak directions, hence it does not need to be specified twice. For more information see Diaphragm Deflection

The **Strong Nail Slip (en)** field specifies the nail slip used for deflection calculations based on the stronger case. For example, while Case 1/3 represents the same panel layout, Case 1 may have less nail slip than Case 3 (based on load direction). For more information see Diaphragm Deflection

The **Weak Nail Slip (en)** field specifies the nail slip used for deflection calculations based on the weaker case. There are many situations where strong and weak nail slips are identical. In these cases the same value must be specified for both fields. For more information see Diaphragm Deflection

Note:

- The Gt and Nail Slip fields are ignored for unblocked diaphragms.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design, capacity or deflection calculations, but are reported on the detail reports for informational purposes only.

The **Framing Width** field identifies the minimum required framing width for the nailing layout. A higher shear capacity can typically be achieved for a diaphragm by using wider supporting framing, thereby reducing the tension perpendicular to the grain of supporting members.

The **Minimum Penetration** field identifies the minimum required nail penetration specified in the IBC/NDS tables.

The **Nail Size** listed in the spreadsheet is intended to refer to the Common nail size, but is reported only for reference purposes and are NOT used in the capacity calculations. If the nail size is changed by the user, then the user should also change the Shear Capacity entry accordingly. The section on shear walls contains a good reference table for common, box and sinker nails.

The **Wind ASIF** field is not currently used within the program at all. It is reserved for a future feature.

Grouping Panel / Nailing Schedules for Design Optimization

The other sheets in the database allow the user to organize multiple nailing schedules into groups or families for design optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The labels specified on these sheets refer only to nailing schedules that have already been defined on the first (full) database sheet. The program is not capable of understanding the complexities of diaphragm construction cost (whether it is cheaper to increase panel thickness versus decreasing nailing spacing for example). Instead the program simply “walks down” the listed labels, looking for the first one that meets the required strength. You may re-order the labels on the Group tabs to meet the most economical arrangement for you.

Help Options

RISA Technologies has, and will, put a great deal of effort into assisting you in getting your work done as quickly as possible. This includes providing many ways that you can get help in understanding the software.

Electronic Help File

The **Help File** is designed to help you get your work done as quickly as possible and is intended to provide:

- Procedures that lead users through the steps of completing tasks
- Troubleshooting topics that guide users through solutions to common problems
- Extensive discussions for a thorough understanding of engineering and modeling topics
- Easy access to related topics

The electronic help file can be accessed by clicking the **Help File** button on the **RISA Toolbar**. A new window containing a Table of Contents will be opened. Click on any item in the Table of Contents for extensive information on the topic.

Context Sensitive Help

Context Sensitive Help is help that a user can access in context while working in a program. It provides you with the information you need where and when you want it.

You may get detailed help when working in some windows by clicking on the **Help** button at the bottom of the window or dialog. This will launch a Help File window displaying the topic that is related to the window in which you are working. The topic will be explained and links to related topics may also be provided.

RISA Technologies Online

Our website, **www.risatech.com**, provides various support information and documents.

Visit RISA Technologies on the web for:

- Frequently Asked Questions
- Download program Manuals (General Reference or Tutorial)
- The latest software updates - When a bug is discovered it is posted on the web site along with possible work-around procedures and/or service releases to update your software.
- Software Verification Problems

Tool-tips

Are you uncertain what a toolbar button is for? Simply hold your mouse pointer over that button without clicking. **Tool-tips** are displayed that will explain what the button will do should you decide to press it.

Tutorial

The comprehensive **Tutorial** (part of the User's Guide - a separate document) guides you through using most features. It is a real-world example of building and solving a model, making changes, and optimizing. This is the best way to quickly get up and running. The User's Guide is designed to be read in two ways. If you are already familiar with structural modeling in general you can skip the supporting text and read only the underlined action items to quickly move through the tutorial. If you want more thorough explanations of the modeling process you may read all or some of the supporting text as you see fit.

Technical Support

Technical support is an important part of the RISAFloor package. There is no charge for technical support for all licensed owners of the **current** version of RISAFloor. Technical support is very important to the staff at RISA Technologies. We want our users to be able to reach us when they are having difficulties with the program. However, this service is not to be used as a way to avoid learning the program or learning how to perform structural modeling in general.

Hours: 6AM to 5PM Pacific Standard Time, Monday through Friday

Before contacting technical support, you should typically do the following:

1. **Please search the Help File or General Reference Manual**. Most questions asked about RISAFloor are already answered in the Help File or General Reference Manual. Use the table of contents or index to find specific topics and appropriate sections. We go to great lengths to provide extensive written and on-line documentation for the program. We do this in order to help you understand the features and make them easier to use. Also be sure to go through the entire User's Guide when you first get the program.
2. If you have access to the Internet, you can visit our website at **www.risatech.com** and check out our **Support** section for release notes, updates, downloads, and frequently asked questions. We list known issues and product updates that you can download. So, if you think the program is in error you should see if the problem is listed and make sure you have the latest release. The **FAQ (Frequently Asked Questions)** section may also address your question.
3. Make sure you understand the problem, and make sure your question is related to the program or structural modeling. Technical Support does not include free engineering consulting. RISA Technologies does provide a consulting service. If you are interested in inquiring about this service, please call RISA Technologies.
4. Take a few minutes to experiment with the problem to try to understand and solve it.

For all modeling support questions, please be prepared to send us your model input file via email or postal mail. We often will need to have your model in hand to debug a problem or answer your questions.

Email: support@risatech.com: This method is the **best** way to send us a model you would like help with. Most email packages support the attachment of files. The input file you would send will have a .RFL extension. Make sure you tell us your name, company name, serial number or Key ID, phone number, and give a decent problem description. If you have multiple members, plates, or load combinations, make sure you specify which ones to look at.

Phone Support:(949) 951-5815: Feel free to call, especially if you need a quick answer and your question is not model specific and therefore doesn't require us to look at your file.

Postal Mail to RISA Technologies: This method works fine as long as you can wait for the postal service. If you don't have email then this is the only way to send us the model that you need help with. Most people who are in a rush will send a floppy disk via overnight mail (email is a lot cheaper and faster, though).

RISA Technologies

Tech Support

26632 Towne Centre Drive

Suite 210

Foothill Ranch, CA 92610

Index

R

S

T

U

V

W

RISA-3D Wood Design - Reference

26632 Towne Centre Drive, Suite 210
Foothill Ranch, California 92610

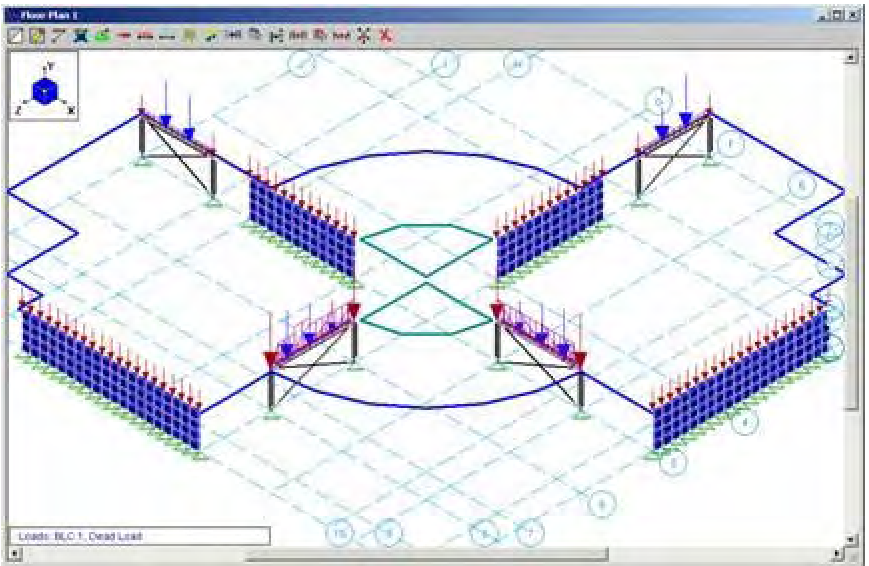
(949) 951-5815
(949) 951-5848 (FAX)

www.risatech.com

Table of Contents

RISAFloor and RISA-3D Integration

While the primary function of RISAFloor is to create and optimize floor systems, another strength is that it can be used to automatically generate a model of the lateral force resisting system in RISA-3D.



Note

- The features described in this section of the manual are only available to users who are running both RISAFloor and RISA-3D.

Lateral System Model Generation

Beams, columns, and walls whose function is set to 'Lateral' on the Primary Data Tab of their respective spreadsheets will automatically be generated in the RISA-3D model when accessed via the Director Menu. To access this model click the Director Menu on the far right end of the Main Menu and choose 'RISA-3D'. The RISA Application Interface will then switch from RISAFloor to that of RISA-3D.

Once in RISA-3D you will notice that you can use the RISA-3D features to edit and solve the model. You can add braces, beams, columns, walls, and additional loads just as you would in a regular RISA-3D model. Refer to the RISA-3D General Reference Manual and User’s Guide for documentation of RISA-3D's features.

The "gravity" model in RISAFloor and the "lateral" model in RISA-3D are fully linked. Subsequently, any changes made to RISAFloor generated members in the RISA-3D model will automatically update those same members in RISAFloor model.

Note

- Beams, columns, and walls whose function is set to 'Gravity' in RISAFloor will NOT be generated in RISA-3D for optimization. These members have been indicated as "gravity-only" members and should not collect any lateral load in the RISA-3D model. Therefore, these members would only "clutter" the lateral system model in RISA-3D and are subsequently left out.
- RISAFloor 'Gravity' members may be viewed in RISA-3D via the Misc Tab of the Plot Options Dialog. These members will be displayed for visual effect only in the model view but will not contribute to the stiffness of the RISA-3D model.

Diaphragms

Diaphragms are created in RISA-3D for every floor slab in RISAFloor. Although you cannot delete these diaphragms, you can make them **inactive** in RISA-3D. The **Mass**, **Mass Moment of Inertia** and **Center of Mass** are automatically calculated based on the RISAFloor loads and on the settings in Global Parameters.

Floor Diaphragms

	Elevati...	Mass[k	Mass ...	Center of Mas...	Plus X Ecc...	Minus X E...	Plus Z Ecc...	Minus Z E...	Inactive	Diaphragm	Type	Region	Design Rule
1	12	62.399	4199.72	10, 10	%5	%5	%5	%5	☐	D2	Flexible	DR1	Lower Level
2									☐			DR2	Lower Level
3									☐			DR3	Roof Level
4									☐			DR4	Typical
5	12	247.759	116858	89.997, 34.453	%5	%5	%5	%5	☐	D3	Flexible	DR1	Typical
6									☐			DR2	Lower Level
7									☐			DR3	Roof Level
8	12	46.582	3964.757	45, 15	%5	%5	%5	%5	☐	D4	Rigid		

The **X and Z eccentricities** are used in the equivalent lateral force method for calculation of seismic loads. This allows you to quickly and easily account for the effects of accidental torsion when calculating your seismic response.

Note:

- Eccentricity does not apply to flexible diaphragms

The **Diaphragm** and **Region** columns are the names of diaphragms and regions in the model.

Note:

- Diaphragms defined as rigid are not designed, thus the Region column is blank.

The **Type** allows you to toggle between flexible and rigid.

Note:

- The program does not have a semi-rigid option at this time. To consider a semi-rigid diaphragm, modeling the diaphragm as a plate model would be the route to go. Here is more information on Plates.

The **Design Rule** allows you to switch between design rules for the diaphragm regions. The Design Rules spreadsheet is where many parameters are defined for your diaphragm. For more information on diaphragm modeling and interaction, see the Diaphragms topic.

Gravity Loads

The gravity loads on the lateral members, including the beam reactions from the gravity only members, become part of the RISA-3D model. The Load Categories in RISAFloor are automatically converted into Basic Load Cases in RISA-3D. The exceptions to this are the Load Categories that deal specifically with construction loads (DL Const, and LL Const), which are not converted.

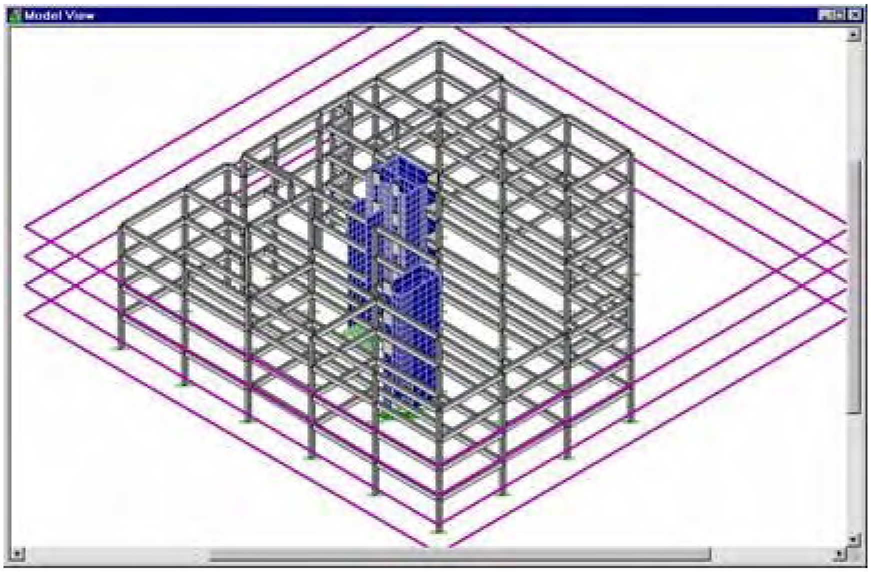
Wind Loads

Wind loads can be automatically generated for the 1995 – 2005 ASCE 7, the IS 875:87 (Indian) code, the 2005 Canadian code, and the Mexican code.

Seismic Loads

Seismic loads can be automatically generated according to the equivalent static methods of the 1997 UBC, 2000 IBC, 2001 CBC (i.e. California Amended UBC), the ASCE 7-2002 and 2005 (which is referenced directly by the 2003 and 2006 IBCs respectively), the 2002 Indian code (IS 1893), the 2005 Candian code, and the 2004 NTC (Mexican) code.

Diaphragms - General



Diaphragms can be defined in both RISAFloor and RISA-3D. The RISAFloor diaphragms are used in RISA-3D when using the Director tool. Here we will first have a general diaphragm discussion. Then we will talk about diaphragms defined directly in 3D. Then we will get into the RISAFloor-RISA-3D diaphragm interaction. Finally, we will discuss some other concepts/ideas to keep in mind.

RISA-3D has two types of rigid diaphragms and a flexible diaphragm. The two types of rigid diaphragms are a rigid membrane and a rigid planar diaphragm. The flexible diaphragm options are described in more detail in the Diaphragms - Flexible topic.

The two different types of diaphragms are provided to handle different modeling situations. The planar diaphragm option is rigid in all 6 degrees of freedom. It has a very large stiffness in plane and out of plane. The membrane diaphragm option is only rigid in the plane of the diaphragm. It has NO stiffness out of plane.

Note

- You should not apply a boundary condition to a joint that is part of a Rigid Diaphragm. See Defining Diaphragms.

Rigid Diaphragm Applications

The rigid diaphragm feature can be used to aid the engineer in quickly modeling the transfer of lateral forces into resisting elements such as shear walls, braced frames, and columns. These loads can be of a static or dynamic nature. Thus, lateral loads can be applied where they really occur on a structure, and the effects of the center of applied force being different from the center of rigidity will be accounted for automatically in the solution. Dynamic mass can also be applied where it actually occurs on the structure, and the differences between the center of mass and the center of stiffness will be accounted for automatically as part of the dynamic solution.

RISA-3D Diaphragm Interface

Defining Rigid Diaphragms

A rigid diaphragm is defined by a master joint and a global plane to be made rigid. Note that only planes parallel to the global axes (XY, YZ, or ZX) may be made rigid. For example, say you enter joint 10 as the master joint and specify ZX as the plane. This means a plane passing through joint 10 and parallel to the global ZX axis will be established. The actual

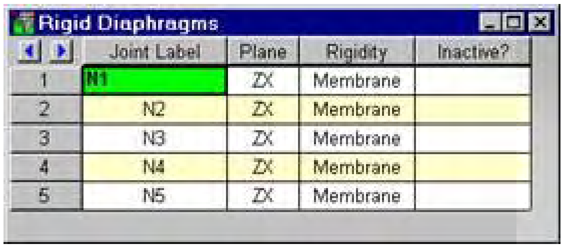
location of the plane relative to the Y-axis will be the Y coordinate of joint 10. Any joints having the same Y coordinate as joint 10 will be on the plane and will be rigidly linked together. Note that the tolerance for other joints to be on the same plane is 0.01 ft. You may create partial diaphragms by detaching joints from the diaphragm. See Partial Diaphragms to learn more about this.

You should not apply a boundary condition to a joint that is part of a Rigid Diaphragm. This will cause numerical problems in the solution and will probably give erroneous results. The stiffness method requires that there be some meaningful deflection in the model to obtain results. Rigid Diaphragms and boundary conditions are both rigid and should not be attached to one another. If these elements are attached together, nothing is left to deflect properly, and anydeformation is likely to be the result of numerical round off.

At this time RISA-3D does not display the center of rigidity for a diaphragm. Although it does display the center of gravity for all applied vertical loads along with the reaction results. In addition, when the load have been applied using the seismic load generator, the program will display the center of mass for the diaphagm. Most design codes require an assumed accidental torsion that is in addition to the natural torsion created by the location of mass with respect to rigidity. While RISA-3D calculates the natural torsion automatically, the accidental torsion is only calculated automatically if you have used the seismic load generator to create your seismic loads. See Modeling Accidental Torsion for more information.

Diaphragms Spreadsheet

The **Rigid Diaphragms** Spreadsheet records the rigid diaphragms for the model and may be accessed by selecting **Diaphragms** on the **Spreadsheets** menu.



Rigid Diaphragms

	Joint Label	Plane	Rigidity	Inactive?
1	N1	ZX	Membrane	
2	N2	ZX	Membrane	
3	N3	ZX	Membrane	
4	N4	ZX	Membrane	
5	N5	ZX	Membrane	

The input for diaphragms is straightforward. Enter the joint to define the location of the diaphragm and then specify a global plane (XZ, XY or YZ) that is to be parallel to the diaphragm.

The “Rigidity” field is used to specify the type of rigid diaphragm. The two options are a fully rigid Plane (enter “P”) or a diaphragm that is rigid only for Membrane action (enter “M”).

The **Inactive** field can be used to turn off the rigid diaphragm without actually deleting the entry. Enter “Y” to inactivate the diaphragm.

To quickly verify that a diaphragm has been assigned to the right joint and in the right plane, view the plot of the structure with the diaphragms activated in the **Plot Options**.

Rigid Membrane Diaphragms

The rigid membrane diaphragm can be used in situations where the engineer wants to model a rigid floor slab that will carry the in-plane loads, but still allow frame action in the beam and columns (i.e., the beam and column joints in the diaphragm are free to rotate out-of-plane). The lateral stiffness for such a model is simply the combined beam and column frame stiffness, with very little contribution from the diaphragm. (There will be a slight contribution to the lateral stiffness from the diaphragm, since the beams cannot deform axially within the plane of the diaphragm).

The membrane diaphragm is also useful if you want to combine out-of-plane loads (typically vertical dead or live loads) with lateral loads (typically seismic or wind). One limitation of the membrane option is that beams or plate elements are needed to provide any out-of-plane stiffness or “frame” type lateral stiffness. For example, a one-story structure composed of 4 columns, fixed at their bases, and a membrane type diaphragm at their tops, will have the lateral stiffness of 4 cantilevers, since there are no beams to prevent tip rotation of the columns.

The behavior of the rigid membrane diaphragm in RISA-3D is defined by observing three “rules”. The first rule for the membrane is that after a model has been solved, all the joints on the diaphragm will have equal in-plane rotations. The out-

of-plane rotations will be based on the stiffness of the attached members and will most likely not be equal. The second rule is that the absolute distances between all joints on the diaphragm must be kept the same at all times. We say absolute distances because the horizontal projected in-plane distances can change due to the different out-of-plane rotations. Following the above two rules will cause a third rule to be satisfied as well. The in-plane translation of any joint on the diaphragm is the sum of the absolute in-plane translation of the diaphragm itself, plus the in-plane diaphragm rotation times the projected in-plane distance between the joint and the diaphragm center of stiffness.

Rigid Planar Diaphragms

The rigid plane diaphragm can be used to model situations such as shear buildings where the floor system is much stiffer than the columns, and in-plane frame action is not desired or unimportant. This option has the advantage of not needing any floor beams or plate elements to provide out-of-plane stiffness. In fact, any existing floor beams or elements will get absorbed into the stiffness of the rigid plane. A rigid plane diaphragm can be defined solely with just joints to connect it to the lateral force resisting elements. A limitation of the plane diaphragm is that lateral loads cannot be directly combined with vertical loads if moment frame action is desired to occur within the diaphragm (i.e., since the diaphragm is rigid in all 6 degrees of freedom, the connection joints between columns and beams cannot rotate). If it is desired to model these effects, you must use the membrane option.

The behavior of the rigid plane diaphragm in RISA-3D is defined by observing three rules. The first rule is that after a model has been solved, all joints on the diaphragm will have equal rotations. The second rule is that the distances between all joints on the diaphragm must be kept the same at all times. Following the above two rules will cause a third rule to be satisfied as well. The translation of any joint on the diaphragm is the sum of the absolute translation of the diaphragm, plus the diaphragm rotation times the distance between the joint and the diaphragm center of stiffness.

RISAFloor and RISA-3D Diaphragm Interaction

Input Interface

The layout (modeling) of the diaphragms is done using the RISAFloor interface, while the actual analysis/design is done within RISA-3D. See the RISAFloor Wood Design Reference for more information

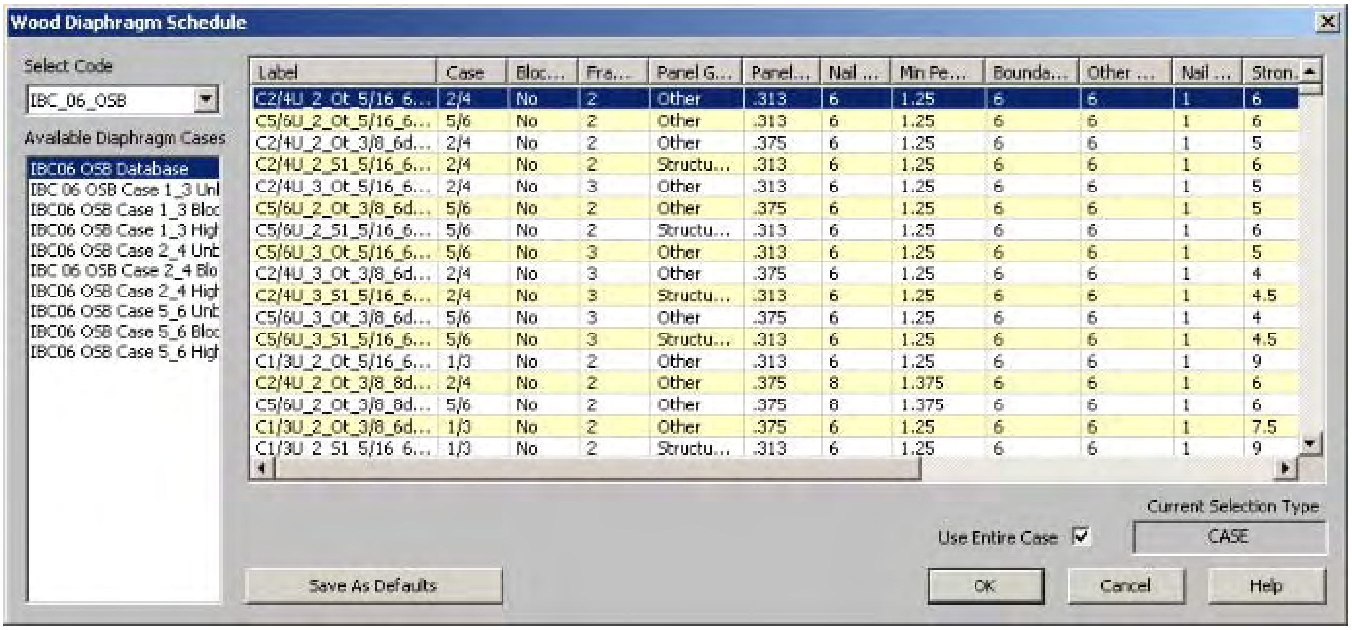
Results Viewing

Below are some general guidelines when reviewing results in a combined RISAFloor-RISA-3D model.

- The diaphragm results can only be viewed in RISA-3D under Results>>Diaphragms. See the Diaphragms - Analysis and Results topic for more information.

General Diaphragm Functionality and Discussion

Diaphragm Optimization



Label	Case	Bloc...	Fra...	Panel G...	Panel...	Nail ...	Min Pe...	Bounda...	Other ...	Nail ...	Stron...
C2/4U_2_Ot_5/16_6...	2/4	No	2	Other	.313	6	1.25	6	6	1	6
C5/6U_2_Ot_5/16_6...	5/6	No	2	Other	.313	6	1.25	6	6	1	6
C2/4U_2_Ot_3/8_6d...	2/4	No	2	Other	.375	6	1.25	6	6	1	5
C2/4U_2_S1_5/16_6...	2/4	No	2	Structu...	.313	6	1.25	6	6	1	6
C2/4U_3_Ot_5/16_6...	2/4	No	3	Other	.313	6	1.25	6	6	1	5
C5/6U_2_Ot_3/8_6d...	5/6	No	2	Other	.375	6	1.25	6	6	1	5
C5/6U_2_S1_5/16_6...	5/6	No	2	Structu...	.313	6	1.25	6	6	1	6
C5/6U_3_Ot_5/16_6...	5/6	No	3	Other	.313	6	1.25	6	6	1	5
C2/4U_3_Ot_3/8_6d...	2/4	No	3	Other	.375	6	1.25	6	6	1	4
C2/4U_3_S1_5/16_6...	2/4	No	3	Structu...	.313	6	1.25	6	6	1	4.5
C5/6U_3_Ot_3/8_6d...	5/6	No	3	Other	.375	6	1.25	6	6	1	4
C5/6U_3_S1_5/16_6...	5/6	No	3	Structu...	.313	6	1.25	6	6	1	4.5
C1/3U_2_Ot_5/16_6...	1/3	No	2	Other	.313	6	1.25	6	6	1	9
C2/4U_2_Ot_3/8_8d...	2/4	No	2	Other	.375	8	1.375	6	6	1	6
C5/6U_2_Ot_3/8_8d...	5/6	No	2	Other	.375	8	1.375	6	6	1	6
C1/3U_2_Ot_3/8_6d...	1/3	No	2	Other	.375	6	1.25	6	6	1	7.5
C1/3U 2 S1 5/16 6...	1/3	No	2	Structu...	.313	6	1.25	6	6	1	9

The procedure that RISA uses for diaphragm optimization is fundamentally based upon the assumption that there is a 'cost' to allowable shear in a product, and therefore the ideal diaphragm would have as little shear capacity as possible to meet code requirements. Once the program has determined the shear force required it will choose the most economical diaphragm thickness and nailing based on that which most closely matches (but does not exceed) the shear demand. The program looks to the **Shear Capacity** field of the diaphragm schedule to choose the design.

Optimization Procedure

For users who are new to diaphragm design within RISA, the best procedure is to utilize the full databases, and to limit the potential designs by utilizing the design rules spreadsheets. This results in a design based on the maximum number of options, which is often the most efficient design.

For experienced users who have more specific limitations in terms of the designs they would like to see, user-defined **Groups** (or families) are the solution. For example, an engineer who prefers to use only one panel layout, or force blocking, can create a custom Group that contains only the arrangements they want. For more information on creating these custom groups see Appendix F-Wood Database Files.

Inactive Diaphragms

Making a diaphragm inactive allows you to analyze the structure without the diaphragm, without having to delete the information that defines it. Putting a "y" in the **Inactive** field makes the diaphragm inactive, i.e. the diaphragm is not included when the model is solved or plotted. This leaves data intact so the diaphragm may be easily reactivated. This is handy if you want to solve a model with and then without certain diaphragms, without having to actually delete the data.

Partial Diaphragms

There may be times when you want to model a partial diaphragm, i.e., a diaphragm that extends over only a portion of a floor or plane. For example, let's say you are trying to model a floor that is composed of a relatively rigid section (thick concrete slab) and a relatively flexible section (corrugated steel decking). You would like a way to model a rigid diaphragm for only the rigid portion of the floor.

To accomplish this you may specified that a joint or group of joints be detached from the diaphragm. This may be accomplished by selecting the **Detach from Diaphragm** option in the **Joints** spreadsheet or double-click a joint and specify it in the **Information** dialog.

Another way this can be done is to offset the elevations of the joints that comprise the rigid floor section so that they are all a little higher or a little lower than the surrounding floor. (The offset only needs to be slightly larger than 0.01ft. since this is the tolerance for other joints to be on the same plane as the master joint) This works because the rigid diaphragm feature will only rigidly connect joints that are at the same elevation as the master joint. The other joints, which are on the flexible portion of the floor and are now at a different elevation than the master joint, will not be incorporated into the diaphragm.

How Rigid Diaphragms Work

Internally both types of rigid diaphragms are implemented by connecting a series of rigid and weightless members between all the joints on the diaphragm plane. There is no slaving involved so no degrees of freedom are lost.

Rigid Diaphragms vs. Slaving

It is not accurate to use joint slaving to try to create a rigid diaphragm. While the joint rotations can be slaved and the correct diaphragm behavior maintained, slaving the in-plane translational degrees of freedom will produce incorrect diaphragm behavior. A diaphragm that has a load applied to a location other than the center of stiffness should experience both a translation and a rotation. A diaphragm that is created by slaving in-plane translations will not rotate under such a loading, it will only translate, and be much stiffer than it should be.

Rigid Diaphragms vs. Plates

The diaphragms in RISA-3D are extremely rigid and will not allow for relative displacement of the joints in that plane. While this is a common modeling assumption, it may not be appropriate for all circumstances. When you want to model diaphragm flexibility or when you are most interested in the force and deflection results of the concrete slab itself, then you would have to model the slab as a mesh of plate elements. Refer to the diaphragms discussed in the Plate Modeling Examples.

Semi-Flexible / Semi-Rigid Diaphragms

Using plate elements to create a concrete slab is an accurate way to model your floor slab. The relative rigidity of the floor slab to the shear walls and lateral frames will determine if you get *rigid*, *semi-rigid*, or *flexible* diaphragm behavior.

Plywood and metal deck, on the other hand, are not as well represented by plate elements. The problem is that the flexibility of a plywood or metal deck diaphragm is greatly influenced by *fastener slip*, which cannot accurately be modeled in a finite element program such as RISA. Since these diaphragms can usually be categorized as *flexible* diaphragms, it is recommended that (if the model originated in RISAFloor) the user classify the diaphram as flexible. Refer to the

Rigid Diaphragm Stiffness

You may alter the stiffness of the diaphragm, though this value should almost never be changed. Arbitrarily changing the diaphragm stiffness without understanding the ramifications on the stiffness solution can produce solution results that are inaccurate. Having said all that, the stiffness of the diaphragm may be adjusted from the **Diaphragm** spreadsheet by clicking the button on the **Window** Toolbar. The default value is 1.E+7. The exponent of the internal stiffness of the diaphragm is twice this displayed value (1.E+14 in the default case) This value should only be adjusted for 3 reasons.

The first reason is that the lateral force resisting elements in your model are so stiff that they are causing the rigid diaphragm to behave semi-rigidly (i.e. the rotations are not all the same for all joints on the diaphragm.). In this case you could try increasing the diaphragm stiffness to 1.E+8, however the internal diaphragm stiffness of 1.E+16 starts to approach the boundary condition reaction stiffness, which is internally modeled as 1.E+20. Stiffness value greater than 1.E+8 can produce unpredictable results that are characterized by ghost reactions, meaning some of the joints in the diaphragm begin to behave as Reaction boundary conditions.

The second reason that you might adjust the diaphragm stiffness is because your dynamics solution will not converge. In this case, you will want to reduce the diaphragm stiffness to 1.E+6 or even lower if necessary. As you lower the stiffness, you will need to watch the joint rotations for joints on the diaphragm to insure that you getting, or at least approximating, rigid diaphragm action. The joint rotations should be the same for all joints in a rigid diaphragm.

The third reason that you might adjust the diaphragm stiffness is because your diaphragm is causing "ghost reactions". This means that load is actually being taken out of your structure at the diaphragm. This might happen if you have large amount of nodes in the diaphragm at an isolated location. With the diaphragm rigid linking all the nodes at that level together, a large amount of rigid links in close proximity to one another can sometimes simulate a reaction. You may drop the diaphragm stiffness to 1.E+6, but be careful that all of the joint rotations at that level stay fairly similar.

Diaphragms - Analysis and Results

RISA-3D currently only handles flexible diaphragms that come in from an integrated Floor / 3D model. This is because RISAFloor contains information about deck edges, deck direction and geometry that RISA-3D would not normally know about.

Analysis / Loading

Seismic Load

The program only applies seismic loads to flexible diaphragms that were generated in RISAFloor using the seismic load generator. The load is internally converted into a one way member area load where the direction of the load attribution is perpendicular to the direction of the applied load. this internal conversion is done at solution time. This area load then gets broken down into a series of "transient" distributed loads that are applied to the members which support the diaphragm. These transient load can be viewed with the basic load cases. Refer the section on Member Area Loads for more information about how these transient loads are generated.

Wind Loads

The program only applies seismic loads to flexible diaphragms that were generated in RISAFloor using the wind load generator. The load is internally converted into a distributed load at the front face of the diaphragm. Then it is broken into area loads of varying magnitudes depending on the depth of the diaphragm. In this way, an L shaped building will have more load applied at the deep section of the L than the skinny section.

These area load then gets broken down into a series of "transient" distributed loads that are applied to the members which support the diaphragm. These transient load can be viewed with the basic load cases. Refer the section on Member Area Loads for more information about how these transient loads are generated.

Diaphragm Results - Detail Reports

Diaphragm results can be accessed by clicking “Diaphragms” within the Results menu. This opens the Diaphragm Detail Report dialog, from which a diaphragm can be chosen to review its results.

The detail report gives detailed information about the diaphragm design. It is split into four main sections: input echo, diagrams and design, design details, and chord forces.

Input Echo

Below is the input echo portion of the detail report:

CRITERIA		GEOMETRY		
Code	: NDS 2005 ASD	Total Length	: 48	ft
Design Rule	: Typical	Total Width	: 24	ft
Panel Grade	: Other	L/W Ratio	: 2	
Panel Schedule	: IBC06 OSB Case 1_3 Blocked	Elevation	: 10	ft

Code – Reports the code used to design the diaphragm.

Design Rule – Reports the design rule assigned to the diaphragm region

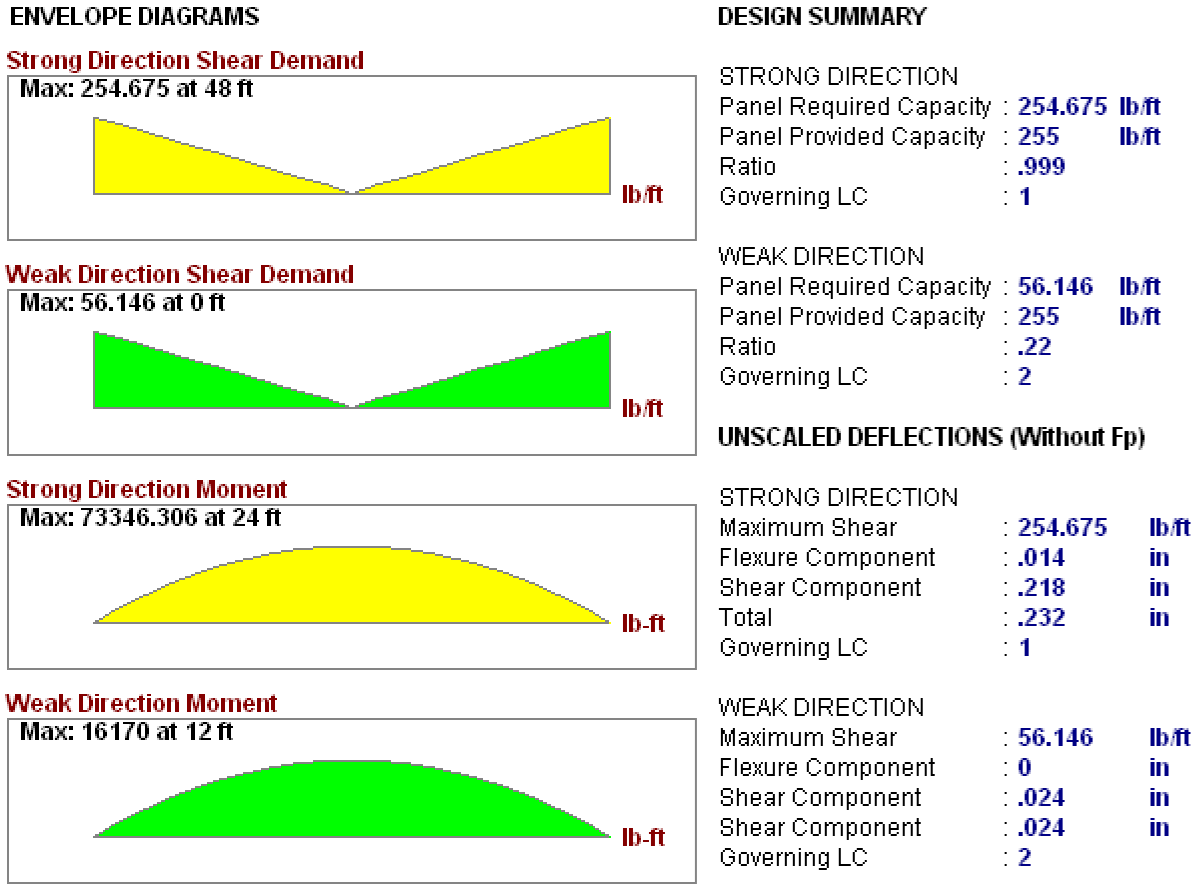
Panel Grade – Reports the grade of the panels used in the design (Structural-I or Other)

Panel Schedule – Reports the nailing schedule database used to optimize panel selection

In addition the basic geometry information for the diaphragm (**Total Length**, **Total Width**, **Elevation**, and **L/W Ratio**) are also reported here.

Diagrams and Design

The detail report presents envelope diagrams for the shear and moment demand seen on the flexible diaphragm for both the strong and weak directions. The diaphragms are color coded yellow for strong direction and green for weak direction for quick reference. The locations of maximum shears/moments are also reported.



Design Summary

The results for design summary are displayed for both strong and weak directions as shown below.

Panel Required Capacity is the maximum shear in the diaphragm region and should also correspond to the shear presented in the envelop diagrams.

Panel Provided Capacity is the capacity of the designed diaphragm. For diapragms with multiple nailng zones, this will be reported as capacity at the point of maximum shear. For more information on how the program selects between the various nailing schedules provided in the database, please refer to Appendix F.

The **Ratio** provides the code check for the diaphragm based on shear demand versus shear capacity.

Governing LC gives the load combination that controls the design. This is based on whichever load combination resulted in the highest raio for shear capacity.

Unscaled Deflections

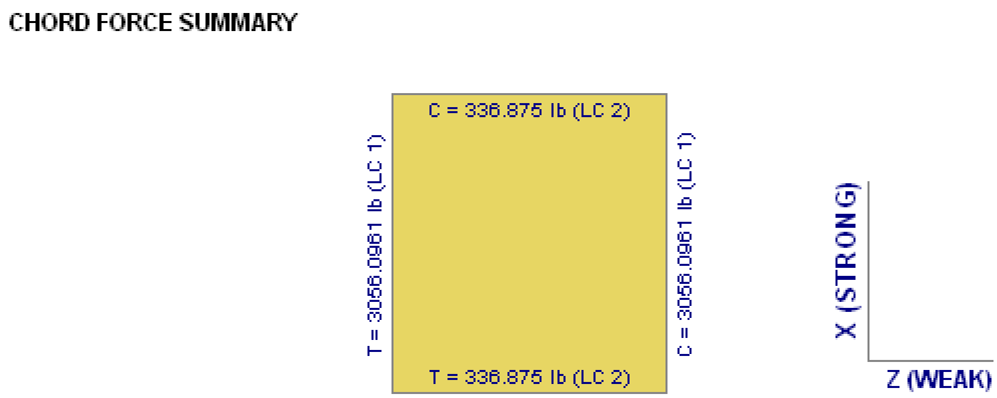
The **Deflection** of a diaphragm is comprised of two main terms, depending on the equation used. The terms are as follows:

The **Flexure Component** accounts for the deflection of the diaphragm based on bending stresses. Refer to Diaphragm Deflections for more information.

The **Shear Component** accounts for the deflection of the diaphragm based on shear stresses. Refer to Diaphragm Deflections for more information.

These values are reported as unscaled, meaning that they do not consider the Fp values assigned for seismic loads per the ASCE-7. The maximum shears reported correspond to the actual shear, not the shear scaled by the Fp value.

Chord Force Summary



A diagram showing the chords as well as their maximum and minimum respective tension and compression forces is shown here. A global axis legend on the right indicates the strong and weak directions of the diaphragm relative to the model's global axes. This is to assist you in understanding the diaphragm's orientation.

Design Details

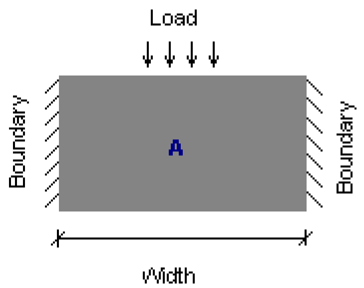
The details of the designed panel are reported here:

DESIGN DETAIL

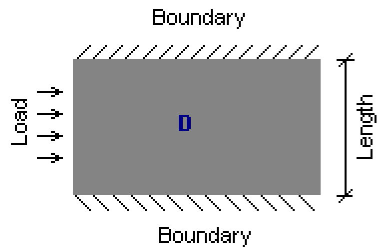
Panel Label	: C1/3B_2_0t_7/16_8d@6/6/1
Layout Case	: 1/3
Blocked	: Yes
Panel Thickness	: .438 in
Required Nail Penetration	: 1.375 in
Nail Size	: 8 d

NAIL SPACING SCHEDULE							
Zone	Location (ft)	Label	Lines	Framing Width (in)	Boundary (in)	Cont Edge (in)	Other Edge (in)
A	0	C1/3B_2_0t_7/16_8d@6/6/1	1	2	6	6	6
D	0	C1/3B_2_0t_7/16_8d@6/6/1	1	2	6	6	6

LAYOUTS



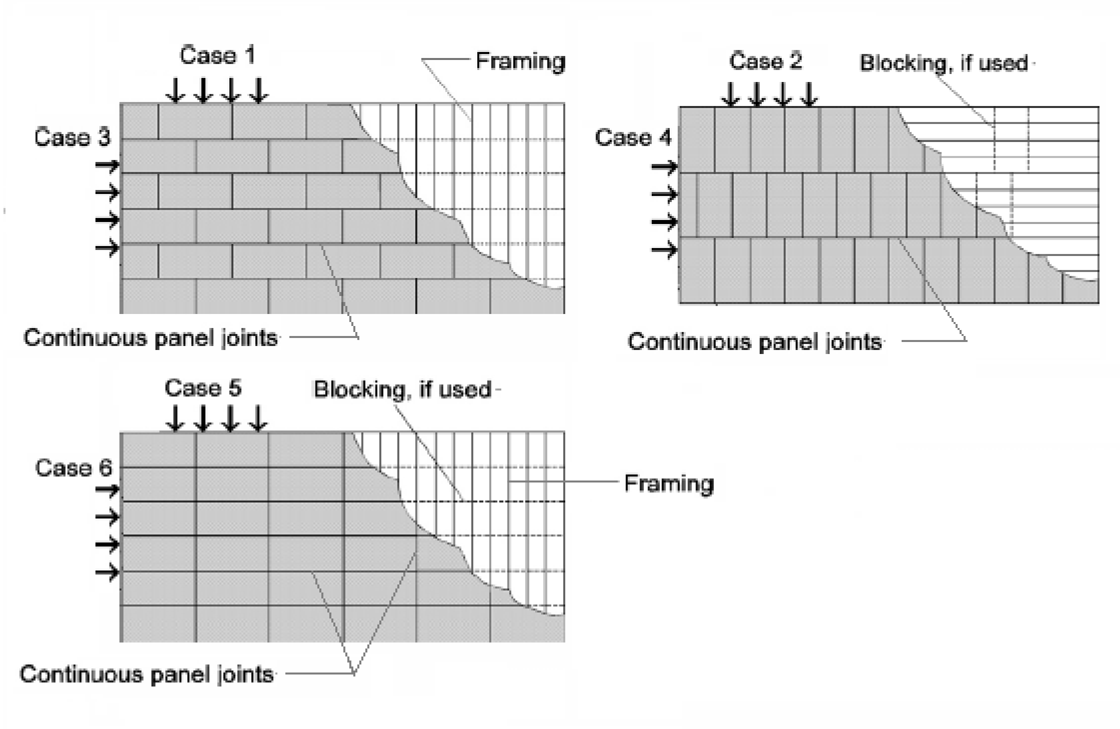
Strong Direction



Weak Direction

Panel Label reports the controlling panel layout that was chosen from the wood diaphragm database (See Appendix F)

Layout Case reports the case (panel layout) used for the diaphragm design. Because each diaphragm has a strong and weak direction, the program will actually report two cases associated with each diaphragm.



Blocked – This reports whether the diaphragm was designed as blocked or unblocked

Panel Thickness – This reports the decimal thickness of the wood panels. For convenience the following table lists the decimals that correspond with common panel thicknesses:

Panel Thickness	Decimal Equivalent
5/16	0.3125
3/8	0.375
7/16	0.4375
15/32	0.4688
19/32	0.5938

Required Nail Penetration – This is the minimum nail penetration as required by the IBC/NDS. This value is taken from the diaphragm nailing schedule/database. For more information on the database refer to Appendix F.

Nail Spacing Schedule

A diaphragm region is split into various nailing zones. The optimum nailing is chosen from the diaphragm nailing database for the highest point of shear of the diaphragm region in each direction. The program considers the Nail Spacing Increment as defined in the Design Rules. If a nailing arrangement is available that has spacing wider than the sum of the nailing increment and the spacing in the "higher" zone then a new nailing zone can be created. The arrangement of the nailing zones is such that they line up with the threshold shears of the capacities of the zones.

The nail spacing schedule reports the nail spacing and panel chosen for each zone. The zones are shown on a legend below.

Note:

- If there is only one zone defined in each direction, their default names will be A and D.

The following information is shown in the nail spacing schedule:

The **Lines** entry gives the number of lines of nails on each panel edge. Normally, this value is greater than one only for High-Load diaphragms. This entry is taken directly from the diaphragm nailing schedule database and is not otherwise used in the diaphragm design.

The **Location** entry defines the distance to that zone from the start of the diaphragm.

Note:

- If there is only one zone, the location value will just be zero.

Req'd Nom Framing Width displays the nominal width required of the framing members that support the diaphragm panels. This entry is taken directly from the diaphragm nailing schedule database and it not otherwise used in the diaphragm design.

The **Boundary** column gives the required boundary edge nail spacing.

The **Cont Edge** column gives the required nail spacing at continuous boundary edges that are parallel to the direction of load.

The **Other Edge** column gives the required nail spacing at all other edges. i.e. edges that are not considered to be boundaries or continuous edges parallel to load.

Deflection Calculations

Flexure Component

If you think of the diaphragm as a beam with the extreme chord members acting as the flanges, then this is the deflection due to the tension and compression forces that develop in the chords.

$$\delta_{flex} = \frac{5vL^3}{8EAW}$$

v = Maximum shear load in diaphragm region

L = Diaphragm dimension perpendicular to the direction of applied load

E = Chord modulus of elasticity

A = Chord cross-sectional area

W = Diaphragm dimension parallel to the direction of applied load

Shear Component

If you think of the diaphragm as a beam with the diaphragm sheathing acting as the web of the beam, then this component represents the deflection due to the shear forces in the sheathing. This term is based on the apparent shear stiffness (Ga) as described in the NDS document Special Design Provisions for Wind and Seismic. As such, this term includes the effects of both elastic shear deformation of the sheathing and nail slip of the panels.

$$\delta_{shear} = \frac{0.25vL}{1000G_a}$$

v = Maximum shear load in diaphragm region

L = Diaphragm dimension perpendicular to the direction of applied load

Ga = Apparent diaphragm shear stiffness

For diaphragm regions that have only one zone in each direction the value of Ga is taken directly from the nailing database. For multi-zone diaphragm regions the program internally calculates an equivalent Ga using the formula below:

$$G_a = \frac{1.4v_s}{\frac{1.4v_s}{Gt} + 0.75\alpha e_n}$$

vs = Diaphragm shear capacity (ASD value taken from nailing database)

Gt = Shear stiffness of panel depth (taken from nailing database)

en = Nail slip (taken from nailing diaphragm nailing schedule / database)

α = Nail slip adjustment factor. This factor accounts for non-uniform nail slip across multiple zones, and is derived from the process outlined in the ATC-7 and further explained in the APA design / Construction Guide on Diaphragms and Shears Walls.

Diaphragm Design Limitations

Rigid Diaphragms - Currently, the program does not provide any design or code check information for Rigid diaphragms. The rigid diaphragms are solely used to attribute load to the supporting frames or walls.

Non-Rectangular Diaphragms - Currently, the program does not provide any design or code check information for Non-Rectangular diaphragms. The rigid diaphragms are solely used to attribute load to the supporting frames or walls.

Skewed Diaphragms - Currently, the program only provides design for diaphragms that are parallel to the global axes.

Unblocked Diaphragm - Currently un-blocked diaphragms are not allowed to have multiple nailing zones.

Chord Slip - The deformation due to chord slip is not considered in the deflections calculations.

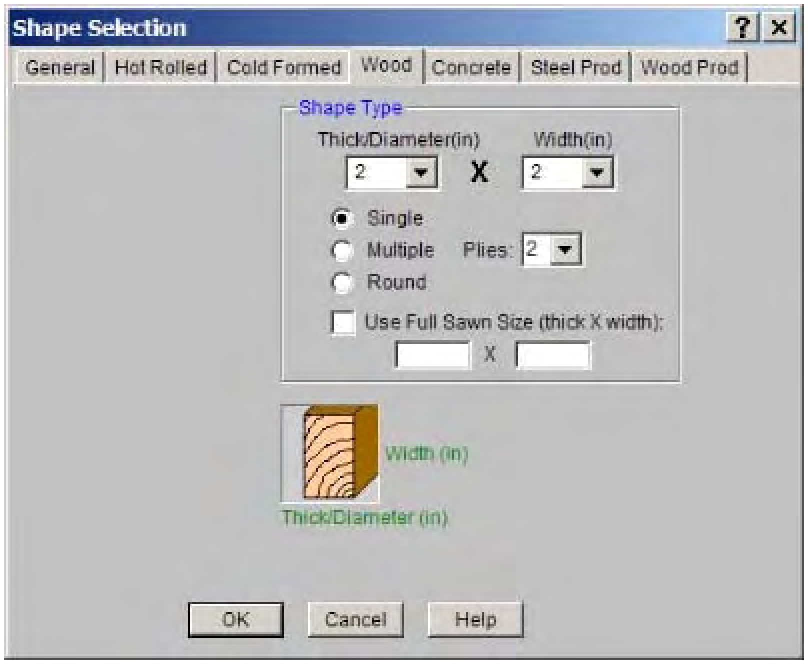
Sloped Diaphragms - Currently the program does not provide any design information for sloped diaphragms. However, it does provide load attribution to the supporting frames or walls.

Openings - Not design is performed for diaphragms with openings. However, diaphragm loads will still be attributed to the supporting frames or walls.

Specific Gravity and Moisture Content - The adjustments to capacity or stiffness related to specific gravity of the framing member or the moisture content of the lumber are not accounted for.

Wood - Database

The Wood Database may be accessed from the **Wood** tab of the **Section Sets** spreadsheet by clicking in the **Shape** field and then clicking , or by clicking the **Shape Database** button on the **RISA** toolbar and clicking the **Wood** tab.



To Select a Wood Database Shape

1. On the **Wood** tab of the **Section Sets Spreadsheet**, move the cursor to the **Shape** field and click .
2. Specify the shape type you wish to use (single, multiple, or round), then select from the lists of thicknesses and widths by clicking on .

Note

- If the value that you need is not given on the drop down list, you may directly enter any whole number for the thickness or width.
- Enter the nominal dimensions (or round diameter) in inches, regardless of what units system you are using. These will be automatically adjusted to the actual (dressed) dimensions for stiffness and stress calculations. For example, if you enter “2X4” as the size, the calculated properties are based on an actual size of “1.5 in. X 3.5 in.”

Custom Wood Sizes

If you would like to enter explicit dimensions of a member or if the member is "Not Dressed", the member must be designated as a "Full Sawn" member by checking the **Use Full Sawn Size** check box and entering the exact dimensions of the member in the boxes below. This applies to regular wood species, custom wood species, Structural Composite Lumber (SCL), and Glu-Lam members. The member thickness should be entered in the box on the left and the member width should be entered in the box on the right.

Wood Design Values

The program allows you to define members and walls as any of the wood species laid out in the NDS. The program does not explicitly call out the design values for these species, but they are taken directly from the NDS manuals. You can also get some verification of these design values by looking at the detail report for individual members.

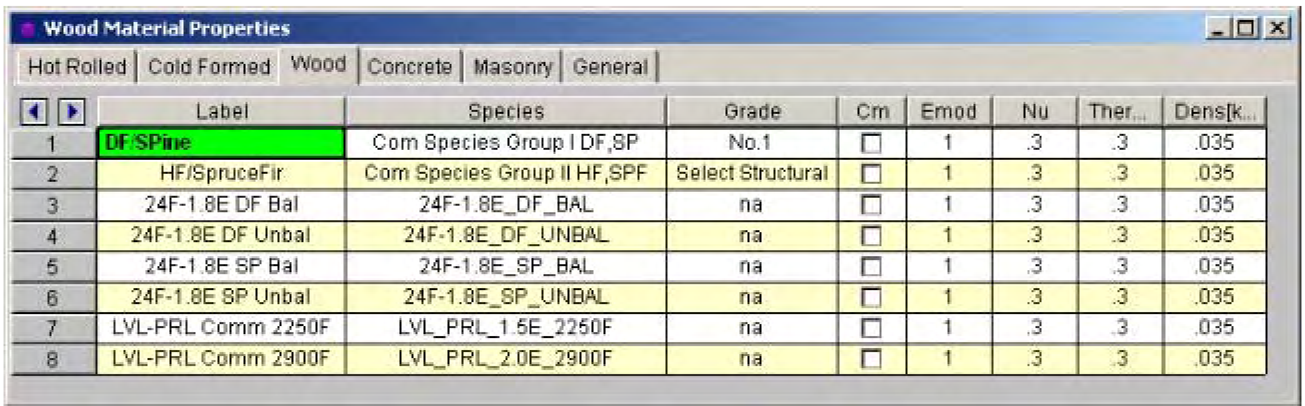
Also, the NDS has created some wood species groupings that will be implemented into the next version of the code. These new material call-outs have been implemented into the program. Here are some new materials defined for both RISAFloor and RISA-3D.

New NDS Wood Material Combinations

The **Com Species Group I DF, SP** and **Com Species Group II HF, SPF** come from the "Commercial Lumber Design Values" tables below.

The **Glulam** species groups come directly from the NDS 2005.

The **LVL** groupings come from the "Proposed PRL Commercial LVL Design Properties" below.



	Label	Species	Grade	Cm	Emod	Nu	Ther...	Dens[k...
1	DF/SPine	Com Species Group I DF,SP	No.1	☐	1	.3	.3	.035
2	HF/SpruceFir	Com Species Group II HF,SPF	Select Structural	☐	1	.3	.3	.035
3	24F-1.8E DF Bal	24F-1.8E_DF_BAL	na	☐	1	.3	.3	.035
4	24F-1.8E DF Unbal	24F-1.8E_DF_UNBAL	na	☐	1	.3	.3	.035
5	24F-1.8E SP Bal	24F-1.8E_SP_BAL	na	☐	1	.3	.3	.035
6	24F-1.8E SP Unbal	24F-1.8E_SP_UNBAL	na	☐	1	.3	.3	.035
7	LVL-PRL Comm 2250F	LVL_PRL_1.5E_2250F	na	☐	1	.3	.3	.035
8	LVL-PRL Comm 2900F	LVL_PRL_2.0E_2900F	na	☐	1	.3	.3	.035

Commercial Lumber Design Values, January 14, 2009

Joists and Rafters

dimension lumber 2" to 4" thick 2" and wider, Use with appropriate Adjustments for size and service conditions

Commercial Lumber Species Group	Grade	Fb (psi)	Ft (psi)	Fv (psi)	Fc-perp (psi)	Fc-para (psi)	E (psi)	Specific Gravity (OD Wt. & OD Vol.)
I	Select Structural	1500	1000	175	565	1700	1,800,000	0.50
	No. 1	1000	675			1500	1,700,000	
	No. 2	900	550			1350	1,600,000	
II	Select Structural	1250	700	135	405	1400	1,500,000	0.42
	No. 1	875	450			1150	1,400,000	
	No. 2	850	450			1150	1,300,000	

Wall Studs

dimension lumber 2" to 4" thick 2" and wider, Use with appropriate Adjustments for size and service conditions

Commercial Lumber Species Group	Grade	Fb (psi)	Ft (psi)	Fv (psi)	Fc-perp (psi)	Fc-para (psi)	E (psi)	Specific Gravity (OD Wt. & OD Vol.)
I	No. 1	1000	675	175	565	1500	1,700,000	0.50
	No. 2	900	550			1350	1,600,000	
	STUD	700	450			850	1,400,000	
II	No. 1	875	450	135	405	1150	1,400,000	0.42
	No. 2	850	450			1150	1,300,000	
	STUD	675	350			725	1,200,000	

Wall Plates

dimension lumber 2" to 4" thick 2" and wider, Use with appropriate Adjustments for size and service conditions

Commercial Lumber Species Group	Grade	Fb (psi)	Ft (psi)	Fv (psi)	Fc-perp (psi)	Fc-para (psi)	E (psi)	Specific Gravity (OD Wt. & OD Vol.)
I	No. 2	900	550	175	565	1350	1,600,000	0.50
	No. 3	525	325			775	1,400,000	
	Standard	575	375			1400	1,400,000	
II	No. 2	850	450	135	405	1150	1,300,000	0.42
	No. 3	500	250			650	1,200,000	
	Standard	550	275			1150	1,200,000	

Timbers - Beams and Columns

Beams and Stringers - 5" and Thicker, Width more than 2" greater than thickness, Use with appropriate Adjustments

Commercial Lumber Species Group	Grade	Fb (psi)	Ft (psi)	Fv (psi)	Fc-perp (psi)	Fc-para (psi)	E (psi)	Specific Gravity (OD Wt. & OD Vol.)
I	Select Structural	1500	950	165	375	950	1,500,000	0.50
	No. 1	1350	675			825	1,500,000	
	No. 2	850	425			525	1,200,000	
II	Select Structural	1300	750	140	405	925	1,300,000	0.43
	No. 1	1050	525			750	1,300,000	
	No. 2	675	350			500	1,000,000	

Posts and Timbers - 5"x5" and Larger, Width not more than 2" greater than thickness, Use with appropriate Adjustments

Commercial Lumber Species Group	Grade	Fb (psi)	Ft (psi)	Fv (psi)	Fc-perp (psi)	Fc-para (psi)	E (psi)	Specific Gravity (OD Wt. & OD Vol.)
I	Select Structural	1500	1000	165	375	950	1,500,000	0.50
	No. 1	1200	825			825	1,500,000	
	No. 2	750	475			525	1,200,000	
II	Select Structural	1200	800	140	405	975	1,300,000	0.43
	No. 1	975	650			850	1,300,000	
	No. 2	575	375			575	1,100,000	

Proposed PRL Commercial LVL Design Properties[a]

Grade/ Species	E (Joist)[b], 10^6 psi	F_b (Joist)[c], psi	F_t [d], psi	$F_{c//}$[e], psi	F_v (Joist)[f], psi	$F_{c\perp}$ (Joist)[g], psi	SG[h]				
							Nails and Wood Screws				Bolts and Lag Screws
							Withdrawal		Lateral		Lateral
							Face	Edge	Face	Edge	Face
1.5E-2250F	1.50	2,250	1,500	1,950	220	575	0.42	0.42	0.42	0.42	0.42
2.0E-2900F	2.00	2,900	1,900	2,750	285	750	0.50	0.47	0.50	0.50	0.50

(a) The tabulated values are design values for normal duration of load. All values, except E and $F_{c\perp}$, are permitted to be adjusted for other load durations as permitted by the code. The design stresses are limited to conditions in which the maximum moisture content is less than 16 percent.

(b) Bending modulus of elasticity (E), which is applicable in either edgewise or flatwise applications, includes shear deflections. For shear-free E, multiply the tabulated value by 1.03.

(c) Allowable bending stress (F_b) is applicable in either edgewise or flatwise applications. For depths of 3-1/2 inches or deeper when loaded edgewise, the tabulated F_b value shall be modified by $(12/d)^{1/6}$, where d is the actual depth in inches. For depths less than 3-1/2 inches when loaded edgewise, use the adjusted F_b for 3-1/2 inches. No adjustment on F_b is required for flatwise applications.

(d) Axial tension (F_t) of the LVL is based on a gauge length of 4 feet. For specimens longer than 4 feet, the tabulated F_t value shall be adjusted by $(4/L)^{1/8}$, where L is the actual length in feet.

(e) Compression parallel to grain ($F_{c//}$) of the LVL.

(f) Allowable shear stress (F_v) of the LVL when loaded edgewise.

(g) Allowable compressive stress perpendicular to grain ($F_{c\perp}$) of the LVL when loaded edgewise.

(h) Equivalent specific gravity for connection design.

Custom Wood Species

Within the program you are also able to define new materials in the Custom Wood Species spreadsheet. You can find this by going to Modify>>Custom Wood Species Database. This spreadsheet exists for those materials not found directly in the NDS. Many standard materials have been added here for your convenience. Any species defined here will appear in the "Species" dropdown in the Wood tab of the Materials spreadsheet.

Custom Wood Properties

	Label	Fb [ksi]	Ft [ksi]	Fv [ksi]	Fc [ksi]	E [ksi]	SCL
1	LVL_PRL_1.5E_2250F	2.25	1.5	.22	1.95	1500	☑
2	LVL_PRL_2.0E_2900F	2.9	1.9	.285	2.75	2000	☑
3	LVL_Microllam_1.9E_2600F	2.6	1.555	.285	2.51	1900	☑
4	PSL_Parallam_2.0E_2900F	2.9	2.025	.29	2.9	2000	☑
5	PSL_Parallam_1.8E	2.4	1.755	.18	2.5	1800	☑
6	LSL_TimberStrand_1.55E_2325F	2.325	1.07	.31	2.05	1550	☑
7	LSL_TimberStrand_1.3E_1700F	1.7	1.075	.4	1.4	1300	☑

Wood - Design

Full code checking can be performed on Dimension Lumber and Post and Timber size wood shapes based on the following codes:

- The 2005 edition of the NDS (National Design Specification)
- The 2001 edition of the NDS
- The 1991 / 1997 editions of the NDS

Note

- When the 1991 / 1997 NDS is selected, the 1991 NDS specification will be used with the 1997 stress tables. This is consistent with the requirements of the 1997 UBC.

Glu-Lams

Glu-Lams are treated as any other wood species and may be selected from the list of species on the Wood Tab of the Materials spreadsheet. When a Glu-Lam is selected, the grade will be listed as "na" or not applicable.

All Glu-Lam members should be dimensioned as "Full Sawn" using the format *wXdFS*, where "w" and "d" are the actual width and depth dimensions. If the size is entered as *wXd* without the *FS* designation, then the size will be dressed down as if the member were regular dimensional lumber.

RISA includes two redesign lists for Glu-Lams: Glu-Lam_Western for Western Species and Hardwoods (HW), and Glu-Lam_SouthernPine for Southern Pine (SP/SP).

Note

- Glu-Lams from Table5A are always assumed to have the special tension laminations. Therefore, the Fbx value is not reduced.
- RISA is NOT applying any of the footnotes to Table 5A and 5C at this time except for the values for Fvx and Fvy in Table 5A.

Custom Wood Materials & Structural Composite Lumber

To use a custom wood material that is not part of the standard NDS database, you will need to define the design properties of a new / custom wood species.

Custom Wood Properties

	Label	Fb [ksi]	Ft [ksi]	Fv [ksi]	Fc [ksi]	E [ksi]	SCL
1	Parallam PSL (DF)	2.5	1.755	.23	2.5	1.8	☑
2	DFL (per testing)	1.25	.85	.15	1.75	1750	☐

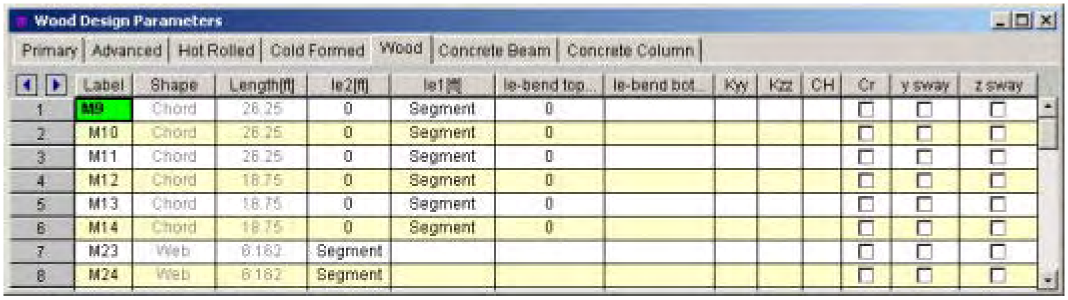
To do this, select **Spreadsheets** ▸ **Custom Wood Species Spreadsheet** from the main menu toolbar. Enter a label for your new custom species, then enter the wood properties (Fb, Fc, etc.) in the columns to the right. This new material will now be included in between the NDS wood species and the glulam types in the **Species** drop down list on the wood materials spreadsheet.

Note

- Fc is the base value for the compressive stress parallel to grain and will be used to calculate the member's ability to resist axial compression.
- The **SCL** checkbox is used to designate whether this new species is Structural Composite Lumber as defined in the 2001/05 NDS. The 2001/05 NDS has a code checking procedure for composite lumber that is slightly different from the procedure used for standard, dimensional lumber.

Wood Member Design Parameters

The **Member Design Parameters** spreadsheet records the design parameters for the material-specific code checks and may be accessed by selecting **Design Parameters** on the **Spreadsheets** menu. These parameters may also be assigned graphically. See Modifying Member Design to learn how to do this.



Wood Design Parameters

Primary | Advanced | Hot Rolled | Cold Formed | Wood | Concrete Beam | Concrete Column

	Label	Shape	Length[ft]	le2[ft]	le1[ft]	le-bend top...	le-bend bot...	Kyy	Kzz	CH	Cr	y sway	z sway
1	M9	Chord	26.25	0	Segment	0					☐	☐	☐
2	M10	Chord	26.25	0	Segment	0					☐	☐	☐
3	M11	Chord	26.25	0	Segment	0					☐	☐	☐
4	M12	Chord	18.75	0	Segment	0					☐	☐	☐
5	M13	Chord	18.75	0	Segment	0					☐	☐	☐
6	M14	Chord	18.75	0	Segment	0					☐	☐	☐
7	M23	Web	6.182	Segment							☐	☐	☐
8	M24	Web	6.182	Segment							☐	☐	☐

These parameters are defined for each member.

Label

You may assign a unique **Label** to all of the members. Each label must be unique, so if you try to enter the same label more than once you will get an error message. You may relabel at any time with the **Relabel** options on the **Tools** menu.

Shape

The member **Shape** or Section Set is reported in the second column. This value is listed for reference only and may not be edited as it is dictated by the entry in the Section/Shape column on the **Primary** tab.

Length

The member **Length** is reported in the third column. This value may not be edited as it is dependent on the member end coordinates listed on the **Primary Data** tab. It is listed here as a reference for unbraced lengths which are discussed in the next section.

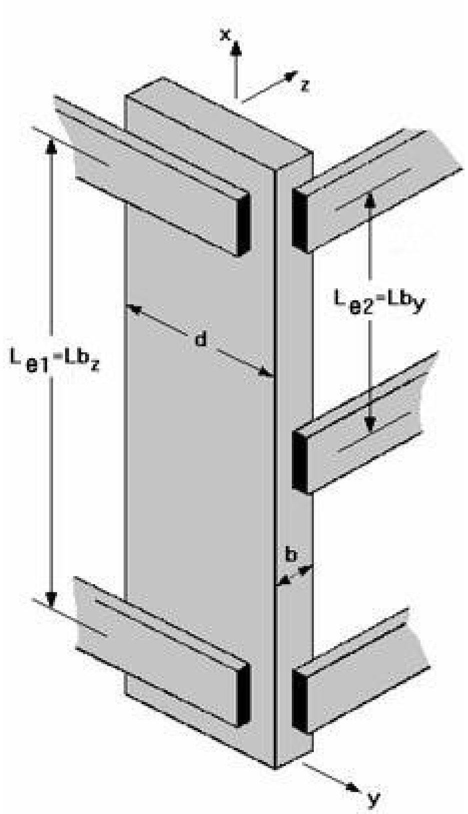
Unbraced Length

You may specify unbraced lengths or have RISA-3D calculate them for you. The unbraced lengths are **Le1**, **Le2**, **Le-bend-top**, and **Le-bend-bot**.

The values **Le1** and **Le2** represent the unbraced length for the member with respect to column type buckling about the member's local z and y axes, respectively. These **Le** values are used to calculate Le1/d and Le2/b, which in turn impact the calculation of Cp, the column stability factor. These length to thickness ratios gauge the vulnerability of the member to buckling. Refer to Section 3.7 of the NDS for more information on this. This section also lists the limiting values of the length to thickness ratios.

The **Le-bend** values, **Le-bend-top** and **Le-bend-bot**, are the effective unbraced lengths of the member for bending. This unbraced length is the length of the face of the member that is in compression from any bending moments. **This value should be obtained from Table 3.3.3 in the NDS code**. The **Le-bend** value is used in the calculation of the slenderness ratio, RB, which is used in the calculation of CL, the beam stability factor. CL is then used to calculate the allowable bending stress. Refer to Section 3.3.3.6 in the NDS for more information on this and note that the value of RB is limited to 50.

For continuous beams the moment will reverse such that the top and bottom faces will be in compression for different portions of the beam span. **Le-bend-top** is the unbraced length of the top face and **Le-bend-bot** is the unbraced length of the bottom face.



If left blank these unbraced lengths all default to the member's full length. The exception to this is if **Le2** is entered and **Le-bend-top** is left blank, **Le-bend-top** will default to the entered value for **Le2**. Since **Le2** and **Le-bend** are often different in wood design, it is likely you should enter the correct value for **Le-bend**.

For physical members, you can enter the code "**Segment**" in the unbraced length fields and the length of each segment will be used. A "segment" is the distance between the joints that are on the physical member. For example, suppose you have a physical member that is 20 feet in length, and there are two joints along the physical member, one 5 feet from the end and one at 15 feet. An unbraced length of 5 feet will be used for the first segment, then a value of 10 feet will be used in the middle segment, and again a value of 5 feet would be used in the last segment.

Note

- If the intermediate framing members are considered to brace the bottom flange, then you can enter “segment” for Le-bend-bot. When the “segment” command is used ALL intermediate points along the beam are viewed as brace points. Therefore, you may have to delete unused or extraneous points.
- The Top Flange is defined as the flange corresponding to the positive local y axis for the member. For more information on setting local axes refer to the Members section.
- The calculated unbraced lengths are listed on the **Member Detail** report.

K Factors (Effective Length Factors)

The **K Factors** are also referred to as effective length factors. **Kyy** is for column type buckling about the member's local y-y axis and **Kzz** is for buckling about the local z-z axis.

If a value is entered for a **K Factor**, that value will be used for the entire length of the physical member. If an entry is not made (left blank), the value will internally default to '1' for that member. See the NDS Appendix G for an explanation of how to calculate K Factors. For wood the **K Factors** are applied to **Le1** and **Le2** to obtain the effective column length. See section 3.7 in the NDS for more on this.

RISA-3D is able to approximate the K values for a member based on the member's sway condition and end release configuration. The K-factor approximation is based on Table G1, found in Appendix G, of the NDS 2005 specification. The following table gives the values used for various conditions.

Table Case	End Conditions	Sidesway?	K-Value
(a)	Fixed-Fixed	No	.65
(b)	Fixed-Pinned	No	.80
(c)	Fixed-Fixed	Yes	1.2
(d)	Pinned-Pinned	No	1.0
(e)	Fixed-Free	Yes	2.1
(f)	Pinned-Fixed	Yes	2.0

RISA-3D will recognize a pinned boundary condition for the K approximation for a full pin, i.e. if all the rotations in the boundary condition are released. If any of the rotations in a boundary condition are restrained, the boundary condition is considered “fixed” for the K approximation.

Any configuration not described here will be given the default value of 1.0.

If any value that influences these K values is changed, the K approximation should be redone. For instance, if you have RISA-3D approximate K values then change some end release designations, you should redo the K approximations.

Remember that the K-values are *approximations*, and you should check to make sure you agree with all K-values RISA-3D assigns. You can always override a K-value after an approximation by directly entering the value that you want in the appropriate field. Keep in mind that a subsequent approximation will overwrite any manually input values so you will need to override the approximation each time it is performed.

Limitation:

RISA-3D will currently neglect the influence of adjoining framing members when those members are connected at a joint that also has degrees of freedom restrained by boundary conditions. For example, suppose a column and beam member connect at a joint that is restrained for translation in all directions (i.e. the joint is “pinned”). The K factor approximation will neglect the beam member when it calculates the K factor for the column and visa-versa. The effect will be that the ends of the members at that joint will be seen as “pinned” and not “fixed” for the K-factor approximation.

Sway Flags

The **Sway Flags** indicate whether the member is to be considered subject to sidesway for bending about its local y and z axes. The **y sway** field is for y-y axis bending and the **z sway** field is for z-z axis bending. Click on the field to check the box and indicate that the member is subject to sway for that particular direction, or leave the entry blank if the member is braced against sway. These sway flags influence the calculation of the K factors.

Timber Design Adjustment Factors

The NDS code has a lot of adjustment factors that you apply to the various allowable stresses, and in some cases, to the Young's modulus (E). All of the adjustment factors are summarized in section 2.3 of the NDS-2005 specification. The following topics help to summarize how adjustment factors are obtained and used. **The CT, Ci, and Cb factors are NOT used.**

Timber Design CH (Shear Stress Factor)

The Shear Stress Factor entry, **CH**, is the shear stress adjustment factor. This design parameter can be set on the Wood tab of the **Members** **Spreadsheet**. If left blank the program will use a default value of 1.0. See the tables in the NDS supplement for information on other CH factors. Note that only tables 4A, 4B, and 4D are used. The CH factor is only available for the 1991/1997 NDS codes. For other codes, this entry will be ignored.

Timber Design Cr (Repetitive Factor)

The Repetitive Factor field, **Cr**, specifies if the beam is one of a group of repetitive members. This design parameter can be set on the **Wood** tab of the **Members Spreadsheet**. If you put a check in the Cr field, a factor of 1.15 will be applied to beam members that are 2" to 4" thick. This flag will be ignored for a NDS shape that is thicker than 4". A value of '1.0' will be used for Wood Products. Different restrictions apply to the use of the Cr factor for Structural Composite Lumber and Glu-Lams.

Timber Design Ct (Temperature Factor)

The Temperature Factor, **Ct**, is calculated internally from the wood temperatures set in the Global Parameters. See section 2.3.3 of the NDS-2005 specification for more information on the temperature factor.

Timber Design Cfu (Flat Use Factor)

The Flat Use Factor, **Cfu**, is automatically applied to the weak axis allowable bending stress of a wood member whenever weak axis moments are present. The flat use factor will only be applied to members that are 2" to 4" thick. See the NDS 2005, Tables 4A, 4B, 4C, 4F, 5A, 5B, 5C, 5D and the footnotes.

Timber Design CF (Size Factor)

The Size Factor, **CF**, is applied automatically when you assign a wood shape from the NDS shape database. See Tables 4A, 4B, 4D, and 4E in the NDS supplement for information on the CF factor.

Timber Design CV (Volume Factor)

The Volume Factor, **CV**, is applied automatically when you assign a Glu-Lam member from the NDS shape database. The user can override the calculated value by inputting the factor on the **Wood** tab of the **Members Spreadsheet**. This entry is only available when using the 2001 or 2005 NDS code.

Note:

- In the calculation of Cv, RISA takes L conservatively as the full length of the member.

Timber Design Cf (Form Factor)

The Form Factor, **Cf**, is applied automatically when designing by the NDS 91/97 or 2001 Specification and a 'Round' shape is selected from the NDS shape database. See section 2.3.8 in the NDS (91/97, 2001) for information on the Cf factor.

Note

- This factor is not applied when design by the NDS 2005 Specification
- This factor is not applied to "diamond" shaped members, which are just rectangular members on edge. This factor is not applied to diamond shapes because any applied moments are transformed internally to the local member axes for the code check calculations, which is the same as applying the “diamond” form factor and NOT transforming the moments.

Timber Design Cm (Wet Service Factor)

The Wet Service Factor, **Cm**, is applied when you check the **Cm** checkbox in the **Materials Spreadsheet**.

Timber Design CP and CL (Column/Beam Stability Factors)

The Column Stability Factor, **CP**, and the Beam Stability Factor, **CL**, are calculated internally. These calculated values are shown on the **Wood** tab of the **Design Results Spreadsheet**, as well as in the **Member Detail Reports**. See NDS 2005 section 3.3.3 for information on the CL factor and NDS 2005 section 3.7.1 for information on the CP factor.

Timber Design CD (Load Duration Factor)

The Load Duration Factor, **CD**, is entered on the **Load Combination Spreadsheet** for each load combination for which you want wood code check results. The CD factor must be entered for each individual load combination because the CD factor is dependent on the types of loads that are applied in each load combination. Therefore, different load combinations could have different CD factors. For example, per the NDS 2005 specification, a load combination that had only dead load, would have a CD factor of “0.9”, while another combination that was comprised of dead load plus wind load would have a CD factor of “1.6”.

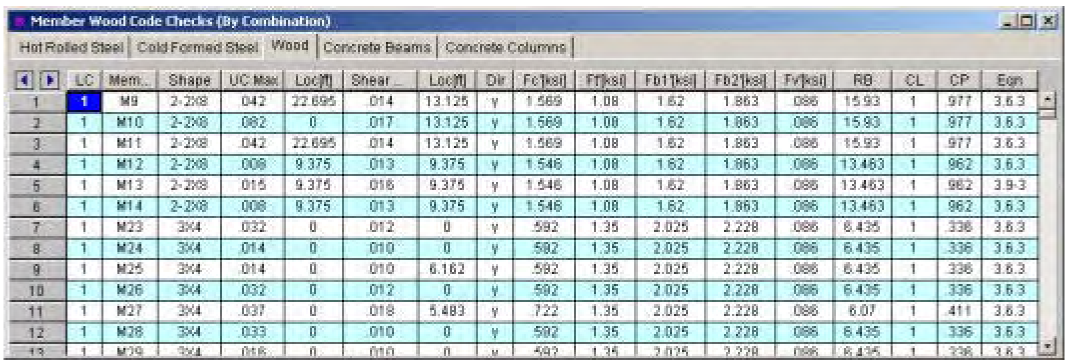
The CD factor will only be applied to wood code checks on wood members. See Table 2.3.2 in the NDS 2005 specification for the CD factors to be applied for typical loads. Appendix B of the NDS has additional information about the Load Duration Factor.

Note

- The CD factor used for a load combination should be for the load with the shortest load duration in that load combination.

Wood Member Code Check Results

Access the **Wood Code Checks Spreadsheet** by selecting the **Results** menu and then selecting **Members** ▸ **Design Results** and then clicking the **Wood** tab.


Member Wood Code Checks (By Combination)

Hot Rolled Steel | Cold Formed Steel | Wood | Concrete Beams | Concrete Columns

	LC	Mem...	Shape	UC Max	Loc[ft]	Shear ...	Loc[ft]	Dir	Fc'[ksi]	Ft'[ksi]	Fb1'[ksi]	Fb2'[ksi]	Fv'[ksi]	RB	CL	CP	Eqn
1	1	M9	2-2X8	.042	22.695	.014	13.125	y	1.569	1.08	1.62	1.863	.086	15.93	1	.977	3.6.3
2	1	M10	2-2X8	.082	0	.017	13.125	y	1.569	1.08	1.62	1.863	.086	15.93	1	.977	3.6.3
3	1	M11	2-2X8	.042	22.695	.014	13.125	y	1.569	1.08	1.62	1.863	.086	15.93	1	.977	3.6.3
4	1	M12	2-2X8	.008	9.375	.013	9.375	y	1.546	1.08	1.62	1.863	.086	13.463	1	.962	3.6.3
5	1	M13	2-2X8	.015	9.375	.016	9.375	y	1.546	1.08	1.62	1.863	.086	13.463	1	.962	3.9-3
6	1	M14	2-2X8	.008	9.375	.013	9.375	y	1.546	1.08	1.62	1.863	.086	13.463	1	.962	3.6.3
7	1	M23	3X4	.032	0	.012	0	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3
8	1	M24	3X4	.014	0	.010	0	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3
9	1	M25	3X4	.014	0	.010	6.162	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3
10	1	M26	3X4	.032	0	.012	0	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3
11	1	M27	3X4	.037	0	.018	5.483	y	.722	1.35	2.025	2.228	.086	6.07	1	.411	3.6.3
12	1	M28	3X4	.033	0	.010	0	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3
13	1	M29	3X4	.016	0	.010	0	y	.592	1.35	2.025	2.228	.086	6.435	1	.336	3.6.3

The final result of the code checking is a code check value. This value represents a ratio of actual stress to allowable stress. So, if this value is less than 1.0, the member passes. If it is greater than 1.0, the member fails. If the value is greater than 9.999 it will be listed as "9.999". The **Loc** field tells at what location the maximum code check occurs measured from the I-joint location of the member.

The Shear Check is the maximum ratio of actual to allowable shear stress. The location for the shear check is followed by "y" or "z" to indicate the direction of the shear.

The remaining columns, discussed below, provide some of the values used in the code check with the equation number itself given in the last column. The Member Detail Report described gives more values used to perform the code check. See Plot Options – Members to learn how to view the code check results graphically.

The values (Fc', Ft', Fb1', Fb2', Fv') are the factored allowable stresses. The unfactored allowable stresses are those listed on the **Wood Properties Spreadsheet**. For the bending stresses (Fb), Fb1' is for bending about the local z-z axis (the strong axis) and Fb2' is for bending about the local y-y axis (the weak axis).RB is the adjustment factor described by Eqn. 3.3-5 of the 2005 NDS Specification. This is a slenderness ratio that is not allowed to exceed 50. CL is the beam stability factor calculated using Eqn. 3.3-6 of the NDS Specifications. CP is the column stability factor calculated using Eqn. 3.7-1 of the NDS Specifications.

Finally, the equation controlling the code check is listed, either Eqn. 3.9-1 or 3.9-3. Eqn. 3.9-2 is not checked since this equation includes the tension stress in a beneficial (non-conservative) manner. All other requirements in Section 3.9 are also checked, such as fc < FcE1, etc. To see ALL the adjustment factors and other information used to calculate the factored allowable stresses, please go to a detail report for the member in question. You can do that from this spreadsheet by clicking Detail Report for Current Member.

For enveloped results the combination that produced the listed code and shear checks is given in the column "lc". The other values are the corresponding values and are not necessarily the maximums across all the combinations.

Note

- The Member Detail Report gives more values used to perform the code check.
- See Spreadsheet Operations to learn how to use **Find**, **Sort** and other options.
- See Plot Options – Members to learn how to plot member results.

Special Messages - Wood Design

In some instances code checks are not performed for a particular member. A message explaining why a code check is not possible will be listed instead of the code check value. You may click the cell that contains the message and look to the status bar to view the full message. Following are the messages that may be listed:

NDS Code Check Not Calculated

This is the general message displayed when code checks were not performed for a member.

RB value is greater than 50

Section 3.3.3.7 of the NDS 1991/1997, 2001 and 2005 codes limits the slenderness ratio RB to a maximum of 50. You need to reduce the effective span length, increase the thickness of the shape, or reduce the depth of the shape.

le/d is greater than 50

Section 3.7.1.4 of the NDS 1991/1997, 2001 and 2005 codes limits the column slenderness ratio of Le1/b or Le2/d to a maximum of 50. You need to reduce your effective length by reducing the actual length between supports or changing the effective length factor “K”. You can also use a thicker shape.

fc is greater than FcE1

Section 3.9.3 of the NDS 1991/1997, 2001 and 2005 codes limits the actual axial compressive stress to be less than the term FcE1. This term is approximately the Euler buckling stress for buckling about the strong axis of the member. (Buckling is in the plane of bending)

fc is greater than FcE2

Section 3.9.3 of the NDS 1991/1997, 2001 and 2005 codes limits the actual axial compressive stress to be less than the term FcE2. This term is approximately the Euler buckling stress for buckling about the weak axis of the member. (Buckling is in the plane of bending)

fb1 is greater than FbE

Section 3.9.3 of the NDS 1991/1997, 2001 and 2005 codes limits the actual strong axis bending compressive stress to be less than the term FbE. This term is approximately the lateral buckling stress.

Limitations - Wood Design

It is assumed that the axial load on the member is occurring through the member's shear center. This means local secondary moments that may occur if the axial load is not applied through the shear center are not considered.

Buckling Stiffness Factor - The buckling stiffness factor, CT, is not currently accounted for.

Incising Factor - The incising factor, Ci, is not currently accounted for.

Bearing Area Factor - The bearing area factor, Cb, is not currently accounted for.

Column Stability Coefficient for bolted and nailed built-up columns - This factor (K_f) is always taken as 1.0 (See NDS 15.3)

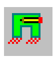
Wall Panels

The wall panel element allows you to easily model walls for in plane and out of plane loads. Wall panel data may be viewed and edited in two ways: graphically in the Wall Panel Editor or in the **Wall Panels Spreadsheet**.

Drawing Wall Panels

There are several graphic-editing features that make the creation and modification of models quite easy. Use the Insert and Modify menus or the Drawing Toolbar to use these features in the model view. To create new wall panels, you can draw them using a drawing grid or draw "dot to dot" existing joints. Once you have created these items you may use other graphic features to apply loads and set boundary conditions.

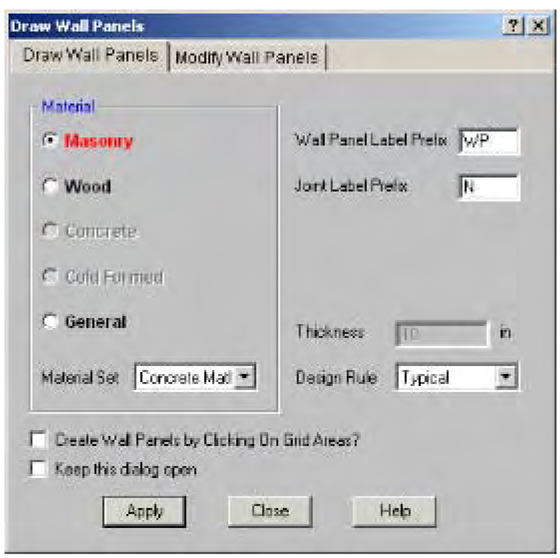
You can set many of the wall panel properties up front or you can modify these properties after you draw them. Modifying properties is discussed in the next sections. See **Wall Panels Spreadsheet** for information on wall panels and their properties.

The **Draw Wall Panels**  button lets you graphically draw wall panels in your model. Enter the appropriate wall panel parameters, click **OK** and draw wall panels between existing joints or on the drawing grid. You will also notice that the coordinates of the joint or grid point that is closest to your cursor are displayed in the lower right hand corner of the model view. The new wall panels will be shown on screen and will be recorded in the **Wall Panels Spreadsheet**.

To actually draw a wall panel, you have two options. One way is to modify your Drawing Grid according to how you wish to lay out your wall panels and use the **Create Wall Panels by Clicking on Grid Areas** option. Wall panels can then be created by clicking in the grid areas formed by the intersecting grid lines. As you click on an area, a wall panel will automatically be created in that area. The second option is to create wall panels by drawing them one joint at a time. First click on the grid point or joint that you want to be the "A" joint for the plate, then the "B" joint, "C" joint, and "D" joint in either clockwise or counter-clockwise order. The wall panel will "stretch" like a rubber band as you draw from joint to joint.

Note:

- You must draw wall panels as rectangular. Wall panels that have a slope at the top can be modeled but you must use the joint coordinates spreadsheet to modify the joint locations at the top of an already defined rectangular wall.
- Wall panels must be oriented vertically in your model.



The parameters shown are the same parameters that you would enter on the **Wall Panels Spreadsheet.**

-

To Draw Wall Panels

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. If you are not drawing between existing joints, you will need to create a drawing grid or define joints on the **Joint Coordinates** spreadsheet.
3. Click the **Draw / Modify Wall Panels** button and select the **Draw Wall Panels** tab. Then set the wall panel properties.
4. Click **Apply** to start drawing wall panels by clicking on the joints or grid points with the left mouse button.
    1. You must click four points in a clockwise or counter-clockwise order.
    2. Or, click in grid areas.
5. To stop drawing altogether right click or press the **Esc** key.

Note

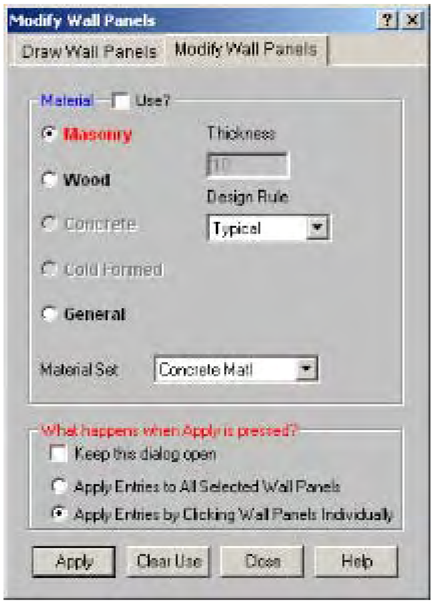
- To draw more wall panels with different properties, press CTRL-D to recall the **Wall Panel Properties** settings.
- You may also specify or edit wall panels in the **Wall Panels Spreadsheet.**
- You may also view and edit wall panel properties by double-clicking on a wall panel.
- You may undo any mistakes by clicking the Undo button.

Modifying Wall Panels

There are a number of ways to modify wall panels. You may view and edit the member data in the **Wall Panel Spreadsheet**, you may double-click a wall panel to view and edit its properties, or you can use the **Modify Wall Panels** tool to graphically modify a possibly large selection of panels.

The graphical **Wall Panel Modify** tool discussed here lets you modify the properties of wall panels that already exist in your model. To use this, you will typically specify the properties you want to change, then select the wall panels that you want to modify. You can modify wall panels one at a time by selecting the **Click to Apply** option and then click on the wall panels you wish to modify. You may also modify entire selections of wall panels by selecting the wall panels first and then use the **Apply to Selected** option. See the Graphic Selection topic for more on selecting.

The parameters shown are the same as those used to define new wall panels.



The **Use?** check boxes next to the data fields indicate whether the particular parameter will be used or not when the modification is applied. If the box next to a field is checked, that parameter will be applied to any selected wall panels. If the box is NOT checked, the parameter will NOT be applied, even if a value is entered in the field. This lets you easily change one or two properties on members without affecting all the rest of the properties. Note that if a no value is entered in a field (i.e. the field is blank) and the corresponding check box is checked, clicking "Apply" will have the effect of clearing the data for these fields.

To Modify Wall Panels

1. If there is not a model view already open then click on the **RISA Toolbar** to open a new view and click to turn on the **Drawing Toolbar** if it is not already displayed.
2. Click the **Draw / Modify Wall Panels** button and select the **Modify Wall Panels** tab. Then set the parameters for the new wall panels. Check the **Use?** Box for the items to apply.
3. You may choose to modify a single wall panel at a time or to an entire selection of wall panels.
    1. To modify a few wall panels choose **Apply Entry by Clicking Items Individually** and click **Apply**. Click on the wall panels with the left mouse button.
    2. To modify a selection of wall panels, choose **Apply Entries to All Selected Items** and click **Apply.**

Note

- To modify more wall panels with different parameters, press CTRL-D to recall the **Modify Wall Panels** settings.
- You may also modify wall panels in the **Wall Panels Spreadsheet**.
- You may undo any mistakes by clicking the **Undo** button.
- The thickness option is only available if you are choosing a General material. Wood and masonry require you to change their thickness in the Design Rules spreadsheet.

Wall Panel Spreadsheets

Another way of adding or editing wall panels is through the **Wall Panel Spreadsheet**. This spreadsheet is accessible through the **Data Entry Toolbar** and includes primary joint, material, and thickness data.


Wall Panel Data

	Label	A Joint	B Joint	C Joint	D Joint	Material Type	Material Set	Thickness	Design Rule	Design Method	SSRF
1	WP1	N1	N2	N3	N4	Wood	DF Larch	5.5	Typical	Perforated	1
2	WP2	N5	N6	N7	N8	Wood	DF Larch	5.5	Typical	Segmented	1
3	WP3	N9	N10	N11	N12	General	gen_Conc3NW	8	Typical	N/A	N/A
4	WP4	N13	N14	N15	N16	Masonry	Concrete Matl	10	Typical	N/A	N/A

The following data columns hold the primary data for the wall panels:

Wall Panel Labels

You may assign a unique label to any or all of the wall panels. You can then refer to the wall panel by its label. Each label has to be unique, so if you try to enter the same label more than once you will get an error message. You may relabel wall panels at any time with the Relabel Wall Panels option on the Tools menu.

Wall Panel Joints

The A, B, C, and D joint entries are used to define the 4 corner joints of a wall panel. The joints must all lie on the same plane, be oriented parallel to the vertical axis and be entered in either a clockwise or counter-clockwise sequence.

Wall Panel Material Type and Material Set

The material set label links the wall panel with the desired material defined on the Material Spreadsheet.

Note

- Currently wall panels can only be made up of masonry, wood, or general materials.

Wall Panel Thickness

The thickness field on the Wall Panels Spreadsheet is the thickness of the element. This thickness is constant over the entire element. Note that the thickness for Masonry and Wood wall panels are set in the Design Rules spreadsheet.

Design Rule

This allows you to choose a specific design from the Design Rules spreadsheet.

Design Method

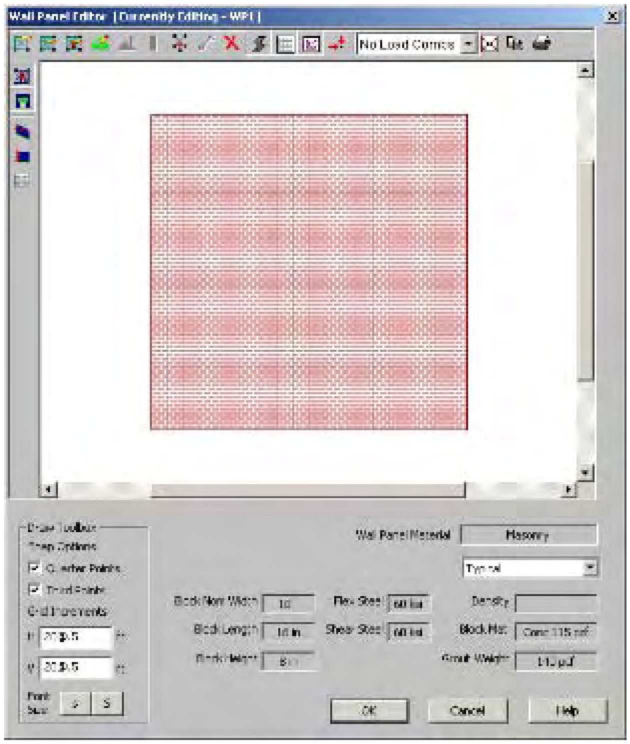
This is a column specific to wood wall panels and allows you to choose which design method you choose to work with: Segmented, Perforated or Force Transfer. See the Wood Wall Panels topic for more information. These design methods are not applicable for masonry or general wall panels.

SSRF (Shear Stiffness Reduction Factor)

This column allows the user to manually reduce the shear stiffness of a particular wall panel. Because the program uses a finite element solution the program does not automatially consider some contributions to the FEM deflections, such as nail slip. With this reduction factor the user can match up the deflections from their hand calculations with the FEM joint deflections at the top nodes in the wall.

Wall Panel Editor

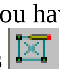
The **Wall Panel Editor** allows the user to edit the detailed properties of a wall panel including openings, regions, boundary conditions, end releases, hold downs and straps. This also gives design options and details for the specific panel. This application is accessible by double clicking on an existing wall panel.



Note:

- There are many icons, dropdown lists and information shown depending on the type of wall panel you are working with. See the Masonry Wall - Design and Wood Wall - Design topics for more information.

Creating Openings

Within the **Wall Panel Editor**, you have the option of adding rectangular openings to the wall panel. To draw an opening, select the **Create New Openings**  button and then select two nodes or grid intersections which make up the two diagonal corners of your opening. Notice that you can view your cursor coordinates in the lower right portion of your screen. To exit this tool right-click your mouse.

Note:

- Drawing an opening in a masonry wall will create a lintel above the opening. For more information on defining lintel geometry and design properties, see the **Masonry Wall - Design** topic on lintels.
- Drawing an opening in wood wall will create a header above the opening. For more information on defining the header properties, see the **Wood Wall - Design** topic on headers.
- When drawing an opening in a general wall panel, there is no header/lintel automatically created. This is because the program does not currently support concrete wall rebar design. The general wall panel is given as an option for analysis only. In a future release wall panel reinforcing will be implemented.
- Openings can not overlap a region. Regions must be deleted before you draw an opening in an area. After the opening is created you can go back and redraw the regions.

Creating Regions

Within the Wall Panel Editor, you also have the option of creating different rectangular regions within your wall panel. Regions are used to further define areas of your wall panel for use in analysis/design. If you do not specify a region in a wall panel without openings, then the entire wall panel will be considered a region.

To automatically draw regions you must first have your openings input. Once you have that you can click the **Generate Wall Regions Automatically** button and the program will define regions as we would expect a user to want them.

Note:

- If the regions defined are not located correctly by the generator, you can delete the generated regions with the Delete button and redraw them manually. See below for more information on this.

To manually draw a region, select the **Create New Regions** button and use your cursor to select two nodes or grid intersections which make up the diagonal corners of the region. To exit this tool right-click your mouse.

Note:

- For masonry wall panels, there is a region editor that allows you to define design properties for the region. See the **Masonry Wall - Design** topic in the General Reference Manual for region information. Note that design and analysis results are displayed by region.
- For wood wall panels using the Segmented method of design, the design and analysis results are displayed by region. The other options, perforated and force transfer around openings, use regions but don't use them for display of results.
- For general wall panels we will not do any design for you. However, you can lay out your regions so that your analysis results will allow you to design your general wall panels much easier.

Boundary Conditions

Within the **Wall Panel Editor**, all boundary conditions are applied as continuous along a wall panel edge. To set boundary conditions within the wall panel editor, select the **Create New Boundary Conditions** button, select your boundary condition criteria, and select **Apply**. To exit out of this tool right-click your mouse. You can also apply boundary conditions to your wall panel outside of the Wall Panel Editor as well, but this is the only place where you can define a continuous boundary condition.

For wood wall panels hold downs and straps are used in the program as well. For more information on adding holddowns and straps, see the Wood Wall - Design topic.

Draw Toolbox

The draw toolbox, which appears in the lower left corner of the Wall Panel Editor screen gives the user options for drawing within the Wall Panel Editor window. The options include:

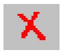
Snap Options allows you to provide snap points at the edges of the wall panel at quarter and third points.

Grid Increments allow you to set a drawing grid within the Wall Panel Editor separate from that in the main model view that you can snap to when drawing openings and regions.

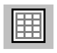
Font Size allows you to increase or decrease the font size associated with the region and opening titles and information shown in the Wall Panel Editor window.

View Controls

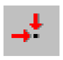
In addition to the wall panel editing tools, the **Wall Panel Editor** window includes the following view controls:

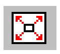
 **Delete** allows you to delete openings or regions from the wall panel.

 **Render** will turn rendering of the current model view on or off, depending on the current setting.

 **Drawing Grid** will turn the display of the Drawing Grid on or off, depending on the current setting.

 **Diaphragm Display** will turn the display of the Diaphragms on or off, depending on the current setting.

 **Loads** will turn the display of the wall panel loads on or off, depending on the current setting.

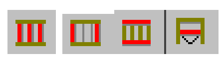
 **Redraw Full Wall View** redraws the wall panel to fit within the **Wall Panel Editor** window.

 **Copy Current View** makes a copy of the current view and saves it to the clipboard.

 **Print Current View** prints your current wall panel view.

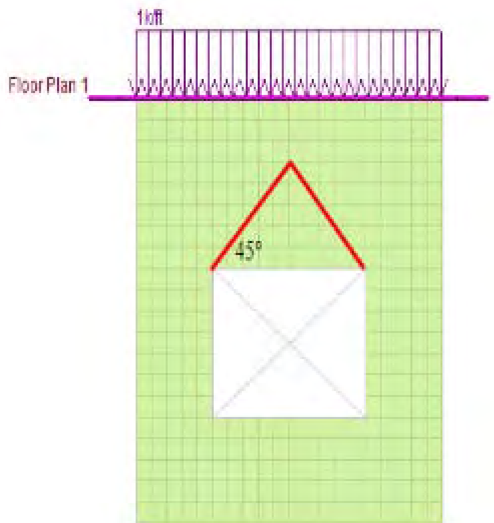
Note:

- There are also view controls specific to wood. For more information see the Wood Wall - Design topic.

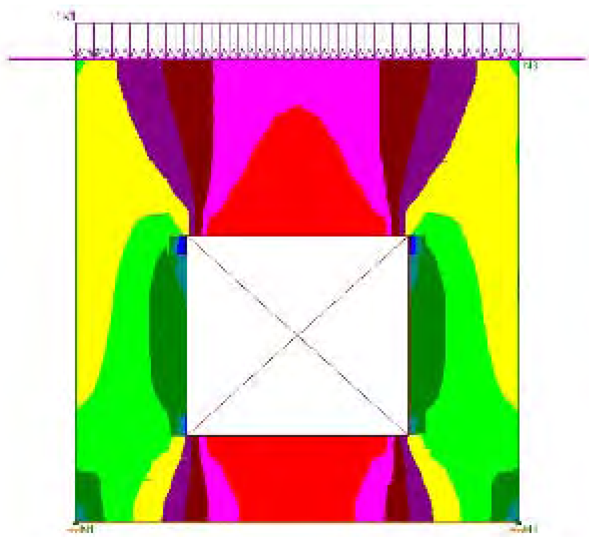


Load Attribution

In RISA-3D, the use of finite elements dictates how loads pass through wall panels. Load is attributed to the structure according to relative stiffnesses of elements. In wood and masonry design, many empirical equations are formed based on approximations or idealizations. Because of this, you may not get your loading in your wall panel elements (regions and lintels) to match hand calculations exactly. A prime example of this occurs with the 45 degree rule for lintels. According to theory, arching action occurs in lintels to the point that, if the top of your wall is a sufficient distance away, only the load in the triangular portion above your lintel would actually be taken into the lintel itself. Also, no load applied at the floor level would be felt by the lintel either. See the image below.



Within RISA-3D, this idealization will not hold true. The wall panel is a FE mesh that attributes load according to the plate mesh FEM behavior. The load that is getting into the lintel is a true representation of how the wall is actually working. There is still arch action taking place as you can tell if you look at the vertical force contours in the wall panel.



In the image above, the red color is an area of very low axial force in the wall. Thus, you can see that, due to plate distribution of force, there is still arching action taking place. This arching action, however, will not be immune to additional loads added to the wall or the opening being located lower in the wall (as is assumed with the idealized arching action in many texts). Thus, though your loads for lintel design may not be identical to what idealized methods might consider, this is a rational loading for the geometry and loading input on the wall panel.

Meshing the Wall Panels

At solution time, the wall panels will be automatically meshed into quadrilateral plate elements. Unlike the plate elements created directly by the user, the automatically generated plate elements are transient in the program and will not be saved in the input file.

The wall panel meshing is treated similar to analysis results. When the results of an analysis are deleted, the wall panel mesh is cleared to be re-built during the next solution. When a solution results file is saved, the meshed elements will be included in that file.

Mesh Size

The global mesh size for the wall panels can be input on the Solution tab of the Global Parameters. The smaller the mesh size, the more accurate the analysis will be. However, smaller mesh size will also lead to longer solution time and more memory usage. The default mesh size is 12 inches RISA-3D.

Localized small mesh sizes are used in the lintel locations for masonry walls, in order to achieve more accuracy for the lintel forces.

Graphical Display of the Wall Panel Mesh

By default, the plate elements associated with the wall panels are not visible to the user. The mesh can be turned on using the setting on the Panels tab of the Plot Options. The Show Mesh check box will turn the display of the wall panel mesh on or off.

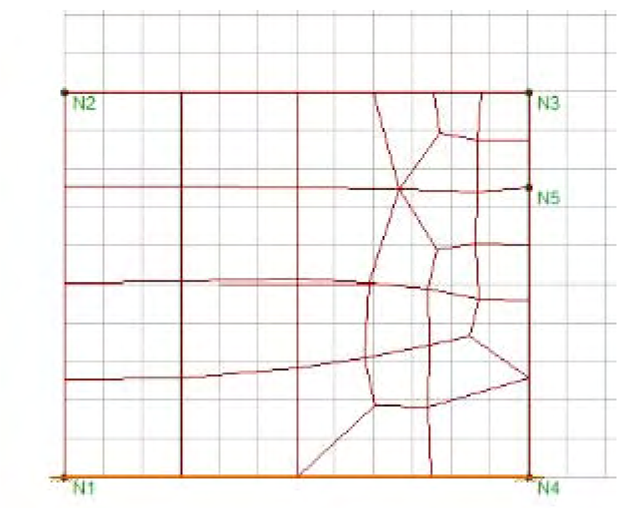
Note

- The display of the mesh is only available when there are active analysis results.

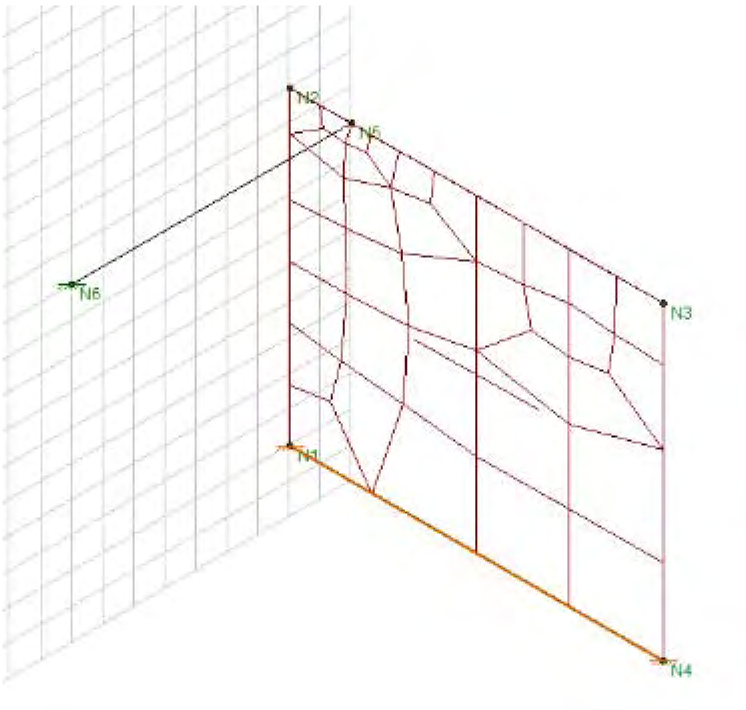
Point Constraints for the Panel Mesh

Point constraints are the locations within the wall panel that require connectivity to the meshed plate elements. The program will automatically generate point constraints at the following locations:

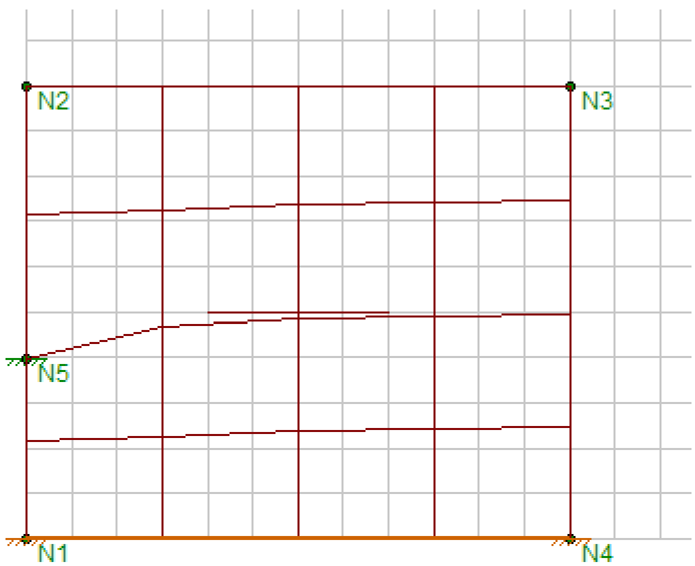
1. Location of an existing joint on the wall panel edges, region boundaries and opening boundaries, as well as within the wall panel.



2. Where beams intersects the wall panels (out-of-plane) on the wall panel edges, region boundaries and opening edges.



3. Location of an external boundary condition.



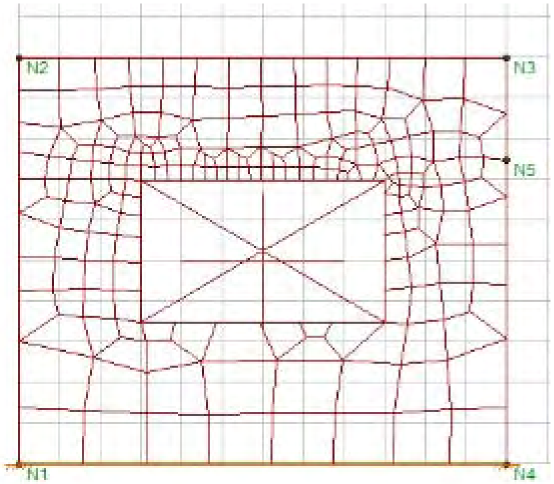
Note:

- Unattached joints that are located on the wall panels can be considered as point constraints and prolong the meshing time. It is highly recommended that the user delete any unattached joints before solving.

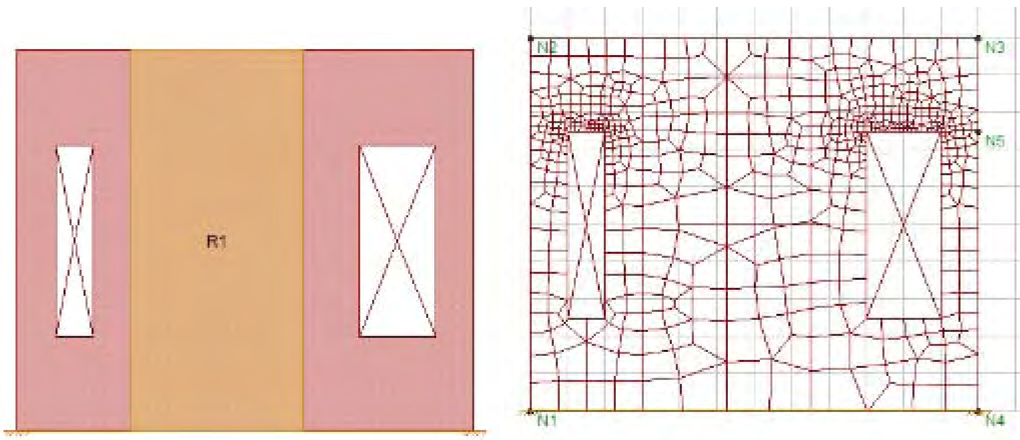
Line Constraints for the Panel Mesh

Line constraints are the locations within the wall panel that require continuous connectivity to plate edges rather than a single point. The program will automatically generate line constraints at the following locations:

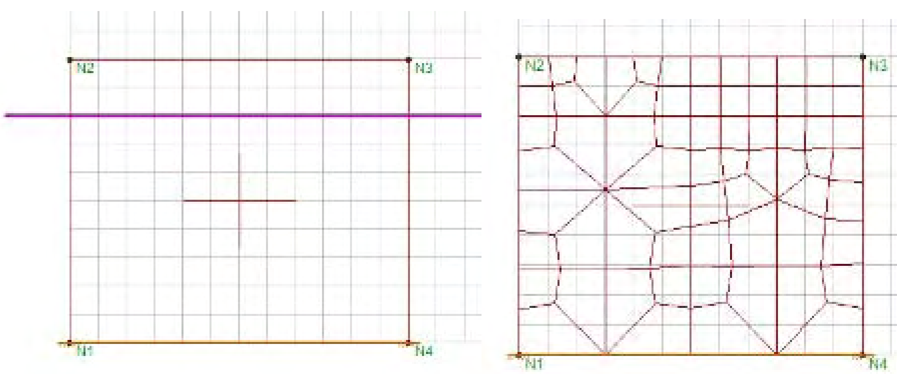
1. Opening edges.



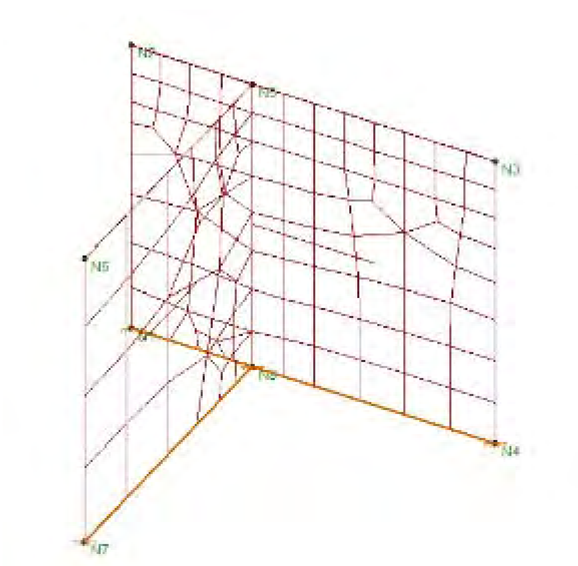
2. The edge and vertical centerline of a defined region.



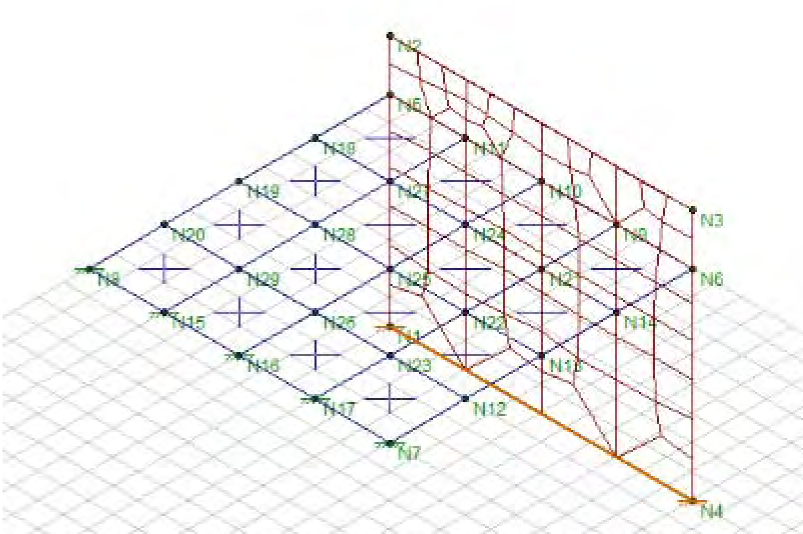
3. Where a diaphragm intersects the wall panel.



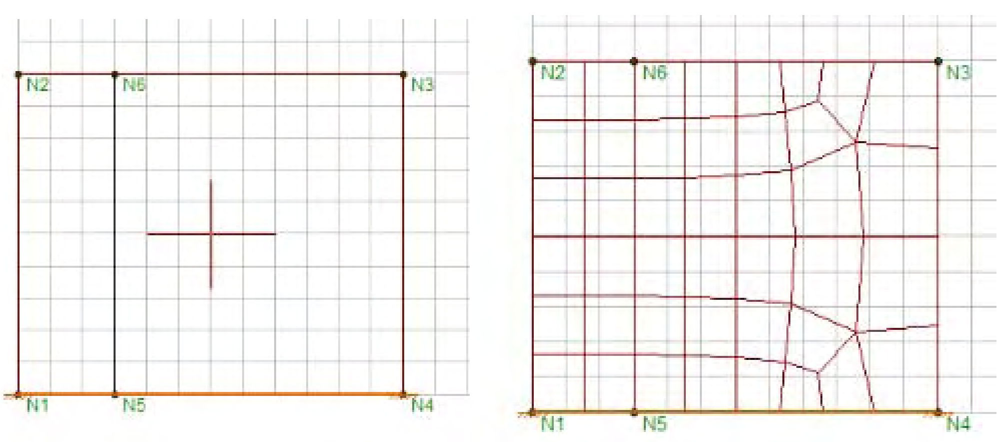
4. At the intersection of multiple wall panels.



5. Where a plate element mesh intersects the wall panel.



6. Where a beam or column element intersects within the plane of the wall panel.



Tips for Ensuring a Healthy Mesh

In order to generate an efficient mesh that gives accurate results, it is critical to place the line constraints and point constraints correctly. If line constraints or point constraints are very close to each other, the auto mesher will be forced to generate small sized elements in order to satisfy the constraints. Therefore, a large number of plate elements will be generated and the solution will be slowed down significantly.

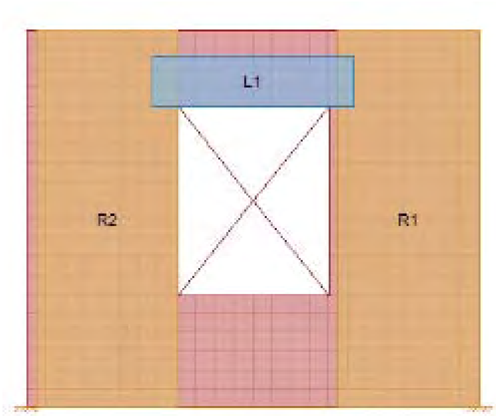
The following guidelines should be followed to ensure a quality mesh:

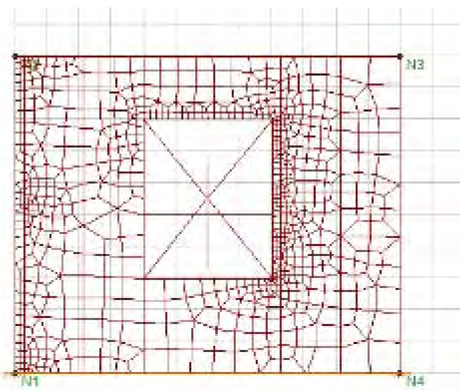
- Avoid generating very narrow regions and openings.
- Avoid small offsets between the external boundary conditions with the location of the region boundaries and wall boundaries.
- Avoid small offsets between opening edges with the region boundaries .
- When a wall panel is intersected by another wall panel, diaphragms, beams or plate elements, keep in mind that the intersection is a line/point constraint. Avoid the small offsets between intersections with the region boundaries or opening edges inside the wall panel.

Example #1: Region Boundaries

1. The left boundary region R1 is placed very close to the opening but not on the opening.
2. The left boundary of region R2 is placed very closed to the left edge of the wall but not exactly on the wall boundary.

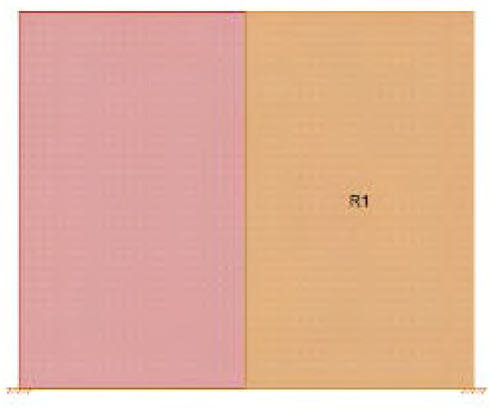
In order to satisfy the line constraints required by the opening edge, region boundaries, and wall boundaries, the program is forced to generate very small meshes in the portion of the wall adjacent to these constraints.

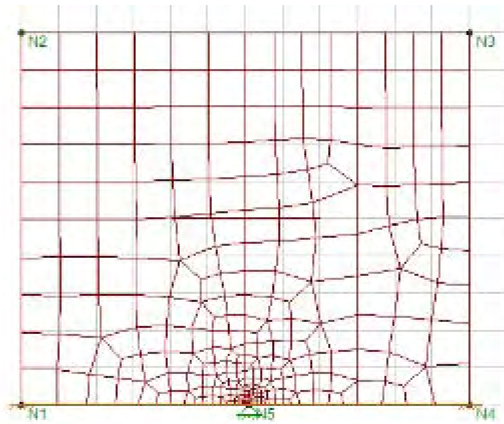




Example #2: Poorly Located Boundary Conditions:

The left boundary of region R1 is at the vertical center line of a wall panel. At the same time, the user placed an external boundary condition at the bottom of the wall panel, which is slightly offset from the center line. In order to accommodate the line constraint of the region boundary and the point constraint of the external boundary condition, the automesher is forced to generate a very small mesh adjacent to these constraints.





Merge Tolerance for Auto-Correction of Mesh

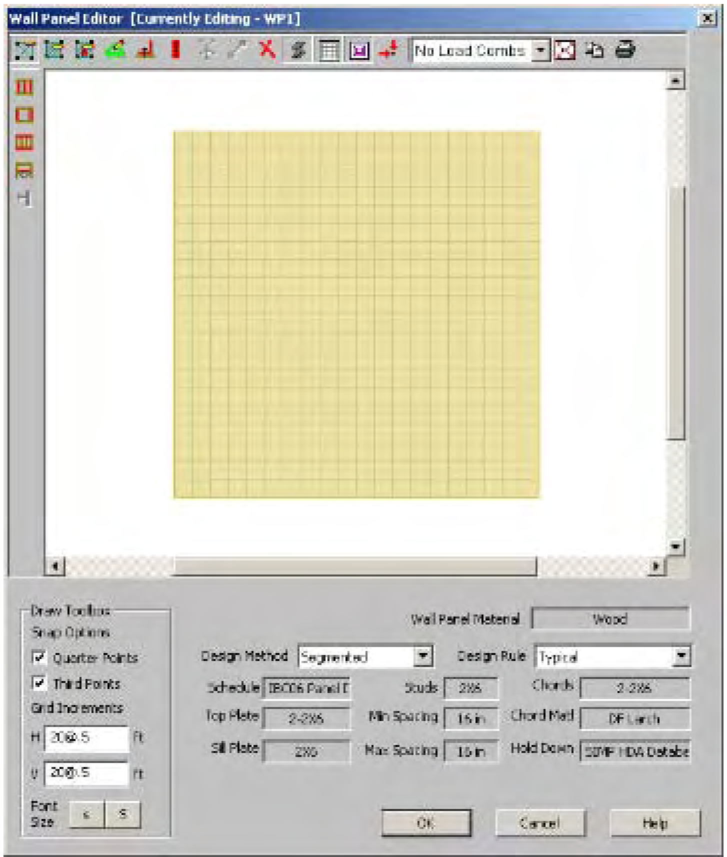
If the distance between the line constraints and point constraints are smaller than the merge tolerance specified on the Global parameters (which defaults to 0.12 inches) then the automesher will automatically snap the constraints together during the meshing. This can eliminate some of the meshing issues that occur in the examples above.

Wood Wall - Design

The wood wall panel element allows you to easily model, analyze and design wood walls for in plane loads. Here we will explain the wood specific inputs and design considerations. For general wall panel information, see the Wall Panels topic. For wood wall results interpretation, see the Wood Wall Results topic.

Wood Wall Input

The **Wall Panel Editor** gives some specific information and options for modeling/analysis of wood walls.



Wood View Controls

 **Toggle Wall Studs Display** allows you to turn the display of the studs on and off.

 **Toggle Wall Chords Display** allows you to turn the display of wall panel region chords on and off.

**Toggle Top/Sill Plate Display** allows you to turn the display of the top/sill plates on and off.

 **Toggle Opening Headers Display** allows you to turn the display of headers on and off.

Creating Openings in Wood Walls with Headers

Within the **Wall Panel Editor**, you have the option of adding rectangular openings to wood wall panels. To draw an opening, select the **Create New Openings**  button and then select two nodes or grid intersections which make up the two diagonal corners of your opening. When an opening is drawn a header beam is automatically created above the opening. To view or

edit the properties of a header beam, double-click inside the boundary of the drawn opening. This will bring up the Editing Properties window for that particular header beam.

Label - This defines the name of this header and shows up in the results output for this header.

Same as Opening - This allows you to define this header with the same properties as another header already defined in this wall panel.

Header Size - This allows you to define the size of the header member. The program will do design checks based on this size.

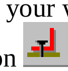
Header Material - This allows you to define the header material.

Trimmer Size - This allows you to define the trimmer size for this opening. This value is only used for the material take off for the wall.

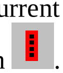
Trimmer Material - This allows you to define the material for the trimmers.

Hold Downs and Straps

Within the **Wall Panel Editor** is where any hold downs and straps must be added to a wood wall panel.

Hold downs represent the anchorage of your wall to the foundation. To add hold downs to the base of your wall, first select the **Add New Wood Hold Downs** button .

Hold downs must be added after regions are created and can only be added at the corners of regions. Hold down requirements depend on the type of wall design you are performing. For a **Segmented** design, you must have hold downs at the bottom corners of each of your design segments. For **Perforated** and **Force Transfer Around Openings (FTAO)**, hold downs are only allowed at the two far corners of the wall panel. The program will not permit the drawing of hold downs at locations where hold downs are not allowed.

Straps represent the anchorage of the current wall panel to a wall panel below. To add straps to the base of your wall, first select the Add New Wood Straps button .

Straps also can only be added after regions are created and can only added at the corners of regions. Strap requirements also follow the same logic as hold downs as to where they must be defined as far as regions are concerned, but there are other restraints that must be followed:

1. Straps can only be drawn on a wall panel that is sitting on top of another wall panel or a column.
2. Straps can only be drawn such that they line up with hold downs from the lower wall or so they line up on a column.

Note:

- All boundary conditions for wall panels should be defined in the wall panel editor. Adding external boundary conditions can create problems.
- Hold downs and straps can only be added in RISA-3D.
- If you have applied your hold downs for the Segmented design with openings in your wall, then hold downs will be required at the interior of the wall panel. However, running Perforated or FTAO does not require hold downs at the interior of the wall panel. Thus, if you toggle between Segmented to Perforated or FTAO, then the hold downs you drew will be removed at the interior of the walls. If you switch back to Segmented the interior hold downs will come back again.

The output for straps and hold downs will show up on the detail report for the wall panel. More information on this can be found in the Wood Wall Results topic.

General Requirements for Shear Walls

The design of wood shear walls within the framework of the NDS requires that many criteria are satisfied before a wall can be considered adequate. For RISA to work within this framework we require that certain modeling practices be followed. Outlined below are many general wall modeling practices and limitations. Also included are specific requirements for each of the three design procedures for wood wall design with openings: segmented, force transfer around openings and perforated.

The three different types of shear walls are defined in Section 4.3 of the NDS Special Design Provisions for Wind and Seismic.

Note:

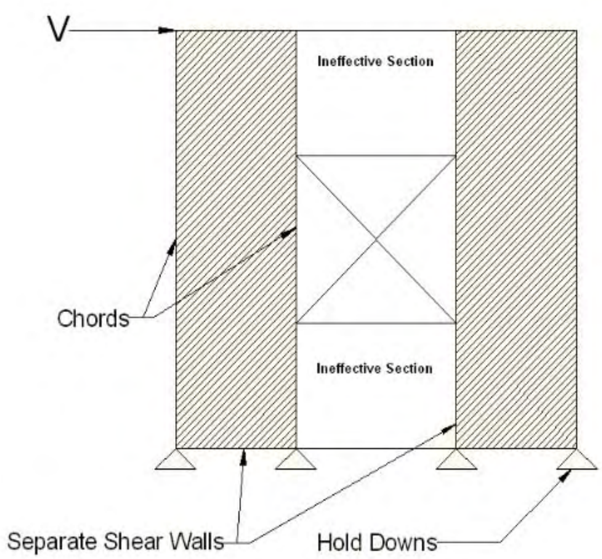
- RISAFloor only considers the Segmented design method. If using RISAFloor together with RISA-3D, simply taking your model into RISA-3D will open up the Perforated and Force Transfer Around Openings design methods.

Segmented Method

Where there is a wall panel with openings, the area above and below the openings is disregarded and the wall is designed as being made up of separate, smaller shear walls.

Like all wall panels, the segmented wood wall is broken into a series of meshed plate elements to represent the overall wall. The portions of the segmented shear wall that are considered "ineffective" in resisting shear are modeled with a plate elements that have a significantly reduced shear stiffness so that they will not receive any significant moment or shear from the FEM analysis.

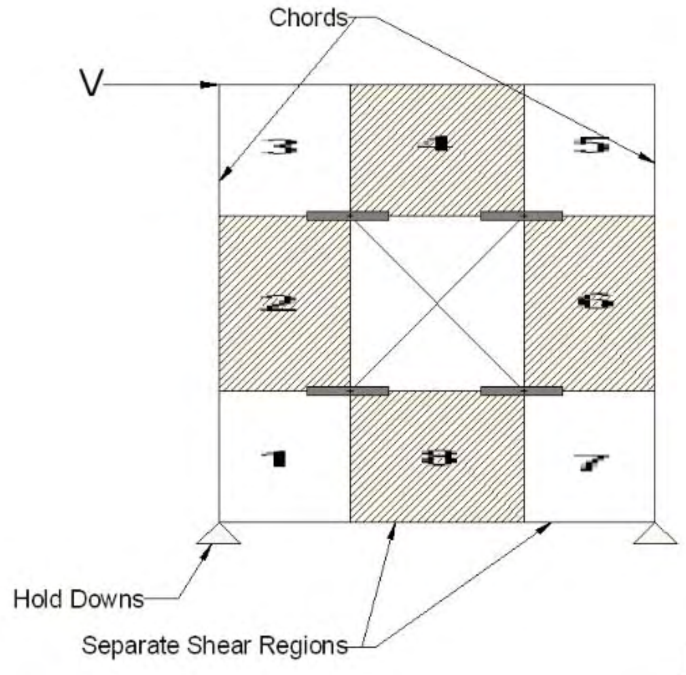
See the diagram below for more information:



In addition, the out of plane stiffness and in plane stiffnesses of the segmented wood wall are modeled separately based on different assumed plate thicknesses. This is done to insure that the shear stiffness is based entirely on the properties of the sheathing and is not influenced by the out-of-plane stiffness of the wall studs.

Force Transfer Around Openings Method

This method is based on a rational analysis of the wall. The assumption being that straps and blocking can added at the corners of the openings to transfer the sheathing forces across these joints. This method essentially allows you to use the entire area of the wall (minus the opening) to resist the shear in the wall.



The basic assumptions made in the shear wall analysis are the following:

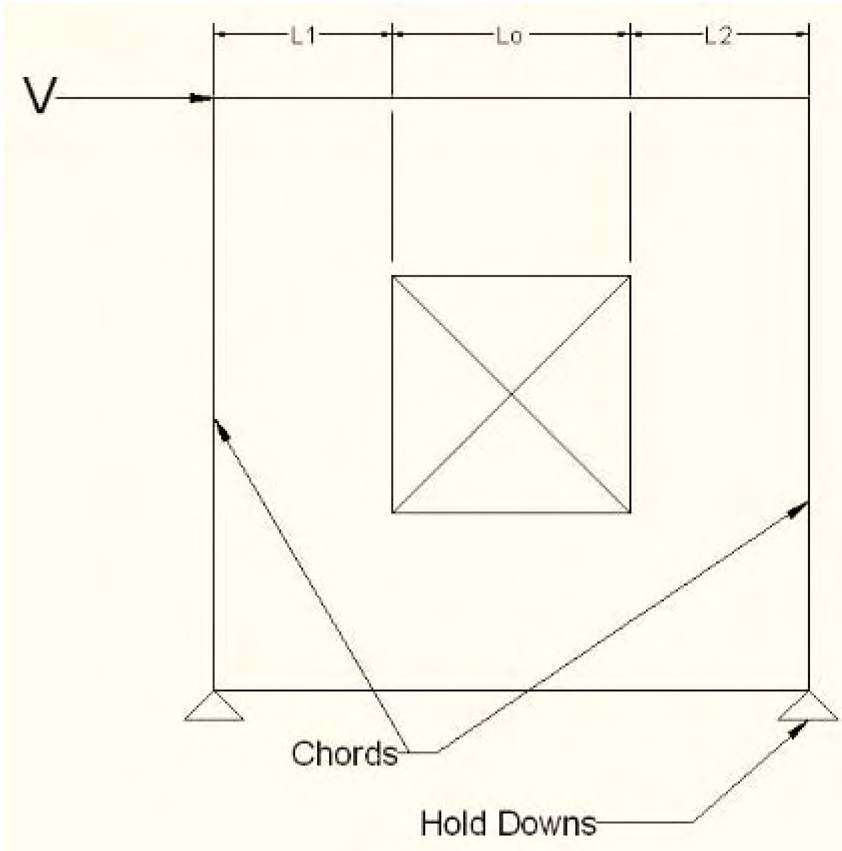
- The sheathing resisting the shear forces. The average shear force in each block of the wall (numbered 1~8 as shown in the image above) is used as the controlling shear force in that location. The maximum shear in each of these locations will control the design of the wall. The program uses an area weighted average of the Fxy plate forces to determine the average shear for each block.
- The moment at edge of each block that is above or below an opening is assumed to be transmitted across the opening interface by horizontal tension straps or compression blocks as shown in the image above. The required force is reported to the user, but the design and length of these elements is left to the engineer.
- The moment at the edge of each block that is to the right or left of an opening is assumed to be transmitted across the opening by tension straps or compression blocks. Since it is likely that the sheathing and king studs will be capable of transmitting these forces, these elements are not shown in the image above. However, these forces are reported so that the design of the studs and sheathing in these regions can be checked by the engineer to consider these effects.

Note:

- The program is limited in the automatic generation of regions for walls with multiple openings that are not aligned. Therefore, it is recommended that complex walls will multiple openings be simplified based on engineering judgement to facilitate easier detailing of the force transfer around these openings.

Perforated Method

The third method for design of wood shear walls with openings may end up being the most cost effective. It only requires hold downs only at the corners of the wall, yet it does not require straps or blocking around the openings. This Perforated shear wall design approach is, however, subject to number code constraints about when it can be used.



The basic design procedure for perforated walls consists, essentially, of ignoring the portions of the wall that do not have full height sheathing and treating the wall instead as a significantly shorter wall. This amplifies the chord and hold down design forces significantly while at the same time increasing the design unit shear as shown in the equations below:

$$T := \frac{V \cdot h}{C_o \cdot \Sigma L_i} \qquad \upsilon_{max} := \frac{V}{C_o \cdot \Sigma L_i}$$

Where:

$$\Sigma L_i := L1 + L2$$

Note:

- The perforated method of design also has many caveats that are given in Section 4.3.5.3 of the NDS 2005 Special Design Provisions for Wind & Seismic. The program will not allow the design of wall panels that do not follow these provisions.

- For multi-story perforated shear walls, the amplified hold down forces required per the code become difficult to interpret for the lower walls. Therefore, RISA when calculating the hold down forces for the lower wall it is assumed that the reduction coefficients for the upper wall are identical to the values for the lower wall. This will result in conservative hold down forces when the lower wall has more openings than the upper wall. But, it may be un-conservative for situations where this is not the case. This assumption does NOT affect the shear design of either wall, nor does it affect the strap force calculations in the upper wall.

Shear Capacity Adjustment Factor, Co:

The NDS Special Design Provisions for Wind & Seismic lists Effective Shear Capacity Ratio (Co) values that are used in calculating the nominal shear capacity of perforated shear walls. Because the tabular values are limited to wall heights of 8’ & 10’, RISA instead uses the equation (from the 2008 NDS SDPWS) to calculate the Co factor for any height wall.

$$Co = \left(\frac{r}{3 - (2 * r)}\right) * \frac{Ltot}{\Sigma Li}$$

Where,

$$r = \frac{1}{1 + \left(\frac{Ao}{h * \Sigma Li}\right)}$$

Note:

- When using these equations RISA makes the assumption that all openings are equal to the maximum opening height.
- When the assumption is made that all opening heights are equal to the maximum opening height then the equation produce values of Co that are within 1% for all the values shown in the NDS table.

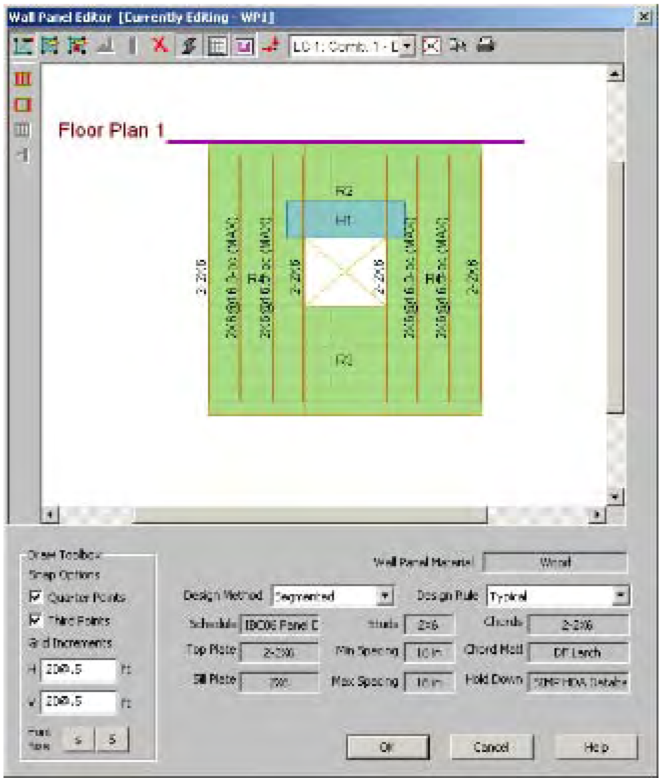
General Program Functionality and Limitations

RISAFloor and RISA-3D Interaction

When using RISAFloor and RISA-3D in combination, the interface transitions nicely between the two programs. Here is a quick walk through of this interaction.

Input Interface

1. Model the entire building (gravity and lateral members) within RISAFloor. Be sure to model all openings and regions for all of the wall panels in the model.



Note:

- You can not modify your openings or regions in RISA-3D. All region and opening modifications must be taken back to RISAFloor to be done.

2. Add loading and solve the model.
3. Take the model into RISA-3D via the Director tool.
4. Once in RISA-3D you must add your hold downs and straps to your wall panels.

Note:

- Hold downs and straps can not be added to wall panels in RISAFloor.
- Hold downs are only allowed to be added to the lower corners of the wall panel for the Perforated or Force Transfer Around Openings design methods.
- Hold downs are required at the corners of all full height regions in the wall panel for the Segmented design method.
- The Design Rule, Design Method and SSRF can be changed in either program at any time.

Results Viewing

Below are some general guidelines when reviewing results in a combined RISAFloor-RISA-3D model.

- Only wall panels designed as Segmented will be designed in RISAFloor. Perforated and Force Transfer Around Openings will give no design values in RISAFloor and must be taken to RISA-3D for design.

Limitations for Hold-downs / Straps (Including deformation)

- The program does not make an adjustment to the chord force calculation based on the eccentricities of the chord and hold-down. The program considers the hold down and the chord to be at the same location in plan.

- The program does not currently have a database for continuous tie rod hold-downs. This will likely be added in a future release. However, the design results that are presented for each floor are intended to provide the type of information necessary for the design of these types of hold downs.
- The $d_a$values (deflection at peak load) from our hold-down database are based entirely on the manufacturers listed values. These do not include any allowance for shrinkage (which is often as much as 1/4" or more per floor), or crushing of the sill plate.
- There is not currently a d_a value used in the calculation for strap deformation for upper floor levels. This will be implemented when a strap database is added into the program.

Automatic Boundary Conditions

- In RISA-3D, if no boundary conditions/hold downs are defined for wood wall panels at the lowest level of the structure, the program will automatically create hold downs at the corners of the wall panel. If you do not want the hold downs to be automatically created, define a "free" boundary condition at the base of the wall panel in the wall panel editor.

Deflection

- There is currently no code check for drift or deflection for shear wall panels.

Chord Design

- RISA-3D models the shear walls using only the sheathing. The vertical resistance occurs only at the tension and compression chords. Thus, if two wall panels are stacked on top of each other, the load transfer will only happen at the chord locations. Therefore, the lateral analysis should agree very well with hand calculations. However, it also means that gravity load design may be more appropriate in RISAFloor.

Stud Design

- Stud design is accomplished by taking the entire axial load in the wall panel and dividing it by the number of studs (based on spacing). This force is then checked against the allowable to give a code check. If you are optimizing for spacing, the program will start with the largest spacing and work its way down until finding a spacing that works.

Force Distribution

- The lateral force distribution between piers is based on the relative stiffness of the sheathing, not on the length of the shear wall. For example, if you have an 8 foot wall and a 4 foot wall, the 8 foot wall will take more than 8/(8+4)*100% of the force. The moment of inertia in the 8 foot wall will allow for a larger proportion of load to go into that pier.

Shear Stiffness Adjustment Factor

- The stiffness adjustment factor for wall panels is intended to allow the user to adjust the stiffness of his walls so that they match the APA and NDS formula for deflection calculations. This adjustment affects the stiffness of the entire wall. Therefore, for segmented walls the engineer may be forced to model the piers separately if they need to adjust the pier stiffnesses independently. In a future release, this may become an automated factor.

Stiffness Assumptions

- RISA uses an orthotropic plate element to de-couple the vertical and shear stiffness of the wood walls. The vertical stiffness will be based on the E value of the studs and chords. Whereas the shear stiffness will be based on the Ga value designated within the specified nailing schedule.

Note:

- Currently the thickness used, for both the vertical and shear stiffnesses, is based on the minimum thickness for the wall panel nailing schedule.

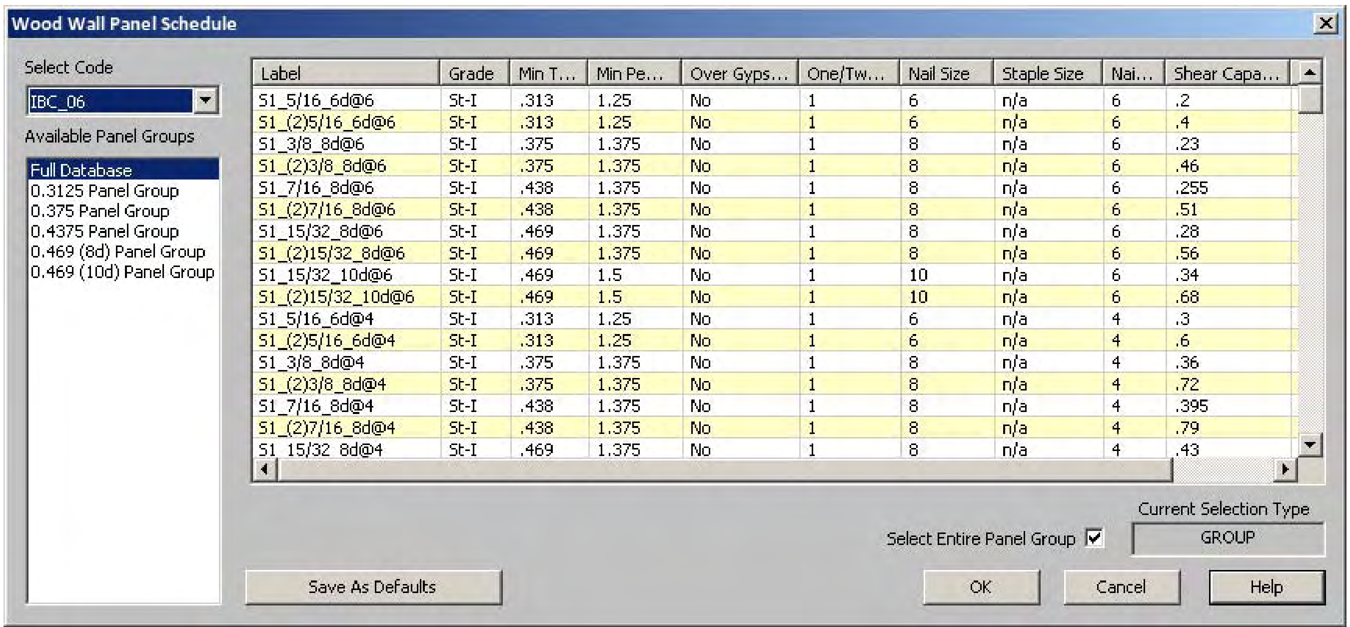
Wood Wall Self Weight

The program will calculate wood wall self weight as a sum of all the weights of the components. The material density is used to calculate the self weight of the studs, chords, top plates, sill plate, and sheathing. These are all then summed together to give the self weight of the entire wall.

Note:

- For this calculation, stud height equals wall height minus the thickness of the sill plate and the top plate.
- The number of studs is calculated using the stud spacing specified in Design Rules.

Panel Optimization

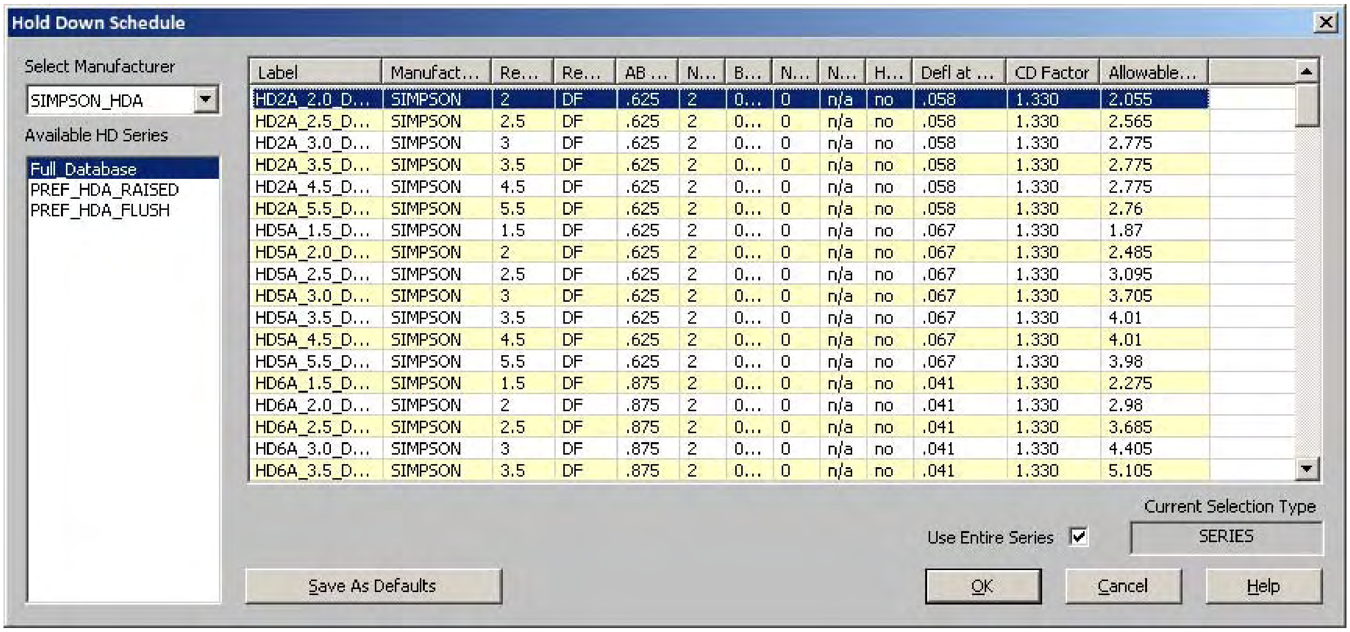


The procedure that RISA uses for design optimization is fundamentally based upon the assumption that there is a 'cost' to shear capacity, and therefore the ideal panel design would have as little shear capacity as possible to meet code requirements. Once the program has determined the shear demand on the wall it will choose the most economical panel configuration based on that which has a **Shear Capacity** closest to, but not exceeding the shear demand.

Note:

- The shear force listed in the XML spreadsheet is currently only tabulated for the seismic values. A future update will use the "wind increase factor" in the database to increase the allowable values whenever a wind load combination is being solved.

Hold Down Optimization



Label	Manufact...	Re...	Re...	AB ...	N...	B...	N...	N...	H...	Defl at ...	CD Factor	Allowable...
HD2A_2.0_D...	SIMPSON	2	DF	.625	2	0...	0	n/a	no	.058	1.330	2.055
HD2A_2.5_D...	SIMPSON	2.5	DF	.625	2	0...	0	n/a	no	.058	1.330	2.565
HD2A_3.0_D...	SIMPSON	3	DF	.625	2	0...	0	n/a	no	.058	1.330	2.775
HD2A_3.5_D...	SIMPSON	3.5	DF	.625	2	0...	0	n/a	no	.058	1.330	2.775
HD2A_4.5_D...	SIMPSON	4.5	DF	.625	2	0...	0	n/a	no	.058	1.330	2.775
HD2A_5.5_D...	SIMPSON	5.5	DF	.625	2	0...	0	n/a	no	.058	1.330	2.76
HD5A_1.5_D...	SIMPSON	1.5	DF	.625	2	0...	0	n/a	no	.067	1.330	1.87
HD5A_2.0_D...	SIMPSON	2	DF	.625	2	0...	0	n/a	no	.067	1.330	2.485
HD5A_2.5_D...	SIMPSON	2.5	DF	.625	2	0...	0	n/a	no	.067	1.330	3.095
HD5A_3.0_D...	SIMPSON	3	DF	.625	2	0...	0	n/a	no	.067	1.330	3.705
HD5A_3.5_D...	SIMPSON	3.5	DF	.625	2	0...	0	n/a	no	.067	1.330	4.01
HD5A_4.5_D...	SIMPSON	4.5	DF	.625	2	0...	0	n/a	no	.067	1.330	4.01
HD5A_5.5_D...	SIMPSON	5.5	DF	.625	2	0...	0	n/a	no	.067	1.330	3.98
HD6A_1.5_D...	SIMPSON	1.5	DF	.875	2	0...	0	n/a	no	.041	1.330	2.275
HD6A_2.0_D...	SIMPSON	2	DF	.875	2	0...	0	n/a	no	.041	1.330	2.98
HD6A_2.5_D...	SIMPSON	2.5	DF	.875	2	0...	0	n/a	no	.041	1.330	3.685
HD6A_3.0_D...	SIMPSON	3	DF	.875	2	0...	0	n/a	no	.041	1.330	4.405
HD6A_3.5_D...	SIMPSON	3.5	DF	.875	2	0...	0	n/a	no	.041	1.330	5.105

The procedure that RISA uses for hold-down optimization is fundamentally based upon the assumption that there is a 'cost' to allowable tension in a product, and therefore the ideal hold-down would have as little tensile capacity as possible to meet code requirements. Once the program has determined the tensile force required to hold-down the wall it will choose the most economical hold-down product based on that which most closely matches (but does not exceed) the tension demand. The program looks to the **Allowable Tension** field of the hold-down schedule to choose the design.

Optimization Procedure

For users who are new to wood wall design within RISA, the best procedure is to utilize the full databases, and to limit the potential designs by utilizing the design rules spreadsheets. This results in a design based on the maximum number of options, which is often the most efficient design.

For experienced users who have more specific limitations in terms of the designs they would like to see, user-defined **Groups** (or families) are the solution. For example, an engineer who prefers to use only one sheathing thickness, or one nail type can create a custom Group that contains only the arrangements they want. For more information on creating these custom groups see Appendix F-Wood Shear Wall Files.

For more information on Wood Walls see Wood Wall Results.

Wall Panels - Results

When the model is solved there is a results spreadsheet specifically for Wall Panel Design. This spreadsheet is divided into different tabs: In plane, Out of Plane, Lintel, Wood Wall Axial and Wood Wall In-Plane. Each tab gives code checks based on the relevant code depending on the material type. These spreadsheets can be used as a summary of all of the panels in your model. To get detailed information about each panel, you can see the Wall Panel Detail Reports.

Wood Wall Spreadsheet Results

- Axial Results
- In Plane / Shear Results

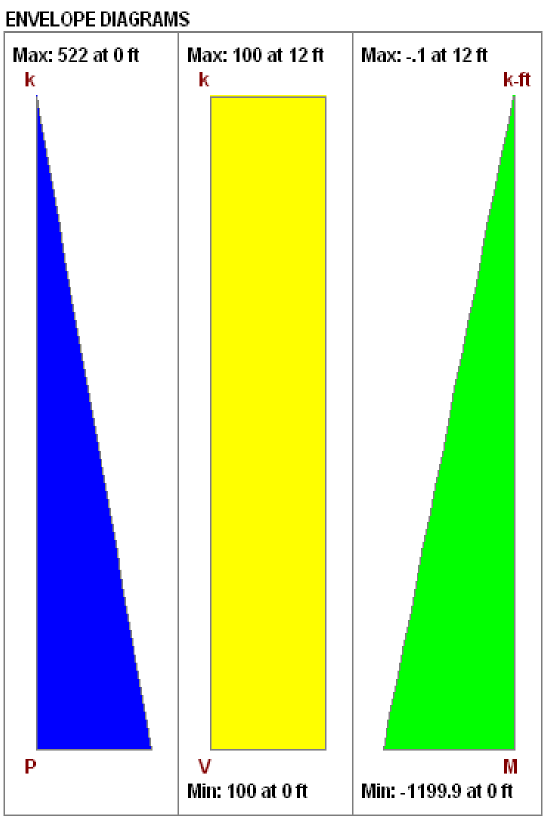
Wood Wall Detail Reports

- In-Plane or Shear Wall Detail Report

General Wall Detail Report

The general wall detail report shows the material type, height, length, and envelope forces for general walls. There is currently no concrete reinforcement design, thus general walls provide an easy way to model concrete walls for analysis.

Material		GEOMETRY		
Wall Material	: gen_Conc3NW	Total Height	: 12	ft
		Total Length	: 30	ft



Note

- No detail report is generated for general walls with openings when there is no region defined.

The **Wall Panel Results Spreadsheet** displays the calculated results for wall elements and may be accessed by selecting **Wall Panel Design** on the **Results** menu. The spreadsheet has five tabs: **In Plane, Out of Plane, Lintel, Wood Wall Axial, and Wood Wall In-Plane.**

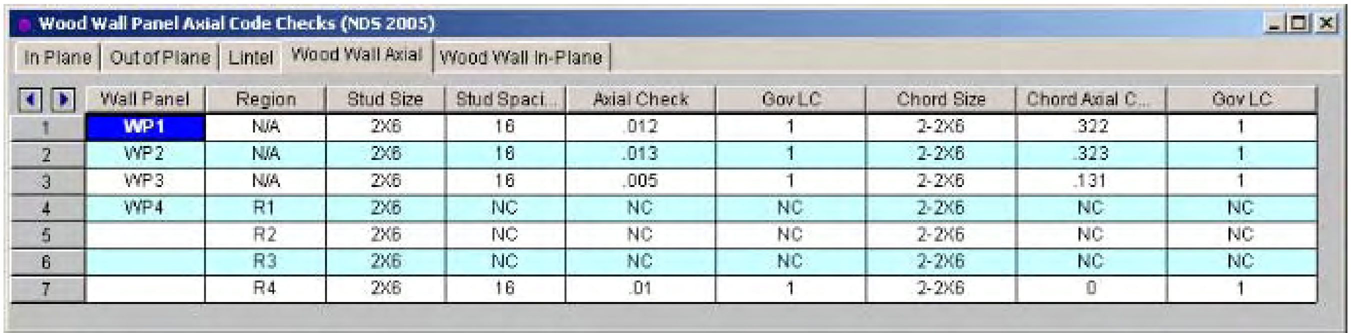
The first three tabs are results of **Masonry Wall** analysis. For more information on these tabs see Masonry Wall Results.

The last two tabs are results of **Wood Wall** analysis. For more information on these tabs see Wood Wall Results.

Wood Wall Results

Wood Wall results are presented in the Wall Panel Design Spreadsheet and the detail reports.

Wood Wall Results Spreadsheets


Wood Wall Panel Axial Code Checks (NDS 2005)

In Plane | Out of Plane | Lintel | Wood Wall Axial | Wood Wall In-Plane

	Wall Panel	Region	Stud Size	Stud Spaci...	Axial Check	Gov LC	Chord Size	Chord Axial C...	Gov LC
1	WP1	N/A	2X6	16	.012	1	2-2X6	.322	1
2	WP2	N/A	2X6	16	.013	1	2-2X6	.323	1
3	WP3	N/A	2X6	16	.005	1	2-2X6	.131	1
4	WP4	R1	2X6	NC	NC	NC	2-2X6	NC	NC
5		R2	2X6	NC	NC	NC	2-2X6	NC	NC
6		R3	2X6	NC	NC	NC	2-2X6	NC	NC
7		R4	2X6	16	.01	1	2-2X6	0	1

The **Wall Panel** column lists the wood wall panels that you have defined.

The **Region** column lists the wall panel region that the results are based on.

Note:

- For Perforated or Force Transfer Around Openings design methods, regions are not used. Thus, an N/A will be displayed.

The **Stud Size and Spacing** columns show the optimum stud size and spacing chosen for your wall, based on the Design Rules you have defined.

The **Axial Check** value is a code check ratio between the member load and the member capacity. The adjacent **Gov LC** column shows the governing load case for the design.

The **Chord Size** column shows the optimum chord size chosen for your wall, based on the Design Rules you have defined. Note that the chords are the vertical hold-down members/posts at the both ends of the wall.

The **Chord Axial Check** value is a code check ratio between the member load and the member capacity. The adjacent **Gov LC** column shows the governing load case for the design.

Note:

- When running the Segmented design method, the wall panel regions above and below the opening are not considered in design. Thus an NC (no calculation) will be displayed.
- If there are some constraints that will not allow a wall to be designed, an NC (no calculation) will be displayed. Check the Warning Log within the program for more information on this.

Wood Wall Panel In Plane Code Checks (NDS 2005)

In Plane | Out of Plane | Lintel | Wood Wall Axial | Wood Wall In-Plane

	Wall Panel	Shear Panel Label	Region	Shear Che...	Gov LC	Hold-Down Label	Tension C...	Gov LC
1	WP1	S1_(2)15/32_10d@3	N/A	1.113	1	HD10A_1.5_DF_F	.994	1
2	WP2	S1_(2)15/32_10d@3	N/A	1.544	1	HD10A_1.5_DF_F	.998	1
3	WP3	S1_(2)15/32_10d@3	N/A	1.268	1	HD5A_1.5_HF_F	.993	1
4	WP4	NC	R1	NC	NC	NC	NC	NC
5		NC	R2	NC	NC	NC	NC	NC
6		NC	R3	NC	NC	NC	NC	NC
7			R4	.574	1	Default	0	1

The **Shear Panel Label** shows the optimum shear panel arrangement chosen for your wall, based on the Design Rules you have defined.

The **Region** gives the region for which the design values are being displayed.

Note:

- The Perforated and Force Transfer Around Openings methods do not consider regions in their design, thus N/A is displayed.

The **Shear Check** value is a code check ratio between the panel shear load and the panel shear capacity. The adjacent **Gov LC** column shows the governing load case for the design.

The **Hold-Down Label** shows the optimum hold-down product chosen for your wall, based on the Design Rules you have defined.

The **Tension Check** value is a code check ratio between the tension load and the hold-down tensile capacity. The adjacent **Gov LC** column shows the governing load case for the design.

Note:

- When running the Segmented design method, the wall panel regions above and below the opening are not considered in design. Thus an NC (no calculation) will be displayed.
- If there are some constraints that will not allow a wall to be designed, an NC (no calculation) will be displayed. Check the Warning Log within the program for more information on this.
- If 'default' is shown in the Hold-Down Label column it means that a hold down is not required for this wall/region.

Wood Wall Self Weight

The program will calculate the self weight of a wood wall based on the weights of the individual components. Using the material density, the self weight is calculated for the studs, chords, top plates, sill plates, and sheathing. These are all then summed for the total self weight of the wall.

Wood Wall Detail Reports

The detail report gives detailed information about the wall design. The detail reports are specifically molded to the type of design specified. Here we will walk through how to access the different information for each of the types of design: Segmented, Perforated and Force Transfer Around Openings.

Note:

- RISAFloor only considers the Segmented design method. If using RISAFloor together with RISA-3D, simply taking your model into RISA-3D will open up the Perforated and Force Transfer Around Openings design methods.

- Many of the values for design checks seen below are not performed in RISAFloor as it is strictly a gravity design program.

Accessing the Detail Reports and the Specific Windows

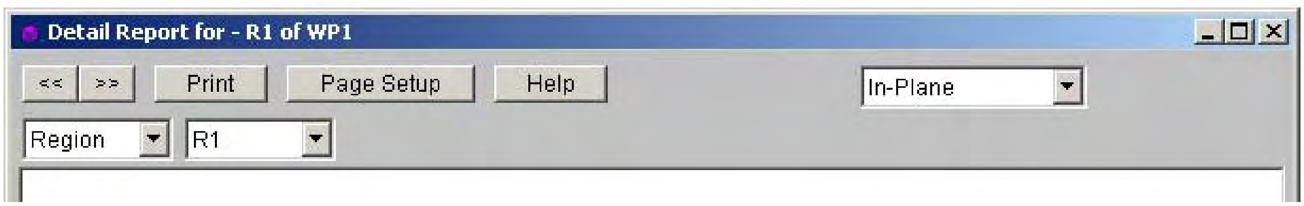
Once you have a solved model, the detail reports become available. They are accessible in two ways:

1. If you have the Wall Panel Design spreadsheet open, there will be a button at the top of the screen: Detail Report for Current Item. This will open up the detail report window.
2. If you are in a graphic view of your model, there is a Detail button that will open up the detail report window.

Note:

- Detail report information is not available for an envelope solution.

Once the detail report window is open, you will see a dialog area at the top.



- 

 - Allows you to click between the different wall panels in your model.
- 

 - This dropdown list allows you to select between the three different parts of the wood wall panel detail report. Below we will explain the importance of each of these sections.
- 

 - This dropdown list allows you to select between different regions or headers defined within the individual wall panel.

The importance of the Opening, Region and Wall detail report sections depends on the type of design you are doing: Segmented, Perforated and Force Transfer Around Openings. Here we will discuss each design method.

Segmented Method

The segmented design method uses each of the three detail report windows to give design information.

Opening Window

This window defines the design of the header beams across the top of the openings in the wall. The different openings can be chosen from the header drop down list. This report is very similar to a wood member detail report. At the top of the detail report the Criteria, Geometry and Materials section give the input parameters defined for the opening. The Envelope Diagrams and the Design Details are given, which provide code check information and give the information required to verify the program values.

Region

The window gives information for your wall on a region basis. Note that only full-height regions of the wall panel will have a region detail report. The segmented method only considers these full height segments in the design of the wall.

The region detail report is split into four portions: input echo, diagrams and design, design details, and cross section detailing. Note that in RISAFloor the detail reports are less detailed because RISAFloor does not consider lateral forces which RISA-3D does.

Input Echo

Below is the input echo portion of the detail report.

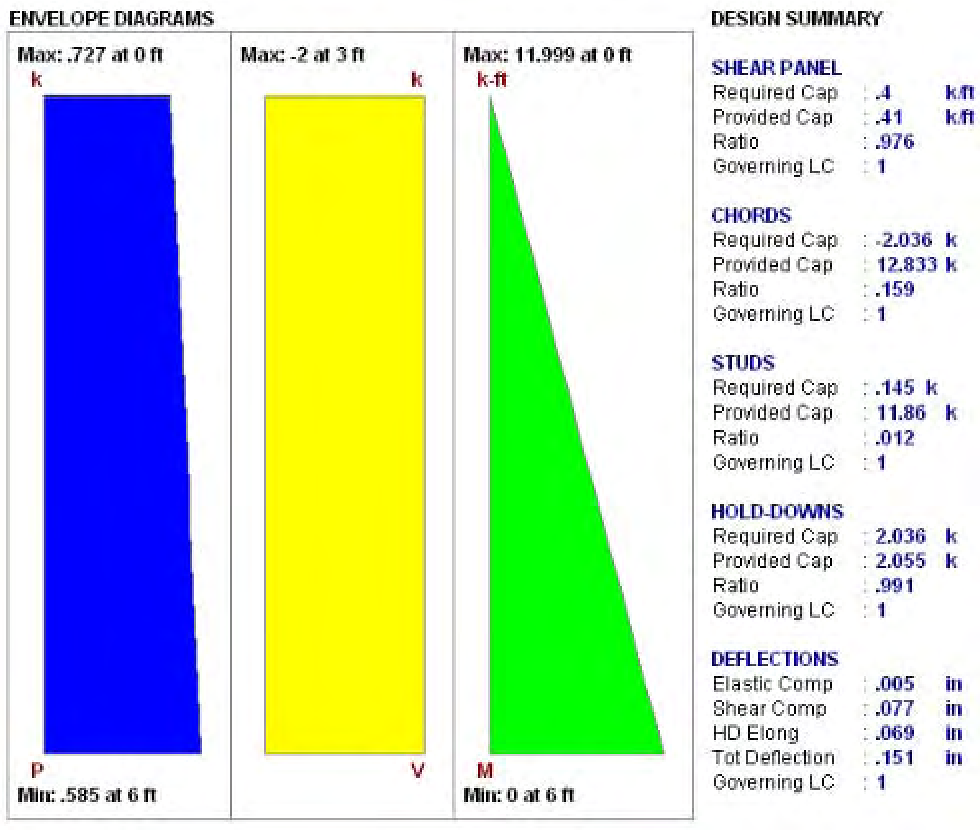
CRITERIA		MATERIALS		GEOMETRY	
Code	: NDS 2005	Wall Studs	: DF Larch	Total Height	: 10 ft
Type of Design	: ASD	Stud Size	: 2X6	Total Length	: 20 ft
Wall Material	: DF Larch	Chord Material	: DF Larch	Wall H/W Ratio	: 0.50
Panel Schedule	: Full Database	Chord Size	: 2-2X6		
Optimize HD	: Yes	Top Pl & Sill	: DF Larch	Stud Spacing	: 16 in
HD Manufacturer	: SIMPSON	Top Pl Size	: 2-2X6		
		Sill Pl Size	: 2X6		

Criteria	Description
Code	Gives the code used to design your wall panel.
Type of Design	Specifies which design method was used (ASD, Strength, etc)
Wall Material	Specifies the wood type assigned to the entire wall
Panel Schedule	Specifies the sheathing/nailing schedule database used to optimize panel selection (set in Design Rules)
Optimize HD	Shows whether or not the program needed to optimize the hold-down, or if the user explicitly defined a hold down
HD Manufacturer	Specifies the manufacturer of chosen hold-down

Materials	Description
Wall Studs	Specifies the wood material type assigned to the wall studs
Stud Size	Specifies the member size used for the wall studs
Chord Material	Specifies the wood material type assigned to the chords (vertical members at both ends of the wall)
Chord Size	Specifies the member size used for the chords (vertical members at both ends of the wall)
Top Plate & Sill	Specifies the wood material type assigned to the top and sill plates
Top Plate Size	Specifies the member size used for the top plate
Sill Plate Size	Specifies the member size used for the sill plate

Geometry	Description
Total Height	This is the height of the wall panel region
Total Length	This is the length of the wall panel region
Wall H/W Ratio	This is the ratio of wall height to length, using the minimum wall height
Stud Spacing	This is the optimized stud spacing based on your Design Rules

Diagrams and Design



Envelope Diagrams

These diagrams show the axial, in-plane shear, and in-plane moments of the wall, as well as the maximum and minimum forces and their locations.

Design Summary

This portion gives you the capacity and strength values at the section in the wall where the combined check is maximum, as well as the governing load combination. Much of this information is also reported in the Wood Wall Panel Design spreadsheets.

Shear Panel

The provided capacity of the shear panel is taken from the **shear capacity** column of the panel database. This is the allowable shear value from Table 2306.4.1 from the 2006 IBC.

Note:

- Shear Panel designations ending in _W indicate shear panels with a 40% increase in shear capacity for wind loads per section 2306.4.1 of 2006 IBC.

Chords, Studs

The provided capacities of these members are calculated using the standard provisions for tension/compression members. These members are assumed to be fully braced in the weak axis, and unbraced in the strong axis.

Hold-Downs

The provided capacity of the hold-down is taken from the **Allowable Tension** column of the hold-down database. This is information supplied by the manufacturer.

Deflections

The deflection listed in the detail report is based on an approximation from the NDS 2005 Special Design Provisions for Wind and Seismic, Equation 4.3-1:

$$\delta_{sw} = \frac{8vh^3}{EAb} + \frac{vh}{1000G_a} + \frac{h\Delta_a}{b}$$

b =Shear wall length, ft

Δ_a = Total vertical elongation of wall anchorage system (including fastener slip, device elongation, rod elongation, etc.) at the induced unit shear in the shear wall, in (This value is taken from the hold down database and scaled per the actual tension force; hence you multiply this value by the holddown ratio given in the output)

E = Modulus of elasticity of end posts (chords), psi

A = Area of end post (chord) cross-section, in^2

G_a = Apparent shear wall shear stiffness from nail slip and panel shear deformation, kips/in. (taken from shear panel database)

h = Shear wall height, ft

ν = Induced unit shear, lbs/ft

δ_{sw} = Total shear wall deflection determined by elastic analysis, in

=

The first term of the above equation determines the Bending Component of the deflection.

The second term of the above equation determines the Shear Component of the deflection.

The third term of the above equation determines the Hold-Down Elongation, which causes additional deflection.

Note

- This is the theoretical deflection of the wall. This may differ from the deflection of the wall as performed by finite element analysis within RISA. Therefore, this deflection value may not coincide with the reported deflection value in the deflections spreadsheets.

Design Details

DESIGN DETAILS

SELECTED SHEAR PANEL : **S1_(2)15/32_10d@3_W**

Panel Grade	: St-I		Nail Size	: 10d		Num Sides	: Two	
Panel Thick	: 0.469	in	Reqd Pen	: 1.500	in	Over Gyp Brd.	: No	
			Reqd. Spacing	: 3	in	Shear Capacity	: 1.860	k/ft

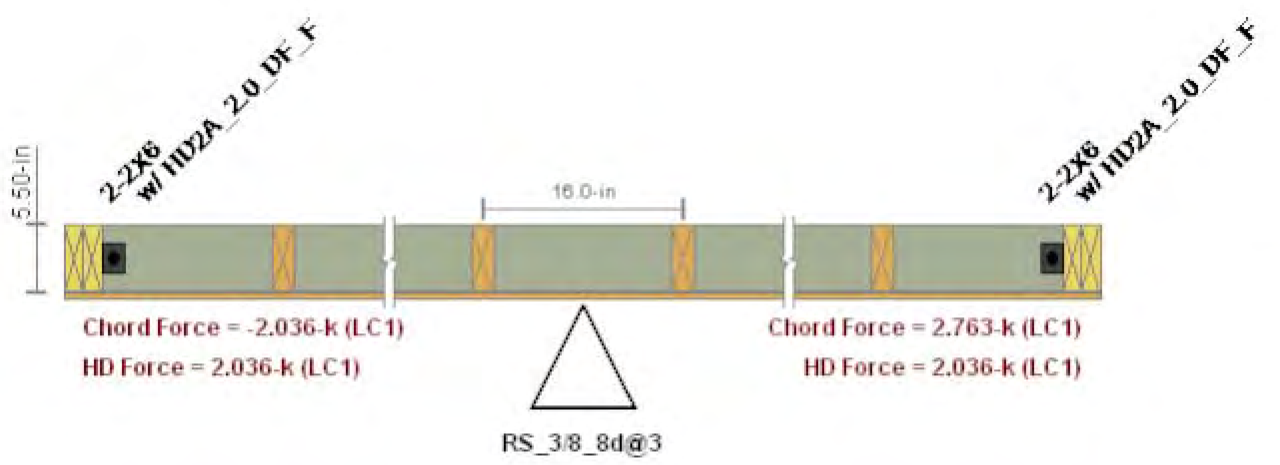
NOTE: defines a 10d nail as being **3.0" x 0.1480" common, or**
3.0" x 0.122" galvanized box

SELECTED HOLD-DOWN : **HD2A_1.5_HF_F**

Raised	: No		Bolt Size	: 0.625	in	Reqd Chord Thk	: 1.50	in
AB Diameter	: 0.625	in	Num Bolts	: 2		Reqd Chord Mat	: Hem Fir	
						Allow Tension	: 1.320	k

The above section of the report echoes the database information for the selected shear panel and hold-down. For more information on these properties refer to Appendix F-Wood Shear Wall Files.

Cross Section Detailing



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. The wall thickness, and stud spacing are shown as dimensions. The triangle shows sheathing on one side of the wall, with the abbreviated panel designation. The chord sizes/forces and hold down designations/forces are shown at either end. If either chord is only experiencing a compression force, the hold down will not be drawn.

Note:

- The displayed chord force is not necessarily the force used in the hold-down design because hold-down optimization only considers the governing tension LC.

Perforated Method

Opening Window

This window is identical to the Segmented Window information.

Wall Window

This is where the majority of the information is located for the perforated method.

GENERAL

Code	:
Design Method	: Perforated
Wall Material	: DF Larch
Panel Schedule	: IBC06 Panel Databas
Optimize HD	: Yes
HD Manufacturer:	SIMPSON

GEOMETRY

Total Height	: 12 ft
Total Length	: 20 ft
Wall H/W Ratio	: 0.60
Max Opening Ht	: 4.00 ft
Open/Wall Ht Ratio	: 0.33
% Full Ht Sheathed	: 72.50

MATERIALS

Description	Material	Size
Top Pl	DF Larch	2-2X6
Sill	DF Larch	2X6
Wall Stud	DF Larch	2X6

The **General** information is the some of the important parameters in designing the wall.

The **Geometry** section gives dimensions and ratios for the wall panel.

- The Wall H/W Ratio is checked against the aspect ratio limits given in Table 4.3.4 of the NDS 2005 Special Design Provisions for Wind & Seismic (SDPWS).
- The Max Opening Ht is used in the calculation of $\mathbf{C_o}$.
- The % Full Ht Sheathed is used in the calculation of $\mathbf{C_o}$.

The **Materials** information just gives the sizes of the members that are not explicitly talked about in the detail report.

Note:

- The top plate, sill plate and trimmer sizes are used only for the Material Take Off.

DESIGN DETAILS
Shear Stiffness Adjustment Factor : **1.00**
Shear Capacity Adjustment Factor (Co): **1.03**

WALL DEFLECTIONS		WALL RESULTS:		
Elastic:	: **-0.00**	Governing LC	: **1**	
HD:	: **0.00**	Total Shear	: **.107**	**k**
Shear:	: **0.00**	Max Unit Shear	: **.003**	**k/ft**
		Shear Ratio	: **.016**	

The **Design Details** section first gives some code check information and then gives the same hold down and shear panel information that was given in the Segmented region report.

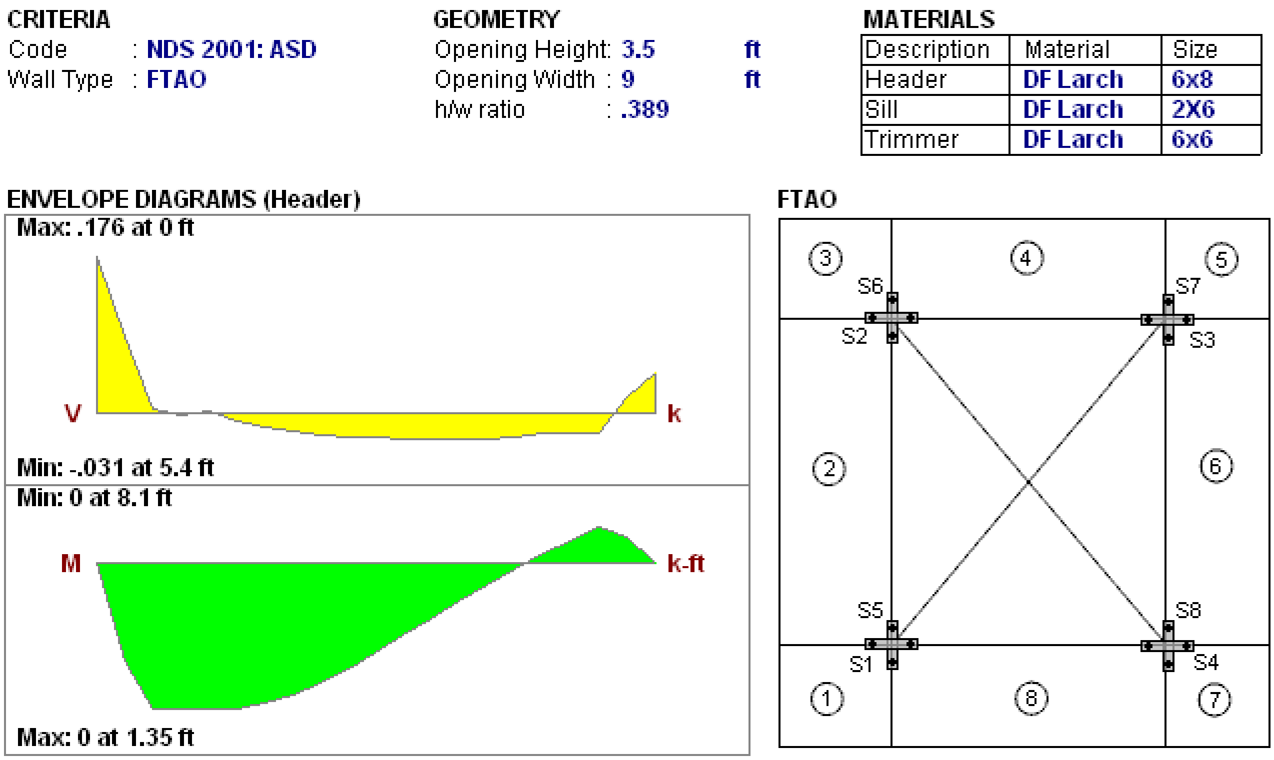
- See the Wall Panels topic for more information on the Shear Stiffness Adjustment Factor.
- See the Wood Wall - Design topic for more information on C_o.
- See above for more information on deflections.
- The **Total Shear** is the total shear in the wall.
- The **Max. Unit Shear** is the maximum shear in the shear panel and it is what is used to optimize the sheathing/nailing selection from the shear panel database.
- The **Shear Ratio** is a ratio of the Max Unit Shear over the Shear Capacity of the shear panel system.

There is also a stud design section that has the same information as the stud design for the Segmented region report.

The **Cross Section Detailing** section gives a detailed view of the wall. For more information see the Segmented section.

Force Transfer Around Openings (FTAO)

Opening Window



The **Criteria** section gives the code being used and which design method used.

The **Geometry** section gives the opening dimensions and the h/w ratio.

The **Materials** section gives dimensions for some of the members in the wall.

The **Envelope Diagrams** give the enveloped shear and moment diagrams for the header beam above the opening.

The **FTAO** graphic shows the design block numbers around the wall panel opening along with the strap numbers.

DESIGN DETAILS

OPENING STRAPS

Name	Location	Direction	Req'd Cap (k)	Gov LC
S1	Bottom, Left	Horizontal	-0.7	1
S2	Upper, Left	Horizontal	0.3	1
S3	Upper, Right	Horizontal	-1.7	1
S4	Bottom, Right	Horizontal	0.9	1
S5	Bottom, Left	Vertical	-0.1	1
S6	Upper, Left	Vertical	-0.0	1
S7	Upper, Right	Vertical	-0.2	1
S8	Bottom, Right	Vertical	0.1	1

ANALYSIS SUMMARY

Block #	Unit Shear (k/ft)	h/w Ratio
1	-0.111	0.917
2	-0.206	0.583
3	-0.156	0.500
4	-0.292	0.333
5	-0.317	0.600
6	-0.190	0.700
7	-0.058	1.100
8	-0.174	0.611

The **Design Details** section is split into two different tables: Opening Straps and Analysis Summary. The opening straps information gives the location, direction, force in the strap and the load combination that caused that force.

Note:

- The program does not design the straps around the opening, just presents the forces.

The analysis summary gives the unit shear and the h/w ratio for each of the blocks.

The information below is the code check information for the header member. This is identical to the information given for Segmented Opening information.

Wall Window

This is the overall wall information and is essentially identical to the Perforated method wall window information. There is some geometry information that is not necessary for FTAO that is omitted.

Appendix F – Wood Database Files

RISA-3D has design databases for wood shear walls and diaphragms which are used to optimize nailing, hold downs, and panels. The criteria used for this optimization is specified on the Wood Wall (Fasteners) and Wood Diaphragms tabs of the **Design Rules** spreadsheet. In addition to this basic criteria, the user may specify a subset of the overall database from which the design / optimization must be performed.

Hold Downs

Each database of hold downs is specified by an XML file in the "Hold Downs" sub-directory of the Wood Wall panels directory. The location of this directory is based on the information in the File Locations tab of the Tools - Preferences dialog.

The program comes pre-loaded with two XML files each of which contain a database of commonly used hold downs: the Simpson HDA/HD hold downs, and the Simpson LTT/MTT/HTT hold downs. The name of the XML file itself will be used in the list of databases in the Hold Down Schedule Dialog.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **SIMP HTT Database**). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, design and code check information used for each hold down. These entries are described below.

Full Database - Required Fields

The following fields are *required* information. If they are not provided or are left blank, then that hold down will not be available for use in that database.

The **Label** field is used to identify the hold down. This field must be referenced on the sheets that identify families or groups of hold downs.

The **Deflection at Peak Load** entry is used to calculate the deflection of the shear wall per APA / NDS formulas. This deflection is then reported on the shear wall detail report for each wall panel.

The **CD Factor** is the assumed load duration factor that was used as the basis for specifying the listed **Allowable Tension** value for that hold down.

A load combination may be solved with a load duration factor different from the CD Factor described above. When this is the case, the **Allowable Tension** for that hold down will be adjusted based on the difference between the assumed and actual load duration factors.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design or capacity calculations, but are reported on the detail reports for reference purposes only.

The **Manufacturer** field is an identifier for the hold down. It is provided so that the engineer can more easily identify the callouts for their final design drawings.

The allowable capacity of the hold down will vary based on the Chord Thickness. Therefore, the **Required Chord Thickness** gives the minimum chord thickness that will yield the listed allowable tension load. However, this field is NOT currently used in the calculations. A future revision may provide a warning message if the actual chord thickness provided is less than required.

The allowable capacity of the hold down will vary based on the density of the wood species being used. Therefore, the **Required Chord Density** lists the density assumed for the entered allowable tension. However, this field is NOT currently used in the calculations. A future revision may provide a warning message if the actual chord density provided is less than required.

The **AB Diameter** is not currently used in the design calculations and is reported for display purposes only.

The **Bolt Size** when specified is used to reduce the axial capacity of the hold down chord itself. The only change to the calculation is that the program will perform the allowable tension check on the net area of the chord member rather than the gross area. The **Nail Size**, on the other hand, is NOT assumed to affect the tension capacity of the hold down chord.

The **Number of Bolts** and **Number of Nails** are not used in the design calculation and are reported for reference purposes only.

The **Is Hold Down Raised?** field is not used in the design calculations and is reported for display purposed only. It is assumed that if the hold down deformation is significantly affected by the connection being flushed or raised, then the **Deflection at Peak Load** entry described in the previous section will adjusted instead.

Grouping Hold Down Schedules for Design Optimization

The other sheets allow the user to group hold downs together into families for optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The hold down labels specified on these sheets refer only to hold downs that have already been defined on the full database sheet. The information in this sheet need not be organized in a specific order. Instead, they will always be optimized based on the assumption that the hold down cost is directly related to the tension capacity. Therefore, when this group is selected, then the hold down within the group with the code check closest to unity, but still less than 1.0 will get selected during the optimization process.

Panel Nailing Schedules

Each database of wall panels is specified by an XML file in the "Shear Panels" sub-directory of the Wood Wall panels directory. This directory is located based on the information File Locations tab of the Tools - Preferences dialog.

The program comes loaded with two XML files one for the tabularized nailing schedules listed in the 1997 UBC and one for the ones listed in the 2006 IBC.

The first sheet of the XML file should always be descriptive of the contents of the database (such as **IBC06 Panel Database**). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, nailing, design and code check information for each nailing schedule. These entries are described below.

Full Database - Required Fields

The following fields are *required* information. If they are not provided or are left blank, then that nailing schedule will not be available for use in the database.

The **Label** field is a used to identify the panel schedule and its nailing requirements. This field must be referenced on the sheets that identify families or groups of panels.

The **Min Panel Thickness** is used during the design optimization to limit the selected panels based on the Design Rules chosen by the user. It is also used to help set the elastic stiffness of the wall panel used during the FEM solution.

The **Ga** value is the **Apparent Shear Stiffness** from nail slip and panel deformation as defined in equation 4.3-1 of the NDS' Special Design Provisions for Wind and Seismic. This value (in combination with the Min Panel Thickness defined above) is used to set the elastic stiffness of the wall panel that will be used during the FEM solution.

Note:

- When a family or group of panels / nailing schedules are assigned to a shear wall, the lowest value of Ga and Min Panel thickness will be used to determine the elastic stiffness of the plate elements in the FEM solution.

The **One/Two Sided** field is used during the design optimization to limit the available panels based on the Design Rules specified by the user.

The **Boundary Nail Spacing** field is used during the design optimization to limit the available panels based on the Design Rules specified by the user.

Note:

- The maximum field spacing is never entered in the program but is generally equal to 12 inches for the nailing schedules defined in the 1997 UBC and 2006 IBC databases. If a different nail spacing is present, then the user should add in a new nailing schedule to the existing database with a user defined shear capacity.

The **Shear Capacity** listed in the spreadsheet is the primary value that controls the code checking of the shear wall.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design or capacity calculations, but are reported on the detail reports for informational purposes only.

The **Panel Grade** and **Min Penetration** fields are identifiers for the engineer, but are not used in the design calculations. They are provided so that the engineer can most easily identify the panels in their design results and drawings.

The **Panel Applied Over Gypsum** field is also a identifier for the engineer that will not be used in the design calculations.

The **Nail Size** listed in the spreadsheet is intended to refer to the Common nail size, but is reported only for reference purposes and are NOT used in the capacity calculations. If the nail size is changed by the user, then the user should also change the Shear Capacity entry accordingly. Below is a reference table for common, box, and sinker nails.

	Diameter				Length	
Penny Weight	Common (in)	Common (Wired Gage)	Box (in)	Sinker (in)	Box and Common (in)	Sinker (in)
6d	0.113	11.5	0.099	0.092	2	1.875
7d	0.113	11.5	0.099	0.099	2.25	2.125
8d	0.131	10	0.113	0.113	2.5	2.375
10d	0.148	9	0.128	0.120	3	2.875
12d	0.148	9	0.128	0.135	3.25	3.125
16d	0.162	8	0.135	0.148	3.5	3.25

The Staple size listed in the database is reported for reference purposes only. If the staple size is entered or changed by the user, then the user should also change the shear capacity entry to the appropriate value.

The **Wind ASIF** field is not currently used within the program at all. It is reserved for a future feature.

Grouping Panel / Nailing Schedules for Design Optimization

The other sheets in the database allow the user to organize multiple nailing schedules into groups or families for design optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The panel labels specified on these sheets refer only to panel / nailing schedules that have already been defined on the full database sheet. The information in this sheet need not be organized in a specific order. Instead, they will always be optimized based on the assumption that the installed cost is directly related to the shear capacity. Therefore, when a group or family is selected, then the nailing schedule within the group with the code check closest to unity, but still less than 1.0 will get selected during the optimization process.

Diaphragm Nailing Schedules

Each database of diaphragms is specified by an XML file in the "Diaphragms" sub-directory of the Wood Wall schedules directory. This directory is located based on the information on the File Locations tab of the Tools - Preferences dialog.

The program comes loaded with two XML files, both based on the shear values of the 2006 IBC and the stiffness values of the 2005 NDS Special Design Provisions for Wind and Seismic. The WSP (Wood Structural Panel) is intended to be a

generic database that could be used used for plywood or OSB panels. However, it uses the Ga (apparent stiffness) values for plywood because they are genearally lower resulting in a more conservative deflection.

Note:

- Diaprhagm design is currently only available for flexible diaphragms that were created in RISAFloor and brought into RISA-3D.

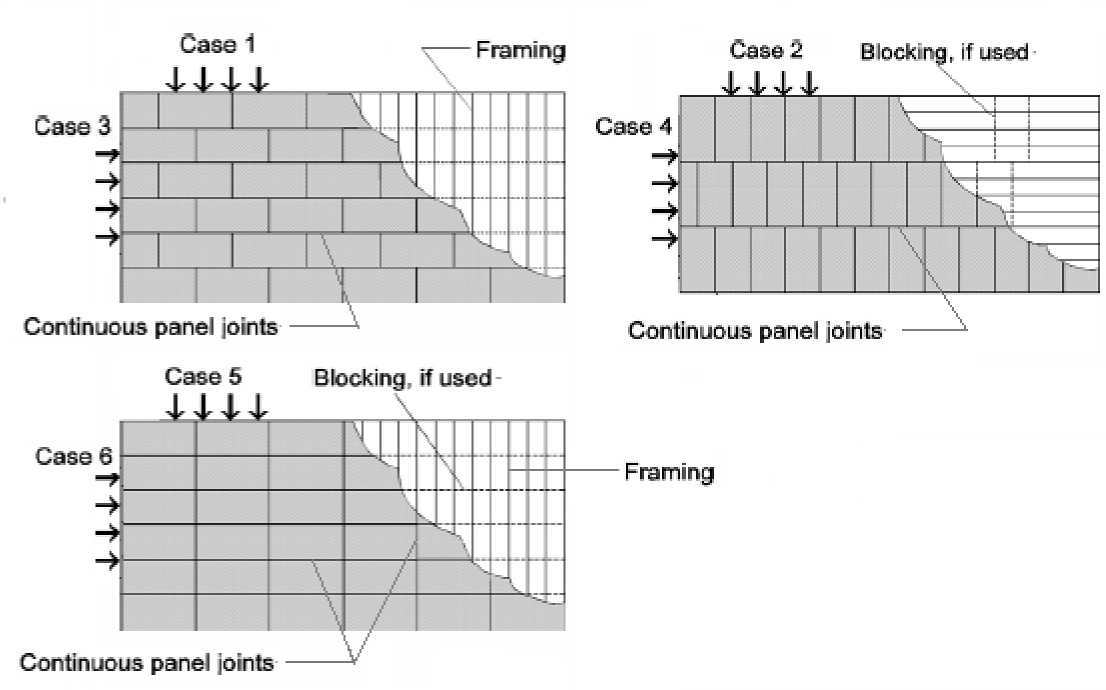
The first sheet of the XML file should always be descriptive of the contents of the database (such as IBC06 OSB Database). This is because the name used here is the name used in the Design Rules spreadsheet. This sheet contains all of the identifier, nailing, design and code check information for each nailing schedule. These entries are described below:

Full Database - Required Fields

The following fields are required information. If they are not provided or are left blank, then that nailing schedule will not be available for use in the database.

The **Label** field is a used to identify the diaphragm nailing. This field must be referenced on the sheets that identify families or groups of panels.

The **Case** field is used to specify the layout of the shear panels as shown below. Any diaphragm that has a Case 1 layout also has a Case 3 layout, and the same goes for 2/4 and 5/6.



The program considers the deck span defined in RISAFloor to coincide with the long direction of the plywood. Therefore, Cases 2, 3 and 6 are considered parallel to the to the RISAFloor deck span. Whereas, Cases 1, 2 and 5 would be considered perpendicular to the same deck span.

The **Blocked** field specifies whether blocking is used to achieve the associated design strength. It also determines the method by which the diaphragm deflection will be calculated. For more information see [Diaphragm Deflection]

The **Panel Grade** field specifies what grade of structural panel is used in the diaphragm. This may be set to either “Structural-I” or “Other” and is used as a criteria in the Design Rules.

The **Panel Thickness** field specifies the thickness of the structural panel used for the diaphragm. This is a decimal value that is rounded to four places for reporting convenience. For example, a 15/32” panel is listed as 0.4688

The **Boundary/Cont Edge Spacing** field specifies the nail spacing at the boundary and along any continuous edges. These must be specified as the same value.

The **Other Edge Spacing** field specifies the nail spacing at non-continuous edges.

The **Nail Lines** field specifies the number of lines of nails along each panel edge. This value is greater than (1) for High Load diaphragms. This value is not currently used in design optimization, but is reported on the output for reference purposes only.

The **Strong Shear Capacity** field specifies the shear strength of the diaphragm (lbs/ft) based on its stronger case. For example, while Case 1/3 represents the same panel layout, Case 1 has greater strength than Case 3 (based on load direction).

The **Weak Shear Capacity** field specifies the shear strength of the diaphragm (lbs/ft) based on its weaker case. There are many situations where strong and weak capacities are identical. In these cases the same value must be specified for both fields.

The **Strong Ga** field specifies the apparent shear stiffness (kips/in) of the diaphragm as specified in the NDS document *Special Design Provisions for Wind and Seismic* Since this is the strong direction it will be based on the stronger direction / case for loading. For example, while Case 1/3 represents the same panel layout, Case 1 has greater stiffness than Case 3 (based on load direction). For more information see Diaphragm Deflection

The **Weak Ga** field specifies the apparent shear stiffness (kips/in) of the diaphragm as specified in the NDS document Special Design Provisions for Wind and Seismic.

The **Gt** field specifies the shear stiffness of panel depth. It is always the same for both strong weak directions, hence it does not need to be specified twice. For more information see Diaphragm Deflection

The **Strong Nail Slip (en)** field specifies the nail slip used for deflection calculations based on the stronger case. For example, while Case 1/3 represents the same panel layout, Case 1 may have less nail slip than Case 3 (based on load direction). For more information see Diaphragm Deflection

The **Weak Nail Slip (en)** field specifies the nail slip used for deflection calculations based on the weaker case. There are many situations where strong and weak nail slips are identical. In these cases the same value must be specified for both fields. For more information see Diaphragm Deflection

Note:

- The Gt and Nail Slip fields are ignored for unblocked diaphragms.

Full Database - Optional Fields

The following fields are optional. They are not currently used in the design, capacity or deflection calculations, but are reported on the detail reports for informational purposes only.

The **Framing Width** field identifies the minimum required framing width for the nailing layout. A higher shear capacity can typically be achieved for a diaphragm by using wider supporting framing, thereby reducing the tension perpendicular to the grain of supporting members.

The **Minimum Penetration** field identifies the minimum required nail penetration specified in the IBC/NDS tables.

The **Nail Size** listed in the spreadsheet is intended to refer to the Common nail size, but is reported only for reference purposes and are NOT used in the capacity calculations. If the nail size is changed by the user, then the user should also change the Shear Capacity entry accordingly. The section on shear walls contains a good reference table for common, box and sinker nails.

The **Wind ASIF** field is not currently used within the program at all. It is reserved for a future feature.

Grouping Panel / Nailing Schedules for Design Optimization

The other sheets in the database allow the user to organize multiple nailing schedules into groups or families for design optimization purposes. These additional sheets CANNOT be the first sheet in the XML file as that first sheet must always be the one where the full database information resides.

The labels specified on these sheets refer only to nailing schedules that have already been defined on the first (full) database sheet. The program is not capable of understanding the complexities of diaphragm construction cost (whether it is cheaper to increase panel thickness versus decreasing nailing spacing for example). Instead the program simply “walks down” the listed labels, looking for the first one that meets the required strength. You may re-order the labels on the Group tabs to meet the most economical arrangement for you.

Help Options

RISA Technologies has, and will, put a great deal of effort into assisting you in getting your work done as quickly as possible. This includes providing many ways that you can get help in understanding the software.

Electronic Help File

The **Help File** is designed to help you get your work done as quickly as possible and is intended to provide:

- Procedures that lead users through the steps of completing tasks
- Context-sensitive Help topics that provide users with quick descriptions of items on their screens
- Troubleshooting topics that guide users through solutions to common problems
- Extensive discussions for a thorough understanding of engineering and modeling topics
- Easy access to related topics

The electronic help file can be accessed by clicking the **Help File** button on the **RISA Toolbar**. A new window containing a Table of Contents will be opened. Click on any item in the Table of Contents for extensive information on the topic.

Context Sensitive Help

Context Sensitive Help is help that a user can access in context while working in a program. It provides you with the information you need where and when you want it. This type of help is provided in two different ways:

The first method provides specific information on every item that you see on the screen. For help on an item such as an entry field in a dialog, click ? and then click the item. This feature is referred to as "**What's This?**" help. This feature may not function properly on Vista computers.

You may also get more detailed help when working in some windows by clicking on the **Help** button at the bottom of the window or dialog. This will launch a Help File window displaying the topic that is related to the window in which you are working. The topic will be explained and links to related topics may also be provided.

RISA Technologies Online

Our website, **www.risatech.com**, provides various support information and documents.

Visit RISA Technologies on the web for:

- Frequently Asked Questions
- Download program Manuals (General Reference or Tutorial)
- The latest software updates - When a bug is discovered it is posted on the web site along with possible work-around procedures and/or service releases to update your software.
- Software Verification Problems

Tool-tips

Are you uncertain what a toolbar button is for? Simply hold your mouse pointer over that button without clicking. **Tool-tips** are displayed that will explain what the button will do should you decide to press it.

Tutorial

The comprehensive **Tutorial** (part of the User's Guide - a separate document) guides you through using most features. It is a real-world example of building and solving a model, making changes, and optimizing. This is the best way to quickly get up and running. The User's Guide is designed to be read in two ways. If you are already familiar with structural modeling in general you can skip the supporting text and read only the underlined action items to quickly move through the tutorial. If you want more thorough explanations of the modeling process you may read all or some of the supporting text as you see fit.

Technical Support

Technical support is an important part of the RISA-3D package. There is no charge for technical support for all licensed owners of the **current** version of RISA-3D. Technical support is very important to the staff at RISA Technologies. We want our users to be able to reach us when they are having difficulties with the program. However, this service is not to be used as a way to avoid learning the program or learning how to perform structural modeling in general.

Hours: 6AM to 5PM Pacific Standard Time, Monday through Friday

Before contacting technical support, you should typically do the following:

1. **Please search the Help File or General Reference Manual**. Most questions asked about RISA-3D are already answered in the Help File or General Reference Manual. Use the table of contents or index to find specific topics and appropriate sections. We go to great lengths to provide extensive written and on-line documentation for the program. We do this in order to help you understand the features and make them easier to use. Also be sure to go through the entire User's Guide when you first get the program.
2. If you have access to the Internet, you can visit our website at **www.risatech.com** and check out our **Support** section for release notes, updates, downloads, and frequently asked questions. We list known issues and product updates that you can download. So, if you think the program is in error you should see if the problem is listed and make sure you have the latest release. The **FAQ** **(Frequently Asked Questions)** section may also address your question.
3. Make sure you understand the problem, and make sure your question is related to the program or structural modeling. Technical Support does not include free engineering consulting. RISA Technologies does provide a consulting service. If you are interested in inquiring about this service, please call RISA Technologies.
4. Take a few minutes to experiment with the problem to try to understand and solve it.

For all modeling support questions, please be prepared to send us your model input file via email or postal mail. We often will need to have your model in hand to debug a problem or answer your questions.

Email: support@risatech.com: This method is the **best** way to send us a model you would like help with. Most email packages support the attachment of files. The input file you would send will have a .R3D extension. Make sure you tell us your name, company name, serial number or Key ID, phone number, and give a decent problem description. If you have multiple members, plates, or load combinations, make sure you specify which ones to look at.

Phone Support:(949) 951-5815: Feel free to call, especially if you need a quick answer and your question is not model specific and therefore doesn't require us to look at your file.

Postal Mail to RISA Technologies: This method works fine as long as you can wait for the postal service. If you don't have email then this is the only way to send us the model that you need help with. Most people who are in a rush will send a floppy disk via overnight mail (email is a lot cheaper and faster, though).

RISA Technologies

Tech Support

26632 Towne Centre Drive

Suite 210

Foothill Ranch, CA 92610

Index

N

O

P

R

S

T

U

V

W

Modeling Tips

Dummy Members

Dummy Members are used to simplify the framing in order to properly attribute load in complex regions in RISAFloor. They must be weightless members and they must run parallel to the deck. Doing those two things assures that these members will not receive any load of their own and will only serve to break a complex region into two smaller regions.

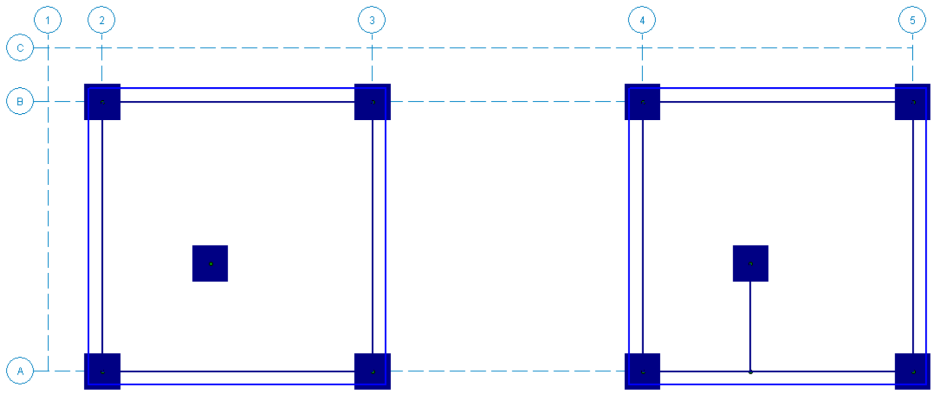
To Make a Dummy Member:

1. On the General tab of the Materials spreadsheet create a material Label called “Dummy.” Use 29000 ksi for the value of E. Blank out the value for G by going to that field and pressing the space bar. Set the Density to zero. Leave all the other values as their defaults.
2. Use the Draw in a General Member with an explicit shape of “BAR1” and a material of “dummy.” Set the end releases to Pinned at both ends.
3. This member must be drawn in parallel to the deck. Frequently, this will require the use of the “Beam to Beam” draw options shown below.

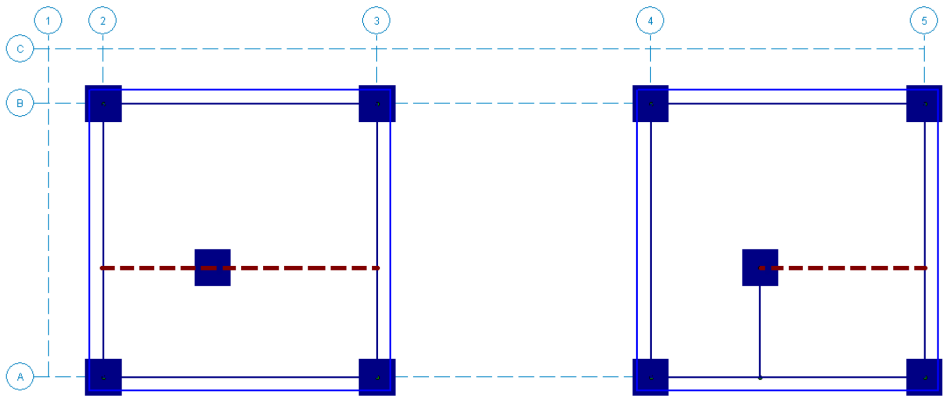
Note: The “Beam to Beam” options can also be used to draw from a wall to a beam or wall to another wall.

Stranded Columns

The image below show two types of typical "stranded" columns. By calling it stranded, we mean to point out that the framing around this column is incomplete; the program requires a closed circuit of members to attribute load properly. The load attribution code will tend to have problems with this type of situation. This sort of framing configuration will halt the solution. If properly detected by the program, it will result in a "framing error" which will identify the exact location of the framing issue. If not properly detected by the program, it could result in a more ambiguous error such as a Load Attribution or Polygon error.

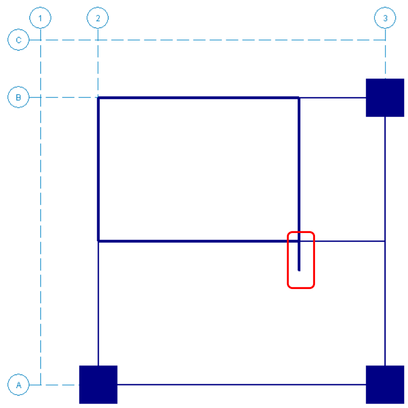


To correct this type of problem, you would typically add in a dummy member parallel to the deck that would frame out the column, but would not itself receive any loading. An example of this type of correction is shown using the dashed lines in the image below. In order to avoid these types of framing problems a column must be connected on at least two sides:

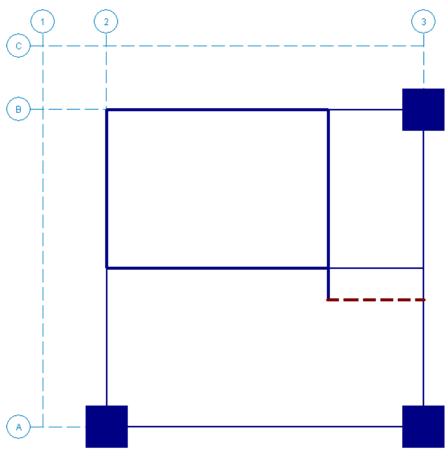


Stranded Walls

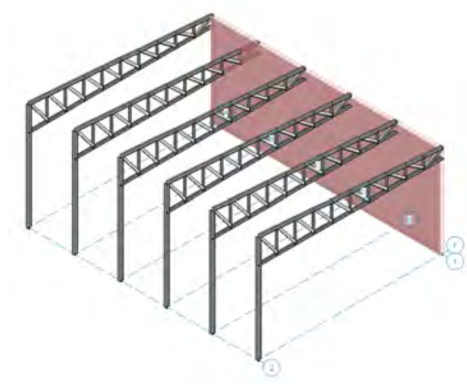
The image below shows a typical "stranded" wall. By calling it stranded, we mean to point out that the framing around this wall is incomplete. The load attribution code will tend to have problems with this type of situation. This sort of framing configuration will halt the solution. If properly detected by the program, it will result in a framing error which will identify the exact location of the framing issue. If not properly detected by the program, it could result in a more ambiguous error such as a Load Attribution or Polygon error.



To correct this type of problem, you would typically add in a dummy member parallel to the deck that would frame out the wall, but would not itself receive any loading. An example of this type of correction is shown using the dashed lines in the image below. In order to avoid these types of framing problems a wall must have at least one beam (or wall) framing into each end joint of the wall.



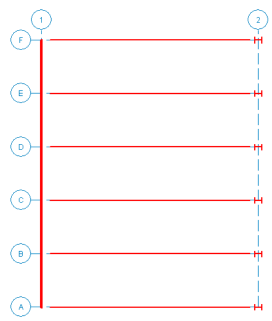
Mono-Slope Roof Truss



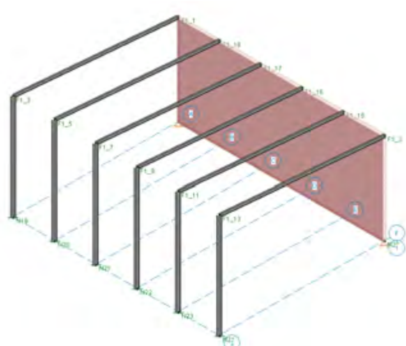
Creating a sloped roof truss is a two step process working first in RISAFloor and then using the Director tool to finish modeling in RISA-3D. Because the user cannot create brace members in RISAFloor, they must first layout the chord "framework" and then finish filling in the web members once in RISA-3D.

Starting in RISAFloor:

1. To start your model in RISAFloor, lay out all walls, columns, and beams as usual.

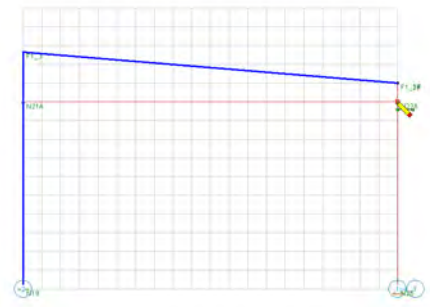


2. Use the Elevate Joints tool to apply the slope to your roof level.
3. Now, apply loads and solve.
4. Use the Director tool to bring your model into RISA-3D.

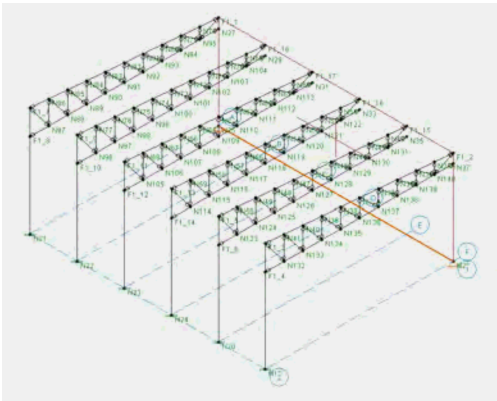


Finishing in RISA-3D:

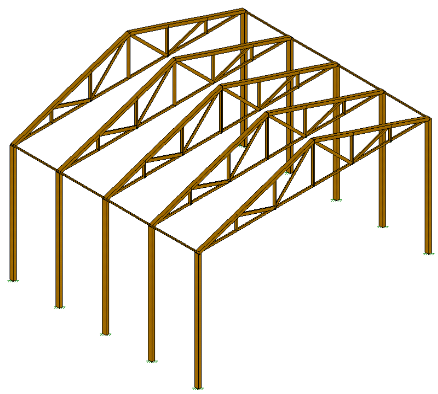
1. Draw in the bottom chord on the end gridline with the assistance of the drawing grid.



2. Copy the new chord member to the other bays.
3. Use the Split Members tool to add in additional joints along the top and bottom chords.
4. Draw in web members to finish.



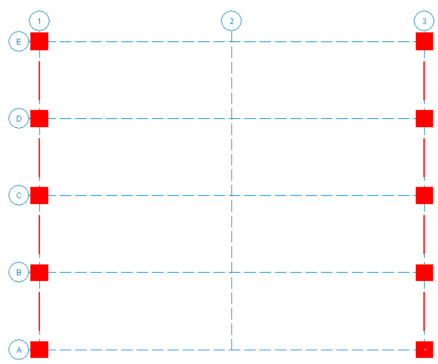
Gabled Roof Truss



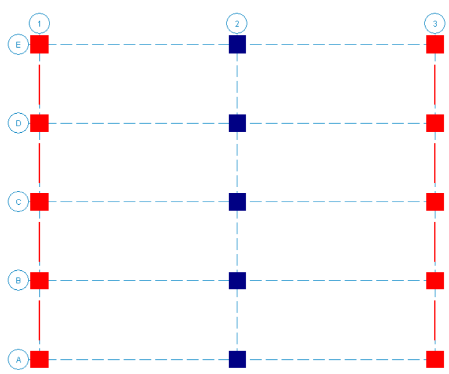
Another option for creating sloped roof trusses is the gabled roof truss. This type of truss requires a little more work, but still can be easily created in the RISAFloor- RISA-3D interaction.

Starting in RISAFloor:

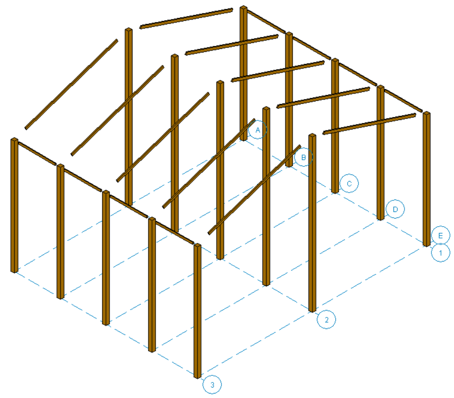
1. Start your model as before by laying out the outer columns. Designate these columns as "Lateral" so that they will be transferred into the RISA-3D model.



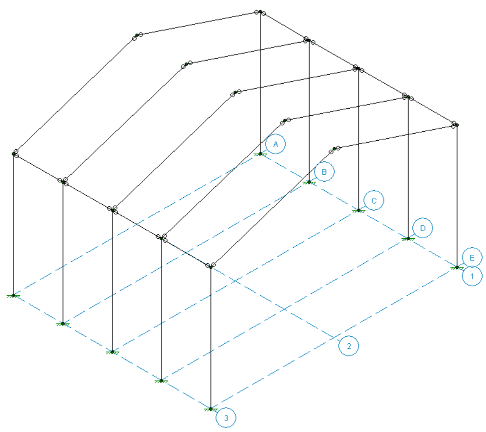
2. Now draw in "Gravity" columns along the center. These will allow us to slope the members on either side but will not transfer across into RISA-3D.



3. Draw in the beams (designated as “Lateral”) and use the Elevate Joints tool  to elevate the center columns and slope the members.

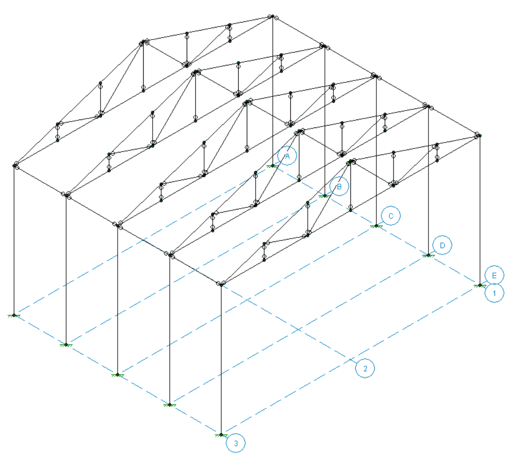


4. Now solve your model and use the Director Tool to transfer the model into RISA-3D.



Finishing in RISA-3D:

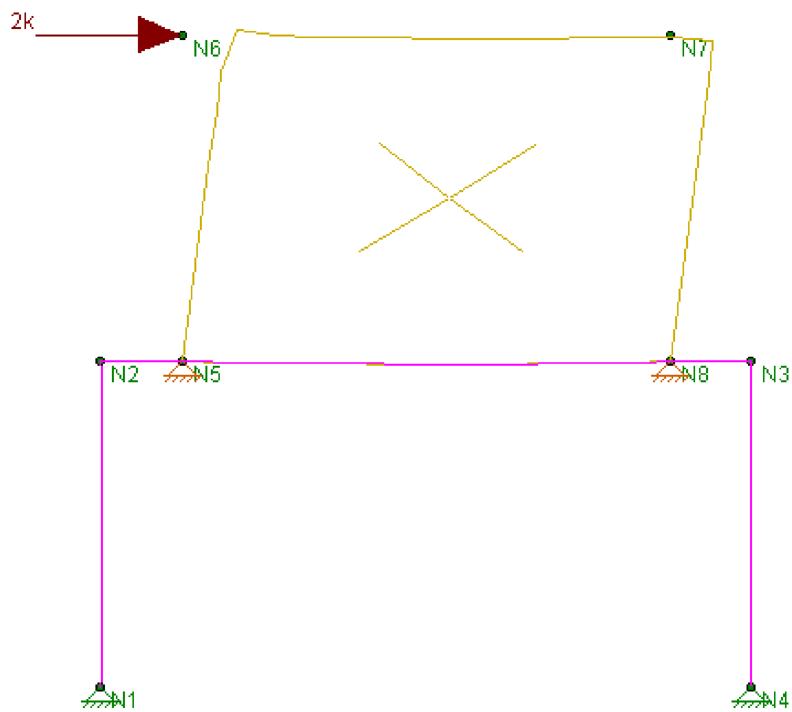
1. Once in RISA-3D you can draw in the remaining lower chord and web members of your trusses.



Walls Sitting on Beams

Wood wall panels are required to have hold-downs for wall panels to be designed. Hold-downs are considered boundary conditions in RISA-3D. There is the rare circumstance where a shear wall panel is supported by something other than the foundation, such as a beam.

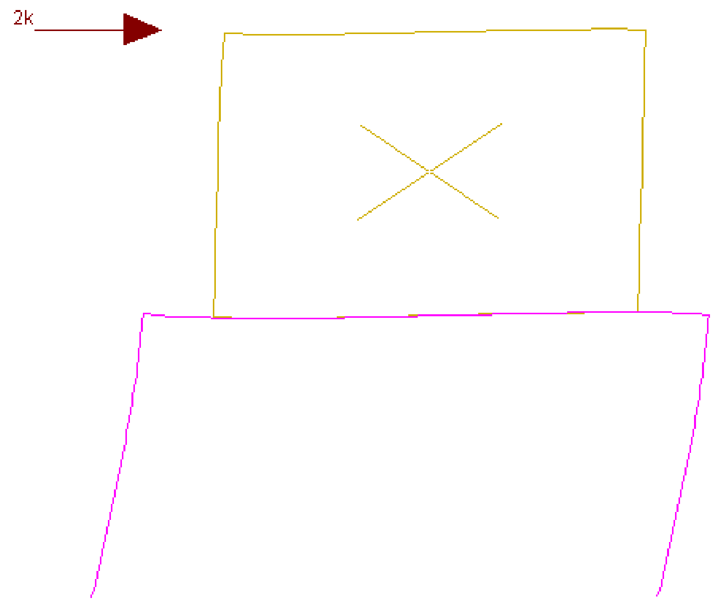
In order to model a situation where a wall panel attaches to a beam, rather than the foundation, an extra step must be taken. Below is an image of how the behavior is by default:



The wall panel's corners are drawn with hold-downs at the corners. All of the lateral load from the wall is being removed from the wall where the hold-downs (orange boundary conditions) are shown. Thus, none of the wall forces are getting into the beam or columns. The solution to this is to use assigned boundary conditions to override the hold-down. If you assign a boundary condition to a joint where a hold-down exists, the assigned boundary condition overrides the hold-down boundary condition.

In the example shown above, apply a "Free" boundary condition to Joints N5 and N8. This is not graphically visible (since there is no graphic for a "Free" condition) but it will appear in the Boundary Conditions spreadsheet. This overrides the condition at Joints N5 and J8 so that instead of behaving as pinned to external boundaries they are free from external boundaries, and instead connect directly to the beam below.

After setting this and solving again, you will see that the load transfers through the beam and columns, down to the base boundary conditions:


2k