# WoodWorks®

# Design Office

***Sizer* | *Shearwalls* | *Connections* | *Database Editor***

# 2016 User Guide – Volume 1 User Manual

**For U.S. Design Office 10 and Canadian Design Office 9**

Canadian Wood Council

American Wood Council

Developed by

Acronym Software Inc.

Limited Warranty

The American Wood Council and the Canadian Wood Council make no representations or warranties with respect to the contents of this user guide and specifically disclaim any implied warranties of merchantability or fitness for any particular purpose. The contents of this documentation may include technical inaccuracies or typographical errors.

The American Wood Council and the Canadian Wood Council disclaim all warranties with respect to the software contained on disk or in printed form, including all expressed or implied warranties of merchantability and fitness; and any stated or expressed warranties are in lieu of all obligations or liabilities of the American Wood Council and the Canadian Wood Council for damages, including, but not limited to special, indirect or consequential damages arising out of or in connection with the performance of the software.

As always, the engineer is ultimately responsible for his or her design.

Refer to Read Me for further information.



07-03-16

10-03-16

# Table of Contents

# 1 Welcome to WoodWorks® Design Office Suite

## 1.1 What Is Design Office?

**WoodWorks®** Design Office is high value software suite suitable for most wood engineering projects. Its component-based operation is intuitive, quick and easy to use, and inexpensive. It does not require a full building information modelling of the structure - whole-building modelling is often very time consuming and not necessary for many projects for which wood is the main structural component.

**Design Office** consists of three independent programs: ***Sizer***, **Shearwalls**, and ***Connections***:

**WoodWorks® *Sizer*** program allows members up to six spans to be designed including cantilevers, with or without notches, and with common load distributions such as point load, partial or full uniform load, triangular load, trapezoidal load, moving load, eccentric axial load, applied bending moments, and load patterning. Simultaneous bi-directional loads can also be applied such as axial dead and live loads and a laterally applied wind load. The program allows for any building code load type to be applied including live, dead, earthquake, snow, wind, and construction. *Sizer* comes with standard databases which are based on all of the available grades in the NDS or the CSA O86 design properties.[1,2] The program can be professionally customized to suit a specific manufacturer or to support a particular product not already included in the standard database. The database can also be edited by the designer using the WoodWorks® *Database Editor* tool. *Sizer* not only design beams, it designs columns, wall studs, sloped members and obliquely angled members. *Sizer* also includes Concept mode which can be utilized to create preliminary gravity load design models up to 6 storeys tall.

**WoodWorks® *Shearwalls*** program allows engineers to quickly model a building by importing a .pdf, .jpg, or .wmf file as a template, or by drawing a model from scratch. Most buildings can be modelled in minutes, and once complete, it requires little more than ensuring the proper wind and seismic data is entered before clicking the *Generate loads* button and then the *Run design* button. The software automatically calculates the wind pressures and component and cladding forces as well as seismic forces following the equivalent static force procedure. Once generated the loads distribute to each level, then within each level to each shearline, and within each shearline to each shearwall segment. It does this using both the flexible and rigid diaphragm distribution methods for both seismic and wind forces. For rigid distribution, *Shearwalls* uses either a “capacity” approach or a “stiffness” approach to distribute forces to shearwalls and along shearlines. *Shearwalls* then designs appropriately strong shearwalls to resist the force.

**The WoodWorks® *Connections*** program is an important part of the WoodWorks® software suite. Its intuitive interface makes it easy to model a connection using bolts, lag screws, wood screws, nails, rivets, and shear plates. WoodWorks® *Connections* not only calculates the number of fasteners in the group, it also draws a CAD quality detail drawing of the connection.

### 1.2 Scope of this 2015 Edition of the User Guide

The purpose of this guide is to help the beginner quickly and efficiently learn how to use *Sizer*, *Shearwalls*, *Connections*, and the *Database Editor*. The user guide also includes step by step tutorials demonstrating how to use the software. More information is available in the On-line Help files that can be opened from within the software by pressing F1 or through the *help* menu.

The 2015 User Guide applies to both Canadian and U.S. versions of WoodWorks® software. Minor differences in the programs do exist and some screen captures may not be exactly as shown. This edition of the guide was based on the U.S. Design Office 10 (originally released July 2013) and Canadian Design Office 9 (originally released September 2014, and revised to include CSA O86-14 in July 2015), and includes guidance about all three programs of the Design Office suite. Some differences between the two countries are due to different code or standard requirements, and some differences are transitional, in which case the more recently released Canadian version will likely migrate to the next U.S. version (see "Design" settings and "Default Values" settings for examples in sections 3.6 and 3.7, respectively).[2]

### 1.3 Technical Support

For questions about engineering assumptions, features and functions, please consult the online help which provides a keyword search feature.

The WoodWorks® website contains additional information which includes product news, frequently asked questions, maintenance releases, and updates for registered software owners.

www.woodworks-software.com

If you have installation or software performance issues, please contact WoodWorks® Support via one of the options listed below.

Email: support@woodworks-software.com

Phone: 1-800-844-1275 ext. 2

## 1.4 Installing Design Office

### 1.4.1 System Requirements

The following are the requirements for running WoodWorks® Design Office on PC-compatible computers:

| System Requirements | Minimum | Recommended (or better) | Notes |
|---|---|---|---|
| Processor speed | 1 GHz | 2GHz | Single core; or equivalent performance in multiple core processors |
| RAM | 512 MB | 1 GB (1024 MB) | |
| Screen resolution (pixels) | 800 x 600/500 | 1280 x 1024/768 | Standard/wide screen |
| Free hard drive space | 80 MB | 150 MB | Minimum is for download and installation; recommended includes 100 typical project files. |
| Operating systems | Windows 10, 8, 7 (32- and 64-bit), and Vista (32- and 64-bit). | | |

Note that WoodWorks® Design Office is not compatible with Mac operating systems, but does work on Mac computers if a Windows emulator is utilized.

### 1.4.2 Downloading Installation File

- Go to **www.woodworks-software.com**
- Go to Software Downloads for either the U.S. or Canadian edition and select the full Design Office installation file or the Sizer Stand-alone installation file
  **Canadian Edition** - http://cwc.ca/woodworks-software/canadian-edition/downloads/
  **U.S. Edition** - http://cwc.ca/woodworks-software/us-edition/downloads/
- **Save** the appropriate .exe file to the hard drive

### 1.4.3 Quick Installation of the Software

- **Run** the **.exe** file and follow the installation procedure instructions. See section 1.4.4 for more information on installing the software
- Once installation is complete, open the software, and you will be prompted to enter a keycode
- If you have already purchased the software, send your software ID which appears at the bottom of the keycode prompt window to sales@woodworks-software.com. See section 1.5 for more information on obtaining a keycode.
- If you would like to access the demo version of the software use "**DEMO**" as the ***keycode.*** The demo version does not allow you to select every available species of lumber and you cannot save or print design results.

### 1.4.4 Additional Installation Information

The installation program allows you to specify the installation folder and the start menu folder on your computer. The installation folder is C:\Program Files\WoodWorks\[USA or Candian Version], but you can modify the location as desired.

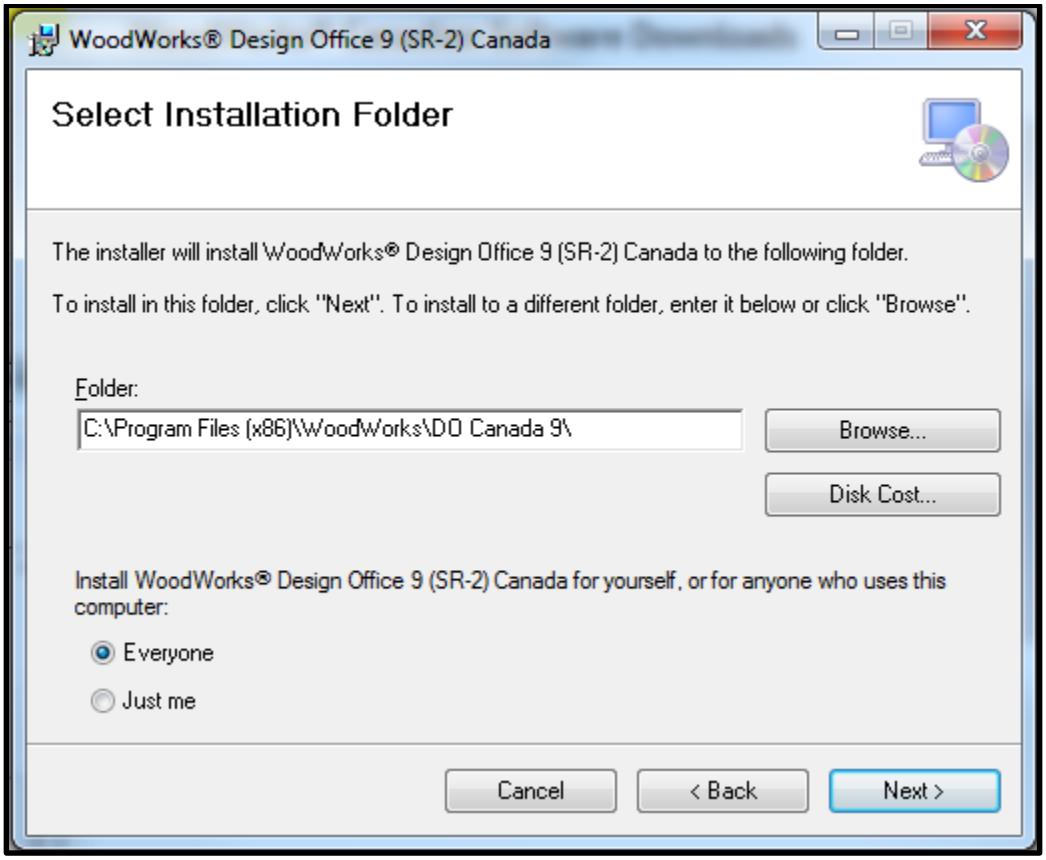



Figure 1: *Select Installation Folder*

During the installation, WoodWork® Design Office Installation options in Figure 2 will appear. If the installation is an upgrade from a previous version of the software, you are given the option of retaining your ***Sizer settings, Shearwalls settings, Connections settings, Standard shearwalls, Material database – Custom, Material database – Standard*** and any ***Hold-down database*** from previous installations of the software. If this is the first installation of WoodWork® Design Office, simply press OK to finish the installation as all components will automatically be installed. Refer to Read Me files for additional instructions regarding installing and uninstalling software.

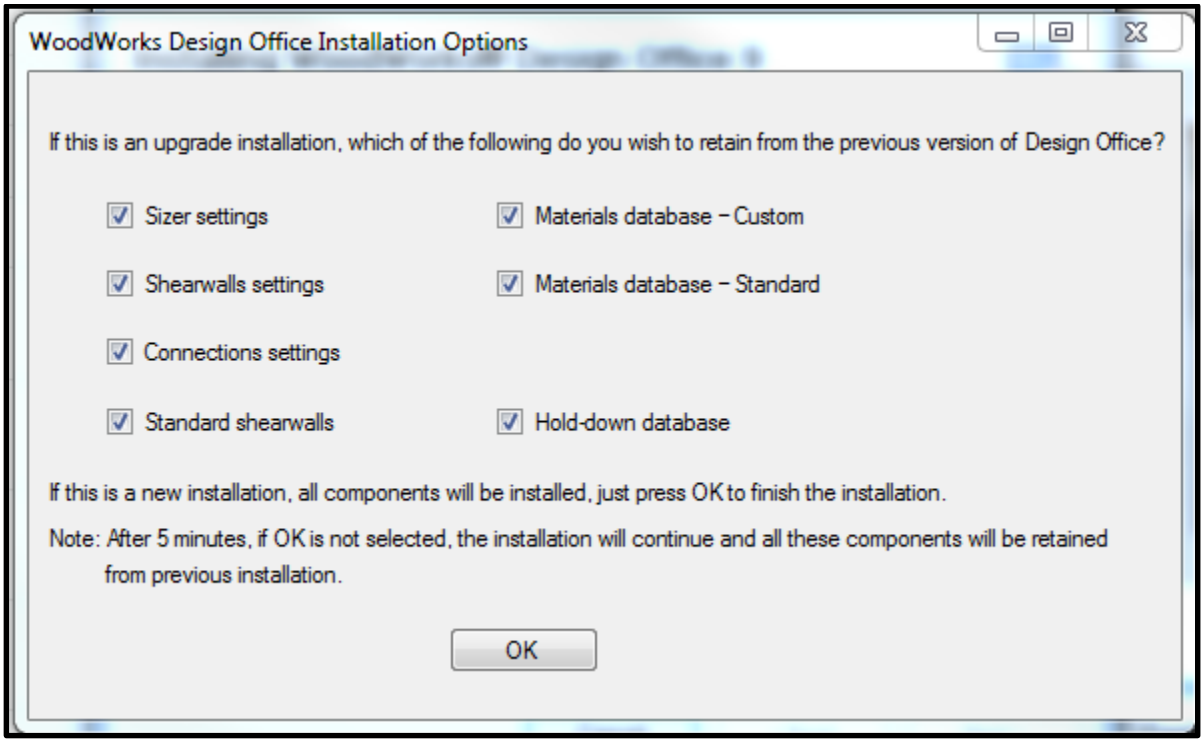



Figure 2: *WoodWork® Design Office Installation Options*

### 1.4.5 Network Installations

This section provides instructions about how to install WoodWorks software on a network server, rather than on individual computers. With a network installation, each Woodworks user opens the program from the server and runs it on their individual "client" computer.

Woodworks does not support "concurrent-use" network licensing, or monitor how many client computers are running the software at any one time. The network installation requires that a software license be purchased for each user who has access to and runs the software. A registration key code is required for each computer that the software is run on.

If any WoodWorks Design Office program is to be run from a server then the network administrator must do the following after installation on the server:

1. Find the *Sizer.ini*, *Shearwalls.ini*, and *Connections.ini* files in the server's WoodWorks *Program Data* folder for *All users*. The location of this folder depends on the server's operating system:

   Windows 7 and 8 (U.S.):

   *C:\ProgramData\WoodWorks\CWC\USA\10*

   Windows 7 and 8 (Canadian):

   *C:\ProgramData\WoodWorks\CWC\Canada\9*

2. Share the WoodWorks *Program Data* folder above, with permissions that allow it to be accessed by the client machines. The server computer's Design Office installation and *Program Data* folders must allow read privileges for all network users.

3. Performing the following step will instruct the client computer to look for the database files on the server as this location. Create a copy of the *Sizer.ini*, *Shearwalls.ini*, and *Connections.ini* files and modify the copied files by:

a) Removing the remark symbols (;) from the following lines:

```
;[Network]
;INI_Location=
```

b) Entering the network path where the database files can be found on the server location:

```
[Network]
INI_Location=\\[Server Name]\[Share on server to Woodworks program data folder]\CWC\USA\10\
```

4. Move the modified copy of the Sizer.ini, Shearwalls.ini. and Connections.ini file into each program's installation folder on the server, that is, the folder into which the program had been installed. For a default installation on Windows 7, the folder is
U.S. Edition:

C:\Program Files (x86)\Woodworks\USA\[program name]\

Canadian Edition:

C:\Program Files (x86)\Woodworks\DO Canada 9\[program name]\

where [program name] is Sizer, Shearwalls, or Connections.

The first time one of the programs is run on the client computer, the .ini file for that program will be automatically copied from the installation folder on the server to the Program Data area of the client machine. This allows the client program to find the shared database resources on the server, and also for the user of the client computer to independently save program settings and preferences

The following steps are necessary for each client machine that you wish to access the server installation:

1. If this is the first time WoodWorks has been set up to run on the client machine, copy and expand the *ShearwallsNetworkSetupNT.exe* self-extracting zip file on each client computer. You can find this file in the in the server's *Shearwalls* installation folder. Then run *ShearwallsNetworkSetup.cmd* from the directory to which the exe file was extracted on each client computer. To remove network capability run *ShearwallsNetworkUninstall.cmd* from the directory to which the exe file was extracted on each client computer.
2. If the client machines do not already have the *Microsoft Visual C++ Redistributable* .dll files in the Windows *System* folder, copy and run the *vcredist_x86.exe* file on each client PC. This file is located in the server's*Sizer* installation folder, and affects the Sizer program only.
3. To run the program on each client machine, you require a separate key code for each. Refer to the section 1.5 about receiving a keycode. If it is known when installing the server software which computers will be running it, we recommend that you run the software on all client machines and order all the keycodes at once.

4. If the program does not operate, check that the *Sizer.ini*, *Shearwalls.ini*, and *Connections.ini* files have indeed been copied to the client machine's *Program Data* folder for the person using the software. The location of this folder depends on the client machine's operating system:

   Windows 7 and 8 (U.S.):

   C:\Users\username\AppData\Local\WoodWorks\CWC\USA\10\

   Windows 7 and 8 (Canadian):

   C:\Users\username\AppData\Local\WoodWorks\CWC\Canada\9\

*Note: It is considered convenient to create shortcusts for the client computers that point to the networked software. WoodWorks Database Editor (DbEdit.exe) will not work on the client computers. Any changes to the database must be made on the server computer.*

### 1.5 Registration Keycodes

When one of the Design Office programs is run for the first time, a keycode is requested. Keycodes will only be given to purchasers of the software. Those wishing to simply evaluate the program may type **demo** instead of a keycode.

To receive your keycode, email the Software ID that appears at the bottom of the keycode prompt screen to sales@woodworks-software.com. This information is also found in the "About WoodWorks..." screen which can be accessed through the help menu.

It is very important that you register with your correct email address, company name and phone number(s). This will allow WoodWorks® to send notification of updates and new releases. Please send contact information updates to sales@woodworks-software.com.

For sales related information or keycode requests, please contact the sales department at:

Email: sales@woodworks-software.com

Phone: 1-800-844-1275 ext. 1

# 2 *Sizer* Operating Modes—Beam, Column and Concept

## 2.1 About WoodWorks® *Sizer*

*Sizer* is a gravity load design tool which comes with three different programs that can be utilized for design purposes (ie. Beam, Column and Concept Modes). *Beam* mode can be utilized to design bending members up to 6 spans in length. *Column* mode can be utilized to design members which are loaded axially and laterally. *Concept* mode can be utilized to create a preliminary gravity load model of a wood structure up to 6 storeys in height. Loads can be added to each storey in concept mode, and will automatically be distributed to the storeys below. From concept mode, it is possible to refine the design of individual members by transferring them into beam or column mode. Along with the three programs, it is also possible to access the *Database Editor* which includes the standard lumber species listed in the NDS 2012 (U.S. only) or the CSA O86-14 (Canada only).[1,2] Figure 3 shows the main icons which appear when *Sizer* is opened. The icons marked by a red square, starting from the left, will lead one to *Concept Mode*, *Column Mode*, *Beam Mode* and the *Database Editor*.

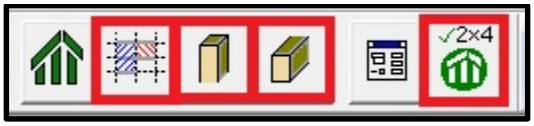



Figure 3: *Sizer Modes*

## 2.2 *Sizer* - Beam Mode

Beam mode allows you to quickly enter and design individual wood bending members. This includes up to six span continuous beams, including cantilevers. Beam mode analyses members following the stiffness method. See online help for more details on the stiffness method. To select Beam mode when first starting *Sizer*, click on the ***Beam*** toolbar button or select ***Beam*** from the ***Mode*** menu.

The main work area allows you to specify a number of parameters for your beam or joist. The toolbar allows you to quickly change between *Beam*, *Load*, *Point of Interest*, *Results* and *Diagram* views and to make *Sizer* design your beam or joist. When *Beam* mode is active, a checkmark is displayed next to ***Beam*** on the ***Mode*** menu.

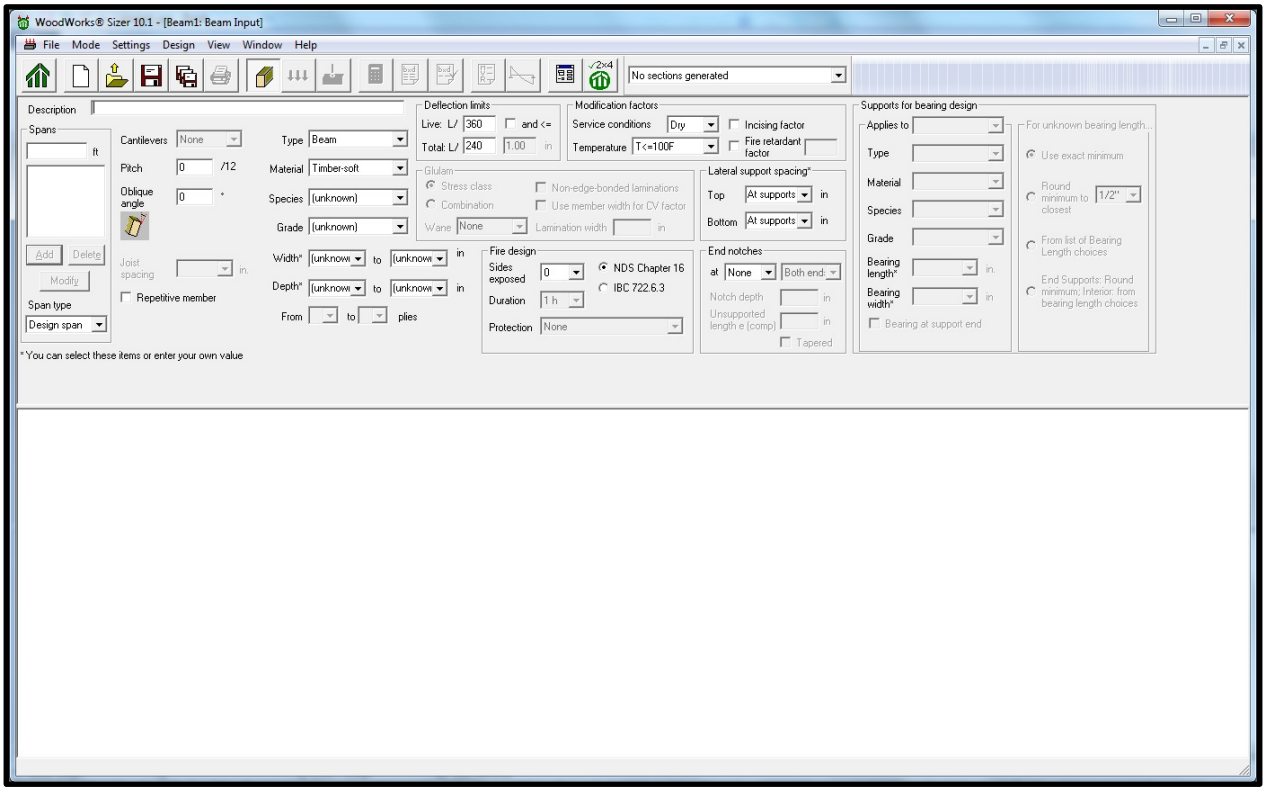



Figure 4: *Beam Mode*

The ***Beam Mode*** toolbar is the box below the menu bar and to the left with sixteen buttons titled ***create or manage project mode, New member file, Open Member file, Save Member file, Save Project/All files, Print, Beam View, Loads View, Points of Interest View, Run, Design Summary, Design Checks, Analysis Results, View Diagrams, Settings, Materials Database Editor, and Select a Section***.


Figure 5: *Beam Mode Toolbar*

## 2.3 *Sizer* - Column Mode

Column mode allows you to quickly enter and design individual wood column or beam-column members. Column mode analyzes members using the stiffness method. See the online help files. To select Column mode when first starting *Sizer*, click on the ***column*** toolbar button or select ***Column*** from the ***Mode*** menu.

The main work area allows you to specify a number of parameters. The toolbar allows you to quickly change between Column, Load, Point of Interest, Results and Diagram views and to make *Sizer* design your column or beam-column. When Column mode is active, a checkmark is displayed next to ***Column*** on the ***Mode*** menu.

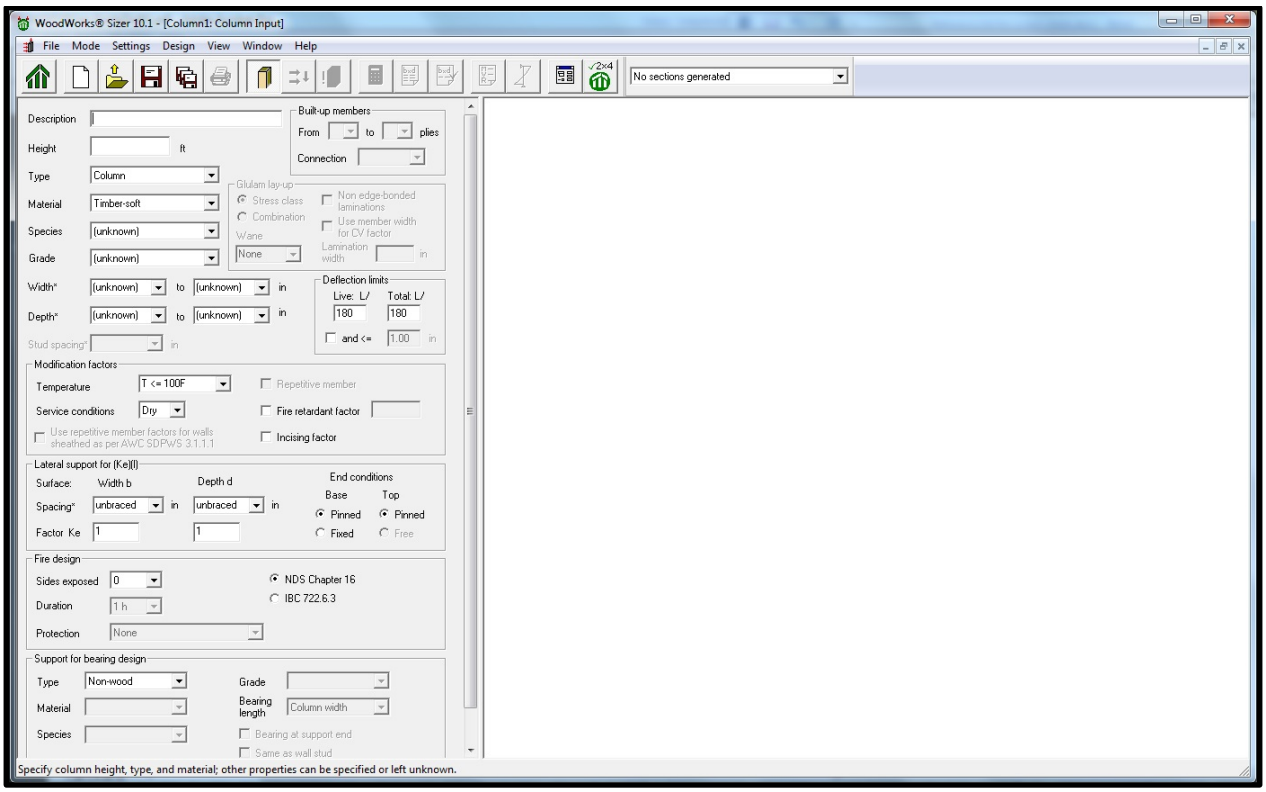



Figure 6: *Column Mode*

The ***Column Mode*** toolbar is the box below the menu bar and to the left with sixteen buttons titled ***create or manage project mode, New member file, Open Member file, Save Member file, Save Project/All files, Print, Column View, Loads View, Points of Interest View, Run, Design Summary, Design Checks, Analysis Results, View Diagrams, Settings, Materials Database Editor, and Select a Section***.


Figure 7: *Column Mode Toolbar*

## 2.4 *Sizer* - Concept Mode

Concept mode is a graphical design and analysis work area for the preliminary design of structures considering gravity loads. To select Concept mode, click on the ***Concept*** toolbar button or select ***Concept*** from the ***Mode*** menu. Concept mode provides a graphical work area where you can configure and design a complete structure in plan. The main work area contains rulers to allow you to position the cursor within the work area accurately and quickly.

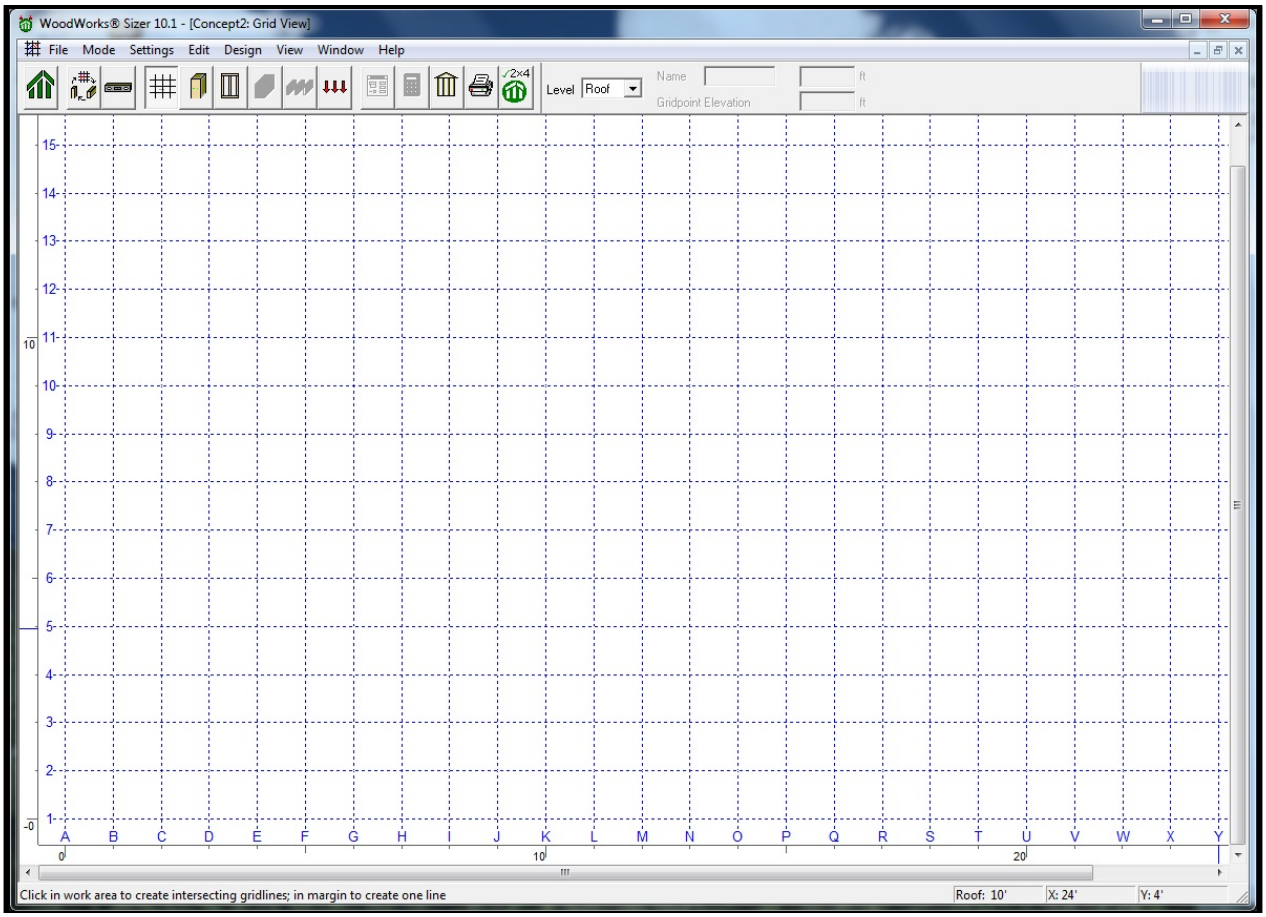



Figure 8: *Concept Mode*

The ***Concept Mode*** toolbar is the box below the menu bar and to the left with fourteen buttons titled ***Create or Manage Project Mode, Level, Grid, Column, Wall, Beam, Joist, Loads, Group, Design, Elevation View, Print, and Database***. The ***Concept Mode*** toolbar allows you to change between *Grid*, *Column*, *Beam*, *Joist* and *Load* views. The ***Run*** button makes *Sizer* design the elements in your structure and display the results.


Figure 9: *Concept Mode Toolbar*

The ***Concept Mode Data*** toolbar is the box below the menu bar and to the right of the ***Concept Mode*** toolbar. This toolbar gives a variety of information depending on the selected view. This includes information such as group names, member names, gridpoint elevations, grid line locations, load magnitudes, load locations or joist direction.


Figure 10: *Concept Mode Load Input toolbar*

The toolbar, status bar, and data bar are turned on or off through the ***View*** menu. When Concept mode is active, a checkmark is displayed next to ***Concept*** in the ***Mode*** menu.

# 3 *Sizer* Settings

## 3.1 Settings Dialog

Choose ***Settings*** from the main menu and then click ***Change*** to open the Settings dialog.

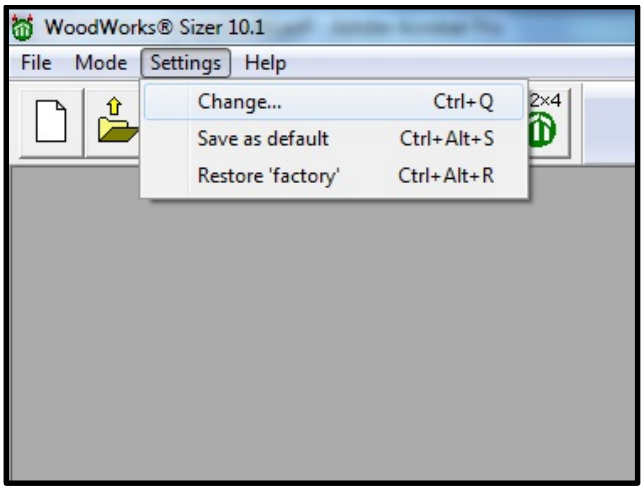



Figure 11: *Open Sizer Settings*

## 3.2 Company Information Tab

The ***Company Information*** tab allows you to enter your company contact information. You only need to enter company information once, if "Save as default for new files" is selected. This option is checked as default, but can be modified at any time.

To create a customized look, a company logo can be added through this tab. The logo will appear in place of lines 1-4 of the output.

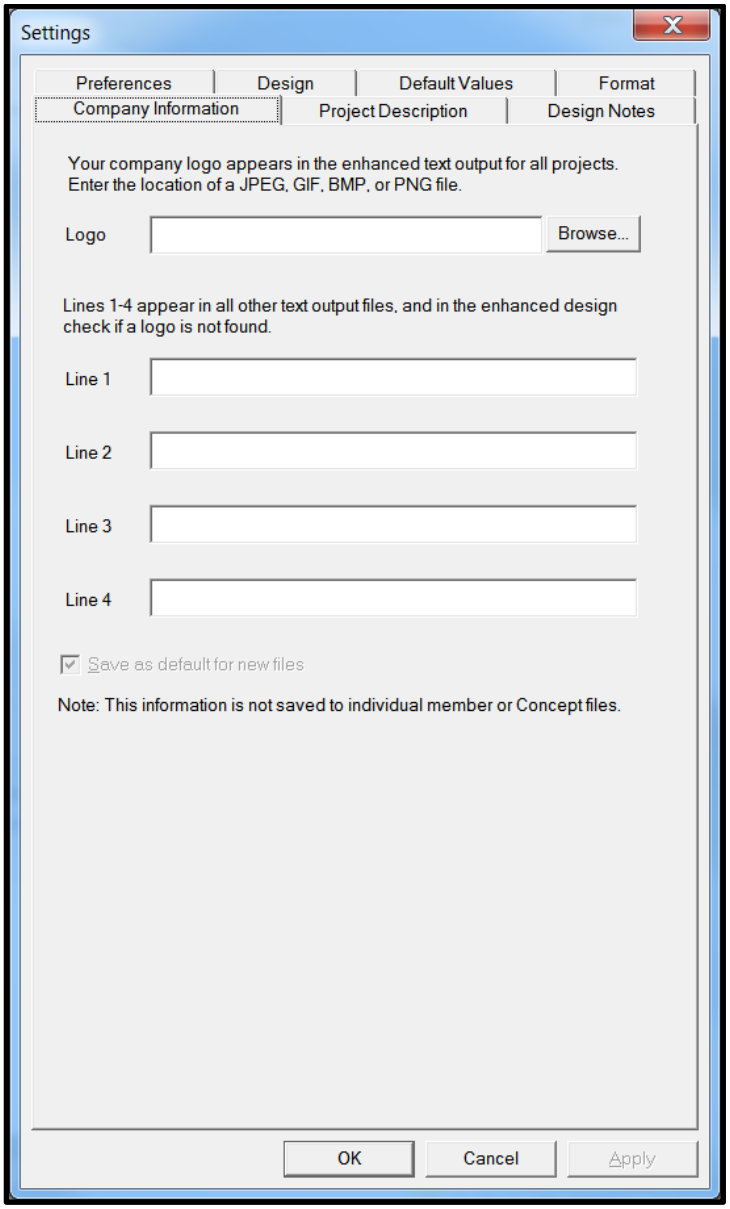



Figure 12: *Sizer Settings - Company Information Tab*

## 3.3 Project Description Tab

The ***Project Description*** tab allows you to enter project information, such as the project name, location, client, job number, etc.

Every time you start a new project, you should enter new project information.

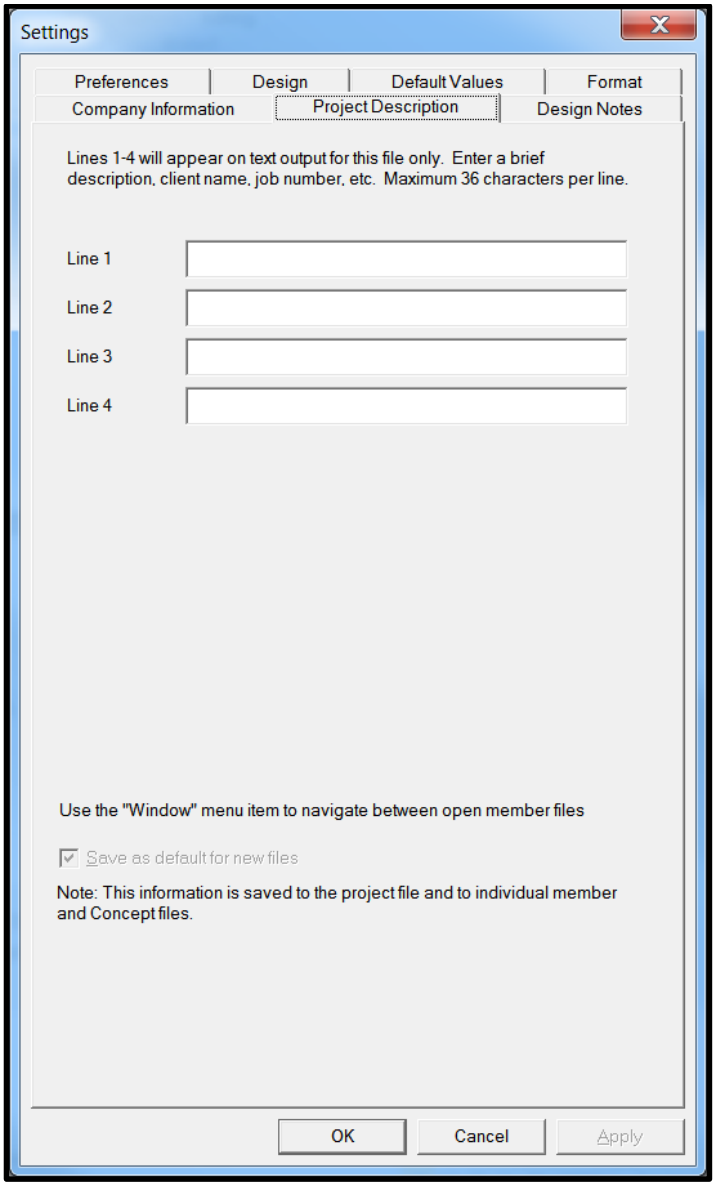



Figure 13: *Sizer Settings - Project Description Tab*

*Note: The workspace feature allows for the creation of a "Project" file meant to correspond to a typical user building project, with one concept mode file and a number of beam and column files to be open simultaneously.*

### 3.4 Design Notes Tab

Choose the *Design Notes* tab from the *Settings Dialog*. This tab allows you to specify standard design notes that are to be added to the *Design Notes* section of the *Design Check* report in *Beam* and *Column* modes. The inclusion (exclusion) of the design notes in the output can be specified by activating, or deactivating the check box to the right of the design note.

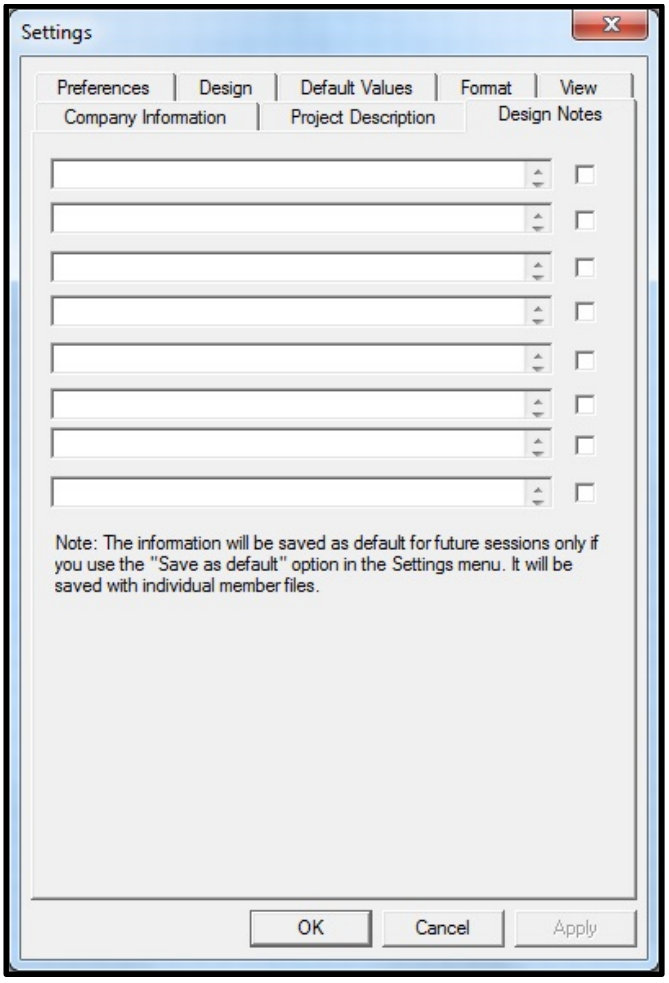



Figure 14: *Sizer Settings – Design Notes*

| | | |
|---|---|---|
| WoodWorks® SOFTWARE FOR WOOD DESIGN | **COMPANY**<br>Company Name<br>Address<br>City<br>phone and email<br>May 6, 2015 14:50 | **PROJECT**<br>Description of project goes here<br>and can be<br>up to 4 lines<br>of information<br>b1.wwb |
| **Design Check Calculation Sheet**<br>WoodWorks Sizer 9.2 | | |

Figure 15: *Company information (middle) and Project description (right) are show in the Design Check Calculation sheet. Design notes (not shown) appear at the bottom of the output.*

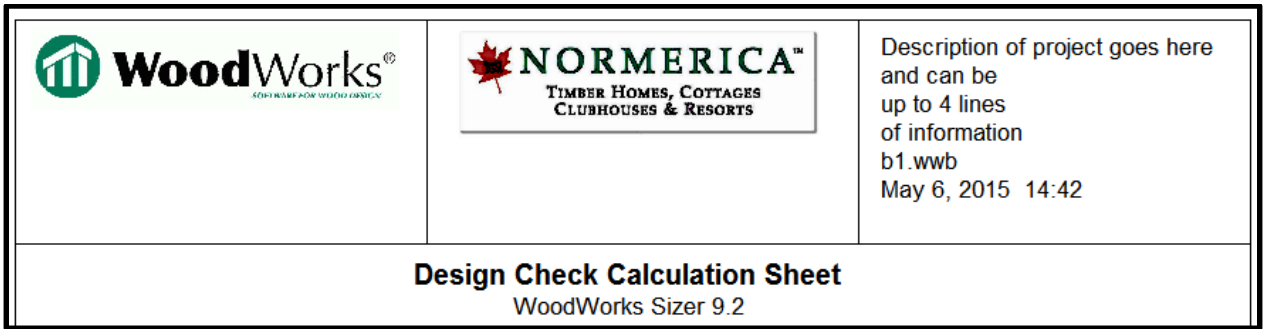



Figure 16: *The Company information is replaced with a logo.*

## 3.5 Preferences Tab

The ***Preferences*** tab allows you to make choices about the operation of the program.

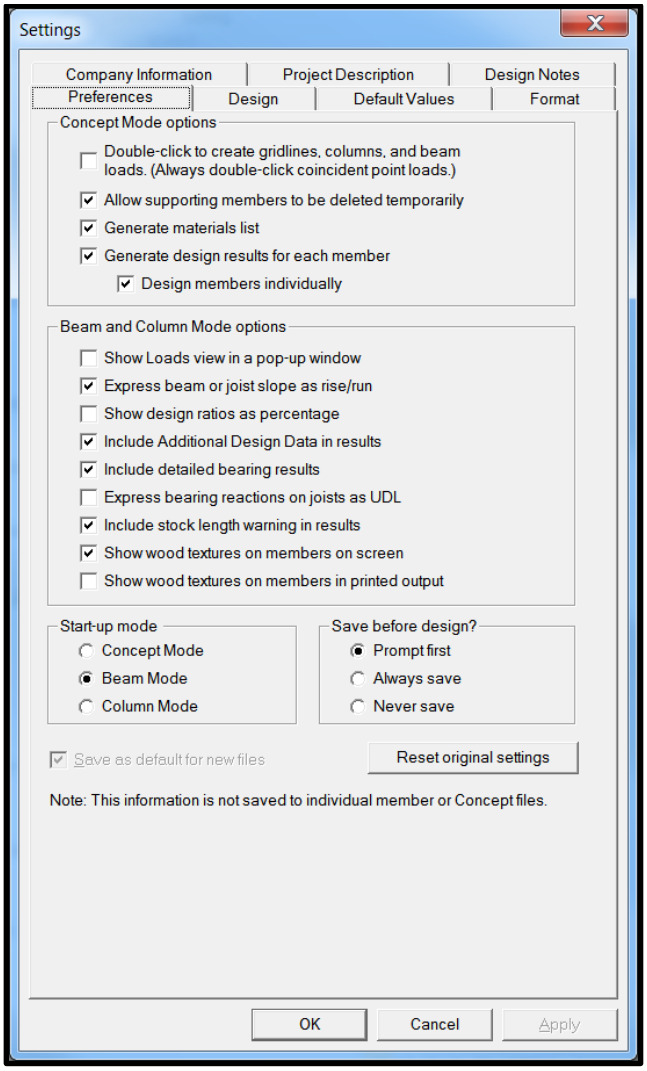



Figure 17: *Sizer Settings - Preferences Tab (Canadian)*

### 3.5.1 Concept Mode Options

These settings apply only to the *Concept* Mode and allow you to specify:

- whether to use a single or double click with the mouse,
- whether or not supporting members can be deleted temporarily while you make changes to the structure. This is useful for modelling purposes and does not affect the design,
- if a list of materials should be generated,
- whether to report Design Results for each individual member and organize the results by group or by individual member.

### 3.5.2 Beam and Column Mode Options

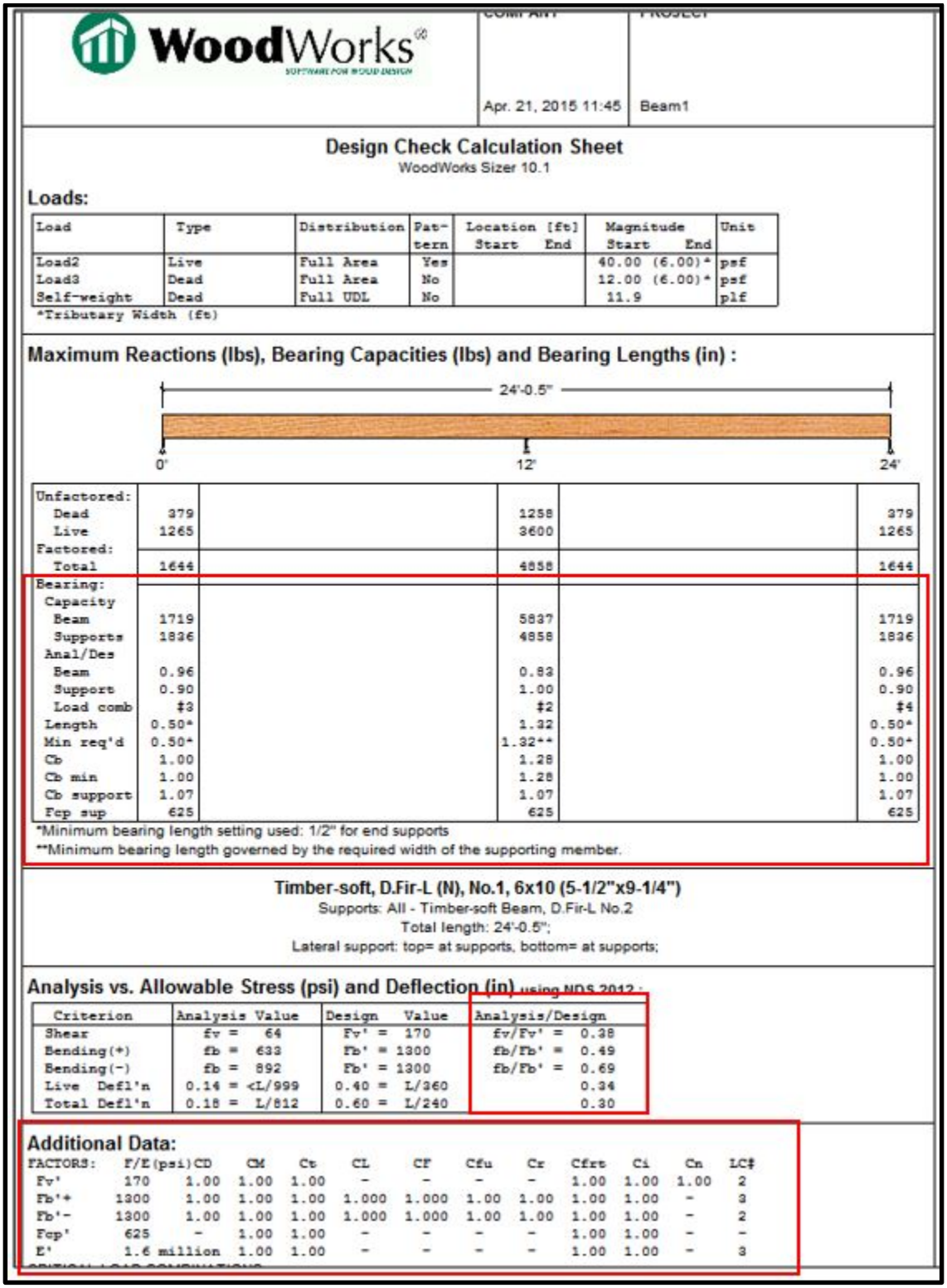
These settings apply only to *Beam* and *Column* mode and allow you to specify:

- whether to display loads in the toolbar or in a pop-up window,
- whether to enter beam or joist slopes:

a) in degrees, or

b) as a value over 12 (i.e. 4 in 12)

- whether to show the Analysis / Design as a percentage, rather than as a ratio (shown as ratio below).
- whether to include the Additional Data section (adjustment factors for reference design values) in the Design Check report (shown below),
- if you want to show detailed bearing results in the Design Check report (shown below …if not selected, only the bearing length and minimum required bearing length is shown; if selected, additional information including the bearing capacity, the governing load combination, and bearing factors are shown),
- if the joist reactions should be shown as a uniform distributed line load to the support below (ie. joist reaction divided by joist spacing), or as the individual total joist reaction, as a beam's reaction would be shown (as shown below, in lbs), whether to include the stock length warning, if the member's wood texture should be shown when viewing the computer screen, and/or when printing (shown below).



WoodWorks

Apr. 21, 2015 11:45 | Beam1

**Design Check Calculation Sheet**
WoodWorks Sizer 10.1

**Loads:**

| Load | Type | Distribution | Pat-tern | Location [ft] Start | End | Magnitude Start | End | Unit |
|---|---|---|---|---|---|---|---|---|
| Load2 | Live | Full Area | Yes | | | 40.00 (6.00)* | | psf |
| Load3 | Dead | Full Area | No | | | 12.00 (6.00)* | | psf |
| Self-weight | Dead | Full UDL | No | | | 11.9 | | plf |

*Tributary Width (ft)

**Maximum Reactions (lbs), Bearing Capacities (lbs) and Bearing Lengths (in) :**




| | | | | | |
|---|---|---|---|---|---|
| Unfactored: | | | | | |
| Dead | 379 | | 1258 | | 379 |
| Live | 1265 | | 3600 | | 1265 |
| Factored: | | | | | |
| Total | 1644 | | 4858 | | 1644 |
| Bearing: | | | | | |
| Capacity | | | | | |
| Beam | 1719 | | 5837 | | 1719 |
| Supports | 1836 | | 4858 | | 1836 |
| Anal/Des | | | | | |
| Beam | 0.96 | | 0.83 | | 0.96 |
| Support | 0.90 | | 1.00 | | 0.90 |
| Load comb | ‡3 | | ‡2 | | ‡4 |
| Length | 0.50* | | 1.32 | | 0.50* |
| Min req'd | 0.50* | | 1.32** | | 0.50* |
| Cb | 1.00 | | 1.28 | | 1.00 |
| Cb min | 1.00 | | 1.28 | | 1.00 |
| Cb support | 1.07 | | 1.07 | | 1.07 |
| Fcp sup | 625 | | 625 | | 625 |

*Minimum bearing length setting used: 1/2" for end supports
**Minimum bearing length governed by the required width of the supporting member.

**Timber-soft, D.Fir-L (N), No.1, 6x10 (5-1/2"x9-1/4")**
Supports: All - Timber-soft Beam, D.Fir-L No.2
Total length: 24'-0.5";
Lateral support: top= at supports, bottom= at supports;

**Analysis vs. Allowable Stress (psi) and Deflection (in)** using NDS 2012

| Criterion | Analysis Value | Design Value | Analysis/Design |
|---|---|---|---|
| Shear | fv = 64 | Fv' = 170 | fv/Fv' = 0.38 |
| Bending(+) | fb = 633 | Fb' = 1300 | fb/Fb' = 0.49 |
| Bending(-) | fb = 892 | Fb' = 1300 | fb/Fb' = 0.69 |
| Live Defl'n | 0.14 = <L/999 | 0.40 = L/360 | 0.34 |
| Total Defl'n | 0.18 = L/812 | 0.60 = L/240 | 0.30 |

**Additional Data:**

| FACTORS: | F/E(psi) | CD | CM | Ct | CL | CF | Cfu | Cr | Cfrt | Ci | Cn | LC# |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Fv' | 170 | 1.00 | 1.00 | 1.00 | - | - | - | - | 1.00 | 1.00 | 1.00 | 2 |
| Fb'+ | 1300 | 1.00 | 1.00 | 1.00 | 1.000 | 1.000 | 1.00 | 1.00 | 1.00 | 1.00 | - | 3 |
| Fb'- | 1300 | 1.00 | 1.00 | 1.00 | 1.000 | 1.000 | 1.00 | 1.00 | 1.00 | 1.00 | - | 2 |
| Fcp' | 625 | - | 1.00 | 1.00 | - | - | - | - | 1.00 | 1.00 | - | - |
| E' | 1.6 million | 1.00 | 1.00 | - | - | - | - | - | 1.00 | 1.00 | - | 3 |

Figure 18: *"Detailed bearing results" and "Additional Design Data" shown in the results; beam is showing "wood texture"*

### 3.5.1 Start Up Mode

Allows the choice of the three design modes—*Concep*t, *Beam*, or *Column*— to start up when *Sizer* opens.

### 3.5.2 Save Before Design

Allows the choice of saving before the design routine begins, after the component(s) have been modelled.

**Prompt first** - *Sizer* prompts you to save the current project prior to starting the design process. This is the default.

**Always** - *Sizer* always saves the current project automatically (without prompting) prior to designing.

**Never** - *Sizer* will not save the current project prior to starting the design process. There will be no prompt.

## 3.6 Design Tab

Choose the *Design* tab from the *Settings* dialog. This tab allows you to specify certain structural design-related options that are used in calculations, and whether or not fire resistance ratings are calculated.

*Note: Some of Design tab settings shown in the Canadian version of the software are available in the US version under the "Default Values" tab.*

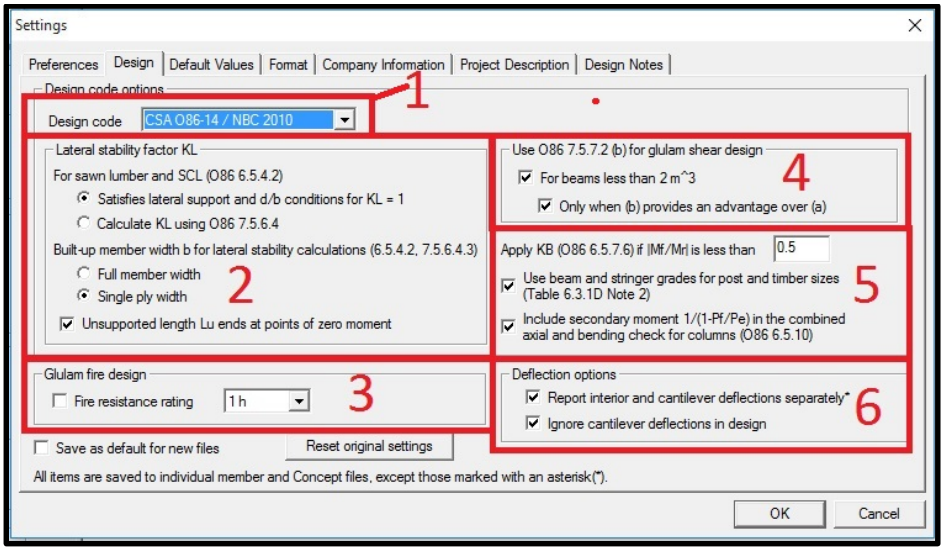



Figure 19: *Sizer Settings - Design Tab comparing Canada*

| Part | Section<br>Figure 19 (Canada) |
|---|---|
| 1 | 3.6.6 |
| 2 | 3.6.5.2, 3.6.5.3, 3.6.7 |
| 3 | 3.6.6.4 |
| 4 | 3.6.6.3 |
| 5 | 3.5.6.1 |
| 6 | 3.6.1 |

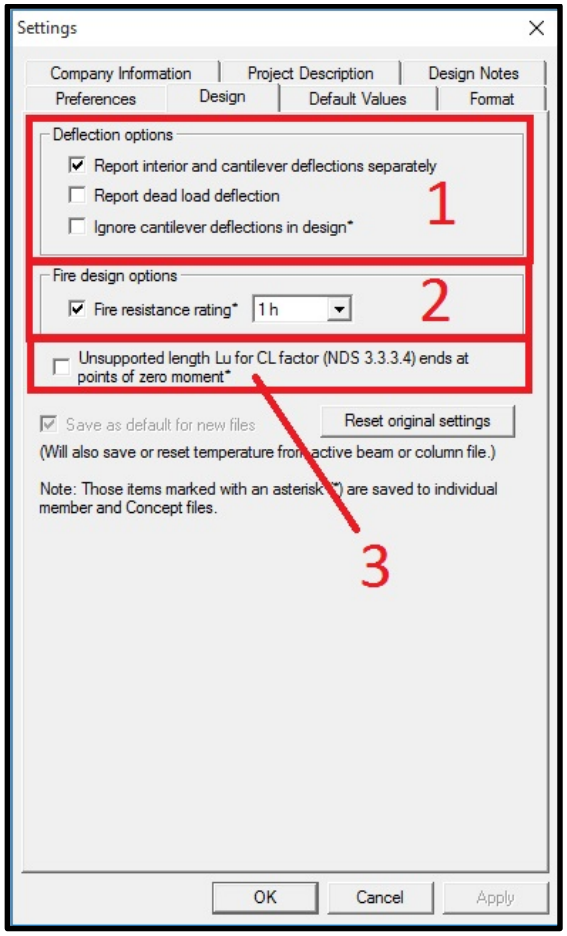



Figure 20: *Sizer Settings - Design Tab comparing U.S.*

| | Section |
|---|---|
| Part | Figure 20 (U.S.) |
| 1 | 3.6.1 |
| 2 | 3.6.2 |
| 3 | 3.6.7 |

### 3.6.1 Deflection Options

- report interior and cantilever deflection separately in Beam mode's Design Check report,
- report the dead load deflection (not shown in the above screen capture); this option is for the U.S. version only, the Canadian version always shows the dead load deflection,
- Ignore cantilever deflections in design so that the deflection of a cantilever never governs a design. For beams or joists with cantilevers, *Sizer* determines the cantilevered deflection limit by taking the specified deflection limit (for example L/360) and assigning the span, L, as equal to twice the length of the cantilever. This is consistent with most model building codes and industry recommendations. For example, for a specified live load deflection limit of L/360 the corresponding deflection limit used

for the cantilevered portion of a beam or column would be 2L/360 which is equivalent to L/180, as indicated in *Sizer*'s output, where L is now the length of the cantilever. This approach is taken because the L/xxx format is based on the deflection profile of a simply supported beam. Older versions of *Sizer* did not report the cantilevered deflection, and when the feature was implemented, the option to ignore cantilever deflection was added to the deflection design as an option to users.

### 3.6.2 Fire Resistance Rating

The fire resistance rating option permits you to specify the minimum required fire resistance in hours for solid beams and columns including sawn lumber and glulam (U.S. version). A similar feature exists for the Canadian version (the setting is towards the bottom of the input), but the design includes only glulam members, not sawn members.

If you enable the Fire Resistance feature, *Sizer* will add fire resistance as another design criterion that must be satisfied. By default, a global fire resistance (also called fire duration or fire endurance) of 1 hour is activated and can globally be modified to a different duration. The fire resistance can also be modified for each component: in Concept mode, in the Groups dialog on a group-by-group basis, and in beam and column mode, within the beam or column view inputs. The number of sides exposed to fire (0, 3 or 4) is set for each component in concept mode, beam mode, and column mode.

### 3.6.3 Minimum Bearing Length

*Sizer* provides designers with the flexibility of setting absolute limits for the minimum bearing length for both exterior and interior support locations. The values entered in these boxes indicate the smallest bearing the program considers. If the program calculates a minimum bearing less than this value, it overrides it with the value entered.

Selecting the *"Use to determine design span"* checkbox uses this absolute bearing length in the determination of beam spans, unless the calculated minimum bearing length is larger. De-selecting this option allows the design span to be based on the calculated minimum bearing length, which will be advantageous if the calculated minimum bearing length is smaller than the user defined absolute minimum bearing length because the design span, calculated from center of bearing, is reduced.

*Note: For the Canadian version, this setting is located in the "Default Values" settings tab.*

### 3.6.4 Column Eccentricity

*Sizer* allows designers to apply an eccentricity proportional to the width or depth of the column, rather than an absolute value. To apply the load eccentricity during calculations, the designer must also select the appropriate box titled "Apply auto-eccentricity % from Design Settings" in the column mode load input window.

*Note: For the Canadian version, this setting is located in the "Default Values" settings tab.*

### 3.6.5 Modification Factor Options (Canada Only)

#### 3.6.5.1 Apply $K_B$ Factor

Specified strength may be multiplied by the length of bearing factor $K_B$ based on CSA O86-14 Table 6.5.7.6 provided the conditions in clause 6.5.7.6 are met.[2] According to the standard, the length of bearing factor should not be applied if the points of bearing occur in areas of high bending stress, although high bending stress is not specifically defined. The software allows the user to specify the maximum ratio of factored bending moment to bending resistance ($M_f/M_r$) for which the $K_B$ factor will be applied, in effect, allowing for

user-determination of what a high bending stress is. As a default setting, the ratio is set to 0.5, meaning that wherever the factored moment is half or more of the moment resistance, regardless of the absolute level of bending moment, the software will consider that areas as high bending stress and not apply $K_B$.

#### 3.6.5.2 Lateral Stability Factor KL

When lateral support capable of preventing lateral displacement and rotation is provided at points of bearing for sawn lumber and SCL, the lateral stability factor KL may be taken as one, provided the maximum depth-to-width ratio meets the conditions listed in CSA O86-14 clause 6.5.4.2.[2] Stocky beams (as determined from the ratios) are laterally stable and therefore KL can be assumed to be 1.0. More slender beams, not meeting the conditions, require a calculation of the lateral stability factor according to the more detailed glulam lateral stability clause 7.5.6.4. **It is up to the software user to determine if the member meets the conditions that allow KL=1, or not.** A safer assumption, and the default assumption, is to always calculate KL based on the glulam clause.

#### 3.6.5.3 Built-up Member Width for $K_L$ Factor (CSA O86-14)

For built-up members, the maximum depth-to-width ratios in 6.5.4.2.1 used to determine if KL is permitted to be 1.0 does not apply, and instead a lower ratio (2.5:1) found in glulam clause 7.5.6.3.1 is used. To determine the depth-to-width ratio for built up members, the designer is allowed to use the total width of the beam, rather than the individual plies, but only if the individual plies are adequately fastened (see O86 6.5.4.2.2 and O86 7.5.6.3.3).[2] The program offers a choice of whether to use the full member width or the width of a single ply in the determination of the ratio. To err on the side of conservative, the default is set to basing the ratio on the width of a single ply. **Again, it is up to the software user to determine if the member meets the conditions that allow KL=1, or not.**

The selection of “single ply width” or “full member width” also is used in the calculation of the slenderness ratio $C_B$ in O86 7.5.6.4.3.[2] When “single ply width” is chosen, the value *b* used in the slenderness ratio is the width of a single ply; that is, assuming no composite action effect. This affects both the limit of 50 for the slenderness ratio itself, and also the use of value $C_B$ in calculating the lateral stability factor $K_L$ in 7.5.6.4.4.

Note Research has shown that nailed and bolted beams have at most 30% composite action effect in terms of resisting torsional buckling, and for this reason it is non-conservative to use the full member width as b in the expression for the slenderness ratio $C_B$ in O86 7.5.6.4.3 which is used to compute the lateral stability factor KL in 7.5.6.4.4.[2]

### 3.6.6 Design Code Options (Canada Only)

This subgroup of settings allow specific Canadian design standard options to be selected.

#### 3.6.6.1 Use Beam and Stringer grades for post and timber sizes…

The CSA O86-14 allows designers to use the higher beam and stringer grade strengths of Table 6.3.1 C instead of the post and timber grade values in Table 6.3.1 D, as per note 2 in O86 Table 6.3.1D, if the posts have been graded according to beam and stringer grade rules.[2] This setting allows the designer to take advantage of the added strength. The default is unchecked.

#### 3.6.6.2 Include secondary moment $1/(1-P_f/P_e)$ in the combined axial and bending check for columns…

For bilateral bending, where a column has both an axial load and a load causing bending, such as eccentric axial loads or loads perpendicular to the column (like wind), selecting the "*Include secondary moment...*" will amplify the primary bending moment. By default, including the secondary moment is selected. In fact there

are no exemptions from including secondary moments, and therefore the benefit of this option is mostly for interest.

WARNING: Combined axial and bending does not include secondary moment 1 / (1 - Pf/PE) required by CSA O86 5.5.10.

**Force vs. Resistance and Deflection using CSA-O86-09:**

| Criterion | Analysis Value | Design Value | Unit | Analysis/Design |
|---|---|---|---|---|
| Shear | Vf = 0.64 | Vr = 18.35 | kN | Vf/Vr = 0.04 |
| Moment(+) | Mf = 1.29 | Mr = 2.25 | kN-m | Mf/Mr = 0.57 |
| Axial | Pf = 55.00 | Pr = 92.99 | kN | Pf/Pr = 0.59 |
| Combined | | (Pf/Pr)^2 + (Mf/Mr) = 0.92 | | |
| Perm. Defl'n | 0.6 = <L/999 | 5.6 = L/360 | mm | 0.10 |
| Live Defl'n | 0.6 = <L/999 | 11.1 = L/180 | mm | 0.05 |
| Total Defl'n | 1.2 = <L/999 | 11.1 = L/180 | mm | 0.10 |

**WARNING: This section violates the following design criteria: Axial+Bending**

**Force vs. Resistance and Deflection using CSA-O86-09:**

| Criterion | Analysis Value | Design Value | Unit | Analysis/Design |
|---|---|---|---|---|
| Shear | Vf = 0.64 | Vr = 18.35 | kN | Vf/Vr = 0.04 |
| Moment(+) | Mf = 1.29 | Mr = 2.25 | kN-m | Mf/Mr = 0.57 |
| Axial | Pf = 55.00 | Pr = 92.99 | kN | Pf/Pr = 0.59 |
| Combined | | (Pf/Pr)^2 + (Mf/Mr)*1/(1-Pf/Pe) = 1.03 | | |
| Perm. Defl'n | 0.6 = <L/999 | 5.6 = L/360 | mm | 0.10 |
| Live Defl'n | 0.6 = <L/999 | 11.1 = L/180 | mm | 0.05 |
| Total Defl'n | 1.2 = <L/999 | 11.1 = L/180 | mm | 0.10 |

Figure 21: *Comparison of Design check results without and with the secondary moment (note: secondary moment is always required)*

### 3.6.6.3 Use CSA O86-14 6.5.7.2.1 (b) for glulam shear design

For glulam beams less than 2.0 $m^3$ in volume, the program offers a choice of using CSA O86-14 7.5.7.2.1 (a) and 7.5.7.2.1 (b) for shear design for beams less than 2 $m^3$ in volume; user selects whichever of these provides an advantage, as per the note to 7.5.7.2.1 (a), The default setting is to use (b) for all members including beams less than 2 $m^3$ if the resulting resistance $V_r$ is greater than that for (a).[2]

### 3.6.6.4 Glulam Fire Design (Canada)

The program uses the procedure in NBC Appendix D, 2.11.2, for fire design of glulam beams and columns. Selecting this setting allows you to specify whether the program designs glulam members for fire resistance, and if so, the default required fire endurance rating for new files.[3]

The fire resistance rating option permits you to specify the minimum required fire resistance in hours for glulam members (Canadian version). A similar feature exists for the US version (the setting is towards the top of the input), but the design includes sawn lumber members as well as glulam members.

If you enable the Fire Resistance feature, *Sizer* will add fire resistance as another design criterion that must be satisfied. By default, a global fire resistance (also called fire duration or fire endurance) of 1 hour is activated and can globally be modified to a different duration. The fire resistance can also be modified for

each component: in *Concept* mode, in the *Groups* dialog on a group-by-group basis, and in *Beam* and *Column* mode, within the *Beam* or *Column View* inputs. The number of sides exposed to fire (0, 3 or 4) is set for each component in *Concept* mode, *Beam* mode, and *Column* mode.

### 3.6.7 Unsupported Length Lu

By default, the software determines the unsupported length Lu, for the purpose of calculating lateral stability based on points of lateral support. In the US, this setting is related to the calculation of $C_L$ in clause 3.3.3.4 of the NDS.[1] In Canada, this setting is related to the calculation of $K_L$ in clause 7.5.6.4 of the CSA O86.[2] The toggle found in the design settings tab, will make sizer calculate lateral stability factors based on points of zero moment instead of points of lateral support.

## 3.7 Default Values Tab

Choose *Default Values* tab from the *Settings* Dialog. This tab allows Designers to specify *Default Deflection Limits for Beams, and solid floor joists*, Columns and wall studs, *Floor I-joists*, and *Roof joists*. In the Canadian edition it is possible to specify default deflection limits for *Live*, *Permanent* and *Total*.

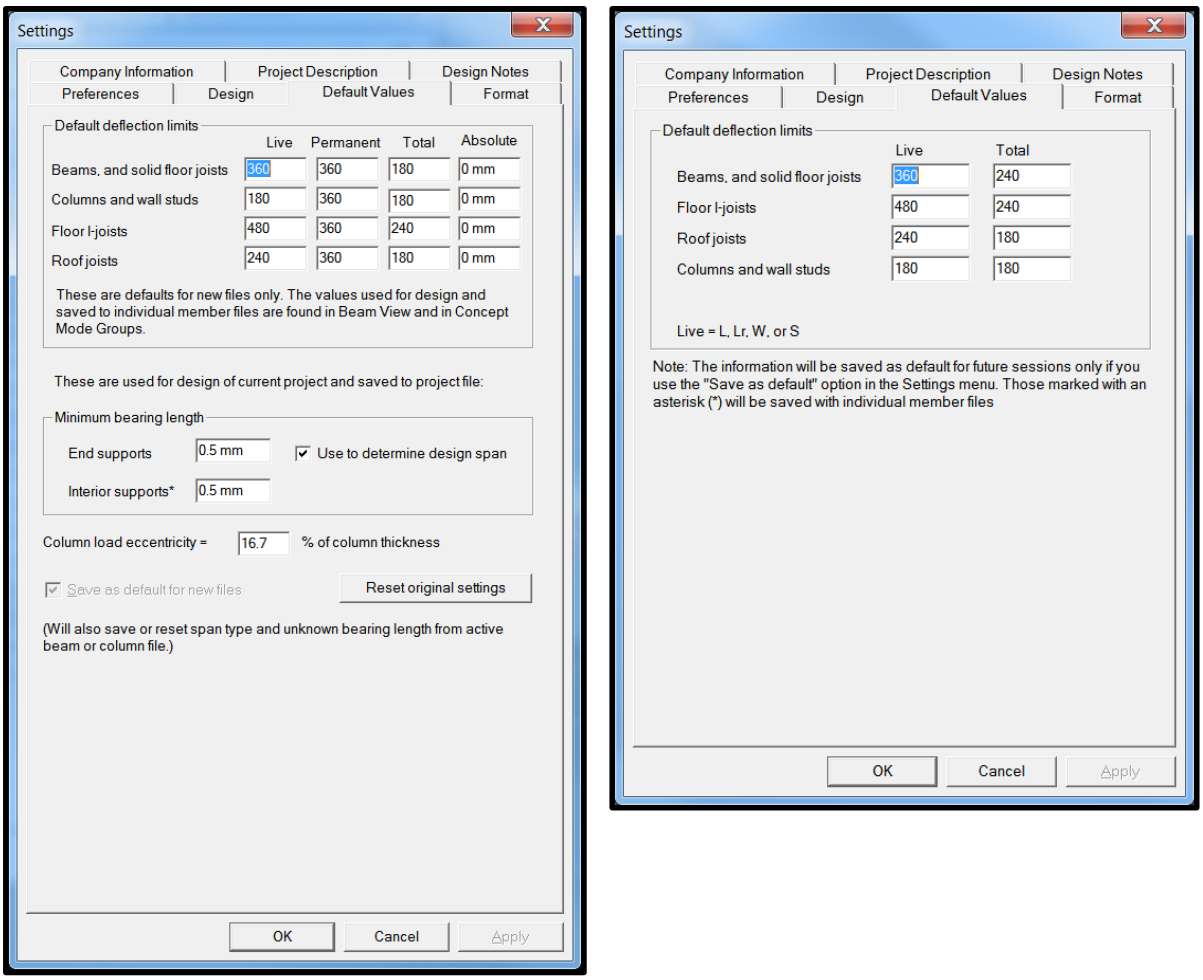



Figure 22: *Sizer Settings - Default Values Tab comparing Canadian (left) and US (right)*

loads or alternatively can input an Absolute deflection limit. In the US edition, it is possible to specify *Live* and *Total* deflection limits. For the Canadian edition only, the *Default Values* tab also includes a section to input the *Minimum bearing length* for *End supports* and I*nterior supports*. The values entered in these boxes

indicate the smallest bearing the program is able to design for, for interior and end supports separately. If the program calculates a minimum bearing less than this value, it overrides it with the value entered. A checkbox also lets the software know whether to use the minimum bearing length to determine the design span. Some users may wish that the program never design a bearing less than some practical amount, say 3.5", however they may wish to take advantage of the reduced design span from using only the minimum required bearing length. For his reason, we allow them to opt out of using the minimum bearing they entered in their calculation of design spans. The program in this case uses the minimum required bearing for design span, but the user entered span for bearing design. Also only in the Canadian edition, the *Default Values* tab includes an input to specify the percentage of column thickness to use for column load eccentricity. By default, when a column is designed *Sizer* automatically applies an eccentricity based on the input value. This feature can be turned off in *Column* mode loads view.

## 3.8 Format Tab

Choose the *Format* tab from the *Settings* Dialog. This tab allows you to enter the unit system to be used and format the font size for printing and viewing.

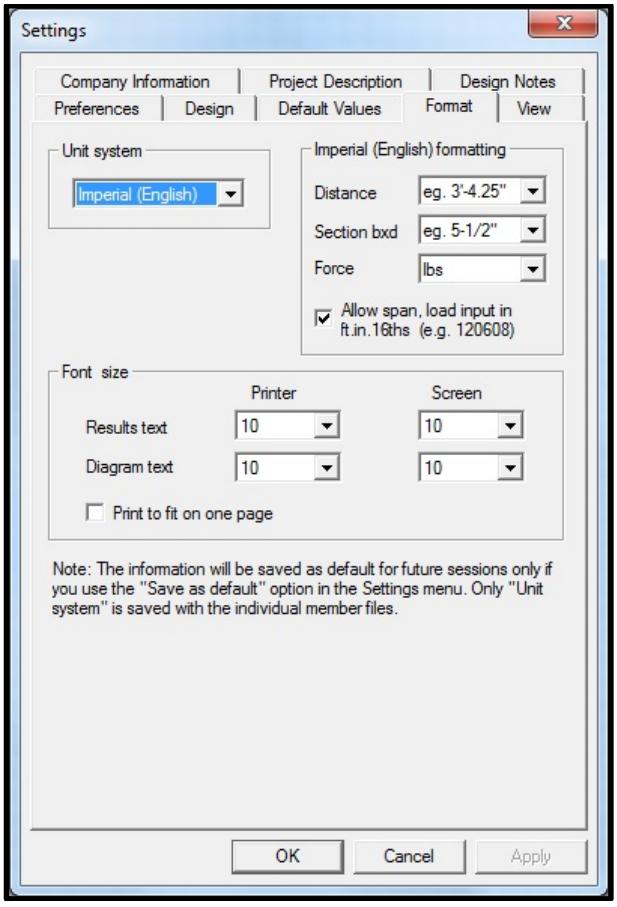



Figure 23: *Sizer Settings - Format Tab*

### 3.8.1 Unit System

This option allows you to select whether *Sizer* will operate in Imperial (English) or metric units

### 3.8.2 Imperial Length Format

Choose Imperial (English) units to be displayed in either decimal feet, in feet with decimal inches, or in feet with inch fractions. This can affect either distances or member sizes.

***Force*** is an option that permits the user to select the Imperial (or English) units for Point Loads as either *lbs* (pounds) or *kips*. This only applies to the input fields for loads – the output unit for analysis and design results are always kips.

Selecting the "*Allow span load input in ft.in.16ths"* provides designers with an easy way to enter, for example, 12' - 6 1/2" as "120608".

### 3.8.3 Font Size

This option allows you to set the font size to be used for results and diagram text for either viewing or printing.

### 3.8.4 Print to fit on one page

This is an option that automatically reduces the printing font to allow the output to fit on a single page. Note that this feature is bypassed if the font size required is less than 4 pt.

## 3.9 View Tab (Concept Mode only)

Choose the *View* tab from the *Settings* Dialog. This tab allows you to specify different viewing options such as the size of the viewing area, the snap increment, the percentage zoom, and member names. This menu is also accessed by right clicking the mouse and selecting *"View option (snap)"*

### 3.9.1 Viewing Area

These fields specify the maximum viewing area in plan for the North-South and East-West directions.

### 3.9.2 Snap Increment

This specifies the smallest increment a newly created gridpoint will move or "snap" to. For example, a snap increment of 2.0 ft allows you to place gridpoints at 10.0, 12.0 and 14.0 ft, but not at 9.0 or 11.0 ft.

### 3.9.3 Display

These options allow you to include additional information on the diagrams for viewing or printing. This includes the snapped coordinate, the member names, and the group names. You can also specify whether snapped or actual mouse co-ordinates appear in the status bar.

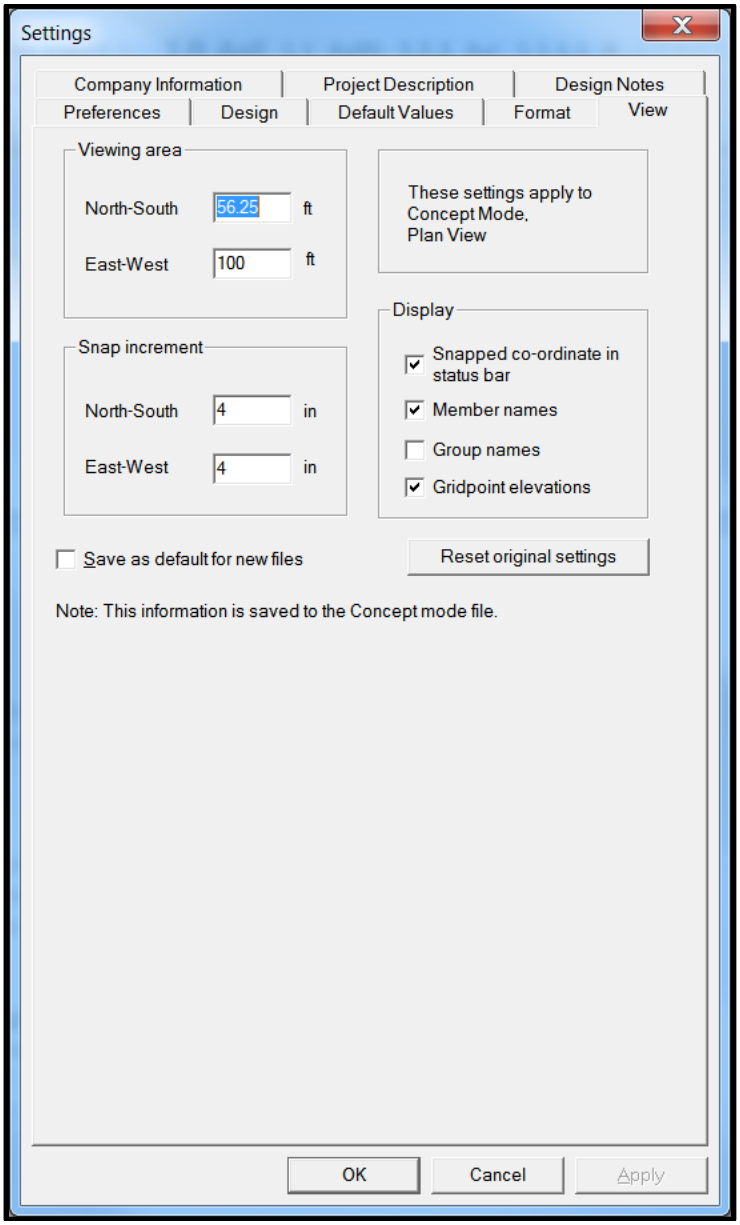



Figure 24: *Sizer Settings – View Tab*

### 3.9.4 Save As Default

Choose ***Save new settings*** under the ***Settings*** menu. This will save any new changes that have been done in the Settings menu. These new settings will be the default for any new files that you create. This custom definition will again be in effect during the next *Sizer* session. However, a previously saved *Sizer* file with different definitions will over-ride these settings. Clicking ***File*** and then ***New*** will restore your definitions at any time.

### 3.9.5 Restore Original Settings

This option restores the original settings that were in effect when *Sizer* was first installed. They will only be in effect for the current session unless you click on ***Save New Settings***.

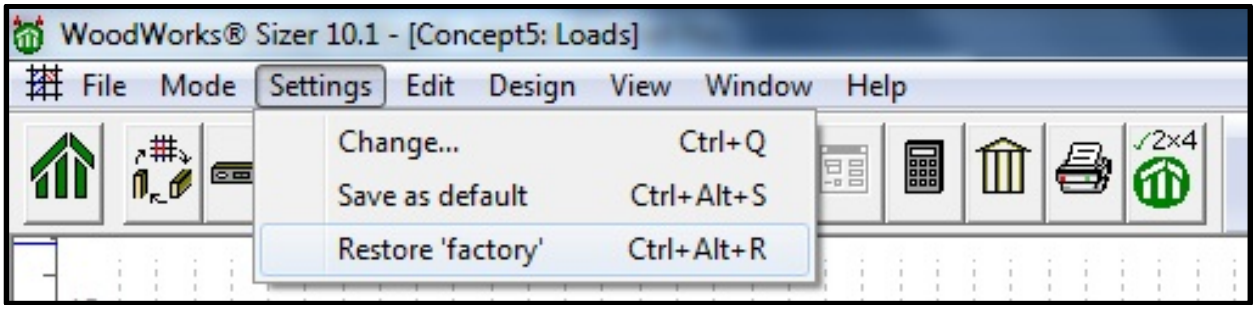



Figure 25: *Restore 'factory' settings*

# 4 *Sizer* Working with Files and Projects

## 4.1 Creating New Files

When you start *Sizer*, it displays an untitled screen, allowing you to create a new file.

Whenever you choose ***New File*** from the ***File*** menu, *Sizer* clears the screen, discarding the current project. If you have modified the current project since you last saved it, *Sizer* asks you whether to save or discard your modifications.

## 4.2 Creating New Projects

In the file menu, there is an option of creating a new project (this can also be done by clicking on the ***Create or Manage Projects*** button on the toolbar), which allows several files to be grouped together from *Sizer* to form a project. When creating a new project, *Sizer* will prompt you to enter a project description that will be shared by each file in the project. The project can contain only one concept mode file and a number of beam and column files.

## 4.3 Saving Files

If you choose ***Save File*** from the ***File*** menu, *Sizer* saves the current file using the last name you gave it. If the file is a new one and you have not given it a name, the ***Save File*** command has the same effect as ***Save File As...*** (see below).

If you choose ***Save File As...*** from the ***File*** menu, the current file is saved with the name you enter. In the *Save As* dialog box, the *File Name* field contains (by default) the last name given to the file. You can specify a different name if you wish.

To change the file type, click the down arrow, select a file type and double-click it.

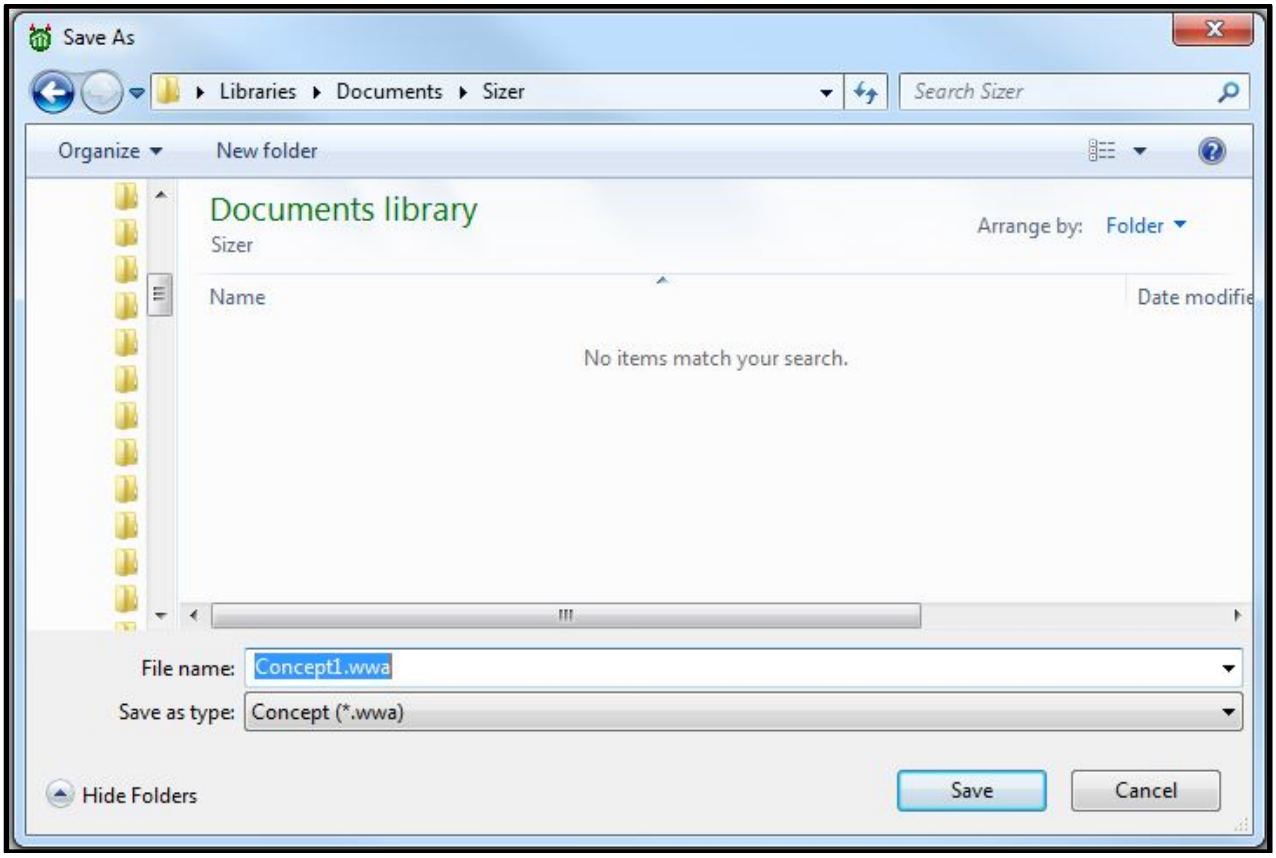



Figure 26: *Save file as*

To change folders, select a folder from the Folders list and simply double-click the folder.

Click on ***Save File As...*** to save the file under the new name. If a file already exists with the name you specified, *Sizer* asks you to confirm that the existing file should be overwritten. Click on ***Cancel*** to exit the ***Save File As...*** dialog without saving your file.

If you choose ***Save Project*** from the File menu, *Sizer* saves the current files in the project with the last name you gave it. If the project is new and or has not been given a name, the ***Save Project*** command has the same effect as ***Save Project As...***

If you have several files open (beam, column and a concept) and wish to save all the open files into a new project, simply click on the ***Create or Manage Project*** button on the toolbar.

The ***Save All*** button has the same effect as ***Save*** for all the open files.

## 4.4 Opening Files

To open an existing project, choose ***Open File*** from the ***File*** menu. The Open dialog appears. Type a file name (or select one from the list) and then click ***Open*** to open that file.

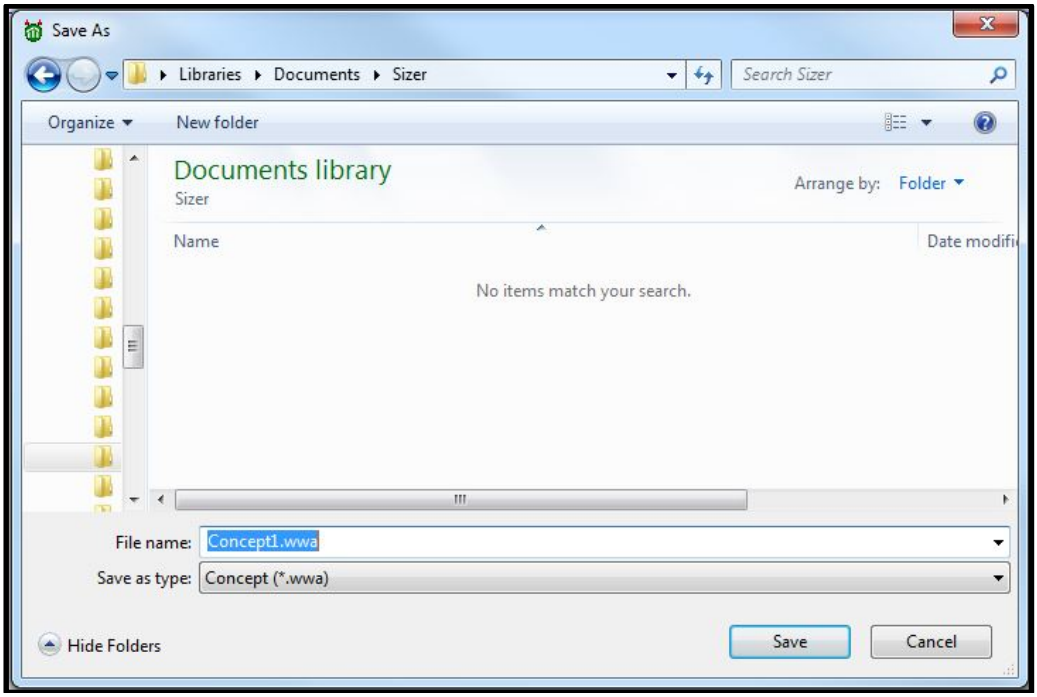



Figure 27: *Open File*

If you select a new folder in the *Folders* list, the selected folder becomes the current folder. Select ***Cancel*** to return to editing the current file without opening a new one.

## 4.5 Opening Projects

To open an existing project, select *Open Project* from the *File* menu. The *Open* dialog appears and the file type is .wprj. When a project is opened, all the files within that project are opened.

## 4.6 Printing

To print a file or on-screen graphics, click on the ***Print*** button from the toolbar or choose ***Print*** from the ***File*** menu. This will print the text or graphics image that is currently shown on screen. Depending on the mode you are in, you will have the choice of printing various text and graphics files as described below.

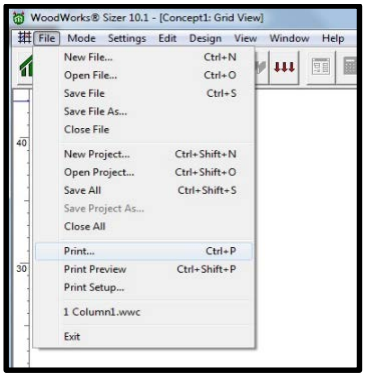



Figure 28: *Printing a file*

# 5 *Sizer* Beam Mode

## 5.1 About *Sizer* Beam Mode

Beam mode allows you to quickly enter and design individual wood bending members. Beam mode provides extensive information about a single beam or joist in your project. Typically you would use *Beam* mode to verify the design of critical members.

To select *Beam* mode, choose ***New File*** from the ***File*** menu and select ***Beam Mode*** or click on the ***Beam*** button on the toolbar.

When *Beam* mode is active, a checkmark is displayed next to ***Beam*** on the ***Mode menu***.

Often Beam mode is used on its own to specify or check bending members. This will be elaborated upon later, but in summary, entering the loads, beam or joist properties (some of which can be left as 'unknown' allowing the software determine the required size and strength), and support conditions is sufficient to determining adequate beam sizes. However, sometimes it is useful to lay out some or all of the wood components, put on the loads, and let the software distribute the loads to the members so you don't have to. This is particularly useful for more complex geometry like hipped roof systems, where the slope of the hip and the triangular load distribution is more difficult to calculate. In this case, using *Concept* mode (described later in more detail) to lay out the components and then transferring individual members and their loads into beam mode for further analysis is beneficial.

## 5.2 Transfer of a Beam or Joist from Concept Mode

If you are operating in Concept mode and you wish to further analyze a particular beam or joist, click on the beam or joist to select it and then choose ***Beam*** from the ***Mode*** menu or click the ***switch modes*** button on the toolbar.

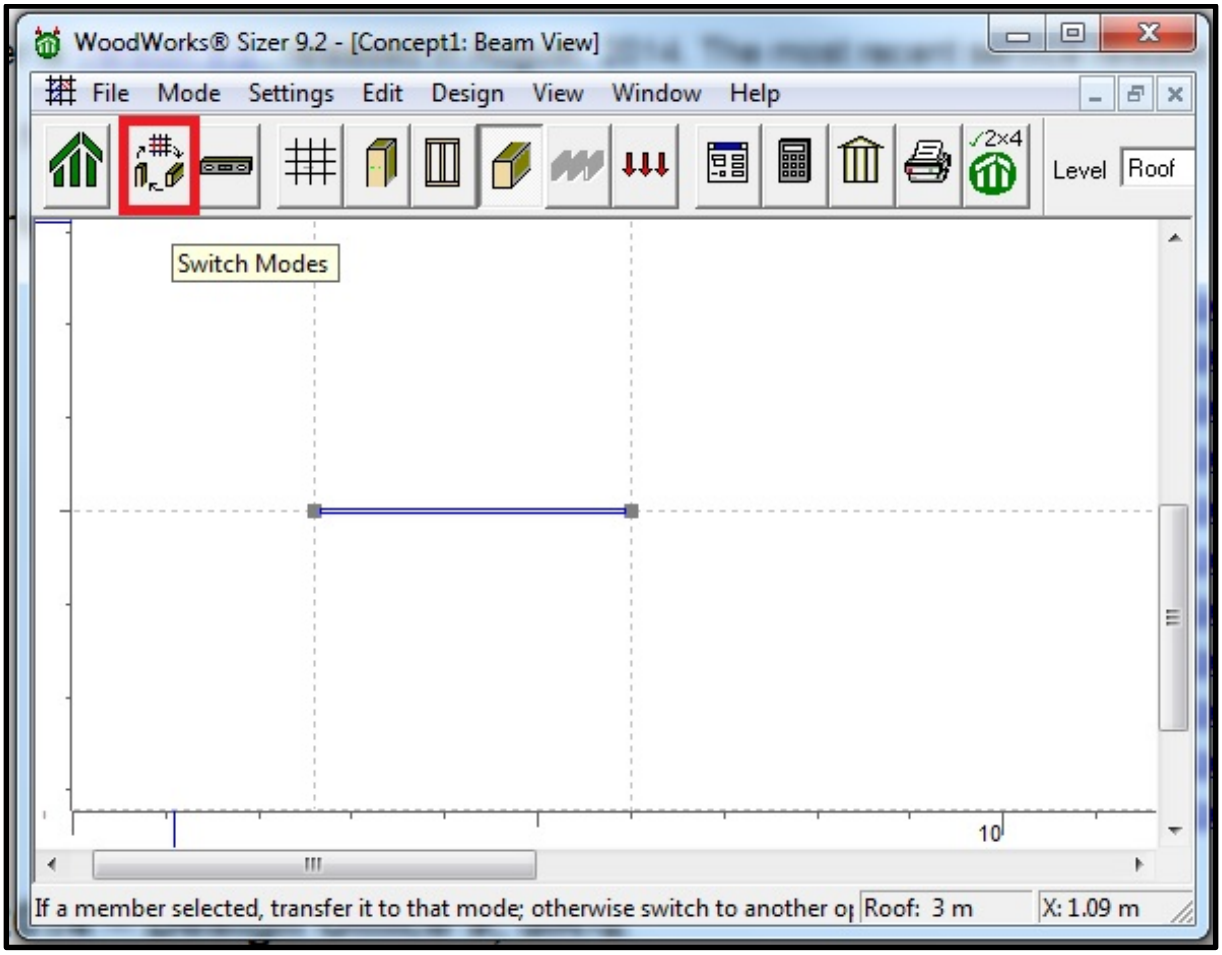



Figure 29: *Transferring information from Concept mode to Beam mode*

*Concept* mode distributes the user-input design loads to each member. At the click of a button, the member and load information is transferred to the input fields in *Beam* mode where additional design criteria (such as notches and bearing details) can be input and designed for.

*Note: The transfer from Concept to Beam mode is generally one-way. Any changes made to the transferred member cannot be exported back to the original member in Concept mode. There are a few exceptions, such as the selection of* ***Self Weight****: "manually input " or "automatically included in loads analysis". Refer to* ***Help*** *on "Apply options to Concept Mode" which refers to what options can be transferred with a click of a button in the* ***Loads V****iew, activated if Concept mode is open.*

Figure 30 shows the beam input for the Canadian edition of the software and Figure 31 shows the beam input for the U.S. edition of the software.

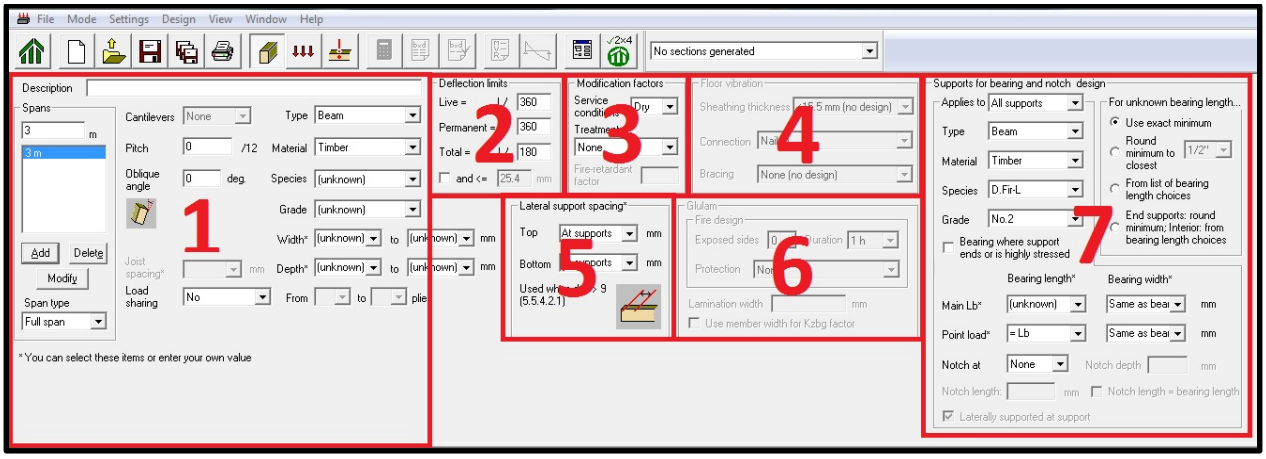



Figure 30: *Beam Mode Input Menu (Canadian Edition)*

The following Table Summarizes where information on each Part of Figure 31 can be found. The section numbers have been hyperlinked for convenience:

| | Section |
|---|---|
| Part | Figure 30 (Canadian) |
| 1 | 5.4 |
| 2 | 5.5 |
| 3 | 5.6 |
| 4 | 5.14 |
| 5 | 5.8 |
| 6 | 5.10 |
| 7 | 5.13 |

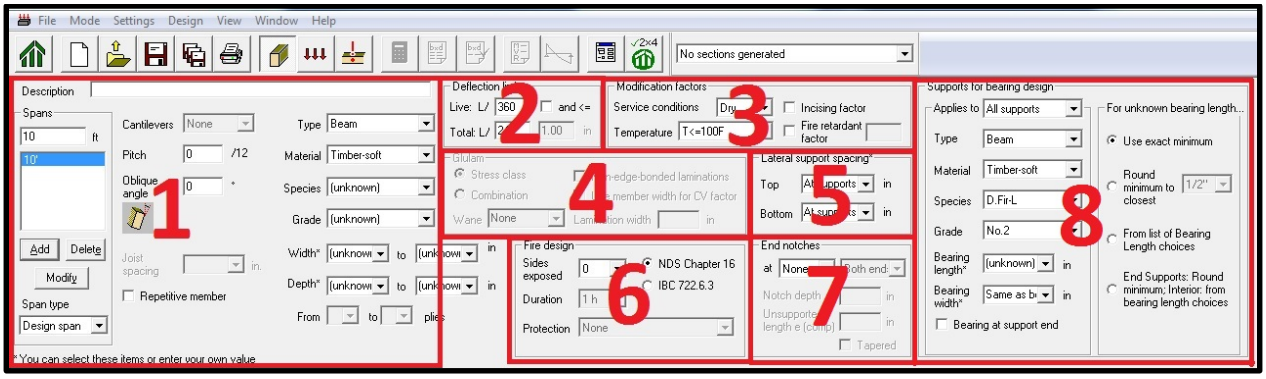



Figure 31: *Beam Mode Input Menu (U.S. Edition)*

The following Table Summarizes where information on each Part of Figure 32 can be found. The section numbers have been hyperlinked for convenience:

| | Section |
|---|---|
| Part | Figure 31 (U.S.) |
| 1 | 5.4 |
| 2 | 5.5 |
| 3 | 5.6 |
| 4 | 5.9 |
| 5 | 5.8 |
| 6 | 5.11 |
| 7 | 5.7 |
| 8 | 5.12 |

## 5.3 Transfer of Beam or Column from Autodesk's Revit® to *Sizer*

WoodWork® *Sizer* is capable of interfacing with cadwork® and Revit®. If you have a structure designed in Revit®, you can import it into *Sizer* for specific design verifications. For more information, visit our website at www.woodworks-software.com.

## 5.4 Beam (or Joist) Parameters

Click the ***beam*** button on the toolbar to describe the beam (or joist) to be designed.

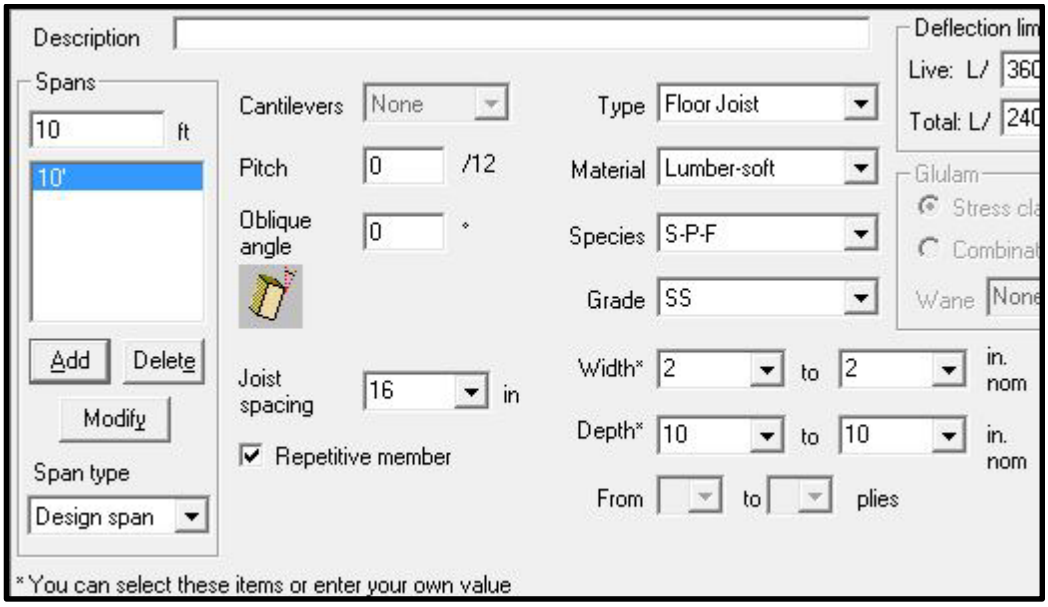



Figure 32: *Beam Mode Input*

*Sizer* displays a diagram of the spans in the lower portion of the screen. The Design Span, Clear span, and Full span are all shown on the diagram. *Sizer* can also display cantilevered and sloped beams.

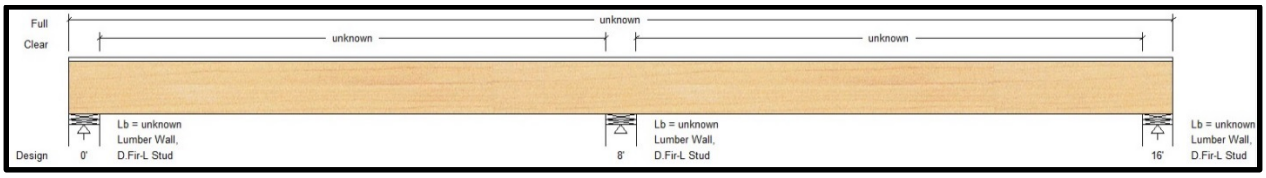



Figure 33: *Beam Mode Display*

### 5.4.1 Description

A description of the beam can be entered which will appear in the *Design Results* output and *Design Check* output. This is over and above the *Project Description* described earlier, and is a useful way to describe the specific component being designed.

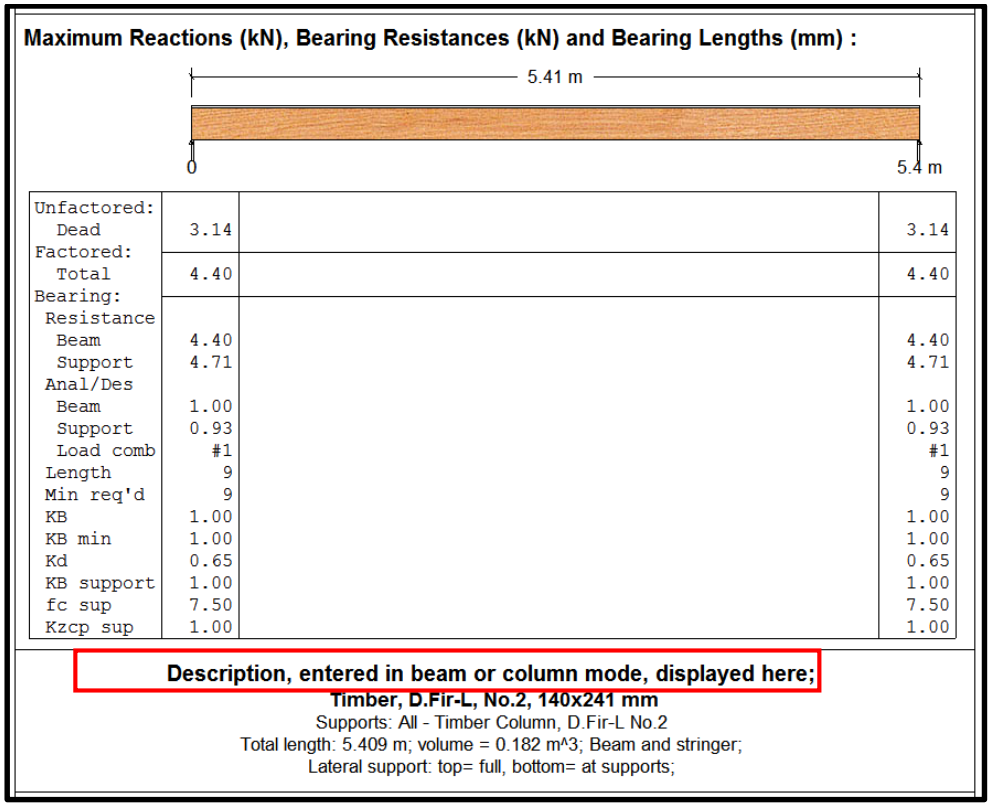



Figure 34: *Description line shown in Design Check*

### 5.4.2 Spans and Span Type

This list box defines (in left-to-right sequence) the length of each span and cantilevers. Before entering spans it is important to understand what type of span is being entered. Immediately below the *Add, Delete* and *Modify* is a "Span type" entry where Design, Full, or Clear span can be selected. "Design span" has been the traditional method used to enter spans in *Sizer*, and is the distance between support points on beams. Support points for exterior supports are ½ the minimum required bearing length from the inside of the support. For interior supports, they are at the centre of the support. The clear span is the distance between the inside faces of the supports, for all spans. Full span is measured from the outside faces of exterior supports, to the centre of interior supports. It is the beam length for single-span members. The graphical representation of the beam accurately illustrates these span types using dimensions.


To change the length of the span, select the span in the list of spans, change the length in the spans box, and then click ***Modify.***

To delete spans, click the span you wish to delete and then click ***Delete***.

In the case of Figure 36, the “Design Span” was used to specify a 1m design span for each beam segment. This translates to a 0.93m clear span, and 1 total full span of 2.28m. If “Clear Span” was selected to enter the 1m length, the spans shown as 0.93, would have been 1.0m. If “Full Span” was selected, the total length of beam would have been 2.0m.

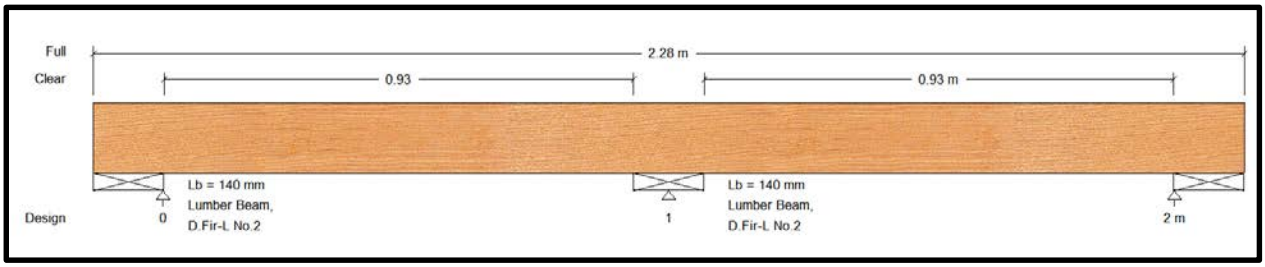



Figure 35: *Span type selection options*

### 5.4.3 Cantilever

This drop-down list is only active when two or more spans have been added and it specifies cantilevers at either end of the beam. The outer end of a cantilevered span is unsupported. The default is none (no cantilever at either end).

### 5.4.4 Slope/Pitch

This field specifies the slope of the beam in degrees or as a pitch. A horizontal beam (the default) has a slope of zero. A beam that slopes up to the right has a positive value. A beam that slopes down to the right has a negative value.

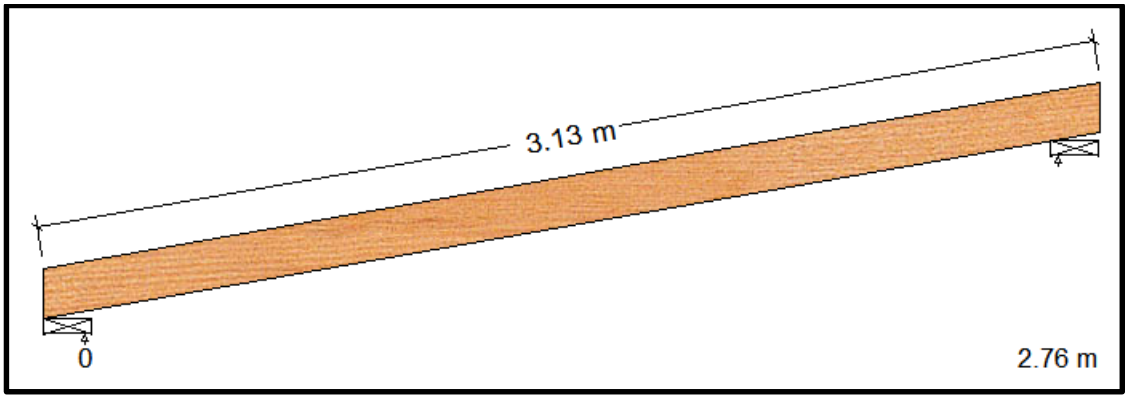



Figure 36: *Slope of 3:12*

Level bearing is assumed and thus there is no horizontal thrust at supports. Sloped members are only designed for flexure. It is assumed that axial force is insignificant in these members.

Joists supported by sloped members are spaced with respect to the longitudinal axis of the sloped member.

### 5.4.5 Oblique Angle

This field specifies the angle for oblique purlins. The default is blank (no angle). By specifying an angle of 90°, you can investigate the use of the selected material database as a plank.

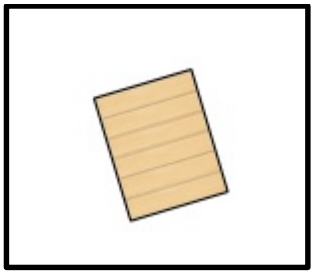


Figure 37: *Oblique angle purlin (representation)*

### 5.4.6 Joist Spacing

Active for joists only, this drop-down list specifies the joist spacing. You can select one of the standard 12 in. (300mm), 16 in. (400mm), or 24 in. (600mm) spacing or enter your own value. The tributary width for uniform area and partial area loads in *Loads View* is equal to the joist spacing set in the *Joist Spacing* field.

### 5.4.7 Repetitive Member (U.S.)/ Load Sharing Factor (Canadian)

The benefit of sharing loads among closely spaced members can be specified in *Sizer*:

U.S. Users have the option of selecting whether to apply the repetitive member factor ($C_r$) when calculating the allowable stress for beams.

Canadian Users have the option of selecting whether to apply the Load sharing factor ($K_L$) when calculating the capacity for beams.

### 5.4.8 Type

This drop-down list specifies the type of bending member to be designed - beam, floor joist or roof joist. The default selection is **Beam**.

### 5.4.9 Material

This drop-down list specifies the material database to use. Standard choices for the Canadian and U.S. editions of the software can be seen in the Figure below. The software includes the standard material types included in the NDS or CSA O86.[1, 2] Some proprietary products are also included with the installation of the software. It is possible to create databases for proprietary products using the *Database Editor*. Should you use a specific proprietary product manufacturer it is recommended to contact them to see if they have a WoodWork® custom database available or would be willing to create one. It is also recommended that you inform them that they should contact WoodWork® Support should they wish to include their product with the installation of the software.

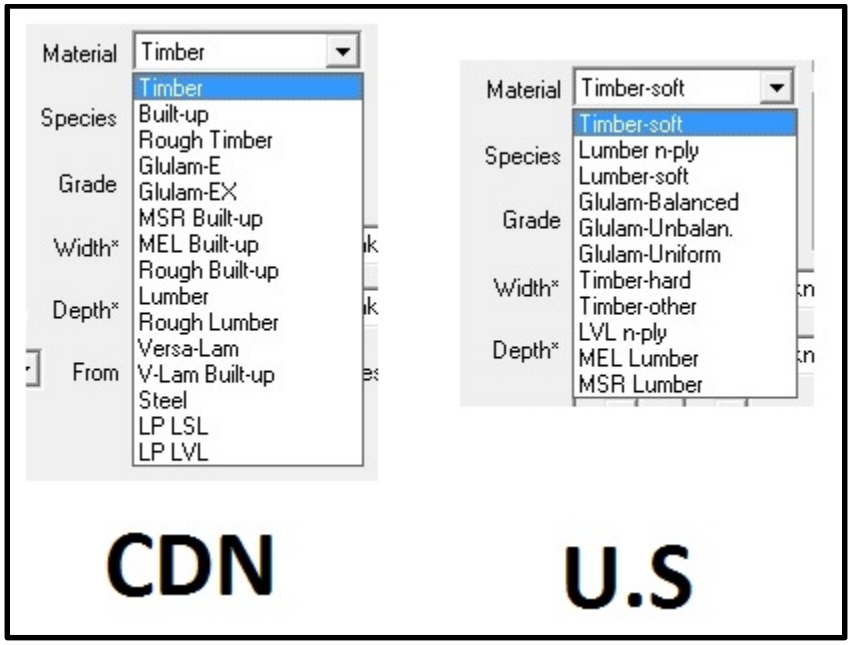



Figure 38: *Default Materials (Canadian and U.S. Editions)*

### 5.4.10 Species (or Strength)

This drop-down list specifies the species of wood (and steel strength, Canada only) to use for member design. Available wood species and steel strength depends on the selected material.

The default selection is (*unknown*), which forces *Sizer* to select suitable sections from each species.

### 5.4.11 Grade (or Combination, or Species Group) US version

This drop-down list specifies the sawn lumber grades (or glulam combinations or glulam species groups) to use to design the member. Available grades, combinations, and species groups depend on the selected database.

The default selection is (*unknown*), which forces *Sizer* to select suitable sections from each grade.

This list box is unavailable if the species is specified as (*unknown*).

### 5.4.12 Grade (or Comb'n, or Shape) Canadian version

Similar to the US version, this drop-down list specifies the sawn lumber grades (or glulam combinations) use to design the member. In Canada when steel material is selected, the shape of steel beam (W shape) is displayed. Available grades and combinations depend on the selected database.

### 5.4.13 Width and Depth

The two width drop-down lists specify the width range of the sections *Sizer* should use when designing the member. Similarly, the two depth drop-down lists specify the depth range. Available widths and depths depend on the selected database, species and grade.

The default selection for each of these drop-down lists is (unknown), which forces *Sizer* to select from a full range of section sizes.

You can limit the section sizes to be considered by selecting appropriate width and depth ranges in the drop-down lists.

These drop-down lists are unavailable if the grade is (*unknown*).

### 5.4.14 Built-up Members

If the selected material can be used in built-up or multi-ply sections, these two drop-down lists specify the range of plies to use. The default selection is *(unknown)*, which forces *Sizer* to select suitable sections for a range of plies.

If the material cannot be used in multiply sections, these lists are disabled.

*To specify a custom section, enter a non-standard size (in actual dimensions) in each drop-down list. Note that a custom size may not be commercially available.*

*Note: Sizer performs a **design check** rather than a **design selection** if both the width range and depth range specify single values. Sizer cannot design a section if only the width or depth is a custom size. For example, you cannot have one field as (unknown) when the other contains a custom size.*

## 5.5 Deflection Limits

These fields allow you to specify the deflection limits to be used for design, based on the span (example L/360) and in absolute terms (1 inch). Default deflection limits are entered in Settings [Design] tab. See also the "Loads" button and "Load types and combinations" for long term deflection (creep) and load reductions (U.S. only).

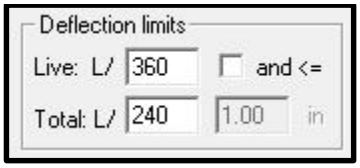



Figure 39: *Deflection Limits Input (U.S.)*

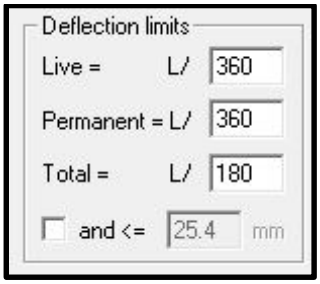



Figure 40: *Deflection Limits Input (Canadian)*

## 5.6 Modification Factors

### 5.6.1 Fire-retardant Factor (U.S. Only)

This field permits you to specify the reduction in bending, shear, and deflection resistance of wood treated with fire-retardant chemicals. In the U.S. version, to specify a value, click the check box.

### 5.6.2 Incising Factor (U.S. Only)

In the U.S. this field allows you to specify whether an incising modification factor should be applied according to NDS 4.3.8.[1]

### 5.6.3 Treatment Factor (Canadian Only)

In Canada, users are given the option of selecting None, Preservative, or Fire-retardant treatment options from a drop-down list.

### 5.6.4 Service Conditions

This drop-down list specifies either Wet or Dry service conditions for the beam. The default is ***Dry***.

### 5.6.5 Temperature (U.S. Only)

The default is T<100°F. Other values for T are 100°F<T<125°F and 125°F<T<150°F.


Figure 41: *Modification Factors (U.S.)*


Figure 42: *Modification Factors (Canadian)*

## 5.7 End Notches (U.S. Only)

*Note: End notches for the Canadian Edition are explained later in Supports for bearing and notch design.*

*Sizer* only designs notches at the ends of a beam. Notches can be specified as being on the **top** or **bottom** of a beam. Both notch depth and length are required as input. Notches can be specified as being on the top or bottom of the beam. Users also have the option of specifying whether the End notches are at both ends, the left end, or the right end. Both notch depth and length are required input.

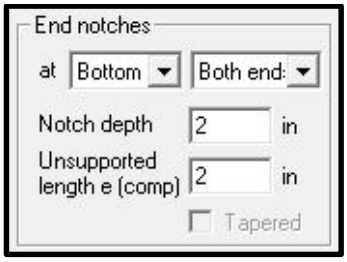



Figure 43: *End-Notch Factors Input*

## 5.8 Lateral Support Spacing

These drop-down lists permit top and bottom support to be specified as "***Full***" (Full Lateral Support), "***At supports***" (Lateral support provided at bearing supports only), or at a specified ***numeric interval*** (specified in inches). The selection of the lateral support determines the lateral support factor ($K_L$ (Canadian) and $C_L$ (U.S.)). The default values depend on the component "Type". For roof joists and floor joists, by default the software specifies "***Full***" for the top and "***At Supports***" for the bottom, assuming sheathing is providing full support along the top, but assuming no sheathing at the bottom of the joists. For beams, both top and bottom are by default "***At Supports***", however the beam may actually be supported at specific intervals by joists, say 4ft on center. Either "***48***" or "***4'***"can be entered. What constitutes "lateral support" is left up to the designer - the size of lateral supports and the connection of the supports to the beam are an important factor.

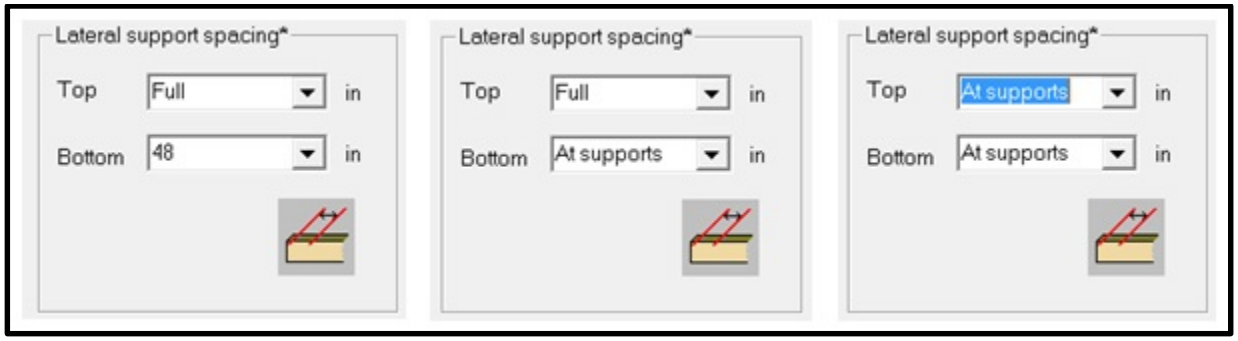



Figure 44: *Lateral support input: Full, At supports, and numerical*

The user is responsible for modifying the selections to match actual conditions if the defaults are not appropriate. Notes in the output indicate the lateral support used in the calculations.

*Note that the factor for lateral support based on the user-specified conditions is overwritten if $K_L$ is assumed to be 1 in the **Settings** menu. The second option, shown below, enables the calculation of $K_L$ based on the lateral support spacing choices*. (**Canadian only**)

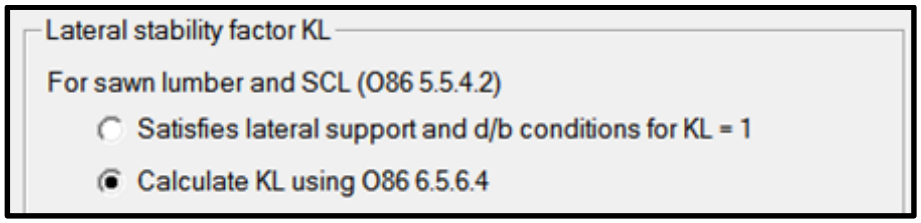



Figure 45: *Lateral stability factor (Canada) must be set to "Calculate…" in order for the user input of lateral support spacing to be effective.*

## 5.9 Glulam (U.S. Only)

This field input allows the user to select between glulam *Stress class* properties or glulam *Combination* properties. For those combinations of Southern Pine which contain wane on one or both sides, the option of specifying whether or not to include the strength reduction is accomplished by selecting or deselecting the check box. The non-edge-bonded laminations check box controls whether a reduction factor is to be applied to the $F_{vy}$ value. The action is only available when the maximum lamination width is less than the member width, so that edge-bonding is possible. The dialog box also allows users the option of specifying whether or not to include the volume factor ($C_v$) in the analysis.


Figure 46: *Glulam Input (US)*

## 5.10 Glulam Design (Canadian)

*Note: The Glulam beam Fire Design options must be activated in the Design Settings*

For glulam beams, users can selected the number of sides exposed to the fire (ie. 0, 3 or 4) based on the equations in NBC D-2.11.2 1) a)-d).[3] There are separate equations in the D-2.11.2 1) a)-d) for columns and beams and for 3- and 4- sided exposure. They depend on the load factor f and the lesser and greater section dimensions B and D. Additional Protection can also be specified by enabling ½" gypsum board (increases endurance by 15 minutes), 5/8" gypsum board (increases endurance by 30 minutes), and two–plies of 5/8 (increases endurance by 1 hr).

With the implementation of the CSA O86-14, the equation for $K_{zbg}$ for glulam beams was updated and is dependent on the width of the member's lamination, the depth and the length.[2] By default *Sizer* will modify the minimum lamination width depending on the width of the member specified based on published typical lamination widths for glulam beams, but users can also modify the lamination width should they have knowledge of the lamination width used by the glulam manufacturer. For glulam beams 130 mm in width or smaller, the lamination width is the same size as the width of the beam. If the width of the glulam beam is left as unknown before the design is run, there is the option of toggling the "Use member width for Kzbg facto and notch $F_f$". Toggling this setting will automatically use the width of the member for the calculation of $K_{zbg}$ and $F_f$, which is a conservative assumption for members which are greater than 130 mm in width.

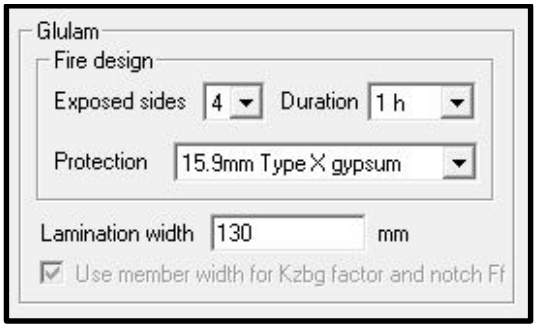



Figure 47: *Glulam Input (Canadian)*

### 5.11 Fire Design Button (U.S. Only)

Clicking on this button opens the Fire Resistance dialog that allows you to specify fire endurance design criteria provided the Fire-endurance rating is activated in the *Settings*/*Design* tab.

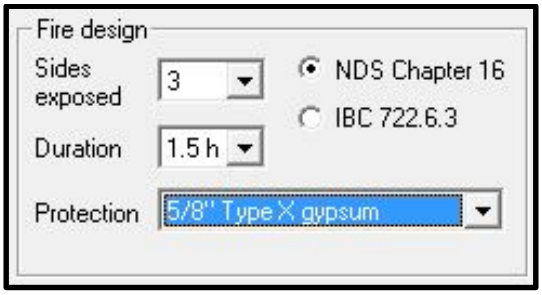



Figure 48: *Fire Design (US)*

*Sizer* checks the Fire Endurance Rating of timber and glulam beams or columns. *Sizer* calculates the *Fire Endurance Rating* based on the number of exposed sides and on the loads applied to a member. To activate the fire endurance check, the number of exposed sides must be defined. If the number of exposed sides is set to zero, the program will not perform the fire endurance check.

Fire Endurance Rating is activated through the *Settings*/*Design* tab.

### 5.12 Supports for Bearing Design (U.S.)

These fields allow you to select the type of bearing as well as the support material for the bearing design. The type can be selected as hanger, beam, column, sill plate or other non-wood. The bearing length or width can be entered or the calculation of the bearing length can be made more specific. Users also have the option of specifying Material, Species and Grade. The bearing design information can be entered for "all supports" at once (as shown in screen capture) or separately for "Left End" versus "Right End" supports, or for each support independently. When the bearing length is not entered (or left as "unknown"), the "For unknown bearing length" entry is activated, and several choices are made available including: using the exact calculated minimum bearing length, rounding the calculated bearing length to the closest user specified designation, automatically rounding up to the next choice in the "Bearing Length" dropdown box, or a combination of the previous two options useful for when a beam sits only partially on end supports but fully on interior supports.

Minimum bearing lengths can be specified in the ***Design*** tab of the ***Settings*** menu.

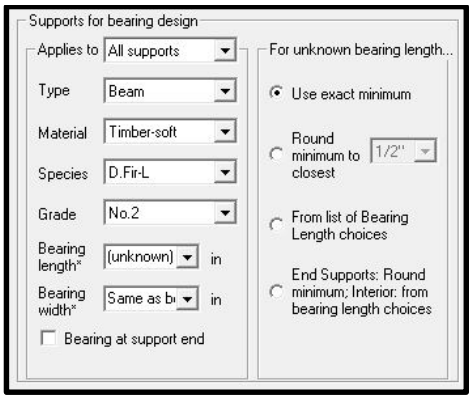



Figure 49: *Support for Bearing Design Input (U.S.)*

### 5.13 Supports for Bearing and notch Design (Canadian)

These fields allow you to select the type of bearing as well as the support material for the bearing design. This section is also used to specify notches at the supports of the beam. For single span beams, the user has the option of specifying bearing and notch details apply to all the supports, the left end, or the right end. If a beam has multiple spans, the user then has the option of specifying support details to all supports, interior supports, end supports, left end, right end, or can select specific supports (ie. “support at #m or #ft” along the length of the beam).

These fields allow you to select the type of bearing as well as the support material for the bearing design. The type of bearing can be selected as hanger, beam, column, sill plate or other non-wood. The bearing length or width can be entered or the calculation of the bearing length can be made more specific. Users also have the option of specifying Material, Species and Grade. When the bearing length is not entered (or left as "unknown"), the "For unknown bearing length" entry is activated to use the exact minimum. Several other choices for unknown bearing length are made available to users, including using the exact calculated minimum bearing length, rounding the calculated bearing length to the closest user specified dimension, automatically rounding up to the next choice in the "Bearing Length" dropdown box, or a combination of the previous two options useful for when a beam sits only partially on end supports but fully on interior supports. Both the bearing length and bearing width can be selected from the drop down list or specific values can be input manually as per the “*” in the beam input.

Users are given the option to select the box *bearing where support ends or is highly stressed.* This is because CSA O86-14 7.5.7.6 includes both these conditions.[2] For main members, both the bending stress and the proximity to the end are detected, but that is not possible for supporting members.

Notches can be specified as being on the **top** or **bottom** of a beam. Both notch depth and length are required as input. Both notch depth and length are required input. If a notch at the bottom is specified, the user has the option of setting the notch length equal to the bearing length instead of inputting a specific value. The ends of a beam must be laterally supported, but the user has the option of specifying whether interior notches are laterally supported by toggling the “Laterally supported at support” box.

Notch length input for interior notches assumes notch is centered at support, that is, there is equal unsupported notch length on either side of the support. If a Glulam member is selected and an interior support is input on the top (or compression) side of the member, the program will reject the input of interior top notches and resets the input fields without notifying you, as top notches are not allowed on interior supports as per CSA O86 commentary.[4]

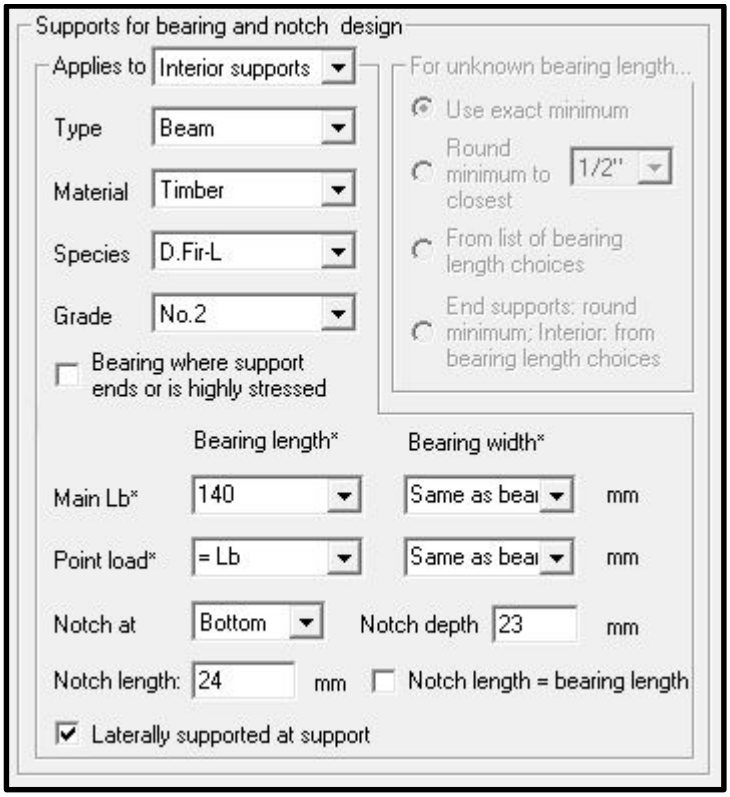



Figure 50: *Supports for bearing and notch design inputs (Canadian)*

### 5.14 Vibration Buttons (Canadian Only, Floor Joists Only)

This button opens the *Vibration Design* dialog box that allows you to specify details that affect floor vibration. Vibration analysis is carried out according to A-9.23.4.2.(2), Appendix A of the National Building Code of Canada.


Figure 51: *Floor Vibration Input (Canadian)*

# 6 Beam Mode Loads

Click the ***Loads View*** button on the toolbar to specify the loading of a beam (or joist) that is to be designed. The *Loads View* opens. Figure 52 shows the beam load input for the Canadian edition of the software and Figure 53 shows the beam load input for the U.S. edition of the software.

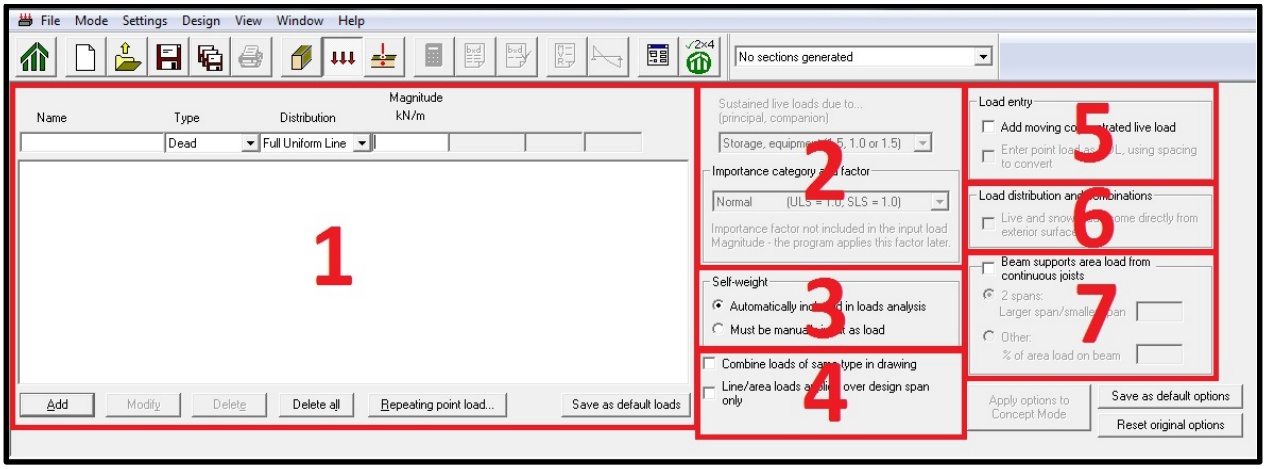



Figure 52: *Beam Mode Load Input Window (Canadian Edition)*

The following Table Summarizes where information on each Part of Figure 52 can be found. The section numbers have been hyperlinked for convenience:

| | Section |
|---|---|
| Part | Figure 52 (Canadian) |
| 1 | 6.1 |
| 2 | 6.9 |
| 3 | 6.4 |
| 4 | 6.6 and 6.7 |
| 5 | 6.2 |
| 6 | 6.10 |
| 7 | 6.5 |

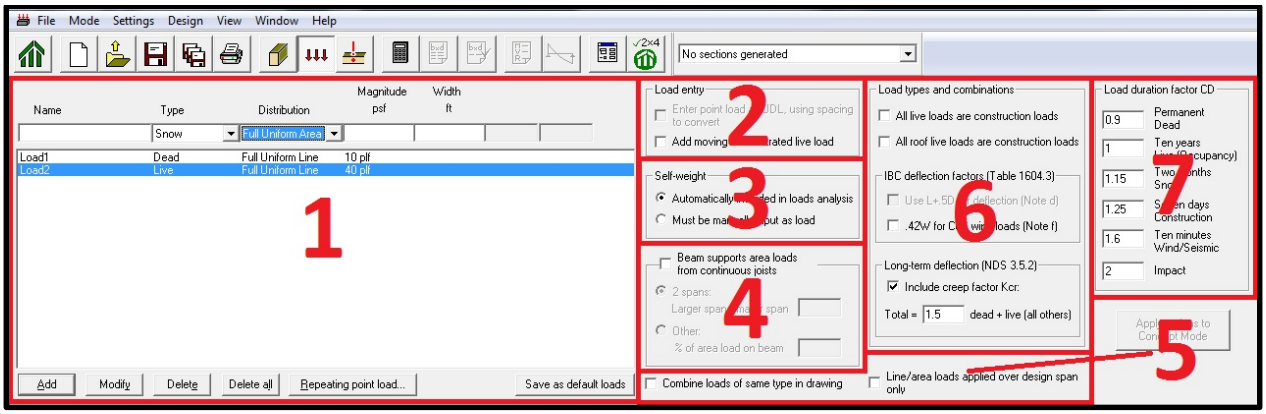



Figure 53: *Beam Mode Load Input window (U.S. Edition)*

The following Table Summarizes where information on each Part of Figure 53 can be found. The section numbers have been hyperlinked for convenience:

| Part | Section Figure 53 (U.S.) |
|---|---|
| 1 | 6.1 |
| 2 | 6.2 |
| 3 | 6.4 |
| 4 | 6.5 |
| 5 | 6.6 and 6.7 |
| 6 | 6.8 |
| 7 | 6.12 |

## 6.1 Load Input

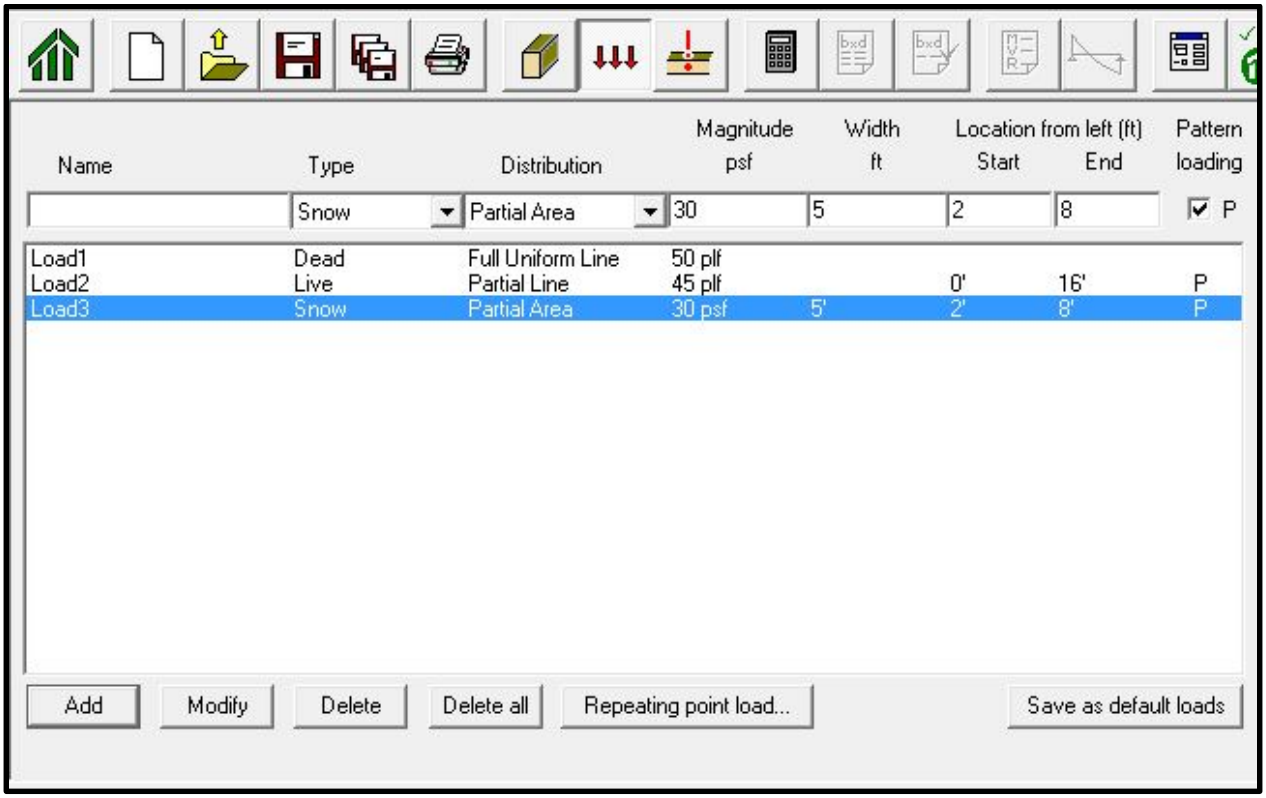



Figure 54: *Beam Mode load input*

### 6.1.1 Name

Use the Name field to enter the name of the load you want to apply to the beam. (*Sizer* generates an appropriate name if you leave this field blank.) The list box beneath this field contains the names of all the loads you have specified for this beam. Click one to select it.

### 6.1.2 Type

This drop-down list specifies the type of load being applied. *Sizer* allows you to select from a variety of load types, including dead, live, roof live (U.S. only), snow, wind, impact (U.S. only), sustained live (storage and contained fluids) and dead (soil) (Canadian only), earthquake, and hydrostatic (Canada only, columns only). The U.S. version also allows all live loads or all roof live loads to be considered a "construction" load, and will use the appropriate duration factor if this is selected.

*Sizer* will apply the correct load duration factor and load combination factor to each load combination.

### 6.1.3 Distribution

This drop-down list specifies the type of load distribution: Full Uniform Line, Full Uniform Area, Partial Line, Partial Area, Triangular, Trapezoidal, Point and Applied Moment.

### 6.1.4 Magnitude

This field specifies the magnitude(s) of the loads being applied. These magnitudes should be entered as specified loads (such as those stated in building codes). When using English units, *Sizer* allows you to change whether point loads are entered in pounds (lbs) or kilopounds (kips) through the Settings Format tab.

*Note that area loads are converted to and displayed as line loads (plf or kN/m) on the load diagram.*

### 6.1.5 Magnitude: Width

This field is only active for Full Uniform Area and Partial Area load distributions and is equal to the tributary width for the bending member. For joists, this value automatically corresponds to the joist spacing selected in *Beam* input view.

### 6.1.6 Location from Left

This field specifies the location of the selected load. It is active if the load distribution is Partial Line, Partial Area, Triangular, Trapezoidal, Point Load and Applied Moment.

For Partial Line loads, specify a start and an end measurement. For Triangular loads, specify a location where the load is zero and a location for the maximum load. For Trapezoidal loads, specify the locations of the minimum and maximum loads.

### 6.1.7 Pattern Loading

Check this box to activate automatic pattern loading. Pattern loading is available in the Beam mode and is applicable to multi-span beams or joists when a live or snow load type is being applied continuously across the member. *Sizer* will take the live load and pattern it on the various spans to find the worst case for design. Pattern loading is an option that can be turned on or off at the Load input stage. Live loads are patterned as full- or no-load intensity. Snow loads are patterned as full- or half-load intensity.

### 6.1.8 Save as default loads:

Once loads have been entered, clicking the "*Save as default loads*" button stores the loads so that the next time the same member type is selected, the default loads will appear.

Saving default loads deletes the default loads that were previously saved for a member type. You can eliminate default loads by pressing ***Save*** as default loads for a blank set of loads.

If repeating loads are included, the program includes only those repeating loads that were applied to the original member when default loads are saved. It is advisable to create the longest possible member to apply default repeating loads to.

Default loads are automatically created for members that have not been saved (and do not have a file location). Once a member has been saved, no loads will be automatically created or deleted.

### 6.1.9 Repeating Point Loads

Click this button to specify multiple point loads with equal spacing and magnitude.

#### 6.1.9.1 Load Direction Assumptions

*Sizer* automatically applies the loads according to the following rules:

- Most loads, including snow, live and dead loads, are applied vertically.
- Wind loads are applied perpendicular to both sloped and horizontal members.

- Along the actual length of a sloped member. All other loads are applied along the horizontal projected length of a sloped member.

Gravity loads (downward) are represented by a positive load magnitude while suction (upward) loads are represented by a negative load magnitude.

### 6.1.10 Sloped Member Load Location

When specifying the location of a load on a sloped member, use the horizontal projected length as a reference.

## 6.2 Load Entry

"***Enter point load as UDL***" uses the spacing to convert a point load to a uniformly distributed load on joists based on the spacing. More typically, this is used to create point loads on joists based on a uniformly distributed load, say from a wall above. This is especially useful when the spacing of joists is being reviewed, in that the line load from above is automatically converted to the correct point load on the joist based on revised spacing. For example a 1200 plf line load from a wall perpendicular to the floor joists imposes a 2400 lb point load on joists spaced at 24" o.c., 1600 lbs per joist when spaced at 16" o.c., and 1200 lbs per joist when spaced at 12" o.c. Instead of manually revising this point load for each spacing run, this load entry tool automates the process.


Figure 55: *Load Entry*

## 6.3 Add moving concentrated live load (Floor Joists Only)

Only applicable to floor joists, this option allows you to design a floor joist for a moving concentrated floor live load that is required by most building codes for certain building categories or floor uses. When selected, *Sizer* automatically creates a concentrated live load with a default magnitude that acts over a default width. The magnitude and width can be modified. During the design process, *Sizer* will determine the worst effects of the concentrated load located anywhere along the length of the member in conjunction with all other applied loads, except live loads, and all required load combinations where the concentrated live load acts in place of any loads of type live specified by the user.

## 6.4 Self-Weight

Users have the option of deciding whether or not to automatically include the self-weight of the beam in the loads analysis. The default setting is to include self-weight in the analysis, but it can be turned off by toggling ***Must be manually input as load***.

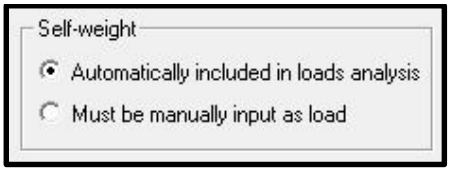



Figure 56: *Self-weight*

## 6.5 Beam supports area loads from continuous Joists

If *"automatically included in loads analysis*" is selected by the user, the self-weight of a member is considered in the design. The actual effects of self-weight are calculated by multiplying the specific gravity of the material times the area of the section. This load is then added to the dead load when checked. Alternatively the self-weight can be ignored or manually added by selecting "*Must be manually input as load*" the beam supports area load from continuous joists.

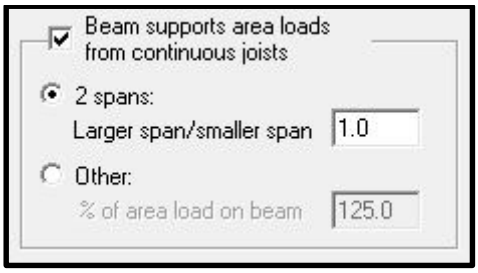



Figure 57: *Beam Supports area load from continuous joists*

When continuous joists, rather than simply supported joists, are loading on a beam, the reaction from the continuous joist is greater than the reaction that would be imposed simply supported joists that end at the beam. In other words the reactions the continuous member imposes are not based on the tributary area alone. For example, when a continuous joist spans over the supporting beam, and the joist length is equal on both sides of the designed beam, the reaction imposed on the designed beam is 1.25 times greater than would be calculated by the tributary area alone, as per below Figure 29 from AWC's *Design Aid No. 6, Beam Design Formulas with Shear and Moment Diagrams*. The general case where the joists are continuous over the designed beam with unequal spans is also shown below, from *Design Aid No. 6,* Figure 31. Alternatively, Beam Design Formulas can be found in Chapter 11 of the CWC publication known as the Wood Design Manual.[5]


Figure 58: *NDS Design Formulas with Shear and Moment Diagrams*[6]

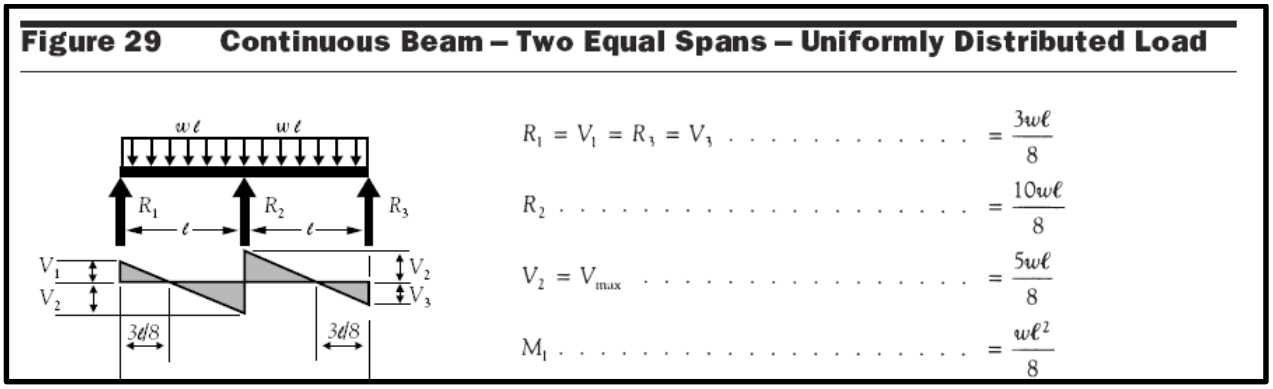



Figure 59: *Continuous beam two equal spans UDL Shear and Moment Diagrams*[6]

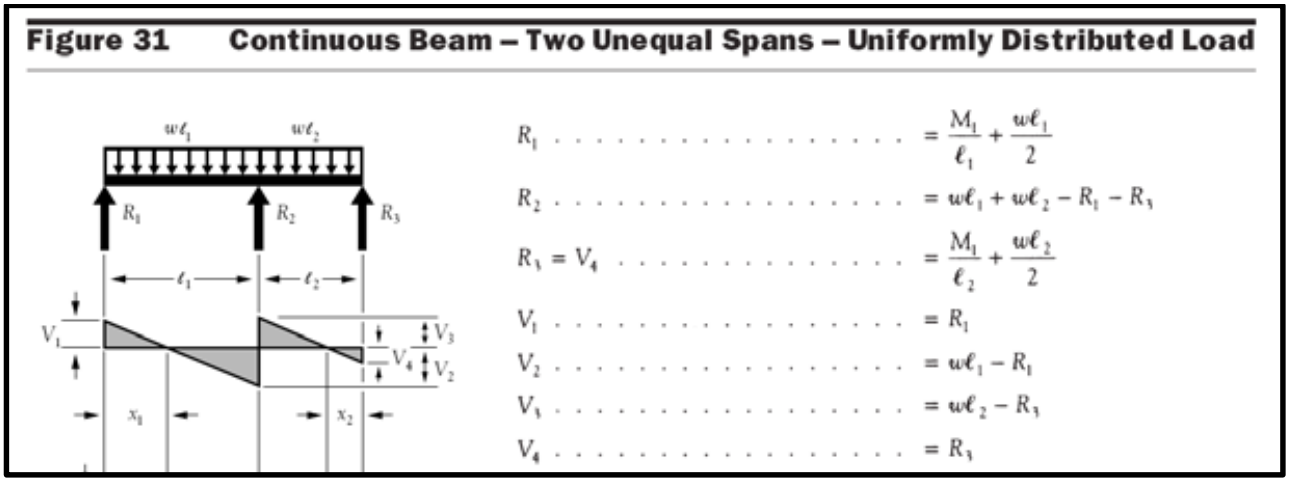



Figure 60: *Continuous beam two unequal spans UDL Shear and Moment Diagrams*[6]

Accounting for the increased load a beam should be designed for when joists are continuous over the beam is done by selecting the "*Beam supports area load from continuous supports*" checkbox, and either indicating the ratio of the joist span on either side of the beam being designed by first selecting "*2-spans*", or indicate the percent of load on the beam manually by selecting "*Other*".

"*Beam supports area load from continuous supports*" is only enabled when a beam is being designed, and only applies to full uniform area and partial area loads.

### 6.6 Combine loads of same type in drawing


Figure 61: *Combine Loads of same type in drawing checkbox*

This option only applies to the loading diagram. When enabled, individual loads are accumulated into a single loading profile for each load type (dead, live, etc.). When not enabled, loads overlap with other loads of the same type. The selected load appears in bold.

## 6.7 Line/Area Loads


Figure 62: *Line/area Load checkbox*

At the bottom of the input section is a "Line loads applied over design span only" checkbox. If checked the software distributes the line and area loads to the design span, not the entire beam. This is more a tool to compare files made in previous versions which only applied loads to the design span.

## 6.8 Load Types and Combinations (U.S. Only)

*Sizer* combines loads based on the Allowable Stress Design method using the basic load combinations in the IBC/ASCE (U.S.) and Limits States Design load combinations (Canadian).[7,8]

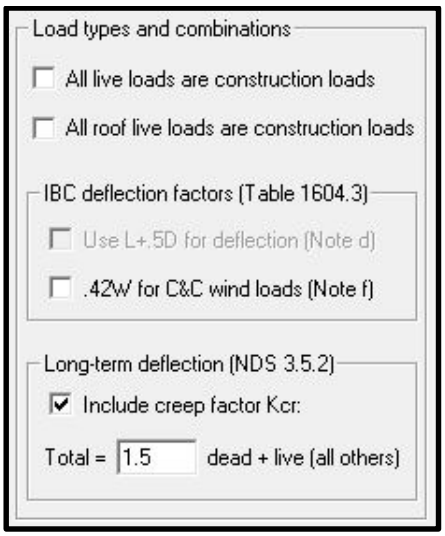



Figure 63: *Load Types and Combinations Input*

In addition to considering the building code load combinations based on the load types present, *Sizer* also creates load combinations to consider the effects of pattern loading live type loads for multi-span beams or joists when pattern loading is active. Refer to the Pattern Loading section for more information on pattern loads. The designer should verify that the load combinations used are adequate. The User has the option of selecting whether or not to consider live and roof loads as construction loads, by clicking the appropriate box.

For Beam mode, *Sizer* output a list of load combinations with an explanation of each load combination number referred to in the results along with the appropriate load duration factor.

When snow loads are present S and s represent full and half snow loads respectively (e.g., pattern: SsS). Wind loads are not patterned since they are assumed to apply to all spans simultaneously.

For more information on load combinations or on pattern loading, refer to online help. You can check obscure load combinations by manually adjusting the load duration factors (U.S. only), or performing individual load combinations and load patterns.

## 6.9 Load Distribution and Combinations (Canadian)

Active whenever there is both a live or sustained live load and a snow load on the member.

To comply with NBC 4.1.5.5, this setting allows the user to create a set of load combinations without live and snow loads in the same combination.[3]

### 6.9.1 Sustained Live Loads

The program now creates separate load combinations for D (+ Ds + H) + Ls or D (+Ds + H) + Lf, in addition to the combinations D + (Ds + H) + Ls/f + L, when occupancy live loads and sustained loads are on the same member. Refer to the section on $K_D$ factor for the reason, and on the on-line help for a complete explanation.

### 6.9.2 Importance Category and factor

Once a Live, Snow, Wind, or Earthquake load type is selected in the load input, the Importance category and factor will come available which allows the importance category for the load to be specified (ie. Low, Normal, High, Post-disaster or Normal Part 9 Snow).

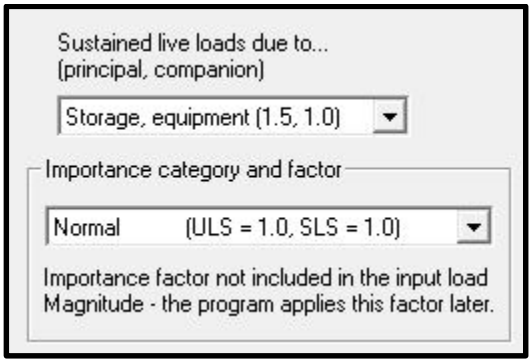



Figure 64: *Sustained Live Loads and Importance category and factor*

## 6.10 New load types

Added all appropriate ULS load combinations containing hydrostatic, earthquake, and dead (soil) loads. Refer to online help for a complete list.

Defaults to unchecked, and it is not possible to set a different default

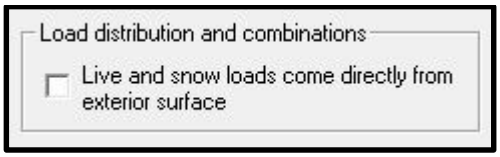



Figure 65: *Load Distribution and combinations (Canadian)*

## 6.11 Deflection Factors (U.S. Only)

### 6.11.1 IBC

There are two leniencies offered in the IBC related to deflection. These optional IBC deflection factors can be selected using two checkboxes.[7]

### 6.11.2 Use L+0.5D for deflection (note d)

The first allows the deflection resulting from L + D to be calculated using L+ 0.5D for wood structural members having a moisture content under 16 percent at the time of installation and used in dry conditions.

### 6.11.3 0.42W for C&C Wind Loads (note f)

The second option is related to components and cladding wind loads, where deflection may be calculated using only 70 percent of the wind load.

### 6.11.4 Long Term Deflection (NDS 3.5.2)

The NDS provides guidance on considering the effects of long term deflection by increasing the contribution of dead load by a creep factor, $K_{cr}$, that typically ranges from 1.5 to 2; any value can be entered if the "*Include creep factor*" is checked.[1]

*Note: The long-term deflection factor of the NDS needs to be deselected in order to enable the IBC L+0.5D leniency.*[7]

## 6.12 Load Duration Factors

*Sizer* applies load duration factors according to the load type. These factors are set to the standard NDS (U.S. version).[1] The Canadian version does not include this input as there is no guidance on modifying load duration factors in the CSA O86-14. For the Canadian version, when D > L, Sizer determines the default duration factor for standard term loads according to the equation shown in CSA O86-14 clause 6.3.2.3.[2]

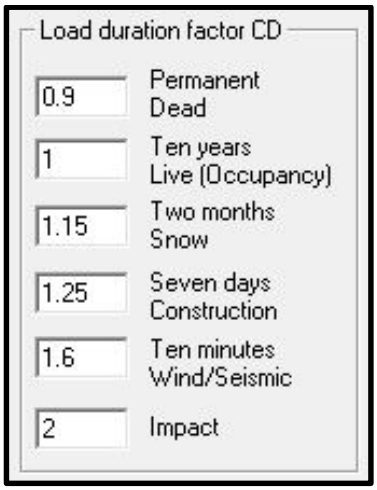



Figure 66: *Load Duration Factor CD*

## 6.13 Apply Options to Concept Mode


Figure 67: *Apply Options to Concept Mode*

This button in *Beam Load View* and *Column Load View*, is only active when there is a *Concept* mode file open, and allows the application of the following settings to have an effect on *Concept* mode. This items can only

be accessed from *Beam* or *Column* mode, and are *Self-weight*, *Line Loads applied over design span only*, *Long-term deflection factor*, *Use L+0.5D for deflection*, *Load duration factors CD*, *Load combinations from…*, and *Temperature* (from *Beam View*).

# 7 Beam Mode Points of Interest

Click the ***points of interest*** button on the toolbar to investigate the shear and moment at any point along the length of a beam or column. A point of interest is generated by specifying a location to perform the analysis. Now click ***Add*** to add this to the list. Several points of interest can be specified.

After performing a design, the point of interest results will be shown in the Diagrams window and in the Analysis results output.

The Points of Interest function can be helpful to designers who want to determine the shear and moment at a location where a hole is to be drilled into a member or in locations where Point Loads are applied. The function can also be used to determine the shear and moment for repeating point loads.

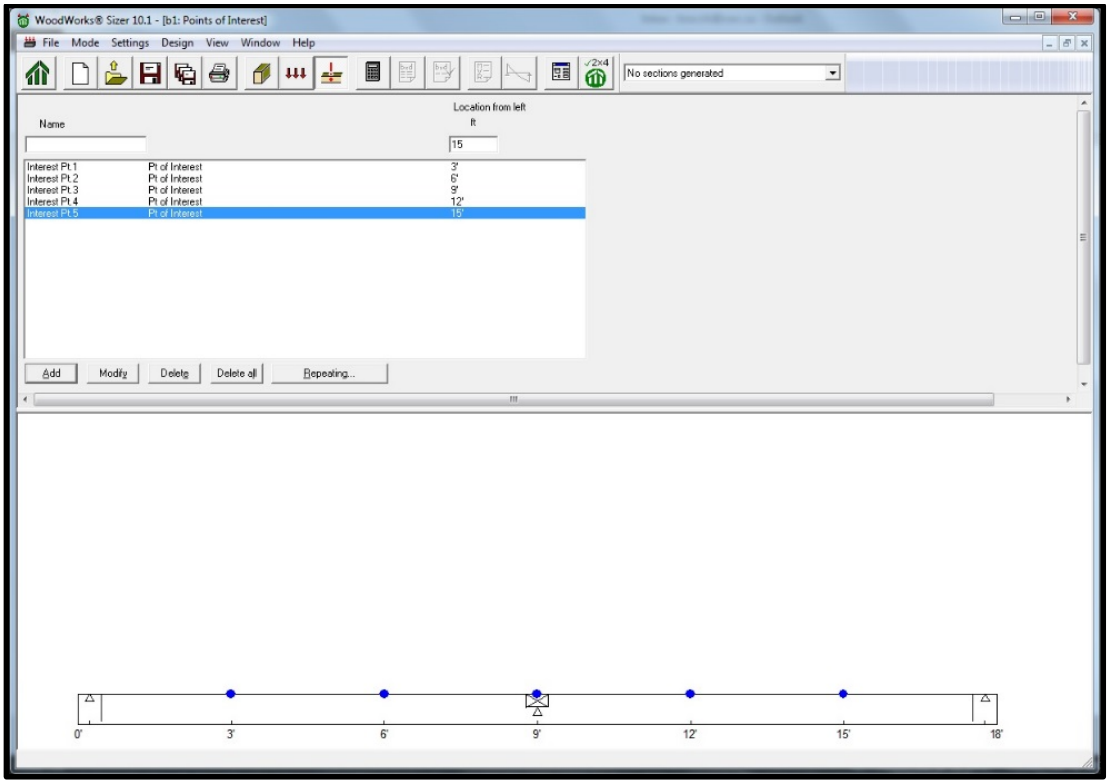



Figure 68: *Inputting Points of Interest*

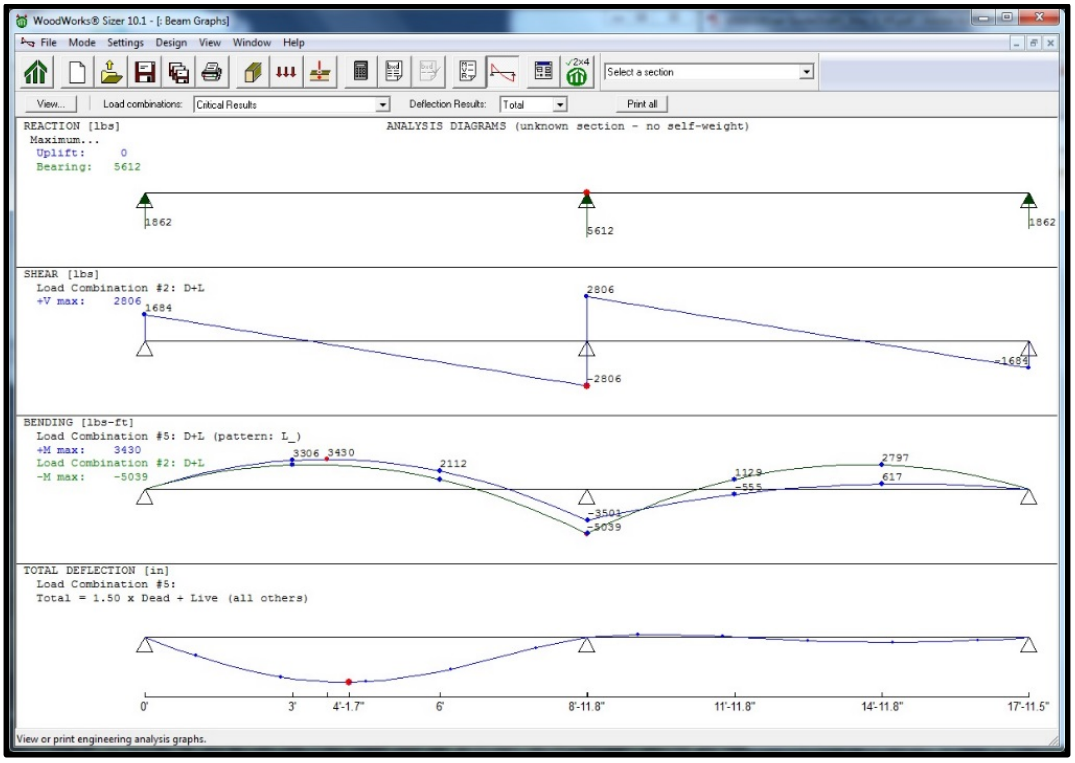



Figure 69: *Points of Interest on Shear and Bending Moment Diagrams*

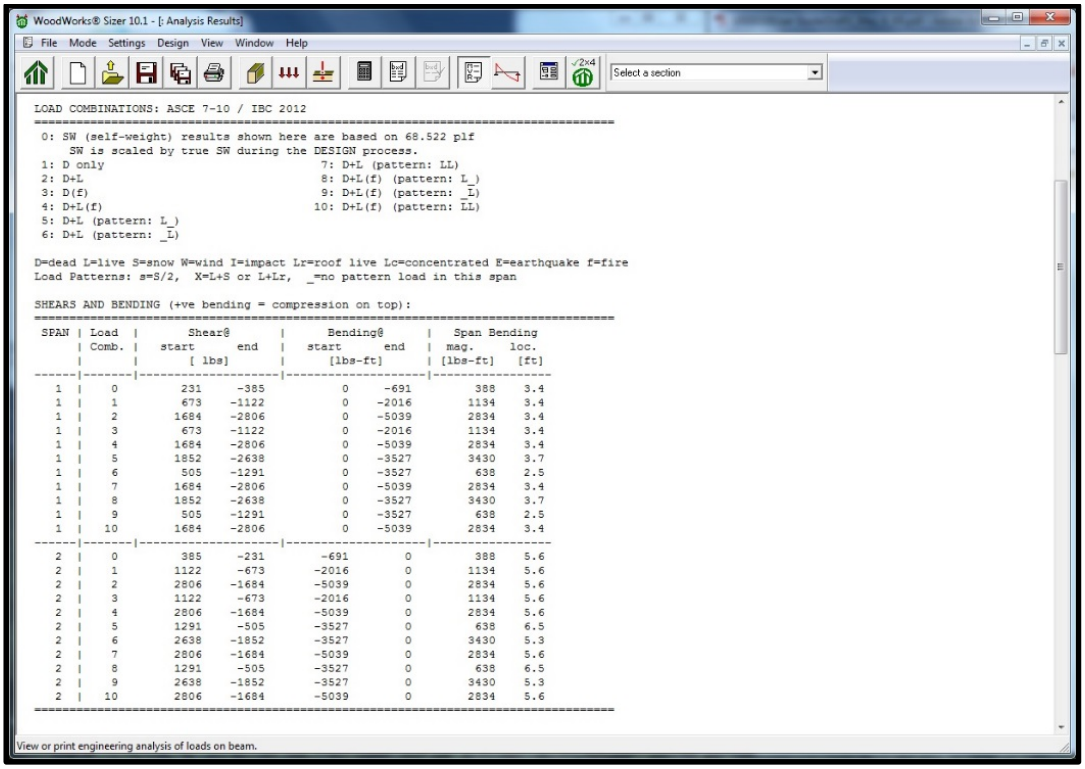



Figure 70: *Points of Interest Analysis Results*

# 8 Beam Mode Design Process

## 8.1 Starting the Design Process

To start the design process, click the ***design*** button on the toolbar. *Sizer* performs an analysis, designs your beam with the information you entered and automatically displays the results.

If you selected *(unknown)* for some entries, *Sizer* selects a series of suitable sections that provide acceptable results. If there are no *(unknowns)* and you have not specified ranges in the width or depth fields, *Sizer* does a design check on the specified section and verifies that the results are within the selected design code's limits.

The number of sections that *Sizer* selects depends on how many fields you specified as *(unknown)* in the beam screen.

*Sizer* normally prompts you to save the current project prior to doing the design. To change this, choose one of the ***Save Before Design*** options from the ***Settings*** dialog.

### 8.1.1 Selecting Sections for Design Check

*Sizer* provides you with an extensive list of suggested sections for the beam it is designing. To get more details about a single section size, you can perform a design check on that section.

After running an initial design, select the section you want to check from the drop-down list entitled ***Suggested Beam Sections*** at the right side of the toolbar.

*Sizer* automatically fills in the Species, Grade, Width and Depth fields with the appropriate values. *Sizer* automatically performs a design check of the selected section.

The list of suggested sections on the toolbar remains available until you perform another design (rather than a design check).

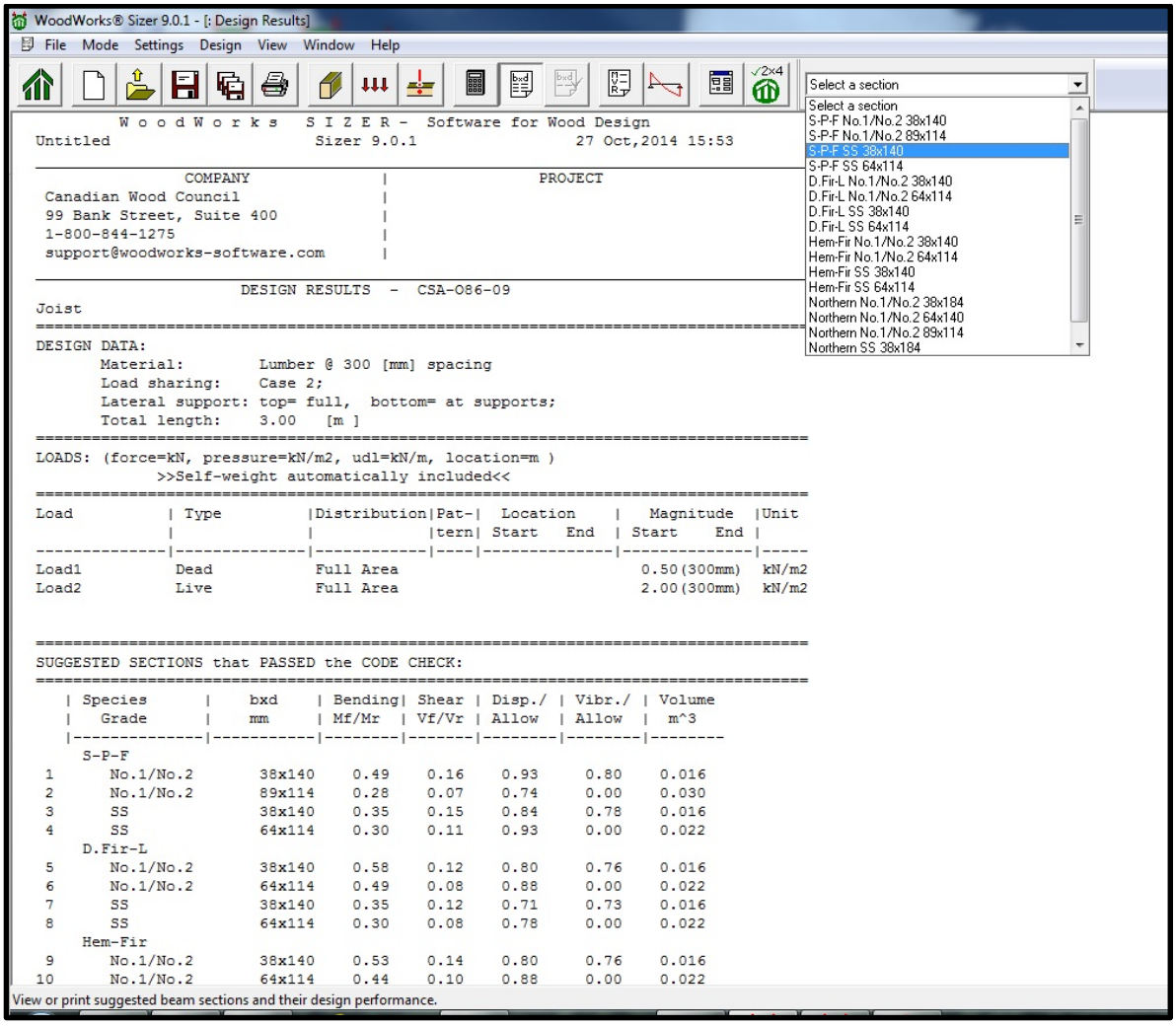



Figure 71: *Selecting Sections for Design Check*

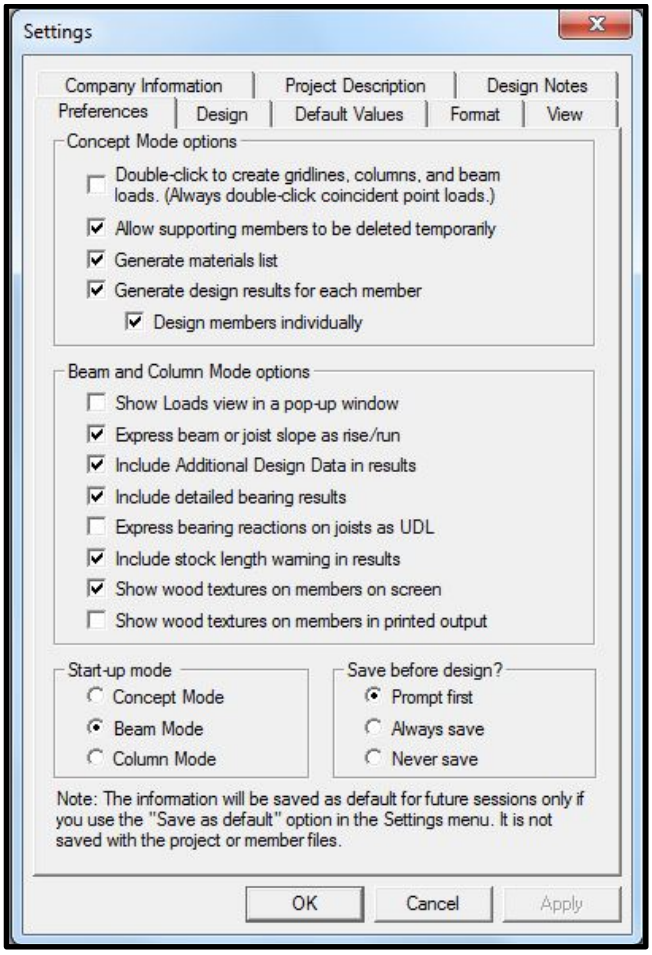



Figure 72: *Beam Mode Settings Preferences*

### 8.1.2 Save Before Design (Prompt first)

Tells *Sizer* to prompt you to save the current worksheet prior to starting the design process. This is the default.

### 8.1.3 Save Before Design (Always save)

Tells *Sizer* always to save the current project automatically (without prompting you) prior to starting the design process.

### 8.1.4 Save Before Design (Never save)

Tells *Sizer* not to save the current project prior to starting the design process. *Sizer* will not even prompt you to save.

## 8.2 Analysis Results

Click the ***analysis*** button on the toolbar to see the analysis results for each load combination (maximum shears, bending moments and so on) for your beam or joist in tabular form.

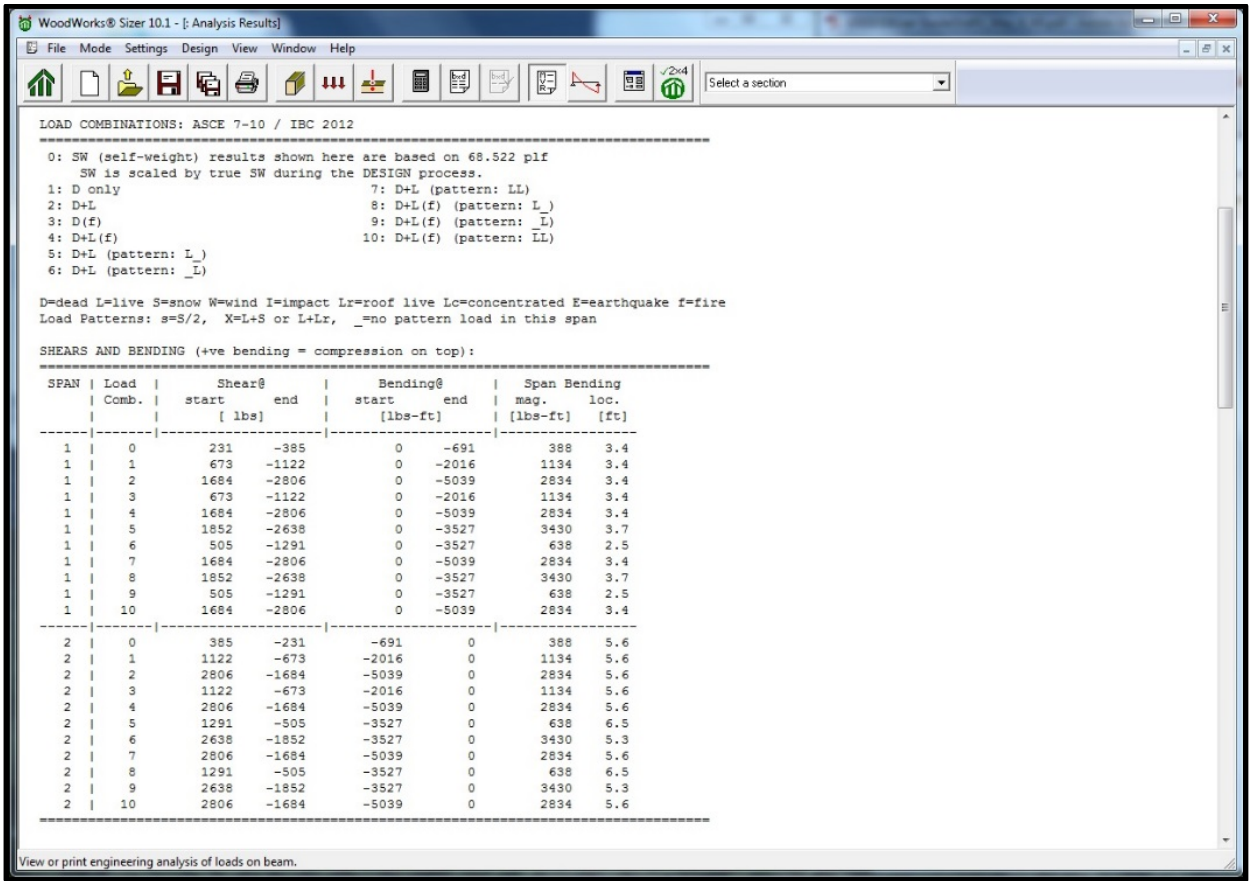



Figure 73: *Beam Mode Analysis Results*

## 8.3 Design Results

If you specified any field as *(unknown)* for your beam or joist, *Sizer* computes all the possible beam or joist sections during the design process. It also computes the ratios of analysis/design values for bending, shear, and displacement for each section.

To see these results, click on the ***Design Summary*** button on the toolbar menu or choose ***Design Results*** from the ***View*** menu.

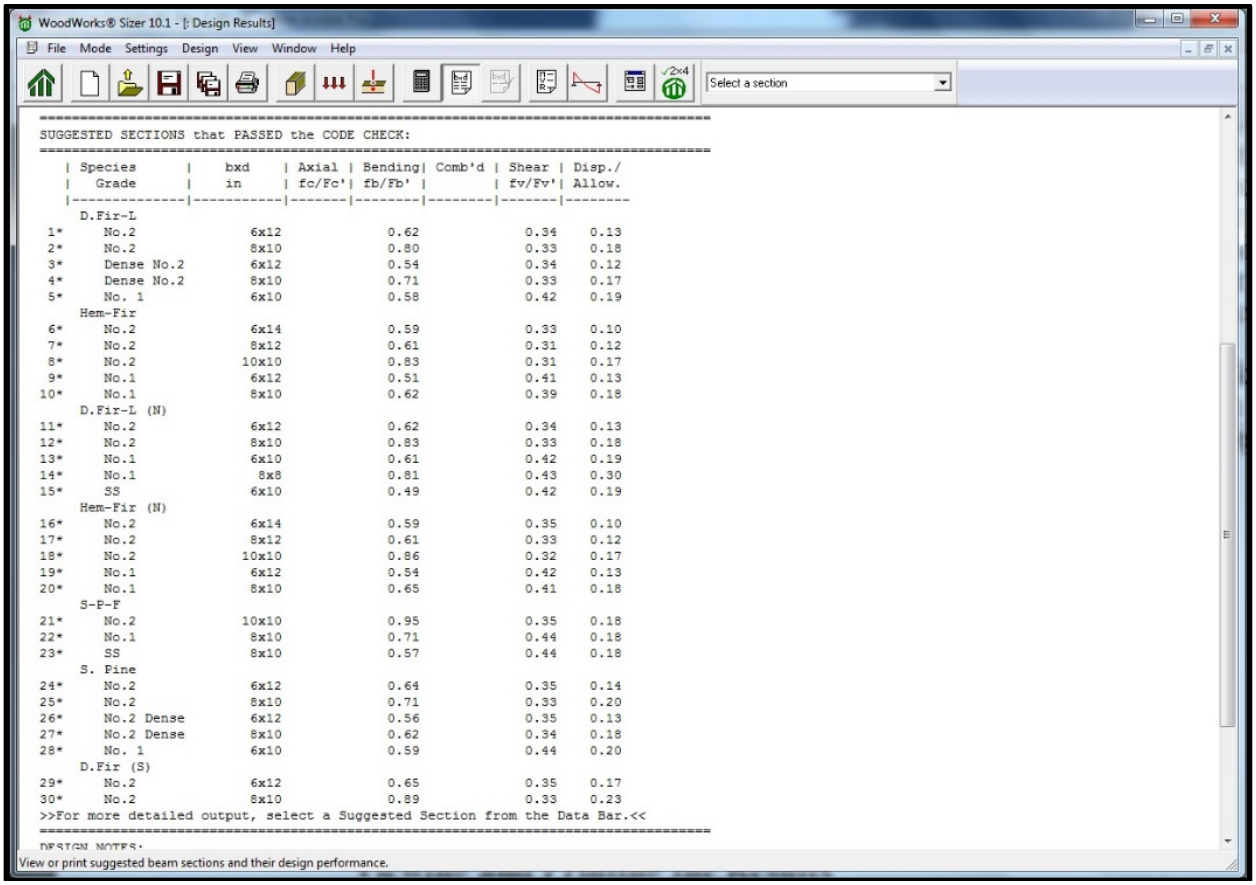



Figure 74: *Beam Mode Design Results*

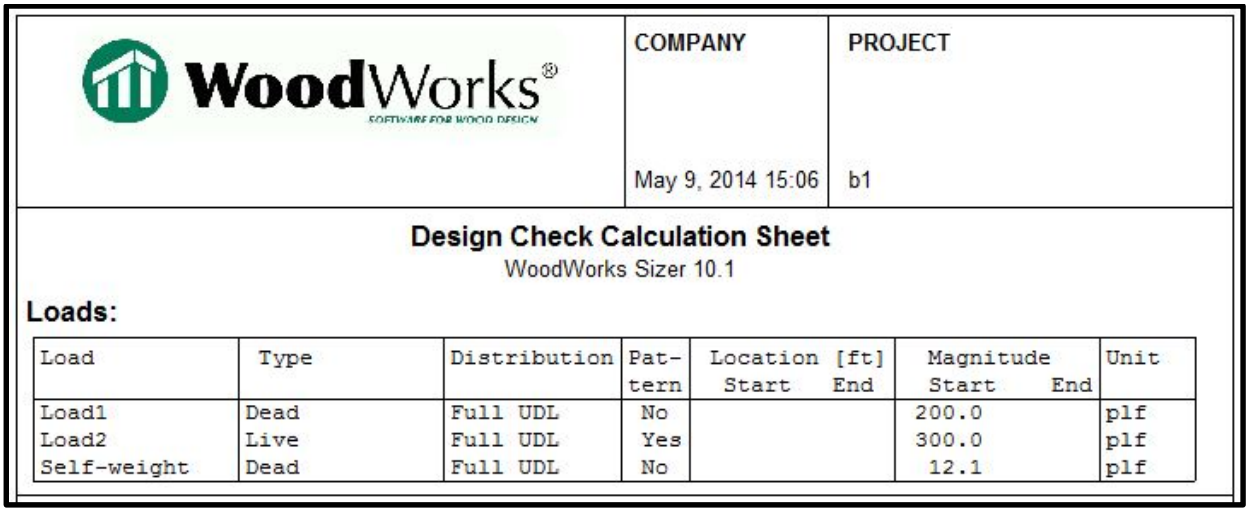
## 8.4 Viewing and Printing the Results

Once *Sizer* has designed the beam, it creates results files. *Sizer* then gives you several options for viewing the results.

## 8.5 Design Check

If you requested a specific beam or joist size, *Sizer* performs a design check and computes analysis and design values (for example shear and bending) for that section.

To see these results, none of the parameters can be left as *(unknown)*. Then click ***check*** on the toolbar.



| COMPANY | PROJECT |
|---|---|
| May 9, 2014 15:06 | b1 |

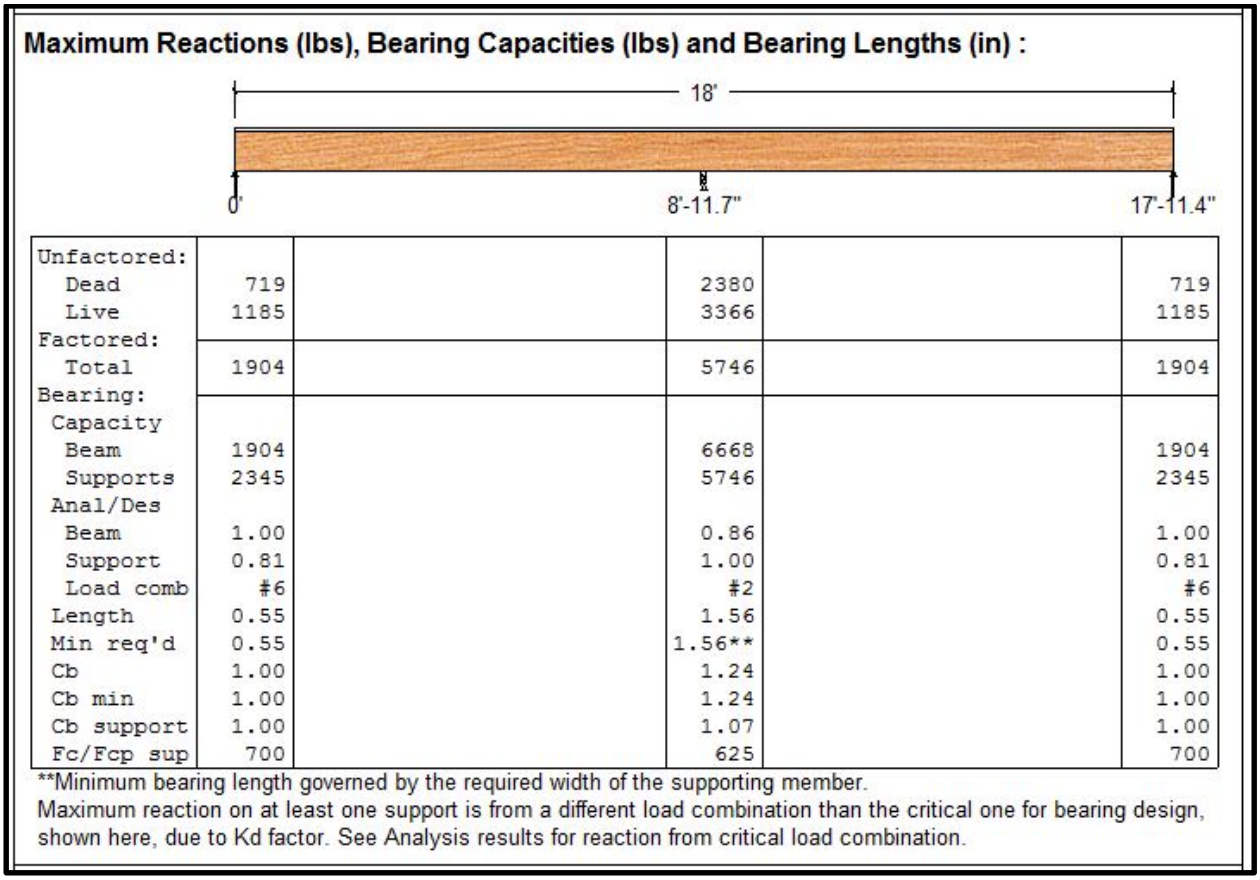
**Design Check Calculation Sheet**
WoodWorks Sizer 10.1

**Loads:**

| Load | Type | Distribution | Pat-tern | Location [ft] Start | End | Magnitude Start | End | Unit |
|---|---|---|---|---|---|---|---|---|
| Load1 | Dead | Full UDL | No | | | 200.0 | | plf |
| Load2 | Live | Full UDL | Yes | | | 300.0 | | plf |
| Self-weight | Dead | Full UDL | No | | | 12.1 | | plf |

Figure 75: *Beam Mode Design Check Part 1*

The Design Check Calculation sheet displays the Company Name, Project Name, and the version of the software that was used. The Loads that were applied to the beam are displayed in a summary table.



**Maximum Reactions (lbs), Bearing Capacities (lbs) and Bearing Lengths (in) :**



| | | | | | |
|---|---|---|---|---|---|
| Unfactored: | | | | | |
| Dead | 719 | | 2380 | | 719 |
| Live | 1185 | | 3366 | | 1185 |
| Factored: | | | | | |
| Total | 1904 | | 5746 | | 1904 |
| Bearing: | | | | | |
| Capacity | | | | | |
| Beam | 1904 | | 6668 | | 1904 |
| Supports | 2345 | | 5746 | | 2345 |
| Anal/Des | | | | | |
| Beam | 1.00 | | 0.86 | | 1.00 |
| Support | 0.81 | | 1.00 | | 0.81 |
| Load comb | #6 | | #2 | | #6 |
| Length | 0.55 | | 1.56 | | 0.55 |
| Min req'd | 0.55 | | 1.56** | | 0.55 |
| Cb | 1.00 | | 1.24 | | 1.00 |
| Cb min | 1.00 | | 1.24 | | 1.00 |
| Cb support | 1.00 | | 1.07 | | 1.00 |
| Fc/Fcp sup | 700 | | 625 | | 700 |

**Minimum bearing length governed by the required width of the supporting member.
Maximum reaction on at least one support is from a different load combination than the critical one for bearing design, shown here, due to Kd factor. See Analysis results for reaction from critical load combination.

Figure 76: *Beam Mode Design Check Part 2*

The design check calculation sheet displays an image of the beam along with the maximum reactions, bearing capacities and bearing lengths at points where the beam is supported. This table allows designers to review the forces at the reactions, as well as the different factors which were used for calculating the bearing at each support.

**b1**

**Timber-soft, D.Fir-L, SS, 6x10 (5-1/2"x9-1/4")**

Supports: 1,3 - Timber-soft Column, D.Fir-L No.2; 2 - Timber-soft Beam, D.Fir-L No.2;
Total length: 18';
Lateral support: top= full, bottom= at supports;
Fire resistance rating: 60 min; No. of exposed sides: 3;

**Analysis vs. Allowable Stress (psi) and Deflection (in) using NDS 2012 :**

| Criterion | Analysis Value | Design Value | Analysis/Design |
|---|---|---|---|
| Shear | fv = 72 | Fv' = 170 | fv/Fv' = 0.42 |
| Bending(+) | fb = 535 | Fb' = 1600 | fb/Fb' = 0.33 |
| Bending(-) | fb = 789 | Fb' = 1600 | fb/Fb' = 0.49 |
| Live Defl'n | 0.05 = <L/999 | 0.30 = L/360 | 0.18 |
| Total Defl'n | 0.09 = <L/999 | 0.45 = L/240 | 0.19 |
| Fire | | | |
| Bending(+) | fb = 2353 | Fb' = 4560 | fb/Fb' = 0.52 |
| Bending(-) | fb = 3462 | Fb' = 4384 | fb/Fb' = 0.79 |

Figure 77: *Beam Mode Design Check Part 3*

The sheet includes the details of the beam which was analyzed. The type of beam, material, species, grade, length, width and depth are displayed. Details on the lateral support and fire resistance rating are also displayed. The results from the Analysis are listed in the "Analysis vs. Allowable Stress (psi) and Deflection (in)" Table. Users can see ratio between the Analysis and Design Values for shear, positive and negative bending, Live and total deflection, as well as the positive and negative bending values from a fire design perspective.

**Additional Data:**

| FACTORS: | F/E(psi) | CD | CM | Ct | CL | CF | Cfu | Cr | Cfrt | Ci | Cn | LC# |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| Fv' | 170 | 1.00 | 1.00 | 1.00 | - | - | - | - | 1.00 | 1.00 | 1.00 | 2 |
| Fb'+ | 1600 | 1.00 | 1.00 | 1.00 | 1.000 | 1.000 | 1.00 | 1.00 | 1.00 | 1.00 | - | 5 |
| Fb'- | 1600 | 1.00 | 1.00 | 1.00 | 1.000 | 1.000 | 1.00 | 1.00 | 1.00 | 1.00 | - | 2 |
| Fcp' | 625 | - | 1.00 | 1.00 | - | - | - | - | 1.00 | 1.00 | - | - |
| E' | 1.6 million | 1.00 | 1.00 | - | - | - | - | 1.00 | 1.00 | - | 5 |
| Fb'+ (f) | 1600 | - | - | - | 1.000 | 1.000 | 1.00 | - | - | - | - | 8 |
| Fb'- (f) | 1600 | - | - | - | 0.961 | 1.000 | 1.00 | - | - | - | - | 4 |

Cn = Notch factor as per NDS 3.4.3

CRITICAL LOAD COMBINATIONS:

Shear : LC #2 = D+L, V = 2873, V design = 2447 lbs
Bending(+): LC #5 = D+L (pattern: L_), M = 3496 lbs-ft
Bending(-): LC #2 = D+L, M = 5158 lbs-ft
Deflection: LC #5 = (live)
LC #5 = (total)
Bending(+): LC #8 = D+L (pattern: L_) (fire), M = 3447 lbs-ft
Bending(-): LC #4 = D+L (fire), M = 5070 lbs-ft
D=dead L=live S=snow W=wind I=impact Lr=roof live Lc=concentrated E=earthquake f=fire
All LC's are listed in the Analysis output
Load Patterns: s=S/2, X=L+S or L+Lr, _=no pattern load in this span
Load combinations: ASCE 7-10 / IBC 2012

CALCULATIONS:

Deflection: EI = 580e06 lb-in2
"Live" deflection = Deflection from all non-dead loads (live, wind, snow…)
Total Deflection = 1.50(Dead Load Deflection) + Live Load Deflection.
Fire design: NDS 2012 Chapter 16
Fire: Residual section = 1.90x7.45 in.; Char rate = 1.80 in./hr; CD = 1.00
Fire design adjustment factor: Fb x 2.85, FbE x 2.03

**Design Notes:**

1. WoodWorks analysis and design are in accordance with the ICC International Building Code (IBC 2012), the National Design Specification (NDS 2012), and NDS Design Supplement.
2. Please verify that the default deflection limits are appropriate for your application.
3. Continuous or Cantilevered Beams: NDS Clause 4.2.5.5 requires that normal grading provisions be extended to the middle 2/3 of 2 span beams and to the full length of cantilevers and other spans.
4. Sawn lumber bending members shall be laterally supported according to the provisions of NDS Clause 4.4.1.

Figure 78: *Beam Mode Design Check Part 4*

The Design check calculation sheet also includes additional data on which design factors were used in the analysis, as well as the critical load cases. Further details on the calculations is also provided. Important design Notes are listed at the bottom of the sheet.

## 8.6 Analysis Diagrams

Click the ***diagram*** button on the toolbar to view the analysis diagrams.

*Sizer* creates four analysis diagrams including Support, Shear, Bending, and Deflection diagrams.

To print the current diagram, click the ***print*** button on the toolbar.

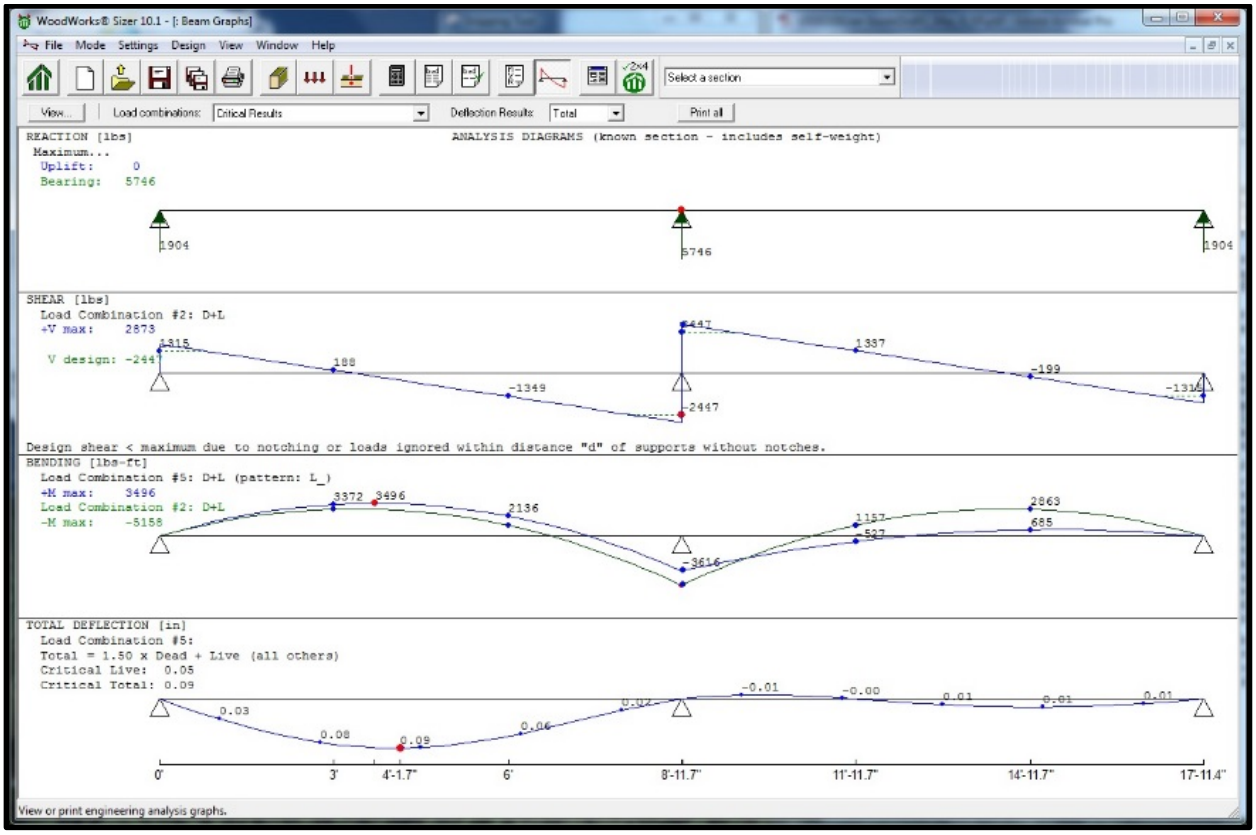



Figure 79: *Analysis Diagrams*

- ***View…*** – This button controls which analysis diagrams are shown, including the Load Envelope, Reaction, Shear, Moment and Deflection diagrams. The Load Envelope diagram is not available when Critical Results is selected as the load combination.

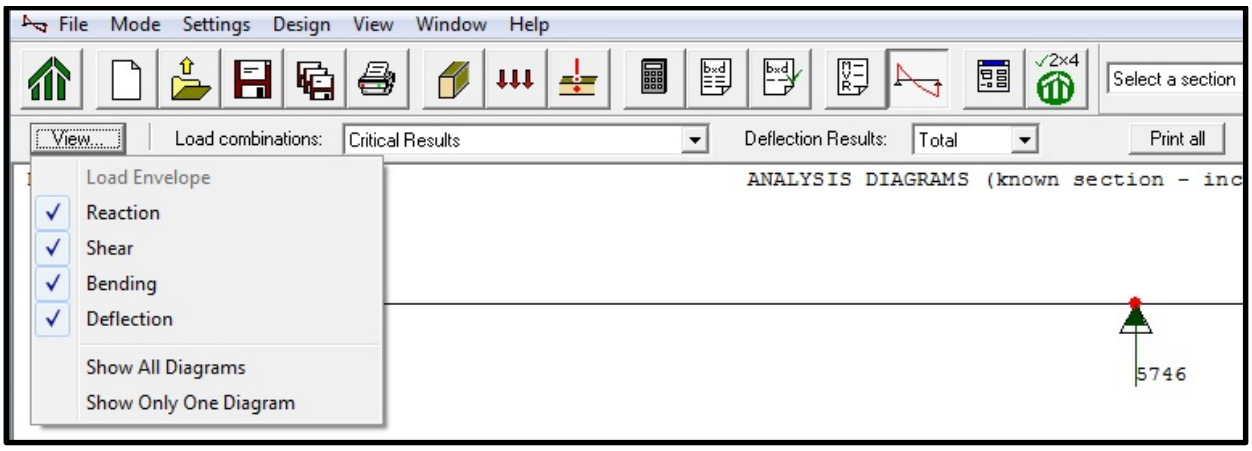



Figure 80: *View Button*

- ***Load Combinations*** – The diagram view results shown are based on the load combination selected in this pull-down menu. When Critical Results is selected, the analysis diagrams shown are for the worst case results of all load combinations. Load combination numbers shown correspond to those used throughout *Sizer* (Analysis Results, Design Check, diagrams, etc.).

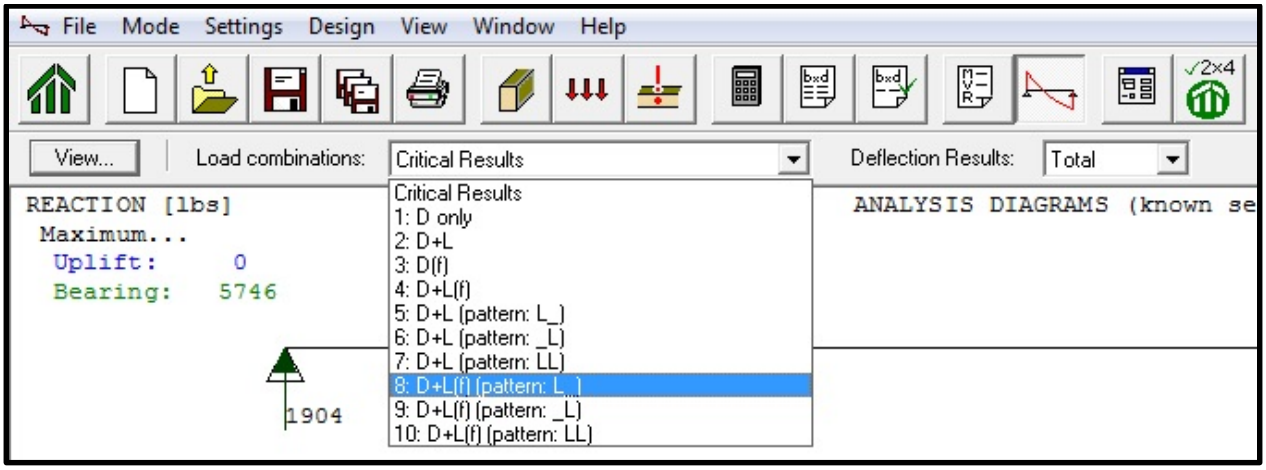



Figure 81: *Load Combination Selection*

- ***Deflection Results*** - The deflection results can be viewed as total load or live load, based on this pull-down menu.

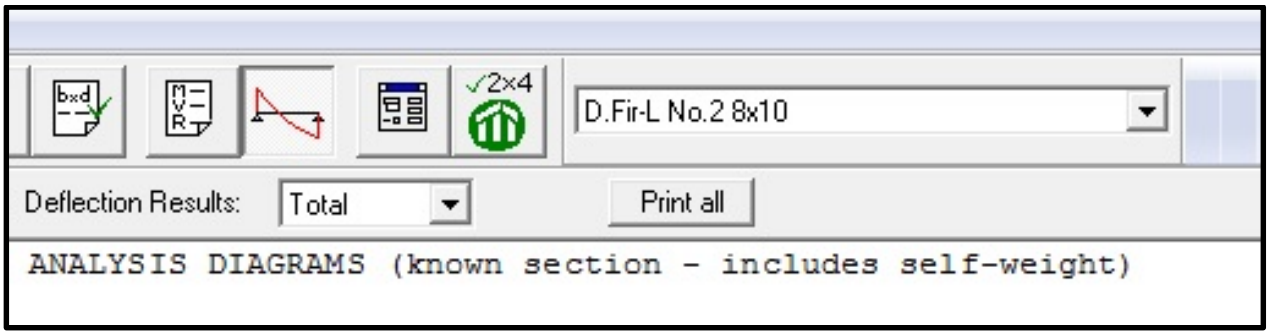



Figure 82: *Deflection Results*

## 8.7 Printing Results

To print results, click on the ***Print All*** button from the toolbar when the result file or graphics screen that you wish to print is on screen. To batch print the analysis diagrams for all load combinations, press ***Print All***.

# 9 *Sizer* Column Mode

## 9.1 What *Sizer* Column Mode Does

Column mode allows you to quickly enter and design individual wood columns, walls and beam-columns under multiple load conditions. Column mode provides extensive information about a single column or wall in your project. Typically, you would use Column mode to verify the design of critical members.

To select *Column* mode, click ***new*** on the toolbar and choose ***column***.

When *Column* mode is active, a checkmark is displayed next to *Column* on the *Mode* menu.

### 9.1.1 Transfer of a members from Concept Mode

If you are operating in *Concept* mode and you wish to further analyze a particular column or wall, click the column or wall to select it and then choose ***Column*** from the ***Mode*** menu, or click the ***mode*** button on the toolbar.

*Sizer* first determines the axial design loads for the column or wall in question, based on the structure entered in Concept mode. *Sizer* automatically transfers the load information to the input fields of Column mode. For walls, it transfers the load information for a single wall stud.

*Note: The transfer from Concept to Column mode is one-way. Any changes you make to the transferred member cannot be exported back to the original member in Concept mode. There are a few exceptions, such as the selection of* ***Self Weight****: "manually input " or "automatically included in loads analysis". Refer to* ***Help*** *on "Apply options to Concept Mode" which refers to what options can be transferred with a click of a button in the* ***Loads View****, activated if Concept mode is open.*

## 9.2 Column (or Wall) Parameters

Click ***column*** on the toolbar to describe the column (or wall) to be designed. Figure 83 shows the column input for the Canadian edition of the software and Figure 84 shows the column input for the U.S. edition of the software.

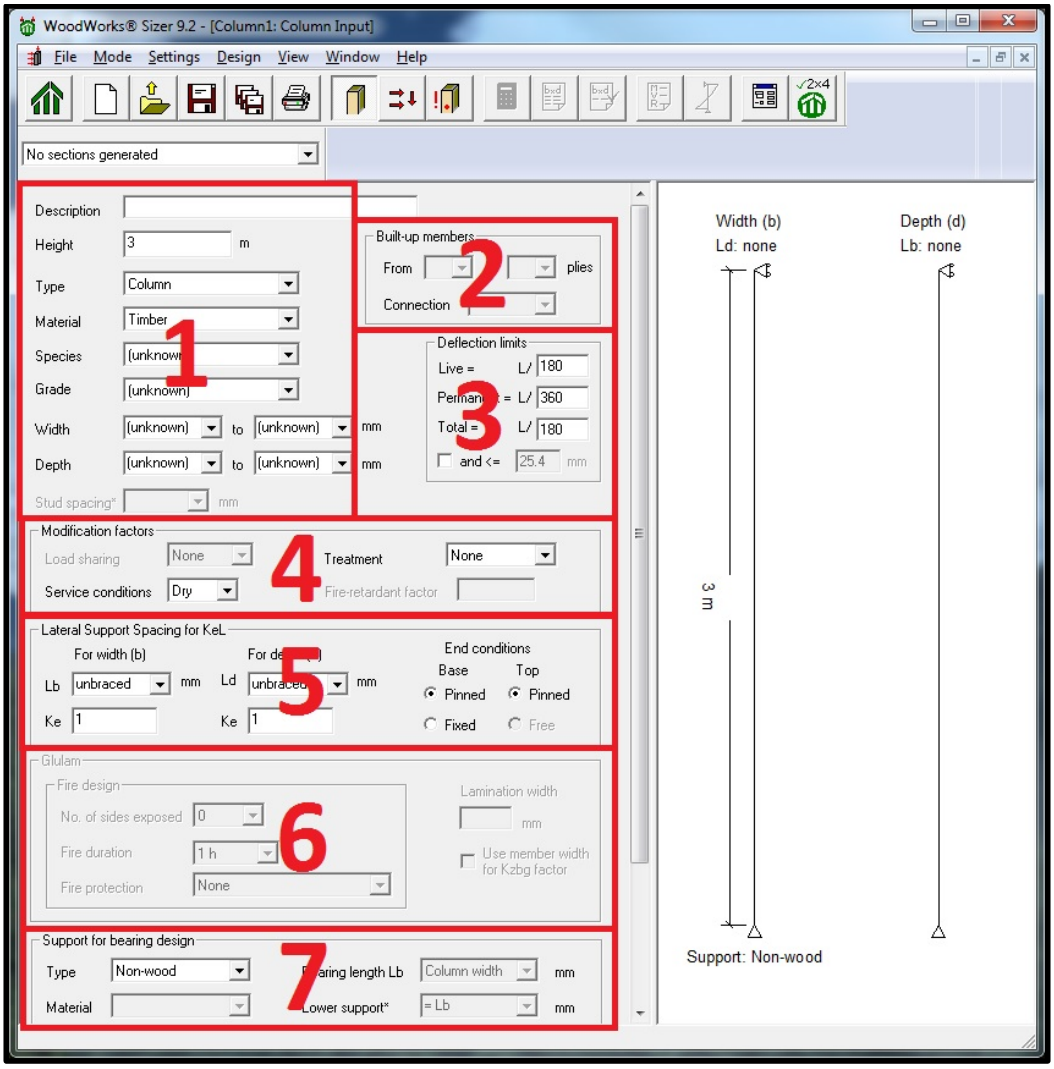



Figure 83: *Column View Input (Canadian Edition)*

The following Table Summarizes where information on each Part of Figure 83 can be found. The section numbers have been hyperlinked for convenience:

| | Section |
|---|---|
| Part | Figure 83 (Canadian) |
| 1 | 11.2.1 |
| 2 | 11.3 |
| 3 | 11.4 |
| 4 | 11.5 |
| 5 | 11.6 |

| | |
|---|---|
| 6 | 11.7 |
| 7 | 11.8 |

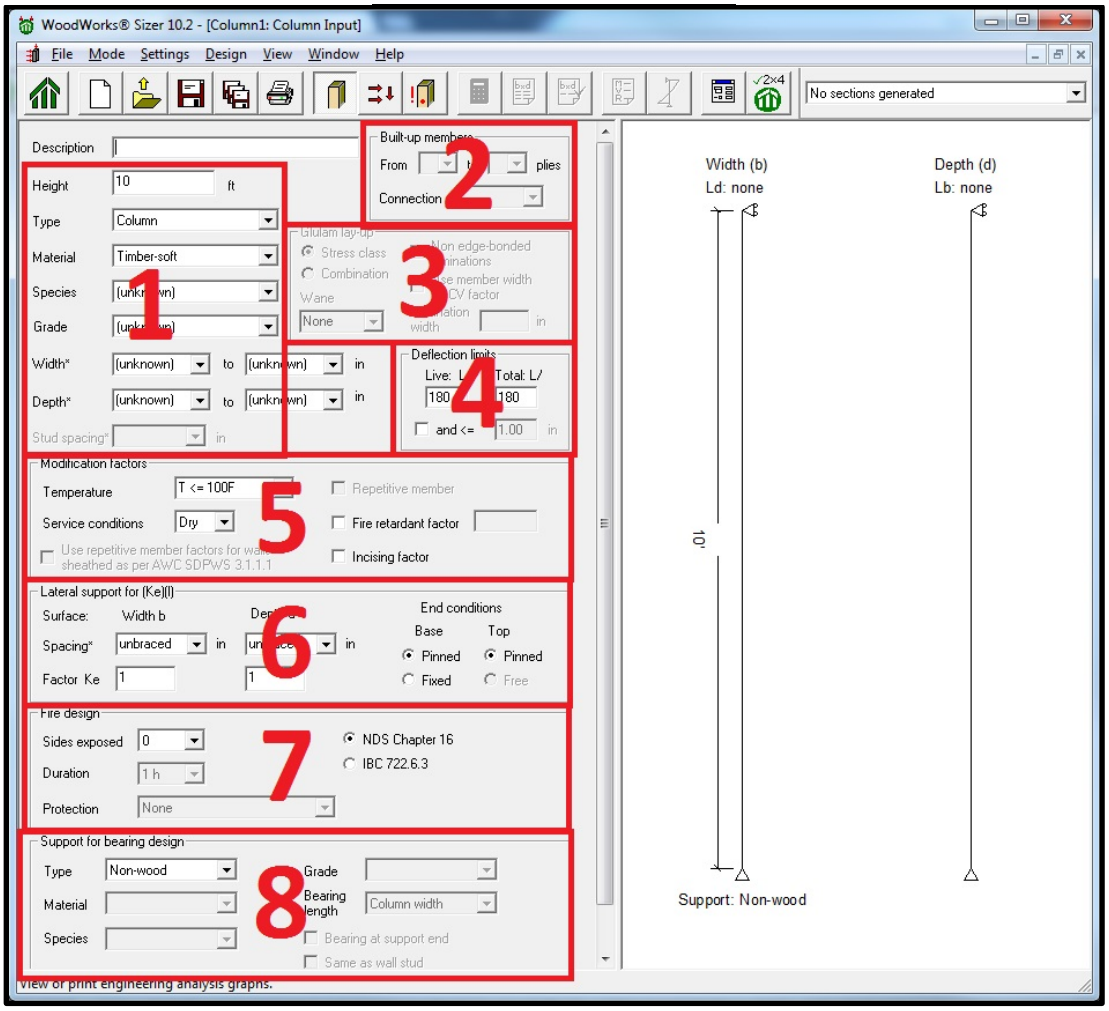



Figure 84: *Column View Input (U.S. Edition)*

The following Table Summarizes where information on each Part of Figure 84 can be found. The section numbers have been hyperlinked for convenience:

| | Section |
|---|---|
| Part | Figure 84 (U.S.) |
| 1 | 11.2.1 |
| 2 | 11.3 |
| 3 | 11.3.1 |
| 4 | 11.4 |
| 5 | 11.5 |

| | |
|---|---|
| 6 | 11.6 |
| 7 | 11.7 |
| 8 | 11.8 |

### 9.2.1 Column Details Input

#### 9.2.1.1 Height

This field specifies the height of the column or wall in the current units.

A height must be specified in order to continue with the design.

#### 9.2.1.2 Type

This drop-down list specifies the type of compression member (Column or Wall) to be designed. The default selection is Column.

#### 9.2.1.3 Description

A description of the column can be entered which will appear in the ***Design Results*** output and ***Design Check*** output.

#### 9.2.1.4 Material

This drop-down list specifies the material database to use. Standard choices include timber post, glulam and a number of others. The default for columns is Timber-soft. The default for walls is **Lumber Stud**.

#### 9.2.1.5 Species

This drop-down list specifies the species of wood to use to design the member. Available species depend on the selected database. For example, common choices for softwood timber are D.Fir-L, Hem-Fir, S-P-F, and S.Pine.

*The default selection is (unknown) which forces Sizer to select suitable sections from each species.*

#### 9.2.1.6 Grade or Combination

This drop-down list specifies the grades of wood to use to design the member. Available grades depend on the selected database. For example, common choices for timber are No.2, No.1, and SS.

*The default selection is (unknown) which forces Sizer to select suitable sections from each grade.*

This list box is unavailable if the species is *(unknown)*.

#### 9.2.1.7 Width and Depth

The two width drop-down lists specify the width range of the sections *Sizer* should use when designing the member. Similarly, the two depth drop-down lists specify the depth range. Available widths and depths depend on the selected database, species and grade.

The default selection for each of these drop-down lists is *(unknown)*, which forces *Sizer* to select from a full range of section sizes.

You can limit the section sizes to be considered by selecting appropriate width and depth ranges in the drop-down lists.

These drop-down lists are unavailable if the grade is *(unknown)*.

To select a custom section, enter some non-standard size (in actual dimensions) in each drop-down list. *Note that a custom size may not be commercially available.*

*Note: Sizer performs a design check rather than a design selection if both the width range and depth range specify single values. Sizer cannot design a section if only the width or depth is a custom size. For example, you cannot have one field as (unknown) when the other contains a custom size.*

#### 9.2.1.8 Stud Spacing

Active for walls only, this drop-down list specifies the stud spacing. You can select one of the three standard spacing's of 12 in. (300mm), 16-in. (400mm), or 24 in. (600mm), or enter your own value.

### 9.3 Built-up Members

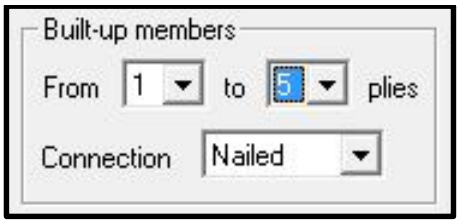



Figure 85: *Built-up Columns*

If the selected material can be used in built-up sections, these two drop-down lists specify the range of plies to use. The default selection is *(unknown)*, which forces *Sizer* to select suitable sections of a range of plies.

For built-up members, the connection type used to assemble the member can also be specified. This affects the resistance of a member.

If the material cannot be used in built-up sections, these lists are disabled.

#### 9.3.1 Glulam (U.S. Only)

This field input allows the user to select between glulam Stress Class properties or glulam Combination properties. For those combinations of Southern Pine which contain wane on one or both sides, the user has the option of specifying whether or not to include the strength reduction by checking the box. The non-edge-bonded laminations check box controls whether a reduction factor is to be applied to the $F_{vy}$ value. The action is only available when the maximum lamination width is less than the member width, so that edge-bonding is possible. The dialog box also allows users the option of specifying whether or not to include the volume factor ($C_v$) in the analysis.

### 9.4 Deflection Limits

These fields allow you to specify the deflection limits to be used for design, based on the span (example L/360) and in absolute terms (1 inch). Default deflection limits are entered in **Settings, Design** tab. See also the "**Loads**" button and "*Load types and combinations*" for long term deflection (creep) and load reductions (US only).

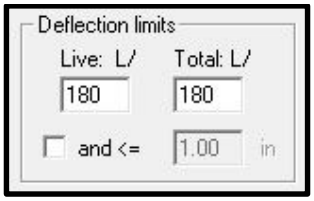


Figure 86: *Deflection Limits (U.S.)*

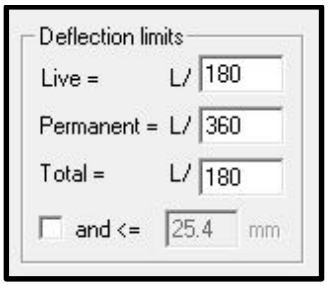



Figure 87: *Deflection Limits (Canadian)*

## 9.5 Modification Factors

### 9.5.1 Temperature (U.S. Only)

The default is T<100°F. Other values for T are 100°F<T<125°F and 125°F<T<150°F.

### 9.5.2 Service Condition

This drop-down list specifies either Wet or Dry service conditions for the column or wall. The default is **Dry**.

### 9.5.3 Fire-Retardant Factor

This field permits you to specify the reduction in compression, bending, shear, and deflection resistance of wood treated with fire-retardant chemicals.

### 9.5.4 Treatment (Canada Only)

When you select Fire-retardant treatment or Preservative treatment from the drop-down, an input field becomes active for you to input the corresponding modification factor.

### 9.5.5 Repetitive Member Factor (U.S.)/Load Sharing (Canadian)

This control allows you to specify if a sharing factor is to be applied. The system factors from tables 2306.2.1 (IBC) and 3.1.1.1 (SDPWS) with factors ranging from 1.5 for 2 x 4 studs and 1.15 for 2 x 12 and higher, are implemented when the following:[7,9]

- the Repetitive Member checkbox is checked
- sheathing conditions checkbox is checked
- the joist spacing is no more than 16 inches

### 9.5.6 Incising Factor (U.S. Only)

This field allows you to specify whether an incising adjustment factor should be applied.

## 9.6 Lateral Support Spacing and End *Connections*

Select *Pinned* if the column base does not resist bending, or *Fixed* for a column base that does resist. For fixed column base, select pinned or free for the column top. Free column tops do not resist bending or translation.

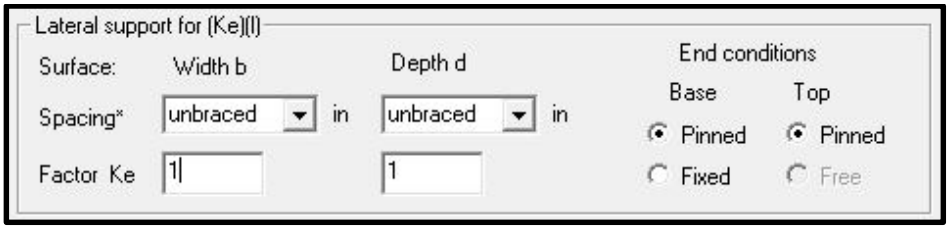



Figure 88: *Lateral Support Input*

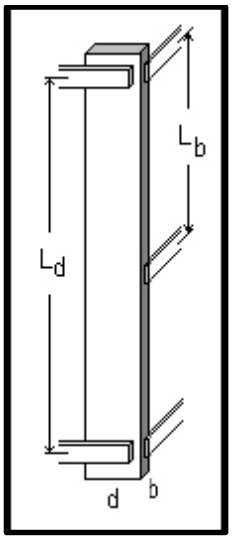



Figure 89: *Column Lateral Support*

#### 9.6.1.1 *$K_eL$ for Width*

The $L_b$ field specifies the unsupported length associated with width b. The $K_e$ field specifies the effective length factor. Its default value is 0.8 for columns with a fixed base, and 1.0 for columns with a pinned base. For free tops, the default value is 2.1 for U.S. and 2.0 for Canada.

#### 9.6.1.2 *$K_eL$ for Depth*

The $L_d$ field specifies the unsupported length associated with depth d. The $K_e$ field specifies the effective length factor. Its default value is 0.8 for columns with a fixed base, and 1.0 for columns with a pinned base. For free tops, the default value is 2.1.

### 9.7 Fire Design

Clicking on this button opens the Fire Resistance and Treatment dialog that allows you to specify fire endurance design criteria and a fire-retardant factor.

### 9.7.1 Fire Design (U.S.)

The User has the option of selecting whether to calculate the fire resistance following Chapter 16 of the NDS or 722.6.3 of the IBC.[1,7] *Sizer* checks the Fire Endurance Rating of timber and glulam beams or columns. *Sizer* calculates the *Fire Endurance Rating* based on the number of exposed sides and on the loads applied to a member. To activate the fire endurance check, the number of exposed sides must be defined. If the number of exposed sides is set to zero, the program will not perform the fire endurance check.

Fire Endurance Rating is activated through the **Settings/Design** tab.


Figure 90: *Column Mode Fire Design Input (U.S.)*

### 9.7.2 Fire Design (Canadian)

In Canada, Fire Design is only available for Glulam members. The Glulam beam Fire Design options must be activated in the Design Settings.

Users have the option of selecting the number of sides exposed to the fire (ie. 0,3 or 4). There are separate equations in the D-2.11.2 1) a)-d) for columns and beams and for 3- and 4- sided exposure.[3] They depend on the load factor f and the lesser and greater section dimensions B and D. Additional Protection can be provided from ½" gypsum board (increases endurance by 15 minutes), 5/8" gypsum board increases endurance by 30 minutes), and two ply 5/8 (increases endurance by 1 hr).

The maximum lamination width used as the value B in CSA O86-14 7.5.6.5.1 for the calculation of the $K_{zbg}$ factor has until now been stored in the standard glulam database, which cannot be edited. To change this value, you had to create a custom glulam database.[2] Since the construction of glulam layups varies from manufacturer to manufacturer, an easier way of changing this value has been added to the program. You can now enter a lamination width in *Beam Input View* to over-ride the lamination width from the database.

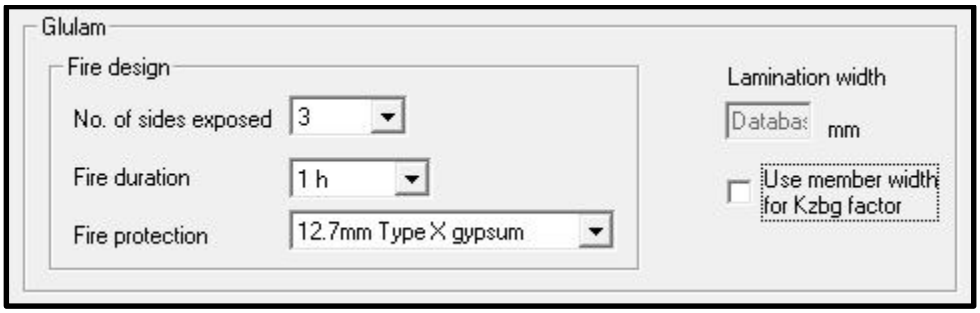



Figure 91: *Column Mode Fire Design Input (Canadian)*

## 9.8 Support for Bearing Design

These fields allow the user to choose the type, material, species, grade and bearing length for the column or wall stud. These fields cannot be left as unknown. The results will be shown in the ***Design Results*** to let the user know if the support bearing is sufficient for the column. For example, *a bottom plate* supporting a lumber stud can be designed if the *Type* is appropriately selected. If "*Same as wall stud*" is not selected, the bottom plate's specific *material, species,* and *grade* can be entered, otherwise the same materials as the wall stud is used.

The checkbox "*Bearing at Support End*" indicates that the supporting beam, sill plate, or wall bottom plate ends at the column or wall, so that the bearing length factor $K_B$ or $C_B$ is not applied.

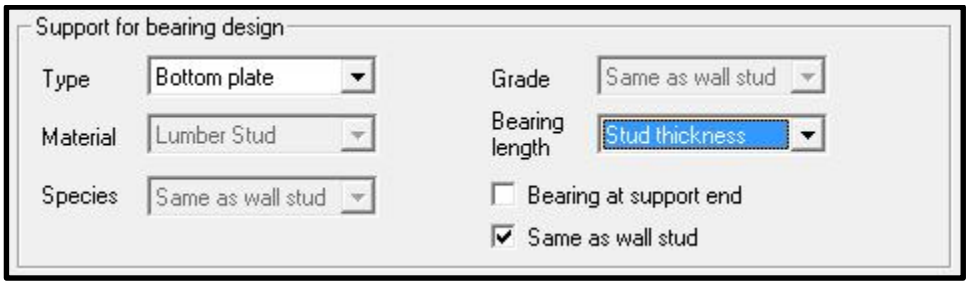



Figure 92: *Column Mode Support for Bearing Design (U.S.)*

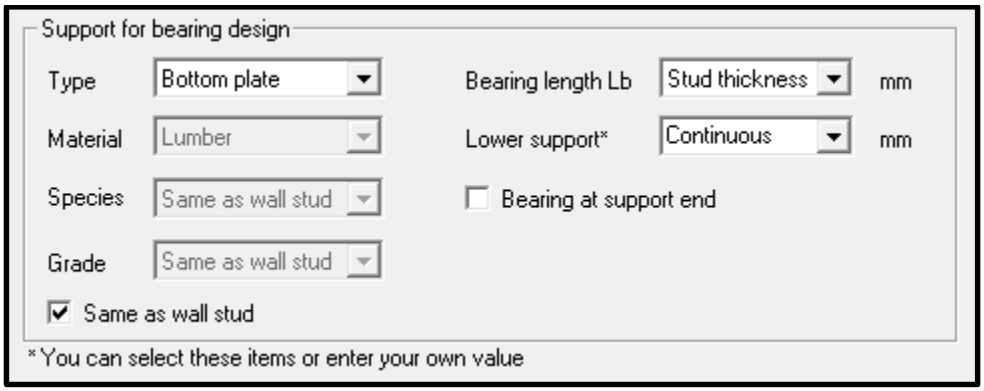



Figure 93: *Column Mode Support for Bearing Design (C*

# 10 Column Mode Loads

Click ***Loads View*** on the toolbar to specify the loading of a column that is to be designed. Figure 94 shows the column load input for the Canadian edition of the software and Figure 95 shows the column load input for the U.S. edition of the software.

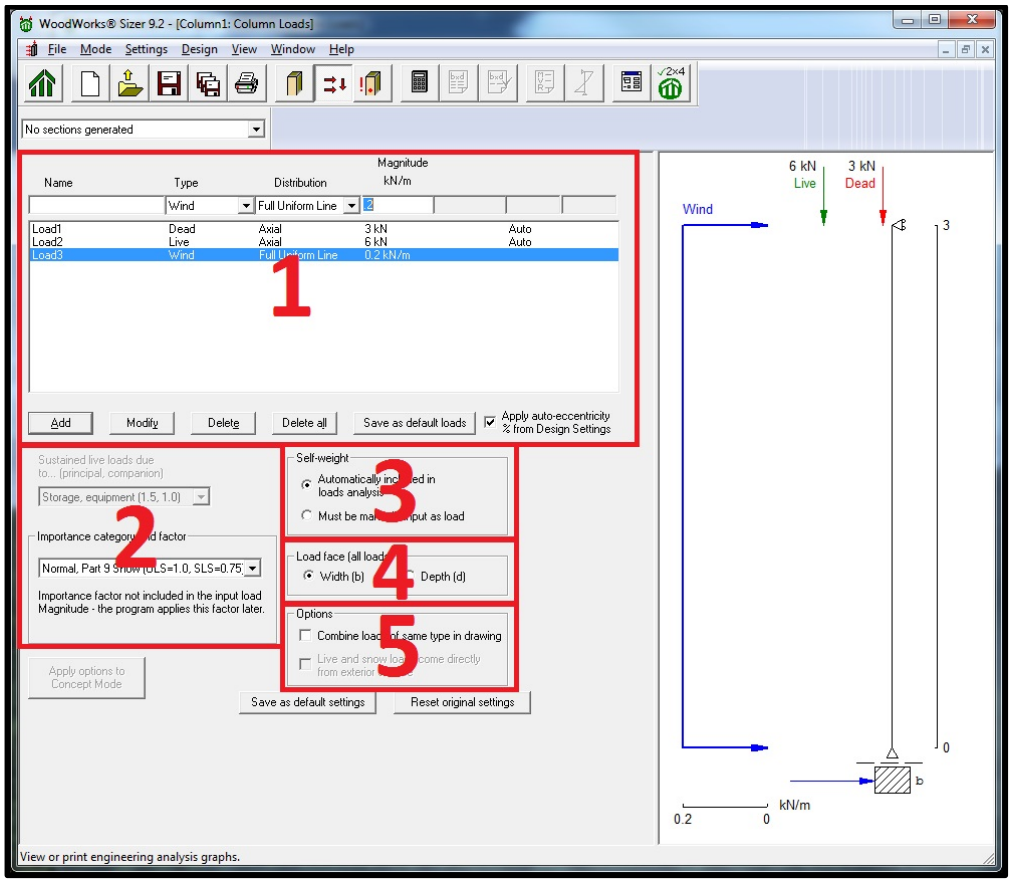



Figure 94: *Column Mode Load View (Canadian Edition)*

The following Table Summarizes where information on each Part of Figure 94 can be found. The section numbers have been hyperlinked for convenience:

| | Section |
|---|---|
| Part | Figure 94 (Canadian) |
| 1 | 12.1 |
| 2 | 12.6 |
| 3 | 12.2 |
| 4 | 12.3 |
| 5 | 12.7 |

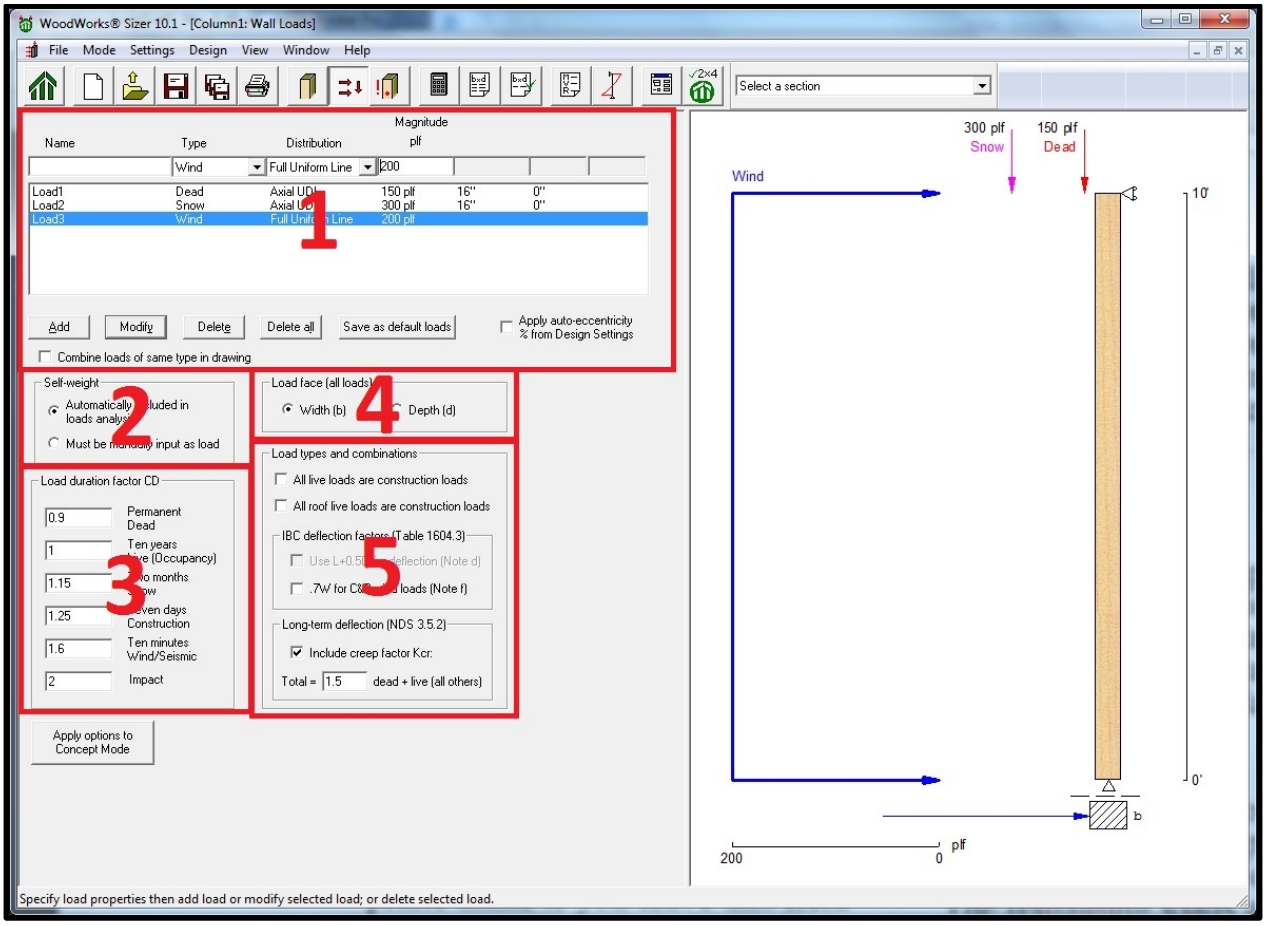



Figure 95: *Column Mode View (U.S. Edition)*

The following Table Summarizes where information on each Part of Figure 95 can be found. The section numbers have been hyperlinked for convenience:

| | Section |
|---|---|
| Part | Figure 95 (U.S.) |
| 1 | 12.1 |
| 2 | 12.2 |
| 3 | 12.4 |
| 4 | 12.3 |
| 5 | 12.5 |

## 10.1 Load Input

### 10.1.1 Name

Use the *Name* field to enter the name of the load you want to apply to the column. (*Sizer* generates an appropriate name if you leave this field blank.) The list box beneath this field contains the names of all the loads you have specified for this column. Click one to select it.

#### 10.1.1.1 Type

This drop-down list specifies the type of load being applied. *Sizer* allows you to select from a variety of load types, including dead, live, roof live (U.S. only), snow, wind, impact (U.S. only), sustained live (storage and contained fluids) and dead (soil) (Canadian only), earthquake, and hydrostatic (Canada only, columns only). The U.S. version also allows all live loads or all roof live loads to be considered a "construction" load, and will use the appropriate duration factor if this is selected.

Depending on the load type specified, *Sizer* will apply the correct load duration factor and load combination factor to each load combination. For the Canadian version, *Sizer* also applies the correct load factor. The default load type is **Dead.**

### 10.1.2 Distribution

This drop-down list specifies the type of load distribution: Axial, Full Uniform Line, Partial Line, Full Uniform Area, Partial Area, Triangular, Trapezoidal, Point and Applied Moment.

Axial loads are applied to the top of the column, where a positive value loads the column in compression. The remaining loads are applied laterally and load the column as a beam.

For walls, an axial load is entered as a uniformly distributed line load along the top of the wall.

*Sizer applies lateral loads to the face identified in the Load Face box of the Column screen.*

### 10.1.3 Magnitude

This field specifies the magnitude(s) of the column load at the beginning and end of the loaded portion of the member (For a Point load, only one load magnitude is specified.). True Uniform loads should have the same magnitude at point a and b. Trapezoidal loads usually have different values for the two magnitudes.

*Note that area loads are converted to and displayed as line loads (plf or kN/m) on the load diagram*

### 10.1.4 Eccentricity

This field applies only to axial loads, with the same eccentricity for all. The bending effect is the same for an eccentricity and a lateral load, when both have positive values.

### 10.1.5 Location From Bottom

This field specifies the location of the selected load. It is active if the load distribution is Partial Line, Partial Area, Triangular, Trapezoidal, Point Load and Applied Moment.

For Partial Line loads, specify a start and an end measurement. For Triangular loads, specify a location where the load is zero and a location for the maximum load. For Trapezoidal loads, specify the locations of the minimum and maximum loads.

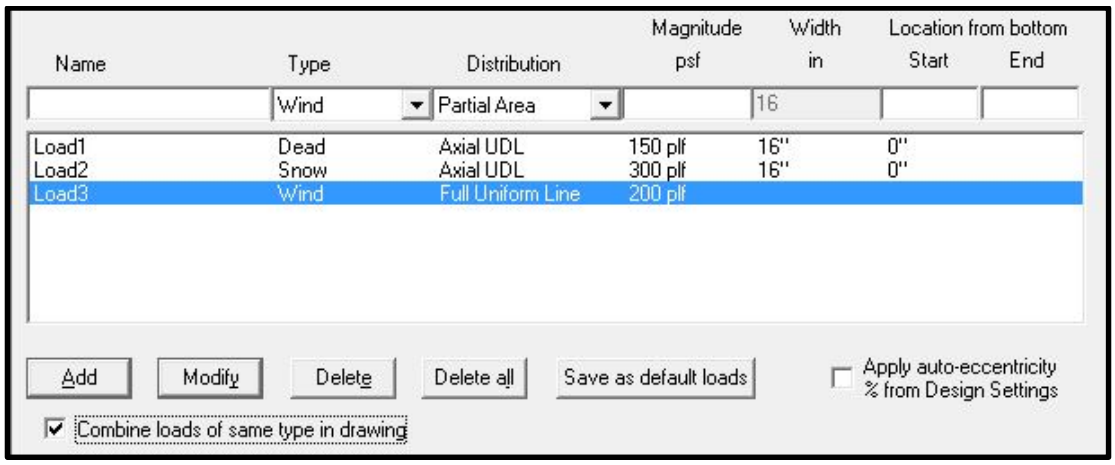



Figure 96: *Column Mode Load Input*

### 10.1.6 Combine Loads of Same Type in Drawing

This option only applies to the loading diagram. When enabled, individual loads are accumulated into a single loading profile for each load type (dead, live, etc.). When not enabled, loads overlap with other loads of the same type. The selected load appears in bold.

## 10.2 Self-Weight

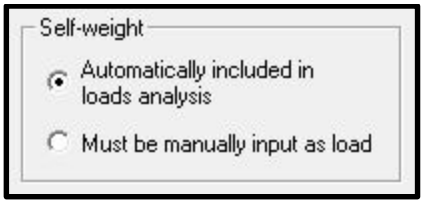



Figure 97: *Column Mode Self-weight Input*

If "*automatically included in loads analysis*" is selected by the user, the self-weight of a member is considered in the design. The actual effects of self-weight are calculated by multiplying the specific gravity of the material times the area of the section. This load is then added to the dead load when checked. Alternatively the self-weight can be ignored or manually added by selecting "*Must be manually input as load*".

## 10.3 Load Face


Figure 98: *Column Mode Load Face*

### 10.4 Column Mode Load types and Combinations

Select ***Width*** if lateral loads such as wind are applied to the narrow face of the member, or ***Depth*** if applied to the wide face. All lateral loads and eccentric axial loads cause bending about the same axis.

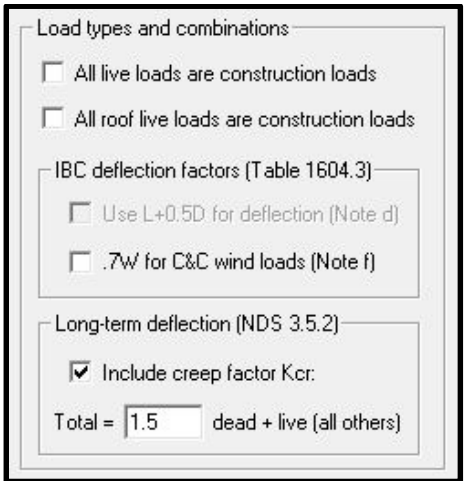



Figure 99: *Column Mode Load types and Combinations*

*Sizer* combines loads based on the Allowable Stress Design method using the basic load combinations in the IBC/ASCE (US) and Limits States Design load combinations (Canada).[7,8]
The User has the option of selecting whether or not to consider live and roof loads as construction loads, by clicking the appropriate box.

### 10.5 Load Duration Factors (U.S. Only)

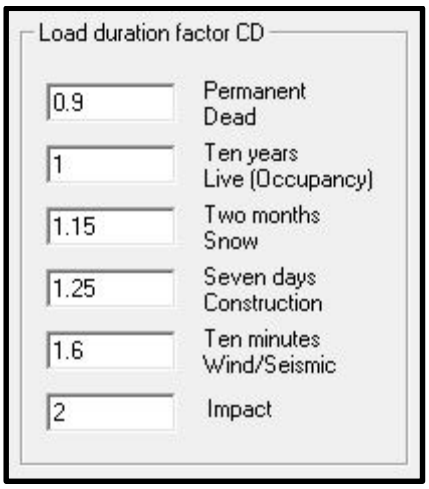



Figure 100: *Column Mode Load duration factor CD*

*Sizer* applies load duration factors according to the load type. These factors are set to the standard NDS (U.S. version) or CSA O86 (Canadian version) values by default but can be changed using this form. (*Note that in the Canadian version, when D > L, Sizer determines the default factor for standard term loads according to the equation shown in CSA O86-14 6.3.2.3 - refer to the online help for details*).[1,2]

## 10.6 Load Distribution and Combinations (Canadian)

Active whenever there is both a live or sustained live load and a snow load on the member.

To comply with NBC 4.1.5.5, this setting allows the user to create a set of load combinations without live and snow loads in the same combination.[3]

### 10.6.1 Sustained Live Loads

The program now creates separate load combinations for D (+ Ds + H) + Ls or D (+Ds + H) + Lf, in addition to the combinations D + (Ds + H) + Ls/f + L, when occupancy live loads and sustained loads are on the same member. Refer to the section on $K_D$ factor for the reason, and on the on-line help for a complete explanation.

### 10.6.2 Importance Category and factor

Once a Live, Snow, Wind, or Earthquake load type is selected in the load input, the Importance category and factor will come available which allows the importance category for the load to be specified (ie. Low, Normal, High, Post-disaster or Normal Part 9 Snow).

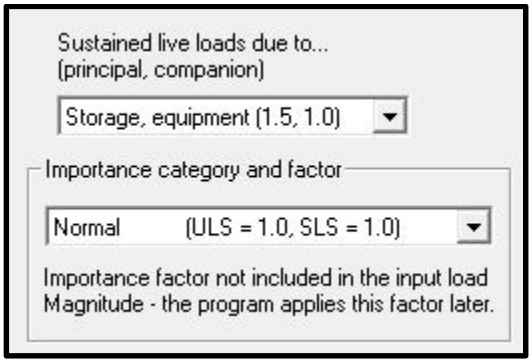



Figure 101: *Sustained Live Loads and Importance category and factor*

## 10.7 Options

Once more than one type of load is added, the option “Combine loads of same type in drawing “ will become available, which affects the output of the software, where loads of the same type will be combined in the drawing of the column.

Whenever a live or sustained live load and a snow load is input on a column, the “Live and snow loads come directly from exterior surfaces” becomes available. By default it will it unchecked, but it is possible to have it checked by default using the “Save as default settings” button found in the *Load Input View*. This option was added as to comply with NBC 4.1.5.5, which allows you to create a set of load combinations without live and snow loads in the same combination.[3]

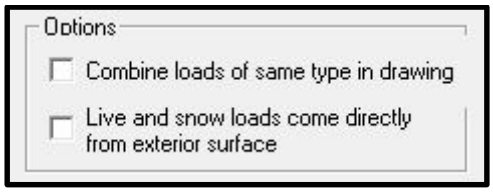



Figure 102: *Column Mode Load input Options*

## 10.8 Deflection Factors (U.S. Only)

### 10.8.1 IBC Deflection Factors (US Only)

There are two leniencies offered in the IBC related to deflection. These optional IBC deflection factors can be selected using two checkboxes.[7]

### 10.8.2 Use L+0.5D for deflection (note d)

The first allows the deflection resulting from L + D to be calculated using L+ 0.5D for wood structural members having a moisture content under 16 percent at the time of installation and used in dry conditions.

### 10.8.3 0.42W for C&C Wind Loads (note f)

The second option is related to components and cladding wind loads, where deflection may be calculated using only 70 percent of the wind load.

### 10.8.4 Long-term deflection (NDS 3.5.2)

The NDS provides guidance on considering the effects of long term deflection by increasing the contribution of dead load by a creep factor, $K_{cr}$, that typically ranges from 1.5 to 2; any value can be entered if the "*Include creep factor*" is checked.[1]

### 10.8.5 Apply Options to Concept Mode


Figure 103: *Column Mode Apply Options to Concept Mode Button*

This button in *Beam Load View* and *Column Load View*, is only active when there is a *Concept* mode file open, and allows the application of the following settings to have an effect on *Concept* mode. This items can only be accessed from *Beam* or *Column* mode, and are *Self-weight*, *Line Loads applied over design span only, Long-term deflection factor*, *Use L+0.5D for deflection*, *Load duration factors CD*, *Load combinations from…*, *Temperature* (from *Beam View*).

# 11 Column Mode Points of Interest

Click the ***points of interest*** button on the toolbar to investigate the shear and moment at any point along the length of column or beam. A point of interest is generated by specifying a location to perform the analysis. Now click ***Add*** to add this to the list. Several points of interest can be specified.

After performing a design, the point of interest result will be shown in the Diagrams window and in the Analysis results output.

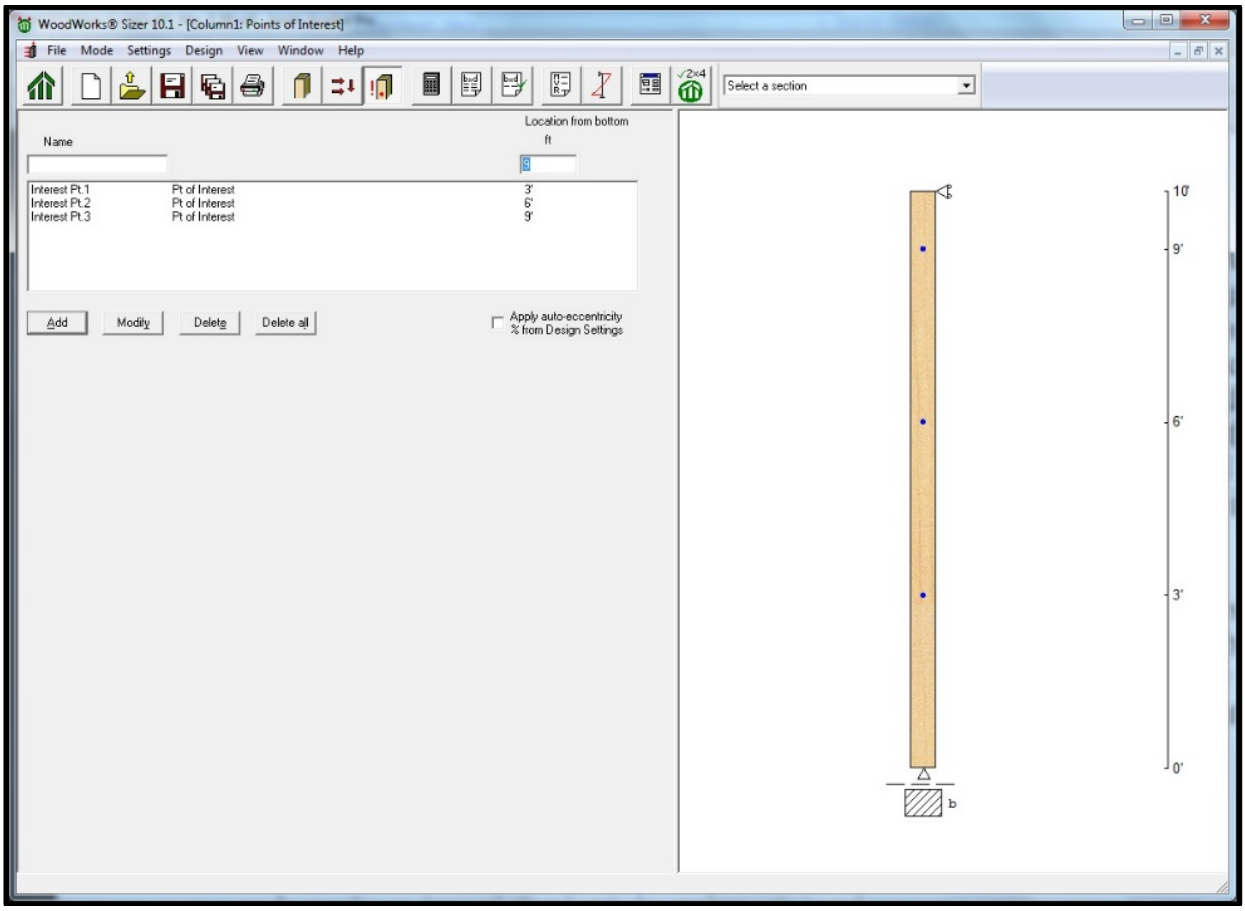



Figure 104: *Inputting Points of Interest*

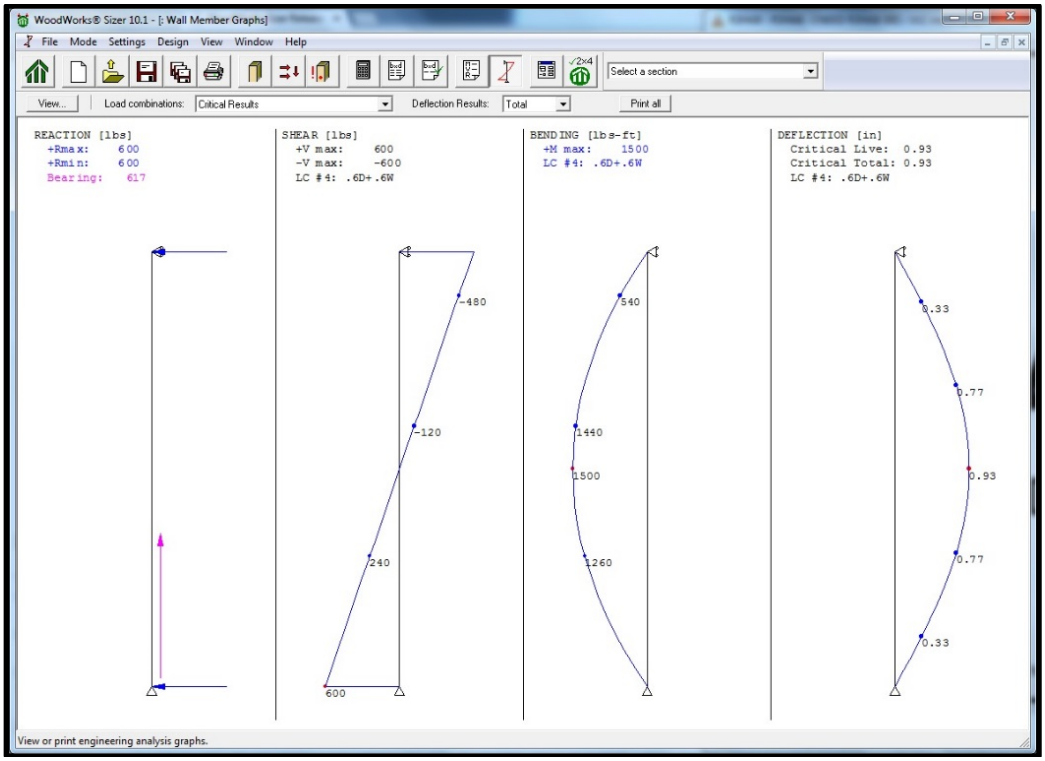



Figure 105: *Points of Interest on Shear and Bending Moment Diagrams*

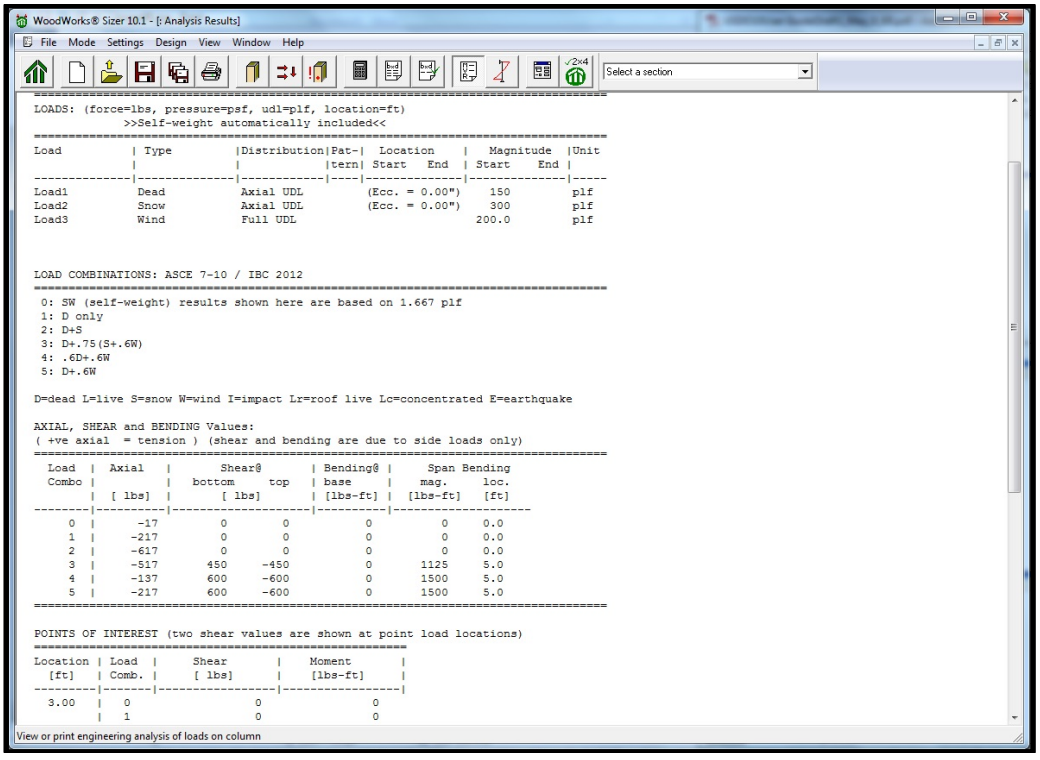



Figure 106: *Points of Interest Analysis Results*

# 12 Column Mode Design Process

## 12.1 Starting the Design Process

To start the design process, click ***design*** on the toolbar. *Sizer* performs an analysis and design of your column with the information you entered and automatically displays the results.

If you specified *(unknown)* for some entries, *Sizer* selects a series of suitable sections that provide acceptable results. If there are no *(unknowns)* and you have not specified ranges in the width or depth fields, *Sizer* does a design check on the specified section and verifies that the results are within the selected design code's limits.

The number of sections that *Sizer* selects depends on how many fields you specified as *(unknown)* in the column screen.

*Sizer prompts you to save the current project prior to doing the design. To change this, choose one of the* ***Save Before Design*** *options from the* ***Settings*** *menu.*

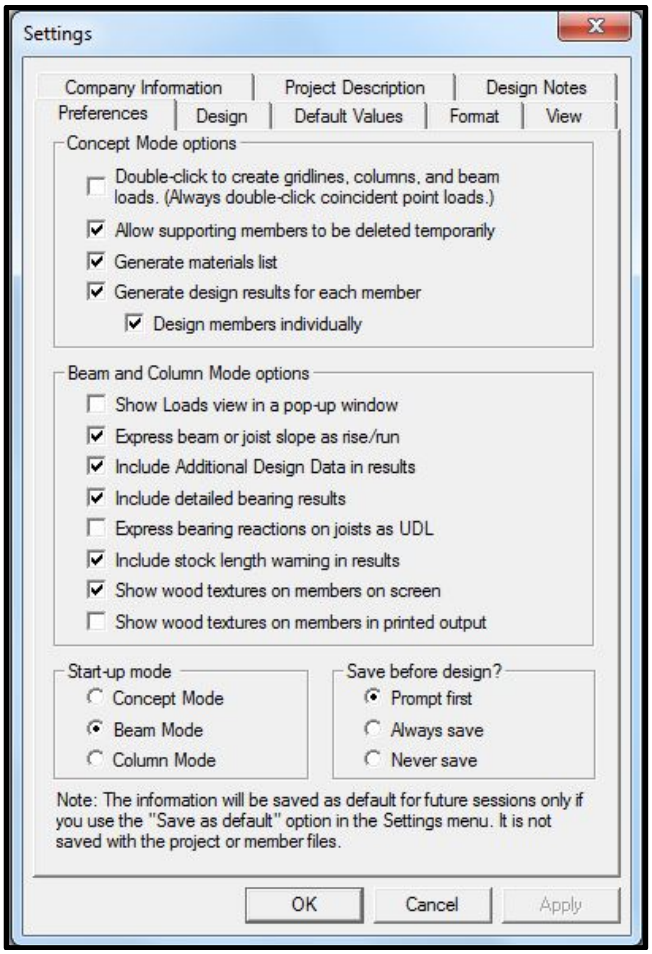



Figure 107: *Column Mode Settings Preferences*

### 12.1.1 Save Before Design (Prompt first)

Tells *Sizer* to prompt you to save the current worksheet prior to starting the design process. This is the default.

### 12.1.2 Save Before Design (Always save)

Tells *Sizer* always to save the current project automatically (without prompting you) prior to starting the design process.

### 12.1.3 Save Before Design (Never save)

Tells *Sizer* not to save the current project prior to starting the design process. *Sizer* will not even prompt you to save.

### 12.1.4 Viewing and Printing the Results

Once *Sizer* has designed the column, it creates results files. *Sizer* then gives you several options for viewing the results.

## 12.2 Selecting Sections for Design Check

*Sizer* provides you with an extensive list of suggested sections for the column it is designing. To get more details about a single section size, you can perform a design check on that section.

After running an initial design, select the section you want to check from the drop-down list entitled ***Suggested Column Sections*** at the right side of the toolbar. *Sizer* automatically fills in the Species, Grade, Width and Depth fields with the appropriate values. *Sizer* also automatically performs a design check on the selected section.

The list of suggested sections on the toolbar remains available until you perform another design (rather than a design check).

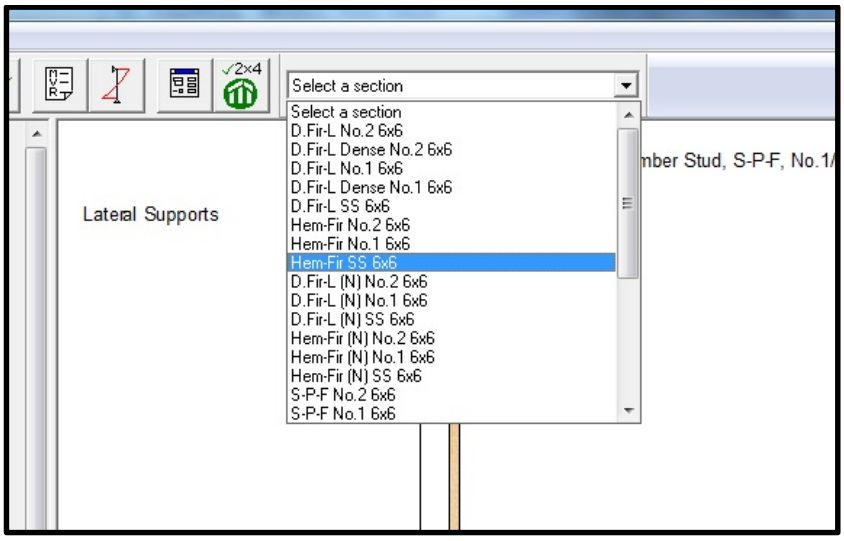



Figure 108: *Selecting Sections for Column Design Check*

## 12.3 Analysis Results

Click the ***analysis*** button on the toolbar to see the analysis results (maximum shears, bending moments and so on) for your column, beam-column or stud wall in tabular form.

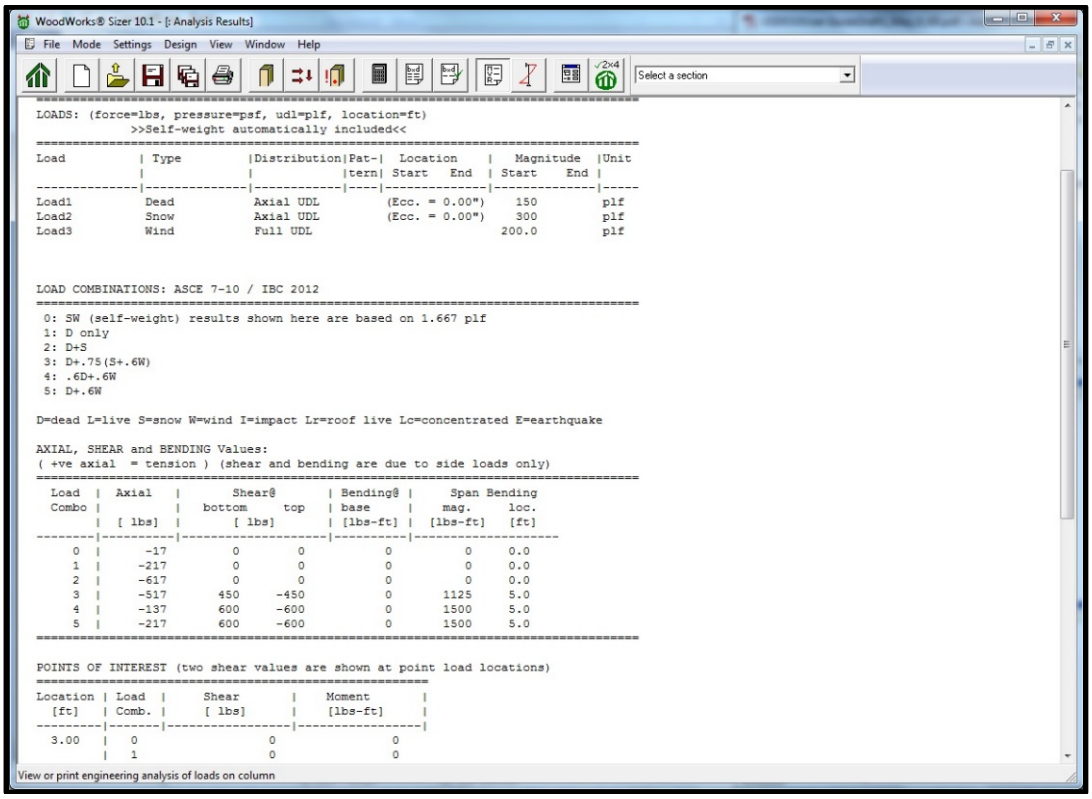



Figure 109: *Column Mode Analysis Results*

## 12.4 Design Results

If you specified any field as *(unknown)* for your column or stud wall, *Sizer* computes all the possible column, beam-column or stud wall sections during the design process. It also computes the ratios of analysis/design values for axial, bending, combined axial and bending, shear, and displacement for each section.

To see these results, click on the ***results*** button on the toolbar menu or choose ***Design Results (Suggested Sections)*** from the ***View*** menu.

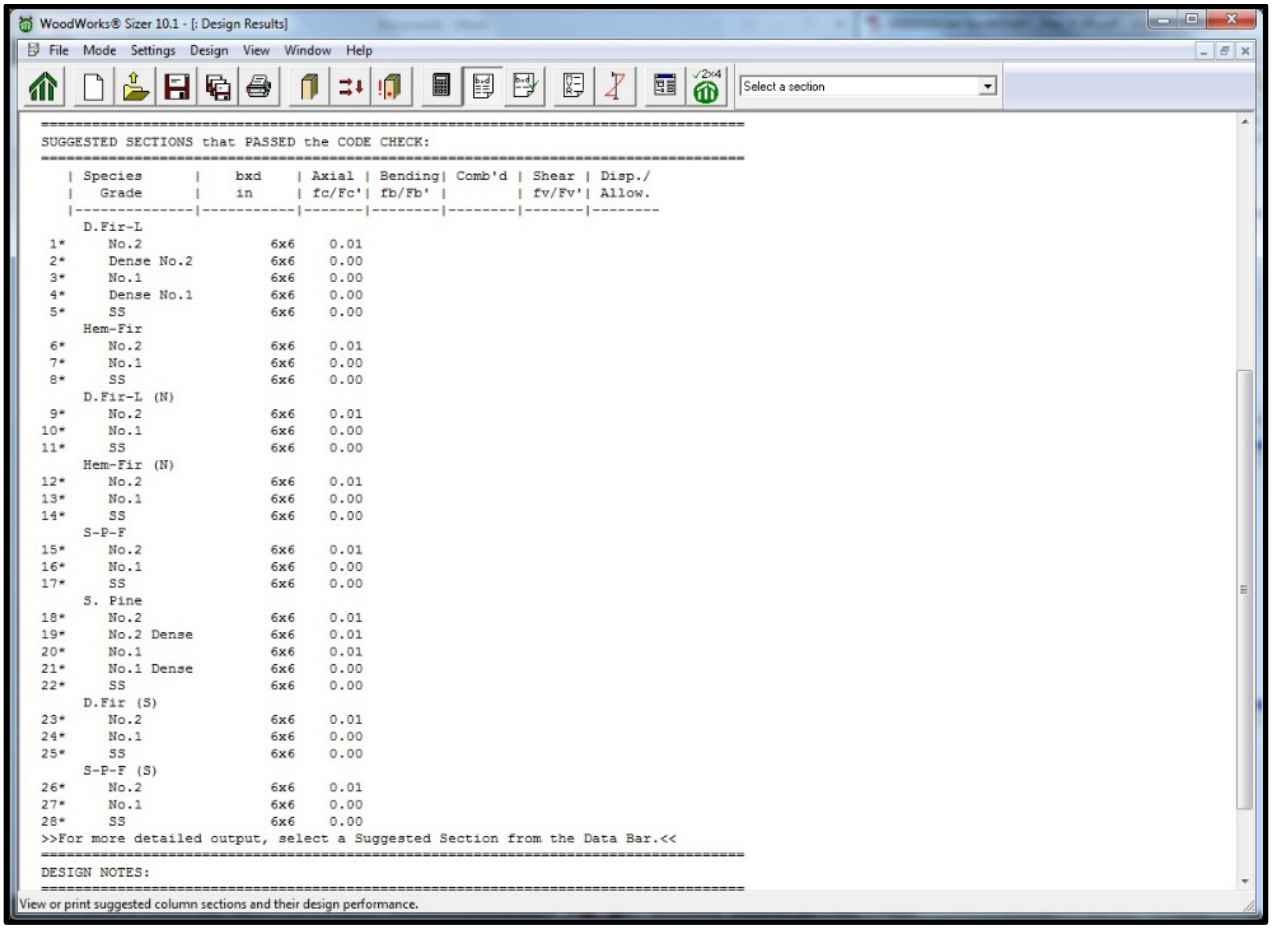



Figure 110: *Column Mode Design Results*

## 12.5 Design Check

If you requested a specific column, beam-column, or stud size, *Sizer* performs a design check and computes analysis and design values (for example axial, shear, and bending moment) for the section.

To see these results, none of the parameters can be left as *(unknown)*. Then click ***check*** on the toolbar.

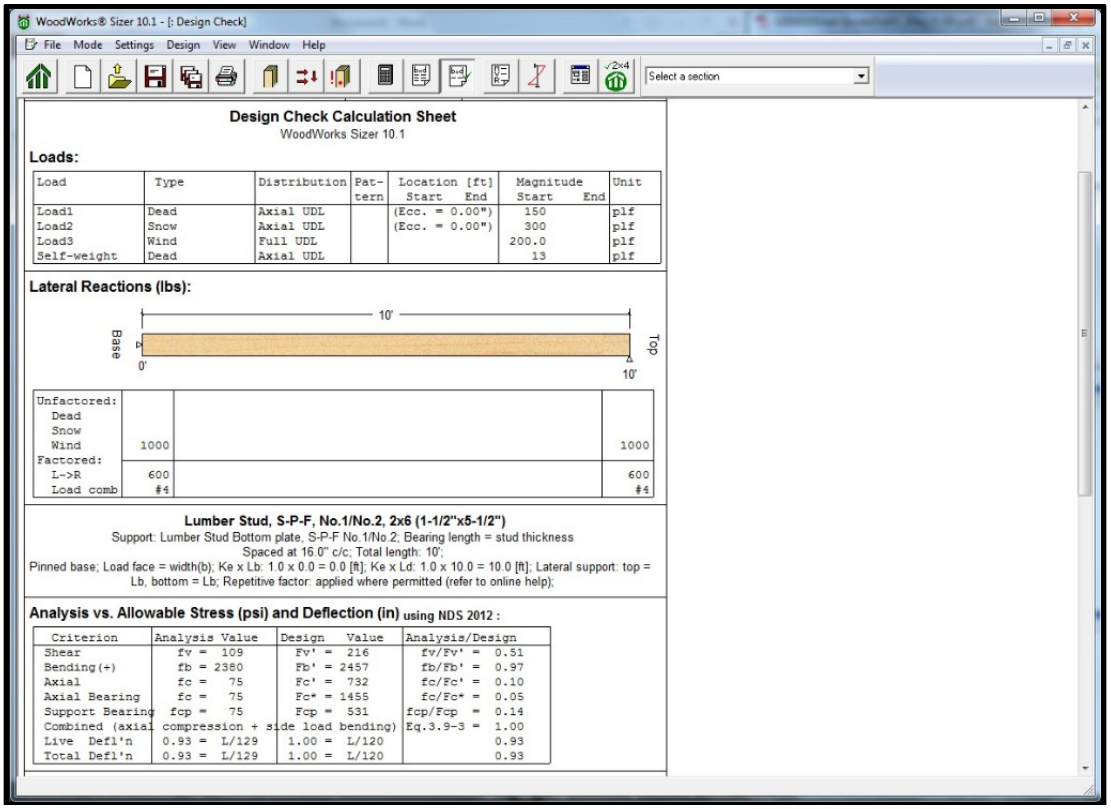



Figure 111: *Column Mode Design Check*

## 12.6 Printing Results

To print results, click on the ***print*** button when the result file or diagram screen that you wish to print is on screen. To batch print the analysis diagrams for all load combinations, press ***Print All.***

## 12.7 Analysis Diagrams

Click the ***diagram*** button on the toolbar to view the analysis diagrams. *Sizer* creates five analysis diagrams; the load Envelope, Support, Shear, Bending, and Deflection diagrams.

To print the current diagram, click the ***print*** button on the toolbar

- ***View…*** – This button controls which analysis diagrams are shown, including the Load Envelope, Reaction, Shear, Moment and Deflection diagrams. The Load Envelope diagram is not available when *Critical Results* is selected as the load combination.
- ***Load Combinations*** – The diagram view results shown are based on the load combination selected in this pull-down menu. When Critical Results is selected, the analysis diagrams shown are for the worst case results of all load combinations. Load combination numbers shown correspond to those used throughout *Sizer* (Analysis Results, Design Check, diagrams, etc.).

# 13 Concept Mode

For a quick overview of how concept mode can be utilized to design structures see the following tutorial - https://www.youtube.com/watch?v=a0AqTKNOW78&feature=youtu.be

By default, *Sizer* starts in *Concep*t mode. If you wish, you can reconfigure *Sizer* to start in *Beam* mode or *Column* mode by changing the default ***Startup Mode*** option in the ***Settings*** menu under ***Preferences***. To switch to Concept mode at any time, choose ***Concept*** from the ***Mode*** menu.

The *Concept* mode graphical work area allows you to quickly design a complete structure using columns, walls, beams and joists. The Concept mode will only design for gravity type loads.

The *Concept* mode is meant to be used as a preliminary design step. For a more detailed design, individual members can be imported from the Concept mode to Beam or Column mode.

Click items in the work area to select them. If you hold down the ctrl key when you click, you can add or remove items from the currently selected set. You can choose ***Select All*** from the ***Edit*** menu to select all of the items in the current view.

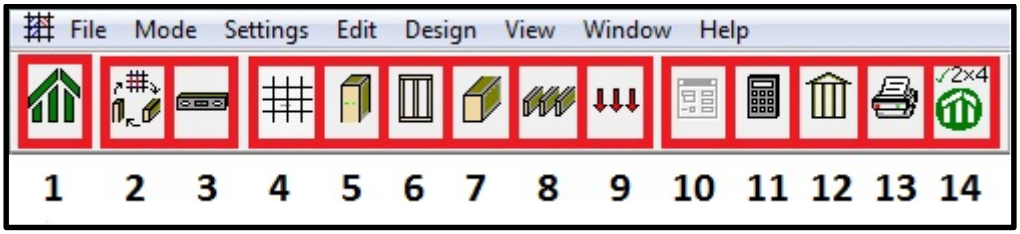



Figure 112: *Column Mode ToolBar*

1. Create or Manage Project
2. Switch Modes
3. Floor and Roof Levels
4. Grid View
5. Column View
6. Wall View
7. Beam View
8. Joist View
9. Load View
10. Material Design Groups
11. Run
12. Elevation View
13. Print
14. Materials *Database Editor*

### 13.1.1 Concept Mode Design Assumptions

Concept Mode Design Assumptions include:

- Columns and Walls are pinned at both ends
- Combined (Axial and lateral) load cannot be applied
- No Bearing Design
- No Pattern Loading
- Cannot define eccentric loading
- Beam fully supported at top edge

It is possible to refine the design of a member after running the initial design, by transferring a member into Beam or Column Mode.

## 13.2 Levels

Click on the ***level*** button on the toolbar menu. The Levels dialog opens and allows you to specify the elevation for each level of your structure.

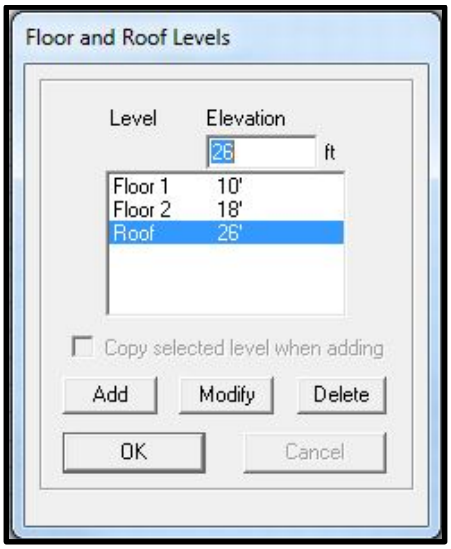



Figure 113: *Floor and Roof Levels*

To add level elevations, enter a new value and click ***Add***. The maximum number of new levels is six.

When design of a structure has already begun, the option of copying the selected levels to the new ones is visible. *Sizer* will not, however, convert roof joists to floor joists or vice versa.

To modify any existing level elevation, select it, enter a new value and click ***Modify***.

To delete a level elevation, select it and click ***Delete***.

Click ***OK*** to save your changes or click ***Cancel*** to discard your changes.

The Levels dialog is also used to change to different levels. Select a level by clicking on it so that it is highlighted. Click ***OK*** and *Sizer* will change to the selected level. Another way to change to different levels is by using the drop-list in the toolbar.

# 14 Concept Mode Design Groups

## 14.1 Why they are used

The *Sizer* design process is based on “design groups”. Design groups allow you to group similar members together for the design process. *Sizer* determines the lightest section size that can be used for each of the design groups, based on the most critical member in each group.

You can make your group divisions as coarse or as fine as you wish. Anything from a single group for all beams to a separate beam group for each beam is acceptable. However, a single group for all beams will likely result in excessively large section size for many beams. Similarly, a separate beam group for each beam (while determining the optimal size for each beam) results in an excessively large number of different sections for construction.

## 14.2 How to create them

Design groups are created using a dialog, as described in the following sections. To help you get started, *Sizer* provides default design groups for each major structural element:

- Column for columns
- Wall for walls
- Beam for beams
- FloorJoist for floor joists
- RoofJoist for roof joists

## 14.3 Group Dialog

To open the group dialog for each structural element, first select the appropriate “View” (ie. Column View, Wall View, Beam View, etc.), then click on the *Material Design* groups button to open the dialog field.

To create a new group, enter appropriate values in the dialog fields and then click ***Add***.

To change a design group, select the group from the list, change any of the values in the dialog fields and then click ***Modify***.

To delete a design group, select the group from the list and then click ***Delete***.

## 14.4 Column Groups

Click ***column*** on the toolbar to change the view to Columns. Now click ***Groups*** on the toolbar. The dialog box named *Column Design Groups* opens.

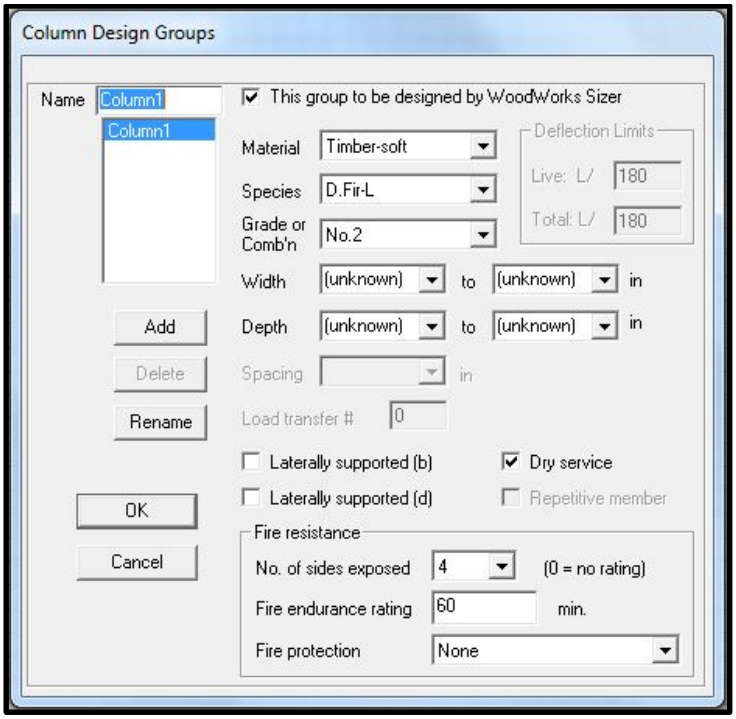



Figure 114: *Column Design Groups*

## 14.5 Column Dialog Items

### 14.5.1 Name

This field contains the name of the currently selected group. The list beneath contains all the column groups created. Click on a group to select it. Once highlighted, you can modify or delete it.

### 14.5.2 To Be Designed

Select this check box if the design group is to-be-designed. Leave it unchecked if the design group is not-to-be-designed. Unchecking it disables all field selections. Either design group supports and transfers loads.

### 14.5.3 Material

This drop-down list specifies the material database. The standard choices include timber, multi-ply lumber, glulam and a number of others. The default for columns is **Softwood Timber.**

### 14.5.4 Species

This drop-down list specifies the species of wood to use for the design process. Available species depend on the selected database, but common choices for timber are D.Fir-L, Hem-Fir, S–P–F and S.Pine. The default is the first species in the list.

### 14.5.5 Grade or Combination

This drop-down list specifies the grades of wood to use for the design process. Available grades depend on the selected database, but common choices for timber are No. 2, Dense No. 2 (U.S. Only), No.1 and Dense No.1 (U.S. Only). The default is No. 2.

### 14.5.6 Width

These two drop-down lists determine the width range of the sections to be considered in the design process. Available widths depend on the selected database, species and grade. The default selection is *(unknown)* for each of the lists, which forces the design process to select from the full range of section widths. To specify a limited range of widths to be considered, select a different width in either or both of the lists.

### 14.5.7 Depth

These two drop-down lists determine the depth range of the sections to be considered in the design process. Available depths depend on the selected database, species, grade and width. The default selection is *(unknown)* for each of the lists, which forces the design process to select from the full range of section depths.

To specify a limited range of depths to be considered, select a different depth in either or both of the lists.

### 14.5.8 Lateral Support

Designers have the option of specifying whether either the width or depth of the column is laterally supported. Leaving the criteria unchecked means no lateral support is to be provided to the column.

### 14.5.9 Dry Service

This check box is checked if the service condition of the design group is dry. If the service condition is wet, this box should be unchecked.

### 14.5.10 No. Of Sides Exposed (U.S. Only)

Select the number of sides of the member that are exposed for calculating fire endurance rating. The selections are 0, 3 or 4 sides. Select 0 to indicate that the member is fully sheathed and no fire resistance calculation is to be performed.

### 14.5.11 Fire Endurance Rating (U.S. Only)

Enter the required fire endurance time. Do not enter a value greater than 60 minutes as this falls outside the research upon which the methodology is based. *Sizer* will attempt to find a section whose fire endurance meets or exceeds the value you enter, according to IBC 721.6.3.[7]

### 14.5.12 Fire Protection

*Sizer* allows user to select whether additional fire protection is to be provided to the column. The Designer has the option of selection none, 15.9 mm Type-X Gypsum, or 2-ply 15.9 mm Type-X Gypsum.

### 14.5.13 Column Assumptions

*Note that Sizer defaults to the following assumptions:*

- Columns are pinned at both ends.
- Columns are laterally restrained only at the ends.

## 14.6 Wall Groups

Click ***wall*** on the toolbar to change to Walls view. Now click ***Groups*** on the toolbar. The dialog box named *Wall Design Groups* opens.

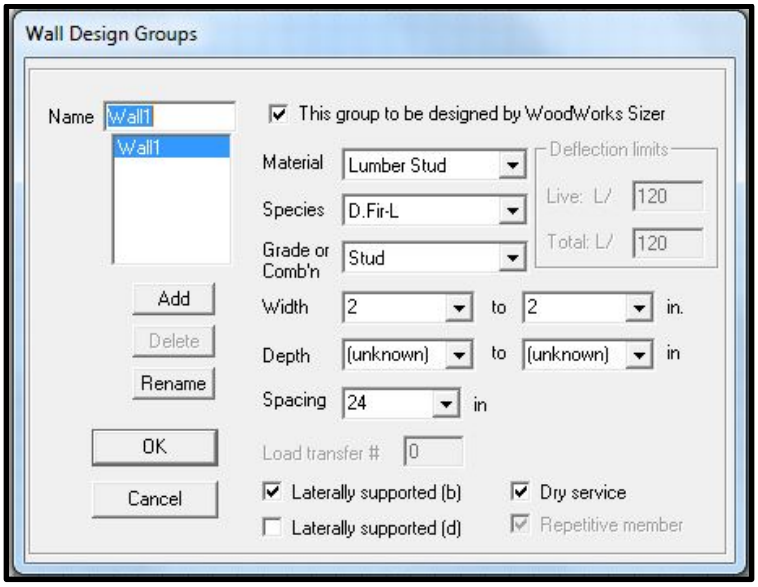



Figure 115: *Wall Design Groups*

### 14.6.1 Wall Dialog Items

When creating design groups for walls, you use a dialog similar to the one used to create design groups for columns. The additional information in the beam dialog is described below.

### 14.6.2 Spacing

This drop-down list specifies the stud spacing. You can select one of three standard spacing's or enter your own value.

### 14.6.3 Repetitive Member Factor (U.S.) Load Sharing System Factor (Canadian)

*Sizer* assumes that all walls act as a system and as a default applies the repetitive member factor Cr for the U.S. or the system factor $K_H$ for Canada.

### 14.6.4 Wall Assumptions

*Note that Sizer defaults to the following assumptions:*

- Walls are pinned at both ends.
- Walls are prevented from buckling about the weak axis by wall sheathing.

## 14.7 Beam Groups

Click ***beam*** on the toolbar to change to Beams view. Now click ***Groups*** on the toolbar. The dialog entitled Beam Design Groups opens.

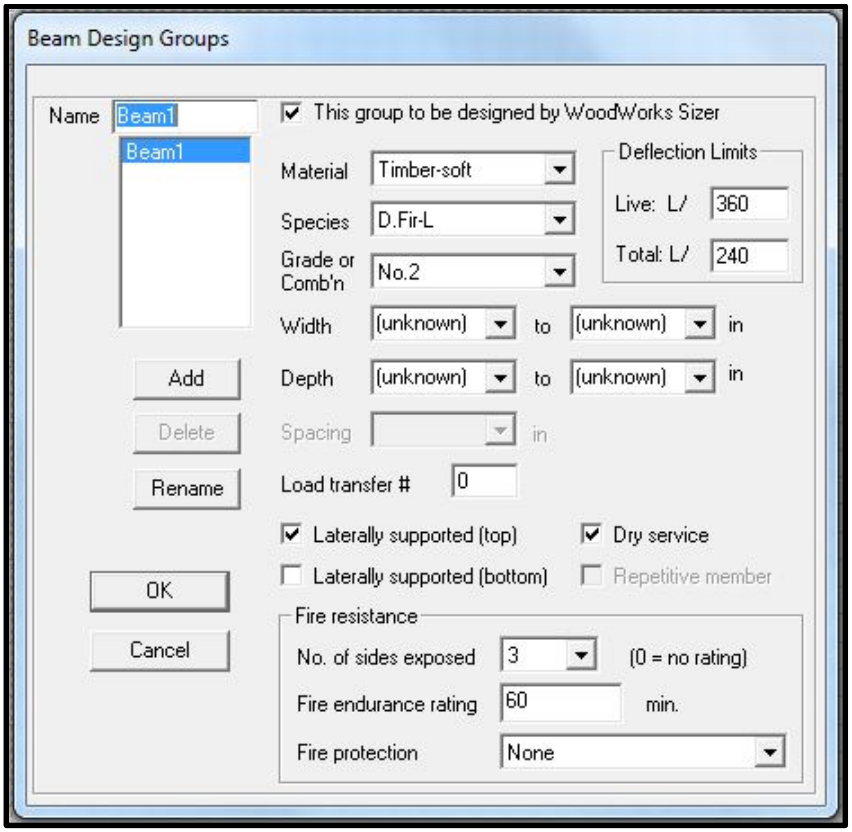



Figure 116: *Beam Design Groups*

### 14.7.1 Beam Dialog Items

When creating design groups for beams, you use a dialog similar to the one used to create design groups for columns and walls. The additional information in the beam dialog is described below.

### 14.7.2 Material

In addition to the standard choices such as timber and glulam, new material databases have been added to include SCL (Structural Composite Lumber) materials for beams only. This material is more commonly known as Parallel Strand Lumber (PSL), Laminated Strand Lumber (LSL) or Laminated Veneer Lumber (LVL).

### 14.7.3 Deflection Limits

The *Live* field specifies the maximum allowed deflection of the design group member for live loads.

The *Total* field specifies the maximum allowed deflection of the design group member for total loads.

### 14.7.4 Laterally Supported

Select this check box if the top of the beam is laterally braced continuously while the bottom is laterally restrained only at supports.

*Note: This selection is only available for glulam (Canadian only). For all other materials, notes are provided in the output specifying the necessary lateral support.*

### 14.7.5 Load Transfer Number (beam-to-beam)

This field determines how loads are transferred from beams of one group to beams of another group. Loads always transfer from a group with a higher load transfer number to a group with a lower load transfer number. For instance, in order to transfer the load from one beam directly to another beam the supported beam needs a higher Load Transfer number than the supporting beam. Use numbers between 0 (default) and 99. The actual Load Transfer number is not significant, a supported member only needs to have a higher number than the supporting member.

## 14.8 Joist Design Group

Click ***joist*** on the toolbar to change the view to Joists. Now click ***Groups*** on the toolbar. The dialog entitled Joist Design Groups opens.

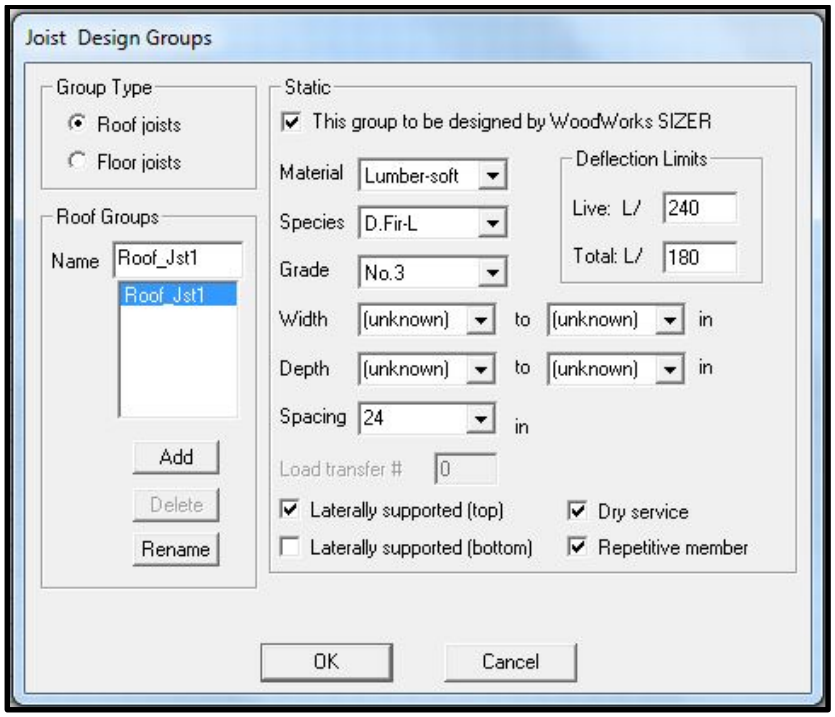



Figure 117: *Joist Design Group*

### 14.8.1 Floor Joist Vibration (Canada Only)

When using floor joists, the sheathing thickness, lateral support and connection to subfloor can be selected from a drop-down list for the determination of floor vibration.

### 14.8.2 Joist Dialog Items

When creating design groups for joists, you use a dialog similar to the one used to create design groups for columns, walls and beams. The additional information in the joist dialog is described below.

### 14.8.3 Material

Standard choices for joists include I–joists and dimension lumber. Visually graded dimension lumber is referred to as Lumber and machine stress rated dimension lumber as MSR Lumber. The default is Softwood Lumber.

### 14.8.4 Grade

Available grades depend on the selected database, but common choices for lumber are Select Structural, No.1 and No. 2. The default is No. 2.

### 14.8.5 Spacing

This drop-down list specifies the joist spacing. Select one of three standard spacing's or enter your own value.

### 14.8.6 Design Groups Summary

Once you have specified your design groups, you can create your structure in the Concept mode.

# 15 Concept Mode Gridlines and Gridpoints

*Gridlines* are horizontal (East-West) or vertical (North-South) lines positioned on the work area. Gridlines appear as dark blue dashed lines in the Grid view. The intersection of a North-South gridline with an East-West gridline is a *gridpoint*.

When you change from *Grid* view to any other view, the gridlines appear as light gray dashed lines.

Under ***View*** in the ***Settings*** menu, a snap increment option is available for gridlines. A snap increment allows you to determine the smallest increment a newly created gridpoint will move or "snap" to.


Figure 118: *Grid View Button in Toolbar*

## 15.1 Automatic Generation

This feature makes a simple task of producing a screen full of gridlines at regular intervals. First start a new **Concept** mode project. Next, from the menu bar select **Settings|Change...|View** and enter the desired gridline interval as the N-S and E-W snap increments. Press the **Ok** button and from the menu bar select **Edit|Generate Grid.**

## 15.2 View Options

Choose the **View** tab from the Settings Dialog. This tab allows you to specify different viewing options such as the size of the viewing area, the snap increment, the percentage zoom, and member names.

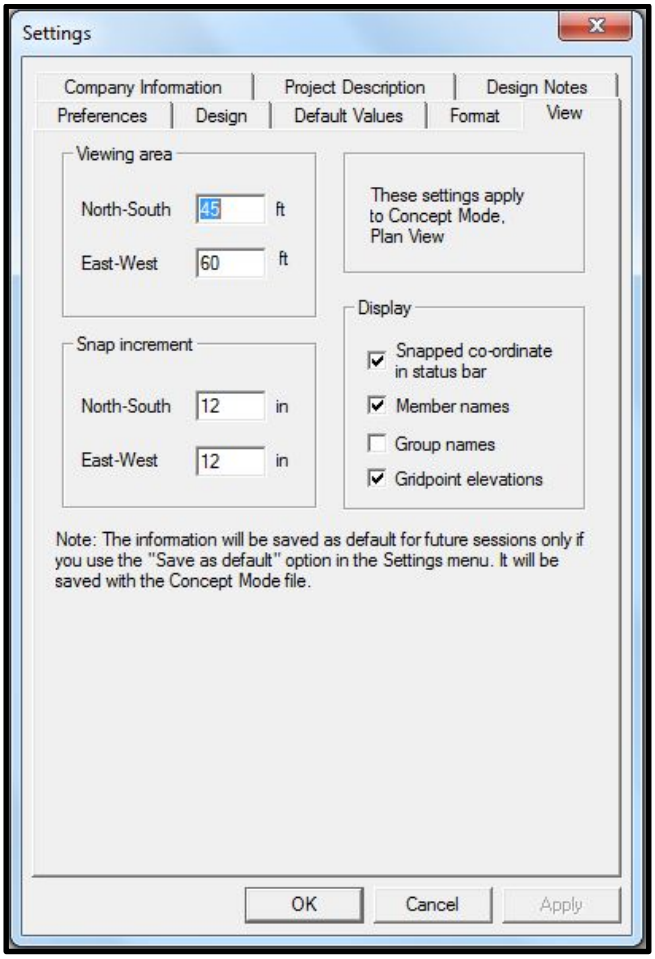



Figure 119: *Concept Mode Settings View*

## 15.3 Viewing Area

These fields specify the maximum viewing area in plan for the North-South and East-West directions.

## 15.4 Snap Increment

This specifies the smallest increment a newly created gridpoint will move or “snap” to. For example, a snap increment of 2.0 ft allows you to place gridpoints at 10.0, 12.0 and 14.0 ft, but not at 9.0 or 11.0 ft.

## 15.5 Display

These options allow you to include additional information on the diagrams for viewing or printing. This includes the snapped coordinates, the member names, and the group names.

## 15.6 Gridlines

Click ***grid*** on the toolbar. The ***grid*** button on the toolbar appears as light gray to indicate the view. The *Sizer* window title also indicates the current view. There are two quick methods for creating gridlines in concept mode, either automatically generation or manual placed.

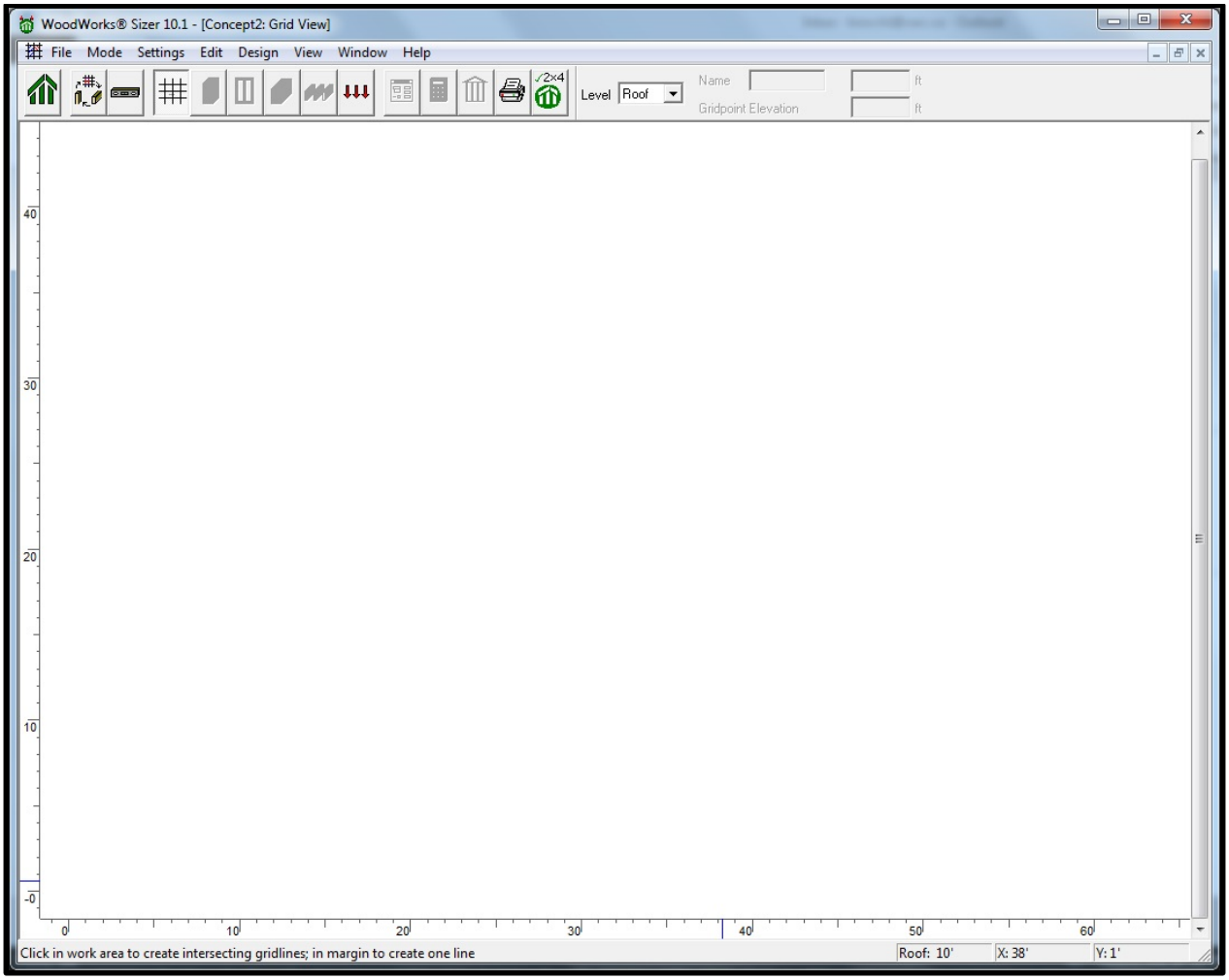



Figure 120: *Grid View before Generating*

### 15.6.1 Automatic Gridline Generation

The automatic gridline generation is dependent on the specified snap increment of the model. Click ***Settings, Change***, and navigate to the ***View*** tab. Modify the ***snap increment*** as desired. Press ***OK***.

Next Click ***Edit***, ***Generate Grid***, or ***Ctrl+shift+g***, to create gridlines based on the specified snap increment.

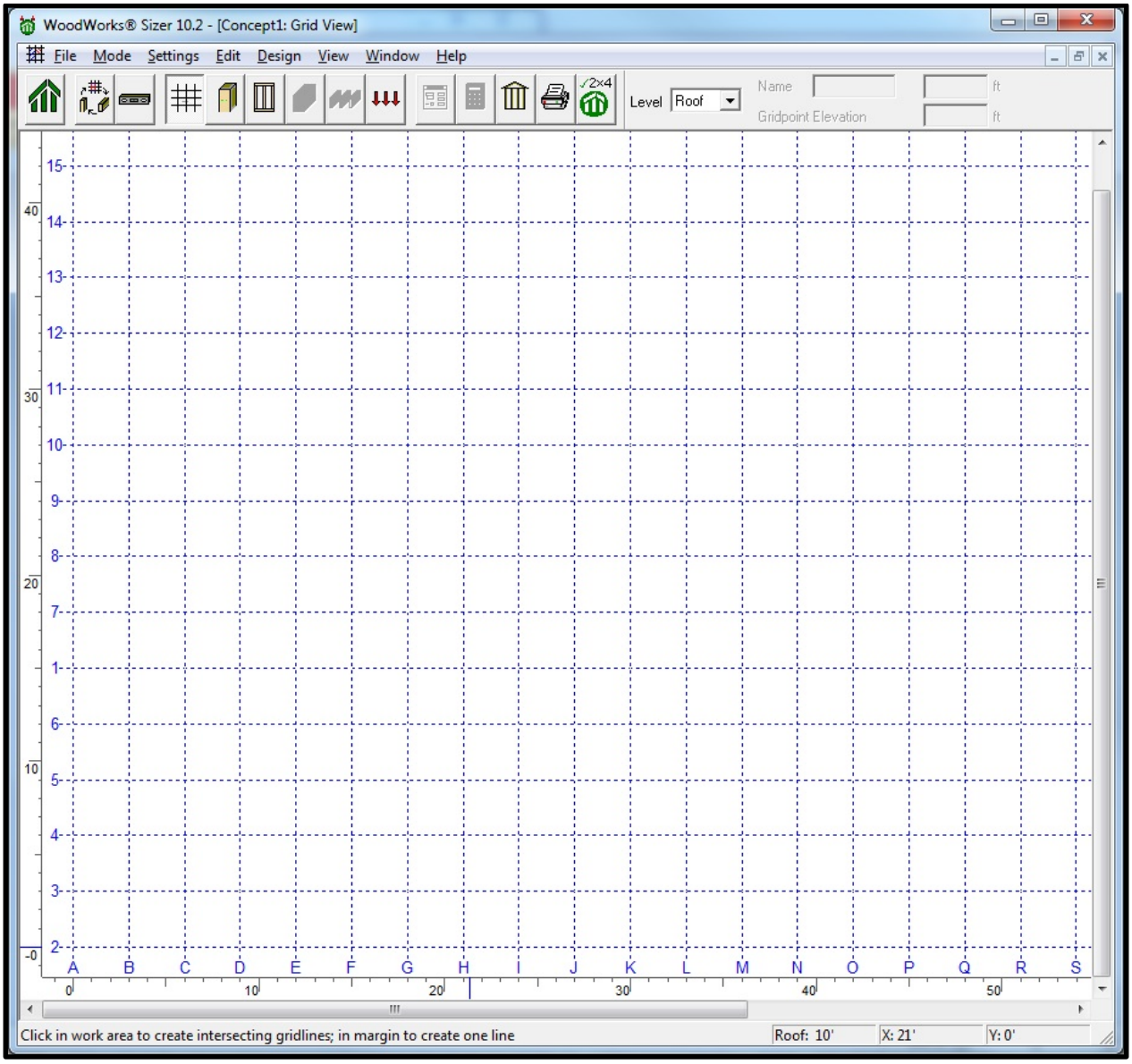



Figure 121: *Grid generated based on specified snap increment*

### 15.6.2 Manually placing Gridlines

Alternatively to utilizing the Generate Grid button, gridlines can be placed manually by clicking on the blank gridline screen. Point close to where you want to place a gridpoint and click to create it.

The first time you click in the work area, *Sizer* creates a North-South gridline designated **A** and an East-West gridline designated **1**. The intersection of these gridlines is at gridpoint **A.1.** (Gridline intersections are referred to as **character.number**.) *Note that gridpoints will snap to the nearest increment specified in the View tab. To create more gridpoints along an existing gridline, click at the required position along the existing gridline. You can place as many gridpoints as you wish in the work area.*

### 15.6.3 Moving Gridlines

To move a gridline, click on it and hold the mouse button and drag it (which changes to red) to the position you want. Use ctrl-click to select multiple gridlines. Dragging any one of the selected gridlines moves all of them.

Alternatively, once a gridline has been placed, its location can be modified using the concept mode data bar (See section 19.6.5 through 19.6.8)

To deselect a gridline, ctrl-click it. To deselect all selected gridlines, click any blank spot in the work area.

### 15.6.4 Removing Unwanted Gridlines

To delete individual gridlines, select them by clicking on them so that they appear in red and then press the DELETE key on the keyboard.

You cannot delete a gridline that has a wall or column located on it.

Should you accidentally generate a screen full of gridlines you can easily delete them. First make sure that the **Grid** button on the toolbar is the active button. From the menu bar select **Edit|Select All**. Then press the Del key. *Sizer* will delete all unused gridlines and will issue a warning for each gridline that is in use and cannot be removed.

Selecting the Generate Gridlines item on the ***Edit*** menu automatically generates gridlines over the whole viewing area. The spacing between gridlines equals the current snap increment, which is set under the **View** tab of the **Settings** options. If you make a mistake and generate gridlines at the wrong increment you need only use the ***Select All*** item of the ***Edit*** menu and then press the DELETE key. This will remove all gridlines that are currently not "in use" by members.

### 15.6.5 Concept Mode Data Bar

The Concept Mode Data toolbar is active in the grid view when a gridline or gridpoint is selected. The Concept Mode Data toolbar will display information on the selected item, including gridline name, gridline position and gridpoint elevation.

### 15.6.6 Gridline Name

If a gridline is selected, the Concept Mode Data toolbar will indicate the name of the gridline.

The gridline name can be changed by typing over the existing name included in the toolbar.

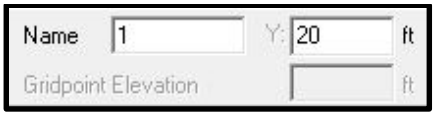



Figure 122: *Gridline Input*

### 15.6.7 Gridline Position

If a gridline is selected, the Concept Mode Data toolbar will indicate the position of the gridline as the distance from the parallel axis (either X or Y). For example, if a gridline runs East-West, the position indicated will be the Y-coordinate.

The gridline position can be changed by typing over the existing name included in the toolbar.

### 15.6.8 Gridpoint Elevation

If a gridpoint is selected, the Concept Mode Data toolbar will indicate the gridpoint elevation.


Figure 123: *Gridpoint Elevation Input*

The gridpoint elevation can be changed by typing over the existing elevation in the toolbar. If a gridpoint elevation is changed, a plus or minus elevation will appear beside the gridpoint on the **Grid** view to indicate its elevation in reference to the elevation of the current level.

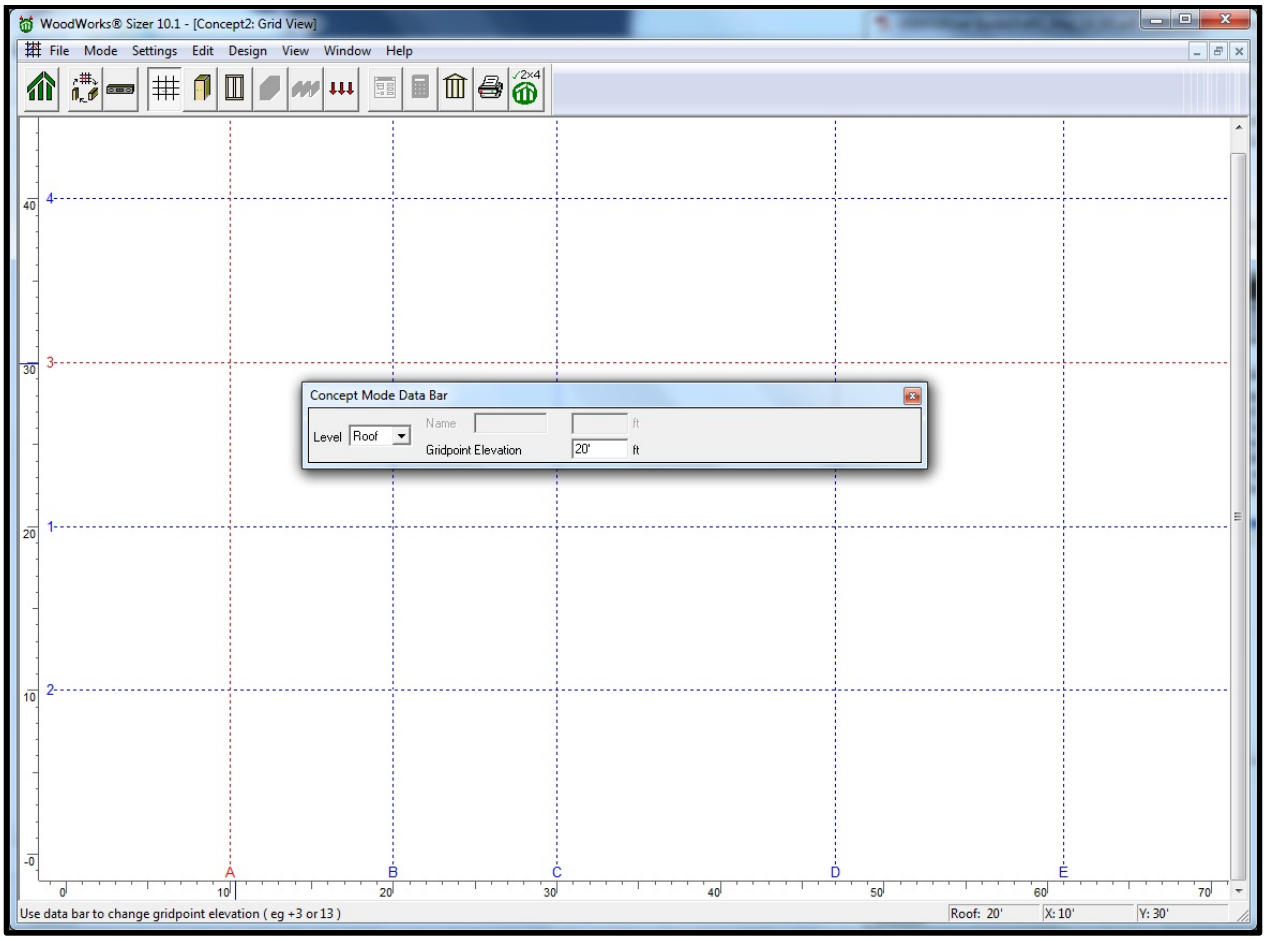



Figure 124: *Modifying Gridlines*

To create a structure with sloped members (such as a gable roof), you need to specify non-standard elevations for some members. In *Sizer* you specify gridpoint elevations from ground level.

Changing the elevation at a gridpoint changes a column's or wall's height at that point. Naturally, this affects the results of the design process.

The elevation of a gridpoint specifies the elevation of all beams and joists supported by the column or wall located at the gridpoint. You cannot have two beams at different elevations supported at the same gridpoint.

# 16 Concept Mode Columns

## 16.1 Creating Columns

Click ***column*** on the toolbar. The ***column*** button on the toolbar appears as light gray to indicate the view. The *Sizer* window title also indicates the current view.

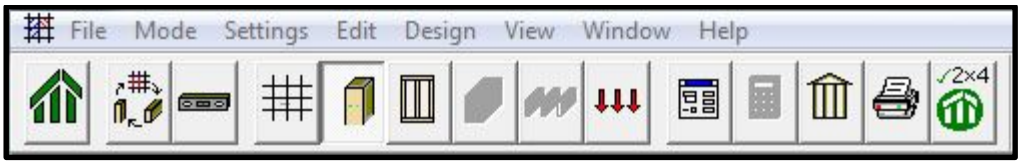



Figure 125: *Column View*

All to-be-designed columns appear as hollow blue squares, while not-to-be-designed columns appear as solid blue squares.

In *Walls*, *Beams*, *Joists*, and *Loads* view, columns appear as light grey squares.

## 16.2 Columns

New columns must be added to existing column groups. If you have created column design groups, simply choose the appropriate group from the drop-down list included in the *Concept Mode Data* toolbar. If you have not created the column design groups, click the ***groups*** button on the toolbar to access the dialog for creating column and wall groups. See section 4.3 for more details.

Columns must be created on a gridpoint. Click a gridpoint to create a column there. Click on any position other than a gridpoint is ignored.

## 16.3 Deleting Columns

Click a column to select it. Selected columns appear red.

To delete a column, select it and press the DELETE key on the keyboard. To delete multiple columns at once, hold the control key, select the desired columns to be deleted (Columns will be highlighted in Red), then press the DELETE key.

## 16.4 Concept Mode Data Bar

The *Concept Mode Data* toolbar is active in the *Column* view when a column is selected. The *Concept Mode Data* toolbar will display information on the selected item, including the column name and the column group name.


Figure 126: *Specifying Column Group ToolBar*

### 16.5 Column Name

If a column is selected, the *Concept Mode Data* toolbar will display the column name.

As a default, the column name is **c1** for the first column, **c2** for the second, and so on. This can be changed however by typing over the existing name included in the toolbar Column Group.

If a column is selected, the Concept Mode Data toolbar indicates which column design group it belongs to.

The group of a column can be changed by selecting a different group from the drop-down list included in the toolbar.

## 17 Concept Mode Walls

### 17.1 Creating Walls

Click ***wall*** on the toolbar. The ***wall*** button on the toolbar appears as light gray to indicate the view. The *Sizer* window title also indicates the current view.

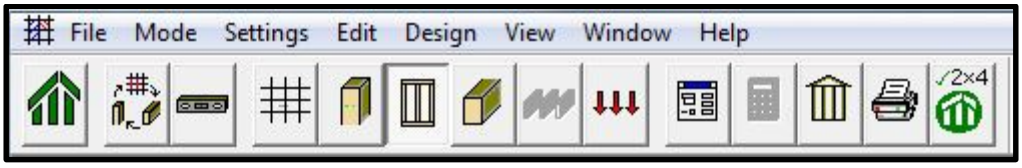



Figure 127: *Wall View*

All to-be-designed walls appear as hollow blue lines, while not-to-be designed walls appear as solid blue lines.

In *Columns*, *Beams*, *Joists*, and *Loads* view, walls appear solid gray lines. In *Grid* view, the walls are shown as light grey lines.

### 17.2 Walls

New walls must be added to existing wall groups. If you have created the wall design groups, simply choose the appropriate group from the drop-down list at the right of the status bar. If you have not created the wall design groups, click ***groups*** on the toolbar to access the dialog for creating column and wall groups.

To create a wall, drag from the start gridpoint to the end gridpoint. As you drag, *Sizer* displays a solid "rubber band" line showing where the wall will go. Release the mouse button at the gridpoint where you wish to end the wall. *Sizer* draws the wall between the two gridpoints.

*You can create walls between any two gridpoints. One wall can cross (or intersect, if you prefer) another wall, but two walls cannot overlap. For example, you cannot create two walls between the same pair of gridpoints. Sizer indicates an error if you try to overlap walls.*

Even if a wall spans several gridpoints, it is treated as continuous. However, if you create several walls in a row, spanning a series of gridpoints along one gridline, the walls are treated as discontinuous. This does not affect the design results unless the discontinuous wall segments belong to different wall design groups.

Previous versions of *Sizer* required that a wall or beam on a lower-story support an upper story wall. This limitation has been removed and joist areas can now support upper story walls in two ways:

### 17.2.1 End of a Cantilevered Joist Area

Walls can sit on the end of cantilevered joist areas. Please refer to *Sizer*'s online help and the examples in this document for further details.

### 17.2.2 Across the Interior of a Joist Area

A joist area where the joist direction is not parallel to a wall can now support that wall. The wall produces a uniformly distributed load which results in a point load on each joist in the joist area. Please refer to *Sizer*'s online help and the examples in this document for further details.

## 17.3 Deleting Walls

Click a wall to select it. Selected walls appear red.

To delete a wall, select it and press the DELETE key on the keyboard. To delete multiple walls at once, hold the control key, select the desired columns to be deleted (Walls will be highlighted in Red), then press the DELETE key.

## 17.4 Concept Mode Data Bar

The Concept Mode Data toolbar is active in the Wall view when a wall is selected. The *Concept Mode Data* toolbar will display information on the selected item, including the wall name and the wall group name.


Figure 128: *Specifying Wall Group ToolBar*

## 17.5 Wall Name

If a wall is selected, the *Concept Mode Data* toolbar will display the wall name.

As a default, the wall name is **w1** for the first wall, **w2** for the second, and so on. This can be changed however by typing over the existing name included in the toolbar.

## 17.6 Wall Group

If a wall is selected, the *Concept Mode Data* toolbar indicates which wall design group it belongs to.

The group of a wall can be changed by selecting a different group from the drop-down list included in the toolbar.

## 17.7 Examples of Walls Supported by Joist Areas

### 17.7.1 Basic Rules

1. A Wall cannot be supported by a 3-sided joist area.
2. A wall can be supported by a 4-sided joist area on the floor below if the wall extends exactly from one edge joist to the other. This ensures that it is fully supported and that it loads all of the joists.

3. If the wall spans the interior of the joist area, it must be parallel to all but possibly one of the underlying supports of the joist area. Thus if there are four supports under a multi-span joist area, the wall must be parallel to at least three of them. This ensures that the outermost (or edge) joists in the joist area are the critical ones.
4. Walls can always be supported at the end of cantilevered joists.

*Refer to the online help for more information about Wall Supports.*

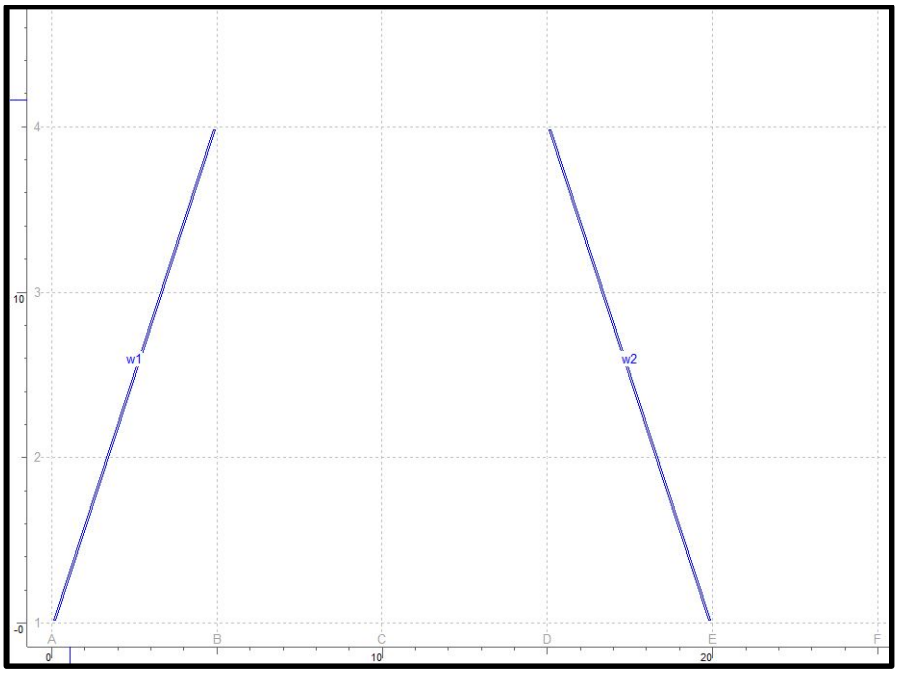



Figure 129: *Concept Mode First Floor Wall View*

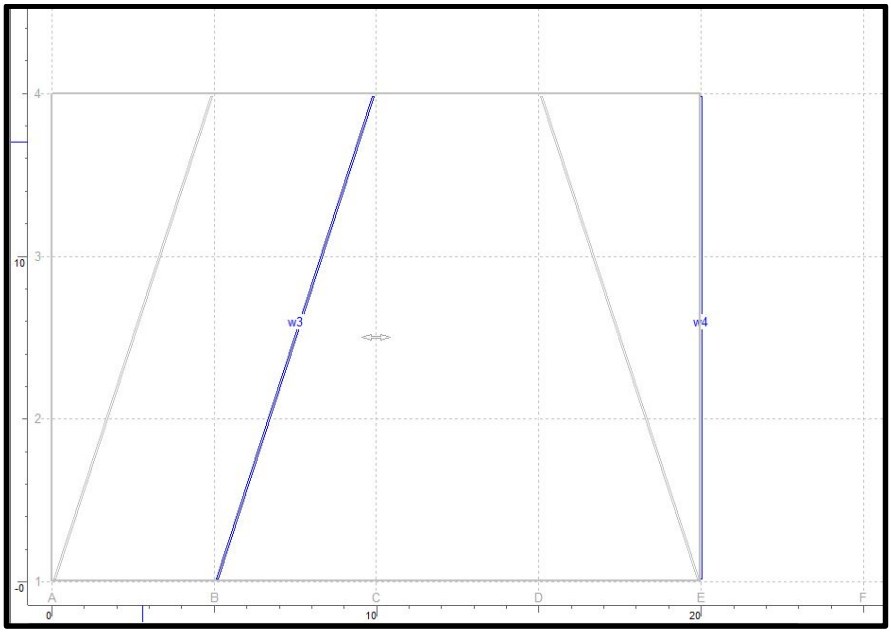



Figure 130: *Concept Mode Second Floor Wall View*

In this example there are two walls (w1 and w2) supporting a joist area on the first level. It is not possible to place a wall along gridline C because it would not be parallel to either w1 or w2 below. A wall on the second level must be parallel to at least 1 (i.e. 2 – 1 = 1) of the joist supports on the first level. This is the case for wall w3.

In the same example, a wall can be placed at the end of the cantilevered joist area, and such is the case for wall w4.

It is worth noting that, as in previous versions of *Sizer*, a second level wall could be placed directly over any of the first level walls. The load would then be transferred directly to the lower wall.

### 17.8 Partial Wall on joist area

*Sizer* allows partial walls to be placed on joist areas if there is a supporting wall below it or if it is at the end of a cantilevered joist area. However, these partial walls will not be carried by the joists, but extend to a wall or the ground below.

In this example, we are on the second floor. Wall w4 and wall w6 are allowed, but the joist area does not support them. They extend down to the ground. You can check to see that w4 and w6 extend from the top storey to the ground by clicking the elevation view for gridlines 2 or 8. Wall w5 is supported by the wall below. An attempt to create a wall from B6 to C6 fails because it does not span the whole extent of the joist area. An attempt to create a wall from B6 to F6 fails, because part of the wall is not supported by the joist area.

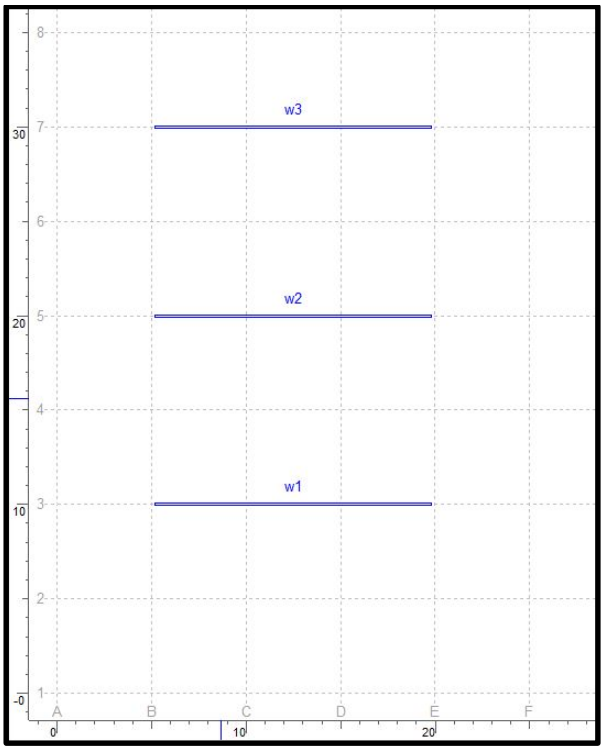



Figure 131: *Partial Wall on Joist Area First Floor Wall View*

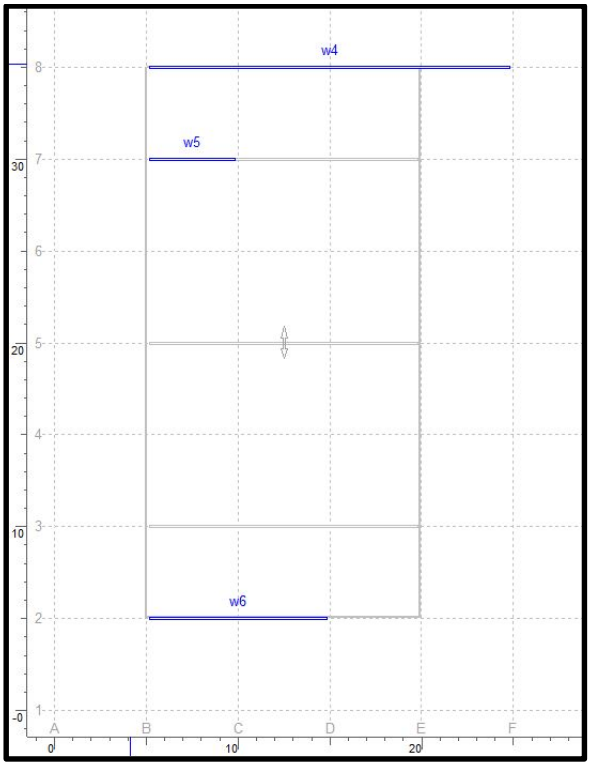



Figure 132: *Partial Wall on Joist Area Second Floor Wall View*

# 18 Concept Mode Beams

## 18.1 Creating Beams

Click ***beam*** on the toolbar. The ***beam*** button on the toolbar appears as light grey to indicate the Beam view. The *Sizer* window title bar also indicates the current view.

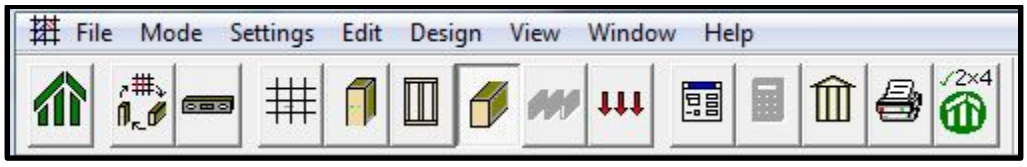



Figure 133: *Beam View*

All to-be-designed beams appear as hollow blue lines, while not-to-be designed beams appear as solid blue lines.

In *Columns*, *Walls*, *Joists*, and *Loads* view, beams appear as solid grey lines. In *Grid* view, beams appear as light grey solid lines.

## 18.2 Beams

New beams must be added to existing beam groups. If you have created the beam design groups, simply choose the appropriate group from the drop-down list included in the *Concept Mode Data* toolbar.

To create a beam, drag from the start gridpoint to the end gridpoint. As you drag, *Sizer* displays a solid green "rubber band" line showing where the beam will go. Release the mouse button at the gridpoint where you wish to end the beam. *Sizer* draws the beam between the two gridpoints.

*You can create beams between any two gridpoints. One beam can cross another beam, but two beams cannot overlap. For example, you cannot create two beams between the same pair of gridpoints. Sizer indicates an error if you try to overlap beams.*

Even if a beam spans several gridpoints, it is treated as continuous. However, if you create several beams in a row, spanning a series of gridpoints along one gridline, the beams are treated as discontinuous.

## 18.3 Deleting Beams

To delete a beam, select it and press the DELETE key on the keyboard. To delete multiple beams at once, hold the control key, select the desired columns to be deleted (Beams will be highlighted in Red), then press the DELETE key.

## 18.4 Concept Mode Data Bar

The Concept Mode Data toolbar is active in the Beam view when a beam is selected. The *Concept Mode Data* toolbar will display information on the selected item, including the beam name and the beam group name.


Figure 134: *Specifying Beam Group Toolbar*

## 18.5 Beam Name

If a beam is selected, the *Concept Mode Data* toolbar will display the beam name.

As a default, the beam name is **b1** for the first beam, **b2** for the second, and so on. This can be changed however by typing over the existing name included in the toolbar.

## 18.6 Beam Group

If a beam is selected, the *Concept Mode Data* toolbar indicates which beam design group it belongs to. The group of a beam can be changed by selecting a different group from the drop-down list included in the toolbar.

## 18.7 Examples of Beam Supports in Concept Mode

### 18.7.1 Beam supported by 2 other beams

In this example, beam group Beam1 has a load transfer number of 0. In order to create b3, its beam group, Beam2, must have a higher load transfer number than the supporting beams. In this case, a load transfer number of 1 or higher for Beam2 would work. The load transfer number is entered in the beam groups dialog (press the **Groups** button after pressing the **Beam** button on the toolbar while in Concept mode).

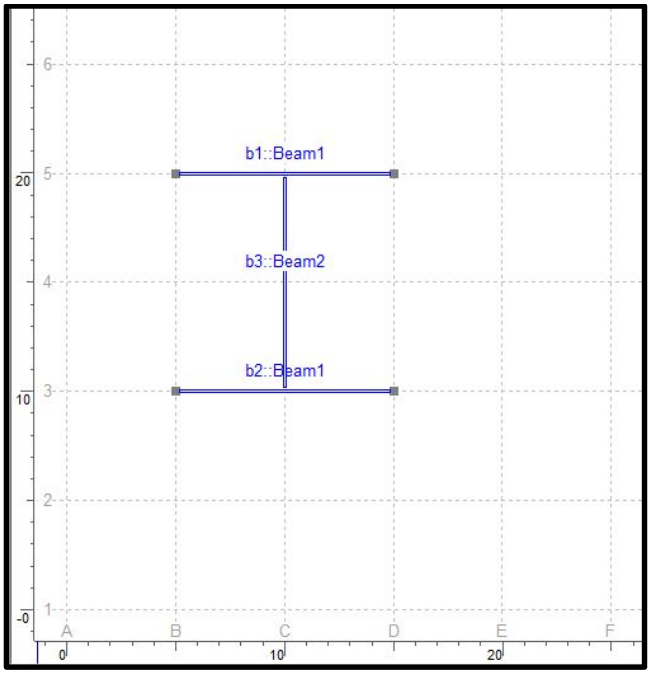



Figure 135: *Beams Supported by Two Other Beams*

### 18.7.2 Sloped beam supported by 2 other beams

Gridpoint elevations of support points must be entered explicitly when they differ from the elevation of the floor level. From the previous example, the elevation of beam b2 has been increased by 2 feet. In order to create beam b3 it is also necessary to raise the elevation of the support point C6 to the same elevation as the rest of the supporting beam b2.

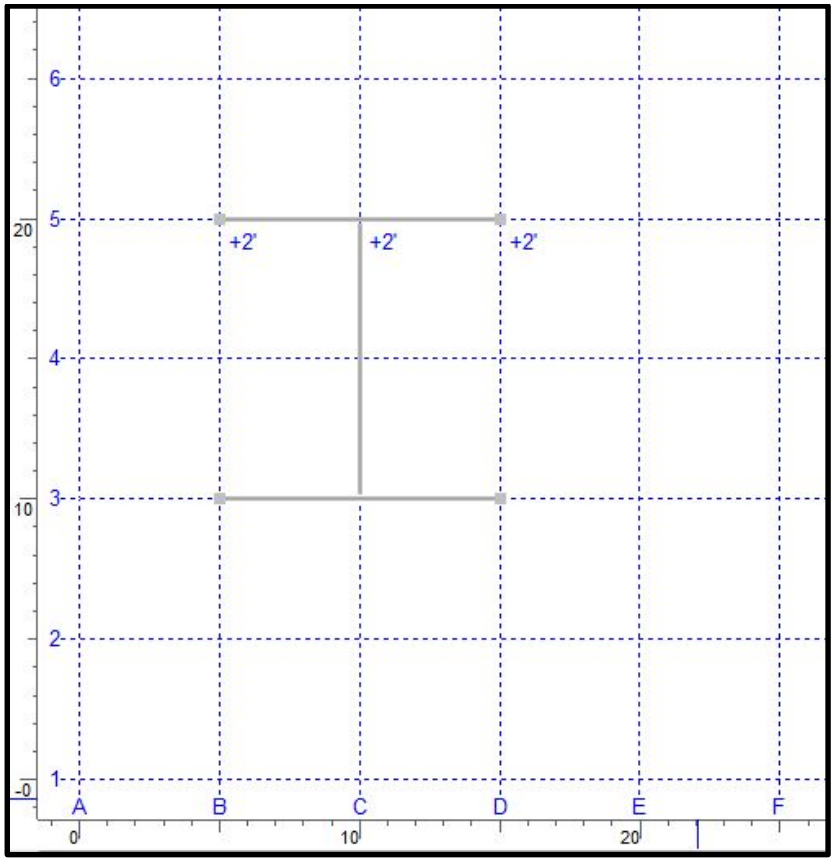



Figure 136: *Beam supported by 2 other beams*

### 18.8 Sloped beam supported by more than 2 other beams

The slope of a beam with more than 2 possible supports is now defined by the elevation of the two extreme supports. In this example, the slope of beam b4 is defined by the elevation of beam b1 at elevation 10 feet and the column at gridpoint C7 with an elevation of 15 feet. This can be done by going to Grid View, Selecting on a specific grid point, and modifying the elevation in the dialog box.

The column at gridpoint C3 and the cantilevered end of beam b2 also support b4 because their elevations lie along the slope of b4, and, as noted in the previous example, the elevations of the support points have been entered explicitly.


Figure 137: *Sloped Beam Supported by more than 2 other Beams Input*

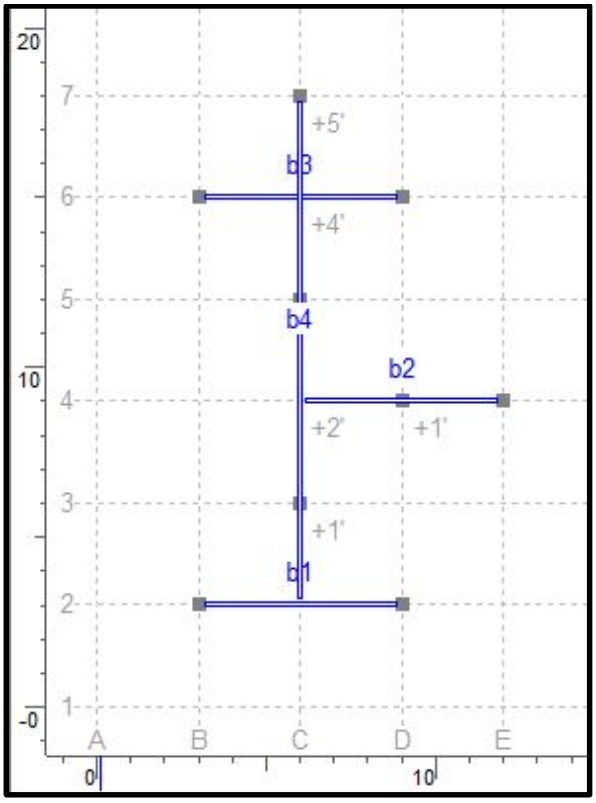



Figure 138: *Sloped Beam Supported by more than 2 other Beams Top View*

Beam b3 does not support b4 even though the support gridpoint has been raised to the correct elevation. This is because the beam b3 is at a lower elevation. Its elevation is defined by the columns at B6 and D6.

The column at C5 also does not support beam b4 because its gridpoint elevation has not been raised to meet the beam. The plan and elevation views shown here illustrates these points.

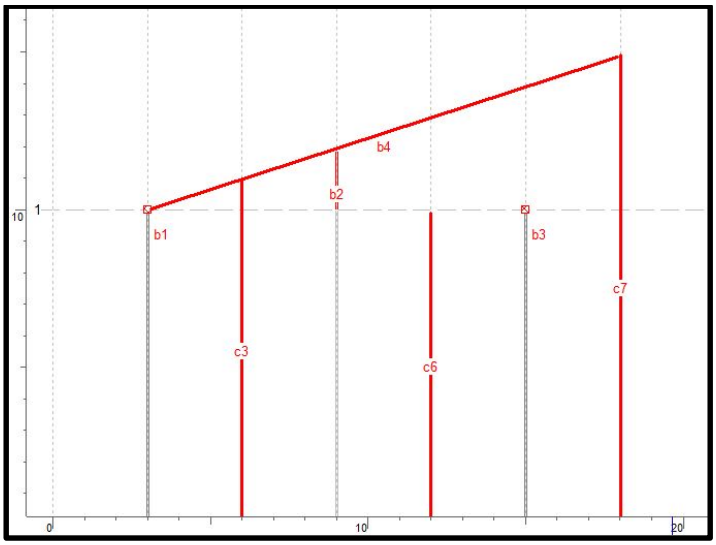



Figure 139: *Sloped Beam Supported by more than 2 other Beams Side View*

# 19 Concept Mode Joists

## 19.1 Creating Joists

Click ***joist*** on the toolbar. The ***joist*** button on the toolbar appears as light grey to indicate the Joists view. The *Sizer* window title also indicates the current view.

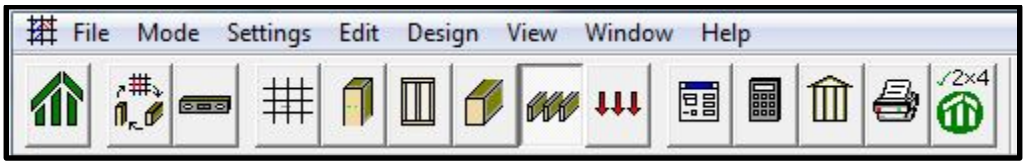



Figure 140: *Joist View*

All to-be-designed joist areas appear as solid blue outlines with a blue direction indicator. All not-to-be-designed joist areas appear as solid blue outlines with a black direction indicator.

## 19.2 Joists

New joists must be added to existing joist groups. If you have created the joist design groups, simply choose the appropriate group from the drop-down list in the *Concept Mode Data* toolbar. To create a rectangular joist area, click on the four gridpoints at the corners of the area in question. As you click, a rubber-band line stretches from one gridpoint to the next. *Sizer* encloses the joist area defined by these four gridpoints with a solid blue outline. Within this outline is a double-headed arrow showing the direction of the joists. The arrow is blue for joists to-be-designed and black for joists not-to-be-designed. To create a triangular joist area, click on the same gridpoint for the first and fourth points.

*Joist areas can be either simply supported, multi-span or cantilevered. Joist areas must have at least two supporting beams or walls. If there are four supporting beams or walls, Sizer assumes the joists span the shorter direction. Where the number of supporting beams or walls exceeds four, you should define several joist areas, each with no more than four supporting beams or walls.*

## 19.3 Deleting Joists

To delete a joist area, select it and press the DELETE key on the keyboard. If the joist area supports a load, *Sizer* does not allow you to delete it. To delete multiple joists at once, hold the control key, select the desired columns to be deleted (Joists will be highlighted in Red), then press the DELETE key.

## 19.4 Concept Mode Data Bar

The Concept Mode Data toolbar is active in the Joist view when a joist area is selected. The *Concept Mode Data* toolbar will display information on the selected item, including the joist area name, the joist areas group name, and the joist areas direction.

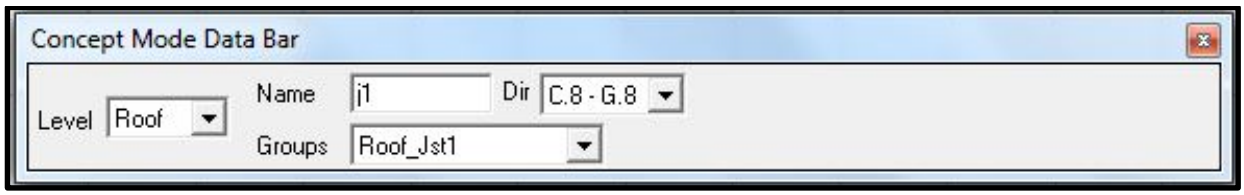



Figure 141: *Specifying Joist Group ToolBar*

### 19.5 Joist Area Name

If a joist area is selected, the *Concept Mode Data* toolbar will display the joist area name.

As a default, the joist area name is **j1** for the first joist area, **j2** for the second, and so on. This can be changed however by typing over the existing name included in the toolbar.

### 19.6 Joist Area Group

If a joist area is selected, the *Concept Mode Data* toolbar indicates which beam design group it belongs to.

The group of a joist area can be changed by selecting a different group from the drop-down list included in the toolbar.

### 19.7 Joist Area Direction

If a beam is selected, the *Concept Mode Data* toolbar indicates the direction which the joist area spans.

The direction of a joist area can be changed by selecting an alternate direction from the drop-down list included in the toolbar. *Note that an alternate direction is only available if there are beams or walls in the alternate direction to support the joists.*

#### 19.7.1 Examples for Joist Areas Sloped Joists and Gridpoint Elevations

The slope of a joist area is determined by the gridpoint elevations of the members that support it rather than by the elevations at the corners of the joist area itself. This is significant when you want to create a cantilevered joist area that sits on 2 beams at different elevations.

### 19.7.2 Maximum of 6 potential supports per member type

Error Message: “Unable to create joist area. There are more than six beams forming potential supports for the joist area in both directions. See on-line help for full explanation.”

*Sizer* is limited to considering a maximum of six potential support members of one type. In this example, an attempt to create the joist area C3-C11-G11-G3 fails because there are 5 beam supports in the E-W direction and 2 beam supports in the N-S direction, creating a total of 7 beams forming potential supports for the joist area in both directions. This example would not have failed had one of the beams been removed or replaced by a wall.

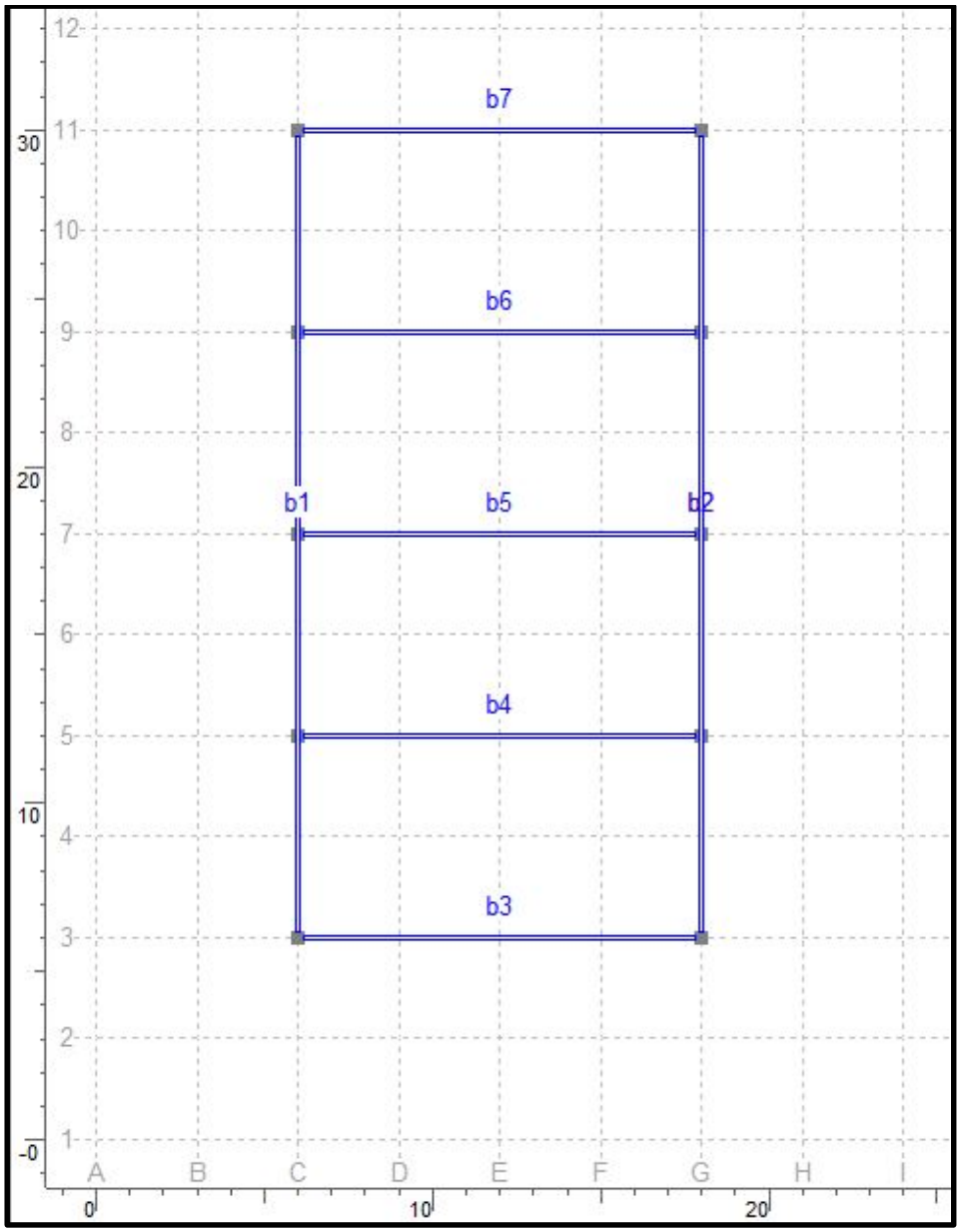



Figure 142: *Maximum of 6 potential supports per member type*

### 19.7.3 Maximum of 6 spans or 6 supports

*Error Message: "*Unable to create joist area. Too many supporting beams and walls in one direction: no more than 6 supports or 6 spans on joists are allowed. See on-line help for dull explanation."

*Sizer* limits multi-span joists to a maximum of 6 supports or 6 spans. In this example, an attempt to create the joist area C3-C9-G9-G3 fails because there are 4 beam supports and 3 wall supports in the E-W direction, exceeding the maximum of 6 supports of both types in that direction, but no supports in the N-S direction.

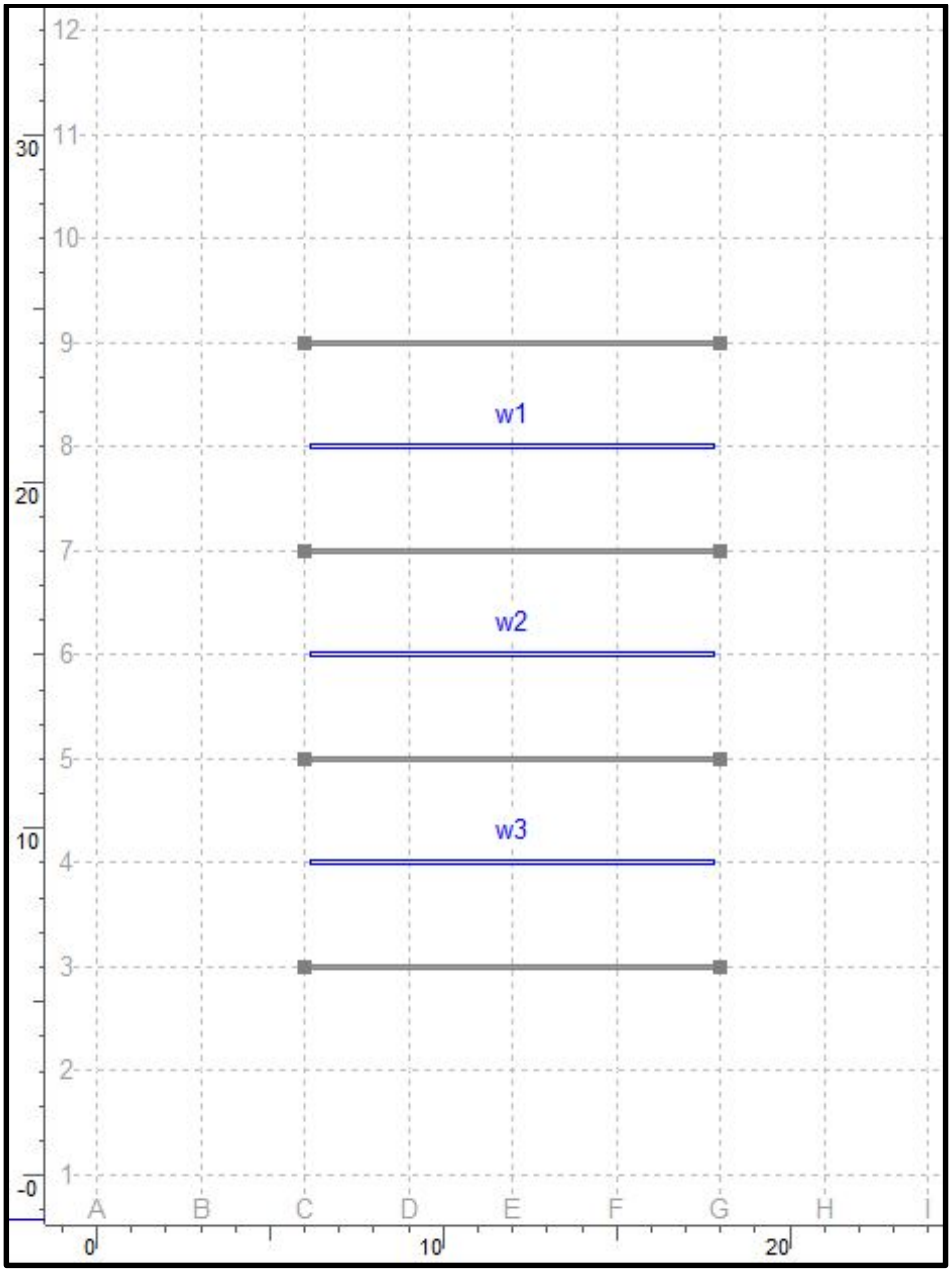



Figure 143: *Maximum of 6 spans or 6 supports*

### 19.7.4 Only one valid span direction

Two walls have been added to the previous example, allowing us to create a joist area running in the E-W direction. E-W is the only feasible joist direction as there are still too many supports in the N-S direction. *Note that we have not exceeded the limit of 6 beams or 6 walls in both directions.*

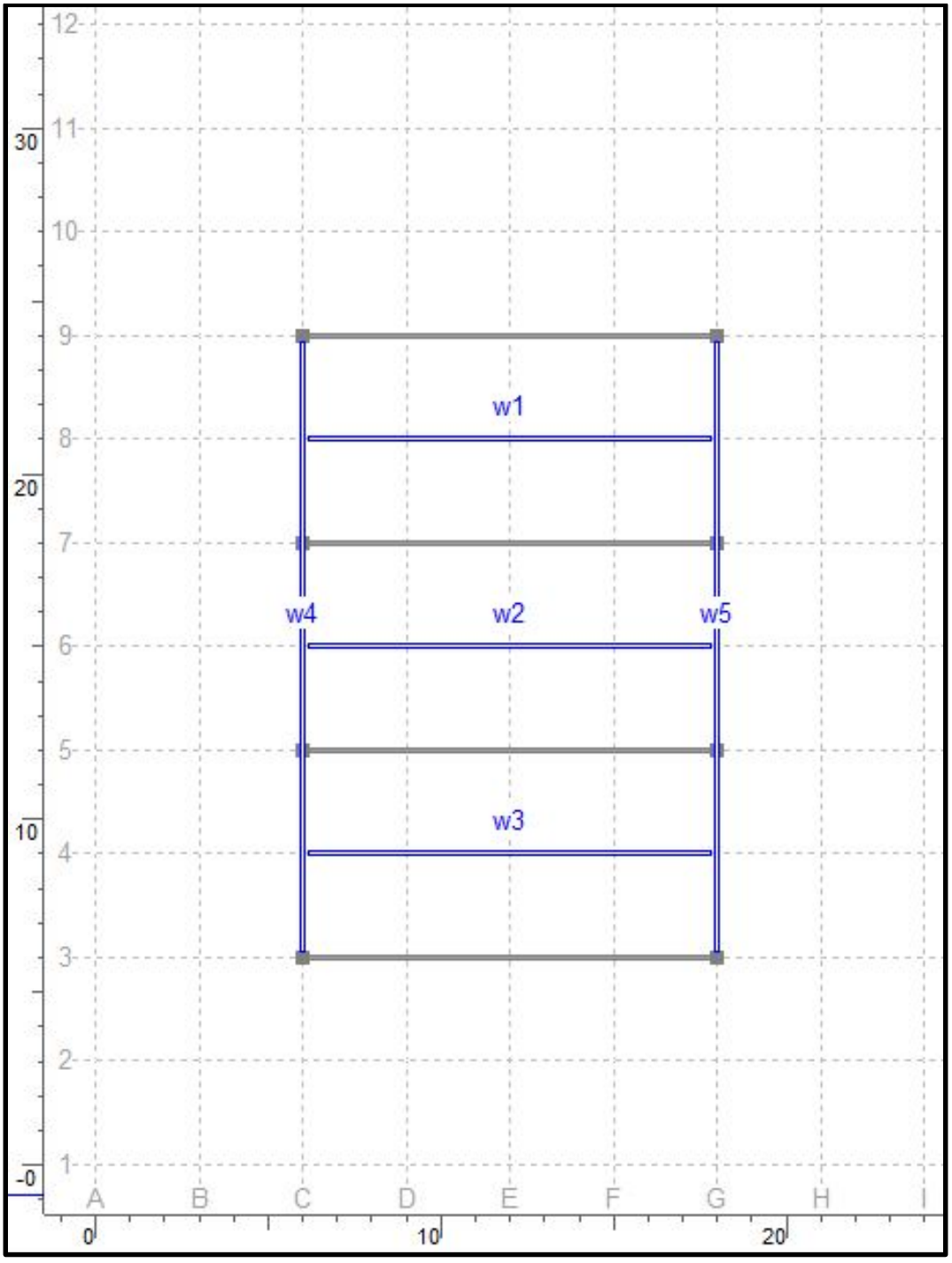



Figure 144: *Only One Valid Span Direction*

### 19.7.5 Partial Supports

*Error Message:* "Unable to create joist area. There is a beam or a wall which supports part of the joist area. See on-line help for full explanation."

*Sizer* does not permit partial supports. In this example, an attempt to create the joist area B2-B4-E4-E2 fails because beam b2 only supports some of the joists in the N-S direction, and there are no supports for joist in the E-W direction.

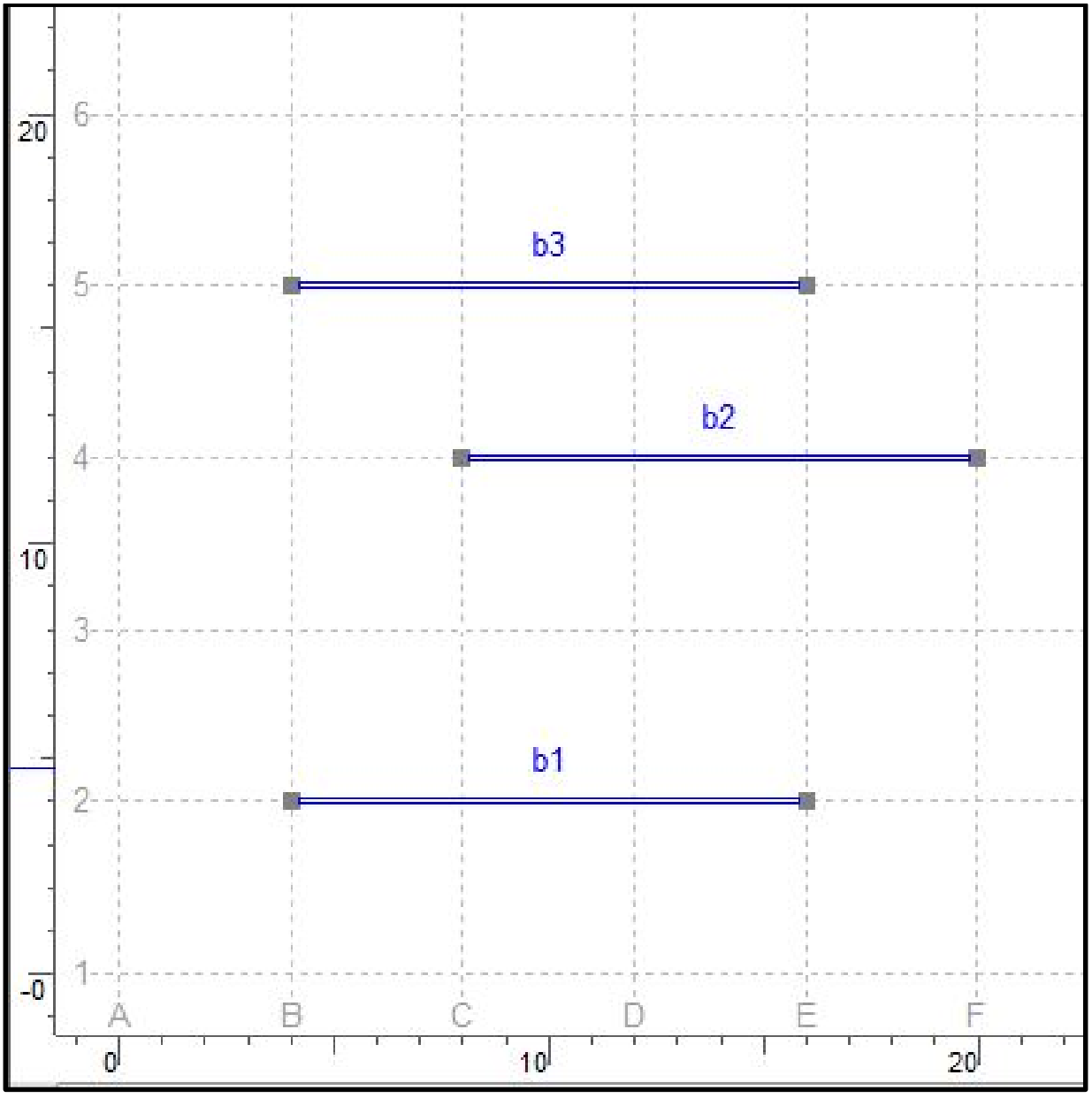



Figure 145: *Partial Supports*

### 19.7.6 Partial Supports are ignored if another direction can be spanned

Two beams have been added to the previous example, running in the N-S direction. Therefore it is possible to create a joist area running in the E-W direction, even though there is a partial support disqualifying the N-S direction.

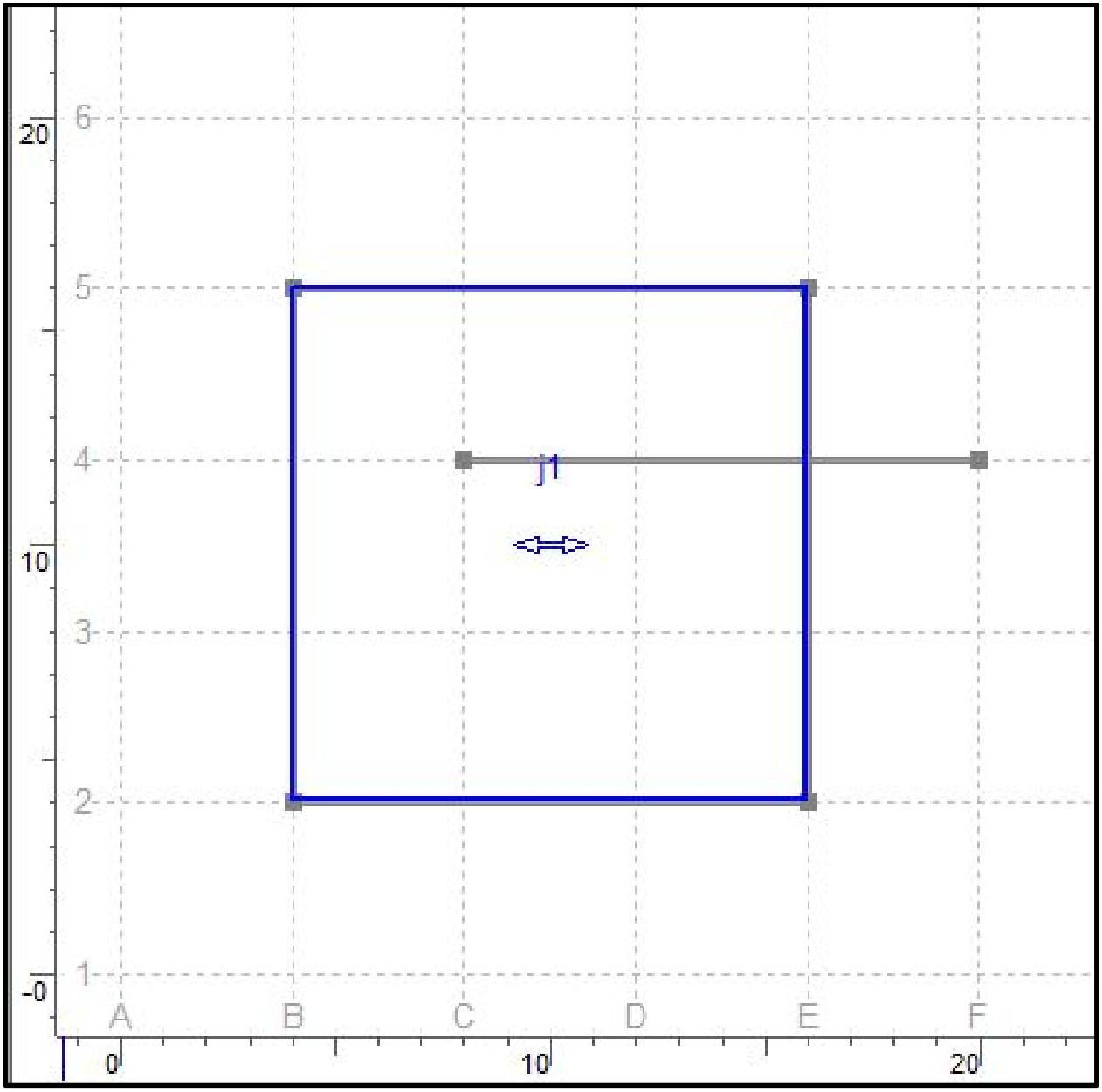



Figure 146: *Partial Supports are ignored if Another Direction can be spanned*

### 19.7.7 Angled Partial Supports

*Error Message:* "Unable to create joist area. There is a beam or a wall which supports only part of the joist area. See on-line help for full explanation."

Joist areas cannot be placed over angled partial supports. In this example, the partial support is at an angle to the joists in both directions so that it supports some of the joists in both directions. This makes it impossible to create a joist area.

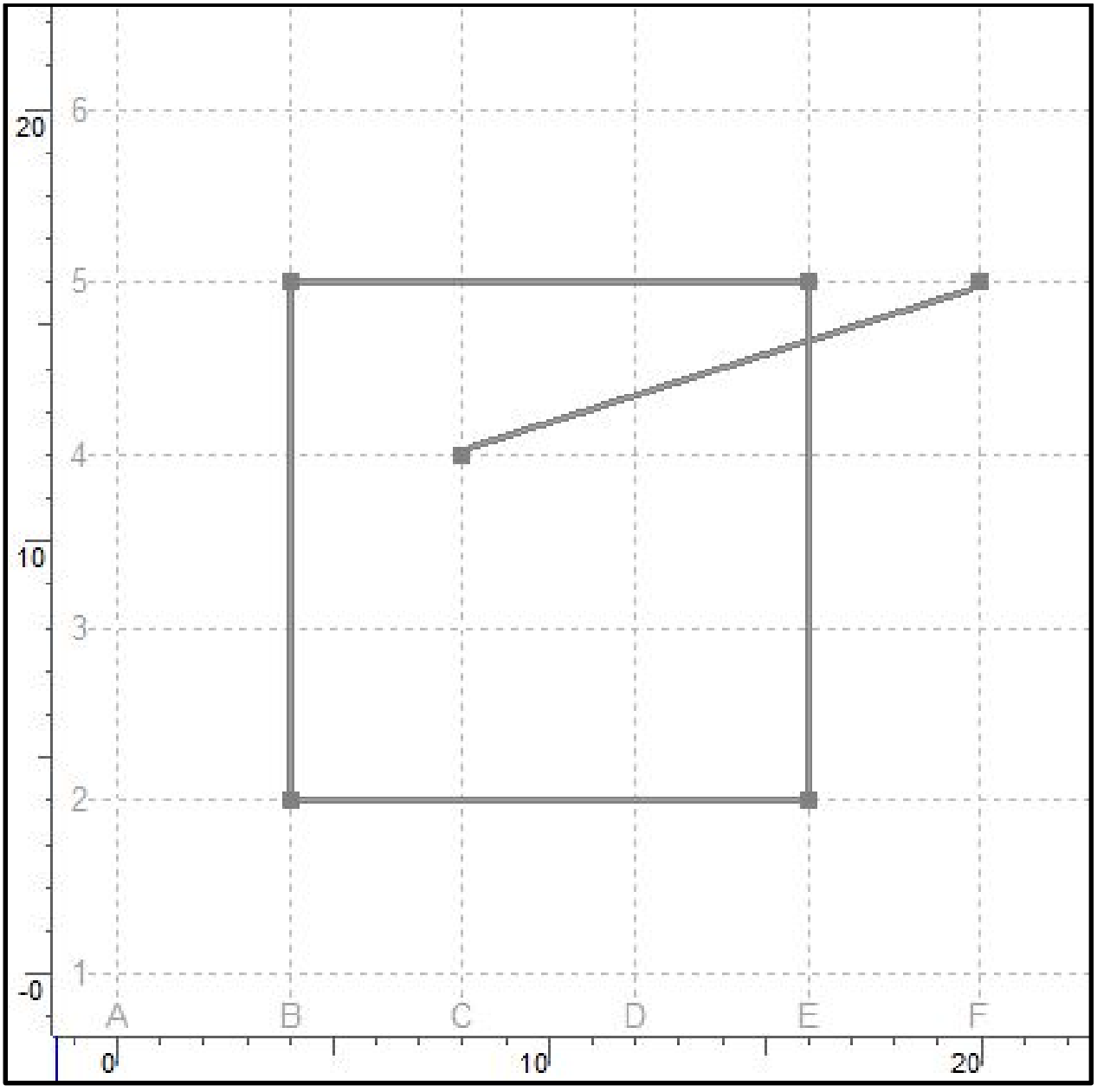



Figure 147: *Angled Partial Supports*

### 19.7.8 Partial Supports are ignored when they lie at a different elevation than the joists

In this example, the end of the partial support has been elevated 2 feet above the other supports so that it no longer is able to support any of the joists. It is now possible to create a joist area with joists running in either direction. *Sizer* assumes that the column supporting the elevated beam pierces the joist area.

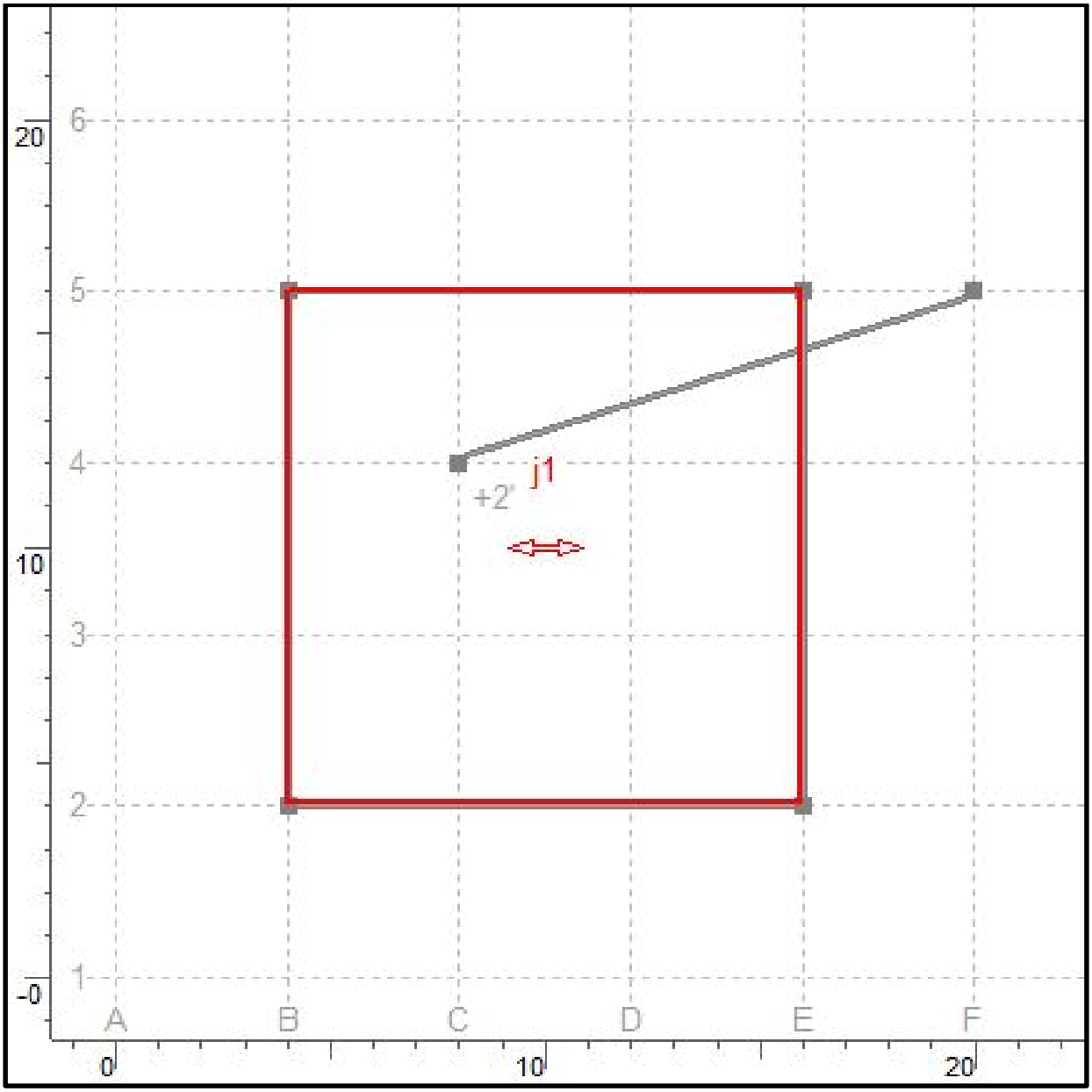



Figure 148: *Partial Supports are ignored when they lie at a different elevation than the joists*

### 19.7.9 Out-of-plane supports

*Error Message: "*Unable to create joist area. There are more than two potential supports and they do not lie in the same plane. This is true either for both joist directions, or for the only direction for which are enough supports. See on-line help for full explanation."

When there are more than two supports, *Sizer* does not permit out-of-plane supports. In this example, an attempt to create the joist area B3-B7-E7-E3fails because all three wall supports do not lie in the same plane. A solution in this case might be to create two separate joist areas, B3-B5-E5-E3 and B5-B7-E7-E5.

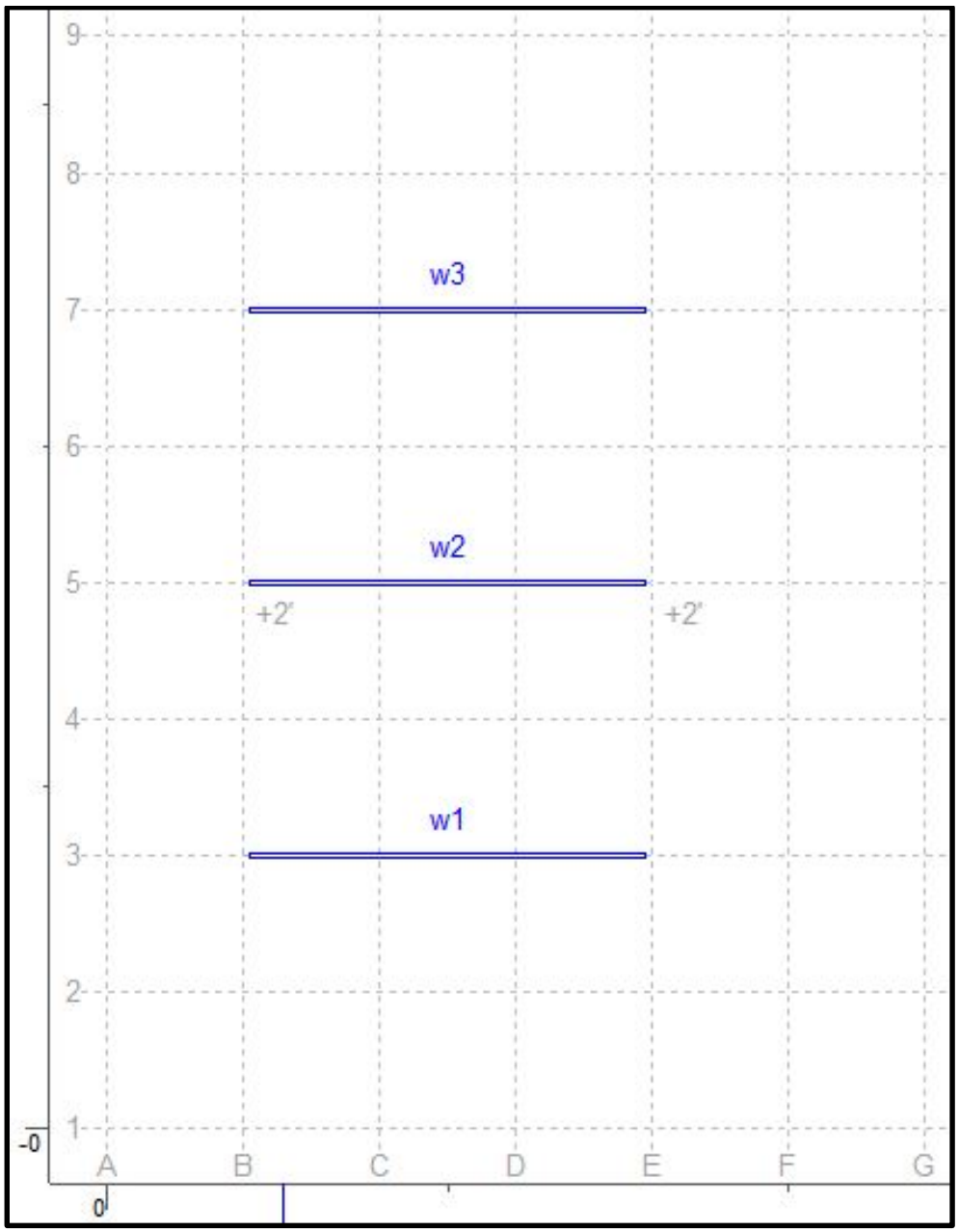



Figure 149: *Out-of-plane Supports*

### 19.7.10 Out-of-plane joist areas

*Error* Message: "The Following Joist areas could not be designed because they do not lie in a plane: j1. However, loads can be transferred from the joist area to supporting members. See on-line help for full explanation."

*Sizer* cannot design out-of-plane joist areas. Despite the fact that the two beam supports do not lie in the same plane – support C6-F6 is sloped and support C3-F3 is horizontal – the joist area C3-C6-F6-F3 can be create, but when completing the design, the Error Message will appear. However, *Sizer* cannot design such out-of-plane joist areas. *Note that if there are more than two supports, they must all lie in the same plane.*

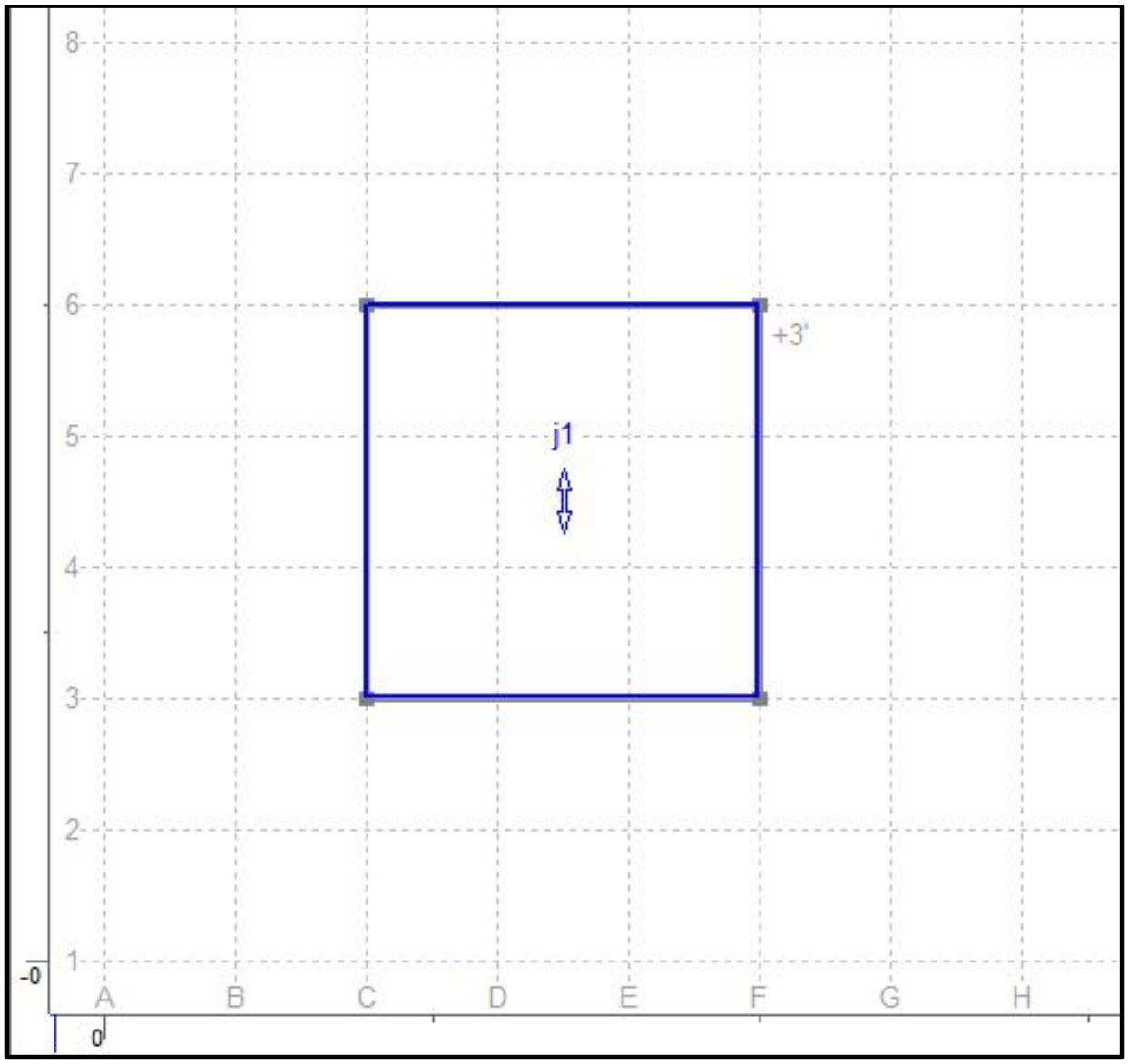



Figure 150: *Out-of-plane joist areas*

### 19.7.11 Supports Intersect under a joist area

*Error* Message: *"Unable to create joist area. Beams or walls supporting joist area in same direction intersect. See on-line help for full explanation."*

In this example, an attempt to create the joist area B2-B6-E6-E2 fails because the potential supports b2 and b3 intersect within the joist area. It is, however, possible to create a joist area that does not include the intersection; B2-B6-C6-C2 is allowed because the intersection is outside the joist area.

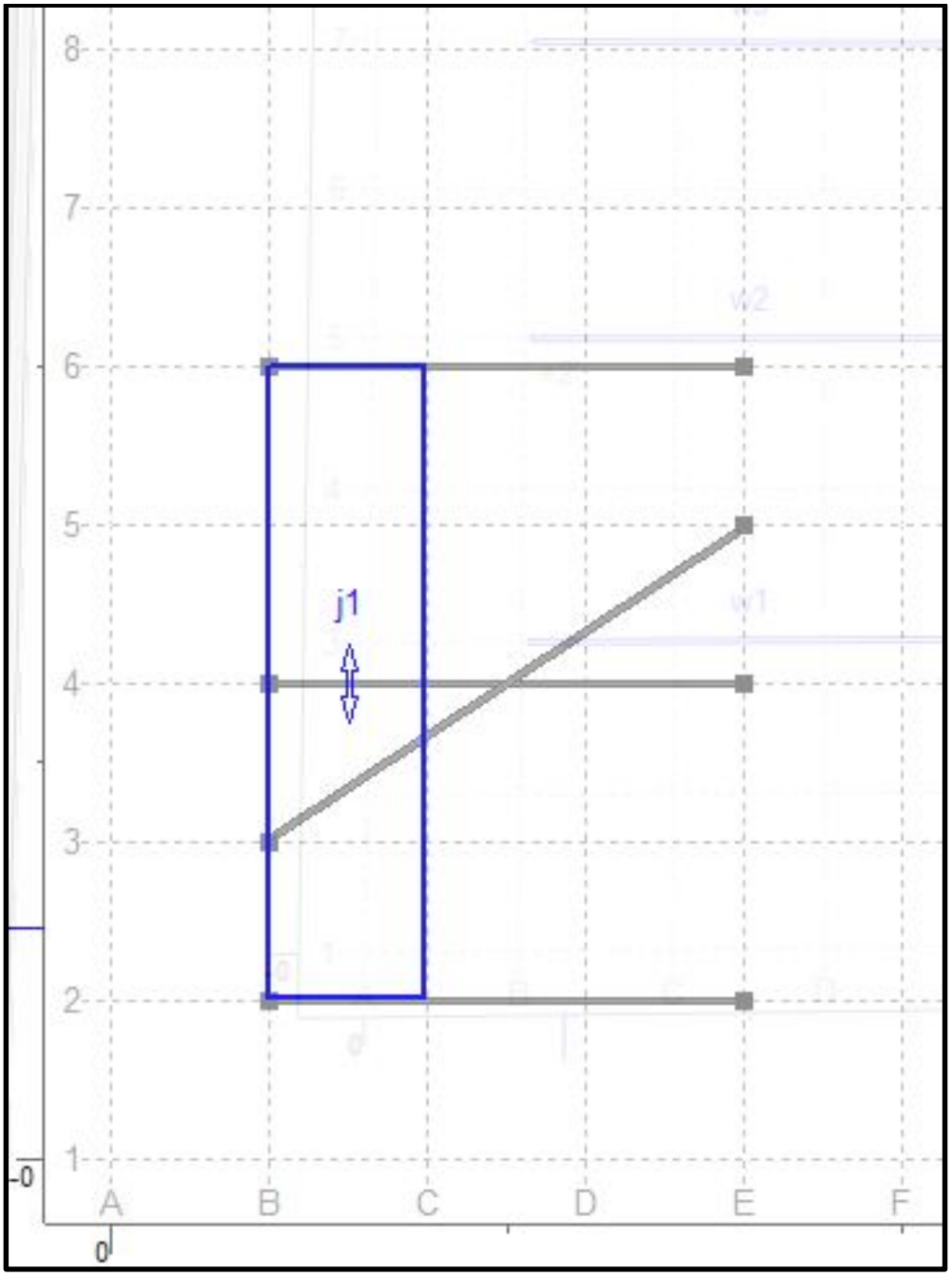



Figure 151: *Supports Intersect Under a Joist Area*

### 19.7.12 Supports Intersect at the edge of a joist area

*Error Message*: "Unable to create joist area. Beams or walls supporting joist area in same direction intersect. See on-line help for full explanation."

In this example, an attempt to create the joist area B2-B6-E6-E2 fails because potential supports b5 and b2 meet on the edge of the joist area. Not all joists would have the same number of supports if this was allowed.

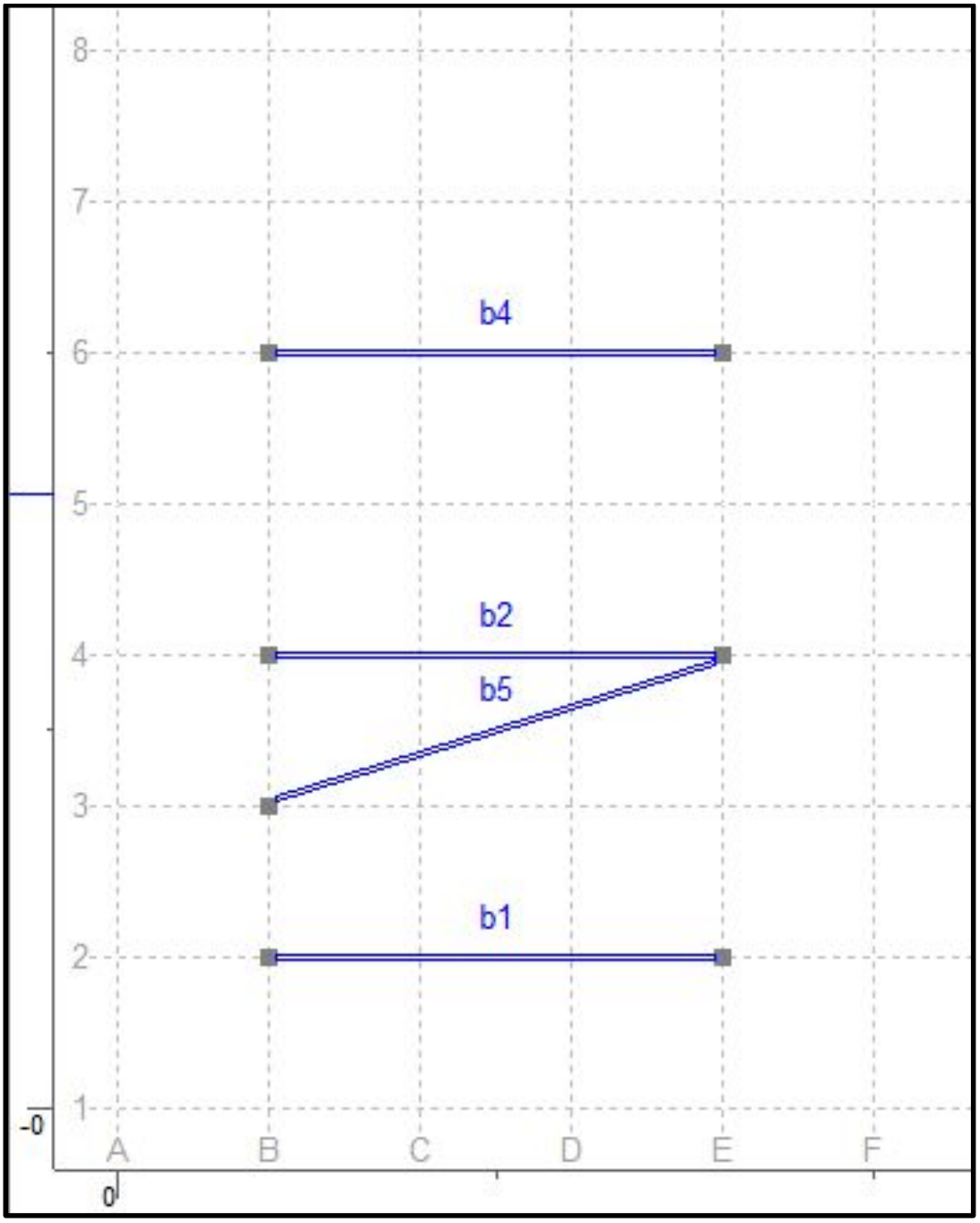



Figure 152: *Supports Intersect at End of Joist Area*

### 19.7.13 Example of Transferring Joists from Concept to Beam Mode

***Select Joist for transfer to Beam mode***

Now that *Sizer* permits multi-span joist areas and wall loads on joist areas, it is no longer the case that the longest joist from the joist area is critical. You are now given a choice between the two joist edges when transferring the joist to beam mode from concept mode. During the design process of the *Concept* mode, the edges are designed separately and the more critical side controls the design.

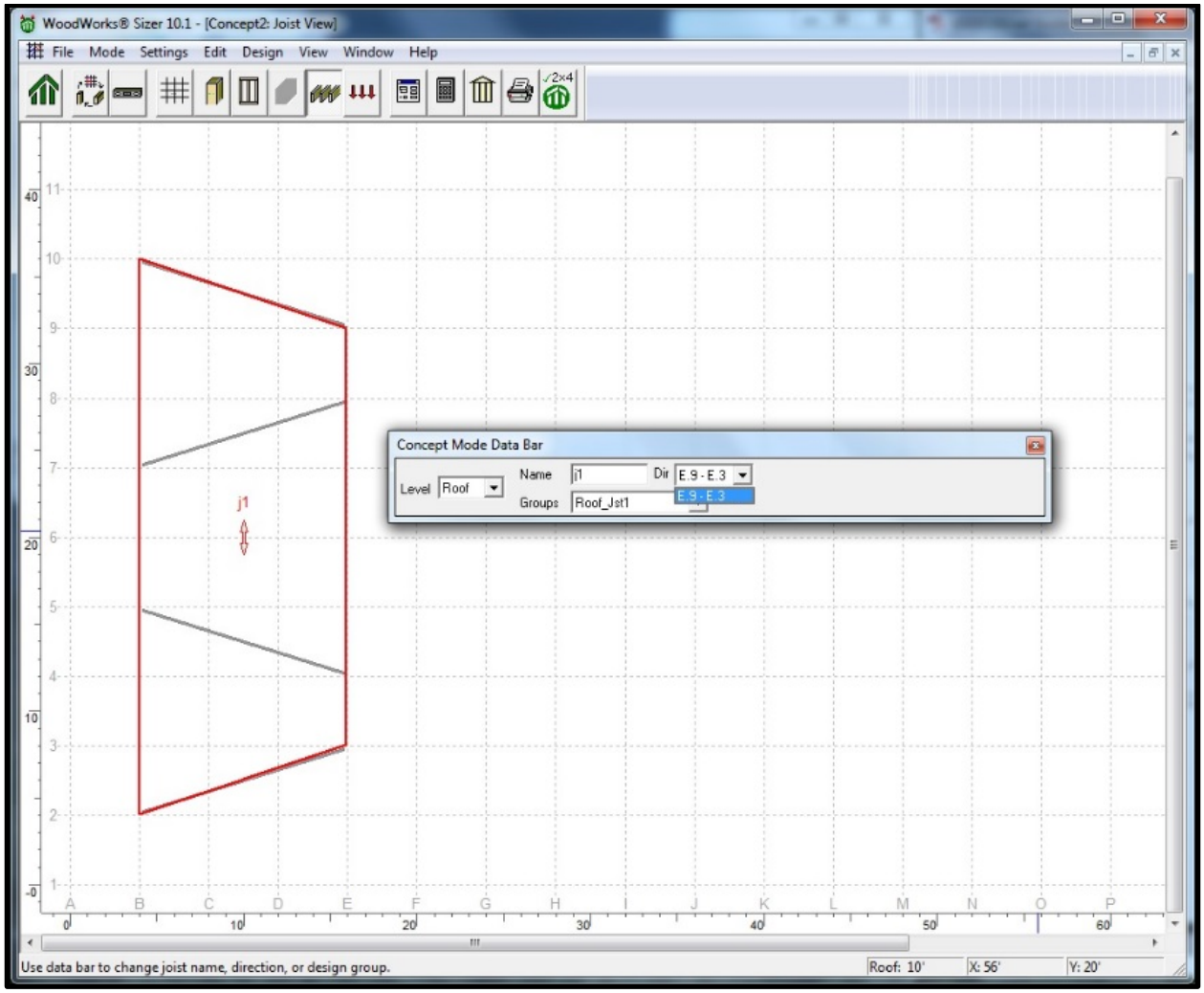



Figure 153: *Example of Transferring Joists from Concept Mode to Beam Mode*

# 20 Concept Mode Loads

## 20.1 Creating Loads

Click ***load*** on the toolbar. The ***load*** button on the toolbar appears as light grey to indicate Loads view. The *Sizer* window title also indicates the current view.

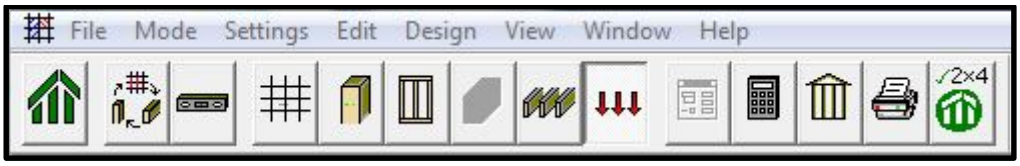



Figure 154: *Load View*

In *Loads* view, area loads appear as diagonal blue lines. Area loads appear as cross-hatched diagonal lines if both live and dead area loads are applied to the same area. Line loads appear as solid blue lines in *Loads* view.

In *Grid*, *Column*, *Wall*, *Beam*, and *Joist* views, loads are not shown.

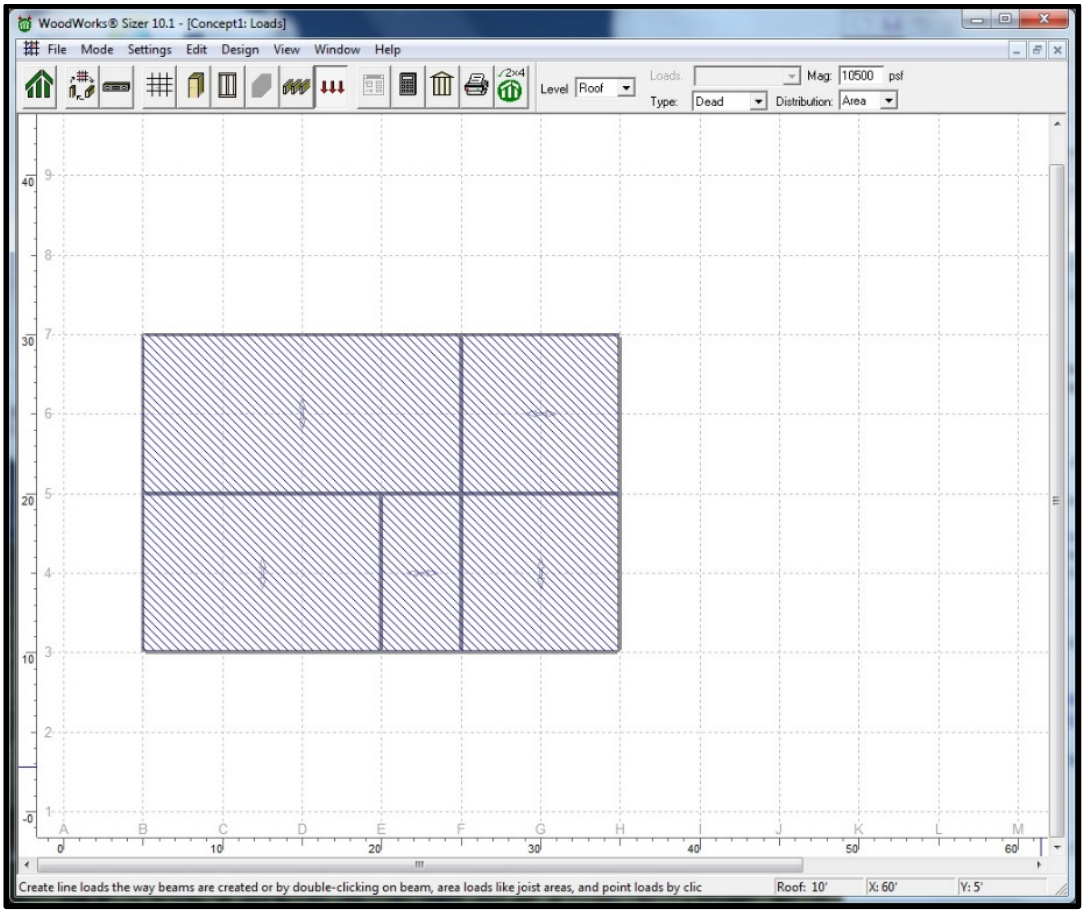



Figure 155: *Loaded Structure*

Before you create a load, you must choose its type and its magnitude. This is done through the *Concept Mode Data* Bar by selecting the load type (dead, live, snow, wind, earthquake etc.) from the **Type** drop-down list and selecting the load distribution (Line, Area or Point). Then enter the magnitude of the load as a numeric value in the field directly to the right of the **Type** drop-down list. To load a rectangular area, click on the four gridpoints at the corners of the area in question. As you click, a rubber-band line stretches from one gridpoint to the next. *Sizer* fills the load area defined by these four gridpoints with diagonal blue lines.

*Since a structure may not be rectangular, Sizer allows you to create a load area larger than the structure. The load is automatically applied only to the joists contained within the selected area. To create a uniformly distributed line load, double-click a beam. The loaded beam is displayed in blue. Load walls the same way.* You can create any number of overlapping loads.

### 20.2 Deleting Loads

To delete an area or beam load, select it and press the ***DELETE*** key on the keyboard

### 20.3 Concept Mode Data Bar

The *Concept Mode Data* toolbar is active in the Load view when a load is selected. The *Concept Mode Data* toolbar allows you to edit and displays information on the selected load. This includes the load name, type, and magnitude.


Figure 156: *Specifying Loads Toolbar*

*Note: To select a load, click on the load. Selected loads appear in red. If loads overlap, simply click repeatedly until you reach the desired load.*

### 20.4 Load Name

If a load is selected, the *Concept Mode Data* toolbar will display the load name.

As a default, the load name is **load1** for the first load, **load2** for the second, and so on. This can be changed however by typing over the existing name included in the toolbar.

### 20.5 Load Type

If a load is selected, the *Concept Mode Data* toolbar will display the load type. This drop-down list contains the types of loads available (dead or live).

*Note: Live loads applied on the roof level are automatically treated as Snow loads. To specify other load types for roofs (e.g. Wind), change the load duration factor for the Snow load type in the Load Duration Factors dialog box (U.S. only).*

### 20.6 Load Magnitude

If a load is selected, the Concept Mode Data toolbar will display the load type. This field allows you to assign the magnitude for an area load (psf or kN/m2) or for a line load (plf or kN/m).

## 20.7 Loads and Load Combinations

### 20.7.1 Load Direction

All live and dead loads are applied vertically downward.

### 20.7.2 Load Location

*Sizer* automatically applies dead loads along the length of a sloped member. Live loads are applied along the horizontal projected length of a sloped member.

### 20.7.3 Load Combinations

Combinations are automatically generated for the following loads:

- dead load only
- dead and live loads

### 20.7.4 Pattern Loads

Pattern loads are only available in Beam mode for live, snow, wind, construction (U.S. only) and impact loads (U.S. only) on multi-span beams.

### 20.7.5 Load Transfer

Loads are transferred in Concept mode using the following rules:

- Joists transfer to supporting beams or walls.
- Beams transfer to supporting beams or columns.
- Walls transfer to supporting walls or beams on a lower floor.
- Columns transfer to columns or beams on a lower floor.

*Sizer determines whether a beam is supporting or supported based solely on the Load Transfer Number (LT#) entered for the beam's design group. A beam with a higher LT# always transfers loads to beams with lower LT#'s. In other words, a beam with a lower LT# always supports a beam with a higher LT#.*

*There are two points to note, however:*

- *Sizer determines maximum joist and beam reactions from a structural analysis of each member. In the case of joist areas, Sizer only analyzes the first and last joists by determining the reactions at the two supports of the first and last joists. Sizer uses these computed reactions as the start and end values of a distributed load on the supporting beams or walls. It is assumed that the resulting distributed load varies linearly between the first and last joists. In the case of a rectangular joist area on parallel supports, this produces a uniformly distributed load on the beam or wall. Any other conditions produce a trapezoidal load on the beam or wall. Should uplift occur at one end of a joist area but not the other, Sizer computes two triangular distributed loads—a positive load over a portion of the supporting member and a negative load over the remainder of the member.*
- *Walls supported by beams on a lower floor cannot span over a column unless the beam is continuous over the column.*

# 21 Concept Mode Elevation View

*Sizer* includes a feature to view the elevation or cross-section of your structure along any gridline. To do this, change to the *Grid* view and select any gridline in either the North-South or East-West directions. The selected gridline will appear in red. Now click on the ***elevation view*** button and the program automatically displays the elevation view along the selected gridline.

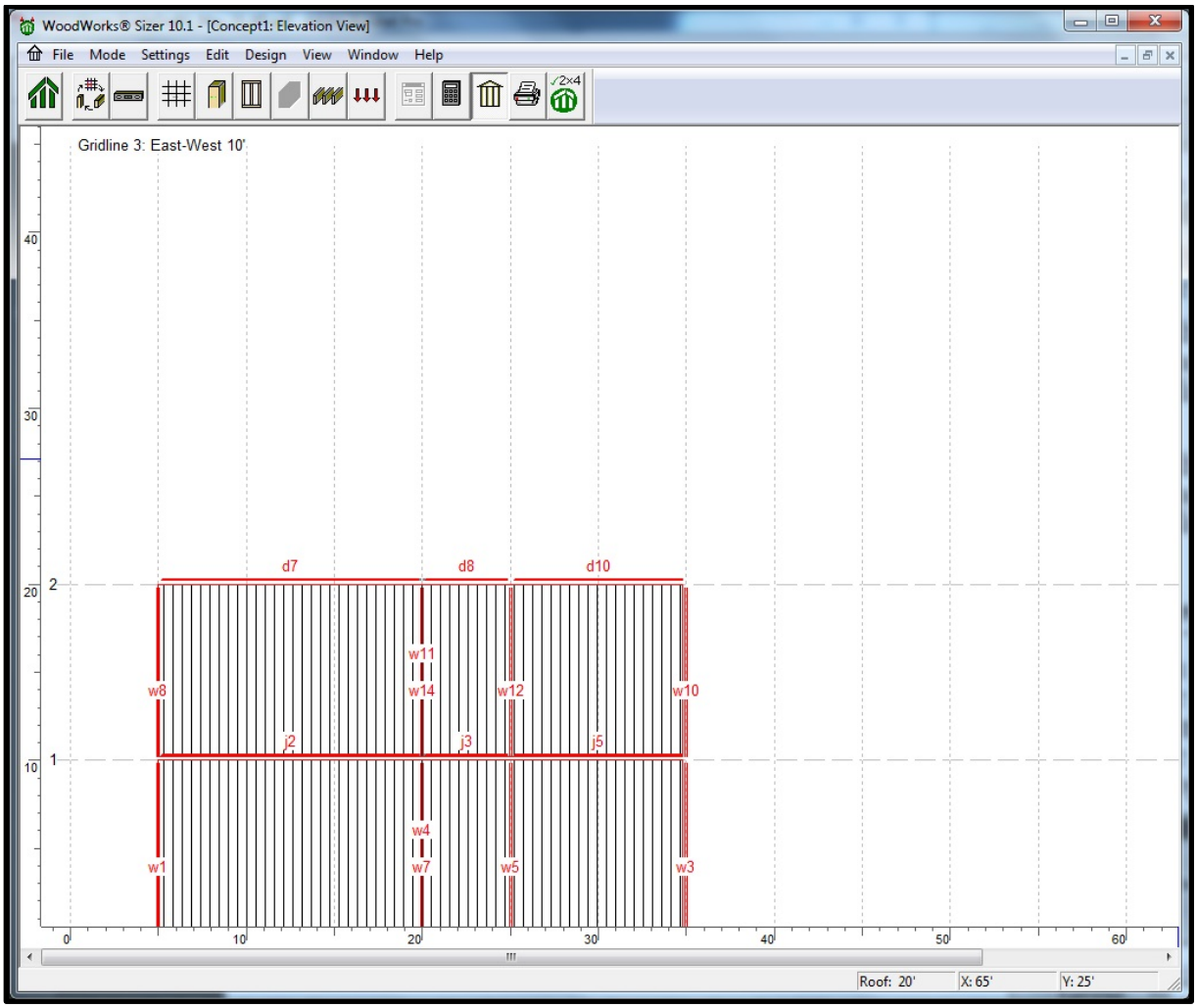



Figure 157: *Example Elevation View*

## 21.1 Member Display

The elevation view displays members in the following fashion:

- Members that are on the selected gridline are shown in red.
- Members that are not on the current gridline and are South or East of the gridline are shown in light grey lines.
- Members that are not on the current gridline and are North or West of the gridline are shown in dark grey lines.

## 21.2 View Options

The elevation view can be enhanced to include the member names and the group names by selecting these options in the ***View*** tab from the ***Settings*** dialog.

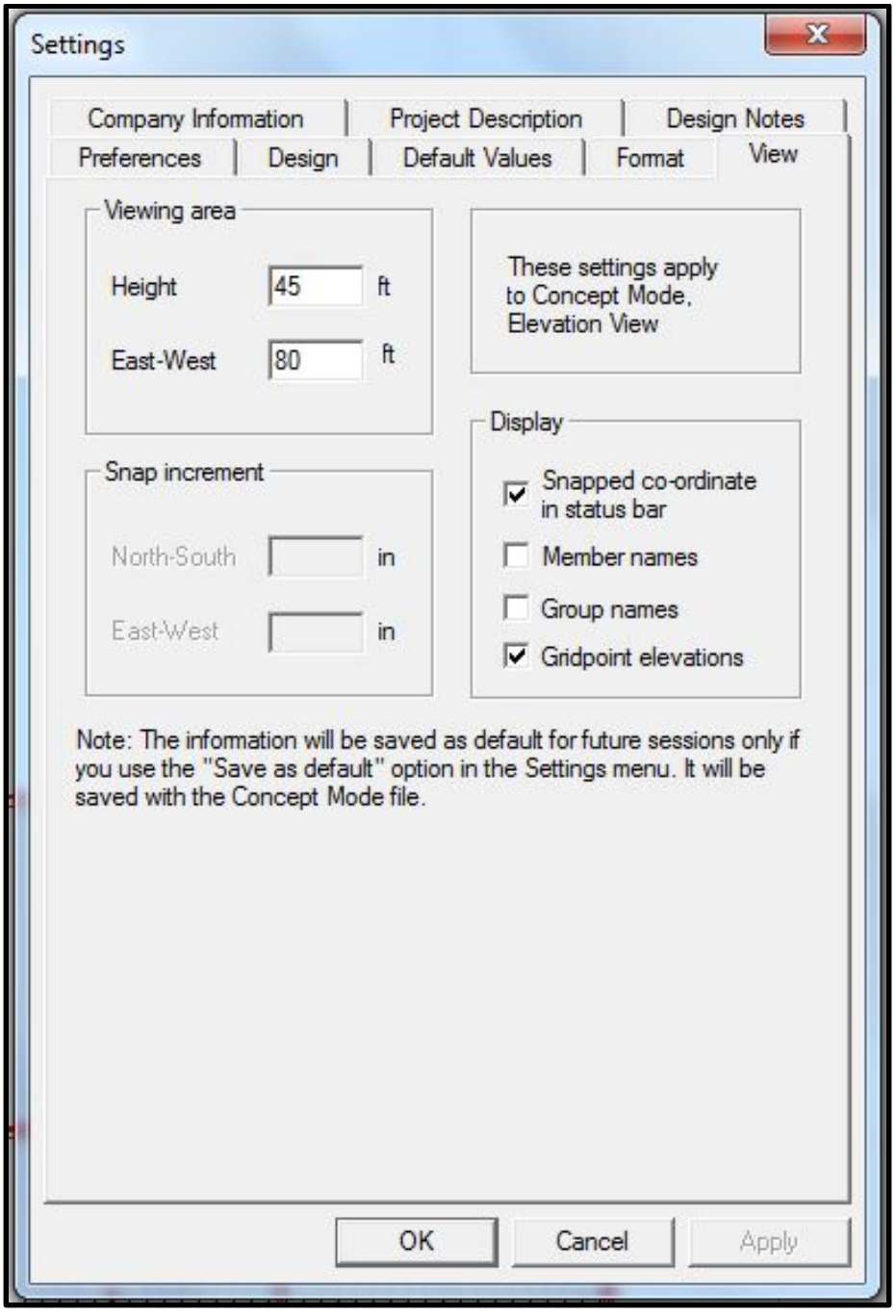



Figure 158: *Concept Mode View Settings*

# 22 Concept Mode Design Process

## 22.1 Starting the Design Process

To start the design process, click ***design*** on the toolbar. *Sizer* performs an analysis and a design of your structure with the information you entered and automatically displays the results.

*Sizer normally prompts you to save the current project prior to doing the design. To change this, choose one of the **Save Before Design** options from the **Preferences** tab under the **Settings** dialog.*

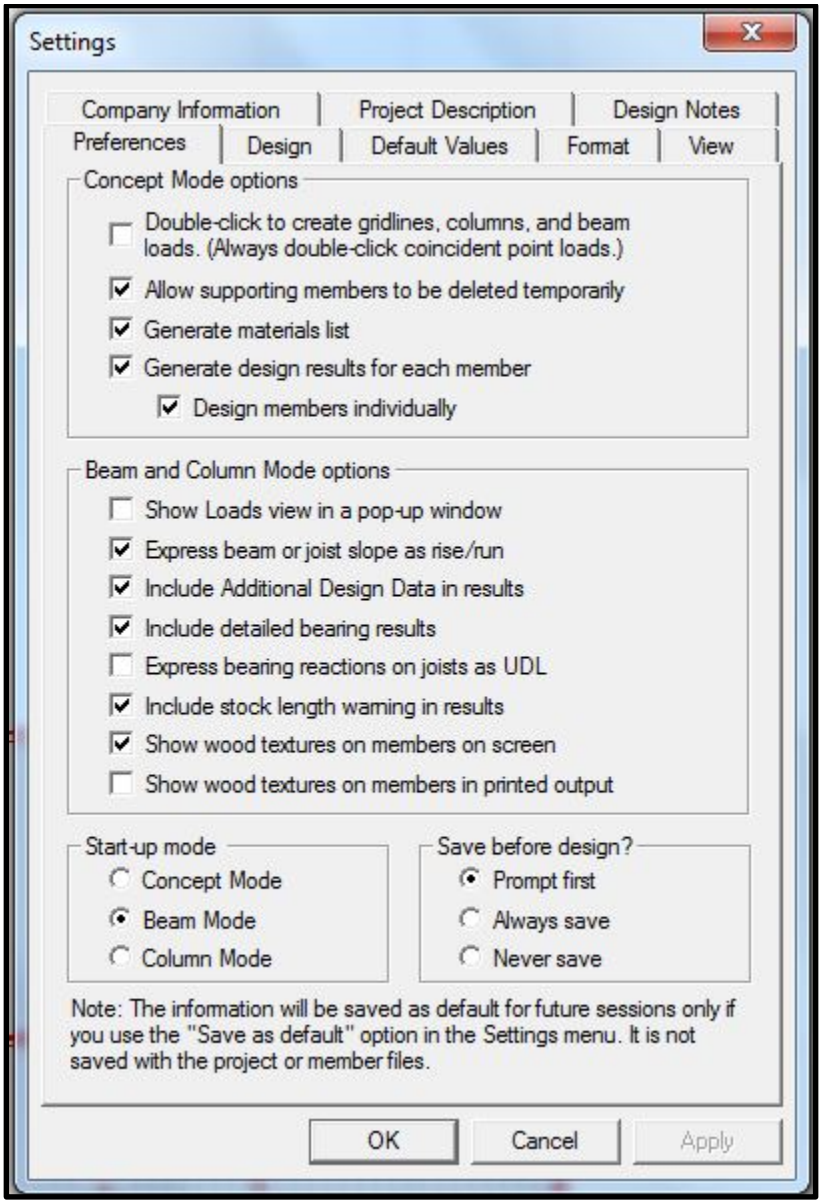



Figure 159: *Sizer Design Settings - Preferences*

## 22.2 Limiting the Design

By default, *Sizer* designs the elements for your entire project. However, you can force *Sizer* to design only the elements on the current level.

To limit the design, choose **Design Current Level Only** from the **Preferences** item under the **Settings** menu.

To design the elements for the entire project, choose **Design Entire Structure** from the **Preferences** tab under the **Settings** dialog.

## 22.3 Viewing and Printing the Results

Once *Sizer* has designed the elements in your project, it automatically displays the design summary on the screen.

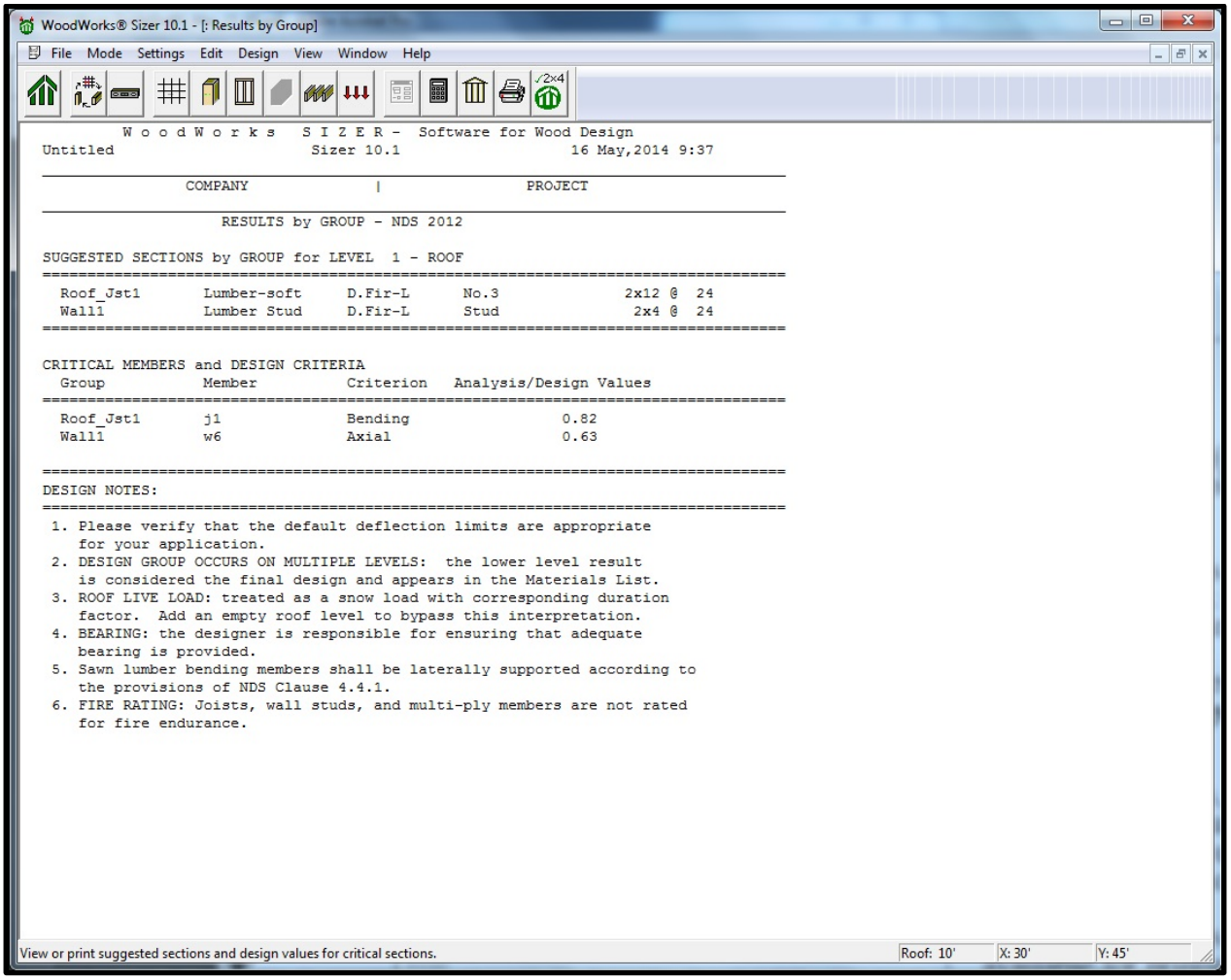



Figure 160: *Results by Group*

Use the scrollbars (or the PgUp and PgDn keys) to scroll though the results.

To close the results window, double-click the close box in the top left corner of the window.

To redisplay the design results at any time, choose ***Design Results*** from the ***View*** menu.

To print the results, choose ***Print*** from the ***File*** menu. The Print submenu provides several options (such as Suggested Sections).

There are new files and diagrams available for printing in Concept mode.

***Reactions at Base*** drawing shows that Column and Wall reactions at the base of a structure.

***Results by Member*** shows the design response of all the members in each design group.

***Design Members Individually*** shows the design of each member in the entire structure.

***Materials List*** tabulates all the material used in the structure.

# 23 Concept Mode Advanced Features

## 23.1 Performing a Detailed Design

The *Concept* mode design process computes the lightest acceptable section size for each design group. However, these results show only a limited amount of detail, and they do not indicate the margins by which the sections passed the various criteria (bending, shear and deflection).

*Sizer can perform a detailed design for single beams, joists, columns and walls. This is done by importing the critical member into the Beam or Column mode. To import, click on the critical member so that it appears in red, and then press the* ***mode*** *button on the main toolbar. This imports the member into the appropriate single member mode for a more detail design and analysis.*

*Sizer* first determines the reactions of the other members to be transferred to the selected one, and converts the reactions to the appropriate types of loads. *Sizer* then transfers these loads (along with all the design group information) to Beam or Column mode, where you can perform a detailed design or code check on the member.

The load names of these transferred loads indicate where each of the loads are coming from. For example, a load named *1_j1* would come from joist area *j1*.

*Note: You cannot transfer a member from Beam mode or Column mode back to Concept mode.*

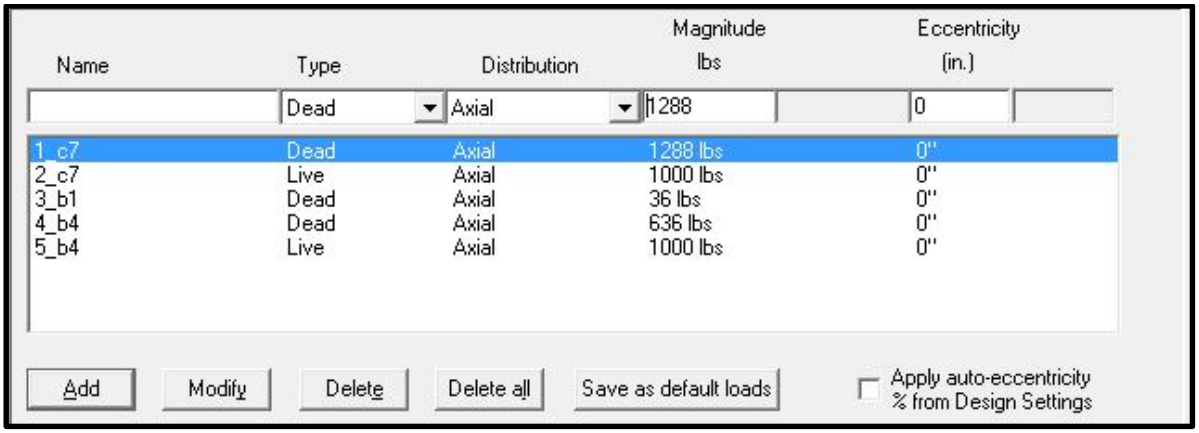



Figure 161: *Beam Mode Load View*

## 23.2 Reactions at Base

Select this option from the View menu (or from the right-button context menu) to display wall and column reactions at the base of the structure. Wall loads are output as line loads (plf or kN/m) and columns loads as point loads (lbs or kN).

Due to the way *Sizer* designs walls it is possible that, for some layouts, the sum of the reactions will exceed the sum of the applied loads. This occurs when the intensity of loads applied to the top of a wall is not uniform along the entire wall length. *Sizer* designs walls by determining the largest load on a single wall stud within the wall. This same design load becomes the reaction that is transferred to a lower story as a uniform load along the entire length of the wall. This is a conservative approach since the single largest load at the top becomes the uniform reaction at the bottom.

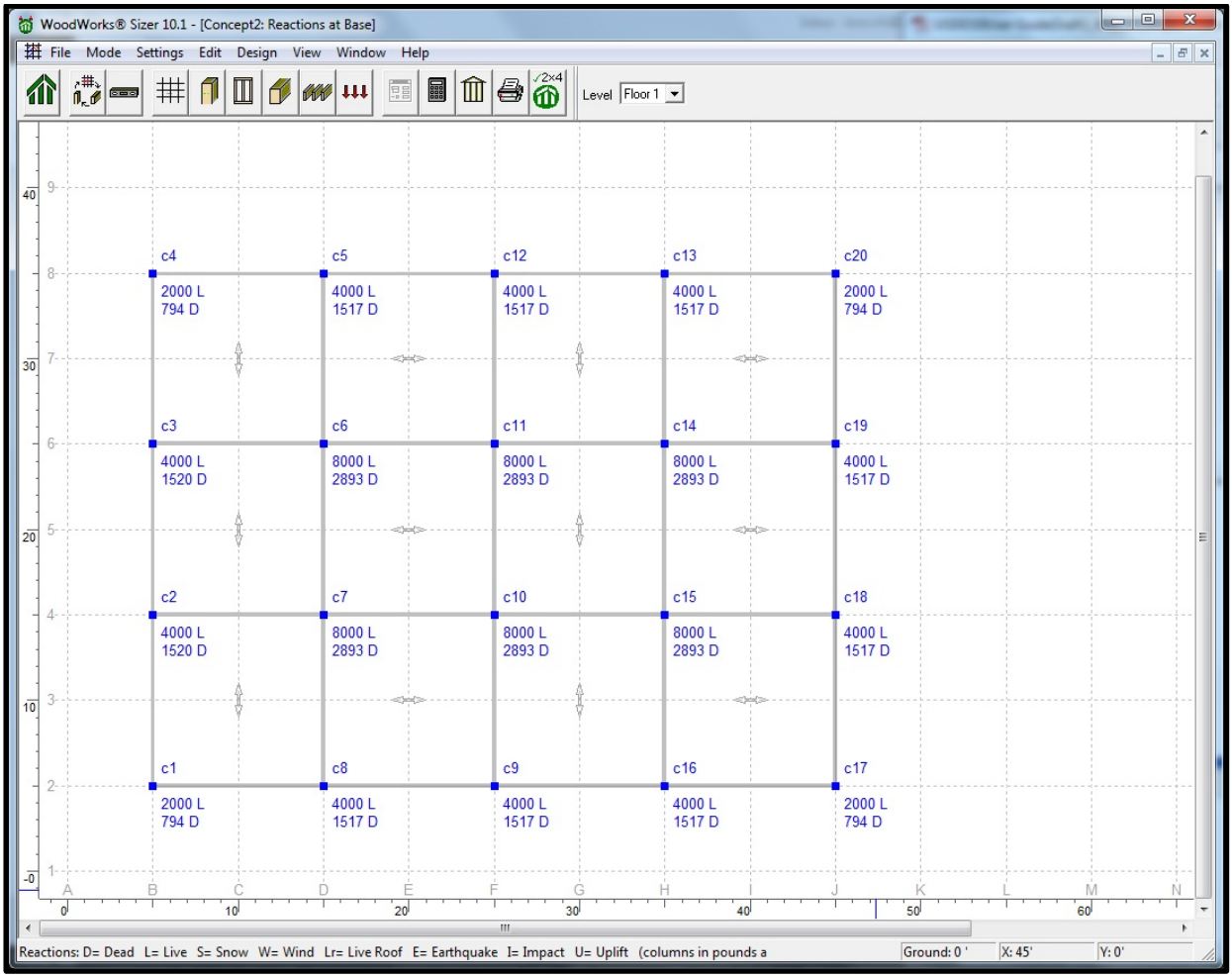
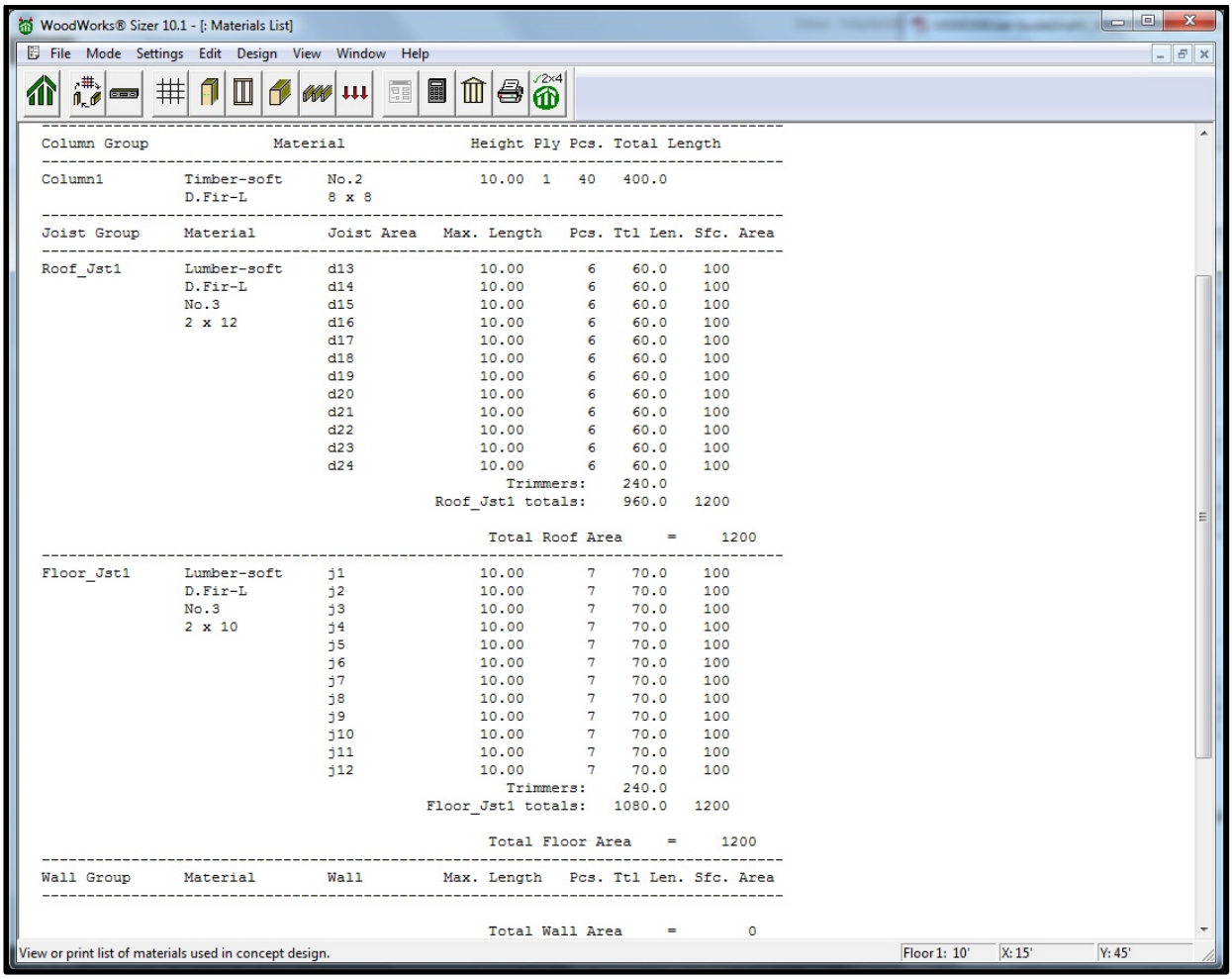



Figure 162: *Reactions at Base*

## 23.3 Materials List

Select this option from the *View* menu (or from the right-button context menu) to display a list of materials. All the materials used in the structure are listed in a table, organized by group. The table includes the number of pieces for each length of the material, area of floors, roofs and walls, total length of joist trimmers and wall plates.




| Column Group | Material | | Height | Ply | Pcs. | Total Length |
|---|---|---|---|---|---|---|
| Column1 | Timber-soft | No.2 | 10.00 | 1 | 40 | 400.0 |
| | D.Fir-L | 8 x 8 | | | | |

| Joist Group | Material | Joist Area | Max. Length | Pcs. | Ttl Len. | Sfc. Area |
|---|---|---|---|---|---|---|
| Roof_Jst1 | Lumber-soft | d13 | 10.00 | 6 | 60.0 | 100 |
| | D.Fir-L | d14 | 10.00 | 6 | 60.0 | 100 |
| | No.3 | d15 | 10.00 | 6 | 60.0 | 100 |
| | 2 x 12 | d16 | 10.00 | 6 | 60.0 | 100 |
| | | d17 | 10.00 | 6 | 60.0 | 100 |
| | | d18 | 10.00 | 6 | 60.0 | 100 |
| | | d19 | 10.00 | 6 | 60.0 | 100 |
| | | d20 | 10.00 | 6 | 60.0 | 100 |
| | | d21 | 10.00 | 6 | 60.0 | 100 |
| | | d22 | 10.00 | 6 | 60.0 | 100 |
| | | d23 | 10.00 | 6 | 60.0 | 100 |
| | | d24 | 10.00 | 6 | 60.0 | 100 |
| | | | Trimmers: | | 240.0 | |
| | | | Roof_Jst1 totals: | | 960.0 | 1200 |
| | | | Total Roof Area = | | | 1200 |
| Floor_Jst1 | Lumber-soft | j1 | 10.00 | 7 | 70.0 | 100 |
| | D.Fir-L | j2 | 10.00 | 7 | 70.0 | 100 |
| | No.3 | j3 | 10.00 | 7 | 70.0 | 100 |
| | 2 x 10 | j4 | 10.00 | 7 | 70.0 | 100 |
| | | j5 | 10.00 | 7 | 70.0 | 100 |
| | | j6 | 10.00 | 7 | 70.0 | 100 |
| | | j7 | 10.00 | 7 | 70.0 | 100 |
| | | j8 | 10.00 | 7 | 70.0 | 100 |
| | | j9 | 10.00 | 7 | 70.0 | 100 |
| | | j10 | 10.00 | 7 | 70.0 | 100 |
| | | j11 | 10.00 | 7 | 70.0 | 100 |
| | | j12 | 10.00 | 7 | 70.0 | 100 |
| | | | Trimmers: | | 240.0 | |
| | | | Floor_Jst1 totals: | | 1080.0 | 1200 |
| | | | Total Floor Area = | | | 1200 |

| Wall Group | Material | Wall | Max. Length | Pcs. | Ttl Len. | Sfc. Area |
|---|---|---|---|---|---|---|
| | | | Total Wall Area = | | | 0 |

Figure 163: *Material List*

## 23.4 Results by Member

### 23.4.1 Designed by Group

Each member is listed in a table that displays how it performed with respect to various design criteria (bending, shear, deflection, axial, and fire endurance). The information in this table can be used to evaluate which members in a group are the least or most “stressed”. You can disable this feature in the *Settings Preferences* tab. Activating this feature increases the time it takes *Sizer* to complete a design.

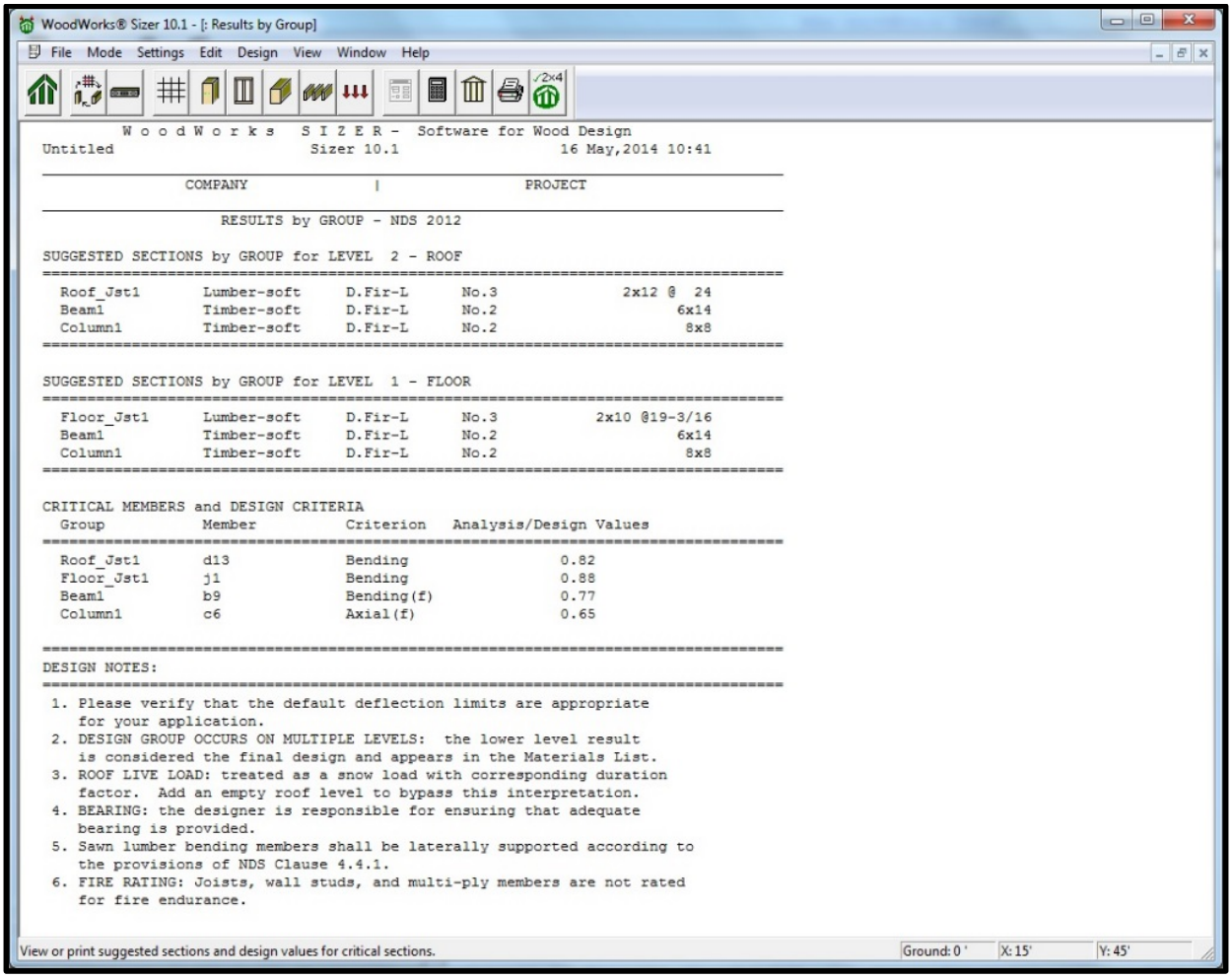



Figure 164: *Designed by Group*

### 23.4.2 Individual Design

*Sizer* includes an option that allows you to design each member of the structure individually rather than designing using the worst case member in the design group. You can enable this feature through the **Settings Preferences** tab by checking the ***Design members individually*** box. This option will modify the *Results by Member* report to include the design for each member along with information on how each member performed.

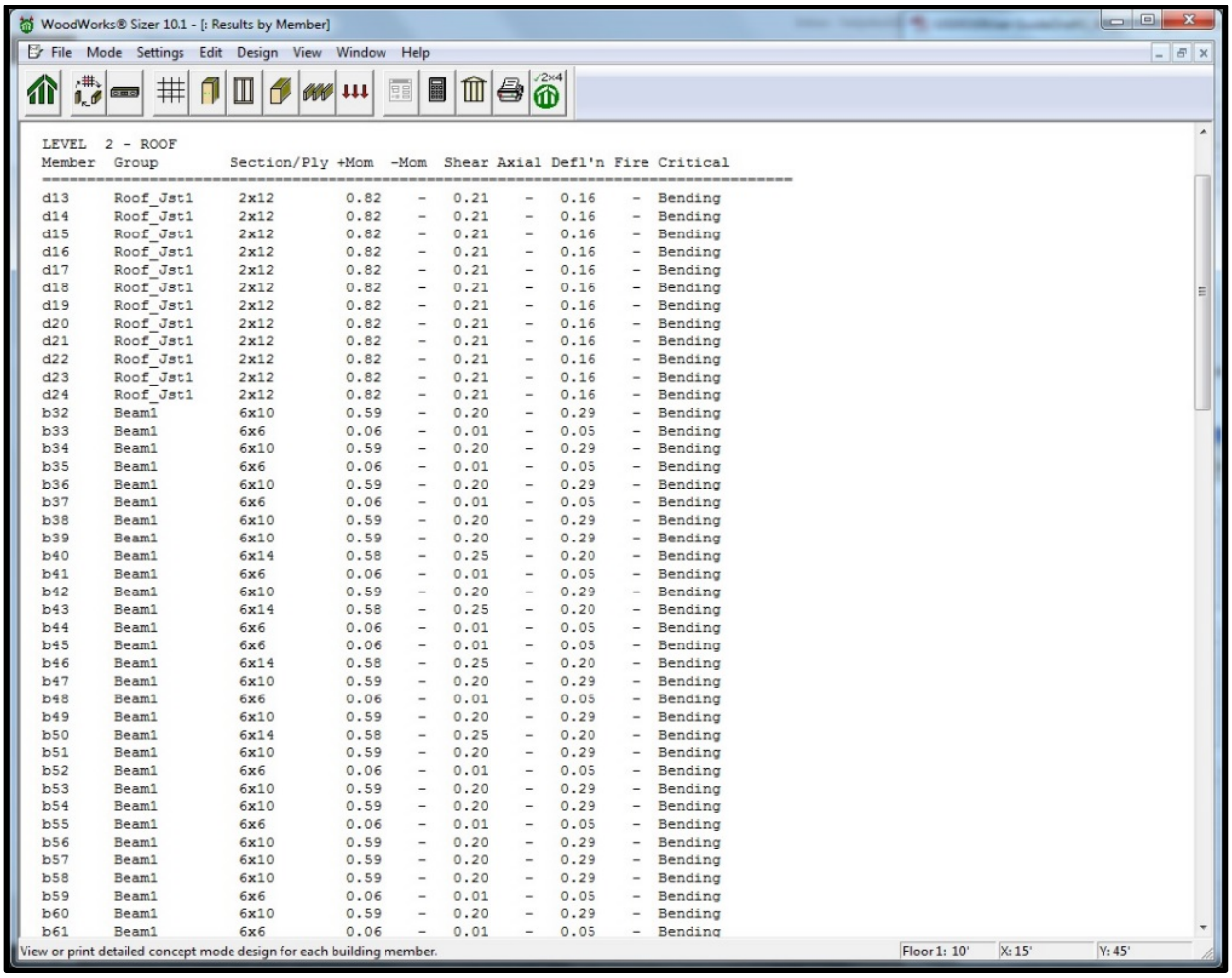



Figure 165: *Individual Design*

# 24 About *Shearwalls*

*Shearwalls* is an engineering design aid for lateral wind and seismic load determination and for the design of shearwalls for buildings subject to these loads. *Shearwalls* will:

- Import CAD drawings in .emf, .wmf, .bmp or .pdf files to establish the building footprint for each level
- Model wood structures with up to six stories
- Automatically calculate seismic and wind loads based on the building code, relevant standards and site information
- Accept user specified seismic and wind loads applied to building surfaces
- Accept input for dead loads and wind uplift loads acting on shearwalls
- Distribute shear forces to shearwalls using either a flexible or rigid diaphragm analysis (including torsional effects)
- Accept input of forces applied directly to shearwalls to adjust the shearline load distributions
- Allow the specification of openings in shearwalls, such as windows and doors

Design shearwalls for their sheathing thickness and nailing requirements

- Determine the capacity of wall sheathing and nailing to resist wind suction
- Specify required hold-down forces for shearwall segments
- Specify required dragstrut forces across openings and at the ends of shearwalls
- Design standard shearwalls with hold-downs at each segment, perforated shearwalls (U.S. only) or shearwalls with anchorages in lieu of hold-downs (Canada only).

***Getting Started*** *Button – Provides instructions on the main steps required to design a building using Shearwalls. Check out the following tutorials which show how to design a simple structure using shearwalls:*

*Canadian - https://www.youtube.com/watch?v=chBez-UFAwQ&feature=youtu.be*

*U.S. - https://www.youtube.com/watch?v=K9X8us4gX-M*

## 24.1 User Interface

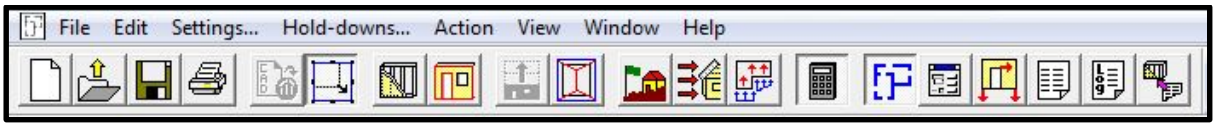



Figure 166: *Shearwalls Toolbar*

The toolbar across the top of the main window is split into three groups of buttons:

- File Operations: *New*, *Open*, *Save*, *Print*

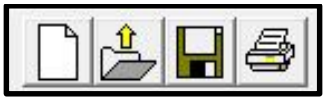


Figure 167: *Shearwalls Toolbar Part 1*

**Actions:** *Import*, *Structure*, *Walls*, *Openings*, *Extend Walls Upward*, *Roof Blocks*, *Building Site*, *Generate Loads*, *Loads and Forces*, *Run Design*

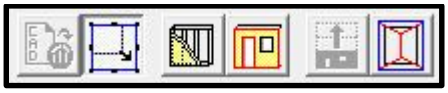


Figure 168: *Shearwalls Toolbar Part 2*

**Views:** *Plan View, Form View, Elevation View, Results View, View Log Files, Accept Design*

Figure 169: *Shearwalls Toolbar Part 3*

For both the ***Action*** and the ***View*** buttons, a depressed button indicates the current activity or view.

### 24.2 Status Bar

The bar at the bottom of the screen displays a line of text that contains hints and instructions to the user, and descriptions of the programs buttons, menu items, and data fields. Consult the status bar when you are uncertain as to how to proceed.


Figure 170: *Shearwalls Status Bar*

## 25 *Shearwalls* Design Settings

### 25.1 General

*Shearwalls* includes a comprehensive *Settings* dialog box that is accessed by pressing the ***Settings*** button in the data bar. It allows the user to control graphical interface options such as the *units*, *format*, *font sizes*, *view area* and *snap increments*; to specify design settings; and to filter what will be shown in the Plan, Elevation and Results views.

The *Settings* menu options are organized into nine tabs:

*Design*, *Default Values*, *View*, *Hold-downs*, *Format*, *Company Information*, *Options*, *Project Description*, and *Loads and Forces.*

A general description of these settings is given in the following. *Note that the settings for each tab of the dialog box are independent of those for all other tabs.*

*Check out the following Tutorials in regards to the Shearwalls Design Settings:*

*Canadian - https://www.youtube.com/watch?v=wHoxqWjfnvg&feature=youtu.be*

*U.S. - https://www.youtube.com/watch?v=4Pap1YfQdEE*

## 25.2 Design Tab

Figure 171 shows the *Shearwalls* Design tab for the Canadian edition of the software and Figure 172 shows the *Shearwalls* Design tab for the U.S. edition of the software.

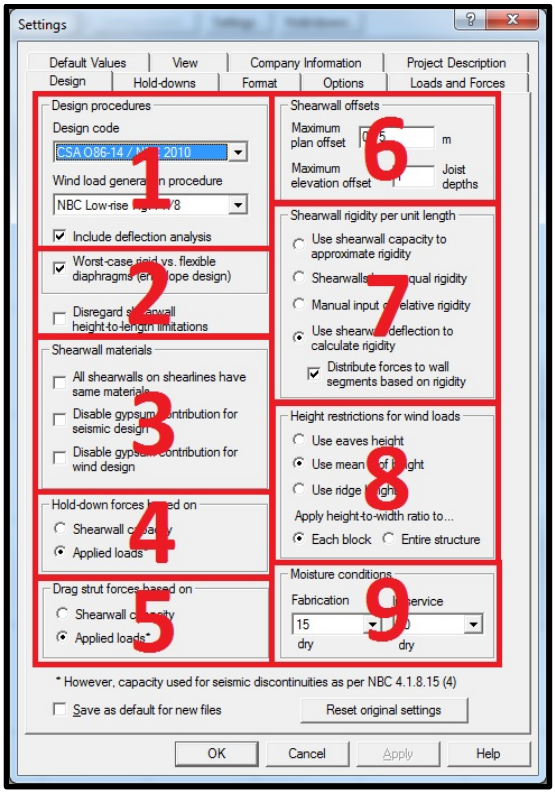



Figure 171: *Shearwalls Settings – Design Tab (Canadian)*

The following Table Summarizes where information on each Part of Figure 239 can be found. The section numbers have been hyperlinked for convenience:

| | Section |
|---|---|
| Part | Figure 171 (Canadian) |
| 1 | 31.2.2 |
| 2 | 31.2.4/31.2.5 |
| 3 | 31.2.10 |
| 4 | 31.2.9 |
| 5 | 31.2.18 |
| 6 | 31.2.11 |
| 7 | 31.2.13 |
| 8 | 31.2.14 |
| 9 | 31.2.16 |

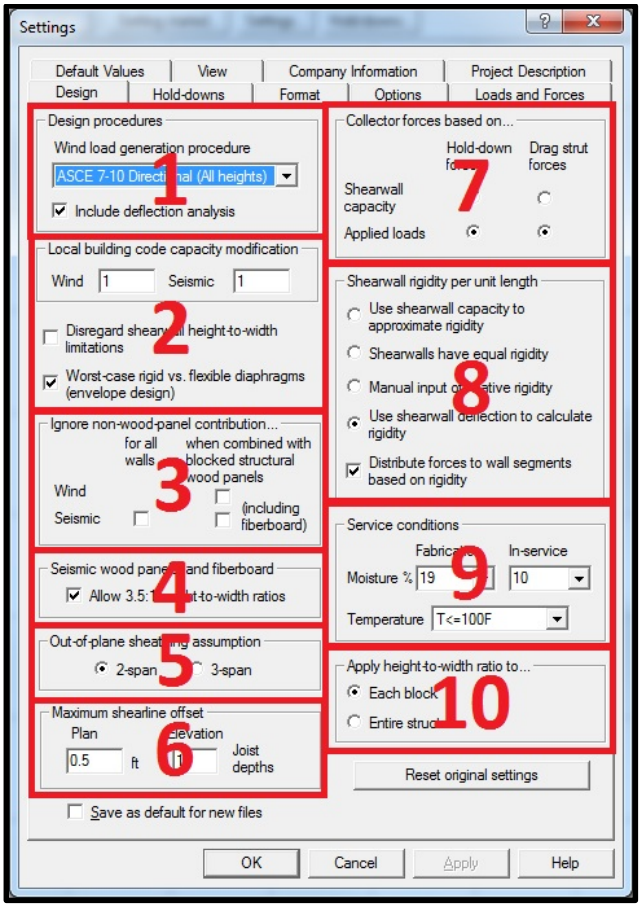



Figure 172: *Shearwalls Settings – Design Tab (U.S.)*

The following Table Summarizes where information on each Part of Figure 172 can be found. The section numbers have been hyperlinked for convenience:

| | Section |
|---|---|
| Part | Figure 172 (U.S.) |
| 1 | 31.2.1 |
| 2 | 31.2.4/31.2.5 |
| 3 | 31.2.6 |
| 4 | 31.2.7 |
| 5 | 31.2.8 |
| 6 | 31.2.11 |
| 7 | 31.2.12 |
| 8 | 31.2.13 |
| 9 | 31.2.15 |
| 10 | 31.2.17 |

### 25.2.1 Design Procedures – Wind Load Design Standard (U.S.)

Design procedures allows the user to select whether to generate wind loads using ASCE 7-10 Directional (All heights) or ASCE 7-10 Envelope (Low-rise).[8] Selecting a deflection analysis not only activates the calculation of shearwall deflections, it also activates the option of distributing forces based on stiffness derived from deflections.

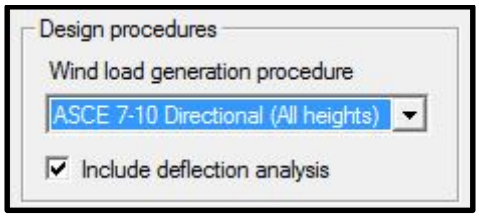



Figure 173: *Wind Load Generation Procedure (U.S.)*

### 25.2.2 Design Procedures (Canadian)

Design procedures allows the user to select the *Design Code* and the *Wind load generation procedure*. The software can perform the design based on the CSA O86-09/NBC 2010 or the CSA O86-14/ NBC 2010.[10,2,3] The software can generate wind loads following NBC Low-rise Figure I-7/8 or following NBC High-rise Figure I-15. Selecting a deflection analysis not only activates the calculation of shearwall deflections, it also activates the option of distributing forces based on stiffness derived from deflections.

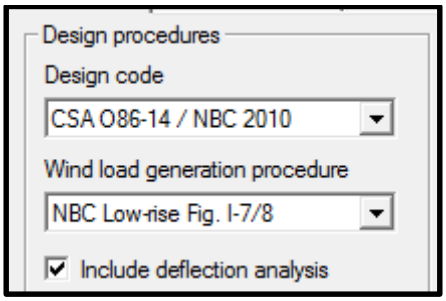



Figure 174: *Design Code and Wind Load Generation Procedure (Canadian)*

### 25.2.3 Local Building Code Capacity Modification (U.S. Only)

The user is given the option of applying local building code reductions for wind and seismic loads during the design process. The values entered here factor the published plywood sheathing shear strengths in conjunction with all other factors. The factor does not apply to gypsum or fiberboard, for which the shear strength depends entirely on the strength of the material and not the nailing. The original default value is 1.0, and values are limited to between 0 and 1, but the default setting to conform with local area conditions. This setting may also be used to simulate the overstrength factor:

The software automatically calculates rho and applies it to the load, as per ASCE 7-10 12.4.2. In certain circumstances, such as irregular buildings (weak stories 12.3.3.2), users are required to multiply the load by omega instead of the redundancy factor rho.[8] WoodWorks® does not include an option of multiplying by omega, but does have a work around solution by reducing the capacity rather than increasing the load. ASCE 7-10 12.4.3.3 permits the capacity to be increased by 1.2 when the overstrength factor omega is used, and this too can be incorporated in WoodWorks® based on the following capacity vs. load equation:[8]

[Capacity x 1.2] = [Load x omega / rho]

This shows we are multiplying the capacity by 1.2, and dividing rho out of the load that WoodWorks® puts in, and multiplying omega into the load. But since WoodWorks® can only handle the modifications to the capacity, we will reduce the capacity (rho / omega) by the same amount we would have increased the load (omega / rho).

Example: In the settings/design menu "local building code capacity modification", seismic cell compute the following, based on omega = 3.0, rho = 1.3, and allowed capacity increase of 1.2:

1.2*(1.3/3.0) = 0.52 (see below)

*Note of caution: This method is meant to give a relative comparison of the load vs. capacity. The ratio of load vs. capacity is correct based on this work-around method, however, the value of the load and capacity are both understated by the same amount.*

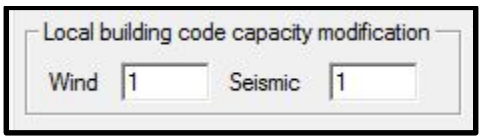



Figure 175: *Local Building Code Modification (U.S.)*

### 25.2.4 Disregard shearwall height-to-width limitations

A checkbox has been added to allow you to *disregard the height to width limitations* for shearwalls entirely. This is ordinarily used to allow for proprietary non-wood shear resisting elements. Narrow wood and fiberboard wall segments are generally not considered to contribute to lateral load resistance. By default, *Shearwalls* implements the height-to-width ratio limitations listed in model building codes for various wall materials subject to wind or seismic loads. This checkbox allows the user to over-ride the program height-to-width limitations to implement proprietary non-wood shear-resisting elements. Normally, the program imposes a maximum height-to-width ratio according to the material and design type using the limits in SDPWS Table 4.3.4 and in CSA O86-14 11.3.3.2, which defines the narrowest full-height sheathing shearwall segment that can contribute to shear resistance.[9,2] When this setting is checked, the limitations on the height-to-width ratio is disregarded.

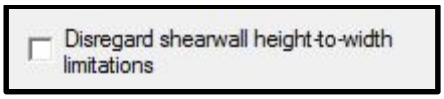



Figure 176: *Disregard shearwall height-to-width limitations*

### 25.2.5 Worst-case rigid vs. flexible diaphragms (envelope design)

Many designers prefer to consider diaphragms to be semi-rigid, so the *Shearwalls* allows for the design of the worst case of rigid or flexible diaphragm distribution, to cover the whole envelope of possible diaphragm rigidities. *Shearwalls* now allows for that approach. If the *Worst case rigid vs. flexible setting* is not selected, program determines the wall parameters needed to resist flexible diaphragm distribution forces, and then does so for rigid forces separately. As a result, the program can create separate wall groups for the same physical wall, one for rigid diaphragm design and one for flexible design.

If the setting is selected, the program first designs a wall for flexible diaphragm forces. When designing for rigid forces, if they are lower than flexible force, the program simply uses the wall designed with the flexible force. If they are higher than the flexible force, it replaces the wall designed for flexible forces with the one designed for rigid forces. For deflection-based intra-shearline distribution, the wall is then processed again

for flexible forces on the next iteration of the design procedure, as the distribution of forces within the shearline may change slightly due to the new wall stiffness. The wall groups are indicated by numbers in the *Shear Design* table, which are defined in the *Sheathing and Framing Materials by Wall Group* tables. If you have selected the *Worst-case rigid vs. flexible diaphragms* design setting, then for a particular wall, the same number appears for rigid and flexible design. If that setting is not selected, they can be different.

*Please note that if the Worst case rigid vs. flexible setting is set, the wall materials appearing in table for rigid diaphragm design may have been designed for a higher force for flexible diaphragm design, and vice-versa. If the program designs walls that appear to be much stronger than needed, this is the most likely reason.*


Figure 177: *Worst-case rigid vs. flexible diaphragms*

The following online tutorials provide more information on Flexible & Rigid Diaphragm Distribution:

*Canadian* - *https://www.youtube.com/watch?v=VZVjN6gdgN8&feature=youtu.be*

*U.S.* - *https://www.youtube.com/watch?v=URYRFtAHHAs*

### 25.2.6 Ignore non-wood-panel contribution… (U.S. Only)

If "*when combined with structural wood panels*" is selected for wind and/or seismic design, then the resistance of two-sided shearwalls will be based only on the allowable shear of wood and fiberboard panels (gypsum and other material will be ignored). This allows higher aspect ratios to be used for wind and/or seismic design for panels where wood or fiberboard is combined with other sheathing materials.

For seismic design, it may be beneficial to *Ignore non-wood panel contribution for all walls*. This removes the penalty of the increased seismic loading that would result from the lower response modification factor, R, which must be used when non-wood materials are part of the seismic resistance system.

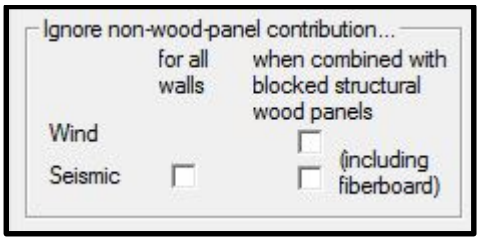



Figure 178: *Ignore non-wood-panel-contribution*

### 25.2.7 Seismic wood panels, and fiberboard (U.S. only)

If "*Allow 3.5:1 height-to-width ratios*" is selected, the program considers full height segments with a height-to-width ratio between 2 and 3.5 as shear resisting elements for seismic design. The allowable shear values are reduced as per the NDS.[1]


Figure 179: *Seismic wood panels and fiberboard (U.S.)*

### 25.2.8 Out-of-plane sheathing assumption (U.S. only)

The *Special Design Provisions* for Wind and Seismic (SDPWS) out-of-plane sheathing values are based on a 2-span configuration.[9] Other publications, such as the Wood Frame Construction Manual, lists these values based on sheathing supported over 3 spans.[11] Furthermore, note under SDPWS table 3.2.2 allows for an increase of values if the sheathing is continuous over 3-spans. Selecting the 3-span here increases the tabulated values by the intended 25%. Refer to SDPWS 2008 C3.2 for the reason for the change from 3-span assumption to two span assumption.[9] The program allows the user to select which assumption are preferred based on the characteristics of the building.

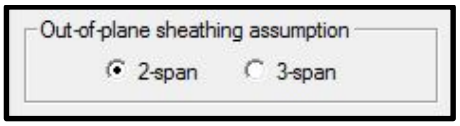



Figure 180: *Out-of-plane sheathing assumption*

### 25.2.9 Hold-down forces based on (Canadian only)

This setting allows the user to select the *Hold-down forces based on Shearwall capacity* or the *Applied loads*, by toggling between the two choices. *Note: The "*" which explains that shearwalls capacity will be used for seismic discontinuities analysis as per NBC 4.1.18.15(4) even if the default applied loads is selected.*[3]

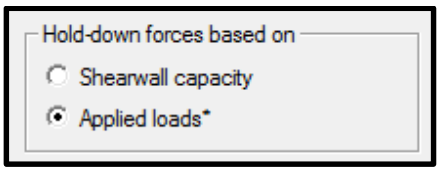



Figure 181: *Material restrictions for anchorages (Canadian)*

### 25.2.10 Shearwall Materials (Canadian only)

Users have the option of enabling design setting to have *all shearwalls on shearlines have the same material*. Leaving this box unchecked allows models to have multiple types of shearwalls along the same shearline.

Users also have the option of disabling the shear resistance contribution from gypsum for both wind and seismic forces. If no gypsum is present in the model, shearwalls warns the user if an Rd value of 2.0 or less has been unnecessarily entered, which corresponds to walls with a combination of sheathing and gypsum materials. The warning allows the user to change the value to 3.0, the value for wooden shearwalls in NBC Table 4.1.8.9.[3]

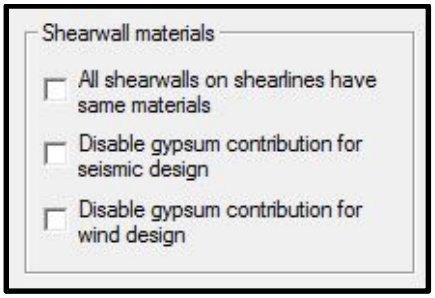



Figure 182: *Shearwall Materials (Canadian)*

### 25.2.11 Maximum shearline offset

The user has the option of defining *Maximum shearline offset* for the *Plan* and *Elevation*. A shearline consists of one or more parallel shearwalls that act as one to resist an applied shear load. Walls can be on the same shearline as long as they are separated by no more than the maximum shearwall plan offset if on the same level, and the maximum shearwall elevation offset if they are on different levels. None of the design codes implemented in the program recommend a shearline offset, as shearwalls are expected to be at the same location if they are to act as a unit. However, a default 0.5 ft (0.15 m) plan offset has been implemented to provide tolerance for automatic shearline generation.

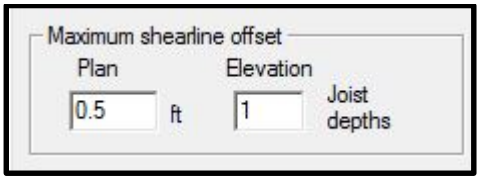



Figure 183: *Maximum Shearline Offset*

### 25.2.12 Collector Forces Based on… (U.S. only)

The user has the ability to indicate whether hold-down forces and drag strut forces are based on the shearwall capacity or applied loads. The drag strut force using applied shear is calculated by taking the absolute value of the difference between the design shear in each segment and the shear flow on the diaphragm at the top of the shearwall. Using shearwall capacity, this value is simply multiplied by the ratio of shearwall capacity of the design shear wall total design shear force. *Note that this means that the correction for perforation factor Co for the critical wall is applied to the entire shearline*. *Also note that if shear force is distributed to the segments via shearwall rigidity (design setting) the critical segment ratio is also applied to the entire line.* If some shearlines do require shearwall capacity design, and some do not, it will be necessary to run the design twice, with different *Drag Strut Force* settings.

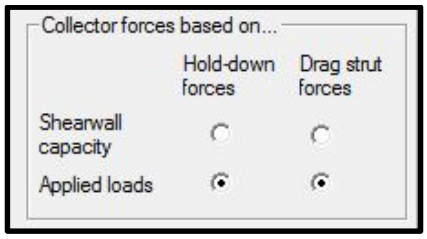



Figure 184: *Collector Forces based on… (U.S.)*

### 25.2.13 Rigid Diaphragm Analysis (Canada) / Shearwall Rigidity per unit length (U.S.)

This design setting allows the user to indicate to the program how it is to determine the relative rigidity of the shearwalls (per unit length). As well as being used in the rigid diaphragm shearline force system, subject to the checkbox below these choices, it is also used in the transfer of shearline force to the walls within the shearline. When checked, the program distributes forces to each wall segment proportionally to the rigidity of the segment, calculated via one of the shearwall rigidity methods below.

1. Use shearwall capacity to approximate rigidity

With this method, the program uses the shearwall rigid diaphragm analysis found from the flexible design method to approximate the relative rigidities. This method is considered by SEAOC and other design code authorities to be an acceptable alternative to more rigorous rigidity analysis that takes into account framing bending, sheathing shear stiffness, and nail and hold-down slippage. Using this method, the rigidities appear as force units (kN, lbs, or kips) in the detailed "log file" output, that is, the rigidity expressed as force/length multiplied by the shearwall length. If *Use shearwall capacity to approximate rigidity* is checked, then force per unit foot will also be equal because of the similar materials requirement, unless there are perforated wall factors or height-to-width factors for short segments along the line that would cause variations in shearwall capacity along the line.

2. Assume shearwalls have equal rigidity

With this method, the program assumes that all shearwalls have the same rigidity per unit length, so that the rigidity of any shearline is proportional to the length of wall on the shearline. Using this method, the rigidities appear as length units (ft or m) in detailed "log file" output. *Note that if Equal rigidities are checked, then it has the same effect as not checking this box, as force per unit foot will be constant along the line in either case.*

3. Manual input of relative rigidities

With this method, the user enters any number using any criterion for relative shearwall rigidity, keeping in mind it is only the ratio of these numbers for the various shearlines that matters. The default that first appears is 1.0, corresponding to the ***shearwalls have equal rigidity*** method. If the user changes from the capacity method, the number that appears is the capacity in lbs/in. If the user changes from the deflection method, the number that appears is the stiffness in lbs/in. Using this method, the rigidities appear as dimensionless numbers in detailed ***log file*** output.

4. Use shearwall deflection to calculate rigidity (Recommended Method)

Using this method, the program uses the sum of segment stiffnesses on the line, or Σ F/D, to calculate rigidity, where F is the force on the segment and D is the deflection calculated for that segment. When deflections are equalized on the line, this is just F/D, where F is the shearline force and D is the common deflection.

*It is possible to select either the shearwall capacity or deflection option, then adjust rigidities using the manual method. This method can help simulate proprietary shearwall sections. The method of distributing loads within the shearline will be the same as the method chosen to distribute loads to the shearlines.*

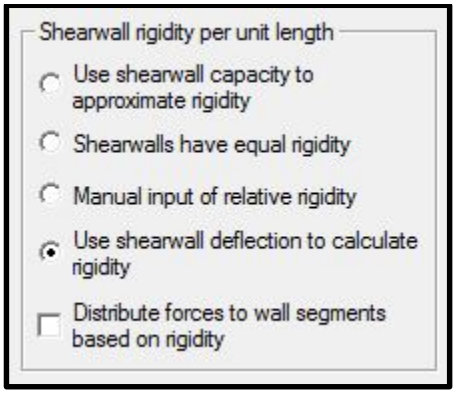



Figure 185: *Shearwall rigidity per unit length*

***Canadian Version:***

*Note that the CSA O86-14 11.4.5.2 mandates that shearwall capacity be used to determine the distribution of forces along a shearline, however, the clause has been interpreted as being the minimum acceptable method, and that an analysis based on stiffness is superior and therefore also acceptable.[2] This option is available if Use shearwall deflection to calculate rigidity is chosen as the method of shearwall rigidity calculation.*

### 25.2.14 Height Restrictions for Wind Loads (Canadian only)

This setting allow you to decide what to use as the "height" of the structure when applying the restrictions regarding the applicability of Figures I-7 for low rise structures and figure I-15 for all-heights. The restrictions are

- High-rise (I-15) buildings muse have a 4-1 height-to-width (H/W) ratio or less (NBC 4.1.7.2). If not, such buildings are disallowed.[3]
- Low-rise buildings (I-7) should have H/W ratio of 0.5 or less, but 1.0 or less is permissible (Commentary I-26). The former case is met with a warning; the latter is disallowed. (Commentary I-26)
- Low-rise buildings must be less than 20 metres in height (Commentary I-26). If not, the low rise choice is are disallowed

The choices for the height to use for these restrictions are, the *Mean* roof height (half way from eave to ridge), the *Ridge* height, and the *Eave* height top of uppermost storey below ridge.

A choice has been added in this regard because the all-heights procedure is intended for flat-roofed structures, but because no other method is available for multi-block structures or narrow low structures, it is used by *Shearwalls* in cases that there is a peaked roof. This setting allows you to apply your discretion as to what height should be used for your structure.

For low-rise structures, it is not clear that the reference height for calculating $C_e$ (Commentary I-7) is an appropriate measure of the height of the structure for this purpose, so the choice applies to restrictions on these structures as well.

*Note that this choice affects only the operation of the program in general and is not saved with the particular project file it has been applied to.*

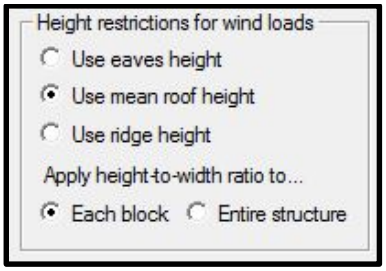



Figure 186: *Height Restrictions for Wind Loads (Canadian)*

### 25.2.15 Service Conditions (U.S.)

Users have the option of specifying service condition factors for temperature and moisture. If Deflection Analysis is not activated then the choices are < 19% or >= 19% moisture content for each of in-service and at time of fabrication. If *Deflection Analysis* is activated, then the program allows the user to enter the precise moisture content percentage.

The program applies the wet service factors from the *Withdrawal* loads section of NDS Table 10.3.3 to the nail withdrawal calculations for C&C design.[1] These factors are shown in the C&C design results table. For deflection analysis, this is used to calculate the shrinkage component of hold-down displacement, via the length subject to shrinkage, as input for each level in *Structure* input view multiplied by .002 x difference in fabrication and in-service moisture content percentage as input in the design settings.

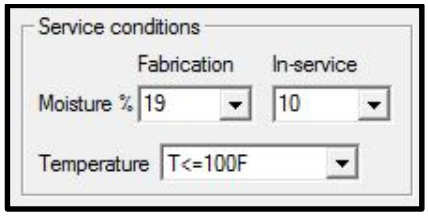



Figure 187: *Service Conditions (U.S.)*

### 25.2.16 Moisture Conditions (Canadian)

If *Deflection analysis* is not activated then the choices are *Wet* or *Dry* service conditions, and *Seasoned* ((<15%) or *Unseasoned* (>15%)) fabrication condition. If *Deflection Analysis* is activated, then the program allows the user to enter the precise moisture content percentage. 15% is also used to define wet in-service conditions. The program applies the service condition $K_{SF}$ factors service factors from CSA O86-14 Table 12.2.1.5.[2] The lateral load factors are used in the equations in CSA O86-14 11.5.1 for shear design, the withdrawal load factors are applied to nail withdrawal for C&C design.[2]

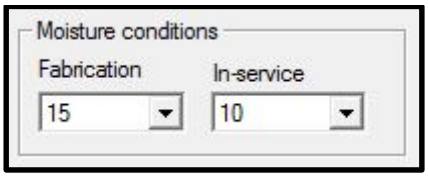



Figure 188: *Moisture Conditions (Canadian)*

### 25.2.17 Apply height-to-width ratio to… (U.S. only)

This checkbox allows the user to choose to not *Apply the height-to-width ratio* limitations, to each block or the entire structure. This allows modeling of proprietary shear elements.

The user is given the option of resetting all of the settings back to default options.

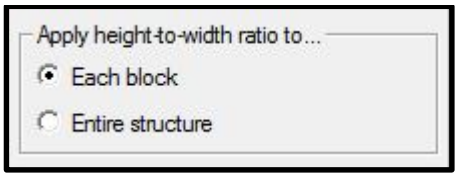



Figure 189: *Apply height-to-width ratio to… (U.S.)*

### 25.2.18 Drag Strut Forces Based on (Canadian only)

This input allows the user to choose whether *Drag strut forces* are based on *Shearwall capacity* or the *Applied loads*, similar to hold-down forces. *Note: The "*" which explains that shearwalls capacity will be used for seismic discontinuities analysis as per NBC 4.1.18.15(4) even if the default applied loads is selected.*[3]

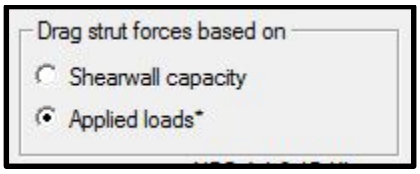



Figure 190: *Drag Strut Forces Based on (Canadian)*

## 25.3 Default Values Tab

Figure 191 shows the *Shearwalls* Design tab for the Canadian edition of the software and Figure 192 shows the *Shearwalls* Design tab for the U.S. edition of the software.

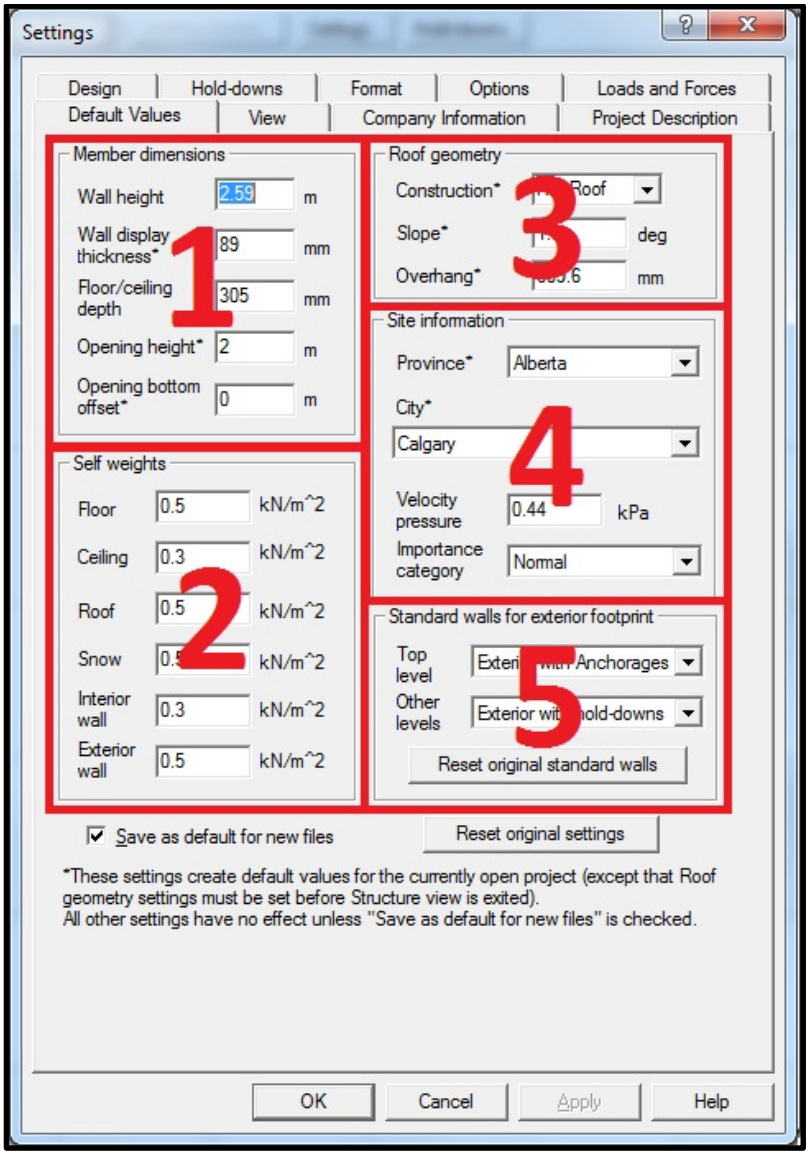



Figure 191: *Shearwalls Settings – Default Values Tab (Canadian)*

The following Table Summarizes where information on each Part of Figure 191 can be found. The section numbers have been hyperlinked for convenience:

| | Section |
|---|---|
| Part | Figure 191 (Canadian) |
| 1 | 31.3.2 |
| 2 | 31.3.3 |
| 3 | 31.3.4 |
| 4 | 31.3.5 |
| 5 | 31.3.6 |

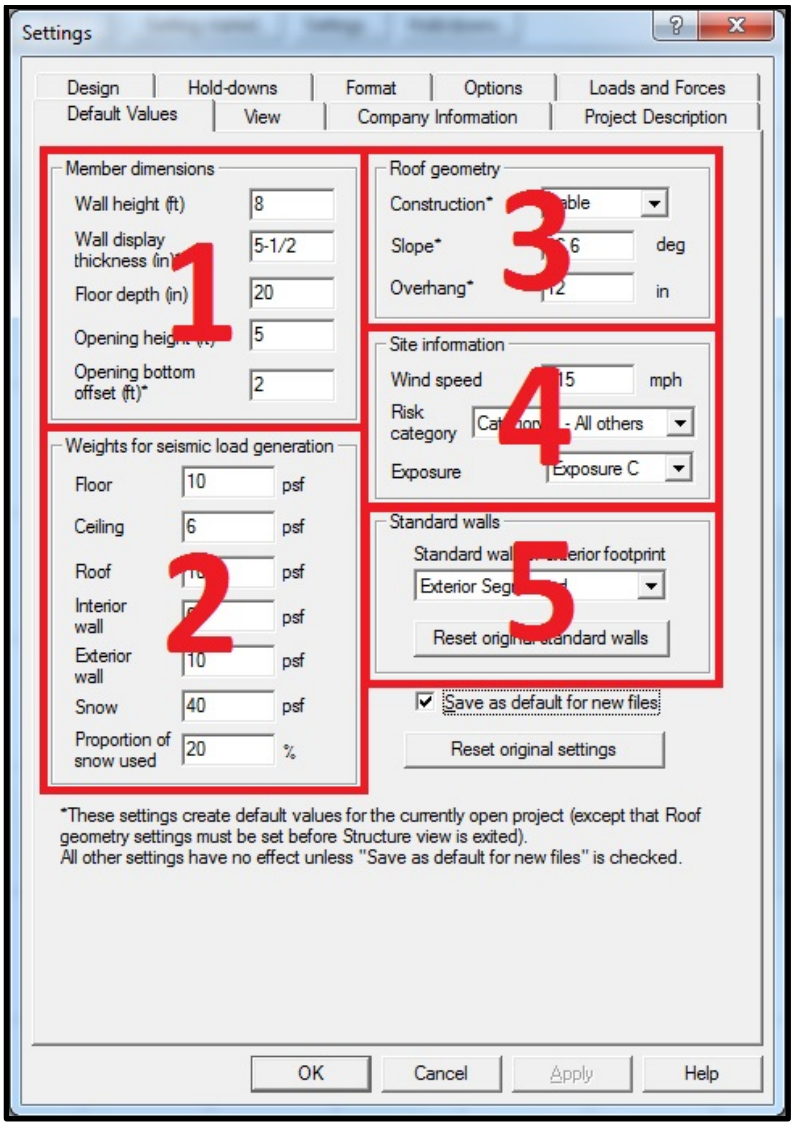



Figure 192: *Shearwalls Settings – Default Values Tab (U.S.)*

The following Table Summarizes where information on each Part of Figure 192 can be found. The section numbers have been hyperlinked for convenience:

| Part | Section<br>Figure 192 (U.S.) |
|---|---|
| 1 | 31.3.2 |
| 2 | 31.3.3 |
| 3 | 31.3.4 |
| 4 | 31.3.5 |
| 5 | 31.3.6 |

Controls the default materials, conditions, and dimensions used to generate the building and the loading model. Some of the settings affect only the values that appear in newly created files. Others also affect the operation of a project while it is running, and in the case of roof geometry settings, if the change is made before any blocks are created. These settings are indicated by an asterisk (*) in both the input form, and also in the descriptions below.

### 25.3.1 Reset Original Settings

If the user clicks this button, the *Standard walls* that came with the installation of *WoodWorks® shearwalls* will be restored. Be very careful using this button, as any other standard walls the user has created and saved will be unavailable to new projects.

### 25.3.2 Member Dimensions

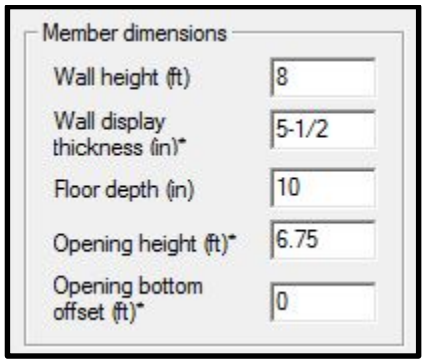



Figure 193: *Shearwalls Member Dimensions*

#### 25.3.2.1 Wall Height

The default height of walls when creating building levels in the *Structure Input* form. Each newly-created level will have this height (not the height of the first level entered).

#### 25.3.2.2 Wall Display Thickness

This does not affect the design in any way; it is here where the user is able to adjust the thickness of the walls that are displayed in *Plan View*. Using the actual wall thickness (4"–6") usually results in a reasonably proportioned diagram.

#### 25.3.2.3 Floor/Ceiling Depth

The default size of the floor joists (and flooring thickness) when creating building levels in the *Structure Input* form. Joist depth affects the elevation offset, which is entered in terms of joist depths, and the overall height of the building for wind and seismic load generation.

#### 25.3.2.4 Opening Height

The height of the very first opening in the project. Created by dragging the mouse in *Plan View*. After that, openings will have the same height as the last one created. The opening heights can then be changed in the *Opening Input Form*.

#### 25.3.2.5 Opening Bottom Offset

The distance from the bottom of the wall to the bottom of the very first opening in the project created by dragging the mouse in *Plan View*. After that, new openings will have the same dimensions as the last one

created. The opening offsets can then be changed in the *Opening Input Form*. The bottom offset does not affect design.

### 25.3.3 Weights for seismic load generation

The values of self-weight listed in this box appear in the *Generate Loads* form. They are used to generate building masses that are used to create seismic loads. Values may be specified for *floor*, *ceiling*, *roof*, *snow*, *interior wall* and *exterior wall* components.

The default for roof self-weight applies to either the roof mass as if it was a load applied to the projected area of the roof or the actual weights of the sloped roof panel according to which of these are set in *the Load Generation* settings.

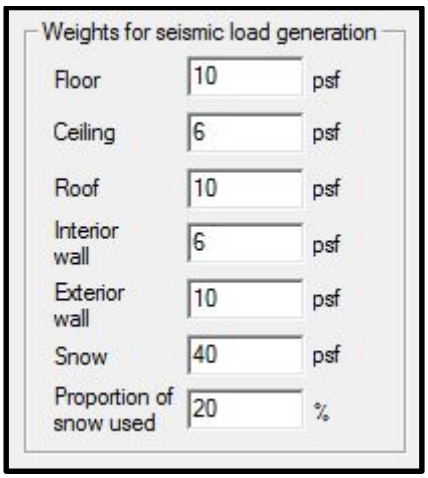



Figure 194: *Weights for seismic Load generation*

#### 25.3.3.1 Snow contribution

***US Version:***

For the U.S. version the user is able to input the total snow mass and the percentage to be used in seismic mass generation. The factory default is 40 lbs, and 20% used according to ASCE 7-10 12.7.2-4.[8]

***Canadian Version:***

For the Canadian version, 25% of the entered value for snow loads contributes to the building mass.

### 25.3.4 Roof Geometry

The default settings of the roof input form may be modified in this box. Default settings for roof construction (flat, hip or gable), roof slope, and length of overhang, apply to each new roof block created in *Plan View*.

*Note that these settings should be set early on in the process, preferably before a file is created. Once a block is drawn, its roof settings are fixed at this point, even though the roof is not visible until user begins the roof action.*

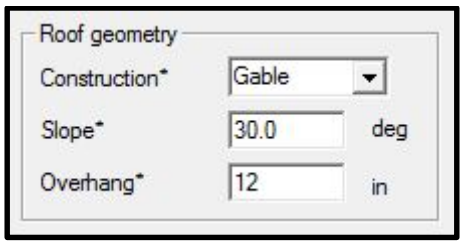



Figure 195: *Shearwalls Roof Geometry*

### 25.3.5 Site Information

The user can enter the default load information that appears in the *Site Information* box for new files. Only those values that can be expected to remain more or less the same for all the structures the user might design are included in this vicinity.

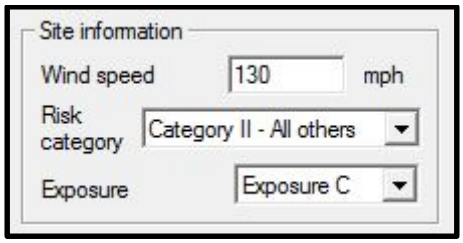



Figure 196: *Shearwalls Site Information (U.S.)*

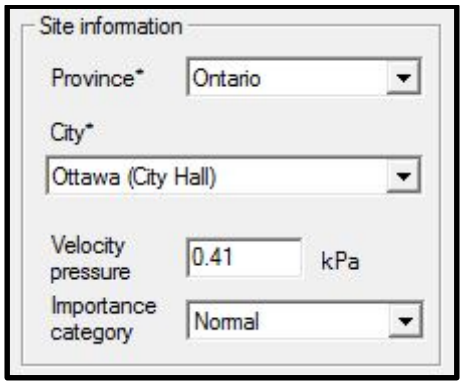



Figure 197: *Shearwalls Site Information (Canadian)*

#### 25.3.5.1 Occupancy/Importance

For all versions of the software, the user can enter the *Occupancy* or *Importance* category that is used for the *Importance Factor* for both wind and seismic design.

#### 25.3.5.2 Wind Velocity

***US Version:***

For the U.S. version, the user is able to enter the design wind speed for the location that the buildings are most likely to be situated. The user is also given the option of selecting the appropriate default exposure.

***Canadian Version:***

For the Canadian version, the user can select the closest city to where the building is situated from a list of cities that is ordered roughly from East to West across Canada. When selected, the design velocity pressures from the NBC Appendix C appear in the control below, and these can be modified to suit the exact location of a structure.[3]

### 25.3.6 Standard Walls

#### 25.3.6.1 Default Standard Wall

The materials from this *Standard walls* will be used when the first walls in the program are created. These walls are placed around the perimeter of the building outline made in the *Block Outline* action. Thereafter, the standard wall to be used to create new walls will be the one that has most recently been selected in the *Wall and Shearline Input Form*.

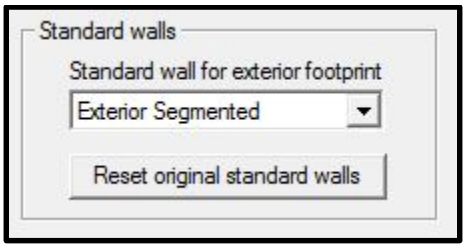



Figure 198: *Shearwalls Standard Walls (U.S.)*

#### 25.3.6.2 Upper and Lower Level *(Canada only)*

The Canadian version has a different default *Standard walls* for the uppermost level than for the rest of the levels. This is because anchorages rather than hold-downs are likely to be placed on the top floor only.

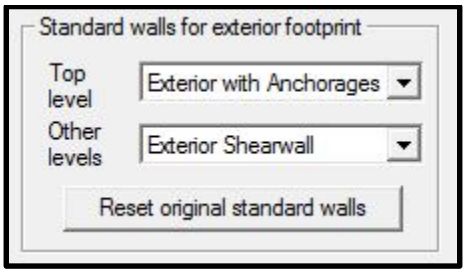



Figure 199: *Shearwalls Standard Walls (Canadian)*

### 25.3.7 Reset Original Settings

If the user clicks this button, the *Standard walls* that came with the installation of *WoodWorks® shearwalls* will be restored. Be very careful using this button, as any other standard walls the user has created and saved will be unavailable for new projects.


Figure 200: *Shearwalls Reset Original Settings*

## 25.4 Hold-downs Settings Tab

The *Hold-down offset* will be used in the calculation of hold-down forces if *Subtract offset from segment length...*" is selected. Left unchecked, the moment arm will be based on the total distance from end to end of the segment, resulting in lower forces in tension and compression at the ends of the panel.

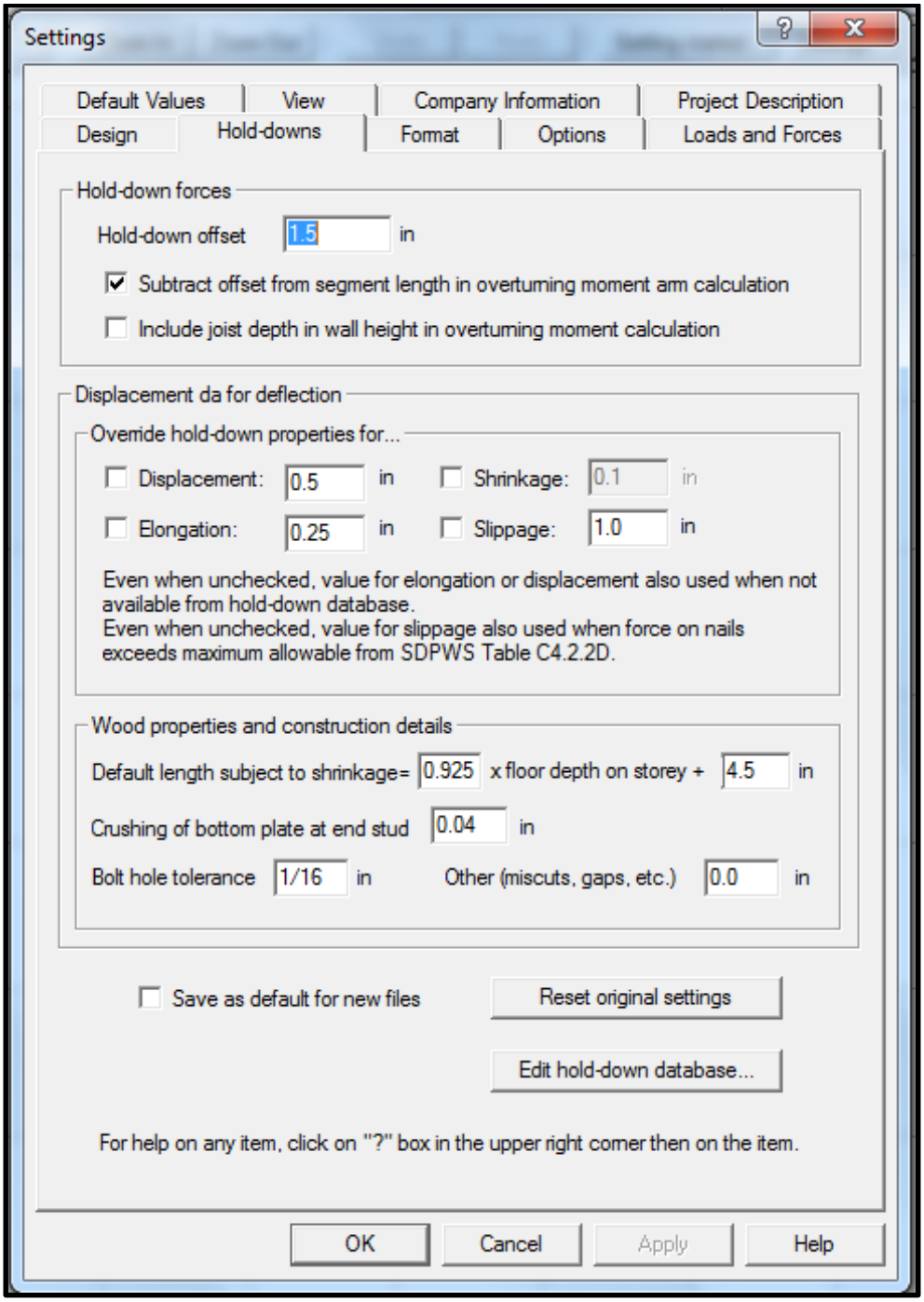



Figure 201: *Shearwalls Settings – Hold-downs Tab*

In the U.S., the joist depth is not included in the overturning calculation, as shown in the SDPWS. However, some engineers prefer to take the more conservative approach of including this extra height, and selecting "*Include joist depth in wall height...*" will give this extra flexibility. The Canadian version automatically includes the floor depth as per the CSA O86-14.[2]

### 25.4.1 Hold-down forces

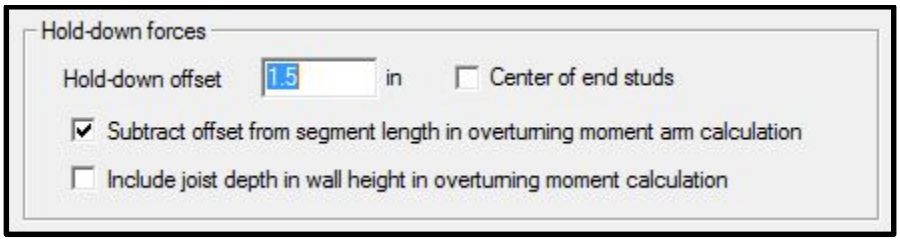



Figure 202: *Shearwalls Hold-down Forces*

#### 25.4.1.1 Hold-down offset

The user is given the option of specifying a hold-down offset. If a value is entered that is greater than or equal to ½ a shearwall segment length, the program reverts to the factory default value of 1.5" for that segment. It issues no warning in this case, it is evident only by the placement of the hold-down in elevation view and its position as listed in the hold-down *Design* table.

#### 25.4.1.2 Subtract offset from segment length in overturning moment arm calculation

A checkbox indicates whether the program subtracts the hold-down offset from the wall length when calculating the overturning moment arm. AWC's SDPWS 4.3.6.1.1 (Eqn. 4.3-4) indicates that this distance is not subtracted, however engineering mechanics, particularly when there is only one shearwall on a line, require that it be subtracted.[9]

#### 25.4.1.3 Include joist depth in wall height in overturning moment calculation

A checkbox indicates whether the program includes the floor depth above the wall in the wall height when calculating the overturning moment arm. AWC's SDPWS 4.3.6. Eqn. 4.3-4.5 indicates that this distance is not included, however this creates gaps in the static analysis of multi-level structures.[9]

### 25.4.2 Displacement da for Deflection – Override Hold-down Properties

The inputs in this data group allow the user to replace the vertical hold-down displacement components with constant values for all hold-downs in the program. They also allow the user to specify values for these components if they cannot be calculated or are not available from the hold-down database for a particular hold-down. A warning appears in the output if this situation occurs.

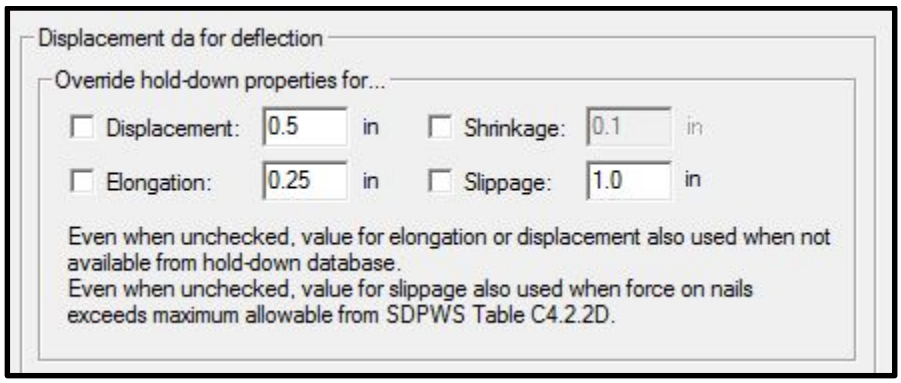



Figure 203: *Shearwalls Override hold-down properties for…*

#### 25.4.2.1 Displacement

If box is checked, the program uses the input value as the elongation for all hold-downs in the structure that have combined elongation/slippage, overriding the hold-down database value. If box is not checked, it uses the override value only when a value is not available from the database for the stud size that the hold-down is attached to.

#### 25.4.2.2 Shrinkage

If box is checked, the program uses the input value as the wood shrinkage value for all hold-downs in the structure, overriding the value calculated using moisture content and length subject to shrinkage on each floor.

#### 25.4.2.3 Elongation (U.S. Only)

If box is checked, the program uses the input value as the elongation for all hold-downs in the structure that have separate elongation/slippage, overriding the hold-down database value. If box is not checked, it uses the override value only when a value is not available from the database for the stud size that the hold-down is attached to.

#### 25.4.2.4 Slippage (U.S. Only)

If box is checked, the program uses the input value as the nail slippage for all nailed hold-downs in the structure that have separate elongation/slippage, overriding the hold-down database value. If box is not checked, it uses the input value only when the force Vn on each nail is greater than the maximum allowed in SDPWS Table C4.2.2D.[9]

### 25.4.3 Wood Properties and Construction Detail Settings

Data for hold-down displacement calculations that cannot be entered independently at each hold-down location is entered here.

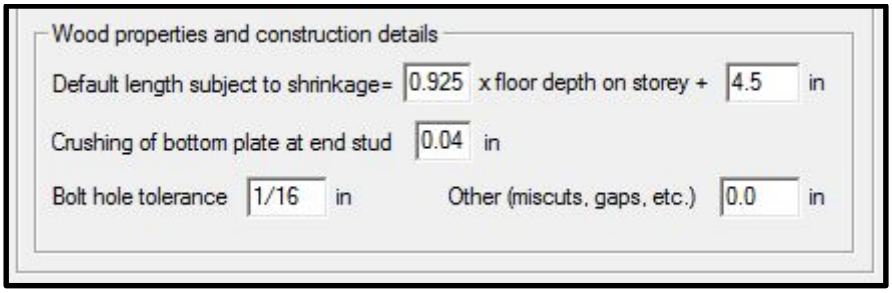



Figure 204: *Shearwalls Hold-downs Wood Properties and Construction Details*

#### 25.4.3.1 Default Length Subject to Shrinkage

Used to enter the proportion of the floor depth as input in the *Structure input* view, plus the depth of other wood members such as wall top and bottom plates that is subject to shrinkage. These values can be adjusted for individual floors in *Structure Input* view, they are of primary use in creating defaults for new files for these values.

#### 25.4.3.2 Crushing of Bottom Plate at End Stud

The deformation of the bottom wall plate beneath the end chord studs at the compression end of the shearwall. The "factory" default is 0.04 corresponding to lumber loaded to capacity for perpendicular compression according to NDS 4.2.6. A value of 0.02 corresponds to lumber loaded to 73% capacity.[1]

#### 25.4.3.3 Bolt hole tolerance

The difference between drilled hole diameter in the studs and the diameter of the horizontal bolt shank. For assembly displacements that include slippage (see 1b, above), any value greater than 1/16" is added to the published displacement, which includes the effect of standard size bolt holes. For separate slippage and elongation, the entire value is added to the calculated slippage.

#### 25.4.3.4 Other (miscuts, gaps, etc.)

Additional sources of vertical shearwall displacement are input here at the discretion of the designer. This could include allowance for studs that are cut too short or without square-cut ends

### 25.4.4 Reset Original Settings

If this button is clicked, the S*tandard walls* that came with the installation of *WoodWorks® Shearwalls* will be restored.

### 25.4.5 Edit hold-down database…

This button is a shortcut to the “Hold-downs Database” Window described in section 37.2 of the user guide.

## 25.5 View Tab

This tab allows the user to specify different viewing options such as the *size of the viewing area*, *the snap increment*, the *percentage zoom*, and *member names*.

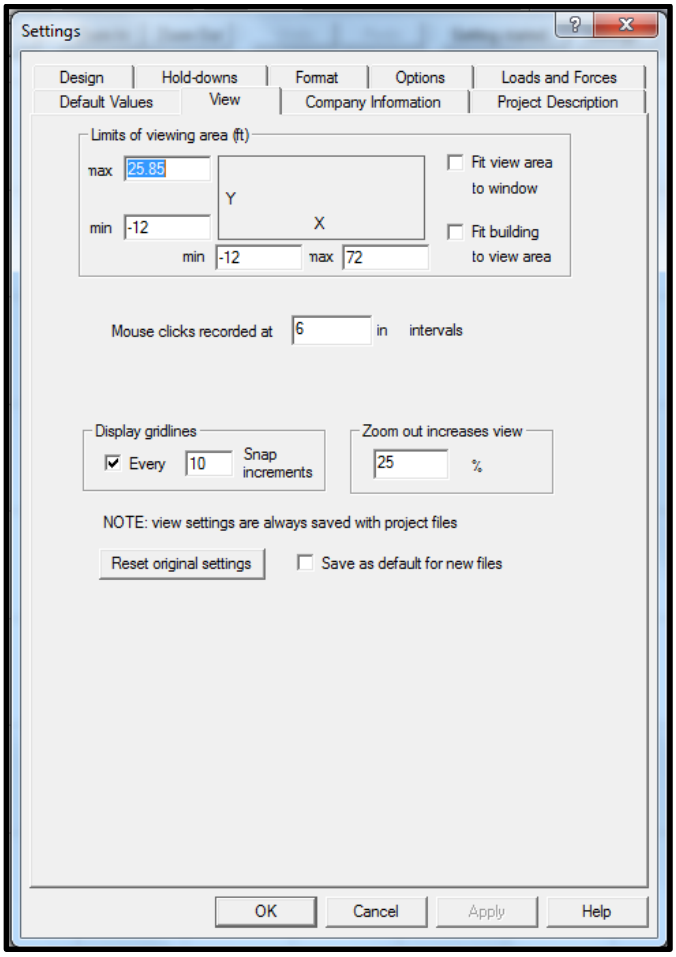



Figure 205: *Shearwalls Settings – View Tab*

### 25.5.1 Limits of Viewing Area (ft)

These fields specify the maximum viewing area in plan for the North-South and East-West directions.

### 25.5.2 Mouse clicks recorded at…

This specifies the smallest increment a newly created gridpoint will move or "snap" to.

### 25.5.3 Display Gridlines

These options allow the user to include additional information on the diagrams for viewing or printing. This includes the snapped coordinate, the member names, and the group names. The user can also specify whether snapped or actual mouse co-ordinates appear in the status bar.

### 25.5.4 Zoom out increases view

When zooming in or out, the program increases or decreases the viewing area by a certain percentage when the buttons are pressed, while maintaining the same west and south view limits. The percentage that the

view is zoomed each time a button is clicked is specified here, and the user can choose a zooming increment anywhere from 1% to 100%.

### 25.5.5 Save As Default for new files

Choose *Save new settings* under the *Settings* menu. This will save any new changes that have been done in the Settings menu. These new settings will be the default for any new files that the user creates. This custom definition will again be in effect during the next *Shearwall* session. However, a previously saved *Shearwall* file with different definitions will over-ride these settings. Clicking *File* and then *new* will restore the users definitions at any time.

### 25.5.6 Restore Factory Settings

This option restores the original settings that were in effect when *Shearwall* was first installed. They will only be in effect for the current session unless the user clicks on *Save New Settings*.

## 25.6 Format Tab

Choose the *Format* tab from the *Settings Dialog*. This tab allows the user to enter the unit system to be used and format the font size for printing and viewing.

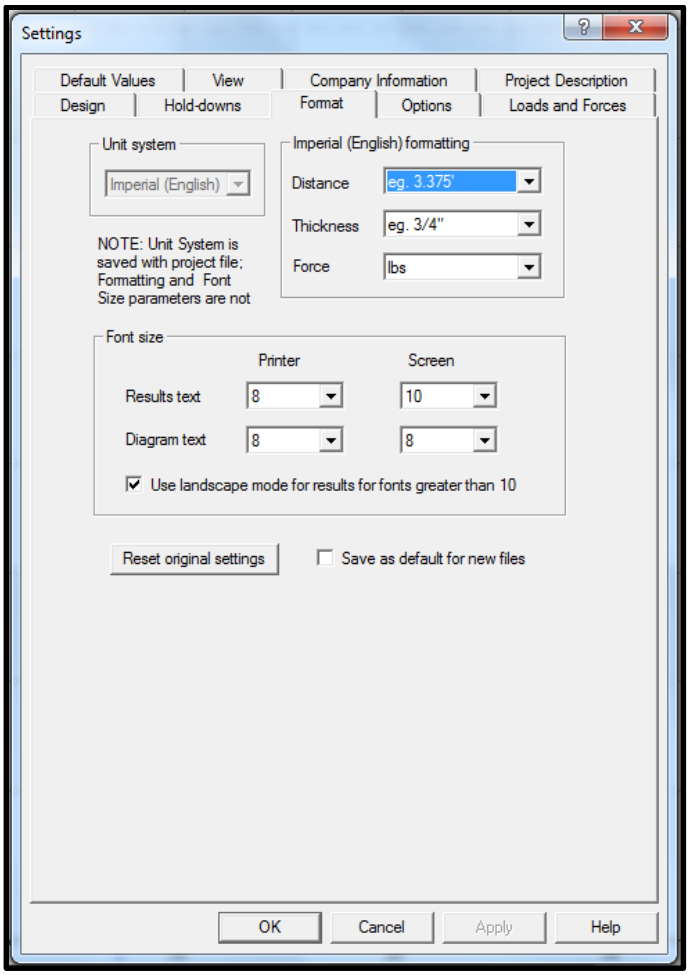



Figure 206: *Shearwall Settings – Format Tab*

### 25.6.1 Unit System

This option allows the user to select whether *Shearwalls* will operate in Imperial (English) or metric units.

### 25.6.2 Imperial (English) Formatting

Choose Imperial (English) units to be displayed in either decimal feet, in feet with decimal inches, or in feet with inch fractions. This can affect either distances or member sizes.

*Force* is an option that permits the user to select the Imperial (or English) units for *Point Loads* as either *lbs* (pounds) or *kips*. This only applies to the input fields for loads – the output unit for analysis and design results are always kips.

### 25.6.3 Font Size

This option allows the user to set the font size to be used for results and diagram text for either viewing or printing.

## 25.7 Company Information Tab

Choose the *Company Information* tab from the Settings Dialog. This tab allows the user to enter appropriate company contact.

The user only needs to enter company information once, and can change it at any time by following the procedure described above.

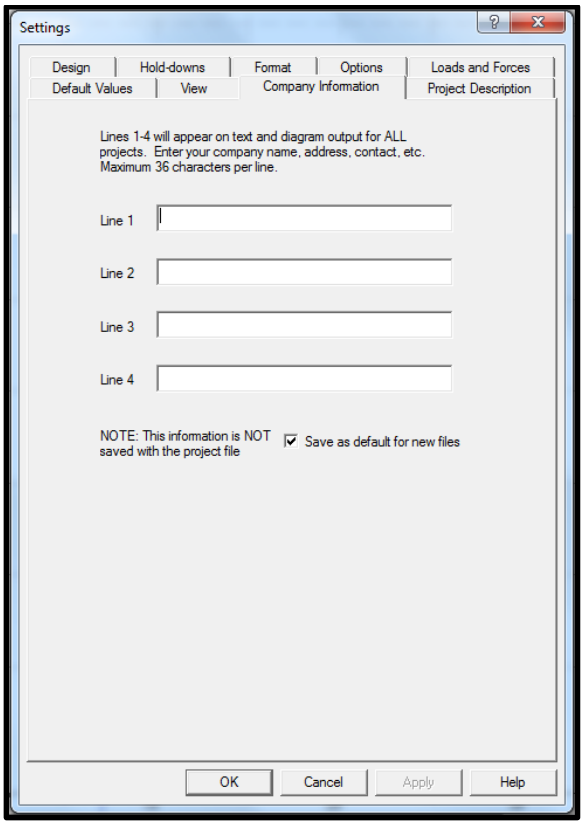



Figure 207: *Shearwalls Settings – Company information*

## 25.8 Options Tab

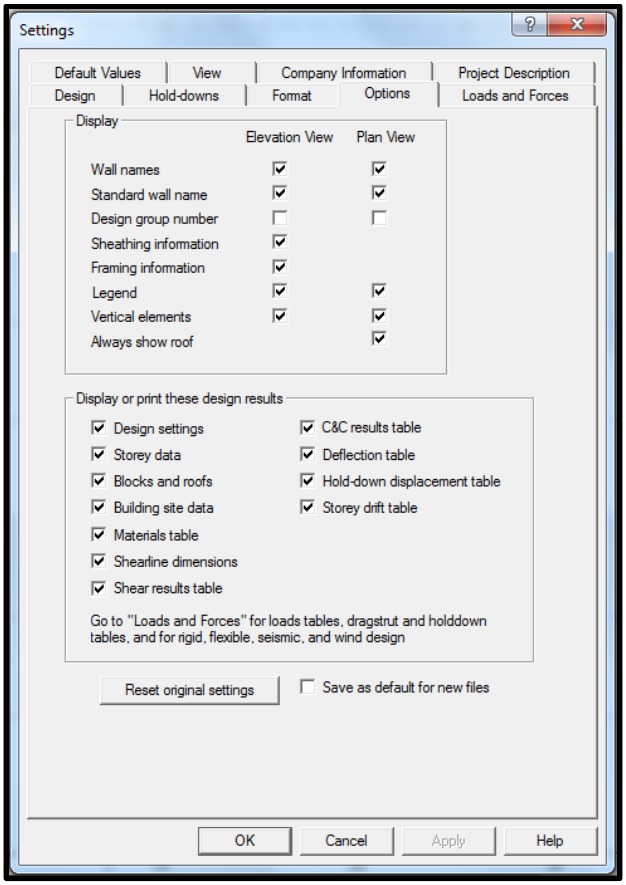



Figure 208: *Shearwalls Settings – Options Tab*

This tab allows the user to decide which preferences regarding what information, apart from loads and forces, will be displayed in *Plan View*, *Elevation view*, and the text output reports. These are not saved with the project file, so they will remain the same when the user opens an existing project.

### 25.8.1 Display

Contains a list of items that can be turned on and off in order to reduce clutter on the users diagrams or to focus on certain information. Applies to both screen display and printed output. There are separate lists for *Plan View* and *Elevation View*.

### 25.8.2 Display or Print these Design Results...

Allows the user to tailor the tables that are printed out in the design results. Can be used to reduce the output to a single page for a subset of the design results.

*The user must go to Loads and Forces Settings to turn on and off the loads and forces that appear on the screen. In the Loads and Forces Settings it is also possible to adjust settings for loads tables, drag strut and hold-down tables, and for rigid, flexible, seismic, and wind design.*

## 25.9 Project Description Tab

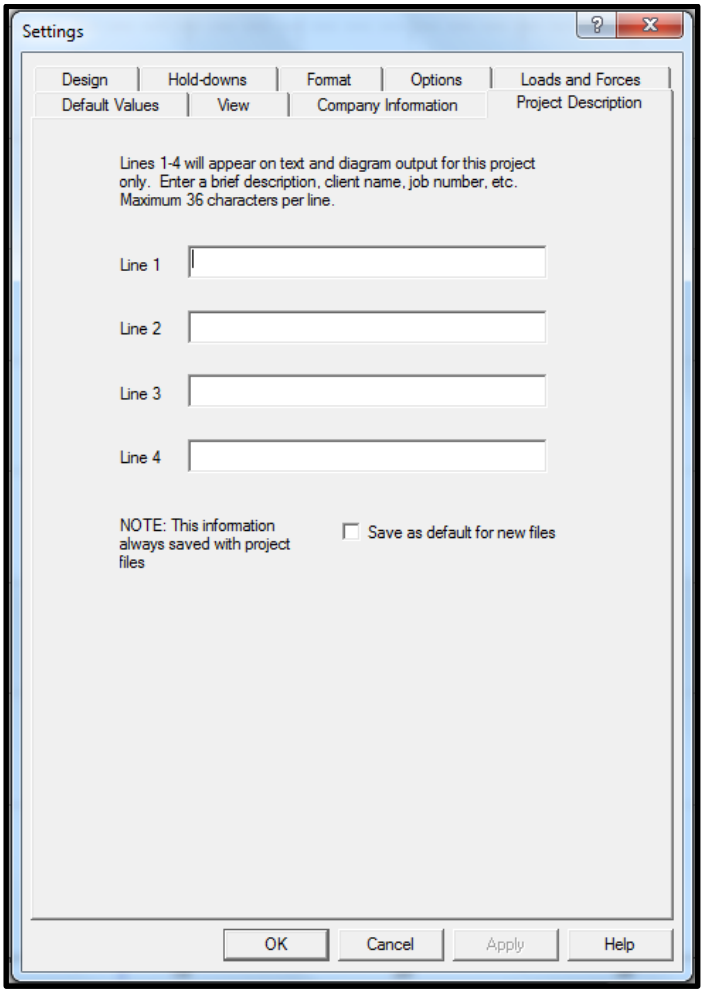



Figure 209: *Shearwalls Settings – Project Description Tab*

Choose the *Project Description* tab from the *Settings Dialog*. This tab allows the user to enter project information, including the *project name*, *location*, *client and job number*. Every time the user starts a new project, the user should enter new project information.

## 25.10 Loads and Forces Tab

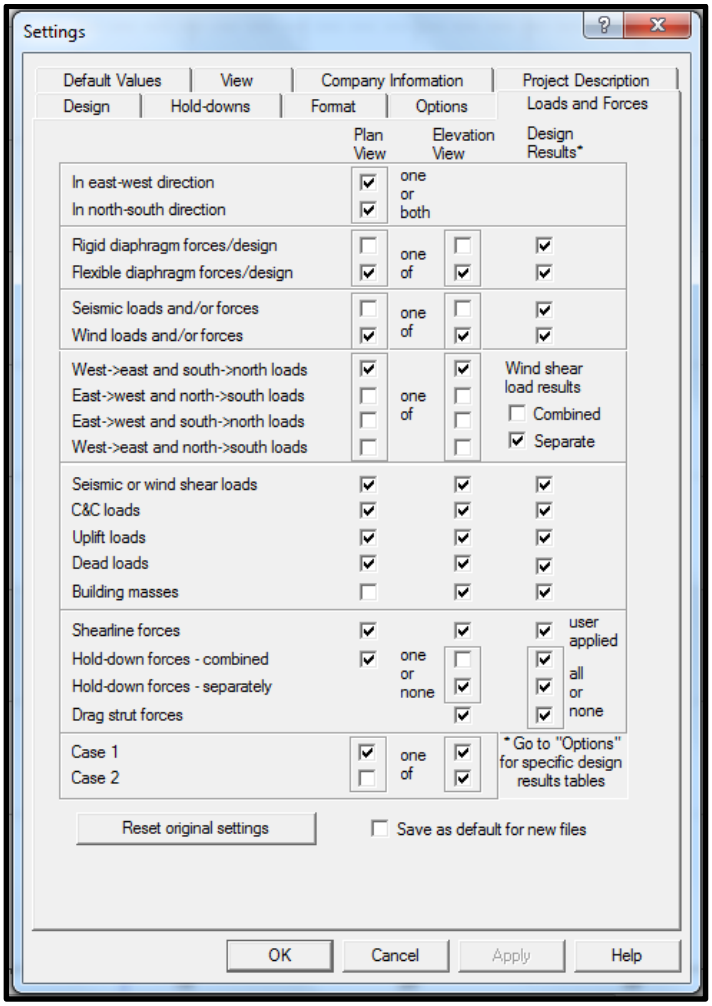



Figure 210: *Shearwalls Settings – Loads and Forces Tab*

This group of check boxes allows the user to turn on and off the loads and forces that appear on the screen, as to reduce clutter and to isolate the analysis results as desired. Some of the settings are exclusive – setting one turns the other on or off. These dependencies are indicated by borders around the interdependent settings with explanatory phrases like "All or none" and "one or both".

For each setting, the user can independently control what appears in *Plan View*, *Elevation View*, and the *Design Results* by means of separate lists of checkboxes.

### 25.10.1 Orientation (Force Direction)

In *Plan View*, choose whether to view loads and forces that ultimately bear on north-south walls or east-west walls, or both. Shear loads bearing on the east building face will be grouped with vertical loads and C&C loads on the south and north faces, because all of these forces go into designing east-west walls.

In the *Show* menu, these selections appear in a submenu under the *Force Direction* item.

### 25.10.2 Distribution Method

In *Plan View* and *Elevation View*, view forces derived from the rigid method or flexible method, but not both. The user can view either one or both in the *Design Results*.

In the *Show* menu, these choices are found in a sub-menu under the main menu item *Forces*.

### 25.10.3 Design Case

The user can choose to view loads (and all selected forces derived from them) for seismic or wind loading, but not both simultaneously. The user can view either one or both in the *Design Results*.

The *Show* menu has these two selections at the top of the menu.

### 25.10.4 Load Directions

For wind loads, the user must choose one of the combinations of load directions, e.g *West->East and South ->North.* These correspond to wind coming from the Northeast, Southeast, Southwest, and Northwest. The *Show* menu item for this is called *Load Direction.*

For the low-rise wind calculation method, the program displays the low-rise wind load case for this windward corner, e.g. the Southwest corner if *West->East and South->North* is selected. When the low-rise method is selected, the *Show* menu names these selections as *Southeast, Northwest,* etc.

For all heights wind load method, the wind loads for the combination of orthogonal directions are defined by the selection. They do not represent a separate load case.

It is possible to view the critical forces in *Plan View* using a menu item in the *Load Directions* menu that is found in the in the *Show* menu only. In this case, no loads will appear on the screen and the forces shown are those derived from the critical wind load case on each shearline.

The design results show only the worst case all the wind load cases, except for the Canadian version in the case that the results are different in opposite directions due to $J_{hd}$ factors, and both are listed.

### 25.10.5 Combined or separate wind loads

The program allows the user to list the loads in the *Design results* as all the loads generated on each element of the structure, or as the combination of loads where they overlap. The first option allows the user to see which building element the load corresponds to, and the second of these creates a more compact list of loads.

The *Show* menu item for this setting appears when the Design Results are showing, under *loads/Wind Shear loads.*

### 25.10.6 Load Type

Turn on and off the main load types – *shear loads* (both seismic and wind), *C&C loads*, *uplift loads*, *dead loads*, *building masses*. Any of these can be on or off in any view.

In the *Show* menu, these choices are found in a sub-menu under the main menu item *loads*.

### 25.10.7 Forces

These settings allow the user to turn on and off the forces created by the program – *Shearline forces*, *Hold-down forces* and *Drag strut forces*. In the *Show* menu, these choices appear in a submenu under *Forces*.

In *Plan view*, only *Shearline forces* and the combined effect of *shear overturning*, *dead*, and *uplift hold-down* forces are shown.

### 25.10.8 Wind Load Generation: Case 1 and Case 2

With ASCE 2010, the terminology "Directional" is used to refer to what used to be called "All Heights" loads (Chapter 27), and the term "Envelope" is used to refer to what was "Low-rise". The old terminology is still used in the captions to the Figures giving the particulars of these methods. This change has been implemented in the program.[8]

The program now generates both Case 1 and Case 2 loads from Figure 27.4-8 simultaneously, and uses the heaviest loading from each of these cases and the minimum load case as the design shearline force on each line. Previously, if the user had to generate these loads in a separate design run and manually compare the resulting designs to determine the critical case.

Case 1: full wind loads without accounting for torsional effects will be used if this option is selected for rigid analysis.

Case 2: 75% of wind loads plus the effect of torsion will be used for rigid analysis.

Both options can potentially govern, depending on the shape of your building, and therefore the design results for both cases should be reviewed.

These settings allow the user to decide which wind load case to display in *Plan* and *Elevation* View (Case 1 or Case 2).

# 26 *Shearwalls* Data Bars

Data bars are located at the top of each window or "view". They allow users to change building levels, to turn data on and off in the corresponding view, and provide for quicker selection of options and settings than by the Settings dialog or the main menu. The buttons vary for each **view**.

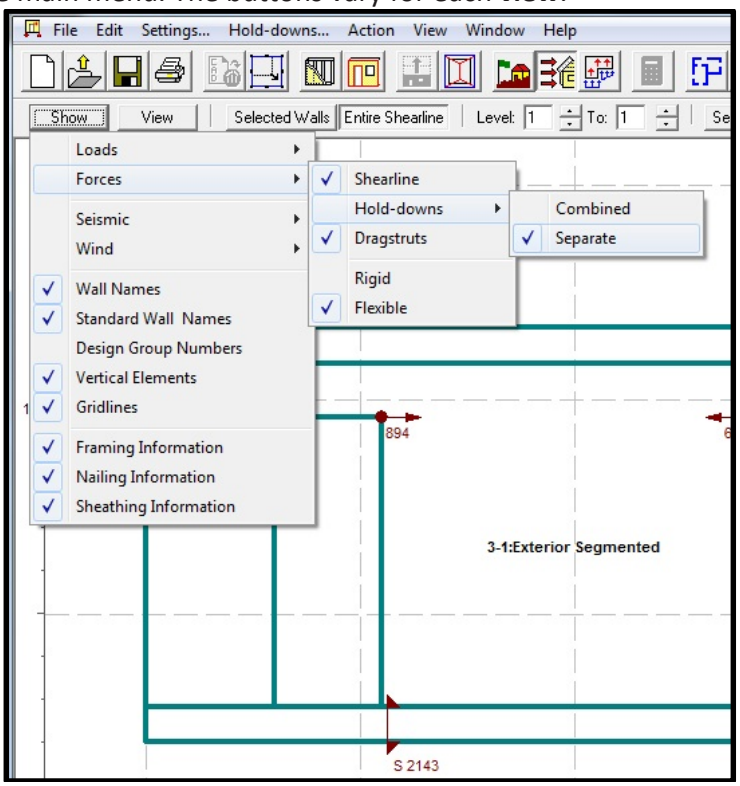



Figure 211: *Show Shearwall Hold-down Forces Combined or Separate*

The data bar for each view is described below:

- **Plan View:** Show, View, Settings, Hold-downs, Log file, Current Level, Zoom In, Zoom Out, Undo, Redo, Getting Started
- **Elevation View:** Show, View, Selected Walls/Entire Shearline, Level
- **Results View:** Show, Settings, Preview, Go To Table

## 26.1 Show Button

This button is used to turn data on and off in the corresponding view. The use of the *Show* button becomes more apparent once loads have been applied to the structure. The button allows the user to quickly change which loads to view in *Plan* and *Elevation* Views. The user has the ability to pick between wind and seismic loads, as well as the direction (ie. North-South, East-West, etc.). The user also has the option of deciding which loads (Shear, Uplift, Dead, C & C, Mass) and forces (Shearline, Hold-down, Drag Strut, Flexible or Rigid Diaphragm) to display in the view.

The button also allows the user to quickly select which information to display in the *View*, such as *Wall Names*, *Standard Wall Names*, *Design Group Numbers*, *Vertical Elements*, *Gridlines*, *Framing Information*, *Nailing Information*, and *Sheathing Information*.

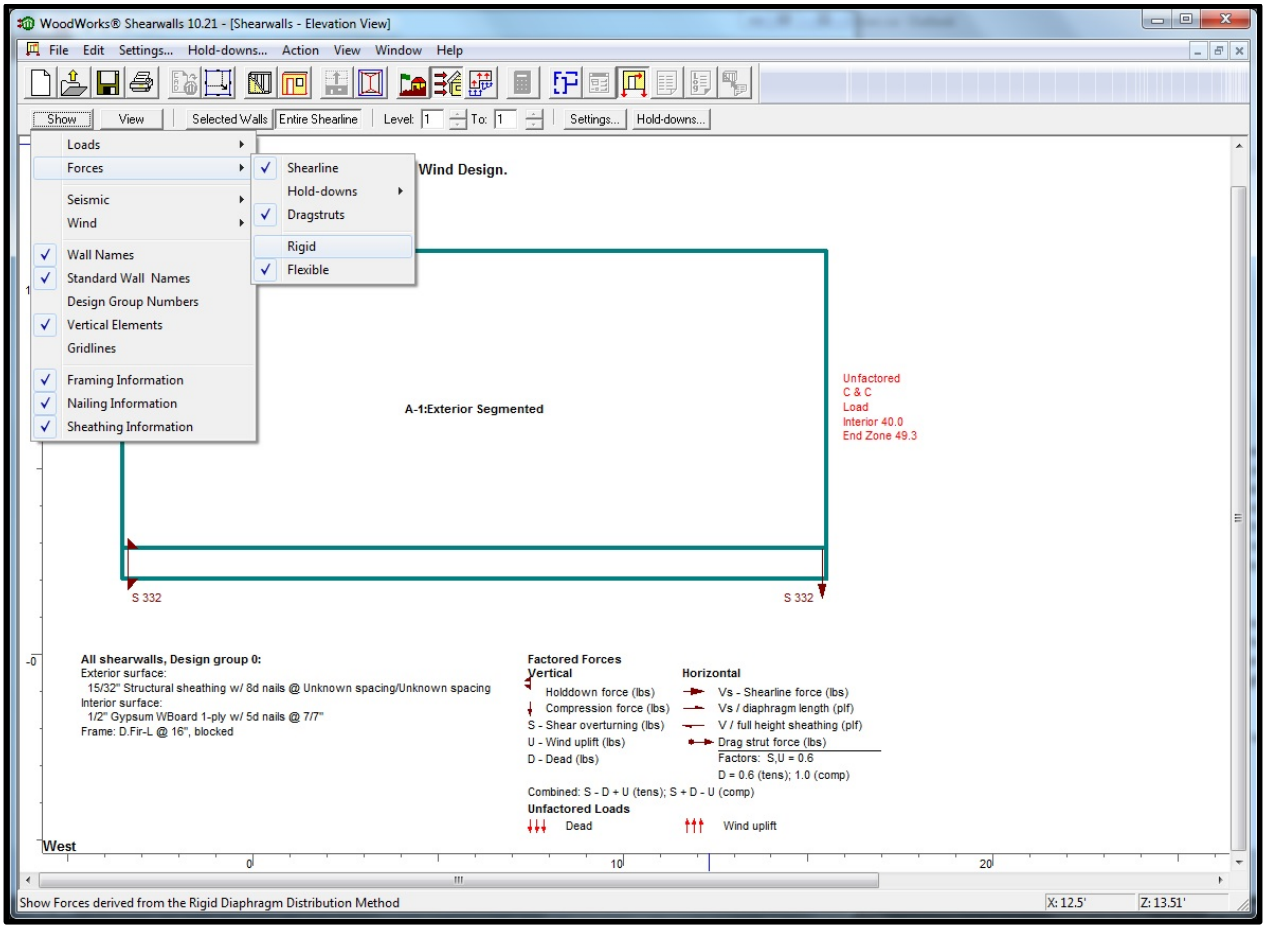



Figure 212: *Show Shearwall Forces Rigid or Flexible*

## 26.2 View Button

This button is used to change viewing options.

## 26.3 Settings Button

This button is used to modify design settings.

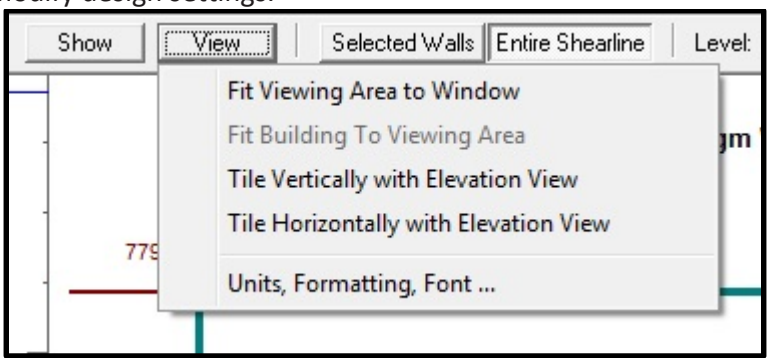



Figure 213: *Shearwalls View Entire Shearline*

## 26.4 Preview Button

This button is used in the *Results* view to create a print preview of the *Design Results* output report.

### 26.5 Log File

This button is used to view the results in a log file. For more information on the log file, see 35.19.2, Results.

### 26.6 Go To Table Button

This button is used in the *Results* view to select which table to view. For more information on accessing *Results*, see section 34.4.

### 26.7 Hold-Down Button

This button opens the hold-down database window where hold-downs can be created and designed. See section 37.2 for more details on hold down design.

### 26.8 Other Data Bar Options

- **Current Level Button** – Controls the active level while in the *Plan View*. Only one level can be active while in this view.
- **Level Buttons** – Control the levels shown in the *Elevation View* and those for which output results are desired in the *Results View*.
- **Selected Walls / Entire Shearline Button** – Controls which walls along a shearline are shown in the *Elevation Vie*w.

## 27 *Shearwalls* Menus

The main menu bar of the program contains all of the commands represented by the toolbar buttons. In addition, it contains several less commonly used commands, such as ***Print Preview***, ***Fit View To Window***, ***Select All Shearwalls***, ***Delete Shearwall***, and ***Merge Shearwalls***.

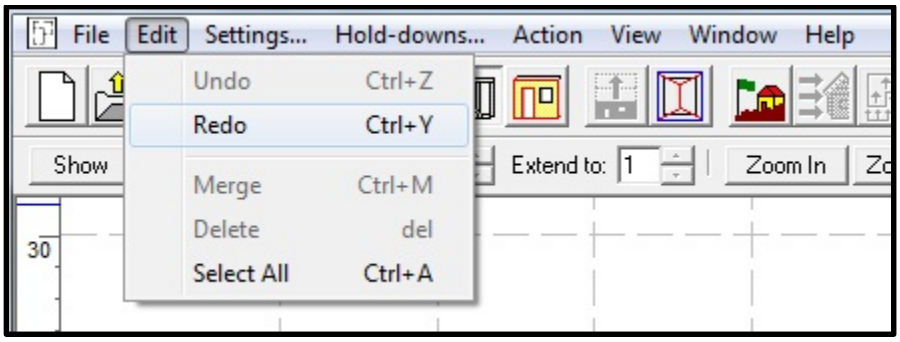



Figure 214: *Main Menu Bar*

### 27.1 Context Menus

Context menus will pop up any time the user performs a right mouse-click. They contain some of the more commonly used menu and toolbar commands, and some extra shortcuts such as tiling the *Plan View* with the *Elevation View*.

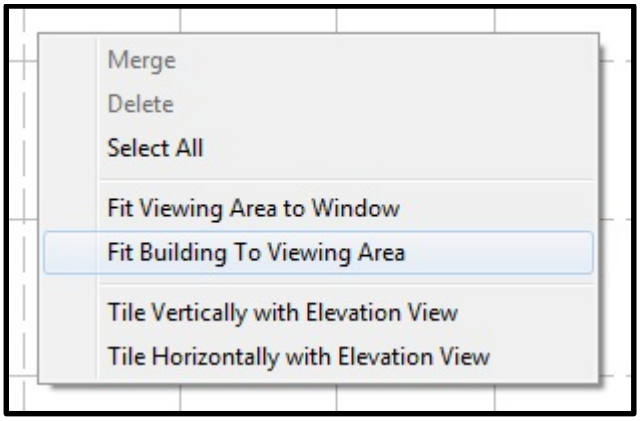



Figure 215: *Context Menu*

# 28 *Shearwalls* Views

*Shearwalls* is divided into three main views in large windows that usually cover the whole viewing area, and several data input views that appear in a smaller window. The three main views are ***Plan View, Elevation View*** and ***Results View***. The data input window is known as the ***Form View,*** which is visible only when the *Plan View* is active, and allows for the input of structure, shearwall, opening and load data.

The data and viewing options for the *Plan*, *Elevation* and *Results Views* are controlled through the Data Bars or the Settings menu.

## 28.1 Plan View

- This is the main working window for interactive graphical input of CAD drawings, structure outline, building levels, shearwalls, openings, and loads.
- This window is always present, but may be hidden by other windows. Pressing the *Plan View* button brings the *Plan View* window back into view.
- In *Plan View*, the user has the option of sizing the building to fit the current size of the window, or setting fixed extents for the window and using scroll bars, if necessary, to navigate around the building. The fixed extents are set in ***Settings/View***.
- Rulers appear along the sides of the plan view to show the scale, and the current mouse position is displayed in the ***Status Bar*** at the bottom right corner of the view.
- Gridlines are not named, or created by the user, as in *Sizer*. Instead, they are shown at regular multiples of the program's snap increment. Both the snap increment and the multiple are specified in the View settings.

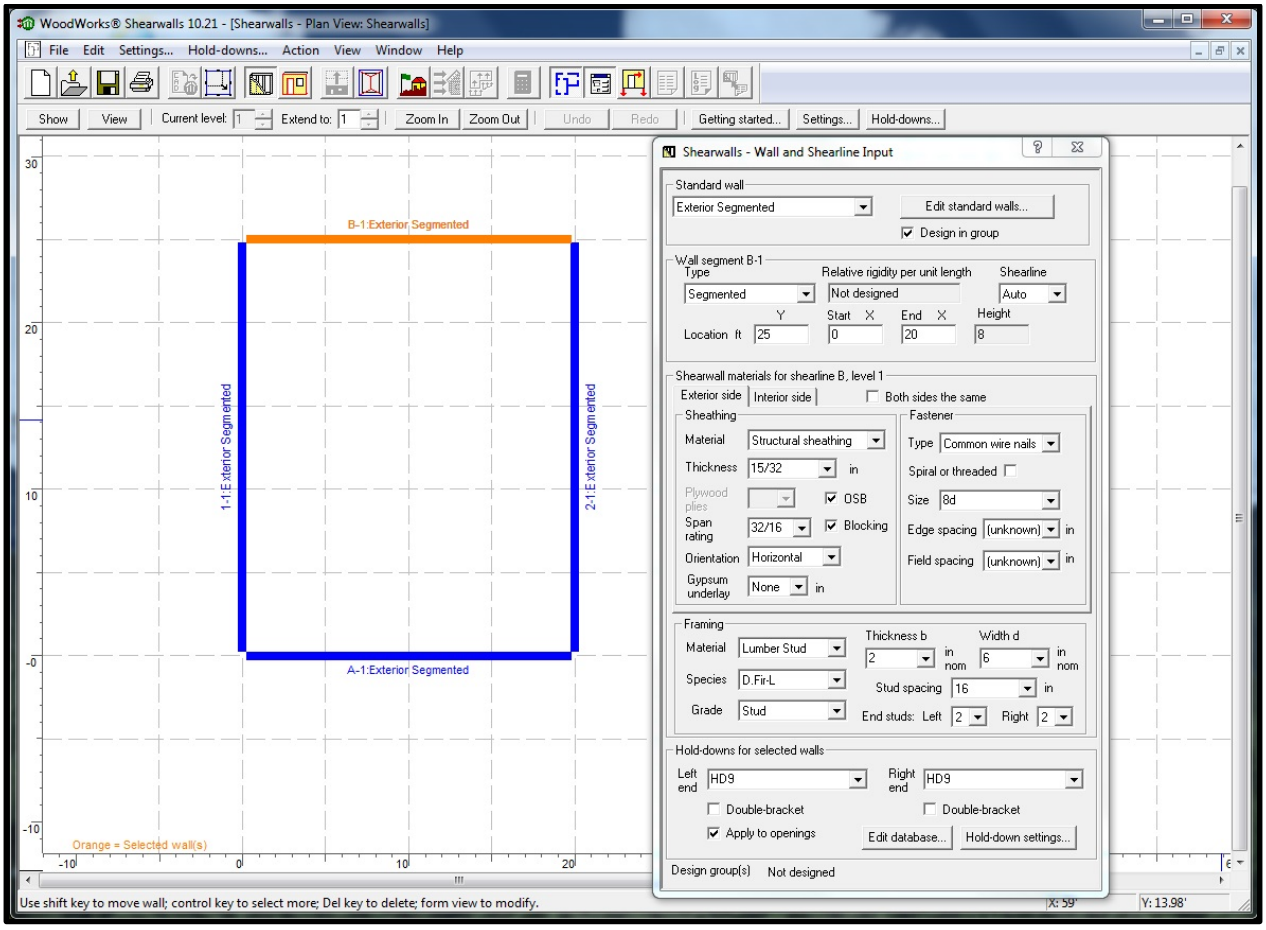



Figure 216: *Shearwalls Plan View*

## 28.2 Form View

This view contains one of a set of forms for the input of data for building elements and loads. Each form contains data fields appropriate to the depressed Action button in the main toolbar:

- **Structure Input Form** – Available when the Structure action button is depressed, and is used to input the structure *blocks*, the number of *levels* for each block, and the *wall heights* and *joist depths* for each level.

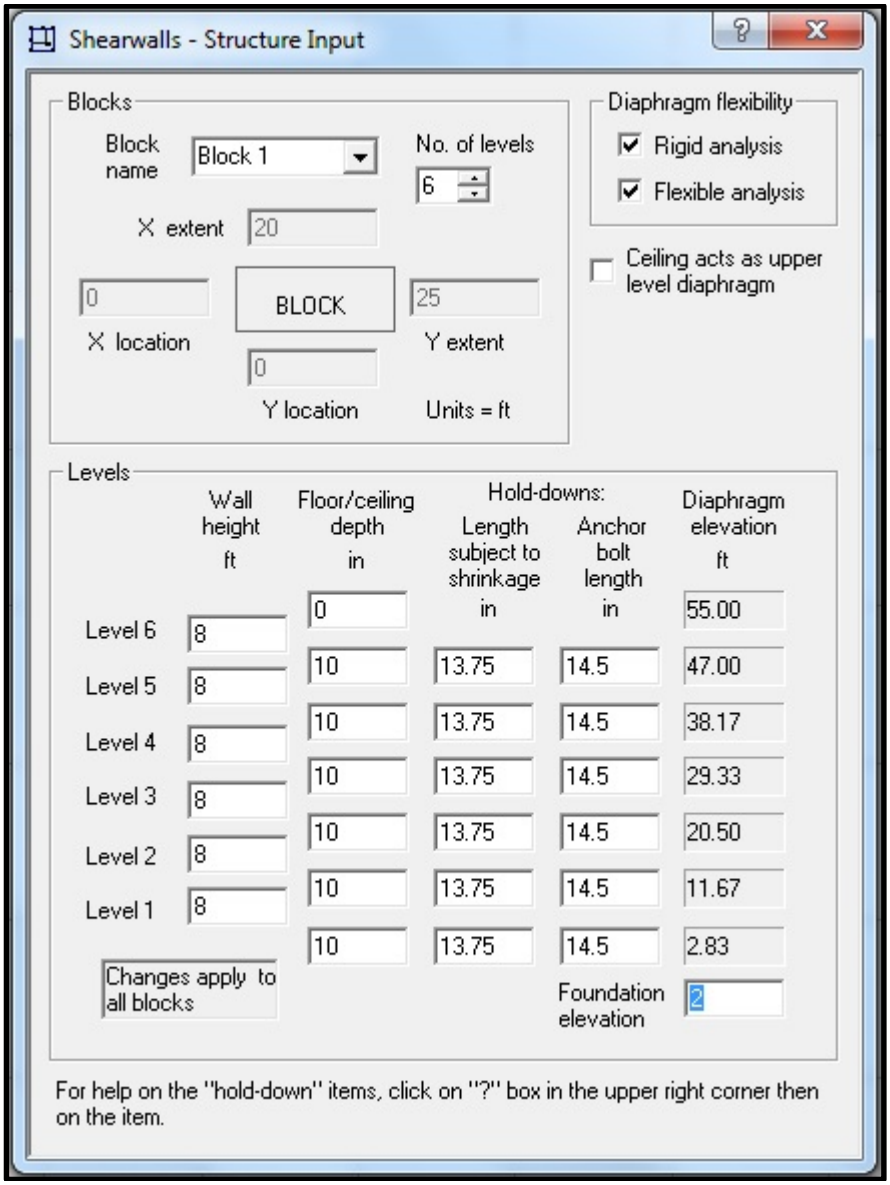



Figure 217: *Shearwalls – Structural Form Input*

- **Wall and Shearline Input Form** – Available when the *Walls* action button is depressed. It is used to modify *wall locations*, and to input *wall material data* for the entire shearline (sheathing, fastening and framing details). It can also be used to create *Standard Walls* for future use.

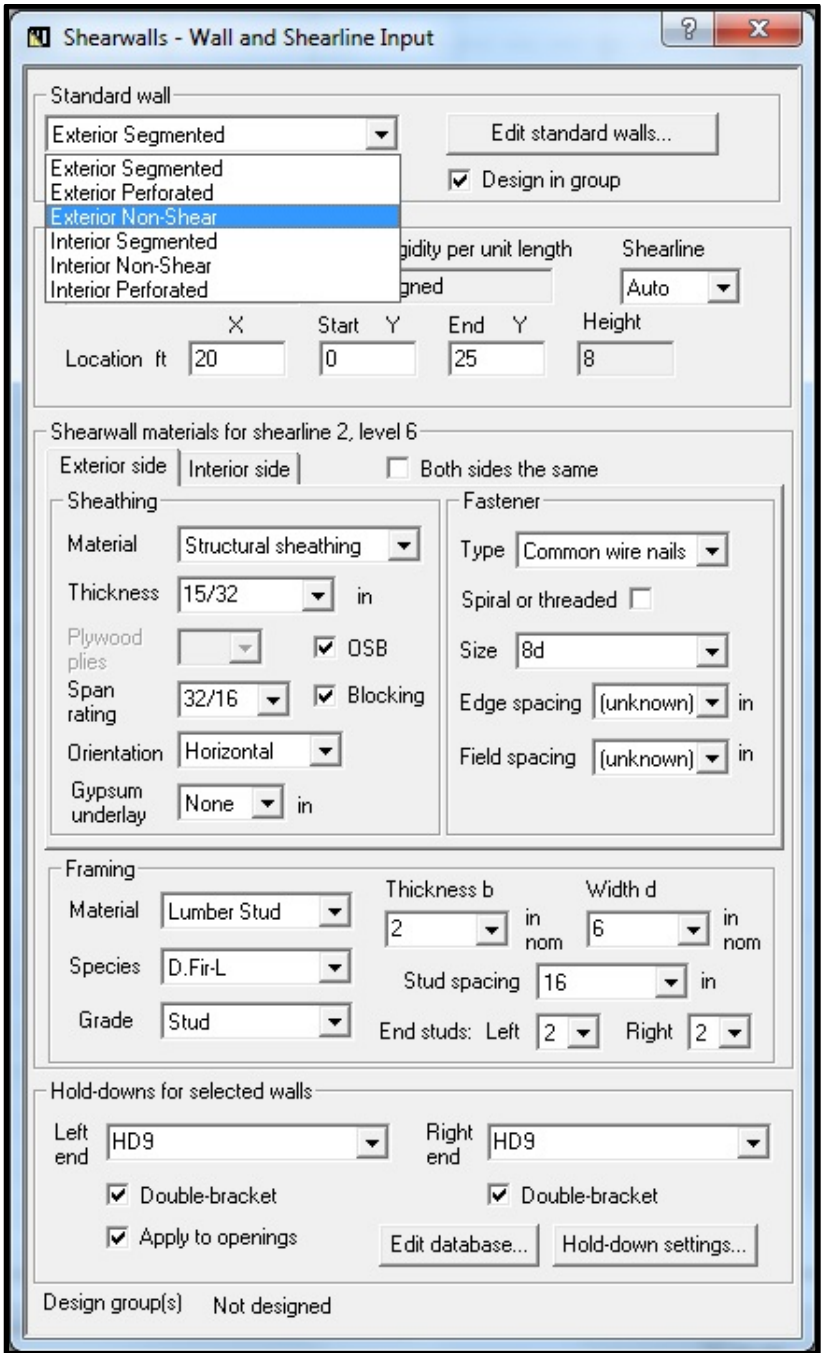



Figure 218: *Shearwalls – Wall and Shearline Input*

- **Opening Input Form** – Available when the *Opening* action button is depressed. It is used to specify opening *locations* and *dimensions*.
- **Roof Input Form** – Available when the *Roof Block* action button is depressed. It is used to specify the *roof geometry*.
- **Generate Loads Form** – Available when the *Generate Loads* action button is depressed. It is used to generate *wind* and *seismic* loads.

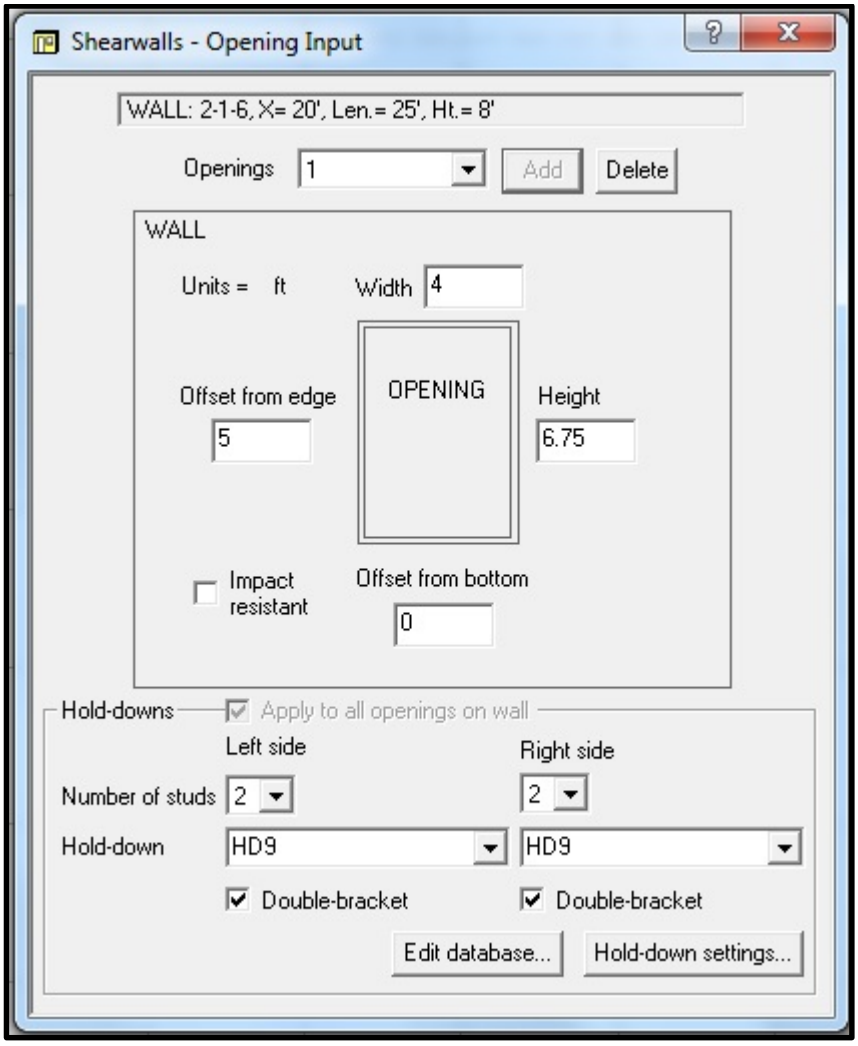



Figure 219: *Shearwalls – Opening Input*

- **Load Input Form** – Available when the Load action button is depressed. It is used to add and modify seismic, wind shear, C&C (suction), building mass, dead and uplift loads to wall lines and building faces.

The *Form View* displays the data for whatever object the user has selected in *Plan View*. (An item is selected by clicking on it with the mouse to highlight it in red). When visible, the *Form View* remains on top of all other windows, so that the user can see the building elements and their data simultaneously. Pressing the *Form View* button makes the form disappear or reappear.

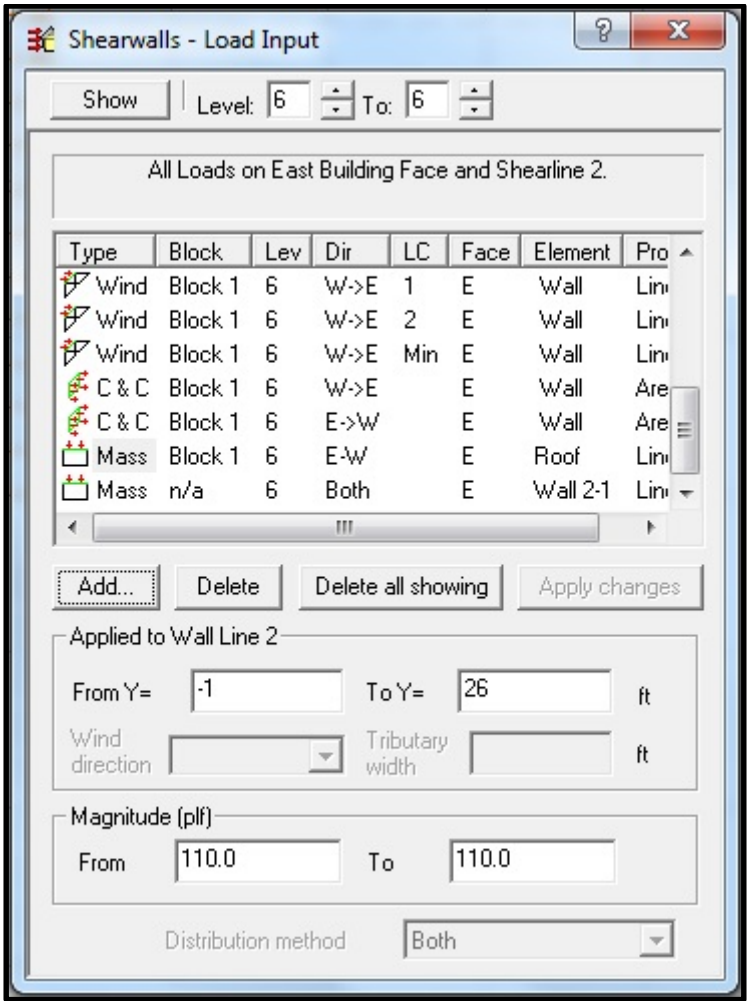



Figure 220: *Shearwalls – Load Input*

## 28.3 Elevation View

- *Elevation* view shows a drawing in elevation of the shearwall or walls selected in *Plan* View.
- It will only show multiple walls if they are along the same shearline (a line of shearwall segments subject to the same loads).
- It displays either the walls selected in *Plan V*iew or all walls on the selected shearline.
- It will show all walls on the shearline for any range of building levels.
- It displays the dimensions of building elements; openings; applied shears, dead and uplift loads; and output data such as shear resistance; hold-down forces; dragstrut forces; nailing patterns required; and sheathing thickness.
- Pressing the ***Elevation View*** button creates the window if it does not already exist, and brings it into view.

- It is useful to tile this window with *Plan View*, where you can select the shearwall you wish to appear in Elevation View.

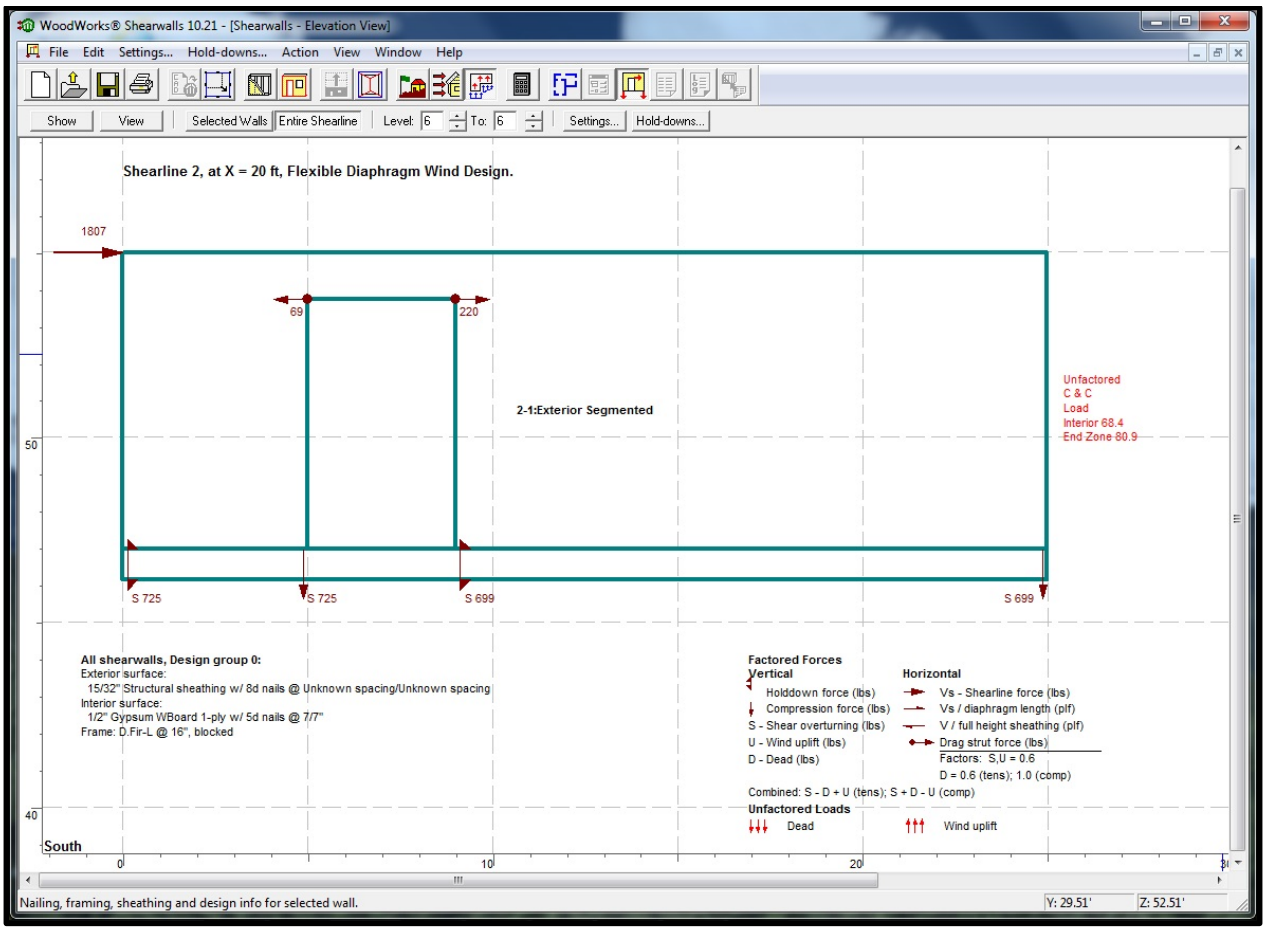



Figure 221: *Shearwalls Elevation View*

## 28.4 Results View

*Results* view shows a detailed design summary organized into the five main sections: *Project Information*; *Structural Data*; *Loads*; and *Design* (*Wind*, *Seismic*). These five sections can be accessed quickly once the Design Results is viewed by using the "***Go to table***" button in the ***Data bar***. Each of these contains several tables, described below.

- **Project Information**: The ***Company Information, Project Description*** and ***Design Settings*** entered by the user in the ***Settings*** dialog are shown in the first two tables of this section. Data entered in the ***Load Site Information*** form is presented in the third table.

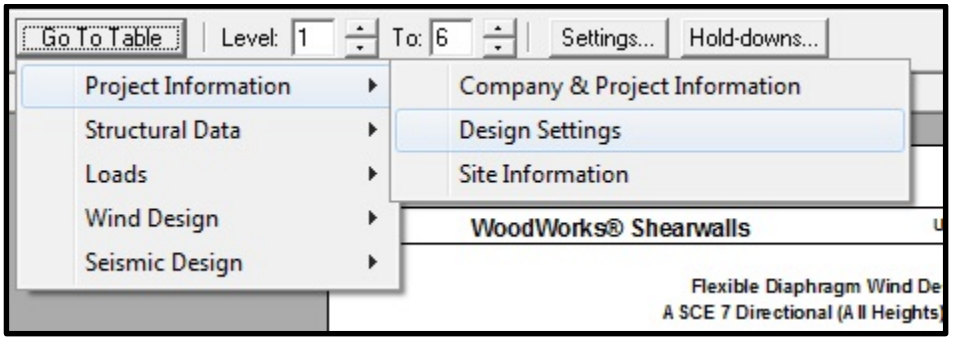



Figure 222: *Results View – Go To Table – Project Information – Design Settings*

- **Structural Data**: The geometry of the building model is summarized in the ***Story Information*** and ***Block and Roof Information*** tables. The program creates Design Groups of walls with identical materials, and lists these in the two separate tables: ***Sheathing Materials by Wall Group;*** and ***Framing Materials by Wall Group***. These two tables, in conjunction with the designed ***Wall groups*** shown in the Shear Results tables, summarize the shearwall construction requirements for the entire building. The dimensions and properties of individual walls, including the group to which they belong, are listed under ***Shearline, Wall*** and ***Opening Dimensions***.

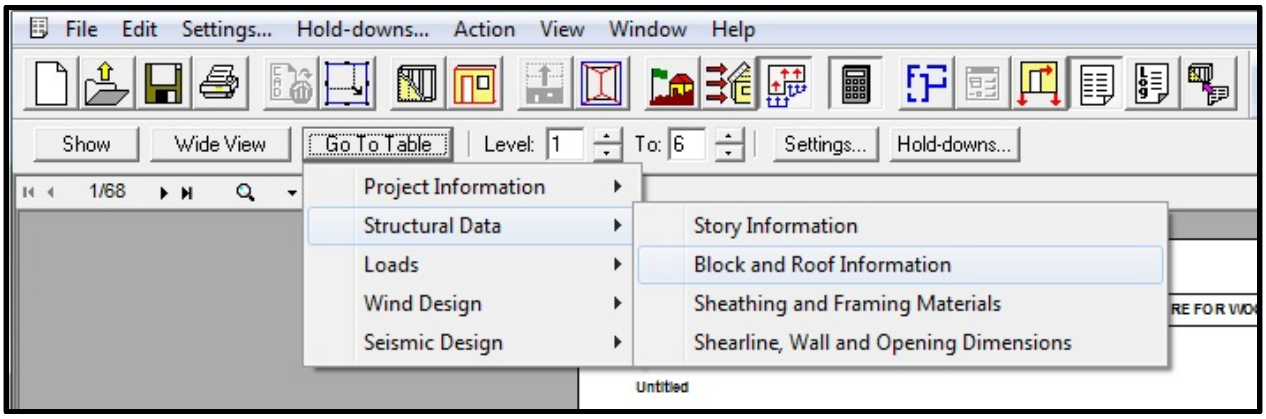
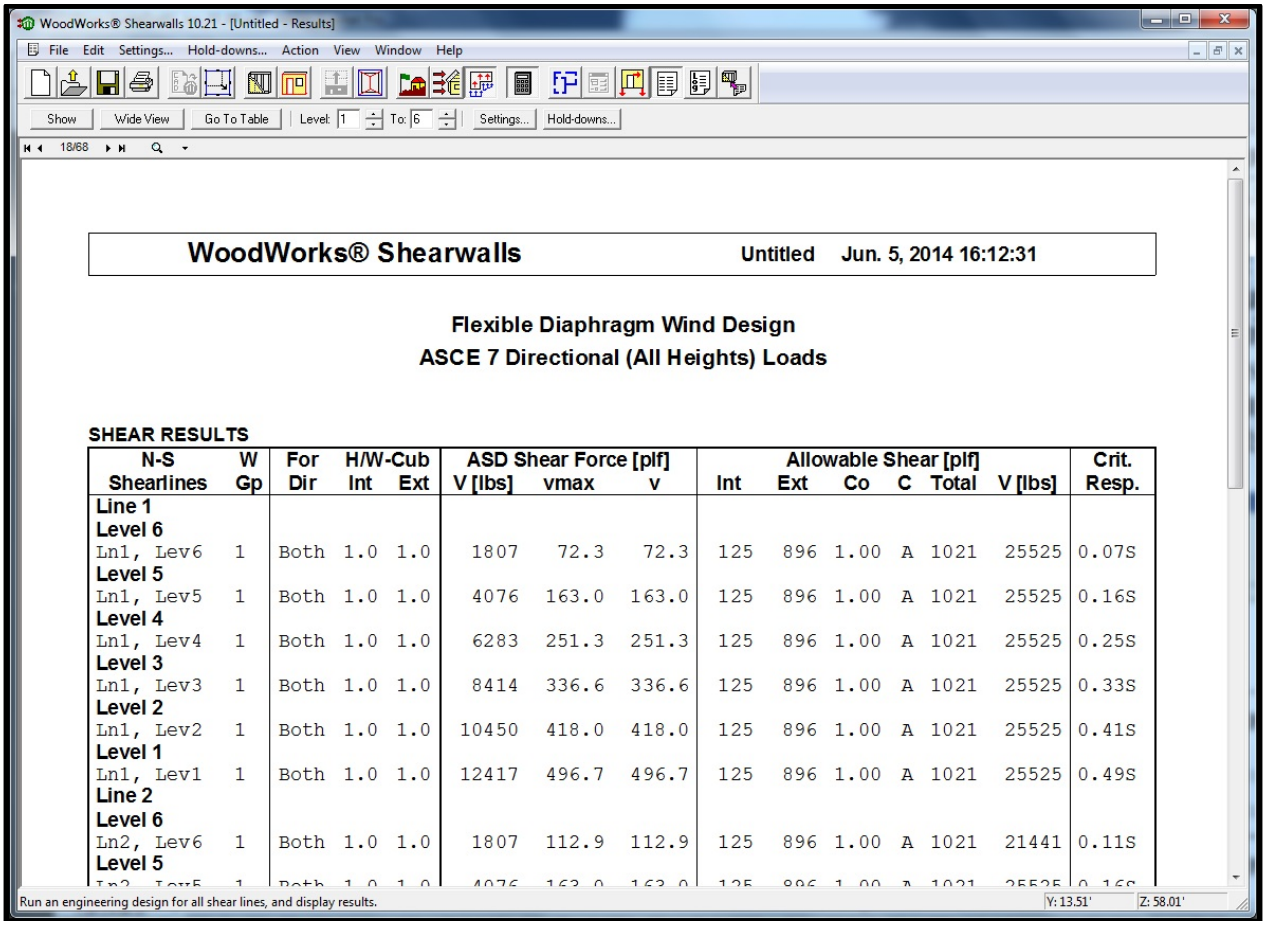



Figure 223: *Results View – Go To Table – Structural Data – Block and Roof Information*




| N-S Shearlines | W Gp | For Dir | H/W-Cub Int | H/W-Cub Ext | ASD Shear Force [plf] V [lbs] | ASD Shear Force [plf] vmax | ASD Shear Force [plf] v | Allowable Shear [plf] Int | Ext | Co | C | Total | V [lbs] | Crit. Resp. |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| **Line 1** | | | | | | | | | | | | | | |
| **Level 6** | | | | | | | | | | | | | | |
| Ln1, Lev6 | 1 | Both | 1.0 | 1.0 | 1807 | 72.3 | 72.3 | 125 | 896 | 1.00 | A | 1021 | 25525 | 0.07S |
| **Level 5** | | | | | | | | | | | | | | |
| Ln1, Lev5 | 1 | Both | 1.0 | 1.0 | 4076 | 163.0 | 163.0 | 125 | 896 | 1.00 | A | 1021 | 25525 | 0.16S |
| **Level 4** | | | | | | | | | | | | | | |
| Ln1, Lev4 | 1 | Both | 1.0 | 1.0 | 6283 | 251.3 | 251.3 | 125 | 896 | 1.00 | A | 1021 | 25525 | 0.25S |
| **Level 3** | | | | | | | | | | | | | | |
| Ln1, Lev3 | 1 | Both | 1.0 | 1.0 | 8414 | 336.6 | 336.6 | 125 | 896 | 1.00 | A | 1021 | 25525 | 0.33S |
| **Level 2** | | | | | | | | | | | | | | |
| Ln1, Lev2 | 1 | Both | 1.0 | 1.0 | 10450 | 418.0 | 418.0 | 125 | 896 | 1.00 | A | 1021 | 25525 | 0.41S |
| **Level 1** | | | | | | | | | | | | | | |
| Ln1, Lev1 | 1 | Both | 1.0 | 1.0 | 12417 | 496.7 | 496.7 | 125 | 896 | 1.00 | A | 1021 | 25525 | 0.49S |
| **Line 2** | | | | | | | | | | | | | | |
| **Level 6** | | | | | | | | | | | | | | |
| Ln2, Lev6 | 1 | Both | 1.0 | 1.0 | 1807 | 112.9 | 112.9 | 125 | 896 | 1.00 | A | 1021 | 21441 | 0.11S |
| **Level 5** | | | | | | | | | | | | | | |

Figure 224: *Results View – Go To Table – Wind Design – Flexible Diaphragm*

- **Loads:** Dead loads, seismic loads and wind loads input by the user or generated by the software are listed in separate tables.

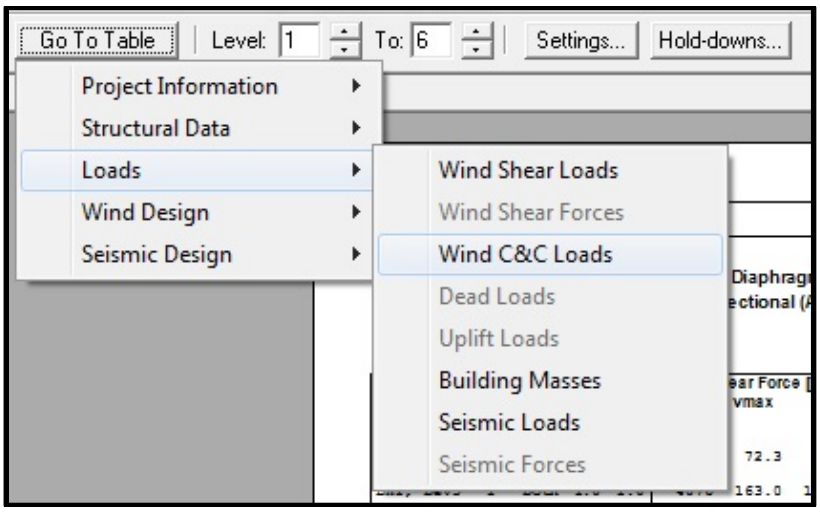



Figure 225: *Results View – Go To Table – Loads – Wind C&C Loads*

- **Design:** The ***Wind*** and ***Seismic Design*** Sections each include both flexible and rigid diaphragm design results. There are separate tables for shear results, hold-down design, dragstrut forces, shearwall deflection, and hold-down displacement. The ***Wind Design*** sections includes an additional table, Components and Cladding by Shearline, that compares the applied C&C wind suction load to the sheathing strength and fastener capacity in withdrawal. Only the ***Seismic design*** tables include story drift.
- The user can tailor the output reports using the **Show** button and can navigate from table to table with the **Go To Table** button.
- Design notes appear at the end of the wall materials table, and warnings are displayed where failed designs occur.
- The Results View window is created and displayed whenever a design is first performed.
- Pressing the **Results View** button brings this window into view for the most recent design run.

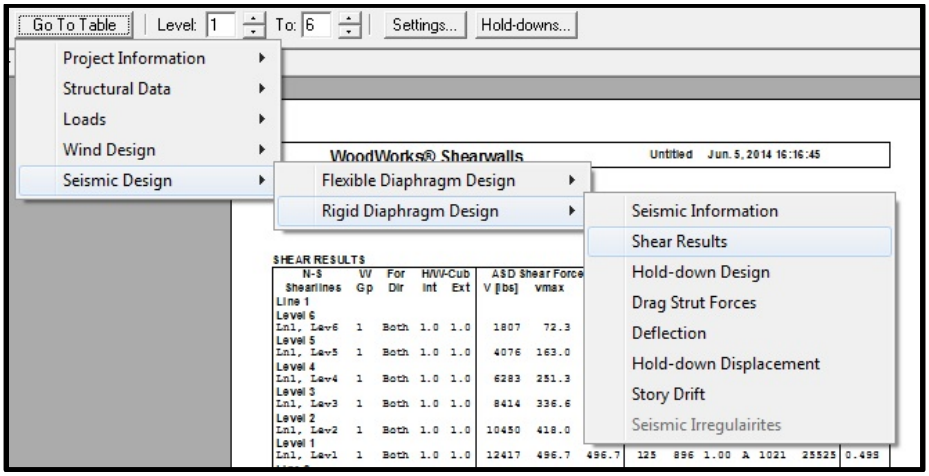



Figure 226: *Results View – Go To Table – Seismic Design – Rigid Dia. Design – Shear Results*

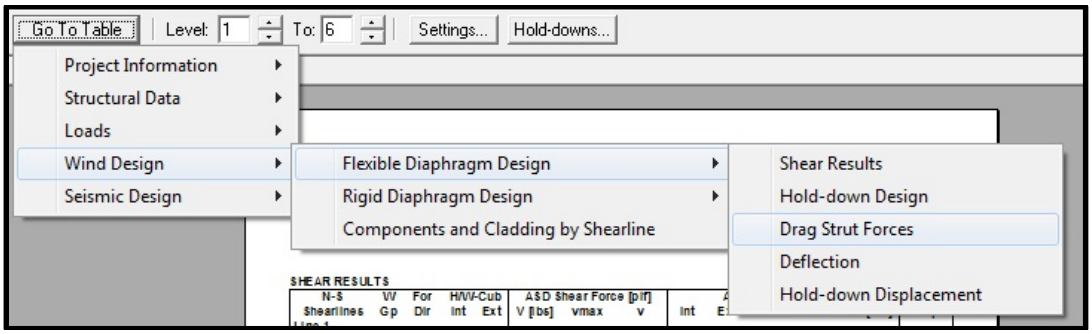



Figure 227: *Results View – Go To Table – Wind Design – Flexible Diaphragm Design – Drag Strut Forces*

# 29 *Shearwalls* Getting Started

## 29.1 General

There are four main steps to designing the shearwalls in a structure with *Shearwalls*:

1. Define the building geometry by placing building blocks, specifying the number of levels and their elevations, and graphically modifying and drawing walls, using the mouse.
2. Using the Form View, provide details for each of the walls, for their openings, and define the roof geometry. Leave those items you wish the program to design as “Unknown”.
3. Apply wind and seismic loads either generated automatically by the program or specified manually by you. Loads can be modified or regenerated at any time.
4. Run the design to determine the unknown values, calculate the forces and capacities, then display diagrams and the design results summary for all of the shearwalls.

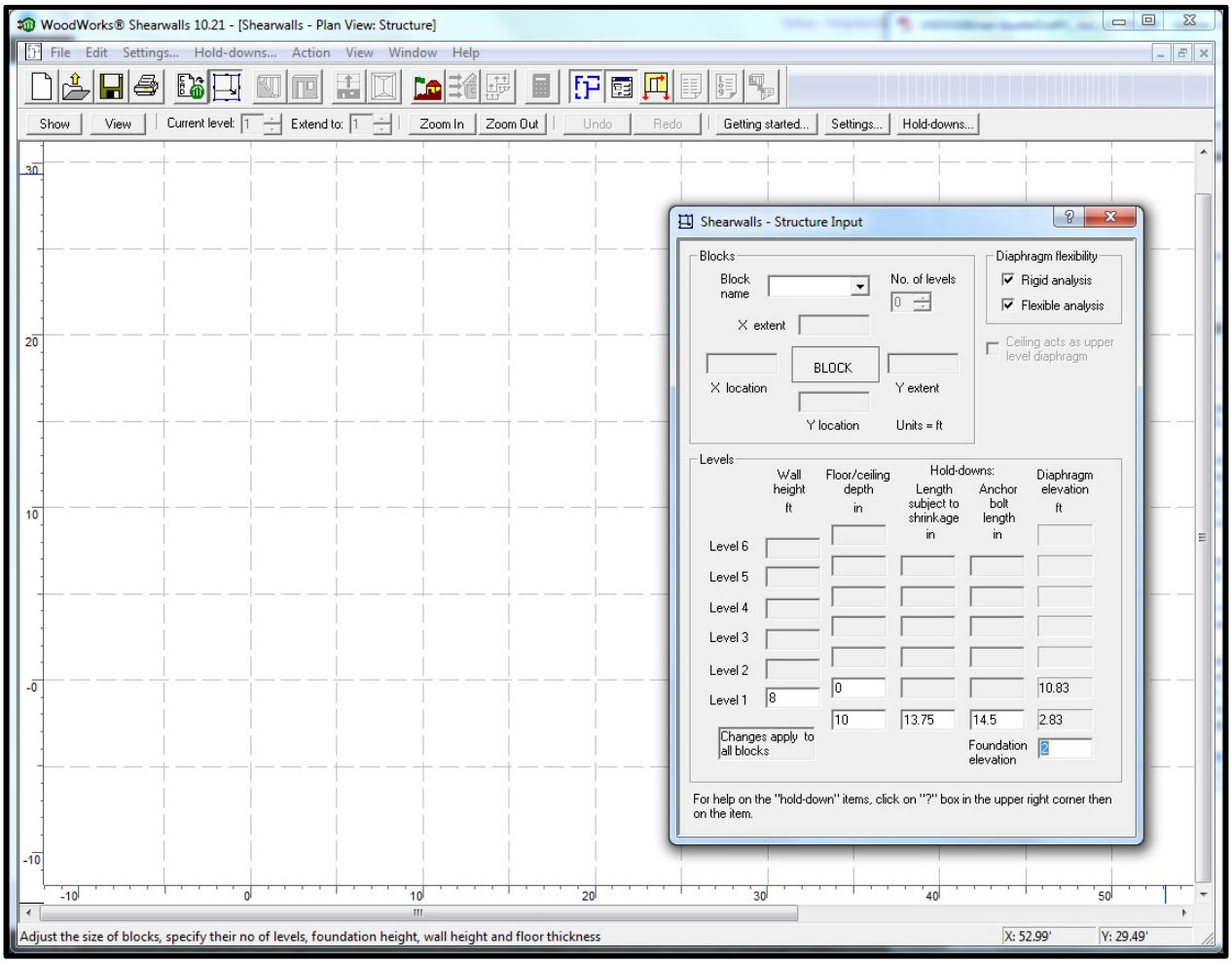



Figure 228: *Shearwalls Default Opening*

## 29.2 Step 1: CAD Import

*Shearwalls* has the capability of importing Windows metafiles (extension .wmf, .emf, and .pdf), independently for each floor. This is a file format that is an export option in CAD programs such as AutoCAD®.

An AutoCAD (.dwg) can quickly be converted into a wmf files by exporting the file using “Other formats”, and then saving the file as a .wmf file. Alternatively, drawings can be converted to a .emf or .pdf files, and then imported into *shearwalls*.

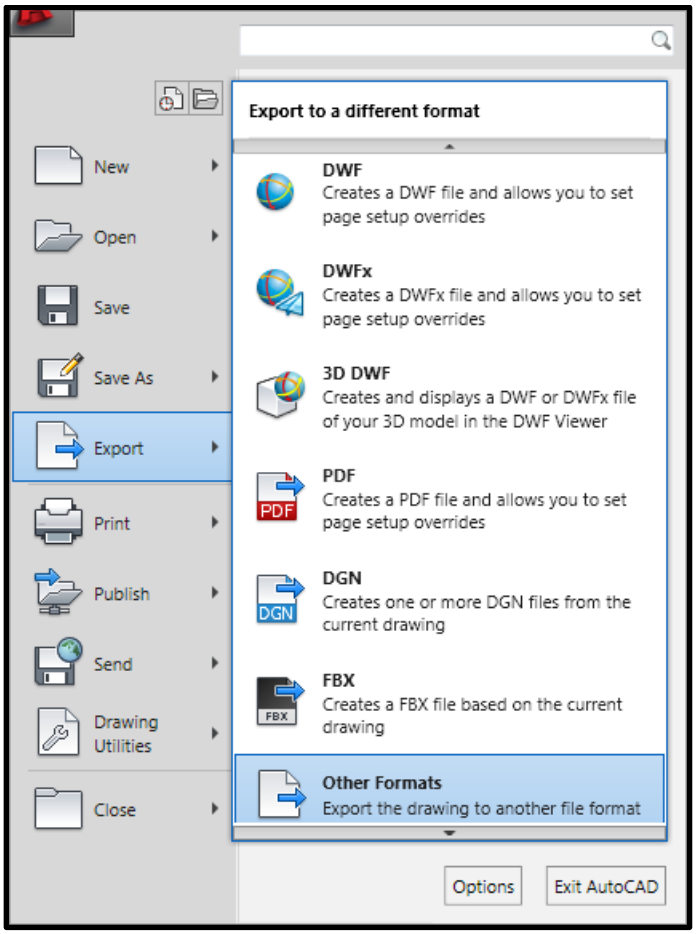



Figure 229: *Exporting CAD Document*

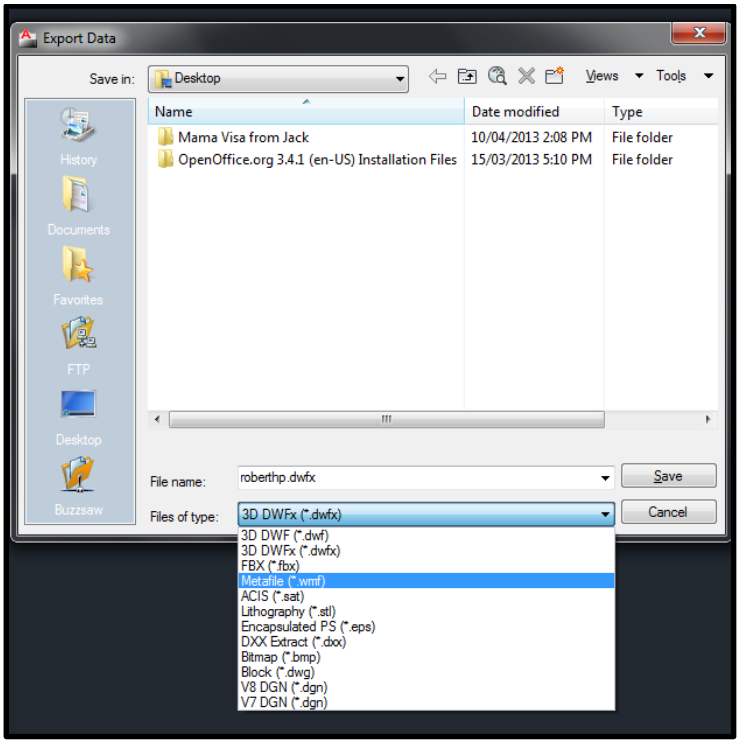



Figure 230: *Exporting CAD file as a Metafile*

Importing a CAD file is an optional step that could allow the following steps of creating blocks, walls, and openings to be completed quicker.

*Shearwalls* allows up to 6 different CAD drawing imports, which can be used for each level of the building that will be drawn in the next step. However, often importing only the first level is most efficient, and avoids the slightly tedious task of lining up each level.

You must use a white or light background with black or dark lines and text for your CAD file. Dark backgrounds are incompatible with the *Shearwalls* screen graphics.

The ***Import*** function opens these files and displays the drawing in *Plan View* for the user to trace over and to recreate the structure. Afterwards, the user can view or hide the original drawing at any time by pressing the Import button.

Click the ***Import CAD Drawing*** button on the toolbar.

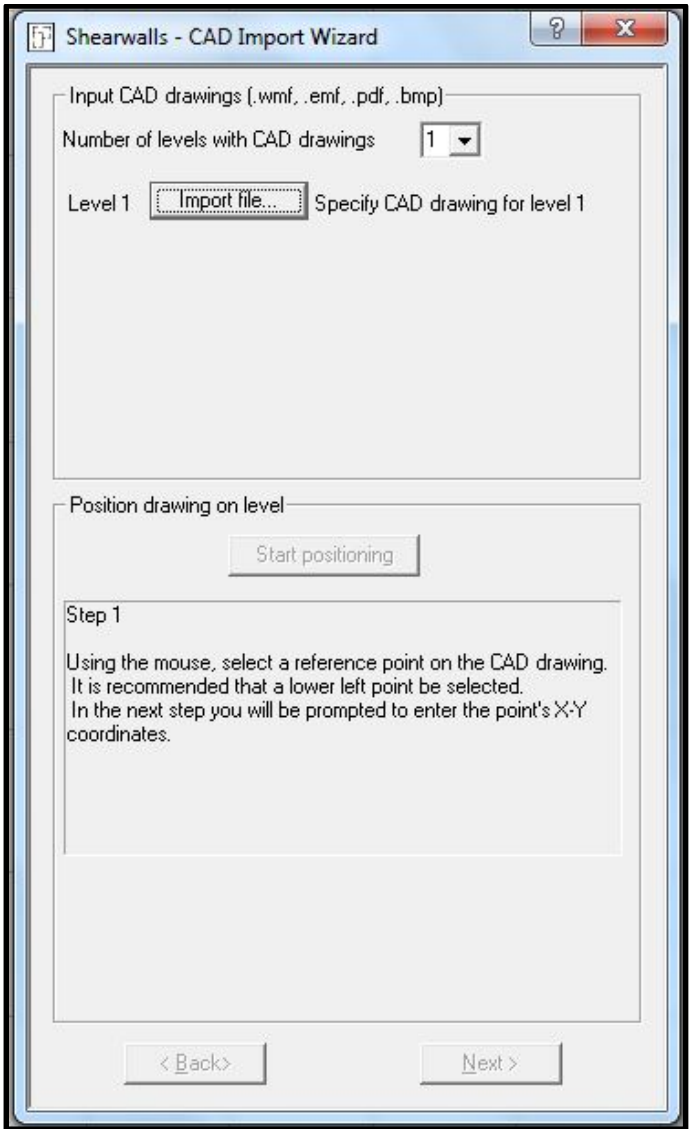



Figure 231: *Import CAD Drawing to Shearwalls Module*

1. Locate your CAD files using the dialog box and ***Open***.
2. The CAD drawing will be displayed in *Plan View*, and the rulers will temporarily disappear.

The Form View will display a form called CAD Import Wizard. To begin, it is necessary to first upload drawings for each floor before beginning to *position drawing on level*. Follow the instructions to position and scale the CAD drawing in the coordinate system of *Plan View*. *Note that when importing floor plans for multiple floors it is important to be consistent when selecting appropriate positions for the extents of each floor plan. Once this is done, rulers will appear on both axes showing the new scale. Use the Zoom buttons, and the scroll bars to resize or reposition the drawing on the screen. The limits of the viewing area and the snap increments can be set from the **View** tab of the **Settings** dialog (select **Setting/View**).*

*Note that it is not necessary to scale your drawing based on the entire width of the structure, just a known dimension of the drawing (ie. between two gridlines).*

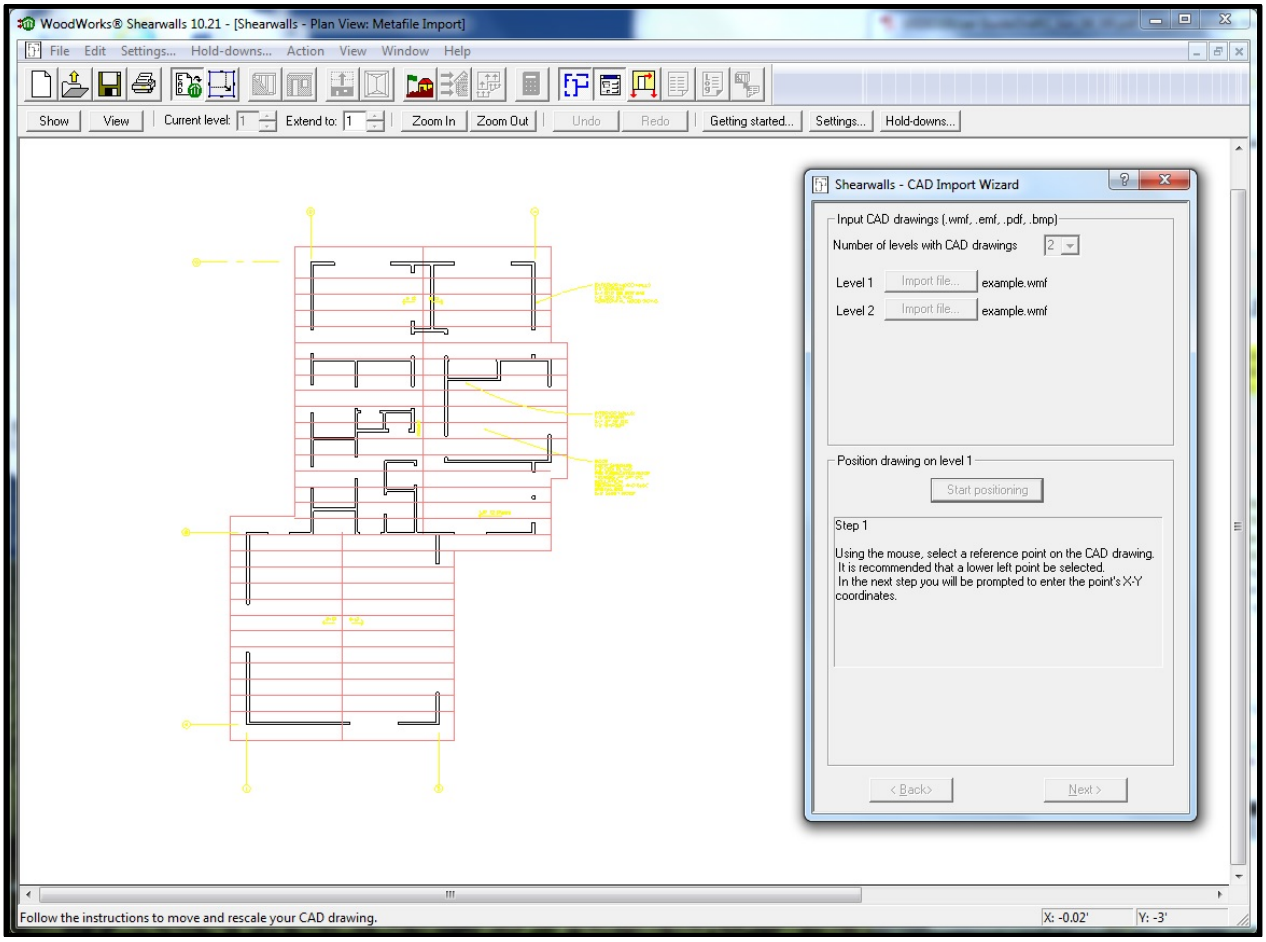



Figure 232: *Positioning CAD drawing*

*Note: Once you have positioned the last level of the structure and exited the CAD Import Wizard, it is not possible to re-enter the Wizard to change a CAD Import file. Choose your imported files carefully!*

## 29.3 Step 2: Building Footprint

Any number of rectangular blocks can be created in Structure Action. The program will ensure that any new blocks intersect with, or about to, one and only one existing block. This block can then have another block or blocks attach themselves to it. These rules ensure proper generation of exterior walls and roof panels. To select one of a number of intersecting blocks, you must click on a portion of the block that does not intersect with any other. Once selected, the block can be deleted, re-sized or moved with the mouse and shift key or in Structure Input Form, or the levels information for that block can be changed.

## 29.4 Creating Building Blocks

1. When the program opens, a new file is started, or you have just imported a CAD drawing, you will be in ***Plan View*** with the ***Structure*** button selected.
2. Click the mouse at the point where you want one of the corners of your first building block to be.
3. Drag the cursor to another point and release the mouse button to create your first block.

4. Resize or reposition the block by highlighting it by clicking on a handle and dragging it, or by changing the location or extents that appear in the **Form View** for **Structure Input**.
5. Block names can be specified by editing the **Block Name** field. The number of levels for a block, hold-down lengths and general level information can also be specified at this point.
6. Additional blocks are created by following steps 2 to 5 above and by ensuring that new blocks abut or overlap an existing block. New blocks being created cannot abut or overlap more than one of the existing blocks.
7. When exiting **Structure Action**, walls will be placed automatically around the edges of the intersecting blocks. You will not be able to move, resize or reposition the blocks while in Walls action, but the blocks will automatically resize to follow movements of the exterior walls.

*Note that if you would like to turn off Rigid or Flexible Diaphragm distribution analysis for your model, either type of distribution can be toggled in the Structure Input screen under Diaphragm Flexibility.*

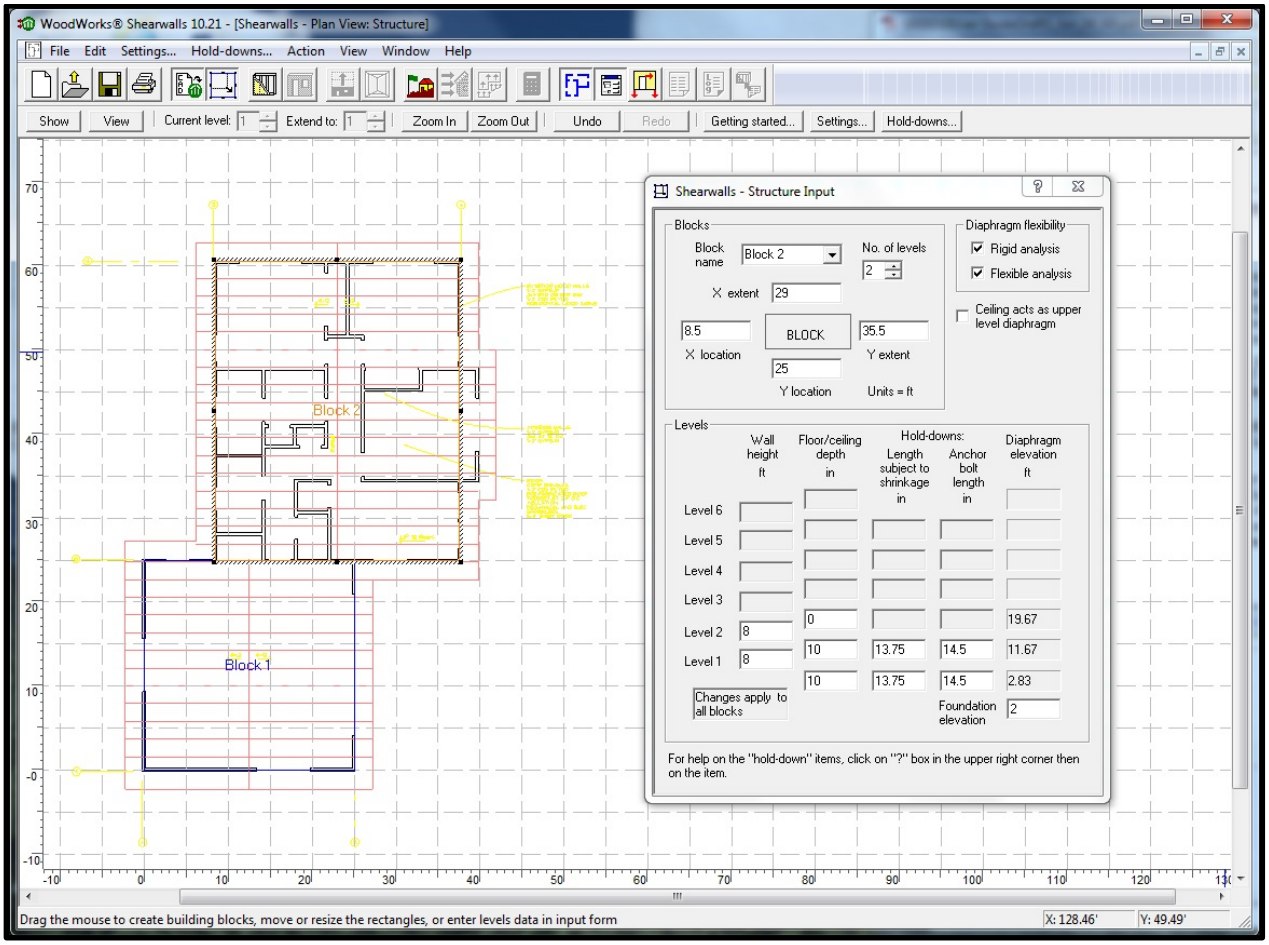



Figure 233: *Creating Building Blocks*

## 29.5 Block Creation Strategy

The decision to model a structure using a single block or several blocks depends on ease of configuring walls and the designers intentions regarding of the generation of lateral loads on roofs.

If a structure is to be designed for wind loads generated by the software using the low-rise method, the designer has no choice but to define the structure using only one block.

Multiple blocks allow for defining different numbers of levels for portions of the structure, so that roofs at different heights are possible.

In the case of small jogs in the building, it is normally not necessary to define separate blocks for these jogs. Instead, the exterior walls of a block can be subdivided and moved to create the jogs by utilizing the shift and mouse cursor.

If a block is contained entirely within one side of another block it is not necessary to be too concerned about the amount of overlap between the blocks, as walls are not created in the interior and the program automatically joins the initial roofs. However, if the blocks are diagonally opposed, the precise placement of the blocks is important to properly position exterior walls and because the roofs in this case overlap.

Since blocks cannot be added or deleted once the user enters Walls Action and the walls are generated on the first level, you must be careful to create all the blocks you need for wall creation before proceeding. Blocks can be added in Roof Action for the purposes of adding new roofs.

Building blocks establish the initial shell of exterior walls for all levels of the structure. Blocks maintain the integrity of the structure by ensuring that this shell does not have any gaps. Multiple blocks make it easier to specify irregular structures without having to segment and then move the exterior walls of the building. A common misconception with *Shearwalls* is that the blocks of the structure must match the roof shape perfectly, but this is not the case. The reason for using multiple blocks in a model would be if you have portions of your structure which vary in height (ie. 2 storey house with a 1 storey garage attached). If your entire structure is consistently the same elevation on each level, then you only need one block, and would modify the block by splitting walls and utilizing the shift key to achieve the shape of the structure. This technique is shown at the 3-4 in the following tutorial linked below where wall 4-1 is split into 3 separate walls, and then portion 4-2 is moved using the shift key. If your roof is not flat and includes multiple hips and valleys, once in Roof view, it is possible to draw multiple roof blocks to achieve the desired shape.

Canadian - https://www.youtube.com/watch?v=RdA0Bel1yeE&feature=youtu.be (8-9 minute mark)

U.S. - https://www.youtube.com/watch?v=XgrdetIHg7I (3-4 minute mark)

## 29.6 Step 3: Building Levels

The building levels are defined in the ***Structure Input Form*** before proceeding to the rest of the design sequence. **The number of levels cannot be modified once the Extend Walls button or the Roof Blocks button has been pressed (See Step 7), but the levels can be changed at any time.**

1. While in the Structure action in *Plan View*, the **Structure Input** Form will appear. (Click on the Form button if this Form is not showing).
2. Highlight one of the building blocks and in the **Levels** portion of the Form, select the number of levels for your block by clicking on the arrows or by entering a value. The maximum number of levels is **six**.
3. Specify the number of levels for all remaining blocks.

4. The program will automatically enter a **2 ft.** US version imperial / **1 meter** (Canadian, metric) foundation elevation, and the default wall height and floor depth values (as defined in Settings/Design) for each level.
5. Edit the foundation elevation, the wall heights and Floor/ceiling depth for each level. Floor diaphragms are typically in the ceiling, but diaphragms are typically not placed at the roof level, so *Shearwalls* assumes a 0 depth for the floor/ceiling depth of the roof. This information applies to all blocks, so that the floor elevations are the same for each block. The blocks can have different numbers of levels.

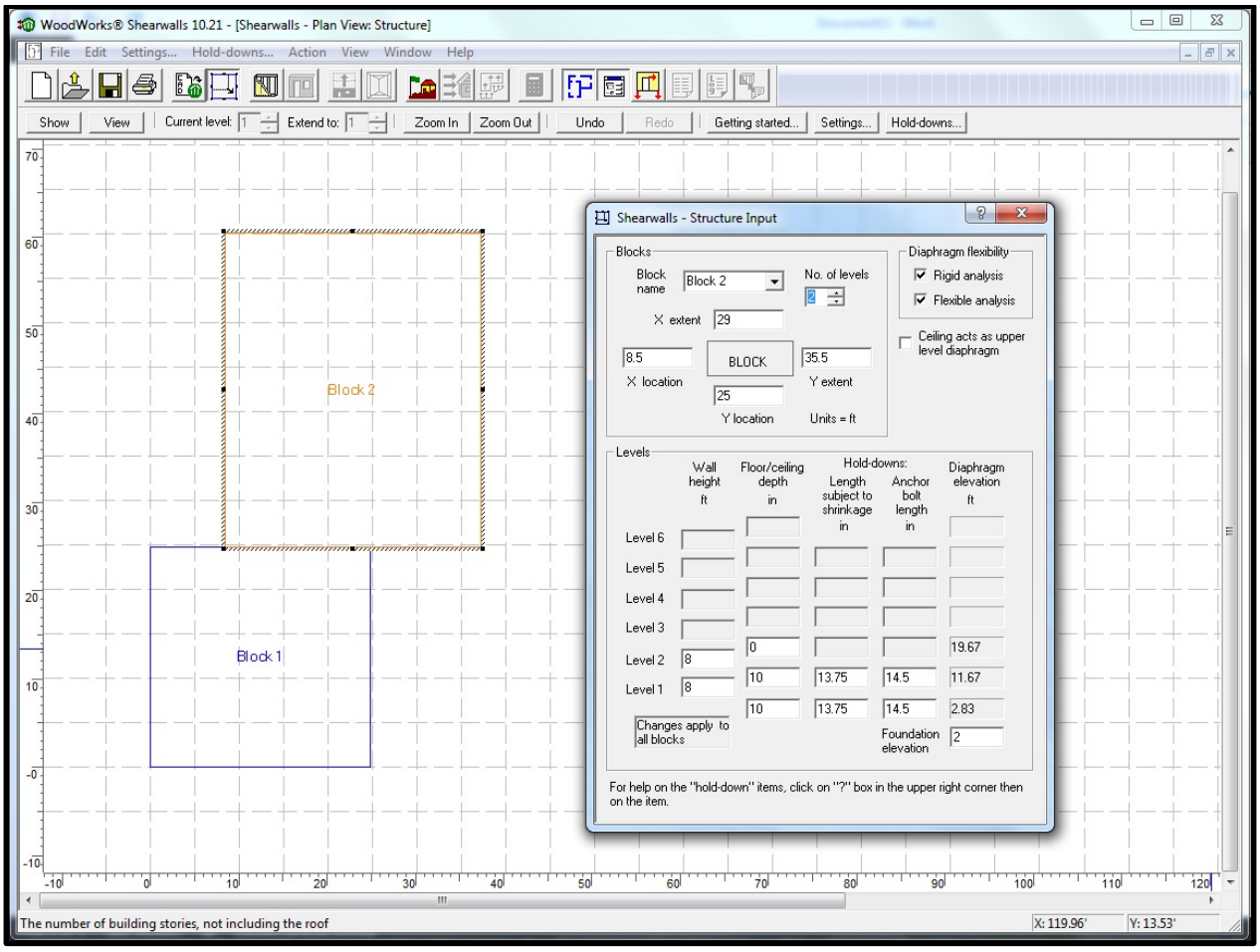



Figure 234: *Selecting Number of Storeys for blocks*

## 29.7 Step 4: Create Walls

Once blocks have been created, and the number of storeys for each block has been specified, it is now necessary to create the exterior walls of the structure. By selecting the Walls icon, the walls of the various blocks will automatically become the exterior walls of the structure. By default, the walls will be considered to be exterior segmented walls for the U.S. version of *Shearwalls*, and exterior with hold-downs for the Canadian version of *Shearwalls*.

## 29.8 Extend Walls Upwards Strategy

If the model only consists of one storey, then the "Extend Walls Upwards" button does not need to be utilized. For multi-storey structures there are essentially two techniques which can be utilized to model a structure and the two techniques have to do with when you utilize the "Extend Walls Upwards" button, as you can only click this button one time during the modelling process. It is recommended to save a model before clicking the "Extend Walls Upwards" button in case it is clicked at the wrong time, so that a previous version can always be reverted back to. Below are descriptions of the two techniques that can be utilize, although a combination of the two would also be possible. Typically the first method would be utilized if each storey is consistent throughout the structure (eg. multi-storey apartments) and the second method would be utilized if the structure is small with inconsistent walls on each storey (eg. single family home).

1. Create a block, specify the number of storeys, go to walls view and move all the exterior walls to match the shape of your structure as described above, draw all the walls which are consistent on each storey, then draw all of the openings in these walls. Go back to "Walls View" and Specify walls which will be considered as *Shearwalls* and those which will be considered non-shearwalls. Click the "Extend Walls Upwards" button. Following this technique will automatically make each storey the same as you go up the structure.
2. Create a block, specify the number of storeys, go to walls view and move all the exterior walls to match the shape of your structure as described above. Add openings in the exterior walls which are consistent on each storey. Click the "Extend Walls Upwards" button. Following this technique will automatically generate the exterior shape of your structure up to the number of storeys you specified in the block view, but you will now need to go in and manually draw all of your interior walls. For this reason, this method is prone to more mistakes when the structure includes multiple storeys, so I would recommend starting from the bottom storey and working your way up so that you can see exactly where you have drawn the walls on the storey below. Also keep in mind that you can modify the locations of the walls in the Walls View input.

## 29.9 Step 5: Edit Walls

Clicking on the ***Walls*** button allows you to manipulate all the walls on the screen, and displays the ***Shearwalls*** data form in the ***Form View***.

### 29.9.1 Selecting Walls

- You can select a wall or wall segment by clicking on it. It will then appear red.
- The data for this wall will appear in the Form View, and you can perform any of the graphical or text editing procedures on the wall that are described below.
- You can select multiple walls by keeping the CTRL key depressed while you select walls. Those data common to the selected wall will appear in the Form View, and certain editing actions will be available.
- You can select all the walls on a level by right clicking the screen and selecting "Select All".

### 29.9.2 Subdividing Walls

- Wall segments can be created from existing walls in order to give them different properties, or to offset portions of a wall from the rest of the wall.
- Click on a point anywhere along an existing wall, drag the mouse to the end of the segment and then release the mouse. The wall will then appear broken into 2 or 3 segments, each with its own name.
- The new wall segment will have the same properties as the wall it was created from.

### 29.9.3 Moving Walls

- Entire walls and wall segments can be moved perpendicular to themselves.
- Hold the SHIFT key, then click the mouse anywhere along the wall to be moved and small arrows will appear around the line.
- Drag the mouse to the new position of the wall, then release. *Note that walls cannot be repositioned past other walls.*
- When moving a wall segment, two new walls will appear perpendicular to the wall being moved, to maintain connectivity with its previous position.
- In this way, extensions to the building can be made quickly.
- When moving an entire wall, the connecting perpendicular walls will be stretched or shrunk to maintain connectivity.

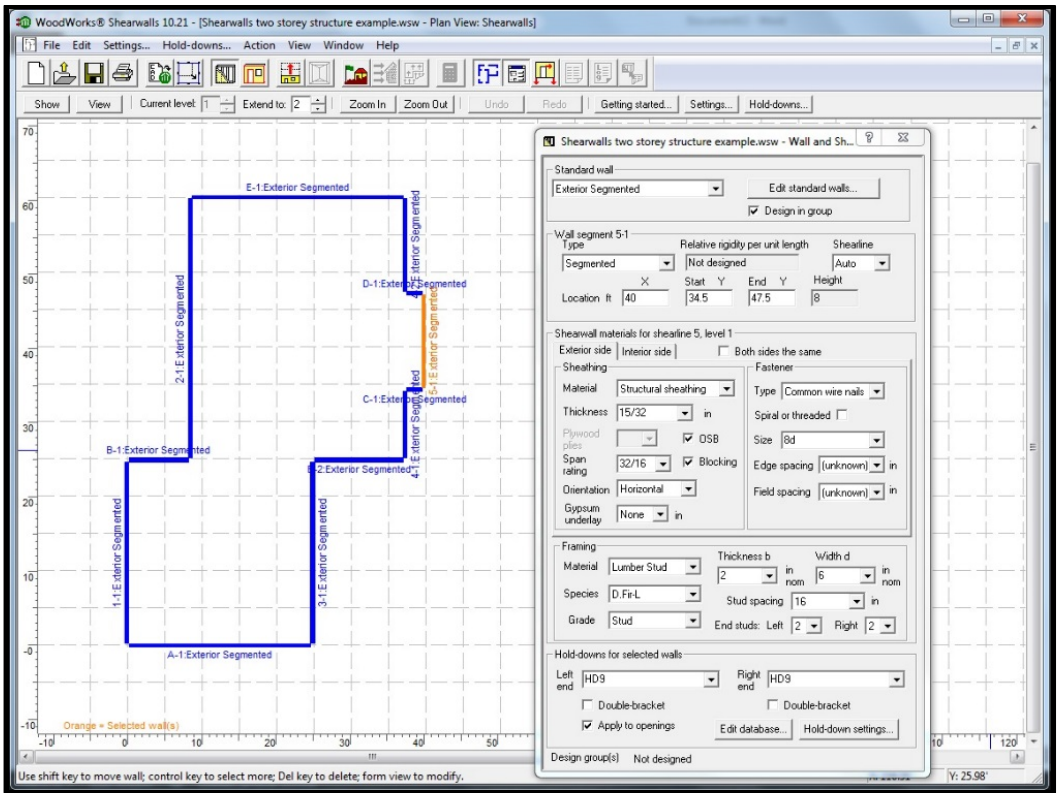



Figure 235: *Moving Walls*

### 29.9.4 Creating Interior Walls

- New interior walls can be created by clicking the start point of the wall, dragging the mouse to the endpoint, and then releasing the mouse button.
- It is not possible to create new walls outside the footprint of exterior walls that were created in the Structure Outline View.
- The wall will have the same shearwall type and be composed of the same materials as the currently selected "*Standard Wall*" in the *Wall Form View*.

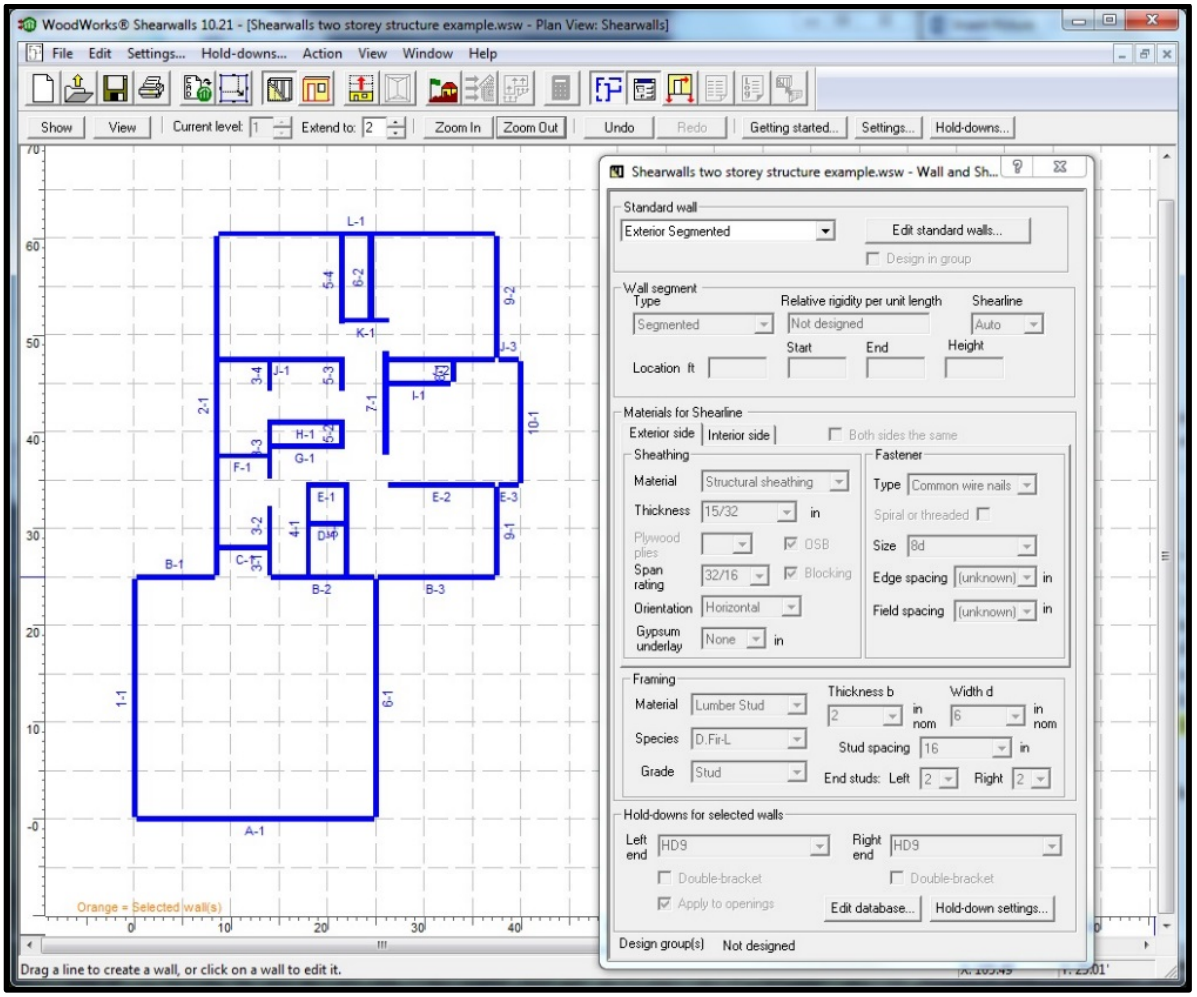



Figure 236: *Creating Interior Walls*

### 29.9.5 Resizing Walls

- Press the SHIFT key, then click on the endpoint of a wall, or where the ends of several walls meet; these walls will be selected and appear red.
- Small arrows will appear indicating the direction that you must move the mouse to resize the walls.
- Drag the mouse in that direction, and release it where you want the new endpoint to be.
- Both walls that are selected in one direction will be resized. Other exterior walls may be moved to maintain the integrity and connectivity of the structure.

### 29.9.6 Merging Walls

- You may wish to undo a subdivision of a wall, or to combine a wall segment with an adjacent one.
- With the CTRL key depressed, select the walls to be combined.
- Select the ***Edit/Merge*** command from the main menu or from the right mouse button context menu.
- The combined wall will remain selected. It will have the properties of the first wall selected.

### 29.9.7 Deleting Walls

- Select any number of walls, and press the DELETE key or select the ***Edit/Delete*** menu item.
- Interior walls will disappear entirely. Exterior walls will be merged with adjacent walls.

## 29.10 Shearlines

A shearline consists of one or more parallel shearwalls that act as one to resist an applied shear load. A shearline may contain gaps or non-shearwall components. A wall can be designated as a non-shearwall should the designer wish to neglect its contribution to a shearline. The walls may be on different wall lines and considered as part of the same shearline as long as they are separated by no more than the maximum shearwall plan offset. The maximum shearwall offset is defined in the **Settings Design** tab, and is defaulted to the following values:

- U.S. – The default offset is **6 inches.**
- Canada – The default offset is **150 mm.**

*Note: A warning is generated in the output if the offset is greater than zero for the Canadian version.*

A bandwidth approach is used to determine which walls belong to a shearline, where the bandwidth is equal to the maximum shearwall offset. For example, N-S running shearlines are created from left to right on the plan layout. The first N-S wall encountered becomes the left limit of the first shearlines bandwidth. Any wall found within the bandwidth is considered to be part of the same shearline.

The next N-S shearline starts at the first wall found beyond the previous shearlines bandwidth. Shearlines are automatically generated by the *Shearwalls* program. They are created or deleted as walls are created and moved about. If a wall can belong to more than one shearline, it can be re-designated using the Wall Input form.

Shearlines are not drawn in any of the views but when you select a wall to load in *Plan View* all walls on the shearline will be highlighted in purple. The text output is generated on a shearline by shearline basis.

### 29.10.1 Changing Wall Properties (Canadian version)

If the *Form View* is not visible, press the **Form View** button on the main toolbar. In *Plan View*, select a wall by clicking on it with the mouse. It will appear in red. Going back to the Form View, you can then change any of the properties of the wall as follows:

#### 29.10.1.1 Wall Type (U.S. Version)

Specify the type of wall by selecting from the drop-down menu for Shearwall Type. It can be a Non-Shearwall (not designed as a shearwall) or a Shearwall. A Shearwall can be identified as a Segmented or Perforated wall.

#### 29.10.1.2 Hold-down Configuration (Canada Only)

Specify the location of hold-down *Connections* by choosing one of the following options from the pull-down list:

#### 29.10.1.3 Non-shearwall

The wall is not designed as a shearwall.

#### 29.10.1.4 All segments

Hold-downs are placed at each end of the wall and on either side of every opening, except where there is no net overturning.

#### 29.10.1.5 Ends of shearwall and where required

Hold-downs are placed at each end of the wall, as well as at the end of any shearwall segment where required (based on material selections and anchorage restriction settings).

#### 29.10.1.6 Ends of shearline and where required

Hold-downs are placed at each end of the shearline, as well as at the end of any shearwall segment where required (based on material selections and anchorage restriction settings).

#### 29.10.1.7 Where required only

Hold-downs are placed only where they are required (based on material selections and anchorage restriction settings).

*Note: There is a trade-off between the use of hold-downs and the materials required for shearwall design. For example, designing without hold-downs may require thicker panels and/or a tighter nail spacing to achieve the required resistance.*

Use the **Hold-down Configuration** pull-down in conjunction with the **Anchorage Restriction Settings** (from the Design Tab of the Settings menu) to specify which should take precedence in the design: use of hold-downs or the materials selected for shearwall design. Refer to the on-line help for a detailed description.

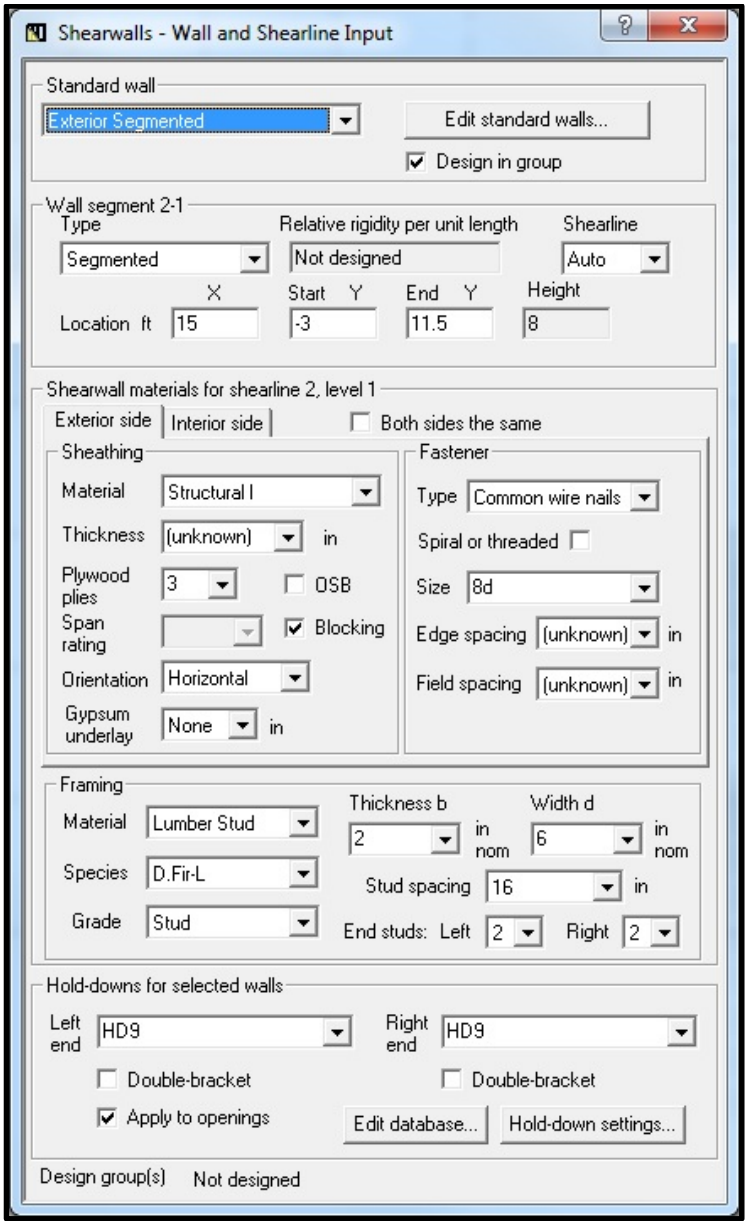



Figure 237: *Shearwall Properties Input Window*

### 29.10.2 Standard Walls

- If you select from the list of *Standard Walls* while a wall is selected, all of the properties for the standard wall are transferred to the selected wall.
- If you change a wall so that it becomes identical to a *Standard Wall*, that *Standard Wall* name will appear. If you make a change so that it is no longer the same as a *Standard Wall*, a blank space will appear.
- Press the "*Edit Standard Walls…*" button in order to add, delete or modify standard walls. Existing walls will not be affected by changes made to *Standard walls* other than to cause the *Standard Wall* field to be cleared.

### 29.10.3 Wall Dimensions

One Can:

- Change the location of the wall. This has the same effect as moving the wall graphically, as described under 'Moving Walls', earlier in this section.
- Change the start or end of the wall, with the same impact as resizing the wall graphically.

### 29.10.4 Wall Materials

To change the material properties of the wall:

- First you must specify which sheathing surface the changes apply to: *exterior*, *interior*, or *both sides*. Selecting *Exterior Only* indicates that there is no interior sheathing.
- Next, specify the type of sheathing material. This causes a new list of choices for sheathing thickness to appear. These thicknesses are the ones that yield differences in design strengths (U.S. only). If you are using a slightly different thickness, just select the next available smaller thickness from the list. You may also specify "*Unknown*" and the program will design a thickness for you. You can also specify the orientation of application.
- Now select *Fastener*, *Type*, *Size*, and the *Nail Spacing* for the edges and for the interior of the sheathing panel. Choosing "Unknown" for edge spacing allows *Shearwalls* to determine this value during the design process.
- Finally, specify the *framing details*, *material*, *species* and *spacing* of the lumber studs, and whether there is *blocking* between them. Stud Spacing may be "Unknown"

### 29.10.5 Multiple Walls

- To change the materials or wall type of several walls at once, select the walls in *Plan View* using the **CTRL** key, or from the *Edit* menu or by right clicking, choose **Select All** to highlight all walls on the level.
- If not all of the selected walls share a certain property, that field on the input form will become blank.
- Any changes to the properties will now affect all the selected walls.

### 29.10.6 Hold-Downs for Selected Walls

Select the type of hold-down for the selected wall. You can also access the hold-down database to create new hold-downs to be used in the design and the hold-down settings window.

Selecting ***Apply to openings*** will specify these same hold downs for each window and door opening in the selected wall(s). Selecting a ***Double bracket*** means that there is a hold down above and below the bottom of the wall, typical for floors above the first level, and will double the hold down elongation/displacement for the purpose of calculating deflection.

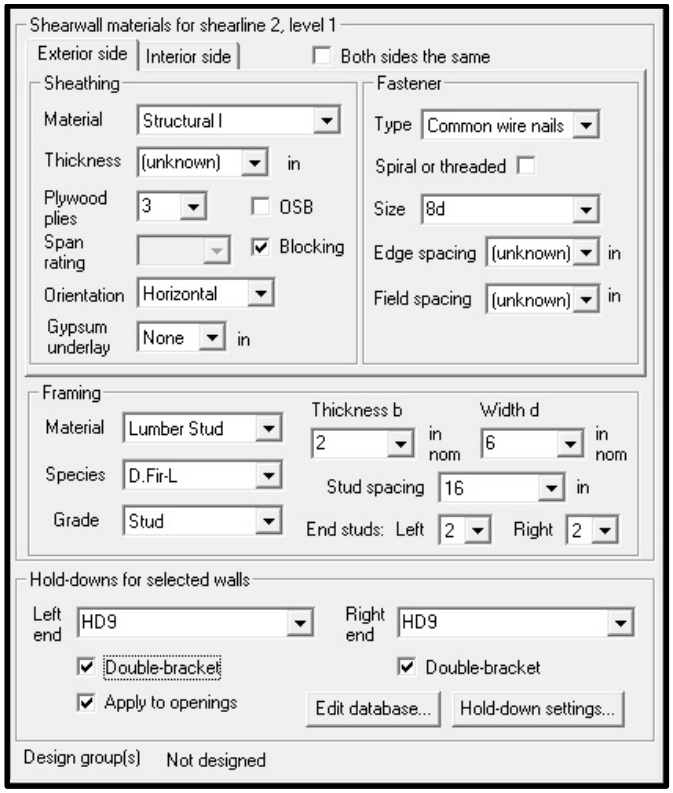



Figure 238: *Selecting Hold-downs for a Wall Segment*

## 29.11 Step 6: Creating Openings

1. Click on the ***Openings*** button. The input form will display information about the openings (windows and doors) in the currently selected wall.
2. In ***Plan View,*** you can draw any number of openings in a selected wall by dragging a line from one point on a wall to another.
   - The opening will appear as a thinner line than the wall.
   - The opening will initially have a height and an offset from the bottom as specified in the ***Settings/Design***.
3. To change the location or the dimensions of an opening:
   - Select the opening number in the dropdown box at the top of the form. The current dimensions will appear in the form.
   - Type over the dimension you wish to change and press **Enter**.
   - *Note that openings cannot be altered graphically after they have been created: they must be resized or relocated using the Form View.*
4. It is possible to create a new opening without using the *Plan View*.
   - Select ***"new opening"*** in the **Openings** drop-down list, enter the dimensions and location of the opening, and then press the ***Add*** button.

- You can add a succession of similarly sized openings in this way.

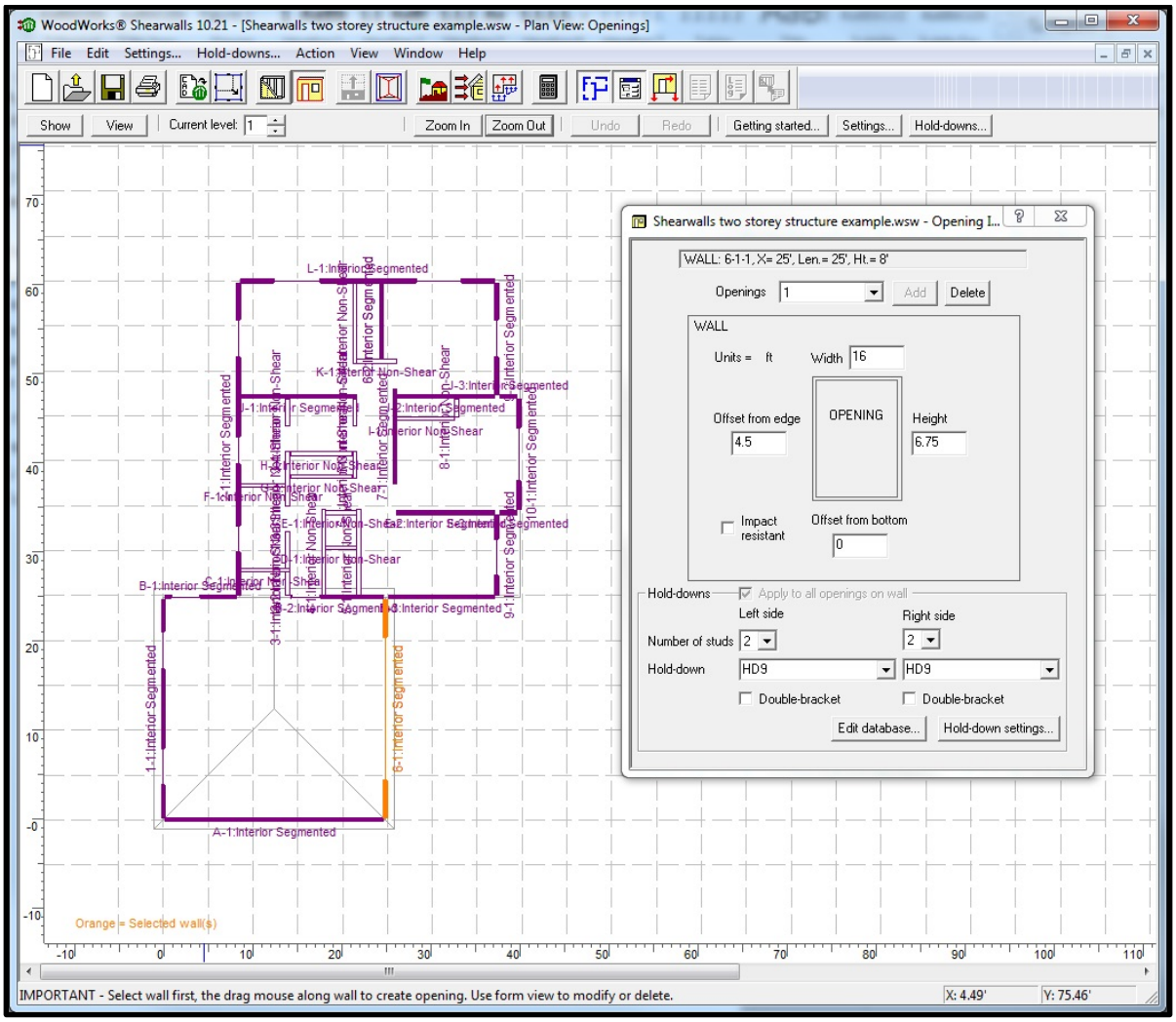



Figure 239: *Creating Openings*

5. Deleting openings:

   - To delete an opening, click on a wall, select an opening from the **Openings** drop-down list, then press the ***Delete*** button.

6. Hold downs are specified for each side of each window and door opening. If the *Apply to all openings on wall* is selected, the choice of hold down will apply to all the openings along the selected wall. Unchecked, unique hold downs can be specified for each opening. By default, this option is disabled, and checked, so that a wall with multiple openings have consistent hold downs. To enable the checkbox, the "*Apply to openings*" in the ***Wall and Shearline Input*** form needs to be unchecked. To reset the hold downs entered in the ***Openings Input*** form to match those entered for the wall, recheck the "*Apply to openings*" or choose a hold down for the left and right end of the selected walls in the ***Wall and Shearline Input*** form. The ***left end*** in the *wall and shearline input* corresponds to the "***left side***" in the ***Opening Input*** form. It would be more typical to specify a consistent hold down for the entire wall system, perhaps even the entire building.

## 29.12 Step 7: Extending Walls

The last step in creating levels is to extend the first floor upwards once the elevations have been specified and walls and openings have been created for the first level.

1. When in *Walls* action or *Openings* action, click on the **Extend Walls** button. All exterior walls defined for the first level will be copied to all levels above for all blocks up to the maximum number of levels specified for each block.
2. *Note that the user may choose to extend the levels after modeling the first floor for only those elements that are common to all levels.*
3. The *Extend Walls* button can only be applied once

## 29.13 Step 8: Creating Roofs

*Roof Action* becomes available for single story buildings when the *Wall Action* button is pressed and walls are created, or for multi-story buildings when the *Extend Walls Upwards* button is pressed. You must press on roof action before proceeding to create loads.

Click on the ***Roof Blocks*** button to specify the building roof geometry. A Roof Block is created for each structure Block. Each Structure *Block* will be assigned an initial roof geometry based on the construction type, slope and overhang settings in ***Settings/Default Values.***

After the *Roof Blocks* button is pressed for the first time, the exterior walls remain associated with the blocks. If these walls are repositioned, the roof block is resized to match their furthest extent. However, the opposite is not true, it is also possible to create roof blocks that are not associated with any walls by dragging them on the screen while in *Roof View*.

Similarly, if the exterior walls associated with a particular block are repositioned, the roof block (and structure block) will adjust itself to match the furthest extents of the exterior walls. The opposite is not true.

If only one block was used to create the structure, only one roof block will be generated in *Roof View*, although it is possible to add more roof blocks to achieve the shape of your roof by clicking the left mouse key and dragging a box on the existing roof block. The new roof block must intersect with an existing roof block, and once a roof block has been created the block can be adjusted accordingly.

### 29.13.1 Adjusting Roof Blocks

The Roof Input Form controls the editing of roof blocks to define the roof blocks location and geometry.

1. Select a Roof Block by clicking on it in *Plan View*, or by choosing it in the *Block* pull-down in the *Roof Input Form*. Selected blocks are highlighted in red.
2. Resize or reposition the block by clicking on a handle and dragging it, or by changing the location or extents that appear in the *Form View* for *Roof Input*.
3. Define the *roof construction*, *ridge direction*, *roof slopes*, *ridge elevation*, *plan location* and *size of overhangs* by editing the appropriate fields in *Form View*.
4. To delete a roof block, select a block and then press the *Delete* button.

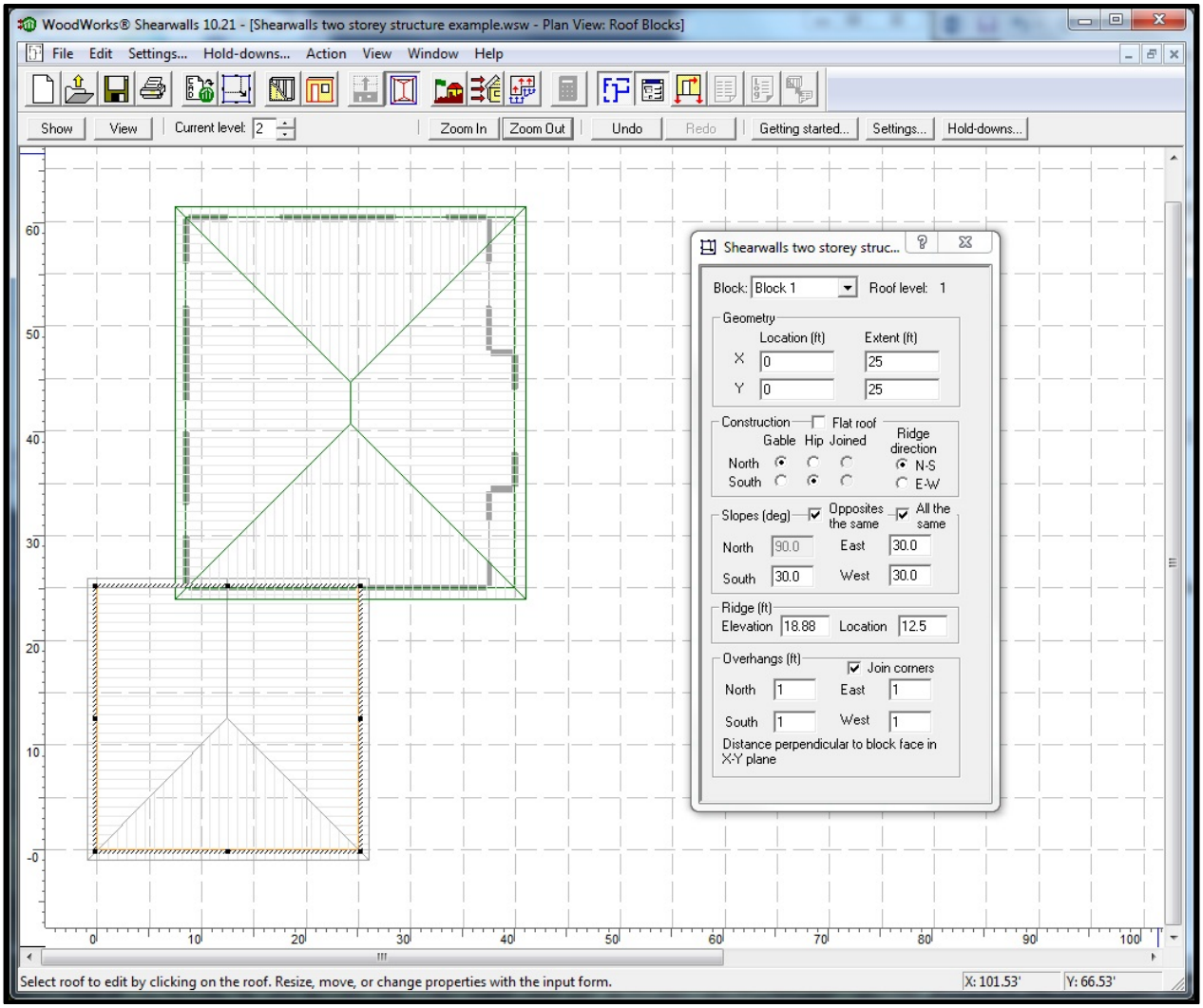



Figure 240: *Creating and Adjusting Roof Blocks*

## 29.14 Step 9: Specifying Site Information

The ***Building Site*** information pertains to characteristics of the building such as period, enclosure, exposure, and occupancy; and climatologic, topographic, seismologic and soil characteristics of the site. The site information is used for the automatic generation of wind and seismic loads.

The initial building site information is based on the settings in **Settings/Default Values** and **Settings/Design.**

The ***Building Site*** dialog box can be accessed while in any of the Views and Actions, and after entering the data you return to the previous state.

### 29.14.1 Occupancy / Importance Category (U.S. / Canadian)

The building *Occupancy* (U.S.) */ Importance category* (Canadian) is selected based on the nomenclature used in the building code selected for design.

### 29.14.2 Wind & Seismic Load Generation

Depending on the design method standard selected for wind generation (refer to ***Settings***), the input fields for wind load generation will vary and certain fields may become inactive. Refer back to section 1.2 ***Design Methods*** for more information

Refer to the ***Help*** for more information on *Terrain* (Canadian)/ *Speed up over hills* (U.S.), *Hurricane prone regions* (U.S.), and *Dynamic Analysis (flexible buildings)* (U.S., wind only), as well as other specific input criteria for wind and seismic generation.

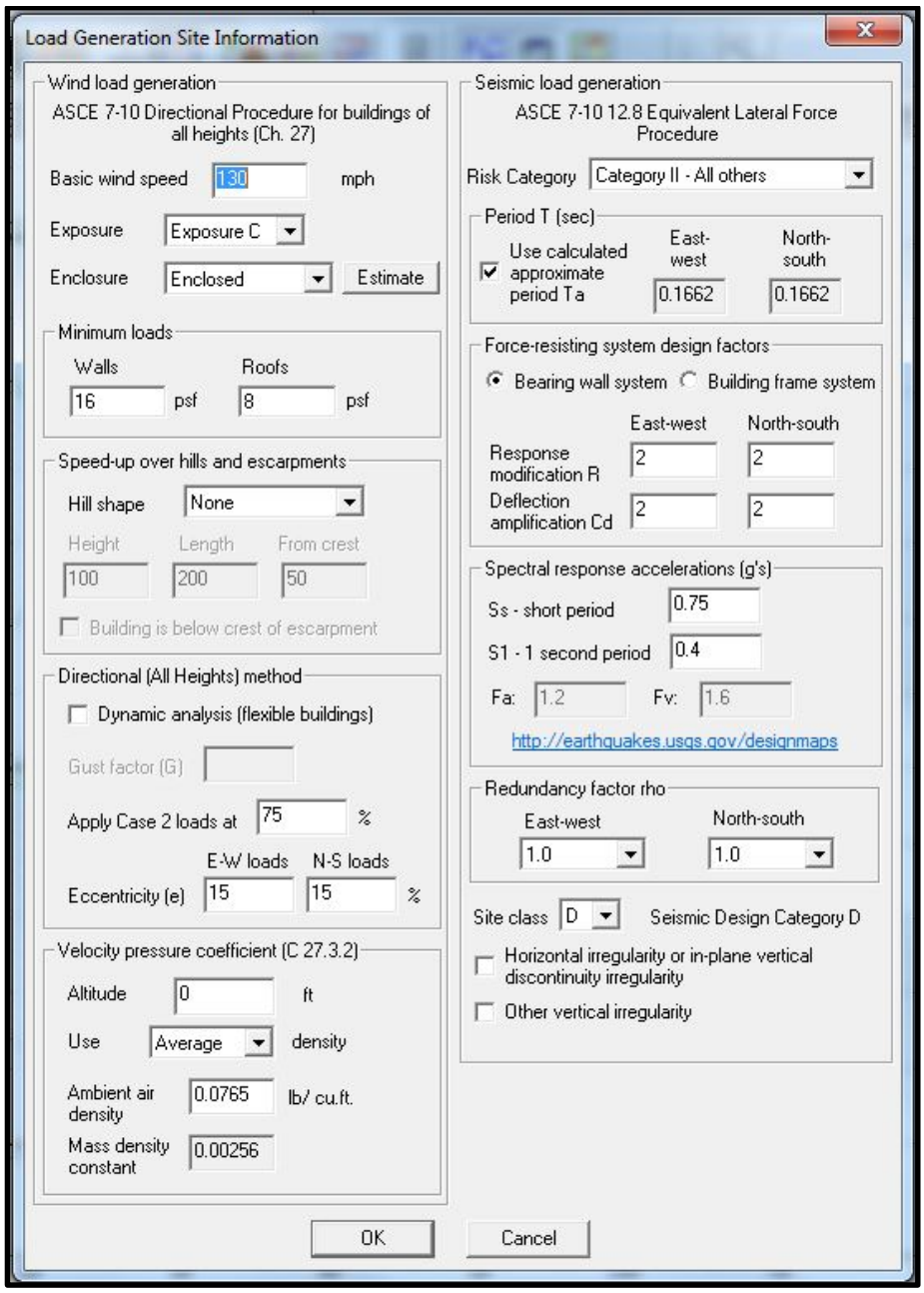



Figure 241: *Load Generation Site Information (U.S. Version)*

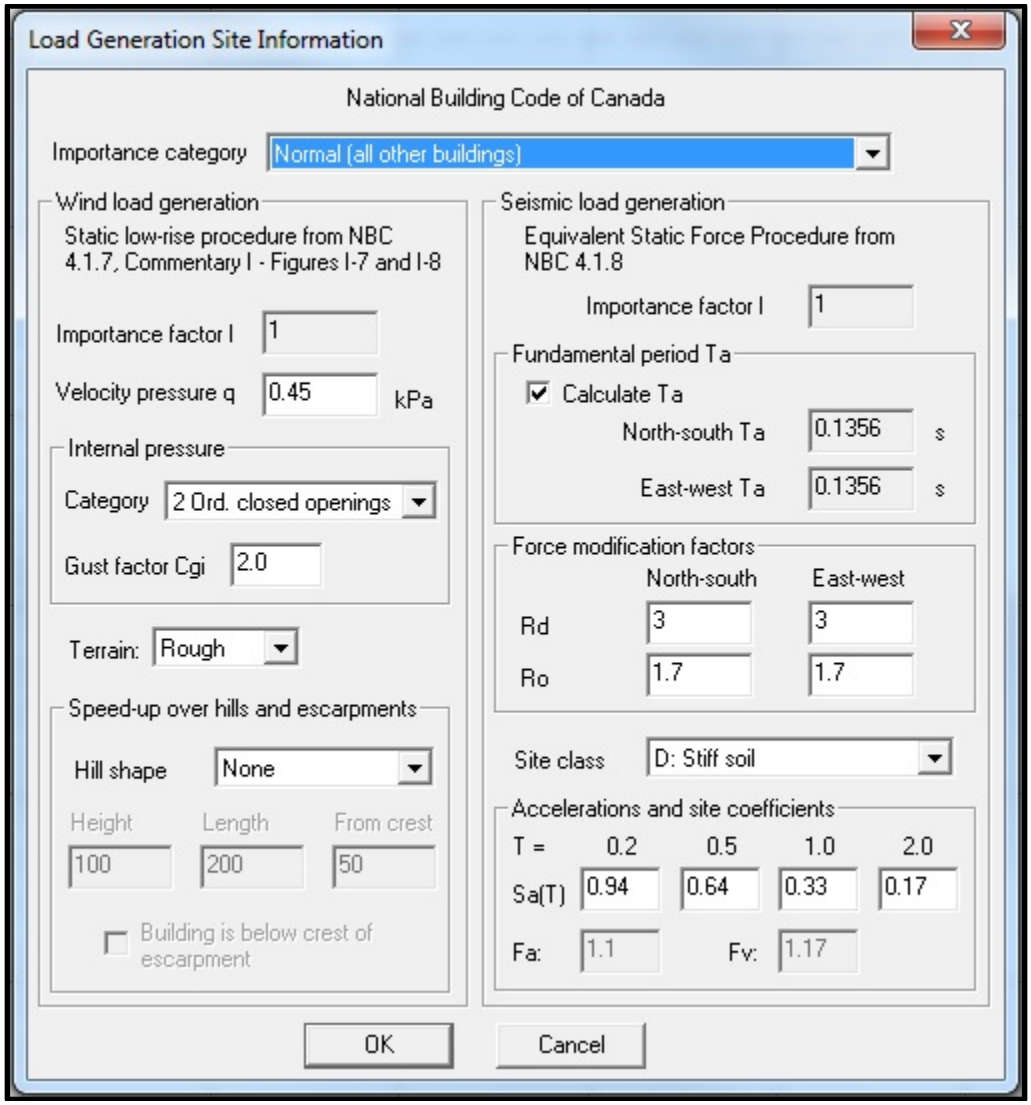



Figure 242: *Load Generation Site Information (Canadian)*

*<u>Note</u>: The Dynamic Analysis (flexible buildings) allows a manually calculated entry of Gust factor suitable for flexible buildings. However, it would be highly unusual that any wood structure, even midrise, to be considered a "flexible" building. "Flexible" buildings are typically defined as being slender with a fundamental natural frequency of less than 1Hz (that is, a period of greater than 1 second). Almost all wood structures are considered "rigid", with a period of less than 1 second. The fundamental (Canadian)/approximate (US) period Ta shown in the Load Generation Site Information input will help determine if the designed building can be considered "Rigid" or "Flexible". Also note that the determination of a "flexible" or "rigid" building is not related to the distribution of loads based on a "flexible" or "rigid" diaphragm.*

## 29.15 Step 10: Automatically Generating Loads

Click on the ***Generate Loads*** button to generate wind and seismic loads. Wind or seismic loads can be generated separately or at the same time by selecting the appropriate type of loads to generate. Loads can be generated on the entire building, on a range of levels or on a single level by choosing the appropriate levels in the *Generate Loads* form.

### 29.15.1 Wind Loads

Use the wind direction pull-down to generate wind loads for all directions at once or for a single direction only. Similarly, use the Building Face pull-down to generate loads on different building faces.

MWFRS (Main Wind Force Resisting System) loads can be generated as either line loads or area loads. Line loads is the default, but both produce the same results.

Loads can be generated individually for walls, roofs and gable ends. Gable ends are considered as the portion of the wall at the gable end that is above the eave. The portion of the wall below the eave is considered in the Walls selection. C&C (components & cladding) wall loads can also be generated.

In the US, WoodWorks® allows you to select the option of generating a 10 psf minimum wind pressure. In the All-heights method, the 10 psf minimum option is selected automatically and the generated pressures will be the higher of the "wind speed" generated loads and the 10 psf pressure. In the Low-Rise method, the 10 psf minimum pressure is not selected as default and must be analyzed separately from the "wind speed" generated cases by selecting and deselecting the 10 psf minimum option.

For Rigid Diaphragm Analysis using the All-heights method, the ***Settings*** form allows you to select between two separate cases: full wind pressures without torsion and reduced wind pressures with torsional effects. This analysis requires separate runs and must be compared to determine the worst case. Refer to Section B.4 for further information on *Shearwalls* Settings.

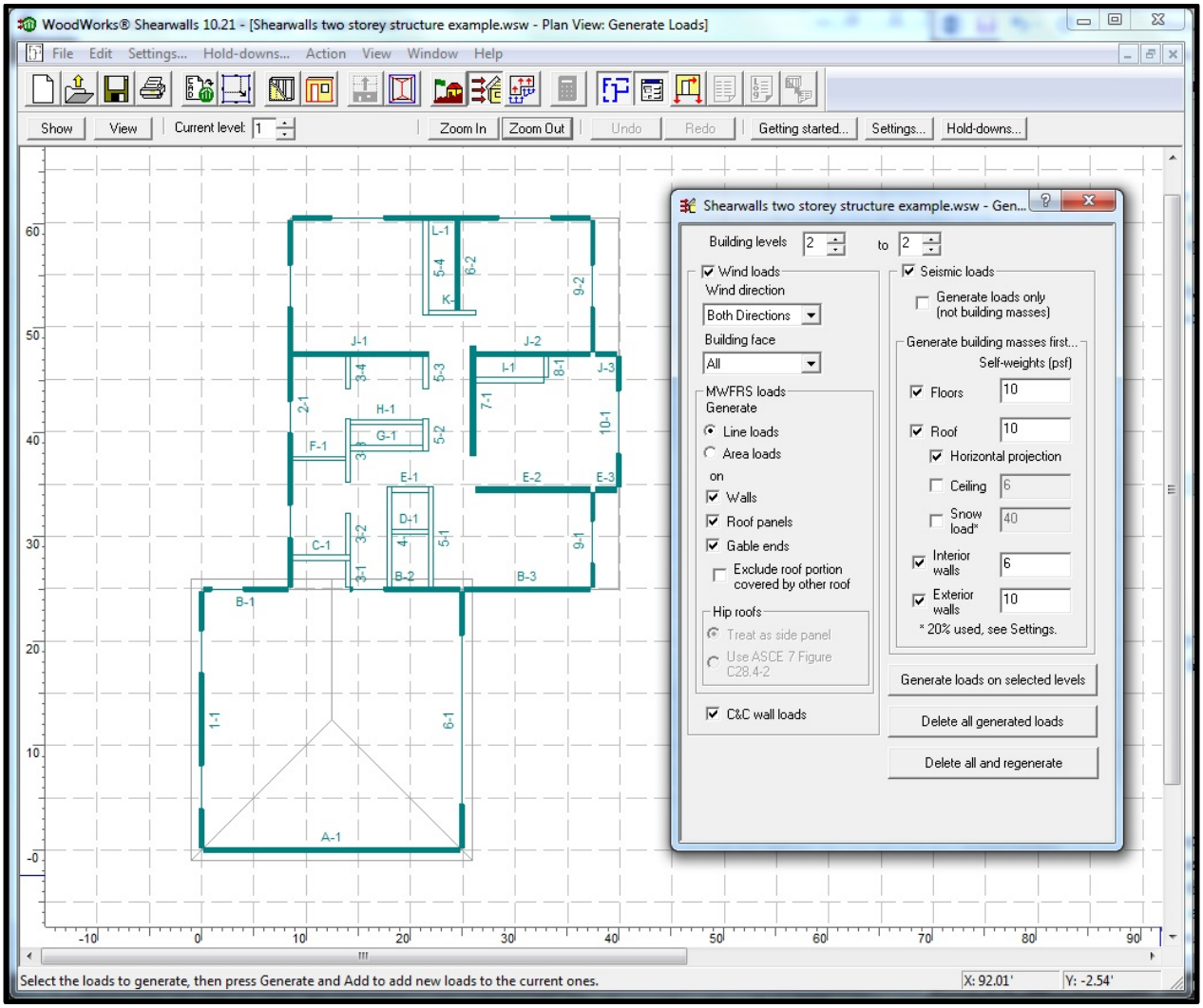



Figure 243: *Generating Loads*

### 29.15.2 Seismic Loads

The loads generated correspond to building masses that are also generated by the program. Check the box for each building element - walls, roofs, floors and ceilings, that you want the masses to be generated for, and specify the area self-weight of these elements. The self-weight specified for the roof can be input as acting along a horizontal projection of the roof or along the surface of the roof.

The ***Generate Loads*** button allows you to use different self-weights for different levels, by selecting each level in turn, changing the self-weight and generating the loads for that level.

The initial self-weights used to generate seismic building masses are based on the values in **Settings/Default Values.**

Refer to Section 31 of the user guide for further information on *Shearwalls* Settings.

#### 29.15.2.1 Use wall self-weights to generate wall dead loads for $J_{hd}$ calculations (Canadian Version)

Activate this checkbox to include the self-weight of walls to calculate wall dead loads that are considered to resist overturning. *Note that the self-weight of floors, ceilings and the roof, entered in the Generate Loads form, are only used to generate building masses which are used to calculate seismic loads – they are not considered to resist overturning.*

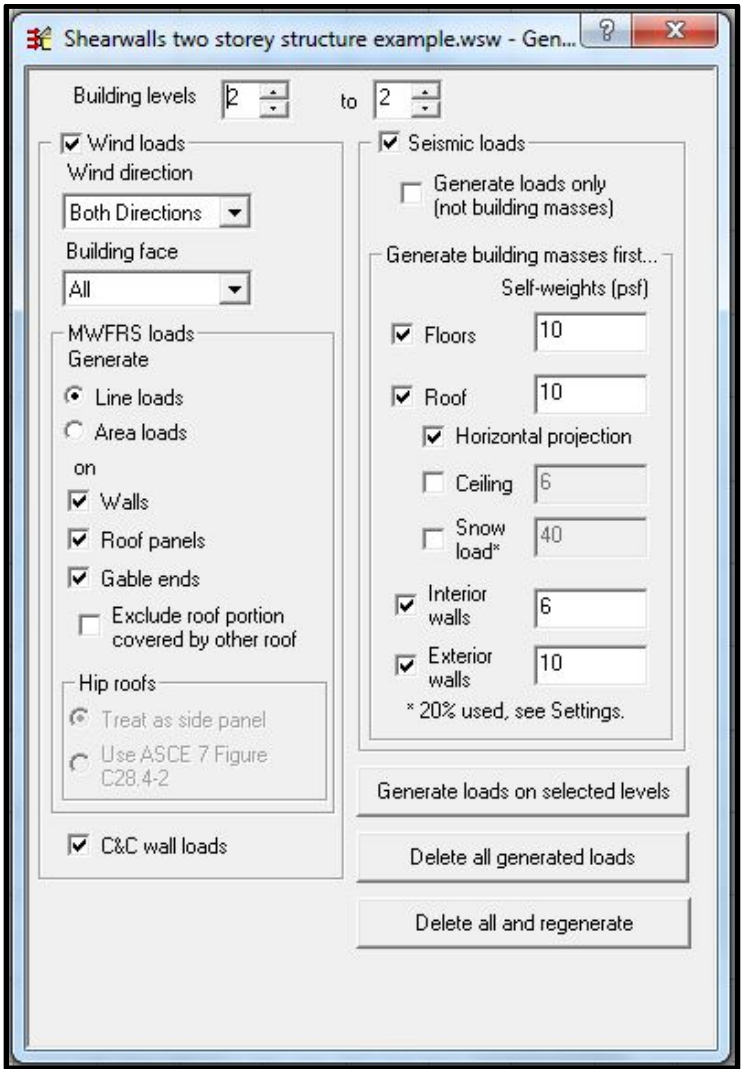



Figure 244: *Generating Loads Input Window*

### 29.16 Step 11: Manually Applying & Modifying Loads

Click on the **Loads and Forces** button to add a variety of load types and profiles to the structure, as described below in the *Load Input Form* section. *Shearwalls* has the flexibility to add loads to one or several levels at once, to individual walls or entire building faces.

### 29.16.1 Load Input Form

This form controls the adding, editing and deleting of loads and forces that can be applied to the structure.

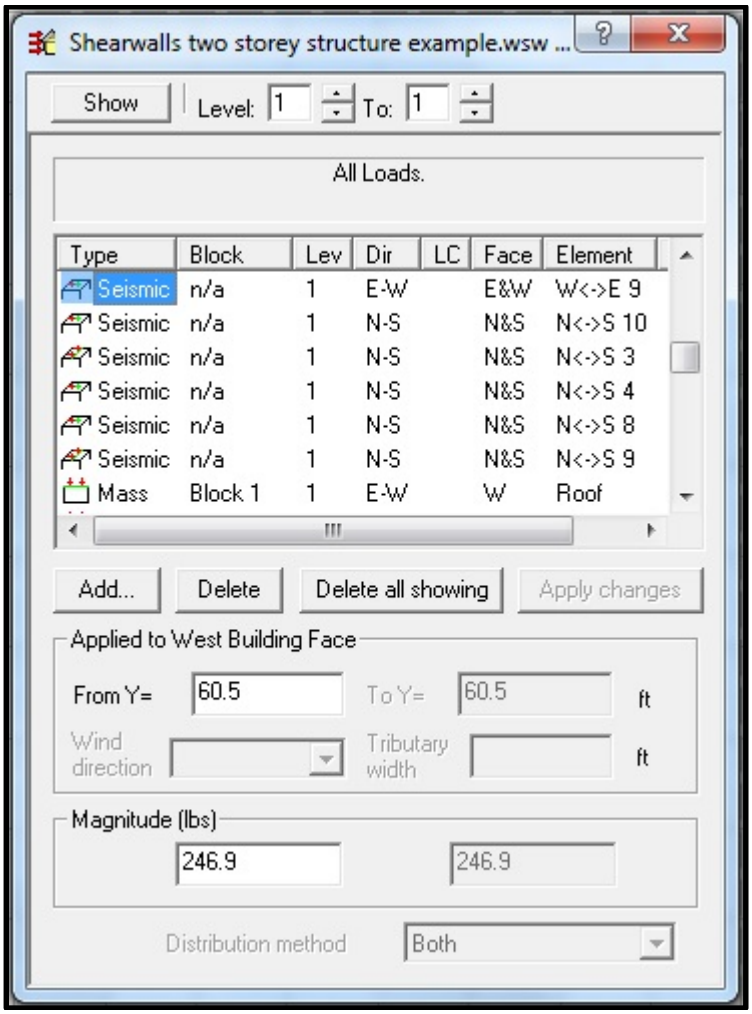



Figure 245: *Load Input Window*

The loads list can be controlled to only show certain categories of loads:

- ***Show… Button*** – Show loads according to their type (Seismic, Wind Shear, Wind Uplift, Dead, C & C).
- ***Level Control*** – Show loads according to the level on which they are applied. This can be done for one or a range of levels.
- ***Selected Building Face*** – Shows loads that are only applied to the selected building face in *Plan View*.

### 29.16.2 Load Types

*Shearwalls* accounts for the following load types:

- ***Seismic*** – Earthquake generated loads restricted by the lateral load resisting system.
- ***Wind Shear*** – External wind loads resisted by the lateral load resisting system.
- ***Wind C&C*** – Wind suction effects on the Components and Cladding for exterior walls only (roofs not included). This includes the bending of sheathing between studs and the withdrawal of nails fastening the sheathing to the wall studs.

- ***Wind Uplift*** – Overturning loads, especially wind uplift load transferred from the roof to supporting walls on the top story.
- ***Building Mass*** – Weight of a building attributable to the diaphragm that transfers loads to the specified level. This weight is used for automatic generation of seismic loads (see Section 2.9) only, and should include all material self-weights along with any roof snow loads or permanent dead loads required by the building code. Building Mass is not considered to resist overturning.

### 29.16.3 Load Profiles

A variety of load profiles can be applied to the structure, including point loads, line loads and area loads. Loads can be applied to one or more floor levels.

### 29.16.4 Load Location

Loads are applied to building faces, wall lines, or selected walls. The location and tributary area can be changed to create partial loads or overhanging loads. Wind shear loads can be applied as windward and/or leeward loads. Highlight a wall in *Plan View* to add corresponding loads.

### 29.16.5 Magnitudes

Load magnitudes represent the intensity of the load applied to the exterior of the structure. These loads are based on the code requirements for your jurisdiction.

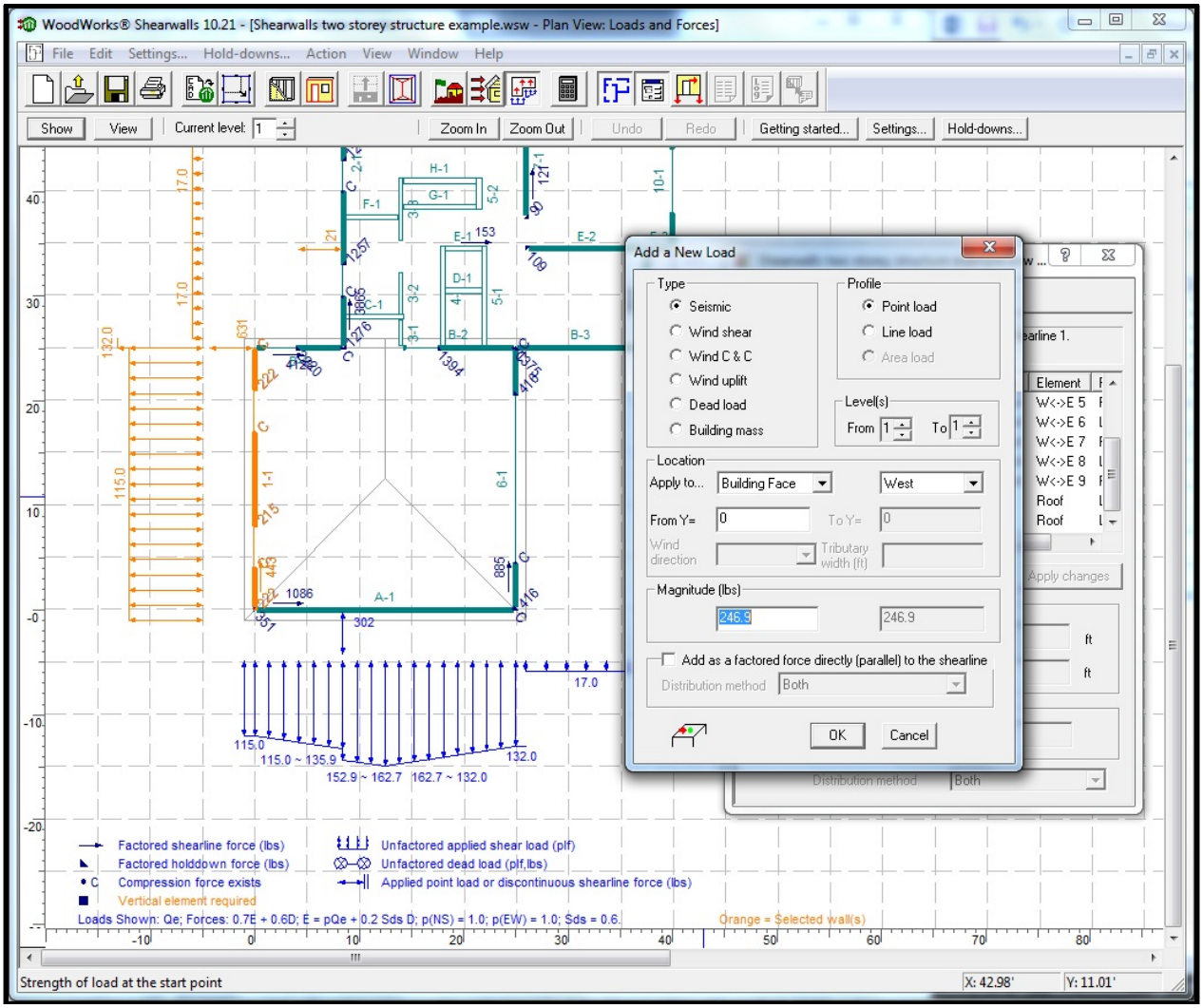



Figure 246: *Applying Loads Manually*

## 29.16.6 Working With Loads

### 29.16.6.1 Adding Loads

- Click on the **Loads Action** button to input loads through the ***Load Input Form***.
- In the Load Input Form select ***Add***… to add loads to the structure. The ***Add a New Load*** input dialog will appear.
- Select the load type, profile, location and floors to which the load applies.
- The magnitude is entered as a From and To value to facilitate trapezoidal and triangular line loads.
- Area loads require a tributary width.
- Click **OK** to add the new load to the list of loads in the *Load Input Form*.

#### 29.16.6.2 Editing Loads

- Select the load to be edited from the *Load Input Form.*
- Edit the load location or magnitude information as required and click on the **Apply Changes** button. The load type or profile cannot be changed.

#### 29.16.6.3 Deleting Loads

- Select the load to be deleted from the *Load Input Form.*
- Click on the **Delete** button.
- To delete all loads, click on **Delete all showing** button.

To edit or delete individual loads, select the wall in *Plan View* to which the loads are applied. The loads list in the Load Input Form will now display only those loads applied to the selected building face.

*Note: After editing generated loads, you must perform a design before you regenerate the loads, otherwise your changes will be lost when the new loads are generated.*

### 29.17 Step 12: Generating Shearline Forces

For the Flexible diaphragm analysis method, *Shearwalls* automatically distributes loads to the shearlines each time a new load is added. It also adjusts the distribution of forces if changes are made to the building's walls.

For the *Rigid* diaphragm analysis method, loads will be distributed to the shearlines once the *Design* button is pressed after creating or adjusting loads or making changes to the building's walls. The rigid method considers the torsional resistance of walls both parallel and perpendicular to the applied load, therefore all walls in all directions must be loaded in order to perform a rigid diaphragm analysis.

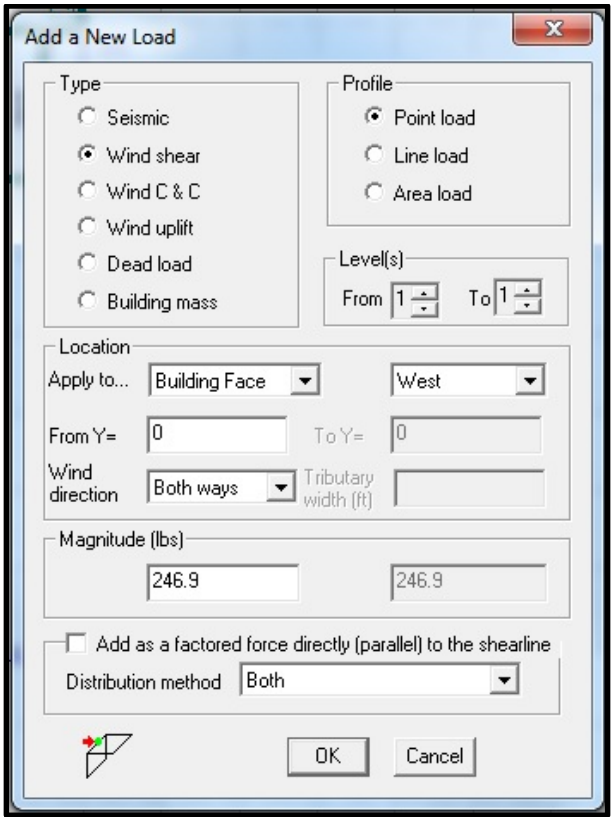



Figure 247: *Add New Load Input*

### 29.17.1 Direct Applied Shearline Forces

As an advanced feature, *Shearwalls* allows forces to be directly applied to shearlines as a manual method of adjusting the load distribution.

A force magnitude can be entered as either a positive or negative value to facilitate redistribution of the loads by adding forces to some shearlines and subtracting forces from others (The load distribution method can be Flexible Diaphragm, Rigid Diaphragm or both. Most often, both methods will be selected).

These direct forces are added in the same manner as for new loads:

- Click on the **Loads** button.
- Select or highlight in *Plan View* the wall in the shearline that the force applies to.
- Click on the **Add…** button and select **Implement as a Force Applied Directly.**
- Specify the force direction in the case of wind, enter the magnitude and select the load distribution method to which the force applies

Direct forces are not shown in the plan or elevation views as applied loads. Rather, the results of applying these loads are shown as adjusted shearline design forces in both the *Plan* and *Elevation Views*. Direct forces appear in the *Results View*.

*There is a lot of additional information in **Help** pertaining to generation and distribution of loads, and the related engineering assumptions used in the software.*

## 29.18 Step 13: Viewing Loads, Forces & Building Masses

Applied loads and corresponding forces can be viewed in the *Plan* and *Elevation Views*.
First click on the **Show Button** to select the loads/forces to be viewed. Viewing is made easier if wall names are turned off.
Forces in *Plan* and *Elevation Views* can be shown based on either the *Flexible* or *Rigid* diaphragm analysis methods for the distribution of lateral loads to shearlines. A torsional analysis is performed for the rigid method.
*Note: Loads appearing are the sum of all loads of a similar type that have been applied to the building face at the current level.*

### 29.18.1 Plan View

Both loads and forces can be viewed in the Loads and Forces view, while forces cannot be viewed in the Generate Loads view.

- A **seismic load** is distributed (point/ uniform/ trapezoidal) over the length of an exterior wall as horizontally applied double-headed arrows with specified magnitude. The corresponding horizontal shearline forces appear as double-headed arrows with calculated magnitude shown at mid-length of the affected shearlines. The calculated magnitudes of the hold-down forces are shown at the end of each wall segment.
- A *wind shear load* is distributed (point/ uniform/ trapezoidal) over the length of an exterior wall as horizontally applied single-headed arrows with specified magnitude. The corresponding horizontal shearline forces appear as single-headed arrows with calculated magnitude shown at one end of the affected shearlines. The calculated magnitudes of the hold-down forces are shown at the end of each wall segment.

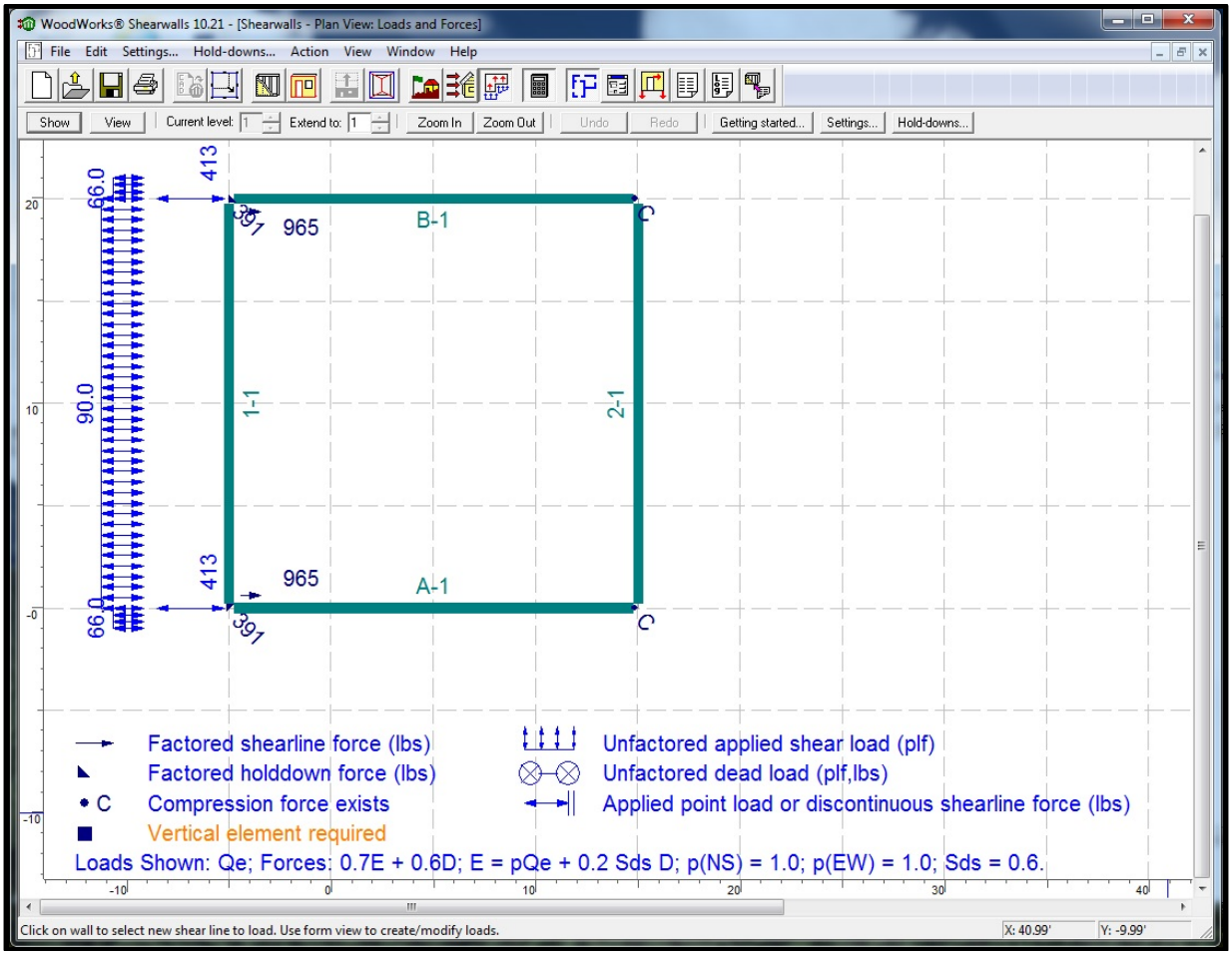



Figure 248: *Seismic Load Distributed Uniformly to a Structure*

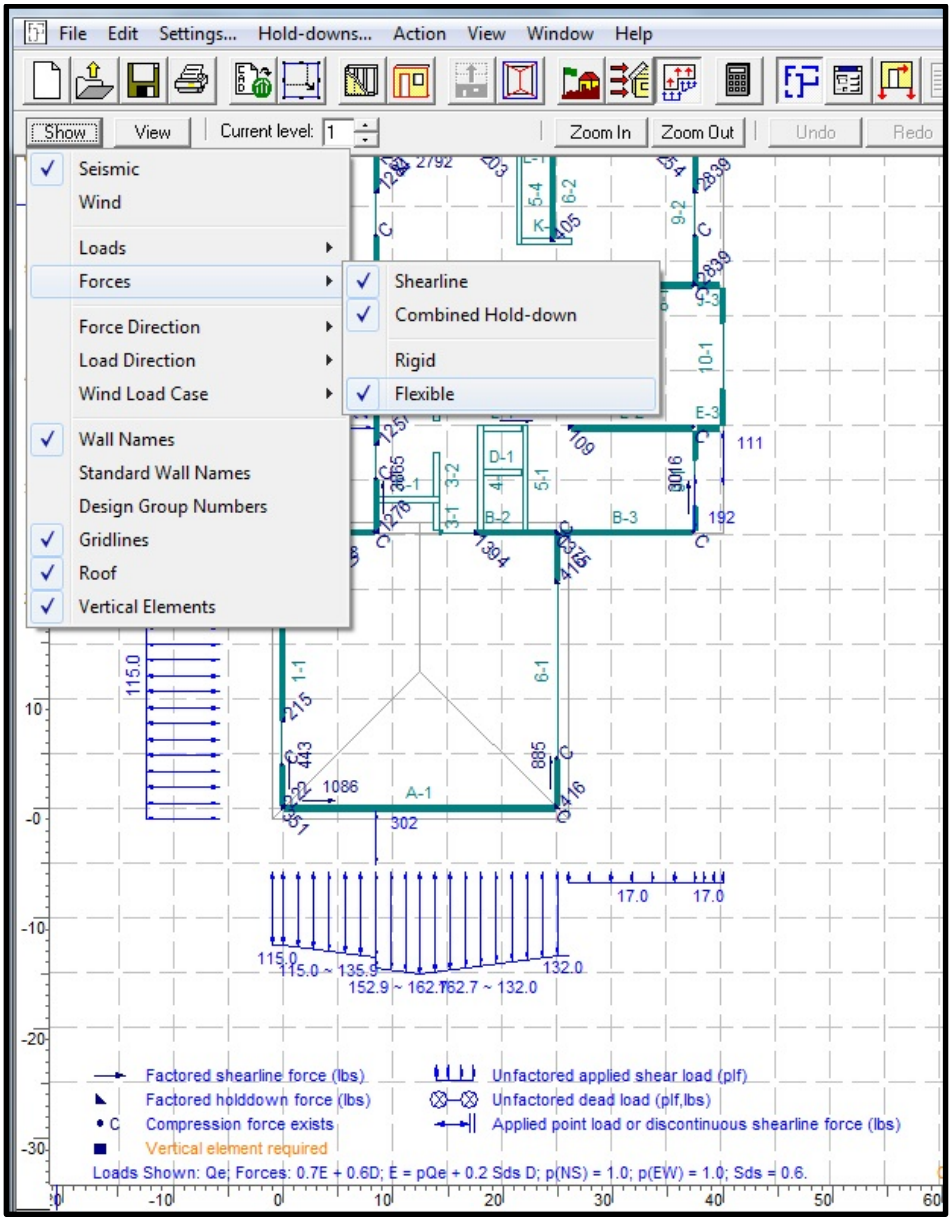



Figure 249: *Show Seismic Forces – Flexible Diaphragm*

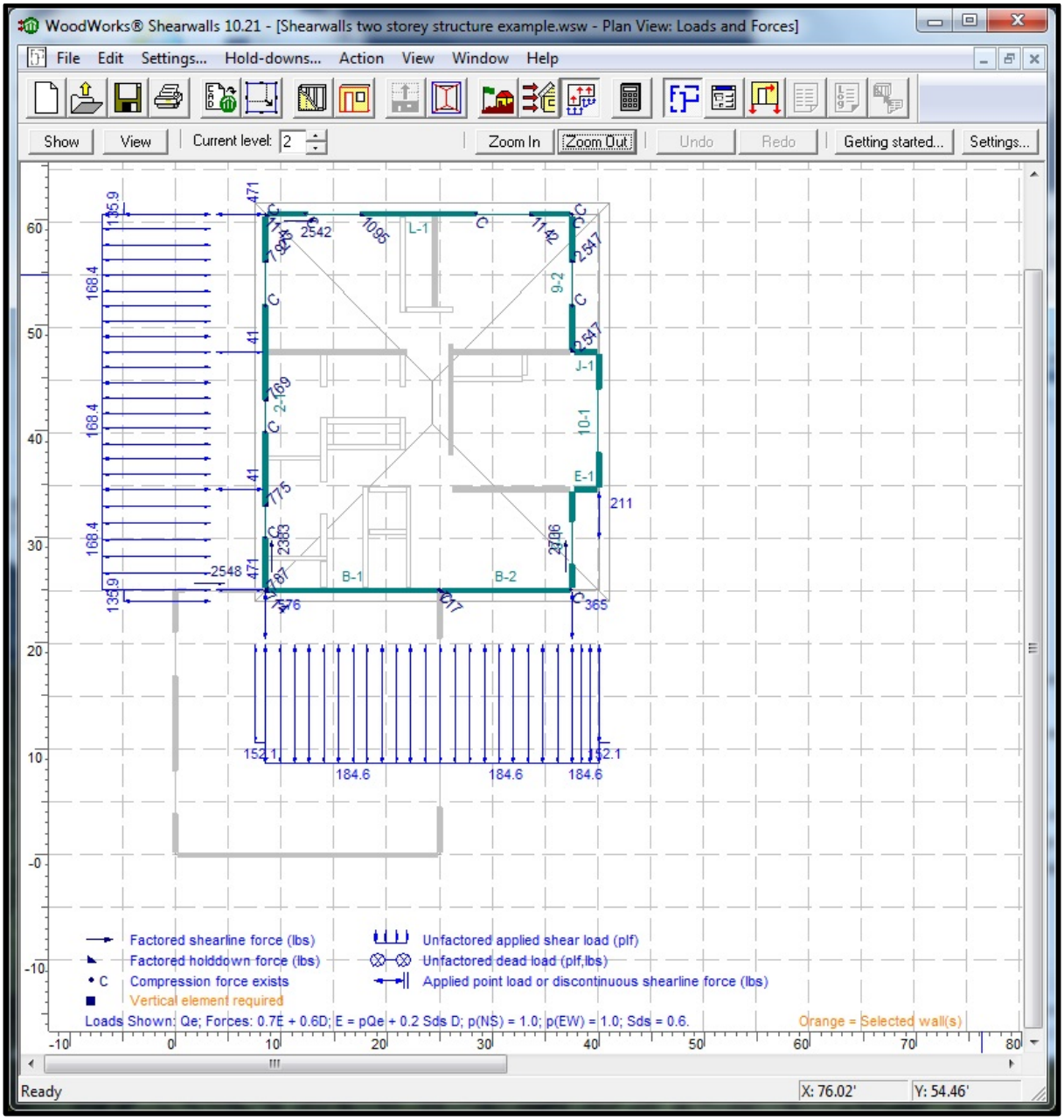



Figure 250: *View Hold-down Forces on a Structure*

### 29.18.2 Elevation View

A *wind and seismic shear* force is shown as horizontal single-headed shear flow arrows distributed along the top and bottom edges of affected shear walls. The total magnitude of the shear force for the shearline is shown, as are the magnitudes of the top and bottom shear flows accounting for openings. Also shown are the magnitudes of horizontal drag strut forces at the top of openings and vertical hold-down forces at the end of wall segments.

- A *wind C&C load* (wind suction on Components and Cladding) is listed beside the affected exterior wall as *Wind C&C Interior Magnitude/ End Zone Magnitude.*

- A *wind uplift load* is indicated by upward-pointing arrowheads distributed along the top edge of the affected wall. The magnitude of the load is shown at mid-length of the wall.
- *Dead loads and building masses* are indicated by downward-pointing arrowheads distributed along the top edge of the affected wall panels and openings. The magnitude of the load is shown at mid-length of the load. Dead loads and building masses cannot be viewed at the same time.
- *Elevation View* can report *hold-down* forces as separate or Combined through the *Show* menu. Separate indicates the individual *shear* (S), *uplift* (U) and *dead load* (D) components, while *Combined* combines all of these affects together along with the appropriate load reduction factors applied.

In the U.S. version, a compression force is indicated at the bottom corners of shearwall segments by downward-pointing arrows, with the magnitude adjacent to the arrows.
A hold- down connection is depicted as two triangles, one on either side of the floor (to represent connector brackets).

- In Canada, an anchorage is depicted in the shape of an uppercase I (to represent a threaded rod with washers and nuts connecting the bottom plate of the upper wall to the top plate of the wall below).
- A *dragstrut force* is indicated by horizontal arrows acting on either side of and at the top of an opening. The magnitude of the force is shown directly under the arrow.

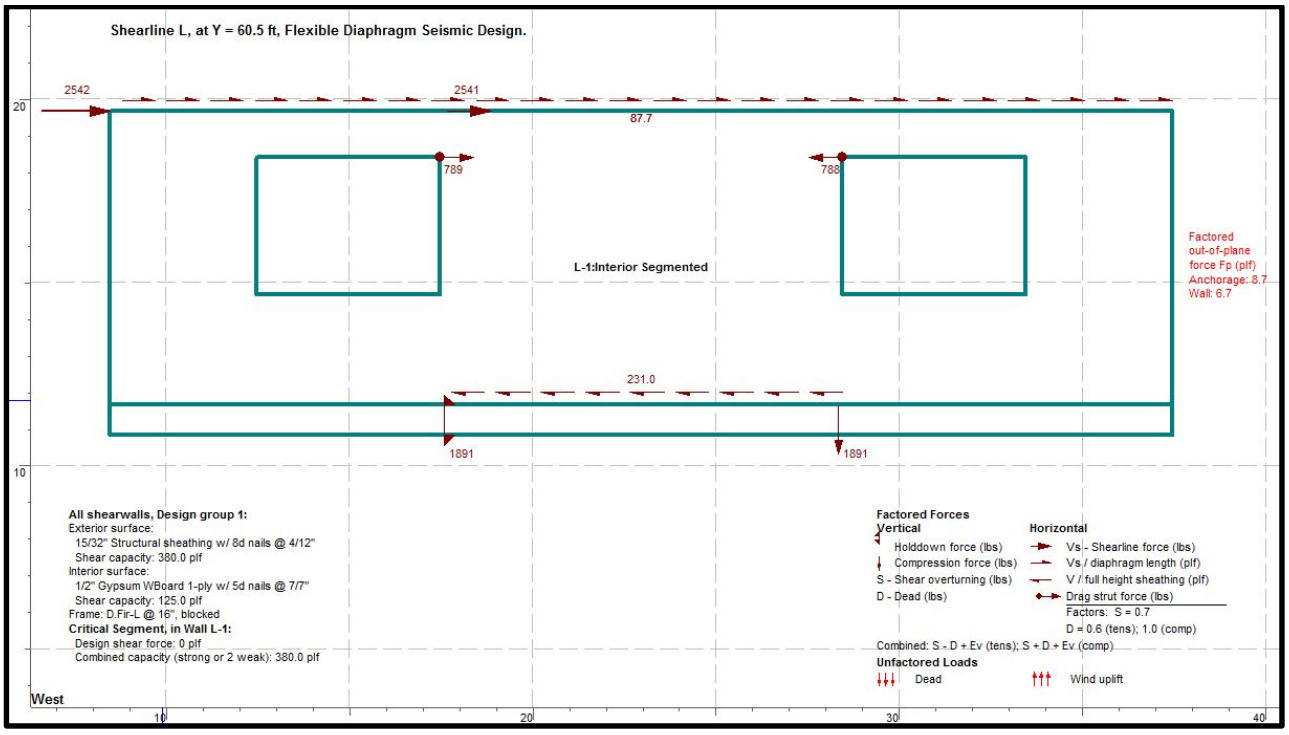



Figure 251: *View Shear Forces in Elevation View*

## 29.19 Step 14: Interpreting Results

The *Results View* produces the design results report for a design run. This view is automatically opened when the user clicks on the *Run Design* action button and generates a new design results report. The report for the last design run is opened when the user directly clicks on the *Results View* action button. In both cases, the design must first be run to activate the Results View button. As well, the design must be re-run to produce an updated results report any time changes are made to the design data.

The design results report is divided into several sections, as described below. The sections which appear depend on the design option and the settings selected through the *Show button* or the *Settings* dialog (both the *Loads and Forces* and the *Option* menu items) boxes.

*There is a lot of additional information in Help pertaining to generation and distribution of loads, and the related engineering assumptions used in the software.*

The basic sections are *Project Information*, *Structural Data*, *Loads*, *Wind Design* and *Seismic Design*.

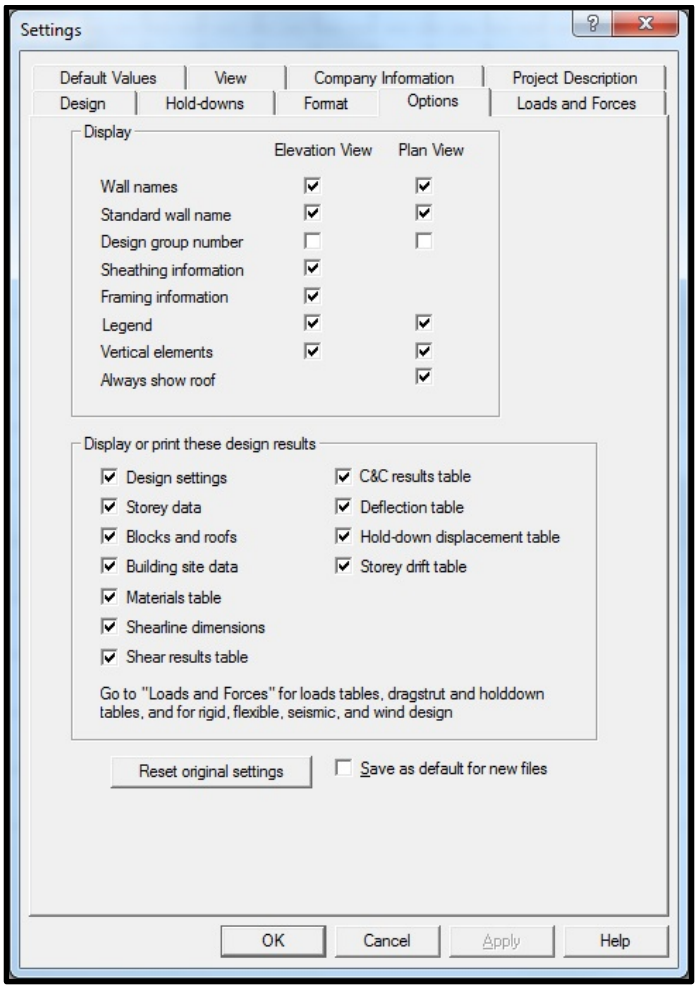



Figure 252: *Settings – Results Display*

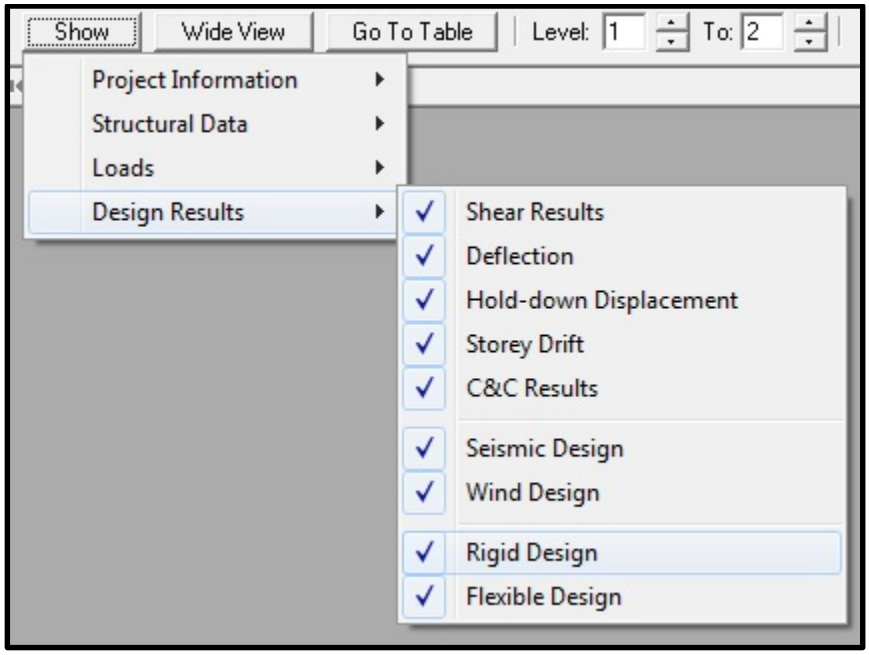



Figure 253: *Show Design Results – Rigid Design*

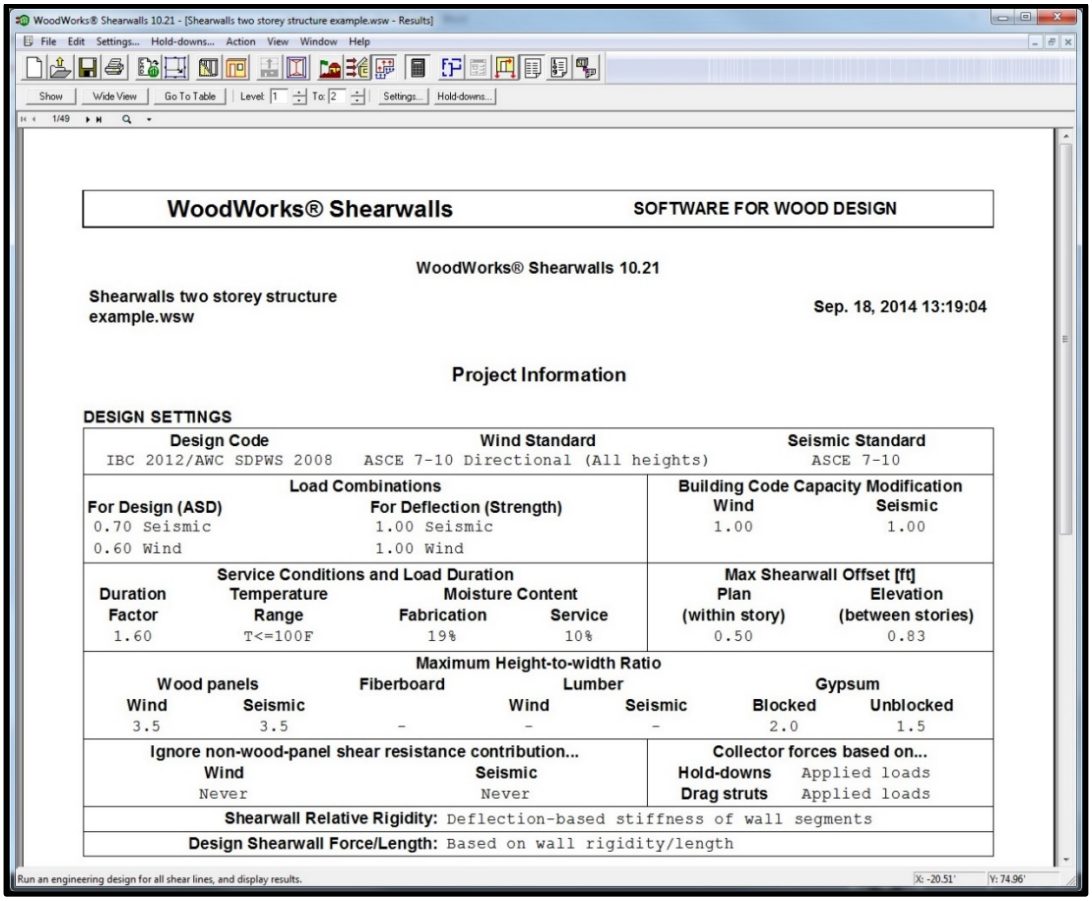



Figure 254: *View Shearwalls Results*

- The *Structural Data* section includes story data, design settings, site information, wall materials by wall groups, and shearline dimensions for the design.
- The Loads section includes the profiles, locations and magnitudes of wind shear, seismic, uplift and C&C loads, and direct forces, specified for the design.
- The *Wind Design* section contains the results of flexible and/or rigid diaphragm design, including shear results by shearline, dragstrut forces, hold-down design and deflection. It also provides component and cladding design results by shearline.
- The *Seismic Design* section contains the results of flexible and/or rigid diaphragm design, including shear results by shearline and dragstrut and hold-down forces.

### 29.19.1 Wall Groups

*Shearwalls* gathers wall assemblies of similar wall materials (sheathing, fastening and framing) into *Wall Groups* and assigns a wall group number to each group. This is summarized in the *Structural Data* section of the design results report.

**WoodWorks® Shearwalls** Shearwalls two storey structure example.wsw Sep. 18, 2014 13:46:02

SHEATHING MATERIALS by WALL GROUP

| Grp | Surf | Material | Sheathing Ratng | Thick in | GU in | Ply | Or | Gvtv lbs/in | Fasteners Size | Type | Df | Eg in | Fd in | Bk | Apply Notes |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 1 | Ext | Structural I | | 3/8 | - | 3 | Horz | 32500 | 8d | Nail | N | 4 | 12 | Y | 3 |
| | Int | Gyp WB 1-ply | | 1/2 | - | - | Horz | 40000 | 5d | Nail | N | 7 | 7 | Y | |
| 2 | Ext | Struct Sh OSB | 32/16 | 3/8 | - | 3 | Horz | 83500 | 8d | Nail | N | 4 | 12 | Y | 3 |
| | Int | Gyp WB 1-ply | | 1/2 | - | - | Horz | 40000 | 5d | Nail | N | 7 | 7 | Y | |
| 3 | Both | Gyp WB 1-ply | 32/16 | 3/8 | - | - | Horz | 40000 | 8d | Nail | N | 4 | 12 | Y | 3 |
| 4 | Both | Gyp WB 1-ply | | 1/2 | - | - | Horz | 40000 | 5d | Nail | N | 7 | 7 | Y | |

*Legend:*
*Grp – Wall Design Group number, used to reference wall in other tables*
*Surf – Exterior or interior surface when applied to exterior wall*
*Ratng – Span rating, see SDPWS Table C4.2.2.2C*
*Thick – Nominal panel thickness*
*GU - Gypsum underlay thickness*
*Ply – Number of plies (or layers) in construction of plywood sheets*
*Or – Orientation of longer dimension of sheathing panels*
*Gvtv – Shear stiffness in lb/in. of depth from SDPWS Tables C4.2.2A-B*
*Type – Fastener type from SDPWS Tables 4.3A-D: Nail – common wire nail for structural panels and lumber, cooler or gypsum wallboard nail for GWB, plasterboard nail for gypsum lath, galvanised nail for gypsum sheathing; Box - box nail; Casing – casing nail; Roof – roofing nail; Screw – drywall screw*
*Size - Common, box, and casing nails: refer to SDPWS Table A1 (casing sizes = box sizes).*
*Gauges: 11 ga = 0.120" x 1-3/4" (gypsum sheathing, 25/32" fiberboard), 1-1/2" (lath & plaster, 1/2" fiberboard); 13 ga plasterboard = 0.92" x 1-1/8".*
*Cooler or gypsum wallboard nail: 5d = .086" x 1-5/8"; 6d = .092" x 1-7/8"; 8d = .113" x 2-3/8"; 6/8d = 6d base ply, 8d face ply for 2-ply GWB.*
*Drywall screws: No. 6, 1-1/4" long.*
*5/8" gypsum sheathing can also use 6d cooler or GWB nail*
*Df – Deformed nails (threaded or spiral), with increased withdrawal capacity*
*Eg – Panel edge fastener spacing*
*Fd – Field spacing interior to panels*
*Bk – Sheathing is nailed to blocking at all panel edges; Y(es) or N(o)*
*Apply Notes – Notes below table legend which apply to sheathing side*

Notes:
3. Shear capacity for current design has been increased to the value for 15/32" sheathing with same nailing because stud spacing is 16" max. or panel orientation is horizontal. See SDPWS T4.3A Note 2.

Figure 255: *Shearwalls – Wall Groups*

Wall group numbers are then referenced throughout the design results output report to avoid repeating the wall assembly materials information at several places in the output.

*The design results can be saved as .pdf and .rtf files. For more information see section 1.3 "**Input and Output**".*

### 29.19.2 Log File

*Shearwalls* creates a detailed log file of the intermediate calculations used to generate wind and seismic loads, and used in the rigid diaphragm analysis. These calculations are stored in a *.log ASCII text file that can be opened and viewed through any text editor, such as Notepad or Word. The log file also includes the definition and value of variables used in the generation of seismic and wind loads.

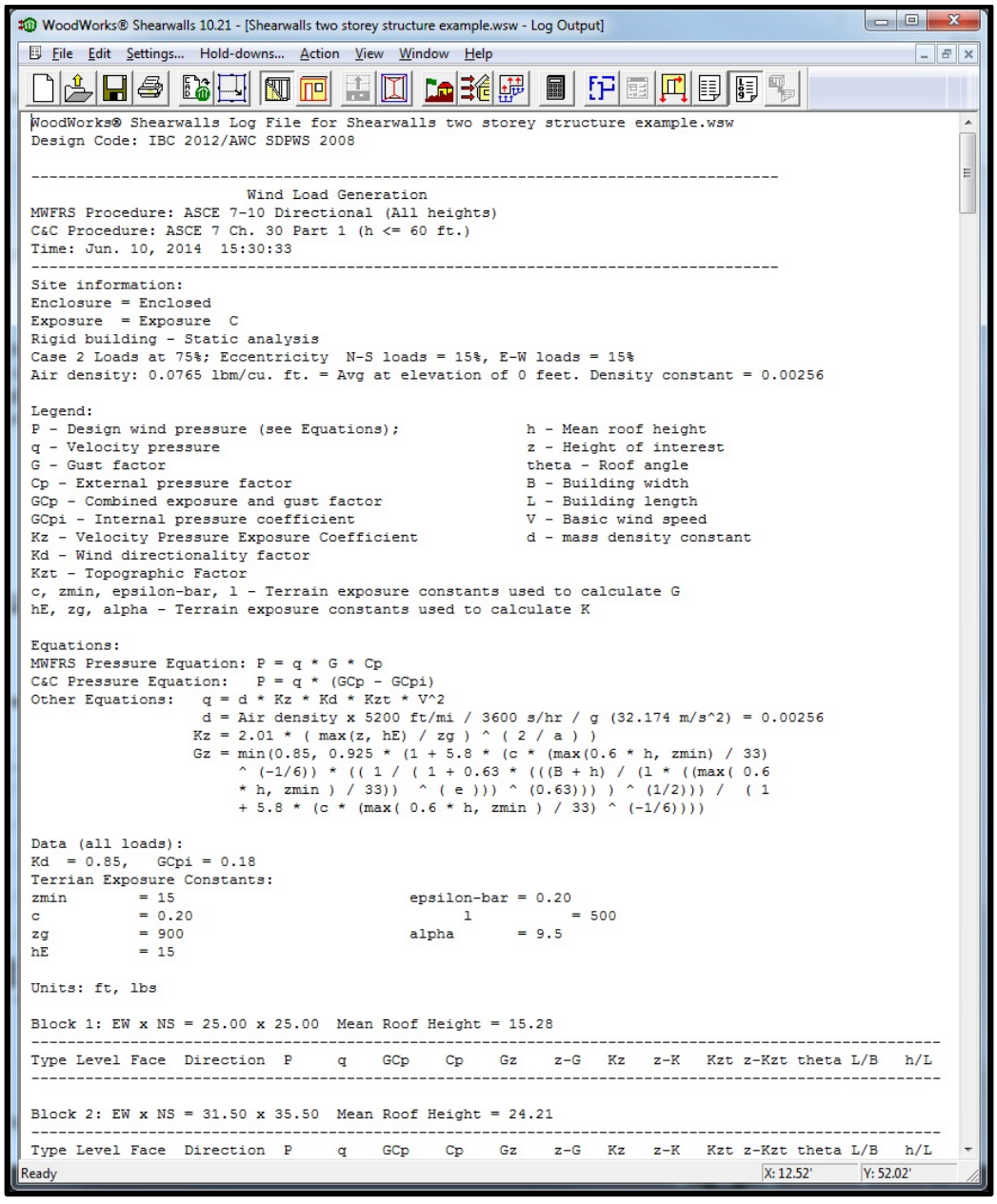



Figure 256: *Shearwalls – Log File*

# 30 *Shearwalls* Wall Types

## 30.1 Perforated Shearwalls (U.S. Only)

### 30.1.1 Traditional Shearwalls (Type I)

Wood frame shearwalls are traditionally designed using shearwall segments that extend the full-height of the wall.

Shearwalls containing openings are treated as multiple shearwall segments, as shown in Figure 325. Each segment requires fasteners to transfer shear and provide overturning restraint resulting in additional hold-downs per wall. The design capacity of the shearwall is assumed to equal the sum of the capacities of each shearwall segment. Sheathing above and below openings typically is not considered to contribute to the overall performance of the wall.[12,13]

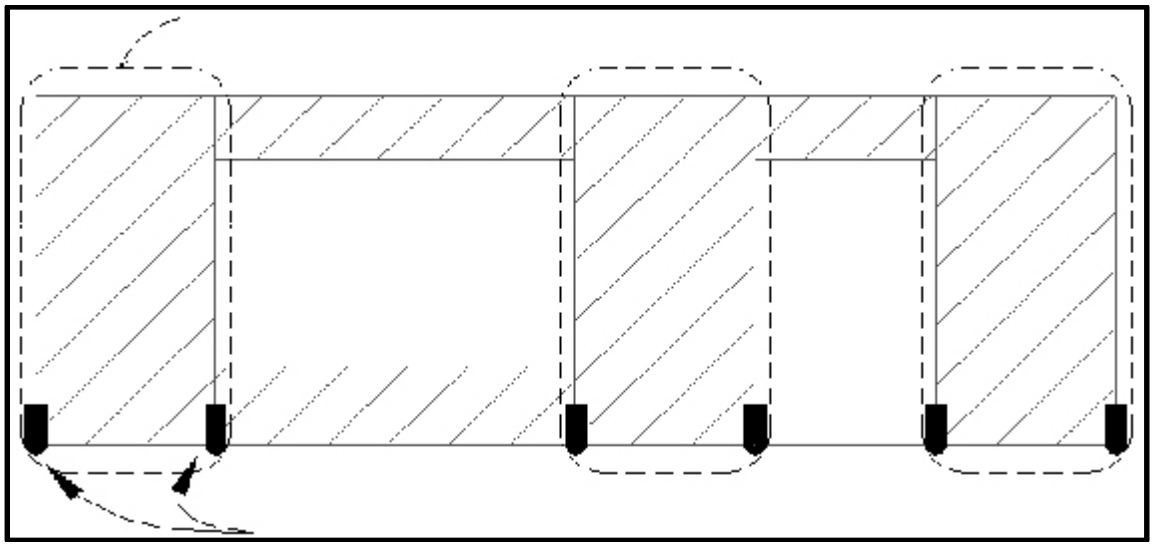


Figure 257: *Typical Shearwall Composed of Three Traditional Shearwall Segments (Type I)*

### 30.1.2 Perforated Shearwalls (Type II)

Another approach considers a shearwall segment with openings, which is illustrated in Figure 326. Here, the wall is identical to the shearwall in Figure 325, but without intermediate overturning restraints next to the openings. This condition is considered a shearwall segment with unrestrained openings (perforated or Type II shearwall). The capacity is clearly less than would be expected from the same shearwall containing multiple shearwall segments, each fully restrained against overturning. However, the capacity of the wall shown in Figure 326 may be sufficient for a particular application while reducing the total number of overturning *Connections*.[12,13]

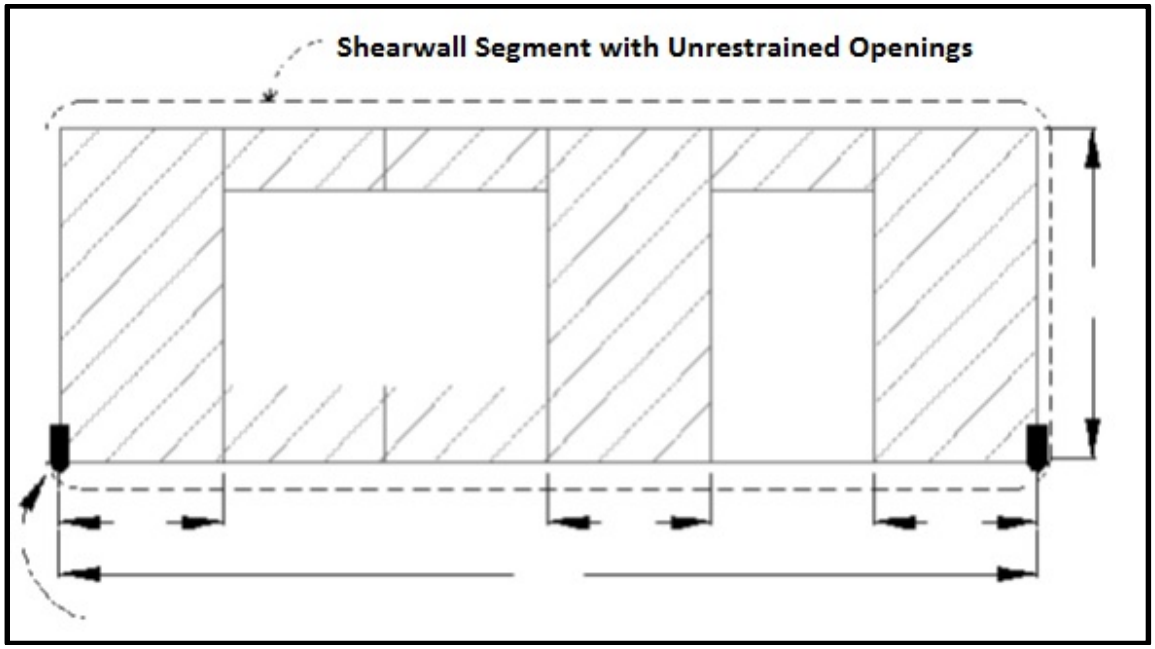



Figure 258: *Typical Perforated Shearwall with Unrestrained Openings (Type II)*

### 30.1.3 Method

Finding the design shear capacity involves relating the strength of a shearwall with unrestrained openings to a similar shearwall consisting of traditional shearwall segments. Empirically derived adjustment factors that relate the strength of shearwalls with unrestrained openings to similar shearwalls containing multiple traditional shearwall segments are presented in Table 1. The perforated shearwall procedure calls the adjustment factor the "opening adjustment factor." This is referred to as the *Perforated Factor* in the Result output.

Opening adjustment factors (referred to as *Perforated Factor* in the *Results* output) for a range of typical opening heights are provided in Table 1 for eight and ten-foot high walls. Factors for other wall heights can be determined from the ratio of opening height to wall height.

Opening Adjustment Factor for Perforated Shearwalls can be seen in the table on the following Table.[9]

Table 1: Perforated Wall Adjustment Factors

| | Maximum Unrestrained Opening Height (Window or Door Height) | | | | |
|---|---|---|---|---|---|
| | H/3 | H/2 | 2H/3 | 5H/6 | H |
| 8’ Wall | | 2’-8” | 4’-0” | 5’-4” | 6’-8” |
| 10’ Wall | | 3’-4” | 5’-0” | 6’-8” | 8’-4” |
| Percent Full-Height Sheathing | Effective Shear Capacity | | | | |
| 0% | 1.00 | 0.67 | 0.50 | 0.40 | 0.33 |
| 10% | 1.00 | 0.69 | 0.53 | 0.43 | 0.36 |
| 20% | 1.00 | 0.71 | 0.56 | 0.45 | 0.38 |
| 30% | 1.00 | 0.74 | 0.59 | 0.49 | 0.42 |
| 40% | 1.00 | 0.77 | 0.63 | 0.53 | 0.45 |
| 50% | 1.00 | 0.80 | 0.67 | 0.57 | 0.50 |
| 60% | 1.00 | 0.83 | 0.71 | 0.63 | 0.56 |
| 70% | 1.00 | 0.87 | 0.77 | 0.69 | 0.63 |
| 80% | 1.00 | 0.91 | 0.83 | 0.77 | 0.71 |
| 90% | 1.00 | 0.95 | 0.91 | 0.87 | 0.83 |
| 100% | 1.00 | 1.00 | 1.00 | 1.00 | 1.00 |

### 30.1.4 Full-Height Sheathing

The percentage of full-height sheathing (FHS) is based on the ratio of full-height sheathing length to shearwall segment length. To be considered full-height sheathing, the length of each full-height sheathed section must equal or exceed the prescribed height-to-length ratios for shearwall segments in the applicable building code.

Values for 0% full-height sheathing are provided for interpolation purposes only. The 100% full-height sheathed case represents a traditional shearwall segment with a corresponding opening adjustment factor of one.

### 30.1.5 Shear *Connections*

Shear *Connections* capable of resisting the allowable unit shear of a traditional shearwall segment are required throughout the segment. While the actual shear connection requirements may be less when using the perforated shearwall method, traditional shearwall segment shear connector requirements account for the non-uniform distribution of shear loads in the perforated wall.

## 30.2 Shearwall Segments Without Hold-downs (Canada Only) (excerpt from Wood Design Manual 2010)[5]

Traditionally, shearwalls have been designed using chords and hold-down *Connections* at the ends of all shearwall segments. Hold-downs are designed to transfer the chord segment overturning force, $T_f$, to the shearwall or foundation below. CSA O86 includes provisions for design of shearwall segments without hold-down *Connections*.[2]

Without hold-downs, the overturning tension force is transferred from the top wall plate to the bottom wall plate through the shearwall sheathing. Since a portion of the shearwall sheathing is used to resist the over-turning force, the shear capacity of the sheathing is reduced. Even though hold-down anchors are not used, anchorage is still required to transfer the uplift force from the wall plate to the foundation or shearwall below (see Commentary to CSA O86).[4]

In some cases, hold-down *Connections* may be placed at one end of a shearwall segment. Where the load is from the end of the segment that contains the hold-down, overturning uplift forces will be resisted by the hold-down. When the load is from the direction opposite the hold-down, the sheathing resists over-turning uplift forces. The shear capacity of the shearwall segment is multiplied by the hold-down factor, $J_{hd}$, which must be calculated for loads acting in opposite directions.

Hold-down *Connections* are not required for shearwall segments where the specified strength is adjusted by the $J_{hd}$, factor, and the following conditions are met:

- the sum of the factored basic shear resistances on both sides of the shearwall, $V_{hd}/L_s$, < 10.3 kN/m;
- shear resistance of the wood-based panel, $v_r$, is based on nail diameters ≤ 3.25 mm; and edge panel nail spacing ≥ 100mm;
- the maximum height of the shearwall, $H_s$, is 3.6m; and
- the factored uplift restraint force at the bottom of the end stud of the shearwall segment, $P_j$, is ≥ 0.

$J_{hd}$ is calculated as:

Case 1

$J_{hd}$ = 1.0 where there is sufficient dead load to resist overturning uplift forces or hold-down *Connections* resist all of the overturning force.

Case 2

Where there is no net uplift at the top of the shearwall segment due to overturning and there is no hold-down connection at the bottom of the shearwall segment to resist overturning:

$$J_{hd} = \sqrt{1 + 2\frac{P_j}{V_{hd}} + \left(\frac{H_S}{L_S}\right)^2} - \left(\frac{H_s}{L_s}\right) < 1.0$$

Where:

$P_j$ = factored uplift restraint force at the bottom of the end stud of the segment calculated as shown below (kN)

$V_{hd}$ = sum of the factored basic shear resistances on both sides of the shearwall segment calculated with $J_{hd}$ = 1.0 (kN)

= $\Sigma(v_r J_n J_{ub} + v_{rg}) \times L_W$

$H_s$ = height of the shearwall segment measured from the bottom of the bottom plate to the top of the top plate (m)

$L_s$ = Length of the shearwall segment (m)

Case 3

Where hold-downs are provided at the bottom wall plate to resist overturning forces but the sheathing is in tension at the top of the shearwall segment due to overturning forces from upper storeys.

$$J_{hd} = \frac{V_{hd} + P_t}{V_{hd}} \leq 1.0$$

Where:

$P_t$ = factored uplift restraint force at the top of the end stud of the segment calculated as shown below (kN)

Note $P_t < 0$

$V_{hd}$ = sum of the factored basic shear resistances on both sides of the shearwall segment calculated with $J_{hd} = 1.0$ (kN) (see above)

# 31 *Shearwalls* Deflection Criteria

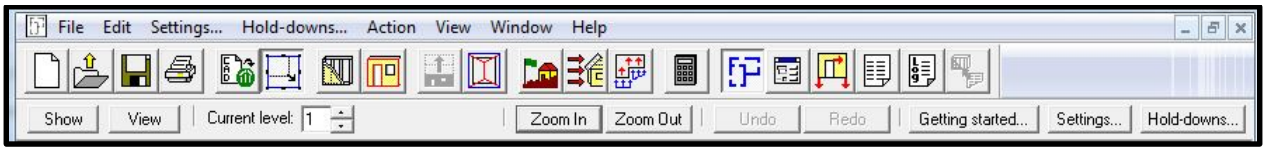



Figure 259: *Shearwalls Display*

*Shearwalls* calculates the deflection of each wall segment between openings for each design case. It uses this deflection to determine the storey drift and check that the drift is within allowable limits for seismic design. *Shearwalls* also distributes loads to shearlines and within shearlines to segments based on equalized deflection of segment. That is, rather than using the capacity of the shear walls to approximate the rigidities for distributing loads to and within shear lines, the software includes the option to determine rigidities based on deflection-derived stiffness. The option to ***Use shearwall deflection to calculate rigidity*** is available only if the ***Settings / Design***: ***Include deflection analysis*** is selected.

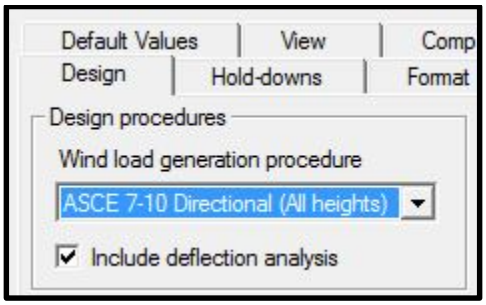



Figure 260: *Default Wind load generation procedure (U.S. Edition)*

*While deflection analysis is considered more accurate in certain circumstances, the added iterative calculations running in the background can slow the run time of Shearwalls. For example, it would not be unusual for a design run to take 3 times longer, in the order of several minutes, when deflection analysis is selected and the distribution of loads is based on rigidity derived from deflection.*

## 31.1 Hold-downs

The ability to input hold-down connectors to a hold-down database for use in design for overturning forces and for deflection analysis gives added flexibility to *Shearwalls*. Previously *Shearwalls* reported hold-down forces at each hold-down location, but did not specify the hold-down *Connections* used.

## 31.2 Hold-downs button

This button opens the hold-down database window where hold-downs can be created and used in the design. Several hold-downs are already created and can be chosen from a drop0down list. The hold-downs specified in the database are selected in the *Wall and Shearline* Input as well as the *Opening* Input at each hold-down location for use in determining the component of deflection attributable to the wall anchorage system's slip.

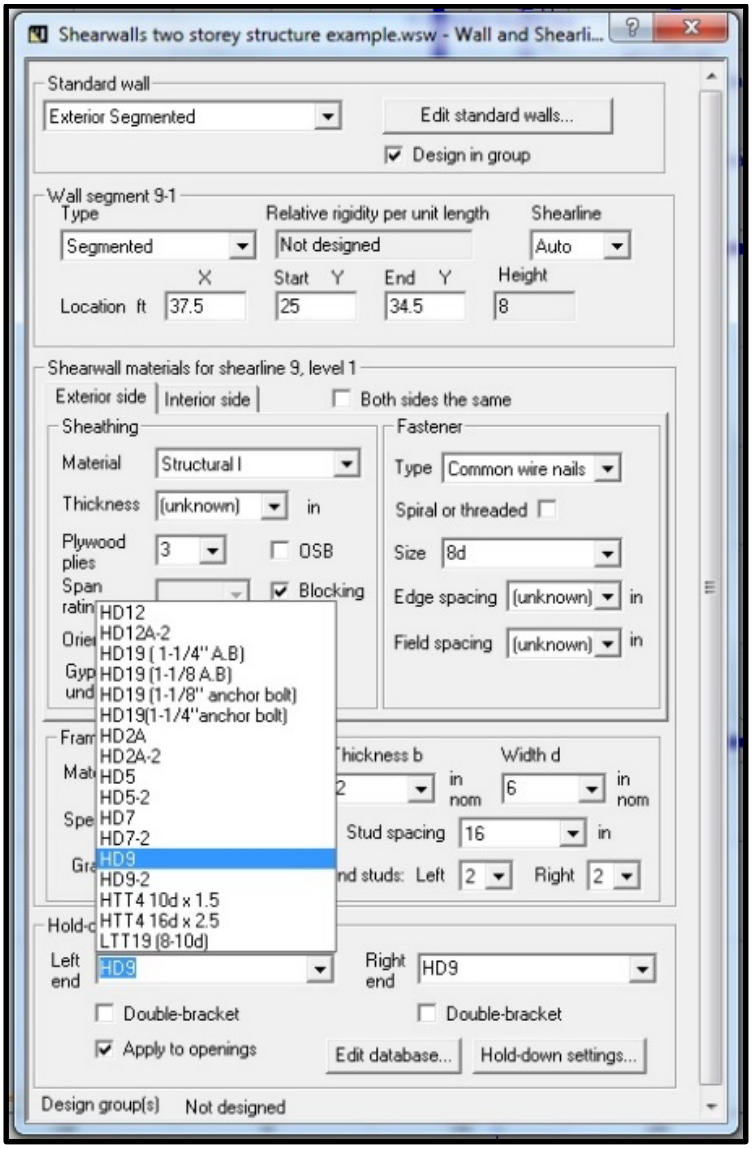



Figure 261: *Shearwalls – Wall Input – Selecting Hold-downs*

Horizontal and vertical fastener information as appropriate for the hold-down is entered in the fastener section.

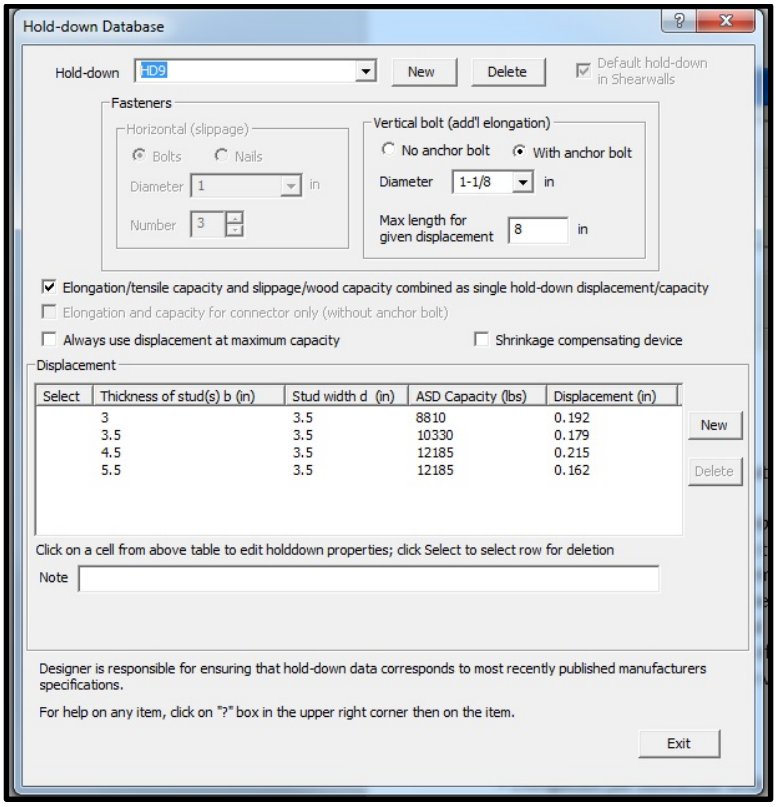



Figure 262: *Shearwalls - Hold-down Database*

There are four check boxes that allow the user to specify what will be taken into account for the displacement.

- *Elongation and slippage combined as single hold-down displacement*: If this is checked, the displacement value entered in the table below includes displacement from all sources of deflection related to the connection assembly, including the elongation of the connector itself, the horizontal fastener slippage used to connect the hold-down bracket to the studs, and the vertical bolt elongation up to the length of vertical bolt shown in the top right of this input form. Manufacturers typically include all these sources of deflection in their tables, therefore selecting this checkbox will be typical. If it is not checked, the value indicates the elongation only and therefore the slippage will be calculated separately.
- *Elongation for connector only (without anchor bolt):* When this box is checked, the elongation value does not represent the anchor bolt elongation, which is calculated using the bolt length indicated in the *Structure* dialog. This option is not viable when the elongation and slippage are combined as single hold-down displacement.
- *Shrinkage compensating device:* If the hold-down is capable to adjust to the wood movement, this box should be checked so the wood shrinkage and crush component are not included in the displacement.
- *Always use elongation/displacement at maximum capacity:* If this box is checked, the displacement is assumed to correspond to the allowable hold-down capacity even if the actual force at the hold-down is less than the listed capacity of the hold down. For example, if *Always use elongation/displacement at maximum capacity* is checked, and if the calculated uplift force that a hold-down is required to resist is 1000 lbs, but the selected hold-down has a capacity of 2000 lbs with a

corresponding displacement of 0.20 inches, the hold-down elongation or displacement will be based on the listed displacement value of 0.20 inches associated with the maximum 2000 lb capacity. Left unchecked, the displacement will be based on the ratio of actual force to capacity, in this case 50% of 0.20 inches, or 0.10 inches.

The deflection of shearwall is important to ensure it remains within the allowable storey drift limits. It also determines the rigidity of the shearwall segments for load distribution within shearlines, and the rigidity of the shearline for use in the rigid diaphragm distribution method. For more information, use the context sensitive help found on this *Hold-down database input* form by selecting the **?** symbol at the top right and clicking on the area for which help is needed.

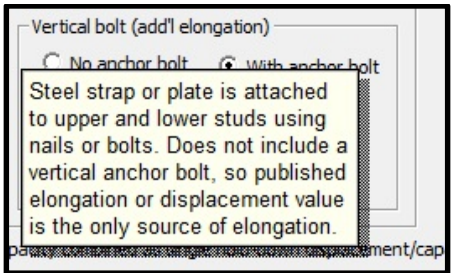



Figure 263: *Shearwalls – selecting ? symbol for further details*

For more detailed information regarding deflection and hold down related information refer to Help and search for keywords as shown below. Also, refer to the *Shearwalls* - New Features file located in the main WoodWorks folder.

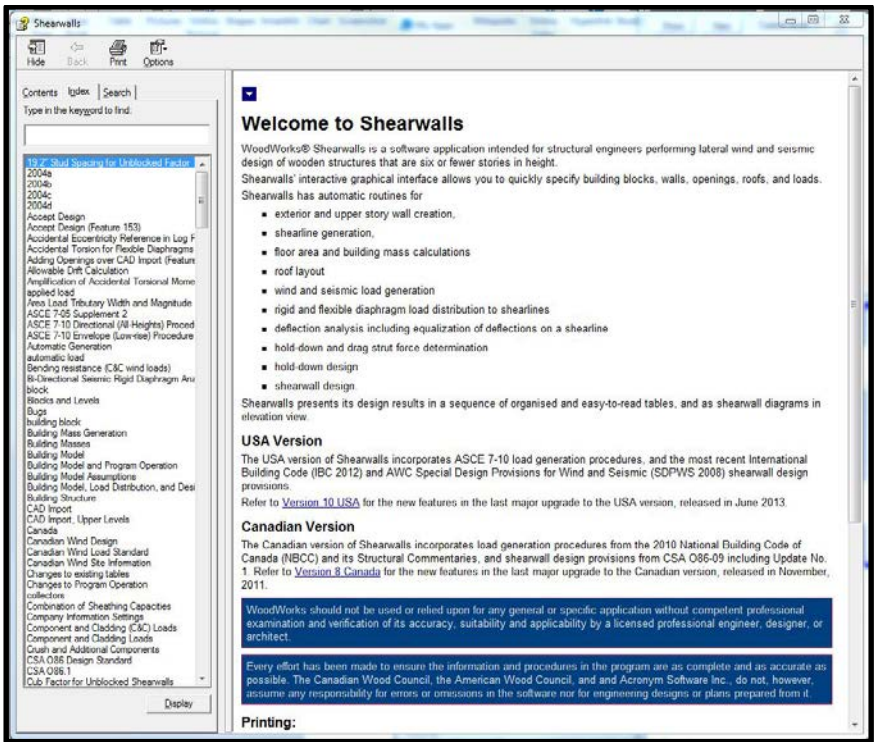



Figure 264: *Shearwalls Help Window*

## 31.3 Design Methods

### 31.3.1 U.S. Version

***Codes and Standards referenced:*** International Building Code (IBC®), Minimum Design Loads for Buildings and Other Structures (ASCE 7-10), National Design Specification for Wood Construction (NDS®), and AF&PA's Special Design Provisions for Wind and Seismic (SDPWS).[7,1,9]

**Design Procedure:** Allowable Stress Design (ASD)

**Wind Procedure:** ASCE 7-10 Method 2 (Analytical Procedure), both Directional (All-heights) and Envelope (Low Rise)

**Seismic Procedure:** ASCE 7-10 Equivalent lateral force procedure

**Horizontal Distribution:** Flexible and rigid diaphragm distribution

**Shearwall types:** Individual full-height wall segment and perforated shear walls. Includes deflection and story drift analysis.

### 31.3.2 Canadian Version

***Codes and Standards referenced:*** National Building Code of Canada, User's Guide - Structural Commentaries, CSA O86 Engineering Design in Wood (Limit States Design).[3,2]

**Wind Procedure:** NBC High-rise (fig I-15) and NBC Low-rise (fig I-7/8)

**Seismic Procedure:** NBC Equivalent static force procedure

**Horizontal Distribution:** Flexible and rigid diaphragm distribution

**Shearwall types**: Segmented with hold-downs on all segments, segmented with hold-downs at ends of shearwalls and where required, segmented with hold-downs at ends of shearlines and where required, and segmented with hold-downs at where required only

## 31.4 Input and Output

*Shearwalls* generates several file types to store general program settings and project information for a design run. This includes the following file types:

- **.wsw** – main file type for *Shearwalls* which stores binary project data and can be opened directly by *Shearwalls* to retrieve previously saved runs.
- **.pdf** – Engineering design results can be saved as Portable Document Format (.pdf) files located in the same folder as the corresponding .wsw file. These can then be opened by Adobe Acrobat or another pdf reader.
- **.rtf** – Engineering design results can be saved as Rich Text Format (.rtf) files located in the same folder as the corresponding .wsw file. These can then be opened and viewed by text editors such as Word and Wordpad, and converted to Word document (.doc) files.
- **.log** – intermediate calculations used to generate wind and seismic loads, and used in the rigid diaphragm analysis are stored in text files with the same name and located in the same folder as the corresponding .wsw file. These files can be opened and viewed through any text editor, such as Notepad or Word.

**shearwalls.wss** – *Standard Walls* are stored in this binary file located in the same folder as the *Shearwalls* program. If lost, it will be regenerated automatically by *Shearwalls*.

# 32 *Connections*

## 32.1 About *Connections*

The WoodWorks® *Connections* software will be referred to as *Connections* throughout this section. This is meant to give a brief introduction to the features and on how to use *Connections*. This brief introduction to *Connections* should be sufficient to help you become an expert user of the software. However, technical support is available if you have further questions. *Connections* is made-up of several different screens which are followed in sequence to complete the design of a connection. You proceed through these screens using Drop-Down Menus and toolbars.

## 32.2 Drop-Down Menus

The *Connections* window always contains a drop-down menu bar and images of the possible *Connections*. Menu selections can be done in two ways:

1. Selecting the desired connection type from the drop-down menu and submenus
2. Selecting the desired connection from the images of the possible *Connections*

*Note: Selecting submenus from the drop-down menu bar will automatically modify the images of possible Connections to the right of the drop-down menu.*

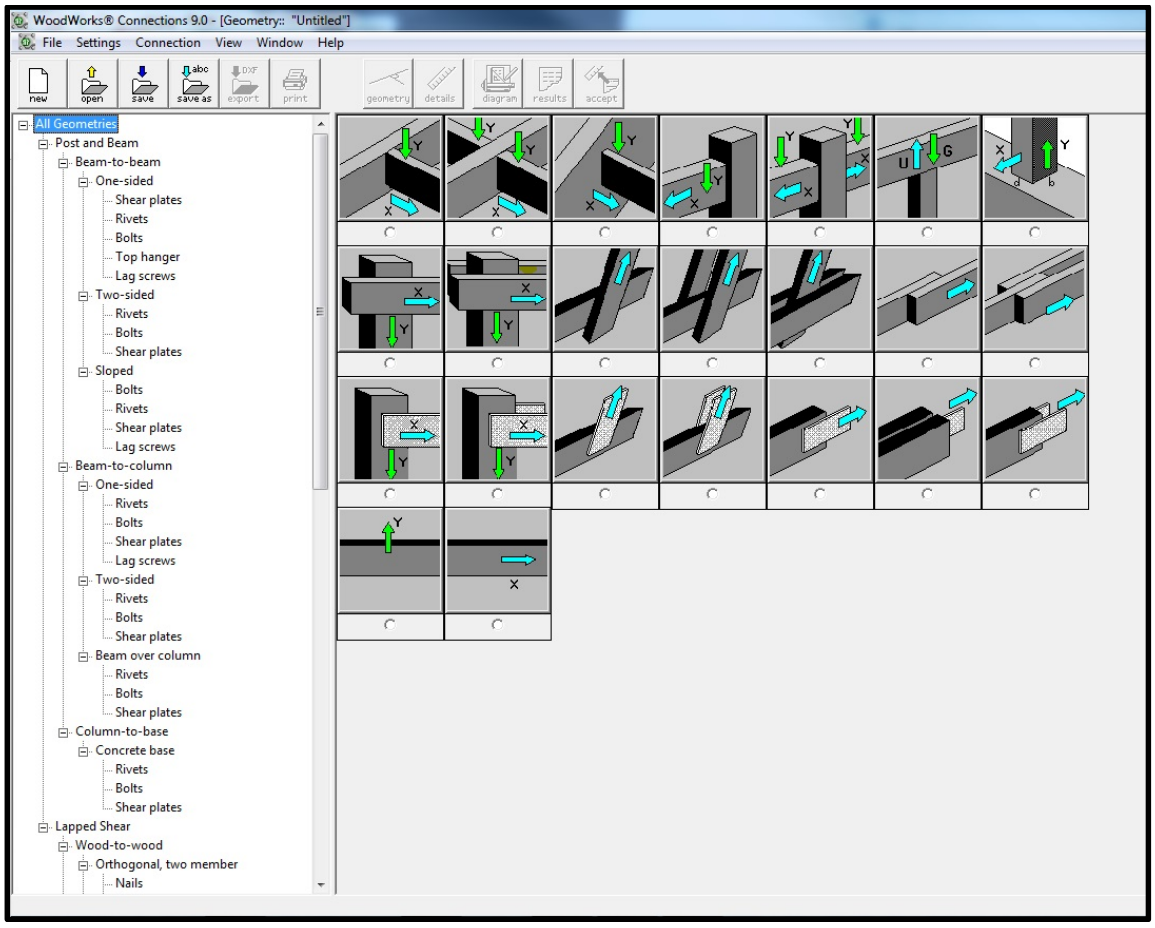



Figure 265: *Connections Drop-down Menu and Connection Images*

## 32.3 Toolbars

*Connections* includes toolbars with toolbar buttons for the main procedures of the program. When performing a design from start to finish, you should proceed from left to right along the toolbar. The toolbar buttons are placed in a sequential order to facilitate designing.

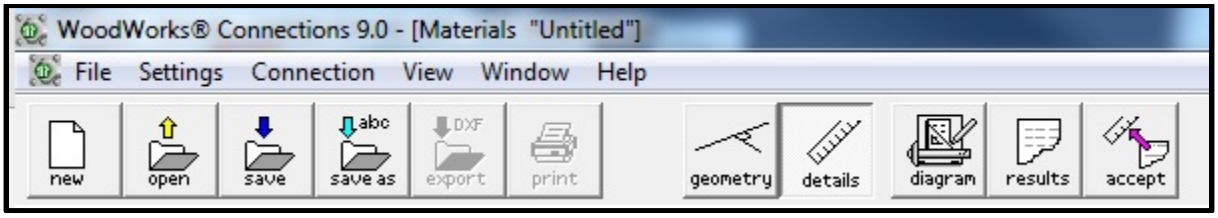



Figure 266: *Connections ToolBar*

### 32.3.1 Toolbar Buttons

The following describes the main features of each of the toolbar buttons in *Connections*.

#### 32.3.1.1 New


Click on the ***New*** button in the toolbar to a start a new connection design file.

#### 32.3.1.2 Open


Click on the ***Open*** button in the toolbar to retrieve an existing file. This command can be used to edit an existing file or to use an existing file as the basis for a new design.

#### 32.3.1.3 Save


Click on the ***Save*** button in the toolbar to save the current design.

#### 32.3.1.4 Save As


Click on the ***Save As*** button in the toolbar to save the current design but with a different name.

#### 32.3.1.5 DXF Export


Selected fasteners can be exported for use in CAD.

#### 32.3.1.6 Print


Click on the ***Print*** button in the toolbar to print the diagram or the design results. These screens must first be open.

#### 32.3.1.7 Geometry


Click on the ***Geometry*** button in the toolbar to change to the ***Geometry*** view. In this view, you can change the current connection geometry.

#### 32.3.1.8 Details


Click on the ***Details*** button in the toolbar to change to the ***Details*** view. In this view, you can specify connector parameters.

#### 32.3.1.9 Diagram


Click on the ***Diagram*** button in the toolbar to change to the ***Diagram*** view. This view displays the current connection detail diagram.

#### 32.3.1.10 Results


Click on the ***Results*** button in the toolbar to change to the ***Results*** view. This view displays the design results output.

#### 32.3.1.11 Accept


Click on the ***Accept*** button to accept the current design results and to input this information into the details screen.

## 32.4 Connection Types

*Connections* can design for both ***Post & Beam*** and ***Lapped Shear*** connection types.

### 32.4.1 Post & Beam *Connections*

The ***Post & Beam*** *Connections* can be designed using any of the following connector types:

- Bolts
- Shear Plates
- Rivets (Canadian only)
- Heavy-duty Hangers
- Lag screws

Post & Beam *Connections* include the following configurations:

#### 32.4.1.1 Beam-to-Beam

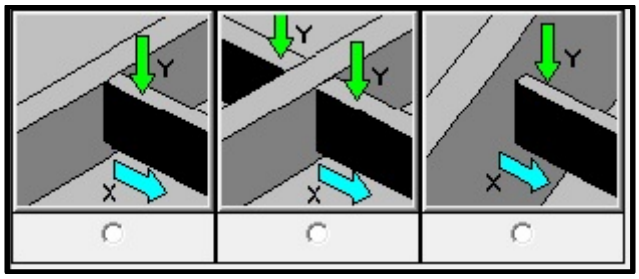



Figure 267: *Beam-to-Beam Connections*

#### 32.4.1.2 Beam-to-Column

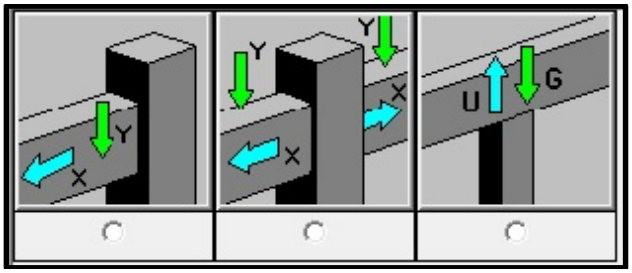



Figure 268: *Beam-to-Column Connections*

#### 32.4.1.3 Column-to-Base

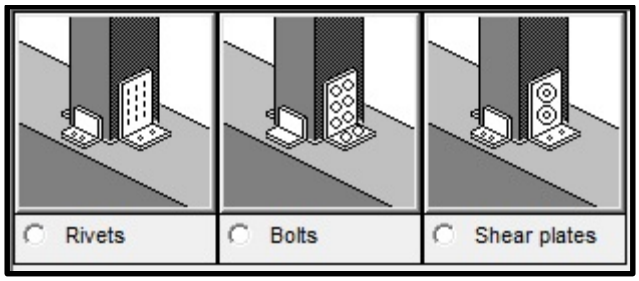



Figure 269: *Column-to-Base Connections*

### 32.4.2 Lapped Shear *Connections*

*Connections* designs for the following Lapped Shear *Connections* using nails or bolts.

#### 32.4.2.1 Wood-to-Wood

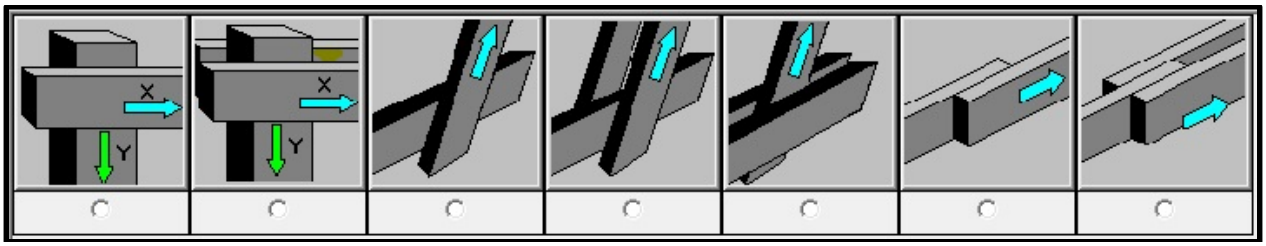



Figure 270: *Wood-to-Wood Connections*

#### 32.4.2.2 Wood-to-Steel

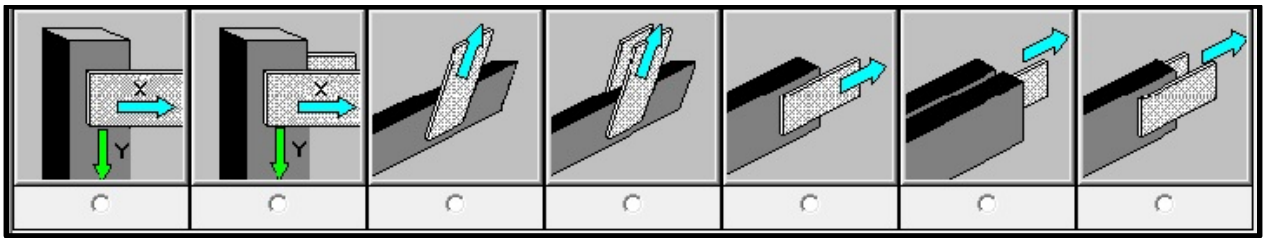



Figure 271: *Wood-to-Steel Connections*

#### 32.4.2.3 Wood-to-Concrete

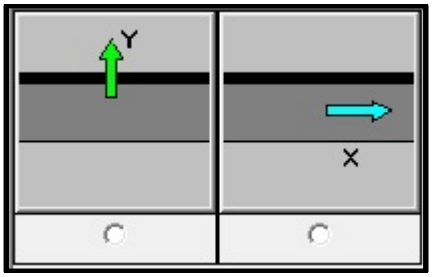



Figure 272: *Wood-to-Concrete Connections*

# 33 *Database Editor*

The *Database Editor* is a simple application that allows you to create new material databases, view existing databases, and customize the material database list for local availability.

## 33.1 Scope of *Database Editor*

The *Database Editor* is currently limited to operating on databases of wood FRAMING materials only. This includes beams, columns, joists and wall studs consisting of solid sawn, glulam and engineered wood products. Sheathing materials and fasteners have not been incorporated at this time. With version 9 of the Canadian edition of the software is now possible to incorporate steel beams, although this feature is not available in the U.S. edition of the software.

Databases used by the Design Office programs:

- ***Sizer*** — all databases
- ***Connections*** — all databases except I-joists, LVL, PSL materials
- ***Shearwalls*** — only the wall stud databases

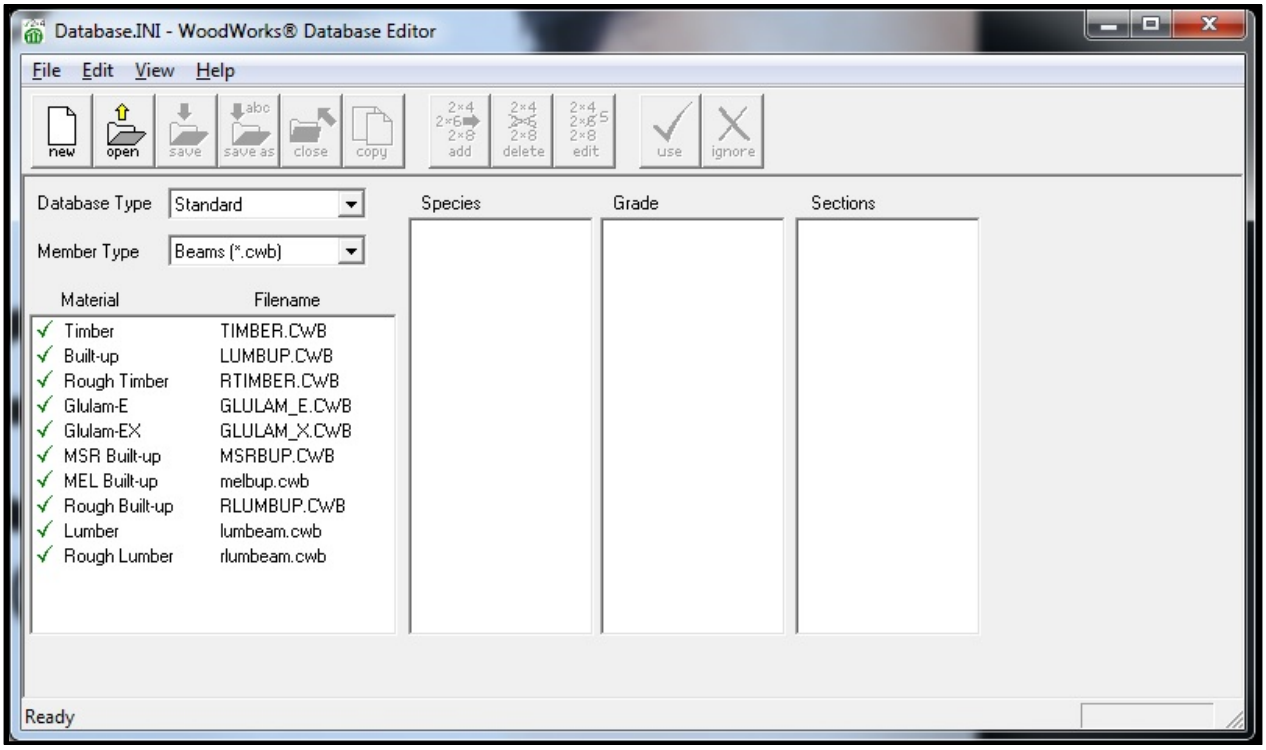



Figure 273: *Database Opening Window*

*Note:*

1. ALL the Design Office programs use a common set of databases that can be viewed and modified by the *Database Editor*.
2. *Sizer* exploits all features of the *Database Editor*.
3. *Connections* and *Shearwalls* do not skip Species, Grades and Sections that have been set to "Ignore" by the *Database Editor*.
4. A change to any database will affect all programs that use it (with the exception of item 3).

## 33.2 File Buttons

### 33.2.1 New


Click on the ***new*** button in the toolbar to create a custom database file. The program will prompt you with a series of dialog boxes to walk you through the process of creating a custom database file.

### 33.2.2 Open


Click on the ***open*** button in the toolbar or click on the database file name in the file list to retrieve an existing database. This command retrieves an existing material database file. You can specify the type of database, such as Beams, Columns, Joists, and Wall Studs.

### 33.2.3 Save


Click on the ***save*** button in the toolbar to update a material database which you applied changes to. The ***save*** button also updates the current ***use*** and ***ignore*** settings for customizing the database list.

### 33.2.4 Save As


Click on the ***save as*** button in the toolbar to make a copy of the current database but with a different name.

### 33.2.5 Close


Click on the ***close*** button in the toolbar to close the current database without saving any changes which may have been done to this file.

### 33.2.6 Copy


Click on the ***copy*** button in the toolbar to copy a current material in the data base for editing. It will be necessary to save the material file with a different name.

## 33.3 Editing Buttons

### 33.3.1 Add


Click on the ***add*** button in the toolbar to add new species, grades or sections to a Custom database file. The ***add*** button is only available for Custom database files.

### 33.3.2 Delete


Click on the ***delete*** button in the toolbar to delete existing species, grades or sections in a Custom database file. The ***delete*** button is only available for Custom database files.

### 33.3.3 Edit


Click on the ***edit*** button in the toolbar to edit the species, grade or section properties of a Custom database file. The ***edit*** button is only available for Custom database files.

### 33.3.4 Use


Click on the ***use*** button in the toolbar to allow a selected species, grade, or section of a material database to be used by *Sizer* and *Shearwalls* during the design process. A check mark will appear beside the species, grade or section to indicate that it can be used by *Sizer* and *Shearwalls*. The ***use*** button applies to both Standard and Custom databases.

### 33.3.5 Ignore


Click on the ***ignore*** button in the toolbar to prevent *Sizer* and *Shearwalls* from using a selected species, grade or section during the design process. An 'X' will appear beside the species, grade or section to indicate that it cannot be used by *Sizer* and *Shearwalls*. The ***ignore*** button applies to both Standard and Custom databases.

# 34 Material Databases

Material databases are described as either being a Standard database or a Custom database. The properties of the Standard databases can be viewed, but not edited. Custom database can be viewed, edited and created.

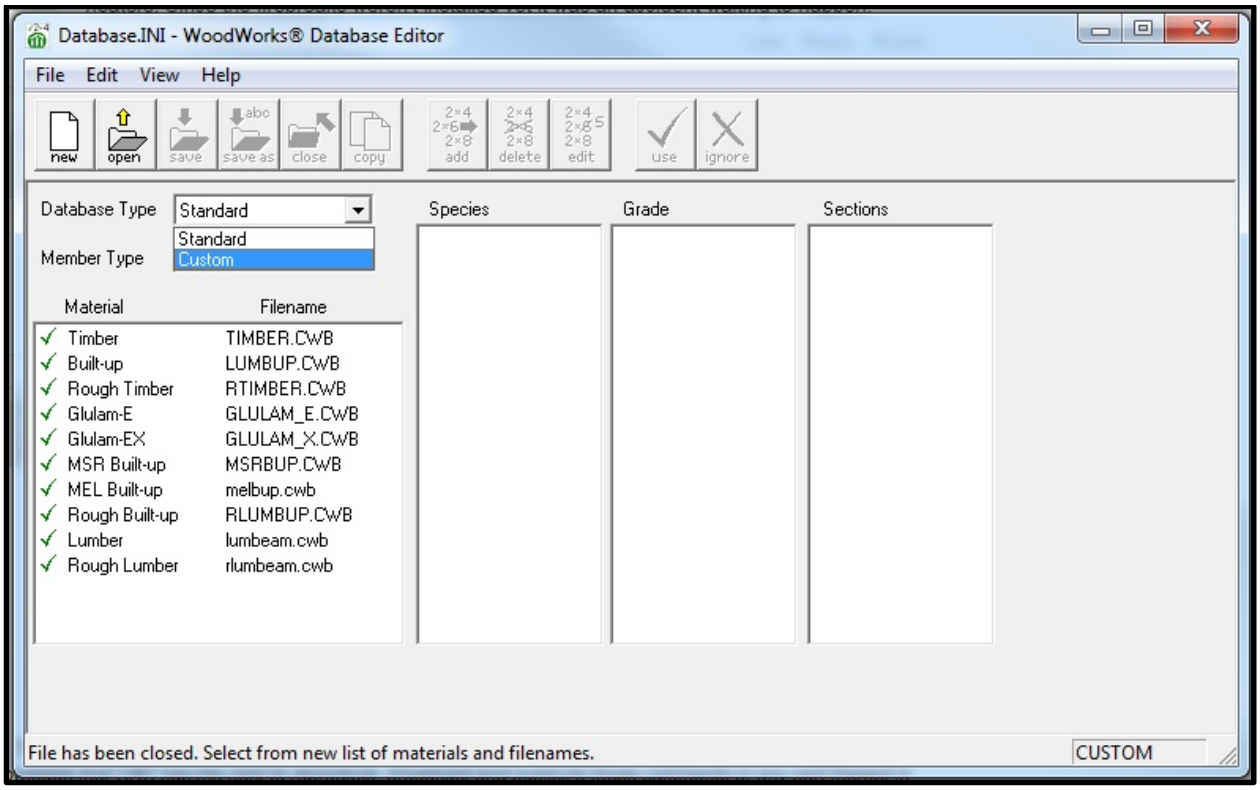



Figure 274: *Switch between Standard and Custom Database*

The ***Database Type*** drop-down list allows you to switch between the *Standard* and *Custom* databases.

## 34.1 Standard Material Databases

Standard material databases include sawn timber and glulam members whose strength properties are based on those published in the applicable design standards. The Standard databases are based on values published in the National Design Specification (NDS) for Wood Construction for the U.S. and based on the CSA O86 for Canada.[1,2]

## 34.2 Custom Material Databases

Custom material databases include I-joists, Parallel Strand Lumber (PSL), Laminated Veneer Lumber (LVL), and Laminated Strand Lumber (LSL) as a default. The strength properties included in these custom databases are based on proprietary listings. The custom databases included with WoodWork® or created by the user should only be used for preliminary sizing of members. Contact the engineered wood product manufacturers directly for an accurate and complete design of proprietary wood products.

**Warning:** You can retain database customizations made with WoodWorks® *Database Editor* from an existing installation by specifying the same installation folder as the existing installation, choosing the ***Custom Install***

*Setup Type*, then unchecking ***Custom Materials Database*** in *Select Components*. Failing to remove the check-mark will result in your custom database being overwritten by the new custom database.

# 35 Creating Custom Databases

The *Database Editor* allows you to create custom material databases for beams, joist, columns, and walls. You can specify material type, species, grade, dimension, and strength properties.

## 35.1 Material

Click the *new* button on the toolbar menu. The *New Material* dialog opens and prompts you to enter a Material name, specify the type of material, specify a filename, define whether this is a multi-ply member or not, and select a *Material Type*.

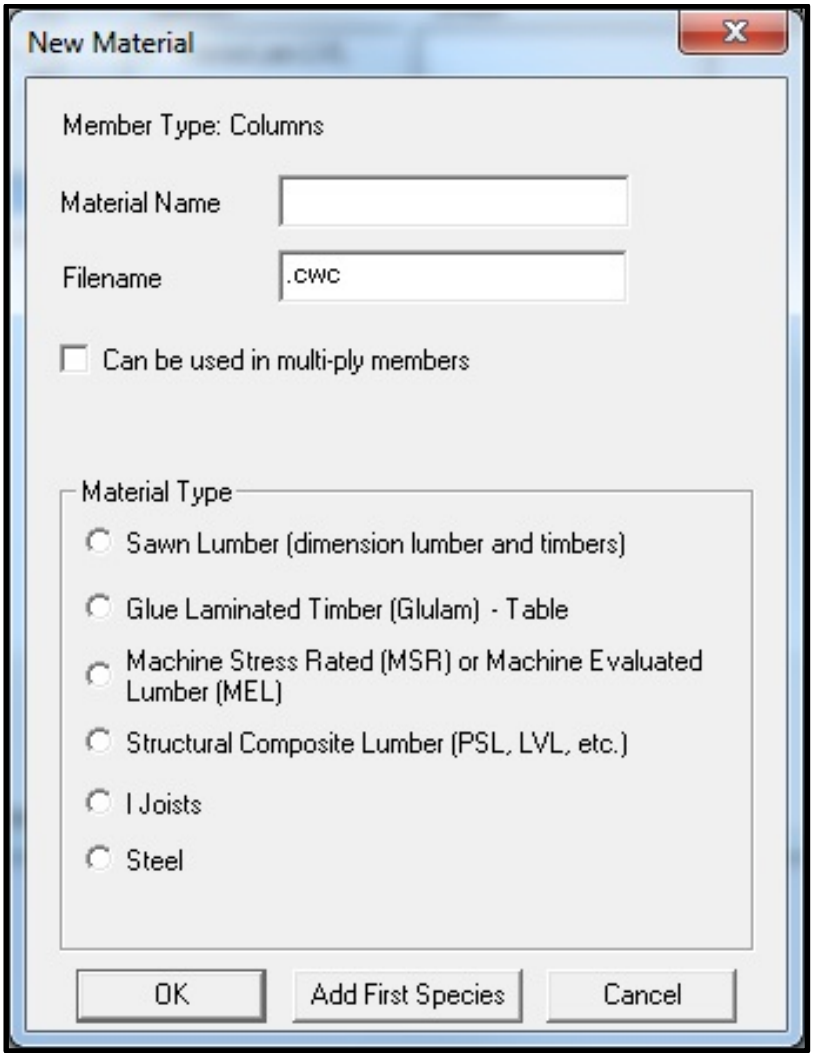



Figure 275: *New Material Dialog Window*

## 35.2 Species

Click on the ***Species*** in the *Database Editor* window, then click the ***edit*** icon. Now the ***Species Properties*** dialog window automatically opens and prompts you to enter a species name, the specific gravity of the material (for self-weight calculations).

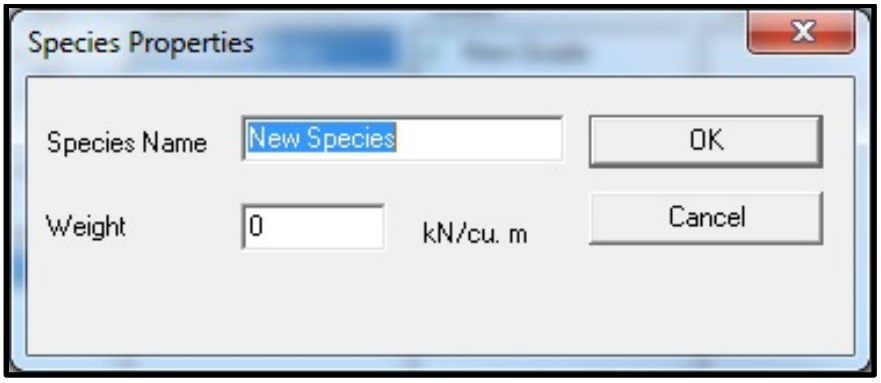



Figure 276: *Species Properties Window*

## 35.3 Grade

Click on the ***Grade*** in the *Database Editor* window, then click the ***edit*** icon. Now the ***Grade Properties*** dialog window automatically opens and prompts you to enter a Grade/Combination name, the specified strengths values, and the modulus of elasticity (E).

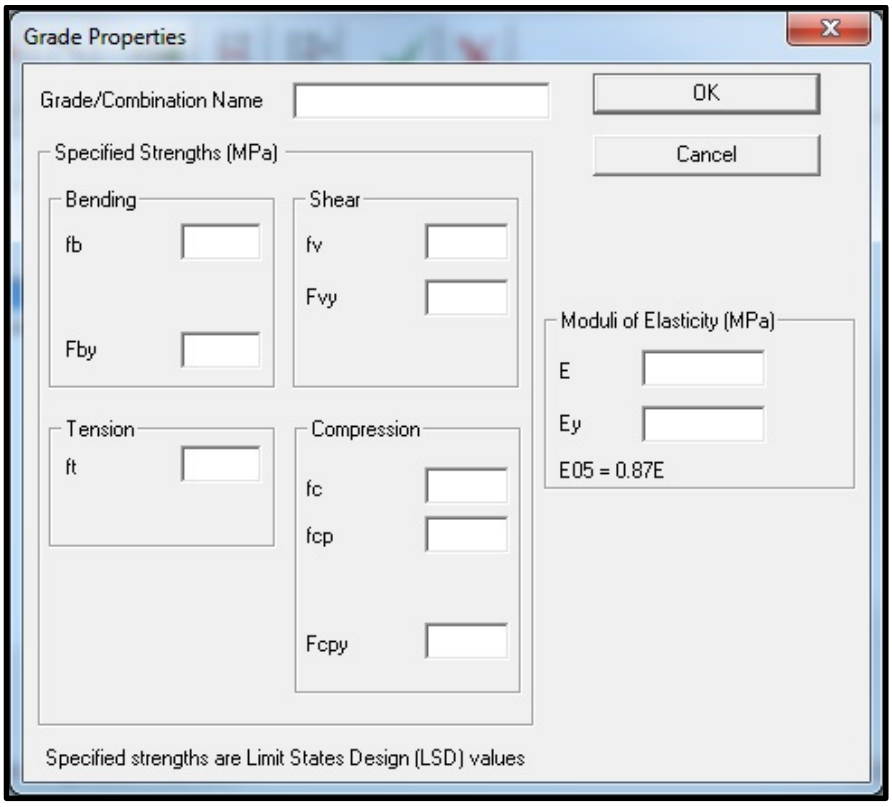



Figure 277: *Grade Properties Window*

### 35.4 Section

Click on ***Section*** in the *Database Editor* window, then click the ***edit*** icon. Now the ***Section Properties*** dialog window automatically opens and prompts you to enter the actual and nominal dimensions, the size factors, and the stock length.

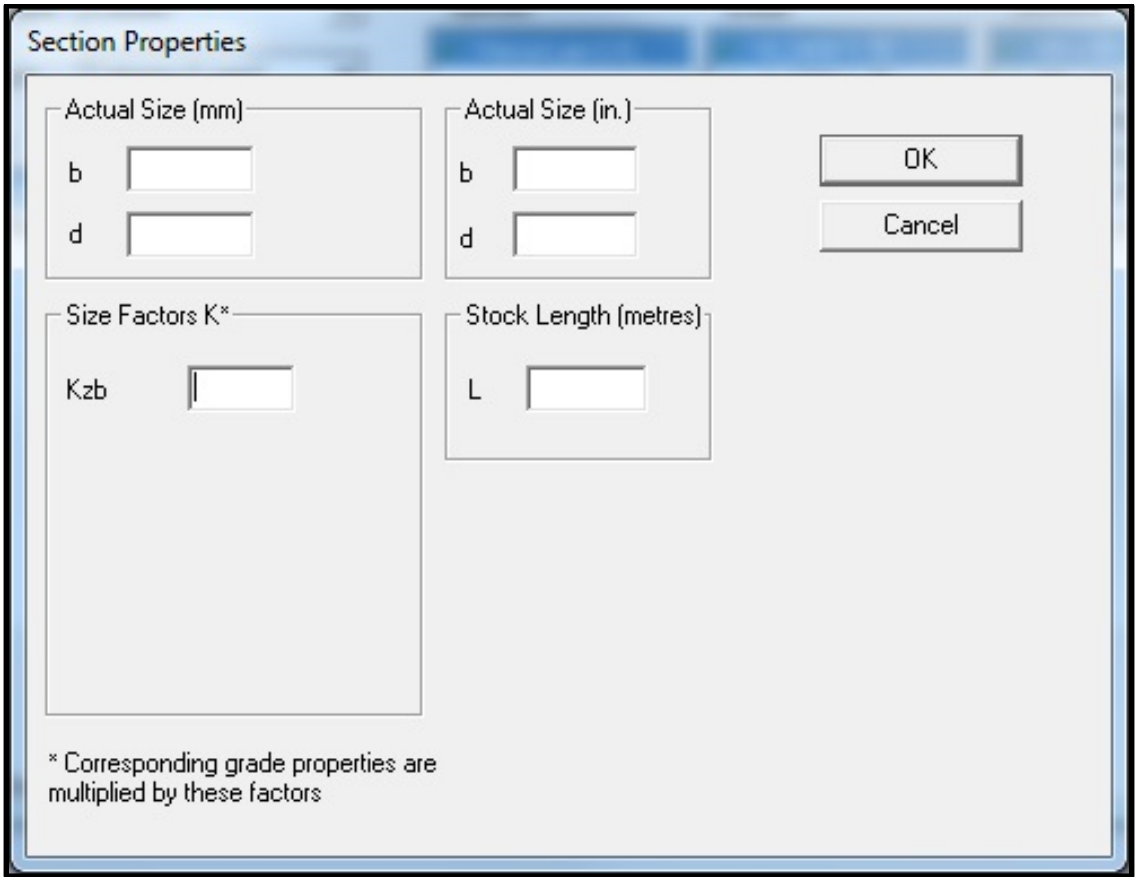



Figure 278: *Section Properties Window*

# 36 Viewing Standard Databases

The *Database Editor* allows you to view the material, species, grade and section properties of the *Standard* databases. The following describes how to do this. First open the *Standard* database file that you wish to view. This is done by using the ***open*** button or by clicking on the file name in the file list.

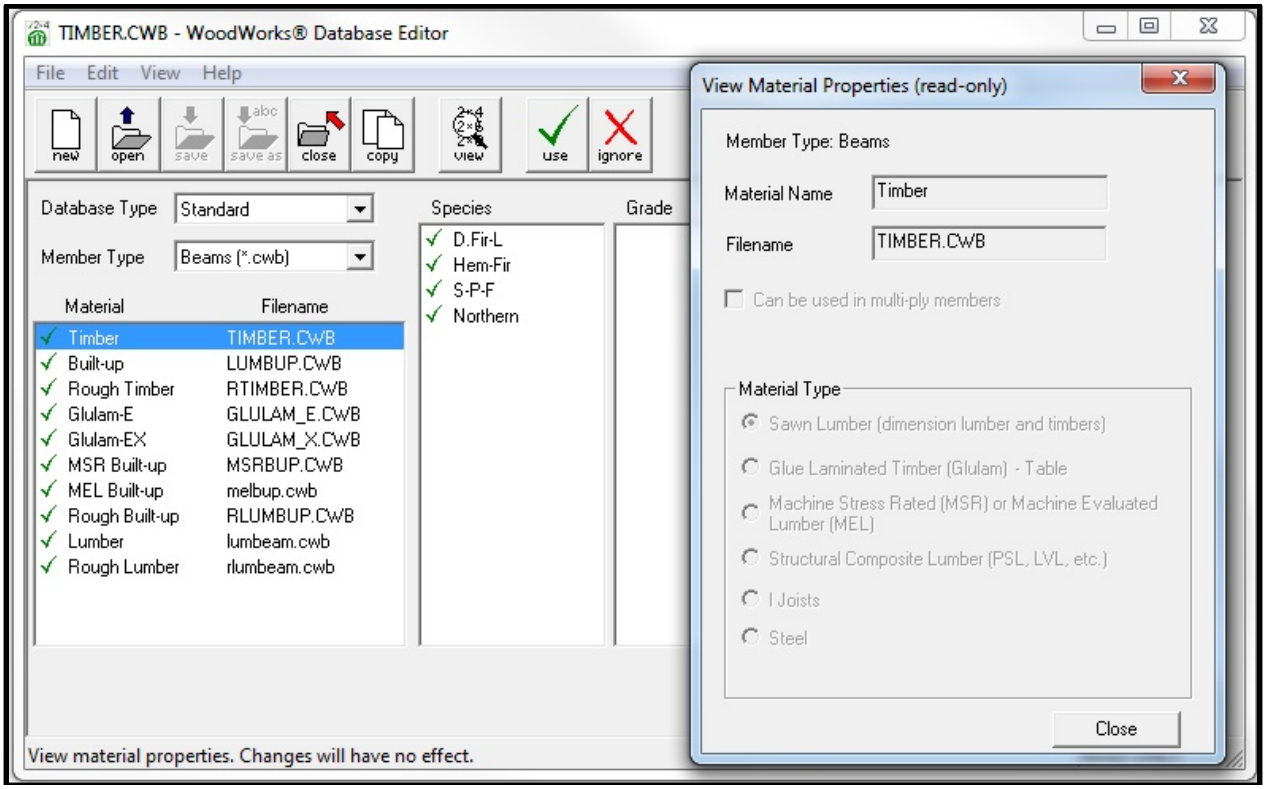



Figure 279: *View Standard Material Properties*

You must now select the property that you want to view. This is done by highlighting the material, species, grade or section that you wish to view and then clicking on the *view* button from the toolbar. Depending on the information that you are viewing, one of the four following dialog boxes will open:

- *Material Properties* dialog
- *Species Properties* dialog
- *Grade Properties* dialog
- *Section Properties* dialog

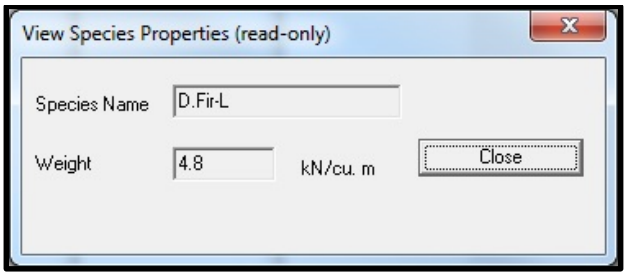



Figure 280: *View Species Properties*

# 37 Customizing the Database List (*Sizer* Only)

The *Database Editor* includes a feature which will allow you to customize the entire database list for your needs by designing for only certain *materials*, *species*, *grades*, or *section sizes*.

This type of customization only affects *Sizer*. *Connections* and *Shearwalls* disregard the *ignore* settings and can use all of the material databases.

The customizing feature simply places a '✓' or an 'X' beside the material, species, grade, or section to let *Sizer* know which properties to *use* or *ignore* during the design process. These properties are not erased by the customizing feature and can therefore be retrieved at a later time. *Sizer* initially sets all properties to *use* (✓) as a default.

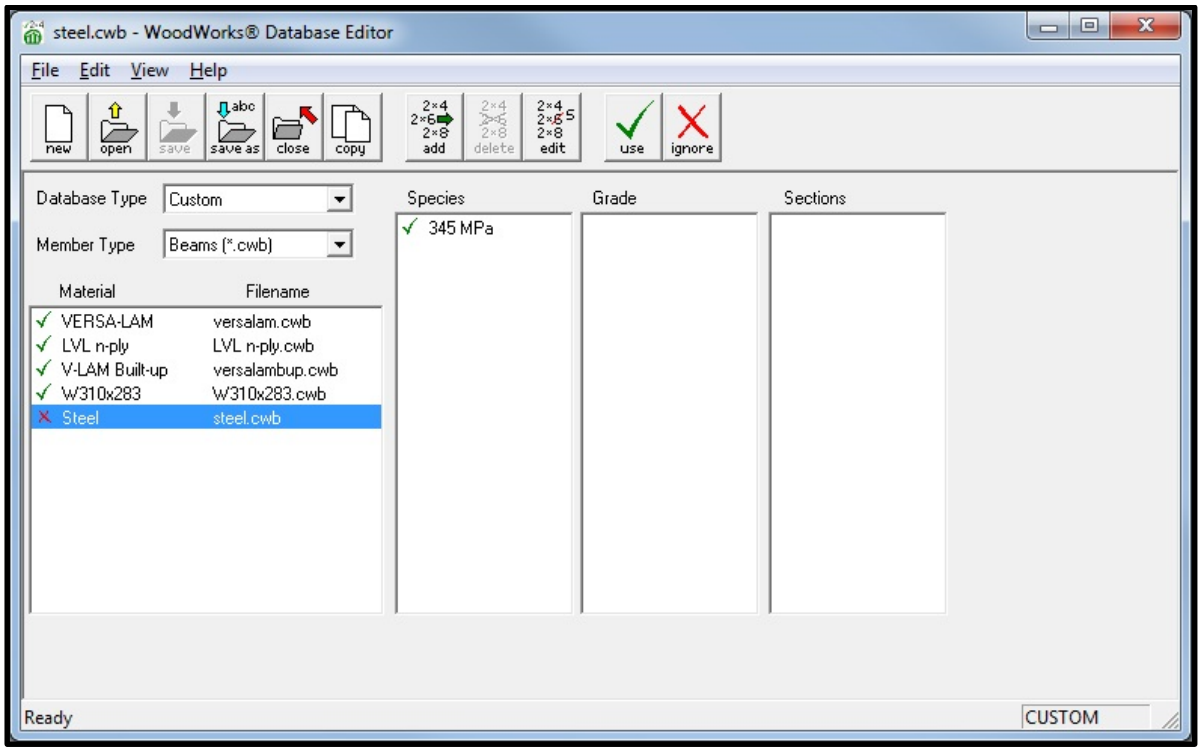



Figure 281: *Tutorial 16: Customizing Data Base List (Ignore Steel)*

## 37.1 Customizing

1. Open the database file (either Standard or Custom) that you want to customize. This is done by using the ***open*** button or by clicking on the file name.
2. Click on the material(s), species, grade(s), or section(s) that you wish to customize so that it is highlighted (*Note that multiple selections can be made at once by clicking while holding down the ctrl and shift key*).
3. Click on the use button (✓) to use this selection during the design process or click on the ignore button (X) to ignore this selection during the design process.
4. Click on the save button to save any customizing done to the materials list.

*Note: Clicking the **save** button at this point will also save any changes done to an open Custom database file.*

# 38 Endnotes

[1] American Wood Council, 2012. National Design Specification for Wood Construction with Commentary 2012 Edition. Leesburg, VA.

[2] Canadian Standards Association, 2014. O86-14 Engineering design in Wood. Mississauga, ON.

[3] National Research Council Canada, 2010. National Building Code of Canada 2010. Ottawa ON.

[4] Canadian Wood Council, 2010. Commentary CSA O86 Engineering Design in Wood 2009 Edition. Ottawa, ON.

[5] Canadian Wood Council, 2010. Wood Design Manual 2010. Ottawa, ON.

[6] American Wood Council, 2007. Beam Formulas with Shear and Moment Diagrams. Washington, DC.

[7] International Code Council INC., 2011. 2012 International Building Code. Country Club Hills, IL.

[8] American Society of Civil Engineers, 2010. Minimum Design Loads for Buildings and other Structures. ASCE/SEI 7-10. Reston, VA.

[9] American Wood Council, 2008. Special Design Provisions for Wind and Seismic with Commentary. Leesburg, VA.

[10] Canadian Standards Association, 2009. O86-09 Engineering design in Wood. Mississauga, ON.

[11] American Wood Council, 2012. Wood Frame Construction Manual for One- and Two-Family Dwellings, 2012 Edition. Leesburg, VA.

[12] Douglas, Bradford; Sugiyama, Hideo., 1995. Perforated shearwall design approach. Washington, DC: American Forest & Paper Association.

[13] Sugiyama, Hideo., 1981. The evaluation of shear strength of plywood-sheathed walls with openings. Mokuzai Kogyo (Wood Industry). 36-7, 1981.