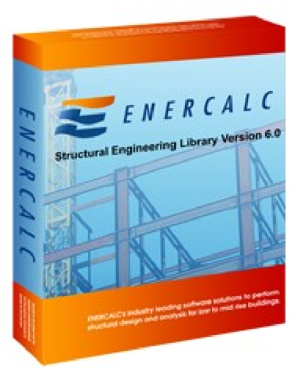
Structural Engineering Library



ENERCALC, INC

Structural Engineering Library

Version 6.0

by Michael D. Brooks, S.E., P.E.

A product of

ENERCALC, INC.

Structural Engineering Library

Publisher

ENERCALC, INC.

Managing Editor

Michael D. Brooks, S.E., P.E.

ENERCALC Engineering Software

Post Office Box 188
Corona del Mar, CA 92625
(949) 645-0151
(800) 424-2252
Fax: (949) 645-3881

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

Vesion 6 User's Reference
March 2010
Corona del Mar, CA, USA

Table of Contents

Foreword 0

Part I Introduction 1

1 Welcome 3
2 Our History 5
3 Warning & Disclaimer 7
4 License & Copyright 8
5 End of Service Policy 9

Part II Installation & Activation 10

1 License Types 12
2 Installation Overview 14
3 Activation Types 15
4 Manual Activation 17
5 Moving & Maintaining Your Activation 19

Part III Software Updates 23

1 Web Update 25
2 Update from Website 28

Part IV Support & Maintenance 29

1 Maintenance & Support Plan 31
2 Getting Assistance 32
3 Viewing Enhancements and Changes to the Software 34

Part V Getting Started 35

1 System Overview & Design Philosophy 37
2 About Our Documentation 38
3 Building Codes Supported 39
4 Learning Structural Engineering Library 40
5 Introductory Videos 41
6 Request for Suggestions 42

Part VI Program Overview 43

1 Introduction 45
2 Typical Worksession 47
3 Program Settings 48
4 Files & File Locations 55

5 Project Assistant ... 56
6 Databases ... 58
7 Screen Layout ... 59
8 Generating Reports ... 62
9 General Calculation Screen Usage ... 64
10 3D Renderings ... 66

Part VII Main Menu 68

1 File ... 70
2 Settings ... 76
Default Values ... 78
3 Databases ... 79
Steel Section Database ... 80
Wood Section Database ... 81
Wood Reference Design Values Database ... 82
Masonry Database ... 83
Load Combination Database ... 84
4 Tools ... 87
5 License ... 89
6 Help ... 90

Part VIII Project Manager 94

1 General Division ... 101
Building Code Information ... 102
Project Information ... 103
Client Information ... 104
Designer Notes ... 105
Revision List ... 106
2 Loads & Forces Division ... 107
3 Calculation List ... 108
Divisions ... 112
Adding, Deleting, Copying ... 116
Sorting by Division, Type & Material ... 118
Changing Calculation Order ... 122
Importing Calculations ... 126
4 Project Printing ... 131

Part IX Sample Session 134

1 Starting the Program ... 136
2 Project Assistant ... 137
3 Creating a Project File ... 139
4 Entering Project Information ... 141
5 Setting up your Title Block ... 143
6 Adding a Calculation ... 145
7 Viewing the Calculation Screen ... 148

8 Changing the Default Values/Settings ... 149
9 Entering Data ... 150
10 Selecting Sections and Materials from Built-in Databases ... 161
11 Displaying a Sketch ... 166
12 Displaying Diagrams ... 167
13 Automatic Member Section Selection ... 169
14 Printing a Calculation ... 172
15 Saving a Calculation ... 175
16 Editing a Division Name and Adding a New Division ... 176
17 Adding Another Calculation ... 180
18 Creating a Technical Support Question ... 183
19 Closing a Project File ... 188

Part X Calculation Modules 189

1 Loads & Forces Division ... 191
- ASCE Seismic Base Shear ... 192
- ASCE Seismic Demands on Nonstructural Components ... 205
- ASCE Seismic Wall Anchorage ... 206
- ASCE Wind Enclosure ... 208
- ASCE 7-10/7-16 Wind Forces, Chapter 27, Part 1 ... 210
- ASCE 7-05 Section 6.4, MWFRS:Simplified Forces ... 225
- IBC Alternate All-Heights Wind Method ... 228
- ASCE Snow Loads ... 234
- ASCE Live and Roof Live Load Reduction ... 239
- Project Load Group Builder ... 240

2 Beams ... 245
- Multiple Simple Beam ... 258
- General Beam Analysis ... 269
- Wood Beam ... 273
- Steel Beam ... 283
- Concrete Beam ... 288
- Beam on Elastic Foundation ... 294
- Composite Steel Beam ... 295
- Wood Ledger ... 307
- Masonry Beam ... 313
- Steel Beam with Torsional Loads ... 321

3 Columns ... 326
- Column Slenderness ... 331
- Wood Column ... 335
- Steel Column ... 344
- Concrete Column ... 350
- Masonry Column ... 366

4 General Walls ... 372
- Concrete Slender Wall ... 373
- Masonry Slender Wall ... 387

5 Shear Walls ... 400
- Concrete Shear Wall ... 401

Masonry Shear Wall ... 411
Wood Shear Wall ... 418
6 Retaining Walls ... 435
Cantilevered Retaining Wall ... 436
7 Foundations ... 467
General Footing ... 468
Combined Footing ... 480
Wall Footing ... 489
Pile Group ... 499
Pole Footing Embedded in Soil ... 503
Beam on Elastic Foundation ... 510
8 2D Frame Analysis ... 511
Frame Wizard ... 516
Joints & Joint Loads ... 521
Joint Data ... 524
Joint Loads & Results ... 528
Members & Member Loads ... 531
Member Data ... 534
Member Loads ... 543
Member Forces ... 549
Sections & Materials ... 550
Section Data ... 550
Material Data ... 552
Load Combinations ... 555
Wood Design ... 558
Results ... 559
Extreme Values ... 559
Joint Displacements & Reactions ... 562
Member End Forces ... 563
Member Details ... 564
Member Check Results ... 565
Sign Convention ... 567
Frame Sketch ... 570
Member Diagrams ... 571
Reports ... 573
Printout ... 574
9 Miscellaneous Calculation Modules ... 580
Torsional Analysis of Rigid Diphragm ... 581
Point Load on Slab ... 595
General Section Property Calculator ... 598
Rebar Development Table ... 605
Steel Bolt Group Analysis ... 607
Steel Base Plate ... 611
10 External Items ... 619
EXCEL Spreadsheet ... 620
WORD Document ... 624
Adobe Acrobat PDF ... 628
Scanned Document ... 632

Part I

1 Introduction

Last Revised: 3 February 2020

1.1 Welcome

Welcome to Structural Engineering Library

You've chosen one of the most respected Structural Engineering software packages available today. In continuous development since 1983, Structural Engineering Library is the culmination of years of development and refinement from suggestions of engineers worldwide.

Structural Engineering Library is developed with the practicing engineer in mind. Although large complex frames are fun projects, structural engineers spend most of their time designing and analyzing the components of structures. Because most of the buildings worldwide rely on simple beams, columns, foundations, walls, and other small items, this software system will quickly become your best friend. **Structural Engineering Library** remembers the mathematics, building code provisions, and standard materials you need to perform a detailed and economical design.

Because we feel that simple, repetitive engineering problems are far more common than extensive 3-D frame analysis, this software package is designed specifically for fast, interactive engineering design of building components. We've combined the typical working methods of engineers, national building code provisions, and construction material databases with the principles of structural mechanics into each "calcsheet" module. You will find that these modules operate very much like an electronic calculation pad.....simply fill in the data entries and the entire calculation will be instantly updated for your review.

To add even more power and utility to the system, we've added detailed design sketches and stress diagrams, automatic design and sizing, an online help system, material databases, and elegant calculation printing to **Structural Engineering Library**.

Structural Engineering Library is designed around a file of calculations called a "**Project**". This single file with the extension "**EC6**" can hold one or thousands of individual calculations. You add, edit, and delete the calculations in your Project File during the in-office design stage. Then, when it's time for submittal to a governing agency, you can print a complete set of calculations.

Because of the ever-expanding number of modules, we invite you to stay in close contact with our website at www.enercalc.com. Maintenance releases, up-to-the-minute technical advice, revised electronic documentation, and new product information will all be provided there FIRST.

ENERCALC has put years of work into this package in support of the highly technical and dedicated service Structural Engineers provide to the people of the world. We continue to enhance this product weekly and are committed to developing this product well into the future. We extend our thanks for choosing ENERCALC, and look forward to using your suggestions to provide you with ever improving tools for your daily work.

1.2 Our History

ENERCALC Engineering Software (now **ENERCALC, INC.**) originated as one young engineer playing with his new T.I. programmable calculator in 1980. In 1981, a set of Lotus 1-2-3 spreadsheet templates was developed to automate the repetitive design of components of tilt-up and small office buildings. The software proved so productive and time saving that it was decided to market it in the newly founded microcomputer industry. The entire set of 26 spreadsheet "templates" was named **Structural Engineering Library** and shipped on three 360K 5 1/4" diskettes, running on a 4.77 MHz 8086 IBM PC. Typical cost of this state of the art engineering hardware and software system was $6,000!

As years went by, sales of the product increased. For engineers to purchase a technical software system based as a pre-programmed spreadsheet "template", it was a testament both to the intelligence of the users and to the easy and simple design of the spreadsheet based software package.

In 1986, Lotus Development introduced a tool for programmers to link programs written in "C" to the very guts of 1-2-3. Called the "Add-In Toolkit", it offered a unique opportunity for **ENERCALC** software designers. A decision was made to rewrite all of the current engineering spreadsheet "templates" into the "C" language, and link these powerful programs to simple 1-2-3 "templates". The Lotus 1-2-3 spreadsheets would become data entry/output screens driving powerful "C" language compiled processing programs. This solved the major problem with a spreadsheet.....lack of iteration capability that was critical to engineering tasks.

Our first application of this technology was **FastFrame** 2-D introduced in 1987. **FastFrame** transformed a simple, off-the-shelf spreadsheet program into a powerful finite-element analysis system with full graphics. Prior to founding RISA Technologies, Bruce Bates worked on a "lightning fast" 16 MHz PC to develop the FastFrame solver. That solver would later become the guts of the first version of RISA 2-D. Users were amazed that the previously complex batch processed frame analysis systems were reduced to entering a number in a spreadsheet and INSTANTLY the entire frame was recalculated. At the 1987 Lotus Developers Conference in Boston, MA, the actual software authors of Lotus 1-2-3 were stunned to see their "business tool" doing complex analysis for multi story buildings!

With the decline of DOS and Lotus 1-2-3 , **ENERCALC** rewrote the entire "user interface" portion that provided the calculation screens and printing. Keeping the same "look and feel" to ease the change for users, our programmers wrote our own user interface program, as simple and fast as 1-2-3, complete with support for hundreds of printers. Version 4.4 for DOS was released in August of 1994, and produced a large increase in ENERCALC's user base. Version 4.4 for DOS has become known as the "Volkswagen of structural engineering software"... simple, enduring, yet designed to get you almost anywhere.

Structural Engineering Library 5.0 for Windows was introduced in 1996 as a completely new rewrite of the legacy systems of the previous 15 years. Although much of the proven "C" language engineering calculation processes were retained, the rest of the system was redesigned and written from scratch for the modern Windows based computer systems. The days of a spreadsheet based program were now gone, and a new system designed to be as easy as 1-2-3 was introduced.

In 2007 ENERCALC released **Version 6** of **Structural Engineering Library**. Over the course of three decades since ENERCALC began we've enjoyed a large, loyal and consistently growing base of users. This new version is a **complete rewrite**....the first of its kind. New solvers, graphics, reporting, user interface, and database designs prepare this new platform with the future in mind. We look forward to years of enhancements based on this new release!

In 2017 ENERCALC released the next generation of Structural Engineering Library. With this release, we de-emphasize the "version" number, and rely solely on the build number to identify the release. The software is now based on new development tools that are available to make processes more efficient and to provide a fresh, new look.

Sincerely,

Michael D. Brooks, P.E., S.E.
President & Founder
ENERCALC, INC.

1.3 Warning & Disclaimer

Although it is our intent that the information contained in this manual and associated software program is accurate and reliable, it is possible that there may be errors, both of omission and commission, that we are not aware of at any time. ENERCALC, Inc. can make no warranties, either express or implied, as to the accuracy of the material in this manual and software nor its suitability for a specific purpose or application for which it is advertised.

ENERCALC, Inc., its owners, directors, and employees, can offer no guarantee and will accept no liability for damages of any kind resulting from the use of the information contained or generated by this document and the accompanying computer software.

If you do not agree to be bound by these conditions and the conditions contained in the License Agreement 8 contained herein, then you may Internet deactivate the program and uninstall the software within the trial period after the date of your order and request a full refund of the License Fee.

1.4 License & Copyright

The complete License Agreement can be viewed by using the following link:

License Agreement

1.5 End of Service Policy

At some point in the future the current version software will undergo a major overhaul of capability increase. At this time the prior version reaches its "**End of Service**" time.

When a version reaches its "End of Service", access to support and maintenance will cease in about 6 months (the time frame will be set by ENERCALC and is at our discretion). The software will continue to operate but technical support, updates, and other support related items will cease to be available.

All software products, Structural Engineering Library included, have a practical commercial lifetime. In order to provide the highest quality products and support to our customers, each product is developed utilizing a product life cycle methodology, which includes an End-of-Service (EOS) phase.

The ENERCALC product EOS policy is to support the current release plus the previous (one back) release for up to six (6) months by default. After this time, ENERCALC's product development ceases active development and support of that software release within the Maintenance and Support Plan. ENERCALC does not create or make available maintenance releases or patches for software that has reached the EOS milestone.

During the EOS phase, ENERCALC will continue to investigate, troubleshoot, and characterize issues in an attempt to provide solutions and workarounds using the production releases. If a solution cannot be found using software that has reached the EOS milestone, ENERCALC will suggest that the system be upgraded to a more recent software release.

Once the EOS process starts on a product release, a notice will be posted on all relevant pages stating that the product release has entered the EOS process.

Note: This policy is subject to revision.

Part II

2 Installation & Activation

General

Structural Engineering Library (**SEL**) must be installed on each computer where it will be used.

SEL has a new license security system that activates the software.

This requires a copy & paste of the **Product Control Code** (**PCC**) into the activation section of the program and then using the [**Internet Activate**] button to obtain activation permission from our Internet Activation server.

You can install SEL on ANY computer that you may wish to use it on, BUT before you can RUN the program, you must Activate it.

This means you can easily move the activation of the software around between all of your computers. You can just use this Internet based activation and deactivation ability to use the software wherever you wish.

Important: Anti-Virus software and Firewalls

Our **Internet Activation** system and Web Update system use the Internet with typical "http over port 80" communications. You may have to tell your anti-virus and firewall software to allow our programs to communicate over the Internet. The important program file names are **EC6.exe** and **ec6webupdate.exe**, and they are typically installed in the Program Files\ENERCALC_6 folder. You will may also need to give your computer permission to communicate with enercalc.net. (Note that it does NOT use the www. prefix.)

If you do not perform this task then your activation request may not receive a response.

There are other methods of manually activating the software if you do not have an Internet Connection. Please see the "For Our Users" page at www.enercalc.com/users.html under ***Manual License Activation*** and also the last section in this guide.

2.1 License Types

Structural Engineering Library can be licensed in a variety of formats:

Annual Subscription (Installed on your computer)

You will receive a Product Control Code that will allow you to activate the software. After you have entered your Product Control Code and performed an activation, the product will be completely operational. Your user registration number and licensee name will appear on all printouts and you will see a subscription expiration date on the licensing screen. Usage ends when the subscription expires or is canceled, but all Project Files remain intact.

Monthly Subscription (Installed on your computer)

You will receive a Product Control Code that will allow you to activate the software. After you have entered your Product Control Code and performed an activation, the product will be completely operational. Your user registration number and licensee name will appear on all printouts and you will see a subscription expiration date on the licensing screen. Usage ends when the subscription expires or is canceled, but all Project Files remain intact.

Perpetual License (Installed on your computer)

You will receive a Product Control Code that will allow you to activate the software. After you have entered your Product Control Code and performed an activation, the product will be completely operational. Your user registration number and licensee name will appear on all printouts and you will see a Maintenance & Support Plan (MSP) expiration date on the licensing screen. When MSP expires, you can choose to renew MSP to keep up to date on new versions. If you do not renew your MSP, then you can continue to use the software indefinitely at whatever version you have when MSP expired.

Academic License

This is a special version/mode for students. Each time an Academic license is granted an expiration date is set after which time the software stops operating. In addition the printouts have a watermark stating that it is an educational version and the student's name and registration number are printed. Click here for an application for an Academic License.

Plan Check License

This is a special version/mode for public plan review agencies. Each time a Plan Check license is granted an expiration date is set after which time the software stops operating. In addition the printouts have a watermark stating that it is a plan check version and the agency's name and registration number are printed. Click here for an application for a Plan Check License.

ENERCALC SE Cloud Subscription

This subscription offering allows access to our cloud-based applications: STRUCTURE, EARTH, and ENERCALC 3D. STRUCTURE is the cloud format of Structural Engineering Library. EARTH is the cloud format of RetainPro. ENERCALC 3D is our general purpose 3D finite element analysis and and design application. No installations are required for ENERCALC SE. The applications run through your Internet browser, and data is stored in our secure online storage for universal availability and ease of collaboration.

2.2 Installation Overview

General

For the most current installation information you need to review this document located on our website:

http://www.enercalc.com/pdf/EC_V6_Install_SingleLicense.pdf

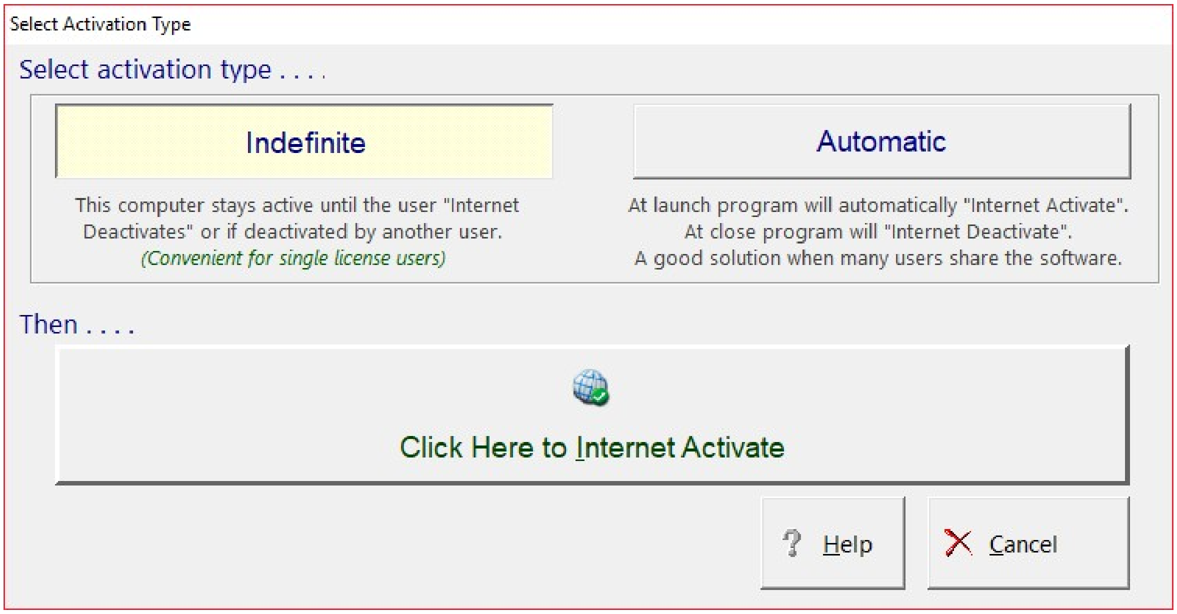
Structural Engineering Library (**SEL**) must be installed on each computer where it will be used. Click here for a video on installation: Installation

Feel free to install the software on any computer where you may want to run it, but keep in mind that you will need to Activate an installation before it will actually run. More on that in the Activation section...

When using our installation program you will be asked to paste in your **Product Control Code** (**PCC**) for security. Then simply follow the on-screen prompts to complete the installation.

2.3 Activation Types

When Internet activating SEL, the following activation types will be offered:



Click here for a video on Internet Activation: Internet Activation

The activation types behave as follows:

Indefinite:

This option saves activation info to your computer, so it will result in an installation that will remain activated until it is Internet deactivated by the user, or until another user forcibly deactivates this computer.

This option is useful in situations where the number of users is equal to the number of available seats, such that there is no need to share seats among multiple computers. It is also useful in situations where a laptop will be taken to a remote location where Internet access is uncertain.

Automatic Activation/Deactivation:

This option automatically activates the program when it is launched (if a seat is available), and it automatically deactivates when the user exits the program.

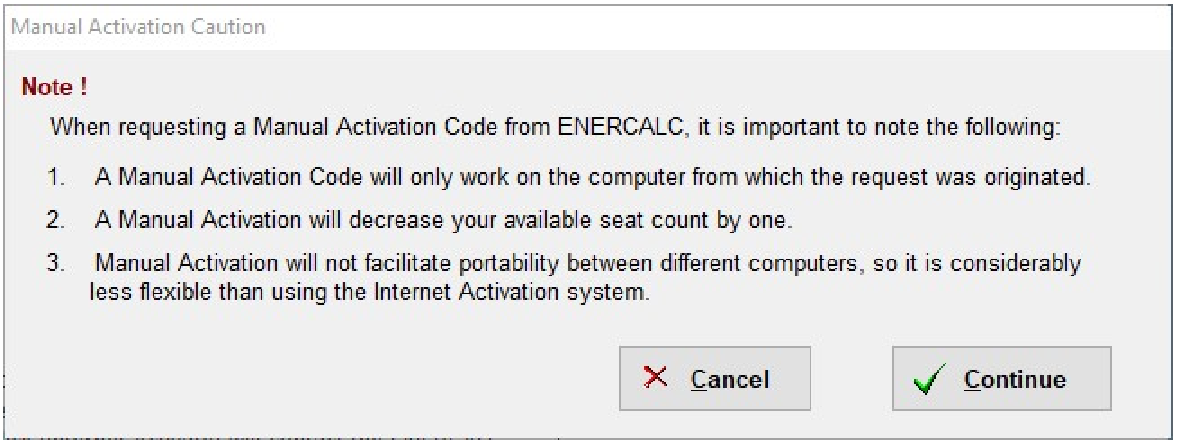
This option is useful in situations where the number of users is greater than the number of available seats, such that there is need to share seats among multiple computers.

Note: To change to a different activation type, such as to change from "Automatic" to "Indefinite", simply Internet Deactivate temporarily. Then Internet Activate once again and you will be offered the "Select Activation Type" dialog where you can choose the desired activation type.

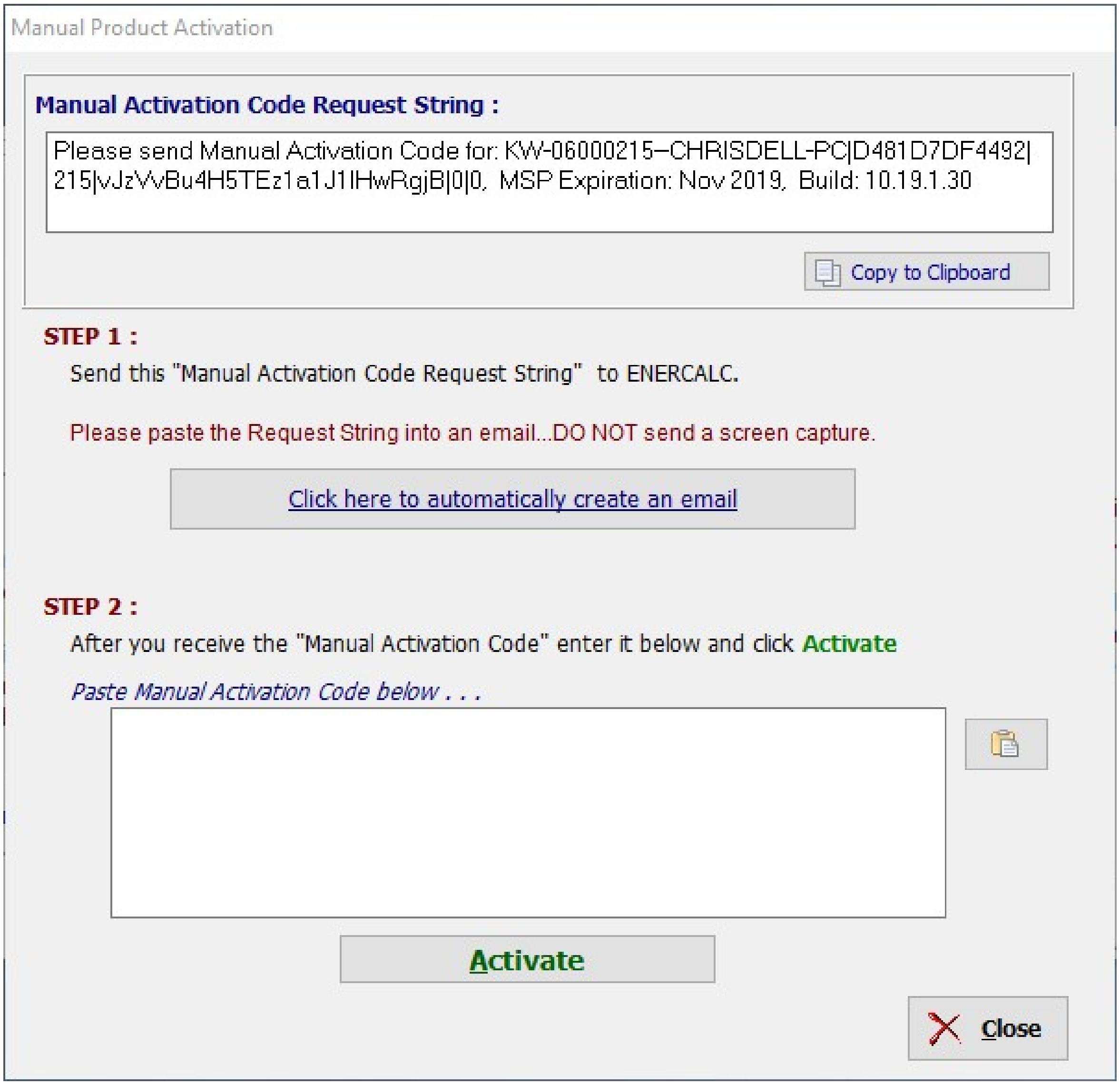
2.4 Manual Activation

If you cannot activate the software using the simple [**Internet Activate**] method (because you have no connection, restricted access to the Internet due to a firewall, or your company prefers not to activate your installation this way), you can activate your software using the [**Manual Activation**] method.

Click the [**Manual Activation**] button. The system will provide some information in a Caution box:



Then instructions will be provided to you using the screen shown below:



NOTE: When using the Manual Activation method, the Manual Activation Code that you receive from ENERCALC can only be used to activate the ***specific machine*** that was used to make the Manual Activation Code request, because the request and the code actually contain the Computer Name and other data that is specific to that particular machine. However, you can still use the Internet Deactivate function to return a manually activated seat to your pool. And then you would be able to use that available seat from your pool to perform a normal Internet Activation on a different computer if desired.

2.5 Moving & Maintaining Your Activation

A Structural Engineering Library license has some number of "seats" associated with it. Each activation uses one of your available seats. When an installation gets deactivated, that seat returns to your pool of available seats.

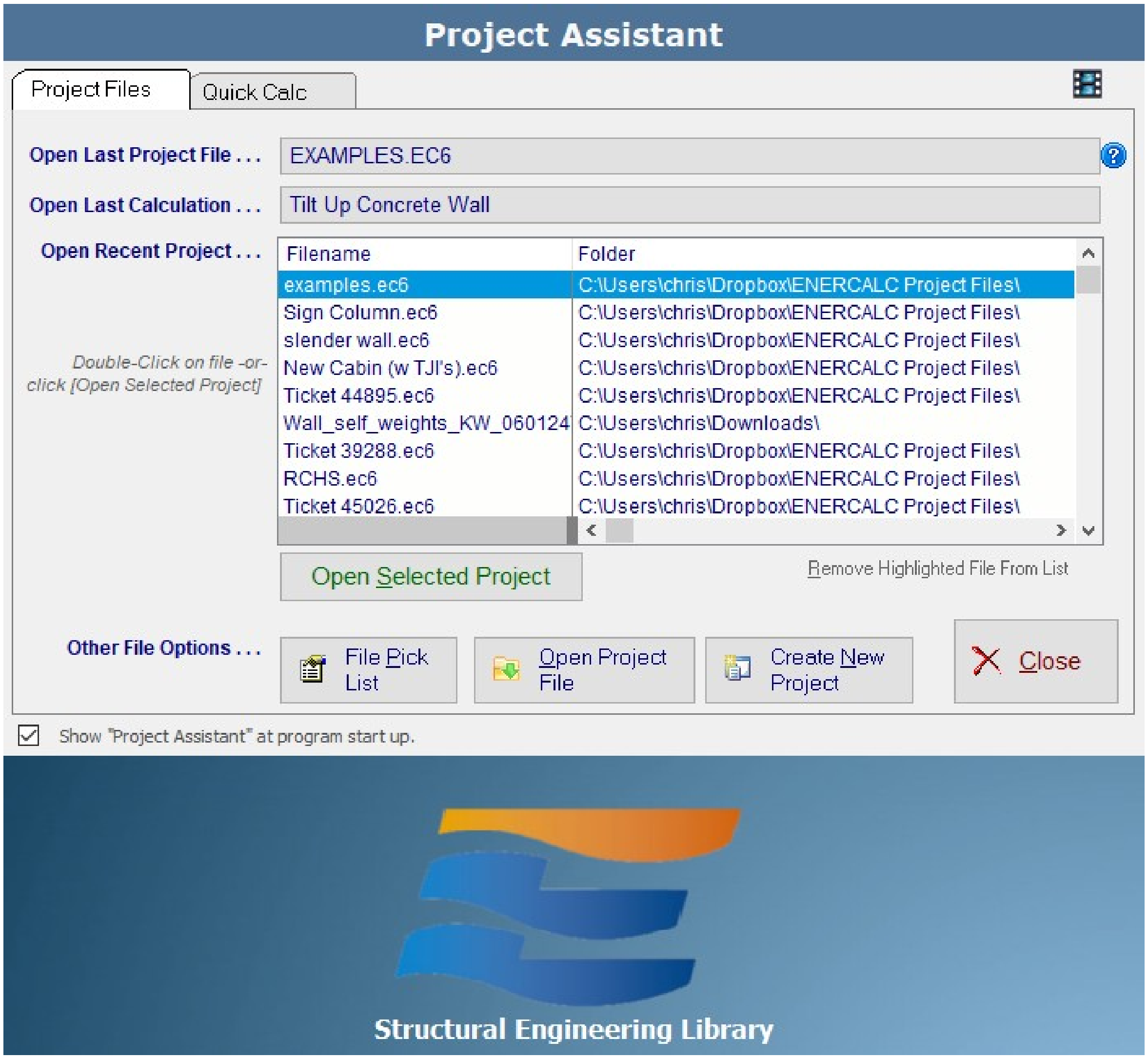
This section will assist you in moving your activation from one computer to another.

Moving Your Activation

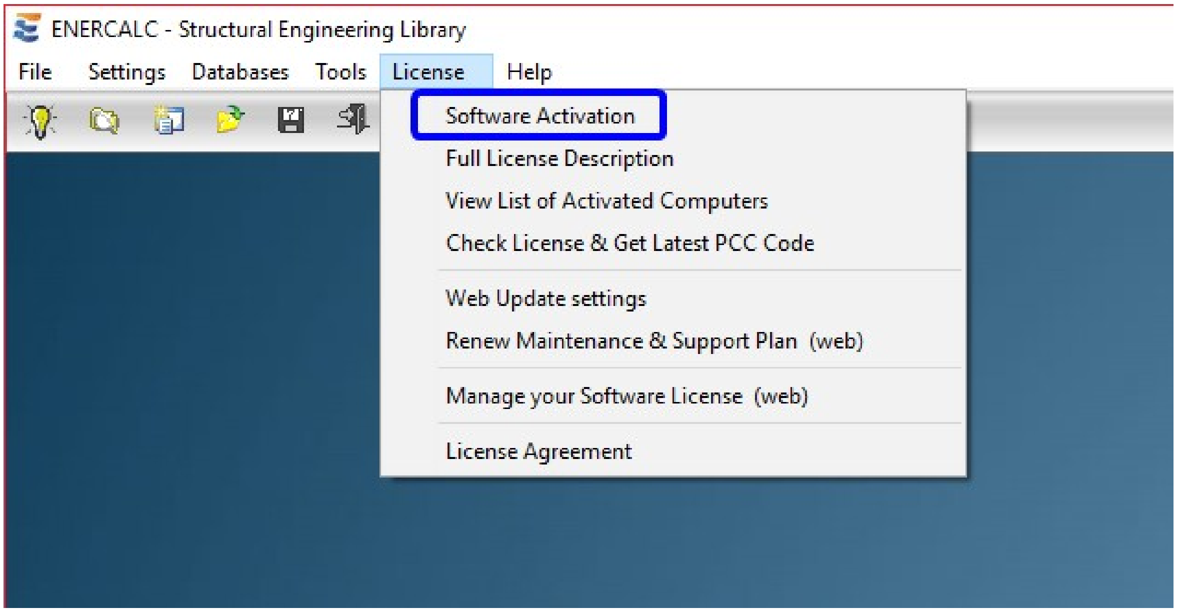
When you originally activated the software, if you used [**Internet Activate**], then you can use [**Internet Deactivate**] to return this activation to our server, so that it can be available to another computer.

Follow these steps:

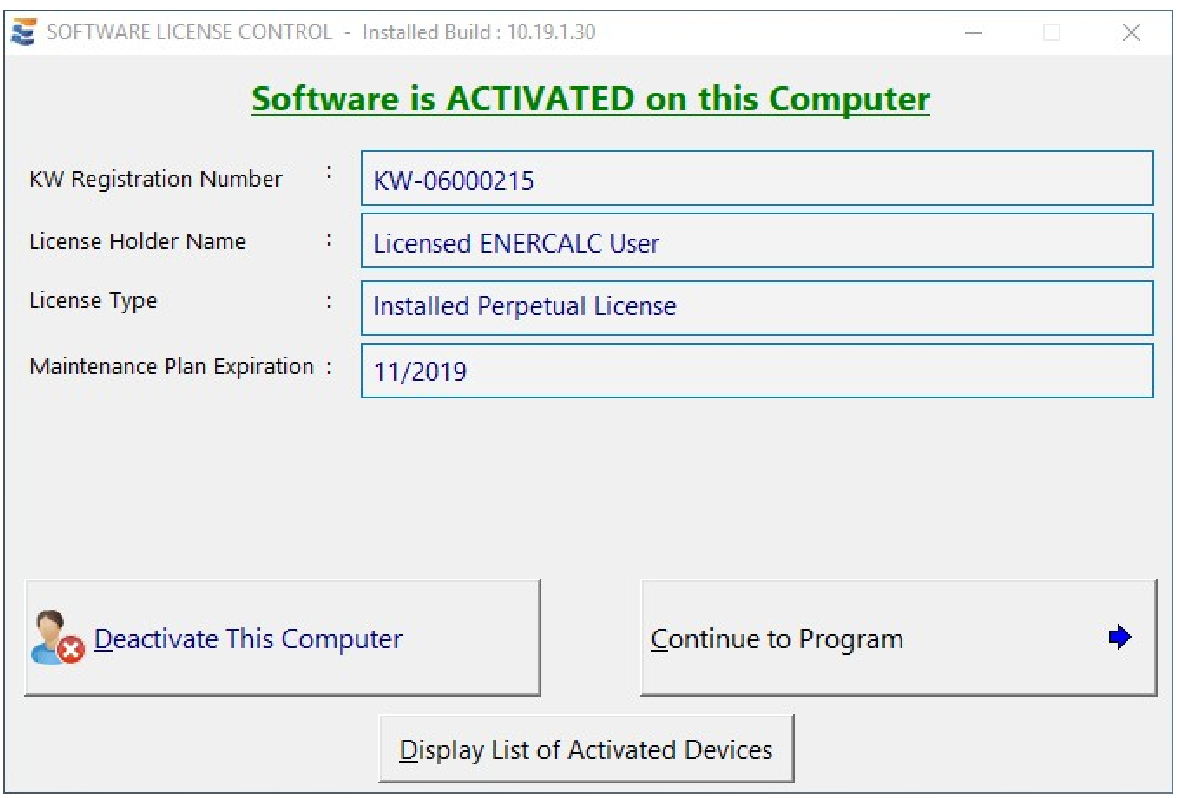
Start **Structural Engineering Library**. You will probably see the Project Assistant screen as shown below:



Click the [**Close**] button. Click **License > Software Activation** from the main menu as shown below:



You will now see the Software License Control dialog as shown below:



.

Click the [**Deactivate This Computer**] button to return your activation to our server.

Note! You must allow the ENERCALC file "EC6.EXE" Internet access through your firewall or anti-virus software. Also, be sure to trust enercalc.net (note no www.).

After a moment, the system will indicate that the installation has been successfully deactivated by offering an [**Internet Activate**] button in the Software License Control dialog as shown below:

Your activation has now been returned to our server, and you can go to another computer where **Structural Engineering Library** is installed, and activate it by clicking **License >Software Activation >** [**Internet Activate**] from the main menu.

Part III

3 Software Updates

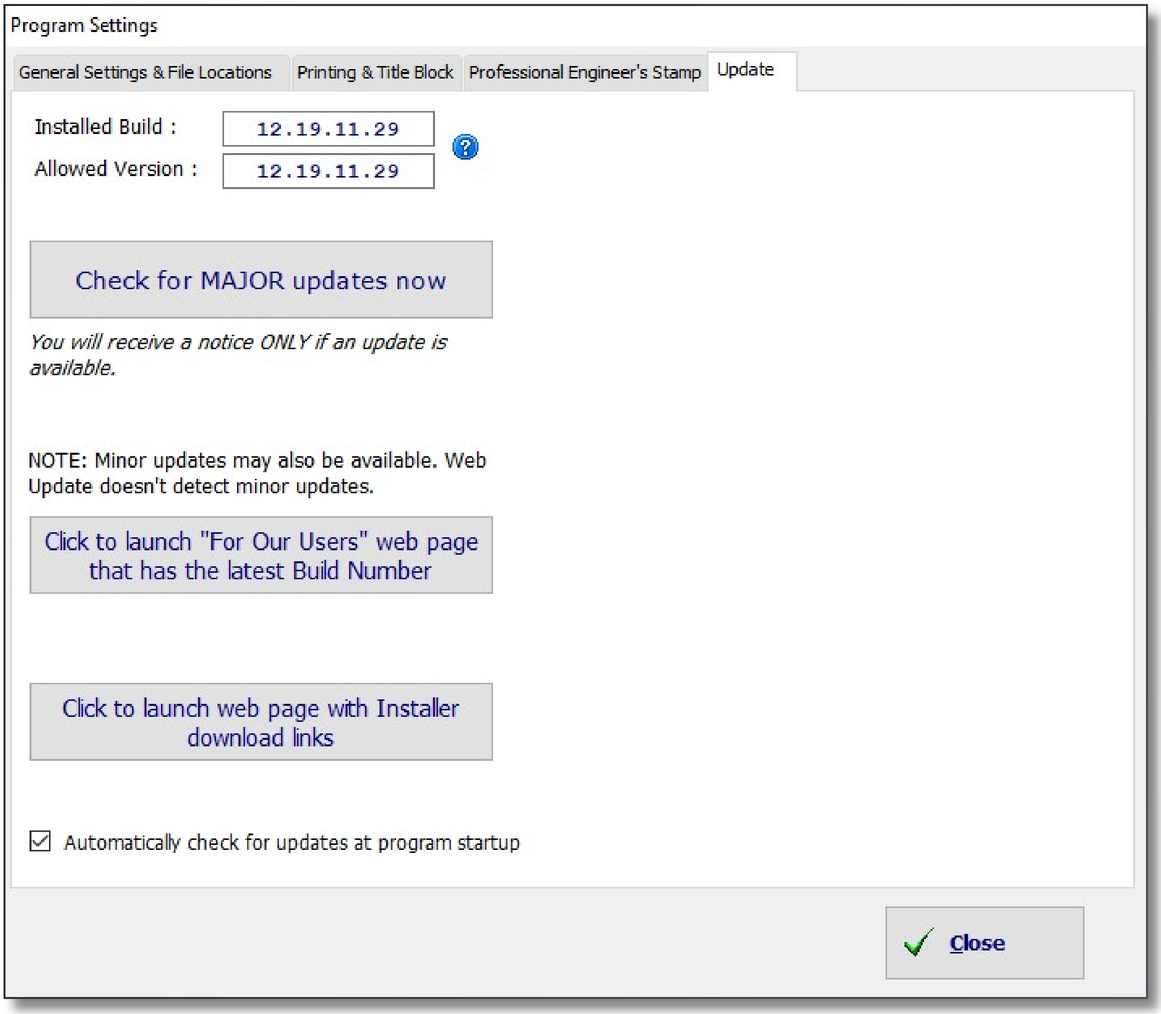
3.1 Web Update

At **ENERCALC** we are continually improving and enhancing our software. It is fairly simple to make changes to the software here in our offices, however it can be overwhelming to ship thousands of CDs to our user base. To distribute the latest software builds to the entire user base, we depend on a Web Update system. The update is provided in two ways:

Web Update System: This system is built into your **Structural Engineering Library** software package. It will check with our server to see if a newer version of the software is available and prompt you with an option to install it if available. See *What Actually Happens* below for a description of how it works.

The Web Update system uses Internet protocol HTTP on port 80 to check if a newer version of the software is available and to transfer those files from our server to your computer as needed. You can configure the program to perform this check automatically every time you start the software, but it can also be performed on an on-demand basis if desired.

Select **Settings > Updates** from the main menu to display the web update screen as shown below:



What actually happens during an update check: What actually happens during an update check is that the software runs a program in the ENERCALC program folder named EC6WebUpdate.exe. This program connects to our Internet servers using the "HTTP" protocol on port 80. It compares the version number you have installed with the current build of the software stored on our servers. If there is a newer build available, you are notified and upon approval, a small update installation program named EC6_WebUpdate.exe is transferred from our server to your computer. This file then executes to complete the file download and software updating/installation process.

EC6WebUpdate.exe and EC6_WebUpdate.exe are digitally signed applications that are secure to run and are virus free.

The actual updating program EC6_WebUpdate.exe that is downloaded from our server will need "write" access to the program installation folder.

Check for MAJOR updates now: This button will download the updating program from our server. This program will check your computer for a non-expired Maintenance & Support Plan and update the software to the latest allowed version.

Note: The "WebUpdate" procedure may send information to ENERCALC, Inc. about your installation and use of the software licensed from ENERCALC, Inc. This may include any of the following: your ENERCALC assigned User Registration Number, the Installed Build Number of your ENERCALC software, Internet IP address of the computer that will receive the updated files, time usage for the various portions of the software, and potentially other information only related specifically to the use of the software license. Absolutely no files, configurations, settings, or other information not specifically regarding the usage of the ENERCALC license will be sent. If you are concerned about this, please contact us for information on what is being sent. We have an open policy on providing you with information showing what might be included.

TECH NOTE: Some users with multiple installations may prefer to set the status of the automatic check directly through the registry entry for convenience:

REG_CURRENT_USER\Software\ENERCALC\V6\DoCheckForUpdates

Yes = 1
No = -1

3.2 Update from Website

In addition to the Web Update system that is built into the software, it is also possible to initiate updates by visiting the "For Our Users" page of www.enercalc.com and using the UPDATE Structural Engineering Library link.

Note:
There are times where a more recent build will be available on the website than is offered/detected by the built-in Web Update system. The reason for this is as follows. The built-in Web Update system offers those updates that are regarded as "major" or that have a significant impact on a majority of the users. On the other hand, the website will always offer an update for the absolute latest available build of Structural Engineering Library, regardless of whether it is categorized as a "major" update or not.

Part IV

4 Support & Maintenance

4.1 Maintenance & Support Plan

Our **Maintenance & Support Plan (MSP)** ensures that you will always have full technical support and access to the latest build of the software during the term of the plan.

While your MSP is current, you will receive every new build, feature enhancement, and improvement for your licensed software, ensuring you're always using the most current technology.

You will have access to voice/fax/email technical assistance from our staff.

You will also receive discounts on new software releases and additional software licenses. This is the easiest and most economical way for you and your company to keep your software investment current.

Please see this web page for details on the Maintenance & Support Plan:

http://www.enercalc.com/support_maintenance.html

4.2 Getting Assistance

There are several ways to get assistance with using **Structural Engineering Library**.

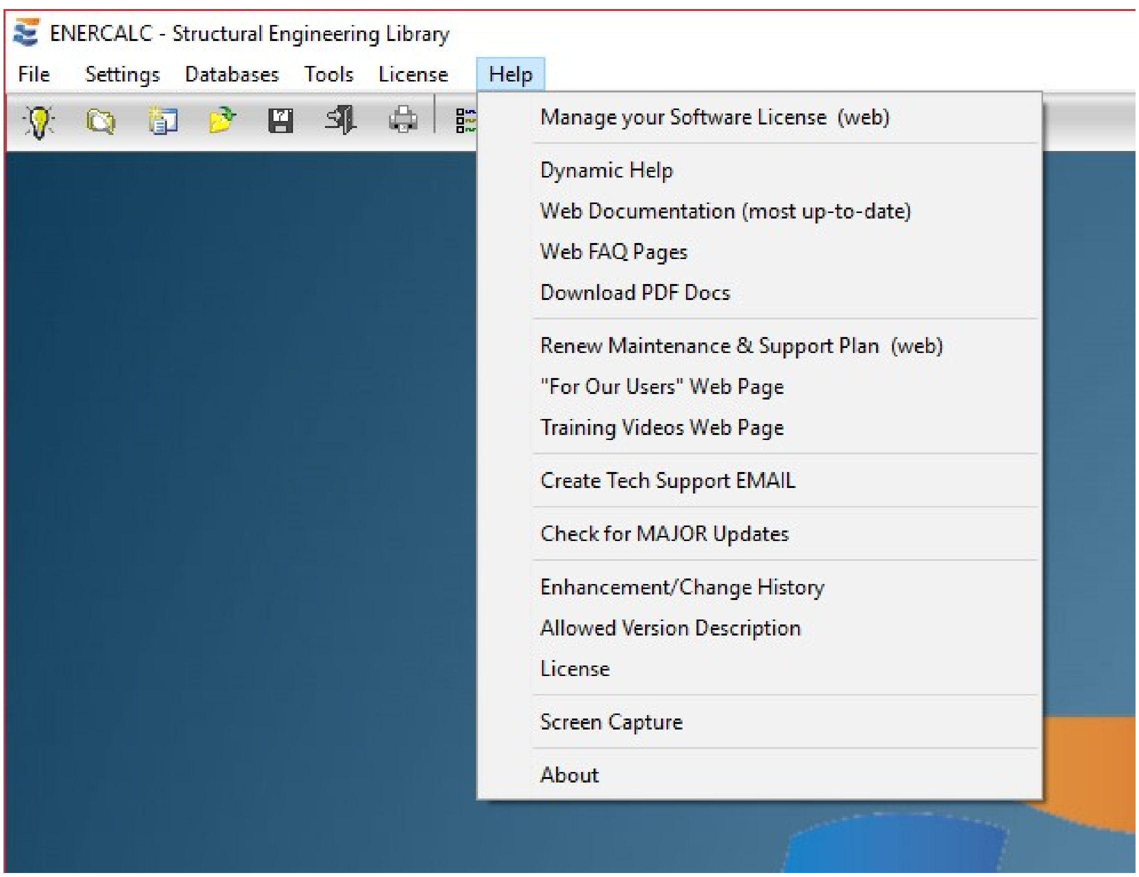
Click here for a video: Getting Assistance

HOWEVER, with the exception of referring to the program documentation and reviewing the FAQ section on the website, all options require that your Maintenance & Support Plan is current.

- Refer to the program documentation.

- Contact ENERCALC Technical Services by email:

  From the main menu:



  From your email program: support@enercalc.com

- Contact ENERCALC Technical Services by fax: 949-645-3881

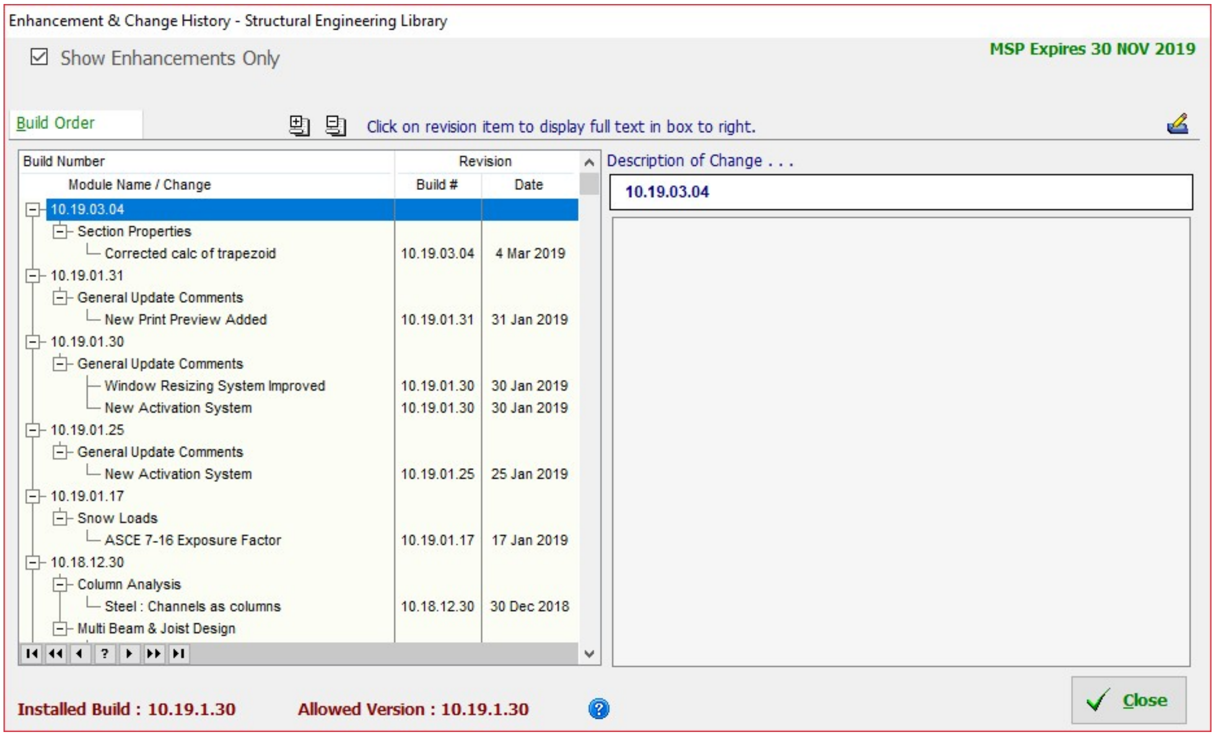
- Contact ENERCALC Technical Services by phone: 949-645-0441, Extension #3

- Review the "Frequently Asked Questions" (FAQ) page on our website:
  http://www.enercalc.com/support_knowledge.html

4.3 Viewing Enhancements and Changes to the Software

The program offers a detailed list of revisions that have been made to the software.

Click **Help > Enhancement/Change History**. The following window will be displayed:



This list can be set to display enhancements ***only***, or it can display enhancements ***and*** changes/corrections. When the list is set to display both enhancements and changes/corrections, it can be sorted by version or by module.

Part V

5 Getting Started

5.1 System Overview & Design Philosophy

Structural Engineering Library is a collection of modules that provides analysis and design functionality for components of buildings.

Walls, columns, beams, footings, diaphragms, frames, and other common elements can be thoroughly engineered through the use of the modules in this package. If you are a typical engineer whose work consists of a regular flow of small and medium-sized projects, ***this package is designed specifically for you***.

As an engineer you will find that each module combines the governing code provisions, mathematical analysis processes, and commonly available construction materials into a simple and effective "calcpad" style fill-in-the-blanks program. You can feel partially relieved that the software will consistently perform all the required checks that may be skipped over when doing repetitive hand calculations....especially when fatigue sets in and a deadline is near! You can enjoy the time to do more exhaustive design studies, come up with safer and more economical designs, and enjoy clearly documented calculations for review and archiving.

This software is not a "black box" program. Each calculation is designed to be a "visible calcpad" where you can work with the data and immediately view the resulting calculations. Automatic design is provided in most modules, and is intended primarily to automate tedious iterative tasks.

You, as an experienced structural engineer or architect, can quickly enter and change member sizes and other design parameters and view the results. In this way, **Structural Engineering Library** maximizes the use of your time and design skills by enabling you to quickly define a concept and then make necessary modifications to refine it into a final design.

5.2 About Our Documentation

Documentation of your software package is essential for your successful and pleasant use of our products. We try hard to supply you, now and on an ongoing basis, with detailed information on all aspects of the software. To support this commitment, we provide documentation of your software in multiple forms:

- A **Windows Help system file** named ENERCALC.CHM is installed with your software. This help system may be accessed by clicking **Help > Dynamic Help** from the main menu.
- A **User's Manual** in Adobe Acrobat PDF file format is available for download at any time from our website http://www.enercalc.com/pdf/SEL60_DOCS.PDF.
- An **Online Help system** is available at www.enercalc.com/sel_help.

Printed Documentation

Printed documentation is not provided with ENERCALC products. This is in keeping with the nearly universal industry move away from printed documentation.

Updating your Documentation

The most up-to-date documentation for our software products is always available in electronic form. Whenever the software is updated with the built-in Web Update system, part of the process includes transferring the latest documentation to your computer. This ensures that the content that you view by clicking **Help > Dynamic Help** from the main menu is current and coordinated.

5.3 Building Codes Supported

As of May 2019, SEL supports the following codes and design standards:

Design Standards:

- American Concrete Institute publication ACI 318-05/08/11/14
- American Concrete Institute publication ACI 530-05/08/11/13
- American Forest & Paper Association publication NDS 2005/2012/2015/2018 Editions
- American Society of Civil Engineers publication ASCE/SEI 7-05/7-10/7-16
- American Institute of Steel Construction publication AISC 360-05, -10, and -16
- The Masonry Society publication TMS 402/602-16

General Building Codes:

- International Code Council publication International Building Code 2006, 2009, 2012, 2015 and 2018 editions. IBC references the above publications in Chapter 35. The modifications to each referenced design standard have been incorporated if applicable to the functionality provided by the individual calculation modules.

- California Building Standards Commission publication 2007 California Building Code, 2010 California Building Code, 2013 California Building Code, 2016 California Building Code and 2019 California Building Code. Chapter 19 references ACI 318, Chapter 21 references ACI 530/TMS 402/602, Chapter 22 references ANSI/AISC 360-05, Chapter 23 references NDS, and Chapter 16 which defines the forces on buildings is essentially the same as ASCE 7-05/ASCE 7-10/ASCE 7-16.

5.4 Learning Structural Engineering Library

There are several sources of information that will assist you with learning to use **Structural Engineering Library**.

- A series of tutorial videos is available at our website:

  http://www.enercalc.com/training_videos.html

- This documentation. An Internet version of the help system is available at:

  http://www.enercalc.com/sel_help

- Responses to "Frequently Asked Questions" are available at our website:

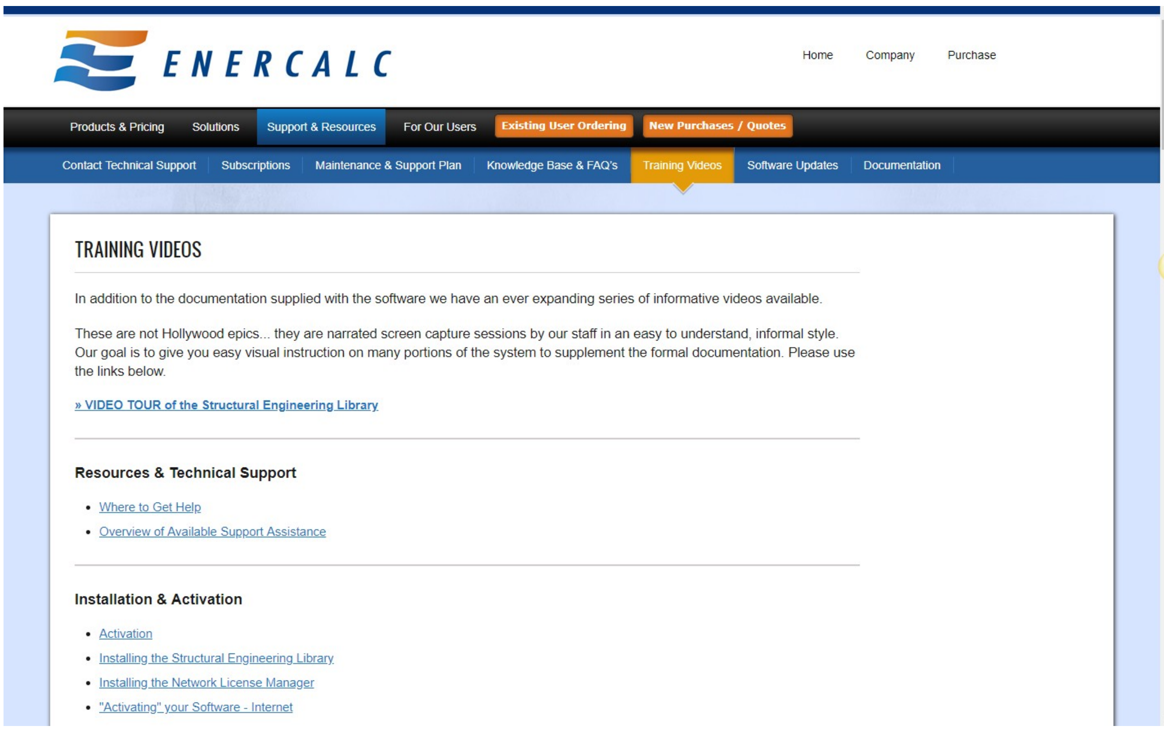
  http://www.enercalc.com/faq_help/

5.5 Introductory Videos

We continue to add topics to a series of videos that discuss all portions of the software.

To access nearly 40 videos please click here: http://www.enercalc.com/training_videos.html

Here is a partial view of what you will see....



5.6 Request for Suggestions

Although our intentions are to provide you with the best product possible, it is likely that there may be areas of this User's Guide and the software itself that could be improved to better suit your needs, be made simpler to operate, easier to understand, or support engineering technologies that have emerged since this publication.

To call these to our attention and to offer suggestions for improvement, we sincerely request that you send us your thoughts. Please address your comments to support@enercalc.com, or:

Michael D. Brooks, P.E., S.E.
President
ENERCALC, Inc.

support@enercalc.com

Post Office Box 2208
Newport Beach, California, USA 92659

Part VI

6 Program Overview

6.1 Introduction

Structural Engineering Library is a collection of modules (also referred to as "calcsheets") that provide functionality for the analysis and design of components of buildings. Walls, columns, beams, footings, diaphragms, frames, and other common elements can be thoroughly engineered through the use of the modules in this package. If you are a typical engineer, whose work consists of a monthly flow of small and medium-sized projects, this package is designed especially for you.

As an engineer, you will find that each module combines the governing code provisions, mathematical analysis processes, and commonly available construction materials into a simple and effective "calcpad" style fill-in-the-blanks program. You can feel partially relieved that the software will consistently perform all the required checks that may be skipped over when doing repetitive hand calculations. You can enjoy the time to do more exhaustive design studies, come up with safer and more economical designs, and enjoy clearly documented calculations for review and archiving.

This software is not a "black box" program. Each calculation is designed to be a "visible calcpad" where you can work with the data and immediately view the resulting calculations. Automatic design is provided in most modules, and is intended primarily to automate tedious iterative tasks.

You, as an experienced structural engineer or architect, can quickly enter and change member sizes and other design parameters and view the results. In this way, **Structural Engineering Library** maximizes the use of your time and design skills by enabling you to quickly define a concept and then make necessary modifications to refine it into a final design.

The "Calcpad" Approach

When **Structural Engineering Library** was designed in 1983, our concept was revolutionary.....design it like an engineer's calculation pad. When an engineer prepares a calculation, the finished product is a neat and organized sheet of paper that follows the design flow from load tabulation, force and stress calculation, to the final adequacy check of the structural component that will satisfy the task.

At the time, all other competing programs were aging versions of mainframe programs that had been modified to run on microcomputers. Many of those programs executed "batch" design, where the user entered all the data, told the computer to run the program, and then opened a crude file to review the results. ENERCALC was unique in that the input and output was mixed on the same screen....easy to see at a glance. But the most revolutionary aspect was the tremendous speed it offered to prepare calculations.

This great speed was due to the fact that you could change a number and instantly see all the updated results on the screen.

Moving forward, the current version of SEL for Windows maintains that same fill-in-the-blanks approach with instant recalculation of results. When using any of the approximately 30 calculation modules, all input data and output results are presented on the same screen and viewed just by selecting a tab that groups the information. **Whenever you change an input value, the entire module is recalculated and the results are immediately visible**. Thanks to efficient programming and fast modern computers, incredibly complex structural analysis and design is performed in a split second.

This instant updating also happens when you are viewing graphical sketches of designs or stress diagrams.....after any change, the graphics are instantly updated.

6.2 Typical Worksession

A Typical Worksession

In order of occurrence, here are the steps in using the software:

1. Start the program.

2. The Project Assistant is displayed where you can choose to use the last calculation, the last project, a recent project, or create a new project.

3. A project is selected and you view the Project Manager. Here you can add new calculations, edit existing calculations, insert non-calculation items into the project (such as Microsoft Word or Excel files, Adobe Acrobat PDF files, or scanned images), and initiate project printing capabilities.

4. Editing or adding a calculation takes you to the calculation sheet for one of the modules.

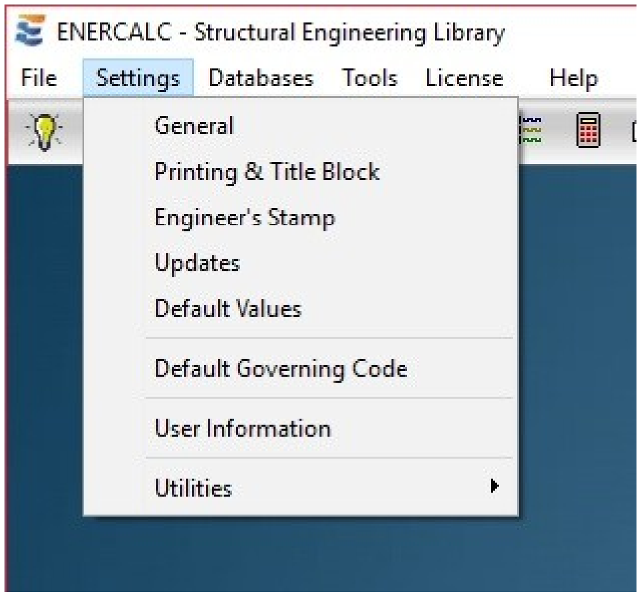
5. Using the calculation sheet, you enter data on the top set of tabs while reviewing results and graphics on the bottom set.

6. When the structural calculation is complete, you can print it and/or save it to the project.

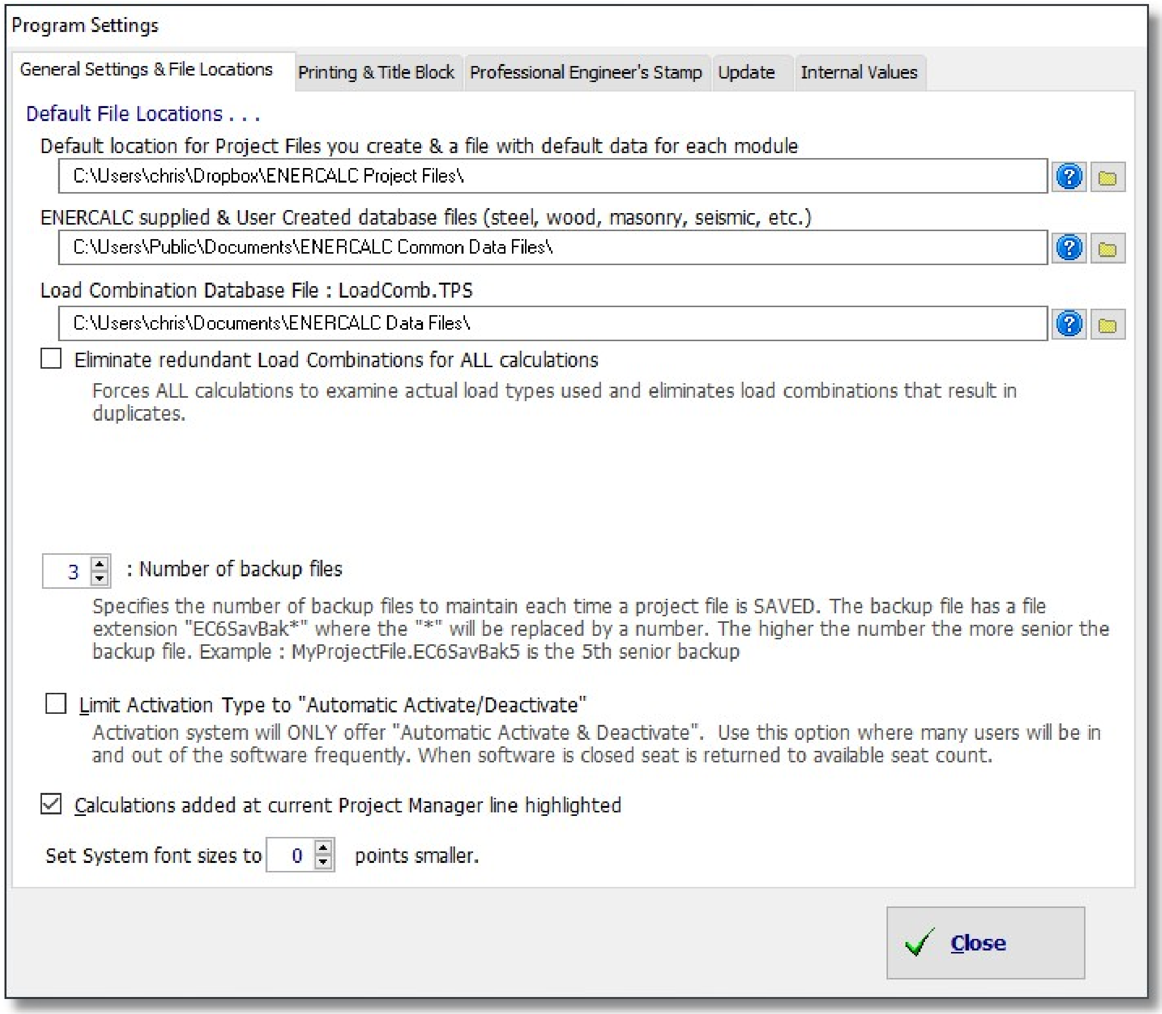
7. You can always return to the Project Manager where you can add/edit/delete/print other items within the current project, or save the current project and open a new one.

6.3 Program Settings

The **Settings** item in the main menu provides access to several selections that control how various aspects of **Structural Engineering Library** operates. A selection of one of the first four items will open the Program Settings window and preselect the appropriate tab for the chosen item.



General Settings & File Locations



Default Project File Location: Specifies the default location where the program will point when using **File > Open** or **File > New** from the main menu.

Because modern operating systems allow multiple users per computer, Microsoft suggests that software manufacturers create their own folder under Documents and set that as the default location for user-created files. The program conforms to this recommendation by suggesting an appropriate directory during the installation process.

Database File Location: Specifies where the program should look for the steel database, wood database, and other files that contain the databases of standard values. This location needs to be accessible to all users on a specific computer. Microsoft suggests that software manufacturers set the following as the default location for data files that are referenced by the software, but not modified by the user: C:\Users\Public\Documents.

Load Database File Location: Specifies where the program should look for the load combination database file and other files that are potentially edited by the user. Because this is a file that can be customized by the user, the default location for this file is the same as the default location offered for storing Project Files.

Eliminate redundant Load Combinations for ALL calculations: With this box checked the software will only run load combinations that are unique. It will automatically eliminate any load combinations that result in a combination that is redundant.

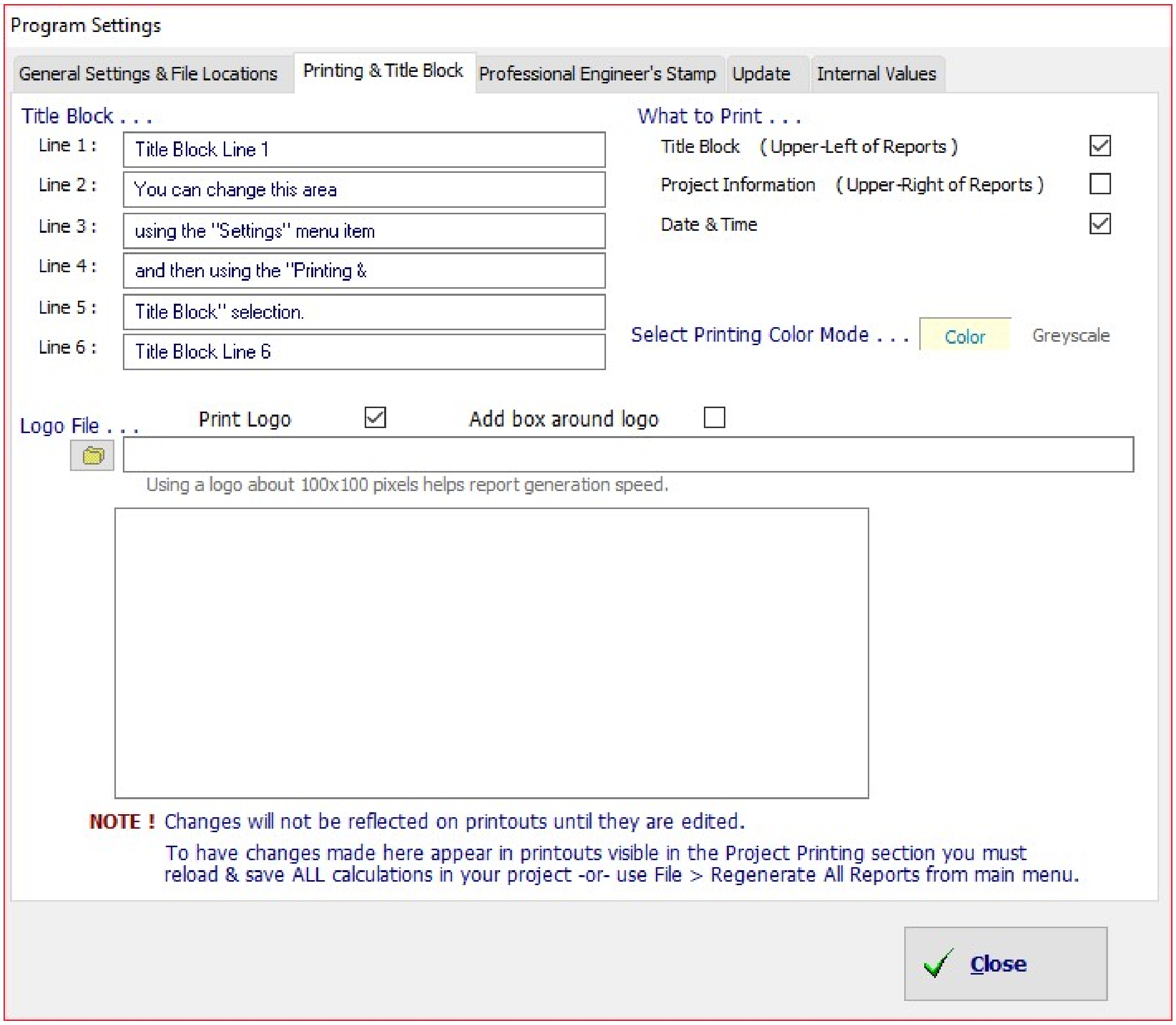
Number of backup files: Specifies the number of backup files to maintain for each Project File. With this box checked the software will automatically create a backup file with the **.EC6SavBak** file extension in the same folder as the original Project File. The backup files are created when a Project File is OPENED.

Limit Activation Type to "Automatic Activate/Deactivate": Removes the option to specify Indefinite activation.

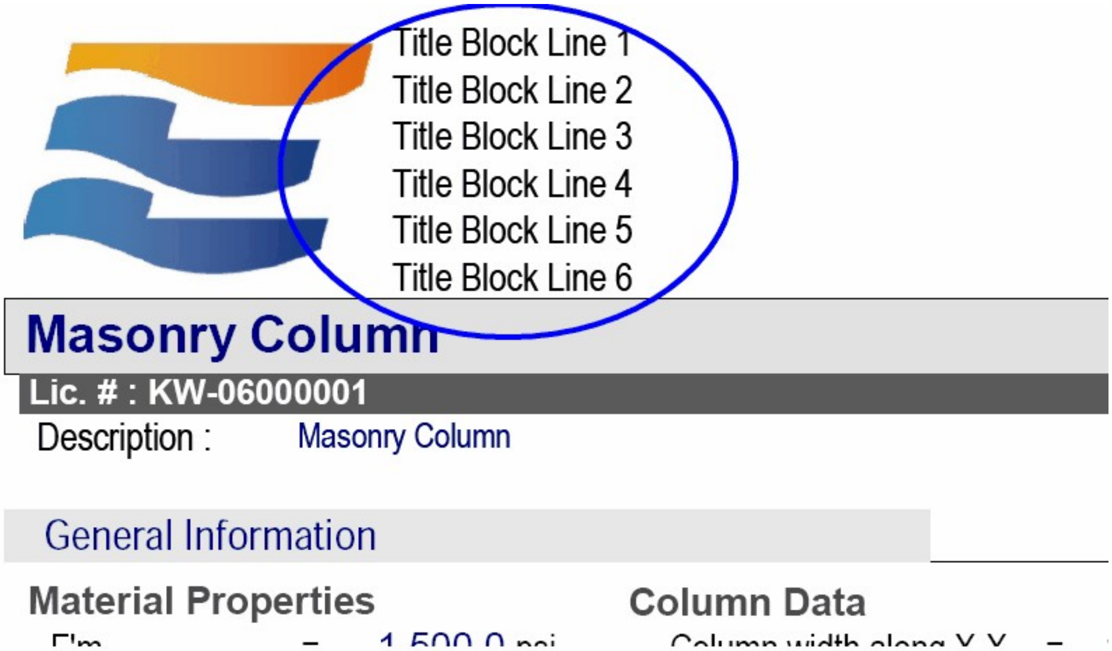
Calculations added at current Project Manager line highlighted: This applies to the way calculations are inserted when using the Project Manager. When this box is checked and you use the [**+Add**] button, it adds a new calculation to the project, and the calculation is inserted directly above that position. What you see is that the new calculation takes that spot in the list, and the highlighted item is moved down. If this box is unchecked and you use the [**+Add**] button, the new calculation is inserted BELOW the highlighted item.

Set System font sizes to X points smaller: Provides a way to scale fonts down, for high resolution displays.

Printing & Title Block



Title Block: These six entries correspond to the six lines in the upper-left corner of the printout.



Logo File: Provides an option to specify a graphics file to be printed at the LEFT edge of the title block. If used, the six lines of Title Block information will be printed immediately to the right of the logo. Logos can be Windows BMP, JPEG, GIF, and WMF formats.

When a logo is specified, the height is adjusted to fit into the title block area and the right-side "floats" according to the width of the image.

What to Print - Title Block: Check this box to print the title block at the top left of the report. With this item unchecked the printout begins with the calculation title bar (shown as "Masonry Column" in the image above)

What to Print - Project Information: Check this box to print the Project Information at the top right of the report.

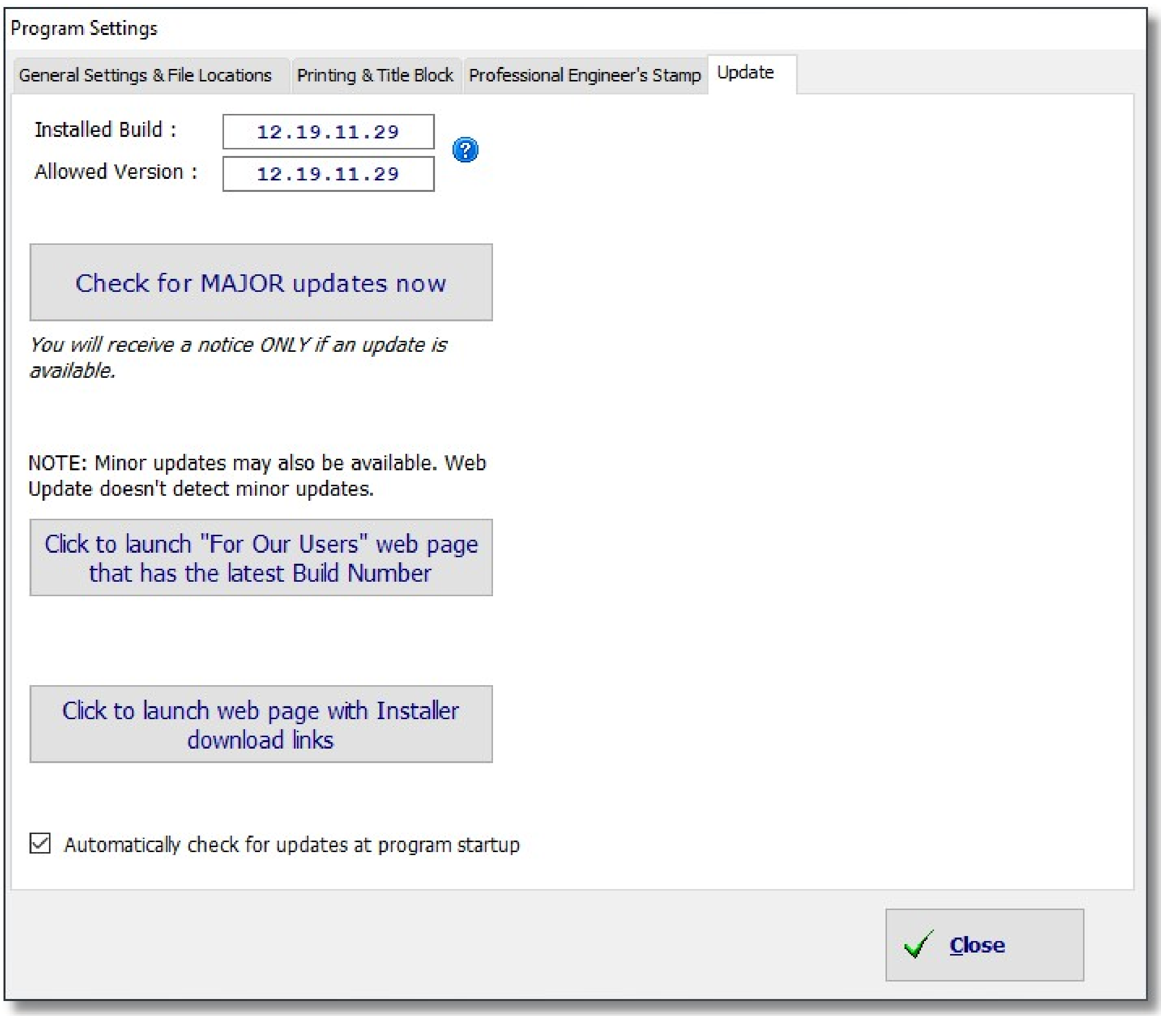
What to Print - Date & Time: Check this box to print the computer date and time on the report as shown below:

Select Printing Color Mode: This sets the default mode for printed reports created from **Structural Engineering Library**.

Professional Engineer's Stamp

This tab allows a graphic image of the user's PE stamp to be uploaded. Then the stamp can be placed on printout by selecting the checkbox.

Update

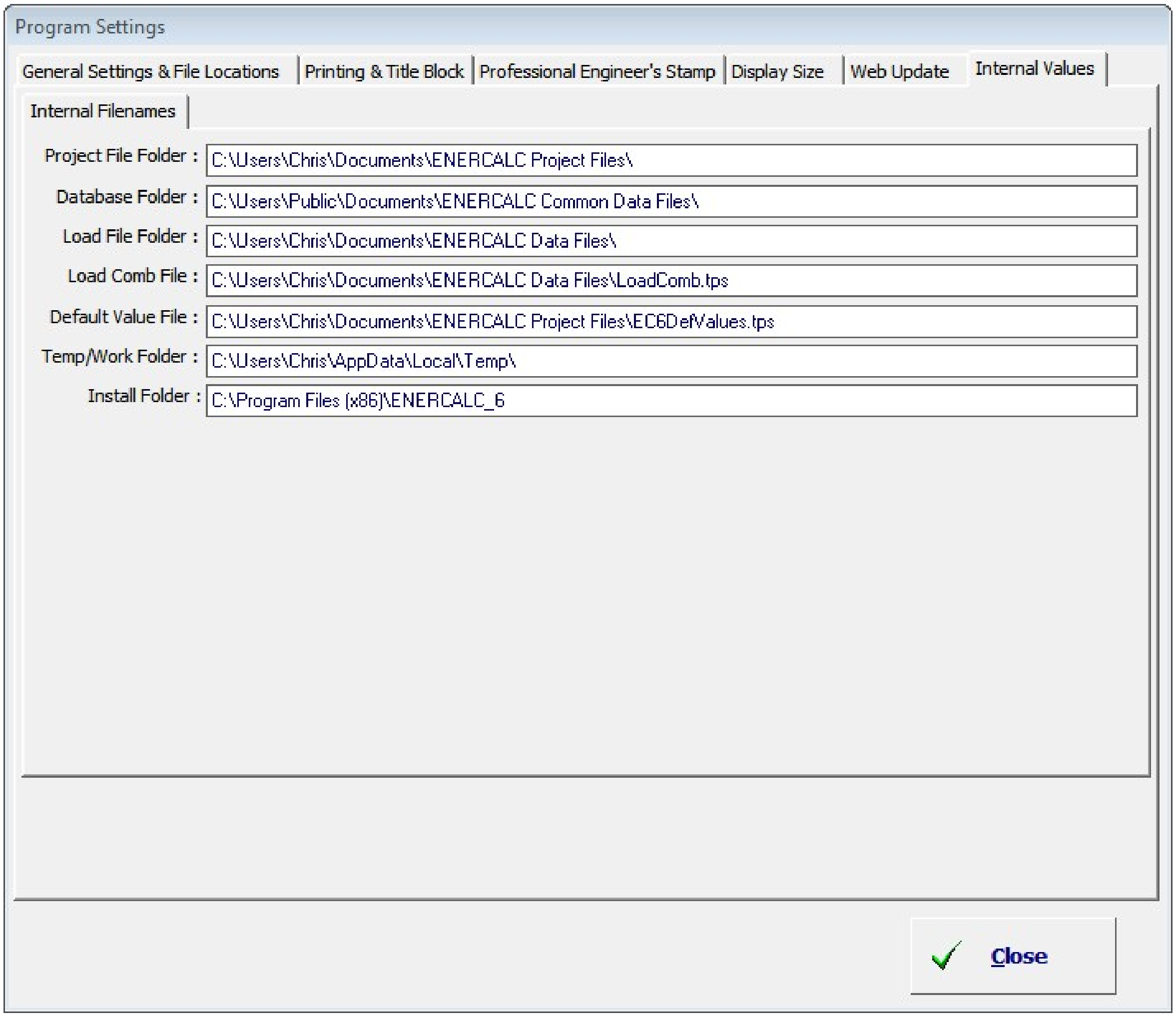


Click here to check for updates: This button will download the updating program from our server. This program will check your computer for a non-expired Maintenance & Support Plan and update the software to the latest version.

Click Here for web update overview 25

Internal Values

Internal Filenames: Displays folder names and file names for the current configuration.



6.4 Files & File Locations

Project Files

Structural Engineering Library uses a single file to store the project information and all the calculations and items that are created as a part of that "Project". This file uses an "EC6" extension.

To backup an ENERCALC Project File simply copy the .EC6 file to the desired backup location.

To have the system automatically create a backup file, activate the checkbox labeled **Automatically create backup files** on the **Settings > General Settings** dialog.

As **Structural Engineering Library** advances in capabilities, the file formats used will change, but we will always provide conversion programs for previously saved Project Files.

General Comments

- Each ENERCALC Project File contains all information on a Project. There are no other files you will need to keep track of.

- Always remember that the Project Manager is showing you the Divisions and calculations for the current Project. It is NOT showing you a disk directory structure.

- There is no **Save** item on the **File** menu. After you have edited a calculation, you can either click [**Save Only**] or [**Save & Exit**] within the individual modules. Either one will save the calculation data to the Project File.

Database Files

A number of database files are supplied with **Structural Engineering Library**. These files contain AISC section properties, NDS stress grades, wood section properties, seismic acceleration data, USA cities & Zip Codes, and other files. These files are not to be edited or modified in any way by the user.

User-Created Database Files

The user can create User Defined database files to store their own steel sections, wood sections and wood stress databases. These files are created and stored in the same folder as the other database files. User defined database files are differentiated from the standard database files that are delivered with the software by inserting the word "_USER" in the filename.

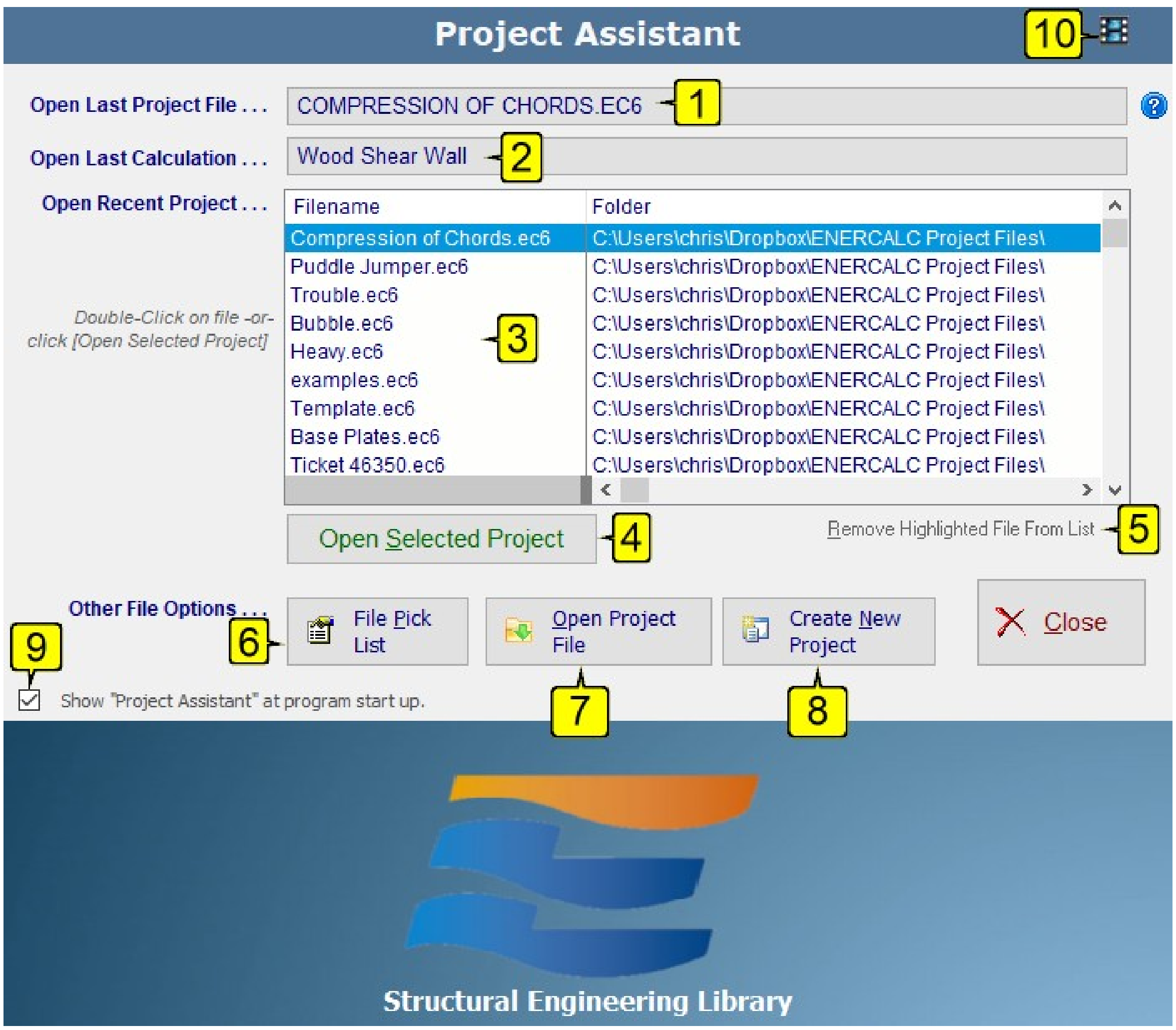
6.5 Project Assistant

When you launch **Structural Engineering Library** the Project Assistant is displayed by default, as shown below.

Click here for a video: Project Assistant

This single dialog allows you instant access to prior Project Files, the last Project File, the last-used calculation, or other functions to create or manage Project Files.

Please see the numbered references below the graphic for specific descriptions.



(1) Click to immediately open the last Project File that you worked on.

(2) Click to immediately open the last Project File that you worked on and open the last calculation you worked on.

(3) This list shows the most recent Project Files that you opened on this computer, with the most recent Project File at the top of the list. Double-click on any item in the list to open that Project File.

(4) Click to open the highlighted Project File in the list.

(5) Click to remove the highlighted file from the Recent Projects list.

(6) Click to open a Pick List that displays ALL Project Files in the various folders that you have used in the past. See the section Main Menu > File 70 for a description of the Pick List.

(7) Click to open a Windows File Open dialog that allows you to navigate through disks and folders to locate and open a Project File.

(8) Click to open the Windows File Create dialog that allows you to navigate through disks and folders and create a new ENERCALC Project File.

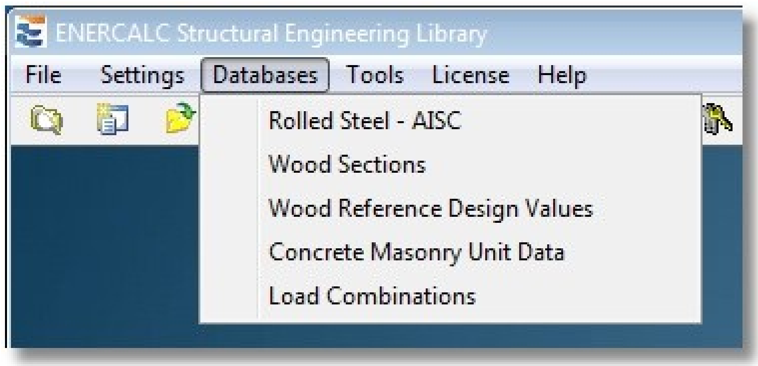
(9) Uncheck this box if you do not wish to automatically display the Project Assistant at startup. To again have the Project Assistant automatically displayed at startup, click **File > Display Project Assistant** and then activate the checkbox to **Show Project Assistant at program startup**.

(10) Link to a help video on the Project Assistant.

6.6 Databases

Structural Engineering Library contains several databases that you can use in the various modules.

To view the databases click **Databases >** (**database**) from the main menu. See below.



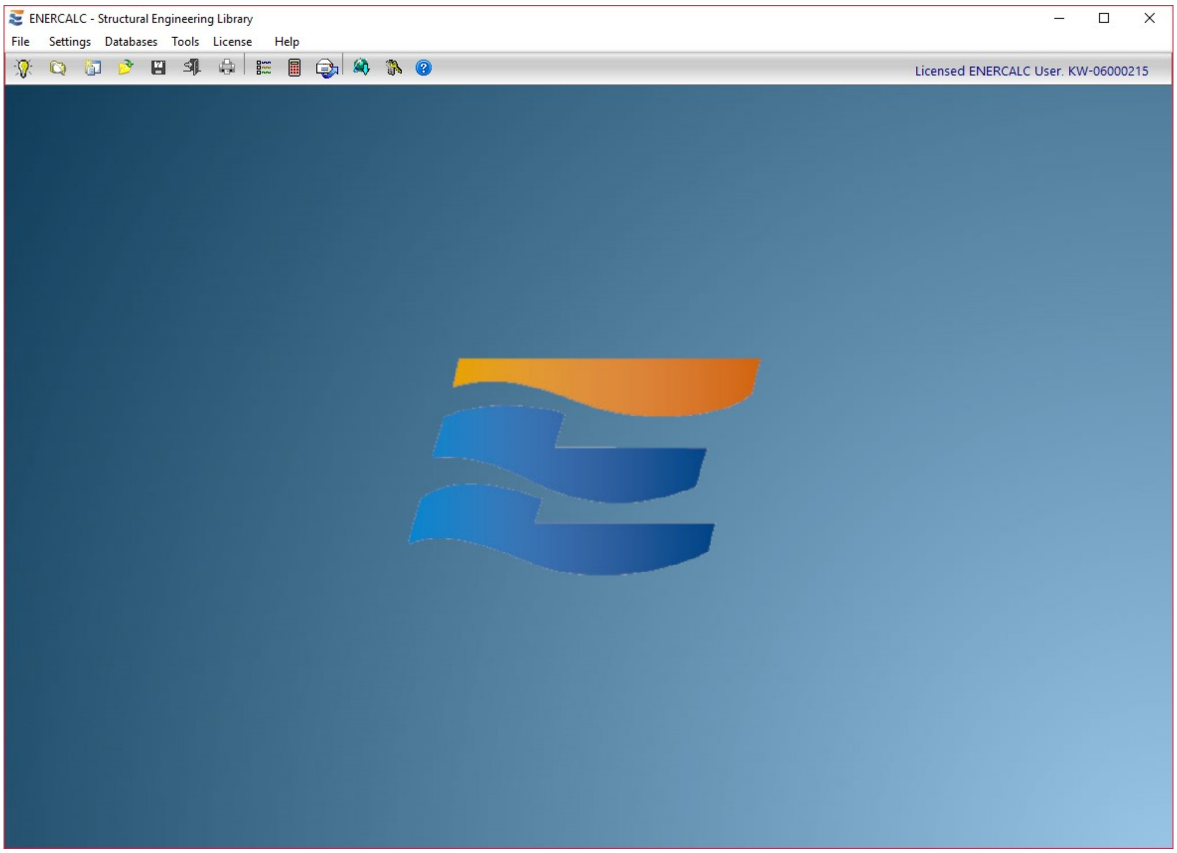
Specific information on each of the databases is provided here: Databases item in Main Menu 79

6.7 Screen Layout

The ENERCALC **Structural Engineering Library** user interface is divided into a Main Menu and a Working Area. Click here for a video: General Layout

The **Working Area** is used to display the calculation module that you are currently editing. When you choose to view the Project Manager, it is displayed on the left side of the Working Area. See more to follow.

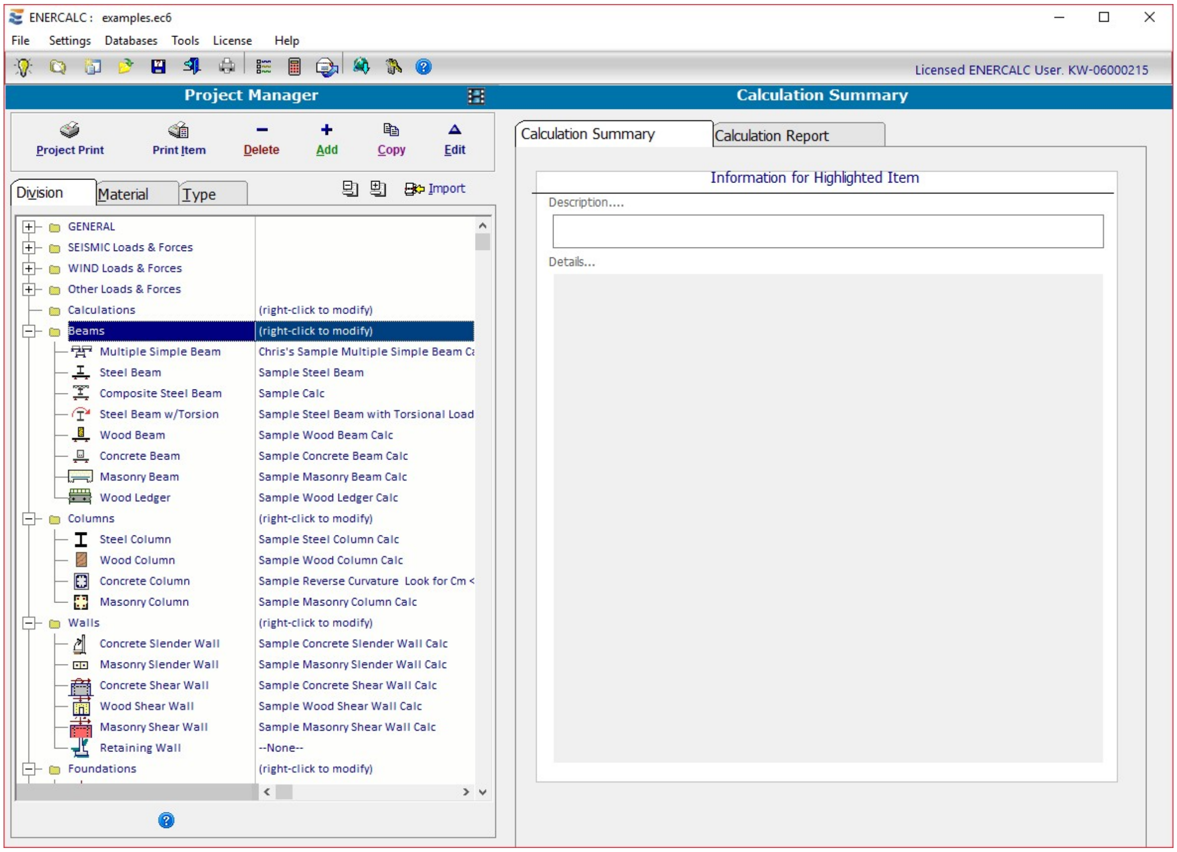
The Main Menu is described in much more detail here: Main Menu description 69



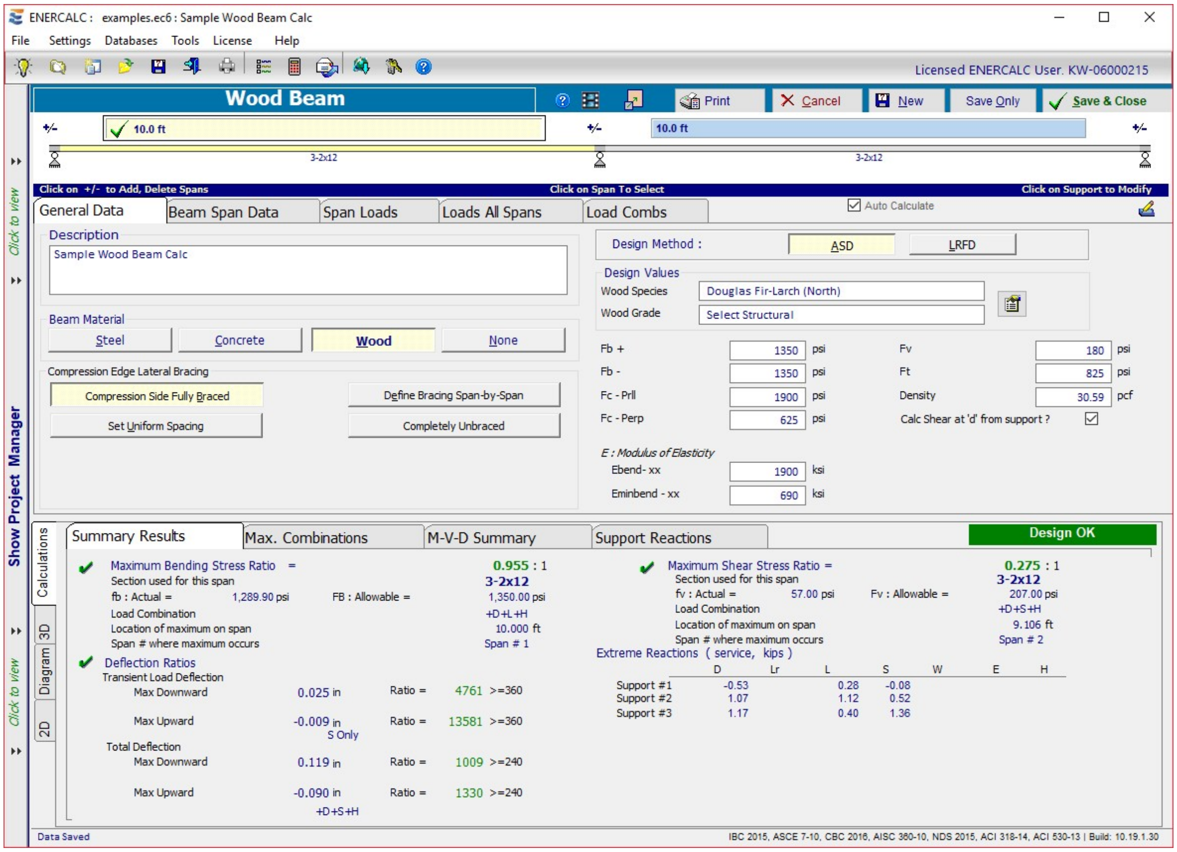
Working Area - Showing Project Manager

Immediately after a Project File is opened the Project Manager is displayed. As you can see in the image below, the Project Manager occupies a portion of the left side of the screen.

The Project Manager provides you with the ability to control the contents of your Project File. Double-click any calculation to open the associated module and edit the parameters of that calculation. Please click here 95 to jump to detailed information for the Project Manager.



After a calculation is selected for editing, the Project Manager automatically collapses, and the window fills with the user interface for that particular calculation module as shown below:



NOTE: It is normal for the Project Manager to hide while a calculation is being edited.

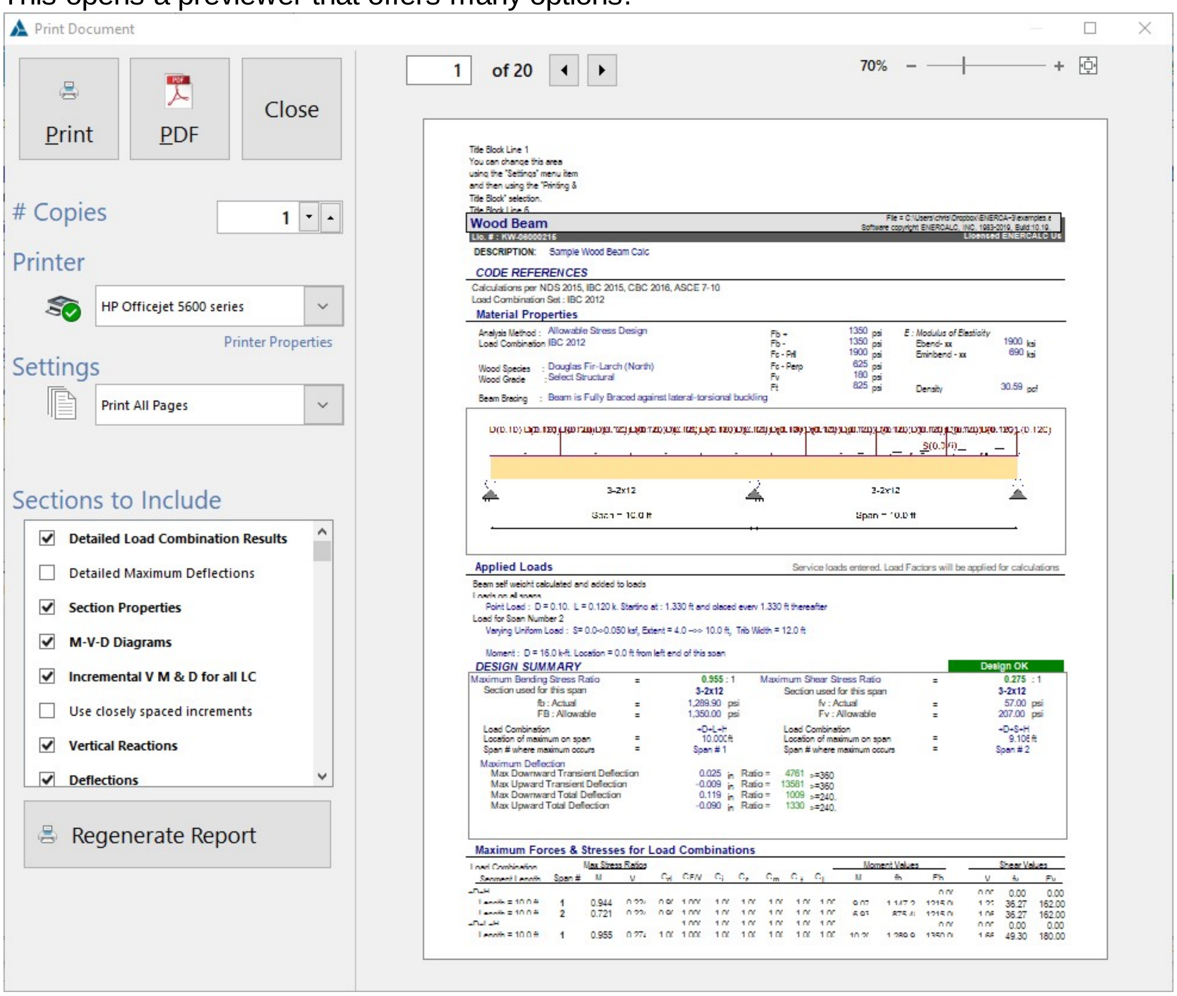
For a review of using a typical calculation screen click here 64.

6.8 Generating Reports

In **Structural Engineering Library**, the term "report" refers to the results documents created by the program in either printed or PDF format.

For each of the ENERCALC modules, the generation of a report starts by clicking the Print button:

This opens a previewer that offers many options:



Print: Create a report and send directly to the currently selected printer.

PDF: Create a report as an Adobe Acrobat PDF file.

Close: Close the previewer without printing anything.

Copies: Set the desired number of copies.

Printer: Select the desired printer.

Settings: Select what is to be printed.

Sections to Include: Select individual report topics to include in the report.

Regenerate Report: Recreate the report after changing the options in the Sections to Include area.

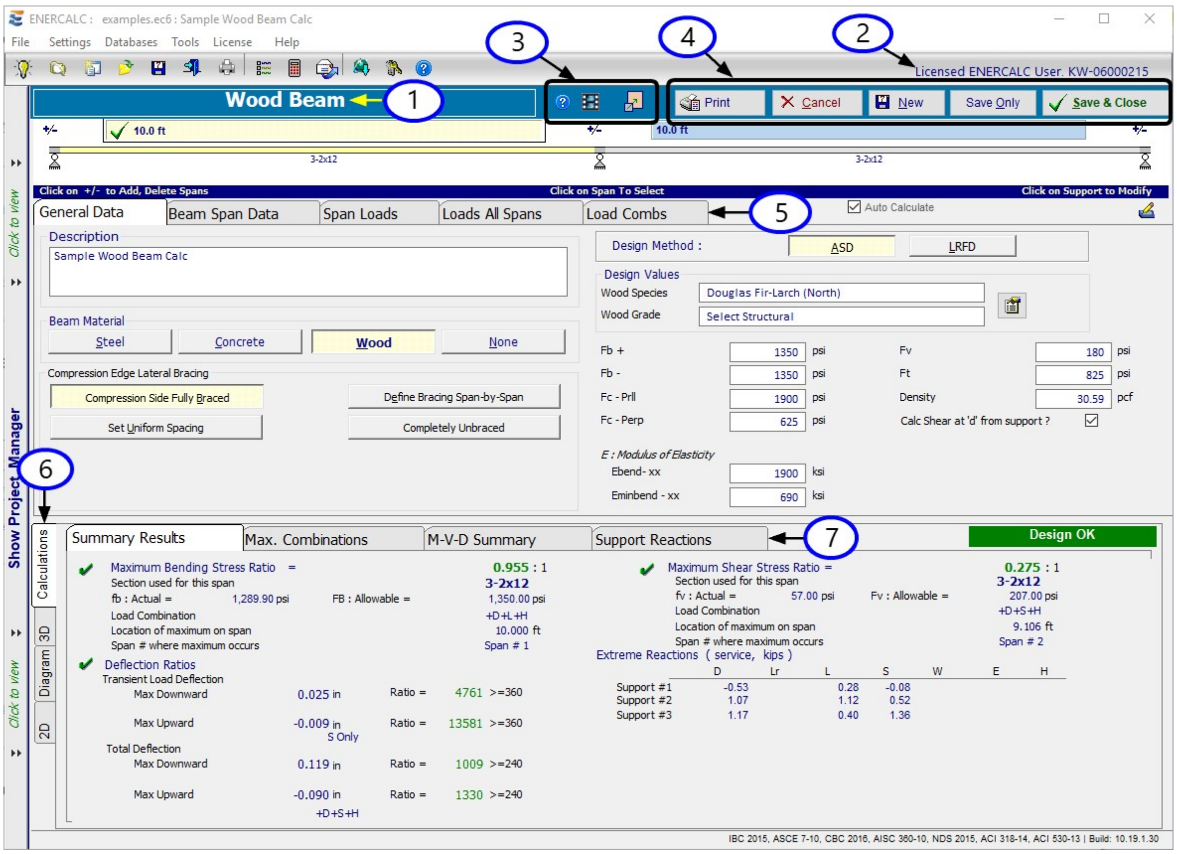
The right side of the screen displays a preview of the current calculation report and offers zooming and page scrolling controls.

In addition to the Regenerate Report button. reports are created and saved when you click the [**Save Only**] or the [**Save & Close**] buttons in the upper-right corner of all modules:

Clicking either of these buttons causes a full report for the current module to be created and saved into the Project File along with all of the input data for that module. **<u>This is how the printouts are prepared for instant viewing and batch printing in the Project Printing system.</u>**

6.9 General Calculation Screen Usage

Please refer to the typical screen below and the numbered items that relate screen areas to their descriptions:



(1) Indicates the module that you are working in.

(2) Indicates the name of the licensed owner of this installation & activation of the software.

(3) From left to right:

Access to help system: Opens the help system and displays the section specific to this module.
Video Tutorial: Opens a video tutorial specific to this module.
Display Enlarged Graphic: Displays an enlarged view of the 2D graphic for the current calculation.

(4) From left to right:

Print: Opens the Print Previewer where the report can be set up, previewed, printed to a printer, or printed to PDF.
Cancel: Cancels all changes made to this module since the last save, closes the module without saving, and returns to the Project Manager. If this calculation module was just Added to the Project File, and if it was never saved, then this option will cause the calculation to be removed without saving.

New: Uses the current calculation data to create a new calculation item in the Project Manager.
Save Only: Saves the current Project File (to capture all the data entered into this module), creates a report (which can be printed at a later time), and keeps the module open for further editing.
Save & Close: Saves the current Project File (to capture all the data entered into this module), creates a report (which can be printed at a later time), closes the module, and returns to the Project Manager.

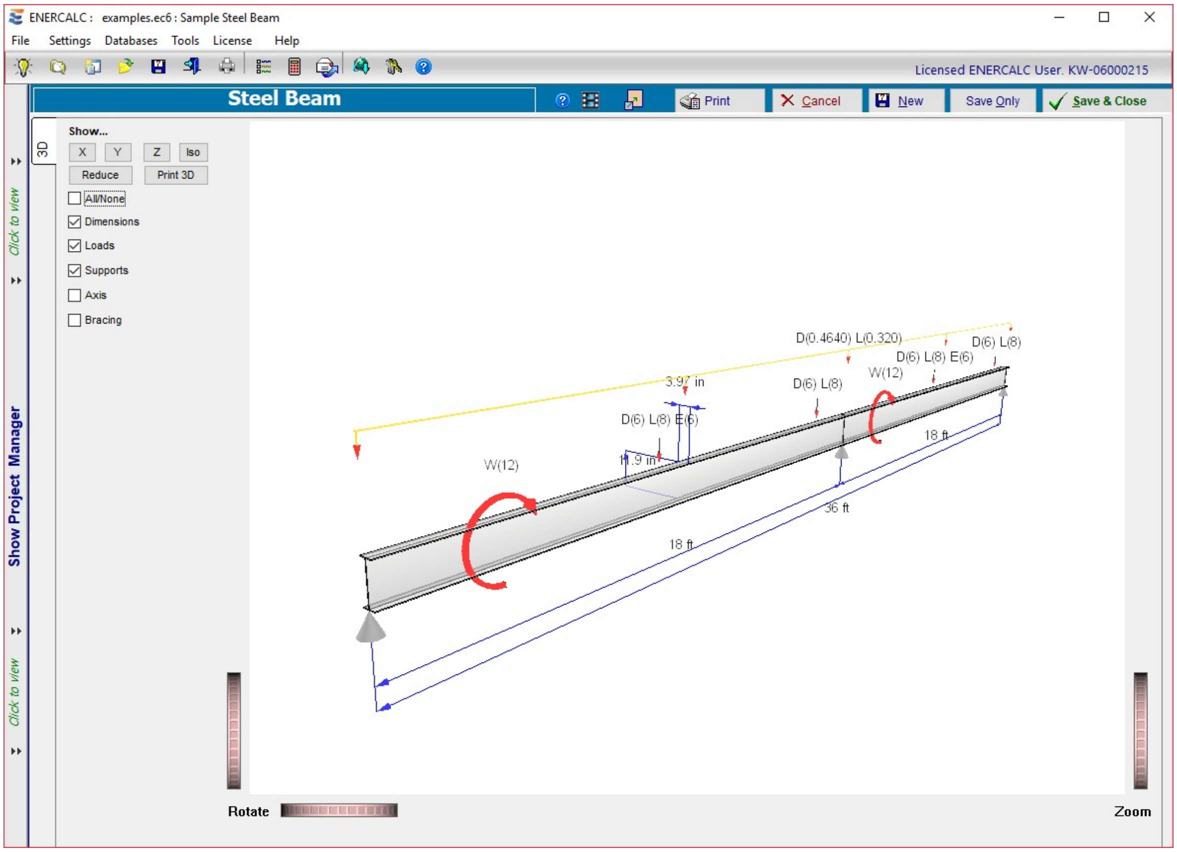
(5) Data input tabs: Click these tabs to move to various categories of data input.

(6) Major Result Category Tabs: Select between major categories of result data to view; numerical values, sketches, diagrams or 3D renderings.

(7) Numeric Result Tabs: Click these tabs to view the various components of calculated results.

6.10 3D Renderings

The 3D tab in most modules offer a 3D rendering of the element in the current calculation.



Each calculation offers "Show" options that are specific to the type of element being displayed. Note that some of the show options will cause some renderings to switch into a semi-transparent mode for clarity. For example, when viewing the rendering for a General Footing, the rendering will automatically switch to semi-transparent mode when the display of rebar is turned on.

The X/Y/Z/Iso buttons allow the rendering to quickly be snapped to specific view angles. Rotate and Zoom controls make it easy to adjust the view, but note that the rendering can also be manipulated by clicking and dragging with the left mouse button to rotate or with the middle mouse button to pan.

The Enlarge/Reduce button toggles between a large and small display window for the rendering.

The Print button allows the 3D rendering to be printed to PDF or paper.

Run menu > sysdm.cpl > Advanced Tab > performance > select adjust for best performance > make sure you do not allow Windows to choose for you because your computer does not know whats best - Make sure Enable Aero Peek is checked.

Part VII

7 Main Menu

The main menu of **Structural Engineering Library** is always displayed.

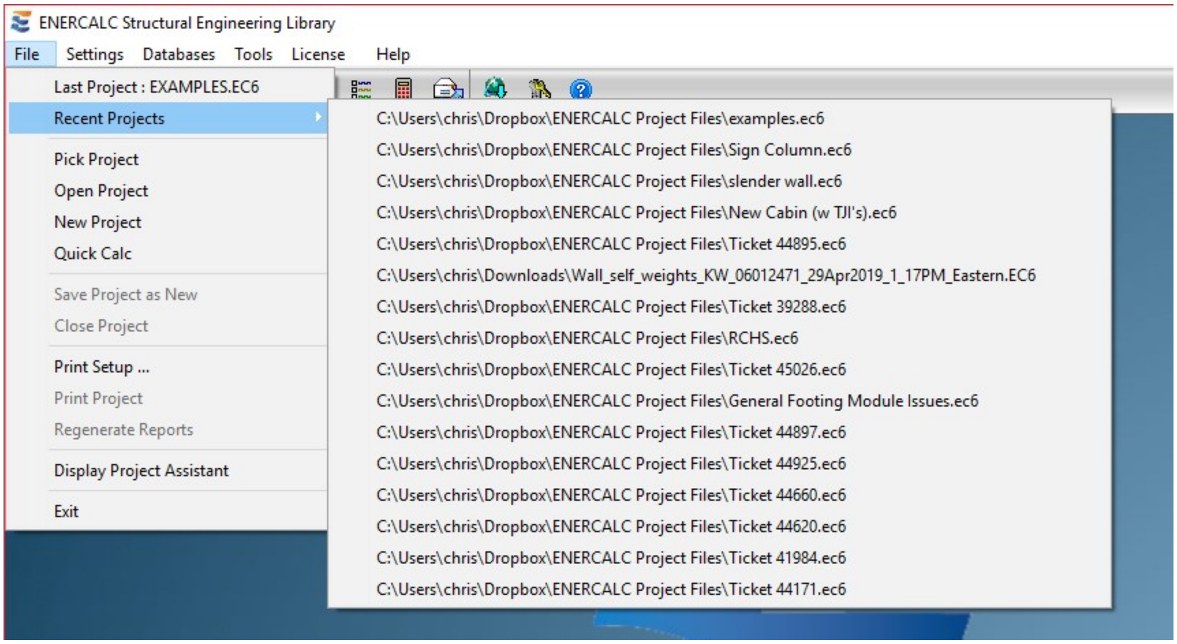
The menu offers the following selections. Click an item below to jump to that help section page, or click here for a video: Main Menu and Toolbar

File 70 : To create, load, close and print Project Files.
Settings 76 : To establish settings, user information and default values
Databases 79 : To view the various databases supplied with the software
Tools 87 : To access utilities provided with the software
License 89 : To directly access the licensing control window
Help 90 : To access the help system, technical support options, Knowledge Base with FAQs, Enhancement/Change History and the ENERCALC website.

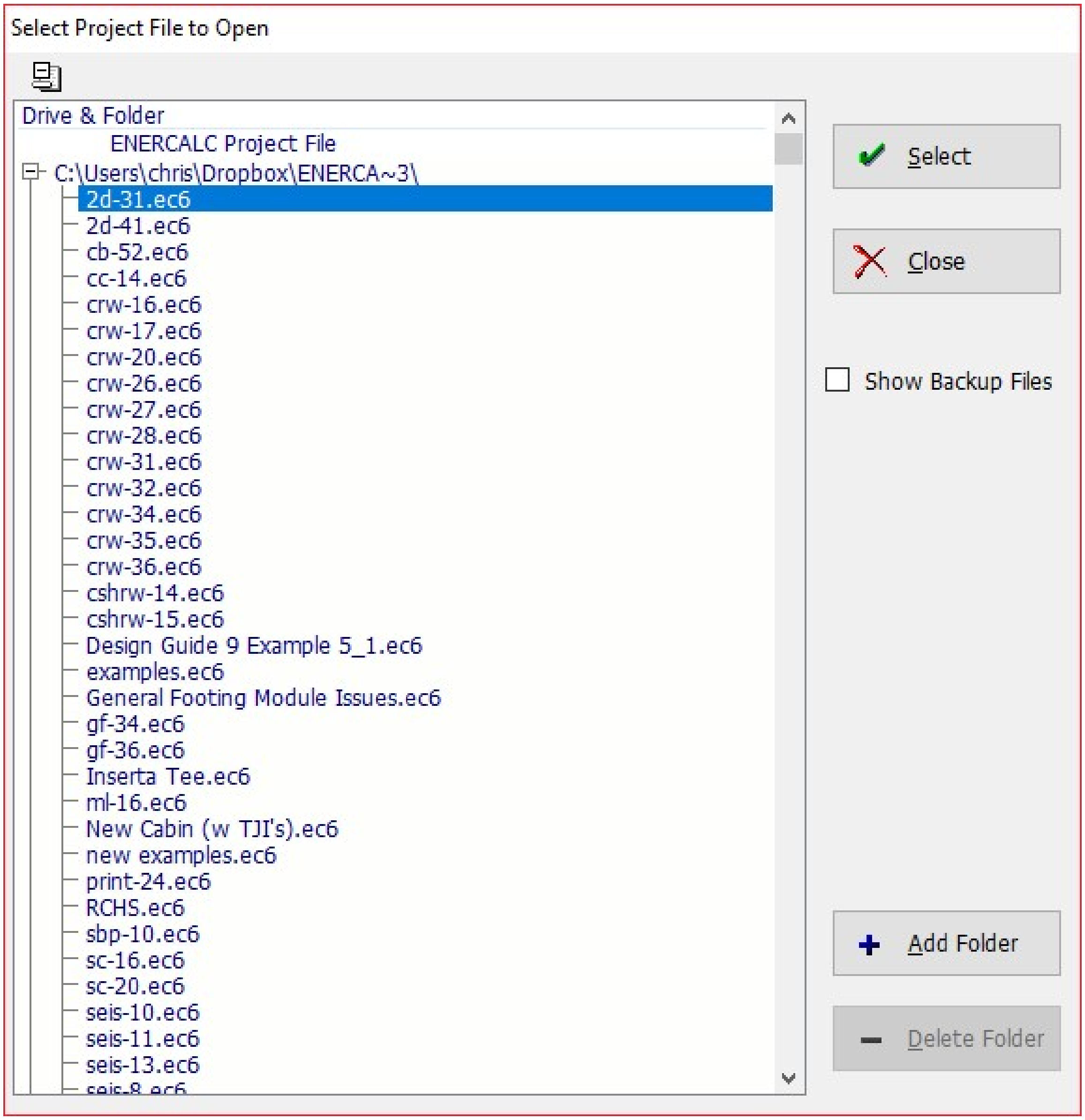
7.1 File

Last Project: Opens the last used Project File.

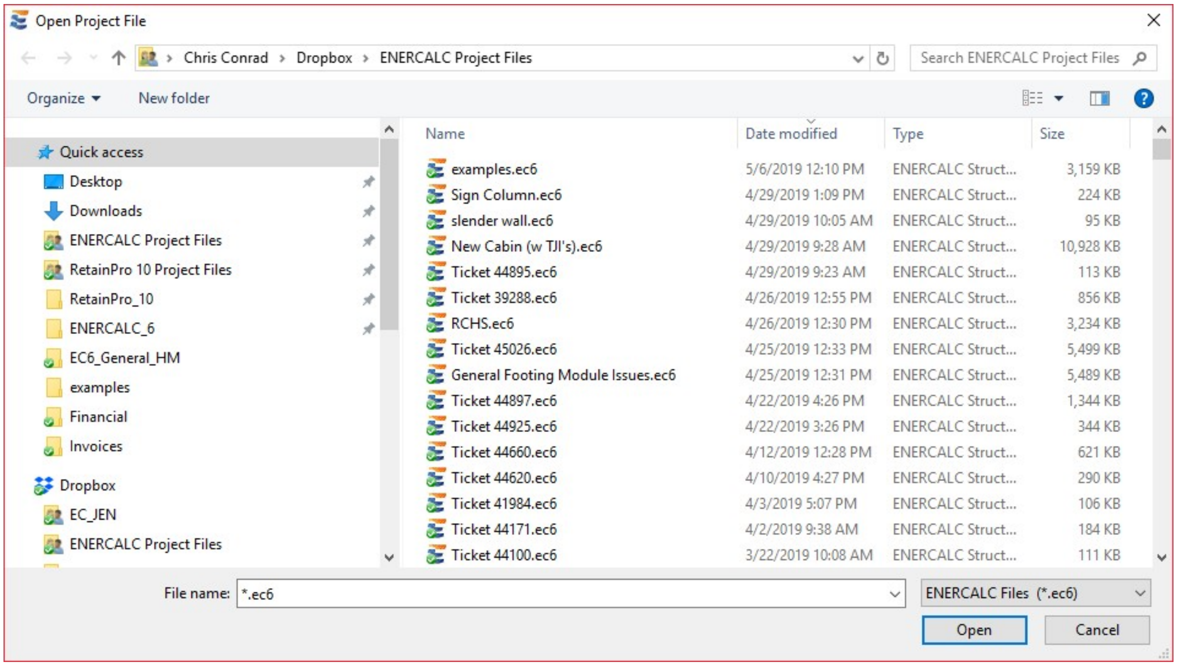
Recent Projects: Provides a list of Project Files that were recently opened.



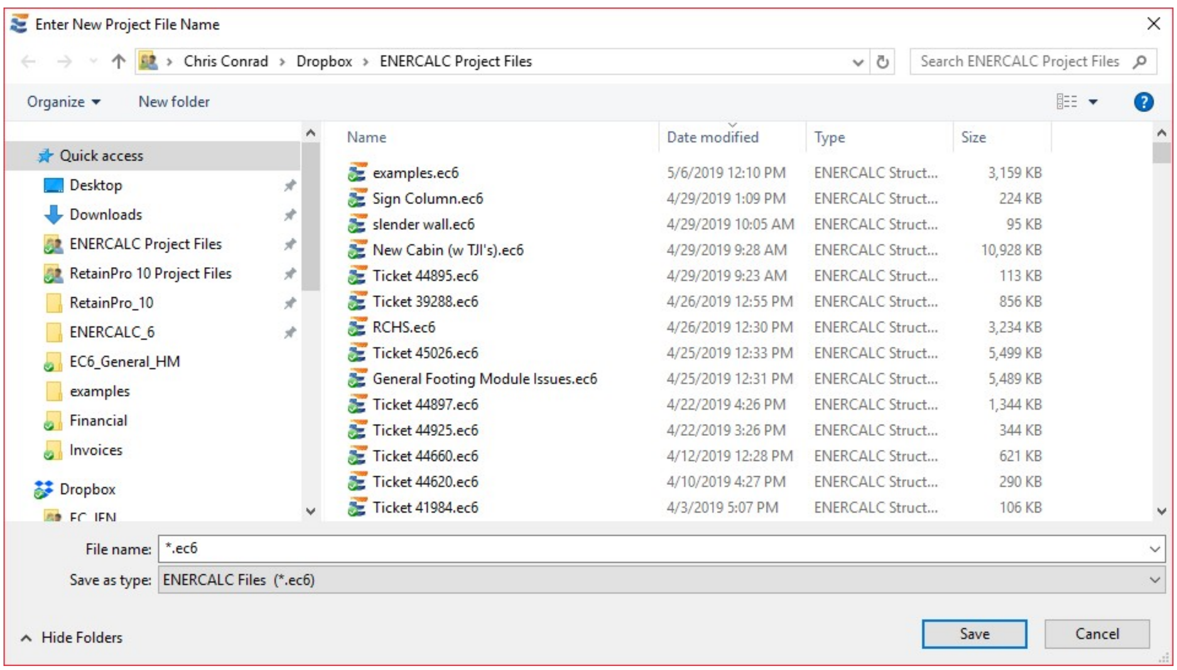
Pick Project: The software remembers all prior folders on your computer (and other computers) that were accessed to load Project Files. This item opens a window that shows those folders and the ENERCALC Project Files contained within them to allow you one-stop selection of a Project File from your commonly used locations.



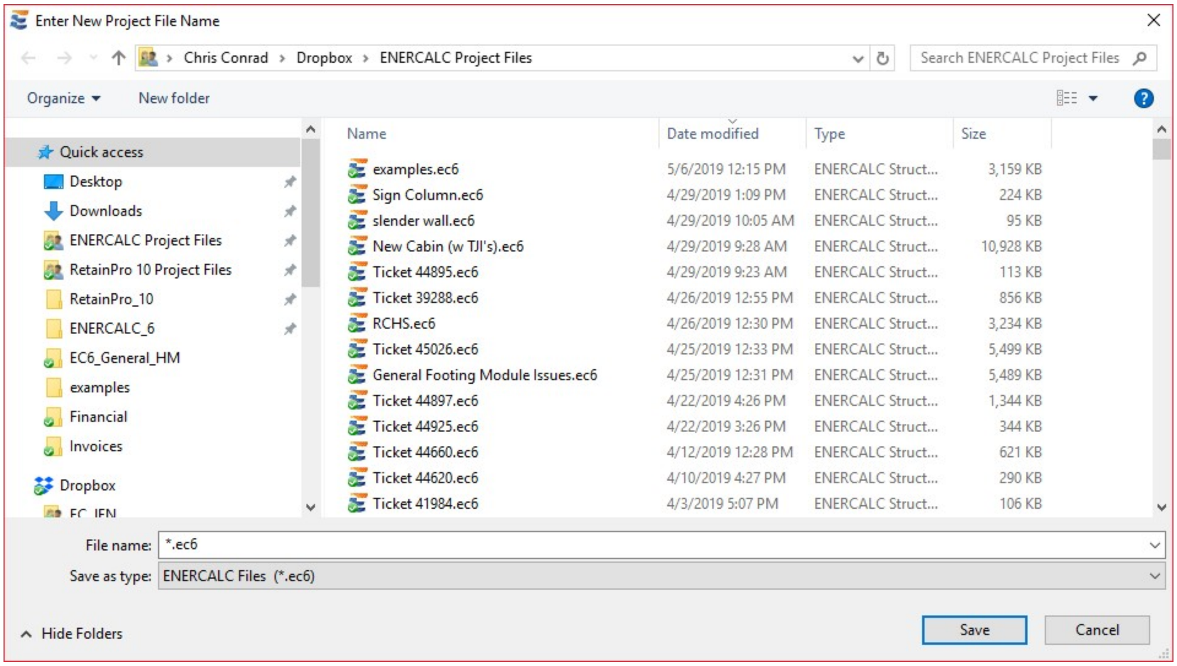
Open Project: Opens a standard Windows File Open dialog allowing you to navigate your disk drives and open a Project File.



New Project: Opens a standard Windows File Create dialog allowing you to navigate your disk drives and specify the name of a Project File to create.

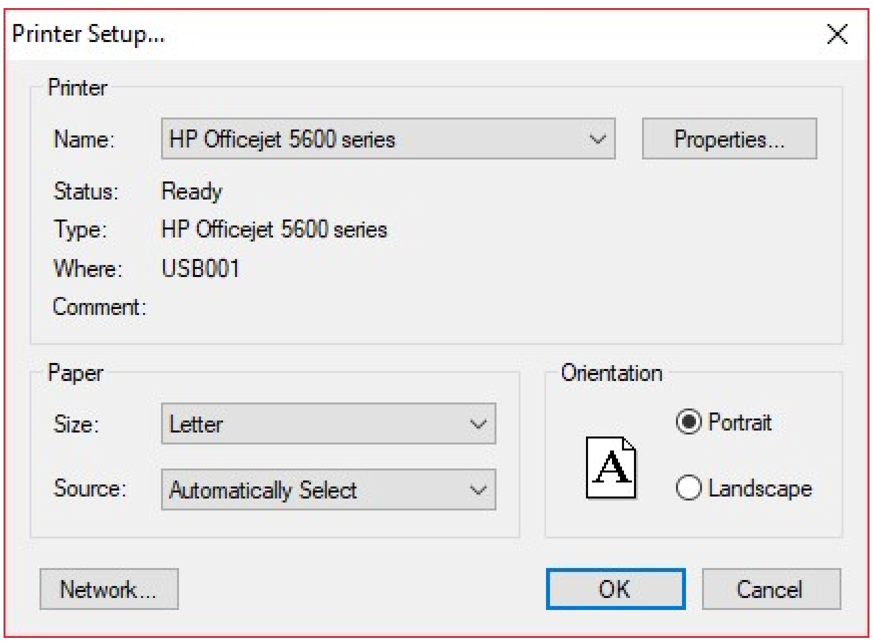


Save Project as New: Allows you to save the currently open Project File as a new file using a different name. Opens a standard Windows File Create dialog allowing you to navigate your disk drives and specify the name of the Project File to create.



Close Project: Closes the currently open Project File (making sure any unsaved data is purged).

Print Setup: Displays a typical Windows Printer Selection dialog to specify the printer to be used for this work session.



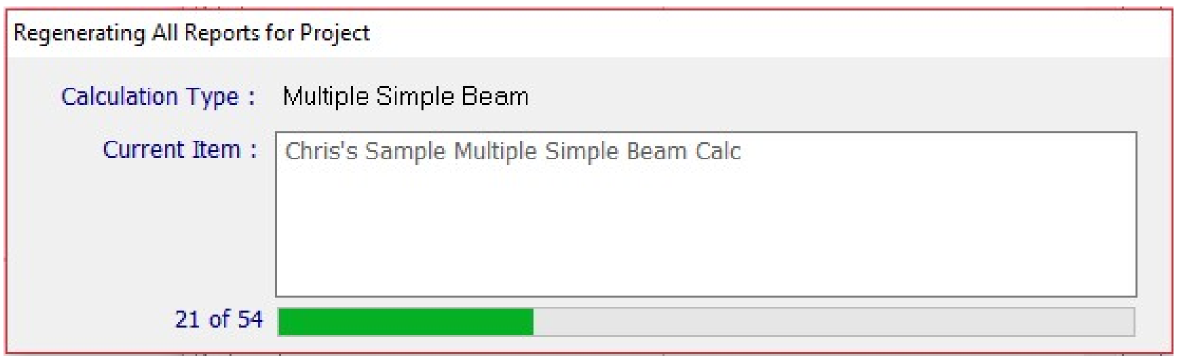
Print Project: Opens the Project Printing 131 Manager.

Regenerate Reports: This selection is used to regenerate printouts so they will be updated and displayed correctly in the Project Printing Manager. It can be used any time the following information has changed:

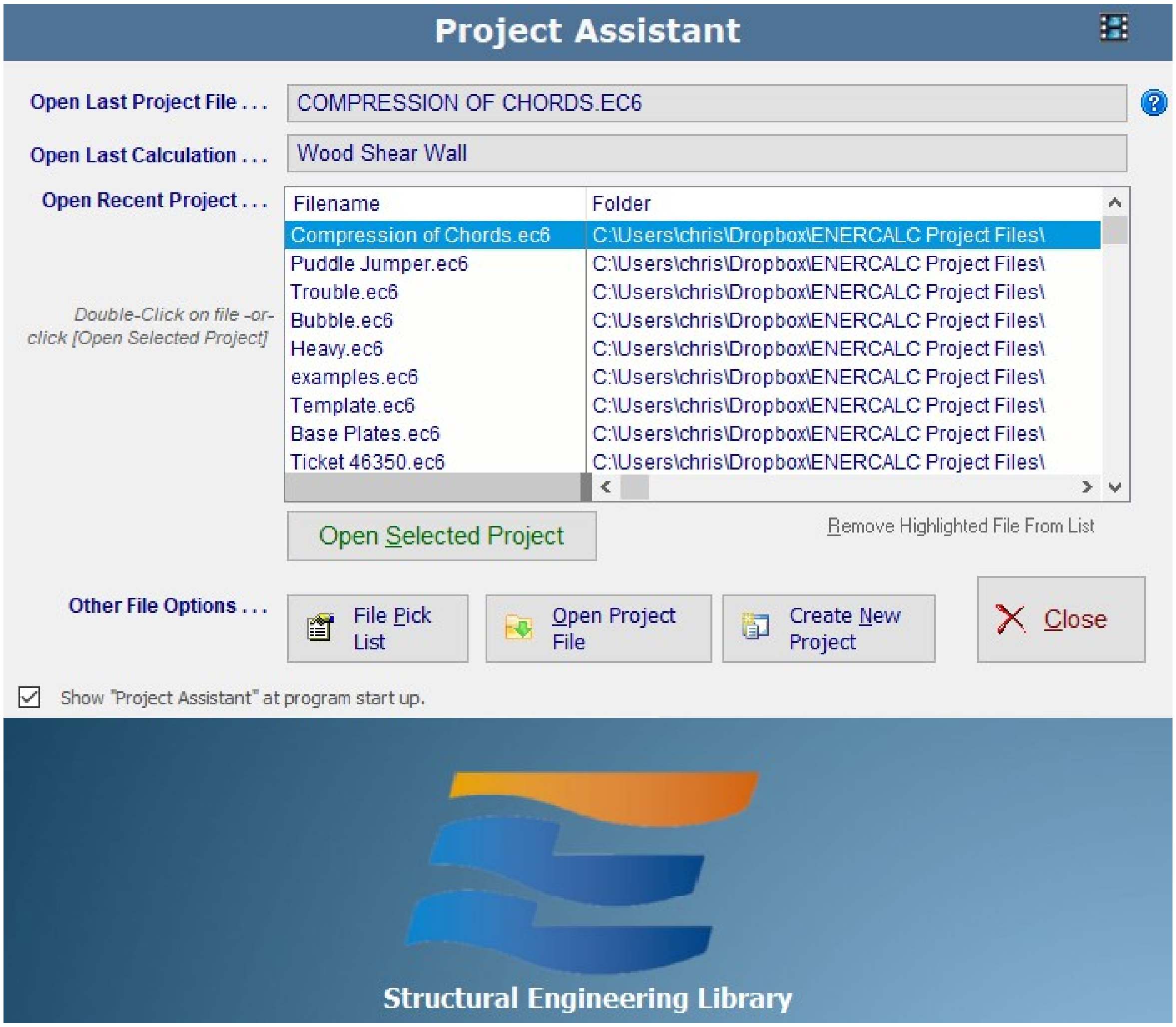
- Project Information in the GENERAL Division of the Project Manager for the currently open Project File
- Title block information filled in using **Settings > Printing & Title Block** from the main menu

There is also a third situation where this command can come in handy. When running in Evaluation mode, the software adds a watermark on the background of all printouts. This watermark says "Evaluation Version" and "Unlicensed Usage", so the report is inconvenient to use for submittal purposes. When the user later runs the software in Licensed mode, those reports with the "Evaluation Version" watermark remain in the Project File until they are reopened and saved. This can be a laborious process if the Project File contains many calculations. So the easier way to remove the watermarks in one pass is to use the Regenerate Reports command.

When you click **File > Regenerate Reports > All**, the program will automatically save and close any open calculations. Then the program will show a progress window as it regenerates all the reports in your Project File:

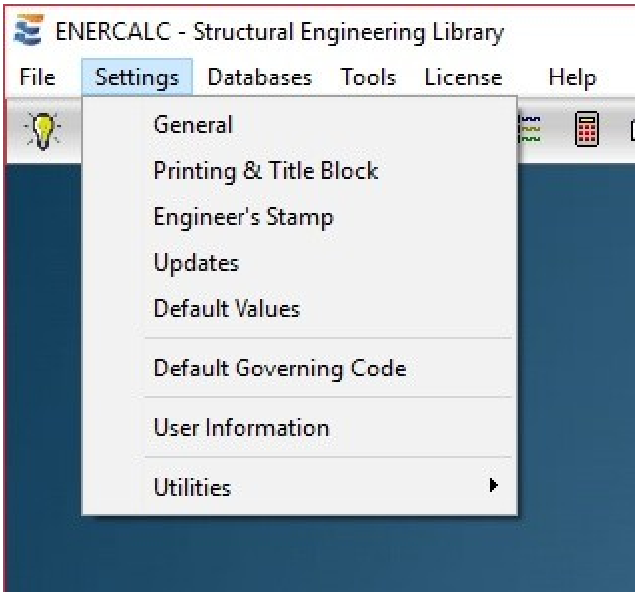


Display Project Assistant: This option displays the Project Assistant which offers a convenient way of beginning work with a Project File.



Exit: Offers the option to save any pending work, then closes the currently open Project File, and exits the software.

7.2 Settings



Most of these items are covered in another section of this help document. Please click on an item in the list below to jump to that section.

General Settings 48: Defines file locations, backup file creation and behavior items in the Project Manager.

Printing & Title Block 50: Provides the ability for the user to define the look of their title block.

Engineer's Stamp: Allows the user to upload a graphic image of their Professional Engineer's stamp for inclusion on printed reports.

Display Settings: Provides options to configure the software for the resolution of the monitor being used.

Web Update Settings 25: Allows control over behavior of web update system and direct access to the update installation program on the web.

Default Values 78: Provides control over the values that serve as the defaults for each individual calculation module. Note that a particular calculation module must be open in order to use the functions provided by this item.

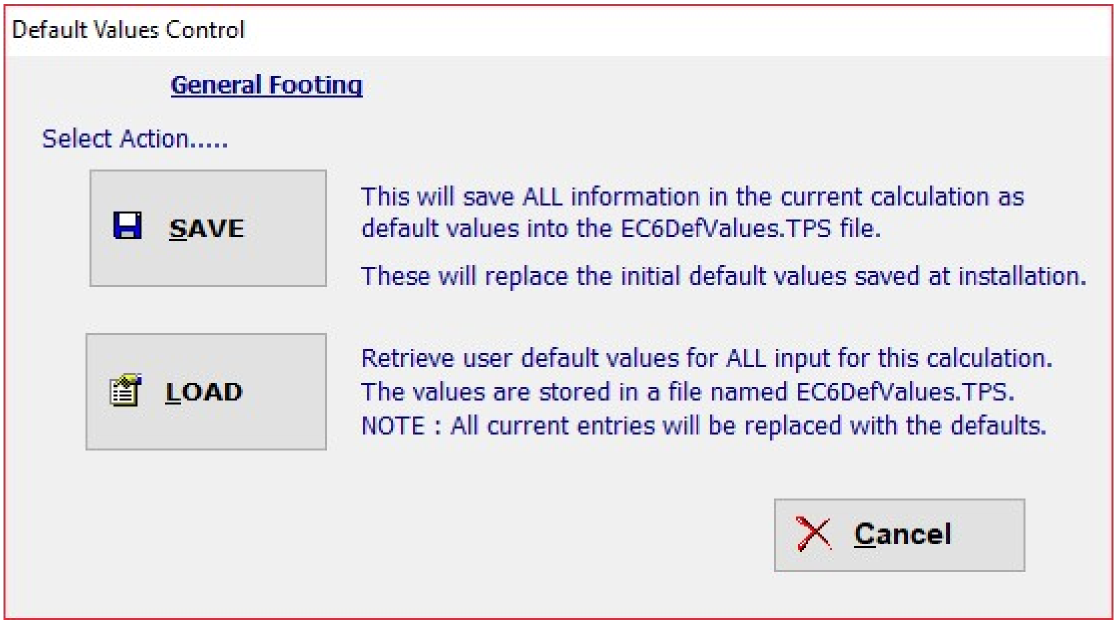
User Information: Provides access to user information.

Utilities: This selection provides items that are mostly used for support and maintenance of the software. At the time of this documentation writing, the only Utility item available is to start a process of registering components used by SEL with the operating system.

7.2.1 Default Values

Most of the user input values in the individual calculation modules have a default value or setting that is stored within the program. Many of these default values can easily be changed by the user. The procedure is as follows:

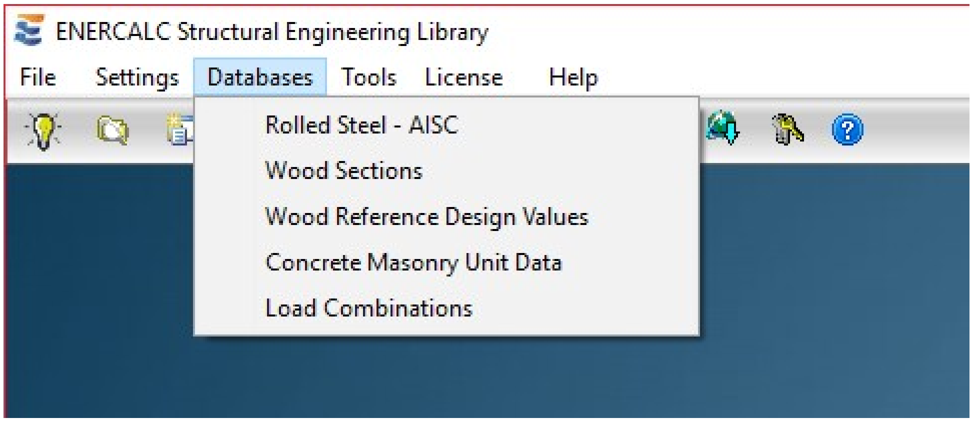
1. Open a new calculation of the particular module of interest.
2. Revise all of the initial values and settings as desired so that they represent the desired default values or settings.
3. Click Settings > Default Values > Save.



The current values will be saved as the new default values for that module, and these initial values and settings will be used when a new instance of this calculation module is added to any Project File.

7.3 Databases

This menu allows viewing of various databases supplied with the software.



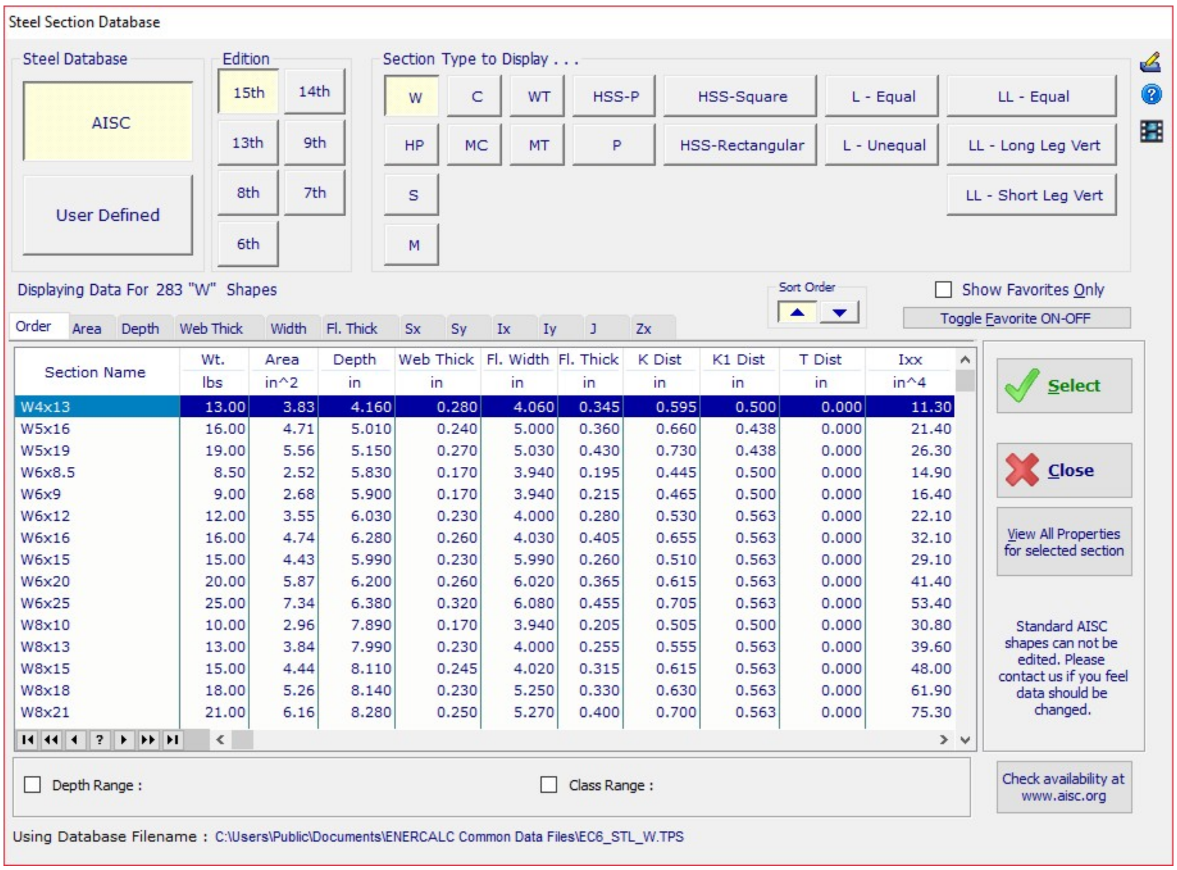
Please see the specific subtopics in this section for more detailed information on the databases.

7.3.1 Steel Section Database

Steel Section Database

The steel section database is available in all applicable steel modules. It contains typical AISC rolled sections from many Editions of the AISC Steel Construction Manual. Click here for a video: Steel Section Database

Note that it is also possible to create User Defined entries that can be referenced in calculations.

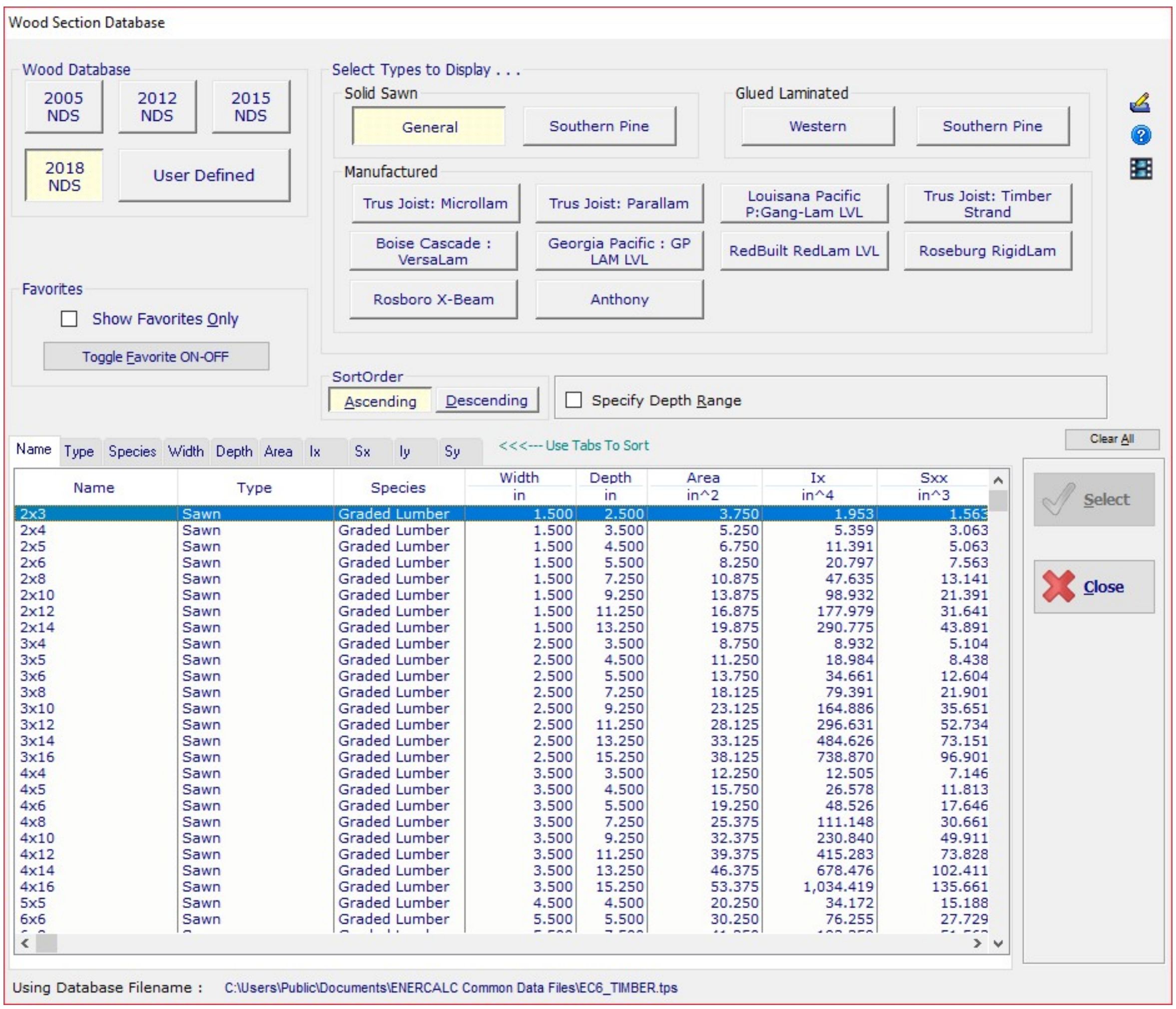


7.3.2 Wood Section Database

Wood Section Database

The wood section database contains the section properties for most of the sections listed in the NDS and for many Engineered Wood Products. Click here for a video: Wood Section Database

The wood section database is available in all applicable wood modules. It contains typical wood sections available in the United States. These sections are also shown in the 2005 NDS. Also included are manufactured sections, however the list of those sections is only updated occasionally. Note that it is also possible to create User Defined entries that can be referenced in calculations.



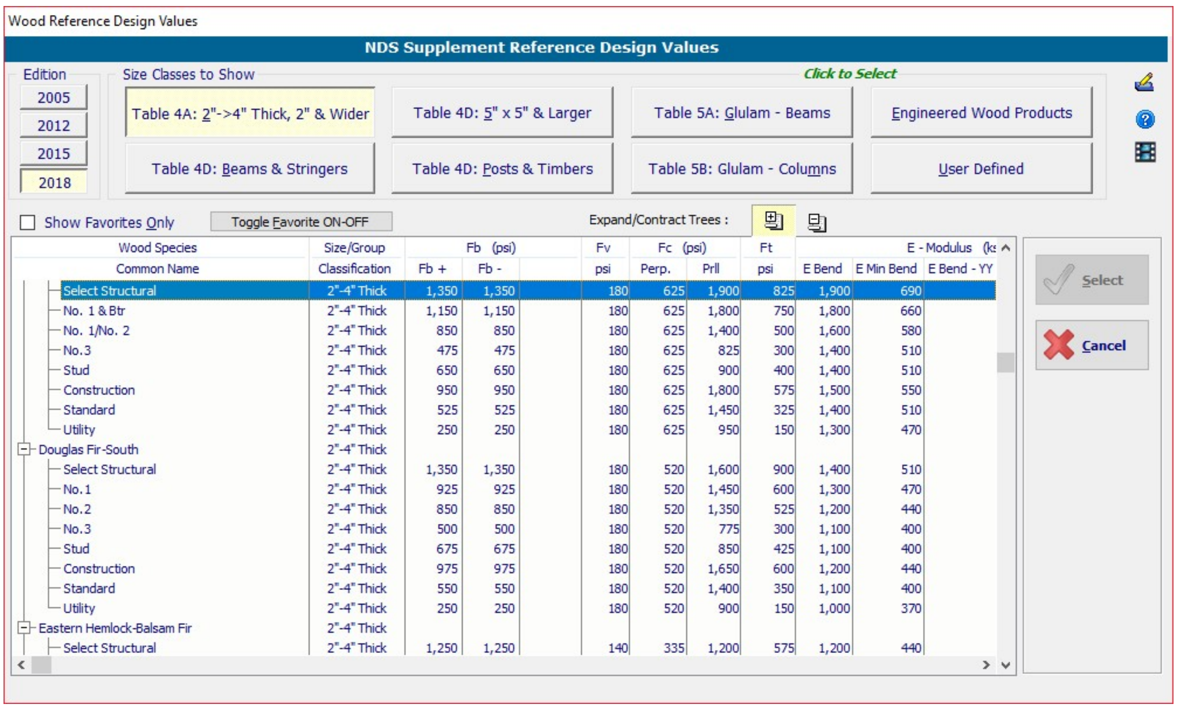
7.3.3 Wood Reference Design Values Database

Wood Reference Design Values Database

The Wood Reference Design Values database is available in all applicable wood modules.

Click here for a video: Wood Reference Design Values Database

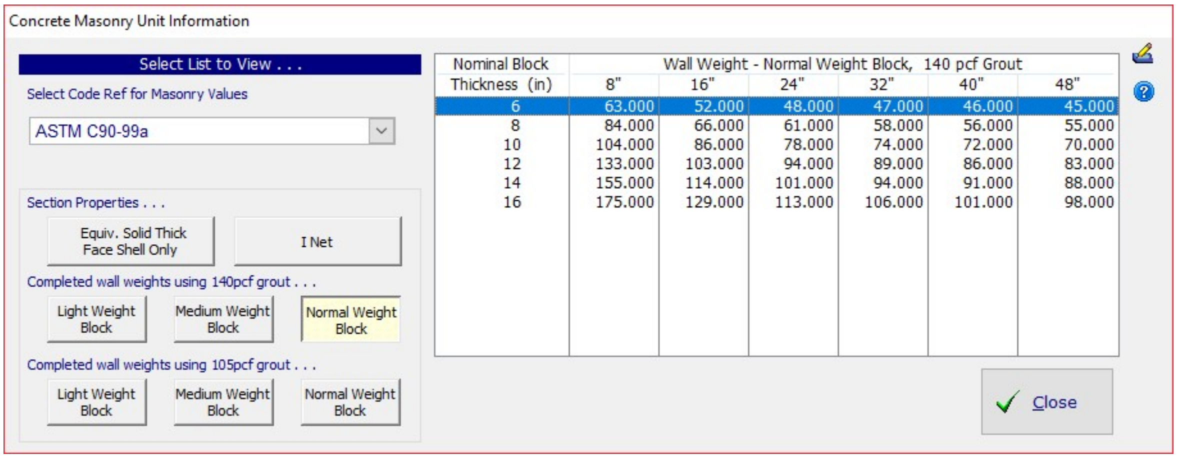
The Wood Reference Design Values database contains typical wood stress grades as defined in the NDS. Also included are manufactured sections, however the list of those sections is only updated occasionally. Note that it is also possible to create User Defined entries that can be referenced in calculations.



7.3.4 Masonry Database

Masonry Database

This is a reference table only. It consists of data for common hollow concrete masonry units:



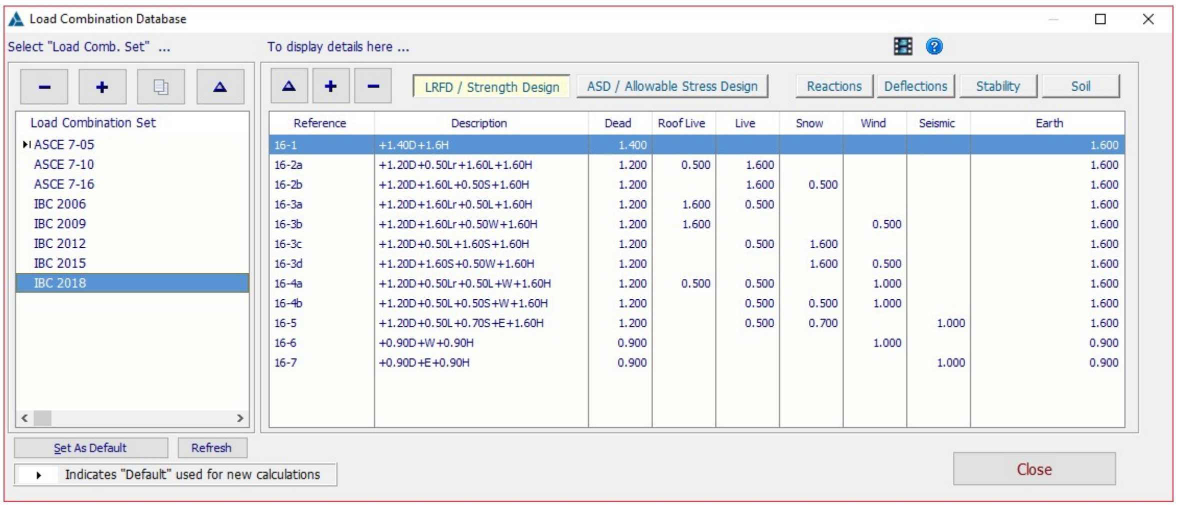
Nominal Block Thickness (in)	Wall Weight - Normal Weight Block, 140 pcf Grout 8"	16"	24"	32"	40"	48"
6	63.000	52.000	48.000	47.000	46.000	45.000
8	84.000	66.000	61.000	58.000	56.000	55.000
10	104.000	86.000	78.000	74.000	72.000	70.000
12	133.000	103.000	94.000	89.000	86.000	83.000
14	155.000	114.000	101.000	94.000	91.000	88.000
16	175.000	129.000	113.000	106.000	101.000	98.000

7.3.5 Load Combination Database

Load Combination Database

This is the database of named load combination sets that can be automatically retrieved into a calculation. Click here for a video: Load Combination Database

A named load combination set can be established as a default and will be used whenever a new calculation is created in the Project File. Note that it is also possible to create new load combination sets and/or revise the load combinations that are in the existing sets.



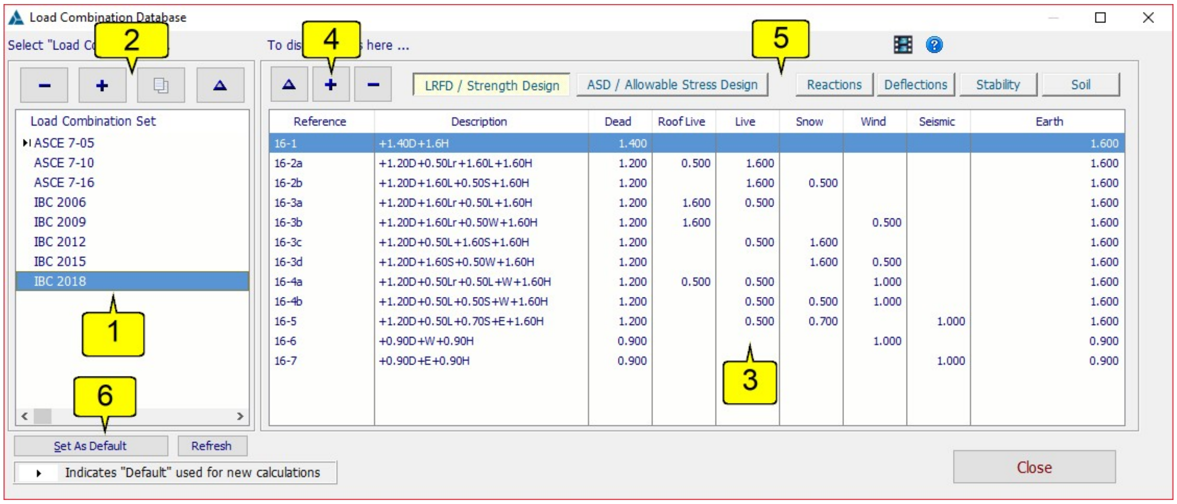
The Load Combination Database is used to manage the load combinations that you can use across all of your Project Files.

The load combinations specified in this section are stored in a separate file and ARE NOT specific to a particular Project File.

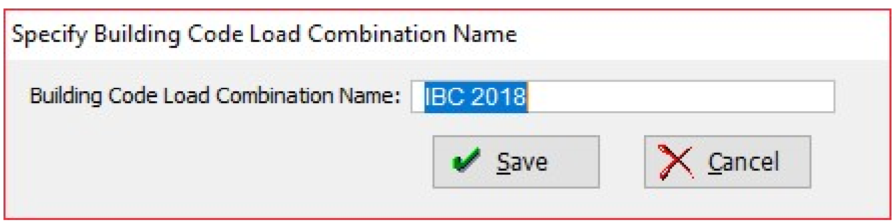
With the Load Combination Database you can create many load combination sets. These are listed in the Code Reference column.

For each set, you can then specify the individual load combinations for both factored load and service load cases, which are used for LRFD and ASD respectively (also referred to as strength design and allowable stress design).

Please see the descriptions for the numbered keynotes on the screen capture below.



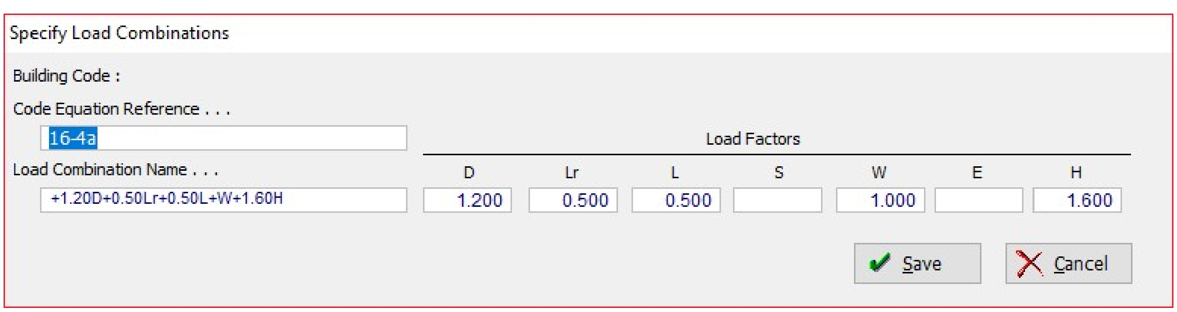
(1) This area lists the load combination sets available for use.
(2) Use these buttons to add, delete, copy or edit the name of load combination sets. The button with the triangle symbol means "Change" or "Edit" and is used to edit the displayed reference name for the set.



(3) This is the area that displays all the load factors for the various load combination types.

(4) Use these buttons to add or delete individual load combinations. The button with the triangle symbol means "Change" or "Edit" and is used to edit the numeric factors applied to each type of load for the load combination that is highlighted.

When clicking the [**Add**] or [**Edit**] button, the following dialog is displayed:



Use this dialog to specify the reference name and the values for each load factor.

(5) Click the various buttons to select the category of load combinations to display.

(6) Use the 

button to make the highlighted load combination set the default that will be used in all new ENERCALC calculations.

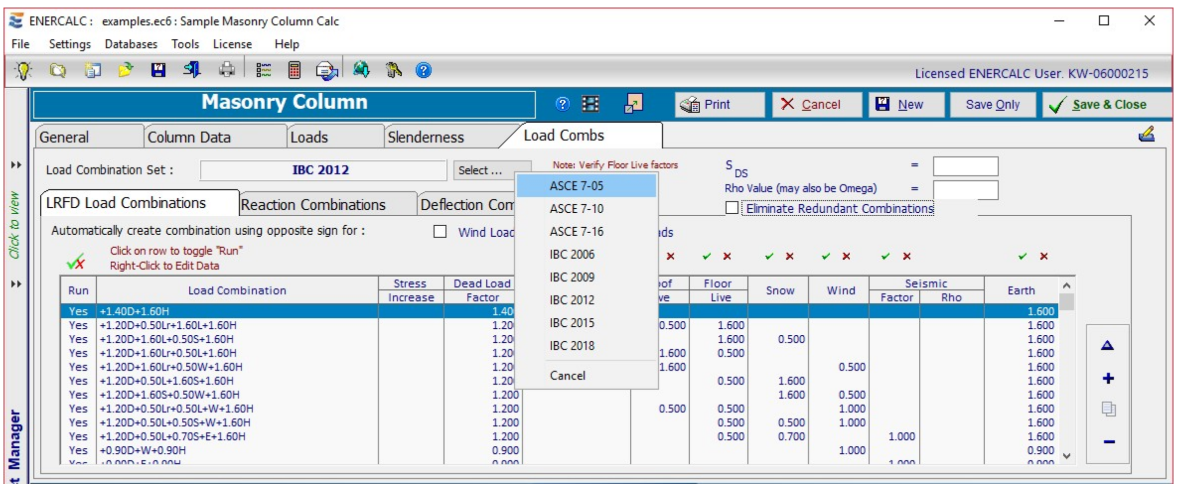
Using Load Combinations

When creating a NEW calculation in the Project Manager, the default load combination set is automatically loaded into the Load Combination tab in the module.

In the image below you can see that the load combinations have been loaded and are displayed on Load Combinations tab in the Masonry Column module.

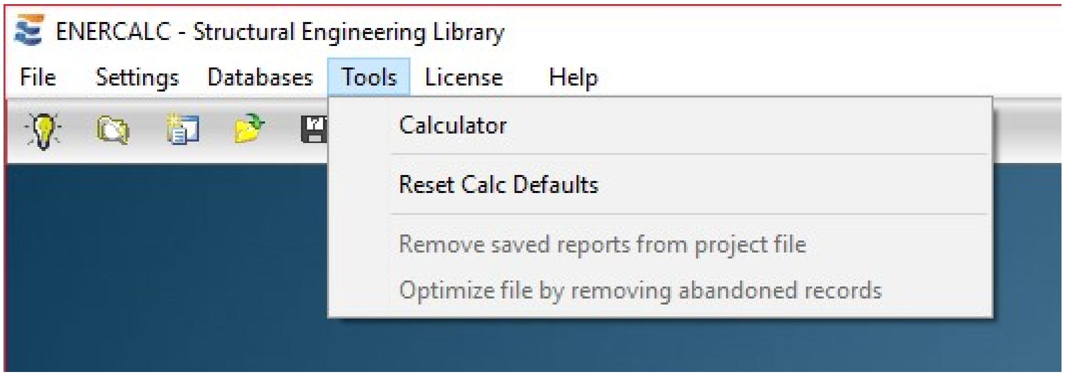
The design method has been set to LRFD, so the software is displaying Factored Combinations.

We have clicked the [**Select**] button to open a list of the available load combination sets from the Load Combination database. Selecting one of them from this dropdown list will load those new load combinations into this particular calculation only.

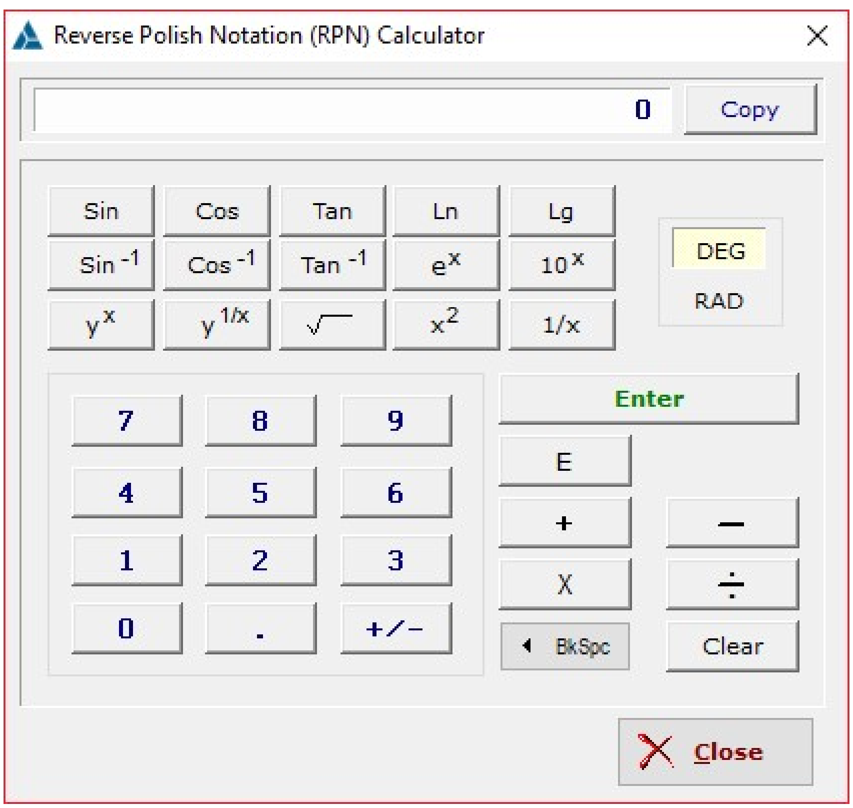


7.4 Tools

This menu item provides access to useful tools that are supplied with **Structural Engineering Library**.



Calculator: We supply a **Reverse Notation Calculator** (RPN) for engineers to use for intermediate calculations when entering data. RPN calculators are the most commonly used among engineers because they allow faster mathematics solutions when nested (parenthetical) calculations are used. It also includes a Copy button, which allows a result value to be copied to the clipboard and then pasted into an input field in Structural Engineering Library.



For more information on Reverse Polish Notation try these links:

http://en.wikipedia.org/wiki/Reverse_Polish_notation

http://www.hpmuseum.org/rpn.htm

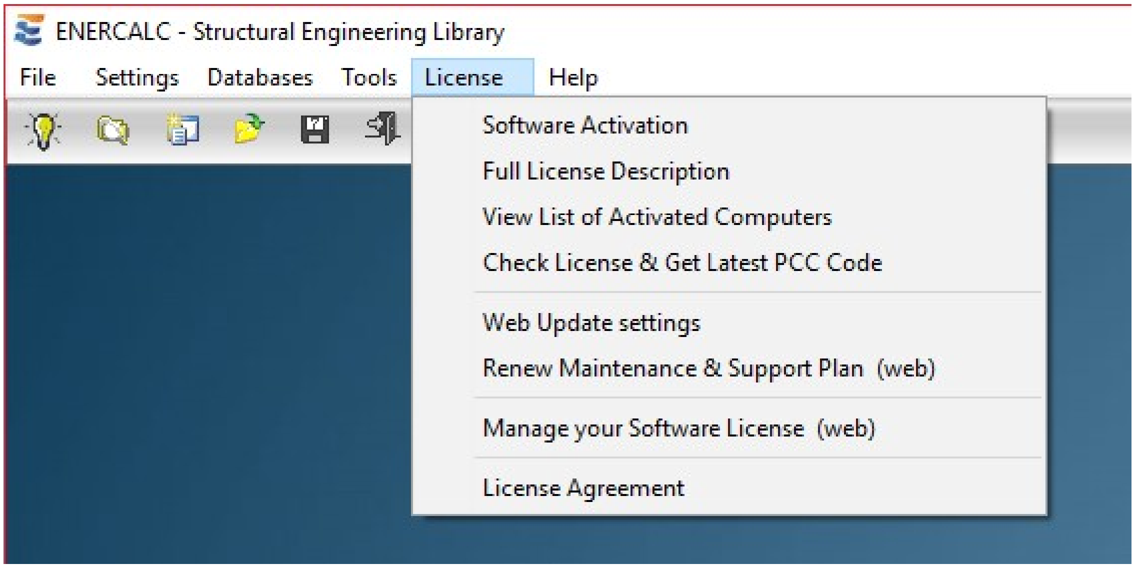
Reset Calc Defaults: This command resets the default values for the current calculation to the hard-coded internal default values. It may be a useful tool in some troubleshooting situations.

Remove Saved Reports from Project File: This command removes any reports that have been saved in the current Project File. It can be helpful in terms of reducing file size if a Project File is to be emailed, but it will require that the reports be regenerated before they can be printed.

Optimize file by removing abandoned records: This command removes lost records in a Project File which could occur if the program is terminated abnormally (perhaps from a power outage, lockup, network failure, lockup of another program, etc).

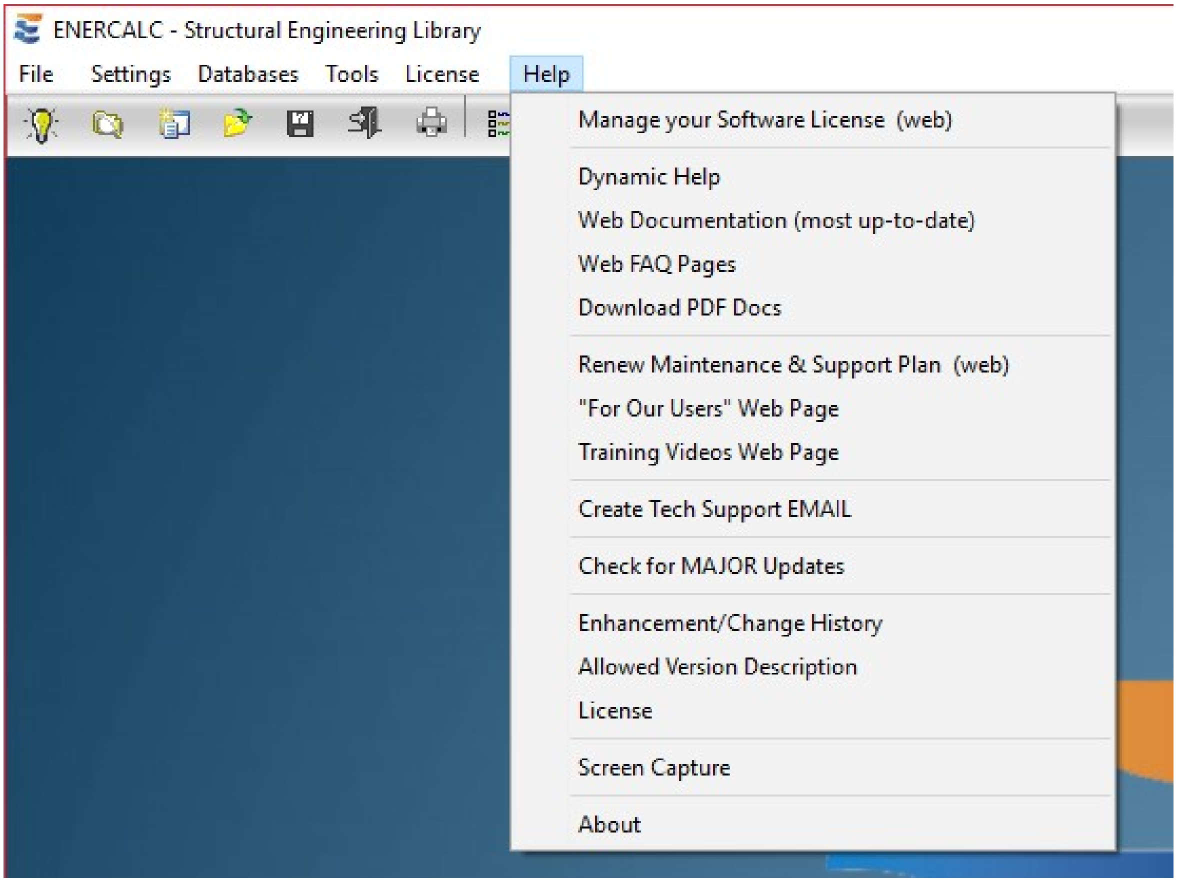
7.5 License

The License item displays the licensing and activation menu.



7.6 Help

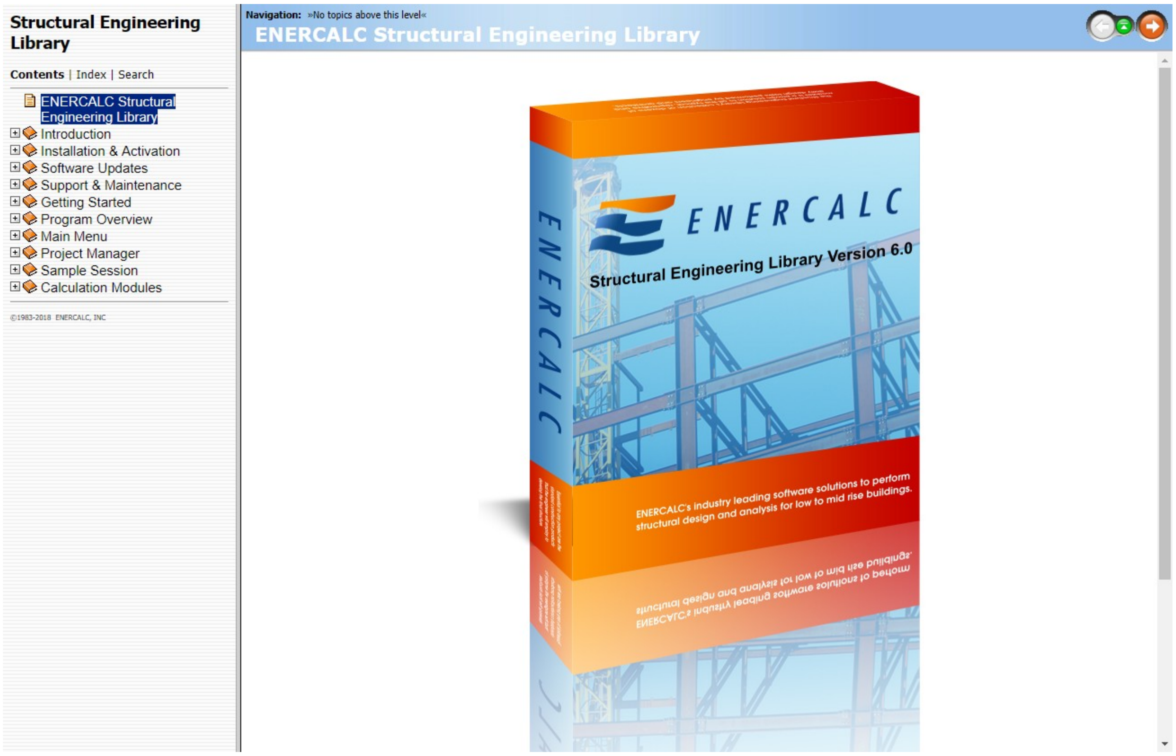
The Help item displays the Help menu.



Manage your Software License: Automatically opens the Manage License page of our website.

Dynamic Help: Displays the help system for the software that is installed on your computer.

Web Documentation (most up-to-date): Displays the most up-to-date help information located on our website.



Web FAQ Pages: Provides a direct link to the **Frequently Asked Questions** section at www.enercalc.com.

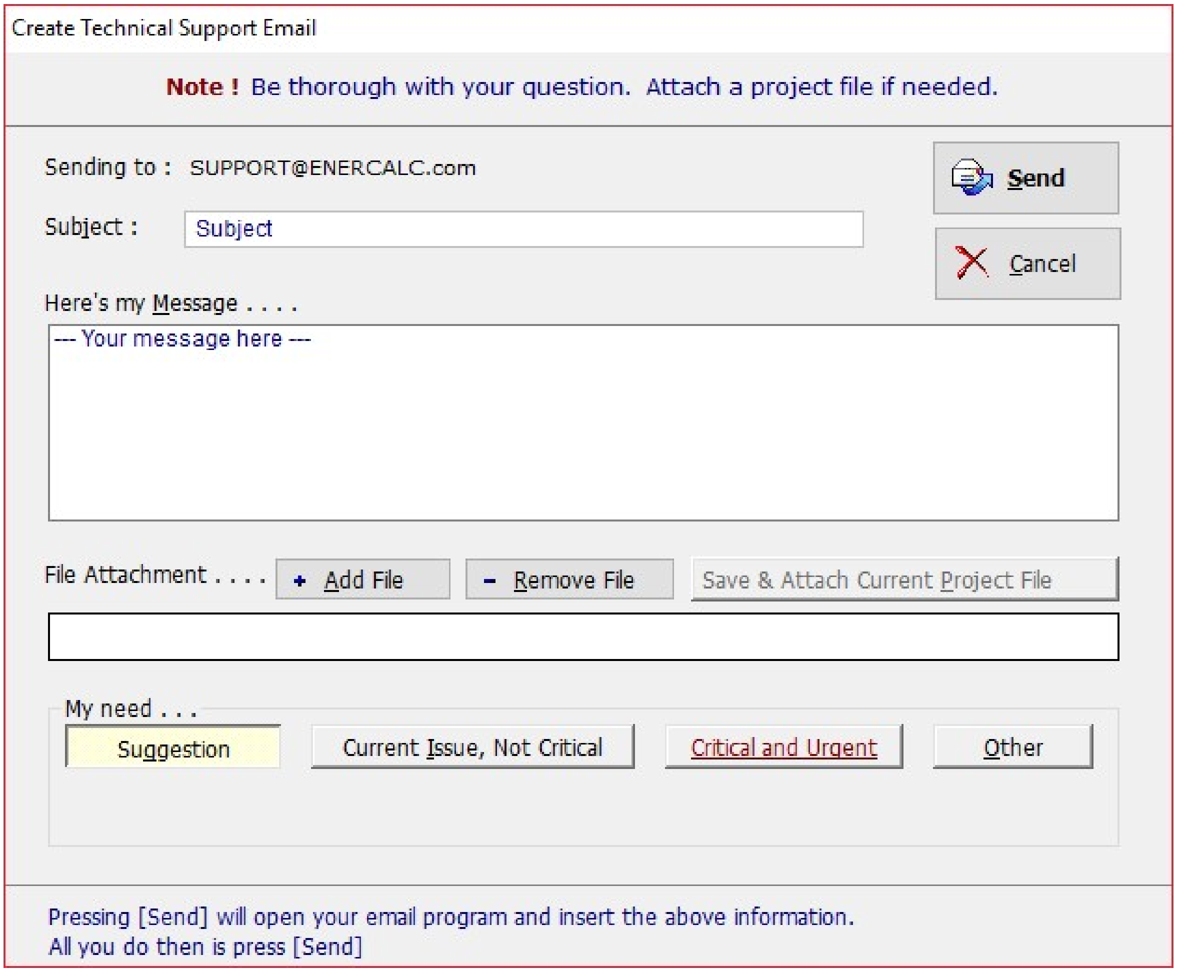
Download PDF Docs: Downloads the PDF version of the documentation for the software currently available at our website. The documentation on our website will always be for the latest available version.

Renew Maintenance & Support Plan: Provides direct access to MSP renewal online.

"For Our Users" Web Page: Provides a link directly to the "For Our Users" page of our website, which offers information on current builds, Customer Information, Software Installation & Update Links, Installation Info & FAQ links, info on Maintenance Plans & Upgrades, Documentation Links, Links to PDF Files of Relevant Information and Useful Forms, and links for Prior Version Software Reinstallation.

"Training Videos" Web Page: Provides a link directly to the "Training Videos" page of our website, which offers videos on topics such as Installation, General Operation, Using Project the Manager, Designing Beams, Columns, Foundations, Slender Walls, working with some of the Miscellaneous modules, and working with External Files.

Create Tech Support Email: Provides you with a Tech Support form to fill out. When finished, the data is transferred to an email form to send to our Tech Support Group. (Note: Some computers and email software will not be able to paste the information into your email program. This is not an ENERCALC issue. In those situations, feel free to compose an email to support@enercalc.com directly from your email program, and please remember to indicate your "KW" User Registration number on all correspondence.)



Check for MAJOR Updates: Immediately runs the update check program named EC6WebUpdate.exe, which resides in the ENERCALC program installation folder. If a newer build is found, a window will be displayed prompting you to decide if you want to continue to update your software. If there is no newer build of the software available, then you will be informed that your installation is currently up-to-date.

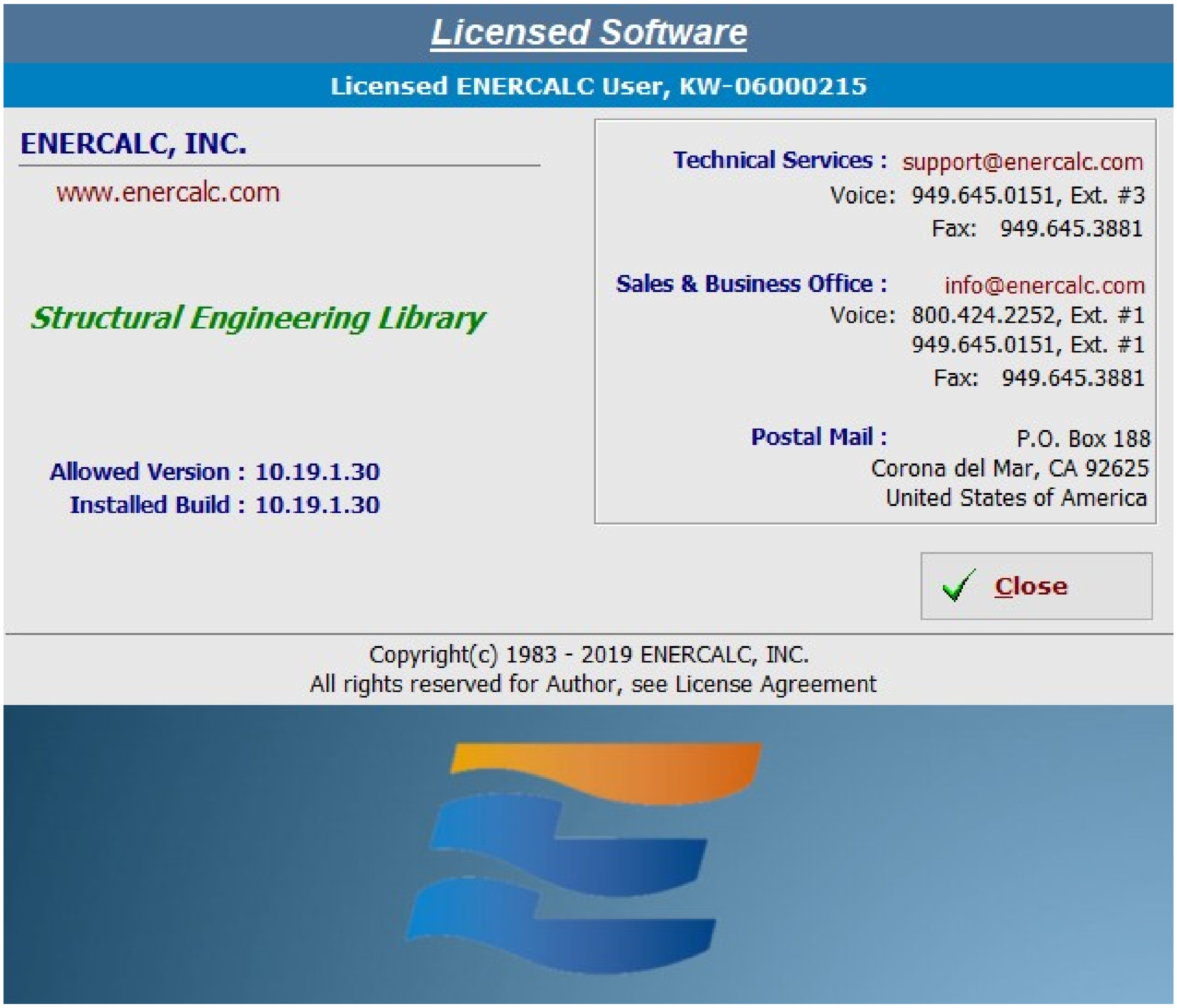
Enhancement/Change History: Opens a window that lists revisions and enhancements that have been performed on the software. Click here 34 to access a specific section with additional information.

Allowed Version Description: Provides a thorough description of the terms "Installed Build" and "Allowed Version" now reported by Structural Engineering Library.

License: Displays the licensing and activation system window. Click here 89 to access a specific section with additional information.

Screen Capture: Provides a utility to create a screen capture, such as to send to us for a tech support question.

About: Displays a general window giving information about software version, copyright, licensed user, activation status, and contact information for ENERCALC.



Part VIII

8 Project Manager

The Project Manager is displayed on the left side of the ENERCALC screen when a Project File has been opened. Click here for a video: Project Manager

The Project Manger provides the ability to create and modify a single Project File that can contain a set of calculations and external items for a specific project. The layout, which will be ever-improving, is designed to allow an ENERCALC Project File to be a single collection point for documents relating to the load development and structural design of a building or other structure.

The behavior of the Project Manager has been enhanced by introducing the ability to display and hide the Project Manager on-demand. This offers the following advantages:

- Larger area is available for use in the Project Manager when it is displayed.
- More verbose calculation descriptions can be provided with fewer cryptic abbreviations.
- Greater selection of "project level" operations buttons can be displayed.
- More efficient organization of GENERAL Division, LOADS & FORCES Division and custom Divisions within the Project Manager.
- Larger area available for the display of input and results when Project Manager is hidden.
- More verbose descriptions of input variables can now be provided.
- Less on-screen congestion and less need for horizontal scroll bars in results tables.

Components that all Project Files AUTOMATICALLY contain are:

GENERAL Division (Only one can exist in a Project File)
A storage location for pieces of information pertaining to areas of the entire project, such as:

- Project Info, Building Department Contact Info, Designer Notes, Revisions, Client, Designer.
- Future additions

LOADS & FORCES Division (Only one can exist in a Project File)
A storage location for pieces of information pertaining to areas of the entire project, such as:

- General forces for the project including Snow Loads, Wind Loads and Seismic Loads.
- Live load reductions.
- Future additions

Components that any Project File CAN contain are:

Custom Divisions (An unlimited number can exist in a Project File)

Storage locations (folders) for organizing user-generated calculations and external items.

- Can be created, named, copied, organized, expanded, collapsed, and deleted.
- Are typically named in ways that are meaningful to the designer and that suit the specific project.
- Are generally populated with calculations and external items.
- Provide a convenient way to selectively control project printing operations for logical sets of calculations.
- Every new Project File is automatically populated with one Division named "Calculations".

Calculations (An unlimited number can exist in a Project File)

User-generated calculations that are created using the built-in ENERCALC modules and are stored in the Custom Divisions.

- Can be created, moved between Divisions, and deleted.
- Can be copied to serve as the basis for a new calculation with similar input data.
- Can be imported from one Project File to another.
- Can be printed singly or in batch mode using the Project Printing Manager.

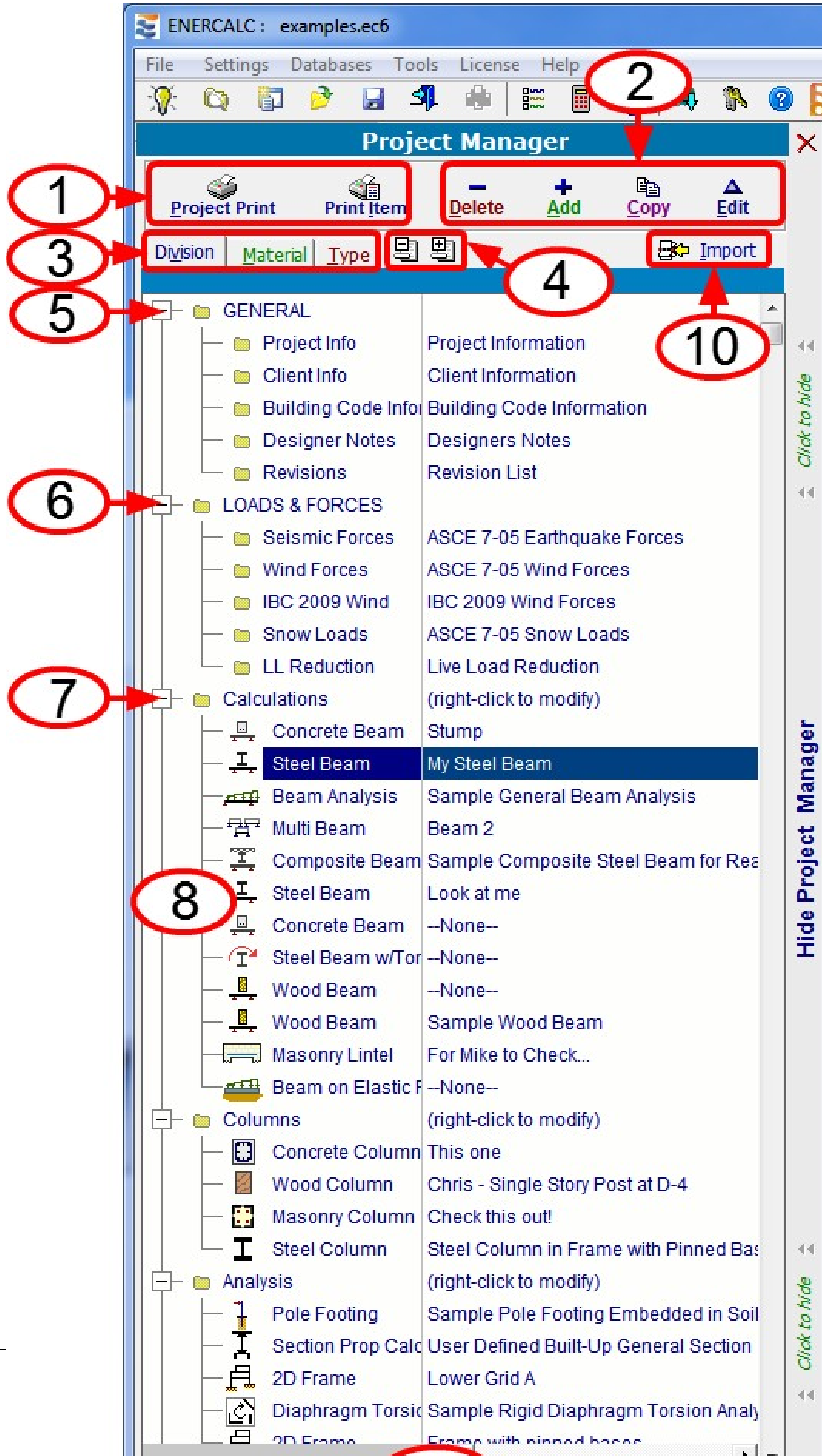
External Items (An unlimited number can exist in a Project File)

User-generated external files that are created using external programs and then stored in the Custom Divisions.

- Can consist of MS Word documents, MS Excel spreadsheets, Adobe Acrobat PDFs, and scanned documents/images.
- Can be embedded within the Project File for maximum portability, or can be linked to the Project File to minimize file size.
- Can be created, moved between Divisions, and deleted.

Please see the descriptive keynotes on the following screen capture for information on various areas of the Project Manager.

Detailed descriptions are contained in the sections of this topic.



ENERCALC : examples.ec6
File Settings Databases Tools License Help
Project Manager
Project Print
Print Item
Delete
Add
Copy
Edit
Division
Material
Type
Import
1
2
3
4
5
6
7
8
10
GENERAL
Project Info Project Information
Client Info Client Information
Building Code Infor Building Code Information
Designer Notes Designers Notes
Revisions Revision List
LOADS & FORCES
Seismic Forces ASCE 7-05 Earthquake Forces
Wind Forces ASCE 7-05 Wind Forces
IBC 2009 Wind IBC 2009 Wind Forces
Snow Loads ASCE 7-05 Snow Loads
LL Reduction Live Load Reduction
Calculations (right-click to modify)
Concrete Beam Stump
Steel Beam My Steel Beam
Beam Analysis Sample General Beam Analysis
Multi Beam Beam 2
Composite Beam Sample Composite Steel Beam for Rea
Steel Beam Look at me
Concrete Beam --None--
Steel Beam w/Tor --None--
Wood Beam --None--
Wood Beam Sample Wood Beam
Masonry Lintel For Mike to Check...
Beam on Elastic F --None--
Columns (right-click to modify)
Concrete Column This one
Wood Column Chris - Single Story Post at D-4
Masonry Column Check this out!
Steel Column Steel Column in Frame with Pinned Bas
Analysis (right-click to modify)
Pole Footing Sample Pole Footing Embedded in Soil
Section Prop Calc User Defined Built-Up General Section
2D Frame Lower Grid A
Diaphragm Torsic Sample Rigid Diaphragm Torsion Analy
Click to hide
Hide Project Manager
Click to hide

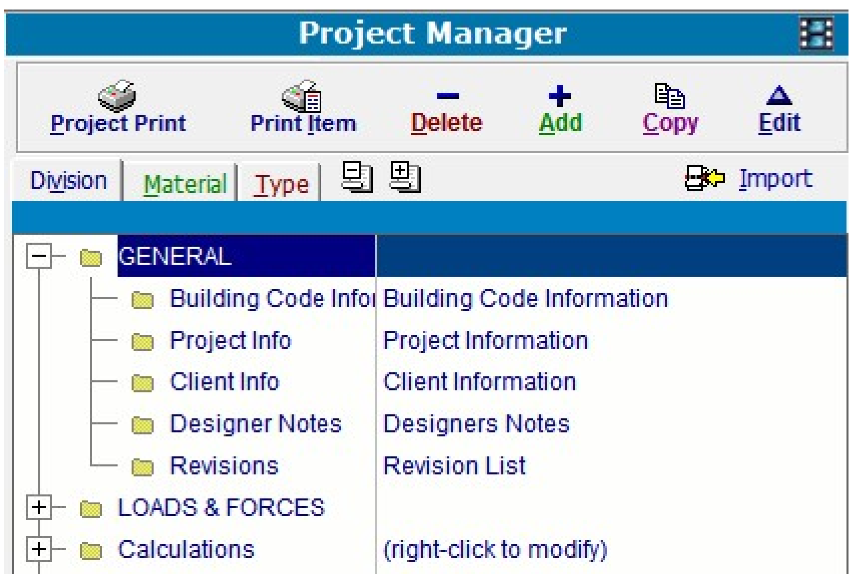
(1) **Project Printing:** These buttons allow you to print the report for the selected calculation, or open the Project Printing system, which allows you to review all report pages for all calculation items in the entire project. Click here 131 to review the specific section on Project Printing.

(2) **Buttons to Add, Copy, Edit & Delete Individual Calculations:** These buttons allow you to add, copy, edit and delete ENERCALC calculations or external source items (such as Excel sheets) in the selected Division. Click here 116 to review specific section.

(3) **Sort Options:** These three tabs sort your Project File content by Division grouping, Type of calculation (Steel Beam, Concrete Column, Excel sheet, etc.) or Material (Concrete, Steel, Wood, etc.). Click here 118 to review specific section. Note that the [**Add**], [**Copy**], and [**Delete**] options are not available in the Type view or the Material view.

(4) **Expand & Contract Tree:** These two buttons fully expand or fully contract the tree structure to display or hide the full contents of the Project File.

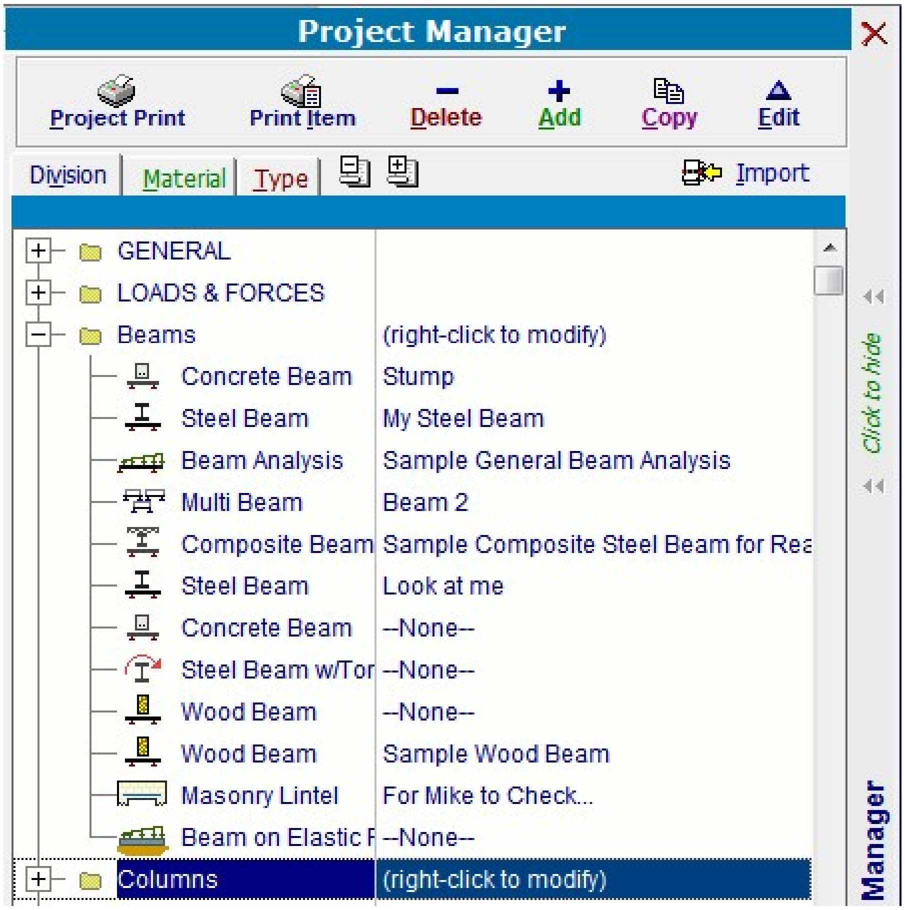
(5) **GENERAL Division:** The GENERAL Division is automatically created in each new Project File. There can only be one GENERAL Division, and its purpose is to contain and organize certain hard-coded pieces of general information that apply to the Project as a whole. Click here 101 to review the section describing these items. The screen capture below shows a view of the Project Manager with the GENERAL Division expanded to display its contents:



(6) **LOADS & FORCES Division:** The LOADS & FORCES Division is also automatically created in each new Project File. There can only be one LOADS & FORCES Division, and its purpose is to contain and organize load related calculations. The screen capture below shows a view of the Project Manager with the LOADS & FORCES Division expanded to display its contents:

(7) **Calculation Division:** The Calculations Division is also automatically created in each new Project File, but it is created merely as a user-convenience. It can be thought of as a "custom" Division, because its name can be changed and it can be moved or deleted. The user is free to create as many custom Divisions as the Project warrants, and to name them as best suits the Project.

(8) **Calculation List:** This is the main list that displays all the calculation items that you have added into your project. It allows you to organize your calculations into Divisions. See image below for another look at two Divisions labeled "Beams" and "Columns". Columns shows a [**+**] button to its left, indicating that the tree containing the calculations in that Division is compressed. Click here 108 to review the section describing these items.



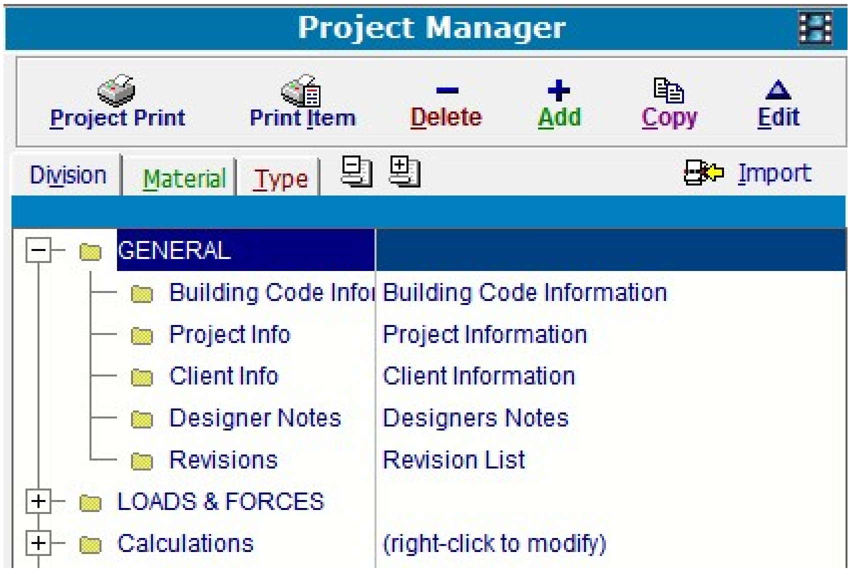
(9) **Move Item in List:** These buttons move the highlighted calculation up and down within the list. Click here 122 to review specific section.

(10) **Import Calculations from other Project Files:** Click here 126 to review specific section.

8.1 General Division

This Division contains a list of items that describe the entire project. Click here for a video:

General Division



Building Code Information 102: Allows entry of information relating to the building code, jurisdiction and building official's contact information.

Project Information 103: Allows entry of general information about the project.

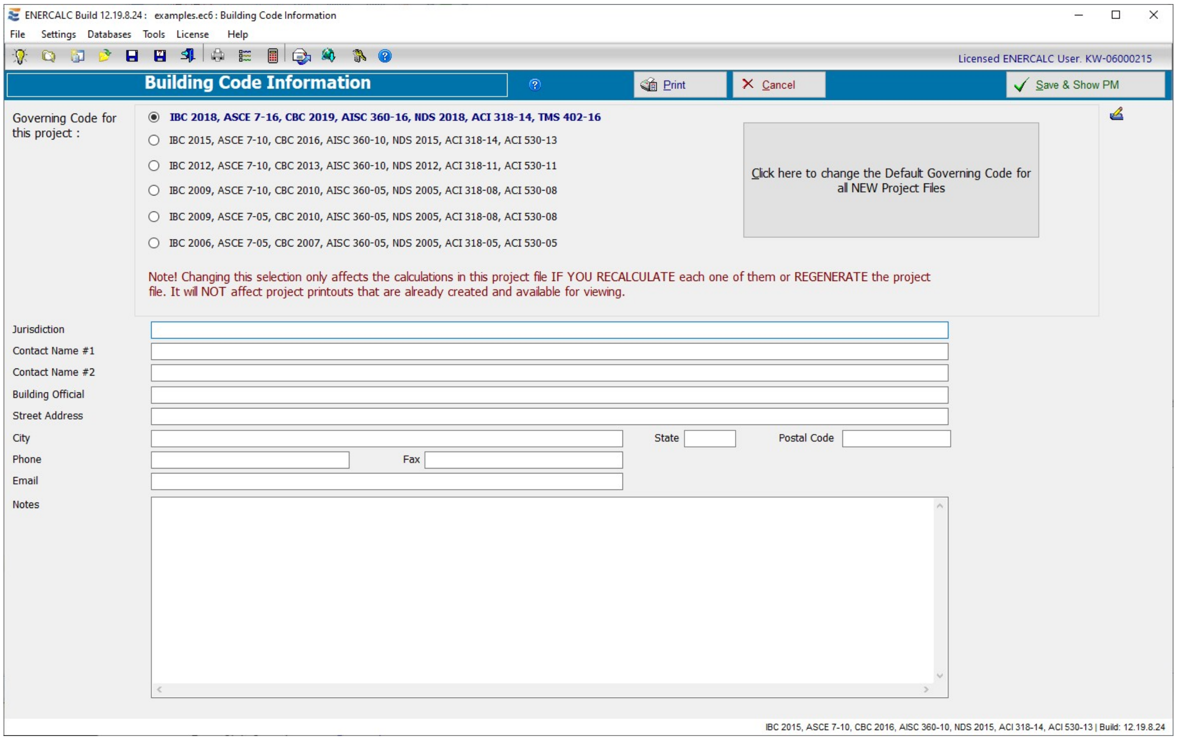
Client Information 104: Allows entry of information specific to the engineer's client.

Designer Notes 105: Allows entry of up to 18 specific notes assigned by a specific person on a certain date.

Revisions 106: Allows entry of up to 18 specific revisions assigned by a specific person on a certain date.

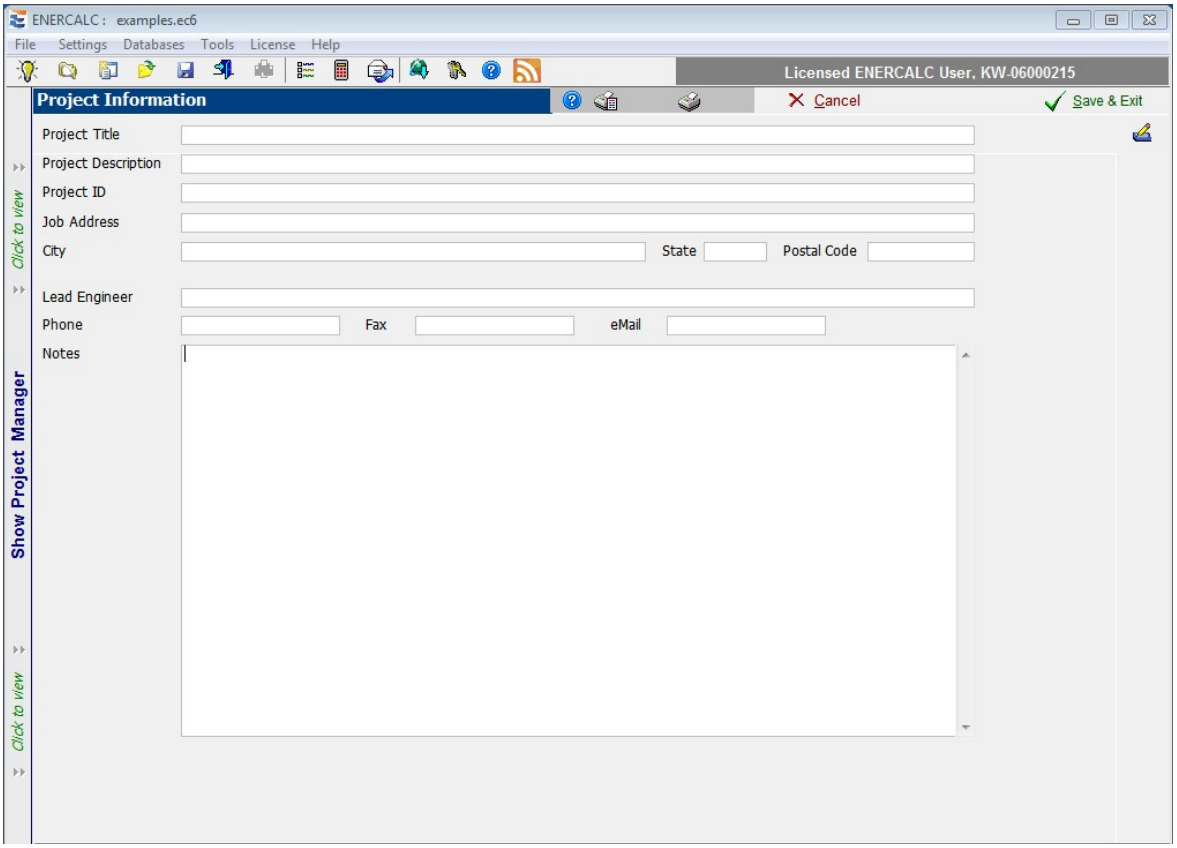
8.1.1 Building Code Information

This form allows you to select the Governing Building Code and Default Load Combination Set, and to enter various information items pertaining to the jurisdiction where your project is located. This data is specific to your Project File.



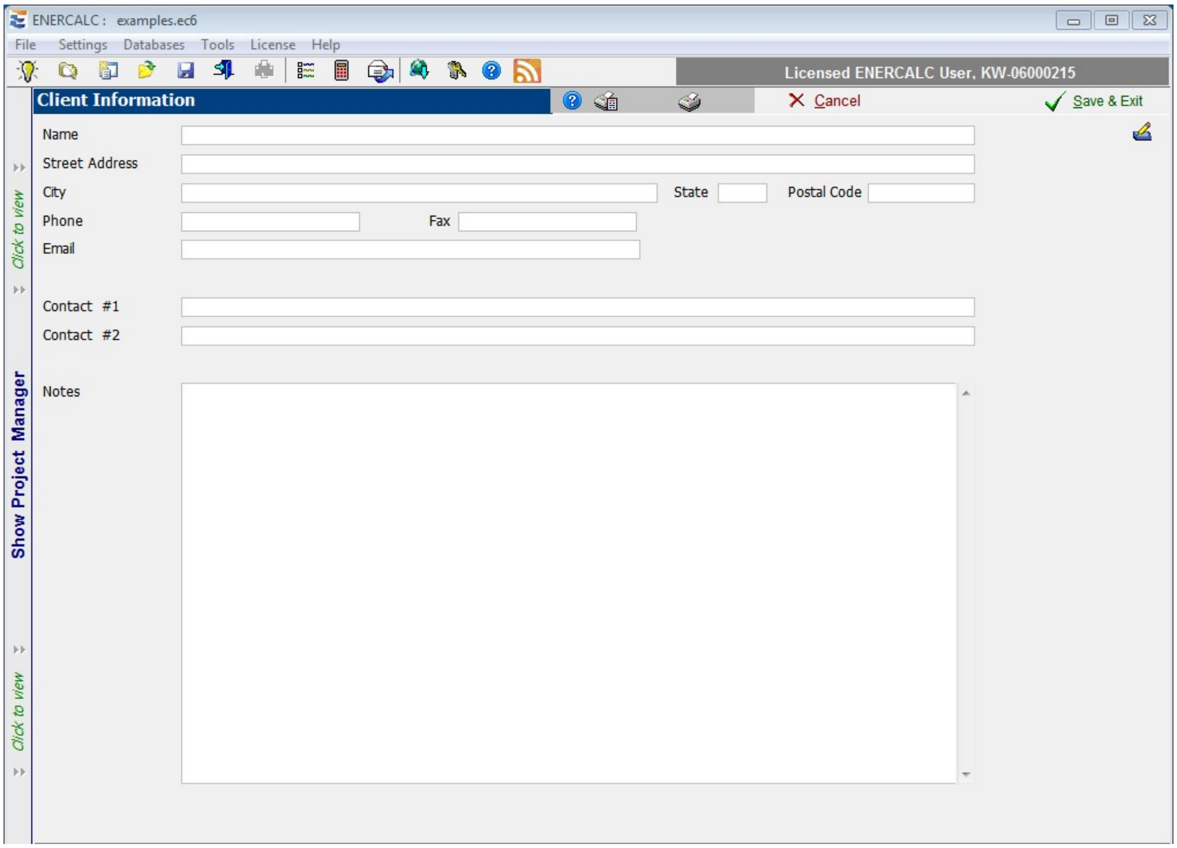
8.1.2 Project Information

This form allows you to enter information specific to your project. This data is specific to your Project File.



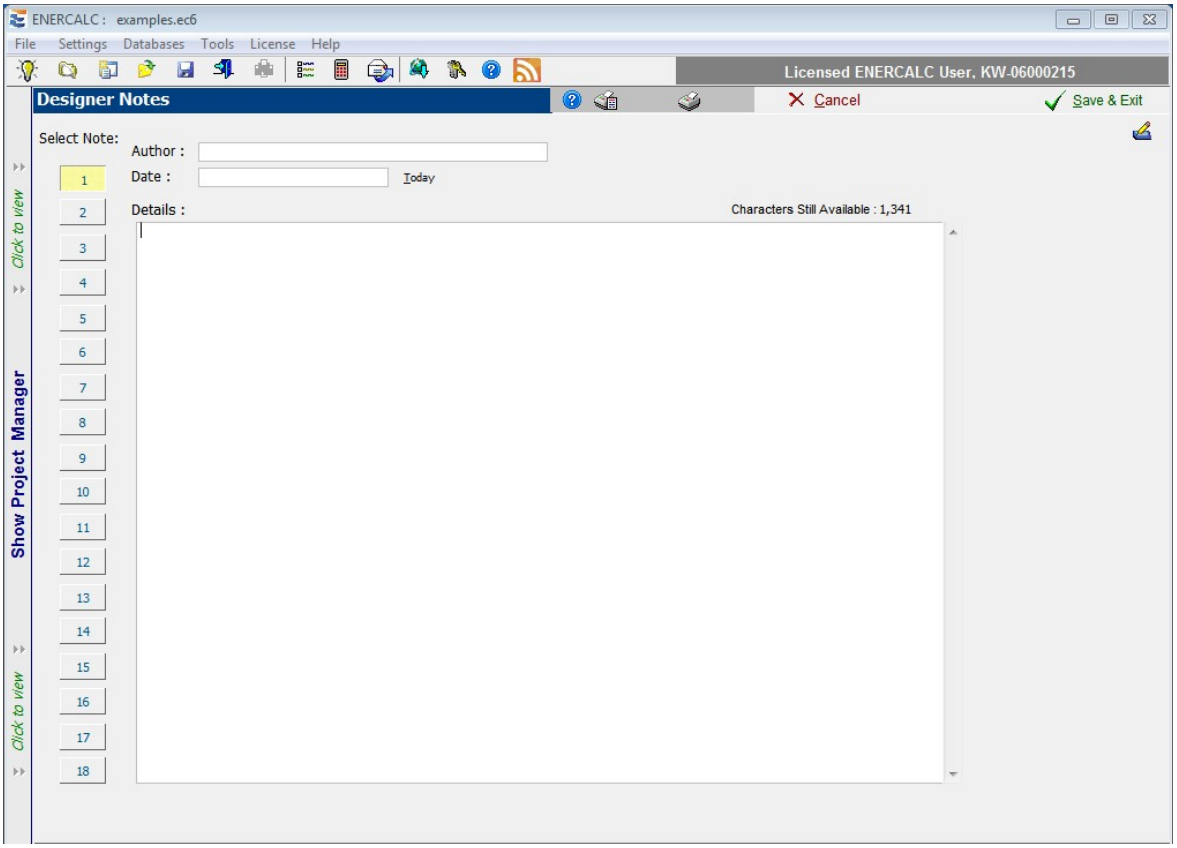
8.1.3 Client Information

This form allows you to enter information on your client for a specific project. This data is specific to your Project File.



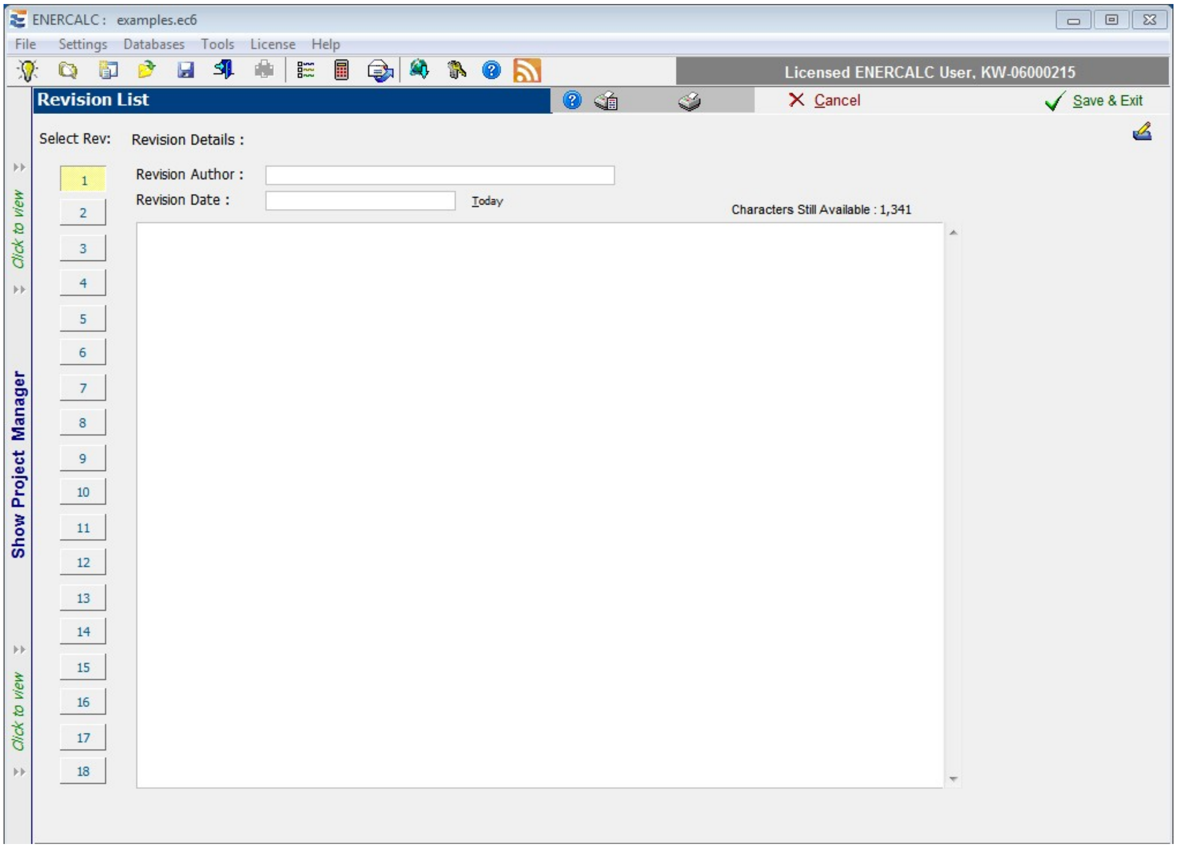
8.1.4 Designer Notes

This form allows you to enter up to 18 distinct notes on the project, each with a specific author and creation date. This data is specific to your Project File.



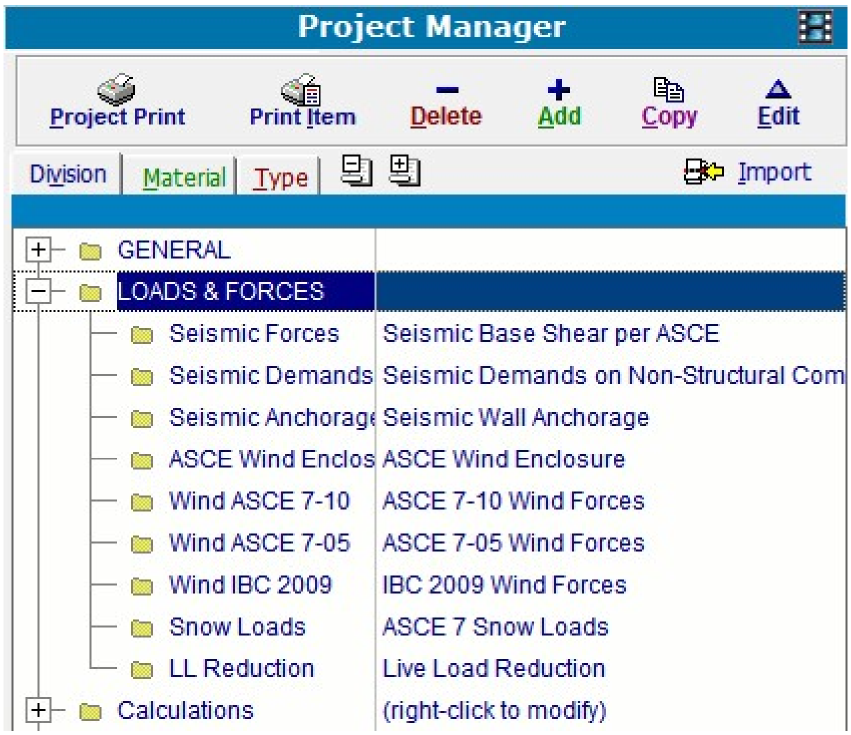
8.1.5 Revision List

This form allows you to enter up to 18 distinct revision explanations on the project, each with a specific author and creation date. This data is specific to your Project File.



8.2 Loads & Forces Division

The LOADS & FORCES Division contains a growing number of calculations including Snow Loads, Live Load Reduction, Wind Loads, and Seismic Loads.



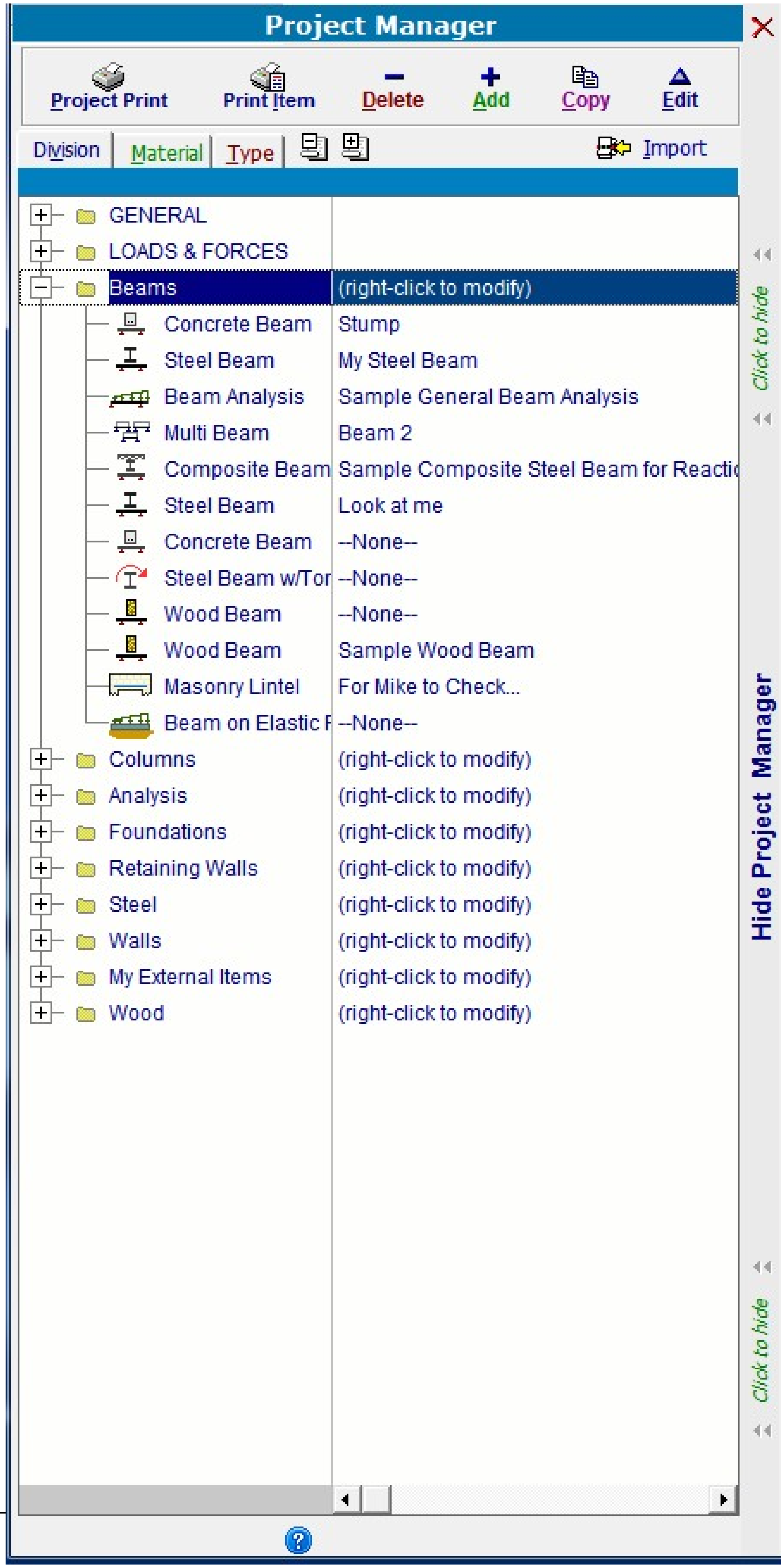
See specific descriptions under Calculation Modules > Loads & Forces Division 107.

8.3 Calculation List

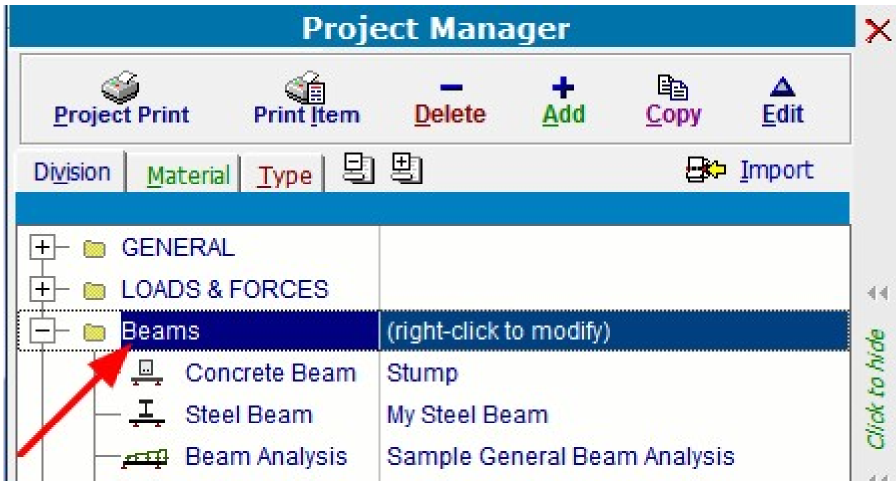
The Calculation List is the main Project Manager view that you will use when building your calculation sets for projects. Click here for a video: Calculations Division

Most of a normal work session consists of adding and editing calculations in the list.

Please see the notes below as well as the following sections to learn what each button provides and how to manage the list of calculations.

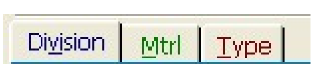

Project Manager
Project Print
Print Item
Delete
Add
Copy
Edit
Division
Material
Type
Import
GENERAL
LOADS & FORCES
Beams
(right-click to modify)
Concrete Beam
Stump
Steel Beam
My Steel Beam
Beam Analysis
Sample General Beam Analysis
Multi Beam
Beam 2
Composite Beam
Sample Composite Steel Beam for Reactic
Steel Beam
Look at me
Concrete Beam
--None--
Steel Beam w/Tor
--None--
Wood Beam
--None--
Wood Beam
Sample Wood Beam
Masonry Lintel
For Mike to Check...
Beam on Elastic F
--None--
Columns
(right-click to modify)
Analysis
(right-click to modify)
Foundations
(right-click to modify)
Retaining Walls
(right-click to modify)
Steel
(right-click to modify)
Walls
(right-click to modify)
My External Items
(right-click to modify)
Wood
(right-click to modify)
Click to hide
Hide Project Manager
Click to hide

Divisions: Divisions are the main organizational category for calculations. These are the left-most text item in the calculation list and will display either a [**+**] or [**-**] to the left of their name. Divisions can be created, copied, renamed, moved, and deleted. Calculations must always exist within a Division, but they can be moved freely between Divisions. Click Here 112 for the specific information section. The screen capture below illustrates an example of how a Division can be renamed with a title that is meaningful to your project:

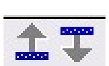


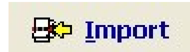
Adding, Deleting, Copying: The four buttons shown below allow you to add, copy, edit, or delete calculations and external items. Click here 116 for the specific information section.

Sorting by Division, Type and Material: Click one of these tabs to change the sorting view of the calculations in your project. Click here 118 for the specific information section.



Changing Calculation and Division Order: The buttons you see below are used to move the highlighted calculation or Division up or down in the list. When moving a calculation downward, if moving a calculation would tend to replace a Division name, then the calculation item is moved into that Division. When moving a calculation upward, if moving a calculation would tend to replace a Division name, then the calculation item is moved into the Division above. Click here 122 for the specific information section.

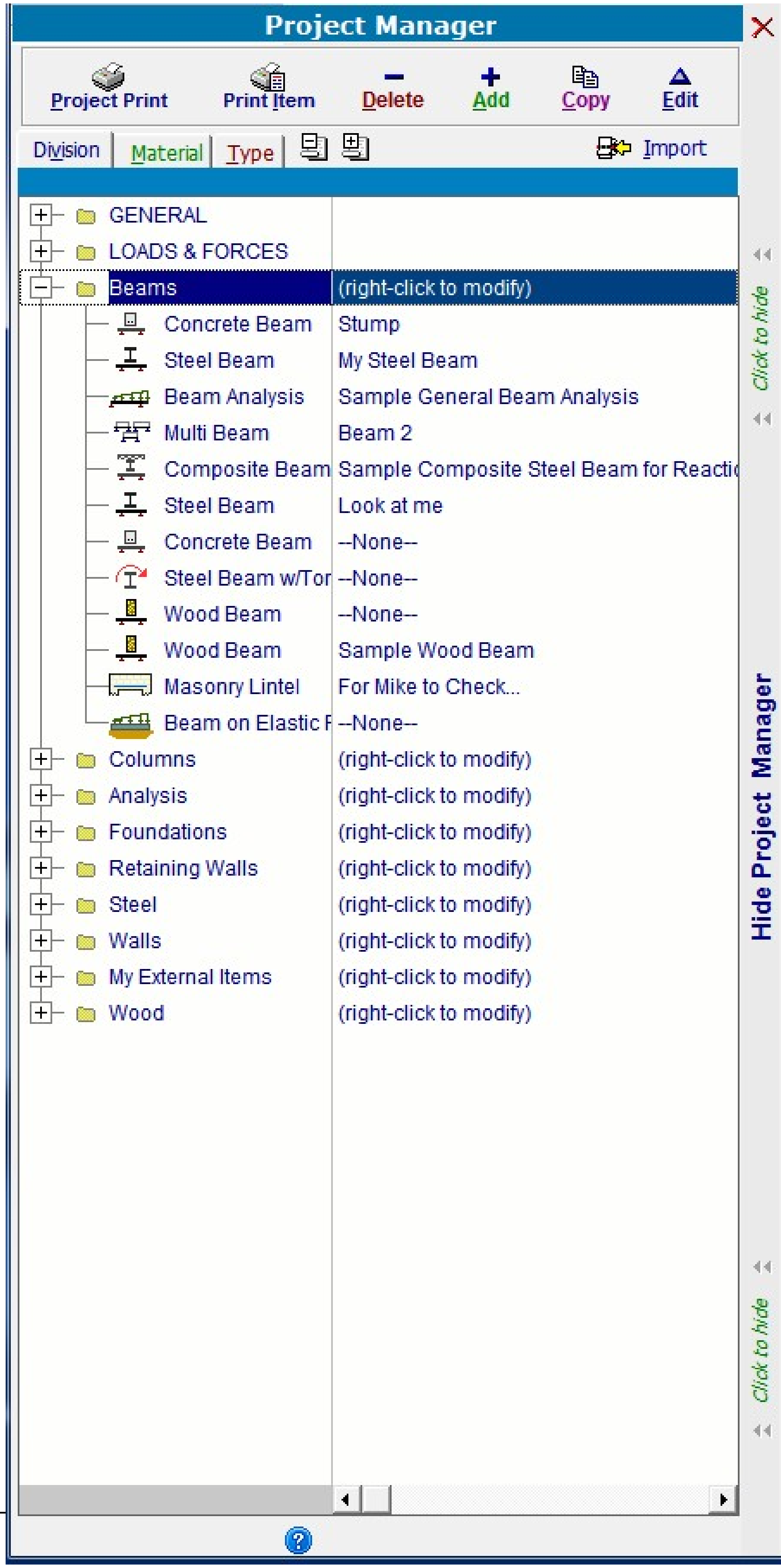


Importing Calculations: Clicking the [**Import**] button 

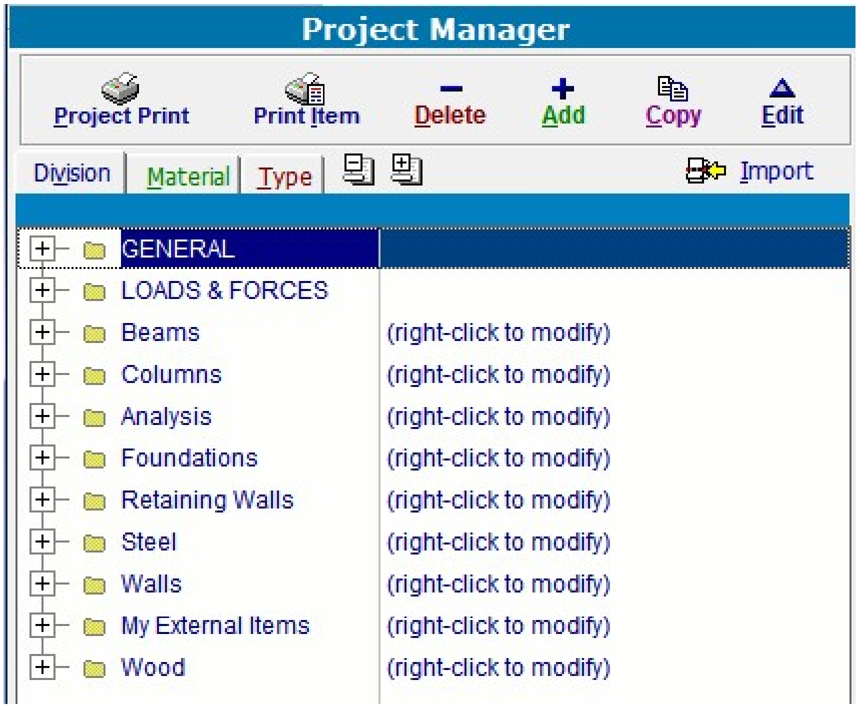
displays the calculation import system. Click here 126 for the specific information section.

8.3.1 Divisions

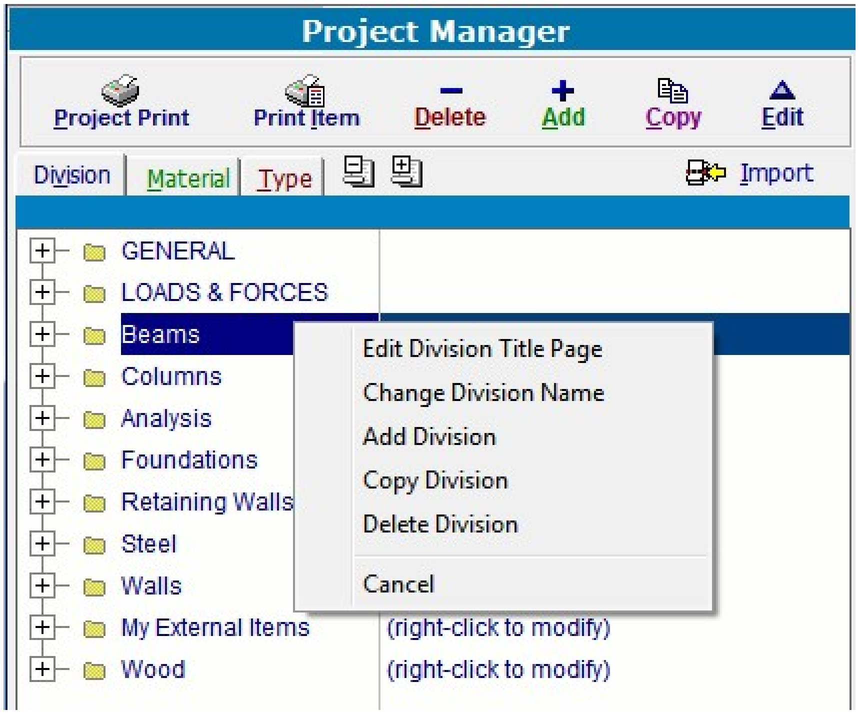
Divisions are used to organize calculations. The image below has custom Divisions named Beams, Columns, Analysis, etc.


Project Manager
Project Print
Print Item
Delete
Add
Copy
Edit
Division
Material
Type
Import
GENERAL
LOADS & FORCES
Beams (right-click to modify)
Concrete Beam Stump
Steel Beam My Steel Beam
Beam Analysis Sample General Beam Analysis
Multi Beam Beam 2
Composite Beam Sample Composite Steel Beam for Reacti
Steel Beam Look at me
Concrete Beam --None--
Steel Beam w/Tor --None--
Wood Beam --None--
Wood Beam Sample Wood Beam
Masonry Lintel For Mike to Check...
Beam on Elastic F --None--
Columns (right-click to modify)
Analysis (right-click to modify)
Foundations (right-click to modify)
Retaining Walls (right-click to modify)
Steel (right-click to modify)
Walls (right-click to modify)
My External Items (right-click to modify)
Wood (right-click to modify)
Click to hide
Hide Project Manager
Click to hide

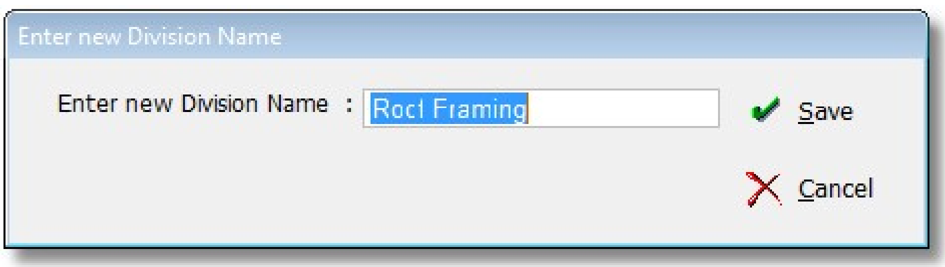
- Clicking the [-] icon to the left of the Division name will compress the Division tree. See image below...note how only the Division names are displayed, but their contents are currently not visible.



Left-clicking on a Division name will display the following pop-up menu:



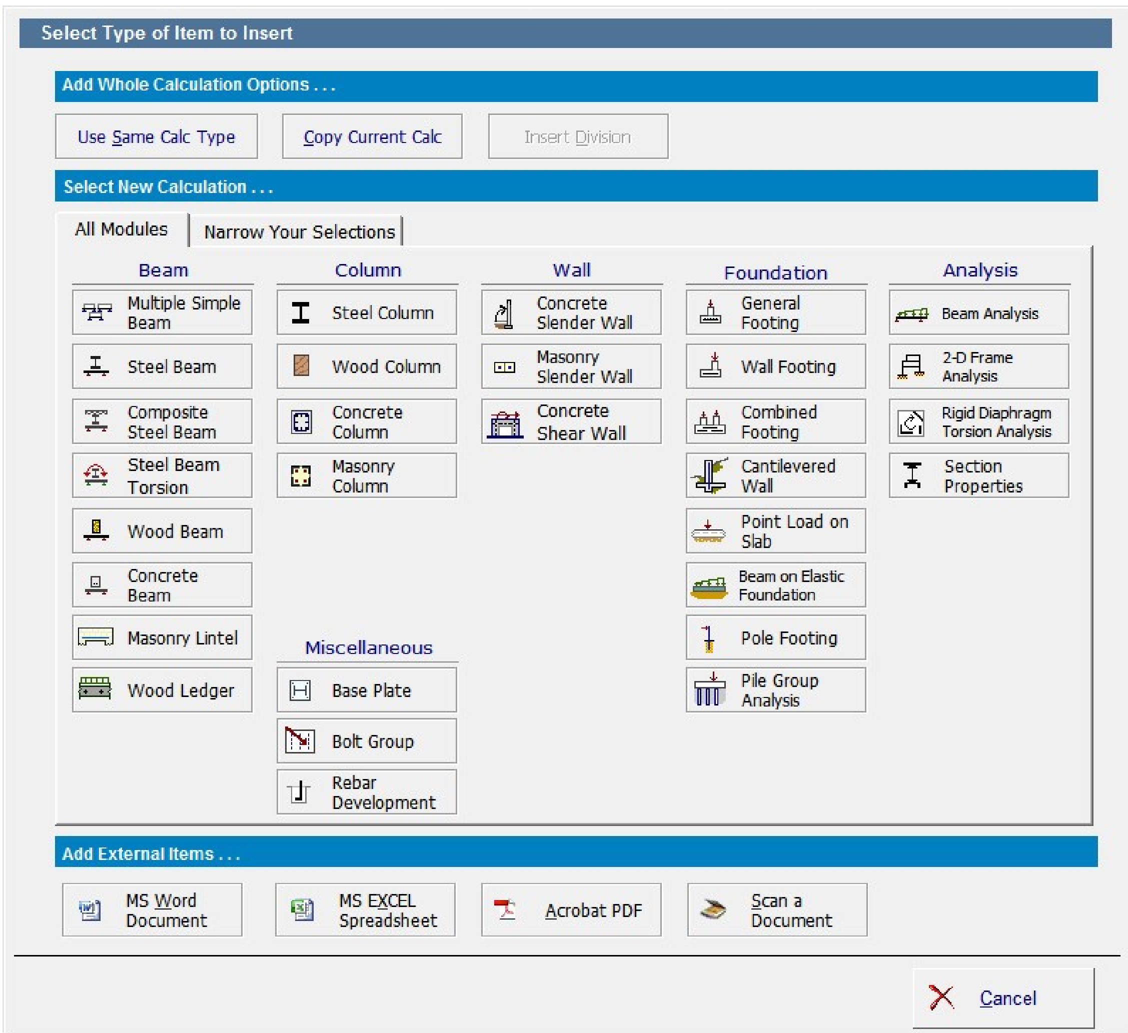
Selecting [**Change Division Name**] will display the following dialog:



8.3.2 Adding, Deleting, Copying

The four buttons shown below are used to manipulate the calculations and Divisions in your current Project File:

Add: Clicking [**Add**] displays the dialog below, where you can select the type of item to add to your Project File. The window contains two categories of items: Calculations and External Items.



The available Calculations are listed in the top portion of the dialog (and there will be more as the product matures). These are ENERCALC-created structural engineering calculations that you can use. The lower portion of the dialog lists the available External

Items that can be created externally from the ENERCALC software package and then inserted into an ENERCALC Project File. These item types are:

Microsoft WORD Document: You can include a Microsoft Word document as an item in your Project File. Click here 620 to go to the specific section.

Microsoft EXCEL Spreadsheet: You can include a Microsoft Excel spreadsheet as an item in your Project File. Click here 624 to go to the specific section.

Adobe Acrobat PDF File: You can include a PDF file as an item in your Project File. Click here 628 to go to the specific section.

Scanned Document: You can set up your scanner and scan documents or images to include as an item in your Project File. Click here 632 to go to the specific section.

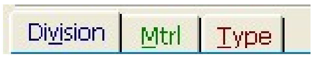
Copy: For the currently highlighted item in the calculation list, clicking [**Copy**] creates a new calculation of the same type using all of the information in the current calculation to make the new calculation. The copy is added to the list and is automatically opened for editing.

Edit: For the currently highlighted item in the calculation list, clicking [**Edit**] opens the calculation for editing.

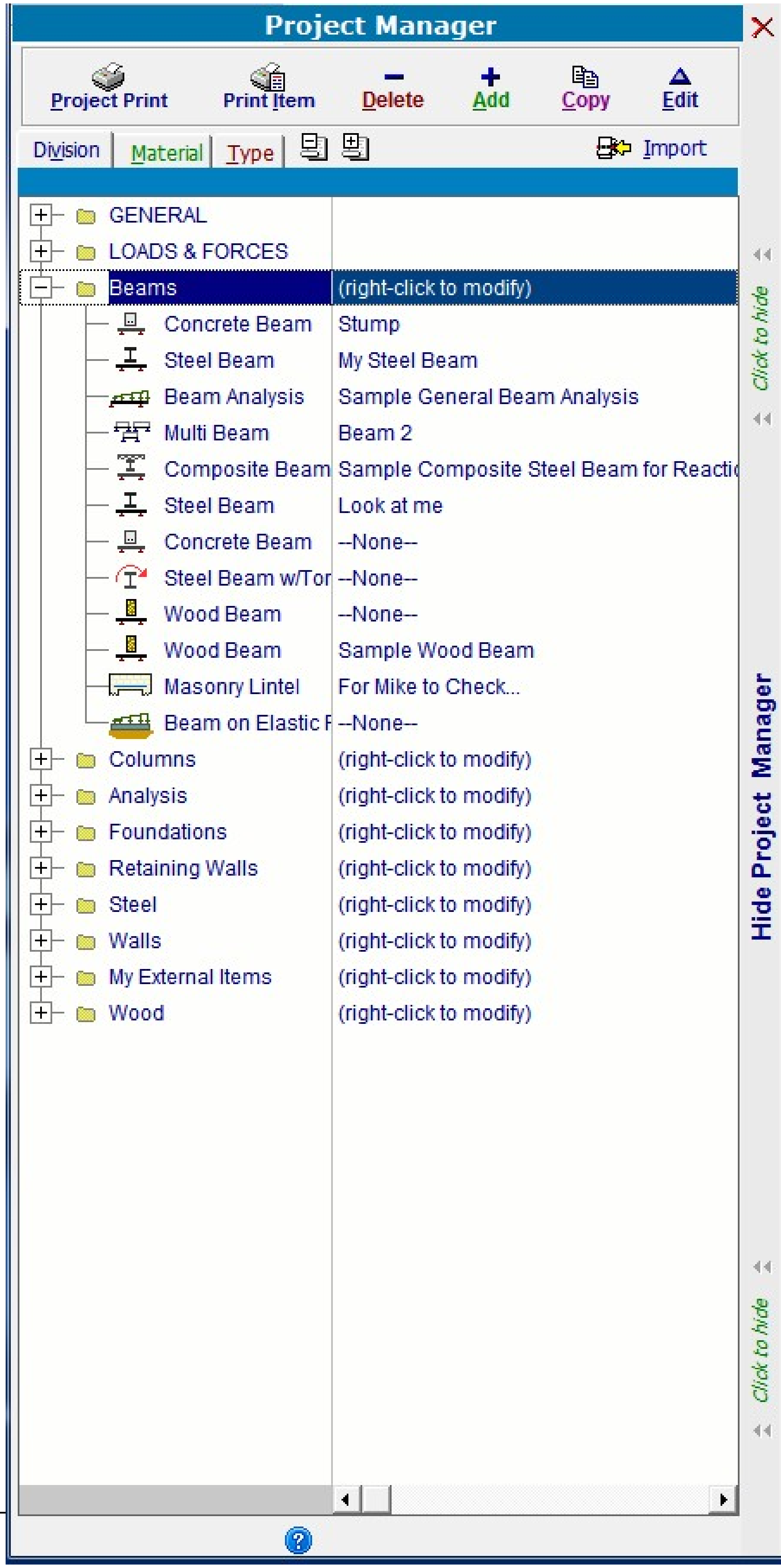
Delete: Deletes the item that is currently highlighted in the calculation list. You are prompted to confirm before the deletion is made.

8.3.3 Sorting by Division, Type & Material

Select the "Division", "Material", or "Type" tab to sort the calculation list based on these three primary organizational hierarchies.

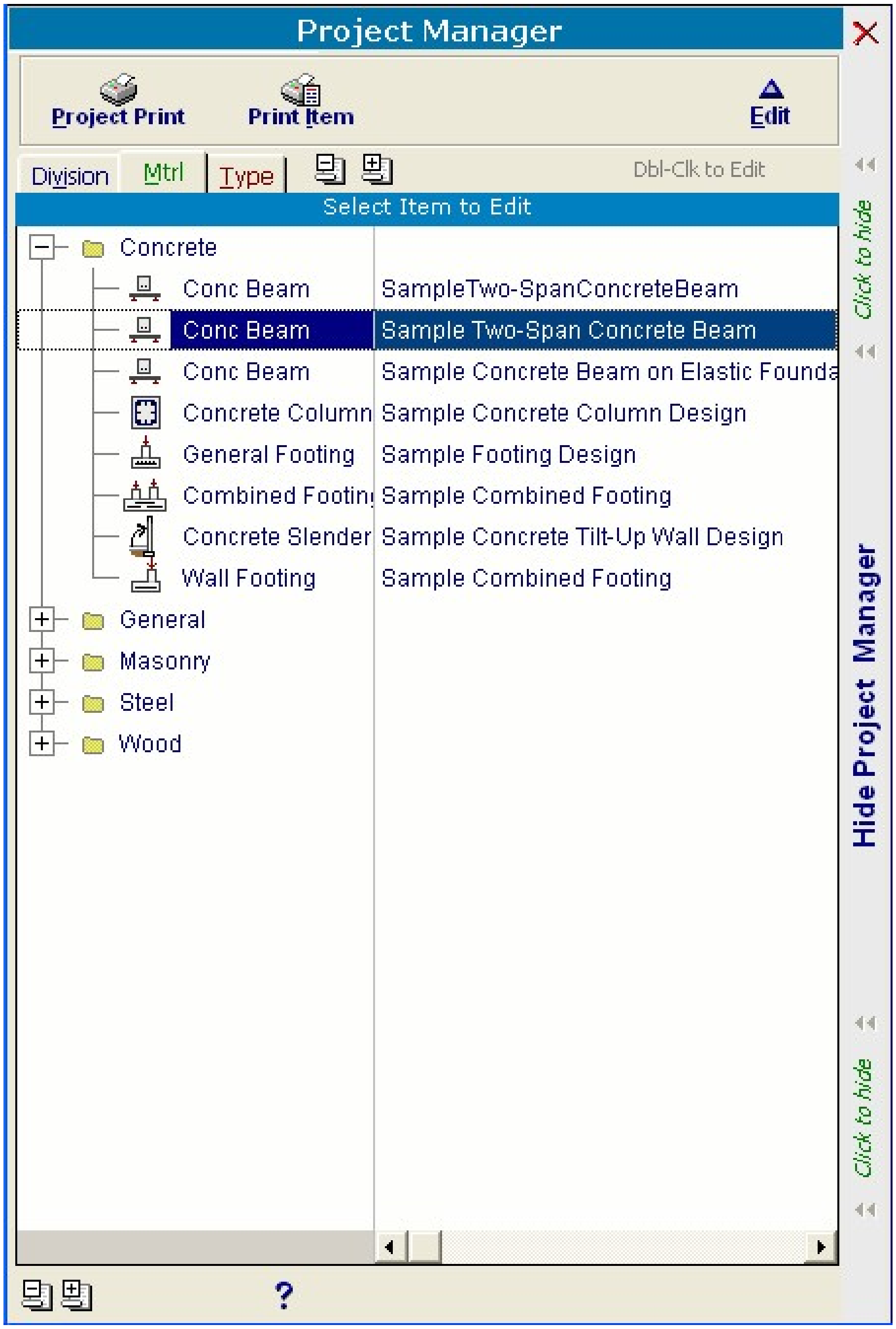


Division: Organizes the calculation list by the Divisions you created, and sorts the calculations within each Division in the order that you established. See below.

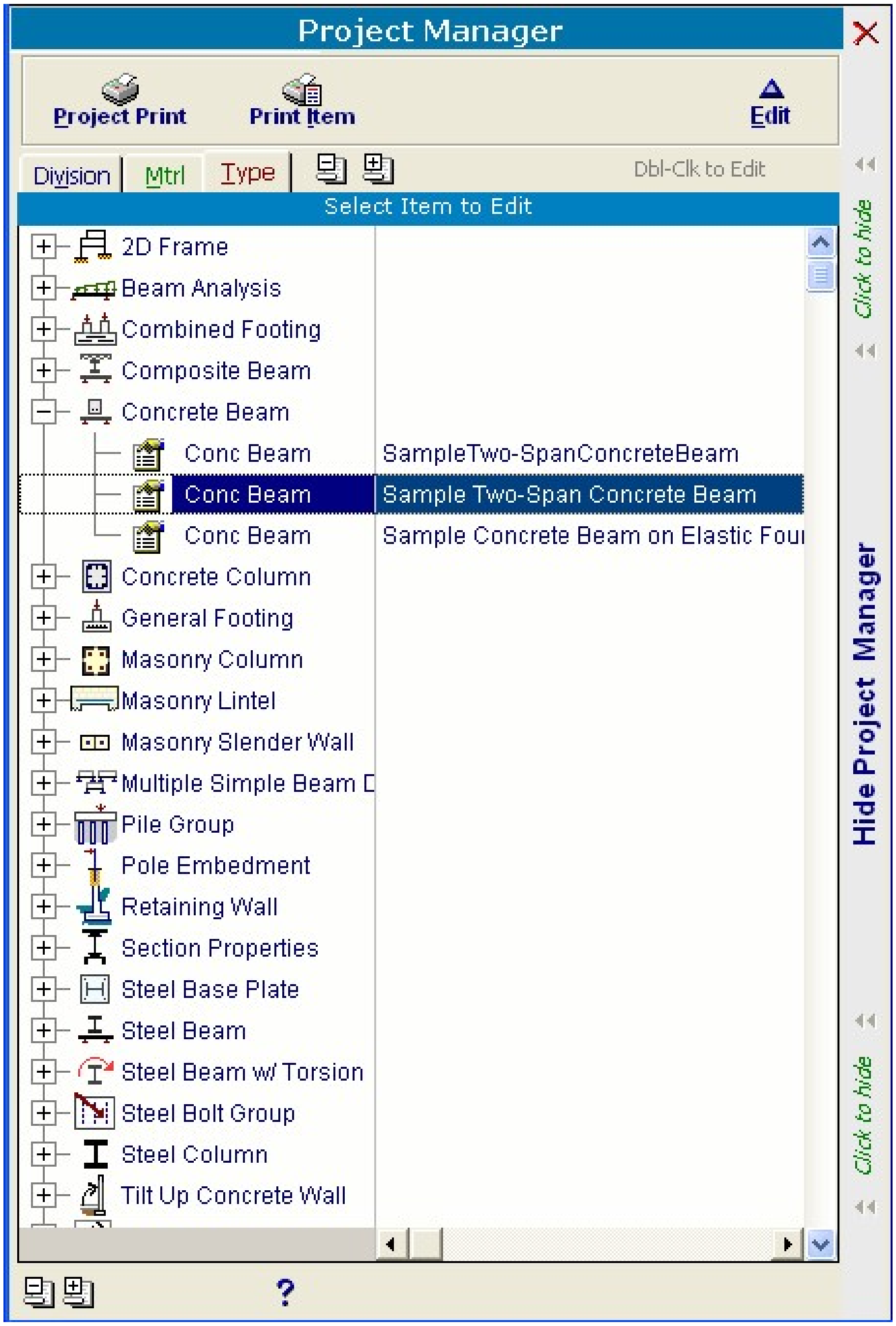


Project Manager
Project Print
Print Item
Delete
Add
Copy
Edit
Division
Material
Type
Import
GENERAL
LOADS & FORCES
Beams
(right-click to modify)
Concrete Beam
Stump
Steel Beam
My Steel Beam
Beam Analysis
Sample General Beam Analysis
Multi Beam
Beam 2
Composite Beam
Sample Composite Steel Beam for Reacti
Steel Beam
Look at me
Concrete Beam
--None--
Steel Beam w/Tor
--None--
Wood Beam
--None--
Wood Beam
Sample Wood Beam
Masonry Lintel
For Mike to Check...
Beam on Elastic F
--None--
Columns
(right-click to modify)
Analysis
(right-click to modify)
Foundations
(right-click to modify)
Retaining Walls
(right-click to modify)
Steel
(right-click to modify)
Walls
(right-click to modify)
My External Items
(right-click to modify)
Wood
(right-click to modify)
Click to hide
Hide Project Manager
Click to hide

Material: Organizes the calculation list by the materials to which they refer, and sorts the calculations within each material in the order that you established. See below.

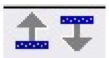


Type: Organizes the calculation list by calculation type, and sorts the calculations within each type in the order that you established. See below.

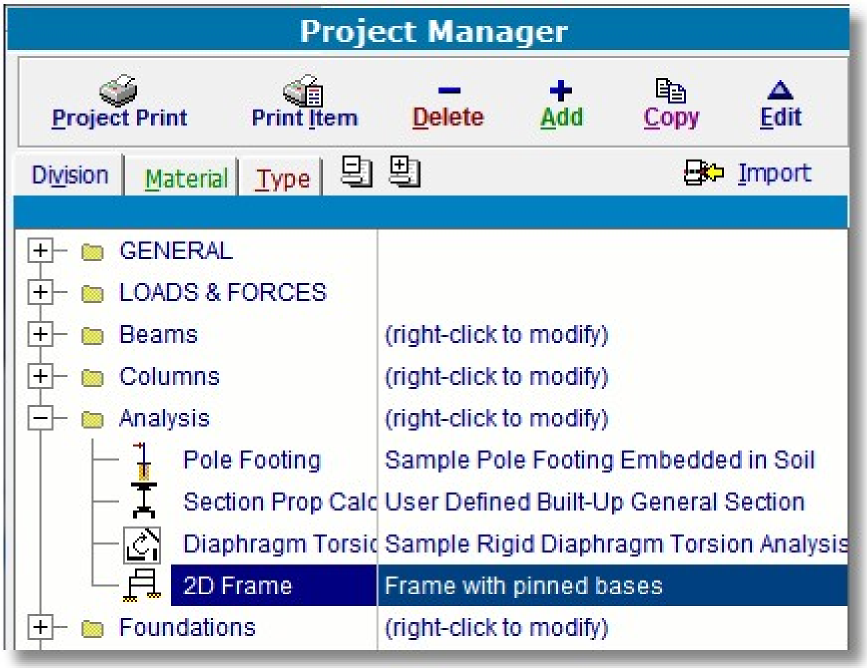

Project Manager
Project Print
Print Item
Edit
Division
Mtrl
Type
Dbl-Clk to Edit
Select Item to Edit
2D Frame
Beam Analysis
Combined Footing
Composite Beam
Concrete Beam
Conc Beam
SampleTwo-SpanConcreteBeam
Conc Beam
Sample Two-Span Concrete Beam
Conc Beam
Sample Concrete Beam on Elastic Fou
Concrete Column
General Footing
Masonry Column
Masonry Lintel
Masonry Slender Wall
Multiple Simple Beam D
Pile Group
Pole Embedment
Retaining Wall
Section Properties
Steel Base Plate
Steel Beam
Steel Beam w/ Torsion
Steel Bolt Group
Steel Column
Tilt Up Concrete Wall
Click to hide
Hide Project Manager
Click to hide

8.3.4 Changing Calculation Order

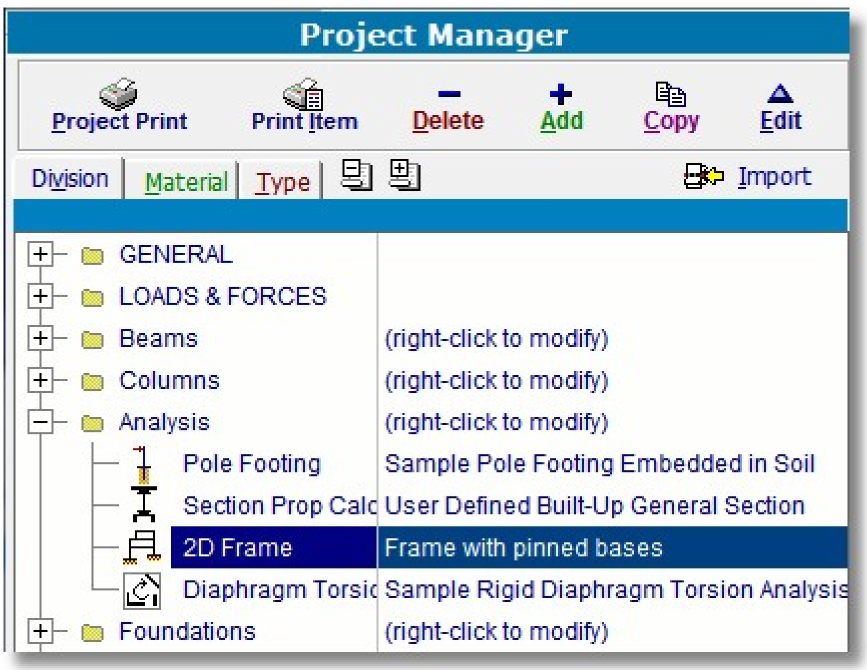
The two buttons shown at the bottom of the calculation list (see image below) are used to move the currently highlighted calculation or Division up or down in the list.



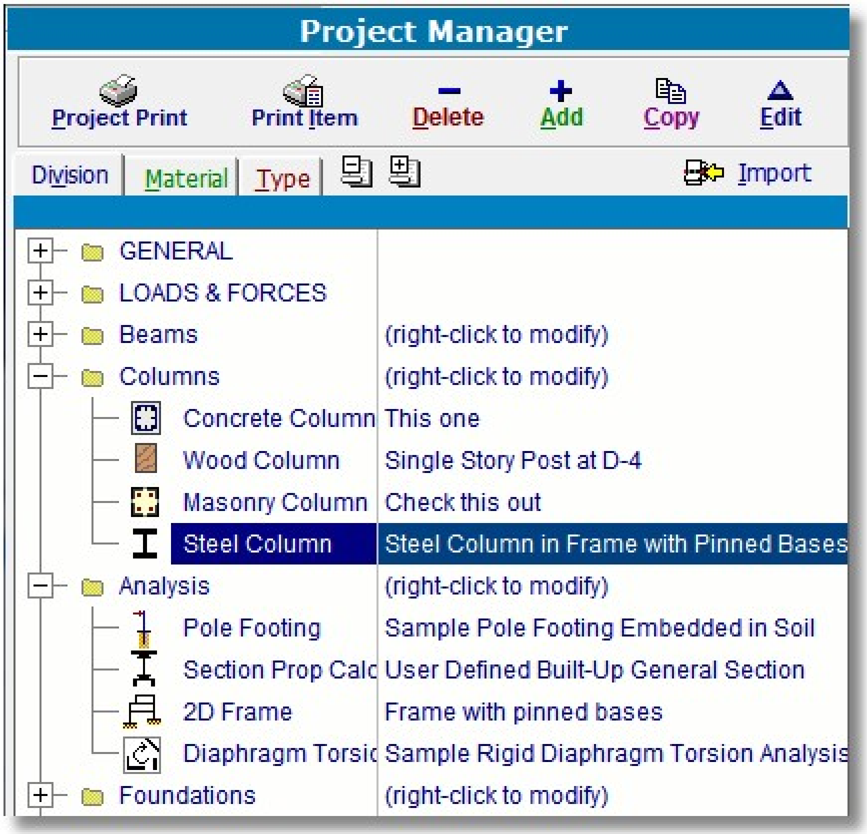
In the images below, note the button that is used to move the item upward in the list:

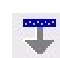


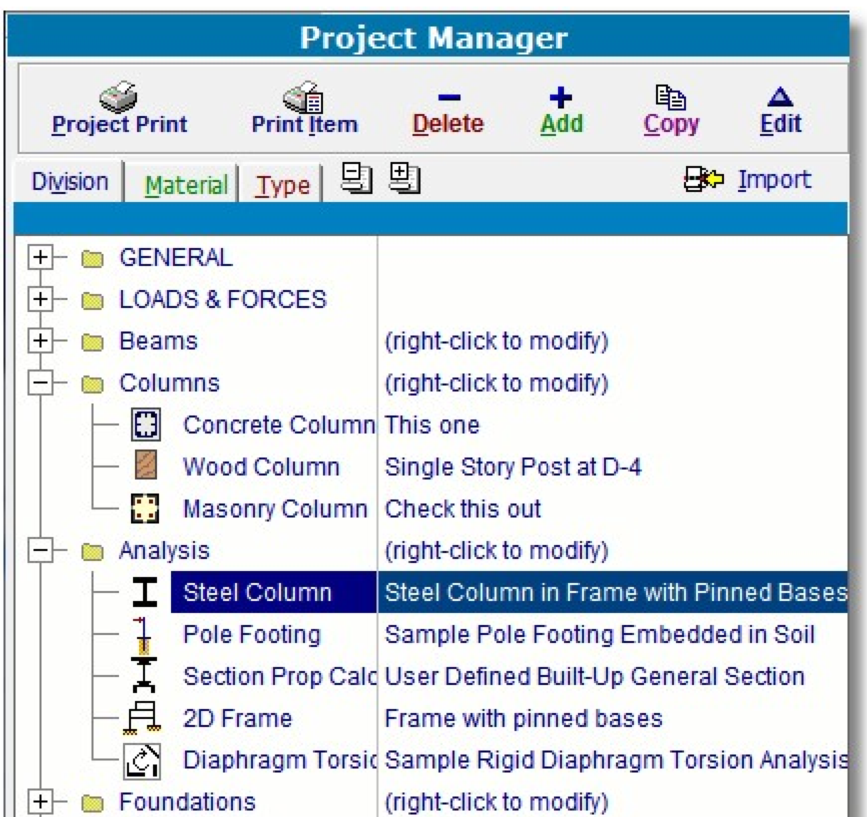
>  >



Now we will show that moving the last item in a Division downward will move the item to the top of the next Division. Notice how the Wood Column calculation moves from the bottom of Floor Framing to the top of Analysis.


Project Manager
Project Print
Print Item
Delete
Add
Copy
Edit
Division
Material
Type
Import
GENERAL
LOADS & FORCES
Beams
(right-click to modify)
Columns
(right-click to modify)
Concrete Column
This one
Wood Column
Single Story Post at D-4
Masonry Column
Check this out
Steel Column
Steel Column in Frame with Pinned Bases
Analysis
(right-click to modify)
Pole Footing
Sample Pole Footing Embedded in Soil
Section Prop Calc
User Defined Built-Up General Section
2D Frame
Frame with pinned bases
Diaphragm Torsic
Sample Rigid Diaphragm Torsion Analysis
Foundations
(right-click to modify)

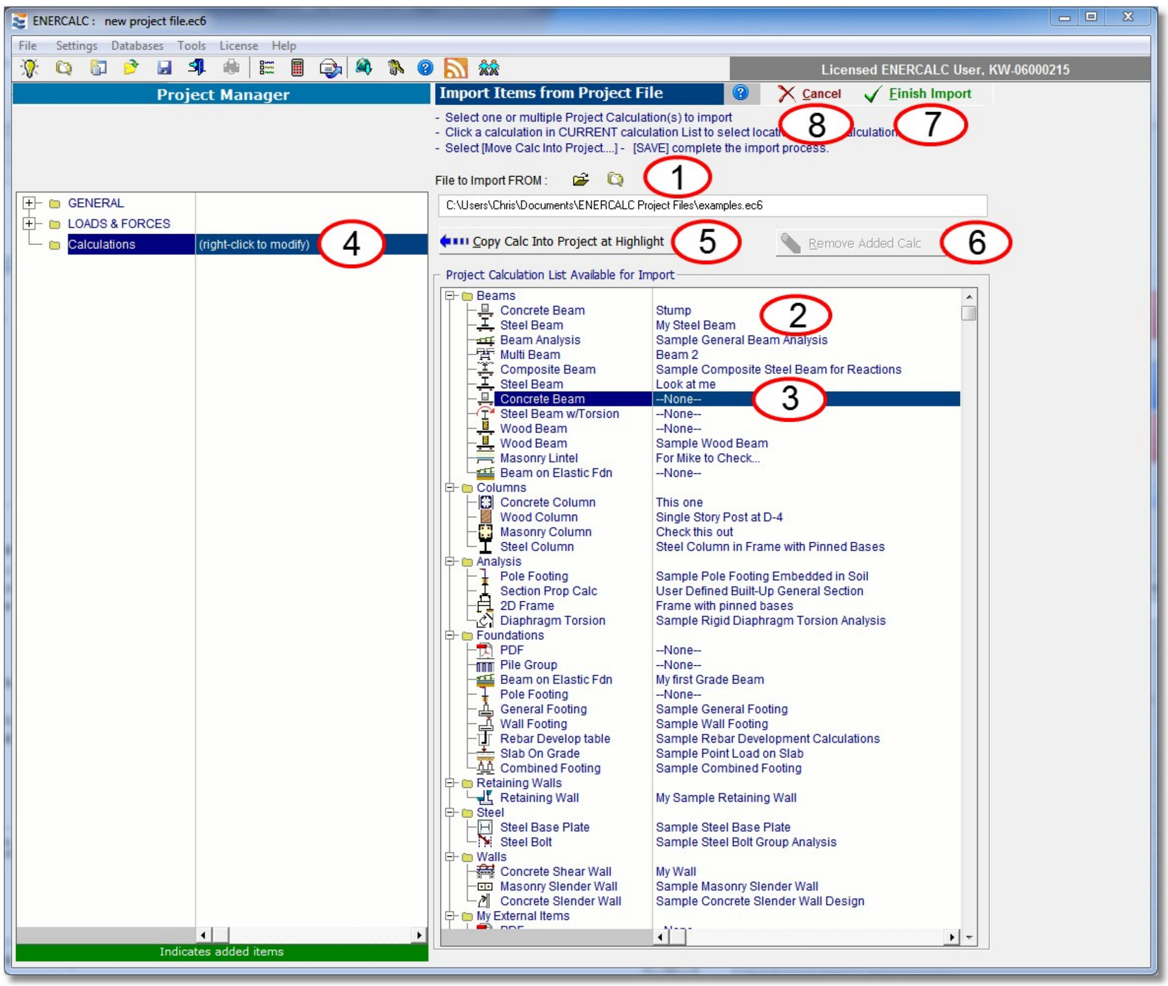
>  >


Project Manager
Project Print
Print Item
Delete
Add
Copy
Edit
Division
Material
Type
Import
GENERAL
LOADS & FORCES
Beams
(right-click to modify)
Columns
(right-click to modify)
Concrete Column
This one
Wood Column
Single Story Post at D-4
Masonry Column
Check this out
Analysis
(right-click to modify)
Steel Column
Steel Column in Frame with Pinned Bases
Pole Footing
Sample Pole Footing Embedded in Soil
Section Prop Calc
User Defined Built-Up General Section
2D Frame
Frame with pinned bases
Diaphragm Torsic
Sample Rigid Diaphragm Torsion Analysis
Foundations
(right-click to modify)

8.3.5 Importing Calculations

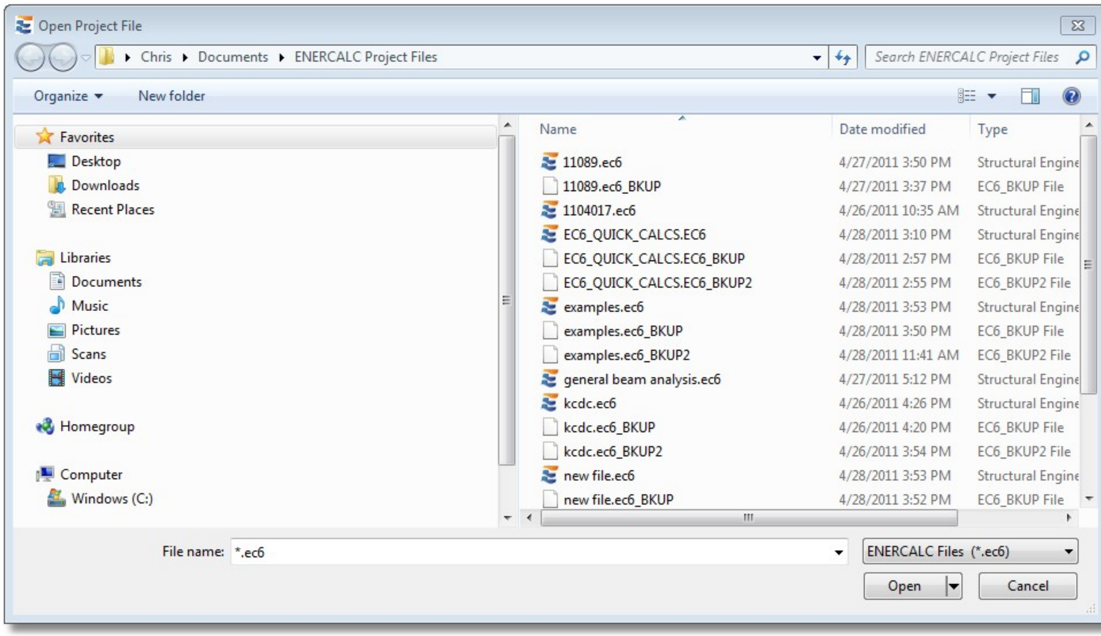
Clicking the [Import] button will change the Project Manager screen to appear as shown in the following image. Click here for a video: [Importing Calculations]

Please see the descriptions following the numbered keynotes. The order of the notes follows the order of usage.

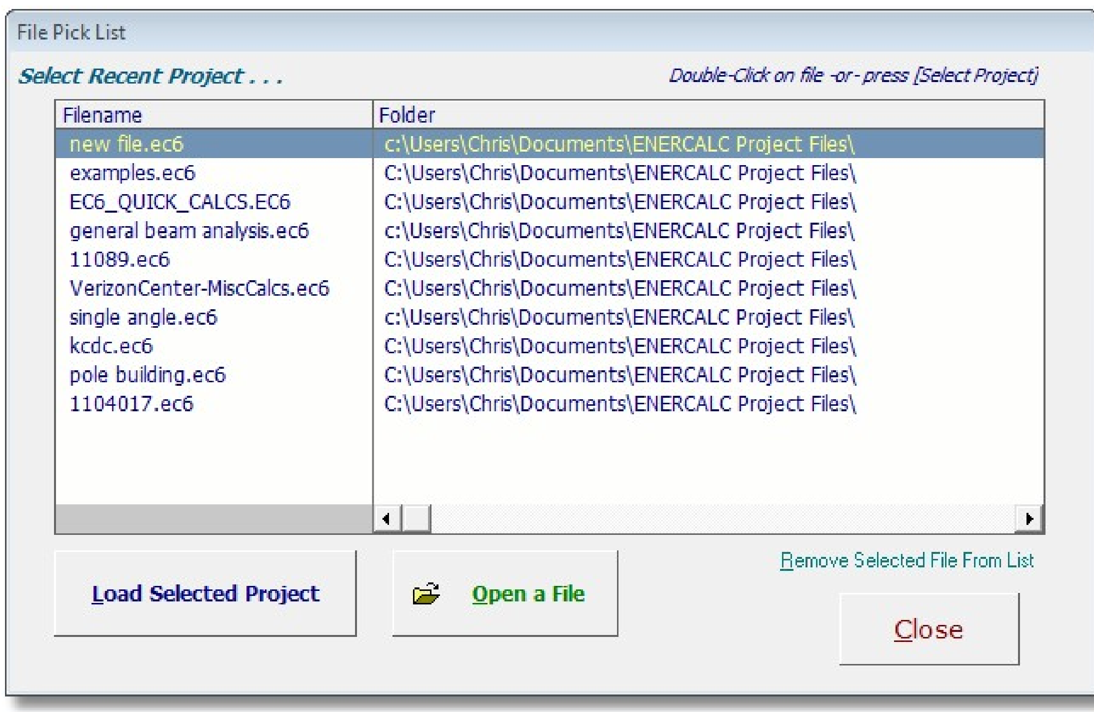


(1) Specify the source ENERCALC Project File from which you will import calculations into the current (destination) Project File.

Clicking the [**Select Source File**] icon will display a standard Windows File Open dialog:



Clicking the [**Use File Pick List**] icon will open a window that lists all the ENERCALC Project Files contained in the last few folders you have accessed:



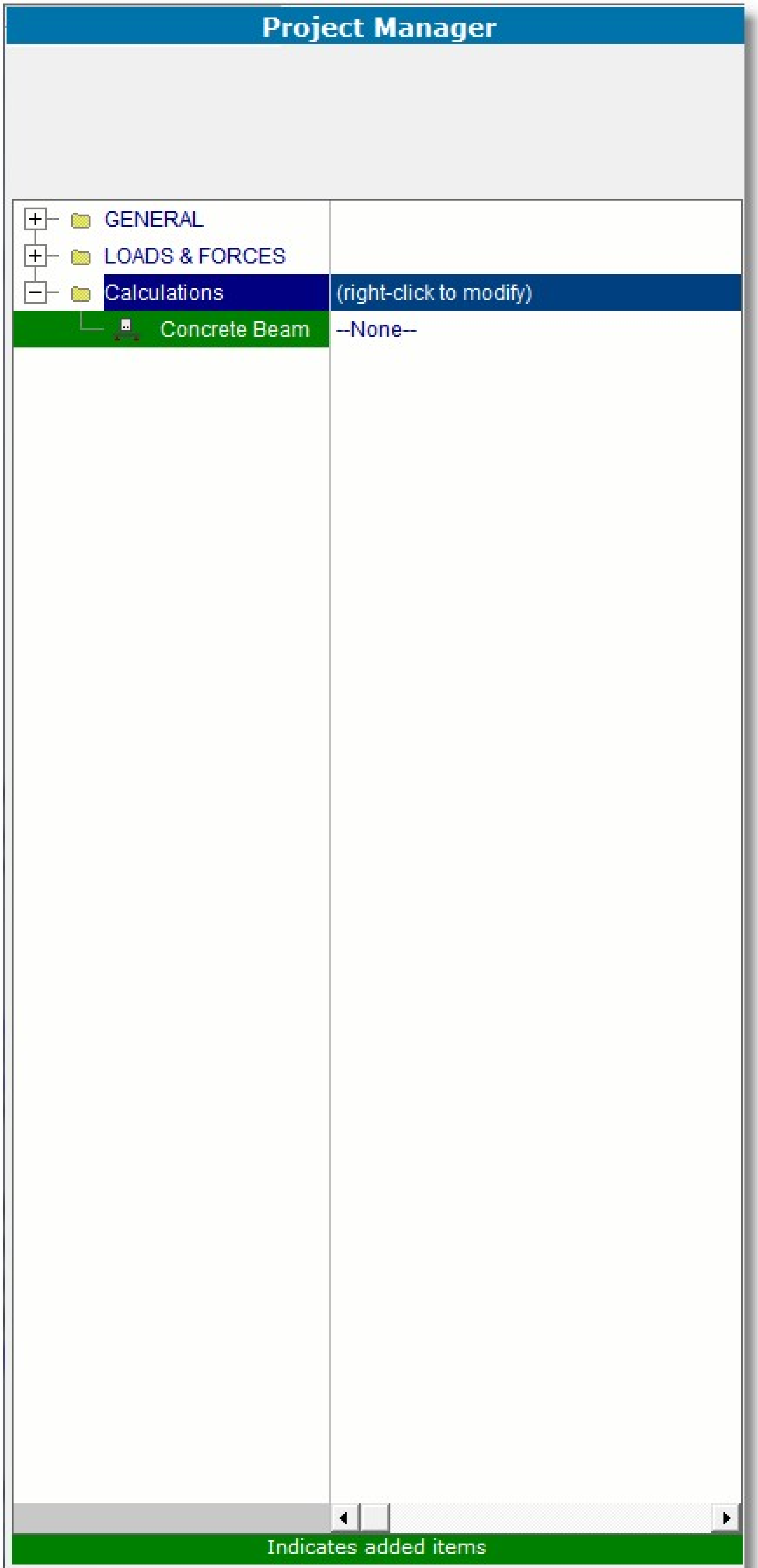
(2) After identifying the source Project File, area **(2)** will show the calculations in that source file in a tree-structure list.

(3) Click on the desired calculation in the source file to select it. It will become highlighted.

(4) Click in the current (destination) Project File to highlight a line where the calculation should be imported. (The only restriction is that calculations cannot be imported into the GENERAL Division or the LOADS & FORCES Division, so be sure to select a location other than those two Divisions or any of their contents.)

(5) Click the [**Copy Calc Into Project at Highlight**] button.

Notice in the image below the [Concrete Beam] calculation is now highlighted in green in the current (destination) file to indicate the location where it will be inserted.


Project Manager
GENERAL
LOADS & FORCES
Calculations
(right-click to modify)
Concrete Beam
--None--
Indicates added items

Note: If you made a mistake and want to cancel the selection, just highlight the calculation in green on the left and click the [**Remove Added Calc**] button **(6)**.

(7) To finish the import process click the 

button.

(8) To completely cancel the import click the 

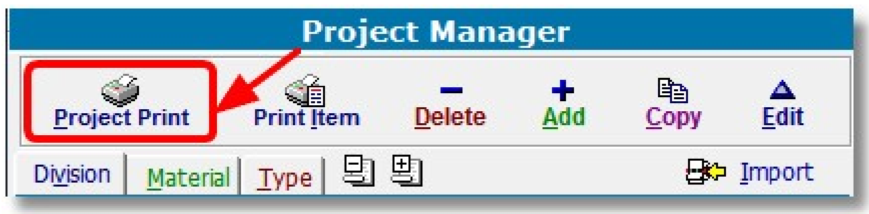
button.

8.4 Project Printing

The **Project Printing** system allows you to print a complete project full of reports in one simple process. Click here for a video: Project Printing

You can review all the reports in your project and then select which reports to print.

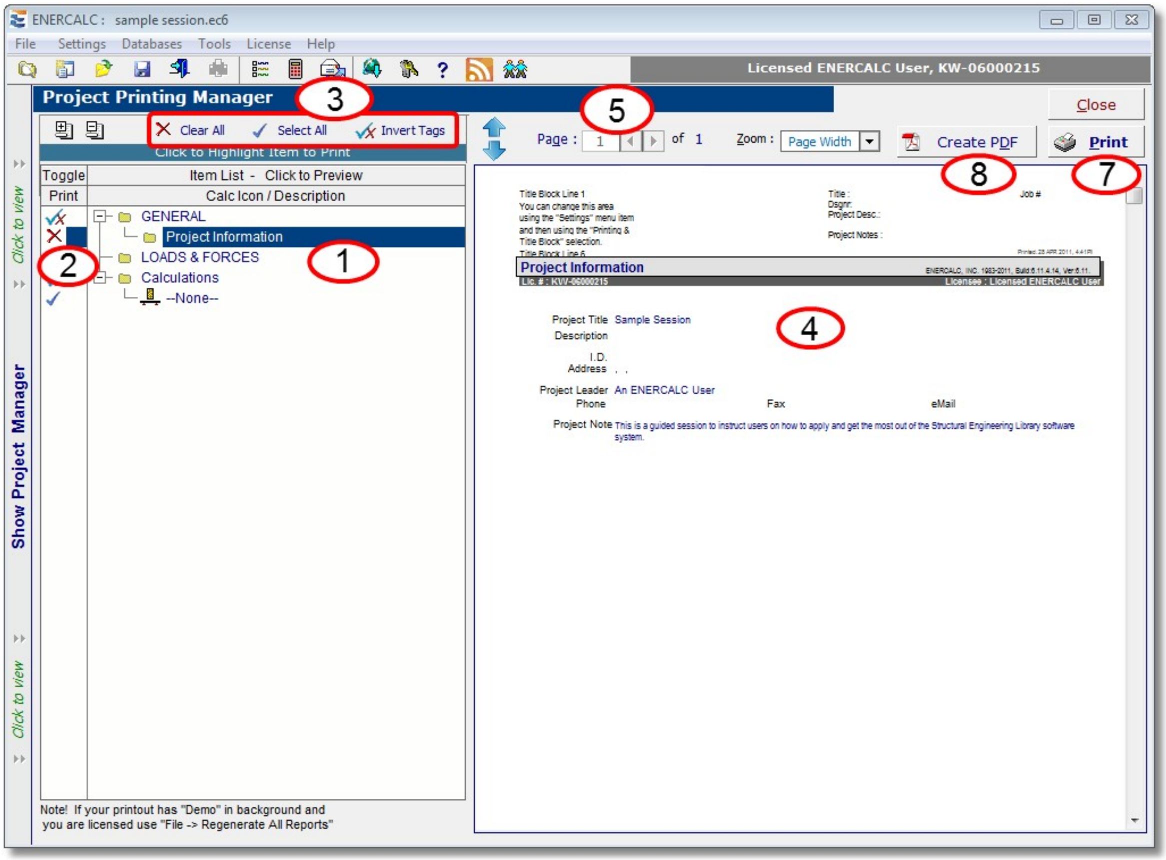
To open the Project Printing system, click the button shown bubbled in the screen capture below:



The screen will change to display the Project Printing Manager window.

On the left will be listed all of the calculations and external items in the Project File. The items in the GENERAL Division are placed at the top and the items in the other Divisions are placed on the bottom. When you highlight an item the full report appears on the right side of the screen.

See the numbered keynotes on the image below as we describe how to use the various controls on the screen. The information is given in order of typical usage.

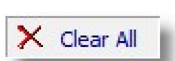


(1) List of project items for which reports are available : This area is similar to the item list in the Project Manager. All available report items are listed here. Simply click an item and the report for that item will be displayed in area **(4)**. Please REMEMBER that the reports shown were created when you clicked [**Save Only**] or [**Save & Exit**] in the specific calculation.

Note: If your software is licensed and activated, and if you see Evaluation Version watermarks on any reports, use the [**Regenerate**] button **(6)** to have all the reports regenerated WITHOUT the watermark.

(2) Print / No-print check mark: This column contains a check mark  or an X  to denote whether that particular report item is scheduled for printing.

(3) Clear All / Select All / Invert Selection buttons: These three buttons perform bulk changes to the print/no-print status of all items in the project.



Clears the print tags on all reports, so none will be printed. A  is displayed to the left of all reports after clicking this button.

Selects the print tags on all reports, so all will be printed. A  is displayed to the left of all reports after clicking this button.

Inverts the print/no-print tags on all reports.

(4) Report Preview area: The complete preview of the report is shown in this area for the report item highlighted in the list.

(5) Report page selection: When the highlighted item is a multi-page report, this box selects the page to view.

(6) (Item removed)

(7) Print selected reports to printer: Displays the Windows standard Print dialog box and begins printing all selected project items.

(8) Create single PDF file for selected reports: Displays a Windows Open File dialog box to request a filename for a PDF file and then creates a PDF of all selected project items.

Click [**Close**] to dismiss the Project Printing Manager.

Part IX

9 Sample Session

This section will guide you through a session of preparing and printing a calculation. We will use the Wood Beam calculation because it contains all the features available in Structural Engineering Library.

9.1 Starting the Program

First....let's start the program

To start **Structural Engineering Library**, navigate to the ENERCALC program group from the Start menu.

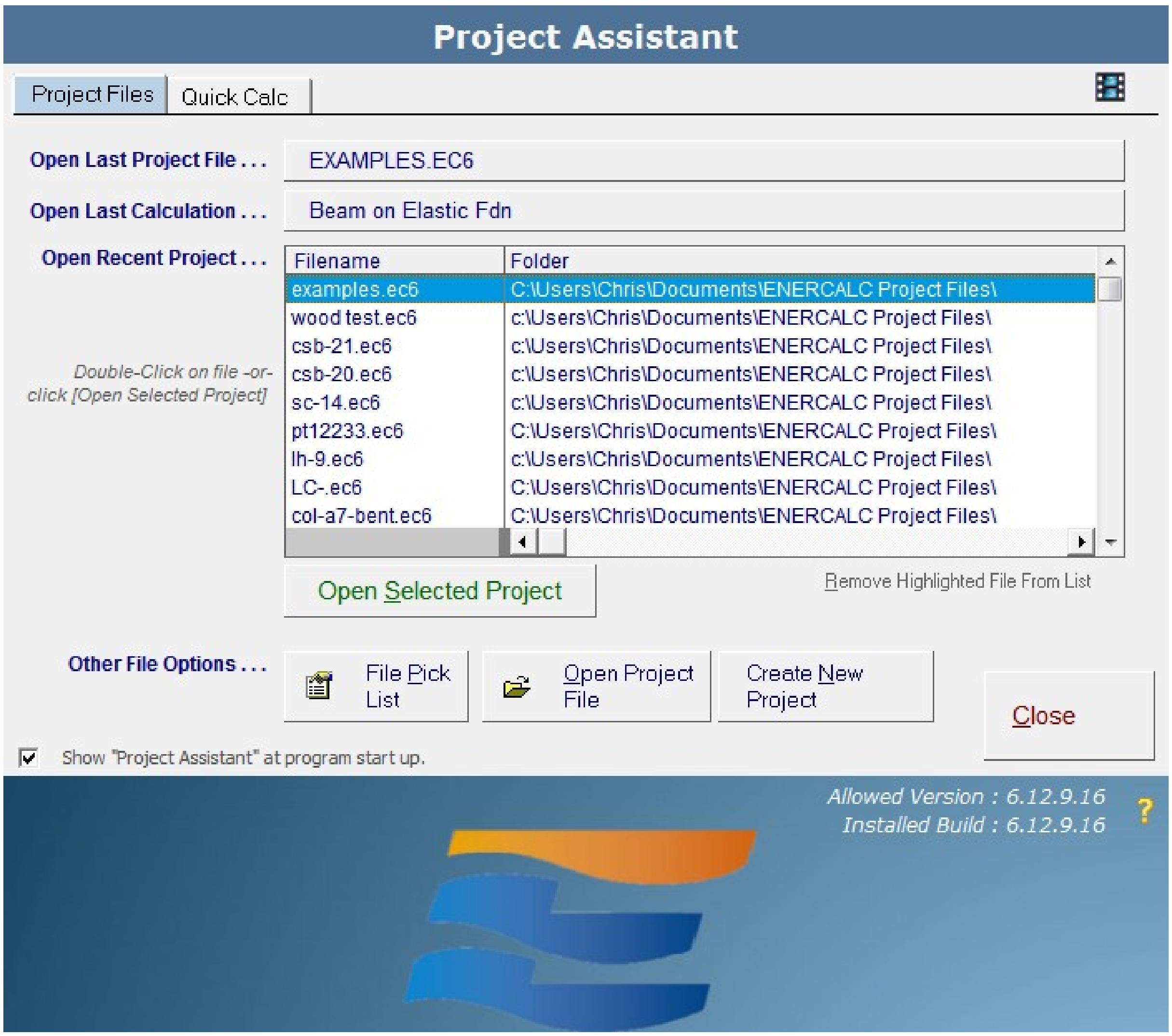
Click the Windows [**Start**] button and move upward to highlight the Programs (or All Programs) item. Locate the ENERCALC program group and highlight it. You will see several selections. Move the cursor to highlight Structural Library and release the mouse button.

Structural Engineering Library will now open, and you will be viewing the Project Assistant as described below.

9.2 Project Assistant

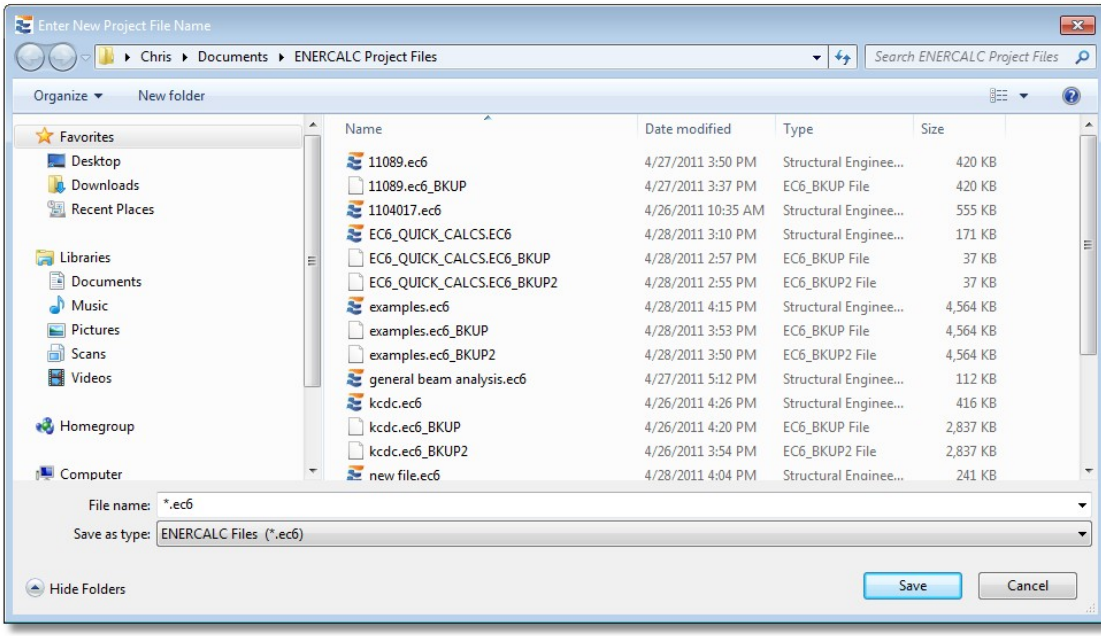
The Project Assistant

This dialog gives you the ability to instantly choose how you wish to begin your work session. All of these options can also be easily chosen from the main menu, and the Project Assistant can also be reopened from the main menu after the program has already started.



For this Sample Session we want to create a new Project File, so the next step will be to click the [**Create Project File**] button.

The next window to be displayed (see below) will allow you to define the file name and select the drive & directory where the file should be placed.



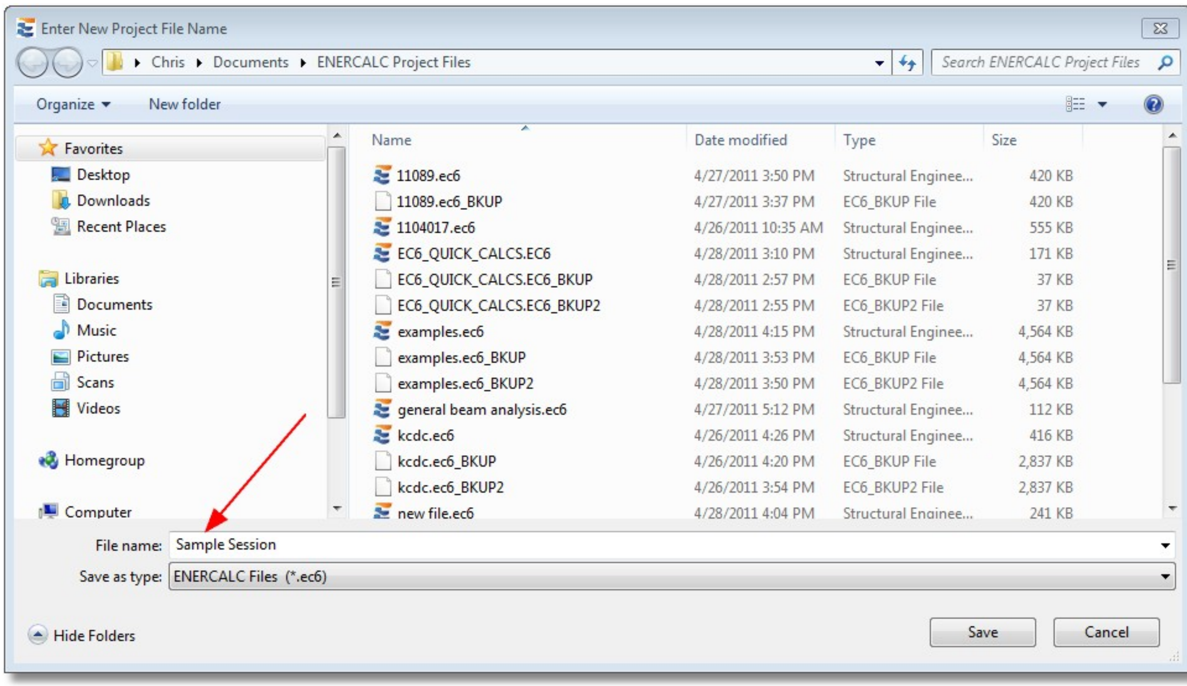
We will use the filename Sample Session and place it in a folder named ENERCALC Project Files that we created specifically to hold Project Files.

Note: You could place the file in other directories, but keep in mind that Microsoft is now recommending specific locations for saving user-generated files with the newer operating systems, and the default ENERCALC data folder conforms to these new best-practice recommendations. The important thing to recognize is that an ENERCALC Project File can hold thousands of calculations, so it is likely that there will only need to be a single ENERCALC Project File for each design project in the office.

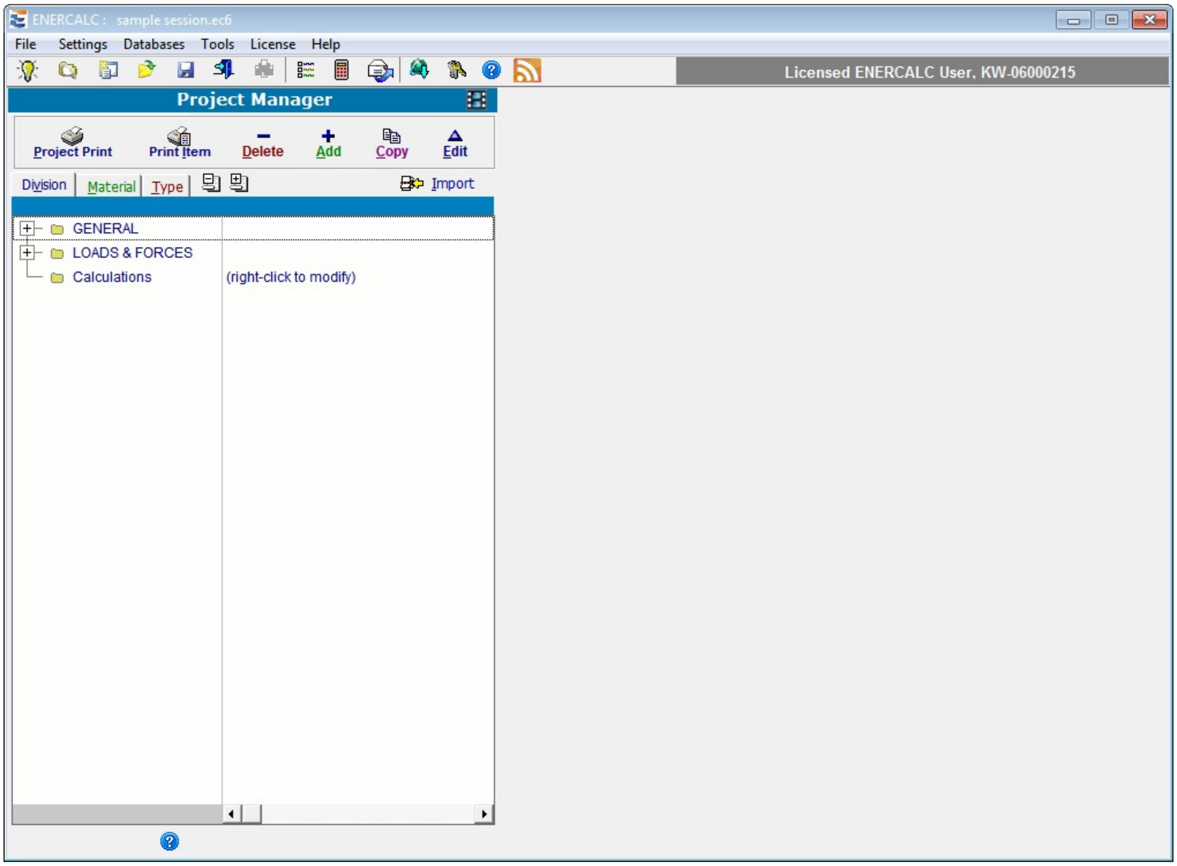
9.3 Creating a Project File

Creating a Project File

Here's the Create NEW Project File dialog box with the new filename entered. NO FILE EXTENSION is necessary....the program will append it. When the dialog appears as it does below, just click [**Save**] and the file will be created.



When a new Project File is created, the new Project File is immediately opened and displayed in the Project Manager window. You can see in the screen image below that the new Project File looks rather plain and simple.



Notice that the ONLY items appearing in the calculation list are the words "GENERAL", "LOADS & FORCES" and "Calculations".

These items are called Divisions. You can create as many Divisions as you like. They serve as a way to organize your calculations into logical groups for your convenience .

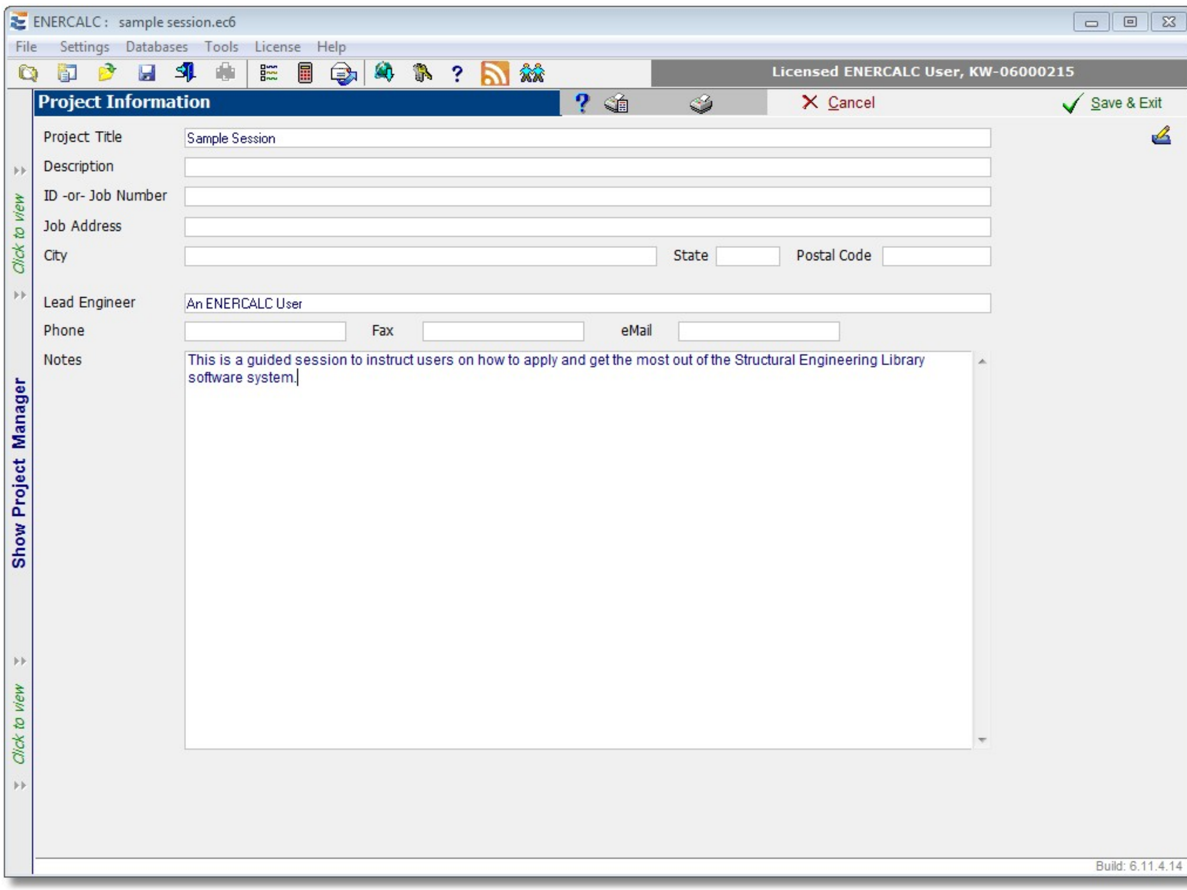
All newly created Project Files will contain these three Divisions by default. The "GENERAL" Division and the "LOADS & FORCES" Division cannot be renamed or deleted, and they serve specific purposes, which we will see shortly. But the "Calculations" Division is just created for convenience. It can be renamed, deleted, copied, etc.

Before we add any calculations to this Project File, the next step will be to enter some general information about the project itself.

9.4 Entering Project Information

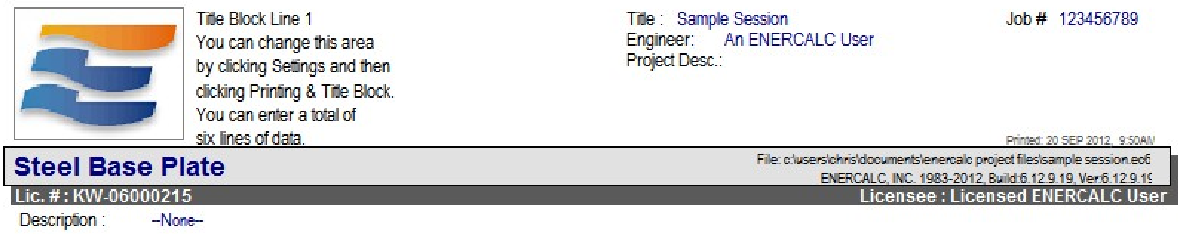
Entering Project Information

Click the [**+**] icon to the left of the **GENERAL** Division in the Project Manager to expand its contents, and then double-click the top item named **Project Information**.



For this sample session you can fill in anything you like, just to see how the Project Information fields can be used. Then click the [**Save & Exit**] button and click [**OK**] in the Reminder dialog. The purpose of the Reminder dialog is to alert us to the fact that some data just changed in the title block. Under normal conditions, **Structural Engineering Library** generates the report for a single calculation at a time, and it does that when the Save command is issued while a calculation is open. This makes for a very efficient use of time by generating reports only on an as-needed basis. However, because we just made a change to title block data, it is likely that we want all reports regenerated and brought up to date at this time. This Reminder dialog guides us to the command in the main menu that manually forces all reports to be immediately regenerated.

NOTE: This Project Information is printed in the upper-right corner of your printouts. All **Structural Engineering Library** printouts have a Title Block area that contains your company's information in the upper-left corner and the Project Information in the upper-right corner. The above project information (and the title block information described in the next section) will look like this on the printouts:


Title Block Line 1
You can change this area
by clicking Settings and then
clicking Printing & Title Block.
You can enter a total of
six lines of data.

Title : Sample Session
Engineer: An ENERCALC User
Project Desc.:

Job # 123456789

Printed: 20 SEP 2012, 9:50AM

Steel Base Plate

File: c:\users\chris\documents\enercalc project files\sample session.ec6
ENERCALC, INC. 1983-2012, Build:6.12.9.19, Ver:6.12.9.19

Lic. # : KW-06000215

Licensee : Licensed ENERCALC User

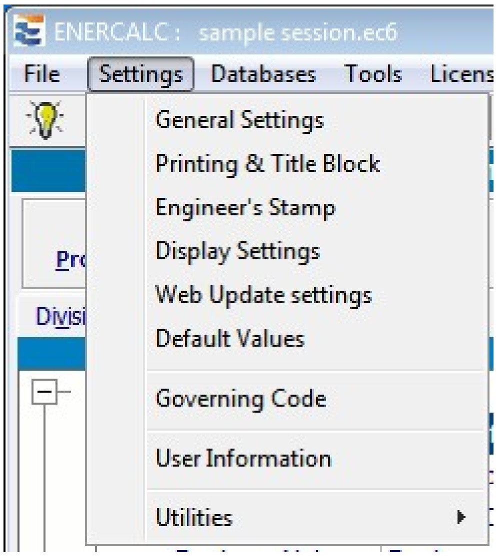
Description : --None--

9.5 Setting up your Title Block

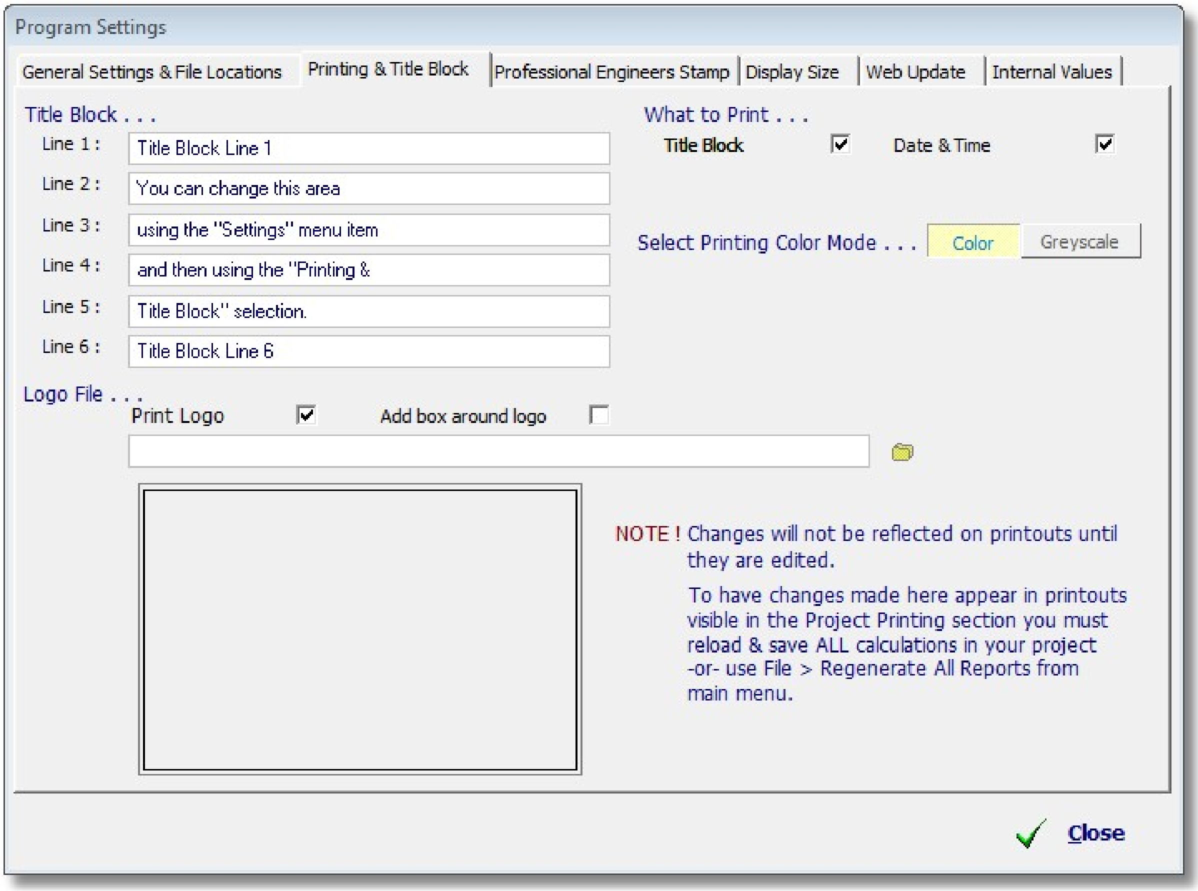
Setting up your Title Block

Before continuing on to add some calculations to the project, we'll demonstrate where to enter company information to be included on ALL calculations produced within **Structural Engineering Library**.

Click **Settings > Printing & Title Block** from the main menu.



The **Printing & Title Block** tab of the Program Settings dialog will be displayed. Here you can use six lines to customize the printout with your own company information. This information will be printed in the upper-left corner of ALL printouts (see sample previously).



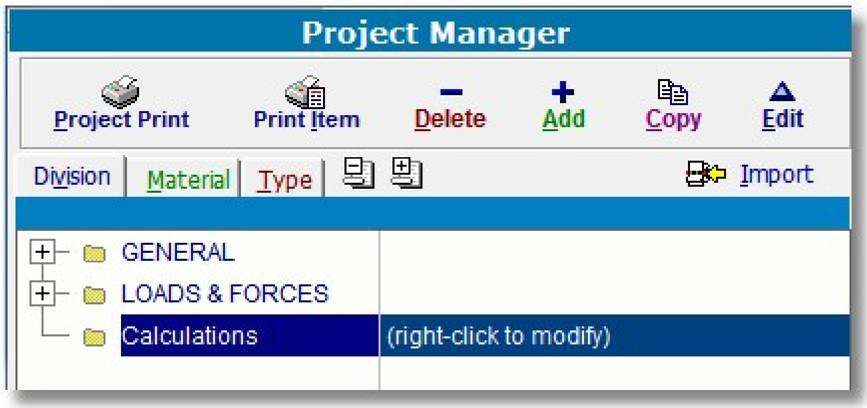
The Logo File area of this dialog also offers the option to identify a graphic file of your company's logo and specify that it be printed as part of the Title Block. Clicking the  button lets you use a file selection dialog to select a Windows Bitmap (BMP), GIF, JPG, or WMF file.

At this time, please enter some information in all six lines so you can see it printed later in this sample session. Click the [**Close**] button when finished, and then click the [**OK**] button in the Reminder dialog.

9.6 Adding a Calculation

Adding a Calculation

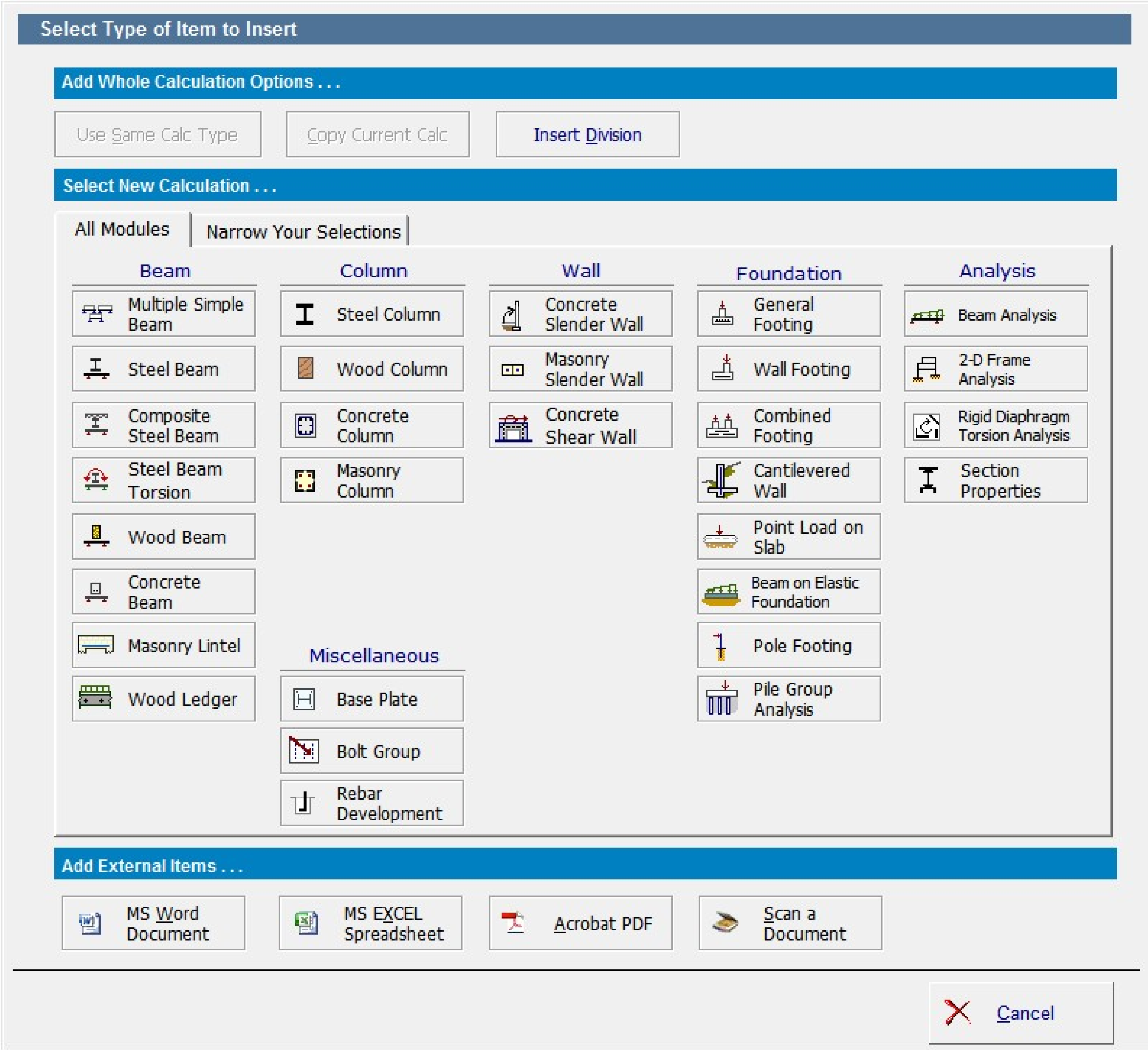
To add a calculation in the Project Manager, first highlight the location where you would like the new calculation to be inserted. At present we have no calculations in the Division named "Calculations", so we will click on "Calculations" to highlight it as shown below:



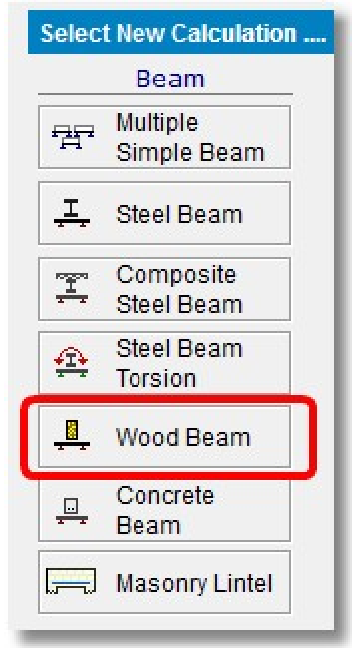
Next, click the [**Add**] 

button.

The next dialog will allow you to select exactly which calculation type you wish to add:



For this sample session we will use the **Wood Beam** module, so click the [**Wood Beam**] button as shown below.

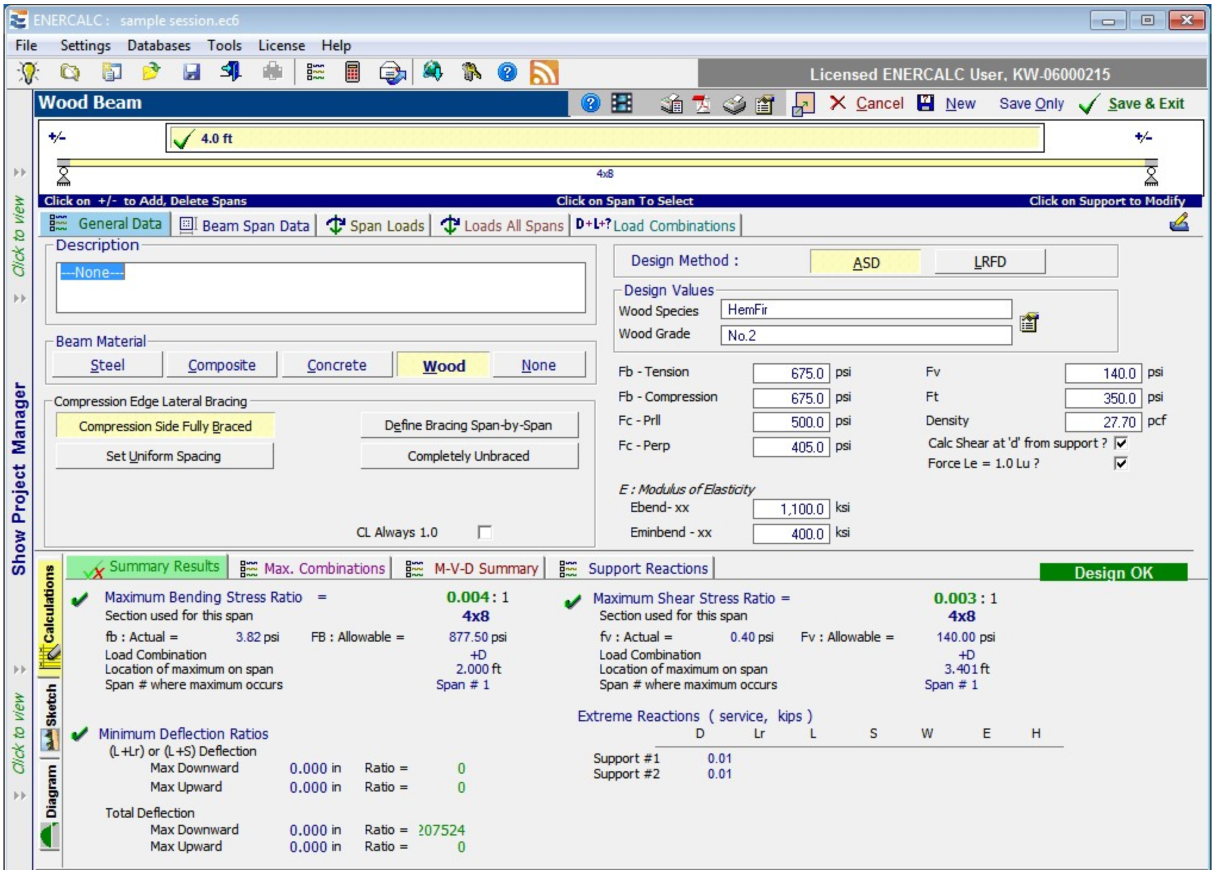


A Wood Beam calculation will be added in the Project Manager, and it will automatically be opened for editing. The screen will reconfigure itself to display the graphical user interface that is specific to the Wood Beam module.

9.7 Viewing the Calculation Screen

Viewing the Calculation Screen

The screen should now appear as shown below. Please observe the following areas of the screen....they are typical of ALL calculations in **Structural Engineering Library**.



The data entry tabs on the top half of the screen are where you enter your input values.

The results and graphics tabs on the bottom half of the screen are where you review the calculated values, sketches, and graphs that result from calculations based on your input values.

This screen layout is consistent throughout the majority of **Structural Engineering Library** calculation modules. This enables you to display the tab of interest on the bottom and see how changes to input data on the top affect it.

9.8 Changing the Default Values/Settings

Changing the Default Values/Settings

When you first add any calculation to a Project File, that calculation will come in with default values/settings for all user input items. If you find that you are almost always having to change certain items, it may make sense for you to revise those defaults to better suit your work. Revising the default values for any module is easy. Just follow these steps:

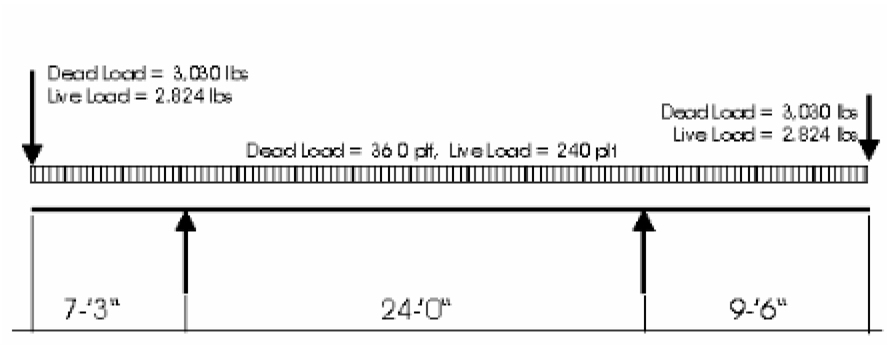
1. Add a new calculation of the module of interest to a Project File. It will come in with the existing default values/settings.
2. Revise the values/settings to suit.
3. Click Settings > Default Values > Save.

This will save all of your current settings/values as the new defaults for that particular module. From that point onward, any time you add a new calculation of that module type, it will automatically be populated with the default values/settings that you just established.

9.9 Entering Data

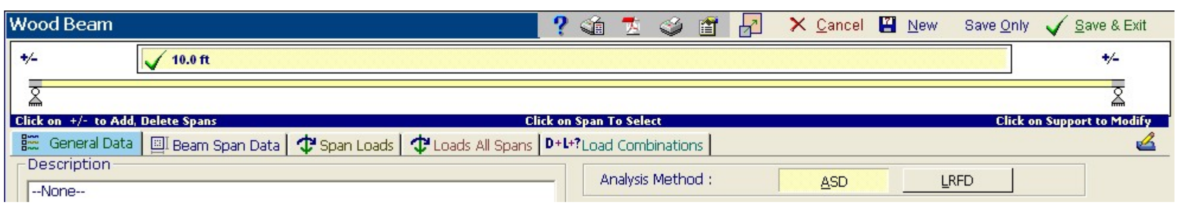
Entering Data

Below is a diagram for a wood roof beam we wish to design.

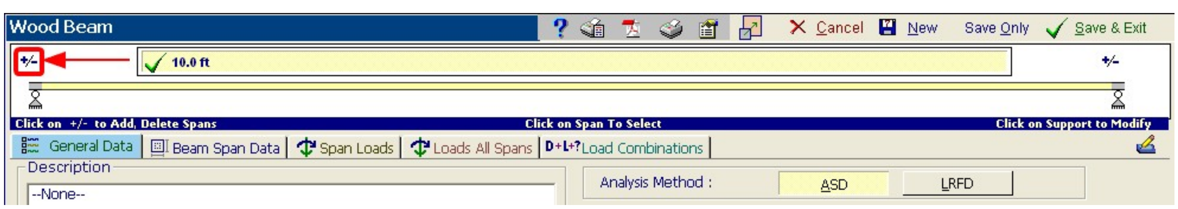


The following steps will guide you through the process of setting the span lengths and entering the loads as shown. (In a subsequent step we will select the member section size and reference design values from the internal databases.)

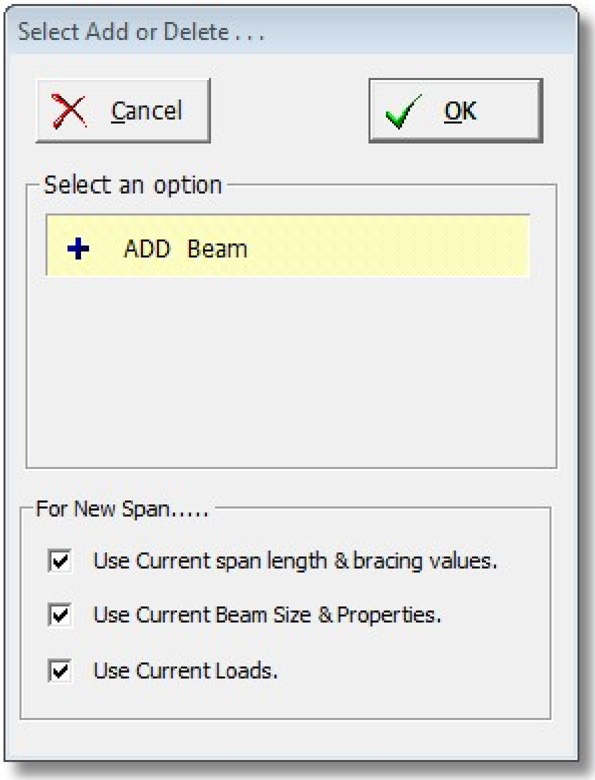
The top portion of the screen should currently look like this:



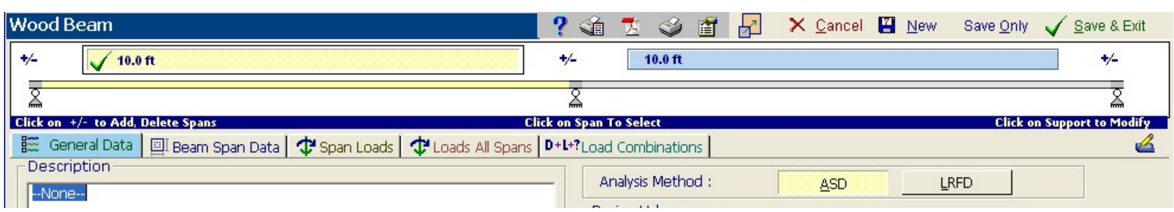
The first thing we will do is add the left cantilever. To do this, click the [**+/-**] icon at the left end of the beam as shown bubbled below:



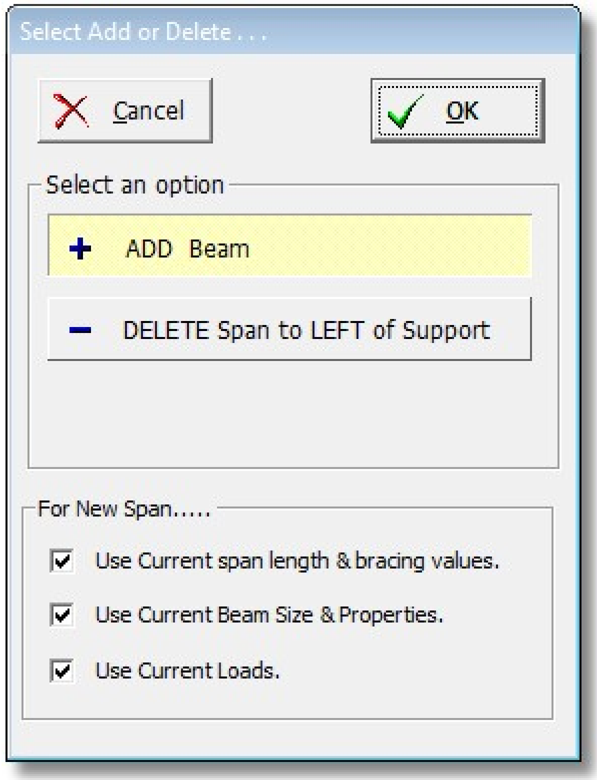
This opens the dialog named Select Add or Delete. It is currently set up to allow us to add a new beam span, which is exactly what we want to do. There are some additional settings offered at the bottom of the dialog in the form of checkboxes. We don't need to worry about these at this point, but it's good to know that they are available, because they can offer some time-saving conveniences in certain modeling situations. See the image of the Select Add or Delete dialog shown below:



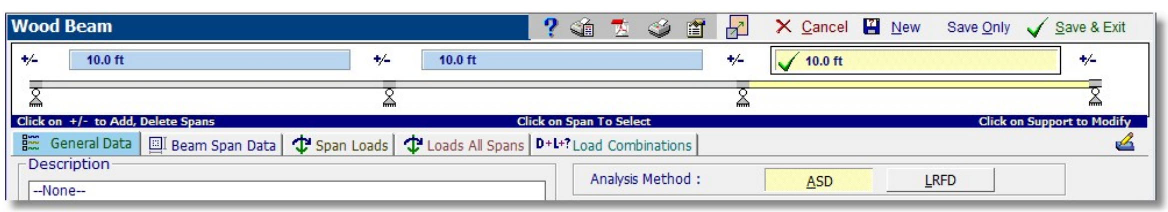
For our purposes, all we need to do is click the [**OK**] button. This inserts a new beam span to the left of the original beam as shown below:



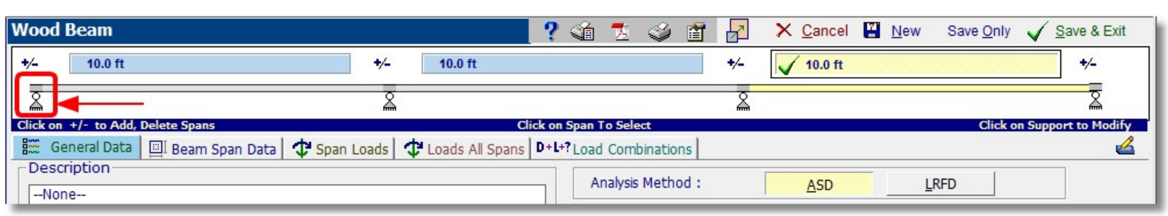
At this point, don't be concerned about the fact that the new span is not a cantilever and that it is not the intended length. We'll correct both of those items in just a moment. For now, let's repeat a similar process to add a new beam span on the right. First click the [**+/-**] icon that appears above the right-most support. The following dialog appears:



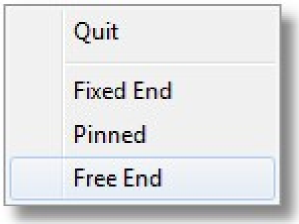
This is very similar to the Select Add or Delete dialog we saw before, except that it now has an extra option to delete the span that is to the left of the support. Although we don't want to do this, it's good to know that this is the way that you would delete a span if necessary. What we want to do is add a new beam span, and that is the option that is currently selected (highlighted in yellow), so we can just click the [**OK**] button. The screen should now display a 3-span continuous beam as shown below:



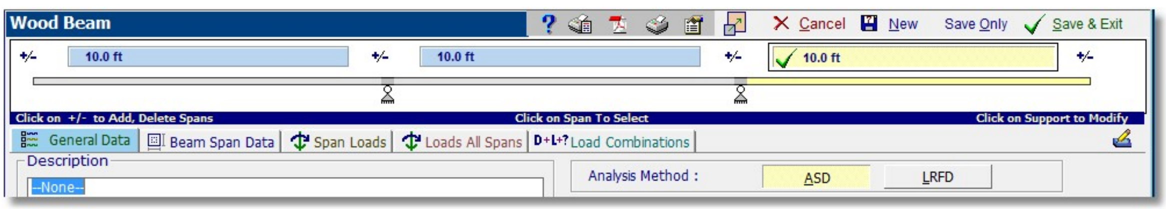
Now that we have three spans to work with, let's modify the support conditions to create the cantilevered spans at the left and the right. To do this, click on the left-most support icon shown bubbled below:



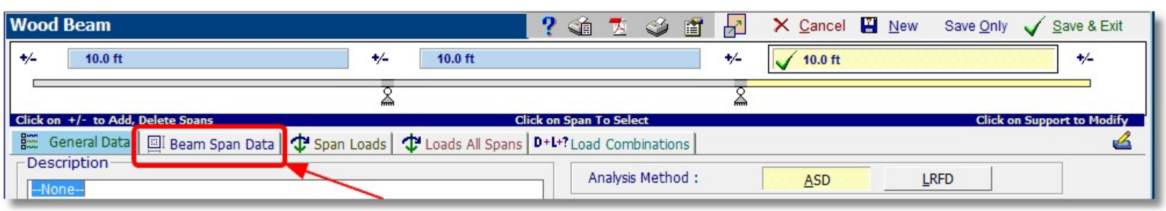
This will open the following pop-up menu that offers many options for defining the support condition at that location:



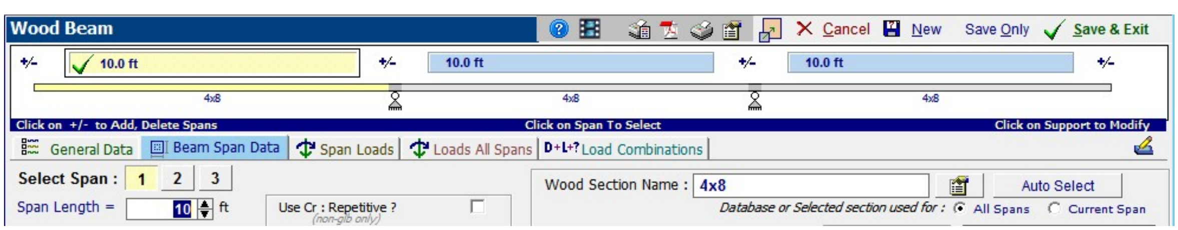
Click the **Free End** option to remove all restraint from the left end, creating a cantilever. Repeat the same process at the right-most support, but leave the remaining two supports as they are currently configured. (They represent pinned conditions that prevent translation in all directions but offer no moment restraint.) The result will be the double-cantilevered beam as shown below:



Now that the support conditions have been established correctly, we can wrap up this exercise by modifying the span lengths to match the problem definition. Start by clicking the Beam Span Data tab shown bubbled below:



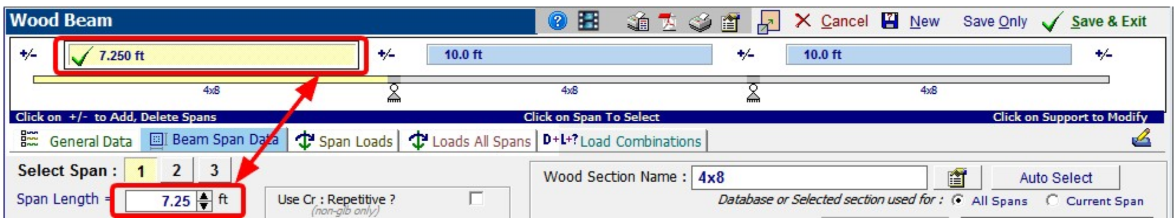
The screen layout will change to display the Beam Span Data tab as shown below:



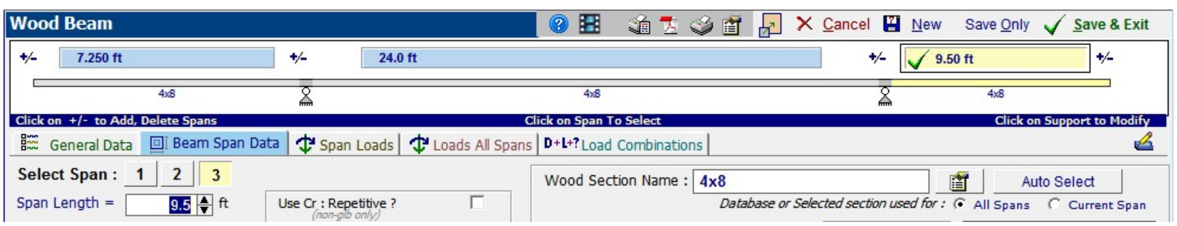
Now, we just need to adjust the span lengths for each of the three spans.

First, note that the graphical depiction of the beam helps us to make the association between the three individual spans and the span numbers. Click any of the three span number buttons in the Select Span area and notice that a corresponding span becomes highlighted in the large graphic at the top of the view. Using this method, we can confirm that the spans are logically ordered from number one on the left to number three on the right. (Note that you can also click the highlighted rectangles shown above each beam span, and they act in exactly the same way that the numbered buttons do.)

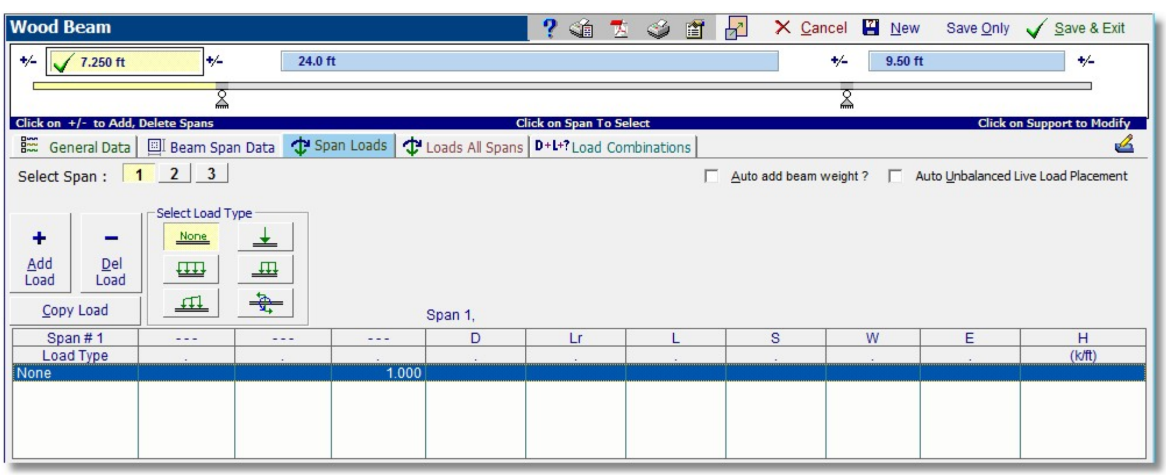
So to finish things off, click the button for span [**1**] and verify that the left span becomes highlighted. According to the problem statement, this span is supposed to be 7'-3", so you can highlight the value that is presently displayed in the Span Length field, manually edit it to say 7.25, and then press the [**Tab**] key to enter that value. The display will update to show the proper span length for the left beam span as shown below:



Repeat the same process to set the Span Length for the middle span to 24'-0" and to set the Span Length for the right span to 9'-6", at which time the display should appear as shown below:



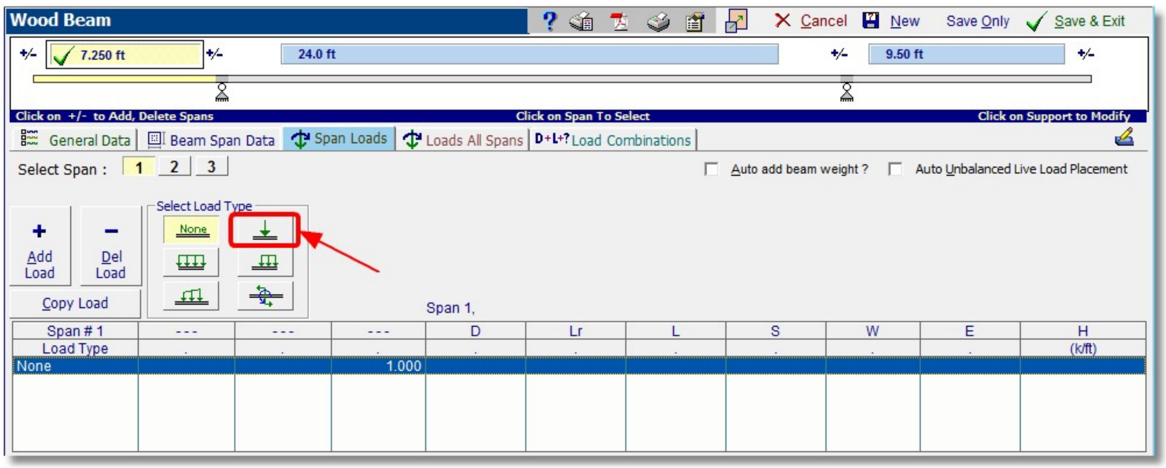
The next step will be to enter the loads on the beam. Let's begin by entering the concentrated load at the free end of the left beam span. Click the Span Loads tab to display the following screen:



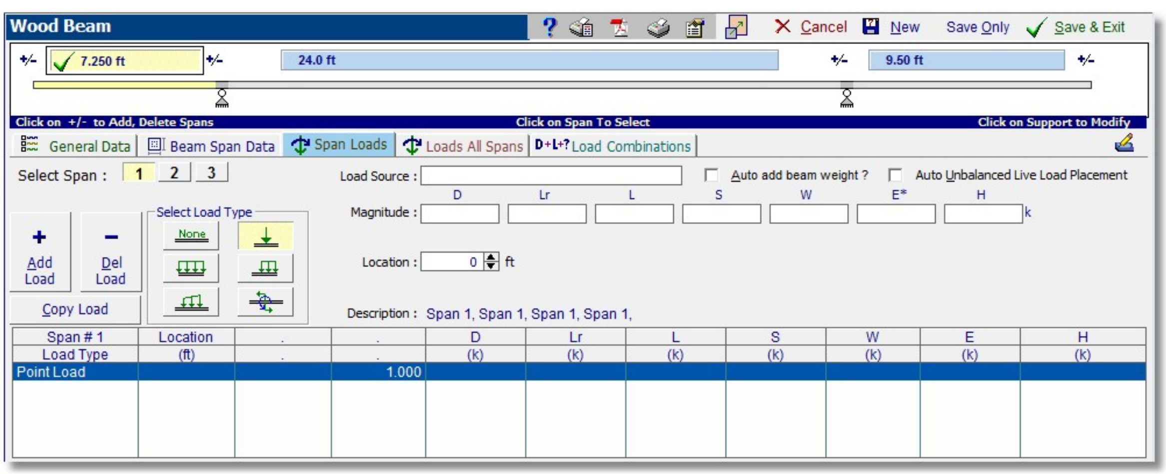
This screen contains tools to apply loads to a selected span at a time. (Shortly, we will see a convenience tool that will allow us to apply loads to ALL spans of a beam at one time.)

Since we want to start by entering the concentrated load at the free end of the left beam span, click the [**1**] button in the Select Span area to put the focus on Span 1.

The blue highlighted band in the table now represents the first load on Span 1. But it currently indicates Load Type = None, because we have not defined the load yet. Click the button indicated in the screen capture below to identify the load as being a concentrated load:



This will reconfigure the screen with the appropriate input fields to fully define the magnitude, direction, and location of the concentrated load as shown below:



The Load Source input field provides a location where you can add some descriptive text for your own use to identify this particular load item. The use of this field is optional, so for this sample session we will leave it blank.

Select the checkbox for the Auto add beam weight option. This will trigger the program to determine the self weight of the beam that we eventually choose, and apply it as dead load when the analysis is performed.

Select the checkbox for the Auto Unbalanced Live load Placement option. This will initiate the generation of additional loading conditions to ensure that all possible permutations of live load placement are properly examined. Note that selecting this option can lead to longer analysis times because of the number of different conditions that must be analyzed. However, with the double-cantilevered beam configuration, it would be important for us to have the module examine all of these potential loading permutations.

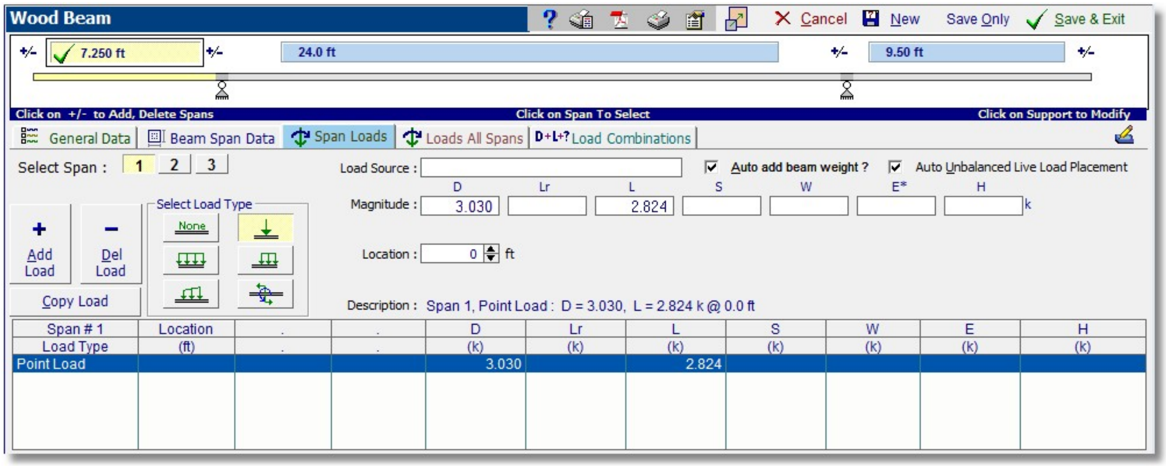
When defining the magnitude of the concentrated load, note that the module is set up to receive this kind of load in units of kips, and notice that it has input fields for the common load cases, such as Dead, Live, Snow, etc.

Click in the input field for Dead, enter a magnitude of 3.03, and then press the [**Tab**] key. This enters the Dead load magnitude and moves the cursor to the next input field.

Note: For convenience this module is configured such that loads with positive magnitudes are assumed to act downward, since the majority of the load items are typically gravity loads of some sort.

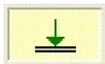
The cursor is currently in the Lr input field, which represents Roof Live load. We don't need to enter any magnitude for Roof Live, so press the [**Tab**] key once again to place the cursor on the Live load input field. Enter a magnitude of 2.824, and then press the [**Tab**] key.

The final step in specifying this concentrated load is to identify its position using the Location input field. Always specify the distance from the left end of this span to the position of the concentrated load. In this case, the correct value is zero, which also happens to be the default value for the Location field, so the Location field can be left as-is. The screen should now appear as shown below:



The next step will be to define the other concentrated load, which occurs at the free end of the cantilever on the right end of the beam, by following a very similar process.

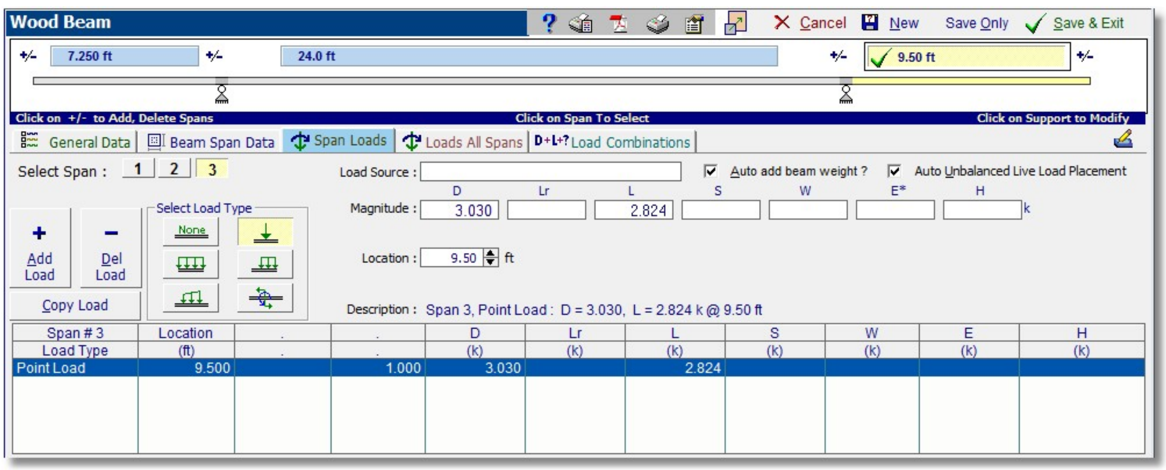
Click the [**3**] button in the Select Span area to put the focus on Span 3.

Click the  button to identify the load as being a concentrated load.

The magnitudes of this load are the same as those used at the opposite end of the beam, so click in the input field for Dead, enter a magnitude of 3.03, and then press the [**Tab**] key twice.

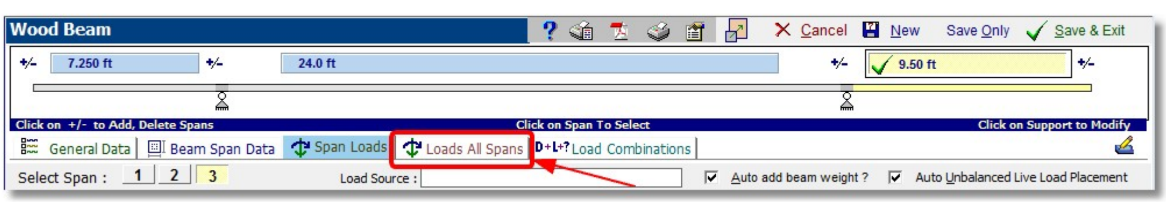
In the Live load input field enter a magnitude of 2.824, and then press the [**Tab**] key.

This concentrated load is to be positioned at the extreme right end of this span, so click in the Location input field, enter a distance of 9.5 (9'-6" from the support at the left end of Span 3), and then press the [**Tab**] key. The screen should now appear as shown below:

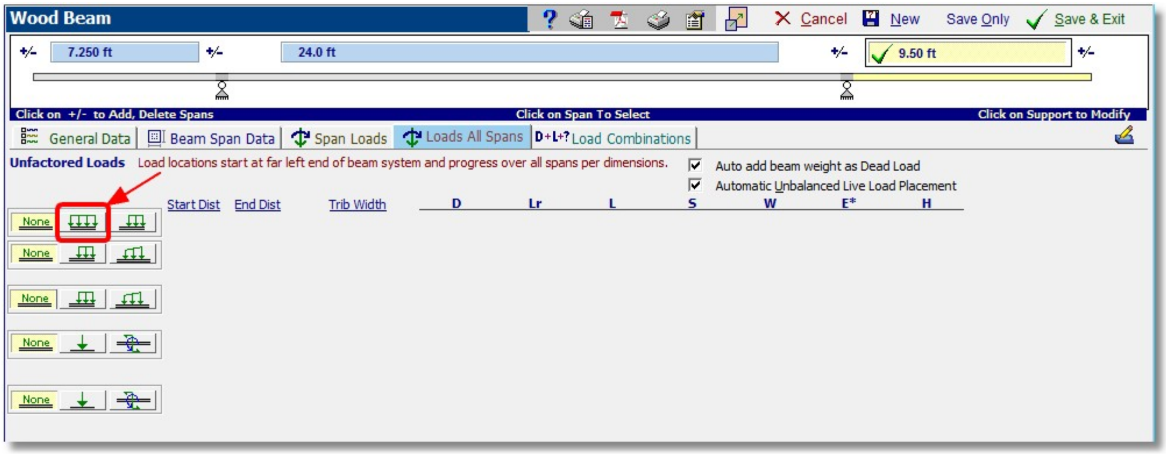


The next step in defining loads is to apply the uniformly distributed load across all three spans. This *could* be done by using the Uniformly Distributed Loading Type here on the Span Loads tab. But this would require some repetition, because the Span Loads tab is specifically set up to facilitate the application of loads to one span at a time.

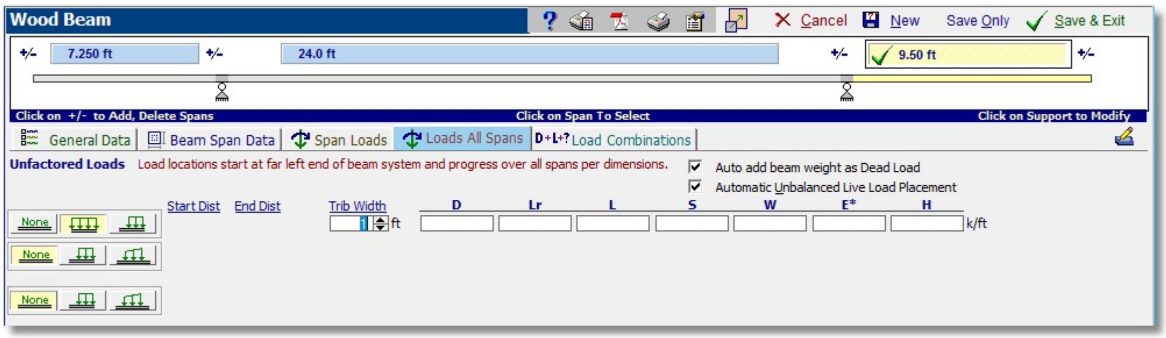
To apply this load in a more efficient manner, click the Loads All Spans tab shown bubbled below:



The screen will reconfigure itself to display different tools for applying various types of loads to all spans of the beam. The one that is most appropriate for our application is the one shown bubbled below:



Click that tool and note that the screen displays input fields for Tributary Width and load magnitudes for the various Load Cases as shown below:



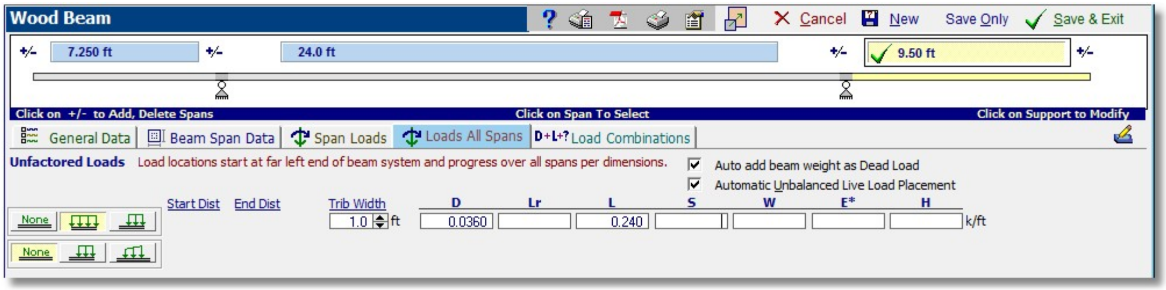
Note: When using this tool, you can either leave the Tributary Width input field at its default value of 1.0, and then enter the magnitudes for the various load cases in units of kips per foot, or you can specify a Tributary Width magnitude *other* than 1.0 foot, and then enter the magnitudes for the various load cases in units of kips per square foot, which will be multiplied by the Tributary Width to determine the magnitude of the effective line load in units of kips per foot.

For the purposes of our exercise, we will leave the Tributary Width input field at its default value of 1.0.

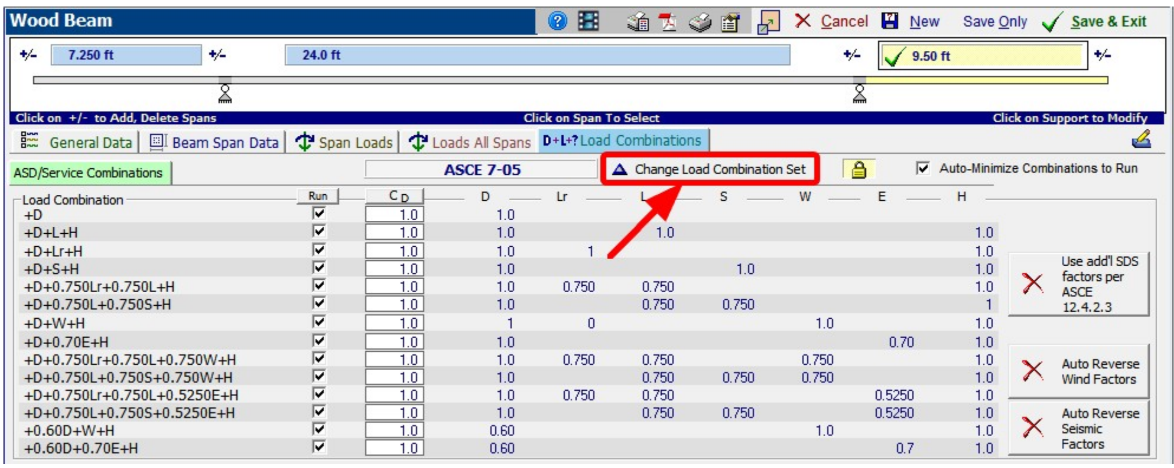
Click in the Dead load input field, enter a magnitude of 0.036, and press the [**Tab**] key twice.

In the Live load input field, enter a magnitude of 0.24, and press the [**Tab**] key once more.

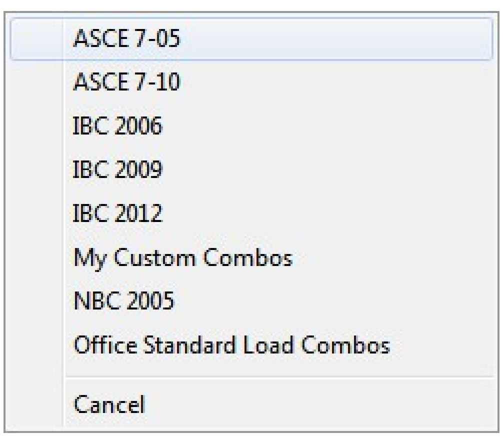
The last item to take care of is to observe the load combinations that will be used for design. Click the Load Combinations tab shown bubbled below:



This will display the Load Combinations screen. On this screen, the first step is to make sure that we are referencing the correct set of load combinations. To do this, click the "Change Load Combination Set" button shown below bubbled below:



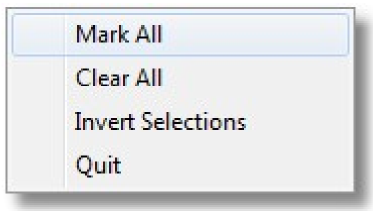
This will open a pop-up menu offering choices of load combination sets that exist on the current machine, similar to the one shown below:



Choose the option named IBC 2009. This will set up the calculation to use load combinations based on IBC 2009.

Next, click the  button, and confirm by clicking the [Yes] button. This is a very convenient option for wood designs, because it automatically sets the C_D value based on the shortest duration load case within each load combination. Note the various values of C_D that result.

The final step in this section is to ensure that all of the listed load combinations will be used in the analysis and design. To do this, click the   button. This will open a pop-up menu offering options for marking and clearing the Run status of all of the load combinations in the list, as shown below:



Simply click the [**Mark All**] option, and all of the load combinations in the list will be marked for use in the analysis and design.

9.10 Selecting Sections and Materials from Built-in Databases

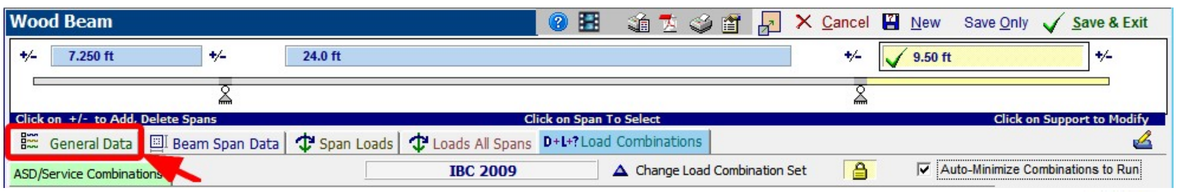
Selecting Sections and Materials from Built-in Databases

In the previous topic you experienced part of the advantage of using **Structural Engineering Library**...simplified data entry. The input screens naturally guide you through all of the necessary items, and you are simply entering data into a form.

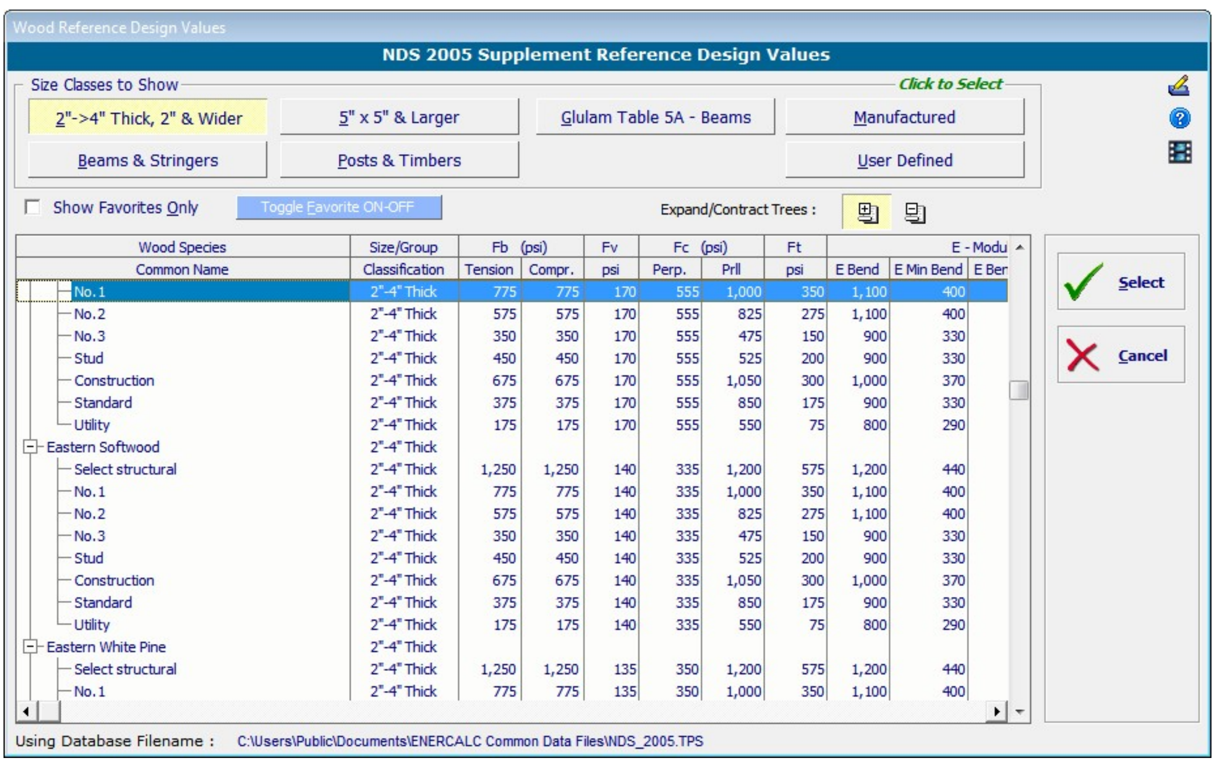
We still have two critical items to specify.....the stress properties of the wood member to use and the physical size of the beam.

First, let's retrieve the reference design values. For this sample we want to use a Douglas Fir glued-laminated beam of type 24F-V8.

Click the General Data tab shown bubbled below:

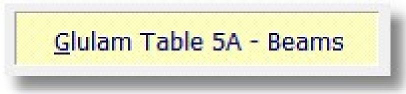


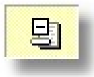
In the Design Values category, click the [**Display Wood Material Database**] button . The Wood Stress Database will be displayed as shown below:



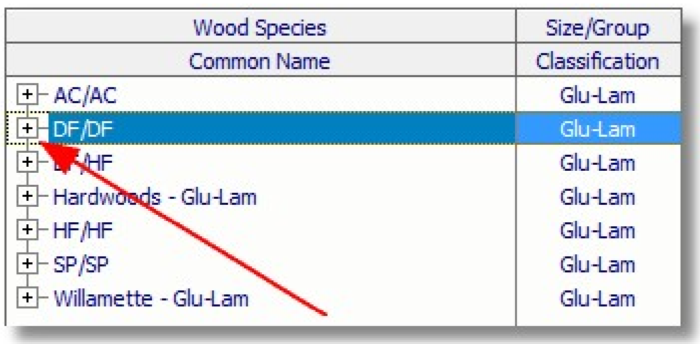
By default the last selection of stress grade is displayed. We want to display ONLY the glued-laminated sections for Douglas Fir.

In the Size Classes to Show category, click the [**Glulam Table 5A Beams**] button:

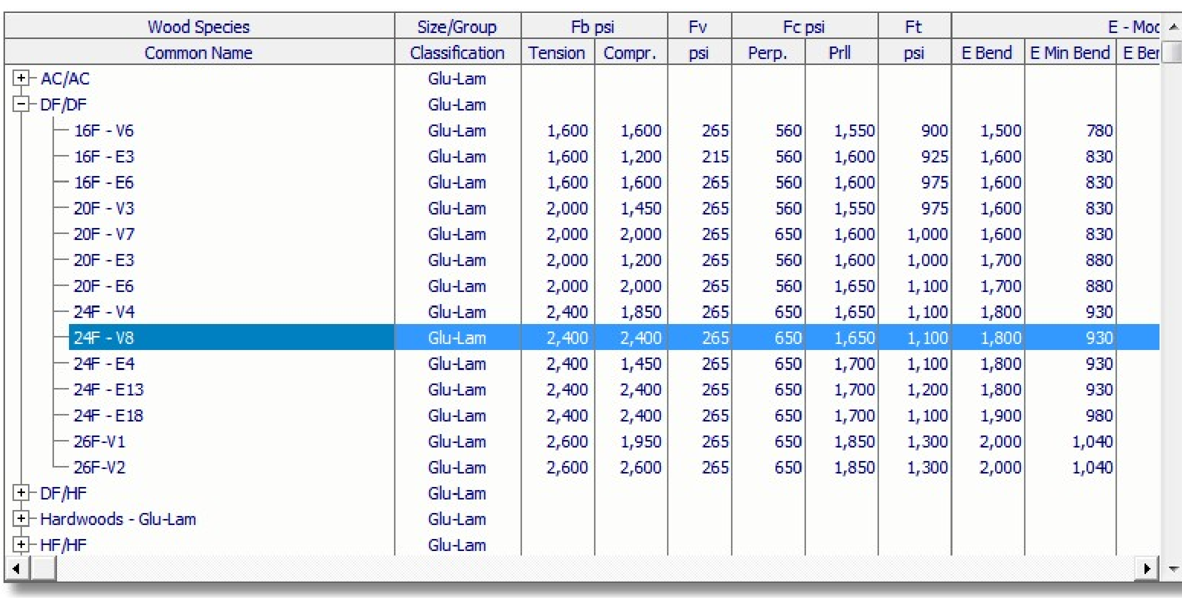


Click the [**Contract Trees**]  button to collapse the display of Wood Species Common Names.

Click the [**+**] in front of the DF/DF item to expand the Douglas Fir list:

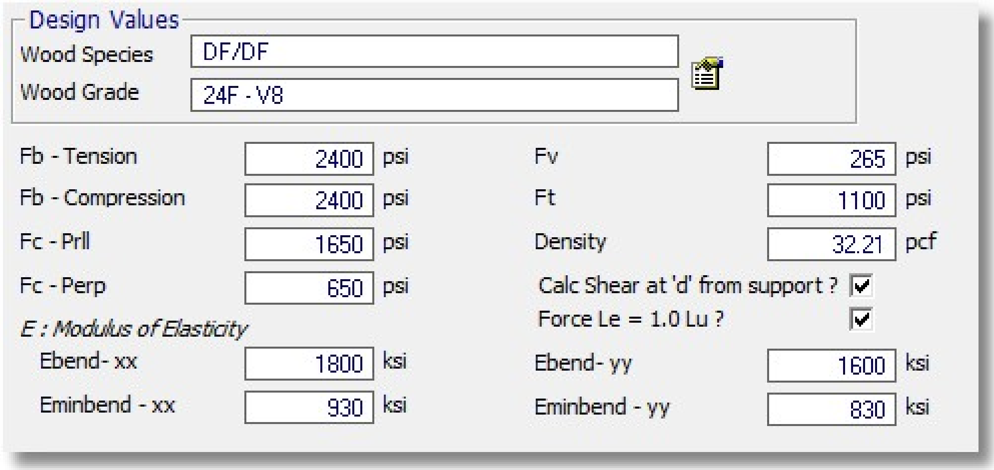


The result of these choices will be a list of stress grades as shown....



Wood Species	Size/Group	Fb psi		Fv	Fc psi		Ft			E - Moc
Common Name	Classification	Tension	Compr.	psi	Perp.	Prll	psi	E Bend	E Min Bend	E Ber
AC/AC	Glu-Lam									
DF/DF	Glu-Lam									
16F - V6	Glu-Lam	1,600	1,600	265	560	1,550	900	1,500	780	
16F - E3	Glu-Lam	1,600	1,200	215	560	1,600	925	1,600	830	
16F - E6	Glu-Lam	1,600	1,600	265	560	1,600	975	1,600	830	
20F - V3	Glu-Lam	2,000	1,450	265	560	1,550	975	1,600	830	
20F - V7	Glu-Lam	2,000	2,000	265	650	1,600	1,000	1,600	830	
20F - E3	Glu-Lam	2,000	1,200	265	560	1,600	1,000	1,700	880	
20F - E6	Glu-Lam	2,000	2,000	265	560	1,650	1,100	1,700	880	
24F - V4	Glu-Lam	2,400	1,850	265	650	1,650	1,100	1,800	930	
24F - V8	Glu-Lam	2,400	2,400	265	650	1,650	1,100	1,800	930	
24F - E4	Glu-Lam	2,400	1,450	265	650	1,700	1,100	1,800	930	
24F - E13	Glu-Lam	2,400	2,400	265	650	1,700	1,200	1,800	930	
24F - E18	Glu-Lam	2,400	2,400	265	650	1,700	1,100	1,900	980	
26F-V1	Glu-Lam	2,600	1,950	265	650	1,850	1,300	2,000	1,040	
26F-V2	Glu-Lam	2,600	2,600	265	650	1,850	1,300	2,000	1,040	
DF/HF	Glu-Lam									
Hardwoods - Glu-Lam	Glu-Lam									
HF/HF	Glu-Lam									

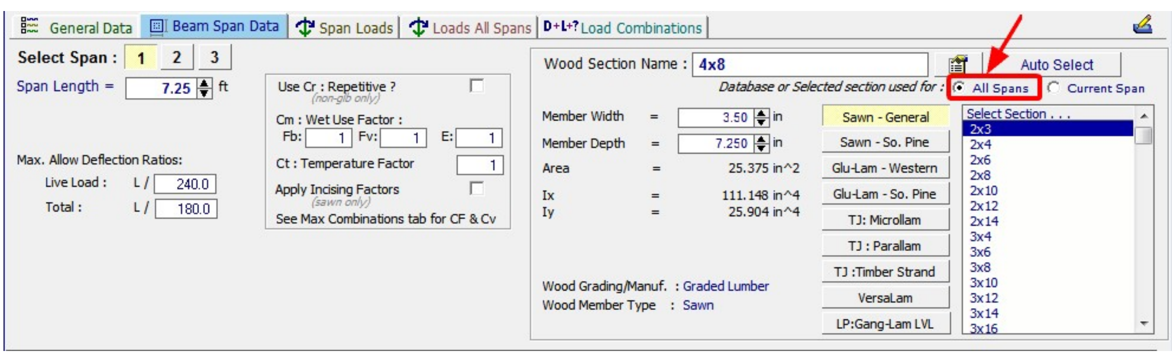
Click the **24F-V8** item as shown, and then click the [**Select**] button. The stress information area on the General tab will immediately be updated to reflect your selection:



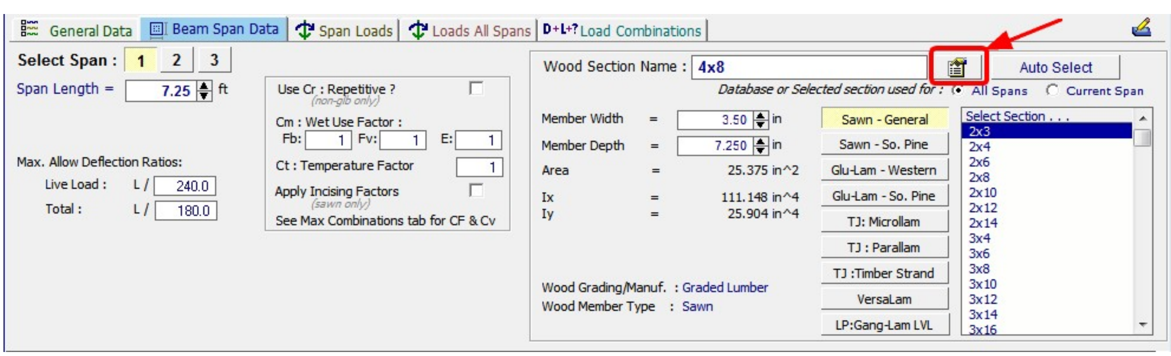
Next, let's retrieve the section property data for a 6.75" x 30" glulam beam

Click the Beam Span Data tab 

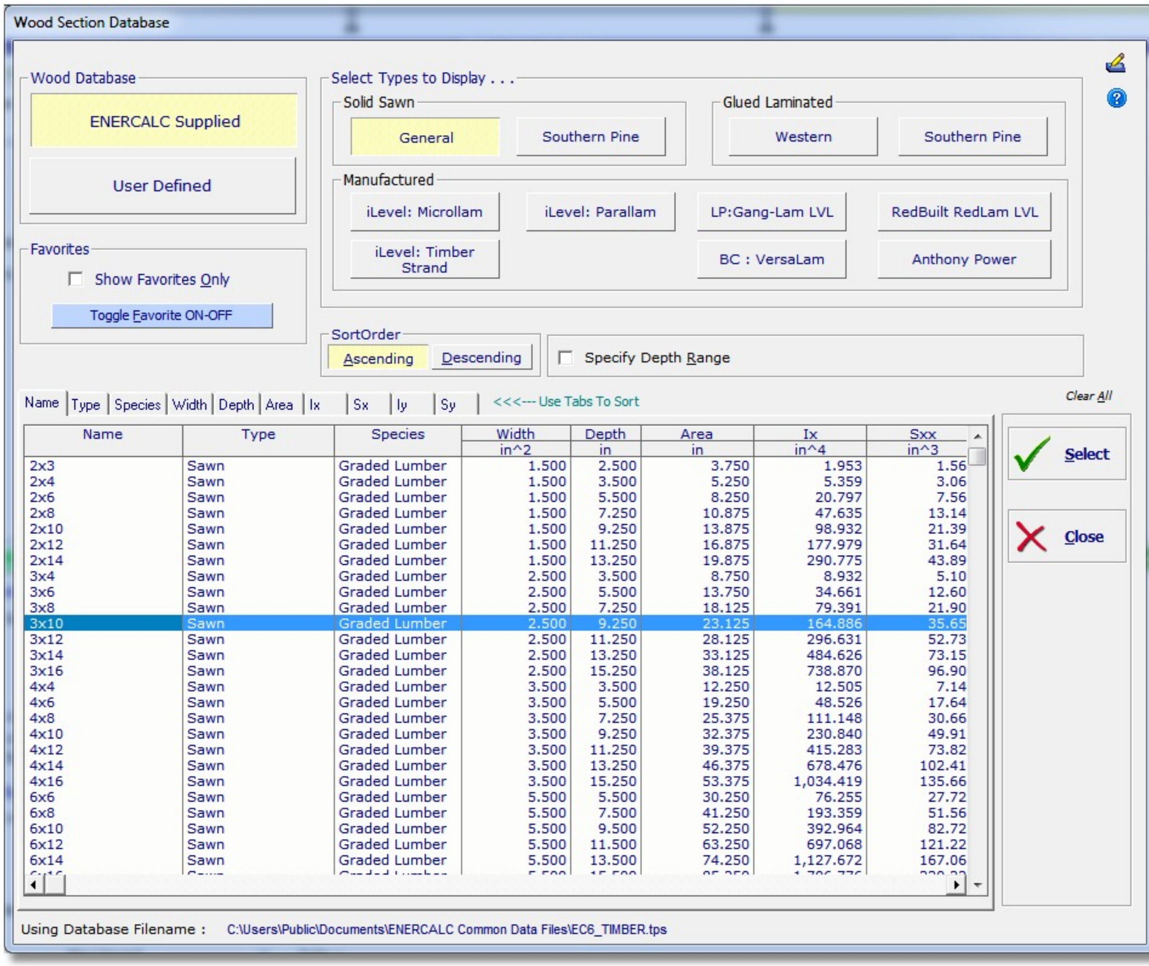
and make sure that the [**All Spans**] option is selected as shown below:



Then click the [**Section Database**] button shown bubbled below:



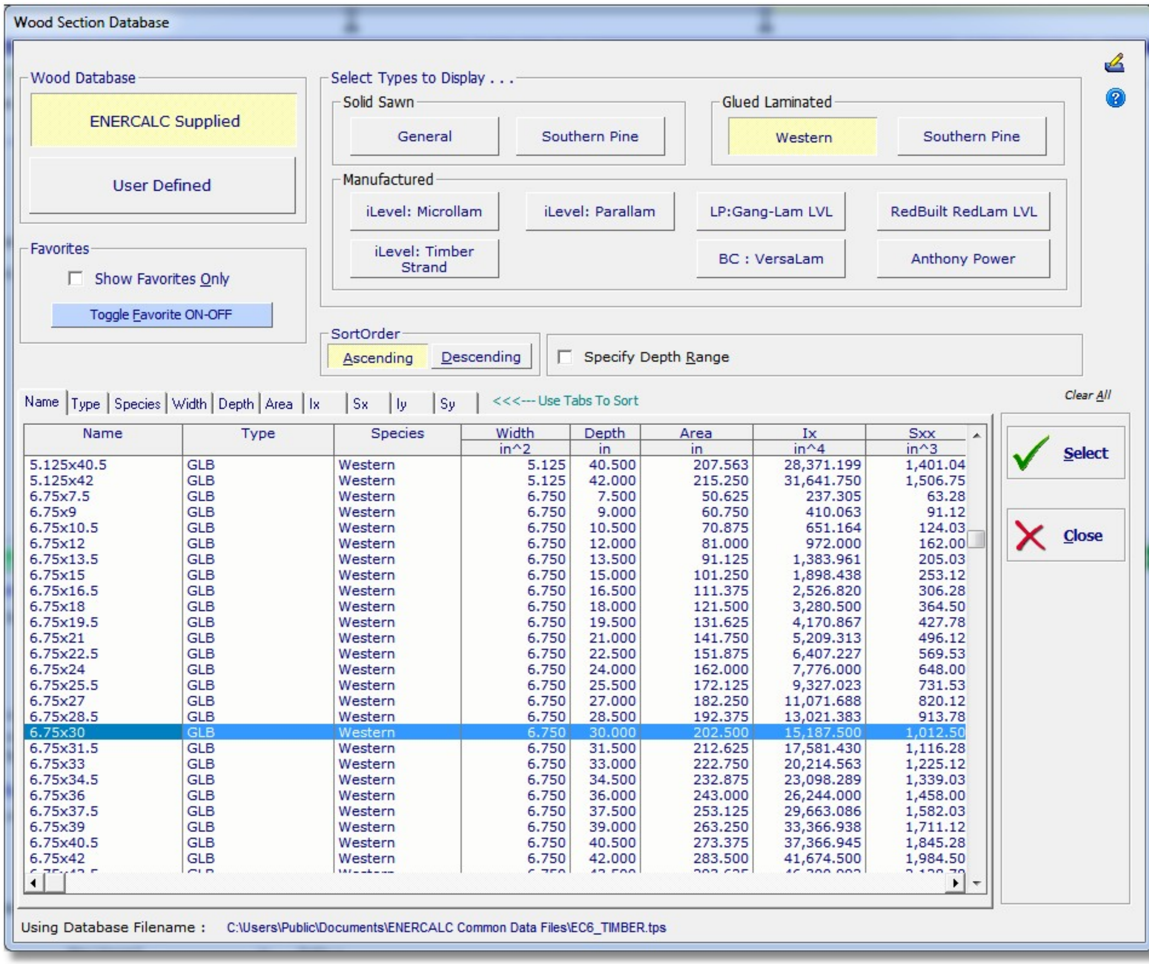
The Wood Section Database will be displayed as shown below:



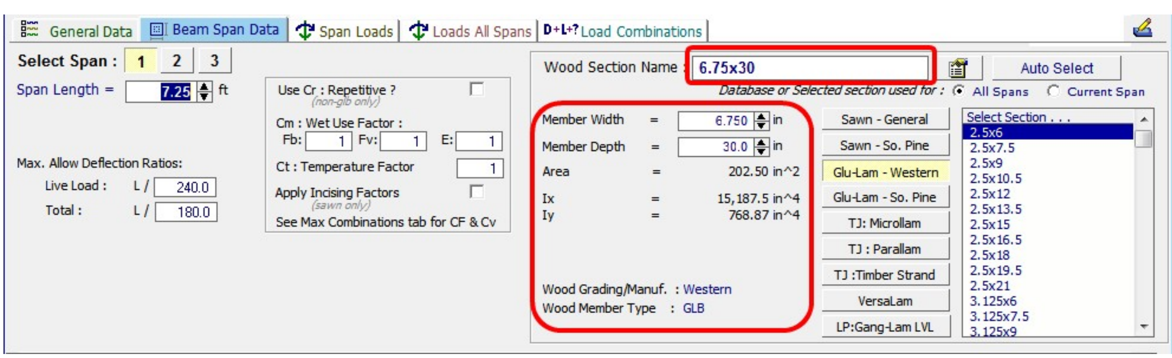
Now click the 

button in the Glued Laminated category so that only those members are displayed. The result of this choice will be a list of glulam beams.

Scroll down through the database and highlight the 6.75" x 30" beam as shown below:



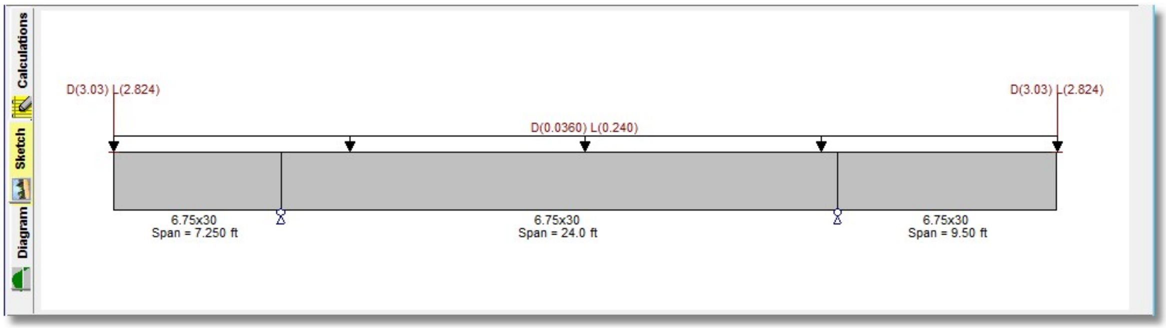
Click the  button. The beam size information area on the Beam Span Data tab will immediately be updated to reflect your selection:



9.11 Displaying a Sketch

Displaying a Sketch

We now have a calculation with span lengths, loads, allowable stresses, and a beam size. It's time to review the sketch to confirm that we have entered the critical span and load information correctly. Click the Sketch tab, and the working area of your screen will be replaced with a simple diagrammatic representation of the beam.



9.12 Displaying Diagrams

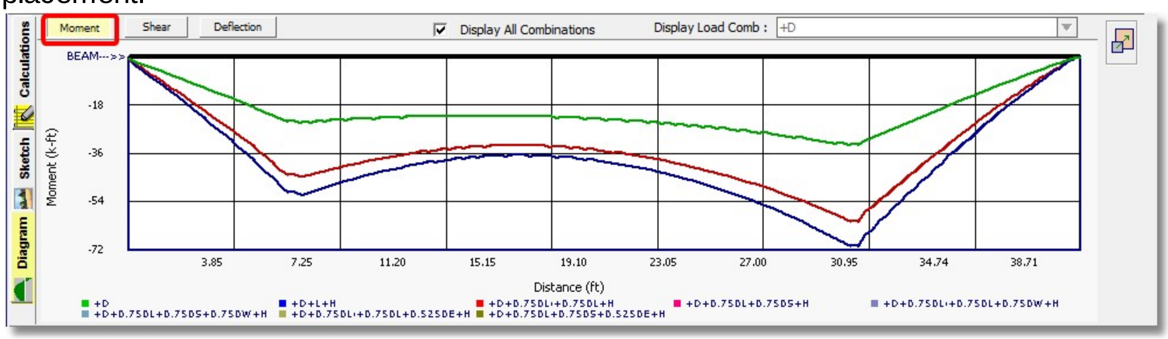
Displaying Diagrams

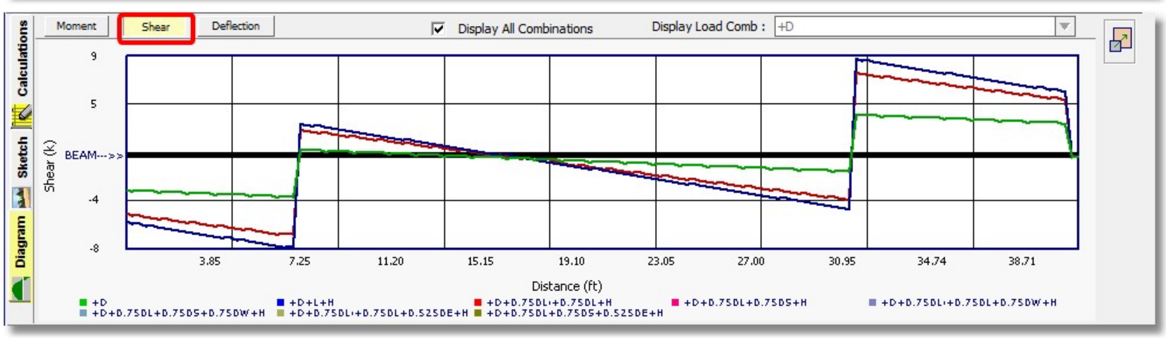
In addition to the Sketch of the beam, we can also display several types of diagrams. Selecting the Diagrams tab will display the Shear/Moment/Deflection diagram as shown below.

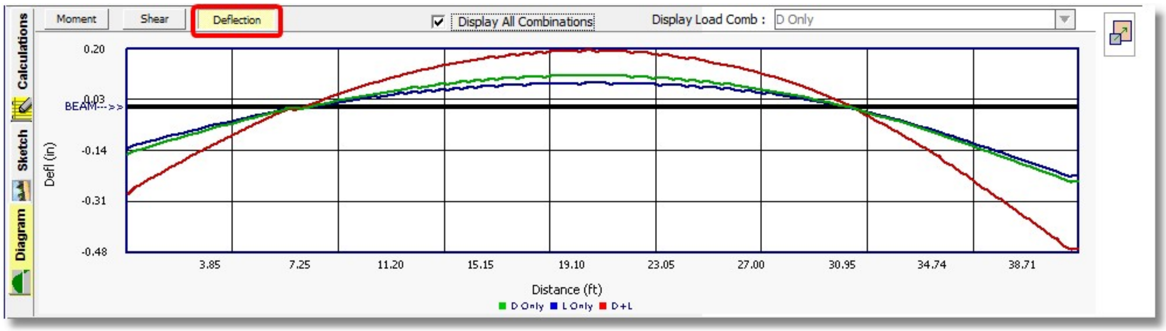
Of special interest are the checkboxes that control the location of live load for the analysis shown in the graphic. Because this program automatically moves the live load around on the beam to obtain the governing values of shear, moment, and deflection on each span for the final calculation results, there really is no single load case that represents the entire beam analysis.

By selecting the live load locations here you can view the detailed shear/moment/deflection variations produced by each load application.

Note: In the previous steps, we turned on the option to have the program automatically handle unbalanced Live load placement. While this is a helpful feature for design, it tends to produce an abundance of results. If you'd like to view shear, moment, and deflection diagrams that are similar to those shown below, click the Span Loads tab and temporarily turn off the option to have the program automatically perform unbalanced Live load placement.






Moment
Shear
Deflection
Display All Combinations
Display Load Comb :
D Only
Calculations
Sketch
Diagram
0.20
0.03
BEAM--->>
-0.14
-0.31
-0.48
Defl (in)
3.85
7.25
11.20
15.15
19.10
23.05
27.00
30.95
34.74
38.71
Distance (ft)
D Only
L Only
D+L

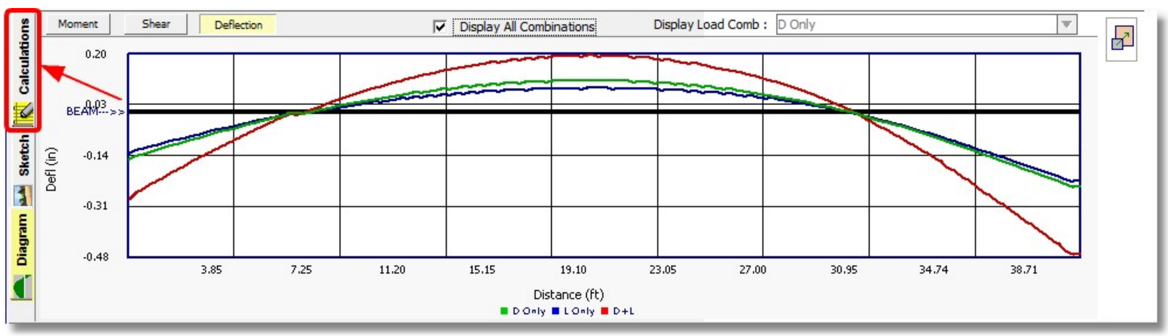
9.13 Automatic Member Section Selection

Automatic Member Section Selection

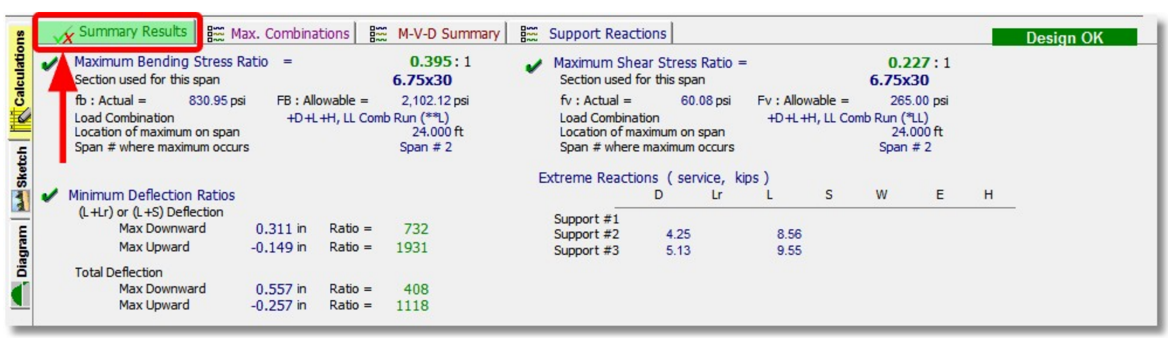
An additional feature of many of **Structural Engineering Library** calculation modules is automatic selection of sizes that satisfy your specified design criteria.

Note: If you turned off the option to have the program automatically perform unbalanced Live load placement in the previous topic, please click back to the Span Loads tab now and make sure the option for Auto Unbalanced Live Load Placement is selected once again.

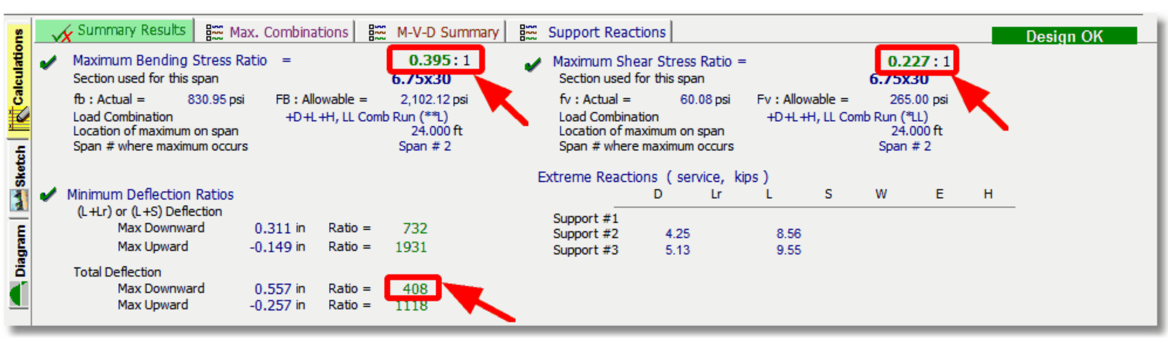
Click the Calculations tab at the left edge of the screen as shown bubbled below:



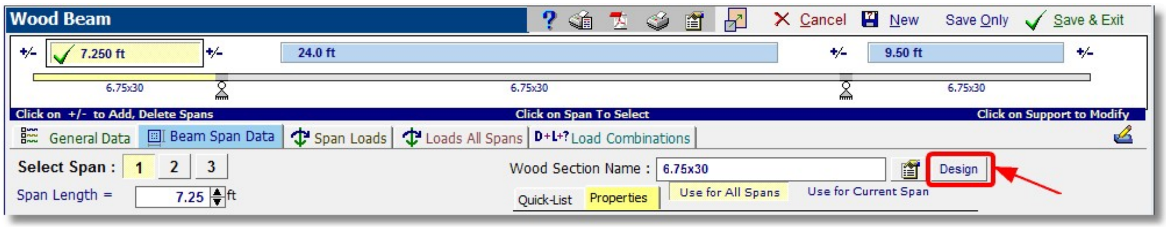
When the screen displays the Calculations environment, click the Summary Results tab as shown bubbled below:



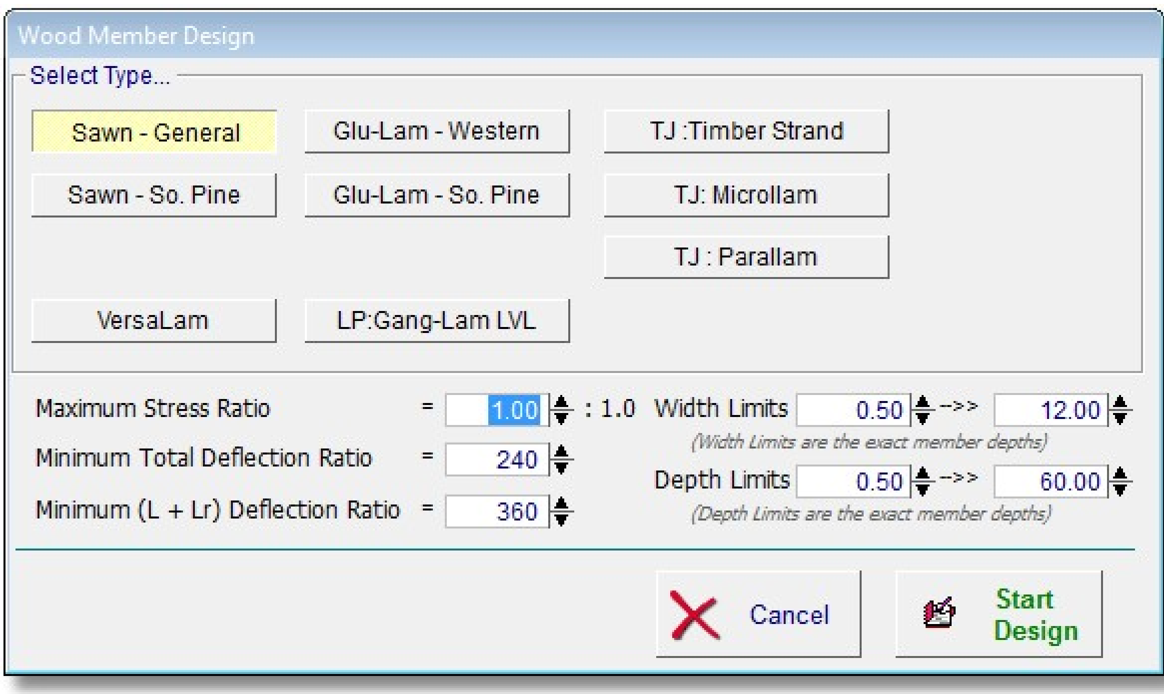
Take a look at the stress and deflection ratios bubbled in the screen capture below:



We can see that our 6.75" x 30" size guess may not be an optimum use of the section. It would be interesting to see if we could find a more efficient section. Rather than manually trying other sizes from the database, we will use the Automatic Design function to select an optimum member size for us. Click the [**Design**] button on the Beam Span Data tab shown bubbled below:

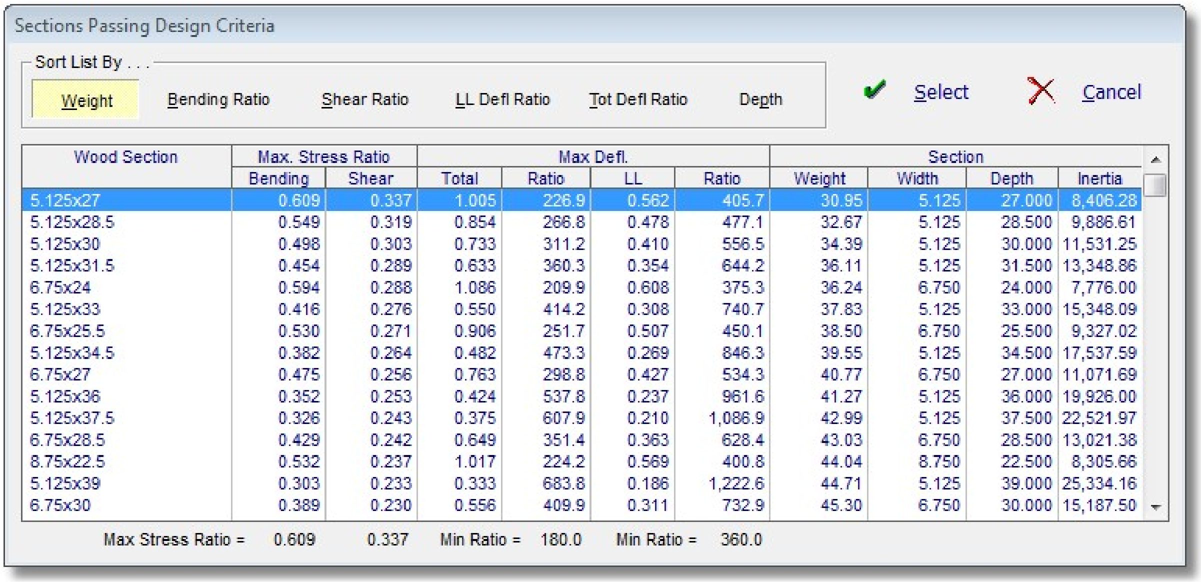


The Wood Member Design dialog is displayed, and it allows you to set the design criteria:



On the Wood Member Design dialog, click the [**Glu-Lam - Western**] section type, set the Maximum Stress Ratio to 1.0, and the Minimum Total Deflection Ratio to 180, then click the [**Start Design**] button.

The software will automatically search the built-in Wood Section Database for all sections that satisfy this criteria using the beam span, stress grade, and loading you have specified. In a few moments the following dialog will be displayed:


Sections Passing Design Criteria

Sort List By . . . Weight | Bending Ratio | Shear Ratio | LL Defl Ratio | Tot Defl Ratio | Depth — Select — Cancel

Wood Section	Max. Stress Ratio		Max Defl.				Section			
	Bending	Shear	Total	Ratio	LL	Ratio	Weight	Width	Depth	Inertia
5.125x27	0.609	0.337	1.005	226.9	0.562	405.7	30.95	5.125	27.000	8,406.28
5.125x28.5	0.549	0.319	0.854	266.8	0.478	477.1	32.67	5.125	28.500	9,886.61
5.125x30	0.498	0.303	0.733	311.2	0.410	556.5	34.39	5.125	30.000	11,531.25
5.125x31.5	0.454	0.289	0.633	360.3	0.354	644.2	36.11	5.125	31.500	13,348.86
6.75x24	0.594	0.288	1.086	209.9	0.608	375.3	36.24	6.750	24.000	7,776.00
5.125x33	0.416	0.276	0.550	414.2	0.308	740.7	37.83	5.125	33.000	15,348.09
6.75x25.5	0.530	0.271	0.906	251.7	0.507	450.1	38.50	6.750	25.500	9,327.02
5.125x34.5	0.382	0.264	0.482	473.3	0.269	846.3	39.55	5.125	34.500	17,537.59
6.75x27	0.475	0.256	0.763	298.8	0.427	534.3	40.77	6.750	27.000	11,071.69
5.125x36	0.352	0.253	0.424	537.8	0.237	961.6	41.27	5.125	36.000	19,926.00
5.125x37.5	0.326	0.243	0.375	607.9	0.210	1,086.9	42.99	5.125	37.500	22,521.97
6.75x28.5	0.429	0.242	0.649	351.4	0.363	628.4	43.03	6.750	28.500	13,021.38
8.75x22.5	0.532	0.237	1.017	224.2	0.569	400.8	44.04	8.750	22.500	8,305.66
5.125x39	0.303	0.233	0.333	683.8	0.186	1,222.6	44.71	5.125	39.000	25,334.16
6.75x30	0.389	0.230	0.556	409.9	0.311	732.9	45.30	6.750	30.000	15,187.50

Max Stress Ratio = 0.609 0.337 Min Ratio = 180.0 Min Ratio = 360.0

Notice how the listed members can be sorted in different orders based on: Weight, Bending Ratio, Shear Ration, LL Deflection Ratio,Total Deflection Ratio, Depth, etc. This allows you to review the beam sizes that pass your criteria in different ways.

Click on the **5.125 x 27** beam and then click the [**Select**] button. The final design (with Summary Results tab showing) looks like this:

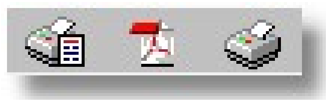
9.14 Printing a Calculation

Printing a calculation

Okay...congratulations! You've created a Project File, added a calculation, entered data, retrieved database information, performed an automatic design, and viewed some graphics. It's time to print this calculation and take a look at what the software provides for documentation.

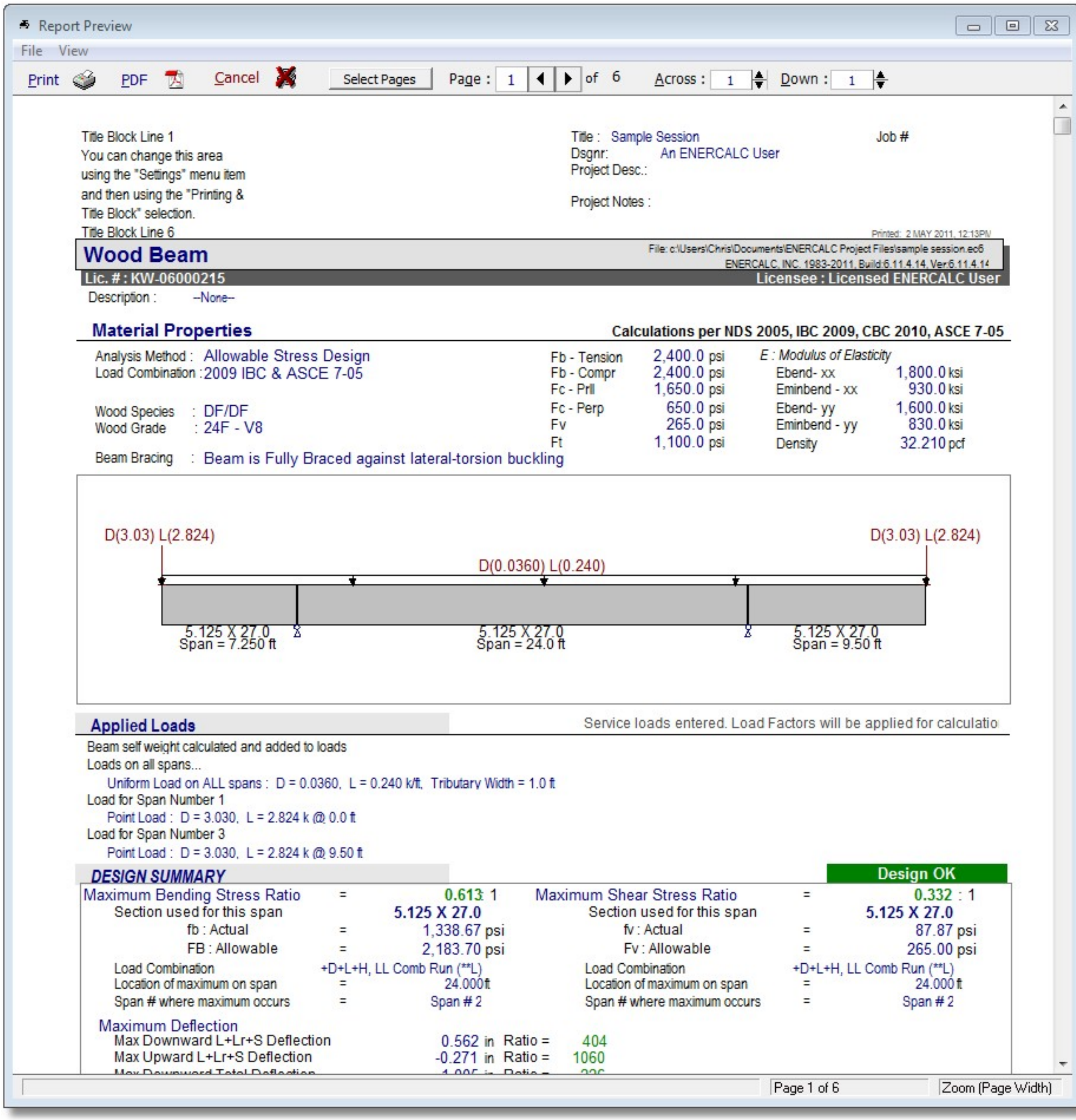
Please remember that you have previously entered project information, title block information, and of course now you have a valid engineering calculation to print.

You have three printing options in **Structural Engineering Library**:



The  icon that looks like a printer sends the report report directly to the printer. The  icon that looks like a printer with a page symbol displays the print preview in a window and allows you to review it before sending it to the printer. The  icon with the Adobe logo creates an Adobe Acrobat PDF file of the report.

The Print Preview window looks like this:



To continue, simply click the [**Print**] button and the standard Windows print dialog will be displayed. Select your printer and you will receive a printout looking very similar to the one below:

Title Block Line 1
You can change this area
using the "Settings" menu item
and then using the "Printing &
Title Block" selection.
Title Block Line 6

Title : Sample Session
Dsgnr: An ENERCALC User
Project Desc.:

Project Notes :

Job #

Printed: 2 MAY 2011, 12:13PM

Wood Beam

File: c:\Users\Chris\Documents\ENERCALC Project Files\sample session.ec6
ENERCALC, INC. 1983-2011, Build:6.11.4.14, Ver:6.11.4.14

Lic. # : KW-06000215 **Licensee : Licensed ENERCALC User**

Description : --None--

Material Properties

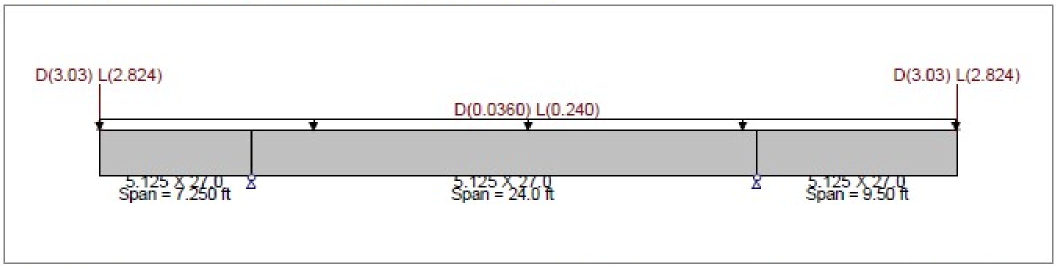
Calculations per NDS 2005, IBC 2009, CBC 2010, ASCE 7-05

Analysis Method : Allowable Stress Design
Load Combination 2009 IBC & ASCE 7-05

Wood Species : DF/DF
Wood Grade : 24F - V8

Beam Bracing : Beam is Fully Braced against lateral-torsion buckling

Fb - Tension	2,400.0 psi	*E : Modulus of Elasticity*	
Fb - Compr	2,400.0 psi	Ebend- xx	1,800.0 ksi
Fc - Prll	1,650.0 psi	Eminbend - xx	930.0 ksi
Fc - Perp	650.0 psi	Ebend- yy	1,600.0 ksi
Fv	265.0 psi	Eminbend - yy	830.0 ksi
Ft	1,100.0 psi	Density	32.210 pcf



Applied Loads

Service loads entered. Load Factors will be applied for calculations.

Beam self weight calculated and added to loads
Loads on all spans...
Uniform Load on ALL spans : D = 0.0360, L = 0.240 k/ft, Tributary Width = 1.0 ft
Load for Span Number 1
Point Load : D = 3.030, L = 2.824 k @ 0.0 ft
Load for Span Number 3
Point Load : D = 3.030, L = 2.824 k @ 9.50 ft

DESIGN SUMMARY

Design OK

Maximum Bending Stress Ratio	=	0.613: 1	Maximum Shear Stress Ratio	=	0.332 : 1
Section used for this span		5.125 X 27.0	Section used for this span		5.125 X 27.0
fb : Actual	=	1,338.67 psi	fv : Actual	=	87.87 psi
FB : Allowable	=	2,183.70 psi	Fv : Allowable	=	265.00 psi
Load Combination		+D+L+H, LL Comb Run (**L)	Load Combination		+D+L+H, LL Comb Run (**L)
Location of maximum on span	=	24.000 ft	Location of maximum on span	=	24.000 ft
Span # where maximum occurs	=	Span # 2	Span # where maximum occurs	=	Span # 2

Maximum Deflection

Max Downward L+Lr+S Deflection	0.562 in	Ratio =	404
Max Upward L+Lr+S Deflection	-0.271 in	Ratio =	1060
Max Downward Total Deflection	1.005 in	Ratio =	226
Max Upward Total Deflection	-0.472 in	Ratio =	610

Maximum Forces & Stresses for Load Combinations

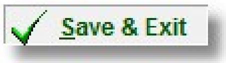
Load Combination		Max Stress Ratios							Summary of Moment Values			Summary of Shear Values		
Segment Length	Span #	M	V	C_d	$C_{F/V}$	C_r	C_m	C_t	Mactual	fb-design	Fb-allow	Vactual	fv-design	Fv-allow
+D														
Length = 7.158 ft	1	0.209	0.159	0.900	1.000	1.000	1.000	1.000	-23.40	451.04	2,160.00	3.51	38.04	238.50
Length = 24.092 ft	2	0.312	0.160	0.900	0.910	1.000	1.000	1.000	-31.81	612.95	1,965.33	3.52	38.17	238.50
Length = 9.50 ft	3	0.284	0.160	0.900	0.998	1.000	1.000	1.000	-31.81	612.95	2,156.18	3.52	38.17	238.50
+D+L+H, LL Comb Run (**L)					0.998	1.000	1.000	1.000						
Length = 7.158 ft	1	0.188	0.144	1.000	1.000	1.000	1.000	1.000	-23.40	451.04	2,400.00	3.51	38.04	265.00
Length = 24.092 ft	2	0.613	0.332	1.000	0.910	1.000	1.000	1.000	-69.46	1,338.67	2,183.70	8.11	87.87	265.00
Length = 9.50 ft	3	0.559	0.332	1.000	0.998	1.000	1.000	1.000	-69.46	1,338.67	2,395.76	8.11	87.87	265.00
+D+L+H, LL Comb Run (*L*)					0.998	1.000	1.000	1.000						

9.15 Saving a Calculation

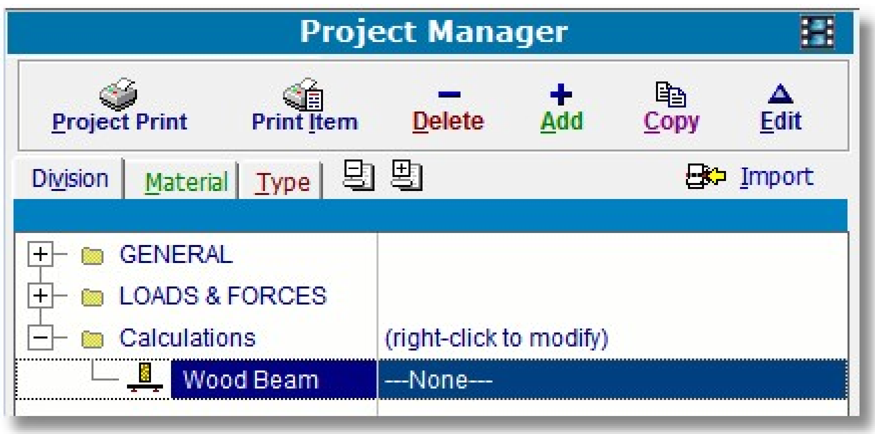
Saving a Calculation

With as much as we have accomplished in this short period of time, it is very important to remember that we have not yet SAVED this calculation. Everything associated with this calculation is still stored in the computer's RAM. A power outage or severe system lockup would cause a loss of your current calculation.

So to save this new calculation to the Project File, click the [**Save & Exit**] button



. The current calculation will be saved and the display will return to the Project Manager. In the image below, you can see that a Wood Beam calculation has been added to the Division named "Calculations":



Congratulations! You calculation is now saved and you have a real Project File with an actual calculation.

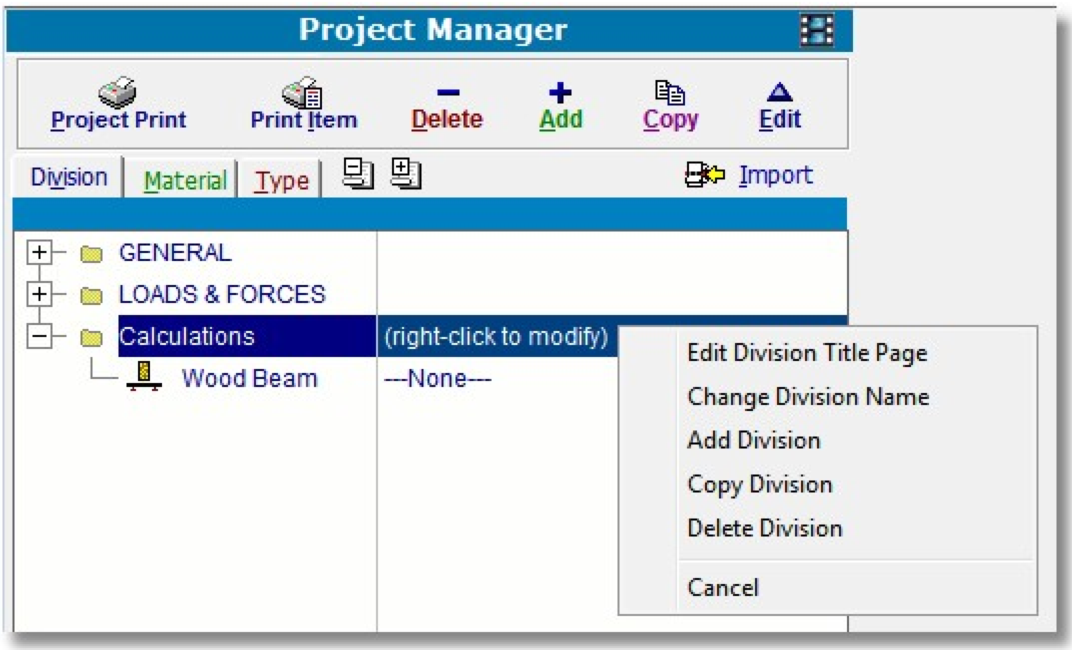
9.16 Editing a Division Name and Adding a New Division

Editing a Division Name

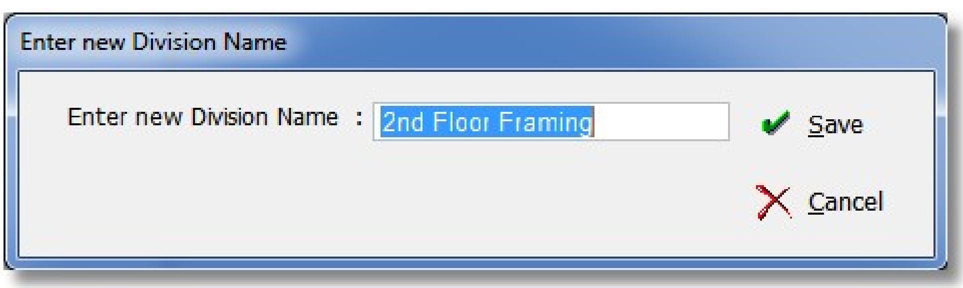
Let's do a little more work here in the current Project File. We will edit the Division named "Calculations", and we will add a new Division.

Edit a Division Name

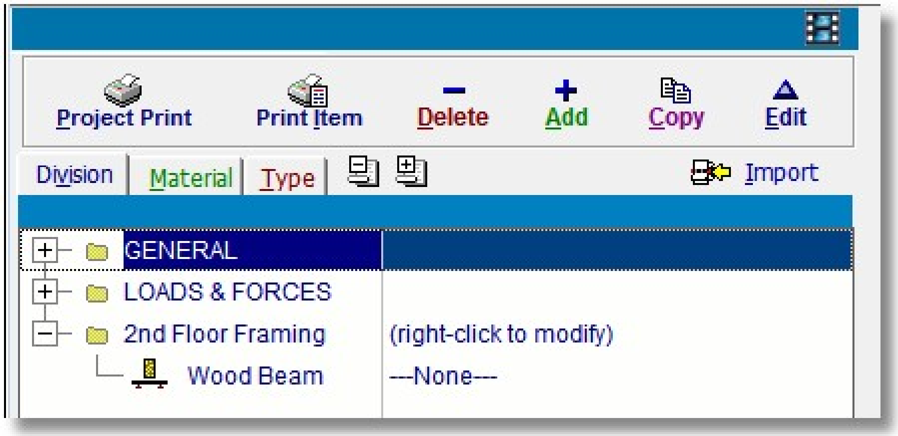
To edit a Division name, simply right-click the name and click **Change Division Name** in the pop-up menu.



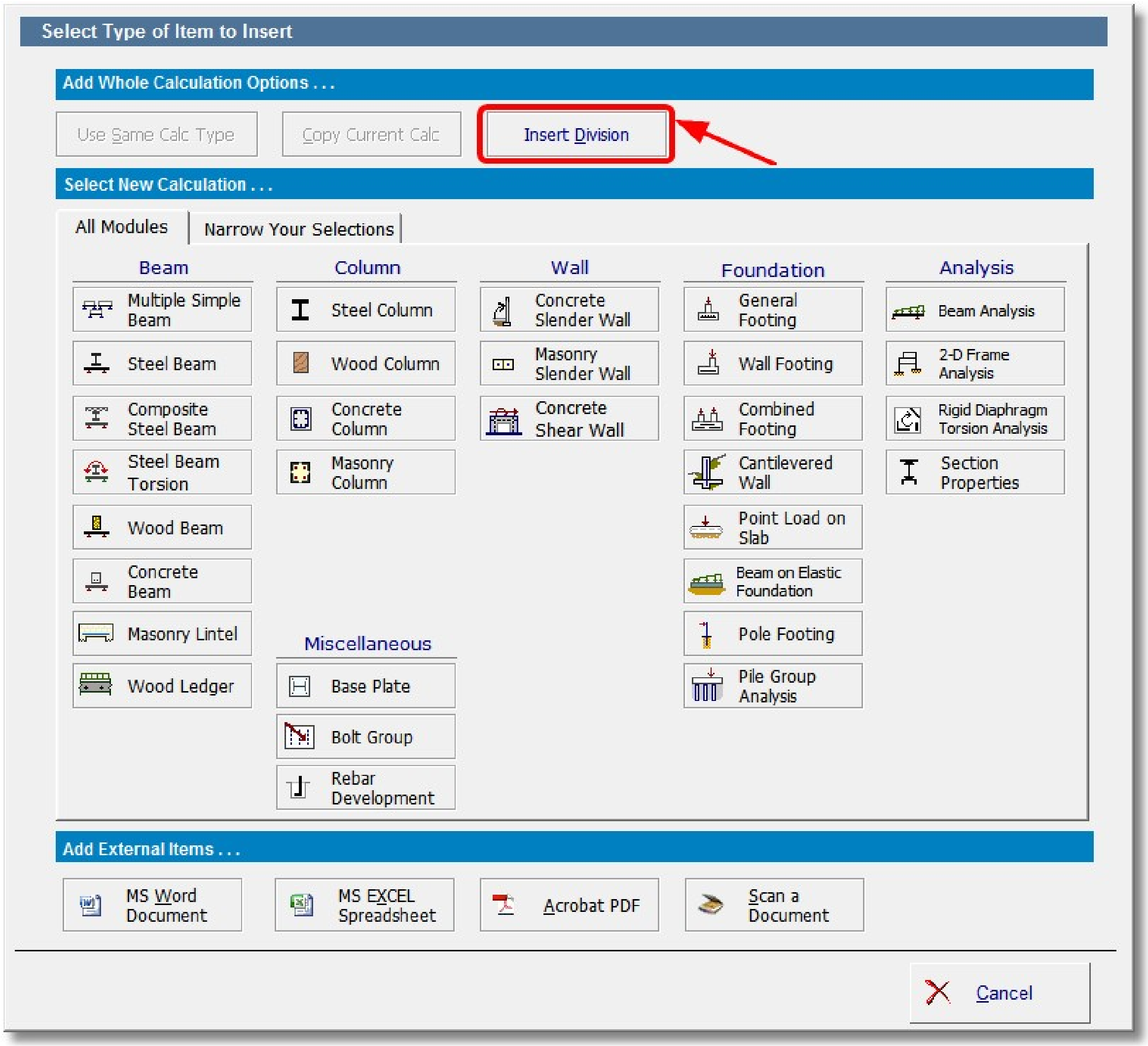
In the Enter New Division Name dialog, revise the Division name to "2nd Floor Framing" as shown below:



Click [**Save**]. You will notice that the Division name has been revised from "Calculations" to "2nd Floor Framing", but it still contains the Wood Beam calculation.

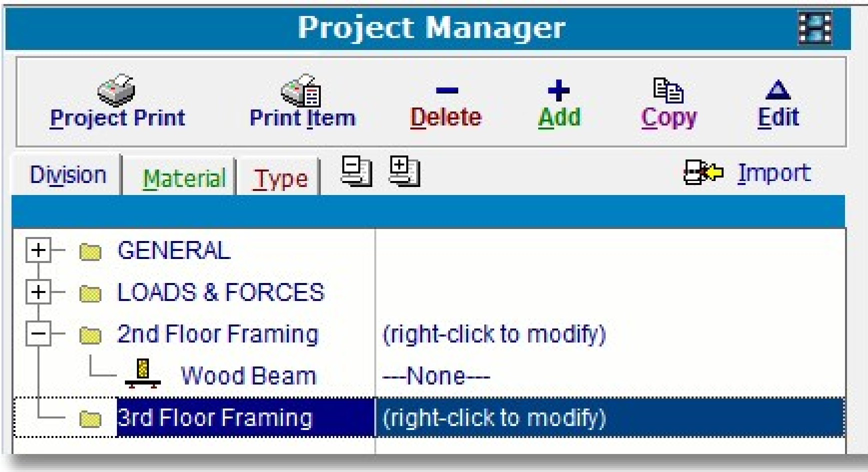


Now we will add a NEW Division named "3rd Floor Framing". To do this, click on the Division named "2nd Floor Framing" and then click the [**Add**] button  . In the Select Item to Insert dialog, click the [**Insert Division**] button shown bubbled below:



Note: There is also an option in the right-click pop-up menu named Add Division.

Then enter the name "3rd Floor Framing" and click [**Save**]. Your Project calculation list should look like this:

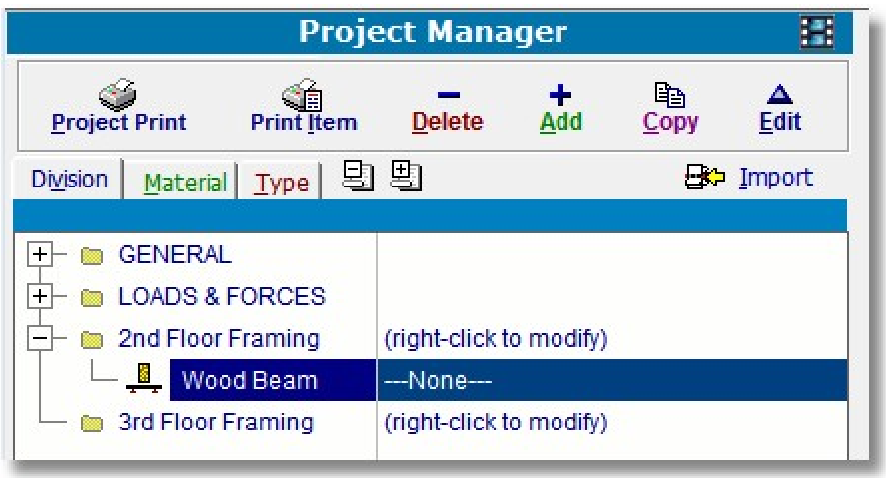

Project Manager
Project Print
Print Item
Delete
Add
Copy
Edit
Division
Material
Type
Import
GENERAL
LOADS & FORCES
2nd Floor Framing
(right-click to modify)
Wood Beam
---None---
3rd Floor Framing
(right-click to modify)

9.17 Adding Another Calculation

Adding Another Calculation in the New Division by using [Copy Calc]

Similar to the way we added our first calculation, we will now add another calculation in the 2nd Floor Framing Division by making a copy of the current Wood Beam calculation and then move it into the 3rd Floor Framing Division.

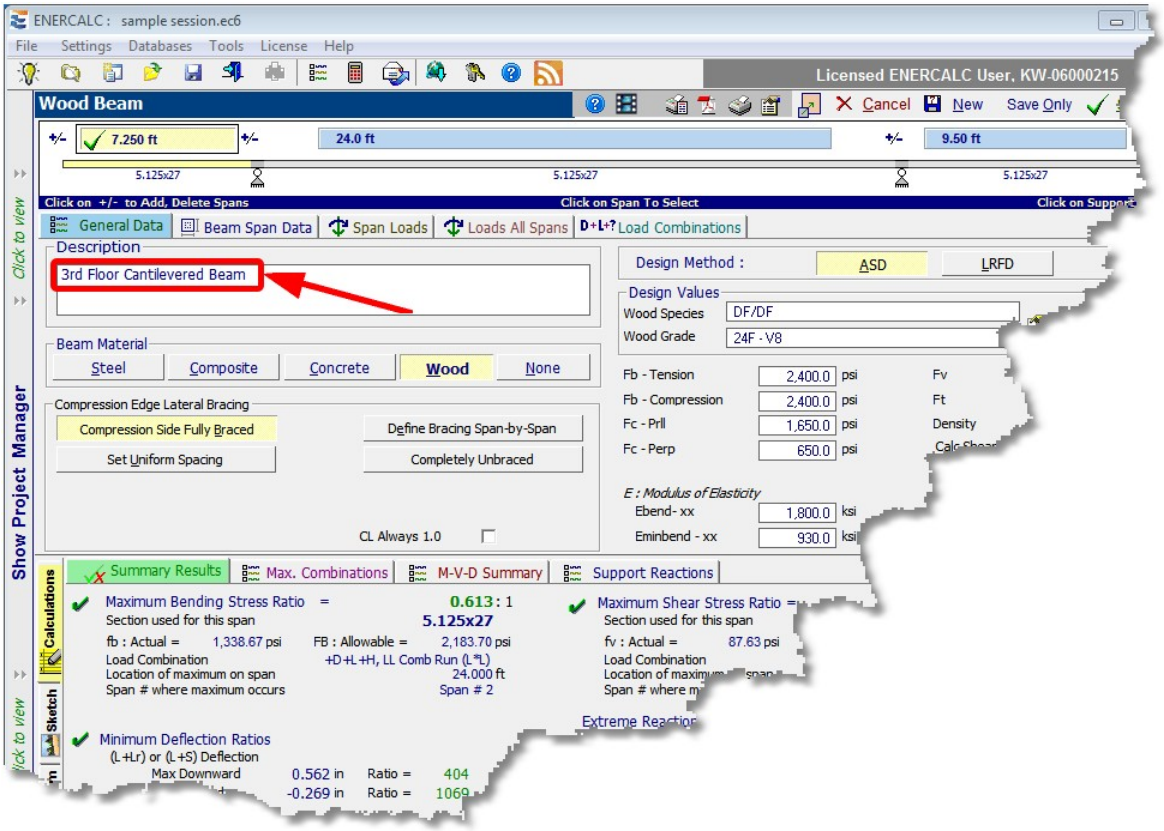
First, click on the Wood Beam calculation as shown in the screen capture below:



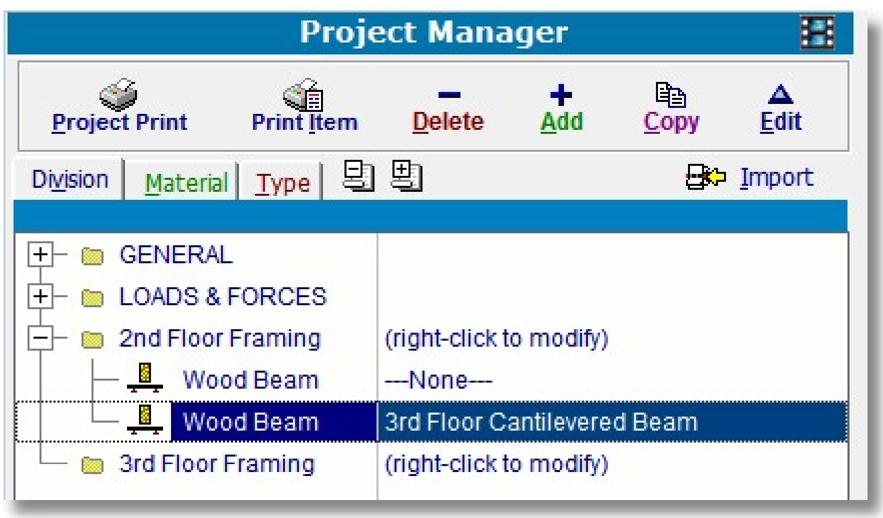
Then, click the 

button. This will read all the data from the highlighted calculation, create an entirely new calculation that is an exact copy of the highlighted one, and open the new calculation so it can be edited as necessary.

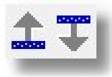
When the Wood Beam calculation module opens, change the Description item to read "3rd Floor Cantilevered Beam" as shown below:

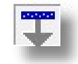


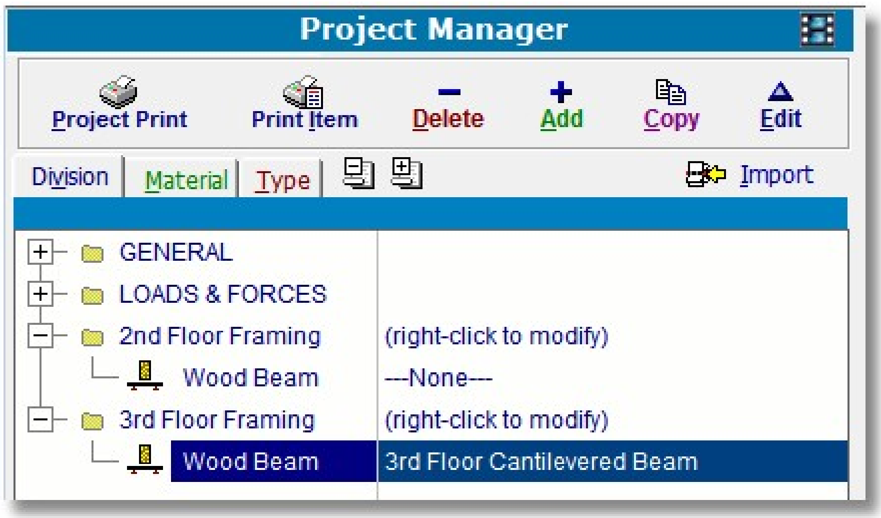
Next click the [**Save & Exit**] button to save this new calculation to the Project File and close the calculation. The Project Manager will appear as follows:



The final step is to move this new calculation into the 3rd Floor Framing Division. This is easily done by using these buttons:



While the new calculation is highlighted, click the Move Highlighted Item Down  button to move it down to the 3rd Floor Framing Division. The Project Calculation list will appear like this:



9.18 Creating a Technical Support Question

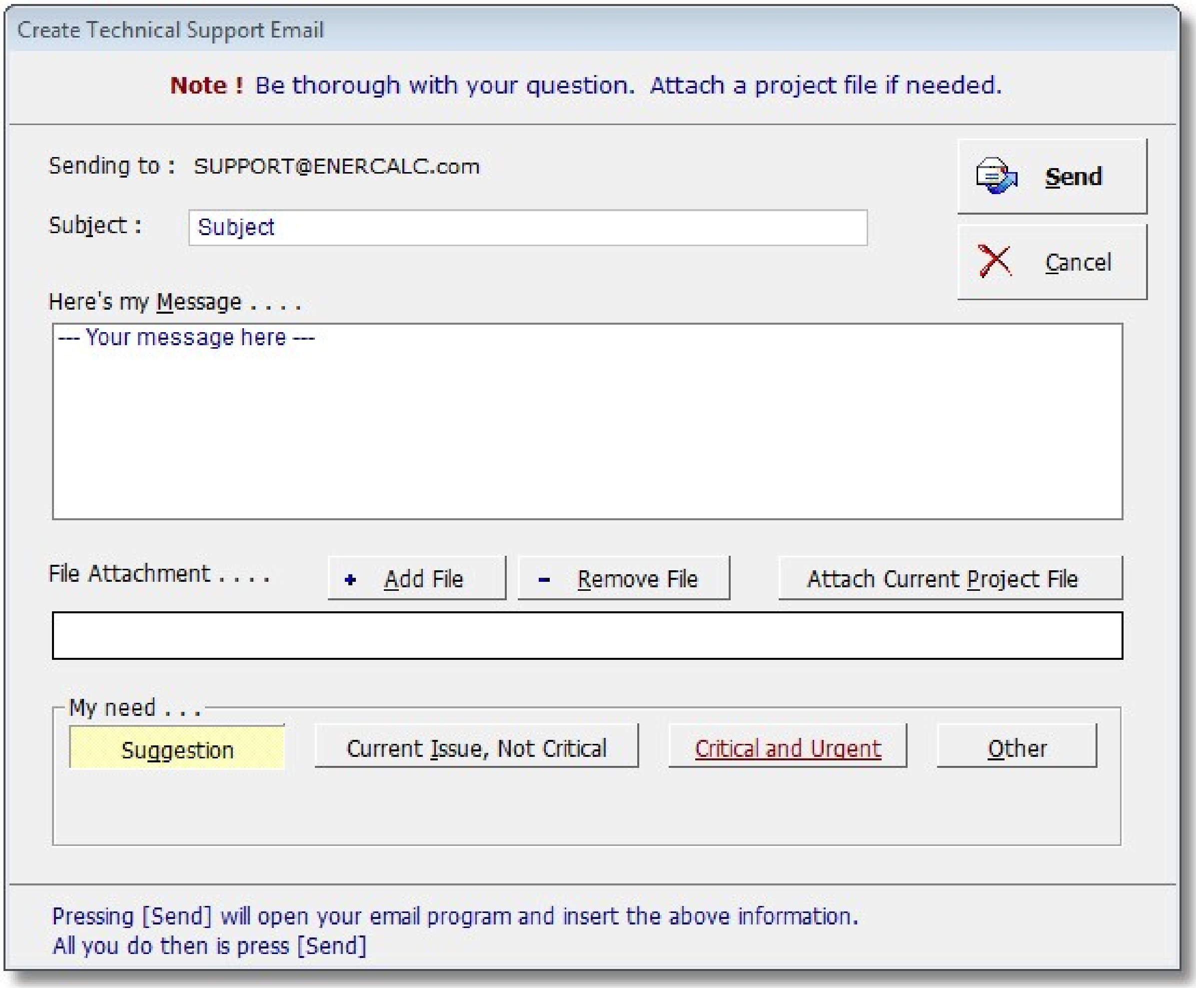
Creating a Technical Support Question

The occasion may arise that you will have a question about the software system. To best serve you, there are certain pieces of information that are essential. Among these are your name, the name of your company, your "KW" user registration number, and the build number of the software you are using.

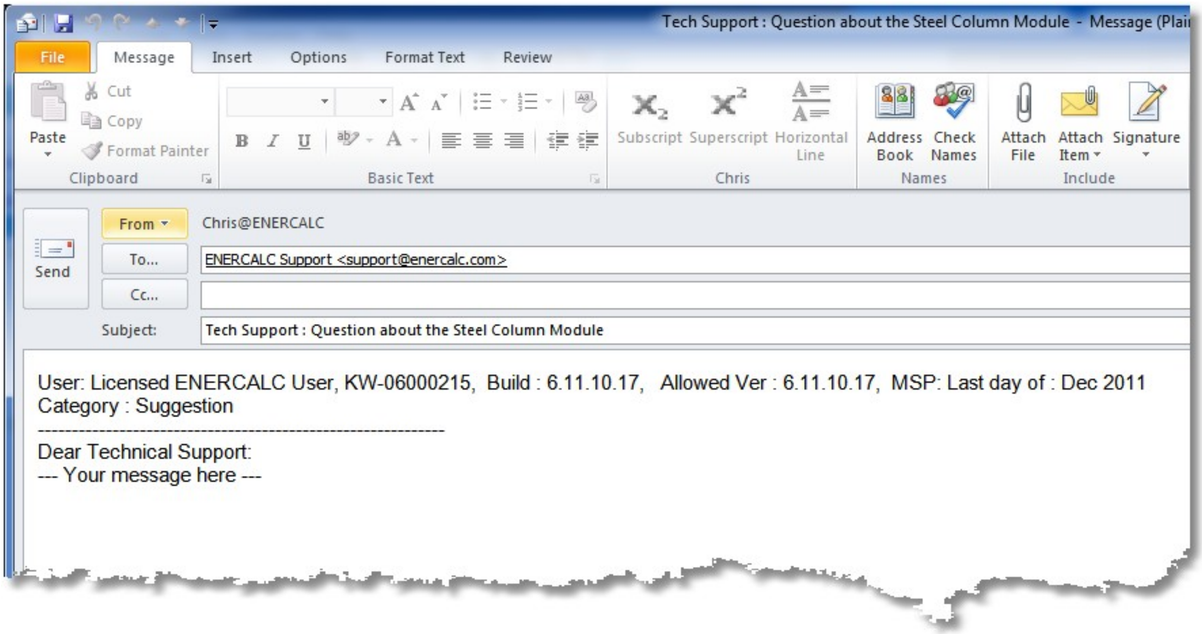
To make it easy for you to give us all this AND state your question, **Structural Engineering Library** has a built-in technical support form. It allows you to simply type in your question and either email it directly to our Technical Support Group or print it out to fax to us.

By emailing or faxing us your Technical Support questions, we can easily identify you, determine if you are in need of a maintenance release, and easily read your questions. In addition, the email method offers the option of attaching the subject ENERCALC file.

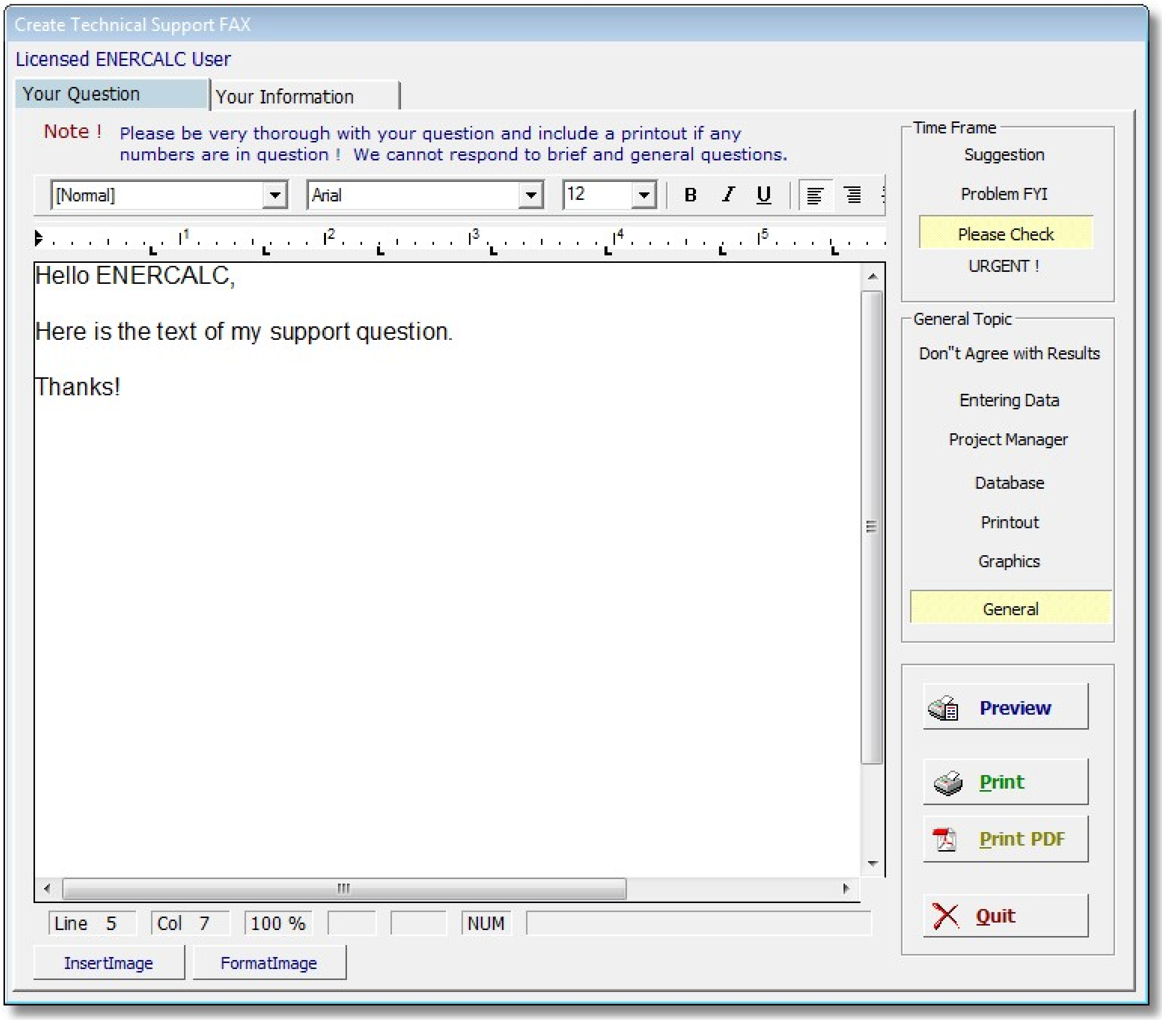
To create an email:
Click **Help > Create Tech Support EMAIL** from the main menu. This opens the form shown below, in which you can enter your question and attach your Project File if desired.



When finished, click the [**Send**] button to transfer the completed message (with attachments) to your email program:

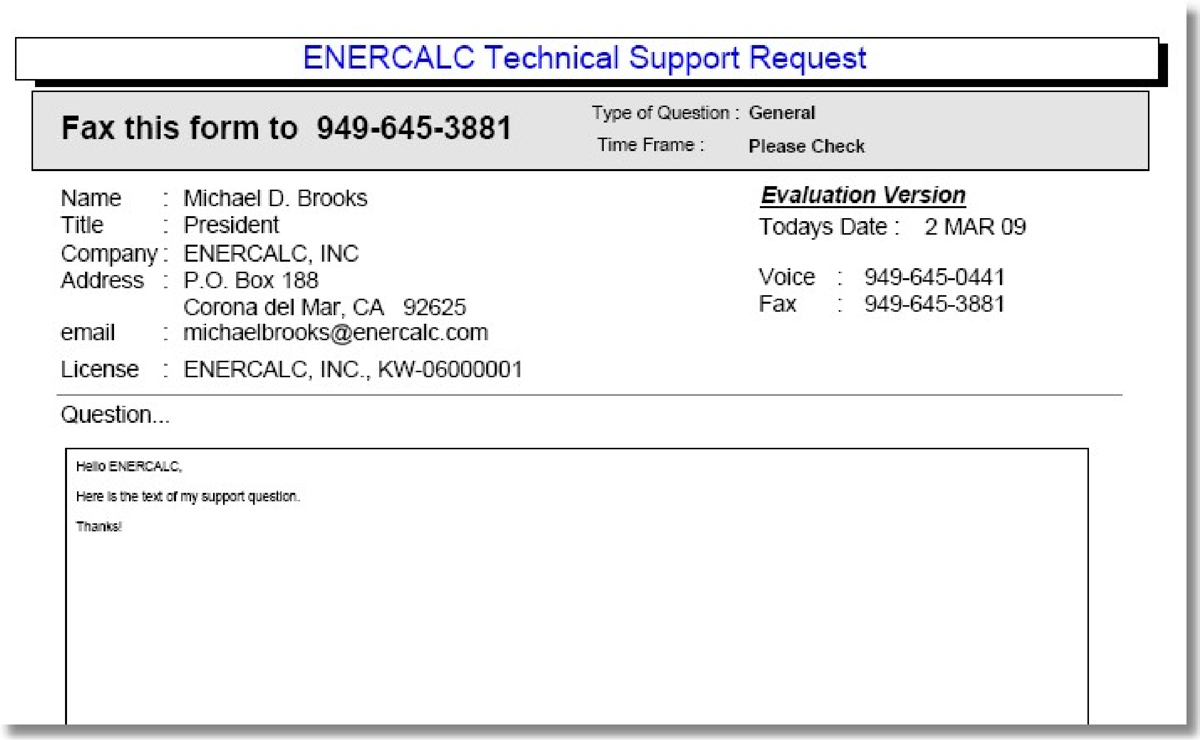


To create a FAX:
Click **Help > Create Tech Support FAX** from the main menu to use the form shown below to enter your question.



When finished, the completed message can be printed for faxing (or it can be sent directly from your computer-based fax software).

Here's the printed form, ready for transmission:



ENERCALC Technical Support Request

Fax this form to 949-645-3881

Type of Question : General
Time Frame : Please Check

Name : Michael D. Brooks
Title : President
Company : ENERCALC, INC
Address : P.O. Box 188
Corona del Mar, CA 92625
email : michaelbrooks@enercalc.com

License : ENERCALC, INC., KW-06000001

Evaluation Version
Todays Date : 2 MAR 09

Voice : 949-645-0441
Fax : 949-645-3881

Question...

Hello ENERCALC,

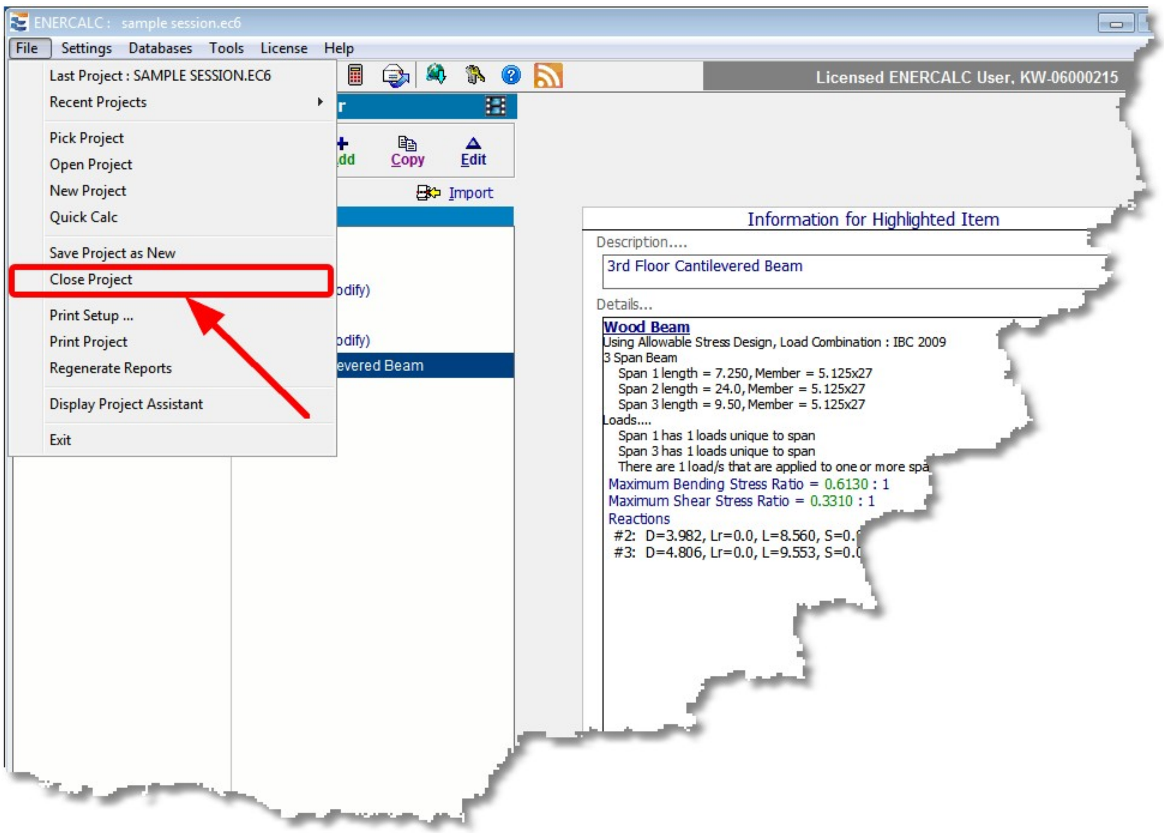
Here is the text of my support question.

Thanks!

9.19 Closing a Project File

Closing a Project File

It is now time to close out our work session. The data is saved in the Project File already, so at this point we can close the current Project File by clicking **File > Close Project** from the main menu:



Part X

10 Calculation Modules

10.1 Loads & Forces Division

The LOADS & FORCES Division in the Project Manager is automatically created for you when a new Project File is started. Click here for a video: LOADS and FORCES Division

There is only one LOADS & FORCES Division for each project, and it is always pre-populated with specific load-related calculation items. Their use is optional, but these calculation modules cannot be moved out of the LOADS & FORCES Division, nor can other calculation modules be moved into or added to the LOADS & FORCES Division.

In particular, the LOADS & FORCES Division includes modules that assist with the development of snow, wind, seismic, and reduced live load calculations.

As **Structural Engineering Library** is enhanced, the LOADS & FORCES Division will contain more and more tools to assist with developing your project load calculations.

10.1.1 ASCE Seismic Base Shear

Need more? Ask Us a Question

This module is a presentation of equivalent lateral force procedure seismic provisions in ASCE 7. Click here for a video: ASCE Seismic Base Shear

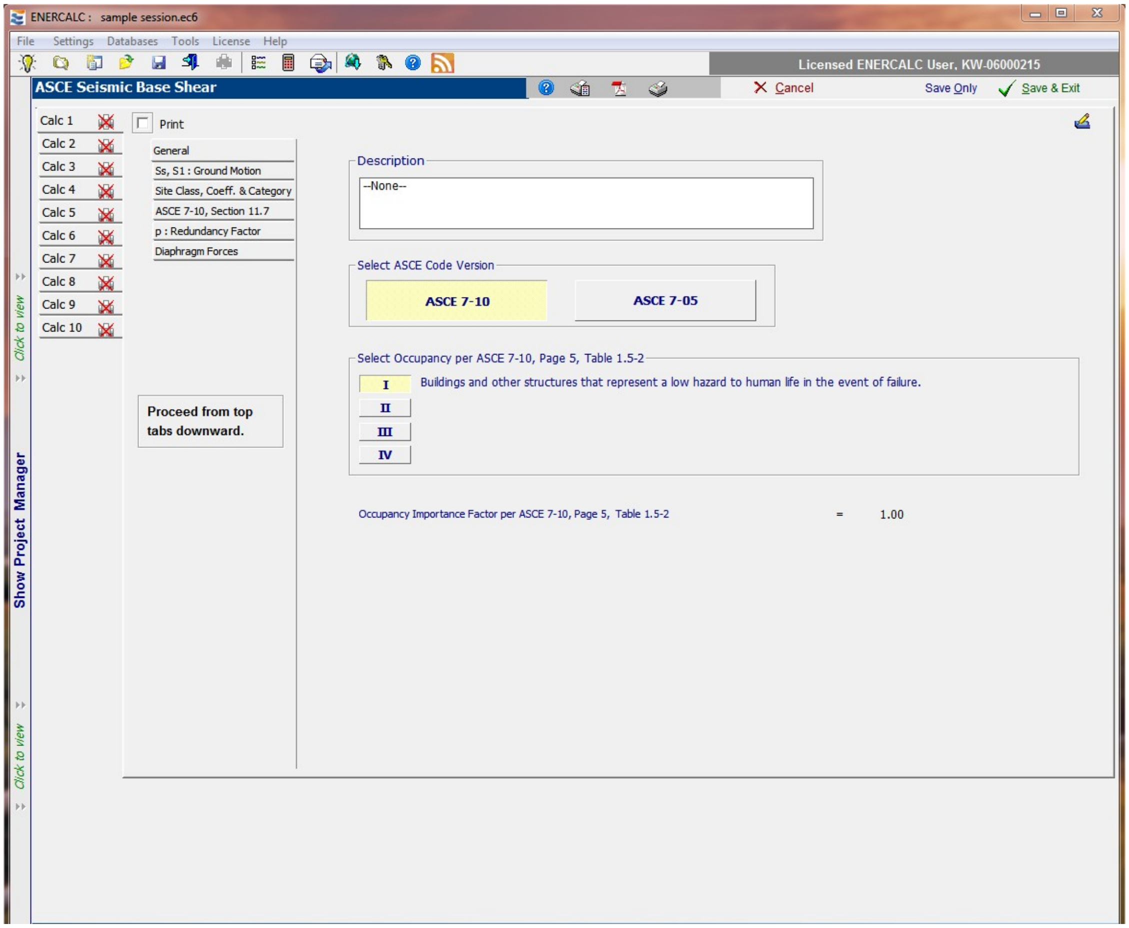
The module currently supports calculations based on ASCE 7-05 and ASCE 7-10. All of the references to ASCE 7 are given on the module screens. Above each screen capture are notes as needed to call attention to entries or explain usage.

The module is designed to allow you to work downwards through the tabs on the left side of the screen. ASCE 7 skips all around in its definitions of things to check. This module simplifies the process by properly ordering the items and the necessary decisions to be made in working toward a final seismic base shear value.

The program includes a complete national zip code database and USGS databases of seismic ground motion specifically for use with ASCE 7. Using city names or zip codes, you can look up the latitude and longitude of the representative center of the zip code.

The seismic ground motion databases consist of "gridded" values for small increments of latitude and longitude. Given the latitude and longitude for the city or zip code of interest, the surrounding grid data points are located and those seismic ground motion values are used to interpolate a value for the latitude and longitude specified. For this reason the values may not match the web-based USGS database values exactly, however the grid point interpolation is proper.

General

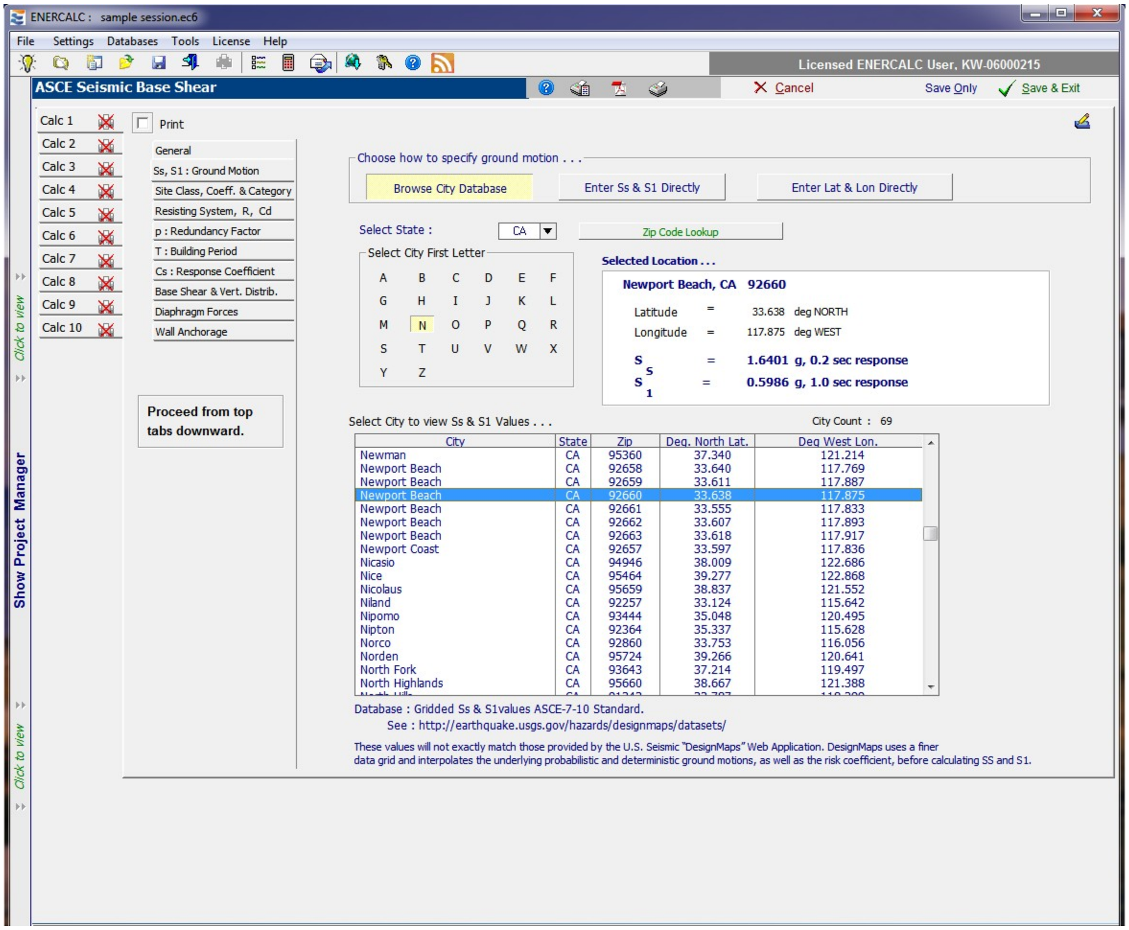


S_s, S_1: Ground Motion

Three options are available to specify the S_S and S_1 values.

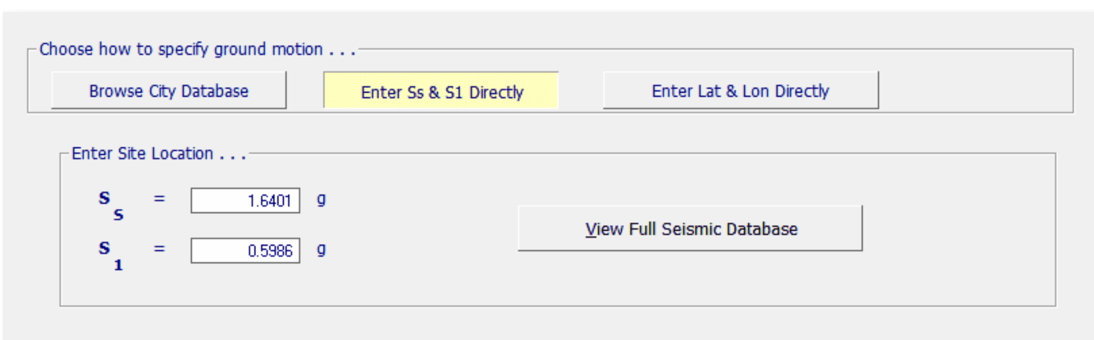
Browse City Database

Use the Select State drop-down list box to select the State. Then click one of the City First Letter buttons to have all cities (in the database) listed in the scrolling box on the right. Then simply click on the city to select. Some city names have have multiple zip codes.



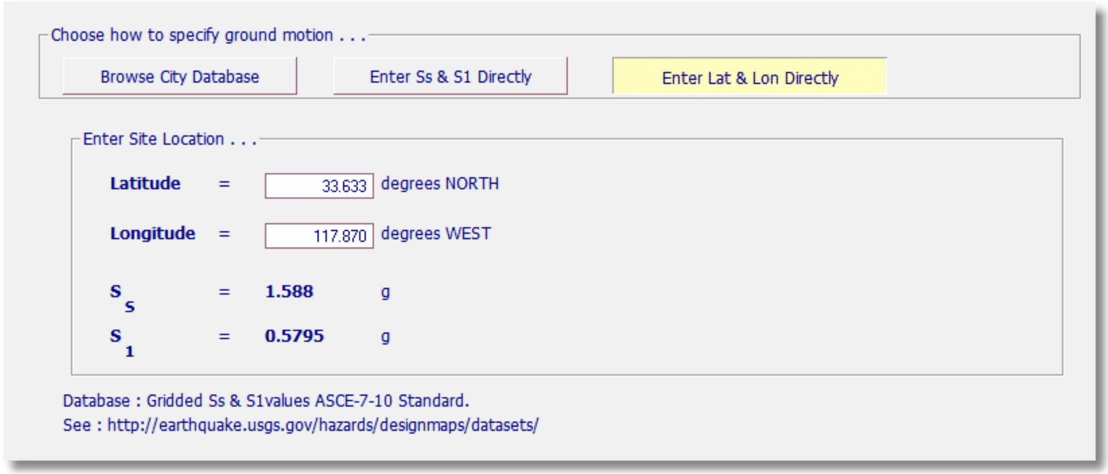
Enter S_S and S_1 Directly

Or, you can enter the S_S and S_1 values directly.



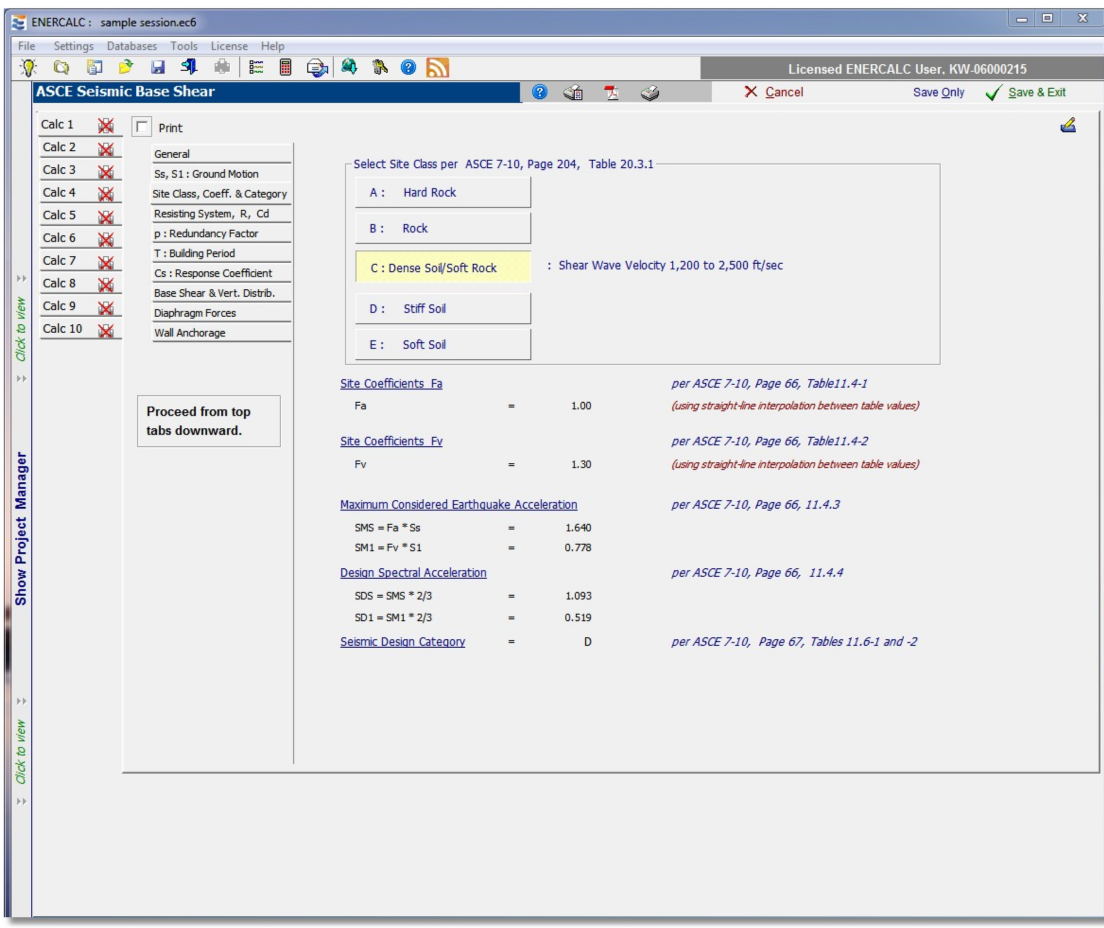
Enter Latitude & Longitude Directly

Or, you have the option to enter the latitude and longitude for the project location. *(Note that longitude is degrees WEST, so do not enter negative values.)*



Site Class, Site Coefficients and Seismic Design Category

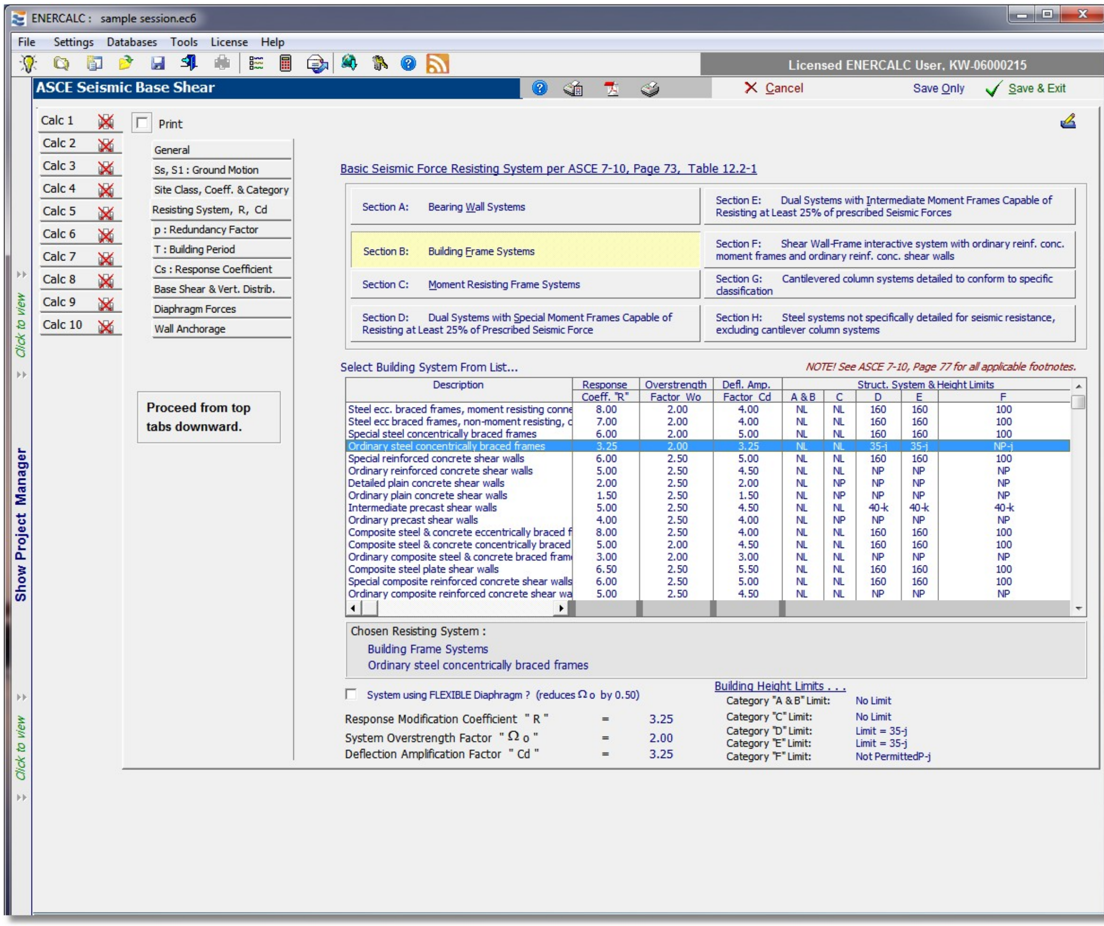
Select the appropriate Site Class based on the geotechnical conditions.



Selection of Seismic Force Resisting System

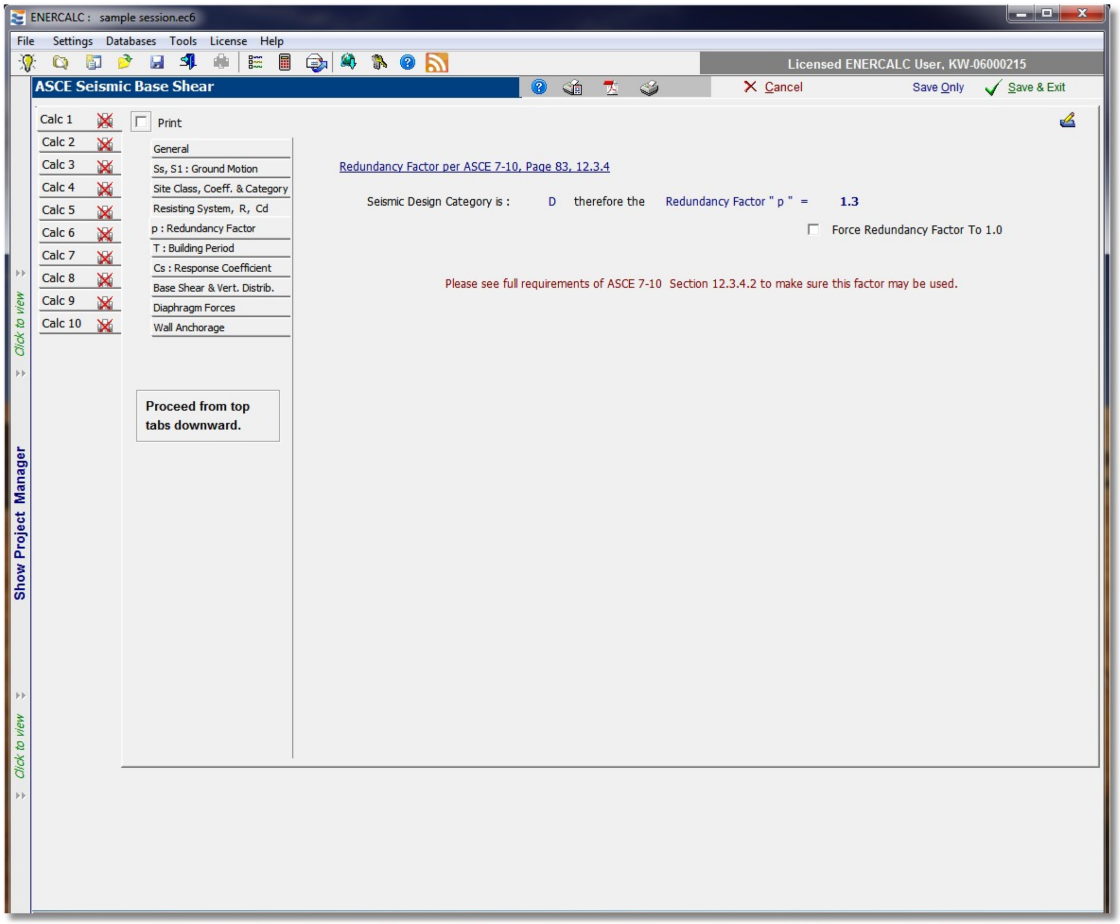
Click one of the large boxes at the top of the tab to load the table with specific selections for that general category of Seismic Force Resisting System.

Note the checkbox for systems with flexible diaphragms. This may not apply to some building systems...check ASCE for details on the particular system of interest.



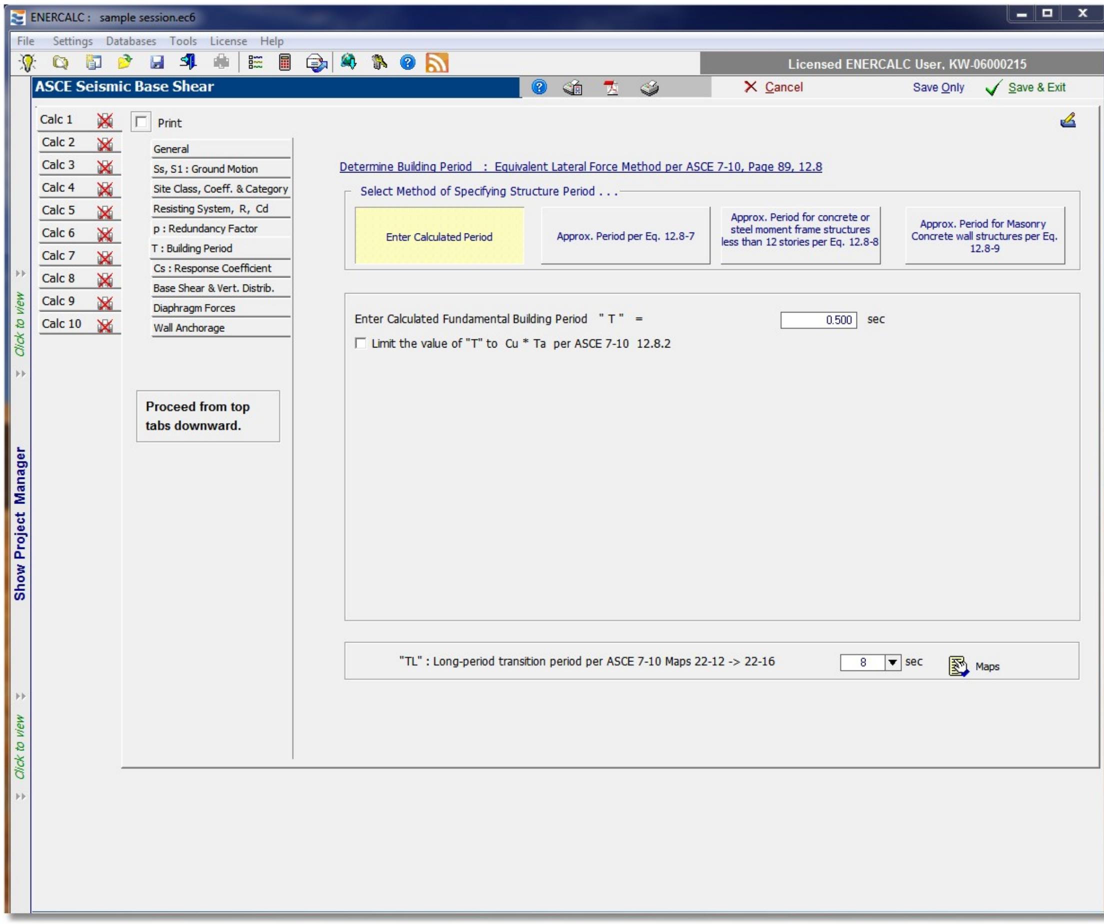
Redundancy Factor

Depending on the Seismic Design Category determined from the prior selections of Occupancy, Seismic Ground Motion and Building System, the Redundancy Factor per ASCE will be shown here.

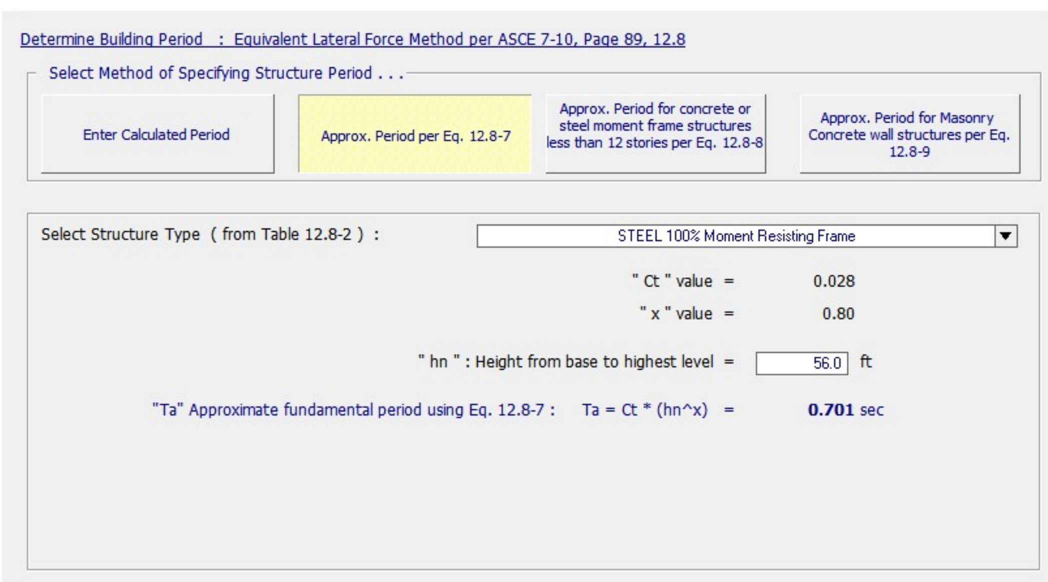


Specification of Building Period

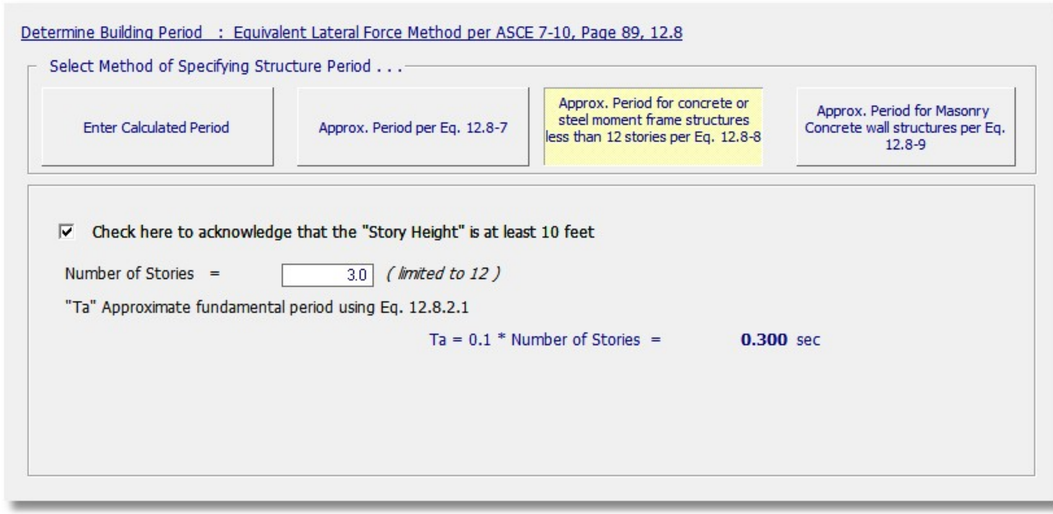
All four ASCE 7 options for building period determination are available here.



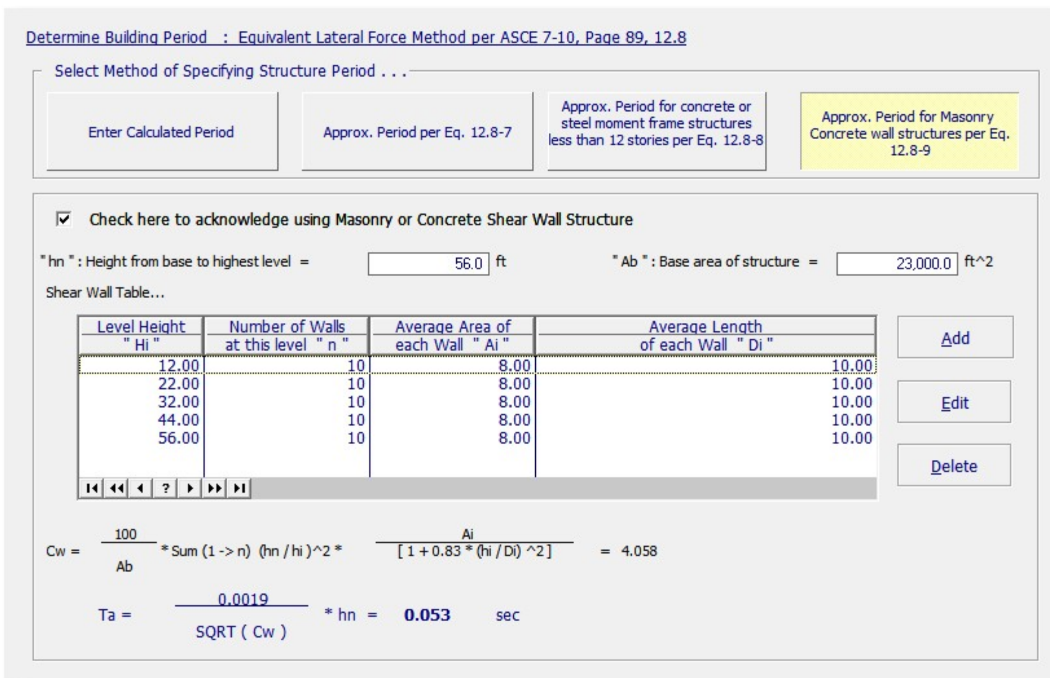
Evaluation of Approximate Building Period



Approximate Period for Steel & Concrete Frames

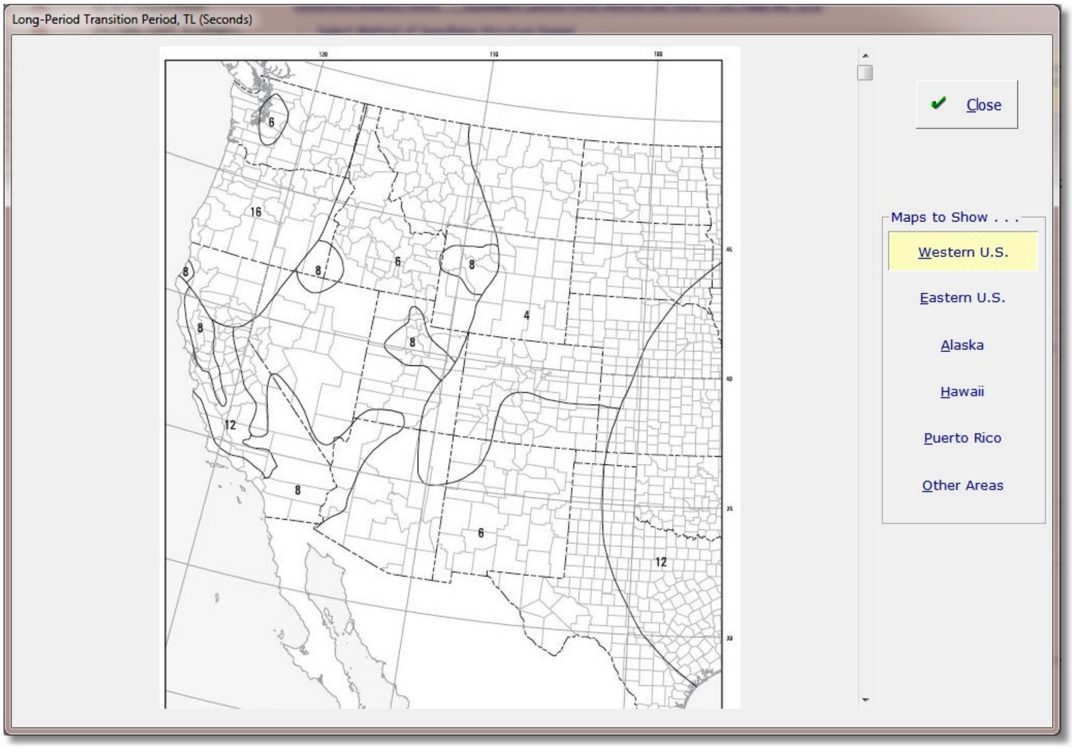


Approximate Period for Masonry Structures



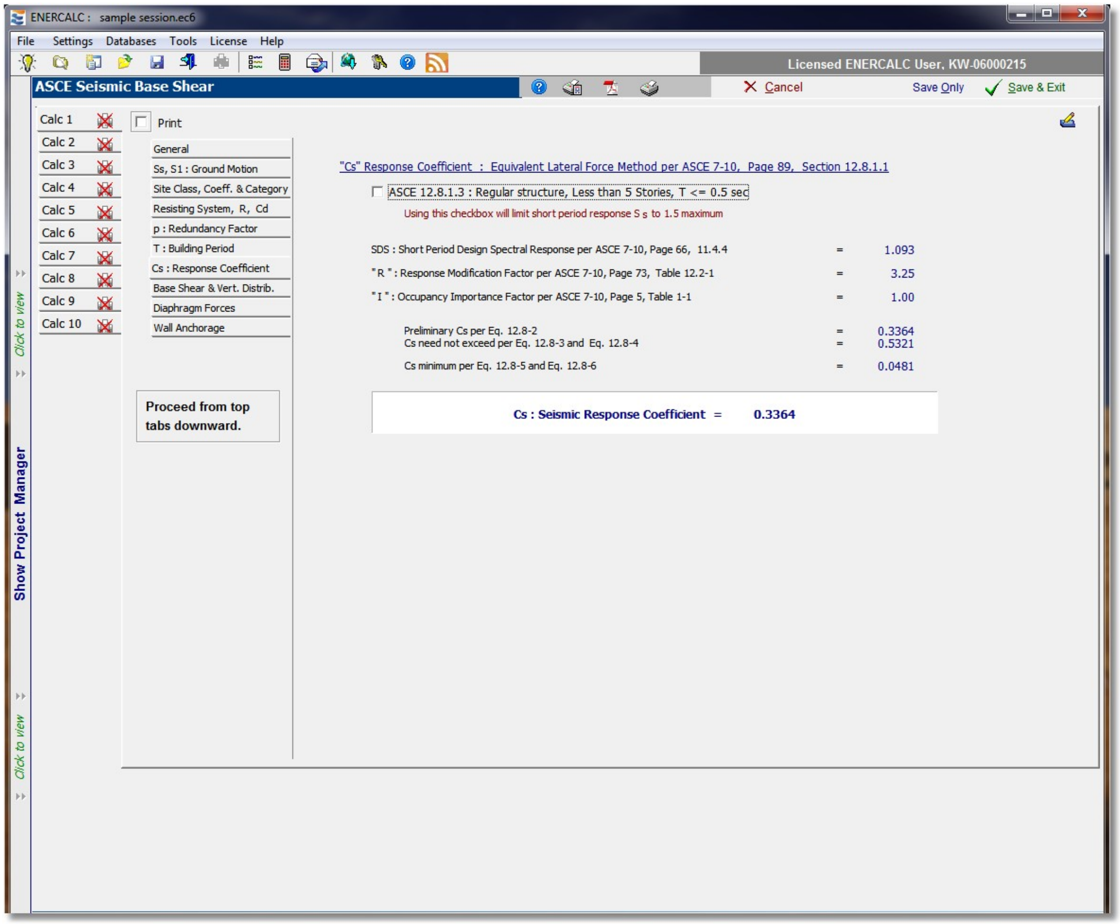
Long-Period Transition Period Reference Maps

Maps copyright ASCE.



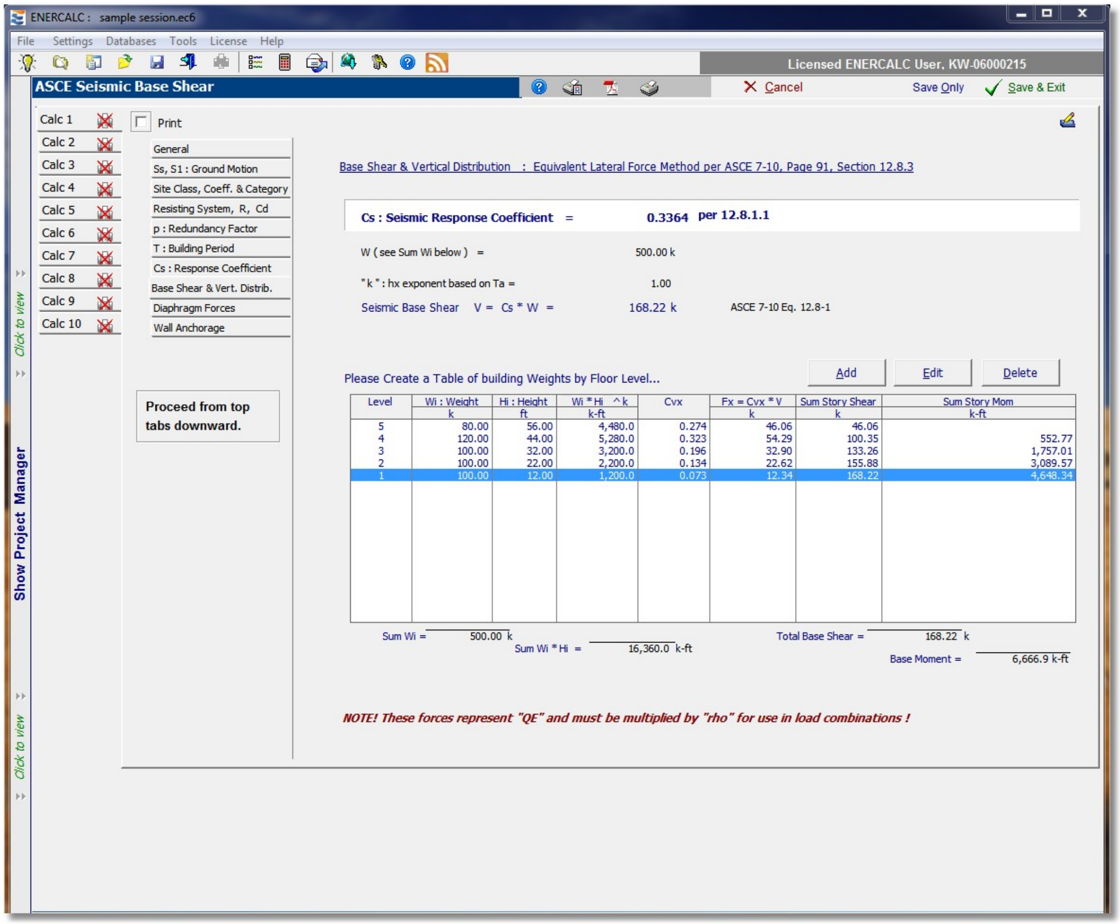
Cs: Seismic Response Coefficient

Please check ASCE 7 section references for details.



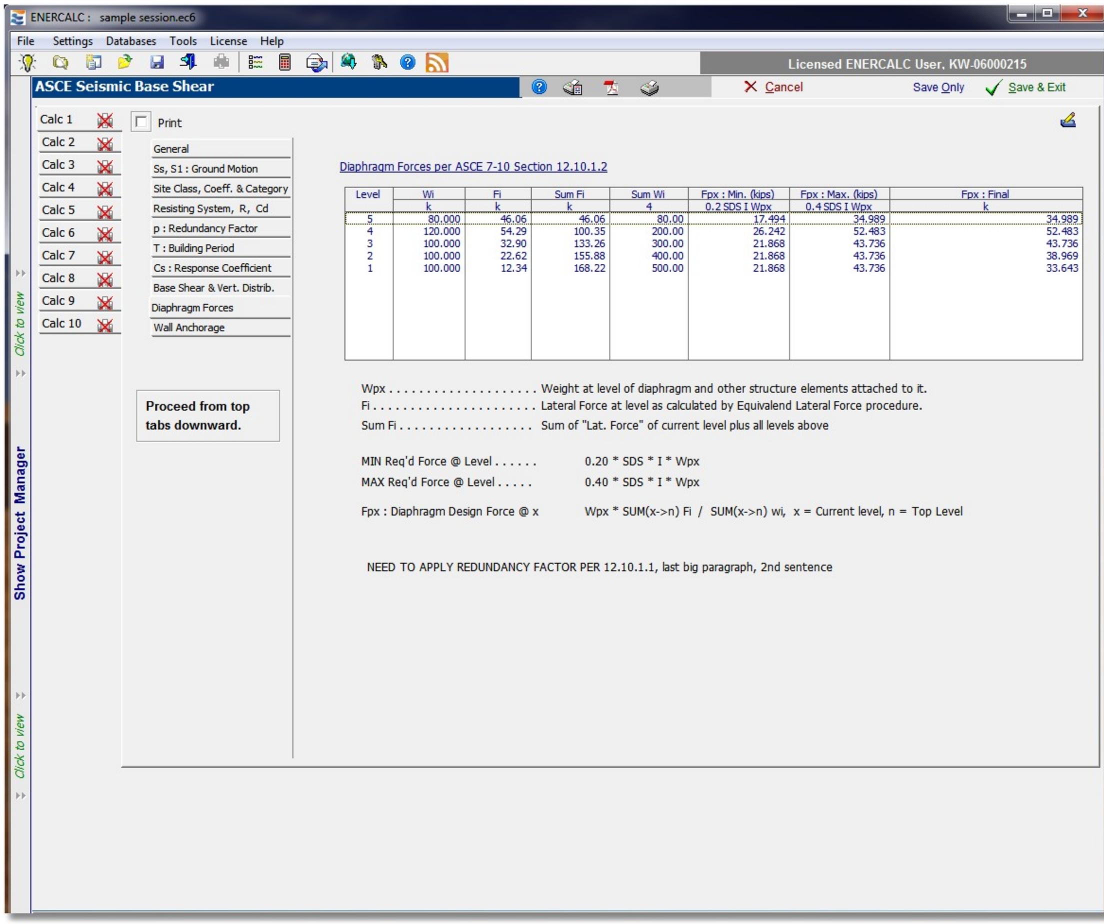
Vertical Distribution of Base Shear

This tab calculates the vertical distribution of seismic forces. Use the [**Add**] button for each new level. To edit information for a level, highlight the corresponding line and click [**Edit**].



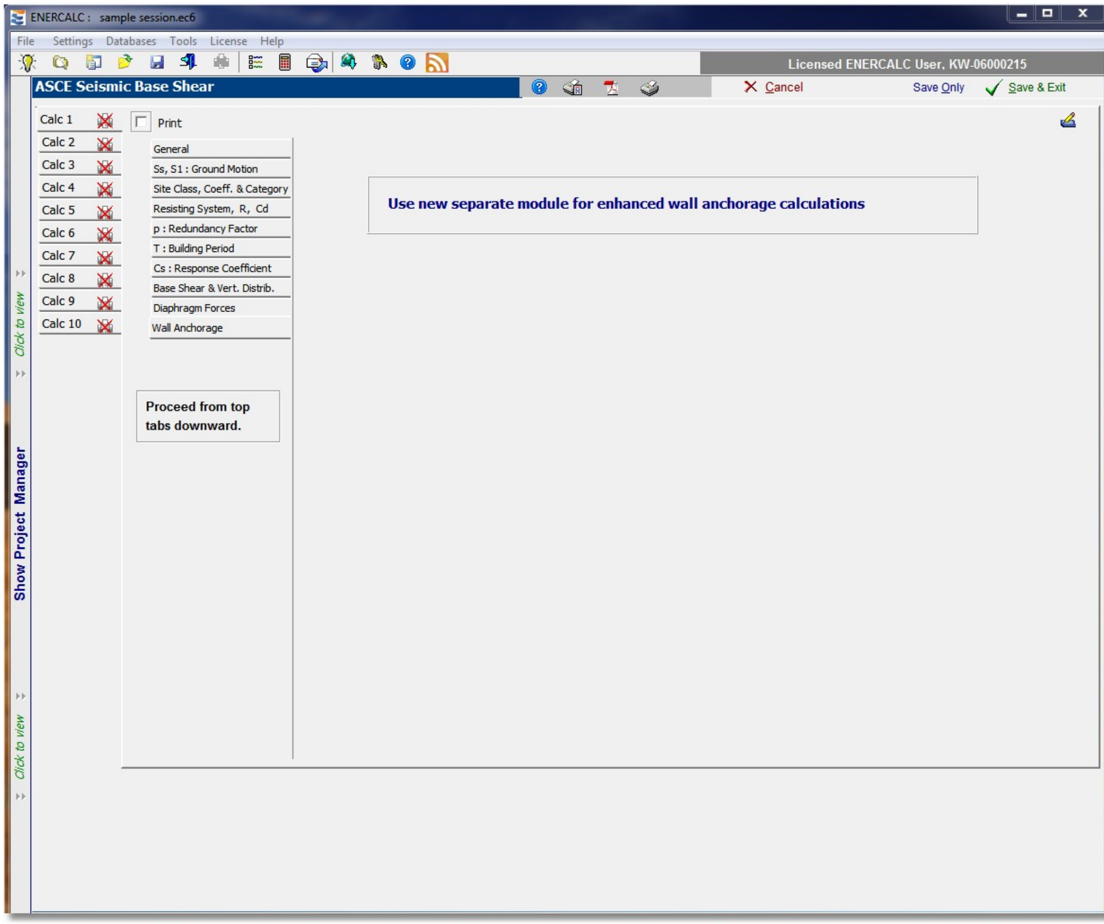
Diaphragm Forces

Using the information entered above (on the prior tab) the force distribution results are shown here.



Wall Anchorage Forces

The calculation of seismic wall anchorage forces has been enhanced and moved to a separate module.

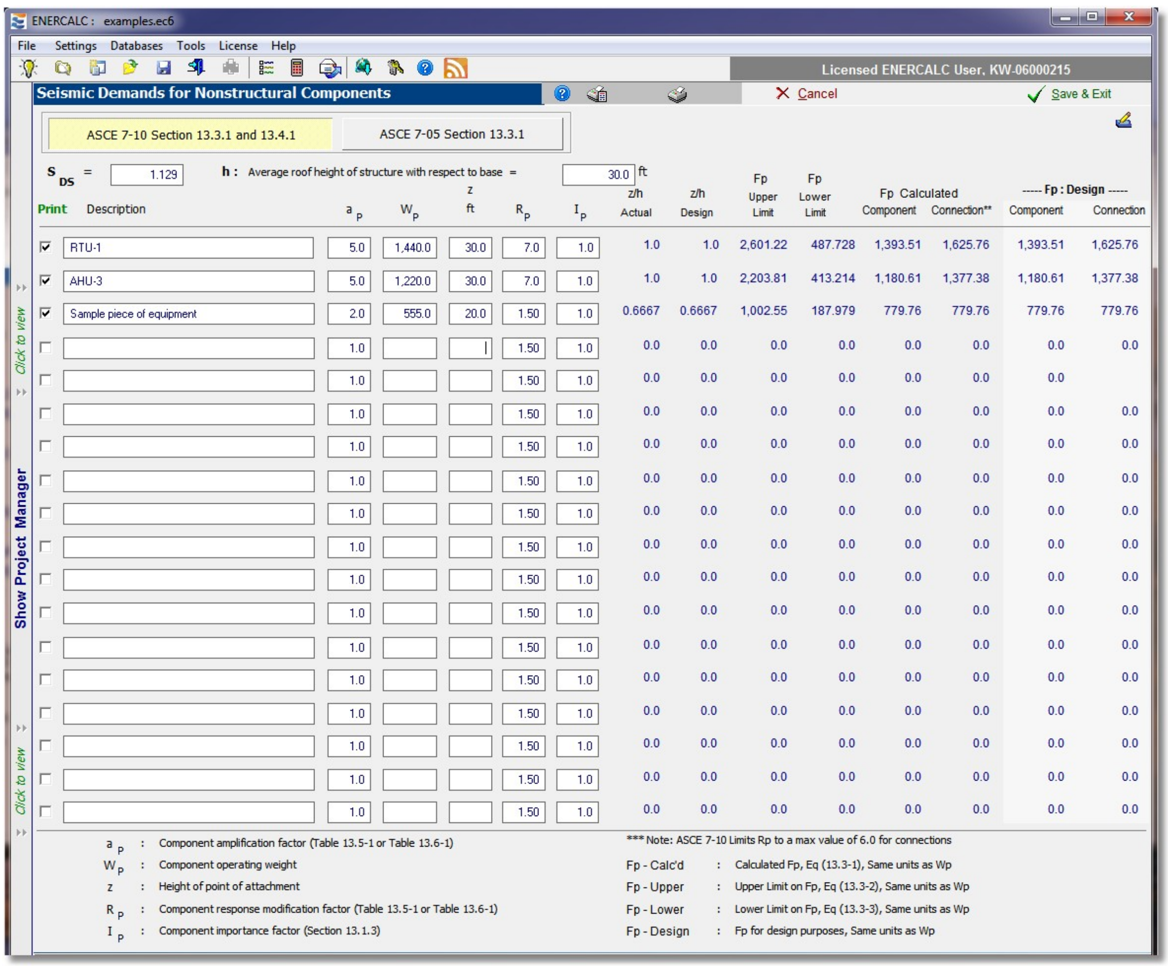

ENERCALC : sample session.ec6
File Settings Databases Tools License Help
Licensed ENERCALC User, KW-06000215
ASCE Seismic Base Shear
Cancel
Save Only
Save & Exit
Calc 1
Calc 2
Calc 3
Calc 4
Calc 5
Calc 6
Calc 7
Calc 8
Calc 9
Calc 10
Print
General
Ss, S1 : Ground Motion
Site Class, Coeff. & Category
Resisting System, R, Cd
p : Redundancy Factor
T : Building Period
Cs : Response Coefficient
Base Shear & Vert. Distrib.
Diaphragm Forces
Wall Anchorage
Use new separate module for enhanced wall anchorage calculations
Proceed from top tabs downward.
Show Project Manager
Click to view
Click to view

10.1.2 ASCE Seismic Demands on Nonstructural Components

This module is an implementation of the provisions for seismic demands on nonstructural components as per ASCE 7. The module currently supports calculations based on ASCE 7-05 and ASCE 7-10. Input parameters are taken directly from the nomenclature presented in ASCE 7.

The ASCE 7-05 version of this calculation produces component design forces using the defined parameters.

The ASCE 7-10 version of this calculation produces both component and connection design forces using the defined parameters.



10.1.3 ASCE Seismic Wall Anchorage

This module is an implementation of the provisions for design and anchorage of walls to resist seismic forces as per ASCE 7. The module currently supports calculations based on ASCE 7-05 and ASCE 7-10. Input parameters are taken directly from the nomenclature presented in ASCE 7.

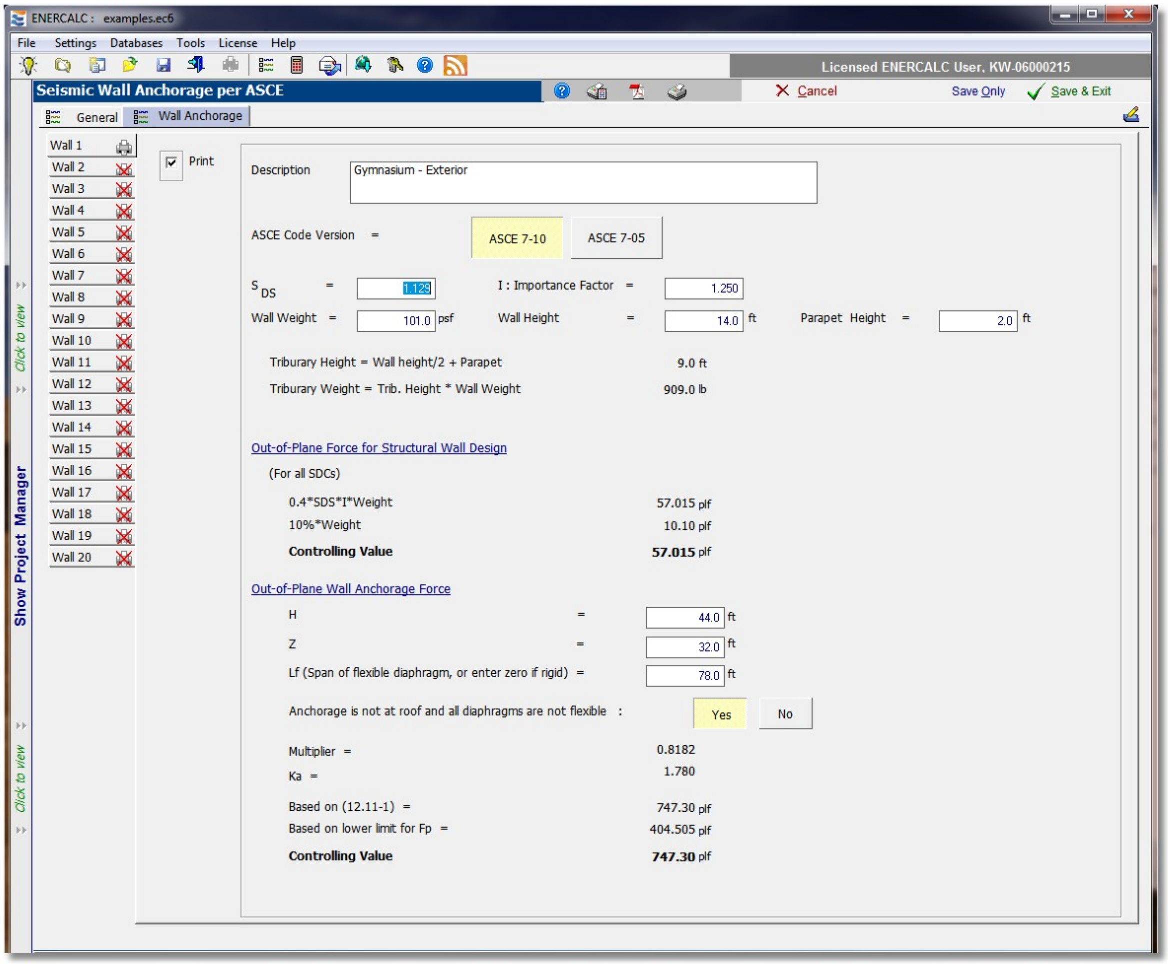
The ASCE 7-05 version of this calculation produces:

- Out-of-Plane Force for Structural Wall Design
- Out-of-Plane Force for Concrete and Masonry Structural Wall Anchorage Design (for all Seismic Design Categories where structural walls connect to rigid diaphragms, and for SDC A and B where walls connect to flexible diaphragms)
- Out-of-Plane Force for Concrete and Masonry Structural Wall Anchorage Design (for Seismic Design Categories C through F where walls connect to flexible diaphragms)

The ASCE 7-10 version of this calculation produces:

- Out-of-Plane Force for Structural Wall Design
- Out-of-Plane Wall Anchorage Force

The module is configured to allow the definition of up to 20 different wall conditions. The user has the option to decide which (if any) of the calculations will be included in a Project Print.


ENERCALC : examples.ec6
File Settings Databases Tools License Help
Licensed ENERCALC User, KW-06000215
Seismic Wall Anchorage per ASCE
Cancel
Save Only
Save & Exit
General
Wall Anchorage
Wall 1
Wall 2
Wall 3
Wall 4
Wall 5
Wall 6
Wall 7
Wall 8
Wall 9
Wall 10
Wall 11
Wall 12
Wall 13
Wall 14
Wall 15
Wall 16
Wall 17
Wall 18
Wall 19
Wall 20
Print
Description
Gymnasium - Exterior
ASCE Code Version =
ASCE 7-10
ASCE 7-05
S DS =
1.129
I : Importance Factor =
1.250
Wall Weight =
101.0 psf
Wall Height =
14.0 ft
Parapet Height =
2.0 ft
Triburary Height = Wall height/2 + Parapet
9.0 ft
Triburary Weight = Trib. Height * Wall Weight
909.0 lb
Out-of-Plane Force for Structural Wall Design
(For all SDCs)
0.4*SDS*I*Weight
57.015 plf
10%*Weight
10.10 plf
Controlling Value
57.015 plf
Out-of-Plane Wall Anchorage Force
H =
44.0 ft
Z =
32.0 ft
Lf (Span of flexible diaphragm, or enter zero if rigid) =
78.0 ft
Anchorage is not at roof and all diaphragms are not flexible :
Yes
No
Multiplier =
0.8182
Ka =
1.780
Based on (12.11-1) =
747.30 plf
Based on lower limit for Fp =
404.505 plf
Controlling Value
747.30 plf
Show Project Manager
Click to view
Click to view

10.1.4 ASCE Wind Enclosure

This module is an implementation of the provisions for determining the Enclosure Classification for the determination of Wind Loads as per ASCE 7. Click here for a video: ASCE 7 Wind Enclosure Classification

The module currently supports calculations based on ASCE 7-05 and ASCE 7-10. Input parameters are taken directly from the nomenclature presented in ASCE 7.

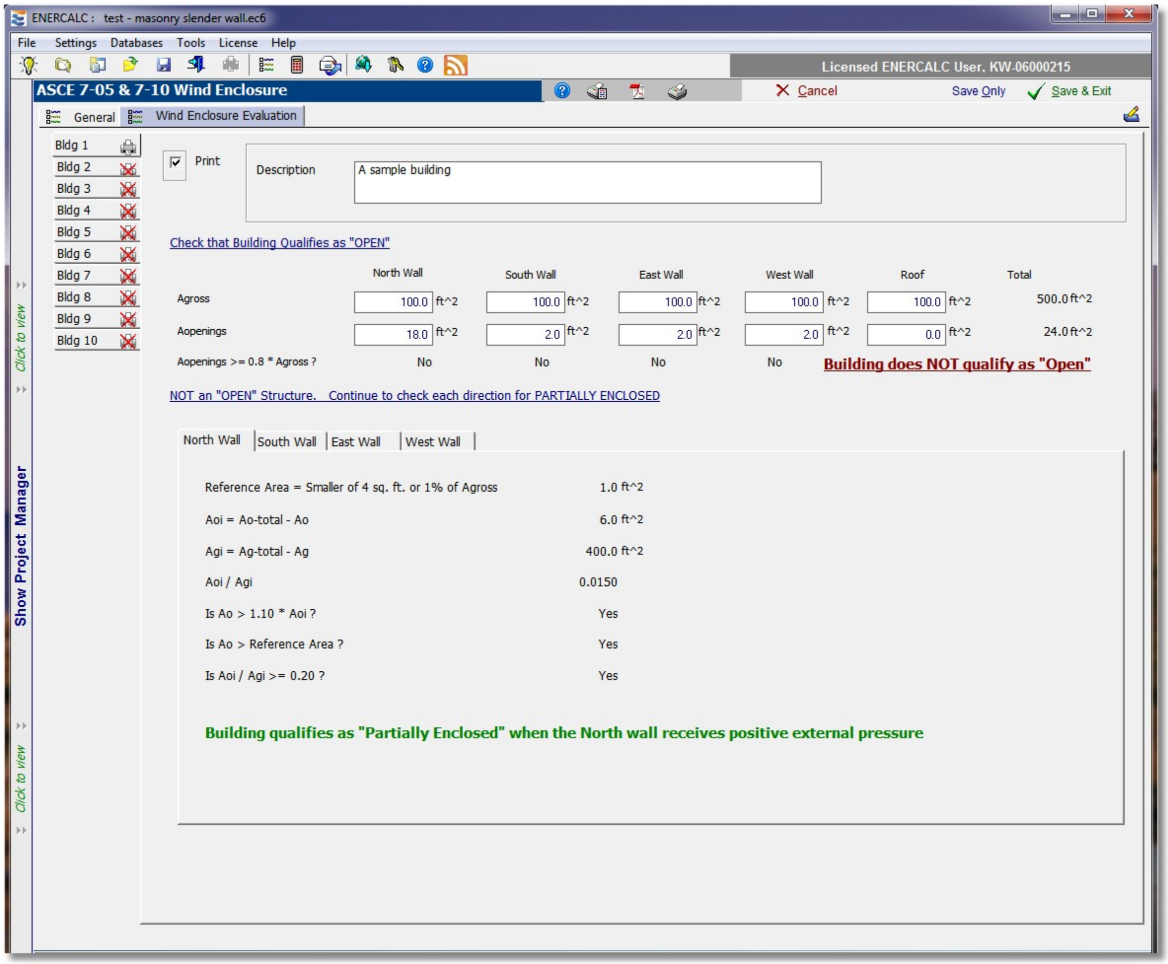
The calculation sheet is set up to permit up to ten different analyses to be documented within a single Project File. This can be useful when a project includes multiple buildings, or in situations where multiple scenarios are being evaluated.

The data collection is presented in such a way as to guide the user through the module in a logical progression.

Values of gross area and area of openings are provided for walls on four sides of the building (arbitrarily designated "North", "South", "East", and "West"). Once these values are provided, the module can determine if the structure qualifies as "Open" or not. If the structure is determined to be "Open", then the module will indicate that result and stop there. If the structure does *not* qualify as an "Open" structure, then input fields for Roof Gross Area and Roof Opening Area will be displayed, and a set of four tabs will appear in the bottom half of the screen. Once the Roof areas are populated, the four tabs thoroughly present the calculations to determine whether the structure qualifies as "Enclosed" or "Partially Enclosed" when each of the four respective walls receives positive external pressure.

The results of this determination are useful for project documentation requirements, as well as in guiding the user to read the appropriate results values from other modules within Structural Engineering Library.

Note: The provisions of the ASCE Wind Enclosure module are already incorporated into the ASCE 7-10 Wind Loads module. So it is not necessary to run the standalone version of this module if the ASCE 7-10 Wind Loads module will be used.


ENERCALC : test - masonry slender wall.ec6
File Settings Databases Tools License Help
Licensed ENERCALC User, KW-06000215
ASCE 7-05 & 7-10 Wind Enclosure
Cancel
Save Only
Save & Exit
General
Wind Enclosure Evaluation
Bldg 1
Bldg 2
Bldg 3
Bldg 4
Bldg 5
Bldg 6
Bldg 7
Bldg 8
Bldg 9
Bldg 10
Print
Description
A sample building
Check that Building Qualifies as "OPEN"
North Wall
South Wall
East Wall
West Wall
Roof
Total
Agross
100.0 ft^2
100.0 ft^2
100.0 ft^2
100.0 ft^2
100.0 ft^2
500.0 ft^2
Aopenings
18.0 ft^2
2.0 ft^2
2.0 ft^2
2.0 ft^2
0.0 ft^2
24.0 ft^2
Aopenings >= 0.8 * Agross ?
No
No
No
No
Building does NOT qualify as "Open"
NOT an "OPEN" Structure. Continue to check each direction for PARTIALLY ENCLOSED
North Wall
South Wall
East Wall
West Wall
Reference Area = Smaller of 4 sq. ft. or 1% of Agross
1.0 ft^2
Aoi = Ao-total - Ao
6.0 ft^2
Agi = Ag-total - Ag
400.0 ft^2
Aoi / Agi
0.0150
Is Ao > 1.10 * Aoi ?
Yes
Is Ao > Reference Area ?
Yes
Is Aoi / Agi >= 0.20 ?
Yes
Building qualifies as "Partially Enclosed" when the North wall receives positive external pressure
Show Project Manager
Click to view
Click to view

10.1.5 ASCE 7-10/7-16 Wind Forces, Chapter 27, Part 1

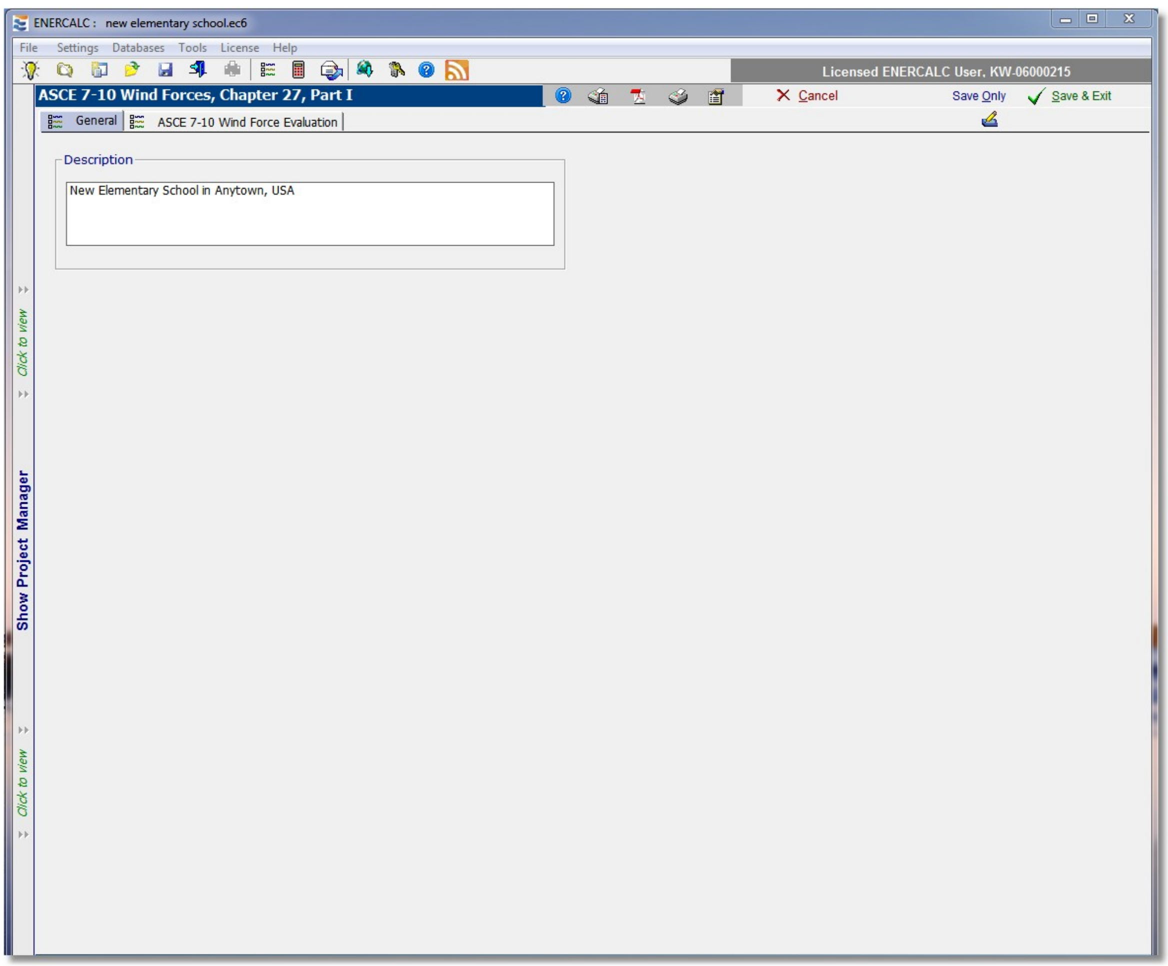
Need more? Ask Us a Question

This module is a presentation of the Wind Forces provisions of Chapter 27, Part 1 of ASCE 7-10 and ASCE 7-16. Click here for a video: ASCE 7-10 Wind Forces

Limited documentation is provided here, because all of the references to ASCE 7 are given on the module screens.

General

The General tab provides an input field for a general description of the project and/or the wind calculations that are being performed.



ASCE 7-10 / 7-16 Wind Force Evaluation

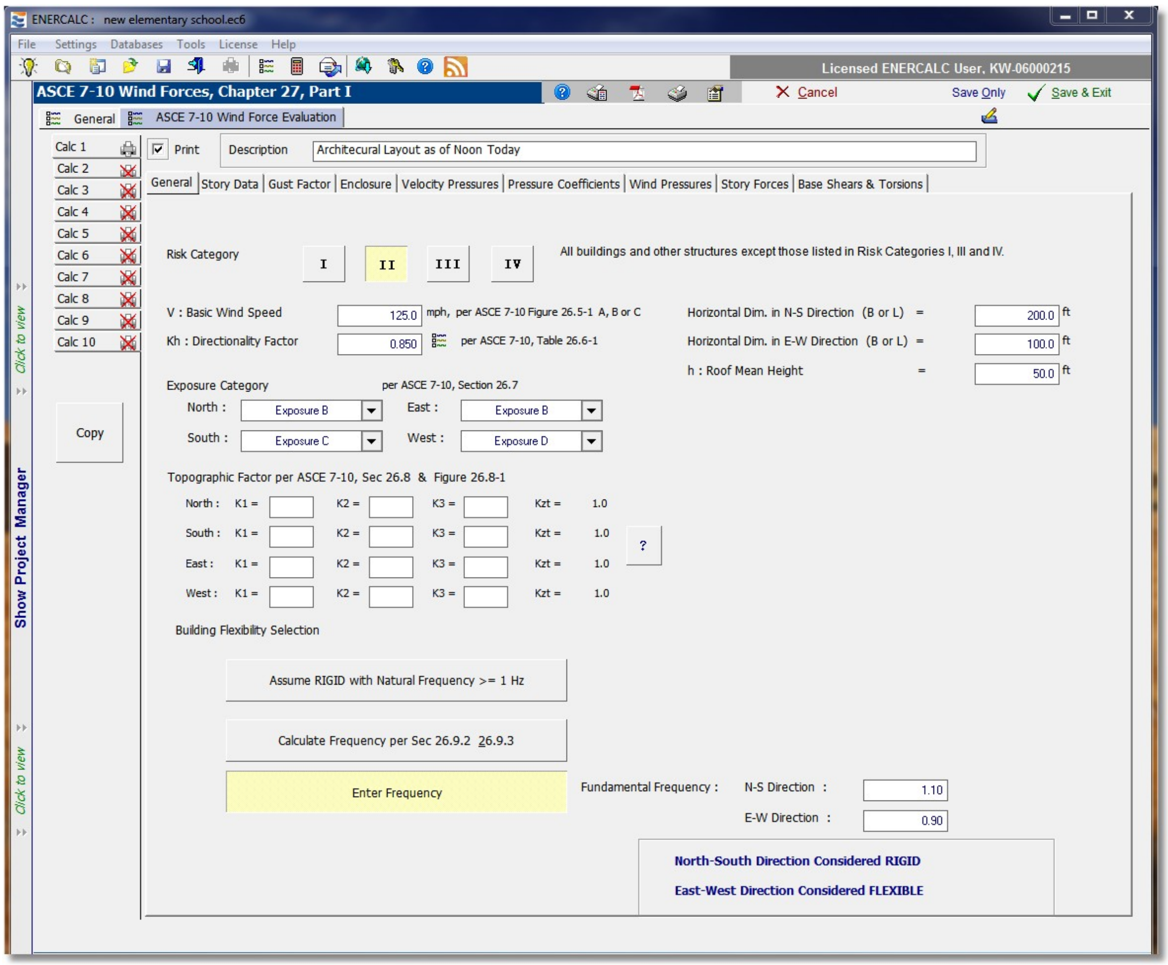
This tab provides access to a series of Calculation tabs that can be used to store up to ten separate wind calculations. These might be useful for studying different project sites, different architectural concepts, or even evaluating separate buildings in a project.

The Description field can be used to distinguish from among the different wind load calculations that might be defined in a single Project File.

The Print checkbox specifies whether or not the particular wind load calculation will or will not be included when a Print command is clicked or when a Project Print is performed.

General (sub-tab of the ASCE 7-10 / 7-16 Wind Force Evaluation tab)

This tab collects basic data such as Risk Category, Basic Wind Speed, Directionality Factor, Building Dimensions, Exposure Category, Topographic Factor, and information to determine how the building frequency will be determined.



The Risk Category is only for reference in documenting the design. It no longer influences the importance factor, but it does dictate which Wind Speed map to use to determine the Basic Wind Speed.

The Exposure Category is dependent upon the upwind characteristics. Therefore, as different building elevations become the windward face of the structure, it is possible that the Exposure Category will change. For this reason, the program allows the Exposure Category to be defined separately for each building face in turn becoming the windward face.

The Building Flexibility Selection offers three different options for defining the frequency of the building.

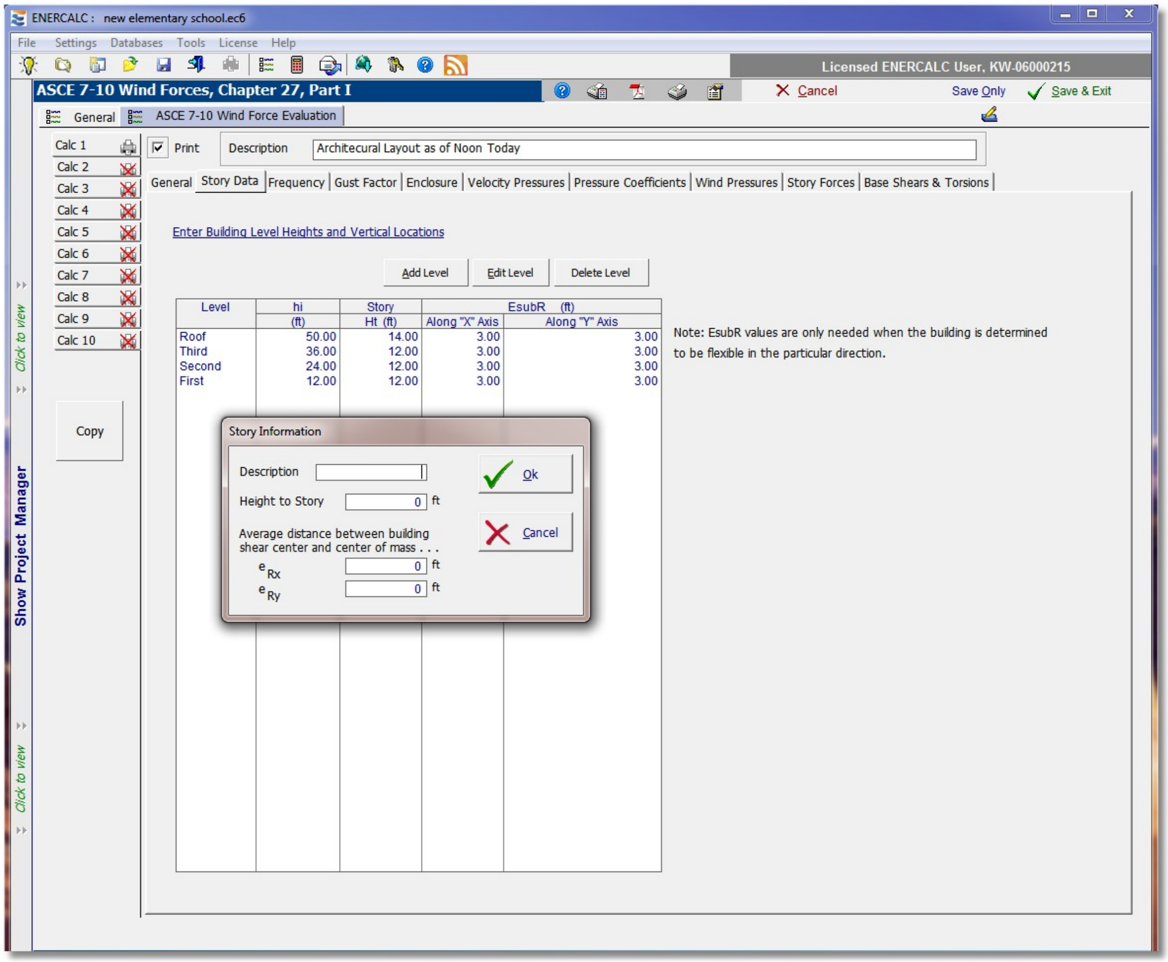
- The first option declares that the building will be assumed to be rigid (having a frequency greater than or equal to 1 Hz).
- The second option applies the prescriptive methods in Sections 26.9.2 and 26.9.3 of ASCE 7-10 to approximate the frequency of the building based on construction type/layout and mean roof height.
- The third option allows the user to explicitly specify the frequency in the North-South direction and in the East-West direction.

Note that if the second option is selected, a new tab named "Frequency" will be introduced between the "Story Data" tab and the "Gust Factor" tab.

This tab contains the only difference between ASCE 7-10 and ASCE 7-16, which is the introduction of Ke, the Ground Elevation Factor in ASCE 7-16.

Story Data

This tab collects the data required to define the vertical locations of the stories with respect to finished grade.

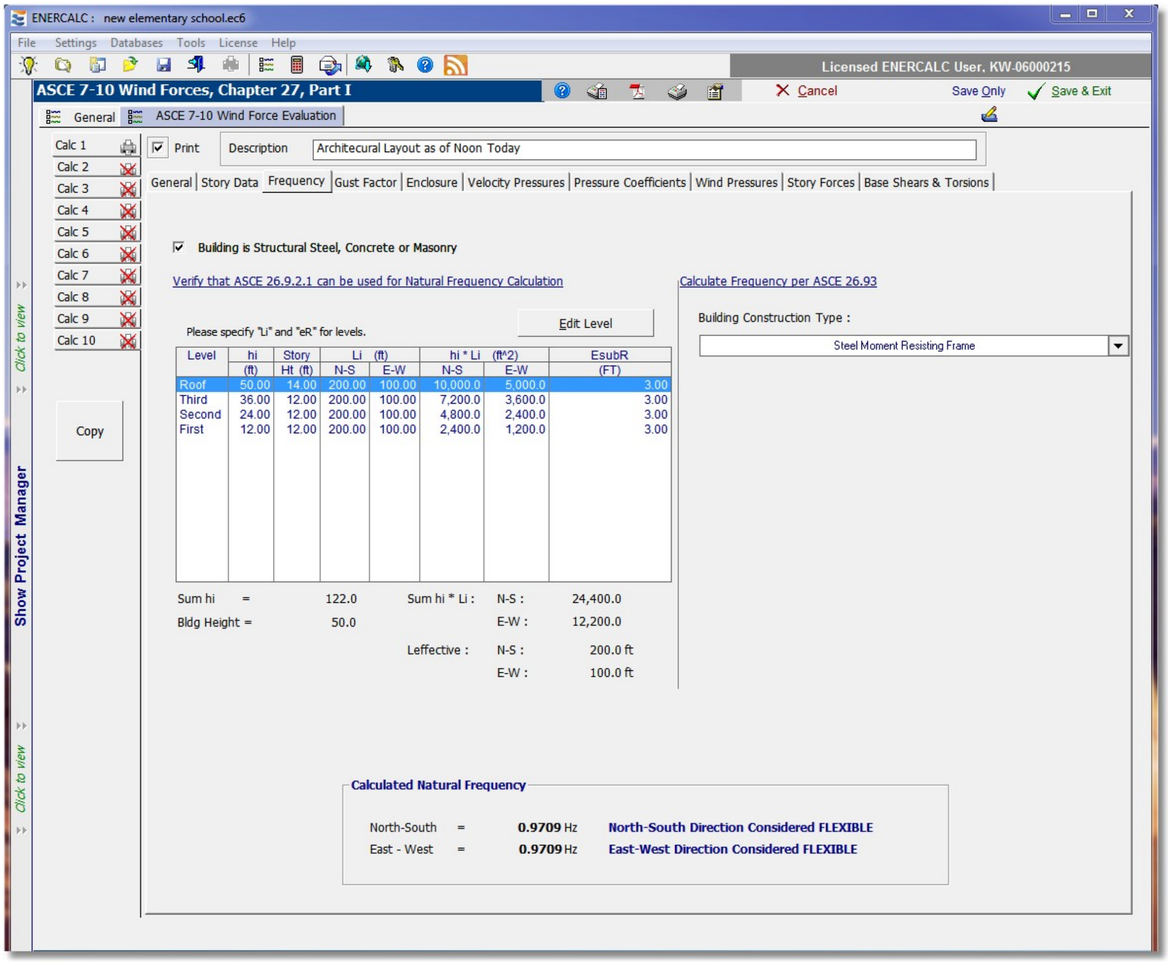


The development of the story data table is controlled by the three buttons: Add Level, Edit Level, and Delete Level, which perform their respective operations on the table of story data.

Clicking Add or Edit opens the Story Information pop-up dialog as shown above. Two items are worthy of note on this dialog. First, the Height to Story is always referenced from grade, so it is looking to collect the height from grade to the story of interest, not the story to story height. Second, the values of e_{Rx} and e_{Ry} are only used if a building is determined (or defined) to be "Flexible" (< 1 Hz) in a particular direction. So if a building turns out to be "Rigid" in one or both directions, then the values of e_{Rx} and e_{Ry} are not required in those directions, and the values can be left at zero or a value can be entered, but the program will ignore it.

Frequency (Only displayed if the option is selected on the General tab to "Calculate Frequency")

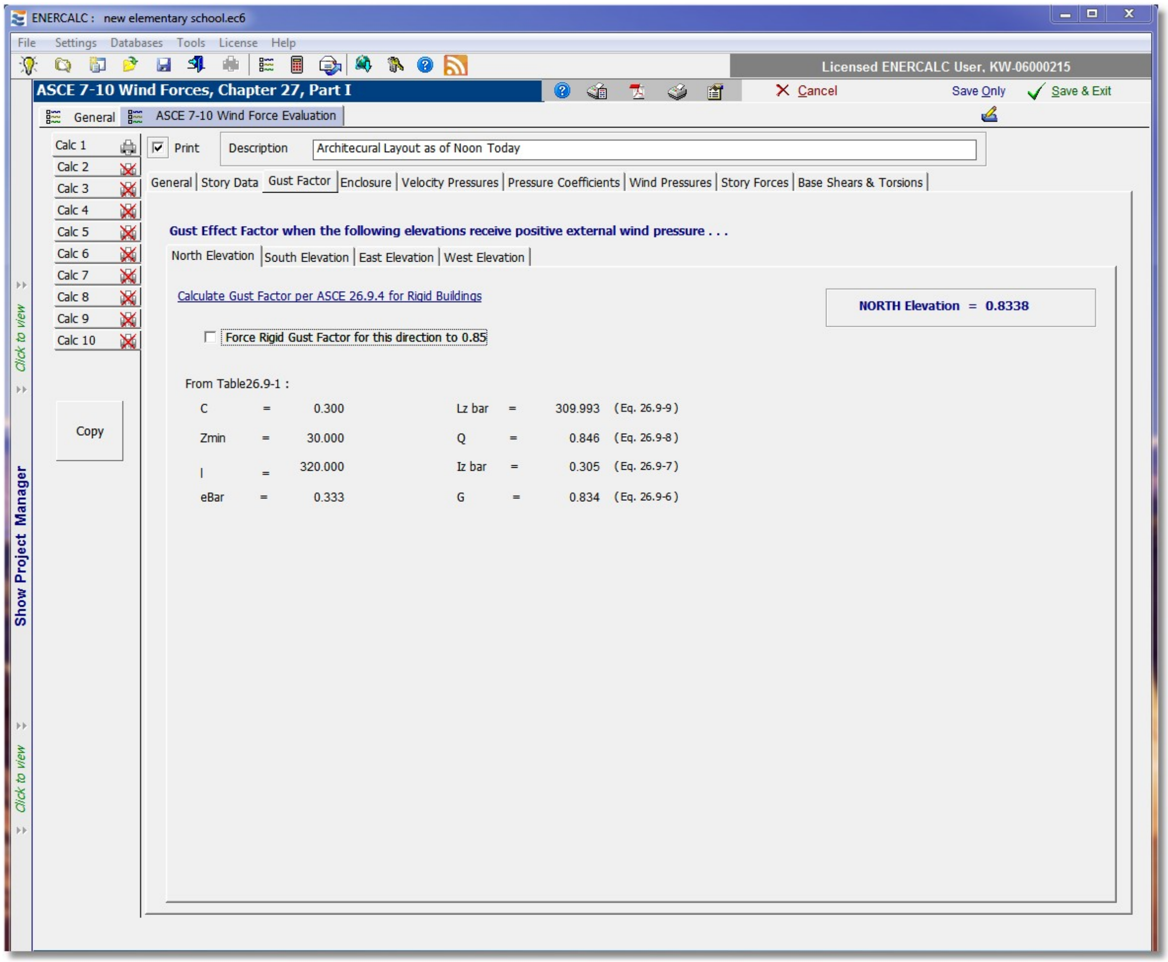
This tab displays the story data that was previously collected, and it requests the Building Construction Type and the values of Li necessary to complete the check to determine if the approximate methods of frequency determination are applicable.



If the approximate methods of frequency determination are found to be applicable, this tab reports the approximate frequency. Otherwise, a message will be displayed to indicate that the approximate methods are not applicable, and that the building frequency must be determined another way.

Gust Factor

This tab displays the results of the Gust Factor determination for each of the four elevations, in turn, being the face of the building that receives positive external pressure.



The North, South, East, and West tabs present the calculation and the resulting Gust Factor that will be used in subsequent calculations when the respective building elevation receives positive external pressure.

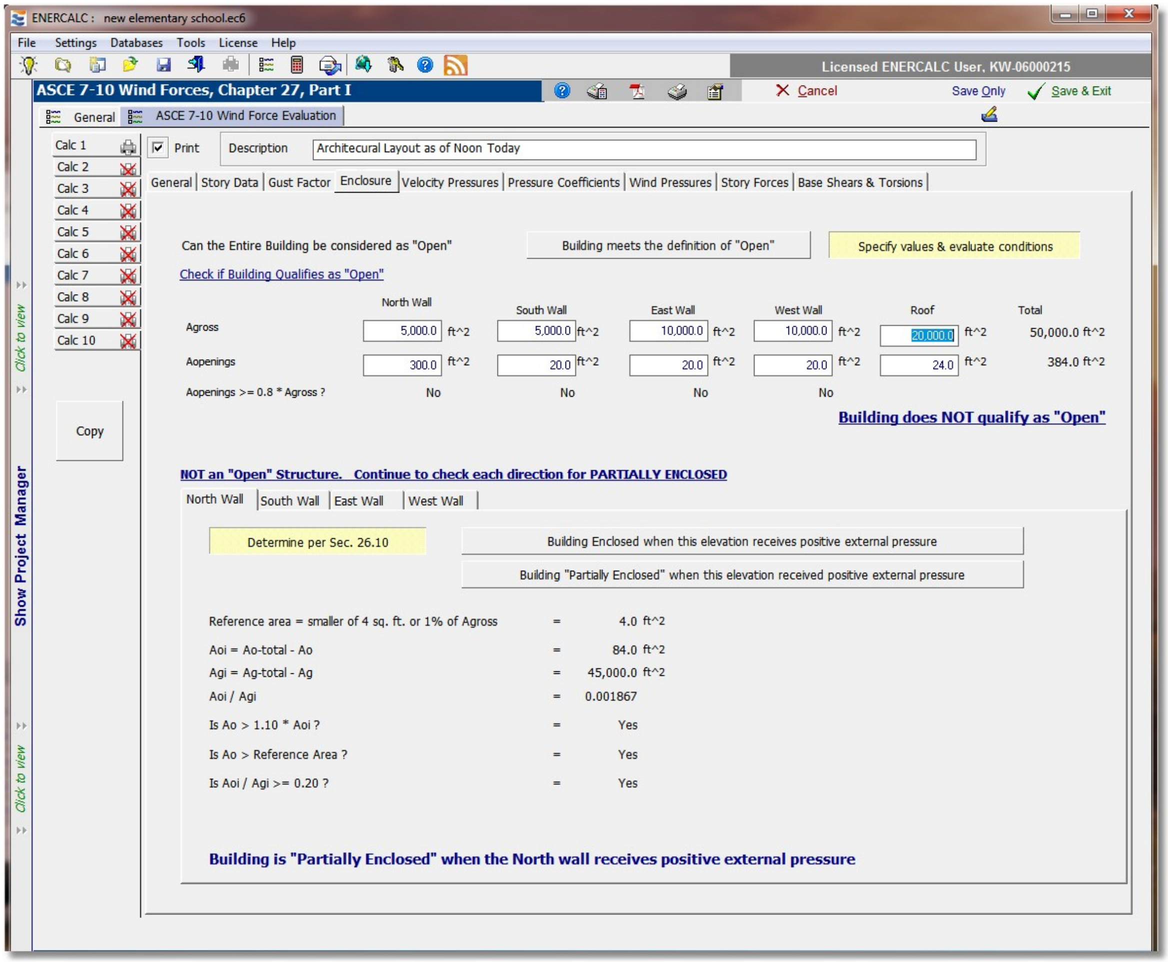
When the building is categorized as "Rigid" in a particular direction, the program offers the checkbox option to force the value of the Gust Factor to 0.85 when the referenced elevation receives positive external pressure.

When the building is categorized as "Flexible" in a particular direction, the program requires one more value, which is the damping ratio. This value is collected in an input box on the respective direction tab when appropriate, and the value is incorporated into the Gust Factor to be used when the referenced elevation receives positive external pressure.

It is worth noting that many of the parameters used in the calculation of the Gust Factor are dependent upon the Exposure category. Since the Exposure category can vary for each of the four cardinal directions, it is possible that the Gust Factor for use when the ***North*** elevation of the building receives positive external pressure may actually be different than the Gust Factor for use when the ***South*** elevation of the building receives positive external pressure.

Enclosure

This tab displays the results of the Enclosure determination for each of the four elevations, in turn, being the face of the building that receives positive external pressure.



The upper half of this tab is dedicated to evaluating the building to determine whether or not it qualifies as an "Open" structure. The module collects the gross areas of each of the four walls along with the areas of opening in each of the four walls. Based on the data provided by the user, the module performs the calculations and checks the criteria to see if the building qualifies as an "Open" structure. If it does, then the module reports that result. If the building does NOT qualify as an "Open" structure" then additional input fields are displayed to collect the gross area of the Roof, and the area of openings in the roof, and the workflow continues to determine whether the building qualifies as "Enclosed" or "Partially Enclosed". This evaluation takes place four times, considering each of the four walls to be the windward wall, in turn. The intermediate calculations are performed and the results are reported on each of the four wall tabs.

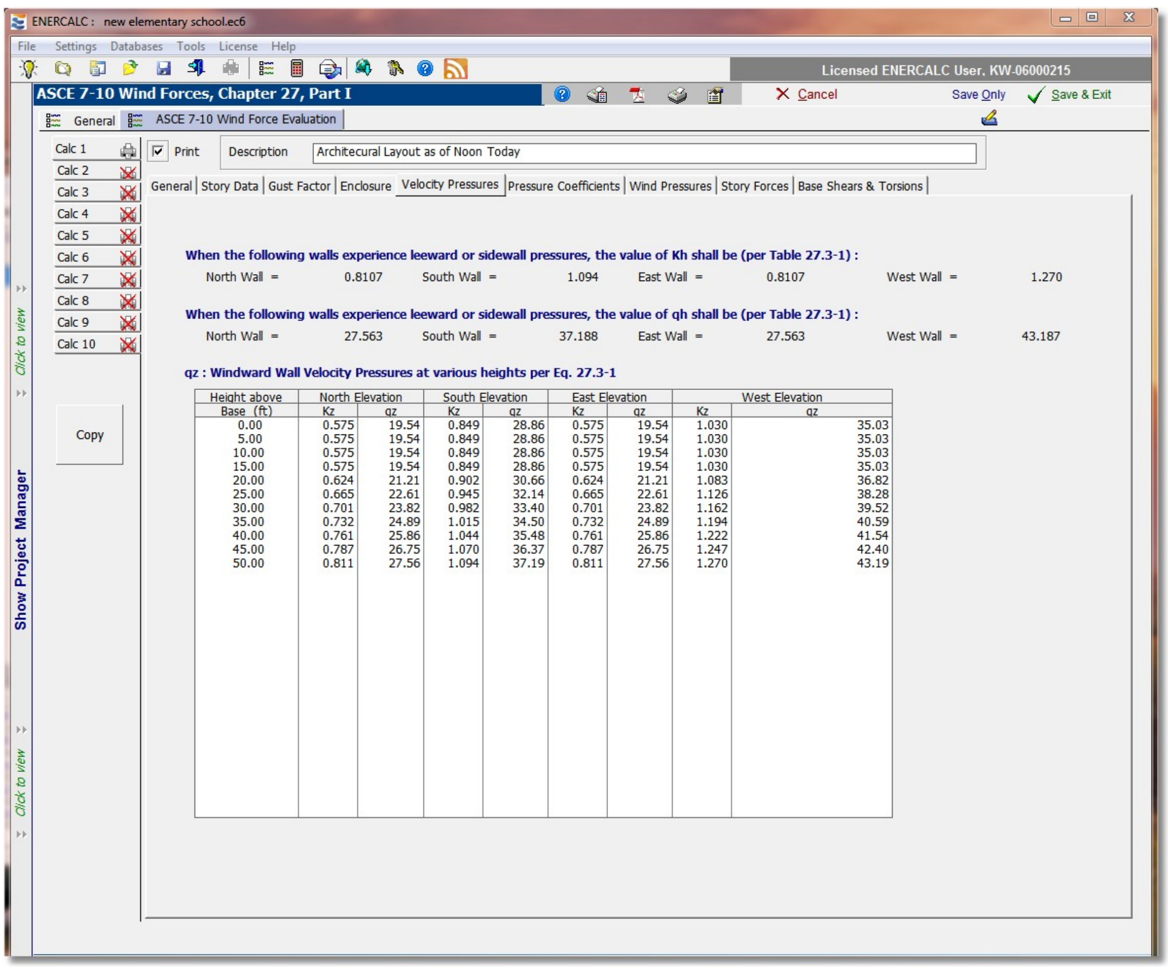
It is worth noting that a building could potentially be considered "Partially Enclosed" when some of its elevations receive positive external pressure, and "Enclosed" when its other elevations receive positive external pressure.

It is also worth noting that convenience buttons have been provided on each of the four wall tabs to allow the user to simply declare the building to be "Enclosed" or "Partially Enclosed" when the selected elevation receives positive external pressure. These have been implemented for situations where the evaluation has already been performed and/or the user is already confident in making a decision by judgment.

The Enclosure classification is used downstream to select appropriate values of GCpi to use when each of the four elevations becomes the windward wall.

Velocity Pressures

This tab displays the results of the Velocity Pressure determination for the various walls when each wall is under leeward, sidewall, and windward wall conditions.



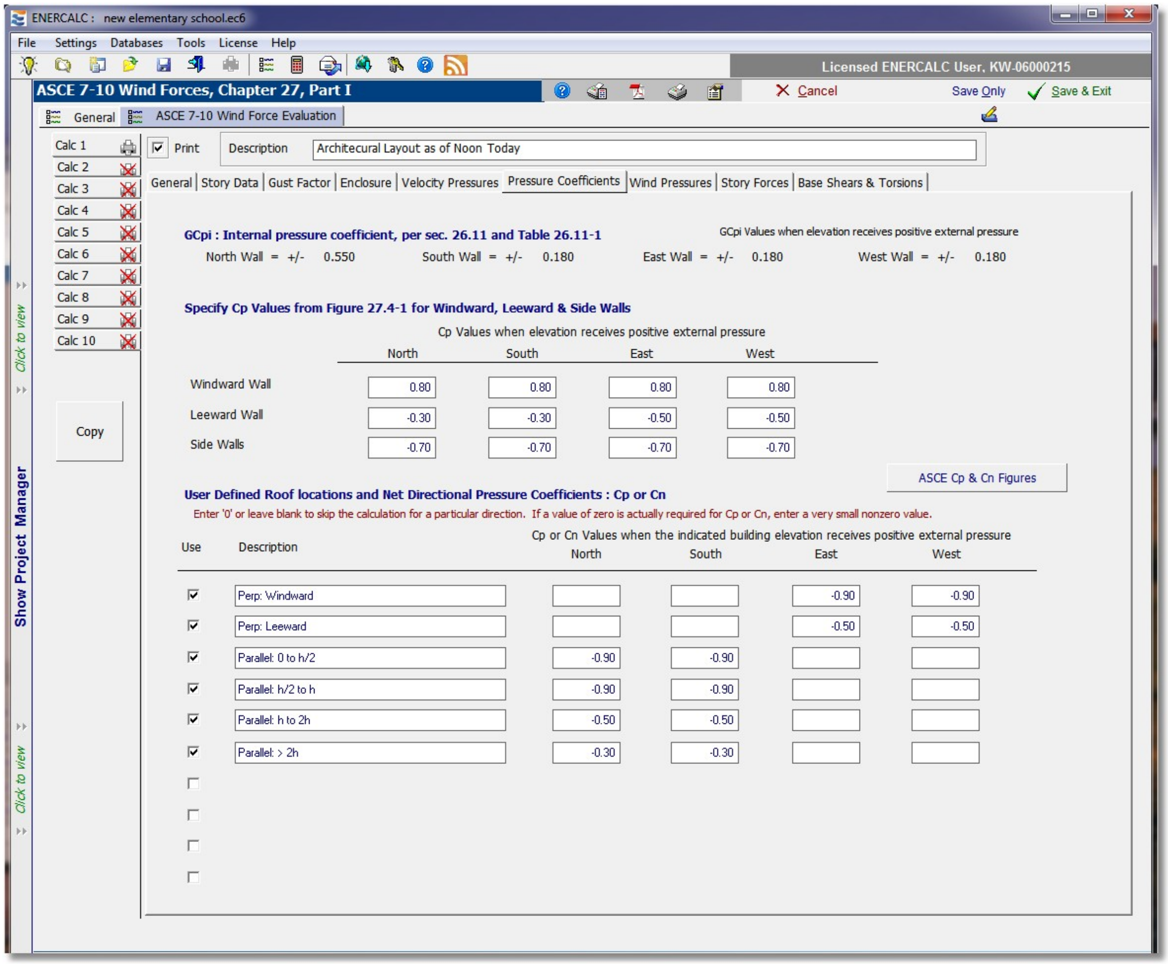
The first row of data reports the values of Kh that are applicable when each of the walls experiences leeward or sidewall pressures.

The second row of data reports the resulting values of qh that are applicable when each of the walls experiences leeward or sidewall pressures.

Next, a table is presented that reports the values of Kz and the resulting values of qz that are applicable, as a function of height, when each of the walls experiences windward wall pressures.

Pressure Coefficients

This tab reports the values of GCpi that are applicable when each of the respective elevations receives positive external pressure. The remainder of this tab is dedicated to collecting the values of Cp or Cn as appropriate for the various surfaces of the building.



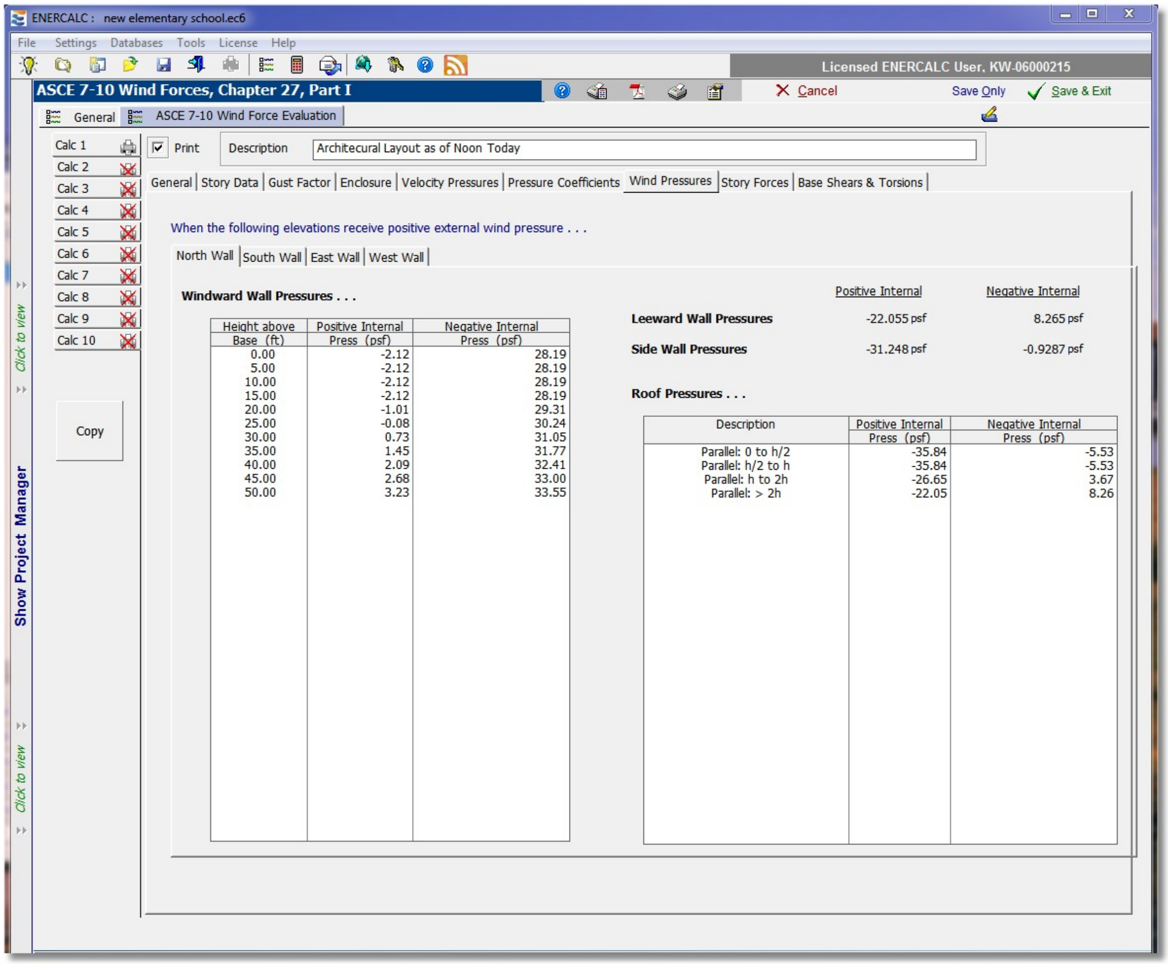
Wall values are collected first, and the input fields collect values of Cp that will be used when each of the four walls is a windward wall, a sidewall, or a leeward wall.

At the bottom of the tab is a customizable table that is set up to receive many lines of Cp or Cn values pertinent to the roof. A convenience button is provided to display the figures that define the Cp or Cn values for various conditions. A glance at the tables reveals that the factors for roofs are frequently dependent upon the wind direction with respect to the ridge, and also dependent upon whether pressures are desired for the windward or leeward surfaces of the roof. So some planning will be required to set this table up to yield the desired results. To make the downstream results most applicable and easy to read, the program has been set up so that a roof pressure value will only be calculated for situations where nonzero values of Cp or Cn have been specified. To put this another way, referring to the table above, there are no values of Cp defined for the North or South elevations for the "Perpendicular: Windward" or "Perpendicular: Leeward" conditions. This is because the ridge is assumed to run in the North-South direction in the hypothetical building being considered. As such, it would not make sense to ask the program to report windward or leeward roof pressures when the wind acts in the north or south directions. So to avoid overpopulating the output with meaningless results, the blank fields will be interpreted by the program as an indication that the corresponding calculation is not required. We will see the benefit of this when we move to the Wind Pressures tab and see how concisely these results are reported.

On a related note, it is worth mentioning that the Cp and Cn tables occasionally report values of zero for certain conditions. These are typically provided for interpolation purposes. But if a situation is ever encountered where a value of zero for Cp or Cn is actually required for design purposes, the user is advised to enter a small nonzero value.

Wind Pressures

This tab reports the values of wind pressures that occur on the various surfaces of the building when the named elevation receives positive external pressure.



Looking at the screen capture above, we see that the North Wall tab is currently selected. Let's work through this tab thoroughly as an example. As indicated in the on-screen note, we interpret all of the results on this tab as being the pressures that occur on the named surface of the building **<u>when the North Wall receives positive external wind pressure</u>**. So when we are focused on the North Wall tab:

- The "Windward Wall Pressures" are those that would apply to the **North** Wall when the North Wall receives positive external pressure.
- The "Leeward Wall Pressures" are those that would apply to the **South** Wall when the North Wall receives positive external pressure.
- The "Sidewall Pressures" are those that would apply to the **East** and **West** Walls when the North Wall receives positive external pressure.
- The "Roof Pressures" are those that would apply to the identified areas measured from the **North** edge of the roof when the North Wall receives positive external pressure.

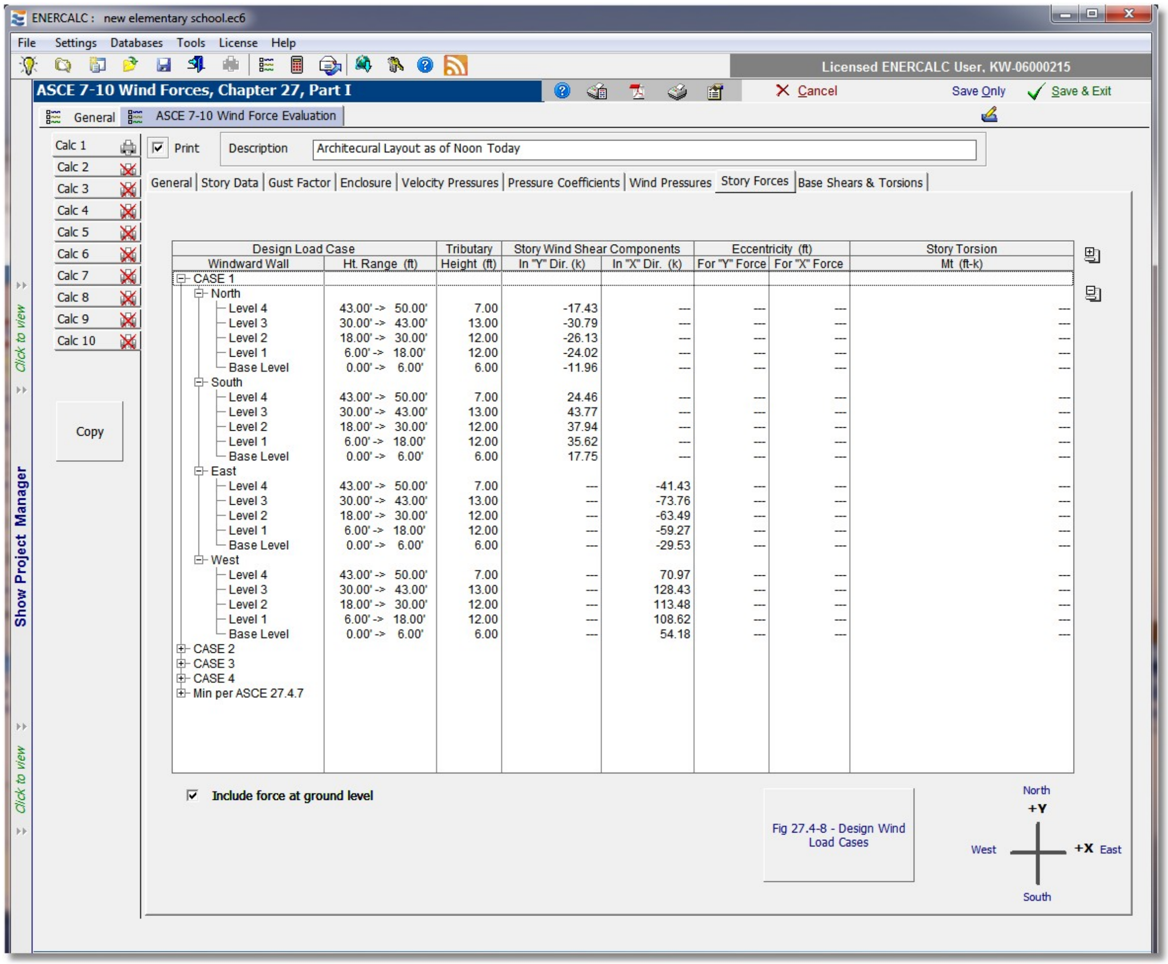
Just to drive home the proper interpretation of the values reported on the Wind Pressures tab, let's work through the East tab, so we can interpret some windward and leeward roof pressures. Remember that this hypothetical building is assumed to have a North-South oriented ridge. So when we are focused on the East Wall tab:

- The "Windward Wall Pressures" are those that would apply to the **East** Wall when the East Wall receives positive external pressure.
- The "Leeward Wall Pressures" are those that would apply to the **West** Wall when the East Wall receives positive external pressure.
- The "Sidewall Pressures" are those that would apply to the **North** and **South** Walls when the East Wall receives positive external pressure.
- The "Perp: Windward Roof Pressures" are those that would apply to the **windward (East)** portion of the roof when the East Wall receives positive external pressure.
- The "Perp: Leeward Roof Pressures" are those that would apply to the **leeward (West)** portion of the roof when the East Wall receives positive external pressure.

Note that all surfaces report pressures based on both the positive and the negative internal pressure conditions, and the algebraic sign convention follows that of ASCE 7, which is to say that positive values are interpreted to act toward the named surface and negative values act away from the named surface. Work this logic through an example such as a windward wall and see that it all makes sense. The negative internal pressure condition produces higher total pressures toward the windward wall, than the positive internal pressure condition does, because the negative internal pressure works in the same direction as the external pressure on a windward wall. Similar logic can be applied to all other surfaces to demonstrate that the mathematics are proper.

Story Forces

This tab reports the values of wind forces tributary to each story in the building.



Using the story heights determined on the Story Data tab, the Story Forces tab determines the tributary heights for each floor by assuming a simply-supported wall construction that spans between adjacent floor/roof levels. Wind pressures are applied to the tributary heights and multiplied by the perpendicular dimension of the building to arrive at forces for each story.

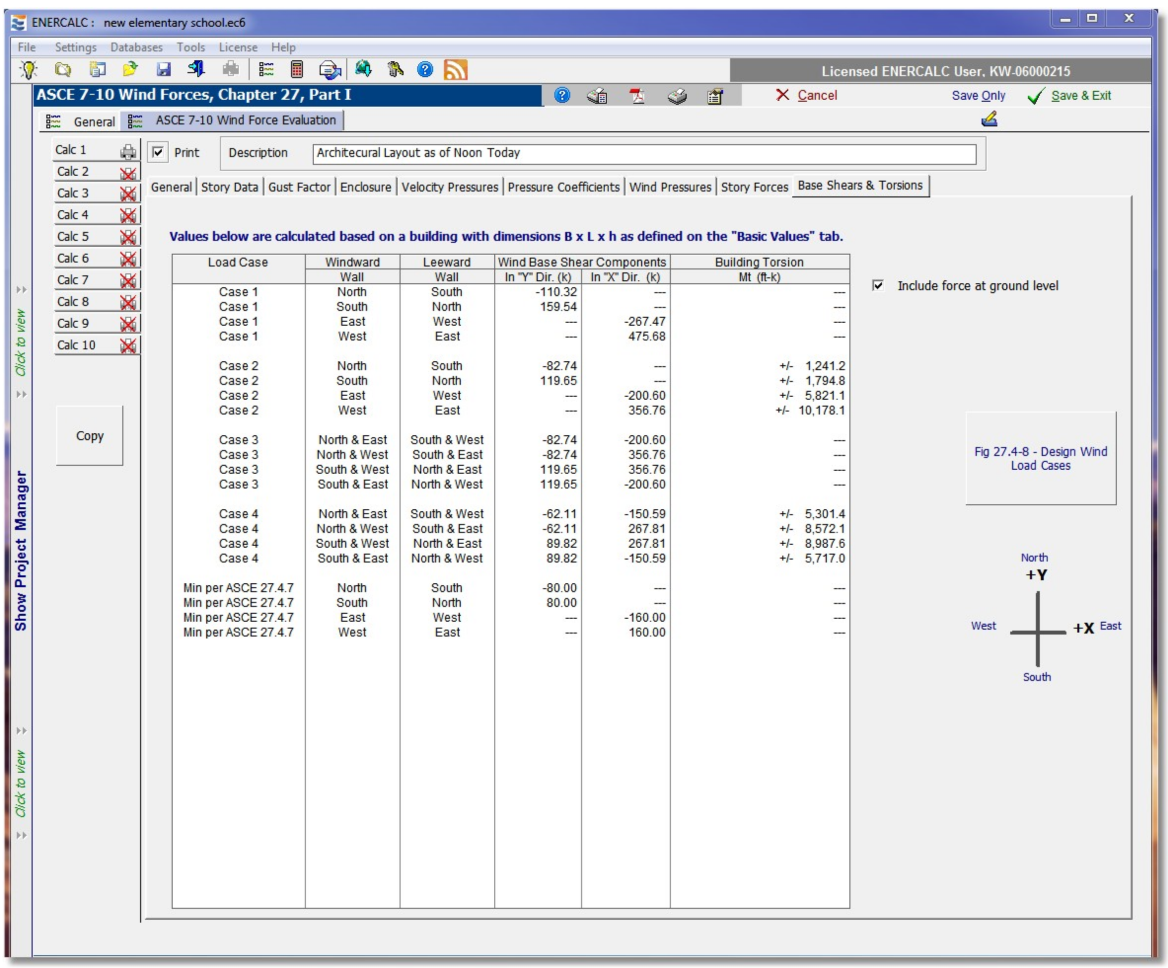
The option is provided to either display or hide the forces tributary to the lower half of the lowest level. In some construction types, this component of load is delivered to a slab on grade and is not a design consideration for the Main Wind Force Resisting System.

The program reports results for each of the four "Cases" as presented by Figure 27.4-8 of ASCE 7-10. Cases 2 and 4 also incorporate a design torsional moment as defined in Figure 27.4-8. In all situations, the program reports the force magnitude. In situations where a torsional moment applies, the program reports the moment arm that is being considered for each component of force, as well as the resulting net moment.

The final item in the results on this tab is based on the minimum required wind loads per Section 27.4.7 of ASCE 7-10. That section requires that the wind load to be used in the design of the Main Wind Force Resisting System for an enclosed or partially enclosed building shall not be less than 16 psf multiplied by the wall area of the building (and 8 psf multiplied by the roof area of the building projected onto a vertical plane normal to the assumed wind direction). So in this last item in the results list, the program reports story forces assuming 16 psf applied to each wall of the building. As of build 6.12.4.24, the program does not collect enough information to consider the 8 psf on the projection of the roof area, so this additional load may need to be considered with supplemental hand calculations on buildings with other than flat or low-slope roofs.

Base Shears & Torsions

This tab reports the summation of the wind story forces and torsions for all levels in the building, for all four "Cases" and for the minimum required wind loads per Section 27.4.7.



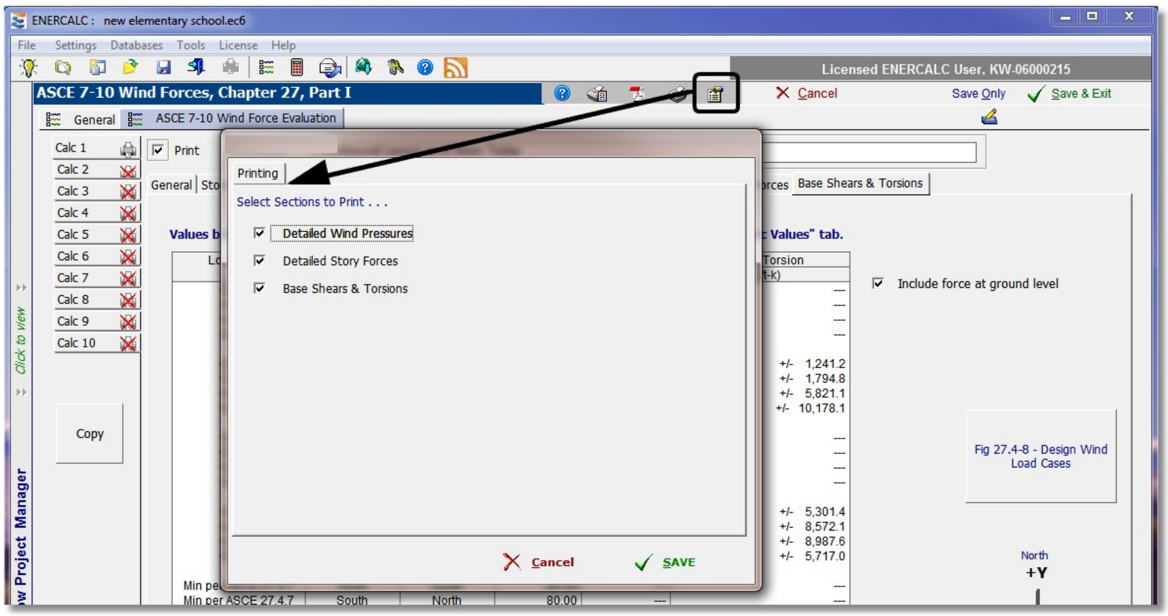
Load Case	Windward Wall	Leeward Wall	Wind Base Shear Components In "Y" Dir. (k)	In "X" Dir. (k)	Building Torsion Mt (ft-k)
Case 1	North	South	-110.32	---	---
Case 1	South	North	159.54	---	---
Case 1	East	West	---	-267.47	---
Case 1	West	East	---	475.68	---
Case 2	North	South	-82.74	---	+/- 1,241.2
Case 2	South	North	119.65	---	+/- 1,794.8
Case 2	East	West	---	-200.60	+/- 5,821.1
Case 2	West	East	---	356.76	+/- 10,178.1
Case 3	North & East	South & West	-82.74	-200.60	---
Case 3	North & West	South & East	-82.74	356.76	---
Case 3	South & West	North & East	119.65	356.76	---
Case 3	South & East	North & West	119.65	-200.60	---
Case 4	North & East	South & West	-62.11	-150.59	+/- 5,301.4
Case 4	North & West	South & East	-62.11	267.81	+/- 8,572.1
Case 4	South & West	North & East	89.82	267.81	+/- 8,987.6
Case 4	South & East	North & West	89.82	-150.59	+/- 5,717.0
Min per ASCE 27.4.7	North	South	-80.00	---	---
Min per ASCE 27.4.7	South	North	80.00	---	---
Min per ASCE 27.4.7	East	West	---	-160.00	---
Min per ASCE 27.4.7	West	East	---	160.00	---

Although "Base Shears" are not technically a part of the ASCE 7-10 design procedure for wind loads, the summation of all story forces is often of interest to designers for a variety of reasons. Some designers like to see how the wind "base shear" compares to the seismic base shear. For some designers, the summation of the wind forces is useful in the checking process to get a confidence level that the calculated pressures are reasonable. Whatever the reason, the values are reported on this tab if they are of interest.

As with the Story Forces tab, the Base Shears & Torsions tab provides the option to either include or exclude the component of shear and torsion that is tributary to the bottom half of the lowest level.

Print Properties

Finally, the complexity of this wind method and its requirements to examine multiple cases and conditions can lead to voluminous output. In recognition of the fact that different designers have different needs, the Print Properties dialog has been provided for this module to enable the output to be tailored to suit.



10.1.6 ASCE 7-05 Section 6.4, MWFRS:Simplified Forces

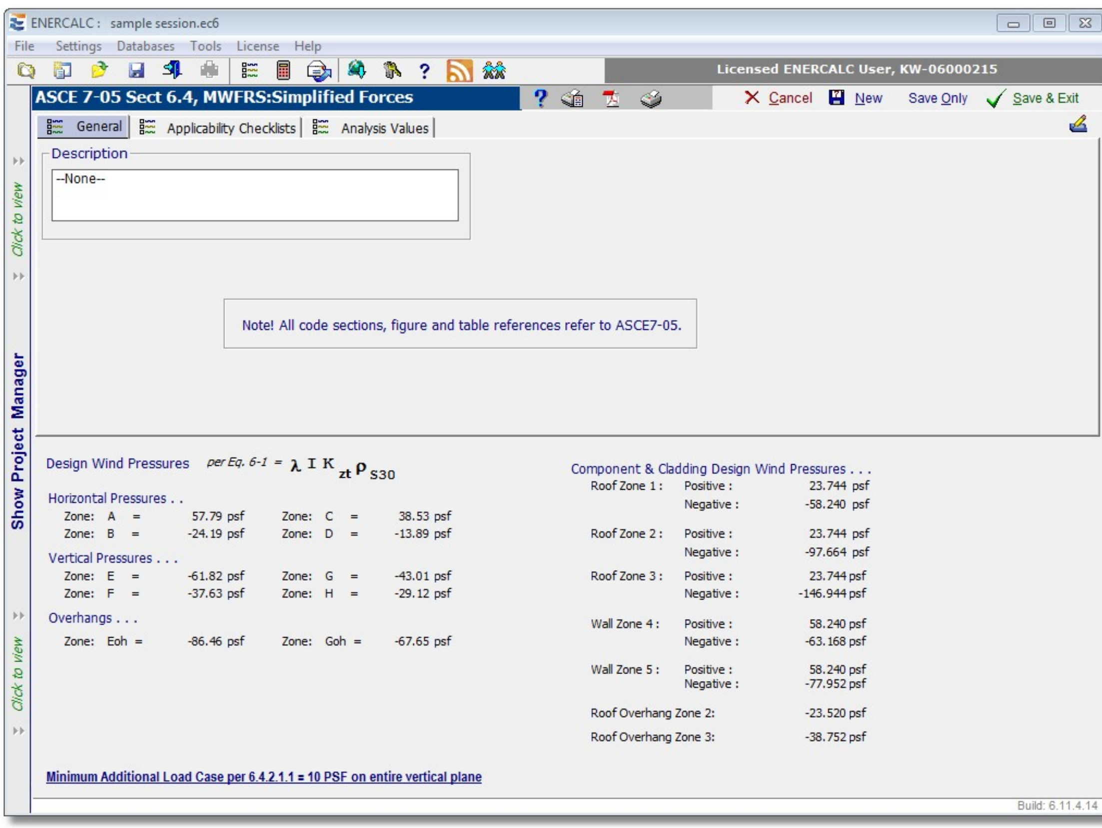
Need more? Ask Us a Question

This module is a presentation of the Simplified Wind Forces provisions of ASCE 7-05.

Click here for a video: ASCE 7-05 Wind Forces

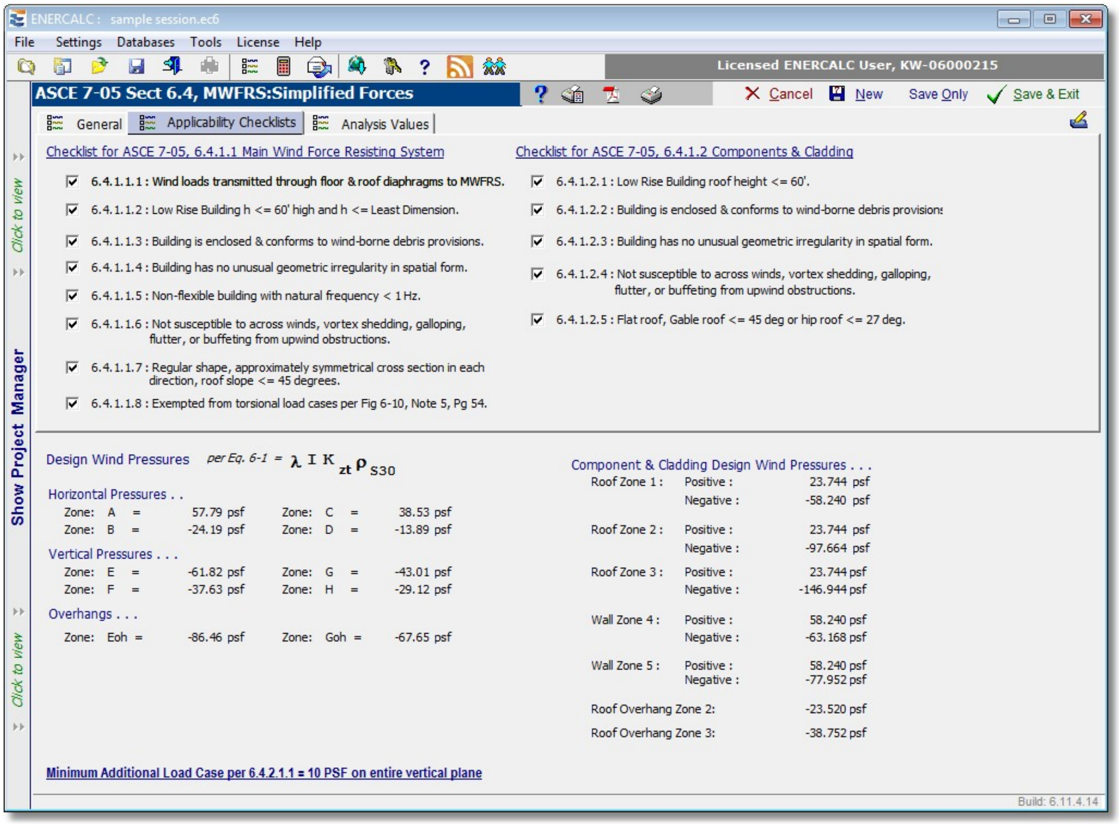
Limited documentation is provided here, because all of the references to ASCE 7-05 are given on the module screens.

General



Applicability Checks

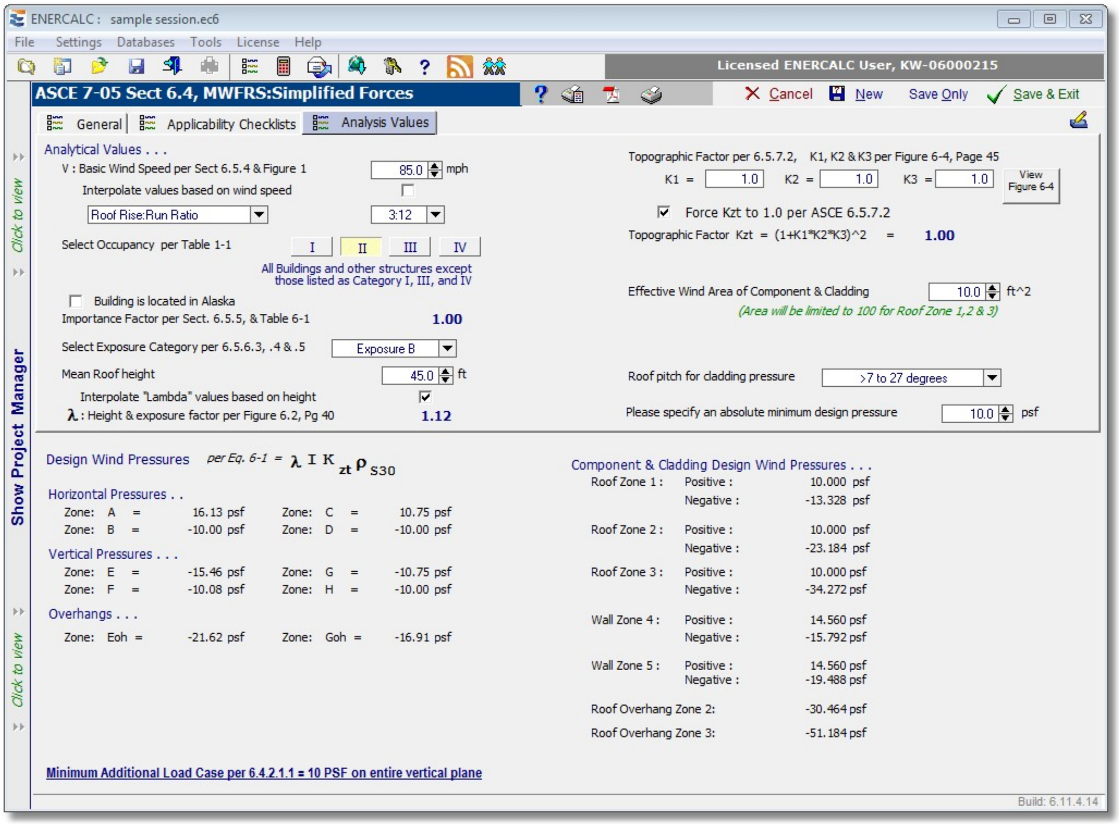
Please see ASCE 7-05 references for descriptions. This module performs wind load calculations for the "Simplified" method. ALL boxes must be checked for module to give results on the Analysis Values tab.



Analysis Values

See ASCE 7-05 sections noted for description of values and other references.

The lower portion of this tab gives the resulting calculated wind pressures for the main wind force resisting system and for components & cladding.


ENERCALC : sample session.ec6
File Settings Databases Tools License Help
Licensed ENERCALC User, KW-06000215
ASCE 7-05 Sect 6.4, MWFRS:Simplified Forces
Cancel New Save Only Save & Exit
General Applicability Checklists Analysis Values
Analytical Values . . .
V : Basic Wind Speed per Sect 6.5.4 & Figure 1 85.0 mph
Interpolate values based on wind speed
Roof Rise:Run Ratio 3:12
Select Occupancy per Table 1-1 I II III IV
All Buildings and other structures except those listed as Category I, III, and IV
Building is located in Alaska
Importance Factor per Sect. 6.5.5, & Table 6-1 1.00
Select Exposure Category per 6.5.6.3, .4 & .5 Exposure B
Mean Roof height 45.0 ft
Interpolate "Lambda" values based on height
λ : Height & exposure factor per Figure 6.2, Pg 40 1.12
Topographic Factor per 6.5.7.2, K1, K2 & K3 per Figure 6-4, Page 45
K1 = 1.0 K2 = 1.0 K3 = 1.0
View Figure 6-4
Force Kzt to 1.0 per ASCE 6.5.7.2
Topographic Factor Kzt = (1+K1*K2*K3)^2 = 1.00
Effective Wind Area of Component & Cladding 10.0 ft^2
(Area will be limited to 100 for Roof Zone 1,2 & 3)
Roof pitch for cladding pressure >7 to 27 degrees
Please specify an absolute minimum design pressure 10.0 psf
Design Wind Pressures per Eq. 6-1 = λ I Kzt P S30
Horizontal Pressures . .
Zone: A = 16.13 psf Zone: C = 10.75 psf
Zone: B = -10.00 psf Zone: D = -10.00 psf
Vertical Pressures . . .
Zone: E = -15.46 psf Zone: G = -10.75 psf
Zone: F = -10.08 psf Zone: H = -10.00 psf
Overhangs . . .
Zone: Eoh = -21.62 psf Zone: Goh = -16.91 psf
Component & Cladding Design Wind Pressures . . .
Roof Zone 1 : Positive : 10.000 psf
Negative : -13.328 psf
Roof Zone 2 : Positive : 10.000 psf
Negative : -23.184 psf
Roof Zone 3 : Positive : 10.000 psf
Negative : -34.272 psf
Wall Zone 4 : Positive : 14.560 psf
Negative : -15.792 psf
Wall Zone 5 : Positive : 14.560 psf
Negative : -19.488 psf
Roof Overhang Zone 2: -30.464 psf
Roof Overhang Zone 3: -51.184 psf
Minimum Additional Load Case per 6.4.2.1.1 = 10 PSF on entire vertical plane
Show Project Manager
Click to view
Build: 6.11.4.14

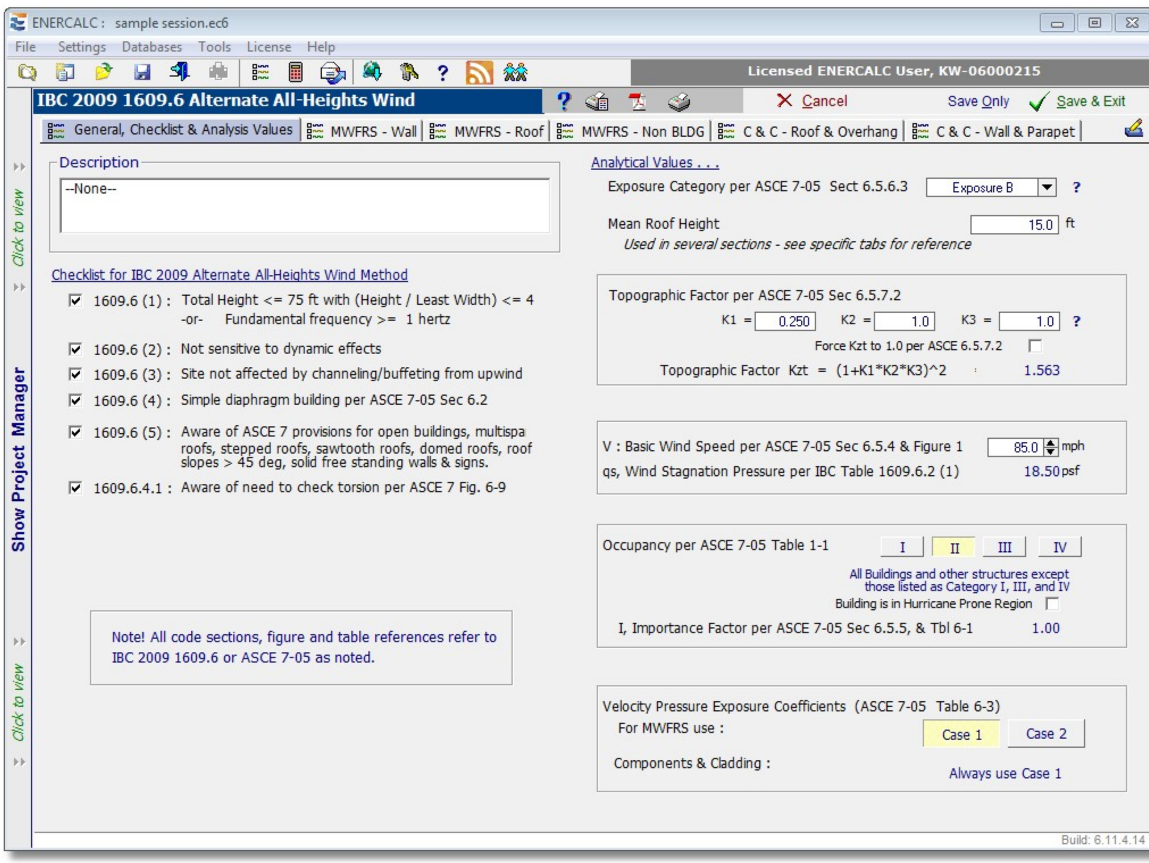
10.1.7 IBC Alternate All-Heights Wind Method

Need more? Ask Us a Question

This module is an implementation of the IBC Alternate All-Heights Wind Method.

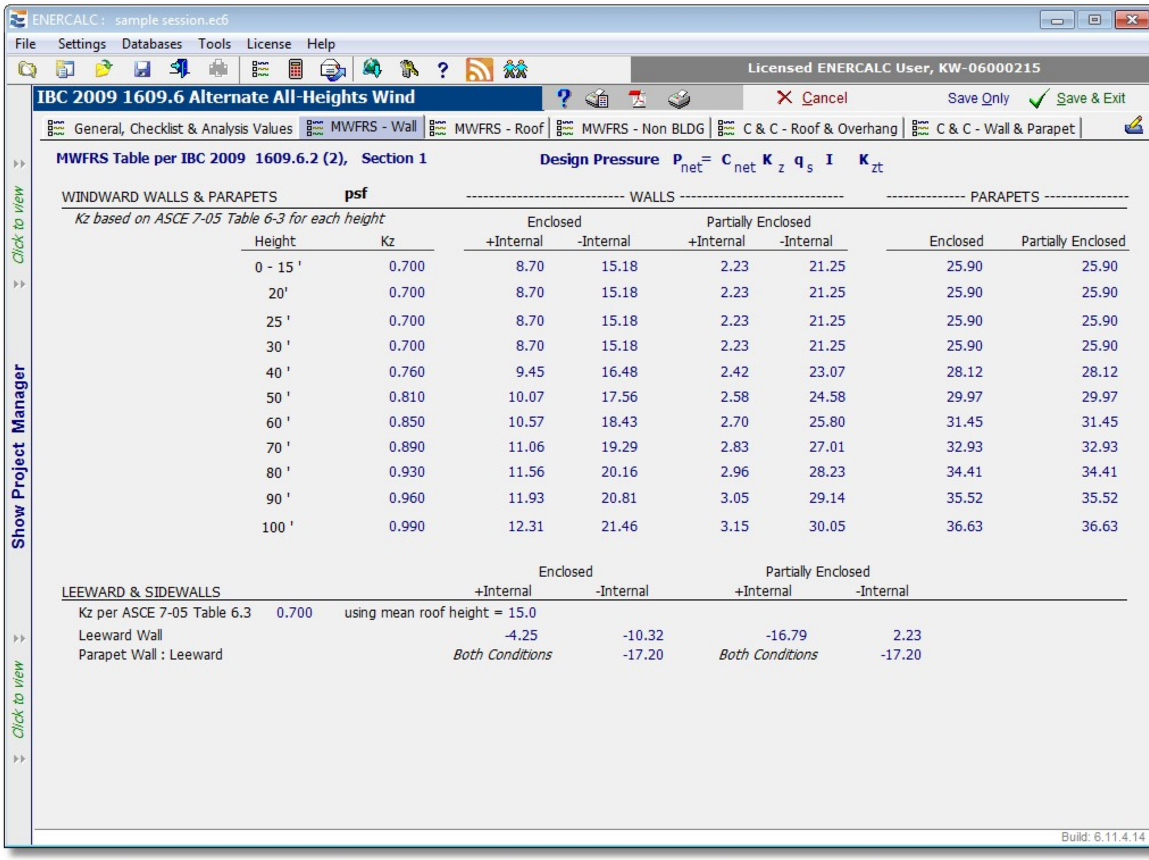
General, Checklist, & Analysis Values

This tab offers a checklist of the criteria that must be satisfied in order to apply the IBC 2009 Alternate All-Heights Wind Method. Once all of the checkboxes are checked, the module will offer input fields for analytical values such as Exposure Category, Mean Roof Height, Topographic Factor parameters, Basic Wind Speed, Occupancy category, and options for the selection of Velocity Pressure Coefficients.



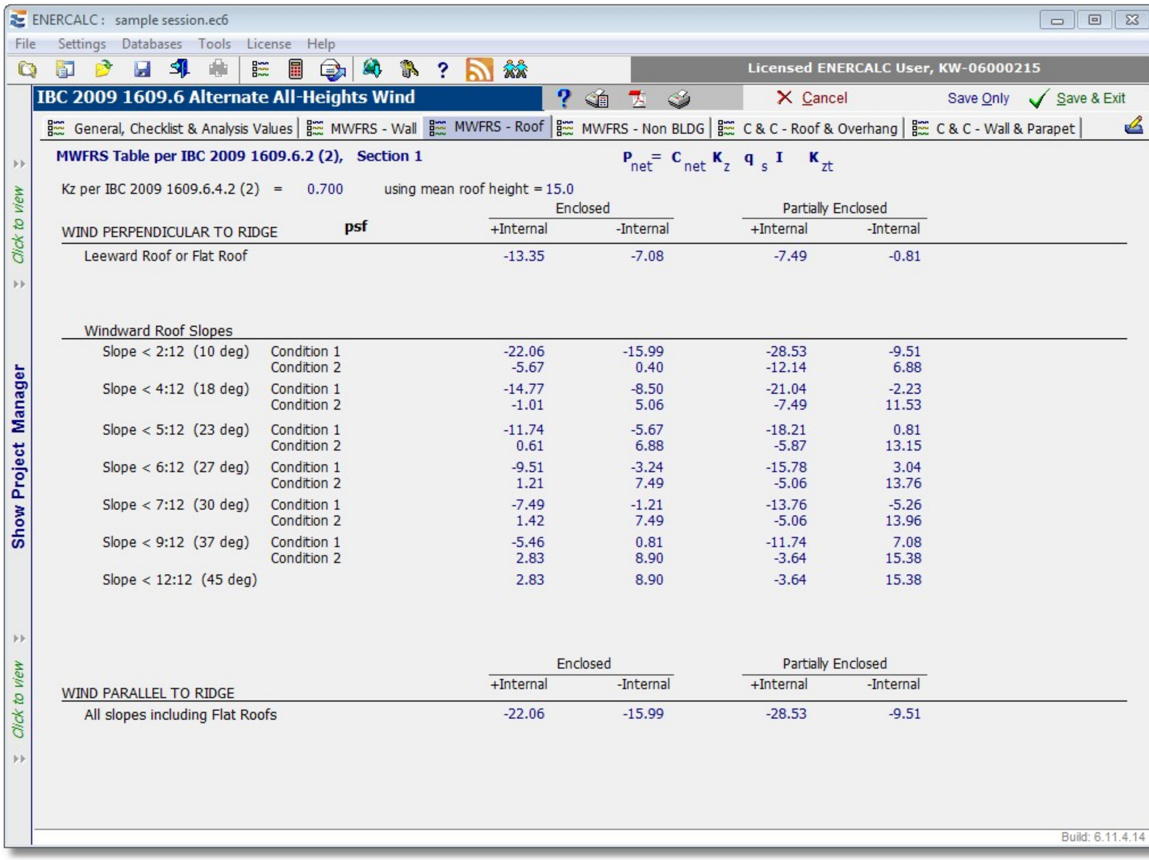
MWFRS - Wall

This tab reports design pressures on windward walls, leeward walls, sidewalls and parapets. Pressure values are distinguished for Enclosed and Partially Enclosed structures, and values are reported for the positive internal pressure condition and for the negative internal pressure condition.



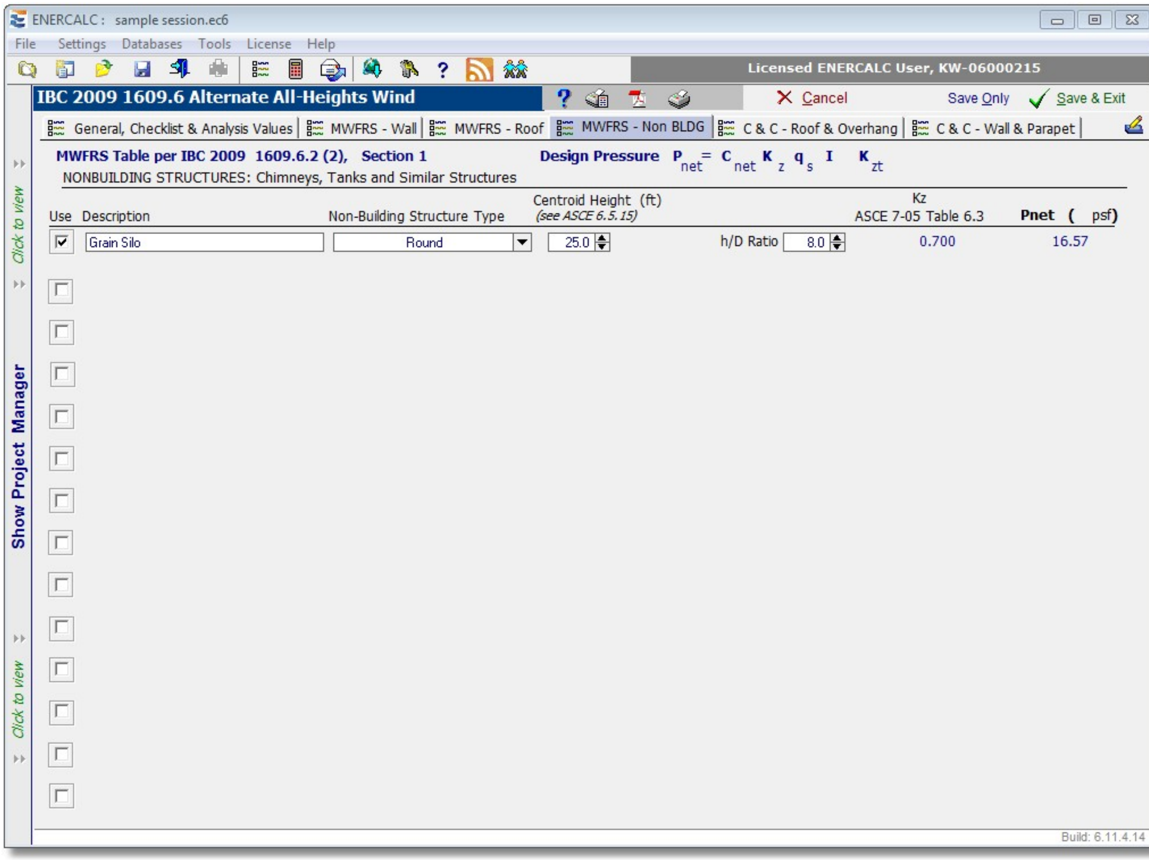
MWFRS - Roof

This tab reports design pressures on roof surfaces. Pressure values are distinguished for the conditions of wind perpendicular to the ridge and wind parallel to the ridge. In the case of wind perpendicular to the ridge, pressure values are reported for flat roofs as well as for various slope angles. In all cases, results are presented for Enclosed and for Partially Enclosed structures, and values are reported for the positive internal pressure condition and for the negative internal pressure condition.



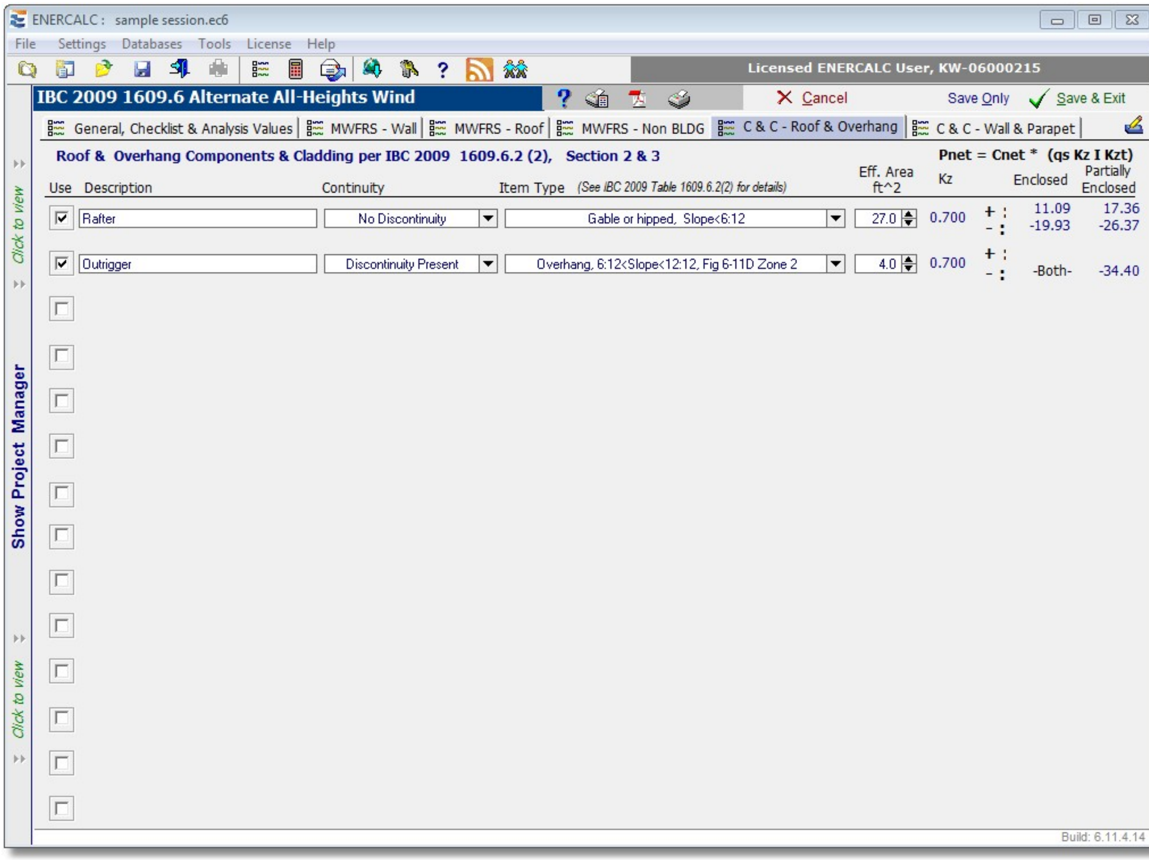
MWFRS - Non BLDG

This tab reports net design pressures on the projection of surfaces of non-building structures.



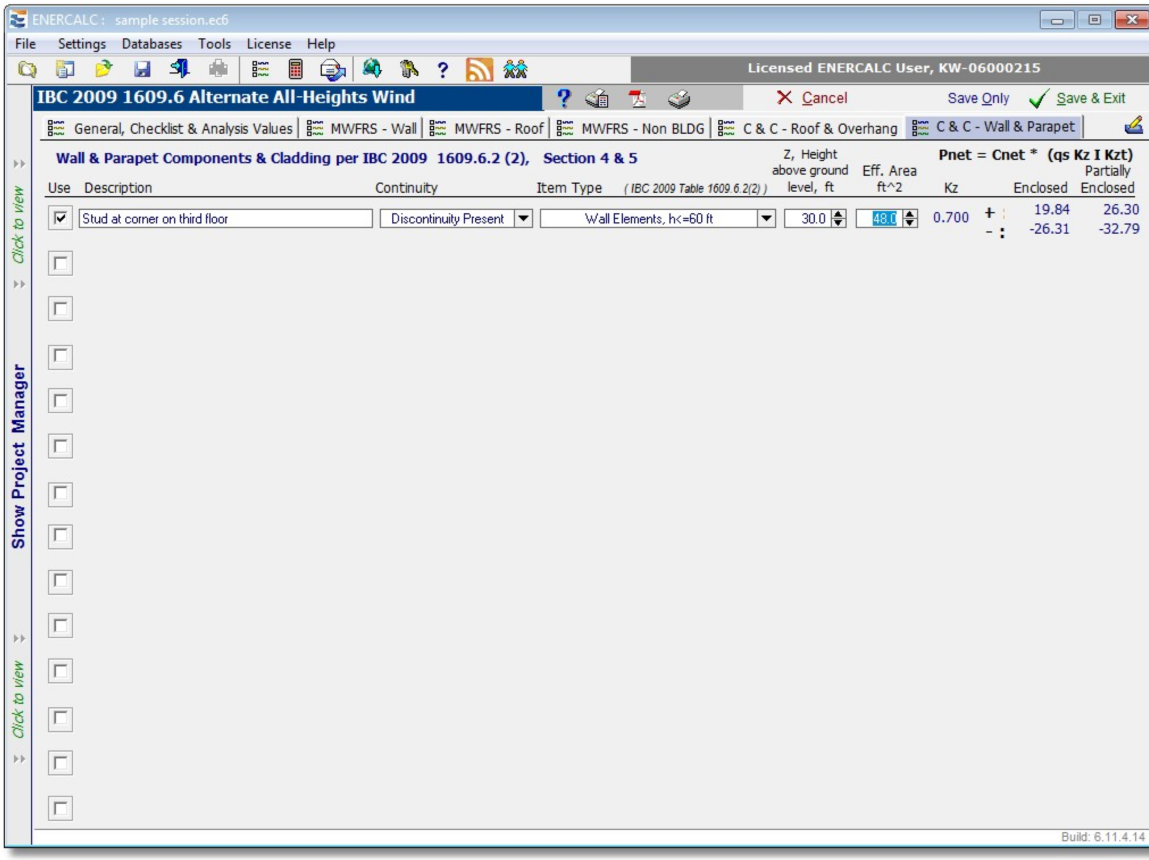
C&C - Roof & Overhang

This tab reports design pressures on components and cladding within roof surfaces. Results are presented for Enclosed and for Partially Enclosed structures, and values are reported for the positive internal pressure condition and for the negative internal pressure condition.



C&C - Wall & Parapet

This tab reports design pressures on components and cladding within wall and parapet surfaces. Results are presented for Enclosed and for Partially Enclosed structures, and values are reported for the positive internal pressure condition and for the negative internal pressure condition.


ENERCALC : sample session.ec6
File Settings Databases Tools License Help
Licensed ENERCALC User, KW-06000215
IBC 2009 1609.6 Alternate All-Heights Wind
Cancel
Save Only
Save & Exit
General, Checklist & Analysis Values
MWFRS - Wall
MWFRS - Roof
MWFRS - Non BLDG
C & C - Roof & Overhang
C & C - Wall & Parapet
Wall & Parapet Components & Cladding per IBC 2009 1609.6.2 (2), Section 4 & 5
Z, Height above ground level, ft
Eff. Area ft^2
Pnet = Cnet * (qs Kz I Kzt)
Use Description
Continuity
Item Type
(IBC 2009 Table 1609.6.2(2))
Kz
Enclosed
Partially Enclosed
Stud at corner on third floor
Discontinuity Present
Wall Elements, h<=60 ft
30.0
48.0
0.700
19.84
26.30
-26.31
-32.79
Show Project Manager
Click to view
Build: 6.11.4.14

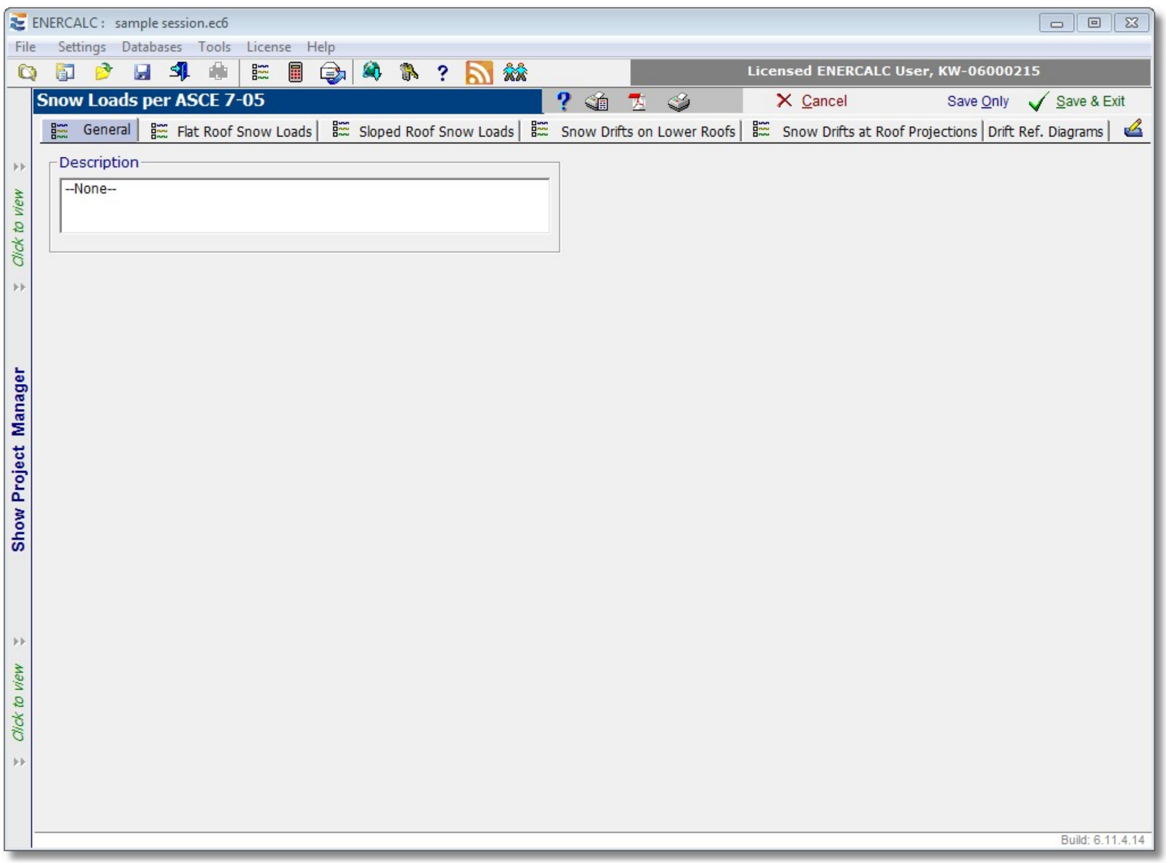
10.1.8 ASCE Snow Loads

[Need more? Ask Us a Question]

This module is an implementation of the ASCE method of determining Flat Roof Snow Load, Sloped Roof Snow Load, and Snow Drifts on Lower Roofs, and Snow Drifts at Roof Obstructions. The module currently supports calculations based on ASCE 7-05 and ASCE 7-10. The Snow Loads module does ***not*** currently address the calculation of unbalanced snow loads, rain on snow surcharges, or sliding snow surcharges.

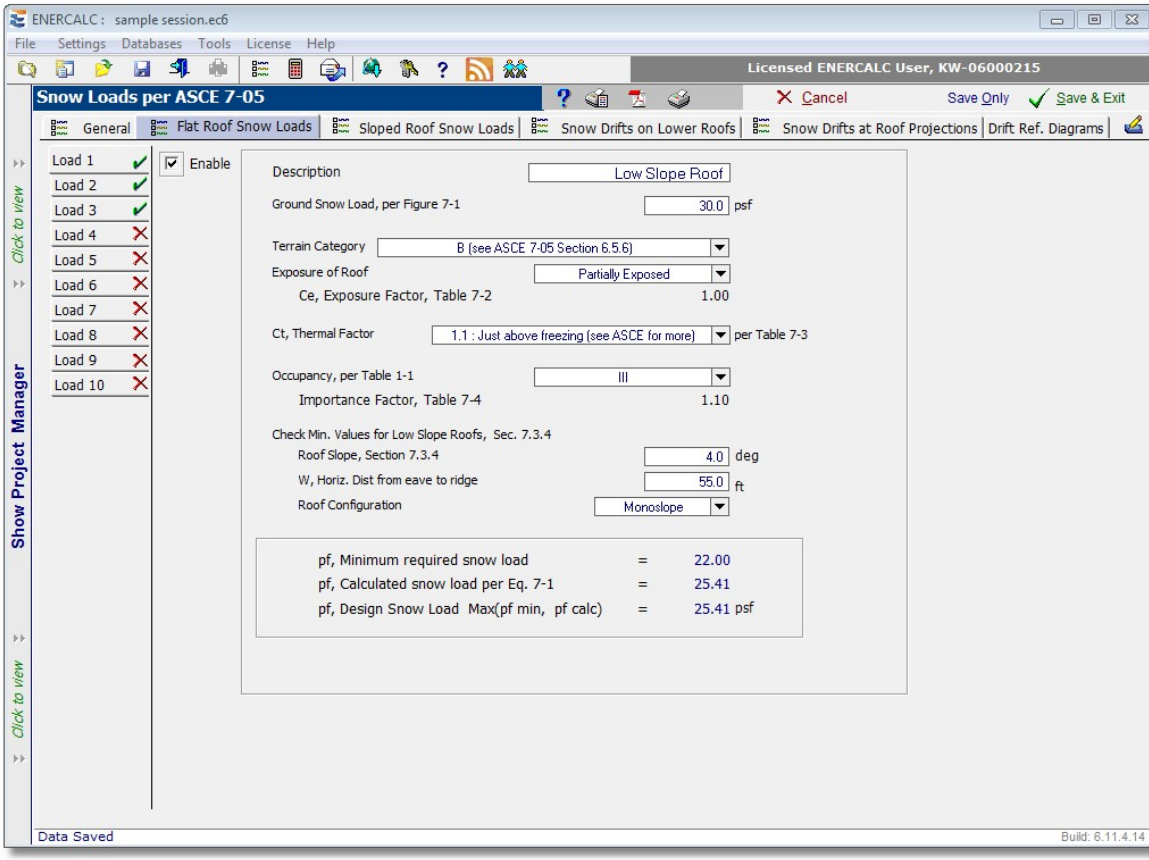
General

Provides a place to enter descriptive text.



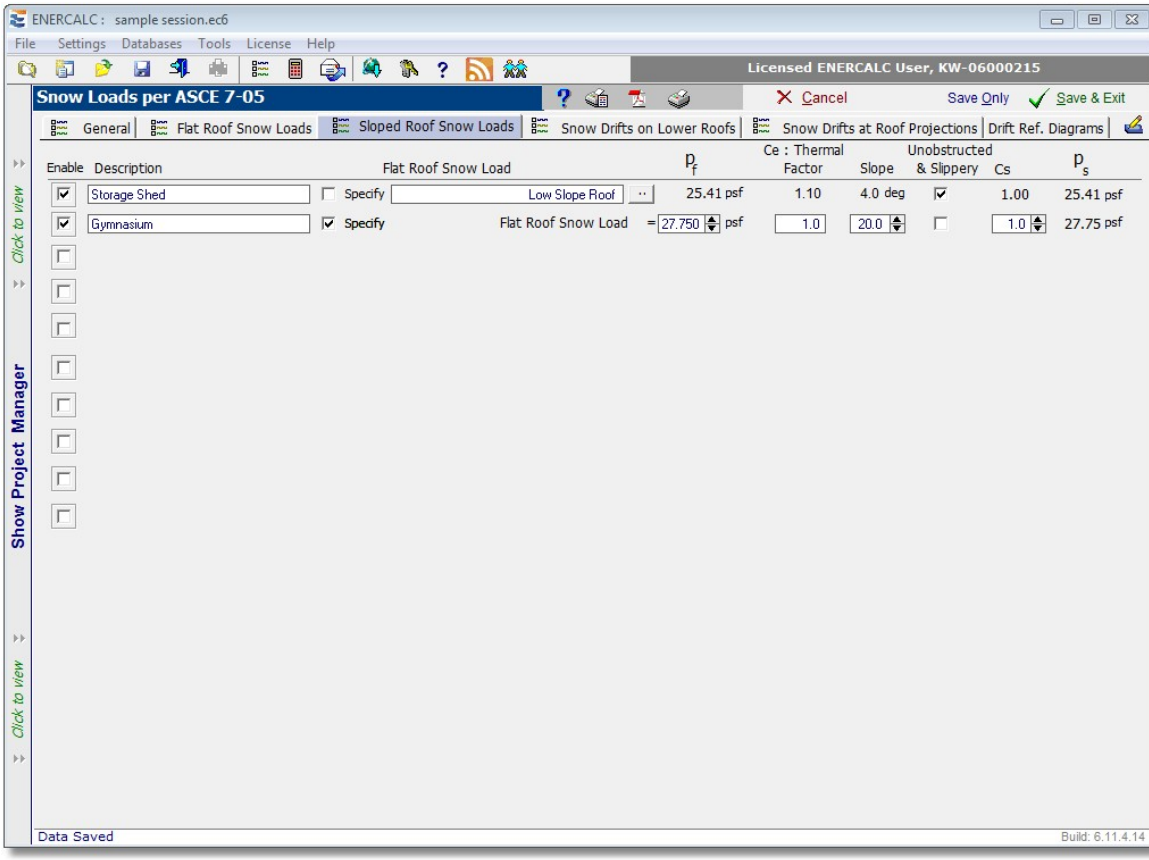
Flat Roof Snow Loads

Allows up to 10 different flat roof snow loads to be calculated by entering values for all of the parameters in accordance with ASCE 7.



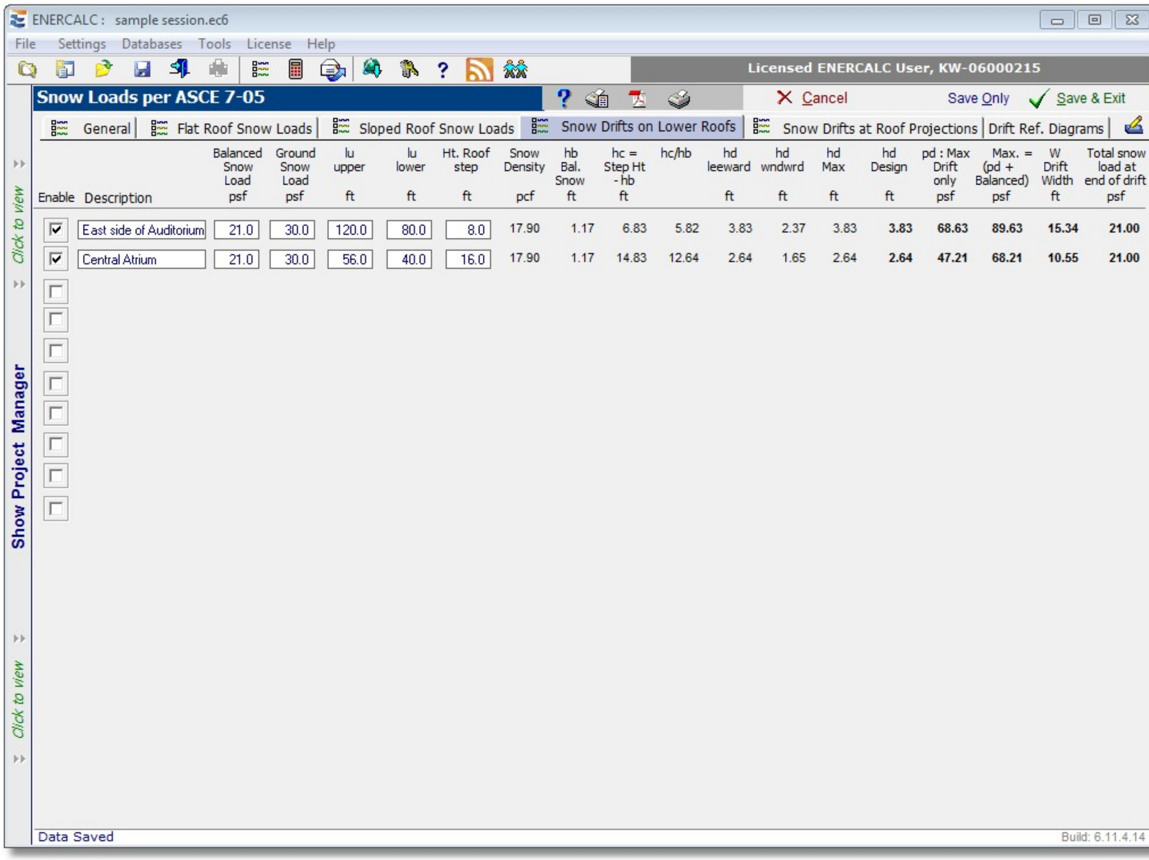
Sloped Roof Snow Loads

Allows up to 10 different sloped roof snow loads. The input form provides the option to refer to a previously defined Flat Roof Snow Load, or to explicitly specify a Flat Roof Snow Load value along with the remaining values for all of the parameters in accordance with ASCE 7.



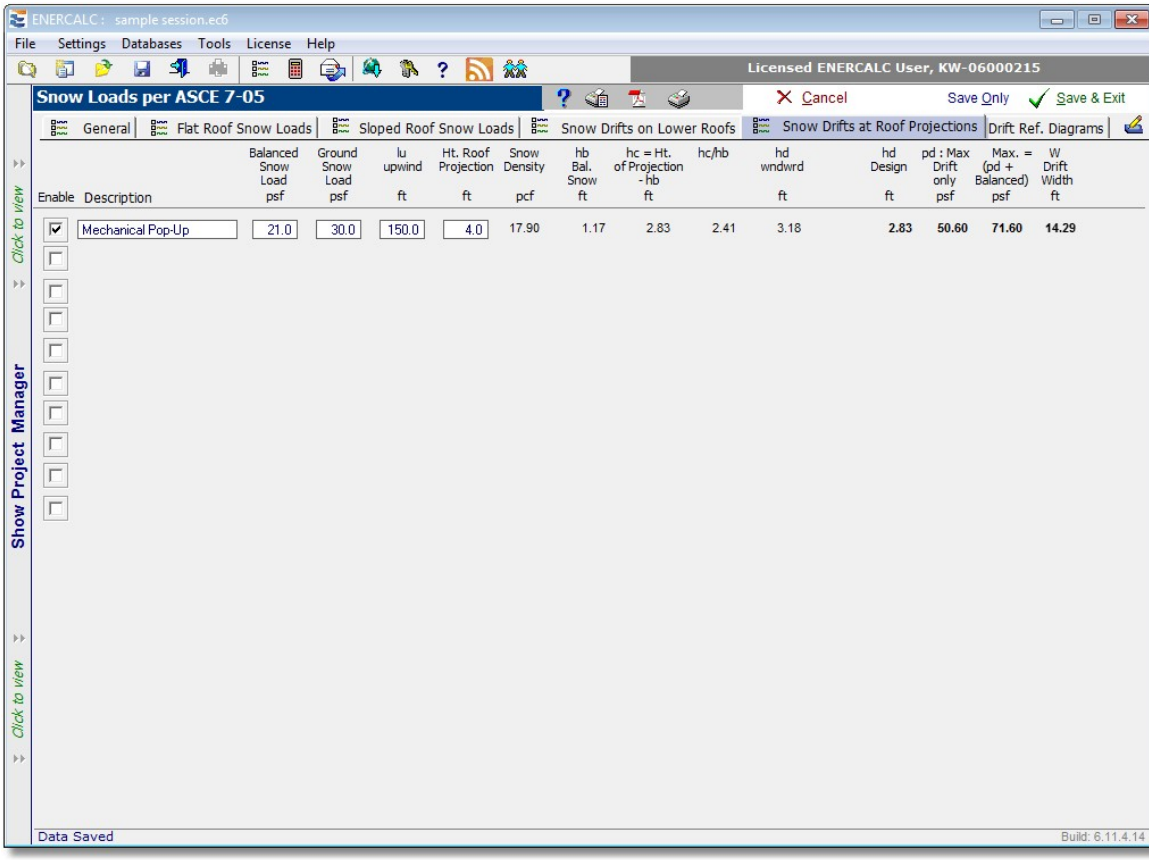
Snow Drifts on Lower Roofs

Allows up to 10 different conditions to be evaluated for snow drifts on lower roofs. The input form collects values for all of the necessary parameters to determine the windward and leeward drifts and selects the controlling case in accordance with ASCE 7.



Snow Drifts at Roof Projections

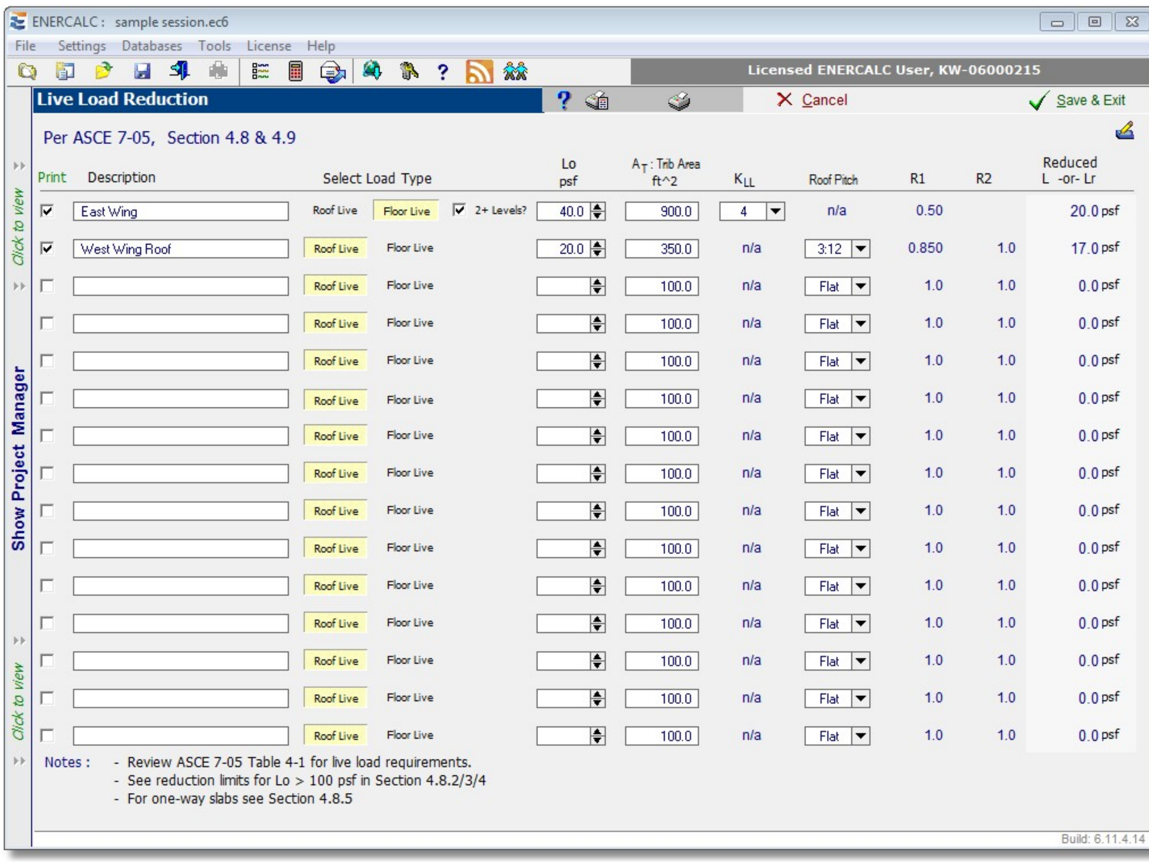
Allows up to 10 different conditions to be evaluated for snow drifts at roof projections. The input form collects values for all of the necessary parameters to determine the windward drift in accordance with ASCE 7.


ENERCALC : sample session.ec6
File Settings Databases Tools License Help
Licensed ENERCALC User, KW-06000215
Snow Loads per ASCE 7-05
Cancel
Save Only
Save & Exit
General
Flat Roof Snow Loads
Sloped Roof Snow Loads
Snow Drifts on Lower Roofs
Snow Drifts at Roof Projections
Drift Ref. Diagrams
Enable
Description
Balanced Snow Load psf
Ground Snow Load psf
lu upwind ft
Ht. Roof Projection ft
Snow Density pcf
hb Bal. Snow ft
hc = Ht. of Projection - hb ft
hc/hb
hd wndwrd ft
hd Design ft
pd : Max Drift only psf
Max. = (pd + Balanced) psf
W Drift Width ft
Mechanical Pop-Up
21.0
30.0
150.0
4.0
17.90
1.17
2.83
2.41
3.18
2.83
50.60
71.60
14.29
Show Project Manager
Click to view
Data Saved
Build: 6.11.4.14

10.1.9 ASCE Live and Roof Live Load Reduction

This section enables the user to calculate reduced live and roof live loads for various parts of a project.

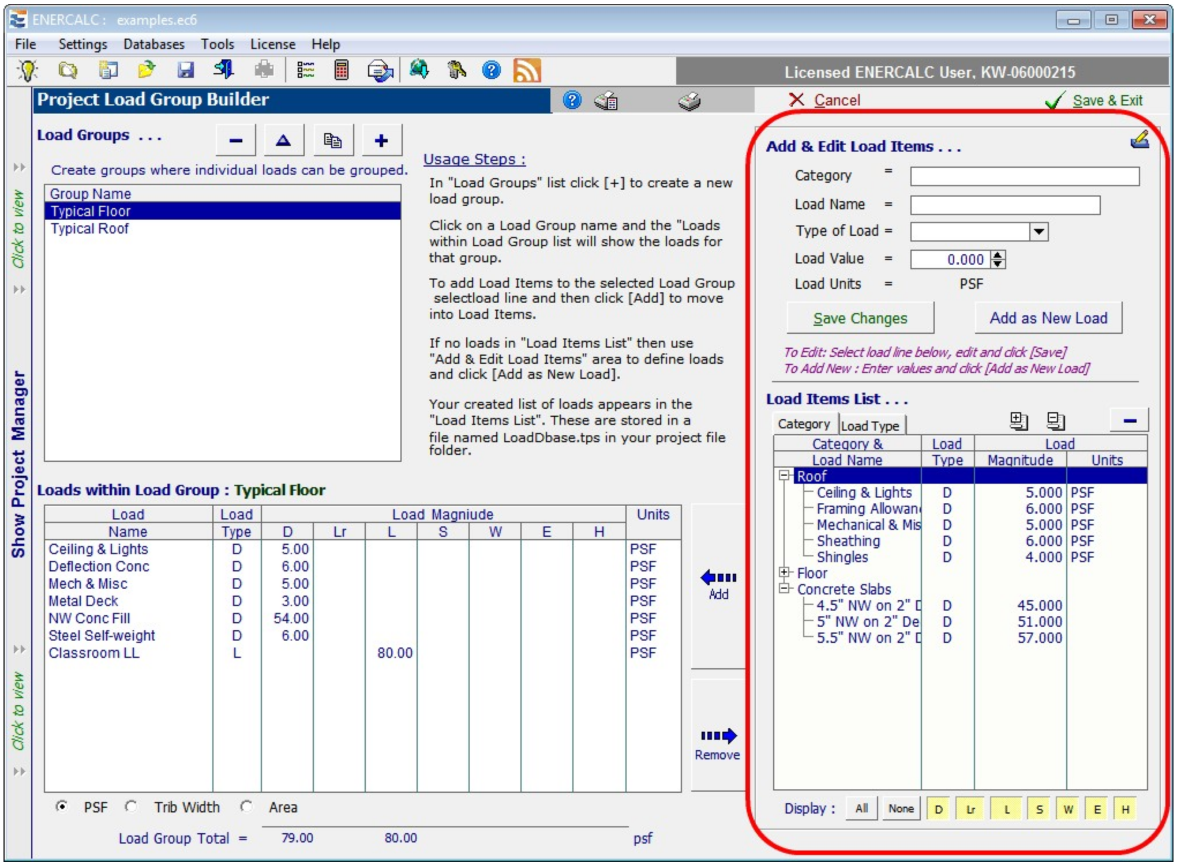
This section is a direct application of ASCE 7-05 Sections 4.8 and 4.9 and ASCE 7-10 Sections 4.7 & 4.8 concerning live and roof live load reductions. We refer the user to those sections of the referenced design standards for specific descriptions of the values used in each column seen below.



10.1.10 Project Load Group Builder

The Project Load Group Builder enables the user to store commonly used "Load Items" and then combine them into various Groups to calculate total loads for the defined Groups.

Load Items are defined and organized on the right-hand side of the screen as indicated in the screen capture below:



You can type in a Category name (discussed in more detail below), a Load Name, choose the Load Type, and specify the Load Value, and then click "Add as New Load". The newly added Load Item will appear in the Load Items List below. To edit an existing Load Item, click on the item of interest in the Load Items List, edit the values in the individual fields above, and then click "Save Changes"

Load Items are very flexible in their definition. A Load Item could represent the weight of a particular building construction material, such as the Dead Load of 3/4" plywood floor sheathing in units of PSF. A Load Item could also represent an allowance of 5 PSF Dead Load for ceiling & lights. But Load Items can also take the form of load types other than Dead Load as well. For instance, you could define a Load Item consisting of 100 PSF Live Load, or 30 PSF Snow Load. In this way, the Project Load Group Builder is capable of tracking loads of all types that may be applicable to a particular area of a building.

One of the powerful features of the Project Load Group Builder is that it takes the Load Item data that you enter and it saves that data in a file that is separate and distinct from your Project File. (The file is named LoadDbase.tps, and it is saved in the folder that you have identified as your default Project File location.) In this way, the system continues to accumulate more and more of your commonly specified materials and loads as you work on different projects. And that ever-growing database of your custom Load Items is always available to you on future projects.

In order to bring some organization to the many load items that you may eventually accumulate, the Project Load Group Builder introduces the concept of the "Category". A Category can contain one or more Load Items. Here again, the way you use Categories can be extremely flexible. Some users may prefer to organize their Load Items into Categories that represent different types of materials or loads. For example, the organizational hierarchy might take on a form like this where the bullet items represent Load Items within the indicated Category:

Category: Roof Sheathing
- 1/2" sheathing
- 5/8" sheathing
- 3/4" sheathing
- 7/8" sheathing
- 1" sheathing

Category: Roof Framing
- 2x6@16" o/c
- 2x8@16" o/c
- 2x10@16" o/c
- 2x12@16" o/c

Category: NW Concrete Slab
- 4.5" total with 2" deck
- 5.0" total with 2" deck
- 5.5" total with 2" deck
- 6.0" total with 2" deck
- 5.5" total with 3" deck
- 6.0" total with 3" deck
- 6.5" total with 3" deck

Category: LW Concrete Slab
- 4.5" total with 2" deck
- 5.0" total with 2" deck
- 5.25" total with 2" deck
- 5.5" total with 2" deck
- 6.0" total with 2" deck
- 5.5" total with 3" deck
- 6.0" total with 3" deck
- 6.25" total with 3" deck

- 6.5" total with 3" deck

Category: Live Load
- Classroom
- First Floor Corridor
- Office
- Stack Rooms

Category: Snow Load
- 30 PSF
- 32 PSF
- 34 PSF
- 36 PSF

And so on...

But remember, this is just one way that Categories could be used. Here is yet another idea:

Category: Typical classroom on structural floor
- DL of flooring
- DL of concrete slab on metal deck
- DL of composite metal deck
- DL allowance for structural steel framing
- DL allowance for ceiling & lights
- DL allowance for mechanical & misc.
- LL for classroom occupancy
- LL for partition allowance

Category: Typical corridor on structural floor
- DL of flooring
- DL of concrete slab on metal deck
- DL of composite metal deck
- DL allowance for structural steel framing
- DL allowance for ceiling & lights
- DL allowance for mechanical & misc.
- LL for corridor above first floor occupancy

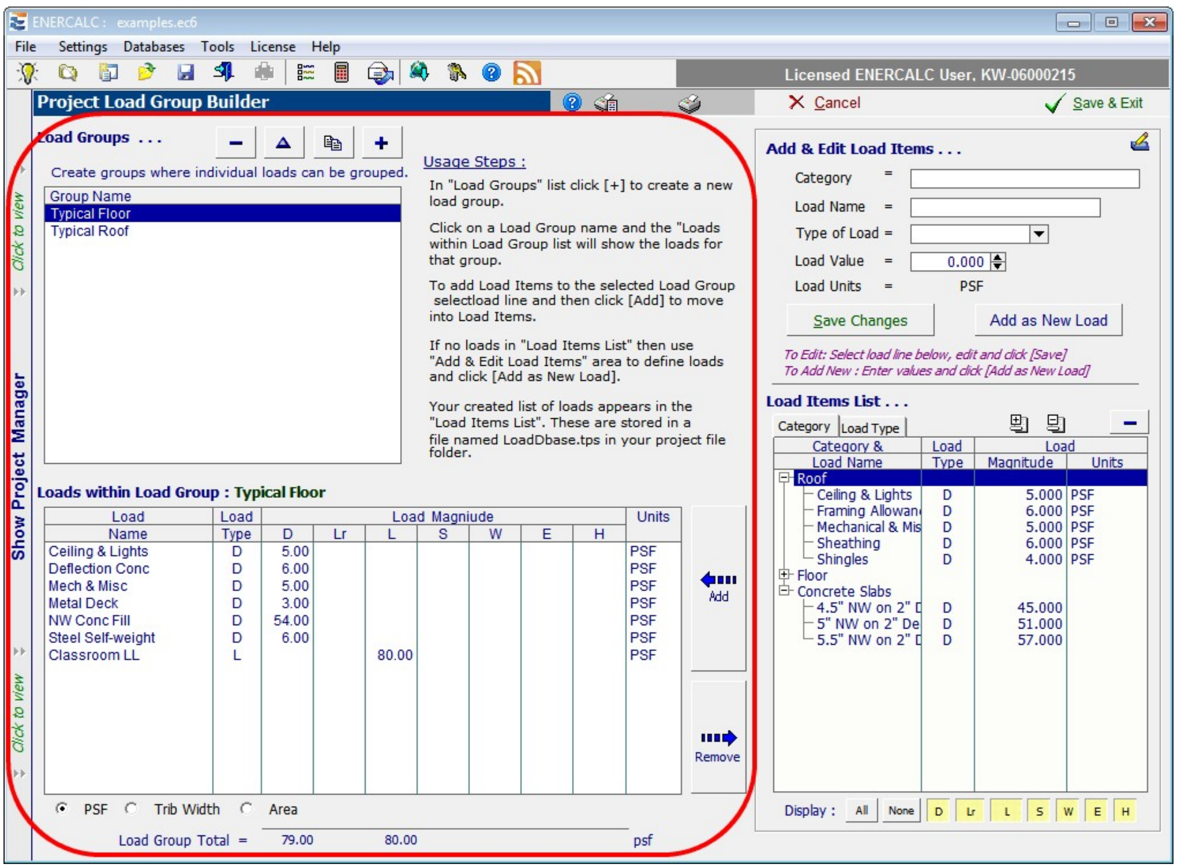
Category: Typical roof structure
- DL of asphalt shingles
- DL allowance for second layer of asphalt shingles
- DL of metal roof deck
- DL structural steel framing
- DL allowance for ceiling & lights
- DL allowance for mechanical & misc.
- SL for sloped roof

And so on...

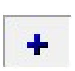
So remember that your Load Items and your Categories are being stored in a separate file, and they are continuing to accumulate from one project to the next as you enter more data. Consider this as you decide on a system of organization that will work best for you.

Now that we have covered Load Items and Categories, we can introduce the *real* purpose for this module, which is the "Load Group". Very simply, a Load Group is a group of Load Items that will be summed together.

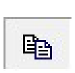
Load Groups are defined and manipulated on the right-hand side of the screen as indicated in the screen capture below:



The Load Groups in the current Project File are listed in the "Group Name" list in the upper left corner of the module. The toolbar buttons above the list perform the following functions:



Adds a new Load Group



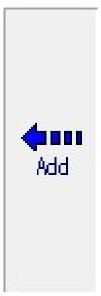
Creates a copy of the selected Load Group

Allows the name of the selected Load Group to be edited

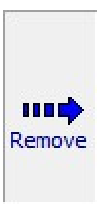
Deletes the selected Load Group

When a Load Group is selected in the "Group Name" list, the details of that Load Group are displayed in the lower left corner of the screen. (A newly created Load Group will display no detailed information until some Load Items are added to the Loaf Group.) The display will show the Load Items that have been added to the current Load Group including the Load Item name, the Load Type (Dead Live, Snow, etc.), the magnitude of the load, and the units.

So we have now covered Load Items and Load Groups. The final detail is...how do we add these Load Items to Load Groups? The answer lies in the buttons shown below:



Adds the selected Load Item to the current Load Group



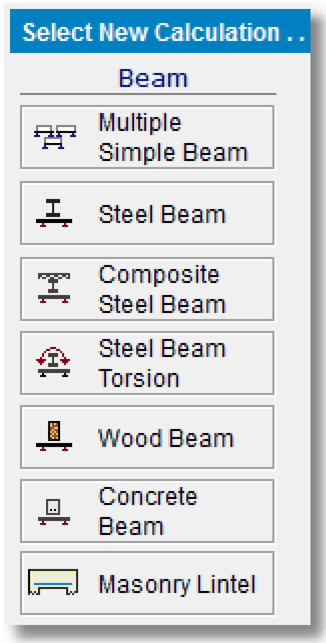
Removes the selected Load Item from the current Load Group

Finally, the Print function will automatically include all defined Load Groups and provide a nice tabular summary of the Load Items that combine to produce each Load Group.

10.2 Beams

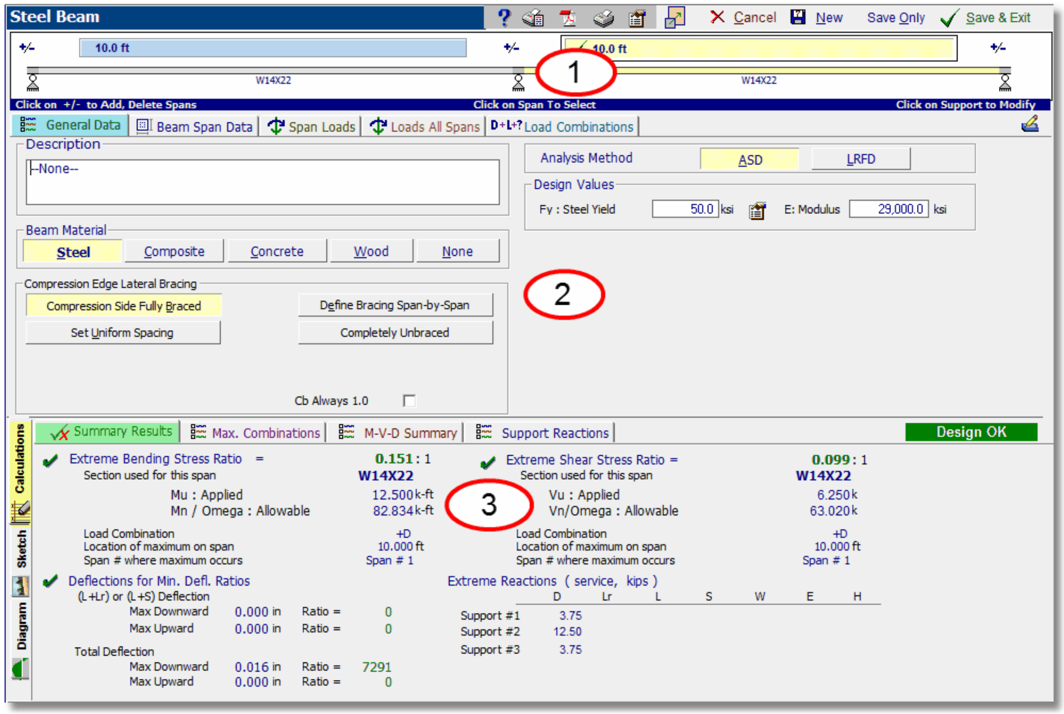
Overview

There is one main module for designing Steel, Concrete and Wood beams. There are separate modules for Composite Steel Beams, Steel Beams with Torsional Loads, and one for Masonry Lintels.

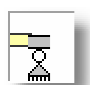


This section deals ONLY with typical single- or multi-span Steel, Concrete & Wood beams.

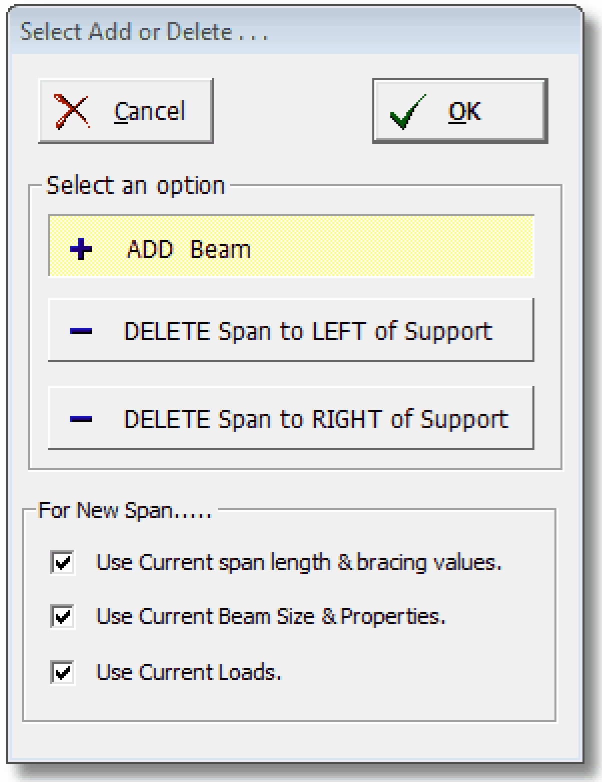
The presentation screen is divided into three areas: **Beam representation & modification**, **Data Entry** & **Calculation Results** as illustrated in the screen capture below:



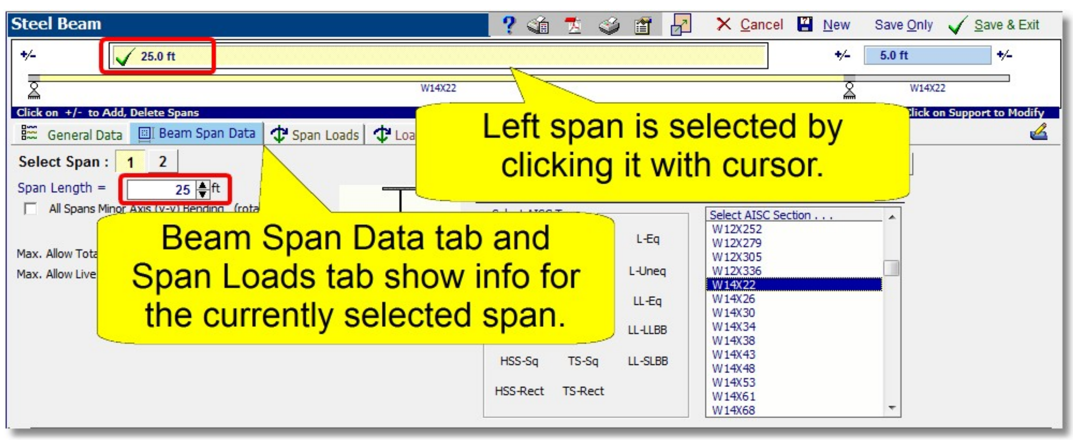
(1) Beam representation & modification: This area allows you to create and modify the beam layout.

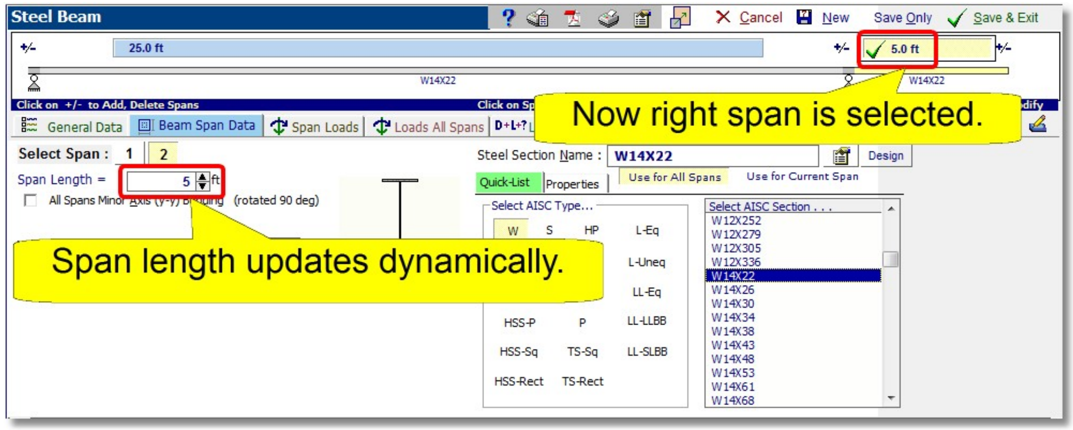
Click on the support icons  to select the type of support fixity: Fixed, Pinned, or Free.

Click on the [**+/-**] icon  to display a window to add or delete beam spans:



Click on the beam representation to change the beam specific data on the **Beam Span Data** and **Span Loads** tabs.



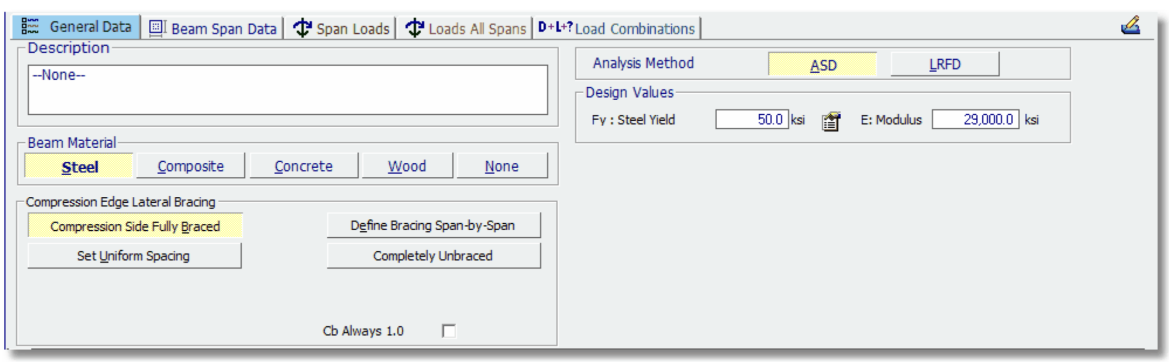


(2) Data Entry: This set of tabs is where you enter all information for the beam. These tabs will show different information according to material type selected. See the specific chapters for the items provided for each material. Click one of the following to jump:

Wood Steel Concrete

Here is a summary of the purpose of each tab:

General Data



Beam Material

Clicking one of these buttons changes the material type used for the beam.

Compression Edge Lateral Bracing

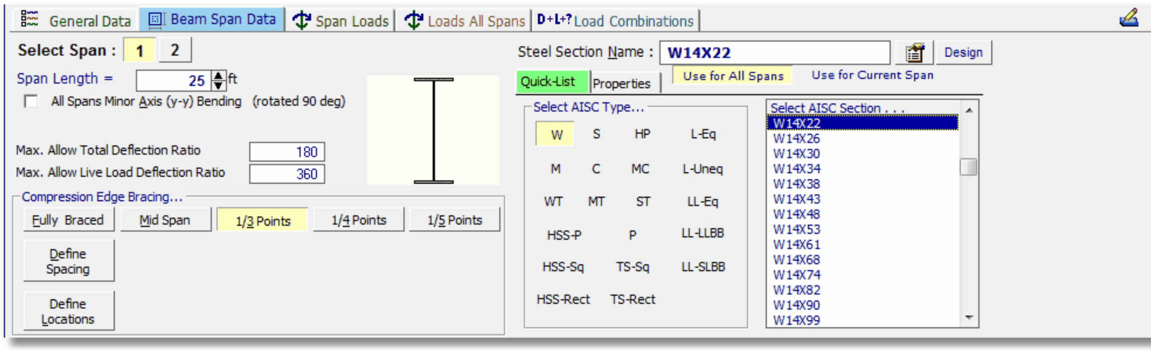
These selections control how the module will evaluate the lateral compression edge bracing for the design. When **Define Bracing Span-by-Span** is selected, the unbraced length is defined on a span-by-span basis on the Beam Span Data tab. When any of the other options are selected, the unbraced length is defined here on the General Data tab, and that definition is applied to all spans of the beam.

Analysis Method

For wood & steel you can select ASD or LRFD design methods. Concrete design is always ultimate strength (LRFD).

Beam Span Data

This tab is used to define span length and beam section information:



Select Span

These buttons allow you to select which span the values in the data entry area apply to.

Span Length

This is where you define the length of the currently selected span.

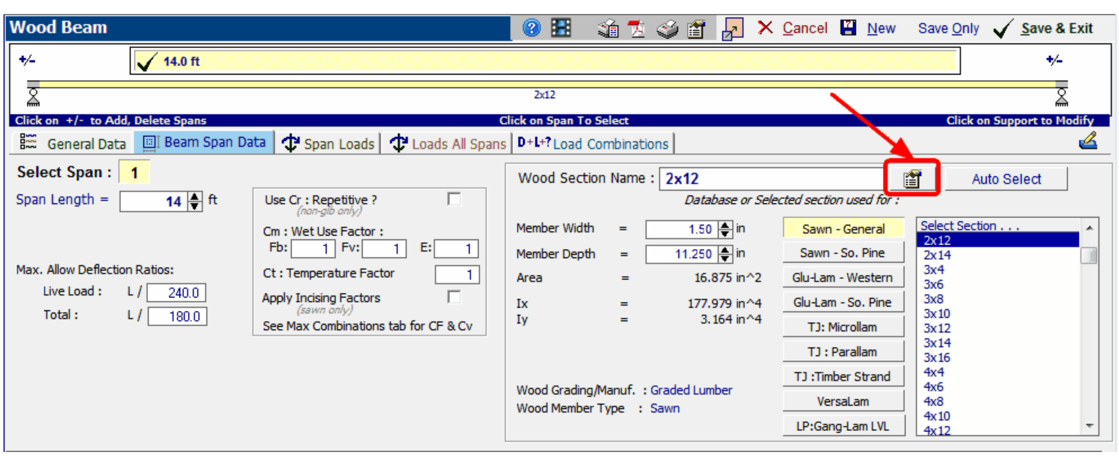
Deflection Ratios

These are used as the basis for the deflection design check and also as the starting point for automatic member selection.

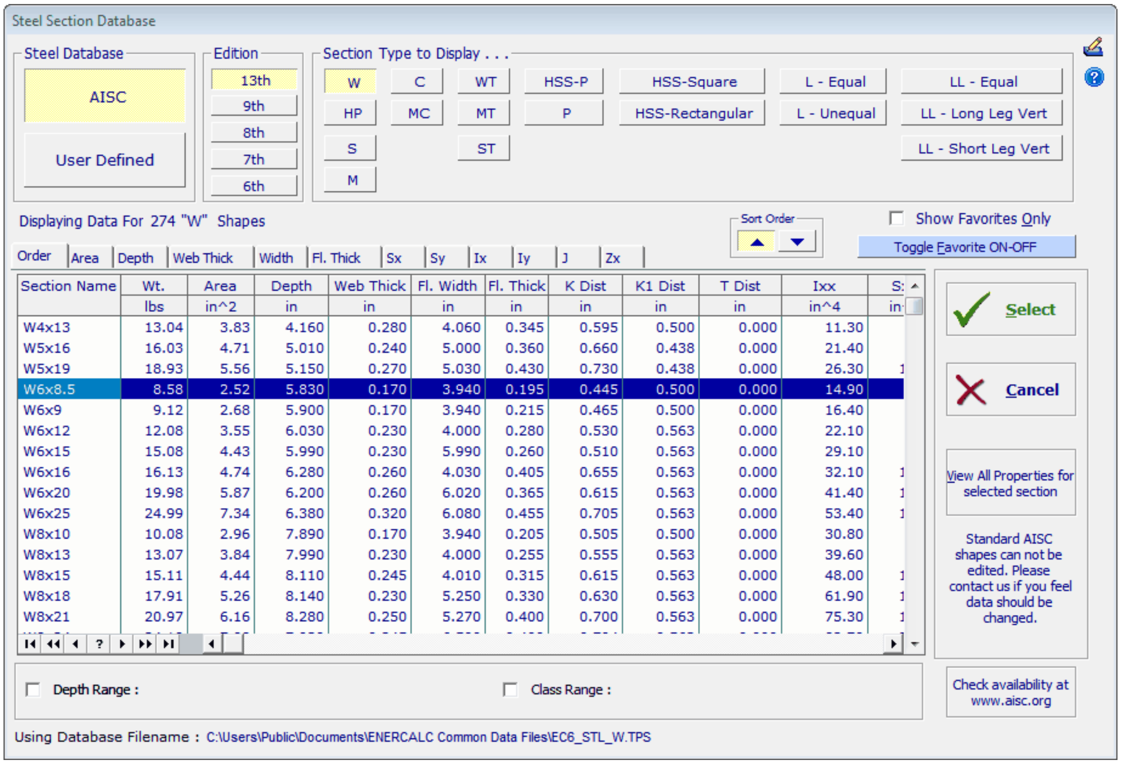
Section Name & Buttons

For steel and wood you can enter the standard beam section designation. You can also type in the section name and the module will search the built-in database for a match. If a match is found, the section properties will be loaded from the database and they will appear on the Properties tab.

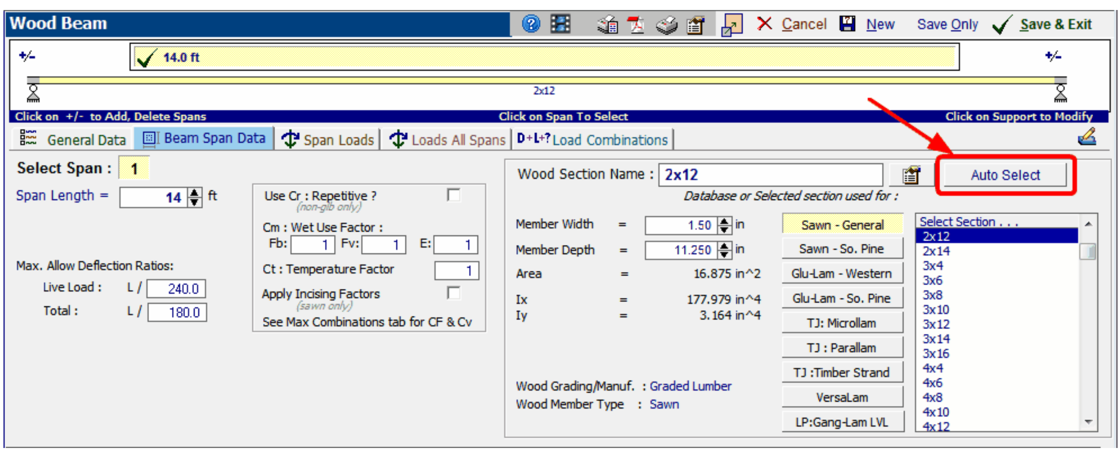
Click the button indicated below to open the section database for rolled steel shapes:



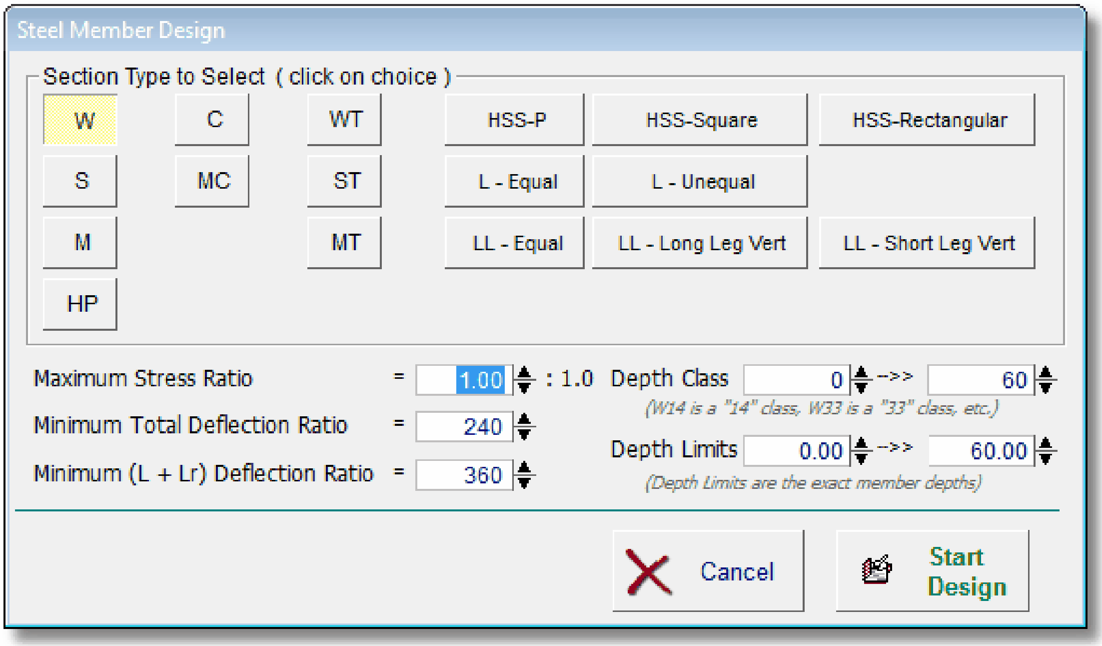
The database contains an extensive number of standard shapes commonly used in the USA.



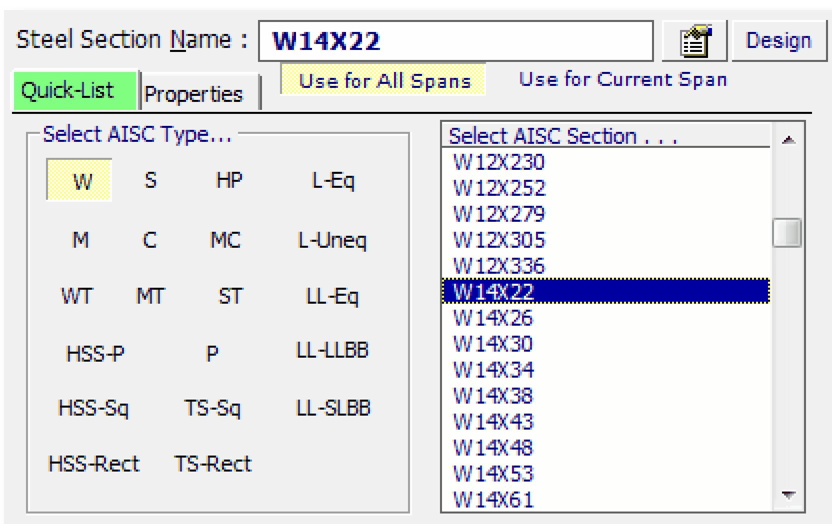
Click the button indicated below to display the Steel Member Design dialog:



This provides control over the type of member to be selected and various stress ratio, deflection ratio and size limits to respect during the automatic member selection process.

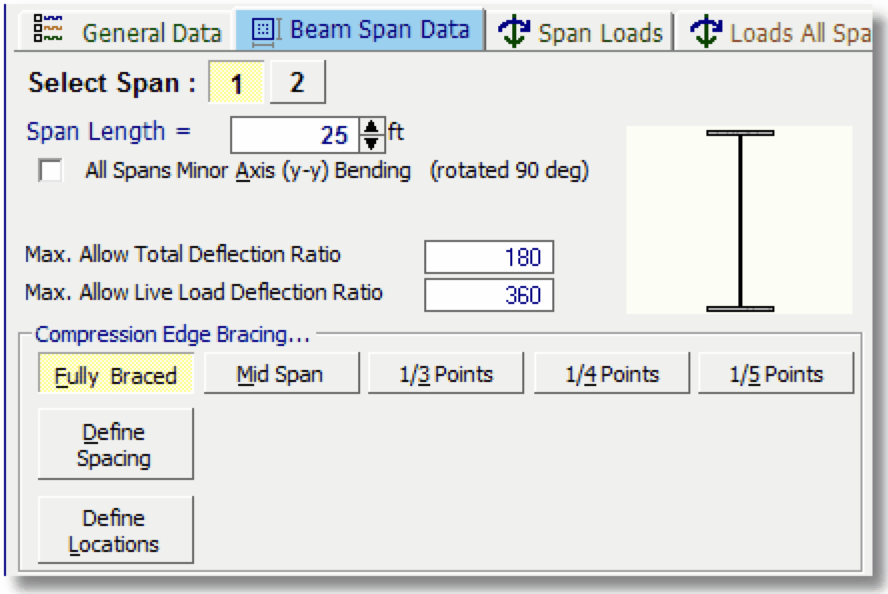


The Quick-List provides a fast way to select a member section from the database. Just select the type of member, scroll through the list and click on your selection.



Compression Edge Lateral Bracing

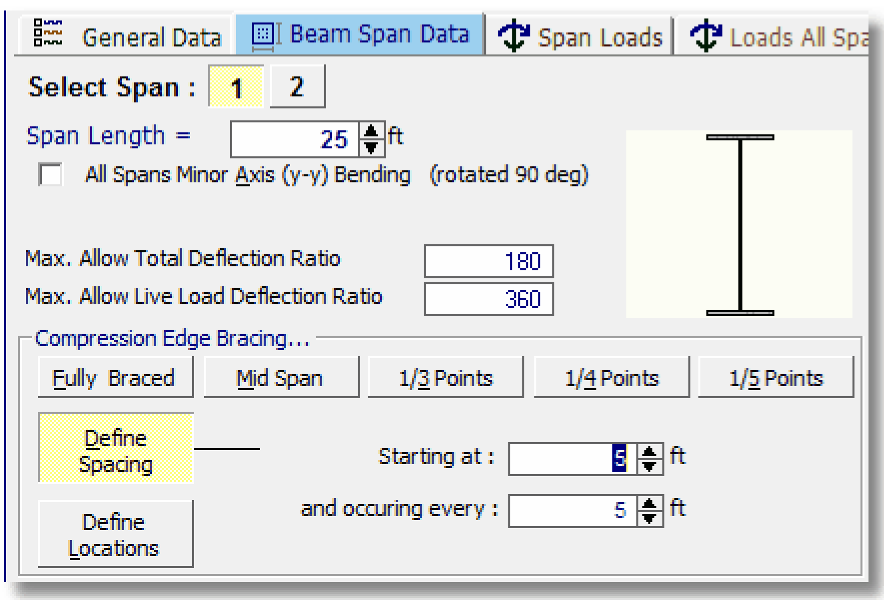
When you have selected Define Bracing Span-by-Span on the General tab you will see the following bracing options on the Beam Span Data tab:



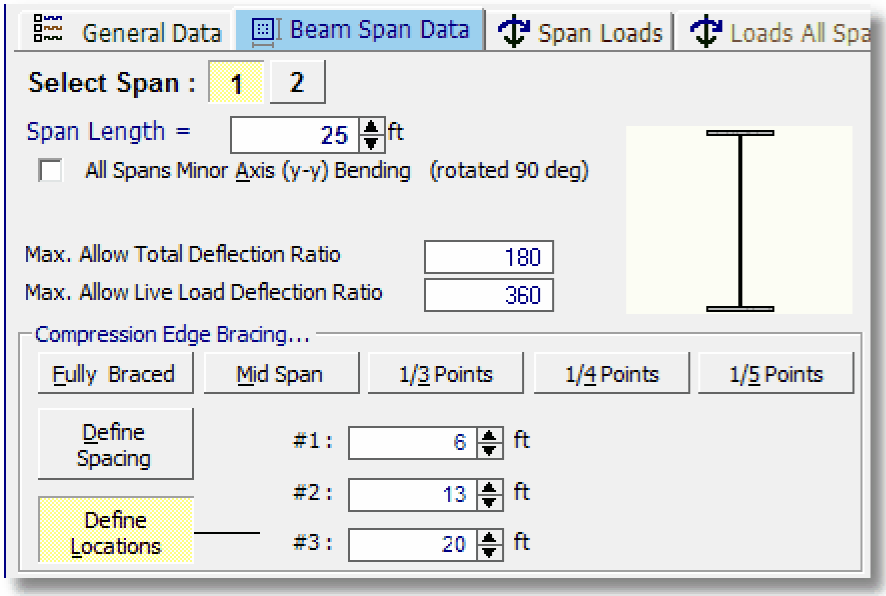
These selections control how the module will evaluate the lateral compression edge bracing for the beam span selected above.

The top row has common selections for fully-braced conditions as well as options to divide the selected span into segments of equal braced length.

The Define Spacing option lets you set a starting point and subsequent spacing for the bracing within the selected span:

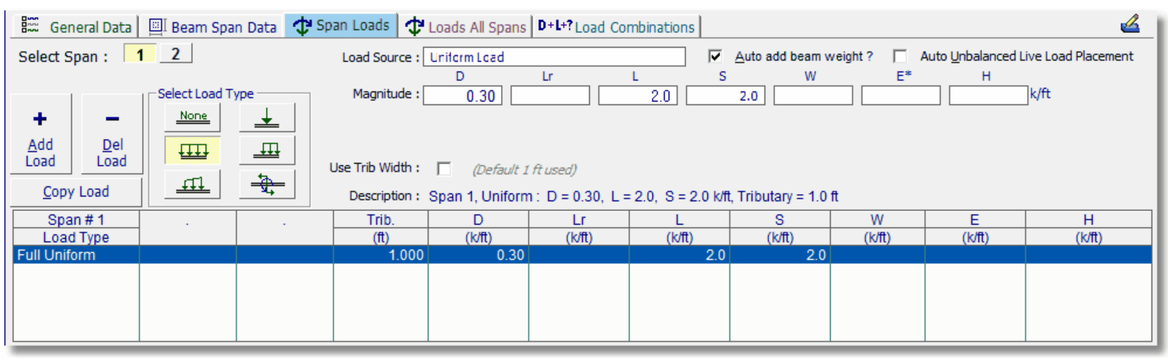


The Define Locations option lets you set up to three specific bracing locations referenced from the left end of the selected span:



Span Loads

This tab is used to specify loads **FOR THE SELECTED SPAN ONLY** (except as noted below) using the tools shown on the following screen capture:



Use the [**Add Load**], [**Copy Load**] and [**Delete Load**] buttons to add, copy, or delete loads on the selected span.

Use the **Load Type** selections to specify the type of load that will be added. This selection affects the **currently highlighted** item in the table of loads. The data entry areas on the right will change based on the type of load selected.

The **Auto Add Beam Weight** option will calculate the weight of the beam and add it to your applied loads as a uniform dead load on the beam. Note that this option applies to the FULL LENGTH of the beam...it is NOT a setting that can be set on a span-by-span basis.

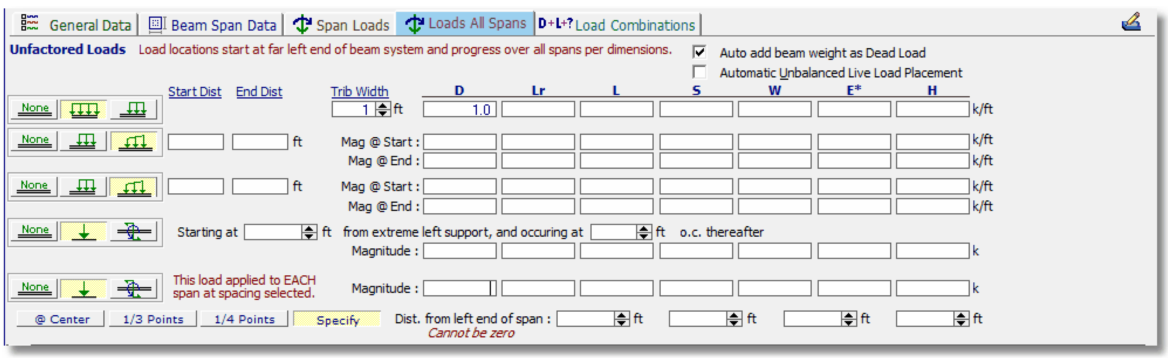
The **Auto Unbalanced Live Load Placement** option is a VERY powerful selection. When you have two or more beam spans, you can select this item and the module will automatically generate load combinations for all possible permutations of patterned live load being placed on alternate spans. For instance, on a two-span beam, it would create conditions that place live load on both spans, live load on the left span only and live load on the right span only. That is a total of three permutations of live load, and it will do this for ALL of the load combinations that are selected to run.

NOTE: This can significantly increase recalculation time needed for beams with many spans.

Load classifications are: D: Dead, Lr: Roof Live, L: Live, S: Snow, W: Wind, E: Earthquake, H: Earth Pressure

Loads All Spans

This tab offers tools that are used to specify loads with spacings or lengths that might cause them to overlap multiple spans as shown in the screen capture below:



Click the load type button on the left and the appropriate load entry items will appear to allow you to define the magnitude, location and extent of the load. With all load items set to [**None**], the tab will be almost entirely blank.

Start Dist and **End Dist** defines the application distance from the FAR LEFT end of the beam. For a beam with two 20' spans where you want to apply a uniform load 5' in from each end use Start Dist = 5.00 and End Dist = 35.00.

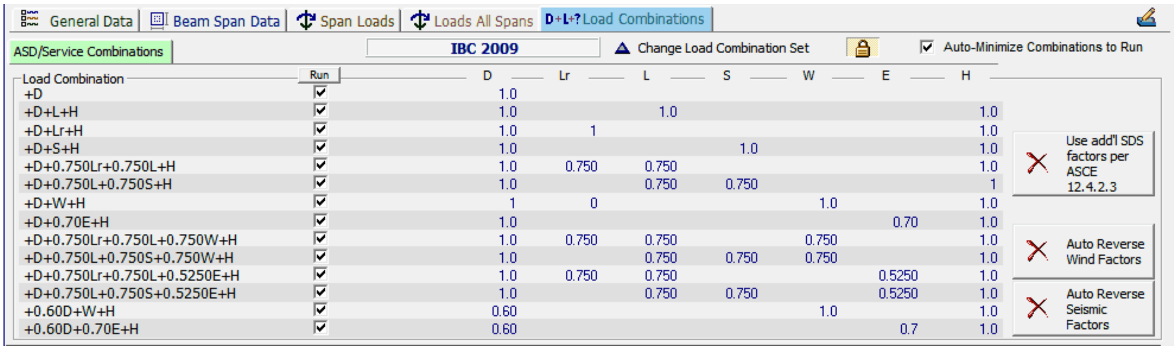
The third item allows you to define repeating point loads or moments. You define the position of the first load and the spacing increment thereafter.

The last item allows you to define repeating loads on EACH span. Load specification is for the load in EACH span. For example, consider a two-span beam where the first span is 25' and the second span is 45'. Selecting [**1/3 Points**] will place loads at 8.33', 16.66' from the left support of the first span and at 15' and 30' from the left support of the second span. The "Specify" option allows you to provide a unique spacing measured from the left end of each span.

Although this tab is not typically used for single-span beams, the tools are perfectly applicable to single-span beams.

Load Combinations

This tab is used to specify the load combinations that will be run for the analysis of this beam, as shown in the screen capture below:



Load Combination	Run	D	Lr	L	S	W	E	H
+D	☑	1.0						
+D+L+H	☑	1.0		1.0				1.0
+D+Lr+H	☑	1.0	1					1.0
+D+S+H	☑	1.0			1.0			1.0
+D+0.750Lr+0.750L+H	☑	1.0	0.750	0.750				1.0
+D+0.750L+0.750S+H	☑	1.0		0.750	0.750			1
+D+W+H	☑	1	0			1.0		1.0
+D+0.70E+H	☑	1.0					0.70	1.0
+D+0.750Lr+0.750L+0.750W+H	☑	1.0	0.750	0.750		0.750		1.0
+D+0.750L+0.750S+0.750W+H	☑	1.0		0.750	0.750	0.750		1.0
+D+0.750Lr+0.750L+0.5250E+H	☑	1.0	0.750	0.750			0.5250	1.0
+D+0.750L+0.750S+0.5250E+H	☑	1.0		0.750	0.750		0.5250	1.0
+0.60D+W+H	☑	0.60				1.0		1.0
+0.60D+0.70E+H	☑	0.60					0.7	1.0

The **[Change Load Combination Set]** button is used to retrieve load combinations sets from the Load Combination 84 database.

The lock symbol is used to enable editing of the load combination values. Click on the lock symbol, and the multipliers for the load types will change to data entry items. After you change an item's value and press [**Tab**], you will see the Load Combination description at the left end of the line change to reflect your entered values.

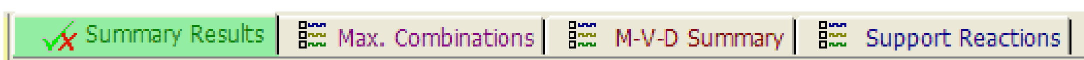
[**Use Add'l...**] button, when clicked, displays a data entry item where you can specify the magnitude of S_{DS} to use as an additional internal factor in the load combinations.

[**Auto Reverse Wind Factors**] and [**Auto Reverse Seismic Factors**] buttons trigger the module to create additional load combinations with the "**W**" and "**E**" factors set to negative values. This has the effect of reversing the direction of application of the wind and seismic loads that have been applied to the member.

(3) Calculation Result: This set of tabs provides detailed results for the current calculation.

The vertical tabs on the left edge of the screen allow you to select the three major areas available for review:

Calculations provides several tabs that let you review the numeric detail of the calculation. The left-most tab is always the Summary where the concise design is given.



See the individual sections for each material type for specific discussions of these result sections.

Sketch provides a scale illustration of the item you are designing, including an indication of support conditions and applied load magnitudes.

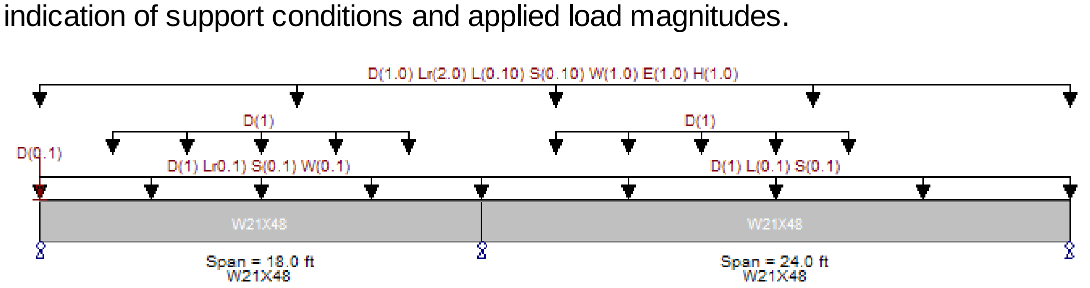
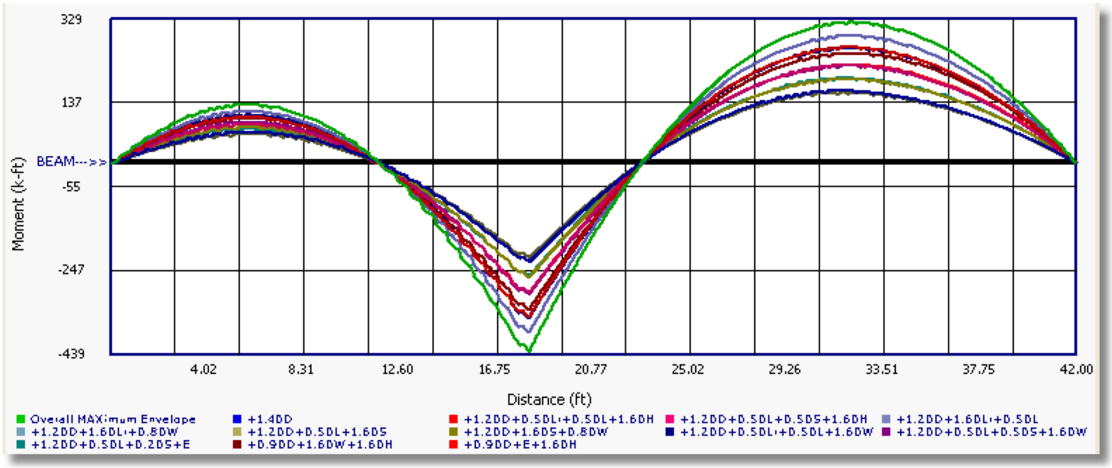


Diagram provides a moment, shear, or deflection diagram for the item you are designing.

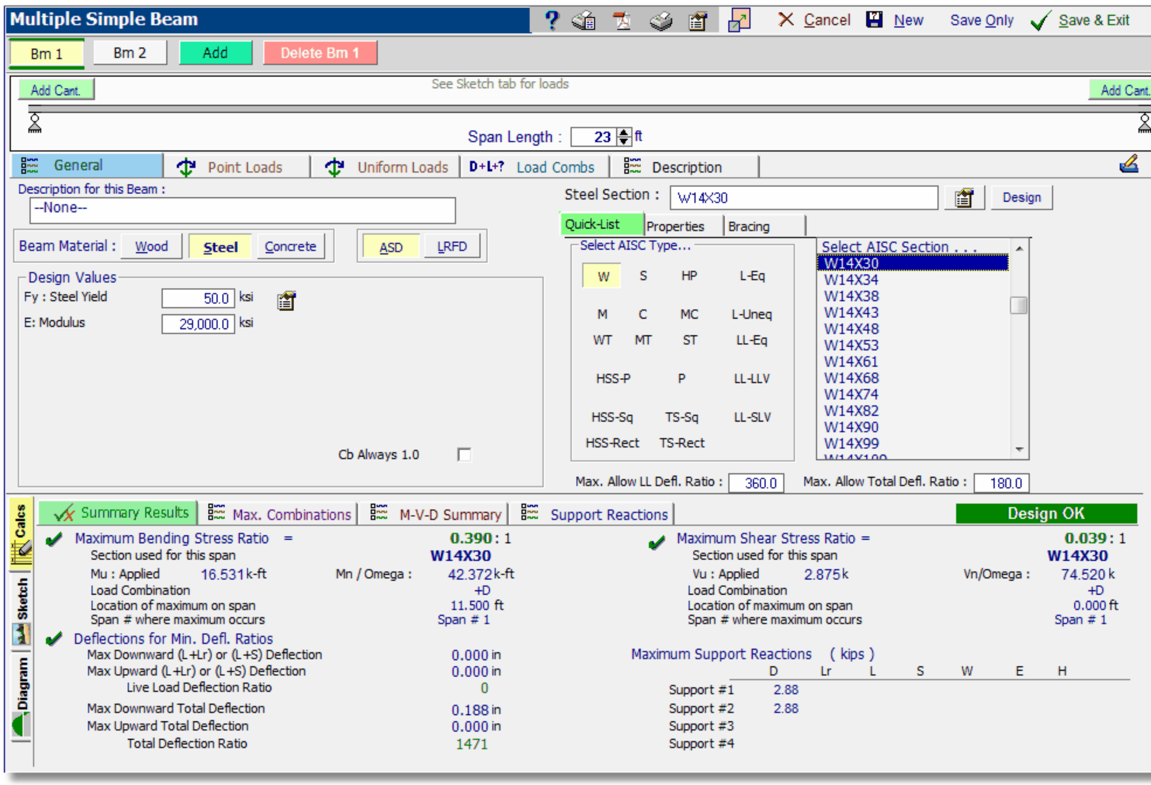

329
137
BEAM--->>
-55
-247
-439
Moment (k-ft)
4.02
8.31
12.60
16.75
20.77
25.02
29.26
33.51
37.75
42.00
Distance (ft)
Overall MAXimum Envelope
+1.40D
+1.20D+0.50Lr+0.50L+1.60H
+1.20D+0.50L+0.50S+1.60H
+1.20D+1.60Lr+0.50L
+1.20D+1.60Lr+0.80W
+1.20D+0.50L+1.60S
+1.20D+1.60S+0.80W
+1.20D+0.50Lr+0.50L+1.60W
+1.20D+0.50L+0.50S+1.60W
+1.20D+0.50L+0.20S+E
+0.90D+1.60W+1.60H
+0.90D+E+1.60H

10.2.1 Multiple Simple Beam

Need more? Ask Us a Question

This module is specifically designed to provide rapid analysis and design of simple beams. For complex, multi-span beams please use the other beam modules for wood, steel and concrete beams.

This module has an added section above the beam representation that allows you to add and select up to 12 beams to design. This can be seen in the screen capture below:



Looking at the screen capture above you can see:

(1) The button in yellow with green lines above and below represents the currently selected beam for which data is displayed.
(2) The button labeled [**Bm 2**] represents the second beam that was defined. Clicking on that button will save all of the data for [**Bm 1**] and display all the data for [**Bm 2**].
(3) [**Add**] is used to add another beam to this calculation. When two beams have already been defined, clicking [**Add**] will add a button labeled [**Bm 3**].
(4) [**Delete Bm X**] is used to delete the noted beam. It will delete the beam that is currently selected.

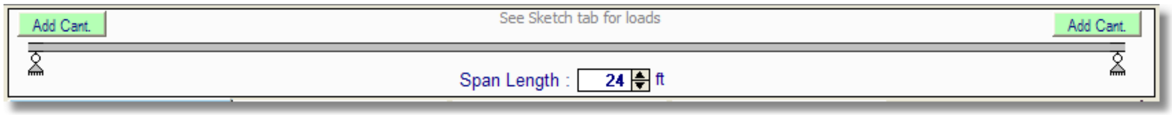
Directly below the band of beam selection buttons is the graphic that shows the basic layout of the currently selected beam. See below for a more detailed description.

Note: In this module, the beam span length is always specified on the beam layout graphic.

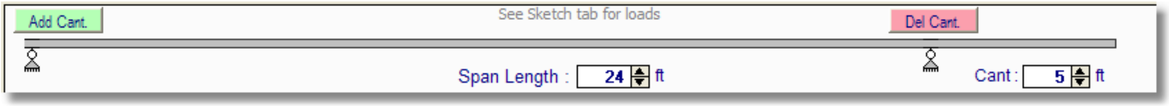
Beam Layout Graphic

This area shows a graphic representation of the selected beam. There are several variations of beams that you can specify using the [**Add Cant**] buttons and also by clicking on the end support graphics.

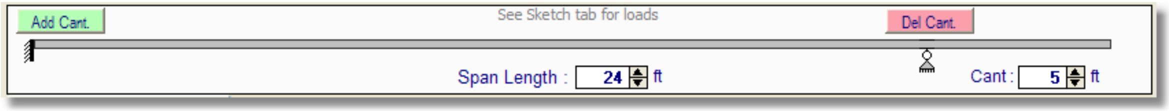
Here is a basic simple span beam:



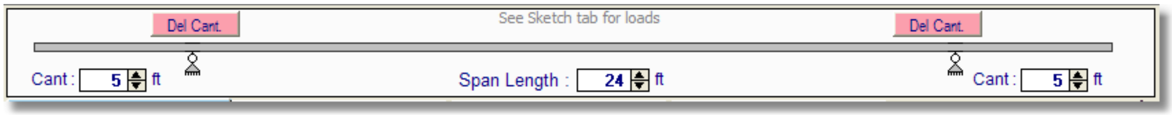
Clicking on the right [**Add Cant**] button in the image above adds a single cantilever at the right support as shown below:



Clicking on the left support icon in the image above displays a selection box so you can select a fixed end. Doing this sets the left support to fixed as shown below:



Clicking on the left [**Add Cant**] button in the image above adds a single cantilever at the left support, resulting in the double cantilever as shown below:

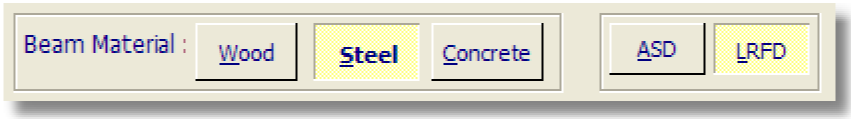


Note that adding the cantilever on the left also had the effect of automatically revising the left support back to a pinned condition.

General Data Tab

This tab is where you select the beam material, allowable stresses, beam size, and set the beam bracing layout.

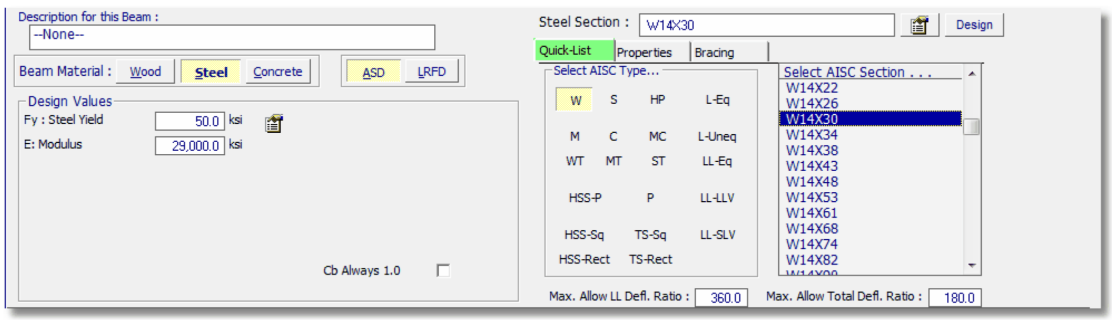
For steel and wood you can select ASD or LRFD design methods. For concrete, only strength design is available.



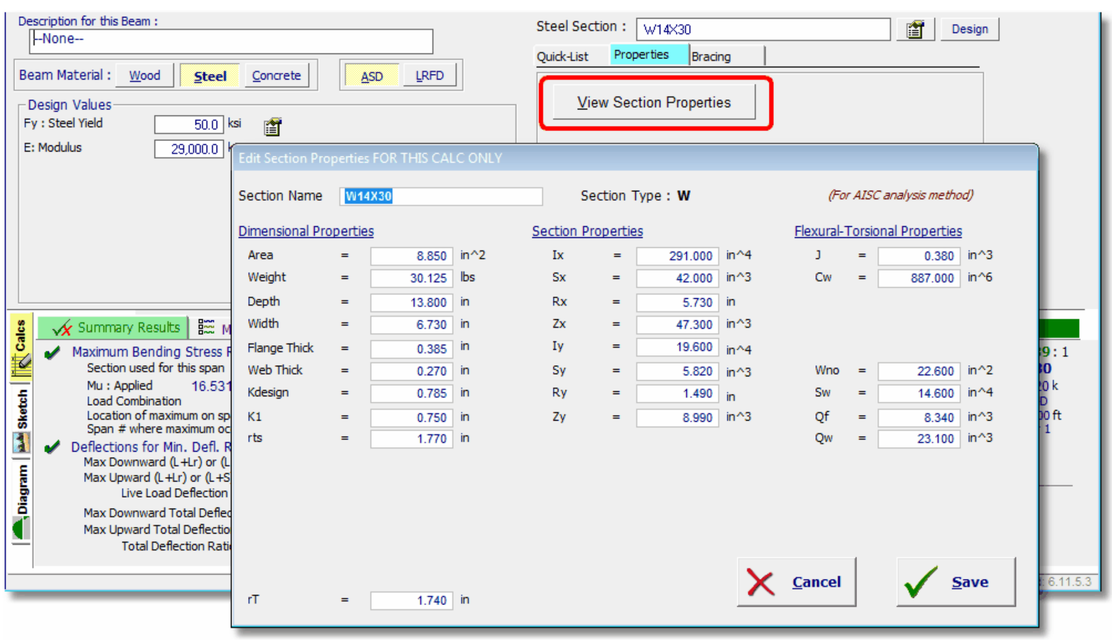
Steel Specific Tab Items

Design Values: This area enables you to enter the yield strength and elastic modulus of the steel. The paper-with-hand icon gives you access to the available AISC steel stress grades.

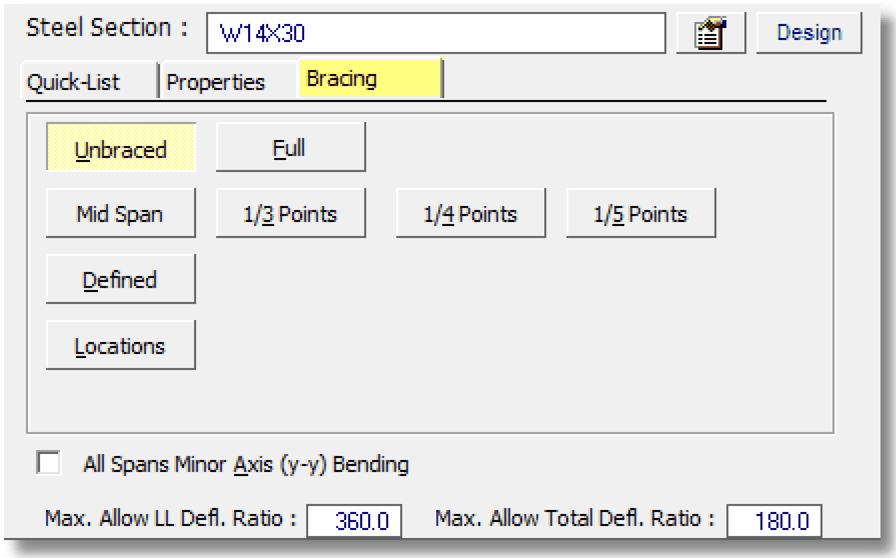
Quick-List Tab: This tab provides quick access to the built-in 13th Edition AISC steel section list. Clicking a section name like [**W**] or [**HP**] in the Select AISC Type category will display all the steel sections of that type in the list to the right.



Properties Tab: This tab provides a [**View Section Properties**] button that when clicked displays all of the design values for the AISC section you have selected.



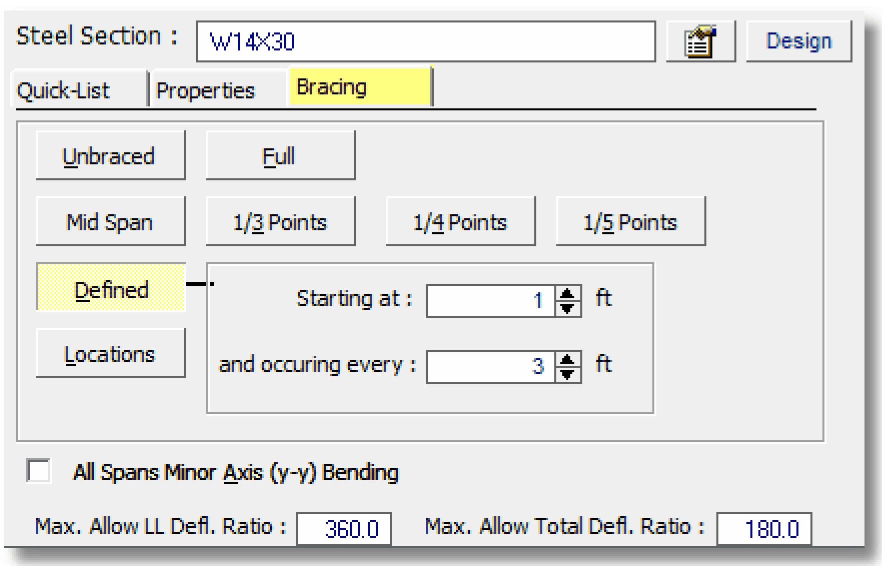
Bracing Tab: This tab allows you to select how your beam is braced against lateral-torsional buckling. Defined brace points are automatically assumed to brace both the top and the bottom flanges.

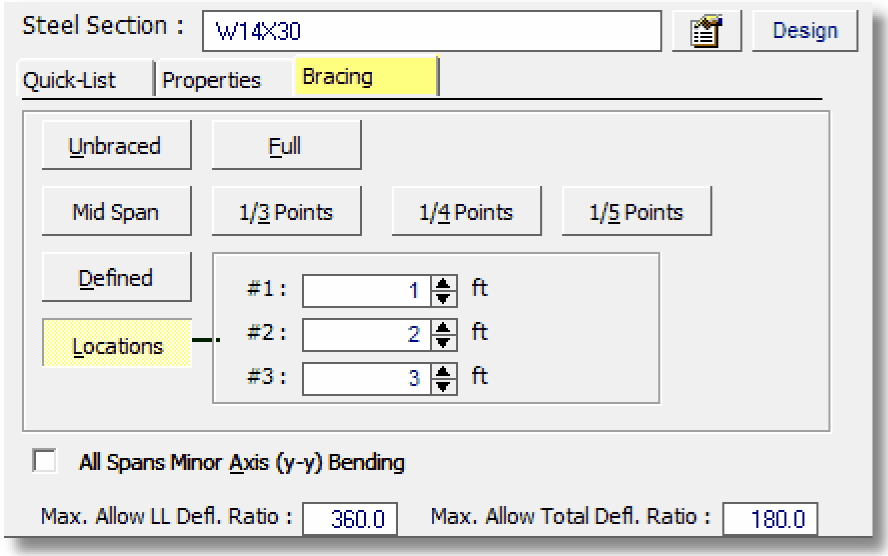


The first line offers the most basic options.....fully unbraced or fully braced.

The next line offers uniform brace spacing options.

The last two buttons offer the ability to specify brace spacings from a starting point or enable you to specify selected brace points, as shown below:

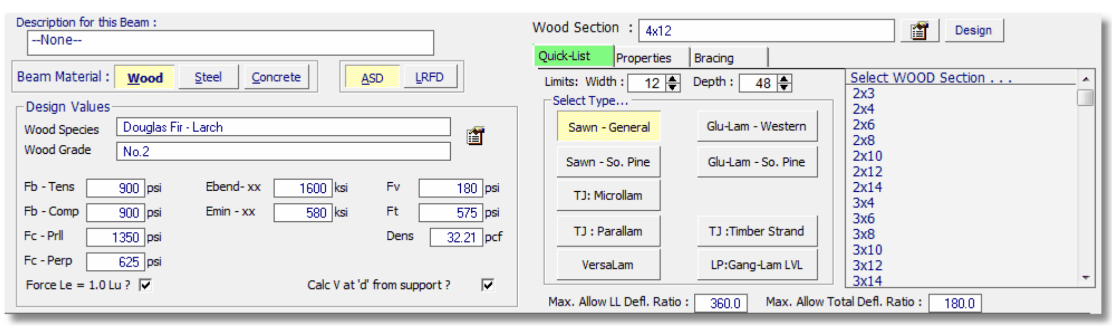




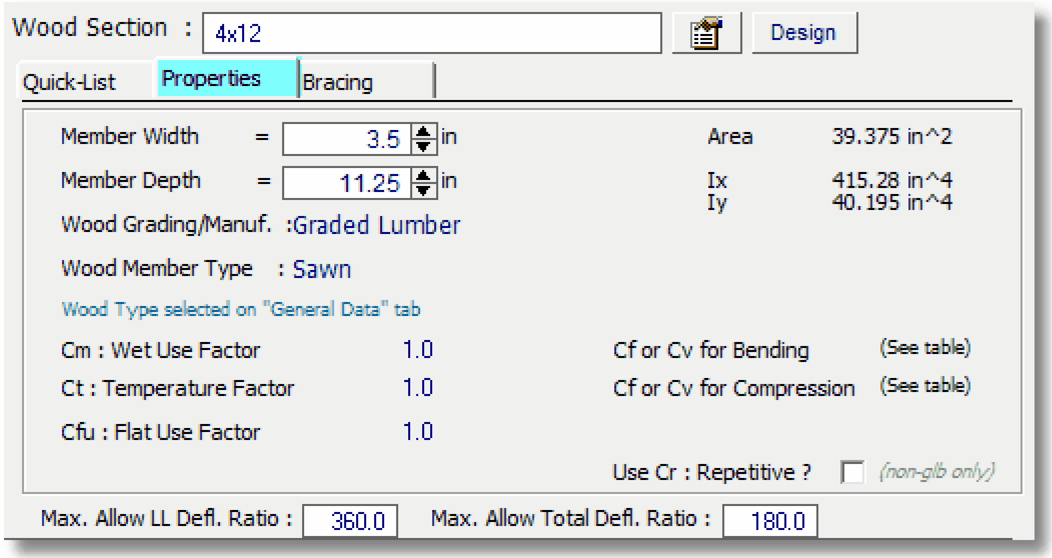
Wood Specific Tab Items

Design Values: This area enables you to specify the design values for the wood species & grade you want to use. Click the  button to access the built-in NDS allowable stresses database.

Quick-List Tab: This tab provides quick access to the database of wood sections. Clicking a section type button like [**Sawn-General**] or [**TJ:Microllam**] in the Select Type category will display all the sections of that type in the list to the right.



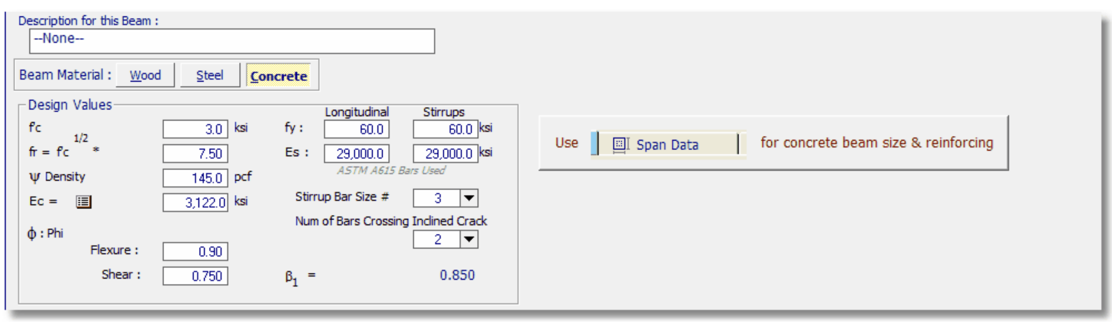
Properties Tab: This tab provides the values for the wood section you have chosen. You can enter different numbers here to modify the section.



Bracing Tab: See information provided above in the Steel section for a summary of bracing options.

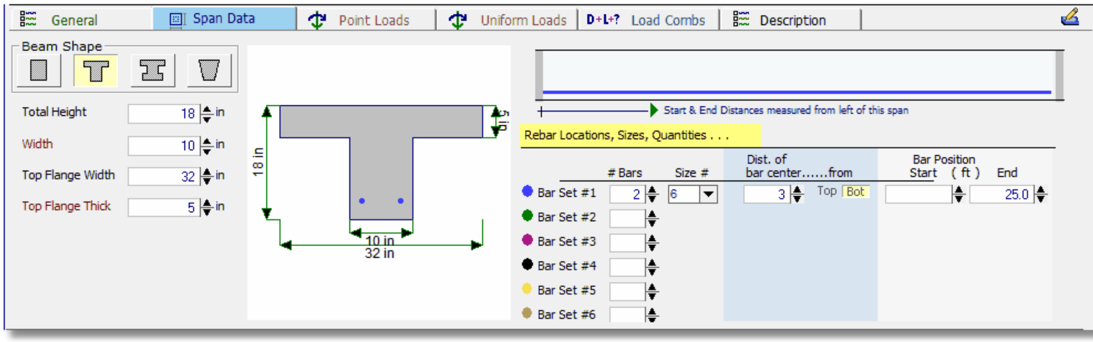
Concrete Specific Tab Items

Design Values: This area enables you to enter concrete and reinforcing strengths for the beam. In addition you can specify the stirrup size and Phi values.

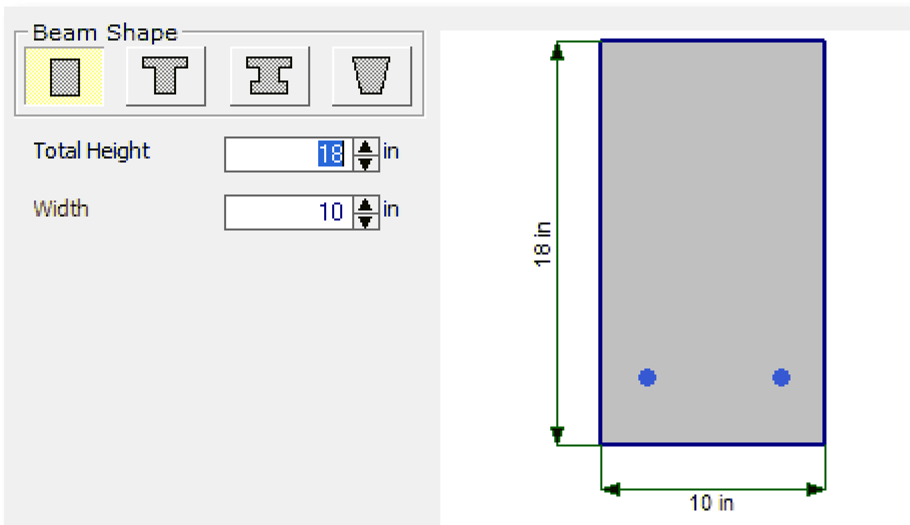


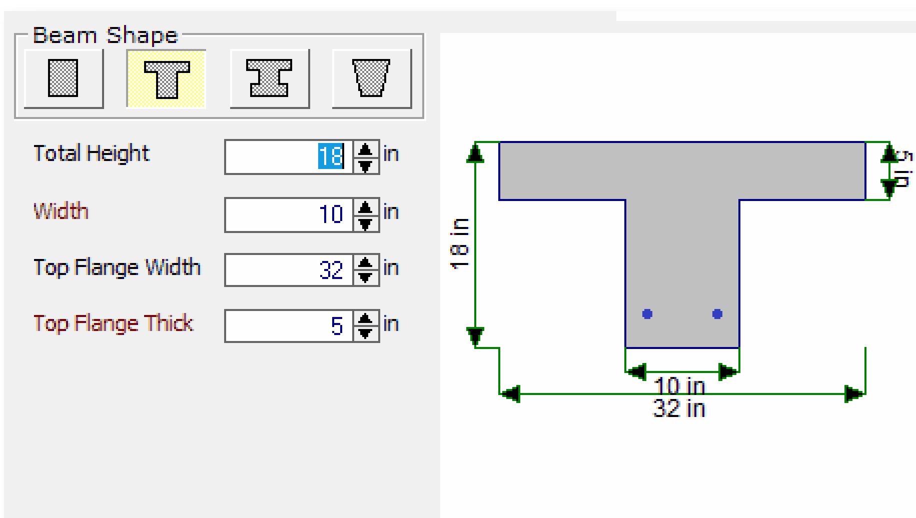
Span Data

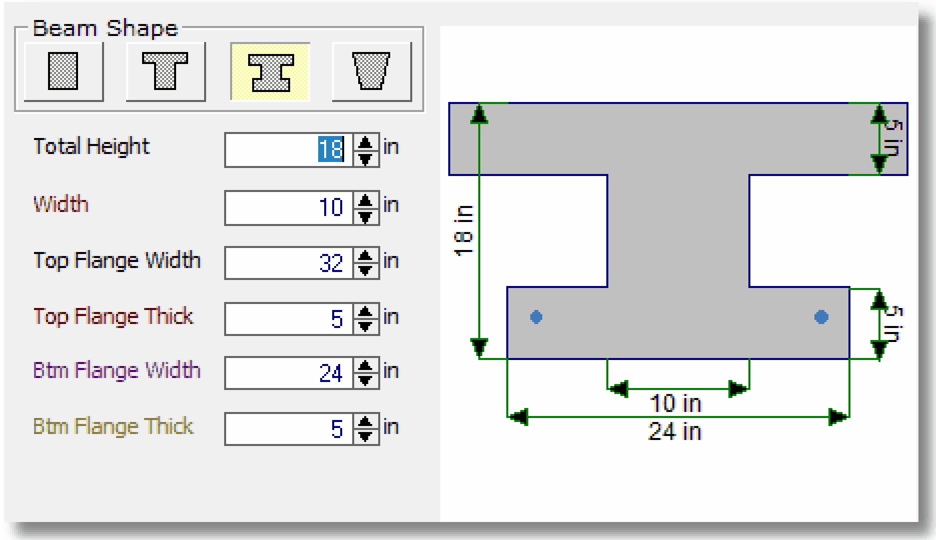
When a concrete beam is chosen, a new tab named **Span Data** will be added to the user interface (see below). This tab is where you specify the beam cross section and reinforcing. This tab will only be shown when the material is set to concrete.

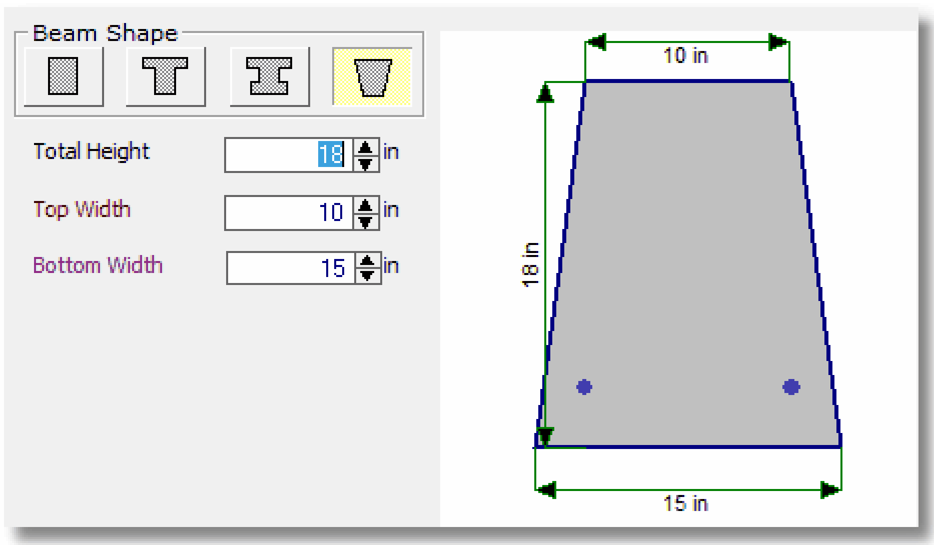


Beam Shape: You can select between 4 beam shapes:









On the right side of the tab you can specify up to 6 bars set (quantity, size, vertical location and start/stop endpoints).
Each bar set is referenced on the sketch with a color shown as a dot to the left of the set description.

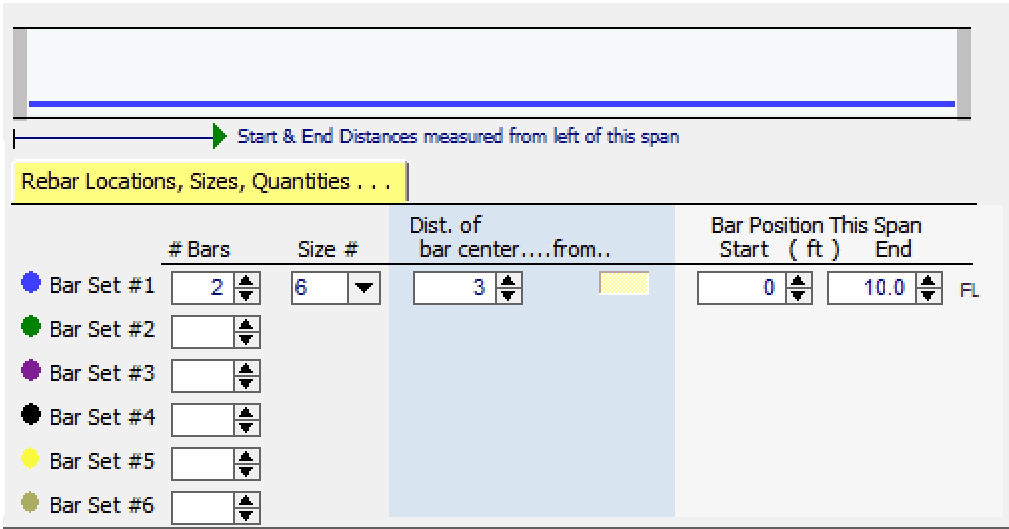
The column highlighted in light blue titled "Dist of bar center....from...." is how you set the vertical position of the bars set in the beam.

When you look at the top one you can read it as "The top bar set is 3 inches from the bottom of the beam". Note that the module will know whether the bars are in tension or compression and handle the calculations properly.

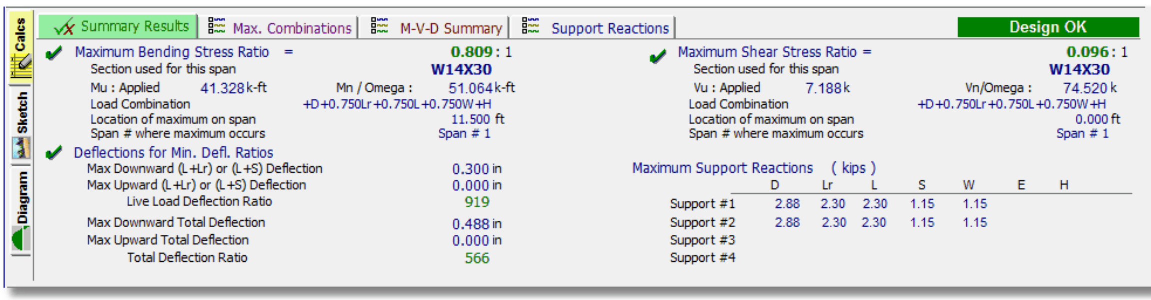
The item named Bar Position This Span defines the starting and ending location of the bar's ends with respect to this span's left support.

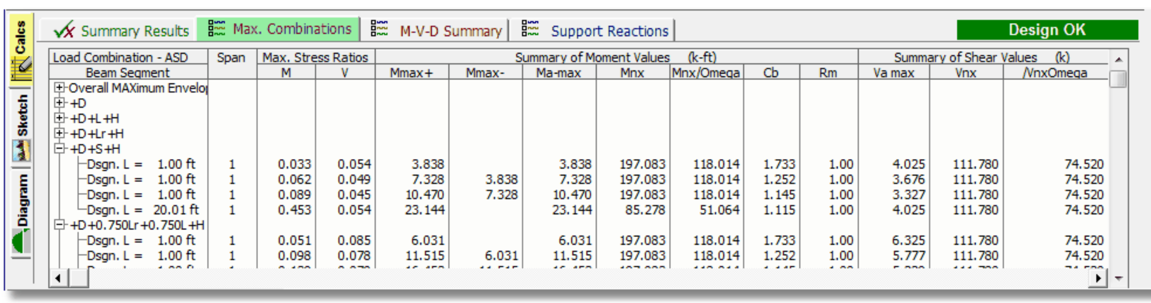
The data in the screen capture below shows that bar set #1 runs from the left end (0.0 ft) to 10.0 ft from the left support.

Using these starting and ending locations you can fine tune the bar layout and end cutoffs.

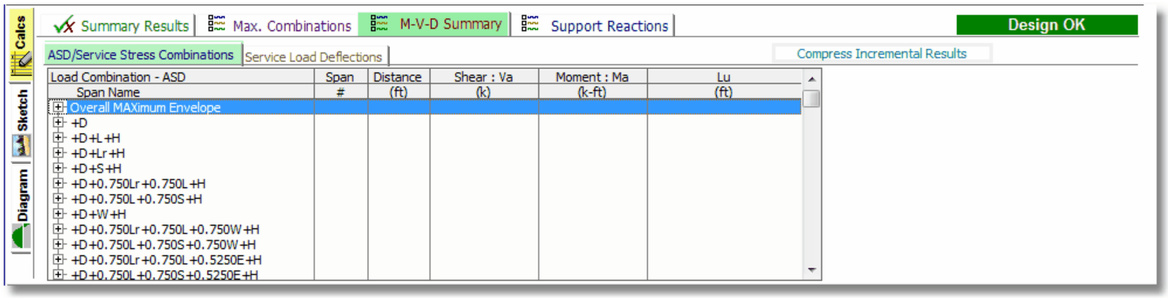


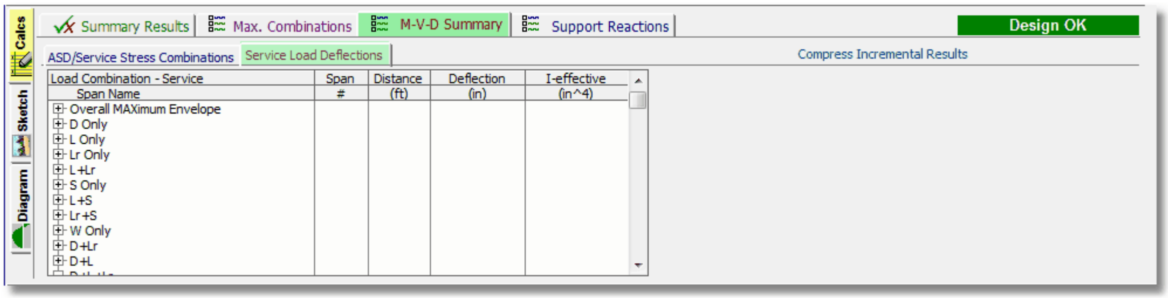
Results Tabs



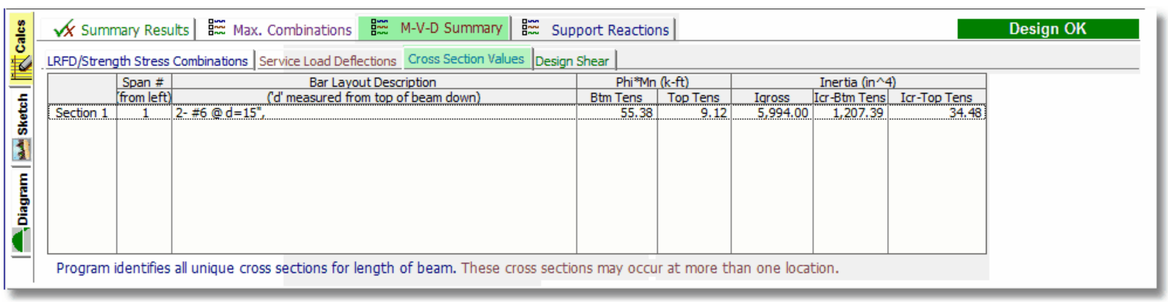


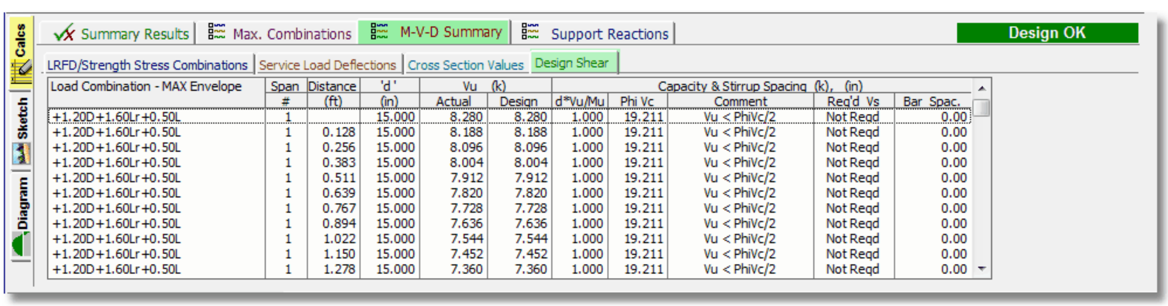
Load Combination - ASD	Span	Max. Stress Ratios		Summary of Moment Values (k-ft)							Summary of Shear Values (k)		
Beam Segment		M	V	Mmax+	Mmax-	Ma-max	Mnx	Mnx/Omega	Cb	Rm	Va max	Vnx	/VnxOmega
⊞Overall MAXimum Envelo													
⊞+D													
⊞+D+L+H													
⊞+D+Lr+H													
⊟+D+S+H													
Dsgn. L = 1.00 ft	1	0.033	0.054	3.838		3.838	197.083	118.014	1.733	1.00	4.025	111.780	74.520
Dsgn. L = 1.00 ft	1	0.062	0.049	7.328	3.838	7.328	197.083	118.014	1.252	1.00	3.676	111.780	74.520
Dsgn. L = 1.00 ft	1	0.089	0.045	10.470	7.328	10.470	197.083	118.014	1.145	1.00	3.327	111.780	74.520
Dsgn. L = 20.01 ft	1	0.453	0.054	23.144		23.144	85.278	51.064	1.115	1.00	4.025	111.780	74.520
⊟+D+0.750Lr+0.750L+H													
Dsgn. L = 1.00 ft	1	0.051	0.085	6.031		6.031	197.083	118.014	1.733	1.00	6.325	111.780	74.520
Dsgn. L = 1.00 ft	1	0.098	0.078	11.515	6.031	11.515	197.083	118.014	1.252	1.00	5.777	111.780	74.520

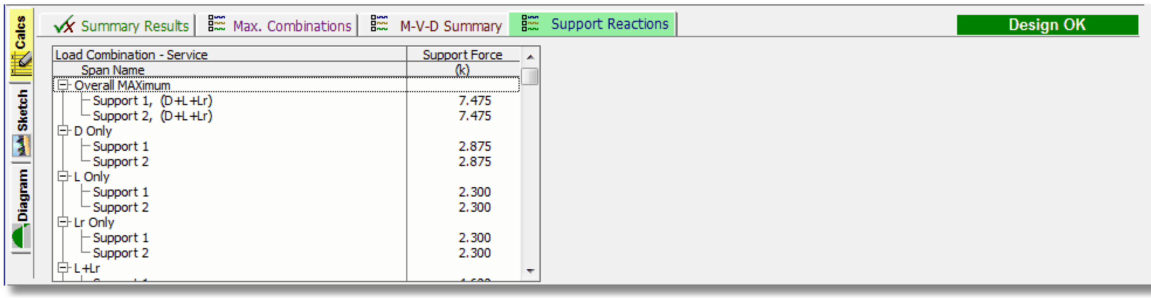




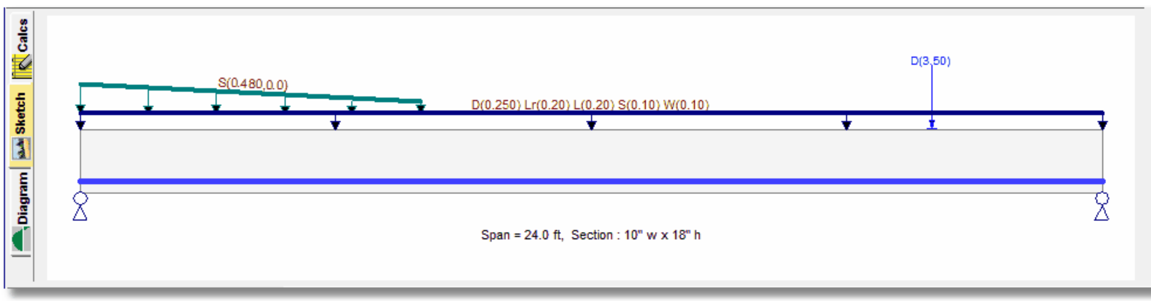
And for concrete beams these two additional sub-tabs are also visible:



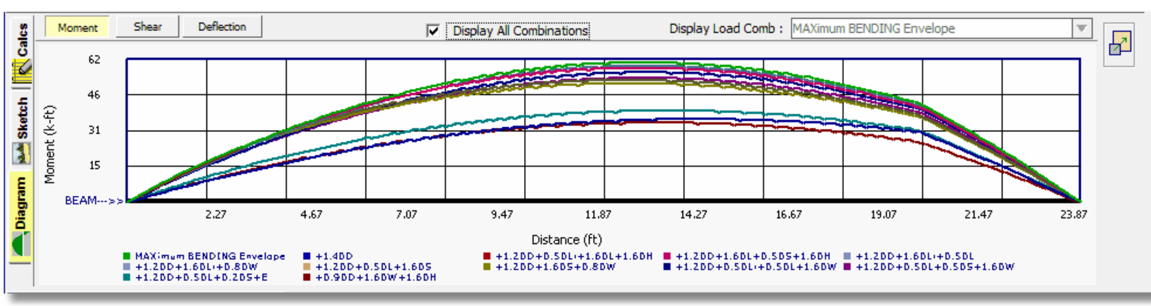




Sketch



Diagrams



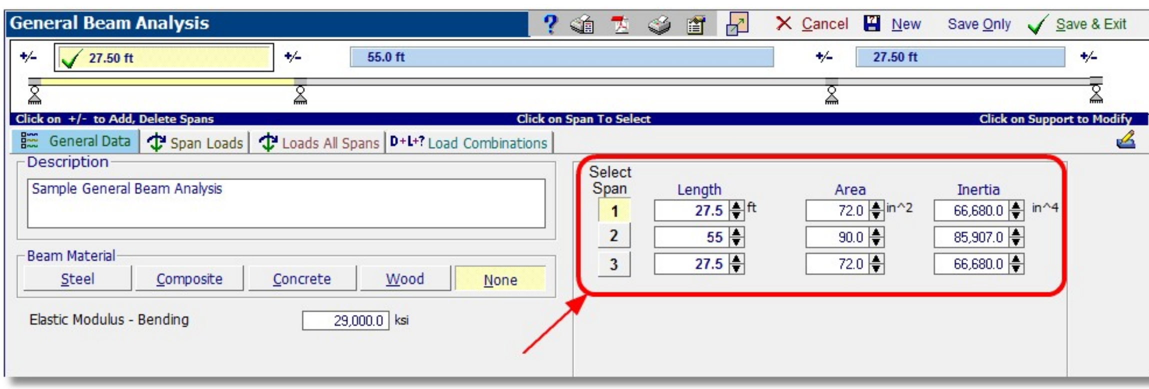
10.2.2 General Beam Analysis

Need more? Ask Us a Question

The General Beam Analysis module offers beam analysis functionality but does not incorporate any design processes. In this way, it can be a useful tool for situations where only analysis results are desired, such as shear, moment, reactions, and deflections.

General Data tab:

The General Data tab allows you to set the span conditions, span lengths, and support conditions in much the same way that this information is provided in the other beam modules. Refer to the Beams topic for additional explanation. In addition to these pieces of data, the General Data tab also provides input fields for the elastic modulus for bending, and the cross sectional area and moment of inertia of each span of the beam, as shown below:



Span Loads tab:

The Span Loads tab allows you to specify loads on one span at a time. The behavior of the tools on this tab is identical to the tools described for use in the other beam modules. Refer to the Beams topic for additional explanation.

Loads All Spans tab:

The Loads All Spans tab allows you to specify loads on all spans at the same time. The behavior of the tools on this tab is identical to the tools described for use in the other beam modules. Refer to the Beams topic for additional explanation.

Load Combinations tab:

The Load Combinations tab provides a view of the load combinations that will be analyzed. It also offers the ability to:

- Select a different set of load combinations,
- Modify the values used as load factors, and
- Turn certain combinations on and off.

Refer to the Beams topic for additional explanation.

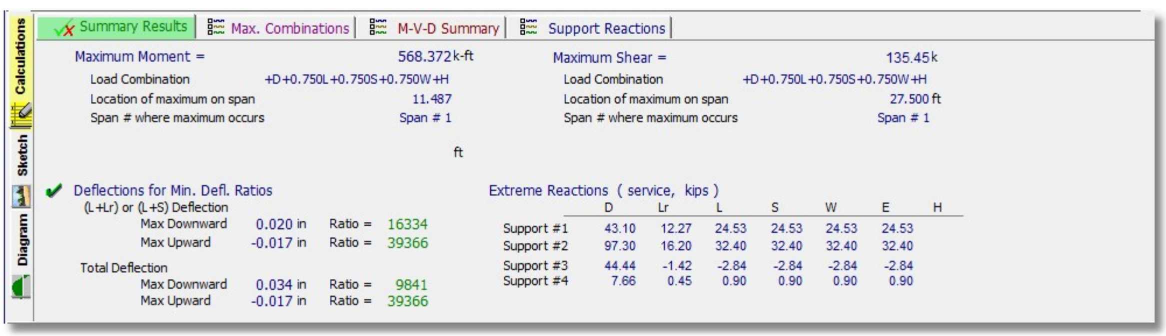
The General Beam Analysis module offers output options that are analogous to the output options provided by the other beam modules, with the exception that no design results are provided.

The lower half of the screen is dedicated to the display of results. The vertical strip of tabs on the left side of the display allow you to choose between Calculations, Sketch, and Diagram as explained below:

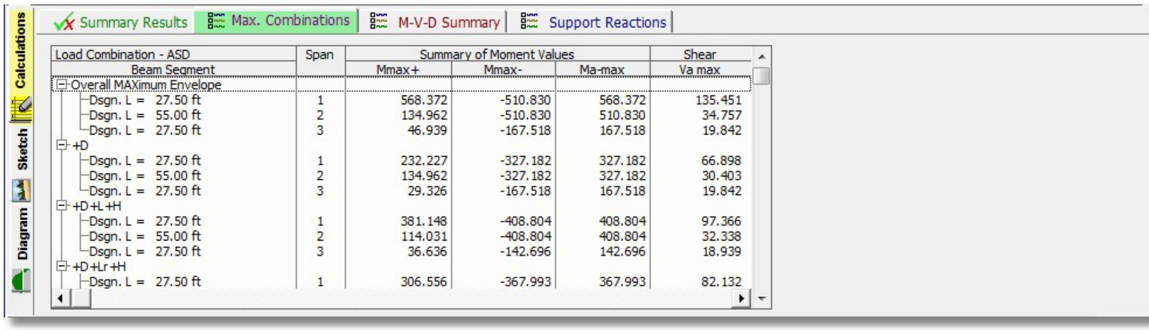
Calculations:

The Calculations tab offers four sub-tabs:

Summary Results: Displays extreme moments, maximum shear, extreme deflections and extreme reactions.

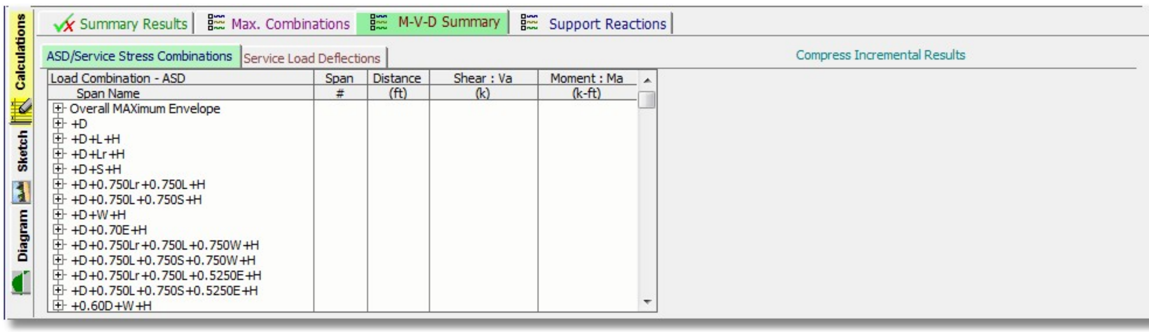


Maximum Combinations: Displays extreme moments and shears, on a span-by-span basis, for all load combinations.

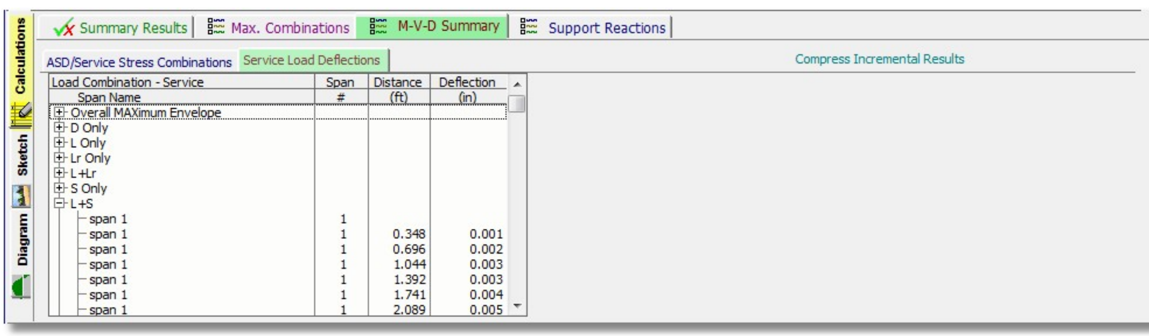


Load Combination - ASD Beam Segment	Span	Summary of Moment Values Mmax+	Mmax-	Ma-max	Shear Va max
Overall MAXimum Envelope					
Dsgn. L = 27.50 ft	1	568.372	-510.830	568.372	135.451
Dsgn. L = 55.00 ft	2	134.962	-510.830	510.830	34.757
Dsgn. L = 27.50 ft	3	46.939	-167.518	167.518	19.842
+D					
Dsgn. L = 27.50 ft	1	232.227	-327.182	327.182	66.898
Dsgn. L = 55.00 ft	2	134.962	-327.182	327.182	30.403
Dsgn. L = 27.50 ft	3	29.326	-167.518	167.518	19.842
+D+L+H					
Dsgn. L = 27.50 ft	1	381.148	-408.804	408.804	97.366
Dsgn. L = 55.00 ft	2	114.031	-408.804	408.804	32.338
Dsgn. L = 27.50 ft	3	36.636	-142.696	142.696	18.939
+D+Lr+H					
Dsgn. L = 27.50 ft	1	306.556	-367.993	367.993	82.132

M-V-D: Summary: Displays moment, shear, and deflections at small increments along all spans. Moment and shear are displayed for all load combinations. Deflection is displayed for service load combinations only.

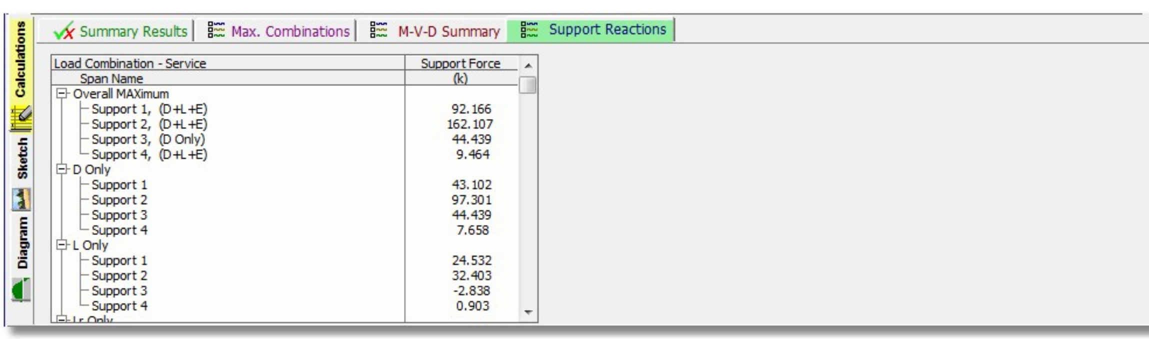


Load Combination - ASD Span Name	Span #	Distance (ft)	Shear : Va (k)	Moment : Ma (k-ft)
Overall MAXimum Envelope				
+D				
+D+L+H				
+D+Lr+H				
+D+S+H				
+D+0.750Lr+0.750L+H				
+D+0.750L+0.750S+H				
+D+W+H				
+D+0.70E+H				
+D+0.750Lr+0.750L+0.750W+H				
+D+0.750L+0.750S+0.750W+H				
+D+0.750Lr+0.750L+0.5250E+H				
+D+0.750L+0.750S+0.5250E+H				
+0.60D+W+H				



Load Combination - Service Span Name	Span #	Distance (ft)	Deflection (in)
Overall MAXimum Envelope			
D Only			
L Only			
Lr Only			
L+Lr			
S Only			
L+S			
span 1	1		
span 1	1	0.348	0.001
span 1	1	0.696	0.002
span 1	1	1.044	0.003
span 1	1	1.392	0.003
span 1	1	1.741	0.004
span 1	1	2.089	0.005

Support Reactions: Displays support reactions for all supports, for all load combinations.



Load Combination - Service Span Name	Support Force (k)
Overall MAXimum	
Support 1, (D+L+E)	92.166
Support 2, (D+L+E)	162.107
Support 3, (D Only)	44.439
Support 4, (D+L+E)	9.464
D Only	
Support 1	43.102
Support 2	97.301
Support 3	44.439
Support 4	7.658
L Only	
Support 1	24.532
Support 2	32.403
Support 3	-2.838
Support 4	0.903
Lr Only	

Sketch:

Displays a sketch of the beam, indicating span lengths, support conditions, and applied loads.

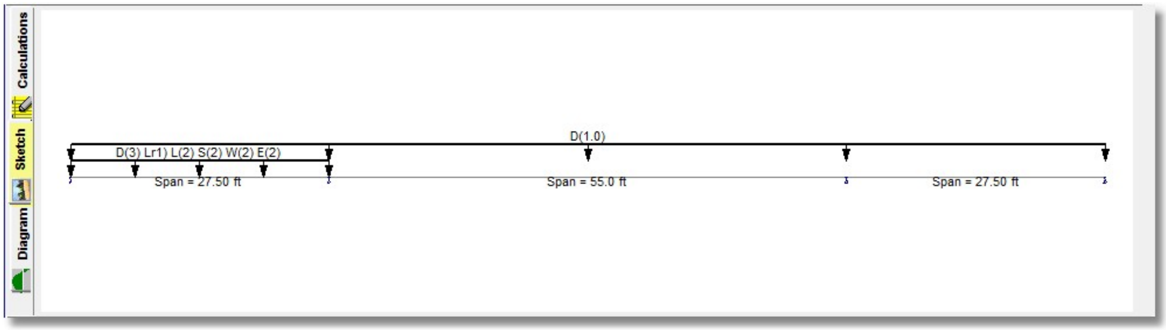
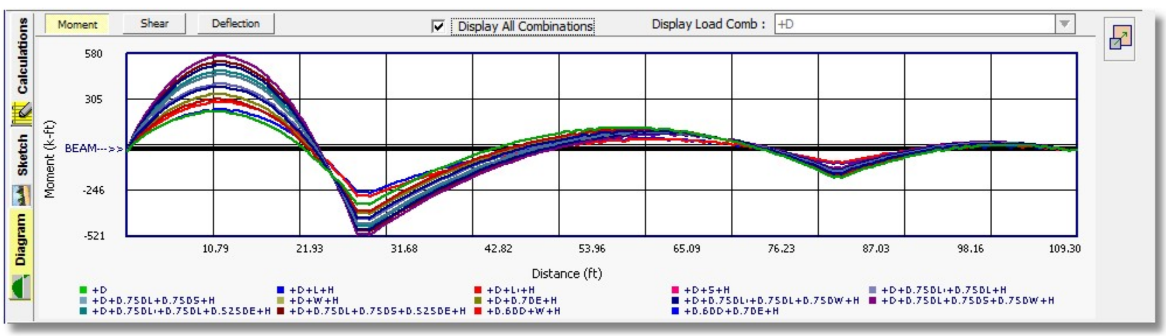


Diagram:

Displays a graphic depiction of the beam with superimposed graphs of Moment, Shear, or Deflection for a selected load combination, or for an envelope of all load combinations.



10.2.3 Wood Beam

Need more? Ask Us a Question

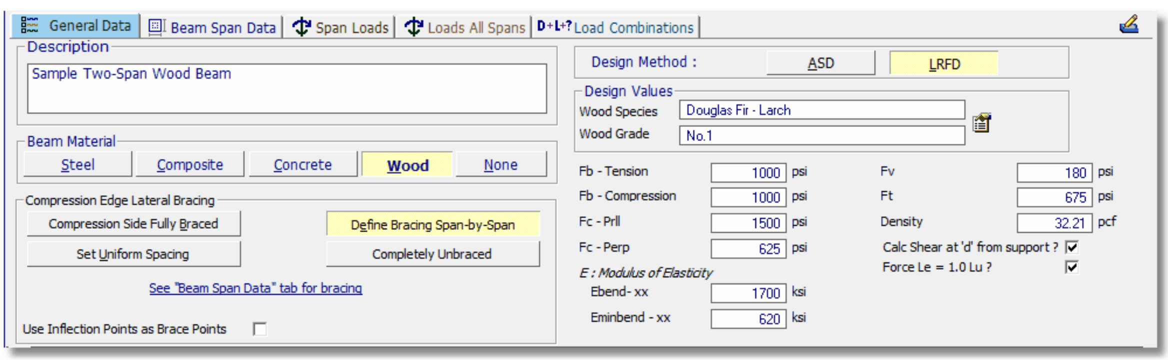
In this section, for each input tab we will review only the items that are unique to the WOOD material type. Click here for a video: Wood Beam

For general information on the typical data input for all beams see the Beams 245 topic.

This module offers complete design of single and multi-span wood members. Among its capabilities are:

- Single or multi-span beams.
- End fixity can be pinned, fixed, free or a combination thereof.
- Analysis is according to 2005 NDS.
- ASD or LRFD design methods can be selected. Values of K_F and phi are automatically determined and applied for the LRFD method.
- A complete wood section database is provided. This includes sawn, glued-laminated and selected manufactured sections.
- A complete wood species database is provided. All values are per 2005 NDS.
- Unbraced compression edge lengths can be specified in a variety of ways.
- Automatic member selection is provided.
- You can specify values for C_M, C_t and C_r. C_F or C_V is automatically provided. In the case of C_F, the value is also based on species stress grade.

General Data



Beam Material

Clicking one of these buttons changes the material type used for the beam.

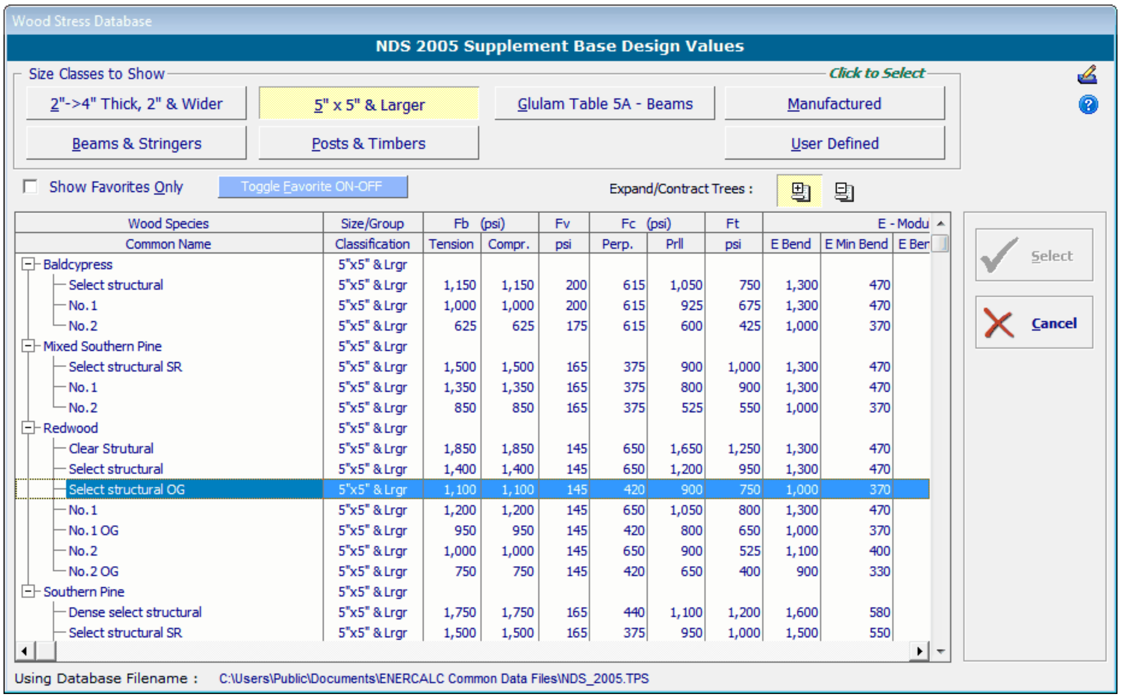
Design Method

For wood & steel you can select ASD or LRFD design methods. Concrete design always uses ultimate strength design (LRFD).

Design Values

This section is used to specify the type of wood that will be used. Use the  button to access the standard wood reference design values database and select a material.

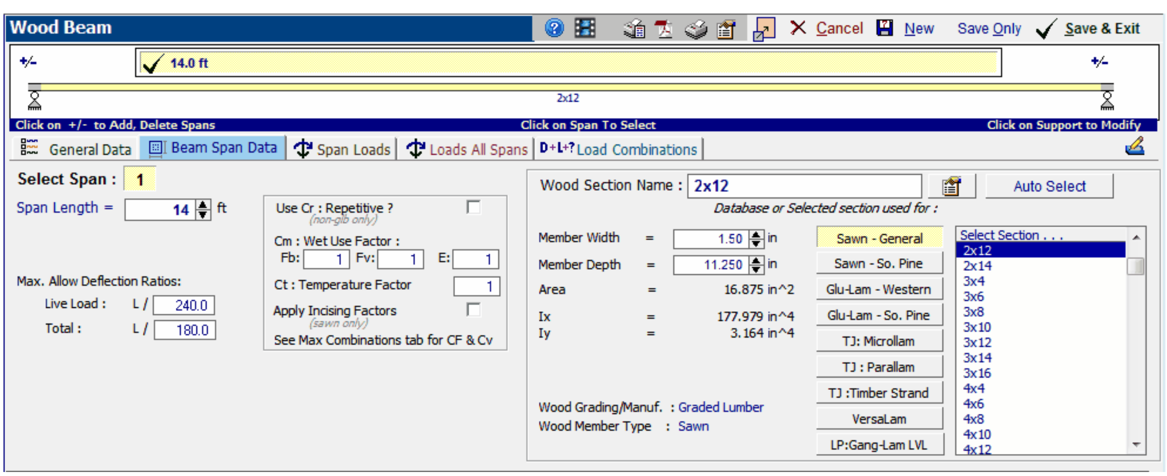
These values can be edited right on the screen. HOWEVER there are other pieces of information, such as size factors for certain sizes of members, that are stored separately.



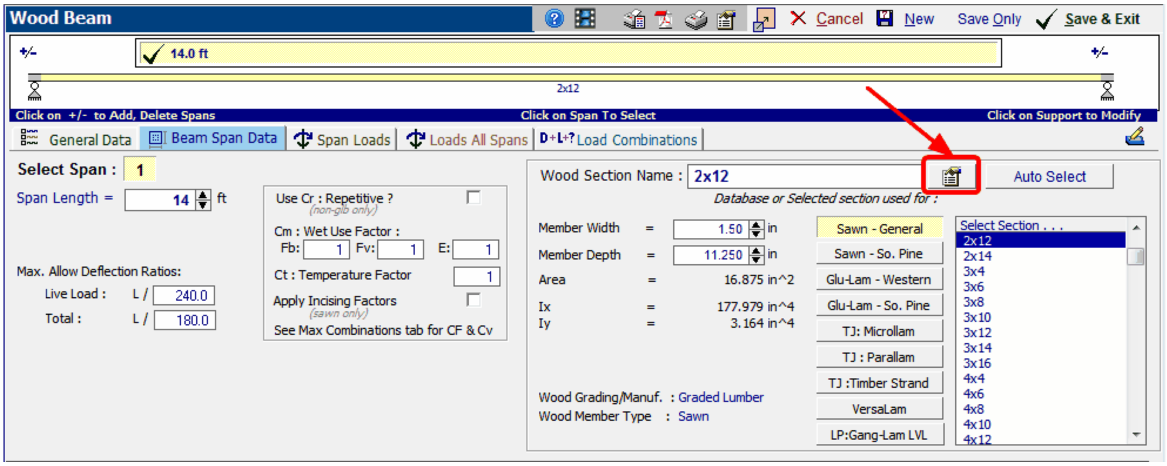
Wood Species Common Name	Size/Group Classification	Fb (psi) Tension	Compr.	Fv psi	Fc (psi) Perp.	Prll	Ft psi	E - Modu E Bend	E Min Bend	E Ber
Baldcypress	5"x5" & Lrgr									
Select structural	5"x5" & Lrgr	1,150	1,150	200	615	1,050	750	1,300	470	
No.1	5"x5" & Lrgr	1,000	1,000	200	615	925	675	1,300	470	
No.2	5"x5" & Lrgr	625	625	175	615	600	425	1,000	370	
Mixed Southern Pine	5"x5" & Lrgr									
Select structural SR	5"x5" & Lrgr	1,500	1,500	165	375	900	1,000	1,300	470	
No.1	5"x5" & Lrgr	1,350	1,350	165	375	800	900	1,300	470	
No.2	5"x5" & Lrgr	850	850	165	375	525	550	1,000	370	
Redwood	5"x5" & Lrgr									
Clear Strutural	5"x5" & Lrgr	1,850	1,850	145	650	1,650	1,250	1,300	470	
Select structural	5"x5" & Lrgr	1,400	1,400	145	650	1,200	950	1,300	470	
Select structural OG	5"x5" & Lrgr	1,100	1,100	145	420	900	750	1,000	370	
No.1	5"x5" & Lrgr	1,200	1,200	145	650	1,050	800	1,300	470	
No.1 OG	5"x5" & Lrgr	950	950	145	420	800	650	1,000	370	
No.2	5"x5" & Lrgr	1,000	1,000	145	650	900	525	1,100	400	
No.2 OG	5"x5" & Lrgr	750	750	145	420	650	400	900	330	
Southern Pine	5"x5" & Lrgr									
Dense select structural	5"x5" & Lrgr	1,750	1,750	165	440	1,100	1,200	1,600	580	
Select structural SR	5"x5" & Lrgr	1,500	1,500	165	375	950	1,000	1,500	550	

Beam Span Data

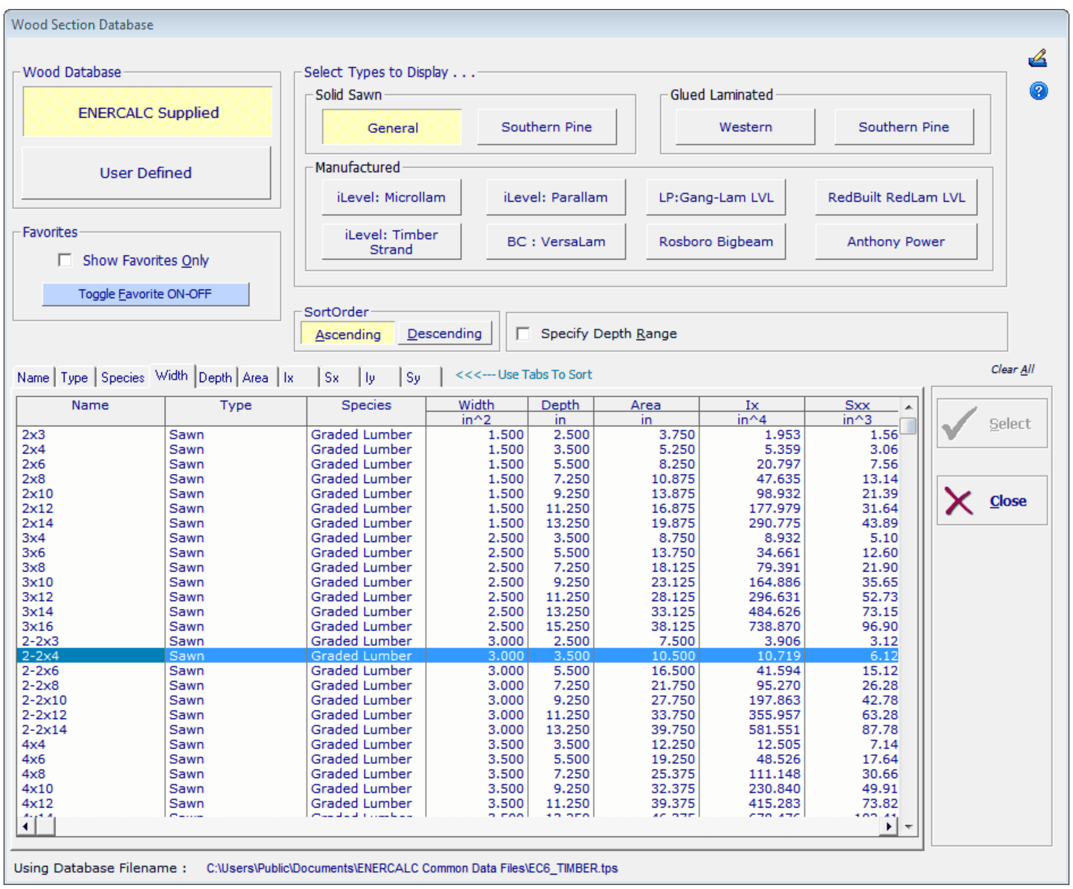
This tab is used to define span length and section information for the beam:



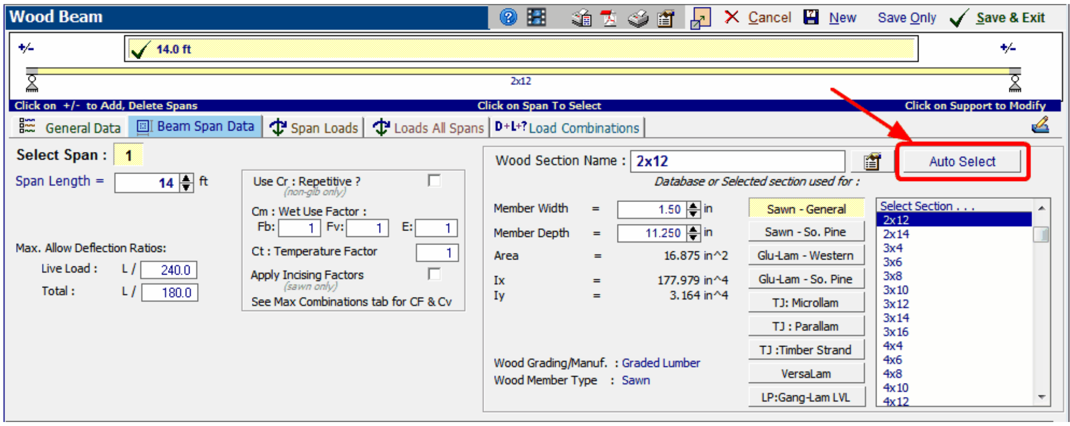
The button shown bubbled in the screen capture below is used to display the Wood Section Database.



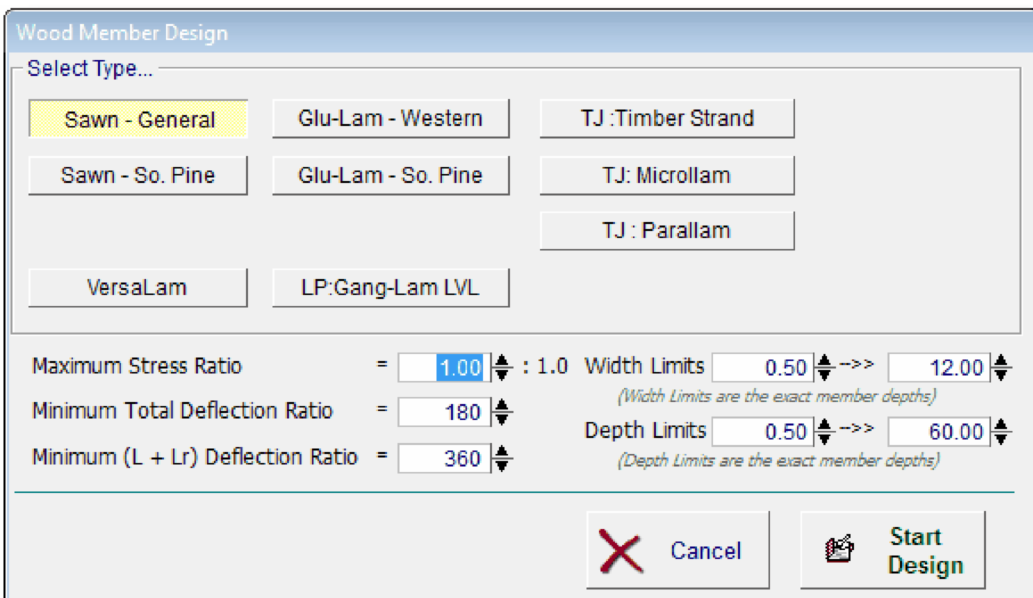
The Wood Section Database contains an extensive number of solid-sawn members, glulam members, and manufactured members commonly used in the USA.



The button shown bubbled in the screen capture below is used to display the Wood Member Design dialog.



The Wood Member Design dialog allows you to choose the type of member to be selected and to specify limits on the permissible stress ratio, deflection ratio and selected member size.



Note: The factor named Maximum Stress Ratio does not act as a multiplier on the specified deflection ratios.

Span Loads

No differences from other materials.

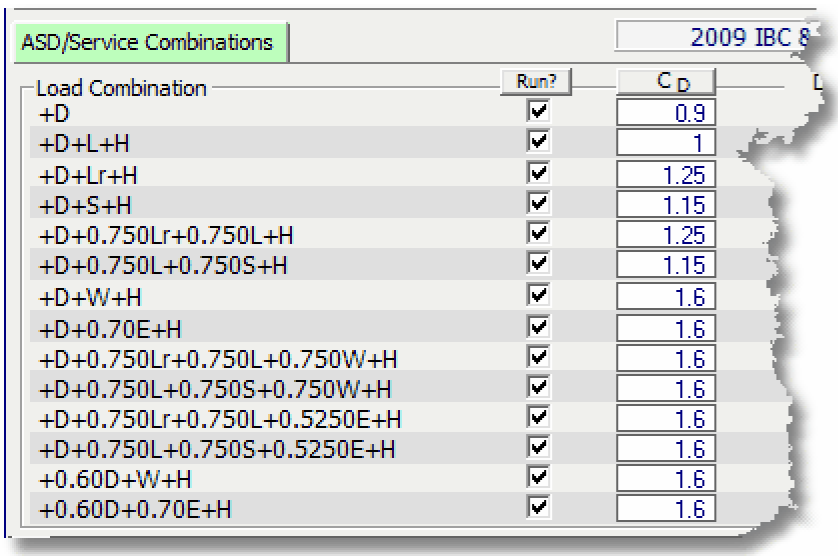
Loads All Spans

No differences from other materials.

Load Combinations

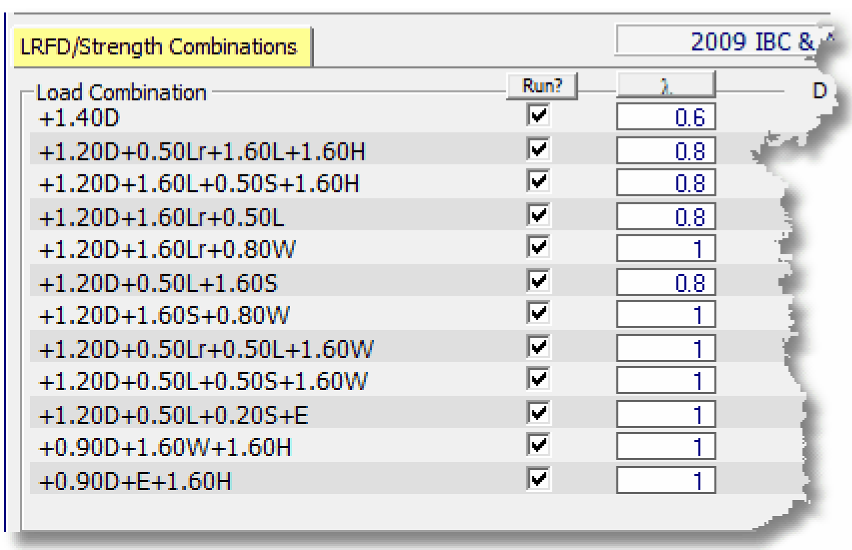
For wood members you will see entries for load duration factors. When ASD is used, the Load Duration Factor is referred to as C_D. When LRFD is used, the Load Duration Factor is referred to as λ.

Note that C_D and λ actually appear on the button at the top of the column of values. When that button is clicked, the program will automatically determine the proper value for C_D or λ according to the NDS based on the shortest duration load type included in each of the load combinations.



ASD/Service Combinations — 2009 IBC &

Load Combination	Run?	C_D
+D	✓	0.9
+D+L+H	✓	1
+D+Lr+H	✓	1.25
+D+S+H	✓	1.15
+D+0.750Lr+0.750L+H	✓	1.25
+D+0.750L+0.750S+H	✓	1.15
+D+W+H	✓	1.6
+D+0.70E+H	✓	1.6
+D+0.750Lr+0.750L+0.750W+H	✓	1.6
+D+0.750L+0.750S+0.750W+H	✓	1.6
+D+0.750Lr+0.750L+0.5250E+H	✓	1.6
+D+0.750L+0.750S+0.5250E+H	✓	1.6
+0.60D+W+H	✓	1.6
+0.60D+0.70E+H	✓	1.6



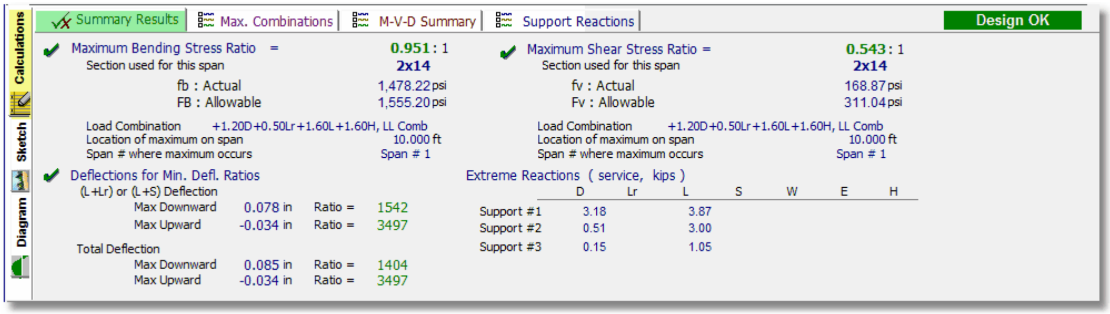
LRFD/Strength Combinations — 2009 IBC &

Load Combination	Run?	λ
+1.40D	✓	0.6
+1.20D+0.50Lr+1.60L+1.60H	✓	0.8
+1.20D+1.60L+0.50S+1.60H	✓	0.8
+1.20D+1.60Lr+0.50L	✓	0.8
+1.20D+1.60Lr+0.80W	✓	1
+1.20D+0.50L+1.60S	✓	0.8
+1.20D+1.60S+0.80W	✓	1
+1.20D+0.50Lr+0.50L+1.60W	✓	1
+1.20D+0.50L+0.50S+1.60W	✓	1
+1.20D+0.50L+0.20S+E	✓	1
+0.90D+1.60W+1.60H	✓	1
+0.90D+E+1.60H	✓	1

Results Tabs: This set of tabs provides detailed results for the current calculation. The vertical tabs on the left edge of the screen allow you to select the three major areas available for review: Calculations, Sketch, and Diagram.

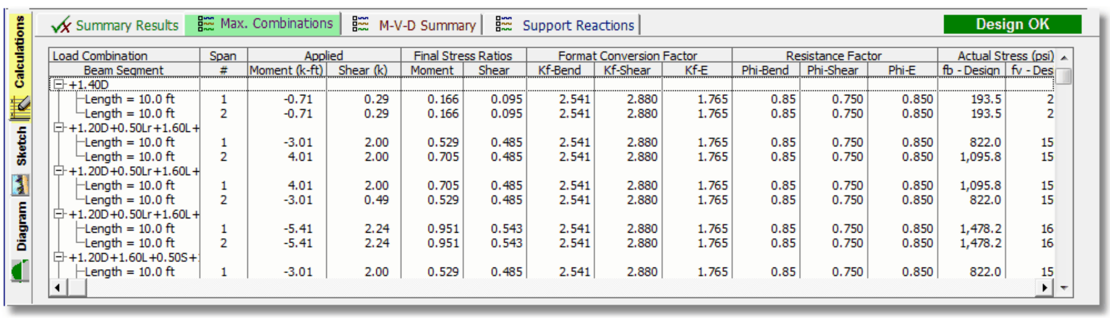
The Calculations tab offers the following results options:

Summary Results provides details for shear, moment and deflection for the governing load combinations.

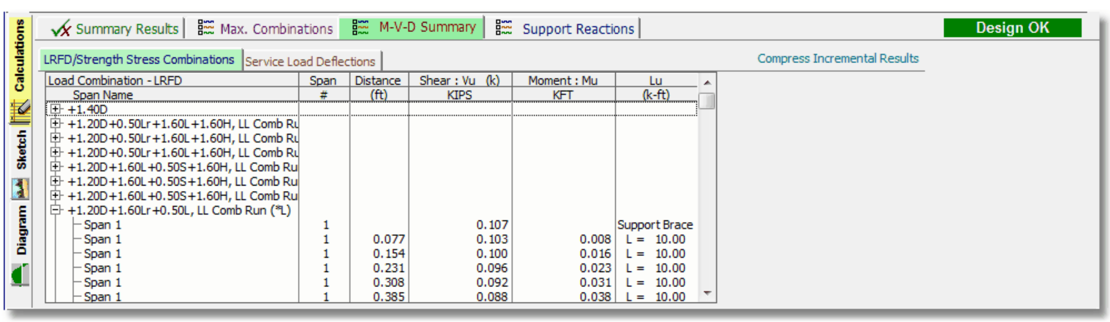


Max. Combinations provides detailed results for each beam segment for each load combination. The leftmost column lists the load combinations and the unbraced length being considered.

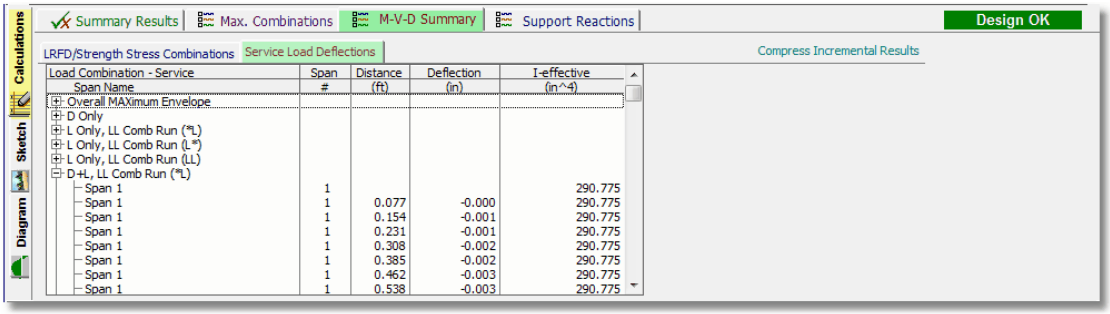
These results are a consolidation of the highly detailed incremental results presented on the M-V-D Summary tab.



M-V-D Summary- Stresses shows highly detailed moment and shear information for each beam and for each load combination. For multi-span beams using Automatic Unbalanced Live Load Placement there may be thousands of lines of results.

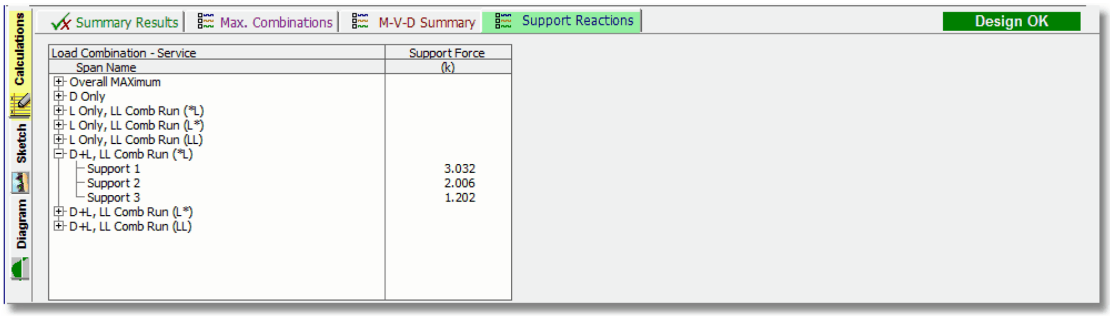


M-V-D Summary - Deflections shows highly detailed deflection results for all load combinations.



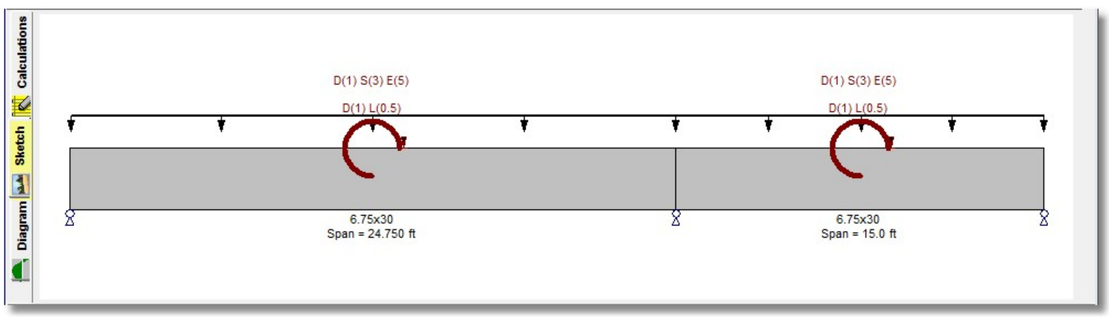
Load Combination - Service Span Name	Span #	Distance (ft)	Deflection (in)	I-effective (in^4)
Overall MAXimum Envelope				
D Only				
L Only, LL Comb Run (*L)				
L Only, LL Comb Run (L*)				
L Only, LL Comb Run (LL)				
D+L, LL Comb Run (*L)				
Span 1	1			290.775
Span 1	1	0.077	-0.000	290.775
Span 1	1	0.154	-0.001	290.775
Span 1	1	0.231	-0.001	290.775
Span 1	1	0.308	-0.002	290.775
Span 1	1	0.385	-0.002	290.775
Span 1	1	0.462	-0.003	290.775
Span 1	1	0.538	-0.003	290.775

Support Reactions shows reactions for each support for each load condition.

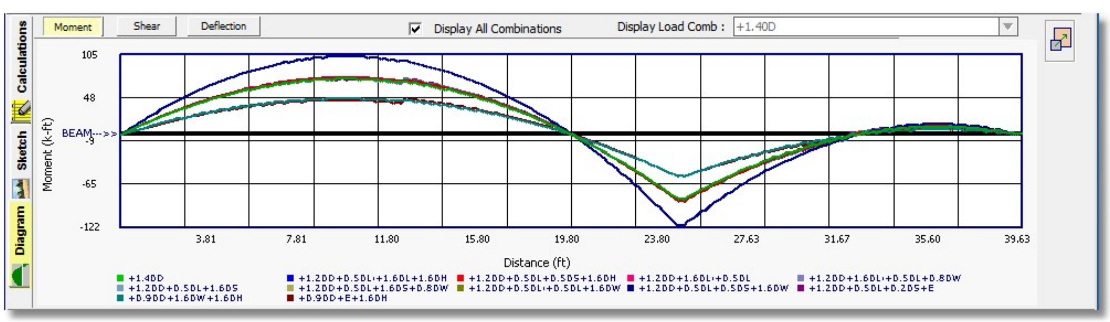


Load Combination - Service Span Name	Support Force (k)
Overall MAXimum	
D Only	
L Only, LL Comb Run (*L)	
L Only, LL Comb Run (L*)	
L Only, LL Comb Run (LL)	
D+L, LL Comb Run (*L)	
Support 1	3.032
Support 2	2.006
Support 3	1.202
D+L, LL Comb Run (L*)	
D+L, LL Comb Run (LL)	

The Sketch tab provides a graphic representation of the beam currently being designed:



The Diagram tab offers the ability to view shear, moment, and deflection diagrams for selected load combinations:



REPORTS

Below is a typical Wood Beam printed report:

Joseph P. Engineer
GREAT ENGINEERING UNLIMITED
1234 Main Street, 1st Floor
Anytown, USA 99221
jpe@GreatEngineering.com
Phone : (800) 111-2222

Title :
Dsgnr:
Project Desc.:

Project Notes :

Job #

Printed: 11 OCT 2009, 5:55PM

Wood Beam Design

File: c:\EC6\AA_TECH_EC6_FILES\examples.ec6
ENERCALC, INC. 1983-2009, Ver: 6.0.225, N:00661

Lic. # : KW-06000215 **License Owner : Licensed ENERCALC User**

Description : Sample Two-Span Wood Beam

Material Properties

Calculations per IBC 2006, CBC 2007, 2005 NDS

Analysis Method : Load Resistance Factor D
Load Combination 2006 IBC & ASCE 7-05

Wood Species : SP/SP
Wood Grade : 26F-V4

Beam Bracing : Beam bracing is defined Beam-by-Beam

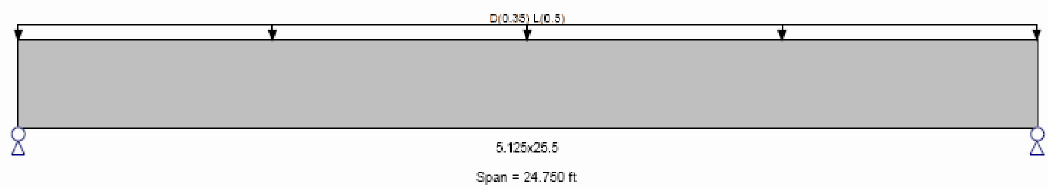
Fb - Tension	2,600.0	psi
Fb - Compr	2,600.0	psi
Fc - Prll	1,600.0	psi
Fc - Perp	740.0	psi
Fv	300.0	psi
Ft	1,200.0	psi

E : Modulus of Elasticity

Ebend- xx	1,900.0	ksi
Eminbend - xx	980.0	ksi
Ebend- yy	1,800.0	ksi
Eminbend - yy	930.0	ksi
Density	35.440	pcf

Unbraced Lengths

Span # 1, Braced @ Mid Span



Applied Loads

Service loads entered. Load Factors will be applied for calculations.

Beam self weight calculated and added to loads
Load for Span Number 1
Uniform Load : D = 0.350, L = 0.50 k/ft, Tributary Width = 1.0 ft

DESIGN SUMMARY

Design OK

Maximum Bending Stress Ratio	=	**0.470**: 1	Maximum Shear Stress Ratio	=	**0.345** : 1
Section used for this span		**5.125x25.5**	Section used for this span		**5.125x25.5**
fb : Actual	=	2,082.12psi	fv : Actual	=	178.77 psi
FB : Allowable	=	4,434.33psi	Fv : Allowable	=	518.40 psi
Load Combination		+1.20D+0.50Lr+1.60L+1.60H	Load Combination		+1.20D+0.50Lr+1.60L+1.60H
Location of maximum on span	=	12.375ft	Location of maximum on span	=	0.000 ft
Span # where maximum occurs	=	Span # 1	Span # where maximum occurs	=	Span # 1

Maximum Deflection

Max Downward L+Lr+S Deflection	0.316 in	Ratio =	939
Max Upward L+Lr+S Deflection	0.000 in	Ratio =	0 <180
Max Downward Total Deflection	0.558 in	Ratio =	532
Max Upward Total Deflection	0.000 in	Ratio =	0 <360

Maximum Forces & Stresses for Load Combinations

Load Combination		Max Stress Ratios			Summary of Moment Values			Summary of Shear Values		
Segment Length	Span #	M	V	λ	Mu	fb-design	Fb-allow	Vu	fv-design	Fv-allow
+1.40D										
Length = 12.375 ft	1	0.266	0.195	0.600	40.97	885.11	3,325.75	6.62	75.99	388.80
Length = 12.375 ft	1	0.266	0.195	0.600	40.97	885.11	3,325.75	6.62	75.99	388.80
+1.20D+0.50Lr+1.60L+1.60H										
Length = 12.375 ft	1	0.470	0.345	0.800	96.37	2,082.12	4,434.33	15.58	178.77	518.40
Length = 12.375 ft	1	0.470	0.345	0.800	96.37	2,082.12	4,434.33	15.58	178.77	518.40
+1.20D+0.50L+0.50S+1.60H										
Length = 12.375 ft	1	0.264	0.194	0.800	54.26	1,172.24	4,434.33	8.77	100.65	518.40
Length = 12.375 ft	1	0.264	0.194	0.800	54.26	1,172.24	4,434.33	8.77	100.65	518.40
+1.20D+1.60Lr+0.50L										
Length = 12.375 ft	1	0.264	0.194	0.800	54.26	1,172.24	4,434.33	8.77	100.65	518.40
Length = 12.375 ft	1	0.264	0.194	0.800	54.26	1,172.24	4,434.33	8.77	100.65	518.40

Joseph P. Engineer
GREAT ENGINEERING UNLIMITED
1234 Main Street, 1st Floor
Anytown, USA 99221
jpe@GreatEngineering.com
Phone : (800) 111-2222

Title :
Dsgnr:
Project Desc.:

Project Notes :

Job #

Printed: 11 OCT 2009, 5:55PM

Wood Beam Design

File: c:\EC6\AA_TECH_EC6_FILES\examples.ec6
ENERCALC, INC. 1983-2009, Ver: 6.0.225, N:00661

Lic. # : KW-06000215 **License Owner : Licensed ENERCALC User**

Description : Sample Two-Span Wood Beam

Load Combination / Segment Length	Span #	Max Stress Ratios M	Max Stress Ratios V	λ	Mu	fb-design	Fb-allow	Vu	fv-design	Fv-allow
+1.20D+1.60Lr+0.50L+0.80W										
Length = 12.375 ft	1	0.211	0.155	1.000	54.26	1,172.24	5,542.92	8.77	100.65	648.00
Length = 12.375 ft	1	0.211	0.155	1.000	54.26	1,172.24	5,542.92	8.77	100.65	648.00
+1.20D+0.50L+1.60S										
Length = 12.375 ft	1	0.264	0.194	0.800	54.26	1,172.24	4,434.33	8.77	100.65	518.40
Length = 12.375 ft	1	0.264	0.194	0.800	54.26	1,172.24	4,434.33	8.77	100.65	518.40
+1.20D+0.50L+1.60S+0.80W										
Length = 12.375 ft	1	0.211	0.155	1.000	54.26	1,172.24	5,542.92	8.77	100.65	648.00
Length = 12.375 ft	1	0.211	0.155	1.000	54.26	1,172.24	5,542.92	8.77	100.65	648.00
+1.20D+0.50Lr+0.50L+1.60W										
Length = 12.375 ft	1	0.211	0.155	1.000	54.26	1,172.24	5,542.92	8.77	100.65	648.00
Length = 12.375 ft	1	0.211	0.155	1.000	54.26	1,172.24	5,542.92	8.77	100.65	648.00
+1.20D+0.50L+0.50S+1.60W										
Length = 12.375 ft	1	0.211	0.155	1.000	54.26	1,172.24	5,542.92	8.77	100.65	648.00
Length = 12.375 ft	1	0.211	0.155	1.000	54.26	1,172.24	5,542.92	8.77	100.65	648.00
+1.20D+0.50L+0.20S+E										
Length = 12.375 ft	1	0.211	0.155	1.000	54.26	1,172.24	5,542.92	8.77	100.65	648.00
Length = 12.375 ft	1	0.211	0.155	1.000	54.26	1,172.24	5,542.92	8.77	100.65	648.00
+0.90D+1.60W+1.60H										
Length = 12.375 ft	1	0.103	0.075	1.000	26.34	569.00	5,542.92	4.26	48.85	648.00
Length = 12.375 ft	1	0.103	0.075	1.000	26.34	569.00	5,542.92	4.26	48.85	648.00
+0.90D+E+1.60H										
Length = 12.375 ft	1	0.103	0.075	1.000	26.34	569.00	5,542.92	4.26	48.85	648.00
Length = 12.375 ft	1	0.103	0.075	1.000	26.34	569.00	5,542.92	4.26	48.85	648.00

Summary of Moment Values: Mu, fb-design, Fb-allow. Summary of Shear Values: Vu, fv-design, Fv-allow.

Overall Maximum Deflections - Unfactored Loads

Load Combination	Span	Max. "-" Defl	Location in Span	Load Combination	Max. "+" Defl	Location in Span
D+L+Lr	1	0.5580	12.499		0.0000	0.000

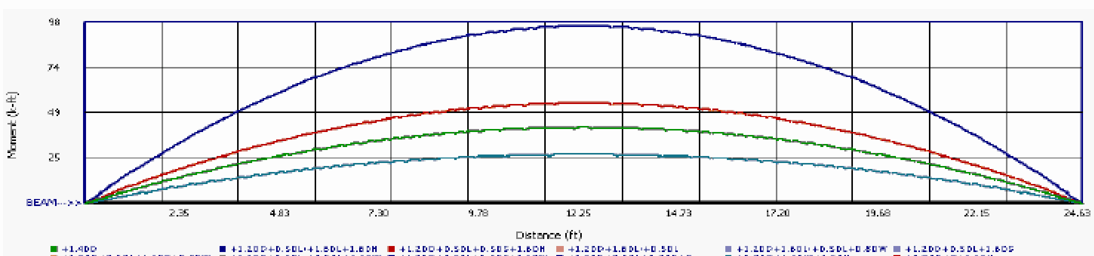
Maximum Deflections for Load Combinations - Unfactored Loads

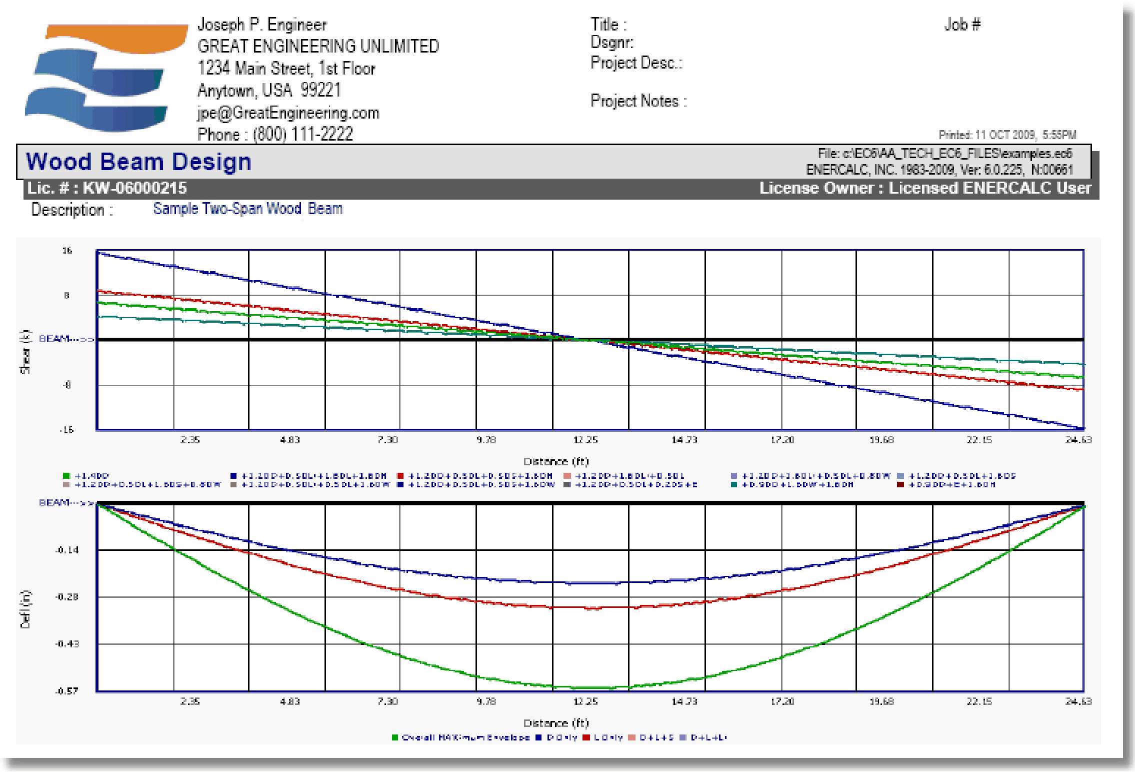
Load Combination	Span	Max. Downward Defl	Location in Span	Max. Upward Defl	Location in Span
D Only	1	0.2417	12.499	0.0000	0.000
L Only	1	0.3162	12.499	0.0000	0.000
D+L+S	1	0.5580	12.499	0.0000	0.000
D+L+Lr	1	0.5580	12.499	0.0000	0.000

Vertical Reactions - Unfactored

Support notation : Far left is #1 Values in KIPS

Load Combination	Support 1	Support 2
Overall MAXimum	10.917	10.917
D Only	4.729	4.729
L Only	6.188	6.188
D+L+S	10.917	10.917
D+L+Lr	10.917	10.917





Joseph P. Engineer
GREAT ENGINEERING UNLIMITED
1234 Main Street, 1st Floor
Anytown, USA 99221
jpe@GreatEngineering.com
Phone : (800) 111-2222
Title :
Dsgnr:
Project Desc.:
Project Notes :
Job #
Printed: 11 OCT 2009, 5:55PM
Wood Beam Design
File: c:\EC6\AA_TECH_EC6_FILES\examples.ec6
ENERCALC, INC. 1983-2009, Ver: 6.0.225, N:00661
Lic. # : KW-06000215
License Owner : Licensed ENERCALC User
Description :
Sample Two-Span Wood Beam
Shear (k)
BEAM--->>
Distance (ft)
Defl (in)
Distance (ft)

10.2.4 Steel Beam

[Need more? Ask Us a Question]

In this section, for each input tab we will review only the items that are unique to the STEEL material type. Click here for a video: Steel Beam

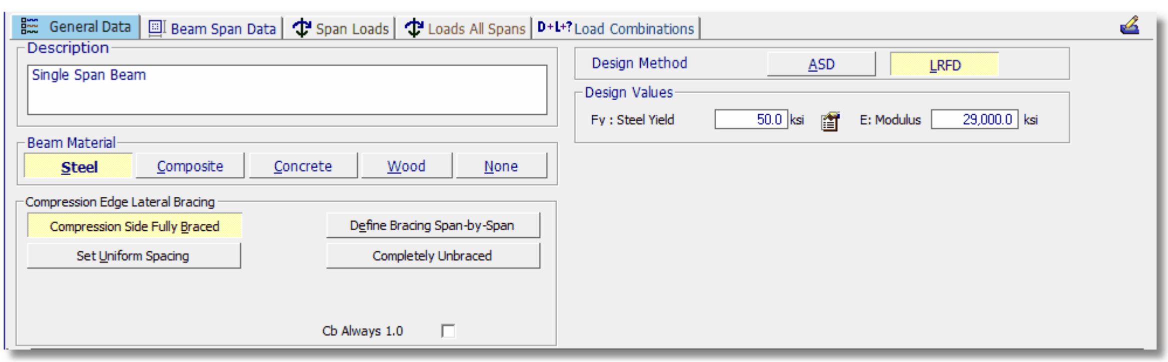
For general information on the typical data input for all beams see the Beams 245 topic.

This module offers complete design of single and multi-span steel members. Among its capabilities are:

- Single or multi-span beams.
- End fixity can be pinned, fixed, free or a combination.
- Steel member analysis and design are according to AISC 360-05.
- ASD or LRFD design methods can be selected.
- A complete steel section database is provided.
- Unbraced compression edge lengths can be specified in a variety of ways.
- Automatic member selection is provided.

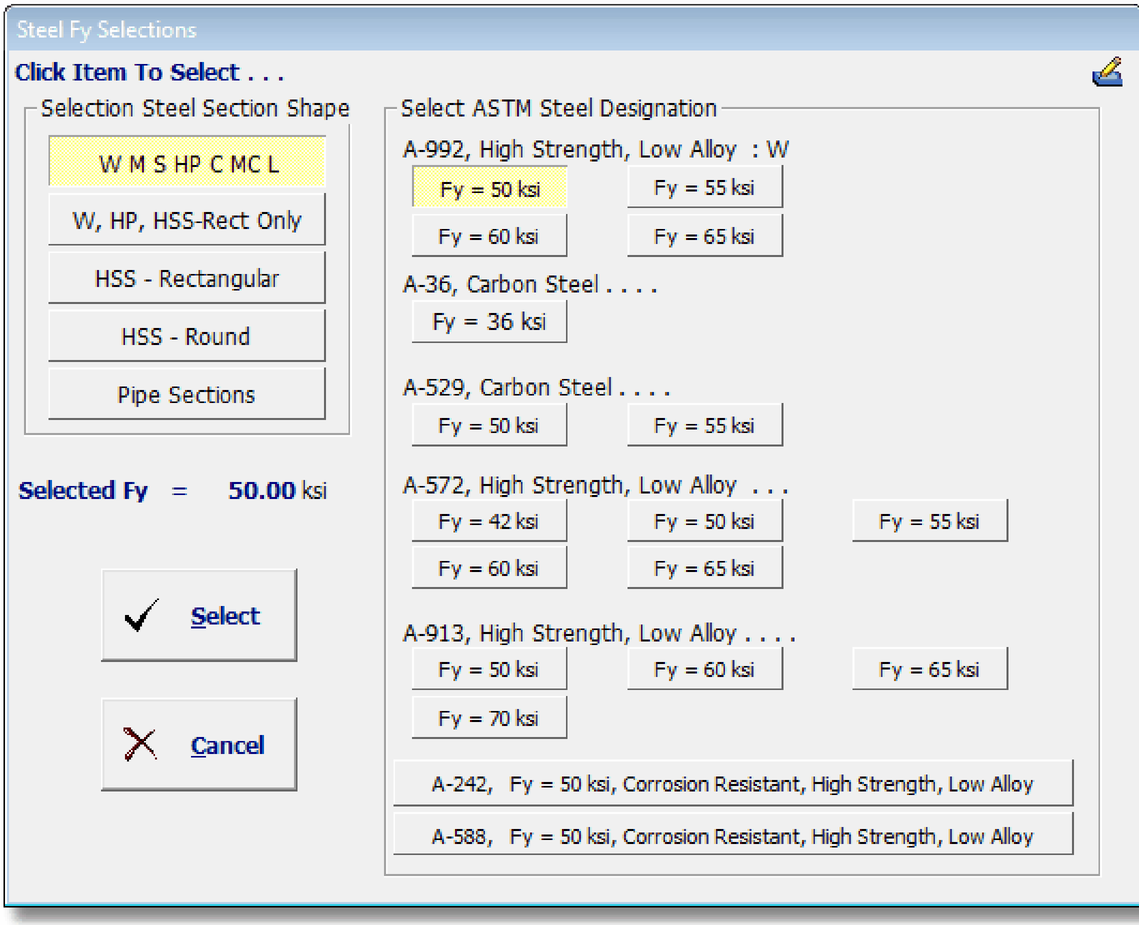
General Data

When steel is used, this tab includes input fields to set the values for yield stress and elastic modulus, as well as an option to set the Cb factor equal to 1, as shown in the screen capture below:



If Cb is not forced to 1, the program automatically calculates Cb based on the moment and direction of curvature at various locations along the beam.

For convenience the module includes a built-in steel database. Click the  button to the right of the Fy entry and you will see the following table, which offers a number of commonly used steel grades:



Beam Span Data

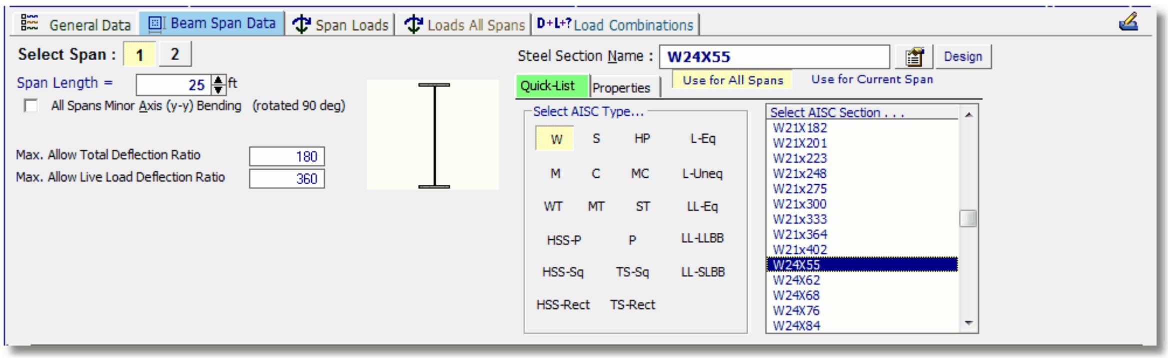
When steel is selected, the beam size selection is specifically for steel. You can select a section in 4 ways:

1 - Simply type the AISC name into the Steel Section Name field and press [**Tab**].

2 - Click the database access button  and select from the built-in AISC section database.

3 - Click the [**Design**] button to have the module evaluate steel sections from the database according to your criteria.

4 - Select a steel section from the Quick-List tab as shown below:



Span Loads

No differences from other materials.

Loads All Spans

No differences from other materials.

Load Combinations

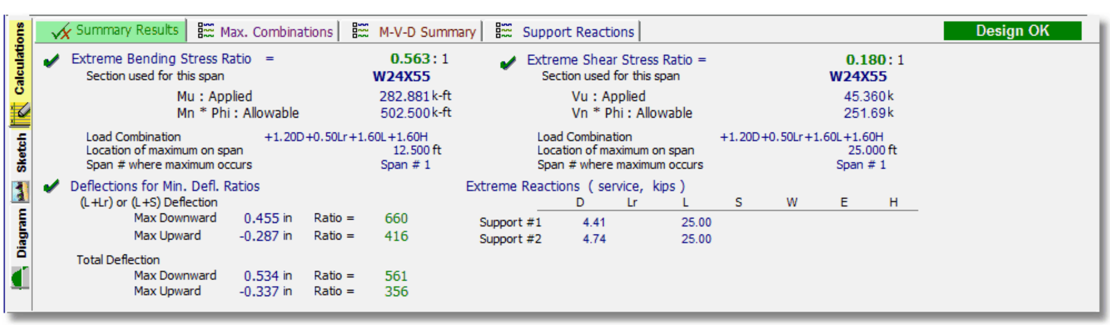
No differences from other materials.

Results Tab

This set of tabs provides detailed results for the current calculation. The vertical tabs on the left edge of the screen allow you to select the three major areas available for review: Calculations, Sketch, and Diagram.

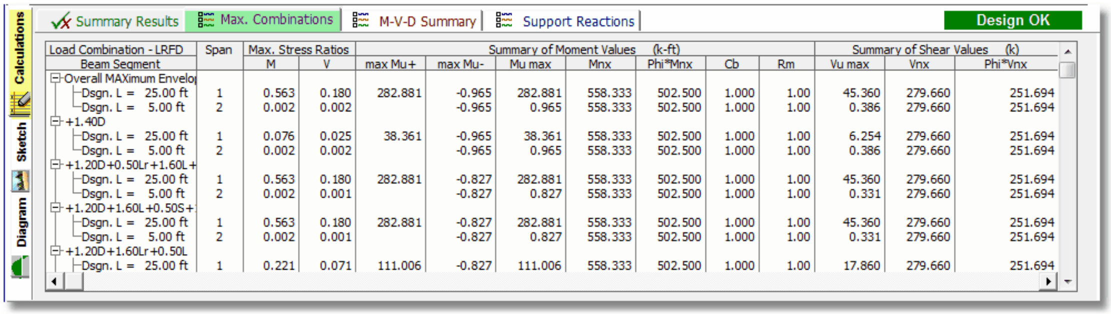
The Calculations tab offers the following results options:

Summary Results provides details for shear, moment and deflection for the governing load combinations.

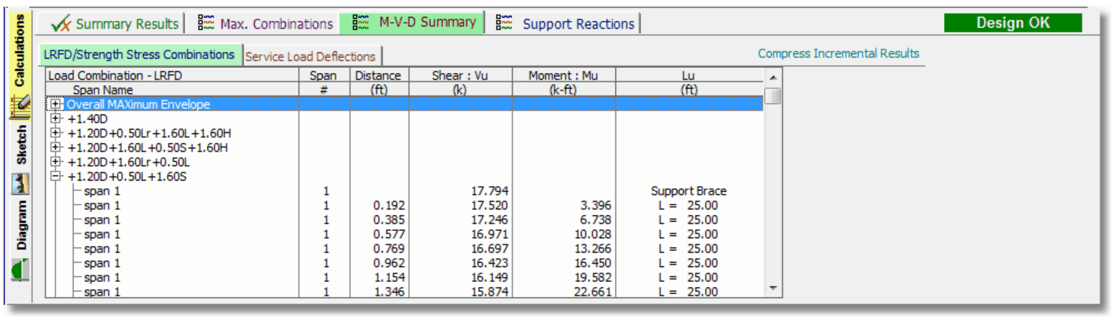


Max. Combinations provides detailed results for each beam segment for each load combination. The leftmost column lists the load combinations and the unbraced length being considered.

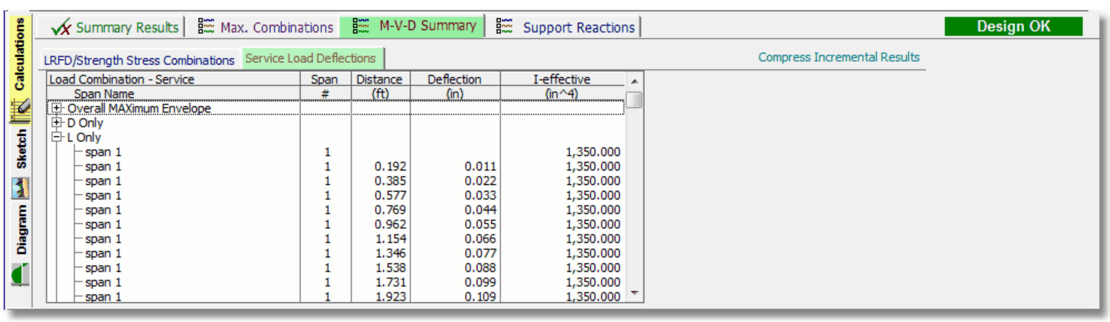
These results are a consolidation of the highly detailed incremental results on the M-V-D Summary tab.



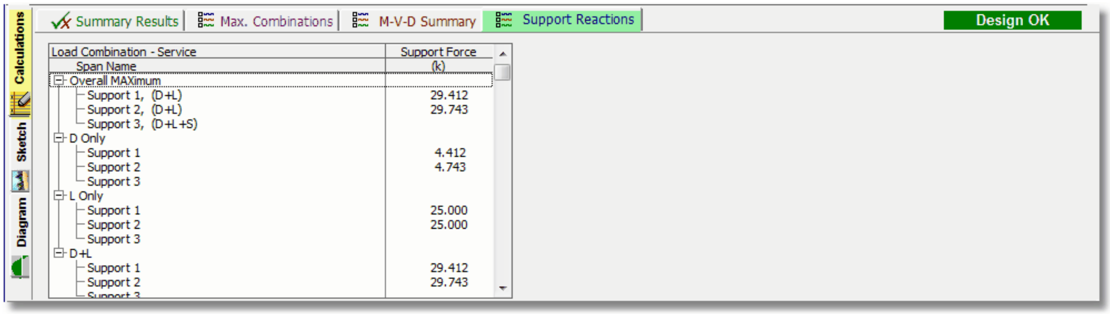
M-V-D Summary- Stresses shows highly detailed moment and shear information for each beam and for each load combination. For multi-span beams using Automatic Unbalanced Live Load Placement there may be thousands of lines of results.



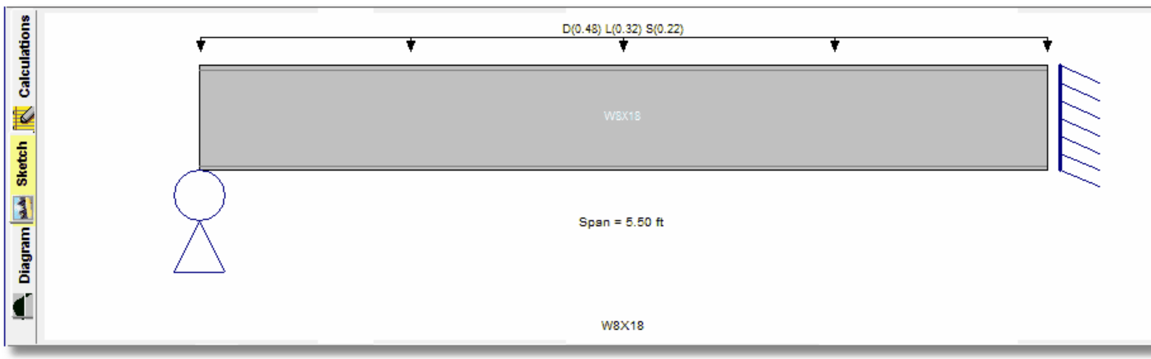
M-V-D Summary - Deflections shows highly detailed deflection results for all load combinations.



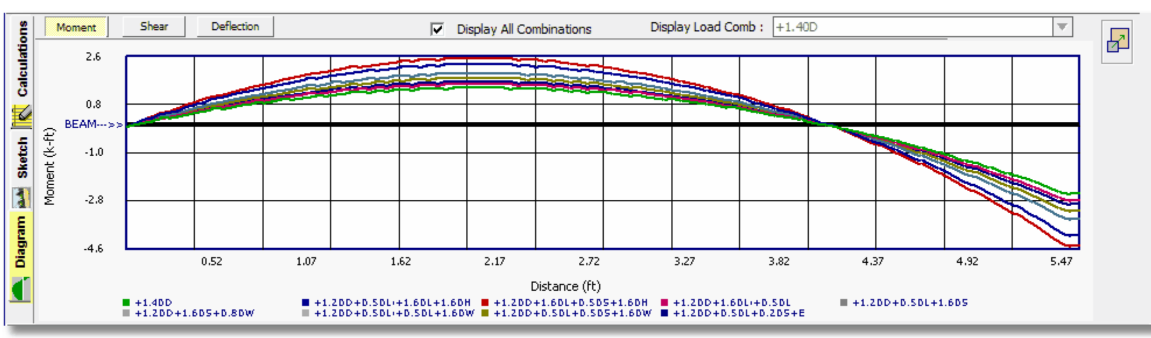
Support Reactions shows reactions for each support for each load condition.



The Sketch tab provides a graphic representation of the beam currently being designed:



The Diagram tab offers the ability to view shear, moment, and deflection diagrams for selected load combinations:



10.2.5 Concrete Beam

[Need more? Ask Us a Question]

In this section, for each input tab we will review only the items that are unique to the CONCRETE material type. Click here for a video: Concrete Beam

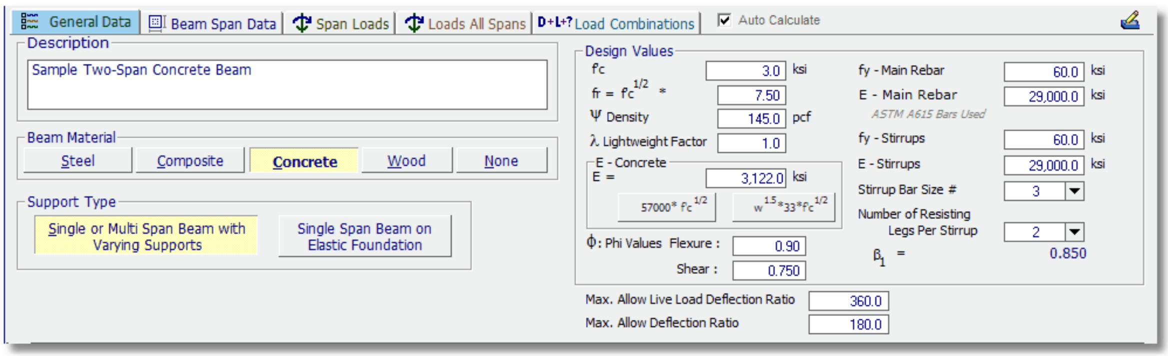
General Data

The concrete beam module handles single- and multiple-span beams using ONE cross section shape. That shape can have up to six groups of reinforcing per span, and the reinforcing can vary on a span-by-span basis.

This module also has a beam on elastic foundation option for single-span beams. In the screen capture below you can see the two large selection boxes. Selecting Single Span Beam on Elastic Foundation will remove the ability to select end support conditions, and it will provide an entry for the Modulus of Subgrade Reaction of the supporting soil.

On the right side of this tab are all the required rebar, concrete strength and elastic modulus entries, shear stirrup data, strength reduction factors and deflection criteria to check.

NOTE: It is important to know how this module operates with regard to beam stiffness along the span, and how this affects multi-span beams. This module divides each span into a series of segments. The effective moment of inertia for each segment (for each load combination) is calculated using the actual unfactored moment on that segment. Thus the module creates a very accurate variable stiffness model of the beam based on actual moments. For multiple-span beams, this will affect the relative stiffness of each beam span. Thus the moment distribution across multiple spans will be properly performed. This will affect factored load moments and shears and service load level deflections and reactions.



Beam Span Data

This tab has some pieces of input that are constant for all spans and some that can vary on a span-by-span basis.

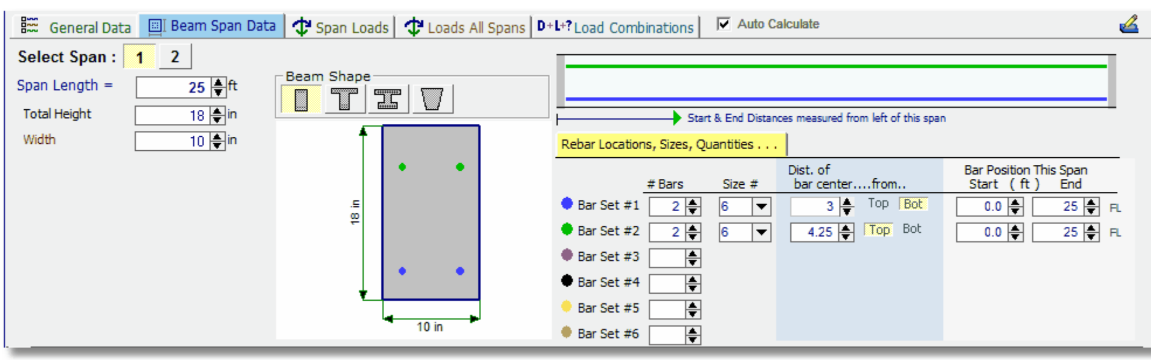
The cross section shape and dimensions are the same for all spans.

When you click on a span (for multi-span beams) in the topmost section of the window, the span length and rebar layout will update to display the arrangement that is specific to that span.

On the right side of this tab you can specify up to 6 bar sets (quantity, size, vertical location and start/stop endpoints). Each bar set is referenced on the sketch with a color shown as a dot to the left of the set description.

The column highlighted in light blue titled "Dist of bar center....from...." is how you set the vertical position of the bar sets in the beam. When you look at the top one, you can read it as "*The top bar set is 3 inches from the bottom of the beam*". Note that the module will know whether the bars are in tension or compression and will handle the calculations properly.

The item labeled Bar Position This Span defines the starting and ending location of the bar ends with respect to the left end of each respective span. The data in the screen capture below shows that bar set #1 and #2 run from the left end (0.0 ft) to 25.0 ft from the left end of Span 1. Using these starting and ending locations you can fine tune the bar layout and end cutoffs.



Note: The module will report an error message if any beam segments are found to be completely unreinforced. Therefore, it is imperative that the rebar be defined in such a way as to prevent completely unreinforced segments. This includes the short segments at the extreme ends of a beam, where rebar is typically terminated. Remember that this module is an analysis tool, not a detailing tool, so don't be tempted to define rebar as starting or ending short of the physical end of the beam. The button labeled "FL" adjacent to each rebar definition has been provided as a convenient way to indicate to the program that the selected rebar runs the Full Length of the span.

Important: Bars are assumed to be fully-effective at the locations where they are defined. This can be seen from the capacity diagram. So in situations where development must be considered, the user must understand this behavior and should define bar locations accordingly.

Span Loads

No differences from other materials.

All Span Loads

No differences from other materials.

Load Combinations

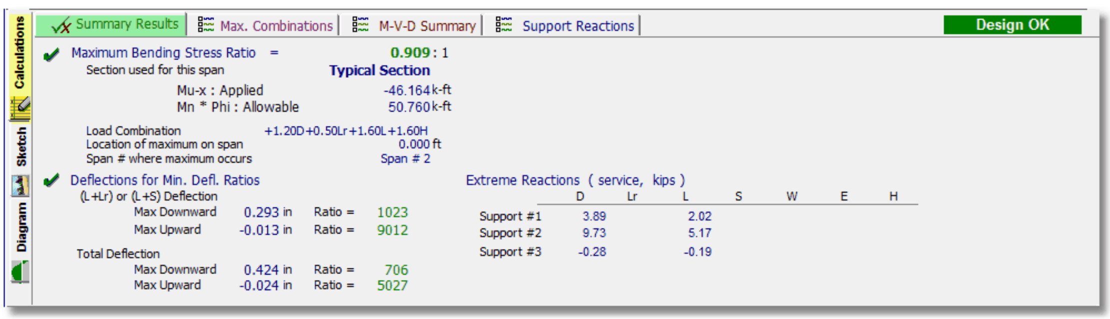
No differences from other materials.

Results Tabs

This set of tabs provides detailed results for the current calculation. The vertical tabs on the left edge of the screen allow you to select the three major areas available for review: Calculations, Sketch, and Diagram.

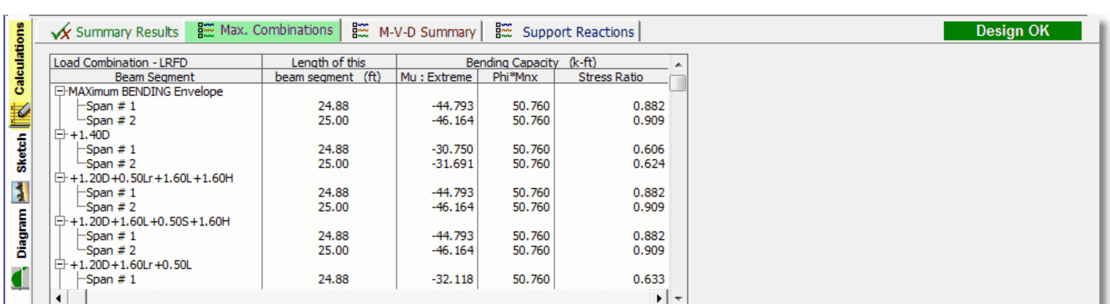
The Calculations tab offers the following results options:

Summary Results provides details for shear, moment and deflection for the governing load combinations. Shear results are not shown here...they are summarized on a separate tab that gives the required stirrup layout.

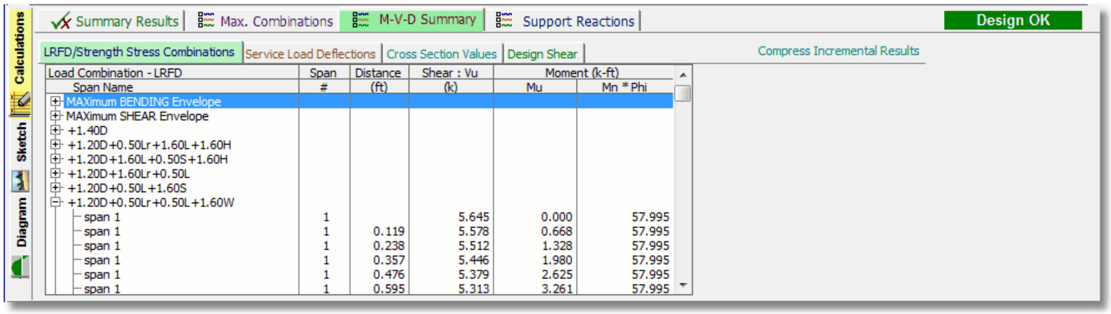


Max. Combinations provides detailed results for each beam segment for each load combination.

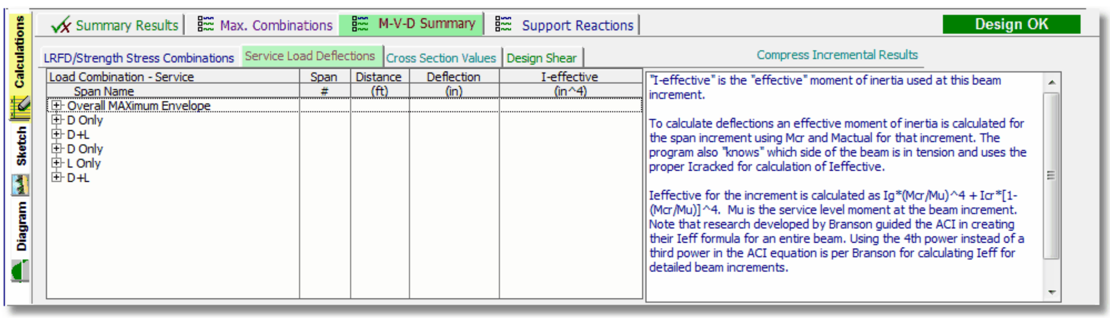
These results are a consolidation to the highly detailed incremental results on the M-V-D Summary tab.



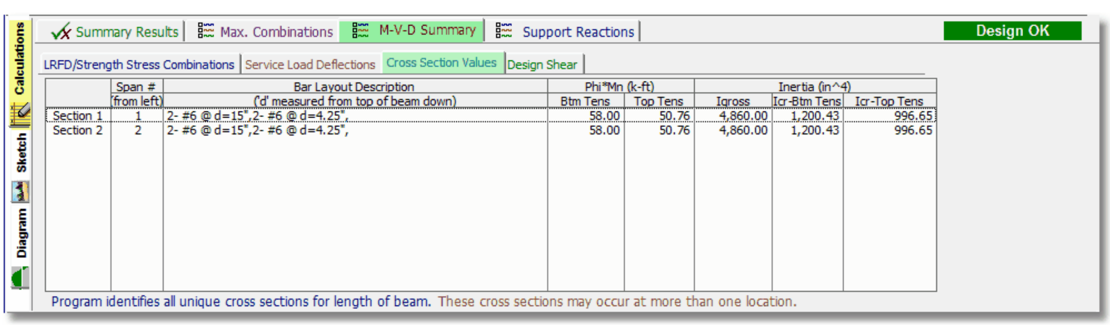
M-V-D Summary - LRFD/Strength Stress Combinations shows highly detailed moment and shear results for each beam and for each load combination. For multi-span beams using Automatic Unbalanced Live Load Placement there may be thousands of lines of results.



M-V-D Summary - Service Load Deflections shows highly detailed deflection results for all load combinations. When each load combination is expanded by clicking the [**+**] icon, you will see the deflections along the entire beam. You will also see the effective moment of inertia used in that region. (Remember that the effective moment of inertia is calculated based on service-level moments at many locations along the length of each span.)

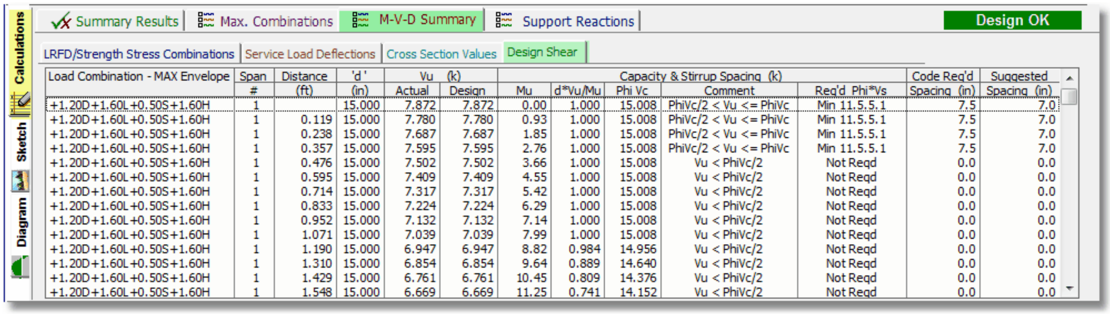


M-V-D Summary - Cross Section Values shows the moment capacities and moment of inertia for all of the identified cross sections. The module has examined all of the spans you defined and looked for identical reinforcing layouts. It has eliminated the duplicates, and for simplicity, it only lists the unique reinforced cross sections here.



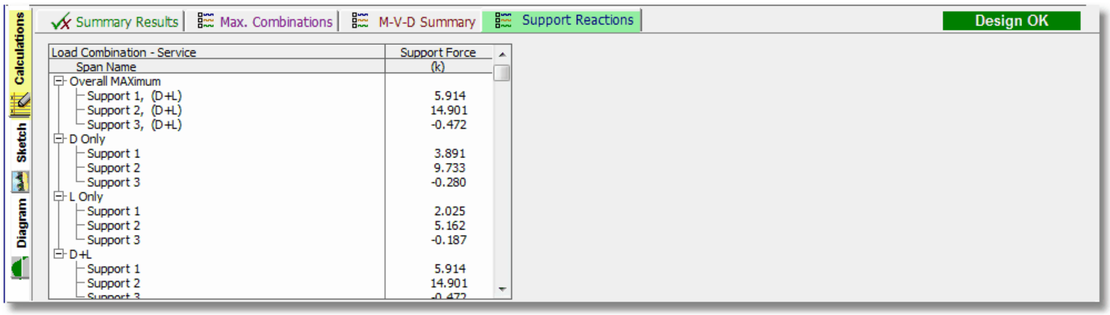
M-V-D Summary - Design Shear shows the shear stirrup requirements along the span(s) as required by the governing load combinations that generate the highest shear at each section.

The column labeled Comment indicates the ACI code condition that governs the requirement for shear reinforcing at each design location.



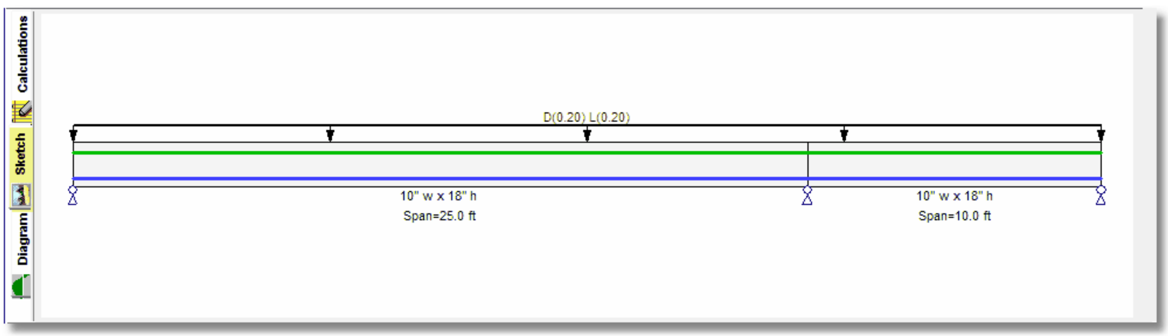
Load Combination - MAX Envelope	Span	Distance	'd'	Vu (k)		Capacity & Stirrup Spacing (k)					Code Req'd	Suggested
	#	(ft)	(in)	Actual	Design	Mu	d*Vu/Mu	Phi Vc	Comment	Req'd Phi*Vs	Spacing (in)	Spacing (in)
+1.20D+1.60L+0.50S+1.60H	1		15.000	7.872	7.872	0.00	1.000	15.008	PhiVc/2 < Vu <= PhiVc	Min 11.5.5.1	7.5	7.0
+1.20D+1.60L+0.50S+1.60H	1	0.119	15.000	7.780	7.780	0.93	1.000	15.008	PhiVc/2 < Vu <= PhiVc	Min 11.5.5.1	7.5	7.0
+1.20D+1.60L+0.50S+1.60H	1	0.238	15.000	7.687	7.687	1.85	1.000	15.008	PhiVc/2 < Vu <= PhiVc	Min 11.5.5.1	7.5	7.0
+1.20D+1.60L+0.50S+1.60H	1	0.357	15.000	7.595	7.595	2.76	1.000	15.008	PhiVc/2 < Vu <= PhiVc	Min 11.5.5.1	7.5	7.0
+1.20D+1.60L+0.50S+1.60H	1	0.476	15.000	7.502	7.502	3.66	1.000	15.008	Vu < PhiVc/2	Not Reqd	0.0	0.0
+1.20D+1.60L+0.50S+1.60H	1	0.595	15.000	7.409	7.409	4.55	1.000	15.008	Vu < PhiVc/2	Not Reqd	0.0	0.0
+1.20D+1.60L+0.50S+1.60H	1	0.714	15.000	7.317	7.317	5.42	1.000	15.008	Vu < PhiVc/2	Not Reqd	0.0	0.0
+1.20D+1.60L+0.50S+1.60H	1	0.833	15.000	7.224	7.224	6.29	1.000	15.008	Vu < PhiVc/2	Not Reqd	0.0	0.0
+1.20D+1.60L+0.50S+1.60H	1	0.952	15.000	7.132	7.132	7.14	1.000	15.008	Vu < PhiVc/2	Not Reqd	0.0	0.0
+1.20D+1.60L+0.50S+1.60H	1	1.071	15.000	7.039	7.039	7.99	1.000	15.008	Vu < PhiVc/2	Not Reqd	0.0	0.0
+1.20D+1.60L+0.50S+1.60H	1	1.190	15.000	6.947	6.947	8.82	0.984	14.956	Vu < PhiVc/2	Not Reqd	0.0	0.0
+1.20D+1.60L+0.50S+1.60H	1	1.310	15.000	6.854	6.854	9.64	0.889	14.640	Vu < PhiVc/2	Not Reqd	0.0	0.0
+1.20D+1.60L+0.50S+1.60H	1	1.429	15.000	6.761	6.761	10.45	0.809	14.376	Vu < PhiVc/2	Not Reqd	0.0	0.0
+1.20D+1.60L+0.50S+1.60H	1	1.548	15.000	6.669	6.669	11.25	0.741	14.152	Vu < PhiVc/2	Not Reqd	0.0	0.0

Support Reactions shows reactions for each support for each load condition.

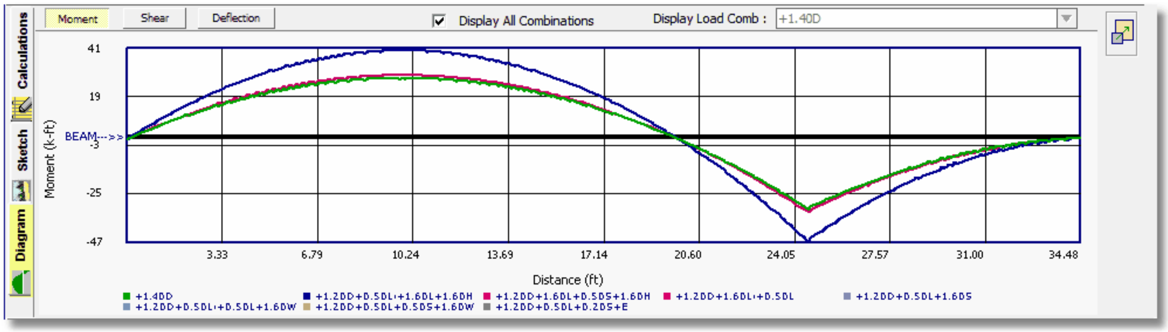


Load Combination - Service / Span Name	Support Force (k)
Overall MAXimum	
Support 1, (D+L)	5.914
Support 2, (D+L)	14.901
Support 3, (D+L)	-0.472
D Only	
Support 1	3.891
Support 2	9.733
Support 3	-0.280
L Only	
Support 1	2.025
Support 2	5.162
Support 3	-0.187
D+L	
Support 1	5.914
Support 2	14.901
Support 3	-0.472

The Sketch tab provides a graphic representation of the beam currently being designed:



The Diagram tab offers the ability to view shear, moment, and deflection diagrams for selected load combinations:


Calculations
Sketch
Diagram
Moment
Shear
Deflection
Display All Combinations
Display Load Comb :
+1.40D
41
19
BEAM--->>
-3
-25
-47
Moment (k-ft)
3.33
6.79
10.24
13.69
17.14
20.60
24.05
27.57
31.00
34.48
Distance (ft)
+1.40D
+1.20D+0.50Lr+1.60L+1.60H
+1.20D+1.60L+0.50S+1.60H
+1.20D+1.60Lr+0.50L
+1.20D+0.50L+1.60S
+1.20D+0.50Lr+0.50L+1.60W
+1.20D+0.50L+0.50S+1.60W
+1.20D+0.50L+0.20S+E

10.2.6 Beam on Elastic Foundation

Need more? Ask Us a Question

The Beam on Elastic Foundation module is a single-span version of the Concrete Beam module that provides special support conditions to represent an idealized, compression-only, continuous elastic support.

The significant variation between the Beam on Elastic Foundation module and the Concrete Beam module is the fact that the Beam on Elastic Foundation module collects a Soil Subgrade Modulus for use in modeling the spring support.

Note: This module will stop if it finds any locations where the soil springs tend to go into tension, so be sure to use realistic load combinations. For instance, don't run +W Only or +Lr Only, without the corresponding dead load that would also be applied concurrently.

Refer to the Concrete Beam module information in this document for other details.

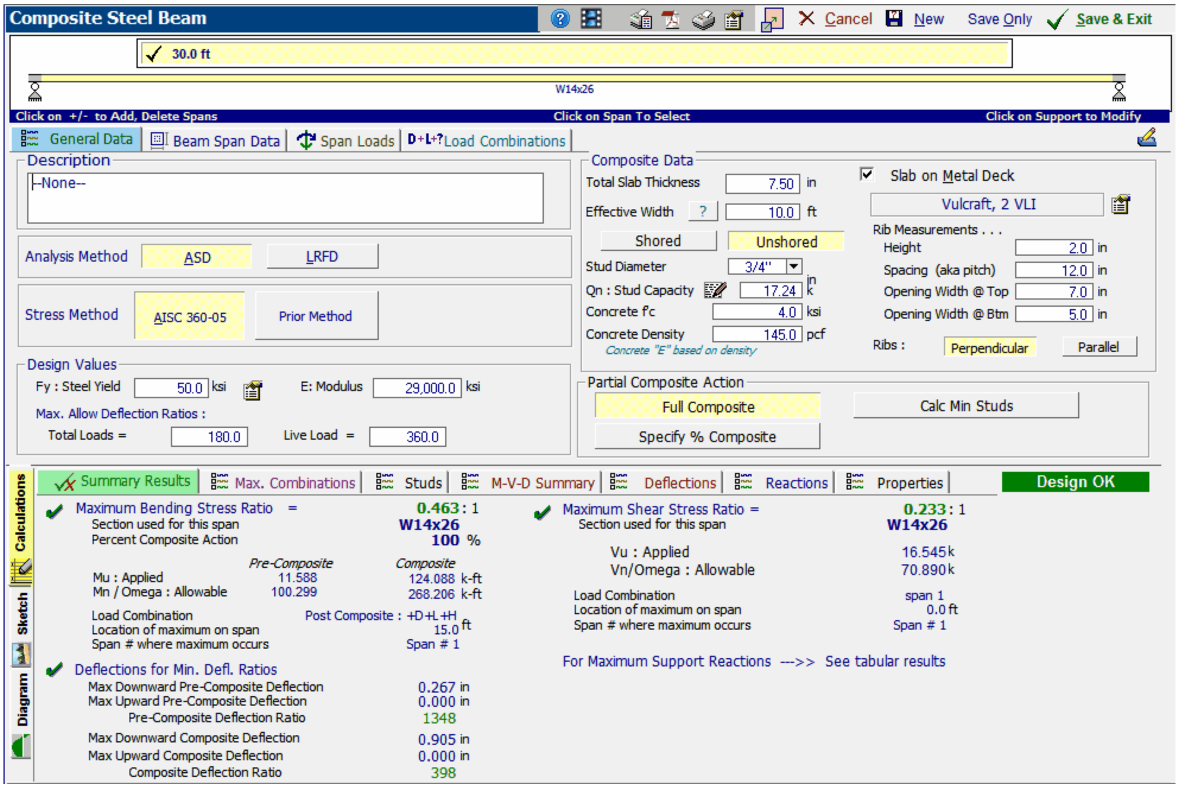
10.2.7 Composite Steel Beam

Need more? Ask Us a Question

This module provides analysis and design of AISC steel sections acting compositely with a concrete slab that is continuously connected to the compression flange of the beam with suitable shear connectors. Click here for a video: Composite Steel Beam

Features of the module include:

- The concrete slab can be either full depth or cast over formed steel decking, with rib orientation perpendicular or parallel to the beam.
- Stud capacity can be calculated by the module using standard AISC procedure.
- Normal or lightweight concrete may be used for both strength and deflection calculations.
- Both shored and unshored construction techniques can be analyzed by the module.
- ASD or LRFD design methods can be chosen.
- Flexible specification of shear studs is available.
- Very flexible loading specification, including the ability to specify construction loads (applied to pre-composite checks only), loads that apply to both the pre-composite and post-composite checks (always applied), and loads that are only applicable to the post-composite checks (applied after curing).
- Extensive load combination capability.
- Can use many sections from the AISC databases.

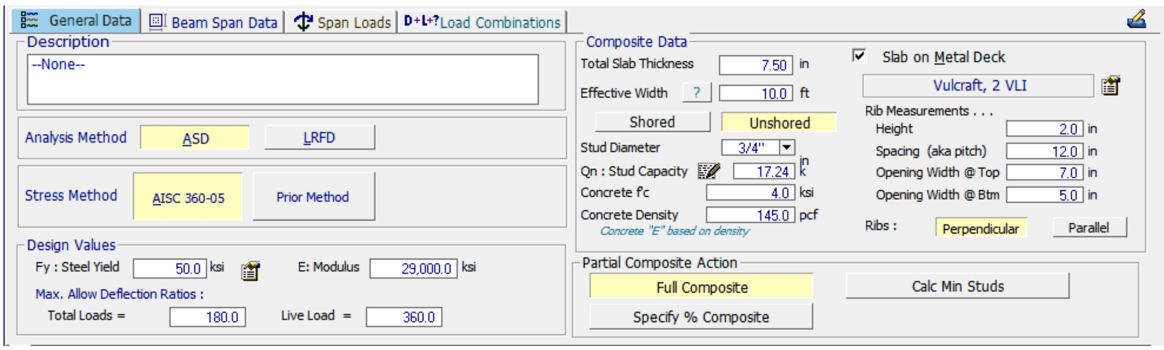


General Data Tab

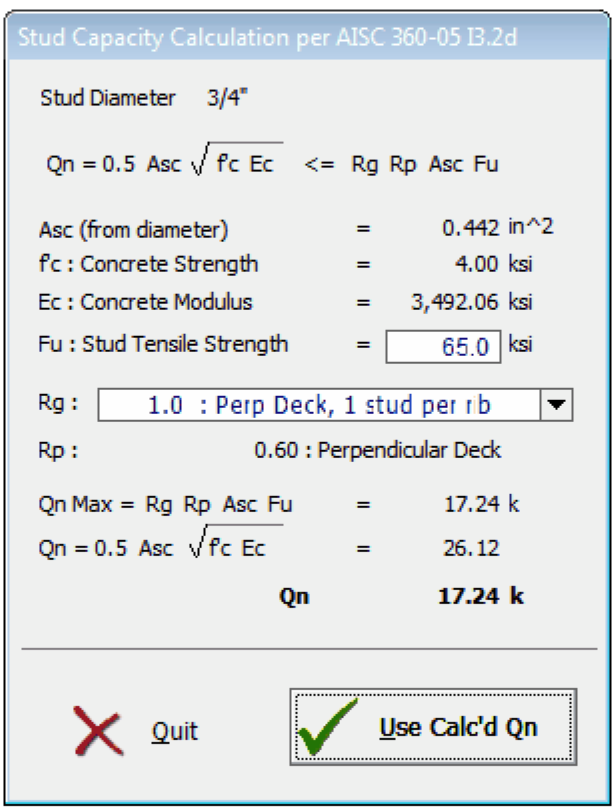
This tab gathers all input except for the beam section size and loads.

The Analysis Method category offers an option of ASD or LRFD methods.

The Composite Data category offers all of the necessary input fields to completely specify the slab, its effective width, stud information and metal deck specification (if applicable).

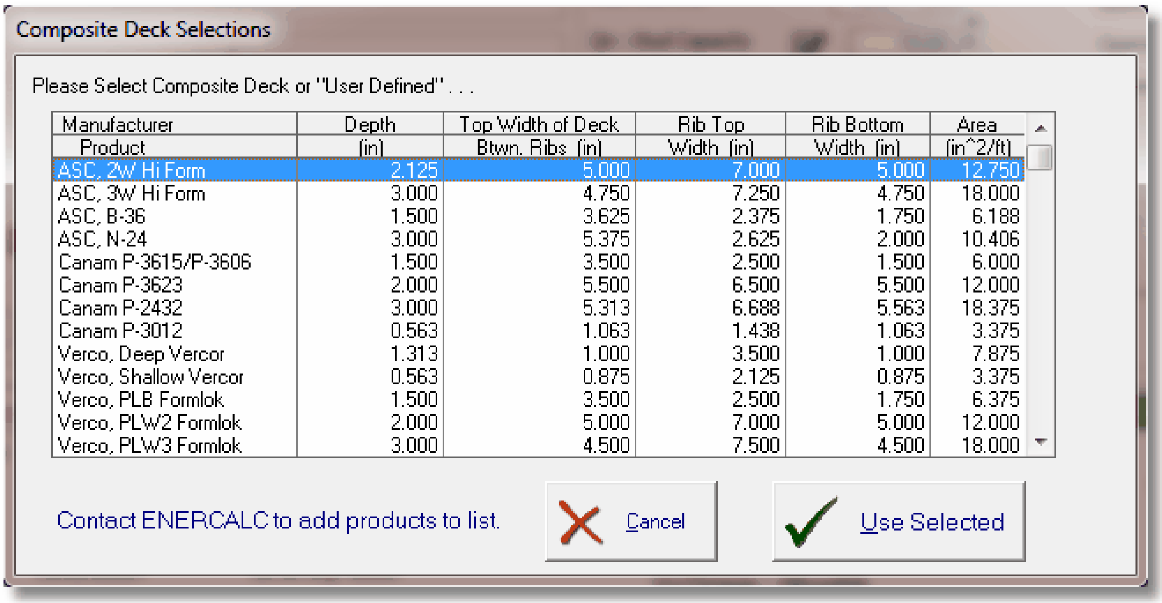


For Qn you can enter the value directly or click the button to display the stud capacity calculation dialog:



When you click the checkbox for Slab on Metal Deck, the associated input fields appear to allow the selection of a manufactured deck product, or the manual specification of the required deck properties.

Click the button to display the metal deck selection window as shown below, or simply enter the deck cross section data in the dimension input fields:



Manufacturer Product	Depth (in)	Top Width of Deck Btwn. Ribs (in)	Rib Top Width (in)	Rib Bottom Width (in)	Area (in^2/ft)
ASC, 2W Hi Form	2.125	5.000	7.000	5.000	12.750
ASC, 3W Hi Form	3.000	4.750	7.250	4.750	18.000
ASC, B-36	1.500	3.625	2.375	1.750	6.188
ASC, N-24	3.000	5.375	2.625	2.000	10.406
Canam P-3615/P-3606	1.500	3.500	2.500	1.500	6.000
Canam P-3623	2.000	5.500	6.500	5.500	12.000
Canam P-2432	3.000	5.313	6.688	5.563	18.375
Canam P-3012	0.563	1.063	1.438	1.063	3.375
Verco, Deep Vercor	1.313	1.000	3.500	1.000	7.875
Verco, Shallow Vercor	0.563	0.875	2.125	0.875	3.375
Verco, PLB Formlok	1.500	3.500	2.500	1.750	6.375
Verco, PLW2 Formlok	2.000	5.000	7.000	5.000	12.000
Verco, PLW3 Formlok	3.000	4.500	7.500	4.500	18.000

Partial Composite Action

This category provides three ways to have the module calculate stud requirements.

Full Composite tells the module to use the number of studs necessary to provide full Vh shear resistance for the slab to beam connection.

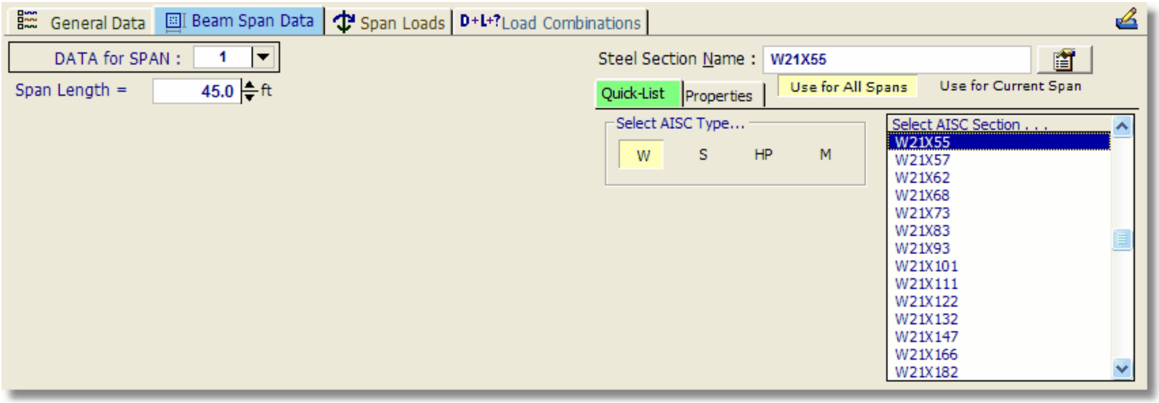
Specify % Composite allows the user to enter a percentage of maximum composite action. The module will then determine the number of studs needed for that Vh' and complete the bending capacity calculations.

Calc Min Studs tells the module to calculate the minimum number of studs (greater than 25% composite per code recommendations) that will adequately supply the required moment capacity based on the applied moment.

Beam Span Data Tab

This is basically the same tab used for a normal steel beam except that there is no ability to specify unbraced compression flange lengths. For regions of positive moment, the top flange is considered to be continuously braced by its composite connection to the concrete slab.

Note: Use caution in situations where the applied loading may result in regions of negative moment, because this module considers the bottom flange to be continuously braced.



Span Loads Tab

This tab is basically the same as the normal load entry for other beam design modules except for two differences:

1- There are two checkboxes that allow the user to indicate whether the module should automatically calculate and apply the beam self weight and slab self weight.

2- There are three "Load Application" options that allow the user to specify the sequencing of the specified loads as follows:

- **Non-Composite Only (Construction):** Dead Load and/or Live Load that will be considered in the pre-composite checks, but not in the post-composite checks.

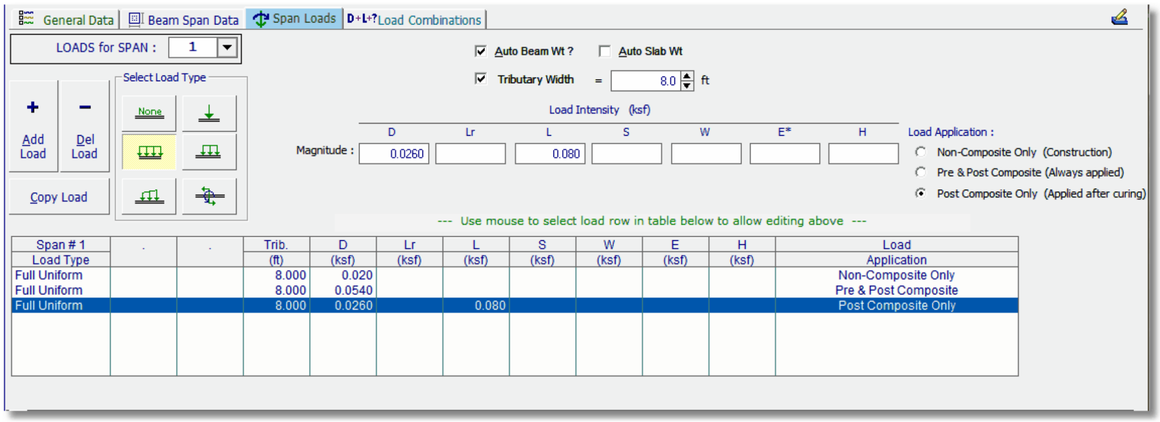
  Use this load application type to specify temporary Construction Live Load or Construction Dead Load that will be removed before the concrete cures. Do NOT use this load application type to specify permanent loads like the weight of the concrete fill on the metal deck. That type of load should be specified using the next option.

- **Pre & Post Composite (Always applied):** Dead Load that will be considered in the pre-composite AND the post-composite checks.

  Use this option for loads that will be in place while the section is non-composite and will REMAIN in place after composite action is achieved. Examples might include the weight of the concrete fill on the metal deck or the self-weight of the steel beam.

- **Post Composite Only (Applied after curing):** Any type of load that will be superimposed on the cured composite section. These will only be considered in the post-composite checks.

Use this option for loads that will be placed on the structure only after composite action has developed. Some examples might include the weight of architectural floor finishes or the weight of permanent mechanical equipment.



Load Combinations Tab

The function of the Load Combinations tab in the Composite Steel Beam module is the same as in other beam modules.

Results Tabs: This set of tabs provides detailed results for the current calculation. The vertical tabs on the left edge of the screen allow the user to select the three major areas available for review: Calculations, Sketch, and Diagram.

The Calculations tab offers the following results options:

Summary Results tab presents the maximum/governing ratio values from all of the results presented on the Max. Combinations tab. The module looks for the maximum bending stress ratio and presents the components that are calculated to create that ratio.

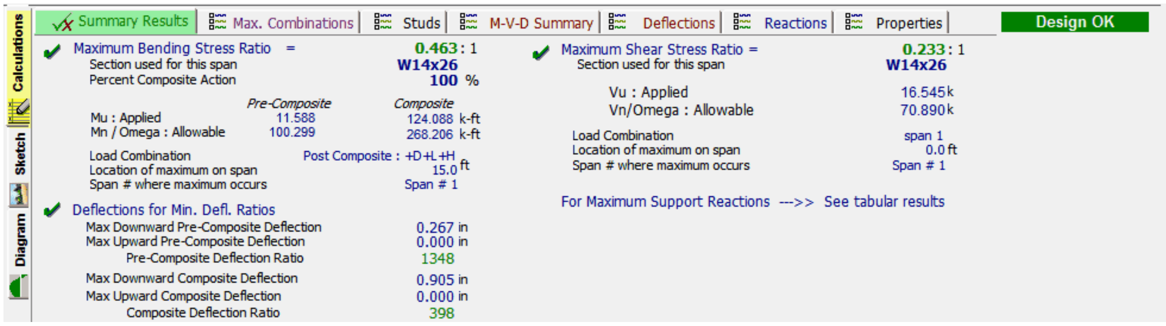
For the bending stress item the module reports the Construction bending ratio which considers all loads specified to be applied to the beam before curing. The resulting moment is compared to the capacity of the steel section acting alone (non-compositely).

For the item reported as Governing Values, the module considers all loads specified to be applied to the composite section. That moment is then compared to the full composite section moment capacity for the percentage of shear connection specified.

The results indicate the load combination that creates the governing values along with the span id and location at which the governing ratio was found to occur.

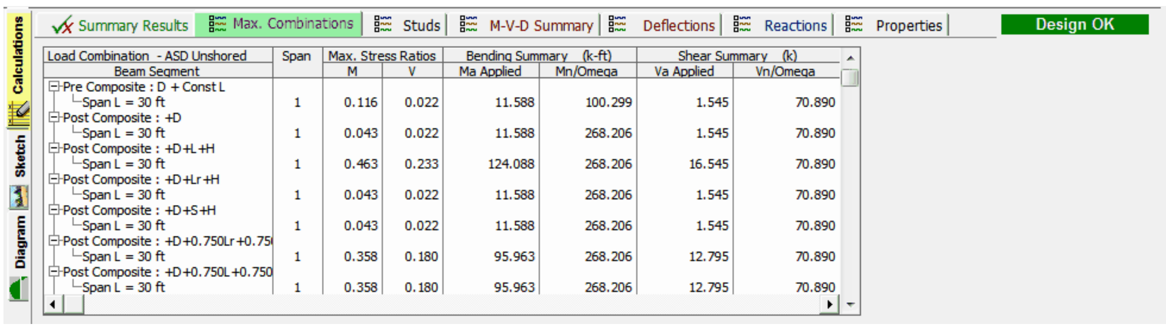
The module calculates the maximum total deflection as the sum of the deflection of the non-composite steel section resisting all of the loads that are specified as Pre & Post Composite (Always applied) plus the deflection of the composite section resisting all of the loads that are specified as Post Composite Only (Applied after curing).

Note! The screen shown below which gives information for Construction loads is only for unshored beams. For shored beams the Construction portion of moments is not applicable, so it is not shown.



Max Combinations tab presents in more concise detail the bending and shear values for all load combinations. The display varies depending upon which Stress Method is selected.

Here is a view of the summary of unshored result values when "AISC 360-05" is selected in the Stress Method category:



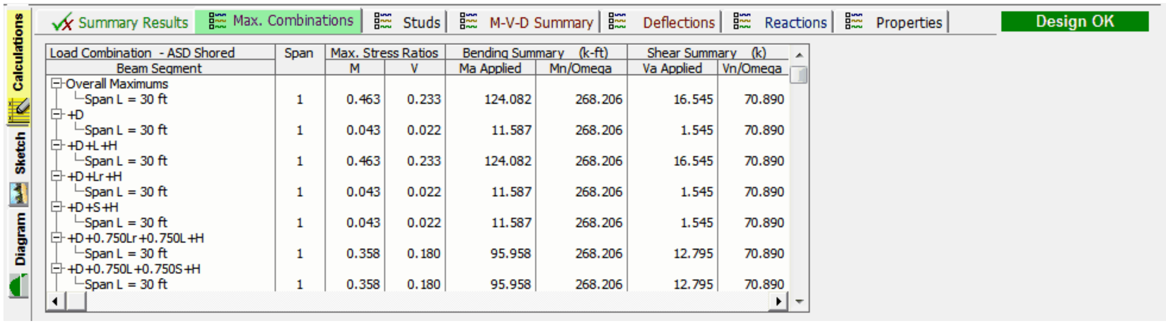
Load Combination - ASD Unshored / Beam Segment	Span	Max. Stress Ratios M	V	Bending Summary (k-ft) Ma Applied	Mn/Omega	Shear Summary (k) Va Applied	Vn/Omega
Pre Composite : D + Const L							
Span L = 30 ft	1	0.116	0.022	11.588	100.299	1.545	70.890
Post Composite : +D							
Span L = 30 ft	1	0.043	0.022	11.588	268.206	1.545	70.890
Post Composite : +D+L+H							
Span L = 30 ft	1	0.463	0.233	124.088	268.206	16.545	70.890
Post Composite : +D+Lr+H							
Span L = 30 ft	1	0.043	0.022	11.588	268.206	1.545	70.890
Post Composite : +D+S+H							
Span L = 30 ft	1	0.043	0.022	11.588	268.206	1.545	70.890
Post Composite : +D+0.750Lr+0.75							
Span L = 30 ft	1	0.358	0.180	95.963	268.206	12.795	70.890
Post Composite : +D+0.750L+0.750							
Span L = 30 ft	1	0.358	0.180	95.963	268.206	12.795	70.890

Ma Applied is the moment demand on the section due to the referenced load combination.

Mn/Omega is the moment capacity of the section at the stage of loading indicated by the referenced load combination (either pre-composite or post-composite condition).

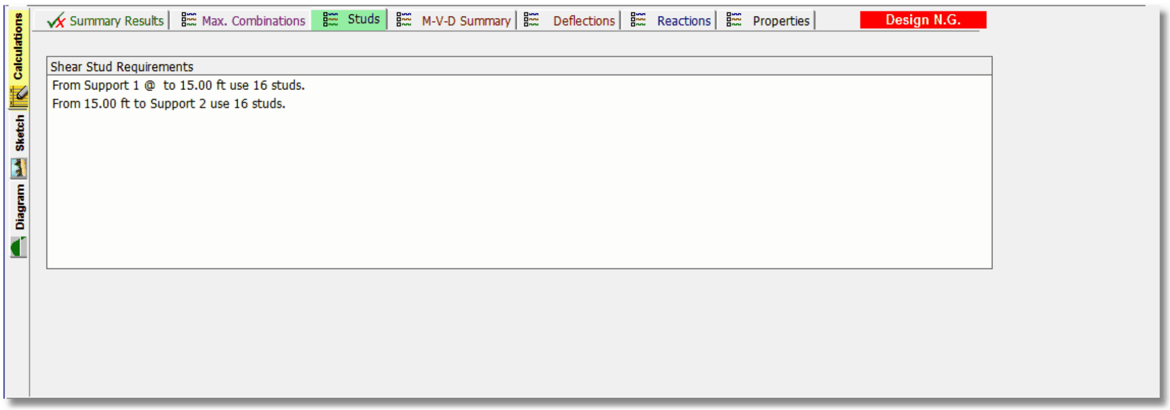
(When LRFD is used, the allowable nomenclature will change to the Mn value times Phi.)

Here is a view of the summary of shored result values when "AISC 360-05" is selected in the Stress Method category:



Load Combination - ASD Shored	Span	Max. Stress Ratios		Bending Summary (k-ft)		Shear Summary (k)	
Beam Segment		M	V	Ma Applied	Mn/Omega	Va Applied	Vn/Omega
Overall Maximums							
Span L = 30 ft	1	0.463	0.233	124.082	268.206	16.545	70.890
+D							
Span L = 30 ft	1	0.043	0.022	11.587	268.206	1.545	70.890
+D+L+H							
Span L = 30 ft	1	0.463	0.233	124.082	268.206	16.545	70.890
+D+Lr+H							
Span L = 30 ft	1	0.043	0.022	11.587	268.206	1.545	70.890
+D+S+H							
Span L = 30 ft	1	0.043	0.022	11.587	268.206	1.545	70.890
+D+0.750Lr+0.750L+H							
Span L = 30 ft	1	0.358	0.180	95.958	268.206	12.795	70.890
+D+0.750L+0.750S+H							
Span L = 30 ft	1	0.358	0.180	95.958	268.206	12.795	70.890

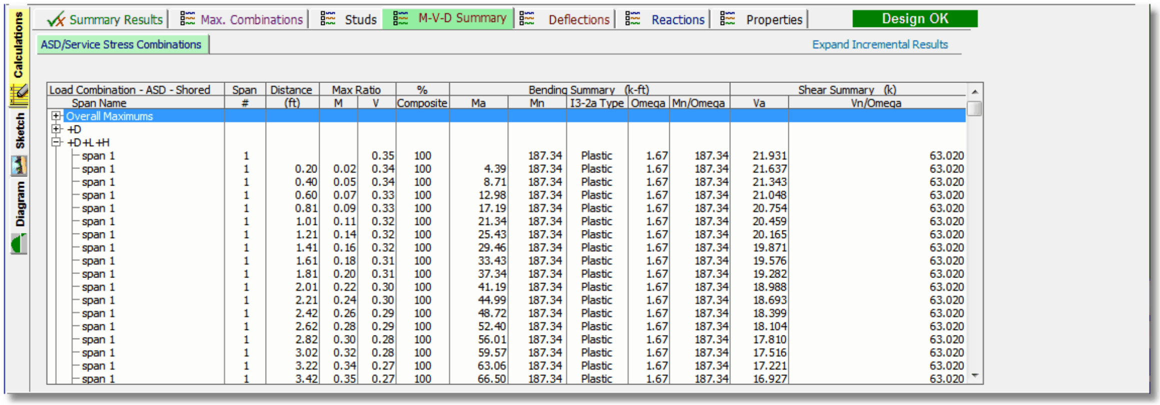
Studs tab presents the shear connector requirements for all span sections. When point loads are present, this chart may list more detailed spacing requirements because of the shear change between applied point loads.



M-V-D Summary presents moment, shear, and deflection results. Four screen captures of this tab are shown below. Depending upon the Analysis Method selection (ASD or LRFD) and the shoring selection (shored or unshored), these lists will present different values and headings.

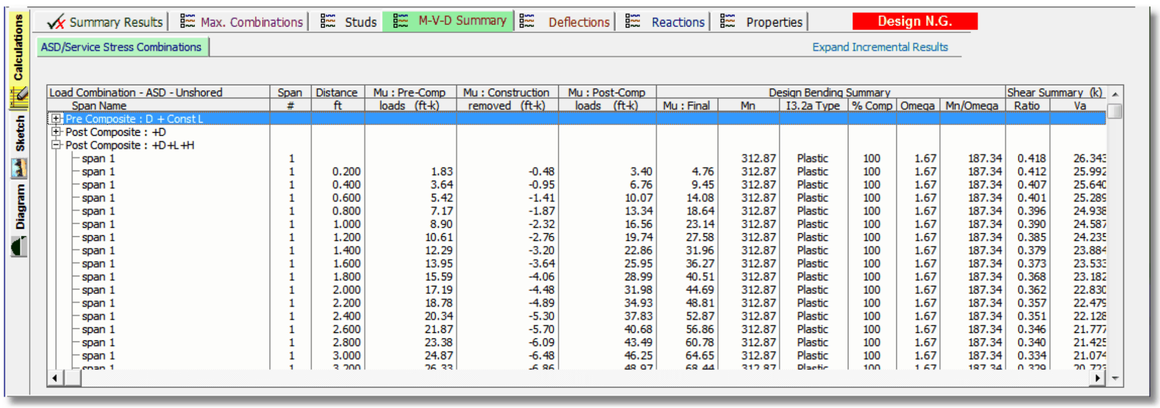
Note: There will also be subtle differences in these displays depending on which option the user has selected for the Stress Method ("AISC 360-05" or "Prior Method"). Since the "Prior Method" is being offered as a transition aid for a limited time, the screen captures below are all based on the AISC 360-05 Stress Method:

ASD/Service Stress Combinations - SHORED:



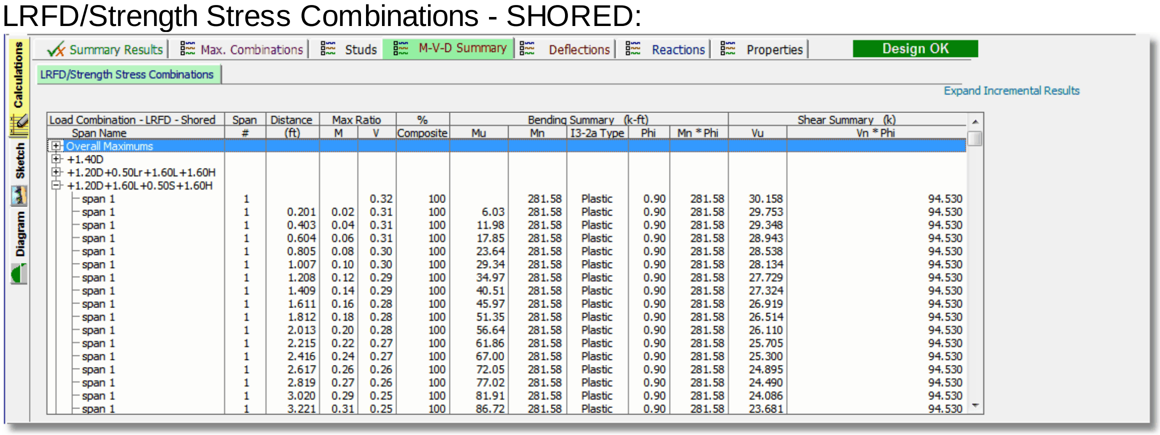
Load Combination - ASD - Shored	Span	Distance	Max Ratio		%	Bending Summary (k-ft)					Shear Summary (k)	
Span Name	#	(ft)	M	V	Composite	Ma	Mn	I3-2a Type	Omega	Mn/Omega	Va	Vn/Omega
Overall Maximums												
+D												
+D+L+H												
span 1	1			0.35	100		187.34	Plastic	1.67	187.34	21.931	63.020
span 1	1	0.20	0.02	0.34	100	4.39	187.34	Plastic	1.67	187.34	21.637	63.020
span 1	1	0.40	0.05	0.34	100	8.71	187.34	Plastic	1.67	187.34	21.343	63.020
span 1	1	0.60	0.07	0.33	100	12.98	187.34	Plastic	1.67	187.34	21.048	63.020
span 1	1	0.81	0.09	0.33	100	17.19	187.34	Plastic	1.67	187.34	20.754	63.020
span 1	1	1.01	0.11	0.32	100	21.34	187.34	Plastic	1.67	187.34	20.459	63.020
span 1	1	1.21	0.14	0.32	100	25.43	187.34	Plastic	1.67	187.34	20.165	63.020
span 1	1	1.41	0.16	0.32	100	29.46	187.34	Plastic	1.67	187.34	19.871	63.020
span 1	1	1.61	0.18	0.31	100	33.43	187.34	Plastic	1.67	187.34	19.576	63.020
span 1	1	1.81	0.20	0.31	100	37.34	187.34	Plastic	1.67	187.34	19.282	63.020
span 1	1	2.01	0.22	0.30	100	41.19	187.34	Plastic	1.67	187.34	18.988	63.020
span 1	1	2.21	0.24	0.30	100	44.99	187.34	Plastic	1.67	187.34	18.693	63.020
span 1	1	2.42	0.26	0.29	100	48.72	187.34	Plastic	1.67	187.34	18.399	63.020
span 1	1	2.62	0.28	0.29	100	52.40	187.34	Plastic	1.67	187.34	18.104	63.020
span 1	1	2.82	0.30	0.28	100	56.01	187.34	Plastic	1.67	187.34	17.810	63.020
span 1	1	3.02	0.32	0.28	100	59.57	187.34	Plastic	1.67	187.34	17.516	63.020
span 1	1	3.22	0.34	0.27	100	63.06	187.34	Plastic	1.67	187.34	17.221	63.020
span 1	1	3.42	0.35	0.27	100	66.50	187.34	Plastic	1.67	187.34	16.927	63.020

ASD/Service Stress Combinations - UNSHORED:



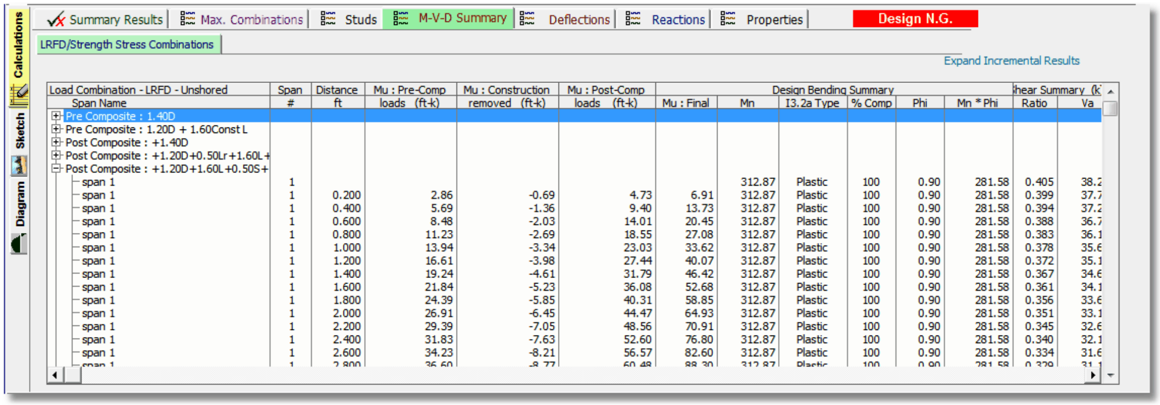
Load Combination - ASD - Unshored	Span	Distance	Mu : Pre-Comp	Mu : Construction	Mu : Post-Comp	Design Bending Summary						Shear Summary (k)	
Span Name	#	ft	loads (ft-k)	removed (ft-k)	loads (ft-k)	Mu : Final	Mn	I3.2a Type	% Comp	Omega	Mn/Omega	Ratio	Va
Pre Composite : D + Const L													
Post Composite : +D													
Post Composite : +D+L+H													
span 1	1						312.87	Plastic	100	1.67	187.34	0.418	26.343
span 1	1	0.200	1.83	-0.48	3.40	4.76	312.87	Plastic	100	1.67	187.34	0.412	25.992
span 1	1	0.400	3.64	-0.95	6.76	9.45	312.87	Plastic	100	1.67	187.34	0.407	25.640
span 1	1	0.600	5.42	-1.41	10.07	14.08	312.87	Plastic	100	1.67	187.34	0.401	25.289
span 1	1	0.800	7.17	-1.87	13.34	18.64	312.87	Plastic	100	1.67	187.34	0.396	24.938
span 1	1	1.000	8.90	-2.32	16.56	23.14	312.87	Plastic	100	1.67	187.34	0.390	24.587
span 1	1	1.200	10.61	-2.76	19.74	27.58	312.87	Plastic	100	1.67	187.34	0.385	24.235
span 1	1	1.400	12.29	-3.20	22.86	31.96	312.87	Plastic	100	1.67	187.34	0.379	23.884
span 1	1	1.600	13.95	-3.64	25.95	36.27	312.87	Plastic	100	1.67	187.34	0.373	23.533
span 1	1	1.800	15.59	-4.06	28.99	40.51	312.87	Plastic	100	1.67	187.34	0.368	23.182
span 1	1	2.000	17.19	-4.48	31.98	44.69	312.87	Plastic	100	1.67	187.34	0.362	22.830
span 1	1	2.200	18.78	-4.89	34.93	48.81	312.87	Plastic	100	1.67	187.34	0.357	22.479
span 1	1	2.400	20.34	-5.30	37.83	52.87	312.87	Plastic	100	1.67	187.34	0.351	22.128
span 1	1	2.600	21.87	-5.70	40.68	56.86	312.87	Plastic	100	1.67	187.34	0.346	21.777
span 1	1	2.800	23.38	-6.09	43.49	60.78	312.87	Plastic	100	1.67	187.34	0.340	21.425
span 1	1	3.000	24.87	-6.48	46.25	64.65	312.87	Plastic	100	1.67	187.34	0.334	21.074
span 1	1	3.200	26.33	-6.86	48.97	68.44	312.87	Plastic	100	1.67	187.34	0.329	[illegible]

LRFD/Strength Stress Combinations - SHORED:



Load Combination - LRFD - Shored	Span	Distance	Max Ratio		%	Bending Summary (k-ft)					Shear Summary (k)	
Span Name	#	(ft)	M	V	Composite	Mu	Mn	I3-2a Type	Phi	Mn * Phi	Vu	Vn * Phi
Overall Maximums												
+1.40D												
+1.20D+0.50Lr+1.60L+1.60H												
+1.20D+1.60L+0.50S+1.60H												
span 1	1			0.32	100		281.58	Plastic	0.90	281.58	30.158	94.530
span 1	1	0.201	0.02	0.31	100	6.03	281.58	Plastic	0.90	281.58	29.753	94.530
span 1	1	0.403	0.04	0.31	100	11.98	281.58	Plastic	0.90	281.58	29.348	94.530
span 1	1	0.604	0.06	0.31	100	17.85	281.58	Plastic	0.90	281.58	28.943	94.530
span 1	1	0.805	0.08	0.30	100	23.64	281.58	Plastic	0.90	281.58	28.538	94.530
span 1	1	1.007	0.10	0.30	100	29.34	281.58	Plastic	0.90	281.58	28.134	94.530
span 1	1	1.208	0.12	0.29	100	34.97	281.58	Plastic	0.90	281.58	27.729	94.530
span 1	1	1.409	0.14	0.29	100	40.51	281.58	Plastic	0.90	281.58	27.324	94.530
span 1	1	1.611	0.16	0.28	100	45.97	281.58	Plastic	0.90	281.58	26.919	94.530
span 1	1	1.812	0.18	0.28	100	51.35	281.58	Plastic	0.90	281.58	26.514	94.530
span 1	1	2.013	0.20	0.28	100	56.64	281.58	Plastic	0.90	281.58	26.110	94.530
span 1	1	2.215	0.22	0.27	100	61.86	281.58	Plastic	0.90	281.58	25.705	94.530
span 1	1	2.416	0.24	0.27	100	67.00	281.58	Plastic	0.90	281.58	25.300	94.530
span 1	1	2.617	0.26	0.26	100	72.05	281.58	Plastic	0.90	281.58	24.895	94.530
span 1	1	2.819	0.27	0.26	100	77.02	281.58	Plastic	0.90	281.58	24.490	94.530
span 1	1	3.020	0.29	0.25	100	81.91	281.58	Plastic	0.90	281.58	24.086	94.530
span 1	1	3.221	0.31	0.25	100	86.72	281.58	Plastic	0.90	281.58	23.681	94.530

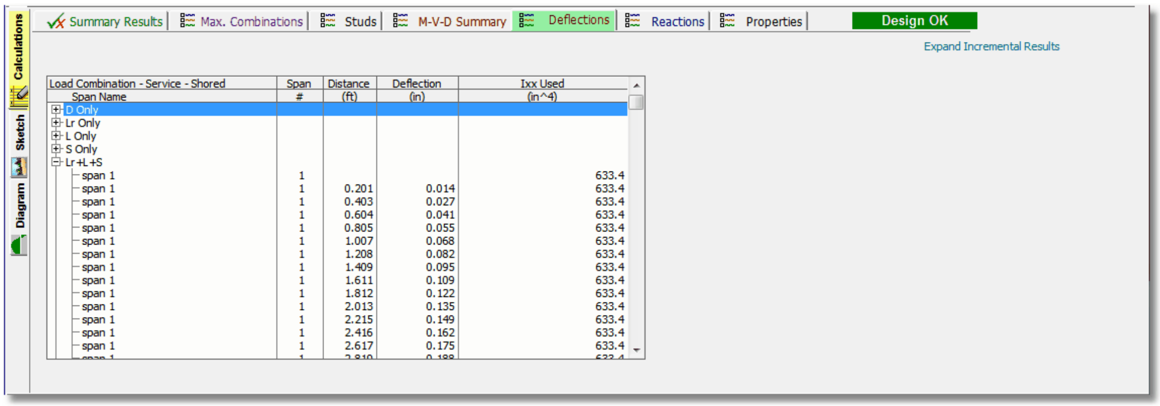
LRFD/Strength Stress Combinations - UNSHORED:



Load Combination - LRFD - Unshored	Span	Distance	Mu : Pre-Comp	Mu : Construction	Mu : Post-Comp	Design Bending Summary						hear Summary (k	
Span Name	#	ft	loads (ft-k)	removed (ft-k)	loads (ft-k)	Mu : Final	Mn	I3.2a Type	% Comp	Phi	Mn * Phi	Ratio	Va
Pre Composite : 1.40D													
Pre Composite : 1.20D + 1.60Const L													
Post Composite : +1.40D													
Post Composite : +1.20D+0.50Lr+1.60L+													
Post Composite : +1.20D+1.60L+0.50S+													
span 1	1						312.87	Plastic	100	0.90	281.58	0.405	38.2
span 1	1	0.200	2.86	-0.69	4.73	6.91	312.87	Plastic	100	0.90	281.58	0.399	37.7
span 1	1	0.400	5.69	-1.36	9.40	13.73	312.87	Plastic	100	0.90	281.58	0.394	37.2
span 1	1	0.600	8.48	-2.03	14.01	20.45	312.87	Plastic	100	0.90	281.58	0.388	36.7
span 1	1	0.800	11.23	-2.69	18.55	27.08	312.87	Plastic	100	0.90	281.58	0.383	36.1
span 1	1	1.000	13.94	-3.34	23.03	33.62	312.87	Plastic	100	0.90	281.58	0.378	35.6
span 1	1	1.200	16.61	-3.98	27.44	40.07	312.87	Plastic	100	0.90	281.58	0.372	35.1
span 1	1	1.400	19.24	-4.61	31.79	46.42	312.87	Plastic	100	0.90	281.58	0.367	34.6
span 1	1	1.600	21.84	-5.23	36.08	52.68	312.87	Plastic	100	0.90	281.58	0.361	34.1
span 1	1	1.800	24.39	-5.85	40.31	58.85	312.87	Plastic	100	0.90	281.58	0.356	33.6
span 1	1	2.000	26.91	-6.45	44.47	64.93	312.87	Plastic	100	0.90	281.58	0.351	33.1
span 1	1	2.200	29.39	-7.05	48.56	70.91	312.87	Plastic	100	0.90	281.58	0.345	32.6
span 1	1	2.400	31.83	-7.63	52.60	76.80	312.87	Plastic	100	0.90	281.58	0.340	32.1
span 1	1	2.600	34.23	-8.21	56.57	82.60	312.87	Plastic	100	0.90	281.58	0.334	31.6

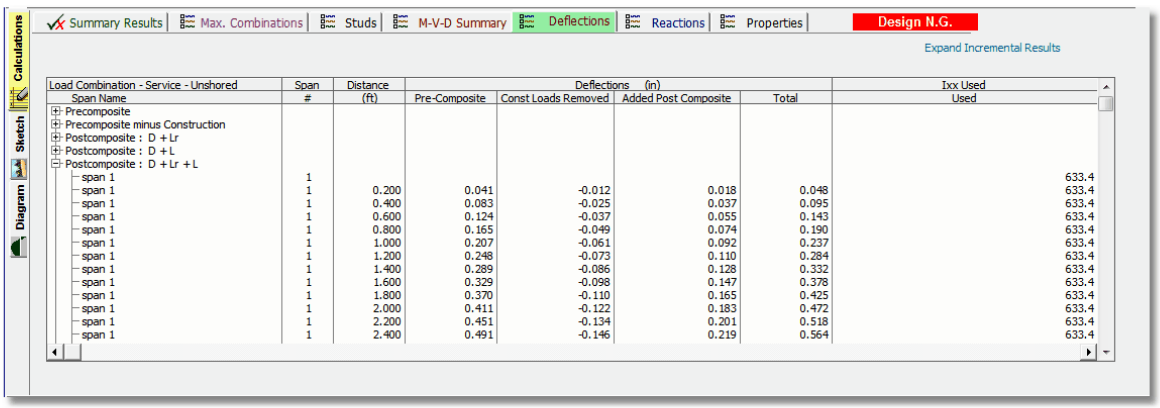
Deflections tab has two versions, one for shored and one for unshored construction. The only differences are the load combinations and the explanations of load applications listed.

Service Deflections - SHORED:



Load Combination - Service - Shored	Span	Distance	Deflection	Ixx Used
Span Name	#	(ft)	(in)	(in^4)
D Only				
Lr Only				
L Only				
S Only				
Lr+L+S				
span 1	1			633.4
span 1	1	0.201	0.014	633.4
span 1	1	0.403	0.027	633.4
span 1	1	0.604	0.041	633.4
span 1	1	0.805	0.055	633.4
span 1	1	1.007	0.068	633.4
span 1	1	1.208	0.082	633.4
span 1	1	1.409	0.095	633.4
span 1	1	1.611	0.109	633.4
span 1	1	1.812	0.122	633.4
span 1	1	2.013	0.135	633.4
span 1	1	2.215	0.149	633.4
span 1	1	2.416	0.162	633.4
span 1	1	2.617	0.175	633.4

Service Deflections - UNSHORED:



Load Combination - Service - Unshored	Span	Distance	Deflections (in)				Ixx Used
Span Name	#	(ft)	Pre-Composite	Const Loads Removed	Added Post Composite	Total	Used
Precomposite							
Precomposite minus Construction							
Postcomposite : D + Lr							
Postcomposite : D + L							
Postcomposite : D + Lr + L							
span 1	1						633.4
span 1	1	0.200	0.041	-0.012	0.018	0.048	633.4
span 1	1	0.400	0.083	-0.025	0.037	0.095	633.4
span 1	1	0.600	0.124	-0.037	0.055	0.143	633.4
span 1	1	0.800	0.165	-0.049	0.074	0.190	633.4
span 1	1	1.000	0.207	-0.061	0.092	0.237	633.4
span 1	1	1.200	0.248	-0.073	0.110	0.284	633.4
span 1	1	1.400	0.289	-0.086	0.128	0.332	633.4
span 1	1	1.600	0.329	-0.098	0.147	0.378	633.4
span 1	1	1.800	0.370	-0.110	0.165	0.425	633.4
span 1	1	2.000	0.411	-0.122	0.183	0.472	633.4
span 1	1	2.200	0.451	-0.134	0.201	0.518	633.4
span 1	1	2.400	0.491	-0.146	0.219	0.564	633.4

In the Unshored Service Deflections table, the columns warrant some detailed explanation:

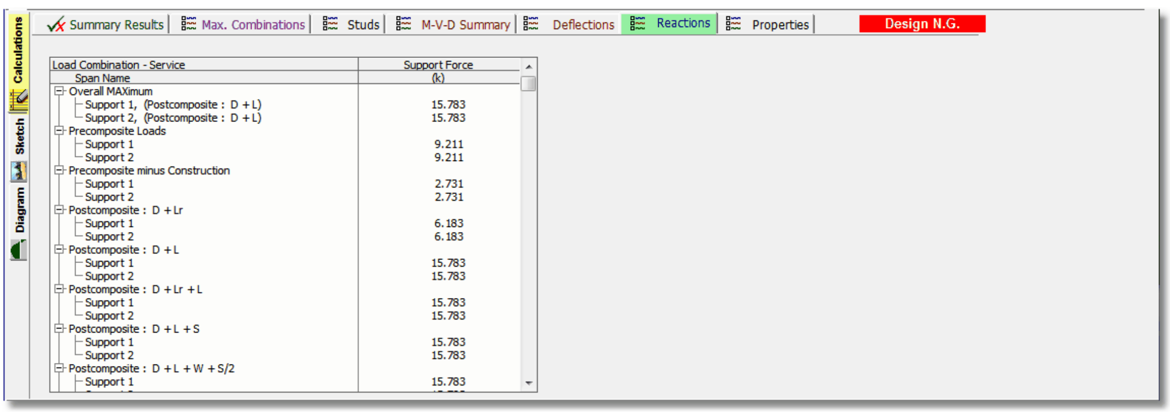
The column labeled "Pre-Composite"shows the deflection of the bare steel beam subjected to all loads that are specified to act on the bare steel beam. This would include all loads defined using the "Non-Composite (Construction)" option and all loads defined using the "Pre & Post Composite (Always Applied)" option on the Span Loads tab.

The column labeled "Construction Loads Removed" shows the deflection of the bare steel beam due all loads defined using the "Non-Composite (Construction)" option on the Span Loads tab. Since these loads are removed before the beam reaches its service condition, these construction load deflections are removed from the total deflection, so that the Total deflection represents a correct net total deflection in the service condition.

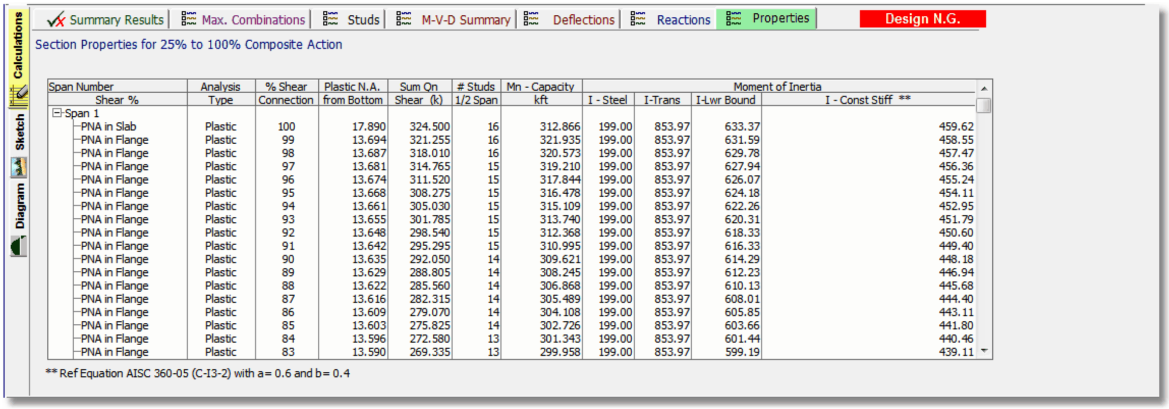
The column labeled "Added Post Composite" shows the deflection of the composite section due to the loads that are superimposed on the section after it has achieved composite action. This would include all loads defined using the "Post Composite Only (Applied after curing)" option on the Span Loads tab.

The column labeled "Total" is calculated by adding the value in the "Pre-Composite" column to the value in the "Added Post Composite" column and then netting out the value in the "Construction Loads Removed" column. This way, the "Total" deflection represents the full anticipated in-service deflection considering all permanent loads and properly accounting for their sequence of application.

Reactions tab has two versions, one for shored and one for unshored construction. The only differences are the load combinations and the explanations of load applications listed.



Properties tab shows the calculations for transformed section properties calculated in increments of 1% shear connection, from 100% down to the code minimum 25%. "I Lower Bound" and "I Constant Stiffness" are terms described in AISC 360-05.


Summary Results | Max. Combinations | Studs | M-V-D Summary | Deflections | Reactions | Properties | Design N.G.

Calculations | Sketch | Diagram

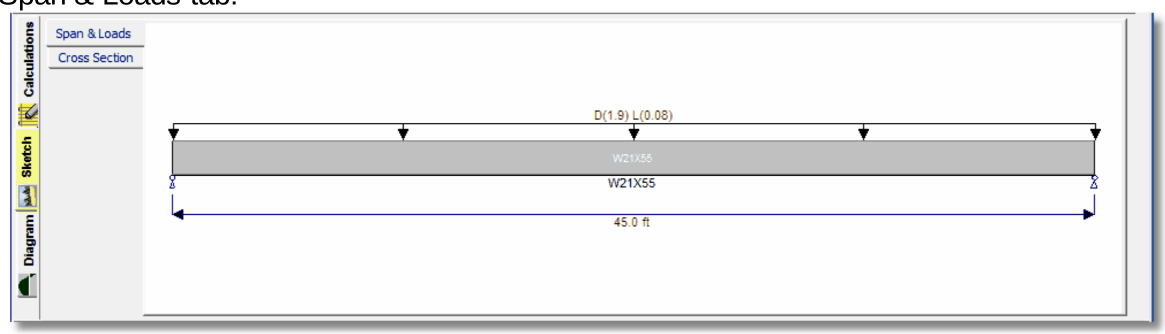
Section Properties for 25% to 100% Composite Action

Span Number	Analysis	% Shear	Plastic N.A.	Sum Qn	# Studs	Mn - Capacity	Moment of Inertia			
Shear %	Type	Connection	from Bottom	Shear (k)	1/2 Span	kft	I - Steel	I-Trans	I-Lwr Bound	I - Const Stiff **
Span 1										
PNA in Slab	Plastic	100	17.890	324.500	16	312.866	199.00	853.97	633.37	459.62
PNA in Flange	Plastic	99	13.694	321.255	16	321.935	199.00	853.97	631.59	458.55
PNA in Flange	Plastic	98	13.687	318.010	16	320.573	199.00	853.97	629.78	457.47
PNA in Flange	Plastic	97	13.681	314.765	15	319.210	199.00	853.97	627.94	456.36
PNA in Flange	Plastic	96	13.674	311.520	15	317.844	199.00	853.97	626.07	455.24
PNA in Flange	Plastic	95	13.668	308.275	15	316.478	199.00	853.97	624.18	454.11
PNA in Flange	Plastic	94	13.661	305.030	15	315.109	199.00	853.97	622.26	452.95
PNA in Flange	Plastic	93	13.655	301.785	15	313.740	199.00	853.97	620.31	451.79
PNA in Flange	Plastic	92	13.648	298.540	15	312.368	199.00	853.97	618.33	450.60
PNA in Flange	Plastic	91	13.642	295.295	15	310.995	199.00	853.97	616.33	449.40
PNA in Flange	Plastic	90	13.635	292.050	14	309.621	199.00	853.97	614.29	448.18
PNA in Flange	Plastic	89	13.629	288.805	14	308.245	199.00	853.97	612.23	446.94
PNA in Flange	Plastic	88	13.622	285.560	14	306.868	199.00	853.97	610.13	445.68
PNA in Flange	Plastic	87	13.616	282.315	14	305.489	199.00	853.97	608.01	444.40
PNA in Flange	Plastic	86	13.609	279.070	14	304.108	199.00	853.97	605.85	443.11
PNA in Flange	Plastic	85	13.603	275.825	14	302.726	199.00	853.97	603.66	441.80
PNA in Flange	Plastic	84	13.596	272.580	13	301.343	199.00	853.97	601.44	440.46
PNA in Flange	Plastic	83	13.590	269.335	13	299.958	199.00	853.97	599.19	439.11

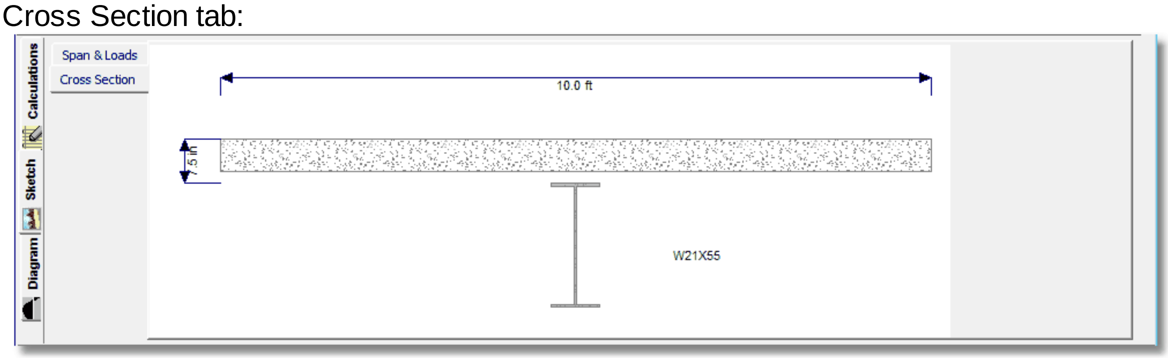
** Ref Equation AISC 360-05 (C-I3-2) with a= 0.6 and b= 0.4

The Sketch tab provides two ways to view a graphic representation of the beam currently being designed:

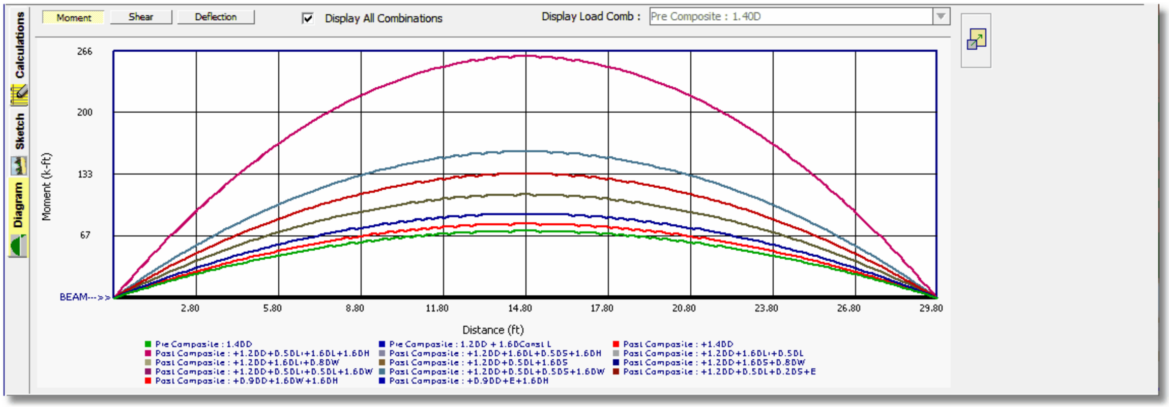
Span & Loads tab:



Cross Section tab:



The Diagram tab offers the ability to view shear, moment, and deflection diagrams for selected load combinations:


Calculations
Sketch
Diagram
Moment
Shear
Deflection
Display All Combinations
Display Load Comb :
Pre Composite : 1.40D
Moment (k-ft)
266
200
133
67
BEAM--->>
2.80
5.80
8.80
11.80
14.80
17.80
20.80
23.80
26.80
29.80
Distance (ft)
Pre Composite : 1.40D
Pre Composite : 1.20D + 1.60Const L
Post Composite : +1.40D
Post Composite : +1.20D+0.50Lr+1.60L+1.60H
Post Composite : +1.20D+1.60L+0.50S+1.60H
Post Composite : +1.20D+1.60Lr+0.50L
Post Composite : +1.20D+1.60Lr+0.80W
Post Composite : +1.20D+0.50L+1.60S
Post Composite : +1.20D+1.60S+0.80W
Post Composite : +1.20D+0.50Lr+0.50L+1.60W
Post Composite : +1.20D+0.50L+0.50S+1.60W
Post Composite : +1.20D+0.50L+0.20S+E
Post Composite : +0.90D+1.60W+1.60H
Post Composite : +0.90D+E+1.60H

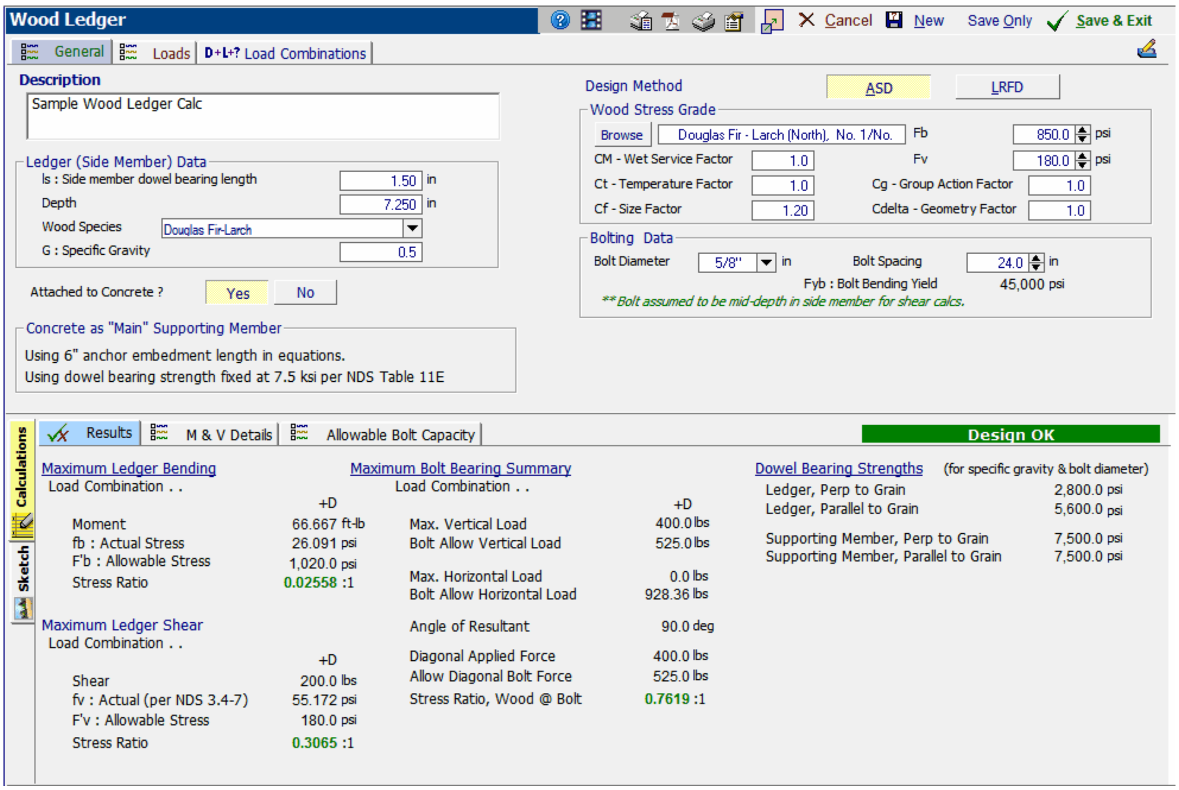
10.2.8 Wood Ledger

Need more? Ask Us a Question

The wood ledger module provides the ability to calculate moments and shears in the ledger, as well as actual and allowable bearing loads on the attaching bolts.

The module allows the ledger to be attached to concrete or another wood member, and it automatically calculates the proper allowable bolt values.

All calculations are according to 2005 NDS and either 2006 IBC or 2009 IBC. Allowable bolt capacity at an angle to the grain is calculated using Hankinson's formula.



General Tab

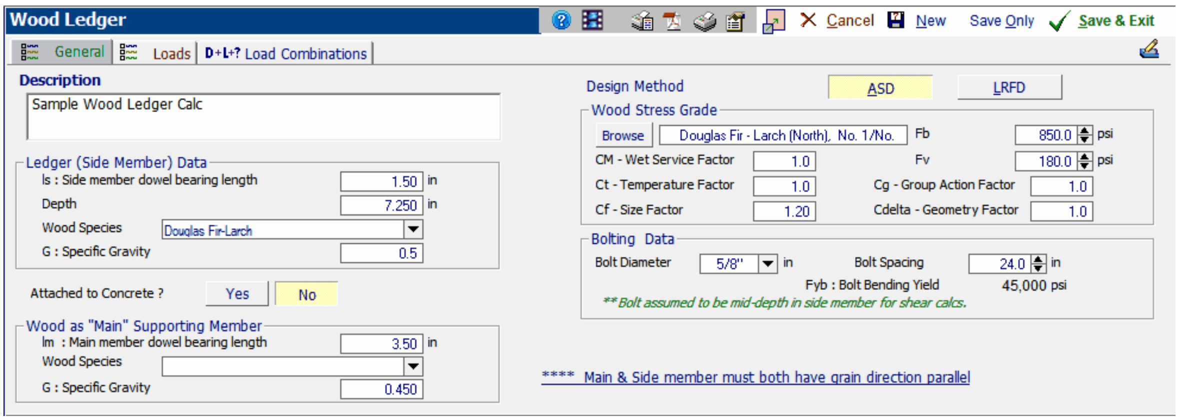
This tab collects information on the ledger size and stress grade, and bolt information. If the ledger in

Ledger Data: Enter the actual dimensions (not nominal) and the wood species of the ledger. The specific gravity will be retrieved from the internal databases (you can also revise the specific gravity).

Wood Stress Grade: Use the [**Browse**] button to access the built-in NDS reference design values database and retrieve the Fb and Fv values. You can also edit these values separately.

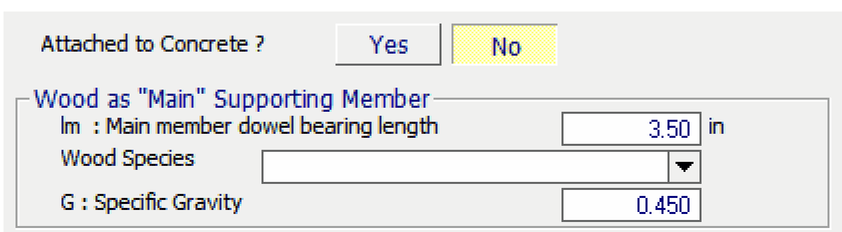
Bolting Data: Enter the bolt diameter and spacing. The yield strength of the bolt is fixed at 45 ksi in this module to remain consistent with the assumptions in the NDS.

ASD or LRFD: Select the design method to be used. Load factoring and allowable stress calculations will applied accordingly. Values of K_F and phi are automatically determined and applied for the LRFD method.

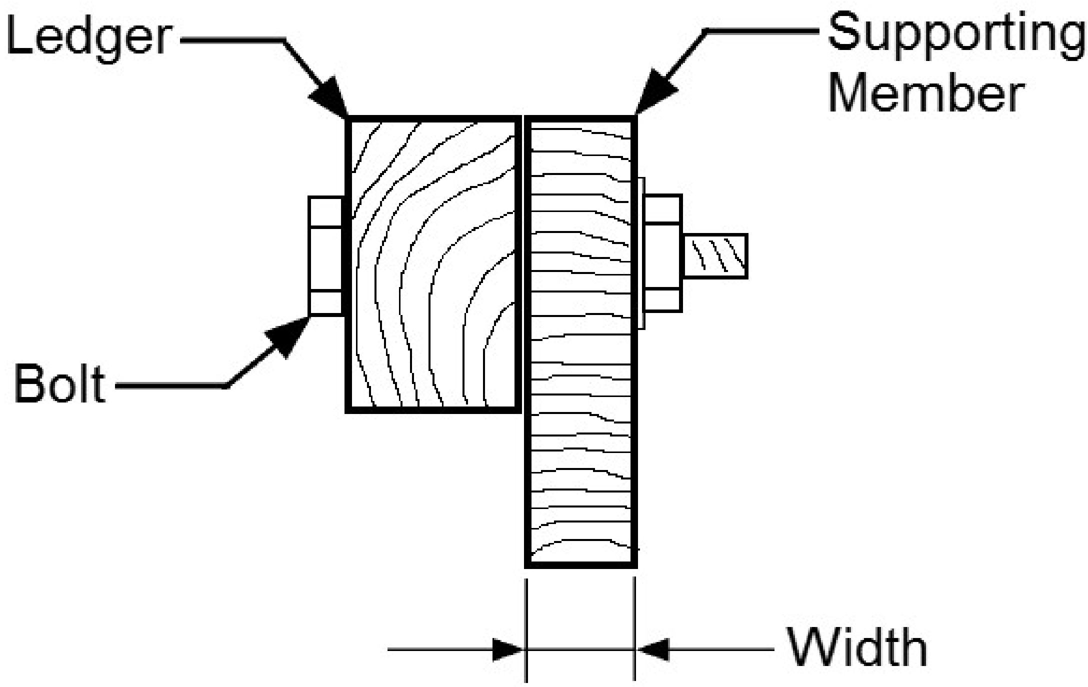


When "Attached to Concrete" is set to [No]

When the user specifies that the ledger is not attached to concrete, this implies that the ledger is attached to a supporting wood member. The display will change to allow the width and species of the supporting member to be entered.



The following sketch clarifies the width dimension required for a wood supporting member:



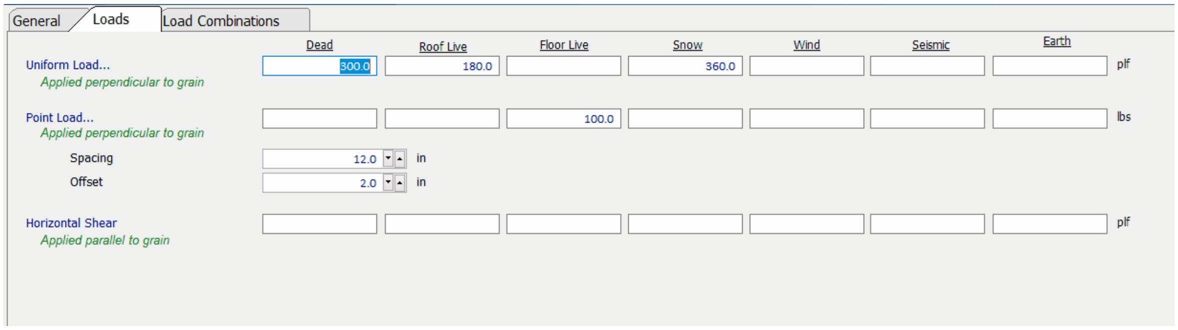
Loads Tab

This tab allows you to apply vertical and horizontal loads to the ledger. All loads are assumed to act in the plane of the wall. Vertical loads might come from gravity loading on supported members. Horizontal loading might come from diaphragm action, such as the floor system dragging lateral wind or seismic load into the ledger. When both vertical and horizontal loads are applied, the resulting load will be at an angle to the bolt that is somewhere between 0 degrees (for purely horizontal loading) and 90 degrees (for purely vertical loading).

One set of uniform load values can be specified, and it will be considered to act consistently along all areas of the ledger. For calculations purposes, this module considers the ledger a continuous beam over multiple supports.

One set of repeating point loads can be specified, where the input collects the magnitude, the starting location, and the spacing between subsequent loads. The module will consider all locations of the point loads over enough ledger spans between bolts to determine the governing case. For example, say your ledger bolting is set to 36" and the point load is set to 15". There will be multiple point loads between bolts, and on the NEXT span, the point loads will be in different relative positions with respect to the bolts. The module analyzes all conditions of that bolt pattern over enough ledger spans to determine the governing point load offset.

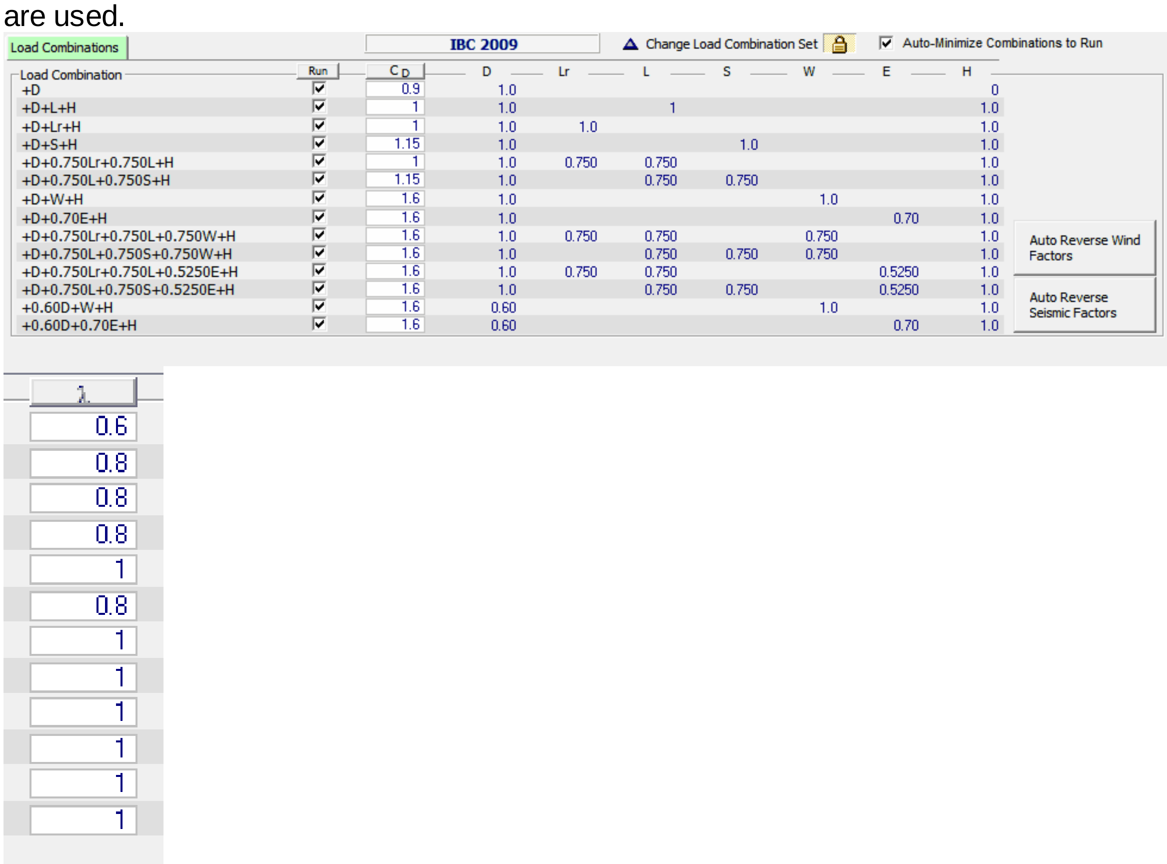
One set of horizontal (lateral) shear loads is allowed, in order to simulate wind or seismic load applied to the ledger and acting in the plane of the supporting wall.



Load Combinations Tab

This is the typical load combination tab with entries for load duration factors C_D (ASD) and λ (LRFD).

Selecting the [**Auto Reverse Wind Factors**] and/or [**Auto Reverse Seismic Factors**] buttons creates additional load combinations that insert "-W" and "-E" whenever W and E are used.



Results Tabs

This set of tabs provides detailed results for the current calculation. The vertical tabs on the left edge of the screen allow you to select the two major areas available for review: Calculations and Sketch.

The Calculations tab offers the following results options:

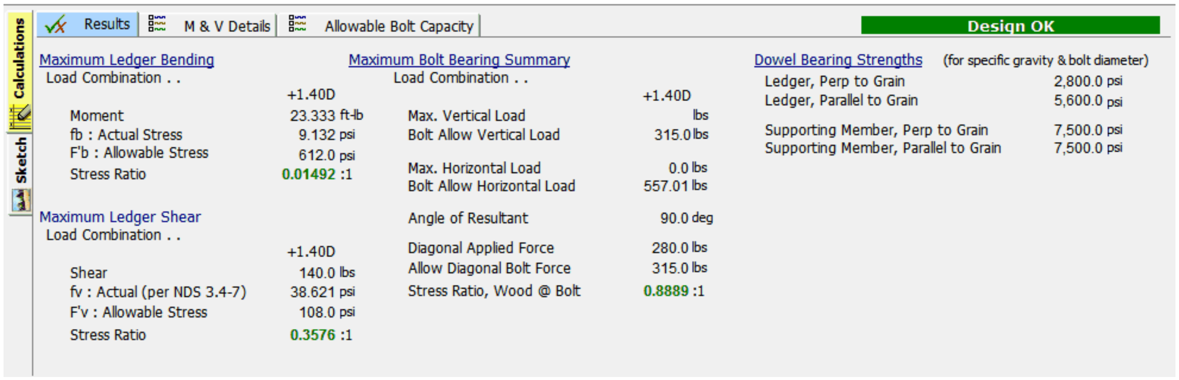
Results tab presents a summary of the design for the current member by reporting the following:

Maximum Ledger Bending shows the load combination, applied moment and actual and allowable bending stresses. Note that no slenderness is considered for the flexural design of the ledger.

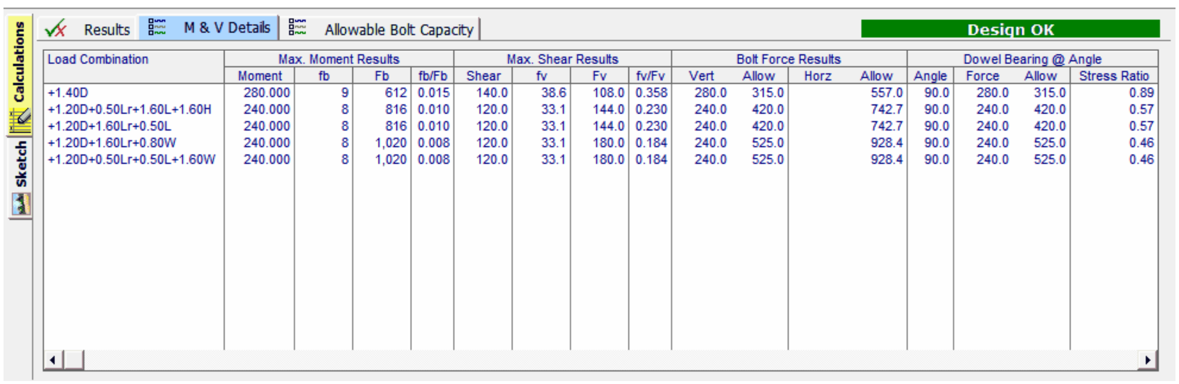
Maximum Ledger Shear shows the load combination, applied shear and actual and allowable shear stresses for vertical loads only. These stresses are taken at the point of bolt support of the ledger and no subtraction of uniform loads within a distance "ledger depth" from that support is considered.

Maximum Bolt Bearing Summary Shows the vertical and horizontal components of force acting on the bolt for the governing condition. "Allow Diagonal Bolt Force" is the result of allowable parallel to grain and perpendicular to grain bolt capacities used for the "Angle of Resultant" in a Hankinson formula calculation.

Dowel Bearing Strengths give the allowable stress and resulting allowable bolt bearing stress considering NDS procedures and strengths and bolt size the user has entered.

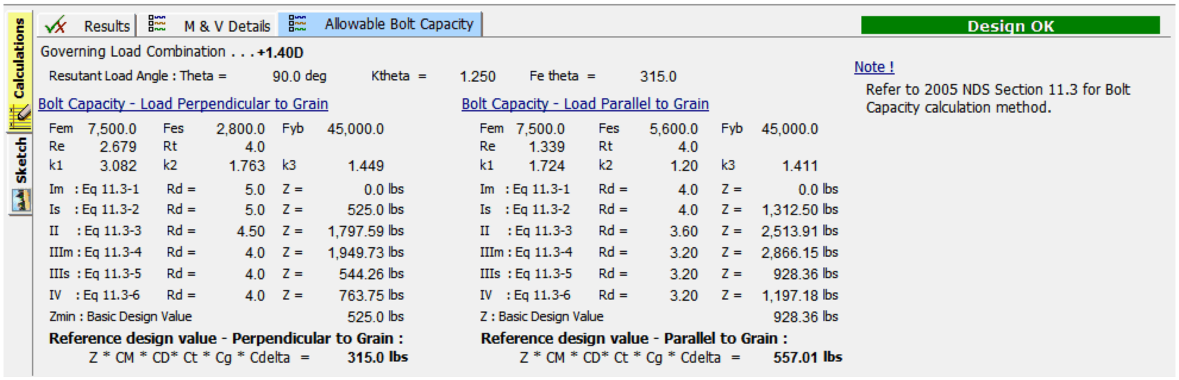


M & V Details tab summarizes the design values according to load combination for moment, shear and bolt forces.

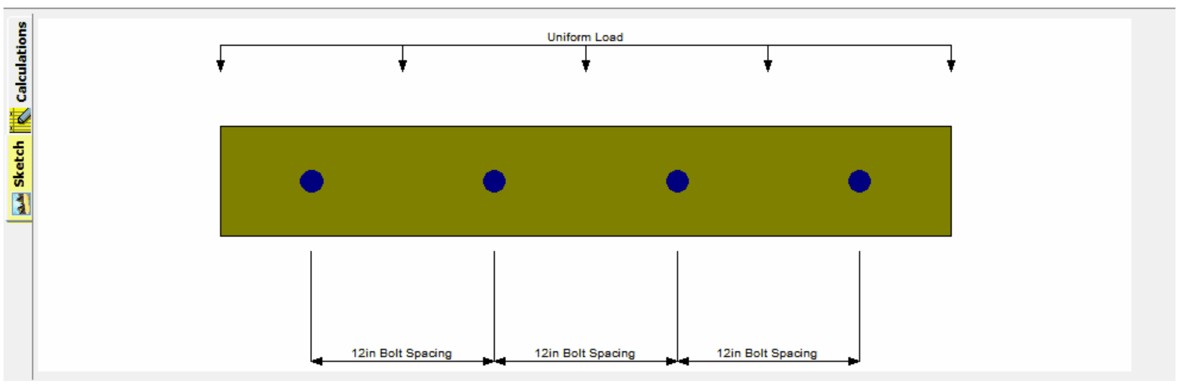


Load Combination	Max. Moment Results				Max. Shear Results				Bolt Force Results				Dowel Bearing @ Angle			
	Moment	fb	Fb	fb/Fb	Shear	fv	Fv	fv/Fv	Vert	Allow	Horz	Allow	Angle	Force	Allow	Stress Ratio
+1.40D	280.000	9	612	0.015	140.0	38.6	108.0	0.358	280.0	315.0		557.0	90.0	280.0	315.0	0.89
+1.20D+0.50Lr+1.60L+1.60H	240.000	8	816	0.010	120.0	33.1	144.0	0.230	240.0	420.0		742.7	90.0	240.0	420.0	0.57
+1.20D+1.60Lr+0.50L	240.000	8	816	0.010	120.0	33.1	144.0	0.230	240.0	420.0		742.7	90.0	240.0	420.0	0.57
+1.20D+1.60Lr+0.80W	240.000	8	1,020	0.008	120.0	33.1	180.0	0.184	240.0	525.0		928.4	90.0	240.0	525.0	0.46
+1.20D+0.50Lr+0.50L+1.60W	240.000	8	1,020	0.008	120.0	33.1	180.0	0.184	240.0	525.0		928.4	90.0	240.0	525.0	0.46

Allowable Bolt Capacity tab provides the details of the bolt capacities according to the chosen NDS calculation method.



The Sketch tab provides a graphic representation of the ledger currently being designed:



10.2.9 Masonry Beam

[Need more? Ask Us a Question](#)

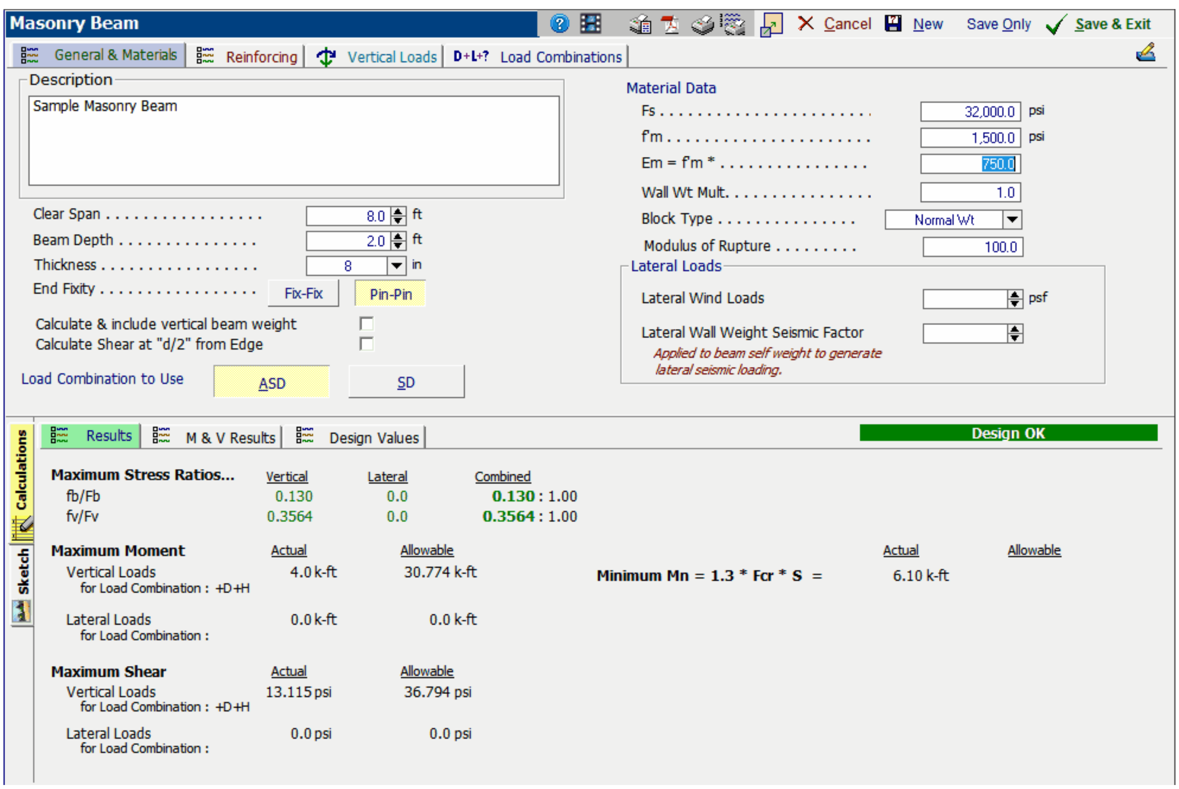
This module provides design analysis for concrete masonry beams and lintels subject to vertical and lateral loads. Beams can have fixed or pinned ends for most typical conditions, and the user can specify rebar sets within the depth of the beam.

Vertical loads can be dead and live uniform and concentrated loads. You can have up to four loads of each type, and the uniform loads can be full or partial length.

The module provides analysis for both seismic and wind loads for each recalculation. You can specify seismic factors that apply to the beam's weight and a wind load.

To allow the module to model different concrete block types, you can specify either lightweight or medium weight block, and additionally enter a self-weight multiplication factor.

For both the vertical and lateral bending and shear directions, the module calculates allowable bending moments and shear stresses. Also, for both directions, actual moments and shears due to all entered loads are calculated. Final results consist of combined stress ratio calculations for all combinations or dead, live, seismic, and wind vertical and lateral moments and shears.



[Unique Features](#)

- This module calculates all vertical and lateral moments and shears and combines them for all possible stress ratios. This provides a thorough evaluation of combined stresses for seismic and wind design.
- The module also provides the ability to modify the material weight, automatically reverse seismic and/or wind loadings, and model both fix-fix and pin-pin end fixity conditions.

Assumptions & Limitations

When the beam's fixity is set to Fixed, both vertical and lateral bending are considered fixed.

General Tab

This tab provides data entry for the lintel dimensions, material properties and lateral loads.

Clear Span and **Lintel Depth** are used to calculate the beam bending and shears.

Thickness is the nominal masonry thickness. The true thickness is determined from internal tables.

End Fixity controls whether the analysis will consider the lintel to be pinned or fixed.

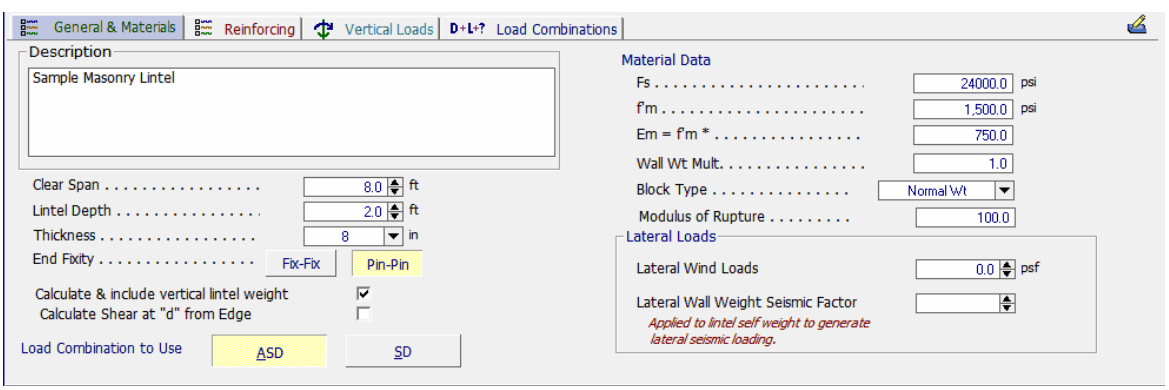
Calculate & Include Vertical Lintel Weight tells the module to calculate the lintel self-weight and add it as a uniform vertical load across the full span.

Material Data defines the allowable stresses for masonry and reinforcing steel.

Wall Weight Multiplier allows the user to factor the lintel self-weight that is pulled from internal tables.

Block Type selects the density of the CMU used for self-weight.

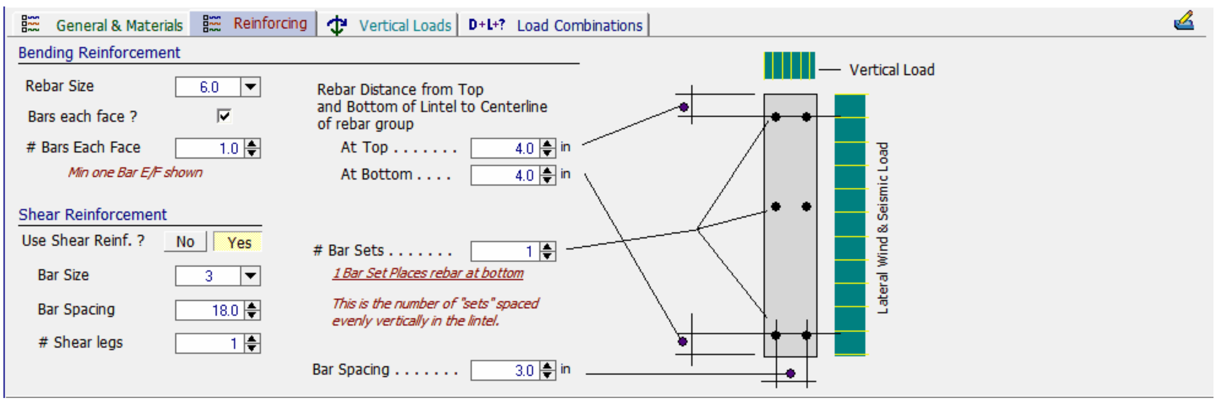
Lateral Loads allows you to specify wind and seismic loads that are applied horizontally, perpendicular to the span of the lintel. Seismic Weight factor is a multiplier applied to the lintel self-weight to create a laterally applied uniform load.



Reinforcing Tab

This tab allows you to specify the longitudinal reinforcing in the beam.

Note: All longitudinal reinforcing bars are assumed to be fully developed. No attempt is made to compare the moment diagram with a longitudinal rebar development diagram, so engineering judgment should be applied in situations where end fixity is assumed and/or where heavy concentrated loads cause significant moments at locations close to the development zone of the provided reinforcement.



Rebar Size

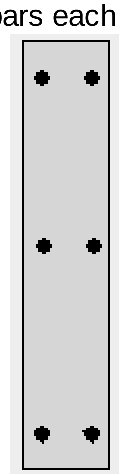
Enter the rebar size used for all longitudinal bar sets to be used in the lintel.

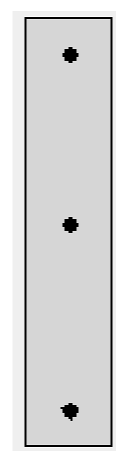
Bars Each Face

Selected implies that the lintel is reinforced with two layers of reinforcing separated by the value specified in the "Bar Spacing" input.

Deselected implies that the lintel is reinforced with one layer of reinforcing located at the middle of the width of the lintel.

The lintel below **DOES** have bars each face: The lintel below does **NOT** have bars each face:

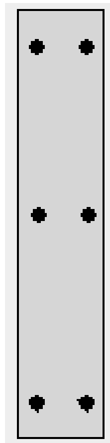


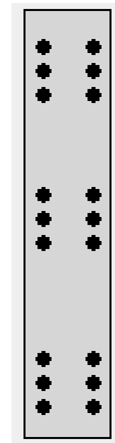


 # Bars Each Location

Enter the number of individual rebars to consider at each reinforced location.

The lintel below has **one** bar at each reinforced location: The lintel below has **three** bars at each reinforced location:





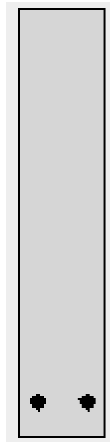
Rebar Distance from Top & Bottom of Lintel to Centerline of rebar group

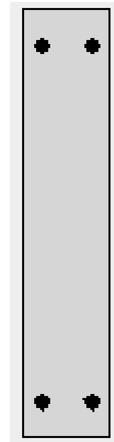
Distance from the top and bottom of the lintel to the center of area of the respective bar set. These distances will be used as "d" for vertical bending strength calculations.

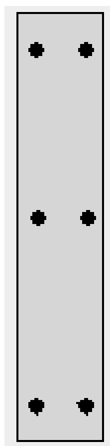
 # Bar Sets

Enter the number of bar sets in the lintel. A value of one indicates that the lintel has bottom reinforcing only.

The lintel below has **one** bar set: The lintel below has **two** bar sets: The lintel below has **three** bar sets:







Bar Spacing

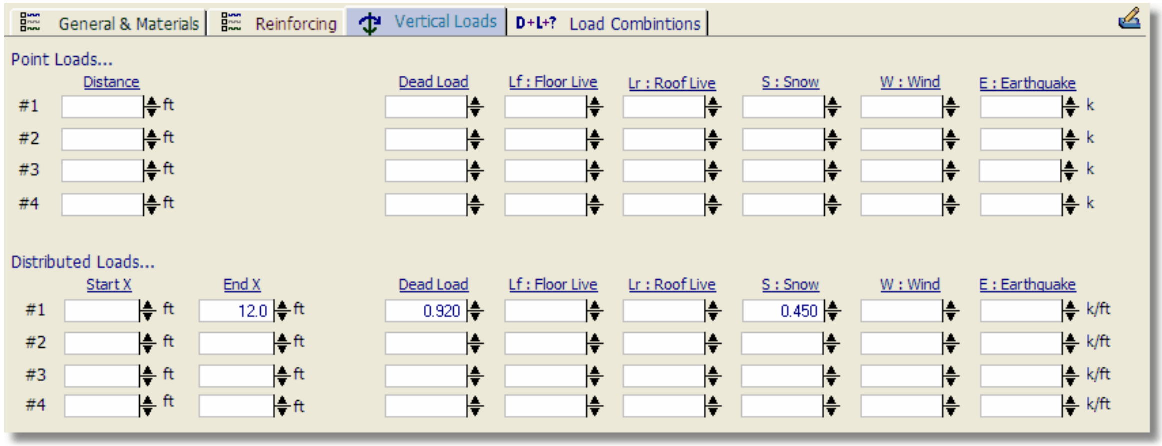
This is the clear distance between the bars on each face in a bar set. It is assumed that the bar set is centered within the width of the lintel. The value of "d" used for lateral bending strength calculations is calculated as: Spacing Between Bars + (Actual Thickness - Spacing Between Bars) / 2

Shear Reinforcement

When you click [**Yes**] you can specify the vertical shear reinforcement used in the lintel. The results will then reflect the allowable & actual shear stress ratios.

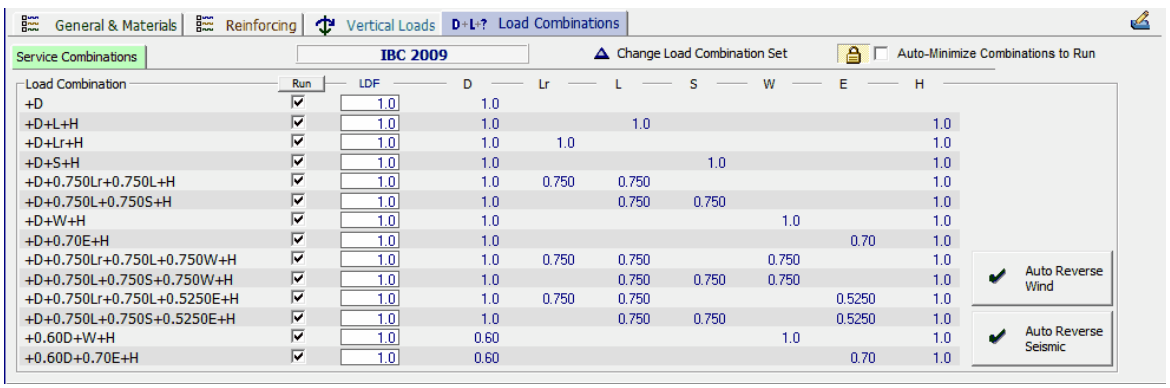
Vertical Load Tab

This tab allows you to specify point loads and distributed loads applied to the lintel.



Load Combinations Tab

This is the typical load combination tab used throughout the **SEL**. "LDF" is a Load Duration Factor that will be applied to allowable stresses and is only available for the ASD design method.

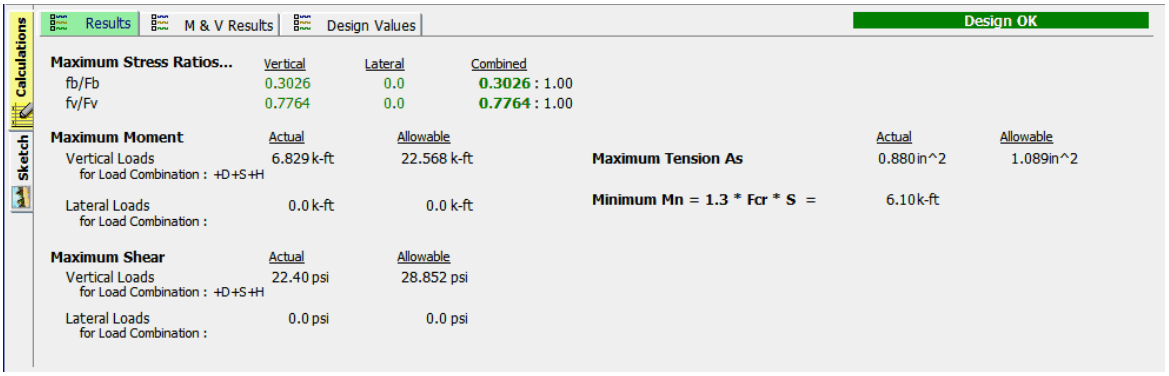


Output & Graphics Tabs

Results Tab

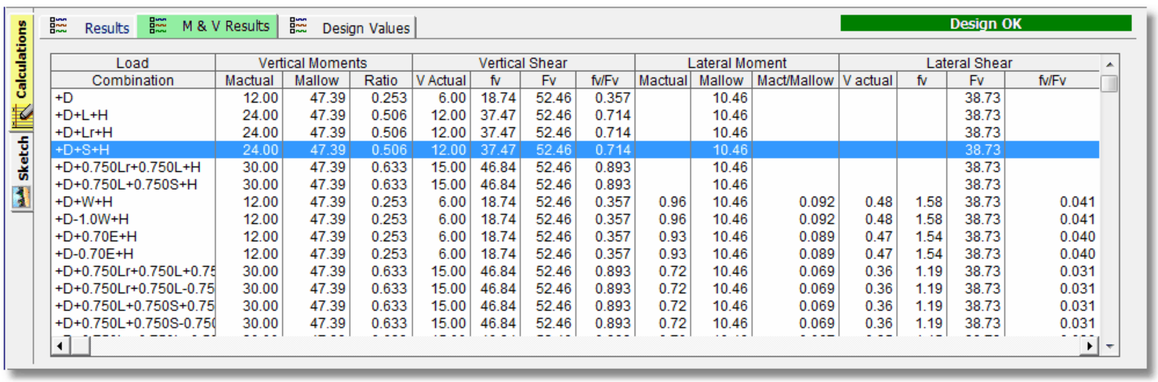
This tab summarizes the calculated moments, shears and combined stress ratios for the lintel.

Ratios for bending and shear are provided for both vertical and lateral load applications. There are no calculations for combined vertical & lateral stresses.



M & V Results Tab

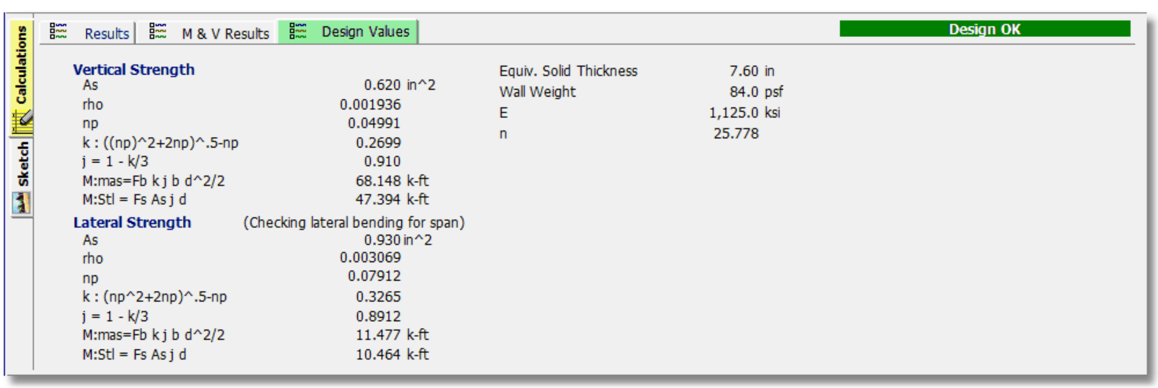
This tab provides full details for all actual and allowable stresses for all load combinations.



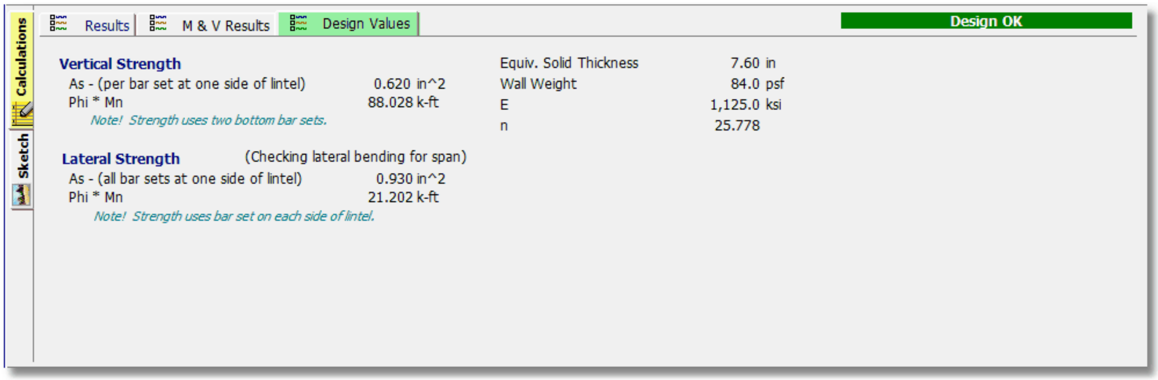
Load	Vertical Moments			Vertical Shear				Lateral Moment			Lateral Shear			
Combination	Mactual	Mallow	Ratio	V Actual	fv	Fv	fv/Fv	Mactual	Mallow	Mact/Mallow	V actual	fv	Fv	fv/Fv
+D	12.00	47.39	0.253	6.00	18.74	52.46	0.357		10.46				38.73	
+D+L+H	24.00	47.39	0.506	12.00	37.47	52.46	0.714		10.46				38.73	
+D+Lr+H	24.00	47.39	0.506	12.00	37.47	52.46	0.714		10.46				38.73	
+D+S+H	24.00	47.39	0.506	12.00	37.47	52.46	0.714		10.46				38.73	
+D+0.750Lr+0.750L+H	30.00	47.39	0.633	15.00	46.84	52.46	0.893		10.46				38.73	
+D+0.750L+0.750S+H	30.00	47.39	0.633	15.00	46.84	52.46	0.893		10.46				38.73	
+D+W+H	12.00	47.39	0.253	6.00	18.74	52.46	0.357	0.96	10.46	0.092	0.48	1.58	38.73	0.041
+D-1.0W+H	12.00	47.39	0.253	6.00	18.74	52.46	0.357	0.96	10.46	0.092	0.48	1.58	38.73	0.041
+D+0.70E+H	12.00	47.39	0.253	6.00	18.74	52.46	0.357	0.93	10.46	0.089	0.47	1.54	38.73	0.040
+D-0.70E+H	12.00	47.39	0.253	6.00	18.74	52.46	0.357	0.93	10.46	0.089	0.47	1.54	38.73	0.040
+D+0.750Lr+0.750L+0.75	30.00	47.39	0.633	15.00	46.84	52.46	0.893	0.72	10.46	0.069	0.36	1.19	38.73	0.031
+D+0.750Lr+0.750L-0.75	30.00	47.39	0.633	15.00	46.84	52.46	0.893	0.72	10.46	0.069	0.36	1.19	38.73	0.031
+D+0.750L+0.750S+0.75	30.00	47.39	0.633	15.00	46.84	52.46	0.893	0.72	10.46	0.069	0.36	1.19	38.73	0.031
+D+0.750L+0.750S-0.75	30.00	47.39	0.633	15.00	46.84	52.46	0.893	0.72	10.46	0.069	0.36	1.19	38.73	0.031

Design Values Tab

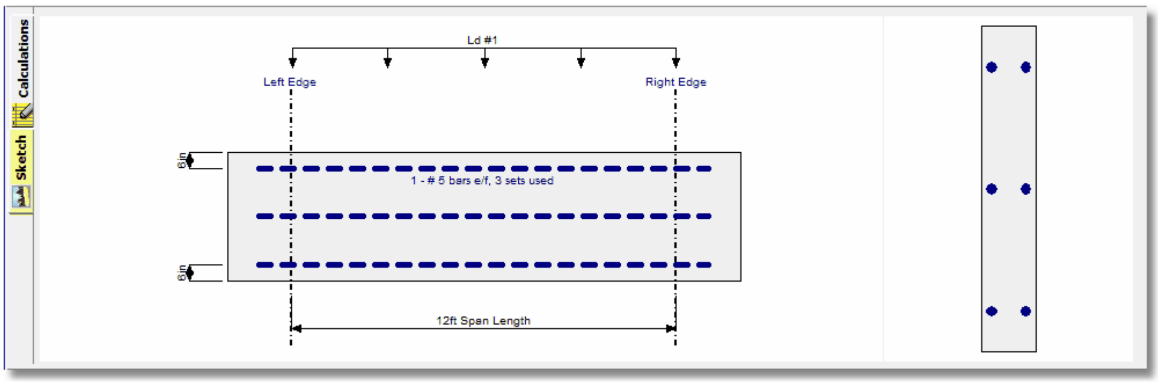
This tab summarizes allowable stress calculations. When ASD is used, the tab looks like this:



And when LRFD is used, the tab looks like this:


Calculations
Sketch
Results
M & V Results
Design Values
Design OK
Vertical Strength
As - (per bar set at one side of lintel) 0.620 in^2
Phi * Mn 88.028 k-ft
Note! Strength uses two bottom bar sets.
Lateral Strength (Checking lateral bending for span)
As - (all bar sets at one side of lintel) 0.930 in^2
Phi * Mn 21.202 k-ft
Note! Strength uses bar set on each side of lintel.
Equiv. Solid Thickness 7.60 in
Wall Weight 84.0 psf
E 1,125.0 ksi
n 25.778

Sketch Tab

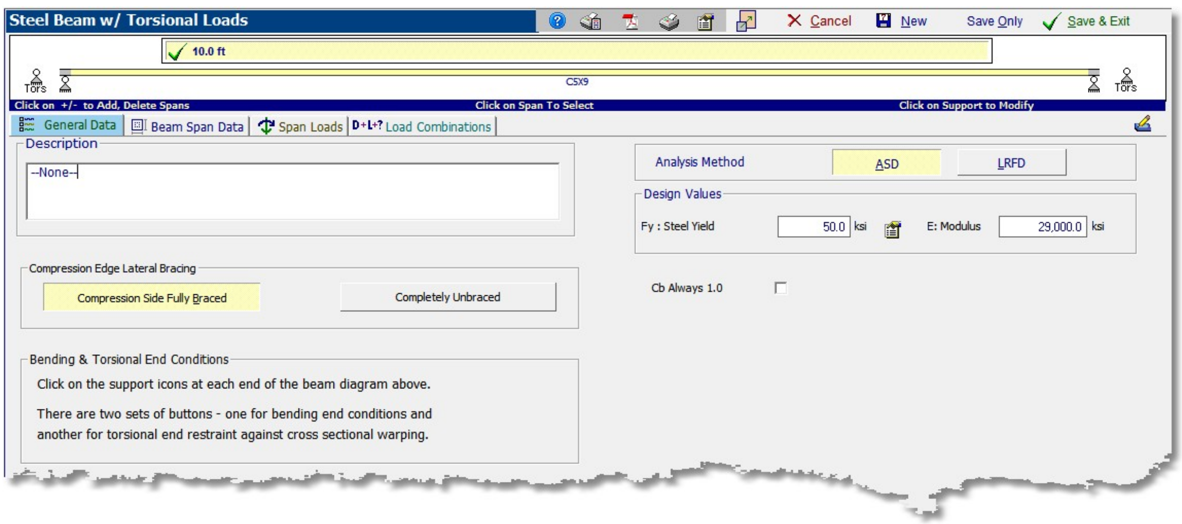

Calculations
Sketch
Ld #1
Left Edge
Right Edge
1 - # 5 bars e/f, 3 sets used
12ft Span Length

10.2.10 Steel Beam with Torsional Loads

The Steel Beam with Torsional Loads module offers the ability to analyze and design a single-span steel beam for applied loads that create shear, bending, and torsion. It can be a useful tool for situations where beams have concentrated or distributed loads that are applied eccentrically, or where beams are subjected to torsional moments.

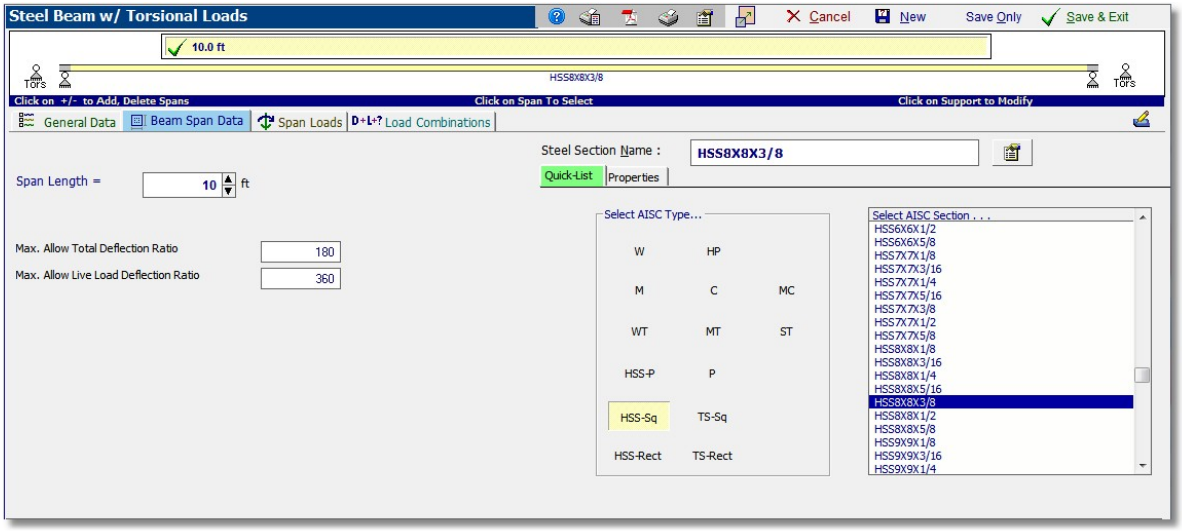
General Data tab:

The General Data tab allows you to set the span length, and support conditions in much the same way that this information is provided in the other beam modules. Refer to the Beams topic for additional explanation, but remember that this particular module is limited to single-span conditions. In addition to these pieces of data, the General Data tab also provides input fields for the compression edge lateral bracing condition, the design method, material properties, and an option to force the Cb value to 1.0, as shown below:



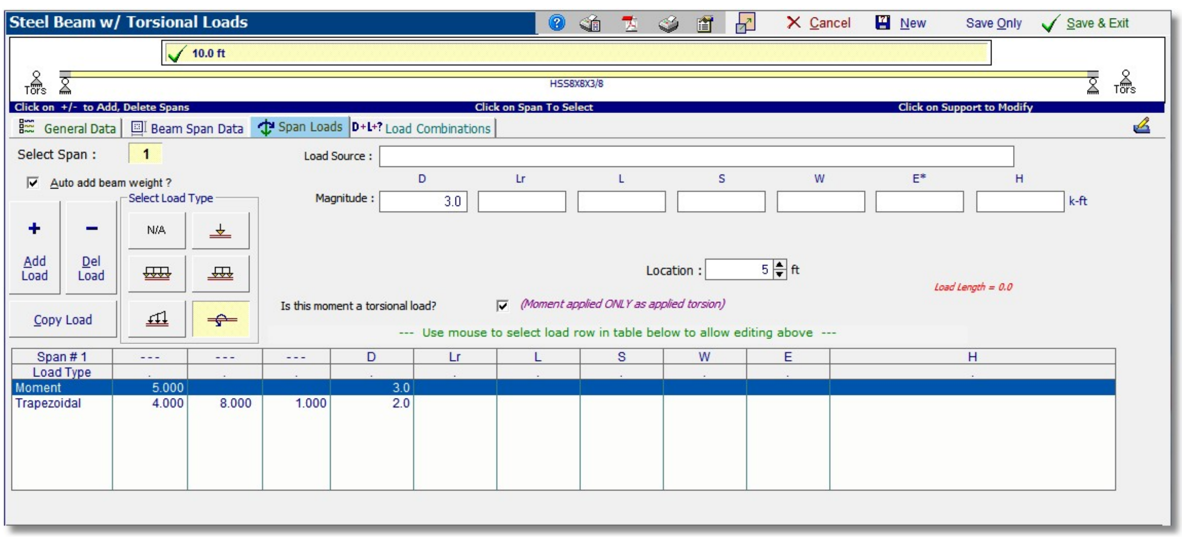
Beam Span Data tab:

The Beam Span Data tab is used to specify the span length, the allowable deflection ratios, and to select the steel section under consideration.



Span Loads tab:

The Span Loads tab allows you to specify loads on one span at a time. The behavior of the tools on this tab is very similar to the tools described for use in the other beam modules, except that it introduces the ability to specify load eccentricities and the ability to indicate that concentrated moments should be considered as torsional moments. Refer to the Beams topic for additional explanation.



Load Combinations tab:

The Load Combinations tab provides a view of the load combinations that will be analyzed. It also offers the ability to:

- Select a different set of load combinations,
- Modify the values used as load factors, and
- Turn certain combinations on and off.

Refer to the Beams topic for additional explanation.

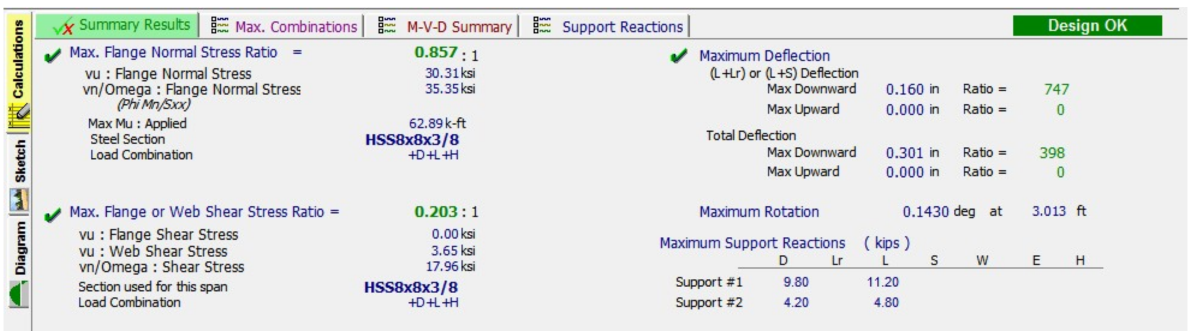
The Steel Beam with Torsional Loads module offers output options that are analogous to the output options provided by the other beam modules, with the exception that the results include torsional design considerations.

The lower half of the screen is dedicated to the display of results. The vertical strip of tabs on the left side of the display allow you to choose between Calculations, Sketch, and Diagram as explained below:

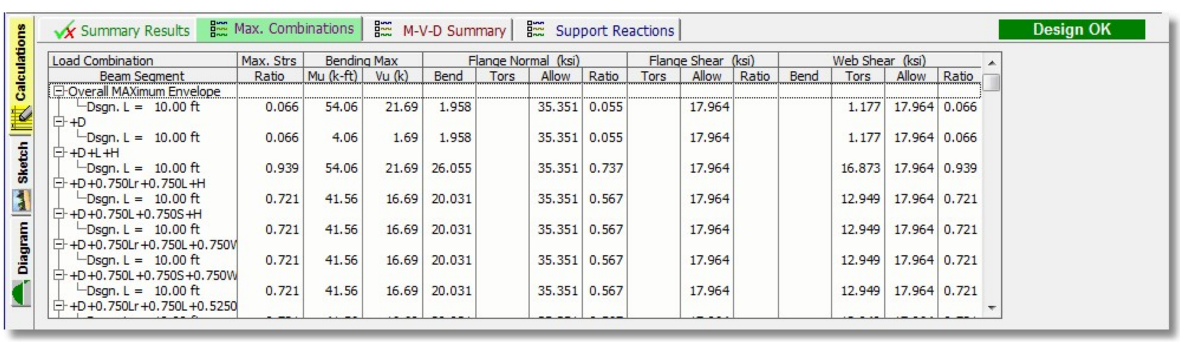
Calculations:

The Calculations tab offers four sub-tabs:

Summary Results: Displays extreme flange normal stress, extreme flange and web shear stress, extreme deflections, extreme rotations, and extreme reactions.

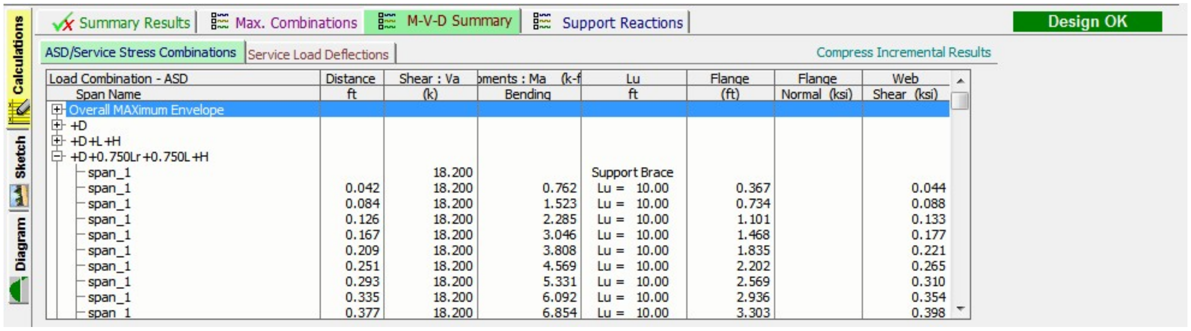


Maximum Combinations: Displays maximum stress ratio, extreme moments and shears, flange normal stresses due to bending and due to torsion, flange shear stress due to torsion, web shear stress due to bending and due to torsion, for all load combinations.



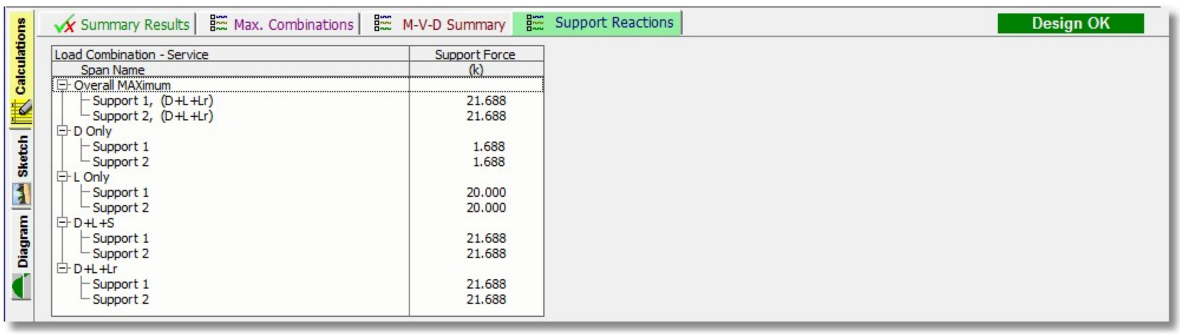
Load Combination	Max. Strs	Bending Max		Flange Normal (ksi)				Flange Shear (ksi)			Web Shear (ksi)			
Beam Segment	Ratio	Mu (k-ft)	Vu (k)	Bend	Tors	Allow	Ratio	Tors	Allow	Ratio	Bend	Tors	Allow	Ratio
Overall MAXimum Envelope														
Dsgn. L = 10.00 ft	0.066	54.06	21.69	1.958		35.351	0.055		17.964			1.177	17.964	0.066
+D														
Dsgn. L = 10.00 ft	0.066	4.06	1.69	1.958		35.351	0.055		17.964			1.177	17.964	0.066
+D+L+H														
Dsgn. L = 10.00 ft	0.939	54.06	21.69	26.055		35.351	0.737		17.964			16.873	17.964	0.939
+D+0.750Lr+0.750L+H														
Dsgn. L = 10.00 ft	0.721	41.56	16.69	20.031		35.351	0.567		17.964			12.949	17.964	0.721
+D+0.750L+0.750S+H														
Dsgn. L = 10.00 ft	0.721	41.56	16.69	20.031		35.351	0.567		17.964			12.949	17.964	0.721
+D+0.750Lr+0.750L+0.750W														
Dsgn. L = 10.00 ft	0.721	41.56	16.69	20.031		35.351	0.567		17.964			12.949	17.964	0.721
+D+0.750L+0.750S+0.750W														
Dsgn. L = 10.00 ft	0.721	41.56	16.69	20.031		35.351	0.567		17.964			12.949	17.964	0.721
+D+0.750Lr+0.750L+0.5250														

M-V-D: Summary: Displays moment, shear, unbraced length, flange normal stress, flange shear stress, web shear stress, and deflections at small increments along all spans. Moment, shear, and stresses are displayed for all load combinations. Deflection is displayed for service load combinations only.



Load Combination - ASD Span Name	Distance ft	Shear : Va (k)	oments : Ma (k-f Bending	Lu ft	Flange (ft)	Flange Normal (ksi)	Web Shear (ksi)
Overall MAXimum Envelope							
+D							
+D+L+H							
+D+0.750Lr+0.750L+H							
span_1		18.200		Support Brace			
span_1	0.042	18.200	0.762	Lu = 10.00	0.367		0.044
span_1	0.084	18.200	1.523	Lu = 10.00	0.734		0.088
span_1	0.126	18.200	2.285	Lu = 10.00	1.101		0.133
span_1	0.167	18.200	3.046	Lu = 10.00	1.468		0.177
span_1	0.209	18.200	3.808	Lu = 10.00	1.835		0.221
span_1	0.251	18.200	4.569	Lu = 10.00	2.202		0.265
span_1	0.293	18.200	5.331	Lu = 10.00	2.569		0.310
span_1	0.335	18.200	6.092	Lu = 10.00	2.936		0.354
span_1	0.377	18.200	6.854	Lu = 10.00	3.303		0.398

Support Reactions: Displays support reactions for all supports, for all load combinations.



Load Combination - Service Span Name	Support Force (k)
Overall MAXimum	
Support 1, (D+L+Lr)	21.688
Support 2, (D+L+Lr)	21.688
D Only	
Support 1	1.688
Support 2	1.688
L Only	
Support 1	20.000
Support 2	20.000
D+L+S	
Support 1	21.688
Support 2	21.688
D+L+Lr	
Support 1	21.688
Support 2	21.688

Sketch:

Displays a sketch of the beam, indicating span lengths, support conditions, and applied loads.

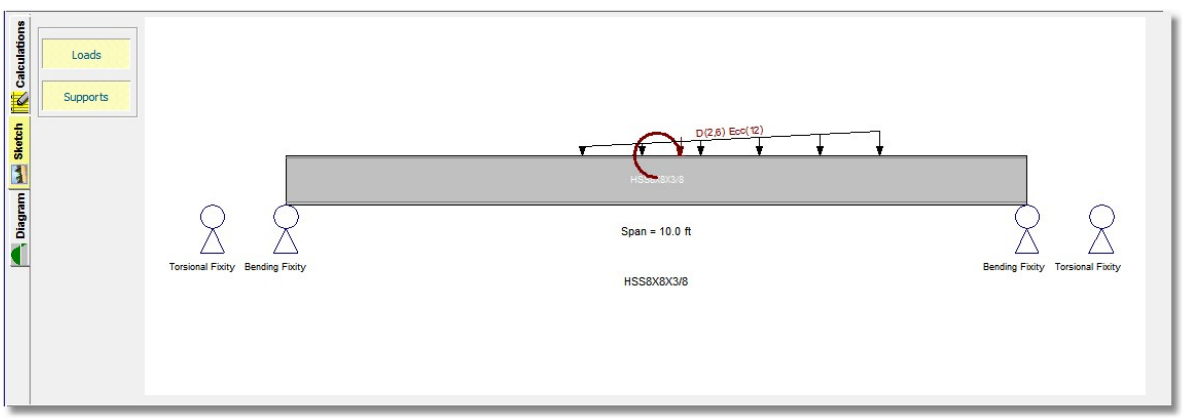
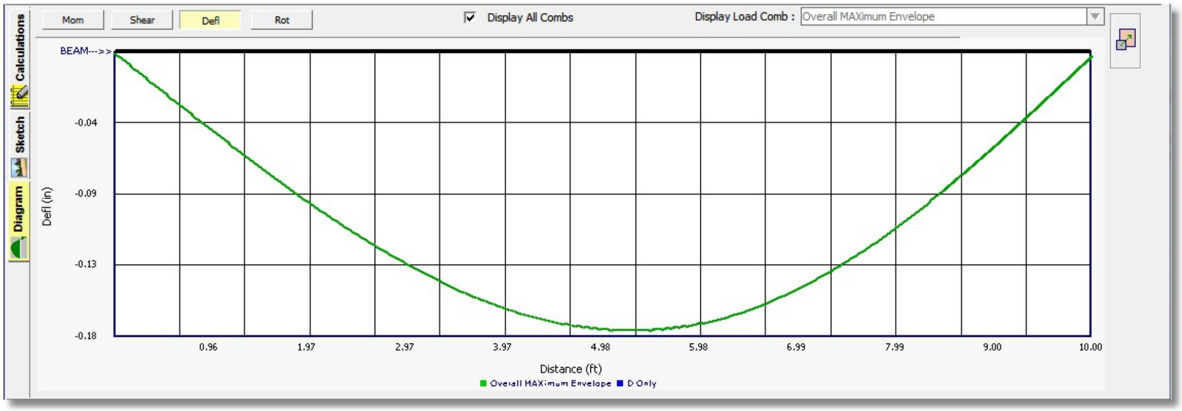


Diagram:

Displays a graphic depiction of the beam with superimposed graphs of Moment, Shear, Deflection or Rotation for a selected load combination, or for an envelope of all load combinations.



Mom
Shear
Defl
Rot
Display All Combs
Display Load Comb :
Overall MAXimum Envelope
Calculations
Sketch
Diagram
BEAM--->>
Defl (in)
-0.04
-0.09
-0.13
-0.18
0.96
1.97
2.97
3.97
4.98
5.98
6.99
7.99
9.00
10.00
Distance (ft)
Overall MAXimum Envelope
D Only

10.3 Columns

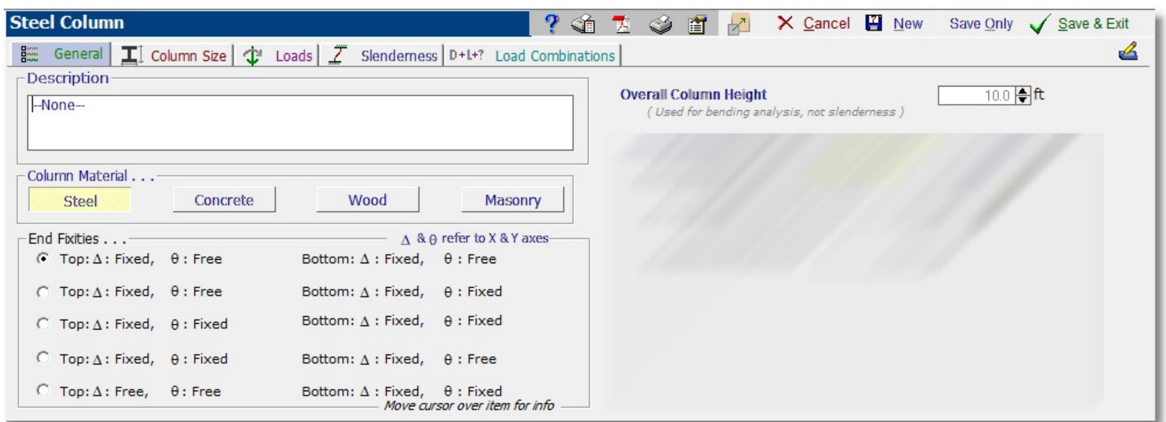
Structural Engineering Library has a single basic column design module that supports Wood, Steel, Concrete and Masonry material types.

There are some portions of the graphical user interface that remain consistent for all materials, and this topic will focus on those items. For detailed information about each of the material-specific calculation modules, please review the respective topics below.

Note: The Column modules are not intended for the design of tension members. Column modules should not be applied in situations where the member experiences net tension.

General Information

The screen capture below shows the portions of the General tab that remain constant for all four material types.



You can easily select a different column material by clicking one of the four material buttons. When you do, the program will load the user interface that is specific to the chosen material.

Overall Column Height is the total height of the column and does not have anything to do with slenderness lengths. This length is used for three things:

- to describe the overall height of the column for the purpose of calculating self weight (if specified),
- to locate the topmost point of load application, and
- to perform the bending analysis when lateral loads are applied.

Rotational End Fixities let you specify how the ends of the columns are or are not attached to boundary conditions. Each condition explicitly describes the translational and rotational boundary conditions at both the top and bottom of the column.

Vertical Loads

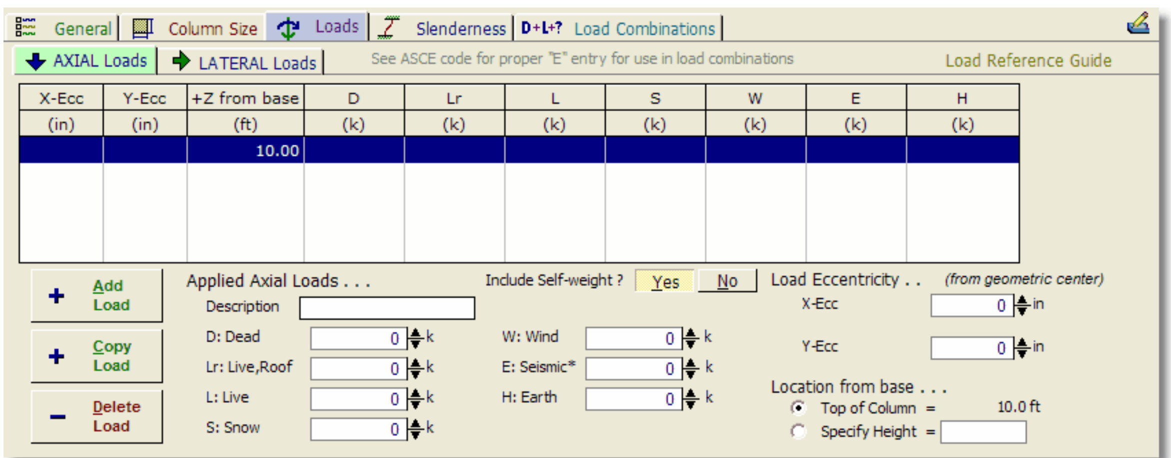
Vertical loads can be applied to any location along the height of the column. You use this tab to build a table of applied loads with the [**Add**], [**Copy**] and [**Delete**] buttons.

Include Self Weight option tells the module to automatically calculate the weight of the column and add it as an additional dead load (which will be factored per "D" load combination factor and applied at the top of the column).

Description lets you describe each load you are applying.

Load Eccentricity has "X" and "Y" eccentricity locations so the loads can create Y-Y and X-X axis moments respectively.

Location from Base is where you specify the vertical location of the axial load with respect to the bottom of the column.



Lateral Loads

On this tab you can specify loads that will be applied along the X or Y axes of the column (non-axial loads).

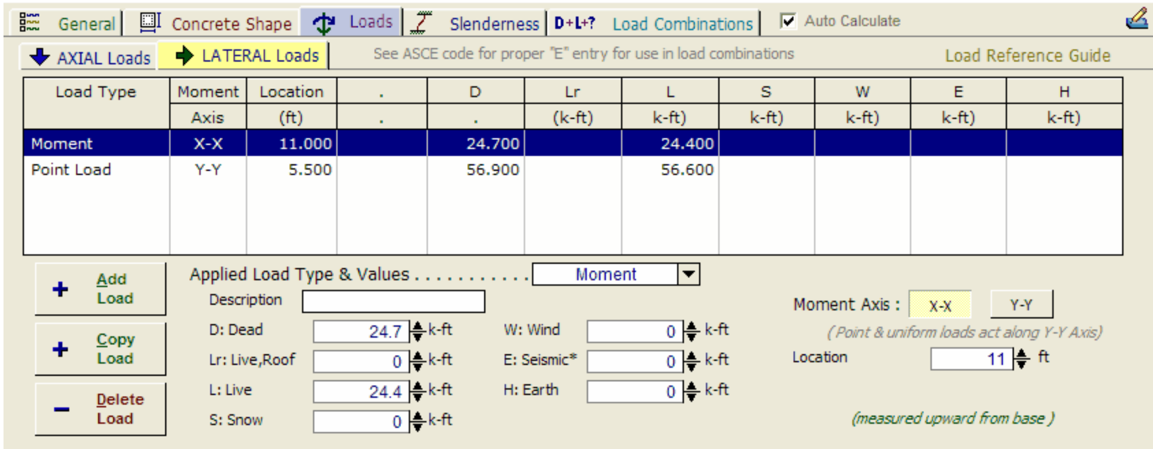
Note! ***Moments applied at fixed ends have no effect on column design, and will be ignored, producing no effect on the column, and will also NOT appear as reactions.***

Applied Load Type & Values has a drop-down list box where you can choose Full Uniform, Partial Uniform, Point Load and Moment load types.

Description lets you describe each load you are applying.

Moment Axis is where you specify about which column axis the load creates its applied moment.

Location from Base is where you specify the vertical location of the lateral load with respect to the bottom of the column.

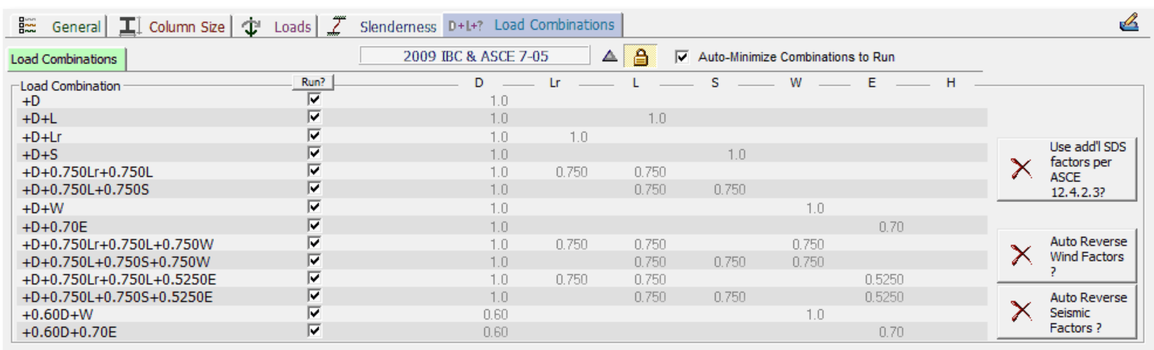


Load Combination Tab

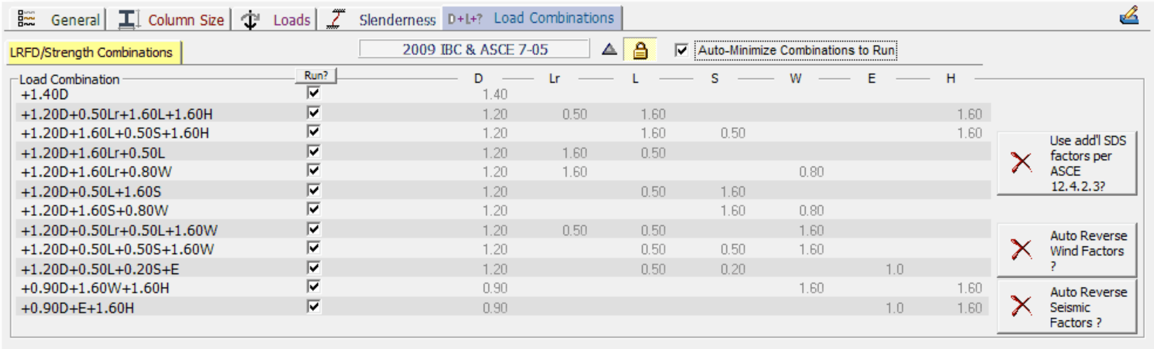
This tab allows you to specify the load combinations to use for the analysis. The tabs change appearance slightly between ASD and LRFD selections. There are also optional load duration factor entries for wood and masonry design.

Please see the screen captures below for variations based on the selected material.

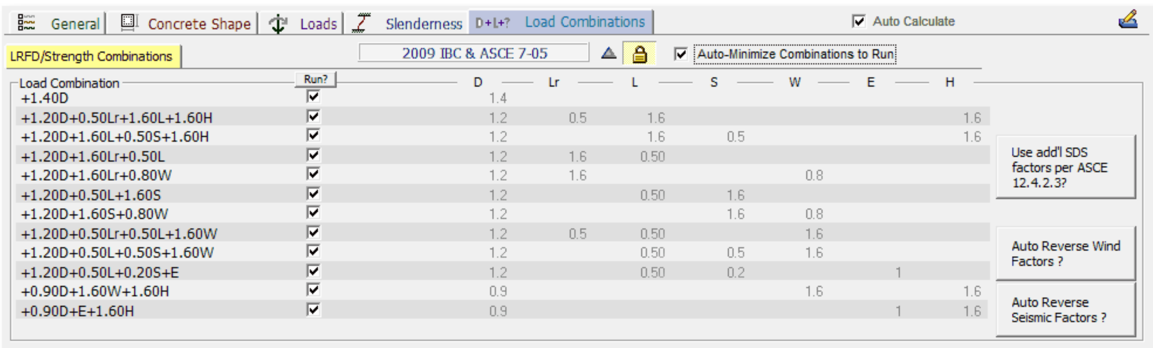
Steel ASD



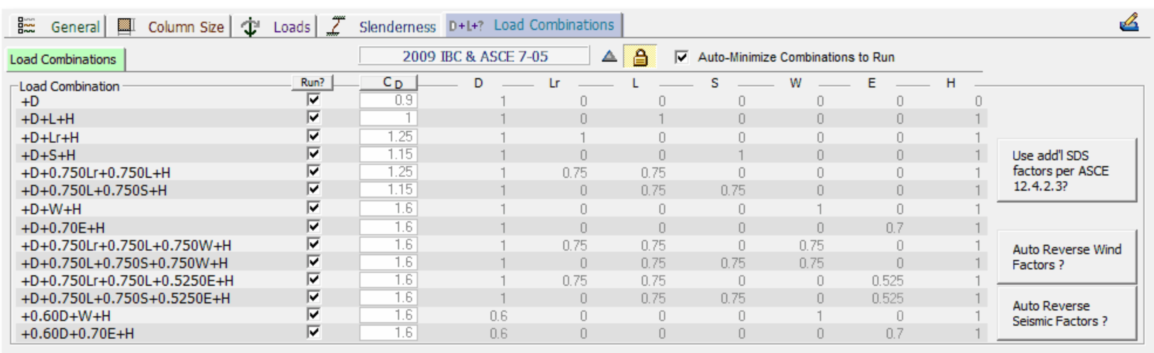
Steel LRFD



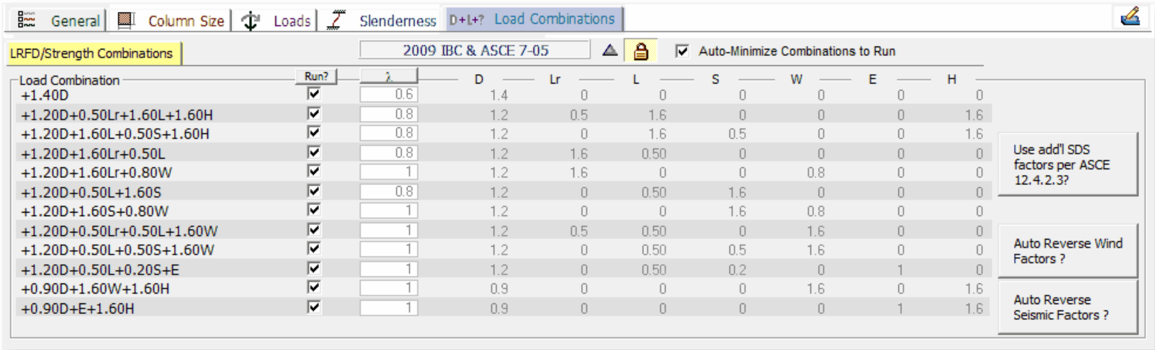
Concrete LRFD



Wood ASD

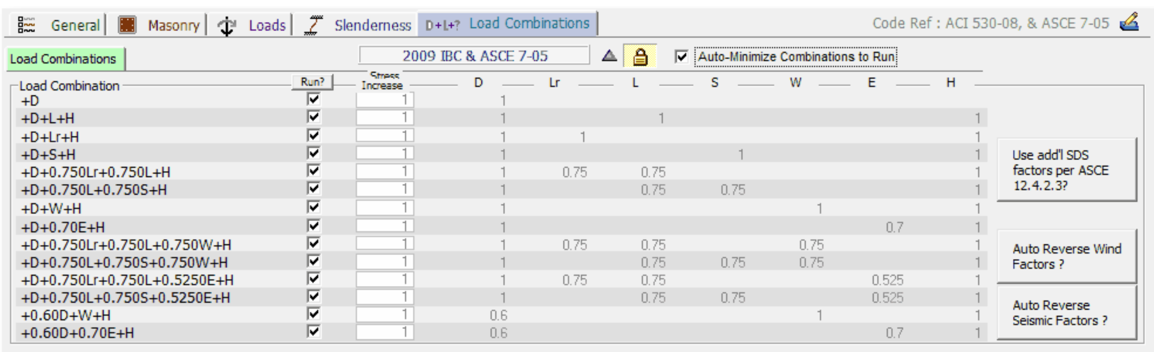


Wood LRFD



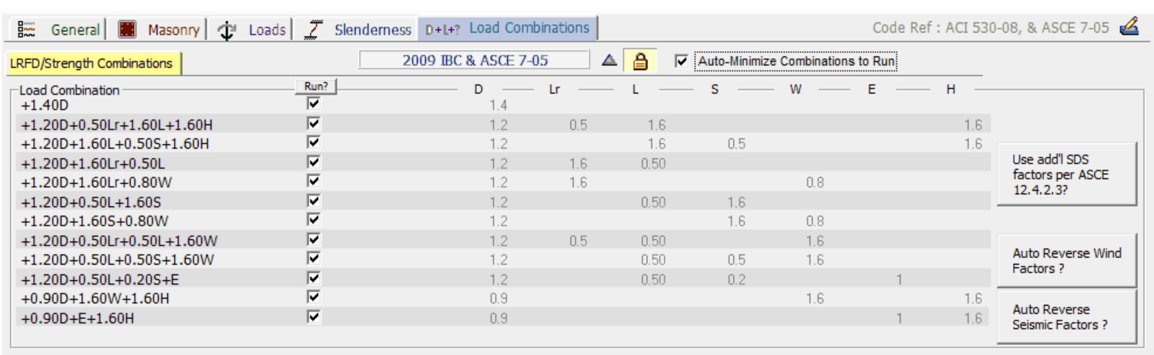
Load Combination	Run?	λ	D	Lr	L	S	W	E	H
+1.40D	✓	0.6	1.4	0	0	0	0	0	0
+1.20D+0.50Lr+1.60L+1.60H	✓	0.8	1.2	0.5	1.6	0	0	0	1.6
+1.20D+1.60L+0.50S+1.60H	✓	0.8	1.2	0	1.6	0.5	0	0	1.6
+1.20D+1.60Lr+0.50L	✓	0.8	1.2	1.6	0.50	0	0	0	0
+1.20D+1.60Lr+0.80W	✓	1	1.2	1.6	0	0	0.8	0	0
+1.20D+0.50L+1.60S	✓	0.8	1.2	0	0.50	1.6	0	0	0
+1.20D+1.60S+0.80W	✓	1	1.2	0	0	1.6	0.8	0	0
+1.20D+0.50Lr+0.50L+1.60W	✓	1	1.2	0.5	0.50	0	1.6	0	0
+1.20D+0.50L+0.50S+1.60W	✓	1	1.2	0	0.50	0.5	1.6	0	0
+1.20D+0.50L+0.20S+E	✓	1	1.2	0	0.50	0.2	0	1	0
+0.90D+1.60W+1.60H	✓	1	0.9	0	0	0	1.6	0	1.6
+0.90D+E+1.60H	✓	1	0.9	0	0	0	0	1	1.6

Masonry ASD



Load Combination	Run?	Stress Increase	D	Lr	L	S	W	E	H
+D	✓	1	1						
+D+L+H	✓	1	1		1				1
+D+Lr+H	✓	1	1	1					1
+D+S+H	✓	1	1			1			1
+D+0.750Lr+0.750L+H	✓	1	1	0.75	0.75				1
+D+0.750L+0.750S+H	✓	1	1		0.75	0.75			1
+D+W+H	✓	1	1				1		1
+D+0.70E+H	✓	1	1					0.7	1
+D+0.750Lr+0.750L+0.750W+H	✓	1	1	0.75	0.75		0.75		1
+D+0.750L+0.750S+0.750W+H	✓	1	1		0.75	0.75	0.75		1
+D+0.750Lr+0.750L+0.5250E+H	✓	1	1	0.75	0.75			0.525	1
+D+0.750L+0.750S+0.5250E+H	✓	1	1		0.75	0.75		0.525	1
+0.60D+W+H	✓	1	0.6				1		1
+0.60D+0.70E+H	✓	1	0.6					0.7	1

Masonry LRFD



Load Combination	Run?	D	Lr	L	S	W	E	H
+1.40D	✓	1.4						
+1.20D+0.50Lr+1.60L+1.60H	✓	1.2	0.5	1.6				1.6
+1.20D+1.60L+0.50S+1.60H	✓	1.2		1.6	0.5			1.6
+1.20D+1.60Lr+0.50L	✓	1.2	1.6	0.50				
+1.20D+1.60Lr+0.80W	✓	1.2	1.6			0.8		
+1.20D+0.50L+1.60S	✓	1.2		0.50	1.6			
+1.20D+1.60S+0.80W	✓	1.2			1.6	0.8		
+1.20D+0.50Lr+0.50L+1.60W	✓	1.2	0.5	0.50		1.6		
+1.20D+0.50L+0.50S+1.60W	✓	1.2		0.50	0.5	1.6		
+1.20D+0.50L+0.20S+E	✓	1.2		0.50	0.2		1	
+0.90D+1.60W+1.60H	✓	0.9				1.6		1.6
+0.90D+E+1.60H	✓	0.9					1	1.6

10.3.1 Column Slenderness

Need more? Ask Us a Question

The Slenderness tab allows you to specify the column bracing, which will be considered in axial capacity calculations. Click here for a video: Column Slenderness

It is **important** to understand that the settings on the Slenderness tab serve double-duty. When the column has applied bending forces, these slenderness settings also define the unbraced compression edge lengths, and therefore they affect the calculation of allowable bending stresses.

In most of the column modules there are two tabs: "X-X" and "Y-Y Axis Column Slenderness". Let's start with the all-important definition of the axis reference for slenderness.

Buckling failure of a column can be thought of as an uncontrolled and excessive deflection in the direction of a particular axis. When defining slenderness, one of the important values is the distance between points that brace the column against movement (or failure) along a particular axis.

The column modules ask you to specify the distance between points of bracing that prevent buckling along the column's local X-X or Y-Y axis.

The **X-X axis** is always parallel to the "**width**" dimension of the column. The **Y-Y axis** is always parallel to the "**depth**" dimension of the column.

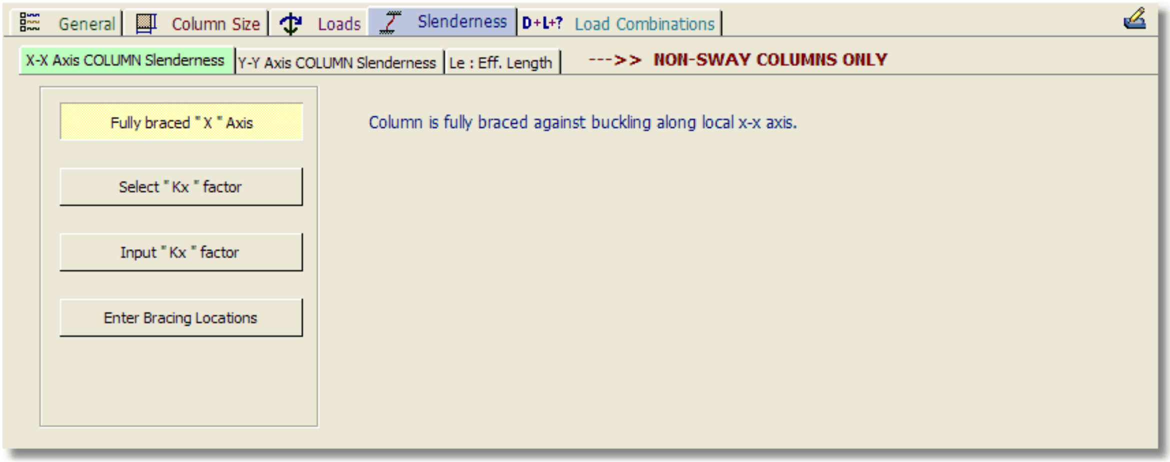
For example, the "X-X" axis of a steel W14 section is parallel to the flanges. The "X-X Axis Column Slenderness" defines the distance between points that prevent the column from buckling along its "X-X" axis.

In Structural Engineering Library, slenderness is entered as the distance between points bracing a member against buckling in the direction of the specified axis.

In the screen capture immediately below, we have selected [**Fully braced "X" Axis**]. This means that the column is fully braced against buckling along its X-X axis, which is parallel to the width dimension. If this was a steel "W" section, then the column is fully braced against buckling parallel to the flanges.

All materials, column fully braced

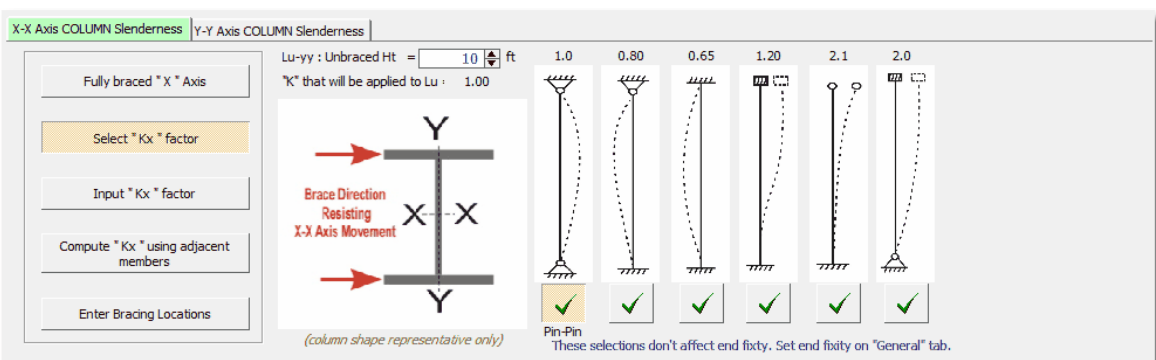
This selection sets the column as fully braced, and no slenderness effects will be evaluated.



All materials, typical simple slenderness specification

This selection allows you to enter the unbraced height to use for the column slenderness calculation. Also available are selections for the typical slenderness multipliers ("K" factors) for various end conditions.

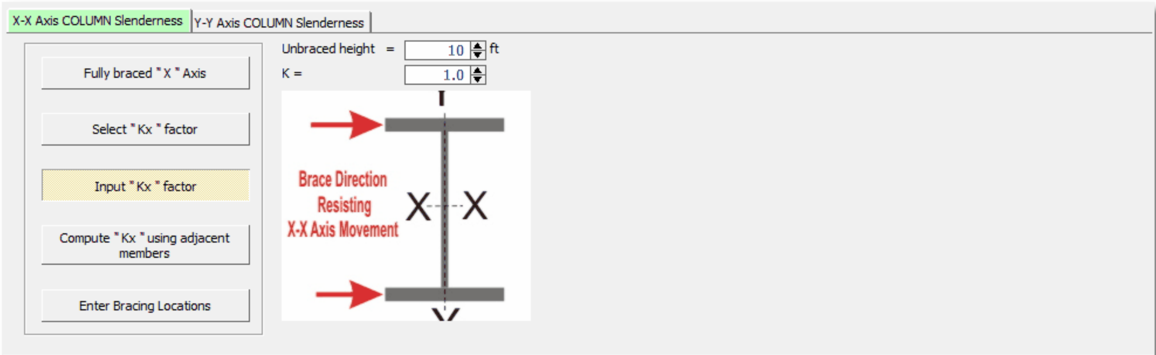
Note! End fixity is specified on a different tab in the column modules. These slenderness factors do not alter the end fixity you specified for the column, nor do they get determined automatically by the end fixities that you specified.



All materials, typical slenderness with user-defined slenderness factor "K"

This selection allows you to enter the unbraced height and the slenderness "K" factor to use for the column slenderness calculation.

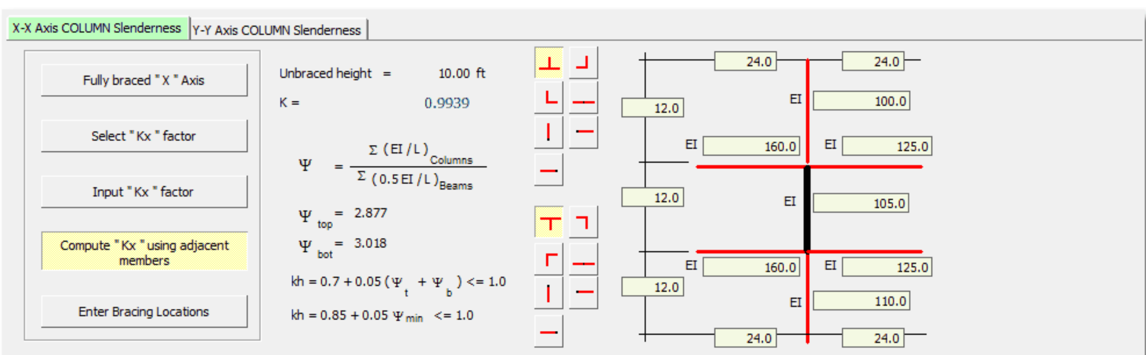
Note! End fixity is specified on a different tab in the column modules. These slenderness factors do not alter the end fixity you specified for the column, nor do they get determined automatically by the end fixities that you specified.



Concrete & Steel Only

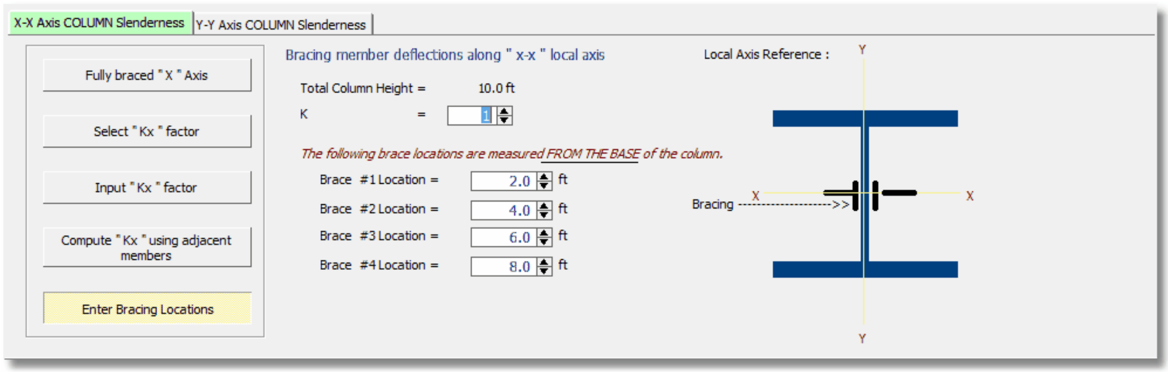
For concrete and steel you will see an option to Compute K using adjacent members. This advanced selection lets you select the framing condition above and below the column, and using the entered lengths and EI values, it will use standard equations for Non-Sway columns to calculate the effective "K" factor.

Keep in mind that this option is used to define the relative stiffnesses of the framing in the plane of buckling that is being considered. For example, when using the X-X Axis Column Slenderness tab, the plane under consideration is the plane in which the column's X-X axis lies.



Wood & Steel Only.....

For wood and steel columns only, there is an option to [**Enter Bracing Location**], which allows you to divide the column up into several segments with different unbraced lengths.


X-X Axis COLUMN Slenderness
Y-Y Axis COLUMN Slenderness
Fully braced " X " Axis
Select " Kx " factor
Input " Kx " factor
Compute " Kx " using adjacent members
Enter Bracing Locations
Bracing member deflections along " x-x " local axis
Total Column Height = 10.0 ft
K =
The following brace locations are measured FROM THE BASE of the column.
Brace #1 Location = 2.0 ft
Brace #2 Location = 4.0 ft
Brace #3 Location = 6.0 ft
Brace #4 Location = 8.0 ft
Local Axis Reference :
Y
X
X
Bracing ------------------->>
Y

10.3.2 Wood Column

[Need more? Ask Us a Question]

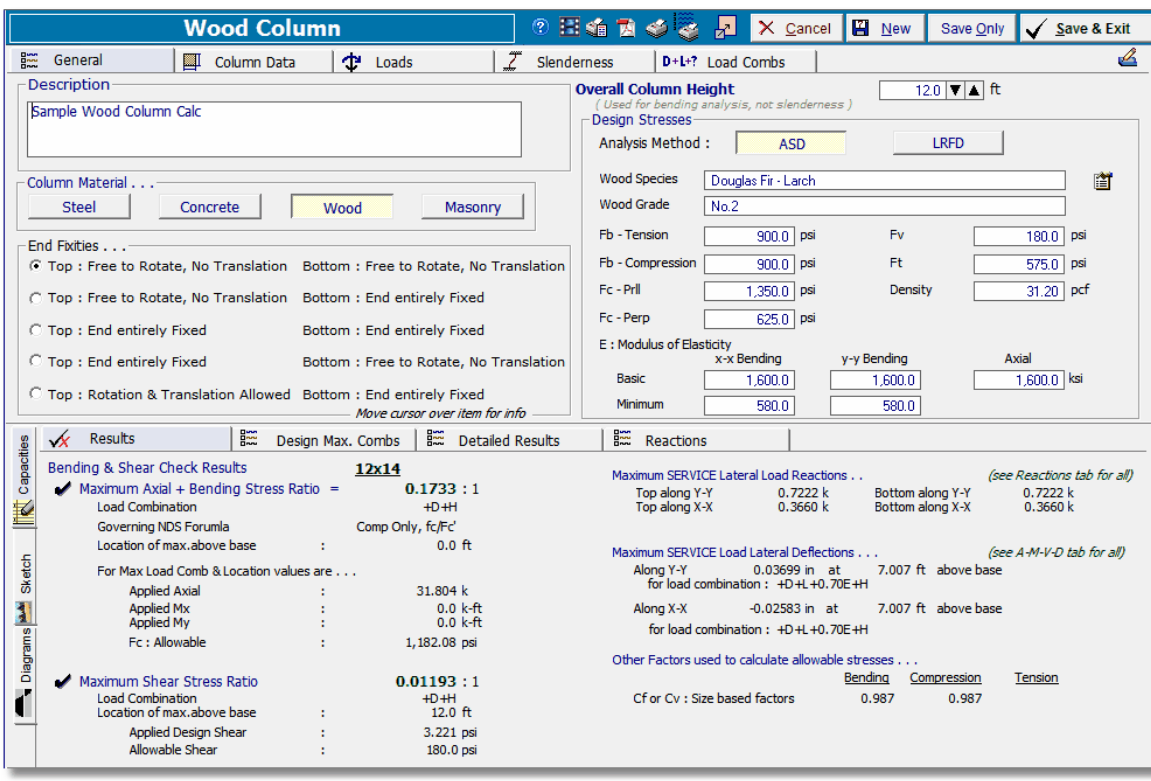
This module designs wood columns that are subject to axial loads and lateral bending loads about both axes. Click here for a video: Wood Column

The user can select ASD or LRFD design methods and has access to a large built-in database of wood sizes and NDS species stress grades. Values of K_F and phi are automatically determined and applied for the LRFD method.

All calculations are according to the NDS code.

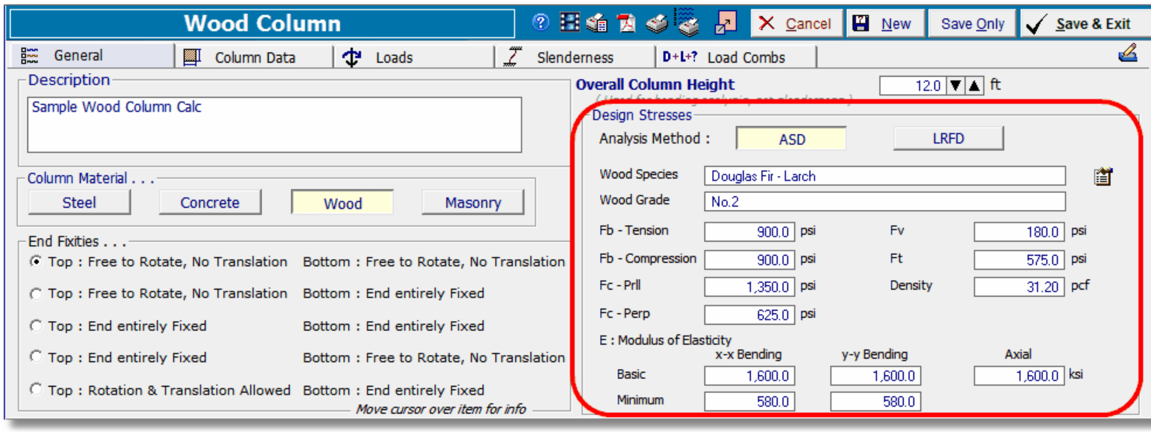
The screen capture below shows the full screen for wood column design. See items below for descriptions of items that are specific to the Wood Column module.

For general description of the module, end fixity, loads, and load combinations click here 326. For slenderness description click here 331.



General Tab

The Design Stresses area bubbled in the screen capture below is unique to the wood column selection. This area enables you to specify the base design values for the wood species and grade of interest.



You can either enter these values manually, or you can click the  button to display the Wood Stress Database.

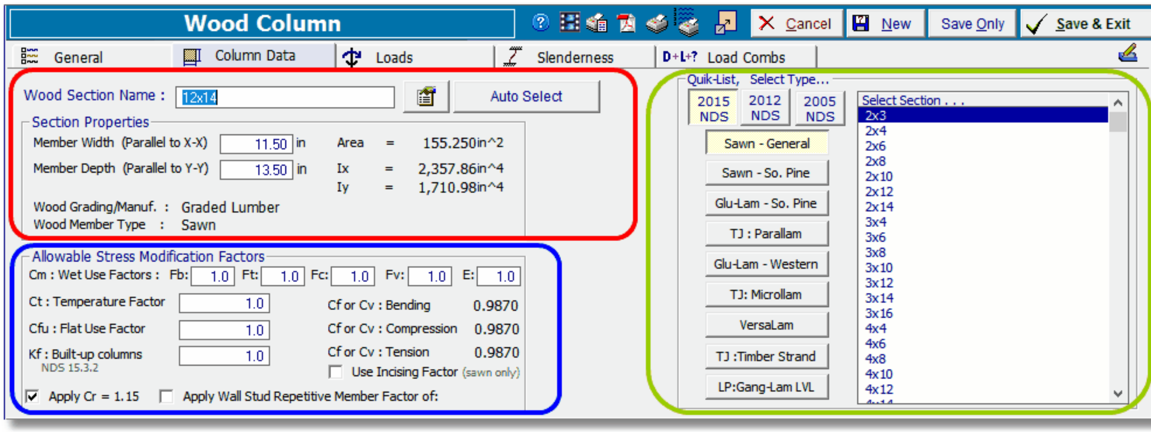
Column Data Tab

All of the information on this tab is unique to a wood column.

The area bubbled in **red** is where you specify the column cross section. Use the  button to display the built-in database of wood sections (solid-sawn, glulam, and manufactured sections are available). You can also enter the values manually.

The area bubbled in **blue** provides allowable stress modification factors that you can specify. Please note that C_F or C_V values are automatically filled in. C_F values are determined from the size and stress grade of the member (No. 1 and Utility grades have different values). C_V values are calculated when a glu-lam section is specified. Note that this section allows for the specification of the Repetitive Member factor. If the Repetitive Member factor option is selected, the program then offers the option to specify a value for the Wall Stud Repetitive Member Factor as defined by the NDS Special Design Provisions for Wind and Seismic (SDPWS). If the option is selected to specify a Wall Stud Repetitive Member Factor, it will only be applied to load combinations that include wind. Be sure to review the appropriate section of SDPWS for requirements on the use of this factor, as well as the values to be used for various sizes of dimension lumber.

The area in **green** provides quick access to the built-in wood section database. Simply click the button of the section type and the list on the right will be populated automatically from the appropriate database. Then just click on a section to have its data loaded into the entries in the **red** area.



Note: The flat use factor will only be applied if it is specified by the user. In addition, it is important to understand how the factor will be applied in situations where built-up columns are designed. The flat use factor is supposed to be applied when bending occurs about the weak axis of the ***individual laminations***. But the program does not actually understand the orientation of individual laminations in a built-up column as of July 2018. So for consistency, the program is set up to look at the ***overall dimensions*** of a built-up column cross section, and then to apply the flat use factor only when considering bending about the overall weak axis. This requires the designer's consideration, because the weak axis of the overall built-up section may or may not correspond to the weak axis of the individual laminations.

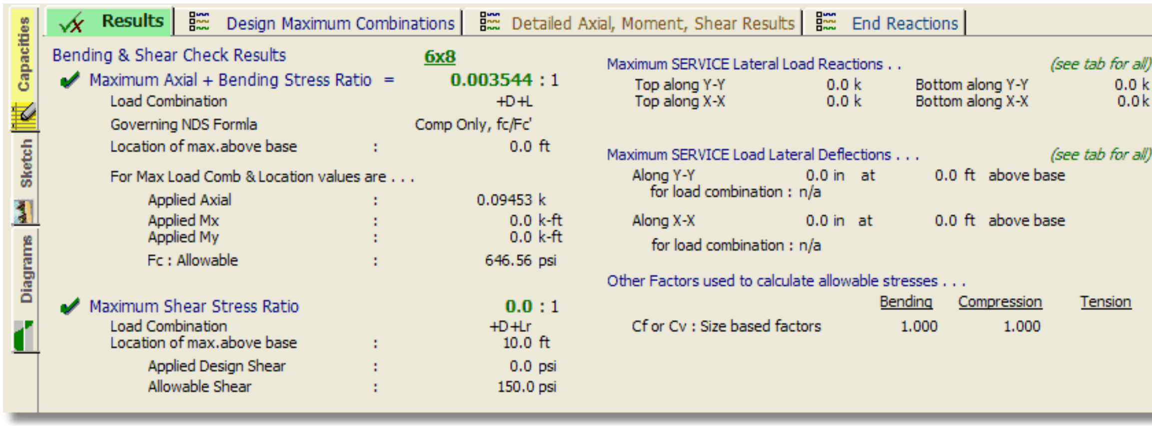
Results Tab

This tab provides a summary of the stress ratios, reactions and deflections for the column.

Max Axial + Bending Stress Ratio is the governing load combination for the column. Listed is the governing load load combination, the NDS formulas that is used and the location of the maximum stress ratio above the base of the column. Please note maximum stress ratio is what is being reported because it governs the design.....not necessarily the highest axial or bending stress.

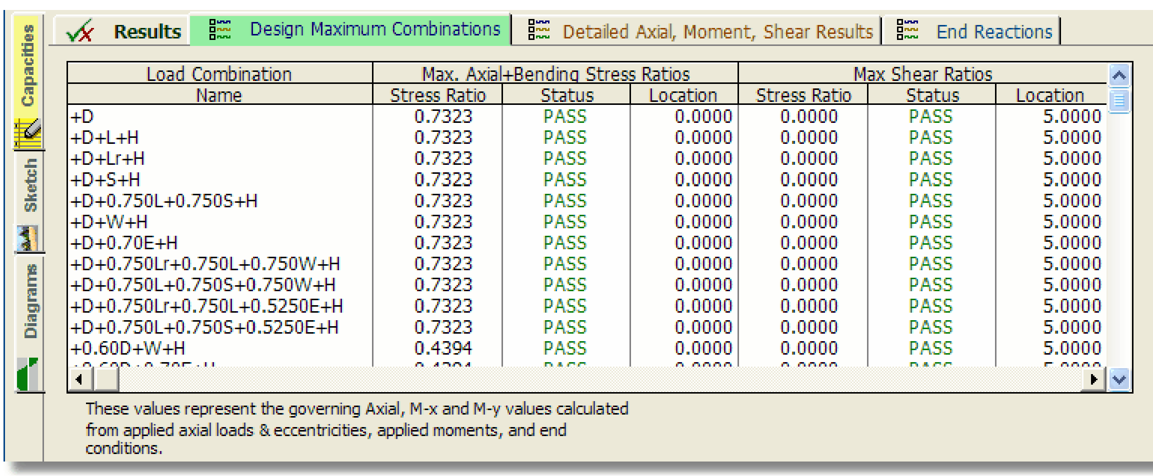
Max Shear Stress Ratio will probably never govern for a column design but is presented here with the governing load combination, location and allowable/actual stress values.

Lateral Load Reactions and **Deflections** are the result of applied lateral loads.



Design Maximum Combinations Tab

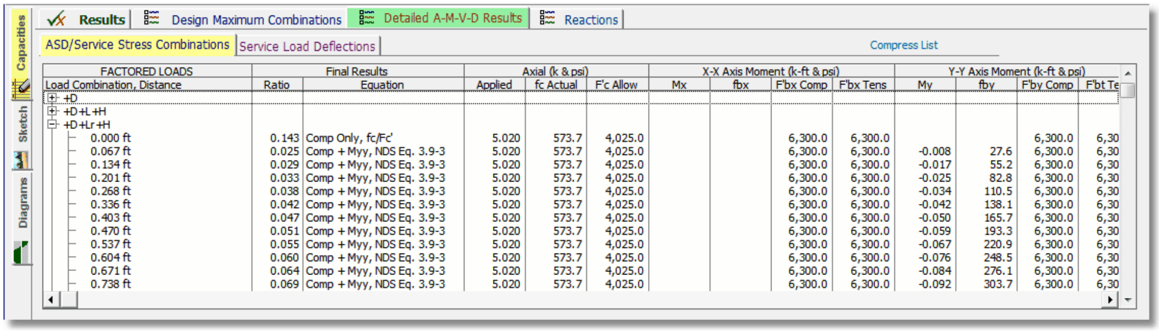
This tab lists the resulting maximum stress ratios for each load combination. This list is created by examining the detailed list (on the next tab) and determing the governing stress ratios for each load combination.



Detailed Results Tab - Stress

This tab lists the detailed results at small increments along the height of the column for each load combination. For consistency, all of the column headings use labels directly from the NDS code.

Note! This list scrolls to the right to display more information. See the successive screen captures below:



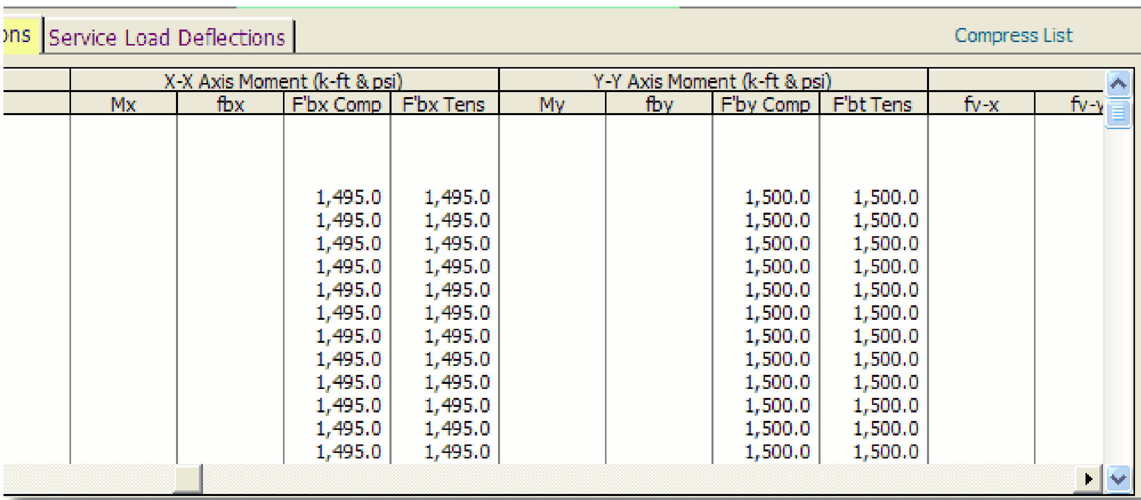
Results | Design Maximum Combinations | Detailed A-M-V-D Results | Reactions

ASD/Service Stress Combinations | Service Load Deflections

Compress List

FACTORED LOADS	Final Results		Axial (k & psi)			X-X Axis Moment (k-ft & psi)				Y-Y Axis Moment (k-ft & psi)			
Load Combination, Distance	Ratio	Equation	Applied	fc Actual	F'c Allow	Mx	fbx	F'bx Comp	F'bx Tens	My	fby	F'by Comp	F'bt Te
+D													
+D+L+H													
+D+Lr+H													
0.000 ft	0.143	Comp Only, fc/Fc'	5.020	573.7	4,025.0			6,300.0	6,300.0			6,300.0	6,30
0.067 ft	0.025	Comp + Myy, NDS Eq. 3.9-3	5.020	573.7	4,025.0			6,300.0	6,300.0	-0.008	27.6	6,300.0	6,30
0.134 ft	0.029	Comp + Myy, NDS Eq. 3.9-3	5.020	573.7	4,025.0			6,300.0	6,300.0	-0.017	55.2	6,300.0	6,30
0.201 ft	0.033	Comp + Myy, NDS Eq. 3.9-3	5.020	573.7	4,025.0			6,300.0	6,300.0	-0.025	82.8	6,300.0	6,30
0.268 ft	0.038	Comp + Myy, NDS Eq. 3.9-3	5.020	573.7	4,025.0			6,300.0	6,300.0	-0.034	110.5	6,300.0	6,30
0.336 ft	0.042	Comp + Myy, NDS Eq. 3.9-3	5.020	573.7	4,025.0			6,300.0	6,300.0	-0.042	138.1	6,300.0	6,30
0.403 ft	0.047	Comp + Myy, NDS Eq. 3.9-3	5.020	573.7	4,025.0			6,300.0	6,300.0	-0.050	165.7	6,300.0	6,30
0.470 ft	0.051	Comp + Myy, NDS Eq. 3.9-3	5.020	573.7	4,025.0			6,300.0	6,300.0	-0.059	193.3	6,300.0	6,30
0.537 ft	0.055	Comp + Myy, NDS Eq. 3.9-3	5.020	573.7	4,025.0			6,300.0	6,300.0	-0.067	220.9	6,300.0	6,30
0.604 ft	0.060	Comp + Myy, NDS Eq. 3.9-3	5.020	573.7	4,025.0			6,300.0	6,300.0	-0.076	248.5	6,300.0	6,30
0.671 ft	0.064	Comp + Myy, NDS Eq. 3.9-3	5.020	573.7	4,025.0			6,300.0	6,300.0	-0.084	276.1	6,300.0	6,30
0.738 ft	0.069	Comp + Myy, NDS Eq. 3.9-3	5.020	573.7	4,025.0			6,300.0	6,300.0	-0.092	303.7	6,300.0	6,30

Scroll a little to the right....

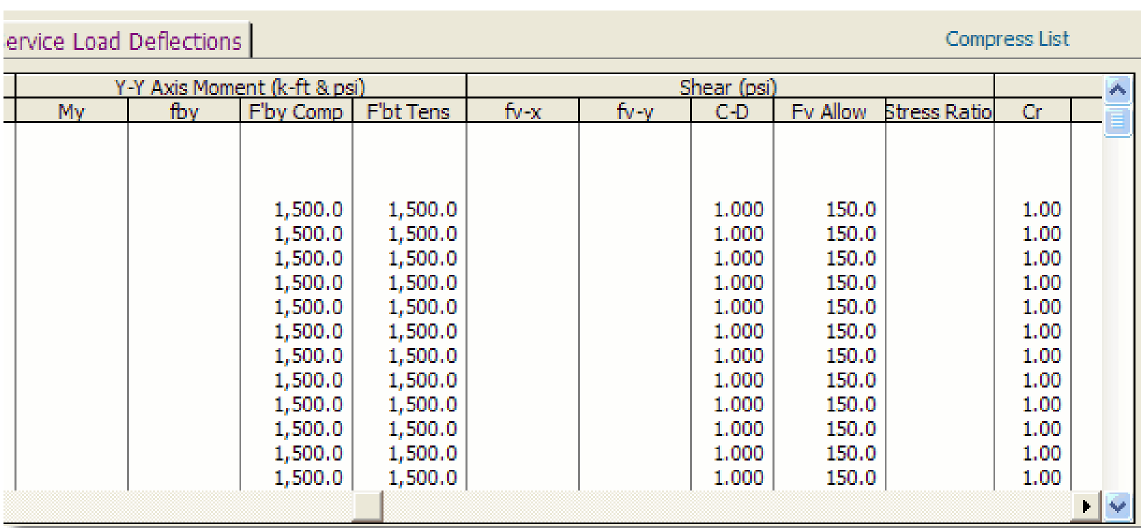


ons | Service Load Deflections

Compress List

X-X Axis Moment (k-ft & psi)				Y-Y Axis Moment (k-ft & psi)					
Mx	fbx	F'bx Comp	F'bx Tens	My	fby	F'by Comp	F'bt Tens	fv-x	fv-y
		1,495.0	1,495.0			1,500.0	1,500.0		
		1,495.0	1,495.0			1,500.0	1,500.0		
		1,495.0	1,495.0			1,500.0	1,500.0		
		1,495.0	1,495.0			1,500.0	1,500.0		
		1,495.0	1,495.0			1,500.0	1,500.0		
		1,495.0	1,495.0			1,500.0	1,500.0		
		1,495.0	1,495.0			1,500.0	1,500.0		
		1,495.0	1,495.0			1,500.0	1,500.0		
		1,495.0	1,495.0			1,500.0	1,500.0		
		1,495.0	1,495.0			1,500.0	1,500.0		
		1,495.0	1,495.0			1,500.0	1,500.0		
		1,495.0	1,495.0			1,500.0	1,500.0		

And a little more....



ervice Load Deflections

Compress List

Y-Y Axis Moment (k-ft & psi)				Shear (psi)					
My	fby	F'by Comp	F'bt Tens	fv-x	fv-y	C-D	Fv Allow	Stress Ratio	Cr
		1,500.0	1,500.0			1.000	150.0		1.00
		1,500.0	1,500.0			1.000	150.0		1.00
		1,500.0	1,500.0			1.000	150.0		1.00
		1,500.0	1,500.0			1.000	150.0		1.00
		1,500.0	1,500.0			1.000	150.0		1.00
		1,500.0	1,500.0			1.000	150.0		1.00
		1,500.0	1,500.0			1.000	150.0		1.00
		1,500.0	1,500.0			1.000	150.0		1.00
		1,500.0	1,500.0			1.000	150.0		1.00
		1,500.0	1,500.0			1.000	150.0		1.00
		1,500.0	1,500.0			1.000	150.0		1.00
		1,500.0	1,500.0			1.000	150.0		1.00

And more still....

ations | Service Load Deflections | Compress List

Bending Stress Variables									
Cr	Cd	Lex	Rbx	FbEx	Clx	Ley	Rby	FbEy	Cly
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000
1.00	1.000	60.000	5.071	23,866.8	0.997	60.000	2.760	80,550.6	1.000

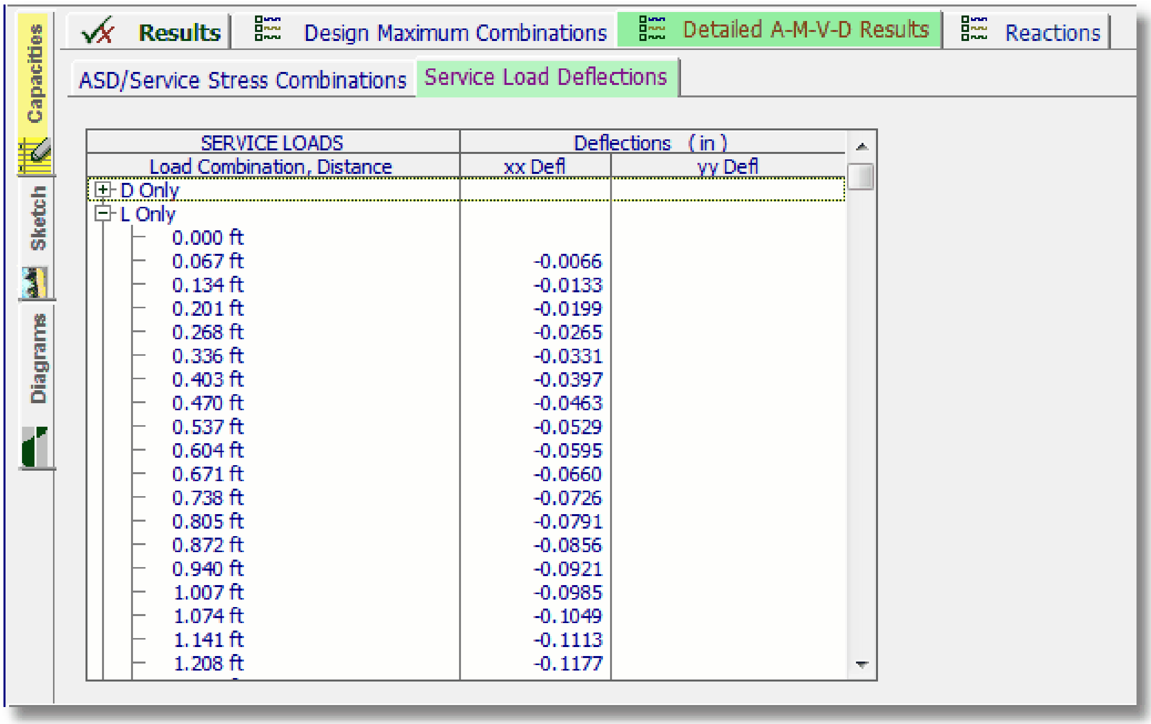
And then the final data column.....

s | Service Load Deflections | Compress List

Column Stress Variables			
Fce - X	Fce - Y	C-D	Cp
729.9	324.4	1.000	0.299
729.9	324.4	1.000	0.299
729.9	324.4	1.000	0.299
729.9	324.4	1.000	0.299
729.9	324.4	1.000	0.299
729.9	324.4	1.000	0.299
729.9	324.4	1.000	0.299
729.9	324.4	1.000	0.299
729.9	324.4	1.000	0.299
729.9	324.4	1.000	0.299
729.9	324.4	1.000	0.299
729.9	324.4	1.000	0.299

Detailed Results Tab - Deflections

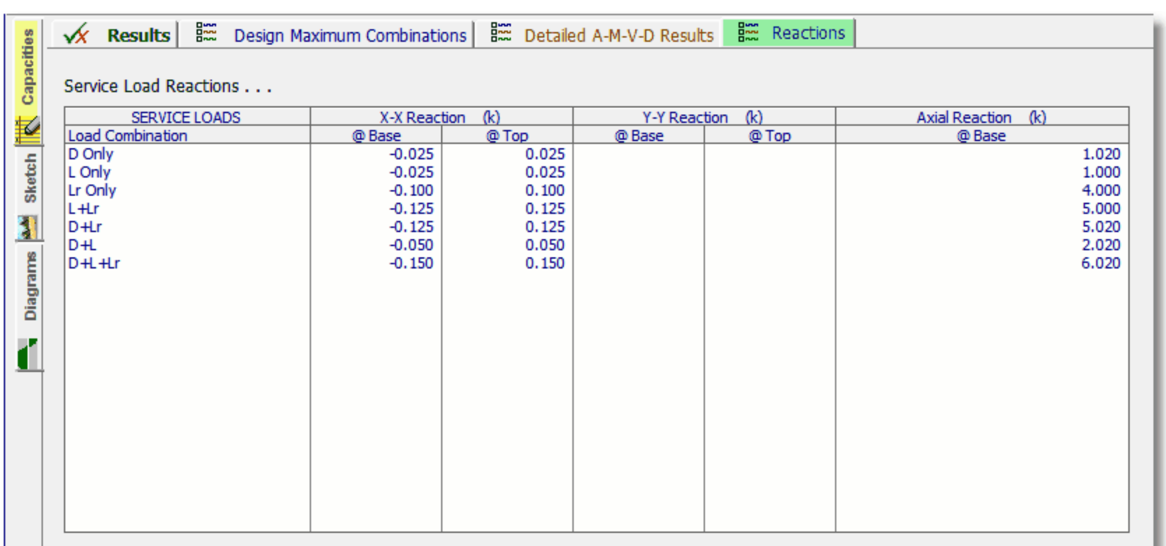
This tab reports the deflections at incremental locations along the height of the column, for each service load condition (i.e. for individual load cases and for a set of built-in service load combinations), along each axis. It is important to understand that the deflection values indicated on this tab represent RELATIVE deflections. That is, they represent the distance from the deflected shape of the column to a straight-line chord drawn between the deflected locations of the top and bottom nodes of the column. This has significance when viewing the deflection data for a column that is fixed at the bottom and free to deflect at the top. If lateral loads are applied to this type of cantilevered column, it is reasonable to expect that the free top will deflect laterally. However, when viewing the results on the Service Load Deflections tab, the deflection at both the bottom AND the top will always be reported as zero, because this tab is reporting RELATIVE deflections, and by definition, the relative deflection of either endpoint of a member is zero. Do not interpret the results on the Service Load Deflections tab as ABSOLUTE deflections.



SERVICE LOADS	Deflections (in)	
Load Combination, Distance	xx Defl	yy Defl
D Only		
L Only		
0.000 ft		
0.067 ft	-0.0066	
0.134 ft	-0.0133	
0.201 ft	-0.0199	
0.268 ft	-0.0265	
0.336 ft	-0.0331	
0.403 ft	-0.0397	
0.470 ft	-0.0463	
0.537 ft	-0.0529	
0.604 ft	-0.0595	
0.671 ft	-0.0660	
0.738 ft	-0.0726	
0.805 ft	-0.0791	
0.872 ft	-0.0856	
0.940 ft	-0.0921	
1.007 ft	-0.0985	
1.074 ft	-0.1049	
1.141 ft	-0.1113	
1.208 ft	-0.1177	

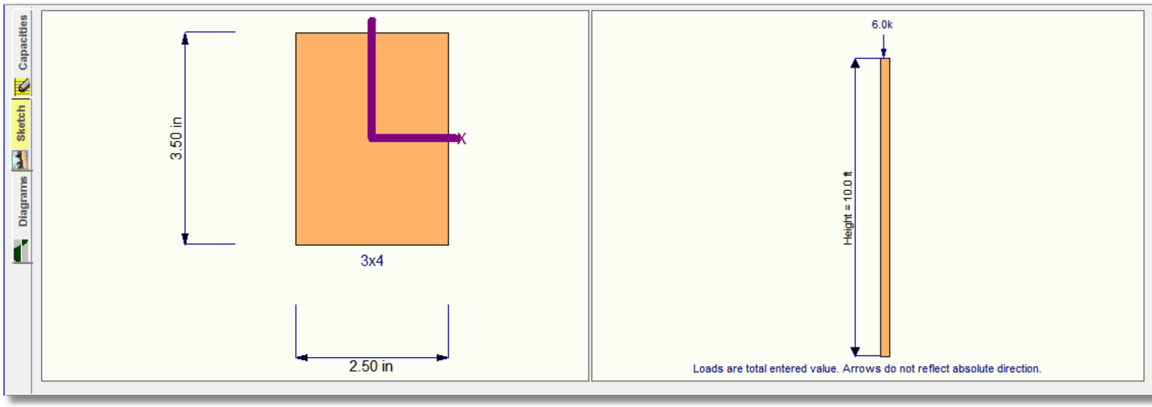
End Reactions Tab

This tab provides the sideways (non-axial) reactions for individual load cases and for a set of built-in service load combinations, along each axis.



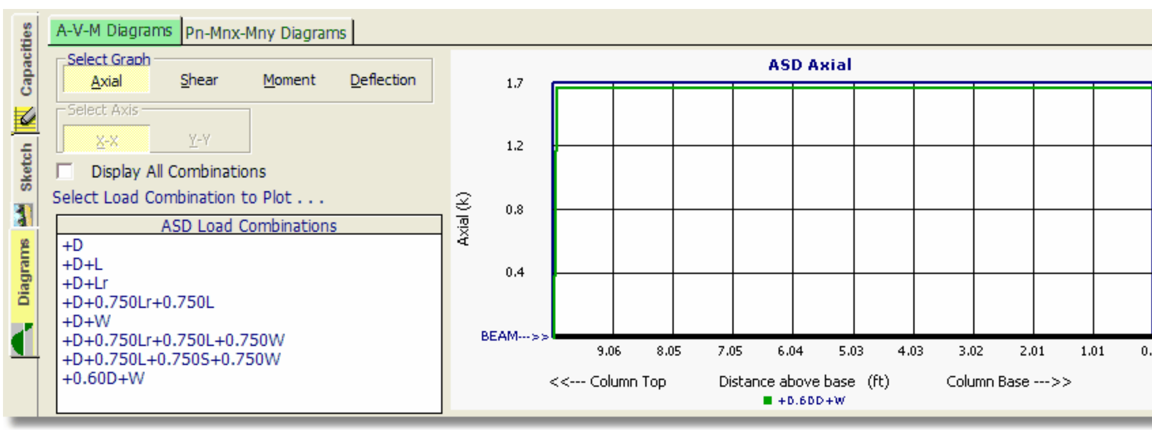
SERVICE LOADS	X-X Reaction (k)		Y-Y Reaction (k)		Axial Reaction (k)
Load Combination	@ Base	@ Top	@ Base	@ Top	@ Base
D Only	-0.025	0.025			1.020
L Only	-0.025	0.025			1.000
Lr Only	-0.100	0.100			4.000
L+Lr	-0.125	0.125			5.000
D+Lr	-0.125	0.125			5.020
D+L	-0.050	0.050			2.020
D+L+Lr	-0.150	0.150			6.020

Sketch Tab



Axial-Shear-Moment Diagrams

This tab provides comprehensive charting capability to view graphs of Axial load, Shear, Moment, and Deflection along the length of the member. Note that the graphs are oriented such that the right end of the graph represents the column base, and the left end of the graph represents the column top. This was done to maximize the scale of the graph based on the screen area available.

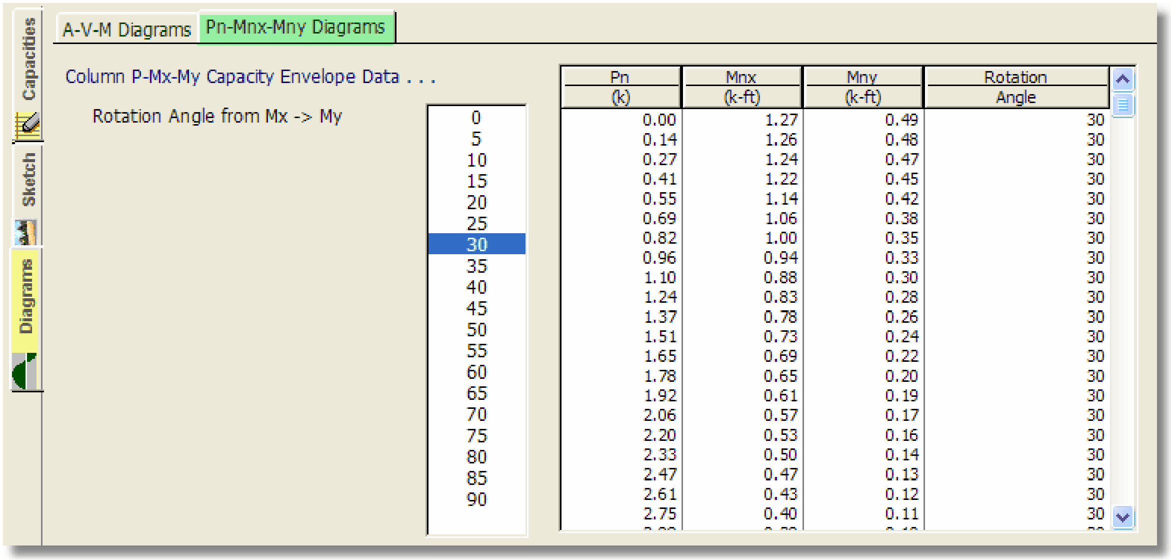


Pn-Pnx-Mny Diagrams

This tab allows you to see the moment capacities about each axis given a certain allowable axial load.

This is mostly for reference and can be considered a reverse application of the load capacity calculations.

In the screen capture below we have selected a 30 degree angle. For a 1.1 kip axial load you can see the moment capacity values, Mnx and Mny.



Pn (k)	Mnx (k-ft)	Mny (k-ft)	Rotation Angle
0.00	1.27	0.49	30
0.14	1.26	0.48	30
0.27	1.24	0.47	30
0.41	1.22	0.45	30
0.55	1.14	0.42	30
0.69	1.06	0.38	30
0.82	1.00	0.35	30
0.96	0.94	0.33	30
1.10	0.88	0.30	30
1.24	0.83	0.28	30
1.37	0.78	0.26	30
1.51	0.73	0.24	30
1.65	0.69	0.22	30
1.78	0.65	0.20	30
1.92	0.61	0.19	30
2.06	0.57	0.17	30
2.20	0.53	0.16	30
2.33	0.50	0.14	30
2.47	0.47	0.13	30
2.61	0.43	0.12	30
2.75	0.40	0.11	30

10.3.3 Steel Column

Need more? Ask Us a Question

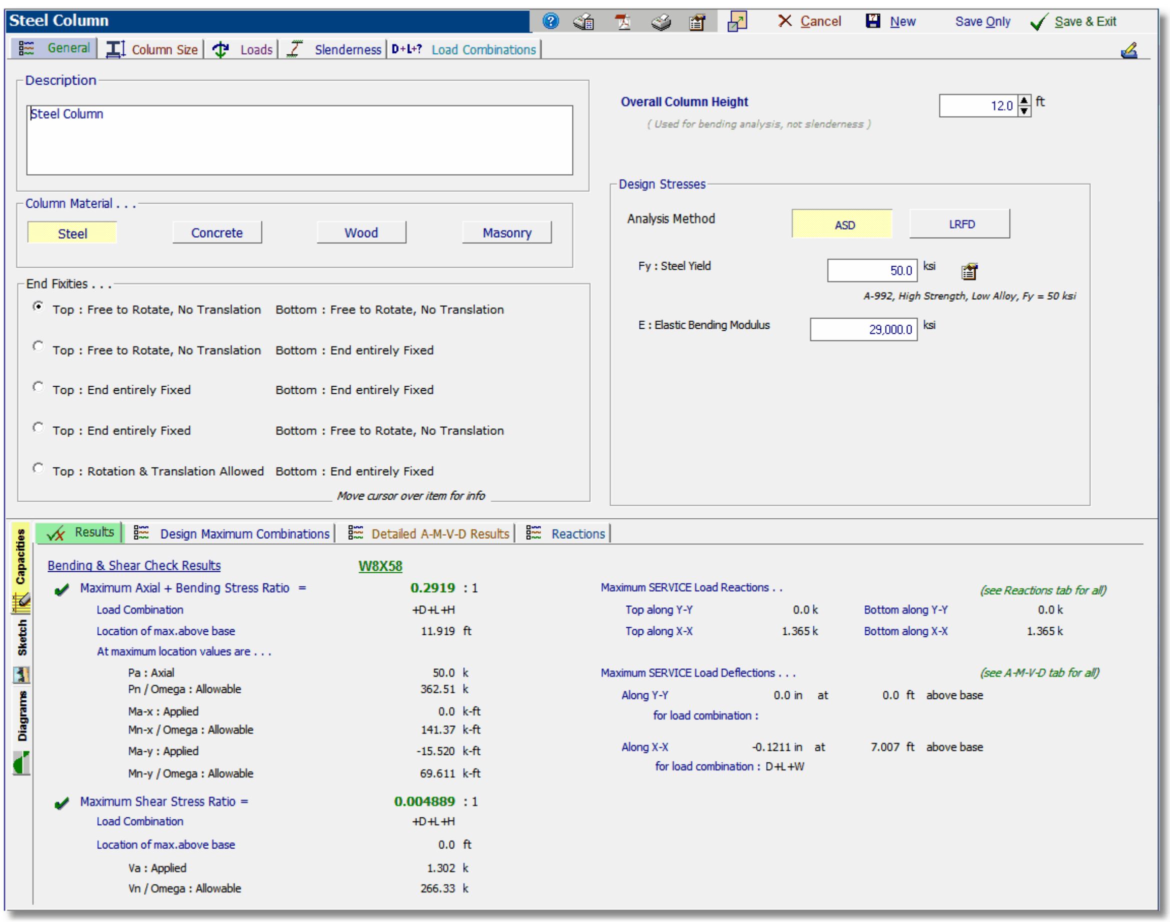
This module designs steel columns that are subject to axial loads and lateral bending loads about both axes. Click here for a video: Steel Column

The user can select ASD or LRFD methods and has access to a complete database of steel section sizes.

All calculations are according to AISC 360-05.

The screen capture below shows the full screen for steel column design. See items below for descriptions of items that are specific to the steel column design module.

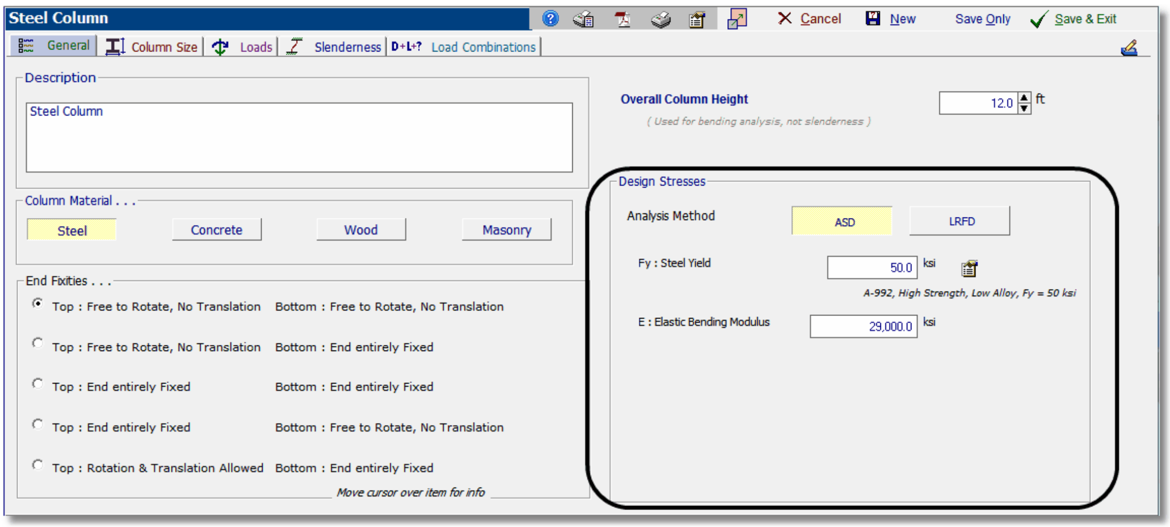
For general description of the module, end fixity, loads, and load combinations click here 326. For slenderness description click here 331.



General Tab

The area shown bubbled in the screen capture below is specific to the steel column selection. Here you can specify ASD or LRFD design procedure and specify the yield strength and elastic modulus of the steel member to be used.

Click the button to display the Steel Fy Selections dialog to select a standard grade.



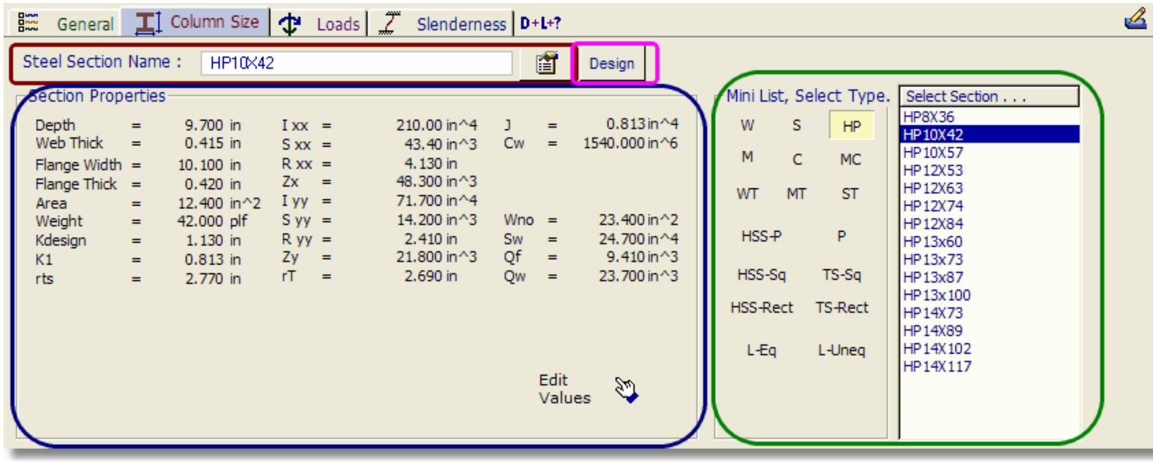
Column Size Tab

All items on this tab are specific to a steel column.

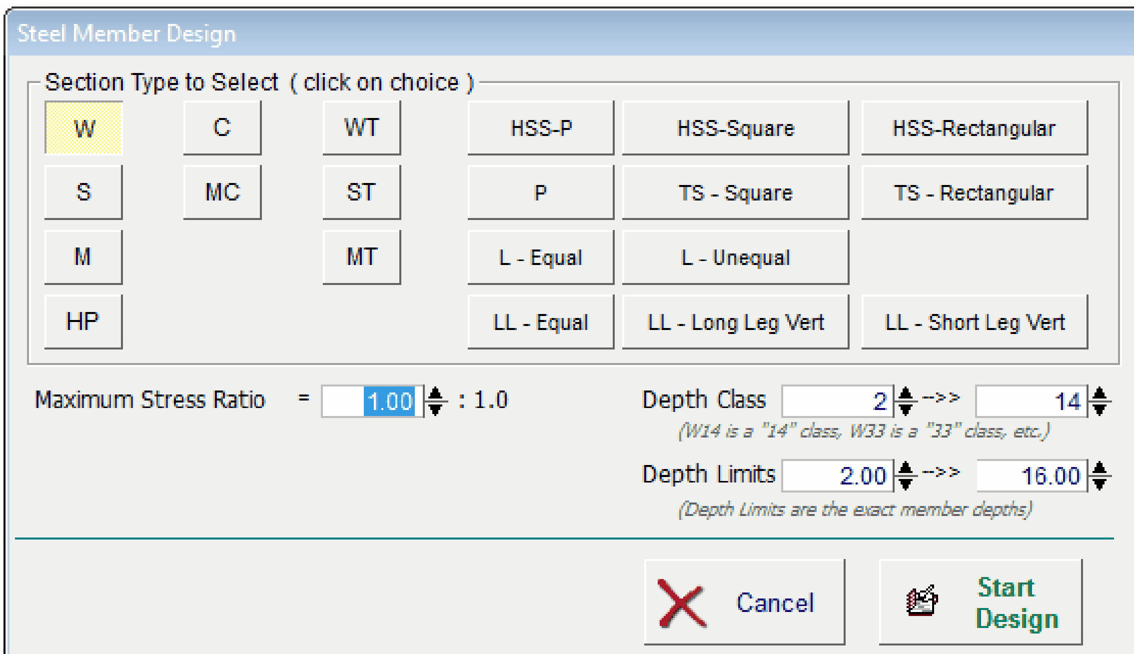
The item circled in **red** is where you can type in the typical section name, press [Tab], and the section will be searched and retrieved from our built-in database. Or you can click the button, and you can select a section from the built-in AISC database.

The items circled in **blue** are the section properties for the section you have chosen.

The item circled in **green** is the Quick-List". Click on one of the many buttons with a section letter and the full list of those sections will be displayed to the right. Simply click on a section and it will be assigned to the red and blue areas.



The button circled in **pink** will display the Steel Member Design dialog (see below).



This tool checks all of the steel sections for the selected type (W is selected in this case). A section will be judged to pass if the maximum stress ratio entered is not exceeded and the depth measurements and depth class are not exceeded.

We use the term "Depth Class" to refer to the nominal dimension of the family to which the section belongs (ex: W14). It does refer to the actual depth of the section. All sections starting with "W14" are of the "14" depth class.

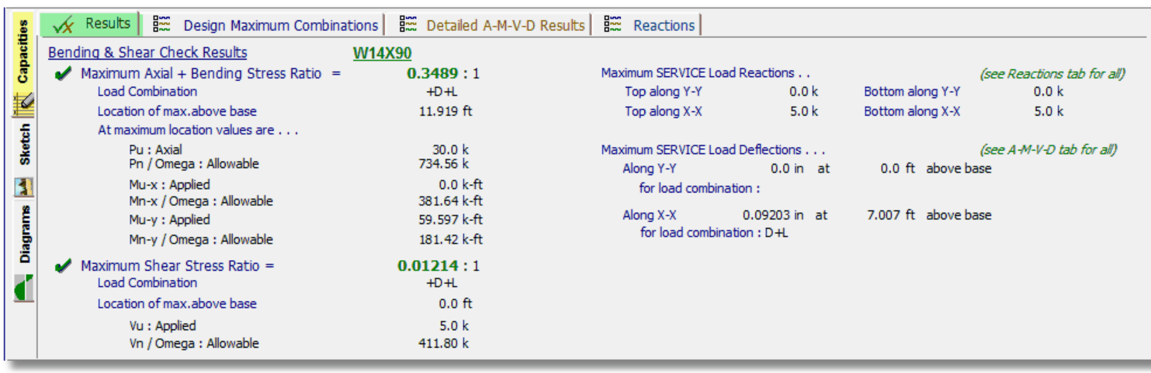
Results Tab

This tab provides a summary of the stress ratios, reactions and deflections for the column.

Max Axial + Bending Stress Ratio is the governing load combination for the column. Listed is the governing load load combination, the AISC formulas that are used, and the location of the maximum stress ratio above the base of the column. Please note that the maximum stress ratio is being reported (along with its location) because it governs the overall design, even though it may not necessarily be the highest axial stress or the highest bending stress experienced anywhere in the column.

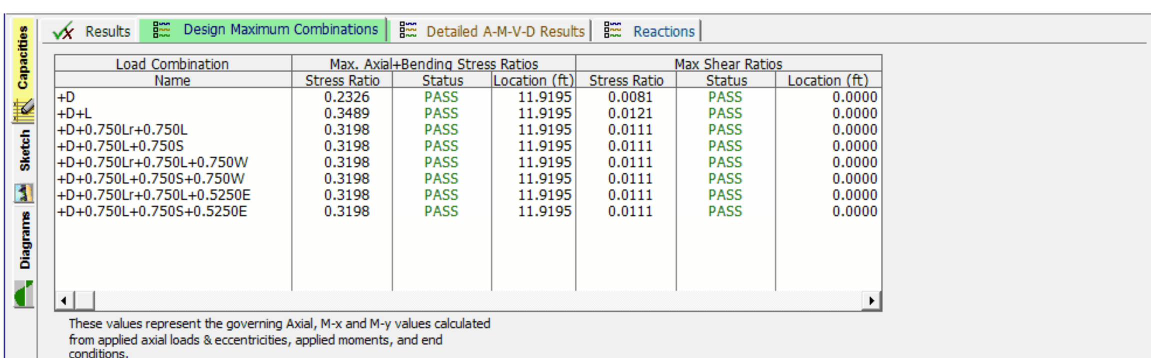
Max Shear Stress Ratio will probably never govern for a normally loaded column design. But it is presented here with the governing load combination, location and allowable/actual stress values.

Lateral Load Reactions and **Deflections** are the result of applied lateral loads.



Design Maximum Combinations Tab

This tab lists the resulting maximum stress ratios for each load combination. This list is created by examining the detailed list (on the next tab) and determining the governing stress ratios for each load combination.



Load Combination	Max. Axial+Bending Stress Ratios			Max Shear Ratios		
Name	Stress Ratio	Status	Location (ft)	Stress Ratio	Status	Location (ft)
+D	0.2326	PASS	11.9195	0.0081	PASS	0.0000
+D+L	0.3489	PASS	11.9195	0.0121	PASS	0.0000
+D+0.750Lr+0.750L	0.3198	PASS	11.9195	0.0111	PASS	0.0000
+D+0.750L+0.750S	0.3198	PASS	11.9195	0.0111	PASS	0.0000
+D+0.750Lr+0.750L+0.750W	0.3198	PASS	11.9195	0.0111	PASS	0.0000
+D+0.750L+0.750S+0.750W	0.3198	PASS	11.9195	0.0111	PASS	0.0000
+D+0.750Lr+0.750L+0.5250E	0.3198	PASS	11.9195	0.0111	PASS	0.0000
+D+0.750L+0.750S+0.5250E	0.3198	PASS	11.9195	0.0111	PASS	0.0000

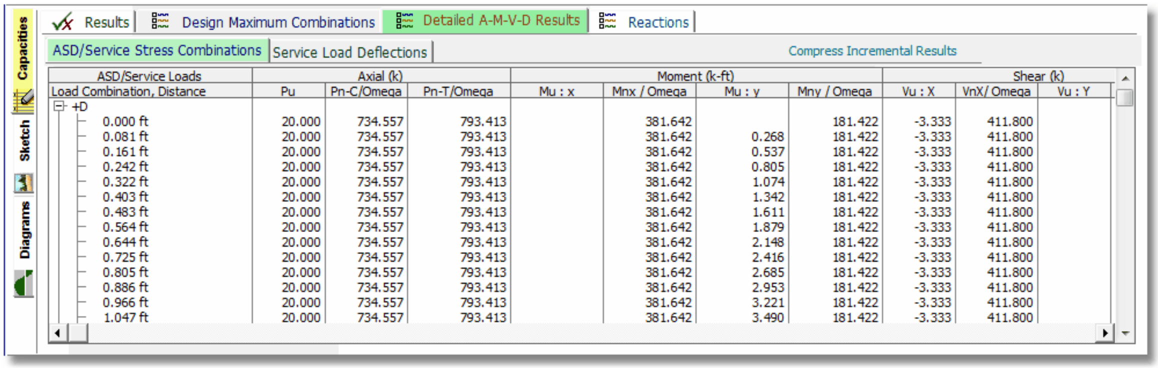
These values represent the governing Axial, M-x and M-y values calculated from applied axial loads & eccentricities, applied moments, and end conditions.

Detailed A-M-V-D Results Tab - Stresses

This tab lists the detailed results at small increments along the height of the column, for each load combination.

For consistency, all of the column headings are taken directly from AISC.

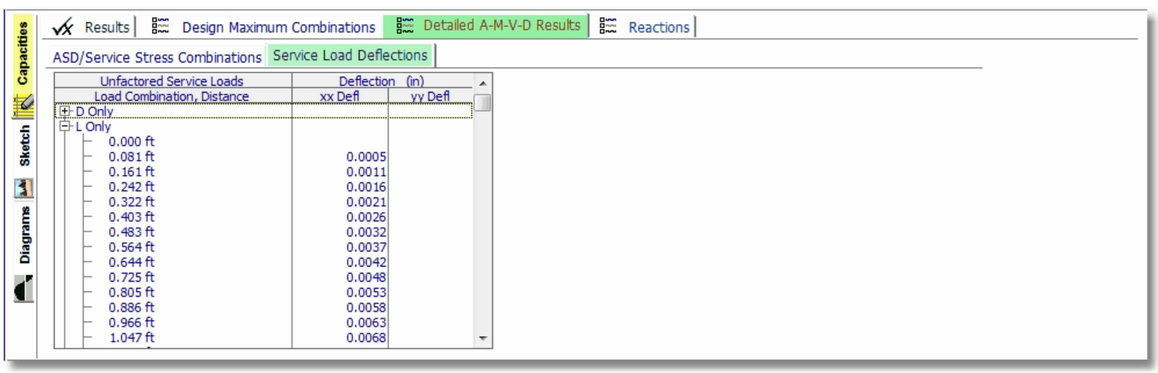
Note! This list scrolls to the right to display more information.



ASD/Service Loads	Axial (k)			Moment (k-ft)				Shear (k)		
Load Combination, Distance	Pu	Pn-C/Omega	Pn-T/Omega	Mu : x	Mnx / Omega	Mu : y	Mny / Omega	Vu : X	VnX/ Omega	Vu : Y
+D										
0.000 ft	20.000	734.557	793.413		381.642		181.422	-3.333	411.800	
0.081 ft	20.000	734.557	793.413		381.642	0.268	181.422	-3.333	411.800	
0.161 ft	20.000	734.557	793.413		381.642	0.537	181.422	-3.333	411.800	
0.242 ft	20.000	734.557	793.413		381.642	0.805	181.422	-3.333	411.800	
0.322 ft	20.000	734.557	793.413		381.642	1.074	181.422	-3.333	411.800	
0.403 ft	20.000	734.557	793.413		381.642	1.342	181.422	-3.333	411.800	
0.483 ft	20.000	734.557	793.413		381.642	1.611	181.422	-3.333	411.800	
0.564 ft	20.000	734.557	793.413		381.642	1.879	181.422	-3.333	411.800	
0.644 ft	20.000	734.557	793.413		381.642	2.148	181.422	-3.333	411.800	
0.725 ft	20.000	734.557	793.413		381.642	2.416	181.422	-3.333	411.800	
0.805 ft	20.000	734.557	793.413		381.642	2.685	181.422	-3.333	411.800	
0.886 ft	20.000	734.557	793.413		381.642	2.953	181.422	-3.333	411.800	
0.966 ft	20.000	734.557	793.413		381.642	3.221	181.422	-3.333	411.800	
1.047 ft	20.000	734.557	793.413		381.642	3.490	181.422	-3.333	411.800	

Detailed A-M-V-D Results Tab - Service Load Deflections

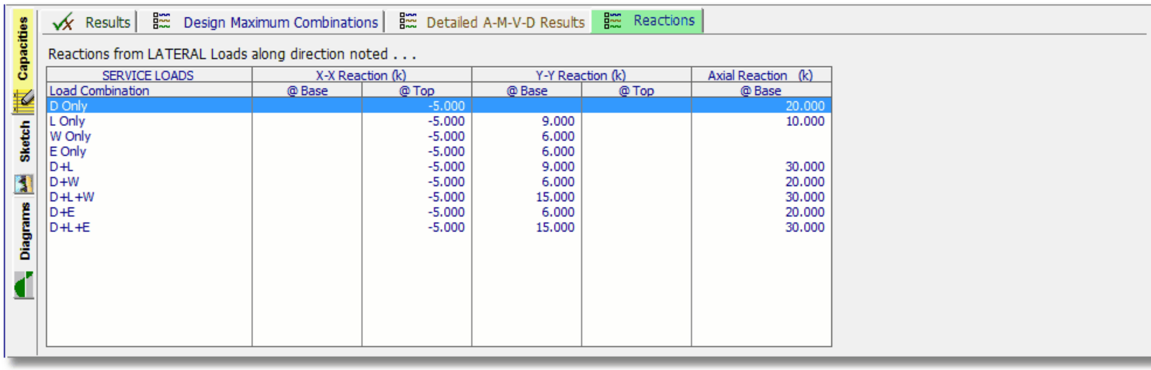
This tab reports the deflections at incremental locations along the height of the column, for each service load condition (i.e. for individual load cases and for a set of built-in service load combinations), along each axis.



Unfactored Service Loads	Deflection (in)	
Load Combination, Distance	xx Defl	yy Defl
D Only		
L Only		
0.000 ft		
0.081 ft	0.0005	
0.161 ft	0.0011	
0.242 ft	0.0016	
0.322 ft	0.0021	
0.403 ft	0.0026	
0.483 ft	0.0032	
0.564 ft	0.0037	
0.644 ft	0.0042	
0.725 ft	0.0048	
0.805 ft	0.0053	
0.886 ft	0.0058	
0.966 ft	0.0063	
1.047 ft	0.0068	

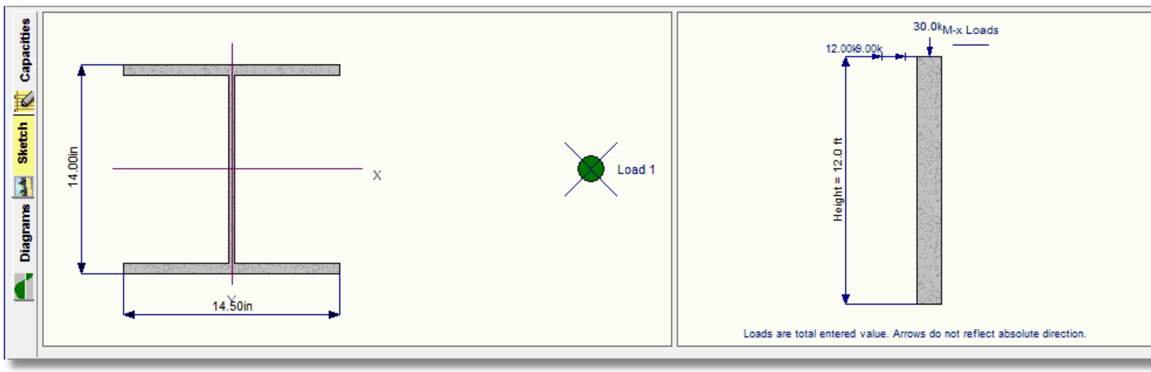
Reactions Tab

This tab provides the sideways (non-axial) reactions for individual load cases and for a set of built-in service load combinations, along each axis.



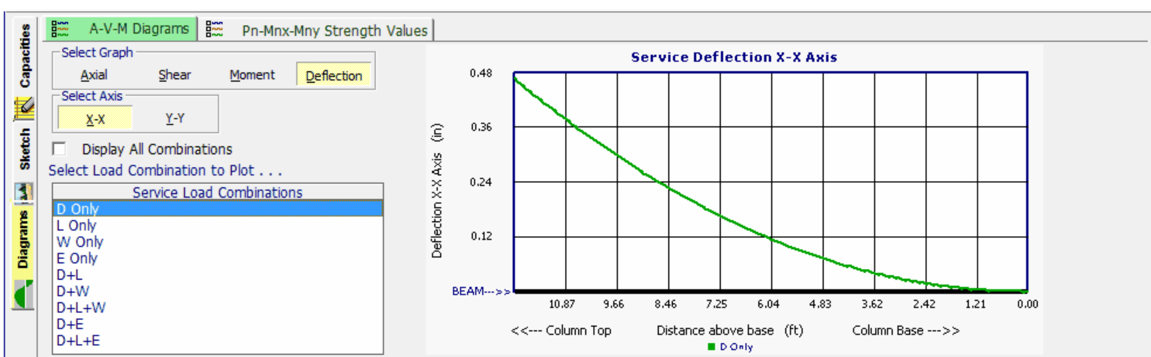
SERVICE LOADS	X-X Reaction (k)		Y-Y Reaction (k)		Axial Reaction (k)
Load Combination	@ Base	@ Top	@ Base	@ Top	@ Base
D Only		-5.000			20.000
L Only		-5.000	9.000		10.000
W Only		-5.000	6.000		
E Only		-5.000	6.000		
D+L		-5.000	9.000		30.000
D+W		-5.000	6.000		20.000
D+L+W		-5.000	15.000		30.000
D+E		-5.000	6.000		20.000
D+L+E		-5.000	15.000		30.000

Sketch Tab



Axial-Shear-Moment Diagrams

This tab provides comprehensive charting capability to view graphs of Axial load, Shear, Moment, and Deflection along the length of the member. Note that the graphs are oriented such that the right end of the graph represents the column base, and the left end of the graph represents the column top. This was done to maximize the scale of the graph based on the screen area available.



10.3.4 Concrete Column

[Need more? Ask Us a Question]

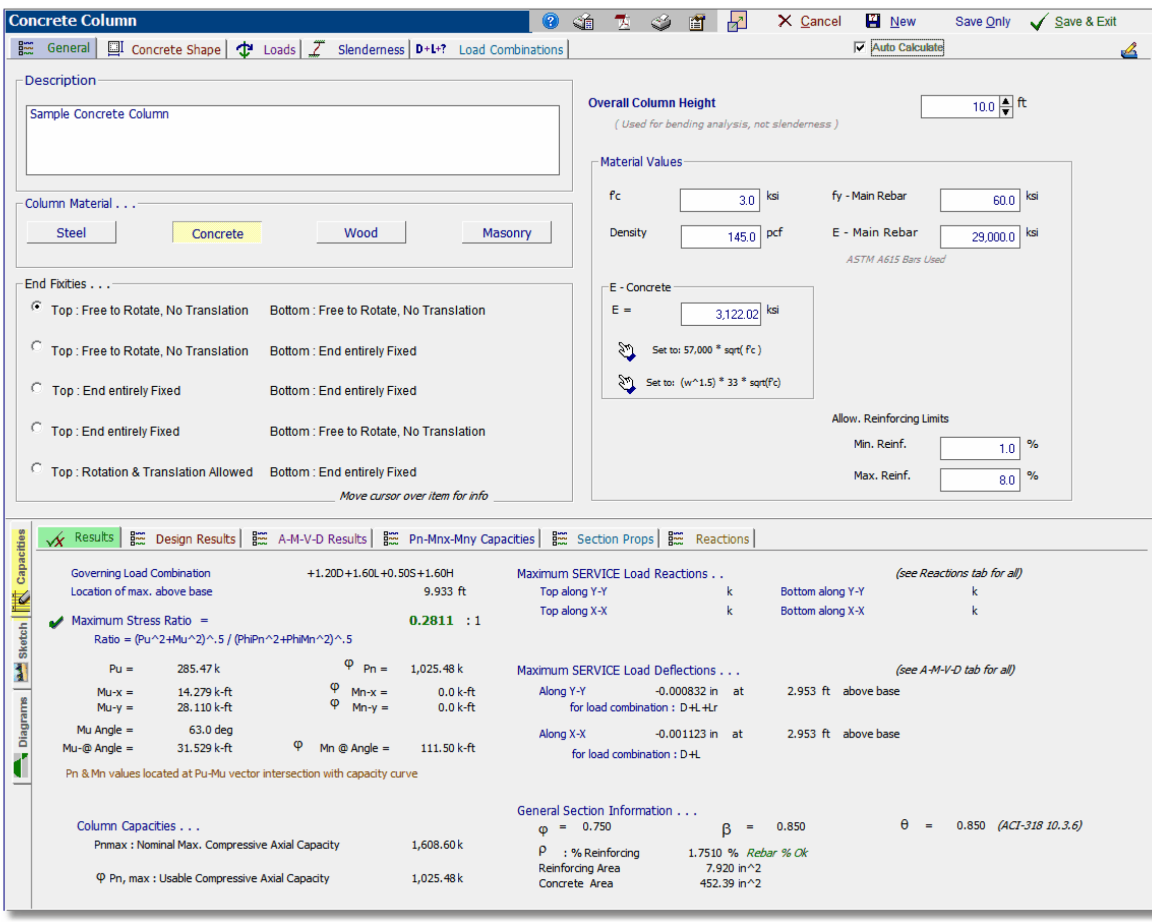
This module designs concrete columns that are subject to axial loads and lateral bending loads about both axes. Click here for a video: Concrete Column

The module only uses strength design for concrete.

All calculations are according to the referenced version of ACI 318 based on the selected governing building code.

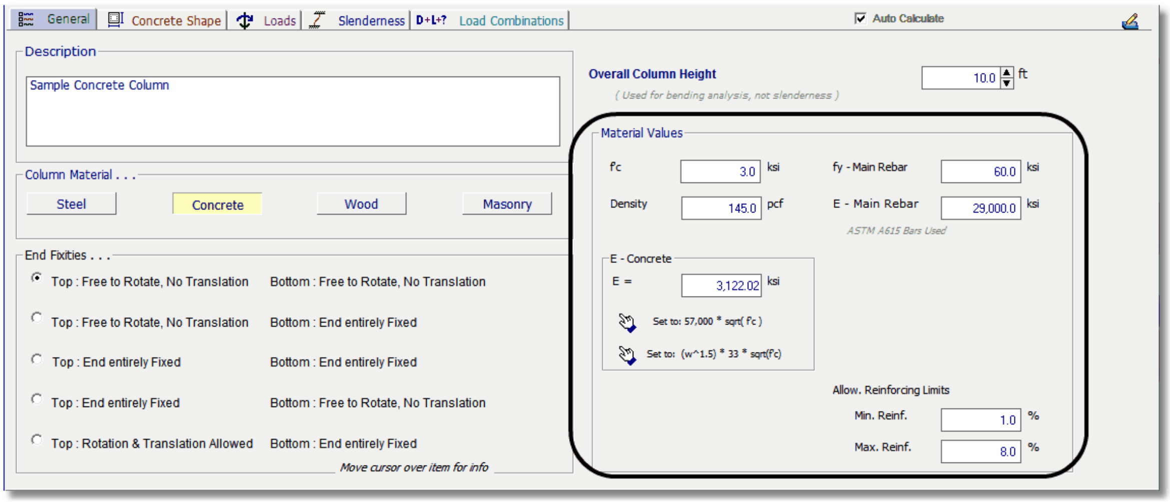
The screen capture below shows the full screen for concrete column design. See items below for descriptions of items that are specific to the concrete column design module.

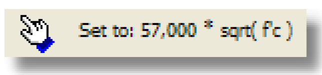
For general description of the module, end fixity, loads, and load combinations click here 326. For slenderness description click here 331.

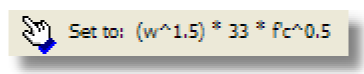


General Tab

The area circled in red in the screen capture below is specific to the concrete column selection.



The two buttons 

and 

immediately set the value for elastic modulus "E" to the values as described on the button.

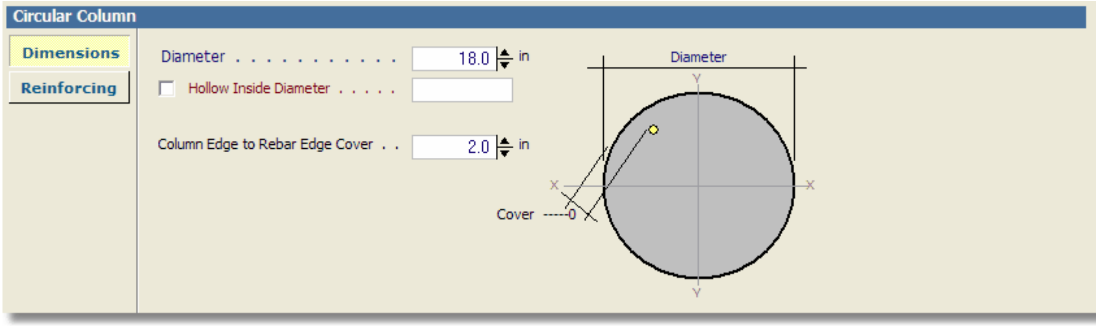
Concrete Shape

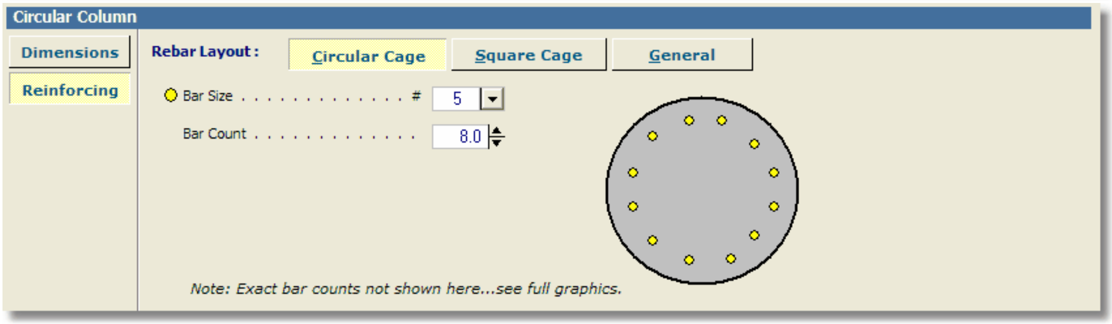
This tab is specific to the concrete column selection. It allows you to select from 12 different column shapes. Simply click the button surrounding the column shape icon and the screen below will change to allow specific data input for measurements and reinforcing layout.

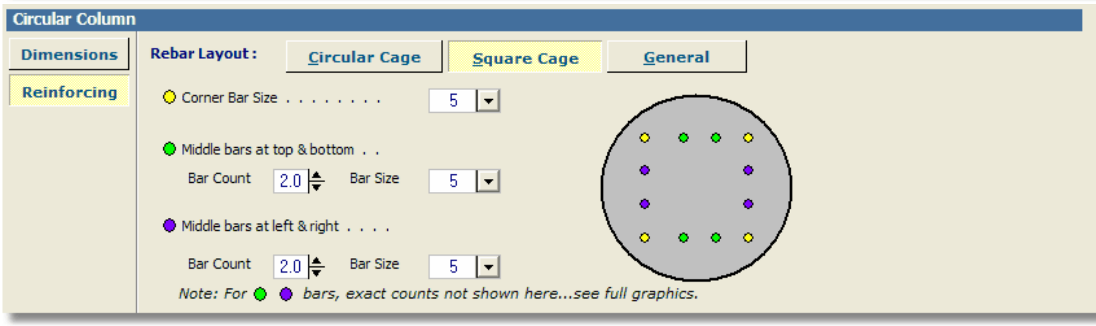
Following the screen capture below we will show ALL the data input areas for ALL the column shapes with descriptions as needed.

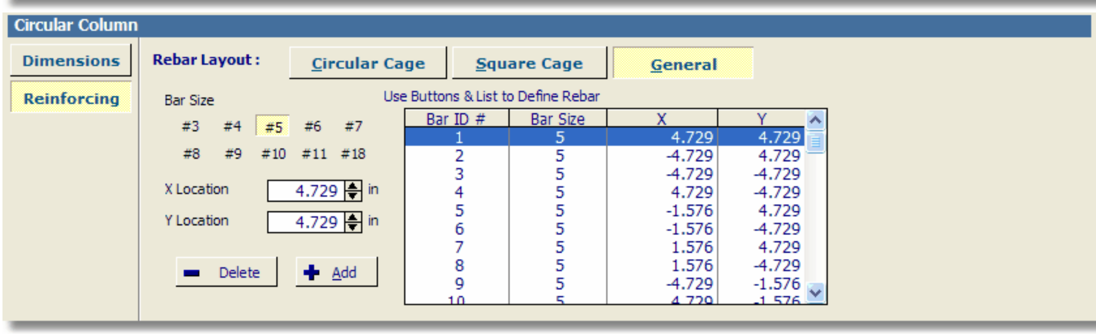
Note that this module does a very detailed biaxial analysis of the column cross section using exact numerical methods.

Circular Column

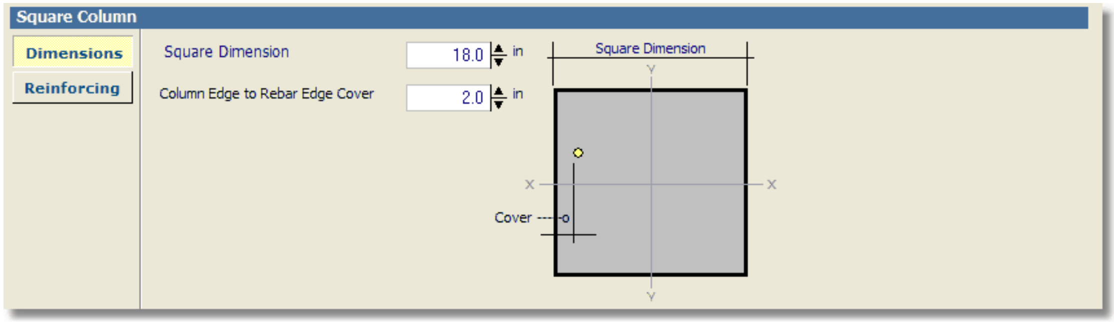


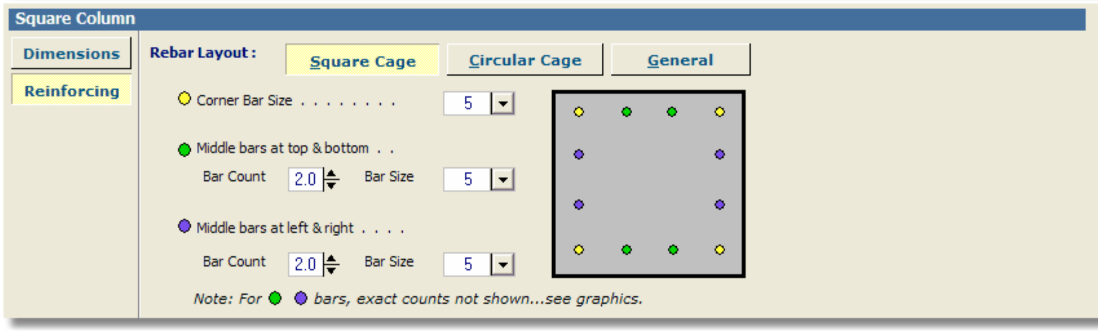


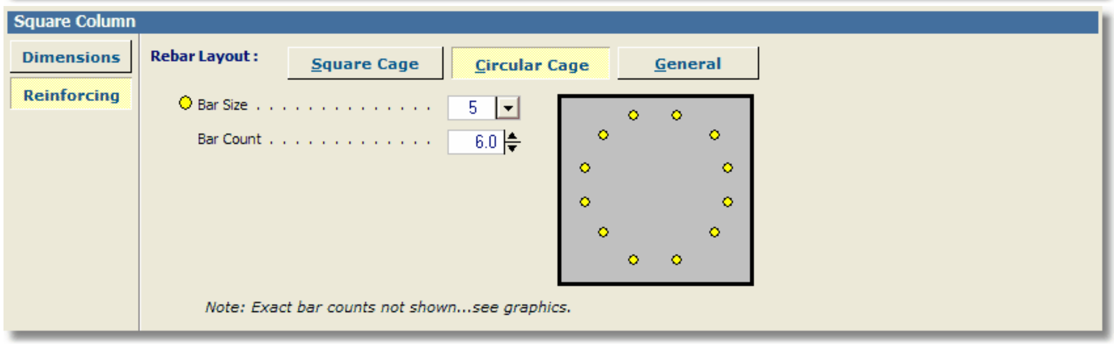


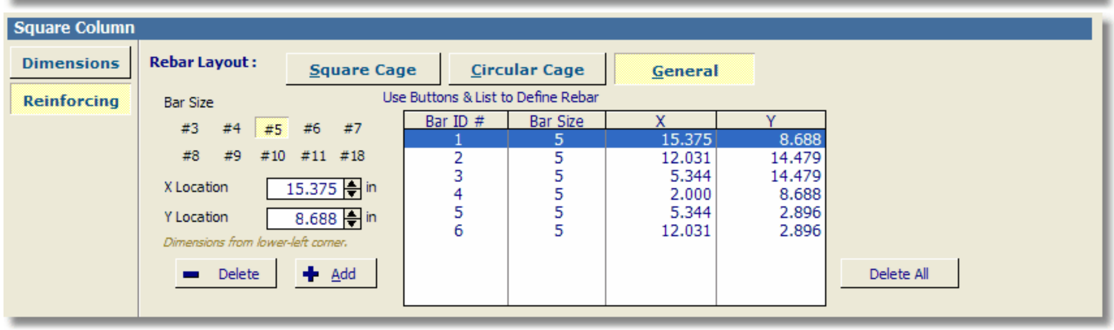


Square Column

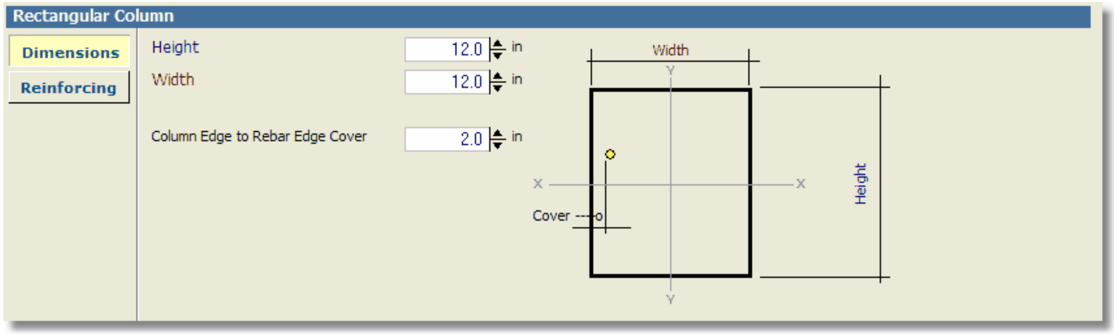


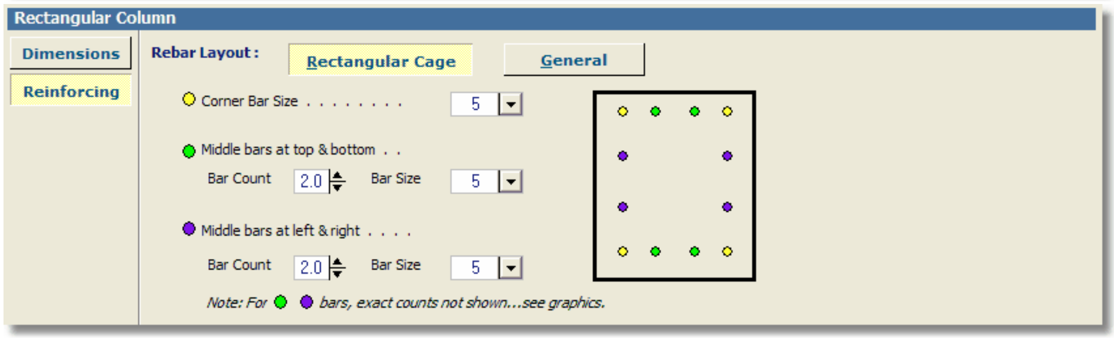


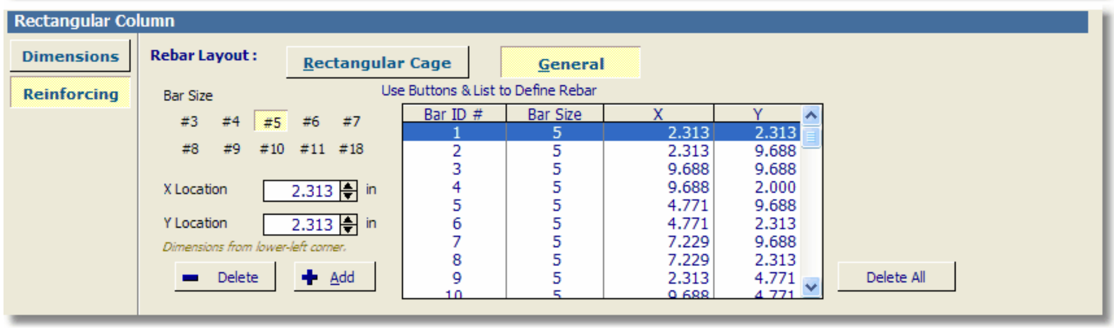




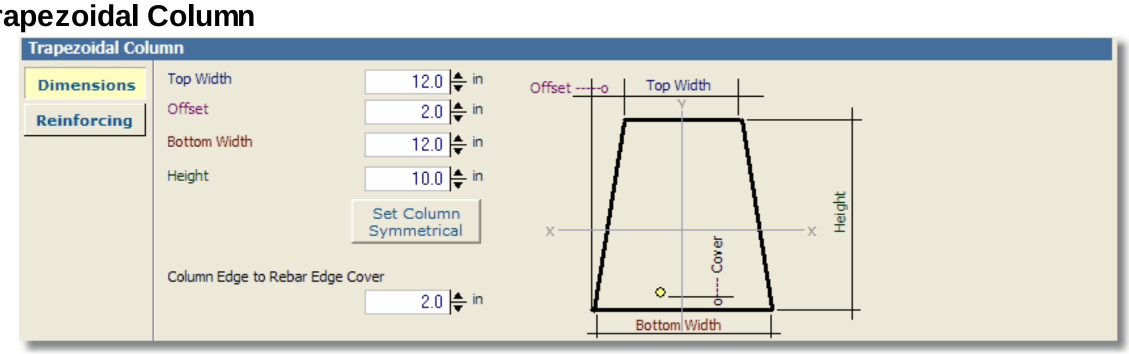
Rectangular Column

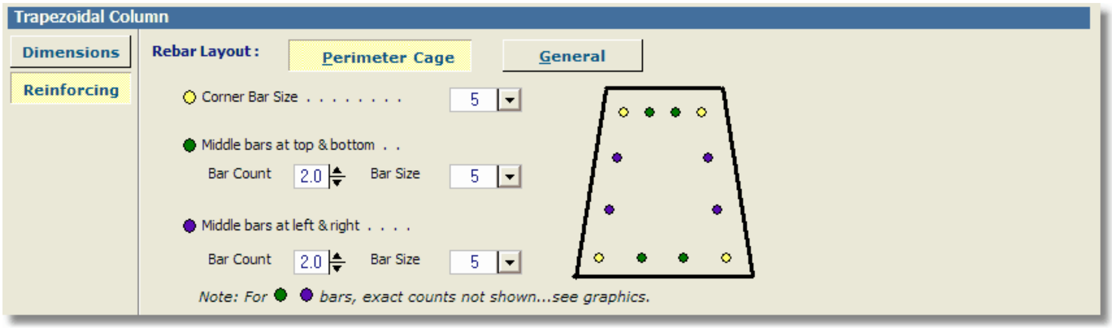


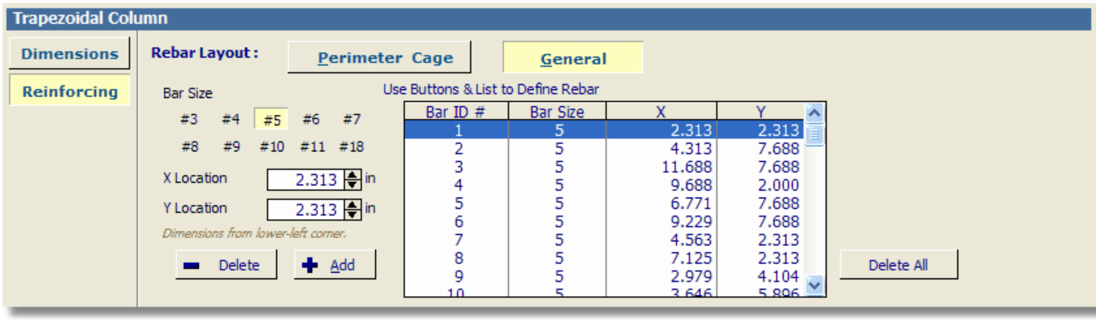




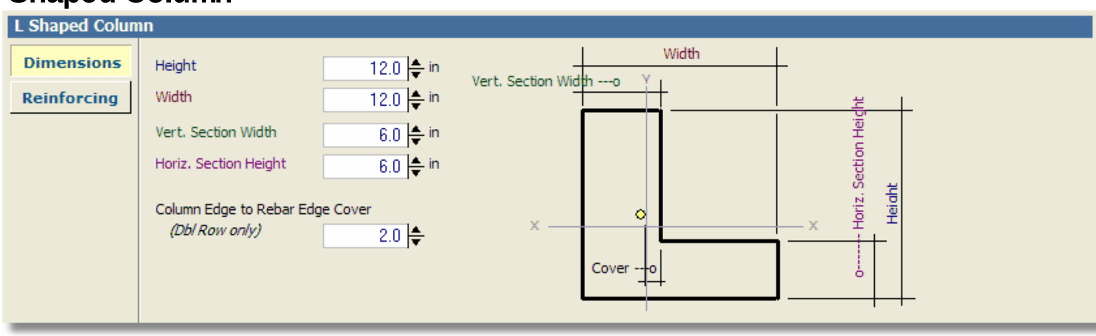
Trapezoidal Column

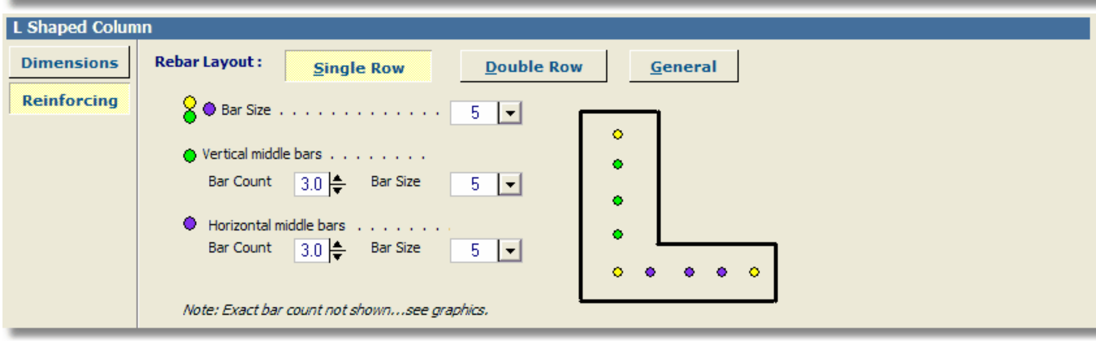


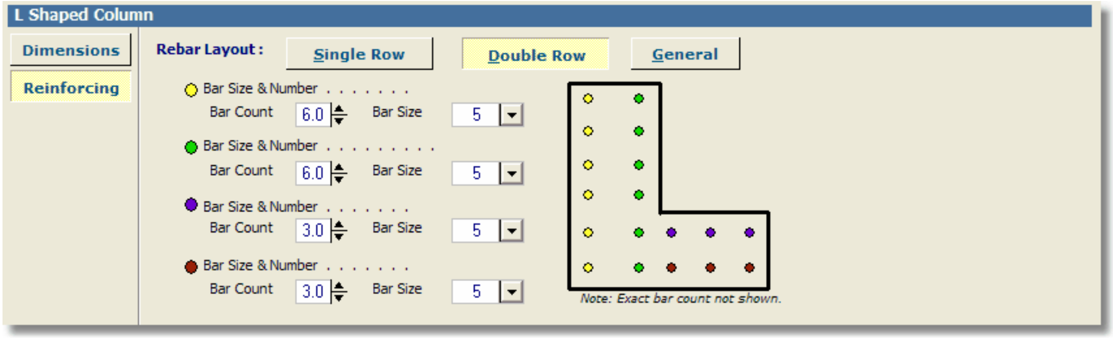


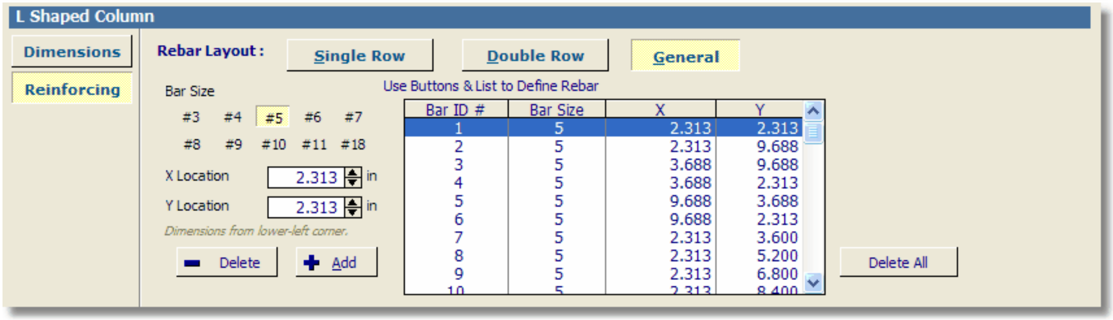


"L" Shaped Column

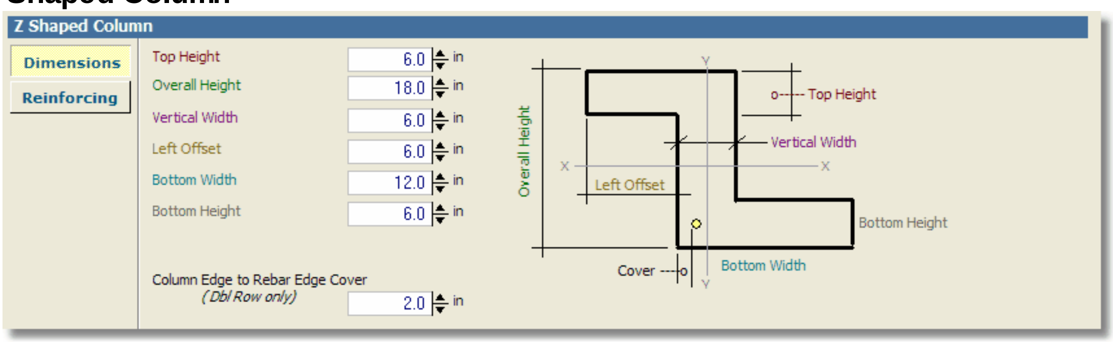


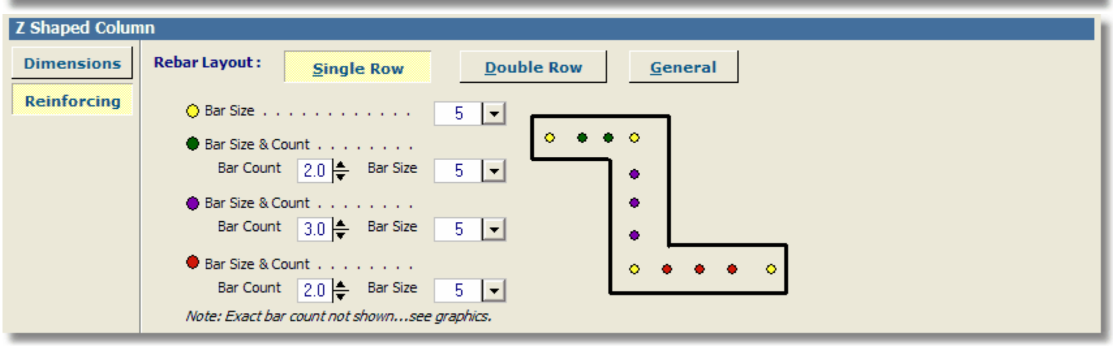


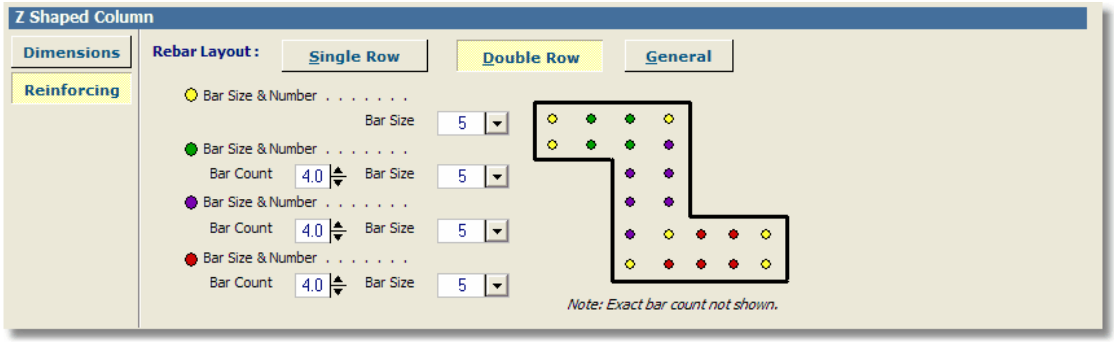


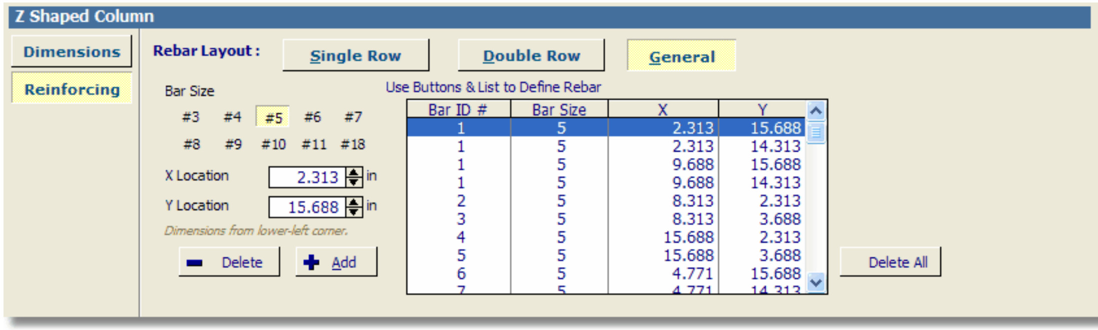


"Z" Shaped Column

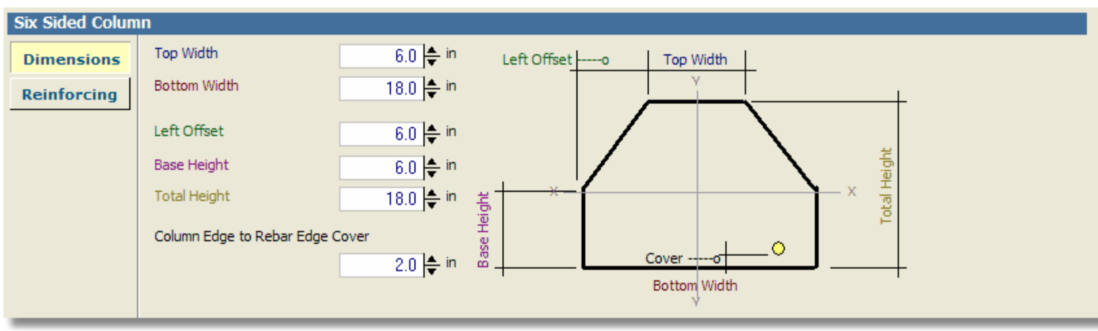


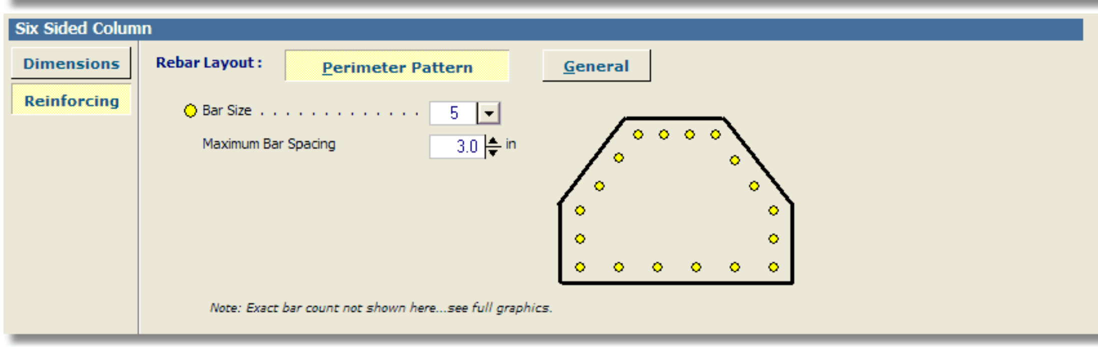


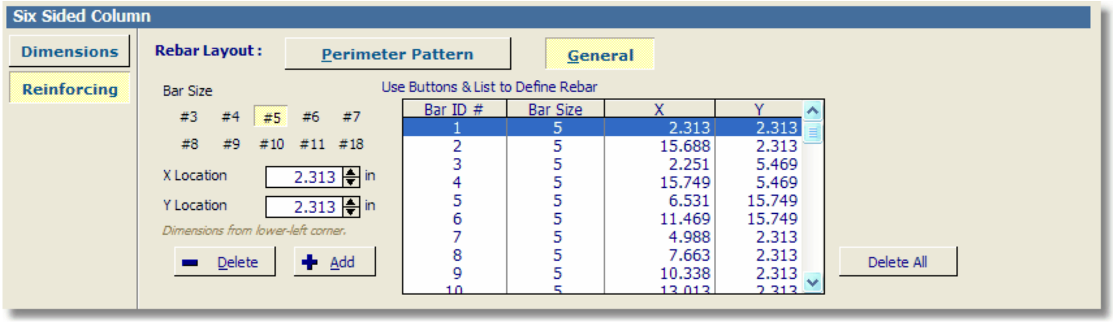




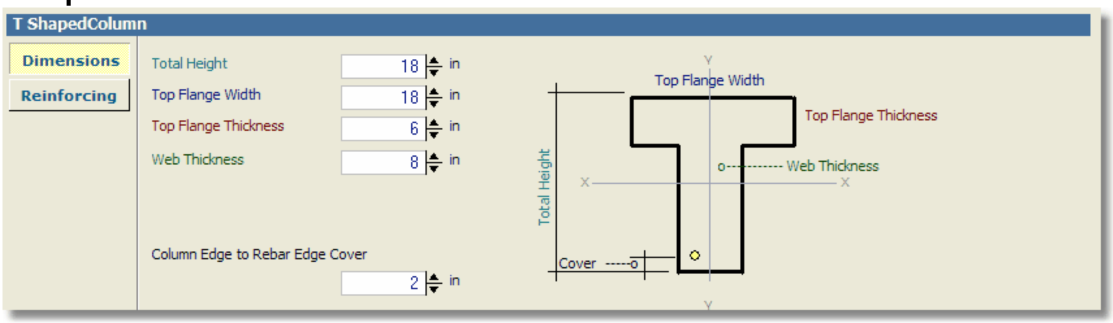
Six Sided Column

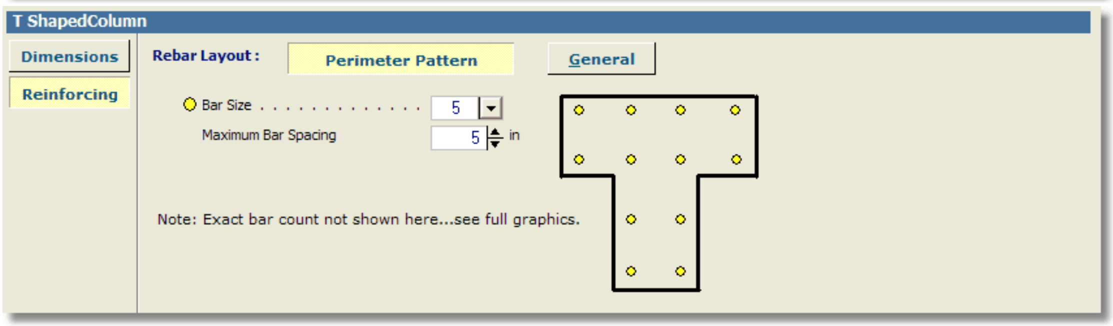


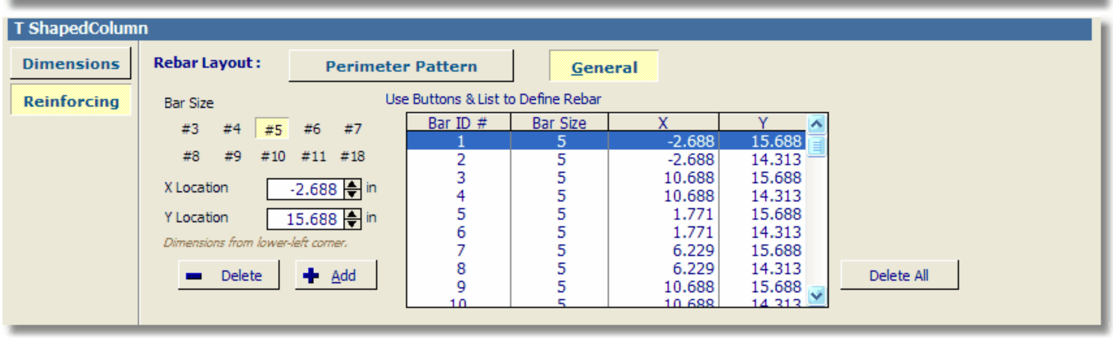




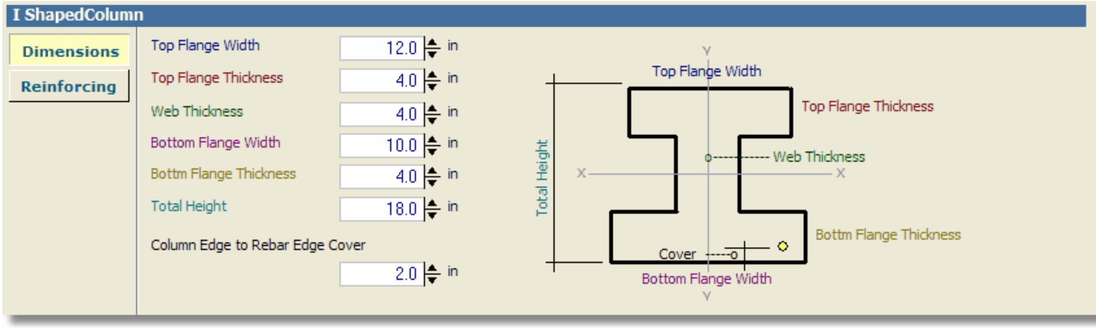
"T" Shaped Column

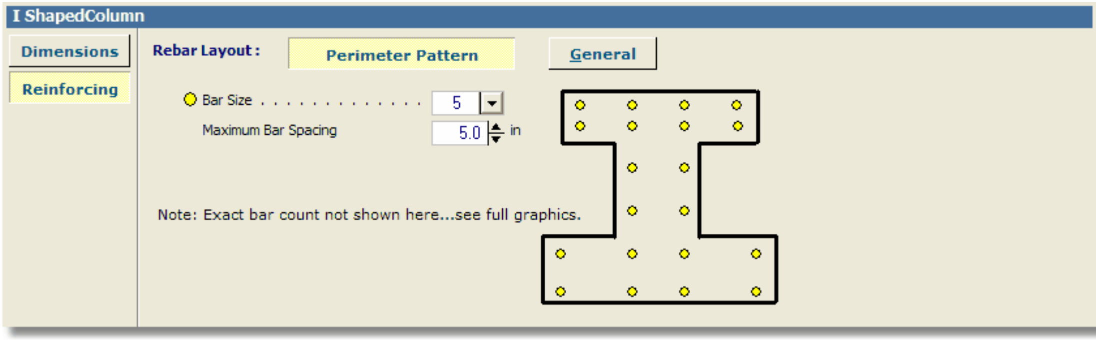


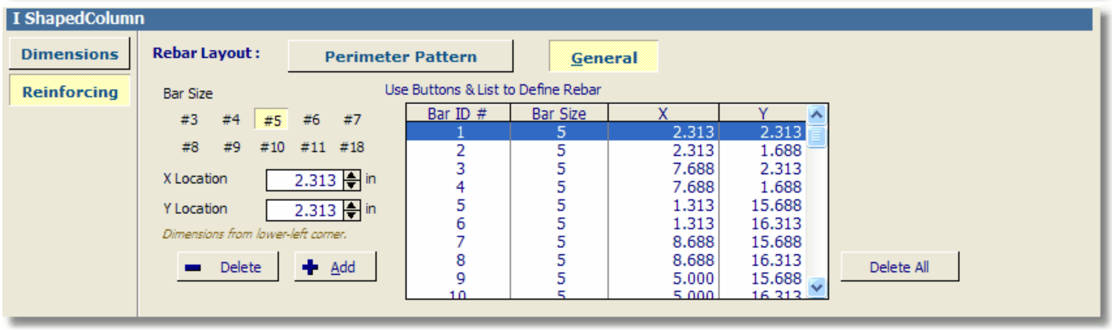




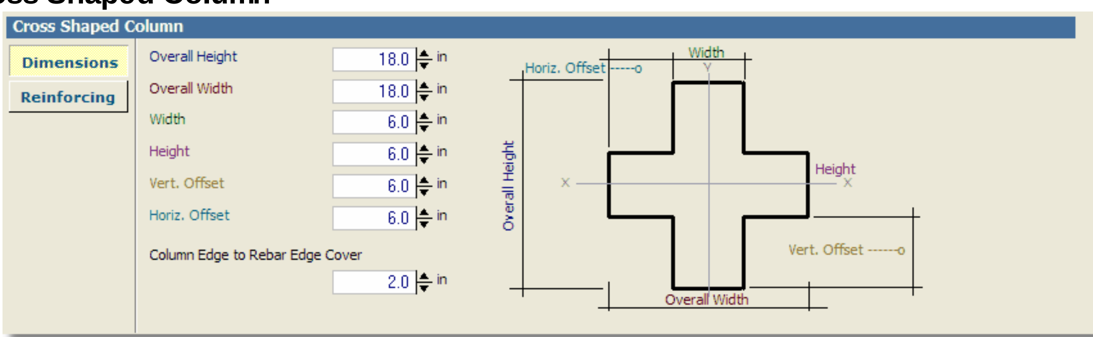
"I" Shaped Column

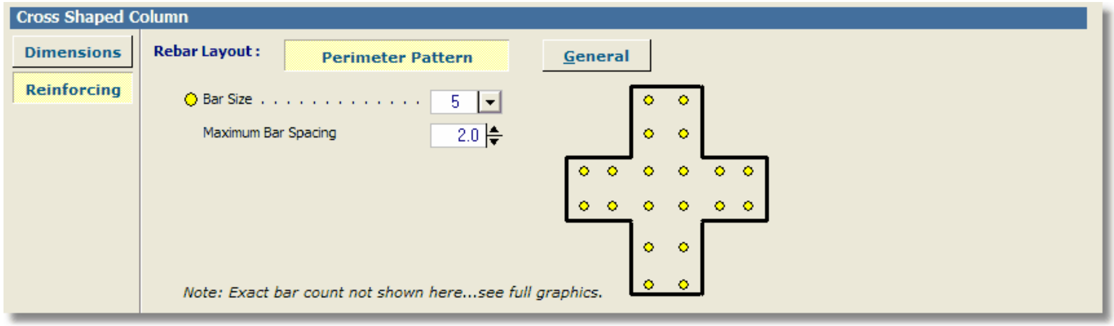


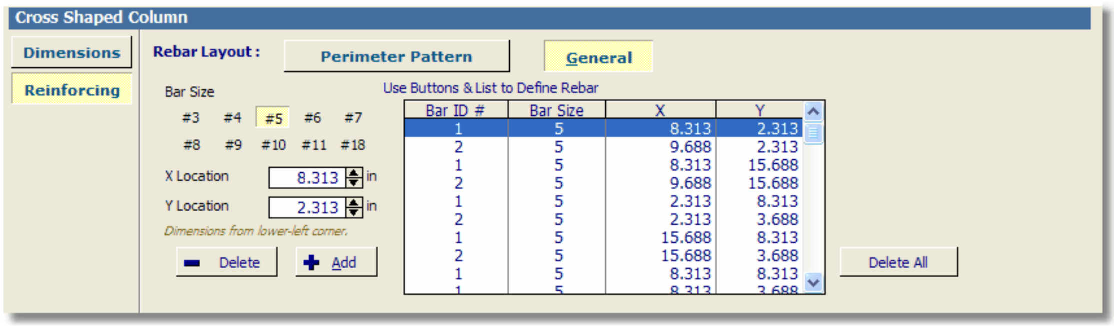




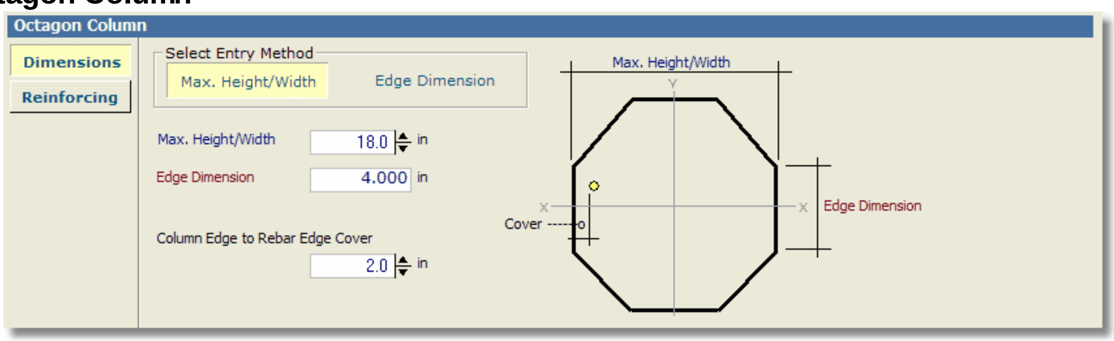
Cross Shaped Column

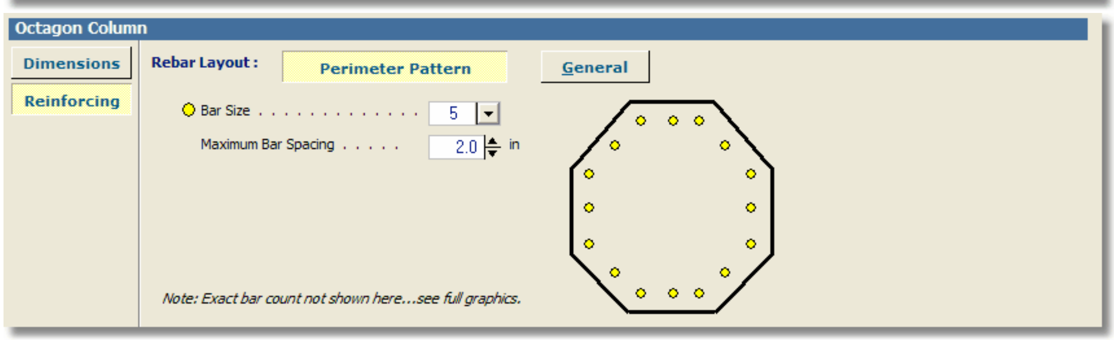


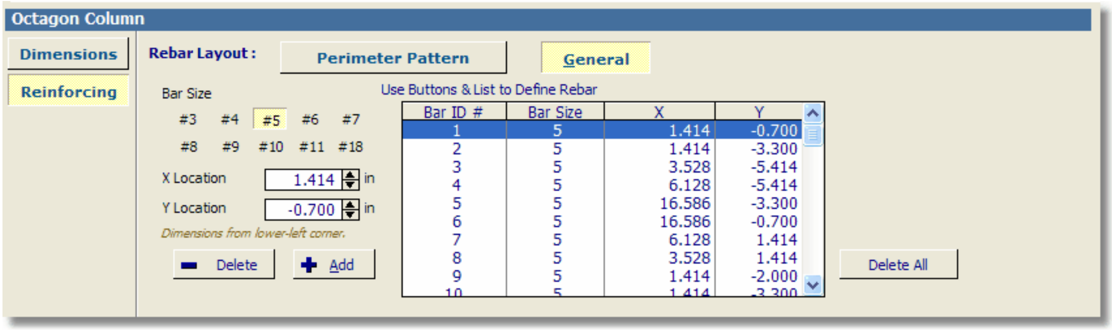




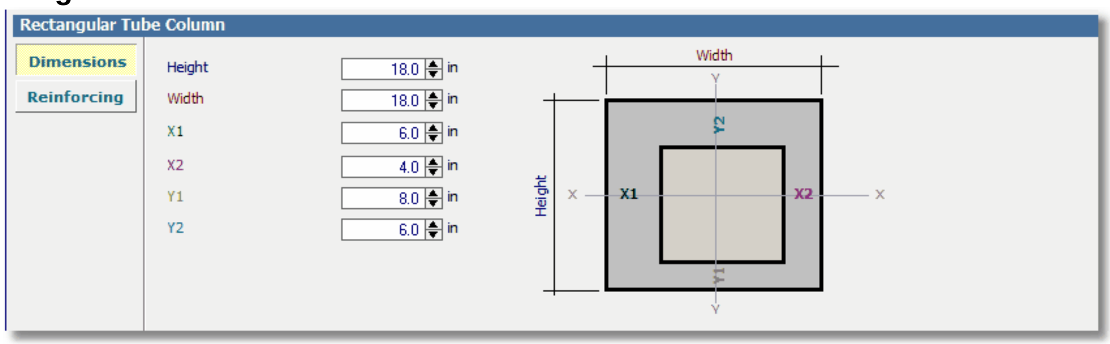
Octagon Column

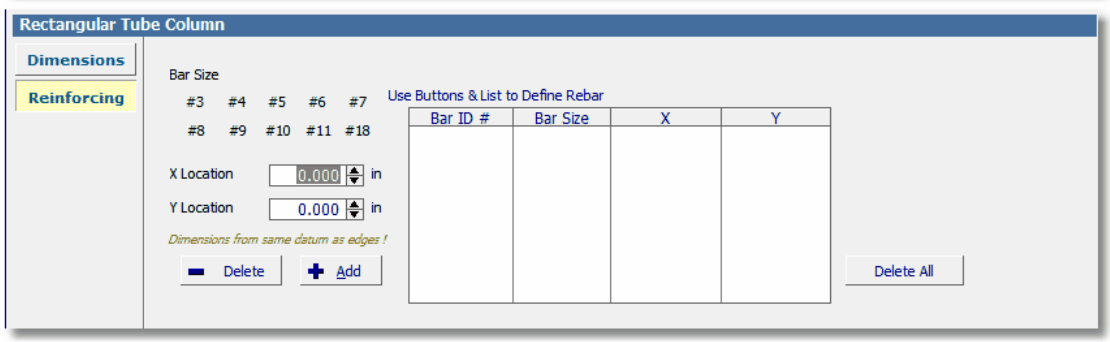




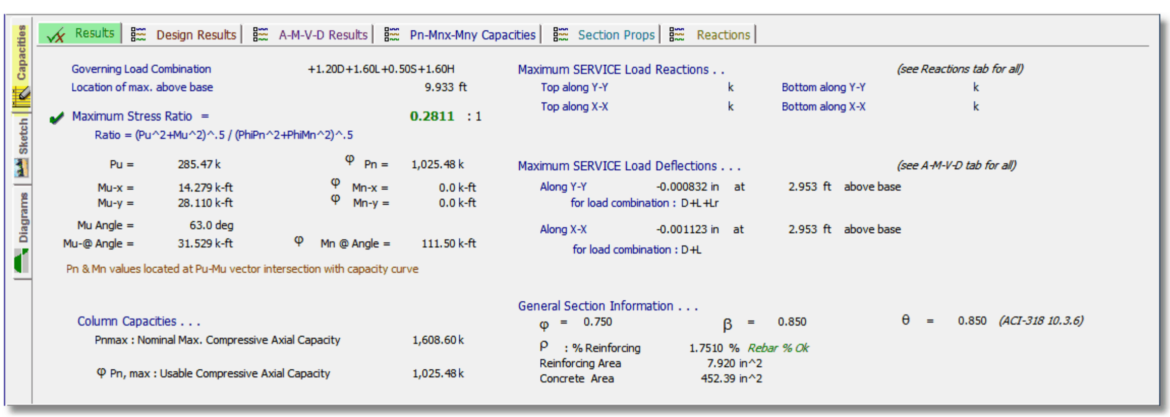


Rectangular Tube Column

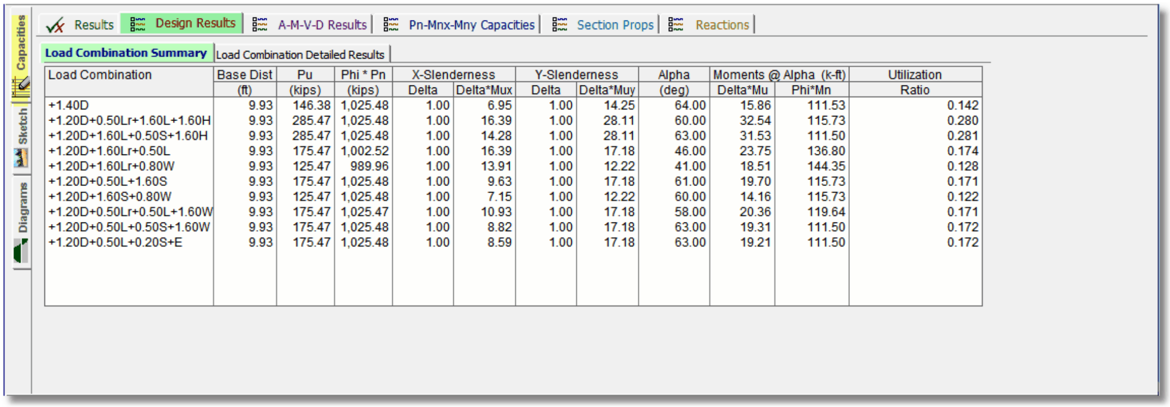




Results Tab

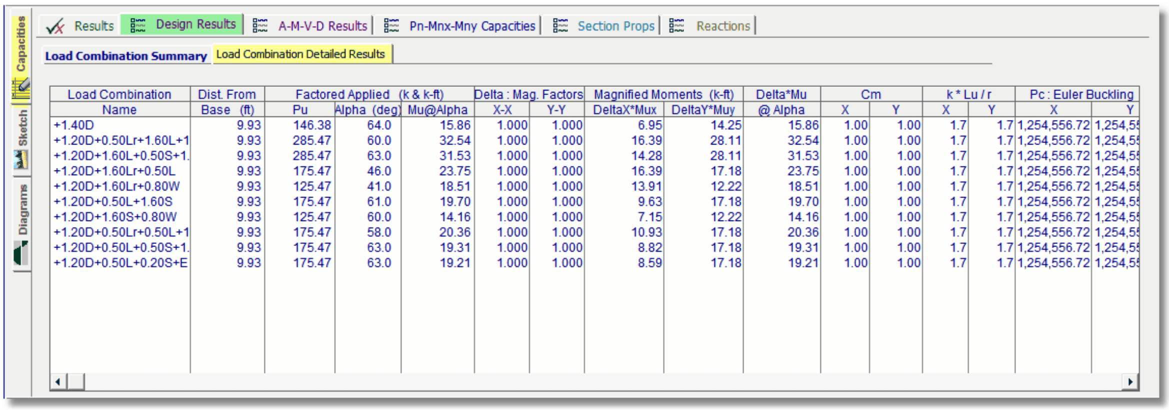


Design Results Tab - Load Combination Summary



Load Combination	Base Dist	Pu	Phi * Pn	X-Slenderness		Y-Slenderness		Alpha	Moments @ Alpha (k-ft)		Utilization
	(ft)	(kips)	(kips)	Delta	Delta*Mux	Delta	Delta*Muy	(deg)	Delta*Mu	Phi*Mn	Ratio
+1.40D	9.93	146.38	1,025.48	1.00	6.95	1.00	14.25	64.00	15.86	111.53	0.142
+1.20D+0.50Lr+1.60L+1.60H	9.93	285.47	1,025.48	1.00	16.39	1.00	28.11	60.00	32.54	115.73	0.280
+1.20D+1.60L+0.50S+1.60H	9.93	285.47	1,025.48	1.00	14.28	1.00	28.11	63.00	31.53	111.50	0.281
+1.20D+1.60Lr+0.50L	9.93	175.47	1,002.52	1.00	16.39	1.00	17.18	46.00	23.75	136.80	0.174
+1.20D+1.60Lr+0.80W	9.93	125.47	989.96	1.00	13.91	1.00	12.22	41.00	18.51	144.35	0.128
+1.20D+0.50L+1.60S	9.93	175.47	1,025.48	1.00	9.63	1.00	17.18	61.00	19.70	115.73	0.171
+1.20D+1.60S+0.80W	9.93	125.47	1,025.48	1.00	7.15	1.00	12.22	60.00	14.16	115.73	0.122
+1.20D+0.50Lr+0.50L+1.60W	9.93	175.47	1,025.47	1.00	10.93	1.00	17.18	58.00	20.36	119.64	0.171
+1.20D+0.50L+0.50S+1.60W	9.93	175.47	1,025.48	1.00	8.82	1.00	17.18	63.00	19.31	111.50	0.172
+1.20D+0.50L+0.20S+E	9.93	175.47	1,025.48	1.00	8.59	1.00	17.18	63.00	19.21	111.50	0.172

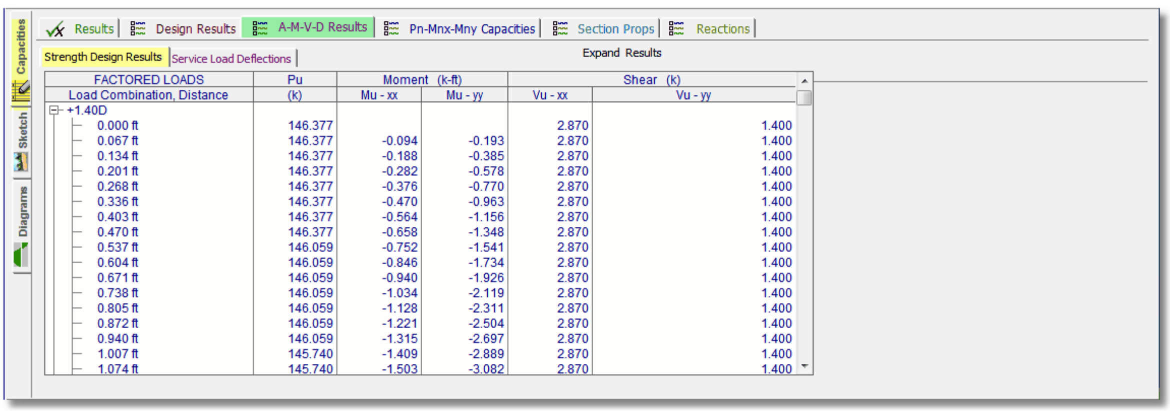
Design Results Tab - Load Combination Detailed Results



Load Combination	Dist. From	Factored Applied (k & k-ft)			Delta : Mag. Factors		Magnified Moments (k-ft)		Delta*Mu	Cm		k * Lu / r		Pc : Euler Buckling	
Name	Base (ft)	Pu	Alpha (deg)	Mu@Alpha	X-X	Y-Y	DeltaX*Mux	DeltaY*Muy	@ Alpha	X	Y	X	Y	X	Y
+1.40D	9.93	146.38	64.0	15.86	1.000	1.000	6.95	14.25	15.86	1.00	1.00	1.7	1.7	1,254,556.72	1,254,5
+1.20D+0.50Lr+1.60L+1	9.93	285.47	60.0	32.54	1.000	1.000	16.39	28.11	32.54	1.00	1.00	1.7	1.7	1,254,556.72	1,254,5
+1.20D+1.60L+0.50S+1.	9.93	285.47	63.0	31.53	1.000	1.000	14.28	28.11	31.53	1.00	1.00	1.7	1.7	1,254,556.72	1,254,5
+1.20D+1.60Lr+0.50L	9.93	175.47	46.0	23.75	1.000	1.000	16.39	17.18	23.75	1.00	1.00	1.7	1.7	1,254,556.72	1,254,5
+1.20D+1.60Lr+0.80W	9.93	125.47	41.0	18.51	1.000	1.000	13.91	12.22	18.51	1.00	1.00	1.7	1.7	1,254,556.72	1,254,5
+1.20D+0.50L+1.60S	9.93	175.47	61.0	19.70	1.000	1.000	9.63	17.18	19.70	1.00	1.00	1.7	1.7	1,254,556.72	1,254,5
+1.20D+1.60S+0.80W	9.93	125.47	60.0	14.16	1.000	1.000	7.15	12.22	14.16	1.00	1.00	1.7	1.7	1,254,556.72	1,254,5
+1.20D+0.50Lr+0.50L+1	9.93	175.47	58.0	20.36	1.000	1.000	10.93	17.18	20.36	1.00	1.00	1.7	1.7	1,254,556.72	1,254,5
+1.20D+0.50L+0.50S+1.	9.93	175.47	63.0	19.31	1.000	1.000	8.82	17.18	19.31	1.00	1.00	1.7	1.7	1,254,556.72	1,254,5
+1.20D+0.50L+0.20S+E	9.93	175.47	63.0	19.21	1.000	1.000	8.59	17.18	19.21	1.00	1.00	1.7	1.7	1,254,556.72	1,254,5

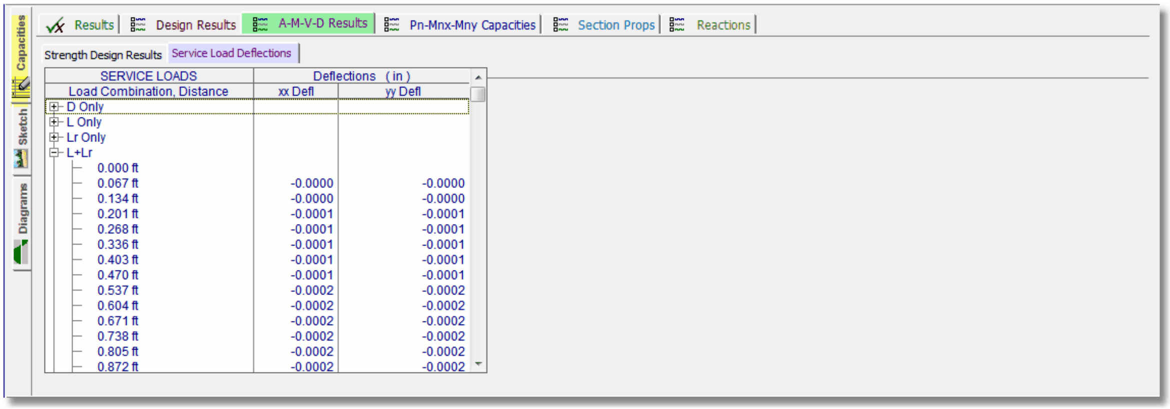
Note: The value of Euler Buckling load (Pc) is calculated using the following formula from ACI 318-11 Section R10.10.6.2: $EI = 0.25E_cI_g$.

A-M-V-D Results - Strength Design Results



FACTORED LOADS	Pu	Moment (k-ft)		Shear (k)	
Load Combination, Distance	(k)	Mu - xx	Mu - yy	Vu - xx	Vu - yy
+1.40D					
0.000 ft	146.377			2.870	1.400
0.067 ft	146.377	-0.094	-0.193	2.870	1.400
0.134 ft	146.377	-0.188	-0.385	2.870	1.400
0.201 ft	146.377	-0.282	-0.578	2.870	1.400
0.268 ft	146.377	-0.376	-0.770	2.870	1.400
0.336 ft	146.377	-0.470	-0.963	2.870	1.400
0.403 ft	146.377	-0.564	-1.156	2.870	1.400
0.470 ft	146.377	-0.658	-1.348	2.870	1.400
0.537 ft	146.059	-0.752	-1.541	2.870	1.400
0.604 ft	146.059	-0.846	-1.734	2.870	1.400
0.671 ft	146.059	-0.940	-1.926	2.870	1.400
0.738 ft	146.059	-1.034	-2.119	2.870	1.400
0.805 ft	146.059	-1.128	-2.311	2.870	1.400
0.872 ft	146.059	-1.221	-2.504	2.870	1.400
0.940 ft	146.059	-1.315	-2.697	2.870	1.400
1.007 ft	145.740	-1.409	-2.889	2.870	1.400
1.074 ft	145.740	-1.503	-3.082	2.870	1.400

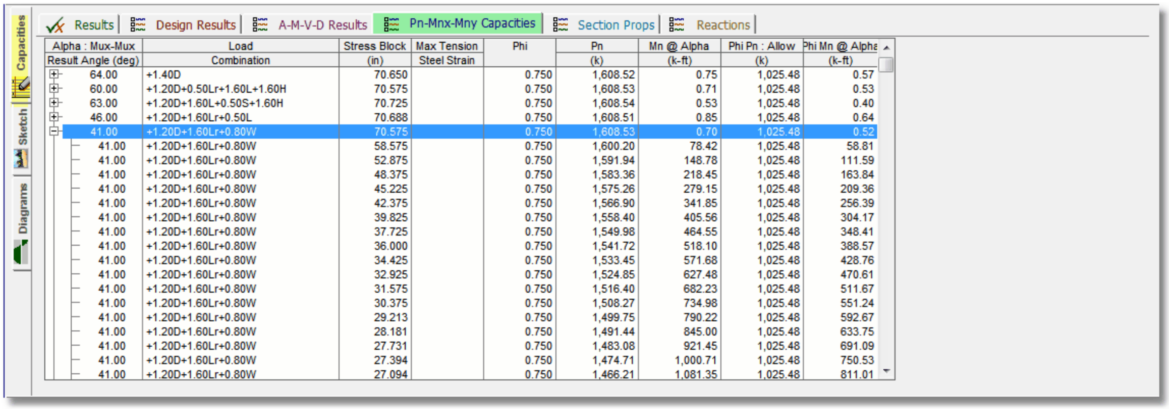
A-M-V-D Results - Service Load Deflections



SERVICE LOADS	Deflections (in)	
Load Combination, Distance	xx Defl	yy Defl
⊞ D Only		
⊞ L Only		
⊞ Lr Only		
⊟ L+Lr		
0.000 ft		
0.067 ft	-0.0000	-0.0000
0.134 ft	-0.0000	-0.0000
0.201 ft	-0.0001	-0.0001
0.268 ft	-0.0001	-0.0001
0.336 ft	-0.0001	-0.0001
0.403 ft	-0.0001	-0.0001
0.470 ft	-0.0001	-0.0001
0.537 ft	-0.0002	-0.0002
0.604 ft	-0.0002	-0.0002
0.671 ft	-0.0002	-0.0002
0.738 ft	-0.0002	-0.0002
0.805 ft	-0.0002	-0.0002
0.872 ft	-0.0002	-0.0002

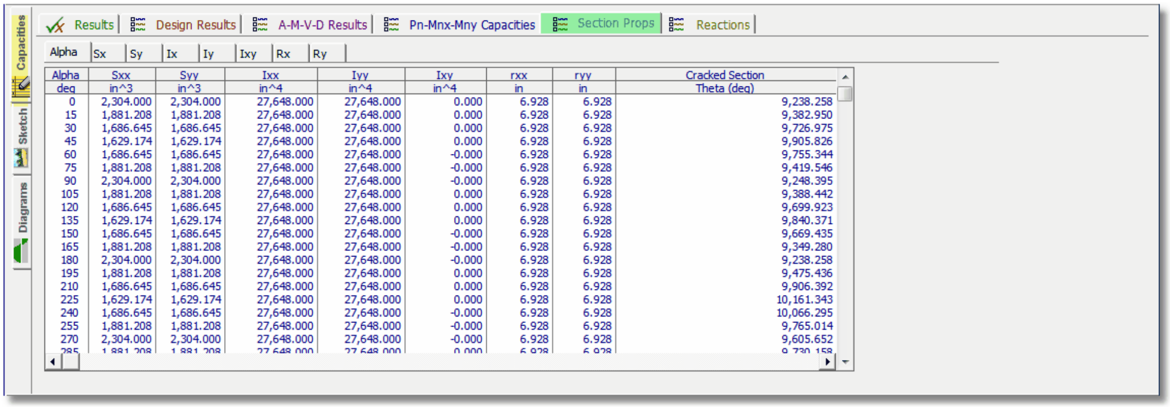
Note: Deflections are based on Ig.

P-Mx-My Capacities



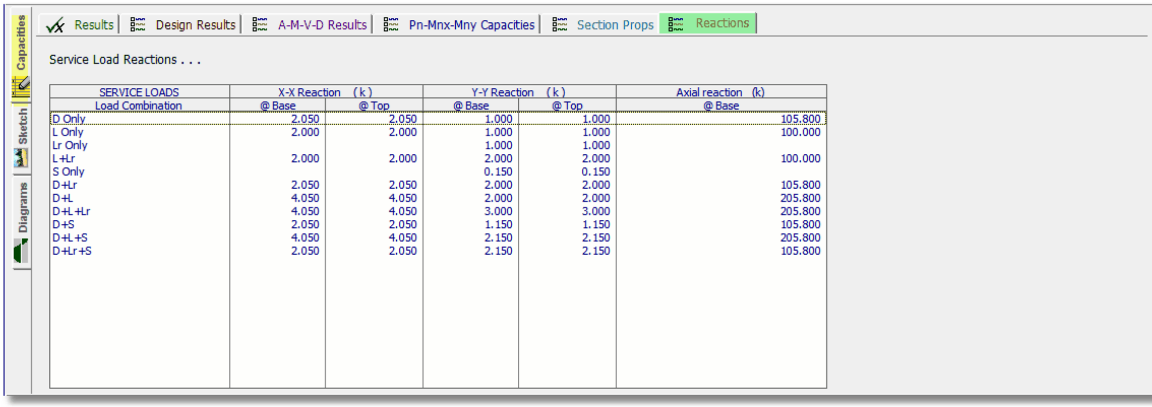
Alpha : Mux-Mux Result Angle (deg)	Load Combination	Stress Block (in)	Max Tension Steel Strain	Phi	Pn (k)	Mn @ Alpha (k-ft)	Phi Pn : Allow (k)	Phi Mn @ Alpha (k-ft)
64.00	+1.40D	70.650		0.750	1,608.52	0.75	1,025.48	0.57
60.00	+1.20D+0.50Lr+1.60L+1.60H	70.575		0.750	1,608.53	0.71	1,025.48	0.53
63.00	+1.20D+1.60L+0.50S+1.60H	70.725		0.750	1,608.54	0.53	1,025.48	0.40
46.00	+1.20D+1.60Lr+0.50L	70.688		0.750	1,608.51	0.85	1,025.48	0.64
41.00	+1.20D+1.60Lr+0.80W	70.575		0.750	1,608.53	0.70	1,025.48	0.52
41.00	+1.20D+1.60Lr+0.80W	58.575		0.750	1,600.20	78.42	1,025.48	58.81
41.00	+1.20D+1.60Lr+0.80W	52.875		0.750	1,591.94	148.78	1,025.48	111.59
41.00	+1.20D+1.60Lr+0.80W	48.375		0.750	1,583.36	218.45	1,025.48	163.84
41.00	+1.20D+1.60Lr+0.80W	45.225		0.750	1,575.26	279.15	1,025.48	209.36
41.00	+1.20D+1.60Lr+0.80W	42.375		0.750	1,566.90	341.85	1,025.48	256.39
41.00	+1.20D+1.60Lr+0.80W	39.825		0.750	1,558.40	405.56	1,025.48	304.17
41.00	+1.20D+1.60Lr+0.80W	37.725		0.750	1,549.98	464.55	1,025.48	348.41
41.00	+1.20D+1.60Lr+0.80W	36.000		0.750	1,541.72	518.10	1,025.48	388.57
41.00	+1.20D+1.60Lr+0.80W	34.425		0.750	1,533.45	571.68	1,025.48	428.76
41.00	+1.20D+1.60Lr+0.80W	32.925		0.750	1,524.85	627.48	1,025.48	470.61
41.00	+1.20D+1.60Lr+0.80W	31.575		0.750	1,516.40	682.23	1,025.48	511.67
41.00	+1.20D+1.60Lr+0.80W	30.375		0.750	1,508.27	734.98	1,025.48	551.24
41.00	+1.20D+1.60Lr+0.80W	29.213		0.750	1,499.75	790.22	1,025.48	592.67
41.00	+1.20D+1.60Lr+0.80W	28.181		0.750	1,491.44	845.00	1,025.48	633.75
41.00	+1.20D+1.60Lr+0.80W	27.731		0.750	1,483.08	921.45	1,025.48	691.09
41.00	+1.20D+1.60Lr+0.80W	27.394		0.750	1,474.71	1,000.71	1,025.48	750.53
41.00	+1.20D+1.60Lr+0.80W	27.094		0.750	1,466.21	1,081.35	1,025.48	811.01

Section Properties



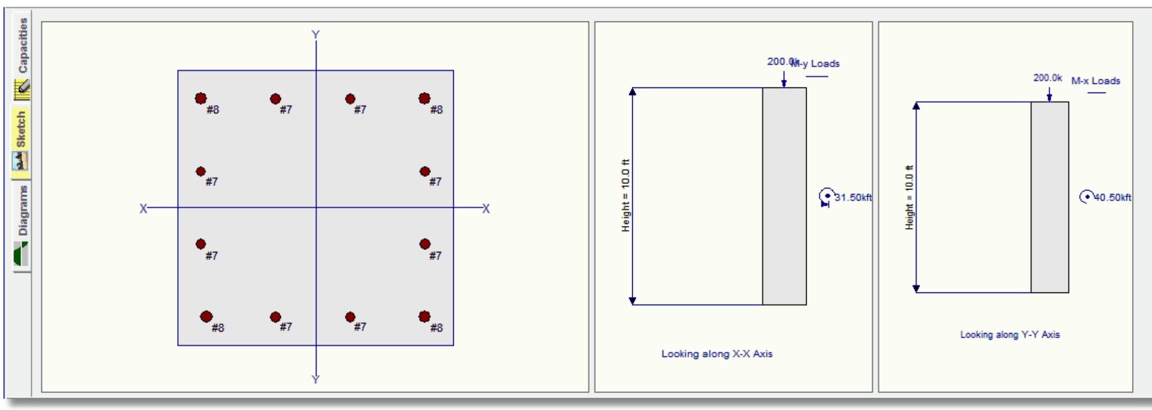
Alpha deg	Sxx in^3	Syy in^3	Ixx in^4	Iyy in^4	Ixy in^4	rxx in	ryy in	Cracked Section Theta (deg)
0	2,304.000	2,304.000	27,648.000	27,648.000	0.000	6.928	6.928	9,238.258
15	1,881.208	1,881.208	27,648.000	27,648.000	0.000	6.928	6.928	9,382.950
30	1,686.645	1,686.645	27,648.000	27,648.000	0.000	6.928	6.928	9,726.975
45	1,629.174	1,629.174	27,648.000	27,648.000	0.000	6.928	6.928	9,905.826
60	1,686.645	1,686.645	27,648.000	27,648.000	-0.000	6.928	6.928	9,755.344
75	1,881.208	1,881.208	27,648.000	27,648.000	-0.000	6.928	6.928	9,419.546
90	2,304.000	2,304.000	27,648.000	27,648.000	-0.000	6.928	6.928	9,248.395
105	1,881.208	1,881.208	27,648.000	27,648.000	0.000	6.928	6.928	9,388.442
120	1,686.645	1,686.645	27,648.000	27,648.000	0.000	6.928	6.928	9,699.923
135	1,629.174	1,629.174	27,648.000	27,648.000	0.000	6.928	6.928	9,840.371
150	1,686.645	1,686.645	27,648.000	27,648.000	-0.000	6.928	6.928	9,669.435
165	1,881.208	1,881.208	27,648.000	27,648.000	-0.000	6.928	6.928	9,349.280
180	2,304.000	2,304.000	27,648.000	27,648.000	-0.000	6.928	6.928	9,238.258
195	1,881.208	1,881.208	27,648.000	27,648.000	0.000	6.928	6.928	9,475.436
210	1,686.645	1,686.645	27,648.000	27,648.000	0.000	6.928	6.928	9,906.392
225	1,629.174	1,629.174	27,648.000	27,648.000	0.000	6.928	6.928	10,161.343
240	1,686.645	1,686.645	27,648.000	27,648.000	-0.000	6.928	6.928	10,066.295
255	1,881.208	1,881.208	27,648.000	27,648.000	-0.000	6.928	6.928	9,765.014
270	2,304.000	2,304.000	27,648.000	27,648.000	-0.000	6.928	6.928	9,605.652

Reactions

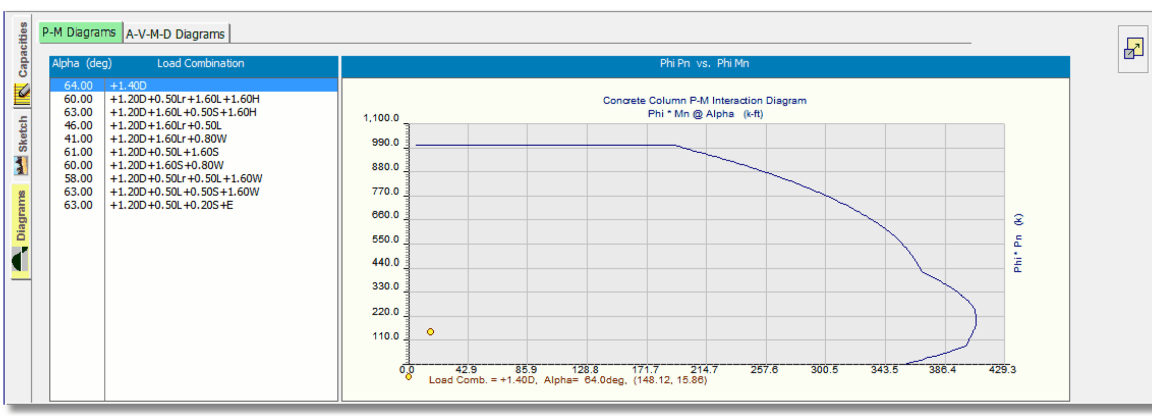


SERVICE LOADS	X-X Reaction (k)		Y-Y Reaction (k)		Axial reaction (k)
Load Combination	@ Base	@ Top	@ Base	@ Top	@ Base
D Only	2.050	2.050	1.000	1.000	105.800
L Only	2.000	2.000	1.000	1.000	100.000
Lr Only			1.000	1.000	
L+Lr	2.000	2.000	2.000	2.000	100.000
S Only			0.150	0.150	
D+Lr	2.050	2.050	2.000	2.000	105.800
D+L	4.050	4.050	2.000	2.000	205.800
D+L+Lr	4.050	4.050	3.000	3.000	205.800
D+S	2.050	2.050	1.150	1.150	105.800
D+L+S	4.050	4.050	2.150	2.150	205.800
D+Lr+S	2.050	2.050	2.150	2.150	105.800

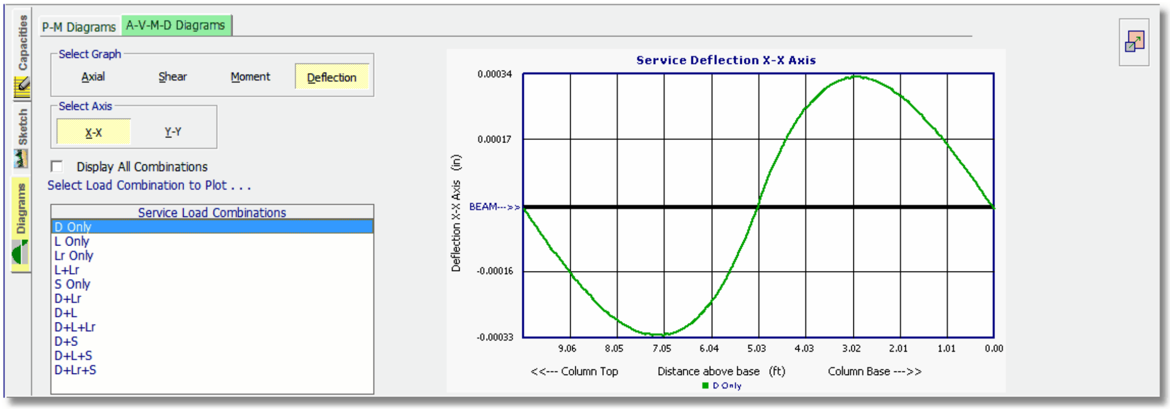
Sketch



P-M Diagrams



A-V-M-D Diagrams



Theoretical Basis

Click here to view Concrete Column Module Theoretical Basis

10.3.5 Masonry Column

Need more? Ask Us a Question

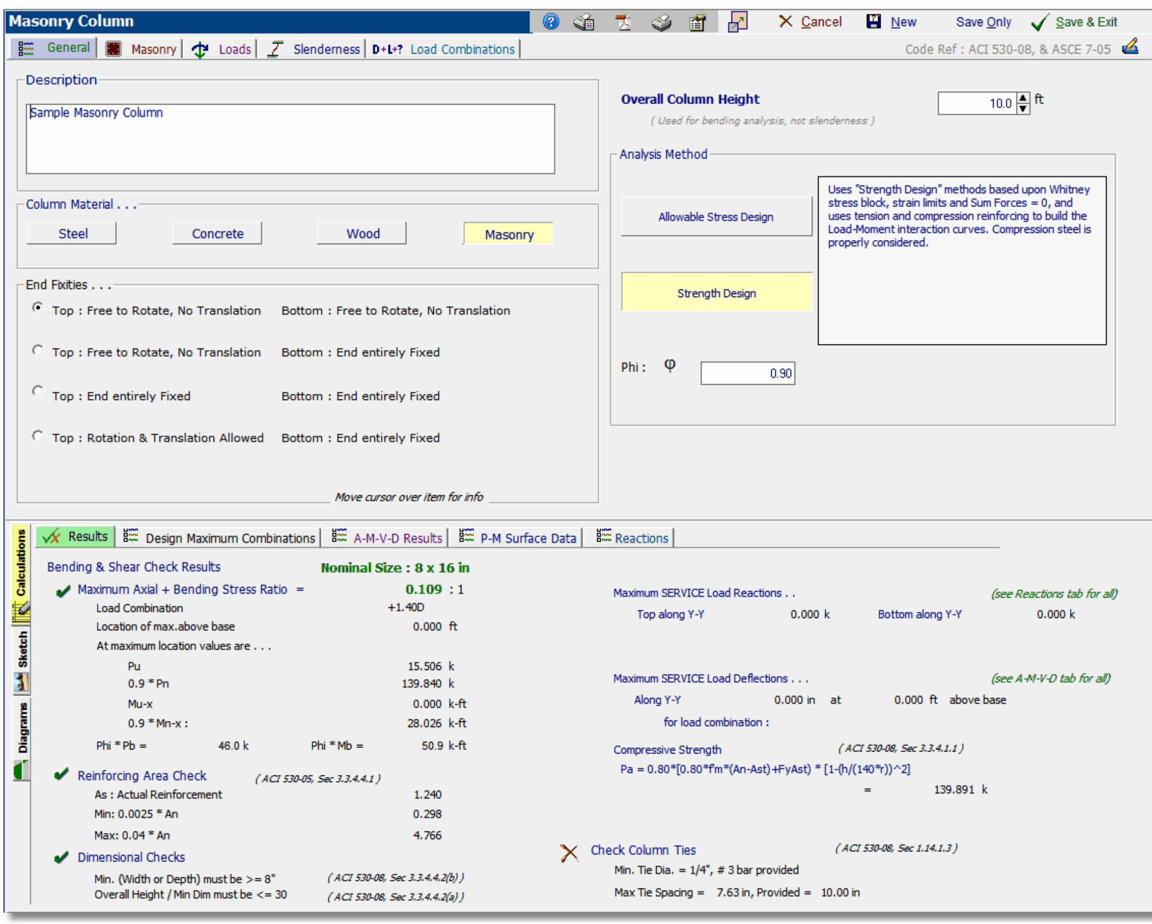
This module designs masonry columns that are subject to axial loads and lateral bending loads about one axis. Click here for a video: Masonry Column

The user can select ASD or LRFD methods.

All calculations are according to the ACI 530-05 or ACI 530-08 code, depending upon whether IBC 06 or IBC 09 has been selected as the governing building code.

The screen capture below shows the full screen for masonry column design. See items below for descriptions of items that are specific to the masonry column design module.

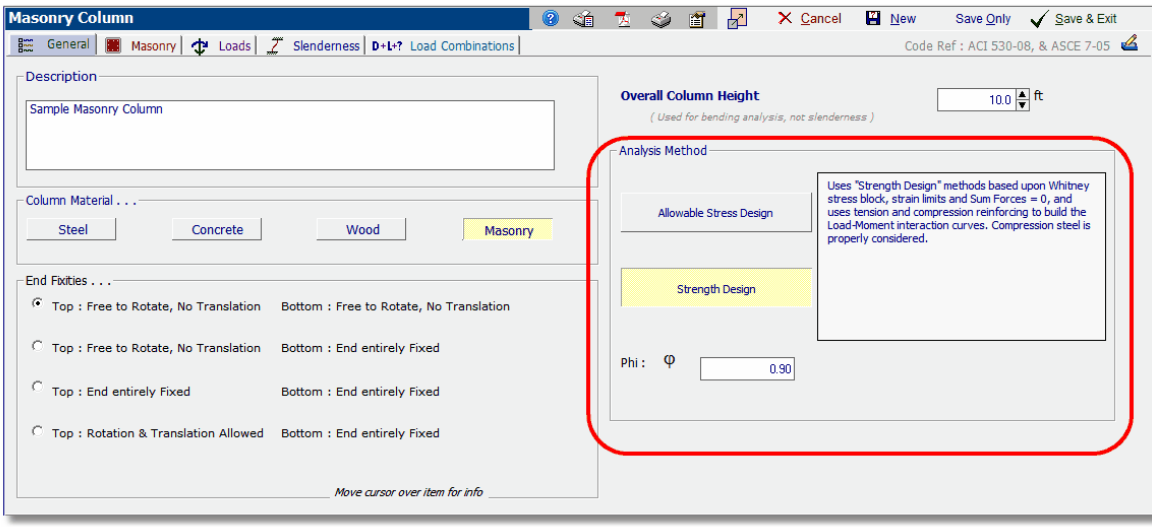
For general description of the module, end fixity, loads, and load combinations click here 326. For slenderness description click here 331.



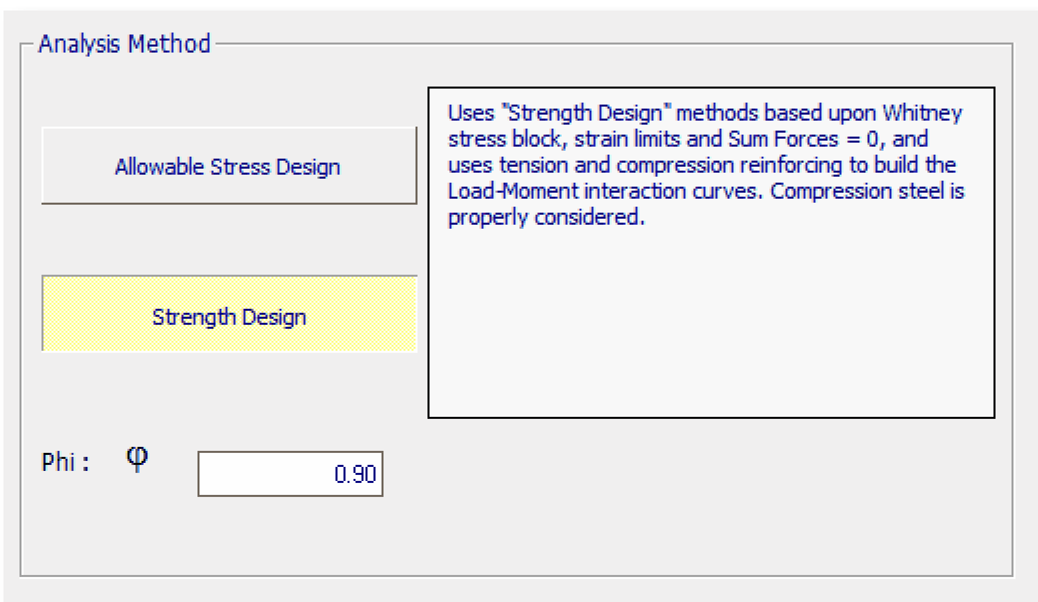
General Tab

The area circled in red below is specific to the masonry column selection. You have the choice of using Working Stress or Strength Design methods.

The column capacity is determined by creating a P-M interaction diagram, so that the effect of compressive force is included in the calculation of allowable moment capacity. For working stress this will result in significantly higher capacities than the older methods that calculated an actual stress ratio using (fa/Fa + fb/Fb).



Here is the description when the Strength Design method is selected...



Masonry Tab

All of the information on this tab is specific to masonry column design.

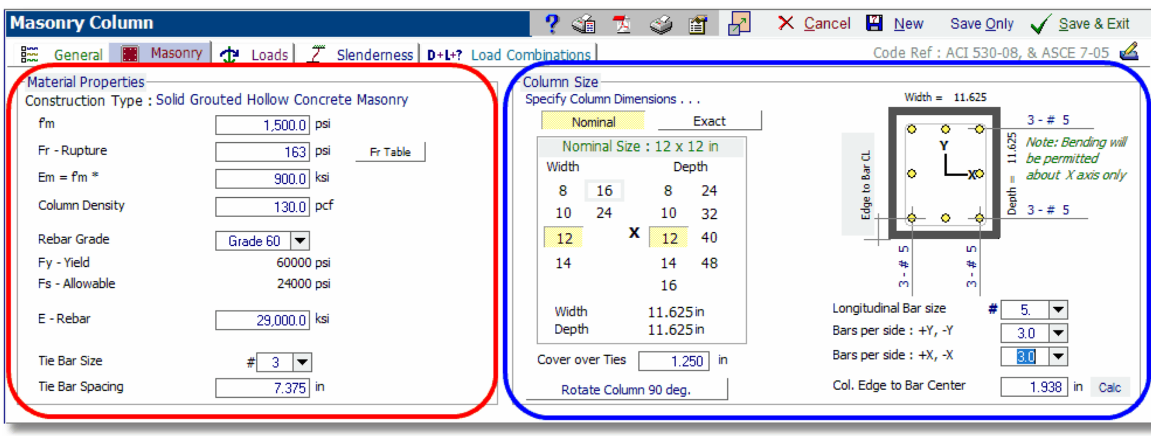
The input items in the **red** bubble define the material properties of the column and reinforcing used.

The input items in the **blue** bubble let you define the cross section size, reinforcing, and orientation of the column.

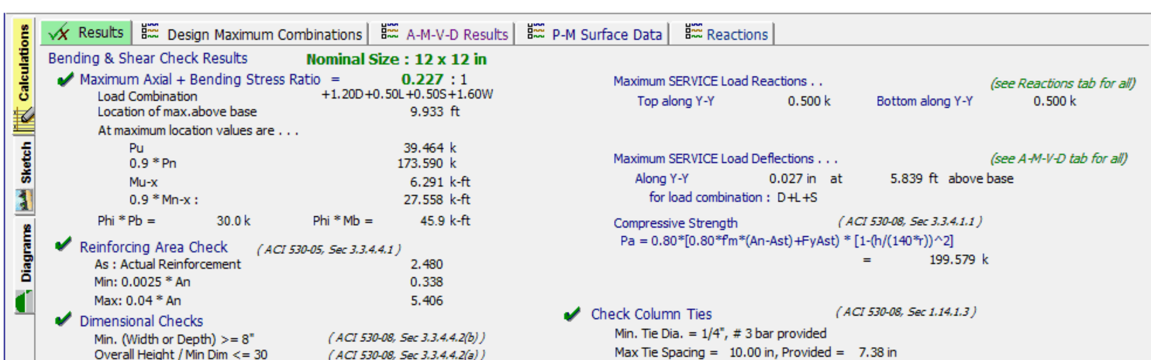
Under **Specify Column Dimensions** we've made it easy by providing buttons for the common nominal dimensions of a masonry column. Click the Width and Depth buttons and you will see the actual dimension appear in the bottom of the area. Note that "Width" is parallel to the "x-x" axis and "Depth" is parallel to the "y-y" axis.

In the lower right you can specify the bar size and bar count to be used on each face of the column.

The graphic will change accordingly as your input changes.

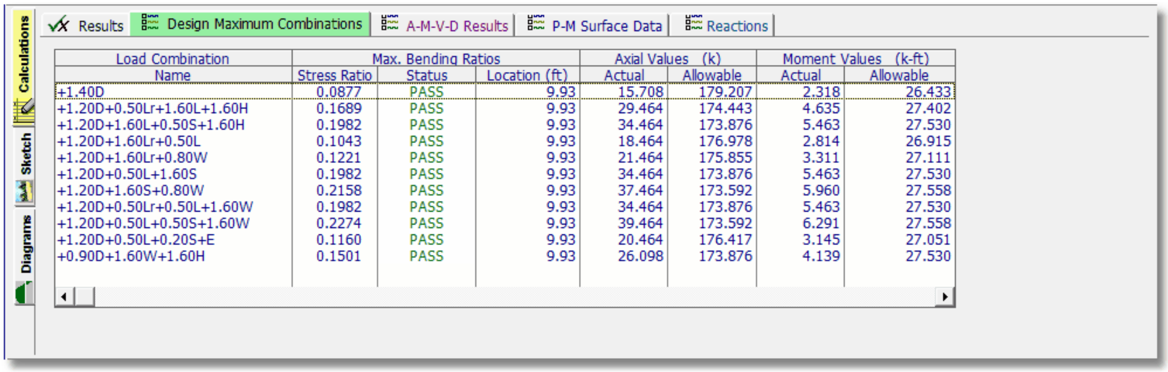


Results Tab



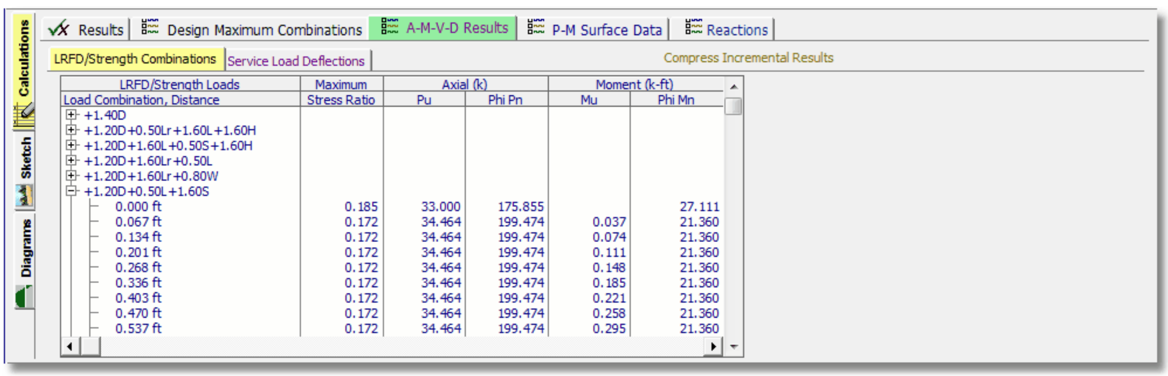
Design Maximum Combinations

This tab summarizes the maximum stress ratios for each load combination.



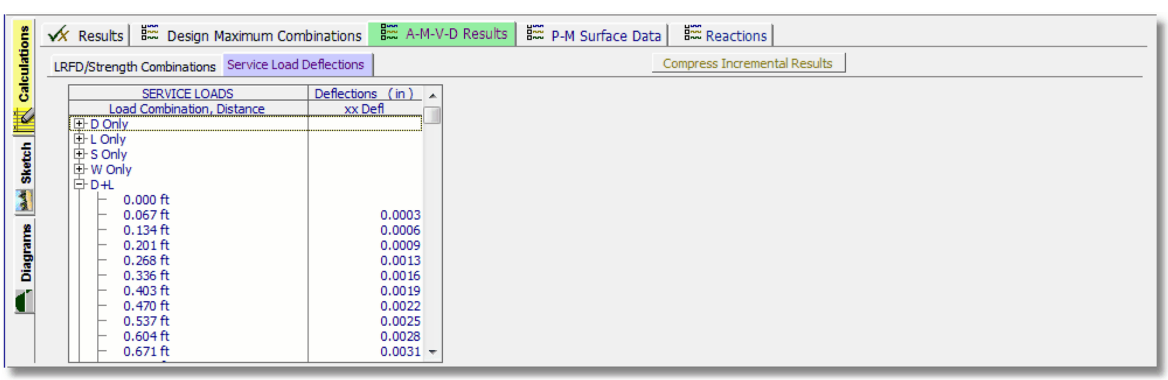
A-M-V-D Results: Strength Combinations

This tab presents the very detailed allowable and actual values for each load combination.



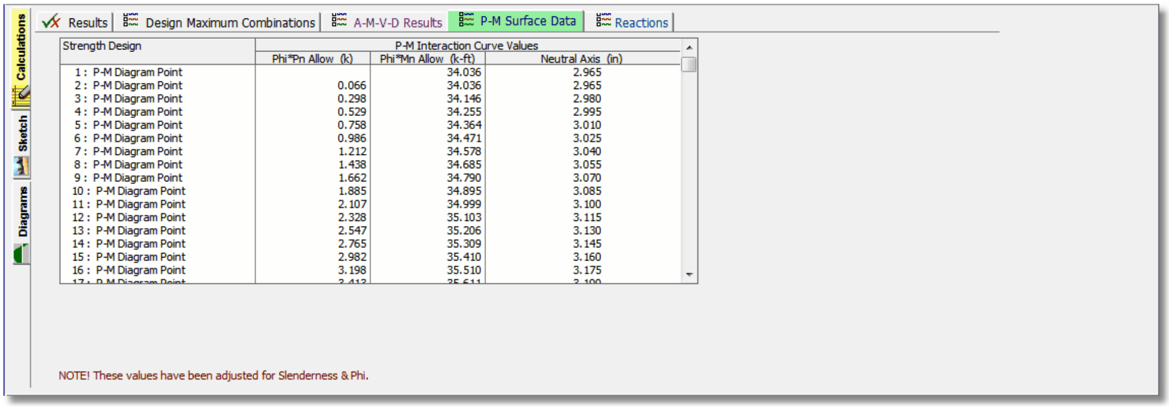
A-M-V-D Results: Service Load Deflections

This tab summarizes the lateral deflections of the column at increments along its height. These values will be nonzero only if lateral loads are applied or the axial load is applied with an eccentricity.



P-M Surface Data

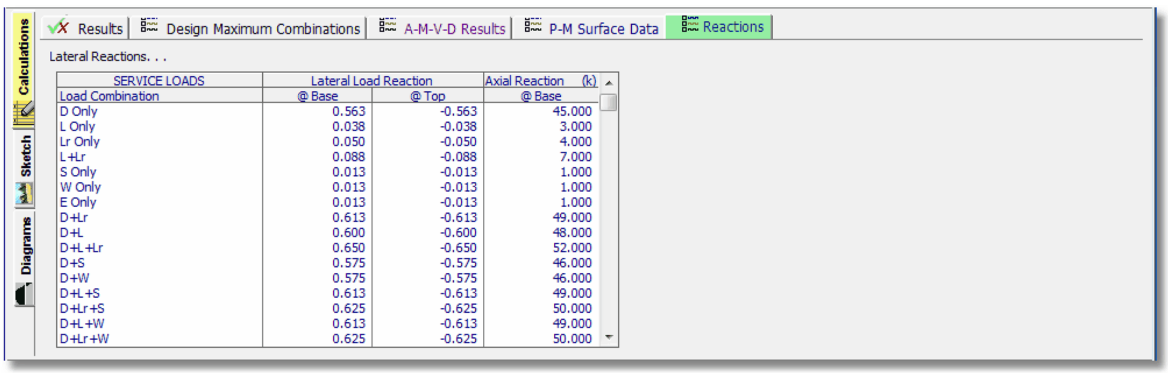
This tab lists the full analysis results for the column.



Strength Design	P-M Interaction Curve Values		
	Phi*Pn Allow (k)	Phi*Mn Allow (k-ft)	Neutral Axis (in)
1 : P-M Diagram Point		34.036	2.965
2 : P-M Diagram Point	0.066	34.036	2.965
3 : P-M Diagram Point	0.298	34.146	2.980
4 : P-M Diagram Point	0.529	34.255	2.995
5 : P-M Diagram Point	0.758	34.364	3.010
6 : P-M Diagram Point	0.986	34.471	3.025
7 : P-M Diagram Point	1.212	34.578	3.040
8 : P-M Diagram Point	1.438	34.685	3.055
9 : P-M Diagram Point	1.662	34.790	3.070
10 : P-M Diagram Point	1.885	34.895	3.085
11 : P-M Diagram Point	2.107	34.999	3.100
12 : P-M Diagram Point	2.328	35.103	3.115
13 : P-M Diagram Point	2.547	35.206	3.130
14 : P-M Diagram Point	2.765	35.309	3.145
15 : P-M Diagram Point	2.982	35.410	3.160
16 : P-M Diagram Point	3.198	35.510	3.175

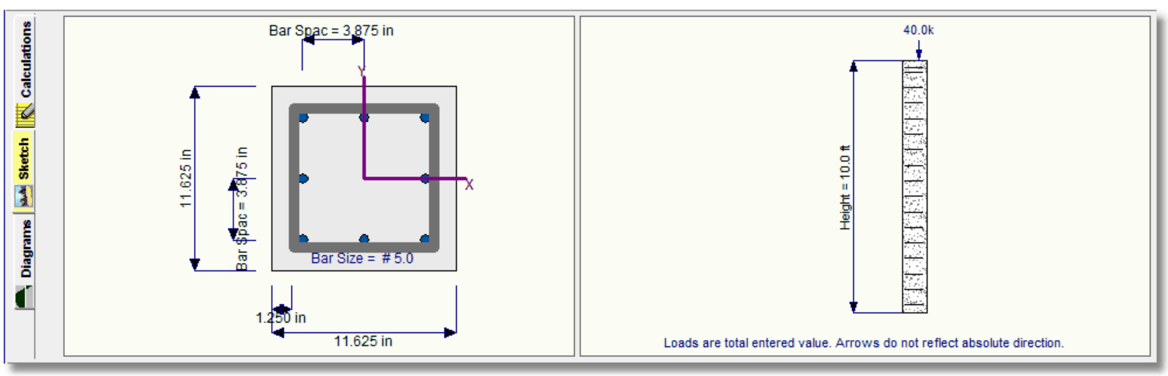
Reactions

This tab reports axial reactions and lateral reactions due to lateral loads or due to axial loads that have an eccentricity.

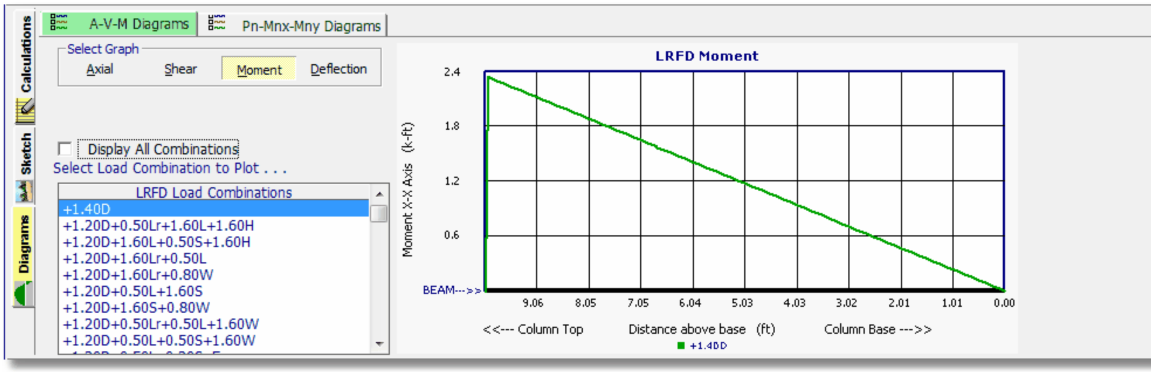


SERVICE LOADS	Lateral Load Reaction		Axial Reaction (k)
Load Combination	@ Base	@ Top	@ Base
D Only	0.563	-0.563	45.000
L Only	0.038	-0.038	3.000
Lr Only	0.050	-0.050	4.000
L+Lr	0.088	-0.088	7.000
S Only	0.013	-0.013	1.000
W Only	0.013	-0.013	1.000
E Only	0.013	-0.013	1.000
D+Lr	0.613	-0.613	49.000
D+L	0.600	-0.600	48.000
D+L+Lr	0.650	-0.650	52.000
D+S	0.575	-0.575	46.000
D+W	0.575	-0.575	46.000
D+L+S	0.613	-0.613	49.000
D+Lr+S	0.625	-0.625	50.000
D+L+W	0.613	-0.613	49.000
D+Lr+W	0.625	-0.625	50.000

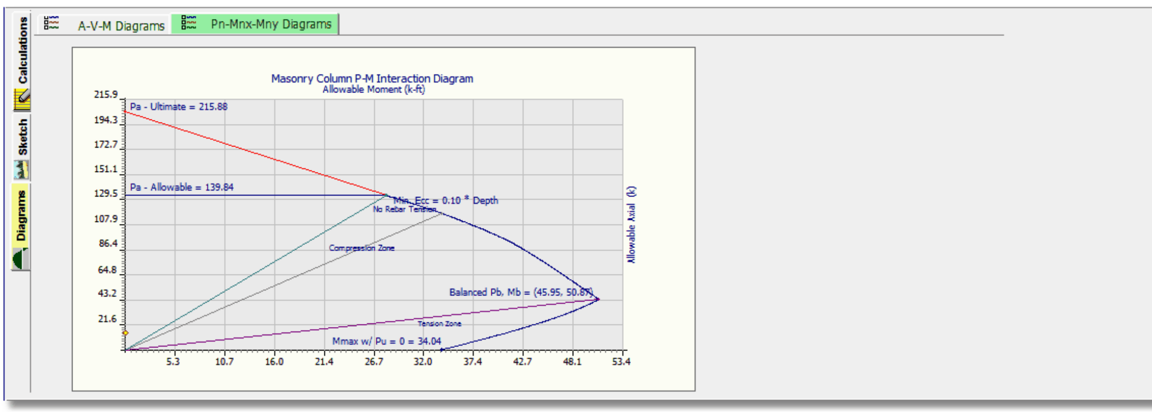
Sketch



A-V-M Diagrams



Pn-Mnx-Mny Diagrams



10.4 General Walls

General Introductory Video:

Click Here for Video

10.4.1 Concrete Slender Wall

Need more? Ask Us a Question

This program provides analysis of concrete wall panels that stand vertically and have applied vertical and out-of-plane lateral loads. Click here for a video: Concrete Slender Wall

The wall panel is analyzed using strength design procedures. Out-of-plane moments within the wall are created by eccentric axial loads, applied lateral loads, lateral self weight loads and moments induced due to the wall weight acting at an eccentricity when it deflects (P-Delta moments).

The ACI slender wall procedure, introduced in ACI 318-99, was first adopted by the IBC 2000 and subsequent code editions. As quoted in ACI 318R-05 Commentary, Section 14.8 is based on the corresponding requirements in 1997 UBC and experimental research presented in the 1982 "Test Report by SCCACI-SEAOSC". Analytical study of the current IBC/ACI provisions for concrete wall panels showed the ACI procedure does not correspond to a bilinear load-deflection characteristic observed in the SEAOSC tests and significantly underestimates the service load deflection.

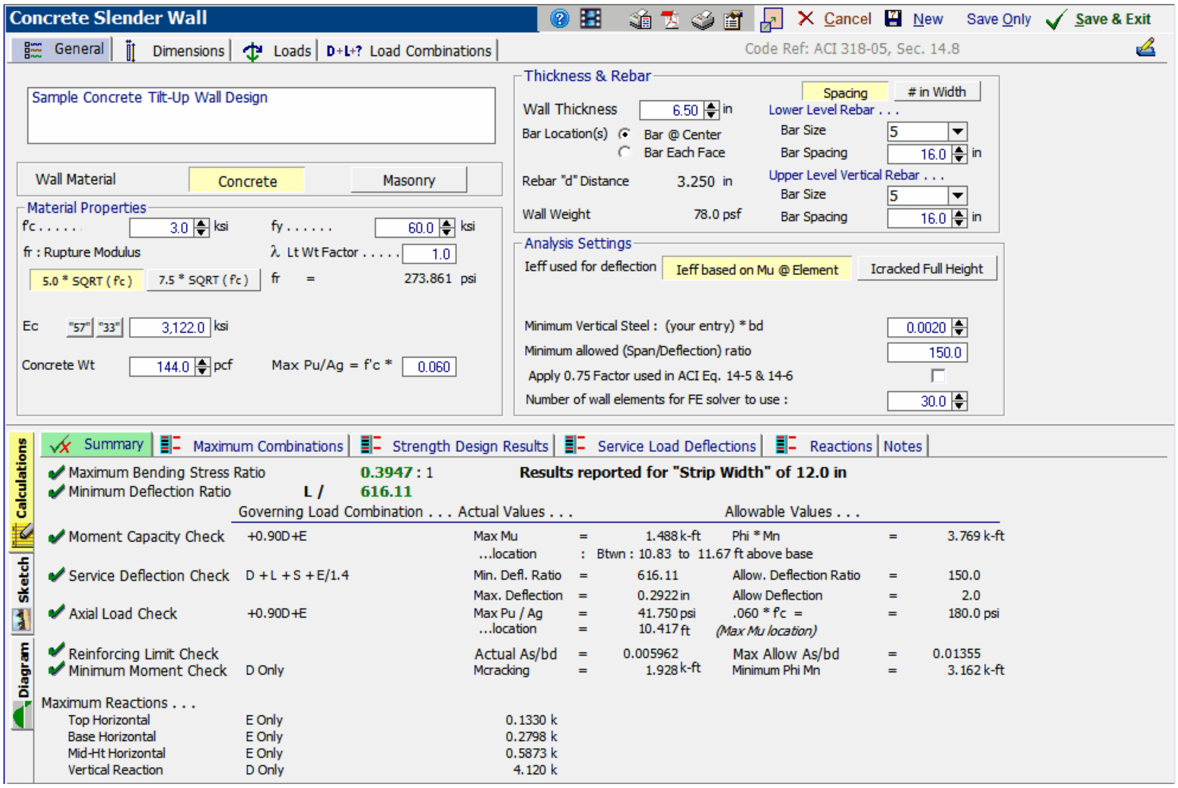
The ENERCALC Concrete Slender Wall program uses basic principles of structural mechanics to model the wall as a series of beam segments. For each segment the actual moment is used to calculate member stiffness using the $I_{effective}$ equations developed by Peter H. Bischoff. Since these changes to wall stiffness affect the wall deflection profile, the program performs an iterative analysis of calculating moments (including P-Delta effects). The results are deflection curves almost exactly matching the SCCACI-SEAOSC test results. This makes this program far more accurate at calculating wall deflections and P-Delta effects than the simple equations in the ACI code.

Capabilities

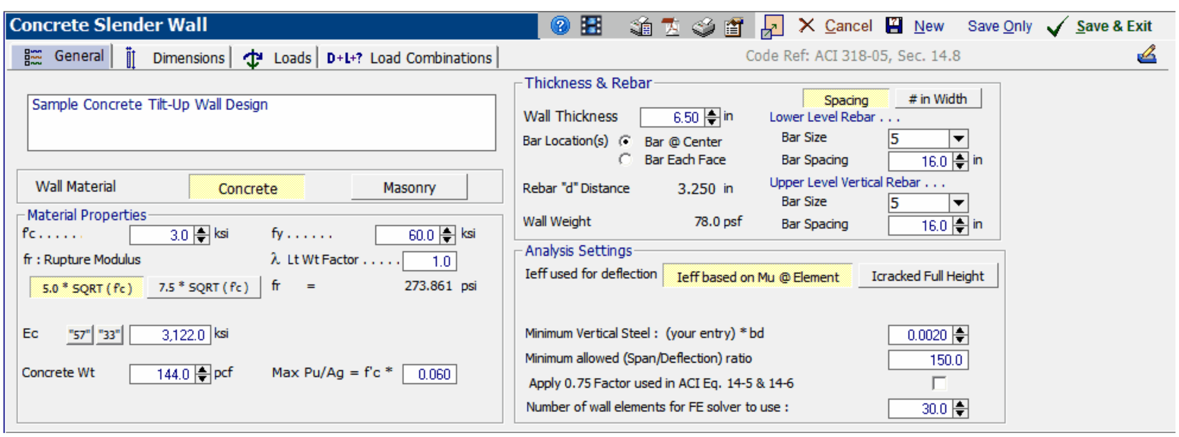
This module provides these capabilities:

- One or two story slender tilt-up concrete walls
- Iterative process accounts for P-delta
- Optional parapet
- Axial loads with optional eccentricities
- Wind, seismic and user defined lateral loads creating bending on the wall panel
- Variable strip width to model the wall panel
- Temperature differential can be specified across thickness of wall to add curvature
- Rebar location at center of wall or two layers of reinforcing at each side
- Bottom of wall can be fixed or pinned for moment resistance
- Top of wall can be pinned or free
- A reveal can be defined and cross section properties modified for reduced thickness and optionally add rebar

- The effects of wall openings can be addressed by modeling the solid panel between or adjacent to openings and then using superposition to apply the loads above and below openings.



General Tab



Material Properties

f'c

28-day compressive strength of the concrete.

Fy

Yield point stress of reinforcing

Fr: Rupture Modulus

Multiplier used in the expression to define the modulus of rupture for the concrete. 5.0 is the original recommended multiplier that was developed as a result of the SEAOSC slender wall tests of the early 1980s. 7.5 is the multiplier provided by ACI 318-05, -08, and -11.

Lambda

Factor to account for lightweight concrete.

Ec

Modulus of elasticity of concrete. You can enter the value or click the
[**"57"**] button to set Ec = 57000 * sqrt(f'c), or click the
[**"33"**] button to set Ec = 33 * sqrt(f'c) * $\text{ConcWeight}^{1.5}$.

Concrete Wt

Weight of concrete in pounds per cubic foot.

Max Pu/Ag = f'c * <entry>

Enter a multiplier less than 1.0 which will be applied to f'c to determine the maximum allowable factored axial stress.

Thickness & Rebar

Wall Thickness

Total wall thickness

Bar Location

You can select bar placement at the center of the wall thickness or at each face. When selecting "Bar Each Face" the module performs calculations considering both bars.

Rebar "d" Distance

Enter the distance between the outside surface of the wall to the centerline of the rebar. For bars each face this measurement can be from either face.

Wall Weight

The internally calculated wall weight considering the concrete weight and wall thickness entered.

Rebar [Spacing] / [# in Width]

These two options indicate how you will specify the rebar quantity in your design strip.

[**Spacing**] will change the entry so that you can enter a spacing in inches for the rebar.

[**# in Width**] changes the entry so you can enter the number of bars in your design strip, where the width of the design strip is entered on the Dimensions tab.

Note: When using the "Bars Each Face" option, the "# Bars in Width" specifies the number of bars **on each face** within the design strip width.

Bar Size

Enter the US customary rebar size number.

Analysis Settings

Ieff used for Deflection

The module offers the option to use $I_{effective}$ based on the moment in the individual wall elements or to use $I_{cracked}$ for the full height of the wall.

Temperature Differential across thickness

This input is used to describe the temperature change between each face of the wall. A temperature change induces a slight curvature into the wall because the hotter side expands, resulting in a slightly higher out-of-plane deflection.

Minimum Vertical Steel: %/100

Minimum steel as a percentage of the gross wall area.

Minimum Allowed (Span/Deflection) Ratio

This setting establishes the minimum allowable ratio of span length to service load deflection. If a lower actual Span/Deflection ratio occurs (meaning greater deflection), a warning message will be displayed.

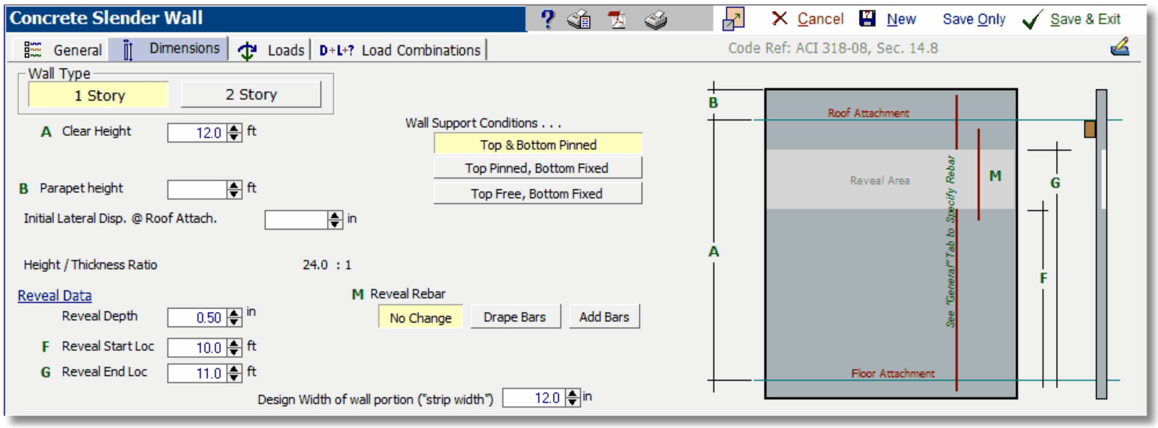
Apply 0.75 Factor used in ACI Eq. 14-5 & 14-6

Please see the code reference for an explanation of this factor. It is typically not used in this module because it is a calibration factor used to curve fit deflection calculations with ACI approximate formulas.

Number of wall elements for FE solver to use

This module divides the wall design strip into segments from the base to the top for analysis purposes. Use this entry to define the number of segments to use. Experience demonstrates that approximately 30 segments gives a good balance between the iterative P-Delta analysis reaching convergence and excessive calculation time.

Dimensions Tab



Fixity Conditions

Controls how the top and bottom of the wall are restrained for moments and lateral movement.

[Top & Bottom Pinned]

Base of wall is restrained against movement out of plane and vertically, rotates freely. Top of wall restrained against out of plane movement and can move vertically and rotate freely.

[Top Pinned, Bottom Fixed]

Base of wall is restrained against movement about all three axes. Top of wall restrained against out of plane movement and can move vertically and rotate freely.

[Top Free, Bottom Fixed]

Base of wall is restrained against movement about all three axes. Top of wall is completely free making this a cantilevered wall.

Clear Height

Span of the wall between the base and the first lateral support. For one story walls this is the top support. For 2 story walls this prompt will change to be "1st story height".

Parapet Height

Distance the wall extends (without a topmost lateral support) above the topmost lateral support (i.e. extension above the clear height for one story wall, extension above the 2nd story height for 2 story walls).

Reveal Data

A reveal is a portion of the wall that is recessed from the rest of the surface. It is formed by placing thin blockout material (typically styrofoam) in the forms prior to concrete placement. It is used to create architectural effects. The reveal reduces the structural thickness of the wall. This module calculates section properties for this reduced section in the portion of the wall where the reveal has been formed.

Reveal Depth

Depth of reveal measured from outside face of wall. A 1" reveal in a 6" wall gives a net structural thickness of 5".

Reveal Start Location, Reveal End Location

Distances measured upwards from bottom of wall that define the start and end points of the reveal.

Reveal Rebar

This selection defines how the module should consider the reveal area to be reinforced.

No Change means that the reinforcing stays where it is as defined by the "Rebar 'd' Distance" entered on the General tab. This option results in an offset rebar location within the remaining structural thickness, because the reveal takes away part of the concrete.

Drape Bars tells the module to move the rebar inward to give the same dimension between the rebar and face of wall. For walls with bars at "Center" this moves the bar to the center of the remaining structural thickness. When bars are specified on each face, this option moves only one of the bars inwards.

Add Bars enables you to add additional reinforcing in the area of the reveal. The location of the main rebar is as described in the "No Change" option above.

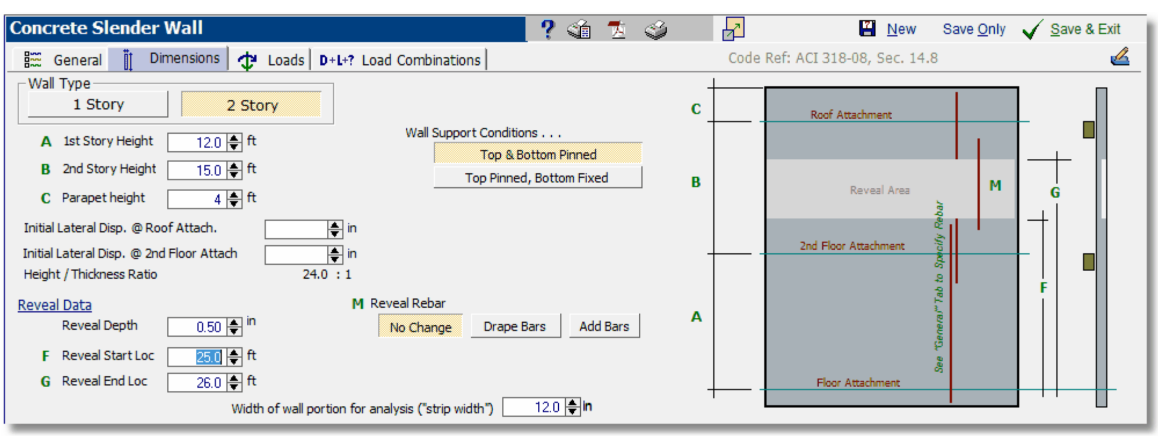
Strip Width: Width of wall portion for analysis

This module performs its analysis for this width. Results are for either this width or a 12" width as noted where the results are provided.

Note that applied loads either are applied to the entire strip width (as for concentrated vertical and lateral loads) or are entered on a per-foot basis when they are uniform loads.

Two Story...

When a two story wall is selected, this tab changes slightly to provide the 2nd story height and remove the Fixed-Free support option.



1st Story Height

Distance from the bottom of the wall to the first lateral support.

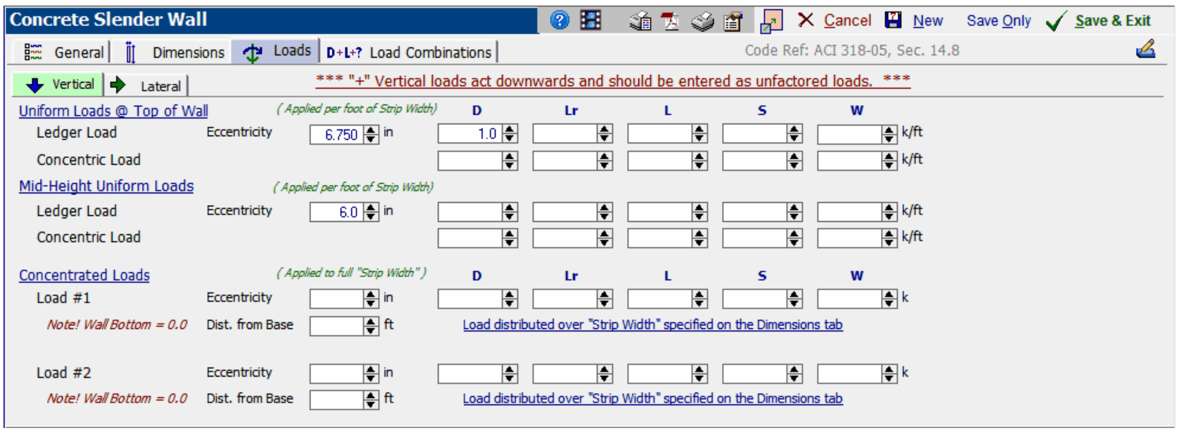
2nd Story Height

Distance from the first lateral support to the top lateral support.

Loads Tab - Vertical Loads

A variety of vertical loads are available. Note the hint in green describing whether the load is per foot or on the entire strip width.

All loads that are entered on this tab will be multiplied by the load factors specified on the Load Combination sub-tabs. So these magnitudes should be specified with those load factors in mind.



Ledger Load

This is a per-foot vertical load applied to the wall at an optional eccentricity. So if you have a 48" strip width and specify a 1 k/ft dead load then the strip will have a total of 4 kip applied due to the 1 k/ft entry.

Eccentricity

Describes an offset from the mid-thickness of the wall panel, which is the default location of application of a vertical load. Enter this value as a positive number when the load is shifted toward the inside of the wall.

Concentric Load

This is a per-foot vertical load applied concentrically to the wall. So if you have a 48" strip width and specify a 1 k/ft dead load then the strip will have a total of 4 kip applied due to the 1 k/ft entry.

Mid-Height Vertical Uniform Load

This load entry is only shown for 2-story walls. It allows you to specify two uniform loads applied at the "1st Story" height, one of which can have an eccentricity from the wall center.

Concentrated Loads

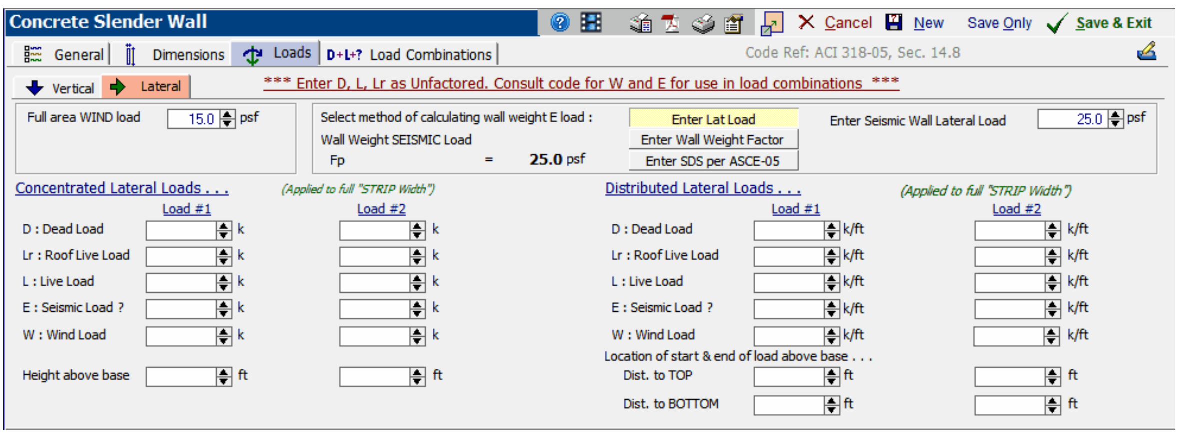
This is a single concentrated vertical load applied to the wall "strip width" with an optional eccentricity.

Distance from Base is the height at which the load is applied.

Loads Tab - Lateral Loads

Lateral loads are applied perpendicular to the plane of the wall and are almost always seismic or wind. These loads create out of plane deflection of the wall, which the module will use to develop P-Delta effects to calculate secondary moments in the wall. Recall that this module divides the wall into small segments and calculates the allowable and actual forces and deflections for each small segment. In this way the lateral loads are properly modeled on what is effectively a beam with variable stiffness due to the state of cracking in each segment.

All loads that are entered on this tab will be multiplied by the load factors specified on the Load Combination sub-tabs. So these magnitudes should be specified with those load factors in mind.



Full area WIND Load

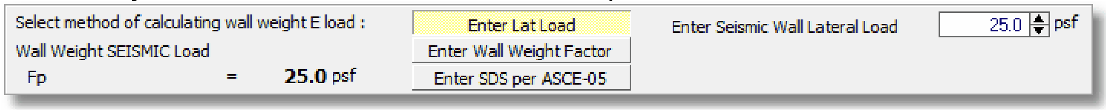
Enter the wind load that will be applied to the wall in the out-of-plane direction. This load will only be applied to one surface of the wall, and as such, the magnitude must take into consideration both the internal and external pressures.

For the purpose of defining the direction of applied wind pressures, consider that pressure will only be applied to the exterior surface of the wall. A net pressure that acts TOWARD the exterior surface of the wall should be entered as a positive value. A net pressure that acts AWAY FROM the exterior surface of the wall should be entered as a negative value.

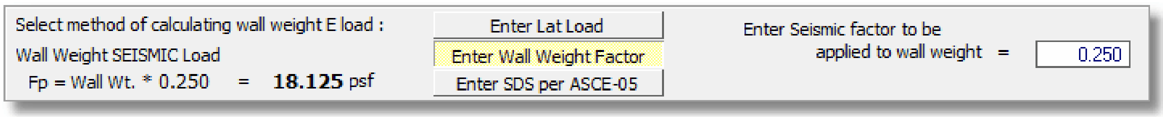
Wall Weight SEISMIC Load

This section offers three options to specify the seismic load that will be applied to the wall in the out-of-plane direction:

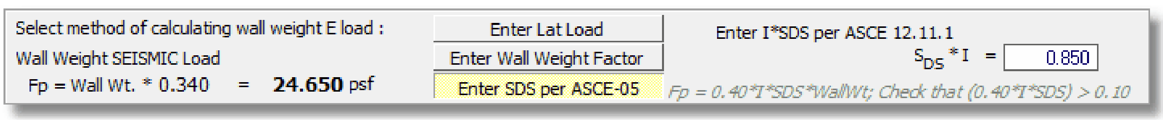
Enter Lateral Load: This entry is a simple net load applied to the wall (but will still be factored by the load combination factors for "E").



Enter Wall Weight Factor: Enter a number that will be multiplied by the self-weight of the wall. For example, if you enter 0.25 and the wall weighs 80 psf, then a 20.00 psf out-of-plane load will be calculated and applied to the wall using the load combination factors for "E".



Enter S_{DS} per ASCE-05: Enter the (S_{DS} * I) value as prescribed by the ASCE code for the building location. The minimum calculated load value of 10 psf or (0.4 * Value Entered * Wall Weight) will be applied to the wall using the load combination factors for "E".



Fp

This is the actual seismic load applied perpendicular to the plane of the wall, which represents the wall's seismic self weight load.

Concentrated Lateral Loads

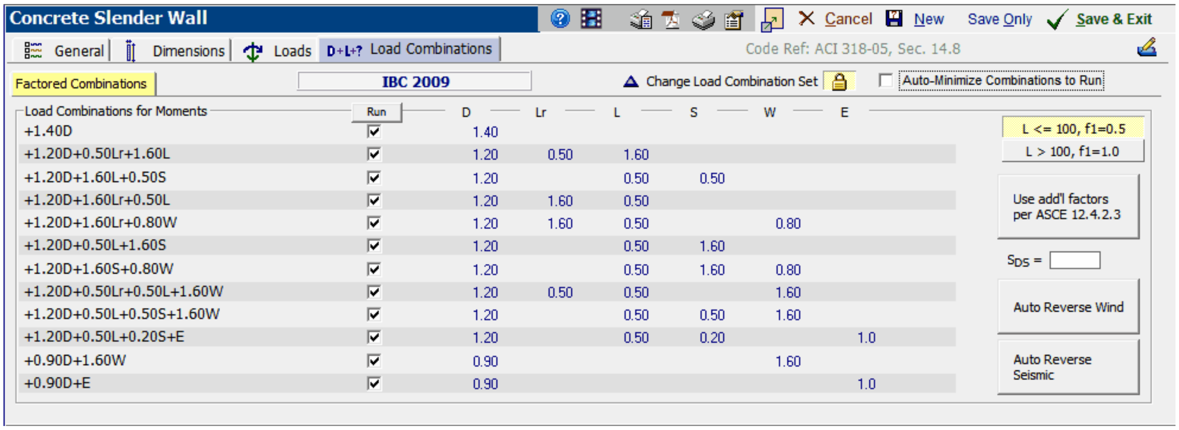
This is an added lateral load applied perpendicular to the plane of the wall. It acts on the full "Strip Width" and is factored by the load combination factors corresponding to the type of load.

Distributed Lateral Loads

This is an added lateral uniform load applied out-of-plane to the wall. It acts on the full "Strip Width" and is factored by the load combination factors corresponding to the type of load. You also enter the start and end distance of the load extent above the base of the wall.

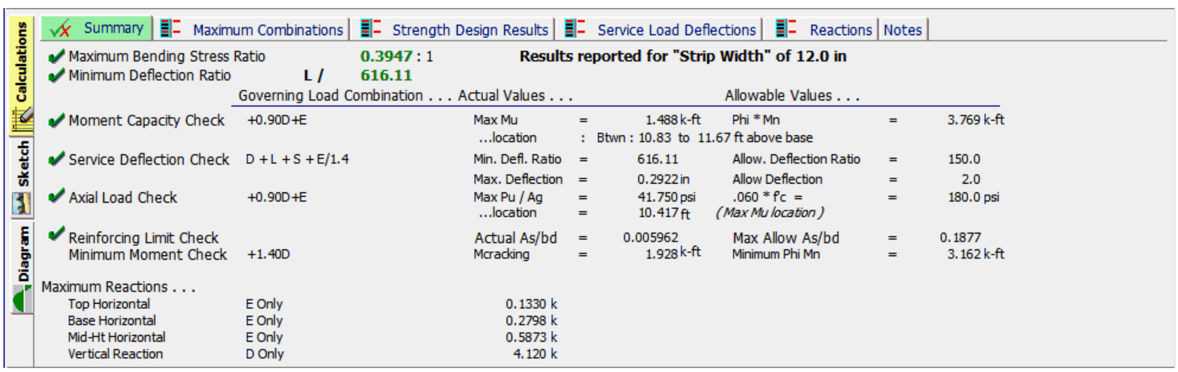
Load Combinations Tab

Typical load combination information as used throughout **Structural Engineering Library**.



Summary Tab

This tab presents the critical results as calculated by the module.



Maximum Bending Stress Ratio

The module looks at the detailed results for ALL strength design load combinations at all "segments" in the wall and pulls out the maximum factored load bending stress ratio to present here as the governing condition.

Minimum Deflection Ratio

The module looks at the detailed results for ALL service load combinations at all "segments" in the wall and pulls out the minimum service load deflection ratio (meaning maximum deflection) to present here as the governing condition.

Moment Capacity Check

For the condition of maximum bending stress ratio, the actual applied and allowable bending moments are given along with the governing load combination.

Service Deflection Check

For the condition of minimum deflection ratio (meaning maximum deflection) the ratio, deflection, allowable minimum ratio, allowable deflection (based on allowable ratio) and governing load combination are reported.

Axial Load Check

The module checks the actual axial stress in all segments for all load combinations and gives the maximum actual stress Pu/Ag. The allowable value is the result of the user's entry for maximum percentage of f'c to use.

Reinforcing Limit Check

The module checks all portions of the wall for reinforcing (including differently reinforced first and second stories and reveal areas). It reports the maximum reinforcing ratio and compares it with the maximum percentage of balanced section analysis As allowed.

Minimum Moment Check

ACI specifies that a wall section in bending shall have a minimum strength Mn that is greater than the cracking strength Mcr = Sgross * Fr.

Maximum Reactions

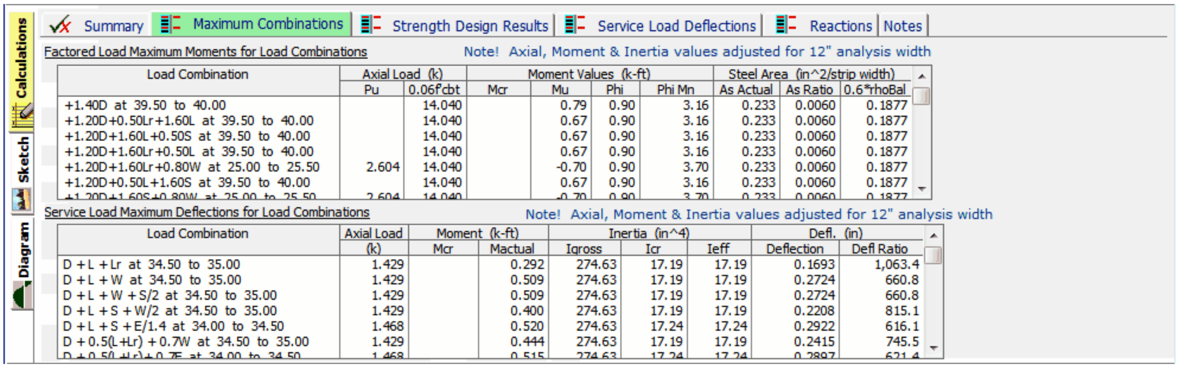
This gives a summary of the maximum reactions (both out-of-plane and vertical) along with the load combination that creates them.

Maximum Combinations Tab

This tab provides a summary of the governing values for each load combination for both factored load axial & bending and service load deflections.

Factored Load Maximum Moments for Load Combinations: The module looks through the result set for each load combination and identifies the location above the base of the wall at which the maximum condition is found. Note that "Aseff" is the effective area of steel and is influenced by the axial compression in that segment.

Service Load Maximum Deflections for Load Combinations: The module looks through the result set for each load combination and identifies the location above the base of the wall at which the maximum out-of-plane deflection is found. The value for "Ieff" is specific to the segment at that location and is based on the actual moment and Bischoff's equation for calculating effective moment of inertia.



Factored Load Maximum Moments for Load Combinations — Note! Axial, Moment & Inertia values adjusted for 12" analysis width

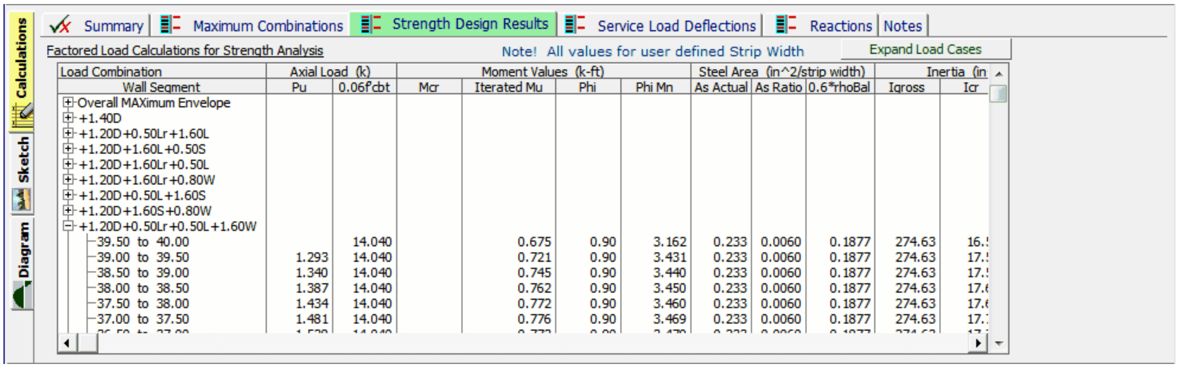
Load Combination	Axial Load (k)		Moment Values (k-ft)				Steel Area (in^2/strip width)		
	Pu	0.06f'cbt	Mcr	Mu	Phi	Phi Mn	As Actual	As Ratio	0.6*rhoBal
+1.40D at 39.50 to 40.00		14.040		0.79	0.90	3.16	0.233	0.0060	0.1877
+1.20D+0.50Lr+1.60L at 39.50 to 40.00		14.040		0.67	0.90	3.16	0.233	0.0060	0.1877
+1.20D+1.60L+0.50S at 39.50 to 40.00		14.040		0.67	0.90	3.16	0.233	0.0060	0.1877
+1.20D+1.60Lr+0.50L at 39.50 to 40.00		14.040		0.67	0.90	3.16	0.233	0.0060	0.1877
+1.20D+1.60Lr+0.80W at 25.00 to 25.50	2.604	14.040		-0.70	0.90	3.70	0.233	0.0060	0.1877
+1.20D+0.50L+1.60S at 39.50 to 40.00		14.040		0.67	0.90	3.16	0.233	0.0060	0.1877
+1.20D+1.60S+0.80W at 25.00 to 25.50	2.604	14.040		-0.70	0.90	3.70	0.233	0.0060	0.1877

Service Load Maximum Deflections for Load Combinations — Note! Axial, Moment & Inertia values adjusted for 12" analysis width

Load Combination	Axial Load	Moment (k-ft)		Inertia (in^4)			Defl. (in)	
	(k)	Mcr	Mactual	Igross	Icr	Ieff	Deflection	Defl Ratio
D + L + Lr at 34.50 to 35.00	1.429		0.292	274.63	17.19	17.19	0.1693	1,063.4
D + L + W at 34.50 to 35.00	1.429		0.509	274.63	17.19	17.19	0.2724	660.8
D + L + W + S/2 at 34.50 to 35.00	1.429		0.509	274.63	17.19	17.19	0.2724	660.8
D + L + S + W/2 at 34.50 to 35.00	1.429		0.400	274.63	17.19	17.19	0.2208	815.1
D + L + S + E/1.4 at 34.00 to 34.50	1.468		0.520	274.63	17.24	17.24	0.2922	616.1
D + 0.5(L+Lr) + 0.7W at 34.50 to 35.00	1.429		0.444	274.63	17.19	17.19	0.2415	745.5
D + 0.5(L+Lr) + 0.7E at 34.00 to 34.50	1.468		0.515	274.63	17.24	17.24	0.2897	621.4

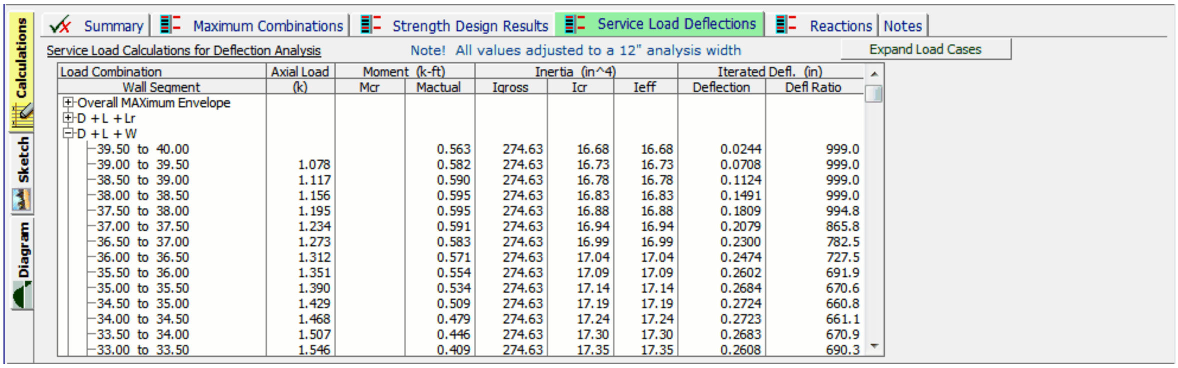
Strength Design Results Tab

This tab provides an extremely detailed summary of the factored axial load, moments, effective steel area and moment of inertia at each wall analysis segment for each load combination.



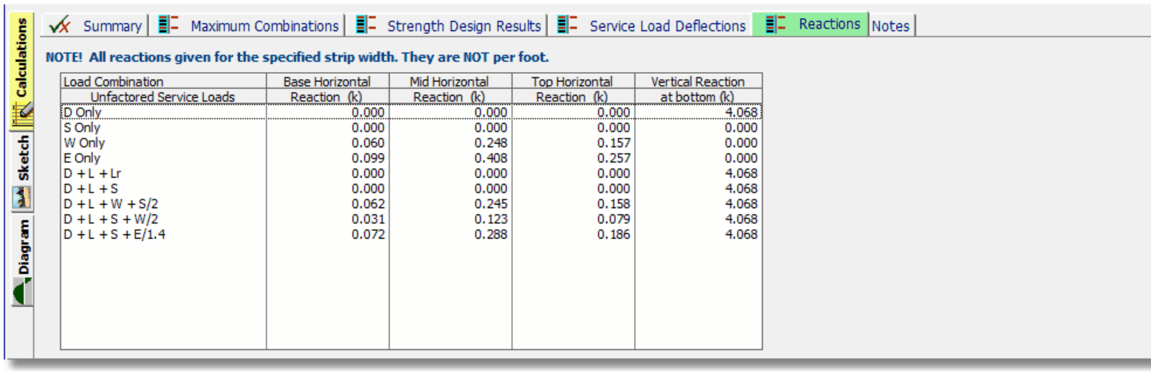
Service Load Deflections Tab

This tab provides an extremely detailed summary of the <u>service</u> axial load, moments, effective moment of inertia and calculated deflection at each wall analysis segment for each load combination.



Reactions Tab

This tab gives a summary of out-of-plane and vertical base reactions for each service load combination.

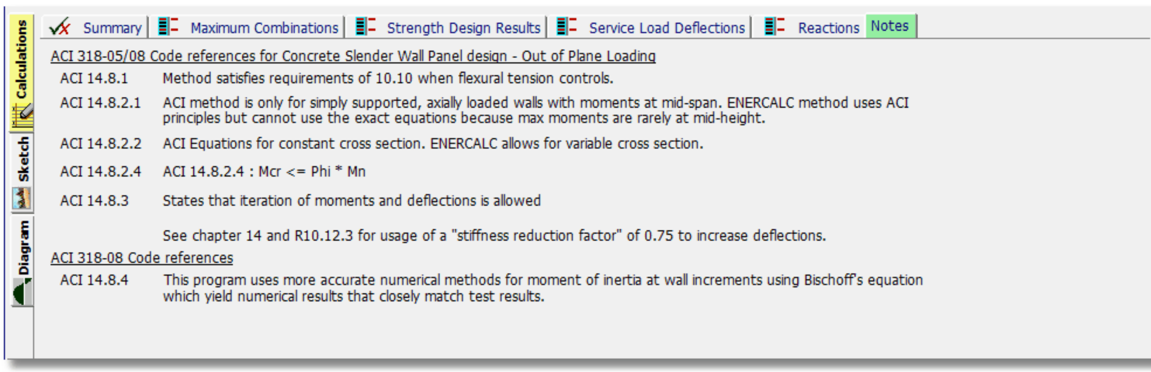


NOTE! All reactions given for the specified strip width. They are NOT per foot.

Load Combination Unfactored Service Loads	Base Horizontal Reaction (k)	Mid Horizontal Reaction (k)	Top Horizontal Reaction (k)	Vertical Reaction at bottom (k)
D Only	0.000	0.000	0.000	4.068
S Only	0.000	0.000	0.000	0.000
W Only	0.060	0.248	0.157	0.000
E Only	0.099	0.408	0.257	0.000
D +L +Lr	0.000	0.000	0.000	4.068
D +L +S	0.000	0.000	0.000	4.068
D +L +W +S/2	0.062	0.245	0.158	4.068
D +L +S +W/2	0.031	0.123	0.079	4.068
D +L +S +E/1.4	0.072	0.288	0.186	4.068

Notes Tab

Included are some excerpts from ACI regarding slender concrete wall design.



ACI 318-05/08 Code references for Concrete Slender Wall Panel design - Out of Plane Loading

ACI 14.8.1	Method satisfies requirements of 10.10 when flexural tension controls.
ACI 14.8.2.1	ACI method is only for simply supported, axially loaded walls with moments at mid-span. ENERCALC method uses ACI principles but cannot use the exact equations because max moments are rarely at mid-height.
ACI 14.8.2.2	ACI Equations for constant cross section. ENERCALC allows for variable cross section.
ACI 14.8.2.4	ACI 14.8.2.4 : Mcr <= Phi * Mn
ACI 14.8.3	States that iteration of moments and deflections is allowed
	See chapter 14 and R10.12.3 for usage of a "stiffness reduction factor" of 0.75 to increase deflections.

ACI 318-08 Code references

ACI 14.8.4	This program uses more accurate numerical methods for moment of inertia at wall increments using Bischoff's equation which yield numerical results that closely match test results.

Sketch Tab

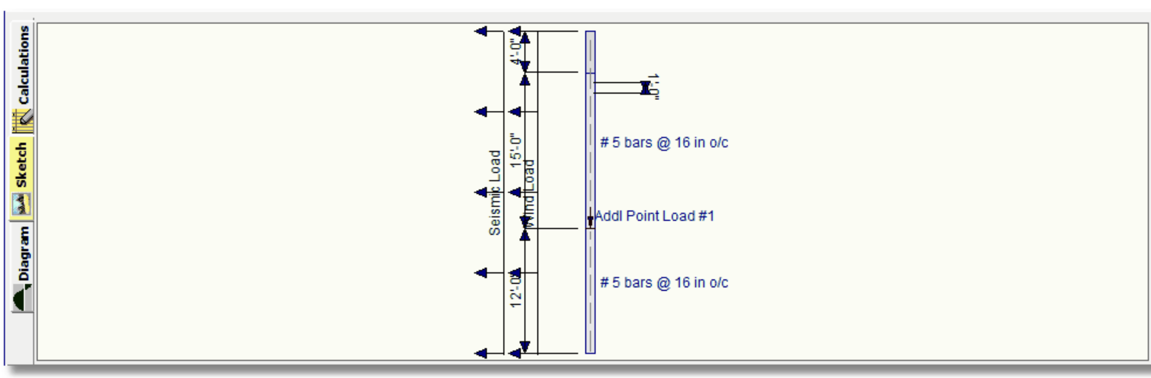
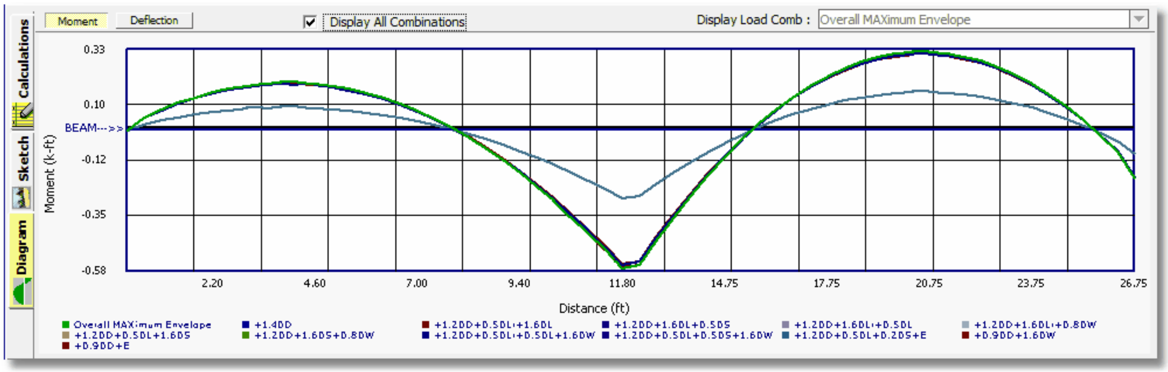


Diagram Tab


Calculations
Sketch
Diagram
Moment
Deflection
Display All Combinations
Display Load Comb :
Overall MAXimum Envelope
0.33
0.10
BEAM--->>
-0.12
-0.35
-0.58
Moment (k-ft)
2.20
4.60
7.00
9.40
11.80
14.75
17.75
20.75
23.75
26.75
Distance (ft)
Overall MAXimum Envelope
+1.4DD
+1.2DD+0.5DL+1.6DL
+1.2DD+1.6DL+0.5DS
+1.2DD+1.6DL+0.5DL
+1.2DD+1.6DL+0.8DW
+1.2DD+0.5DL+1.6DS
+1.2DD+1.6DS+0.8DW
+1.2DD+0.5DL+0.5DL+1.6DW
+1.2DD+0.5DL+0.5DS+1.6DW
+1.2DD+0.5DL+0.2DS+E
+0.9DD+1.6DW
+0.9DD+E

10.4.2 Masonry Slender Wall

Need more? Ask Us a Question

This module provides design and analysis according to the new provisions for design of masonry walls, using the P-Delta deflection considerations now included in the IBC. Click here for a video: Masonry Slender Wall

This method lifts the restriction on H/t ratios, and performs wall analysis using the principles of ultimate strength design. The design method is very similar to that of the Concrete Slender Wall module.

This module uses a variable width strip of wall section to represent a typical section of wall. The module has the ability to apply a lateral wind load, seismic load, partial length uniform lateral load, and a lateral point load to the clear span of the wall section. This variety of loadings should take care of almost every lateral loading case possible.

The user may specify masonry and reinforcing strengths, seismic factor, wind load, vertical and lateral loads, vertical load eccentricities, and wall construction. The module determines the wall capacity, actual deflections considering P-Delta effects, and solves for the final moments obtained through iteration of the P-Delta effects. Deflection analysis is provided for both service and factored load cases.

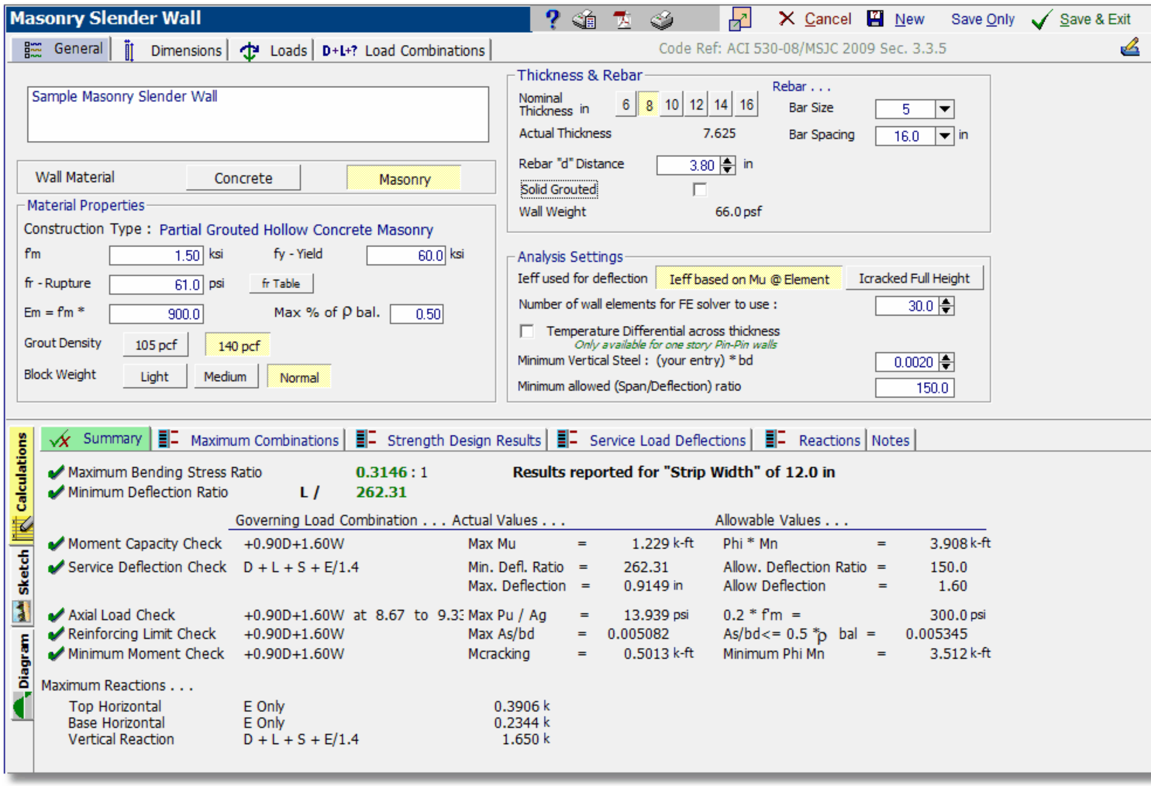
The user reaches a final design by modifying wall thickness, rebar size, and rebar spacing until no overstress condition exists and the deflection limits prescribed in the code are satisfied.

This module uses basic principles of structural mechanics to model the wall as a series of beam segments. For each segment the actual moment is used to calculate member stiffness using the $I_{effective}$ equations developed by Peter H. Bischoff. Since these changes to wall stiffness affect the wall deflection profile, the program performs an iterative analysis of calculating moments (including P-Delta effects). The results are deflection curves almost exactly matching the SCCACI-SEAOSC test results. This makes this module far more accurate at calculating wall deflections and P-Delta effects than the simple equations in the ACI code.

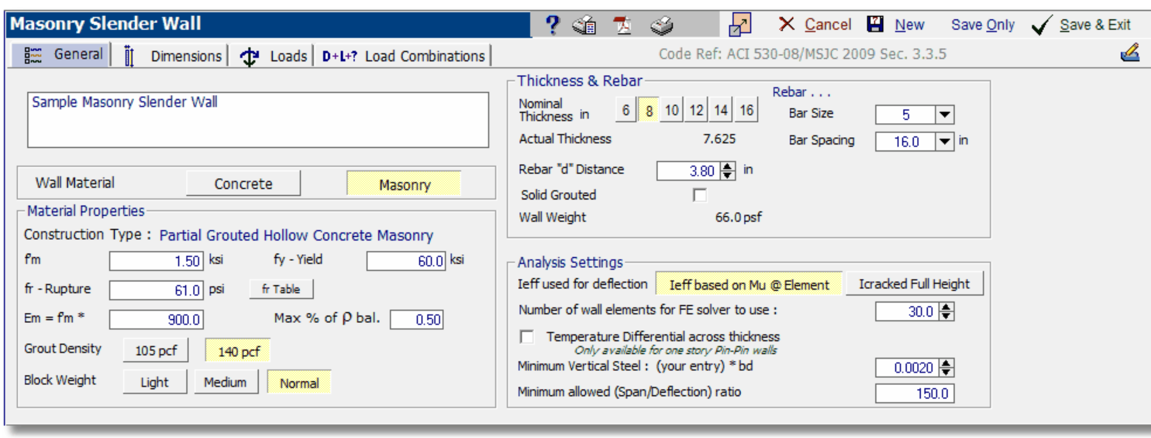
Capabilities

This module provides these capabilities:

- One or two story slender masonry walls
- Iterative process accounts for P-delta
- Optional parapet
- Axial loads with optional eccentricities
- Wind, seismic and user defined lateral loads creating bending on the wall panel
- Variable strip width to model the wall panel
- Temperature differential can be specified across thickness of wall to add curvature



General Tab



Material Properties

f'm

Enter the allowable masonry strength to be used in the analysis. The allowable bending and axial stresses calculated from f'm are outlined in a later section.

fy

Yield point stress of reinforcing.

fr - Rupture & Fr-Table

Modulus of rupture for the masonry wall system.

Em = f'm * [value]

The modulus of elasticity of the masonry wall system is specified by this value acting as a multiplier to f'm.

Max % of Rho Balanced

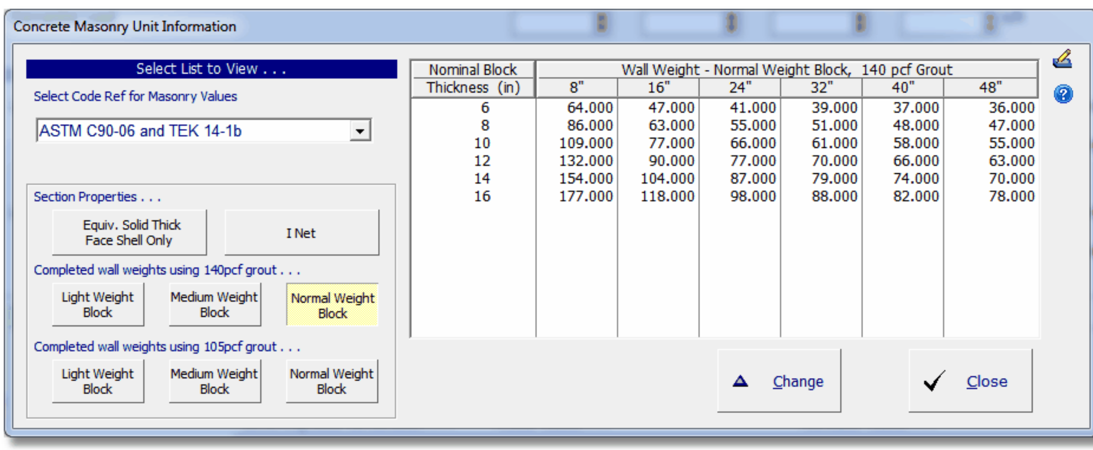
This value sets the maximum percentage of reinforcing the module will allow without giving a warning message. Enter this as a value less than 1.0 which will be applied to the reinforcing area calculated for a balanced section to determine the maximum allowable reinforcing ratio.

Grout Density

Choose from two different options for the density of the grout.

Block Weight

Select light, medium and normal weight block. The weight of a completed wall is determined from the masonry database, depending on the block weight, grout density, and grouted cell spacing. To view the database values click **Databases > Concrete Masonry Unit Data** from the main menu. Here is what you will see:



Nominal Block Thickness (in)	Wall Weight - Normal Weight Block, 140 pcf Grout 8"	16"	24"	32"	40"	48"
6	64.000	47.000	41.000	39.000	37.000	36.000
8	86.000	63.000	55.000	51.000	48.000	47.000
10	109.000	77.000	66.000	61.000	58.000	55.000
12	132.000	90.000	77.000	70.000	66.000	63.000
14	154.000	104.000	87.000	79.000	74.000	70.000
16	177.000	118.000	98.000	88.000	82.000	78.000

Thickness & Rebar

Nominal Thickness

Select the nominal thickness for the concrete masonry units used in the wall construction. This selection will pull the values for wall weight, equivalent solid thickness and Igross from the masonry database (see above).

Actual Thickness

The actual wall thickness for the nominal selection.

Rebar Size & Spacing

Enter the rebar size and spacing.

Rebar "d" Distance

Enter the distance from the extreme compression fiber to the center of the rebar.

Solid Grouted

Select this checkbox if the wall is to be grouted solid . If unchecked the module will calculate the wall weight considering that grouted cells only occur at the spacing of the reinforcing.

Wall Weight

Weight of wall as retrieved from the masonry database. Value is based on specified wall thickness, grout density, block type, and grouting frequency.

Analysis Settings

P-Delta Deflection Method

The module always performs an iterative analysis for moments and deflections using progressively greater wall deflections due to increasing P-Delta effects.

Temperature Differential across thickness

This input is used to describe the temperature change between each face of the wall. A temperature change induces a slight curvature into the wall because the hotter side expands, resulting in a slightly higher out-of-plane deflection.

Enter temperature differentials as positive values. The effects of a specified temperature differential are always additive with the bending and deflection resulting from other applied loads.

Minimum Vertical Steel: %/100

Minimum steel percentage as a portion of the gross wall area.

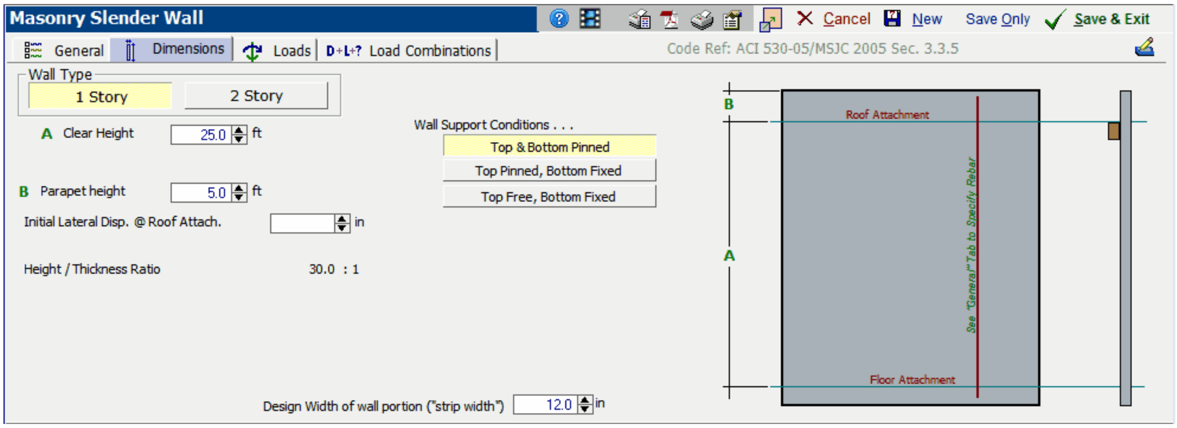
Minimum Out-of-Plane Deflection Ratio

This setting establishes the minimum allowable ratio of span length to service load deflection. If a lower actual Span/Deflection ratio occurs (meaning greater deflection), a warning message will be displayed.

Number of wall elements for FE solver to use

This module divides the wall design strip into segments from the base to the top for analysis purposes. Use this entry to define the number of segments to use. Experience demonstrates that approximately 30 segments gives a good balance between the iterative P-Delta analysis reaching convergence and excessive calculation time.

Dimensions Tab



Fixity Conditions

Controls how the top and bottom of the wall are restrained for moments and lateral movement.

[Top & Bottom Pinned]

Base of wall is restrained against movement out of plane and vertically, rotates freely. Top of wall restrained against out of plane movement and can move vertically and rotate freely.

[Top Pinned, Bottom Fixed]

Base of wall is restrained against movement about all three axes. Top of wall restrained against out of plane movement and can move vertically and rotate freely.

[Top Free, Bottom Fixed]

Base of wall is restrained against movement about all three axes. Top of wall is completely free making this a cantilevered wall.

Clear Height

Span of the wall between the base and the first lateral support. For one story walls this is the top support. For 2 story walls this is this prompt will change to be "1st story height".

Parapet Height

Distance the wall extends (without a topmost lateral support) above the topmost lateral support (clear height for one story wall, 2nd story height for 2 story walls)

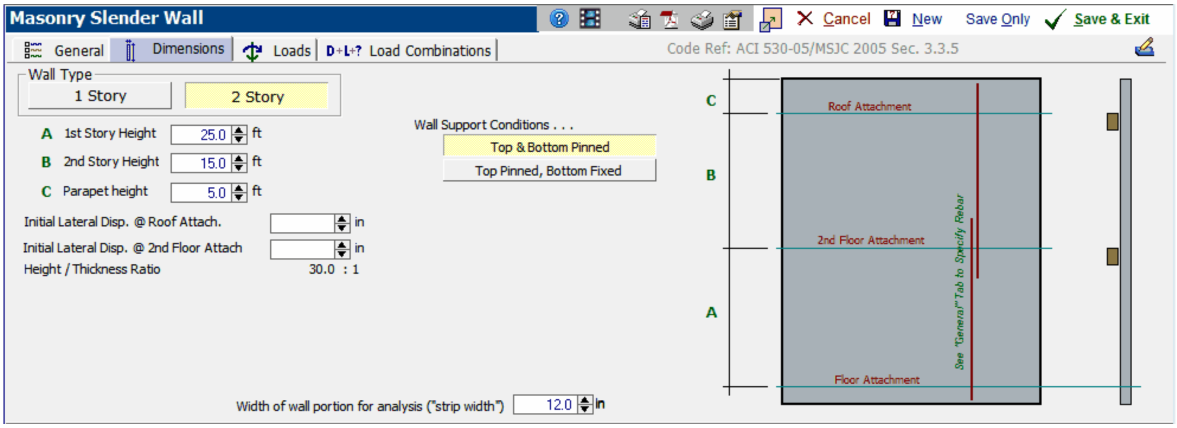
Strip Width: Width of wall portion for analysis

This module performs its analysis for this width. Results are for either for this width or a 12" width as noted where the results are provided.

Note that applied loads either are applied to the entire strip width (as for concentrated vertical and lateral loads) or are entered on a per-foot basis when they are uniform loads.

Two Story...

When a two story wall is selected this tab changes slightly to provide the 2nd story height and remove the Fixed-Free support option.



1st Story Height

Distance from the bottom of the wall to the first lateral support.

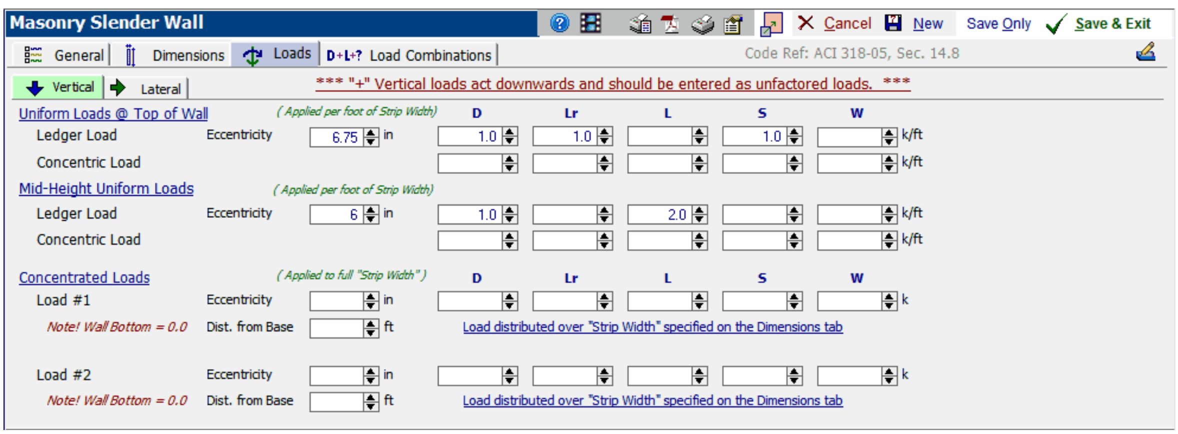
2nd Story Height

Distance from the first lateral support to the top lateral support.

Loads Tab - Vertical Loads

A variety of vertical loads are available. Note the hint in green describing whether the load is per foot or on the entire strip width.

All loads that are entered on this tab will be multiplied by the load factors specified on the Load Combination sub-tabs. So these magnitudes should be specified with those load factors in mind.



Ledger Load

This is a per-foot vertical load applied to the wall at an optional eccentricity. So if you have a 48" strip width and specify a 1 k/ft dead load then the strip will have a total of 4 kip applied due to the 1 k/ft entry.

Concentric Load

This is a per-foot vertical load applied concentrically to the wall. So if you have a 48" strip width and specify a 1 k/ft dead load then the strip will have a total of 4 kip applied due to the 1 k/ft entry.

Mid-Height Vertical Uniform Load

This load entry is only shown for 2-story walls. It allows you to specify two uniform loads applied at the "1st Story" height, one of which can have an eccentricity from the wall center.

Concentrated Loads

These are single concentrated vertical loads applied to the wall "strip width" with an optional eccentricity.

Distance from Base is the height at which the load is applied.

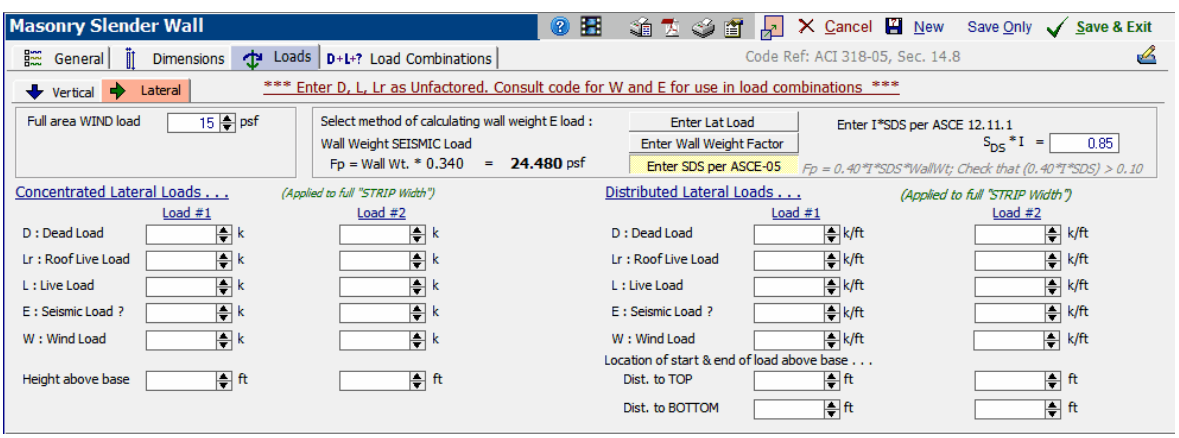
Eccentricity

Describes an offset from the mid-thickness of the wall panel, which is the default location of application of a vertical load. Enter this value as a positive number when the load is shifted toward the inside of the wall.

Loads Tab - Lateral Loads

Lateral loads are applied perpendicular to the plane of the wall and are almost always seismic or wind. These loads create out-of-plane deflection of the wall, which the module will use to develop P-Delta effects to calculate secondary moments in the wall. Recall from other explanations that the module divides the wall into small segments and calculates the allowable and actual forces and deflections for each small segment. In this way the lateral loads are properly modeled on what is a beam with variable stiffness due to the state of cracking in each segment.

All loads that are entered on this tab will be multiplied by the load factors specified on the Load Combination sub-tabs. So these magnitudes should be specified with those load factors in mind.



Full area WIND Load

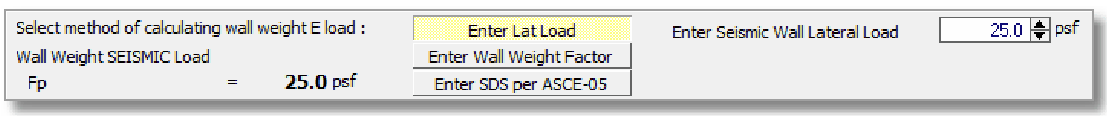
Enter the wind load that will be applied to the wall in the out-of-plane direction. This load will only be applied to one surface of the wall, and as such, the magnitude must take into consideration both the internal and external pressures.

For the purpose of defining the direction of applied wind pressures, consider that pressure will only be applied to the exterior surface of the wall. A net pressure that acts TOWARD the exterior surface of the wall should be entered as a positive value. A net pressure that acts AWAY FROM the exterior surface of the wall should be entered as a negative value.

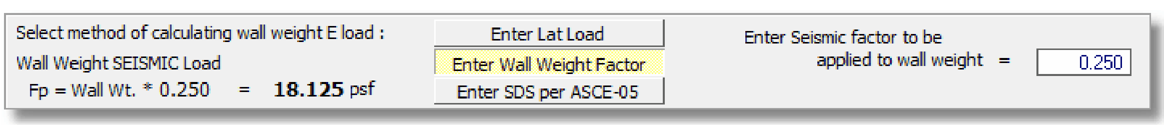
Wall weight seismic load

This section offers three options to specify the seismic load that will be internally applied to the wall in the out-of-plane direction:

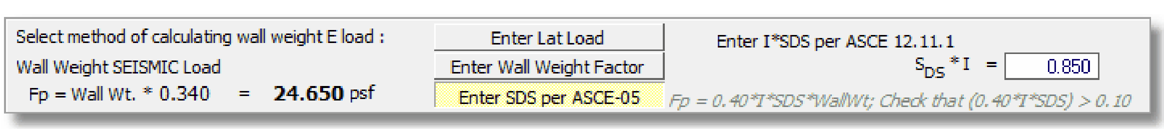
Enter Lateral Load: This entry is a simple net load applied to the wall (but will still be factored by the load combination factors for "E").



Enter Wall Weight Factor: Enter a number that will be multiplied by the self-weight of the wall. For example, if you enter 0.25 and the wall weighs 80 psf, then a 20.00 psf out-of-plane load will be calculated and applied to the wall using the load combination factors for "E".



Enter S_{DS} per ASCE-05: Enter the (S_{DS} * I) value as prescribed by the ASCE code for the building location. The minimum calculated load value of 10 psf or (0.4 * Value Entered * Wall Weight) will be applied to the wall using the load combination factors for "E".



Fp

This is the actual seismic load applied perpendicular to the plane of the wall, which represents the wall's seismic self weight load.

Concentrated Lateral Loads

This is an added lateral load applied perpendicular to the plane of the wall. It acts on the full "Strip Width" and is factored by the load combination factors corresponding to the type of load.

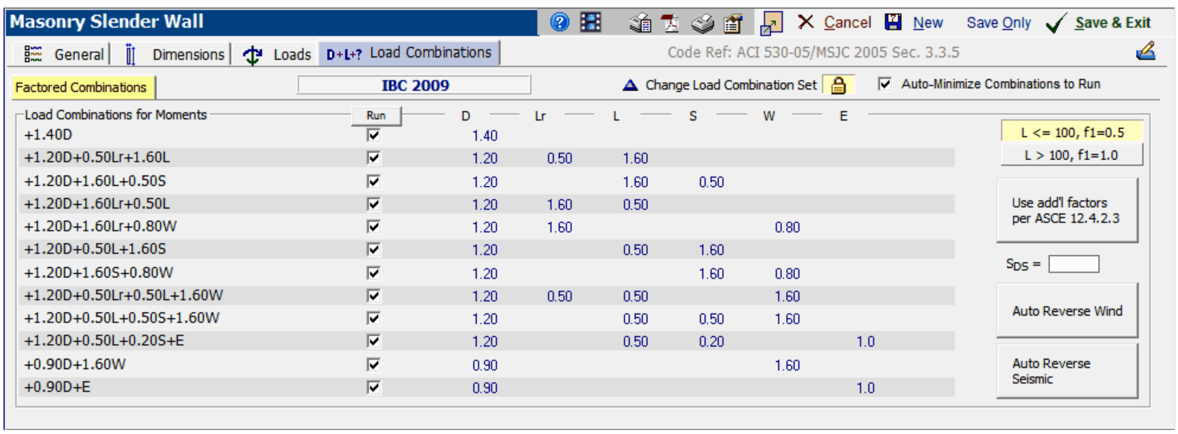
Distributed Lateral Loads

This is an added lateral uniform load applied perpendicular to the plane of the wall. It acts on the full "Strip Width" and is factored by the load combination factors corresponding to the type of load.

You also enter the start and end distance of the load extent above the base of the wall.

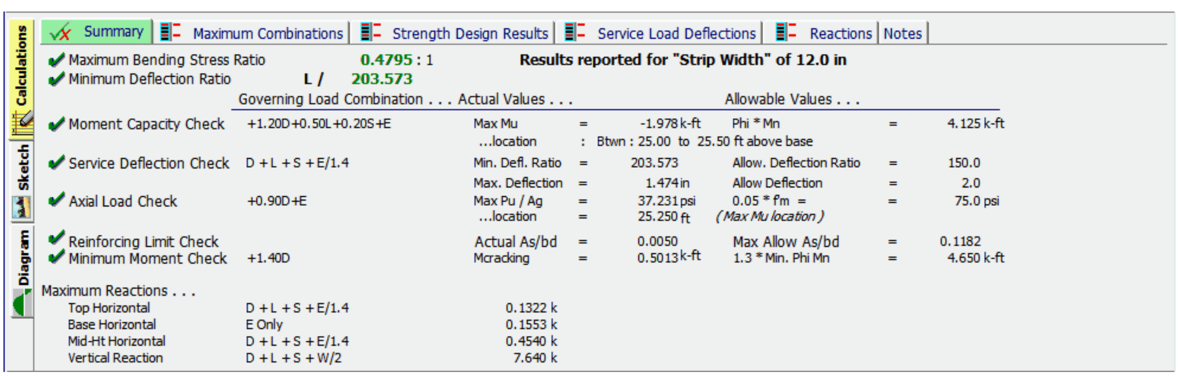
Load Combination Tab

Typical load combination information as used throughout **Structural Engineering Library**.



Results Summary Tab

This tab presents the critical results as calculated by the module.



Maximum Bending Stress Ratio

The module looks at the detailed results for ALL strength design load combinations at all "segments" in the wall and pulls out the maximum factored load bending stress ratio to present here as the governing condition.

Minimum Deflection Ratio

The module looks at the detailed results for ALL service load combinations at all "segments" in the wall and pulls out the minimum service load deflection ratio (meaning maximum deflection) to present here as the governing condition.

Moment Capacity Check

For the condition of maximum bending stress ratio, the actual applied and allowable bending moments are given along with the governing load combination.

Service Deflection Check

For the condition of minimum deflection ratio (meaning maximum deflection) the ratio, deflection, allowable minimum ratio, allowable deflection (based on allowable ratio) and governing load combination are reported.

Axial Load Check

The module checks the factored axial stress in all segments for all load combinations and gives the maximum actual stress Pu/Ag. The allowable value is the result of the wall slenderness. If the slenderness is less than or equal to 30, the allowable value of factored axial stress is 0.20f'm. If the slenderness is greater than 30, the allowable value of factored axial stress is 0.05f'm.

Reinforcing Limit Check

The module checks all portions of the wall for reinforcing (including differently reinforced first and second stories) and reports the maximum reinforcing ratio and compares it with the maximum percentage of balanced section analysis As allowed.

Minimum Moment Check

ACI specifies that a wall section in bending shall have a minimum strength Mn that is greater than the cracking strength Mcr = Sgross * fr.

Maximum Reactions

This gives a summary of the maximum reactions (both out-of-plane and vertical) along with the load combination that creates them.

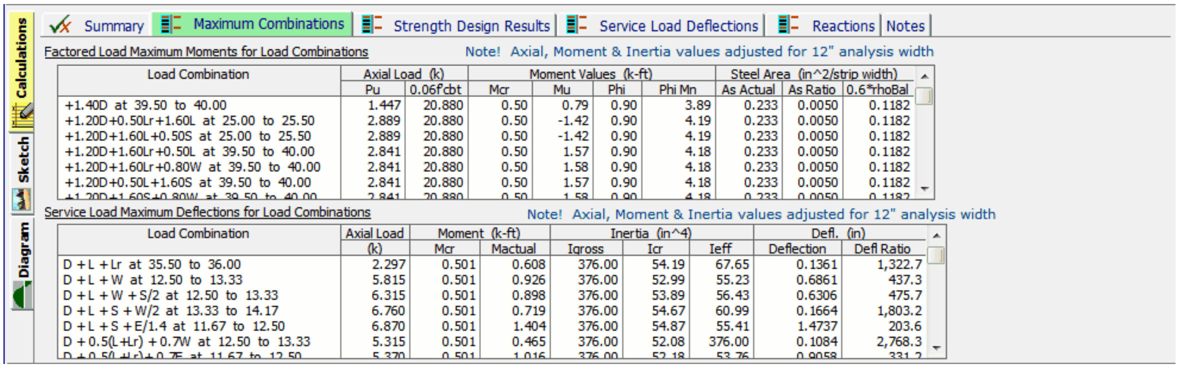
Maximum Combinations

This tab provides a summary of the governing values for each load combination for both factored load axial & bending and service load deflections.

Factored Load Maximum Moments for Load Combinations: The module looks through the result set for each load combination and identifies the location above the base of the wall at which the maximum condition is found. Note that "Aseff" is the effective area of steel and is influenced by the axial compression in that segment.

Service Load Maximum Deflections for Load Combinations: The module looks through the result set for each load combination and identifies the location above the base of the wall at which the maximum out-of-plane deflection is found. The value for

"Ieff" is specific to the segment at that location and is based on the actual moment and Bischoff's equation for calculating effective moment of inertia.



Summary | Maximum Combinations | Strength Design Results | Service Load Deflections | Reactions | Notes

Factored Load Maximum Moments for Load Combinations — Note! Axial, Moment & Inertia values adjusted for 12" analysis width

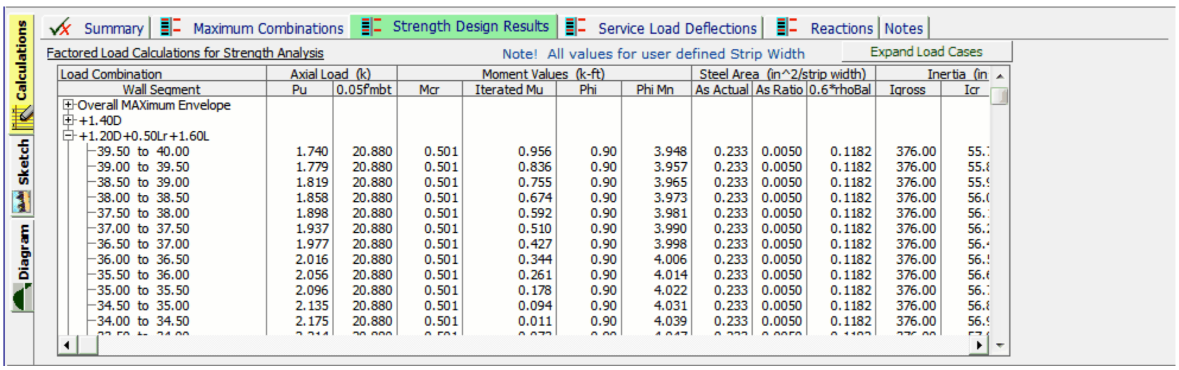
Load Combination	Axial Load (k)		Moment Values (k-ft)				Steel Area (in^2/strip width)		
	Pu	0.06f'cbt	Mcr	Mu	Phi	Phi Mn	As Actual	As Ratio	0.6*rhoBal
+1.40D at 39.50 to 40.00	1.447	20.880	0.50	0.79	0.90	3.89	0.233	0.0050	0.1182
+1.20D+0.50Lr+1.60L at 25.00 to 25.50	2.889	20.880	0.50	-1.42	0.90	4.19	0.233	0.0050	0.1182
+1.20D+1.60L+0.50S at 25.00 to 25.50	2.889	20.880	0.50	-1.42	0.90	4.19	0.233	0.0050	0.1182
+1.20D+1.60Lr+0.50L at 39.50 to 40.00	2.841	20.880	0.50	1.57	0.90	4.18	0.233	0.0050	0.1182
+1.20D+1.60Lr+0.80W at 39.50 to 40.00	2.841	20.880	0.50	1.58	0.90	4.18	0.233	0.0050	0.1182
+1.20D+0.50L+1.60S at 39.50 to 40.00	2.841	20.880	0.50	1.57	0.90	4.18	0.233	0.0050	0.1182

Service Load Maximum Deflections for Load Combinations — Note! Axial, Moment & Inertia values adjusted for 12" analysis width

Load Combination	Axial Load	Moment (k-ft)		Inertia (in^4)			Defl. (in)	
	(k)	Mcr	Mactual	Igross	Icr	Ieff	Deflection	Defl Ratio
D +L +Lr at 35.50 to 36.00	2.297	0.501	0.608	376.00	54.19	67.65	0.1361	1,322.7
D +L +W at 12.50 to 13.33	5.815	0.501	0.926	376.00	52.99	55.23	0.6861	437.3
D +L +W +S/2 at 12.50 to 13.33	6.315	0.501	0.898	376.00	53.89	56.43	0.6306	475.7
D +L +S +W/2 at 13.33 to 14.17	6.760	0.501	0.719	376.00	54.67	60.99	0.1664	1,803.2
D +L +S +E/1.4 at 11.67 to 12.50	6.870	0.501	1.404	376.00	54.87	55.41	1.4737	203.6
D +0.5(L+Lr) +0.7W at 12.50 to 13.33	5.315	0.501	0.465	376.00	52.08	376.00	0.1084	2,768.3

Strength Design Results

This tab provides an extremely detailed summary of the factored axial load, moments, effective steel area and moment of inertia at each wall analysis segment for each load combination.



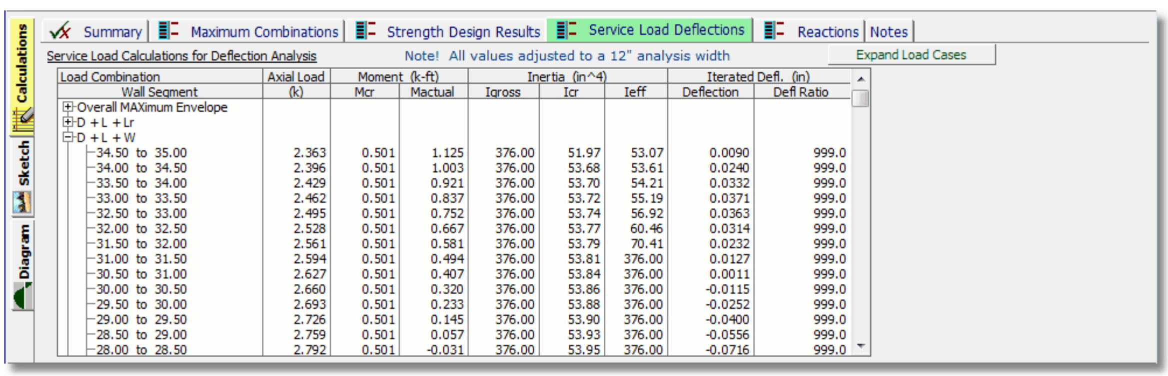
Summary | Maximum Combinations | Strength Design Results | Service Load Deflections | Reactions | Notes

Factored Load Calculations for Strength Analysis — Note! All values for user defined Strip Width — Expand Load Cases

Load Combination	Axial Load (k)		Moment Values (k-ft)				Steel Area (in^2/strip width)			Inertia (in	
Wall Segment	Pu	0.05f'mbt	Mcr	Iterated Mu	Phi	Phi Mn	As Actual	As Ratio	0.6*rhoBal	Igross	Icr
⊞ Overall MAXimum Envelope											
⊞ +1.40D											
⊟ +1.20D+0.50Lr+1.60L											
39.50 to 40.00	1.740	20.880	0.501	0.956	0.90	3.948	0.233	0.0050	0.1182	376.00	55.[illegible]
39.00 to 39.50	1.779	20.880	0.501	0.836	0.90	3.957	0.233	0.0050	0.1182	376.00	55.[illegible]
38.50 to 39.00	1.819	20.880	0.501	0.755	0.90	3.965	0.233	0.0050	0.1182	376.00	55.[illegible]
38.00 to 38.50	1.858	20.880	0.501	0.674	0.90	3.973	0.233	0.0050	0.1182	376.00	56.[illegible]
37.50 to 38.00	1.898	20.880	0.501	0.592	0.90	3.981	0.233	0.0050	0.1182	376.00	56.[illegible]
37.00 to 37.50	1.937	20.880	0.501	0.510	0.90	3.990	0.233	0.0050	0.1182	376.00	56.[illegible]
36.50 to 37.00	1.977	20.880	0.501	0.427	0.90	3.998	0.233	0.0050	0.1182	376.00	56.[illegible]
36.00 to 36.50	2.016	20.880	0.501	0.344	0.90	4.006	0.233	0.0050	0.1182	376.00	56.[illegible]
35.50 to 36.00	2.056	20.880	0.501	0.261	0.90	4.014	0.233	0.0050	0.1182	376.00	56.[illegible]
35.00 to 35.50	2.096	20.880	0.501	0.178	0.90	4.022	0.233	0.0050	0.1182	376.00	56.[illegible]
34.50 to 35.00	2.135	20.880	0.501	0.094	0.90	4.031	0.233	0.0050	0.1182	376.00	56.[illegible]
34.00 to 34.50	2.175	20.880	0.501	0.011	0.90	4.039	0.233	0.0050	0.1182	376.00	56.[illegible]

Service Load Deflections

This tab provides an extremely detailed summary of the service axial load, moments, effective moment of inertia and calculated deflection at each wall analysis segment for each load combination.



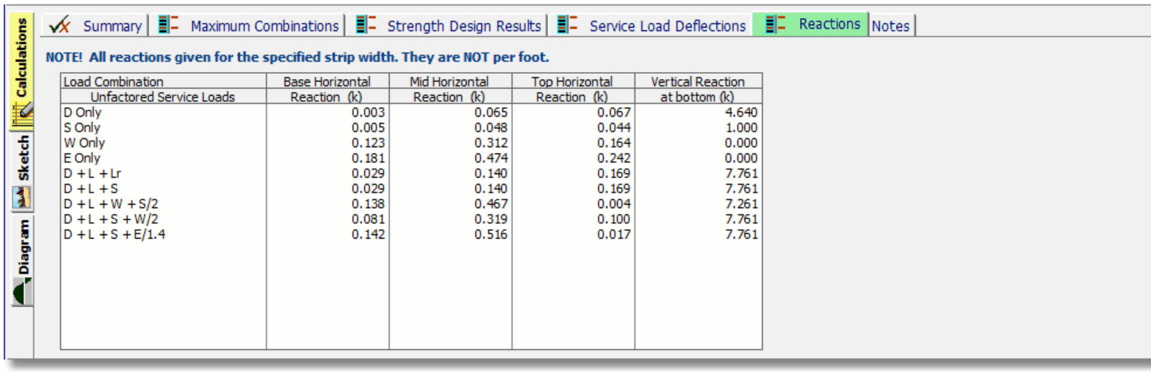
Summary | Maximum Combinations | Strength Design Results | Service Load Deflections | Reactions | Notes

Service Load Calculations for Deflection Analysis — Note! All values adjusted to a 12" analysis width — Expand Load Cases

Load Combination	Axial Load	Moment (k-ft)		Inertia (in^4)			Iterated Defl. (in)	
Wall Segment	(k)	Mcr	Mactual	Igross	Icr	Ieff	Deflection	Defl Ratio
⊞ Overall MAXimum Envelope								
⊞ D +L +Lr								
⊟ D +L +W								
34.50 to 35.00	2.363	0.501	1.125	376.00	51.97	53.07	0.0090	999.0
34.00 to 34.50	2.396	0.501	1.003	376.00	53.68	53.61	0.0240	999.0
33.50 to 34.00	2.429	0.501	0.921	376.00	53.70	54.21	0.0332	999.0
33.00 to 33.50	2.462	0.501	0.837	376.00	53.72	55.19	0.0371	999.0
32.50 to 33.00	2.495	0.501	0.752	376.00	53.74	56.92	0.0363	999.0
32.00 to 32.50	2.528	0.501	0.667	376.00	53.77	60.46	0.0314	999.0
31.50 to 32.00	2.561	0.501	0.581	376.00	53.79	70.41	0.0232	999.0
31.00 to 31.50	2.594	0.501	0.494	376.00	53.81	376.00	0.0127	999.0
30.50 to 31.00	2.627	0.501	0.407	376.00	53.84	376.00	0.0011	999.0
30.00 to 30.50	2.660	0.501	0.320	376.00	53.86	376.00	-0.0115	999.0
29.50 to 30.00	2.693	0.501	0.233	376.00	53.88	376.00	-0.0252	999.0
29.00 to 29.50	2.726	0.501	0.145	376.00	53.90	376.00	-0.0400	999.0
28.50 to 29.00	2.759	0.501	0.057	376.00	53.93	376.00	-0.0556	999.0
28.00 to 28.50	2.792	0.501	-0.031	376.00	53.95	376.00	-0.0716	999.0

Reactions

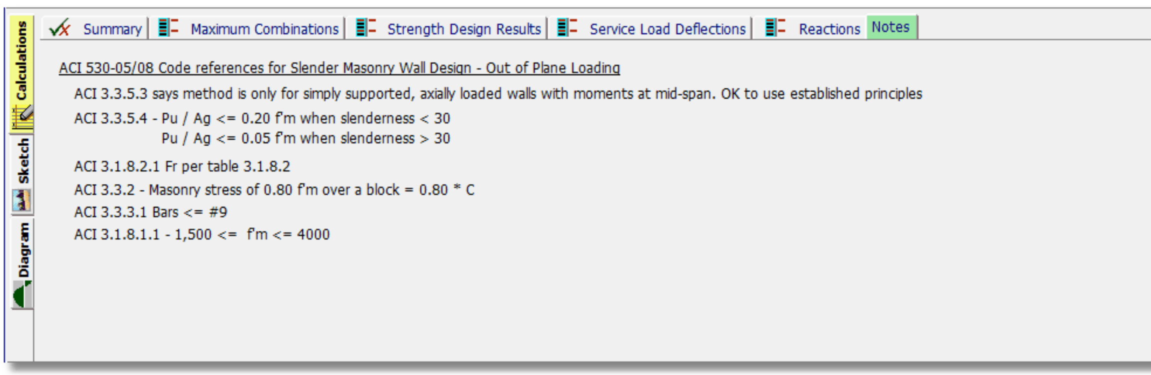
This tab gives a summary of out-of-plane and vertical base reactions for each service load combination.



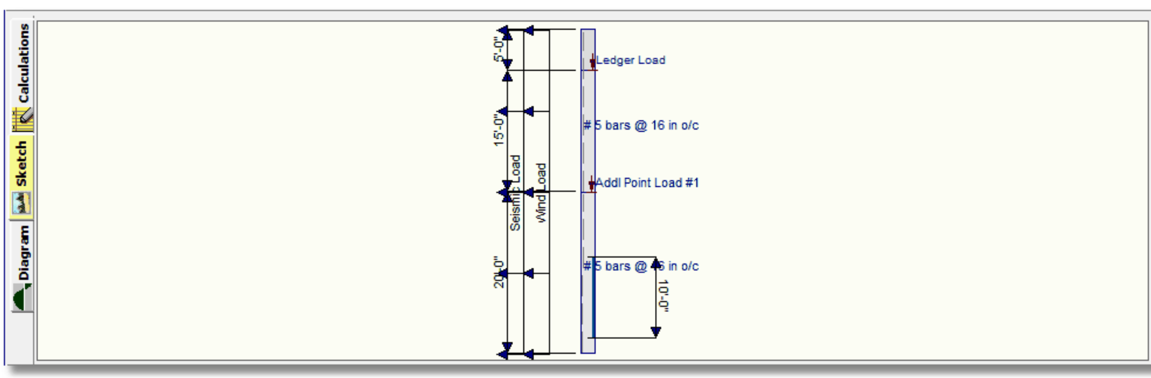
Load Combination Unfactored Service Loads	Base Horizontal Reaction (k)	Mid Horizontal Reaction (k)	Top Horizontal Reaction (k)	Vertical Reaction at bottom (k)
D Only	0.003	0.065	0.067	4.640
S Only	0.005	0.048	0.044	1.000
W Only	0.123	0.312	0.164	0.000
E Only	0.181	0.474	0.242	0.000
D +L +Lr	0.029	0.140	0.169	7.761
D +L +S	0.029	0.140	0.169	7.761
D +L +W +S/2	0.138	0.467	0.004	7.261
D +L +S +W/2	0.081	0.319	0.100	7.761
D +L +S +E/1.4	0.142	0.516	0.017	7.761

Notes

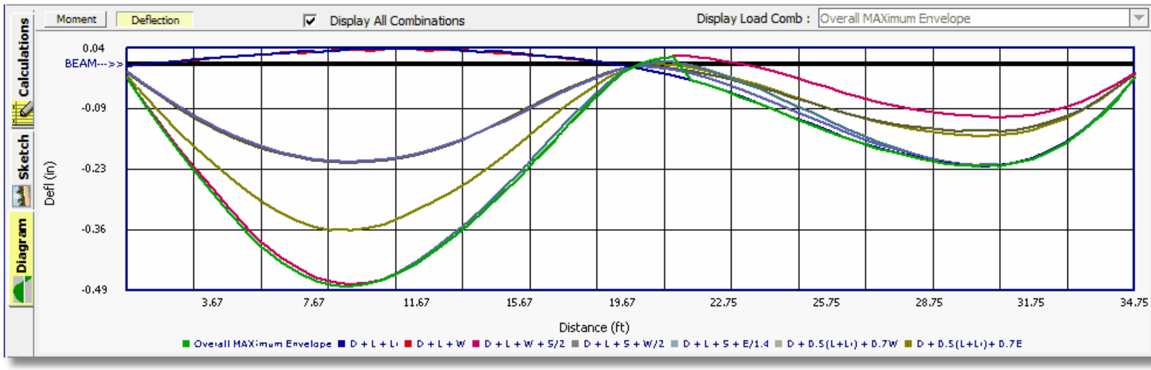
Included are some excerpts from ACI regarding slender masonry wall design.



Sketch



Diagram


Calculations
Sketch
Diagram
Moment
Deflection
Display All Combinations
Display Load Comb :
Overall MAXimum Envelope
0.04
BEAM--->>
-0.09
-0.23
-0.36
-0.49
Defl (in)
3.67
7.67
11.67
15.67
19.67
22.75
25.75
28.75
31.75
34.75
Distance (ft)
Overall MAXimum Envelope
D + L + Lr
D + L + W
D + L + W + S/2
D + L + S + W/2
D + L + S + E/1.4
D + 0.5(L+Lr) + 0.7W
D + 0.5(L+Lr)+ 0.7E

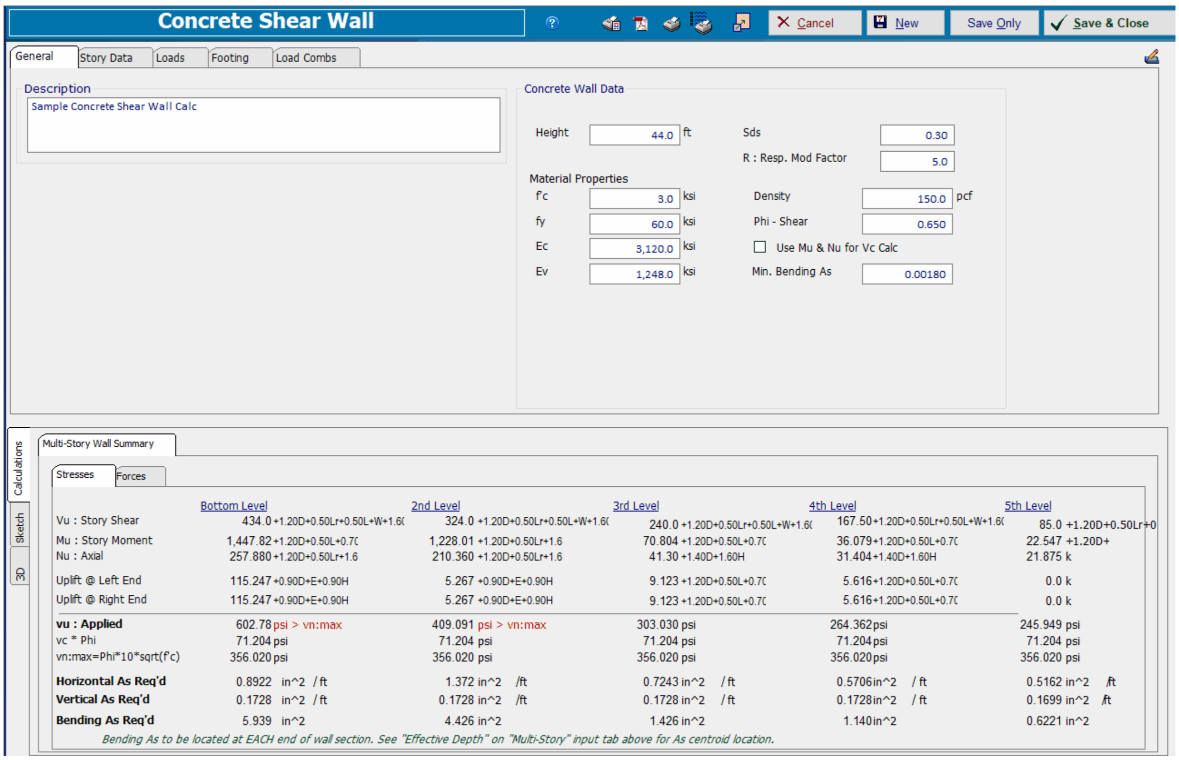
10.5 Shear Walls

Please select a material chapter.

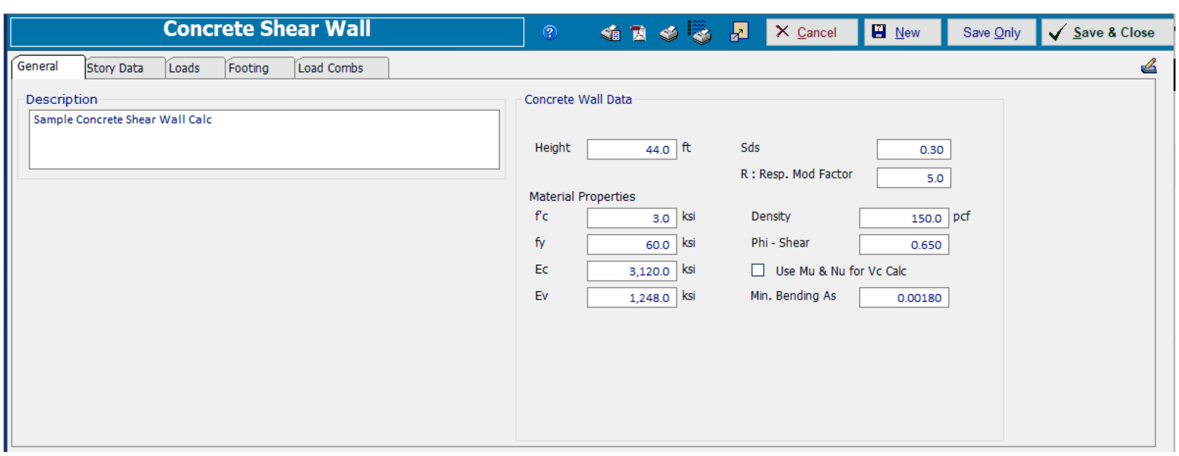
10.5.1 Concrete Shear Wall

[Need more? Ask Us a Question]

This module allows the design of concrete shear walls including multi-story walls with no openings but with up to five levels of differing length, height and thickness.



General Tab



Height specifies the total height of the wall. On the next tab you can divide that total height into up to five different wall portions.

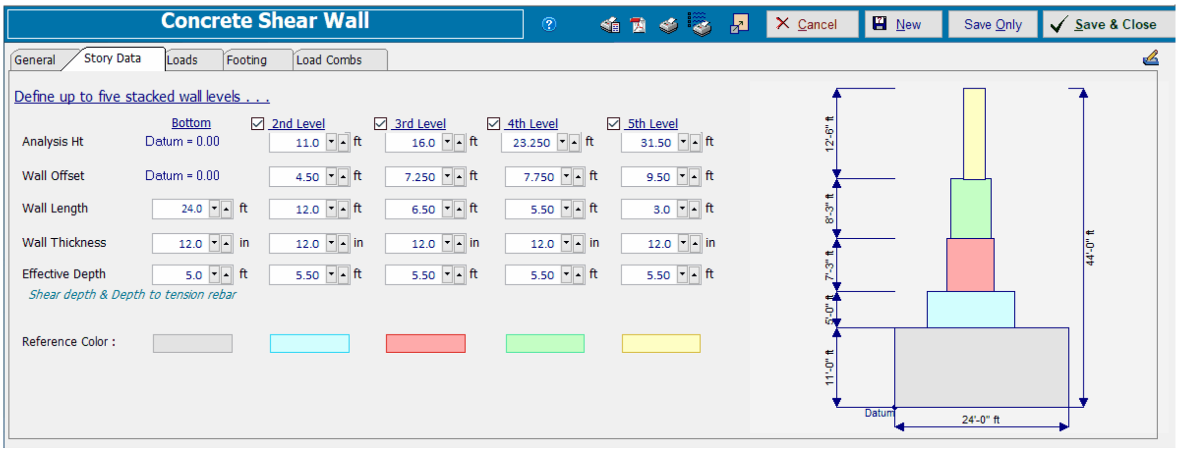
Ec is the bending modulus of elasticity and **Ev** is the shear modulus of elasticity.

S_{DS} and **R** are used to calculate the in-plane portion of the wall weight to be applied as a seismic load.

Ec is the bending modulus of elasticity, and **Ev** is the shear modulus of elasticity.

Story Data Tab

This tab is where you specify the distinct wall levels for the wall.



Analysis Height locates the bottom edge of the wall section and is where the maximum shear and bending stress will be calculated. This is the user-defined height at which the analysis of a particular wall section will be performed. All moments, shears, and vertical loads at this height will be calculated using all applied lateral and vertical loads and the wall self weight above this point. The other wall data items specified in the same column will be used between this analysis height and the next higher level indicated in the column to the right.

ALWAYS WORK WITH THE HIGHER ANALYSIS HEIGHT IN THE COLUMN TO THE RIGHT OF A LOWER HEIGHT. This is needed due to the manner in which the module calculates the heights by comparing heights of adjacent sections.

Wall Offset is the distance that this wall section is offset from the left-most edge of the bottom-most wall section. Please refer to the diagram to further understand this item. Because this module can be used with a walls that have their length changes with height, you must enter the offset from the bottom wall section to the LEFT EDGE of the wall section. This enables the module to calculate the actual X-Distance to the center of gravity of the wall.

Wall Length is the length of the wall section. Maximum length is the overall wall length - offset. Enter the length to be used in the analysis of the particular wall section. Please note that if the Wall length + Offset is greater than the Wall Length + Offset for the level below, this indicates that the section OVERHANGS the section below it. This is not allowed.

Wall thickness is the thickness of this wall section. Enter the thickness to be used in the analysis of a particular wall section. This thickness will be used only between the Analysis Height for that section up to the analysis of the next higher section (or Total Wall Height if it is the highest section).

Effective Depth locates the tension rebar in the panel, and is used to calculate "shear depth" for calculation of actual shear stresses. As with beams, the Effective Depth in a shear wall is measured from the compression edge of the wall to the centroid of the tension chord rebar.

Loads Tab

This main tab has four sub-tabs that allow you to enter four types of loads.

Vertical loads can be of dead, live, roof live and snow types.

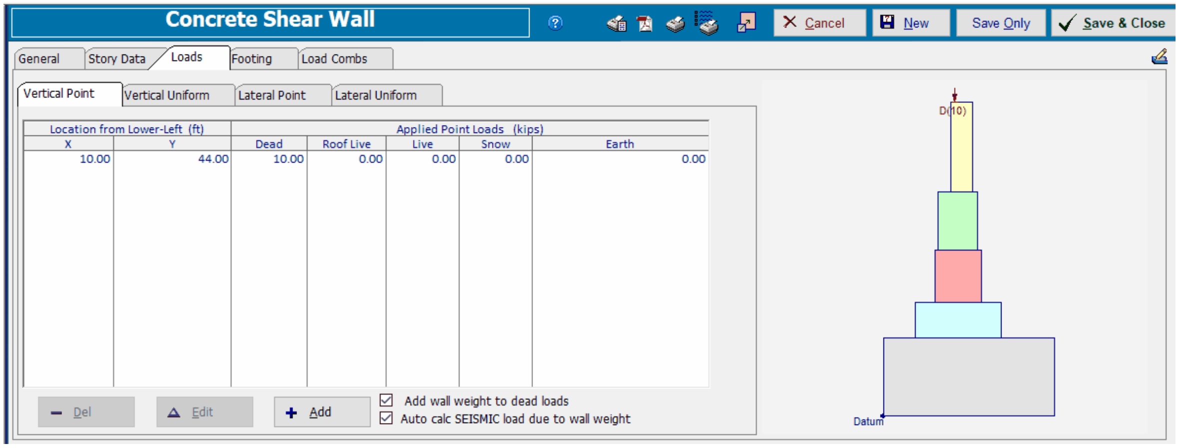
Lateral loads can be of seismic and wind types.

Add wall weight to dead loads will tell the module to calculate the weight of the wall above each analysis height and include it in the vertical dead loads to calculate applied axial stress. It also is used for footing design when that option is selected.

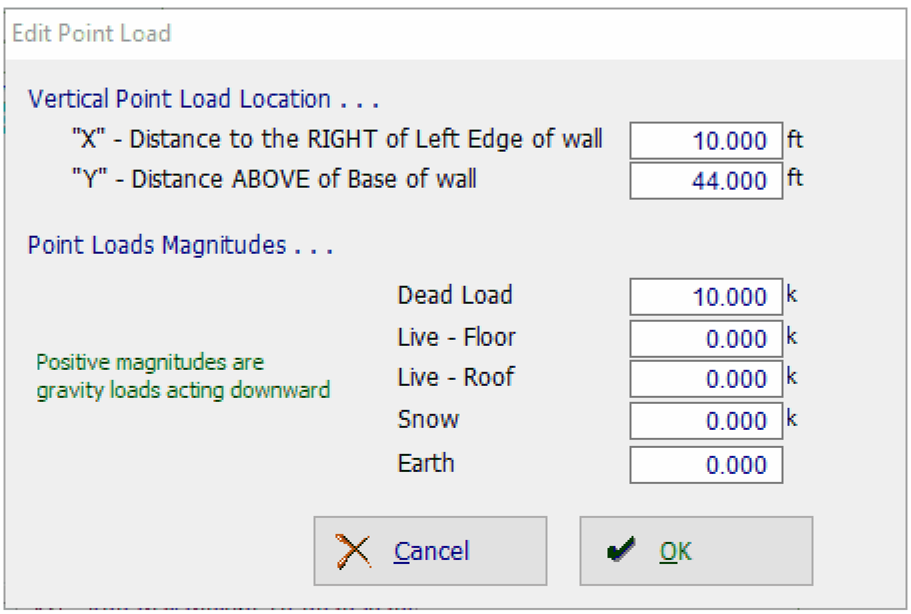
Add wall weight as SEISMIC load will calculate the wall self weight, apply the lateral weight seismic factor and "E" load combination factor. The resulting load will be applied at the wall center to calculate shear and overturning due to that portion of the wall. This is used for values at the analysis height, for the effect of that level's seismic weight on the levels below, and for footing overturning and sliding calculations.

Vertical Point Loads

Use this tab to apply point loads to the wall. You can specify an "X" and "Y" distance from the lower-left corner of the lower wall so that the load can be located anywhere on the defined walls.

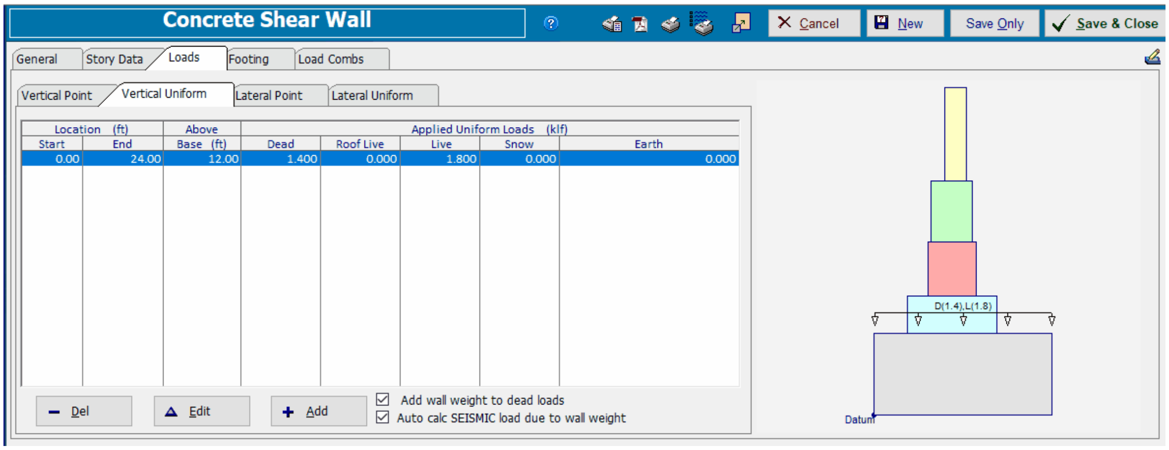


Use the [**Add**] and [**Edit**] buttons to change the values of applied loads. Clicking either button displays this window:

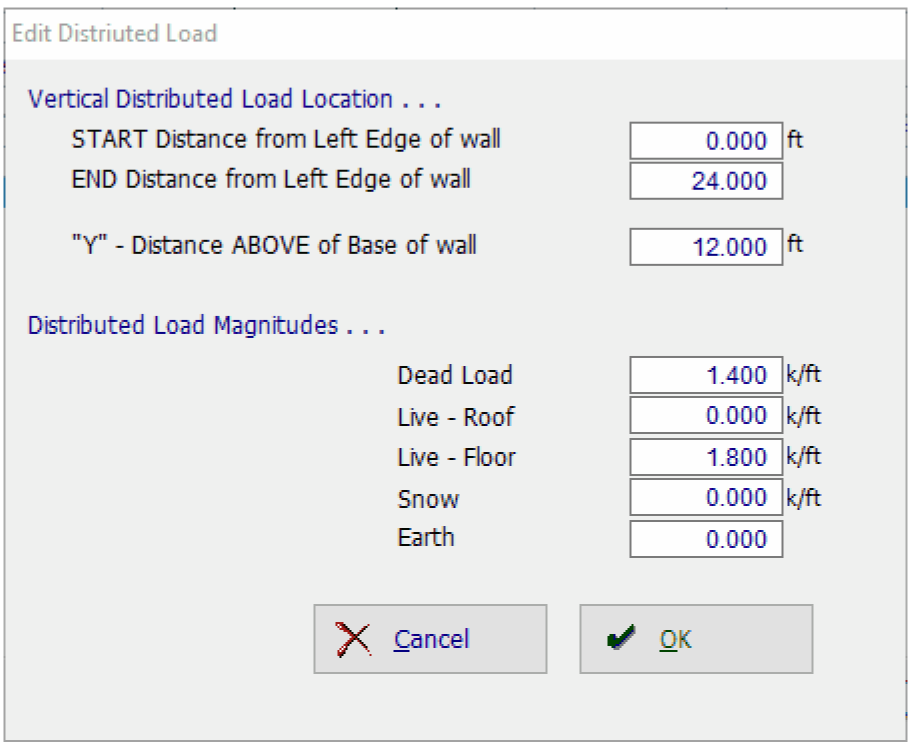


Vertical Uniform Loads

Use this tab to apply uniform loads to the wall. You can specify a "Y" distance from the bottom of the lower wall so that the load can be located at any height.

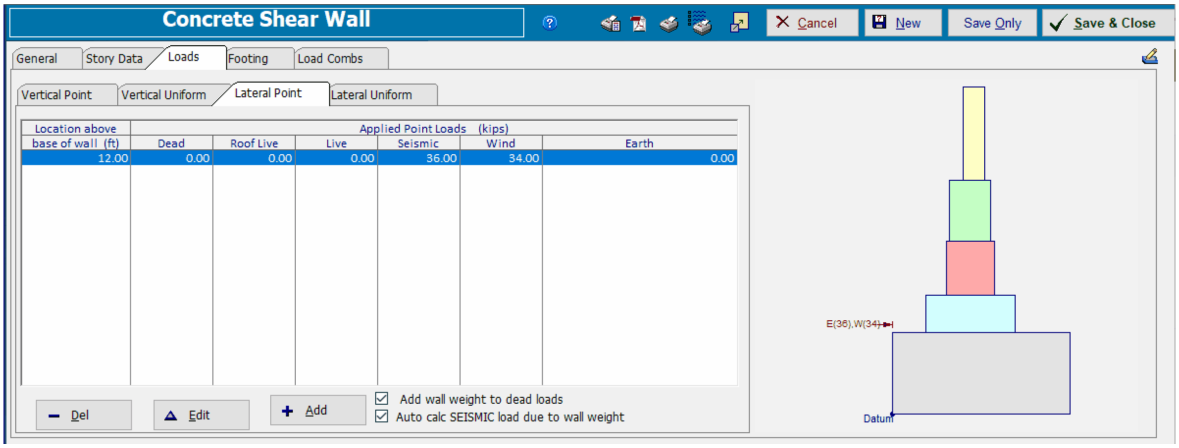


Use the [**Add**] and [**Edit**] buttons to change the values of applied loads. Clicking either button displays this window:

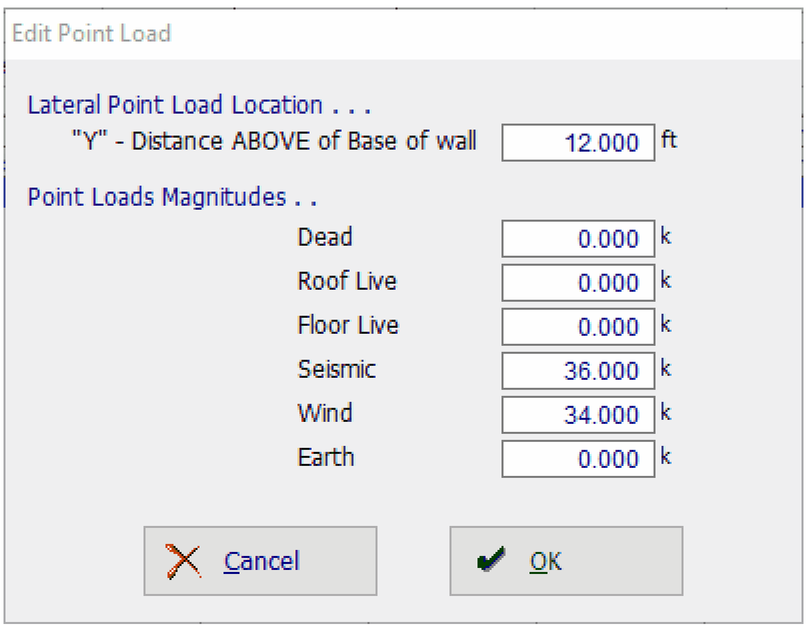


Lateral Point Loads

Use this tab to apply point lateral loads to the wall. You can specify a "Y" distance from the bottom of the lower wall so that the load can be located at any height.

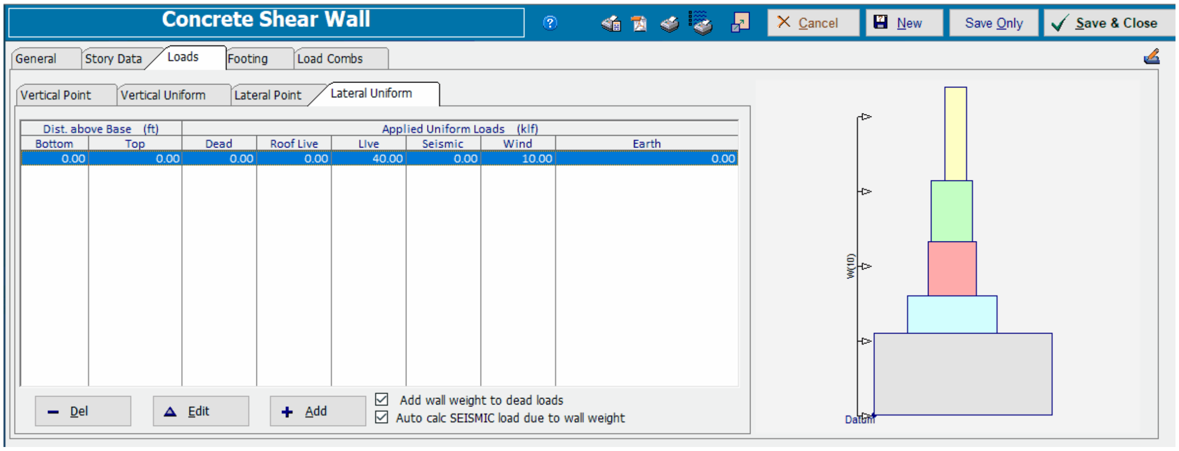


Use the [**Add**] and [**Edit**] buttons to change the values of applied loads. Clicking either button displays this window:

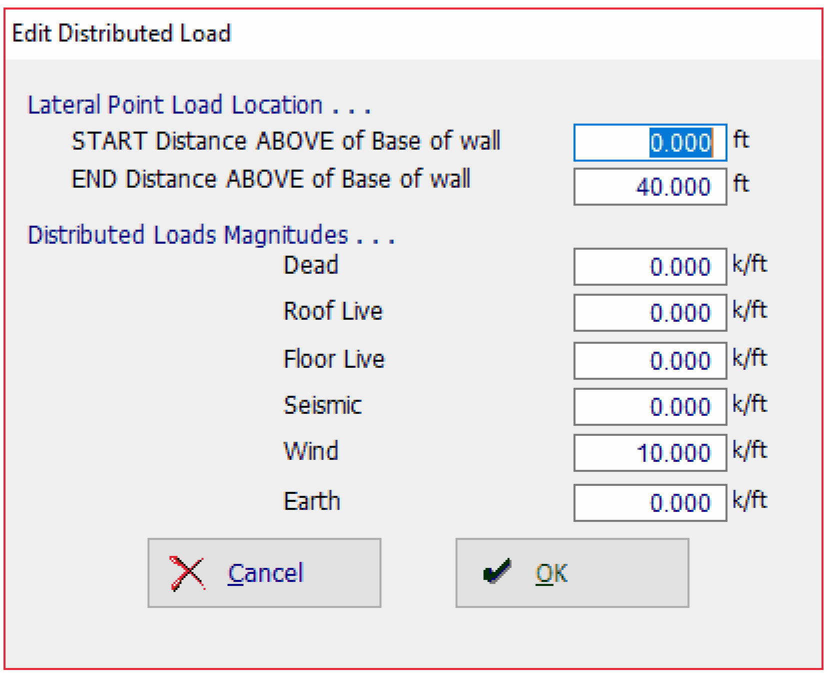


Lateral Uniform Loads

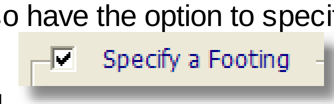
Use this tab to apply uniform lateral loads to the wall. You can specify a Start and End location to define the extent of the lateral load.



Use the [**Add**] and [**Edit**] buttons to change the values of applied loads. Clicking either button displays this window:



Footing Tab

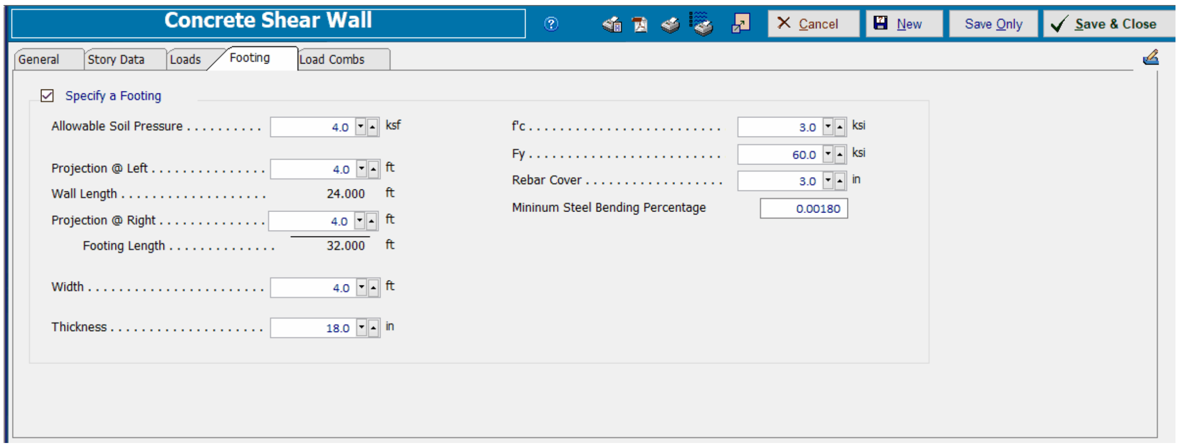
You also have the option to specify a footing under a shear wall by selecting the Specify a Footing  checkbox.

The dimensions of the footing are specified using Distance @ Left and Distance @ Right.

Distance @ Left is the distance that the footing projects past the left edge of the wall (meaning the lower wall level for a multi-story type wall).

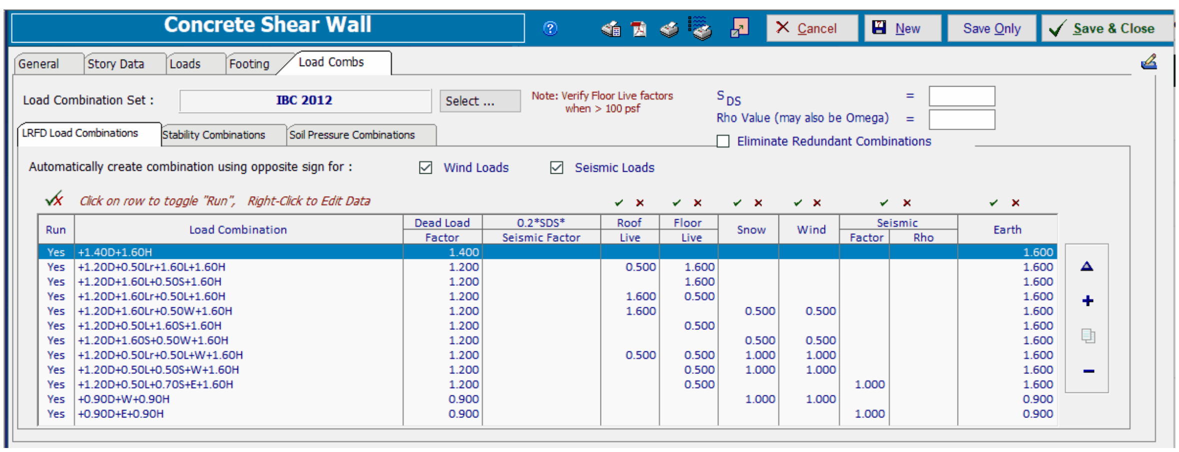
Distance @ Right is the distance that the footing projects past the right edge of the wall (meaning the lower wall level for a multi-story type wall).

Rebar is assumed to exist only at the bottom of the footing to resist tensile forces from the vertical loads and increased pressure due to overturning forces. Tension in the top of the footing in cases where no upward soil pressure exists and the footing weight creates a downward net force IS IGNORED.



Load Combination Tab

The typical load combination tab for strength design of concrete is provided.

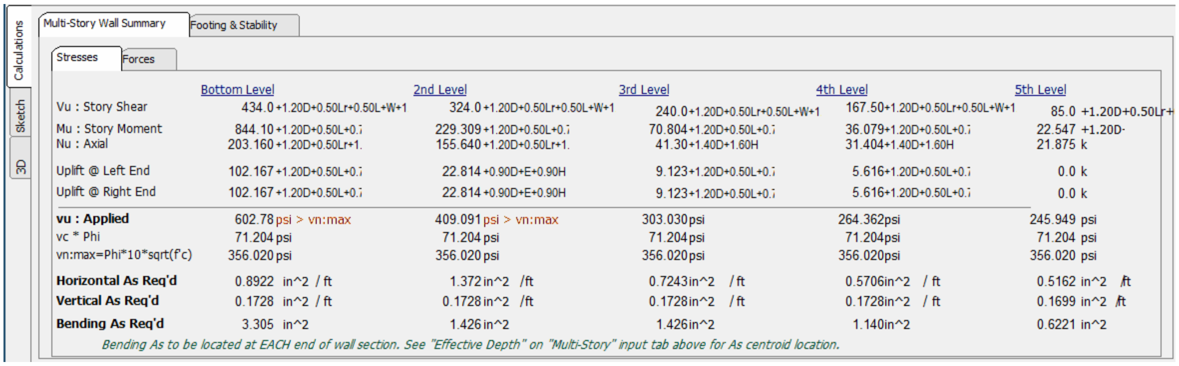


Wall Summary - Stresses Tab

Provides a summary of each level.

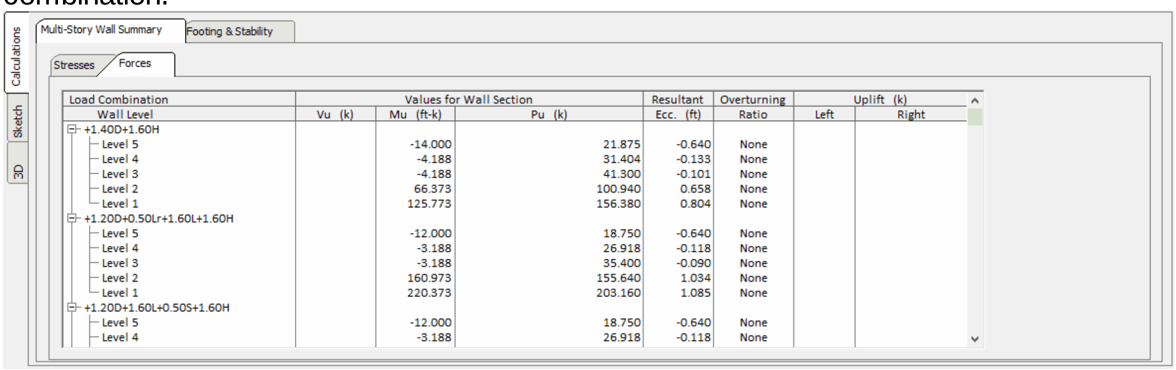
In the top portion you will see the calculated shear, moment and axial loads at the analysis height you have specified. These values are due to wall self weight and applied vertical and lateral loads from that analysis height and above.

In the bottom portion of the screen, the unit shear stresses, shear steel required and end reinforcing for bending tension in that wall section are reported. All calculations are per ACI.



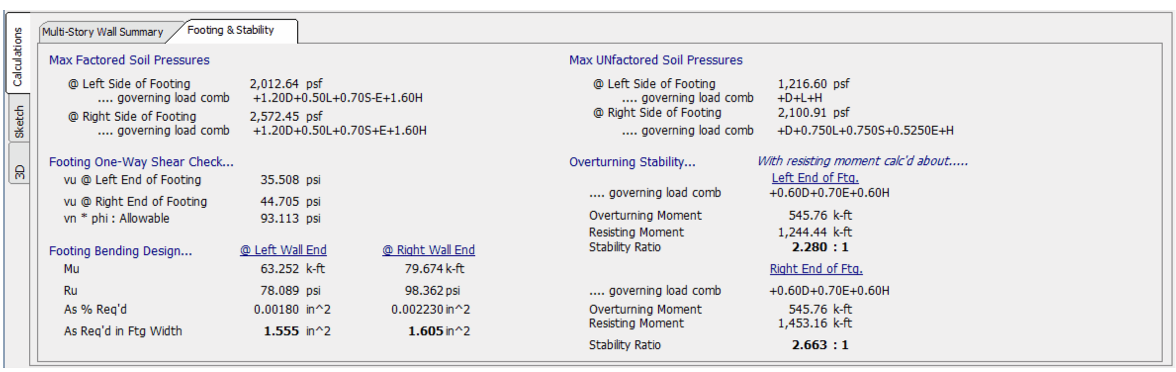
Wall Summary - Forces Tab

This tab provides the detailed force values for each wall level and for each load combination.

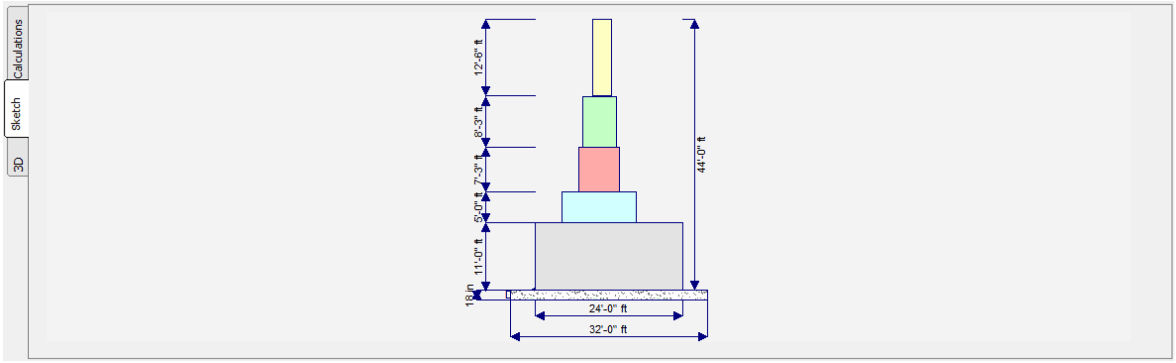


Footing & Stability Tab

This tab provides the calculated service and factored load soil pressure, one-way shear and bending at the end of the wall.



Sketch Tab


Calculations
Sketch
3D
12'-6" ft
8'-3" ft
7'-3" ft
5'-0" ft
11'-0" ft
18 in
44'-0" ft
24'-0" ft
32'-0" ft

10.5.2 Masonry Shear Wall

General

This module allows the design of masonry shear walls including multi-story walls with no openings but with up to five levels of differing length, height, thickness, grouting and reinforcing patterns.

The wall will be composed of two zones:

Chord Zone: The areas at both ends of each level that contain chord rebar in every cell. (Always solid grouted.)

Field of Wall: The area between chord zones that may have a different reinforcing/grouting pattern. (May be partially-grouted or solid-grouted.)

Convenience Features

The module collects data on a level-by-level basis to allow the user to account for varying reinforcing and grouting patterns. It also allows some convenience features such as the ability to specify:

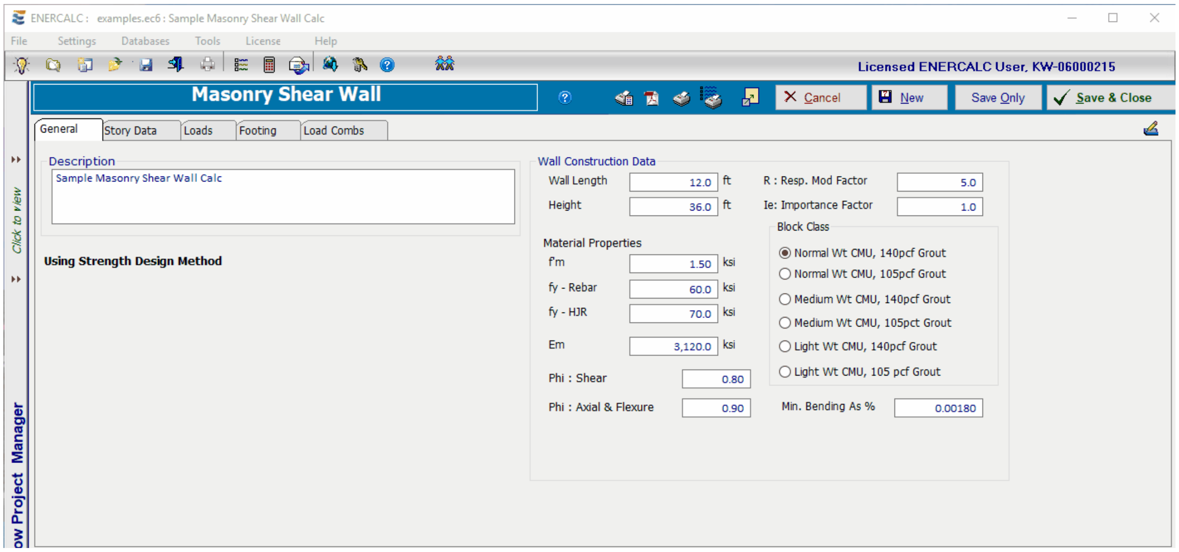
- reinforcing and grouting that can vary from one story to the next,
- wall offsets to account for conditions where the length of wall at one story is shorter than the length of the wall below,
- a continuous footing,
- solid grouting,
- chord rebar that is different from the rebar in the field of the wall,
- bond beams and/or horizontal joint reinforcing for shear reinforcing.

Limitations

Masonry Shear Wall does not allow for the design of openings. It also does not allow for the design of special boundary elements, but it does incorporate the checks to verify that special boundary elements are or are not required.

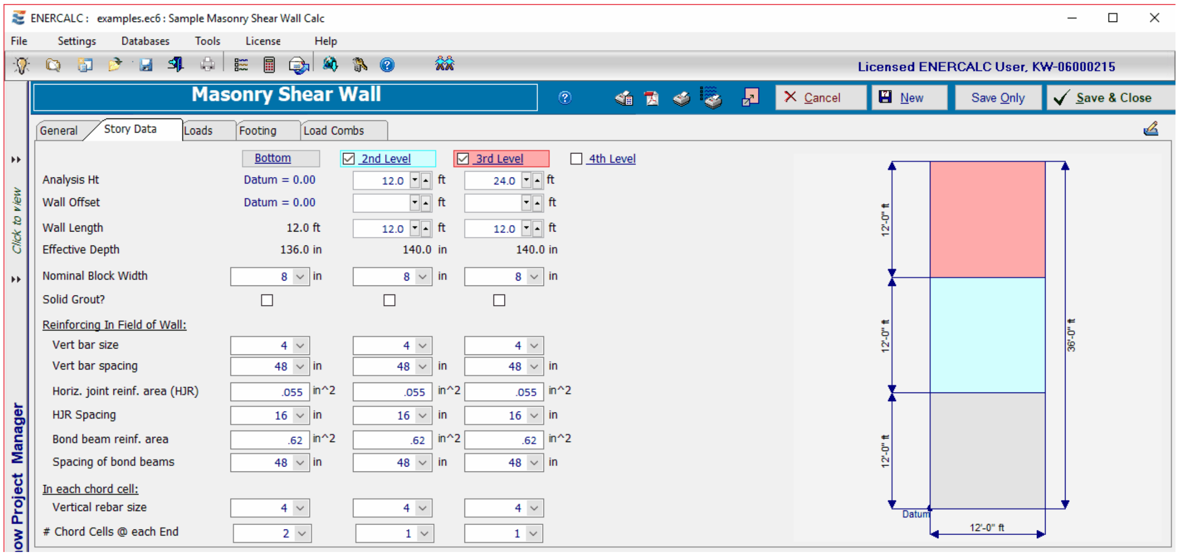
Input Parameters

General tab



Collects overall wall length and height, material properties, and strength reduction factors.

Story Data tab



Select checkboxes as necessary to specify the number of stories. Shear wall is assumed to be laterally braced at each defined story level.

Analysis Height:	The heights at which the story framing is assumed to brace the shear wall.
Wall Offset:	A lateral offset dimension that can be used to shift the left edge of the wall at the selected level.
Wall Length:	The distance from the left end of the wall to the right end, at the selected level. This can be used to shift the right edge of the wall.

Effective Depth: The dimension from the compression edge of the wall to the centroid of the chord steel.

Nominal Block Width: Use the dropdown to select the block width.

Chord Cells @ each End: Specify the number of solid-grouted reinforced chord cells at each end of the selected level.

Solid Grout: Reinforced cells will always be assumed to be grouted, but this option provides a way to tell the program that ALL cells in the selected level will be grouted, regardless of whether they contain rebar or not.

Vertical Bar Size: Use the dropdown to select the size of the vertical rebar that will be used in the field of the wall.

Vertical Bar Spacing: Specify the spacing of vertical rebar in the field of the wall.

Horizontal Joint Reinforcing (HJR) Area: Specify the effective cross-sectional area of one piece of horizontal joint reinforcing, if it is to be considered as shear reinforcing.

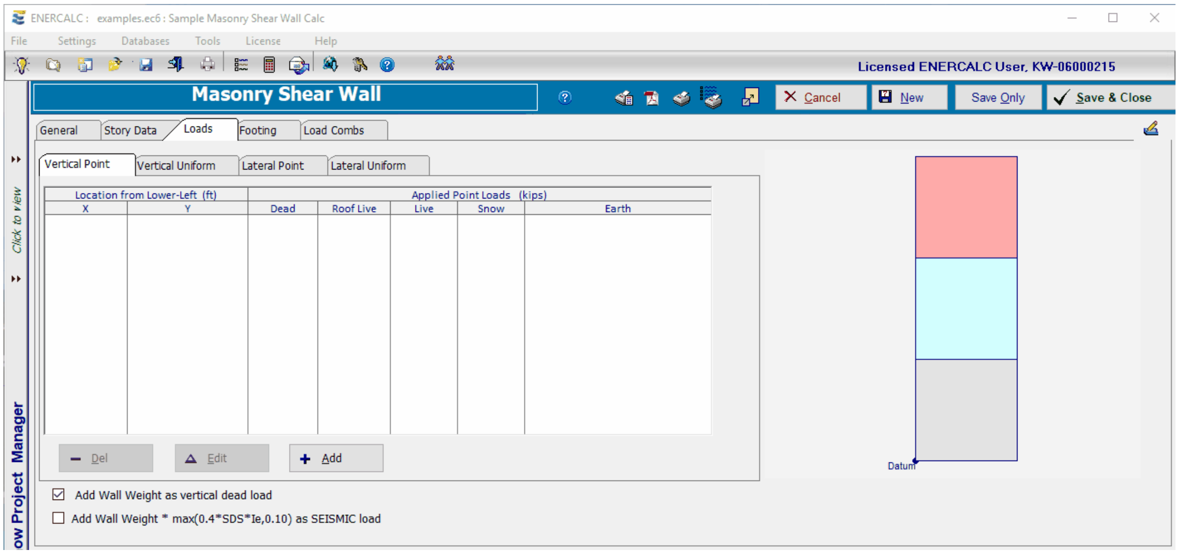
Horizontal Joint Reinforcing (HJR) Spacing: Specify the vertical spacing of horizontal joint reinforcing, if it is to be considered as shear reinforcing.

Bond Beam Reinforcing Area: Specify the effective cross-sectional area of rebar in one bond beam, if it is to be considered as shear reinforcing.

Spacing of Bond Beams: Specify the vertical spacing of bond beam reinforcing, if it is to be considered as shear reinforcing.

Vertical Rebar Size (Chords): Use the dropdown to select the size of the vertical rebar that will be used in each solid-grouted chord cell.

Loads

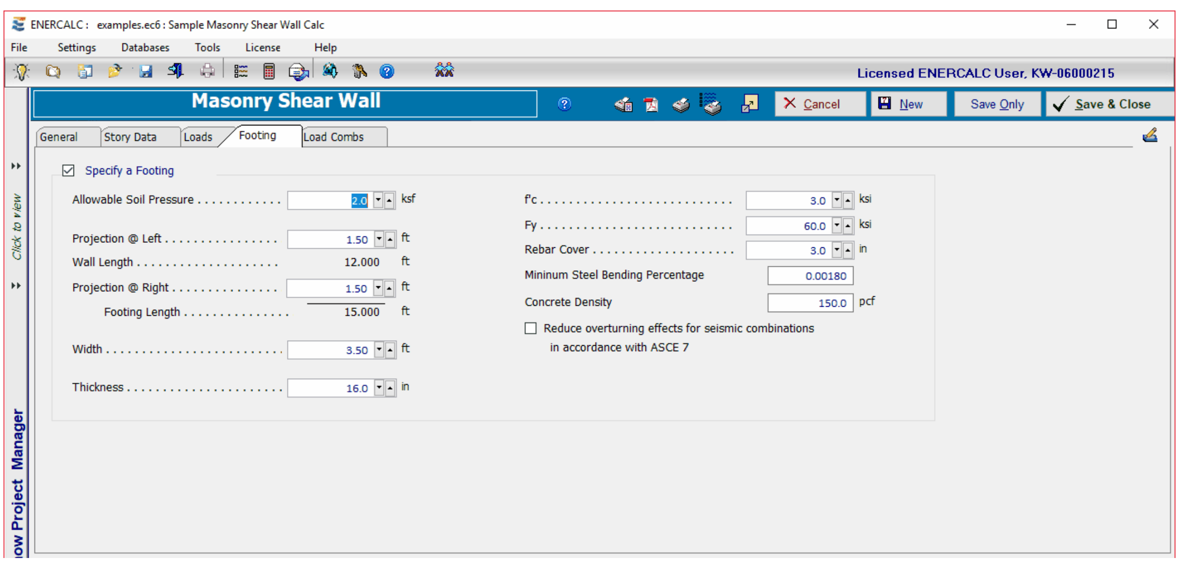


Use the tabs to specify magnitude and location of:

- Vertical Point Loads
- Vertical Uniform Loads
- Lateral Point Loads, and
- Lateral Uniform Loads.

Also offers the option to automatically consider wall weight as vertical dead load and/or automatically calculate seismic load due to wall weight.

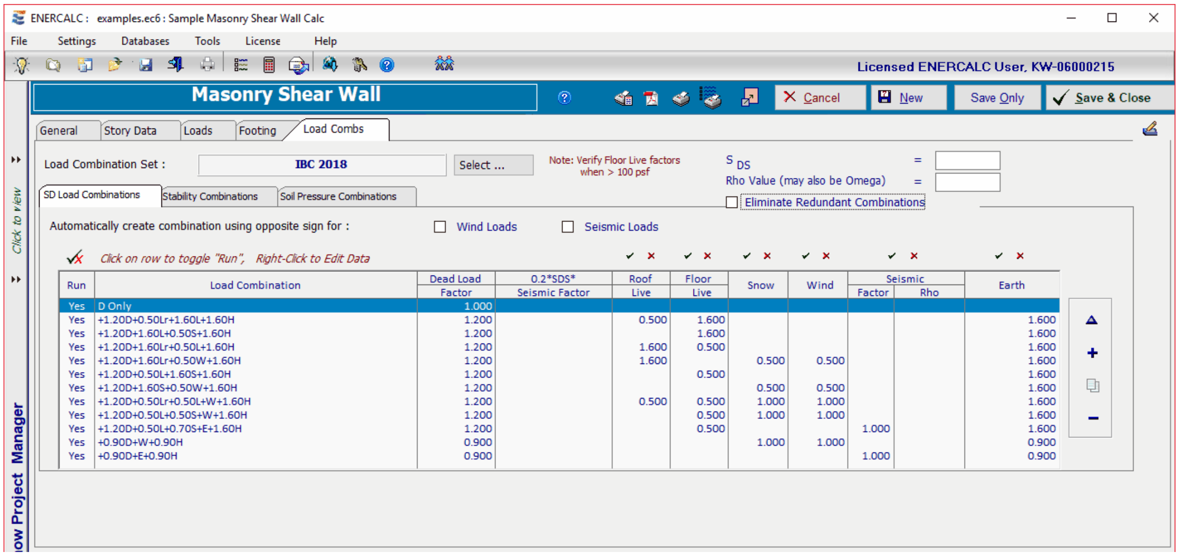
Footing



Optional tab to design a footing that behaves as a continuous grade beam that cantilevers past the end of the wall.

Specify geometry, allowable soil pressure, and material properties to arrive at checks for soil pressure and footing flexure.

Load Combinations

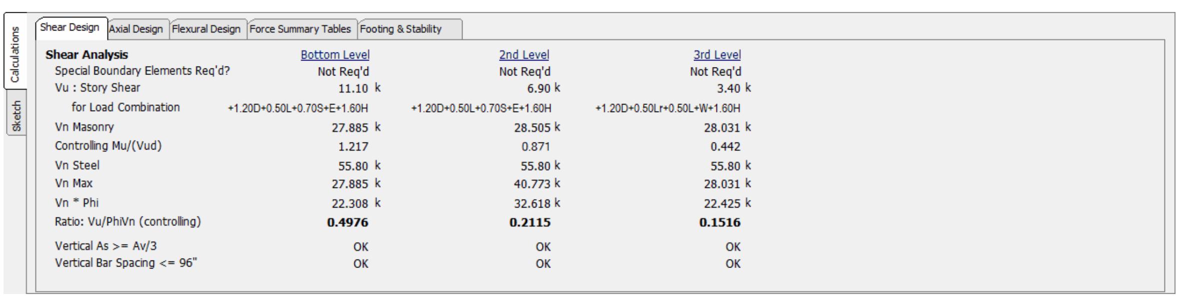


Collects settings that control how load combinations will be generated and applied to the analysis/design of:

- Masonry and reinforcing
- Footing and reinforcing
- Stability
- Soil Pressure

Output

Shear Design

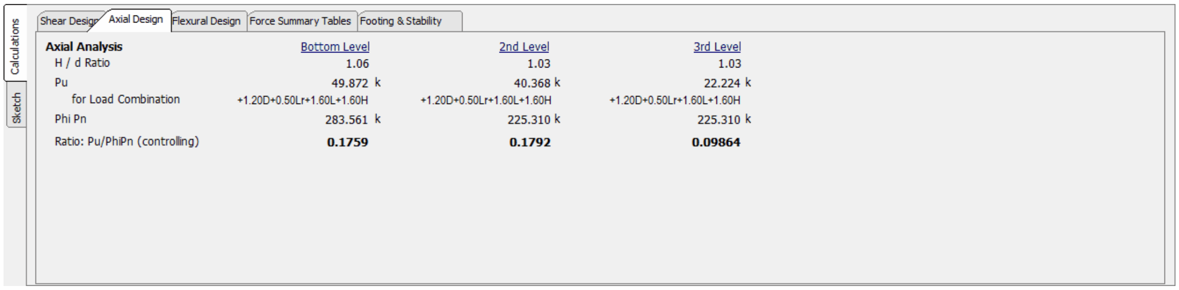


Reports:

- Factored shear force and controlling load combination
- Shear strength from masonry
- Shear strength from reinforcing

- Limiting shear strength
- Design shear strength
- Design ratio
- Need for special boundary elements
- Code requirements for area and spacing of vertical rebar.

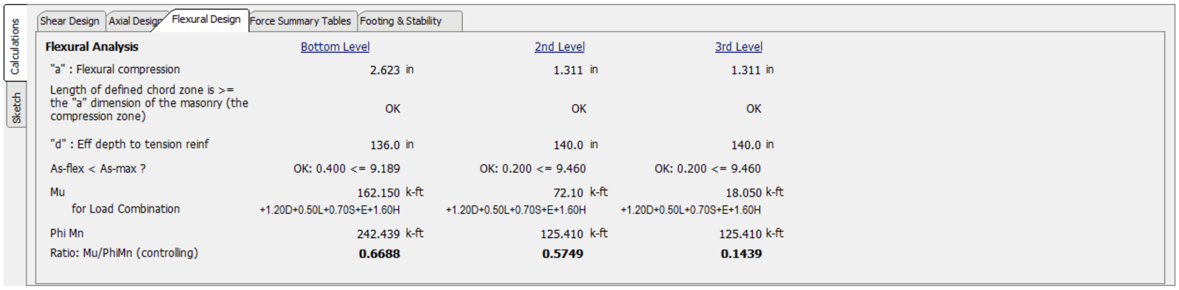
Axial Design



Reports:

- H/d ratio
- Factored axial load and controlling load combination
- Design axial load
- Design ratio

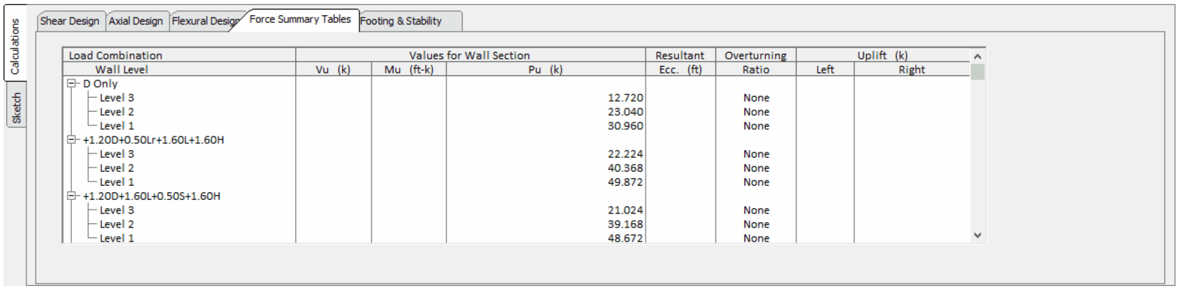
Flexural Design



Reports:

- Length of chord zone
- Comparison of As flex to As max
- Factored moment and controlling load combination
- Design moment
- Design ratio

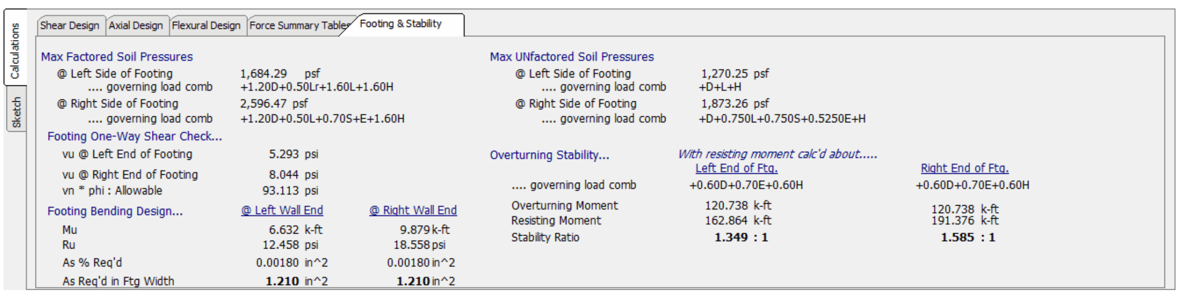
Force Summary Table



For each load combination, this table reports:

- Vu
- Mu
- Pu
- Eccentricity
- Overturning Ratio
- Uplift

Footing & Stability



Reports:

- Maximum Factored Soil Pressures and controlling load combinations
- Maximum Unfactored Soil Pressures and controlling load combinations
- Footing One-Way Shear Check
- Footing Bending Design Values
- Overturning Stability Results and controlling load combinations

10.5.3 Wood Shear Wall

Basis

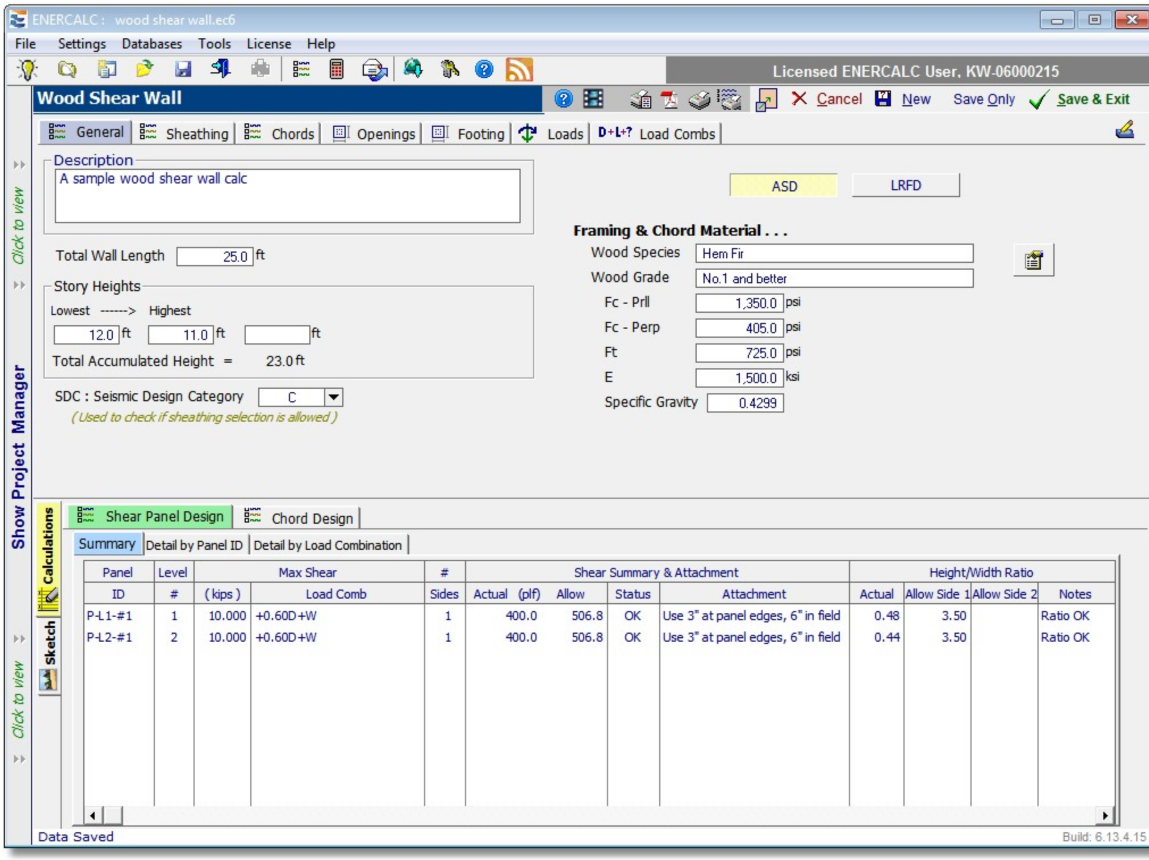
The NDS Special Design Provisions for Wind & Seismic allows for three approaches to wood shear wall design:

1. Force Transfer Shear Walls
2. Perforated Shear Walls
3. Individual Full-Height Wall Segment Shear Walls.

Force Transfer Shear Walls and Perforated Shear Walls allow the designer to use more favorable analysis assumptions, and offers some benefits in terms of construction details. But it comes at a cost of the designer having to perform more calculations and designs to follow the load path through all of the headers, jambs, sills, etc., and their connections.

At present, the Wood Shear Wall module implements the Individual Full-Height Wall Segment Shear Walls method. This is the most straightforward method from the design standpoint. This method still requires the evaluation of h/b ratios to decide whether to eliminate slender segments or suffer the imposed reduction on the nominal shear capacity based on the slenderness of each panel in the wall. (Section 4.3.5.1 refers back to Section 4.3.4, which includes this reduction.) But this method does not require consideration of the reduction factor (Co) that applies to Perforated Shear Walls.

The implementation within Structural Engineering Library goes a little further by allowing the user to specify openings. The module then takes the defined openings and considers only the solid stacked portions of remaining wall as being effective at resisting lateral loads. Section 4.3.3.4 provides the justification for summing the resistance of multiple Individual Full-Height Wall Segments because, by our definition within the module, all segments will be “sheathed with the same materials and construction”.



Overview

The Wood Shear Wall module allows the user to define overall geometry, openings (if any), sheathing type, chord member species, grade, and size, applied loads, and a footing (if desired). The module then evaluates the resulting shear panels, chords, and footing (if defined).

Sheathing is evaluated for aspect ratio and unit shear due to load combinations that include either wind or seismic. The module considers the selected sheathing type, sheathing thickness, fastener size, blocking condition, and the species of the supporting framing. It then automatically incorporates any necessary adjustments to the nominal unit shear values from the 2008 Special Design Provisions for Wind and Seismic on the basis of sheathing type, aspect ratio, and the species of the supporting framing. The result is a required fastener spacing for each panel in the wall.

Chords are evaluated for tension and compression due to load combinations that include either wind or seismic. The moment in a given shear panel is assumed to be coupled out at the location of the chords, resulting in tension and compression forces. Any applied vertical loads that may be present are combined with the wind or seismic chord forces, and the resulting loads on the chords are evaluated as per the requirements of the NDS.

Footings (if defined) are evaluated for soil bearing pressure, overturning, and one-way shear and flexure of the cantilevered end projection.

There are certain items that are not explicitly evaluated by this module, and the user should be aware of the following exclusions: bending or shear in the top plate, bearing on the top plate, out of plane design of the wall sheathing or framing, gravity-only loading on the common studs or chords, bearing on the bottom plate, design of anchorage hardware or connections.

Workflow Process

The general workflow proceeds as follows:

1. The overall height and length of the shear wall are defined on the General tab, along with selections such as the Seismic Design Category, the Design Method, and the Framing and Chord Species and Grade.
2. The sheathing type, thickness, fastener size and blocking conditions is selected on the Sheathing tab. This tab also allows for the specification of sheathing on the second side of the framing when necessary.
3. The chord size is specified on a level-by-level basis on the Chords tab. This tab also allows for the specification of the bracing assumption to be applied in the compression design of the chords.
4. If openings are present in the wall, they can be defined on the Openings tab.
5. Loads of many types can be defined and applied on the Loads tab.
6. If a continuous footing design is desired, the footing geometry and material properties can be entered on the Footing tab.
7. The Load Combinations tab allows for the definition of the load combinations that will be used for the design.
8. The results for the shear panels, chords, and footing (if designed) can be reviewed in the Results panel in the lower portion of the screen.

General tab

Description: Enter a freeform description of the current wall design for reference.

Total Wall Length: Enter the overall length of the shear wall in units of feet.

Story Heights: Enter the heights of up to five stories in units of feet. Whenever a value is entered, the input field for the next story height is displayed in case it is needed. The total accumulated height is automatically reported.

Seismic Design Category: Enter the appropriate Seismic Design Category for the wall being designed. This is used to check the selected sheathing type to be sure it is permissible for use in that SDS according to the Special Design Provisions for Wind and Seismic.

Design Method: Select ASD or LRFD to dictate which method will be applied when designing shear panels and chords. Footing design (when requested) is always by LRFD methods.

Framing & Chord Material: Use the icon to access the Wood Reference Design Values database in order to select the species and grade of wood used for the chords and common framing in the wall.

Sheathing tab

Select SDPWS Construction Table: Choose the table from which the sheathing will be selected.

Select Main Sheathing: Select a tabular entry to represent the sheathing type, sheathing thickness, fastener size/penetration, and in some cases the blocking condition.

Nominal Shear Capacities: The program displays the nominal shear capacities for seismic design and for wind design directly from the selected table. These are nominal values which still need to be modified for use in design, such as with a phi factor or factor of safety and with any applicable adjustments such as for aspect ratio or specific gravity of the supporting framing.

Sheathing is Blocked: (Only visible for some tabular selections) Use this checkbox to indicate if the sheathing is blocked or not. This setting has an effect on some of the allowable aspect ratios.

Table 4.3A Footnote 2 is applicable: (Only visible for some tabular selections) Use this checkbox to indicate if a specific condition exists as described in detail in the referenced footnote in Table 4.3A of the 2008 Special Design Provisions for Wind and Seismic.

Sheathing on 2nd Side: Use this checkbox to indicate that there is a sheathing of some sort on the other side of the framing.

Use Same as Main Sheathing: A convenience option that automatically sets the Sheathing on 2nd Side to be identical to the sheathing on the Main side.

Note: If sheathing is specified on the second side, and if the sheathing comes from the same Construction Table as the Main Sheathing, then the blocking setting for the 2nd side will automatically be assumed to be the same as the blocking setting for the Main side.

Chords tab

The Chords tab will automatically display one row of chord definition data for each story that was defined on the General tab.

Chord Size: Use the drop-down list box to select the size of the sawn lumber member(s) that will be used at the chord locations for the given level. Note that the program will automatically determine the *number* of chord members required at each location. So for example, if the wall was generally going to be constructed of 2x4 framing, then this input should be set to "2x4". Once the analysis and design has been completed, the program will report how many 2x4s should be ganged together to safely resist the imposed chord forces at each location.

CF: Size Factor: Enter the appropriate Size Factor for the chord member size, species, and grade being entered.

Area per Chord: The program reports that cross sectional area of one member of the selected size for reference.

Maximum Chord Stress Ratio: Enter the maximum permissible chord stress ratio. Typically 1.0 using current design methods and load combinations.

Chord Strength Calculation: This setting offers two options for defining the bracing of chord members when the allowable compression stress is calculated.

- The option named "*Treat all chords as fully braced about both axes*" implies that all chords are braced against column buckling. The physical model for this option might be a situation where chord members always occur at "L" or "T" shaped intersections, such that the sheathing prevents buckling in the plane of the shear wall, and the intersecting wall prevents buckling out of the plane of the shear wall. (These conditions are probably not common.)
- The option named "*Assume all chords unbraced out of plane of wall for story height*" implies that the sheathing prevents buckling in the plane of the shear wall, but nothing prohibits column buckling of the chord member out of the plane of the wall. This is the condition for the chords at the end of an isolated straight panel of wall where no perpendicular walls intersect the shear wall at the chord locations.

Openings tab

The Openings tab allows openings to be defined in the shear wall.

Add: The Add button opens the Add Opening dialog. The dialog automatically numbers the openings that are added, and it collects the geometric information necessary to locate and size the opening. The Add Opening dialog also checks the alignment of defined openings to ensure that all jambs are aligned.

Edit: By selecting an existing opening in the list, the Edit button allows the selected opening to be revised.

Delete: By selecting an existing opening in the list, the Delete button allows the selected opening to be removed.

Renumber: The Renumber button allows existing openings to be renumbered in a logical order by working from left to right and then from bottom to top.

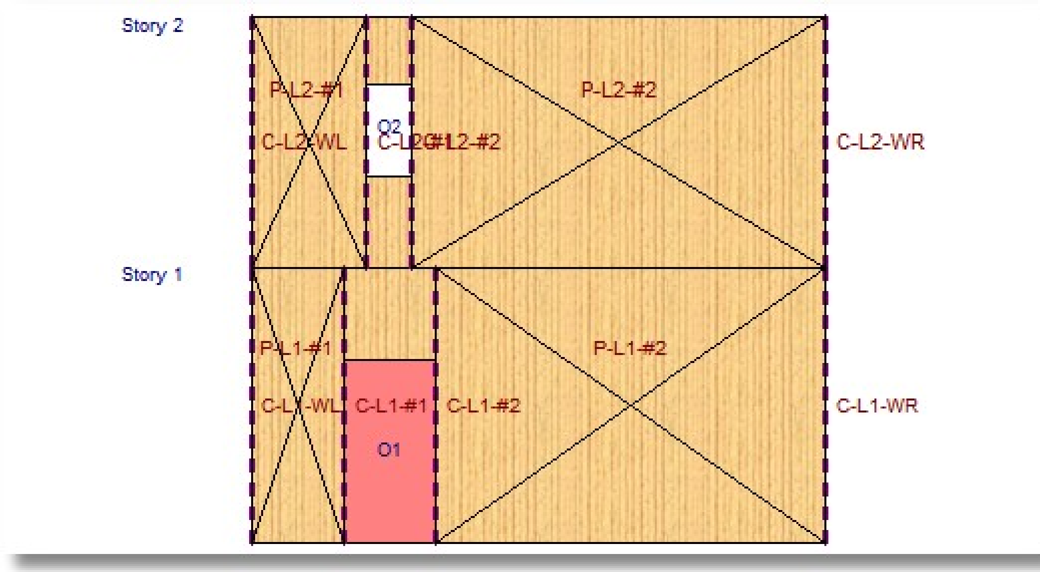
The graphic display offers checkboxes to independently display or hide:
- Openings
- Shear Panel callouts
- Chord graphics and callouts
- Panel and opening dimensions
- All loads

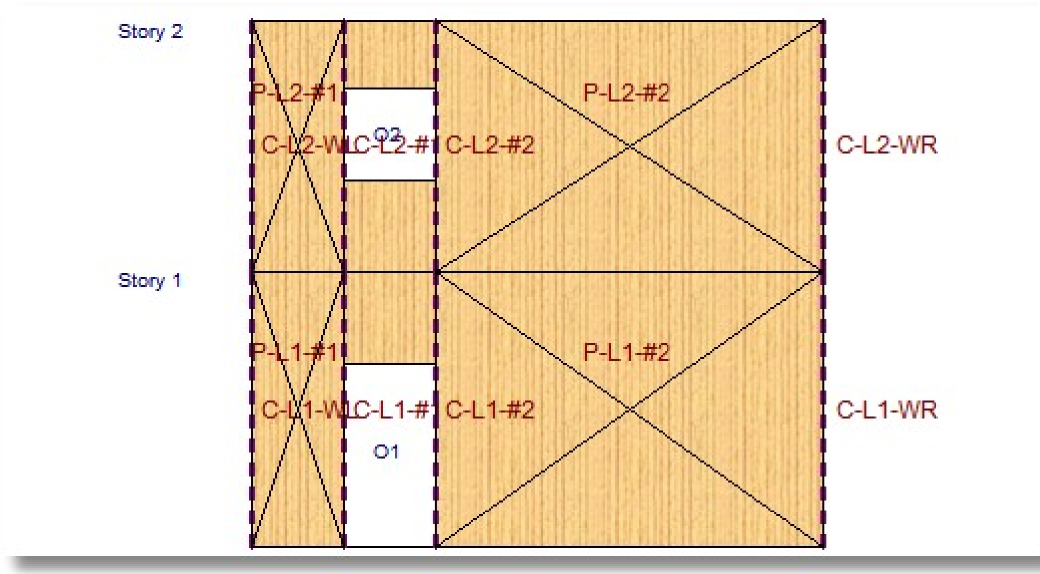
Note: This module is predicated on the design assumptions of the Individual Full-Height Wall Segment Shear Walls method. This method disregards the shear resistance of portions of shear walls above and below openings. This leads to two **important** considerations regarding the use of openings:

1. In multi-story models, any opening in one story must not partially overlap an opening in a different story. Envision vertical lines at the jambs of an opening, and project them to the top and bottom of the entire wall. Those jamb reference lines must not cross any other opening. The Add Opening dialog will warn if this condition is being violated by having partially-overlapping openings in any story. This may require enlarging some openings so that they respect the jamb locations of adjacent openings, or it may require applying the module only to the solid portions of the wall that remain after the openings have been omitted.

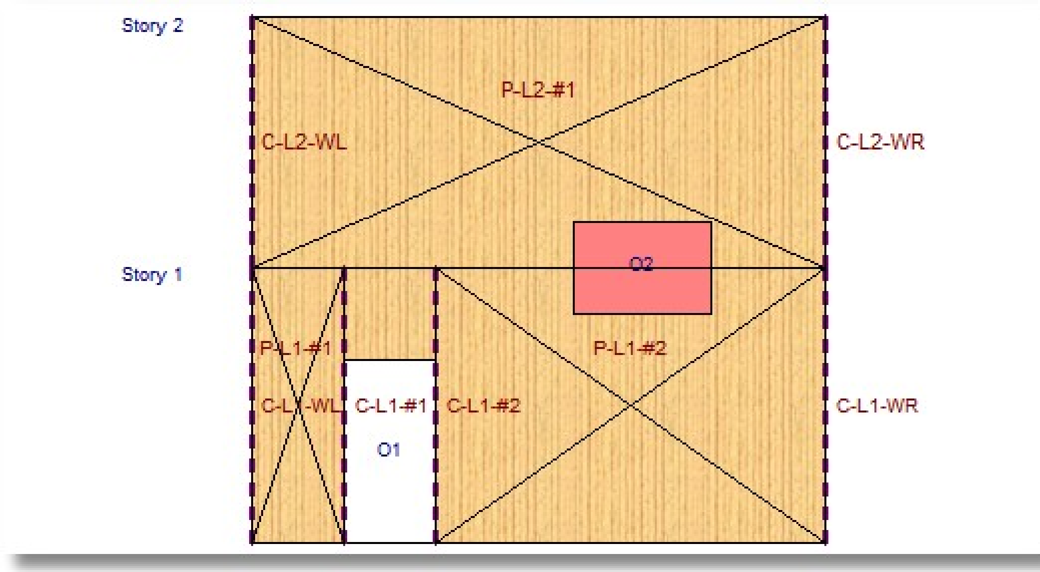
2. The presence of an opening within a given story, creates a panel that is ineffective at resisting shear. The ineffective panel is defined by the width of the opening and it extends for the full height of that story. This means that it is not necessary to model stacked openings within a given story. In fact, doings so will cause an error.

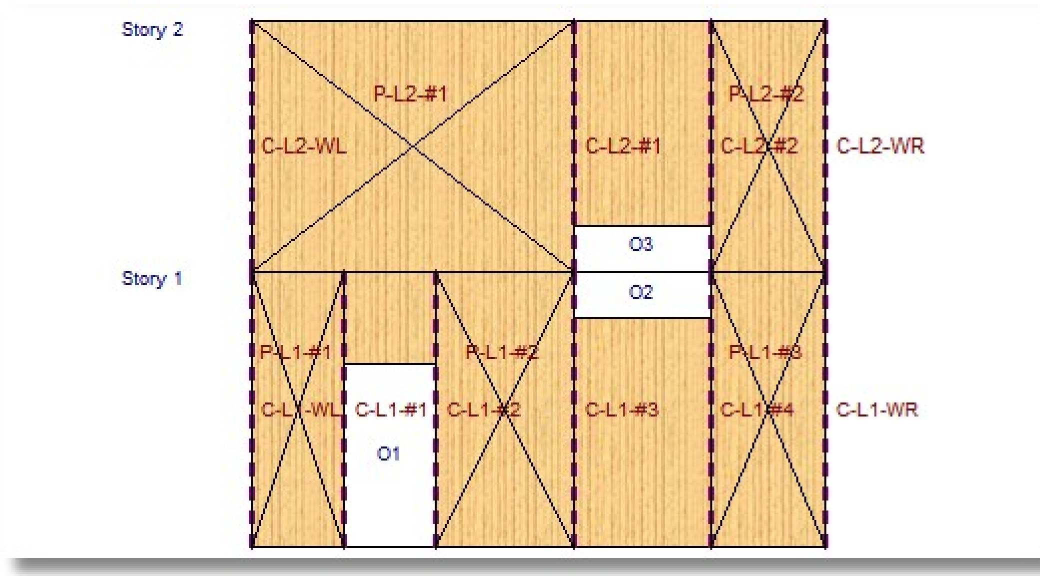
See the following diagrams for some acceptable and unacceptable opening geometries:



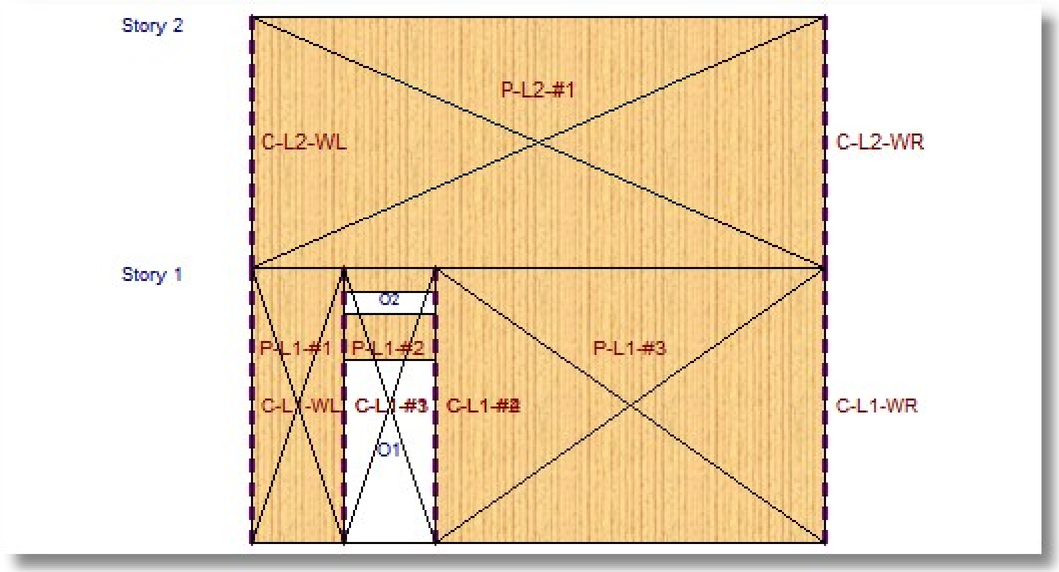


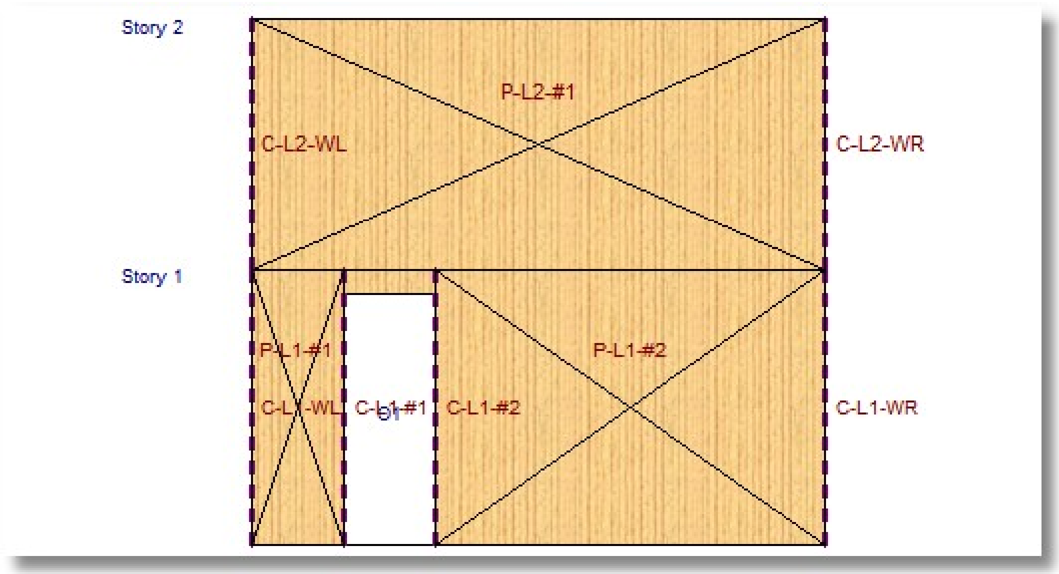
NG - Jambs of openings do NOT align --> Here is how to fix it --> OK - Jambs of openings align properly





NG - Opening crosses top of story --> Here is how to fix it --> OK - Large opening modeled as 2 separate openings





NG - Stacked openings within a story --> Here is how to fix it --> OK - One opening accomplishes the task

Note: When using the self-weight functions for Dead Load and for seismic load, the program does not deduct any weight for openings. In other words, the program assumes that the unit weight of the window, door, louver, etc. in the opening, is approximately the same as the unit weight of the wood-framed construction it is replacing.

Footing tab

Specify a Footing: If this checkbox is selected, then the input fields for defining a footing are displayed. Note: This checkbox also causes some footing-related input to be shown or hidden as appropriate on the Loads tab.

Allowable Soil Pressure: Specify the allowable soil bearing pressure in units of ksf.

f'c: Specify the compressive stress of concrete in units of ksi.

Fy: Specify the yield stress of rebar in units of ksi.

Rebar Cover: Specify the cover over rebar in units of inches. The program will use this value and an allowance for the rebar size to calculate the effective depth of the concrete section.

Minimum Steel Reinforcing Percentage Based on Thickness: Specify the minimum permissible ratio of rebar area to gross area of concrete.

Concrete Density: Specify the density of concrete in units of pcf.

Footing Width: Specify the width of the footing in units of ft. This is the dimension perpendicular to the wall length.

Footing Thickness: Enter the thickness of the footing in units of inches.

Projection @ Left: Enter the projection of the footing beyond the left end of the wall in units of ft.

Wall Length: The program reports the shear wall length in units of ft for reference.

Projection @ Right: Enter the projection of the footing beyond the right end of the wall in units of ft.

Footing Length: The program reports the footing length in units of ft for reference.

Loads tab

The Loads tab provides several sub-tabs for the convenient definition of many different types of loads that could potentially act on a shear wall system.

General sub-tab

The Wall Self Weight category offers inputs for the following:

Consider Wall Weight: This category provides options to automatically consider the wall self-weight:

- As vertical Dead load - a way to request that the program use the unit weight described above to calculate a dead load

- As Seismic load - a way to request that the program use the unit weight described above to calculate a lateral seismic load using the value of Cs described below

Weight: (Only displayed when one of the options above has been selected.) Enter the unit weight of shear wall construction in psf.

Cs: Seismic Response Coefficient: (Only displayed when the "As Seismic load" option above has been selected.) Collects the value of Cs by which the unit weight would be multiplied to determine the seismic load due to the self-weight of the wall if that option is selected.

The Soil over Footing category is only displayed if the "Specify a Footing" option is selected on the Footing tab. This category offers inputs for the following:

Soil depth over footing: Enter the depth of soil above the top of the footing in units of inches. This value is used to determine loading on concrete footings when one is designed. This value will affect the reinforced concrete design, the soil bearing pressure, and the overturning ratio.

Soil density: Enter the density of soil above the top of the footing in units of pcf. This value is used to determine loading on concrete footings when one is designed. This value will affect the reinforced concrete design, the soil bearing pressure, and the overturning ratio. If no footing design is requested, this value has no influence on the remaining calculations.

The Added Overburden Load over Footing category is only displayed if the "Specify a Footing" option is selected on the Footing tab. This category allows the user to specify superimposed load on the top of the footing for all of the common load cases. This value will affect the reinforced concrete design, the soil bearing pressure, and the overturning ratio.

Vertical Point sub-tab

Add: The Add button opens the Add Point Load dialog. The dialog allows vertical point loads of all load cases to be defined and located with respect to the lower left corner of the wall. Note: Positive magnitudes are assumed to act downward.

Edit: By selecting an existing load in the list, the Edit button allows the selected load to be revised.

Delete: By selecting an existing load in the list, the Delete button allows the load opening to be removed.

Vertical Uniform sub-tab

Add: The Add button opens the Add Uniform Load dialog. The dialog allows vertical uniform loads of all load cases to be defined and located with respect to the lower left corner of the wall. Note: Uniform loads can be specified as partial-length loads by specifying the start and end locations of the load with respect to the left edge of the wall. Positive magnitudes are assumed to act downward.

Edit: By selecting an existing load in the list, the Edit button allows the selected load to be revised.

Delete: By selecting an existing load in the list, the Delete button allows the load opening to be removed.

Lateral Point sub-tab

Add: The Add button opens the Add Point Load dialog. The dialog allows lateral point loads of all load cases to be defined and located with respect to the lower edge of the wall. Note: Positive magnitudes are assumed to act to the right.

Edit: By selecting an existing load in the list, the Edit button allows the selected load to be revised.

Delete: By selecting an existing load in the list, the Delete button allows the load opening to be removed.

Lateral Uniform sub-tab

Add: The Add button opens the Add Uniform Load dialog. The dialog allows lateral uniform loads of all load cases to be defined and located with respect to the lower edge of the wall. Note: Uniform loads can be specified as partial-height loads by specifying the start and end locations of the load with respect to the bottom edge of the wall. Positive magnitudes are assumed to act to the right.

Edit: By selecting an existing load in the list, the Edit button allows the selected load to be revised.

Delete: By selecting an existing load in the list, the Delete button allows the load opening to be removed.

The graphic display offers checkboxes to independently display or hide:
- Openings
- Shear Panel callouts
- Chord graphics and callouts
- Panel and opening dimensions
- All loads - otherwise only the loads associated with the selected tab are displayed

Load Combinations tab

The Load Combinations tab indicates the currently selected load combination set.

The load combination set can be changed by clicking the Change Load Combination Set button.

The padlock icon can be used to unlock the individual load factors so they can be edited. Note: Editing load factors in this way will not alter the load factors that are stored in the selected load combination set. The edits will only affect the currently active calculation.

The Load Combinations tab has two sub-tabs: Service Combinations and Factored Combinations.

Service Combinations sub-tab

The Service Combinations tab offers the service-level combinations from the selected load combination set. These load combinations will be used for wood design when the ASD Design Method is selected. They will also be used to determine soil bearing pressures when the option is selected to design a footing.

A selected checkbox in the "Run" column indicates that the associated load combination will be considered. The "Run" button offers quick convenience options for changing the selection status of many load combinations at one time.

The C_D column is only visible when the ASD Design Method is selected. It indicates the value of the Load Duration Factor for each load combination. Click the C_D button to quickly set the values of C_D for all load combinations based on the load case with the shortest duration.

Use additional factors per ASCE 12.4.2.3: Select this button to instruct the module to consider the additional contribution to the Dead Load due to the vertical component of seismic load prescribed in ASCE 12.4.2.3.

Auto Reverse Wind: Select this button to instruct the module to also consider the algebraic negative of the defined Wind loads. (This may be useful for quickly creating a load combinations that reverse the direction of application of applied wind loads by creating a "sister" load combination that uses the negative version of the wind component for each load combo that normally incorporates +W.)

Auto Reverse Seismic: Select this button to instruct the module to also consider the algebraic negative of the defined Seismic loads. (This may be useful for quickly creating a load combinations that reverse the direction of application of applied seismic loads by creating a "sister" load combination that uses the negative version of the seismic component for each load combo that normally incorporates +E.)

Factored Combinations sub-tab

The Factored Combinations tab offers the factored- or ultimate-level combinations from the selected load combination set. These load combinations will be used for wood design when the LRFD Design Method is selected. They will also be used for reinforced concrete design when the option is selected to design a footing.

A selected checkbox in the "Run" column indicates that the associated load combination will be considered. The "Run" button offers quick convenience options for changing the selection status of many load combinations at one time.

The Lambda column is only visible when the LRFD Design Method is selected. It indicates the value of the Time Effect Factor for each load combination. Click the Lambda button to quickly set the values of Lambda for all load combinations based on Table N3.

L <= 100, f1=0.5: Select this option to set appropriate load factors on Live Load when the specified Live Load is less than or equal to 100 psf.

L > 100, f1=1.0: Select this option to set appropriate load factors on Live Load when the specified Live Load is greater than 100 psf.

Use additional factors per ASCE 12.4.2.3: Select this button to instruct the module to consider the additional contribution to the Dead Load due to the vertical component of seismic load prescribed in ASCE 12.4.2.3.

Auto Reverse Wind: Select this button to instruct the module to also consider the algebraic negative of the defined Wind loads. (This may be useful for quickly creating a load combinations that reverse the direction of application of applied wind loads by creating a "sister" load combination that uses the negative version of the wind component for each load combo that normally incorporates +W.)

Auto Reverse Seismic: Select this button to instruct the module to also consider the algebraic negative of the defined Seismic loads. (This may be useful for quickly creating a load combinations that reverse the direction of application of applied seismic loads by creating a "sister" load combination that uses the negative version of the seismic component for each load combo that normally incorporates +E.)

The lower portion of the screen offers the following options for reviewing results:

Shear Panel Design

Summary

On a panel-by-panel, level-by-level basis, the Summary tab presents the following information:

Max Shear: Reports the maximum shear force and the load combination associated with that maximum shear force.

# of Sides: Indicates the number of sides to which sheathing has been applied. Will either be 1 or 2.

Shear Summary & Attachment: Reports the actual unit shear, the allowable unit shear, the design status, and the required attachment pattern.

Height/Width Ratio: Reports the actual height-to-width ratio, the allowable height-to-width ratio for Side 1 (Main Sheathing), the allowable height-to-width ratio for Side 2 (2nd Side Sheathing), and notes regarding the status of the height-to-width ratio and any necessary adjustments.

Detail by Panel ID

On a panel-by-panel, load combination-by-load combination basis, the Detail by Panel ID tab presents the following information:

Panel Data: Reports the story in which the panel exists, the distance from the left edge of the overall wall to the left edge of the panel, the width of the panel, the distance from the bottom edge of the overall wall to the bottom edge of the panel, the height of the panel, and the height-to-width ratio of the panel.

Shear Forces: Reports the "Tributary Width" = (Panel Width/∑Panel Widths)*Overall Wall Length, Tributary % = Panel Width/∑Panel Widths, Shear tributary to each panel, panel width, and the maximum unit shear.

Capacity Factors: Reports phi for LRFD designs or the reciprocal of the Factor of Safety for ASD designs, a capacity adjustment factor due to aspect ratio of the sheathing on the Main Sheathing side of the wall, a capacity adjustment factor due to aspect ratio of the sheathing on the 2nd Side of the wall, and a capacity adjustment factor due to the specific gravity of the framing.

Adjusted Allowable Shear: Reports the allowable unit shears for all relevant fastener spacings incorporating any applicable capacity adjustment factors. In situations where two sides of sheathing are being considered, this table will correctly determine the capacity considering both sides of sheathing by applying the rules of 4.3.3.3 and 4.3.3.3.2.

Panel Moment: Reports the moment in the panel.

Chord ID: Reports the identifying labels for the chords at the left and at the right end of each panel.

Detail by Load Combination

On a load combination-by-load combination, panel-by-panel basis, the Detail by Load Combination tab presents the following information:

"Tributary Width" = (Panel Width/∑Panel Widths)*Overall Wall Length

Tributary % = Panel Width/∑Panel Widths

Shear Force = shear tributary to each panel

A convenient nomenclature key is presented on this tab as well.

Chord Design

Chord Data

For each chord in the wall, the Chord Data tab presents the following information:

Location: Reports the Level in which the chord occurs and the distance from the left edge of the overall wall to the chord.

Chord Design: Reports the chord force associated with the controlling design ratio, the load combination responsible for producing the controlling design ratio, then number of chord members required to resist the applied load, the size of the chord member indicated by the user, the governing design ratio, the governing design consideration (tension or compression), and the design status.

Chord Compression Stress: Reports the maximum chord compression force, the load combination responsible for producing the maximum chord compression force, the maximum compressive stress, and the allowable compressive stress.

Chord Tension Stress: Reports the maximum chord tension force, the load combination responsible for producing the maximum chord tension force, the maximum tensile stress, and the allowable tensile stress.

Chord Forces by Chord ID

For each chord in the wall, on a load combination-by-load combination basis the Chord Forces by Chord ID tab presents the following information:

Story: The story in which the chord occurs.

ID: The identification label assigned to the chord.

Axial: Reports the load in the chord due to the sum of applied vertical loads, the load in the chord that can be added or subtracted due to the overturning moment in the panel, the force with the largest (tendency for) tension, and the force with the largest (tendency for) compression.

Chord Location: Reports the distance from the left edge of the overall shear wall to the chord location, the distance from the bottom edge of the overall shear wall to the bottom of the chord, and the distance from the bottom edge of the overall shear wall to the top of the chord.

Footing & Stability

Max Factored Pressures: Reports the maximum factored soil pressures at the left and right sides of the footing, along with the load combination responsible for producing each of those pressures. Note: Factored soil pressures are reported for reference, as they are used in the reinforced concrete design calculation for the footing.

Footing One-Way Shear Check: Reports the results of the one-way shear calculation for both ends of the footing.

Footing Bending Design: Reports the results of the flexural analysis and design including recommended reinforcing options to satisfy the required area of steel.

Max Unfactored Soil Pressures: Reports the maximum unfactored soil pressures at the left and right sides of the footing, along with the load combination responsible for producing each of those pressures. Unfactored soil pressures are compared to the allowable soil bearing pressure provided by the user.

Overturning Stability: Reports the overturning analysis performed about each end of the footing including the overturning moment, the resisting moment, the stability ratio, and the governing load combination.

Sketch

The sketch tab provides a convenient graphical view of the shear wall with options to selectively view or hide the following:

- Openings
- Shear Panel callouts
- Chord graphics and callouts
- Panel and opening dimensions
- All loads

10.6 Retaining Walls

Please select a subtopic.

10.6.1 Cantilevered Retaining Wall

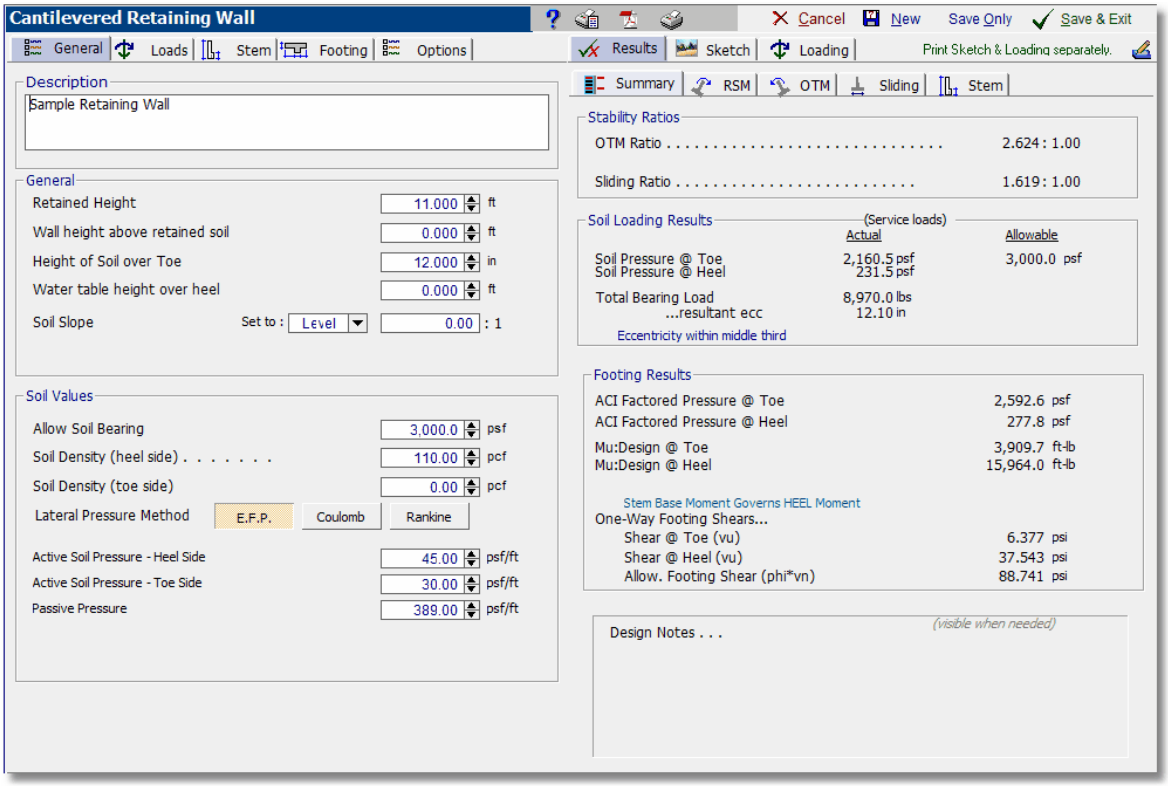
[Need more? Ask Us a Question]

Here is a listing of design capabilities contained in this Cantilevered Retaining Wall module:

- Cantilevered stem wall can have up to five different stem sections, of either masonry or concrete, each with a different thickness and/or reinforcing size and spacing. You may also include a weightless fence on top of the wall for the purpose of collecting additional wind load.
- Surcharges on either side of the wall.
- Sloped backfill.
- Axial dead and live load applied to the top of the wall, with eccentricity.
- Wind acting on a wall projection above grade.
- Add lateral loads against the stem -- uniform or concentrated (impact) loads.
- Effect of an adjacent footing behind the wall, line or point loading.
- Option to use user-defined active and passive pressure or input angle of internal friction and module will compute pressures using the Rankine or Coulomb formulas.
- Specify percent passive and frictional resistance to be used to prevent sliding.
- Option to specify sliding resistance using cohesion, in lieu of friction.

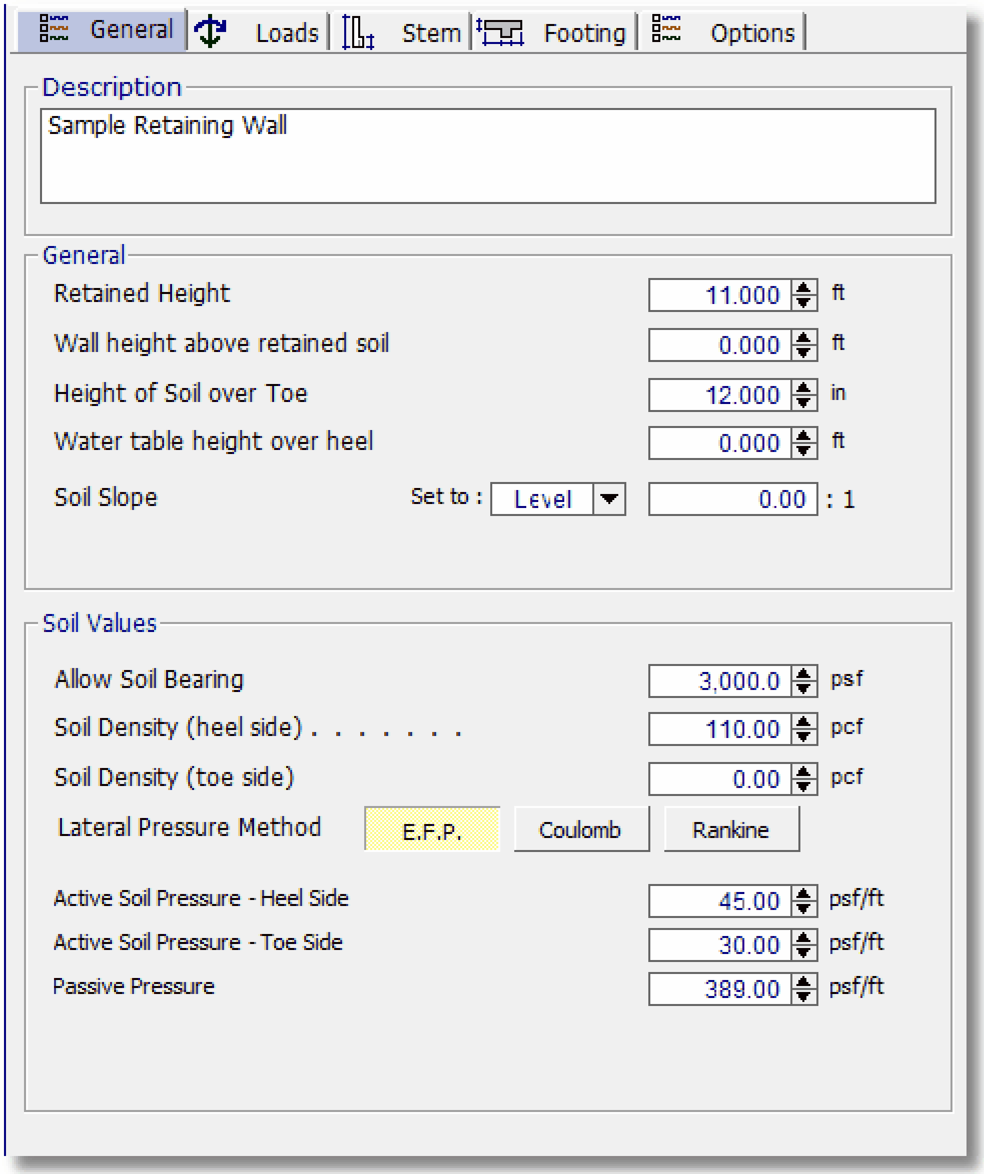
Window Layout

The retaining wall module divides the screen into a left and right portion. The left portion contains all of the input data (and in some cases intermediate calculated values). The right portion contains the calculated results and sketches.



GENERAL WALL INFORMATION

This tab allows you to enter the general information affecting the retaining wall. More specific data will be entered on other tabs dedicated specifically to the stem, footing and loads.



Retained Height

This is the height of retained earth measured from top of footing to the top of soil behind the stem (over the heel). When the backfill is sloped, the soil will slope away and upwards from this height. The actual retained height used for overturning and soil pressure calculations will be the retained height projected at the vertical plane of the back of the heel, but for stem moments, no such increase will be made.

Using the spin-buttons you can vary the retained height in 3-inch increments. You can also type in any number. After each entry, you can press [**Tab**] to advance to the next entry, or use your mouse to reposition the cursor.

Wall Height above Retained Soil

Use this entry to specify if the wall extends above the retained height. This entry is typically used to define a "screen wall" projection above the retained soil. This extension can be used as a weightless "Fence", or it can be defined as a concrete or masonry stem section without any soil retained behind it. You can enter wind load on this projection using the entry labeled "Load @ stem above soil" on the Loads tab. We'll handle the fence when we get to the Stem design screen. The total height of the wall (above the footing) will be equal to the Retained Height plus the Wall Height Above Retained Soil.

Height of Soil over Toe

Measured from top of footing to top of soil on the toe side, this may vary from a few inches to a few feet depending upon site conditions. (Note that it is specified in inches.) It is used to calculate passive soil resistance (but its effective depth can be modified by the "Soil over toe to Neglect" entry in the Sliding Resistance category on the Footing tab). This depth of soil is also used to calculate the resisting moment, and to reduce the net lateral sliding force. You can negate the latter effects on the Options screen if desired.

Water table height over heel

If a portion of the retained height is below a water table, the active pressure of the saturated soil will increase below that level. This additional pressure for the saturated soil is equal to the pressure of water, plus the submerged weight of the soil (its saturated weight - 62.4), plus the surcharge of the soil above the water table. The submerged weight of a soil can be approximated as 62% of its dry unit weight.

If you want to design for a water table condition, enter the maximum height from top of footing to water table level. The module will then compute the added pressures for saturated soil on the heel side of the footing, including buoyancy effect. It will also calculate increased moments and shears on the stem, and an increased overturning moment. Don’t enter a height more than the retained height, nor a liquid other than water. If the water table is near the top of the retained height, it may be advisable to enter the saturated soil density and specify the resulting active pressure for the full retained height.

Soil Slope

You may enter any backfill slope behind the wall. Use the drop-down list box or type the slope ratio as Horiz/Vert. The soil must be level or slope upward. Negative backfill slopes (grade sloping downward, away from the wall) are not allowed.

The module will use this slope to:

1. include the weight of a triangular wedge of soil over the heel as vertical load, and
2. compute overturning based upon an assumed vertical plane at the back face of the footing extending from the bottom of the footing to the ground surface – a steeper slope will result in a higher overturning moment.

When the Rankine or Coulomb method is used, the final calculated pressures do include the effect of the slope on those Rankine or Coulomb equations.

The module will not accept a backfill slope steeper than the angle of internal friction.

Allow Soil Bearing

The maximum allowable soil bearing pressure for static conditions. Using the spin buttons you can increment in 50 psf steps. Typical values vary from 1,000 psf to 4,000 psf or more.

Soil Density (heel side)

Enter the soil density for all earth (or water if applicable) above the heel of the footing. This weight is used to calculate overturning resistance forces and soil pressures using the weight of the soil block over the projecting heel of the footing. When surcharges are applied over the soil, the surcharges are transformed to equivalent uniform lateral loads acting on the wall by the ratio force = (Surcharge/ Density) *Lateral Load. Input this value in lbs. per cubic foot. Usual values are 110 pcf to 120 pcf. More if saturated soil. Water is usually assumed to be 64 pcf.

Soil Density (toe side)

Enter the soil density on the toe side, which may be different than the heel side. When surcharges are applied over the soil on the toe side, the surcharge is transformed to equivalent uniform lateral loads acting on the wall by the ratio force = (Surcharge/ Density)*Lateral Load. Input this value in lbs. per cubic foot. Typical values are 110 pcf to 120 pcf.

Lateral Pressure Method

Here you can choose between E.F.P. (Equivalent Fluid Pressure), Rankine formula or Coulomb formula. Based on your choice for Lateral Pressure Method, you will be offered the following input fields to fully define the lateral forces acting on the wall and footing.

When the EFP Method is selected:

Active Soil Pressure - Heel Side

Enter the equivalent fluid pressure (EFP) for the soil being retained that acts to overturn and slide the wall toward the toe side. This pressure acts on the stem for stem section calculations, and on the total footing+wall+slope height for overturning, sliding, and soil pressure calculations.

Commonly used values, assuming an angle of internal friction of 34°, are 30 pcf for a level backfill; 35 pcf for a 4:1 slope; 38 pcf for a 3:1 slope; 43 pcf for a 2:1 slope; and 55 pcf for a 1.5:1 slope. These values are usually provided by the geotechnical engineer.

When the retained soil is sloped, a vertical component of the lateral earth pressure over the heel can be applied vertically downward in the plane of the back of the footing. You can choose to apply this force for overturning resistance, sliding resistance, and/or for soil pressure calculations, by checking the boxes on the Options tab.

Active Soil Pressure - Toe Side

Enter the active pressure to be used on the toe side of the wall. This active pressure is used along with the "Soil Height over Toe" value (entered on the Sliding tab) to calculate a stabilizing soil force on the wall. This front side of the wall is assumed to be level. The active pressure from soil over the toe counteracts the heel-side active pressure to reduce net overturning and net sliding force.

This action is arguable, therefore the default is set to not use this counteracting force.

Passive Pressure
This is the resistance of the soil in front of the wall and footing to being pushed against to resist sliding. Its value is in psf per foot of depth (pcf). This value is usually obtained from the geotechnical engineer. Its value usually varies from 100 pcf to about 350 pcf.

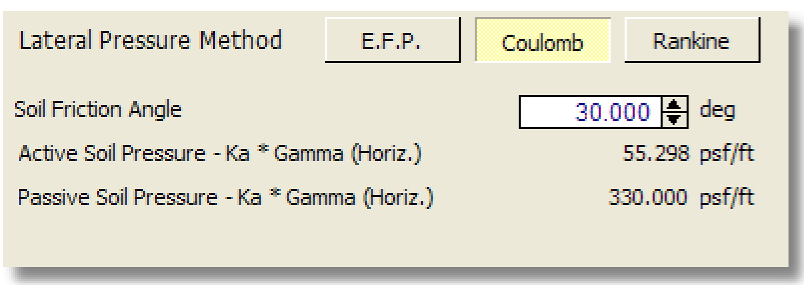
<u>When the Rankine or Coulomb Method is selected:</u>

Soil Friction Angle
This value is entered in degrees and is the angle of internal friction of the soil. This value is usually provided by a geotechnical engineer from soils tests, but can also be found in reference books or building codes for various typical soil classifications. This value is used along with Soil Density within the standard Rankine and Coulomb equations to determine "Ka" and "Kp" multipliers of density to give active and passive soil pressure values.

Active Soil Pressure
This value will be computed using the Rankine or Coulomb formulas. This represents the lateral earth pressure acting to slide and overturn the wall toward the toe side. The result will be presented in units of psf/ft. This pressure acts on the stem for stem section calculations, and on the total footing+wall+slope height for overturning, sliding, and soil pressure calculations.

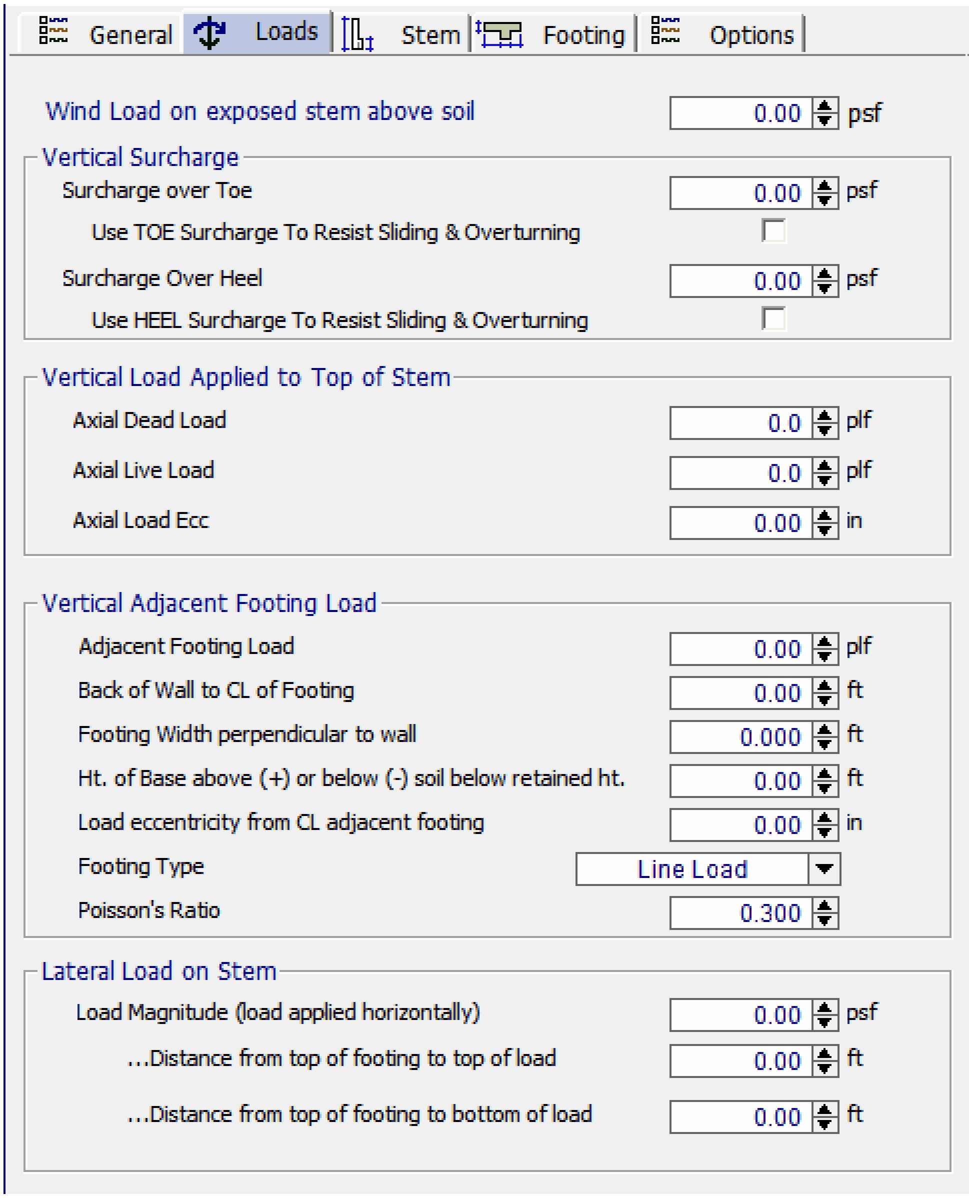
When the retained soil is sloped, a vertical component of the lateral earth pressure over the heel can be applied vertically downward in the plane of the back of the footing. You can choose to apply this force for overturning resistance, sliding resistance, and/or for soil pressure calculations, by checking the boxes on the Options tab.

Passive Soil Pressure
This value will also be computed using the Rankine or Coulomb formulas. This is the resistance of the soil in front of the wall to being pushed against to resist sliding. Its value is in psf per foot of depth (pcf). Common values usually vary from 100 pcf to about 350 pcf.



APPLIED LOADS

This tab allows you to enter all the loads that will be applied to your retaining wall in addition to lateral earth pressure.



Wind Load on exposed stem above soil

This wind force will be applied to that part of the stem projecting above the retained height defined by the entry "Wall height above retained soil." It is used to calculate overturning moment and sliding, stem design moment and shear, and soil pressures. Only positive values of wind load should be specified. This will ensure that the wind load acts in the direction of the active soil pressure, increasing the overturning moment, the sliding force, the soil bearing pressure, and shear and moment in the stem.

Vertical Surcharge (Surcharges will be factored as Earth Load, H for LRFD designs.)

Surcharge over Toe

This surcharge is treated as additional soil weight – if the surcharge is 240 psf and the density is 120 pcf, then the module uses two feet of additional soil. Similarly, if 50 psf is added for the weight of a slab over the footing, this will be equivalent to 0.41 feet of soil (50 / 120). This surcharge will affect sliding resistance and active toe pressure. Keep this in mind if modeling a point load toe surcharge.

Use TOE Surcharge to resist sliding & overturning

Checking this box will include the weight of soil overburden on the toe to resist overturning and add to its weight for frictional resistance.

Surcharge over Heel

This surcharge is considered uniformly applied to the top surface of the soil over the heel. It may be entered whether or not the ground surface is sloped, but it is unlikely a surcharge could apply to a sloped backfill. This surcharge is always taken as a vertical force. This surcharge is divided by the soil density and multiplied by the Active Pressure to create a uniform lateral load applied to the wall. You can choose to use this surcharge to resist sliding and overturning by clicking the box on the Options tab. Typical live load surcharges are 100 psf for light traffic and parking, and 250 psf for highway traffic.

Use HEEL Surcharge to resist sliding & overturning

Checking this box will include heel surcharge. If the surcharge includes live load, then using it to resist sliding and overturning could be non-conservative. In this situation, it might be advisable to deselect this checkbox.

Vertical Load Applied to Top of Stem

These loads are considered uniform load along the length of the wall. They are applied to the top of the topmost stem section and affect the design of masonry stems only. The dead and live loads are used to calculate stem design values and factored soil reaction pressures used for footing design. Only the dead load is used to resist overturning and sliding of the retaining wall.

If a wall is subjected to a high axial load (say more than 3 kips/ft) it could cause a reversal of the bending moment in the heel. Under these conditions, it might be advisable to investigate the design with and without the high axial load, to be sure that an acceptable design is found for all conditions.

Since slenderness ratios (h/t) for retaining walls are generally small, usually less than 10, and axial stresses are low, slenderness effects are checked but usually have a small effect.

If a point load is applied to the top of a wall, such as a beam reaction, the point load is typically assumed to distribute itself laterally at a rate that is based on engineering judgment for the materials under consideration. As a result of this distribution, the point load will result in a uniformly distributed load of some relatively low magnitude by the time it reaches the base of the stem. This module does not have an explicit input field for point loads, so they must be represented as uniformly distributed loads. To properly account for the lateral distribution that is characteristic of an axial point load applied to a wall, the magnitude that is entered to represent the point load should consider this distribution effect. The top of the wall may also need to be checked by appended calculations for the localized effects of the full magnitude of the concentrated load.

Axial Load Eccentricity

This is the eccentricity of the axial load with respect to the centerline of the uppermost stem section. Positive values of eccentricity move the load toward the toe, causing bending moments that are additive to those caused by the lateral soil pressure over the heel. Negative eccentricities are not accepted.

Vertical Adjacent Footing Load

This entry gives you the option of placing a footing (line or square) adjacent and parallel to the back face of the wall, and have its effect on the wall included in both the vertical and horizontal forces on the wall and footing. Refer to the General Reference Diagram for locations where input measurements should be taken.

For "Line (Strip) Load" the entry is the total load per ft. parallel to the wall (not psf). If the adjacent footing is specified as "Square Footing" (not line load), the load entered should be the adjacent footing load divided by its dimension parallel to the wall, giving a pounds per lineal foot value, as for a continuous (line) footing.

A Boussinesq analysis is used to calculate the vertical and lateral pressures acting on the stem and footing. The module uses equation (11-20a) in Bowles' Foundation Analysis and Design, 5th Edition, McGraw-Hill, pages 630. When the Boussinesq analysis is used, the module may require additional computing time, depending upon the speed of your computer (hundreds of internal calculations are done after each entry). To avoid this delay (which occurs any time any entry is changed) we suggest you use a vertical load of zero until your data entry is nearly finalized. Then enter the actual footing load and modify your final values.

For adjacent truck or highway loading, it may be preferable to use a heel surcharge (uniform) of 250 psf (or more), instead of treating it as an adjacent footing.

It is generally not necessary to use this feature if the adjacent footing load is farther from the stem than the retained height, less the depth of the adjacent footing below the retained height, since at this distance it will not have significant effect on the wall.

Wall to Footing Centerline Distance

This is the horizontal distance from the center of the adjacent footing to the back face of the stem (measured at the top of retaining wall footing). The nearest edge of the footing should be at least a foot away from the wall face – otherwise it is suggested to use an equivalent heel surcharge instead.

Note: If the horizontal distance from the center of the adjacent footing to the back face of the stem is greater than the vertical distance from the top of the retaining wall footing to the bottom of the adjacent footing, then the effect on the retaining wall will be insignificant.

Footing Width

Width of the adjacent footing measured perpendicular to the wall. This is necessary to create a one-foot long by Width wide area over which the load is applied.

Height of Base Above (+) or Below (-) Retained Height

Use this entry to locate the bottom of the adjacent footing with respect to the Retained Height. Entering a negative number places the footing below the Retained Height. A positive entry would typically only be used when the soil is sloped and the adjacent footing resides uphill. To insert a negative number, first type the number, then press the "-" (minus) sign.

Note: If the Adjacent Footing is another retaining wall at a higher elevation, the Boussinesq analysis may be used for the vertical load applied to the soil from the wall, however the design must also consider the lateral (sliding) loads from that adjacent wall. This load could be applied as Added Lateral Load, however this is at the discretion of the designer and is not within the scope of the module. Caution is urged for this condition. See discussion in the companion book: Basics of Retaining Wall Design. For questionable soil or site conditions a global stability analysis is advised.

Eccentricity

This entry is provided in case the soil pressure under the adjacent footing is not uniform. Enter the eccentricity of the resultant force under the adjacent footing from the centerline of the adjacent footing. Positive eccentricity shifts the load toward the toe, resulting in greater pressure at the side of the adjacent footing closest to the stem of the retaining wall. The module will use the vertical load and eccentricity and create a trapezoidal pressure distribution under the adjacent footing for use with the Boussinesq analysis of vertical and lateral pressures.

Footing Type

This drop-down list box allows you to enter either an isolated footing using the "Square Footing" selection, or a continuous footing using the "Line Load" selection.

Poisson's Ratio

Since the resulting pressures are sensitive to Poisson’s Ratio, there is an entry allowing you to select a ratio from 0.30 to 0.55. This value should be provided by the geotechnical engineer. A value of 0.50 is often assumed.

Lateral Load on Stem

This input allows you to specify an additional uniformly distributed lateral load applied to the stem.

This is for an impact point load, such as due to an impact of a car or similar force. Enter the load as a one-foot high increment, separating the "Height to Bottom" and "Height to Top" by one-half foot (or meter).

Note: **This load is not factored.** To apply a load factor (such as for an impact load), increase the applied load proportionately (e.g. an impact load of 1000 lbs requiring a load factor of 2.0 would be entered as 2,000 lbs). You may need to do several designs to check load factor combinations.

Keep in mind that when considering a concentrated lateral load, it may be possible to reduce the magnitude to account for the fact that the load distributes horizontally at levels below the point of application.

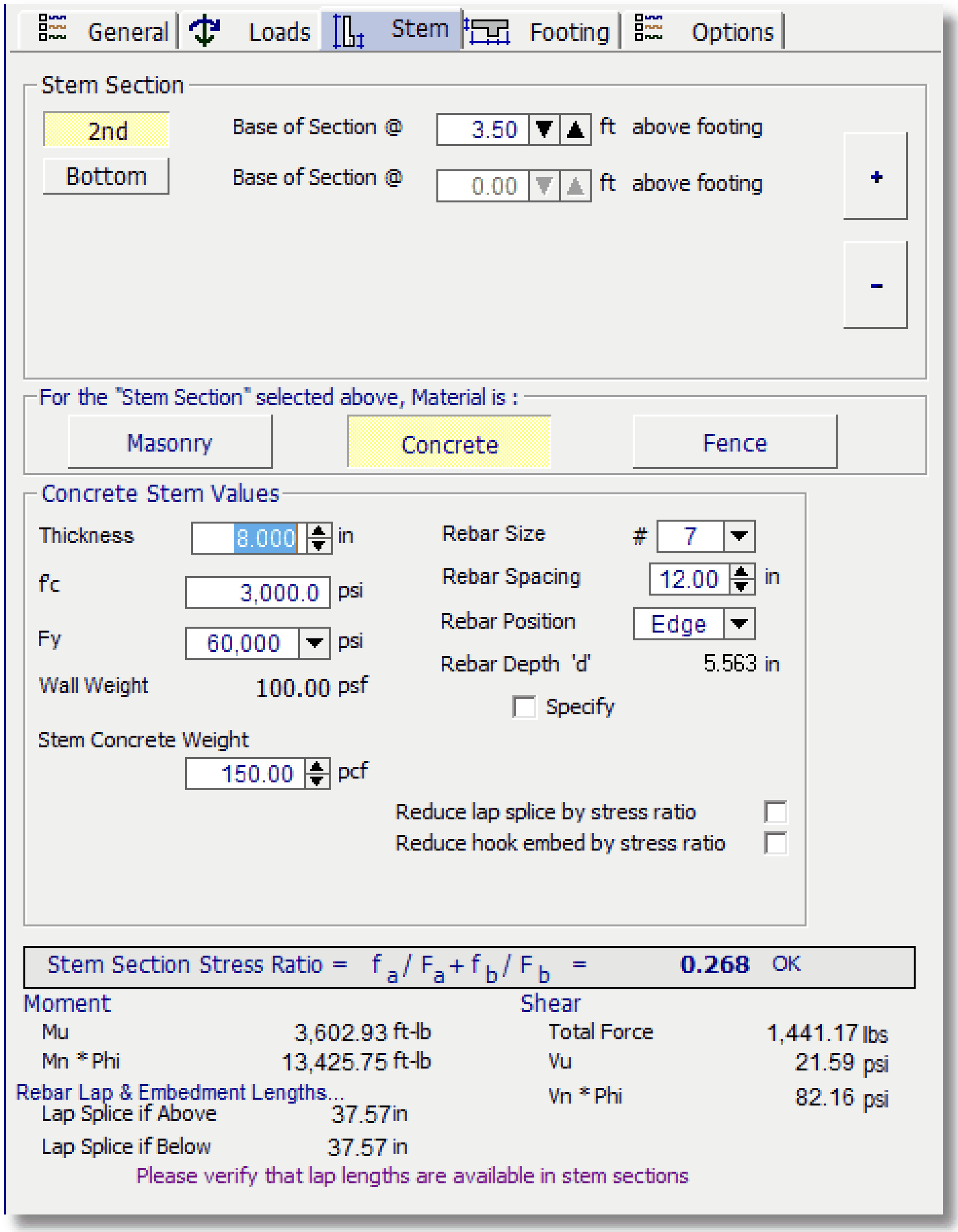
Height to Top

This dimension defines the upper extent of the added lateral load measured from the top of the footing. Do not enter a dimension that exceeds "retained height" plus "Wall height above retained soil".

Height to Bottom

This dimension defines the lower extent (or bottom) of the added lateral load measured from the top of the footing.

STEM DESIGN TAB



Material

Use the buttons to select Masonry, Concrete, or Fence. Fence is only allowed on top of the wall, higher than the Retained Height, and is considered weightless.

Thickness

Use the drop-down list box to input the wall thickness. If masonry is chosen, you will be given standard masonry thickness (e.g. 6", 8", 12"). If concrete is chosen, you can increment in one-inch steps. If Fence is chosen, this entry is unavailable since the fence is assumed to be weightless.

Wall Weight

This displayed value is based upon the wall data entered earlier. For concrete stems, the unit weight of concrete can be specified on the Stem tab. For masonry stems, 140 pcf grout is assumed, and the unit weight of the completed stem is a function of the specified thickness, CMU Type, and the status of the Solid Grouting checkbox, all of which are located on the Stem tab. A multiplier is also available on the Options tab to modify the tabular weights for masonry walls.

The industry standard masonry unit weight values used by this module may be modified by clicking **Databases > Concrete Masonry Unit Data** from the main menu and then clicking the [**Change**] button.

Design Method

When a masonry stem section is chosen, this allows a choice of ASD or LRFD methods. When the latter is selected the input notations change (e.g. fs to fy) and all calculations are based upon LRFD.

Rebar Size

Make your selection from the pull-down menu for bar sizes #3 to #10. “Soft Metric” sizes will be displayed in parentheses alongside.

Rebar Position

Chose between Center or Edge. If Center is chosen, the rebar d distance will be 1/2 the actual wall thickness. If Edge is chosen it will be located at the heel side of the stem.

For masonry design, the module contains a table of the appropriate "d" values to use for various block sizes and center/edge locations, as shown in the table below.

Rebar Position Depth for Masonry, Default Values.

Nominal Thickness	**Rebar Depth (in)**	
	Center	**Edge**
6"	2.75"	2.75"
8"	3.75"	5.25"
10"	4.75"	7.25"
12"	5.75"	9.0"
14"	6.75"	11.0"
16"	7.75"	13.0"

For concrete, the edge rebar depth is always stem thickness less 1.5" for #5 and smaller bars (or 2" for #6 or larger), less one-half the bar diameter.

Specify Position Box

Click this box to change the default "d" value.

Fs

Enter the allowable steel stress, based on working stress design, which should be used for design of the masonry stem section. The spin button changes this value in increments, and is not visible when a concrete wall has been specified.

Short Term Increase

This factor is applied to masonry ASD design and allowable soil bearing as permitted by IBC 2009, section 1806.1, and ACI 530-08, section 2.1.2.3. This is applicable only when wind and/or seismic is applied.

Solid Grout

This applies to masonry only. If this box is checked, the weight of the wall will be based upon industry standard solid-grout weight for either lightweight, medium weight, or normal weight block regardless of the specified spacing of reinforcing. If this box is not checked, the module will calculate the weight based on the assumption that only cells containing reinforcing are grouted.

This also affects equivalent solid thickness for stem shear calculations, and area for axial stress calculations (combined with moment for masonry stems).

Modular "n", Ratio

This is the multiplier used to calculate the modulus of elasticity of masonry. The ACI 530-05 and ACI 530-08 both specify Em = 900*f'm which is the default value. The multiplier can be modified on the Options tab.

Equivalent Solid Thickness

If partially grouted (not solid grout), this value is generated from an internal database that is accessible by clicking **Database > Concrete Masonry Unit Data**.

Stem Design Heights

IMPORTANT! The term “Stem Design Height” used in this module is the height <u>above the top of the footing</u> (i.e. above the base of the stem). It is the height above the bottom of the stem where you want the module to compute moments and shears.

You can divide the stem into up to five sections (increments of height). Each section represents either a different material (concrete, masonry, or fence), a change in thickness, or a change in reinforcing size or spacing.

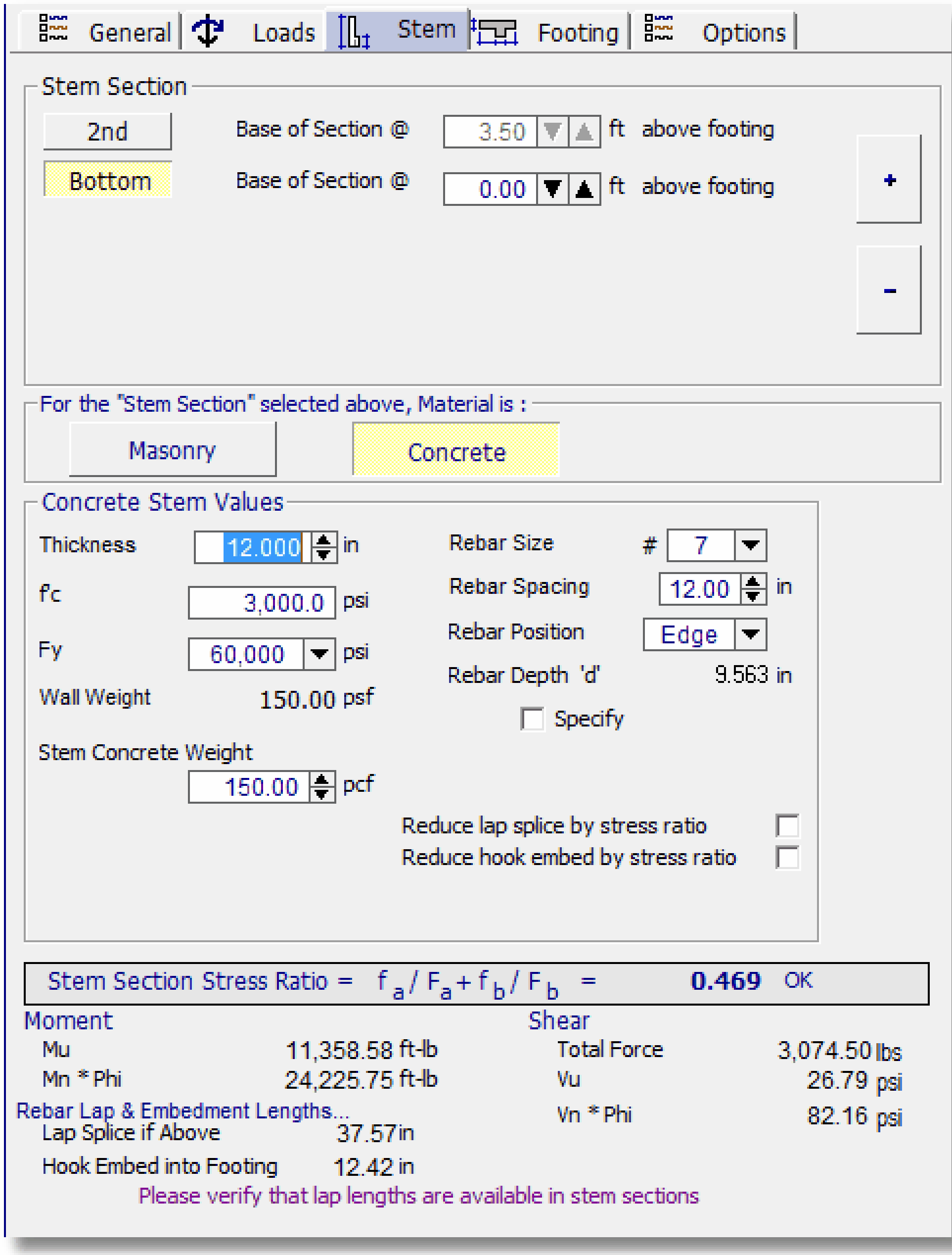
For most walls, only two or three changes in stem sections are used. For example, it would be logical to create a change of section at the top of the dowels projecting into the stem from the footing and perhaps another change in section further up the wall where a more economical design is desired.

Bottom

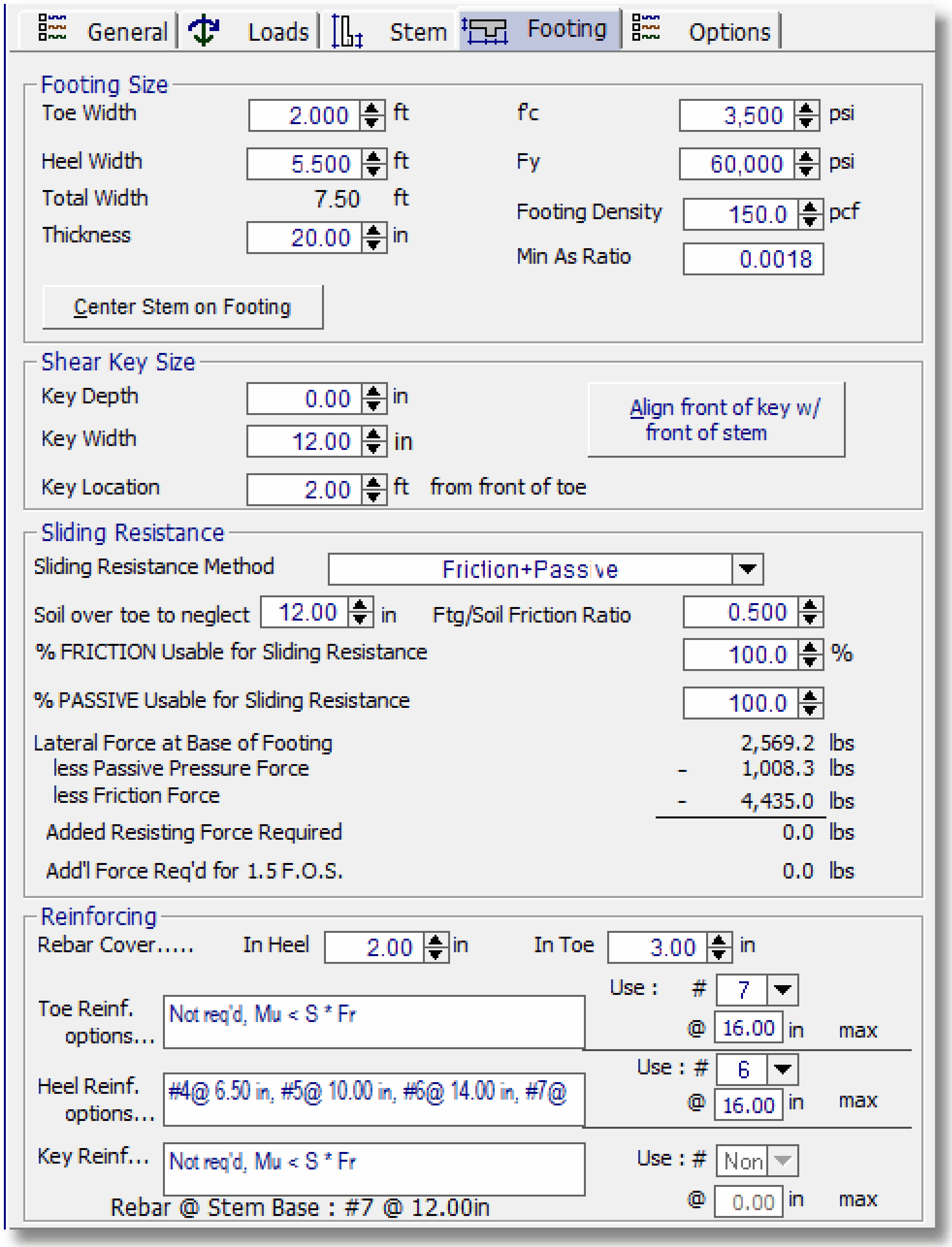
You must start the stem design here, at the base (height above footing = 0.00), where the stem moment and shear are maximum. As you manipulate the bar sizes, spacing, and position (you first, of course, will have selected a wall material and trial thickness) until the Summary box shows you an acceptable stress ratio (the higher and closer to 1.0, the more efficient).

To check the wall at a higher Design Height, such as at least the LAP REQ'D IF ABOVE distance, where reinforcing or thickness can be reduced, click the [**Insert Stem**] button and enter the next higher section. Advance the spin button to the desired height above the top of the footing or enter it by typing. This will move (and dim) the Bottom Section and you can now design this new section.

Continue this way, clicking [**Insert Stem**] button after each stem section design is completed, up to a maximum of five heights. A new Design Height should only be entered when you want to change the material, thickness, or reinforcing, and should never be less than about two foot intervals.


General
Loads
Stem
Footing
Options
Stem Section
2nd
Base of Section @
3.50
ft above footing
Bottom
Base of Section @
0.00
ft above footing
+
-
For the "Stem Section" selected above, Material is :
Masonry
Concrete
Concrete Stem Values
Thickness
12.000
in
f'c
3,000.0
psi
Fy
60,000
psi
Wall Weight
150.00 psf
Stem Concrete Weight
150.00
pcf
Rebar Size
7
Rebar Spacing
12.00
in
Rebar Position
Edge
Rebar Depth 'd'
9.563 in
Specify
Reduce lap splice by stress ratio
Reduce hook embed by stress ratio
Stem Section Stress Ratio = f a / F a + f b / F b = 0.469 OK
Moment
Mu
11,358.58 ft-lb
Mn * Phi
24,225.75 ft-lb
Shear
Total Force
3,074.50 lbs
Vu
26.79 psi
Vn * Phi
82.16 psi
Rebar Lap & Embedment Lengths...
Lap Splice if Above
37.57 in
Hook Embed into Footing
12.42 in
Please verify that lap lengths are available in stem sections

FOOTING TAB



Toe Width

This is the width of the Toe of the footing, and is measured from the front edge of the footing to the front face of the stem. Can be set to 0.00 for a property line condition. All overturning and resisting moments are taken about the bottom-front edge of the toe.

Heel Width

Distance from front face of stem to back of heel projection. If a dimension is entered that is less than the stem width at the base, the module will automatically reset the heel dimension to at least the stem width. For a property line at the rear face of the stem, this dimension would be the stem width.

Total Width

The calculated width of the footing, Toe Width + Heel Width.

Thickness

Total footing thickness, NOT including the key depth (if used). For bending and shear design of the footing, the rebar depth "d" is taken as Footing Depth - Rebar Cover - ½" (to account for the rebar radius). If footing thickness is inadequate for shear capacity a red warning indicator will appear.

The footing thickness must be sufficient to allow for rebar development (for hooked dowels) plus rebar cover (adjacent to the soil). If you enter a dimension less than required for stem bar development, a red message will appear at the top of the screen. If the thickness is inadequate, increase the footing thickness, or change the stem dowels, until this message disappears.

f'c

Enter concrete compressive stress for footing.

Fy

Allowable rebar yield stress to be used for design of footing bending reinforcement.

Footing Density

This option is necessary since, if there is any buoyancy effect, it will reduce the effective weight of the footing concrete.

Min "As" Ratio

Enter the absolute minimum steel percentage to be used to calculate rebar spacing requirements (commonly 0.0018 Ag for Fy=60,000 psi, but code applicability for footings is arguable). If the % steel required by stress analysis is less that 200/Fy, the minimum of (200/Fy or 1.333 * bending percentage required) is calculated and compared with the Minimum As% entered here, and the greater of the two is used to calculate rebar spacing requirements.

Key Depth

Depth of the key below the bottom of footing. The bottom of the key is used as the lower horizontal plane for determining the size of the passive pressure block from the soil in front of the footing. Adjust this depth so the sliding safety factor is acceptable (typically a value of 1.5 is used).

Key Width

Width of the key, measured along the same direction as the footing width. This is usually 12"-14", but generally not less than one-half the key depth so flexural stress in the key is usually minimal.

Key Location

Enter the distance from the front edge of the toe to the beginning of the key. Do not enter a distance greater than the footing width minus key width.

Sliding Resistance Method

Enter whether sliding resistance will be by friction and passive pressure or by cohesion and passive pressure.

Soil over toe to neglect

Since the soil over the toe of the footing may be loose and uncompacted, it may have little or no passive resistance. This entry gives you the option of neglecting any or all of the Height of Soil Over Toe that you entered in the Criteria tab. You can neglect the soil over toe plus the footing thickness, if desired.

Ftg/Soil Friction Ratio

Enter the friction factor here. It is generally provided by the geotechnical engineer and usually varies from 0.25 to 0.45.

% FRICTION Usable for Sliding Resistance

This may be a stated restriction in the geotechnical report. Enter a value from zero to 100%.

% PASSIVE Usable for Sliding Resistance

This may be a stated restriction in the geotechnical report. Enter a value from zero to 100%

Lateral Forces at Base of Footing

This is the total lateral force against the stem and footing which causes the wall to slide and which must be resisted. It is the total active pressure on the heel side less the active pressure on the toe side.

less Passive Pressure Force

This uses the allowable passive pressure in pcf times the available depth (footing thickness plus soil above toe minus height to neglect) and multiplied by the percent usable that you indicated, to compute the total passive resistance. Weight due to toe surcharge, if applicable, will also be added. If a key is used, the available passive pressure depth will be to the bottom of the key.

less Friction Force

This is the total vertical reaction multiplied by the friction factor and multiplied by the percent usable that you indicated.

Added resisting force required

If this is 0.0 lbs., the forces balance, but there may be no safety factor. Watch the Sliding Factor of Safety for an adequate value (usually 1.5). Consider adding a key or revising the footing dimensions if required.

Additional force required for a 1.5 Factor of Safety

This is the additional resisting force that would be required in order to achieve a 1.5 safety factor. If this value indicates zero, then the sliding factor of safety is already greater than or equal to 1.5.

Rebar Cover in Heel & Toe

These input fields allow you to specify the clear cover that will be used at the heel and at the toe. When specifying these values, keep in mind that the toe rebar is placed closest to the bottom of the footing, and the heel rebar is placed closest to the top of the footing. When calculating the "d" dimension for bending and shear strength calculations, this module will consider the footing thickness and then deduct the specified clear cover and an additional 1/2" to account for the radius of the rebar.

Toe Reinforcing Options

This list gives you choices for reinforcing sizes and spacing for the bottom toe bars. Typically the toe bars are extensions of the stem dowels, which are bent out toward the toe. Therefore, it may be most efficient to simply verify that the bar size and spacing used for the stem dowels is within the range of the selections offered for toe reinforcing options.

NOTE: If “No reinf’ req’d” message appears, it means the flexural capacity of the footing (modulus of rupture times the section modulus, with 2” deducted from the thickness for crack allowance per code) is adequate to resist the applied moment. However, the designer in some cases may consider it prudent to add reinforcing regardless of the theoretical flexural capacity. For plain concrete per ACI 22.5.1, Fr = phi(5)(f’c)$^{1/2}$, where phi = 0.55.

Heel Reinforcing Options

This list gives you choices for acceptable sizes and spacing of top heel bars. It is desirable to select a spacing that is modular with the stem dowel bars for ease of construction. Note: The module does not calculate the heel bar development length inward from the back face of the stem (where the moment is maximum).

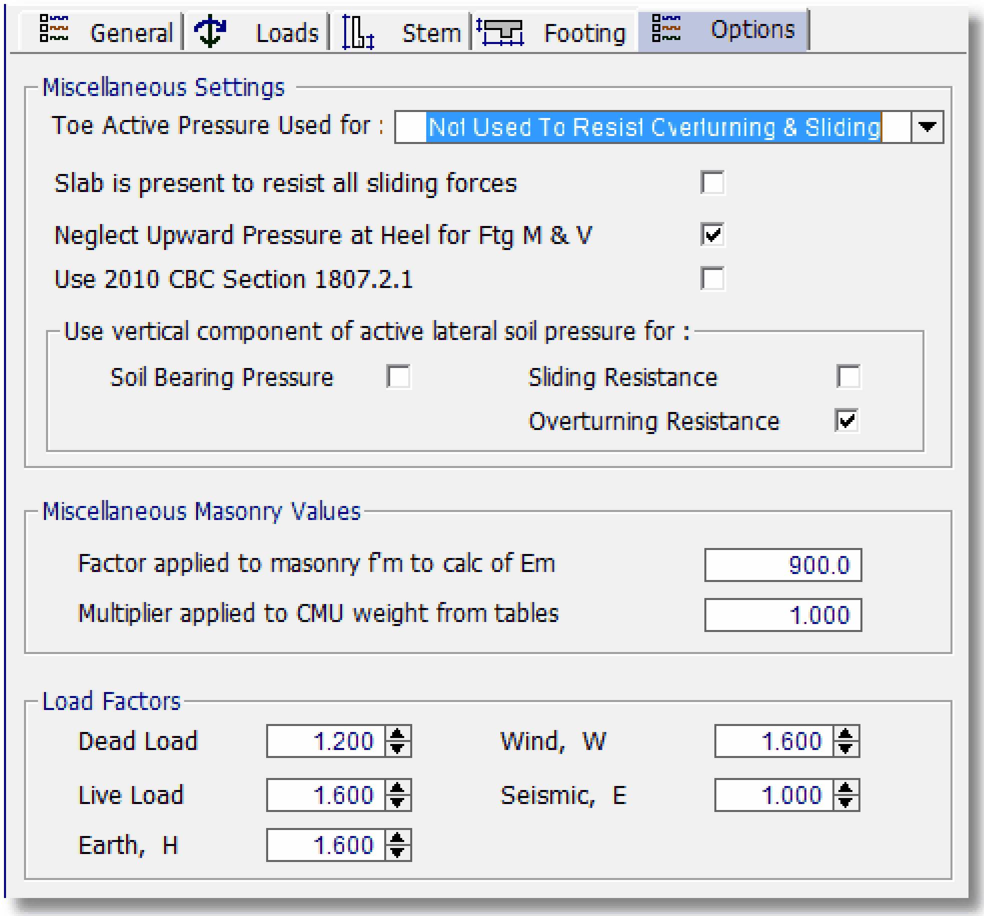
NOTE: If “No reinf’ req’d” message appears, it means the flexural capacity of the footing (modulus of rupture times the section modulus, with 2” deducted from the thickness for crack allowance per code) is adequate to resist the applied moment. However, the designer in some cases may consider it prudent to add reinforcing regardless of the theoretical flexural capacity.

ALSO NOTE: The heel design moment may be influenced by the setting used for the item labeled "Neglect Upward Pressure at Heel for Ftg M & V" on the Options tab. See the section on the Options tab for additional information.

Key Reinforcing Options

If flexural tension is insufficient to resist bending in key, a message will appear indicating reinforcing required. You can vary the width of the key until the message disappears. If reinforcing is required options will be shown on the Footing tab.

OPTIONS TAB



Toe Active Pressure Used

This drop-down list box provides the option to specify whether the module should or should not apply the toe side horizontal active pressure to reduce the overturning moment and sliding force to be resisted. Typically this is NEVER used. It was added to assist in cases where the footing was buried very deeply in soil.

Slab is Present to Resist all Sliding Forces

Check this box when a slab is in front of the wall to resist lateral sliding. When this box is checked, sliding is not a design issue – passive and friction resistance are ignored -- but the lateral sliding force is displayed for checking the resistance offered by the slab.

The slab is assumed to be at the top of the footing – not higher, so selecting this checkbox will not reduce the design shear or moment in the stem.

Neglect Upward Pressure at Heel for Ftg M & V

When the user DESELECTS the option to “Neglect Upward Pressure at Heel for Footing Moment and Shear”:

- The program is considering the upward pressure at heel.

- The upward pressure tends to reduce the moment caused by the weight of the soil and the self-weight of the heel itself.
- The program is reporting the actual net moment found to exist in the heel.

When the user SELECTS the option to “Neglect Upward Pressure at Heel for Footing Moment and Shear”:

- The program is NOT considering the upward pressure at heel.
- The program determines the heel moment caused by the weight of the soil and the self-weight of the heel itself.
- The program then applies the following logic:

- The calculated heel moment is conservative when the upward pressure at heel is neglected.

- The calculated heel moment would act in the same direction as the toe moment.

- The sum of the heel moment and the toe moment can’t be any greater than the stem design moment, which is delivering the moment to that joint in the first place.

- So the program conservatively assumes that the heel design moment is the calculated heel moment but not to exceed the stem design moment.

Use 2010 CBC Section 1807.2.1

Section 1807.2.1 of 2010 CBC and IBC 2009 requires the designer to consider in the sliding analysis the effects of the active pressure extending all the way to the bottom of a keyway when one is used. Selecting this option will ensure that the analysis properly considers the full extent of the active pressure on a keyway.

Choices for Use of Vertical Component of Active Pressure

The vertical component of the lateral pressure is applied at a vertical plane at the back of the heel. You can choose whether or not to use this force to resist overturning, to resist sliding, and to reduce soil bearing pressure.

If you choose the option to use this force to resist overturning, then for a level backfill, the module will back-solve the EFP method to find the equivalent internal friction angle, and then apply this vertical component equal to tanß. If either the Rankine or Coulomb method had been chosen, this vertical component would be tangent of $\phi/2$.

If you choose the option to use this force to resist sliding, then the sliding calculation will incorporate the additional frictional force that can be generated as a result of the additional vertical force.

If you choose the option named "Soil Bearing Pressure", then the the effect of this vertical component at the back of the heel will be considered in the calculation of the soil bearing pressures.

Factor Applied to Masonry f'm for Calculation of Em

The modulus of elasticity for masonry is 900 times f'm per ACI 530-05. This field allows you to enter a multiplier of other than 900 if necessary.

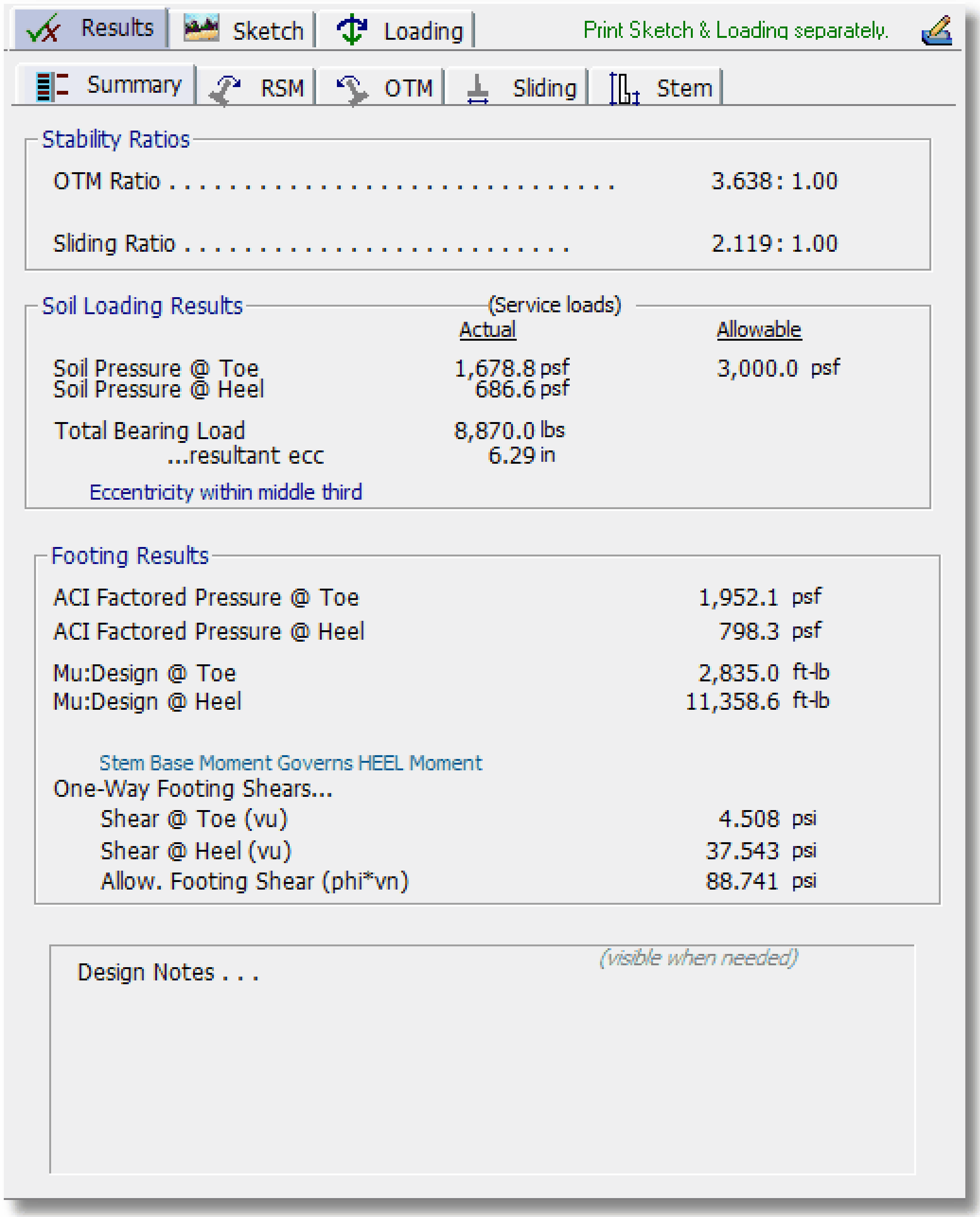
Multiplier Applied to CMU Weight from Tables

This entry allows you to increase or decrease the internal default values of CMU weights, as displayed on the Stem tab.

Load Factors

For each type of load (DL, LL, etc) the default factor will be displayed. You can change them and set new defaults, but remember to review them for a new design since they may have been changed.

RESULTS TAB



Stability Ratios

These are displayed for both overturning and sliding.

Soil Pressure @ Toe and Heel

This is the resulting unfactored soil pressure for both the toe and heel. If the eccentricity is outside the middle third, the heel pressure will show 0.00. (Note: when the resultant is outside the middle-third, the module calculates the toe pressure assuming no tension at heel).

Allowable Soil Pressure

This is for your reference as input on the Criteria tab.

Total Bearing Load

This is the sum of all vertical forces.

Resultant Eccentricity

Distance from center of footing to resultant soil pressure.

Eccentricity Within/Outside Middle Third

The resultant is outside the middle third of the footing width if the eccentricity is greater than one-sixth the footing width. (If outside the middle third, the module computes the toe soil pressure assuming no tension at heel.)

ACI Factored Soil Pressure @ Toe and Heel

Load factors are applied to all dead and live loads to determine total vertical load for soil pressure used in calculating footing moments and shears. This load is then applied at the same eccentricity calculated for service load soil pressures to yield the actual factored soil pressures for footing design using ultimate strength design principles. Note that since only factored vertical loads are applied at the non-factored resultant eccentricity, a true 1.6 load factor applied to lateral earth pressure is not used for footing design. If resultant vertical load eccentricity were to be calculated using factored loads, the distance would not truly represent a correct state of stress in the soil. ACI load factors are intended to give conservative results for stress. Calculation of a factored load eccentricity would give soil pressure diagrams that would not always represent the actual soil pressure distribution under the footing, and yield unreasonable results. Factored lateral earth pressure, however, is always used for concrete stem design.

Mu Design @ Toe/Heel

These are the factored (by 1.2) moments at face of stem for toe and heel moments. Since neither can be greater than the stem base moment (factored if concrete stem), the latter may govern. These moments will be reduced if you choose to neglect the upward soil pressure on the Options tab.

A message will indicate which controls.

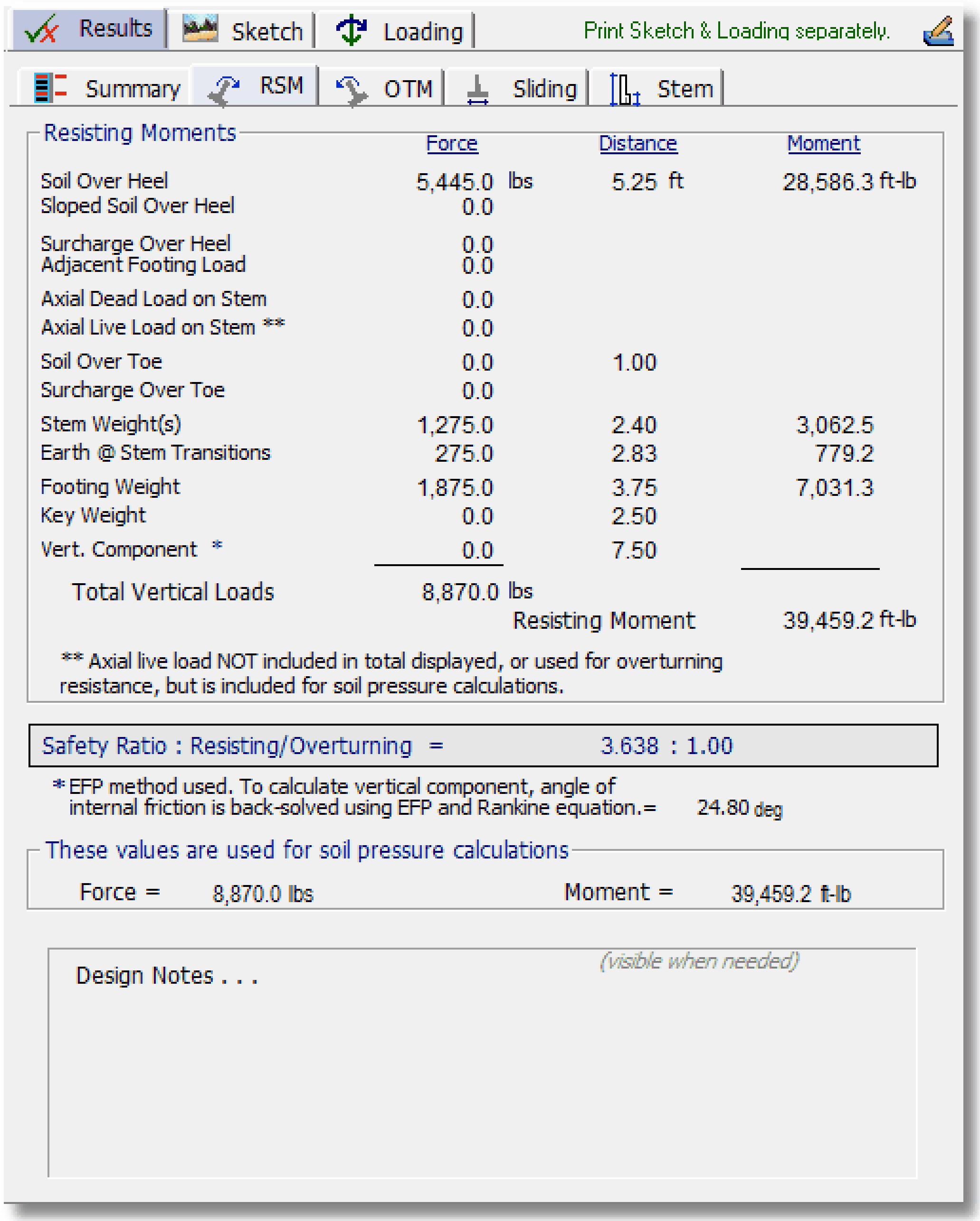
Shear @ Toe and Heel

The actual shear is calculated from the one-way action in the footing at a distance "d" (footing thickness - rebar cover) from the toe side of the bottom stem section, and at the face of the stem on the heel side. If "d" is greater than the projecting toe or heel length, then the one-way shear is zero.

Allowable Footing Shear

The allowable unit shear equals (0.75 * 2 * $f'c^{1/2}$).

RSM - RESISTING MOMENTS TAB

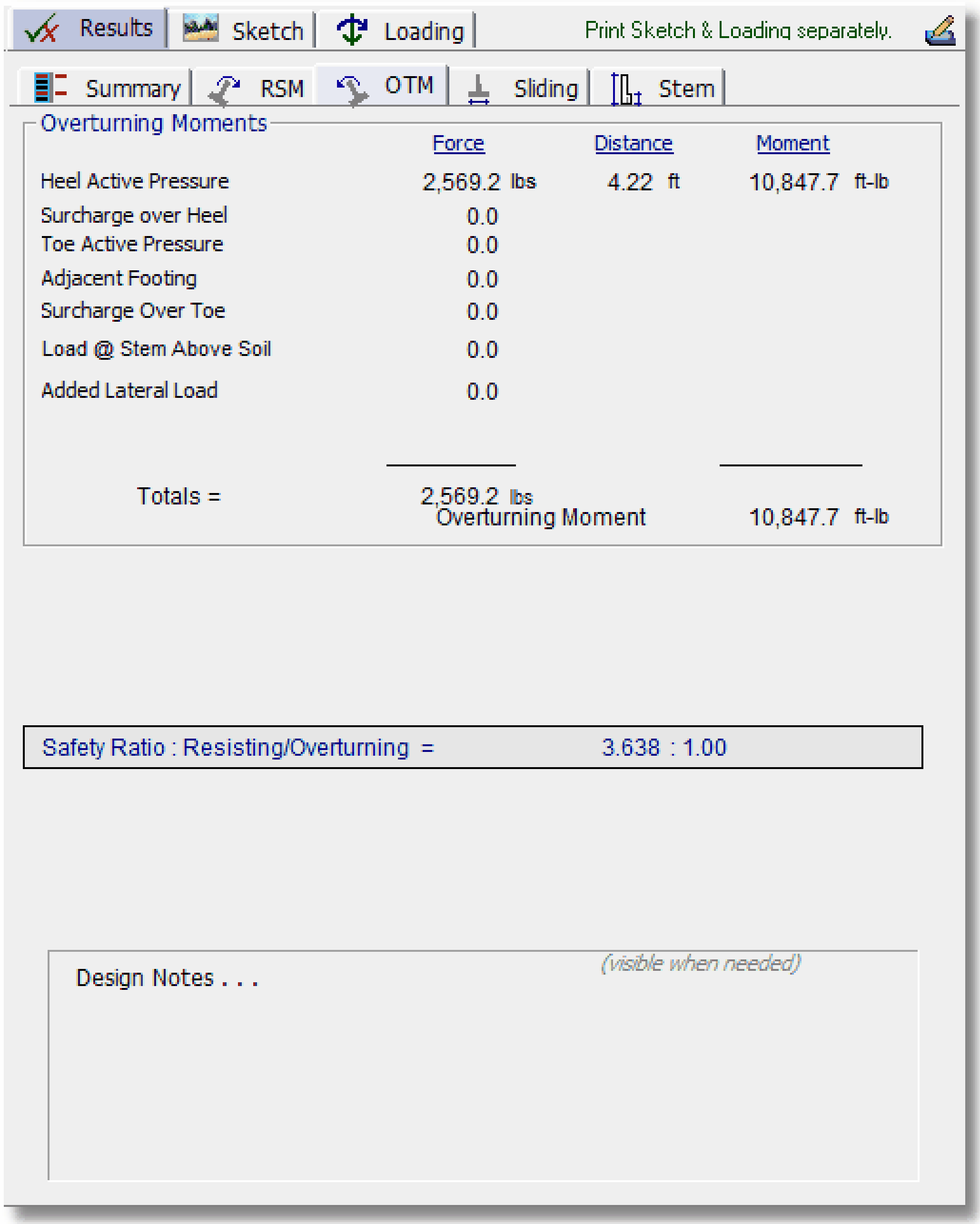


This screen presents in tabular form each component contributing to resisting moment, giving weights and lever arms from the front edge of the toe to the centroid of the weight.

For calculating the vertical component, if checked on the OPTIONS screen, and if the EFP method was chosen, the module will back-solve using the Rankine formula to obtain an equivalent internal friction angle.

The force and moment displayed at the bottom accounts for deduction of effect of vertical component, if box on Options tab has been checked.

OTM - OVERTURNING MOMENTS TAB



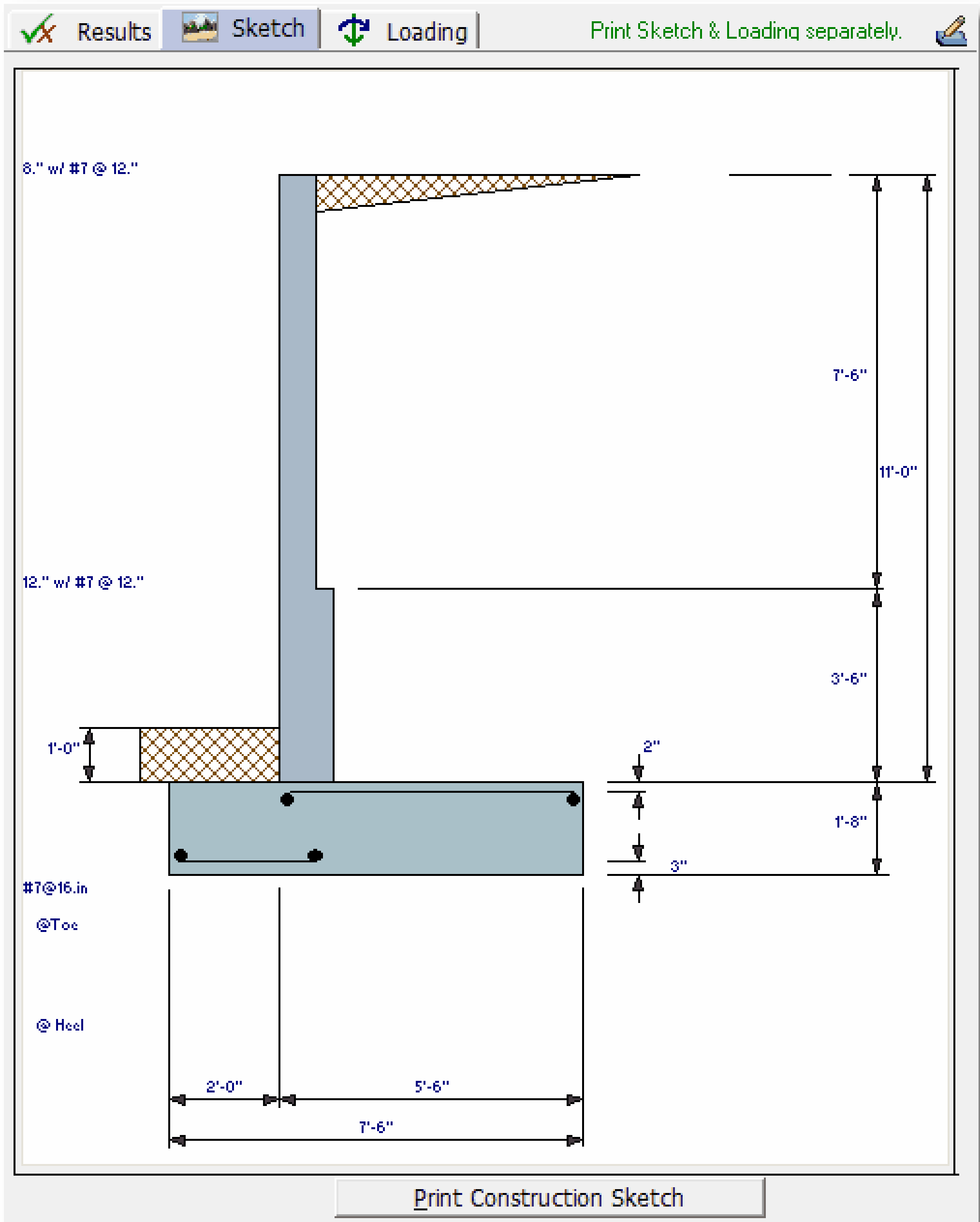
This screen presents in tabular form each component acting horizontally to overturn the wall/footing system. The centroid of each force is multiplied by its distance up from the bottom of the footing. The Heel Active Pressure includes the effect of surcharges and water table, if applicable, and its Distance is to the centroid of the total lateral force.

The total overturning moment is displayed, along with the Resisting/Overturning ratio. The overturning moment is reduced by the toe side active pressure, if this option is selected on the Options tab.

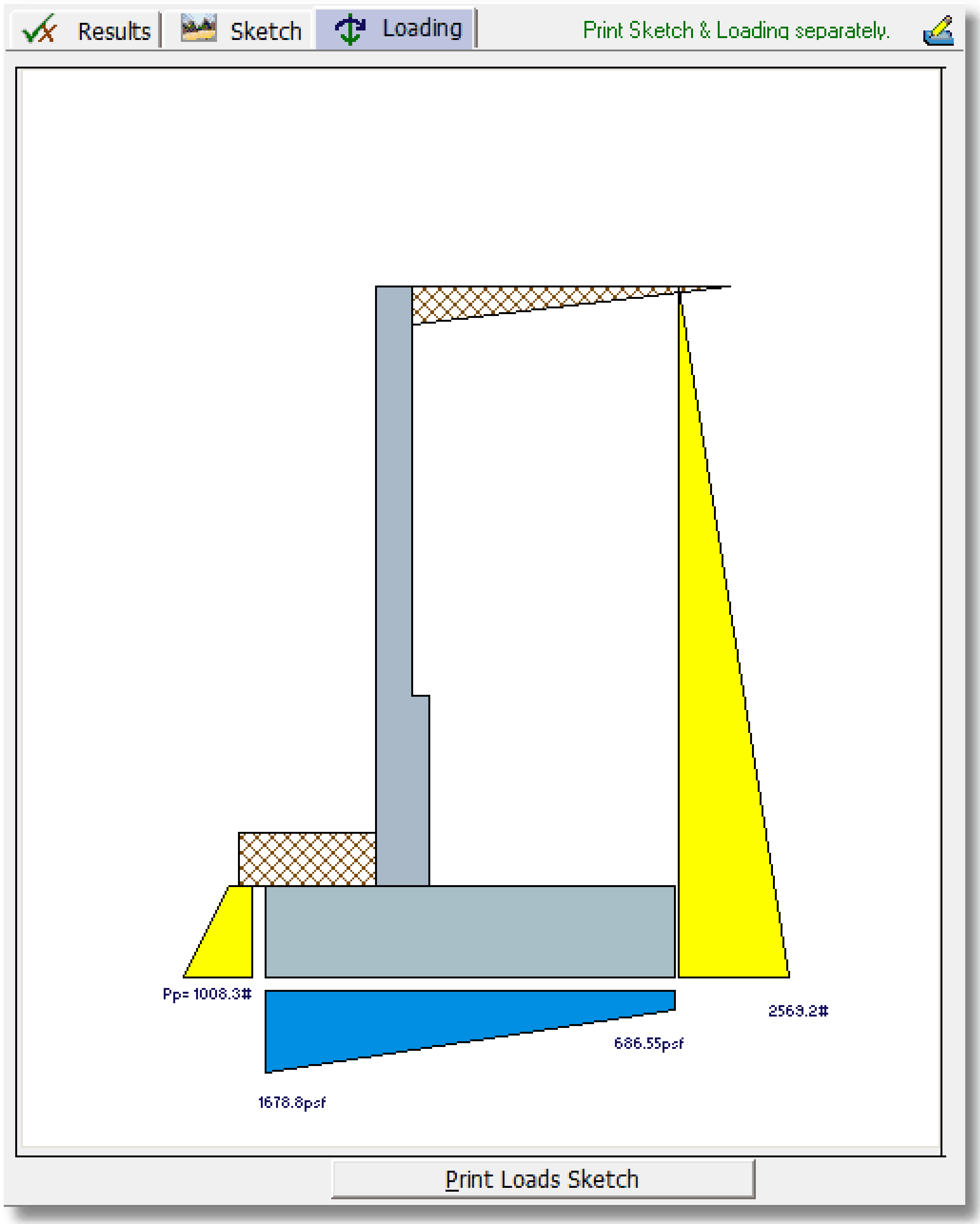
SLIDING TAB

STEM TAB

SKETCH TAB


Results
Sketch
Loading
Print Sketch & Loading separately.
8." w/ #7 @ 12."
12." w/ #7 @ 12."
7'-6"
11'-0"
3'-6"
1'-0"
2"
1'-8"
3"
#7@16.in
@Toe
@ Heel
2'-0"
5'-6"
7'-6"
Print Construction Sketch

LOADING TAB


Results
Sketch
Loading
Print Sketch & Loading separately.
Pp= 1008.3#
2569.2#
686.55psf
1678.8psf
Print Loads Sketch

10.7 Foundations

Please select a subtopic.

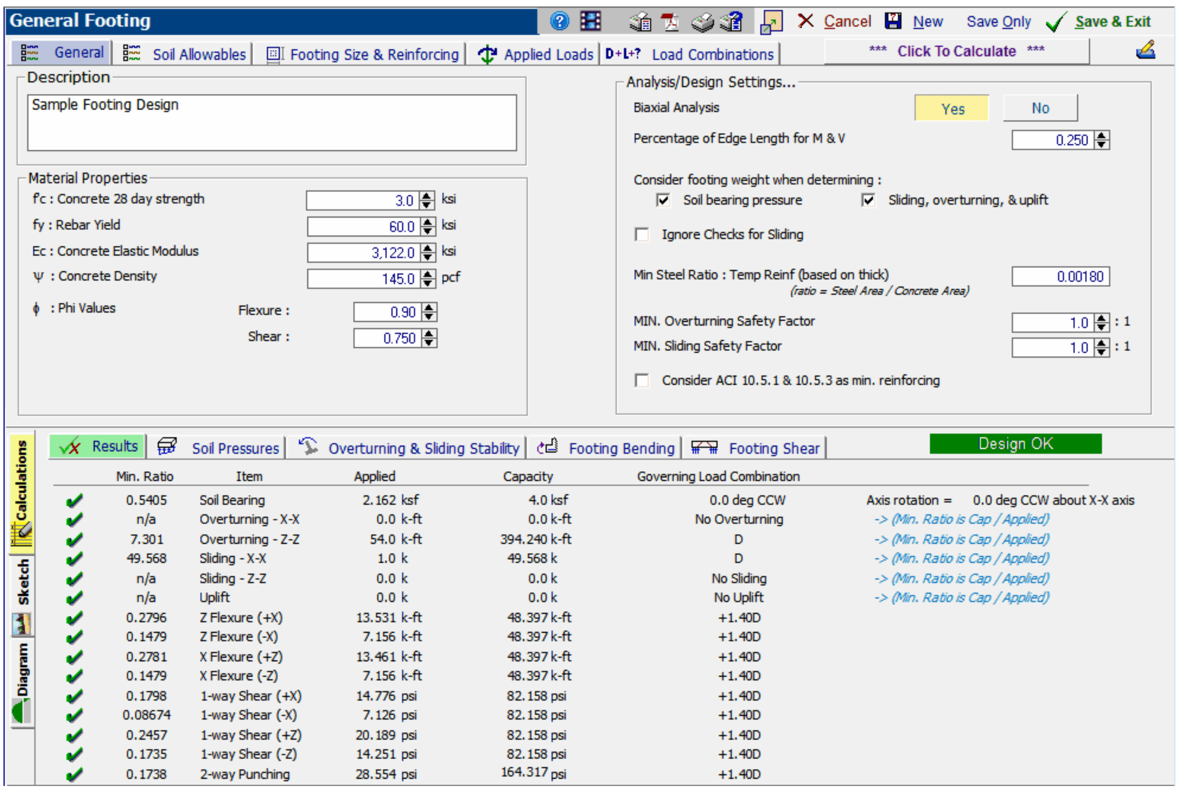
10.7.1 General Footing

[Need more? Ask Us a Question]

This module provides analysis of a rectangular footing with applied axial load, overburden, moment and shear loads. Click here for a video: General Footing

The module allows you to move the axial load application position off-center of the footing, and provides automatic calculation of allowable soil bearing pressure increases based on footing dimensions and/or depth below surface.

The module checks service load soil pressure, overturning stability, sliding stability, flexure at each of the four pedestal faces, 1-way shear at 'd' from each of the four pedestal faces, and punching shear along a perimeter located 'd/2' from the pedestal faces.



General

f'c

28-day compressive strength of the concrete.

fy

Yield point stress of reinforcing.

Ec

Modulus of elasticity of concrete.

Concrete Density

The density of the concrete is used to calculate the self-weight of the pedestal and footing when that option is selected.

Phi Values

Enter the capacity reduction values to be applied to Vn and Mn.

Biaxial Analysis

Select Yes or No to indicate if a biaxial analysis is to be performed. If a biaxial analysis is performed, the solution will consider moments applied simultaneously about the two orthogonal axes of the footing. If a biaxial analysis is NOT performed, the solution will consider moments applied about the two orthogonal axes to be acting non-concurrently.

Amount of Edge Length for M & V (Only displayed when biaxial analysis is selected)

When calculating shear and moment for footings where the maximum soil pressure values occur at the corners, this value specifies the fraction (as a decimal) of the footing dimension from the edge to use when calculating moments and shears due to variable soil pressure in that region. A smaller value for this variable will produce a more conservative design, because it will focus on a narrower strip which is experiencing the highest soil bearing pressure.

Click to Calculate (Button is only visible when biaxial analysis is selected)

Due to the iterative nature of the calculations that are required for a biaxial analysis, it would be undesirable to rerun the entire analysis and design every time an input parameter changes. So for reasons of efficiency, the program automatically goes into manual recalculation mode when biaxial analysis is selected. Click this button any time you wish to recalculate with the current input parameters.

Consider footing weight when determining soil bearing pressure

Select this option to have the module calculate the self-weight of the footing and apply it as a downward load when determining soil bearing pressures. The self-weight will be multiplied by the dead load factor in each of the Soil Bearing Pressure load combinations.

Note: This option should generally be selected. Deselecting this option can lead to incorrect soil bearing pressure calculations in footings with moment. If the goal is to try to compare soil bearing pressures to net allowable pressures, then it would be advisable to use the option on the Soil Allowables tab to "Increase Bearing by Footing Weight".

Consider footing weight when determining sliding, overturning and uplift

Select this option to have the module calculate the self-weight of the footing and apply it as a downward load when determining sliding, overturning and uplift factors of safety. The self-weight will be multiplied by the dead load factor in each of the Stability load combinations.

Ignore Checks for Sliding

Select this option if sliding is not a design consideration for any particular reason.

Min Steel Ratio - Temperature/Shrinkage

Enter the minimum ratio for temperature/shrinkage steel, calculated using the full footing thickness. This will trigger a warning message if the section is under-reinforced.

Note: This check is performed assuming that only one mat of the defined rebar will be provided. If the design has net uplift, such that a top mat is warranted, or if a top mat will be provided anyway, then be aware that the program will still only consider the contribution of one mat toward meeting the Temperature & Shrinkage requirement. In this case, it may be more convenient to set the T&S ratio to a value that represents one-half of the total, knowing that the two mats will be sufficient to provide the full amount required.

Minimum overturning safety factor

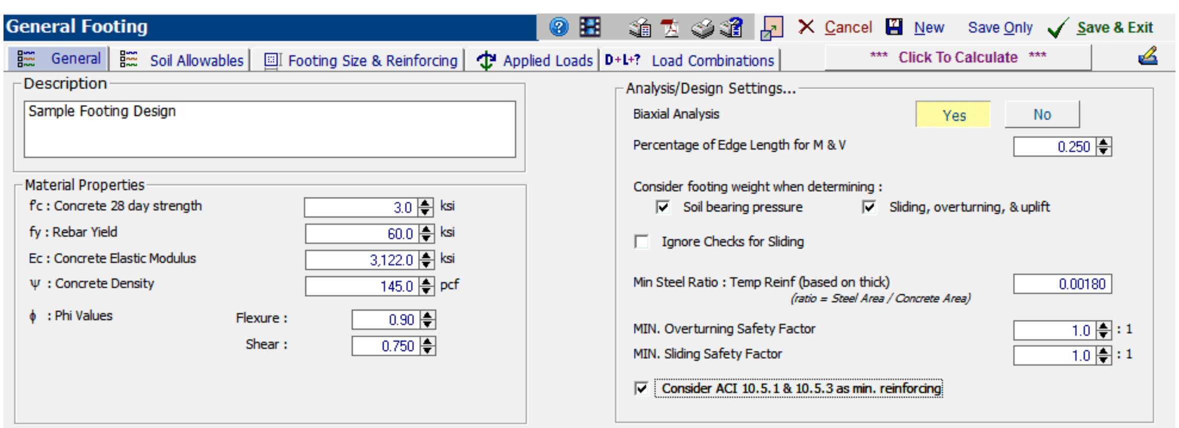
Enter the minimum allowable ratio of resisting moment to overturning moment. If the actual ratio is less than the specified minimum ratio, it will trigger a message that overturning stability is not satisfied.

Minimum sliding safety factor

Enter the minimum allowable ratio of resisting force to sliding force. If the actual ratio is less than the specified minimum ratio, it will trigger a message that sliding stability is not satisfied.

Consider ACI 10.5.1 & 10.5.3 as minimum reinforcing

Select this checkbox if you wish to have the module consider ACI 318 Sections 10.5.1 and 10.5.3 in the determination of minimum reinforcing.



Soil Allowable Values

Allowable Soil Bearing

Enter the allowable soil bearing pressure that the soil can resist. This is a service load resistance and will be compared to calculated service load soil pressures (loads not factored as in strength design).

Increase bearing by footing weight

Click [**Yes**] to tell the module to calculate the weight of one square foot (plan view) of footing weight and add it to the allowable soil bearing value. This has the effect of not penalizing the soil for the self weight of the footing, and is useful for situations where the geotechnical engineering report provides allowable net bearing pressures.

Soil passive sliding resistance

Enter the value of passive soil pressure resistance to sliding. This value will be used to determine a component of sliding resistance that is generated by the passive pressure of the soil. The sliding resistance due to passive pressure is then added to the sliding resistance due to friction to determine the total resistance to sliding for each load combination.

Coefficient of Soil/Concrete Friction

Enter the coefficient of friction between soil and footing to use in sliding resistance calculations.

Soil Bearing Increase

This section allows you to specify some dimensions that, when exceeded, will automatically increase the allowable soil bearing pressure.

Footing base depth below soil surface: The distance from the bottom of the footing to the top of the soil. This value is used to determine allowable soil bearing pressure increases and soil passive sliding resistance, but it is not used in any other calculations in this module.

Increases based on footing depth: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing depth below some reference depth. Collects the following parameters:

Allowable pressure increase per foot: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of depth below some reference depth.

When base of footing is below: Specifies the required depth in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing depth.

Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing base is 6'-0" below soil surface. The Geotechnical report specifies that a 0.15 ksf increase in bearing pressure is allowed for each foot of depth when the base is deeper than 4' below top of soil. Since you've indicated that the footing is 6' below the soil surface, the module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (6' - 4') * 0.15 ksf = 3.30 ksf.

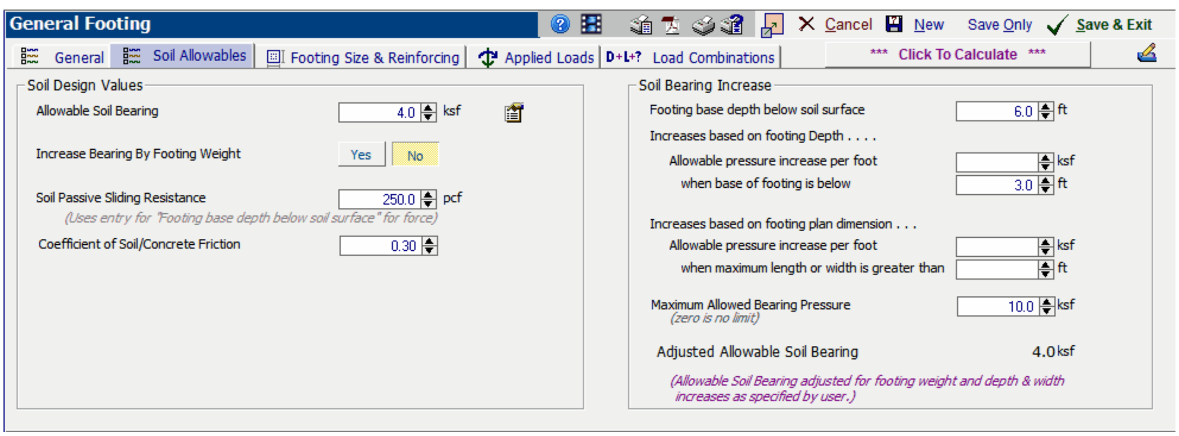
Increases based on footing plan dimension: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing dimensions greater than some reference dimension. Collects the following parameters:

Allowable pressure increase per foot: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of length or width greater than some reference dimension.

When maximum length or width is greater than: Specifies the required dimension in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing dimension.

Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing measures 12'-0" x 6'-0". The Geotechnical report specifies that a 0.15 ksf increase in soil bearing pressure is allowed for each foot when the largest plan dimension of the footing is greater than 4'. The module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (12' - 4') * 0.15 ksf = 4.2 ksf.

<u>Note: Increases based on footing depth and plan dimensions are cumulative.</u>



Footing Dimensions

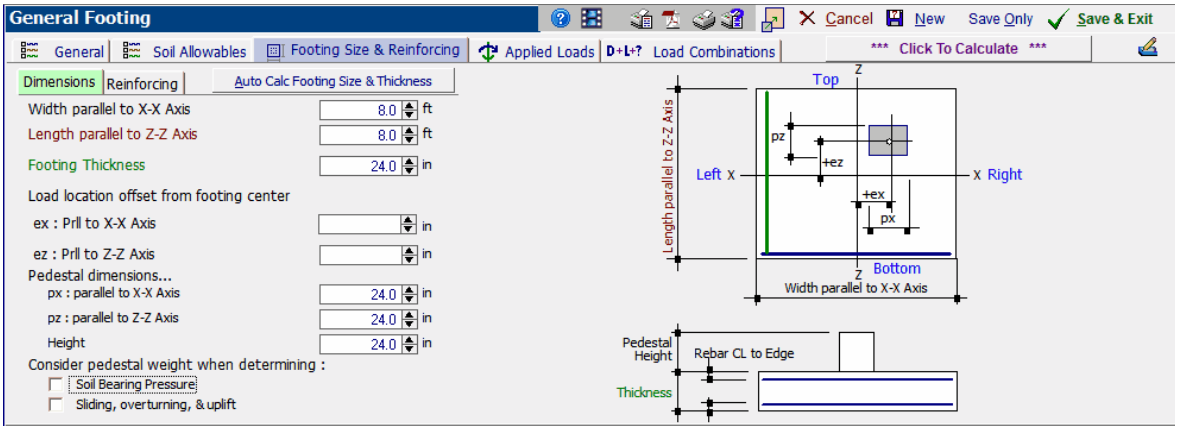
This tab is where you enter the footing and pedestal dimensions.

Width, Length & Thickness: defines the overall dimensions of the footing

Load Location: defines the offset from the center of the footing where the axial load is applied. If a biaxial analysis is NOT being used, then only one direction can be used.

Pedestal dimensions: If a concrete pedestal bears on the footing, its dimensions can be specified here. The px and pz dimensions are used to define the locations on all four sides where one-way shear, two-way shear and bending moment are calculated. If you enter a nonzero height, then you can choose to have the weight of that prism calculated and added as dead load. Any applied overburden loads will be omitted from the area defined as the pedestal dimension along the xx and yy axis, regardless of the specified height of the prism.

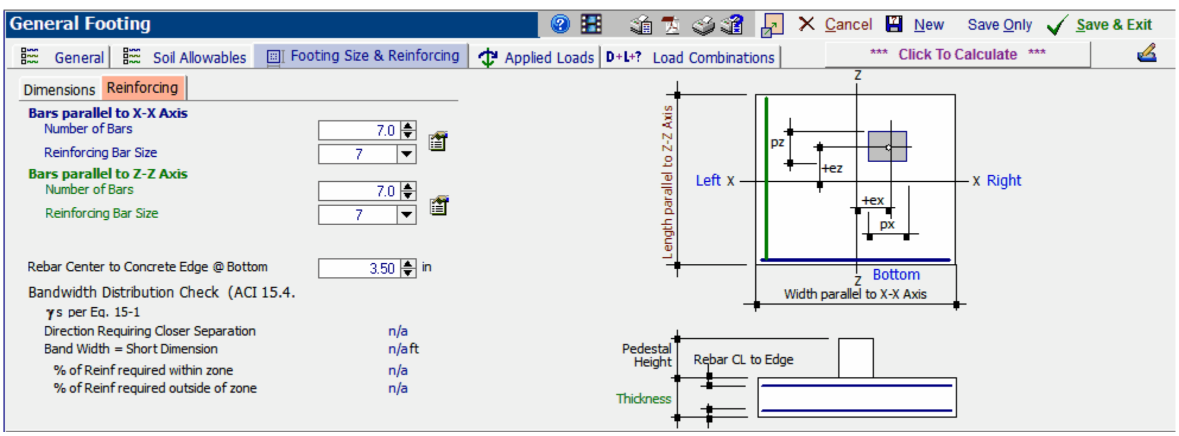
Note: If no pedestal is defined, then the center of the footing will be treated as the face of the pedestal when determining the critical locations to check shear and flexure.



Consider Pedestal Weight When Determining: This option allows the user to specify whether or not the self-weight of the pedestal is to be considered when determining the soil bearing pressure, and separately, whether or not the self-weight of the pedestal is to be considered when performing the checks for sliding, overturning, and uplift.

Footing Reinforcing

This tab allows you to specify the reinforcing in each direction of the footing.

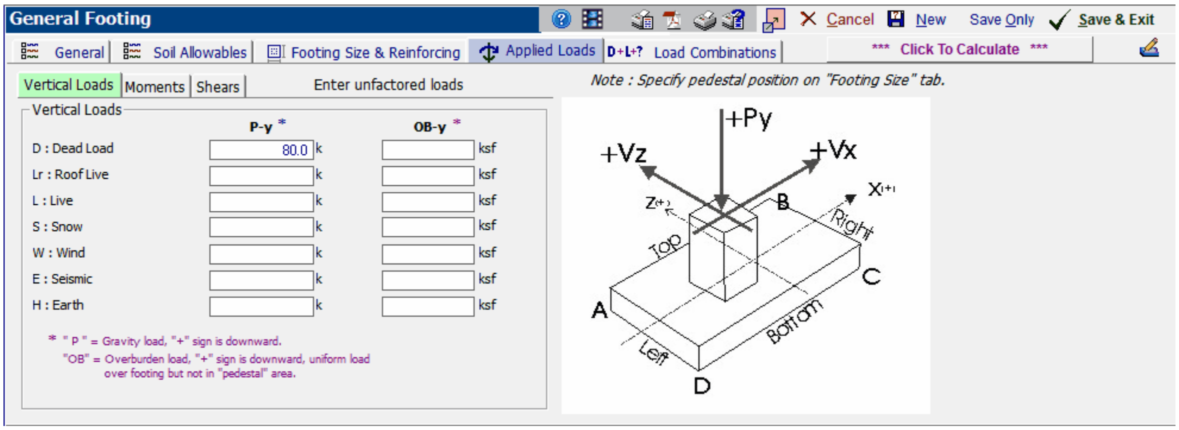


Applied VERTICAL Loads

This tab allows you to specify the axial load applied to the pedestal location and an overburden load applied to the entire plan dimension of the footing (except the area designated as the pedestal).

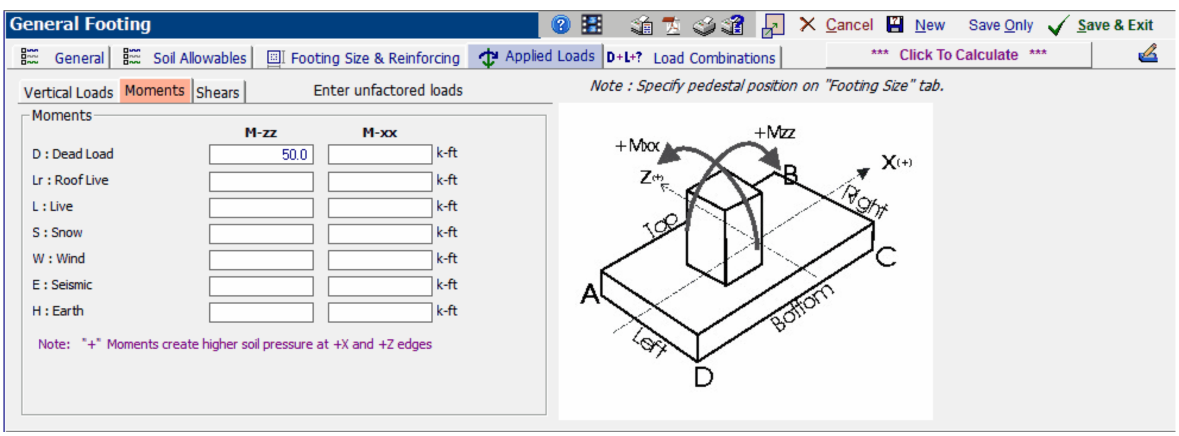
Enter loads with a positive sign for downward direction.

Note! This module will not allow a net uplift on the footing. If the result of the factored axial loads (dead, live, wind, etc) produces a negative load sign, the module will not recalculate and will notify you of which load combination resulted in net uplift.



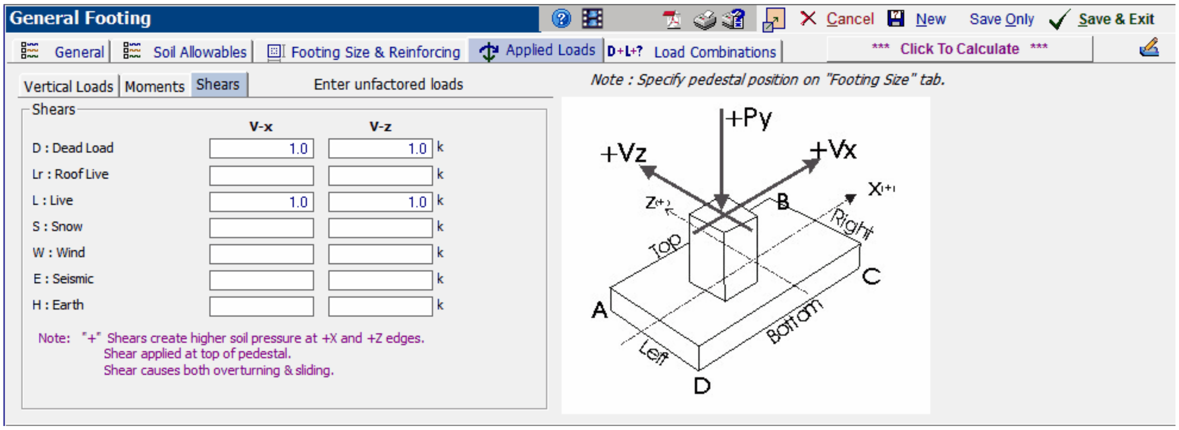
Applied BENDING Loads

This tab allows you to enter applied moments.



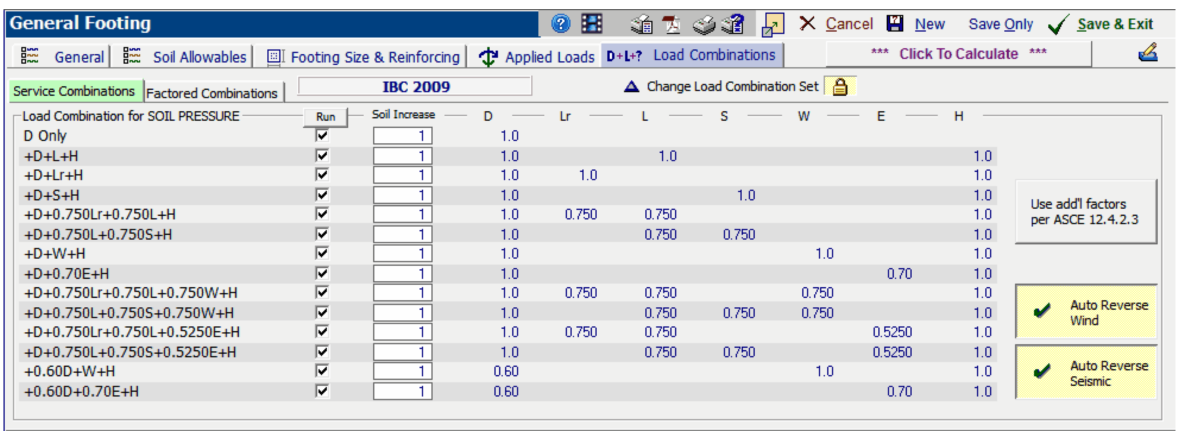
Applied SHEAR Loads

This tab allows you to enter applied shear forces. These loads are applied at the location of the pedestal. If the pedestal is specified to have a height, the shear will be applied at that height and will create a moment on the footing equal to Shear Load * (Footing Thickness + Pedestal Height).



Load Combinations - Service

This is the typical load combination tab used throughout **Structural Engineering Library**. The Service Combinations tab is used to calculate soil pressures to be compared to the allowable soil bearing pressure. The "Soil Increase" is a factor that can be specified individually for each load combination and is applied to the allowable soil bearing pressure.

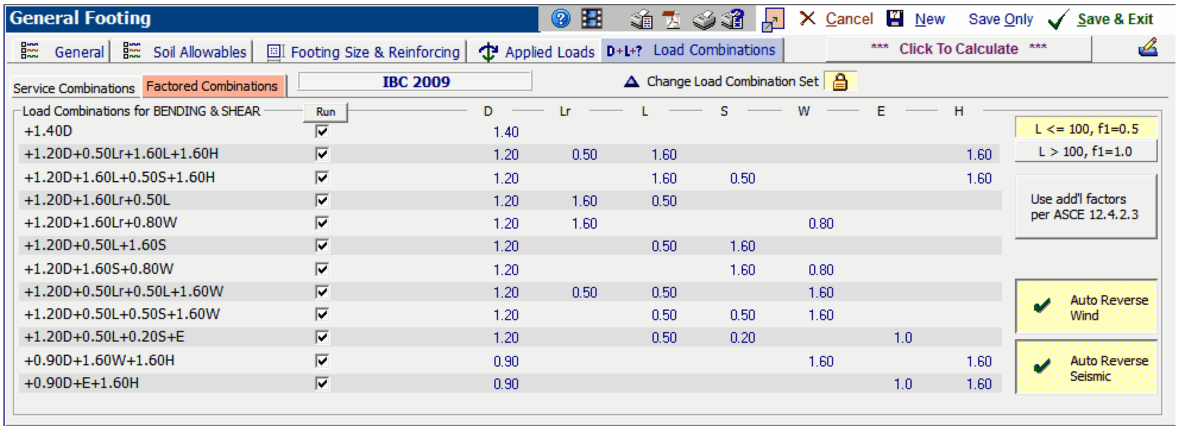


Load Combination for SOIL PRESSURE	Run	Soil Increase	D	Lr	L	S	W	E	H
D Only	☑	1	1.0						
+D+L+H	☑	1	1.0		1.0				1.0
+D+Lr+H	☑	1	1.0	1.0					1.0
+D+S+H	☑	1	1.0			1.0			1.0
+D+0.750Lr+0.750L+H	☑	1	1.0	0.750	0.750				1.0
+D+0.750L+0.750S+H	☑	1	1.0		0.750	0.750			1.0
+D+W+H	☑	1	1.0				1.0		1.0
+D+0.70E+H	☑	1	1.0					0.70	1.0
+D+0.750Lr+0.750L+0.750W+H	☑	1	1.0	0.750	0.750		0.750		1.0
+D+0.750L+0.750S+0.750W+H	☑	1	1.0		0.750	0.750	0.750		1.0
+D+0.750Lr+0.750L+0.5250E+H	☑	1	1.0	0.750	0.750			0.5250	1.0
+D+0.750L+0.750S+0.5250E+H	☑	1	1.0		0.750	0.750		0.5250	1.0
+0.60D+W+H	☑	1	0.60				1.0		1.0
+0.60D+0.70E+H	☑	1	0.60					0.70	1.0

Load Combinations - Factored

This is the typical load combination tab used throughout **Structural Engineering Library** for strength design. These load combinations are used to calculate moments and shears in the footing for use in determining stresses and required reinforcing.

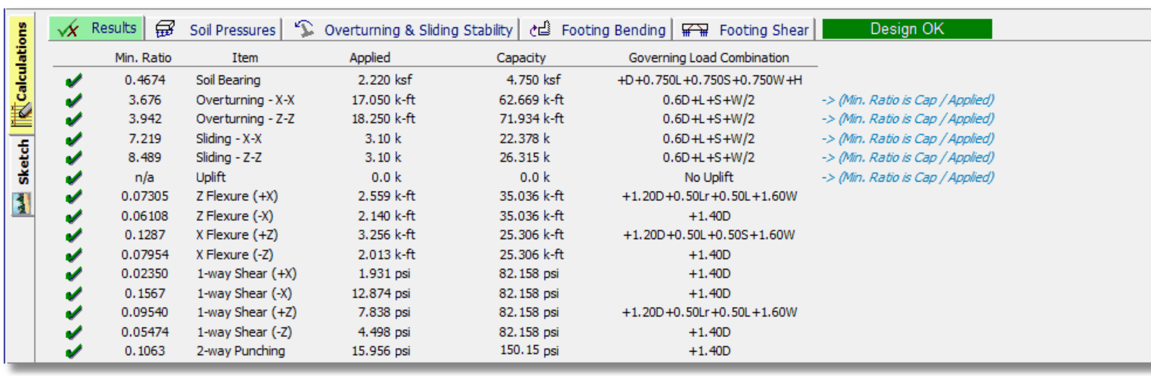
Note: The General Footing module is applying the factored loads to the footing and determining a different eccentricity than was determined using the service loads for the soil bearing pressure check.



Load Combinations for BENDING & SHEAR	Run	D	Lr	L	S	W	E	H
+1.40D	☑	1.40						
+1.20D+0.50Lr+1.60L+1.60H	☑	1.20	0.50	1.60				1.60
+1.20D+1.60L+0.50S+1.60H	☑	1.20		1.60	0.50			1.60
+1.20D+1.60Lr+0.50L	☑	1.20	1.60	0.50				
+1.20D+1.60Lr+0.80W	☑	1.20	1.60			0.80		
+1.20D+0.50L+1.60S	☑	1.20		0.50	1.60			
+1.20D+1.60S+0.80W	☑	1.20			1.60	0.80		
+1.20D+0.50Lr+0.50L+1.60W	☑	1.20	0.50	0.50		1.60		
+1.20D+0.50L+0.50S+1.60W	☑	1.20		0.50	0.50	1.60		
+1.20D+0.50L+0.20S+E	☑	1.20		0.50	0.20		1.0	
+0.90D+1.60W+1.60H	☑	0.90				1.60		1.60
+0.90D+E+1.60H	☑	0.90					1.0	1.60

Results Tab

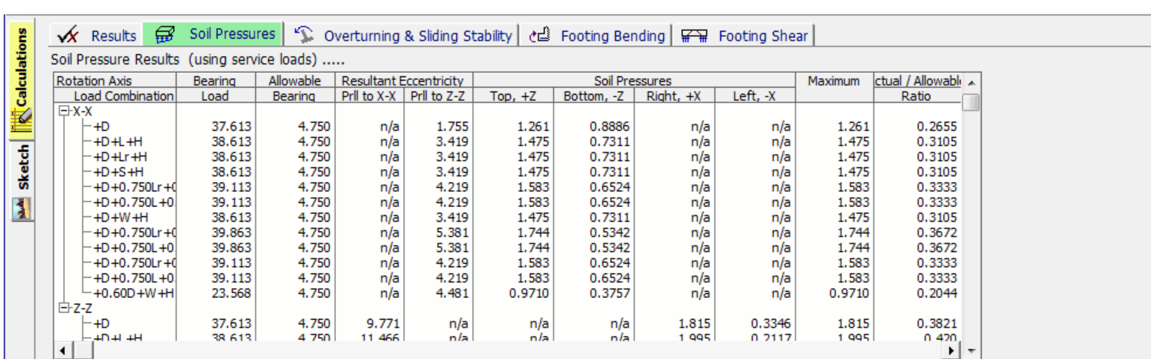
This tab provides a summary of all calculated values. The stress ratios, applied & allowable values and load combination for those governing values are reported.



	Min. Ratio	Item	Applied	Capacity	Governing Load Combination	
✔	0.4674	Soil Bearing	2.220 ksf	4.750 ksf	+D+0.750L+0.750S+0.750W+H	
✔	3.676	Overturning - X-X	17.050 k-ft	62.669 k-ft	0.6D+L+S+W/2	-> (Min. Ratio is Cap / Applied)
✔	3.942	Overturning - Z-Z	18.250 k-ft	71.934 k-ft	0.6D+L+S+W/2	-> (Min. Ratio is Cap / Applied)
✔	7.219	Sliding - X-X	3.10 k	22.378 k	0.6D+L+S+W/2	-> (Min. Ratio is Cap / Applied)
✔	8.489	Sliding - Z-Z	3.10 k	26.315 k	0.6D+L+S+W/2	-> (Min. Ratio is Cap / Applied)
✔	n/a	Uplift	0.0 k	0.0 k	No Uplift	-> (Min. Ratio is Cap / Applied)
✔	0.07305	Z Flexure (+X)	2.559 k-ft	35.036 k-ft	+1.20D+0.50Lr+0.50L+1.60W	
✔	0.06108	Z Flexure (-X)	2.140 k-ft	35.036 k-ft	+1.40D	
✔	0.1287	X Flexure (+Z)	3.256 k-ft	25.306 k-ft	+1.20D+0.50L+0.50S+1.60W	
✔	0.07954	X Flexure (-Z)	2.013 k-ft	25.306 k-ft	+1.40D	
✔	0.02350	1-way Shear (+X)	1.931 psi	82.158 psi	+1.40D	
✔	0.1567	1-way Shear (-X)	12.874 psi	82.158 psi	+1.40D	
✔	0.09540	1-way Shear (+Z)	7.838 psi	82.158 psi	+1.20D+0.50Lr+0.50L+1.60W	
✔	0.05474	1-way Shear (-Z)	4.498 psi	82.158 psi	+1.40D	
✔	0.1063	2-way Punching	15.956 psi	150.15 psi	+1.40D	

Soil Pressures Tab

This tab summarizes the calculated service load soil bearing pressure for moments & shears applied about the specified axis, for each load combination.

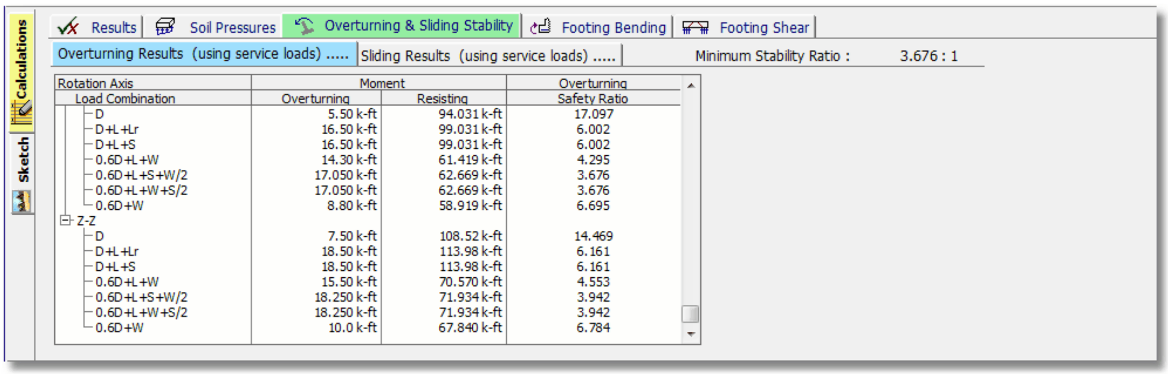


Rotation Axis	Bearing	Allowable	Resultant Eccentricity		Soil Pressures				Maximum	ctual / Allowable
Load Combination	Load	Bearing	Prll to X-X	Prll to Z-Z	Top, +Z	Bottom, -Z	Right, +X	Left, -X		Ratio
X-X										
+D	37.613	4.750	n/a	1.755	1.261	0.8886	n/a	n/a	1.261	0.2655
+D+L+H	38.613	4.750	n/a	3.419	1.475	0.7311	n/a	n/a	1.475	0.3105
+D+Lr+H	38.613	4.750	n/a	3.419	1.475	0.7311	n/a	n/a	1.475	0.3105
+D+S+H	38.613	4.750	n/a	3.419	1.475	0.7311	n/a	n/a	1.475	0.3105
+D+0.750Lr+0	39.113	4.750	n/a	4.219	1.583	0.6524	n/a	n/a	1.583	0.3333
+D+0.750L+0	39.113	4.750	n/a	4.219	1.583	0.6524	n/a	n/a	1.583	0.3333
+D+W+H	38.613	4.750	n/a	3.419	1.475	0.7311	n/a	n/a	1.475	0.3105
+D+0.750Lr+0	39.863	4.750	n/a	5.381	1.744	0.5342	n/a	n/a	1.744	0.3672
+D+0.750L+0	39.863	4.750	n/a	5.381	1.744	0.5342	n/a	n/a	1.744	0.3672
+D+0.750Lr+0	39.113	4.750	n/a	4.219	1.583	0.6524	n/a	n/a	1.583	0.3333
+D+0.750L+0	39.113	4.750	n/a	4.219	1.583	0.6524	n/a	n/a	1.583	0.3333
+0.60D+W+H	23.568	4.750	n/a	4.481	0.9710	0.3757	n/a	n/a	0.9710	0.2044
Z-Z										
+D	37.613	4.750	9.771	n/a	n/a	n/a	1.815	0.3346	1.815	0.3821
+D+L+H	38.613	4.750	11.466	n/a	n/a	n/a	1.995	0.2117	1.995	0.420

Overturning Stability Tab

This tab provides the calculations for overturning and resisting moment stability of the footing about each axis and for each load combination. Please note that the load combinations used here are internally generated and are NOT from the Service Load Combinations you have entered for the purpose of evaluating soil bearing pressures.

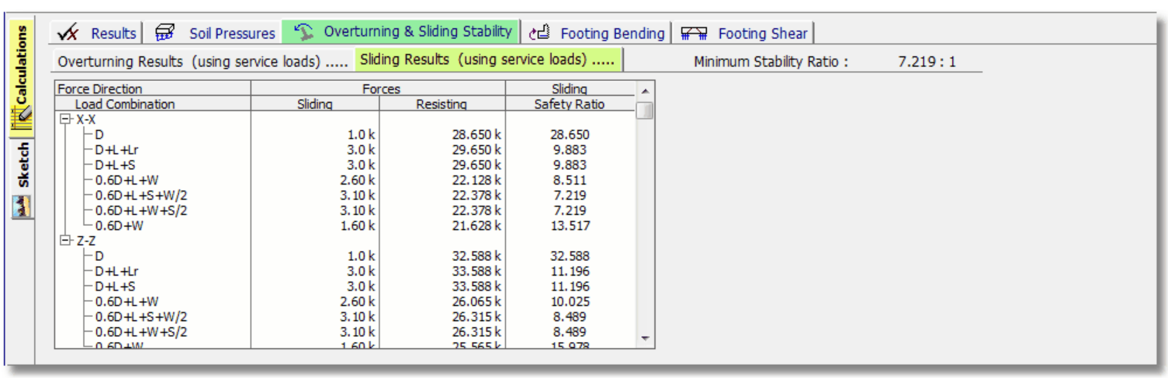
Note that the program is set up to look for overturning and resisting forces individually. For example, take the situation where the footing is subjected to equal and opposite shears at a given elevation. Common sense dictates that these forces cancel each other, and the footing experiences no net applied overturning moment from them. But the program treats one of the two equal and opposite forces as an *overturning* force, and the other as a *resisting* force. So for these two forces, there IS a net overturning moment reported, but the resisting moment ALSO considers the effect of the opposing load, so the accounting used to determine the overturning ratio is proper.



Rotation Axis / Load Combination	Moment: Overturning	Moment: Resisting	Overturning Safety Ratio
D	5.50 k-ft	94.031 k-ft	17.097
D+L+Lr	16.50 k-ft	99.031 k-ft	6.002
D+L+S	16.50 k-ft	99.031 k-ft	6.002
0.6D+L+W	14.30 k-ft	61.419 k-ft	4.295
0.6D+L+S+W/2	17.050 k-ft	62.669 k-ft	3.676
0.6D+L+W+S/2	17.050 k-ft	62.669 k-ft	3.676
0.6D+W	8.80 k-ft	58.919 k-ft	6.695
Z-Z			
D	7.50 k-ft	108.52 k-ft	14.469
D+L+Lr	18.50 k-ft	113.98 k-ft	6.161
D+L+S	18.50 k-ft	113.98 k-ft	6.161
0.6D+L+W	15.50 k-ft	70.570 k-ft	4.553
0.6D+L+S+W/2	18.250 k-ft	71.934 k-ft	3.942
0.6D+L+W+S/2	18.250 k-ft	71.934 k-ft	3.942
0.6D+W	10.0 k-ft	67.840 k-ft	6.784

Sliding Stability Tab

This tab provides the calculations for applied and resisting sliding stability of the footing in each axis direction and for each load combination. Please note the load combinations used here are internally generated and are NOT from the Service Load Combinations you have entered for the purpose of evaluating soil bearing pressures.

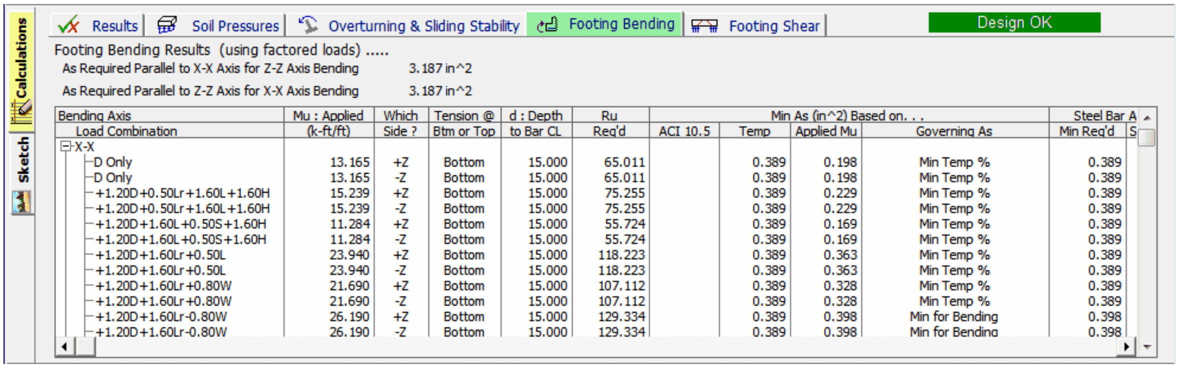


Force Direction / Load Combination	Forces: Sliding	Forces: Resisting	Sliding Safety Ratio
X-X			
D	1.0 k	28.650 k	28.650
D+L+Lr	3.0 k	29.650 k	9.883
D+L+S	3.0 k	29.650 k	9.883
0.6D+L+W	2.60 k	22.128 k	8.511
0.6D+L+S+W/2	3.10 k	22.378 k	7.219
0.6D+L+W+S/2	3.10 k	22.378 k	7.219
0.6D+W	1.60 k	21.628 k	13.517
Z-Z			
D	1.0 k	32.588 k	32.588
D+L+Lr	3.0 k	33.588 k	11.196
D+L+S	3.0 k	33.588 k	11.196
0.6D+L+W	2.60 k	26.065 k	10.025
0.6D+L+S+W/2	3.10 k	26.315 k	8.489
0.6D+L+W+S/2	3.10 k	26.315 k	8.489
0.6D+W	1.60 k	25.565 k	15.978

Footing Bending Tab

This tab provides a summary of the calculated factored load moment at all four edges of the pedestal perimeter for each load combination. It indicates whether the named load combination produces tension on the top surface of the footing or the bottom.

Note: In cases where tension occurs on the top of the footing, the flexural check will be based on the assumption that the defined rebar mat is provided on the top surface of the footing. The user must review the results and determine if any load combinations actually require a top mat of reinforcing, or if the footing could be reinforced with a bottom mat only.



Results | Soil Pressures | Overturning & Sliding Stability | Footing Bending | Footing Shear | Design OK

Footing Bending Results (using factored loads) .....

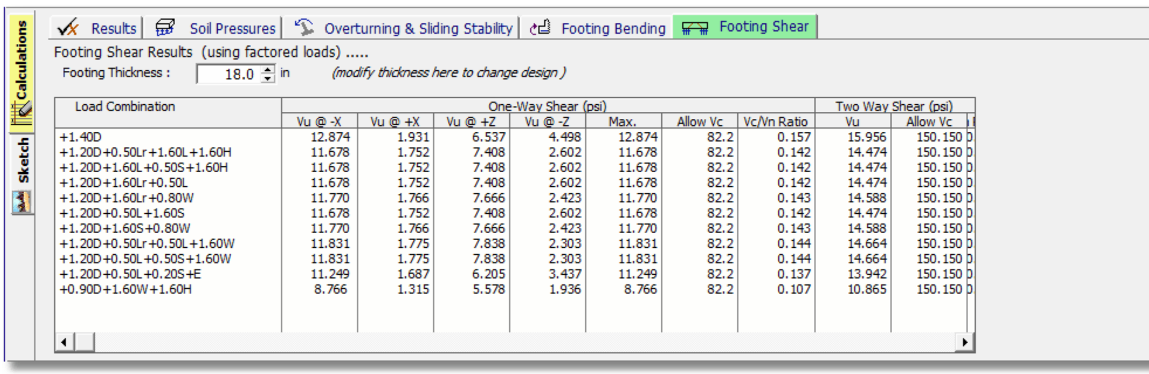
As Required Parallel to X-X Axis for Z-Z Axis Bending 3.187 in^2

As Required Parallel to Z-Z Axis for X-X Axis Bending 3.187 in^2

Bending Axis / Load Combination	Mu : Applied (k-ft/ft)	Which Side ?	Tension @ Btm or Top	d : Depth to Bar CL	Ru Req'd	Min As (in^2) Based on. . . ACI 10.5	Temp	Applied Mu	Governing As	Steel Bar A Min Req'd
X-X										
D Only	13.165	+Z	Bottom	15.000	65.011		0.389	0.198	Min Temp %	0.389
D Only	13.165	-Z	Bottom	15.000	65.011		0.389	0.198	Min Temp %	0.389
+1.20D+0.50Lr+1.60L+1.60H	15.239	+Z	Bottom	15.000	75.255		0.389	0.229	Min Temp %	0.389
+1.20D+0.50Lr+1.60L+1.60H	15.239	-Z	Bottom	15.000	75.255		0.389	0.229	Min Temp %	0.389
+1.20D+1.60L+0.50S+1.60H	11.284	+Z	Bottom	15.000	55.724		0.389	0.169	Min Temp %	0.389
+1.20D+1.60L+0.50S+1.60H	11.284	-Z	Bottom	15.000	55.724		0.389	0.169	Min Temp %	0.389
+1.20D+1.60Lr+0.50L	23.940	+Z	Bottom	15.000	118.223		0.389	0.363	Min Temp %	0.389
+1.20D+1.60Lr+0.50L	23.940	-Z	Bottom	15.000	118.223		0.389	0.363	Min Temp %	0.389
+1.20D+1.60Lr+0.80W	21.690	+Z	Bottom	15.000	107.112		0.389	0.328	Min Temp %	0.389
+1.20D+1.60Lr+0.80W	21.690	-Z	Bottom	15.000	107.112		0.389	0.328	Min Temp %	0.389
+1.20D+1.60Lr-0.80W	26.190	+Z	Bottom	15.000	129.334		0.389	0.398	Min for Bending	0.398
+1.20D+1.60Lr-0.80W	26.190	-Z	Bottom	15.000	129.334		0.389	0.398	Min for Bending	0.398

Footing Shear Tab

This tab provides a summary of the calculated factored load shear at all four edges of the pedestal perimeter for each load combination. Two-way or punching shear is also calculated.



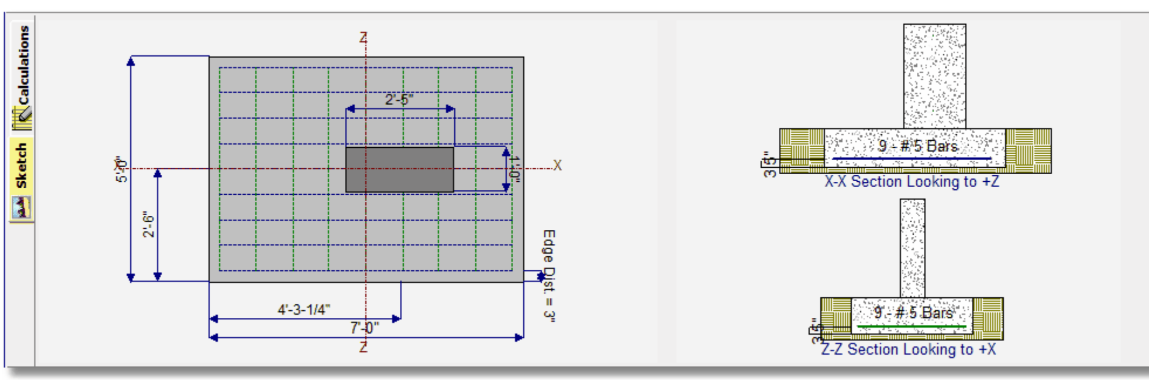
Results | Soil Pressures | Overturning & Sliding Stability | Footing Bending | Footing Shear

Footing Shear Results (using factored loads) .....

Footing Thickness : 18.0 in *(modify thickness here to change design)*

Load Combination	One-Way Shear (psi) Vu @ -X	Vu @ +X	Vu @ +Z	Vu @ -Z	Max.	Allow Vc	Vc/Vn Ratio	Two Way Shear (psi) Vu	Allow Vc
+1.40D	12.874	1.931	6.537	4.498	12.874	82.2	0.157	15.956	150.150
+1.20D+0.50Lr+1.60L+1.60H	11.678	1.752	7.408	2.602	11.678	82.2	0.142	14.474	150.150
+1.20D+1.60L+0.50S+1.60H	11.678	1.752	7.408	2.602	11.678	82.2	0.142	14.474	150.150
+1.20D+1.60Lr+0.50L	11.678	1.752	7.408	2.602	11.678	82.2	0.142	14.474	150.150
+1.20D+1.60Lr+0.80W	11.770	1.766	7.666	2.423	11.770	82.2	0.143	14.588	150.150
+1.20D+0.50L+1.60S	11.678	1.752	7.408	2.602	11.678	82.2	0.142	14.474	150.150
+1.20D+1.60S+0.80W	11.770	1.766	7.666	2.423	11.770	82.2	0.143	14.588	150.150
+1.20D+0.50Lr+0.50L+1.60W	11.831	1.775	7.838	2.303	11.831	82.2	0.144	14.664	150.150
+1.20D+0.50L+0.50S+1.60W	11.831	1.775	7.838	2.303	11.831	82.2	0.144	14.664	150.150
+1.20D+0.50L+0.20S+E	11.249	1.687	6.205	3.437	11.249	82.2	0.137	13.942	150.150
+0.90D+1.60W+1.60H	8.766	1.315	5.578	1.936	8.766	82.2	0.107	10.865	150.150

Sketch Tab

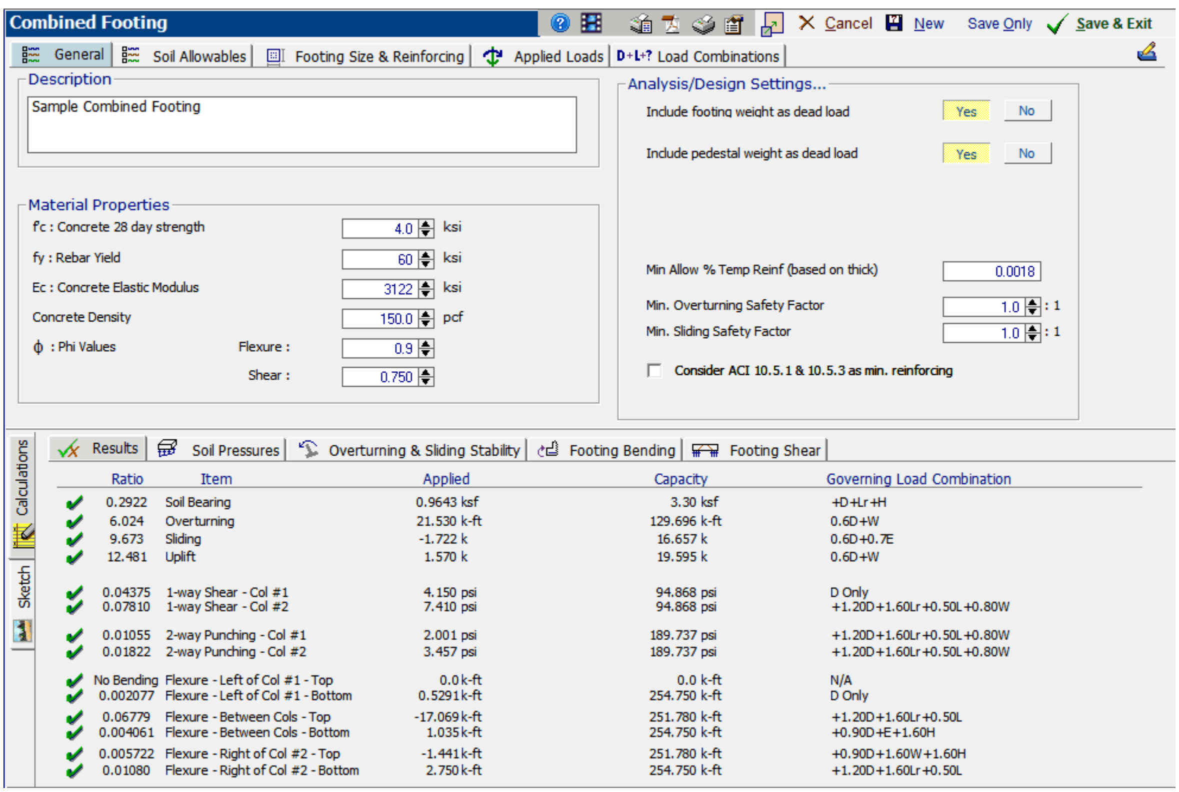


10.7.2 Combined Footing

Need more? Ask Us a Question

This module provides analysis of a rectangular footing with two applied axial, moment and shear loads. Overburden loads can also be specified, and will apply to the surface area of the footing (except the areas covered by the two piers). The module allows you to position the application of the pier loads as necessary, and provides automatic calculation of allowable soil bearing pressure increases based on footing dimensions and/or depth below surface.

The module checks service load soil pressure, overturning stability, sliding stability, uplift stability, flexure left and right of each pedestal, 1-way shear at 'd' from each of the pedestals, and punching shear along a perimeter located 'd/2' from the pedestal faces. The module does not evaluate the footing for flexure about the Length axis.



General

f'c

28-day compressive strength of the concrete.

fy

Yield point stress of reinforcing.

Ec

Modulus of elasticity of concrete.

Concrete Density

The density of the concrete is used to calculate the self weight of the pedestals and footing when those options are selected.

Phi Values

Enter the capacity reduction values to be applied to Vn and Mn.

Include footing weight as dead load

Click [**Yes**] to have the module calculate the weight of the footing and apply it as a downward load. The footing self weight will be multiplied by the dead load factor in each load combination.

Include pedestal weight as dead load

Click [**Yes**] to have the module calculate the weight of the pedestals and apply it as downward loads. The pedestal self weight will be multiplied by the dead load factor in each load combination.

Min Steel Ratio - Temperature/Shrinkage

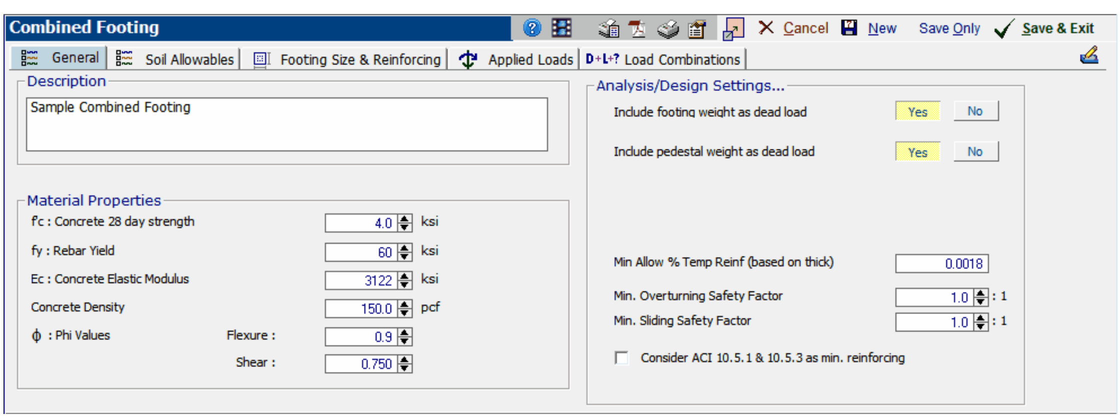
Enter the minimum ratio for temperature/shrinkage steel, calculated using the footing thickness. This will trigger a warning message if the section is under-reinforced.

Minimum Overturning Safety Factor

Enter the minimum allowable ratio of resisting moment to overturning moment. If the actual ratio is less than the specified minimum ratio, it will trigger a message that overturning stability is not satisfied.

Minimum Sliding Safety Factor

Enter the minimum allowable ratio of resisting force to sliding force. If the actual ratio is less than the specified minimum ratio, it will trigger a message that sliding stability is not satisfied.



Soil Allowables

Allowable Soil Bearing

Enter the allowable soil bearing pressure that the soil can resist. This is a service load resistance and will be compared to calculated service load soil pressures (loads not factored as in strength design).

Increase Bearing by Footing Weight

Click [**Yes**] to tell the module to calculate the weight of one square foot (plan view) of footing and add it to the allowable soil bearing value. This has the effect of not penalizing the soil for the self weight of the footing, and is useful for situations where the geotechnical engineering report provides allowable net bearing pressures.

Soil Passive Sliding Resistance

Enter the value of passive soil pressure resistance to sliding. This value will be used to determine a component of sliding resistance that is generated by the passive pressure of the soil. The sliding resistance due to passive pressure is then added to the sliding resistance due to friction to determine the total resistance to sliding for each load combination.

Coefficient of Soil/Concrete Friction

Enter the coefficient of friction between soil and footing to use in sliding resistance calculations.

Soil Bearing Increase

This section allows you to specify some dimensions that, when exceeded, will automatically increase the allowable soil bearing pressure.

Footing base depth below soil surface: The distance from the bottom of the footing to the top of the soil. This value is used to determine allowable soil bearing pressure increases and soil passive sliding resistance, but it is not used in any other calculations in this module.

Increases based on footing depth: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing depth below some reference depth. Collects the following parameters:

> **Allowable pressure increase per foot**: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of depth below some reference depth.
>
> **When base of footing is below**: Specifies the required depth in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing depth.

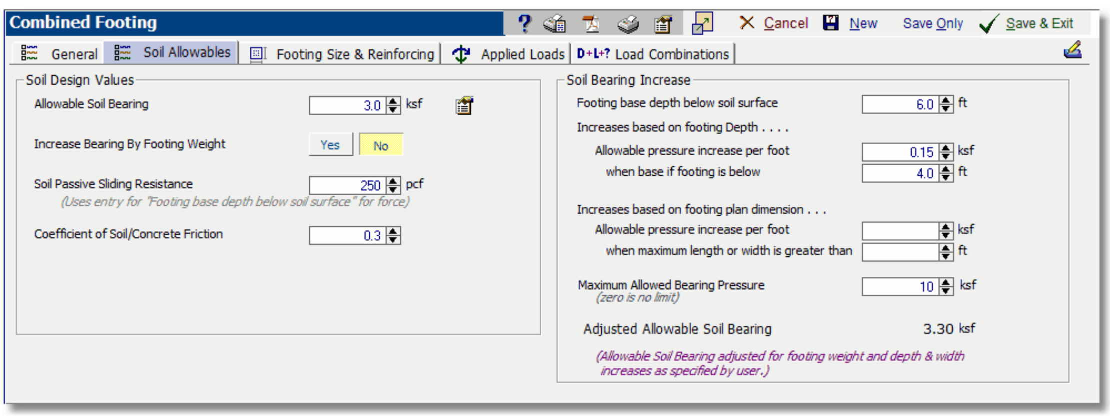
Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing base is 6'-0" below soil surface. The Geotechnical report specifies that a 0.15 ksf increase in bearing pressure is allowed for each foot of depth when the base is deeper than 4' below top of soil. Since you've indicated that the footing is 6' below the soil surface, the module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (6' - 4') * 0.15 ksf = 3.30 ksf.

Increases based on footing plan dimension: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing dimensions greater than some reference dimension. Collects the following parameters:

> **Allowable pressure increase per foot**: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of width or length greater than some reference dimension.
>
> **When maximum length or width is greater than**: Specifies the required dimension in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing dimension.

Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing measures 12'-0" x 6'-0". The Geotechnical report specifies that a 0.15 ksf increase in soil bearing pressure is allowed for each foot when the largest plan dimension of the footing is greater than 4'. The module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (12' - 4') * 0.15 ksf = 4.2 ksf.

Note: Increases based on footing depth and plan dimensions are cumulative.



Footing Size & Reinforcing

Dimensions tab

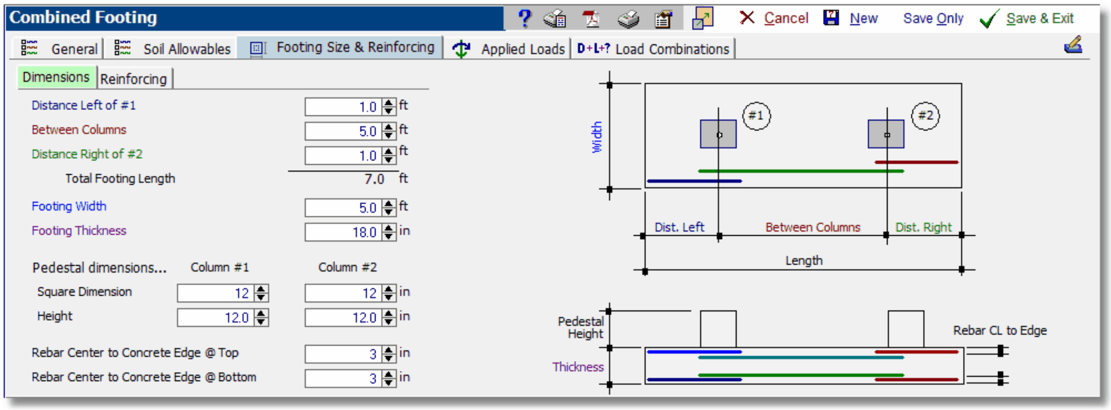
Projection on Left, Distance Between Columns, Projection on Right: Define the dimensions of the footing in the Length direction.

Footing Width: Define the dimension of the Width direction.

Footing Thickness: Define the total thickness of the footing.

Pedestal dimensions: If concrete pedestals bear on the footing, their dimensions can be specified here. Pedestals are assumed to be square, and they are assumed to be centered on the Width dimension of the footing.

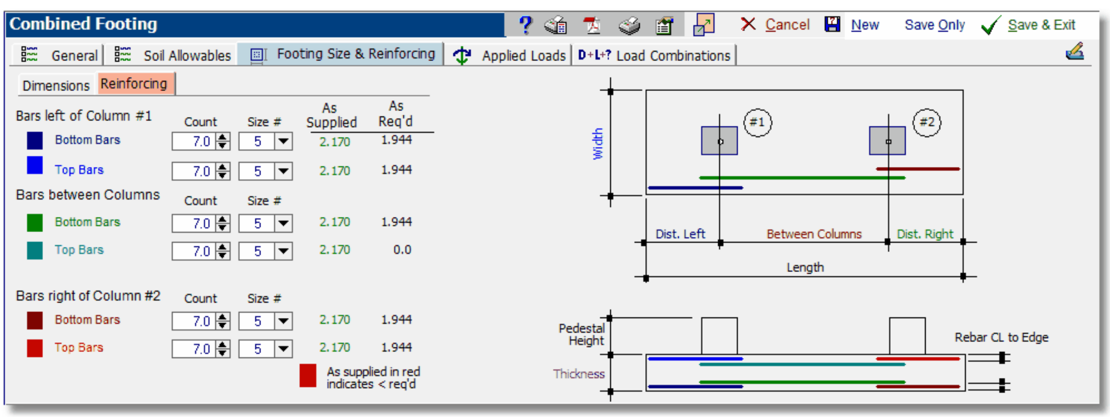
Note: Any applied overburden loads will be omitted from the area occupied by the pedestals.



Reinforcing tab

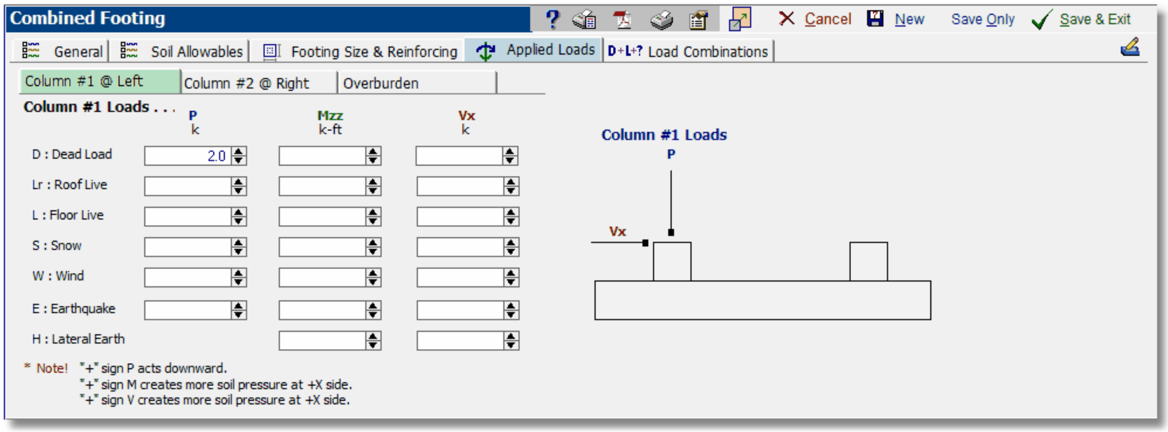
Reinforcing parallel to the Length dimension can be defined separately for the left and right projections of the footing and for the area between the columns. Input fields are provided to define top bars and bottom bars separately.

Note: Bars are assumed to be fully developed at the locations where they are required. It is the engineer's responsibility to validate that assumption. The program is not taking rebar development length into consideration.



Applied Loads

The Applied Loads tab provides sub-tabs for Column #1 (the column on the left), Column #2 (the column on the right), and Overburden. The two column load tabs offer input fields for vertical loads, moment about the Width axis, and shear in the Length direction. The Overburden tab provides input fields for a uniform vertical pressure that will be applied to the entire surface area of the footing with the exception of the areas occupied by the pedestals.

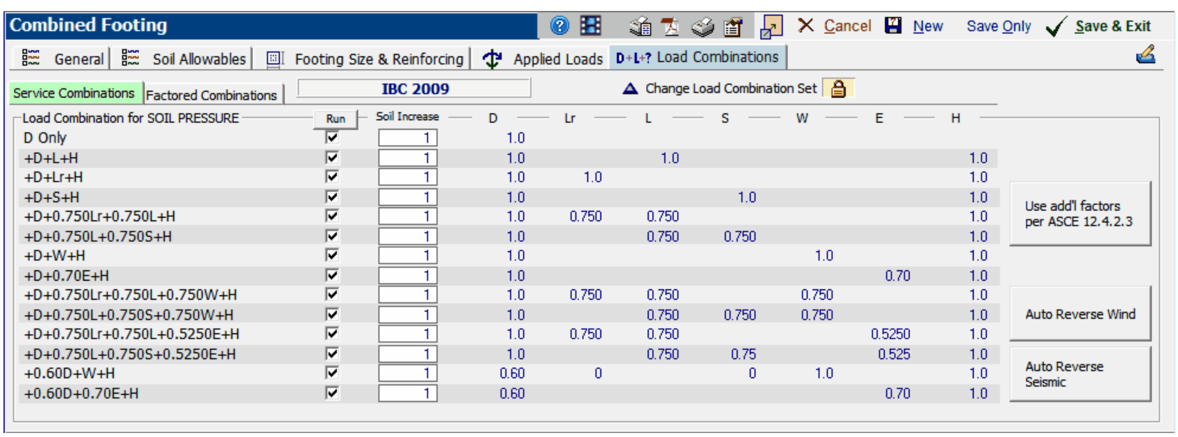


Load Combinations

The Load Combinations tab is used to specify the load combinations to be used in the design. The Service Combinations tab controls the load combinations that are used to perform the serviceability checks for Soil Bearing, Overturning, Sliding, and Uplift. The Factored Combinations tab controls the load combinations that are used to perform the strength checks for Flexure, One-Way Shear, and Two-Way Punching Shear.

These tabs allow the user to select from load combination sets that are supplied with the program or to select from custom load combination sets that have been created and saved on the user's machine. It is also possible to unlock the selected load combination set and make edits to the factors directly in this view. The user has control over which combinations are run and which are ignored. A Soil Increase factor can be applied on a load combination by load combination basis, as permitted by the geotechnical engineering report.

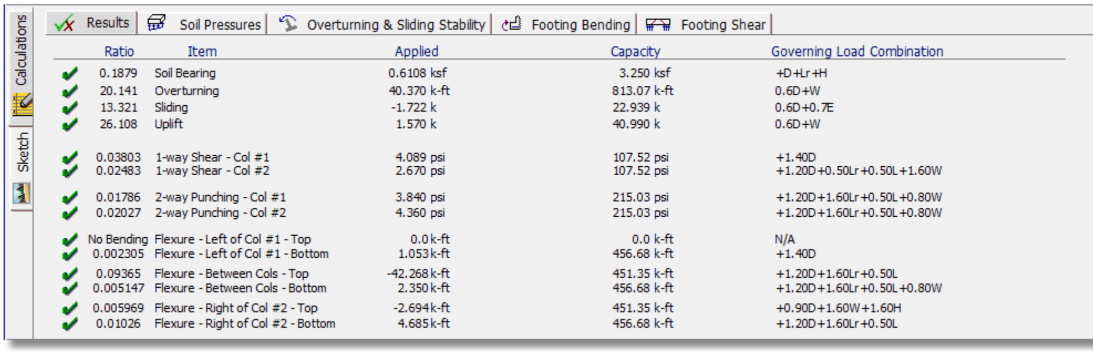
Finally, this tab allows the user to specify whether the program should consider the algebraic sign of the specified load factors on wind loads and seismic loads to be reversible or not. This can be a convenient way to ensure that these loads are investigated as acting in both positive and negative directions, if that is the design intent. Note, however, that if selected, the algebraic sign reversal will be applied to ALL wind loads and/or ALL seismic loads including horizontally AND vertically applied loads.



Calculations

Results tab

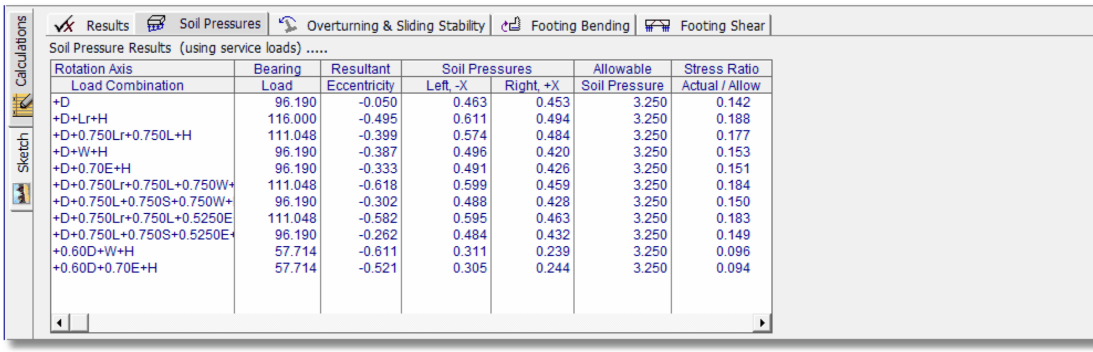
This tab summarizes the controlling values (highest utilization ratio) for each design consideration, from all of the load combinations that have been run. For the controlling load combination, it presents the Applied load, the Capacity or available resisting load, the ratio of the Applied to the Capacity, and the governing load combination that produces this controlling ratio.



Ratio	Item	Applied	Capacity	Governing Load Combination
0.1879	Soil Bearing	0.6108 ksf	3.250 ksf	+D+Lr+H
20.141	Overturning	40.370 k-ft	813.07 k-ft	0.6D+W
13.321	Sliding	-1.722 k	22.939 k	0.6D+0.7E
26.108	Uplift	1.570 k	40.990 k	0.6D+W
0.03803	1-way Shear - Col #1	4.089 psi	107.52 psi	+1.40D
0.02483	1-way Shear - Col #2	2.670 psi	107.52 psi	+1.20D+0.50Lr+0.50L+1.60W
0.01786	2-way Punching - Col #1	3.840 psi	215.03 psi	+1.20D+1.60Lr+0.50L+0.80W
0.02027	2-way Punching - Col #2	4.360 psi	215.03 psi	+1.20D+1.60Lr+0.50L+0.80W
No Bending	Flexure - Left of Col #1 - Top	0.0 k-ft	0.0 k-ft	N/A
0.002305	Flexure - Left of Col #1 - Bottom	1.053 k-ft	456.68 k-ft	+1.40D
0.09365	Flexure - Between Cols - Top	-42.268 k-ft	451.35 k-ft	+1.20D+1.60Lr+0.50L
0.005147	Flexure - Between Cols - Bottom	2.350 k-ft	456.68 k-ft	+1.20D+1.60Lr+0.50L+0.80W
0.005969	Flexure - Right of Col #2 - Top	-2.694 k-ft	451.35 k-ft	+0.90D+1.60W+1.60H
0.01026	Flexure - Right of Col #2 - Bottom	4.685 k-ft	456.68 k-ft	+1.20D+1.60Lr+0.50L

Soil Pressures tab

For each service load combination, this tab presents the total vertical load, the resultant eccentricity, the soil pressures on the left and the right ends of the footing, the allowable soil pressure, and the ratio of the actual to the allowable soil pressure.

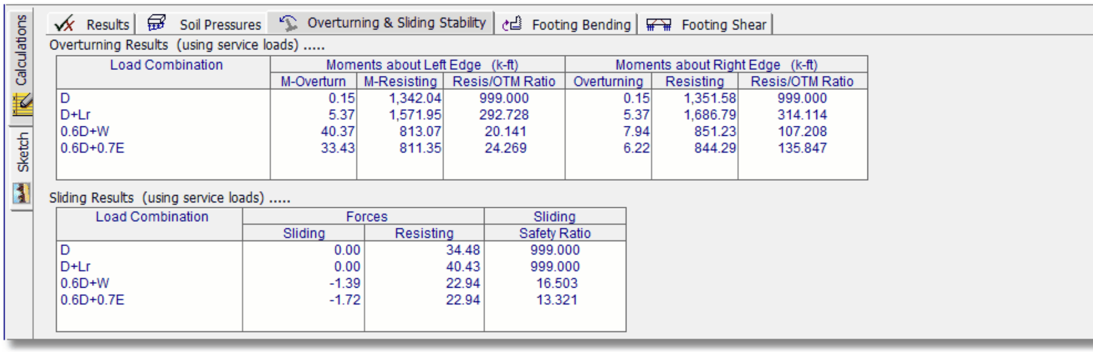


Rotation Axis	Bearing	Resultant	Soil Pressures		Allowable	Stress Ratio
Load Combination	Load	Eccentricity	Left, -X	Right, +X	Soil Pressure	Actual / Allow
+D	96.190	-0.050	0.463	0.453	3.250	0.142
+D+Lr+H	116.000	-0.495	0.611	0.494	3.250	0.188
+D+0.750Lr+0.750L+H	111.048	-0.399	0.574	0.484	3.250	0.177
+D+W+H	96.190	-0.387	0.496	0.420	3.250	0.153
+D+0.70E+H	96.190	-0.333	0.491	0.426	3.250	0.151
+D+0.750Lr+0.750L+0.750W+	111.048	-0.618	0.599	0.459	3.250	0.184
+D+0.750L+0.750S+0.750W+	96.190	-0.302	0.488	0.428	3.250	0.150
+D+0.750Lr+0.750L+0.5250E	111.048	-0.582	0.595	0.463	3.250	0.183
+D+0.750L+0.750S+0.5250E+	96.190	-0.262	0.484	0.432	3.250	0.149
+0.60D+W+H	57.714	-0.611	0.311	0.239	3.250	0.096
+0.60D+0.70E+H	57.714	-0.521	0.305	0.244	3.250	0.094

Overturning & Sliding Stability tab

For each service load combination, this tab presents the overturning moment, the resisting moment and the ratio of the resisting to overturning moment about the left and right edges of the footing. It also reports the sliding force, the resisting force, and the ratio of the resisting to sliding force.

Note that the program is set up to look for overturning and resisting forces individually. For example, take the situation where the footing is subjected to equal and opposite shears at a given elevation. Common sense dictates that these forces cancel each other, and the footing experiences no net applied overturning moment from them. But the program treats one of the two equal and opposite forces as an overturning force, and the other as a resisting force. So for these two forces, there IS a net overturning moment reported, but the resisting moment ALSO considers the effect of the opposing load, so the accounting used to determine the overturning ratio is proper.



Overturning Results (using service loads) .....

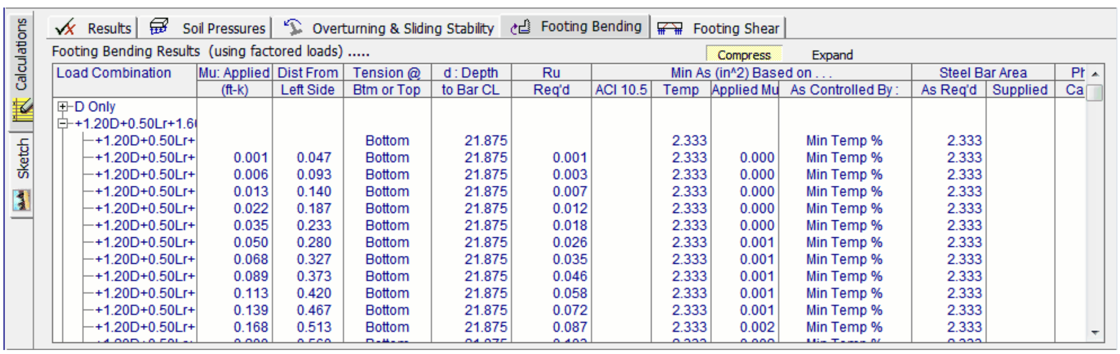
Load Combination	Moments about Left Edge (k-ft)			Moments about Right Edge (k-ft)		
	M-Overturn	M-Resisting	Resis/OTM Ratio	Overturning	Resisting	Resis/OTM Ratio
D	0.15	1,342.04	999.000	0.15	1,351.58	999.000
D+Lr	5.37	1,571.95	292.728	5.37	1,686.79	314.114
0.6D+W	40.37	813.07	20.141	7.94	851.23	107.208
0.6D+0.7E	33.43	811.35	24.269	6.22	844.29	135.847

Sliding Results (using service loads) .....

Load Combination	Forces		Sliding
	Sliding	Resisting	Safety Ratio
D	0.00	34.48	999.000
D+Lr	0.00	40.43	999.000
0.6D+W	-1.39	22.94	16.503
0.6D+0.7E	-1.72	22.94	13.321

Footing Bending tab

This tab reports the results of the flexural design on a load combination by load combination basis, at small increments along the length of the footing.

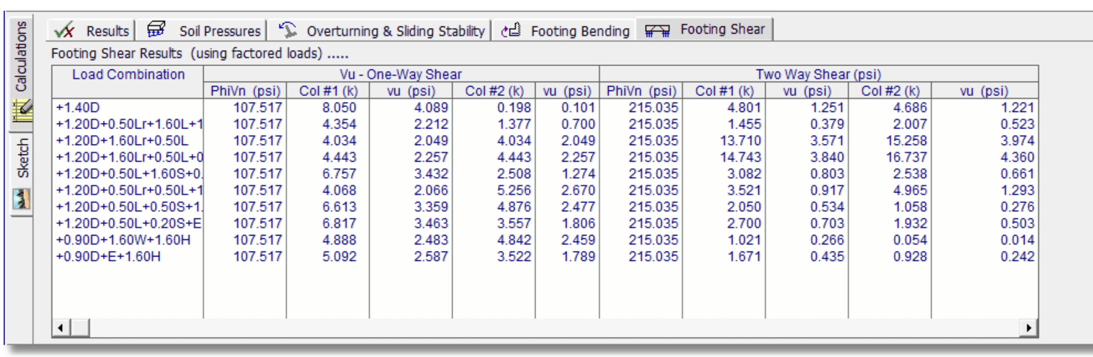


Footing Bending Results (using factored loads) .....

Load Combination	Mu: Applied	Dist From	Tension @	d : Depth	Ru	Min As (in^2) Based on . . .				Steel Bar Area		Ph
	(ft-k)	Left Side	Btm or Top	to Bar CL	Req'd	ACI 10.5	Temp	Applied Mu	As Controlled By :	As Req'd	Supplied	Ca
⊞-D Only												
⊟-+1.20D+0.50Lr+1.6												
+1.20D+0.50Lr+			Bottom	21.875			2.333		Min Temp %	2.333		
+1.20D+0.50Lr+	0.001	0.047	Bottom	21.875	0.001		2.333	0.000	Min Temp %	2.333		
+1.20D+0.50Lr+	0.006	0.093	Bottom	21.875	0.003		2.333	0.000	Min Temp %	2.333		
+1.20D+0.50Lr+	0.013	0.140	Bottom	21.875	0.007		2.333	0.000	Min Temp %	2.333		
+1.20D+0.50Lr+	0.022	0.187	Bottom	21.875	0.012		2.333	0.000	Min Temp %	2.333		
+1.20D+0.50Lr+	0.035	0.233	Bottom	21.875	0.018		2.333	0.000	Min Temp %	2.333		
+1.20D+0.50Lr+	0.050	0.280	Bottom	21.875	0.026		2.333	0.001	Min Temp %	2.333		
+1.20D+0.50Lr+	0.068	0.327	Bottom	21.875	0.035		2.333	0.001	Min Temp %	2.333		
+1.20D+0.50Lr+	0.089	0.373	Bottom	21.875	0.046		2.333	0.001	Min Temp %	2.333		
+1.20D+0.50Lr+	0.113	0.420	Bottom	21.875	0.058		2.333	0.001	Min Temp %	2.333		
+1.20D+0.50Lr+	0.139	0.467	Bottom	21.875	0.072		2.333	0.001	Min Temp %	2.333		
+1.20D+0.50Lr+	0.168	0.513	Bottom	21.875	0.087		2.333	0.002	Min Temp %	2.333		

Footing Shear tab

This tab reports the results of the one-way and two-way shear design on a load combination by load combination basis.

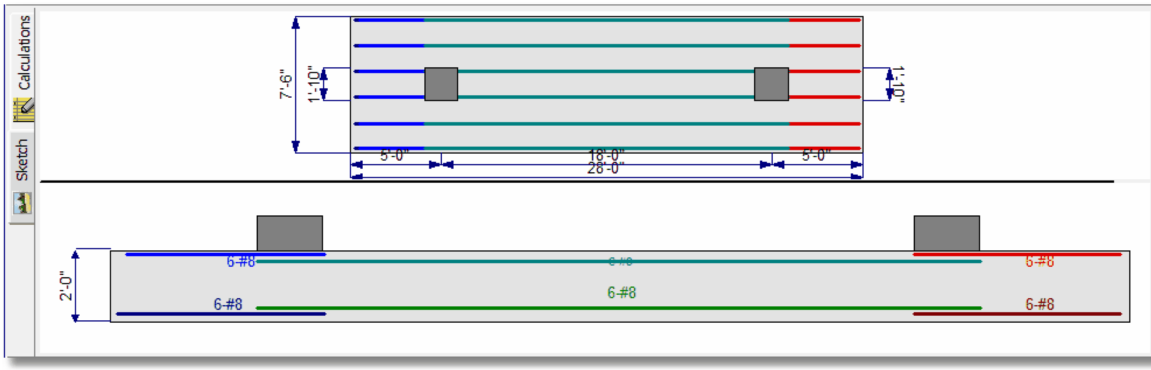


Footing Shear Results (using factored loads) .....

Load Combination	Vu - One-Way Shear					Two Way Shear (psi)				
	PhiVn (psi)	Col #1 (k)	vu (psi)	Col #2 (k)	vu (psi)	PhiVn (psi)	Col #1 (k)	vu (psi)	Col #2 (k)	vu (psi)
+1.40D	107.517	8.050	4.089	0.198	0.101	215.035	4.801	1.251	4.686	1.221
+1.20D+0.50Lr+1.60L+1	107.517	4.354	2.212	1.377	0.700	215.035	1.455	0.379	2.007	0.523
+1.20D+1.60Lr+0.50L	107.517	4.034	2.049	4.034	2.049	215.035	13.710	3.571	15.258	3.974
+1.20D+1.60Lr+0.50L+0	107.517	4.443	2.257	4.443	2.257	215.035	14.743	3.840	16.737	4.360
+1.20D+0.50L+1.60S+0.	107.517	6.757	3.432	2.508	1.274	215.035	3.082	0.803	2.538	0.661
+1.20D+0.50Lr+0.50L+1	107.517	4.068	2.066	5.256	2.670	215.035	3.521	0.917	4.965	1.293
+1.20D+0.50L+0.50S+1.	107.517	6.613	3.359	4.876	2.477	215.035	2.050	0.534	1.058	0.276
+1.20D+0.50L+0.20S+E	107.517	6.817	3.463	3.557	1.806	215.035	2.700	0.703	1.932	0.503
+0.90D+1.60W+1.60H	107.517	4.888	2.483	4.842	2.459	215.035	1.021	0.266	0.054	0.014
+0.90D+E+1.60H	107.517	5.092	2.587	3.522	1.789	215.035	1.671	0.435	0.928	0.242

Sketch tab

This tab provides a plan view and a longitudinal section view of the footing with rebar callouts and overall dimensions.

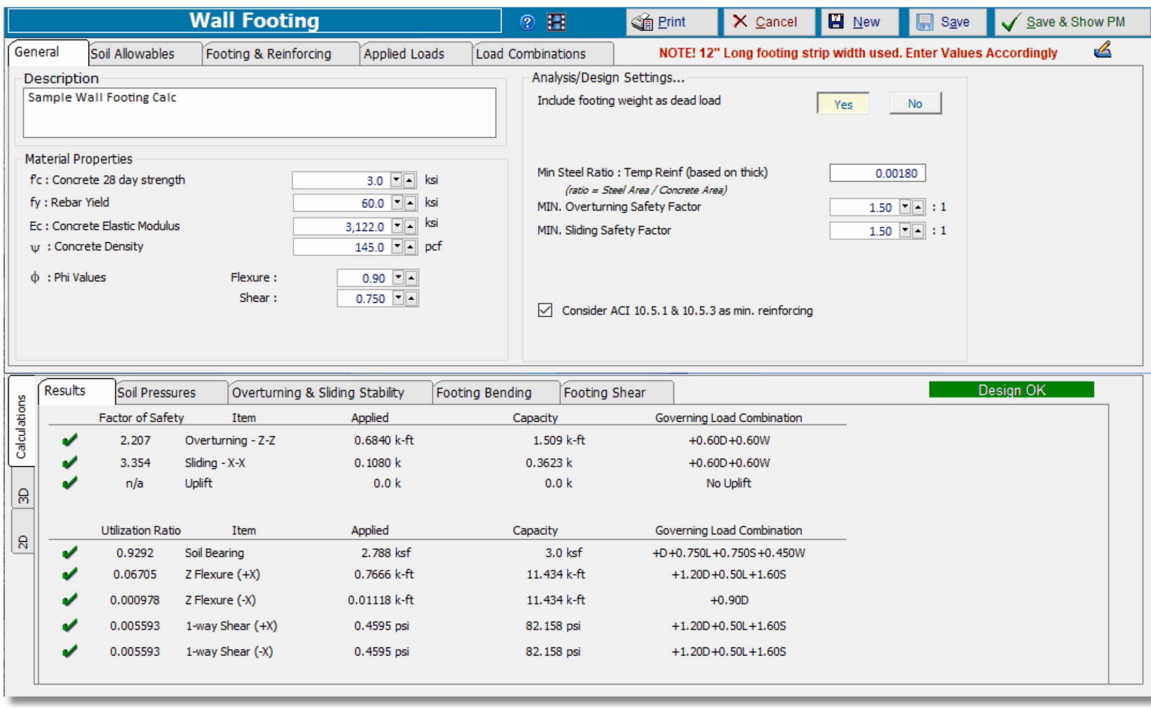


10.7.3 Wall Footing

Need more? Ask Us a Question

This module provides analysis of a unit strip of a continuous wall footing with applied axial, moment and shear loads. Overburden loads can also be specified, and will apply to the surface area of the footing (except the area covered by the wall). The module also provides automatic calculation of allowable soil bearing pressure increases based on footing width and/or depth below surface.

The module checks service load soil pressure, overturning stability, sliding stability, uplift stability, footing flexure and one-way footing shear.



General

This tab collects material property values, strength reduction factors, and other settings that influence the design.

f'c

28-day compressive strength of the concrete.

fy

Yield point stress of reinforcing.

Ec

Modulus of elasticity of concrete.

Concrete Density

The density of the concrete is used to calculate the self weight of the footing when the option is selected.

Phi Values

Enter the capacity reduction values to be applied to Vn and Mn.

Include footing weight as dead load

Click [**Yes**] to have the module calculate the weight of the footing and apply it as a downward load. The footing self weight will be multiplied by the dead load factor in each load combination.

Min Steel Ratio - Temperature/Shrinkage Reinf.

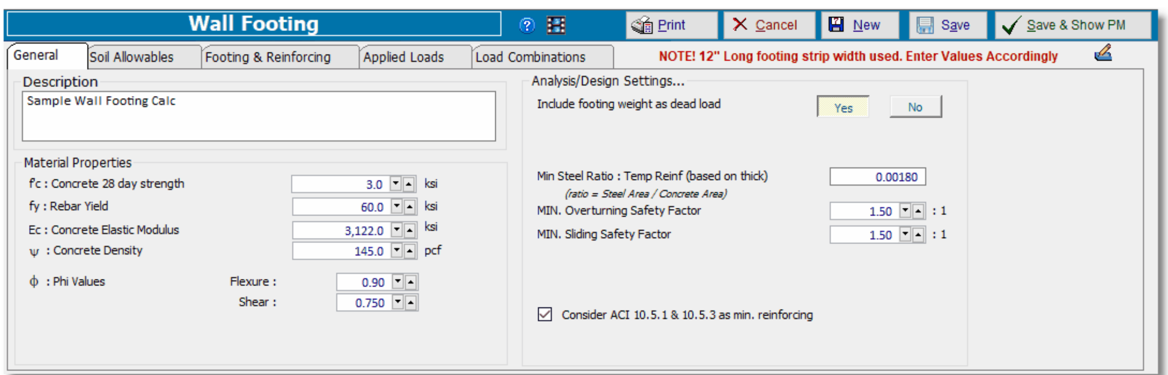
Enter the minimum ratio for temperature/shrinkage steel, calculated using the footing thickness. This will trigger a warning message if the section is under-reinforced.

Minimum Overturning Safety Factor

Enter the minimum allowable ratio of resisting moment to overturning moment. If the actual ratio is less than the specified minimum ratio, it will trigger a message that overturning stability is not satisfied.

Minimum Sliding Safety Factor

Enter the minimum allowable ratio of resisting force to sliding force. If the actual ratio is less than the specified minimum ratio, it will trigger a message that sliding stability is not satisfied.



Soil Allowables

Allowable Soil Bearing Pressure

Enter the allowable soil bearing pressure. This is a service load resistance and will be compared to calculated service load soil pressures (loads not factored as in strength design).

Increase Bearing by Footing Weight

Click [**Yes**] to tell the module to calculate the weight of one square foot (plan view) of footing and add it to the allowable soil bearing value. This has the effect of not penalizing the soil for the self weight of the footing, and is useful for situations where the geotechnical engineering report provides allowable net bearing pressures.

Soil Passive Sliding Resistance

Enter the value of passive soil pressure resistance to sliding. This value will be used to determine a component of sliding resistance that is generated by the passive pressure of the soil. The sliding resistance due to passive pressure is then added to the sliding resistance due to friction to determine the total resistance to sliding for each load combination.

Coefficient of Soil/Concrete Friction

Enter the coefficient of friction between soil and footing to use in sliding resistance calculations.

Soil Bearing Increase

This section allows you to specify some dimensions that, when exceeded, will automatically increase the allowable soil bearing pressure.

Footing base depth below soil surface: The distance from the bottom of the footing to the top of the soil. This value is used to determine allowable soil bearing pressure increases and soil passive sliding resistance, but it is NOT used in any other calculations in this module.

Increases based on footing depth: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing depth below some reference depth. Collects the following parameters:

Allowable pressure increase per foot: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of depth below some reference depth.

When base of footing is below: Specifies the required depth in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing depth.

Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing base is 6'-0" below soil surface. The Geotechnical report specifies that a 0.15 ksf increase in bearing pressure is allowed for each foot of depth when the base is deeper than 4' below top of soil. Since you've indicated that the footing is 6' below the soil surface, the module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (6' - 4') * 0.15 ksf = 3.30 ksf.

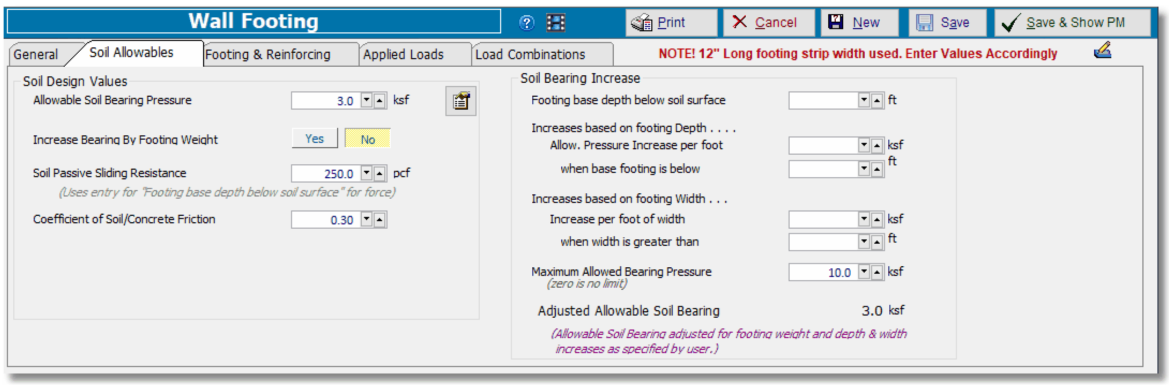
Increases based on footing width: Provides a method to automatically apply increases to the basic allowable soil bearing pressure based on footing width greater than some reference dimension. Collects the following parameters:

Allowable pressure increase per foot: Specifies the amount that the basic allowable soil bearing pressure can be increased for each foot of width greater than some reference dimension.

When maximum length or width is greater than: Specifies the required dimension in order to start realizing incremental increases in the allowable soil bearing pressure on the basis of footing width.

Example: Assume the following: Basic Allowable Soil Bearing Pressure = 3 ksf. Footing measures 6'-0" wide. The Geotechnical report specifies that a 0.15 ksf increase in soil bearing pressure is allowed for each foot when the width of the footing is greater than 4'-0". The module will automatically calculate the adjusted allowable soil bearing pressure to be 3 ksf + (6' - 4') * 0.15 ksf = 3.3 ksf.

Note: Increases based on footing depth and width are cumulative.



Footing Size & Reinforcing

Dimensions tab

Footing Width: Define the width of the footing.

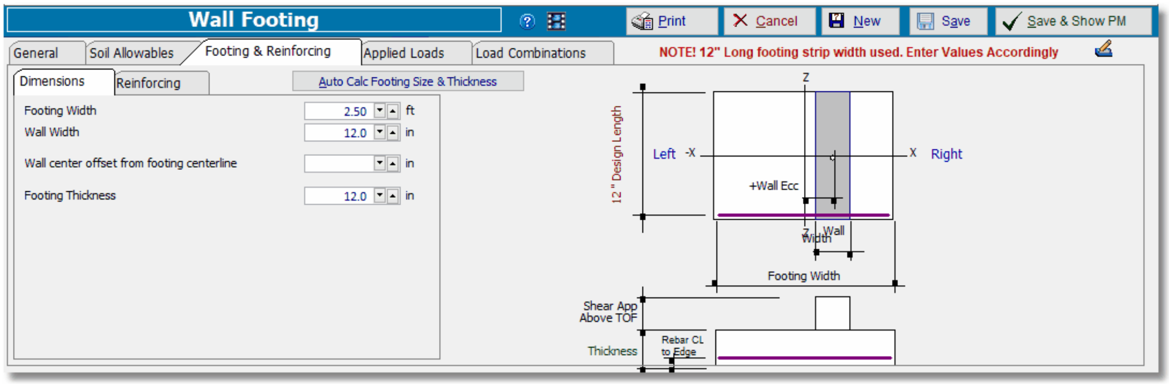
Wall Width: Define the width of the supported wall.

Wall center offset from footing centerline: Define the dimension between the centerline of the wall and the centerline of the footing. Positive offsets shift the wall toward the right edge of the footing.

Footing Thickness: Define the thickness of the footing.

Auto Calculate Footing Size & Thickness: Provides an automated routine to increase footing dimensions until soil pressures are satisfied and one-way shear is acceptable.

Note: Any applied overburden loads will be omitted from the area occupied by the wall.

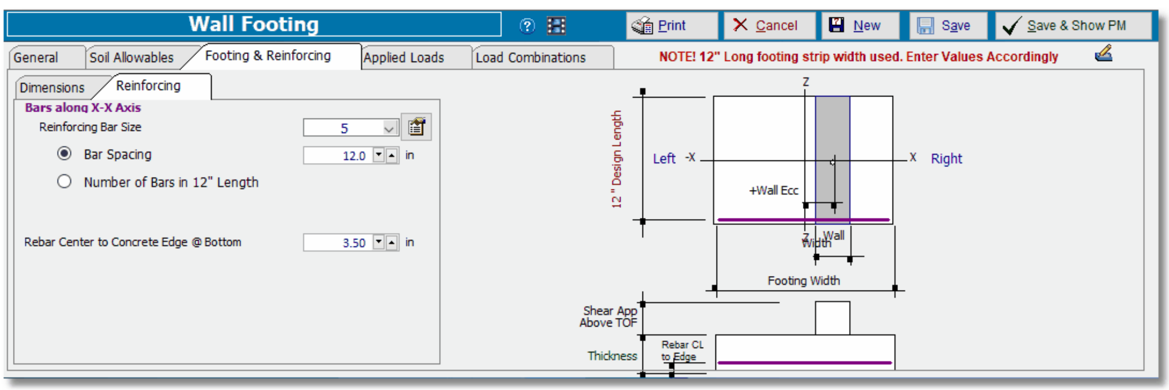


Reinforcing tab

Reinforcing Bar Size: Indicate the rebar size to consider for the bars that run parallel to the footing width.

Rebar Spacing: Provides an option to either specify an explicit value for the rebar spacing, or to specify the number of bars in a 12-inch length.

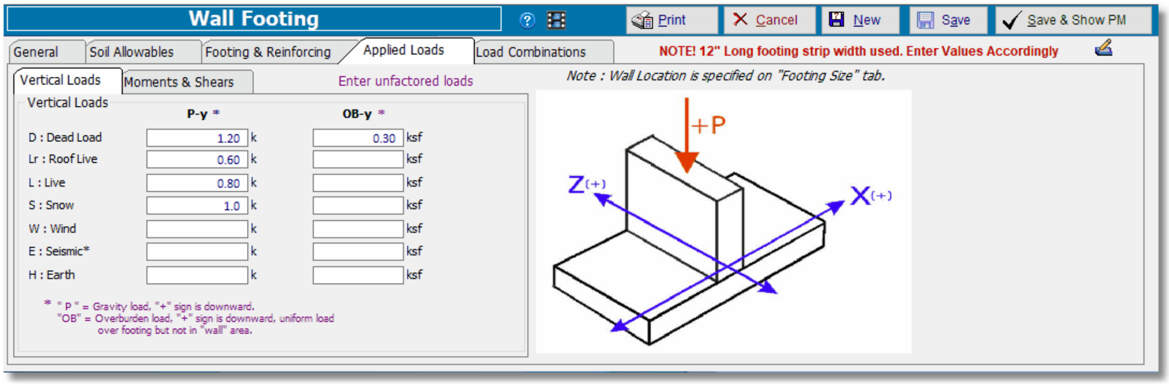
Rebar Center to Concrete Edge @ Bottom: Specify the clear cover plus 1/2 the diameter of the rebar.



Applied Loads

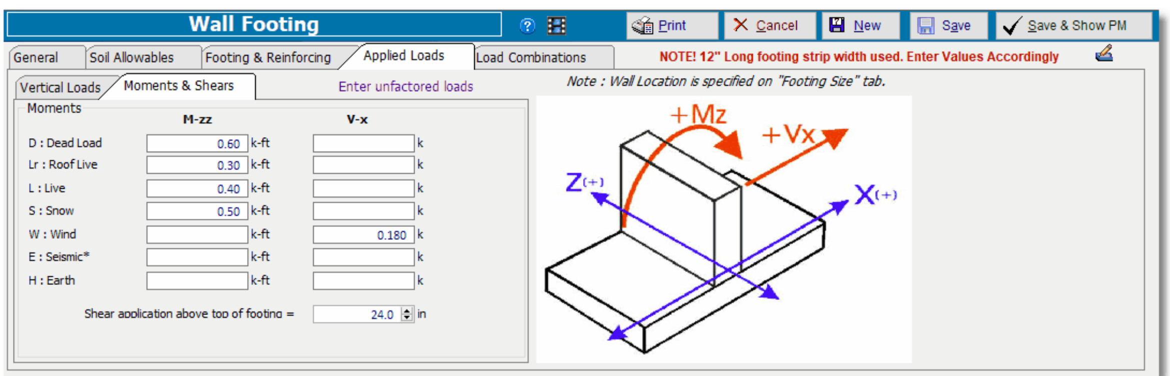
Vertical Loads tab

Provides input fields for vertical loads and overburden pressures. Vertical loads are specified in units of kips per foot, and they are considered to act at the center of the width of the wall. Overburden loads are specified in units of kips per square foot, and they are considered to act on the top surface area of the footing, excluding the area occupied by the wall.



Moments & Shears tab

Provides input fields for moments and shears. Moments are specified in units of foot kips per foot. Shears are specified in units of kips per foot, and they are considered to act at the height specified in the field named **Shear application above top of footing**. Shears will produce a moment equal to the shear force times the distance from the bottom of the footing to the location of application of the shear force.

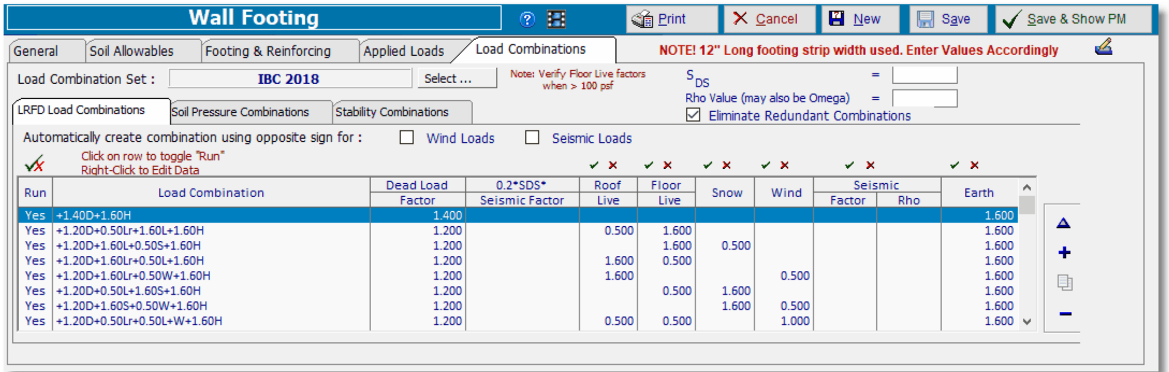


Load Combinations

The Load Combinations tab is used to specify the load combinations to be used in the design. The LRFD Load Combinations tab controls the combinations that are used for reinforced concrete design checks. The Soil Pressure Combinations tab controls the combinations that are used for evaluating soil bearing pressure. A Soil Increase factor can be applied on a load combination by load combination basis, as permitted by the geotechnical engineering report. The Stability Combinations tab controls the load combinations that are used to perform the serviceability checks for Overturning, Sliding, and Uplift.

These tabs allow the user to select from load combination sets that are supplied with the program or to select from custom load combination sets that have been created and saved on the user's machine. It is also possible to unlock the selected load combination set and make edits to the factors directly in this view.

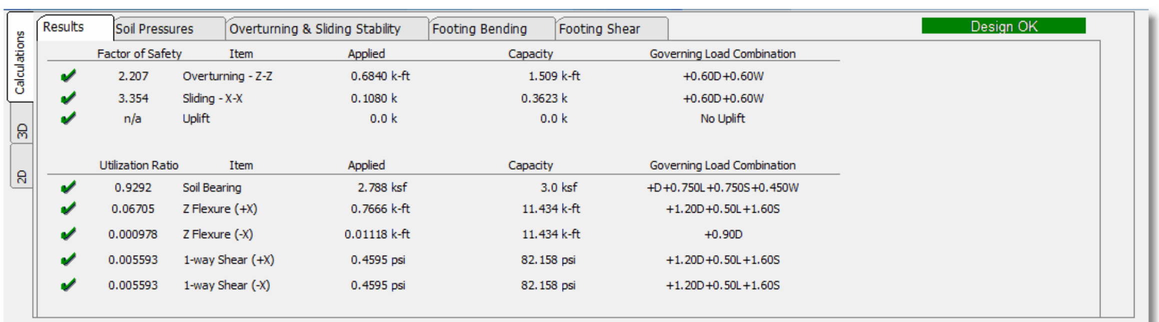
The user has control over which combinations are run and which are ignored. Finally, these tabs allow the user to specify whether the program should consider the algebraic sign of the specified load factors on wind loads and seismic loads to be reversible or not. This can be a convenient way to ensure that these loads are investigated as acting in both positive and negative directions, if that is the design intent. Note, however, that if selected, the algebraic sign reversal will be applied to ALL wind loads and/or ALL seismic loads including horizontally AND vertically applied loads.



Calculations

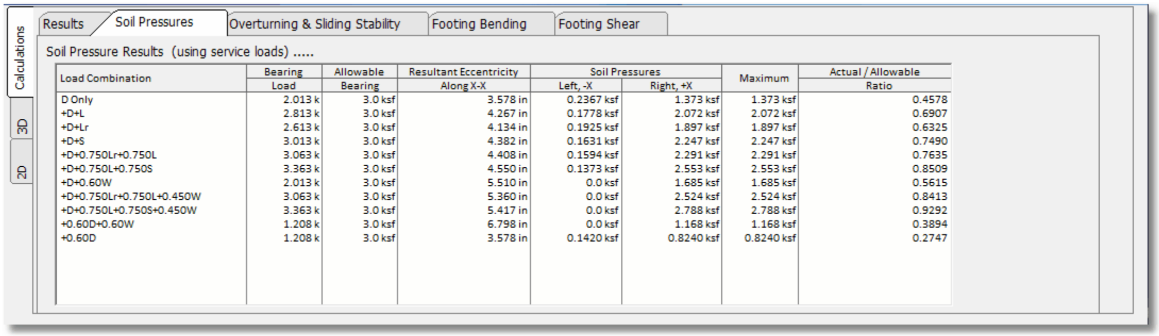
Results tab

This tab summarizes the controlling values (highest utilization ratio) for each design consideration, from all of the load combinations that have been run. For the controlling load combination, it presents the Applied load, the Capacity or available resisting load, the ratio of the Applied to the Capacity, and the governing load combination that produces this controlling ratio.



Soil Pressures tab

For each service load combination, this tab presents the total vertical load, the resultant eccentricity, the soil pressures on the left and the right ends of the footing, the allowable soil pressure, and the ratio of the actual to the allowable soil pressure.

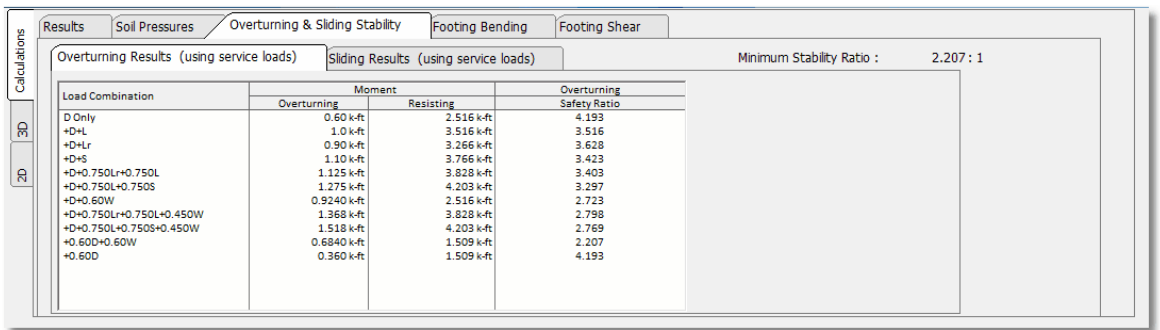


Soil Pressure Results (using service loads) .....

Load Combination	Bearing Load	Allowable Bearing	Resultant Eccentricity Along X-X	Soil Pressures Left, -X	Soil Pressures Right, +X	Maximum	Actual / Allowable Ratio
D Only	2.013 k	3.0 ksf	3.578 in	0.2367 ksf	1.373 ksf	1.373 ksf	0.4578
+D+L	2.813 k	3.0 ksf	4.267 in	0.1778 ksf	2.072 ksf	2.072 ksf	0.6907
+D+Lr	2.613 k	3.0 ksf	4.134 in	0.1925 ksf	1.897 ksf	1.897 ksf	0.6325
+D+S	3.013 k	3.0 ksf	4.382 in	0.1631 ksf	2.247 ksf	2.247 ksf	0.7490
+D+0.750Lr+0.750L	3.063 k	3.0 ksf	4.408 in	0.1594 ksf	2.291 ksf	2.291 ksf	0.7635
+D+0.750L+0.750S	3.363 k	3.0 ksf	4.550 in	0.1373 ksf	2.553 ksf	2.553 ksf	0.8509
+D+0.60W	2.013 k	3.0 ksf	5.510 in	0.0 ksf	1.685 ksf	1.685 ksf	0.5615
+D+0.750Lr+0.750L+0.450W	3.063 k	3.0 ksf	5.360 in	0.0 ksf	2.524 ksf	2.524 ksf	0.8413
+D+0.750L+0.750S+0.450W	3.363 k	3.0 ksf	5.417 in	0.0 ksf	2.788 ksf	2.788 ksf	0.9292
+0.60D+0.60W	1.208 k	3.0 ksf	6.798 in	0.0 ksf	1.168 ksf	1.168 ksf	0.3894
+0.60D	1.208 k	3.0 ksf	3.578 in	0.1420 ksf	0.8240 ksf	0.8240 ksf	0.2747

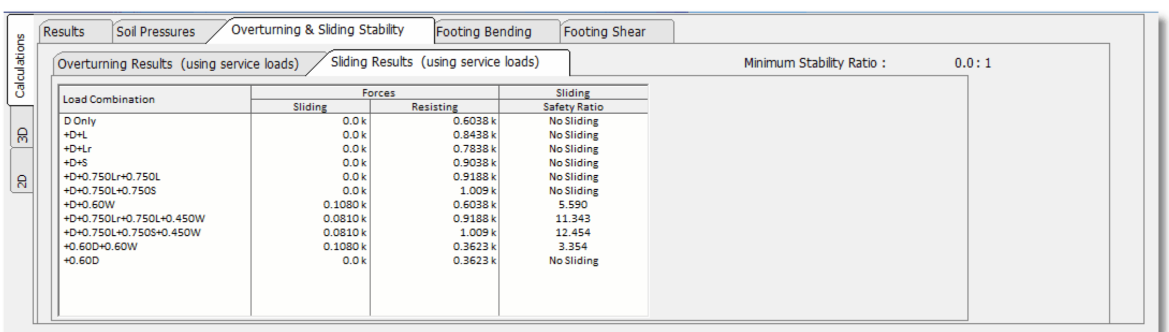
Overturning & Sliding Stability tab

For each service load combination, this tab presents the overturning moment, the resisting moment and the ratio of the resisting to overturning moment about the left and right edges of the footing.



Load Combination	Moment Overturning	Moment Resisting	Overturning Safety Ratio
D Only	0.60 k-ft	2.516 k-ft	4.193
+D+L	1.0 k-ft	3.516 k-ft	3.516
+D+Lr	0.90 k-ft	3.266 k-ft	3.628
+D+S	1.10 k-ft	3.766 k-ft	3.423
+D+0.750Lr+0.750L	1.125 k-ft	3.828 k-ft	3.403
+D+0.750L+0.750S	1.275 k-ft	4.203 k-ft	3.297
+D+0.60W	0.9240 k-ft	2.516 k-ft	2.723
+D+0.750Lr+0.750L+0.450W	1.368 k-ft	3.828 k-ft	2.798
+D+0.750L+0.750S+0.450W	1.518 k-ft	4.203 k-ft	2.769
+0.60D+0.60W	0.6840 k-ft	1.509 k-ft	2.207
+0.60D	0.360 k-ft	1.509 k-ft	4.193

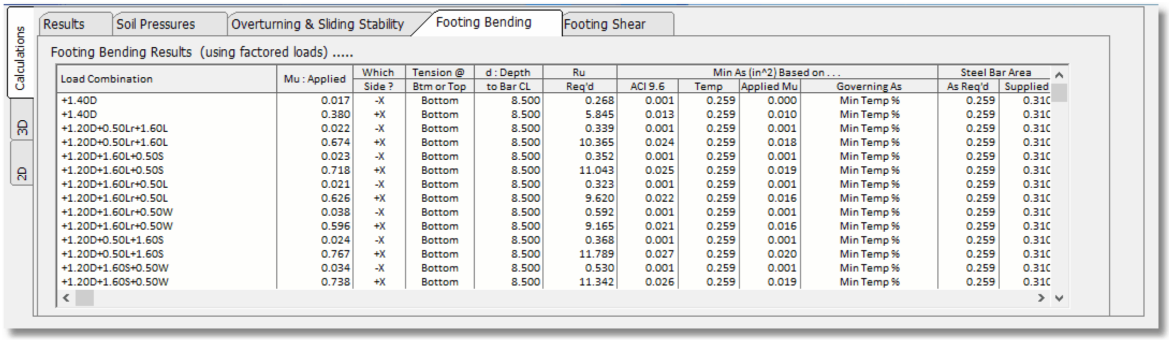
It also reports the sliding force, the resisting force, and the ratio of the resisting to sliding force.



Load Combination	Forces Sliding	Forces Resisting	Sliding Safety Ratio
D Only	0.0 k	0.6038 k	No Sliding
+D+L	0.0 k	0.8438 k	No Sliding
+D+Lr	0.0 k	0.7838 k	No Sliding
+D+S	0.0 k	0.9038 k	No Sliding
+D+0.750Lr+0.750L	0.0 k	0.9188 k	No Sliding
+D+0.750L+0.750S	0.0 k	1.009 k	No Sliding
+D+0.60W	0.1080 k	0.6038 k	5.590
+D+0.750Lr+0.750L+0.450W	0.0810 k	0.9188 k	11.343
+D+0.750L+0.750S+0.450W	0.0810 k	1.009 k	12.454
+0.60D+0.60W	0.1080 k	0.3623 k	3.354
+0.60D	0.0 k	0.3623 k	No Sliding

Footing Bending tab

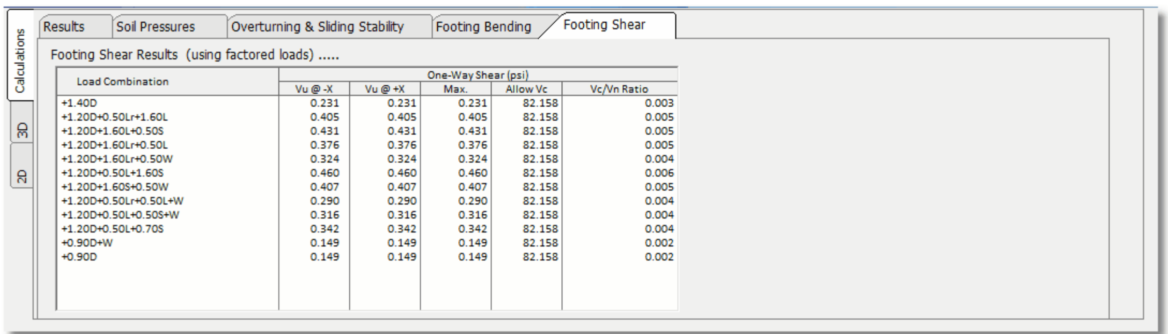
This tab reports the results of the flexural design on a load combination by load combination basis.



Load Combination	Mu : Applied	Which Side ?	Tension @ Btm or Top	d : Depth to Bar CL	Ru Req'd	Min As (in^2) Based on . . . ACI 9.6	Temp	Applied Mu	Governing As	Steel Bar Area As Req'd	Supplied
+1.40D	0.017	-X	Bottom	8.500	0.268	0.001	0.259	0.000	Min Temp %	0.259	0.31C
+1.40D	0.380	+X	Bottom	8.500	5.845	0.013	0.259	0.010	Min Temp %	0.259	0.31C
+1.20D+0.50Lr+1.60L	0.022	-X	Bottom	8.500	0.339	0.001	0.259	0.001	Min Temp %	0.259	0.31C
+1.20D+0.50Lr+1.60L	0.674	+X	Bottom	8.500	10.365	0.024	0.259	0.018	Min Temp %	0.259	0.31C
+1.20D+1.60L+0.50S	0.023	-X	Bottom	8.500	0.352	0.001	0.259	0.001	Min Temp %	0.259	0.31C
+1.20D+1.60L+0.50S	0.718	+X	Bottom	8.500	11.043	0.025	0.259	0.019	Min Temp %	0.259	0.31C
+1.20D+1.60Lr+0.50L	0.021	-X	Bottom	8.500	0.323	0.001	0.259	0.001	Min Temp %	0.259	0.31C
+1.20D+1.60Lr+0.50L	0.626	+X	Bottom	8.500	9.620	0.022	0.259	0.016	Min Temp %	0.259	0.31C
+1.20D+1.60Lr+0.50W	0.038	-X	Bottom	8.500	0.592	0.001	0.259	0.001	Min Temp %	0.259	0.31C
+1.20D+1.60Lr+0.50W	0.596	+X	Bottom	8.500	9.165	0.021	0.259	0.016	Min Temp %	0.259	0.31C
+1.20D+0.50L+1.60S	0.024	-X	Bottom	8.500	0.368	0.001	0.259	0.001	Min Temp %	0.259	0.31C
+1.20D+0.50L+1.60S	0.767	+X	Bottom	8.500	11.789	0.027	0.259	0.020	Min Temp %	0.259	0.31C
+1.20D+1.60S+0.50W	0.034	-X	Bottom	8.500	0.530	0.001	0.259	0.001	Min Temp %	0.259	0.31C
+1.20D+1.60S+0.50W	0.738	+X	Bottom	8.500	11.342	0.026	0.259	0.019	Min Temp %	0.259	0.31C

Footing Shear tab

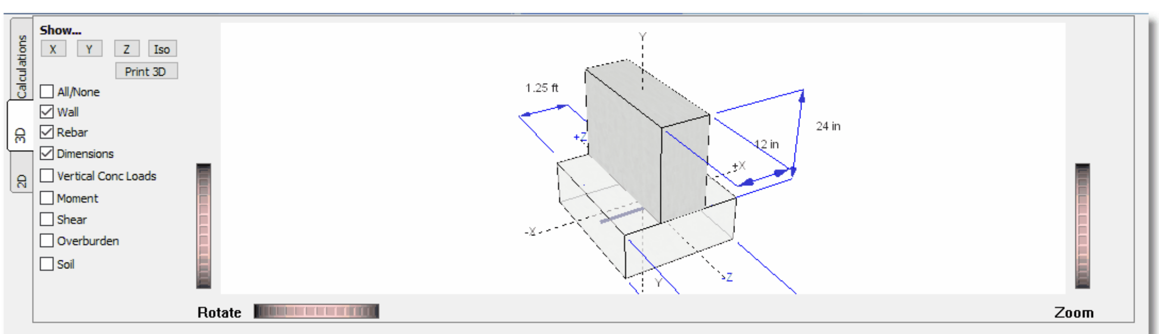
This tab reports the results of the shear design on a load combination by load combination basis.



Load Combination	One-Way Shear (psi) Vu @ -X	Vu @ +X	Max.	Allow Vc	Vc/Vn Ratio
+1.40D	0.231	0.231	0.231	82.158	0.003
+1.20D+0.50Lr+1.60L	0.405	0.405	0.405	82.158	0.005
+1.20D+1.60L+0.50S	0.431	0.431	0.431	82.158	0.005
+1.20D+1.60Lr+0.50L	0.376	0.376	0.376	82.158	0.005
+1.20D+1.60Lr+0.50W	0.324	0.324	0.324	82.158	0.004
+1.20D+0.50L+1.60S	0.460	0.460	0.460	82.158	0.006
+1.20D+1.60S+0.50W	0.407	0.407	0.407	82.158	0.005
+1.20D+0.50Lr+0.50L+W	0.290	0.290	0.290	82.158	0.004
+1.20D+0.50L+0.50S+W	0.316	0.316	0.316	82.158	0.004
+1.20D+0.50L+0.70S	0.342	0.342	0.342	82.158	0.004
+0.90D+W	0.149	0.149	0.149	82.158	0.002
+0.90D	0.149	0.149	0.149	82.158	0.002

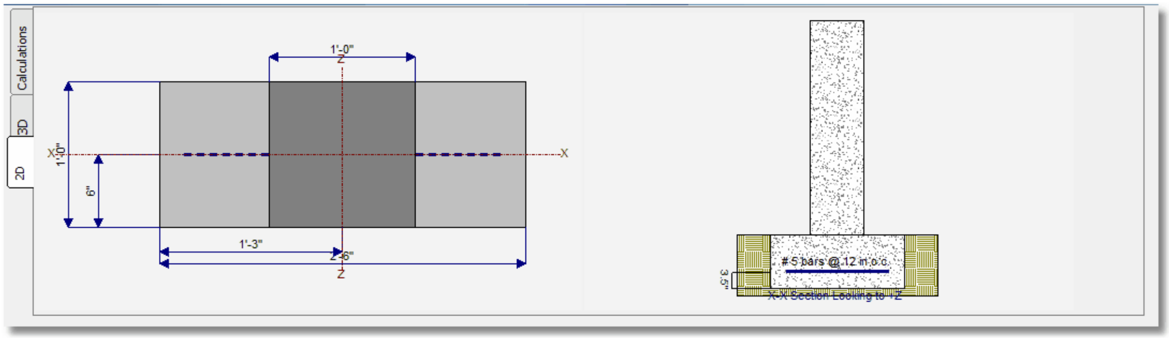
3D tab

This tab presents the 3D rendering of the footing:



2D tab

This tab presents plan and section views of the footing:


Calculations
3D
2D
1'-0"
1'-0"
6"
1'-3"
2'-6"
5 bars @ 12 in o.c.
3.5"
X-X Section Looking to +Z

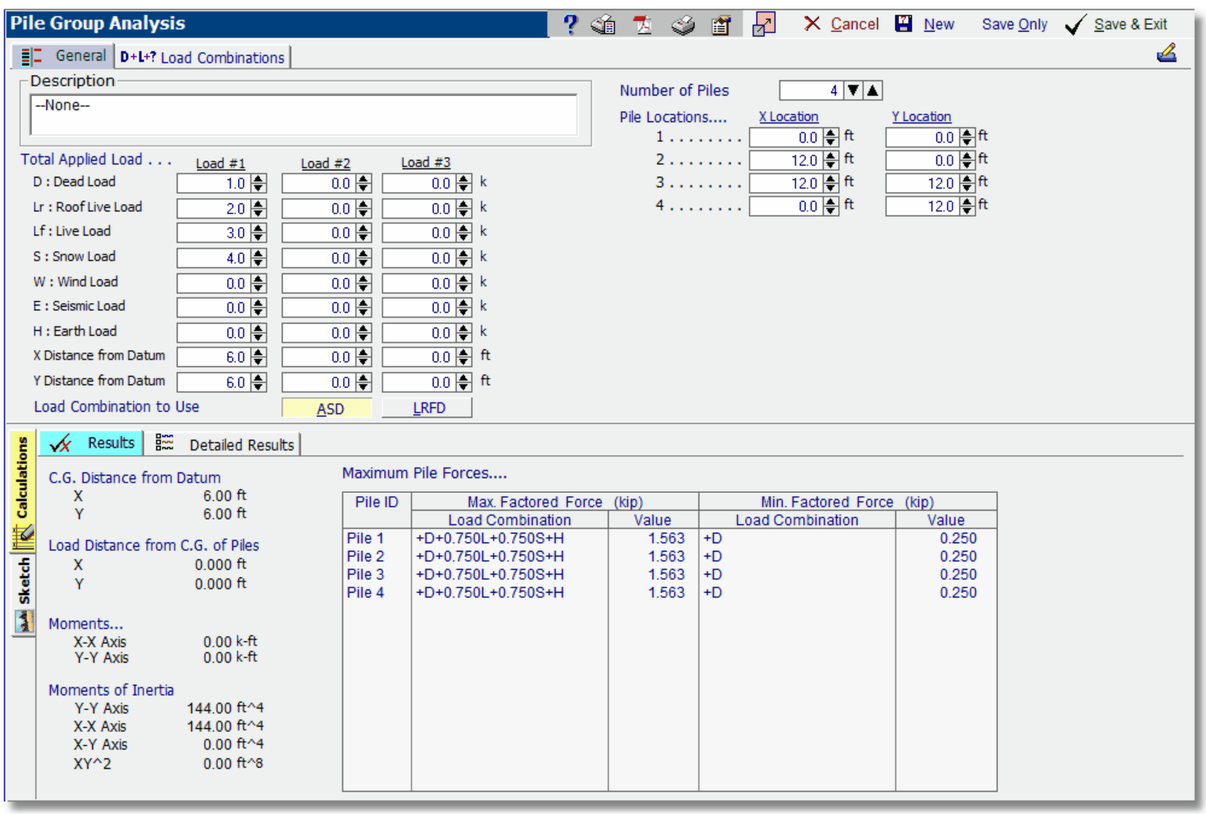
10.7.4 Pile Group

Need more? Ask Us a Question

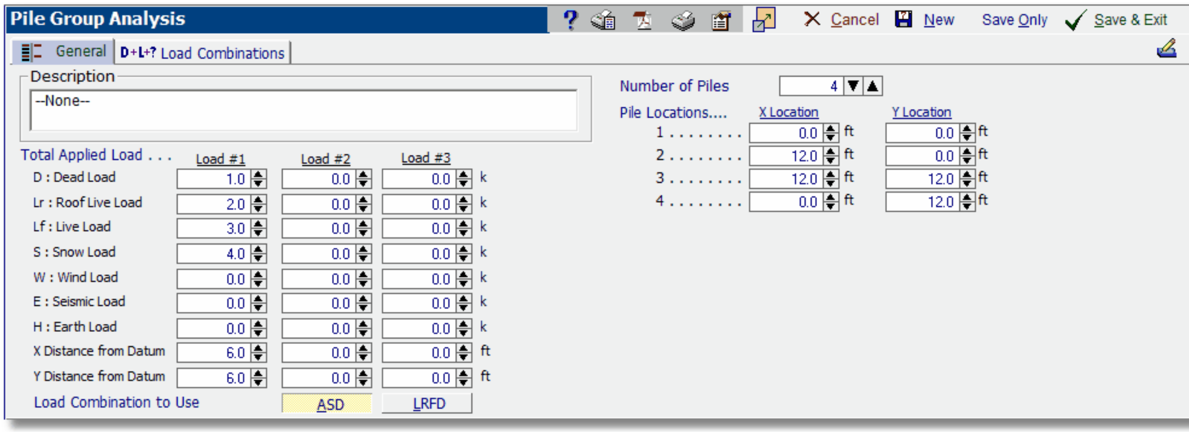
This module considers a concentrated load applied to a rigid pile cap and distributes it to a group of piles. Force distribution is performed assuming a rigid pile cap and that all piles have equal vertical load resistance.

Distribution of loads to each pile due to the effect of load eccentricity is determined using a skew bending analysis. This considers simultaneous action about both X and Y axes.

The module is also an efficient method for determining loads on a pile group in the as-driven arrangement.



General Tab



NOTE! Establish an X & Y Coordinate system prior to entering pile and load locations. Module requires a 2-dimensional pile group to be defined. It will not report results for a collinear group, i.e. a single line of piles.

Total Applied Axial Load

Enter the total vertical load to be distributed to the piles in the pile group using the coordinate system you have defined.

Note: Only vertical loads are allowed; no lateral shears.

X & Y Distance to Load

Enter the distance from the X & Y datum (0,0) to the location of the applied vertical load.

Number of Piles

This entry defines the number of piles in the group. As you change the number of piles, the number of data entry locations will match the specified number of piles.

Pile Locations: distance from Datum to the pile

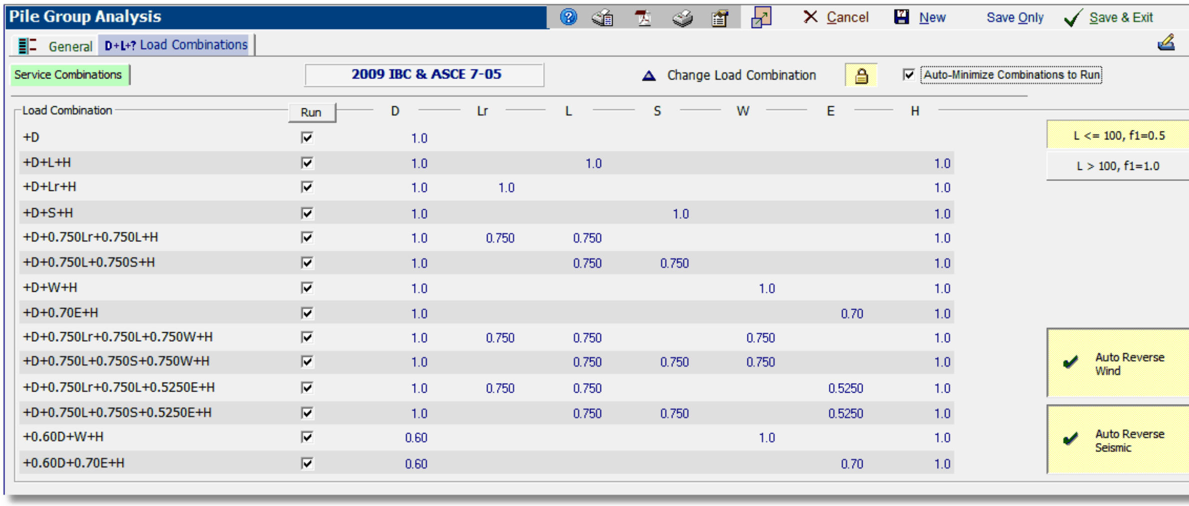
Enter the distance from the X & Y datum (0,0) to the center of each pile location.

Load Combination to use

This selection will switch the load combinations shown on the Load Combinations tab between Service and Factored design combinations.

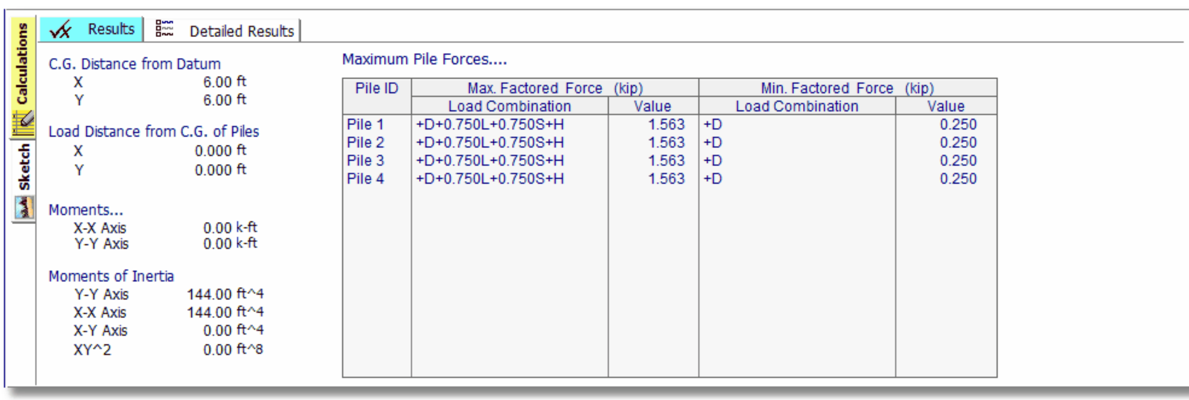
Load Combination Tab

This tab allows you to specify the load combinations to be considered.



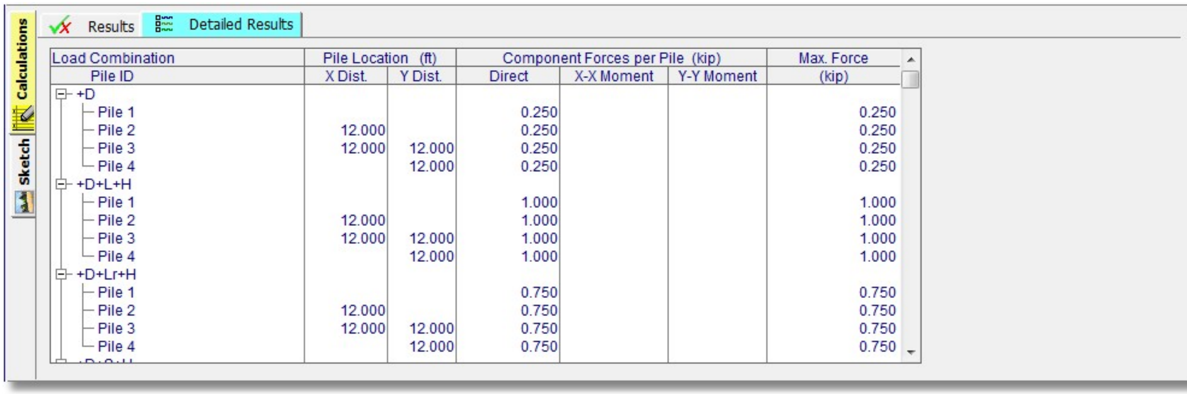
Results Tab

This tab summarizes the overall calculated values for the pile group and lists the maximum factored load for each pile and the load combination that produced the maximum load.



Detailed Results Tab

This tab gives the detailed calculations for each pile for each load combination. It indicates the Direct force as well as the component of axial load that is due to the net moment applied to the entire group. This latter effect will be observed for any pile that is located at a distance from the center of gravity of the pile group.

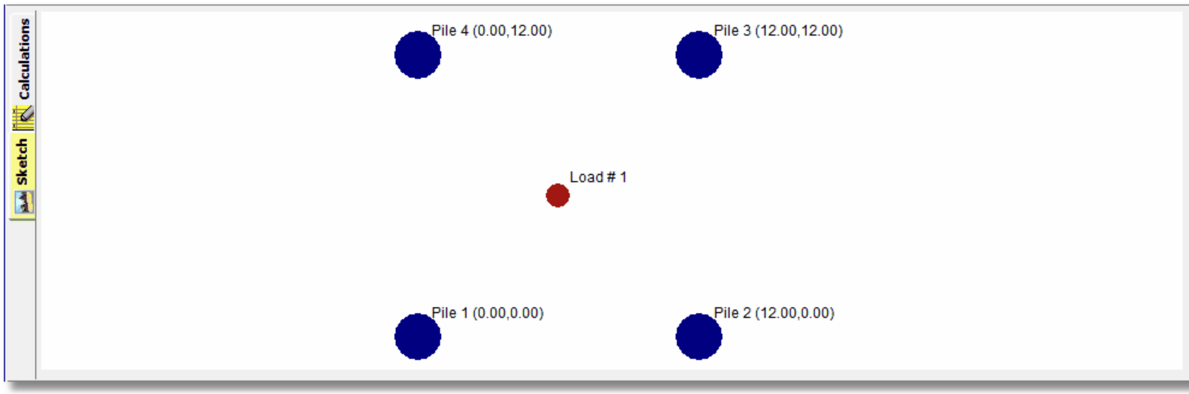


Calculations | Sketch

Results | Detailed Results

Load Combination / Pile ID	Pile Location (ft) X Dist.	Y Dist.	Component Forces per Pile (kip) Direct	X-X Moment	Y-Y Moment	Max. Force (kip)
+D						
Pile 1			0.250			0.250
Pile 2	12.000		0.250			0.250
Pile 3	12.000	12.000	0.250			0.250
Pile 4		12.000	0.250			0.250
+D+L+H						
Pile 1			1.000			1.000
Pile 2	12.000		1.000			1.000
Pile 3	12.000	12.000	1.000			1.000
Pile 4		12.000	1.000			1.000
+D+Lr+H						
Pile 1			0.750			0.750
Pile 2	12.000		0.750			0.750
Pile 3	12.000	12.000	0.750			0.750
Pile 4		12.000	0.750			0.750

Sketch Tab



Design Basis: Pile Group Analysis

10.7.5 Pole Footing Embedded in Soil

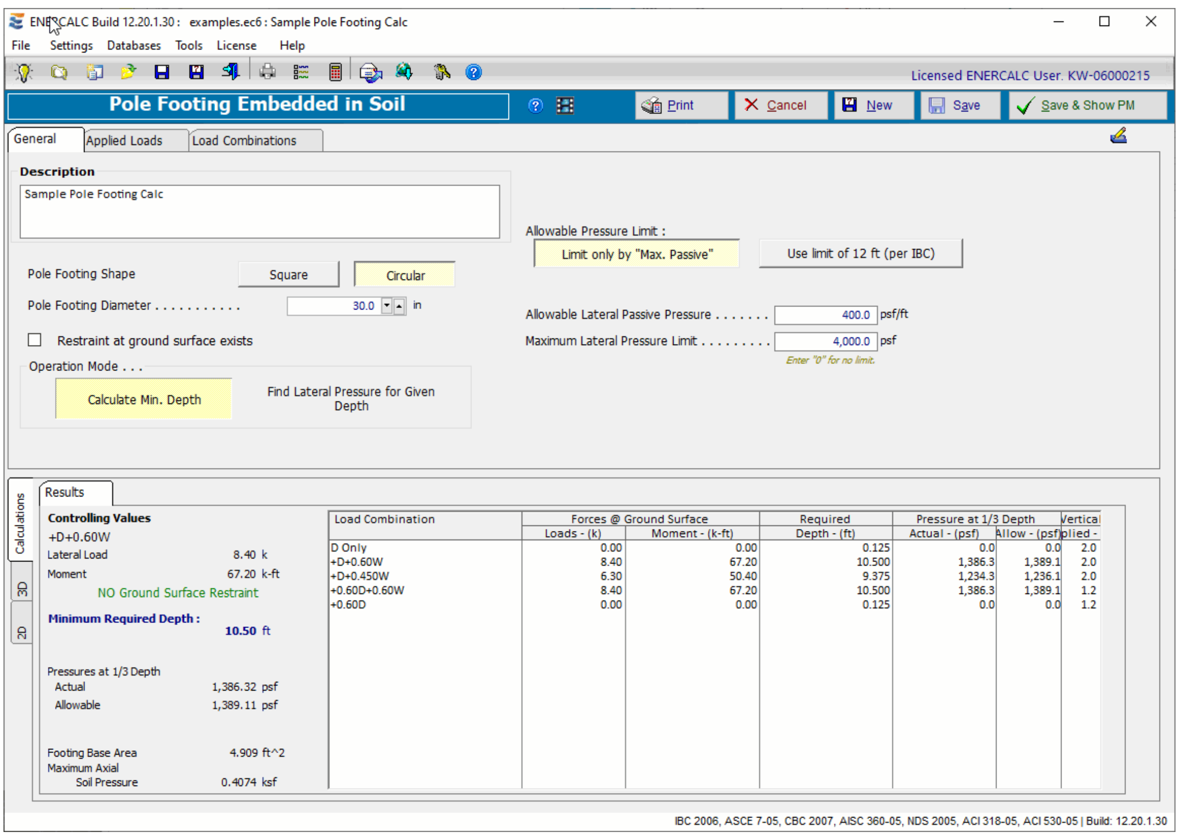
[Need more? Ask Us a Question]

This module determines actual soil pressures and required depths for pole footings primarily supporting lateral loads. Such footings are commonly called "flagpole footings".

Click here for a video: Pole Footing Embedded in Soil

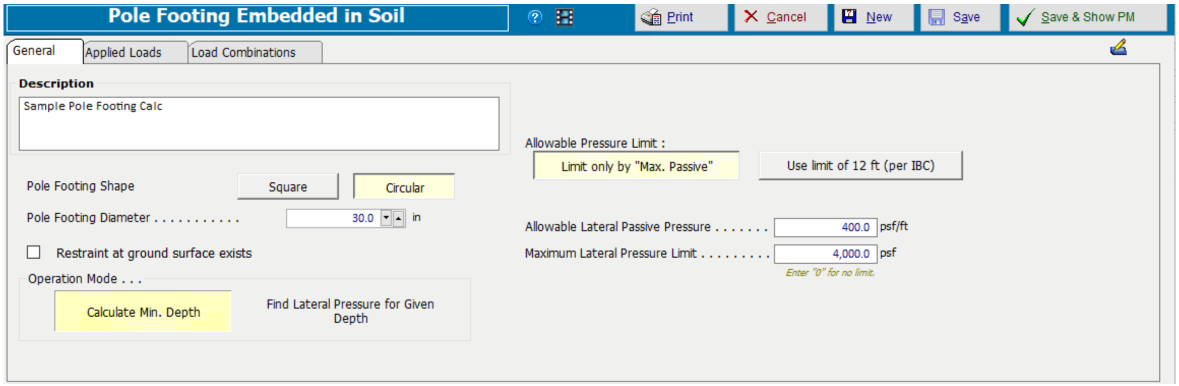
Since applied top moment generates lateral soil pressures that usually govern the design, these footings typically have a depth/width ratio of 2:1 and greater.

Cases with and without lateral restraint at the ground surface are allowed. Evaluation of actual and allowable pressures is in accordance with the IBC Section entitled "Embedded posts and poles".



Load Combination	Forces @ Ground Surface Loads - (k)	Moment - (k-ft)	Required Depth - (ft)	Pressure at 1/3 Depth Actual - (psf)	Allow - (psf)	Vertical ...plied -
D Only	0.00	0.00	0.125	0.0	0.0	2.0
+D+0.60W	8.40	67.20	10.500	1,386.3	1,389.1	2.0
+D+0.450W	6.30	50.40	9.375	1,234.3	1,236.1	2.0
+0.60D+0.60W	8.40	67.20	10.500	1,386.3	1,389.1	1.2
+0.60D	0.00	0.00	0.125	0.0	0.0	1.2

General Data Tab



Pole Footing Shape

Use this section to specify whether the pole is round or rectangular (assumed square).

Footing Width/Diameter

Enter the width or diameter of the pole footing. Width is measured perpendicular to force direction. If the pole is specified as rectangular, the module will multiply the value entered for footing width 1.41 to determine an equivalent width dimension for calculations.

Restraint at Ground Surface

Specify whether the footing is free at the ground surface or restrained and cannot translate. A restrained footing indicates that a concrete slab or other rigid element prevents translation of the pole footing at the ground surface, but does not prevent rotation. When specifying a restrained footing, you must assure yourself that the final force required to provide the restraint can actually be developed by the restraining construction.

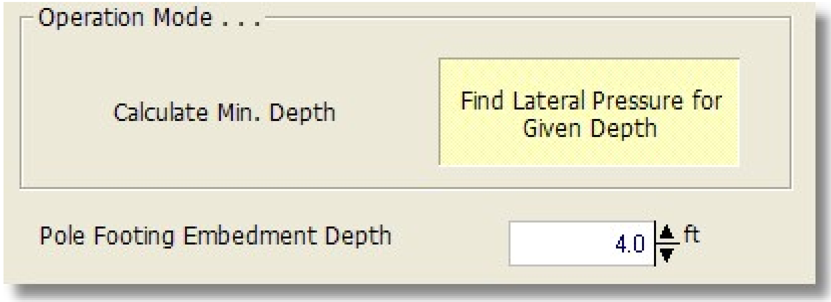
When ground surface restraint is present, the lateral pressure value at the bottom of the pole will govern the design.

Operation Mode

This setting provides an option to select from two different modes of operation as follows:

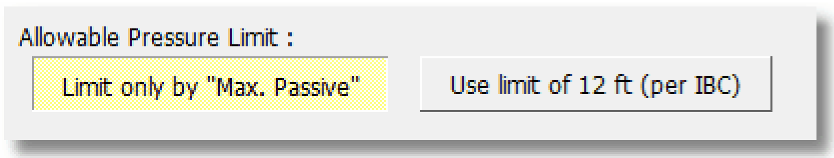
Calculate Minimum Depth: In this mode, the module will iterate to determine the minimum embedment depth required to make the actual lateral soil pressure lower than the allowable soil pressure.

Find Lateral Pressure for Given Depth: In this mode, the module will calculate the lateral earth pressures caused by the specified pole size, embedment depth and applied loads. When this option is selected, a Pole Footing Embedment Depth input field will appear as shown below:



Allowable Pressure Limit

Two options are provided as indicated below:



Limit only by "Max. Passive": Solves for a design that allows the passive pressure to approach the value specified in the Allowable Lateral Passive Pressure field below (limited to the value specified in the Maximum Lateral Pressure Limit field).

Example: Assume allowable lateral passive pressure is 200 psf/ft with an upper limit of 3000 psf.

<u>When the **Limit only by "Max. Passive"** option is selected, the solution will progress as follows:</u>

- The program will start with a shallow assumed depth and calculate the 1/3 embedment depth.
- Then it will calculate an allowable lateral passive pressure for that 1/3 embedment depth.
- Next, the program will compare this calculated allowable lateral passive pressure value to the specified upper limit on the allowable passive pressure and select the smaller of the two.
- The IBC formula is then used to determine the actual pressure for the assumed embedment depth.
- If the actual pressure is higher than the allowable pressure, the program increments the length and repeats the above process.
- For illustration, assume that the iterations have progressed to the point where the embedment depth is now 42 feet.
- The program will calculate the 1/3 embedment depth as (42 feet / 3) = 14 feet.
- Then it will calculate an allowable lateral passive pressure of (200 psf/ft * 14 ft) = 2800 psf.
- Next, the program will compare this calculated allowable lateral passive pressure value to the specified upper limit on the allowable passive pressure and determine

that 2800 psf < 3000 psf, therefore it will use 2800 psf as the Allowable Lateral Passive Pressure.
- When the program finds an embedment depth for which the actual pressure is lower than the allowable pressure, it rounds the embedment depth up slightly and reports that value.

Use limit of 12 ft (per IBC): Solves for a design that achieves a passive pressure that does not exceed the Allowable Lateral Passive Pressure, where the Allowable Lateral Passive Pressure is calculated based on 1/3 of the embedment depth but not to exceed 12 feet (and limited to the value specified in the Maximum Lateral Pressure Limit field).

Example: Assume allowable lateral passive pressure is 200 psf/ft with an upper limit of 3000 psf.

When the **Use limit of 12 ft (per IBC)** option is selected, the solution will progress as follows:
- The program will start with a shallow assumed depth and calculate the 1/3 embedment depth.
- Next, it will compare the 1/3 embedment depth to 12 feet and base the allowable lateral passive pressure calculation on the smaller of the two.
- Next, the program will compare this calculated allowable lateral passive pressure value to the specified upper limit on the allowable passive pressure and select the smaller of the two.
- The IBC formula is then used to determine the actual pressure for the assumed embedment depth.
- If the actual pressure is higher than the allowable pressure, the program increments the length and repeats the above process.
- For illustration, assume that the iterations have progressed to the point where the embedment depth is now 42 feet.
- The program will calculate the 1/3 embedment depth as (42 feet / 3) = 14 feet.
- Next, it will compare the 1/3 embedment depth to 12 feet and determine that 14 feet > 12 feet, therefore it will base the allowable lateral passive pressure calculation on 12 feet.
- Then it will calculate an allowable lateral passive pressure of (200 psf/ft * 12 ft) = 2400 psf.
- Next, the program will compare this calculated allowable lateral passive pressure value to the specified upper limit on the allowable passive pressure and determine that 2400 psf < 3000 psf, therefore it will use 2400 psf as the Allowable Lateral Passive Pressure.
- When the program finds an embedment depth for which the actual pressure is lower than the allowable pressure, it rounds the embedment depth up slightly and reports that value.

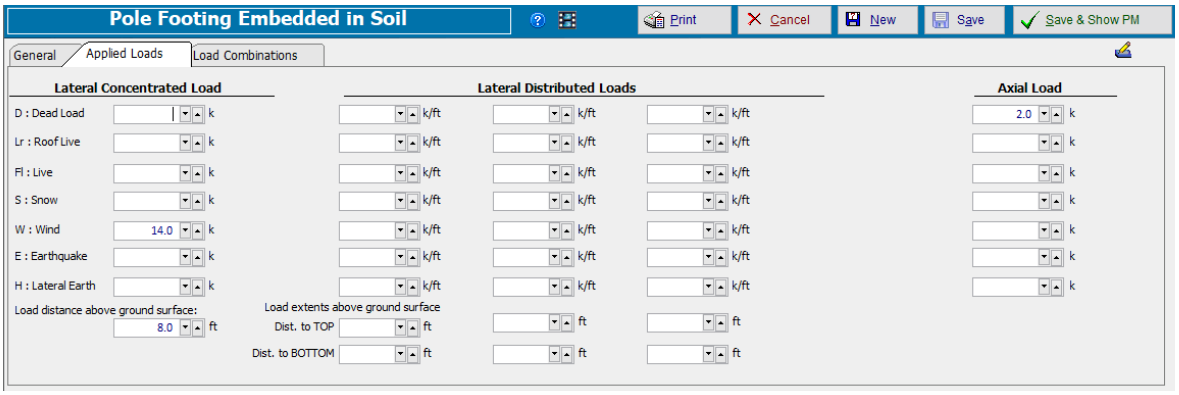
Allowable Lateral Passive Pressure

The allowable lateral passive pressure that the soil can withstand. This value is entered as pounds per square foot per foot of embedment depth.

Maximum Lateral Pressure Limit

This value is used to specify an upper limit on the Allowable Lateral Passive Pressure, so that it does not increase in an uncontrolled manner as the embedment depth increases. This value is entered as pounds per square foot.

Applied Loads Tab



This module allows many types of loads to be applied to a pole footing embedded in soil.

Lateral Concentrated Loads

Module allows one concentrated load with various load types to be applied at a specified distance above the surface of the soil.

Lateral Distributed Loads

You can apply a uniform lateral load to the pole by specifying the magnitude of the load and the starting and ending locations.

Applied Moments

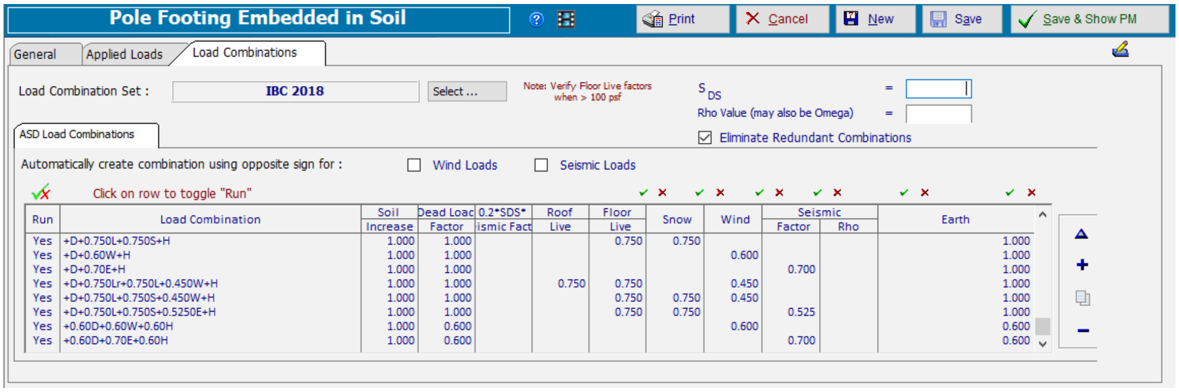
You can apply a concentrated moment. No "height" entry is required, because it is purely a rotational force.

Vertical Load

You can also apply a vertical load so that the module can calculate the vertical bearing load on the footing for each load combination.

Load Combinations Tab

Use this tab to specify the load combinations you want the module to analyze.

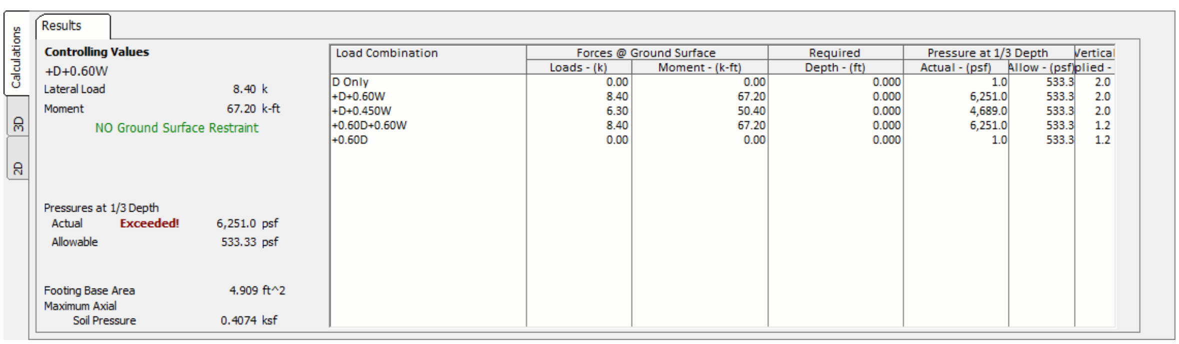


Results Tab

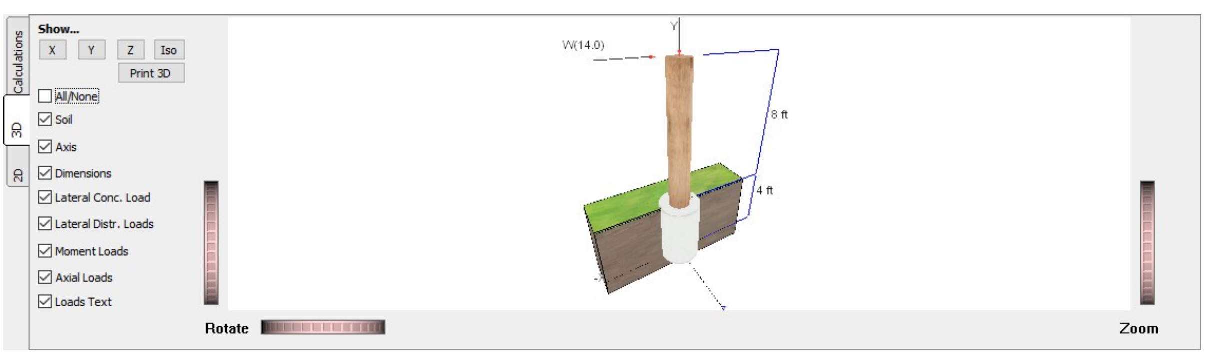
The results tab provides a summary of the calculations.

The table reports the resulting forces, moments and required depth for each load combination.

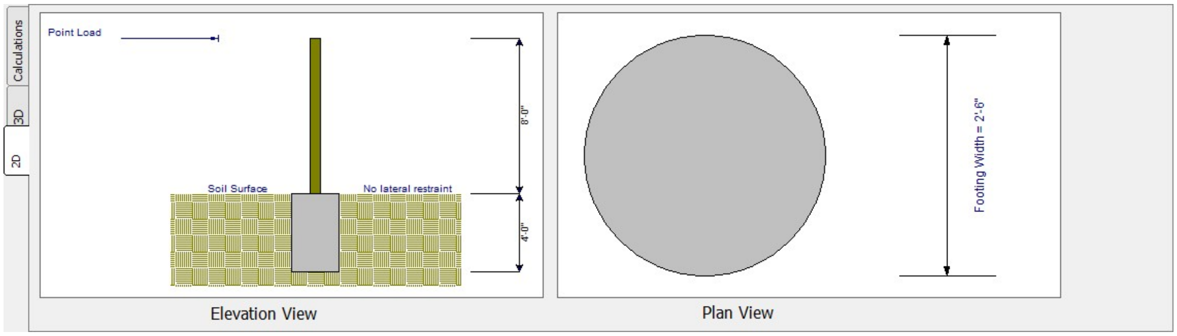
The Controlling Values area provides information for the most severe load combination.



3D Tab



2D Tab


Calculations
3D
2D
Point Load
Soil Surface
No lateral restraint
8'-0"
4'-0"
Footing Width = 2'-6"
Elevation View
Plan View

10.7.6 Beam on Elastic Foundation

See Beam on Elastic Foundation 294 in the Beams section.

10.8 2D Frame Analysis

Need more? Ask Us a Question

This module provides force & deflection analysis, steel member stress analysis and wood stress analysis for two dimensional frames. Click here for the first in a series of videos:

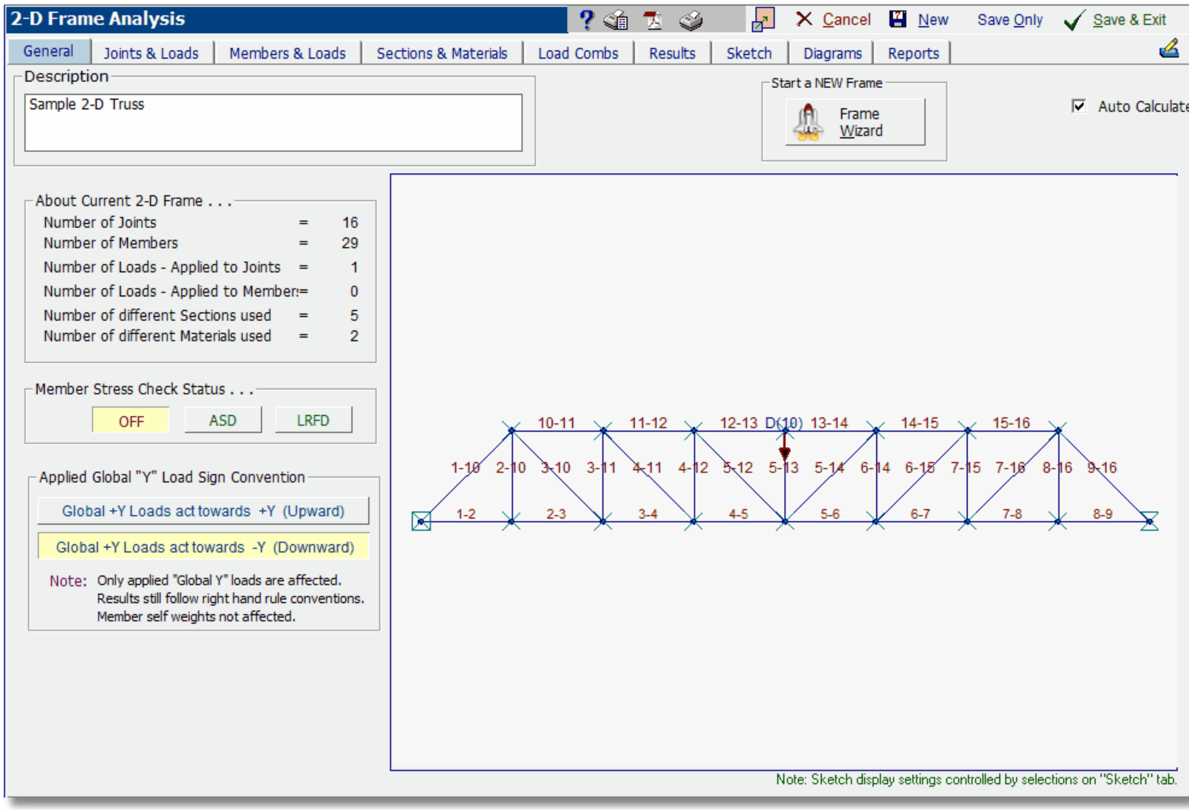
2-D Frame Analysis Intro

The frame you create can be general in nature...the module is not limited to trusses or frames with rigid connections. You have the ability to specify the connection between all members in terms of axial, shear and bending restraints or releases.

Here are the highlights of this module:

- Use the Frame Wizard to automatically create common trusses and frames.
- Generic joint & member method of specifying the geometry of the frame.
- Completely flexible way to specify how the X, Y & Z degrees of freedom for joints are connected to the earth and how the members are connected to the joints (three degrees of freedom are available at each end of each member).
- Apply forces and moments directly to joints. You can also specify joint start temperatures.
- All loads can have dead, live, roof live, snow, wind, seismic and earth types.
- Apply concentrated and distributed loads to beams. These loads can be Global or local in direction and can be applied as forces, moments or temperature changes.
- Members are easily linked to section properties. For stress analysis you can set unbraced length, slenderness factors, Cb & Cm for members.
- Loads and members can be deactivated for quick modeling of alternatives.
- Complete AISC and NDS section databases are available.
- Specify material properties used by sections.
- Specify an extensive set of load combinations.
- Complete graphics of frame with deflected shapes.
- Complete stress & deflection diagrams of members individually.
- Extensive reporting capability and control.
- Module uses beam elements and a very fast matrix solver. With each change of the input data, the frame can be completely recalculated in an instant. (For convenience with large models or when running many load combinations, this module has been enhanced with the ability to turn the Automatic Recalculation feature off, and to trigger manual recalculations only when desired.)

Note: This module is intended to be a simple alternative to complex frame programs. The intent is to provide a simple and fast tool to perform simple indeterminate analysis & member design. No P-Delta effects, exotic elements, or non-linear analysis will ever be added to this module.



Program Limitations

Maximum Number of Allowed:

Joints = 300 Joint Loads

=600Beams=600BeamPointLoads=

600BeamDistributedLoa

ds = 600
Sections = 100
Materials = 50
LoadCombinations

=

1
0
0

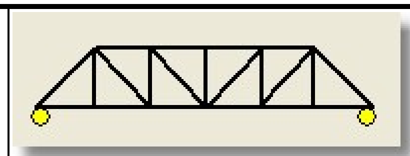
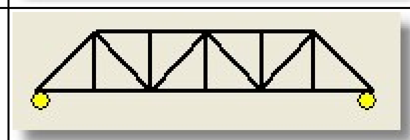
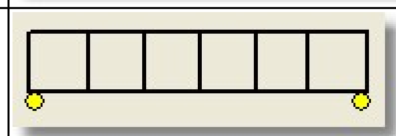
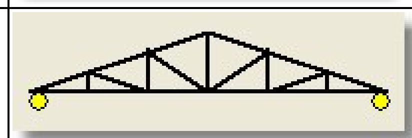
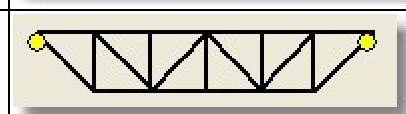
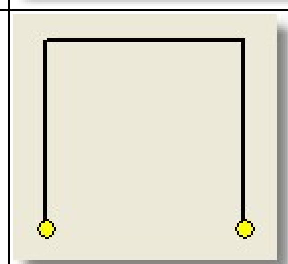
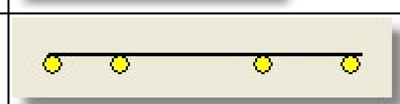
10.8.1 Frame Wizard

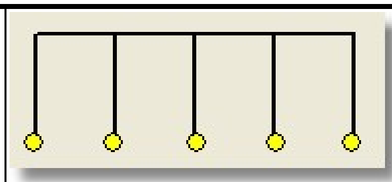
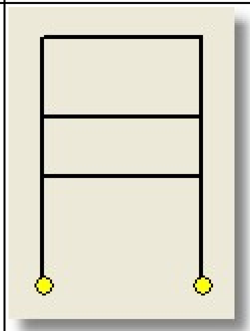
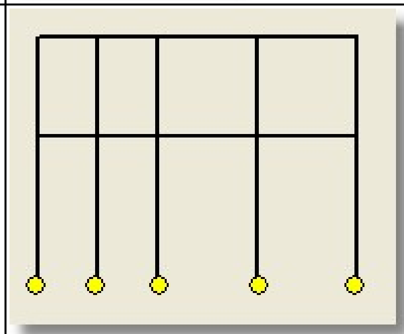
Overview

The **Frame Wizard** can create complete 2D Frame models for many typical framing configurations and truss layouts with just a few simple inputs.

The Frame Wizard is also especially helpful when you are just beginning to use this module and want to learn about how to specify joint end restraints and member end releases.

Types of Models the Frame Wizard Can Generate

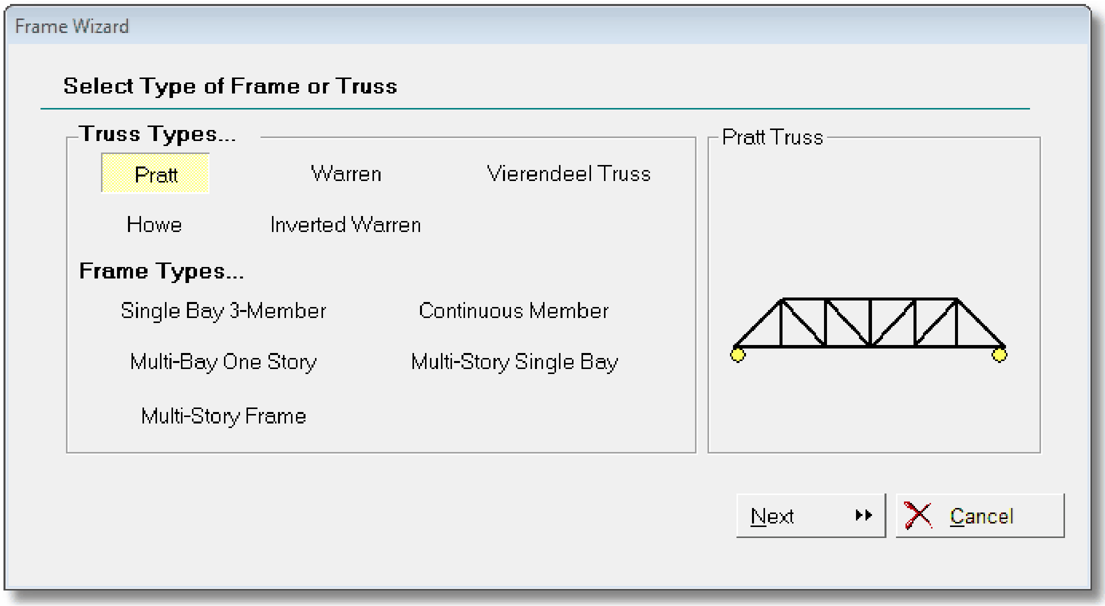
Pratt Truss	
Warren Truss	
Vierendeel Truss	
Howe Truss	
Inverted Warren Truss	
Single Bay 3-Member Frame	
Continuous Member	

Multi-Bay One Story Frame	
Multi-Story Single Bay Frame	
Multi-Story Frame	

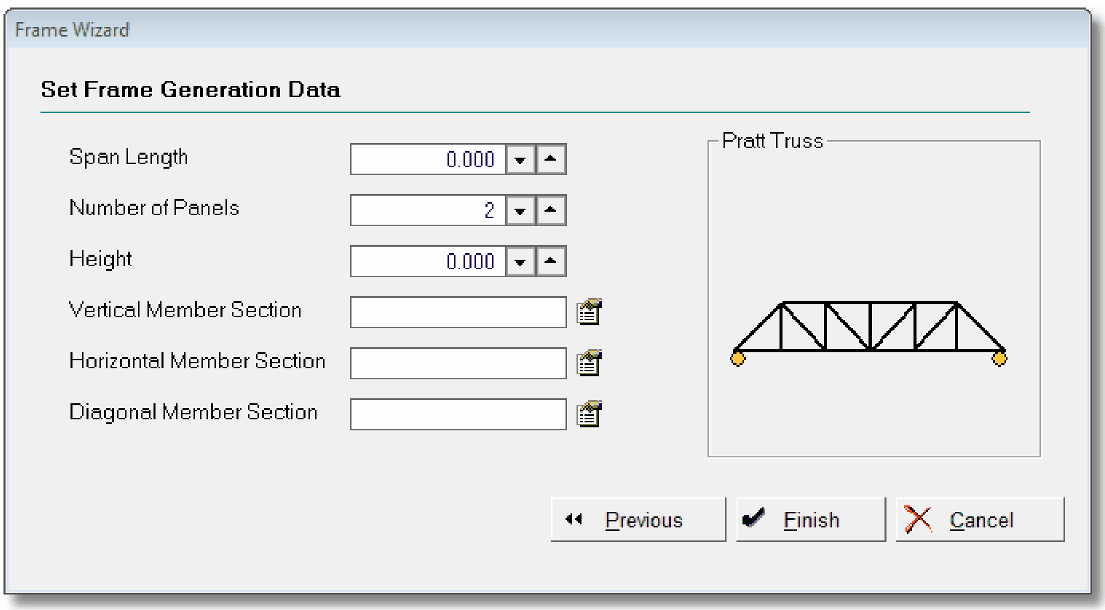
Please send additional requests to support@enercalc.com.

Example: Using the Frame Wizard to create a Pratt Truss

Creating a truss with the Frame Wizard is extremely simple. Just start by selecting [**Pratt**] and clicking [**Next**].



The screen you see below will be displayed.



Simply enter the overall length and height of the truss and the number of panels the truss will have. The sample image shown has 6 panels.

The next entries allow you to easily specify a member size for the three main members that are typically the same on a truss like this (Verticals, Horizontals, and Diagonals).

The button to the right of each data entry area provides access to the built-in section database in **Structural Engineering Library**.

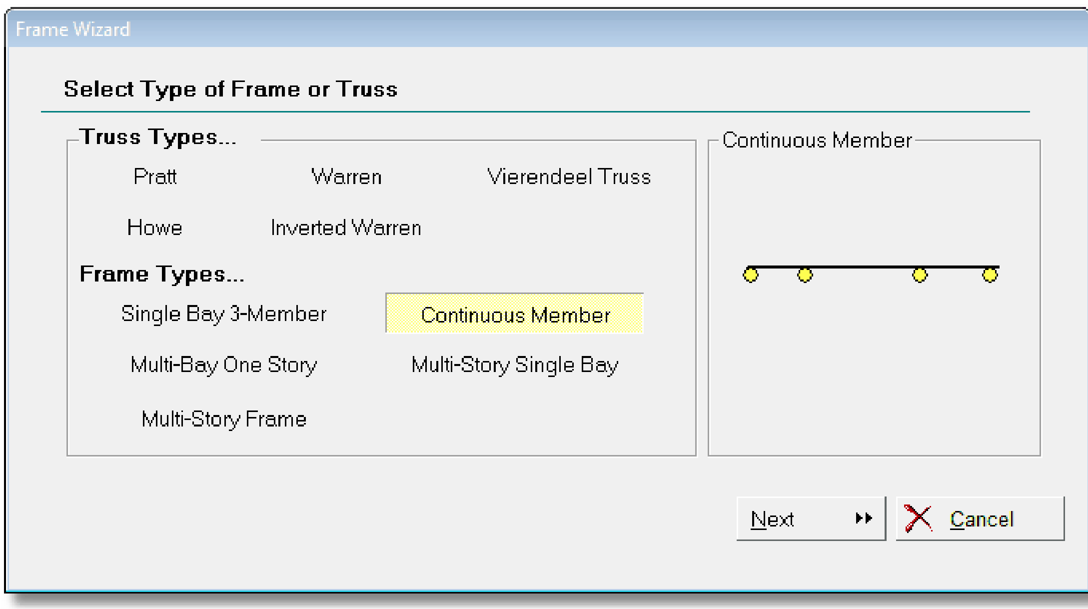
If you **DO** specify a database section, all of the associated section properties will be brought into the frame model for you.

If you **DO NOT** specify a database section, then the members will be assigned a name of "Vertical", "Diagonal" or "Chord" as a section group. Then all you need to do is enter property values for those three sections.

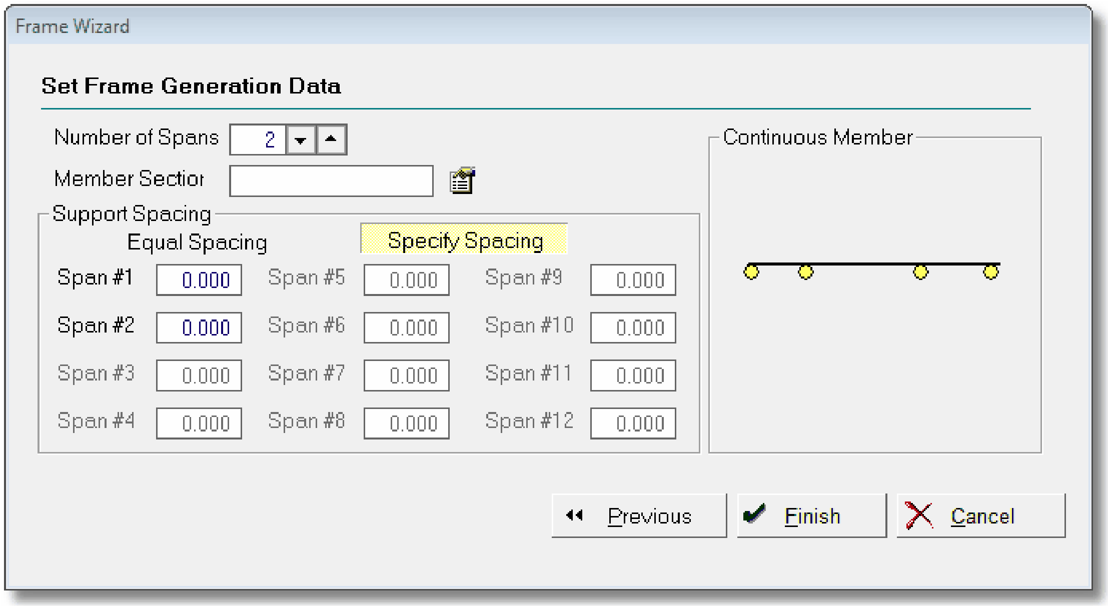
To finish just click [**Finish**]. The model will be instantly created.

Example: Using the Frame Wizard to create a Continuous Beam

The Frame Wizard is an extremely efficient way to model a continuous beam. Just start by selecting [**Continuous Member**] and then click [**Next**].



The screen you see below will be displayed:



Note: This wizard allows just one beam to be entered BUT you can easily change that beam specification for each span after the model is built.

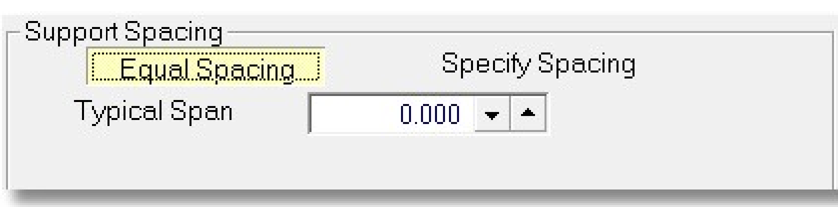
Enter the Number of Spans first.

The next entry allows you to easily specify a member section name. The button to the right of each data entry area provides access to the built-in section database in **Structural Engineering Library**.

If you **DO** specify a database section, all of the associated section properties will be brought into the continuous beam model for you.

If you **DO NOT** specify a database section then the members will be assigned a name of "Beam" as a section group. Then all you need to do is enter property values for that section.

For specification of span lengths you can either use [**Equal Spacing**]:



-or- [**Specify Spacing**] which allows you to enter the span lengths for each span.

Then just click [**Finish**], and the model will be instantly created.

10.8.2 Joints & Joint Loads

[Need more? Ask Us a Question]

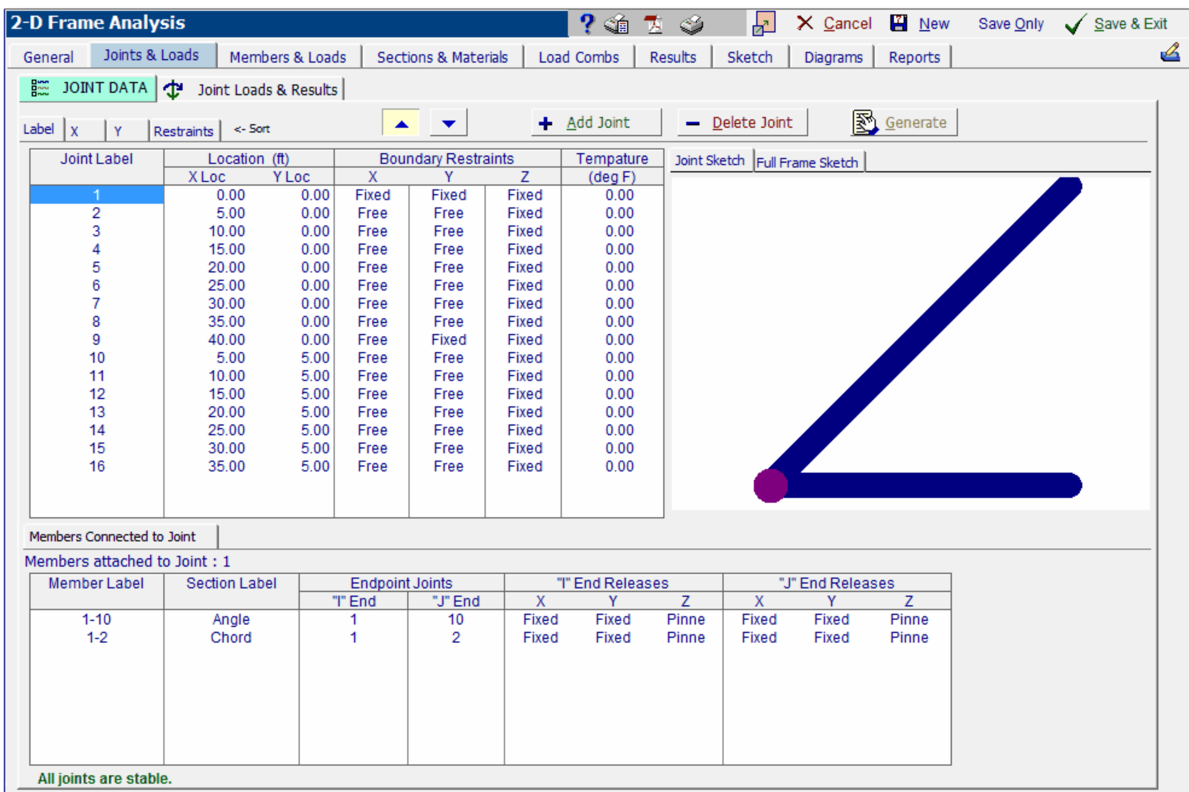
Joints are the points in space where members are interconnected. Members can ONLY connect to another member or the boundary by first connecting to a joint.

IMPORTANT: See **JOINT RESTRAINTS** below.

Joints & Loads Tab

This tab contains two sub-tabs that allow you to specify joints and loads applied to the joints.

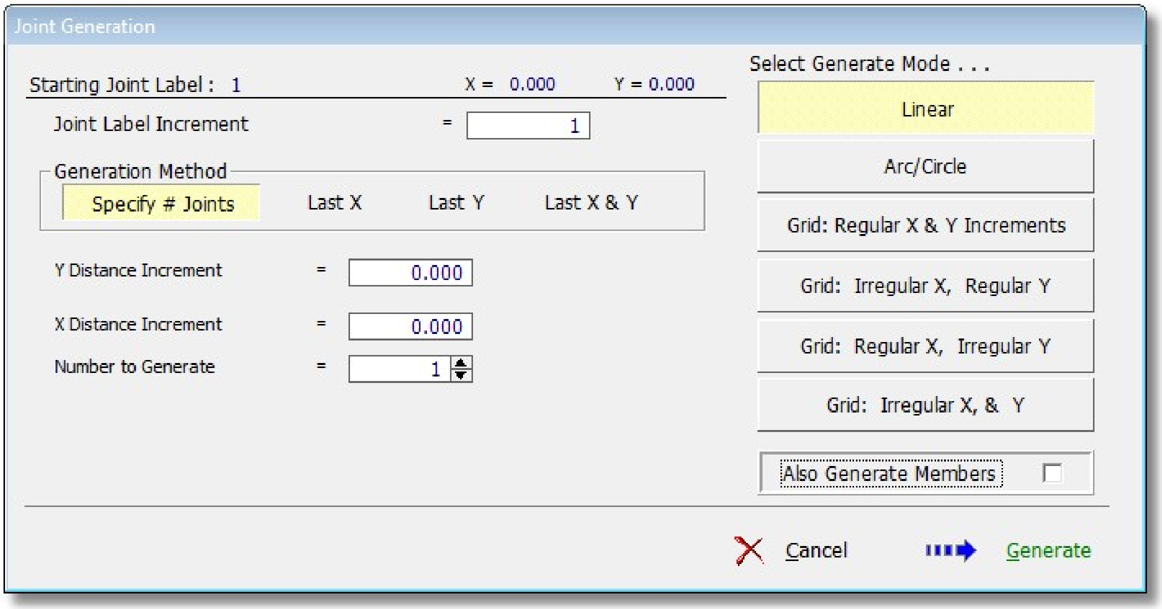
Note: The data entered into this module is linked by a database system. The effects of this linking can be seen in the screen capture seen below. Notice that the table in the upper half of the view lists available joints for which Joint Data can be viewed. As a result of the data linking, selecting Joint #1 in the upper table automatically causes the lower table to display a list of members connected to Joint #1.



Automatic Joint Generation

You can use the [**Generate**] button to have joints automatically generated. The joint generation popup is self explanatory....just select from the mode of generation, specify a label increment, the number of joints to generate, and the x and y distance increments to use. The "Grid Irregular X & Y" can generate some very complex joint layouts.

Another powerful tool is the Also Generate Members option, which will generate members to interconnect between the joints for the generation grid or arc/circle you have specified.



JOINT RESTRAINTS

These interconnections are extremely important, and when set incorrectly, they lead to the majority of the instabilities and other unexpected results when using this 2-D Frame Analysis module. The connections between members and the connections to "ground" have a significant effect on structural behavior, and as such, they warrant a thorough discussion and complete understanding. Always keep in mind the following concepts related to joint boundary restraints and the end releases of the member(s) connecting to the joint:

- Joints have Degrees of Freedom (DOF) associated with them.
- In the 2-D Frame Analysis program, each joint has two translational and one rotational DOF.
- An individual DOF may be restrained or released.
- DOF can be defined as Boundary Restraints or as Member End Releases.
- When the degrees of freedom are defined as Boundary Restraints, they are defined with respect to the global coordinate axis system.
- When the degrees of freedom are defined as Member End Releases, they are defined with respect to the member local coordinate axis system.

The term Degrees of Freedom refers to the capability of a joint to move in the X and Y direction and to rotate about the Z axis.

An X Restraint prevents a joint from moving in the Global X direction.
A Y Restraint prevents a joint from moving in the Global Y direction.
A Z Restraint prevents a joint from rotating about the Global Z axis.

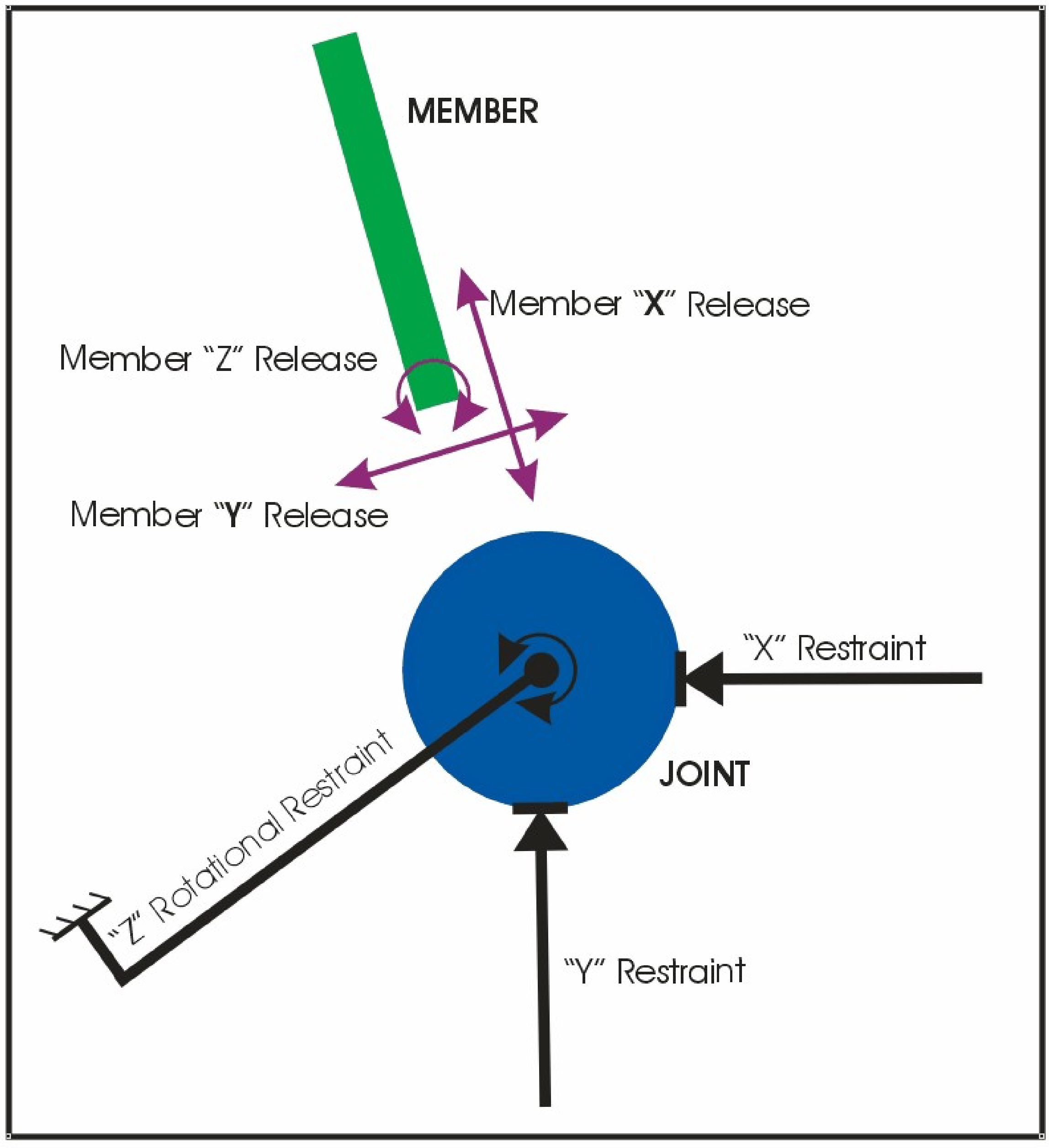
Please see the diagram below for a brief primer on joint restraints and member releases.

The **dark black** items define the joint restraints. **Joint restraints attach the joint to the boundary.** The boundary is an infinitely stiff support and is most commonly called earth. When a joint is restrained against translation in the X and Y directions and against rotation about the Z axis, it is held firmly in place. Any forces or moments applied to it will result in a reaction which is the boundary applying a force to the joint to keep it from moving in a particular direction.

THE MOST IMPORTANT PIECE OF INFORMATION: Each joint must be prevented from moving in/rotating about each of its three degrees of freedom, either by a joint restraint or by a positive connection to member. If a joint can translate or rotate in an unrestrained manner, then it is unstable.

NEXT MOST IMPORTANT PIECE OF INFORMATION: When a joint restraint is specified, then any connected members that are not released in that particular direction will not deflect in that direction. If a joint has an "X" restraint and a member framing into that joint does not have an "X" release, then the member is actually rigidly connected to the boundary through the joint.

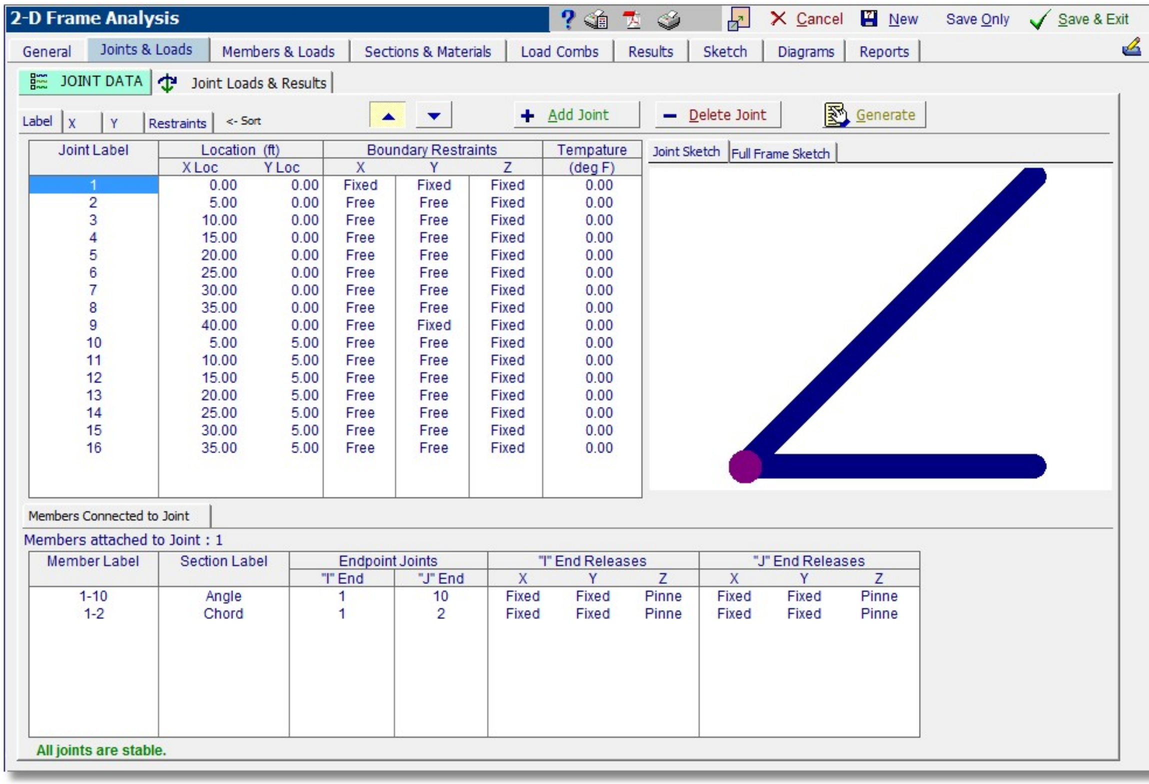
ALSO IMPORTANT: When you are modeling a frame, nearly all joints will be free from the boundary. This means that those joints will need to have their X, Y & Z degrees of freedom restrained by a connection to one or more members. Because members can only attach to each other through a joint, this is usually handled by virtue of the member having all three degrees of freedom "fixed" at its ends. BUT FOR TRUSS CONNECTIONS, THIS CAN CAUSE PROBLEMS WITH THE "Z" rotational degree of freedom. For truss models (where moments are not applied to nodes), it may be easier to achieve stability for all joints by declaring all joints as "fixed" against rotation about the Z-axis, and then "pinning" both ends of all members to prevent the transfer of moment from one member to another.



10.8.2.1 Joint Data

Joint Data Tab

This tab is where you add and edit the joints that define the member interconnections for the frame.



Joint Label

The label for a joint can be up to 25 characters long. They are CASE INSENSITIVE meaning the module internally converts all joint names to all lower case for internal usage.

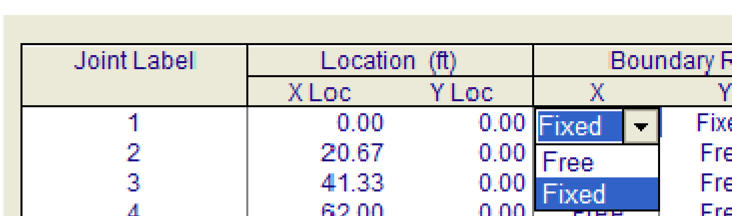
Location

This is the X and Y location of the joint in a Cartesian coordinate system.

Boundary Restraints

These specify how the joint is connected to the boundary. The boundary is an infinitely rigid item and is typically earth. See the Joints & Joint Loads 521 topic for more information.

When you click on these items (or use [Tab] to move between them) the entry turns into a drop-down list box with the available selections.



Temperature

This specifies the base temperature for the joint. It is only used when you are defining temperature loads for the members.

[+Add Joint]

Prompts you for the label to assign to the newly added joint. Joint labels can be up to 25 characters long.

[- Delete Joint]

Deletes the highlighted joint after your confirmation. Keep in mind that members depend upon joints for connectivity. So if there are any members connected to a joint, the members will be deleted if the joint is deleted. Any joint loads that were assigned to a joint will also be deleted if the joint is deleted.

Generate

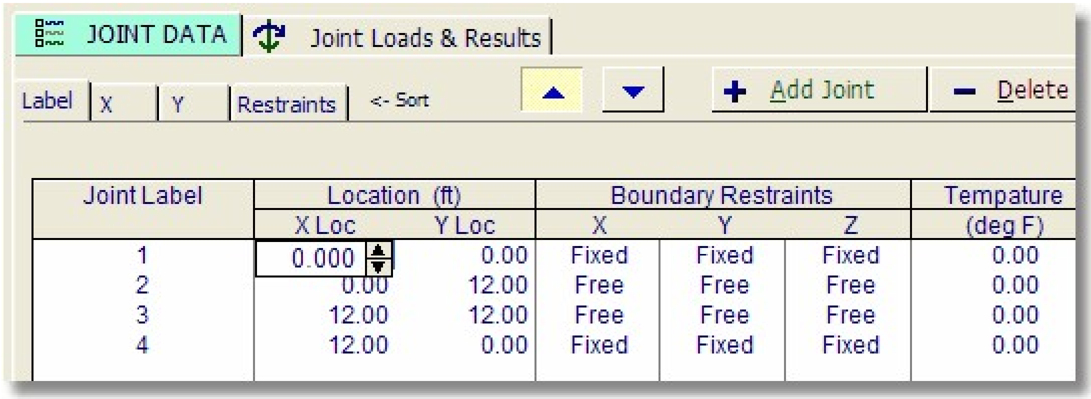
See bottom of this section.

Joint Sketch/Full Frame Sketch

The Joint Sketch displays a graphic representation of the individual joint that is currently selected in the list of joints. The Full Frame Sketch displays a graphic representation of the entire model with the currently selected joint highlighted for easy recognition.

Editing Joint Information

Click on any data item when the joint list first appears, and the list will switch into an editing mode. See the image below where we have clicked on the "X Loc" for joint 1.



You can type in a numeric value or use the "spin" buttons to change the value by a fixed decimal amount.

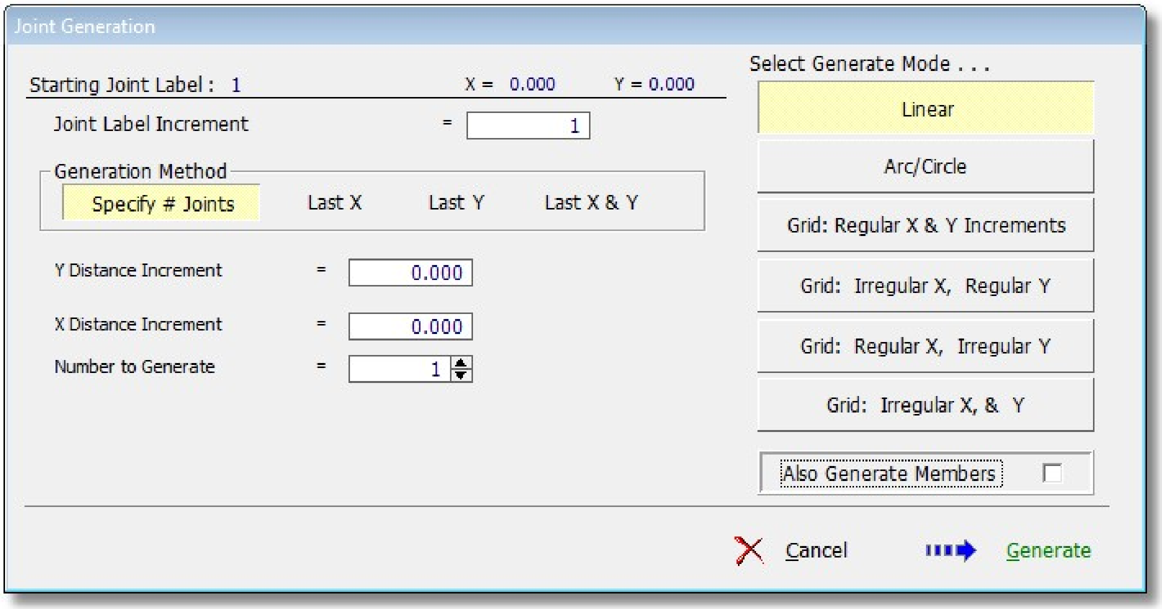
To finish the entry either press [**Tab**], [**Enter**] or click on another data item in the list.

When your entry is completed the entire frame will be recalculated if the Auto Calculate option is selected.

Automatic Joint Generation

You can use the [**Generate**] button to initiate the automatic generation of joints. Just select from the mode of generation, and you can specify the label and the x and y distance increments to use. The "Grid Irregular X & Y" can generate some very complex joint layouts.

Another powerful tool is the Also Generate Members option, which will generate members to interconnect the joints for the generation grid or arc/circle you have specified.



10.8.2.2 Joint Loads & Results

Joint Loads & Results Tab

This tab provides the input locations for all loads applied to the joints in the frame EXCEPT member self weights.

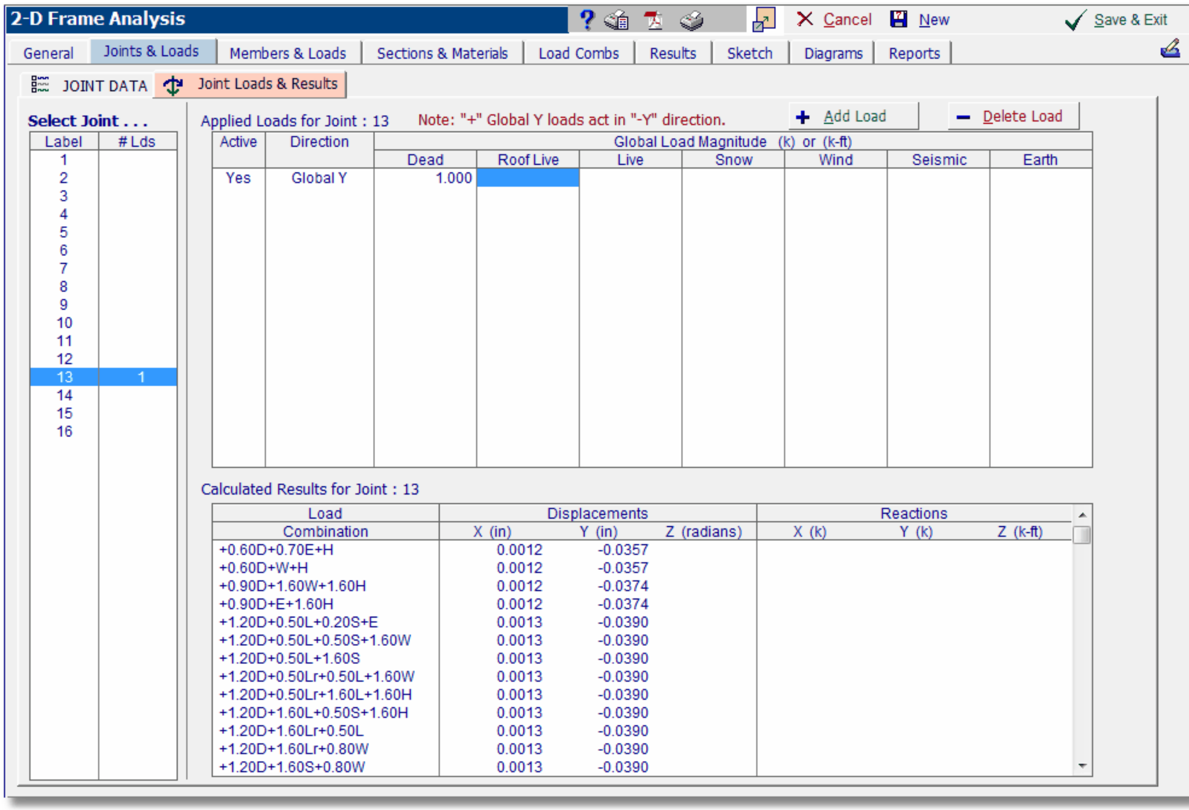
This tab is divided into three sections:

Select Joint: list is where you select the joint for which you wish to modify loads. This selection controls what is visible in the other two lists.

Applied Loads for Joint: is where you build and edit the loads applied to that joint

Calculated Results for Joint: displays the calculated displacements and reactions for the joint selected in the Select Joint list.

The only list that can be edited is the Applied Loads for Joint list.

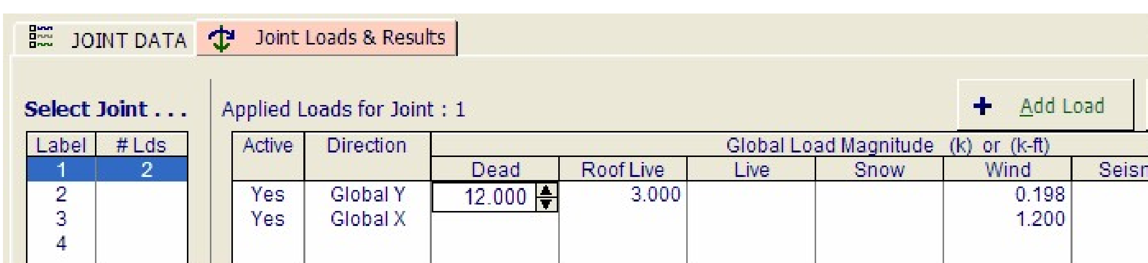


Adding & Deleting Loads

To add a joint load just click the [**Add Load**] button and a new load entry line will be added at the bottom of the list for the currently selected joint.
To delete a load just click on the load line you wish to delete and click the [**Delete Load**] button.

Editing Joint Load Information

Click on any data item in the joint load list, and it will switch into an editing mode. See the image below where we have clicked on the Dead load column in the first line in the list of loads associated with joint 1.



You can type in a numeric value or use the "spin" buttons to change the value by a fixed decimal amount.

To finish the entry either press [**Tab**], [**Enter**] or click on another data item in the list.

When your entry is completed the entire frame will be recalculated if the Auto Calculate option is selected.

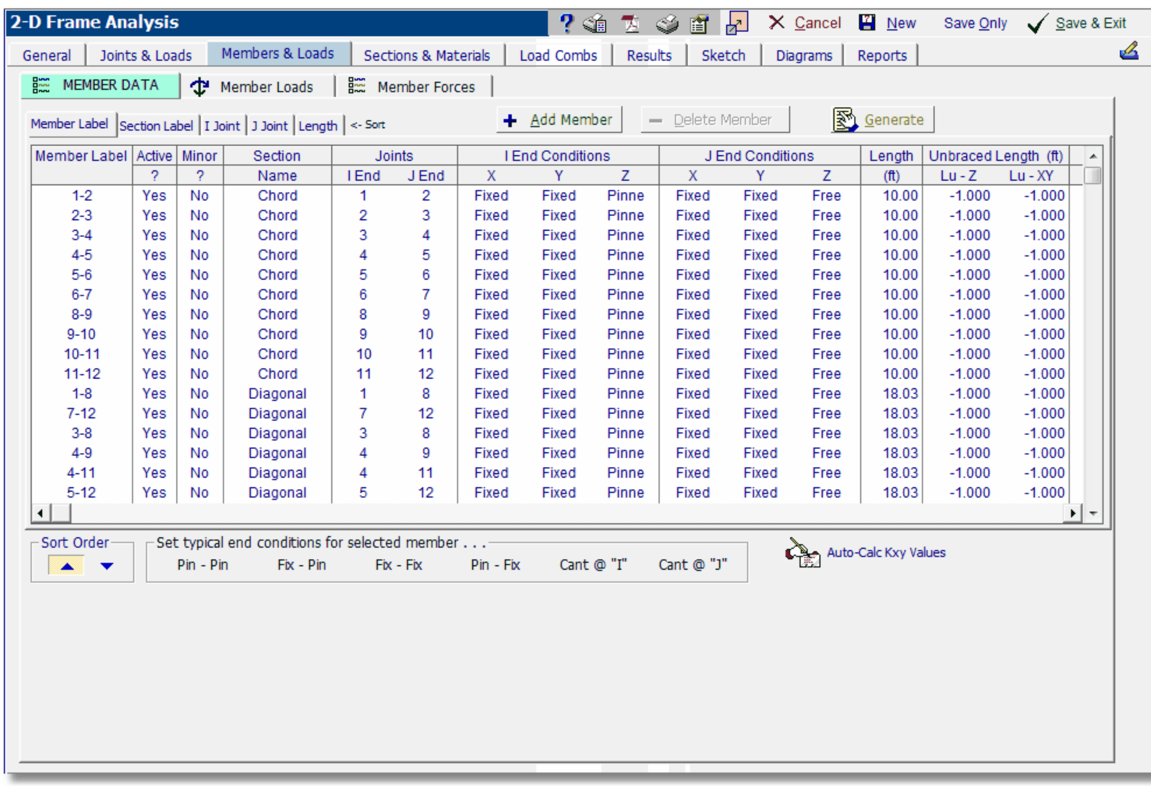
Note: If a load is applied in the same direction that a Joint Restraint is specified, that load will immediately be "absorbed" by the joint restraint, so it will not have any effect on the frame.

10.8.3 Members & Member Loads

[Need more? Ask Us a Question]

"Member" is the generic term that describes the beams, columns, struts, braces, diagonals, hangers, and other structural entities that make up the frame. This module uses a general stiffness matrix approach to solve for forces and deflections. Members are modeled by one-dimensional (length-only) entities that offer resistance to axial loads, bending moment about an axis perpendicular to the plane of the frame, and shear forces acting in the plane of the frame.

Members & Loads Tab



Member End Releases

These interconnections lead to the greatest potential for instabilities and other unexpected effects from the use of the 2-D module. The connections between members and the connections to "ground" have a significant effect on structural behavior, and as such, they warrant a thorough discussion and complete understanding. Always keep in mind the following concepts related to joint boundary restraints and the end releases of the member(s) connecting to the joint:

- Joints have Degrees of Freedom (DOF) associated with them.
- In the 2-D Frame Analysis program, each joint has two translational and one rotational DOF.

- An individual DOF may be restrained or released.
- DOF can be defined as Boundary Restraints or as Member End Releases.
- When the degrees of freedom are defined as Boundary Restraints, they are defined with respect to the global coordinate axis system.
- When the degrees of freedom are defined as Member End Releases, they are defined with respect to the member local coordinate axis system.

The term Degrees of Freedom refers to the capability of a joint to move in the X and Y direction and to rotate about the Z axis.

An X Restraint prevents a joint from moving in the Global X direction.
A Y Restraint prevents a joint from moving in the Global Y direction.
A Z Restraint prevents a joint from rotating about the Global Z axis.

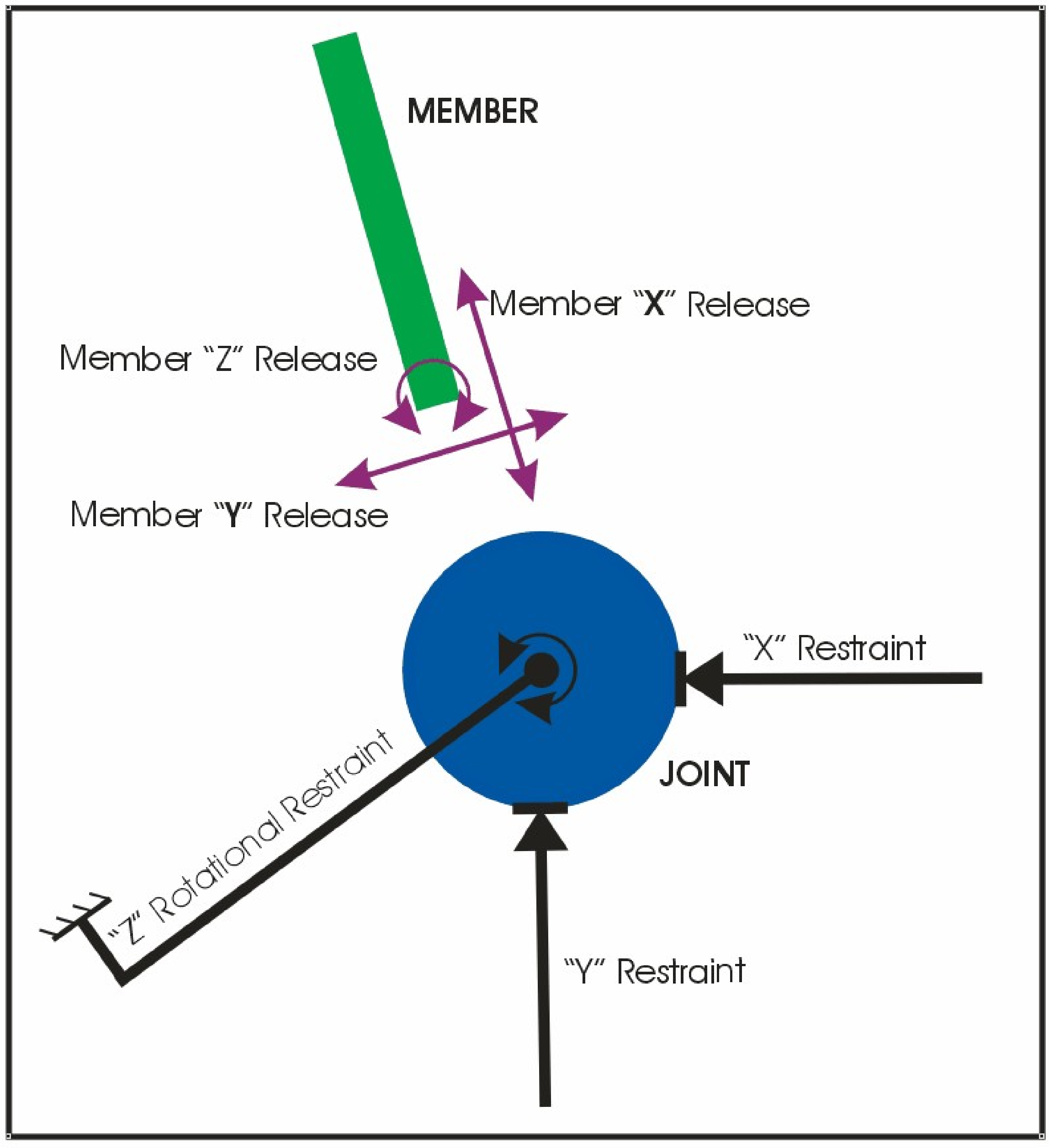
Please see the diagram below for a brief primer on joint restraints and member releases.

The **dark black** items define the joint restraints. **Joint restraints attach the joint to the boundary.** The boundary is an infinitely stiff support and is most commonly called earth. When a joint is restrained against translation in the X and Y directions and against rotation about the Z axis, it is held firmly in place. Any forces or moments applied to it will result in a reaction which is the boundary applying a force to the joint to keep it from moving in a particular direction.

THE MOST IMPORTANT PIECE OF INFORMATION: Each joint must be prevented from moving in/rotating about each of its three degrees of freedom, either by a joint restraint or by a positive connection to member. If a joint can translate or rotate in an unrestrained manner, then it is unstable.

NEXT MOST IMPORTANT PIECE OF INFORMATION: When a joint restraint is specified, then any connected members that are not released in that particular direction will not deflect in that direction. If a joint has an "X" restraint and a member framing into that joint does not have an "X" release, then the member is actually rigidly connected to the boundary through the joint.

ALSO IMPORTANT: When you are modeling a frame, nearly all joints will be free from the boundary. This means that those joints will need to have their X, Y & Z degrees of freedom restrained by a connection to one or more members. Because members can only attach to each other through a joint, this is usually handled by virtue of the member having all three degrees of freedom "fixed" at its ends. BUT FOR TRUSS CONNECTIONS, THIS CAN CAUSE PROBLEMS WITH THE "Z" rotational degree of freedom. For truss models (where moments are not applied to nodes), it may be easier to achieve stability for all joints by declaring all joints as "fixed" against rotation about the Z-axis, and then "pinning" both ends of all members to prevent the transfer of moment from one member to another.


MEMBER
Member "X" Release
Member "Z" Release
Member "Y" Release
"X" Restraint
JOINT
"Z" Rotational Restraint
"Y" Restraint

10.8.3.1 Member Data

Member Data Tab

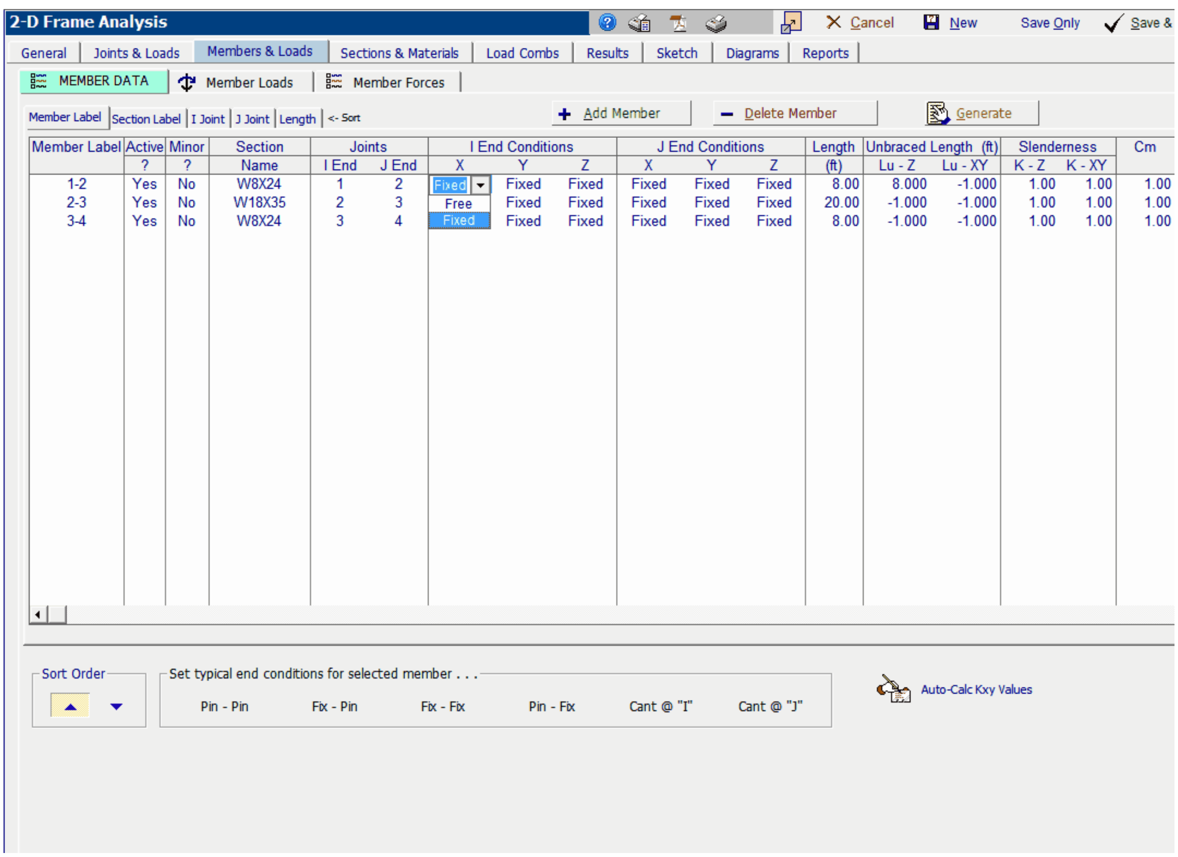
Members always span between two joints. The Member End Conditions tell the module how each of the three degrees of freedom at each end of the member are attached to the joint. To connect two members rigidly together you must specify the two connecting ends as "fixed" conditions for all three degrees of freedom. That locks each beam end to the joint.

For each member you can specify the section to use for its properties, and values to be used when stress analysis is performed (unbraced length, slenderness factor, Cm and Cb).

You can also set the member to be inactive to test force distributions and stresses for alternate framing conditions.

Editing Member Information

Click on any data item when the member list first appears, and the list will switch into an editing mode. See the image below where we have clicked on the "X" degree of freedom for the I: end of member 1-2.



You can type in a numeric value or use the "spin" buttons to change the value by a fixed decimal amount.

To finish the entry, either press [**Tab**], [**Enter**] or click on another data item in the list.

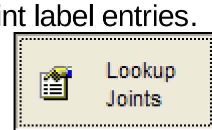
Note: Each column in the Member Data table can have its own type of editing mode. The Active column turns into a checkbox for yes/no selections. The Section Name column turns into a button so you can select or add a section from the Section list. See descriptions for each item below.

Adding & Deleting Members

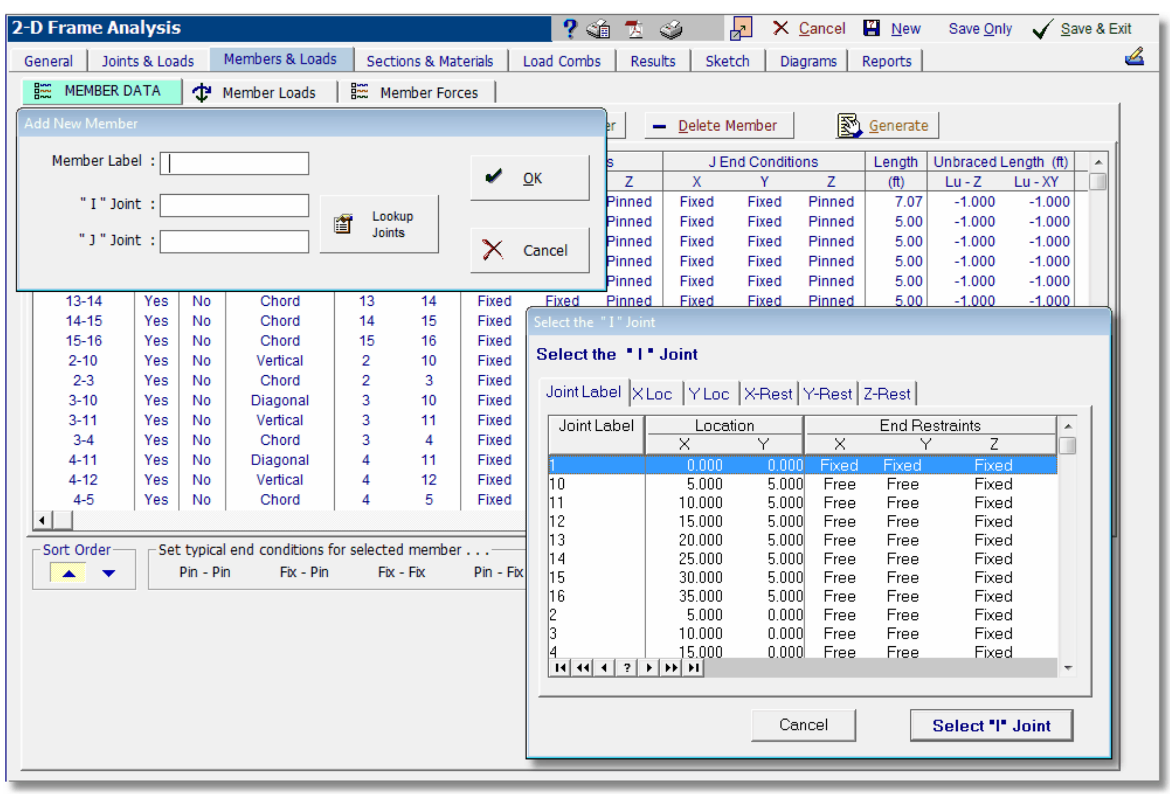
To add a member click the [**Add Member**] button. You will be prompted for three pieces of information:

1) A label for the member.
2) The member's "I" joint number.
3) The member's "J" joint number.

In the screen capture below you will notice a button to the right of the joint label entries.

Just click in the "I" or "J" joint entry and click the [**Lookup Joints**] button 

. This will display a window where you can scroll through the list of created joints and select one. Simply click on one of the listed joints and then click [**Select**].

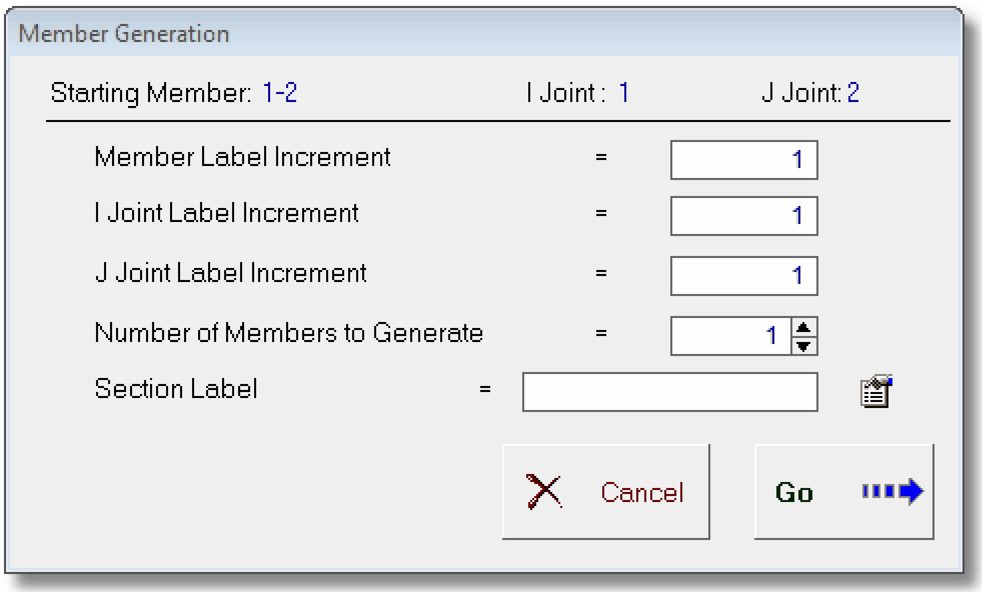


To delete a member, select the member and click [**Delete Member**]. After you approve the deletion, that member and all loads applied to that member will be deleted.

Generating Members

You can use the [**Generate**] button to initiate the automatic generation of members. This is a simple process that automates member creation by generating members between existing nodes in the specified order.

Note: This tool will generate new members, but it will not generate new nodes.



Detailed Item Descriptions (*see 2 screen captures below*)

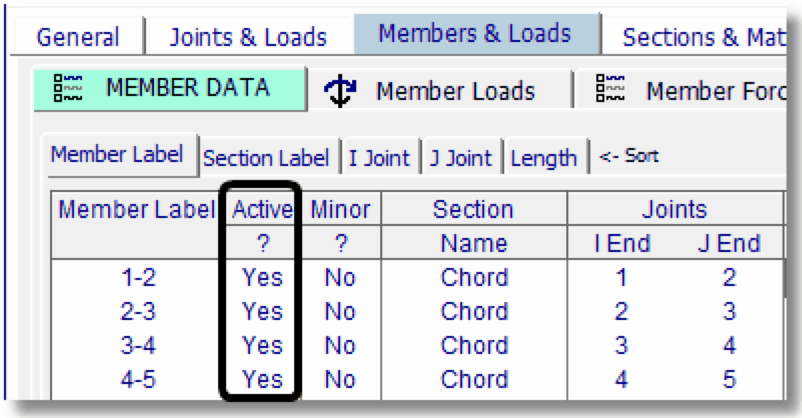
Member Label

This label can be any combination of letters and numbers. It is common to use numbers for joints and letters for members, but a very convenient member labeling convention is to use the I and J joint numbers separated by a dash. So the member between joint 5 and 12 would be labeled "**5-12**".

The member label can ONLY be specified when Adding the member. After that point, the name can't be changed unless the member is deleted and added again.

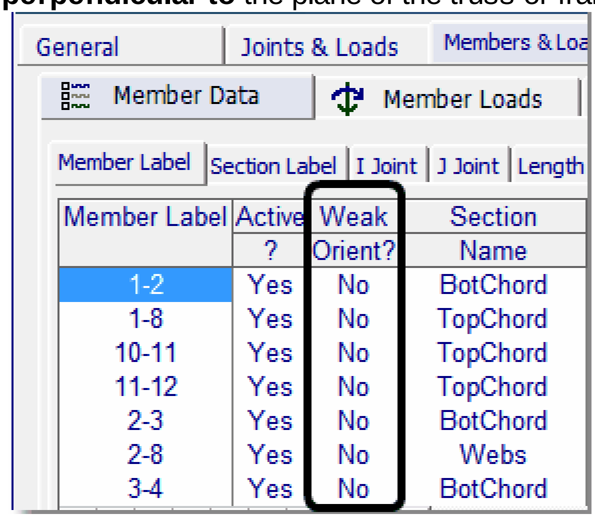
Active

This item is a yes/no selection. When you click on it or [**Tab**] to it, the entry changes to a checkbox. When the box is checked, the member will be considered active, meaning that it will contribute stiffness to the model. When the box is not checked, the member will be considered inactive, meaning that it will be ignored in the analysis and will not contribute stiffness to the model.



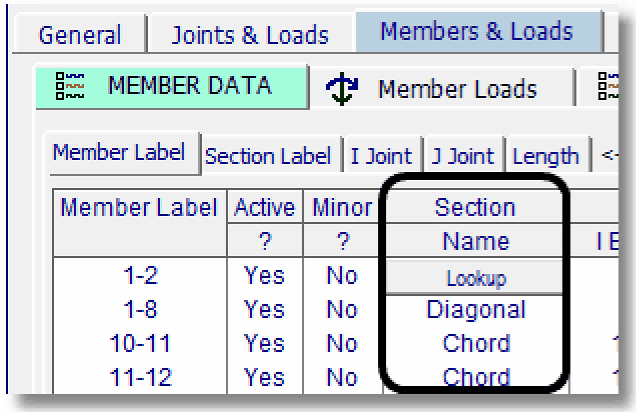
Weak Orientation

This item allows the member to be rotated so that its strong axis is **in** the plane of the truss or frame. By default. the member is assumed to be oriented so that its strong axis is **perpendicular to** the plane of the truss or frame.

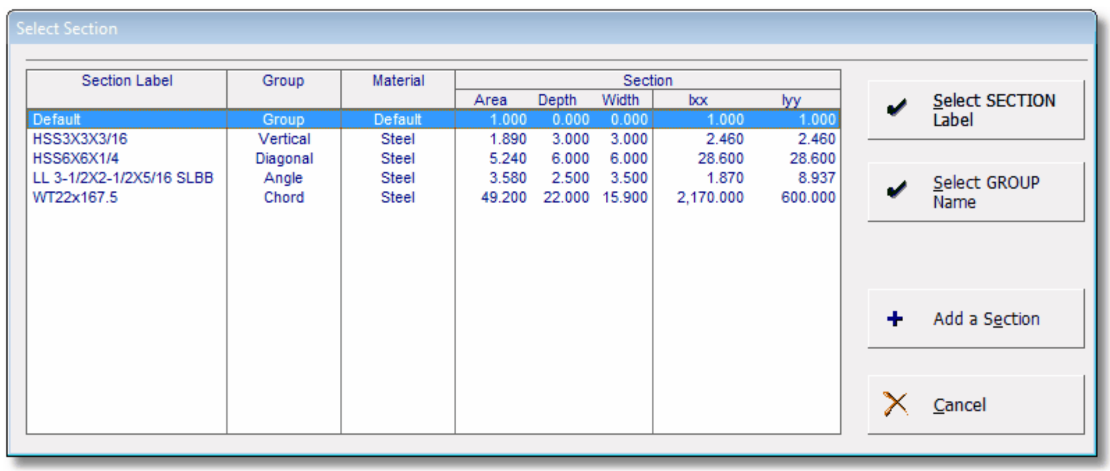


Section Name

This establishes what section is to be used for that member. When you click on it or [**Tab**] to it, the entry changes to a button labeled Lookup.



When you click the [Lookup] button, a selection window appears as shown below:

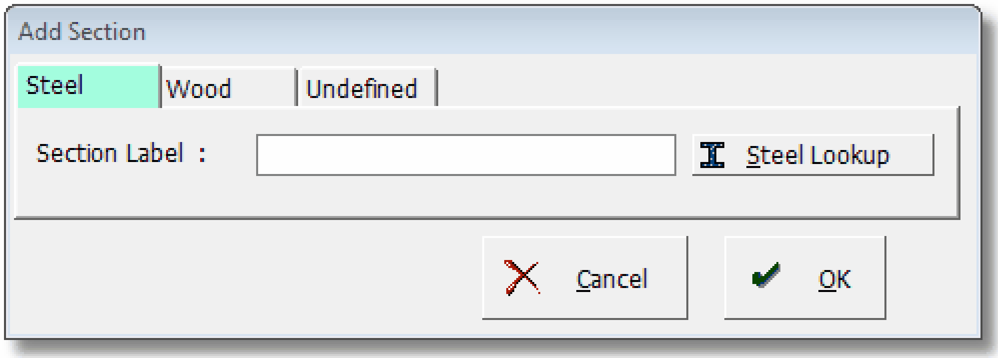


On this window you can scroll through the sections you have already added to the section list.

Note that the sections are listed with both Section labels and Group labels. A Group label is used when you want to have more than one member have the same section properties. In the screen capture above, notice that there are groups named "Chord", "Diagonal", and "Vertical", all of which have the same section assigned to them. This offers two convenient benefits:

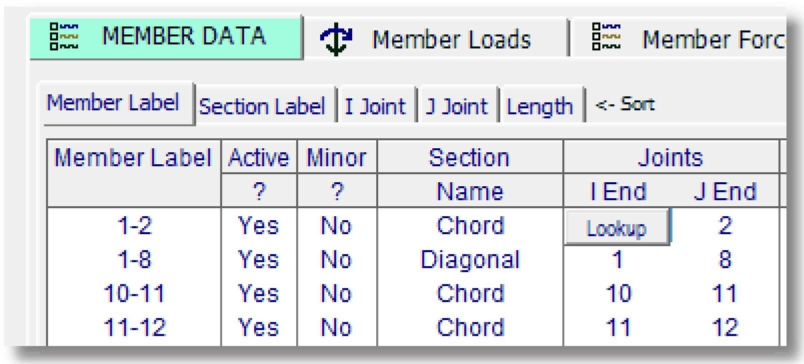
- First, it provides a handy way to manage multiple sets of members, some of which may have the same section.
- Second, by assigning a group name to the appropriate members in the model, it is possible to change the AISC or NDS section assigned to all members of that group by simply assigning a new section to the group, rather than having to assign the new section to many individual members.

The [**+Add Section**] button provides access to the built-in section property databases. It displays the screen below, which allows you to either type in a typical section name or click a button and display the database to select the desired section from the database.

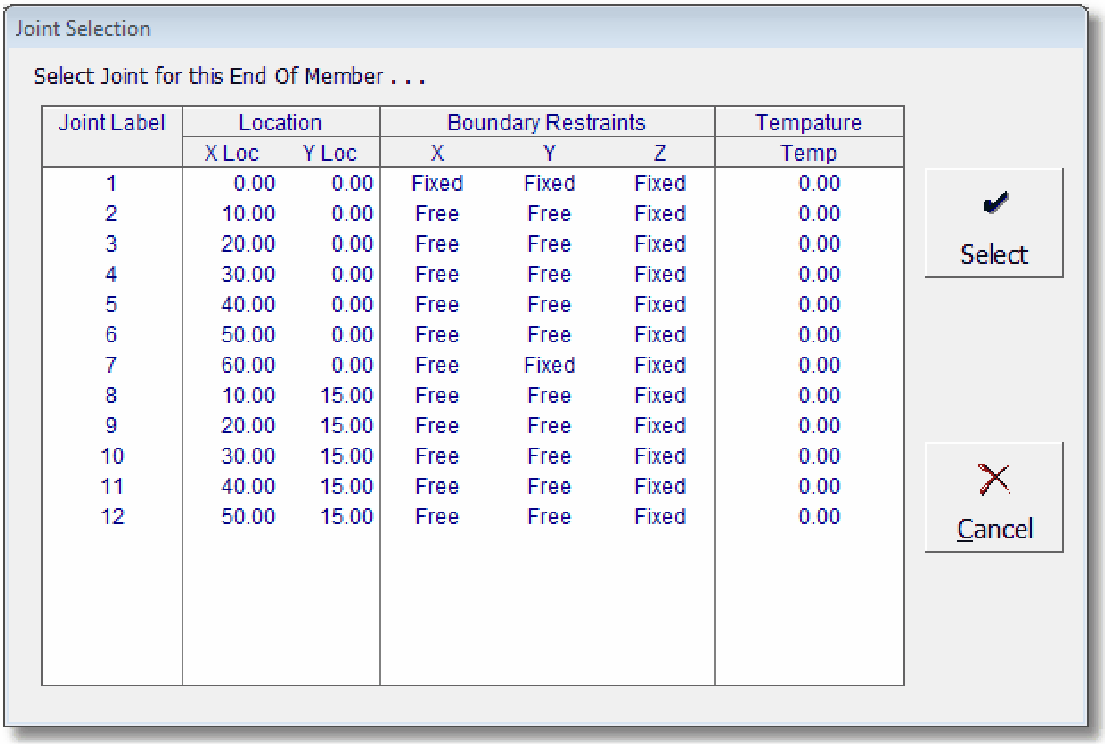


Joints

These two columns let you specify and change the i and j end joints of a member. When you click on these columns, the entry will change to a [**Lookup**] button as shown below:



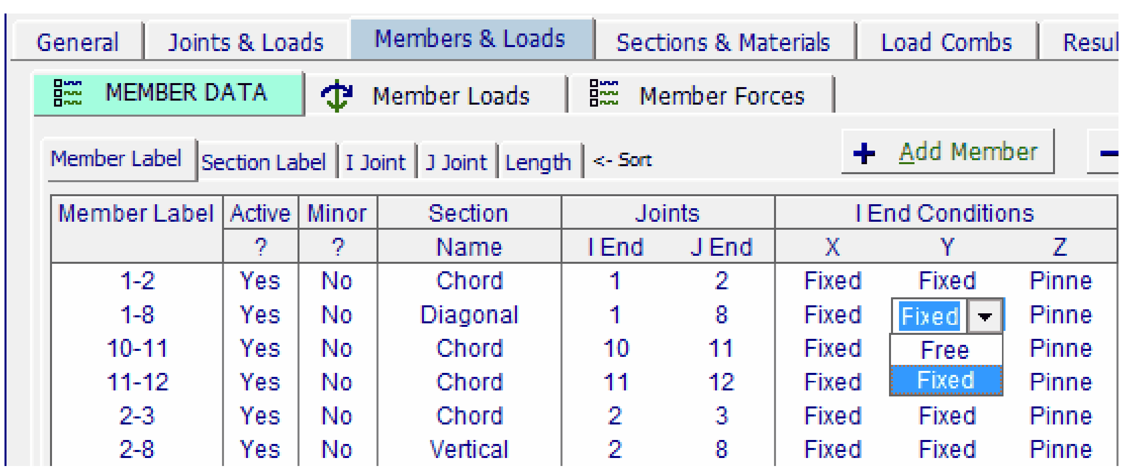
Clicking that button displays a window where you can scroll through the joint list and click to identify the desired joint.



Joint Label	Location		Boundary Restraints			Tempature
	X Loc	Y Loc	X	Y	Z	Temp
1	0.00	0.00	Fixed	Fixed	Fixed	0.00
2	10.00	0.00	Free	Free	Fixed	0.00
3	20.00	0.00	Free	Free	Fixed	0.00
4	30.00	0.00	Free	Free	Fixed	0.00
5	40.00	0.00	Free	Free	Fixed	0.00
6	50.00	0.00	Free	Free	Fixed	0.00
7	60.00	0.00	Free	Fixed	Fixed	0.00
8	10.00	15.00	Free	Free	Fixed	0.00
9	20.00	15.00	Free	Free	Fixed	0.00
10	30.00	15.00	Free	Free	Fixed	0.00
11	40.00	15.00	Free	Free	Fixed	0.00
12	50.00	15.00	Free	Free	Fixed	0.00

I & J End Conditions

These six columns allow you to specify how the ends of the member are attached to the joints. When you click on or [**Tab**] to one of these columns, the entry will change to a drop-down list box offering appropriate fixity options:



Member Label	Active	Minor	Section	Joints		I End Conditions		
	?	?	Name	I End	J End	X	Y	Z
1-2	Yes	No	Chord	1	2	Fixed	Fixed	Pinne
1-8	Yes	No	Diagonal	1	8	Fixed	Fixed	Pinne
10-11	Yes	No	Chord	10	11	Fixed	Free	Pinne
11-12	Yes	No	Chord	11	12	Fixed	Fixed	Pinne
2-3	Yes	No	Chord	2	3	Fixed	Fixed	Pinne
2-8	Yes	No	Vertical	2	8	Fixed	Fixed	Pinne

In order to most efficiently describe the I & J End Conditions, it helps to introduce the concept of the member local axis system. Each member can be thought of as having its own x, y, and z coordinate axes that are mutually perpendicular and follow the right-hand rule. The orientation of the member local axes can be determined as follows:

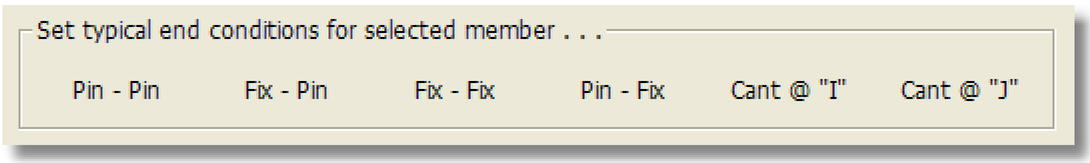
1. The local x axis is always parallel to a vector from the "I" node to the "J" node. (This axis is parallel to the longitudinal axis of the member.)
2. The local z axis is always parallel to the Global Z axis and points out of the plane of the screen.
3. The local y axis can be found by taking the vector cross product of local z cross local x. (Envision using the right hand to rotate the local z axis into the local x axis, and the right thumb will automatically indicate the positive direction of the local y axis.)

Note: It is important to understand that I & J End Conditions are defined with respect to the member local axes, not the Global axis system.

A "Fixed" status for a particular end of a member, for a particular degree of freedom (X, Y or Z) means that end of the member is locked to the joint for that degree of freedom. A "Free" or "Pinned" status means that end of the member is disconnected from the joint for that degree of freedom. Here are some examples:

X, Y, Z Setting	How it will work
Fixed, Fixed, Fixed	The member is locked to the joint. If this joint was completely restrained to the boundary, then this end of the member would have an X and Y force reaction and a Z moment reaction.
Fixed, Fixed, Pinned	The member is locked to the joint in its local X and Y directions, meaning that this end of the member cannot translate with respect to the joint, however, this end of the member is free to rotate independently of the joint about the member's local Z axis (which is always parallel to the Global Z axis).
Fixed, Free, Fixed	The member is locked to the joint with respect to translation in the local X axis direction. It is disconnected from the joint with respect to translation in the local Y axis direction. It is locked to the joint with respect to rotation about the member's local Z axis. If this member is oriented horizontally, then this combination of end conditions could be thought of as a vertical roller. When a member is in the horizontal orientation, its member local axes are parallel to the corresponding Global axes. Therefore, the fixed condition in the local X axis direction means that this end of this horizontal member cannot move left or right in the Global X direction. The free condition in the local Y direction means that this end of this horizontal member is free to move up and down in the Global Y direction. Finally, the fixed condition about the local Z axis means that this end of the member is fixed against rotation about the local Z axis, which is parallel to the Global Z axis.

Along the bottom of the list is a set of buttons that allows you to quickly set the Z-axis rotation end conditions. Clicking one of these buttons will set the end releases for both ends of the member that is currently selected in the list.



NOTE! A truss connection is unique. You must look at the joint where the truss members intersect. If they are all free to rotate, you will set the "X" and "Y" conditions of all member ends at that joint to "Fixed". And for all "Z" conditions you will set them to "Pinned" so they can rotate freely. For that joint you will set its "X" and "Y" restraint to "Free" (assuming it is not a support location) and set its "Z" restraint to "Fixed", so that the joint will be stable.

Length

This value is automatically calculated for you from the distance between the I and J joints.

Unbraced Length

Enter the unbraced compression edge length that should be used for allowable stress analysis of this member. Entering a "-1" means to use the full length of the member. Any other number (0.00 or greater) is used as the unbraced length. Note that the unbraced length can be assigned a value that is greater than the node-to-node length of a member.

Lu-y is used as the unbraced compression edge length for flexural design. It is also used in K-y * Lu-y to define the unbraced length for column buckling about the y axis of the member.

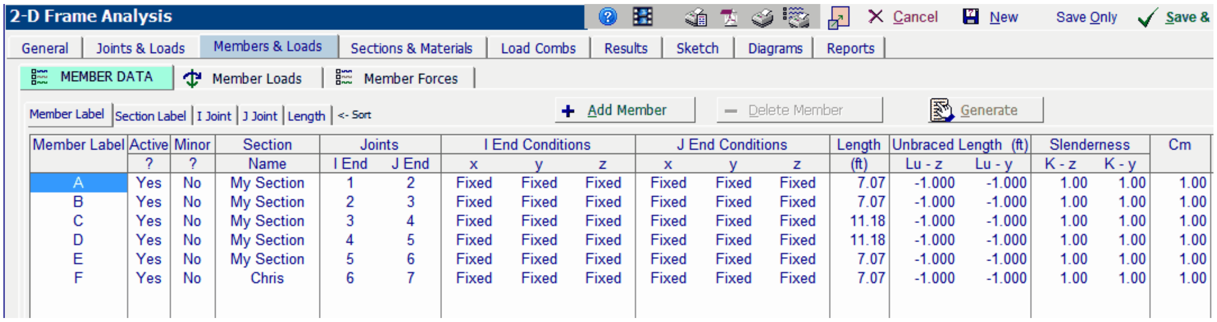
Lu-z is used in Kz * Lu-z to define the unbraced length for column buckling about the z axis of the member.

Slenderness

This entry is a simple multiplier to be applied to the Unbraced Length you have entered.

Cm

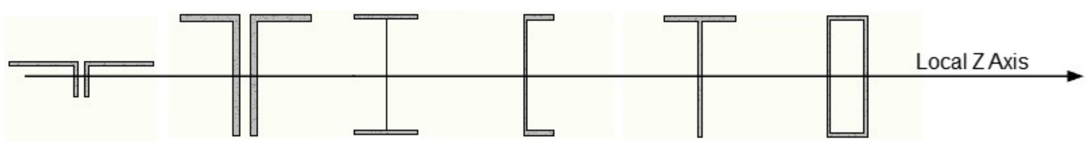
You can specify these values for use in allowable stress calculations.



Member Label	Active	Minor	Section	Joints		I End Conditions			J End Conditions			Length	Unbraced Length (ft)		Slenderness		Cm
	?	?	Name	I End	J End	x	y	z	x	y	z	(ft)	Lu - z	Lu - y	K - z	K - y	
A	Yes	No	My Section	1	2	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed	7.07	-1.000	-1.000	1.00	1.00	1.00
B	Yes	No	My Section	2	3	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed	7.07	-1.000	-1.000	1.00	1.00	1.00
C	Yes	No	My Section	3	4	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed	11.18	-1.000	-1.000	1.00	1.00	1.00
D	Yes	No	My Section	4	5	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed	11.18	-1.000	-1.000	1.00	1.00	1.00
E	Yes	No	My Section	5	6	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed	7.07	-1.000	-1.000	1.00	1.00	1.00
F	Yes	No	Chris	6	7	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed	7.07	-1.000	-1.000	1.00	1.00	1.00

Default Member Orientation

Based on the concept of the member local axis system introduced above, it is now meaningful to describe the default orientation of the various sections that can be assigned to a member in a 2-D Frame Analysis model. Of most interest is the default orientation of steel sections. When a steel section is assigned to a member, it will be assumed to be oriented as shown in the following diagram, which references the member local Z axis, the axis that is perpendicular to the plane of the model by default:



10.8.3.2 Member Loads

Member Loads Tab

This tab provides the input locations for all loads applied to the frame EXCEPT joint loads and member self weights. This module can automatically generate member self weight loads using Member Self Wt item in the load combination lists.

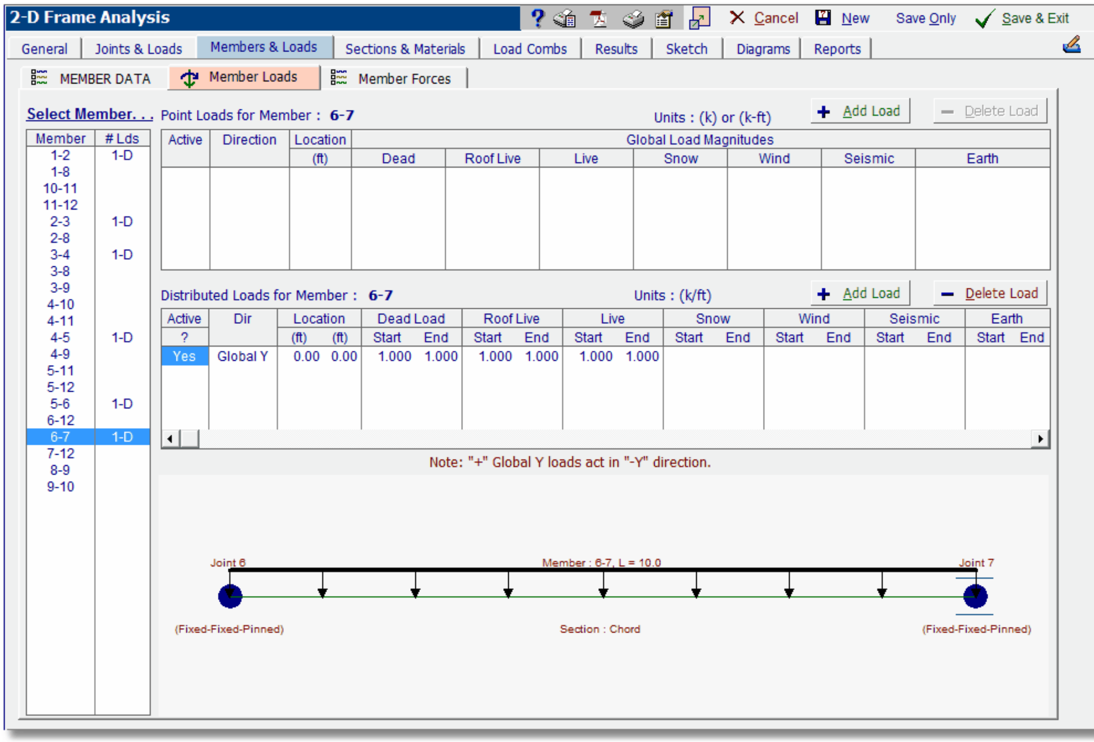
This tab contains three lists and one graphic area.

Select Member: lists all members in the model. This is where you click to select a member for which to add/delete/modify loads. This selection controls what is visible on the other two lists and in the graphic image at the bottom. The column labeled # Lds indicates the number and type of loads that have been applied to each member. "P" represents a point load, and "D" is for distributed loads. "1-P, 2-D" means that the member has one point load and two distributed loads applied to it.

Point Loads for Member X: shows the point loads that are applied to the selected member.

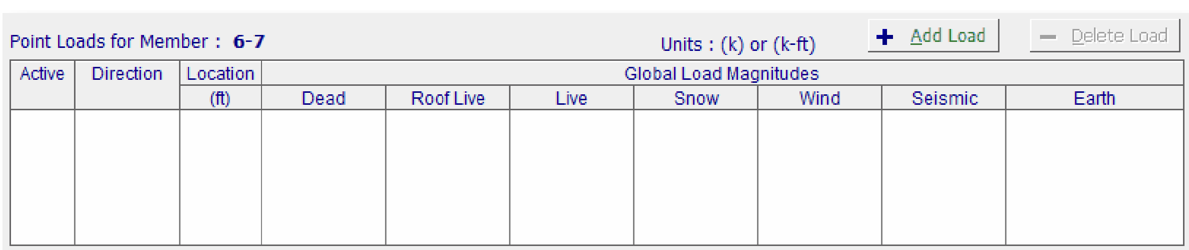
Distributed Loads for Member X: shows the distributed loads that are applied to the selected member.

At the bottom is a sketch showing the member and applied loads for reference.



When you click on any item in the two load entry lists, that location in the list will change into a data entry item. The columns in the list have their own specific entry types that will be described below.

Point Loads Entry List



Active

This is a Yes/No checkbox that allows you to turn the load on and off. This is useful when you want to quickly see the effect of removing the load without actually having to delete the load and then potentially have to redefine the load at a later time.

Direction

This specifies the direction of application of the load. Here is a description of each direction:

Global X: This point load acts in a direction parallel to the Global X Axis. Entering a positive value will apply the load to the right (in the positive X direction).

Global Y: This point load acts in a direction parallel to the Global Y Axis. The algebraic sign on the magnitude will affect the direction of application based on the Applied Global Y Load Sign Convention setting on the General tab as follows:

> When the Applied Global Y Load Sign Convention is set to "Global +Y Loads act towards +Y (Upward)", then loads applied in the Global Y direction with a positive algebraic sign act upward, and loads applied in the Global Y direction with a negative algebraic sign act downward.
>
> When the Applied Global Y Load Sign Convention is set to "Global +Y Loads act towards -Y (Downward)", then loads applied in the Global Y direction with a positive algebraic sign act downward, and loads applied in the Global Y direction with a negative algebraic sign act upward.

In order to most efficiently describe the direction of application of the "Local" load types, it helps to refer to the member local axis system. Each member can be thought of as having its own x, y, and z coordinate axes that are mutually perpendicular and follow the right-hand rule. The orientation of the member local axes can be determined as follows:

1. The local x axis is always parallel to a vector from the "I" node to the "J" node. (This axis is parallel to the longitudinal axis of the member.)
2. The local z axis is always parallel to the Global Z axis.
3. The local y axis can be found by taking the vector cross product of local z cross local x. (Envision using the right hand to rotate the local z axis into the local x axis, and the right thumb will automatically indicate the positive direction of the local y axis.)

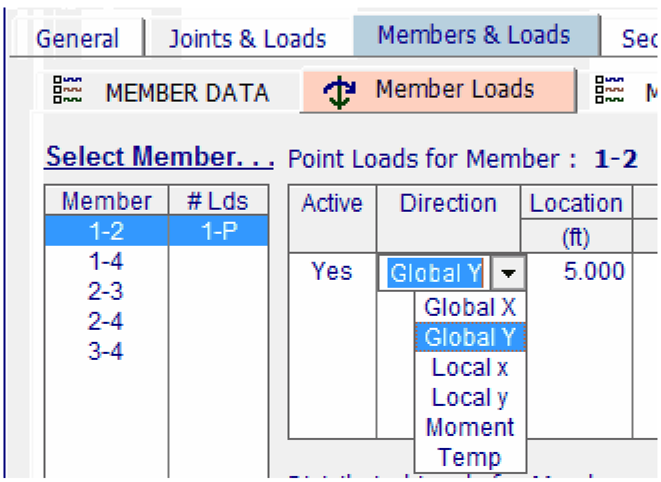
Local x: This point load acts parallel to the member's local x axis. If this load is specified with a positive magnitude, the load will act in the positive direction of the local x axis. If this load is specified with a negative magnitude, the load will act in the negative direction of the local x axis.

Local y: This point load acts parallel to the member's local y axis. If this load is specified with a positive magnitude, the load will act in the positive direction of the local y axis. If this load is specified with a negative magnitude, the load will act in the negative direction of the local y axis.

It should now be obvious that it is VERY important to have a thorough understanding of member orientation when using "Local" load types.

Moment: This specifies that the load is a concentrated moment. Positive moments follow the right-hand rule and apply a counter-clockwise rotational force to the member (when viewing the model from the positive Z direction).

Temperature: This is used to specify temperatures at particular locations along the length of a member. The module then uses these spot temperatures, in conjunction with the joint temperatures specified in the Joint Data list, to establish temperature gradients along the member(s). The module will calculate the effects of the specified temperature gradient from the end joint to the location along the member at which the temperature was specified. If you apply more temperature loads, the gradients are developed between each adjacent point of temperature load.



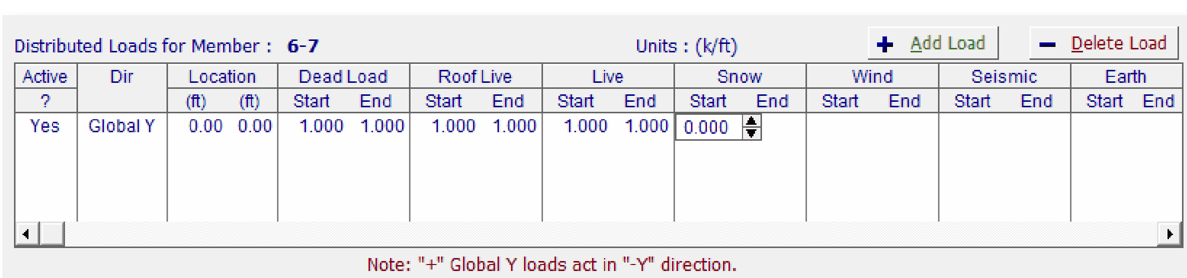
Location

This specifies the distance from the "I" joint where the point load is located.

Load Type & Magnitude

You can enter seven different types of loads on the frame and combine them using the factors on the Load Combinations tab.

Distributed Load Entry List

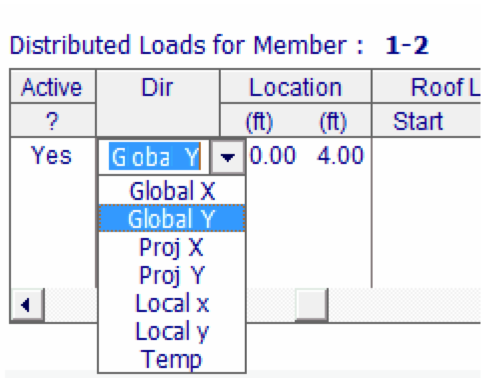


Active

This is a Yes/No checkbox that allows you to turn the load on and off. This is useful when you want to quickly see the effect of removing the load.

Direction

This specifies the direction of application of the load.



Here is a description of the different directions:

Global X: This load acts in a direction parallel to the Global X Axis and is distributed along the full length of the member. If the member is sloped, the load will be applied for the full length of the member. So if a member has a 10-foot rise and a 10-foot run, the length of the load will equal 14.14 feet. Entering a positive value will apply the load to the right (in the positive X direction).

Global Y: This load acts in a direction parallel to the Global Y Axis and is distributed along the full length of the member. If the member is sloped, the load will be applied for the full length of the member. So if a member has a 10-foot rise and a 10-foot run, the length of the load will equal 14.14 feet. The algebraic sign on the magnitude will affect the direction of application based on the Applied Global Y Load Sign Convention setting on the General tab as follows:

> When the Applied Global Y Load Sign Convention is set to "Global +Y Loads act towards +Y (Upward)", then loads applied in the Global Y direction with a positive algebraic sign act upward, and loads applied in the Global Y direction with a negative algebraic sign act downward.
>
> When the Applied Global Y Load Sign Convention is set to "Global +Y Loads act towards -Y (Downward)", then loads applied in the Global Y direction with a positive algebraic sign act downward, and loads applied in the Global Y direction with a negative algebraic sign act upward.

Projected X: This load acts in a direction parallel to the Global X Axis, but it is applied only to the length of the member projected onto the Global Y Axis. So if a member has a 10-foot rise and a 20-foot run, the length of the load will equal 10.00 feet (the rise of the member).

Projected Y: This load acts in a direction parallel to the Global Y Axis, but it is applied only to the length of the member projected onto the Global X axis. So if a member has a 10-foot rise and a 20-foot run, the length of the load will equal 20.00 feet (the run of the member).

In order to most efficiently describe the direction of application of the "Local" load types, it helps to refer to the member local axis system. Each member can be thought of as having its own x, y, and z coordinate axes that are mutually perpendicular and follow the right-hand rule. The orientation of the member local axes can be determined as follows:

1. The local x axis is always parallel to a vector from the "I" node to the "J" node. (This axis is parallel to the longitudinal axis of the member.)
2. The local z axis is always parallel to the Global Z axis.
3. The local y axis can be found by taking the vector cross product of local z cross local x. (Envision using the right hand to rotate the local z axis into the local x axis, and the right thumb will automatically indicate the positive direction of the local y axis.)

Local x: This distributed load acts parallel to the member's local x axis. If this load is specified with a positive magnitude, the load will act in the positive direction of the local x axis. If this load is specified with a negative magnitude, the load will act in the negative direction of the local x axis.

Local y: This distributed load acts parallel to the member's local y axis. If this load is specified with a positive magnitude, the load will act in the positive direction of the local y axis. If this load is specified with a negative magnitude, the load will act in the negative direction of the local y axis.

It should now be obvious that it is VERY important to have a thorough understanding of member orientation when using "Local" load types.

Temperature: This is used to specify temperatures at particular locations along the length of a member. The module then uses these spot temperatures, in conjunction with the joint temperatures specified in the Joint Data list, to establish temperature gradients along the member(s). The module will calculate the effects of the specified temperature gradient from the end joint to the location along the member at which the temperature was specified. If you apply more temperature loads, the gradients are developed between each adjacent point of temperature load.

Location - Start, End

This specifies the distance from the "I" joint to the beginning and end of the load. Leaving BOTH values as zero (0.0) will cause the load to be applied to the full length of the member.

Load Type & Magnitude

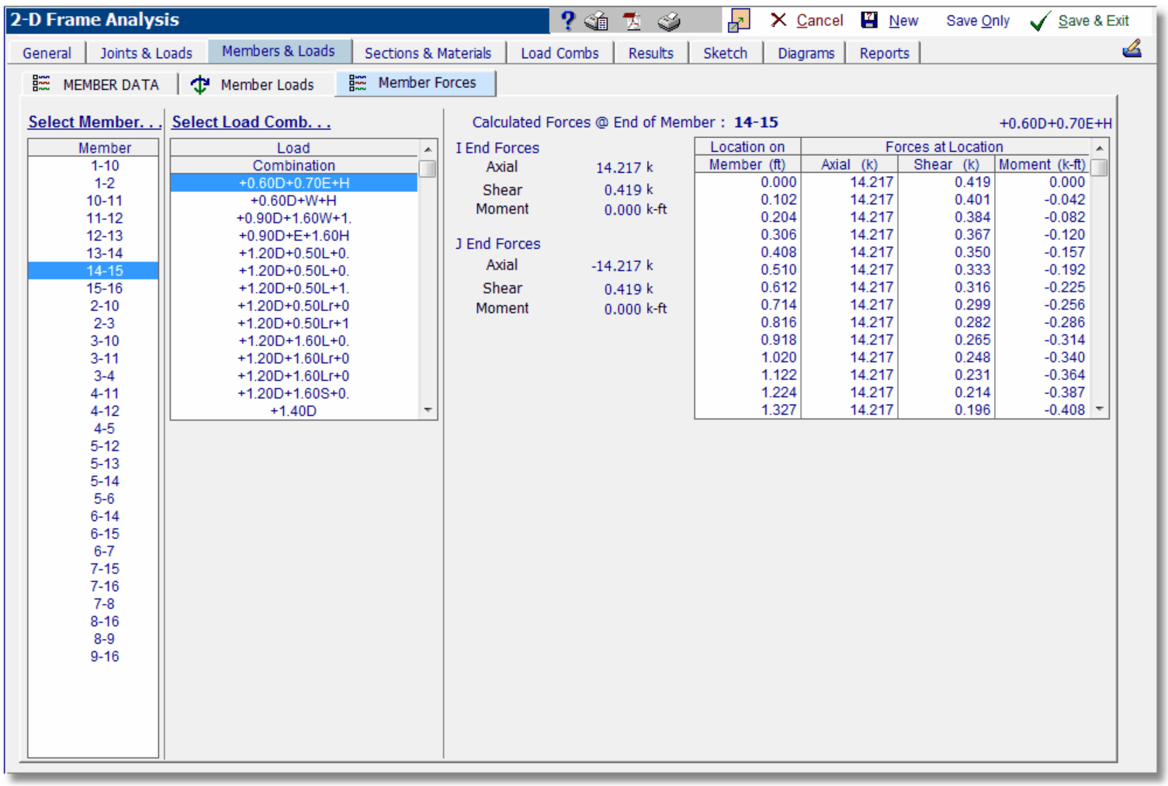
You can enter seven different types of loads on the frame and combine them using the values on the Load Combinations tab.

10.8.3.3 Member Forces

This tab allows you to review the final calculated forces for a member.

To review member forces, first click the member of interest in the Select Member list. Next, click on a load combination in the Select Load Comb. list. The member end forces will be displayed for the chosen load combination, and a table of member forces at increments along the length of the member will be shown.

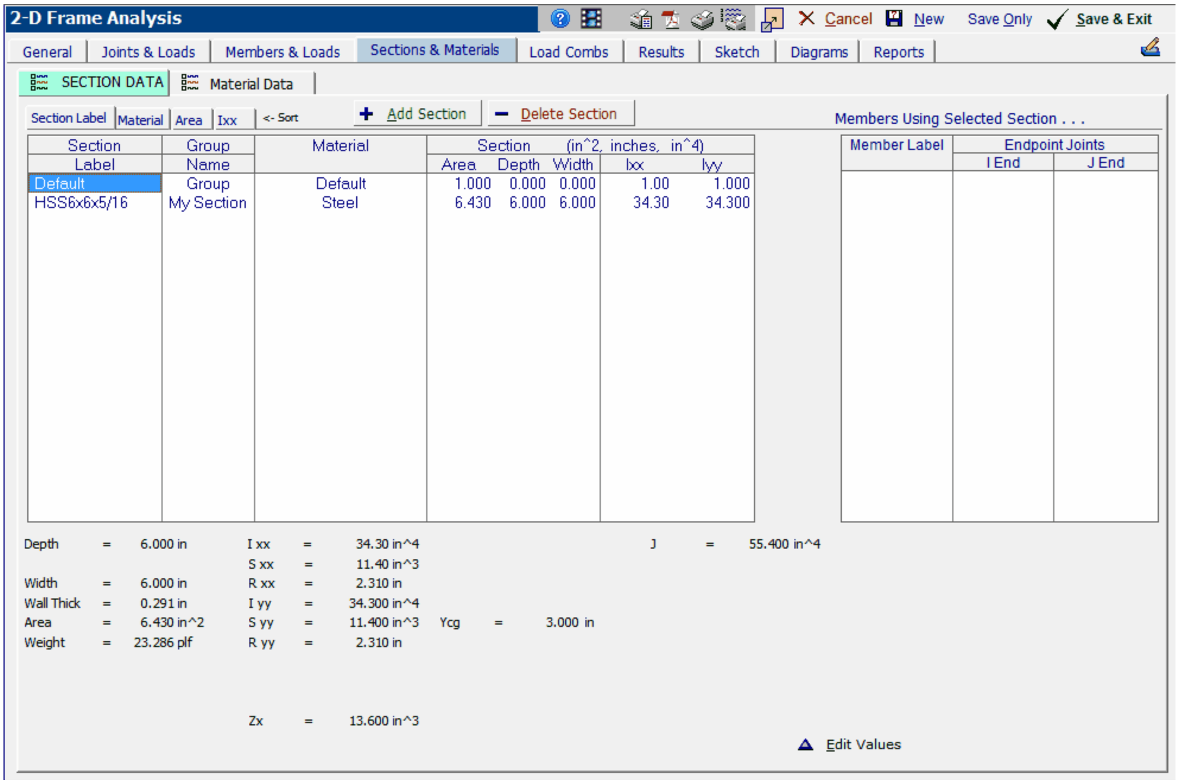
This tab provides a more simplified view of member forces than the large tables displayed on the Results tab.



10.8.4 Sections & Materials

[Need more? Ask Us a Question](#)

The two tabs under Sections & Materials provide the ability to define the section properties and material properties for the members used in the frame.



10.8.4.1 Section Data

The SECTION DATA tab allows you to specify sections to be used for the frame.

Section Label vs. Group Label

These two ways of labeling a section are very helpful and should be understood.

The Section Label is the actual name of the section, whether it is an AISC section name like W14x22 or a wood section like 4x10. The Section Label always represents something that can be retrieved from the internal databases.

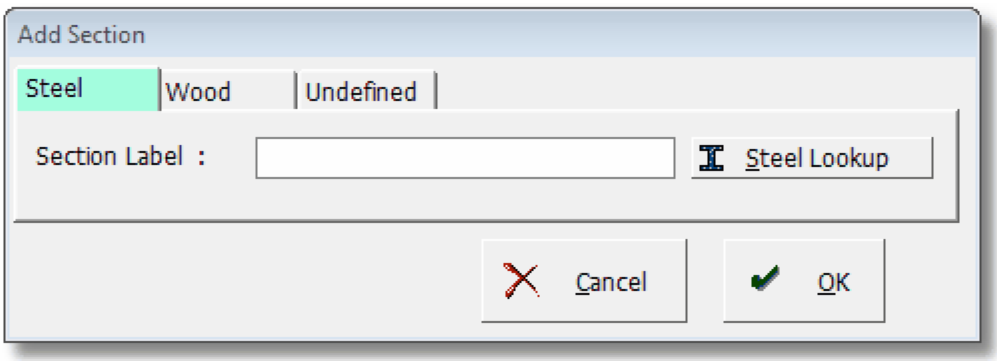
It can also be the name of a section that you create by entering a name and some properties.

The Group Label lets you associate a section name with a label that is meaningful to you and that can be used on multiple members. This makes it so you can easily revise the section that is assigned to a set of members without the need to change the section name individually for all the members where it is used.

For example, say that members 1,3,5,7,9,11,13,15 and 17 will all use the same section. You don't know which AISC section will be selected, but you DO know it will be the same section. You can assign a preliminary section name of HSS 3x3x1/4 and a group label of "diagonal_1". If the frame analysis shows that the HSS section fails, you can simply change it to a different section (with new properties). Because that section is linked to those members with the group label, it simplifies the section assignment and modification for the entire group.

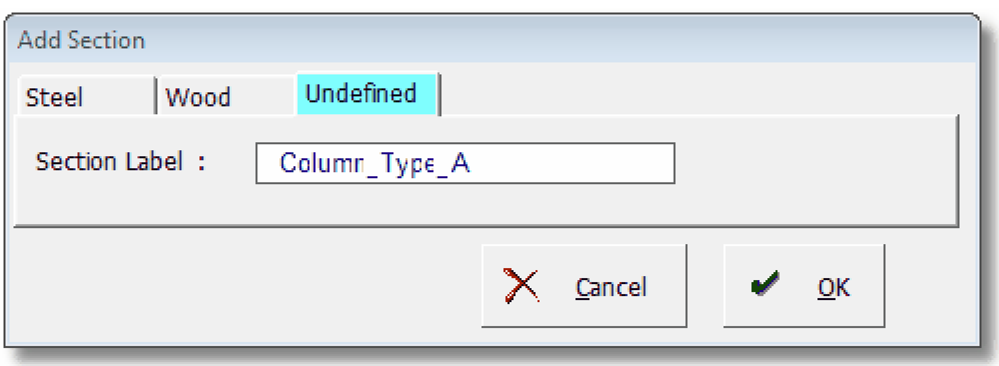
Adding and Deleting Sections

When you click the [**Add Section**] button, the following window appears:



To select a section from the built-in AISC or NDS databases, just click the appropriate tab and either type in the section name or click the [**Lookup**] button.

If you want to add your own section name and type in the properties, click the Undefined tab, type in the desired section name, and click [**OK**].



[**Delete Section**] will delete the section you've highlighted in the list. Any members using that section will be changed to reference the Default section.

Editing the List Columns

Section label

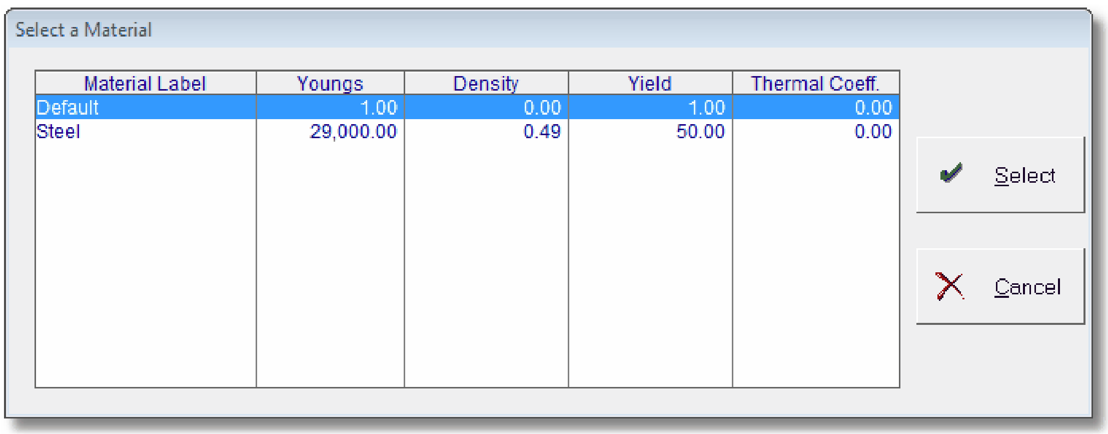
When you click on the entries in this column the entry position will change to a button labeled [**Lookup**]. This button provides access to the built-in section databases.

Group Label

When you click on the entries in this column the entry position changes to a text editing box.

Material

When you click on the entries in this column the entry position changes to a [**Lookup**] button. This button provides access to a list of the material properties that you have already defined on the Materials tab.



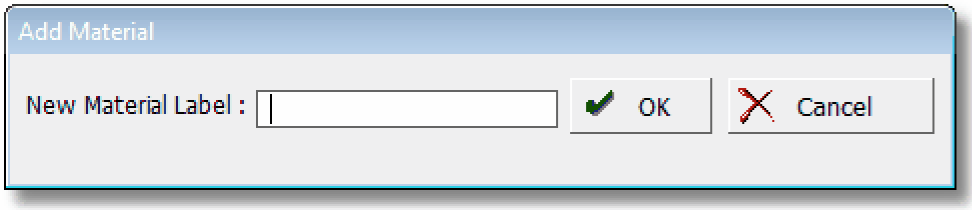
Section property values

These entry columns change to numeric entries when selected to allow entry of the values.

10.8.4.2 Material Data

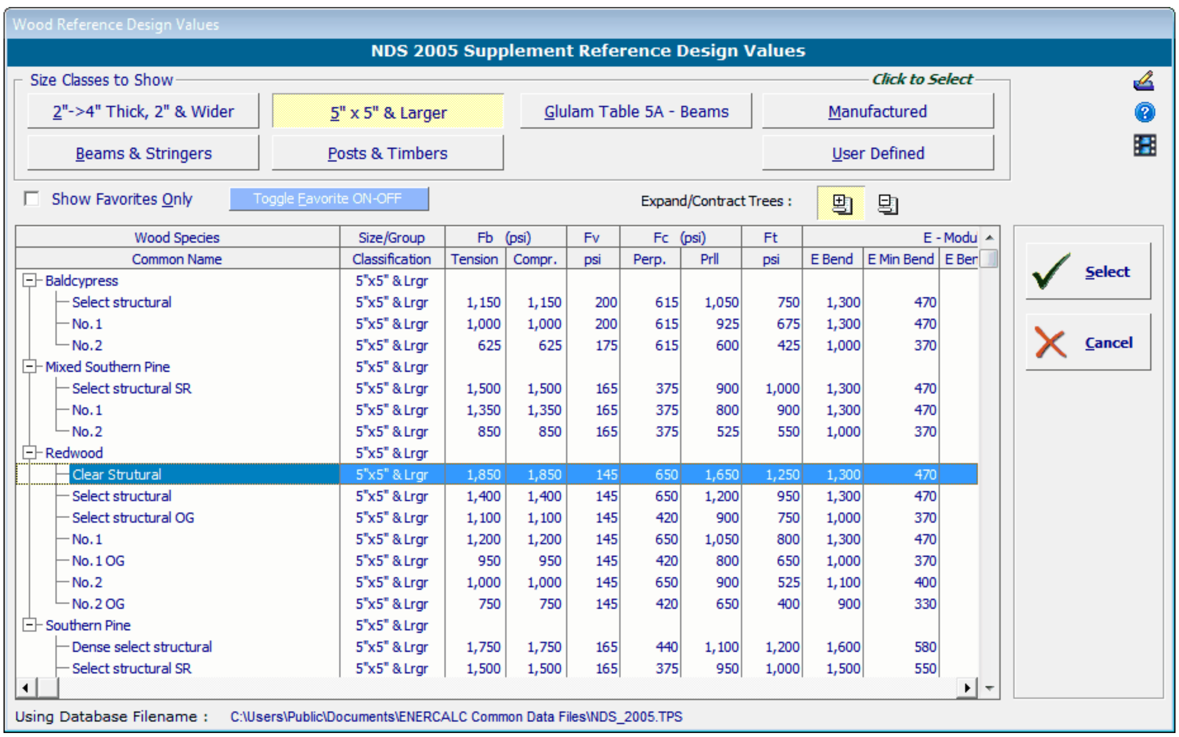
This screen provides the ability to define the material values used by the sections you define.

Clicking [**Add General Material**] displays the following input box for you to enter the material name:

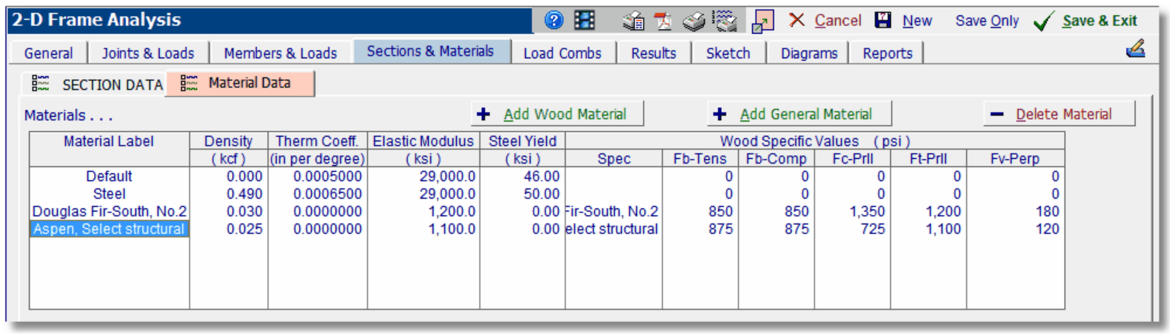


To add a new material, enter the name of the material and then click [**OK**]. The new material will appear in the Materials list on the Material Data tab.

Clicking [**Add Wood Material**] displays the Wood Reference Design Values database where you can select a wood species and grade combination:



To select a wood material to use in the current calculation, click on the desired material in the Wood Reference Design Values database and then click [**Select**]. The new wood material will appear in the Materials list on the Material Data tab as shown below:



Material Label	Density	Therm Coeff.	Elastic Modulus	Steel Yield	Wood Specific Values (psi)					
	(kcf)	(in per degree)	(ksi)	(ksi)	Spec	Fb-Tens	Fb-Comp	Fc-Prll	Ft-Prll	Fv-Perp
Default	0.000	0.0005000	29,000.0	46.00		0	0	0	0	0
Steel	0.490	0.0006500	29,000.0	50.00		0	0	0	0	0
Douglas Fir-South, No.2	0.030	0.0000000	1,200.0	0.00	Fir-South, No.2	850	850	1,350	1,200	180
Aspen, Select structural	0.025	0.0000000	1,100.0	0.00	elect structural	875	875	725	1,100	120

Clicking on the Elastic Modulus, Density, Yield Stress and Thermal Coefficient entry items will change them into numeric entry boxes where you can type in a value and press [**Tab**] or click off of the value to complete the data entry.

Elastic Modulus

The elastic modulus defines how the members will react to forces by defining the relationship between stress and strain.

Density

The density entry is used only when the module calculates and applies member self weights using the **Member Self Weight** entry column on the Load Combination tab.

Yield Stress

The use of this column and additional allowable stress information will be enhanced as the module matures. For steel members this property is used to perform the AISC member allowable stress evaluation. When wood stress evaluation is added, the module will store other pertinent values from the built-in database.

Thermal Coefficient

This defines the rate of thermal expansion per degree of temperature change. This value is only used when temperature loads are defined for a member.

Wood-Specific Values

The Wood-Specific Values are only populated when the selected material comes from the Wood Reference Design Values database.

10.8.5 Load Combinations

This data table controls how all of the loads are applied to the frame and also enables you to have member self weight loads automatically considered in specified directions.

The current list of load combinations comes from ASCE 7-05. It represents most of the commonly used load combinations suggested for both Allowable Stress Design and Strength Design. This table will be enhanced to provide more controlled specification of load combinations including templates for serviceability checks.

Auto-Minimize Combinations to Run

This checkbox instructs the module to eliminate load combinations that have a duplicate set of factored loads or specific combinations that include wind, seismic, earth, and snow load types when none of those load types is present in the model.

The purpose is simply to suppress duplicate result sets.

Add

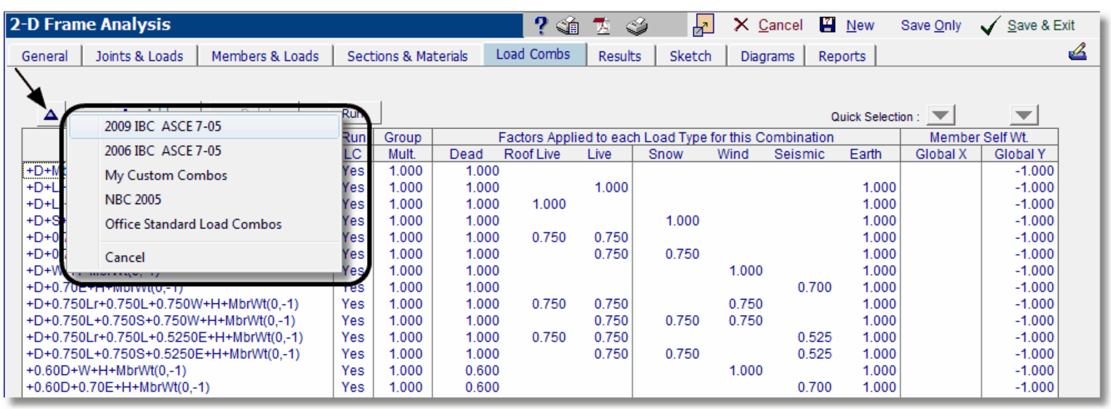
This button adds a load combination line to the list. You can then edit the factors.

Delete

This button deletes the currently selected load combination from the list.

(Load a new load combination set)

This button allow you to retrieve load combination sets from the built-in load combination database and place them in the list. Clicking it will display a popup menu listing the load combination sets currently included in the database.



Description of Items in List

Load Combination Name

This is the name of the load combination. It consists of abbreviations for each included load type along with the associated numeric values that represent the respective load factors. You cannot edit this name. It is constructed automatically based on the entries you make in the following columns.

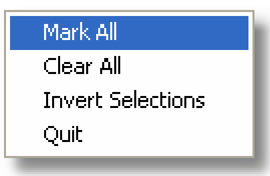
Note in the following image how the Load Combination Name is constructed from the specified values:

Load Combination Name	Run	Group	Factors Applied to each Load Type for this Combination							Member Self Wt.	
	LC	Mult.	Dead	Roof Live	Live	Snow	Wind	Seismic	Earth	Global X	Global Y
0.75(+D)+MbrWt(0,-1)	No	0.750	1.000								-1.000
+D+L+H+MbrWt(0,-1)	No	1.000	1.000		1.000				1.000		-1.000
+D+Lr+H+MbrWt(0,-1)	No	1.000	1.000	1.000					1.000		-1.000
+D+S+H+MbrWt(0,-1)	No	1.000	1.000			1.000			1.000		-1.000

Run LC

Clicking on this entry changes it to a checkbox, which controls whether that load combination is used or ignored.

This button allows you to change the Run setting for all the load combinations at once using several options.



Group Multiplier

This value acts as a multiplier on the entire load combination.

Lets say your values describe a combination of 1.0D + 0.7L + 0.3E and you want to reduce that entire combined loading by 50%. You would enter a value of 0.5 for the Group Multiplier. The resulting load combination would be 0.50 * (1.0D + 0.7L + 0.3E).

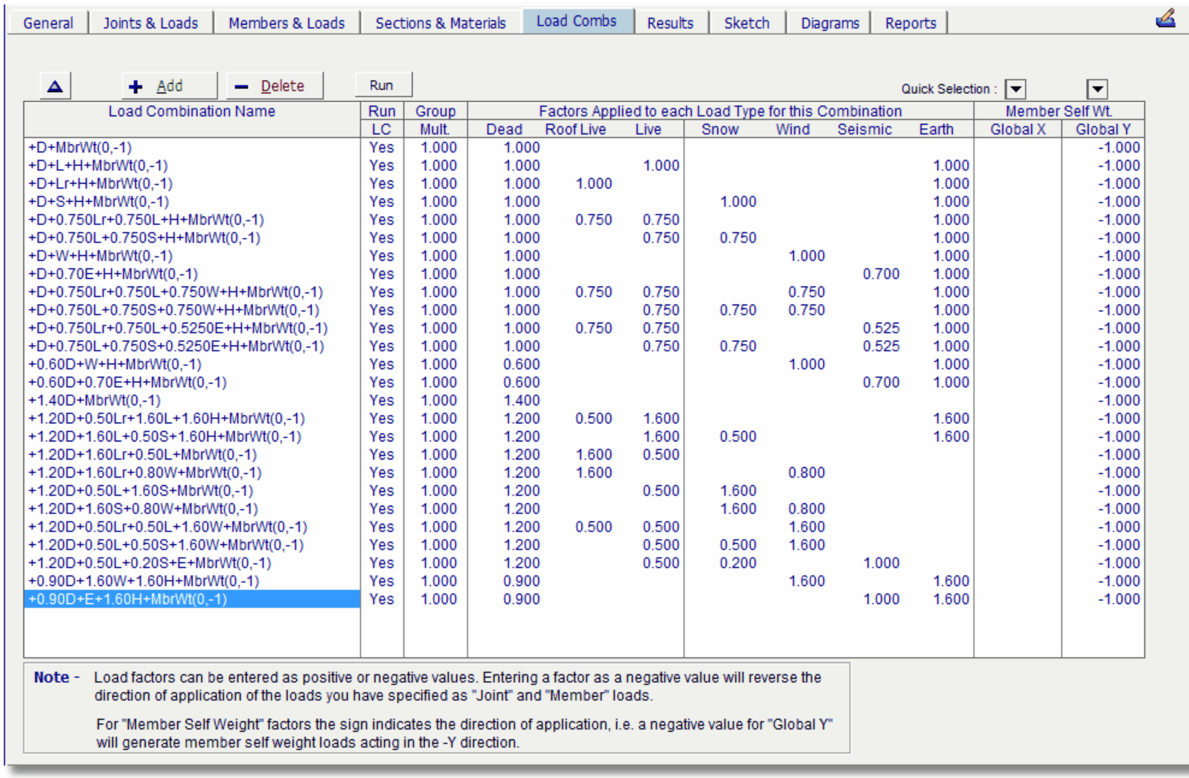
Individual Load Factors

Enter the numeric value to be applied to each type of load.

Member Self Weight

When either of these two values is non-zero, the module will calculate the weight of each member (as cross sectional area * density * length) and consider that weight during the analysis.

A "-1" value for "Global Y" will apply the member self weight downward (in the -Y direction). A value of "1" for "Global X" will have the member self weight applied to the right (the +X direction).



Load Combination Name	Run LC	Group Mult.	Factors Applied to each Load Type for this Combination: Dead	Roof Live	Live	Snow	Wind	Seismic	Earth	Member Self Wt. Global X	Member Self Wt. Global Y
+D+MbrWt(0,-1)	Yes	1.000	1.000								-1.000
+D+L+H+MbrWt(0,-1)	Yes	1.000	1.000		1.000				1.000		-1.000
+D+Lr+H+MbrWt(0,-1)	Yes	1.000	1.000	1.000					1.000		-1.000
+D+S+H+MbrWt(0,-1)	Yes	1.000	1.000			1.000			1.000		-1.000
+D+0.750Lr+0.750L+H+MbrWt(0,-1)	Yes	1.000	1.000	0.750	0.750				1.000		-1.000
+D+0.750L+0.750S+H+MbrWt(0,-1)	Yes	1.000	1.000		0.750	0.750			1.000		-1.000
+D+W+H+MbrWt(0,-1)	Yes	1.000	1.000				1.000		1.000		-1.000
+D+0.70E+H+MbrWt(0,-1)	Yes	1.000	1.000					0.700	1.000		-1.000
+D+0.750Lr+0.750L+0.750W+H+MbrWt(0,-1)	Yes	1.000	1.000	0.750	0.750		0.750		1.000		-1.000
+D+0.750L+0.750S+0.750W+H+MbrWt(0,-1)	Yes	1.000	1.000		0.750	0.750	0.750		1.000		-1.000
+D+0.750Lr+0.750L+0.5250E+H+MbrWt(0,-1)	Yes	1.000	1.000	0.750	0.750			0.525	1.000		-1.000
+D+0.750L+0.750S+0.5250E+H+MbrWt(0,-1)	Yes	1.000	1.000		0.750	0.750		0.525	1.000		-1.000
+0.60D+W+H+MbrWt(0,-1)	Yes	1.000	0.600				1.000		1.000		-1.000
+0.60D+0.70E+H+MbrWt(0,-1)	Yes	1.000	0.600					0.700	1.000		-1.000
+1.40D+MbrWt(0,-1)	Yes	1.000	1.400								-1.000
+1.20D+0.50Lr+1.60L+1.60H+MbrWt(0,-1)	Yes	1.000	1.200	0.500	1.600				1.600		-1.000
+1.20D+1.60L+0.50S+1.60H+MbrWt(0,-1)	Yes	1.000	1.200		1.600	0.500			1.600		-1.000
+1.20D+1.60Lr+0.50L+MbrWt(0,-1)	Yes	1.000	1.200	1.600	0.500						-1.000
+1.20D+1.60Lr+0.80W+MbrWt(0,-1)	Yes	1.000	1.200	1.600			0.800				-1.000
+1.20D+0.50L+1.60S+MbrWt(0,-1)	Yes	1.000	1.200		0.500	1.600					-1.000
+1.20D+1.60S+0.80W+MbrWt(0,-1)	Yes	1.000	1.200			1.600	0.800				-1.000
+1.20D+0.50Lr+0.50L+1.60W+MbrWt(0,-1)	Yes	1.000	1.200	0.500	0.500		1.600				-1.000
+1.20D+0.50L+0.50S+1.60W+MbrWt(0,-1)	Yes	1.000	1.200		0.500	0.500	1.600				-1.000
+1.20D+0.50L+0.20S+E+MbrWt(0,-1)	Yes	1.000	1.200		0.500	0.200		1.000			-1.000
+0.90D+1.60W+1.60H+MbrWt(0,-1)	Yes	1.000	0.900				1.600		1.600		-1.000
+0.90D+E+1.60H+MbrWt(0,-1)	Yes	1.000	0.900					1.000	1.600		-1.000

Note - Load factors can be entered as positive or negative values. Entering a factor as a negative value will reverse the direction of application of the loads you have specified as "Joint" and "Member" loads.

For "Member Self Weight" factors the sign indicates the direction of application, i.e. a negative value for "Global Y" will generate member self weight loads acting in the -Y direction.

10.8.6 Wood Design

The 2D Frame module now incorporates wood stress checking according based on ASD or LRFD methods according to NDS methods.

Adjustment Factors for Sawn Lumber

The module only collects C_D or Lambda from the user.

The module assumes a value of 1.0 for C_M, C_t, C_{fu}, C_i, C_r and C_T.

The module calculates values for C_L, C_F, and C_P.

The bearing area factor C_b never comes into play in the functions performed within this module.

Adjustment Factors for Glued Laminated Timber

The module only collects C_D or Lambda from the user.

The module assumes a value of 1.0 for C_M, C_t, C_{fu}, C_c, C_I, and C_{vr}.

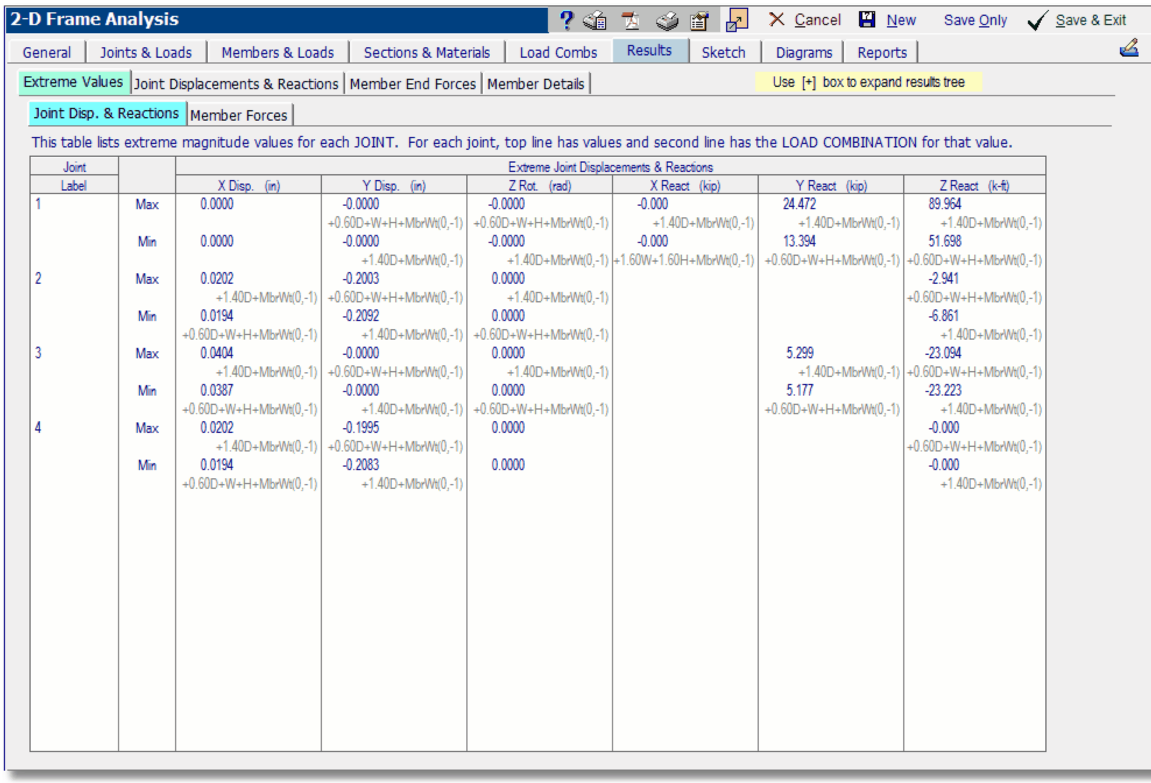
The module calculates values for C_L, C_V, and C_P.

The bearing area factor C_b never comes into play in the functions performed within this module.

10.8.7 Results

[Need more? Ask Us a Question](#)

The RESULTS tab provides extensive lists of detailed and summarized results from the frame analysis.

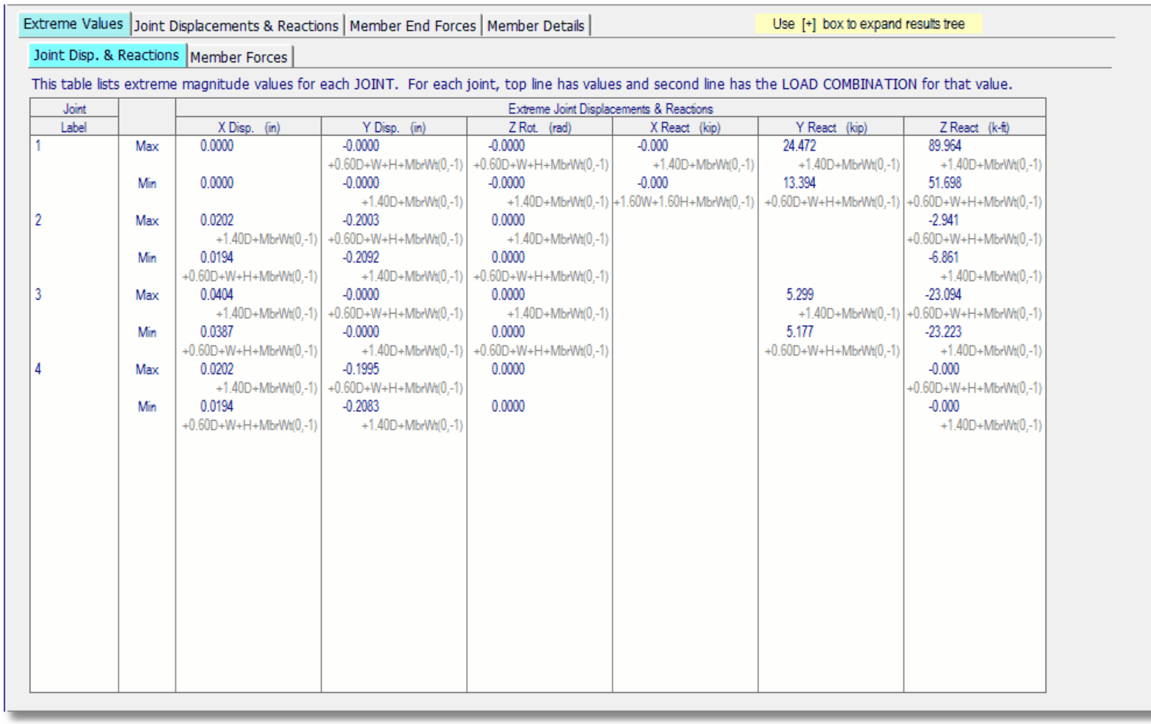


Joint Label		Extreme Joint Displacements & Reactions X Disp. (in)	Y Disp. (in)	Z Rot. (rad)	X React (kip)	Y React (kip)	Z React (k-ft)
1	Max	0.0000	-0.0000	-0.0000	-0.000	24.472	89.964
			+0.60D+W+H+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)
	Min	0.0000	-0.0000	-0.0000	-0.000	13.394	51.698
			+1.40D+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)	+1.60W+1.60H+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)
2	Max	0.0202	-0.2003	0.0000			-2.941
		+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)			+0.60D+W+H+MbrWt(0,-1)
	Min	0.0194	-0.2092	0.0000			-6.861
		+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)			+1.40D+MbrWt(0,-1)
3	Max	0.0404	-0.0000	0.0000		5.299	-23.094
		+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)		+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)
	Min	0.0387	-0.0000	0.0000		5.177	-23.223
		+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)		+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)
4	Max	0.0202	-0.1995	0.0000			-0.000
		+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)				+0.60D+W+H+MbrWt(0,-1)
	Min	0.0194	-0.2083	0.0000			-0.000
		+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)				+1.40D+MbrWt(0,-1)

10.8.7.1 Extreme Values

Joint Displacements & Reactions

This list summarizes the extreme joint displacements and support reactions. For each joint there is a four-line result display. The first line shows the joint label and the maximum values for each displacement and reaction direction. The second line indicates the load combinations that create the maximum values. The third line indicates the minimum (most negative) values for each displacement and reaction direction. The fourth line indicates the load combinations that create the minimum values.



Extreme Values | Joint Displacements & Reactions | Member End Forces | Member Details

Use [+] box to expand results tree

Joint Disp. & Reactions | Member Forces

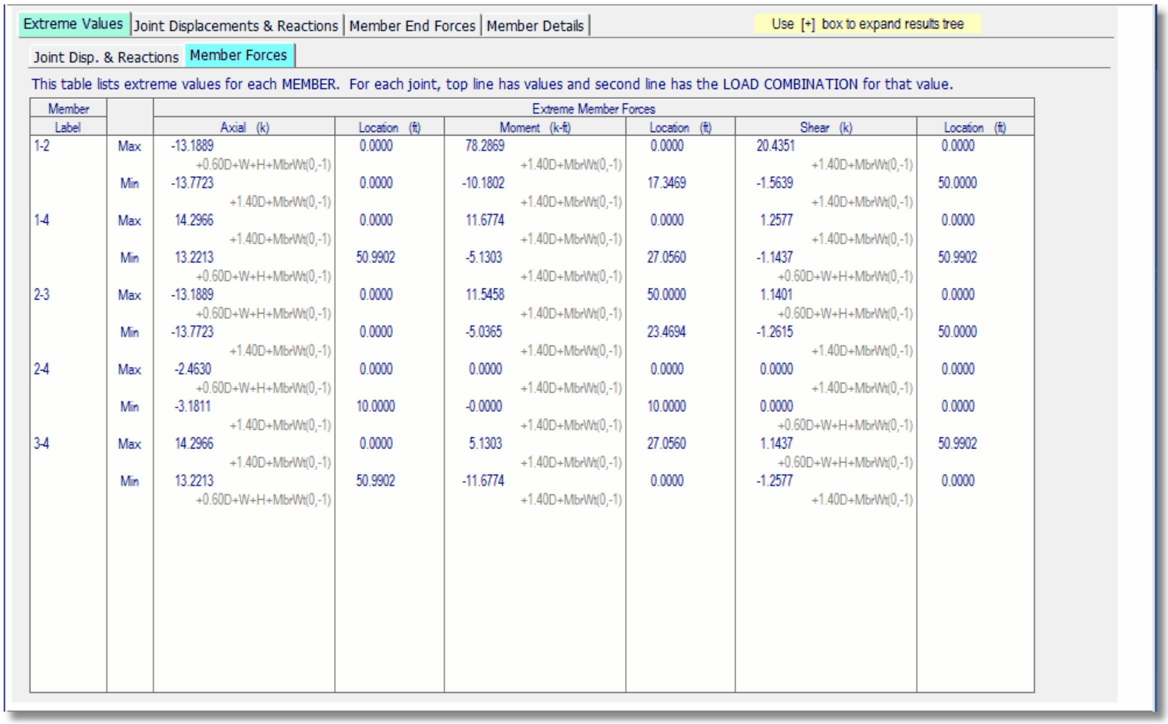
This table lists extreme magnitude values for each JOINT. For each joint, top line has values and second line has the LOAD COMBINATION for that value.

Joint		Extreme Joint Displacements & Reactions					
Label		X Disp. (in)	Y Disp. (in)	Z Rot. (rad)	X React (kip)	Y React (kip)	Z React (k-ft)
1	Max	0.0000	-0.0000	-0.0000	-0.000	24.472	89.964
			+0.60D+W+H+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)
	Min	0.0000	-0.0000	-0.0000	-0.000	13.394	51.698
			+1.40D+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)	+1.60W+1.60H+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)
2	Max	0.0202	-0.2003	0.0000			-2.941
		+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)			+0.60D+W+H+MbrWt(0,-1)
	Min	0.0194	-0.2092	0.0000			-6.861
		+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)			+1.40D+MbrWt(0,-1)
3	Max	0.0404	-0.0000	0.0000		5.299	-23.094
		+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)		+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)
	Min	0.0387	-0.0000	0.0000		5.177	-23.223
		+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)		+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)
4	Max	0.0202	-0.1995	0.0000			-0.000
		+1.40D+MbrWt(0,-1)	+0.60D+W+H+MbrWt(0,-1)				+0.60D+W+H+MbrWt(0,-1)
	Min	0.0194	-0.2083	0.0000			-0.000
		+0.60D+W+H+MbrWt(0,-1)	+1.40D+MbrWt(0,-1)				+1.40D+MbrWt(0,-1)

Member Forces

This table summarizes the extreme forces that occur anywhere along the length of each member. (The detailed forces ALONG the length of the member are given in another list within this Results section.)

For each member there is a four-line result display. The first line shows the member label and the maximum member force values that occur anywhere along the length of the member. The second line indicates the load combinations that create the maximum values. The third line indicates the minimum (most negative) member force values that occur anywhere along the length of the member. The fourth line indicates the load combinations that create the minimum values.



Extreme Values | Joint Displacements & Reactions | Member End Forces | Member Details

Use [+] box to expand results tree

Joint Disp. & Reactions | Member Forces

This table lists extreme values for each MEMBER. For each joint, top line has values and second line has the LOAD COMBINATION for that value.

Member Label		Extreme Member Forces Axial (k)	Location (ft)	Moment (k-ft)	Location (ft)	Shear (k)	Location (ft)
1-2	Max	-13.1889 +0.60D+W+H+MbrWt(0,-1)	0.0000	78.2869 +1.40D+MbrWt(0,-1)	0.0000	20.4351 +1.40D+MbrWt(0,-1)	0.0000
	Min	-13.7723 +1.40D+MbrWt(0,-1)	0.0000	-10.1802 +1.40D+MbrWt(0,-1)	17.3469	-1.5639 +1.40D+MbrWt(0,-1)	50.0000
1-4	Max	14.2966 +1.40D+MbrWt(0,-1)	0.0000	11.6774 +1.40D+MbrWt(0,-1)	0.0000	1.2577 +1.40D+MbrWt(0,-1)	0.0000
	Min	13.2213 +0.60D+W+H+MbrWt(0,-1)	50.9902	-5.1303 +1.40D+MbrWt(0,-1)	27.0560	-1.1437 +0.60D+W+H+MbrWt(0,-1)	50.9902
2-3	Max	-13.1889 +0.60D+W+H+MbrWt(0,-1)	0.0000	11.5458 +1.40D+MbrWt(0,-1)	50.0000	1.1401 +0.60D+W+H+MbrWt(0,-1)	0.0000
	Min	-13.7723 +1.40D+MbrWt(0,-1)	0.0000	-5.0365 +1.40D+MbrWt(0,-1)	23.4694	-1.2615 +1.40D+MbrWt(0,-1)	50.0000
2-4	Max	-2.4630 +0.60D+W+H+MbrWt(0,-1)	0.0000	0.0000 +1.40D+MbrWt(0,-1)	0.0000	0.0000 +1.40D+MbrWt(0,-1)	0.0000
	Min	-3.1811 +1.40D+MbrWt(0,-1)	10.0000	-0.0000 +1.40D+MbrWt(0,-1)	10.0000	0.0000 +0.60D+W+H+MbrWt(0,-1)	0.0000
3-4	Max	14.2966 +1.40D+MbrWt(0,-1)	0.0000	5.1303 +1.40D+MbrWt(0,-1)	27.0560	1.1437 +0.60D+W+H+MbrWt(0,-1)	50.9902
	Min	13.2213 +0.60D+W+H+MbrWt(0,-1)	50.9902	-11.6774 +1.40D+MbrWt(0,-1)	0.0000	-1.2577 +1.40D+MbrWt(0,-1)	0.0000

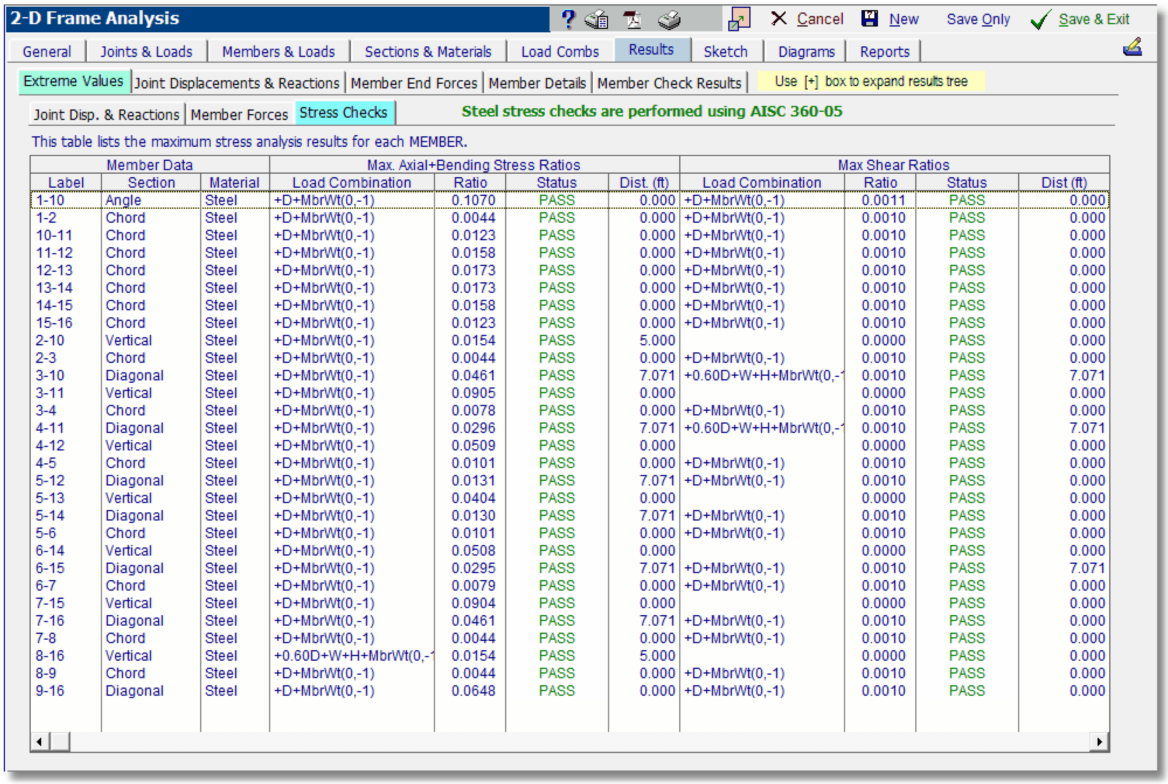
Stress Checks

The Stress Checks tab is only displayed when the Member Stress Check Status item on the General tab has been set to ASD or LRFD stress checks.

For each member this list shows the following information:

- member label
- section or group label
- material
- governing load combination that results in the maximum Axial plus Bending stress ratio
- maximum stress ratio for the Axial plus Bending check
- pass/fail status for the Axial plus Bending check
- location along the member length where the critical Axial plus Bending result was found to occur
- governing load combination that results in the maximum Shear stress ratio
- maximum stress ratio for the Shear check
- pass/fail status for the Shear check
- location along the member length where the critical Shear result was found to occur.

The image below shows the results for the steel members used in this frame:



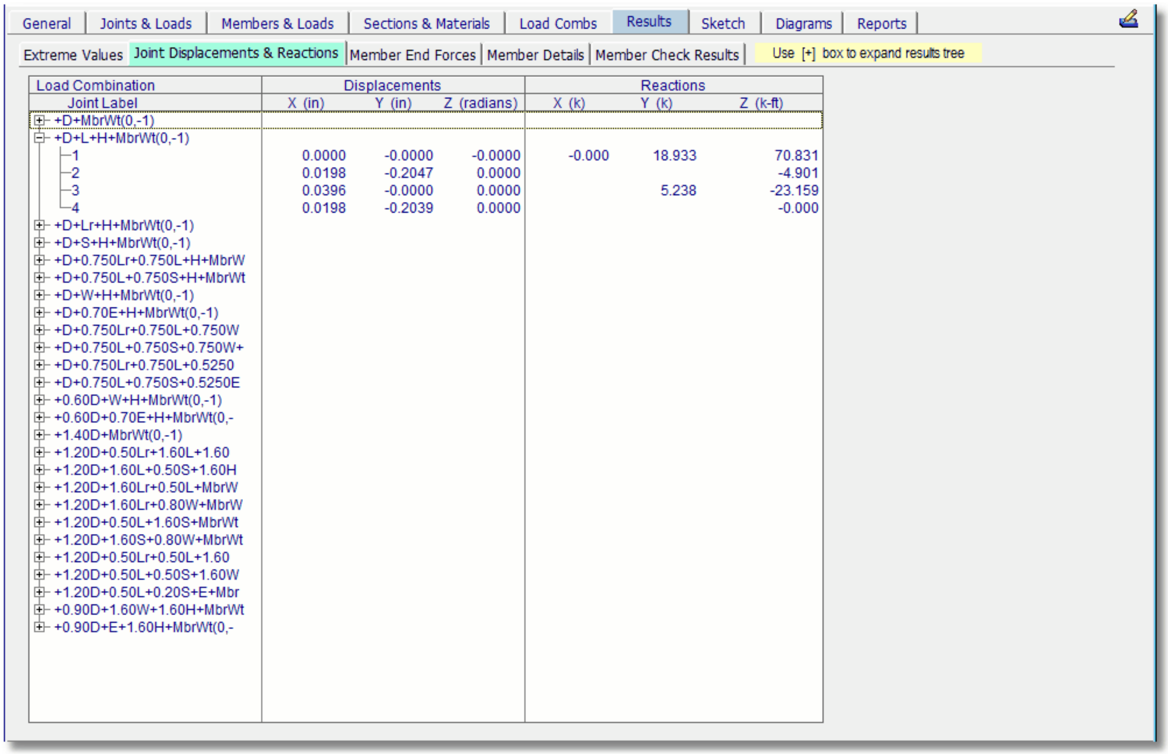
Member Data			Max. Axial+Bending Stress Ratios				Max Shear Ratios			
Label	Section	Material	Load Combination	Ratio	Status	Dist. (ft)	Load Combination	Ratio	Status	Dist (ft)
1-10	Angle	Steel	+D+MbrWt(0,-1)	0.1070	PASS	0.000	+D+MbrWt(0,-1)	0.0011	PASS	0.000
1-2	Chord	Steel	+D+MbrWt(0,-1)	0.0044	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
10-11	Chord	Steel	+D+MbrWt(0,-1)	0.0123	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
11-12	Chord	Steel	+D+MbrWt(0,-1)	0.0158	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
12-13	Chord	Steel	+D+MbrWt(0,-1)	0.0173	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
13-14	Chord	Steel	+D+MbrWt(0,-1)	0.0173	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
14-15	Chord	Steel	+D+MbrWt(0,-1)	0.0158	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
15-16	Chord	Steel	+D+MbrWt(0,-1)	0.0123	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
2-10	Vertical	Steel	+D+MbrWt(0,-1)	0.0154	PASS	5.000		0.0000	PASS	0.000
2-3	Chord	Steel	+D+MbrWt(0,-1)	0.0044	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
3-10	Diagonal	Steel	+D+MbrWt(0,-1)	0.0461	PASS	7.071	+0.60D+W+H+MbrWt(0,-1	0.0010	PASS	7.071
3-11	Vertical	Steel	+D+MbrWt(0,-1)	0.0905	PASS	0.000		0.0000	PASS	0.000
3-4	Chord	Steel	+D+MbrWt(0,-1)	0.0078	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
4-11	Diagonal	Steel	+D+MbrWt(0,-1)	0.0296	PASS	7.071	+0.60D+W+H+MbrWt(0,-1	0.0010	PASS	7.071
4-12	Vertical	Steel	+D+MbrWt(0,-1)	0.0509	PASS	0.000		0.0000	PASS	0.000
4-5	Chord	Steel	+D+MbrWt(0,-1)	0.0101	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
5-12	Diagonal	Steel	+D+MbrWt(0,-1)	0.0131	PASS	7.071	+D+MbrWt(0,-1)	0.0010	PASS	0.000
5-13	Vertical	Steel	+D+MbrWt(0,-1)	0.0404	PASS	0.000		0.0000	PASS	0.000
5-14	Diagonal	Steel	+D+MbrWt(0,-1)	0.0130	PASS	7.071	+D+MbrWt(0,-1)	0.0010	PASS	0.000
5-6	Chord	Steel	+D+MbrWt(0,-1)	0.0101	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
6-14	Vertical	Steel	+D+MbrWt(0,-1)	0.0508	PASS	0.000		0.0000	PASS	0.000
6-15	Diagonal	Steel	+D+MbrWt(0,-1)	0.0295	PASS	7.071	+D+MbrWt(0,-1)	0.0010	PASS	7.071
6-7	Chord	Steel	+D+MbrWt(0,-1)	0.0079	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
7-15	Vertical	Steel	+D+MbrWt(0,-1)	0.0904	PASS	0.000		0.0000	PASS	0.000
7-16	Diagonal	Steel	+D+MbrWt(0,-1)	0.0461	PASS	7.071	+D+MbrWt(0,-1)	0.0010	PASS	0.000
7-8	Chord	Steel	+D+MbrWt(0,-1)	0.0044	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
8-16	Vertical	Steel	+0.60D+W+H+MbrWt(0,-1	0.0154	PASS	5.000		0.0000	PASS	0.000
8-9	Chord	Steel	+D+MbrWt(0,-1)	0.0044	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000
9-16	Diagonal	Steel	+D+MbrWt(0,-1)	0.0648	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS	0.000

10.8.7.2 Joint Displacements & Reactions

This list summarizes the joint displacements resulting from each load combination.

Joint displacements are reported with respect to the global coordinate system.

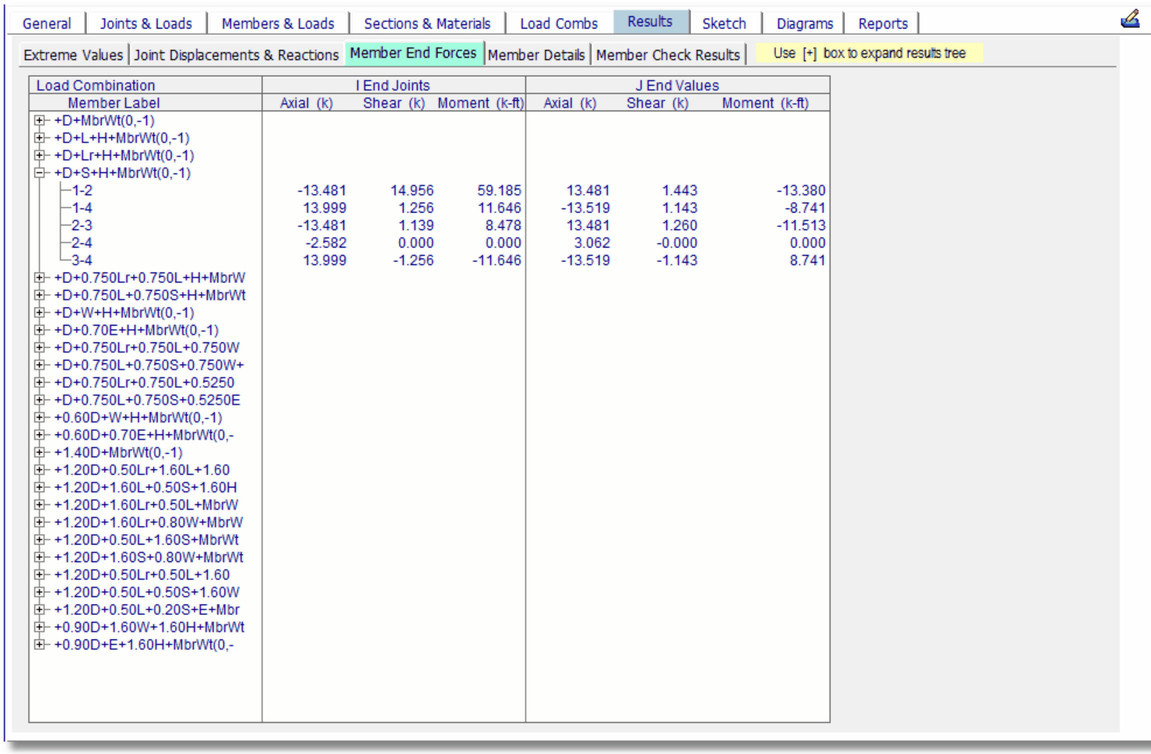
Clicking on the [**+**] icon to the left of each combination will expand the sub-list to show the detailed values.



10.8.7.3 Member End Forces

This list summarizes the member end forces resulting from each load combination.

Clicking on the [**+**] icon to the left of each combination will expand the sub-list to show the detailed values.



Load Combination / Member Label	I End Joints Axial (k)	Shear (k)	Moment (k-ft)	J End Values Axial (k)	Shear (k)	Moment (k-ft)
+D+MbrWt(0,-1)						
+D+L+H+MbrWt(0,-1)						
+D+Lr+H+MbrWt(0,-1)						
+D+S+H+MbrWt(0,-1)						
1-2	-13.481	14.956	59.185	13.481	1.443	-13.380
1-4	13.999	1.256	11.646	-13.519	1.143	-8.741
2-3	-13.481	1.139	8.478	13.481	1.260	-11.513
2-4	-2.582	0.000	0.000	3.062	-0.000	0.000
3-4	13.999	-1.256	-11.646	-13.519	-1.143	8.741
+D+0.750Lr+0.750L+H+MbrW						
+D+0.750L+0.750S+H+MbrWt						
+D+W+H+MbrWt(0,-1)						
+D+0.70E+H+MbrWt(0,-1)						
+D+0.750Lr+0.750L+0.750W						
+D+0.750L+0.750S+0.750W+						
+D+0.750Lr+0.750L+0.5250						
+D+0.750L+0.750S+0.5250E						
+0.60D+W+H+MbrWt(0,-1)						
+0.60D+0.70E+H+MbrWt(0,-						
+1.40D+MbrWt(0,-1)						
+1.20D+0.50Lr+1.60L+1.60						
+1.20D+1.60L+0.50S+1.60H						
+1.20D+1.60Lr+0.50L+MbrW						
+1.20D+1.60Lr+0.80W+MbrW						
+1.20D+0.50L+1.60S+MbrWt						
+1.20D+1.60S+0.80W+MbrWt						
+1.20D+0.50Lr+0.50L+1.60						
+1.20D+0.50L+0.50S+1.60W						
+1.20D+0.50L+0.20S+E+Mbr						
+0.90D+1.60W+1.60H+MbrWt						
+0.90D+E+1.60H+MbrWt(0,-						

10.8.7.4 Member Details

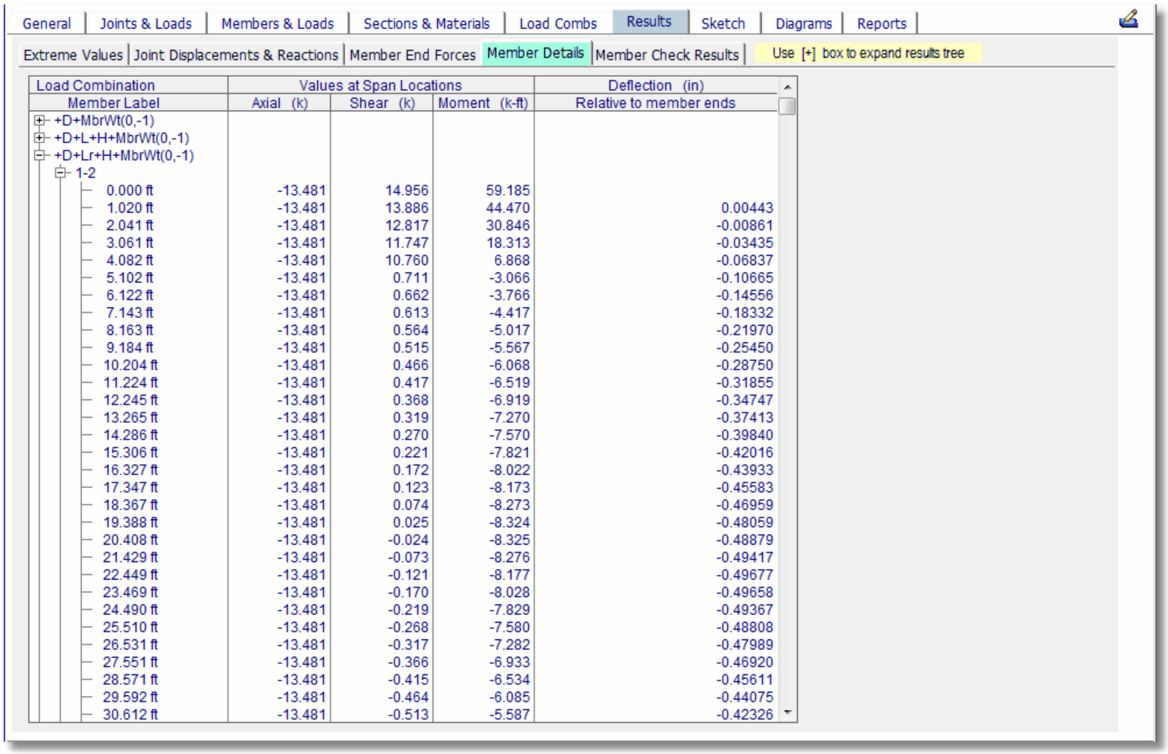
This list gives a very detailed presentation of the member forces and member deflections at small increments along the member length.

Note that the Member Deflections provided in this list are reported relative to the straight-line chord drawn between the *deflected* position of the two end nodes of the member. In other words, these Member Deflections will ALWAYS report a value of zero at both ends of all members.

The list is a tree with two sub-levels:

- The main level allows a choice of the load combination.
- The next level down offers the choice of which member to observe.

Click on the [**+**] icons to expand the list to show more details. Click on the [**-**] icon will compress the level.



Load Combination	Values at Span Locations			Deflection (in)
Member Label	Axial (k)	Shear (k)	Moment (k-ft)	Relative to member ends
+D+MbrWt(0,-1)				
+D+L+H+MbrWt(0,-1)				
+D+Lr+H+MbrWt(0,-1)				
1-2				
0.000 ft	-13.481	14.956	59.185	
1.020 ft	-13.481	13.886	44.470	0.00443
2.041 ft	-13.481	12.817	30.846	-0.00861
3.061 ft	-13.481	11.747	18.313	-0.03435
4.082 ft	-13.481	10.760	6.868	-0.06837
5.102 ft	-13.481	0.711	-3.066	-0.10665
6.122 ft	-13.481	0.662	-3.766	-0.14556
7.143 ft	-13.481	0.613	-4.417	-0.18332
8.163 ft	-13.481	0.564	-5.017	-0.21970
9.184 ft	-13.481	0.515	-5.567	-0.25450
10.204 ft	-13.481	0.466	-6.068	-0.28750
11.224 ft	-13.481	0.417	-6.519	-0.31855
12.245 ft	-13.481	0.368	-6.919	-0.34747
13.265 ft	-13.481	0.319	-7.270	-0.37413
14.286 ft	-13.481	0.270	-7.570	-0.39840
15.306 ft	-13.481	0.221	-7.821	-0.42016
16.327 ft	-13.481	0.172	-8.022	-0.43933
17.347 ft	-13.481	0.123	-8.173	-0.45583
18.367 ft	-13.481	0.074	-8.273	-0.46959
19.388 ft	-13.481	0.025	-8.324	-0.48059
20.408 ft	-13.481	-0.024	-8.325	-0.48879
21.429 ft	-13.481	-0.073	-8.276	-0.49417
22.449 ft	-13.481	-0.121	-8.177	-0.49677
23.469 ft	-13.481	-0.170	-8.028	-0.49658
24.490 ft	-13.481	-0.219	-7.829	-0.49367
25.510 ft	-13.481	-0.268	-7.580	-0.48808
26.531 ft	-13.481	-0.317	-7.282	-0.47989
27.551 ft	-13.481	-0.366	-6.933	-0.46920
28.571 ft	-13.481	-0.415	-6.534	-0.45611
29.592 ft	-13.481	-0.464	-6.085	-0.44075
30.612 ft	-13.481	-0.513	-5.587	-0.42326

10.8.7.5 Member Check Results

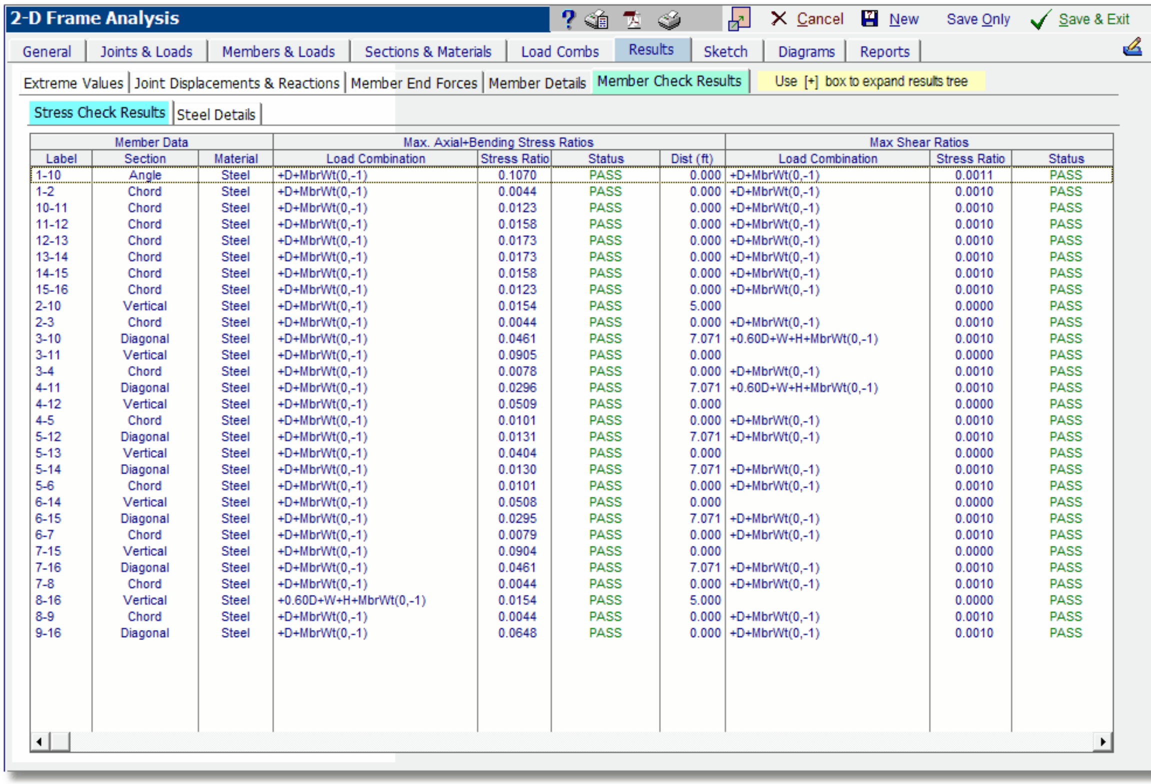
The Member Check Results tab is only shown when the Member Stress Check Status item on the General tab has been set to ASD or LRFD stress checks.

Stress Check Results

For each member this list shows the following information:

- member label
- section or group label
- material
- governing load combination that results in the maximum stress ratios
- maximum stress ratio for the Axial plus Bending check
- pass/fail status for the Axial plus Bending check
- location along the member length where the critical Axial plus Bending result was found to occur
- maximum stress ratio for the Shear check
- pass/fail status for the Shear check
- location along the member length where the critical Shear result was found to occur.

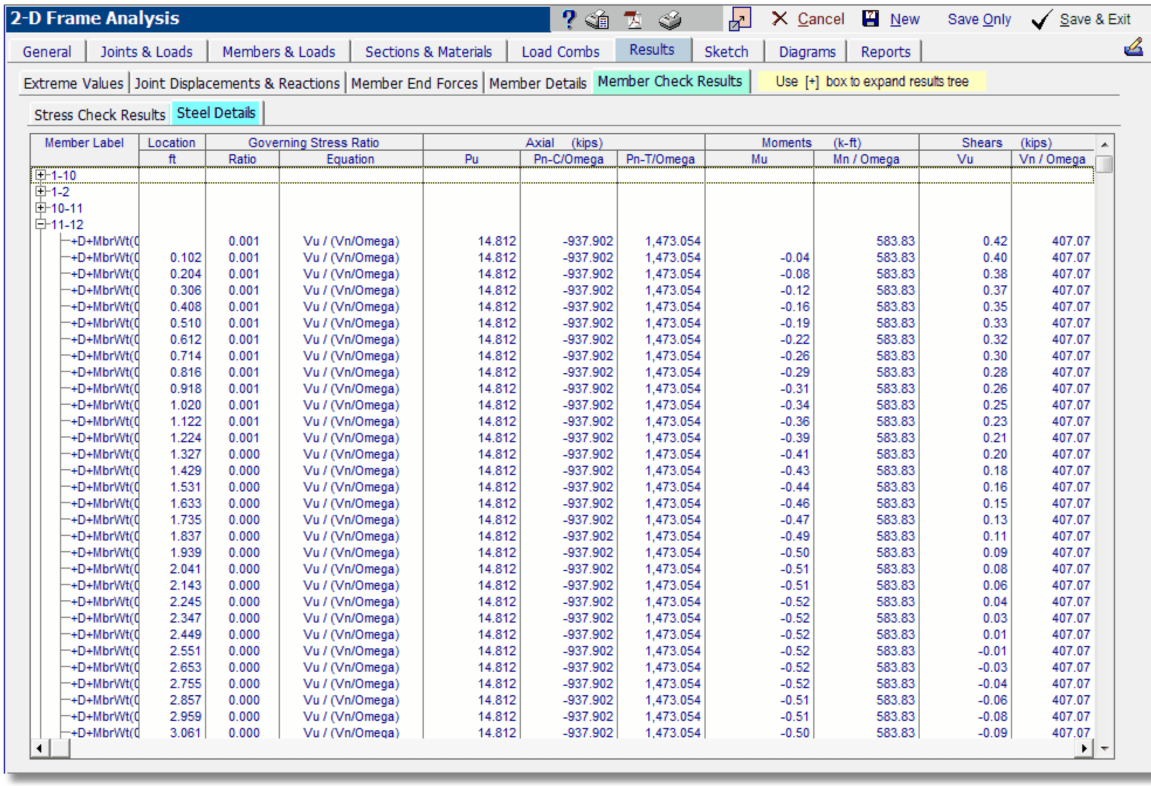
The image below shows the results for the steel members used in this frame:



Member Data			Max. Axial+Bending Stress Ratios				Max Shear Ratios		
Label	Section	Material	Load Combination	Stress Ratio	Status	Dist (ft)	Load Combination	Stress Ratio	Status
1-10	Angle	Steel	+D+MbrWt(0,-1)	0.1070	PASS	0.000	+D+MbrWt(0,-1)	0.0011	PASS
1-2	Chord	Steel	+D+MbrWt(0,-1)	0.0044	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
10-11	Chord	Steel	+D+MbrWt(0,-1)	0.0123	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
11-12	Chord	Steel	+D+MbrWt(0,-1)	0.0158	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
12-13	Chord	Steel	+D+MbrWt(0,-1)	0.0173	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
13-14	Chord	Steel	+D+MbrWt(0,-1)	0.0173	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
14-15	Chord	Steel	+D+MbrWt(0,-1)	0.0158	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
15-16	Chord	Steel	+D+MbrWt(0,-1)	0.0123	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
2-10	Vertical	Steel	+D+MbrWt(0,-1)	0.0154	PASS	5.000		0.0000	PASS
2-3	Chord	Steel	+D+MbrWt(0,-1)	0.0044	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
3-10	Diagonal	Steel	+D+MbrWt(0,-1)	0.0461	PASS	7.071	+0.60D+W+H+MbrWt(0,-1)	0.0010	PASS
3-11	Vertical	Steel	+D+MbrWt(0,-1)	0.0905	PASS	0.000		0.0000	PASS
3-4	Chord	Steel	+D+MbrWt(0,-1)	0.0078	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
4-11	Diagonal	Steel	+D+MbrWt(0,-1)	0.0296	PASS	7.071	+0.60D+W+H+MbrWt(0,-1)	0.0010	PASS
4-12	Vertical	Steel	+D+MbrWt(0,-1)	0.0509	PASS	0.000		0.0000	PASS
4-5	Chord	Steel	+D+MbrWt(0,-1)	0.0101	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
5-12	Diagonal	Steel	+D+MbrWt(0,-1)	0.0131	PASS	7.071	+D+MbrWt(0,-1)	0.0010	PASS
5-13	Vertical	Steel	+D+MbrWt(0,-1)	0.0404	PASS	0.000		0.0000	PASS
5-14	Diagonal	Steel	+D+MbrWt(0,-1)	0.0130	PASS	7.071	+D+MbrWt(0,-1)	0.0010	PASS
5-6	Chord	Steel	+D+MbrWt(0,-1)	0.0101	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
6-14	Vertical	Steel	+D+MbrWt(0,-1)	0.0508	PASS	0.000		0.0000	PASS
6-15	Diagonal	Steel	+D+MbrWt(0,-1)	0.0295	PASS	7.071	+D+MbrWt(0,-1)	0.0010	PASS
6-7	Chord	Steel	+D+MbrWt(0,-1)	0.0079	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
7-15	Vertical	Steel	+D+MbrWt(0,-1)	0.0904	PASS	0.000		0.0000	PASS
7-16	Diagonal	Steel	+D+MbrWt(0,-1)	0.0461	PASS	7.071	+D+MbrWt(0,-1)	0.0010	PASS
7-8	Chord	Steel	+D+MbrWt(0,-1)	0.0044	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
8-16	Vertical	Steel	+0.60D+W+H+MbrWt(0,-1)	0.0154	PASS	5.000		0.0000	PASS
8-9	Chord	Steel	+D+MbrWt(0,-1)	0.0044	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS
9-16	Diagonal	Steel	+D+MbrWt(0,-1)	0.0648	PASS	0.000	+D+MbrWt(0,-1)	0.0010	PASS

Steel Details

This list pertains only to the stress checks for steel members, and it provides detailed information about the stress checks at small increments along the length of each member.



Member Label	Location	Governing Stress Ratio		Axial (kips)			Moments (k-ft)		Shears (kips)	
	ft	Ratio	Equation	Pu	Pn-C/Omega	Pn-T/Omega	Mu	Mn / Omega	Vu	Vn / Omega
⊞1-10										
⊞1-2										
⊞10-11										
⊟11-12										
+D+MbrWt(		0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054		583.83	0.42	407.07
+D+MbrWt(	0.102	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.04	583.83	0.40	407.07
+D+MbrWt(	0.204	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.08	583.83	0.38	407.07
+D+MbrWt(	0.306	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.12	583.83	0.37	407.07
+D+MbrWt(	0.408	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.16	583.83	0.35	407.07
+D+MbrWt(	0.510	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.19	583.83	0.33	407.07
+D+MbrWt(	0.612	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.22	583.83	0.32	407.07
+D+MbrWt(	0.714	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.26	583.83	0.30	407.07
+D+MbrWt(	0.816	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.29	583.83	0.28	407.07
+D+MbrWt(	0.918	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.31	583.83	0.26	407.07
+D+MbrWt(	1.020	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.34	583.83	0.25	407.07
+D+MbrWt(	1.122	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.36	583.83	0.23	407.07
+D+MbrWt(	1.224	0.001	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.39	583.83	0.21	407.07
+D+MbrWt(	1.327	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.41	583.83	0.20	407.07
+D+MbrWt(	1.429	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.43	583.83	0.18	407.07
+D+MbrWt(	1.531	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.44	583.83	0.16	407.07
+D+MbrWt(	1.633	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.46	583.83	0.15	407.07
+D+MbrWt(	1.735	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.47	583.83	0.13	407.07
+D+MbrWt(	1.837	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.49	583.83	0.11	407.07
+D+MbrWt(	1.939	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.50	583.83	0.09	407.07
+D+MbrWt(	2.041	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.51	583.83	0.08	407.07
+D+MbrWt(	2.143	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.51	583.83	0.06	407.07
+D+MbrWt(	2.245	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.52	583.83	0.04	407.07
+D+MbrWt(	2.347	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.52	583.83	0.03	407.07
+D+MbrWt(	2.449	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.52	583.83	0.01	407.07
+D+MbrWt(	2.551	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.52	583.83	-0.01	407.07
+D+MbrWt(	2.653	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.52	583.83	-0.03	407.07
+D+MbrWt(	2.755	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.52	583.83	-0.04	407.07
+D+MbrWt(	2.857	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.51	583.83	-0.06	407.07
+D+MbrWt(	2.959	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.51	583.83	-0.08	407.07
+D+MbrWt(	3.061	0.000	Vu / (Vn/Omega)	14.812	-937.902	1,473.054	-0.50	583.83	-0.09	407.07

10.8.7.6 Sign Convention

Coordinate Axis Convention

The right-hand rule states that if you take the vector cross product of X cross Y, the result is in the Z direction. This is what is used to establish the positive Z direction if you know the positive X and Y directions. It applies to the Global Coordinate Axis system and the member local coordinate axis system in the 2-D Frame Analysis module, in the 2-D Frame Analysis module.

The Global Coordinate Axis system is oriented such that X points to the right, Y points upward, and Z is perpendicular to the screen in the 2-D Frame Analysis module.

The member local coordinate axis system is established as follows. A vector from the I node to the J node establishes the member local x axis. The vector cross product of local x cross Global Y produces the member local z direction. This works for all member orientations except for vertically oriented members, because it is not possible to take the vector cross product of two parallel vectors. So in those cases the module adopts the convention that the member local z axis will be parallel to the Global Z axis. These are just mathematical rules that establish that member local z will be perpendicular to the screen (unless you have specified that the member is working in weak axis bending). The local z axis is perpendicular to the web of a wide flange section. It typically represents the “strong” axis of a member. Finally, we need to establish the orientation of the local y axis. Vector cross product rules for a right-hand coordinate system also state that local z cross local x produces local y. So this pins down the orientation of the local y axis. The local y axis of a wide flange member lies in the plane of the web and is mutually perpendicular to the local x and local z axes. For a horizontally oriented beam member, the local y axis points straight up, parallel to the Global Y axis.

Result Sign Convention

Now for the sign conventions of the various results that are available.

Joint Displacements

Joint Displacements are reported with respect to the Global Coordinate Axis system. A positive displacement indicates a displacement in the direction of the positive corresponding axis. A positive rotation indicates a positive rotation about the Global Z axis. (Using the thumb of the right hand, point the thumb in the direction of the positive Global Z axis, and the natural curl of the right fingers will indicate the direction of a positive rotation.)

Reactions

Reactions are also reported with respect to the Global Coordinate Axis system. A positive force reaction indicates a force in the direction of the positive corresponding axis. A positive moment reaction indicates a positive moment about the Global Z axis. (Using the thumb of the right hand, point the thumb in the direction of the positive Global Z axis, and the natural curl of the right fingers will indicate the direction of a positive moment.)

Member End Forces

Member End Forces are reported with respect to the member local coordinate axis system. A positive value of axial load at the I end of the member means that the force acts in the member local x direction, so it is pushing into the starting end of the member, so it represents a compressive force. Likewise, a negative value of axial load at the J end of the member means that the force acts in the member local -x direction, so it is pushing into the ending end of the member, so again this represents a compressive force.

Shears are also reported with respect to the member local coordinate axis system. A positive value of shear at the I end of the member means that the force acts in the member local y direction. Likewise, a negative value of shear at the J end of the member means that the force acts in the member local -y direction.

A positive value of moment at either end of the member means that the moment acts in the member local z direction. (Using the thumb of the right hand, point the thumb in the direction of the positive Global Z axis, and the natural curl of the right fingers will indicate the direction of a positive moment.)

Member Forces at Sections

The physical sense of the member forces at sections can best be established by relating to the physical sense of the member end forces at the starting end of the member as described above.

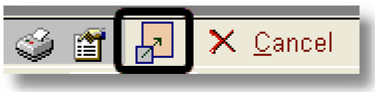
Deflections (Relative to member ends)

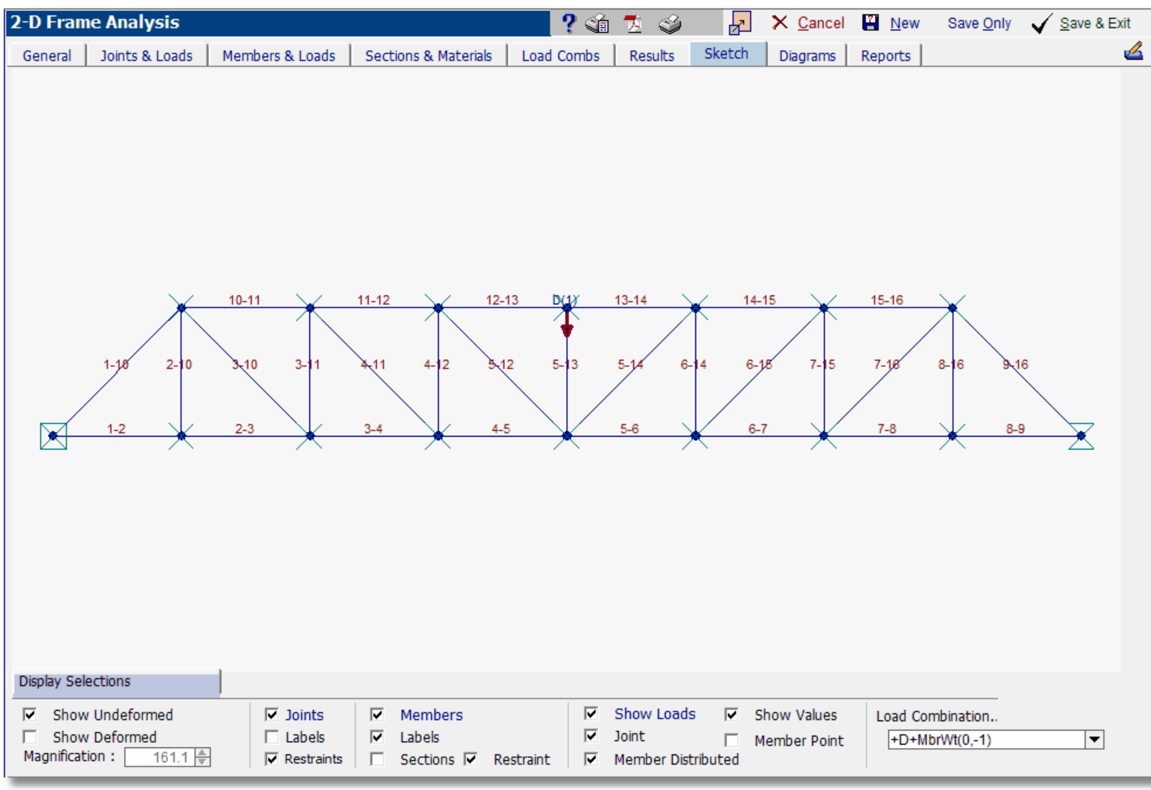
The deflections relative to member ends are measured parallel to the member local y axis and are referenced from the straight-line chord connecting the undeflected end node locations. Positive values represent deflections in the positive local y axis direction from that straight-line chord.

10.8.8 Frame Sketch

This tab provides a graphical display of the frame. You can use various check boxes, displacement magnification and load combination selection items to control the display.

You can quickly display a sketch of the frame when in another tab by hovering the cursor over the double rectangle icon in the upper right corner of the window. See small image below:





10.8.9 Member Diagrams

This tab allows you to display axial load, moment, shear and deflection diagrams for each individual member in the frame.

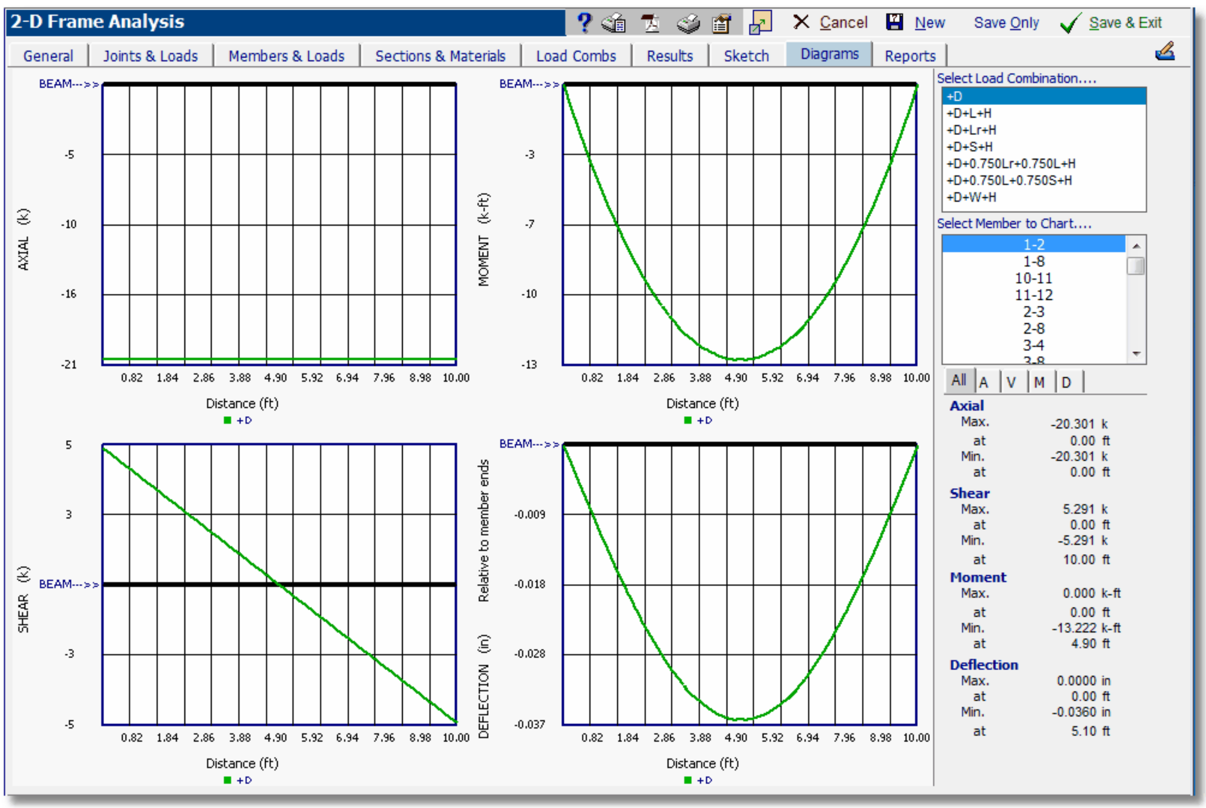
You have two selection areas to use...

Select Load Combination

This box lists all of the load combinations that are being run. Click on any of the available load combinations in the list to view diagrams based on that load combination.

Select Member to Chart

This box lists all of the members in the frame. Click on any of the members in the list to view diagrams for that member.

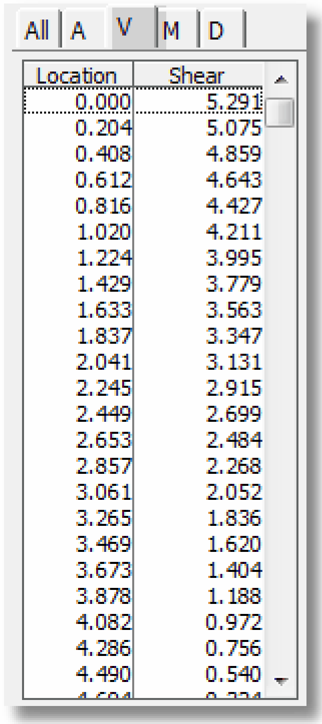


Tabs All, A, V, M, D

These tabs let you explore the force details for the selected member and load combination in different ways.

When the All tab is selected, the extreme values of axial load, moment, shear, and deflection are presented for the currently selected member.

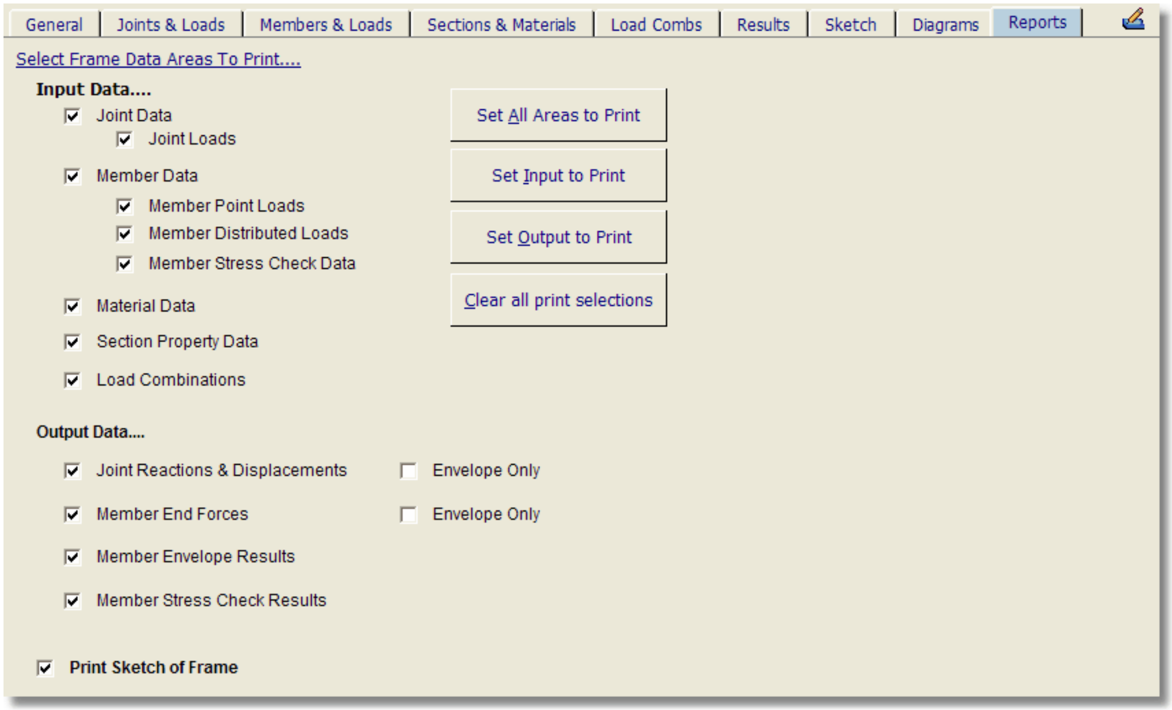
The other tabs provide detailed list of the values for the currently selected which are used to create the respective diagrams.



Location	Shear
0.000	5.291
0.204	5.075
0.408	4.859
0.612	4.643
0.816	4.427
1.020	4.211
1.224	3.995
1.429	3.779
1.633	3.563
1.837	3.347
2.041	3.131
2.245	2.915
2.449	2.699
2.653	2.484
2.857	2.268
3.061	2.052
3.265	1.836
3.469	1.620
3.673	1.404
3.878	1.188
4.082	0.972
4.286	0.756
4.490	0.540

10.8.10 Reports

This tab lets you select what data to print from the frame analysis.



Under Output Data you have the choice of compressing the output to an Envelope Only status. If Envelope Only is not checked the full details will be printed which can results in many pages of output. It is advisable to look at a print preview to see what it generates.

Using Envelope Only will examine each table and print only the extreme values for each load combination.

10.8.11 Printout

Here is a sample printout for the current frame:

Joseph P. Engineer
GREAT ENGINEERING UNLIMITED
1234 Main Street, 1st Floor
Anytown, USA 99221
jpe@GreatEngineering.com
Phone : (800) 111-2222

Title :
Dsgnr:
Project Desc.:

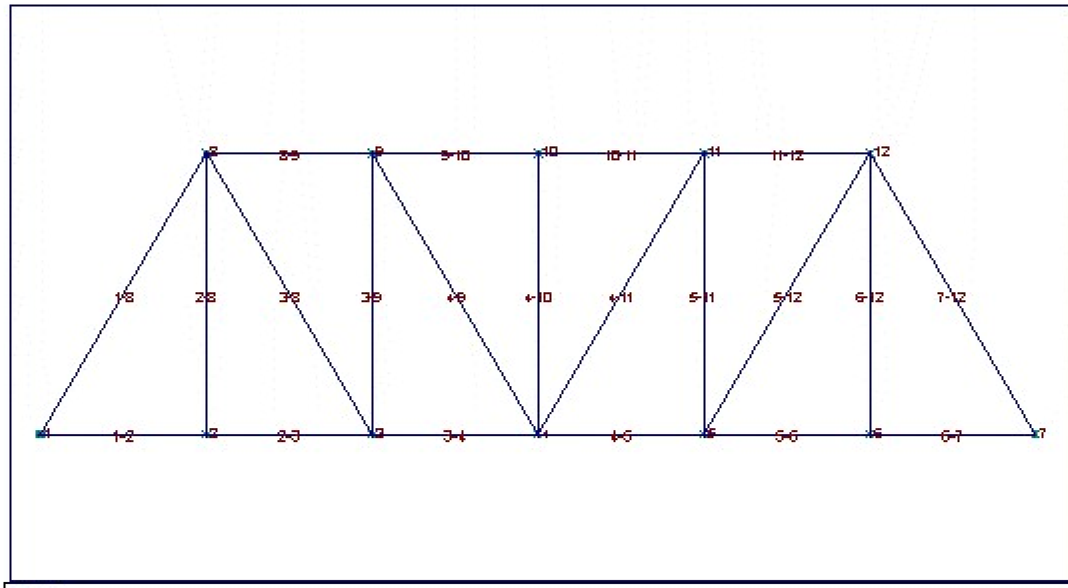
Project Notes:

Job #

2-D Frame Analysis

Lic # : KW-06000216

License Owner : licensed enercalc user



Joints...

Joint Label	Joint Coordinates X ft	Joint Coordinates Y ft	X Restraint	Y Restraint	Z Restraint	Joint Temp deg F
1	0.0	0.0	Fixed	Fixed	Fixed	0
2	20.667	0.0			Fixed	0
3	41.333	0.0			Fixed	0
4	62.0	0.0			Fixed	0
5	82.667	0.0			Fixed	0
6	103.33	0.0			Fixed	0
7	124.0	0.0		Fixed	Fixed	0
8	20.667	35.0			Fixed	0
9	41.333	35.0			Fixed	0
10	62.0	35.0			Fixed	0
11	82.667	35.0			Fixed	0
12	103.33	35.0			Fixed	0

Members...

Member Label	Property Label	Endpoint Joints I Joint	Endpoint Joints J Joint	Member Length ft	I End Releases X	I End Releases Y	I End Releases Z	J End Releases X	J End Releases Y	J End Releases Z
1-2	Chord	1	2	20.667	Fixed	Fixed	Free	Fixed	Fixed	Free
1-8	Diagonal	1	8	40.646	Fixed	Fixed	Free	Fixed	Fixed	Free
10-11	Chord	10	11	20.667	Fixed	Fixed	Free	Fixed	Fixed	Free
11-12	Chord	11	12	20.667	Fixed	Fixed	Free	Fixed	Fixed	Free
2-3	Chord	2	3	20.667	Fixed	Fixed	Free	Fixed	Fixed	Free
2-8	Vertical	2	8	35.000	Fixed	Fixed	Free	Fixed	Fixed	Free
3-4	Chord	3	4	20.667	Fixed	Fixed	Free	Fixed	Fixed	Free
3-8	Diagonal	3	8	40.646	Fixed	Fixed	Free	Fixed	Fixed	Free
3-9	Vertical	3	9	35.000	Fixed	Fixed	Free	Fixed	Fixed	Free
4-10	Vertical	4	10	35.000	Fixed	Fixed	Free	Fixed	Fixed	Free
4-11	Diagonal	4	11	40.646	Fixed	Fixed	Free	Fixed	Fixed	Free
4-5	Chord	4	5	20.667	Fixed	Fixed	Free	Fixed	Fixed	Free
4-9	Diagonal	4	9	40.646	Fixed	Fixed	Free	Fixed	Fixed	Free

Joseph P. Engineer
GREAT ENGINEERING UNLIMITED
1234 Main Street, 1st Floor
Anytown, USA 99221
jpe@GreatEngineering.com
Phone : (800) 111-2222

Title :
Job #
Dsgnr:
Project Desc.:

Project Notes :

2-D Frame Analysis

File: C:\ECWA_TECH_ECX_FILES\examples.ecx
ENERCALC, INC.
Lic. # : KW-06000216
License Owner : licensed enercalc user

Members...

Member Label	Property Label	Endpoint Joints I Joint	J Joint	Member Length ft	I End Releases X	Y	Z	J End Releases X	Y	Z
5-11	Vertical	5	11	35.000	Fixed	Fixed	Free	Fixed	Fixed	Free
5-12	Diagonal	5	12	40.646	Fixed	Fixed	Free	Fixed	Fixed	Free
5-6	Chord	5	6	20.667	Fixed	Fixed	Free	Fixed	Fixed	Free
6-12	Vertical	6	12	35.000	Fixed	Fixed	Free	Fixed	Fixed	Free
6-7	Chord	6	7	20.667	Fixed	Fixed	Free	Fixed	Fixed	Free
7-12	Diagonal	7	12	40.646	Fixed	Fixed	Free	Fixed	Fixed	Free
8-9	Chord	8	9	20.667	Fixed	Fixed	Free	Fixed	Fixed	Free
9-10	Chord	9	10	20.667	Fixed	Fixed	Free	Fixed	Fixed	Free

Materials...

Member Label	Youngs ksi	Density kcf	Thermal in/degr	Yield ksi
Default	1.00	0.000	0.000000	1.00
Steel	29,000.00	0.490	0.000650	50.00
wood	1.00	65.000	1.000000	1.50

Section Sections...

Prop Label	Group Tag	Material	Area	Depth	Width	Ixx	Iyy
Default	Group	Default	1.0 in^2	0.0 in	0.0 in	1.0 in^4	1.0 in^4
W14x145	Chord	Steel	42.70 in^2	14.80 in	15.50 in	1,710.0 in^4	677.0 in^4
W14x145	Diagonal	Steel	42.70 in^2	14.80 in	15.50 in	1,710.0 in^4	677.0 in^4
W14x145	Vertical	Steel	42.70 in^2	14.80 in	15.50 in	1,710.0 in^4	677.0 in^4
S-2x8		Wood	18.750 in^2	2.50 in	7.50 in	9.766 in^4	87.891 in^4

Joint Loads....

Joint Label	Load Direction	Load Magnitude Dead	Roof Live	Live	Snow	Seismic	Wind	Earth	
1	Global Y	1.0							k
2	Global Y	2.0							k
3	Global Y	3.0							k
4	Global Y	4.0							k
5	Global Y	5.0							k
6	Global Y	6.0							k
7	Global Y	7.0							k
8	Global Y	8.0							k
9	Global Y	9.0							k

Member Point Loads....

Member Label	Load Direction	Distance from "I" Joint	Load Magnitude Dead	Roof Live	Live	Snow	Seismic	Wind	Earth	
8-9	Global Y	5 ft	1.0							k

Member Distributed Loads....

Member Label	Load Direction	Load Extents Start / End ft		Load Magnitude Dead	Roof Live	Live	Snow	Seismic	Wind	Earth	
8-9	Global Y	0.0	Start Mag :	1.0							k/ft
		20.667	End Mag :	1.0							k/ft

Load Combinations...

Load Combination Description	Group Multiplier	Gravity Load Factors X	Y	Load Combination Factors Dead	Roof Live	Live	Snow	Seismic	Wind	Earth
+D	1.0		1.0	1.0						
+D+L+H	1.0		1.0	1.0		1.0				1.0
+D+Lr+H	1.0		1.0	1.0	1.0					1.0
+D+S+H	1.0		1.0	1.0			1.0			1.0
0.3(+D+2.0Lr+3.0L+4.0S+5.0W+6.0E+7.0H)	0.30		-1.0	1.0	2.0	3.0	4.0	6.0	5.0	7.0
+D+0.750Lr+0.750L+H	1.0		1.0	1.0	0.750	0.750				1.0
+D+0.750L+0.750S+H	1.0		1.0	1.0		0.750	0.750			1.0

Joseph P. Engineer
GREAT ENGINEERING UNLIMITED
1234 Main Street, 1st Floor
Anytown, USA 99221
jpe@GreatEngineering.com
Phone : (800) 111-2222

Title :
Dsgnr:
Project Desc.:

Project Notes:

Job #

Printed: 18 APR 2018, 11:30AM

File: C:\ECI\WA_TECH_ECI_FILES\examples\
ENERCALC, INC. 1983-2018, Ver: 6.18

2-D Frame Analysis

Lic. # : KW-06000216

License Owner : licensed enercalc user

Load Combinations...

Load Combination Description	Group Multiplier	Gravity Load Factors X	Gravity Load Factors Y	Dead	Roof Live	Live	Snow	Seismic	Wind	Earth
+D+W+H	1.0		1.0	1.0					1.0	1.0
+D+0.70E+H	1.0		1.0	1.0				0.70		1.0
+D+0.750Lr+0.750L+0.750W+H	1.0		1.0	1.0	0.750	0.750			0.750	1.0
+D+0.750L+0.750S+0.750W+H	1.0		1.0	1.0		0.750	0.750		0.750	1.0
+D+0.750Lr+0.750L+0.5250E+H	1.0		1.0	1.0	0.750	0.750		0.5250		1.0
+D+0.750L+0.750S+0.5250E+H	1.0		1.0	1.0		0.750	0.750	0.5250		1.0
+0.60D+W+H	1.0		1.0	0.60					1.0	1.0
+0.60D+0.70E+H	1.0		1.0	0.60				0.70		1.0
+1.40D	1.0		1.0	1.40						
+1.20D+0.50Lr+1.60L+1.60H	1.0		1.0	1.20	0.50	1.60				1.60
+1.20D+1.60L+0.50S+1.60H	1.0		1.0	1.20		1.60	0.50			1.60
+1.20D+1.60Lr+0.50L	1.0		1.0	1.20	1.60	0.50				
+1.20D+1.60Lr+0.80W	1.0		1.0	1.20	1.60				0.80	
+1.20D+0.50L+1.60S	1.0		1.0	1.20		0.50	1.60			
+1.20D+1.60S+0.80W	1.0		1.0	1.20			1.60		0.80	
+1.20D+0.50Lr+0.50L+1.60W	1.0		1.0	1.20	0.50	0.50			1.60	
+1.20D+0.50L+0.50S+1.60W	1.0		1.0	1.20		0.50	0.50		1.60	
+1.20D+0.50L+0.20S+E	1.0		1.0	1.20		0.50	0.20	1.0		
+0.90D+1.60W+1.60H	1.0		1.0	0.90					1.60	1.60
+0.90D+E+1.60H	1.0		1.0	0.90				1.0		1.60
+D+Lr+L+S+W+E+H	1.0		-1.0	1.0	1.0	1.0	1.0	1.0	1.0	1.0

Joint Displacements & Reactions for Load Combination :

Joint Label	LoadCombination	Joint Displacements X (in)	Joint Displacements Y (in)	Joint Displacements Z (Radians)	Joint Reactions X (k)	Joint Reactions Y (k)	Joint Reactions Z (k-ft)
1	+D	0.0	.0000	0.0	.0000	-7.152	0.0
2	+D	-0.000437	0.008754	0.0	0.0	0.0	0.0
3	+D	-0.000875	0.01756	0.0	0.0	0.0	0.0
4	+D	-0.002411	0.030	0.0	0.0	0.0	0.0
5	+D	-0.005928	0.02812	0.0	0.0	0.0	0.0
6	+D	-0.008248	0.01780	0.0	0.0	0.0	0.0
7	+D	-0.01057	.0000	0.0	0.0	-17.072	0.0
8	+D	-0.009461	0.007551	0.0	0.0	0.0	0.0
9	+D	-0.007925	0.01885	0.0	0.0	0.0	0.0
10	+D	-0.004630	0.03188	0.0	0.0	0.0	0.0
11	+D	-0.001334	0.03036	0.0	0.0	0.0	0.0
12	+D	0.002182	0.01795	0.0	0.0	0.0	0.0
1	+D+L+H	0.0	.0000	0.0	.0000	-7.152	0.0
2	+D+L+H	-0.000437	0.008754	0.0	0.0	0.0	0.0
3	+D+L+H	-0.000875	0.01756	0.0	0.0	0.0	0.0
4	+D+L+H	-0.002411	0.030	0.0	0.0	0.0	0.0
5	+D+L+H	-0.005928	0.02812	0.0	0.0	0.0	0.0
6	+D+L+H	-0.008248	0.01780	0.0	0.0	0.0	0.0
7	+D+L+H	-0.01057	.0000	0.0	0.0	-17.072	0.0
8	+D+L+H	-0.009461	0.007551	0.0	0.0	0.0	0.0
9	+D+L+H	-0.007925	0.01885	0.0	0.0	0.0	0.0
10	+D+L+H	-0.004630	0.03188	0.0	0.0	0.0	0.0
11	+D+L+H	-0.001334	0.03036	0.0	0.0	0.0	0.0
12	+D+L+H	0.002182	0.01795	0.0	0.0	0.0	0.0
1	+D+Lr+H	0.0	.0000	0.0	.0000	-7.152	0.0
2	+D+Lr+H	-0.000437	0.008754	0.0	0.0	0.0	0.0
3	+D+Lr+H	-0.000875	0.01756	0.0	0.0	0.0	0.0
4	+D+Lr+H	-0.002411	0.030	0.0	0.0	0.0	0.0
5	+D+Lr+H	-0.005928	0.02812	0.0	0.0	0.0	0.0
6	+D+Lr+H	-0.008248	0.01780	0.0	0.0	0.0	0.0
7	+D+Lr+H	-0.01057	.0000	0.0	0.0	-17.072	0.0
8	+D+Lr+H	-0.009461	0.007551	0.0	0.0	0.0	0.0
9	+D+Lr+H	-0.007925	0.01885	0.0	0.0	0.0	0.0
10	+D+Lr+H	-0.004630	0.03188	0.0	0.0	0.0	0.0
11	+D+Lr+H	-0.001334	0.03036	0.0	0.0	0.0	0.0
12	+D+Lr+H	0.002182	0.01795	0.0	0.0	0.0	0.0
1	+1.40D	0.0	.0000	0.0	0.0	8.165	0.0

Joseph P. Engineer
GREAT ENGINEERING UNLIMITED
1234 Main Street, 1st Floor
Anytown, USA 99221
jpe@GreatEngineering.com
Phone : (800) 111-2222

Title :
Dsgnr:
Project Desc.:

Project Notes :

Job #

2-D Frame Analysis

ENERCALC, INC.

Lic. # : KW-06000216 — License Owner : licensed enercalc user

Joint Displacements & Reactions for Load Combination :

Joint Label	Load Combination	Joint Displacements X (in)	Joint Displacements Y (in)	Joint Displacements Z (Radians)	Joint Reactions X (k)	Joint Reactions Y (k)	Joint Reactions Z (k-ft)
2	+1.40D	0.001327	-0.005306	0.0	0.0	0.0	0.0
3	+1.40D	0.002654	-0.003453	0.0	0.0	0.0	0.0
4	+1.40D	0.003648	0.007761	0.0	0.0	0.0	0.0
5	+1.40D	0.001869	0.01133	0.0	0.0	0.0	0.0
6	+1.40D	0.000560	0.007353	0.0	0.0	0.0	0.0
7	+1.40D	-0.000749	.0000	0.0	0.0	-5.722	0.0
8	+1.40D	0.000469	-0.006237	0.0	0.0	0.0	0.0
9	+1.40D	-0.000525	-0.003957	0.0	0.0	0.0	0.0
10	+1.40D	0.000542	0.009642	0.0	0.0	0.0	0.0
11	+1.40D	0.001608	0.01217	0.0	0.0	0.0	0.0
12	+1.40D	0.003386	0.008321	0.0	0.0	0.0	0.0

Member End Forces for Load Combination :

Member Label	Load Combination	Joint "I" End Forces Axial (k)	Joint "I" End Forces Shear (k)	Joint "I" End Forces Moment (k-ft)	Joint "J" End Forces Axial (k)	Joint "J" End Forces Shear (k)	Joint "J" End Forces Moment (k-ft)
1-2	+D	2.184	-1.501	0.0	-2.184	-1.501	0.0
1-8	+D	-6.837	-1.501	0.0	1.752	-1.501	0.0
10-11	+D	-16.453	-1.501	0.0	16.453	-1.501	0.0
11-12	+D	-17.560	-1.501	0.0	17.560	-1.501	0.0
2-3	+D	2.184	-1.501	0.0	-2.184	-1.501	0.0
2-8	+D	1.003	0.0	0.0	-6.088	0.0	0.0
3-4	+D	7.671	-1.501	0.0	-7.671	-1.501	0.0
3-8	+D	8.250	1.501	0.0	-13.336	1.501	0.0
3-9	+D	-6.338	0.0	0.0	1.253	0.0	0.0
4-10	+D	-8.088	0.0	0.0	3.003	0.0	0.0
4-11	+D	-4.720	-1.501	0.0	-0.3650	-1.501	0.0
4-5	+D	17.560	-1.501	0.0	-17.560	-1.501	0.0
4-9	+D	14.729	1.501	0.0	-19.814	1.501	0.0
5-11	+D	-9.166	0.0	0.0	4.081	0.0	0.0
5-12	+D	9.212	-1.501	0.0	-14.297	-1.501	0.0
5-6	+D	11.584	-1.501	0.0	-11.584	-1.501	0.0
6-12	+D	-2.997	0.0	0.0	-2.088	0.0	0.0
6-7	+D	11.584	-1.501	0.0	-11.584	-1.501	0.0
7-12	+D	-25.325	1.501	0.0	20.239	1.501	0.0
8-9	+D	-7.671	9.590	0.0	7.671	9.074	0.0
9-10	+D	-16.453	-1.501	0.0	16.453	-1.501	0.0
1-2	+D+L+H	2.184	-1.501	0.0	-2.184	-1.501	0.0
1-8	+D+L+H	-6.837	-1.501	0.0	1.752	-1.501	0.0
10-11	+D+L+H	-16.453	-1.501	0.0	16.453	-1.501	0.0
11-12	+D+L+H	-17.560	-1.501	0.0	17.560	-1.501	0.0
2-3	+D+L+H	2.184	-1.501	0.0	-2.184	-1.501	0.0
2-8	+D+L+H	1.003	0.0	0.0	-6.088	0.0	0.0
3-4	+D+L+H	7.671	-1.501	0.0	-7.671	-1.501	0.0
3-8	+D+L+H	8.250	1.501	0.0	-13.336	1.501	0.0
3-9	+D+L+H	-6.338	0.0	0.0	1.253	0.0	0.0
4-10	+D+L+H	-8.088	0.0	0.0	3.003	0.0	0.0
4-11	+D+L+H	-4.720	-1.501	0.0	-0.3650	-1.501	0.0
4-5	+D+L+H	17.560	-1.501	0.0	-17.560	-1.501	0.0
4-9	+D+L+H	14.729	1.501	0.0	-19.814	1.501	0.0
5-11	+D+L+H	-9.166	0.0	0.0	4.081	0.0	0.0
5-12	+D+L+H	9.212	-1.501	0.0	-14.297	-1.501	0.0
5-6	+D+L+H	11.584	-1.501	0.0	-11.584	-1.501	0.0
6-12	+D+L+H	-2.997	0.0	0.0	-2.088	0.0	0.0
6-7	+D+L+H	11.584	-1.501	0.0	-11.584	-1.501	0.0
7-12	+D+L+H	-25.325	1.501	0.0	20.239	1.501	0.0
8-9	+D+L+H	-7.671	9.590	0.0	7.671	9.074	0.0
9-10	+D+L+H	-16.453	-1.501	0.0	16.453	-1.501	0.0
1-2	+D+Lr+H	2.184	-1.501	0.0	-2.184	-1.501	0.0
1-8	+D+Lr+H	-6.837	-1.501	0.0	1.752	-1.501	0.0
10-11	+D+Lr+H	-16.453	-1.501	0.0	16.453	-1.501	0.0
11-12	+D+Lr+H	-17.560	-1.501	0.0	17.560	-1.501	0.0

Joseph P. Engineer
GREAT ENGINEERING UNLIMITED
1234 Main Street, 1st Floor
Anytown, USA 99221
jpe@GreatEngineering.com
Phone : (800) 111-2222

Title :
Dsgnr:
Project Desc.:

Project Notes :

Job #

2-D Frame Analysis

File:C:\ECWA_TECH_ECA_FILES\examples.ecl
ENERCALC, INC.
Lic. # : KW-06000216
License Owner : licensed enercalc user

Member End Forces for Load Combination :

Member Label	Load Combination	Joint "I" End Forces Axial k	Shear k	Moment k-ft	Joint "J" End Forces Axial k	Shear k	Moment k-ft
2-3	+D+Lr+H	2.184	-1.501	0.0	-2.184	-1.501	0.0
2-8	+D+Lr+H	1.003	0.0	0.0	-6.088	0.0	0.0
3-4	+D+Lr+H	7.671	-1.501	0.0	-7.671	-1.501	0.0
3-8	+D+Lr+H	8.250	1.501	0.0	-13.336	1.501	0.0
3-9	+D+Lr+H	-6.338	0.0	0.0	1.253	0.0	0.0
4-10	+D+Lr+H	-8.088	0.0	0.0	3.003	0.0	0.0
4-11	+D+Lr+H	-4.720	-1.501	0.0	-0.3650	-1.501	0.0
4-5	+D+Lr+H	17.560	-1.501	0.0	-17.560	-1.501	0.0
4-9	+D+Lr+H	14.729	1.501	0.0	-19.814	1.501	0.0
5-11	+D+Lr+H	-9.166	0.0	0.0	4.081	0.0	0.0
5-12	+D+Lr+H	9.212	-1.501	0.0	-14.297	-1.501	0.0
5-6	+D+Lr+H	11.584	-1.501	0.0	-11.584	-1.501	0.0
6-12	+D+Lr+H	-2.997	0.0	0.0	-2.088	0.0	0.0
6-7	+D+Lr+H	11.584	-1.501	0.0	-11.584	-1.501	0.0
7-12	+D+Lr+H	-25.325	1.501	0.0	20.239	1.501	0.0
8-9	+D+Lr+H	-7.671	9.590	0.0	7.671	9.074	0.0
9-10	+D+Lr+H	-16.453	-1.501	0.0	16.453	-1.501	0.0
1-2	+1.40D	-6.625	-1.501	0.0	6.625	-1.501	0.0
1-8	+1.40D	10.486	-1.501	0.0	-15.572	-1.501	0.0
10-11	+1.40D	-5.323	-1.501	0.0	5.323	-1.501	0.0
11-12	+1.40D	-8.881	-1.501	0.0	8.881	-1.501	0.0
2-3	+1.40D	-6.625	-1.501	0.0	6.625	-1.501	0.0
2-8	+1.40D	0.2028	0.0	0.0	-5.288	0.0	0.0
3-4	+1.40D	-4.964	-1.501	0.0	4.964	-1.501	0.0
3-8	+1.40D	0.7243	1.501	0.0	-5.810	1.501	0.0
3-9	+1.40D	-1.057	0.0	0.0	-4.028	0.0	0.0
4-10	+1.40D	-8.088	0.0	0.0	3.003	0.0	0.0
4-11	+1.40D	-9.539	-1.501	0.0	4.454	-1.501	0.0
4-5	+1.40D	8.881	-1.501	0.0	-8.881	-1.501	0.0
4-9	+1.40D	17.689	1.501	0.0	-22.775	1.501	0.0
5-11	+1.40D	-5.017	0.0	0.0	-0.06889	0.0	0.0
5-12	+1.40D	2.070	-1.501	0.0	-7.156	-1.501	0.0
5-6	+1.40D	6.535	-1.501	0.0	-6.535	-1.501	0.0
6-12	+1.40D	-5.397	0.0	0.0	0.3117	0.0	0.0
6-7	+1.40D	6.535	-1.501	0.0	-6.535	-1.501	0.0
7-12	+1.40D	-15.396	1.501	0.0	10.311	1.501	0.0
8-9	+1.40D	4.964	14.027	0.0	-4.964	13.304	0.0
9-10	+1.40D	-5.323	-1.501	0.0	5.323	-1.501	0.0

Maximum Member Forces — Only Load Combinations giving maximum values are listed

Member Label	Axial	Distance from "I" Joint	Moment	Distance from "I" Joint	Shear	Distance from "I" Joint
1-2	-6.625 k	0.0 ft *+1.40D*	7.754 k-ft	0.8435 ft *+D*	-1.501 k	0.0 ft *+D*
1-8	15.572 k	3.387 ft *+1.40D*	15.250 k-ft	1.728 ft *+1.40D*	-1.501 k	0.0 ft *+1.40D*
10-11	-16.453 k	0.0 ft *+D*	7.754 k-ft	0.8435 ft *+D*	-1.501 k	0.0 ft *+D*
11-12	-17.560 k	0.0 ft *+D*	7.754 k-ft	0.8435 ft *+D*	-1.501 k	0.0 ft *+D*
2-3	-6.625 k	0.0 ft *+1.40D*	7.754 k-ft	0.8435 ft *+D*	-1.501 k	0.0 ft *+D*
2-8	6.088 k	2.917 ft *+D*	0.0 k-ft	0.0 ft	0.0 k	0.0 ft
3-4	7.671 k	0.0 ft *+D*	7.754 k-ft	0.8435 ft *+D*	-1.501 k	0.0 ft *+D*
3-8	13.336 k	3.387 ft *+D*	-15.250 k-ft	1.728 ft *+1.40D*	1.501 k	0.0 ft *+1.40D*
3-9	-6.338 k	0.0 ft *+D*	0.0 k-ft	0.0 ft	0.0 k	0.0 ft

Joseph P. Engineer
GREAT ENGINEERING UNLIMITED
1234 Main Street, 1st Floor
Anytown, USA 99221
jpe@GreatEngineering.com
Phone : (800) 111-2222

Title :
Job #
Dsgnr:
Project Desc.:

Project Notes :

Printed: [illegible]

2-D Frame Analysis

File: [illegible]
ENERCALC, INC. [illegible], Ver: [illegible]

Lic. # : KW-06000216
License Owner : licensed enercalc user

Maximum Member Forces
Only Load Combinations giving maximum values are listed

Member Label	Axial	Distance from "I" Joint	Moment	Distance from "I" Joint	Shear	Distance from "I" Joint
4-10	-8.088 k	0.0 ft +1.40D	0.0 k-ft	0.0 ft	0.0 k	0.0 ft
4-11	-9.539 k	0.0 ft +1.40D	15.250 k-ft	1.728 ft +1.40D	1.501 k	3.387 ft +D
4-5	17.560 k	0.0 ft +D	7.754 k-ft	0.8435 ft +D	-1.501 k	0.0 ft +D
4-9	22.775 k	3.387 ft +1.40D	-15.250 k-ft	1.728 ft +1.40D	-1.501 k	3.387 ft +D
5-11	-9.166 k	0.0 ft +D	0.0 k-ft	0.0 ft	0.0 k	0.0 ft
5-12	14.297 k	3.387 ft +D	15.250 k-ft	1.728 ft +D	-1.501 k	0.0 ft +D
5-6	11.584 k	0.0 ft +D	7.754 k-ft	0.8435 ft +D	-1.501 k	0.0 ft +D
6-12	-5.397 k	0.0 ft +1.40D	0.0 k-ft	0.0 ft	0.0 k	0.0 ft
6-7	11.584 k	0.0 ft +D	7.754 k-ft	0.8435 ft +D	-1.501 k	0.0 ft +D
7-12	-25.325 k	0.0 ft +D	-15.250 k-ft	1.728 ft +D	1.501 k	0.0 ft +D
8-9	-7.671 k	0.0 ft +D	-70.531 k-ft	0.8435 ft +1.40D	14.027 k	0.0 ft +1.40D
9-10	-16.453 k	0.0 ft +D	7.754 k-ft	0.8435 ft +D	-1.501 k	0.0 ft +D

10.9 Miscellaneous Calculation Modules

10.9.1 Torsional Analysis of Rigid Diphragm

Need more? Ask Us a Question

This module provides horizontal force distribution analysis for a rigid diaphragm laterally supported by up to 160 resisting elements (walls, columns or generic resisting elements).

Click here for a video: Torsional Analysis of Rigid Diaphragm

The lateral shear force is applied to the rigid diaphragm, and that force is distributed to all elements after the rotational stiffness analysis has been completed.

All lateral forces are distributed to each element on the basis of relative rigidities and resisting element locations. Lateral shear forces, direct torsional forces, and accidental eccentricity torsional forces are considered after determining the location of the center of rigidity.

The module provides analysis for one level only. For structures where elements are symmetrically placed on many levels, a calculation may be performed for each level and results added to determine shears and overturning moments for each element. When determining center of mass (where the lateral force is applied) on successively lower levels when elements are NOT all aligned vertically, a new center of mass position should be calculated based upon element forces acting from the diaphragm from the level above and combined with the force at that level.

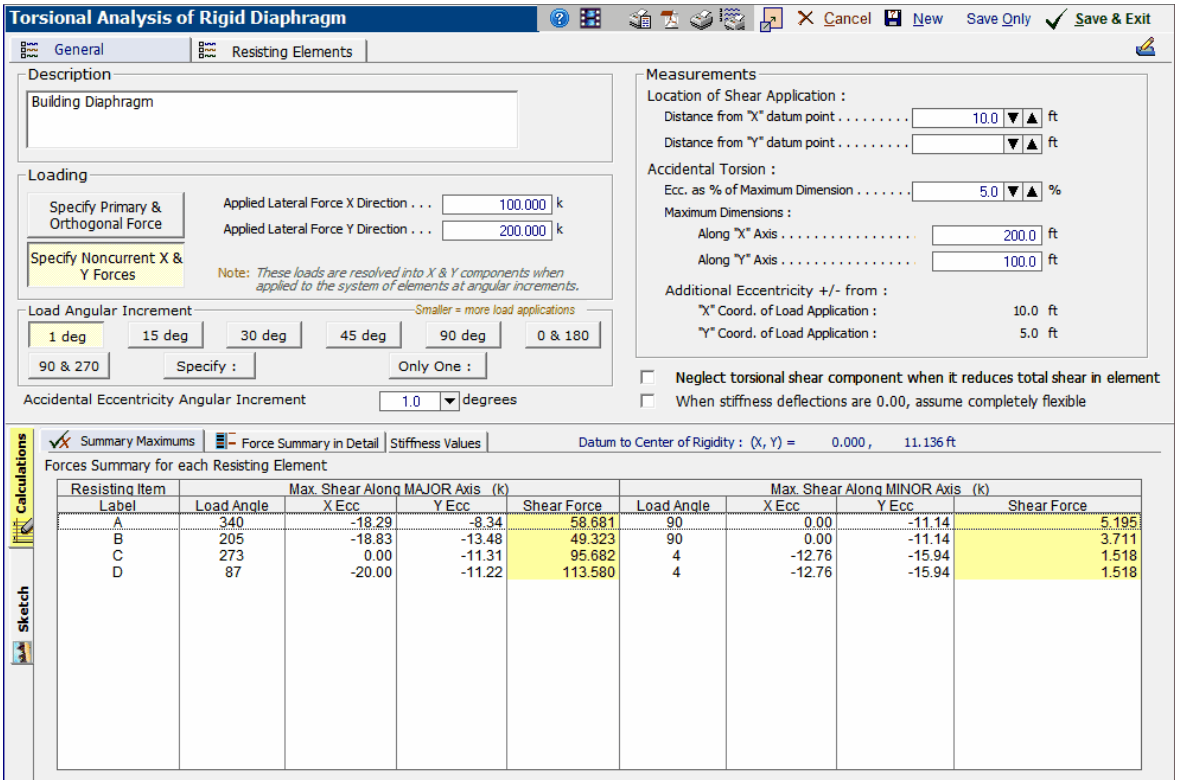
A very unique capability of this module is to have the applied lateral load applied at angular increments for a full 360 degree rotation. The prior version of this module in our Version 5.8 software only applied the lateral load at 90 degree increments. Because seismic or wind loads can occur at any angle, we provide the ability for the user to define the angles at which the lateral load is applied to the rigid diaphragm for distribution to the resisting elements.

When the lateral force is rotated around the specified number of angular increments, the user has two options for specifying the magnitude of the force to consider at each angular orientation. One option is to specify the magnitude of the applied lateral force and an optional orthogonal force magnitude that will be considered to act concurrently. When this option is selected, the program uses the same magnitude for the resultant lateral force at all angular increments, and that magnitude is calculated as the SRSS of the applied lateral force and the orthogonal force. The second option is to specify the magnitude of the applied lateral force when the force points in the X direction and the magnitude of the applied lateral force when the force points in the Y direction. When this option is selected, the program considers the lateral force to vary in an elliptical manner as the angular orientation of the force changes. When the force is considered to act at the zero-degree orientation, the magnitude will be exactly equal to the specified lateral force in the X direction. As the angular orientation changes, the lateral force will vary in that elliptical manner. When the force is considered to act at the 90-degree orientation, the lateral force will be exactly equal to the specified force in the Y direction, and so on.

Another unique feature is the handling of the accidental eccentricity. The code specifies that an accidental eccentricity must be considered, as it will have an effect on the total torsional moment applied to the diaphragm. The minimum eccentricity is typically specified as 5% of the building dimension measured perpendicular to the direction of load application. To thoroughly address the eccentricity requirements, this module creates an ellipse measuring 5% (or the specified value) of the building dimension on each axis, around which the lateral load is applied.

Technical note: Prior to build 6.15.7.24, this module neglected the shear due to the torsional component of load if it was of the opposite algebraic sign to the direct shear component, because considering that component would reduce the total shear on the particular element being considered. However, build 6.15.7.24 introduced a user option to change this behavior. It appears in the form of a checkbox labeled "Neglect torsional shear component when it reduces total shear in element". If this option is selected, the program will operate in the way that it used to prior to build 6.15.7.24. If this option is DEselected, the program will ***always*** *consider the shears due to torsion (inherent and accidental), even if they are of opposite algebraic sign to the direct shears and therefore tend to reduce the total shears in an element.*

So to recap.....the applied lateral load is applied at the angular increments you specify for a full 360 degrees, and this is performed for the number of angular locations you specify around the minimum eccentricity ellipse. This means if you use 15 degree angular increments for load direction and 15 degree increments for accidental eccentricity, then the lateral load is actually applied (360/15+1) * (360/15+1) = 625 times in various locations and directions. This can provide a very accurate calculation of applied torsions and direct shears to all resisting elements connected to a rigid diaphragm.



Basic Usage

- The most important step for successful use of this module is to properly enter the X and Y location of the center of rigidity of each resisting element and its angle in degrees counterclockwise from a normal Cartesian "0" degree orientation.

- For each resisting element, its center of rigidity will be at the centroid of the element.

- Default angular orientation of elements is as follows:
- Walls: When rotation is zero, length (local y) is parallel to Global X (points right on the screen), and local x points downward on the screen.
- Bending Members: When rotation is zero, local y is parallel to Global X (points right on the screen), and local x points downward on the screen.
- Generic Resisting Elements: When rotation is zero, local y is parallel to Global X (points right on the screen), and local x points downward on the screen.

- When rotation angles are applied to resisting elements, the angle increases positively in a counterclockwise direction. Enter all angles as positive.

- Lateral shears are typically the force at the diaphragm level due to wind or seismic forces at that level. Location of Shear Application specifies the X-Y coordinates of the center of the load ellipse where the lateral shears act. If lateral forces must be added to the diaphragm from the level above or below, you must combine all forces to calculate an adjusted mass application point. Maximum Dimensions are used to calculate the minimum additional eccentricity that will be added to and subtracted from the inherent eccentricity to calculate governing forces for each resisting element.

- When defining walls as resisting elements, the thickness, length, and height are required for each wall providing lateral support to the diaphragm. These values are used with the elastic modulus to establish the relative stiffness of each wall. For other resisting elements you can enter the section information or just enter the resisting element deflection under the same load for all elements.

- The Elastic Modulus does not have to be an exact value if all of the elements are of identical construction. In this situation, it may be simpler to just use a value of 1.

- X & Y Distances for each resisting element define the location of the center of stiffness of each element in plan view. This location will be used when combining all stiffnesses and calculating the overall center of rigidity for all elements acting as a system.

- Enter the fixity condition that best describes the element's top and bottom restraint against rotation about the longitudinal and transverse axes. Fix-Pin would be appropriate for an inverted pendulum condition (where walls or columns cantilever up from a fixed base condition, but are free to rotate at their tops). Fix-Pin would also be appropriate for a moment frame structure with pinned column bases (a structure that behaves like a table). Fix-Fix would be appropriate for conditions where both the tops and the bottoms of the columns and/or walls are fixed against rotation about their longitudinal and transverse axes. This setting results in double curvature in the vertical lateral force resisting elements.

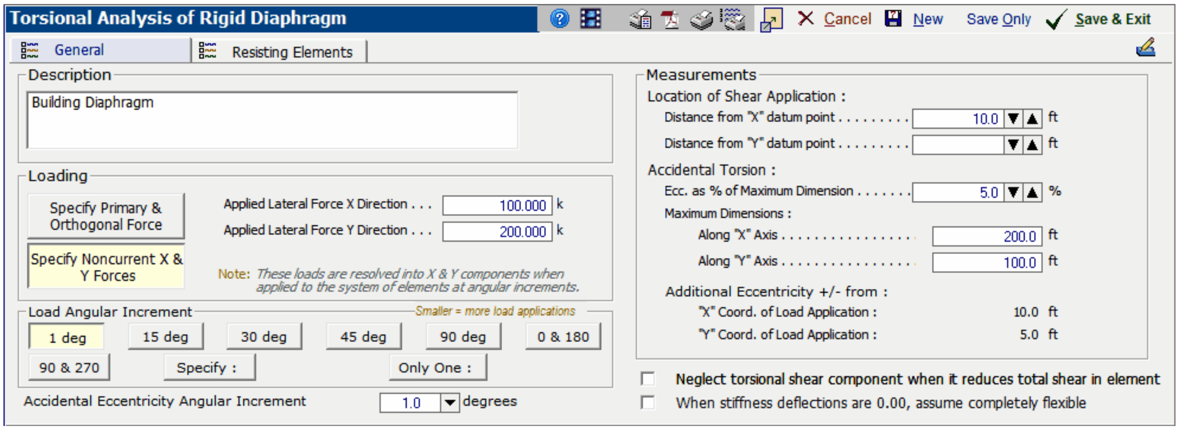
Unique Features

This module uses a numerical approach to determine center of rigidity location and to distribute lateral forces to each resisting element. Because resisting elements may be located at any angle, a rigorous stiffness analysis is performed, calculating each element's stiffness about both axes and combining the stiffnesses of all the elements to determine a center of rigidity location.

Coordinate System

Please note that a strict X-Y coordinate system should be used to ensure that the analysis is properly carried out. When setting up a model, remember that Global X increases to the right and Global Y increases up the screen.

General Input Tab



Loading

Specify Primary & Orthogonal Force

Applied Lateral Force

This is the main force applied to the rigid diaphragm. The location of application is defined by the load ellipse, the center of which is specified in the input item labeled Location of Shear Application.

Additional Orthogonal Force

This is an optional force that is applied at a 90-degree angle to the main force. Some codes specify that this force must be applied concurrently with the main force.

Maximum Load Used for Analysis

This is the resultant force applied to the diaphragm, calculated as sqrt(Main^2 + Orthogonal^2).

Specify Nonconcurrent X & Y Forces

Applied Lateral Force X Direction

This is the magnitude of the lateral force applied to the rigid diaphragm when the load is oriented at exactly zero or 180 degrees.

Applied Lateral Force Y Direction

This is the magnitude of the lateral force applied to the rigid diaphragm when the load is oriented at exactly 90 or 270 degrees.

When the load orientation is anywhere between the cardinal directions, the magnitude of the applied lateral force is determined by assuming that the lateral force follows a smoothly varying elliptical function.

The location of application is defined by the load ellipse, the center of which is specified in the input item labeled Location of Shear Application.

Load Angular Increment

This module allows the force to be applied to the rigid diaphragm in almost all angular directions.

According to the entry for angular increment, the module will apply the load to the diaphragm at multiple angular increments. For example, if you select "15 deg", the module will apply the lateral load at 0, 15, 30 degrees, etc. When the Load Angular Increment is set to smaller values, it will result in slightly longer calculation times, but it wall also allow the module to "zero in" more accurately on the actual maximum shear forces in all of the resisting elements.

Note that there is also an option named "Specify". This allows you to specify an angular increment for the direction of load.

Accidental Eccentricity Angular Increment

Most building codes require the consideration of an "accidental eccentricity". This is a prescribed additional amount of moment arm that must be compounded with the inherent eccentricity that already exists in the system; i.e. the distance between the center of rigidity and the center of mass for seismic loads or the distance between the center of rigidity and the center of exposure for wind loads. This additional eccentricity accounts for the variability of the exact location of the center of rigidity in normal as-built conditions.

Normally an "X direction" and a "Y direction" accidental eccentricity would be determined as a function (typically 5%) of the overall building dimension perpendicular to each direction. Then, the X directed force would be applied at two locations:

- center of mass PLUS "Y direction" eccentricity, and
- center of mass MINUS "Y direction" eccentricity.

And the Y directed force would be applied at two locations:

- center of mass PLUS "X direction" eccentricity, and
- center of mass MINUS "X direction" eccentricity.

However, in this module the "X direction" and "Y direction" eccentricities are used to specify the dimensions of an ellipse that encircles the center of mass. This ellipse creates a continuous path that smoothly incorporates the "X direction" and "Y direction" eccentricities. In this way, it defines all possible locations where the load should be applied to account for all possible accidental eccentricity locations.

The entry for Accidental Eccentricity Angular Increment specifies the angular increment that will be used to subdivide the ellipse into a number of locations where the force will be applied to the diaphragm.

Summary of Angular Increment & Accidental Eccentricity Angular Increment

The module applies the lateral load at the "Load Angular Increments" at each location of "Accidental Eccentricity Angular Increment" to generate an extensive set of results from which the maximum force values for each resisting element may be inspected.

For example, setting both "Load Angular Increment" and "Accidental Eccentricity Angular Increment" to 15 degrees tells the module to run (360/15 + 1) * (360/15 + 1) = 625 separate analyses of force distributions to the resisting elements.

Location of Shear Application

This specifies the X and Y location of the center of mass. The Accidental Eccentricity ellipse will be circumscribed around this location.

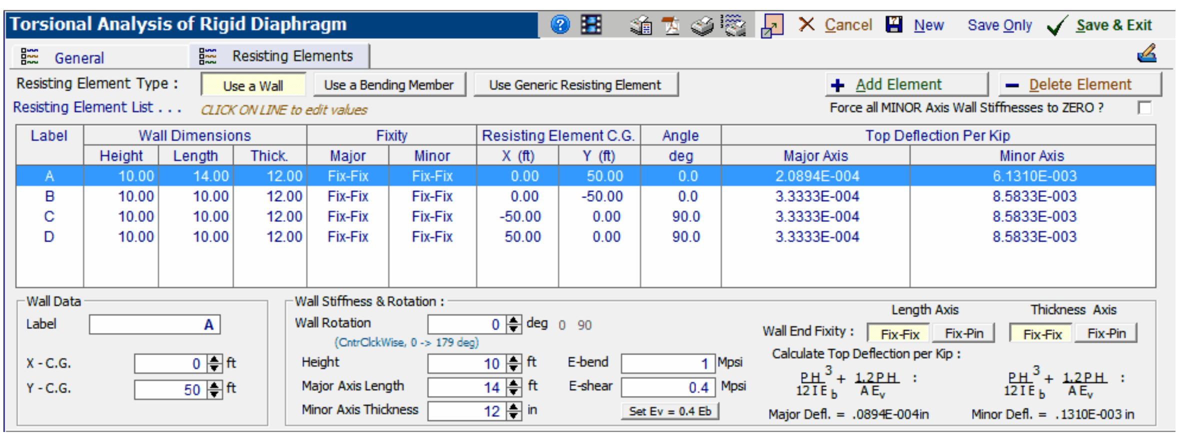
Accidental Torsion Values

Accidental torsion is defined as a percentage of overall constructed diaphragm dimension in each of two orthogonal directions. Therefore enter the necessary eccentricity percentage and both maximum diaphragm dimensions here.

When Stiffness deflections are 0.00, assume completely flexible

Note the option named "When Stiffness deflections are 0.00, assume completely flexible". This option can be used if your intent is to specify that an element is completely flexible in the weak direction. In this situation, you would need to specify an infinite deflection in that direction. So as a convenience, the system has been configured such that when this option is selected, it will interpret a deflection value of 0.00 as meaning that the element is completely flexible in that direction (i.e. has no ability to resist an applied force in that direction).

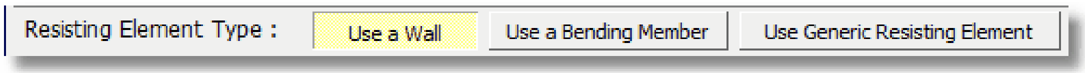
Resisting Elements Tab



Resisting Element Type

This module allows you to use three types of resisting elements. In past versions of this module, only walls were allowed. But many users wanted to enter information for braced frames or cantilevered columns for open buildings. So we've expanded this module to allow more general types of lateral resisting elements.

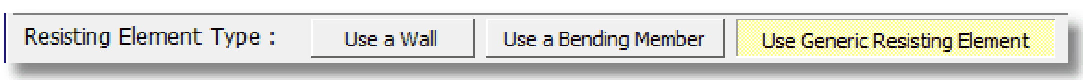
WALL: Click the [**Use a Wall**] button to define a wall as a resisting element. The wall must be rectangular in plan and must have a non-zero height. The selections for "Fix" and "Pin" will alter the equation used to calculate deflection in BOTH directions of the wall (unless the option is selected to "Consider all minor axis wall stiffnesses to be negligible"). Using the entered height, length, thickness, and modulus of elasticity for bending and shear, the module will calculate the bending and shear stiffness of the wall and report the deflection for a unit 1 kip applied load.



Note: The option to "Consider all minor axis wall stiffnesses to be negligible" allows walls to be modeled with a weak spring stiffness resisting flexure about the weak axis. It will tend to minimize the stiffness of walls about that axis, so they will not pick up much loading in the weak direction.

BENDING MEMBER: Click the [**Use a Bending Member**] button to define a bending member (such as a column) as a resisting element. This will be a linear member whose stiffness is specified simply by its X and Y axis moments of inertia. You must also provide a value for the modulus of elasticity of the Bending Member for bending. Finally, you must make a fixity selection, which dictates the equation used to calculate deflection in BOTH directions of the member (unless the option is selected to "Force all MINOR AXIS Stiffnesses to ZERO"). Using these settings, the module will calculate the bending stiffness of the member and report the deflection for a unit 1 kip applied load.

GENERIC ELEMENT: Click the [**Use Generic Resisting Element**] button to specify a generic resisting element whose lateral deflection is known for an applied 1 kip load. This selection is intended for complex resisting elements like braced or moment frames, where another analysis module has determined the unit deflection.



Add & Delete Buttons

Use the [**Add**] and [**Delete**] buttons to add a new resisting element or delete the one currently highlighted in the list.

Element Data

This area allows you to specify a label and location of the center of resistance for a resisting element.

Resisting Element List

This is the list that you create to define the resisting element locations that give lateral force resistance to the rigid diaphragm.

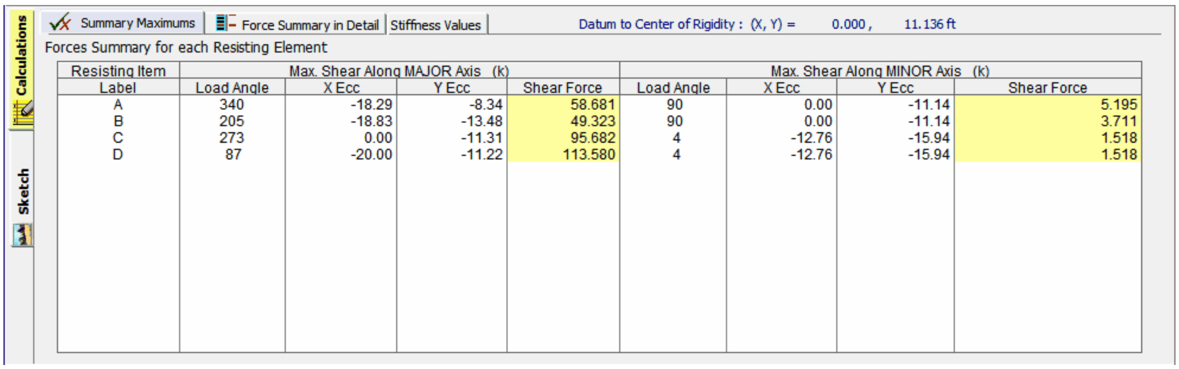
This table serves to give a summary of the deflections, location and major axis angle for each element. When you click to highlight a line in the table, the information for that resisting element is brought into the variables on the input area.

Summary Maximum Tab

Please note that a STRICT X-Y coordinate system should be used to ensure that the analysis is properly carried out. When setting up an X-Y coordinate axis, please follow the standard Cartesian model with the diaphragm.

Recall that the module calculates the forces to each resisting element by rotating the force about its point of application. That point of application is in increments around an accidental eccentricity ellipse.

This Summary Maximums tab provides the maximum forces for each resisting element along the major and minor axis of the element.

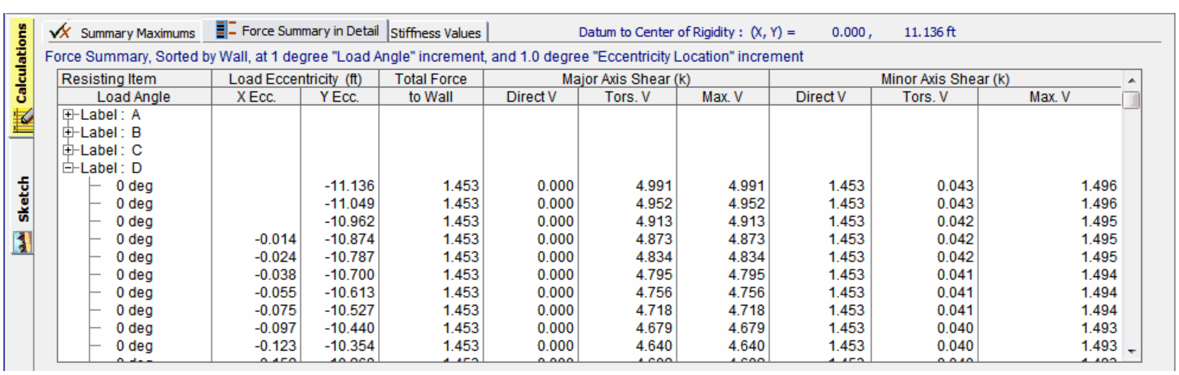


Resisting Item	Max. Shear Along MAJOR Axis (k)				Max. Shear Along MINOR Axis (k)			
Label	Load Angle	X Ecc	Y Ecc	Shear Force	Load Angle	X Ecc	Y Ecc	Shear Force
A	340	-18.29	-8.34	58.681	90	0.00	-11.14	5.195
B	205	-18.83	-13.48	49.323	90	0.00	-11.14	3.711
C	273	0.00	-11.31	95.682	4	-12.76	-15.94	1.518
D	87	-20.00	-11.22	113.580	4	-12.76	-15.94	1.518

Force Envelope Detail Tab

This tab provides the main table that shows all of the force calculations for each resisting element. It is tree structured, so clicking the [**+**] sign to the left of each item name will expand the result set for that item.

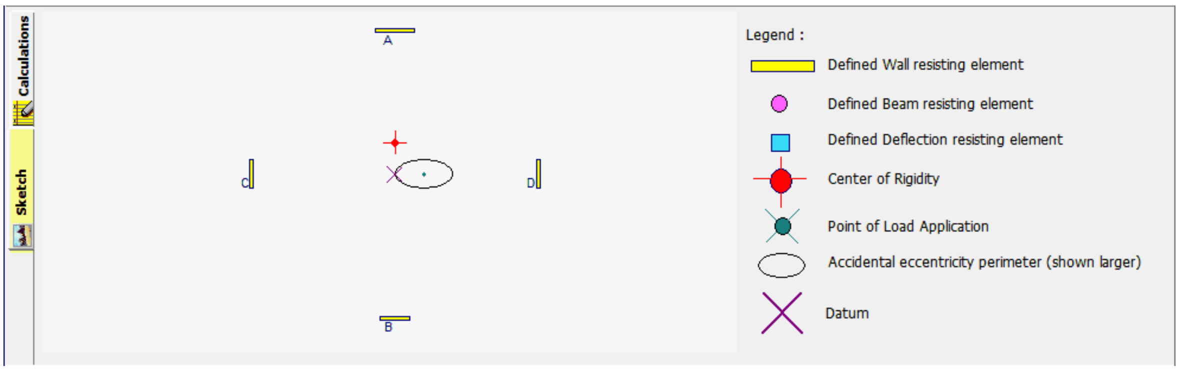
In the image below we see that the data for the wall labeled "D" is expanded. Below "Label : D" we see many lines labeled "0 deg". These are the results for the load applied at an orientation of 0 degrees. On each "0 deg" line, observe that the "X Ecc" and "Y Ecc" values are changing. These values are the locations of the applied load as it moves its way around the accidental eccentricity ellipse. The note at the top of the table indicates that the analysis is based on 15-degree "Eccentricity Location" increments. This implies that there will be (360 degrees/15 degrees) = 24 lines of data based on the "0 deg" force orientation. Then, if we scrolled down through the table, we would see that the load application angle has also been set to change in 15-degree increments as well.



Resisting Item	Load Eccentricity (ft)		Total Force	Major Axis Shear (k)			Minor Axis Shear (k)		
Load Angle	X Ecc.	Y Ecc.	to Wall	Direct V	Tors. V	Max. V	Direct V	Tors. V	Max. V
⊞-Label : A									
⊞-Label : B									
⊞-Label : C									
⊟-Label : D									
0 deg		-11.136	1.453	0.000	4.991	4.991	1.453	0.043	1.496
0 deg		-11.049	1.453	0.000	4.952	4.952	1.453	0.043	1.496
0 deg		-10.962	1.453	0.000	4.913	4.913	1.453	0.042	1.495
0 deg	-0.014	-10.874	1.453	0.000	4.873	4.873	1.453	0.042	1.495
0 deg	-0.024	-10.787	1.453	0.000	4.834	4.834	1.453	0.042	1.495
0 deg	-0.038	-10.700	1.453	0.000	4.795	4.795	1.453	0.041	1.494
0 deg	-0.055	-10.613	1.453	0.000	4.756	4.756	1.453	0.041	1.494
0 deg	-0.075	-10.527	1.453	0.000	4.718	4.718	1.453	0.041	1.494
0 deg	-0.097	-10.440	1.453	0.000	4.679	4.679	1.453	0.040	1.493
0 deg	-0.123	-10.354	1.453	0.000	4.640	4.640	1.453	0.040	1.493

Note that direct shear is always considered as a positive value, and the algebraic sign on the torsional shear component will be **positive** if its effect is **additive** to the effects of the direct shear, or **negative** if its effects are **subtractive** from the direct shear. The algebraic sign on the torsional shear component does NOT indicate its direction with respect to the overall Cartesian coordinate system.

Sketch Tab



Analysis Procedure

Please see the following description for the procedure used to calculate the system stiffness matrix and resolve the forces for each resisting element.

76.2.0 THEORY

Consider the displacements of the single wall assembly shown in Figure 1. The magnitude of the displacements are assumed to be very small. The assembly rotates through an angle Δ_θ about the origin of coordinates Y and Z, and translates Δ_y and Δ_z. The diaphragm is assumed to be very rigid compared to the walls. When the diaphragm rotates without translation, displacements occur which generate the following forces:

$$F_y = -K_{yy}\Delta_y + K_{yz}\Delta_z \tag{5}$$

$$F_z = -K_{zy}\Delta_y + K_{zz}\Delta_z \tag{6}$$

$$M_x = J\Delta_\theta \tag{7}$$

$$M_p = -zF_y + yF_z + J\Delta_\theta \tag{8}$$

in which the stiffness coefficient K_{yz} is the force in the Y direction due to a unit displacement in the Z direction. The rotational stiffness J is the moment about the X axis due to a unit rotation of the element about the X axis. All values are plotted using the right hand cartesian coordinate axis system; thus a negative sign indicates a displacement in the negative direction or a clockwise moment. F_x, M_y and M_z are zero for this type of system.

It can be shown that for the displacement in Figure 1, $\Delta_y = z\Delta\theta$ and $\Delta_z = y\Delta\theta$; thus, Equations 5 and 6 can be revised as follows: Also note that $K_{zy} = K_{yz}$

$$F_y = -zK_{yy}\Delta_\theta + yK_{yz}\Delta_\theta \tag{9}$$

$$F_z = -zK_{yz}\Delta_\theta + yK_{zz}\Delta_\theta \tag{10}$$

Equation 8 now becomes:

$$M_p = \left[z^2K_{yy} + y^2K_{zz} - 2yzK_{yz} + J\right]\Delta_\theta \tag{11}$$

When the diaphragm translates without rotation, see Figure 2, displacements occur which generate the following forces.

$$F_y = K_{yy}\Delta_y + K_{yz}\Delta_z \qquad (12)$$

$$F_z = K_{yz}\Delta_y + K_{zz}\Delta_z \qquad (13)$$

The element forces acting on the diaphragm may now be written in matrix notation.

$$\begin{Bmatrix} F_y \\ F_z \\ M_p \end{Bmatrix} = \begin{bmatrix} K_{yy} & K_{yz} & (yK_{yz} - zK_{yy}) \\ K_{yz} & K_{zz} & (yK_{zz} - zK_{yz}) \\ 0 & 0 & (z^2K_{yy} + y^2K_{zz} - 2yzK_{yz} + J) \end{bmatrix} \begin{Bmatrix} \Delta_y \\ \Delta_z \\ \Delta_\theta \end{Bmatrix} \qquad (14)$$

If the applied loads on the diaphragm are P_y and P_z in the Y and Z directions, and a torsional moment T_p, is also applied to the diaphragm in the positive, counter-clockwise direction,

$$P_y = \Sigma F_y \qquad (15)$$

$$P_z = \Sigma F_z \qquad (16)$$

$$T_p = \Sigma M_p \qquad (17)$$

Substituting the values from Equation 14 into Equation 15, 16 and 17,

$$P_y = \Delta_y \Sigma K_{yy} + \Delta_z \Sigma K_{yz} + \Delta_\theta (\Sigma yK_{yz} - \Sigma zK_{yy}) \qquad (18)$$

$$P_z = \Delta_y \Sigma K_{yz} + \Delta_z \Sigma K_{zz} + \Delta_\theta (\Sigma yK_{zz} - \Sigma zK_{yz}) \qquad (19)$$

$$T_p = \Delta_\theta (\Sigma z^2K_{yy} + \Sigma y^2K_{zz} - 2\Sigma yzK_{yz} + \Sigma J) \qquad (20)$$

Select a coordinate axis system such that;

$$\Sigma y K_{yz} - \Sigma z K_{yy} = 0 \quad (21)$$

$$\Sigma y K_{zz} - \Sigma z K_{yz} = 0 \quad (22)$$

Then,

$$P_y = \Delta_y \Sigma K_{yy} + \Delta_z \Sigma K_{yz} \quad (23)$$

$$P_z = \Delta_y \Sigma K_{yz} + \Delta_z \Sigma K_{zz} \quad (24)$$

$$T_p = \Delta_\theta J_P \quad (25)$$

$$\text{where } J_p = \Sigma z^2 K_{yy} + \Sigma y^2 K_{zz} - 2\Sigma yz K_{yz} + \Sigma J \quad (26)$$

The solution for F_y, F_z and M_x forces and torsional moment acting on a wall, is then found as follows:

1. Compute values for K_{yy}, K_{zz}, K_{yz}. See Example 1.
2. Locate coordinate axis system as defined in Equations 21 and 22.
3. Compute displacements Δ_y, Δ_z, and Δ_θ from Equations 23, 24 and 25.
4. Solve for forces from Equations 14 and 7.

The solution for the origin of coordinate axis as defined in Equation 21 and 22, can be accomplished if any set of trial axes are taken y_1 and z_1 with distances $\bar{y}$ and $\bar{z}$ to the desired origin. From Figure 3,

$$y = y_1 - \bar{y} \quad (27)$$

$$z = z_1 - \bar{z} \quad (28)$$

substituting Equation 27 and 28 into Equation 21 and 22

$$\Sigma\ (y_1-\bar{y})\ K_{yz} - \Sigma\ (z_1-\bar{z})\ K_{yy} = 0 \quad (29)$$
$$\Sigma\ (y_1-\bar{y})\ K_{zz} - \Sigma\ (z_1-\bar{z})\ K_{yz} = 0 \quad (30)$$

Then,

$$\bar{z}\ \Sigma\ K_{yy} - \bar{y}\Sigma K_{yz} = \Sigma z_1 K_{yy} - \Sigma y_1 K_{yz} \quad (31)$$
$$\bar{z}\ \Sigma\ K_{yz} - \bar{y}\Sigma K_{zz} = \Sigma z_1 K_{yz} - \Sigma y_1 K_{zz} \quad (32)$$

Now must solve two simultaneous equations (Equation 31 and 32) for $\bar{y}$ and $\bar{z}$.
If $K_{yz} = 0$, the problem is greatly simplified.

$$\Sigma y K_{zz} = 0$$

$$\Sigma\ z K_{yy} = 0$$

$$\Delta_y = P_y \div \Sigma K_{yy}$$

$$\Delta_z = P_z \div \Sigma K_{zz}$$

$$\Delta_\theta = Tp \div Jp$$

$$F_y = \Delta_y K_{yy} - \Delta_\theta (z K_{yy})$$
$$= \frac{P_y K_{yy}}{\Sigma K_{yy}} - \frac{T_p (z K_{yy})}{J_p}$$

$$F_z = \Delta_z K_{zz} + \Delta_\theta\ (y K_{zz})$$
$$= \frac{P_z\ K_{zz}}{\Sigma K_{zz}} + \frac{T_p (y K_{zz})}{J_p}$$

$$Ma = J\Delta_\theta$$
$$= \frac{T_p J}{J_p}$$

The stiffness coefficients K_{yy}, K_{zz} and K_{yz} may be found by inverting the flexibility matrix of the lateral load resisting wall system. See Example Problem 1.

10.9.2 Point Load on Slab

Need more? Ask Us a Question

This module calculates the capacity of an unreinforced concrete slab to support isolated concentrated loads. Typical use is for legs of storage racks not supported by a building structure, and is not within the scope of the ACI code.

The design method is based on the recent research of Shentu, Jiang and Hsu. For further information see (1) "Load carrying capacity for concrete slabs on grade" in the ASCE Journal of Structural Engineering January 1997; (2) Acceptable Design & Analysis methods for use of slabs-on-grade foundations, City of Los Angeles LAMC91.1806 and (3) Seismic considerations for steel storage racks, FEMA 460 September 2005.

The work of Shentu and colleagues has shown that load carrying capacity, verified with test results, can be very closely predicted using the formulas given below.

Rather than historical elastic methods, the method used here is elasto-plastic which is more applicable to ultimate capacity determination.

Allowable load capacity is given by this equation:

$$P_n = 1.72\ [(k_s\ R_1\ /\ E_c)\ 10{,}000 + 3.60] * f_t' * d^2$$

Where

k_s is the modulus of subgrade reaction of the soil, pci

R_1 is sqrt(Plate Width * Plate Length) / 2, inches

E_c is the concrete elastic modulus, psi

f_t' is the tensile strength of the concrete = 7.5 sqrt(f'c), psi

d is the slab thickness, inches

The above equation assumes that the load acting on the slab is unique and no other nearby loads are affecting the calculation.

To assist in the evaluation of slabs-on-grade, this module also provides a calculation of the distance that the closest load may be without affecting the calculated slab capacity. The calculation given below is based on research of Packard, Pickett & Ray and more recently by Spears and Panarese. It is also discussed in ACI 360R-92(4).

In this module the distance is calculated as 1.5 * "Radius of Relative Stiffness" given by the following equation:

$$b = [\ E_c\ d^3\ /\ (12 * (1-u^2) * k_s)]^{\ 0.25}$$

Where

b is the radius of relative stiffness

E_c is the concrete elastic modulus, psi

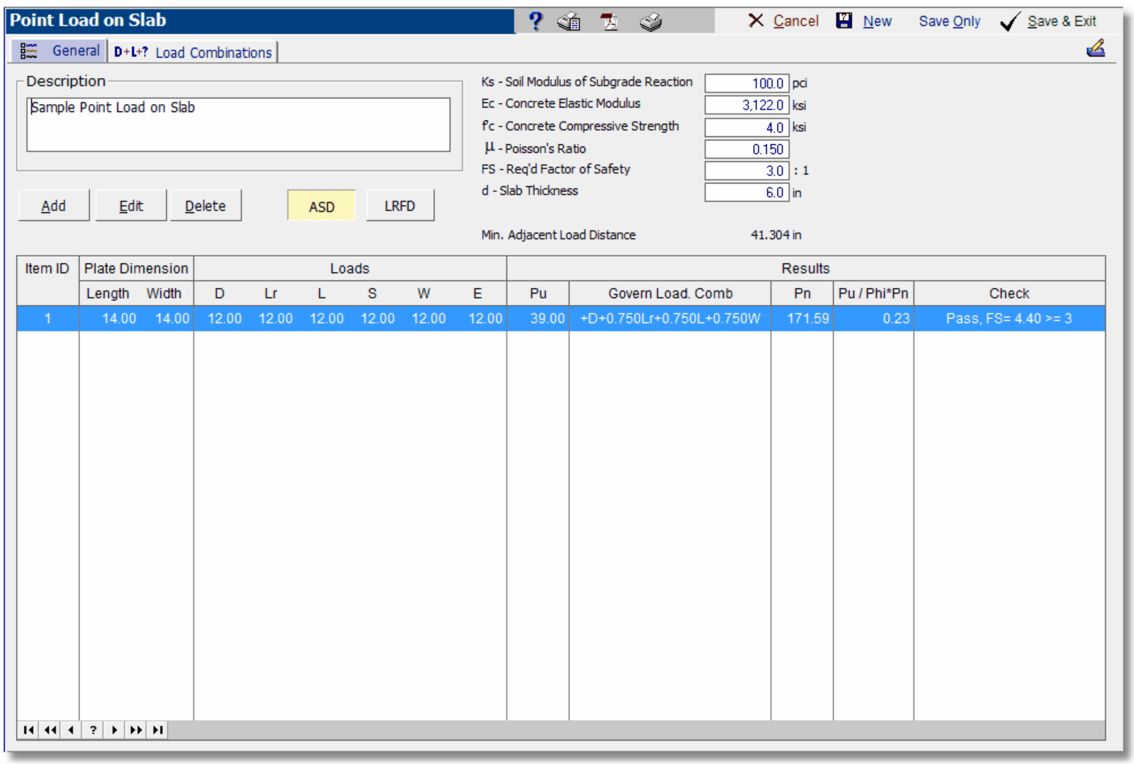
d is the slab thickness, inches

u is Poisson's ratio which is set to 0.15 in this module

k_s is the modulus of subgrade reaction of the soil, pci

Additionally this module allows the user to enter a Factor of Safety that is used when the module reports the adequacy of each applied load.

Tabular Entry Screen

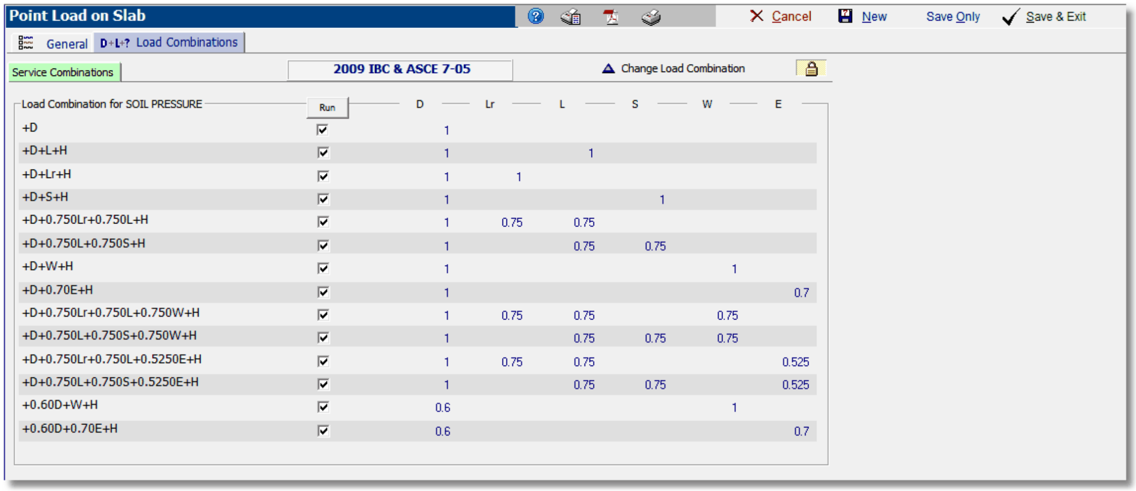


This module is designed to allow the user to create a table of loads applied to a particular concrete slab and supporting soil with one set of material properties.

You can then use the [**Add**], [**Edit**] and [**Delete**] buttons to add a set of applied loads and base plate dimensions. From this data all load combinations are used to determine the maximum axial force. For the plate dimension you specify, the maximum load capacity for the point load application is calculated and compared with your required factor of safety.

The option for ASD or LRFD analysis only changes the load combination set used. Because this is a non-ACI design process, you need to enter a Factor of Safety to determine the final design status. Research material suggests a F.S. of 3.0.

Load Combinations



Load Combination for SOIL PRESSURE	Run	D	Lr	L	S	W	E
+D	☑	1					
+D+L+H	☑	1		1			
+D+Lr+H	☑	1	1				
+D+S+H	☑	1			1		
+D+0.750Lr+0.750L+H	☑	1	0.75	0.75			
+D+0.750L+0.750S+H	☑	1		0.75	0.75		
+D+W+H	☑	1				1	
+D+0.70E+H	☑	1					0.7
+D+0.750Lr+0.750L+0.750W+H	☑	1	0.75	0.75		0.75	
+D+0.750L+0.750S+0.750W+H	☑	1		0.75	0.75	0.75	
+D+0.750Lr+0.750L+0.5250E+H	☑	1	0.75	0.75			0.525
+D+0.750L+0.750S+0.5250E+H	☑	1		0.75	0.75		0.525
+0.60D+W+H	☑	0.6				1	
+0.60D+0.70E+H	☑	0.6					0.7

10.9.3 General Section Property Calculator

Need more? Ask Us a Question

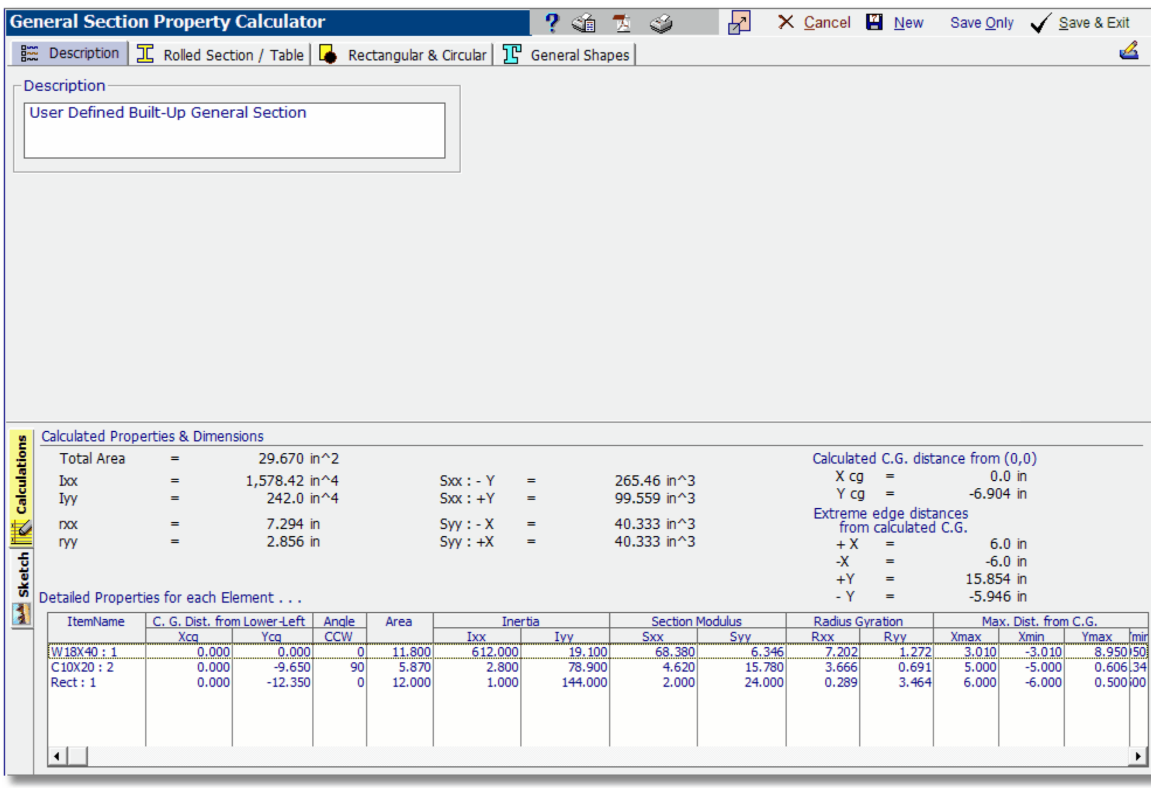
Overview

This module determines section properties for built-up sections with rectangles, hollow circles, solid circles, standard AISC steel sections and general multi-sided solid shapes.

Click here for a video: General Section Property Calculator

AISC sections can be recalled from the database files and can be included in the built-up section. All sections from the 13th Edition AISC Steel Construction Manual are available, and can be reoriented as necessary.

The calculated section property values include: area, moments of inertia, center of gravity location, extreme fiber distances, section moduli, and radius of gyration.



Basic Usage

- Before starting data entry, be sure you have set up an X-Y coordinate system to consistently reference all component locations.

- For each rectangular shape, enter the height, width, and center of area measured from the datum.

- Hollow circular sections are entered by specifying the outside radius and thickness. Solid circular sections are entered by specifying the outside radius and setting the thickness to zero.

- For AISC sections, you can use the Xcg and Ycg input fields to locate the section's centroid position with respect to the datum. The module knows the centroid location of AISC members with respect to their own extreme fiber locations. However, you need to enter the location of the member's centroid in relation to the other members in the built-up section. Be careful, as this can be tricky when entering channels, angles, and tee sections that are rotated.

- A unique feature allows the user to specify that AISC sections can be rotated in 90-degree increments, and steel angle sections can also be mirrored about their Y axis.

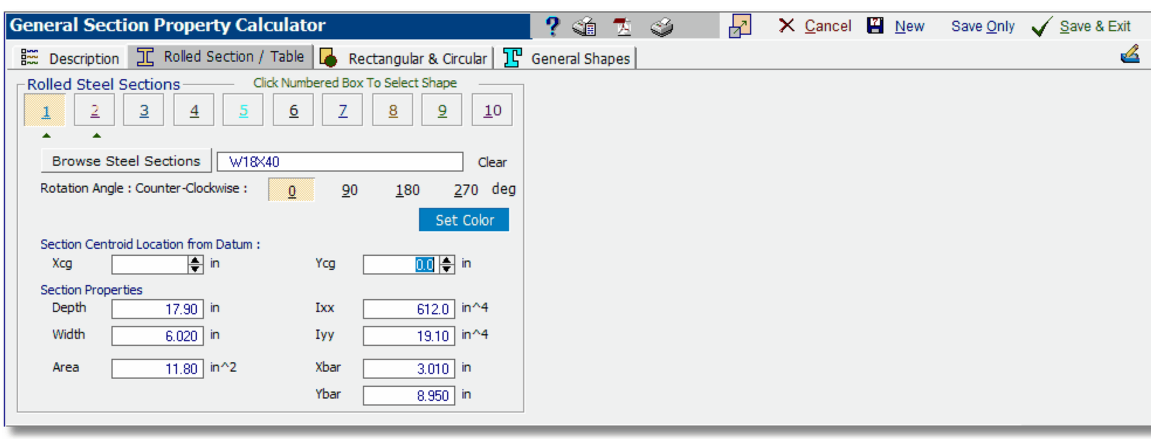
Assumptions & Limitations

The module operates on a simple calculation procedure:

- Calculate the moment of inertia of each shape,
- Calculate the neutral axis of the group of shapes, and
- Calculate the moment of inertia of the group using I + A*D^2 equations.

More complex analysis such as polar moment of inertia, plastic moduli, and buckling constants are beyond the scope of the module at this time but continuing development will add these items in future updates.

Rolled Section / Table tab



This tab enables you to specify up to 10 sections from the AISC 13th Edition database to use in a built-up member.

The square buttons across the top of the tab are used to represent the component sections that comprise your built-up shape. When a section has been specified for a particular button, a small green upward facing triangle will be shown under the corresponding button. Click on any button to add a section or view and modify the section that has already been assigned to that button.

Note: It is important to understand that the numbered buttons on the various tabs DO NOT represent different built-up shapes. Instead, each instance of this module only creates ONE built-up shape, and the overall built-up shape consists of a composite of ALL sections that currently exist on ANY buttons in the Rolled Section / Table tab, the Rectangular & Circular tab, and the General Shapes tab.

To insert an AISC section you can:

- Type in the section name and press [**Tab**]. The module will search the database and retrieve the information.
- Use the [**Browse Steel Sections**] button to display the steel database where you can navigate and select the desired section.

Rotation Angle: Counter-Clockwise

If you need to rotate a section, click one of the four angular rotation buttons.

Rotate Section 180 degrees about its own y-y Axis

This checkbox option will only be displayed for single-angle sections. It offers the ability to mirror a single-angle section if needed.

Xcg & Ycg

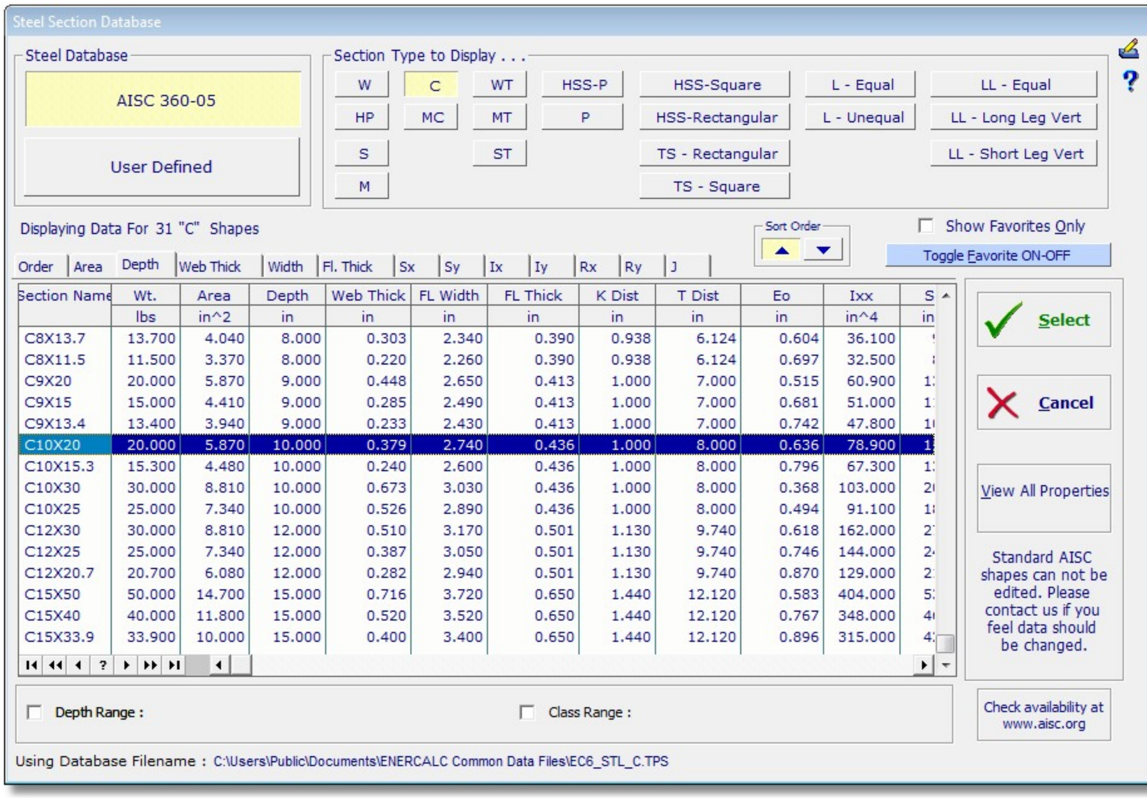
Enter the location of the section's centroidal axis measured from the datum (the origin of your assumed X-Y Cartesian coordinate system).

Section Properties

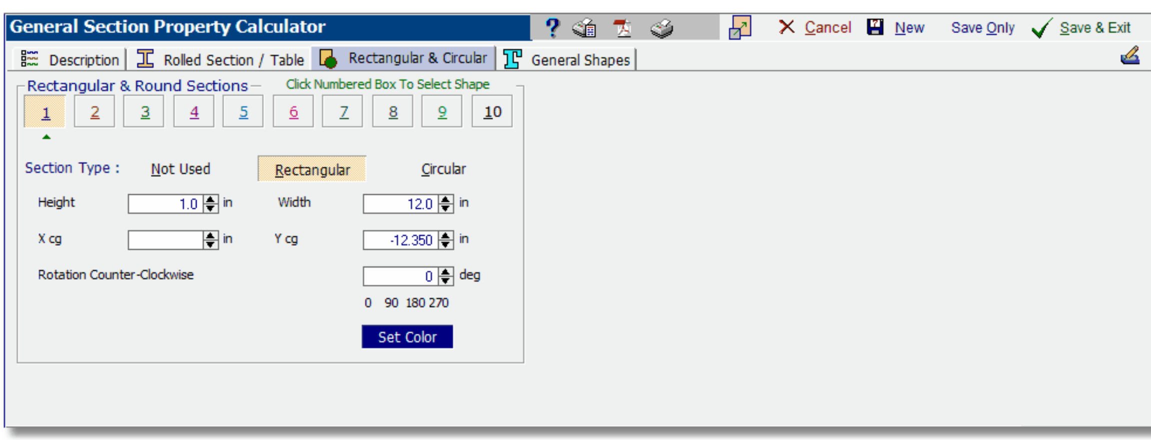
These values will be filled in after you make your choice from the AISC database. HOWEVER you can alter these values yourself. Of particular importance for unsymmetrical sections is entering the correct "Xbar" and "Ybar" location. This is the distance from the lower-left edge of the section, measured upwards and to the right, to the centroidal axis position of the section.

Steel Section Database

Click the [**Browse Steel Sections**] button to display the AISC database window:



Rectangular & Circular tab



This tab allows you to specify simple rectangular and circular shapes.

The square buttons across the top of the tab are used to represent the component sections that comprise your built-up shape. When a section has been specified for a particular button, a small green upward facing triangle will be shown under the corresponding button. Click on any button to add a section or view and modify the section that has already been assigned to that button.

Not Used / Rectangular / Circular

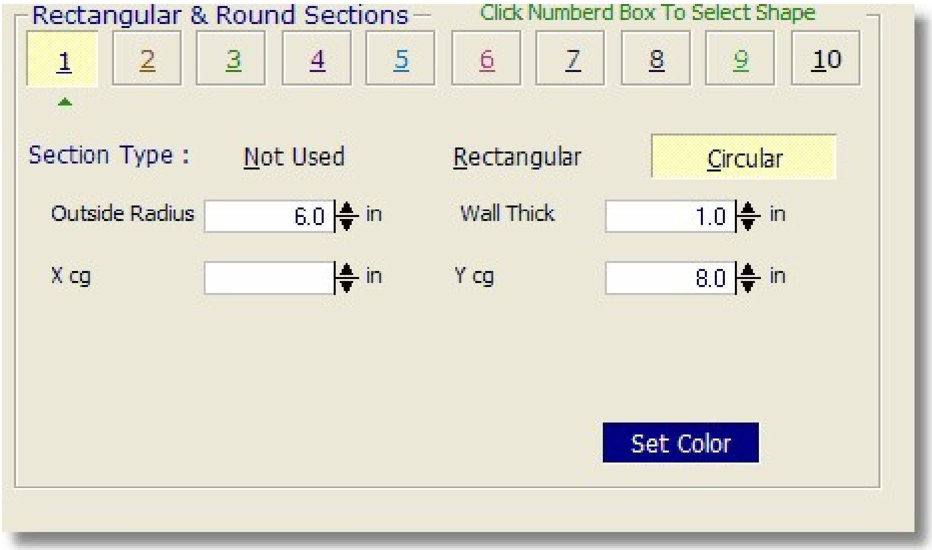
Select the shape you wish to use for this item.

Rectangular Data Entry

When a rectangular shape is chosen the data entry consists of height and width.

Circular Data Entry

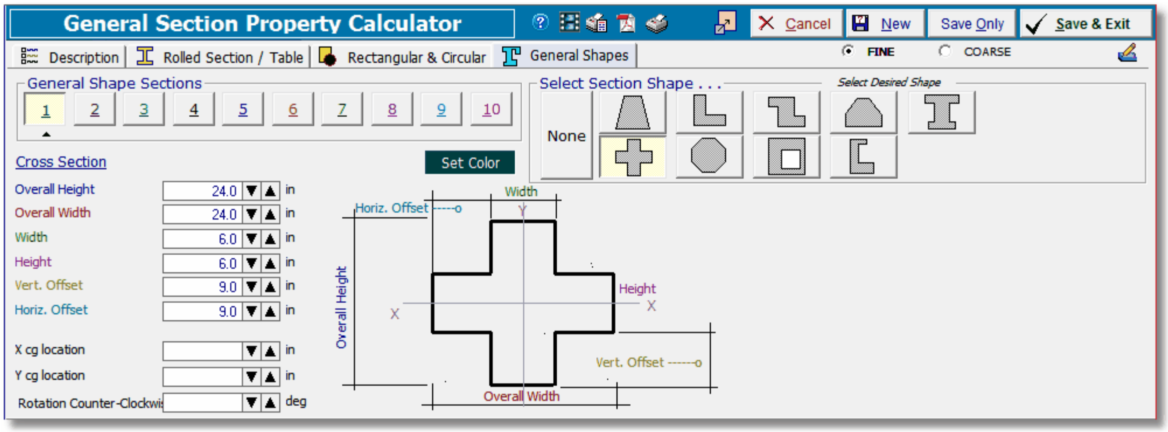
When a circular shape is selected the data entry consists of Outside Radius and Wall Thickness (not inside radius). To model a solid circular section, enter the appropriate Outside Radius and set the Wall Thickness to zero.



Xcg & Ycg

Enter the location of the section's centroidal axis measured from the datum (the origin of your assumed X-Y Cartesian coordinate system).

General Shapes tab



This tab allows you to select from a number of common polygonal shapes. With each selection the reference drawing and data entry prompts will change.

The square buttons across the top of the tab are used to represent the component sections that comprise your built-up shape. When a section has been specified for a particular button, a small green upward facing triangle will be shown under the corresponding button. Click on any button to add a section or view and modify the section that has already been assigned to that button.

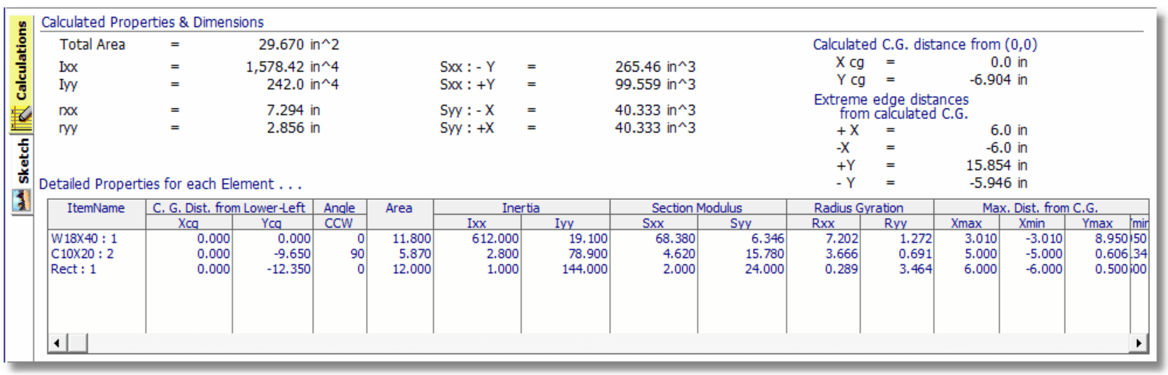
Xcg & Ycg

Enter the location of the section's centroidal axis measured from the datum (the origin of your assumed X-Y Cartesian coordinate system).

Rotation Angle: Counter-Clockwise

For these shapes you can rotate the section in one-degree increments. Positive angles represent counter-clockwise rotation.

Results Tab



Detailed Properties Table

This table summarizes each of the component items you have added to the section. It reports their individual locations, properties and maximum distance from CG for each of the four edges.

Note: This table scrolls to the right. Just use the scroll bar along the bottom of the table.

Total Area

The total area of all defined shapes, including the area of any AISC sections which have been included in the built-up shape.

Inertia: Ixx & Iyy

The overall moment of inertia of the composite section is determined by applying the following equation to all the defined shapes:

$$Ixx = Iox + (A * d_y^{\ 2}) \text{ and } Iyy = Ioy + (A * d_x^{\ 2})$$

where d = Distance from the shape's C.G. to the overall C.G. of the composite section, measured in the direction indicated by the subscript.

Section Modulus: Sxx and Syy

These values are the calculated section moduli of the composite section. The values are determined by dividing Ixx or Iyy by the extreme fiber distances above, below, right, and left of the center of gravity of the section.

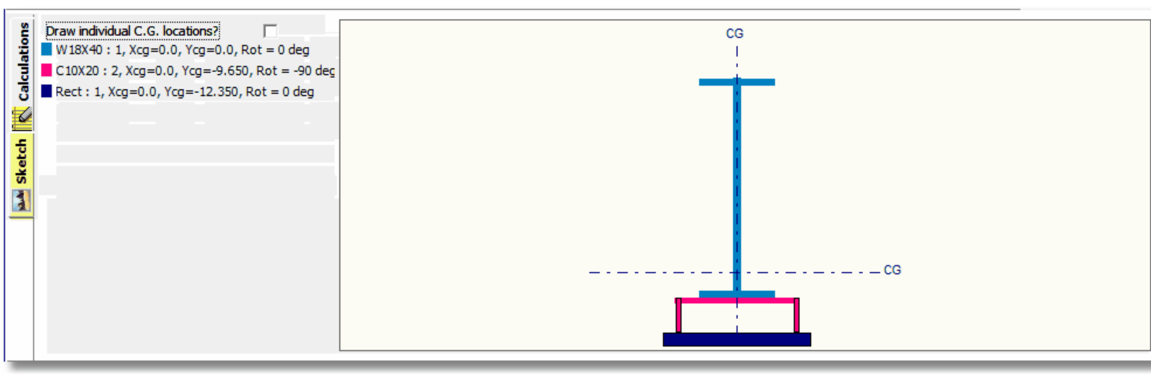
Radius of Gyration

The radius of gyration of the composite section is determined using the typical equation: $rxx = (Ixx/A)^{1/2}$ and $ryy = (Iyy/A)^{1/2}$.

Max Distance from CG

For each of the sections in the built-up shape, these columns report the distance from the extreme fibers of that section to the C.G. of the composite section.

Sketch



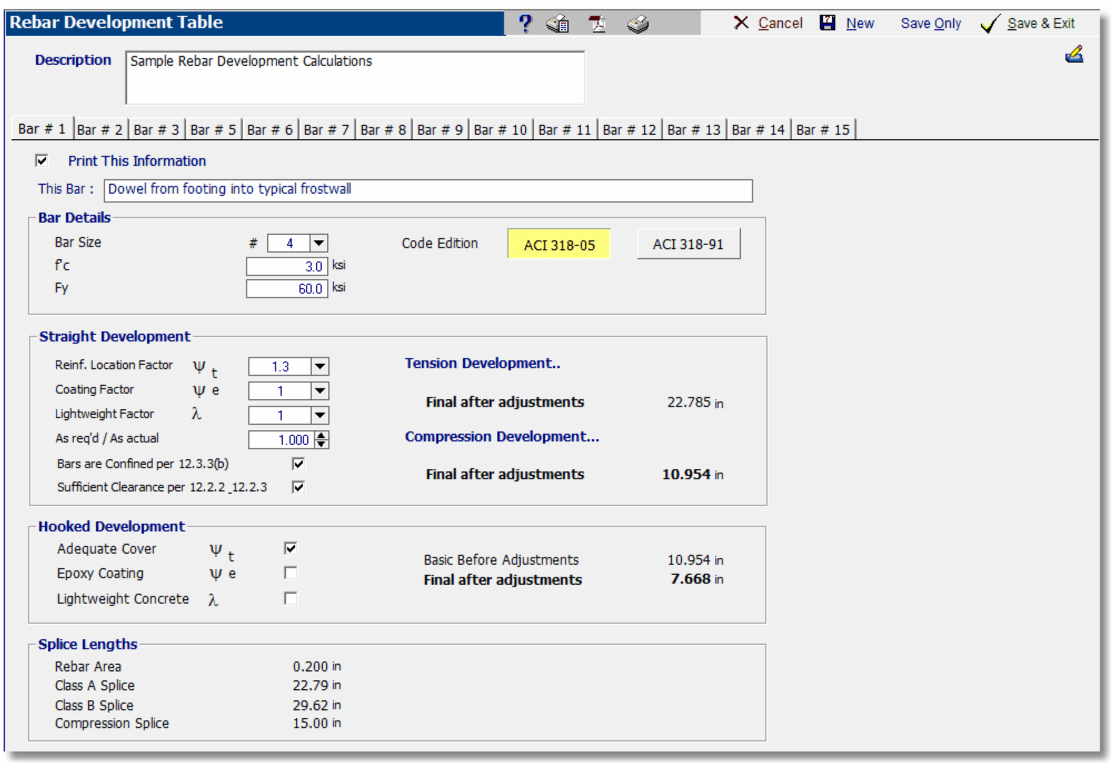
10.9.4 Rebar Development Table

Need more? Ask Us a Question

This module calculates development and splice lengths for deformed steel rebar in concrete. Since typical development lengths are readily available in tables, this module is primarily for developing tables of development lengths when unusual conditions affect the lengths.

The module uses ACI 318-05 sections 12.1, 12.2, 12.3 and 12.5 for straight tension & compression development and hooked tension development.

Two major categories of development calculation are available; straight development and hooked development. The module calculates the basic tension development length for straight and hooked cases, and compression development for straight cases only.



Item Descriptions

Print This Information

Check this box if you want the development calculation on the current tab to appear on the printout.

This Bar

Enter a description for the current development calc.

Bar Details

Enter the basic stress info for this bar. You can also select to perform the calculation according to ACI 318-91.

Straight Development

This section performs tension & compression bar development according to ACI section 12.2 and 12.3. Please refer to the ACI code for the proper definitions of these entries.

Hooked Development

This section performs tension hooked rebar development according to ACI section 12.5. Please refer to the ACI code for the proper definitions of these entries.

10.9.5 Steel Bolt Group Analysis

Need more? Ask Us a Question

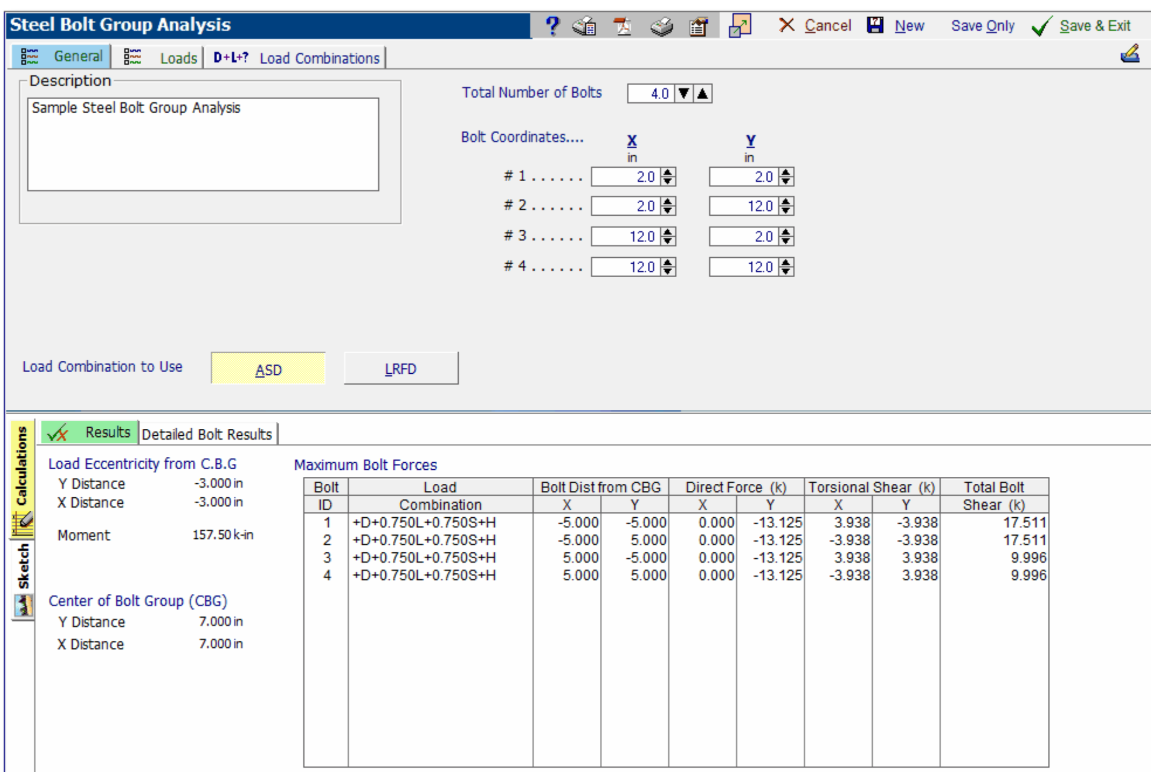
This module applies the Elastic Method to determine the force distribution from loads applied to a group of up to 16 bolts.

The required input data includes vertical and horizontal load magnitudes and their location with respect to a datum point, along with the coordinates of up to sixteen bolts.

Using these force and bolt coordinates, the module calculates direct shears and torsional shears on each bolt due to its relative location within the group.

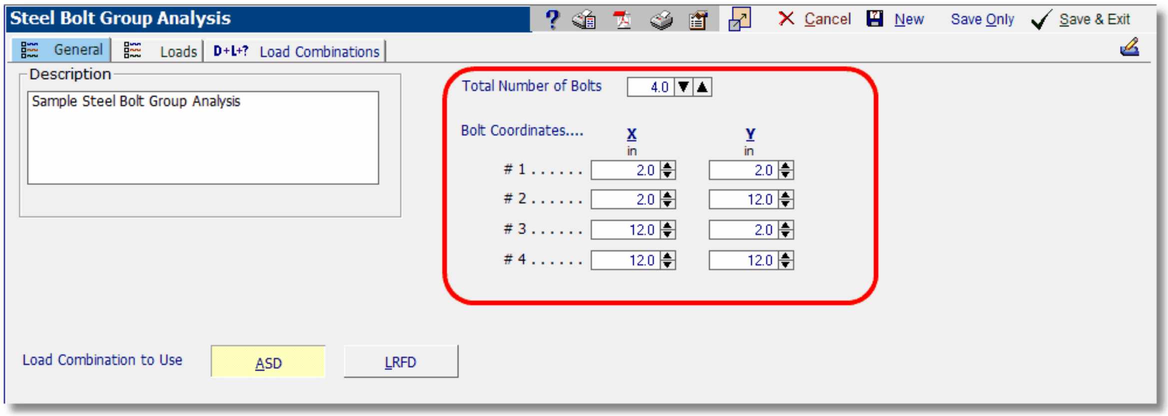
Note:

- At least two bolts should be specified.
- All bolts are assumed to be of the same deformation characteristics when loads are distributed.
- Vertical and Horizontal forces are divided by the number of fasteners to give direct shears.



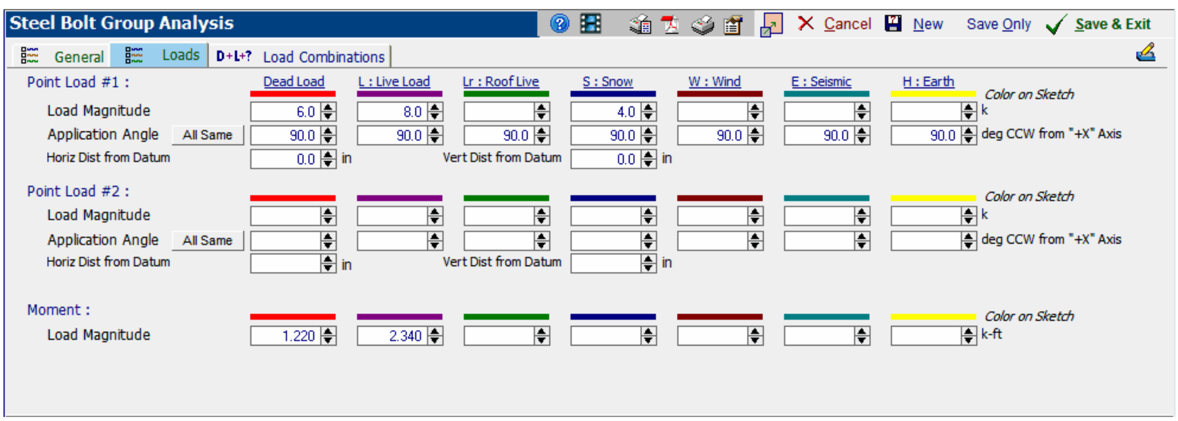
General Tab

Specify the locations up to 16 bolts. Use the Total Number of Bolts entry to indicate bolt quantity. The correct number of data entry locations will be displayed.



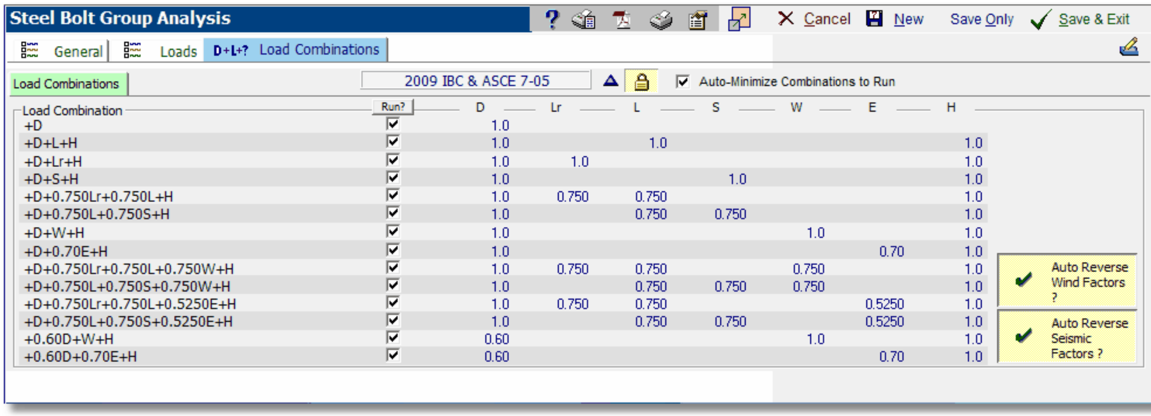
Loads Tab

Enter the magnitude of the applied loads, the angle of load application measured CCW from Cartesian zero degrees and the location of the point of load application relative to the Datum point. The module now accepts applied moments as well.



Load Combinations Tab

Based upon the selection of ASD or LRFD factoring, the appropriate load combinations will be displayed on the load combinations tab.

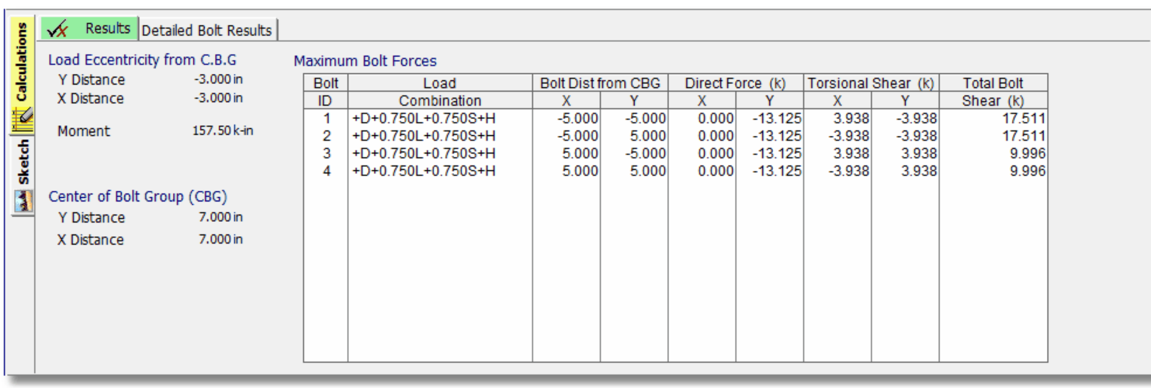


Load Combination	Run?	D	Lr	L	S	W	E	H
+D	☑	1.0						
+D+L+H	☑	1.0		1.0				1.0
+D+Lr+H	☑	1.0	1.0					1.0
+D+S+H	☑	1.0			1.0			1.0
+D+0.750Lr+0.750L+H	☑	1.0	0.750	0.750				1.0
+D+0.750L+0.750S+H	☑	1.0		0.750	0.750			1.0
+D+W+H	☑	1.0				1.0		1.0
+D+0.70E+H	☑	1.0					0.70	1.0
+D+0.750Lr+0.750L+0.750W+H	☑	1.0	0.750	0.750		0.750		1.0
+D+0.750L+0.750S+0.750W+H	☑	1.0		0.750	0.750	0.750		1.0
+D+0.750Lr+0.750L+0.5250E+H	☑	1.0	0.750	0.750			0.5250	1.0
+D+0.750L+0.750S+0.5250E+H	☑	1.0		0.750	0.750		0.5250	1.0
+0.60D+W+H	☑	0.60				1.0		1.0
+0.60D+0.70E+H	☑	0.60					0.70	1.0

Results Tab

On the left side of this tab are the calculated Center of Bolt Group location, calculated moment applied to the group from the load combination that creates the extreme moment and the eccentricity of the load application from the Center of Bolt Group.

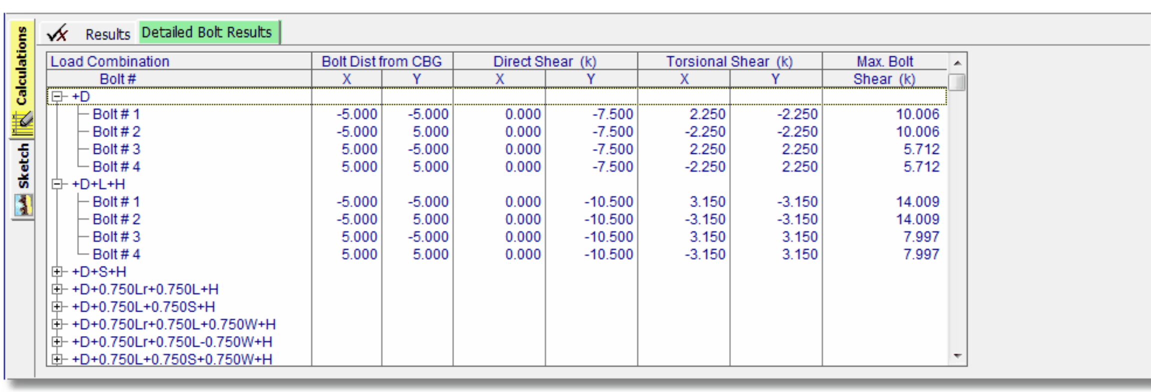
In the table on the right are presented the calculated of force for each bolt due to direct and torsional shears for each axis (after the applied load is reduced to X & Y components).



Bolt ID	Load Combination	Bolt Dist from CBG X	Bolt Dist from CBG Y	Direct Force (k) X	Direct Force (k) Y	Torsional Shear (k) X	Torsional Shear (k) Y	Total Bolt Shear (k)
1	+D+0.750L+0.750S+H	-5.000	-5.000	0.000	-13.125	3.938	-3.938	17.511
2	+D+0.750L+0.750S+H	-5.000	5.000	0.000	-13.125	-3.938	-3.938	17.511
3	+D+0.750L+0.750S+H	5.000	-5.000	0.000	-13.125	3.938	3.938	9.996
4	+D+0.750L+0.750S+H	5.000	5.000	0.000	-13.125	-3.938	3.938	9.996

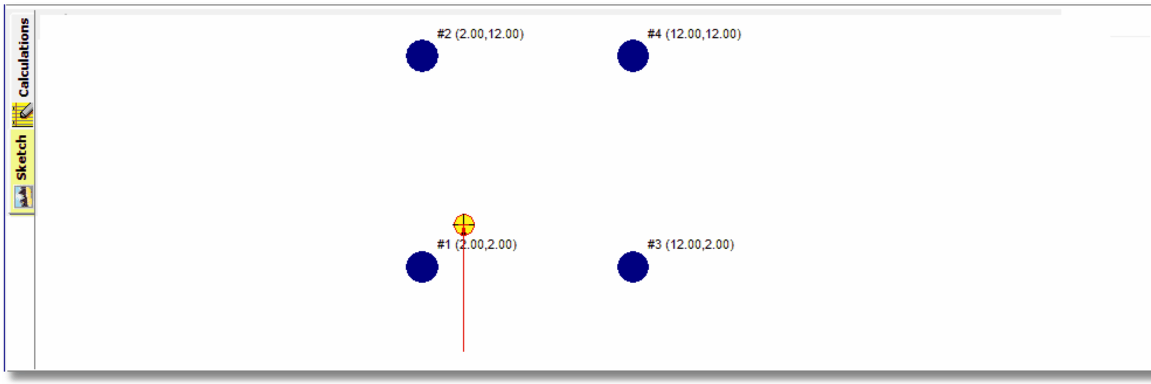
Detailed Bolt Results

This tab provides the detailed calculations for all load combinations and all bolts.



Load Combination / Bolt #	Bolt Dist from CBG X	Bolt Dist from CBG Y	Direct Shear (k) X	Direct Shear (k) Y	Torsional Shear (k) X	Torsional Shear (k) Y	Max. Bolt Shear (k)
+D							
Bolt # 1	-5.000	-5.000	0.000	-7.500	2.250	-2.250	10.006
Bolt # 2	-5.000	5.000	0.000	-7.500	-2.250	-2.250	10.006
Bolt # 3	5.000	-5.000	0.000	-7.500	2.250	2.250	5.712
Bolt # 4	5.000	5.000	0.000	-7.500	-2.250	2.250	5.712
+D+L+H							
Bolt # 1	-5.000	-5.000	0.000	-10.500	3.150	-3.150	14.009
Bolt # 2	-5.000	5.000	0.000	-10.500	-3.150	-3.150	14.009
Bolt # 3	5.000	-5.000	0.000	-10.500	3.150	3.150	7.997
Bolt # 4	5.000	5.000	0.000	-10.500	-3.150	3.150	7.997
+D+S+H							
+D+0.750Lr+0.750L+H							
+D+0.750L+0.750S+H							
+D+0.750Lr+0.750L+0.750W+H							
+D+0.750Lr+0.750L-0.750W+H							
+D+0.750L+0.750S+0.750W+H							

Sketch


Calculations
Sketch
#2 (2.00,12.00)
#4 (12.00,12.00)
#1 (2.00,2.00)
#3 (12.00,2.00)

10.9.6 Steel Base Plate

Need more? Ask Us a Question

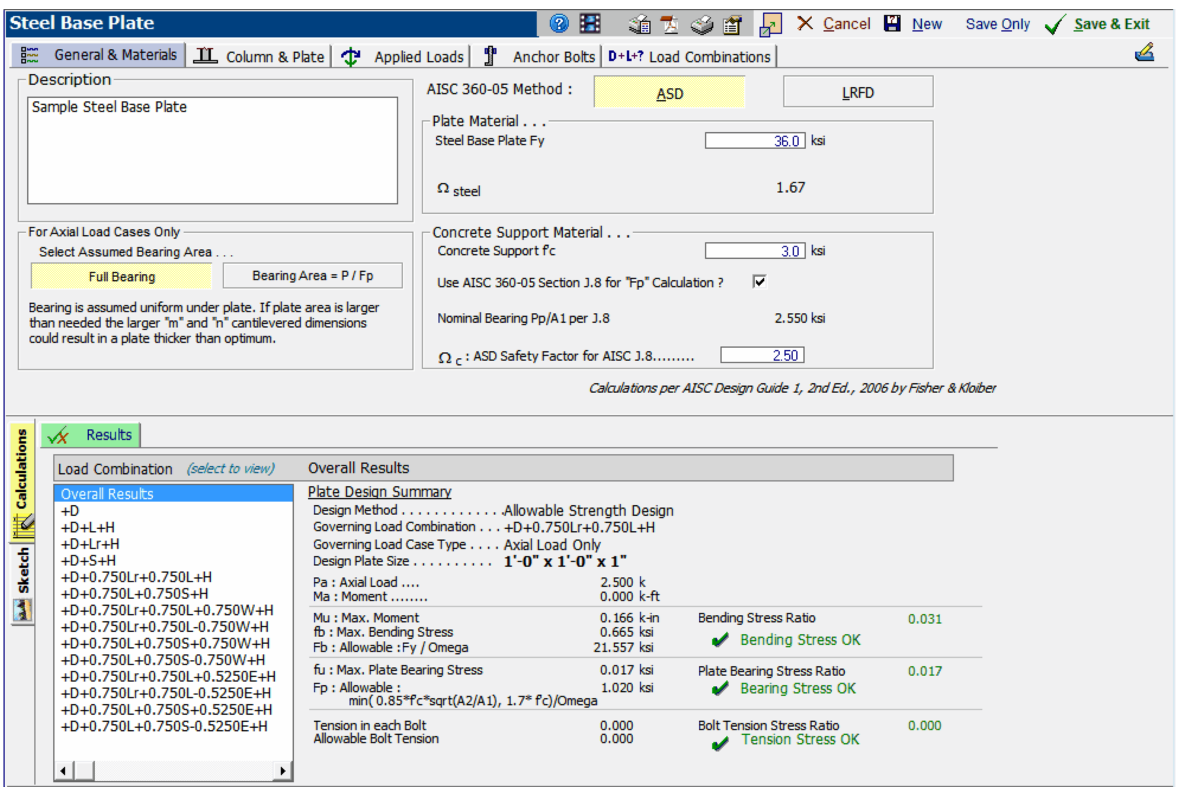
This module designs steel column base plates according to the 13th Edition AISC Steel Construction Manual and the AISC Design Guide 1, Second Edition. Click here for a video:

Steel Base Plate

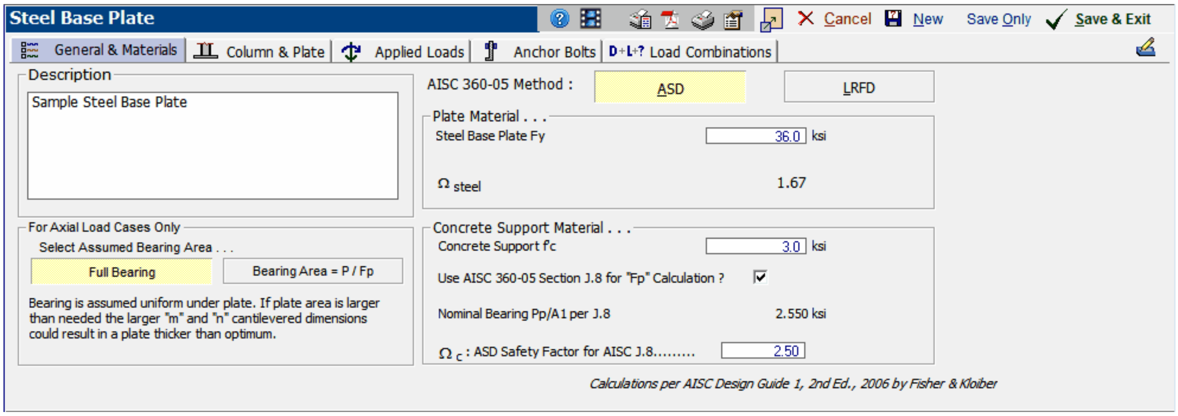
This module handles base plate design for the following conditions:

- where the resultant eccentricity is zero,
- where the resultant eccentricity is within the middle third for full bearing pressure,
- where the resultant eccentricity is outside the middle third resulting in a triangular pressure distribution on part of the base plate, and
- extreme eccentricity conditions where anchor bolts are required.

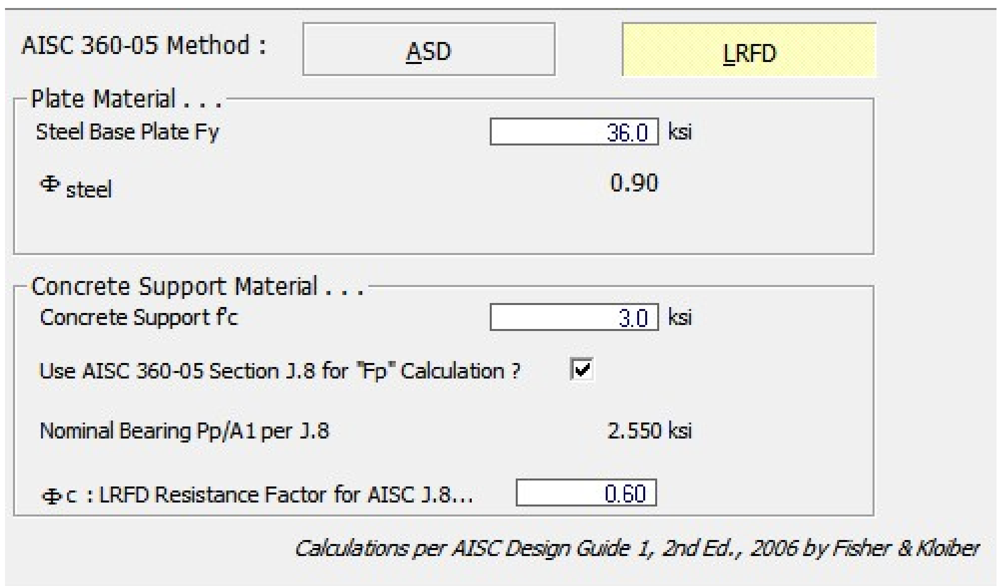
This module does not handle conditions where NET UPLIFT is present. When the summation of factored axial loads in a load combination is negative, then uplift controls and the module will not calculate. Under these conditions, a red error message will be displayed at the bottom of the window to notify you of this condition.



General & Materials Tab



The right side of this tab will look slightly different for LRFD:



For Axial Load Cases Only....

This selection controls the upward pressure used to design the plate. A brief description of each choice is provided below the buttons for each selection.

Full Bearing: Bearing is assumed uniform under plate. If plate area is larger than needed, the larger "m" and "n" cantilevered dimensions could result in a thicker plate than would be required if the pressure was calculated by the other option.

Bearing Area = P / Fp: Base plate is considered flexible with bearing concentrated close to column. Maximum Fp is used to calculate "design" minimum plate size. Determining the bearing pressure by this method might result in thicker plates if bending stress is high between webs or within pipe or tube walls.

Steel Design Method

Select between ASD or LRFD design methods.

Steel Base Plate Fy

Specify the yield strength of the base plate material.

ASD: Omega

Enter the capacity reduction factor, Omega, to be used in ASD per AISC 360.

LRFD: Phi

Enter the capacity reduction factor, Phi, to be used in LRFD per AISC 360.

Concrete Support f'c

28-day compressive strength of concrete used to support the base plate.

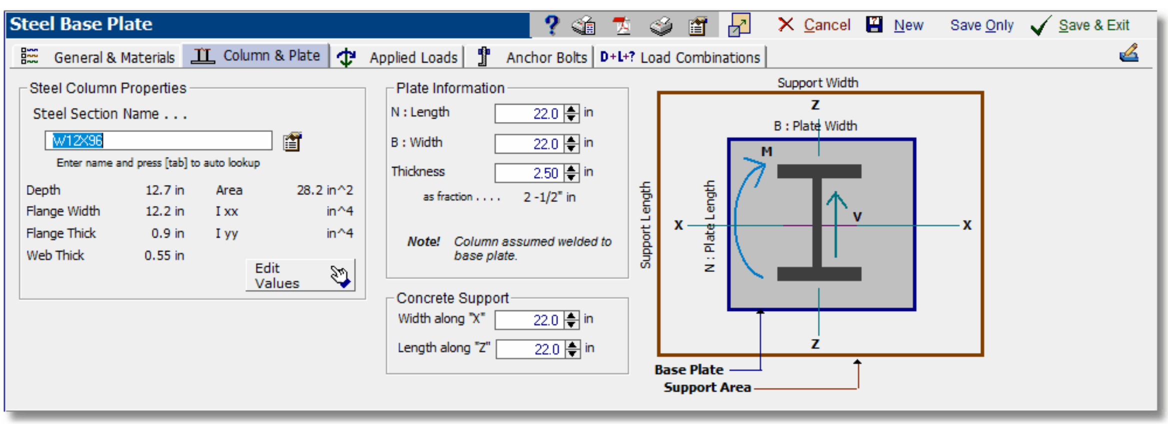
ASD: Omega per AISC J.8

AISC 360 Section J.8 specifies Omega as 2.5. This entry allows the value to be modified.

LRFD: Phi per AISC J.8

AISC 360 Section J.8 specifies Phi as 0.6. This entry allows the value to be modified.

Column & Plate Tab



Steel Section Name & Database Button

Type the AISC section name in the entry and press [**Tab**]. The module will look up the section in the Steel database and, if found, will retrieve the values. The name must be typed just as it appears in the 13th Edition AISC Steel Construction Manual.

Or click the [**Section Database**]  button to display the built-in steel database and select a section.

[Edit Values] Button

Clicking this button will allow you to enter the steel properties.

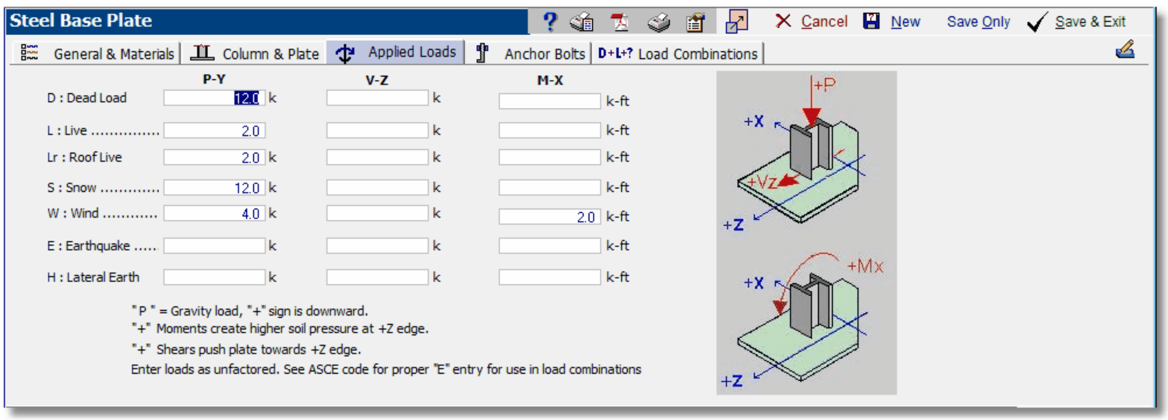
Plate Information

Enter the length, width and thickness of the base plate. Use the  buttons to quickly change the values...the results are instantly recalculated.

Concrete Support

Enter the support dimensions that will be used to calculate the allowable bearing pressure increase, A1 * sqrt(A2/A1).

Applied Loads Tab



Py - Axial Load

This column of entries specifies the axial load applied to the base plate. Note that positive values represent downward loads.

Vz - Shear

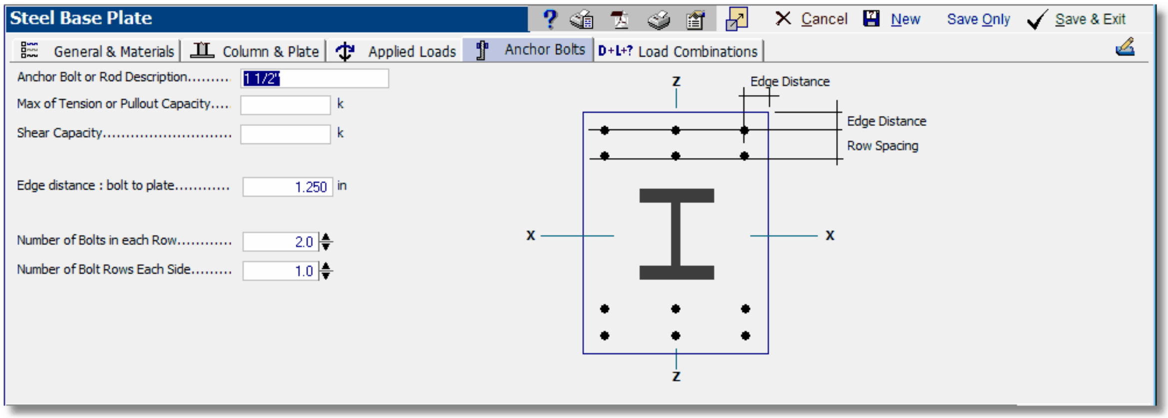
This column of entries specifies the shear applied parallel to the plate and to be resisted by the anchor bolts.

Mx - Moment

This column of entries specifies the moment that the column applies to the plate.

Anchor Bolts Tab

Items on this tab specify the strength and location of the anchor bolts that will resist shear and tension.



Description

Description of the bolt for your reference. Not used by module.

Tension Capacity

Net tension capacity of bolt after all capacity factors are applied.

Shear Capacity

Net shear capacity of bolt after all capacity factors are applied.

Edge Distance

Distance from edge of plate to center of bolt closest to edge.

Number of Bolts in a Row

The "Row" referred to is a row of bolts at the plate edge that will take any tension force.

Number of Bolt Rows

Number of rows of bolts.

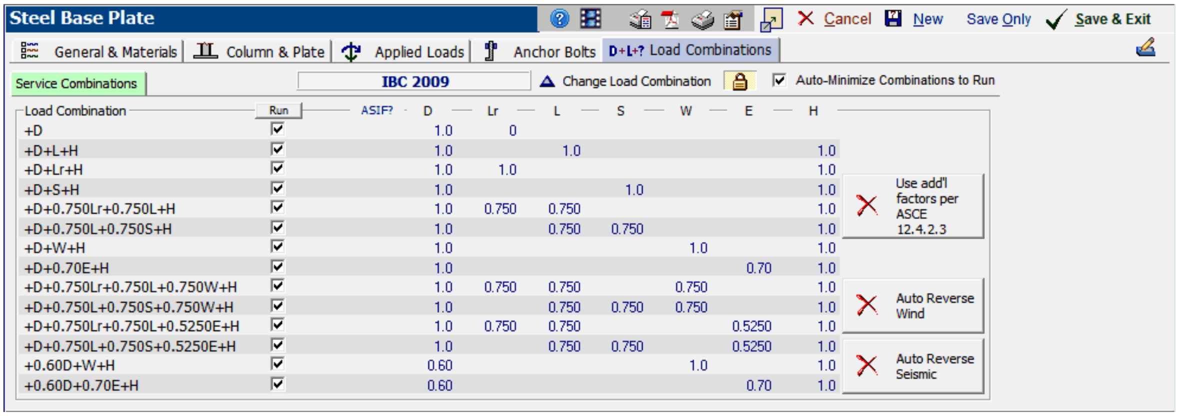
Row Spacing

Spacing of rows when more than one row is used.

Load Combinations Tab

This tab displays the load combinations used for either the ASD or LRFD selections.

The screen capture below shows a sample of the load combinations factors:



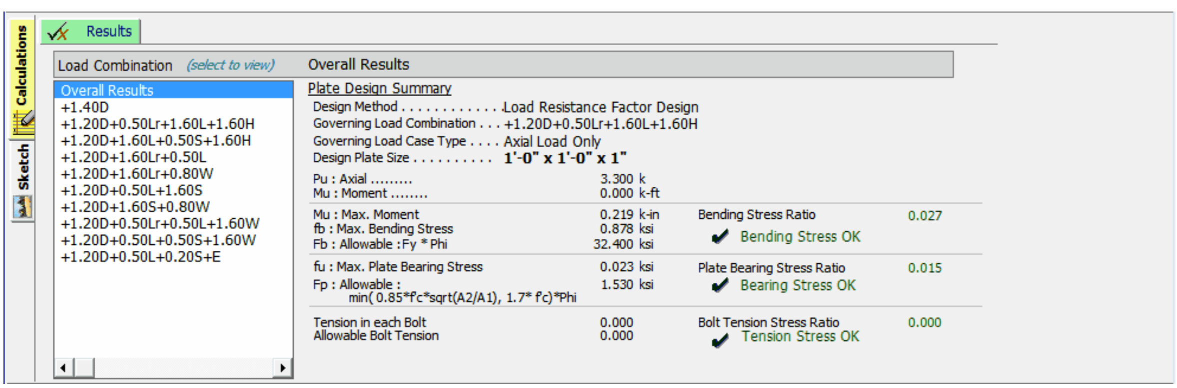
Load Combination	Run	ASIF?	D	Lr	L	S	W	E	H
+D	✓		1.0	0					
+D+L+H	✓		1.0		1.0				1.0
+D+Lr+H	✓		1.0	1.0					1.0
+D+S+H	✓		1.0			1.0			1.0
+D+0.750Lr+0.750L+H	✓		1.0	0.750	0.750				1.0
+D+0.750L+0.750S+H	✓		1.0		0.750	0.750			1.0
+D+W+H	✓		1.0				1.0		1.0
+D+0.70E+H	✓		1.0					0.70	1.0
+D+0.750Lr+0.750L+0.750W+H	✓		1.0	0.750	0.750		0.750		1.0
+D+0.750L+0.750S+0.750W+H	✓		1.0		0.750	0.750	0.750		1.0
+D+0.750Lr+0.750L+0.5250E+H	✓		1.0	0.750	0.750			0.5250	1.0
+D+0.750L+0.750S+0.5250E+H	✓		1.0		0.750	0.750		0.5250	1.0
+0.60D+W+H	✓		0.60				1.0		1.0
+0.60D+0.70E+H	✓		0.60					0.70	1.0

Results Tab - Overall

The **Results** tab contains a list of load combinations on the left and a summary of all the calculated values for each load combination on the right. Click one of the load combinations in the list to view results specifically for that load combination.

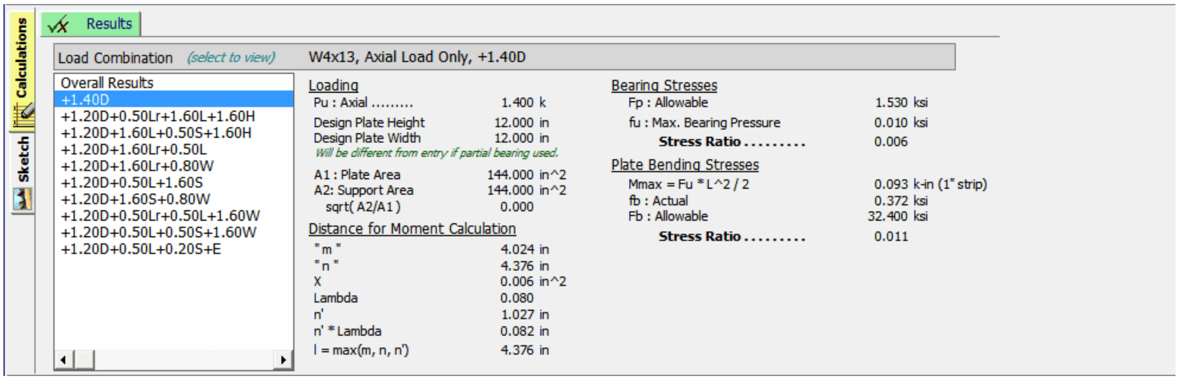
The very first item in the list will always say **Overall Results**. The module will examine the results for all of the load combinations and determine which gives the extreme condition of stress and presents it the **Overall Results** item. This is the governing case for the base plate.

Note that Maximum Bending Stress is calculated using the *plastic* section modulus, in keeping with AISC Design Guide 1, Second Edition.



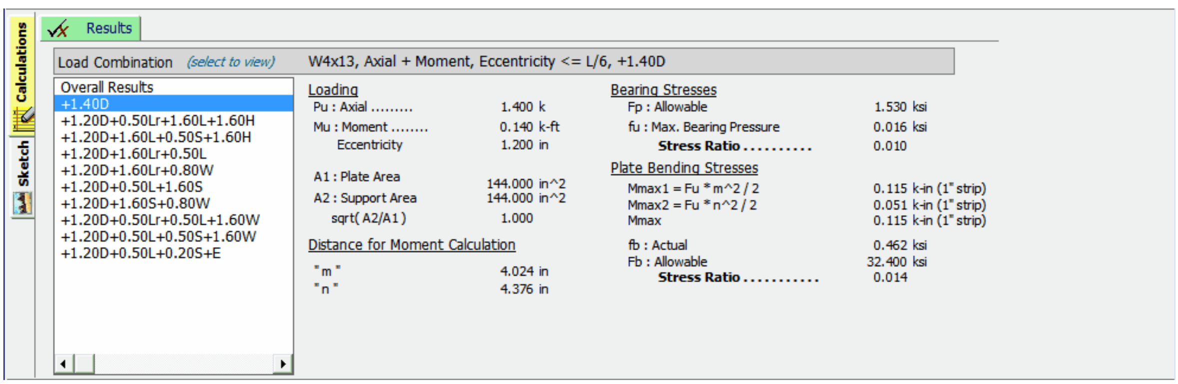
Results Tab - Axial Load Only

When a load combination results in only an axial load being applied, the summary of information below is displayed. See the AISC Design Guide #1 starting on page 4 for a description of the values present here.



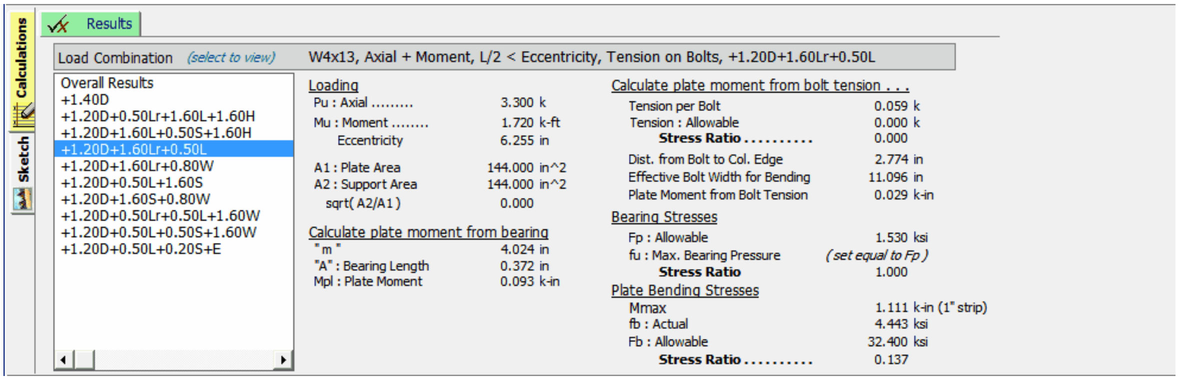
Results Tab - Small Eccentricity

When a load combination results in an axial load and very small moment being applied, the summary of information below is displayed. This condition is caused when the moment causes the resultant eccentricity of the axial load to be within the middle 1/3rd of the base plate. See the AISC Design Guide #1 starting on page 19 for a description of the values present here.

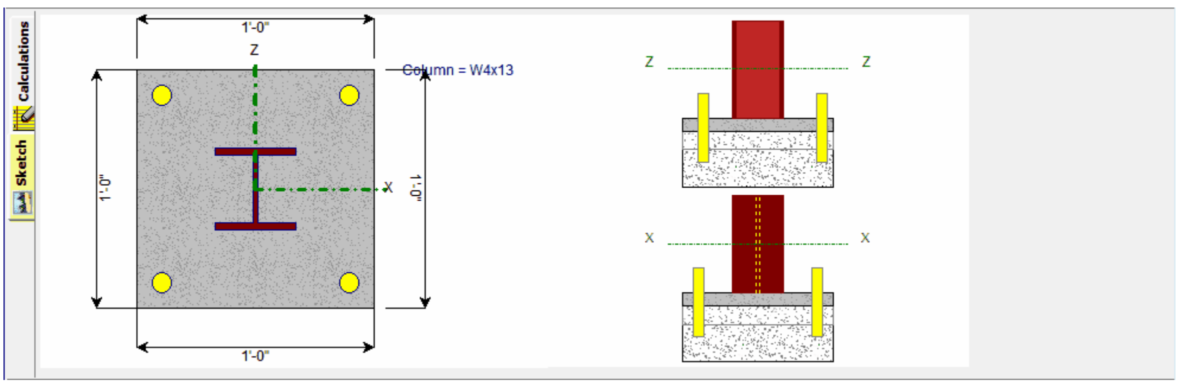


Results Tab - Large Eccentricity

When a load combination results in an axial load and large moment being applied, the summary of information below is seen. This condition is caused when the moment causes the resultant eccentricity of the axial load to be outside the middle 1/3rd of the base plate. See the AISC Design Guide #1 starting on page 21 for a description of the values present here.


Calculations
Sketch
Results
Load Combination (select to view)
W4x13, Axial + Moment, L/2 < Eccentricity, Tension on Bolts, +1.20D+1.60Lr+0.50L
Overall Results
+1.40D
+1.20D+0.50Lr+1.60L+1.60H
+1.20D+1.60L+0.50S+1.60H
+1.20D+1.60Lr+0.50L
+1.20D+1.60Lr+0.80W
+1.20D+0.50L+1.60S
+1.20D+1.60S+0.80W
+1.20D+0.50Lr+0.50L+1.60W
+1.20D+0.50L+0.50S+1.60W
+1.20D+0.50L+0.20S+E
Loading
Pu : Axial .... 3.300 k
Mu : Moment .... 1.720 k-ft
Eccentricity 6.255 in
A1 : Plate Area 144.000 in^2
A2 : Support Area 144.000 in^2
sqrt(A2/A1) 0.000
Calculate plate moment from bearing
"m" 4.024 in
"A" : Bearing Length 0.372 in
Mpl : Plate Moment 0.093 k-in
Calculate plate moment from bolt tension . . .
Tension per Bolt 0.059 k
Tension : Allowable 0.000 k
Stress Ratio 0.000
Dist. from Bolt to Col. Edge 2.774 in
Effective Bolt Width for Bending 11.096 in
Plate Moment from Bolt Tension 0.029 k-in
Bearing Stresses
Fp : Allowable 1.530 ksi
fu : Max. Bearing Pressure (set equal to Fp)
Stress Ratio 1.000
Plate Bending Stresses
Mmax 1.111 k-in (1" strip)
fb : Actual 4.443 ksi
Fb : Allowable 32.400 ksi
Stress Ratio 0.137

Sketch Tab


Calculations
Sketch
1'-0"
Z
Column = W4x13
1'-0"
X
1'-0"
1'-0"
Z
Z
X
X

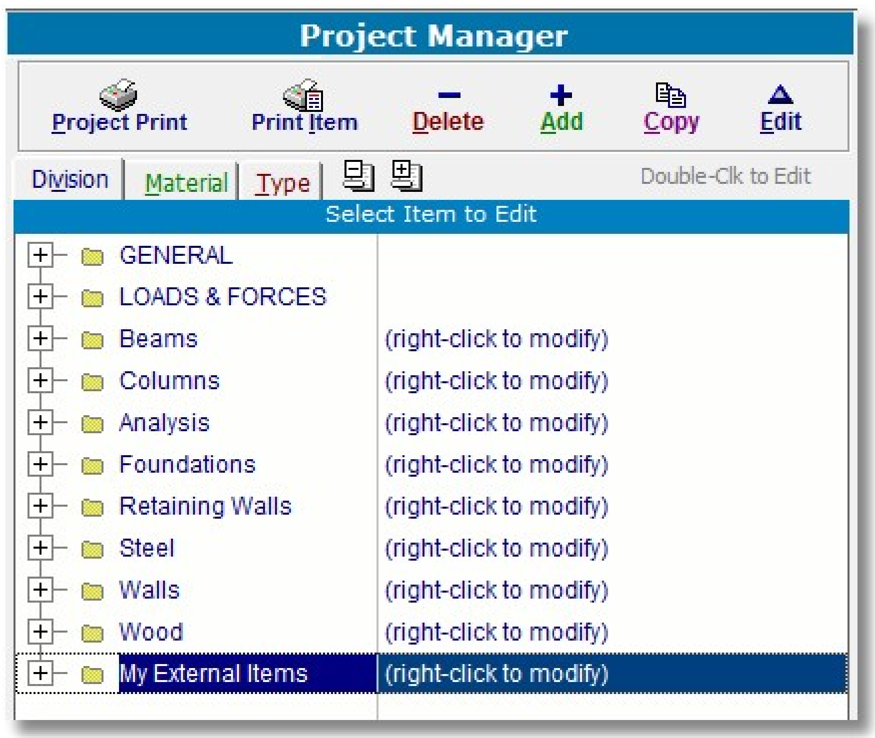
10.10 External Items

Structural Engineering Library allows you to add non-ENERCALC items into a Project File, such as:

- MS Excel Spreadsheets
- MS Word Documents
- Adobe Acrobat PDF Files
- Scanned Images

The ability to include these types of items allows you to use the ENERCALC Project File as a central point of document preparation and storage for structural project calculations.

These external items can be interspersed within the ENERCALC calculations. Here is an image of the Project Manager showing a Division that has been created specifically to contain the external items we are about to add:



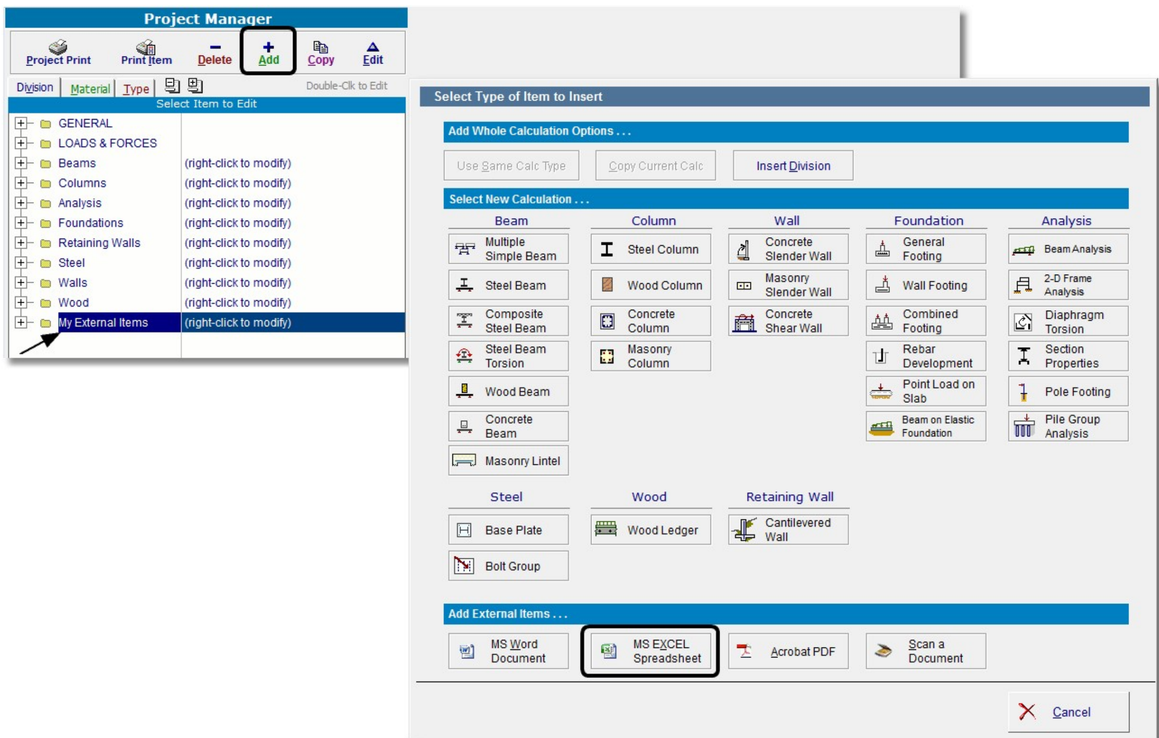
10.10.1 EXCEL Spreadsheet

[Need more? Ask Us a Question]

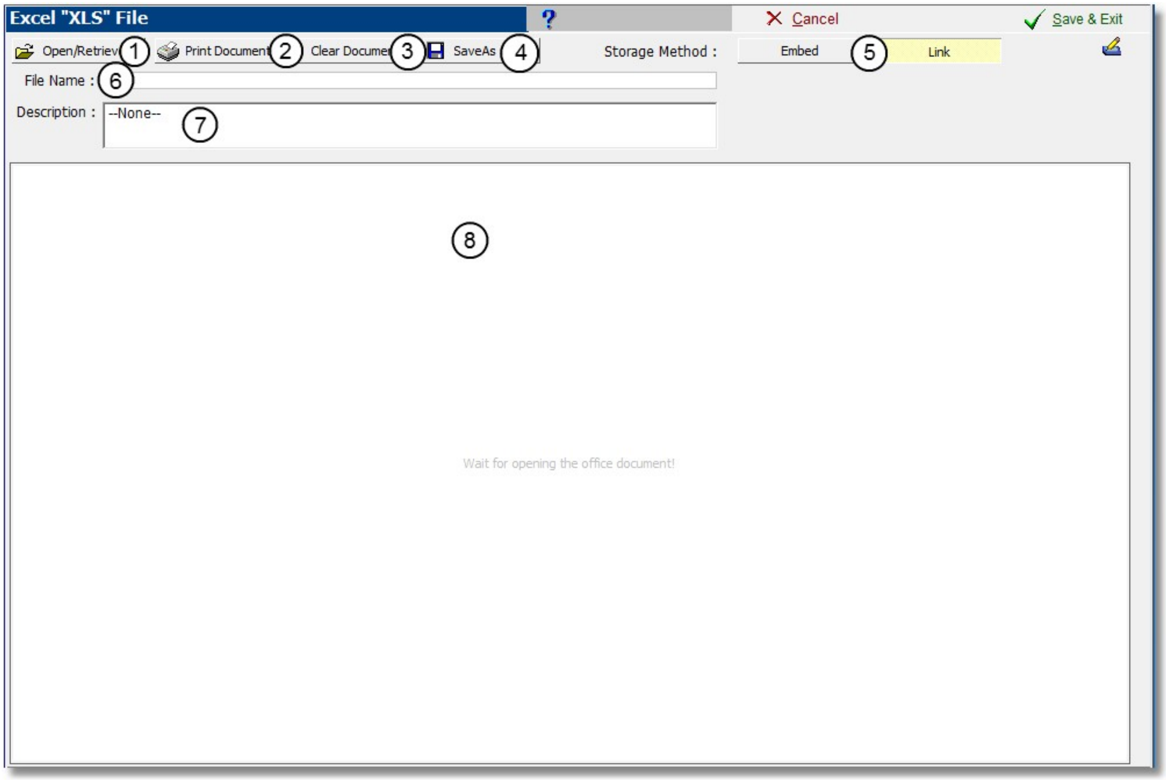
Click here for a video: Inserting an EXCEL Spreadsheet

Adding a Microsoft Excel spreadsheet to the project requires that you already have Excel loaded on your computer.

To add an Excel spreadsheet to your Project, click on the Division where you would like to insert the spreadsheet, click the [**Add**] button, and then click [**MS Excel Spreadsheet**]:



The screen will appear like this, ready for you to load the spreadsheet of your choice:



(1) Use the [**Open/Retrieve**] button to load the desired Excel spreadsheet.

(2) Use the [**Print Document**] button to print this item using the Excel printing functionality.

(3) Use the [**Clear Document**] button to remove the currently loaded Excel spreadsheet from the Project File.

(4) Use the [**SaveAs**] button to save the currently loaded Excel spreadsheet to a new filename.

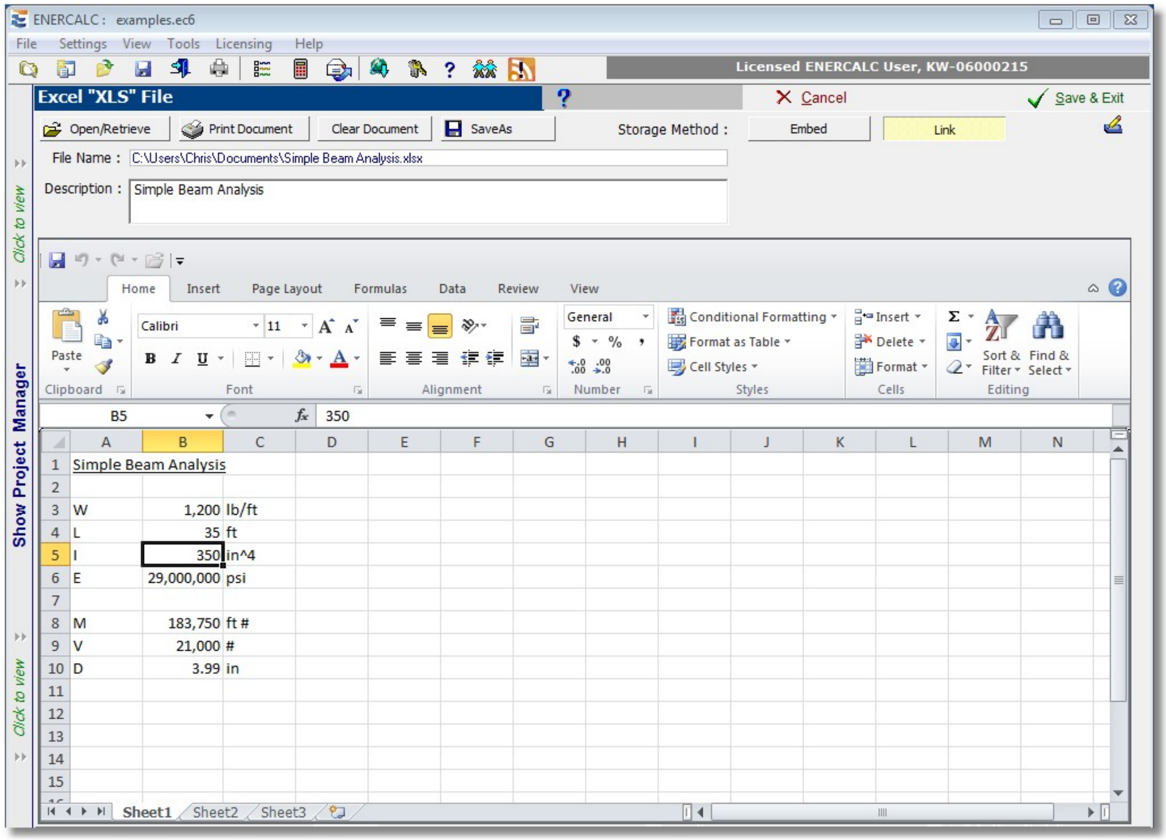
(5) The Storage Method buttons are used to specify how the Excel spreadsheet will be stored for use within ENERCALC.

- [**Embed**] saves a *copy* of the item in the Project File. (Easiest for portability, results in larger Project File size, changes to original item will **not** be visible.)
- [**Link**] saves a *link* to the item in Project File. (Changes to original item **will** be visible, keeps Project File size small, not easily portable.)

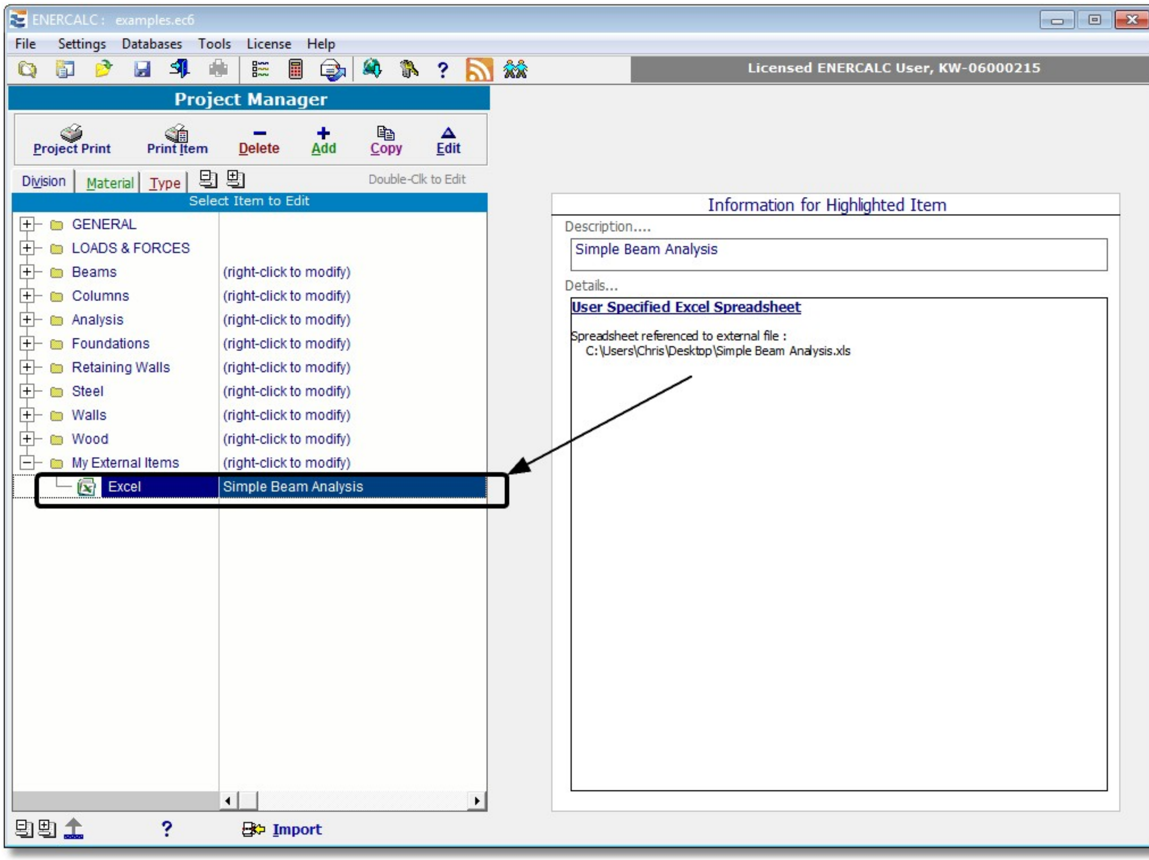
(6) This area is used to define the name of the Excel spreadsheet when it is stored externally from the Project File.

(7) Enter a description of this item as needed. It will be displayed in the Descriptions column of the Project Manager for reference.

(8) This is the area where the Excel spreadsheet is loaded, as shown below:



The last step is to click the [**Save & Exit**] button. The display will return to the Project Manager, and you will see an indication that the selected spreadsheet has been attached as shown in the image below:


ENERCALC : examples.ec6
File Settings Databases Tools License Help
Licensed ENERCALC User, KW-06000215
Project Manager
Project Print
Print Item
Delete
Add
Copy
Edit
Division
Material
Type
Double-Clk to Edit
Select Item to Edit
GENERAL
LOADS & FORCES
Beams
Columns
Analysis
Foundations
Retaining Walls
Steel
Walls
Wood
My External Items
(right-click to modify)
Excel
Simple Beam Analysis
Import
Information for Highlighted Item
Description....
Simple Beam Analysis
Details...
User Specified Excel Spreadsheet
Spreadsheet referenced to external file :
C:\Users\Chris\Desktop\Simple Beam Analysis.xls

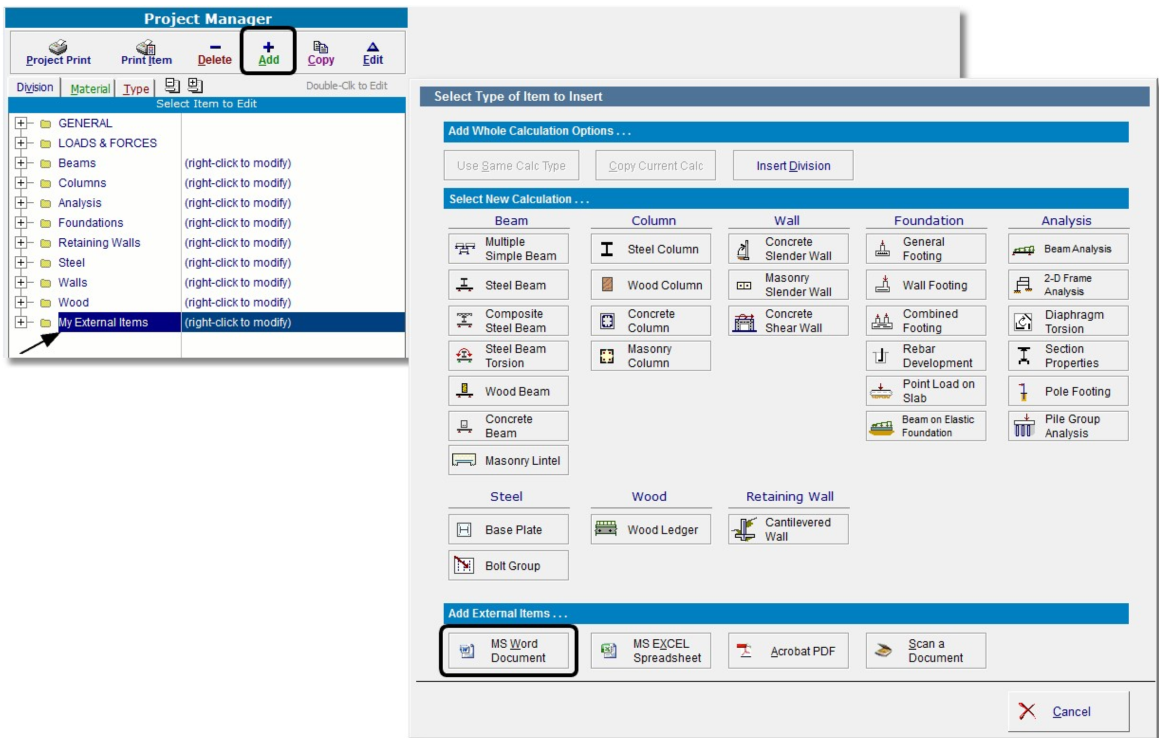
10.10.2 WORD Document

[Need more? Ask Us a Question]

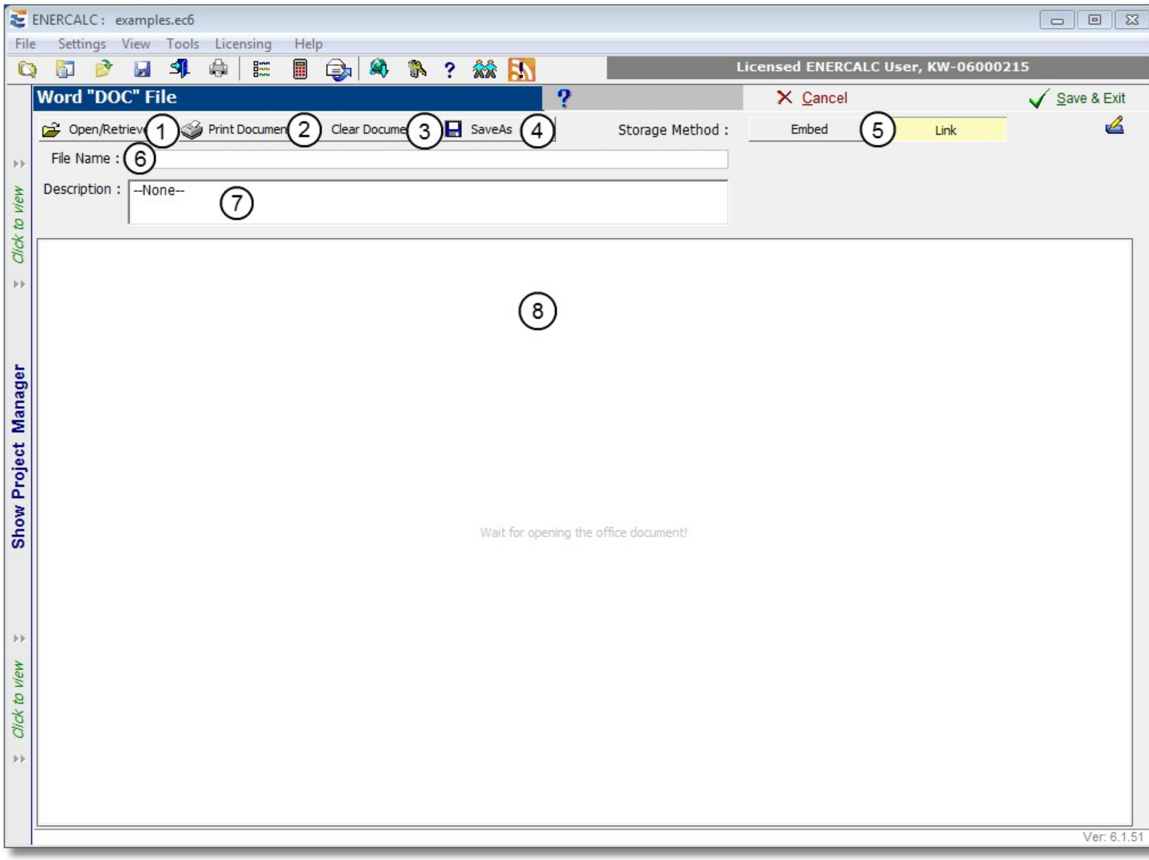
Click here for a video: Inserting a WORD Document

Adding a Microsoft Word document to the Project requires that you already have Word loaded on your computer.

To add a Word document to your Project, click on the Division where you would like to insert the document, click the [**Add**] button, and then click [**MS Word Document**]:



The screen will appear like this, ready for you to load the document of your choice:



(1) Use the [**Open/Retrieve**] button to load the desired Word document.

(2) Use the [**Print Document**] button to print this item using the Word printing functionality.

(3) Use the [**Clear Document**] button to delete the currently loaded Word document from the Project File.

(4) Use the [**SaveAs**] button to save the currently loaded Word document to a new filename.

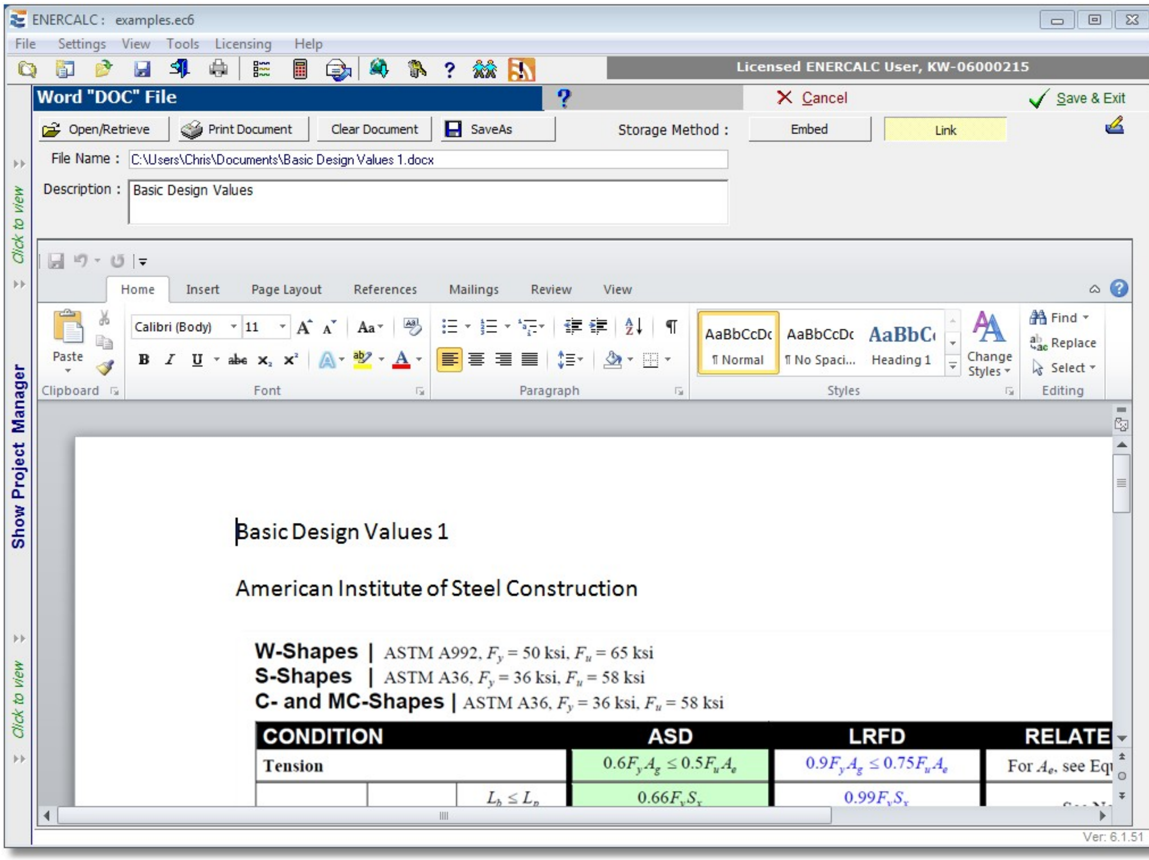
(5) The Storage Method buttons are used to specify how the Word document will be stored for use within ENERCALC.

- [**Embed**] saves a *copy* of the item in the Project File. (Easiest for portability, results in larger Project File size, changes to original item will **not** be visible.)
- [**Link**] saves a *link* to the item in Project File. (Changes to original item **will** be visible, keeps Project File size small, not easily portable.)

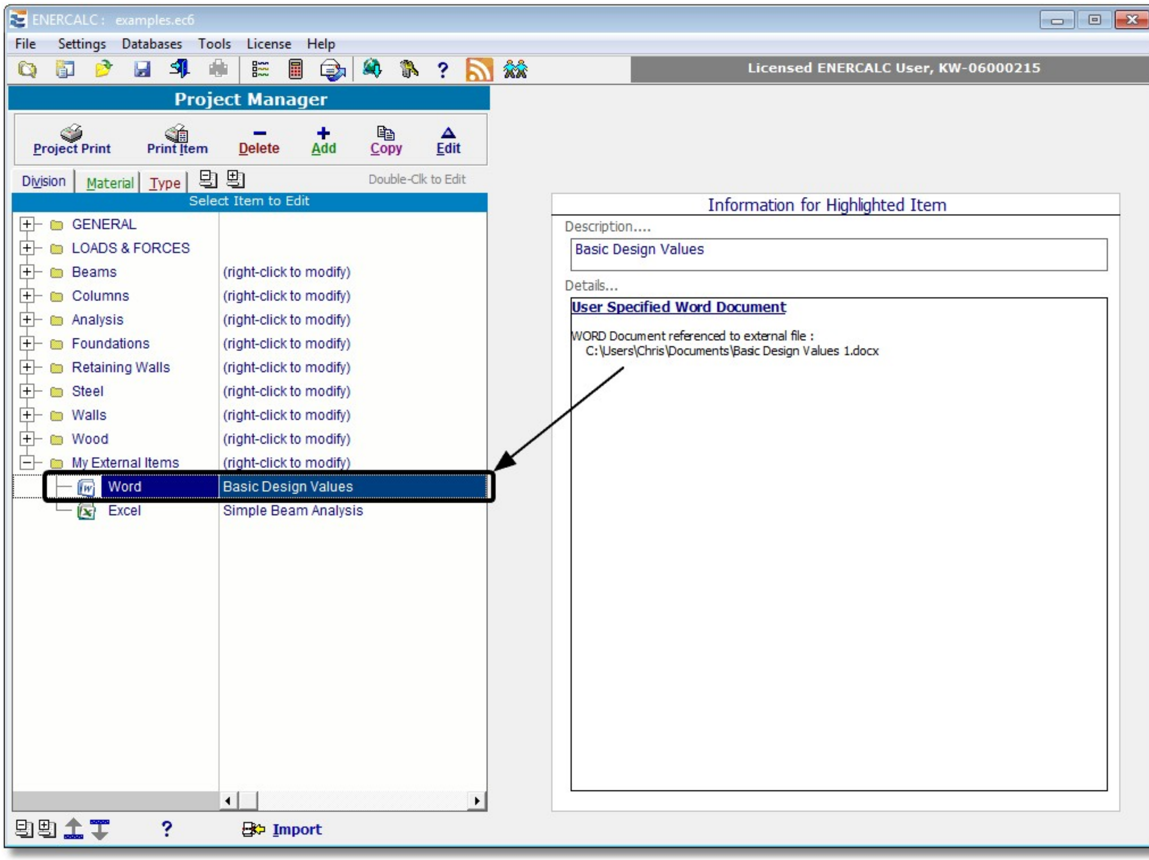
(6) This area is used to define the name of the Word document when it is stored externally from the Project File.

(7) Enter a description of this item as needed. It will be displayed in the Descriptions column of the Project Manager for reference.

(8) This is the area where the Word document is loaded, as shown below:



The last step is to click the [**Save & Exit**] button. The display will return to the Project Manager, and you will see an indication that the selected document has been attached as shown in the image below:


ENERCALC : examples.ec6
File Settings Databases Tools License Help
Licensed ENERCALC User, KW-06000215
Project Manager
Project Print
Print Item
Delete
Add
Copy
Edit
Division
Material
Type
Double-Clk to Edit
Select Item to Edit
GENERAL
LOADS & FORCES
Beams (right-click to modify)
Columns (right-click to modify)
Analysis (right-click to modify)
Foundations (right-click to modify)
Retaining Walls (right-click to modify)
Steel (right-click to modify)
Walls (right-click to modify)
Wood (right-click to modify)
My External Items (right-click to modify)
Word Basic Design Values
Excel Simple Beam Analysis
Import
Information for Highlighted Item
Description....
Basic Design Values
Details...
User Specified Word Document
WORD Document referenced to external file :
C:\Users\Chris\Documents\Basic Design Values 1.docx

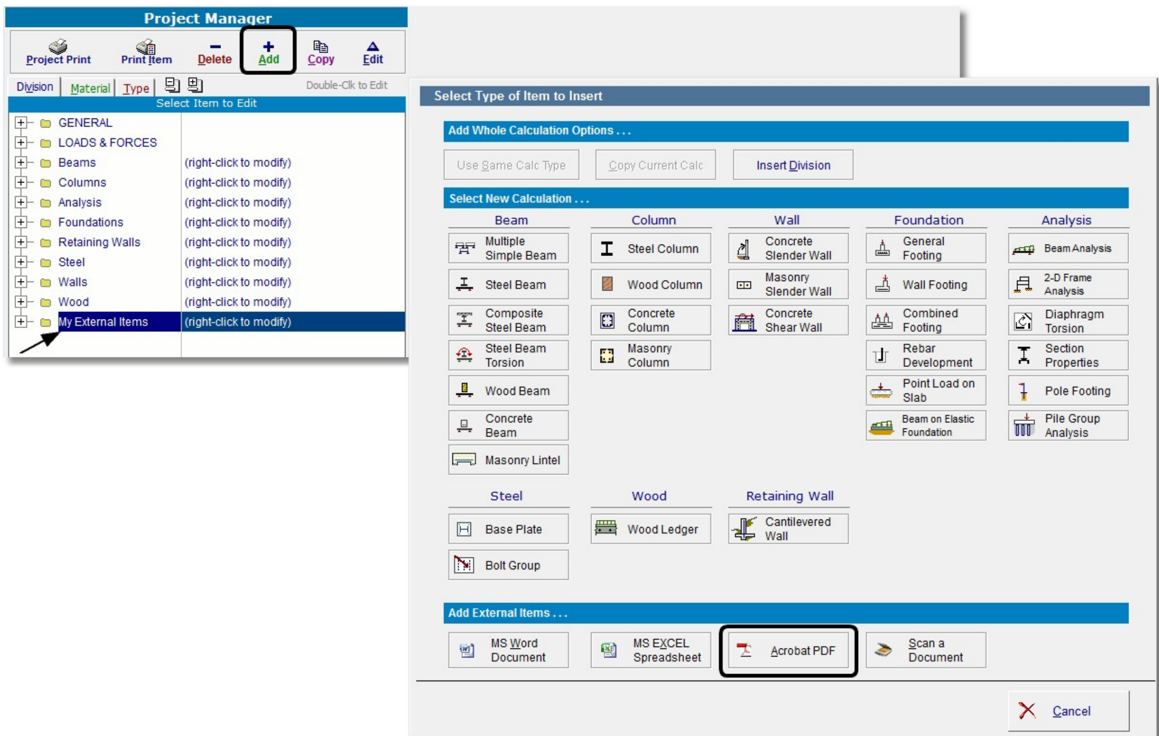
10.10.3 Adobe Acrobat PDF

Need more? Ask Us a Question

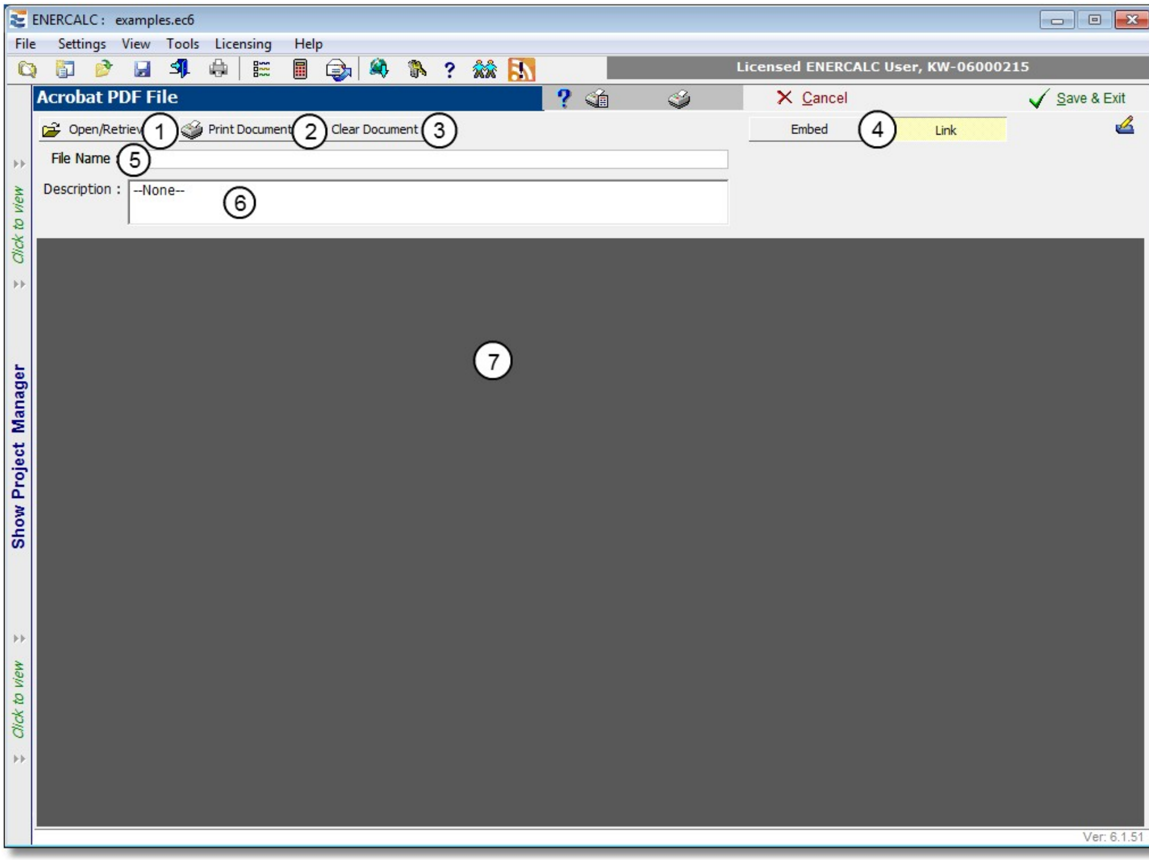
Click here for a video: Inserting an Adobe Acrobat PDF

Note: When using Adobe Reader X, it is necessary to disable Protected Mode if PDFs are to be used as External Items in Structural Engineering Library. To disable Protected Mode in Adobe Reader X, go to Edit > Preferences > General > Application Startup, and disable the option named “Enable Protected Mode at startup”.

To add an Adobe Acrobat PDF file to your Project, click on the Division where you would like to insert the document, click the [**Add**] button, and then click [**Acrobat PDF**]:



The screen will appear like this, ready for you to load the PDF file of your choice:



(1) Use the [**Open/Retrieve**] button to load the desired PDF file.

(2) Use the [**Print Document**] button to print this item using the PDF printing functionality.

(3) Use the [**Clear Document**] button to delete the currently loaded PDF file from the Project File.

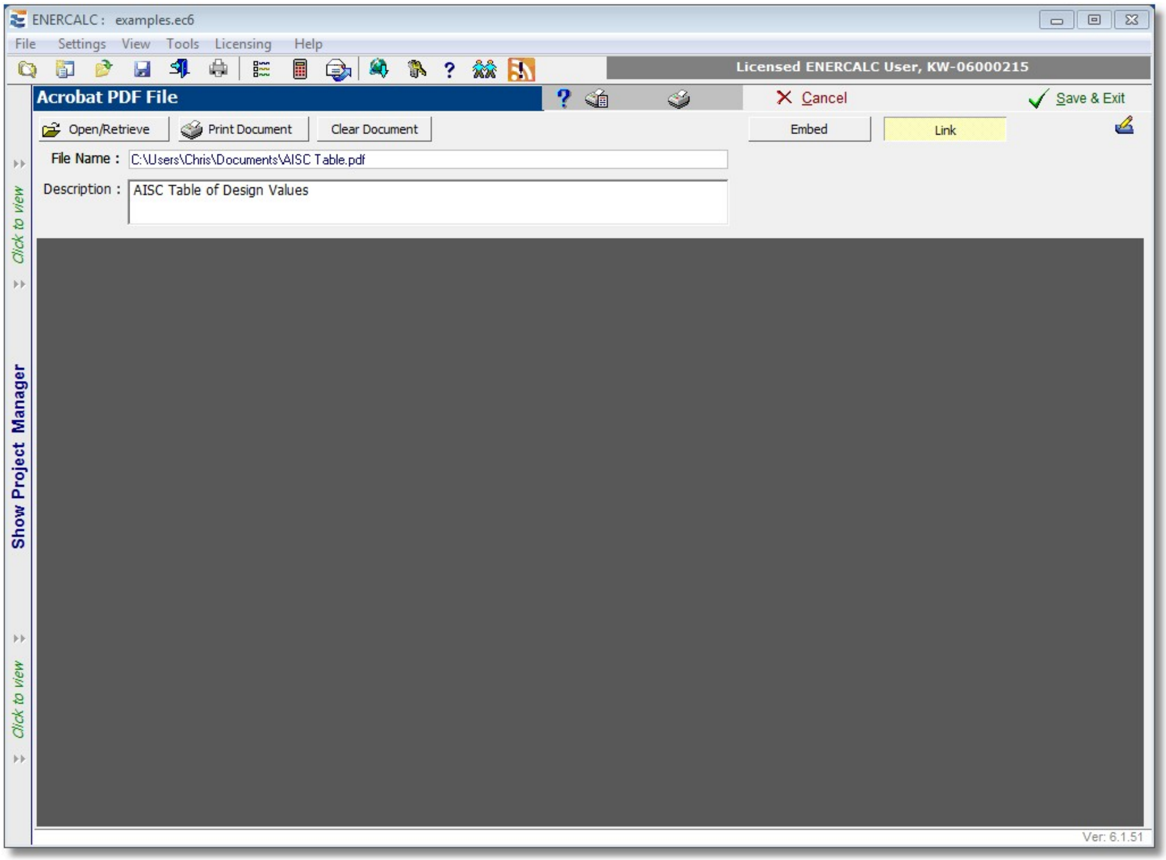
(4) The Storage Method buttons are used to specify how the PDF file will be stored for use within ENERCALC.

- [**Embed**] saves a copy of the item in the Project File. (Easiest for portability, results in larger Project File size, changes to original item will **not** be visible.)
- [**Link**] saves a link to the item in Project File. (Changes to original item **will** be visible, keeps Project File size small, not easily portable.)

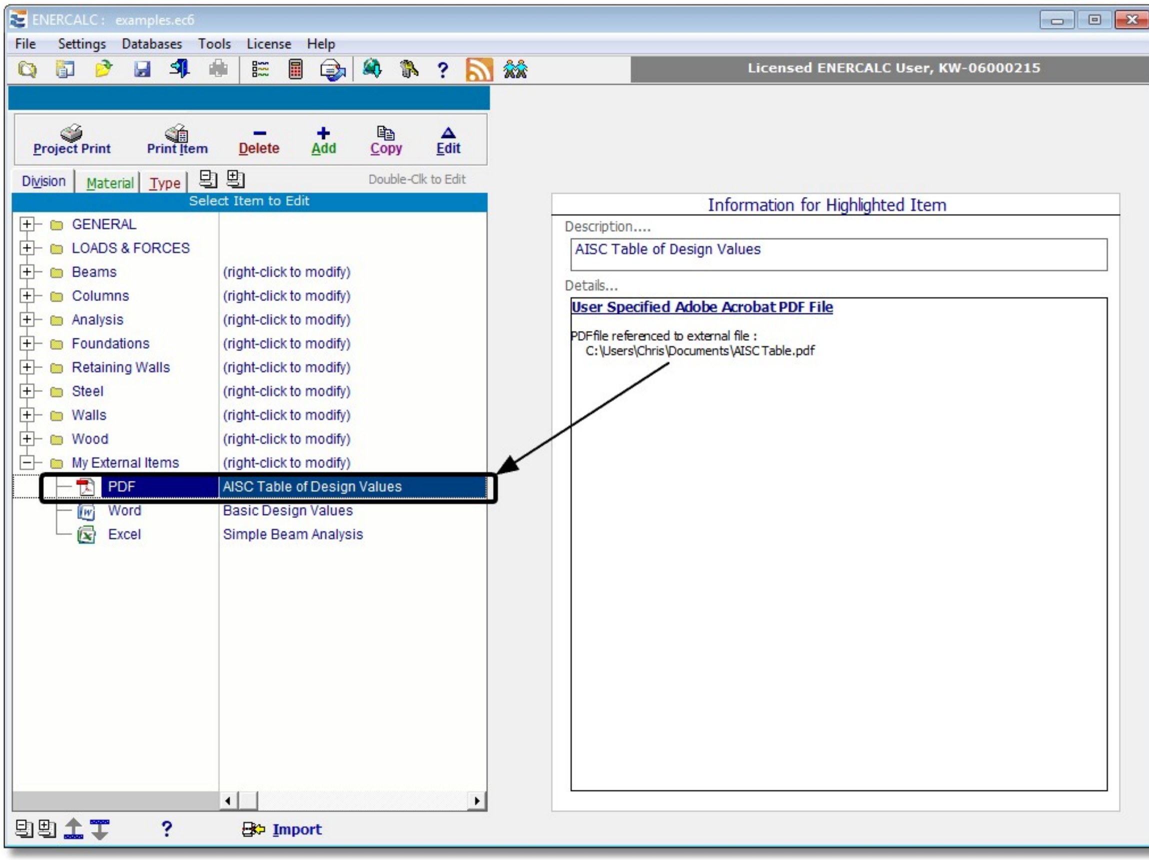
(5) This area is used to define the name of the PDF file when it is stored externally from the Project File.

(6) Enter a description of this item as needed. It will be displayed in the Descriptions column of the Project Manager for reference.

(7) This is the area where the PDF file is loaded, as shown below:



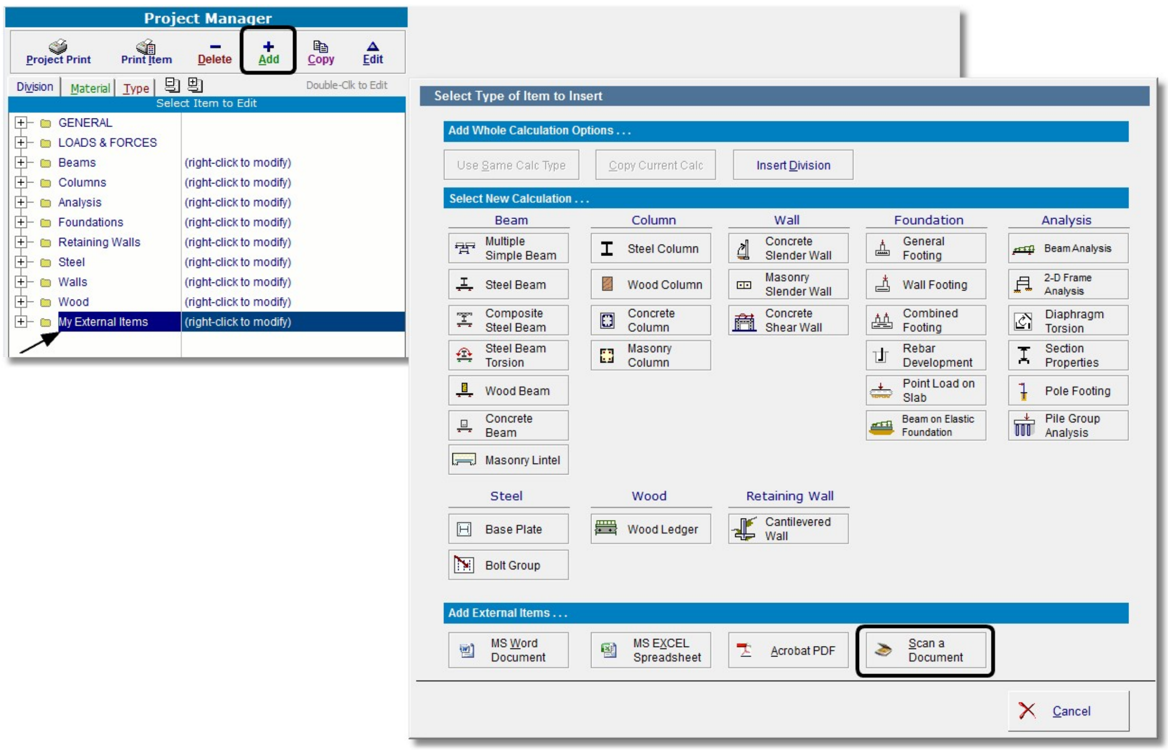
The last step is to click the [**Save & Exit**] button. The display will return to the Project Manager, and you will see an indication that the selected PDF file has been attached as shown in the image below:


ENERCALC : examples.ec6
File Settings Databases Tools License Help
Licensed ENERCALC User, KW-06000215
Project Print
Print Item
Delete
Add
Copy
Edit
Division
Material
Type
Double-Clk to Edit
Select Item to Edit
GENERAL
LOADS & FORCES
Beams
Columns
Analysis
Foundations
Retaining Walls
Steel
Walls
Wood
My External Items
(right-click to modify)
PDF
AISC Table of Design Values
Word
Basic Design Values
Excel
Simple Beam Analysis
Import
Information for Highlighted Item
Description....
AISC Table of Design Values
Details...
User Specified Adobe Acrobat PDF File
PDF file referenced to external file :
C:\Users\Chris\Documents\AISC Table.pdf

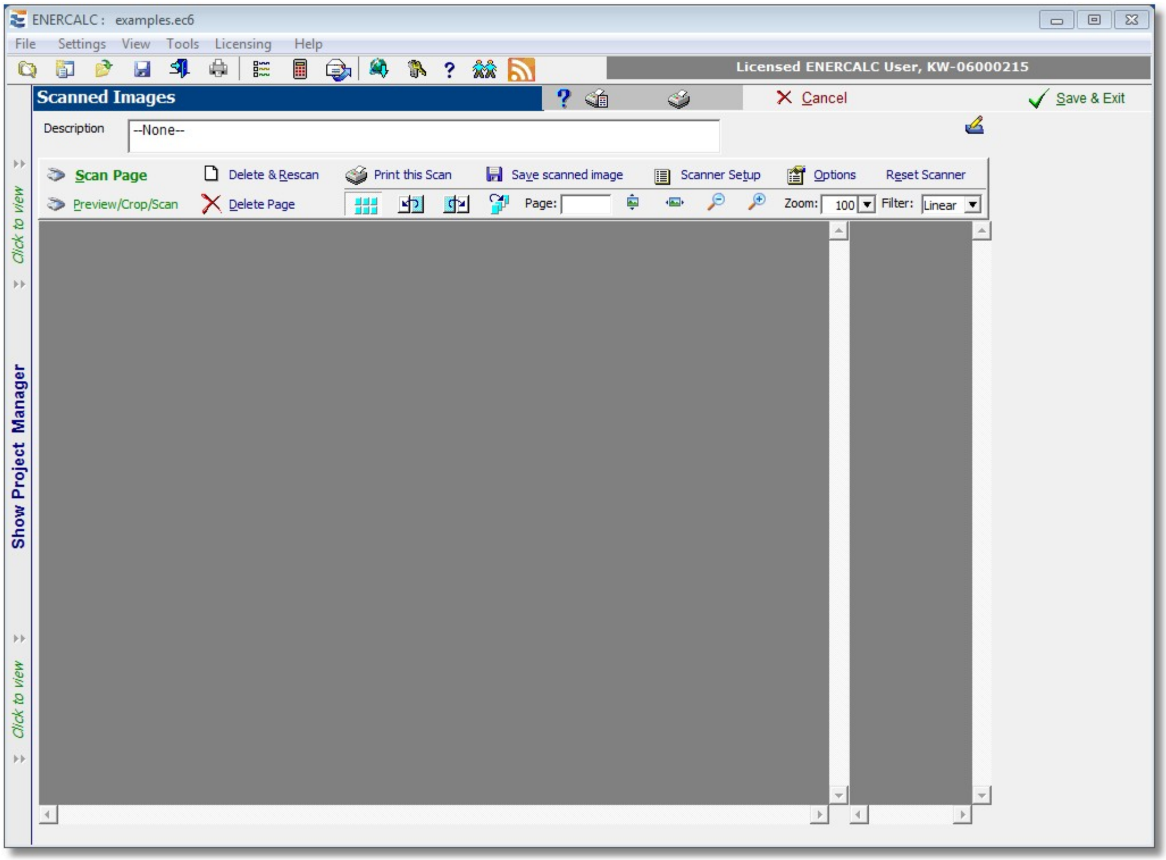
10.10.4 Scanned Document

Need more? Ask Us a Question

Structural Engineering Library now offers you the ability to use your scanner to create items and insert them into your Project File. To scan an item and add it to your Project File, click on the Division where you would like to insert the scanned item, click the [**Add**] button, and then click [**Scan a Document**]:



When the module is loaded you will see the screen below:



Here is a description of the various controls in this module:

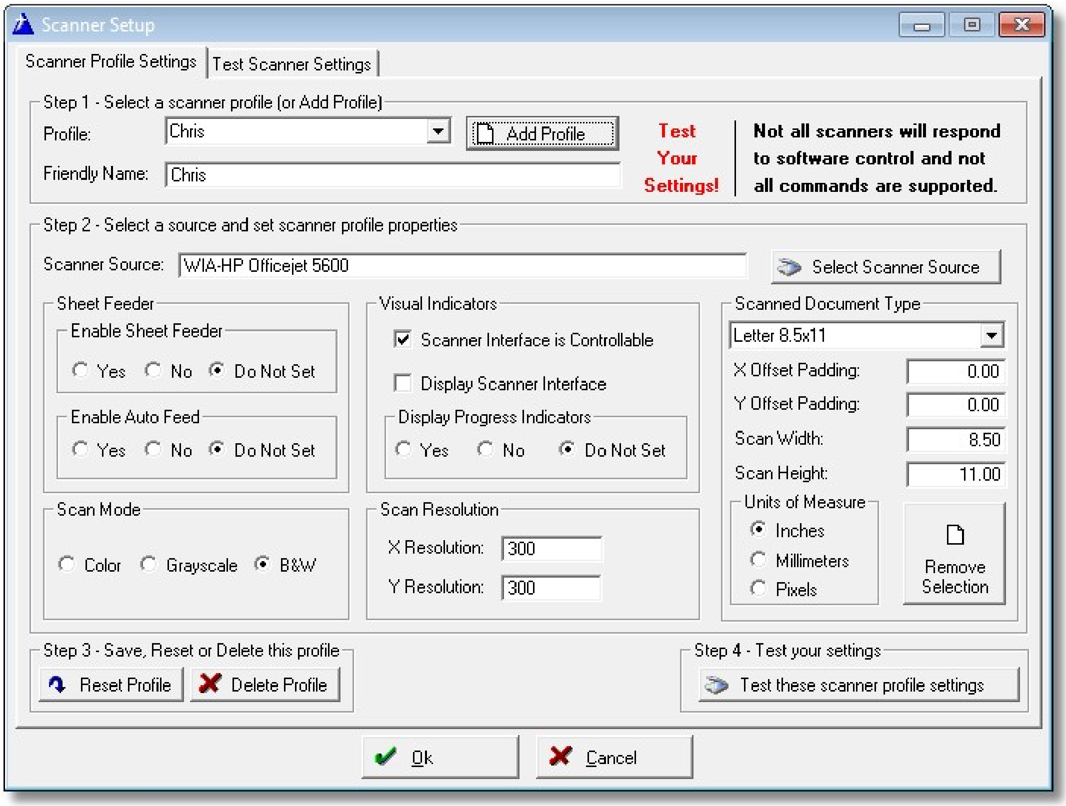
Scan Page: Using all the settings on the "Scanner Setup" page, perform a high resolution scan.

Delete & Rescan: Delete the currently visible scan and rescan the image at a different resolution.

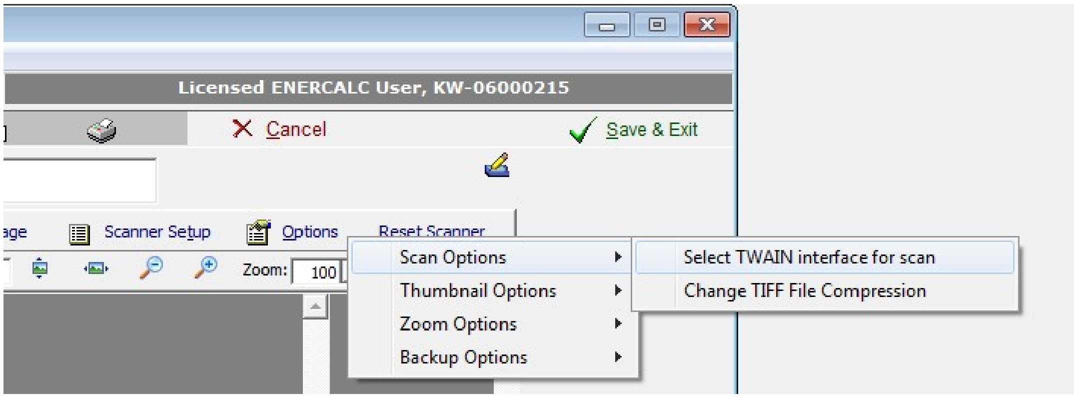
Print this Scan: Print the currently visible image to a printer.

Save scanned image: Save the current scan to a file.

Scanner Setup: Access the scanner setup screen (see image below)



Options:



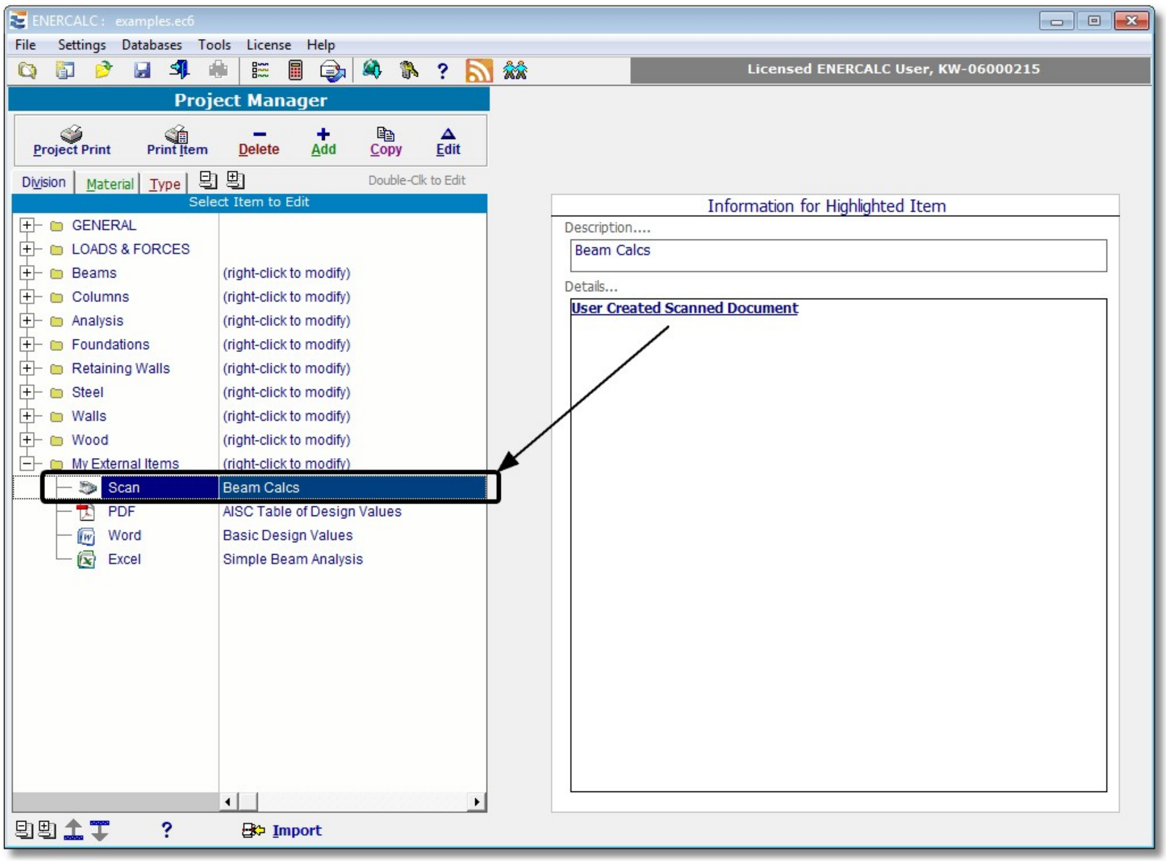
Reset Scanner: Send signals to the scanner to reset itself to its default state.

Preview/Crop/Scan: Perform a fast, low resolution "preview" scan and display it for your use in defining the final region that will be used for the high resolution scan.

Delete Page: Remove the current scan from this viewer.

Image Manipulation buttons: Allow you to change the way you view the scans in this document.

To create a scanned item, place the item on the scanner and click the [**Scan Page**] button. If desired, enter text in the Description field. Then, click the [**Save & Exit**] button. The display will return to the Project Manager, and you will see an indication that the scanned item has been attached as shown in the image below:




ENERCALC
Structural Engineering Library Version 6.0
ENERCALC's industry leading software solutions to perform structural design and analysis for low to mid rise buildings.
ENERCALC