RC-PIER®

AASHTO LFD and LRFD Analysis and Design of Reinforced Concrete Bridge Substructures and Foundations

LEAP Software, Inc.

P.O. Box 16827

Tampa, Florida 33687

(800) 451-5327

(813) 985-9170

www.leapsoft.com

Fax (813) 980-3642

This software program has been developed for use by professional engineers. It is the user's responsibility to ensure that the input for the software program is complete and correct, and that the results provided by the software program are interpreted correctly and conform to any design codes and government regulations that may apply.

Although this program has been written and tested by LEAP Software, Inc., no warranty, expressed or implied, is made as to the accuracy or functioning of the program or related program material. In no event will LEAP Software be liable for any damages for lost data or profits arising out of the use of or inability to use the licensed program.

The RC-PIER® *software is Copyright LEAP Software, Inc., 1988-2005. All rights reserved.*
RC-PIER® *is licensed to the original licensee for use only on the terms set forth in the License and Confidentiality Agreement.*

The RC-PIER® *user manual is Copyright LEAP Software, Inc., 1988-2005. All rights reserved. Unauthorized reproduction, display, modification, or distribution of this manual, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the full extent permitted by law.*

Trademarks

RC-PIER® *and the* RC-PIER® *logo are registered in the U.S. Patent and Trademark Office and are registered trademarks of LEAP Software, Inc., 1988-2005. All rights reserved. LEAP Bridge™, the LEAP Bridge logo, Integrated Bridge Software™, IBS™, and the IBS logo are trademarks of LEAP Software, Inc., 2005. LEAP® and LEAP Software® are registered trademarks of LEAP Software, Inc.*

Microsoft®, MS-DOS®, Windows®, and the Microsoft logo are either registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries.

Version: RC-PIER v4.1.0

Published by: LEAP Software, Inc.
Tampa, Florida
Updated January 2006

Editor: Robert Prancevic

Contributing Authors: Abdul Rauf, P.E.
Rob Ray
David A. Tomley, P.E.
Jia Wang, Ph.D.

Table of Contents

Chapter IN Introduction

Chapter GS Getting Started

Chapter GO General Operation

Chapter LB Using IBS Technology

Chapter T Tutorial Sessions

Chapter T1 Two-Column Pier, Auto Load Generation (AASHTO LFD)

Chapter T2 Multiple Column Pier (AASHTO LRFD)

Chapter T3 Hammerhead Pier, Strut and Tie Model Design (AASHTO LRFD)

Chapter TH Theory

Appendix A Nomenclature and Printout Explanation

Appendix B Hand Calculations for Selected Items

Appendix C References

Index

Introduction

Chapter

IN

About RC-PIER

RC-PIER® is an integrated tool for the AASHTO Standard and LRFD analysis and design of reinforced concrete bridge substructures and foundations. By incorporating both LFD and LRFD specifications in one interface, RC-PIER makes the transition to LRFD simple and efficient. RC-PIER allows users to design multi-column and hammerhead piers, straight, tapered or variable caps, and circular, rectangular (tapered and non-tapered) or drilled-shaft columns. Footing types include isolated or combined, supported on either soil or piles. There is no limit to the number of loads, bearings and piles that may be included in the design. Analysis results are presented in a variety of easy-to-view formats.

The following highlights some of RC-PIER's main features:

- Ability of dual design codes, AASHTO Standard and LRFD, to design reinforced concrete. No prestressing design is involved.
- Ability of dual units, U.S. and Metric. Allowed to switch between the two units at any time during program execution.
- Quickly specify project and job descriptions, and select working units.
- Ability to specify multi-column or hammerhead pier.
- Specify pier to be at any skew angle.
- Ability to select cap shape, either straight, tapered, or variable.
- Specify horizontal or inclined cap top surface.
- Ability to select circular, rectangular, rectangular chamfered, rectangular filleted, octagonal and hexagonal column shapes.
- Specify column shape in X-direction, Z-direction, or tapered in both X and Z directions.
- Specify up to two lines of bearings.
- Specify different materials for column, cap, and footing.
- Ability to specify up to two levels of intermediate struts.
- Specify checkpoints where you want to obtain results in addition to member nodes.
- Easy modeling of structures with the help of 3-D graphics.
- Specify loads in individual load cases.
- Self-load of substructure is considered automatically.
- Automatically generate superstructure dead loads, live loads with or without permit vehicles, wind loads, wind load on live load, longitudinal/braking load, centrifugal load, temperature load, earthquake load, and vessel collision (LRFD).
- Ability to save the live loads for live load generation in a library for repeated use.
- Specify load groups to be considered.
- Specify user-defined load groups in a library for repeated use.
- Carry out the three dimensional frame analysis of the bridge substructure.
- Ability to use either impact factor calculated by the program or manually input the impact factor.

- Select which cap, column, and footing design will be based on analysis results with impact factor.
- Ability to define other analysis/design parameters, such as strength reduction factor, cover, multi-presence factor, and crack control factor.
- Ability to check crack control in LRFD either based on Z factor (LRFD 3rd Edition and earlier) or based on Exposure (LRFD Interims of 2005).
- View individual load case results, load combination results, and envelopes for load groups/limits.
- View member forces in global or member local axis system.
- View analysis result plot diagrams.
- Ability to automatically design cap/strut. The program will automatically generate the required reinforcement and produce satisfying flexural bar placement.
- Ability to specify stirrups in cap and check for adequacy or do auto design.
- Analyze existing piers and footings, specify reinforcements, and then calculate the capacity and compare it to the requirement.
- Ability to check the cap/strut for shear, torsion, cracking, and fatigue.
- Ability to perform either P-delta or Moment Magnification analysis for slender columns.
- Automatically design the column for applied loads and generate the required reinforcement pattern.
- Define rectangular, circular, intersecting hoops or general rebar pattern.
- Define column reinforcement which is either vertical or parallel to column faces.
- Define reinforcement and determine section capacity for applied loads for a column.
- View interaction diagram(s) for the column.
- Specify and design isolated, combined, or strap footings.
- Specify and design spread or pile footings.
- Specify user defined pile patterns in a library for repeated use.
- Determine the soil pressure for service and factored loads.
- Either use program computed or your own input of soil pressures/pile reactions for design.
- Design the footing for flexure.
- Ability to check the footing for one-way or two-way shear.
- Option to check the footing for crack control and fatigue.
- Optional strut-and-tie method for analysis of hammerhead cap and isolated pile cap using the LRFD design code.
- Ability to import geometry data from other LEAP Software applications
- Ability to check overturning of pier about X and Z axis of pier for service or strength combinations.

Using the Manual

The RC-PIER Manual is intended to help you become familiar with the program and its capabilities. We recommend that you first read the General Operation chapter, and then complete the tutorials. After that, review the Theory chapter and the Hand Calculations for Selected Items contained in the Appendix.

The manual is divided into seven major chapters:

- Introduction
- Getting Started
- General Operation
- Tutorials
- Theory
- Appendices
- Index

Introduction. This chapter contains information that is helpful to know before attempting to use RC-PIER, such as program capabilities. It also includes information about using the manual. Reviewing this material prior to running the program will help ensure smooth operation.

Getting Started. This chapter contains information relating to program installation, authorization, and network setup. It also includes how to transfer authorization and how to contact technical support. You should read this chapter completely before installing and running the program.

General Operation. This chapter includes information and steps to using RC-PIER. It also includes many common facts and tips for using the program more effectively. If you would like specific information about any of RC-PIER’s features, you can find detailed operating information in the online help. You can activate help at any time while running the program by pressing the **F1** key.

Tutorials. The best way to become acquainted with the program is by actually using it to solve a few problems. In the Tutorials chapter, we walk you through a number of actual design projects. We encourage you to run the tutorials on your own computer and compare monitor displays with the illustrations in this manual. By closely following these tutorials, you can quickly become familiar with a number of typical design situations and most of the computer data input/output process.

It is advisable for you to check several previous designs using RC-PIER. This will serve to confirm your knowledge of the operation and assumptions of the program.

Theory. This chapter contains details on the assumptions and methods used in RC-PIER. Most in-depth technical questions can be answered by referring to this chapter.

Appendices. This chapter contains the nomenclature and detailed printout explanation, hand calculations for Tutorials 1-3, and also for seismic auto load generation. It also includes a list of cited references used in the manual.

Typographical Conventions

For your convenience, many of this manual's commonly used terms and references appear in special styles. The following are examples of these typographical conventions that will allow you to easily recognize key information.

Button References	**Add**
Menu References	**File | New**
Screen References	*Project* screen

Unless stated otherwise, references to "Art." refer to an article of the AASHTO Standard and LRFD code.

Units

The program works with either U.S. or metric units. Following is a list of units and the conversion between both systems.

English	**Metric**	**Conversion Factor***
ft	m	0.3048
in	mm	25.4
in^2/ft	mm^2/m	2116.6667
in^2	mm^2	645.16
in^4	$mm^4 \times 10^6$	0.41623
k	kN	4.44822
kft	kNm	1.35582
klf	kN/m	14.5939
ksi	Mpa	6.89476
psi	kPa	6.89476
pcf	kg/m^3	16.0185
plf	N/m	14.4939
psf	Pa	47.8803

* For example, 1ft = 0.3048m

Note: Although you can switch between U.S. and metric units at any time during program execution, some round-off errors may be introduced due to the fact that for converting between both systems of units, the program uses only the number of decimals that are shown on the screen.

Program Interface

The program uses the standard Windows interface conventions such as tabbed dialog boxes, menus, dialog boxes, buttons, scroll bars, etc. This interface is very intuitive; it eliminates the need to memorize keystrokes or commands.

Since the program uses a common interface for both design specifications, some of the screen captures shown in the manual may vary according to the design code selected.

Using Online Help

The program uses the standard Windows Help interface. To access help, select the **Contents** item from the **Help** menu. While running the program, you may access context-sensitive help information by pressing the **F1** key.

Chapter

GS

Getting Started

This chapter contains information about installing and authorizing the program, network setup and technical support. Before installing this program, it is recommended to read this chapter.

System Requirements

The minimum system requirements are as follows:

- Microsoft® Windows® NT, 2000, or XP
- 128 MB RAM
- Mouse or other pointing device
- 100 MB available uncompressed hard disk space
- CD-ROM drive
- Internet access (recommended)
- 800 × 600 dpi with 256 colors minimum (1024 × 768 dpi with thousands or millions of colors recommended)
- Microsoft® .NET Framework v1.1 (Refer *Installing and Configuring Microsoft .NET Framework* on page GS-6 for more information.)

LEAP Software Security

LEAP Software protects all programs with software security keys which prevent the use of unauthorized copies.

To run this program, an authorization code must be obtained from LEAP Software. This procedure is detailed in *Managing Product Site Keys* on page GS-7.

Types of Licenses

LEAP Software offers multiuser network capabilities. All RC-PIER® licenses are network licenses, allowing either a single user or multiple users the flexibility to access the program simultaneously. A predetermined number of copies of the program may run concurrently on client machines from a single network server hosting the source copy. The program may be run from a local hard drive after obtaining the licensing information from the server.

With a multiuser license, anyone who has access to the location where the program is installed can use the program on any networked computer; however, the number of people actually running the program at any given time may not exceed the maximum number of users specified by the license.

Network Setup

RC-PIER supports the following configurations:

Host/Server	**Client Computers**
Windows® NT/2000/XP	Windows® NT, 2000, and XP

When installing the program for network use, select one computer to be the host. The host will be the computer from which the program will run on network clients. The program must be installed on the host.

After installation of the program, the program must be authorized for network use. (Refer to *Managing Product Site Keys* on page GS-7.) When requesting an authorization code, make sure to notify LEAP Software of the number of licensed users being requested.

LEAP Software does not make any guarantees regarding program installation on a Novell Netware server. (Occasionally, network license users have had difficulties with compatibility in Novell Netware environments.)

Authorization Warnings

It is possible to permanently lose an authorization when changing configurations without taking the necessary precautionary steps. A few actions which may adversely affect an authorization are listed below. If any of these actions need to be taken, contact LEAP Software for assistance.

- Formatting the hard drive.
- Replacing the hard drive.
- Defragmenting the hard drive, without excluding the program directory from the defragmentation path.
- Installing a new operating system.
- Upgrading the operating system.
- Changing the system time by more than 75 minutes.
- Compressing the directory where the program is located.
- Moving the program to another directory or computer without properly transferring authorization.
- Installing the program on mirrored drives.

Note: The restoration of a license lost resulting from any of the above actions, without first consulting LEAP Software, may result in additional reauthorization fees.

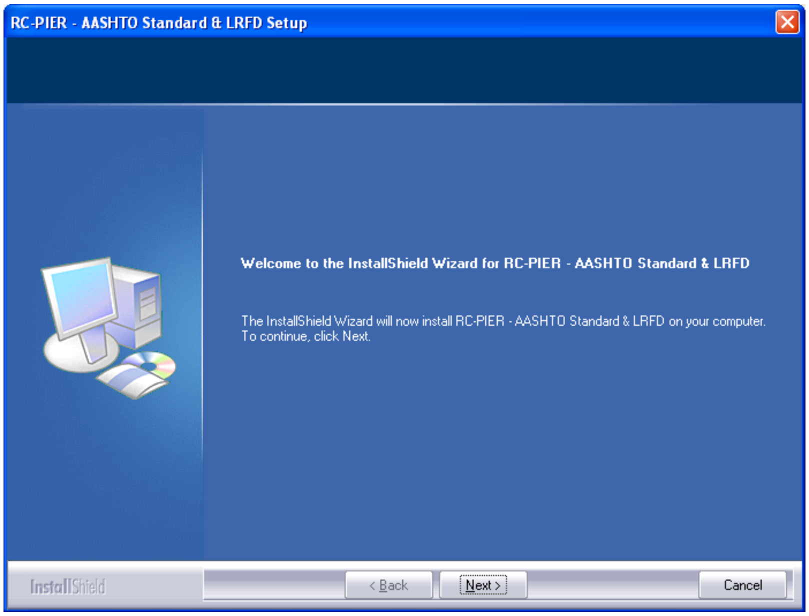
Installing the Program

Before installing the application it is recommend to determine the type of installation. Stand-alone and server installations require no options to be selected. However, a client installation requires that the application on the client be provided a path to the security file on the server, as discussed in step 5 on page GS-3.

Note: This program requires Microsoft .NET Framework v1.1. For more information, please refer to *Installing and Configuring Microsoft .NET Framework* on page GS-6.

Note: To install the program, login as an Administrator on the machine you want to do the installation. Remote installation is not allowed.

1. Download the program from the LEAP Software web site (*www.leapsoft.com*) and run the executable and follow the on-screen instructions to install the program.
2. When the *Welcome* screen displays, click **Next** and follow the on-screen instructions to complete the installation.



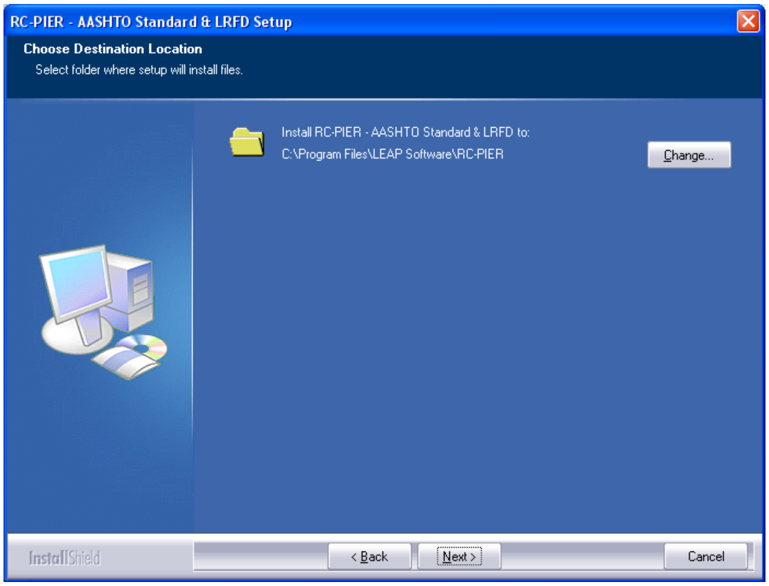
RC-PIER - AASHTO Standard & LRFD Setup

Welcome to the InstallShield Wizard for RC-PIER - AASHTO Standard & LRFD

The InstallShield Wizard will now install RC-PIER - AASHTO Standard & LRFD on your computer. To continue, click Next.

Figure GS-1 Welcome Screen

3. The *Choose Destination Location* screen displays. Browse to the location where the application will be installed and click **Next**.



RC-PIER - AASHTO Standard & LRFD Setup

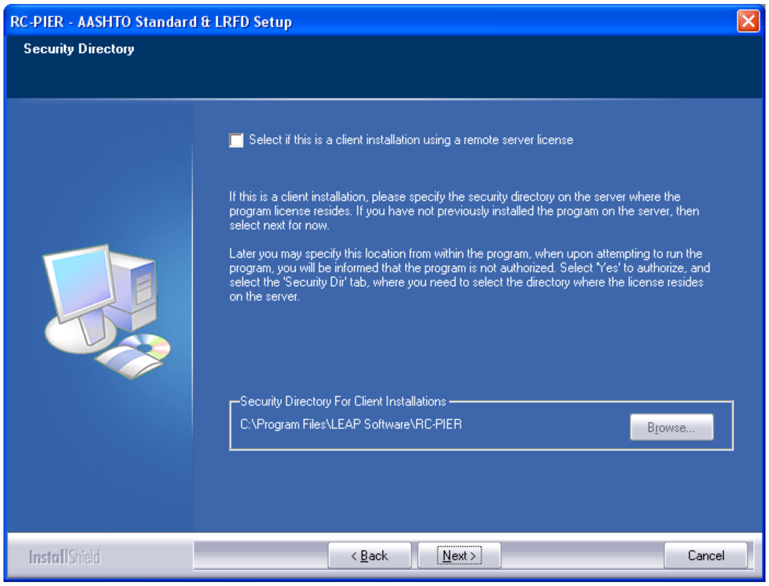
Choose Destination Location

Select folder where setup will install files.

Install RC-PIER - AASHTO Standard & LRFD to:
C:\Program Files\LEAP Software\RC-PIER

Figure GS-2 Choose Destination Location

4. The *Security Directory* screen displays. This screen is used to select the location of security files for a client-server configuration.
   - If the installation is in a stand-alone configuration or on a Server, do not select the check box and click **Next**.
   - If the installation is on a client, select the check box and use the **Browse** button to select the license directory location on the Server. Click **Next**.



RC-PIER - AASHTO Standard & LRFD Setup

Security Directory

☐ Select if this is a client installation using a remote server license

If this is a client installation, please specify the security directory on the server where the program license resides. If you have not previously installed the program on the server, then select next for now.

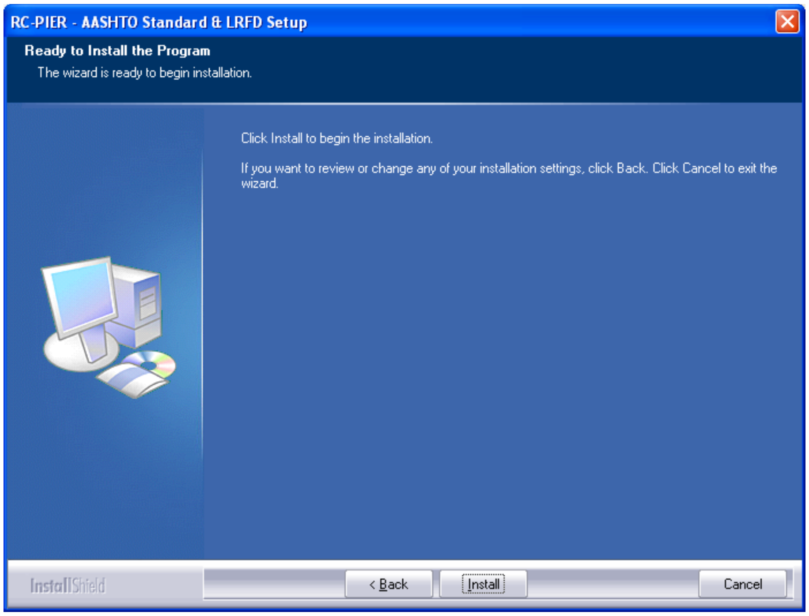
Later you may specify this location from within the program, when upon attempting to run the program, you will be informed that the program is not authorized. Select 'Yes' to authorize, and select the 'Security Dir' tab, where you need to select the directory where the license resides on the server.

Security Directory For Client Installations

C:\Program Files\LEAP Software\RC-PIER

Figure GS-3 Security Directory

5. The *Ready to Install the Program* window displays. Click **Install** to start the installation.



RC-PIER - AASHTO Standard & LRFD Setup

Ready to Install the Program

The wizard is ready to begin installation.

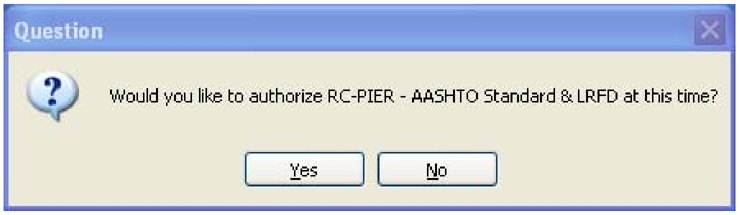
Click Install to begin the installation.

If you want to review or change any of your installation settings, click Back. Click Cancel to exit the wizard.

Figure GS-4 Read to Install the Program

6. Upon successful installation the *Authorize now?* dialog is displayed, as shown in Figure GS-5 on page GS-5.
   - If you select **Yes**, refer to *Authorizing a Program* on page GS-7.
   - If you select **No**, the program will prompt for authorization when started.

Note: The program must be authorized before it can be used.

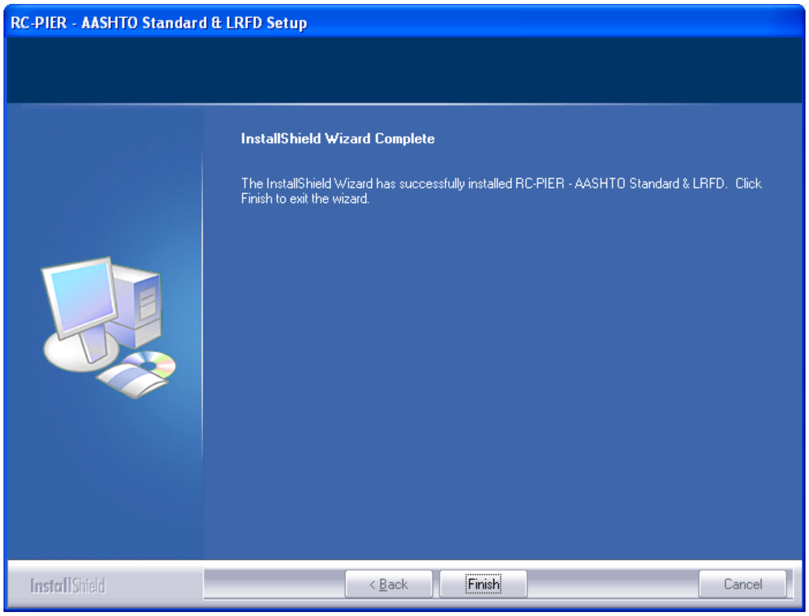


Question

Would you like to authorize RC-PIER - AASHTO Standard & LRFD at this time?

Figure GS-5 Authorize Now? Dialog

7. When the last screen is displayed click **Finish** to exit the install wizard.



RC-PIER - AASHTO Standard & LRFD Setup

InstallShield Wizard Complete

The InstallShield Wizard has successfully installed RC-PIER - AASHTO Standard & LRFD. Click Finish to exit the wizard.

Figure GS-6 Installation Complete

Note: Give all users of this program full access to the directory where the program installs and authorizes.

Novell Installation

LEAP Software does not make any guarantees about installations in Novell Netware environments. (Occasionally, network license users have experienced difficulties with compatibility of workstations using a Windows operating system in Novell Netware environments.)

The installation process is the same as described above for a Novell installation. However, one file needs to be loaded as described in the note below.

Note: A driver called CKSERVER.NLM is copied to the program directory. On the Novell Server, load the driver and reference it in the AUTOEXEC.NCF file for automatic startup.

Installing and Configuring Microsoft .NET Framework

To determine if this framework is installed select: **Start > Settings > Control Panel > Add/Remove Programs**. Scroll through the list of programs to see if the Microsoft .NET Framework v1.1 is listed. If it is, proceed with installation and authorization of the program. If it is not listed, please go to **www.leapsoft.com/dotnet.html** and follow the link to download and install the latest .NET *Framework* files.

Once the Microsoft .NET installation is complete, the next step is configuration. Refer to following table for instructions on how to access the .NET Framework v1.1 Wizards.

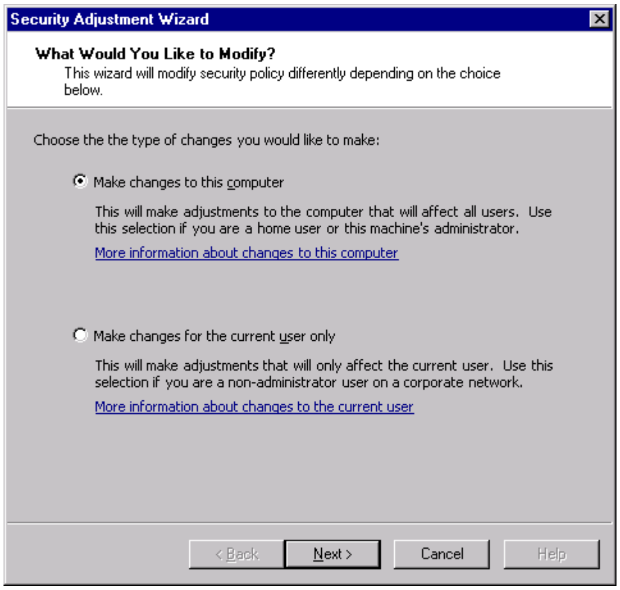
Operating System	Access Configuration Wizard
Windows 98	Search for ConfigWizards.exe and double-click to start (typically found in C:\Windows\Microsoft.NET\ Framework\v1.1 folder).
Windows NT	Start > Programs > Administrative tools
Windows 2000	Start > Settings > Control Panel > Administrative tools
Windows XP	Start > Settings > Control Panel > Administrative tools

Once the .NET Framework v1.1 Wizard is open, perform the following steps to configure the framework.

The wizards below can help you make simple changes to applications. Click on the icon to launch the wizard.

Figure GS-7 .NET Framework v1.1 Wizard

1. Select the **Adjust .NET Security** icon to open the *Security Adjustment Wizard*.



What Would You Like to Modify?

This wizard will modify security policy differently depending on the choice below.

Choose the the type of changes you would like to make:

● Make changes to this computer

This will make adjustments to the computer that will affect all users. Use this selection if you are a home user or this machine's administrator.

More information about changes to this computer

○ Make changes for the current user only

This will make adjustments that will only affect the current user. Use this selection if you are a non-administrator user on a corporate network.

More information about changes to the current user

Figure GS-8 Security Adjustment Wizard

2. Select the **Make changes to this computer** option and click **Next**.
3. Select the **Local Intranet** icon, change the level to **Full Trust**, and click **Next**.
4. Click **Finish** to accept the changes and close the *.NET Framework Wizard*.

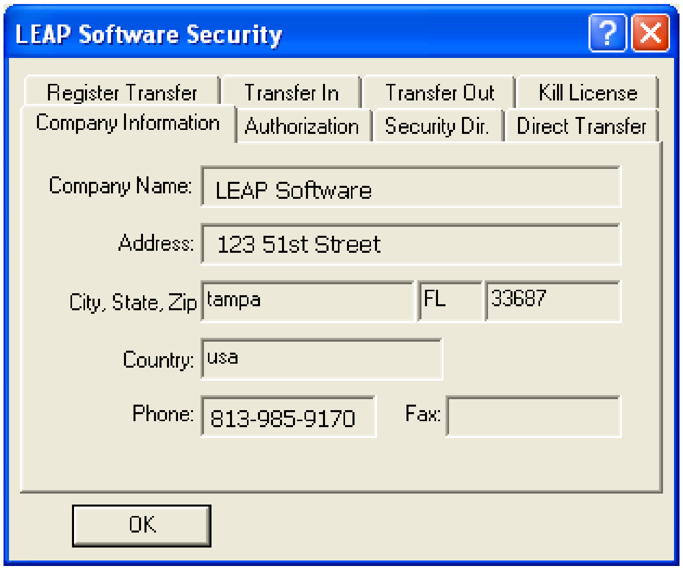
Managing Product Site Keys

The LEAP Software Online Authorization System is the recommended method for obtaining an authorization code.

Note: Only the Primary Key Holder may login on the Online Authorization System.

Authorizing a Program

1. From the **Start** menu select the program from the LEAP Software folder.
2. When the Authorization warning dialog opens, click **Yes**.
3. When the *LEAP Software Security* screen displays, complete the information in the *Company Information* tab.

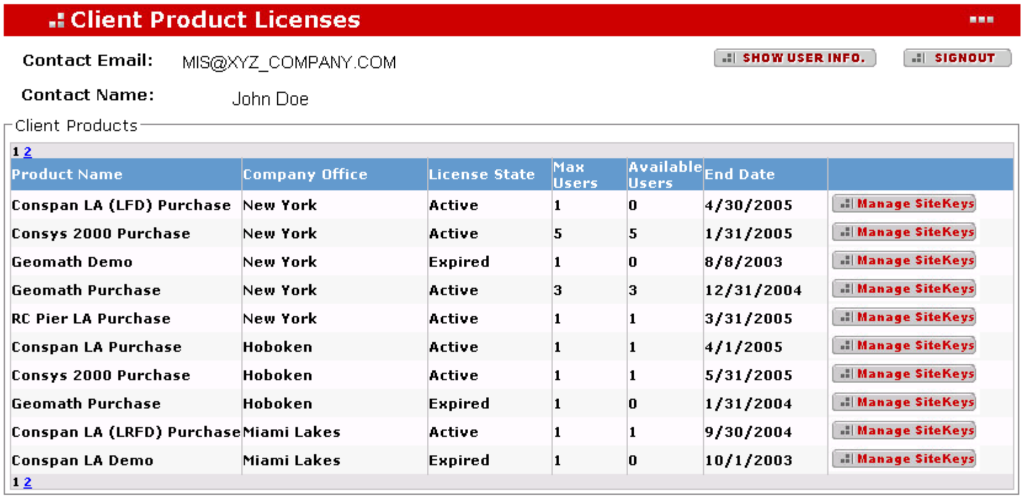


Company Name: LEAP Software
Address: 123 51st Street
City, State, Zip: tampa FL 33687
Country: usa
Phone: 813-985-9170
Fax:

Figure GS-9 Security - Company Information Tab

4. Go to the LEAP Software web site (*www.leapsoft.com*) and login.
5. Select **Online Authorization.**
6. When prompted, login with the appropriate email address and password.

7. At the Client Product Licenses window, click **Manage Site Keys** for the product for which the Site Key is required.



Client Product Licenses

Contact Email: MIS@XYZ_COMPANY.COM

Contact Name: John Doe

Client Products

1 2

Product Name	Company Office	License State	Max Users	Available Users	End Date	
Conspan LA (LFD) Purchase	New York	Active	1	0	4/30/2005	Manage SiteKeys
Consys 2000 Purchase	New York	Active	5	5	1/31/2005	Manage SiteKeys
Geomath Demo	New York	Expired	1	0	8/8/2003	Manage SiteKeys
Geomath Purchase	New York	Active	3	3	12/31/2004	Manage SiteKeys
RC Pier LA Purchase	New York	Active	1	1	3/31/2005	Manage SiteKeys
Conspan LA Purchase	Hoboken	Active	1	1	4/1/2005	Manage SiteKeys
Consys 2000 Purchase	Hoboken	Active	1	1	5/31/2005	Manage SiteKeys
Geomath Purchase	Hoboken	Expired	1	0	1/31/2004	Manage SiteKeys
Conspan LA (LRFD) Purchase	Miami Lakes	Active	1	1	9/30/2004	Manage SiteKeys
Conspan LA Demo	Miami Lakes	Expired	1	0	10/1/2003	Manage SiteKeys

1 2

Figure GS-10 Client Product Licenses Window

8. At the Create Site Key window, click **Authorize**. The Create License window opens.

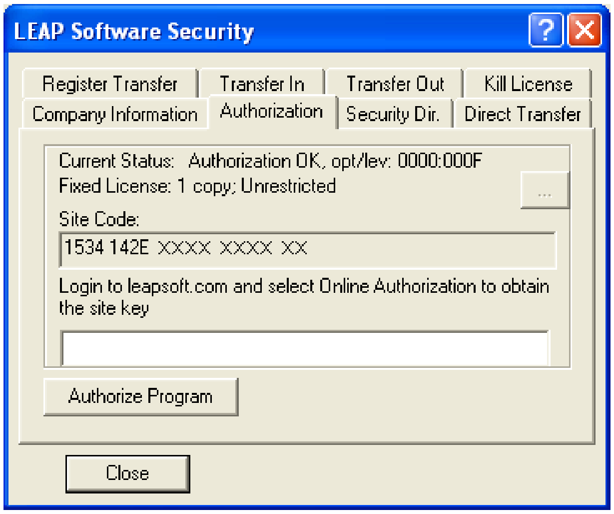
Create License

Site Code:

User Count: 5

Figure GS-11 Create Site Key Window

9. Cut and paste the site code from the Authorization tab of the Security window. Enter the number of user accounts required and click **GetSiteKey**.



LEAP Software Security

Current Status: Authorization OK, opt/lev: 0000:000F

Fixed License: 1 copy; Unrestricted

Site Code:

1534 142E XXXX XXXX XX

Login to leapsoft.com and select Online Authorization to obtain the site key

Figure GS-12 Security - Authorization Tab

10. Cut and paste the Site Key into the Site Key field of the Authorization tab. Click **Authorize Program**.

Demo Authorization

Upon request, LEAP Software issues special demonstration licenses. A demonstration license is a preview of the fully functional version of the program. The demonstration version of the program is the same as the purchased version of the program; however, a demonstration version only lasts for a limited time. To learn more about obtaining a demonstration license, please contact LEAP Software Sales Representatives at (800) 451-5327.

To authorize a demonstration version of the program, follow the instructions above for Program Authorization.

Transferring Authorization

Authorizations may be transferred to different directories and computers. Essentially, there are three ways to transfer authorizations:

- **Direct Transfer on a Single Computer**: Transfer the authorization from one directory to another directory on the same computer.
- **Floppy Disk Transfer**: Transfer the authorization from one computer to another computer when a network license is not present.
- **Direct Transfer to a Network Computer**: Transfer the authorization from one computer to another computer on a network when a network license is present.

Direct Transfer on a Single Computer

To move the program to a new directory, authorization must be transferred before the program in the new location can be run.

For example, if **RC-PIER** is installed in *C:\Programs Files\LEAP Software\RC-PIER* (source directory) and it needs to be moved to *D:\Programs* (target directory), first install the program on the new target directory (that is, *D:\Programs*). After the installation completes, transfer authorization to the target directory as follows:

1. Start the original authorized copy of the program located in the source directory. Make sure the copy of the program on the target directory is not running.
2. Select **Security** from the **File** menu to open the *LEAP Software Security* screen. Select the *Direct Transfer* tab.

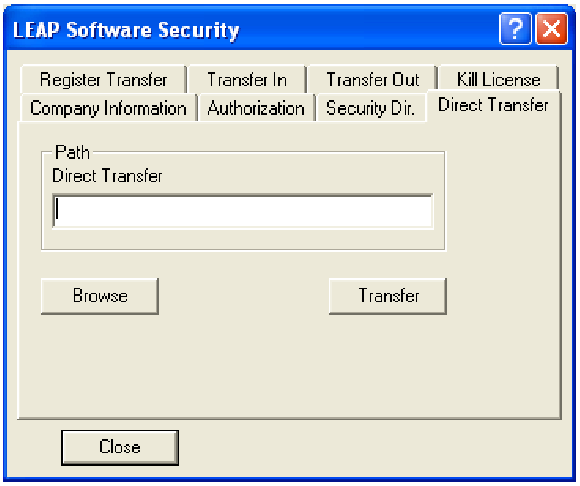


Figure GS-13 Security - Direct Transfer Tab

3. Enter the location of the target directory in the text box, or use the **Browse** button, and click **Transfer**. The authorization instantly transfers from the original installation (source directory) to the new installation (target directory). Now the original program is the unauthorized program and the new program is authorized.
4. Uninstall the copy of the program no longer using (in this case, *C:\Program Files\LEAP Software\RC-PIER*).

Floppy Disk Transfer

If a network license is not available and the program must be moved to another computer, first transfer the authorization. This can be easily accomplished via floppy disk transfer by performing the following steps:

1. Install the program on the target computer.
2. Take a blank, formatted floppy disk to the target computer that contains the unauthorized copy of the program. This blank floppy disk will be called the transfer disk.
3. Insert the transfer disk into the floppy drive and start the unauthorized copy of the program.
4. Authorize the program when prompted.
5. When the *LEAP Software Security* screen displays, select the *Register Transfer* tab to open the appropriate screen.

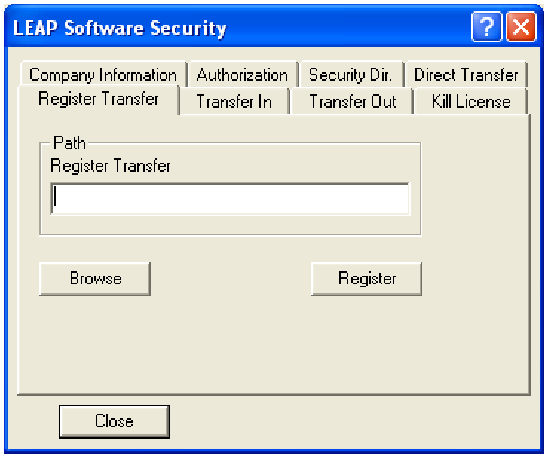


Figure GS-14 Security - Register Transfer Tab

6. Enter the path to the transfer disk (A:\ on most computers), and click **Register**.
7. Once the disk automatically initializes, a notification appears to transfer authorization from an authorized version of the program. When this message appears, remove the transfer disk and exit the program.
8. Insert the transfer disk into the floppy drive of the source computer and start the authorized copy of the program.
9. Select **Security** from the *File* menu to activate the *LEAP Software Security* screen. Select the *Transfer Out* tab.
10. Enter the path location of the transfer disk (A:\), and click **Transfer**. At this point, the original authorized copy of the program is unauthorized and the authorization is on the transfer disk.
11. Take this transfer disk back to the target computer and insert it into the floppy drive.
12. Start the program. Click **Yes** when prompted to Authorize the program.
13. When the *LEAP Software Security* screen appears, select the *Transfer In* tab.
14. Enter the location path of the transfer disk (A:\), and click **Transfer**.
15. The authorization instantly transfers from the original installation (source directory) to the new installation (target directory). At this point, the original program is the unauthorized program and the new program is the authorized program.

Direct Transfer to a Network Computer

To transfer the authorization to another server/host computer on the network, follow the steps below. (A network license is required.)

1. Make sure that the program is installed on the source computer using a network license. To check this, open the program and activate the *LEAP Software Security* screen. Select the *Authorization* tab and confirm that “Floating License” is in the *Current Status* field. A Fixed License is not a network license.
2. Install the program on the target computer using the same setup type as on the source computer (e.g., shared). If necessary, restart the target computer to enable the network protocols.
3. Depending on the operating system, follow the steps outlined in *Network Setup* on page GS-1, to share (with full access) the directory on the target computer where the program was installed.
4. From the source program, open the *LEAP Software Security* screen and select the *Direct Transfer* tab. Type the location of the newly mapped network drive or click **Browse**.
5. Make sure the program on the target computer is not active. Click **Transfer** on the *LEAP Software Security* screen to transfer the license from the source computer to the new target server/host computer.
6. At this point, the program authorized on the server/host computer becomes unauthorized and the new server/host computer becomes authorized.
7. Follow the setup procedures outlined in Network Setup to complete the setup process.

Multiple File Copies

Please note that LEAP Software allows multiple users to simultaneously use a single file. This also applies to other types of **RC-PIER** files such as libraries and other savable files.

Note: To prevent users from accidentally overwriting files, it is recommended to establish different directories for each individual user.

Program Crash/Temporary Loss of License

In case of an unexpected event such as a program crash, the problem may be corrected by deleting the .tb2 files from the program directory. In most cases, this restores authorization. If deleting this file does not correct the problem, contact the Technical Support Department at LEAP Software.

Contacting Technical Support

Technical support is available Monday through Friday (except holidays) from 8:00 AM to 5:00 PM EST at the following numbers:

Tampa	(813) 985-9170
U.S./Canada Toll-Free	(800) 451-5327
International	(813) 985-9170
FAX	(813) 980-3642
E-mail	**techsup@leapsoft.com**
Web Site	**www.leapsoft.com**

Calls are always welcome, but before calling, please consider the following:

- Have you checked the user manual and online help system?
- If you encounter a problem, retrace your steps and check for incorrectly entered information.

Also, please have the following items with you:

- Have your input data ready.
- Have the user manual at hand.
- If you think that you have found an error in the program, have the steps that we should follow to reproduce the error.

Chapter

GO

General Operation

This chapter covers the concepts and procedures of program operation for RC-PIER®, screen-by-screen. Whenever appropriate, refer to the *AASHTO Standard Specifications for Highway Bridge*s, Seventeenth Edition, 2002 and *AASHTO LRFD Bridge Design Specifications,* Third Edition, 2004.

RC-PIER Tab Screens

RC-PIER uses a tab screen interface. This means that each major screen has a corresponding tab. You can instantly access a tab screen by simply clicking on its tab with your mouse.

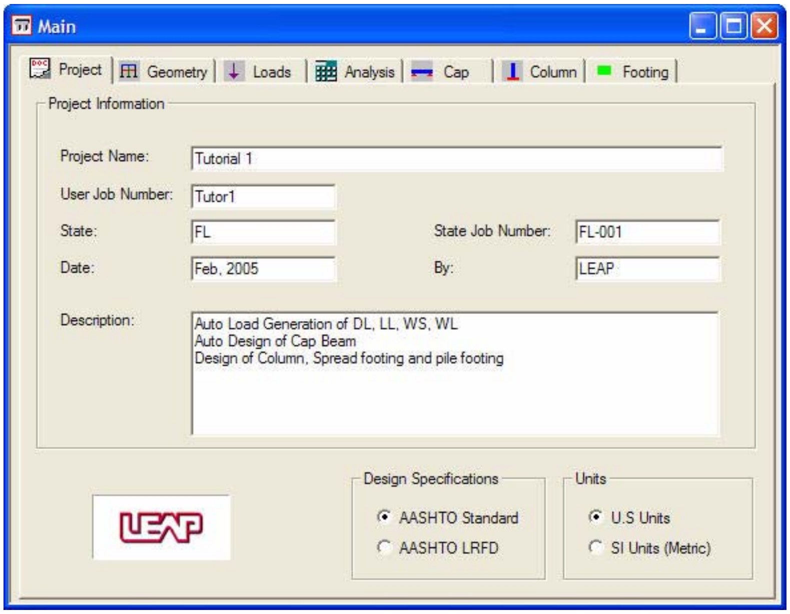
For your convenience, RC-PIER has seven major tab screens: *Project*, *Geometry*, *Loads*, *Analysis*, *Cap*, *Column*, and *Footing*. Each tab screen contains the relevant data. The tab screens appear in the order in which they should be completed.

Project Tab Screen

This is the first screen that you encounter in RC-PIER. (Figure GO-1) It contains specific information related to the project, design code specification, and units selection.

If you return to the *Project* screen at any time to change units, each screen instantaneously converts its values, thereby updating the units to the new selection.

Also, some screens will appear differently depending on the design code selected. For example, changing the specification may reset all load data, void cap, column, and footing designs.



Project Information

Project Name: Tutorial 1

User Job Number: Tutor1

State: FL

State Job Number: FL-001

Date: Feb, 2005

By: LEAP

Description:
Auto Load Generation of DL, LL, WS, WL
Auto Design of Cap Beam
Design of Column, Spread footing and pile footing

Design Specifications
● AASHTO Standard
○ AASHTO LRFD

Units
● U.S Units
○ SI Units (Metric)

Figure GO-1 Project Tab Screen

Geometry Tab Screen

This is the second screen in the tab series, as shown in Figure GO-2. Using the buttons located on the left side of the screen, input the pier layout information, which includes:

- Pier Configuration
- Superstructure information (used for auto load generation)
- Cap, column, and strut geometry
- Bearings (used to locate bearing points; ability to select single or double bearing lines)
- Materials specification
- Structural Model
- Hammer STM (hammerhead strut-and-tie model for LRFD only)

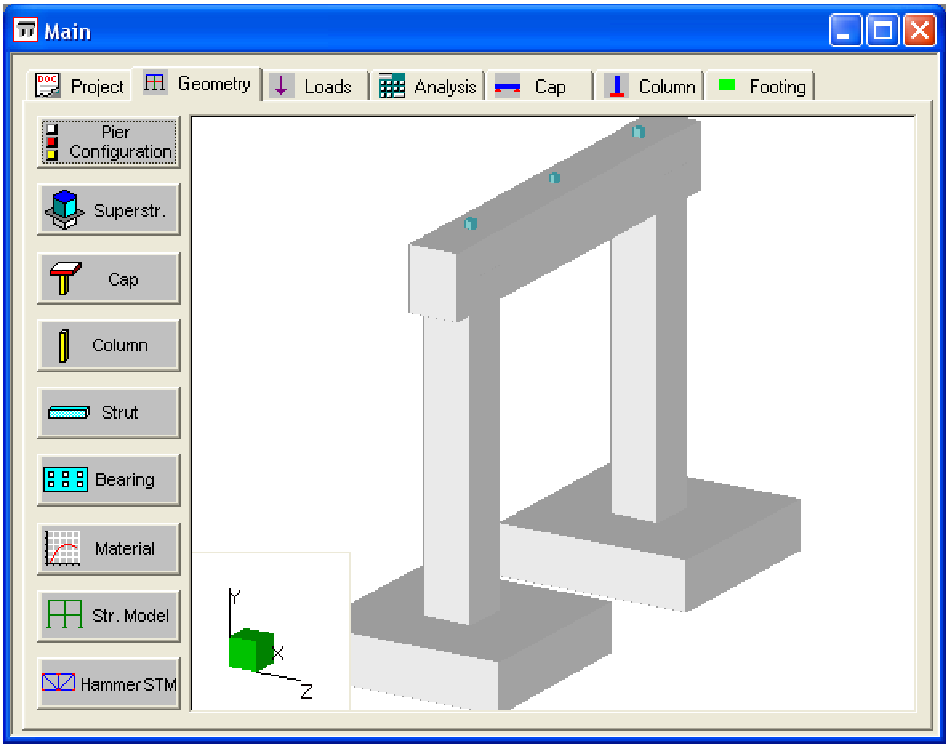


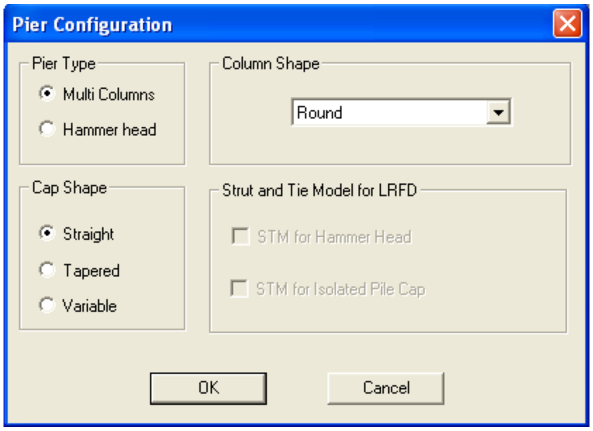
Figure GO-2 Geometry Tab Screen

The buttons appear in the order in which they should be completed. Each button is used to input different required information for the structure.

Notice on the right side of the screen is a model area. This area will display a 3-D representation of the structure as it is created. You can rotate this model by clicking and dragging the mouse over it.

Pier Configuration Button

This button is used to define the basic configuration of the pier structure. Clicking this button will activate the *Pier Configuration* screen, as shown in Figure GO-3. Here, specify the pier type, cap shape, and column shape. For LRFD design code, you can select the strut-and-tie (STM) model for analysis of hammerhead piers or isolated pile/shaft cap.



Pier Configuration

Pier Type
- ● Multi Columns
- ○ Hammer head

Column Shape: Round

Cap Shape
- ● Straight
- ○ Tapered
- ○ Variable

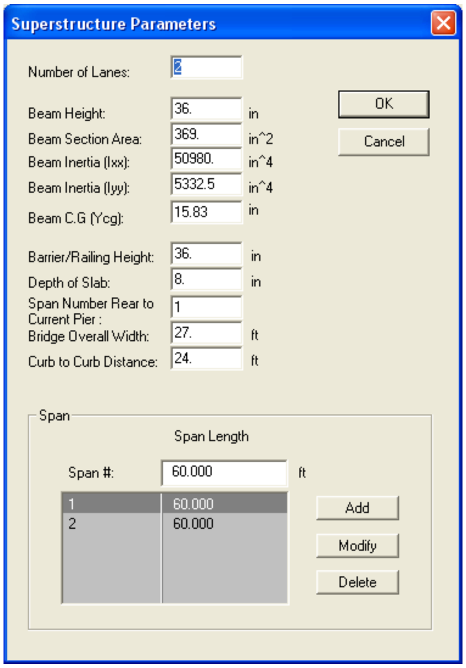
Strut and Tie Model for LRFD
- ☐ STM for Hammer Head
- ☐ STM for Isolated Pile Cap

Figure GO-3 Pier Configuration Screen

You can create a hammerhead pier using a multi-column pier with the number of columns equal to one. This is the same, for the analysis other than STM, as choosing the pier type as hammerhead. However, if you want to use the STM, you must select the pier type as hammerhead.

Superstr. Button

Click **Superstr**. to activate the *Superstructure Parameters* screen, as shown in Figure GO-4. In the appropriate fields, specify the number of lanes, beam height, area, inertia, C.G., barrier/railing height, slab depth, bridge overall width, curb-to-curb distance, and span lengths.



Superstructure Parameters

Number of Lanes: 2

Beam Height: 36. in
Beam Section Area: 369. in^2
Beam Inertia (Ixx): 50980. in^4
Beam Inertia (Iyy): 5332.5 in^4
Beam C.G (Ycg): 15.83 in

Barrier/Railing Height: 36. in
Depth of Slab: 8. in
Span Number Rear to Current Pier : 1
Bridge Overall Width: 27. ft
Curb to Curb Distance: 24. ft

Span

Span Length

Span #: 60.000 ft

1	60.000
2	60.000

Figure GO-4 Superstructure Parameters Screen

Note that the superstructure information is only required if you will be using the auto load generation feature (see page GO-21 for more details) and the impact factor for live load calculated by the program. If you manually input the loads and impact factor, you do not need to input this information.

If you would like to generate live loads for a continuous superstructure, please specify all the spans that are continuous. Please also specify the location of the pier with respect to the spans. However, if you will not be generating any seismic loads and will be doing live load generation using simple span approach, you may specify only two spans adjacent to the pier.

Cap Button

This button allows you to define the dimensions of the pier cap, to include the size, start and end elevation, skew angle, and factor of reduced moment of inertia. Click this button to activate the *Cap Parameters* screen. Notice that this screen will appear differently depending on the pier type selected. Regardless of the choice made, RC-PIER will define sections that are symmetrical about the centerline of the cap in the Z-direction.

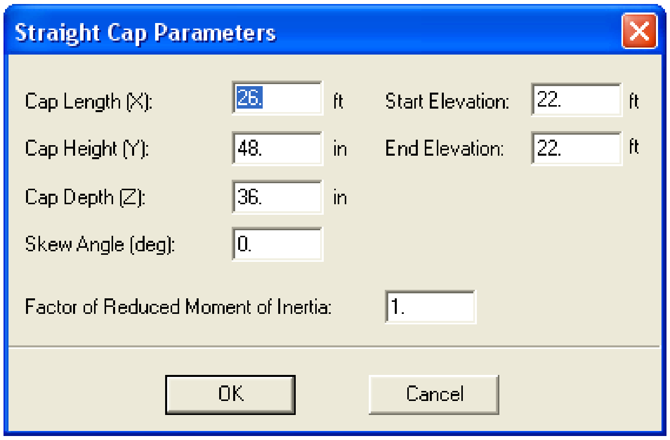
After you entered all information and click **OK**, a three dimensional representation of the cap will appear in the model area on the *Geometry* screen.

RC-PIER assumes that the top surface of a cap is straight. The start and end elevation is at the top of the cap. The skew angle of a cap is only used for auto load generation.

The section of a pier cap must be rectangular. RC-PIER does not allow a circular section or rectangular section with rounded corners. The cap size can be from 5 in. to 200 in., using U.S. units, and for metric units, it can be from 100 mm to 5000 mm.

Straight Cap Parameters Screen

If you selected the cap shape as straight, the *Cap Parameters* screen will resemble Figure GO-5. Here, input the cap's length, height, depth, skew angle, start and end elevation, and factor of reduced moment of inertia in the appropriate fields. Click **OK** to accept all data and return to the *Geometry* screen.


Straight Cap Parameters

Cap Length (X): 26. ft

Cap Height (Y): 48. in

Cap Depth (Z): 36. in

Skew Angle (deg): 0.

Start Elevation: 22. ft

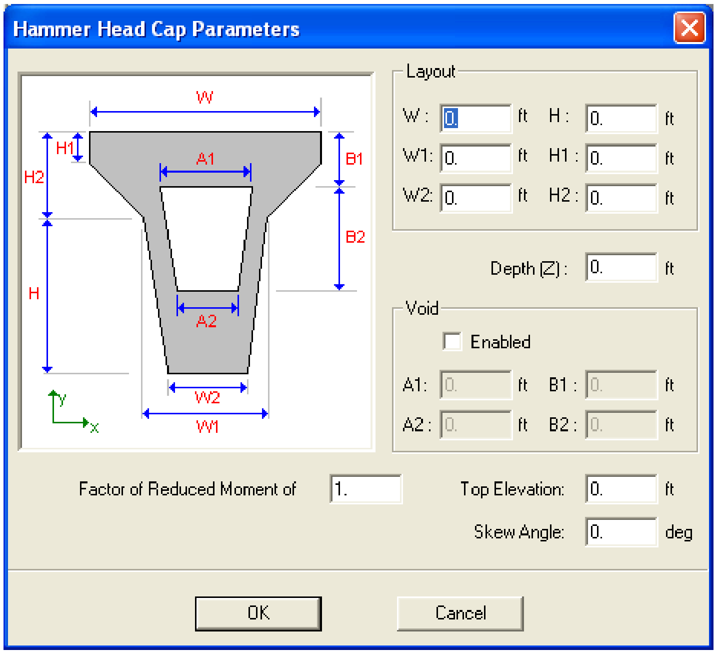
End Elevation: 22. ft

Factor of Reduced Moment of Inertia: 1.

Figure GO-5 Straight Cap Parameters Screen

Hammer Head Cap Parameters Screen

If you selected the pier type as hammerhead, the *Cap Parameters* screen will resemble Figure GO-6. Here, input the cap's width, height, and depth in the appropriate fields. You can also input voids by selecting the Enabled check box and entering the dimensions in the corresponding fields. Click **OK** to accept all data and return to the *Geometry* screen.



Layout
W : 0. ft H : 0. ft
W1: 0. ft H1 : 0. ft
W2: 0. ft H2 : 0. ft
Depth (Z) : 0. ft

Void
☐ Enabled
A1: 0. ft B1 : 0. ft
A2 : 0. ft B2 : 0. ft

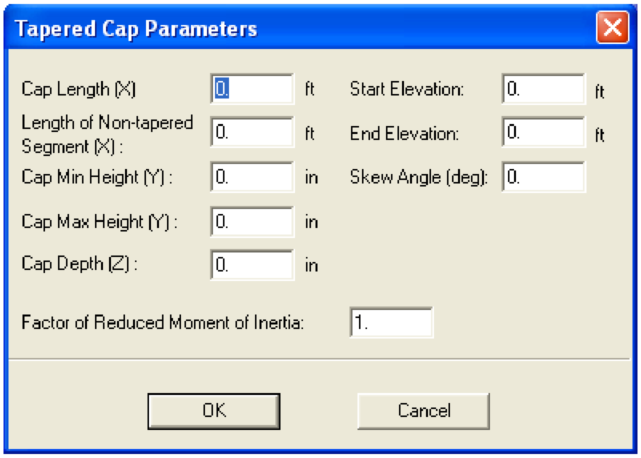
Factor of Reduced Moment of 1.
Top Elevation: 0. ft
Skew Angle: 0. deg

Figure GO-6 Hammer Head Cap Parameters Screen

Note that the cap depth and size in the global Z-direction of the hammerhead must be the same as the column depth and size.

Tapered Cap Parameters Screen

If you selected the cap shape as tapered, the *Cap Parameters* screen will resemble Figure GO-7. A tapered cap is defined in terms of the maximum and minimum cap height, as well as cap non-tapered segment lengths.



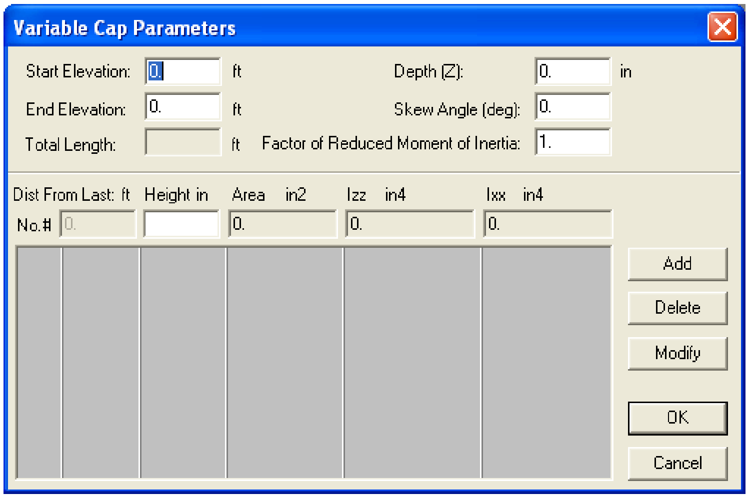
Cap Length (X) 0. ft
Length of Non-tapered Segment (X) : 0. ft
Cap Min Height (Y) : 0. in
Cap Max Height (Y) : 0. in
Cap Depth (Z) : 0. in
Start Elevation: 0. ft
End Elevation: 0. ft
Skew Angle (deg): 0.
Factor of Reduced Moment of Inertia: 1.

Figure GO-7 Tapered Cap Parameters Screen

This button is used to define symmetrical tapers; and therefore, the length of the tapered section will be the same on the left and right sides of the cap.

Variable Cap Parameters Screen

If you selected the cap shape as variable, the *Cap Parameters* screen will resemble Figure GO-8. A variable cap has different section heights along the cap, and RC-PIER assumes that the centerline of a variable cap is a straight line passing through the mid-height of the smaller of the two ends and parallel to the top surface of the cap



Variable Cap Parameters

Start Elevation: 0. ft

End Elevation: 0. ft

Total Length: ft

Depth (Z): 0. in

Skew Angle (deg): 0.

Factor of Reduced Moment of Inertia: 1.

Dist From Last: ft	Height in	Area in2	Izz in4	Ixx in4
No.# 0.		0.	0.	0.

Figure GO-8 Variable Cap Parameters Screen

To define a section, input the elevations of the extremities of the cap, depth, and skew angle. Then, input the height of a section in the Height field and its location from left end of cap and each successive section in the Dist. From Last field. Note that in the Dist. From Last field, you can specify the current section location referring to the last section. Click **Add**. The section will be added to the list on the screen. Repeat this procedure to add additional sections.

To modify any section location and height displayed in the list, highlight it, make the appropriate changes, and click **Modify**. To delete a section, highlight it and click **Delete**.

Cap Parameters Screen Terms

Cap Length (X), ft. The total length of the cap. For piers on skew, the cap length measured along the centerline of the pier cap must be entered.

Cap Height (Y), in. The total height of the cap.

Cap Depth (Z), in. The thickness of the cap.

Skew Angle (deg). Angle between the centerline of a pier (in X-direction) and the normal to the centerline of the bridge. It is positive if measured from the normal to centerline of bridge in the counterclockwise direction. To further illustrate the skew angle, see Figure TH-6.

Start Elevation, ft. Elevation at the left end of the cap, measured at its top surface.

End Elevation, ft. Elevation at the right end of the cap, measured at its top surface.

Centerline of a cap. A straight line that passes through the mid-height of the smaller of two ends of the cap and parallel to its top surface.

Factor of Reduced Moment of Inertia. Used to model the cracking effect. This factor will be multiplied with the moment of inertia of the original section. It is used for analysis only.

Length of Non-Tapered Segment (X), ft. For tapered cap. Length of non-tapered segment.

Cap Min. Height (Y), in. Minimum height of a tapered cap. (Usually at the ends of the tapered pier.)

Cap Max. Height (Y), in. Maximum height of a tapered cap.

Depth (Z), in. The depth of a cap. Measured in the global Z-direction.

Dist From Last, ft. Distance measured from the last section.

Height, in. Height of a section of a variable cap. Always measured in global Y-direction.

Izz, in^4. Moment of inertia of a section about the Z-axis (calculated by the program).

Area, in^2. Total area of a section (calculated by the program).

Ixx, in^4. Moment of inertia of a section about the X-axis (calculated by the program).

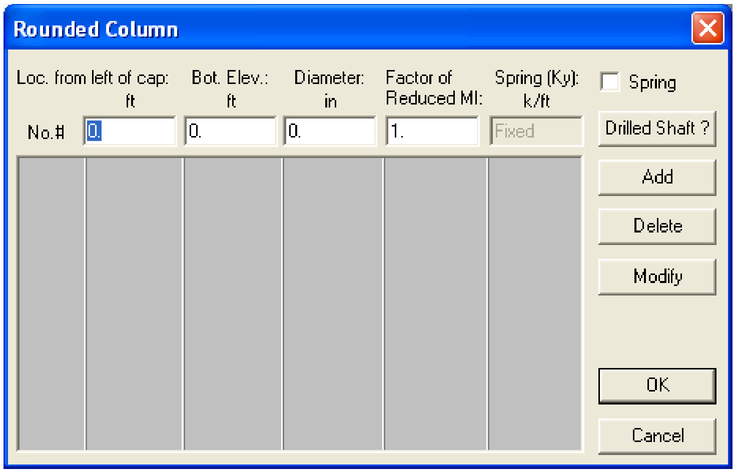
Column Button

Column allows adds columns to the pier structure by activating the *Column* screen. This screen will appear differently depending on the selected column shape (rectangular, round, rectangular tapered, rectangular chamfered, hexagonal, or octagonal).

In RC-PIER, the column size can be from 12 in. to 1000 in., using U.S. units, and from 300 mm to 254000 mm for metric (SI) units. The program does not allow a column to have a hollow section. Note that **Column** is not available for hammerhead pier designs.

Rounded Column Screen

If you selected the column shape as round, the *Column* screen will resemble Figure GO-9. RC-PIER only allows constant circular column sections.



No.# Loc. from left of cap: ft 0.
Bot. Elev.: ft 0.
Diameter: in 0.
Factor of Reduced MI: 1.
Spring (Ky): k/ft Fixed

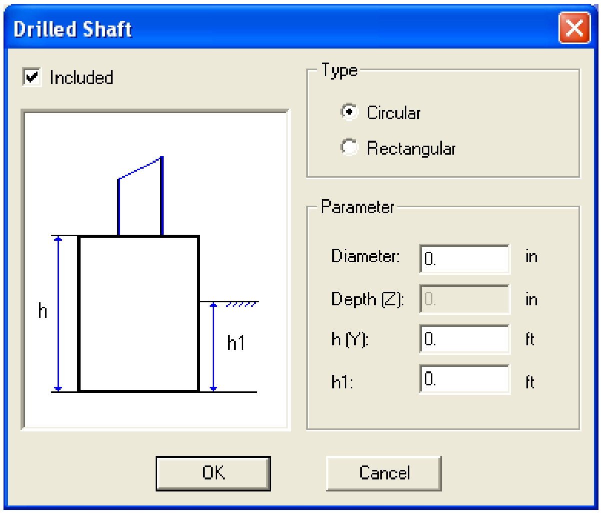
Figure GO-9 Rounded Column Screen

To input a column, input the values in the Loc from left of cap, Bot Elev, Diameter, Factor of Reduced MI fields. The Spring field is available for non-fixed bottom of column conditions only and is only applied in vertical direction (i.e., global Y-direction). In such case, select the Spring check box and input the spring constant in the field. Click **Add** and the column will appear in the list on the screen. Repeat this procedure to add additional columns to the pier structure.

To modify a column, highlight it in the list, make the appropriate changes, and click **Modify**. To delete a column, highlight it and click **Delete**.

To define a drilled shaft as part of a column, click **Drilled Shaft**. This will activate the *Drilled Shaft* screen, as shown in Figure GO-10. Select the type of drilled shaft, either circular or rectangular, then input the values for diameter, depth, width (rectangular only), and height. Select the Included check box. This will include the drilled parameters in the calculations. Click **OK** to accept the values and return to the *Rounded Column* screen. Click **OK** to return to the *Geometry* screen. Notice that the image displayed in the model area will be updated to include the drilled shaft.

Note that the point of fixity and drilled shaft member length used in the frame analysis is defined by the term (h-h1).


Drilled Shaft

☑ Included

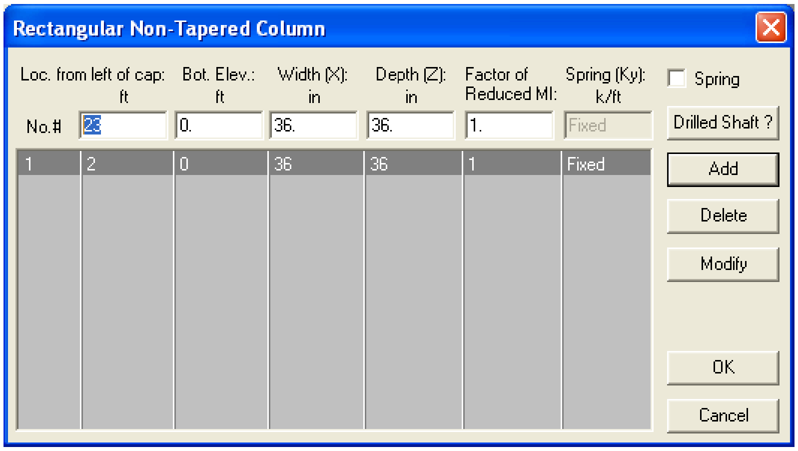
Type
● Circular
○ Rectangular

Parameter
Diameter: 0. in
Depth (Z): 0. in
h (Y): 0. ft
h1: 0. ft

Figure GO-10 Drilled Shaft Screen

Rectangular Non-Tapered Column Screen

If you select the column shape as Rect. Non-Tapered, the *Rectangular Non-Tapered Column* screen will display, as shown in Figure GO-11.


Rectangular Non-Tapered Column

No.#	Loc. from left of cap: ft	Bot. Elev.: ft	Width (X): in	Depth (Z): in	Factor of Reduced MI:	Spring (Ky): k/ft
	23	0.	36.	36.	1.	Fixed
1	2	0	36	36	1	Fixed

☐ Spring

Figure GO-11 Rectangular Non-Tapered Column Screen

Adding a column is similar to the procedure for a rounded column. Input the values in the text boxes at the top of the screen. If this is non-fixed bottom of column conditions, select the Spring check box and input the spring constant in the text box. Click **Add** to add the column to the list. Repeat this procedure to add additional columns to the pier structure.

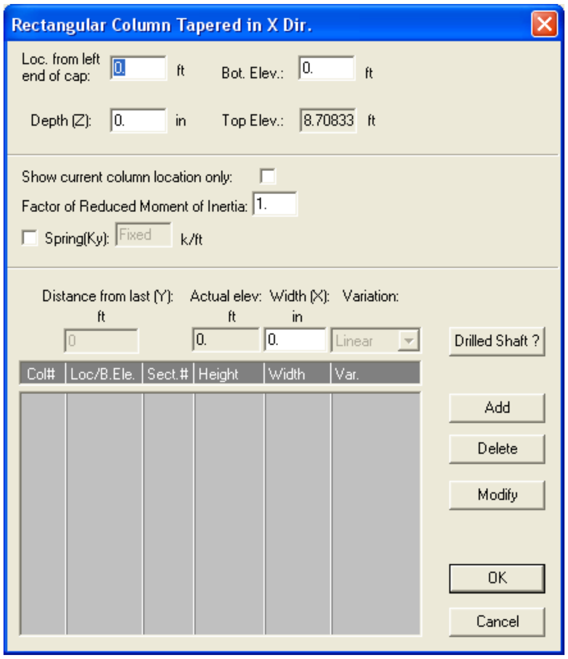
To modify a column, highlight it, make the appropriate changes, and click **Modify**. To delete a column, highlight it and click **Delete**.

When all columns are entered click **OK**.

To add a drilled shaft as part of a column, click **Drilled Shaft**. See page GO-8 for further details.

Rectangular Column Tapered in X-Dir, Z-Dir, and X, Z-Dir Screen

For tapered columns, you have three options: Rect. Tapered in X-dir, Rect. Tapered in Z-dir, or Rect. with both Tapered (X, Z dir). The screens are similar in appearance as shown in Figure GO-12, Figure GO-13, and Figure GO-14, respectively.



Loc. from left end of cap: 0 ft

Bot. Elev.: 0. ft

Depth (Z): 0. in

Top Elev.: 8.70833 ft

Show current column location only: ☐

Factor of Reduced Moment of Inertia: 1.

☐ Spring(Ky): Fixed k/ft

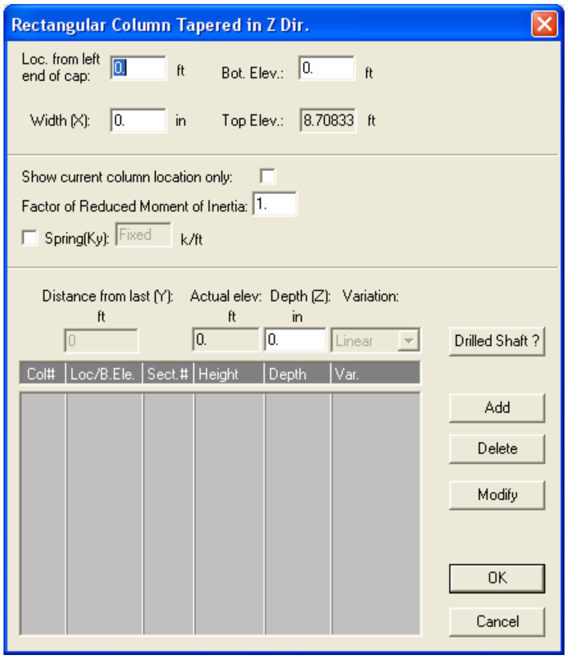
Distance from last (Y) ft: 0

Actual elev ft: 0.

Width (X) in: 0.

Variation: Linear

Figure GO-12 Rectangular Column Tapered in X-Dir


Rectangular Column Tapered in Z Dir.

Loc. from left end of cap: 0. ft

Bot. Elev.: 0. ft

Width (X): 0. in

Top Elev.: 8.70833 ft

Show current column location only: ☐

Factor of Reduced Moment of Inertia: 1.

☐ Spring(Ky): Fixed k/ft

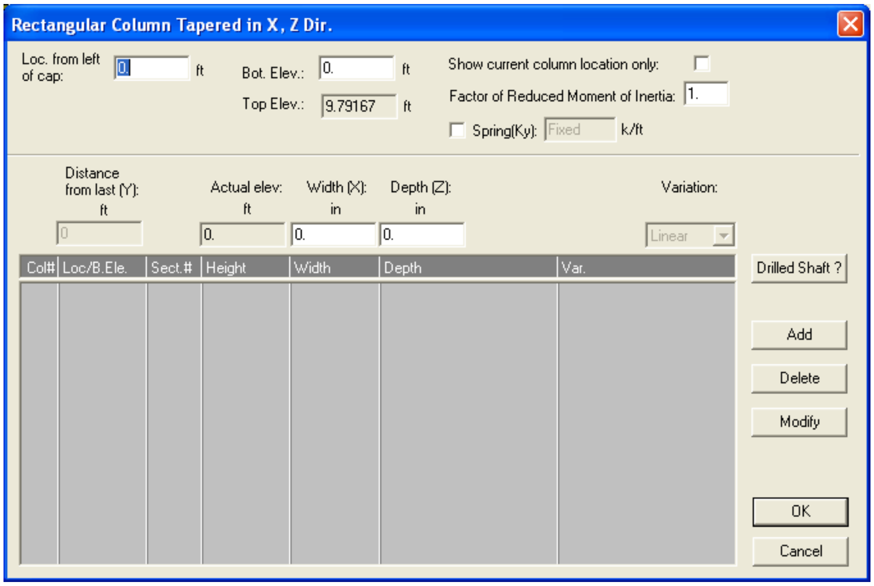
Distance from last (Y): ft: 0

Actual elev: ft: 0.

Depth (Z): in: 0.

Variation: Linear

Figure GO-13 Rectangular Column Tapered in Z Dir Screen


Rectangular Column Tapered in X, Z Dir.

Loc. from left of cap: 0. ft

Bot. Elev.: 0. ft

Top Elev.: 9.79167 ft

Show current column location only: ☐

Factor of Reduced Moment of Inertia: 1.

☐ Spring(Ky): Fixed k/ft

Distance from last (Y): ft: 0

Actual elev: ft: 0.

Width (X): in: 0.

Depth (Z): in: 0.

Variation: Linear

Figure GO-14 Rectangular Column Tapered in X, Z Dir Screen

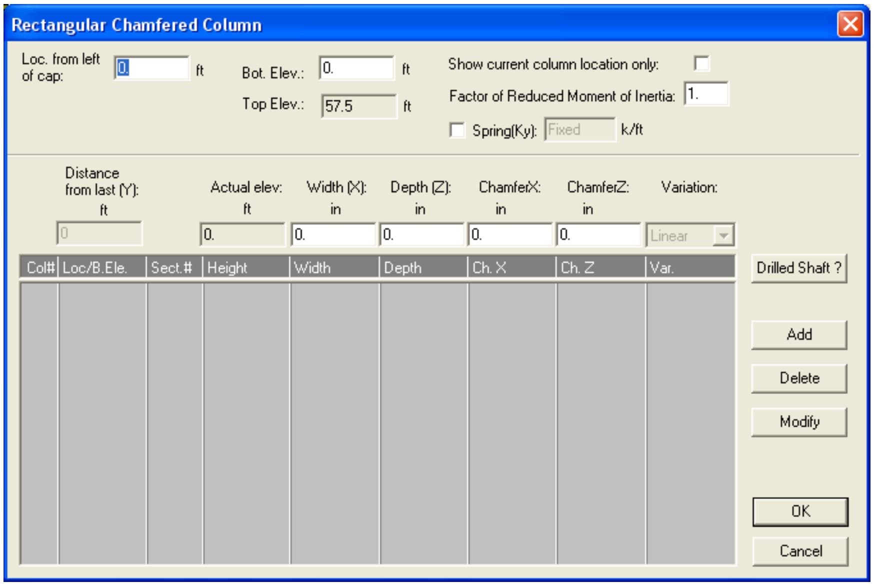
For every column in the pier at least two cross-sections must be specified. To enter the first column, input the values in the Loc from Left of Cap, Bot. Elev, Depth, and Width fields. Click **Add** to add the column to the list. RC-PIER will calculate the value in the Top Elev field, which is the location where the top of the column intersects the pier cap centerline, using the information provided in the Cap Parameters screen (for more information refer to *Cap Button* on page GO-4).

To add sections to the columns, input the Distance From Last field. Enter a new value in the Width (x) or Depth (z) field and select either linear or parabolic variation. Click **Add**. The new column section will appear in the list. At least two sections must be specified for a column. However, first define the section at the bottom of the column and then add additional segments. For each segment you can specify the segment height. When all columns are entered click **OK**.

To modify a column, select it and change the parameters as desired. Click **Modify** to apply the changes.

For multiple columns, to show only the current column on the screen, select the “Show current column location only” check box. Note that on the *Rectangular Column Tapered in X, Z-Dir* screen, you will need to enter the values in the width and depth fields for both directions.

Rectangular Chamfered Column Screen



Loc. from left of cap: 0. ft
Bot. Elev.: 0. ft
Top Elev.: 57.5 ft
Show current column location only: ☐
Factor of Reduced Moment of Inertia: 1.
☐ Spring(Ky): Fixed k/ft

Distance from last (Y): ft	Actual elev: ft	Width (X): in	Depth (Z): in	ChamferX: in	ChamferZ: in	Variation:
0	0.	0.	0.	0.	0.	Linear

Col#	Loc/B.Ele.	Sect.#	Height	Width	Depth	Ch. X	Ch. Z	Var.

Figure GO-15 Rectangular Chamfered Column Screen

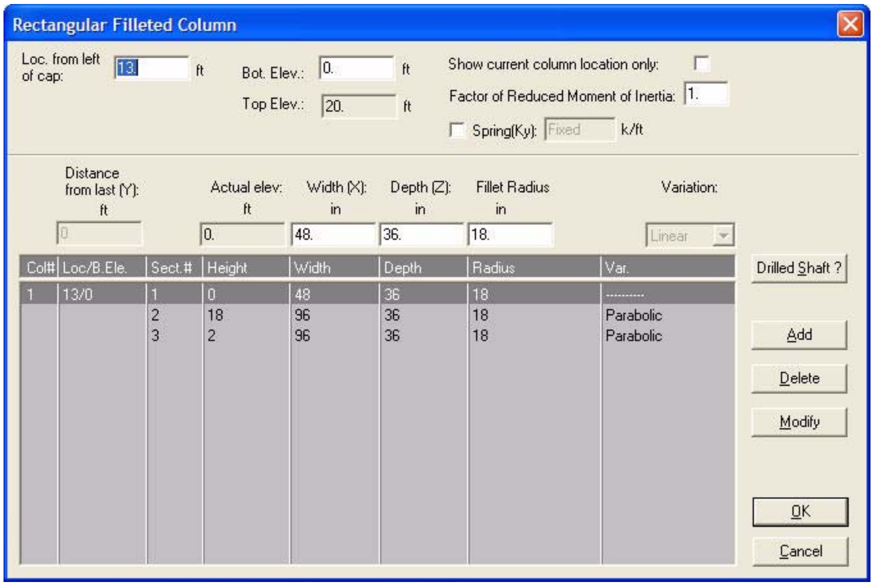
For every column in the pier at least two cross-sections must be specified. To enter the first column, input the values in the Loc from Left of Cap, Bot. Elev, Width, Depth, ChamferX, and ChamferZ fields. Click **Add** to add the column to the list. RC-PIER will calculate the value in the Top Elev field, which is the location where the top of the column intersects the pier cap centerline, using the information provided in the Cap Parameters screen (for more information refer to *Cap Button* on page GO-4).

To add sections to the columns, enter the Distance From Last field. If desired enter new values in the Width, Depth, and Chamfer (X or Z) fields and select either linear or parabolic variation. Click **Add**. The new column segment will appear in the list. Add any other segments if needed. Then add other columns in the pier in the same way. When all columns are entered click **OK**.

To modify a column, select it and change the parameters as desired. Click **Modify** to apply the changes.

For multiple columns, to show only the current column on the screen, select the “Show current column location only” check box.

Rectangular Filleted Column Screen



Loc. from left of cap: 13 ft

Bot. Elev.: 0. ft

Top Elev.: 20. ft

Show current column location only: ☐

Factor of Reduced Moment of Inertia: 1.

☐ Spring(Ky): Fixed k/ft

Distance from last (Y): ft	Actual elev: ft	Width (X): in	Depth (Z): in	Fillet Radius in	Variation:
0	0.	48.	36.	18.	Linear

Col#	Loc/B.Ele.	Sect.#	Height	Width	Depth	Radius	Var.
1	13/0	1	0	48	36	18	---------
		2	18	96	36	18	Parabolic
		3	2	96	36	18	Parabolic

Figure GO-16 Rectangular Filleted Column Screen

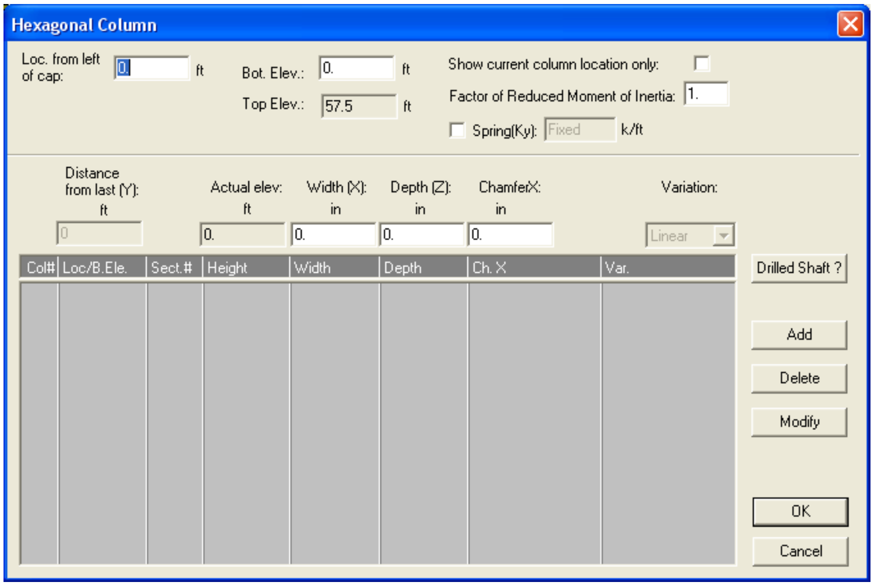
For every column in the pier at least two cross-sections must be specified. To enter the first column, input the values in the Loc from Left of Cap, Bot. Elev, Width, Depth, Fillet Radius fields. Click **Add** to add the column to the list. RC-PIER will calculate the value in the Top Elev field, which is the location where the top of the column intersects the pier cap centerline, using the information provided in the Cap Parameters screen (for more information refer to *Cap Button* on page GO-4).

To add sections to the columns, enter the Distance From Last field. If desired enter new values in the Width, Depth, and Chamfer (X or Z) fields and select either linear or parabolic variation. Click **Add**. The new column segment will appear in the list. Add any other segments if needed. Then add other columns in the pier in the same way. When all columns are entered click **OK**.

To modify a column, select it and change the parameters as desired. Click **Modify** to apply the changes.

For multiple columns, to show only the current column on the screen, select the “Show current column location only” check box.

Hexagonal Column Screen



Loc. from left of cap: 0. ft

Bot. Elev.: 0. ft

Top Elev.: 57.5 ft

Show current column location only: ☐

Factor of Reduced Moment of Inertia: 1.

☐ Spring(Ky): Fixed k/ft

Distance from last (Y): 0 ft

Actual elev: 0. ft

Width (X): 0. in

Depth (Z): 0. in

ChamferX: 0. in

Variation: Linear

Figure GO-17 Hexagonal Column Screen

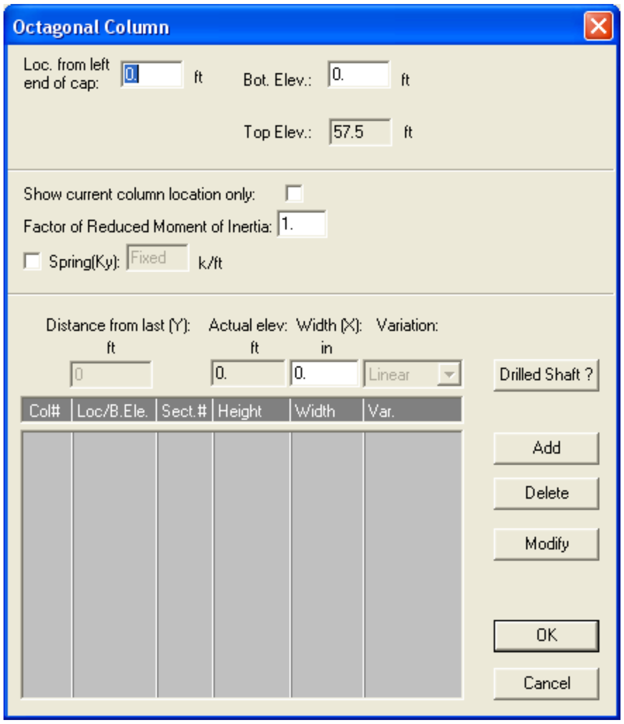
For every column in the pier at least two cross-sections must be specified. To enter the first column, input the values in the Loc from Left of Cap, Bot. Elev, Width, Depth, and Chamfer fields. Click **Add** to add the column to the list. RC-PIER will calculate the value in the Top Elev field, which is the location where the top of the column intersects the pier cap centerline, using the information provided in the Cap Parameters screen (for more information refer to *Cap Button* on page GO-4).

To add sections to the columns, enter the Distance From Last field. If desired enter a new value in the Width, Depth, or Chamfer fields and select either linear or parabolic variation. Click **Add**. The new column segment will appear in the list. Add any other segments if needed. Then add other columns in the pier in the same way. When all columns are entered click **OK**.

To modify a column, select it and change the parameters as desired. Click **Modify** to apply the changes.

For multiple columns, to show only the current column on the screen, select the “Show current column location only” check box.

Octagonal Column Screen



Loc. from left end of cap: 0 ft
Bot. Elev.: 0. ft
Top Elev.: 57.5 ft
Show current column location only: ☐
Factor of Reduced Moment of Inertia: 1.
☐ Spring(Ky): Fixed k/ft
Distance from last (Y): ft 0
Actual elev: ft 0.
Width (X): in 0.
Variation: Linear

Figure GO-18 Octagonal Column Screen

For every column in the pier at least two cross-sections must be specified. To enter the first column, input the values in the Loc from Left of Cap, Bot. Elev, and Width fields. Click **Add** to add the column to the list. RC-PIER will calculate the value in the Top Elev field, which is the location where the top of the column intersects the pier cap centerline, using the information provided in the Cap Parameters screen (for more information refer to *Cap Button* on page GO-4).

To add sections to the columns enter the Distance From Last field and if desired, a new value in the Width field and select either linear or parabolic variation. Click **Add**. The new column segment will appear in the list. Add other segments if needed. Then add other columns in the pier in the same way. When all columns are entered click **OK**.

To modify a column, select it and change the parameters as desired. Click **Modify** to apply the changes.

For multiple columns, to show only the current column on the screen, select the “Show current column location only” check box.

Column Screen Terms

Loc. from left end of cap, ft. Define the centerline location of a column, measured from the left end of the cap.

Diameter, in. Diameter of a column.

Factor of Reduced Moment of Inertia. Used to model the cracking effect. This factor will be multiplied by the moment of inertia of the original section.

Spring (Ky), k/ft. This option allows you to apply a spring to the bottom of the column. This term refers to the stiffness of the spring.

Depth (Z). The depth of a column, measured in global Z-direction.

Bot. Elev., ft. The elevation at the bottom of a column.

Distance from last (Y), ft. Distance from the last section of a column.

Actual Elev, ft. The actual elevation of the present section of a column (calculated by the program).

Width (X), in. The width of a column, measured in Global X-direction.

Izz, in^4. Moment of inertia of a section about Z-axis (calculated by the program).

Ixx, in^4. Moment of inertia of a section about X-axis (calculated by the program).

Chamfer x: The chamfer in the global X-direction.

Chamfer z: The chamfer in the global Z-direction.

Fillet Radius: Radius of the corner fillet for a Rectangular Filleted Column.

Drilled Shaft Button. Click this button to open the *Drilled Shaft* screen, to define the parameters of the drilled shaft for a column.

The following terms relate to the Drilled Shaft Screen.

Type. The type or shape of the drilled shaft, either circular or rectangular.

Diameter. The diameter of the drilled shaft.

Width. The width of a rectangular drilled shaft in the global X-direction.

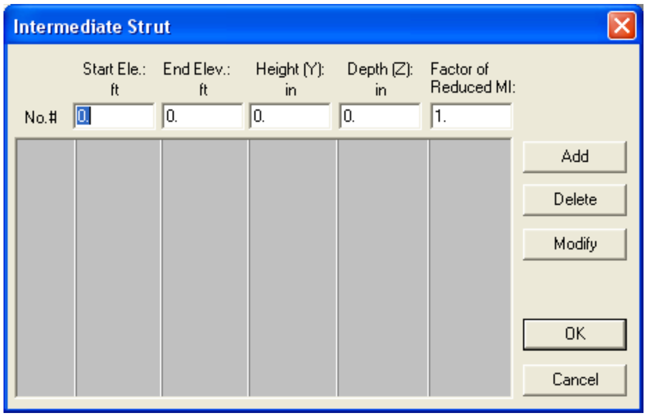
Depth. The depth of a rectangular drilled shaft in the global Z-direction.

h. Total length of a drilled shaft.

h1. Part of the drilled shaft underground level. Note that the drilled shaft is fixed at ground level in the structure model ($h – h_1$).

Strut Button

Strut adds intermediate struts to the pier structure. Note, a maximum of two intermediate struts are allowed. To add intermediate struts, click **Strut** to activate the *Intermediate Strut* screen, as shown in Figure GO-19.



Start Ele.: ft	End Elev.: ft	Height (Y): in	Depth (Z): in	Factor of Reduced MI:
0.	0.	0.	0.	1.

Figure GO-19 Intermediate Strut Screen

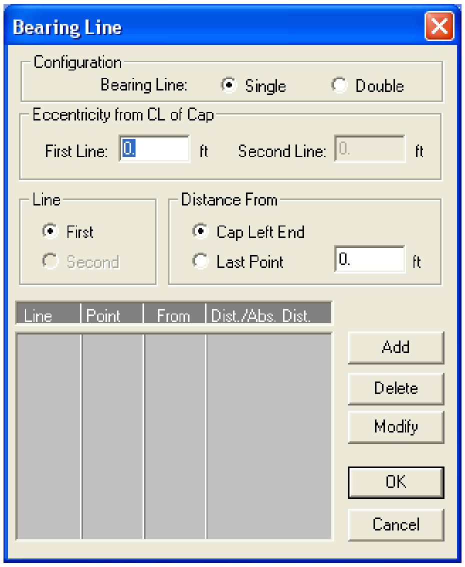
Input the values for start and end elevation, height, depth, and factor of reduced moment of inertia. Note that the start and end elevations for a strut are referred to as its top surface. Click **Add** and it will appear in the list on the screen. Repeat this procedure to add additional struts.

When finished, click **OK** and the program returns to the *Geometry* screen. To modify an existing strut in the list, highlight it, change the appropriate fields, and click **Modify**. To delete an existing strut, highlight it and click **Delete**.

Note that this option is not available for hammerhead piers.

Bearing Button

Bearing defines the configuration of the bearing line(s), eccentricity, and location of each bearing point. Click **Bearing** to activate the *Bearing Line* screen, as shown in Figure GO-20.



Bearing Line: ● Single ○ Double

First Line: 0 ft Second Line: 0. ft

● First ○ Second

● Cap Left End ○ Last Point 0. ft

Figure GO-20 Bearing Line Screen

To input the bearing points follow the steps below.

1. Select the bearing line by clicking the **Single** or **Double** option under Configuration.
2. Input the values of eccentricity for the first and second lines, where eccentricity is the distance of the bearing line from the centerline of the cap. A positive value is measured in the positive global Z-axis from the centerline of cap. A negative value is measured in the negative global Z-axis.
3. Select either the **First** or **Second** option under line. Note if you selected a single bearing line (Single option under Configuration), only the First option will be available.
4. Add evenly spaced bearings. Under **Distance From**, select the **Cap Left End** or **Last Point** option. For the first bearing, the **Cap Left End** option must be used. For all other bearing points, select the **Last Point** option. This simplifies the input of the same value multiple times; each new bearing spaced evenly apart from the previous bearing.
5. Click **Add**. The bearing point will appear in the list on the screen. Repeat this procedure to add additional bearing points.
6. Click **OK** when you have finished inputting all the bearing points. The program returns to the *Geometry* screen.
7. Modify an existing bearing point by highlighting it, making the appropriate changes, and clicking **Modify**. To delete a bearing point from the list, highlight it and click **Delete**.

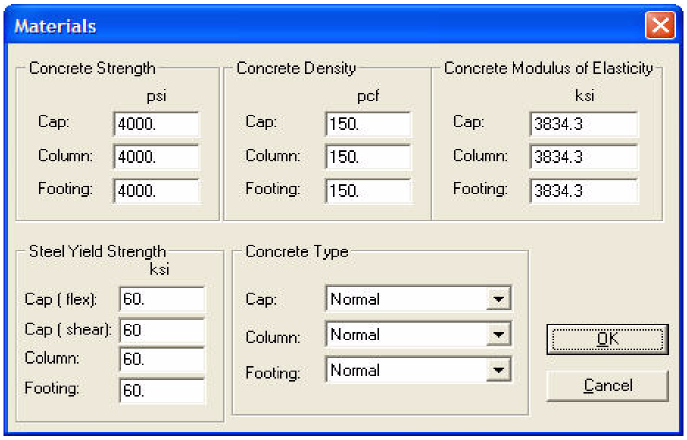
Note in the case of a sloped cap, the bearing location is measured horizontally, instead of along the cap (i.e., parallel to the global x-axis).

Also note, for bearings at an eccentricity on a skewed pier, distance along X is measured separately for two bearing lines. This will result distinct values of bearing location in two lines.

Material Button

Use this button to define the concrete strength, concrete density, concrete modulus of elasticity, steel yield strength, and concrete type.

To start, click **Material** to activate the *Materials* screen as shown in Figure GO-21, and input the desired values in the appropriate fields.



	Concrete Strength psi	Concrete Density pcf	Concrete Modulus of Elasticity ksi
Cap:	4000.	150.	3834.3
Column:	4000.	150.	3834.3
Footing:	4000.	150.	3834.3

Steel Yield Strength ksi	
Cap (flex):	60.
Cap (shear):	60
Column:	60.
Footing:	60.

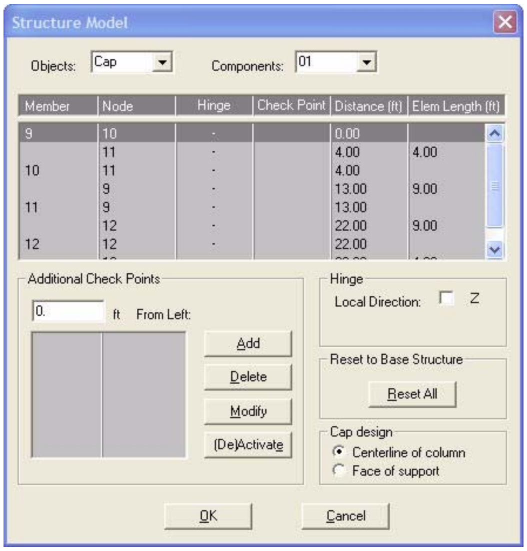
Concrete Type	
Cap:	Normal
Column:	Normal
Footing:	Normal

Figure GO-21 Materials Screen

Note that RC-PIER automatically calculates the elasticity; but these values can be edited manually. Also, there are three concrete types available in RC-PIER: normal, sand-lightweight, and all-lightweight. There are separate options to specify the steel yield strength for cap flexure steel, cap shear steel, column steel and footing steel.

Str. Model Button

Use this button to activate the *Structure Model* screen, as shown in Figure GO-22. While defining each of the pier parameters, the program automatically generates a structural model, to include all the nodes, members, and connectivity conditions (results shown in the list at the top of the *Structure Model* screen). In addition, you can add or remove new checkpoints to the pier structure, as well as define hinges at existing nodes, or choose to do cap design at face of supports or column center lines.



Objects: Cap

Components: 01

Member	Node	Hinge	Check Point	Distance (ft)	Elem Length (ft)
9	10	-		0.00	
	11	-		4.00	4.00
10	11	-		4.00	
	9	-		13.00	9.00
11	9	-		13.00	
	12	-		22.00	9.00
12	12	-		22.00	

0. ft From Left:

Local Direction: ☐ Z

● Centerline of column
○ Face of support

Figure GO-22 Structure Model Screen

To add additional checkpoints, input a value in the From Left text box (cap and struts) or from bottom (column) and click **Add**. It will appear as active in the list under Additional Check Points.

You can modify an existing checkpoint from the list by highlighting it, changing the value, and clicking **Modify**. To delete a checkpoint, highlight it and click **Delete**.

Additional checkpoints, once added, can be active or inactive mode. To change the mode of the checkpoint from active to deactive mode, highlight the checkpoint in the list and click **Deactivate**. Note that only active checkpoints are considered for later analysis. To reset all node settings to the default setting, click **Reset All**.

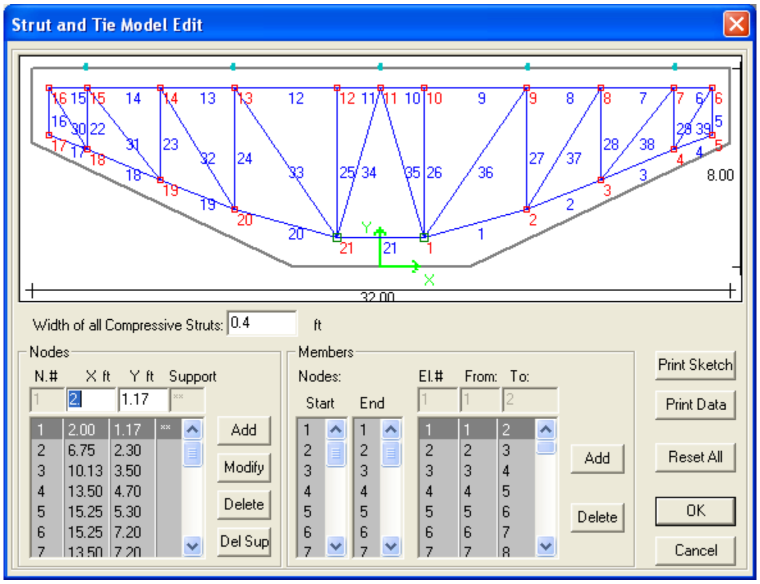
The hinge can only be added in Z-direction to any node. For example, if you assign the hinge to node number 3, the moment about Z-direction (M_z) will be zero.

To do the cap design for moment and shear at the column centerline locations, choose "centerline of column" option. However, if you choose, "face of support" option for cap design, program will automatically add check points at both faces of columns and do the design for those values ignoring any values in cap over the supported portion of columns.

To see a graphical model of the structure, select **Model** from the **Show** menu, or its representative icon on the toolbar. This option allows you to view the node and member numbers and checkpoints. (See page GO-65 for more information on this option.)

Hammer STM Button

This option is only available for hammerhead pier types (LRFD only). Click **Hammer STM** to activate the *Strut-and-Tie Model Edit* screen, as shown in Figure GO-23, and add or modify the nodes of a strut and tie model. Note the diagram at the top of the screen shows a model of the STM with all nodes and members (taken from Tutor3.rcp), where the dimension of the hammerhead pier is outlined and the position of existing bearing points are shown by small green squares on top of the cap.



Width of all Compressive Struts: 0.4 ft

Nodes

N.#	X ft	Y ft	Support
1	2.	1.17	**
1	2.00	1.17	**
2	6.75	2.30	
3	10.13	3.50	
4	13.50	4.70	
5	15.25	5.30	
6	15.25	7.20	
7	13.50	7.20	

Members

Nodes:

Start	End
1	1
2	2
3	3
4	4
5	5
6	6
7	7

El.#	From:	To:
1	1	2
1	1	2
2	2	3
3	3	4
4	4	5
5	5	6
6	6	7
7	7	8

Figure GO-23 Strut and Tie Model Edit Screen

To define a strut-and-tie model, first add the node coordinates, then connect them to create members within the cap. There are two ways you can define a strut-and-tie model, as follows.

The first method is using the input boxes under Nodes and Members. To add a node, input the values in the X and Y text boxes under Nodes and click **Add**. It will appear in the list below. Modify a node in the list by highlighting it, making the appropriate changes, and clicking **Modify**. To delete a node, highlight it and click **Delete**.

Add a support to a node by clicking **Set Sup**. If the node is already designated as a support node, this button will be **Del Sup**; click to remove the support from the node. For trusses, at least two nodes must be designated as Support Nodes (e.g., in Figure GO-23, nodes 1 and 21 are support nodes).

To add a member, select a node from the Start and End lists and click **Add**. It will appear in the list under Members. You can delete a member by highlighting it and clicking **Delete**

The second method involves using the mouse to create the nodes and members directly in the diagram at the top of the screen. Point the mouse where you want the first node to be in the diagram. Click the left mouse button. A square will appear at this point to indicate a new node. Now, drag the mouse to another location on the diagram. A line appears, extending from the first node. Click the right mouse button. A square appears at this point to indicate another new node has been created, and also, a new member between the two nodes. To stop between creating nodes and members, click the right mouse button.

To remove all data from the screen, click **Reset All**. Click **OK** to accept any changes and return to the *Geometry* screen.

Loads Tab Screen

The *Loads* screen, as shown in Figure GO-24, is where all load information is entered. You can add preset individual loads and load groups to the pier structure from the two lists on the screen, Load Type and Available Groups.

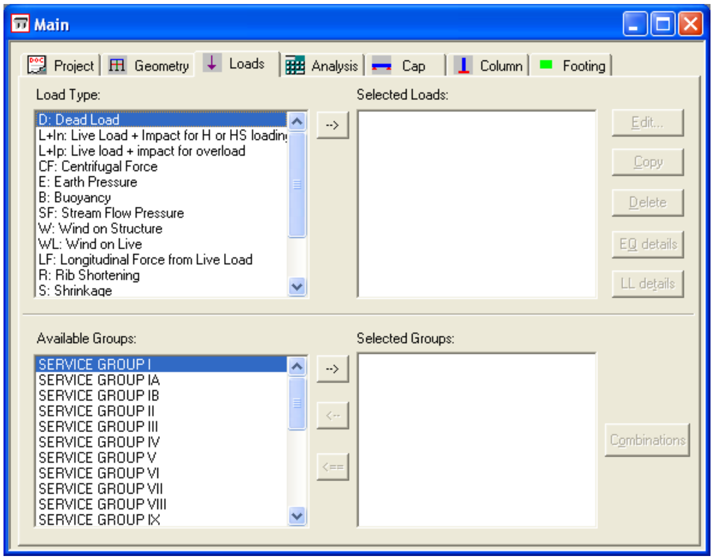


Figure GO-24 Loads Tab Screen

Load Types are preset individual loads that you can apply to the pier structure for analysis and design. To add a load, highlight the load type in the Load Type list and click the -> button. It will appear in the list under Selected Loads. To delete a selected load, highlight it and click **Delete**.

RC-PIER allows you to define specific parameters for a selected load. Simply, highlight the load in the list under Selected Loads and click **Edit**. This will bring up the *Loads: Load Data* screen, as shown in Figure GO-25. Here, you can manually edit the parameters for a load, or allow the program to automatically generate the loads. This is one of the most powerful features available in RC-PIER. For more details on auto load generation, see page TH-9.

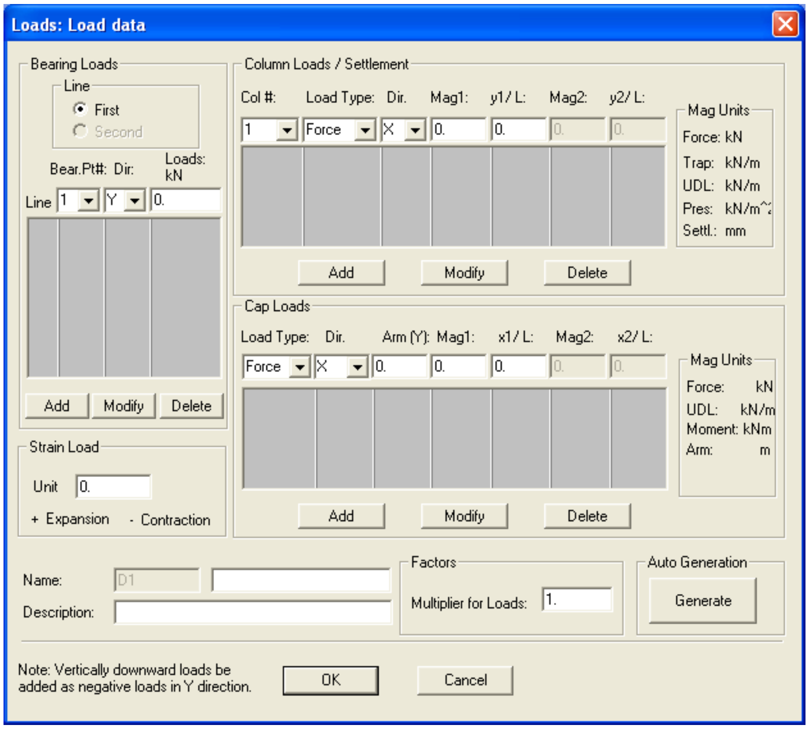
Click **EQ Details** to view the details of the last auto-generated earthquake load. For the last auto generated live load, click **LL Details**.

Available Groups consist of preset load combinations that can be added to the pier structure. To add a load group to the Selected Groups list, highlight a group from the Available Groups list and click the -> button. Click the <- to remove a group from the Selected Groups list or <== to remove all selected groups. If you want to create and add your own load group, select the **Load Groups/Limit States** item from the **Libraries** menu. (For more information on this option, see page GO-69). Note that RC-PIER always provides the load groups/limit states according to AASHTO Standard or LRFD as default.

After an analysis has been performed, you can view the details of all load combinations created by the program by clicking **Combinations.**

Loads: Load Data Screen

As mentioned above, RC-PIER allows you to define specific parameters for a selected load. Highlight the selected load and click **Edit** to activate the *Loads: Load Data* screen, as shown in Figure GO-25.



Bearing Loads: Line 1, Dir. Y, Loads 0.

Strain Load: Unit 0.

Column Loads / Settlement: Col # 1, Load Type Force, Dir. X, Mag1 0., y1/L 0.

Cap Loads: Load Type Force, Dir. X, Arm (Y) 0., Mag1 0., x1/L 0.

Name: D1

Multiplier for Loads: 1.

Note: Vertically downward loads be added as negative loads in Y direction.

Figure GO-25 Loads: Load Data Screen

You can either manually input the parameters for load type, direction, and bearing loads or allow the program to automatically generate the load. (See Auto Load Generation Screens on page GO-21 for more information on this feature.)

The fields under Column Loads/Settlement will vary depending on the load type selected (Force, Trapezoid, UDL, Pressure, and Settlement). Also, this screen uses the global coordinate system, e.g., loads acting down in the Y direction must be negative.

Bearing loads (X, Y, and Z directions) will be applied at the top of the pier cap.

Cap loads can have applied forces in the X, Y or Z-direction (positive or negative) applied moments about X, Y, or Z direction. Uniform distributed load (UDL) can be applied to the cap in the X, Y or Z-direction (positive or negative). Note that the UDL maybe applied over the full or part cap length. When concentrated force in X or Z direction is applied, RC-PIER allows you to specify Arm (Y). This is the height above the cap centerline at which the load acts.

Cap and column loads will be applied at the centerline of the cap and column respectively.

Each load can have an individual name and description. Input a name in the Name text box and a brief description of the load in the Description text box. Click **OK** to accept all information and return to the *Loads* screen.

The location of column loads is measured with respect to the fixity point. The fixity point will be at the bottom of a column, or at the fixity point for a drilled shaft. The location of loads on a column is specified as a ratio of y1/L or y2/L. If a distributed load starts a quarter point from the bottom, y1/L will be equal to 0.25. If this load ends at the mid-height of the column, y2/L will be equal to 0.5. A unit strain load can be entered over the entire pier; end a positive value for expansion and a negative value for contraction.

Additional load factors can be entered for each individual load type by modifying the value in the Multiplier for Loads text box under Factors on the *Loads: Load Data* screen. Note that all loads entered will be increased accordingly by this value (default value = 1.0).

For manually input live loads in AASHTO LRFD, this screen behaves slightly different. When a live load case is added in LRFD mode and edited, the program fills the bearing loads list. This contains two sets of loads. The first set of bearing loads corresponds to truck loads and the second set corresponds to lane load part of Design Truck loads. Initially, all these loads are set equal to zero. Users can only modify loads for this case but cannot delete the entries. To apply a truck load, select the corresponding bearing location within the first set, edit the load value, and click **Modify**. To specify a lane load part on a bearing, scroll down in the list and select the bearing location in the second set of loads and update it. To remove a previously specified load, set that load equal to zero.

Note that the impact factor and multiple presence factors for live load should be defined in the *Analysis/Design Parameters* screen on the *Analysis* tab (Figure GO-36 and Figure GO-37). If you are manually inputting the live load, enter the loads without the impact factor.

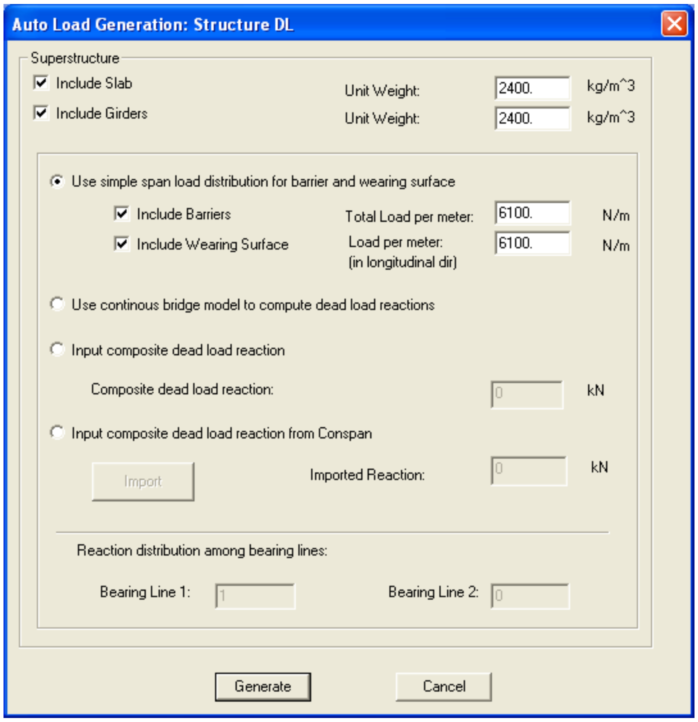
Auto Load Generation Screens

One of the most powerful features in RC-PIER is the auto load generation. This feature allows the program to automatically generate the loads for dead load, live load, wind load on structure, wind load on live load, earthquake load, and vessel collision (for LRFD only). Select one of these loads from the Selected Loads list on the *Loads* screen and click **Edit** to activate the *Loads: Load Data* screen. Then, click **Generate** to activate the *Auto Load Generation* screen. This screen will be specific to the load type selected, as explained below. The loads generated will be used later for the analysis of the pier.

Note before using this feature, you must define the superstructure parameters on the *Geometry* screen.

Auto Load Generation: Structure DL Screen

The *Auto Load Generation: Structure DL* screen, as shown in Figure GO-26, is used for auto generation of dead loads. Select the appropriate check boxes to include or not include the dead load of slabs and girders as superstructure dead loads acting on the pier cap at the bearing point locations. Then, input the appropriate unit weights.



☑ Include Slab — Unit Weight: 2400. kg/m^3

☑ Include Girders — Unit Weight: 2400. kg/m^3

● Use simple span load distribution for barrier and wearing surface

☑ Include Barriers — Total Load per meter: 6100. N/m

☑ Include Wearing Surface — Load per meter: (in longitudinal dir) 6100. N/m

○ Use continous bridge model to compute dead load reactions

○ Input composite dead load reaction

Composite dead load reaction: 0 kN

○ Input composite dead load reaction from Conspan

Imported Reaction: 0 kN

Reaction distribution among bearing lines:

Bearing Line 1: 1 Bearing Line 2: 0

Figure GO-26 Auto Load Generation: Structure DL Screen

There are four options available that allow users to select how they wish to generate the composite dead loads:

- **Using Simple Span Distribution**

  To generate the dead load using the simple span distribution for barriers and wearing surface, select the appropriate check boxes to include or not include the dead load. Then, enter the appropriate unit weights and load per foot.

- **Using Continuous Beam Model**

  If you would like to compute dead loads of the superstructure considering the continuity of spans and supports, select this option.

- **Entering a Composite Reaction**

  To generate the dead load by entering a total composite reaction, select this option and enter the value of the composite dead load reaction. This option may be used for bridges which are either continuous over the pier and a DL reaction from superstructure is already computed.

- **Importing the Composite Dead Load Reaction**

Using this option you can import CONSPAN computed support reactions into the program. However, you will first need to export the results from CONSPAN using File: Export to RC-PIER. Then in RC-PIER to import the composite dead load reaction from CONSPAN, select this option and click **Import** to open the *Import Load Reactions from CONSPAN* screen as explained on page GO-26.

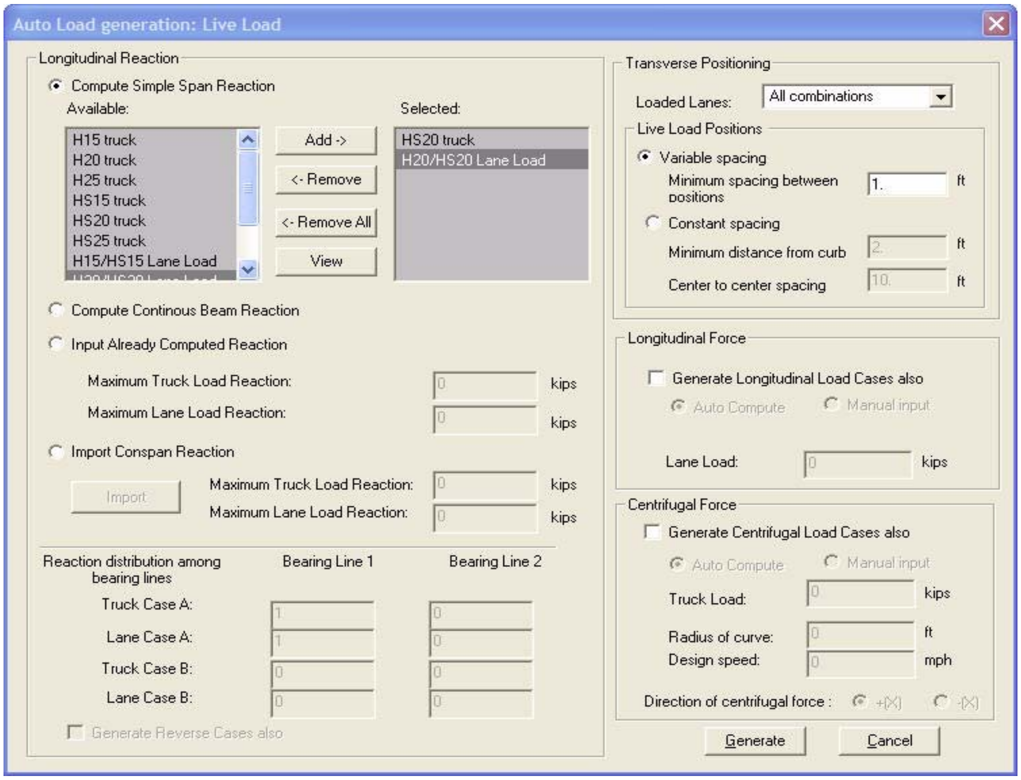
When there are two bearing lines present, the *Auto Load Generation: Structure DL* also allows you to specify the distribution of loads among bearing lines. A factor from 0.0 to 1.0 may be specified for either of the two bearing lines. If there is only one bearing line, all the load must be supported by it.Therefore, the default value is 1.0; if there are two lines, the default value for both is 0.5 which will distribute the load equally between two lines.

Once all required data is entered, click **Generate**. RC-PIER will automatically generate the loads and return to the *Loads: Load Data* screen. If any of the loads generated automatically needs to be changed, select the particular load on the *Loads: Load Data* screen, make the appropriate changes, and click **Modify**.

Note that auto dead load generation with first option uses simple span analysis. As such, you can adjust the values from auto generation by a continuity factor for a continuous bridge. A continuity factor can be entered in the Multiplier for Loads text box under Factors on the *Loads: Load Data* screen.

Auto Load Generation: Live Load Screen

For auto generating live loads, click **Generate** to activate the *Auto Load Generation: Live Load* screen, as shown in Figure GO-27.



Longitudinal Reaction

● Compute Simple Span Reaction

Available:
- H15 truck
- H20 truck
- H25 truck
- HS15 truck
- HS20 truck
- HS25 truck
- H15/HS15 Lane Load

Selected:
- HS20 truck
- H20/HS20 Lane Load

○ Compute Continous Beam Reaction

○ Input Already Computed Reaction

Maximum Truck Load Reaction: 0 kips

Maximum Lane Load Reaction: 0 kips

○ Import Conspan Reaction

Maximum Truck Load Reaction: 0 kips

Maximum Lane Load Reaction: 0 kips

Reaction distribution among bearing lines	Bearing Line 1	Bearing Line 2
Truck Case A:	1	0
Lane Case A:	1	0
Truck Case B:	0	0
Lane Case B:	0	0

☐ Generate Reverse Cases also

Transverse Positioning

Loaded Lanes: All combinations

Live Load Positions

● Variable spacing

Minimum spacing between positions 1. ft

○ Constant spacing

Minimum distance from curb 2 ft

Center to center spacing 10. ft

Longitudinal Force

☐ Generate Longitudinal Load Cases also

● Auto Compute ○ Manual input

Lane Load: 0 kips

Centrifugal Force

☐ Generate Centrifugal Load Cases also

● Auto Compute ○ Manual input

Truck Load: 0 kips

Radius of curve: 0 ft

Design speed: 0 mph

Direction of centrifugal force : ● +[X] ○ -[X]

Figure GO-27 Auto Load Generation: Live Load Screen

There are four options available that allow users to select how they wish to generate the longitudinal reactions for live loads:

- **Compute Simple Span Reaction**

To use simple span superstructure live load reactions, highlight the live load(s) from the list under Available and click **Add**. The load will appear in the list under Selected. To view the configuration of the live load, click **View**.

RC-PIER always provides the standard live loads according to the AASHTO Specification. However, you can add custom live loads by selecting the **Vehicular Loads** option from the **Libraries** menu. See page GO-67 for more information about adding custom loads.

- **Compute Continuous Beam Reactions**

  If you would like to compute reactions on the pier considering the continuity of the spans and the supports, select this option. To generate live loads using this option, select **Live Loads** from the list and click **Add**.

- **Input Computed Reaction**

  If you have already performed live load analysis of superstructure and already know the live load reactions on the pier, you can specify those using this option, In LFD, you may choose to specify either truck load or lane load or both. However in case of LRFD you will need to specify both lane and truck reactions. Program will then try to position this reaction in transverse direction for rest of the live load generation process.

  To generate a live load by entering an already computed longitudinal reaction, select this option and specify the maximum truck and lane load reactions.

- **Import CONSPAN Reaction**

  If you use CONSPAN for superstructure analysis, you can export the results of longitudinal reactions from it on supports to a file and then import those in RC-Pier for further LL generation.

  To import the longitudinal live load reaction from CONSPAN, select this option and click **Import** to open the *Import Load Reactions from CONSPAN* screen as explained on page GO-26.

The *Auto Load Generation: Live Load* screen also allows you to specify the distribution of loads among bearing lines. Any value from 0.0 to 1.0 may be specified for a maximum of two bearing lines.

When you choose either option 2 or 3 on this dialog, you will also need to specify the factors in which load will be distributed among bearing lines if here are two bering lines. Program allows you input factors for two cases (Case A and Case B) for truck and lane loads.

To generate live loads using transverse positioning, selected the number of lanes to be loaded from the Loaded Lanes list.

Using the live load generation, RC-PIER first creates live load positions, where live load position is defined by the center of a truck (resultant of two axles) or center of a lane load. After the program has all available live load positions, it selects these positions producing maximum load effect (see page TH-10 in the Theory chapter for more information). There are two choices in which you can have live load positions: 1) variable spacing and 2) constant spacing. Variable spacing is where RC-PIER creates the positions according to a couple of predefined rules (criteria). For an explanation of these rules, see page TH-12. Constant spacing is where RC-PIER creates the positions using two inputs provided by the user and assumes even spacing of live load positions. (For further details, see page TH-11.) One input is the minimum distance of the nearest truck wheel or tandem load from the curb, and the second is the center-to-center spacing between two consecutive positions (truck and lane). Note that the edge of the lane load is always placed against the face of the curb.

The live loads generated by RC-PIER do not include the impact factor. The live load generation can be very time-consuming, since the program includes the comprehensive live load positions. Also, the auto live load generation uses simple span analysis for load distribution to the bearings.

This dialog now allows you to optionally generate longitudinal (braking) load and centrifugal load at the same time along with live load. These however, can also be generated separately from respective load case dialogs (i.e. longitudinal and centrifugal force dialogs).

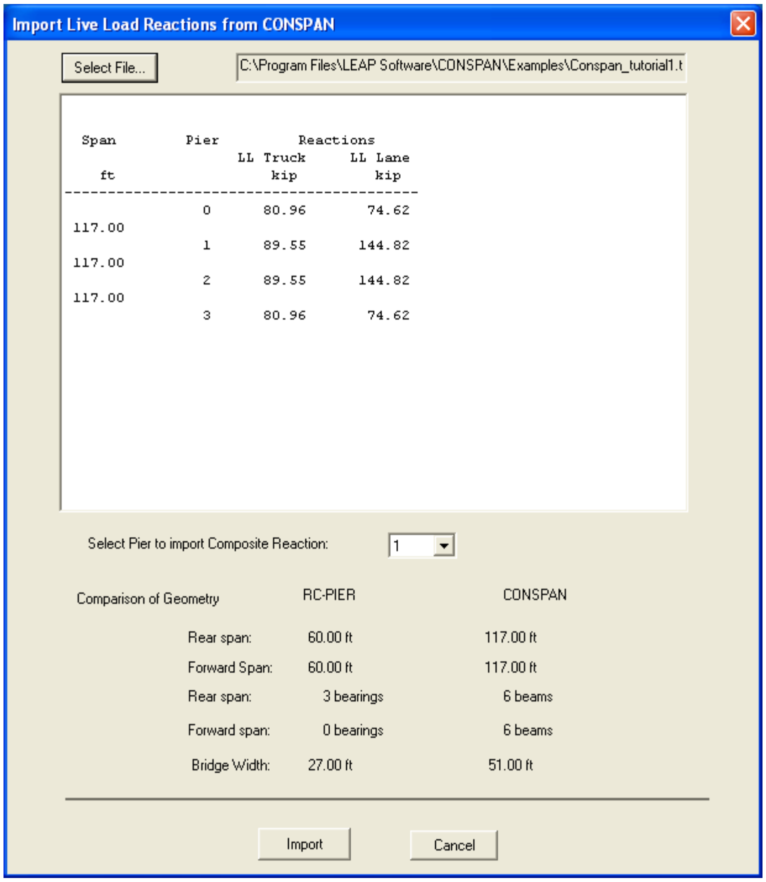
To generate longitudinal forces at the same time along with live load, check the correspondingly option. Then if you would like to generate longitudinal loads for the live loads selected on this dialog, choose auto compute. For this option program will use contributing length to be equal to average of the two adjacent spans. However, you can also switch to manual input and then specify the total live load for one lane to be considered. Please read more details about longitudinal load generation in the section Auto Load Generation: Longitudinal/Braking load on page GO-26 later in this chapter and Auto Longitudinal/Braking Load Generation on page TH-20 in the Theory chapter.

To generate centrifugal load at the same time along with live load, check the corresponding option. If you would like to do this generation based on already selected live loads, select the Auto compute option. Then specify the Radius of curve and the design speed. Depending upon the curvature choose to apply the load in the + X direction or - X direction. If you would like to use a different value of total live load, then choose the manual input in this area and specify the total live load to be considered for centrifugal loads for one lane. Please read more details about centrifugal load generation in the section Auto Load Generation: Centrifugal Force on page GO-27 later in this chapter and Auto Centrifugal Load generation on page TH-21 in the Theory chapter.

Once all required data is entered, click **Generate**. RC-PIER will automatically generate all live loads and return to the *Loads: Load Data* screen.

Import Live Load Reactions from CONSPAN Screen

This screen allows you to import composite dead load reactions and longitudinal live load reactions from CONSPAN, a LEAP Software program used for the analysis and design of Simple and Continuous Precast/Prestressed Bridge Beams.



C:\Program Files\LEAP Software\CONSPAN\Examples\Conspan_tutorial1.t

```
  Span         Pier          Reactions
                       LL Truck     LL Lane
   ft                     kip          kip
-----------------------------------------
                 0      80.96        74.62
117.00
                 1      89.55       144.82
117.00
                 2      89.55       144.82
117.00
                 3      80.96        74.62
```

Select Pier to import Composite Reaction: 1

Comparison of Geometry	RC-PIER	CONSPAN
Rear span:	60.00 ft	117.00 ft
Forward Span:	60.00 ft	117.00 ft
Rear span:	3 bearings	6 beams
Forward span:	0 bearings	6 beams
Bridge Width:	27.00 ft	51.00 ft

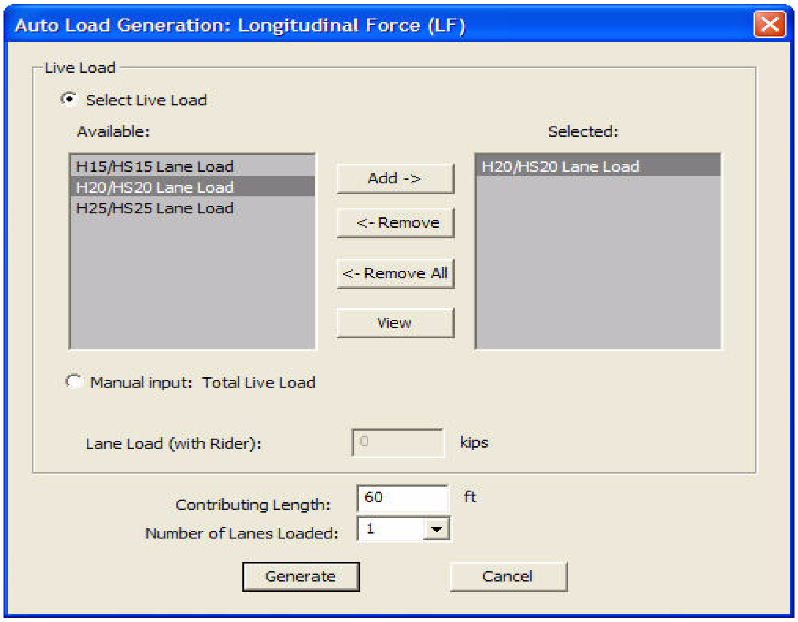
Figure GO-28 Import Live Load Reactions from CONSPAN Screen

Once you have selected a file, RC-PIER will display the selected CONSPAN output and the corresponding load reactions. Select the pier number for which you want to import the reactions and the program displays a geometry comparison of the rear and forward spans related to the selected pier as defined in each program.

To import the pier reaction data, click **Import**. If there are significant differences in the geometry, a warning message will display to confirm whether the data should be imported. If you select **Yes**, the imported reaction will display either as dead load reactions or the Maximum Truck Load Reaction and Maximum Lane Load Reactions fields depending upon if you are importing dead loads or live loads. If the user selects **No**, the dialog box will close and the import will be cancelled.

Auto Load Generation: Longitudinal/Braking Force Screen

This screen is used for auto generating longitudinal/braking force, as shown in Figure GO-29. Click Generate on the Loads: Load Data dialog of type LF in AASHTO standard code or type BR in LRFD code.



● Select Live Load

Selected: H20/HS20 Lane Load

○ Manual input: Total Live Load

Lane Load (with Rider): 0 kips

Contributing Length: 60 ft

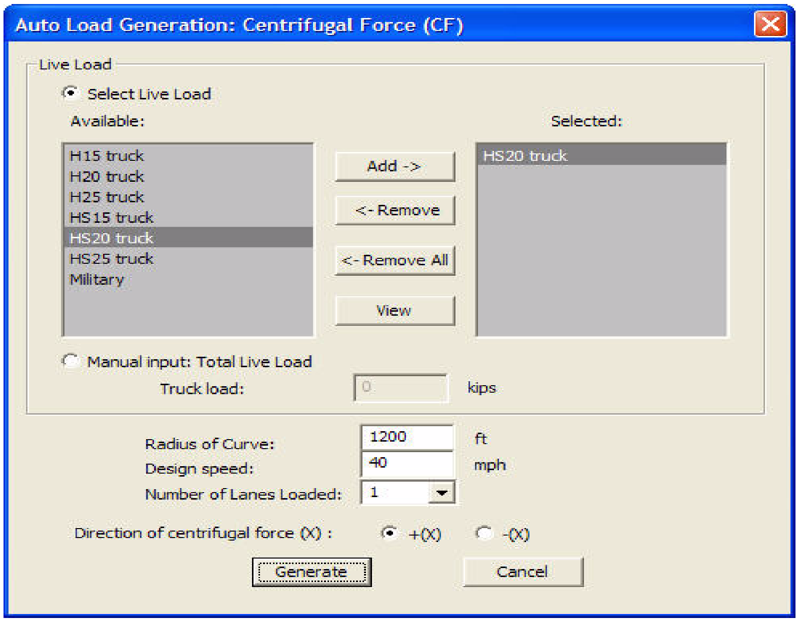
Number of Lanes Loaded: 1

Figure GO-29 Auto Load Generation: Longitudinal/Braking Force Screen

For this generation, you can select live loads specified in the program Vehicular Load library and specify the contributing length along with the number of lanes loaded. Note that as per AASHTO standard code, the braking force is determined based on lane load only. Therefore, in AASHTO standard only lane loads are available for generation. As per AASHTO LRFD, the braking force depends on axle weights as well as lane load therefore all the live loads are available for generation. Contributing length is the length of the bridge for which the loads should be generated. This is mainly used with lane load intensity. The other option allows you to specify the total live load (per lane) to use for computation of longitudinal/braking force.

Auto Load Generation: Centrifugal Force Screen

This screen is used for auto generating centrifugal force, as shown in Figure GO-30. Click Generate on the Loads: Load Data dialog for a CF case in AASHTO Standard or CE in AASHTO LRFD code.



● Select Live Load

Available:
- H15 truck
- H20 truck
- H25 truck
- HS15 truck
- HS20 truck
- HS25 truck
- Military

Selected: HS20 truck

○ Manual input: Total Live Load

Truck load: 0 kips

Radius of Curve: 1200 ft

Design speed: 40 mph

Number of Lanes Loaded: 1

Direction of centrifugal force (X) : ● +(X) ○ -(X)

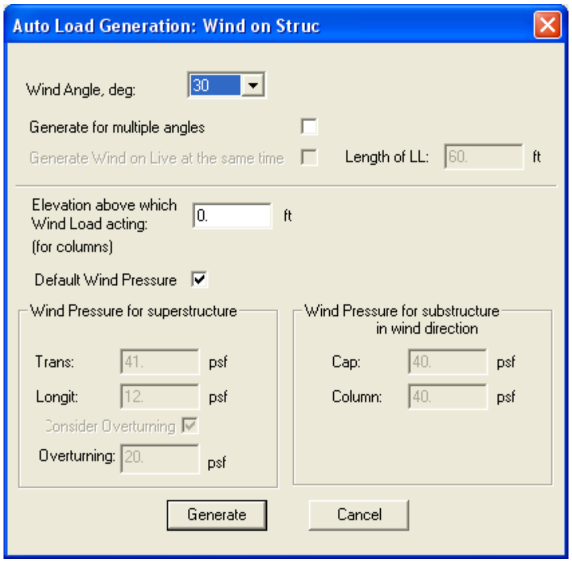
Figure GO-30 Auto Load Generation: Centrifugal Force Screen

For this generation, you can select one of the available live loads. The available list is based on the loads in the vehicular library. You can modify/add vehicle loads in the library. See page GO-55 for details about that. In addition to vehicular load, you need to specify the radius of the curve and the direction in which to apply the loads. If the center of curve lies to the right of bridge when looking upstation, specify the load to be in +X direction. Otherwise specify it to be in –X direction. If the pier is at skew program will compute load as two components. One along X and one along Z.

The manual input option allows you to specify the total live load (per lane) to use for computations of centrifugal force.

Auto Load Generation: Wind on Structure Screen

For auto generating wind loads on structure, click **Generate** to activate the *Auto Load Generation: Wind on Structure* screen, as shown in Figure GO-31.



Auto Load Generation: Wind on Struc

Wind Angle, deg: 30

Generate for multiple angles ☐

Generate Wind on Live at the same time ☐ Length of LL: 60. ft

Elevation above which Wind Load acting: (for columns) 0. ft

Default Wind Pressure ☑

Wind Pressure for superstructure

Trans: 41. psf

Longit: 12. psf

Consider Overturning ☑

Overturning: 20. psf

Wind Pressure for substructure in wind direction

Cap: 40. psf

Column: 40. psf

Figure GO-31 Auto Load Generation: Wind on Structure Screen (LRFD)

Select the wind angle from the Wind Angle list. The portion of the column subjected to wind load is measured by the elevation defined in the “Elevation above which Wind Load acting” text box. RC-PIER applies the wind load to the column between this elevation and top of the column. To manually change the default wind pressure, deselect the check box and input the values for wind pressure in the text boxes under Wind Pressure for Superstructure and Wind Pressure for Substructure. Overturning pressure is used only when the wind angle is zero. The bridge location option is only applicable for AASHTO LRFD.

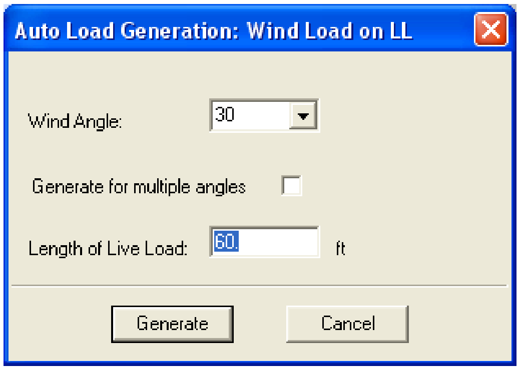
You can have the program generate a few wind loads on structure simultaneously for different wind angles by selecting the Generate for multiple angles check box and then inputting the start and end wind angle values. In addition, you can generate the wind on live load at the same time as the wind load on structure for the same wind angle by selecting the Generate Wind on Live at the same time check box.

Click **Generate** and the program will automatically generate the wind load on structure and return to the *Loads: Load Data* screen.

Note that the wind angle is measured from the normal of the bridge centerline (traffic direction) to the wind direction. It is positive if counterclockwise. The component of X-direction of a positive wind load is along the global X-direction.

Auto Load Generation: Wind Load on Live Load Screen

This screen is used for auto generating wind load on live loads, as shown in Figure GO-32. Select the wind angle from the Wind Angle list and input the length of the live load in the Length of the Live Load text box. Click **Generate** and the program will automatically generate the loads and return to the *Loads: Load Data* screen.



Wind Angle: 30

Generate for multiple angles ☐

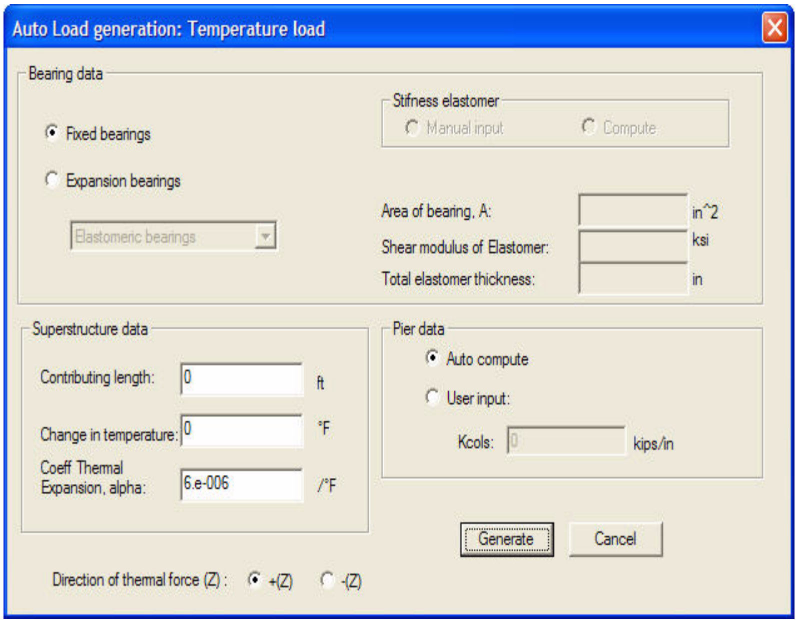
Length of Live Load: 60. ft

Figure GO-32 Auto Load Generation: Wind Load on Live Load Screen

You can have the program generate multiple wind loads on live load for different angles by selecting the Generate for multiple angles check box. Note that the default value in the Length of Live Load field is the average span length from two adjacent spans.

Auto Load Generation: Temperature Load Screen

This screen is used for auto generating temperature loads from superstructure. These loads are based on the approach listed in the Theory chapter on page TH-25. Click Generate on the Loads: Load Data dialog of type Temperature to display the screen as shown in Figure GO-33.



● Fixed bearings

○ Expansion bearings

Elastomeric bearings

○ Manual input ○ Compute

Area of bearing, A: in^2

Shear modulus of Elastomer: ksi

Total elastomer thickness: in

Contributing length: 0 ft

Change in temperature: 0 °F

Coeff Thermal Expansion, alpha: 6.e-006 /°F

● Auto compute

○ User input:

Kcols: 0 kips/in

Direction of thermal force (Z) : ● +(Z) ○ -(Z)

Figure GO-33 Auto Load Generation: Temperature Load Screen

For Auto Temperature generation click the generate button on the Load Data screen. User has to select the type of bearing on the pier. The selected bearing will enable the data required to be filled in by the user.

Fixed bearings required the super structure data and the pier data to calculate the thermal force. Pier data can either be input manually or it can be auto computed. If the user selects the auto compute then RC-PIER internally calculates the stiffness of all the columns in the pier and use for the thermal force. The user input option allows the user to control the stiffness of the columns in the pier, in this the user can change the properties of the column e.g., may used cracked section for calculating the thermal forces. Super Structure data requires the user to input the contributing length that is involved in the thermal movement, RC-PIER internally doesn't calculate the contributing length user has to input the appropriate value to control the thermal effects. The user has to consider the zero point movement on the whole and also take into consideration the construction joints to decide about the contributing length. The change in temperature could be the one provided in the code for different regions or it can be the one required by the state. Alpha is the coefficient of thermal expansion and it depends on the material of the super structure, the default value is for concrete but if the super structure is of steel, then user should change it appropriately.

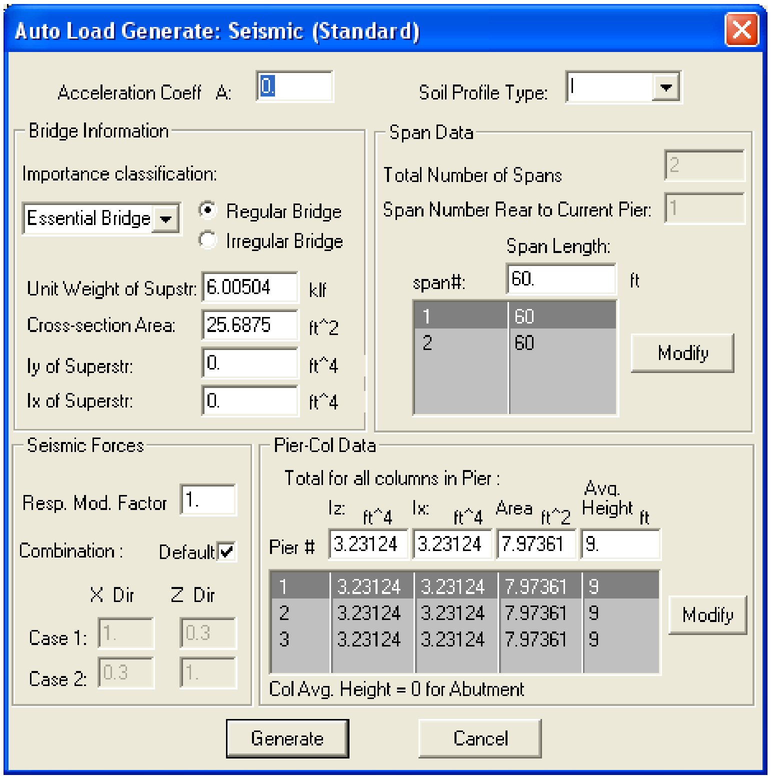
Expansion bearing requires the user to input super structure, pier and Elatomeric bearing data. The pier and super structure data is same as for fixed bearing. Elastomer data has two options for the user manual or compute. If the option for compute then user has to input the area of the bearing, the shear modulus of elastomer and the total thickness of the elastomeric bearing. These values depend upon the properties of the elastomer provided. RC-PIER only generates the forces on the bearings and will not check any bearings design. The thickness of the elastomeric bearing is without the steel plates. RC-PIER will calculate the stiffness of the bearing depending upon these properties or user can input it manually.

For sliding bearing the user has to input the friction coefficient (μ). By default it is set to 0.06, normally it ranges from 0.04 to 0.08. It depends upon the type of material used in the bearings. If the user selects the auto calculate option then RC-PIER takes the total DL applied to the pier or user can input manually.

For Rocker bearing, RC-PIER requires the radius of the Rocker pin and the rocker. The DL option is either manual or auto, in case of auto RC-PIER takes the total dead load applied to the pier.

Auto Load Generation: EQ Screen

This screen is used for auto generating earthquake loads. These loads are obtained from single-mode spectral analysis, as shown in Figure GO-34. Note that the screen shown is the same for either design code.



Acceleration Coeff A: 0.

Soil Profile Type: I

Importance classification: Essential Bridge

● Regular Bridge

○ Irregular Bridge

Unit Weight of Supstr: 6.00504 klf

Cross-section Area: 25.6875 ft^2

Iy of Superstr: 0. ft^4

Ix of Superstr: 0. ft^4

Total Number of Spans 2

Span Number Rear to Current Pier: 1

Span Length: span#: 60. ft

1	60
2	60

Resp. Mod. Factor 1.

Combination : Default ☑

	X Dir	Z Dir
Case 1:	1.	0.3
Case 2:	0.3	1.

Total for all columns in Pier :

	Iz: ft^4	Ix: ft^4	Area ft^2	Avg. Height ft
Pier #	3.23124	3.23124	7.97361	9.

1	3.23124	3.23124	7.97361	9
2	3.23124	3.23124	7.97361	9
3	3.23124	3.23124	7.97361	9

Col Avg. Height = 0 for Abutment

Figure GO-34 Auto Load Generation: Seismic (Standard) Screen

Start by entering the values in the Acceleration Coefficient (A) and Soil Profile Type text boxes. Then, input the bridge information, as follows. Select the importance classification and input the moment of inertia of superstructure about the global X- and Y-axis. The program will automatically calculate the unit weight and cross-section area of superstructure based on available superstructure data, but you may edit these values as desired.

Next, you can modify the span length under Span Data by highlighting the span #, inputting a new value in the Span Length text box, and clicking **Modify**.

Now, enter the required information under Pier-Col Data. You must input the total moment of inertia and total area for all columns, about the global X, Z directions, for each pier. For example, if a pier has three columns, Iz will be the sum of the moment of inertia about Z-axis from these three columns, or Area will be the sum of that of each column. Note that the default values for Iz, Ix, and Area are already provided in the fields under Pier-Col Data. These values are calculated on the assumption that all other piers in the bridge considered has the same number and size of columns, as those of the current pier under consideration. To modify these values, highlight the desired pier in the list, make the appropriate changes in the text boxes, and click **Modify**.

Click **Generate** when finished entering all necessary information, and the program will automatically calculate the loads and return to the *Loads: Load Data* screen.

Note: Column Avg Height = 0 at the abutment, which is only allowed at the end of the bridge. The default value of Resp. Mod Factor (Response Modification Factor) = 1; but you can modify this factor.

Loads Screen Terms

EQ Details Button. Displays the details of the last auto-generated earthquake load.

LL Details Button. Displays the details of the last auto-generated live load.

Combinations Button. Displays the details of all load combinations. This is enabled after the analysis has been performed.

Bearing Loads. Loads acting on bearing points.

Column Loads/Settlement. Loads acting on a column. Settlement refers to the movement of a column at the bottom.

Cap Loads. Loads acting on the cap.

Strain Load. Volume change of a member. It can be due to temperature, shrinkage, etc.

Multiplier for loads. A factor multiplied to the current load.

Auto Load Generation Screen Terms

Loaded Lanes. In live load generation, specifies the number of lanes to be loaded by the live load.

Minimum Spacing Between Positions. Minimum spacing between two live load positions to be considered in the analysis.

Minimum Distance From Curb. Minimum distance of the nearest truck or tandem wheel load from the curb.

Center to Center Spacing. Distance between the center of two live loads.

Wind Angle. Angle of wind acting on a bridge, measured from the normal to a bridge longitudinal direction to the wind pressure direction. Positive if anti-clockwise. (Figure TH-6)

Elevation above which Wind Load acting. Part of a column where wind load is acting. Wind Load is applied to the column from this elevation to the top of the column.

Trans. Transverse direction of wind pressure component (measured normal/perpendicular to the centerline of bridge).

Longit. Longitudinal direction of wind pressure component (measured parallel to the flow of traffic or centerline of a bridge).

Acceleration Coeff A. Acceleration coefficient for a bridge.

Soil Profile Type. Soil condition used in calculation of seismic load.

Unit Weight of Supstr., klf. Unit weight of the superstructure, in klf.

Resp. Mod. Factor. Response modification factor (R).

Cross-section Area, ft2. Cross-section area for the superstructure.

Iy of Superstr. Moment of inertia about the global Y-axis for the superstructure.

Ix of Superstr. Moment of inertia about the global X-axis for the superstructure.

Avg. Height. Average height of all columns for a pier.

Contributing Length: Length of the bridge that contributes to the calculation of longitudinal or temperature load.

Lane Load (with Rider): For manual input option on longitudinal force generation dialog, specify total lane load value to use instead of program computed value for a vehicular lane load.

Truck Load: For manual input options on longitudinal and centrifugal force generation, specify a total truck load value to use instead of program computed value for a vehicular truck load.

Radius of Curve: Radius of the curve to use in centrifugal force calculations measured in feet (or meters).

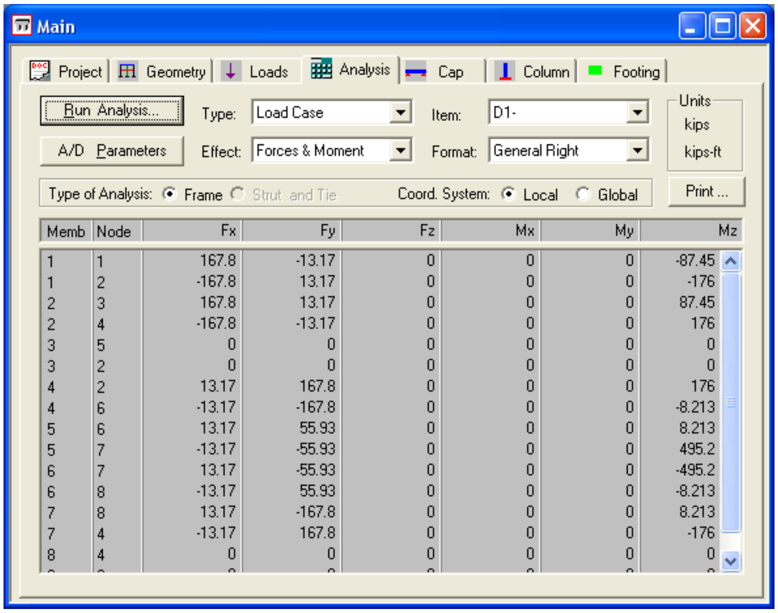
Design Speed: For use in centrifugal load generation. Measured in miles per hour (or km/hr).

Direction of Centrifugal Force (X): If the center of curve lies to the right of the bridge looking upstation, specify the direction as + (X).

Analysis Tab Screen

This is the fourth screen in the tab series. Here, you can run the analysis of the pier structure based on all the data you have entered up to this point. Note that any changes made to the data after the analysis is run will reset the analysis results.

To run an analysis, click **Run Analysis**. When the analysis is finished, the results will appear on-screen, as shown in Figure GO-35. If necessary, use the scrollbar on the right side of the screen to view results at the bottom.



Type: Load Case

Item: D1-

Units kips kips-ft

Effect: Forces & Moment

Format: General Right

Type of Analysis: ● Frame ○ Strut and Tie

Coord. System: ● Local ○ Global

Memb	Node	Fx	Fy	Fz	Mx	My	Mz
1	1	167.8	-13.17	0	0	0	-87.45
1	2	-167.8	13.17	0	0	0	-176
2	3	167.8	13.17	0	0	0	87.45
2	4	-167.8	-13.17	0	0	0	176
3	5	0	0	0	0	0	0
3	2	0	0	0	0	0	0
4	2	13.17	167.8	0	0	0	176
4	6	-13.17	-167.8	0	0	0	-8.213
5	6	13.17	55.93	0	0	0	8.213
5	7	-13.17	-55.93	0	0	0	495.2
6	7	13.17	-55.93	0	0	0	-495.2
6	8	-13.17	55.93	0	0	0	-8.213
7	8	13.17	-167.8	0	0	0	8.213
7	4	-13.17	167.8	0	0	0	-176
8	4	0	0	0	0	0	0

Figure GO-35 Analysis Tab Screen (After Analysis is Performed)

Using the Type, Effect, Item, and Format lists, you can select the specific type of results you want to view. For Type, load case results are raw data results and the envelope and combination results have already been multiplied by the appropriate g factors, impact factors, etc. Note that the analysis results for individual live loads do not include the impact factor and multi-lane factor. These factors are only included in the analysis results for load combinations and envelopes.

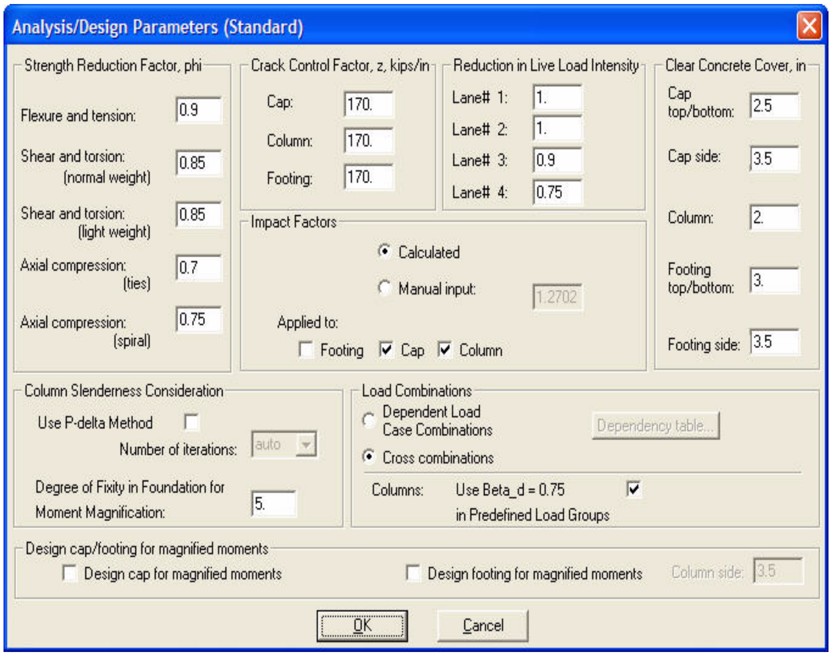
The results are based on two coordinate systems: local and global. Local coordinates are specific to individual components of the pier structure. Global coordinates are general to the entire structure.

To print the results shown on the screen, click **Print**. You can also view graphic analysis results by selecting the **Diagrams** option from the **Show** menu. (see page GO-66 for more information on this menu option). On this screen, you can choose which member you want to view from the **Objects** and **Components** lists. Either shear or moment can be selected from the **Type** list. Note that the graphic results are shown for the item selected from the Item list on the *Analysis* screen.

RC-PIER allows the option to manually modify the analysis and design parameters. To do this, click **A/D Parameters** to activate the *Analysis/Design Parameters* screen. This screen will appear differently depending on the design code selected, LFD or LRFD, as follows.

Analysis/Design Parameters (Standard) Screen

For the LFD design code, the *Analysis/Design Parameters* screen will resemble Figure GO-36. You can modify or review the strength reduction factor, crack control factor, reduction in live load intensity, clear concrete cover, and impact factors.



Strength Reduction Factor, phi
Flexure and tension: 0.9
Shear and torsion: (normal weight) 0.85
Shear and torsion: (light weight) 0.85
Axial compression: (ties) 0.7
Axial compression: (spiral) 0.75

Crack Control Factor, z, kips/in
Cap: 170.
Column: 170.
Footing: 170.

Reduction in Live Load Intensity
Lane# 1: 1.
Lane# 2: 1.
Lane# 3: 0.9
Lane# 4: 0.75

Clear Concrete Cover, in
Cap top/bottom: 2.5
Cap side: 3.5
Column: 2.
Footing top/bottom: 3.
Footing side: 3.5

Impact Factors
● Calculated
○ Manual input: 1.2702
Applied to: ☐ Footing ☑ Cap ☑ Column

Column Slenderness Consideration
Use P-delta Method ☐
Number of iterations: auto
Degree of Fixity in Foundation for Moment Magnification: 5.

Load Combinations
○ Dependent Load Case Combinations
● Cross combinations
Columns: Use Beta_d = 0.75 in Predefined Load Groups ☑

Design cap/footing for magnified moments
☐ Design cap for magnified moments
☐ Design footing for magnified moments
Column side: 3.5

Figure GO-36 Analysis/Design Parameters (Standard) Screen

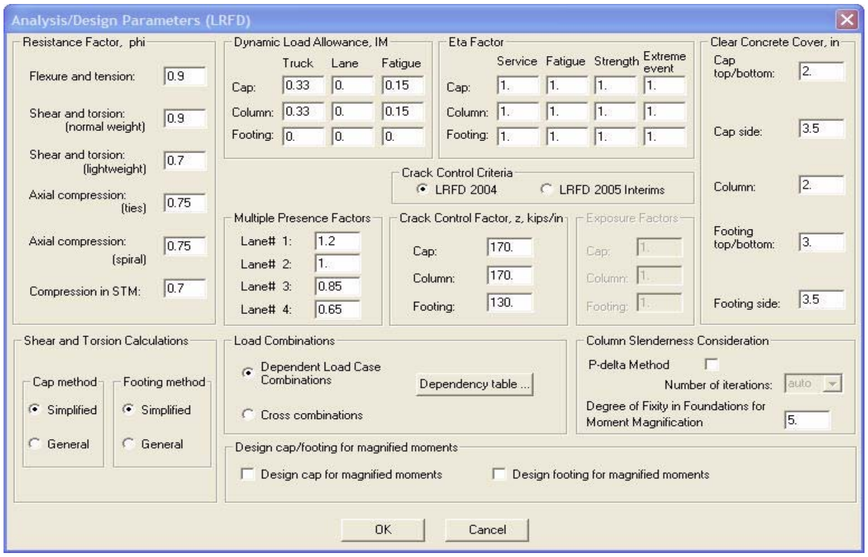
For slender columns, you can select the P-delta Method check box under Column Slenderness Consideration. You can assign up to 5 number of iterations or have RC-PIER perform iterations by selecting the Auto option from the Number of Iterations list. If you selected the Auto option, RC-PIER will stop iteration once the results are convergent or the number of iterations reaches 5. If you do not select the P-delta method, you can use the moment magnification method (on the *Column* screen, see page GO-43) to design the slender column. If you have an unbraced column and want to use the moment magnification method, specify the value for degree of fixity in foundation (under Column Slenderness Consideration). RC-PIER uses this value to calculate the effective column length factor, k. For more details about degree of fixity in foundation, see page TH-38 in the Theory chapter.

RC-PIER follows two different algorithms to generate load combinations. If the Use Cross Combinations option is selected, the program will generate comprehensive load combinations. If the Dependent Load Case combination option is selected, the program will use the concept of load dependency to generate the combinations. In this case, the number of combinations will be less than if you selected the Use Cross Combinations check box. See page TH-31 in the Theory chapter for more information. Program now allows you to view or modify the dependency. To do this, you may click the Dependency Table button. Please see page GO-38 for information on how to set/modify the dependency table.

Other two options on this dialog allows you to do the cap and footing design for the magnified column moments at the cap-column or column-footing connection. If you would like do this, check the appropriate box. However, for that case, you should first do the design of all the columns with magnified moments and then do the design of cap or footing.

Analysis/Design Parameters (LRFD) Screen

For LRFD design code, the *Analysis/Design Parameters* screen will resemble Figure GO-37. You can modify or review the resistance factor, Eta factor, multiple presence factor, dynamic load allowance, shear and torsion calculations, and clear concrete cover. You can choose to perform crack control check either as per AASHTO LRFD 2004 or as per Interims of 2005. You can also switch between the simplified and general method for shear and torsion design.



Resistance Factor, phi
Flexure and tension: 0.9
Shear and torsion: (normal weight) 0.9
Shear and torsion: (lightweight) 0.7
Axial compression: (ties) 0.75
Axial compression: (spiral) 0.75
Compression in STM: 0.7

Dynamic Load Allowance, IM

	Truck	Lane	Fatigue
Cap:	0.33	0.	0.15
Column:	0.33	0.	0.15
Footing:	0.	0.	0.

Eta Factor

	Service	Fatigue	Strength	Extreme event
Cap:	1.	1.	1.	1.
Column:	1.	1.	1.	1.
Footing:	1.	1.	1.	1.

Clear Concrete Cover, in
Cap top/bottom: 2.
Cap side: 3.5
Column: 2.
Footing top/bottom: 3.
Footing side: 3.5

Crack Control Criteria
● LRFD 2004 ○ LRFD 2005 Interims

Multiple Presence Factors
Lane# 1: 1.2
Lane# 2: 1.
Lane# 3: 0.85
Lane# 4: 0.65

Crack Control Factor, z, kips/in
Cap: 170.
Column: 170.
Footing: 130.

Exposure Factors
Cap: 1.
Column: 1.
Footing: 1.

Shear and Torsion Calculations
Cap method: ● Simplified ○ General
Footing method: ● Simplified ○ General

Load Combinations
● Dependent Load Case Combinations
○ Cross combinations

Column Slenderness Consideration
P-delta Method ☐
Number of iterations: auto
Degree of Fixity in Foundations for Moment Magnification 5.

Design cap/footing for magnified moments
☐ Design cap for magnified moments ☐ Design footing for magnified moments

Figure GO-37 Analysis/Design Parameters (LRFD) Screen

The remaining options on this screen are the same as the standard specification.

Analysis Screen Terms

Type. Select the type of results to view. Includes load case, load combination, and various envelopes.

Effect. View the results for Forces & Moment or Displacement & Rotation.

Item. The items under this menu will vary depending on the type selected and will include a selection of results for that particular type. For example, if Load Combination was selected under Type, then you can select any combination from the Item menu.

Format. Select the precision and exponentiation of result values from many available formats.

Type of Analysis. Select the type of analysis and includes two options, Frame Results and Strut-and-Tie. By default, frame results are only available; however, if a hammerhead was designed using STM (AASHTO LRFD) then both options are available.

Coordinate System. This option is available only for Moment and Forces. There are two types of coordinate systems available: Local coordinates (specific to member) or Global coordinates (general to entire structure). This option is not available if you selected Displ. and Rotations from the Effect list.

Run Analysis Button. Starts the analysis. This will launch the solver engine and you will see a progress bar showing you the progress of the analysis. After the analysis is performed, the results will appear on-screen. To print the analysis results, click the right mouse button.

A/D Parameters Button. Brings up the *Analysis/Design Parameters* screen. The screen will appear differently depending on the design code selected (LFD or LRFD). Here, you can modify or review the analysis and design parameters.

Print Button. Opens a dialog box that gives you the option to either send the displayed analysis results to a Microsoft Excel file or to a printer.

Analysis/Design Parameters (Standard) Screen Terms

Strength Reduction Factor, phi. Strength reduction factors may be entered as per AASHTO LFD Art. 8.16.1.2.2 or as suggested by other governing agencies.

Crack Control: LRFD 2004. Choose this option to do crack control check as per LRFD 2004 3rd Edition which is based on Z factor.

Crack Control: LRFD 2005 Interims. Choose this option to check crack control as per the new approach specified in AASHTO LRFD 2005 interims using exposure factor.

Crack Control Factor, z, kips/in. Crack control coefficient specified in AASHTO LFD Art. 8.16.8.4.

Exposure Factor: Specify the exposure factor as per AASHTO LRFD 2005 interims Article 5.7.3.4.

Clear Concrete Cover, in. The clear cover to the stirrup from the exterior face for various components of the pier.

Reduction in Live Load Intensity: Lane#. Reduction in live load factors. Default values are as specified in AASHTO LFD Art. 3.12.1 but you can modify them.

Impact Factors. RC-PIER gives you two options for the impact factor: calculated or manual. If you select the Calculated option, RC-PIER will compute the impact factor as specified in AASHTO LFD Art. 3.8.2. If you select the Manual option, you must input the value for the impact factor in the appropriate field. Both of these options can be applied to results for any of the pier components, i.e., cap, column, and footing.

Column Slenderness Consideration. This option allows you to select what kind of analysis you want performed for slender members. Either select the p-delta method or manually input the fixity coefficient for moment magnification (for unbraced columns). See page TH-37 in the Theory chapter for more information on the moment magnification method.

Load Combinations. If cross combinations option is selected, program will generate all possible combinations for included load cases and load group. If the Dependent Load combinations option is selected, the program will generate load combinations based on the concept of load dependency. See page TH-31 in the Theory chapter for more information. See page GO-38 for information on load dependency dialog.

Columns. If selected and per AASHTO Table 3.22.1A, beta_d factor = 0.75 will be used when checking member for minimum axial load and maximum moment or maximum eccentricity for column design.

Design Cap for Magnified Moments: Select this option if you would like to design cap for magnified moments of columns at a cap-column joint.

Design Footing for Magnified Moments: Select this option if you would like to design footing for magnified moments at the base of column.

Analysis/Design Parameters (LRFD) Screen Terms

Resistance Factor, phi. Resistance factors may be entered as per AASHTO LRFD Art. 5.5.4.2.1 or as suggested by other governing agencies.

Crack Control Factor, z, kips/in. Crack control coefficient specified in AASHTO LRFD Art. 5.7.3.4.

Clear Concrete Cover, in. The clear cover to the stirrup from the exterior face for various components of the pier.

Eta Factor. Load multiplier. See AASHTO LRFD Art. 3.4.1.

Multiple Presence Factor. Multiple presence factor for live load. The default values are as specified in AASHTO LRFD Art. 3.6.1.1.2, but you can modify them as desired.

Dynamic Load Allowance, IM. AASHTO LRFD requires different dynamic load allowance factors to be applied to truck, lane, and fatigue loading. The default values are as specified in AASHTO LRFD Art. 3.6.2.1, but you may modify them.

Shear and Torsion. There are two methods available to perform shear and torsion calculations, simplified or general. If the simplified method is selected, the program performs the shear and torsion calculations with β and θ, as specified in AASHTO LRFD Art. 5.8.3.4.1. For the general method, the program uses the general procedure, as specified in AASHTO LRFD, Art. 5.8.3.4.2, to compute these parameters. The program allows separate options for cap and footing shear calculations.

Column Slenderness Consideration. This option allows you to select what kind of analysis you want performed for slender members. Either select the p-delta method or manually input the fixity coefficient for moment magnification (for unbraced columns). See page TH-37 in the Theory chapter for more information on the moment magnification method.

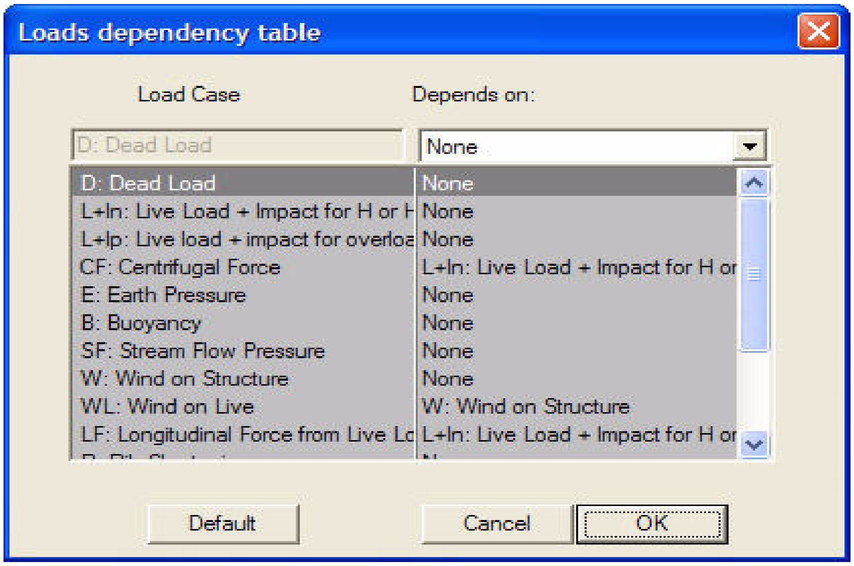
Load Combinations. If cross combinations option is selected, program will generate all possible combinations for included load cases and load group. If the Dependent Load combinations option is selected, the program will generate load combinations based on the concept of load dependency. If selected, cross combinations will be used. See page TH-31 in the Theory chapter for more information. See page GO-39 for information on load dependency dialog.

Design Cap for magnified moments: Select this option if you would like to design cap for magnified moments using the same magnification as the columns at a cap-column joint.

Design Footing for Magnified Moments: Select this option if you would like to design footing for magnified moments at the base of column.

Load Dependency Table Screen

The load dependency table dialog resembles Figure GO-38. On this dialog, you can modify the dependency between various loads.



Load Case: D: Dead Load

Depends on: None

D: Dead Load	None
L+In: Live Load + Impact for H or H	None
L+Ip: Live load + impact for overloa	None
CF: Centrifugal Force	L+In: Live Load + Impact for H or
E: Earth Pressure	None
B: Buoyancy	None
SF: Stream Flow Pressure	None
W: Wind on Structure	None
WL: Wind on Live	W: Wind on Structure
LF: Longitudinal Force from Live Lo	L+In: Live Load + Impact for H or

Figure GO-38 Load Dependency Table Screen

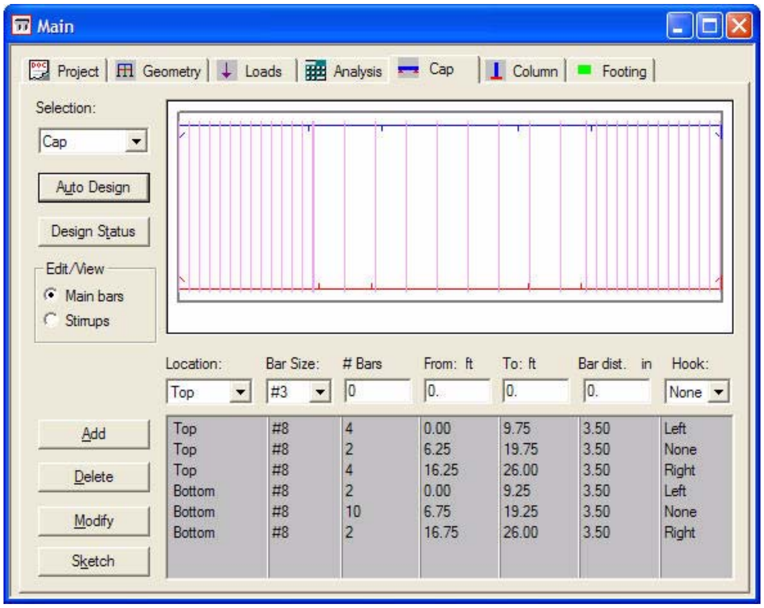
On this screen, program will show current dependency relationship between loads. To change the dependency of a load, click the load case for which you want to modify its dependency. That will then be shown in two boxes at the top of the list. Simply select the load case on which this load depends on.

To remove a dependency, change the depends on to be "None". Click OK to accept all changes or click Cancel to ignore all change. If you click Default, all the dependencies will be reset to a

predefined dependency as explained in the Theory chapter on page TH-24.

Cap Tab Screen

This is the fifth screen in the tab series, as shown in Figure GO-39. You can input the reinforcement of the pier cap manually or use the auto design feature.



Selection: Cap

● Main bars
○ Stirrups

Location:	Bar Size:	# Bars	From: ft	To: ft	Bar dist. in	Hook:
Top	#3	0	0.	0.	0.	None
Top	#8	4	0.00	9.75	3.50	Left
Top	#8	2	6.25	19.75	3.50	None
Top	#8	4	16.25	26.00	3.50	Right
Bottom	#8	2	0.00	9.25	3.50	Left
Bottom	#8	10	6.75	19.25	3.50	None
Bottom	#8	2	16.75	26.00	3.50	Right

Figure GO-39 Cap Tab Screen Showing Main Bars

To begin, select the part of the structure you want to design from the Selection list, e.g., cap or desired strut. Then, input the reinforcement manually or use the Auto Design feature, as follows.

Clicking **Auto Design** will have the program automatically design the selected component of the structure. You must select the main bar size, stirrup size, number of legs and stirrup increment after clicking this button. Auto design is based on the structural requirement, not the serviceability requirement. Also, a maximum of three layers of main rebars, either at the top or bottom of cap, are designed using this feature.

Cap Reinforcement

Main bar size:	Stirrups size:	Nr. of legs:	Stirrup Increment,: in
#8	#4	4	6.

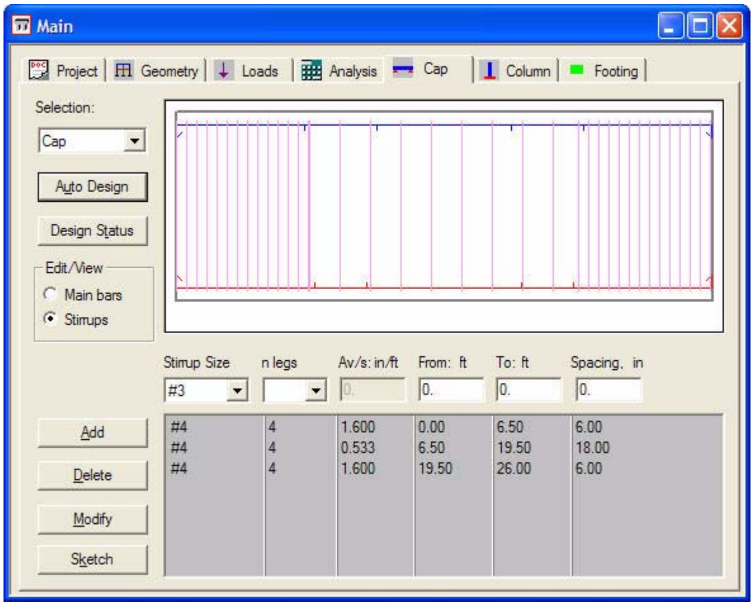
Figure GO-40 Auto Design

You can switch to view either the main bars or the stirrups.

To manually input the main bars, switch to **Edit** main bars and then enter the values in the text boxes at the bottom of the screen (below the diagram) and click **Add**. Note that you must select the bar size from the list. You can modify existing reinforcement in the list by highlighting it, making the appropriate changes, and clicking **Modify**. To delete an existing reinforcement, highlight it and click **Delete**.

Note that the location of all the bars is measured from the left end of the cap and the hook information is used to compute the development length of the bars

To manually input stirrups, first switch to **Edit/View** stirrups by selecting the stirrups option on this tab. The view will change to show the stirrup data as shown in Figure GO-41. Then specify the starting location and ending location of the stirrups, their size, spacing and number of legs and click **Add.** The stirrup set will get added and will be displayed on the sketch. To add additional sets, repeat the same. To modify any existing set, select that set, make changes to it and then click **Modify**. To delete an existing set, highlight the row and click **Delete.**



Selection: Cap

Edit/View
○ Main bars
● Stirrups

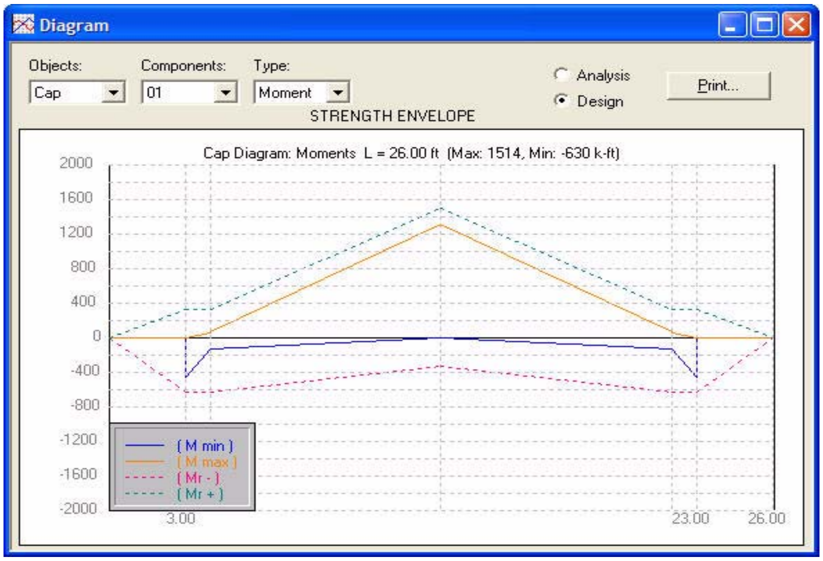
Stirrup Size	n legs	Av/s: in/ft	From: ft	To: ft	Spacing, in
#3		0.	0.	0.	0.
#4	4	1.600	0.00	6.50	6.00
#4	4	0.533	6.50	19.50	18.00
#4	4	1.600	19.50	26.00	6.00

Figure GO-41 Cap Tab Showing Stirrup Data

Note that From and To distances must be specified such that those do not overlap for different sets of stirrups.

Once the reinforcement is entered, an updated view of the component will be displayed in the top portion of the screen. You can click **Design Status** to display a design summary of the selected component, which can be printed out. Clicking **Sketch** will display a graphical representation with section details of the selected structure.

You can also view the design results by selecting the **Diagrams** option from the **Show** menu (or selecting its respective icon on the toolbar). This activates the *Diagrams* screen. Select the Design option to display the graphic design results, as shown in Figure GO-42. Note the graphic design results are only available after you complete the design. See page GO-66 for more information on the **Show | Diagrams** option.



Objects: Cap
Components: 01
Type: Moment
○ Analysis
● Design

Figure GO-42 Diagram Screen - Showing Graphic Design Results

Cap Screen Terms

Selection. Select the component you want to design, i.e., the cap or strut.

Auto Design Button. Makes the program automatically design the selected component of the structure. You must select the main bar size, stirrup size, number of legs and stirrup increment.

Design Status Button. Displays the *Design Status* screen, showing the design status summary of the selected component.

Edit/View: Main Bars. Select this to view or edit main bars (flexural bars) in the component.

Add Button. Manually input the reinforcement.

Modify Button. Modify any existing reinforcement in the list.

Delete Button. Delete previously defined reinforcement in the list.

Sketch Button. Displays a graphical representation of the reinforcement in elevation and section for the selected component.

Location. Specifies the location of a main bar. It can be either a top bar or bottom bar.

Bar Size. Bar Size refers to all U.S. customary bars. However, when SI units are used, they are designated in metric units (e.g., #8 U.S. bars is shown in #25 metric bar).

 # Bars. Number of main bars per layer.

From, ft. Starting location of the bars measured from left end of the cap. A zero is left end.

To, ft. Ending location of the bars measured from left end of cap.

Bar dist, in. Distance to the center of the main bars in that layer. For top bars, it is the distance from the top surface to the center of the bar. For bottom bars, it is the distance from the bottom surface to the center of the bar.

Hook. Specify if the bar is hooked at the left end, right end, both ends, or neither end (i.e., None).

S. Size. Select the size of the stirrup ties.

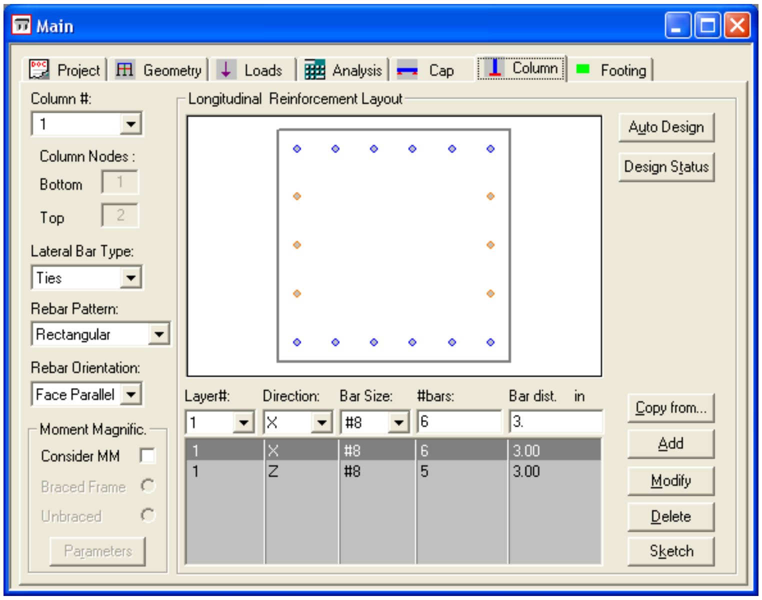
n legs: Specify the number of legs for the stirrups. Minimum of 2 legs and maximum of 10 legs are allowed.

Spacing: Specify center to center spacing of stirrups for this set.

Av/s: This is a read only column and program computes and displays this based on stirrup size, spacing and number of legs.

Column Tab Screen

From the Column Tab, as shown in Figure GO-43, you can specify the reinforcements for the different columns of the pier structure.



Column #: 1

Column Nodes :
Bottom 1
Top 2

Lateral Bar Type: Ties

Rebar Pattern: Rectangular

Rebar Orientation: Face Parallel

Consider MM ☐
Braced Frame ○
Unbraced ○

Layer#:	Direction:	Bar Size:	#bars:	Bar dist. in
1	X	#8	6	3.
1	X	#8	6	3.00
1	Z	#8	5	3.00

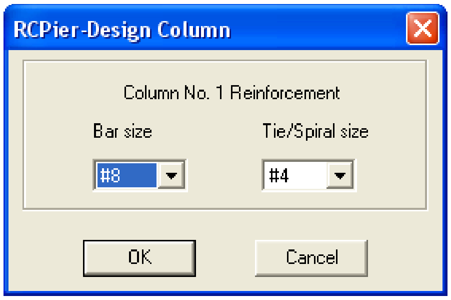
Figure GO-43 Column Tab Screen

To start, select the column of the structure you want to design from the Columns # list and the type of stirrup (ties or spiral) from the Lateral Bar Type list. Under Moment Magnific, if you want to use the moment magnification method (MM) for the selected column, select the Consider MM check box. If selected, specify if the column is braced or unbraced. RC-PIER does not determine if a column is braced or unbraced.

Now, you can either manually input the reinforcement for the selected column or click **Auto Design** to have the program automatically design the selected column.

Click **Auto Design** to open the *Design Column* screen, as shown in Figure GO-44. Select the bar size and tie/spiral size from the lists and click **OK**. RC-PIER will automatically design the selected column and display the design status before returning to the *Column* screen. Then, the required reinforcement will appear in the list at the bottom of the screen.

When Auto Design is done RC-PIER resets the Rebar Pattern to rectangular for all column shapes except circular and octagonal columns. For circular and octagonal columns, the pattern is reset to circular. Rebar orientation is reset to face parallel.



Bar size: #8

Tie/Spiral size: #4

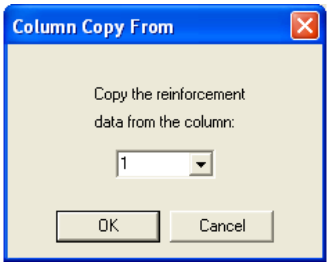
Figure GO-44 Design Column Screen

To manually input the reinforcement, first select the rebar pattern. Depending upon the column shape, RC-PIER allows rectangular, circular, intersecting hoops and general rebar pattern. (For more information about these patterns, refer to "Column Rebar Patterns" on page TH-41.)

If the rebar pattern is rectangular or circular select the rebar orientation. RC-PIER allows bars to be either vertical or face parallel. For face parallel bars, the cover remains constant at all locations. Intersecting hoops and general pattern allow only vertical bars.

After selecting rebar pattern and orientation, enter the values in the text boxes at the bottom of the screen and click **Add**. You can modify existing reinforcement in the list by highlighting it, making the appropriate changes, and clicking **Modify**. To delete an existing reinforcement, highlight it and click **Delete**. A maximum of two layers of reinforcement steel are allowed in the column design for circular or rectangular patterns.

A valuable timesaving feature available is **Copy From**. Clicking this button will copy the reinforcement from another column to the selected column. Simply, click **Copy From** to activate the *Column Copy From* screen, as shown in Figure GO-45.



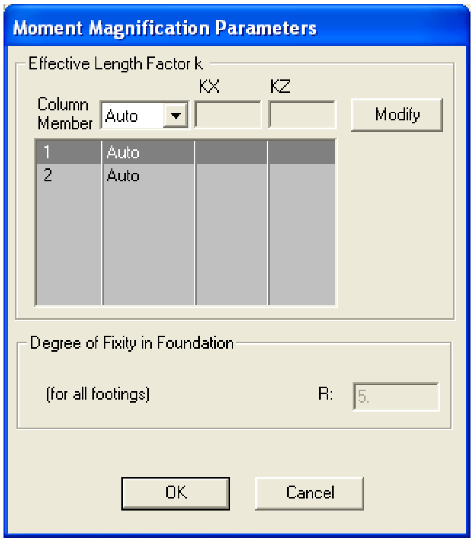
Copy the reinforcement data from the column: 1

Figure GO-45 Column Copy From Screen

Select the column you would like to copy the reinforcement from and click **OK**. It will appear in the list on the *Column* screen.

Once the reinforcement is entered, an updated view of the component will be displayed in the top portion of the screen. You can click **Design Status** to display a design summary of the selected component. To print the summary, click **Print**. Clicking **Sketch** will display a graphical representation of the selected columns with reinforcements.

Clicking **Parameters** will activate the *Moment Magnification Parameters* screen, as shown in Figure GO-46. This button is only enabled when you select the Consider MM check box. If you do not want the program to calculate the effective length factor (k), then you can manually input the value for k in the X and Z-directions.



Column Member: Auto

1	Auto		
2	Auto		

(for all footings) R: 5.

Figure GO-46 Moment Magnification Parameters Screen

Column Screen Terms

Column #. Column selected for design.

Lateral Bar Type. Either ties or spiral.

Consider MM. Selecting this check box indicates that you want to use the moment magnification method for the selected column.

Degree of Fixity in Foundation. Used to calculate the effective length factor, k. For more information see page TH-38 in the Theory chapter.

Bar dist. Location from the face of a column to the center of reinforcement.

Auto Design Button. The program to automatically designs the selected reinforcement of the column.

Design Status Button. Brings up the *Design Status* screen showing the design status summary of the selected column.

Copy From Button. Brings up the *Column Copy From* screen, where you can copy the reinforcement from one of the other columns to the selected column.

Sketch Button. Brings up a screen showing a graphical representation of the selected column with reinforcement.

Footing Tab Screen

This is the final screen in the tab series, as shown in Figure GO-47 and is a unique feature available only in RC-PIER. Here, you can design and analyze multiple footings or footing types for each column, to include single spread footing under all columns, isolated spread footings under each column, single pile/shaft cap under all columns, and isolated pile/shaft cap under each column You can also check overturning from this tab. For more details about that, refer to Overturning Check Screen on page GO-56

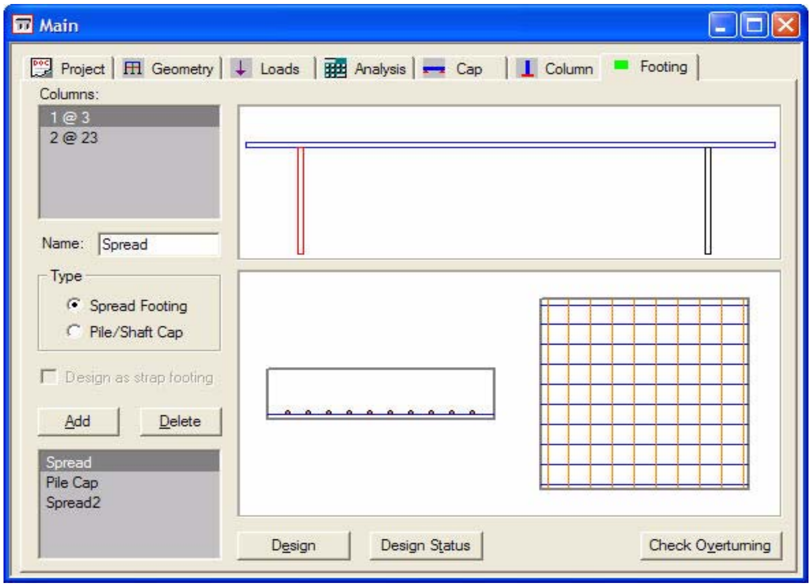


Figure GO-47 Footing Tab Screen (Showing Spread Footing Design)

First input the footing specifications for the pier structure. Each footing section must have a name and type selected before it can be designed. From the Columns list, highlight the column(s) you want to add a footing (or select the column(s) in the graphical display at the top of the screen). The selected column(s) will turn red. Next, input a name for the footing in the Name text box and select the type of footing (spread or pile/shaft cap) under Type. Click **Add**. The name will appear in the list on the screen.

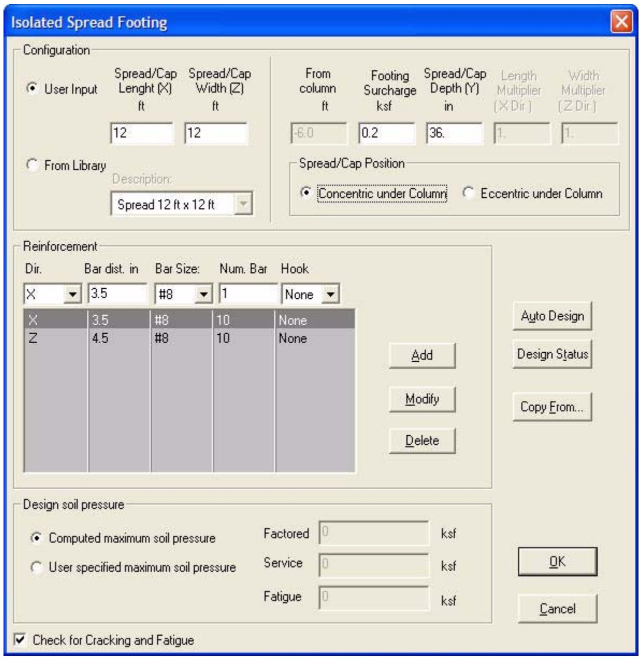
Note that you can analyze additional footings by inputting another name in the Name text box and clicking **Add**. This is usually done at the preliminary design stage when alternative footings need to be investigated.

Once the footing is named, highlight it in the list and click **Design**. The *Footing Design* screen will display. You can specify the size and reinforcement of the selected footing either manually or allow the program to automatically design the selected footing. Note this screen will appear differently depending on the footing type selected.

Before you start the footing design, you must define the footing geometry by selecting the **Footing Configuration** option from the **Libraries** menu. See page GO-70 for more information on this option. Any previously defined footings will be available in the Description list described in the next section.

Isolated Spread Footing Screen

If you selected one column and the spread footing option, the *Isolated Spread Footing* screen will display, as shown in Figure GO-48.



Configuration

● User Input — Spread/Cap Lenght (X) ft: 12 — Spread/Cap Width (Z) ft: 12

From column ft: -6.0 — Footing Surcharge ksf: 0.2 — Spread/Cap Depth (Y) in: 36. — Length Multiplier (X Dir.): 1. — Width Multiplier (Z Dir.): 1.

○ From Library — Description: Spread 12 ft x 12 ft

Spread/Cap Position: ● Concentric under Column ○ Eccentric under Column

Reinforcement

Dir.	Bar dist. in	Bar Size:	Num. Bar	Hook
X	3.5	#8	1	None
X	3.5	#8	10	None
Z	4.5	#8	10	None

Design soil pressure

● Computed maximum soil pressure
○ User specified maximum soil pressure
Factored: 0 ksf
Service: 0 ksf
Fatigue: 0 ksf

☑ Check for Cracking and Fatigue

Figure GO-48 Isolated Spread Footing Screen

First, input the configuration information You can specify the footing size in one of two ways. First option is to select User Input and then specify length (X) and width (Z). Second option is to select the From Library and then choose one of the footings from the Description list (which you defined under the **Libraries | Footing Configuration** option). If the footing is symmetrically placed (concentric) under the column (which is the most common case) select concentric under column option. However, if the footing is not concentric under columns, select Eccentric under column option. In this case, input the values in the text boxes where the From Column field is the distance between the left edge of the footing to the center of the column. This distance must be negative. For example, if you have a 10x10 ft square footing and the column is at the center of the footing, then the value in the From Column field would be -5 ft. Note that, RC-PIER only allows the footing to have eccentricity with respect to the column centerline in the global X-direction.

Spread/Cap Depth is the depth of the bottom of footing. Length and width multipliers can be specified only when footing From Library option is selected. These are the factors that will be multiplied to the original footing length and width defined under the **Libraries | Footing Configuration** option. For example, if you have a 10x10 square footing in Library and input 2 and 3 for the length and width multipliers, the size of the footing used in the design will be 20x30 ft. Footing surcharge is the uniform pressure acting downwards on the top surface of the footing. No negative value is allowed as the footing surcharge. Note that the footing surcharge is applied everywhere on the footing surface, including the area occupied by the column.

For the structural design of the footing, program provides two options for Design Soil Pressure. First option which is the default option, computes the maximum soil pressure from the controlling load combinations and uses in design calculations. Second option allows user to specify the soil pressures manually. When this option is selected, program carries out all the design calculations (flexure, cracking, fatigue, one way and two way shear) using the user specified values.

Program can check cracking and fatigue for load factored or LRFD designs if service and fatigue load groups are also included in analysis. Program will do this only if check for Cracking box is selected on this dialog. If this box is not selected, program will not do the check for cracking and fatigue in footing.

Next, input the reinforcement of the selected footing, either manually or allow the program to automatically generate it. To have the program generate the reinforcement, click **Auto Design** to activate the *Design Footing* screen, as shown in Figure GO-49. Select the bar size from the list and click **OK**. RC-PIER will generate the footing reinforcement and immediately displays the *Design Status* screen. Click **Close**, or the **X** in the top right corner of the screen, to exit this screen and return to the *Isolated Spread Footing* screen. The reinforcement will appear in the list on the screen. You can click **Design Status** to display the design status of the selected footing.

Bar size: #8

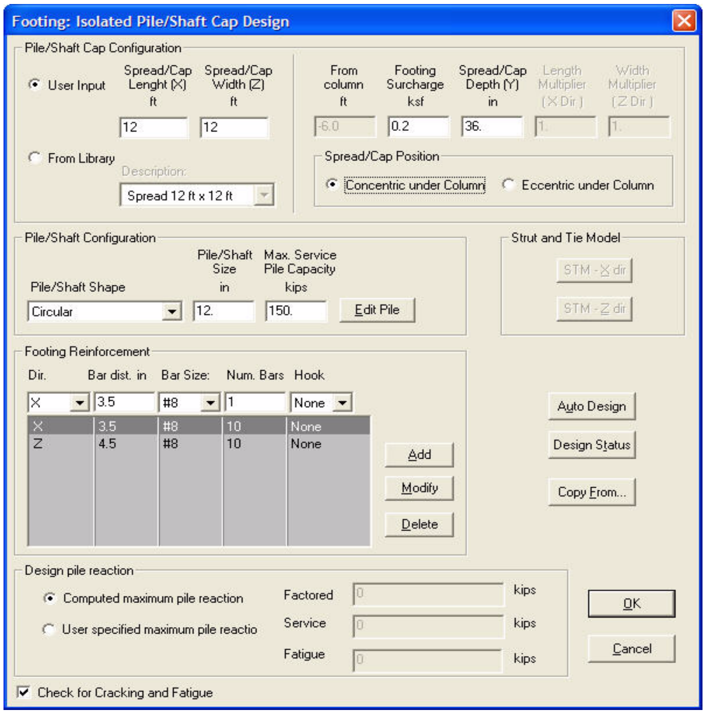
Figure GO-49 Design Footing Screen

To manually add reinforcement, input the data in the text boxes under Reinforcement for Direction, Bar distance, Bar Size, Num Bar, and Hook. Click **Add** to add the reinforcement to the list.

You can easily modify existing reinforcement by highlighting it the list, making the appropriate changes, and clicking **Modify**. To delete existing reinforcement, highlight it and click **Delete**.

Footing: Isolated Pile/Shaft Cap Design Screen

If you select one column and the pile/shaft option to open the *Footing: Isolated Pile/Shaft Cap Design* screen, as shown in Figure GO-50 will be shown. Begin by inputting the information for footing size You can specify the footing size in one of two ways. First option is to select User Input and then specify length (X) and width (Z). Second option is to select the From Library and then choose one of the footings. Select the footing in the Description list and input the values in the text boxes. If the footing is symmetrically placed (concentric) under the column (which is the most common case) select concentric under column option. However, if the footing is not concentric under columns, select Eccentric under column option. In this case, specify the From Column distance.



Footing: Isolated Pile/Shaft Cap Design

Pile/Shaft Cap Configuration

● User Input — Spread/Cap Lenght (X) ft: 12 — Spread/Cap Width (Z) ft: 12

○ From Library — Description: Spread 12 ft x 12 ft

From column ft: -6.0 — Footing Surcharge ksf: 0.2 — Spread/Cap Depth (Y) in: 36. — Length Multiplier (X Dir): 1. — Width Multiplier (Z Dir): 1.

Spread/Cap Position: ● Concentric under Column ○ Eccentric under Column

Pile/Shaft Configuration

Pile/Shaft Shape: Circular — Pile/Shaft Size in: 12. — Max. Service Pile Capacity kips: 150.

Footing Reinforcement

Dir.	Bar dist. in	Bar Size:	Num. Bars	Hook
X	3.5	#8	1	None
X	3.5	#8	10	None
Z	4.5	#8	10	None

Design pile reaction

● Computed maximum pile reaction

○ User specified maximum pile reactio

Factored: 0 kips

Service: 0 kips

Fatigue: 0 kips

☑ Check for Cracking and Fatigue

Figure GO-50 Footing: Isolated Pile/Shaft Cap Design Screen

Next, input the configuration for the pile/shaft. Note that this section is only available for pile footings. Select the shape of the pile/shaft from the Pile/Shaft Shape list, and input the values for the pile/shaft size and maximum pile capacity Click **Edit Pile** to edit the pile locations. For information on how to specify the pile pattern, see page GO-51.

For the structural design of a pile cap, program provides two options for Design Pile Reaction. First option which is the default option, computes the maximum pile reaction from the controlling load combinations and uses in design calculations. Second option allows user to specify the Pile reaction manually. When this option is selected, program carries out all the design calculations (flexure, cracking, fatigue, one way and two way shear) using the specified values.

Program can check cracking and fatigue for load factored or LRFD designs if service and fatigue load groups are also included in analysis. Program will do this only if check for Cracking box is selected on this dialog. If this box is not selected, program will not do the check for cracking and fatigue in footing.

Now, input the reinforcement for the selected footing. You can either input the values manually or allow the program to automatically generate the reinforcement for the selected footing. The procedure is the same as the *Isolated Spread Footing* screen.

If you design a new footing, you can copy the configuration of an existing footing to the new footing by clicking **Copy From**. If it is a pile cap footing, the pile locations will also be copied to the new footing.

Note: Eccentric columns can only be entered in the X-direction. All columns are centered on the footing in the Z-direction.

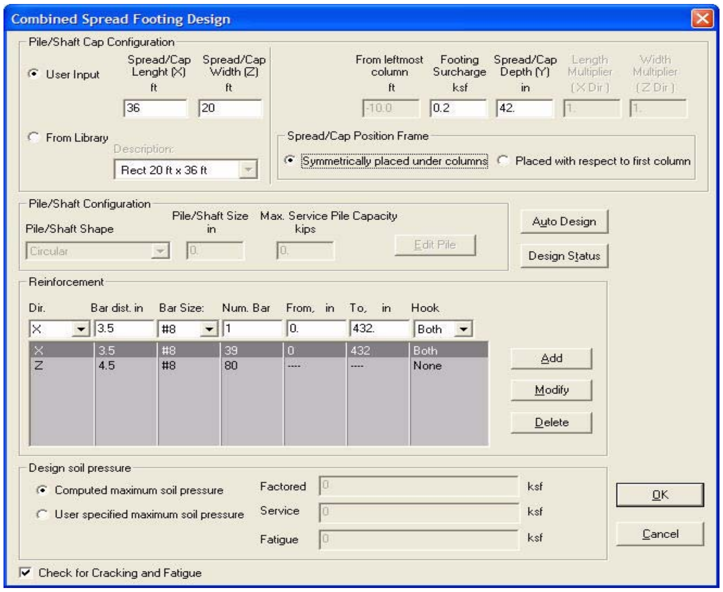
Combined Footing Design Screens

One unique feature available in RC-PIER is Combined Footing Design. This feature allows the ability to design and analyze a footing under multiple columns. It includes three options: Combined Rectangular Footing, Combined Trapezoid Footing, and Strap Footing. Note that strap footing is only available for a footing covered by two columns.

Before you can design a combined footing, you must select more than one column under Columns on the *Footing* screen (Figure GO-47). If the combined spread or Combined pile footing is a rectangular or trapezoidal footing, you may specify that directly on the footing screen. However, if the footing is a strap, you must define the footing geometry before any design can take place (using the **Libraries | Footing Configuration** option).

Combined Spread Footing Design Screen

If you selected more than one column and the spread footing type, the *Combined Spread Footing Design* screen will display, as shown in Figure GO-51.



Pile/Shaft Cap Configuration

● User Input
Spread/Cap Lenght (X) ft: 36
Spread/Cap Width (Z) ft: 20
○ From Library
Description: Rect 20 ft x 36 ft
From leftmost column ft: -10.0
Footing Surcharge ksf: 0.2
Spread/Cap Depth (Y) in: 42.
Length Multiplier (X Dir): 1.
Width Multiplier (Z Dir): 1.

Spread/Cap Position Frame
● Symmetrically placed under columns ○ Placed with respect to first column

Pile/Shaft Configuration
Pile/Shaft Shape: Circular
Pile/Shaft Size in: 0.
Max. Service Pile Capacity kips: 0.

Reinforcement

Dir.	Bar dist. in	Bar Size:	Num. Bar	From, in	To, in	Hook
X	3.5	#8	1	0.	432.	Both
X	3.5	#8	39	0	432	Both
Z	4.5	#8	80	----	----	None

Design soil pressure
● Computed maximum soil pressure
○ User specified maximum soil pressure
Factored: 0 ksf
Service: 0 ksf
Fatigue: 0 ksf

☑ Check for Cracking and Fatigue

Figure GO-51 Combined Spread Footing Design Screen

Begin by inputting the pile/shaft cap configuration information. You can specify the footing size in one of two ways. First option is to select User Input and then specify length (X) and width (Z). Second option is to select the From Library and then choose one of the footings. The Description list (previously defined using the **Libraries | Footing Configuration** option). If the footing is symmetrically placed (concentric) under the column which is the most common case, select that option. However, if the footing is not symmetrically placed under columns, select placed with respect to first column option. In this case, input the values in the text boxes where From Leftmost Column is the distance between the left edge of the footing to the center of the leftmost column. Spread/Cap Depth is the depth of the footing, in global Y-direction, measured from bottom of footing. Length and width multipliers can be specified only when footing From Library option is selected. These are the factors that will be multiplied to the original footing length and width, defined under the **Libraries | Footing Configuration** option.

Note: For trapezoidal combined footing, you must first define a trapezoidal footing under the **Libraries | Footing Configuration** option. Then select that same footing in the Description list on the *Combined Spread Footing Design* screen. RC-PIER will then generate a trapezoidal footing.

For the structural design of the footing, program provides two options for Design Soil Pressure. First option which is the default option, computes the maximum soil pressure from the controlling load combinations and uses in design calculations. Second option allows user to specify the soil pressures manually. When this option is selected, program carries out all the design calculations (flexure, cracking, fatigue, one way and two way shear) using the specified values.

Program can check cracking and fatigue for load factored or LRFD designs if service and fatigue load groups are also included in analysis. Program will do this only if check for Cracking box is selected on this dialog. If this box is not selected, program will not do the check for cracking and fatigue in footing.

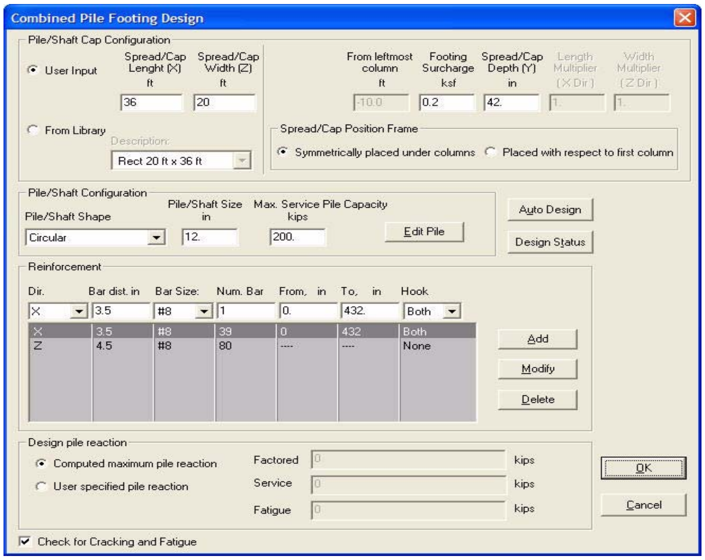
Next, input the reinforcement for the selected footing. The procedure is similar to the *Isolated Spread Footing* screen, as follows. To automatically generate the footing reinforcement, click **Auto Design** to activate the *Design Footing* screen. Select the bar size and stirrup size from the list and click **OK**. The program will automatically generate the footing reinforcement and immediately display the *Footing Design Results* screen. To close this screen and return to the *Combined Spread Footing Design* screen, click **Close** or the **X** in the top right corner of the screen. The reinforcement will appear in the list under Reinforcement.

To manually input the reinforcement, input the values in the text boxes under Reinforcement and click **Add**. You can enter bars that extend to only a portion of the footing by inputting the starting bar location in the From text box and ending location in the To text box. To modify existing reinforcement, highlight it in the list, make the appropriate changes, and click **Modify**. You can delete existing reinforcement by highlighting it in the list and clicking **Delete**.

Click **Design Status** to display the *Footing Design Results* screen. Here you can see a design summary of the selected footing. To print the summary, click **Print**. To exit this screen, click **Close** (or the **X** in the top right corner of the screen).

Combined Pile Footing Design Screen

If you select more than one column and the pile/shaft cap type, the *Combined Pile Footing Design* screen will display, as shown in Figure GO-52.



● User Input

Spread/Cap Lenght (X) ft: 36

Spread/Cap Width (Z) ft: 20

○ From Library

Description: Rect 20 ft x 36 ft

From leftmost column ft: -10.0

Footing Surcharge ksf: 0.2

Spread/Cap Depth (Y) in: 42.

Length Multiplier (X Dir): 1.

Width Multiplier (Z Dir): 1.

● Symmetrically placed under columns ○ Placed with respect to first column

Pile/Shaft Shape: Circular

Pile/Shaft Size in: 12.

Max. Service Pile Capacity kips: 200.

Dir.	Bar dist. in	Bar Size:	Num. Bar	From, in	To, in	Hook
X	3.5	#8	1	0.	432.	Both
X	3.5	#8	39	0	432	Both
Z	4.5	#8	80	----	----	None

● Computed maximum pile reaction

○ User specified pile reaction

Factored: 0 kips

Service: 0 kips

Fatigue: 0 kips

☑ Check for Cracking and Fatigue

Figure GO-52 Combined Pile Footing Design Screen

Data input is similar to the *Isolated Pile/Shaft Cap Footing Design* screen, as follows.

Start by inputting the footing size. You can specify the footing size in one of two ways. First option is to select User Input and then specify length (X) and width (Z). Second option is to select the From Library and then choose one of the footings in the Description drop-down list (previously defined under the **Libraries | Footing Configuration** option) and inputting the values in the text boxes.

Next, input the pile/shaft configuration information. Note that this section is only available for pile footings; that is, if you selected the pile/shaft footing type on the *Footing* screen. Select the shape of the pile/shaft from the Pile/Shaft Shape list, and input the values for the pile/shaft size and maximum pile capacity. Then, click **Edit Pile** to activate the *Edit: Pile Locations* screen. Specify the piles. For more information about this screen, refer to page GO-51 later in this chapter.

For the structural design of a pile cap, program provides two options for Design Pile Reaction. First option which is the default option, computes the maximum pile reaction from the controlling load combinations and uses in design calculations. Second option allows user to specify the Pile reaction manually. When this option is selected, program carries out all the design calculations (flexure, cracking, fatigue, one way and two way shear) using the specified values.

Program can check cracking and fatigue checks for load factored or LRFD designs if service and fatigue load groups are also included in analysis. Program will do this only if check for Cracking box is selected on this dialog. If this box is not selected, program will not do the check for cracking and fatigue in footing.

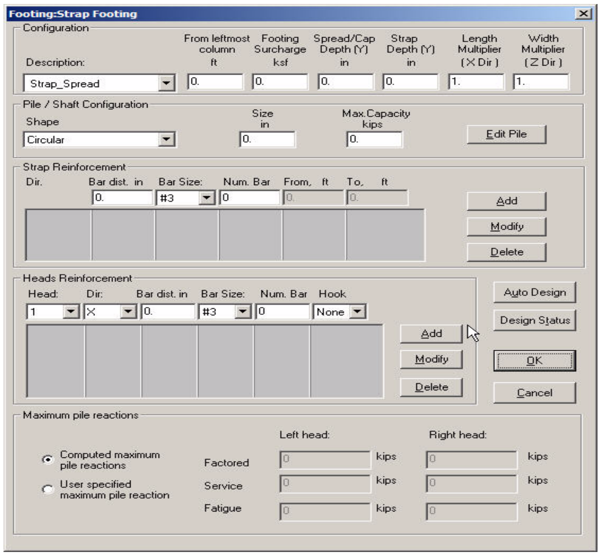
Now input the reinforcement for the selected footing either manually or using the auto design feature. To automatically generate the footing reinforcement, click **Auto Design** to activate the *Design Footing* screen. Select the bar size and stirrup size from the list and click **OK**. The program will automatically generate the footing reinforcement and immediately display the *Footing Design Results* screen. Click **Close** to exit this screen and return to the *Combined Pile Footing Design* screen. The reinforcement will appear in the list under Reinforcement.

To manually input the reinforcement, input the values in the text boxes under Reinforcement and click **Add**. You can enter bars that extend to only a portion of the footing by inputting the starting bar location in the **From** box and ending location in the **To** box. To modify existing reinforcement, highlight it in the list, make the appropriate changes, and click **Modify**. Delete existing reinforcement by highlighting it in the list and clicking **Delete**.

You can click **Design Status** to display a design summary of the selected footing. To print the summary, click **Print**. To exit this screen, click **Close**.

Footing: Strap Footing Screen

If you selected more than one column for the footing design and the Design as Strap Footing check box, the *Footing: Strap Footing* screen displays, as shown in Figure GO-53. Here, you can specify the size and reinforcement of a strap footing. A strap footing consists of two heads beneath two selected columns. These two heads are joined together with a strap. You can design/check both heads for soil pressure and reinforcement and design/check reinforcement for the strap. Note that a strap footing is only available if you selected two columns on the *Footing* screen.


Footing:Strap Footing

Configuration
Description: Strap_Spread
From leftmost column ft: 0.
Footing Surcharge ksf: 0.
Spread/Cap Depth (Y) in: 0.
Strap Depth (Y) in: 0.
Length Multiplier (X Dir): 1.
Width Multiplier (Z Dir): 1.

Pile / Shaft Configuration
Shape: Circular
Size in: 0.
Max.Capacity kips: 0.

Strap Reinforcement
Dir.
Bar dist. in: 0.
Bar Size: #3
Num. Bar: 0
From, ft: 0.
To, ft: 0.

Heads Reinforcement
Head: 1
Dir: X
Bar dist. in: 0.
Bar Size: #3
Num. Bar: 0
Hook: None

Maximum pile reactions
● Computed maximum pile reactions
○ User specified maximum pile reaction

	Left head:		Right head:	
Factored	0	kips	0	kips
Service	0	kips	0	kips
Fatigue	0	kips	0	kips

Figure GO-53 Footing: Strap Footing Screen

First, input the footing size by selecting the footing from the Description menu (previously defined under the **Libraries | Footing Configuration** option) and inputting the values in the text boxes.

Next, input the pile/shaft configuration information. This section is only available if you selected the pile/shaft footing type. You can design a strap footing with or without piles (make sure to select the strap footing check box in either case). If you selected the pile/shaft footing type, select the shape of the pile/shaft from the Shape list and input the values in the text boxes. Click **Edit Pile** to edit the coordinates of the pile locations. In the *Edit: Pile Locations* screen, input the values for the X and Z coordinates and click **Add**. It will appear in the list on the screen. Once all locations have been entered, click **OK** to return to the *Footing: Strap Footing* screen.

Now, input the reinforcement for strap and heads. You can have the program generate the reinforcement for both by clicking **Auto Design** to activate the *Design Footing* screen. Input the values for bar size and stirrup size and click **OK**. The program will automatically generate the reinforcement for Strap and Heads and return to the *Footing: Strap Footing* screen.

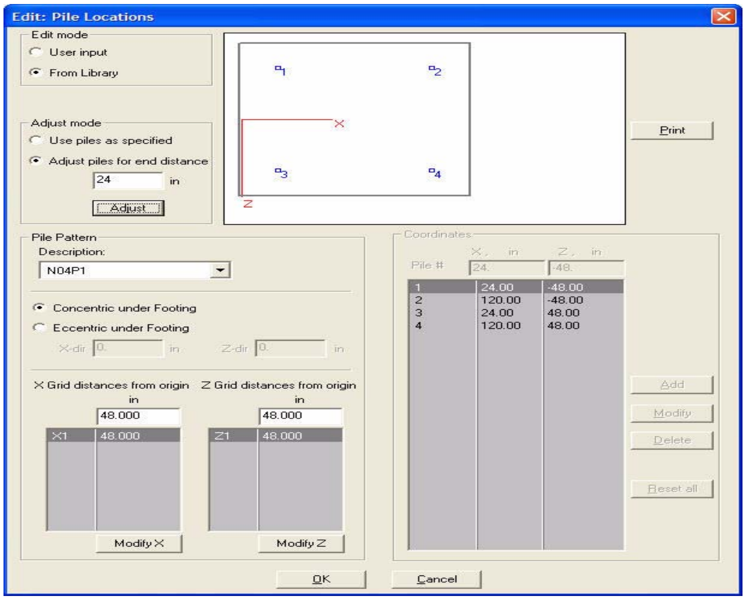
To manually input the reinforcement, input the data in the text boxes and click **Add**. You can enter bars that extend to only a portion of the strap; however, the bars for heads extend to the full length of the heads. To specify the bars under Strap Reinforcement for partial length, input the starting bar location in the From text box and the ending location in the To field. You can modify either reinforcement by highlighting it in the list, making the appropriate changes, and clicking **Modify**. To delete existing reinforcement, highlight it and click **Delete**.

Program can check cracking and fatigue checks for load factored or LRFD designs if service and fatigue load groups are also included in analysis. Program will do this only if check for Cracking box is selected on this dialog. If this box is not selected, program will not do the check for cracking and fatigue in footing.

Program can check cracking and fatigue checks for load factored or LRFD designs if service and fatigue load groups are also included in analysis. Program will do this only if check for Cracking box is selected on this dialog. If this box is not selected, program will not do the check for cracking and fatigue in footing.

Edit: Pile Locations Screen

When you click Edit Pile button on Isolated Pile Cap or Combined Pile Cap or Strap Pile Cap, the *Edit: Pile Location* Screen, as shown in Figure GO-53, will display.



Edit mode
○ User input
● From Library

Adjust mode
○ Use piles as specified
● Adjust piles for end distance
24 in

Pile Pattern
Description: N04P1
● Concentric under Footing
○ Eccentric under Footing
X-dir 0 in Z-dir 0 in

X Grid distances from origin in: 48.000

X1	48.000

Z Grid distances from origin in: 48.000

Z1	48.000

Coordinates
X, in 24 Z, in -48

Pile #	X, in	Z, in
1	24.00	-48.00
2	120.00	-48.00
3	24.00	48.00
4	120.00	48.00

Figure GO-54 Edit: Pile Locations Screen

On this screen you can specify the piles in two ways. First option is the user input option. If you select this option, you can specify each pile coordinate by inputting the value in the X & Z text boxes and clicking add. The pile location will get added to the list and will be displayed in the graphical display. Note the coordinates for piles are measured with the origin as shown in the figure. Repeat these steps to add additional pile locations. You can easily modify an existing pile location by highlighting it in the list, making the appropriate changes, and clicking **Modify**. To delete a pile location, highlight it and click **Delete**.

The second option allows you to specify a pattern from Pile Pattern Library. If you want to use this feature, you must define the pile pattern using the **Libraries | Pile Pattern Configuration** option. Please see page GO-58 on how to define Pile Patterns and save them for a repetitive use. You can save regular (grid based) as well as irregular patterns in the Pile pattern library.

Select the From Library option to use the pattern from the Pile Pattern library. Then select a pattern from the Description list.

Program will read the pattern and show in the graphics area. If the origin of the pile group (defined in library) is at the cg of the footing, choose concentric under footing. However, if Pile group origin is not concentric with footing cg, choose the Eccentric under footing and specify the eccentricity in X and Z direction. The pile coordinates and graphics will be adjusted for the specified eccentricity.

For a grid based (regular) pattern, program provides additional features. You can adjust the location of grid lines at X and Z intervals. If you would like to revise, simply select the entry and modify it. Note that X1 is the distance of a grid location which is farthest from the pattern origin. X2 is the second farthest and so on. Similarly Z1 is the distance of the grid farthest in Z direction. Z2 is the second farthest and so on.

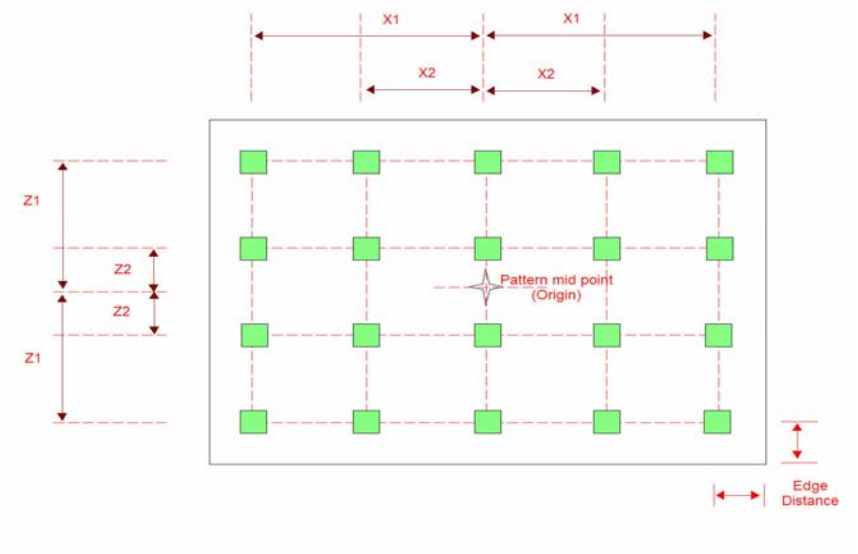


Figure GO-55 Pile Pattern Grid Location

The Adjust Mode feature allows you to adjust the grid based pattern keeping the edge distance constant for the farthest defined grid lines and then equispacing all interior grid locations.

Let us clarify this with an example: assume that the pattern in the Figure GO-54 is defined with c/c spacing of grids at 48 inch in both directions. This means X1 = 96”, X2 = 48”, Z1 = 72” and Z2 = 24”. If the footing size is 28 ft along X and 20 ft along Z the distance between the X1 grid line and the edge of the footing will be

$\frac{28'}{2} \times 12 \quad - 96" = 72$ inches. And the distance between the Z1 and the edge of the footing will be

$\frac{20'}{2} \times 12 \quad - 72" = 48$ inch.

Program will use the grid as defined in the pile library with X1 = 96”, X2 = 48”, Z1 = 72” and Z2 = 24” if the Use Piles as specified option is selected.

If the Adjust piles for edge distance is selected, program will first recomputed X1 and Z1 distance and then equispace the grid locations within. If you select this for the above case with edge distance = 36” X1 will be recomputed as

$$X1 = \frac{28' \times 12}{2} - 36 = 132".$$

X2 for this will then be computed as 132/2 = 66 inch.

$$Z1 = \frac{20' \times 12}{2} - 36 = 84".$$

$$Z \text{ grid spacing} = \frac{84 + 84}{3} = 56".$$

$$Z2 = 84 - 56 = 28"$$

After you specify the piles using either of the two options, click OK and return to the appropriate footing design screen.

Overturning Check Screen

After you have completed design of one isolated spread footing below each column, click on the **Overturning Check** button to open the Overturning Check Screen, as shown in Figure GO-56.

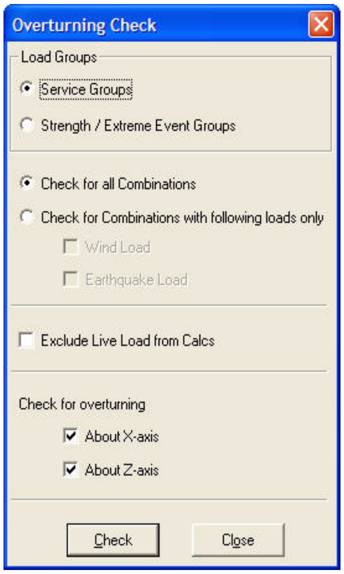
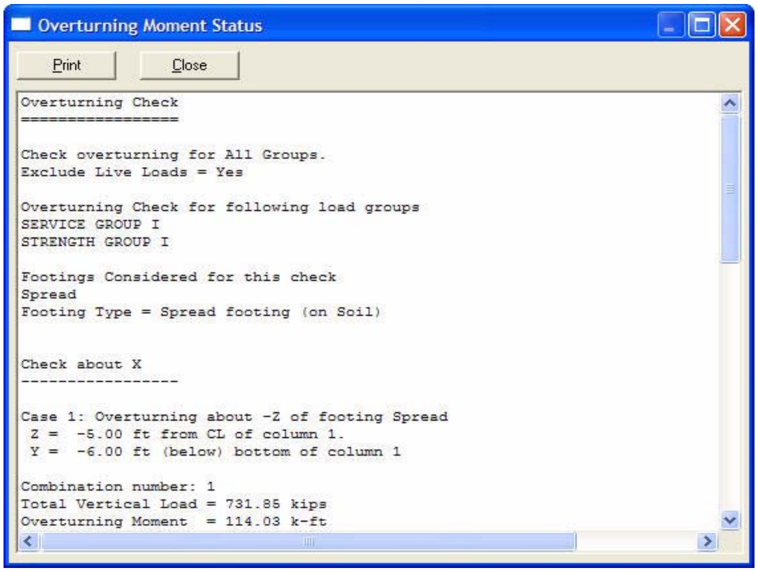


Figure GO-56 Overturning Check Dialog

On this dialog first choose if you would like to check overturning for service groups or factored groups. Note that in LRFD mode, strength as well as extreme event combinations will be checked when factored option is selected. Then choose if you would like to check for all service or factored combinations or only those which have wind or earthquake as one of the load in the combination. If you would like to ignore the effects of live loads from the combinations, select the appropriate box.

Program can check overturning about pier X axis as well as pier Z axis. You may choose to check overturning about both or only one axis.

After making the choices, click on the check button. Program will then display the Overturning check report as shown in Figure GO-57.



```
Overturning Check
=================

Check overturning for All Groups.
Exclude Live Loads = Yes

Overturning Check for following load groups
SERVICE GROUP I
STRENGTH GROUP I

Footings Considered for this check
Spread
Footing Type = Spread footing (on Soil)


Check about X
-----------------

Case 1: Overturning about -Z of footing Spread
 Z =  -5.00 ft from CL of column 1.
 Y =  -6.00 ft (below) bottom of column 1

Combination number: 1
Total Vertical Load = 731.85 kips
Overturning Moment  = 114.03 k-ft
```

Figure GO-57 Overturning Design Status

In the report, program computes and reports two cases of overturning about each axis. For each case, program reports the point about which overturning is being checked, Summation of vertical loads, summation of horizontal loads, total overturning moment, total stabilizing moment, ratio of moments and eccentricity.

Review the report. Click **Print** to print it. Click **Close** and return to the Overturing Check Screen.

Footing Tab Screen Terms

Columns. This option lists all the columns in the pier with their location measured from the left end of the cap. You can click any column in this list to select/deselect it from the footing list. Select only one column if you want to design an isolated spread footing or isolated pile cap. For combined footing, with or without piles, select two or more adjacent columns.

Name. This field allows you to specify the name of each footing before you add it to the footing list.

Type. Select the type of footing you want to design, either on ground (spread footing option) or on piles (pile/shaft cap option).

Design as Strap Footing. This check box is available only if you selected two adjacent columns. If this check box is selected, the program will design a strap footing instead of a combined footing.

Add Button. Adds a footing type to the list of footings. You must first select the column, input a unique name, and select a type before you can add it to the list. Note that more than one footing can be designed for preliminary or alternate study.

Delete Button. Delete any previously added footing in the list.

Design Button. Design the footing selected in the list on the *Footing* screen. The appropriate design screen corresponding to the type of footing selected will display.

Design Status Button. Select a footing in the footing list and click this button to display the *Footing Design Results* screen. The design results for the selected footing will be displayed. This is only possible if the footing has already been designed. See Design Button.

Footing Design Screen Terms

Description. Displays the footing size that you defined previously under the **Libraries | Footing Configuration** option.

From Column. This is the location where the left edge of the footing starts with reference to the centerline of the column. A negative number specifies that the footing starts to the left (negative X-direction) of the column.

From Leftmost Column. Same as From Column. This is the distance between the left edge of the footing and the centerline of the leftmost column. It is used for combined footing.

Footing Surcharge. Specify the amount of surcharge above the footing. This load is in force per unit area, so convert any height of soil into a surface load by multiplying it with the height before specifying it as surcharge. For example a 2 ft soil height with soil weight equal to 120 pcf will have a surcharge load equal to the following:

$$\frac{2 \times 120\text{pcf}}{1000} = 0.24\text{ ksf}$$

The surcharge is applied over the entire footing area including all column areas.

Spread/Cap Depth. For spread footing, the thickness of the footing in global Y-direction. For pile/shaft footing, thickness of the cap.

Length Multiplier. Used to change the footing length dimension in the global X-direction, defined under the **Libraries | Footing Configuration** option. To change this length, multiply it by the length multiplier

Width Multiplier. Used to change the footing width dimension in the global Z-direction. To change this width, multiply it by the width multiplier.

Pile/Shaft Shape. Select the shape of the pile/shaft, e.g., circular, square, or H-pile.

Pile/Shaft Size. The size of the pile/shaft. This is the diameter for a circular pile, or side width for a square, or H-pile.

Max. Pile Capacity. This field allows you to specify the maximum capacity of the pile.

Edit Pile Button. Specify the pile location(s).

Dir. This field specifies if the bar defined is along X or Z direction.

STM X-Dir. Strut and Tie Model in X-direction (globally).

STM Z-Dir. Strut and Tie Model in Z-direction (globally).

Bar dist. Location from the footing base (i.e., bottom surface of the footing in global Y-direction) to the center of reinforcement.

Bar Size. Specify the bar size, i.e., specify U.S. bars for U.S. units and specify metric designations of U.S. bars for metric units.

Num Bars. Number of bars in layer at specified distance.

Hook. This field allows you to specify if the reinforcing bars are to be hooked (either none or both).

Add Button. Add reinforcement to the footing.

Modify Button. Modify any previously existing reinforcement.

Delete Button. Remove any previously existing reinforcement.

Auto Design Button. Once all the required information is provided on the *Footing Design* screen. Clicking this button will allow the program to automatically design the footing. Note, at any time, you can manually change the parameters.

Design Status Button. After a design has been performed, click this button to display a design summary of the selected footing.

Design Soil Pressure: This area allows you to either choose to use the computed soil pressure/ Pile reaction for footing design or specify your own values.

Computed Maximum Soil Pressure: For a spread footing, when this option is selected, program will compute the maximum soil pressure due to all the load combinations. Program will then use that for the structural design of footing.

Computed Maximum Pile Reaction: For a pile cap footing, when this option is selected, program will compute the maximum pile reaction due to all the load combinations. Program will then use that for the structural design of footing.

User specified Maximum Soil Pressure: For a spread footing, when this option is selected, program lets you specify values of factored, service and fatigue soil pressures to be used in the structural design of the footing.

User specified Maximum Pile Reaction: For a pile cap footing, when this option is selected, program lets you specify values of factored, service and fatigue pile reactions to be used in the structural design of the footing.

Factored: For a spread footing, specify the factored soil pressure that will be used in design. For a pile cap footing, specify the factored pile reaction to be used in design.

Service: For a spread footing, specify the service soil pressure that will be used in design. For a pile cap footing, specify the service pile reaction to be used in design.

Fatigue: For a spread footing, specify the fatigue soil pressure that will be used in design. For a pile cap footing, specify the fatigue pile reaction to be used in design.

Overturning Screen Terms

Load Groups: Service Groups. Choose this option if you would like to check overturning for service load groups.

Load Groups: Factored Groups. Choose this option if you would like to check overturning for factored load groups. In LRFD mode, this will include strength as well as extreme event groups.

Check for all Combinations: Use this option to check all combinations.

Check for Combinations with following loads only: Choose this option if you want to check overturning only for wind or EQ combinations.

Wind Load: Select this option to check overturning for only the wind load combinations.

Earthquake Load: Use this option to check overturning for only the Earthquake load combinations.

Exclude Live Load from Calcs: Use this option if you want to ignore the live load from the combinations.

About x-axis: Use this option to do overturning check about x-axis of the pier.

About z-axis: Use this option to do overturning check about z-axis of the pier.

Program Menu Commands

RC-PIER comes with a large collection of timesaving commands and functions. Please make yourself familiar with these features in order to maximize your ability to take advantage of RC-PIER's many features. Some of the commands may be accessed by selecting their respective icons on the toolbar located at the top of the screen.

File | New

This command clears the workspace and prepares RC-PIER for a new project. If there is a file with new data in the workspace when you activate the **New** command, a warning message will appear to ask if any current changes in the file are to be saved. Selecting **Yes** will save the existing data to a file. When the *Save As* screen appears, enter a file name in the File Name field and click **Save**. Click **No** to open a new file without saving any existing data.

File | Open

This command opens a previously saved file. Browse through drives, directories, and files to locate a file. Select by double-clicking on the directory or file of your choice. Clicking **Cancel** will close the screen without opening any files.

Note: RC-PIER allows you to open a data file over the network; however, it does not check how many people open the same data file concurrently. As a result, if two users open the same data file over the network at the same time, they can both update the data file. This may cause confusion and adverse effect on your design work.

If there was existing data in the workspace when activating the **Open** command, a message box will appear asking if you would like to save the latest changes to the existing file in the workspace. Click **Yes** to save the data. The *Save As* screen will display, input a file name and click **Save**. Click **No** to open the new file without saving the current data.

File | Save

This command saves the existing project using the current name. No warnings will be issued from this command. This command is useful for quickly saving intermediate steps during a work session.

If the data was not saved previously, using the **Save As** command, the file will not have a specified name and the **Save** command will (by default) name the file UNTITLED.rcp. In this case, the user will be asked for a file name to save the data under a different name.

File | Save As

This command allows you to save a project file under a different name. The previous data file will be retained under the original name. To save a file, select the appropriate directory and drive and input a name in the File Name field. Click **Save** to save the file under the new name. Clicking **Cancel** will cancel the file saving and return to the program.

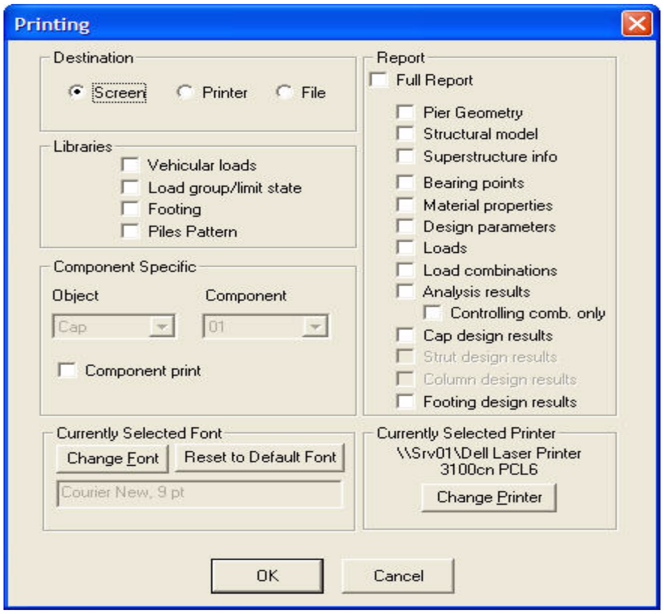
File | Save Setting

The **Save Setting** command allows you to save material properties as default data. After clicking this command, a message will appear asking if you want to save the current settings as default settings. Click **OK** and the next time the program is started or the **New** command is activated, these previously saved settings will exist as defaults. These values are stored in a default file specified under the **Show | Preferences** option.

File | Print

The **Print** command allows you to print the project data and results summary. (Figure GO-58) You can view the results on-screen, send to a printer, or print to a file. You can select a specific component of the pier to print or customize which parts of the results or input data to print. Pier specific results can only be printed for piers for which a design has been completed and the results are available.

A typical printout contains your company header, to include file name, date, etc. The bottom of a printout includes the program name and input file path, and the design code used.



Printing

Destination
● Screen ○ Printer ○ File

Libraries
☐ Vehicular loads
☐ Load group/limit state
☐ Footing
☐ Piles Pattern

Component Specific
Object: Cap
Component: 01
☐ Component print

Report
☐ Full Report
☐ Pier Geometry
☐ Structural model
☐ Superstructure info
☐ Bearing points
☐ Material properties
☐ Design parameters
☐ Loads
☐ Load combinations
☐ Analysis results
☐ Controlling comb. only
☐ Cap design results
☐ Strut design results
☐ Column design results
☐ Footing design results

Currently Selected Font: Courier New, 9 pt

Currently Selected Printer: \\Srv01\Dell Laser Printer 3100cn PCL6

Figure GO-58 File | Print

File | Print Setup

Select this command to activate the *Print Setup* screen. Different printers or plotters can be selected from this screen. Once the output device is selected, click **Properties** to configure the selected output device.

Note: For detailed information about configuring printers or plotters, see the appropriate sections in the *Windows Operating System Manual* and/or the installation documentation for your printer.

File | Convert Old data

Selecting this command will activate the *RC-PIER Upgrade Utility* screen, as shown in Figure GO-59. This command allows you to convert old RC-PIER (DOS version) files to the new format used in RC-PIER.

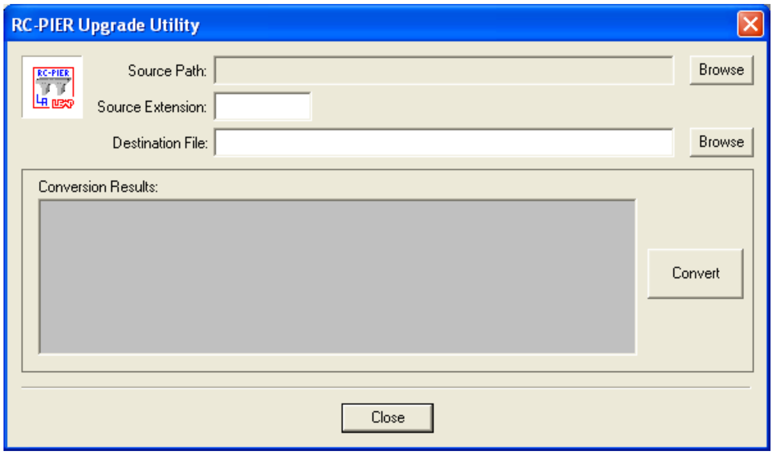


Figure GO-59 File | Convert All the Data

First, input the directory where the old RC-PIER DOS project resides in the Source Path text box, or click **Browse** to locate the directory. Second, input the directory where you want to save the converted file in the format used by RC-PIER in the Destination text box, or you can click **Browse** to locate a directory where the file will be saved. Third, click **Convert**. The program will convert all old data to the current format. The results will be displayed on the screen under Conversion Results. Click **Close** to exit the screen.

File Menu

File | Security

This command will open the *Security* screen, as shown in Figure GO-60. This screen gives the user access to RC-PIER's built-in, electronic security. By selecting the appropriate tab, you can transfer authorization to different directories or computers, or kill an existing license.

Company Name: leap
Address: office
City, State, Zip tampa FL 33687
Country: usa
Phone: 123-456-7890 Fax:

Figure GO-60 File | Security

One valuable feature is the **Security Dir** tab, which allows the user to install the program on a local computer but use the security from another directory, usually from a network. This option allows you to have multiple users without the server or network running slow. See the *Getting Started* chapter for detailed information about installation, authorization, network setup, and authorization warnings.

File | Exit

The **Exit** command exits RC-PIER. If there is any data not saved, the program will display a message asking if you want to save the existing data before exiting the program.

Show Menu

Show | Image

Click the **Image** option from the **Show** menu, or select its respective icon, to activate the *Image* screen, as shown in Figure GO-61. All components of the pier associated with the project at hand, will appear in the diagram once they have been defined.

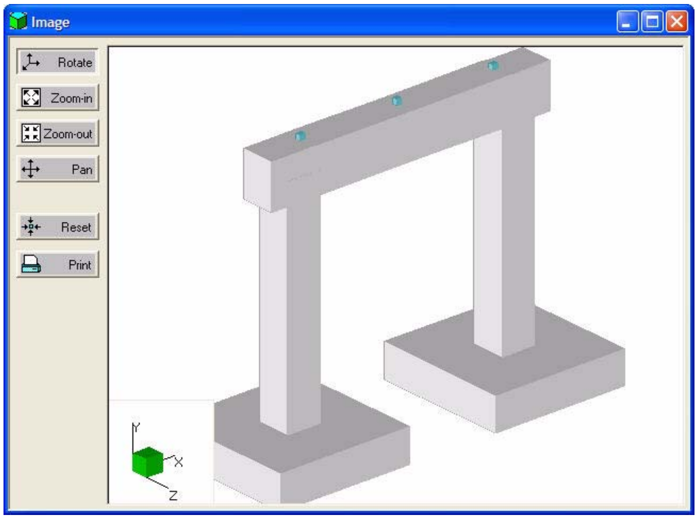


Figure GO-61 Show | Image

Here, the pier image can be generated after the geometry information is provided and without a design analysis being performed. It is encouraged to view the image before proceeding with the design, to confirm the pier geometry.

The control buttons on the left side of the *Image* screen are explained below.

Image Button Summaries

The **Rotate** button allows you to rotate the image. After selecting this icon, you simply bring the mouse pointer anywhere within the image area. Hold down the left mouse button while dragging the pointer. The image is regenerated as you move the pointer.

The **Zoom-in** button allows you to zoom in on a part of the image. After clicking this icon, select the area to enlarge by holding down the left mouse button and dragging the pointer over the part to be enlarged. While dragging the mouse pointer, a selection area will be defined. Release of the button will complete the task, and the selected part of the image will appear magnified.

The **Zoom-out** button allows you to zoom out. As with the zoom-in feature, select an area to zoom out by holding down the left mouse button and dragging the pointer over the part to be reduced. Release of the button will complete the task, and the selected part of the image will appear reduced.

The **Pan** button allows you to pan the image. After clicking on this button, place the mouse pointer anywhere in the image, hold down the left mouse button, and drag the pointer in order to pan the view.

The **Reset** button allows you to reset the image. If you have “zoomed-in” on part of the image, for example, clicking this button will restore the image back to its original size and location.

The **Print** button allows you to print the image displayed on the Image screen. Clicking this button will send the image to the default printer. Note that RC-PIER will print in a “WYSIWYG” format, that is, if the image appears “zoomed-in” on your monitor, then it will print out as shown on the monitor, enlarged.

Show | Model

Click the **Model** option from the **Show** menu, or select its respective icon, to display the *Model* screen, as shown in Figure GO-62. All components of the pier associated with the project at hand, will appear as line members in the diagram once they have been defined.

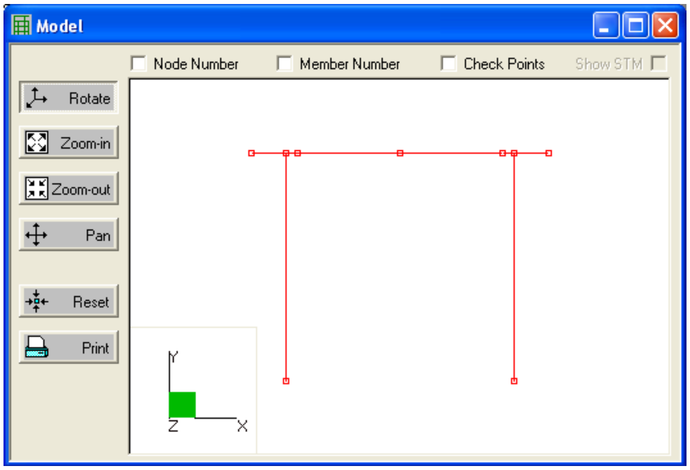
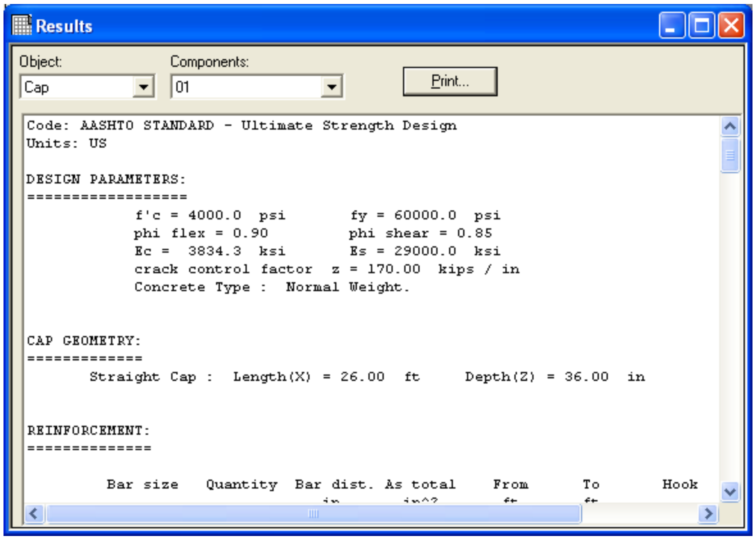


Figure GO-62 Show | Model

You can view the node, member numbers, checkpoints, or STM by selecting the appropriate check boxes at the top of this screen. The control buttons located on the left side of the screen perform the same actions as described under Image Button Summaries, page -64.

Show | Results

This menu option will display the results in tabular format, as shown in Figure GO-63, and is accessible from the **Show** menu, or selecting its respective icon. You can view specific results with regards to the specific object and component number selection.



Object: Cap

Components: 01

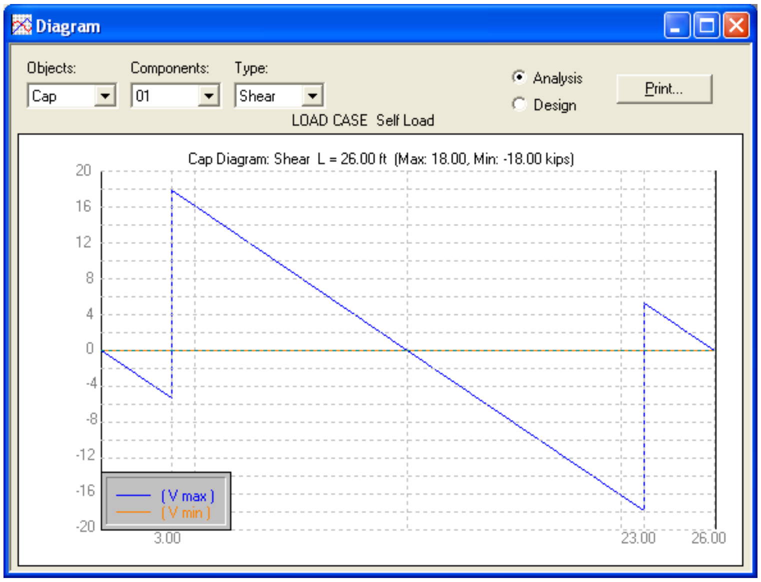
```
Code: AASHTO STANDARD - Ultimate Strength Design
Units: US

DESIGN PARAMETERS:
==================
            f'c = 4000.0  psi        fy = 60000.0  psi
            phi flex = 0.90          phi shear = 0.85
            Ec =  3834.3  ksi        Es = 29000.0  ksi
            crack control factor  z = 170.00  kips / in
            Concrete Type :  Normal Weight.


CAP GEOMETRY:
=============
       Straight Cap :  Length(X) = 26.00  ft     Depth(Z) = 36.00  in


REINFORCEMENT:
==============

         Bar size   Quantity  Bar dist. As total    From      To       Hook
```

Figure GO-63 Show | Results

Any of the results displayed on the screen can be printed individually by clicking **Print**. To close the *Results* screen, click the X located in the top right hand corner of the results display.

Show | Diagram

This menu option will display the results in a graphical format and is accessible from the **Show** menu or selecting its respective icon, as shown in Figure GO-64. You can select specific information from the lists located at the top of the screen to view the graphics.



Diagram

Objects: Cap

Components: 01

Type: Shear

● Analysis

○ Design

LOAD CASE Self Load

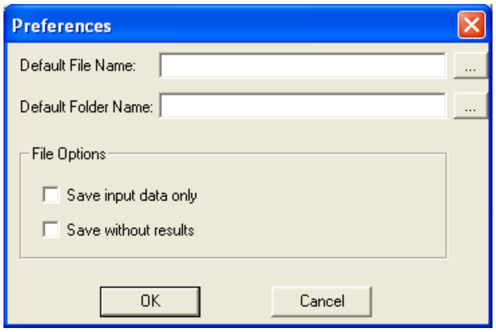
Figure GO-64 Show | Diagram

After the analysis has been performed, Moment and Shear diagrams are available for all components. After the column has been designed, the Moment Interaction diagram is available. To see this diagram, select Column from the Objects list, the component number from the Component list and then, select the Design option. The Column Interaction Diagram will be displayed on the screen. Any of the diagrams displayed on this screen can be printed individually by clicking **Print**.

Note: RC-PIER will print in a "WYSIWYG" format. That is, the image will print exactly as shown on the monitor.

Show | Preferences

This command allows the user to specify a default settings file name or default location for data files and is accessible by selecting the **Preferences** option from the **Show** menu or selecting its respective icon. (Figure GO-65) The Default Folder Name allows the user to specify the default folder in which he or she would like to save the data files when the **Open** and **Save As** commands are used.



Preferences

Default File Name:

Default Folder Name:

File Options

☐ Save input data only

☐ Save without results

Figure GO-65 Show | Preferences

The Save Input Data Only field allows you to save only the data you manually input. Checking this option will make the file size smaller since data and analysis that are generated by the program will not be saved. However, if you select the Save Without Results option, it will keep your input data and the data generated by the program, but the analysis results will be lost.

Libraries

RC-PIER groups commonly used data together in the form of libraries. Libraries are used to store data that does not change often and is to be used repeatedly across projects. The three library types available in RC-PIER are Vehicular Loads, Load Groups/Limit States, and Footing Configuration, all of which store information as suggested by their titles.

RC-PIER is released with a set of default libraries namely Lrfdtrk.rp1 (vehicular loads - LRFD), stdtrk.rp1 (vehicular loads - LFD), Lrfdload.rp2 (Load Groups/Limit States - LRFD), stdload.rp2 (Load Groups/Limit States - LFD), and Default.rp3 (footing). Most users may find it convenient to work with a single library file, modifying, adding, and deleting items in these files rather than creating and setting up different/multiple library files.

Libraries | Setup

The **Setup** command brings up the *Library Setup* screen, as shown in Figure GO-66. The user can select the specific library files to be used as program defaults, for each of the three library types. Once the libraries have been set up in this screen, RC-PIER will keep the library setup selections in the computer registry, and the next time the program is started, it will automatically load the appropriate library files.

Type	File Name
Std Truck	StdTrk.rp1
Std Load	StdLoad.rp2
Footing	Footing.rp3
Piles Pattern	PilePatterns.rp4

C:\Program Files\LEAP Software\RCPIER300\StdTrk.rp1

Figure GO-66 Libraries | Setup

If the user has defined other libraries, click **Select** to activate the *Open* screen. Select the appropriate file and click **Open**. The program will automatically update the current library setup.

Different library files can be created for different library types from the Vehicular Loads, Load Groups/Limit States, and Footing Configuration menus, as explained in the following sections.

Libraries | Vehicular Loads

By selecting the **Vehicular Loads** option from the **Libraries** menu, or its respective icon, the *Library: Truck Main* screen will display, as shown in Figure GO-67. This option allows you to access and manage the vehicles in your library (lrfdtrk.rp1 or stdtrk.rp1). More specifically, you can input the specifics of the load placed on the pier structure by different types of trucks and vehicles. A list of previously defined trucks will display in the screen. This list contains the classes of trucks available in RC-PIER.

In the case of AASHTO LRFD, the values of two design trucks + lane have been reduced to 90% of their original values, which are hard-coded in the LRFD vehicular library.

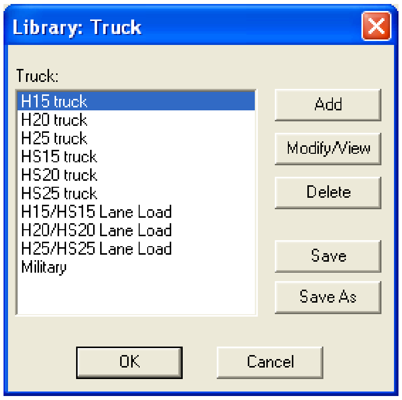
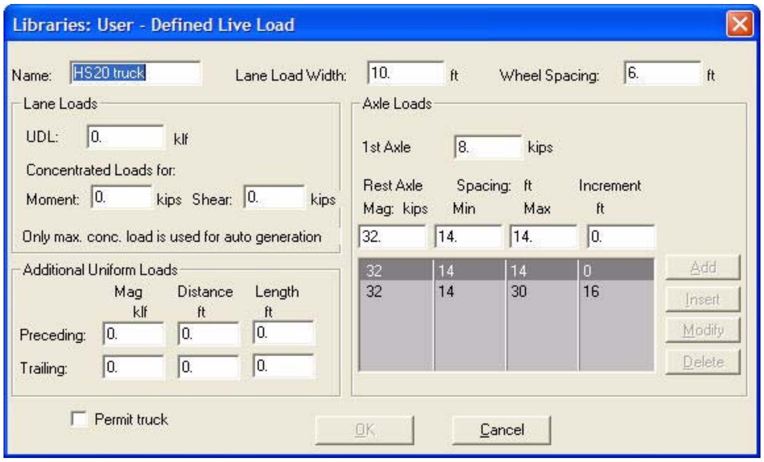


Figure GO-67 Libraries | Vehicular Loads (LFD)

To add a new truck to the list, click **Add**. The program will insert a truck called “Vehicular Load” to the list. This is an undefined load. You can modify this load with your own specifications, by clicking **Modify/View** to activate the *Libraries: User-Defined Live Load* screen, as shown in Figure GO-68. Enter a name in the Name text box and input the values in the text boxes as they apply. If this is a permit vehicle, check the appropriate box for it. Click **OK** to accept the changes and return to the *Library: Truck Main* screen.



Name: HS20 truck
Lane Load Width: 10. ft
Wheel Spacing: 6. ft

UDL: 0. klf
Moment: 0. kips
Shear: 0. kips

	Mag klf	Distance ft	Length ft
Preceding:	0.	0.	0.
Trailing:	0.	0.	0.

1st Axle 8. kips

Rest Axle Mag: kips	Spacing: ft Min	Max	Increment ft
32.	14.	14.	0.
32	14	14	0
32	14	30	16

☐ Permit truck

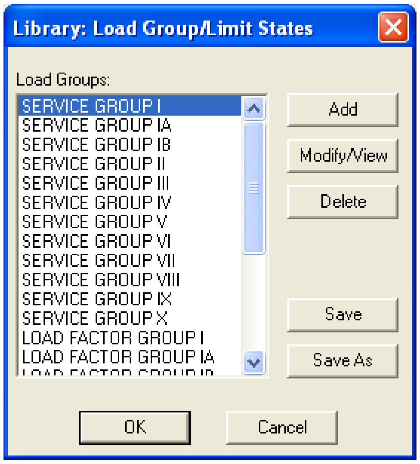
Figure GO-68 Libraries: User-Defined Live Load Screen

Note that **Modify/View** can also be used to review or modify existing truck types, except for default (preloaded) load truck types.

Click **Save** to save the new truck (or any changes to existing truck) to the existing lrfdtrk.rp1 or stdtrk.rp1 library (overriding the existing data). To save under a new file name, click **Save As**. Input a new name in the File Name field and click **Save**.

Libraries | Load Groups/Limit States

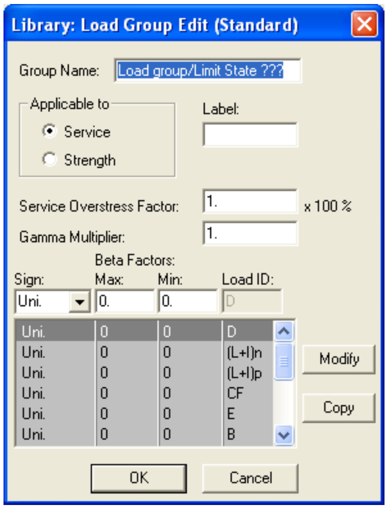
By selecting the **Load Groups/Limit States** option from the Libraries menu, the *Library: Load Group/Limit States Main* screen displays, as shown in Figure GO-69. This library (lrfdload.rp2 or stdload.rp2) contains all the information pertaining to the Load Groups. Here you can modify existing load groups or define your own groups. A list of the available load groups in RC-PIER are displayed on the screen under Load Groups.



SERVICE GROUP I
SERVICE GROUP IA
SERVICE GROUP IB
SERVICE GROUP II
SERVICE GROUP III
SERVICE GROUP IV
SERVICE GROUP V
SERVICE GROUP VI
SERVICE GROUP VII
SERVICE GROUP VIII
SERVICE GROUP IX
SERVICE GROUP X
LOAD FACTOR GROUP I
LOAD FACTOR GROUP IA
LOAD FACTOR GROUP IB

Figure GO-69 Libraries | Load Groups/Limit States

To add a new load group, click **Add**. The program will add a load group called “Load Group/Limit State(?)” to the list. This is an undefined load group. You can modify this load group with your own specifications by selecting **Modify/View** to activate the *Library: Load Group Edit* screen, as shown in Figure GO-70.



Group Name: Load group/Limit State ???

Applicable to: ● Service ○ Strength

Label:

Service Overstress Factor: 1. x 100 %

Gamma Multiplier: 1.

Beta Factors:

Sign:	Max:	Min:	Load ID:
Uni.	0.	0.	D

Uni.	0	0	D
Uni.	0	0	(L+I)n
Uni.	0	0	(L+I)p
Uni.	0	0	CF
Uni.	0	0	E
Uni.	0	0	B

Figure GO-70 Library: Load Group Edit Screen

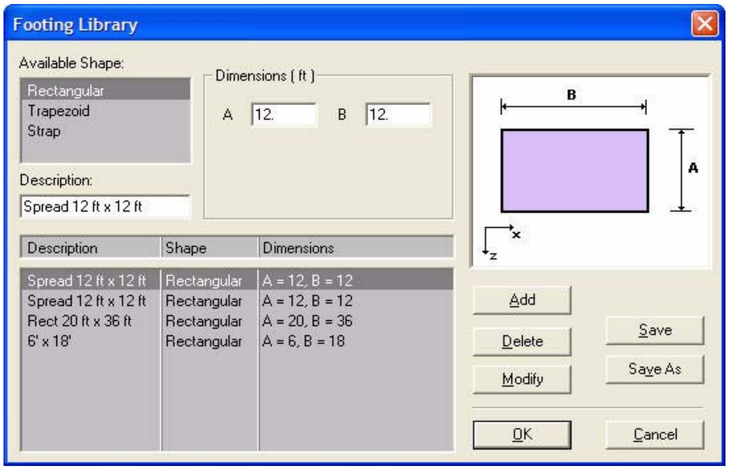
Input a name in the Group Name field and the values in the text boxes as they apply. Click **OK** to accept changes and return to the *Library: Load Groups/Limit States Main* screen.

You can copy an existing load group library to the one you are editing by clicking **Copy**.

Click **Save** to save the new load group (or any changes to existing load groups) to the existing lrfdload.rp2 or stdload.rp2 library (overriding the existing data). To save under a new file name, click **Save As**. Input a new name in the File Name field and click **Save**.

Libraries | Footing Configuration

Selecting the **Footing Configuration** option from the **Libraries** menu, or its respective icon, will display the *Footing Library* screen, as shown in Figure GO-71. This command allows you to manage and define the different footing types used in the pier structure.



Available Shape:
- Rectangular
- Trapezoid
- Strap

Dimensions (ft): A 12. B 12.

Description: Spread 12 ft x 12 ft

Description	Shape	Dimensions
Spread 12 ft x 12 ft	Rectangular	A = 12, B = 12
Spread 12 ft x 12 ft	Rectangular	A = 12, B = 12
Rect 20 ft x 36 ft	Rectangular	A = 20, B = 36
6' x 18'	Rectangular	A = 6, B = 18

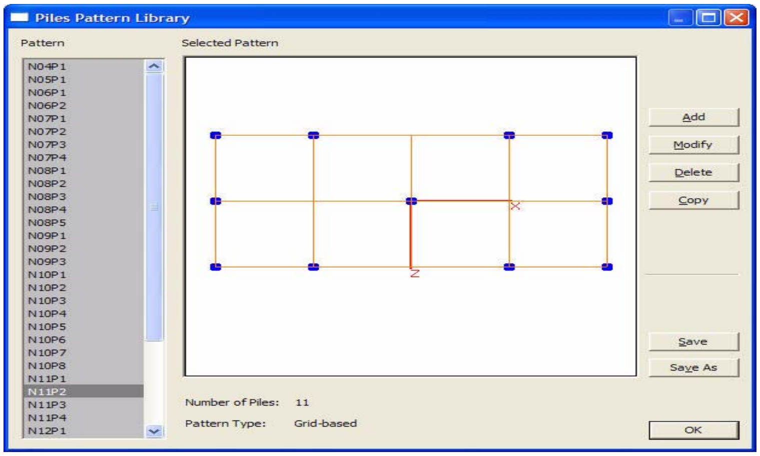
Figure GO-71 Libraries | Footing Configuration

To define a new footing type, select the shape from the Available Shape list. Then, referring to the diagram on the right side of the screen, input the footing size in the text boxes under Dimensions and a name for the footing in the Description text box. Click **Add**. The name will appear in the list on the screen. You can modify existing footing types in the list by highlighting it, making the appropriate changes, and clicking **Modify**. To delete a footing type in the list, highlight it and click **Delete**.

Click **Save** to save the new footing type (or any changes to existing footing types) to the existing default.rp3 library (overriding the existing data). To save under a new file name, click **Save As**. Input a new name in the File Name field and click **Save**.

Libraries | Pile Pattern Configuration

Selecting the Pile Pattern Configuration option from the Libraries menu, or its respective icon, will display the Pile Pattern Library Screen as shown in Figure GO-72. This Library allows you to define and manage pile pattern configurations (pile groups).



Pattern:
- N04P1
- N05P1
- N06P1
- N06P2
- N07P1
- N07P2
- N07P3
- N07P4
- N08P1
- N08P2
- N08P3
- N08P4
- N08P5
- N09P1
- N09P2
- N09P3
- N10P1
- N10P2
- N10P3
- N10P4
- N10P5
- N10P6
- N10P7
- N10P8
- N11P1
- N11P2
- N11P3
- N11P4
- N12P1

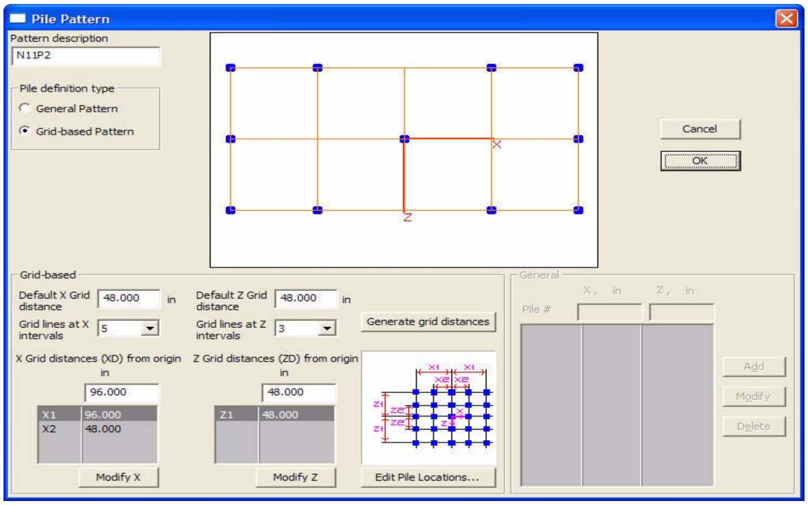
Number of Piles: 11

Pattern Type: Grid-based

Figure GO-72 Libraries | Pile Pattern Configuration

To define a new pile pattern, click Add. The program will insert a pattern called “Pile Pattern???” to the bottom of the pattern list.

Select newly added item and click modify to activate the Pile Pattern screen as shown in Figure GO-73.



Pattern description: N11P2

Pile definition type:
○ General Pattern
● Grid-based Pattern

Default X Grid distance: 48.000 in

Default Z Grid distance: 48.000 in

Grid lines at X intervals: 5

Grid lines at Z intervals: 3

X Grid distances (XD) from origin in: 96.000

X1	96.000
X2	48.000

Z Grid distances (ZD) from origin in: 48.000

Z1	48.000

Figure GO-73 Library: Pile Pattern Screen

Input a unique name of the Pile pattern. Then choose the Pattern type. Pattern could either be grid based on which the piles lie on a regular grid, or it can be a General pattern in which piles can be placed at any location with X and Z coordinates.

If you choose general patterns, simply specify X and Z coordinates of each pile and then click Add. If you would like to modify a previously added pile location, first click that in the list. Then modify the X and Z values and then click the modify button again. To delete a pile simply select it and click delete.

To specify a grid based pattern, first choose that option. Then specify the spacing of grid lines in X and Z direction. Then select number of grid lines in two directions and then click Regen Grid Locations button. Program will recomputed the grid locations and show those as X1, X2, X3… and Z1, Z2, Z3…etc. and show in the two lists. It will also update the grid layout in the graphics area.

All the grid based patterns must have symmetric grid placed with respect to the origin (mid) of the pattern as shown in Figure GO-74.

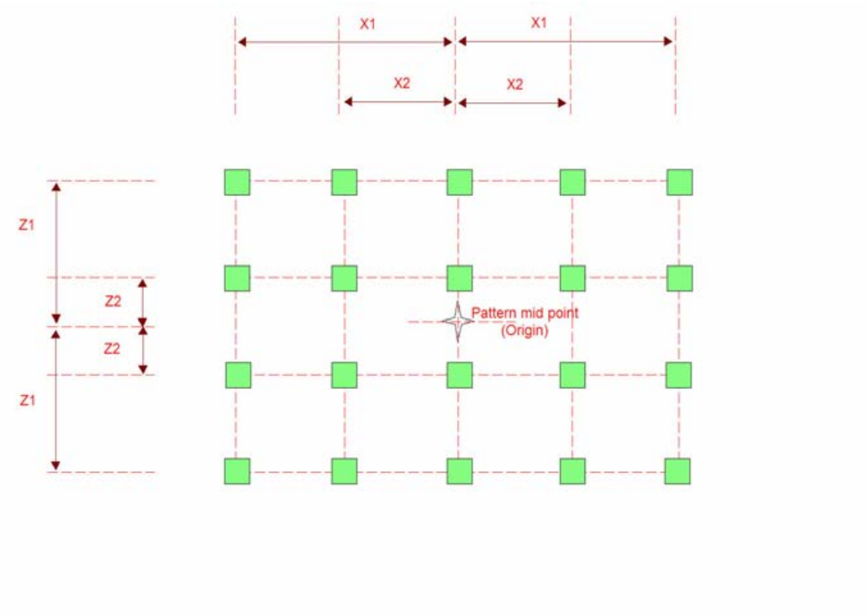
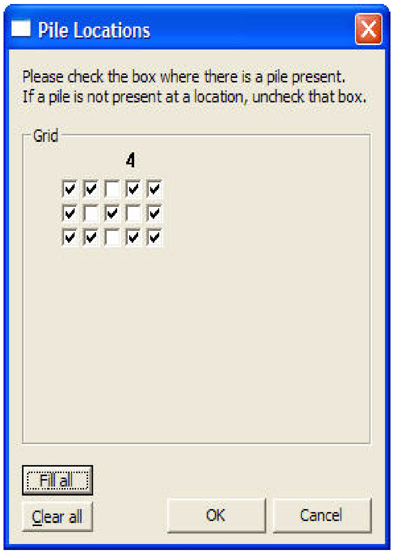


Figure GO-74 Pile Pattern Grid

X1 is the distance of grid lines at farthest X location from the mid (origin) of the pattern X2 is the distance of the next location. Similar is true for Z locations.

If some of the grid lines are not at constant spacing, you can modify the grid locations after initial grid generation. To do this, first generate regular grid and then modify the X1, X2, Z1, Z2,… etc. locations. To do this simply select a location (X1, X2, etc.) and then revise the corresponding value and click modify. Program will modify the location and update the graphics display.

All the grid intersection points are possible pile locations. To Specify the intersections on which the piles are really present, click the Edit Pile Locations button. This will display the Pile Locations screen as shown in Figure GO-71.


Pile Locations

Please check the box where there is a pile present.
If a pile is not present at a location, uncheck that box.

Grid

4

☑ ☑ ☐ ☑ ☑
☑ ☐ ☑ ☐ ☑
☑ ☑ ☐ ☑ ☑

Figure GO-75 Library: Pile Locations

This screen has a grid of check boxes. Each check box represents an intersection of grid lines. To place a pile at a grid intersection check the corresponding box. If you do not want to place a pile at a location, make sure the corresponding box is unchecked. To set piles at all locations click Fill All. To remove all piles, click Clear All. Click OK once all the pile locations are specified. Click cancel if you do not want to make any changes to the placement positions.

To save the pattern, click OK to get back to the library main dialog and then click Save button.

Help Menu

Help | Contents

This menu option opens the Help file, which lists the items contained in the RC-PIER Help. Select an item and the help routine will display all information associated with that item. You can access the help file anytime within the program by pressing the **F1** key.

Help | User Manual

This menu option opens the User Manual. The User Manual is in PDF format and can be printed.

Help | Search for Help On

This menu option opens RC-PIER’s Help search engine. Type in the topic(s) to search for information on, and the Help program will compile a listing of the matches it finds. Click on an item in the list to display the associated information.

Help | How to Use Help

This menu option gives you some general information and tips on how to use RC-PIER’s Help routines.

Help | Tech. Support

This menu option provides information about how to contact our technical support team, to include LEAP’s phone number (1-800-451-5327) and e-mail address (techsup@leapsoft.com). For additional information, see "Contacting Technical Support" on page GS-11.

Help | Check for Updates

Select this option to check the LEAP Software web site for updates of RC-PIER.

Help | E-Mail Tech Support

This menu option starts your e-mail program if it is not already open and creates a message addressed to LEAP Technical Support (techsup@leapsoft.com). The program automatically attaches the currently open data file along with information about your system. You may edit the message if necessary and add a description of the problem before sending the e-mail message.

Help | Visit LEAP Web Site

If you have Internet access, clicking this menu option will activate the dial-up feature, load your browser, and direct you to LEAP’s Web site. Check this site frequently for important program updates, news, technical support questions, and more.

Help | About RC-PIER

This menu option contains general information and disclaimer on the RC-PIER program, as well as the computer's physical memory.

Using IBS Technology

Chapter
LB

This chapter explains the basics of the application interoperability process inherent in the LEAP Bridge™ configurations, referred to as Integrated Bridge Software™ (IBS). LEAP Bridge applications share data with each other and react to changes to shared data.

Introduction

Because we at LEAP Software are engineers first, we understand your world. Our professional staff encompasses more than 200 years of combined structural engineering experience, and our products are specifically designed to address the challenges you face, such as the need for an integrated bridge design and analysis software tool.

For more than 30 years, structural engineers have wanted computer applications to readily share information with each other. All too often, for example, users must enter geometric information to define a roadway in a design program, then must re-enter that same data into an analysis program. Or, as specific components of a bridge change, they must go back into their design program to implement the changes already suggested by their design application. The common lament has always been, “Wouldn’t it be great if these applications would share what they know with each other?”

As the leader in bridge analysis and design software, we forged this frontier with the development of LEAP Bridge, the first software package that parametrically adjusts to changes in component design and populates those changes throughout the geometry, substructure and superstructure of a project. For example, spans that are designed in CONSPAN may be passed to GEOMATH. Any subsequent changes made by either application can be shared between the two, such that the project is consistently maintained throughout its life-cycle.

You will find that IBS:

- Reduces repetitive data entry as the applications draw the necessary information from each other;
- Virtually eliminates errors associated with repetitive data entry;
- Contributes to a reduction in total project time and costs.

Conceptual Overview

The basic operating model for LEAP Bridge with IBS Technology is that an engineer typically uses a number of applications to solve analysis and design problems. The current toolbox of analysis and design software is based on a structural component model in that each piece of software solves a piece of the overall problem. There are separate applications for bridge longitudinal analysis, and girder analysis, as well as for abutment, pier, and footing analysis and design.

Therefore, LEAP Bridge is tailored to the sharing of data between the different applications that engineers use in their daily analysis and design activities. There is an external project file which is shared between applications; each application writes to this external project file. This transfer file is also referred to as a database file.

LEAP Bridge applications pass all geometry data to the external project file and each application reads the appropriate data for its use. The data which will be read *from* the database is selected by the user and subscribed by the project. Any changes made to the selected data are reflected within the application. This includes changes by other LEAP Software applications.

Once an application writes data to the database (or subscribes to data), there is a mapping created between the data in the application and the data in the database. Any changes made to a project while working in a particular LEAP Bridge application - including deletions - are concurrently made in the external project file, and will be reflected as changed when other LEAP Bridge applications read data from that same project file. In situations where this is not desired, make sure that the data sharing mode is disabled.

All LEAP Software applications maintain a native data file with an application specific extension, including LEAP Bridge applications. For example, CONSPAN uses the *.csl* extension, GEOMATH uses the *.gmd* extension, RC-PIER uses the *.rcp* extension, and CONBOX uses the *.cbx* extension. Within the database file, data selected for synchronization replaces like data. Other data is generally not affected unless the native data integrity of the program is affected.

LEAP Bridge is not intended to be the final repository of all persistent data for a company or agency. The database file is an XML file and the file is accessed without the services of a database management system.

Note: It is recommend that multiple users make use of LEAP Bridge applications on a sequential basis only.

Application Interaction

The database file and method of data synchronization must be selected for LEAP Bridge enabled applications. This is done by selecting the **Setup** command from the **LEAP Bridge** menu shown below to open the ***IBS Setup*** dialog box.

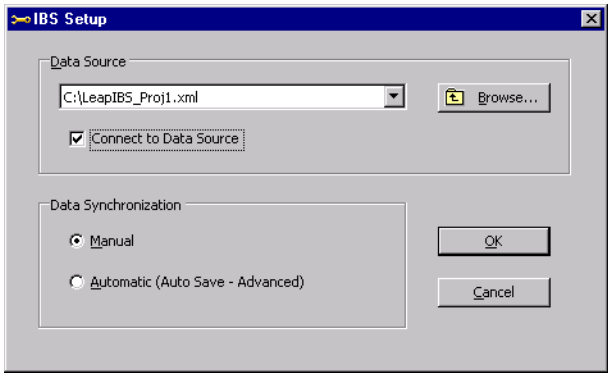
Figure LB-1 LEAP Bridge Menu

Interacting applications must connect to the same database file (XML file) to share data. An application may switch database files at any time, thus giving additional flexibility in data sharing. Typically, a database file is created for each project (e.g., “Bridge 15-10.xml”). Each application then should subscribe to the entire bridge project.

Once an application has connected to a database file, a message will be sent to the application whenever any modifications are made to that file. For example, if both LEAP Bridge applications subscribe to a four-span bridge, modifying the precast beam lengths in one span, adding a pier to the bridge, or changing the alignment in either application would be reflected in the other in real time.

IBS Setup Dialog Box

Each LEAP Software application utilizing LEAP Bridge has the LEAP Bridge menu available, as shown in Figure LB-1. Select **Setup** to open the *IBS Setup* dialog box, as shown in Figure LB-2.



C:\LeapIBS_Proj1.xml

☑ Connect to Data Source

● Manual

○ Automatic (Auto Save - Advanced)

Figure LB-2 IBS Setup Dialog Box

Database File

The *Data Source* list provides the name and location of all XML files which have been written. Use **Browse** to navigate to the location of an existing file, or to the location where the new file is to be created.

After selecting the Data Source (XML file), select the **Connect to Data Source** check box to put the application in a data sharing mode. This box may be unchecked at any time, allowing you to work in a connected or unconnected mode as desired.

Note: An option to clear history is presented when right clicking the menu. It is recommended that this option be used only after saving a copy of the log file. In this manner, each log file will be associated with a specific project, thereby retaining a record of all XML files associated with a project. The log file is created and maintained in the following location: C:\Documents and Settings\user\Local Settings\Application Data\Leap Software, Inc\IBS\1.0.0.0

Location

The Database file (XML file) may be located in any desired location. The database files may be placed on the local hard drive or on a network drive to facilitate group collaboration or take advantage of enterprise file backup procedures.

Format

The database file, shown in the “ProjectInfo” example below, is an XML file. XML files are readable and editable in any text editor. However, if incorrectly edited (such as deleting tags) the file may become invalid for subsequent processing by IBS.

```
<ProjectInfo>
  <Name />
  <ObjectID>1</ObjectID>
  <Health>Healthy</Health>
  <ChangeLog>Created: 6/23/2004 11:49:38 AM; By: LEAPSOFT\AR; Using: Geomath</ChangeLog>
  <GEOMATH_UNITS xmlns="">ft</GEOMATH_UNITS>
  <ProjName>Single-Span Bridge Layout</ProjName>
  <ProjDescription />
```

```
    <ProjNumber>Tutorial 1</ProjNumber>
    <State />
    <Date>6/22/01</Date>
    <Designer>LEAP</Designer>
    <JobNumber />
    <Units>US</Units>
  </ProjectInfo>
```

Data Synchronization

Data synchronization refers to the process where an application synchronizes data with the database. There are two types of data synchronization in LEAP Bridge applications: **Manual** and **Automatic**. The default type of data synchronization is **Manual**.

Manual

When Manual synchronization is selected, select the **Write To DB** command in the **LEAP Bridge** menu (or press **F4**) to write data to the database file and select the **Read From DB** command (or press **F5**) to read subscribed data back into an application.

For example, when creating profiles, data transfer would not be required. When a structural analysis needs to be performed to obtain dead or live load results and confirm the appropriateness of various bridge components specified in **GEOMATH**, perform the following steps:

Note: Also refer to *Typical Scenarios* on page LB-7 for scenarios using LEAP Bridge applications.

---

1. Select the **Setup** command menu option from the **LEAP Bridge** menu and browse to the appropriate directory and create a new database file.
2. Click **Save** and then select the **Connect to Data Source** option in the **IBS Setup** dialog box and click **OK.**
3. Select the **Write To DB** command (or press **F4**) to write the data to the database file.
4. Start another LEAP Bridge application and select the **Setup** command from the **LEAP Bridge** menu in that program.
5. Select the **Connect to Data Source** option; the same database file specified in **GEOMATH** will be listed in the field. Click **OK.**
6. Select the **Select Components** command from the **LEAP Bridge** menu, expand the Bridges tree item and select the bridge (the default name is “BR01”)
7. Click **OK**. The program will read the database file and import the data written by **GEOMATH**. At this point, the program is “subscribing” to the data associated with “BR01” and any changes made to it by another application will be reflected when the **Read From DB** command is selected.

Note: GEOMATH currently is not subscribing to the data in the database file, so changes made by the second application will not be reflected in GEOMATH upon a read. In order for GEOMATH to see any changes made to the data by the current program, subscribe to BR01 in GEOMATH using this procedure.

---

8. Select the materials, add dead loads, live loads, or make other changes to the loadings, combinations, post-tensioning, etc. to BR01 and perform an analysis.

9. Review the results. Make changes to the structure geometry in the current application. For example, depending which LEAP Bridge application in use, modify the structure depth or the beam spacing and perform additional analyses until the design is appropriate. Save the data in the current application.
10. Select the **Write To DB** command to update the information in the database file.
11. Activate **GEOMATH** and open the project file. Connect to the database file and select bridge “BR01.” The changes made in the other application will now be reflected in **GEOMATH**.

Automatic

This type of data synchronization means that when an application is *deactivated* (closed or moved in the background window) it will automatically write its data to the database file and, when the database file is modified in any way, the application will update with any selected or subscribed data.

This type of data synchronization is an advanced mode that allows a user to utilize multiple applications to solve a bridge design problem by using one application to make changes to data, selecting another application, and making additional changes. Changes to subscribed data are reflected in each application in real time.

Application Status Bar

Each LEAP Bridge application includes buttons in the application status bar to show the status of the database connection and also the status of the type of data synchronization. The status bar shown in Figure LB-3 indicates the application is connected to a database file and the type of data synchronization is set to Manual.

Figure LB-3 Application Status Bar with Data Synchronization Set to Manual

The state of the database connection is **IBS: Not Connected** with an unknown file when an application is first started. Clicking **IBS: Not Connected** in this state opens the **IBS Setup** dialog box. Clicking **IBS Connected** in the status bar thereafter will toggle the connection on and off.

Click **IBS: Manual** to toggle the type of data synchronization between Automatic and Manual. This behavior occurs even if the application has not yet been put in data sharing mode.

Selecting Components for Synchronization

Once connected to a database file, applications will write to the database file, either automatically or manually. The specific data written to the database is controlled by the application.

The data read from the database file selected by the user from the **Project Tree**. The application will read the selected data from the database and synchronize accordingly. The Project Tree is opened by selecting **Select Components** from the **LEAP Bridge** menu. Note that this command will be disabled until connected to a database file.

The Project Tree presents a structured view of the data in the database file, as shown in Figure LB-4. The title bar displays the name of the currently connected database. Use the check box next to each tree node to select data to be read from the database file. Once selected, the application will update the native data file on each read.

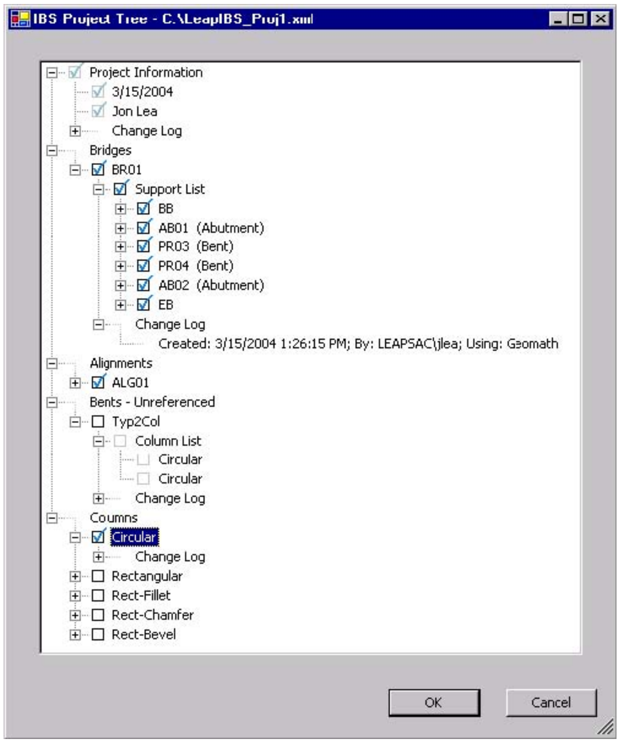


Figure LB-4 Project Tree List

Note: Selecting data items from within the **Project Tree** marks those items as “selected.” If a particular application does not recognize bents or alignments, those items will not be read.

Certain project information is selected by default. These default selections cannot be canceled, therefore applications will always synchronize with this Project Information. This behavior is consistent with the concept that the IBS database is a *project* level database.

A project level database may contain one or more bridges, alignments, bents, etc. A particular project may be an intersection which is first described in GEOMATH with several alignments, and later several bridges may be placed on those alignments with many piers. Another LEAP Bridge application can be used to add loads, and perform analyses and designs, on selected bridges.

The **Project Tree** also shows that subsets of data items cannot be selected because the check box is unavailable. For example, the Column List for the bent named “Typ2Col” and the two “Circular” columns in the list are disabled; therefore, they cannot be selected. When “Typ2Col” is selected, the Column List and the two columns will be automatically selected. This means that the Column List (or the two columns) cannot be selected independently of the bent. A bent and its list of columns is a single entity relative to selection and data synchronization.

Renaming & Deleting Components from the Project Tree

Items within the **Project Tree** may be renamed or deleted. This is done by right-clicking on enabled items and selecting either **Rename** or **Delete**. Click **OK** to commit. The deletions will be propagated to all running applications.

Note: There is no “undo” once data is written to, or deleted from, the database file.

Typical Scenarios

Defining A Bridge Using GEOMATH And CONBOX

To define a Bridge using GEOMATH and CONBOX, perform the following steps:

1. Start GEOMATH and open the *.gmd* file associated with the project (e.g., “ProjABC.gmd”).
2. Select the **Setup** command menu option from the **LEAP Bridge** menu and browse to the appropriate directory and create a new database file named: “ProjABC.xml”.
3. Click **Save** and select the **Connect to Data Source** option in the **IBS Setup** dialog box. Click **OK**.
4. Select the **Write To DB** command (or press **F4**); GEOMATH will write to the database file.
5. Start CONBOX and select the **Setup** command from the **LEAP Bridge** menu.
6. Select the **Connect to Data Source** option. The same database file specified in GEOMATH will be listed in the field. Click **OK**.
7. Select the **Select Components** command from the **LEAP Bridge** menu, expand the Bridges tree item and select the bridge (the default name is “BR01”).
8. Click **OK**. CONBOX will read the database file and import the data written by GEOMATH. At this point, CONBOX is “subscribing” to the data associated with “BR01”. Any changes made to it by another application will be reflected in CONBOX when the **Read From DB** command is selected.

Note: GEOMATH is currently not subscribing to the database file; therefore, changes made by CONBOX (or another application) will not be reflected in GEOMATH upon a read. In order for GEOMATH to read changes made to the database file subscribe to BR01 in GEOMATH.

9. Add a dead load and other loadings and combinations, post-tensioning, etc. to BR01 and perform an analysis.
10. Review the results. Make changes to the structure in CONBOX, for example, modify the structure depth and perform additional analyses until the design is appropriate. Save the data in CONBOX to a file, e.g., “ProjABC.cbx”.
11. Select the **Write To DB** command to update the structure depth information in the database file.
12. Either activate GEOMATH or start it up and open “ProjABC.gmd,” connect to the database file and select bridge “BR01.” The changes made to the structure depth in CONBOX will be reflected in GEOMATH**.**

Defining A Bridge Using GEOMATH And CONSPAN

To define a bridge. using GEOMATH and CONSPAN, perform the following steps:

1. Start GEOMATH and open the *.gmd* file associated with the project (e.g., “ProjABC.gmd”).
2. Select the **Setup** command menu option from the **LEAP Bridge** menu and browse to the appropriate directory and create a new database file named: “ProjABC.xml”.
3. Click **Save** and select the **Connect to Data Source** option in the **IBS Setup** dialog box. Click **OK**.
4. Select the **Write To DB** command (or press **F4**); GEOMATH will write to the database file.
5. Start CONSPAN and select the **Setup** command from the **LEAP Bridge** menu.

6. Select the **Connect to Data Source** option. The same database file specified in GEOMATH will be listed in the field. Click **OK**.
7. Select the **Select Components** command from the **LEAP Bridge** menu, expand the Bridges tree item and select the bridge (the default name is “BR01”).
8. Click **OK**. CONSPAN will read the database file and import the data written by GEOMATH. At this point, CONSPAN is “subscribing” to the data associated with “BR01” and any changes made to it by another application will be reflected in CONSPAN when the **Read From DB** command is selected.
9. Select the materials, and add dead and live loads to the bridge and run analysis.
10. Review the results. Make changes to the structure in CONSPAN, for example, modify the precast beam length and perform additional analyses until the design is appropriate. Save the data in CONSPAN to a file, e.g., “ProjABC.csl”.
11. Select the **Write To DB** command to update the precast beam length information in the database file.
12. Either activate GEOMATH or start it up and open “ProjABC.gmd,” connect to the database file and select bridge “BR01.” The changes made to the precast beam length will be reflected in GEOMATH**.**

DEFINING A PIER USING GEOMATH AND RC-PIER

To define a Pier using GEOMATH and RC-PIER, perform the following steps:

1. Start GEOMATH and open the *.gmd* file associated with the project (e.g., “ProjABC.gmd”).
2. Select **Setup** from the **LEAP Bridge** menu. The IBS Setup dialog will open. Browse to the appropriate directory and create a new database file named: “ProjABC.xml”. Click **Save.**
3. Select **Connect to Data Source,** select **Manual** and click **OK**.
4. Select the **Write To DB** command (or press **F4**); GEOMATH will write its data to the database file.
5. Start **RC-Pier** and select **Setup** from the **LEAP Bridge** menu.
6. The same database file specified in GEOMATH will be listed in the field. If not, select it and click **OK**. Select the **Connect to Data Source** option and select **Manual**.
7. Select the **Select Components** command from the **LEAP Bridge** menu, expand the following tree items: Bridges, BR01, Support List, and select **PR01**.
8. Click **OK**. RC-PIER will read the database file and import the data written by GEOMATH. At this point, RC-PIER is “subscribing” to the data associated with Pier PR01 “BR01” and any changes made to it by another application will be reflected in RC-PIER when the **Read from DB** command is selected.

Note: GEOMATH currently is not subscribing to the data in the database file so changes made by RC-PIER will not be reflected in GEOMATH upon a read.

---

9. In RC-PIER, add and generate loads and load groups and perform an analysis.
10. Review the results. Make changes to the structure in RC-PIER, for example, modify the cap depth and perform additional analyses until the design is appropriate. Save the data in RC-PIER to a file, e.g., “ProjABC.rcp”.
11. Select the **Write To DB** command to update the cap depth information in the database file.
12. Either activate GEOMATH or start it up and open “ProjABC.gmd,” connect to the database file and select bridge **“BR01” PRO1**. The changes made to the PR01 cap depth in RC-PIER will be reflected in GEOMATH.

IBS Shared Data Items

As each LEAP Software application is designed to solve specific engineering issues, each application has specific internal data structures and assumptions. These internal data structures and assumptions dictate how data is exchanged with the database file and if the data can be shared with other applications.

For example, an application that has a span-based internal representation for support locations will be able to *read* support locations from the database file for its use, converting them to span lengths as appropriate. However, it will not be able to make any changes to support locations in the database (i.e., write span lengths) when changes are made to span lengths since the application will not know which support to move in order to realize the new span length.

For instance, you are using one application that is able to modify support locations (such as GEOMATH) and another application which is unable to modify support locations (such as CONSPAN). If you make a change to a span length in CONSPAN, and then re-synchronize CONSPAN with the data in the database file, the original span length will be transferred back into CONSPAN. To change a span length, make a support location change in the application that can modify support locations or disconnect from the database file in the application that cannot make the support location change after data from the database file has been initially transferred.

Applications cannot alter or delete any unknown data items in the database file. For example, if an application supports multi-column bents, it will write the data about the type of columns and column spacings for those bent types to the database file, since that information is part of the database schema. Another application that supports bents (i.e., data items such as cap width), but does not support columns, will be able to read and write some data about the bent, but not all. The fact that the bent it is being shared with other applications which might have one or more columns will not be considered.

Table 1 on page IB-10 lists the data items currently defined in the database file. Data items which are utilized are indicated by a dot. Explanatory notes are provided as necessary to explain application specific data interpretations.

Table LB-1 LEAP BRIDGE Database Data Items

	GEOMATH (GMD)			CONBOX (CBX)			CONSPAN (CSL)			RC-PIER (RCP)		
Database Data Items	**Write**	**Read**	**Note**	**Write**	**Read**	**Note**	**Write**	**Read**	**Note**	**Write**	**Read**	**Note**
Project Data	●	●		●	●		●	●		●	●	
Name	●	●		●	●		●	●		●	●	
Description	●	●		●	●		●	●		●	●	
Project number	●	●		●	●		●	●		●	●	
State	●	●		●	●		●	●		●	●	
Date	●	●		●	●		●	●		●	●	
Designer	●	●		●	●		●	●		●	●	
Units	●	●		●	●		●	●		●	●	
Design specifications				●	●		●	●		●	●	
Alignment Data	●	●		●	●	**CBX-01**	●	●				
Basic PI data	●	●		●	●		●	●				
POB stationing	●	●		●	●		●	●				
Bridge Data	●	●		●	●	**CBX-02**	●	●		●	●	
Beginning and ending station	●	●		●	●		●	●				
Offset from CL to alignment	●		**GMD-01**	●	●		●	●				
Box girder Superstructures	●	●		●	●				**CSL-01**			
Cross sections	●	●		●	●							
Location	●	●		●	●							
By distance or percent	●	●		●	●							
From beginning or end	●	●		●	●							
Span number				●	●							
Longitudinal variation	●	●		●	●							
Linear	●	●		●	●							

Parabolic				●	●							
Jump	●	●		●	●							
Box girder Superstructures	●	●		●	●				**CSL-01**			
Section dimensions	●	●		●	●							
Full-type (19 parameters)	●	●	**GMD-02**	●	●							
Variant-type (5 parameters)				●	●							
Recall Sections				●	●							
Section properties (A, I)	●	●		●	●							
Material reference				●	●							
Open Girder Superstructures	●	●		●	●		●	●				
Section dimensions	●	●		●	●		●	●				
Rectangular	●	●		●	●		●	●				
Section properties (A, I)	●	●		●	●		●	●				
Material reference	●	●		●	●		●	●				
Left/Right barrier distances	●	●		●	●		●	●				
Supplemental thickness	●	●		●	●		●	●				
Sacrificial thickness	●	●		●	●							
Girders	●	●		●	●		●	●				
Beg/End spacings	●	●		●	●		●	●				
Normal measurement	●	●		●	●		●	●				
Skewed measurement	●	●		●	●		●	●				
Beg/End longitudinal offset	●	●		●	●		●	●				
Haunch dimensions	●	●		●	●		●	●				
Beg/End girder group transverse offset	●	●		●	●		●	●		●	●	
Normal measurement	●	●		●	●		●	●		●	●	
Skewed measurement	●	●		●	●		●	●		●	●	

Supports	●	●		●	●		●	●		●	●	
Type	●	●		●	●		●	●		●	●	
Abutments	●	●		●	●		●	●				
Bents	●	●		●	●		●	●		●	●	
Stationing	●	●		●	●		●	●				
Bearing	●	●		●	●					●	●	
Connection Left/Center/Right to				●	●					●	●	
Keyword (e.g., PIN, FIX)				●	●					●	●	
Stiffness matrix (Full)				●	●					●	●	
Multiple transverse bearing										●	●	
Spacings										●	●	
Beg/End bearing group transverse										●	●	
Normal measurement										●	●	
Skewed measurement										●	●	
Multiple lines of bearings				●	●					●	●	
Spacings				●	●					●	●	
Transverse Offset from CL bridge				●	●					●	●	
Normal measurement				●	●							
Skewed measurement				●	●					●	●	
Longitudinal offset from CL				●	●					●	●	
Intermediate hinges				●	●							
Stationing				●	●					●	●	
Bearing				●	●					●	●	
Connection to superstructure				●	●					●	●	
Keyword (e.g., PIN, FIX)				●	●					●	●	
Stiffness matrix (Full)				●	●					●	●	

Multiple Points of Bearing										●	●	
Offset from CL bridge										●	●	
Normal measurement										●	●	
Skewed measurement										●	●	
Girder Definition Data	●	●		●	●		●	●				
Cross sections	●	●		●	●		●	●	**CSL-02**			
Location	●	●		●	●		●	●				
By distance or percent	●	●		●	●		●	●				
From beginning or end	●	●		●	●		●	●				
Longitudinal variation	●	●		●	●		●	●				
Linear	●	●		●	●		●	●				
Parabolic	●	●		●	●		●	●				
Jump	●	●		●	●		●	●				
Section dimensions	●	●		●	●		●	●				
Flange type (I, Tee, etc.)	●	●		●	●		●	●				
Box beam type	●	●		●	●		●	●				
Rectangular	●	●		●	●		●	●				
U-Type	●	●		●	●		●	●				
Section group (I-Girder)	●	●		●	●		●	●				
Section member (AASHTO-I)	●	●		●	●		●	●				
Bent or Pier Definition Data	●	●	**GMD-03**	●	●	**CBX-03**				●	●	
Drop-cap bent	●	●	**GMD-04**	●	●					●	●	**RCP-01**
Length	●	●	**GMD-05**							●	●	
Width	●	●								●	●	
Depth	●	●		●	●					●	●	
Soffit-to-bearing distance				●	●							

Bearing-to-cap distance				●	●							
Left overhang										●	●	
Right overhang										●	●	
Cross sections	●	●		●	●					●	●	
Location	●	●		●	●					●	●	
By distance or percent	●	●		●	●					●	●	
From beginning or end	●	●		●	●					●	●	
Span number	●	●		●	●					●	●	
Longitudinal variation	●	●		●	●					●	●	
Linear	●	●		●	●					●	●	
Parabolic	●	●		●	●							
Jump	●	●		●	●							
Section dimensions	●	●		●	●					●	●	
Rectangular	●	●		●	●					●	●	
Elevation top left center	●	●		●	●					●	●	
Elevation top right center	●	●		●	●					●	●	
Offset from CL bridge										●	●	
Normal measurement												
Skewed measurement										●	●	
Integral bent	●	●	**GMD-06**	●	●							
List of Columns				●	●					●	●	
Reference name				●	●					●	●	
Spacings				●	●					●	●	
Length				●	●					●	●	
Type				●	●					●	●	
Dimensions for standard types				●	●					●	●	

Bottom support type				●	●					●	●	
Keyword (.e.g. PIN, FIX)				●	●					●	●	
Stiffness matrix (Full)				●	●					●	●	
Material reference				●	●					●	●	
Offset from CL bridge												
Normal measurement												
Skewed measurement												
Physical location in XYZ space	●	●								●	●	
Column Definition Data				●	●					●	●	
Type				●	●					●	●	
General (A, I)				●	●							
Circular				●	●					●	●	**RCP-02**
Rectangular				●	●					●	●	
Rectangular fillet				●	●							
Rectangular chamfer				●	●					●	●	
Rectangular bevel				●	●					●	●	
Hammerhead				●	●					●	●	
Cross Sections				●	●							
Location				●	●					●	●	
By distance				●	●					●	●	
Longitudinal variation				●	●					●	●	
Linear				●	●					●	●	
Parabolic				●	●					●	●	
Jump				●	●							
Abutment Definition Data	●	●	**GMD-07**	●	●							
Material Data				●	●					●	●	

Generic				●	●					●	●	
Elastic modulus				●	●					●	●	
Poisson’s ratio				●	●					●	●	
Density				●	●					●	●	
Thermal coefficient				●	●					●	●	
Yield stress				●	●					●	●	
Concrete				●	●					●	●	
Fci				●	●					●	●	
Fc				●	●					●	●	
Epcu				●	●					●	●	
Time-Dependent Concrete				●	●							
Humidity				●	●							
Age when shrinkage starts				●	●							
Age when concrete is placed				●	●							
Concrete type				●	●							
Curing method				●	●							
Weight type (e.g. Normal)				●	●							
Ultimate creep factor				●	●							
Ultimate shrinkage strain				●	●							
ACI a/b ratio				●	●							
CEB S-factor				●	●							

Notes

Table LB-2 GEOMATH (GMD)

Note #	**Description**
GMD-01	This is computed using the average distance of all abutments and piers from the alignment.
GMD-02	Top slab width is based on the roadway; if there isn’t a roadway then the width is based on the average abutment and pier widths.
GMD-03	All layout placement methods are supported.
GMD-04	Database drop-cap bents are generated for piers not associated with a box girder bridge.
GMD-05	This is computed using L1 and L2.
GMD-06	Database integral bents are generated for piers associated with a box girder bridge.
GMD-07	Abutments are a type of pier in GEOMATH.

Table LB-3 CONBOX (CBX)

Note #	**Description**
CBX-01	CONBOX works with a single alignment and it is always present.
CBX-02	CONBOX works with a single bridge and it is always present.
CBX-03	A default bent named “Typ2Col” is created at program startup.

Table LB-4 CONSPAN (CSL)

Note #	**Description**
CSL-01	Box Girder Bridges being referred to here are only cast-in-place Box Girders bridges being capable of designed by CONBOX.
CSL-02	Only uniform cross section is read into CONSPAN.

Table LB-5 RC-PIER (RCP)

Note #	**Description**
RCP-01	RC-PIER assumes cap top as straight line connected to ends of cap. This will be based on the elevation of first and last sections ignoring any intermediate sections.
RCP-02	RC-PIER can read only prismatic circular columns. It will not read columns with varying circular sections with height.

Chapter

T

Tutorial Sessions

This chapter consists of three tutorial sessions for RC-PIER$^{®}$. These tutorials are designed to familiarize the user with RC-PIER's capabilities through a "hands-on" approach.

Each tutorial takes you through an entire design session, screen-by-screen, for AASHTO Standard and LRFD design codes. To assist you in completing the tutorials, we have included the output generated from each tutorial.

Upon completing these tutorials, you should be familiar with most of the features in RC-PIER.

Chapter

T1

Two-Column Pier, Auto Load Generation (AASHTO LFD)

Two Column Pier, Auto Load Generation (AASHTO LFD)

Tutorial 1 demonstrates the basic features of RC-PIER®. This will take you through a RC-PIER design, step-by-step, illustrating the design of a project entry for AASHTO LFD. It also shows you how to auto generate loads.

The information for Tutorial 1 is contained in a RC-PIER input file called “Tutor1.rcp”. As an alternative to inputting all information manually, as directed by this tutorial, you can load this file and work on only specific sections. To load this file, select **Open** from the **File** menu (**File | Open**). Move to the RC-PIER\Examples subdirectory, if it is not already showing, and select the file “Tutor1.rcp”.

Diagrams

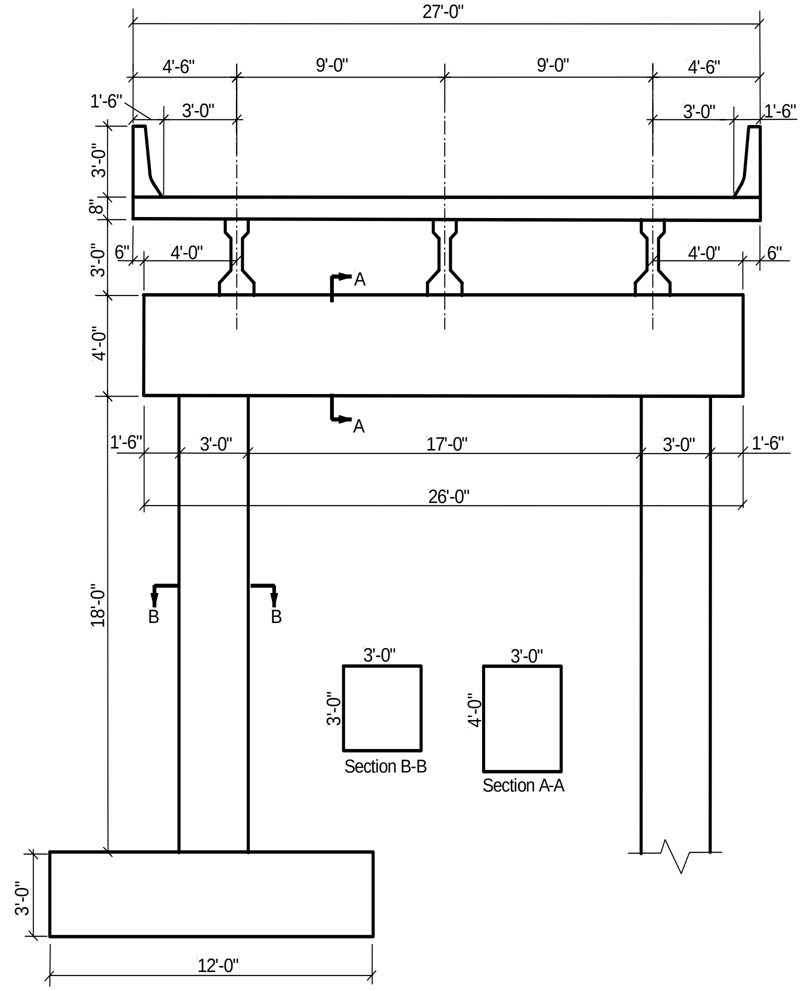
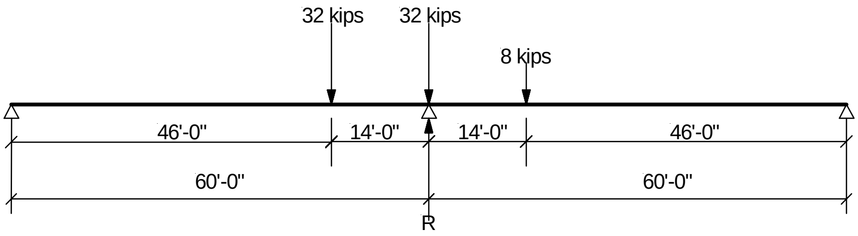
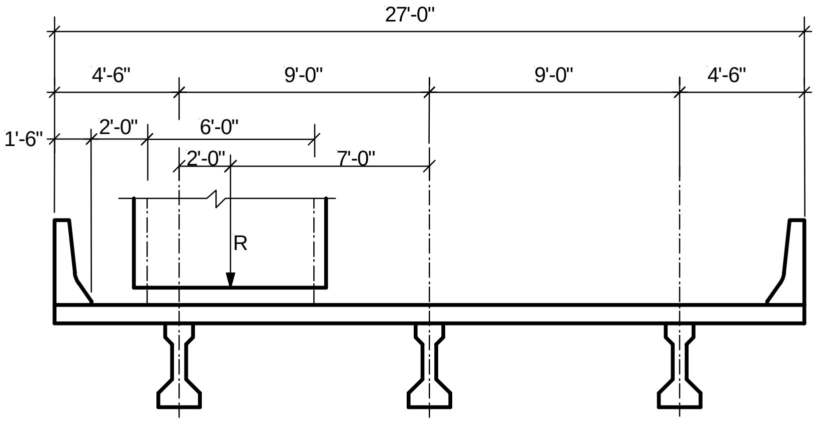


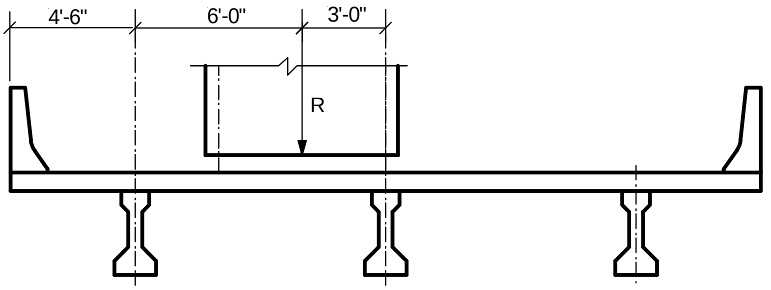
Figure T1-1 Elevation of Bridge Pier with Cross Section for Cap and Column



(a) Longitudinal View

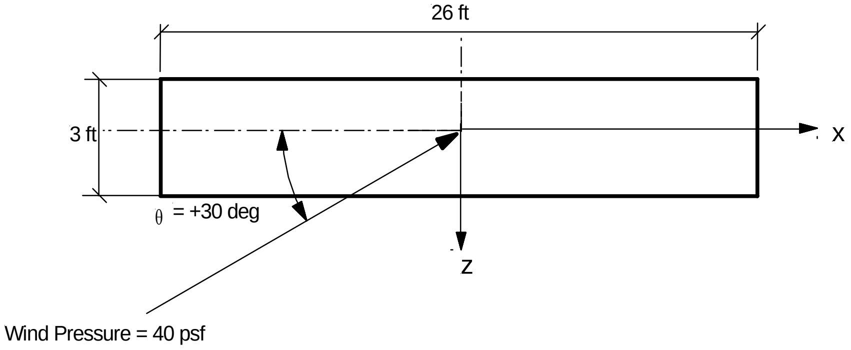


(b) Cases LL1 and LL4

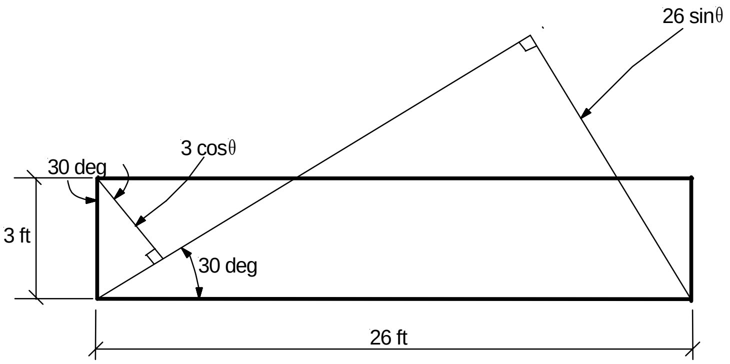


(c) Cases LL2 and LL3

Figure T1-2 Live Load Truck Positions



(a) Plan View of the Pier Cap where wind pressure acts on



(b) Calculation of projected area for wind pressure acts on a pier cap

Figure T1-3 Wind Load on the Substructure

Problem Data

Table T1-1

Concrete Strength	
Cap	f’c = 4000 psi
Columns	f’c = 4000 psi
Footings	f’c = 4000 psi
Modulus of Elasticity	E_c = 3834 ksi
Concrete Density	
Cap	ρ = 150 pcf
Columns	ρ = 150 pcf
Footings	ρ = 150 pcf
Steel Yield Strength	
Cap	f_y = 60 ksi
Columns	f_y = 60 ksi
Footings	f_y = 60 ksi
Modulus of Elasticity	E_s = 29000 ksi
Concrete Type	
Cap	Normal
Columns	Normal
Footings	Normal
Other Parameters	
Crack Control Factor	z = 170 kips/in
Pier Configuration	
Rectangular non-tapered multi-columns, with straight cap	
Superstructure Parameters	
Number of lanes	= 2
(Note that in this tutorial, the pier is only designed for 1 lane loaded)	
Beam Height	= 36 in
Beam Section Area	= 369 in^2
Beam Inertia Ixx	= 50980 in^4
Beam Inertia Iyy	= 5332.5 in^4
Beam Ycg	=15.83 in
Barrier Height	= 36 in
Slab Depth	= 8.0 in
Total number of spans	2
Span Information	
Bridge Overall Width, ft	27 ft
Curb to Curb Distance, ft	24 ft
Span Length, Span 1, ft	60 ft

Table T1-1

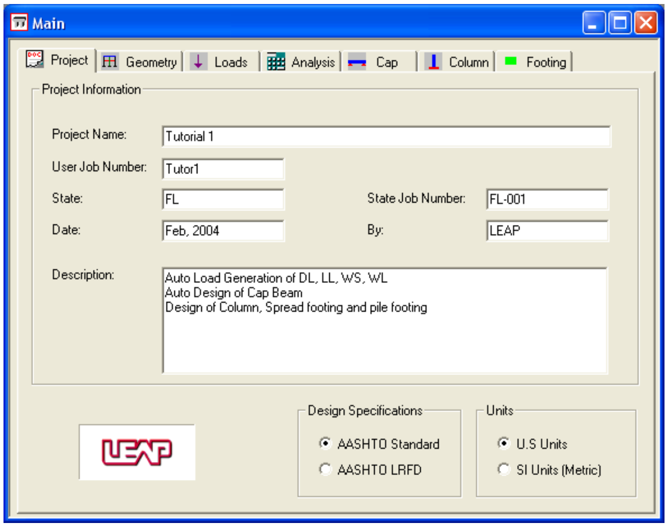
Span Length, Span 2, ft	60 ft
Substructure Parameters	
Cap: Length × Height × Depth	= 26 ft × 48 in × 36 in
Skew Angle	= 0 degrees
Start Elevation	= 22 ft
End Elevation	= 22 ft
Factor of Reduced Moment of Inertia	= 1.0 (non-cracked section)
Columns: Fixed at base	
Column Height	= 20 ft
Width × Depth	= 36 in × 36 in
Factor of Reduced Moment of Inertia:	= 1.0 (non-cracked section)
Bearings	= one line with no eccentricity
Loads	
Dead Load	
Self-weight	= 150 pcf
Slab	= 150 pcf
Girder Weight	= 150 pcf
Barrier Weight	= 600 plf each side
Total Barrier Weight	= 2 × 600 = 1200 plf
Future Wearing Surface Load	= 20 psf (20 × 27 = 540 plf)
Live Load	
HS20 Truck	
Wind on Structure	
Direction of wind	= + 30°
Elevation above which wind acts	= 0 ft
Trans. wind pressure on superstructure	= 41 psf
Longitudinal pressure on superstructure	= 12 psf
Wind pressure on cap	= 40 psf
Wind pressure on column	= 40 psf
Wind on Live Load	
Wind Angle	= +30°
Trans. wind pressure on live load	= 82 plf
Longitudinal wind pressure on live load	= 24 plf
Length of live load	= 60 ft
Footing Surcharge	
Footing	σ = 0.200 ksf

Starting the Tutorial

RC-PIER is installed in Program Files/LEAP Software/RC-PIER. Start RC-PIER by clicking **Start** > **Programs** > **LEAP Software** > **RC-PIER**.

Step 1

The *Project* screen will be displayed, as shown in Figure T1-4. Fill in the general project information. Select the **AASHTO Standard** option under Design Specifications and **U.S. Units** under Units.



Project Information

Project Name: Tutorial 1

User Job Number: Tutor1

State: FL

State Job Number: FL-001

Date: Feb, 2004

By: LEAP

Description: Auto Load Generation of DL, LL, WS, WL
Auto Design of Cap Beam
Design of Column, Spread footing and pile footing

Design Specifications
● AASHTO Standard
○ AASHTO LRFD

Units
● U.S Units
○ SI Units (Metric)

Figure T1-4 Project Tab Screen

Step 2

Click the **Geometry** tab to open the *Geometry* screen, as shown in Figure T1-5. Click and drag to rotate the model.

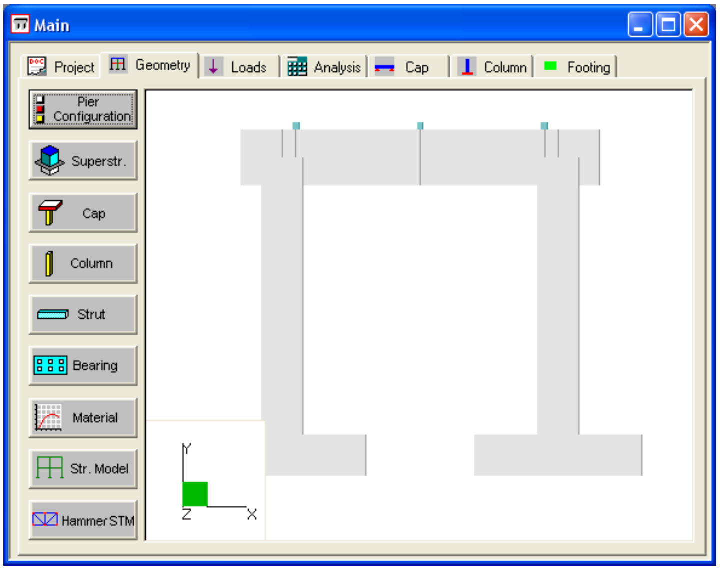
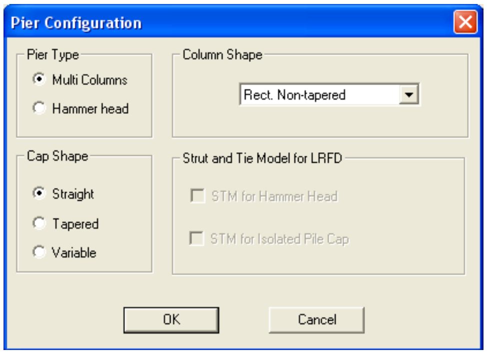


Figure T1-5 Geometry Tab Screen

Step 3

Click **Pier Configuration** to open the *Pier Configuration* screen, as shown in Figure T1-6. Under Pier Type, select **Multi-columns**, select **Straight** under Cap Shape, and **Rect. Non-tapered** from the Column Shape list. Click **OK**.



Pier Type
● Multi Columns
○ Hammer head

Column Shape: Rect. Non-tapered

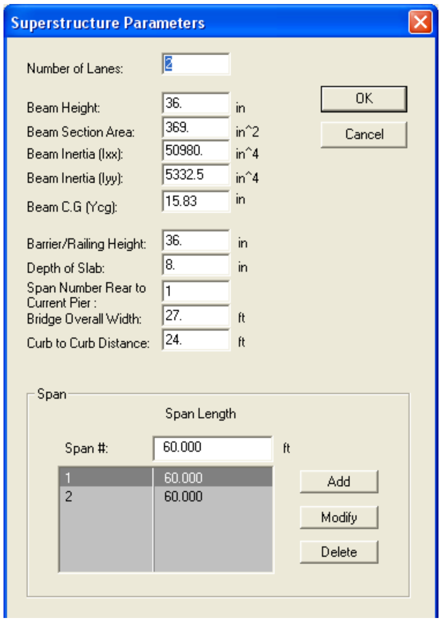
Cap Shape
● Straight
○ Tapered
○ Variable

Strut and Tie Model for LRFD
☐ STM for Hammer Head
☐ STM for Isolated Pile Cap

Figure T1-6 Pier Configuration Screen

Step 4

Click **Superstr**. to open the *Superstructure Parameters* screen. Input the values for the number of lanes, beam height and area, barrier/railing height, slab depth, total number of spans, and total length and width of spans, as shown in Figure T1-7, and then all add all span lengths. Click **OK** to the *Geometry* screen. Note that this screen is optional; however, it is required for auto load generation, as illustrated later in this tutorial.



Number of Lanes: 2

Beam Height: 36. in
Beam Section Area: 369. in^2
Beam Inertia (Ixx): 50980. in^4
Beam Inertia (Iyy): 5332.5 in^4
Beam C.G (Ycg): 15.83 in

Barrier/Railing Height: 36. in
Depth of Slab: 8. in
Span Number Rear to Current Pier : 1
Bridge Overall Width: 27. ft
Curb to Curb Distance: 24. ft

Span

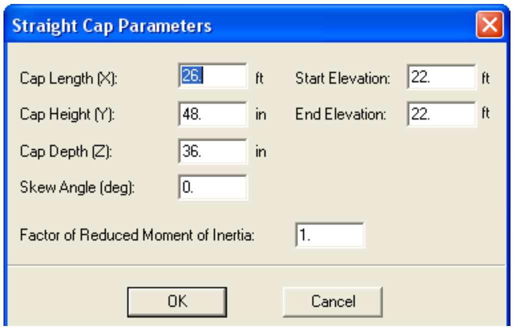
Span #: Span Length 60.000 ft

Span #	Span Length
1	60.000
2	60.000

Figure T1-7 Superstructure Parameters Screen

Step 5

Click **Cap** to open the *Straight Cap Parameters* screen, as shown in Figure T1-8. Enter **26** in the Cap Length field, **48** in the Cap Height field, **36** in the Cap Depth field, and **22** in the Start and End Elevation fields. Click **OK**.

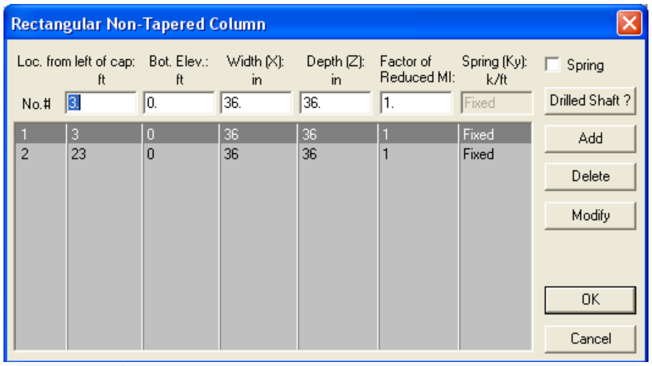


Cap Length (X): 26. ft
Cap Height (Y): 48. in
Cap Depth (Z): 36. in
Skew Angle (deg): 0.
Start Elevation: 22. ft
End Elevation: 22. ft
Factor of Reduced Moment of Inertia: 1.

Figure T1-8 Straight Cap Parameters Screen

Step 6

Click **Column** to open the *Rectangular Non-Tapered Column* screen, as shown in Figure T1-9. (This screen will be specific to the type of column shape selected.) For this tutorial input two columns, as follows.



No.#	Loc. from left of cap: ft	Bot. Elev.: ft	Width (X): in	Depth (Z): in	Factor of Reduced MI:	Spring (Ky): k/ft
	3.	0.	36.	36.	1.	Fixed
1	3	0	36	36	1	Fixed
2	23	0	36	36	1	Fixed

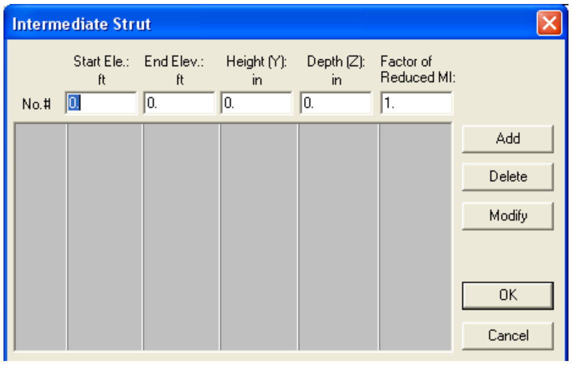
Figure T1-9 Rectangular Non-Tapered Column Screen

1. Enter **3** in the Loc from left of cap field, **0** in the Bot. Elev field, **36** in the Width and Depth fields, and **1** in the Factor of Reduced MI field. Click **Add**. The column will appear in the list as shown.
2. Input the second column. Enter **23** in the Loc from left of cap field, **0** in the Bot. Elev field, **36** in the Width and Depth fields, and **1** in the Factor of Reduced MI field. Click **Add** to add the column to the list.
3. Click **OK** and return to the *Geometry* screen.

To modify a column, highlight it in the list, make the necessary changes, and click **Modify**. To delete a column, highlight it and click **Delete**.

Step 7

Click **Strut** to open the *Intermediate Strut* screen, as shown in Figure T1-10. This screen is used to add intermediate struts to the pier structure; however, in this tutorial, modifications are not required. Click **Cancel**.

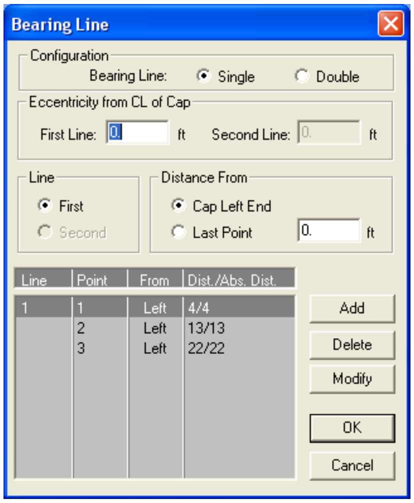


	Start Ele.: ft	End Elev.: ft	Height (Y): in	Depth (Z): in	Factor of Reduced MI:
No.#	0.	0.	0.	0.	1.

Figure T1-10 Intermediate Strut Screen

Step 8

Click **Bearing** to open the *Bearing Line* screen, as shown in Figure T1-11. This screen is used to define the configuration of the bearing line, eccentricity, and distance from left end of pier cap to individual bearings.



Configuration
Bearing Line: ● Single ○ Double

Eccentricity from CL of Cap
First Line: 0. ft Second Line: 0. ft

Line
● First
○ Second

Distance From
● Cap Left End
○ Last Point 0. ft

Line	Point	From	Dist./Abs. Dist.
1	1	Left	4/4
	2	Left	13/13
	3	Left	22/22

Figure T1-11 Bearing Line Screen

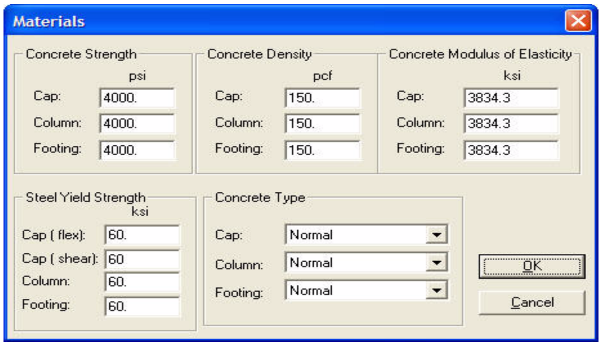
1. Select **Single** under Configuration. This specifies that there is only one bearing line on the pier.
2. Input the first bearing point by selecting the **Cap Left End** option and entering **4** in the text box under Distance From. Click **Add**. A bearing line will appear in the list on the screen.
3. Enter the second bearing point. Select **Cap Left End** under Distance From and enter **13.** Click **Add** to add the second bearing line to the list.
4. Repeat the above steps to add the third bearing point, enter **22** under Distance From. Click **Add**. When completed, your screen should resemble Figure T1-11. Click **OK**.

To modify a bearing line in the list, highlight it, make the appropriate changes, and click **Modify**. To delete a bearing line, highlight it and click **Delete**.

Note: The first bearing point must be measured from Cap Left End. For all other points, you can select the Last Point option. This allows you to input the same value multiple times; each new bearing spaced evenly from the previous bearing.

Step 9

Click **Material** to activate the *Materials* screen, as shown in Figure T1-12. This screen defines the strength, density, concrete modulus of elasticity, and reinforcing steel strength as well as the concrete type. Notice that the program defaults to certain values. You can override these values by typing over them. Input the values shown in figure and click **OK** to return to the *Geometry* screen.



	Concrete Strength psi	Concrete Density pcf	Concrete Modulus of Elasticity ksi
Cap:	4000.	150.	3834.3
Column:	4000.	150.	3834.3
Footing:	4000.	150.	3834.3

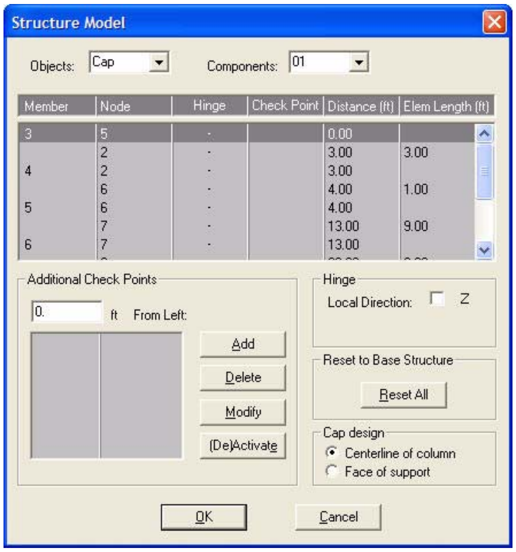
Steel Yield Strength ksi
Cap (flex): 60.
Cap (shear): 60
Column: 60.
Footing: 60.

Concrete Type
Cap: Normal
Column: Normal
Footing: Normal

Figure T1-12 Materials Screen

Step 10

Click **Str. Model** to open the *Structure Model* screen, as shown in Figure T1-13. Use this screen to keep track of all nodes of the pier structure, add or remove nodes to the pier structure for use as reference points (checkpoints), and define hinges at existing points. This screen is not used in this tutorial. Click **OK** to return to the *Geometry* screen.



Objects: Cap
Components: 01

Member	Node	Hinge	Check Point	Distance (ft)	Elem Length (ft)
3	5	-		0.00	
	2	-		3.00	3.00
4	2	-		3.00	
	6	-		4.00	1.00
5	6	-		4.00	
	7	-		13.00	9.00
6	7	-		13.00	

Additional Check Points
0. ft From Left:

Hinge
Local Direction: ☐ Z

Cap design
● Centerline of column
○ Face of support

Figure T1-13 Structure Model Screen

Step 11

Hammer STM is only available for hammerhead structures (LRFD). This option is not used in Tutorial 1.

Step 12

Select **Image** from the **Show** menu to activate the *Image* screen (or the corresponding icon on the toolbar at the top of the screen). A 3-D image of the structure will be displayed on the screen, as shown in Figure T1-14.

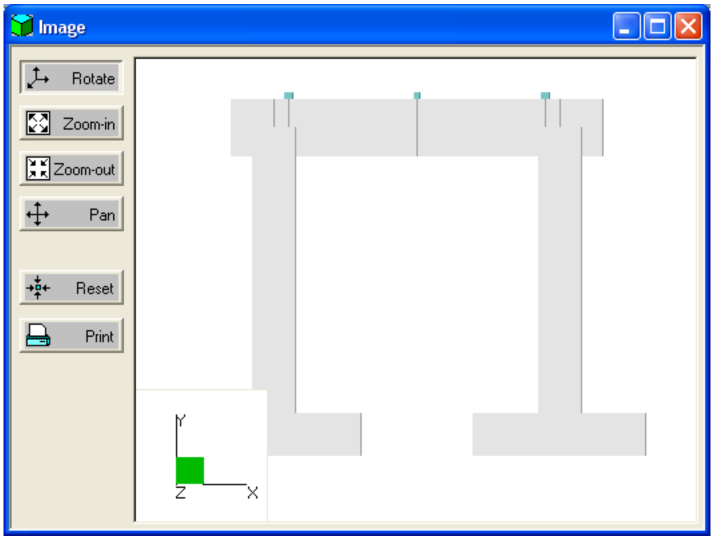


Figure T1-14 Image Screen

Use the buttons on the left of the screen to manipulate your view of the image (e.g., rotate, pan, zoom-in or out). Experiment with the buttons to become more familiar with their functions. Once you have become familiar with this screen, close or minimize the screen and return to the *Geometry* screen.

Step 13

Select **Model** from the **Show** menu to bring up the *Model* screen (or its corresponding icon on the toolbar at the top of the screen), as shown in Figure T1-15. A 3-D model of the nodes, element number, etc will be displayed on the screen.

The type of model displayed depends on which characteristics you select from the check boxes at the top of the screen (Node Number, Member Number, and Checkpoints). Use the buttons on the left side of the screen to manipulate your view of the model (e.g., rotate, pan, zoom-in or out). Experiment with these buttons to become familiar with their functions. Once you have become familiar with this screen, close or minimize the screen and return to the *Geometry* screen.

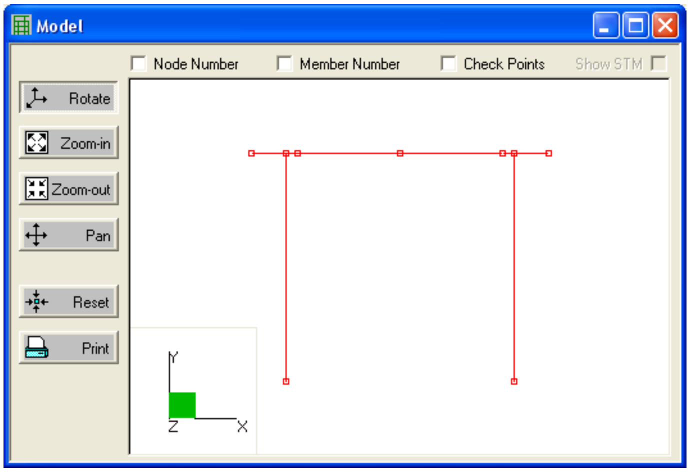
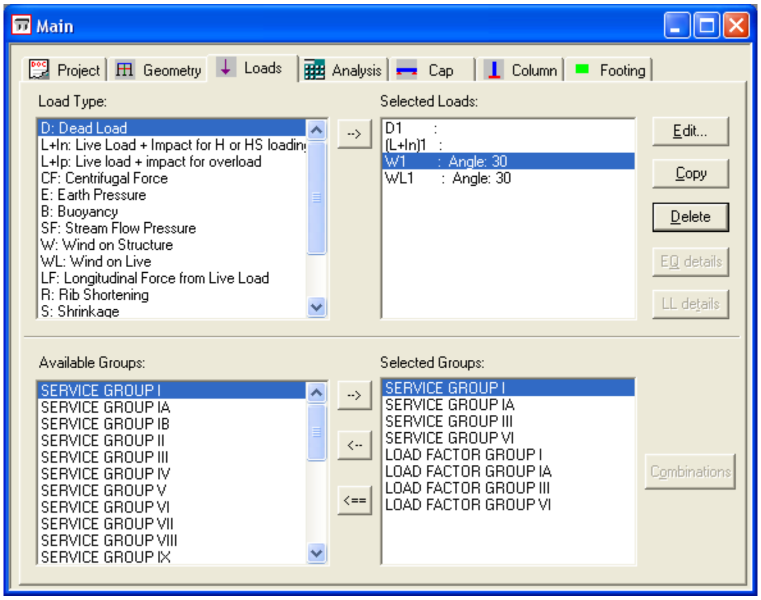


Figure T1-15 Model Screen

Step 14

Click the **Loads** tab to display the *Loads* screen, as shown in Figure T1-16. This is where you enter all load information.



Load Type:

D: Dead Load
L+In: Live Load + Impact for H or HS loadin
L+Ip: Live load + impact for overload
CF: Centrifugal Force
E: Earth Pressure
B: Buoyancy
SF: Stream Flow Pressure
W: Wind on Structure
WL: Wind on Live
LF: Longitudinal Force from Live Load
R: Rib Shortening
S: Shrinkage

Selected Loads:

D1 :
(L+In)1 :
W1 : Angle: 30
WL1 : Angle: 30

Available Groups:

SERVICE GROUP I
SERVICE GROUP IA
SERVICE GROUP IB
SERVICE GROUP II
SERVICE GROUP III
SERVICE GROUP IV
SERVICE GROUP V
SERVICE GROUP VI
SERVICE GROUP VII
SERVICE GROUP VIII
SERVICE GROUP IX

Selected Groups:

SERVICE GROUP I
SERVICE GROUP IA
SERVICE GROUP III
SERVICE GROUP VI
LOAD FACTOR GROUP I
LOAD FACTOR GROUP IA
LOAD FACTOR GROUP III
LOAD FACTOR GROUP VI

Figure T1-16 Loads Tab Screen

Notice there are two lists of loads: Load Types and Available Groups. Load Types are preset individual loads that you can add to the pier structure for calculations. Available groups are preset load combinations that can be added to the pier structure. First, add the load types (shown in Figure T1-16) to the Selected Load list:

1. Highlight **D - Dead Load** in the list under Load Type.
2. Click the right-arrow button (–>). The load type will appear in the list under Selected Loads.
3. Repeat the above steps until all the required loads have been entered, as shown in Figure T1-16.

Next, add the load groups to the Selected Groups list:

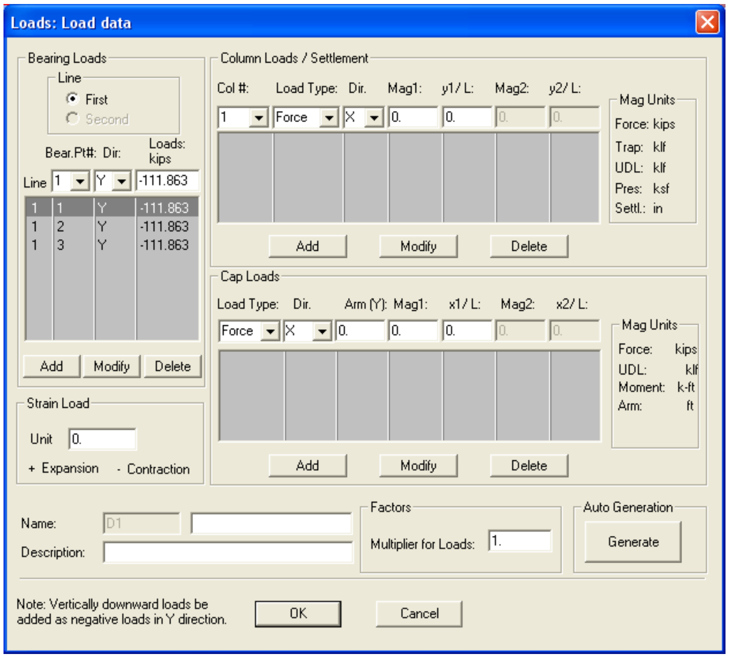
1. Highlight **Service Group I** in the list under Available Groups.
2. Click the right-arrow button (–>). The load group will appear in the list under Selected Groups.
3. Repeat the above steps until all required load groups are entered, as shown in Figure T1-16.

Remove a selected group from the Selected Groups list by clicking the left-arrow button (<-). To remove all groups from the Selected Groups list, click the <== button. To define groups, select the **Load Groups/Limit States** item from the **Libraries** menu.

Step 15

Auto-generate loads for dead load, live load, wind load on structure, and wind load on live load.

1. Highlight **D1** in the Selected Loads list and click **Edit**. The *Loads: Load Data* screen will display, as shown in Figure T1-17.



Line: ● First ○ Second

Line	Bear.Pt#:	Dir:	Loads: kips
1	1	Y	-111.863
1	2	Y	-111.863
1	3	Y	-111.863

Col #: 1 Load Type: Force Dir.: X Mag1: 0. y1/L: 0. Mag2: 0. y2/L: 0.

Mag Units: Force: kips Trap: klf UDL: klf Pres: ksf Settl.: in

Cap Loads: Load Type: Force Dir.: X Arm (Y): 0. Mag1: 0. x1/L: 0. Mag2: 0. x2/L: 0.

Mag Units: Force: kips UDL: klf Moment: k-ft Arm: ft

Strain Load: Unit 0. + Expansion - Contraction

Name: D1

Description:

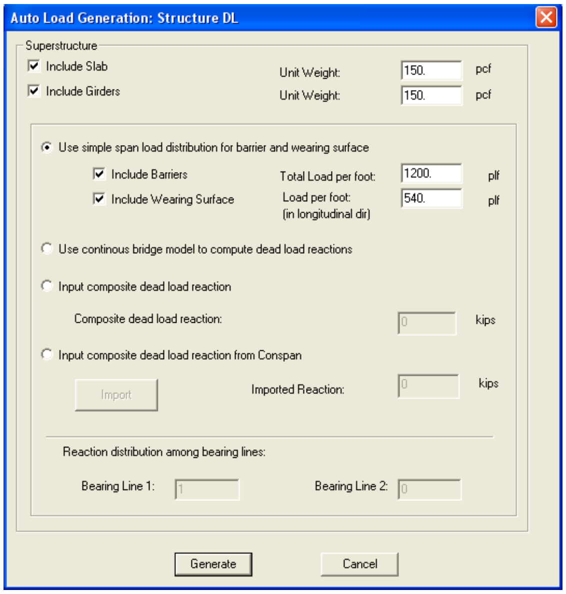
Multiplier for Loads: 1.

Note: Vertically downward loads be added as negative loads in Y direction.

Figure T1-17 Loads: Load Data Screen

2. Click **Generate** to bring up the *Auto Load Generation: Structure DL* screen, as shown in Figure T1-18. (Note that the auto load generation screen will be specific to the load type selected).

3. Select the check boxes to include slab and girders.
4. Enter **150** in the Slab and Girder Unit Weight fields.
5. Select the option “use simple span distribution for barrier and wearing surface”.
6. Check the boxes for barriers and wearing surface.
7. Specify 1200 in the total load per foot field for barriers, and **540** in the Load per foot field for wearing surface.



☑ Include Slab Unit Weight: 150. pcf

☑ Include Girders Unit Weight: 150. pcf

● Use simple span load distribution for barrier and wearing surface

☑ Include Barriers Total Load per foot: 1200. plf

☑ Include Wearing Surface Load per foot: (in longitudinal dir) 540. plf

○ Use continous bridge model to compute dead load reactions

○ Input composite dead load reaction

Composite dead load reaction: 0 kips

○ Input composite dead load reaction from Conspan

Imported Reaction: 0 kips

Reaction distribution among bearing lines:

Bearing Line 1: 1 Bearing Line 2: 0

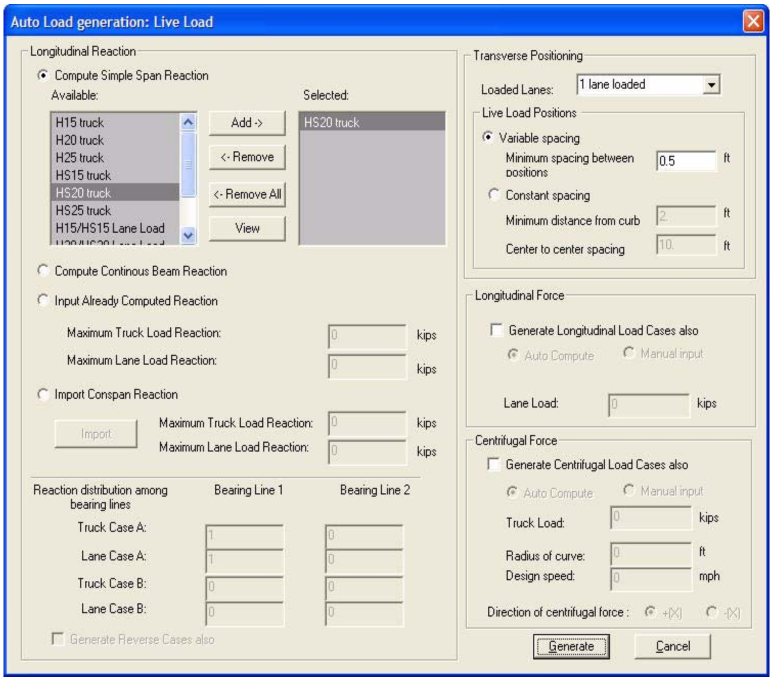
Figure T1-18 Auto Load Generation: Structure DL Screen

8. Click **Generate**. The program will automatically generate the loads and return to the *Loads: Load Data* screen.
9. Click **OK** to return to the *Loads* screen.

Step 16

Auto generate the live load:

1. Highlight **(L+In)1** in the Selected Loads list and click **Edit** to bring up the *Loads: Load Data* screen.
2. Click **Generate** to activate the *Auto Load Generation: Live Load* screen, as shown in Figure T1-20.



Auto Load generation: Live Load

Longitudinal Reaction

● Compute Simple Span Reaction

Available: H15 truck, H20 truck, H25 truck, HS15 truck, HS20 truck, HS25 truck, H15/HS15 Lane Load

Selected: HS20 truck

○ Compute Continous Beam Reaction

○ Input Already Computed Reaction

Maximum Truck Load Reaction: 0 kips

Maximum Lane Load Reaction: 0 kips

○ Import Conspan Reaction

Maximum Truck Load Reaction: 0 kips

Maximum Lane Load Reaction: 0 kips

Reaction distribution among bearing lines	Bearing Line 1	Bearing Line 2
Truck Case A:	1	0
Lane Case A:	1	0
Truck Case B:	0	0
Lane Case B:	0	0

☐ Generate Reverse Cases also

Transverse Positioning

Loaded Lanes: 1 lane loaded

Live Load Positions

● Variable spacing

Minimum spacing between positions 0.5 ft

○ Constant spacing

Minimum distance from curb 2. ft

Center to center spacing 10. ft

Longitudinal Force

☐ Generate Longitudinal Load Cases also

● Auto Compute ○ Manual input

Lane Load: 0 kips

Centrifugal Force

☐ Generate Centrifugal Load Cases also

● Auto Compute ○ Manual input

Truck Load: 0 kips

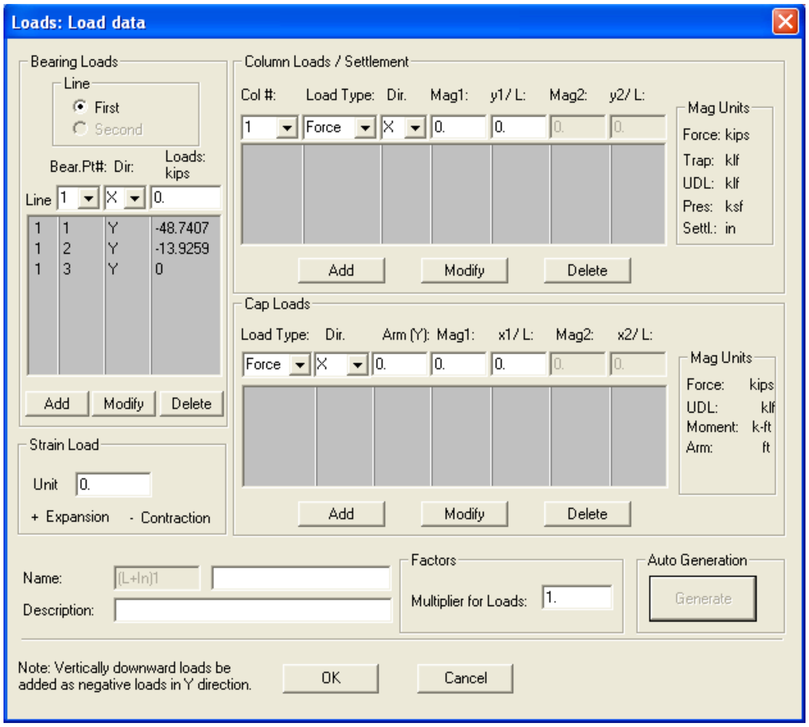
Radius of curve: 0 ft

Design speed: 0 mph

Direction of centrifugal force : ● +[X] ○ -[X]

Figure T1-19 Auto Load Generation: Live Load Screen - Prior to Auto Generation

3. In the longitudinal reaction area, select the option “compute simple span longitudinal reaction using following trucks”.
4. Highlight **HS20 Truck** under Available and click **Add** to add it to the list under Selected.
5. In the “Transverse Positioning” area select **1 lane loaded** from the Loaded Lanes list.
6. Select the **Variable Spacing Method** option and input **0.5** in the Minimum Spacing Between Locations text box.
7. Click **Generate**. The program will automatically generate the loads and return to the *Loads: Load Data* screen.



Line: 1, X, 0.

1	1	Y	-48.7407
1	2	Y	-13.9259
1	3	Y	0

Column Loads: 1, Force, X, 0., 0., 0., 0.

Cap Loads: Force, X, 0., 0., 0., 0., 0.

Strain Load Unit: 0.

Name: (L+In)1

Description:

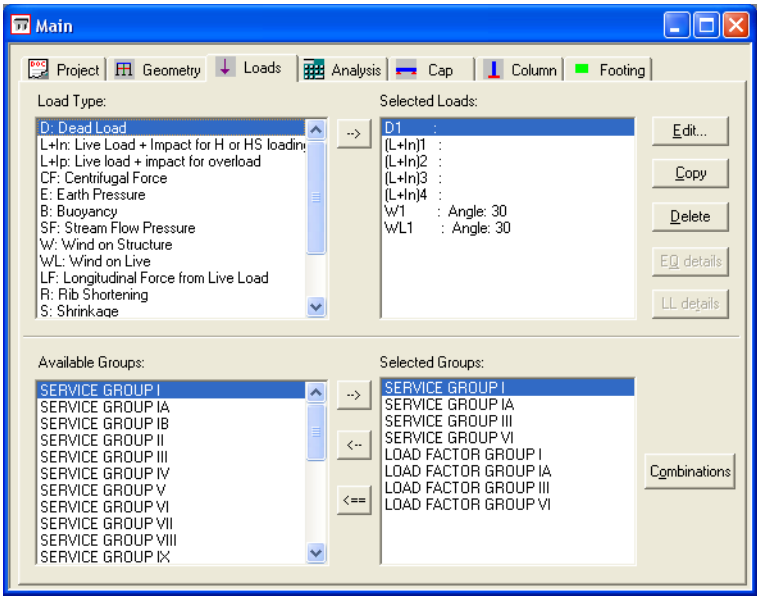
Multiplier for Loads: 1.

Note: Vertically downward loads be added as negative loads in Y direction.

Figure T1-20 Auto Load Generation: Live Load Screen - Post to Auto Generation

8. If desired, input a name in the Name text box (e.g., llcase1) and a description in the Description text box.
9. Click **OK** and return to the *Loads* screen.

Note that when you return to the *Loads* screen after generating the live loads, RC-PIER adds (L+In)2, (L+In)3, and (L+In)4 load cases to the list under Selected Loads. The *Loads* screen will look similar to Figure T1-21.



Load Type:
D: Dead Load
L+In: Live Load + Impact for H or HS loadin
L+Ip: Live load + impact for overload
CF: Centrifugal Force
E: Earth Pressure
B: Buoyancy
SF: Stream Flow Pressure
W: Wind on Structure
WL: Wind on Live
LF: Longitudinal Force from Live Load
R: Rib Shortening
S: Shrinkage

Selected Loads:
D1 :
(L+In)1 :
(L+In)2 :
(L+In)3 :
(L+In)4 :
W1 : Angle: 30
WL1 : Angle: 30

Available Groups:
SERVICE GROUP I
SERVICE GROUP IA
SERVICE GROUP IB
SERVICE GROUP II
SERVICE GROUP III
SERVICE GROUP IV
SERVICE GROUP V
SERVICE GROUP VI
SERVICE GROUP VII
SERVICE GROUP VIII
SERVICE GROUP IX

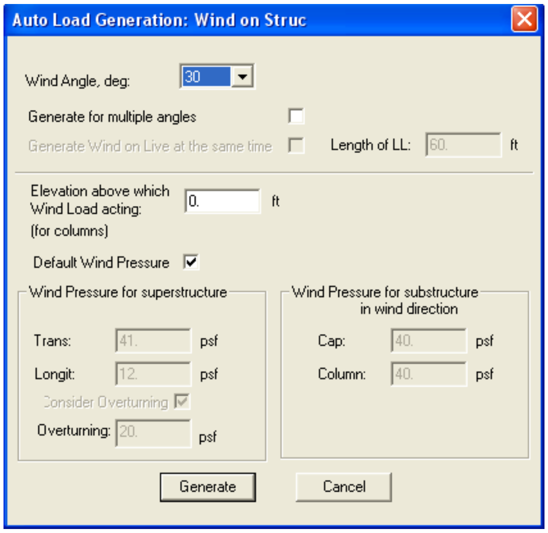
Selected Groups:
SERVICE GROUP I
SERVICE GROUP IA
SERVICE GROUP III
SERVICE GROUP VI
LOAD FACTOR GROUP I
LOAD FACTOR GROUP IA
LOAD FACTOR GROUP III
LOAD FACTOR GROUP VI

Figure T1-21 Loads Screen - After Generating the Live Loads

Step 17

Now, auto generate the Wind Load on Structure:

1. Highlight **W1** in the Selected Loads list and click **Edit** to bring up the *Loads: Load Data* screen.
2. Click **Generate** to open the *Auto Load Generation: Wind on Structure* screen, as shown in Figure T1-22.



Auto Load Generation: Wind on Struc

Wind Angle, deg: 30

Generate for multiple angles ☐

Generate Wind on Live at the same time ☐ Length of LL: 60. ft

Elevation above which Wind Load acting: (for columns) 0. ft

Default Wind Pressure ☑

Wind Pressure for superstructure

Trans: 41. psf

Longit: 12. psf

Consider Overturning ☑

Overturning: 20. psf

Wind Pressure for substructure in wind direction

Cap: 40. psf

Column: 40. psf

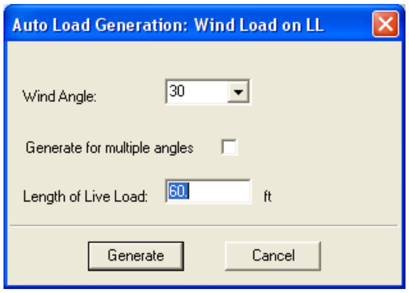
Figure T1-22 Auto Load Generation: Wind on Structure Screen

3. Select **30** from the Wind Angle list and enter **0** in the Elevation Above Which Wind Load Acting field.
4. Select the **Default Wind Pressure** check box. (Note that when this option is selected, the remaining fields are grayed out.)
5. Click **Generate**. The program automatically generates the loads and returns to the *Loads: Load Data* screen.
6. Click **OK** and return to the *Loads* screen.

Step 18

Auto generate the Wind Load on Live Load:

1. Highlight **WL1** in the Selected Loads list and click **Edit** to bring up the *Loads: Load Data* screen.
2. Click **Generate** to open the *Auto Load Generation: Wind on Live Load* screen, as shown in Figure T1-23.



Auto Load Generation: Wind Load on LL

Wind Angle: 30

Generate for multiple angles ☐

Length of Live Load: 60. ft

Figure T1-23 Auto Load Generation: Wind on Live Load Screen

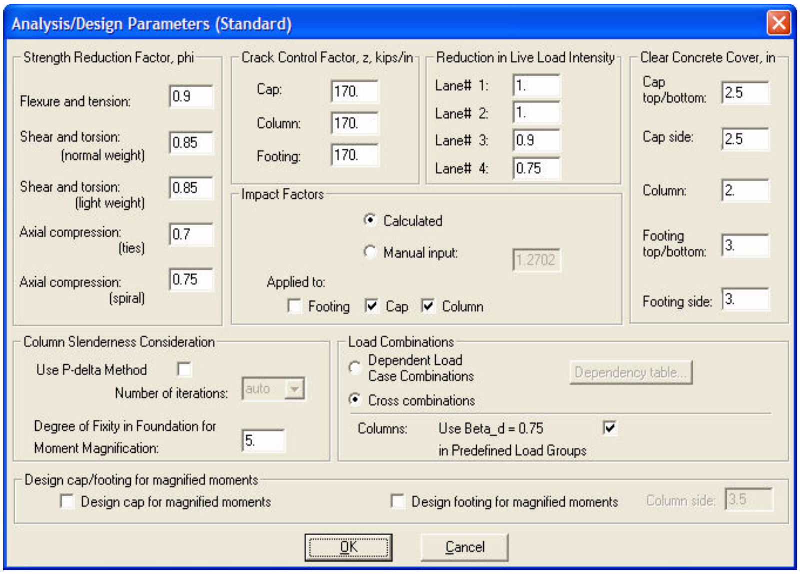
3. Select **30** from the Wind Angle list and leave **60** in the Length of Live Load field.
4. Click **Generate**. The program automatically generates the loads.

5. Click **OK** and return to the *Loads* screen.

Step 19

Click the **Analysis** tab to activate the *Analysis* screen. This screen is used to perform an analysis and also specify various factors relating to the analysis and design.

Click **A/D Parameters** to open the *Analysis/Design Parameters (Standard)* screen and input the values as shown in Figure T1-24. Click **OK** and return to the *Analysis* screen.



Strength Reduction Factor, phi
Flexure and tension: 0.9
Shear and torsion: (normal weight) 0.85
Shear and torsion: (light weight) 0.85
Axial compression: (ties) 0.7
Axial compression: (spiral) 0.75

Crack Control Factor, z, kips/in
Cap: 170.
Column: 170.
Footing: 170.

Reduction in Live Load Intensity
Lane# 1: 1.
Lane# 2: 1.
Lane# 3: 0.9
Lane# 4: 0.75

Clear Concrete Cover, in
Cap top/bottom: 2.5
Cap side: 2.5
Column: 2.
Footing top/bottom: 3.
Footing side: 3.

Impact Factors
● Calculated
○ Manual input: 1.2702
Applied to: ☐ Footing ☑ Cap ☑ Column

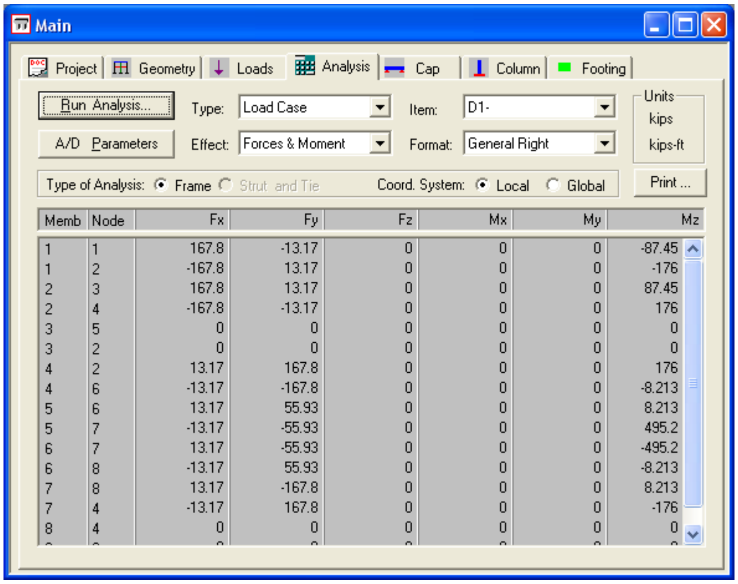
Column Slenderness Consideration
Use P-delta Method ☐
Number of iterations: auto
Degree of Fixity in Foundation for Moment Magnification: 5.

Load Combinations
○ Dependent Load Case Combinations
● Cross combinations
Columns: Use Beta_d = 0.75 in Predefined Load Groups ☑

Design cap/footing for magnified moments
☐ Design cap for magnified moments
☐ Design footing for magnified moments
Column side: 3.5

Figure T1-24 Analysis/Design Parameters Screen

Click **Run Analysis** to perform the analysis for the pier structure based on all the data entered up to this point. The results will appear on the screen, as shown in Figure T1-25. If necessary, use the scroll bar on the right side of the screen to view all the results. Specify the type of results to view by using the lists at the top of the screen.



Type: Load Case
Item: D1-
Effect: Forces & Moment
Format: General Right
Units kips kips-ft
Type of Analysis: ● Frame ○ Strut and Tie
Coord. System: ● Local ○ Global

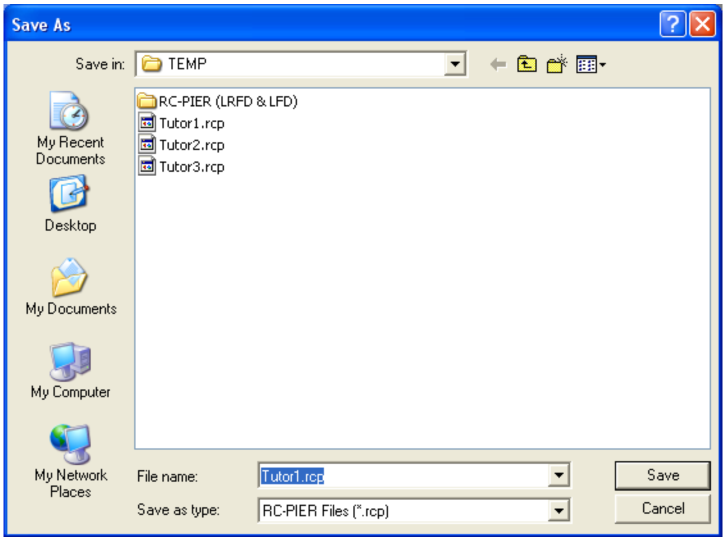
Memb	Node	Fx	Fy	Fz	Mx	My	Mz
1	1	167.8	-13.17	0	0	0	-87.45
1	2	-167.8	13.17	0	0	0	-176
2	3	167.8	13.17	0	0	0	87.45
2	4	-167.8	-13.17	0	0	0	176
3	5	0	0	0	0	0	0
3	2	0	0	0	0	0	0
4	2	13.17	167.8	0	0	0	176
4	6	-13.17	-167.8	0	0	0	-8.213
5	6	13.17	55.93	0	0	0	8.213
5	7	-13.17	-55.93	0	0	0	495.2
6	7	13.17	-55.93	0	0	0	-495.2
6	8	-13.17	55.93	0	0	0	-8.213
7	8	13.17	-167.8	0	0	0	8.213
7	4	-13.17	167.8	0	0	0	-176
8	4	0	0	0	0	0	0

Figure T1-25 Analysis Tab Screen (After Analysis is Performed)

Print the analysis results by right-clicking in the results area of the *Analysis* screen and selecting the **Print**.

Step 20

Save your project. Select **Save As** from the **File** menu. The *Save As* screen, will open, as shown in Figure T1-26. Enter a name for the file in the File Name field (e.g., Mytutor1). The default extension is “*.rcp”. Click **Save**.



Save in: TEMP

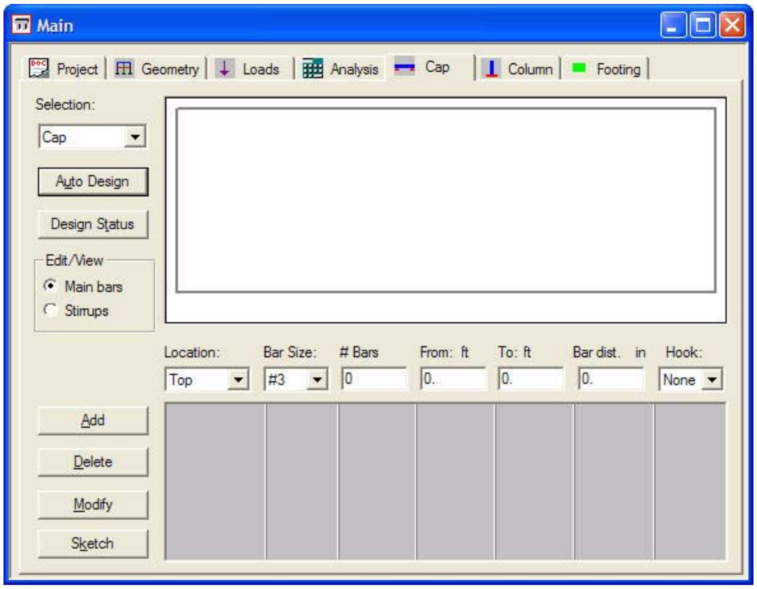
RC-PIER (LRFD & LFD)
Tutor1.rcp
Tutor2.rcp
Tutor3.rcp

File name: Tutor1.rcp
Save as type: RC-PIER Files (*.rcp)

Figure T1-26 Save As Screen

Step 21

Click the **Cap** tab to open the *Cap* screen, as shown in Figure T1-27. Use this screen to have RC-PIER design the cap. Clicking **Auto Design** or you can manually input the cap design. The following steps illustrate the auto design feature.



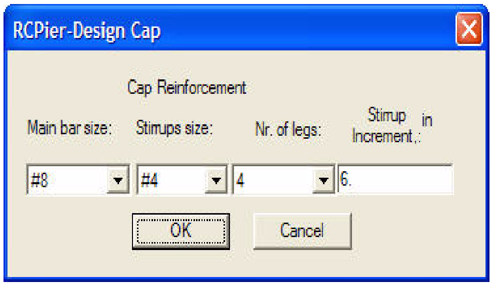
Selection: Cap

● Main bars
○ Stirrups

Location: Top
Bar Size: #3
Bars 0
From: ft 0.
To: ft 0.
Bar dist. in 0.
Hook: None

Figure T1-27 Cap Tab Screen

1. Select **Cap** from the Selection list.
2. Click **Auto Design** to bring up the *Design Cap* screen, as shown in Figure T1-28.



Main bar size: #8

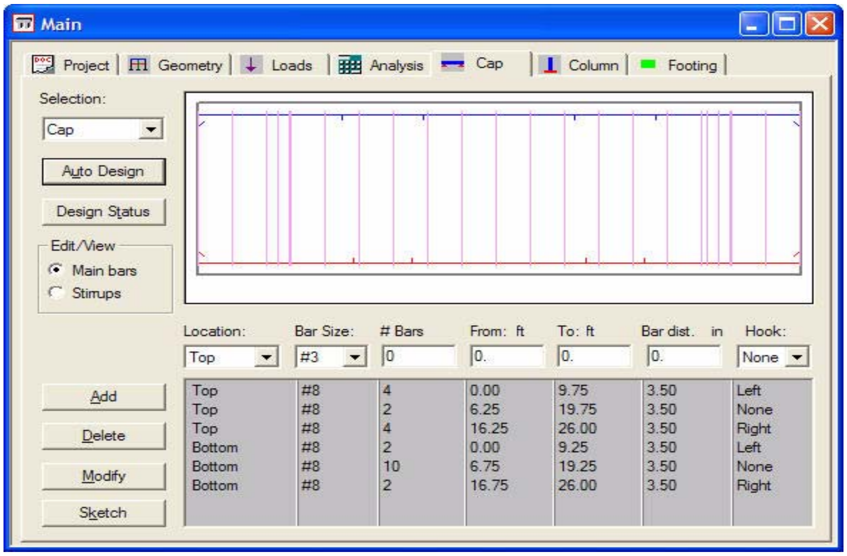
Stirrups size: #4

Nr. of legs: 4

Stirrup Increment.: 6. in

Figure T1-28 Design Cap Screen

3. Select **#8** from the Main Bar Size list, **#4** from the Stirrup Size list, **2** from the nlegs list and specify 6 inch in the spacing increment box.
4. Click **OK**.
5. The *Design Status - Cap* screen will immediately display. Click **Close** (or the **X** in the top right corner of the screen) to exit this screen and return to the *Cap* screen.
6. The generated cap design main reinforcement will appear on the screen, as shown in Figure T1-29.



Selection: Cap

● Main bars
○ Stirrups

Location: Top

Bar Size: #3

Bars: 0

From: ft 0.

To: ft 0.

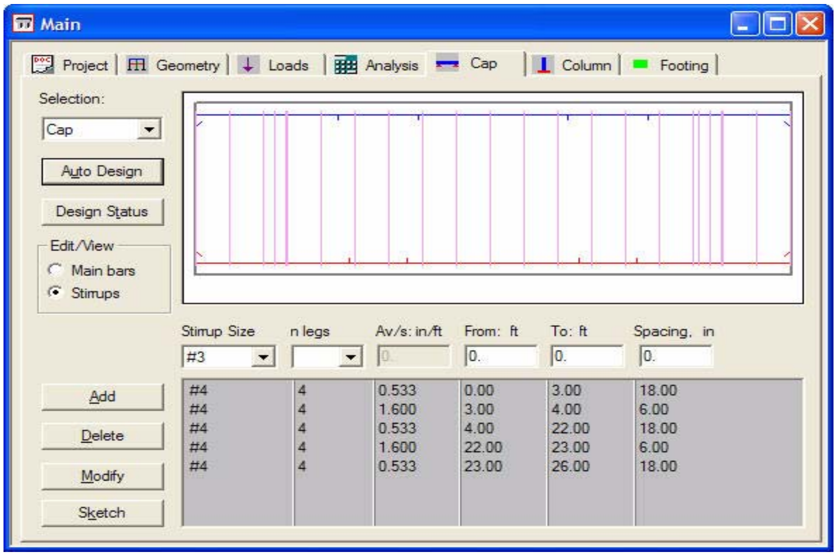
Bar dist. in 0.

Hook: None

Location:	Bar Size:	# Bars	From: ft	To: ft	Bar dist. in	Hook:
Top	#8	4	0.00	9.75	3.50	Left
Top	#8	2	6.25	19.75	3.50	None
Top	#8	4	16.25	26.00	3.50	Right
Bottom	#8	2	0.00	9.25	3.50	Left
Bottom	#8	10	6.75	19.25	3.50	None
Bottom	#8	2	16.75	26.00	3.50	Right

Figure T1-29 Cap Tab Screen - Showing Cap Main Steel

7. To see the shear stirrups design, choose the option to view stirrups, as shown in Figure T1-30..



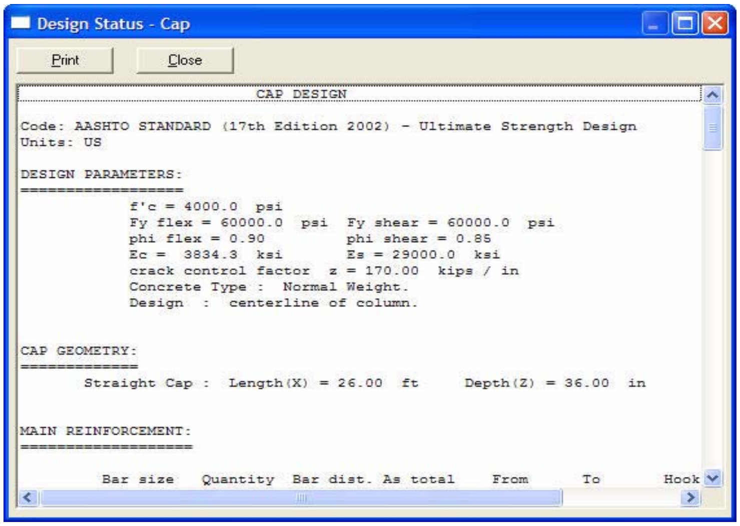
Selection: Cap

Edit/View
○ Main bars
● Stirrups

Stirrup Size	n legs	Av/s: in/ft	From: ft	To: ft	Spacing, in
#3		0.	0.	0.	0.
#4	4	0.533	0.00	3.00	18.00
#4	4	1.600	3.00	4.00	6.00
#4	4	0.533	4.00	22.00	18.00
#4	4	1.600	22.00	23.00	6.00
#4	4	0.533	23.00	26.00	18.00

Figure T1-30 Cap Tab Screen - Showing Stirrup

To see a design summary of the cap, click **Design Status** to display the cap summary, as shown in Figure T1-31. Click **Print** to print the design summary.



```
                              CAP DESIGN

Code: AASHTO STANDARD (17th Edition 2002) - Ultimate Strength Design
Units: US

DESIGN PARAMETERS:
==================
            f'c = 4000.0  psi
            Fy flex = 60000.0  psi  Fy shear = 60000.0  psi
            phi flex = 0.90         phi shear = 0.85
            Ec =  3834.3  ksi       Es = 29000.0  ksi
            crack control factor  z = 170.00  kips / in
            Concrete Type :  Normal Weight.
            Design  :  centerline of column.


CAP GEOMETRY:
=============
       Straight Cap :  Length(X) = 26.00  ft     Depth(Z) = 36.00  in


MAIN REINFORCEMENT:
===================


         Bar size   Quantity  Bar dist. As total    From      To        Hook
```

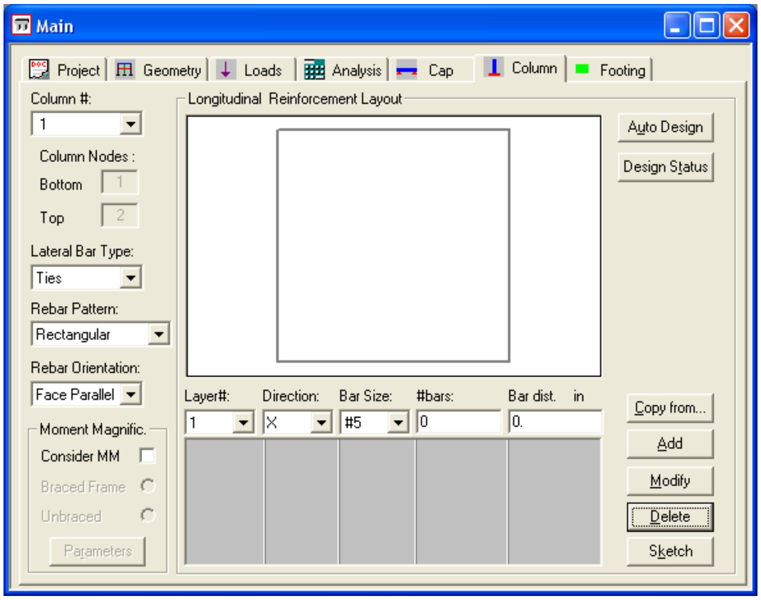
Figure T1-31 Design Status - Cap Screen

To see a graphical representation of the cap, click **Sketch**.

Note: The printout for shear/torsion design is given in terms of the sides of a section. For the section other than the start or end of a span, it has two sides: left and right. The start and end of the span have only one side.

Step 22

Click the **Column** tab to open the *Column* screen, as shown in Figure T1-32. Either manually input the column reinforcement or have the program automatically design it. For this tutorial, the Auto Design feature is used.



Column #: 1

Column Nodes :
Bottom 1
Top 2

Lateral Bar Type: Ties

Rebar Pattern: Rectangular

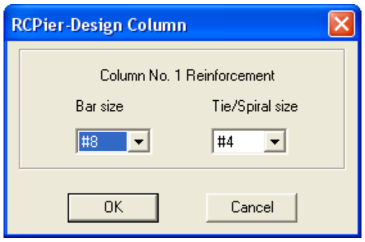
Rebar Orientation: Face Parallel

Consider MM ☐

Layer#:	Direction:	Bar Size:	#bars:	Bar dist. in
1	X	#5	0	0.

Figure T1-32 Column Tab Screen

1. Select **1** from the Column# list and **Ties** from the Lateral Bar Type list.
2. Click **Auto Design** to bring up the *Design Column* screen, as shown in Figure T1-33.



Column No. 1 Reinforcement

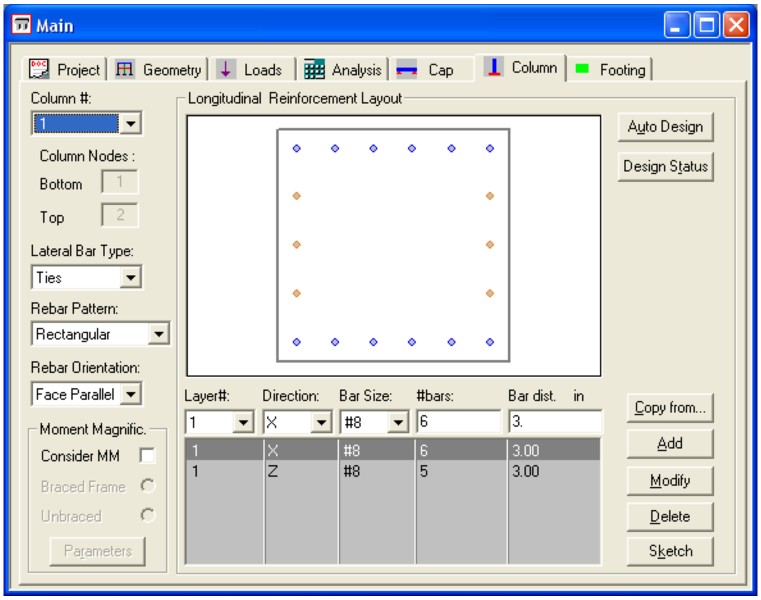
Bar size: #8

Tie/Spiral size: #4

Figure T1-33 Design Column Screen

3. Select **#8** from the Bar Size list and **#4** from the Stirrup Size list.
4. Click **OK**.
5. The *Design Status* **-** *Column* screen will immediately display. Click **Close** or the **X** in the top right corner of the screen to exit this screen and return to the Column screen.

6. The column design will appear on the screen, as shown in Figure T1-34.



Column #: 1

Column Nodes :
Bottom 1
Top 2

Lateral Bar Type: Ties

Rebar Pattern: Rectangular

Rebar Orientation: Face Parallel

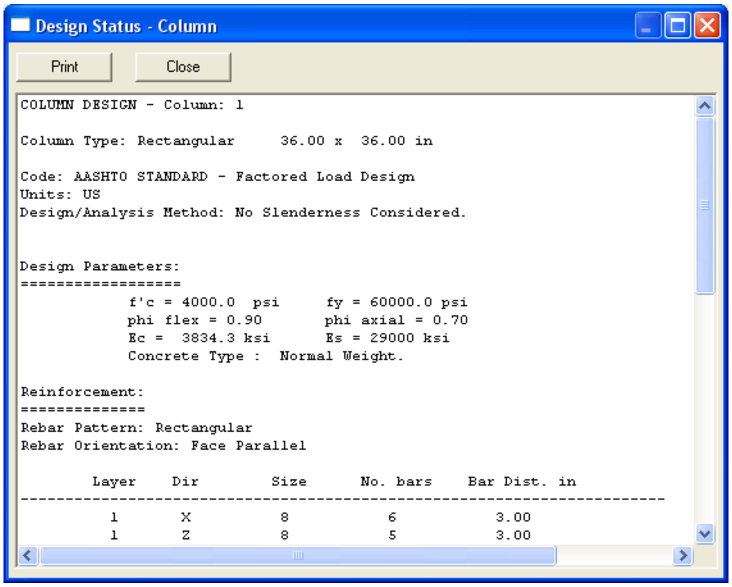
Consider MM ☐
Braced Frame ○
Unbraced ○

Layer#	Direction:	Bar Size:	#bars:	Bar dist. in
1	X	#8	6	3.
1	X	#8	6	3.00
1	Z	#8	5	3.00

Figure T1-34 Column Tab Screen - Showing Column 1 Design

7. Repeat steps 1-6 for the second column. Select **2** from the Column# list and click **Auto Design**. Select **#8** from the Bar Size list and **4** from the Stirrup Size list. Click **OK**.

To see a design summary of the selected component, click **Design Status** on the *Column* screen to activate the *Design Status - Column* screen, as shown in Figure T1-35. Click **Print** to print the design summary or click **Sketch** to see a graphical representation of the selected column.

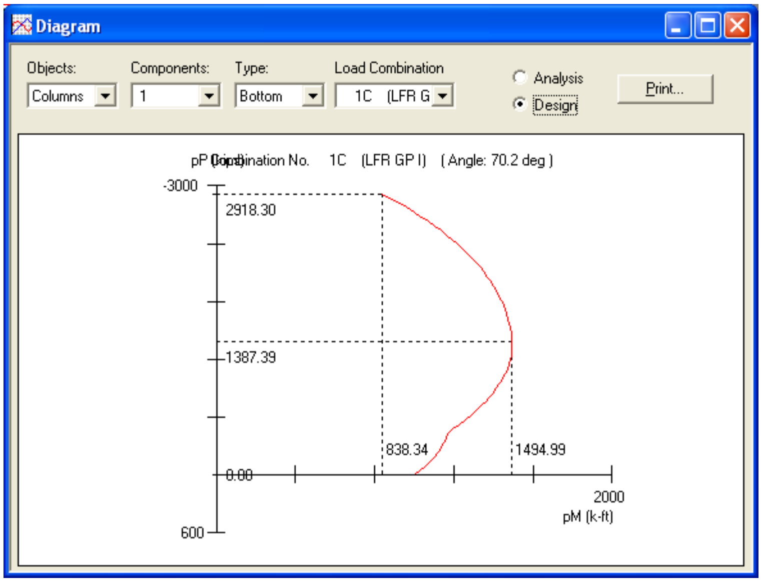


```
COLUMN DESIGN - Column: 1

Column Type: Rectangular     36.00 x  36.00 in

Code: AASHTO STANDARD - Factored Load Design
Units: US
Design/Analysis Method: No Slenderness Considered.


Design Parameters:
==================
            f'c = 4000.0  psi     fy = 60000.0 psi
            phi flex = 0.90       phi axial = 0.70
            Ec =  3834.3 ksi      Es = 29000 ksi
            Concrete Type :  Normal Weight.

Reinforcement:
==============
Rebar Pattern: Rectangular
Rebar Orientation: Face Parallel

        Layer    Dir        Size      No. bars    Bar Dist. in
------------------------------------------------------------------------
          1       X          8          6           3.00
          1       Z          8          5           3.00
```

Figure T1-35 Design Status - Column Screen

Step 23

After the column is designed view the column interaction diagram by selecting **Diagrams** from the **Show** menu. Then, select the column to view from the Objects list and the Design option. The column interaction diagram will appear on the screen as shown in Figure T1-36. View different load combinations by selecting from the Load Combination list.,



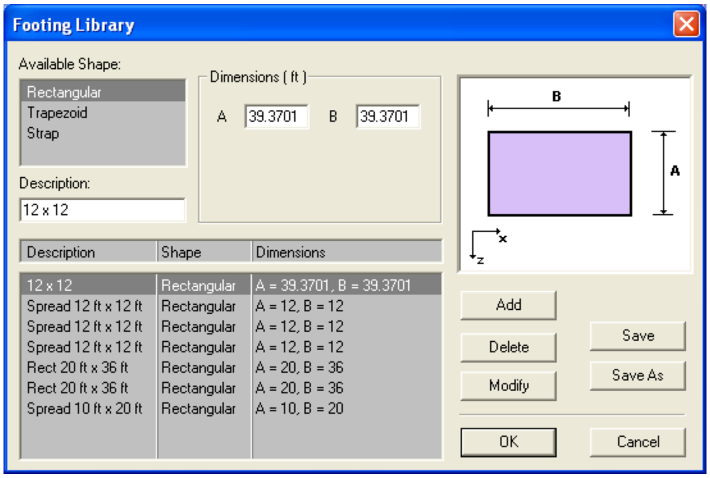
Diagram

Objects: Columns
Components: 1
Type: Bottom
Load Combination: 1C (LFR G
○ Analysis
● Design

Figure T1-36 Diagrams Screen - Showing Moment Interaction Diagram

Step 24

Before you can design the footing, you must first add the footing to the library. Select the **Footing Configuration** item from the **Libraries** menu to activate the *Footing Library* screen, as shown in Figure T1-37.



Footing Library

Available Shape:
- Rectangular
- Trapezoid
- Strap

Dimensions (ft)
A 39.3701 B 39.3701

Description: 12 x 12

Description	Shape	Dimensions
12 x 12	Rectangular	A = 39.3701, B = 39.3701
Spread 12 ft x 12 ft	Rectangular	A = 12, B = 12
Spread 12 ft x 12 ft	Rectangular	A = 12, B = 12
Spread 12 ft x 12 ft	Rectangular	A = 12, B = 12
Rect 20 ft x 36 ft	Rectangular	A = 20, B = 36
Rect 20 ft x 36 ft	Rectangular	A = 20, B = 36
Spread 10 ft x 20 ft	Rectangular	A = 10, B = 20

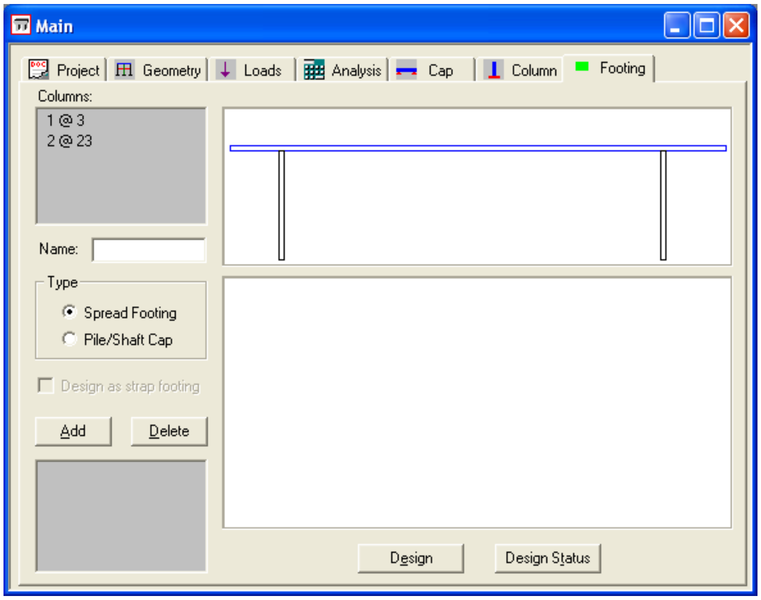
Figure T1-37 Footing Library Screen

Select **Rectangular** under Available Shape and input **Spread 12 ft × 12 ft** in the Description text box. Under Dimensions, input **12** in the A field and **12** in the B field. Click **Add**. It will appear in the list on the screen.

Click **Save** to save the footing library. Click **OK** to exit the *Footing Library* screen.

Step 25

Select the **Footing** tab to activate the *Footing* screen, similar to the one shown in Figure T1-38. Notice that no footings have been defined. The following illustrates how to define a spread footing and pile/cap footing.



Columns:
1 @ 3
2 @ 23

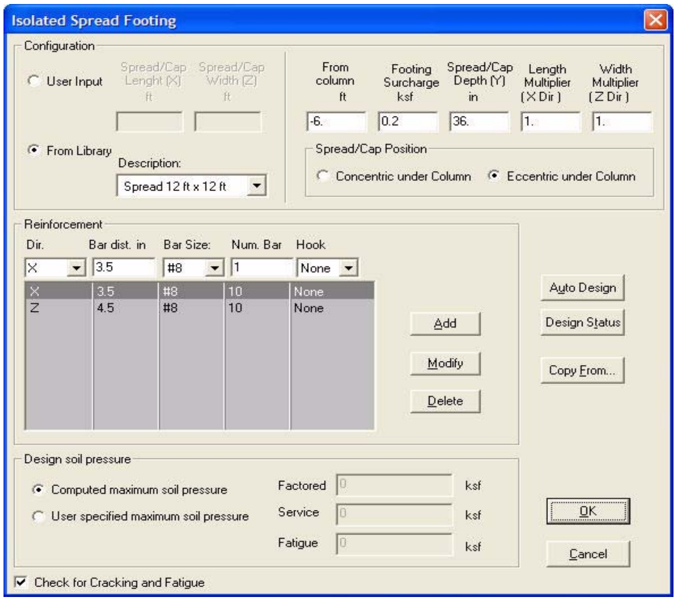
Name:

Type
● Spread Footing
○ Pile/Shaft Cap

☐ Design as strap footing

Figure T1-38 Footing Tab Screen

1. Select **1@3** under Columns, input **Spread** in the Name field, and select **Spread Footing** under Type. Click **Add**. The name **Spread** will appear in the list at the bottom of the screen.
2. Click **Design** to activate the *Isolated Spread Footing* screen, as shown in Figure T1-39.



Configuration

○ User Input
Spread/Cap Lenght (X) ft:
Spread/Cap Width (Z) ft:

● From Library
Description: Spread 12 ft x 12 ft

From column ft	Footing Surcharge ksf	Spread/Cap Depth (Y) in	Length Multiplier (X Dir)	Width Multiplier (Z Dir)
-6.	0.2	36.	1.	1.

Spread/Cap Position
○ Concentric under Column ● Eccentric under Column

Reinforcement

Dir.	Bar dist. in	Bar Size:	Num. Bar	Hook
X	3.5	#8	1	None
X	3.5	#8	10	None
Z	4.5	#8	10	None

Design soil pressure
● Computed maximum soil pressure
○ User specified maximum soil pressure
Factored 0 ksf
Service 0 ksf
Fatigue 0 ksf

☑ Check for Cracking and Fatigue

Figure T1-39 Isolated Spread Footing Screen

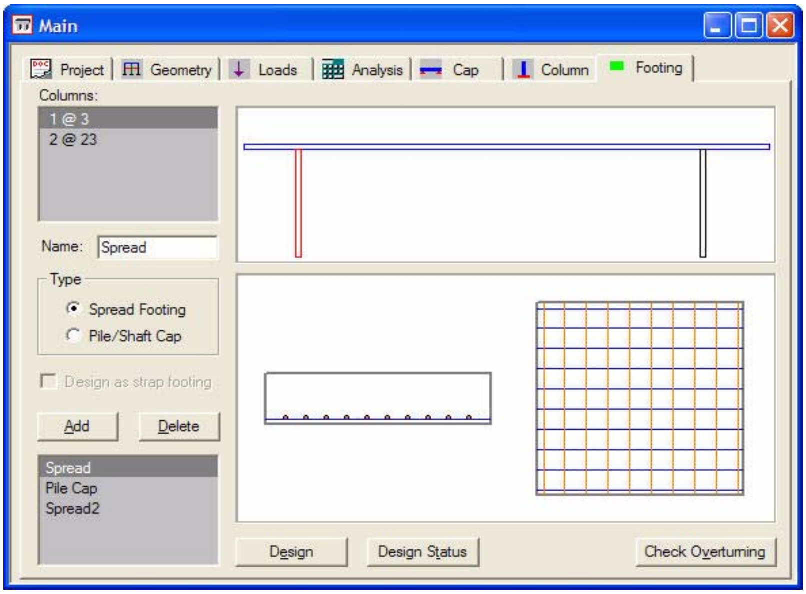
3. Select **Spread 12 ft × 12 ft** from the Description list. (This is the footing defined earlier.) Input **-6** in the From Column field, **0.2** in the Footing Surcharge field, **36** in the Spread/Cap Depth field, and **1** in the Length and Width Multiplier fields.

4. Click **Auto Design** to activate the *Design Footing* screen, as shown in Figure T1-40. Select **#8** from the Bar Size list. Click **OK** and the program will automatically generate the footing reinforcement.

Bar size: #8

Figure T1-40 Design Footing Screen

5. The *Footing Design Results - Spread* screen will immediately display. Click **Close** to exit this screen and return to the *Isolated Spread Footing* screen.
6. Highlight the first line (X) under Reinforcement. Change the value in the Num Bar field to **10** and click **Modify**. Repeat this for the second line, entering **10** in the Num Bar field.
7. Click the **Design Status** button to display a design summary of the selected component and verify the data. Click **Close** to exit this screen.
8. Click **OK** and return to the *Footing* screen. Notice that a diagram of the spread footing will be displayed at the bottom of the screen, as shown in Figure T1-41.



Name: Spread

● Spread Footing
○ Pile/Shaft Cap

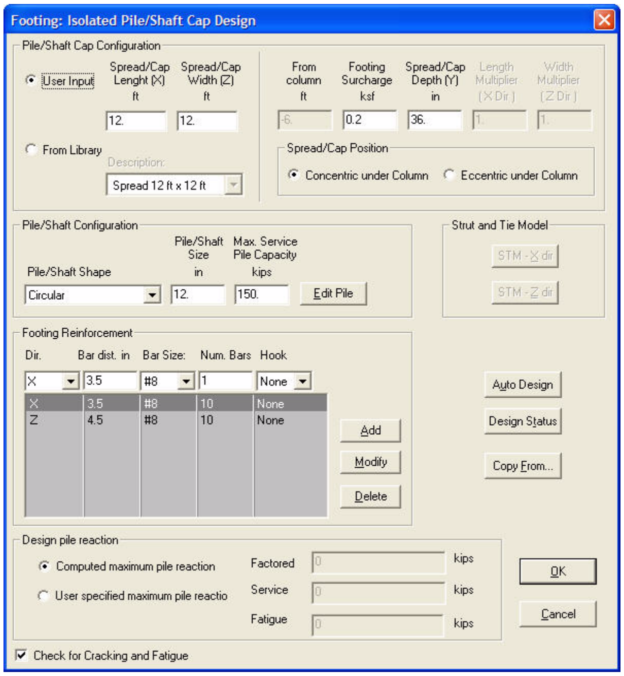
☐ Design as strap footing

Figure T1-41 Footing Tab Screen - Spread Footing Design

Define the Pile/Cap Footing Design, as follows:

1. Select **1@3** under Columns, input **Pile cap** in the Name field, and select **Pile/Shaft Cap** under Type. Click **Add**. The name **Pile cap** will appear in the list at the bottom of the screen.

2. Click **Design** to activate the *Footing: Isolated Pile/Shaft Cap Design* screen, as shown in Figure T1-42.



Footing: Isolated Pile/Shaft Cap Design

Pile/Shaft Cap Configuration

● User Input ○ From Library

Spread/Cap Lenght (X) ft	Spread/Cap Width (Z) ft	From column ft	Footing Surcharge ksf	Spread/Cap Depth (Y) in	Length Multiplier (X Dir.)	Width Multiplier (Z Dir.)
12.	12.	-6.	0.2	36.	1.	1.

Description: Spread 12 ft x 12 ft

Spread/Cap Position: ● Concentric under Column ○ Eccentric under Column

Pile/Shaft Configuration

Pile/Shaft Shape	Pile/Shaft Size in	Max. Service Pile Capacity kips
Circular	12.	150.

Footing Reinforcement

Dir.	Bar dist. in	Bar Size:	Num. Bars	Hook
X	3.5	#8	1	None
X	3.5	#8	10	None
Z	4.5	#8	10	None

Design pile reaction

● Computed maximum pile reaction

○ User specified maximum pile reactio

Factored 0 kips

Service 0 kips

Fatigue 0 kips

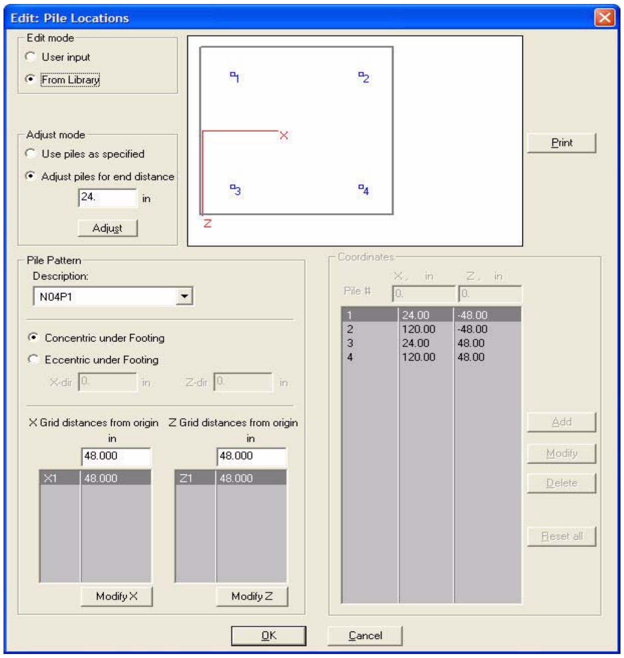
☑ Check for Cracking and Fatigue

Figure T1-42 Footing: Isolated Pile/Shaft Cap Design Screen

3. Select **Spread 12 ft × 12 ft** from the Description list. Input **-6** in the From Column field, **0.2** in the Footing Surcharge field, **36** in the Spread/Cap Depth field and **1** in the Length and Width Multiplier fields.

4. Under Pile/Shaft configuration, select **Circular** from the Pile/Shaft Shape list and input **12** in the Pile/Shaft Size field and **150** in the Max. Pile Capacity field.

5. In the Edit Mode, select From Library, to use the pile pattern defined in the library..



Edit: Pile Locations

Edit mode: ○ User input ● From Library

Adjust mode: ○ Use piles as specified ● Adjust piles for end distance 24. in

Pile Pattern

Description: N04P1

● Concentric under Footing

○ Eccentric under Footing

X-dir 0. in Z-dir 0. in

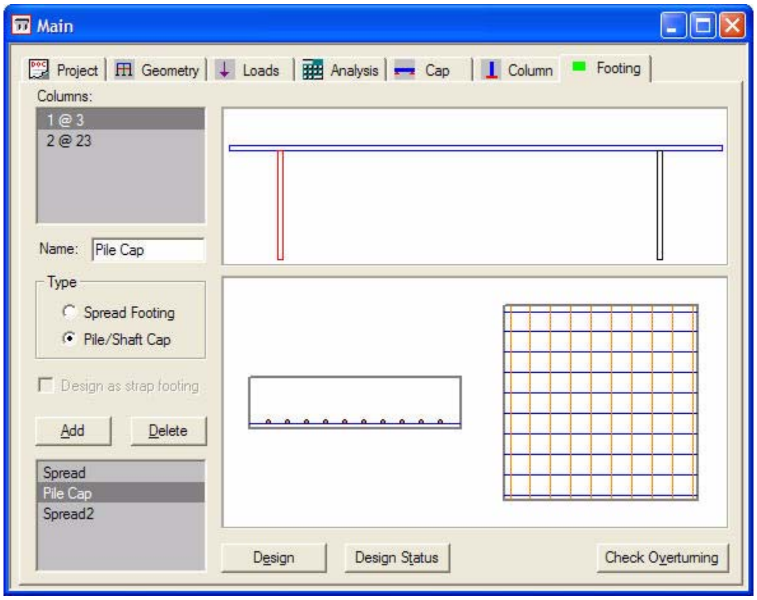
X Grid distances from origin in	Z Grid distances from origin in
48.000	48.000
X1 48.000	Z1 48.000

Coordinates

Pile #	X, in	Z, in
	0.	0.
1	24.00	-48.00
2	120.00	-48.00
3	24.00	48.00
4	120.00	48.00

Figure T1-43 Edit: Pile Locations Screen

6. In the Pile Pattern area, click on the Description drop down field and select the pattern N04P1. (Note: If you do not see any pile pattern available in the list, this means that Pile Library is not correctly selected. Please see page GO-63 on how to setup the Pile Pattern Library. You can also add custom pile patterns to the pile library as explained on page GO-66 in section, Libraries | Pile Pattern Configuration.)
7. Select to keep the pattern origin to be concentric under the footing. For that, choose the option, Concentric under footing.
8. Choose the Adjust Piles for end distance option and specify 24 in the text box.
9. When completed, there should be four pile locations, as shown in Figure T1-43. Click **OK** and return to the *Footing: Isolated Pile/Shaft Cap Design* screen.
10. Now input the reinforcement. Select **X** from the Direction list, input **3.5** in the Bar Dist field, **#8** from the Bar Size list, **10** in the Num Bars field, and **None** from the Hook list.
11. Click **Add**. It will appear in the list under Footing Reinforcement.
12. Select **Z** from the Direction list and input **4.5** in the Bar dist field, **#8** from the Bar Size list, **10** in the Num Bars field, and **None** from the Hook list.
13. Click **Add** to add it to the list under Footing Reinforcement.
14. Click **Design Status** to display the design summary of the selected component. Click **Close** to exit this screen.
15. Click **OK** and return to the *Footing* screen. The screen will resemble Figure T1-44.



Columns:
1 @ 3
2 @ 23

Name: Pile Cap

Type
○ Spread Footing
● Pile/Shaft Cap

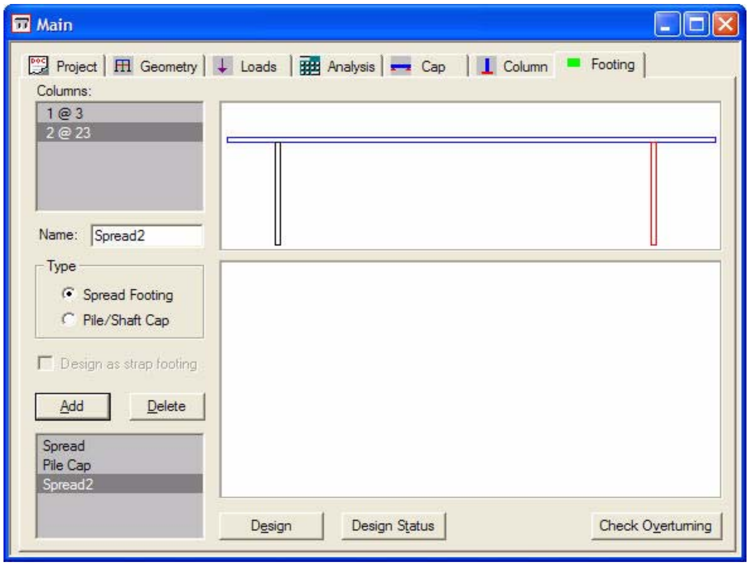
☐ Design as strap footing

Spread
Pile Cap
Spread2

Figure T1-44 Footing Tab Screen - Pile Cap Design

Step 26

In this step we will check to see if the isolated spread designed under column 1 works for column 2. On the Footing tab, select column 2 in the columns list. Specify the footing name as Spread2 and click **Add**, as shown in Figure T1-45.



Columns:
1 @ 3
2 @ 23

Name: Spread2

Type
● Spread Footing
○ Pile/Shaft Cap

☐ Design as strap footing

Spread
Pile Cap
Spread2

Figure T1-45 Adding Spread2 to the Footing List

Click the **Design** button. On the Isolated Spread Footing dialog click on the **Copy From** button, as shown in Figure T1-42. The Copy Footing dialog will appear, as shown in Figure T1-46. Select **Spread** from in the list and click **Copy**.

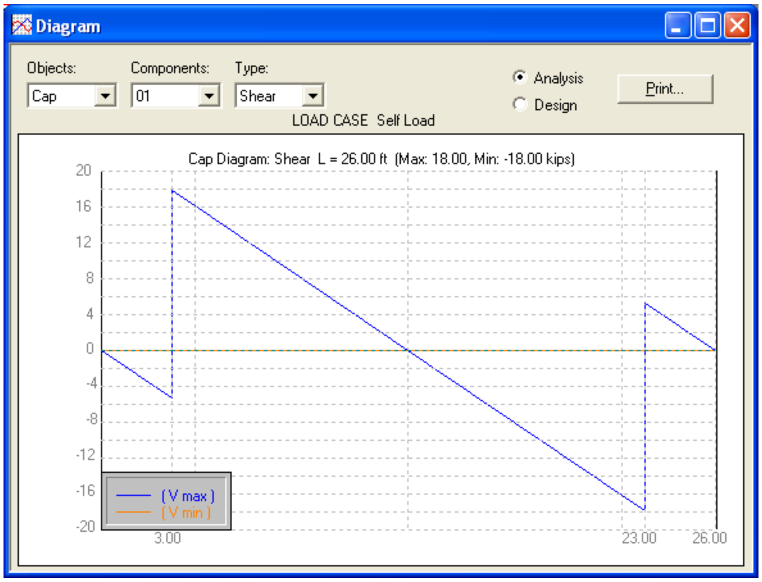
Isolated Spread Footing
to be copied from:
Spread
Spread

Figure T1-46 Copy Footing Dialog

All the information about spread will be copied to the Spread2 footing. Click the **Design Status** button and review the results to see if this footing will work. Click **OK** on this dialog to return to the Footing tab.

Step 27

Select **Diagrams** from the **Show** menu to activate the *Diagram* screen, as shown in Figure T1-47. Experiment with the lists and buttons to become familiar with the options of this feature.



Diagram

Objects: Cap

Components: 01

Type: Shear

● Analysis

○ Design

LOAD CASE Self Load

Cap Diagram: Shear L = 26.00 ft (Max: 18.00, Min: -18.00 kips)

Figure T1-47 Diagram Screen

Step 28

This completes Tutorial 1. To print the output of the project, select **Print** from the **File** menu and the *Print* screen will display. Select the appropriate options and click **OK**.

Following this tutorial is a printout of the output for selected items from Tutorial 1.

```
PROJECT DATA
============

Project     : Tutorial 1
User Job No.: Tutor1
State       : FL                             State Job No. : FL-001
Code        : AASHTO STANDARD (17th Edition 2002)
Comments    : Auto Load Generation of DL, LL, WS, WL
              Auto Design of Cap Beam
              Design of Column, Spread footing and pile footing




PIER GEOMETRY
=============

Pier Type: Multi Column

Cap Shape: Straight   Top Elevations: start = 22.00 ft   end = 22.00 ft
   Depth(Z) = 36.00 in   Skew angle = 0.00   Reduction of I = 1.000
   Length(X) = 26.00 ft   Height(Y) = 48.00 in

Column Shape : Rectangular
   Number of columns: 2


   Column number 1:

      Location from the left edge of the cap(X): 3.00 ft
      Elevations: bottom  = 0.00 ft   top = 20.00 ft   Reduction of I = 1.000
      Column section dimensions:
         Width(X) = 36.00 in   Depth(Z) = 36.00 in

   Column number 2:

      Location from the left edge of the cap(X): 23.00 ft
      Elevations: bottom  = 0.00 ft   top = 20.00 ft   Reduction of I = 1.000
      Column section dimensions:
         Width(X) = 36.00 in   Depth(Z) = 36.00 in


STRUCTURE MODEL

================

FRAME Model:
                Member   Node   Hinge Check Pt Dist(ft) Memb length(ft)
                -------------------------------------------------------
   Column No. 1
                  1        1       -              0.00
                           2       -             20.00      20.00
   Column No. 2
                  2        3       -              0.00
                           4       -             20.00      20.00
   Cap
                  3        5       -              0.00
                           2       -              3.00       3.00
                  4        2       -              3.00
```

```
                           6       -              4.00       1.00
                  5        6       -              4.00
                           7       -             13.00       9.00
                  6        7       -             13.00
                           8       -             22.00       9.00
                  7        8       -             22.00
                           4       -             23.00       1.00
                  8        4       -             23.00
                           9       -             26.00       3.00

   Node coordinates:
      Number         X(ft)        Y(ft)          Node type
      ---------------------------------------------------------
         1            3.00         0.00        fixed at ground
         2            3.00        20.00             column-cap
         3           23.00         0.00        fixed at ground
         4           23.00        20.00             column-cap
         5            0.00        20.00
         6            4.00        20.00                bearing
         7           13.00        20.00                bearing
         8           22.00        20.00                bearing
         9           26.00        20.00


SUPERSTRUCTURE INFO
===================

   Total number of spans: 2   Span number rear to current pier: 1
   Number of traffic lanes: 2

   Beam:  height : 36.00 in   section area : 369.00 in^2
   Beam Inertia (Ixx): 50980.00 in^4  Beam inertia (Iyy): 5332.50 in^4
   Beam CG:15.83 in  Barrier height : 36.00 in  Depth of slab : 8.00 in

 Bridge width:    27.00  ft         Curb to curb distance:   24.000 ft
    Span #          Span length
 ------------------------------------------------
      1                60.000 ft
      2                60.000 ft


BEARING POINTS
==============

   Number of bearing lines: 1
      First bearing line      Eccentricity = 0.00 ft
                 Point      Distance ft
                 ----------------------
                   1            4.00
                   2           13.00
                   3           22.00


MATERIAL PROPERTIES
===================

                                  Cap          Column      Footing
                         ------------------------------------------
   Concrete Type                  normal       normal       normal
   Concrete Strength  (psi)      4000.00      4000.00      4000.00
```

```
   Concrete Density (lb/ft3)      150.00       150.00       150.00
   Concrete Modulus Ec  (ksi)    3834.30      3834.30      3834.30
   Steel Strength Fy  (ksi)        60.00        60.00        60.00


DESIGN PARAMETERS
=================

   AASHTO STANDARD Code
      Strength Reduction factors for reinf. concrete:   Multi presence factors for live load:
         Flexure and tension          0.90                      1  Lane       1.00
         Shear and torsion (normal)   0.85                      2  Lanes      1.00
                      (lightweight)   0.85                      3  Lanes      0.90
         Axial compression (ties)     0.70             more than 3  Lanes      0.75
         Axial compression (spiral)   0.75

               Crack control factor    Min clear cover    Impact factors
                     kip/ft                      in        (auto calculation)
         ----------------------------------------------------------------
         Cap            170.00                  2.50          1.27
         Column         170.00                  2.00          1.27
         Footing        170.00                  3.00          1.00

      Degree of fixity in foundations for Moment Magnify Method:  R =  5.00
```

```
LOADS
=====

   Load Cases: 7


      Loadcase ID: D1     Name:
      Multiplier =  1.000

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          1         Y      -111.86
           1          2         Y      -111.86
           1          3         Y      -111.86

      Auto generation details:

      Generated Dead Load
         Slab weight    =   150.00     pcf   Girder weight =   150.00     pcf
         Wearing weight =   540.00     plf   Barrier load  =  1200.00     plf


      Loadcase ID: (L+In)1     Name:
      Multiplier =  1.000

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          1         Y       -48.74
           1          2         Y       -13.93
           1          3         Y         0.00

      Auto generation details:

      Generated Live Load
                Load:  HS20 truck
            Number of loaded lanes = 1
                Total number of considered truck positions = 9


      Loadcase ID: (L+In)2     Name:
      Multiplier =  1.000

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          1         Y       -20.89
           1          2         Y       -41.78
           1          3         Y         0.00

      Auto generation details:

      Generated Live Load
                Load:  HS20 truck
            Number of loaded lanes = 1
                Total number of considered truck positions = 9


      Loadcase ID: (L+In)3     Name:
```

```
Multiplier =  1.000

Bearing loads:
   Line #    Bearing #   Dir.    Load,  kip
   -----------------------------------------
     1          1         Y         0.00
     1          2         Y       -41.78
     1          3         Y       -20.89

Auto generation details:

Generated Live Load
          Load:  HS20 truck
      Number of loaded lanes = 1
          Total number of considered truck positions = 9


Loadcase ID: (L+In)4     Name:
Multiplier =  1.000

Bearing loads:
   Line #    Bearing #   Dir.    Load,  kip
   -----------------------------------------
     1          1         Y         0.00
     1          2         Y       -13.93
     1          3         Y       -48.74

Auto generation details:

Generated Live Load
          Load:  HS20 truck
      Number of loaded lanes = 1
          Total number of considered truck positions = 9


Loadcase ID: W1     Name: Angle: 30
Multiplier =  1.000

Cap loads:
   Type     Dir       Arm          Mag1          x1/L        Mag2        x2/L
                       ft     kip,kip/ft, kft           kip,kip/ft, kft
   --------------------------------------------------------------------------------
   Force     X        0.00         2.16          0.50        ----        ----
   UDL       Z        ----        -0.05          0.00        ----        1.00

Column loads:
   Col #      Type     Dir      Mag1        y1/L        Mag2       y2/L
   --------------------------------------------------------------------
      1        UDL     X       0.142  k/ft  0.00       ----        0.90
      1        UDL     Z      -0.082  k/ft  0.00       ----        0.90
      2        UDL     X       0.142  k/ft  0.00       ----        0.90
      2        UDL     Z      -0.082  k/ft  0.00       ----        0.90

Bearing loads:
   Line #    Bearing #   Dir.    Load,  kip
   -----------------------------------------
     1          1         X         5.47
     1          1         Y         3.04
     1          1         Z        -1.60
     1          2         X         5.47
```

```
          1          2         Y         0.00
          1          2         Z        -1.60
          1          3         X         5.47
          1          3         Y        -3.04
          1          3         Z        -1.60

     Auto generation details:
     Generated Wind Load on Structure
        Angle of wind = 30.00 deg    Elevation above which wind load acts =   0.00 ft
        Default wind pressure
        Wind pressure for superstructure:           Wind pressure for substructure:
             Transverse     41.000 psf                     Cap        40.000 psf
           Longitudinal     12.000 psf                  Column        40.000 psf
            Overturning   not considered


     Loadcase ID: WL1     Name: Angle: 30
     Multiplier =  1.000

     Bearing loads:
        Line #    Bearing #   Dir.    Load,  kip
        -----------------------------------------
          1          1         X         1.64
          1          1         Y         2.64
          1          1         Z        -0.48
          1          2         X         1.64
          1          2         Y         0.00
          1          2         Z        -0.48
          1          3         X         1.64
          1          3         Y        -2.64
          1          3         Z        -0.48

     Auto generation details:
     Generated Wind Load on Live Load
        Angle of wind = 30.00 deg   Live load length =  60.00 ft


  Selected load groups:

     SERVICE GROUP I
     SERVICE GROUP IA
     SERVICE GROUP III
     SERVICE GROUP VI
     LOAD FACTOR GROUP I
     LOAD FACTOR GROUP IA
     LOAD FACTOR GROUP III
     LOAD FACTOR GROUP VI
```

```
CAP DESIGN
==========

Code: AASHTO STANDARD (17th Edition 2002) - Ultimate Strength Design
Units: US

DESIGN PARAMETERS:
==================
            f'c = 4000.0  psi
            Fy flex = 60000.0  psi  Fy shear = 60000.0  psi
            phi flex = 0.90         phi shear = 0.85
            Ec =  3834.3  ksi       Es = 29000.0  ksi
            crack control factor  z = 170.00  kips / in
            Concrete Type :  Normal Weight.
            Design  :  centerline of column.


CAP GEOMETRY:
=============
       Straight Cap :  Length(X) = 26.00  ft     Depth(Z) = 36.00  in


MAIN REINFORCEMENT:
===================

         Bar size   Quantity  Bar dist. As total    From      To       Hook
                                 in       in^2       ft       ft
------------------------------------------------------------------------------
TOP
          #  8          4       3.50     3.160     0.00      9.75      Left
          #  8          2       3.50     1.580     6.25     19.75      None
          #  8          4       3.50     3.160    16.25     26.00      Right
BOTTOM
          #  8          2       3.50     1.580     0.00      9.25      Left
          #  8         10       3.50     7.900     6.75     19.25      None
          #  8          2       3.50     1.580    16.75     26.00      Right


STIRRUPS:
==========

    From       To   Stirrup Size       n legs    Spacing        Aprv/s
     ft        ft                                  in         in^2 / ft
------------------------------------------------------------------------------
     0.00      3.00     # 4              4        18.00         0.53
     3.00      4.00     # 4              4         6.00         1.60
     4.00     22.00     # 4              4        18.00         0.53
    22.00     23.00     # 4              4         6.00         1.60
    23.00     26.00     # 4              4        18.00         0.53

Clear Cover on Sides =   3.50  in

FLEXURE DESIGN:
===============

                             Span 1: From 0.00  ft  To 3.00  ft

 Loc  AbsLoc  H    Mmax    pMn   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
                   Mmin    pMn   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
  ft    ft    in  kips-ft  kips-ft       in^2     in^2     in^2     in^2     in^2     in^2
```

```
-------------------------------------------------------------------------------------------
-
 0.0   0.0   48     0.0     0.0     0    0.00     1.58     0.00     0.00     3.16     0.00
                    0.0     0.0     0    0.00     1.58     0.00     0.00     3.16     0.00
 3.0   3.0   48     0.0   337.2     0    0.00     1.58     1.58     0.00     3.16     3.16
                  -10.5  -630.0    41    0.00     1.58     1.58     0.07     3.16     3.16


                             Span 2: From 3.00  ft  To 23.00  ft

 Loc  AbsLoc  H    Mmax    pMn   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
                   Mmin    pMn   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
  ft    ft   in  kips-ft  kips-ft       in^2     in^2     in^2     in^2     in^2     in^2
-------------------------------------------------------------------------------------------
-
 0.0   3.0   48     0.0   337.2     0    0.00     1.58     1.58     0.00     3.16     3.16
                 -459.9  -630.0    47    0.00     1.58     1.58     3.11     3.16     3.16
 1.0   4.0   48    76.7   337.2    49    0.51     1.58     1.58     0.00     3.16     3.16
                 -141.7  -630.0    63    0.00     1.58     1.58     0.95     3.16     3.16
10.0  13.0   48  1326.8  1513.6    46    6.89     7.90     7.90     0.00     1.58     1.58
                    0.0  -342.0     0    0.00     7.90     7.90     0.00     1.58     1.58
19.0  22.0   48    76.7   337.2    64    0.51     1.58     1.58     0.00     3.16     3.16
                 -141.7  -630.0    50    0.00     1.58     1.58     0.95     3.16     3.16
20.0  23.0   48     0.0   337.2     0    0.00     1.58     1.58     0.00     3.16     3.16
                 -459.9  -630.0    46    0.00     1.58     1.58     3.11     3.16     3.16


                             Span 3: From 23.00  ft  To 26.00  ft

 Loc  AbsLoc  H    Mmax    pMn   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
                   Mmin    pMn   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
  ft    ft   in  kips-ft  kips-ft       in^2     in^2     in^2     in^2     in^2     in^2
-------------------------------------------------------------------------------------------
-
 0.0  23.0   48     0.0   337.2     0    0.00     1.58     1.58     0.00     3.16     3.16
                  -10.5  -630.0    41    0.00     1.58     1.58     0.07     3.16     3.16
 3.0  26.0   48     0.0     0.0     0    0.00     1.58     0.00     0.00     3.16     0.00
                    0.0     0.0     0    0.00     1.58     0.00     0.00     3.16     0.00



SHEAR AND TORSION DESIGN:
=========================

                            Span 1: From 0.00  ft  To 3.00  ft

  Loc  AbsLoc Pos   Vu   Comb    Tu   Comb phi*Vc   T-lim   Avs/s 2Ats/s  Av/s    Aprv/s   Alt
   ft    ft      kips      kips-ft      kips   kips-ft  <--------- in^2/ft -------->   in^2
-------------------------------------------------------------------------------------------
-
  0.00   0.00 R    0.0    0     0.0    0   172.2    79.6   0.00   0.00   0.00     0.53    0.00
  3.00   3.00 L    7.0   41     0.0    0   172.2    79.6   0.00   0.00   0.00     0.53    0.00


                            Span 2: From 3.00  ft  To 23.00  ft

  Loc  AbsLoc Pos   Vu   Comb    Tu   Comb phi*Vc   T-lim   Avs/s 2Ats/s  Av/s    Aprv/s   Alt
   ft    ft      kips      kips-ft      kips   kips-ft  <--------- in^2/ft -------->   in^2
-------------------------------------------------------------------------------------------
-
```

```
  0.00   3.00 R   435.0   45     3.7   49   172.2    79.6   1.39   0.00   1.39     1.60     0.00
  1.00   4.00 L   432.6   45     3.7   61   172.2    79.6   1.38   0.00   1.38     1.60     0.00
              R   173.5   47     1.2   51   172.2    79.6   0.36   0.00   0.36     0.53     0.00
 10.00  13.00 L   152.4   47     1.2   63   172.2    79.6   0.36   0.00   0.36     0.53     0.00
              R   152.4   46     1.2   62   172.2    79.6   0.36   0.00   0.36     0.53     0.00
 19.00  22.00 L   173.5   46     1.2   50   172.2    79.6   0.36   0.00   0.36     0.53     0.00
              R   432.6   48     3.7   64   172.2    79.6   1.38   0.00   1.38     1.60     0.00
 20.00  23.00 L   435.0   48     3.7   52   172.2    79.6   1.39   0.00   1.39     1.60     0.00


                         Span 3: From 23.00  ft  To 26.00  ft

  Loc  AbsLoc Pos   Vu   Comb    Tu   Comb phi*Vc   T-lim   Avs/s 2Ats/s  Av/s    Aprv/s   Alt
   ft    ft      kips       kips-ft       kips   kips-ft  <--------- in^2/ft -------->   in^2
-----------------------------------------------------------------------------------------
-
  0.00  23.00 R     7.0   41     0.0    0   172.2    79.6  0.00   0.00   0.00     0.53     0.00
  3.00  26.00 L     0.0    0     0.0    0   172.2    79.6  0.00   0.00   0.00     0.53     0.00


Note:

-  Pos is the design position. L suggests the calculation is done at immediate left
   of "Loc" and R suggests at immediate right of it.
-  T-lim is the limiting value of torsion for the concrete section. If actual torsion
   is higher than this value, torsional steel has to be provided.
-  Avs/s is the required area of steel per unit length for shear force.
-  2Ats/s is the required area of steel per unit length for two legs of torsional
   reinforcement.
-  Av/s is the total required area of steel per unit length due to shear plus torsion.
-  Aprvs/s is the total provided area of steel per unit length due to shear (stirrups).
-  Alt is the total longitudinal steel required due to torsion in addition to the REQUIRED
   flexural steel.


CRACKING/FATIGUE CHECK:
=======================

                           Span 1: From 0.00   ft  To 3.00   ft

                      <------ Cracking ------>     <------ Fatigue ------>
 Loc    AbsLoc   H      fs-t  ratio fs-t  Comb     fs-t   ratio fs-t
                        fs-b  ratio fs-b  Comb     fs-b   ratio fs-b
  ft      ft     in     ksi                         ksi
-----------------------------------------------------------------------
  0.00    0.0   48.0    0.0       0.00      0       0.0       0.00
                        0.0       0.00      0       0.0       0.00
  3.00    3.0   48.0    0.7       0.02      1       0.0       0.00
                        0.0       0.00      0       0.0       0.00


                           Span 2: From 3.00   ft  To 23.00   ft

                      <------ Cracking ------>     <------ Fatigue ------>
 Loc    AbsLoc   H      fs-t  ratio fs-t  Comb     fs-t   ratio fs-t
                        fs-b  ratio fs-b  Comb     fs-b   ratio fs-b
  ft      ft     in     ksi                         ksi
-----------------------------------------------------------------------
  0.00    3.0   48.0   30.7       0.87      7       6.0       0.37
                        0.0       0.00      0       0.0       0.00
```

```
  1.00    4.0   48.0    9.8       0.28     23       3.9       0.17
                       10.5       0.38      9       6.7       0.29
 10.00   13.0   48.0    0.0       0.00      0       0.0       0.00
                       36.2       1.00*     6       7.9       0.50
 19.00   22.0   48.0    9.8       0.28     10       3.9       0.17
                       10.5       0.38     24       6.7       0.29
 20.00   23.0   48.0   30.7       0.87      6       6.0       0.37
                        0.0       0.00      0       0.0       0.00


                           Span 3: From 23.00   ft  To 26.00   ft

                      <------ Cracking ------>     <------ Fatigue ------>
 Loc    AbsLoc   H      fs-t  ratio fs-t  Comb     fs-t   ratio fs-t
                        fs-b  ratio fs-b  Comb     fs-b   ratio fs-b
  ft      ft     in     ksi                         ksi
------------------------------------------------------------------------
  0.00   23.0   48.0    0.7       0.02      1       0.0       0.00
                        0.0       0.00      0       0.0       0.00
  3.00   26.0   48.0    0.0       0.00      0       0.0       0.00
                        0.0       0.00      0       0.0       0.00


Note:
*  Cracking / fatigue checking failed.
```

```
COLUMN DESIGN
=============

COLUMN DESIGN - Column: 1

Column Type: Rectangular     36.00 x  36.00 in

Code: AASHTO STANDARD (17th Edition 2002) - Factored Load Design
Units: US
Design/Analysis Method: No Slenderness Considered.


Design Parameters:
==================
            f'c = 4000.0  psi     fy = 60000.0 psi
            phi flex = 0.90       phi axial = 0.70
            Ec =  3834.3 ksi      Es = 29000 ksi
            Concrete Type :  Normal Weight.

Reinforcement:
==============
Rebar Pattern: Rectangular
Rebar Orientation: Face Parallel

        Layer    Dir        Size      No. bars    Bar Dist. in
------------------------------------------------------------------------
          1       X          8           6           3.00
          1       Z          8           5           3.00

          Main bars summary:                 Ties   size: # 4
                            18 #  8 bars

          Total number of bars in the column: 18


Design values used - ( e-min effect included ).
===============================================
                      (global coordinates)
 Loc      Comb      Fx        Fy        Fz        Mx        My        Mz
  ft               kips      kips      kips    kips-ft   kips-ft   kips-ft
--------------------------------------------------------------------------
  0.00    13C       33.6     359.9       0.0     -50.4       0.0    -222.9
 20.00    13C      -33.6     328.3       0.0     -46.0       0.0    -449.3
  0.00    45C       33.7     328.3      -2.7     -51.2       1.2    -251.6
 20.00    45C      -32.7     296.8       2.1      41.5       1.2    -411.2


Column Design
=============

 Loc     Comb    Pu         Mux        Muz       pMn       Incl   pPn/Pu  pMn/Mu
  ft            kips      kips-ft    kips-ft    kips-ft     deg
--------------------------------------------------------------------------------------
  0.00   45C   328.3       51.2      251.6     1150.8      78.49    1.00   4.48
 20.00   13C   328.3       46.0      449.3     1156.3      84.16    1.00   2.56
```

```
COLUMN DESIGN
=============

COLUMN DESIGN - Column: 2

Column Type: Rectangular     36.00 x  36.00 in

Code: AASHTO STANDARD (17th Edition 2002) - Factored Load Design
Units: US
Design/Analysis Method: No Slenderness Considered.


Design Parameters:
==================
            f'c = 4000.0  psi     fy = 60000.0 psi
            phi flex = 0.90       phi axial = 0.70
            Ec =  3834.3 ksi      Es = 29000 ksi
            Concrete Type :  Normal Weight.

Reinforcement:
==============
Rebar Pattern: Rectangular
Rebar Orientation: Face Parallel

        Layer    Dir        Size      No. bars    Bar Dist. in
------------------------------------------------------------------------
          1       X          8           6           3.00
          1       Z          8           5           3.00

          Main bars summary:                  Ties   size: # 4
                            18 #  8 bars

          Total number of bars in the column: 18


Design values used - ( e-min effect included ).
===============================================
                      (global coordinates)
 Loc      Comb      Fx        Fy        Fz        Mx        My        Mz
  ft               kips      kips      kips    kips-ft   kips-ft   kips-ft
--------------------------------------------------------------------------
  0.00    11C     -33.6     359.9       0.0     -50.4       0.0     222.9
 20.00    11C      33.6     328.3       0.0     -46.0       0.0     449.3
  0.00    19C     -33.7     328.3       2.7      51.2       1.2     251.6
 20.00    19C      32.7     296.8      -2.1     -41.5       1.2     411.2


Column Design
=============

 Loc     Comb    Pu         Mux        Muz       pMn       Incl   pPn/Pu  pMn/Mu
  ft           kips      kips-ft    kips-ft    kips-ft     deg
--------------------------------------------------------------------------------------
  0.00   19C   328.3       51.2      251.6     1150.8      78.49    1.00   4.48
 20.00   11C   328.3       46.0      449.3     1156.3      84.16    1.00   2.56
```

```
ISOLATED FOOTING DESIGN

Code: AASHTO STANDARD (17th Edition 2002) - Ultimate Strength Design
Units: US

Geometry:
=========
   Name  : Spread
   Shape : Rectangular, Type : Spread

   Bf(X) = 12.00 ft, Hf(Z) = 12.00 ft, Thickness(Y) = 36.00 in

   Footing concentric.
   Columns located on the footing:
   Column No.  1  at x =   0.00 ft,  Rectangular  36.00 in x  36.00 in

   Ag = 144.00 ft^2, Ix = 1728.00 ft^4, Iz = 1728.00 ft^4

   Surcharge =   0.20 ksf


Design Parameters:
==================     f'c =  4000.00  psi      fy = 60000.00  psi
                       phi flex = 0.90          phi shear = 0.85
                       Ec =    3834.3  ksi      Es =   29000.0 ksi
                       Crack control factor z = 170.00 kips/in
                       Concrete Type :  Normal Weight.



Max Soil Pressures, Service (Without the reduction of overstress allowance):
============================

Corner     X       Z   |---------------- Column Loads ----------------| Soil press.
           ft      ft   comb    Ovs      P, kips   Mxx, kft   Mzz, kft     ksf
------------------------------------------------------------------------------------
   1      6.00   -6.00    5    1.500    -321.99       0.00     125.81      2.45
                         24    1.250    -234.12      39.41     173.67      1.54
   2     -6.00   -6.00    5    1.500    -321.99       0.00     125.81      3.32
                         12    1.250    -215.70     -39.41      50.68      2.46
   3     -6.00    6.00    5    1.500    -321.99       0.00     125.81      3.32
                         12    1.250    -215.70     -39.41      50.68      2.19
   4      6.00    6.00    5    1.500    -321.99       0.00     125.81      2.45
                         24    1.250    -234.12      39.41     173.67      1.81


Max Soil Pressures, Factored:
=============================

Corner     X       Z   |---------------- Column Loads ----------------| Soil press.
           ft      ft   comb    Ovs      P, kips   Mxx, kft   Mzz, kft     ksf
------------------------------------------------------------------------------------
   1      6.00   -6.00   45      ---     -432.44       0.00     167.01      3.27
                         80      ---     -292.65      49.26     217.09      1.92
   2     -6.00   -6.00   45      ---     -432.44       0.00     167.01      4.43
                         68      ---     -269.63     -49.26      63.35      3.08
   3     -6.00    6.00   45      ---     -432.44       0.00     167.01      4.43
                         68      ---     -269.63     -49.26      63.35      2.73
   4      6.00    6.00   45      ---     -432.44       0.00     167.01      3.27
                         80      ---     -292.65      49.26     217.09      2.26
```

```
Max Soil Pressures, Service (After the reduction of overstress allowance):
============================

Corner     X       Z   |---------------- Column Loads ----------------| Soil press.
          ft      ft   comb    Ovs      P, kips   Mxx, kft   Mzz, kft     ksf
------------------------------------------------------------------------------------
   1      6.00   -6.00    1    1.000    -268.74       0.00     112.51      2.13
                         40    1.400    -234.12      39.41     173.67      1.10
   2     -6.00   -6.00    1    1.000    -268.74       0.00     112.51      2.91
                         28    1.400    -215.70     -39.41      50.68      1.76
   3     -6.00    6.00    1    1.000    -268.74       0.00     112.51      2.91
                         28    1.400    -215.70     -39.41      50.68      1.56
   4      6.00    6.00    1    1.000    -268.74       0.00     112.51      2.13
                          8    1.500    -234.33       0.00     125.14      1.23


Note:
   Only max. positive pressure is considered for design.


Max. Soil Pressure Used in Design:  (without selfweight and surcharge)
==================================

 Factored soil pressure  =   3.62 ksf

 Service soil pressure   =   2.89 ksf

 Fatigue soil pressure   =   0.42 ksf



Reinforcement Schedule:
=======================:

 Dir  Quantity  Size  Bar dist.  As total    Spacing    Hook
                         in        in^2         in
------------------------------------------------------------
  X      10     #  8    3.50         7.90     15.22     None
  Z      10     #  8    4.50         7.90     15.22     None



Flexure:
========

Dir   Loc    d       Mmax   Comb  Asb_req  Asb_prv   Asb_eff   Ast_req  Ast_prv   Ast_eff
       ft    in       kft           in^2     in^2      in^2      in^2     in^2      in^2
-----------------------------------------------------------------------------------------
--
 X  -1.50  32.50     439.3   45     4.03     7.90      7.90      0.00     0.00      0.00
 X   1.50  32.50     439.3   45     4.03     7.90      7.90      0.00     0.00      0.00
 Z  -1.50  31.50     439.3   45     4.16     7.90      7.90      0.00     0.00      0.00
 Z   1.50  31.50     439.3   45     4.16     7.90      7.90      0.00     0.00      0.00
```

```
Cracking/Fatigue
================
                        <-------- Cracking ---------->   <---------- Fatigue --------->
 Dir    Loc     d       Mmax   Comb     fs    ratio fs    Mmax    Comb      fs    ratio fs
         ft     in       kft            ksi               kft             ksi
----------------------------------------------------------------------------------------
  X   -1.50   32.50     351.1    5    17.26    0.57      50.5      1     2.48    0.13
  X    1.50   32.50     351.1    5    17.26    0.57      50.5      1     2.48    0.13
  Z   -1.50   31.50     351.1    5    17.82    0.59      50.5      1     2.57    0.13
  Z    1.50   31.50     351.1    5    17.82    0.59      50.5      1     2.57    0.13




One Way Shear:
==============

 Col   Dir     Dist      Comb       d          Vu      phi*Vc
                ft                  in        kips       kips
----------------------------------------------------------------
   1    X     -4.21       45      32.50       77.7      503.2
        X      4.21       45      32.50       77.7      503.2
        Z     -4.13       45      31.50       81.3      487.7
        Z      4.13       45      31.50       81.3      487.7




Two Way Shear:
==============

  #     Bo       Ao     Comb     Avg. d      Vu      phi*Vc
          ft       ft^2              in       kips       kips
----------------------------------------------------------------
Columns:
  1      22.67      32.11   45     32.00      404.5     1871.7


Note:
TWO WAY SHEAR IN FOOTING IS NOT DESIGNED AND STIRRUPS ARE NOT CONSIDERED.
```

```
ISOLATED FOOTING DESIGN

Code: AASHTO STANDARD (17th Edition 2002) - Ultimate Strength Design
Units: US

Geometry:
=========
   Name  : Pile Cap
   Shape : Rectangular, Type : Pile/Shaft Cap

   Bf(X) = 12.00 ft, Hf(Z) = 12.00 ft, Thickness(Y) = 36.00 in

   Footing concentric.
   Columns located on the footing:
   Column No.  1  at x =   0.00 ft,  Rectangular  36.00 in x  36.00 in

   Ag = 144.00 ft^2, Ix = 64.00 ft^2, Iz = 64.00 ft^2

   Surcharge =   0.20 ksf

   Piles:   Circular      Size: 12.00 in   Capacity: 150.00 kips

   Pile Pattern Name: N04P1
   Pile Pattern concentric.
   Pile Pattern Type: Grid-based
   Number of Piles: 4

   Grid distances:
      X1 = 48.000
      Z1 = 48.000

 1  1
 1  1

Design Parameters:
==================     f'c =  4000.00  psi      fy = 60000.00  psi
                       phi flex = 0.90          phi shear = 0.85
                       Ec =    3834.3  ksi      Es =   29000.0 ksi
                       Crack control factor z = 170.00 kips/in
                       Concrete Type :  Normal Weight.



Pile Reactions, Service (Without the reduction of overstress allowance):
========================

Pile Loc(X)   X       Z   |---------------- Column Loads ----------------| Pile Reac.
       ft     in      in   comb    Ovs      P, kips    Mxx, kft   Mzz, kft    kips
------------------------------------------------------------------------------------
  1  -4.00   24.0   -48.0    5    1.500    -321.99       0.00     125.81      111.76
                            12    1.250    -215.70     -39.41      50.68       82.96
  2   4.00  120.0   -48.0    5    1.500    -321.99       0.00     125.81       96.04
                            24    1.250    -234.12      39.41     173.67       68.61
  3  -4.00   24.0    48.0    5    1.500    -321.99       0.00     125.81      111.76
                            12    1.250    -215.70     -39.41      50.68       78.03
  4   4.00  120.0    48.0    5    1.500    -321.99       0.00     125.81       96.04
                            12    1.250    -215.70     -39.41      50.68       71.69
```

```
Pile Reactions, Factored:
=========================

Pile Loc(X)   X       Z   |---------------- Column Loads ----------------| Pile Reac.
       ft     in      in   comb    Ovs      P, kips    Mxx, kft   Mzz, kft    kips
------------------------------------------------------------------------------------
  1  -4.00   24.0   -48.0   45     ---     -432.44       0.00     167.01      148.97
                            68     ---     -269.63     -49.26      63.35      103.69
  2   4.00  120.0   -48.0   45     ---     -432.44       0.00     167.01      128.09
                            80     ---     -292.65      49.26     217.09       85.77
  3  -4.00   24.0    48.0   45     ---     -432.44       0.00     167.01      148.97
                            68     ---     -269.63     -49.26      63.35       97.54
  4   4.00  120.0    48.0   45     ---     -432.44       0.00     167.01      128.09
                            68     ---     -269.63     -49.26      63.35       89.62


Pile Reactions, Service (After the reduction of overstress allowance):
========================

Pile Loc(X)   X       Z   |---------------- Column Loads ----------------| Pile Reac.
       ft     in      in   comb    Ovs      P, kips    Mxx, kft   Mzz, kft    kips
------------------------------------------------------------------------------------
  1  -4.00   24.0   -48.0    1    1.000    -268.74       0.00     112.51       97.62
                            28    1.400    -215.70     -39.41      50.68       59.25
  2   4.00  120.0   -48.0    1    1.000    -268.74       0.00     112.51       83.55
                            40    1.400    -234.12      39.41     173.67       49.01
  3  -4.00   24.0    48.0    1    1.000    -268.74       0.00     112.51       97.62
                            28    1.400    -215.70     -39.41      50.68       55.74
  4   4.00  120.0    48.0    1    1.000    -268.74       0.00     112.51       83.55
                             8    1.500    -234.33       0.00     125.14       49.44


Note:

   Only max. force in piles is considered for design.
   Pile coordinates X and Z are from the most left edge of the footing.


Max. Pile Reaction Used in Design: (without selfweight and surcharge)
==================================

 Factored pile reaction  = 118.55 kips

 Service pile reaction   =  88.36 kips

 Fatigue pile reaction   =  14.14 kips



Reinforcement Schedule:
=======================:

 Dir  Quantity  Size  Bar dist.  As total    Spacing    Hook
                         in        in^2         in
------------------------------------------------------------
  X      10     #  8    3.50         7.90     15.22     None
  Z      10     #  8    4.50         7.90     15.22     None
```

```
Flexure:
========

Dir   Loc    d       Mmax   Comb  Asb_req  Asb_prv   Asb_eff   Ast_req  Ast_prv   Ast_eff
       ft    in       kft           in^2     in^2      in^2      in^2     in^2      in^2
----------------------------------------------------------------------------------------
--
 X  -1.50  32.50     592.7   45    5.45     7.90      7.90      0.00     0.00      0.00
 X   1.50  32.50     592.7   45    5.45     7.90      7.90      0.00     0.00      0.00
 Z  -1.50  31.50     592.7   45    5.62     7.90      7.90      0.00     0.00      0.00
 Z   1.50  31.50     592.7   45    5.62     7.90      7.90      0.00     0.00      0.00




Cracking/Fatigue
================
                        <-------- Cracking ---------->   <---------- Fatigue --------->
 Dir    Loc     d       Mmax   Comb    fs    ratio fs    Mmax    Comb    fs    ratio fs
         ft     in       kft           ksi               kft             ksi
---------------------------------------------------------------------------------------
  X   -1.50   32.50     441.8    5    21.72    0.72      70.7      1     3.48    0.19
  X    1.50   32.50     441.8    5    21.72    0.72      70.7      1     3.48    0.19
  Z   -1.50   31.50     441.8    5    22.42    0.74      70.7      1     3.59    0.20
  Z    1.50   31.50     441.8    5    22.42    0.74      70.7      1     3.59    0.20




One Way Shear:
==============

 Col   Dir     Dist      Comb       d          Vu      phi*Vc
                ft                  in        kips       kips
----------------------------------------------------------------
   1    X     -4.21       45      32.50       62.8      503.2
        X      4.21       45      32.50       62.8      503.2
        Z     -4.13       45      31.50       85.1      487.7
        Z      4.13       45      31.50       85.1      487.7




Two Way Shear:
==============

  #       Bo        Ao     Comb     Avg. d      Vu      phi*Vc
          ft       ft^2              in        kips       kips
----------------------------------------------------------------
Columns:
  1      22.67      32.11   45     32.00      474.2     1871.7

Piles - max:
  1      11.52      10.56   45     32.00      118.5      951.2
Piles - min:
  1      11.52      10.56   45     32.00      118.5      951.2


Note:
TWO WAY SHEAR IN FOOTING IS NOT DESIGNED AND STIRRUPS ARE NOT CONSIDERED.
```

```
ISOLATED FOOTING DESIGN

Code: AASHTO STANDARD (17th Edition 2002) - Ultimate Strength Design
Units: US

Geometry:
=========
   Name  : Spread2
   Shape : Rectangular, Type : Spread

   Bf(X) = 12.00 ft, Hf(Z) = 12.00 ft, Thickness(Y) = 36.00 in

   Footing concentric.
   Columns located on the footing:
   Column No.  2  at x =   0.00 ft,  Rectangular  36.00 in x  36.00 in

   Ag = 144.00 ft^2, Ix = 1728.00 ft^4, Iz = 1728.00 ft^4

   Surcharge =   0.20 ksf


Design Parameters:
==================     f'c =  4000.00  psi      fy = 60000.00  psi
                       phi flex = 0.90          phi shear = 0.85
                       Ec =    3834.3  ksi      Es =   29000.0 ksi
                       Crack control factor z = 170.00 kips/in
                       Concrete Type :  Normal Weight.



Max Soil Pressures, Service (Without the reduction of overstress allowance):
============================

Corner     X       Z   |---------------- Column Loads ----------------| Soil press.
          ft      ft   comb    Ovs      P, kips   Mxx, kft   Mzz, kft     ksf
---------------------------------------------------------------------------------
   1      6.00   -6.00    8    1.500    -321.99       0.00    -125.81      3.32
                         21    1.250    -215.70      39.41     -50.68      2.19
   2     -6.00   -6.00    8    1.500    -321.99       0.00    -125.81      2.45
                          9    1.250    -234.12     -39.41    -173.67      1.81
   3     -6.00    6.00    8    1.500    -321.99       0.00    -125.81      2.45
                          9    1.250    -234.12     -39.41    -173.67      1.54
   4      6.00    6.00    8    1.500    -321.99       0.00    -125.81      3.32
                         21    1.250    -215.70      39.41     -50.68      2.46


Max Soil Pressures, Factored:
=============================

Corner     X       Z   |---------------- Column Loads ----------------| Soil press.
          ft      ft   comb    Ovs      P, kips   Mxx, kft   Mzz, kft     ksf
---------------------------------------------------------------------------------
   1      6.00   -6.00   48     ---     -432.44       0.00    -167.01      4.43
                         77     ---     -269.63      49.26     -63.35      2.73
   2     -6.00   -6.00   48     ---     -432.44       0.00    -167.01      3.27
                         65     ---     -292.65     -49.26    -217.09      2.26
   3     -6.00    6.00   48     ---     -432.44       0.00    -167.01      3.27
                         65     ---     -292.65     -49.26    -217.09      1.92
   4      6.00    6.00   48     ---     -432.44       0.00    -167.01      4.43
                         77     ---     -269.63      49.26     -63.35      3.08
```

```
Max Soil Pressures, Service (After the reduction of overstress allowance):
============================

Corner     X       Z   |---------------- Column Loads ----------------| Soil press.
           ft      ft   comb    Ovs      P, kips   Mxx, kft   Mzz, kft     ksf
-------------------------------------------------------------------------------------
   1      6.00   -6.00    4    1.000    -268.74       0.00    -112.51      2.91
                         37    1.400    -215.70      39.41     -50.68      1.56
   2     -6.00   -6.00    4    1.000    -268.74       0.00    -112.51      2.13
                          5    1.500    -234.33       0.00    -125.14      1.23
   3     -6.00    6.00    4    1.000    -268.74       0.00    -112.51      2.13
                         25    1.400    -234.12     -39.41    -173.67      1.10
   4      6.00    6.00    4    1.000    -268.74       0.00    -112.51      2.91
                         37    1.400    -215.70      39.41     -50.68      1.76


Note:
   Only max. positive pressure is considered for design.


Max. Soil Pressure Used in Design:  (without selfweight and surcharge)
==================================

 Factored soil pressure  =   3.62 ksf

 Service soil pressure   =   2.89 ksf

 Fatigue soil pressure   =   0.42 ksf



Reinforcement Schedule:
=======================:

 Dir  Quantity  Size  Bar dist.  As total    Spacing    Hook
                         in        in^2         in
------------------------------------------------------------
  X      10     #  8    3.50         7.90     15.22     None
  Z      10     #  8    4.50         7.90     15.22     None



Flexure:
========

Dir   Loc    d       Mmax   Comb  Asb_req  Asb_prv   Asb_eff   Ast_req  Ast_prv   Ast_eff
       ft    in       kft           in^2     in^2      in^2      in^2     in^2      in^2
----------------------------------------------------------------------------------------
--
 X  -1.50  32.50     439.3   48     4.03     7.90      7.90      0.00     0.00      0.00
 X   1.50  32.50     439.3   48     4.03     7.90      7.90      0.00     0.00      0.00
 Z  -1.50  31.50     439.3   48     4.16     7.90      7.90      0.00     0.00      0.00
 Z   1.50  31.50     439.3   48     4.16     7.90      7.90      0.00     0.00      0.00
```

```
Cracking/Fatigue
================
                        <-------- Cracking ---------->   <---------- Fatigue --------->
 Dir    Loc     d       Mmax   Comb    fs    ratio fs    Mmax    Comb    fs    ratio fs
         ft     in       kft           ksi               kft           ksi
---------------------------------------------------------------------------------------
  X   -1.50   32.50     351.1    8    17.26    0.57       50.5      4     2.48    0.13
  X    1.50   32.50     351.1    8    17.26    0.57       50.5      4     2.48    0.13
  Z   -1.50   31.50     351.1    8    17.82    0.59       50.5      4     2.57    0.13
  Z    1.50   31.50     351.1    8    17.82    0.59       50.5      4     2.57    0.13




One Way Shear:
==============

 Col  Dir     Dist      Comb       d          Vu      phi*Vc
                ft                  in        kips       kips
----------------------------------------------------------------
   1    X     -4.21       48      32.50       77.7      503.2
        X      4.21       48      32.50       77.7      503.2
        Z     -4.13       48      31.50       81.3      487.7
        Z      4.13       48      31.50       81.3      487.7




Two Way Shear:
==============

  #       Bo        Ao     Comb     Avg. d      Vu      phi*Vc
          ft       ft^2              in        kips       kips
----------------------------------------------------------------
Columns:
  1      22.67      32.11   48     32.00      404.5     1871.7


Note:
TWO WAY SHEAR IN FOOTING IS NOT DESIGNED AND STIRRUPS ARE NOT CONSIDERED.
```

Chapter

T2

Multiple Column Pier (AASHTO LRFD)

Tutorial 2 demonstrates the basic features of RC-PIER® according to LRFD. This tutorial will take you through a RC-PIER design, step-by-step, illustrating the design of a cap beam, column, rectangular combined spread footing, and rectangular combined pile footing.

The information for Tutorial 2 is contained in a RC-PIER input file called “Tutor2.rcp.” As an alternative to inputting all information manually, as directed by this tutorial, you can load this file and work on only specific sections. To load this file, select the **Open** item from the **File** menu (**File | Open**). Move to the RC-PIER/Examples subdirectory, if it is not already showing and select the file “Tutor2.rcp”.

Diagrams

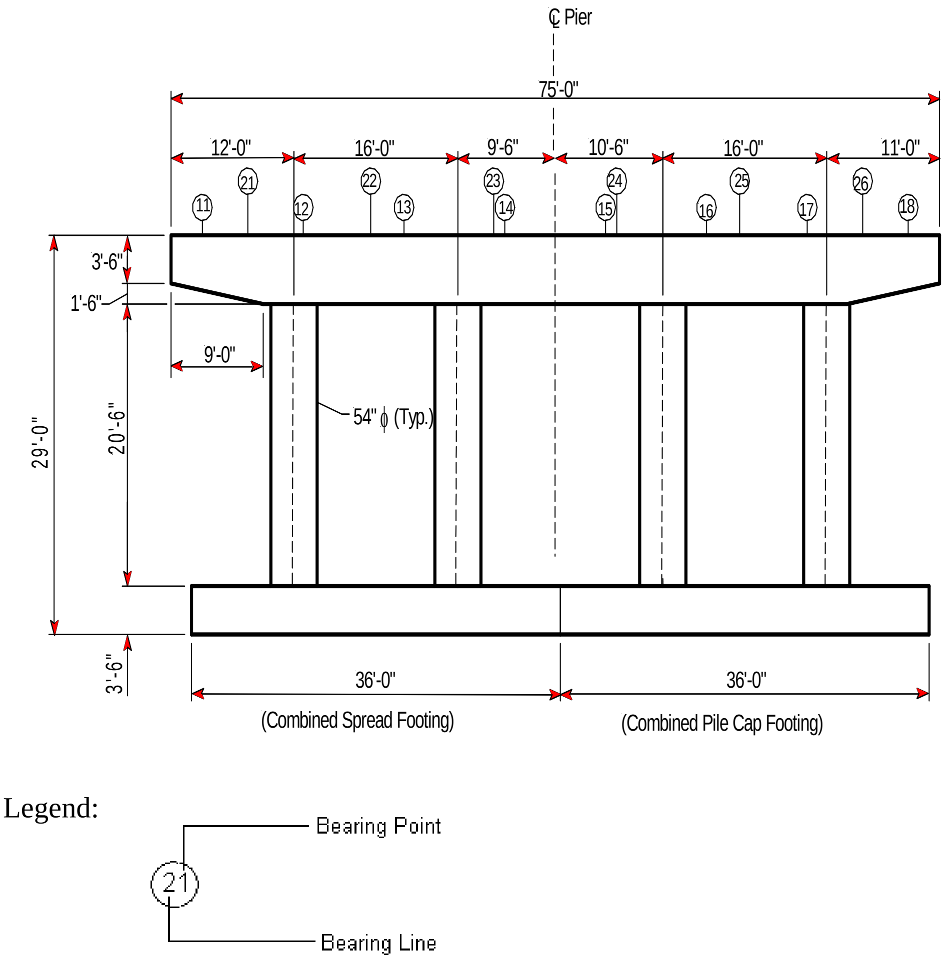


Figure T2-1 Longitudinal Pier Elevation

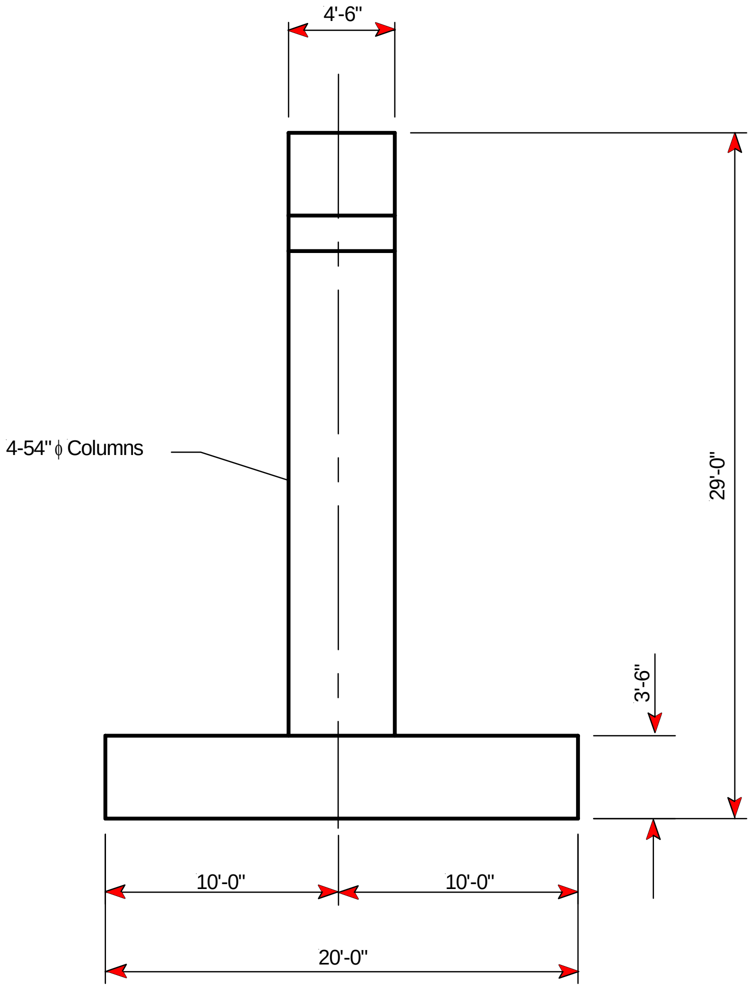


Figure T2-2 Transverse Pier Elevation

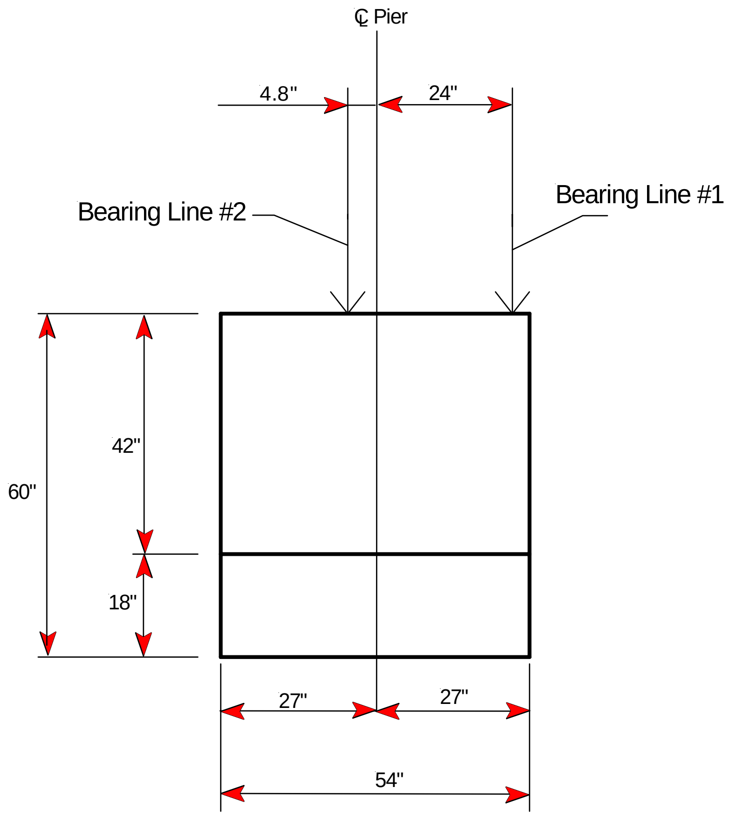


Figure T2-3 Section of Pier Cap

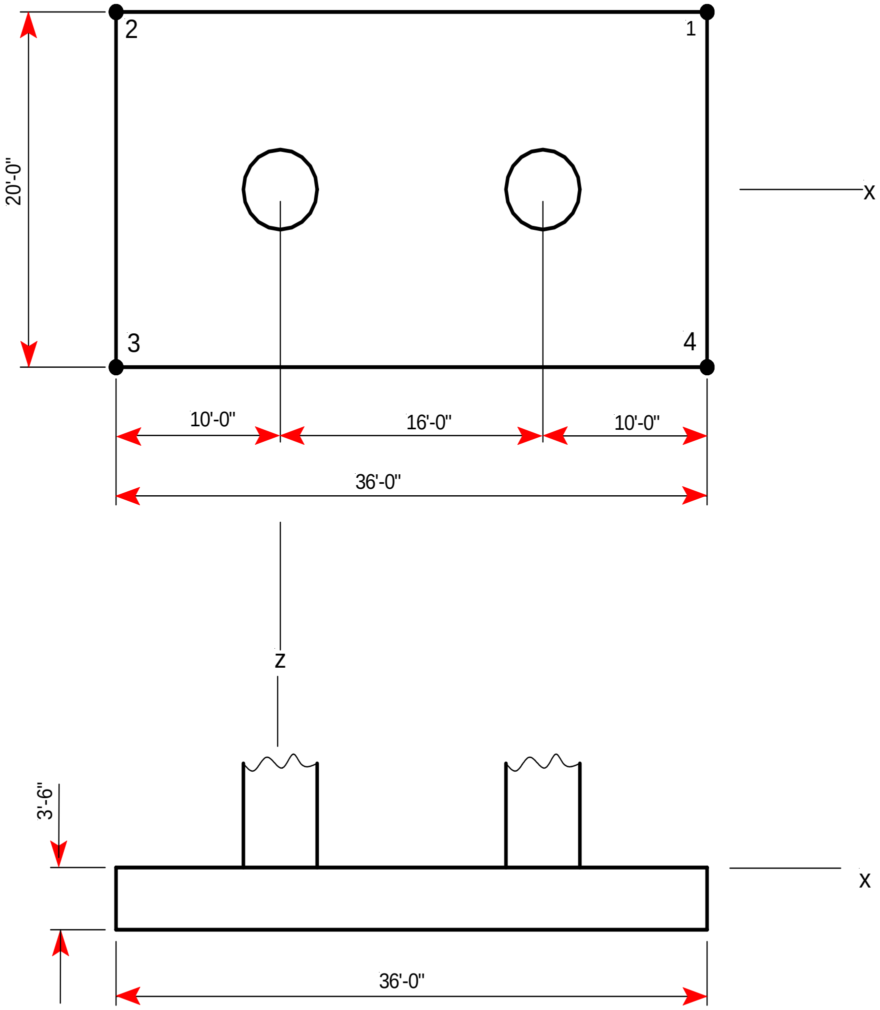


Figure T2-4 Plan and Elevation of Combined Spread Footing

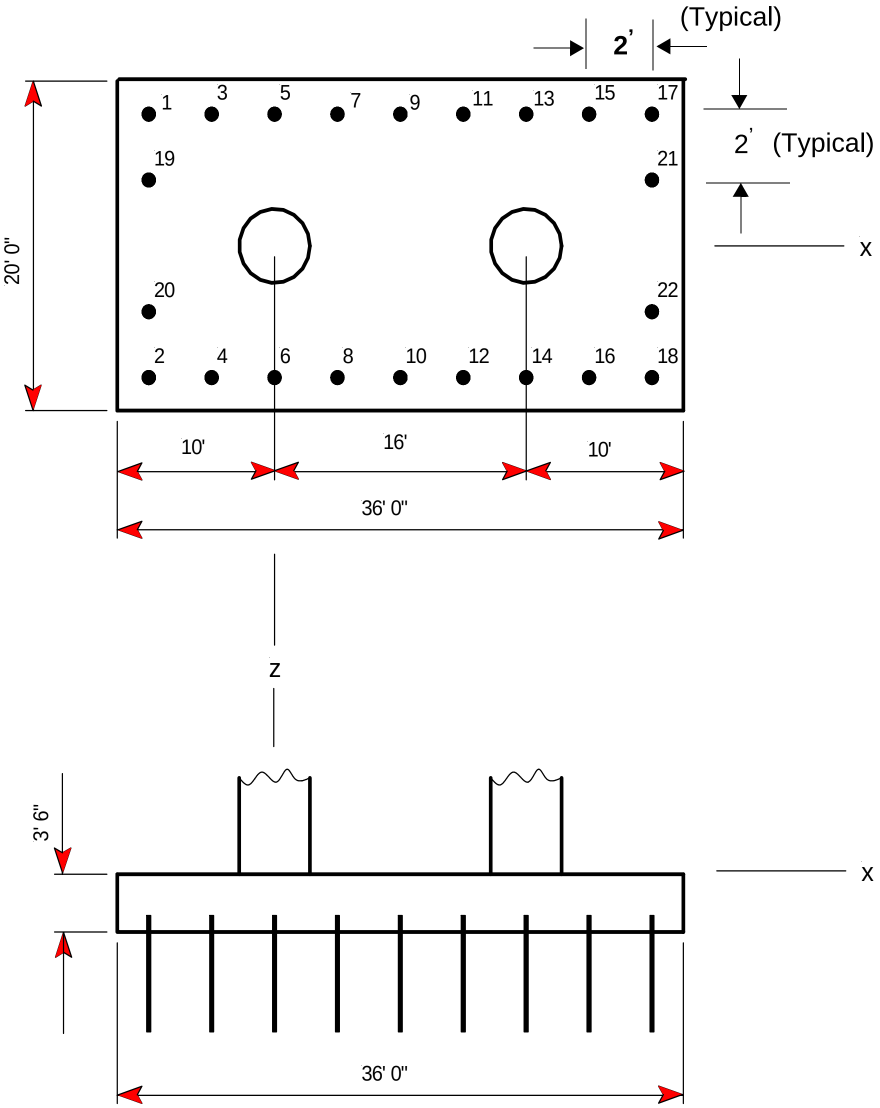


Figure T2-5 Plan and Elevation of Combined Pile Footing

Problem Data

Table T2-1

Concrete Strength	
Cap	f’c = 4 ksi
Columns	f’c = 4 ksi
Footings	’c = 4 ksi
Concrete Density	
Cap	ρ = 150 pcf
Columns	ρ = 150 pcf
Footings	ρ = 150 pcf
Steel Yield Strength	
Cap	f_y = 60 ksi
Columns	f_y = 60 ksi
Footings	f_y = 60 ksi
Concrete Type	
Cap	Normal
Columns	Normal
Footings	Normal
Loads	
Dead Load	
Bearing Loads	
Bearing Line 1	P = -300 kips (all bearings)
Bearing Line 2	P = -100 kips (all bearings)
Vehicular Live Load	
Bearing Loads (LL1)	
Line 1	
Bearing 1	P = -20 kips
Bearing 2	P = -7 kips
Line 2	
Bearing 1	P = -45 kips
Bearing 2	P = 6 kips
Bearing Loads (LL2)	
Line 1	
Bearing 7	P = -10 kips
Bearing 8	P = -29 kips
Line 2	

Table T2-1

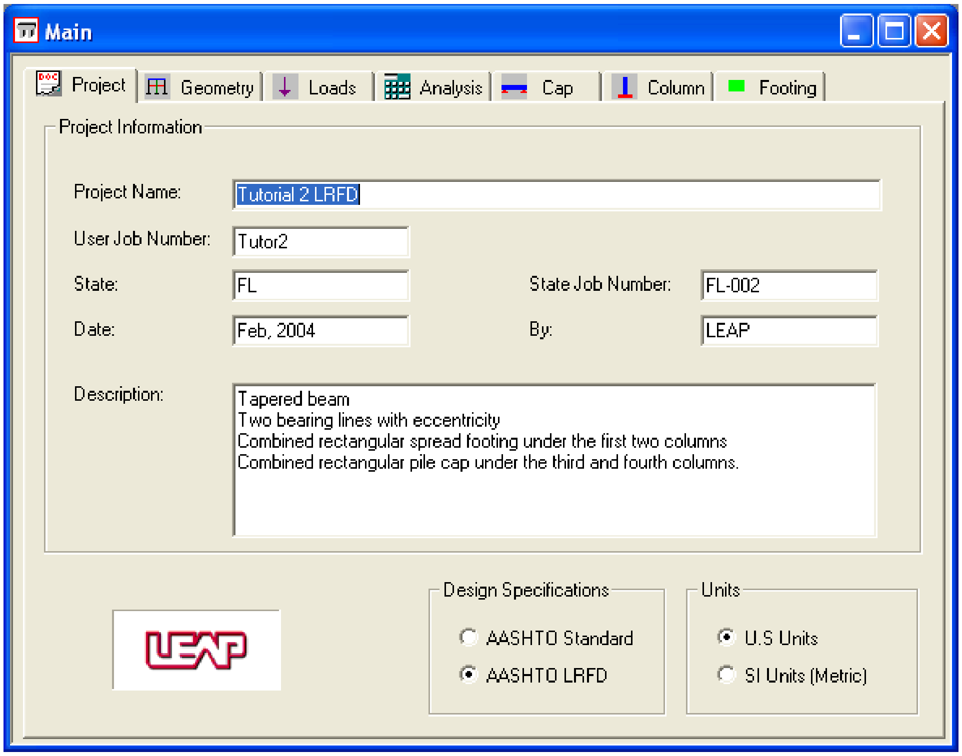
Bearing 5	P = 4 kips
Bearing 6	P = -30 kips
Bearing Loads (LL3)	
Line 1	
Bearing 2	P = -43 kips
Bearing 3	P = -19 kips
Bearing Loads (LL4)	
Line 1	
Bearing 6	P = -19 kips
Bearing 7	P = -43 kips
Wind on Structure	
Bearing Loads	
Line 1	
Bearing 1	P = 35 kips
Bearing 8	P = -35 kips
Line 2	
Bearing 1	P = 42 kips
Bearing 6	P = -42 kips
Cap Loads	
X-dir	P = 40 kips
Z-dir	
UDL	ϖ = 0.4667 klf
Creep	
Creep Strain Load	ε_{cr} = 0.0003
Shrinkage	
Shrinkage Strain Load	ε_{sh} = - 0.00027
Footing Surcharge	
Footing	σ = 0.200 ksf

Starting the Tutorial

By default, RC-PIER is installed in Program Files\LEAP Software\RC Pier. Activate the program by clicking **Start>Programs>LEAP Software** and selecting the RC Pier icon.

Step 1

The *Project* screen will be displayed, as shown in Figure T2-6. The fields will be blank. Fill in the general project information in the text boxes. Select the **AASHTO LRFD** option under Design Specifications and **U.S. Units** under Units.



Project Information

Project Name: Tutorial 2 LRFD

User Job Number: Tutor2

State: FL

State Job Number: FL-002

Date: Feb, 2004

By: LEAP

Description:
Tapered beam
Two bearing lines with eccentricity
Combined rectangular spread footing under the first two columns
Combined rectangular pile cap under the third and fourth columns.

Design Specifications
○ AASHTO Standard
● AASHTO LRFD

Units
● U.S Units
○ SI Units (Metric)

Figure T2-6 Project Tab Screen

Step 2

Click the **Geometry** tab to activate the *Geometry* screen, as shown in Figure T2-7. Use the buttons on the left of the screen to input the pier layout information as follows.

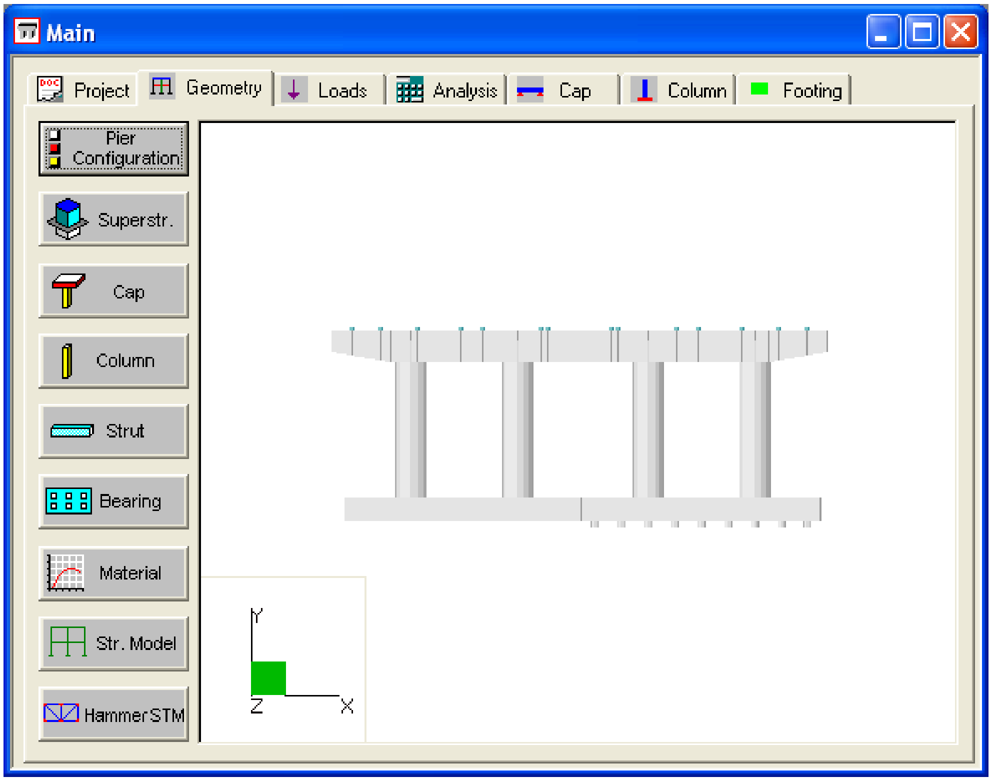
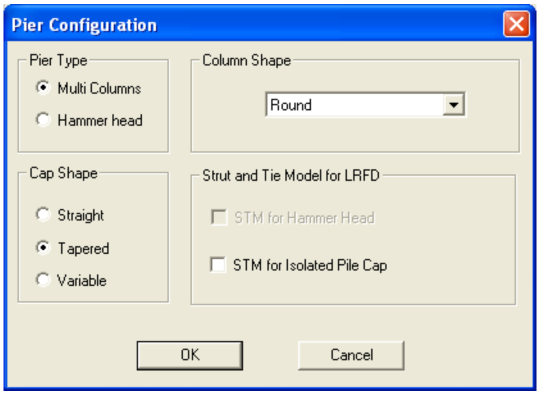


Figure T2-7 Geometry Tab Screen

Step 3

Click **Pier Configuration** to open the *Pier Configuration* screen, as shown in Figure T2-8. Under Pier Type, select **Multi-Columns**, under Cap Shape, select **Tapered**, and from the Column Shape list, select **Round**. Click **OK** and return to the *Geometry* screen.



Pier Configuration

Pier Type
● Multi Columns
○ Hammer head

Column Shape
Round

Cap Shape
○ Straight
● Tapered
○ Variable

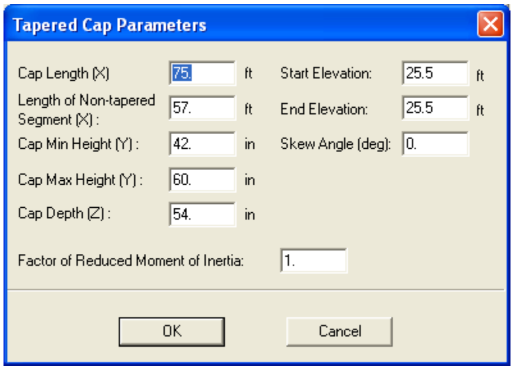
Strut and Tie Model for LRFD
☐ STM for Hammer Head
☐ STM for Isolated Pile Cap

OK Cancel

Figure T2-8 Pier Configuration Screen

Step 4

Click **Cap**. The *Tapered Cap Parameters* screen will open, as shown in Figure T2-9. Enter **75** in the Cap Length field, **57** in the Length of Non-tapered Segment field, **42** in the Cap Min. Height field, **60** in the Cap Max Height field, **54** in the Cap Depth field, and **25.5** in the Start and End Elevation fields. Click **OK**.



Tapered Cap Parameters

Cap Length (X) 75. ft

Length of Non-tapered Segment (X) : 57. ft

Cap Min Height (Y) : 42. in

Cap Max Height (Y) : 60. in

Cap Depth (Z) : 54. in

Start Elevation: 25.5 ft

End Elevation: 25.5 ft

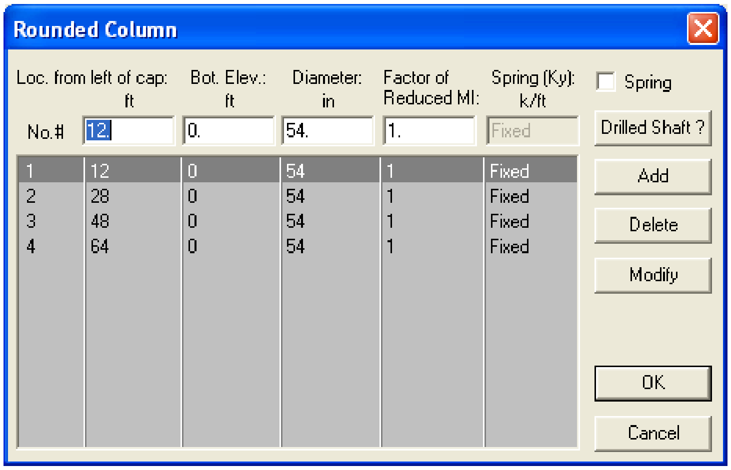
Skew Angle (deg): 0.

Factor of Reduced Moment of Inertia: 1.

Figure T2-9 Tapered Cap Parameters Screen

Step 5

Click **Column** to bring up the *Rounded Column* screen, as shown in Figure T2-10.



Rounded Column

No.#	Loc. from left of cap: ft	Bot. Elev.: ft	Diameter: in	Factor of Reduced MI:	Spring (Ky): k/ft
	12.	0.	54.	1.	Fixed
1	12	0	54	1	Fixed
2	28	0	54	1	Fixed
3	48	0	54	1	Fixed
4	64	0	54	1	Fixed

☐ Spring

Figure T2-10 Rounded Column Screen

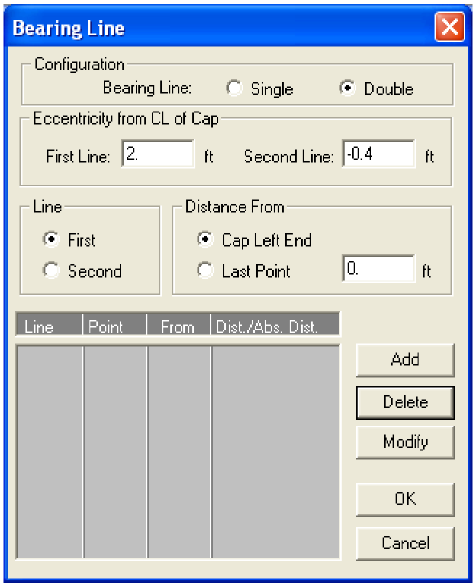
1. To input the first column, enter **12** in the Loc from left cap field, **0** in the Bot. Elev field, **54** in the Diameter field, and **1** in the Factor of Reduced MI field. Click **Add**. The column will appear in the list on the screen (Figure T2-10).
2. Repeat the above step to input the remaining three columns, as shown in Figure T2-10.
3. Click **OK** and return to the *Geometry* screen.

To modify a column, highlight it in the list, make the necessary changes, and click **Modify**. To delete a column, highlight it and click **Delete**.

Step 6

Click **Bearing** to bring up the *Bearing Line* screen. You will be entering two bearing lines, as shown in Figure T2-11 and Figure T2-12, starting with the first bearing line:

1. Select the **Double** option under Configuration and input **2** in the First Line field and **-0.4** in the Second Line field (under Eccentricity from CL of Cap).
2. Input the bearing points for the first line. Select **First** under Line, and then select **Cap Left End** under Distance From and input **3.06** in the text box.
3. Click **Add**. The first bearing point for the first line will appear in the list at the bottom of the screen.



Bearing Line: ○ Single ● Double

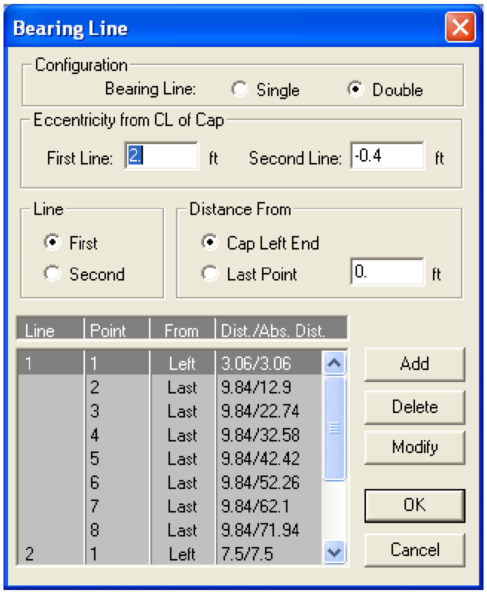
First Line: 2. ft Second Line: -0.4 ft

● First ○ Second

● Cap Left End ○ Last Point 0. ft

Figure T2-11 Bearing Line Screen - Showing First Bearing Line

4. For the remaining 7 points, repeat the above steps except under Distance From, select the **Last Point** option and input **9.84** in the text box (for each point). When completed, there will be a total of 8 bearing points for the first line (Figure T2-11).
5. To input the bearings for the second line, select **Second** under Line and then select **Cap Left End** and input **7.5** in the text box under Distance From.
6. Click **Add** to add it to the list on the screen.
7. For the remaining 5 points, repeat #5-6 except select the **Last Point** option and input **12** in the text box (for each point).
8. When completed, there will be a total of 6 bearing points on the second line (Figure T2-12).



Configuration

Bearing Line: ○ Single ● Double

Eccentricity from CL of Cap

First Line: 2 ft Second Line: -0.4 ft

Line

● First

○ Second

Distance From

● Cap Left End

○ Last Point 0. ft

Line	Point	From	Dist./Abs. Dist.
1	1	Left	3.06/3.06
	2	Last	9.84/12.9
	3	Last	9.84/22.74
	4	Last	9.84/32.58
	5	Last	9.84/42.42
	6	Last	9.84/52.26
	7	Last	9.84/62.1
	8	Last	9.84/71.94
2	1	Left	7.5/7.5

Figure T2-12 Bearing Line Screen - Showing Second Bearing Line

9. Click **OK** and return to the *Geometry* screen.

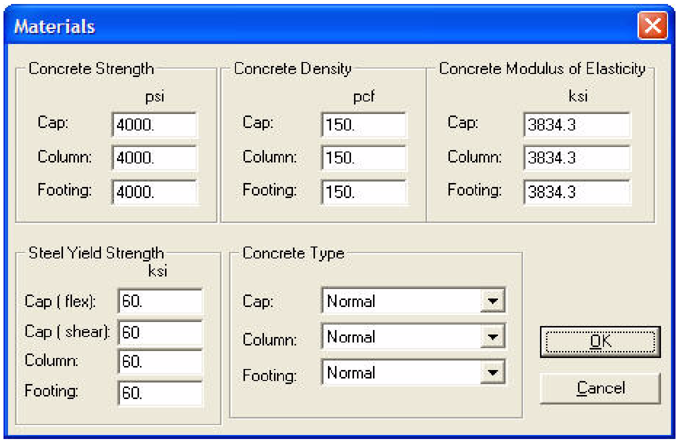
The following table shows all the bearing points for each line and the distance from the left edge of the pier cap.

Table T1-1

Bearing Line #1		Bearing Line #2	
Bearing Point	Distance from left edge of pier cap (ft)	Bearing Point	Distance from left edge of pier cap (ft)
1	3.06	1	7.50
2	12.90	2	19.50
3	22.74	3	31.50
4	32.58	4	43.50
5	42.42	5	55.50
6	52.26	6	67.50
7	62.10		
8	71.94		

Step 7

Click **Material** to open the *Materials* screen, as shown in Figure T2-13. This screen defines the strength, density, type of elasticity of concrete, and reinforcing steel of the pier.



Materials

Concrete Strength
psi
Cap: 4000.
Column: 4000.
Footing: 4000.

Concrete Density
pcf
Cap: 150.
Column: 150.
Footing: 150.

Concrete Modulus of Elasticity
ksi
Cap: 3834.3
Column: 3834.3
Footing: 3834.3

Steel Yield Strength
ksi
Cap (flex): 60.
Cap (shear): 60
Column: 60.
Footing: 60.

Concrete Type
Cap: Normal
Column: Normal
Footing: Normal

Figure T2-13 Materials Screen

Notice that the program defaults to certain values. Override these values by typing over them. Input the values shown in Figure T2-13 and click **OK** when completed and return to the *Geometry* screen.

Step 8

Select **Image** from the **Show** menu to open the *Image* screen. A 3-D image of the structure will be displayed on the screen, as shown in Figure T2-14. Experiment with the buttons left of the pane to become more familiar with their functions. Close or minimize the screen and return to the *Geometry* screen.

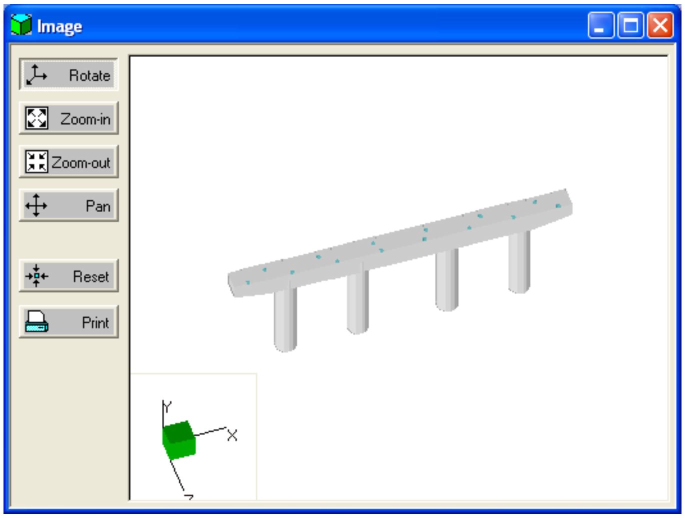


Figure T2-14 Image Screen

Step 9

Select **Model** from the **Show** menu to open the *Model* screen, as shown in Figure T2-15. A 3-D model of the nodes and elements will be displayed on the screen.

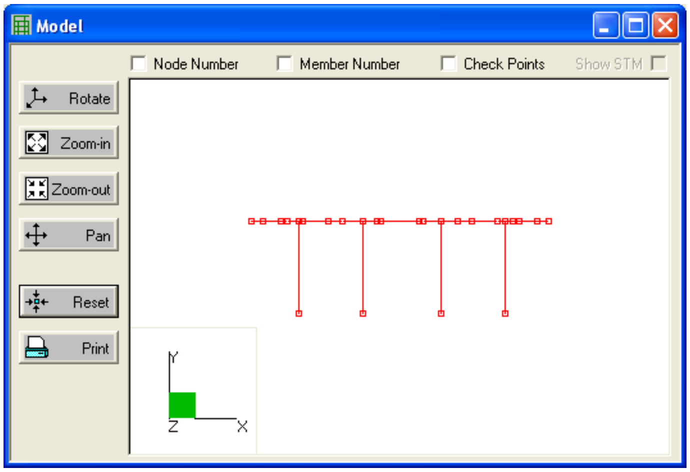
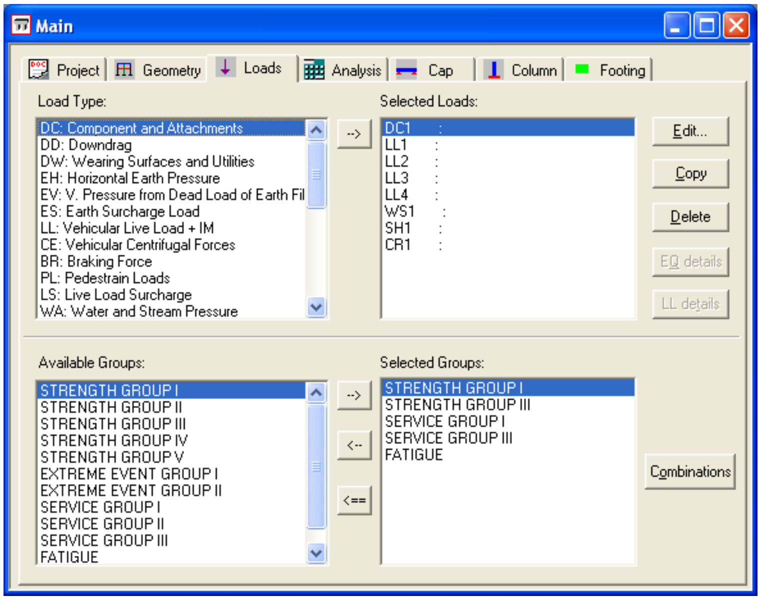


Figure T2-15 Model Screen

The type of model displayed depends on which characteristics you select from the check boxes at the top of the screen (Node Number, Member Number, and Check Points). Use the buttons on the left side of the screen to manipulate the view of the model. Close or minimize the screen and return to the *Geometry* screen.

Step 10

Click the **Loads** tab to open the *Loads* screen. Add the load types to the Selected Load list, by highlighting the load type in the list under Load Type and clicking the -> button. The load type will appear in the list under Selected Loads. Repeat the above steps until you enter all the required loads as shown in Figure T2-16.



Load Type:

DC: Component and Attachments
DD: Downdrag
DW: Wearing Surfaces and Utilities
EH: Horizontal Earth Pressure
EV: V. Pressure from Dead Load of Earth Fil
ES: Earth Surcharge Load
LL: Vehicular Live Load + IM
CE: Vehicular Centrifugal Forces
BR: Braking Force
PL: Pedestrain Loads
LS: Live Load Surcharge
WA: Water and Stream Pressure

Selected Loads:

DC1 :
LL1 :
LL2 :
LL3 :
LL4 :
WS1 :
SH1 :
CR1 :

Available Groups:

STRENGTH GROUP I
STRENGTH GROUP II
STRENGTH GROUP III
STRENGTH GROUP IV
STRENGTH GROUP V
EXTREME EVENT GROUP I
EXTREME EVENT GROUP II
SERVICE GROUP I
SERVICE GROUP II
SERVICE GROUP III
FATIGUE

Selected Groups:

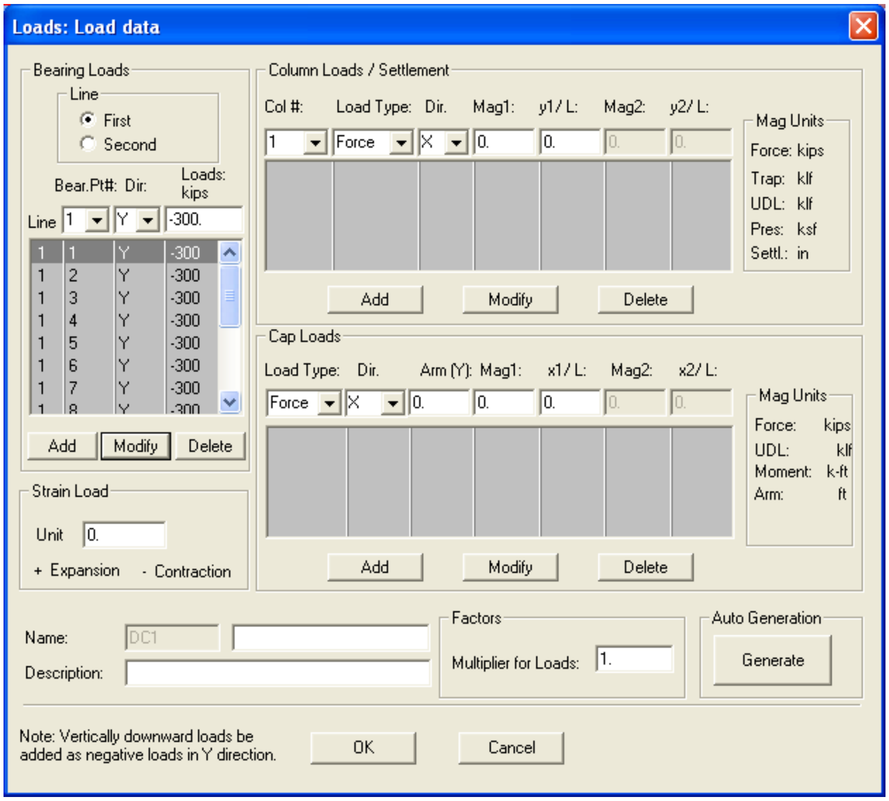
STRENGTH GROUP I
STRENGTH GROUP III
SERVICE GROUP I
SERVICE GROUP III
FATIGUE

Figure T2-16 Loads Tab Screen

Next, add the load groups to the Selected Groups list in a similar manner as above. Highlight the load group in the list under Available Groups and click the --> button. The load group will appear in the list under Selected Groups. Repeat the above steps until you select all required load groups as shown in Figure T2-16.

To manually add Bearing Loads follow the steps below.

1. Select the Load Type and click the --> button to place the selected load in the Selected Loads list.
2. Select the Load in the Selected Loads list and click **Edit**. The Load Data screen will appear as shown in Figure T2-17.



Loads: Load data

Bearing Loads

Line: ● First ○ Second

Line 1, Bear.Pt#: 1, Dir: Y, Loads: kips -300.

1	1	Y	-300
1	2	Y	-300
1	3	Y	-300
1	4	Y	-300
1	5	Y	-300
1	6	Y	-300
1	7	Y	-300
1	8	Y	-300

Strain Load

Unit: 0.

+ Expansion - Contraction

Column Loads / Settlement

Col #: 1, Load Type: Force, Dir.: X, Mag1: 0., y1 / L: 0., Mag2: 0., y2/ L: 0.

Mag Units

Force: kips

Trap: klf

UDL: klf

Pres: ksf

Settl.: in

Cap Loads

Load Type: Force, Dir.: X, Arm (Y): 0., Mag1: 0., x1/ L: 0., Mag2: 0., x2/ L: 0.

Mag Units

Force: kips

UDL: klf

Moment: k-ft

Arm: ft

Name: DC1

Description:

Factors

Multiplier for Loads: 1.

Auto Generation

Note: Vertically downward loads be added as negative loads in Y direction.

Figure T2-17 Load Data Screen

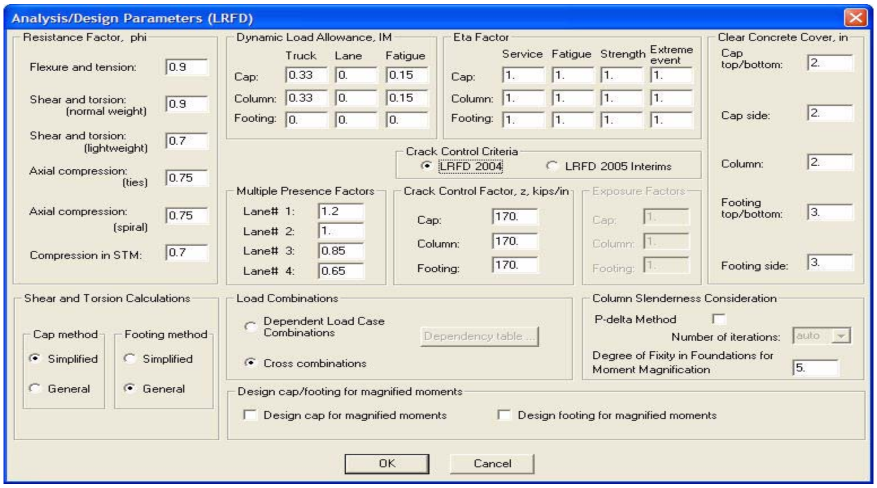
3. In the Bearing Loads section, select the Line, enter the Bearing Point number (Bear.Pt#), Direction (Dir), and Loads (kips). The values used in the example are: Line: **First**, Bear.Pt#: **1**, Dir: **Y**, Loads: **-300**.
4. Click **Add** to add the Bearing Loads list. Continue this process for all bearing points.

To remove a selected group from the Selected Groups list by clicking the <- button. To remove all groups from the Selected Groups list, click the <== button. To define all groups, select the **Load Groups/Limit States** item from the **Libraries** menu. (See page GO-69 for more information on this command.) Click on each load case in the Selected Loads list and click **Edit**. Then manually input the loads for each load case.

Step 11

Click the **Analysis** tab to activate the *Analysis* screen. Perform an analysis and, if desired, specify various factors relating to the analysis and design.

Click **A/D Parameters** to bring up the *Analysis/Design Parameters (LRFD)* screen and input the values as shown in Figure T2-18. Click **OK** to accept any changes and return to the *Analysis* screen. (Click **Cancel** to ignore any changes.)



Resistance Factor, phi

Flexure and tension:	0.9
Shear and torsion: (normal weight)	0.9
Shear and torsion: (lightweight)	0.7
Axial compression: (ties)	0.75
Axial compression: (spiral)	0.75
Compression in STM:	0.7

Dynamic Load Allowance, IM

	Truck	Lane	Fatigue
Cap:	0.33	0.	0.15
Column:	0.33	0.	0.15
Footing:	0.	0.	0.

Eta Factor

	Service	Fatigue	Strength	Extreme event
Cap:	1.	1.	1.	1.
Column:	1.	1.	1.	1.
Footing:	1.	1.	1.	1.

Clear Concrete Cover, in

Cap top/bottom:	2.
Cap side:	2.
Column:	2.
Footing top/bottom:	3.
Footing side:	3.

Crack Control Criteria
● LRFD 2004 ○ LRFD 2005 Interims

Multiple Presence Factors

Lane# 1:	1.2
Lane# 2:	1.
Lane# 3:	0.85
Lane# 4:	0.65

Crack Control Factor, z, kips/in

Cap:	170.
Column:	170.
Footing:	170.

Exposure Factors

Cap:	1.
Column:	1.
Footing:	1.

Shear and Torsion Calculations
Cap method: ● Simplified ○ General
Footing method: ○ Simplified ● General

Load Combinations
○ Dependent Load Case Combinations
● Cross combinations

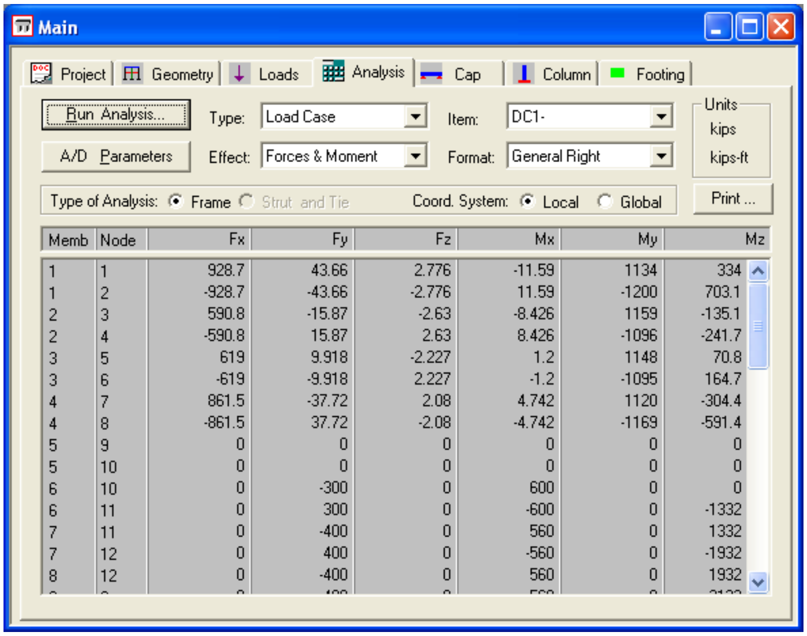
Column Slenderness Consideration
P-delta Method ☐
Number of iterations: auto
Degree of Fixity in Foundations for Moment Magnification: 5.

Design cap/footing for magnified moments
☐ Design cap for magnified moments ☐ Design footing for magnified moments

Figure T2-18 Analysis/Design Parameters (LRFD) Screen

Step 12

Click **Run Analysis** to perform the analysis for the pier structure based on all the data entered up to this point. The results will appear on the screen, as shown in Figure T2-19. If necessary, use the scroll bar on the right side of the screen to view all the results. Specify the type of results to view by using the lists at the top of the screen.



Type: Load Case
Effect: Forces & Moment
Item: DC1-
Format: General Right
Units: kips, kips-ft
Type of Analysis: ● Frame ○ Strut and Tie
Coord. System: ● Local ○ Global

Memb	Node	Fx	Fy	Fz	Mx	My	Mz
1	1	928.7	43.66	2.776	-11.59	1134	334
1	2	-928.7	-43.66	-2.776	11.59	-1200	703.1
2	3	590.8	-15.87	-2.63	-8.426	1159	-135.1
2	4	-590.8	15.87	2.63	8.426	-1096	-241.7
3	5	619	9.918	-2.227	1.2	1148	70.8
3	6	-619	-9.918	2.227	-1.2	-1095	164.7
4	7	861.5	-37.72	2.08	4.742	1120	-304.4
4	8	-861.5	37.72	-2.08	-4.742	-1169	-591.4
5	9	0	0	0	0	0	0
5	10	0	0	0	0	0	0
6	10	0	-300	0	600	0	0
6	11	0	300	0	-600	0	-1332
7	11	0	-400	0	560	0	1332
7	12	0	400	0	-560	0	-1932
8	12	0	-400	0	560	0	1932

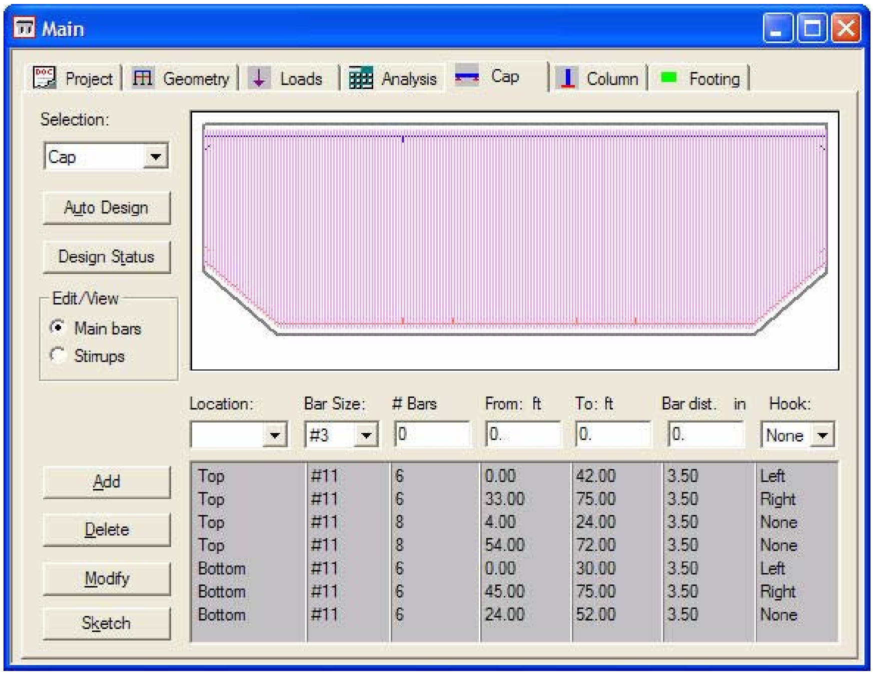
Figure T2-19 Analysis Tab Screen (After Analysis is Performed)

Step 13

At this point, it is recommended to save your project. Select the **Save As** item from the **File** menu. This will display the *Save As* screen. Enter a name for the file in the File Name field (e.g., Mytutor2). The default extension is "*.rcp". Click **Save**.

Step 14

Click the **Cap** tab to activate the *Cap* screen, as shown in Figure T2-20. First, select **Cap** from the Selection list and then choose to edit main rebars.



Selection: Cap

Edit/View
● Main bars
○ Stirrups

Location:	Bar Size:	# Bars	From: ft	To: ft	Bar dist. in	Hook:
	#3	0	0.	0.	0.	None
Top	#11	6	0.00	42.00	3.50	Left
Top	#11	6	33.00	75.00	3.50	Right
Top	#11	8	4.00	24.00	3.50	None
Top	#11	8	54.00	72.00	3.50	None
Bottom	#11	6	0.00	30.00	3.50	Left
Bottom	#11	6	45.00	75.00	3.50	Right
Bottom	#11	6	24.00	52.00	3.50	None

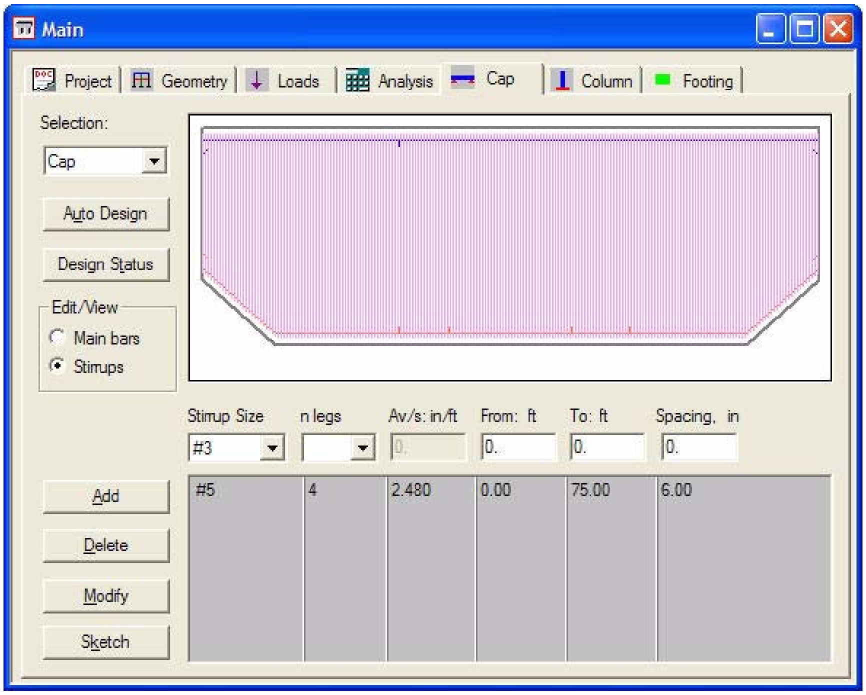
Figure T2-20 Cap Tab Screen Showing Main Rebars

First manually input the main reinforcement of the cap:

1. First choose the Main rebar option in the **Edit/View** frame.
2. Select **Top** from the Location list and **11** from the Bar Size list. Input **6** in the #Bars field, **0** in the From field, **42** in the To field, **3.5** in the Bar Dist field, and select **Left** from the Hook list.
3. Click **Add**. The cap reinforcement will appear in the list on the screen. Also, a graphical sketch of the cap will appear in the top portion of the screen
4. Repeat the above steps for the remaining bars, as shown in Figure T2-20.

Next manually input the stirrups:

5. Choose Stirrups option in the **Edit/View** frame to manually specify transverse reinforcement.
6. Specify the first set from 0 ft to 75 ft. Choose **#5** as the stirrups and **4** as nlegs with spacing of 6 inches.
7. Click **Add**. The stirrup set will appear in the list on the screen. Also the graphic will be updated.



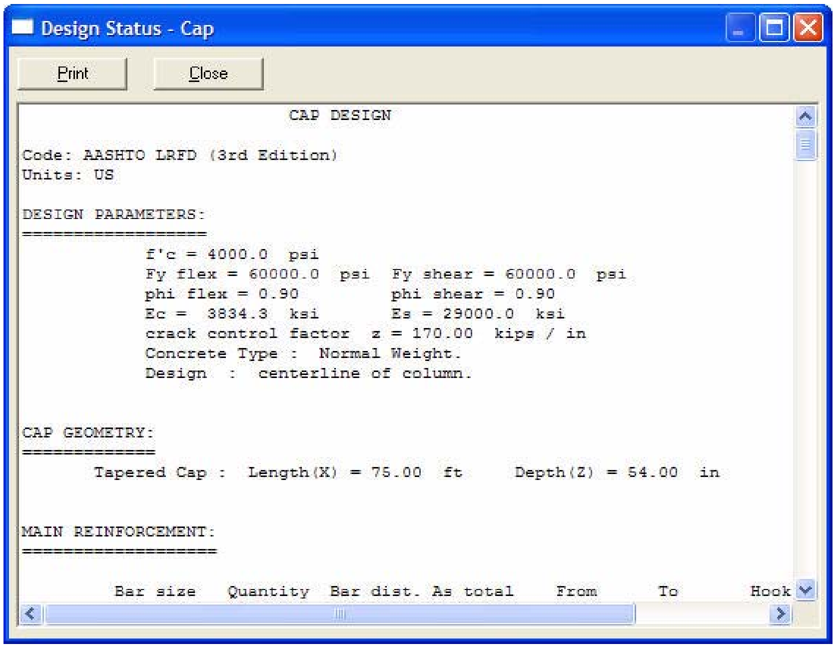
Selection: Cap

Edit/View
○ Main bars
● Stirrups

Stirrup Size	n legs	Av/s: in/ft	From: ft	To: ft	Spacing, in
#3		0.	0.	0.	0.
#5	4	2.480	0.00	75.00	6.00

Figure T2-21 Cap Tab Screen Showing Stirrups

Click **Design Status** on the *Cap* screen to see a design summary of the cap as shown in Figure T2-22.



```
                          CAP DESIGN

Code: AASHTO LRFD (3rd Edition)
Units: US

DESIGN PARAMETERS:
==================
            f'c = 4000.0  psi
            Fy flex = 60000.0  psi  Fy shear = 60000.0  psi
            phi flex = 0.90         phi shear = 0.90
            Ec =  3834.3  ksi       Es = 29000.0  ksi
            crack control factor  z = 170.00  kips / in
            Concrete Type :  Normal Weight.
            Design  :  centerline of column.


CAP GEOMETRY:
=============
       Tapered Cap :  Length(X) = 75.00  ft     Depth(Z) = 54.00  in


MAIN REINFORCEMENT:
===================

         Bar size   Quantity  Bar dist. As total    From      To       Hook
```

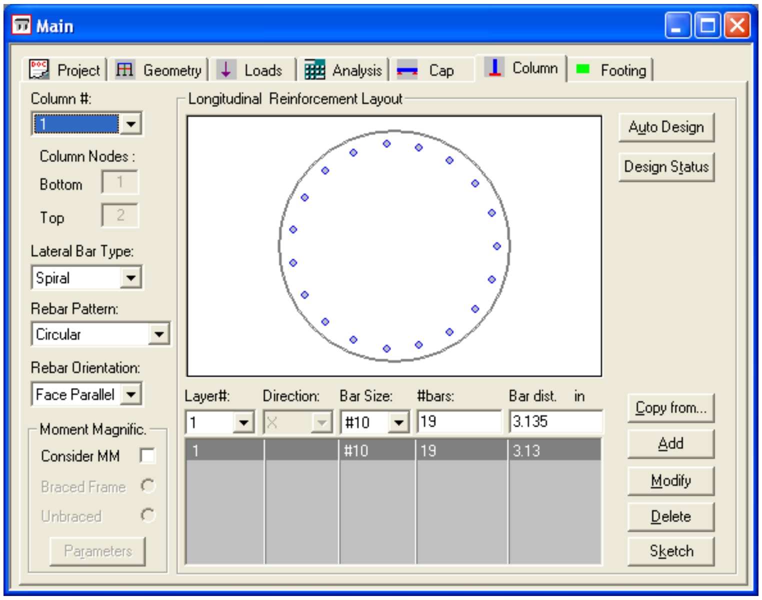
Figure T2-22 Design Status - Cap Screen

To view a graphical representation of the cap with multiple views, click **Sketch** on the *Cap* screen.

Note: The printout for shear/torsion design is given in terms of the sides of a section. For the section other than the start or end of a span, it has two sides: left and right. The start and end of the span have only one side.

Step 15

Click the **Column** tab to activate the *Column* screen. For this tutorial, manually input the column design, as follows. First, select **1** from the Column# list, **Spiral** from the Lateral Bar Type list. Select Rebar Pattern as Circular and Rebar Orientation as Face Parallel. Uncheck the **Consider MM** check box and the **Braced Framed** option. Then, input the values for the column reinforcement. Select **1** from the Layer# list, **#10** from the Bar Size list and input **19** in the #Bars field and **3.135** in the Bar Dist field. Click **Add**. It will appear in the list on the screen, as shown in Figure T2-23. A diagram of the column design will appear in the display window on the screen.



Column #: 1

Column Nodes :

Bottom 1

Top 2

Lateral Bar Type: Spiral

Rebar Pattern: Circular

Rebar Orientation: Face Parallel

Consider MM ☐

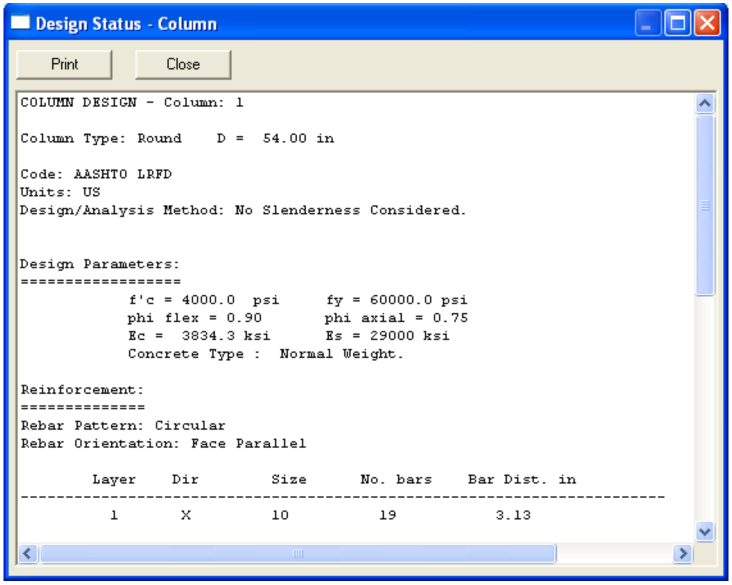
Braced Frame ○

Unbraced ○

Layer#:	Direction:	Bar Size:	#bars:	Bar dist. in
1	X	#10	19	3.135
1		#10	19	3.13

Figure T2-23 Column Tab Screen

Click the **Design Status** on the *Column* screen to see a design summary of the selected column components, as shown in Figure T2-24. To print the design summary, click **Print**.



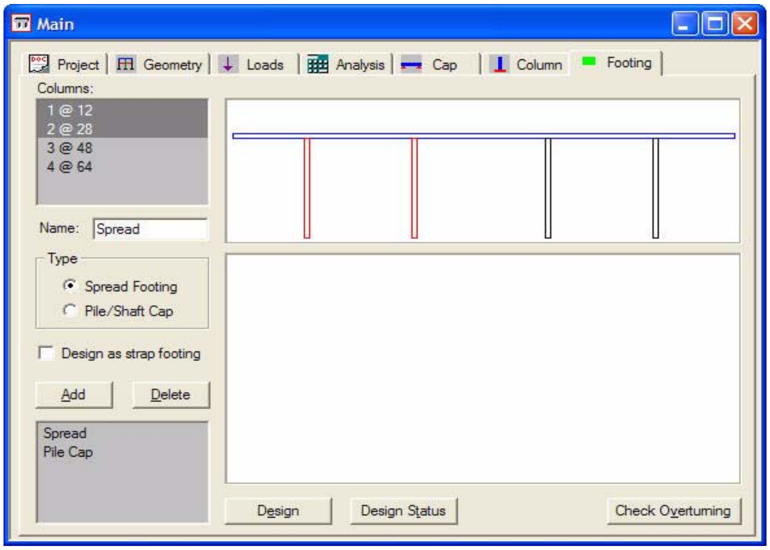
```
COLUMN DESIGN - Column: 1

Column Type: Round    D =  54.00 in

Code: AASHTO LRFD
Units: US
Design/Analysis Method: No Slenderness Considered.


Design Parameters:
==================
            f'c = 4000.0  psi      fy = 60000.0 psi
            phi flex = 0.90        phi axial = 0.75
            Ec =  3834.3 ksi       Es = 29000 ksi
            Concrete Type :  Normal Weight.

Reinforcement:
==============
Rebar Pattern: Circular
Rebar Orientation: Face Parallel

        Layer    Dir        Size      No. bars    Bar Dist. in
------------------------------------------------------------------------
          1       X         10          19           3.13
```

Figure T2-24 Design Status - Column Screen

To view a graphical representation of the selected column, click **Sketch** on the *Column* screen.

Step 16

Select the **Footing** tab to activate the *Footing* screen, as shown in Figure T2-25. For this tutorial, you will define a rectangular spread footing under the first two columns, and a combined rectangular pile footing under the last two columns. Start with the rectangular spread footing design.



Columns:
1 @ 12
2 @ 28
3 @ 48
4 @ 64

Name: Spread

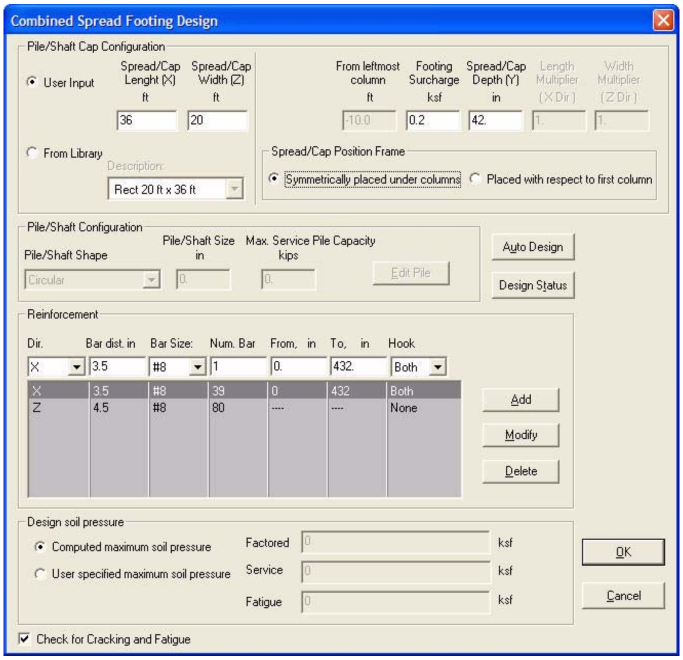
Type
● Spread Footing
○ Pile/Shaft Cap

☐ Design as strap footing

Spread
Pile Cap

Figure T2-25 Footing Tab Screen

1. Select the first two columns, **1@12** and **2@28** under Columns for a combined footing. Input **Spread** in the Name field, and select **Spread Footing** under Type. Click **Add**. The name “Spread” will appear in the list at the bottom of the screen.
2. Click **Design** to activate the *Combined Spread Footing Design* screen, as shown in Figure T2-26.



Pile/Shaft Cap Configuration

● User Input

○ From Library

Description: Rect 20 ft x 36 ft

Spread/Cap Lenght (X) ft	Spread/Cap Width (Z) ft	From leftmost column ft	Footing Surcharge ksf	Spread/Cap Depth (Y) in	Length Multiplier (X Dir)	Width Multiplier (Z Dir)
36	20	-10.0	0.2	42.	1.	1.

Spread/Cap Position Frame

● Symmetrically placed under columns ○ Placed with respect to first column

Pile/Shaft Configuration

Pile/Shaft Shape	Pile/Shaft Size in	Max. Service Pile Capacity kips
Circular	0.	0.

Reinforcement

Dir.	Bar dist. in	Bar Size:	Num. Bar	From, in	To, in	Hook
X	3.5	#8	1	0.	432.	Both
X	3.5	#8	39	0	432	Both
Z	4.5	#8	80	----	----	None

Design soil pressure

● Computed maximum soil pressure

○ User specified maximum soil pressure

Factored 0 ksf

Service 0 ksf

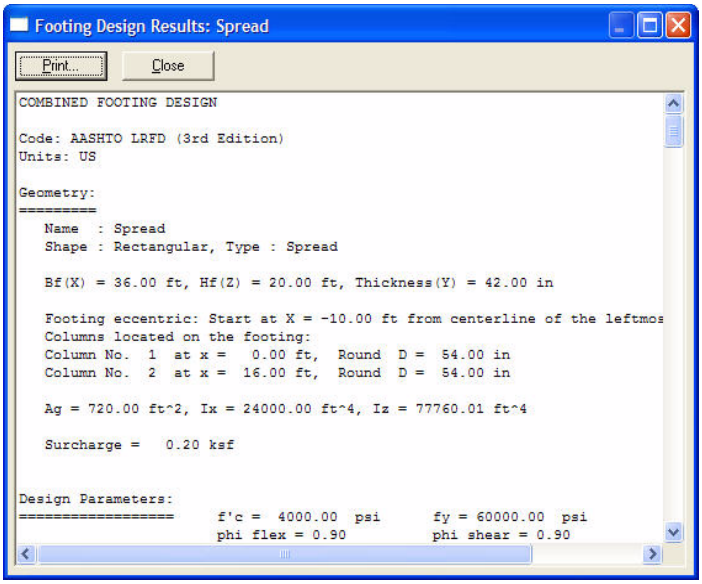
Fatigue 0 ksf

☑ Check for Cracking and Fatigue

Figure T2-26 Combined Spread Footing Design Screen

3. Select User Input under Pile / Shaft Cap Configuration.
4. Specify Spread Cap Length (X) as 36 ft and Width (Z) as 20 ft.
5. Choose symmetrically placed under columns in Spread/Cap Position.
6. Input **0.2** in the Footing Surcharge field and **42** in the Spread/Cap Depth.
7. Input the reinforcement. Select **X** from the Direction list. Input **3.5** in the Bar Dist field, **#8** in the Bar Size field, **39** in the Num Bar field, **0** in the From field, **432** in the To field, and select **Both** from the Hook list.
8. Click **Add**. It will appear in the list on the screen.
9. Select **Z** from the Direction list. Input **4.5** in the Bar Dist field, **#8** from the Bar Size list, **80** in the Num Bar field, and select **None** from the Hook list.
10. Click **Add** to add it to the list on the screen.

11. Click **Design Status** to display a design summary of the selected footing, as shown in Figure T2-27. Click **Close** to exit this screen and return to the *Combined Spread Footing Design* screen.



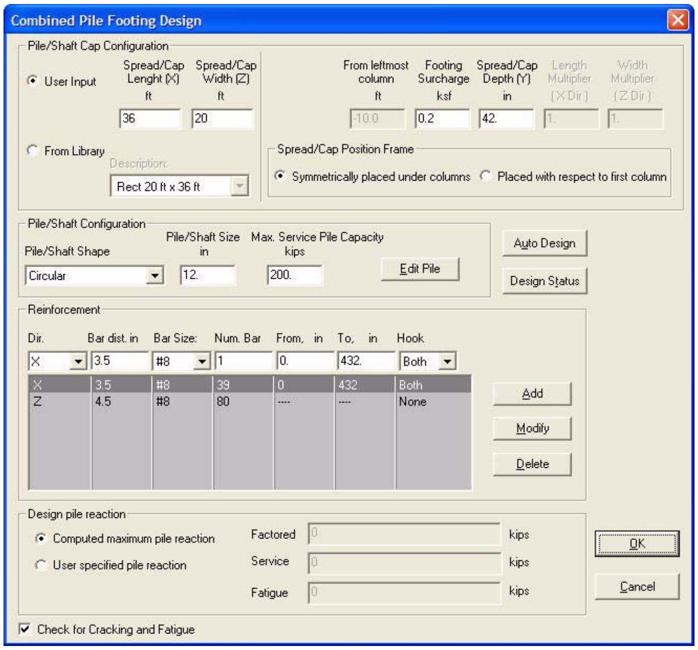
```
COMBINED FOOTING DESIGN

Code: AASHTO LRFD (3rd Edition)
Units: US

Geometry:
=========
   Name  : Spread
   Shape : Rectangular, Type : Spread

   Bf(X) = 36.00 ft, Hf(Z) = 20.00 ft, Thickness(Y) = 42.00 in

   Footing eccentric: Start at X = -10.00 ft from centerline of the leftmos
   Columns located on the footing:
   Column No.  1  at x =   0.00 ft,  Round  D =  54.00 in
   Column No.  2  at x =  16.00 ft,  Round  D =  54.00 in

   Ag = 720.00 ft^2, Ix = 24000.00 ft^4, Iz = 77760.01 ft^4

   Surcharge =   0.20 ksf


Design Parameters:
==================      f'c =  4000.00  psi      fy = 60000.00  psi
                        phi flex = 0.90          phi shear = 0.90
```

Figure T2-27 Footing Design Results - Spread Screen

12. Click **OK** and return to the *Footing* screen. A graphical representation of the footing will appear in the display window on the screen.

13. Now, define the Combined Rectangular Pile Footing Design. Select **3@48** and **4@64** under Columns, input **Pile cap** in the Name field, and select **Pile/Shaft Cap** under Type. Click **Add**. The name **Pile cap** will appear in the list at the bottom of the screen.

14. Click **Design** to activate the *Combined Pile Footing Design* screen, as shown in Figure T2-28.



● User Input

Spread/Cap Lenght (X) ft: 36

Spread/Cap Width (Z) ft: 20

○ From Library

Description: Rect 20 ft x 36 ft

From leftmost column ft: -10.0

Footing Surcharge ksf: 0.2

Spread/Cap Depth (Y) in: 42.

Length Multiplier (X Dir) 1.

Width Multiplier (Z Dir) 1.

● Symmetrically placed under columns ○ Placed with respect to first column

Pile/Shaft Shape: Circular

Pile/Shaft Size in: 12.

Max. Service Pile Capacity kips: 200.

Dir.	Bar dist. in	Bar Size:	Num. Bar	From, in	To, in	Hook
X	3.5	#8	1	0.	432.	Both
X	3.5	#8	39	0	432	Both
Z	4.5	#8	80	----	----	None

● Computed maximum pile reaction

○ User specified pile reaction

Factored 0 kips

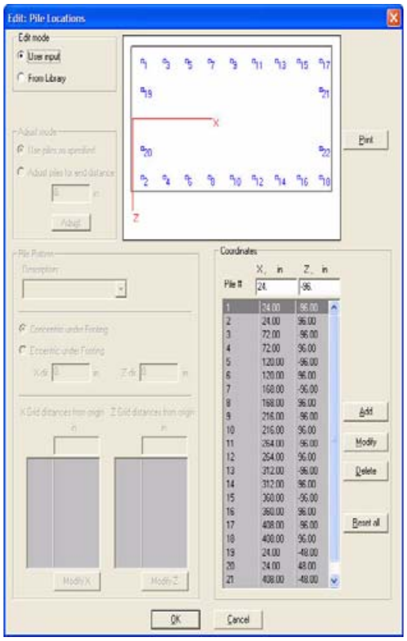
Service 0 kips

Fatigue 0 kips

☑ Check for Cracking and Fatigue

Figure T2-28 Combined Pile Footing Design Screen

15. Select User Input under Pile / Shaft Cap Configuration.
16. Specify Spread Cap Length (X) as 36 ft and Width (Z) as 20 ft.
17. Choose symmetrically placed under columns option as the Spread / Cap Position.
18. Input **0.2** in the Footing Surcharge field, **42** in the Spread/Cap Depth field.
19. Select **Circular** from the Pile/Shaft Shape list and input **12** in the Pile/Shaft Size field and **200** in the Max. Pile Capacity field.
20. Click Edit Pile to activate the Edit: Pile locations screen, as sown in Figure T2-28.
21. Select User Input in the Edit Mode.
22. Input 24 in the X field and -96 in the Z field. Click Add. This is the first pile location.
23. Click **Edit Pile** to activate the *Edit: Pile Locations* screen, as shown in Figure T2-29.



Pile # X, in 24. Z, in -96.

Pile #	X, in	Z, in
1	24.00	-96.00
2	24.00	96.00
3	72.00	-96.00
4	72.00	96.00
5	120.00	-96.00
6	120.00	96.00
7	168.00	-96.00
8	168.00	96.00
9	216.00	-96.00
10	216.00	96.00
11	264.00	-96.00
12	264.00	96.00
13	312.00	-96.00
14	312.00	96.00
15	360.00	-96.00
16	360.00	96.00
17	408.00	-96.00
18	400.00	96.00
19	24.00	-48.00
20	24.00	48.00
21	408.00	-48.00

Figure T2-29 Edit: Pile Locations Screen

24. Select Usr Input.
25. Input **24** in the X field and **-96** in the Z field. Click **Add**. This is the first pile location.
26. Repeat this for the remaining pile locations listed in the following table. There should be a total of 22 pile locations.

Table T2-27

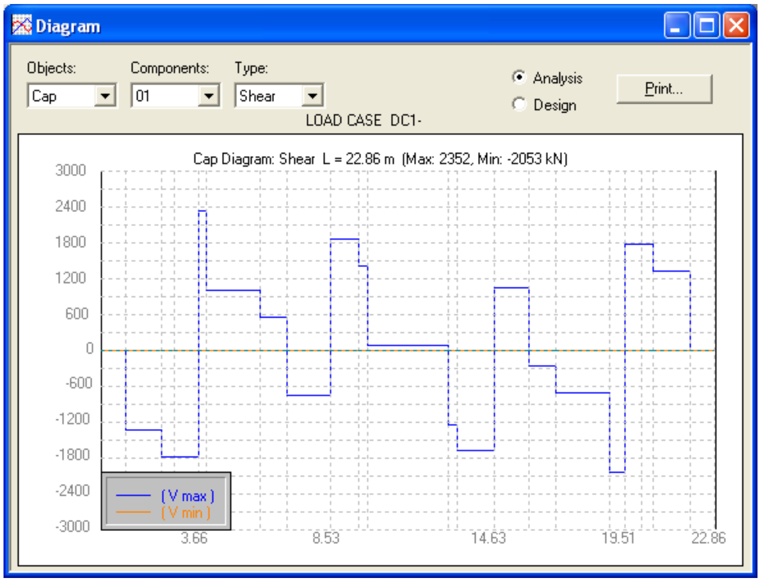
Pile Locations for Tutorial 2					
Pile #	X (in)	Z (in)	Pile #	X (in)	Z (in)
1	24	-96	12	264	96
2	24	96	13	312	-96
3	72	-96	14	312	96
4	72	96	15	360	-96

5	120	-96	16	360	96
6	120	96	17	408	-96
7	168	-96	18	408	96
8	168	96	19	24	-48
9	216	-96	20	24	48
10	216	96	21	408	-48
11	264	-96	22	408	48

28. Click **OK** to return to the *Combined Pile Footing Design* screen.
29. Input the reinforcement. Select **X** from the Direction list, and input **3.5** in the Bar Dist field, **#8** in the Bar Size field, **39** in the Num Bar field, **0** in the From field, **432** in the To field, and select **Both** from the Hook list.
30. Click **Add**. It will appear in the list on the screen.
31. Select **Z** from the Direction list. Input **4.5** in the Bar Dist field, **#8** in the Bar Size field, **80** in the Num Bar field, and **None** from the Hook list.
32. Click **Add**. It will appear in the list on the screen.
33. When completed, click **OK** and return to the *Footing* screen.

Step 17

RC-PIER provides you with many diagrams for your project. Select **Diagrams** from the **Show** menu to activate the *Diagram* screen, as shown in Figure T2-30. Experiment with the lists and buttons to become familiar with the options of this feature.



Objects: Cap

Components: 01

Type: Shear

● Analysis

○ Design

LOAD CASE DC1-

Cap Diagram: Shear L = 22.86 m (Max: 2352, Min: -2053 kN)

Figure T2-30 Diagram Screen

Step 18

This completes RC-PIER Tutorial 2. To print the output of the project, select **Print** from the **File** menu. The *Print* screen will display. Select the appropriate options and click **OK**.

Save your file by selecting **Save** from the **File** menu. Exit the program by selecting **Exit** from the **File** menu.

Following this tutorial, is a printout of the output for selected items from Tutorial 2.

```
PROJECT DATA
============

Project     : Tutorial 2 LRFD
User Job No.: Tutor2
State       : FL                             State Job No. : FL-002
Code        : AASHTO LRFD (3rd edition)
Comments    : Tapered beam
              Two bearing lines with eccentricity
              Combined rectangular spread footing under the first two columns
              Combined rectangular pile cap under the third and fourth columns.




PIER GEOMETRY
=============

Pier Type: Multi Column

Cap Shape: Tapered   Top Elevations: start = 25.50 ft   end = 25.50 ft
   Depth(Z) = 54.00 in   Skew angle = 0.00   Reduction of I = 1.000
   Length(X) = 75.00 ft   Non tapered length = 57.00 ft
   Height max(Y) = 60.00 in  Height min(Y) = 42.00 in

Column Shape : Round
   Number of columns: 4


   Column number 1:

      Location from the left edge of the cap(X): 12.00 ft
      Elevations: bottom  = 0.00 ft   top = 23.75 ft   Reduction of I = 1.000
      Column section dimensions:
            Diameter = 54.00 in

   Column number 2:

      Location from the left edge of the cap(X): 28.00 ft
      Elevations: bottom  = 0.00 ft   top = 23.75 ft   Reduction of I = 1.000
      Column section dimensions:
            Diameter = 54.00 in

   Column number 3:

      Location from the left edge of the cap(X): 48.00 ft
      Elevations: bottom  = 0.00 ft   top = 23.75 ft   Reduction of I = 1.000
      Column section dimensions:
            Diameter = 54.00 in

   Column number 4:

      Location from the left edge of the cap(X): 64.00 ft
      Elevations: bottom  = 0.00 ft   top = 23.75 ft   Reduction of I = 1.000
      Column section dimensions:
            Diameter = 54.00 in
```

```
STRUCTURE MODEL
================

FRAME Model:
                Member   Node   Hinge Check Pt Dist(ft) Memb length(ft)
                -------------------------------------------------------
   Column No. 1
                  1        1       -              0.00
                           2       -             23.75      23.75
   Column No. 2
                  2        3       -              0.00
                           4       -             23.75      23.75
   Column No. 3
                  3        5       -              0.00
                           6       -             23.75      23.75
   Column No. 4
                  4        7       -              0.00
                           8       -             23.75      23.75
   Cap
                  5        9       -              0.00
                          10       -              3.06       3.06
                  6       10       -              3.06
                          11       -              7.50       4.44
                  7       11       -              7.50
                          12       -              9.00       1.50
                  8       12       -              9.00
                           2       -             12.00       3.00
                  9        2       -             12.00
                          13       -             12.90       0.90
                 10       13       -             12.90
                          14       -             19.50       6.60
                 11       14       -             19.50
                          15       -             22.74       3.24
                 12       15       -             22.74
                           4       -             28.00       5.26
                 13        4       -             28.00
                          16       -             31.50       3.50
                 14       16       -             31.50
                          17       -             32.58       1.08
                 15       17       -             32.58
                          18       -             42.42       9.84
                 16       18       -             42.42
                          19       -             43.50       1.08
                 17       19       -             43.50
                           6       -             48.00       4.50
                 18        6       -             48.00
                          20       -             52.26       4.26
                 19       20       -             52.26
                          21       -             55.50       3.24
                 20       21       -             55.50
                          22       -             62.10       6.60
                 21       22       -             62.10
                           8       -             64.00       1.90
                 22        8       -             64.00
                          23       -             66.00       2.00
                 23       23       -             66.00
                          24       -             67.50       1.50
                 24       24       -             67.50
                          25       -             71.94       4.44
                 25       25       -             71.94
```

```
                          26       -            75.00       3.06

   Node coordinates:
      Number         X(ft)        Y(ft)          Node type
      ---------------------------------------------------------
         1           12.00         0.00        fixed at ground
         2           12.00        23.75             column-cap
         3           28.00         0.00        fixed at ground
         4           28.00        23.75             column-cap
         5           48.00         0.00        fixed at ground
         6           48.00        23.75             column-cap
         7           64.00         0.00        fixed at ground
         8           64.00        23.75             column-cap
         9            0.00        23.75
        10            3.06        23.75                bearing
        11            7.50        23.75                bearing
        12            9.00        23.75
        13           12.90        23.75                bearing
        14           19.50        23.75                bearing
        15           22.74        23.75                bearing
        16           31.50        23.75                bearing
        17           32.58        23.75                bearing
        18           42.42        23.75                bearing
        19           43.50        23.75                bearing
        20           52.26        23.75                bearing
        21           55.50        23.75                bearing
        22           62.10        23.75                bearing
        23           66.00        23.75
        24           67.50        23.75                bearing
        25           71.94        23.75                bearing
        26           75.00        23.75


SUPERSTRUCTURE INFO
===================

             No superstructure defined!


BEARING POINTS
==============

   Number of bearing lines: 2
      First bearing line      Eccentricity = 2.00 ft
                 Point      Distance ft
                 ----------------------
                   1            3.06
                   2           12.90
                   3           22.74
                   4           32.58
                   5           42.42
                   6           52.26
                   7           62.10
                   8           71.94

      Second bearing line      Eccentricity = -0.40 ft
             Point          Distance ft
             ------------------------
                   1            7.50
                   2           19.50
```

```
                   3            31.50
                   4            43.50
                   5            55.50
                   6            67.50


MATERIAL PROPERTIES
===================

                                  Cap          Column      Footing
                         ------------------------------------------
   Concrete Type                  normal       normal       normal
   Concrete Strength  (psi)      4000.00      4000.00      4000.00
   Concrete Density (lb/ft3)      150.00       150.00       150.00
   Concrete Modulus Ec  (ksi)    3834.30      3834.30      3834.30
   Steel Strength Fy  (ksi)        60.00        60.00        60.00


DESIGN PARAMETERS
=================

   AASHTO LRFD Code
      Resistance factors for reinf. concrete:         Multi presence factors for live load:
         Flexure and tension           0.90                          1  Lane       1.20
         Shear and torsion (normal)    0.90                          2  Lanes      1.00
                      (lightweight)    0.70                          3  Lanes      0.85
         Axial compression (ties)      0.75              more than 3  Lanes      0.65
         Axial compression (spiral)    0.75
         Compression in STM            0.70

      Eta factor:
                    Service  Fatigue  Strength  Extreme event
         ----------------------------------------------------
         Cap          1.00     1.00      1.00      1.00
         Column       1.00     1.00      1.00      1.00
         Footing      1.00     1.00      1.00      1.00

      Dynamic load allowance IM:
                     Truck     Lane    Fatigue
         --------------------------------------
         Cap          0.33     0.00      0.15
         Column       0.33     0.00      0.15
         Footing      0.00     0.00      0.00

               Crack control factor      Min clear cover
                     kip/ft                     in
         -----------------------------------------------
         Cap            170.00                 2.00
         Column         170.00                 2.00
         Footing        170.00                 3.00

      Degree of fixity in foundations for Moment Magnify Method:  R =  5.00
```

```
LOADS
=====

   Load Cases: 8


      Loadcase ID: DC1     Name:
      Multiplier =  1.000

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          1         Y      -300.00
           1          2         Y      -300.00
           1          3         Y      -300.00
           1          4         Y      -300.00
           1          5         Y      -300.00
           1          6         Y      -300.00
           1          7         Y      -300.00
           1          8         Y      -300.00
           2          1         Y      -100.00
           2          2         Y      -100.00
           2          3         Y      -100.00
           2          4         Y      -100.00
           2          5         Y      -100.00
           2          6         Y      -100.00


      Loadcase ID: LL1     Name:
      Multiplier =  1.000

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          1         Y       -20.00
           1          2         Y        -7.00
           2          1         Y       -45.00
           2          2         Y         6.00


      Loadcase ID: LL2     Name:
      Multiplier =  1.000

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          7         Y       -10.00
           1          8         Y       -29.00
           2          5         Y         4.00
           2          6         Y       -30.00


      Loadcase ID: LL3     Name:
      Multiplier =  1.000

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          2         Y       -43.00
           1          3         Y       -19.00
```

```
      Loadcase ID: LL4     Name:
      Multiplier =  1.000

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          6         Y       -19.00
           1          7         Y       -43.00


      Loadcase ID: WS1     Name:
      Multiplier =  1.000

      Cap loads:
         Type     Dir       Arm          Mag1          x1/L        Mag2        x2/L
                             ft     kip,kip/ft, kft           kip,kip/ft, kft
         ---------------------------------------------------------------------------
----
         Force     X        0.00        40.00          0.50        ----        ----
         UDL       Z        ----         0.47          0.00        ----        1.00

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          1         Y        35.00
           1          8         Y       -35.00
           2          1         Y        42.00
           2          6         Y       -42.00


      Loadcase ID: SH1     Name:
      Multiplier =  1.000

      Unit strain:  -0.000270



      Loadcase ID: CR1     Name:
      Multiplier =  1.000

      Unit strain:  0.000300



   Selected load groups:

      STRENGTH GROUP I
      STRENGTH GROUP III
      SERVICE GROUP I
      SERVICE GROUP III
      FATIGUE
```

```
CAP DESIGN
==========

Code: AASHTO LRFD (3rd Edition)
Units: US

DESIGN PARAMETERS:
==================
            f'c = 4000.0  psi
            Fy flex = 60000.0  psi  Fy shear = 60000.0  psi
            phi flex = 0.90         phi shear = 0.90
            Ec =  3834.3  ksi       Es = 29000.0  ksi
            crack control factor  z = 170.00  kips / in
            Concrete Type :  Normal Weight.
            Design  :  centerline of column.


CAP GEOMETRY:
=============
       Tapered Cap :  Length(X) = 75.00  ft     Depth(Z) = 54.00  in


MAIN REINFORCEMENT:
===================

         Bar size   Quantity  Bar dist. As total    From      To       Hook
                                 in       in^2       ft       ft
------------------------------------------------------------------------------
TOP
          # 11          6       3.50     9.360     0.00     42.00      Left
          # 11          6       3.50     9.360    33.00     75.00      Right
          # 11          8       3.50    12.480     4.00     24.00      None
          # 11          8       3.50    12.480    54.00     72.00      None
BOTTOM
          # 11          6       3.50     9.360     0.00     30.00      Left
          # 11          6       3.50     9.360    45.00     75.00      Right
          # 11          6       3.50     9.360    24.00     52.00      None


STIRRUPS:
==========

    From       To   Stirrup Size       n legs    Spacing        Aprv/s
     ft        ft                                  in         in^2 / ft
------------------------------------------------------------------------------
     0.00     75.00     # 5              4         6.00         2.48

Clear Cover on Sides =   2.00  in


FLEXURE DESIGN:
===============

                             Span 1: From 0.00  ft  To 12.00  ft

 Loc  AbsLoc  H    Mmax     Mr   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-
eff
                 Mmin     Mr   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
  ft    ft   in  kips-ft  kips-ft       in^2     in^2     in^2     in^2     in^2     in^2
```

```
------------------------------------------------------------------------------------------
--
 0.0   0.0   42     0.0     0.0     0   2.08     9.36*    0.00*    2.08     9.36*    0.00*
                    0.0     0.0     0   2.08     9.36*    0.00*    2.08     9.36*    0.00*
 3.1   3.1   48     0.0  1815.1     0   2.38     9.36     9.36     2.38     9.36     9.36
                  -14.8 -1815.1     1   2.38     9.36     9.36     2.38     9.36     9.36
 7.5   7.5   57     0.0  2189.2     0   2.82     9.36     9.36     2.82    21.84    21.84
                -1976.9 -3627.8    10   2.82     9.36     9.36     8.43    21.84    15.76
 9.0   9.0   60     0.0  2315.5     0   2.97     9.36     9.36     2.97    21.84    21.84
                -2986.7 -4488.4     1   2.97     9.36     9.36    12.18    21.84    18.50
12.0  12.0   60     0.0  2315.5     0   2.97     9.36     9.36     2.97    21.84    21.84
                -5056.4 -5269.8     1   2.97     9.36     9.36    21.19    21.84    21.84


                             Span 2: From 12.00  ft  To 28.00  ft

 Loc  AbsLoc  H    Mmax     Mr   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
                   Mmin     Mr   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
  ft    ft   in  kips-ft  kips-ft       in^2     in^2     in^2     in^2     in^2     in^2
------------------------------------------------------------------------------------------
--
 0.0  12.0   60     0.0  2315.5     0   2.97     9.36     9.36     2.97    21.84    21.84
                -3956.5 -5269.8    10   2.97     9.36     9.36    16.33    21.84    21.84
 0.9  12.9   60     0.0  2315.5     0   2.97     9.36     9.36     2.97    21.84    21.84
                -3276.0 -5269.8    10   2.97     9.36     9.36    13.40    21.84    21.84
 7.5  19.5   60     0.0  2315.5     0   2.97     9.36     9.36     2.97    21.84    21.84
                 -928.2 -4272.9     1   2.97     9.36     9.36     4.93    21.84    17.59
10.7  22.7   60    74.1  2315.5     7   2.97     9.36     9.36     2.97    21.84    16.44
                 -226.9 -2866.1     1   2.97     9.36     9.36     2.97    21.84    11.66
16.0  28.0   60     0.0  2831.7     0   2.97    18.72    11.52     2.97     9.36     9.36
                -1414.4 -2315.5     9   2.97    18.72    17.79     6.15     9.36     9.36


                             Span 3: From 28.00  ft  To 48.00  ft

 Loc  AbsLoc  H    Mmax     Mr   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
                   Mmin     Mr   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
  ft    ft   in  kips-ft  kips-ft       in^2     in^2     in^2     in^2     in^2     in^2
------------------------------------------------------------------------------------------
--
 0.0  28.0   60     0.0  2831.7     0   2.97    18.72    11.52     2.97     9.36     9.36
                -1676.1 -2315.5    10   2.97    18.72    17.79     6.72     9.36     9.36
 3.5  31.5   60   439.5  2315.5     9   2.97     9.36     9.36     2.97     9.36     9.36
                    0.0 -2315.5     0   2.97     9.36     9.36     2.97     9.36     9.36
 4.6  32.6   60   887.1  2315.5     9   4.70     9.36     9.36     2.97     9.36     9.36
                    0.0 -2315.5     0   2.97     9.36     9.36     2.97     9.36     9.36
14.4  42.4   60  1145.9  2315.5    10   6.10     9.36     9.36     2.97     9.36     9.36
                    0.0 -2315.5     0   2.97     9.36     9.36     2.97     9.36     9.36
15.5  43.5   60   755.6  2315.5    10   4.00     9.36     9.36     2.97     9.36     9.36
                    0.0 -2315.5     0   2.97     9.36     9.36     2.97     9.36     9.36
20.0  48.0   60     0.0  3289.4     0   2.97    18.72    13.44     2.97     9.36     9.36
                -1696.2 -2315.5     9   2.97    18.72    18.72     6.81     9.36     9.36


                             Span 4: From 48.00  ft  To 64.00  ft

 Loc  AbsLoc  H    Mmax     Mr   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
                   Mmin     Mr   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
  ft    ft   in  kips-ft  kips-ft       in^2     in^2     in^2     in^2     in^2     in^2
```

```
-------------------------------------------------------------------------------------
-----
 0.0  48.0   60    0.0  3289.4    0    2.97    18.72    13.44    2.97    9.36    9.36
              -1514.6 -2315.5    10    2.97    18.72    18.72    6.15    9.36    9.36
 4.3  52.3   60   89.2  2315.5    8    2.97     9.36     9.36    2.97    9.36    9.36
               -125.1 -2315.5    2    2.97     9.36     9.36    2.97    9.36    9.36
 7.5  55.5   60   11.7  2315.5   12    2.97     9.36     9.36    2.97   21.84   17.79
               -502.1 -2970.8    2    2.97     9.36     9.36    2.97   21.84   12.10
14.1  62.1   60    0.0  2315.5    0    2.97     9.36     9.36    2.97   21.84   21.84
              -2227.6 -5269.8    9    2.97     9.36     9.36    9.00   21.84   21.84
16.0  64.0   60    0.0  2315.5    0    2.97     9.36     9.36    2.97   21.84   21.84
              -3485.1 -5269.8    9    2.97     9.36     9.36   14.30   21.84   21.84


                         Span 5: From 64.00  ft  To 75.00  ft

 Loc  AbsLoc  H    Mmax     Mr   Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-
eff
                  Mmin     Mr  Comb  Asb-req  Asb-prv  Asb-eff  Ast-req  Ast-prv  Ast-eff
  ft    ft   in  kips-ft  kips-ft      in^2     in^2     in^2     in^2     in^2     in^2
-------------------------------------------------------------------------------------
-----
 0.0  64.0   60    0.0  2315.5    0    2.97     9.36     9.36    2.97   21.84   21.84
              -4406.4 -5269.8    2    2.97     9.36     9.36   18.30   21.84   21.84
 2.0  66.0   60    0.0  2315.5    0    2.97     9.36     9.36    2.97   21.84   21.84
              -3058.8 -4917.5    2    2.97     9.36     9.36   12.48   21.84   20.33
 3.5  67.5   57    0.0  2189.2    0    2.82     9.36     9.36    2.82   21.84   21.84
              -2059.0 -4035.5    2    2.82     9.36     9.36    8.79   21.84   17.59
 7.9  71.9   48    0.0  1815.1    0    2.38     9.36     9.36    2.38   21.84    9.70
                -14.8 -1835.5    1    2.38     9.36     9.36    2.38   21.84    9.47
11.0  75.0   42    0.0     0.0    0    2.08     9.36*    0.00*   2.08    9.36*   0.00*
                    0.0     0.0    0    2.08     9.36*    0.00*   2.08    9.36*   0.00*


Note:
*  The provided reinforcement is not adequate, either less than required
   or larger than maximum allowed.


SHEAR AND TORSION DESIGN:
=========================
Simplified Method used for design.


                         Span 1: From 0.00  ft  To 12.00  ft

  Loc  AbsLoc Pos   Vu   Comb    Tu   Comb phi*Vc   T-lim   Avs/s 2Ats/s  Av/s    Aprv/s
Alx
   ft     ft       kips       kips-ft       kips   kips-ft  <--------- in^2/ft -------->
in^2
-------------------------------------------------------------------------------------
-----
  0.00   0.00 R     0.0    0     0.0    0   236.7   125.6   0.00   0.00   0.00     2.48
0.00
  3.06   3.06 L     9.7    1     0.0    0   264.9   155.0   0.00   0.00   0.00     2.48
0.00
              R   433.7   10   848.0   10   264.9   155.0   0.87   1.25   2.12     2.48
0.00
  7.50   7.50 L   450.1   10   848.0   10   313.1   200.0   0.60   1.04   1.63     2.48
0.87
```

```
             R   677.4    1   774.5   10   313.1   200.0   1.59   0.95   2.54     2.48**   2.61
  9.00   9.00 L   683.6    1   774.5   10   328.7   215.8   1.47   0.89   2.37     2.48     3.86
             R   683.6    1   774.5   10   328.7   215.8   1.47   0.89   2.37     2.48     3.86
 12.00  12.00 L   696.2    1   774.5   10   325.4   215.8   1.56   0.89   2.45     2.48     9.42


                              Span 2: From 12.00  ft  To 28.00  ft

  Loc  AbsLoc Pos   Vu   Comb    Tu   Comb phi*Vc   T-lim   Avs/s 2Ats/s  Av/s    Aprv/s   Alx
   ft    ft      kips      kips-ft      kips   kips-ft  <--------- in^2/ft -------->   in^2
----------------------------------------------------------------------------------------------
-
  0.00  12.00 R   808.8    3   934.2    3   325.4   215.8   2.03   1.08   3.11     2.48**   3.27
  0.90  12.90 L   805.0    3   934.2    3   325.4   215.8   2.01   1.08   3.09     2.48**   0.19
             R   379.3   10    85.4    4   325.4   215.8   0.68   0.00   0.68     2.48     0.00
  7.50  19.50 L   351.4   10    85.4    4   329.7   215.8   0.68   0.00   0.68     2.48     0.00
             R   226.4   10   135.4    4   329.7   215.8   0.68   0.00   0.68     2.48     0.00
 10.74  22.74 L   212.8   10   135.4    4   335.6   215.8   0.68   0.00   0.68     2.48     0.00
             R   269.2    9   804.4    3   335.6   215.8   0.00   0.93   0.93     2.48     0.00
 16.00  28.00 L   291.4    9   804.4    3   335.8   215.8   0.00   0.93   0.93     2.48     2.82


                              Span 3: From 28.00  ft  To 48.00  ft

  Loc  AbsLoc Pos   Vu   Comb    Tu   Comb phi*Vc   T-lim   Avs/s 2Ats/s  Av/s    Aprv/s   Alx
   ft    ft      kips      kips-ft      kips   kips-ft  <--------- in^2/ft -------->   in^2
----------------------------------------------------------------------------------------------
-
  0.00  28.00 R   576.7   10   791.9    4   335.8   215.8   0.98   0.91   1.89     2.48     6.58
  3.50  31.50 L   561.9   10   791.9    4   337.9   215.8   0.91   0.91   1.82     2.48     0.93
             R   436.9   10   841.9    4   337.9   215.8   0.40   0.97   1.37     2.48     0.00
  4.58  32.58 L   432.4   10   841.9    4   337.9   215.8   0.38   0.97   1.35     2.48     1.83
             R    57.4   10    91.9    4   337.9   215.8   0.00   0.00   0.00     2.48     0.00
 14.42  42.42 L    15.8   10    91.9    4   337.9   215.8   0.00   0.00   0.00     2.48     0.00
             R   379.3    9   801.7    3   337.9   215.8   0.17   0.93   1.09     2.48     2.37
 15.50  43.50 L   383.9    9   801.7    3   337.9   215.8   0.19   0.93   1.11     2.48     0.74
             R   508.9    9   751.7    3   337.9   215.8   0.69   0.87   1.56     2.48     1.70
 20.00  48.00 L   527.9    9   751.7    3   333.8   215.8   0.79   0.87   1.66     2.48     6.20


                              Span 4: From 48.00  ft  To 64.00  ft

  Loc  AbsLoc Pos   Vu   Comb    Tu   Comb phi*Vc   T-lim   Avs/s 2Ats/s  Av/s    Aprv/s   Alx
   ft    ft      kips      kips-ft      kips   kips-ft  <--------- in^2/ft -------->   in^2
----------------------------------------------------------------------------------------------
-
  0.00  48.00 R   372.9    4   848.8    4   333.8   215.8   0.16   0.98   1.14     2.48     4.14
  4.26  52.26 L   355.0    4   848.8    4   337.9   215.8   0.07   0.98   1.05     2.48     0.00
             R   116.4    9   107.8   10   337.9   215.8   0.00   0.00   0.00     2.48     0.00
  7.50  55.50 L   130.0    9   107.8   10   335.2   215.8   0.00   0.00   0.00     2.48     0.00
             R   255.0    9    63.7    8   335.2   215.8   0.68   0.00   0.68     2.48     0.00
 14.10  62.10 L   282.9    9    63.7    8   325.4   215.8   0.68   0.00   0.68     2.48     0.00
             R   705.9    4   889.8    4   325.4   215.8   1.60   1.03   2.62     2.48**   0.00
 16.00  64.00 L   713.9    4   889.8    4   325.4   215.8   1.63   1.03   2.66     2.48**   0.80


                              Span 5: From 64.00  ft  To 75.00  ft

  Loc  AbsLoc Pos   Vu   Comb    Tu   Comb phi*Vc   T-lim   Avs/s 2Ats/s  Av/s    Aprv/s   Alx
```

```
   ft      ft        kips        kips-ft        kips    kips-ft  <--------- in^2/ft -------->
in^2
------------------------------------------------------------------------------------------
--
  0.00  64.00 R    678.0     2    807.1     2    325.4    215.8    1.48    0.93    2.41      2.48
6.59
  2.00  66.00 L    669.6     2    807.1     2    326.9    215.8    1.43    0.93    2.36      2.48
2.32
               R    669.6     2    807.1     2    326.9    215.8    1.43    0.93    2.36      2.48
2.32
  3.50  67.50 L    663.4     2    807.1     2    311.2    200.0    1.55    0.99    2.53      2.48**
1.16
               R    468.6     2    885.0     2    311.2    200.0    0.69    1.08    1.77      2.48
0.00
  7.94  71.94 L    452.2     2    885.0     2    264.8    155.0    0.97    1.30    2.27      2.48
0.00
             R     9.7    1     0.0    0   264.8   155.0   0.00   0.00   0.00     2.48     0.00
 11.00  75.00 L      0.0     0      0.0     0    236.7    125.6    0.00    0.00    0.00      2.48
0.00


Note:
** Provided stirrup area ( Aprv/s ) is not adequate.

-  Pos is the design position. L suggests the calculation is done at immediate left
   of "Loc" and R suggests at immediate right of it.
-  T-lim is the limiting value of torsion for the concrete section. If actual torsion
   is higher than this value, torsional steel has to be provided.
-  Avs/s is the required area of steel per unit length for shear force.
-  2Ats/s is the required area of steel per unit length for two legs of torsional
   reinforcement.
-  Av/s is the total required area of steel per unit length due to shear plus torsion.
-  Aprvs/s is the total provided area of steel per unit length due to shear (stirrups).
-  Alx is the EFFECTIVE longitudinal steel required in addition to the PROVIDED EFFECTIVE
   flexural steel.

CRACKING/FATIGUE CHECK:
=======================
Cracking check as per AASHTO LRFD 3rd Edition (2004)

                           Span 1: From 0.00   ft  To 12.00   ft

                      <------ Cracking ------>     <------ Fatigue ------>
 Loc    AbsLoc    H      fs-t  ratio fs-t  Comb     fs-t   ratio fs-t  Comb
                         fs-b  ratio fs-b  Comb     fs-b   ratio fs-b  Comb
  ft      ft     in     ksi                          ksi
------------------------------------------------------------------------------
  0.00    0.0   42.0    0.0       0.00      0       0.0       0.00      0
                        0.0       0.00      0       0.0       0.00      0
  3.06    3.1   48.1    0.4       0.01     13       0.0       0.00      0
                        0.0       0.00      0       0.0       0.00      0
  7.50    7.5   57.0   24.1       0.67     17       0.9       0.05     25
                        0.0       0.00      0       0.0       0.00      0
  9.00    9.0   60.0   29.4       0.82     17       1.7       0.10     25
                        0.0       0.00      0       0.0       0.00      0
 12.00   12.0   60.0   42.2       1.17*    17       3.5       0.30     25
                        0.0       0.00      0       0.0       0.00      0


                           Span 2: From 12.00   ft  To 28.00   ft
```

```
                      <------ Cracking ------>     <------ Fatigue ------>
 Loc    AbsLoc   H      fs-t  ratio fs-t  Comb     fs-t   ratio fs-t  Comb
                        fs-b  ratio fs-b  Comb     fs-b   ratio fs-b  Comb
  ft      ft     in     ksi                         ksi
------------------------------------------------------------------------------
  0.00   12.0   60.0   33.5       0.93     17       2.5       0.18     25
                        0.0       0.00      0       1.6       0.06     26
  0.90   12.9   60.0   27.7       0.77     17       2.4       0.15     25
                        0.0       0.00      0       1.8       0.07     27
  7.50   19.5   60.0    9.6       0.27     17       1.5       0.07     25
                        0.0       0.00      0       1.5       0.06     27
 10.74   22.7   60.0    3.2       0.09     17       1.1       0.05     25
                        1.1       0.03     15       1.3       0.06     27
 16.00   28.0   60.0   25.7       0.77     15       1.1       0.07     27
                        0.0       0.00      0       0.8       0.03     25


                           Span 3: From 28.00   ft  To 48.00   ft

                      <------ Cracking ------>     <------ Fatigue ------>
 Loc    AbsLoc   H      fs-t  ratio fs-t  Comb     fs-t   ratio fs-t  Comb
                        fs-b  ratio fs-b  Comb     fs-b   ratio fs-b  Comb
  ft      ft     in     ksi                         ksi
------------------------------------------------------------------------------
  0.00   28.0   60.0   31.8       0.95     19       0.6       0.05     27
                        0.0       0.00      0       0.4       0.01     26
  3.50   31.5   60.0    0.0       0.00      0       0.5       0.02     27
                        7.9       0.24     14       0.3       0.02     26
  4.58   32.6   60.0    0.0       0.00      0       0.5       0.02     27
                       16.6       0.50     14       0.3       0.02     26
 14.42   42.4   60.0    0.0       0.00      0       0.5       0.02     28
                       21.7       0.65     17       0.3       0.02     25
 15.50   43.5   60.0    0.0       0.00      0       0.5       0.02     28
                       14.0       0.42     17       0.3       0.02     25
 20.00   48.0   60.0   32.1       0.96     16       0.7       0.05     28
                        0.0       0.00      0       0.3       0.01     25


                           Span 4: From 48.00   ft  To 64.00   ft

                      <------ Cracking ------>     <------ Fatigue ------>
 Loc    AbsLoc   H      fs-t  ratio fs-t  Comb     fs-t   ratio fs-t  Comb
                        fs-b  ratio fs-b  Comb     fs-b   ratio fs-b  Comb
  ft      ft     in     ksi                         ksi
------------------------------------------------------------------------------
  0.00   48.0   60.0   27.9       0.84     20       1.1       0.08     28
                        0.0       0.00      0       0.6       0.02     26
  4.26   52.3   60.0    1.9       0.06     14       1.2       0.05     26
                        1.4       0.04     20       1.1       0.05     28
  7.50   55.5   60.0    7.4       0.21     14       1.4       0.06     26
                        0.0       0.00      0       1.5       0.06     28
 14.10   62.1   60.0   18.8       0.52     14       2.0       0.11     26
                        0.0       0.00      0       2.2       0.08     28
 16.00   64.0   60.0   29.5       0.82     14       2.3       0.15     26
                        0.0       0.00      0       1.5       0.06     25


                           Span 5: From 64.00   ft  To 75.00   ft
```

```
                      <------ Cracking ------>     <------ Fatigue ------>
 Loc    AbsLoc   H      fs-t  ratio fs-t  Comb     fs-t   ratio fs-t  Comb
                        fs-b  ratio fs-b  Comb     fs-b   ratio fs-b  Comb
  ft      ft     in     ksi                         ksi
------------------------------------------------------------------------------
  0.00   64.0   60.0   36.7       1.02*    14       3.1       0.23     26
                        0.0       0.00      0       0.0       0.00      0
  2.00   66.0   60.0   27.3       0.76     14       2.0       0.12     26
                        0.0       0.00      0       0.0       0.00      0
  3.50   67.5   57.0   22.4       0.62     14       1.2       0.07     26
                        0.0       0.00      0       0.0       0.00      0
  7.94   71.9   48.1    0.4       0.01     13       0.0       0.00      0
                        0.0       0.00      0       0.0       0.00      0
 11.00   75.0   42.0    0.0       0.00      0       0.0       0.00      0
                        0.0       0.00      0       0.0       0.00      0


Note:
*  Cracking / fatigue checking failed.





COLUMN DESIGN
=============

COLUMN DESIGN - Column: 1

Column Type: Round    D =  54.00 in

Code: AASHTO LRFD (3rd edition)
Units: US
Design/Analysis Method: No Slenderness Considered.


Design Parameters:
==================
            f'c = 4000.0  psi     fy = 60000.0 psi
            phi flex = 0.90       phi axial = 0.75
            Ec =  3834.3 ksi      Es = 29000 ksi
            Concrete Type :  Normal Weight.

Reinforcement:
==============
Rebar Pattern: Circular
Rebar Orientation: Face Parallel

        Layer    Dir        Size      No. bars    Bar Dist. in
------------------------------------------------------------------------
          1       X         10          19           3.13

          Main bars summary:                  Spiral size: # 4
                            19 # 10 bars

          Total number of bars in the column: 19


Design values used - ( e-min effect included ).
===============================================
```

```
                      (global coordinates)
 Loc      Comb      Fx        Fy        Fz        Mx        My        Mz
  ft             kips      kips      kips    kips-ft   kips-ft   kips-ft
--------------------------------------------------------------------------
  0.00     1      -69.4    1514.6       4.2   -1447.3     -20.2     498.3
 23.75     1       69.4    1453.4      -4.2    1547.5     -20.2    1150.2
  0.00     9      -58.8    1149.4     -10.4   -1705.8     -16.3     525.7
 23.75     9       58.8    1088.2      10.4    1458.3     -16.3     871.6


Column Design
=============

 Loc     Comb    Pu         Mux        Muz       pMn       Incl   pPn/Pu  pMn/Mu
  ft           kips      kips-ft    kips-ft    kips-ft     deg
--------------------------------------------------------------------------------------
  0.00    9   1149.4     1705.8      525.7     3252.7      17.13    1.00   1.82
 23.75    1   1453.4     1547.5     1150.2     3458.7      36.62    1.00   1.79




COLUMN DESIGN
=============

COLUMN DESIGN - Column: 2

Column Type: Round    D =  54.00 in

Code: AASHTO LRFD (3rd edition)
Units: US
Design/Analysis Method: No Slenderness Considered.




Design Parameters:
==================
            f'c = 4000.0  psi     fy = 60000.0 psi
            phi flex = 0.90       phi axial = 0.75
            Ec =  3834.3 ksi      Es = 29000 ksi
            Concrete Type :  Normal Weight.

Reinforcement:
==============
Rebar Pattern: Circular
Rebar Orientation: Face Parallel

        Layer    Dir        Size      No. bars    Bar Dist. in
------------------------------------------------------------------------
          1       X         10          19           3.13

          Main bars summary:                  Spiral size: # 4
                            19 # 10 bars

          Total number of bars in the column: 19


Design values used - ( e-min effect included ).
===============================================
```

```
                      (global coordinates)
 Loc      Comb      Fx       Fy        Fz         Mx        My        Mz
  ft               kips      kips      kips    kips-ft   kips-ft   kips-ft
--------------------------------------------------------------------------
  0.00     9       -0.3     909.0     -14.9   -1722.2      -4.9     168.2
 23.75     9        0.3     847.9      14.9    1369.1      -4.9    -156.9
  0.00    10       45.5     822.2       8.3   -1174.1     -16.1    -469.8
 23.75    10      -45.5     761.1      -8.3    1371.1     -16.1    -611.5


Column Design
=============

 Loc     Comb    Pu         Mux        Muz       pMn       Incl   pPn/Pu  pMn/Mu
  ft            kips      kips-ft    kips-ft    kips-ft     deg
--------------------------------------------------------------------------------------
  0.00    9    909.0     1722.2      168.2     3022.2        5.58    1.00   1.75
 23.75   10    761.1     1371.1      611.5     2886.1      24.04    1.00   1.92




COLUMN DESIGN
=============

COLUMN DESIGN - Column: 3

Column Type: Round    D =  54.00 in

Code: AASHTO LRFD (3rd edition)
Units: US
Design/Analysis Method: No Slenderness Considered.




Design Parameters:
==================
            f'c = 4000.0  psi     fy = 60000.0 psi
            phi flex = 0.90       phi axial = 0.75
            Ec =  3834.3 ksi      Es = 29000 ksi
            Concrete Type :  Normal Weight.

Reinforcement:
==============
Rebar Pattern: Circular
Rebar Orientation: Face Parallel

        Layer    Dir        Size      No. bars    Bar Dist. in
------------------------------------------------------------------------
          1       X         10          19           3.13

          Main bars summary:                  Spiral size: # 4
                            19 # 10 bars

          Total number of bars in the column: 19


Design values used - ( e-min effect included ).
===============================================
```

```
                      (global coordinates)
 Loc      Comb      Fx        Fy        Fz        Mx        My        Mz
  ft               kips      kips      kips    kips-ft   kips-ft   kips-ft
--------------------------------------------------------------------------
  0.00     9      -37.2     865.6     -14.4   -1724.8      10.6     380.5
 23.75     9       37.2     804.5      14.4    1382.7      10.6     502.8


Column Design
=============

 Loc     Comb    Pu         Mux        Muz       pMn       Incl   pPn/Pu  pMn/Mu
  ft            kips      kips-ft    kips-ft    kips-ft     deg
--------------------------------------------------------------------------------------
  0.00    9    865.6     1724.8      380.5     2988.4      12.44    1.00   1.69
 23.75    9    804.5     1382.7      502.8     2920.6      19.98    1.00   1.99




COLUMN DESIGN
=============

COLUMN DESIGN - Column: 4

Column Type: Round    D =  54.00 in

Code: AASHTO LRFD (3rd edition)
Units: US
Design/Analysis Method: No Slenderness Considered.






Design Parameters:
==================
            f'c = 4000.0  psi     fy = 60000.0 psi
            phi flex = 0.90       phi axial = 0.75
            Ec =  3834.3 ksi      Es = 29000 ksi
            Concrete Type :  Normal Weight.

Reinforcement:
==============
Rebar Pattern: Circular
Rebar Orientation: Face Parallel

        Layer    Dir        Size      No. bars    Bar Dist. in
------------------------------------------------------------------------
          1       X         10          19           3.13

          Main bars summary:                  Spiral size: # 4
                            19 # 10 bars

          Total number of bars in the column: 19


Design values used - ( e-min effect included ).
===============================================
```

```
                      (global coordinates)
 Loc      Comb      Fx        Fy        Fz        Mx        My        Mz
  ft               kips      kips      kips    kips-ft   kips-ft   kips-ft
--------------------------------------------------------------------------
  0.00     2       59.3    1412.2       3.9   -1453.1      16.0    -444.7
 23.75     2      -59.3    1351.1      -3.9    1546.2      16.0    -964.6
  0.00     9       40.3    1375.5      -9.3   -1711.0      19.7    -254.5
 23.75     9      -40.3    1314.4       9.3    1489.9      19.7    -735.8


Column Design
=============

 Loc     Comb    Pu         Mux        Muz       pMn       Incl   pPn/Pu  pMn/Mu
  ft            kips      kips-ft    kips-ft    kips-ft     deg
-------------------------------------------------------------------------------------
-
  0.00    9   1375.5     1711.0      254.5     3407.2       8.46    1.00   1.97
 23.75    2   1351.1     1546.2      964.6     3405.8      31.96    1.00   1.87




COMBINED FOOTING DESIGN

Code: AASHTO LRFD (3rd Edition)
Units: US

Geometry:
=========
   Name  : Spread
   Shape : Rectangular, Type : Spread

   Bf(X) = 36.00 ft, Hf(Z) = 20.00 ft, Thickness(Y) = 42.00 in


Footing concentric.
   Columns located on the footing:
   Column No.  1  at x =   0.00 ft,  Round  D =  54.00 in
   Column No.  2  at x =  16.00 ft,  Round  D =  54.00 in

   Ag = 720.00 ft^2, Ix = 24000.00 ft^4, Iz = 77760.01 ft^4

   Surcharge =   0.20 ksf


Design Parameters:
==================     f'c =  4000.00  psi      fy = 60000.00  psi
                       phi flex = 0.90          phi shear = 0.90
                       Ec =    3834.3  ksi      Es =   29000.0 ksi
                       Crack control factor z = 170.00 kips/in
                       Concrete Type :  Normal Weight.



Max Soil Pressures, Service:
============================

Corner     X      Z   |------------------ Column Loads ------------------| Soil press.
           ft      ft   col#  comb   Ovs      P, kips   Mxx, kft   Mzz, kft      ksf
```

```
-----------------------------------------------------------------------------------------
   1     26.00  -10.00   1    15    1.000   -1062.31    1237.96    -293.89      1.56
                         2    15    1.000    -716.94    1254.54     112.34
                         1    17    1.000   -1174.30    1084.81    -288.53      1.41
                         2    17    1.000    -658.60    1112.56     266.65
   2    -10.00  -10.00   1    17    1.000   -1174.30    1084.81    -288.53      3.31
                         2    17    1.000    -658.60    1112.56     266.65
                         1    14    1.000   -1016.03    1203.31    -318.91      2.63
                         2    14    1.000    -698.98    1231.81      98.66
   3    -10.00   10.00   1    17    1.000   -1174.30    1084.81    -288.53      5.14
                         2    17    1.000    -658.60    1112.56     266.65
                         1    14    1.000   -1016.03    1203.31    -318.91      4.66
                         2    14    1.000    -698.98    1231.81      98.66
   4     26.00   10.00   1    15    1.000   -1062.31    1237.96    -293.89      3.64
                         2    15    1.000    -716.94    1254.54     112.34
                         1    17    1.000   -1174.30    1084.81    -288.53      3.24
                         2    17    1.000    -658.60    1112.56     266.65


Max Soil Pressures, Factored:
=============================

Corner     X       Z   |------------------ Column Loads ------------------| Soil press.
           ft      ft   col#  comb   Ovs      P, kips   Mxx, kft   Mzz, kft      ksf
-----------------------------------------------------------------------------------------
   1     26.00  -10.00   1     9     ---    -1149.37    1705.85    -525.71      2.05
                         2     9     ---     -909.02    1722.20     -74.59
                         1     5     ---    -1097.84    1043.05    -350.91      1.47
                         2     5     ---     -585.66    1064.98     199.72
   2    -10.00  -10.00   1    10     ---    -1471.95    1128.40    -275.45      4.40
                         2    10     ---     -822.22    1174.14     469.80
                         1    11     ---     -783.22    1309.05    -402.56      1.94
                         2    11     ---     -665.81    1316.71    -125.23
   3    -10.00   10.00   1     1     ---    -1463.98    1439.85    -474.07      6.45
                         2     1     ---     -828.87    1470.47     250.36
                         1    11     ---     -783.22    1309.05    -402.56      4.13
                         2    11     ---     -665.81    1316.71    -125.23
   4     26.00   10.00   1     9     ---    -1149.37    1705.85    -525.71      4.91
                         2     9     ---     -909.02    1722.20     -74.59
                         1    12     ---    -1105.80     731.60    -152.30      2.85
                         2    12     ---     -579.01     768.66     419.16


Note:
   Only max. positive pressure is considered for design.


Max. Soil Pressure Used in Design:  (without selfweight and surcharge)
==================================

 Factored soil pressure  =   5.52 ksf

 Service soil pressure   =   4.41 ksf

 Fatigue soil pressure   =   0.23 ksf
```

```
Reinforcement Schedule:
=======================:

 Dir  Quantity  Size  Bar dist.  As total    From     To      Hook
                         in        in^2       ft      ft
-----------------------------------------------------------------
  X      39     #  8    3.50        30.81    0.00   36.00     Both
  Z      80     #  8    4.50        63.20    ----    ----     None



Flexure:
========

X direction

  Loc        Mmax    Comb   Asb_req   Asb_prv   Asb_eff   Ast_req   Ast_prv   Ast_eff
             Mmin    Comb   Asb_req   Asb_prv   Asb_eff   Ast_req   Ast_prv   Ast_eff
   ft         kft             in^2      in^2      in^2      in^2      in^2      in^2
-------------------------------------------------------------------------------------
-10.00         0.0     1      7.56     30.81 *    0.00 *    7.56      0.00 *    0.00
               0.0     1      7.56     30.81 *    0.00 *    7.56      0.00 *    0.00
 -9.11        43.7     1      7.56     30.81     17.31      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     17.40      7.56      0.00 *    0.00
 -8.22       174.8     1      7.56     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 -7.33       393.2     1      7.56     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 -6.44       699.1     1      7.56     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 -5.11      1321.4     1     10.24     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 -4.66      1572.9     1     12.22     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 -3.77      2140.9     1     16.70     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 -2.88      2796.3     1     19.69     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 -1.99      3539.1     1     20.84     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
  0.00         0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
  1.99      3979.4     1     23.50     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
  3.50      3108.1     1     19.69     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
  5.11      2449.6     1     19.16     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
  6.50      2112.2     1     16.47     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
  8.00      1987.7     1     15.49     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
  9.50      2112.2     1     16.47     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 10.89      2449.6     1     19.16     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 12.50      3108.1     1     19.69     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 14.01      3979.4     1     23.50     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
```

```
 16.00         0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 17.99      3539.1     1     20.84     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 18.88      2796.3     1     19.69     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 19.77      2140.9     1     16.70     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 20.66      1572.9     1     12.22     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 21.11      1321.4     1     10.24     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 22.44       699.1     1      7.56     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 23.33       393.2     1      7.56     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 24.22       174.8     1      7.56     30.81     30.81      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     30.81      7.56      0.00 *    0.00
 25.11        43.7     1      7.56     30.81     17.31      7.56      0.00 *    0.00
               0.0     1      7.56     30.81     17.40      7.56      0.00 *    0.00
 26.00         0.0     1      7.56     30.81 *    0.00 *    7.56      0.00 *    0.00
               0.0     1      7.56     30.81 *    0.00 *    7.56      0.00 *    0.00

Z direction

  Loc        Mmax    Comb   Asb_req   Asb_prv   Asb_eff   Ast_req   Ast_prv   Ast_eff
   ft         kft             in^2      in^2      in^2      in^2      in^2      in^2
-------------------------------------------------------------------------------------
 -1.99      6370.3     1     38.56     63.20     63.20     13.61      0.00 *    0.00
  1.99      6370.3     1     38.56     63.20     63.20     13.61      0.00 *    0.00


Note:
*  The provided reinforcement is not adequate, either less than required
   or larger than maximum allowed.



Cracking/Fatigue
================
Cracking check as per AASHTO LRFD 3rd Edition (2004)


X direction
             <----------- Cracking ----------->    <----------- Fatigue ----------->
 Loc         Mmax    Comb     fs-t   ratio fs-t    Mmax    Comb     fs-t   ratio fs-t
 Loc         Mmin    Comb     fs-b   ratio fs-b    Mmin    Comb     fs-b   ratio fs-b
  ft          kft              ksi                  kft              ksi
------------------------------------------------------------------------------------
-10.00         0.0     0        0.00    0.00         0.00    0        0.00     0.00
               0.0     0        0.00    0.00         0.00    0        0.00     0.00
 -9.11         0.0     0        0.00    0.00         0.00    0        0.00     0.00
              34.9    17        0.66    0.02         1.79   25        0.03     0.00
 -8.22         0.0     0        0.00    0.00         0.00    0        0.00     0.00
             139.6    17        1.51    0.04         7.16   25        0.08     0.00
 -7.33         0.0     0        0.00    0.00         0.00    0        0.00     0.00
             314.2    17        3.41    0.09        16.11   25        0.17     0.01
 -6.44         0.0     0        0.00    0.00         0.00    0        0.00     0.00
             558.5    17        6.06    0.17        28.64   25        0.31     0.01
 -5.11         0.0     0        0.00    0.00         0.00    0        0.00     0.00
```

```
            1055.7    17      11.45     0.32       54.14    25       0.59     0.03
 -4.66         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1256.6    17      13.62     0.38       64.45    25       0.70     0.04
 -3.77         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1710.4    17      18.54     0.52       87.72    25       0.95     0.05
 -2.88         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            2234.0    17      24.22     0.67      114.57    25       1.24     0.08
 -1.99         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            2827.4    17      30.65     0.85      145.00    25       1.57     0.11
  0.00         0.0     0       0.00     0.00        0.00     0       0.00     0.00
               0.0     0       0.00     0.00        0.00     0       0.00     0.00
  1.99         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            3179.2    17      34.47     0.96      163.05    25       1.77     0.14
  3.50         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            2483.1    17      26.92     0.75      127.34    25       1.38     0.09
  5.11         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1957.0    17      21.22     0.59      100.36    25       1.09     0.06
  6.50         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1687.5    17      18.30     0.51       86.54    25       0.94     0.05
  8.00         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1588.0    17      17.22     0.48       81.44    25       0.88     0.05
  9.50         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1687.5    17      18.30     0.51       86.54    25       0.94     0.05
 10.89         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1957.0    17      21.22     0.59      100.36    25       1.09     0.06
 12.50         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            2483.1    17      26.92     0.75      127.34    25       1.38     0.09
 14.01         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            3179.2    17      34.47     0.96      163.05    25       1.77     0.14
 16.00         0.0     0       0.00     0.00        0.00     0       0.00     0.00
               0.0     0       0.00     0.00        0.00     0       0.00     0.00
 17.99         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            2827.4    17      30.65     0.85      145.00    25       1.57     0.11
 18.88         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            2234.0    17      24.22     0.67      114.57    25       1.24     0.08
 19.77         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1710.4    17      18.54     0.52       87.72    25       0.95     0.05
 20.66         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1256.6    17      13.62     0.38       64.45    25       0.70     0.04
 21.11         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1055.7    17      11.45     0.32       54.14    25       0.59     0.03
 22.44         0.0     0       0.00     0.00        0.00     0       0.00     0.00
             558.5    17       6.06     0.17       28.64    25       0.31     0.01
 23.33         0.0     0       0.00     0.00        0.00     0       0.00     0.00
             314.2    17       3.41     0.09       16.11    25       0.17     0.01
 24.22         0.0     0       0.00     0.00        0.00     0       0.00     0.00
             139.6    17       1.51     0.04        7.16    25       0.08     0.00
 25.11         0.0     0       0.00     0.00        0.00     0       0.00     0.00
              34.9    17       0.66     0.02        1.79    25       0.03     0.00
 26.00         0.0     0       0.00     0.00        0.00     0       0.00     0.00
               0.0     0       0.00     0.00        0.00     0       0.00     0.00

Z direction

 -1.99      5089.3    17      27.76     0.77      261.00    25       1.42     0.06
  1.99      5089.3    17      27.76     0.77      261.00    25       1.42     0.06
```

```
One Way Shear:
==============
(General Method)

 Col      Dist      Comb      dv        Vu          Mu     theta      beta       phi*Vc
           ft                 in       kips        kft      deg                   kips
----------------------------------------------------------------------------------------

X direction

   1     -5.11        1     37.37     540.2      1321.4    52.18      1.76       899.8
          5.11        1     37.37     319.4      2449.6    55.36      1.47       751.7
   2     10.89        1     37.37     319.4      2449.6    55.36      1.47       751.7
         21.11        1     37.37     540.2      1321.4    52.18      1.76       899.8

Z direction

         -5.01        1     36.21     991.6      2473.3    51.14      1.84      1642.7
          5.01        1     36.21     991.6      2473.3    51.14      1.84      1642.7




Two Way Shear:
==============

  #       Bo        Ao     Comb    Avg. dv      Vu      phi*Vc
           ft      ft^2              in        kips       kips
----------------------------------------------------------------
Columns:
  1      23.77      44.96    1     36.79     1739.5     2379.7
  2      23.77      44.96    1     36.79     1739.5     2379.7


Note:
TWO WAY SHEAR IN FOOTING IS NOT DESIGNED AND STIRRUPS ARE NOT CONSIDERED.


COMBINED FOOTING DESIGN

Code: AASHTO LRFD (3rd Edition)
Units: US

Geometry:
=========
   Name  : Pile Cap
   Shape : Rectangular, Type : Pile/Shaft Cap

   Bf(X) = 36.00 ft, Hf(Z) = 20.00 ft, Thickness(Y) = 42.00 in

   Footing concentric.
   Columns located on the footing:
   Column No.  3  at x =   0.00 ft,  Round  D =  54.00 in
   Column No.  4  at x =  16.00 ft,  Round  D =  54.00 in

   Ag = 720.00 ft^2, Ix = 1216.00 ft^2, Iz = 2944.00 ft^2

   Surcharge =   0.20 ksf

   Piles:   Circular      Size: 12.00 in   Capacity: 200.00 kips
```

```
Design Parameters:
==================     f'c =  4000.00  psi      fy = 60000.00  psi
                       phi flex = 0.90          phi shear = 0.90
                       Ec =    3834.3  ksi      Es =   29000.0 ksi
                       Crack control factor z = 170.00 kips/in
                       Concrete Type :  Normal Weight.



Pile Reactions, Service:
========================

Pile Loc(X)   X       Z   |------------------ Column Loads ------------------| Pile Reac.
       ft     in      in   col# comb   Ovs      P, kips    Mxx, kft   Mzz, kft    kips
----------------------------------------------------------------------------------------
  1  -8.00   24.0   -96.0   1   20    1.000    -749.40    1121.76     -42.19       78.93
                            2   20    1.000    -987.39    1095.78     262.81
                            1   14    1.000    -694.13    1232.46    -192.37       71.86
                            2   14    1.000   -1095.50    1209.42     247.80
  2  -8.00   24.0    96.0   1   20    1.000    -749.40    1121.76     -42.19      108.11
                            2   20    1.000    -987.39    1095.78     262.81
                            1   14    1.000    -694.13    1232.46    -192.37      103.99
                            2   14    1.000   -1095.50    1209.42     247.80
  3  -4.00   72.0   -96.0   1   20    1.000    -749.40    1121.76     -42.19       81.22
                            2   20    1.000    -987.39    1095.78     262.81
                            1   14    1.000    -694.13    1232.46    -192.37       76.15
                            2   14    1.000   -1095.50    1209.42     247.80
  4  -4.00   72.0    96.0   1   16    1.000    -733.27    1246.04    -157.42      110.98
                            2   16    1.000   -1054.05    1229.10     202.87
                            1   18    1.000    -710.26    1108.17     -77.14      107.70
                            2   18    1.000   -1028.84    1076.10     307.74
  5   0.00  120.0   -96.0   1   20    1.000    -749.40    1121.76     -42.19       83.51
                            2   20    1.000    -987.39    1095.78     262.81
                            1   15    1.000    -709.55    1238.04    -165.29       79.50
                            2   15    1.000   -1012.83    1202.85     213.39
  6   0.00  120.0    96.0   1   16    1.000    -733.27    1246.04    -157.42      114.40
                            2   16    1.000   -1054.05    1229.10     202.87
                            1   19    1.000    -725.69    1113.75     -50.06      109.90
                            2   19    1.000    -946.17    1069.54     273.33
  7   4.00  168.0   -96.0   1   20    1.000    -749.40    1121.76     -42.19       85.80
                            2   20    1.000    -987.39    1095.78     262.81
                            1   15    1.000    -709.55    1238.04    -165.29       82.73
                            2   15    1.000   -1012.83    1202.85     213.39
  8   4.00  168.0    96.0   1   16    1.000    -733.27    1246.04    -157.42      117.83
                            2   16    1.000   -1054.05    1229.10     202.87
                            1   17    1.000    -728.25    1093.98     -27.74      111.93
                            2   17    1.000    -944.49    1057.65     293.34
  9   8.00  216.0   -96.0   1   14    1.000    -694.13    1232.46    -192.37       89.01
                            2   14    1.000   -1095.50    1209.42     247.80
                            1   19    1.000    -725.69    1113.75     -50.06       85.36
                            2   19    1.000    -946.17    1069.54     273.33
 10   8.00  216.0    96.0   1   16    1.000    -733.27    1246.04    -157.42      121.25
                            2   16    1.000   -1054.05    1229.10     202.87
                            1   17    1.000    -728.25    1093.98     -27.74      113.92
                            2   17    1.000    -944.49    1057.65     293.34
 11  12.00  264.0   -96.0   1   14    1.000    -694.13    1232.46    -192.37       93.30
                            2   14    1.000   -1095.50    1209.42     247.80
                            1   19    1.000    -725.69    1113.75     -50.06       87.45
                            2   19    1.000    -946.17    1069.54     273.33
```

```
 12  12.00  264.0    96.0  1   14    1.000    -694.13   1232.46    -192.37      125.43
                            2   14    1.000   -1095.50    1209.42     247.80
                           1   17    1.000    -728.25   1093.98     -27.74      115.91
                            2   17    1.000    -944.49    1057.65     293.34
 13  16.00  312.0   -96.0  1   14    1.000    -694.13   1232.46    -192.37       97.58
                            2   14    1.000   -1095.50    1209.42     247.80
                           1   19    1.000    -725.69   1113.75     -50.06       89.54
                            2   19    1.000    -946.17    1069.54     273.33
 14  16.00  312.0    96.0  1   14    1.000    -694.13   1232.46    -192.37      129.71
                            2   14    1.000   -1095.50    1209.42     247.80
                           1   17    1.000    -728.25   1093.98     -27.74      117.90
                            2   17    1.000    -944.49    1057.65     293.34
 15  20.00  360.0   -96.0  1   14    1.000    -694.13   1232.46    -192.37      101.87
                            2   14    1.000   -1095.50    1209.42     247.80
                           1   17    1.000    -728.25   1093.98     -27.74       91.57
                            2   17    1.000    -944.49    1057.65     293.34
 16  20.00  360.0    96.0  1   14    1.000    -694.13   1232.46    -192.37      134.00
                            2   14    1.000   -1095.50    1209.42     247.80
                           1   17    1.000    -728.25   1093.98     -27.74      119.89
                            2   17    1.000    -944.49    1057.65     293.34
 17  24.00  408.0   -96.0  1   14    1.000    -694.13   1232.46    -192.37      106.16
                            2   14    1.000   -1095.50    1209.42     247.80
                           1   17    1.000    -728.25   1093.98     -27.74       93.56
                            2   17    1.000    -944.49    1057.65     293.34
 18  24.00  408.0    96.0  1   14    1.000    -694.13   1232.46    -192.37      138.29
                            2   14    1.000   -1095.50    1209.42     247.80
                           1   17    1.000    -728.25   1093.98     -27.74      121.87
                            2   17    1.000    -944.49    1057.65     293.34
 19  -8.00   24.0   -48.0  1   20    1.000    -749.40   1121.76     -42.19       86.23
                            2   20    1.000    -987.39    1095.78     262.81
                           1   14    1.000    -694.13   1232.46    -192.37       79.89
                            2   14    1.000   -1095.50    1209.42     247.80
 20  -8.00   24.0    48.0  1   20    1.000    -749.40   1121.76     -42.19      100.82
                            2   20    1.000    -987.39    1095.78     262.81
                           1   14    1.000    -694.13   1232.46    -192.37       95.96
                            2   14    1.000   -1095.50    1209.42     247.80
 21  24.00  408.0   -48.0  1   14    1.000    -694.13   1232.46    -192.37      114.19
                            2   14    1.000   -1095.50    1209.42     247.80
                           1   17    1.000    -728.25   1093.98     -27.74      100.64
                            2   17    1.000    -944.49    1057.65     293.34
 22  24.00  408.0    48.0  1   14    1.000    -694.13   1232.46    -192.37      130.26
                            2   14    1.000   -1095.50    1209.42     247.80
                           1   17    1.000    -728.25   1093.98     -27.74      114.80
                            2   17    1.000    -944.49    1057.65     293.34


Pile Reactions, Factored:
=========================

Pile Loc(X)   X       Z   |------------------ Column Loads ------------------| Pile Reac.
       ft     in      in   col# comb   Ovs      P, kips    Mxx, kft   Mzz, kft    kips
---------------------------------------------------------------------------------------
-
  1  -8.00   24.0   -96.0  1   10     ---     -940.92   1144.82     157.19      104.98
                            2   10     ---    -1064.45    1088.80     501.20
                           1   11     ---     -611.86   1323.08    -353.97       68.06
                            2   11     ---    -1034.78    1318.99     109.33
  2  -8.00   24.0    96.0  1    4     ---     -942.32   1497.89     -94.82      137.59
                            2    4     ---    -1291.96    1474.33     345.38
                           1    6     ---     -620.07   1072.37    -129.39      102.42
```

```
                                2    6     ---    -1023.75    1047.91     311.83
     3  -4.00   72.0   -96.0   1   10     ---     -940.92    1144.82     157.19      105.43
                                2   10     ---    -1064.45    1088.80     501.20
                               1   11     ---     -611.86    1323.08    -353.97       72.99
                                2   11     ---    -1034.78    1318.99     109.33
     4  -4.00   72.0    96.0   1    4     ---     -942.32    1497.89     -94.82      141.05
                                2    4     ---    -1291.96    1474.33     345.38
                               1   12     ---     -687.17     743.07     183.77      105.06
                                2   12     ---     -723.69     696.84     389.03
     5   0.00  120.0   -96.0   1   10     ---     -940.92    1144.82     157.19      105.88
                                2   10     ---    -1064.45    1088.80     501.20
                               1   11     ---     -611.86    1323.08    -353.97       77.92
                                2   11     ---    -1034.78    1318.99     109.33
     6   0.00  120.0    96.0   1    4     ---     -942.32    1497.89     -94.82      144.51
                                2    4     ---    -1291.96    1474.33     345.38
                               1   12     ---     -687.17     743.07     183.77      104.68
                                2   12     ---     -723.69     696.84     389.03
     7   4.00  168.0   -96.0   1    4     ---     -942.32    1497.89     -94.82      108.86
                                2    4     ---    -1291.96    1474.33     345.38
                               1   11     ---     -611.86    1323.08    -353.97       82.85
                                2   11     ---    -1034.78    1318.99     109.33
     8   4.00  168.0    96.0   1    9     ---     -865.62    1724.83    -380.55      149.03
                                2    9     ---    -1375.53    1710.96     221.50
                               1   12     ---     -687.17     743.07     183.77      104.30
                                2   12     ---     -723.69     696.84     389.03
     9   8.00  216.0   -96.0   1    2     ---     -873.83    1474.12    -155.97      112.88
                                2    2     ---    -1364.51    1439.88     424.00
                               1   12     ---     -687.17     743.07     183.77       84.97
                                2   12     ---     -723.69     696.84     389.03
    10   8.00  216.0    96.0   1    9     ---     -865.62    1724.83    -380.55      154.79
                                2    9     ---    -1375.53    1710.96     221.50
                               1   12     ---     -687.17     743.07     183.77      103.92
                                2   12     ---     -723.69     696.84     389.03
    11  12.00  264.0   -96.0   1    2     ---     -873.83    1474.12    -155.97      117.85
                                2    2     ---    -1364.51    1439.88     424.00
                               1   12     ---     -687.17     743.07     183.77       84.59
                                2   12     ---     -723.69     696.84     389.03
    12  12.00  264.0    96.0   1    9     ---     -865.62    1724.83    -380.55      160.55
                                2    9     ---    -1375.53    1710.96     221.50
                               1   12     ---     -687.17     743.07     183.77      103.54
                                2   12     ---     -723.69     696.84     389.03
    13  16.00  312.0   -96.0   1    2     ---     -873.83    1474.12    -155.97      122.82
                                2    2     ---    -1364.51    1439.88     424.00
                               1   12     ---     -687.17     743.07     183.77       84.21
                                2   12     ---     -723.69     696.84     389.03
    14  16.00  312.0    96.0   1    9     ---     -865.62    1724.83    -380.55      166.31
                                2    9     ---    -1375.53    1710.96     221.50
                               1   12     ---     -687.17     743.07     183.77      103.15
                                2   12     ---     -723.69     696.84     389.03
    15  20.00  360.0   -96.0   1    2     ---     -873.83    1474.12    -155.97      127.79
                                2    2     ---    -1364.51    1439.88     424.00
                               1   12     ---     -687.17     743.07     183.77       83.83
                                2   12     ---     -723.69     696.84     389.03
    16  20.00  360.0    96.0   1    9     ---     -865.62    1724.83    -380.55      172.06
                                2    9     ---    -1375.53    1710.96     221.50
                               1   12     ---     -687.17     743.07     183.77      102.77
                                2   12     ---     -723.69     696.84     389.03
    17  24.00  408.0   -96.0   1    2     ---     -873.83    1474.12    -155.97      132.76
                                2    2     ---    -1364.51    1439.88     424.00
                               1   12     ---     -687.17     743.07     183.77       83.45
```

```
                            2   12     ---     -723.69     696.84     389.03
 18  24.00  408.0    96.0  1    9     ---     -865.62   1724.83    -380.55      177.82
                            2    9     ---    -1375.53    1710.96     221.50
                           1   12     ---     -687.17    743.07     183.77      102.39
                            2   12     ---     -723.69     696.84     389.03
 19  -8.00   24.0   -48.0  1    10    ---     -940.92   1144.82     157.19      112.33
                            2   10     ---    -1064.45    1088.80     501.20
                           1   11     ---     -611.86   1323.08    -353.97       76.75
                            2   11     ---    -1034.78    1318.99     109.33
 20  -8.00   24.0    48.0  1    4     ---     -942.32   1497.89     -94.82      127.81
                            2    4     ---    -1291.96    1474.33     345.38
                           1   11     ---     -611.86   1323.08    -353.97       94.13
                            2   11     ---    -1034.78    1318.99     109.33
 21  24.00  408.0   -48.0  1    9     ---     -865.62   1724.83    -380.55      143.92
                            2    9     ---    -1375.53    1710.96     221.50
                           1   12     ---     -687.17    743.07     183.77       88.18
                            2   12     ---     -723.69     696.84     389.03
 22  24.00  408.0    48.0  1    9     ---     -865.62   1724.83    -380.55      166.52
                            2    9     ---    -1375.53    1710.96     221.50
                           1   12     ---     -687.17    743.07     183.77       97.66
                            2   12     ---     -723.69     696.84     389.03


Note:

   Only max. force in piles is considered for design.
   Pile coordinates X and Z are from the most left edge of the footing.


Max. Pile Reaction Used in Design: (without selfweight and surcharge)
==================================

 Factored pile reaction  = 147.51 kips

 Service pile reaction   = 114.56 kips

 Fatigue pile reaction   =   5.66 kips

Reinforcement Schedule:
=======================:

 Dir  Quantity  Size  Bar dist.  As total    From     To     Hook
                         in        in^2       ft      ft
-----------------------------------------------------------------
  X      39     #  8    3.50        30.81    0.00   36.00     Both
  Z      80     #  8    4.50        63.20    ----    ----     None



Flexure:
========

X direction

  Loc        Mmax    Comb   Asb_req   Asb_prv   Asb_eff   Ast_req   Ast_prv   Ast_eff
             Mmin    Comb   Asb_req   Asb_prv   Asb_eff   Ast_req   Ast_prv   Ast_eff
   ft         kft             in^2      in^2      in^2      in^2      in^2      in^2
-------------------------------------------------------------------------------------
-10.00         0.0     9      7.56     30.81 *    0.00 *    7.56      0.00 *    0.00
               0.0     9      7.56     30.81 *    0.00 *    7.56      0.00 *    0.00
```

```
   -9.11         0.0     9      7.56     30.81     17.31      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     17.40      7.56      0.00 *    0.00
   -8.22         0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   -7.33       394.5     9      7.56     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   -6.44       919.4     9      7.56     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   -5.11      1706.4     9     13.27     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   -4.66      1969.1     9     15.34     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   -3.77      2560.9     9     19.69     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   -2.88      3348.3     9     19.70     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   -1.99      4135.6     9     24.44     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
    0.00         0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
    1.99      5018.0     9     29.81     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
    3.50      4353.5     9     25.76     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
    5.11      3966.8     9     23.42     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
    6.50      3761.7     9     22.18     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
    8.00      3540.2     9     20.85     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
    9.50      3761.7     9     22.18     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   10.89      3966.8     9     23.42     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   12.50      4353.5     9     25.76     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   14.01      5018.0     9     29.81     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   16.00         0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   17.99      4135.6     9     24.44     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   18.88      3348.3     9     19.70     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   19.77      2560.9     9     19.69     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   20.66      1969.1     9     15.34     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   21.11      1706.4     9     13.27     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   22.44       919.4     9      7.56     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   23.33       394.5     9      7.56     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   24.22         0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     30.81      7.56      0.00 *    0.00
   25.11         0.0     9      7.56     30.81     17.31      7.56      0.00 *    0.00
                 0.0     9      7.56     30.81     17.40      7.56      0.00 *    0.00
   26.00         0.0     9      7.56     30.81 *    0.00 *    7.56      0.00 *    0.00
                 0.0     9      7.56     30.81 *    0.00 *    7.56      0.00 *    0.00
```

```
Z direction

  Loc        Mmax    Comb   Asb_req   Asb_prv   Asb_eff   Ast_req   Ast_prv   Ast_eff
   ft         kft             in^2      in^2      in^2      in^2      in^2      in^2
-------------------------------------------------------------------------------------
 -1.99      8565.2     9     52.24     63.20     63.20     13.61      0.00 *    0.00
  1.99      8565.2     9     52.24     63.20     63.20     13.61      0.00 *    0.00


Note:
*  The provided reinforcement is not adequate, either less than required
   or larger than maximum allowed.


Cracking/Fatigue
================
Cracking check as per AASHTO LRFD 3rd Edition (2004)


X direction
             <----------- Cracking ----------->    <----------- Fatigue ----------->
 Loc         Mmax    Comb     fs-t   ratio fs-t    Mmax    Comb     fs-t   ratio fs-t
 Loc         Mmin    Comb     fs-b   ratio fs-b    Mmin    Comb     fs-b   ratio fs-b
  ft          kft              ksi                  kft              ksi
------------------------------------------------------------------------------------
-10.00         0.0     0       0.00     0.00        0.00     0       0.00     0.00
               0.0     0       0.00     0.00        0.00     0       0.00     0.00
 -9.11         0.0     0       0.00     0.00        0.00     0       0.00     0.00
               0.0     0       0.00     0.00        0.00     0       0.00     0.00
 -8.22         0.0     0       0.00     0.00        0.00     0       0.00     0.00
               0.0     0       0.00     0.00        0.00     0       0.00     0.00
 -7.33         0.0     0       0.00     0.00        0.00     0       0.00     0.00
             306.4    14       3.32     0.09       15.15    26       0.16     0.01
 -6.44         0.0     0       0.00     0.00        0.00     0       0.00     0.00
             714.0    14       7.74     0.22       35.30    26       0.38     0.02
 -5.11         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1325.3    14      14.37     0.40       65.52    26       0.71     0.04
 -4.66         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1529.3    14      16.58     0.46       75.61    26       0.82     0.04
 -3.77         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            1988.9    14      21.56     0.60       98.33    26       1.07     0.06
 -2.88         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            2600.4    14      28.19     0.78      128.56    26       1.39     0.09
 -1.99         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            3211.8    14      34.82     0.97      158.79    26       1.72     0.13
  0.00         0.0     0       0.00     0.00        0.00     0       0.00     0.00
               0.0     0       0.00     0.00        0.00     0       0.00     0.00
  1.99         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            3897.1    14      42.25     1.17 *    192.67    26       2.09     0.19
  3.50         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            3381.1    14      36.66     1.02 *    167.16    26       1.81     0.14
  5.11         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            3080.8    14      33.40     0.93      152.31    26       1.65     0.12
  6.50         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            2921.5    14      31.67     0.88      144.44    26       1.57     0.11
  8.00         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            2749.5    14      29.81     0.83      135.93    26       1.47     0.10
  9.50         0.0     0       0.00     0.00        0.00     0       0.00     0.00
            2921.5    14      31.67     0.88      144.44    26       1.57     0.11
```

```
 10.89         0.0     0       0.00     0.00       0.00     0       0.00     0.00
            3080.8    14      33.40     0.93     152.31    26       1.65     0.12
 12.50         0.0     0       0.00     0.00       0.00     0       0.00     0.00
            3381.1    14      36.66     1.02 *   167.16    26       1.81     0.14
 14.01         0.0     0       0.00     0.00       0.00     0       0.00     0.00
            3897.1    14      42.25     1.17 *   192.67    26       2.09     0.19
 16.00         0.0     0       0.00     0.00       0.00     0       0.00     0.00
               0.0     0       0.00     0.00       0.00     0       0.00     0.00
 17.99         0.0     0       0.00     0.00       0.00     0       0.00     0.00
            3211.8    14      34.82     0.97     158.79    26       1.72     0.13
 18.88         0.0     0       0.00     0.00       0.00     0       0.00     0.00
            2600.4    14      28.19     0.78     128.56    26       1.39     0.09
 19.77         0.0     0       0.00     0.00       0.00     0       0.00     0.00
            1988.9    14      21.56     0.60      98.33    26       1.07     0.06
 20.66         0.0     0       0.00     0.00       0.00     0       0.00     0.00
            1529.3    14      16.58     0.46      75.61    26       0.82     0.04
 21.11         0.0     0       0.00     0.00       0.00     0       0.00     0.00
            1325.3    14      14.37     0.40      65.52    26       0.71     0.04
 22.44         0.0     0       0.00     0.00       0.00     0       0.00     0.00
             714.0    14       7.74     0.22      35.30    26       0.38     0.02
 23.33         0.0     0       0.00     0.00       0.00     0       0.00     0.00
             306.4    14       3.32     0.09      15.15    26       0.16     0.01
 24.22         0.0     0       0.00     0.00       0.00     0       0.00     0.00
               0.0     0       0.00     0.00       0.00     0       0.00     0.00
 25.11         0.0     0       0.00     0.00       0.00     0       0.00     0.00
               0.0     0       0.00     0.00       0.00     0       0.00     0.00
 26.00         0.0     0       0.00     0.00       0.00     0       0.00     0.00
               0.0     0       0.00     0.00       0.00     0       0.00     0.00

Z direction

 -1.99      6652.1    14      36.28     1.01 *   328.88    26       1.79     0.14
  1.99      6652.1    14      36.28     1.01 *   328.88    26       1.79     0.14


Note:
*  Cracking / fatigue ratio exceeds allowable.



One Way Shear:
==============
(General Method)

 Col      Dist      Comb      dv        Vu          Mu     theta      beta       phi*Vc
           ft                 in       kips        kft      deg                   kips
----------------------------------------------------------------------------------------

X direction

   1     -5.11        9     37.37     590.0      1706.4    53.83      1.61       821.8
          5.11        9     37.37     147.5      3966.8    58.98      1.20       610.8
   2     10.89        9     37.37     147.5      3966.8    58.98      1.20       610.8
         21.11        9     37.37     590.0      1706.4    53.83      1.61       821.8

Z direction

         -5.01        9     36.21    1327.6      3967.6    54.64      1.52      1351.9
          5.01        9     36.21    1327.6      3967.6    54.64      1.52      1351.9
```

```
Two Way Shear:
==============

  #       Bo        Ao     Comb    Avg. dv      Vu      phi*Vc
          ft       ft^2              in        kips       kips
----------------------------------------------------------------
Columns:
  1      23.77      44.96    9     36.79     1622.6     2379.7
  2      23.77      44.96    9     36.79     1622.6     2379.7

Piles - max:
  3       7.19       9.75    9     36.79      147.5      720.2
Piles - min:
  1       5.60      12.94    9     36.79      147.5      560.3


Note:
TWO WAY SHEAR IN FOOTING IS NOT DESIGNED AND STIRRUPS ARE NOT CONSIDERED.
```

Chapter T3

Hammerhead Pier, Strut and Tie Model Design (AASHTO LRFD)

This tutorial will take you through a strut and tie model design, step-by-step, illustrating the design of a hammerhead pier using the strut and tie model for pier cap and pile cap. Tutorial 3 does not cover column design; however, you can refer to the previous tutorials for an example of column design.

The information for Tutorial 3 is contained in an RC-PIER® input file called "Tutor3.rcp." As an alternative to inputting all information manually, as directed by this tutorial, you can load this file and work on only specific sections. To load this file, select the **Open** item from the **File** menu (**File | Open**). Move to the RC-PIER/Examples subdirectory, if it is not already showing and select the file "Tutor3.rcp".

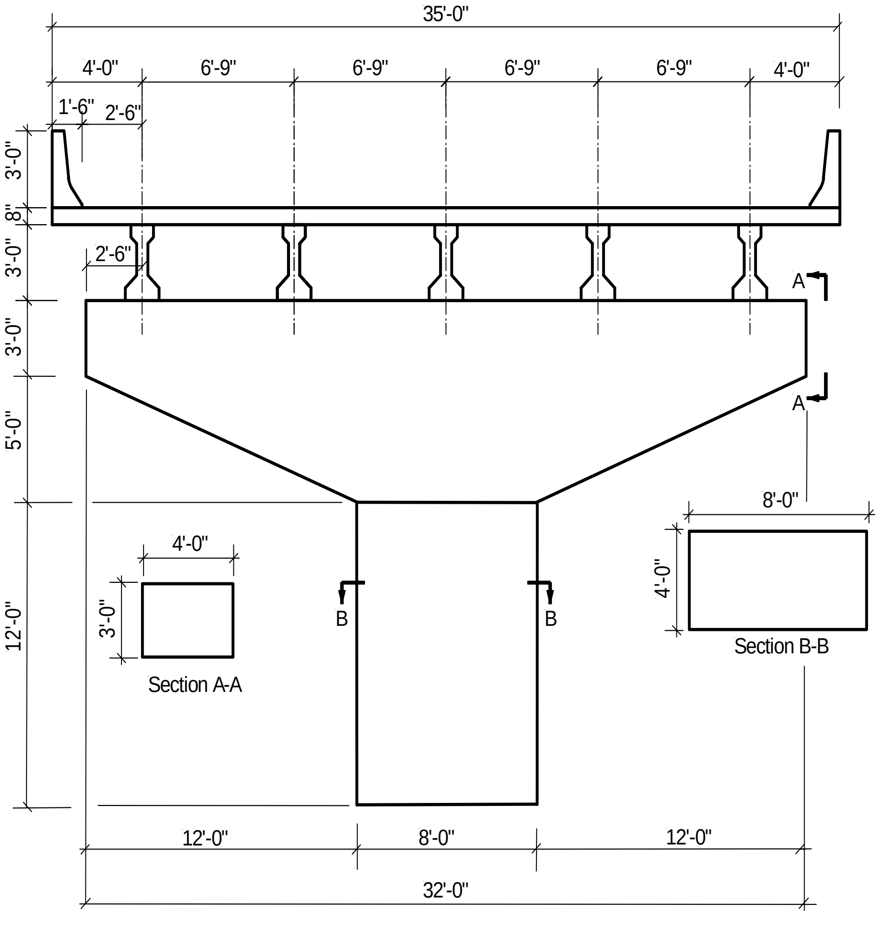


Figure T3-1 Hammerhead Bridge Cross Section at the Pier

Problem Data

Table T3-1

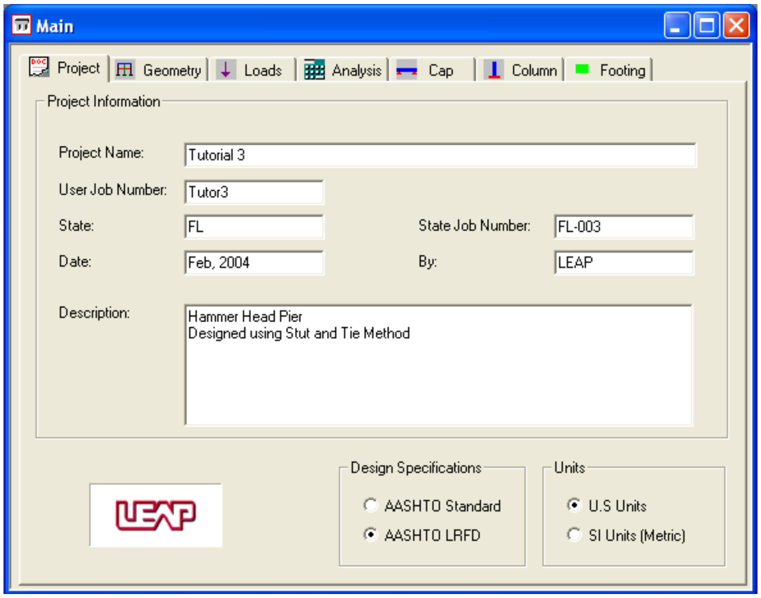
Concrete Strength	
Cap	f’c = 4 ksi
Columns	f’c = 4 ksi
Footings	f’c = 4 ksi
Modulus of Elasticity	E_c = 3834.3 ksi
Concrete Density	
Cap	ρ = 150 pcf
Columns	ρ = 150 pcf
Footings	ρ = 150 pcf
Steel Yield Strength	
Cap	f_y = 60 ksi
Columns	f_y = 60 ksi
Footings	f_y = 60 ksi
Modulus of Elasticity	E_s = 29000 ksi
Other Parameters	
Crack Control Factor	z = 170 kips/in
Superstructure Parameters	
Number of Lanes	= 2
Beam Height	= 36 in
Beam Section Area	= 369 in^2
Beam Inertia Ixx	= 50980 in^4
Beam Inertia Iyy	=5332.5 in^4
Beam Ycg	= 15.83 in
Barrier Height	= 36 in
Slab Depth	= 8.0 in
Total number of spans	= 2
Span Information	
Bridge Overall Width	= 35 ft
Curb to Curb Distance	= 32 ft
Span Length, Span 1	= 60 ft
Span Length, Span 2	= 60 ft
Loads	
Dead Load	
Self Load	= 150 pcf
Wearing Surface Load	= 20 psf × 35 ft = 700 plf
Barrier Load	= 600 plf each side
Live Loads	
Design Truck + Design Lane Load (only one lane loaded)	

Starting the Tutorial

RC-PIER is installed in Program Files\LEAP Software\RC-PIER. Start the program by clicking **Start** > **Programs** > **LEAP Software** > **RC-PIER**.

Step 1

The *Project* screen, shown in Figure T3-2, will open when the program is started. Fill in the general project information in the text boxes. Select the **AASHTO LRFD** option under Design Specifications and **U.S. Units** under Units.



Project Name: Tutorial 3
User Job Number: Tutor3
State: FL
State Job Number: FL-003
Date: Feb, 2004
By: LEAP
Description: Hammer Head Pier
Designed using Stut and Tie Method

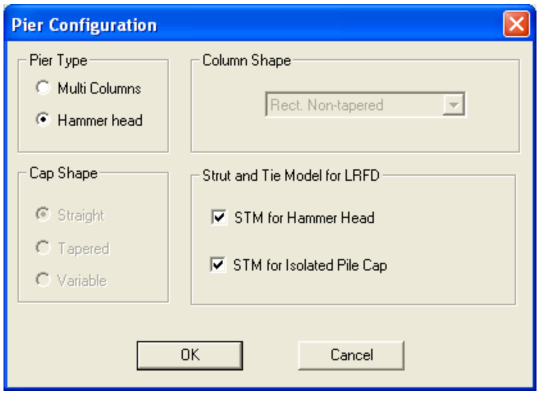
Design Specifications
○ AASHTO Standard
● AASHTO LRFD

Units
● U.S Units
○ SI Units (Metric)

Figure T3-2 Project Tab Screen

Step 2

Click the **Geometry** tab to open the *Geometry* screen. Click **Pier Configuration** to bring up the *Pier Configuration* screen, as shown in Figure T3-3. Under Pier Type, select **Hammerhead**. Notice that when you select Hammerhead, the options under Cap Shape and Column Shape are grayed out; however, both STM options become enabled. Select both options under Strut and Tie Model for LRFD. Click **OK**.



Pier Type
○ Multi Columns
● Hammer head

Column Shape
Rect. Non-tapered

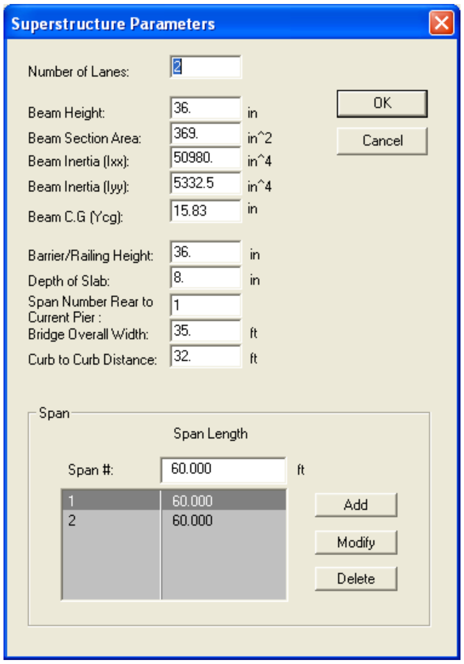
Cap Shape
● Straight
○ Tapered
○ Variable

Strut and Tie Model for LRFD
☑ STM for Hammer Head
☑ STM for Isolated Pile Cap

Figure T3-3 Pier Configuration Screen

Step 4

Click **Superstr**. to activate the *Superstructure Parameters* screen, as shown in Figure T3-4. Input the values for number of lanes, beam height and area, barrier/railing height, slab depth, total number of spans, bridge width and then add all span lengths. Note this screen is optional; however input is required for auto generating loads. Once you entered all the data, click **OK** and return to the *Geometry* screen.



Superstructure Parameters

Number of Lanes: 2

Beam Height: 36. in
Beam Section Area: 369. in^2
Beam Inertia (Ixx): 50980. in^4
Beam Inertia (Iyy): 5332.5 in^4
Beam C.G (Ycg): 15.83 in

Barrier/Railing Height: 36. in
Depth of Slab: 8. in
Span Number Rear to Current Pier : 1
Bridge Overall Width: 35. ft
Curb to Curb Distance: 32. ft

Span

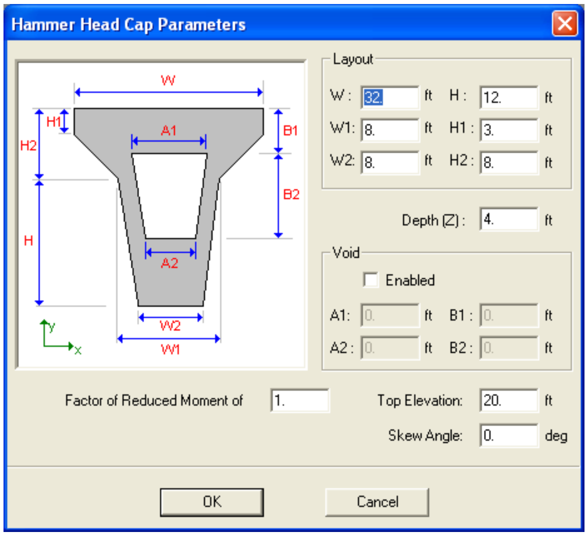
Span #: 60.000 ft

	Span Length
1	60.000
2	60.000

Figure T3-4 Superstructure Parameters Screen

Step 5

Click **Cap** to open the *Hammer Head Cap Parameters* screen, as shown in Figure T3-5. Input **32** in the W field, **8** in the W1 and W2 fields, **12** in the H field, **3** in the H1 field, **8** in the H2 field, **4** in the Depth field, **20** in the Top Elevation field, and **0** in the Skew Angle field. Click **OK** and return to the *Geometry* screen.



Hammer Head Cap Parameters

Layout

W : 32. ft H : 12. ft
W1: 8. ft H1 : 3. ft
W2: 8. ft H2 : 8. ft

Depth (Z) : 4. ft

Void

☐ Enabled

A1: 0. ft B1 : 0. ft
A2 : 0. ft B2 : 0. ft

Factor of Reduced Moment of 1.

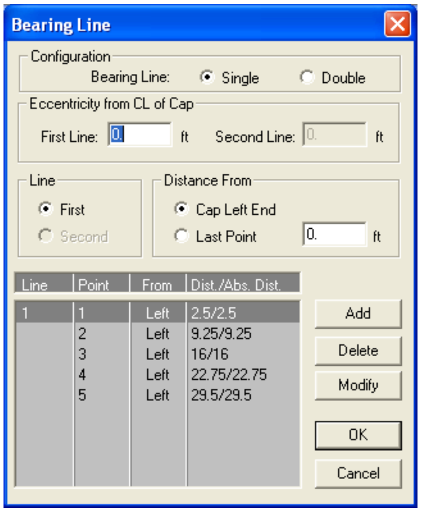
Top Elevation: 20. ft
Skew Angle: 0. deg

Figure T3-5 Hammer Head Cap Parameters Screen

Step 6

Click **Bearing** to bring up the *Bearing Line* screen. For this tutorial, entering one bearing line, as shown in Figure T3-6:

1. Select the **Single** option under Configuration and the **First** option under Line.
2. Select the **Cap Left End** option and input **2.5** in the text box under Distance From.
3. Click **Add**. The first bearing point for the first line will appear in the list at the bottom of the screen.



Configuration

Bearing Line: ● Single ○ Double

Eccentricity from CL of Cap

First Line: 0. ft Second Line: 0. ft

Line

● First

○ Second

Distance From

● Cap Left End

○ Last Point 0. ft

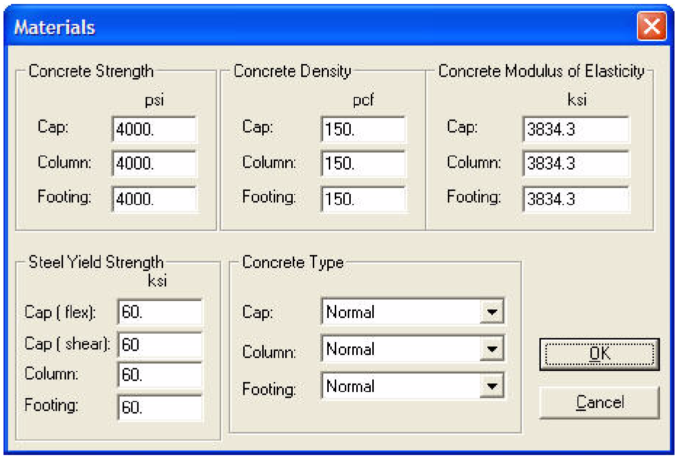
Line	Point	From	Dist./Abs. Dist.
1	1	Left	2.5/2.5
	2	Left	9.25/9.25
	3	Left	16/16
	4	Left	22.75/22.75
	5	Left	29.5/29.5

Figure T3-6 Bearing Line Screen

4. Enter the second point. Select the **Single** option under Configuration, the **First** option under Line, and then select the **Cap Left End** option and input **9.25** in the text box under Distance From.
5. Click **Add** to add it to the list on the screen.
6. Repeat the above steps to enter the remaining points, entering the following values in the text box under Distance From: **16** for the third point, **22.75** for the fourth point, and **29.5** for the fifth point.
7. When completed, there will be a total of 5 bearing points on the first line (Figure T3-6).
8. Click **OK** and return to the *Geometry* screen.

Step 7

Click **Material** to open the *Materials* screen, as shown in Figure T3-7. Use this screen to define the strength, density, type of elasticity of concrete, and reinforced steel of the pier. Notice that the program defaults to certain values. These values can be overwritten by typing over them. Input the values shown in Figure T3-7 and click **OK**.



Materials

	Concrete Strength psi	Concrete Density pcf	Concrete Modulus of Elasticity ksi
Cap:	4000.	150.	3834.3
Column:	4000.	150.	3834.3
Footing:	4000.	150.	3834.3

Steel Yield Strength ksi

Cap (flex): 60.
Cap (shear): 60
Column: 60.
Footing: 60.

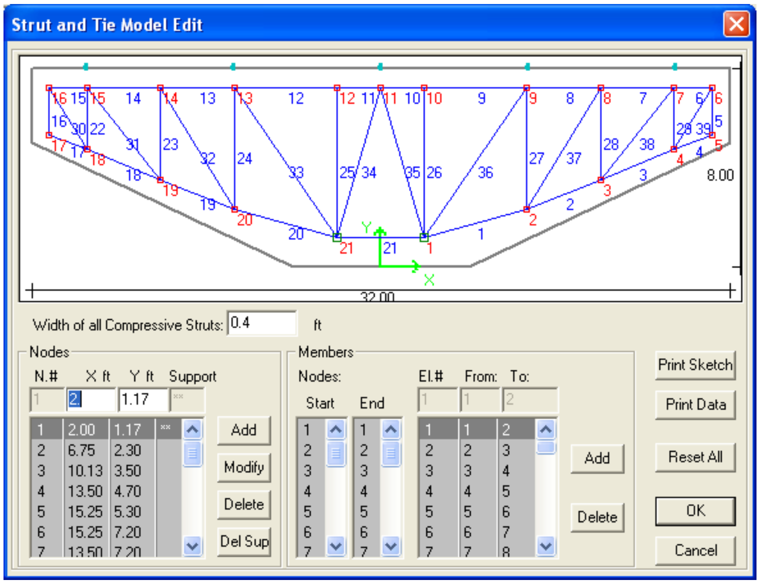
Concrete Type

Cap: Normal
Column: Normal
Footing: Normal

Figure T3-7 Materials Screen

Step 8

Click **Hammer STM** to open the *Strut and Tie Model Edit* screen, as shown in Figure T3-8. The following steps will illustrate how to define a strut and tie model.



Strut and Tie Model Edit

Width of all Compressive Struts: 0.4 ft

Nodes

N.# 1 X ft 2 Y ft 1.17

1	2.00	1.17	**
2	6.75	2.30	
3	10.13	3.50	
4	13.50	4.70	
5	15.25	5.30	
6	15.25	7.20	
7	13.50	7.20	

Members

Nodes: El.# 1 From: 1 To: 2

El.#	From	To
1	1	2
2	2	3
3	3	4
4	4	5
5	5	6
6	6	7
7	7	8

Figure T3-8 Strut and Tie Model Edit Screen

1. Enter **0.4** in the Width of All Compressive Struts field
2. Define the nodes. Input **2** in the X field and **1.17** in the Y field under Nodes.
3. Click **Add**. This is the first node. It will appear in the list on the screen.

4. Highlight node #1 (one you just created) and click **Set Sup**. An asterisk will appear in the list next to the node. This button allows you to specify a node as a support node. At least two nodes must be designated as support nodes for stability of the strut and tie model. For this model, nodes #1 and #21 are support nodes. You can delete a support by clicking **Del Sup**.
5. Repeat the above steps for the remaining node listed in Table T3-2.

Table T3-2

Nodes for Tutorial 3			
N#	**Xft**	**Yft**	**Support**
1	2	1.17	Yes
2	6.75	2.3	No
3	10.13	3.5	No
4	13.5	4.7	No
5	15.25	5.3	No
6	15.25	7.2	No
7	13.5	7.2	No
8	10.13	7.2	No
9	6.75	7.2	No
10	2	7.2	No
11	0	7.2	No
12	-2	7.2	No
13	-6.75	7.2	No
14	-10.12	7.2	No
15	-13.5	7.2	No
16	-15.25	7.2	No
17	-15.25	5.3	No
18	-13.5	4.7	No
19	-10.12	3.5	No
20	-6.75	2.3	No
21	-2	1.17	Yes

6. Define the members that will form the STM. You can add the members in one of two ways: (a) using the lists under Members, or (b) using the mouse to draw the members in the diagram at the top of the screen.
7. For the first member, use the boxes. Select **1** from the Start column and **2** from the End column. Click **Add**. The member will be added to the strut and tie model and will appear in the list under Members and in the diagram in the display area.
8. For the second member, try drawing the member in the diagram. Move the cursor over to Node #2 and left-click. Notice that one end (Node #2) is marked with a square and the other end is free. From Node #2, move the mouse over to Node #3 and left-click. This is the second member and it will appear in the list under Members.
9. Using either of these methods, add the remaining members listed in Table T3-3 on page T3-8. When completed, click **OK** and return to the *Geometry* screen.

Table T3-3

Members for Tutorial 3					
Element#	**From**	**To**	**Element#**	**From**	**To**
1	1	2	21	21	1
2	2	3	22	15	18
3	3	4	23	14	19
4	4	5	24	13	20
5	5	6	25	12	21
6	6	7	26	10	1
7	7	8	27	9	2
8	8	9	28	8	3
9	9	10	29	7	4
10	10	11	30	16	18
11	11	12	31	15	19
12	12	13	32	14	20
13	13	14	33	13	21
14	14	15	34	11	21
15	15	16	35	11	1
16	16	17	36	9	1
17	17	18	37	8	2
18	18	19	38	7	3
19	19	20	39	6	4
20	20	21			

Step 9

Select **Image** from the **Show** menu to open the *Image* screen (or the corresponding icon on the toolbar). A 3-D image of the structure will be displayed on the screen, as shown in Figure T3-9.

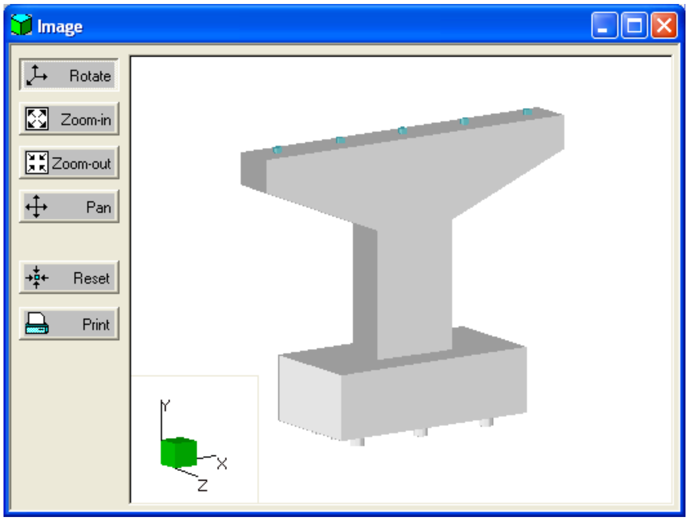
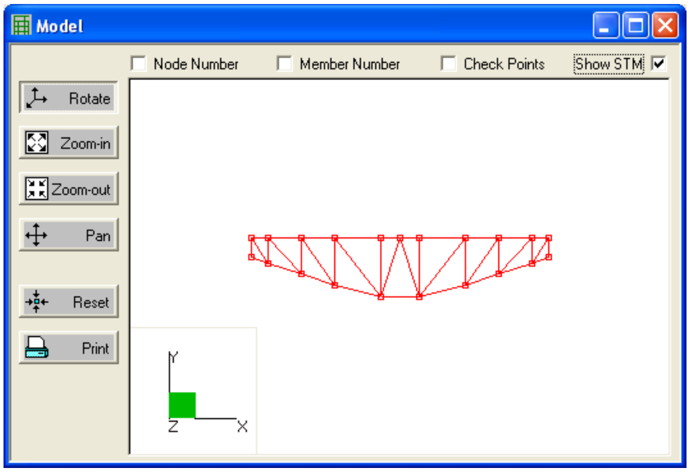


Figure T3-9 Image Screen

Use the buttons on the left of the screen to manipulate your view of the image. When finished, close or minimize the screen and return to the *Geometry* screen.

Step 10

Select **Model** from the **Show** menu to open the *Model* screen, as shown in Figure T3-10. A 3-D model of the nodes, element number, etc. will be displayed on the screen. To see the model shown in the figure, select Show STM.



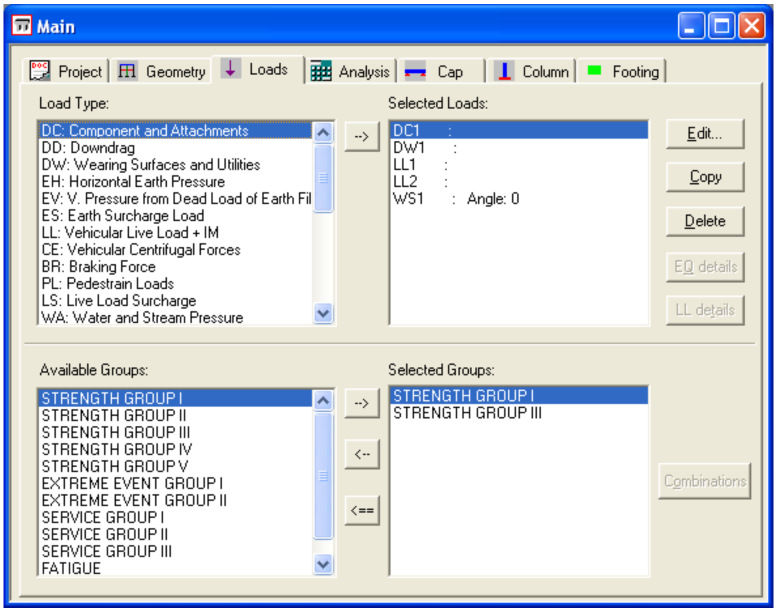
☐ Node Number ☐ Member Number ☐ Check Points Show STM ☑

Figure T3-10 Model Screen

The type of model displayed depends on which characteristics you select from the check boxes at the top of the screen (Node Number, Member Number, Checkpoints, and STM). Use the buttons on the left side of the screen to manipulate your view of the model (e.g., rotate, pan, zoom-in or out).When finished, close or minimize the screen and return to the *Geometry* screen.

Step 11

Click the **Loads** tab to open the *Loads* screen, as shown in Figure T3-11. Add the load types to the Selected Load list by highlighting **DC: Component and Attachments** in the list under Load Type and clicking the -> button. The load DC1 will move to the list under Selected Loads. Repeat the above steps until all the required loads are entered.



Load Type:

DC: Component and Attachments
DD: Downdrag
DW: Wearing Surfaces and Utilities
EH: Horizontal Earth Pressure
EV: V. Pressure from Dead Load of Earth Fil
ES: Earth Surcharge Load
LL: Vehicular Live Load + IM
CE: Vehicular Centrifugal Forces
BR: Braking Force
PL: Pedestrain Loads
LS: Live Load Surcharge
WA: Water and Stream Pressure

Selected Loads:

DC1 :
DW1 :
LL1 :
LL2 :
WS1 : Angle: 0

Available Groups:

STRENGTH GROUP I
STRENGTH GROUP II
STRENGTH GROUP III
STRENGTH GROUP IV
STRENGTH GROUP V
EXTREME EVENT GROUP I
EXTREME EVENT GROUP II
SERVICE GROUP I
SERVICE GROUP II
SERVICE GROUP III
FATIGUE

Selected Groups:

STRENGTH GROUP I
STRENGTH GROUP III

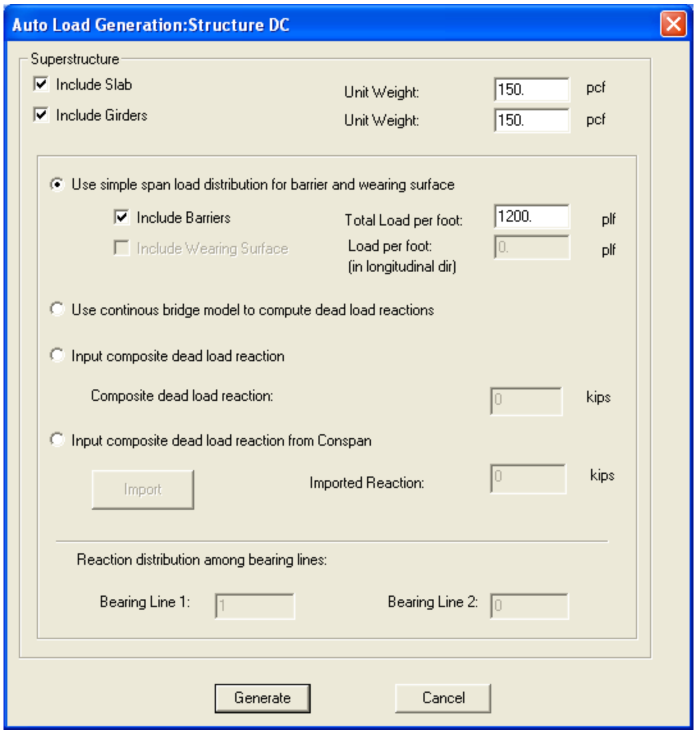
Figure T3-11 Loads Tab Screen

Add the load groups to the Selected Groups list in a similar manner as above. Highlight **Strength Group I** in the list under Available Groups. Click the -> button. It will move to the Selected Groups list. Repeat the above steps to add **Strength Group III** to the Selected Groups list.

Remove a selected group from the Selected Groups list by clicking the <- button. To remove all groups from the Selected Groups list, click the <== button. To define all groups, select the **Load Groups/Limit States** item from the **Libraries** menu.

Step 12

1. Auto generate the loads, beginning with the dead load. Highlight **DC1** in the Selected Loads list and click **Edit**. The *Loads: Load Data* screen will display. Click **Generate** to activate the *Auto Load Generation: Structure DC* screen, as shown in Figure T3-12.



☑ Include Slab — Unit Weight: 150. pcf

☑ Include Girders — Unit Weight: 150. pcf

● Use simple span load distribution for barrier and wearing surface

☑ Include Barriers — Total Load per foot: 1200. plf

☐ Include Wearing Surface — Load per foot: (in longitudinal dir) 0. plf

○ Use continous bridge model to compute dead load reactions

○ Input composite dead load reaction

Composite dead load reaction: 0 kips

○ Input composite dead load reaction from Conspan

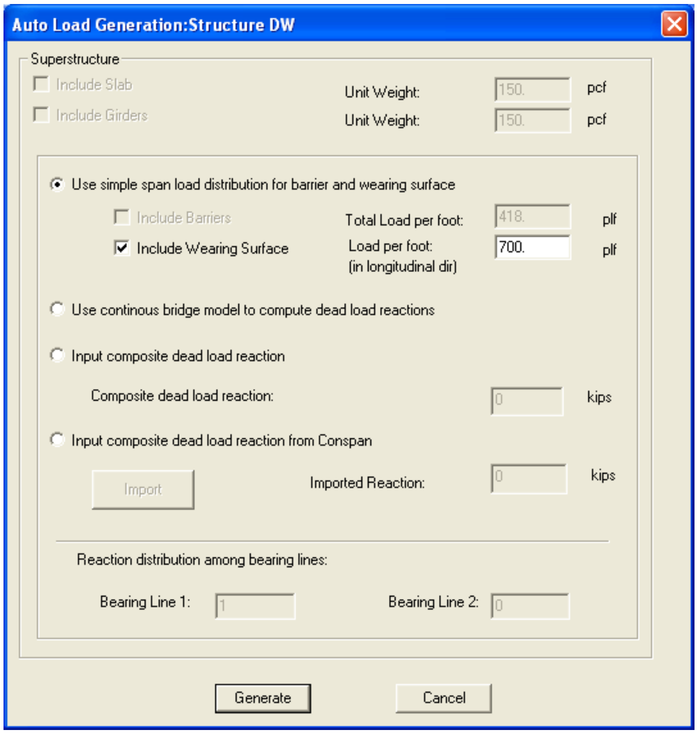
Imported Reaction: 0 kips

Reaction distribution among bearing lines:

Bearing Line 1: 1 — Bearing Line 2: 0

Figure T3-12 Auto Load Generation: Structure DC Screen

2. Select the check boxes for **Include Slab** and **Include Girders**. Input **150** in the Unit Weight field.
3. Select the option “Use simple span load distribution for barrier and wearing surface”.
4. Select Include Barrier and specify 1200 in the total load per foot field.
5. Click **Generate**. The program will automatically generate the loads and return to the *Loads: Load Data* screen. Click **OK** to return to the *Loads* screen.
6. Highlight **DW1** in the Selected Loads list and click **Edit**. The *Loads: Load Data* screen will display. Click **Generate** to activate the *Auto Load Generation: Structure DW* screen, as shown in Figure T3-13.



Auto Load Generation:Structure DW

Superstructure

☐ Include Slab Unit Weight: 150. pcf

☐ Include Girders Unit Weight: 150. pcf

● Use simple span load distribution for barrier and wearing surface

☐ Include Barriers Total Load per foot: 418. plf

☑ Include Wearing Surface Load per foot: (in longitudinal dir) 700. plf

○ Use continous bridge model to compute dead load reactions

○ Input composite dead load reaction

Composite dead load reaction: 0 kips

○ Input composite dead load reaction from Conspan

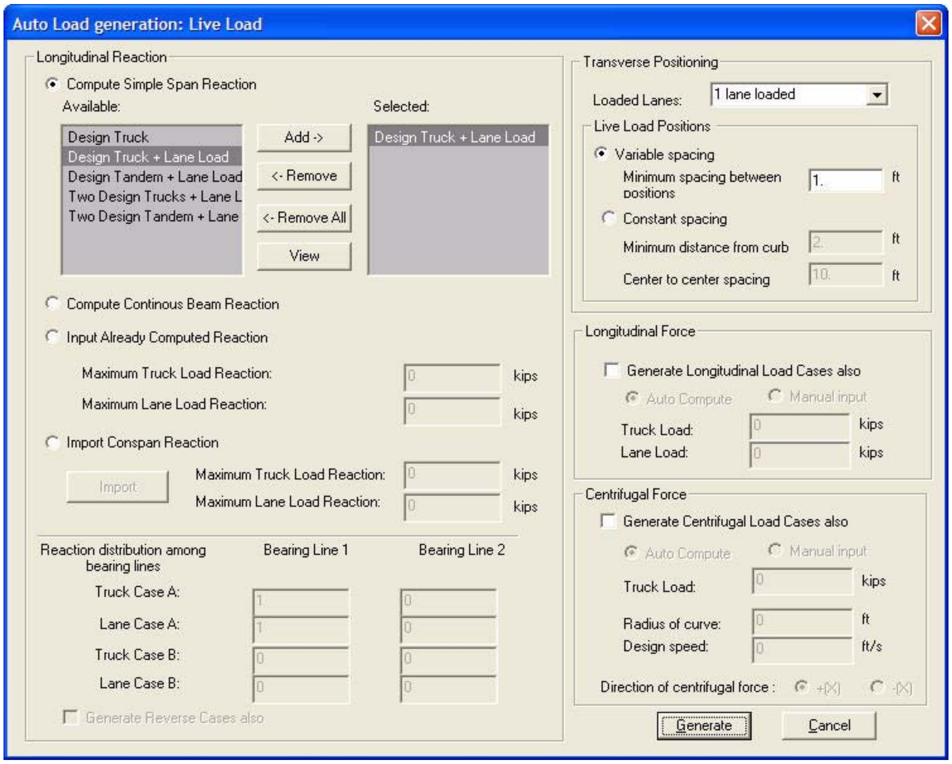
Imported Reaction: 0 kips

Reaction distribution among bearing lines:

Bearing Line 1: 1 Bearing Line 2: 0

Figure T3-13 Auto Load Generation: Structure DW Screen

7. Select the option “Use simple span load distribution for barrier and wearing surface”.
8. Select **Include Wearing Surface** and input **700** in the Load per foot text box. (Note that all other options are grayed out because, in LRFD, dead load due to component and dead load due to wearing surface are separate load cases.)
9. Click **Generate**. The program will automatically generate the loads. Click **OK** to return to the *Loads* screen.
10. Highlight **LL1** on the Selected Loads list and click **Edit**. The *Loads: Load Data* screen will display. Click **Generate** to activate the *Auto Load Generation: Live Load* screen, as shown in Figure T3-14.
11. In the longitudinal reaction area, select “Compute simple span longitudinal reaction using following trucks”.



Longitudinal Reaction

● Compute Simple Span Reaction

Available:
Design Truck
Design Truck + Lane Load
Design Tandem + Lane Load
Two Design Trucks + Lane L
Two Design Tandem + Lane

Selected:
Design Truck + Lane Load

○ Compute Continous Beam Reaction

○ Input Already Computed Reaction
Maximum Truck Load Reaction: 0 kips
Maximum Lane Load Reaction: 0 kips

○ Import Conspan Reaction
Maximum Truck Load Reaction: 0 kips
Maximum Lane Load Reaction: 0 kips

Reaction distribution among bearing lines	Bearing Line 1	Bearing Line 2
Truck Case A:	1	0
Lane Case A:	1	0
Truck Case B:	0	0
Lane Case B:	0	0

☐ Generate Reverse Cases also

Transverse Positioning
Loaded Lanes: 1 lane loaded

Live Load Positions
● Variable spacing
Minimum spacing between positions: 1. ft
○ Constant spacing
Minimum distance from curb: 2. ft
Center to center spacing: 10. ft

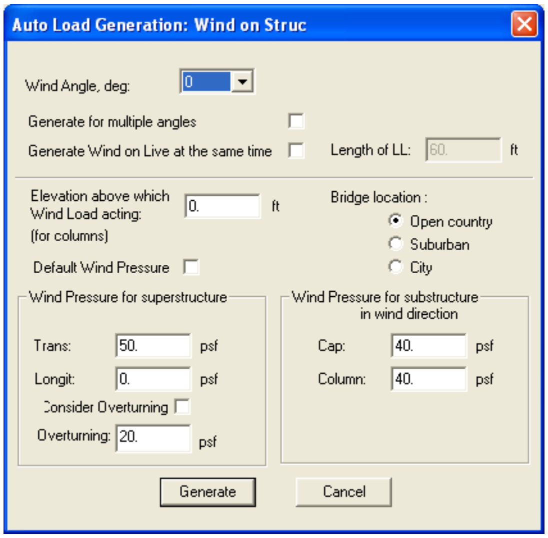
Longitudinal Force
☐ Generate Longitudinal Load Cases also
● Auto Compute ○ Manual input
Truck Load: 0 kips
Lane Load: 0 kips

Centrifugal Force
☐ Generate Centrifugal Load Cases also
● Auto Compute ○ Manual input
Truck Load: 0 kips
Radius of curve: 0 ft
Design speed: 0 ft/s
Direction of centrifugal force : ● +(X) ○ -(X)

Figure T3-14 Auto Load Generation: Live Load Screen

12. Select **Design Truck + Lane Load** under Available and click **Add**. It will appear in the list under Selected.
13. In Transverse Positioning, select **1 lane loaded** from the Loaded Lanes list. Select the **Variable Spacing** option and input **0.5** in the Minimum Spacing Between Positions text box.
14. Click the **Generate**. The program will automatically generate the loads and return to the *Loads: Load Data* screen. Click **OK** and return to the *Loads* screen.

15. Highlight **WS1** on the Selected Loads list and click **Edit**. The *Loads: Load Data* screen displays. Click **Generate** to activate the *Auto Load Generation: Wind on Structure* screen, as shown in Figure T3-15.



Wind Angle, deg: 0

☐ Generate for multiple angles

☐ Generate Wind on Live at the same time

Length of LL: 60. ft

Elevation above which Wind Load acting: (for columns) 0. ft

☐ Default Wind Pressure

Bridge location:
● Open country
○ Suburban
○ City

Wind Pressure for superstructure
Trans: 50. psf
Longit: 0. psf
☐ Consider Overturning
Overturning: 20. psf

Wind Pressure for substructure in wind direction
Cap: 40. psf
Column: 40. psf

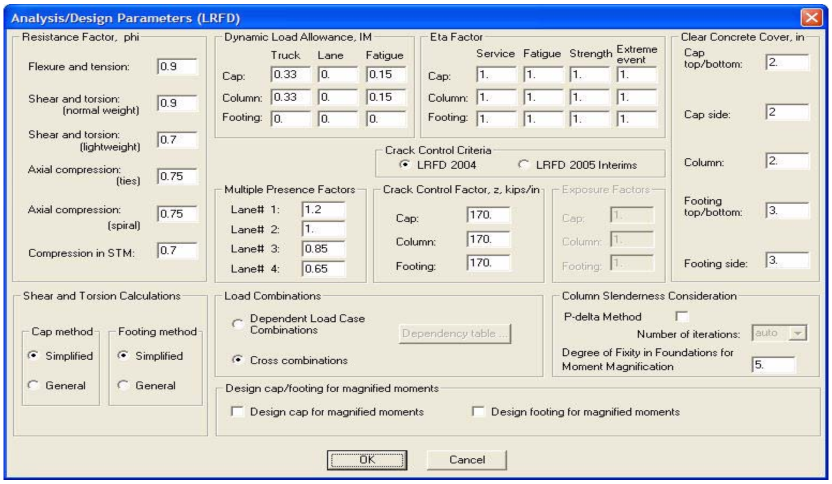
Figure T3-15 Auto Load Generation: Wind on Structure Screen

16. Select **0** from the Wind Angle list, **City** under Bridge Location, and the **Default Wind Pressure** check box is unchecked. Then specify Transverse Longitudinal cap and column pressures as shown in the figure. Make sure the **Generate for Multiple Angles** and **Generate Wind on Live at same time** check boxes are not checked.

17. Click **Generate**. The program will automatically generate the loads and return to the *Loads: Load Data* screen. Click **OK** and return to the *Loads* screen.

Step 13

Click the **Analysis** tab to open the *Analysis* screen. This screen is used to perform an analysis and specify various factors relating to the analysis and design by clicking **A/D Parameters**. This will activate the *Analysis/Design Parameters (LRFD)* screen, as shown in Figure T3-16. Adjust all the values, and click **OK** to return to the *Analysis* screen, or click **Cancel** to close the screen without making any changes.



Resistance Factor, phi
Flexure and tension: 0.9
Shear and torsion: (normal weight) 0.9
Shear and torsion: (lightweight) 0.7
Axial compression: (ties) 0.75
Axial compression: (spiral) 0.75
Compression in STM: 0.7

Dynamic Load Allowance, IM

	Truck	Lane	Fatigue
Cap:	0.33	0.	0.15
Column:	0.33	0.	0.15
Footing:	0.	0.	0.

Eta Factor

	Service	Fatigue	Strength	Extreme event
Cap:	1.	1.	1.	1.
Column:	1.	1.	1.	1.
Footing:	1.	1.	1.	1.

Clear Concrete Cover, in
Cap top/bottom: 2.
Cap side: 2
Column: 2.
Footing top/bottom: 3.
Footing side: 3.

Crack Control Criteria
● LRFD 2004
○ LRFD 2005 Interims

Multiple Presence Factors
Lane# 1: 1.2
Lane# 2: 1.
Lane# 3: 0.85
Lane# 4: 0.65

Crack Control Factor, z, kips/in
Cap: 170.
Column: 170.
Footing: 170.

Exposure Factors
Cap: 1.
Column: 1.
Footing: 1.

Shear and Torsion Calculations
Cap method: ● Simplified ○ General
Footing method: ● Simplified ○ General

Load Combinations
○ Dependent Load Case Combinations
● Cross combinations

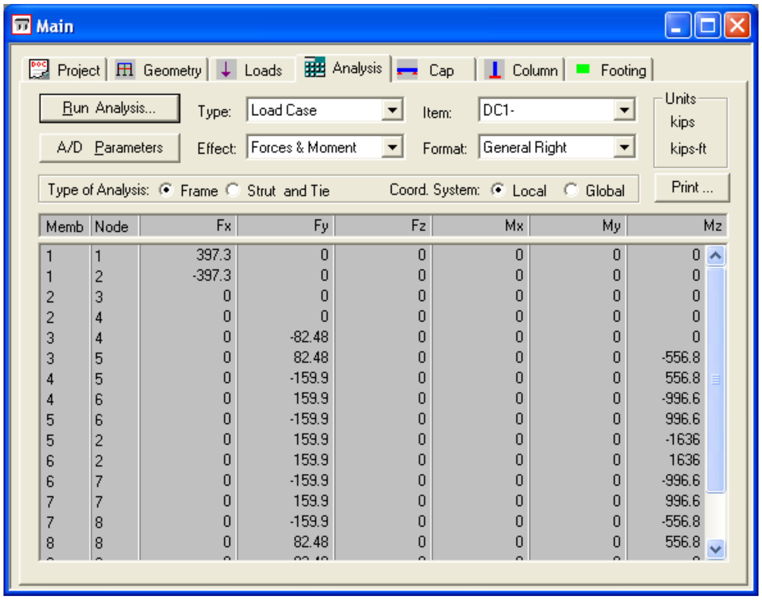
Column Slenderness Consideration
☐ P-delta Method
Number of iterations: auto
Degree of Fixity in Foundations for Moment Magnification: 5.

Design cap/footing for magnified moments
☐ Design cap for magnified moments
☐ Design footing for magnified moments

Figure T3-16 Analysis/Design Parameters (LRFD) Screen

Step 14

Click **Run Analysis** to perform the analysis for the pier structure based on all the data entered up to this point. The results will appear on the screen, as shown in Figure T3-17.



Type: Load Case
Item: DC1-
Effect: Forces & Moment
Format: General Right
Units: kips, kips-ft
Type of Analysis: ● Frame ○ Strut and Tie
Coord. System: ● Local ○ Global

Memb	Node	Fx	Fy	Fz	Mx	My	Mz
1	1	397.3	0	0	0	0	0
1	2	-397.3	0	0	0	0	0
2	3	0	0	0	0	0	0
2	4	0	0	0	0	0	0
3	4	0	-82.48	0	0	0	0
3	5	0	82.48	0	0	0	-556.8
4	5	0	-159.9	0	0	0	556.8
4	6	0	159.9	0	0	0	-996.6
5	6	0	-159.9	0	0	0	996.6
5	2	0	159.9	0	0	0	-1636
6	2	0	159.9	0	0	0	1636
6	7	0	-159.9	0	0	0	-996.6
7	7	0	159.9	0	0	0	996.6
7	8	0	-159.9	0	0	0	-556.8
8	8	0	82.48	0	0	0	556.8

Figure T3-17 Analysis Tab Screen (After Analysis is Performed)

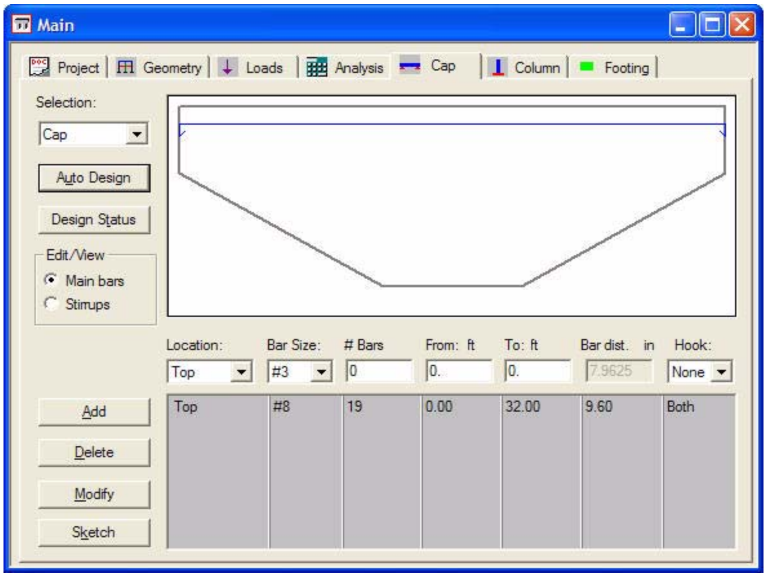
If necessary, use the scroll bar on the right side of the screen to view all the results. To view the results for strut and tie model, select the **Strut and Tie** option under Type of Analysis.

Step 15

At this point, it would be a good idea to save your project. Select the **Save As** item from the **File** menu. This will activate the *Save As* screen. Enter a name for the file in the File Name field (e.g., Mytutor3). The default extension is “*.rcp”. Click **Save**.

Step 16

Click the **Cap** tab to activate the *Cap* screen, as shown in Figure T3-18. Select **Cap** from the Selection list, and then click **Auto Design**. The *Design Cap* screen displays. Select **8** from the Bar Size list and **4** from the Stirrup Size list. Click **OK**. The program will automatically generate the cap design and immediately displays the *Design Status* **-** *Cap* screen (Figure T3-19). Click **Close** to exit this screen and return to the *Cap* screen. A diagram of the cap will appear in the display area at the top of the screen.



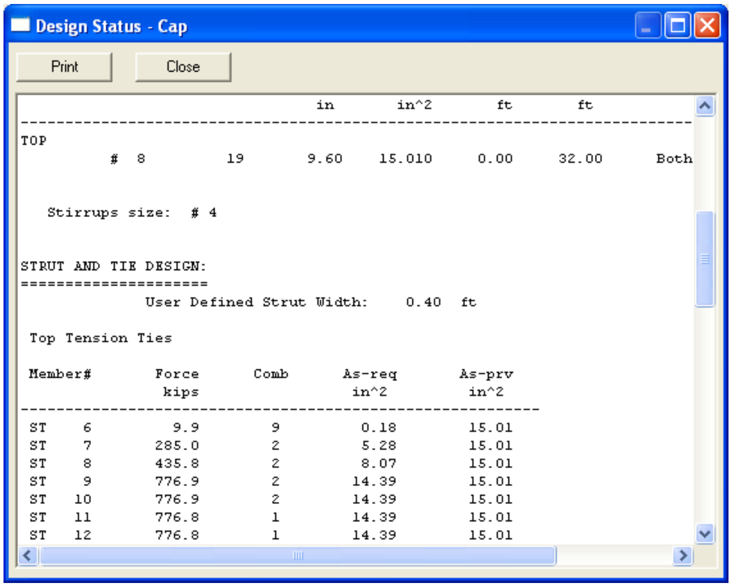
Selection: Cap

Edit/View
● Main bars
○ Stirrups

Location:	Bar Size:	# Bars	From: ft	To: ft	Bar dist. in	Hook:
Top	#3	0	0.	0.	7.9625	None
Top	#8	19	0.00	32.00	9.60	Both

Figure T3-18 Cap Tab Screen

Click **Design Status** on the *Cap* screen to display a design summary of the selected component, as shown in Figure T3-19. To see a graphical representation of the selected structure, click **Sketch** on the *Cap* screen.

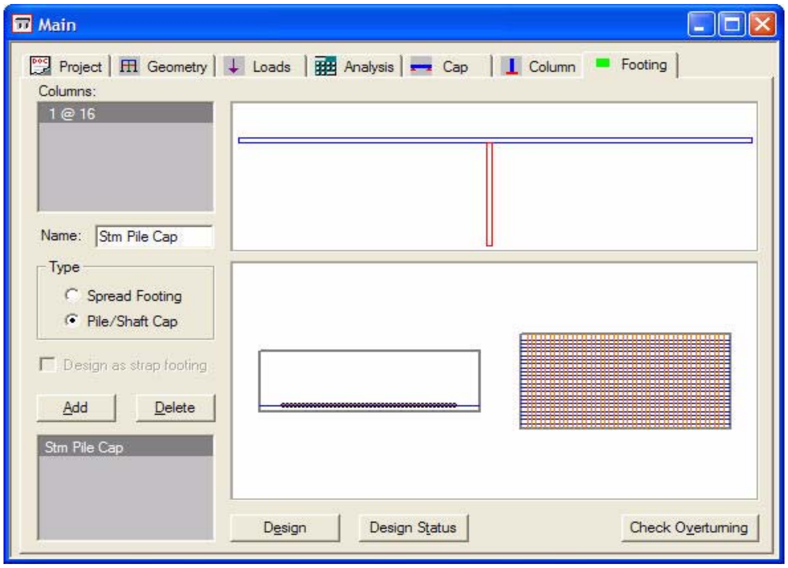


```
                                 in       in^2       ft       ft
---------------------------------------------------------------------------
TOP
          #  8         19       9.60    15.010      0.00     32.00      Both


   Stirrups size:   # 4


STRUT AND TIE DESIGN:
=====================
              User Defined Strut Width:    0.40   ft

 Top Tension Ties

 Member#       Force      Comb      As-req       As-prv
                kips                 in^2         in^2
----------------------------------------------------------
 ST     6        9.9        9         0.18        15.01
 ST     7      285.0        2         5.28        15.01
 ST     8      435.8        2         8.07        15.01
 ST     9      776.9        2        14.39        15.01
 ST    10      776.9        2        14.39        15.01
 ST    11      776.8        1        14.39        15.01
 ST    12      776.8        1        14.39        15.01
```

Figure T3-19 Design Status - Cap Screen

Step 17

Select the **Footing** tab to activate the *Footing* screen, as shown in Figure T3-20.



Columns: 1 @ 16

Name: Stm Pile Cap

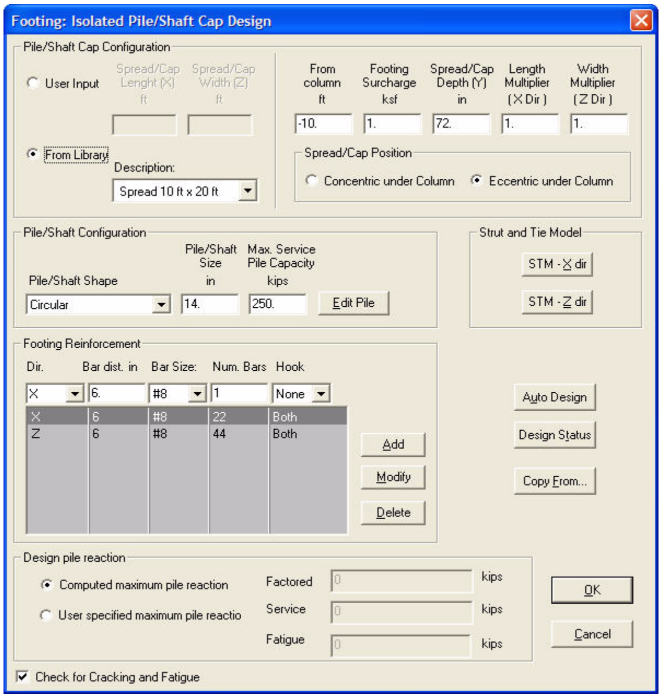
Type
○ Spread Footing
● Pile/Shaft Cap

☐ Design as strap footing

Stm Pile Cap

Figure T3-20 Footing Tab Screen

1. To begin, select **1@16** under Columns, input **STM Pile Cap** in the Name field, and select **Pile/Shaft Cap** under Type. Click **Add**. The name **STM Pile Cap** will appear in the list, as shown in Figure T3-20.
2. Highlight **STM Pile Cap** and click **Design** to activate the *Footing: Isolated Pile/Shaft Cap Design* screen, as shown in Figure T3-21.



Pile/Shaft Cap Configuration

○ User Input — Spread/Cap Lenght (X) ft: ; Spread/Cap Width (Z) ft:

● From Library — Description: Spread 10 ft x 20 ft

From column ft	Footing Surcharge ksf	Spread/Cap Depth (Y) in	Length Multiplier (X Dir)	Width Multiplier (Z Dir)
-10.	1.	72.	1.	1.

Spread/Cap Position: ○ Concentric under Column ● Eccentric under Column

Pile/Shaft Configuration

Pile/Shaft Shape	Pile/Shaft Size in	Max. Service Pile Capacity kips
Circular	14.	250.

Footing Reinforcement

Dir.	Bar dist. in	Bar Size:	Num. Bars	Hook
X	6.	#8	1	None
X	6	#8	22	Both
Z	6	#8	44	Both

Design pile reaction
● Computed maximum pile reaction
○ User specified maximum pile reactio

Factored 0 kips
Service 0 kips
Fatigue 0 kips

☑ Check for Cracking and Fatigue

Figure T3-21 Footing: Isolated Pile/Shaft Cap Design Screen

3. Select **Spread 10 ft** x **20 ft** from the Description list and input **-10** in the From Column field, **1** in the Footing Surcharge field, **72** in the Spread/Cap Depth field, and **1** in the Length and Width Multiplier fields

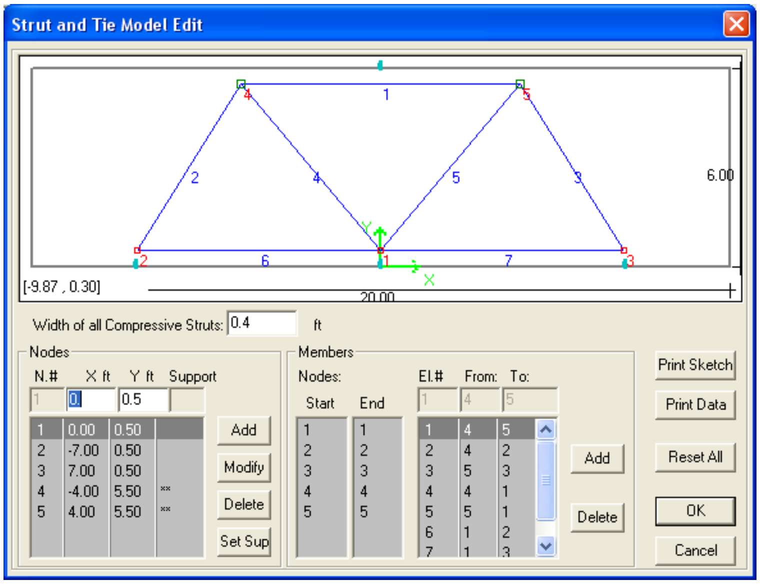
4. Under Pile/Shaft Configuration, select **Circular** from the Pile/Shaft Shape list, input **14** in the Pile/Shaft Size field, **250** in the Max. Pile Capacity field.

5. Then, click **Edit Pile** to activate the *Edit: Pile Locations* screen. For the first pile location, input **36** in the X field and **36** in the Z field. Click **Add**. It will appear in the list on the screen. Repeat this for the remaining pile locations, as listed in the following table.

Pile Locations		
Pile #	**X (in)**	**Z (in)**
1	36	36
2	120	36
3	204	36
4	36	0
5	120	0
6	204	0
7	36	-36
8	120	-36
9	204	-36

6. When completed, there will be nine pile locations. Click **OK** and return to *Footing: Isolated Pile/Shaft Cap Design* screen.

7. Click **STM X-Dir** to activate the *Strut and Tie Model Edit* screen, as shown in Figure T3-22.



Width of all Compressive Struts: 0.4 ft

Nodes

N.# 1 X ft 0. Y ft 0.5 Support

N.#	X ft	Y ft	Support
1	0.00	0.50	
2	-7.00	0.50	
3	7.00	0.50	
4	-4.00	5.50	**
5	4.00	5.50	**

Members

Nodes: Start	End
1	1
2	2
3	3
4	4
5	5

El.# 1 From: 4 To: 5

El.#	From:	To:
1	4	5
2	4	2
3	5	3
4	4	1
5	5	1
6	1	2
7	1	3

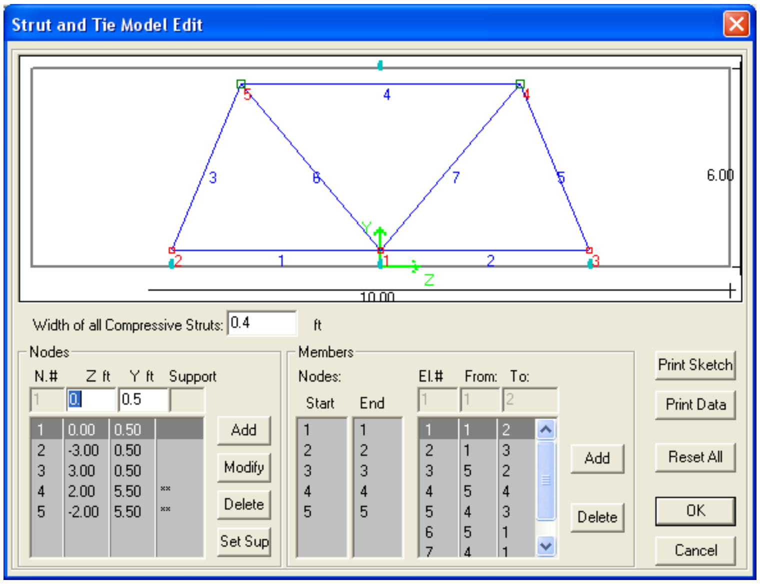
Figure T3-22 Strut and Tie Model Edit Screen in X-Direction

8. Input **0.4** in the Width of all Compressive Struts field. Under Nodes, input **0** in the X field and **0.5** in the Y field. Click **Add** to add the first node to the list under Nodes.

9. Using the above procedure, add the remaining nodes shown in Figure T3-22.

10. Highlight **Node #4** in the list and click **Set Sup**. An asterisk will appear next to the node. Repeat this for Node #5.

11. Add the members to design the strut and ties. Select **4** from the Start column and **5** from the End column. Click **Add**. This is the first member and it will appear in the list under Members.
12. Repeat the above step to add the remaining members as shown in Figure T3-22.
13. When completed, click **OK** and return to the *Footing: Isolated Pile/Shaft Cap Design* screen.
14. Click **STM Z-Dir** to activate the *Strut and Tie Model Edit* screen, as shown in Figure T3-23. Repeat steps 8-13 above to add the nodes and members shown in Figure T3-23.



Width of all Compressive Struts: 0.4 ft

Nodes

N.#	Z ft	Y ft	Support
1	0.	0.5	
1	0.00	0.50	
2	-3.00	0.50	
3	3.00	0.50	
4	2.00	5.50	**
5	-2.00	5.50	**

Members

Nodes: Start	End
1	1
2	2
3	3
4	4
5	5

El.#	From:	To:
1	1	2
1	1	2
2	1	3
3	5	2
4	5	4
5	4	3
6	5	1
7	4	1

Figure T3-23 Strut and Tie Model Edit Screen in Z-direction

15. Click **Auto Design** to activate the *Design Footing* screen, as shown in Figure T3-24. Select **#8** from the Bar Size list. Click **OK**.

Footing Reinforcement

Bar size #8

Figure T3-24 Design Footing Screen

16. The program will design the footing and immediately displays the *Design Status* screen. Close this screen to return to the *Footing: Isolated Pile/Shaft Cap Design* screen.
17. Click **OK** to return to the *Footing* screen. The diagram at the top of the screen shows the selected footing design.

Step 18

This completes RC-PIER Tutorial 3. To print the output of the project, select **Print** from the **File** menu.

Save your file by selecting **Save** from the **File** menu. Exit the program by selecting **Exit** from the **File** menu.

Following this tutorial is a printout of the output for selected items from Tutorial 3.

Theory

Chapter

TH

This chapter explains the assumptions and procedures used in RC-PIER®. It explains such topics as pier types, pier components, sign conventions, geometry, loads, analysis, and design.

RC-PIER has the ability to work with dual codes. This means that the user has the option of using either the AASHTO LFD design code or the LRFD design code, in accordance with *AASHTO Standard Specifications for Highway Bridges*, Seventeenth Edition, 2003, and *AASHTO LRFD Bridge Design Specifications*, Third Edition, 2004.

The program can also work with dual units, either metric (SI) or U.S. units. Although you can switch between metric and U.S. units at any time during program execution, some round-off errors may occur due to the conversion between both system of units.

Pier Types

The pier types available in RC-PIER are hammerhead and multi-column. Each type is explained below.

Hammerhead Pier

A unique feature available in RC-PIER is the capability to design hammerhead piers. RC-PIER simplifies the structural model of a hammerhead pier to two perpendicular lines, representing the centerline of the cap and column. Thus, the simplified structural model of a hammerhead looks like a T-shape. If there is a void in the hammerhead, the same structural model is applied, but the reduced section area and stiffness of the members is used to model the void. Note if a large opening is introduced, the T-shape model may produce inaccurate results. In this case, it is recommended to use the strut-and-tie model (STM).

After the structural analysis is performed, moments and forces at the sections for the cap or hammerhead are used in the design. If there is an opening, the beam design will provide the results for the segment between end of the cap and the face of opening. The section between the faces of the opening is not considered in the design and the user should check these sections by other methods. Column design will be based on the section moments and axial force. A reduced section property will be used if an opening exists. For such hammerhead columns with void/opening, the program designs the column sides as a single column and not as two separate columns.

For more information on the design of hammerhead piers by strut-and-tie model, see page TH-33 in this chapter.

Multi-Column Pier

Multi-column piers use the conventional frame analysis. The centerline of each component is used in the structural model. The structural model is composed of members and nodes. The proper section geometry, moment of inertia, and material properties are defined for the members. The bottom of the column is fixed. However, RC-PIER allows the option to add a spring in the vertical direction to simulate the settlement of the pier structure.

Multi-column designs are performed for cap, column, and footing individually. Note that you can design hammerhead piers by selecting the pier type as multi-column. In this case, you need to provide a single column.

Pier Components

Pier Caps

The pier cap is defined by dimensions, i.e., size, start and ending elevation, skew angle, and factor of reduced moment of inertia. There are four types of pier caps: Hammerhead, Straight, Tapered, and Variable. Regardless of the type of cap selected, the program will define sections that are symmetrical about the centerline of the cap in the Z-direction. Bearing points sit on the top of the cap. The eccentricity of the bearing points can not be more than half of the cap width.

Column

In RC-PIER, you can add columns to the pier structure, except for hammerhead structures. The shape of the column can be rectangular, round, tapered, rectangular chamfered, hexagonal and octagonal. Columns could also comprise of several segments with linear or parabolic variation for each segment. You can also add a drilled shaft to the column. The minimum and maximum column sizes are 12 in. and 1000 in., respectively, for U.S. units, and 300 mm and 25400 mm for metric units.

Drilled Shaft

There are two types of drilled shaft available: circular and rectangular. The section of a drilled shaft is defined by the width and depth or diameters. To determine the location of a drilled shaft, the program uses the total height of the drilled shaft and the partial height of the drilled shaft below fixity point.

A drilled shaft is divided into two portions: above fixity point and below fixity point. The drilled shaft is fixed (no displacements and rotations) at fixity point, in the frame analysis. The specifications for the structural design of the drilled shaft are in accordance with AASHTO LFD Art. 4.6.6 and AASHTO LRFD Art. 10.8.3.

RC-PIER takes the portions of the drilled shaft below fixity point as a member supported laterally. In this case, only the axial load is considered. For the portion of the drilled shaft above fixity point, RC-PIER assumes there are no lateral supports. Therefore, it is designed exactly the same as a reinforced column, taking into consideration both axial load and moments.

Footing

There are four footing types available that you can design for each column or combination of columns: spread, pile/shaft cap, combined, and strap. The forces and moments at the bottom of the column (or top of the footing) are used in the footing design. To calculate the bearing pressure or pile reactions, the assumption of infinite rigid footing is assumed.

Sign Conventions

The sign conventions for Bending Moment Diagram (BMD) and Shear Force Diagram (SFD) are as follows:

For Bending Moment Diagram (BMD), RC-PIER follows the right-hand rule, that is, if the moment vector is acting along the positive direction of any axis, it is a positive moment about that axis. In other words, when considering the moment acting about any axis, if the moment is counter-clockwise looking from the positive direction of that particular axis, the moment is positive.

Figure TH-1 on page TH-3 illustrates the bending moment diagram (drawn on tension side) as per the above sign convention for the frame in Tutorial 1 (load combination #46). Figure TH-2 on page TH-3 shows the sign convention for BMD (right-hand rule).

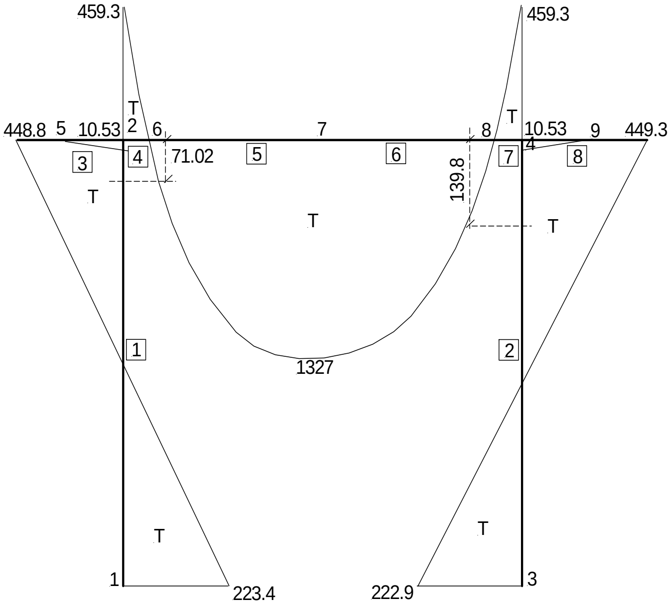


Figure TH-1 Bending Moment Diagram for Load Combination #46 from Tutorial 1 (Moments shown in k-ft) (T indicates tension)

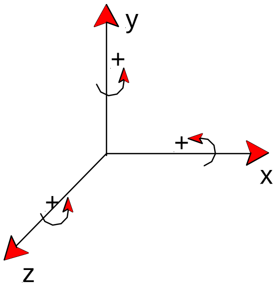


Figure TH-2 Sign Convention for BMD

For Shear Force Diagram (SFD), the program follows the standard left up/right down positive sign convention, as shown in Figure TH-3 on page TH-3.

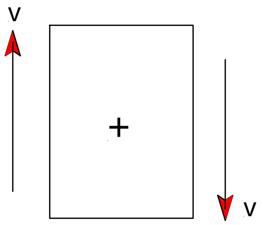


Figure TH-3 Sign Convention for SFD

For axial loads, if a load acts along the positive direction of the axis, it is positive.

Note: In all three cases, the sign for a particular load may vary depending on whether the local or global coordinate system is being considered.

Geometry

This section covers some important concepts of the terms used in RC-PIER.

RC-Pier assumes that bridge superstructure is centered on pier cap. Also, it assumes that left and right curb distances are equal

Bridge Span Information

When you define bridge spans, you must define all the spans that will be continuous and will affect the load generation. To do that you should start specifying span lengths starting from the least station and move towards largest station. Please see Figure TH-4 and Figure TH-6, as how spans should be defined for a three span bridge with respect to the direction of stationing. This data is used for the generation of loads.

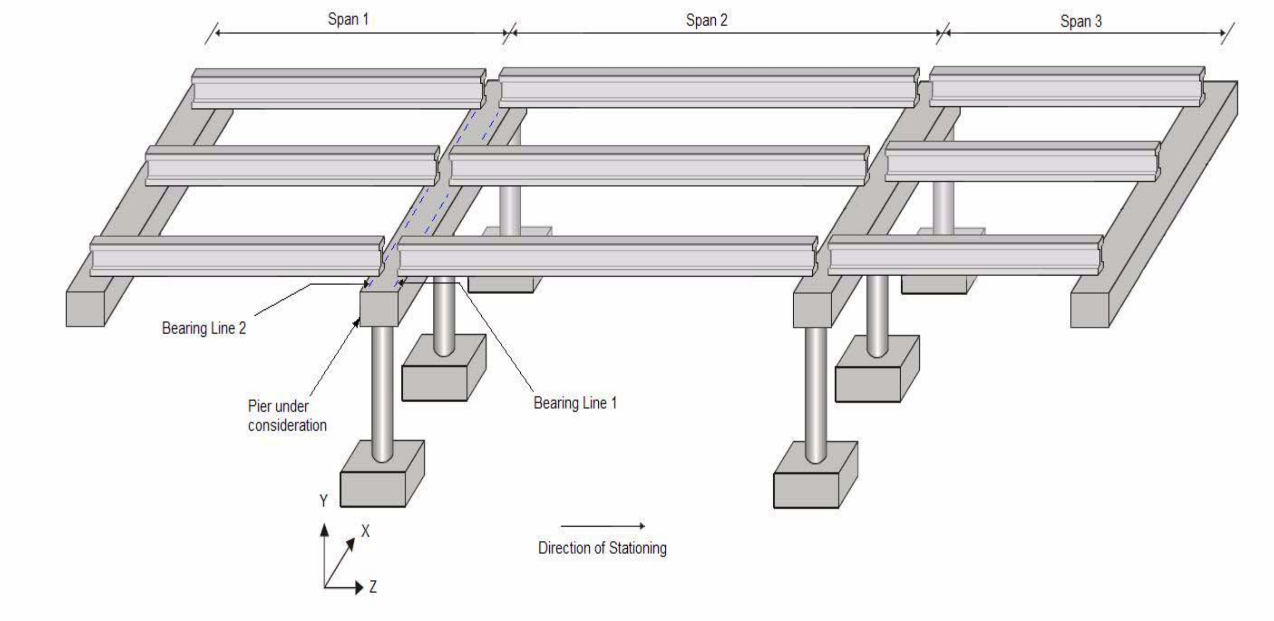
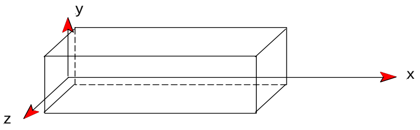


Figure TH-4 Bridge Span and Spacing

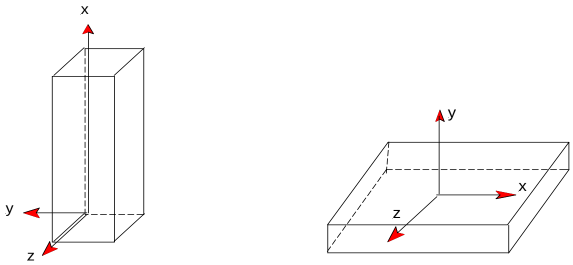
Global and Local Coordinates

There are two different types of coordinate systems used in RC-PIER: global and local coordinates. Global coordinates refer to the coordinate system for the entire structure, where positive X-direction is along the length of the pier cap (i.e., centerline) from the left end to the right end when looking the pier downstation, positive Z-direction is in the direction of the traffic flow, and positive Y-direction is in the upward vertical direction (see Figure TH-4 and Figure TH-6). Global coordinates are the same for each of the three components of the structure, namely the pier cap, column, and footing. For a skewed bridge, the positive X-direction is along the centerline of the pier cap from left to right and the other two axes are oriented accordingly.

Local coordinates refer to the individual coordinate system for each of the three components of the structure, namely, the pier cap, column, and footing, as shown in Figure TH-5 on page TH-5 (a, b, and c). Note that for the pier cap and footing, the local and global coordinates are identical but for the column, the local positive X-direction is along the length of column from the bottom to the top.



a) Local Coordinates for Pier Cap Only



b) Local Coordinates for Column Only

c) Local Coordinates for Footing Only

Figure TH-5 Local Coordinates for Pier Cap, Column, and Footing

Cap Length

Cap length refers to the horizontal length of the cap, or the length of the horizontal projection, if the cap is inclined.

Cap Elevation

Cap elevation is defined at its top surface, at left or rightmost ends of the cap.

Centerline of a Cap

The centerline of a cap is a straight line passing through the mid-height of the smaller of the two ends of the cap and is parallel to the top of the cap.

Skew Angle (in degrees)

The skew angle is defined as the angle between the normal to the centerline of the bridge and the centerline of the pier cap (in positive X-direction). It is positive if measured in counterclockwise direction, as shown in Figure TH-6. Note that the skew angle is used only for auto load generation.

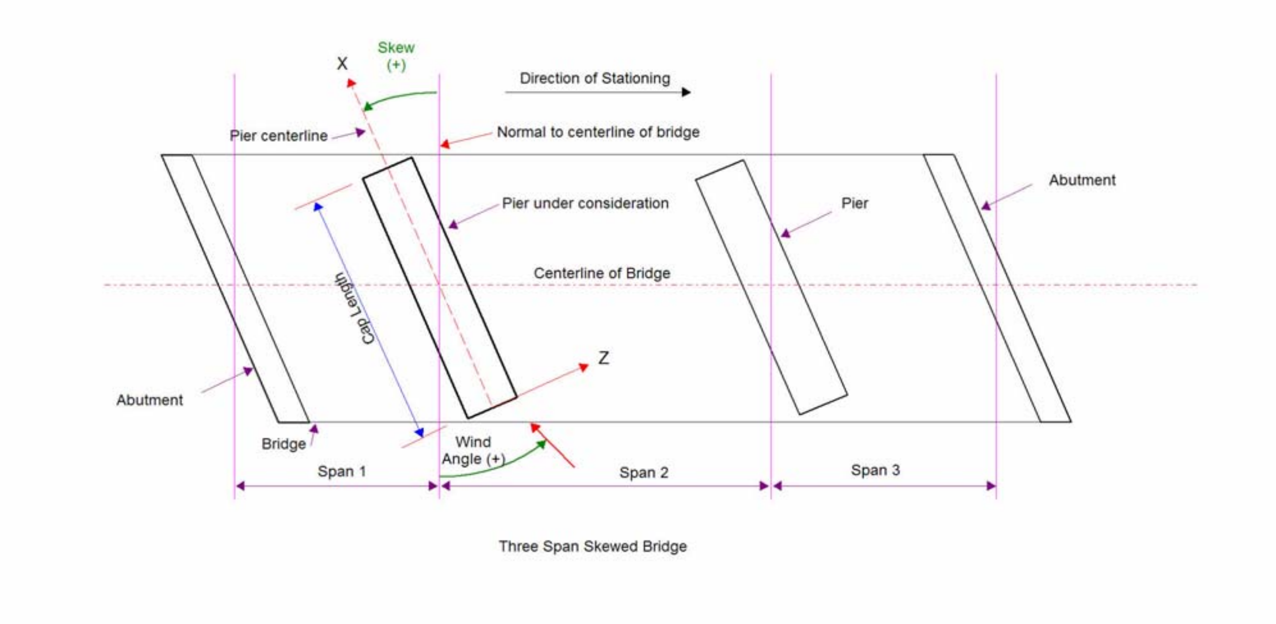


Figure TH-6 Bridge with Piers at Skew

Column Shapes

RC-PIER allows the following column shapes, as shown in Figure TH-7.

1. Round (Prismatic)
2. Rectangular (Prismatic)
3. Rectangular Tapered in X (Non-Prismatic)
4. Rectangular Tapered in Z (Non-Prismatic)
5. Rectangular Tapered in X and Z (Non-Prismatic
6. Rectangular Chamfered (Prismatic or Non-Prismatic)
7. Hexagonal (Prismatic and Non-Prismatic)
8. Octagonal (Prismatic and/or Non-Prismatic)
9. Rectangular Filleted (Prismatic and Non-Prismatic)

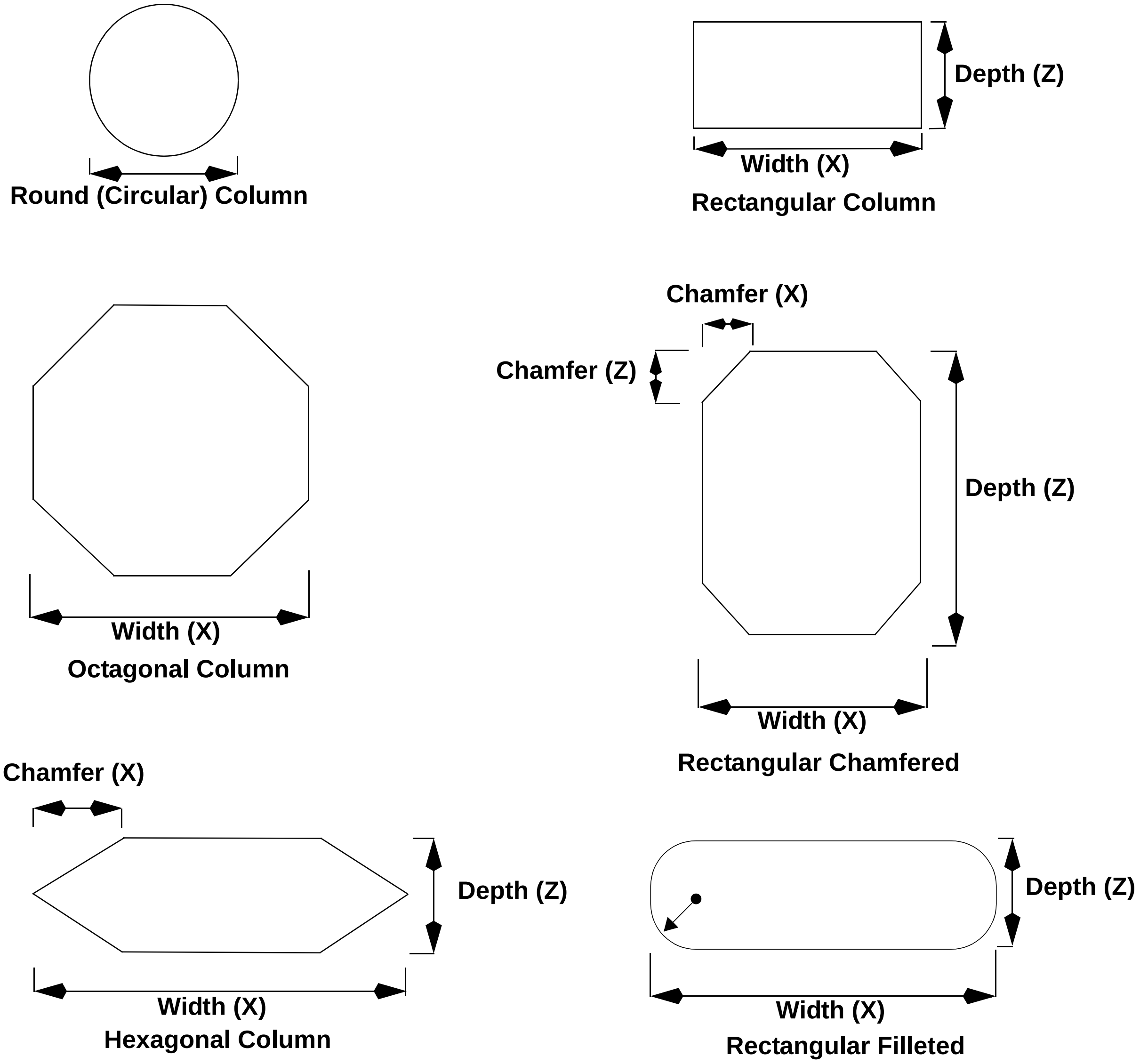


Figure TH-7 Column Shapes

Column Variation

Each segment for a non-prismatic column could either have a linear variation or a parabolic variation. If the variation is parabolic, RC-PIER internally converts that to multiple linear segments for analysis and design purpose.

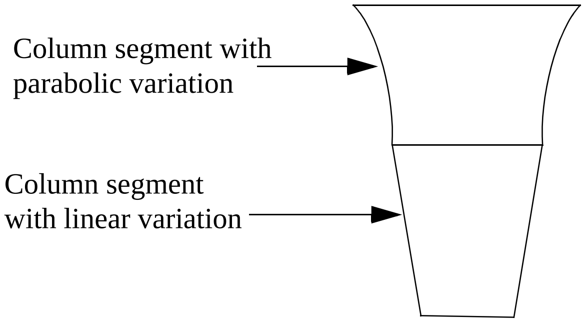


Figure TH-8 Column Variation (Tapering)

Column location is measured with respect to the left end of the cap. For each column you can specify bottom elevation. However, top of column is automatically extended to the centerline of the cap. In structural analysis as well as design, the length to the cap center line is used as the column length.

Auto Node Generation Pier Model

The connection point between different components is automatically assigned as a node. Any additional checkpoints you add will also be a node. By default, the node at the bottom of the column is fixed. This means there are no displacements and rotation at this node. You can add a spring at the bottom of the column in the global Y-direction, and also a hinge at any node in the global Z-direction. When adding a hinge at the node in Z-direction, the moment about this direction will be zero.

Bearing Lines

Bearing lines is where the loads from superstructure are transferred to the substructure. RC-PIER allows a maximum of two bearing lines. Eccentricity can be given to the bearing line. Positive eccentricity is measured from the pier cap centerline to the positive global Z-axis. Negative eccentricity is measured from the pier cap centerline to the negative global Z-axis. For an inclined pier cap, the bearing point location is measured horizontally, instead of along the pier cap.

For skewed piers with double bearing lines, distinct locations along X are to be specified even when girders are parallel to bridge.

Loads

Loads in RC-Pier are specified in distinct load cases corresponding to load types specified by AASHTO Standard or LRFD. All loads may be manually input. However, RC-Pier can generate most commonly encountered loads automatically.

The program generates the load listed in section *Auto Load Generation* on page TH-9

It is, however, the user's choice to use this generation. For specific cases, the user might want to input already computed loads directly, even for the load types that the program can generate. All other loads, when needed should be manually input.

Loads can be applied at bearing points, on the column or at the cap directly.

Manual Input Loads

In RC-PIER, you can manually input the loads for any load types. Bearing loads, column loads and cap loads are applied in pier global axis system. In case of a pier on a skew, the global X and Z-directions are not the same as the bridge transverse and longitudinal directions. As a result, you need to decompose longitudinal or transverse loads acting on the skewed bridge to those in the global X and Z direction of the pier and then use the decomposed loads in RC-PIER.

Bearing loads are applied at the top of the pier cap. Therefore, if you have a bearing load in Z-direction, this load will produce torsion in the cap. On the other hand, cap and column loads are applied at the centerline of the cap and column, respectively.

When inputting the live load manually, please specify the loads without the live load impact factor. If the superstructure information is specified, the program computes the appropriate impact for the loads. However, you can change this option in the Analysis/Design parameters on the *Analysis* tab. Note that in AASHTO LRFD, different impact factors are applied to Truck and Lane loads. For manually input LRFD live loads, the program allows two sets of bearing loads. In this, the first set corresponds to the truck loads and the second set corresponds to the lane loads. For auto generated LRFD live loads, the program keeps two separate sets of loads also. In both manual or auto generated loads, the program appropriately applies impact factors when the analysis is performed. Further, note that when inputting loads manually, reduce the loads by the multi-presence factor and then input the loads.

Column loads are located with respect to the fixity point.

Any number of cap loads can also be specified. The location of loads is measured as a ratio with respect to the length of the cap. If a load is applied in the middle of the cap, its location should be a X/L = 0.5. For concentrated force in X or Z direction, RC-PIER also allows you to specify moment arm. Arm should be measured from the load application point to the cap centerline. (Please refer to *Centerline of a Cap* on page TH-5). In addition to concentrated loads, distributed loads or point moments can also be specified at any location along the length of the cap.

Auto Load Generation

RC-PIER can automatically generate the following loads:

- Self load of the pier
- Dead Load from the superstructure
- Live Load
- Longitudinal Force/Braking Load
- Centrifugal Load
- Wind Load on Structure
- Wind Load on Live Load
- Temperature Loads from Superstructure
- Earthquake Load
- Vessel Collision (LRFD)

Self Load

The program can determine the self load of the pier. The program maintains this load internally. The results due to these loads are visible to the user after a successful completion of the analysis. These analysis results can be seen under the “self load” cases.

In the generation of these loads, the entire cap load is applied to the internal model on cap elements. For column members, the load is generated to the bottom of the cap and applied to the column elements. The weight of the column is generated from the bottom of the column to the cap bottom where the centerline of the column intersects. In rare cases where the column is wider than the cap and extended above the cap bottom, the self load of the part wider than the cap is ignored for the part above the cap bottom line.

Auto Dead Load Generation

Program provides four options for the computation of bearing loads due to dead load from superstructure.

The first option uses the simple span distribution to compute bearing loads. For slab and girders you will specify material unit weight. Program will use the girder cross sectional area, bridge width and slab thickness and determine the load on each bearing location. For barriers, the load should be the total load for two barriers (when two barriers are present, one on each side.) The load from the slab, barriers and wearing surface is distributed among bearings based on each bearing/girder tributory width. If you want to distribute barriers differently, please specify loads as manual input.

When the second option is selected, RC-PIER computes maximum dead load reactions for the girder, slab, barrier and wearing surface using a continuous beam model at the pier and then generates the bearing loads.

The third option lets user specify a computed superimposed DL reaction on the pier. If the pier supports double bearing lines, program first computes the load that will be supported by each line. If the spans supported by two bearings are not equal, user should change the factors for correct distribution of load among two bearings. For single bearing line, all load is supported by the bearings on this line. Once total load to be supported by each bearing is computed, program then distributed this load among bearings using their corresponding tributory width. When this option is selected, user can still at the same time choose to generate girder and slab load based on simple span approach (this may be helpful for pre-stressed concrete superstructures made continuous). For steel bridges, the entire reaction be specified including the slab and girder acting on composite and no slab or girder loads generated.

The fourth option lets user read the composite dead load reactions from Conspan. In order to do that, you must first export the analysis results from within Conspan to a file. Afterwards, you can import that in RC-PIER. Conspan computes separate reactions for DL and ADL and supplemental composite load on each pier. When imported into RC-PIER, program first determines total reaction and then distributes among bearing lines and bearings using a very similar approach explained earlier in this section for a manually input reaction. Please note that in Conspan, slab and girder loads act on simple span. For that reason, those are not imported. In this case, it is suggested that you generate the girder and slab loads in addition to importing the reaction from Conspan.

If you have a continuous bridge and you generate loads using simple span approach, you can apply a continuity factor to the dead load and live load generated by the program. For example, a three span continuous bridge having equal spans of 100 ft would have a continuity factor of 1.10. You can input this value manually in the Multiplier for Loads text box on the *Loads: Load Data* screen (see Fig. GO-25 on page GO-20). The continuity factor is then multiplied to each of the loads under the *Loads: Load Data* screen.

Auto Live Load Generation

Program proceeds through following steps to generate live load on piers.

- Determine longitudinal reaction by moving truck in both directions
- Determine all possible transverse positions of trucks and solve for member responses
- Determine possible combinations of transverse positions and combined responses
- Select all sets that cause critical effects in any member

For the very first step program can either calculate the reaction internally or user can specify already computed lane and truck reactions or import the governing reactions from Conspan.

When the option to compute live loads longitudinal reaction for simple span is selected, program determines the maximum live load reactions for the selected trucks using simple span assumption (without the impact) along the longitudinal bridge direction. For double bearing lines, the program places each axle consecutively on each bearing line longitudinally and determines the different live load cases for reaction and torsion. Only those live load cases, which produce maximum reaction(s) and/or torsion(s) on the bearing lines, are selected for the next step of analysis. The uniform lane live load is multiplied by the length of live load and the shear rider (concentrated force) is added for calculating the reaction.

When you select the second option, RC-PIER computes the maximum continuous beam live load reactions at the pier for selected trucks. For double bearing line case, this load must be divided among the two lines. It is recommended to modify the reaction distribution among bearing lines. RC-PIER will then use the these factors to compute the load on each bearing line and then after transverse positioning, determine the critical load cases with critical bearing loads.

If you have already performed the live load analysis for superstructure which may be done using a continuous analysis, and you know the reaction at the pier, you can specify that using the third option. When this is specified, program computes the load on each bearing line using line factors. Further use of these reactions is somewhat different in AASHTO LRFD or AASHTO standard. When in AASHTO standard, if truck and lane reactions are specified, those are treated separately and are individually used for Step 2. However, in LRFD mode, live load contains load from truck axles as well as from Design Lane. Therefore, program uses both Truck and Lane and treats them as a single live load. When reversible cases are computed, program reverses the computed loads on bearings 1and 2 and may result in additional cases. When pier supports two unequal spans, you may generate load twice with separate factors rather than selecting the reverse option.

The fourth option lets you import the Conspan computed reactions. In order for you to use this option, you must first run analysis in Conspan and export the results using *File: Export check results.* When results are imported in RC-PIER, program provides information about supported spans for all piers but the program does not restrict the use of information if the information for the adjacent spans does not match. User must decide carefully about the reactions being imported. In case of LFD design, program will import controlling load which could either be lane or truck load. In case of LRFD, program will import lane load as well as truck load. Truck load may be the controlling load of Design Truck, Design Tandem, Design Double Load or Design Double Tandem whichever trucks were included. Once these loads are imported, the remaining procedure is very similar to the second option of input of reaction.

In the second step, the reaction(s) computed along the longitudinal direction in step 1 are distributed transversely, using the lever rule, across the bridge width. This is done by using either the Constant Spacing Method or the Variable Spacing Method.

Multi-presence factor is considered in auto live load generation and the generated bearing loads are already reduced by the multi-presence factor for that load case. This is done on a load case basis because different load cases might have a different number of truck lanes present for which different values of the multi-presence factor should be used.

Constant Spacing Method

Using this method, the user specifies the Minimum Distance from Curb (see definition below) between the left curb and the left wheel of the truck or the right curb and the right wheel of the truck. The Constant Center to Center Spacing (see definition below) between two consecutive live load positions is also specified. Once the user inputs these two values, the program will generate the first position by placing the truck load at the minimum distance from the left curb. The next position is obtained by moving the live load (truck) to the right of the first position by the constant center to center spacing. This process is continued transversely across the bridge width as long as the distance between the right wheel of the truck and the right curb is greater than or equal to the minimum spacing allowed.

Note that for lane loads, the first position is adjacent to the left curb (without any minimum spacing from the curb). The next position is obtained by moving the lane load to the right by the constant center-to-center spacing. The process is continued as long as there is enough space to accommodate another lane load between the rightmost end of the previous position and the right curb. The same process is repeated by placing the lane load adjacent to the right curb and moving it transversely from right to left across the bridge until no more space for the lane load to be placed.

Once all the live load positions are determined, RC-PIER creates the different live load combinations depending upon the number of loaded lanes selected.

Minimum Distance from Curb (For Constant Spacing)

This is the minimum user-specified distance from either the left curb of the bridge to the left wheel of the live load or the right curb to the right wheel of the live load. This input value is only used for truck or tandem loads and not for lane loads.

Default values: 2 ft or 0.6 m.

Range of values that can be input: 0 to 10 ft (or 0 to 3 m).

Note: For the constant spacing method, the lane load is always placed adjacent to the left or right curb.

Center to Center Spacing (For Constant Spacing)

This is the user-specified, center-to-center spacing for placement of various live loads (lane, truck, or tandem) for the generation of different live load positions. For example, if center-to-center spacing = 10 ft, RC-PIER will first, determine the initial position of the live load from the left end of the bridge (first live load position). Assume that the first position is 6.5 ft from the left end, the program will then add the center-to-center spacing to the first position to obtain the second live load position, which is 16.5 ft away from the left end of the bridge. RC-PIER will repeat the same process by starting from the right side of the bridge.

Default values: 10 ft or 3 m.

Range of values that can be input: 1 to 20 ft (or 0.3 to 6.0 m).

Variable Spacing Method

The Variable Spacing Method places the live loads at specific positions along the transverse direction of the bridge based on the value of the Minimum Spacing Between Locations. The algorithm used for the generation of individual live load positions is explained after the Minimum Spacing Between Locations (for Variable Spacing) section.

Minimum Spacing Between Locations (for Variable Spacing)

This is a user-defined spacing value that specifies the minimum center to center distance that should be maintained between any two consecutive live load (LL+I) positions that are considered by the program while auto generating the various (LL+I) cases.

In other words, RC-PIER will first generate all the possible (LL+I) positions internally (according to the live load generation algorithm discussed later). Then, based on the Minimum Spacing Between Locations value specified by the user, it will combine only those live load positions, which are separated by at least that minimum distance from the last considered position (going from left side of the bridge to the right). The remaining positions are not considered for the purpose of combination. This option is provided to reduce the total number of (LL+I) cases used for analysis when the generated positions are very close to each other. Thus, by specifying the value for Minimum Spacing Between Locations, the user may reduce the total number of live load positions used for combinations and consequently the analysis time.

Default values: 0.5 ft or 0.15 m.

Range of values that can be input: 0 to 20 ft (or 0 to 6 m).

The above definition may be illustrated with an example taken from Tutorial 1. In Tutorial 1, if the user does an auto live load generation with HS20 truck (one-lane loaded), the live load positions generated by the program corresponding to Minimum Spacing Between Locations values of 0.5 ft, 1 ft and 2 ft are shown in the following table (for a different bridge, the positions would be different).

All possible positions (ft) / Min. spacing cases	**6.5**	**7.5**	**9.5**	**10.5**	**13.0**	**13.5**	**16.5**	**17.5**	**19.5**	**20.5**
a) Min. spacing between locations = 0.5 ft	√	√	√	√	√	√	√	√	√	√
b) Min. spacing between locations = 1 ft	√	√	√	√	√	×	√	√	√	√
c) Min. spacing between locations = 2 ft	√	×	√	×	√	×	√	×	√	×

√ - LL+I positions considered for combination

× - LL+I positions not considered for combination

From the table above, when the input value is 0.5 ft, all 10 generated live load positions are considered since none of those positions are less than 0.5 ft from each other. When the input value is 1 ft, only the position corresponding to 13.5 ft is not considered, since this position is at a distance of less than 1 ft from the immediate previous live load position at 13 ft. Similarly, when the input value is 2 ft, only 5 positions are considered for combination such that all those positions are at least at a minimum spacing of 2 ft from each other.

Once the program has all the available live load positions, it will generate the final live load cases according to the criteria discussed above.

The individual live load positions generated by the variable spacing method are obtained using the following algorithm:

Truck Loads

Case 1: The first position is obtained by placing the left wheel of the truck at a distance of 2 feet from the face of the left curb per AASHTO. The next position is obtained by moving the centerline of the truck 10 feet to the right so that a minimum of 4 feet is maintained between the right wheel of the original position and the left wheel of the new position. This process is repeated and additional positions are created until the minimum distance requirement of 2 feet between the right wheel of the truck and the face of the right curb is not violated.

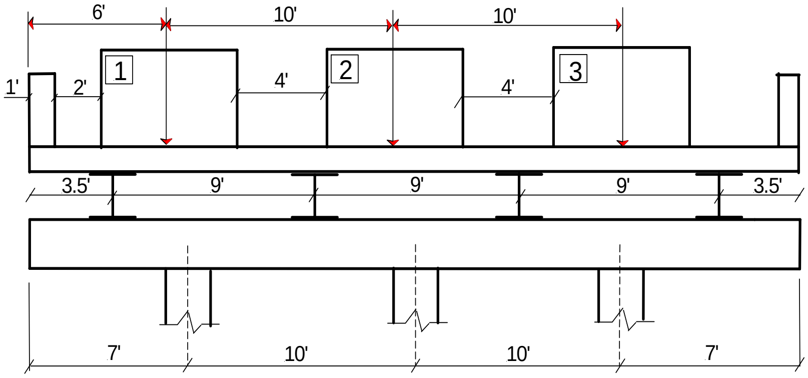


Figure TH-9 Truck Loads - Case 1

Case 2: RC-PIER repeats the same process as Case 1, but starts by placing the right wheel of the truck at a distance of 2 feet from the face of the right curb. Additional positions are obtained by moving the centerline of the truck in 10 feet increments transversely across the bridge until the minimum distance requirement of 2 feet between the left wheel of the truck and the face of the left curb is not violated.

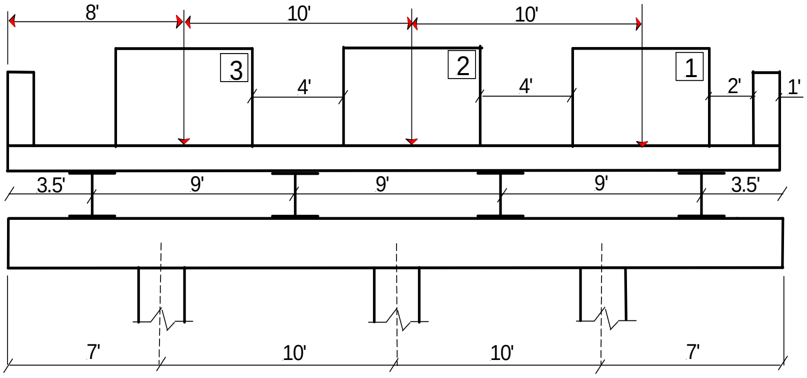


Figure TH-10 Truck Loads - Case 2

Case 3: The left wheel of the truck is placed directly on each of the bearings and additional positions are created by moving the centerline of the truck 10 feet to the left and to the right of the previous positions until the minimum distance requirement of 2 feet between either of the two wheels and the face of the curb is not violated.

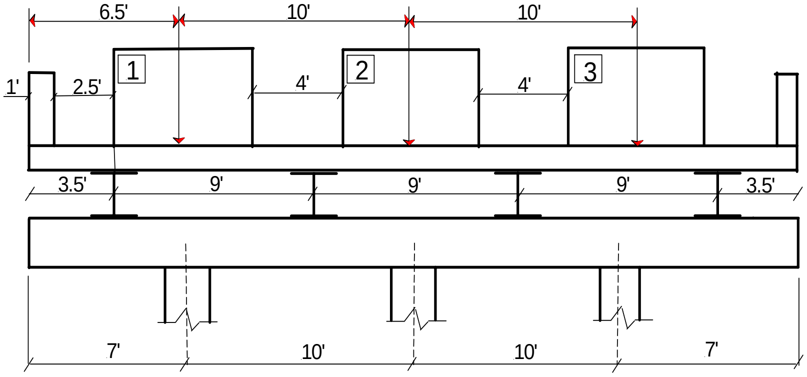


Figure TH-11 Truck Loads - Case 3

Case 4: The right wheel of the truck is placed directly on each of the bearings and additional positions are created by moving the centerline of the truck 10 feet to the left and to the right of the previous positions until the minimum distance requirement of 2 feet between either of the two wheels and the face of the curb is not violated.

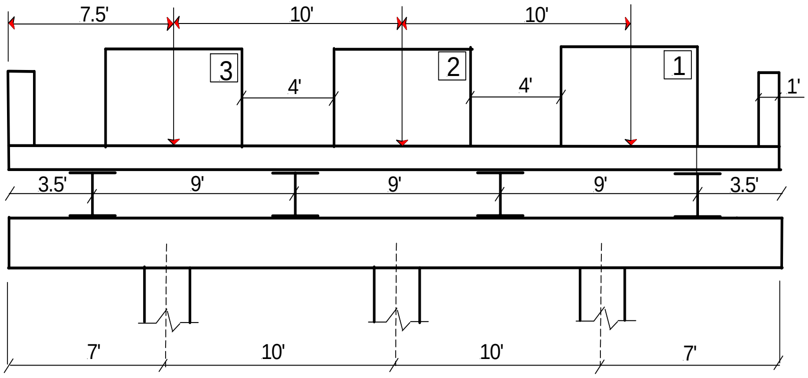


Figure TH-12 Truck Loads - Case 4

Case 5: The centerline of the truck is placed directly on each of the bearings and additional positions are created by moving the centerline of the truck 10 feet to the left and to the right of the previous positions until the minimum distance requirement of 2 feet between either of the two wheels and the face of the curb is not violated.

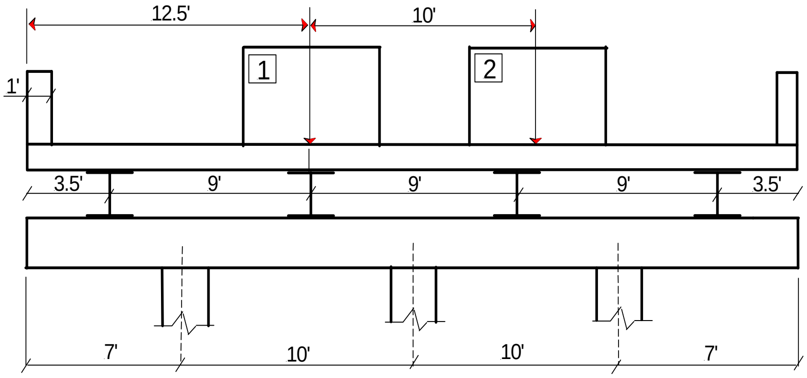


Figure TH-13 Truck Loads - Case 5

Case 6: The centerline of the truck is placed coincident with the centerline of each of the columns and additional positions are created by moving the centerline of the truck 10 feet to the left and to the right of the previous positions until the minimum distance requirement of 2 feet between either of the two wheels and the face of the curb is not violated.

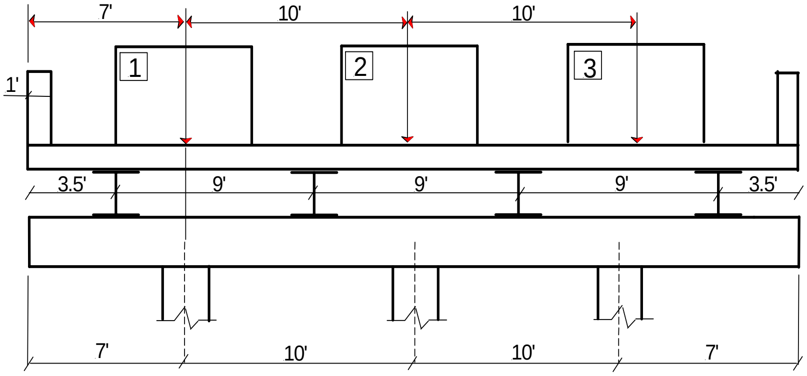


Figure TH-14 Truck Loads - Case 6

Lane Loads

Case 1: The first position is obtained by placing the left edge of the lane load adjacent to the left curb. The next position is obtained by moving the centerline of the lane 10 feet to the right so that the left edge is adjacent to the right edge of the previous position. This process is repeated until there is no more available space to place another lane load between the right edge of the last position and the face of the right curb.

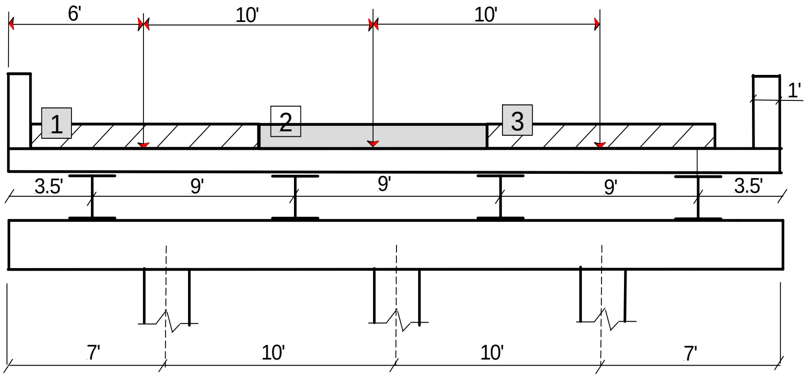


Figure TH-15 Lane Loads - Case 1

Case 2: The right edge of the lane load is placed adjacent to the right curb. Additional positions are obtained by moving the centerline of the lane 10 feet to the left so that the right edge is adjacent to the left edge of the previous position. This process is repeated until there is no more available space to place another lane load between the left edge of the last position and the face of the left curb.

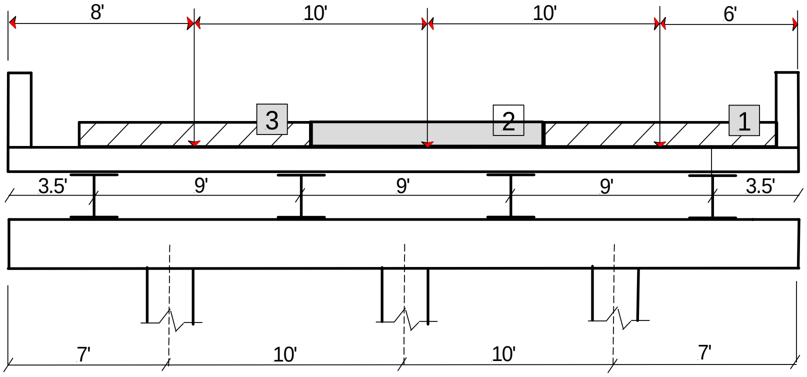


Figure TH-16 Lane Loads - Case 2

Case 3: The left edge of the lane is placed directly on each of the bearings and additional positions are created by moving the centerline of the lane 10 feet to the left and to the right of the previous positions until there is no more available space to place another lane between the left or right edge of the last position and the face of either curb.

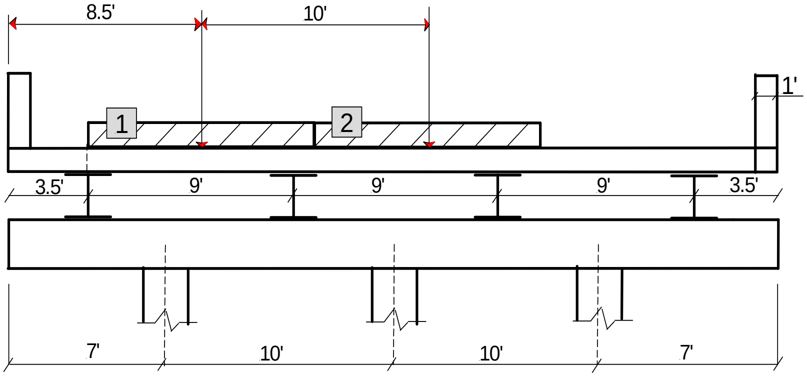


Figure TH-17 Lane Loads - Case 3

Case 4: The right edge of the lane is placed directly on each of the bearings and additional positions are created by moving the centerline of the lane 10 feet to the left and to the right of the previous positions until there is no more available space to place another lane between the left or right edge of the last position and the face of either curb.

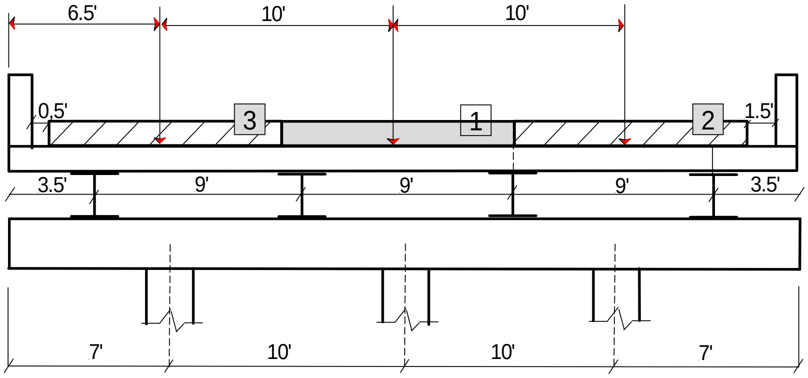


Figure TH-18 Lane Loads - Case 4

Case 5: The centerline of the lane is placed directly on each of the bearings and additional positions are created by moving the centerline of the lane 10 feet to the left and to the right of the previous positions until there is no more available space to place another lane between the left or right edge of the last position and the face of either curb.

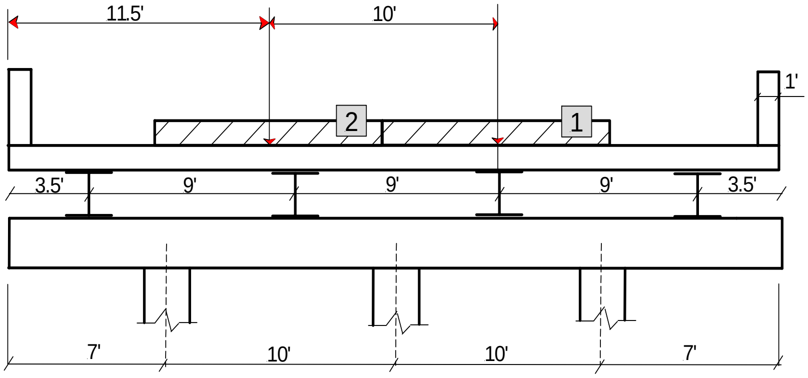


Figure TH-19 Lane Loads - Case 5

Case 6: The centerline of the lane load is placed coincident with the centerline of each of the columns and additional positions are created by moving the centerline of the lane 10 feet to the left and to the right of the previous positions until there is no more available space to place another lane load between the left or right edge of the last position and the face of either curb.

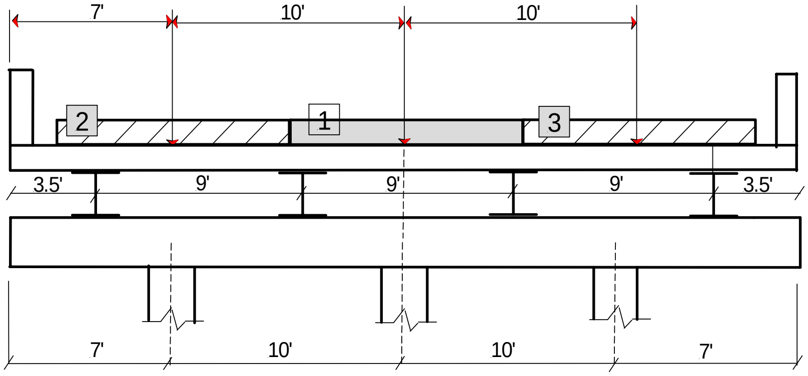


Figure TH-20 Lane Loads - Case 6

The above algorithm for live load positions are further used by RC-PIER as follows.

In the *Auto Load Generation: Live Load* screen, there are several loaded lane options (depending on the number of lanes entered by the user) in the Loaded Lane list box (e.g., 1 lane loaded, 2 lanes loaded, All combinations, etc.). For example, if you select 1 lane loaded, the program will investigate the effects of every live load position created by the algorithm. If you select 2 lanes loaded, the program will only investigate the effects of any two live load positions simultaneously. In this case, RC-PIER will select any two live load positions that are not overlapped. If you select All Combinations, the program will consider the effects for both the 1 lane loaded and 2 lanes loaded and report the maximum cases. The final live load cases for any specific number of loaded lanes are determined by the maximum load effect in each member. The maximum load effect means that the position of the specific loading(s) produces the maximum or minimum internal forces/moments in any of the frame members.

Note, for auto generation of live loads, RC-PIER internally applies the reduction of live load intensity/multiple presence factors to the effects on the structure model. If the live load reactions are entered manually, the user must include the reduction in live load intensity/multiple presence factors accordingly.

Permit Vehicle Side by Side

Program allows to have permit vehicle placed side by side with other regular trucks. In library user can specify if a live load is a permit vehicle or not. When such a truck is included for load generation along with other trucks which are not permit, program will consider placement of permit truck side by side with regular trucks. For such generation, however, the width of the trucks included should be same. After determining the transverse positions, program will try to first determine the critical loads as explained earlier for all regular trucks. After that, program will try placing one permit truck with all other regular trucks. It will generate additional load cases for such permit + regular trucks loaded. Live load details provide specific information as what truck positions are controlling for generated loads.

Live Load Generation for Skewed Bridges

For skewed bridges, the live loads are moved transversely across the width of the bridge (i.e., perpendicular to the centerline of the bridge), as described in the algorithm above. The horizontal projection (or cosine component) of the distance between bearing points measured along the centerline of the pier cap is used for calculating the live load positions and distribution (i.e., lever rule).

Impact Factor for Live Load

The impact factor is defined in the *Analysis/Design Parameters* screen under the *Analysis* screen (see pages GO-35 and GO-36 for further information). RC-PIER uses the impact factor for both the manually input live load and automatically generated live load.

The impact factor for live load is applied slightly differently for AASHTO LFD and LRFD. For both AASHTO LFD and LRFD, the generated bearing loads do not have the effect of impact factor applied. In addition, the analysis results of moments and forces due to the individual live loads shown on the *Analysis* screen are those without the impact factor. The effect of the impact factor is only applied to the analysis results of moments and forces for the load combinations and envelopes.

For AASHTO LFD, only one impact factor, for both the truck and lane, is used for both manually input and program generated live loads. However, for AASHTO LRFD, RC-PIER applies, by default, 33% and 0% as the truck and the lane impact factors, respectively. But, you can change the default impact factors by clicking **A/D Parameters** on the *Analysis* screen. Furthermore, RC-PIER distinguishes between truck and lane loads. When in LRFD, live load cases always contain two sets of bearing loads. This means if you have only 3 bearings on a pier, you will see a total of 6 loads. The first set of loads is treated as truck loads and the second set is treated as lane loads. Program applies the truck impact factor to first set of loads during analysis and the lane impact factor to second set of loads.

Results

To view the comprehensive results of the last auto generated live load, click **LL Details** on the *Loads* screen. This will activate the *Live Load Generation Details* screen. By selecting the options at the top of the screen, you can see results of the last generated live load as follows:

- Selected Item - displays the results for each controlling live load case.
- Selected Combinations - displays the results for all controlling live load cases, and
- All Load Positions - displays the results for all live load positions.

The results include the maximum effects produced in the members of the structure model, maximum live load reactions, truck and/or lane positions measured from the left edge of the bridge superstructure to the centerline of the truck/lane position.

The number of lane load positions vary according to the geometry of the superstructure, such as curb to curb distance, number of lanes, number of bearings, single or double bearing lines, and type of live load (military, truck, lane, or combination). Therefore, auto generation of the live load can be very extensive. The time involved depends on the number of live load positions created by the algorithm explained earlier, number of structural members, and speed of the computer. For a wide bridge (4 or more lanes), the number of live load positions can be very large. To limit the amount of execution time required by analysis, the maximum number of loaded lanes allowed is 4.

For AASHTO LRFD, the design truck and design lane have to be evaluated separately since the dynamic load allowance for the design truck and design lane are different. RC-PIER calculates individual design truck and design lane load results for each bearing. The first value(s) are for the design truck while the second value(s) are for the design lane.

Note that you are not allowed to delete the auto generated live load values but you can modify the loads or add additional loads.

Auto Longitudinal/Braking Load Generation

Longitudinal / braking load can be generated from two locations in the program. First is the generation from Longitudinal / Braking load case dialog. Secondly it can be generated along with Live load generation. When longitudinal load is generated from its own load case, only one longitudinal load case is generated by the program. When the longitudinal load is generated along with the live load generation, program generates same number of LF/BR cases as it generates live load cases. For this situation, every LF/BR case is generated for same number of live load lanes loaded as in the live load case. For example, if program generates 5 live load cases, it will generate 5 LF/BR cases. If the first live load case generated is based on two lanes loaded simultaneously, then first LF/BR case will also have two lanes loaded.

As per AASHTO Standard Art. 3.9, the longitudinal load as a percent of live load is,

C = 5% of lane load

As per AASHTO LRFD Art. 3.6.4., the braking force as a percent of live load is,
The braking force shall be taken as greater of:

C = 25% of the axle weights of the design truck/tandem, or

C = 5% of the design truck + lane load, or

5% of the design tandem + lane load

When lane load is used in computing the LF/BR force, the uniform load intensity is multiplied by a contributing length to compute the total force. This total force is then used to compute the LF/BR percent (C) above. RC-PIER assumes that average of the two spans adjacent to the pier as the default contributing length. However, there may be cases when a different length may be adequate. User must adjust this length in that case.

In LRFD mode, when a truck load is considered for generation, all axles of the truck are considered in computing the total load even if the span is too short to accommodate the entire length of the truck.

Program also provides an option in which the total live load values for lane and truck can be specified rather than selecting a truck or lane load. In AASHTO Standard since only lane is used, user can specify only lane load total. In AASHTO LRFD, both lane as well as truck total live load per lane have to be specified. In that case, program uses the value input as total for one loaded lane.

Once the percent C is computed, program then determines the total longitudinal load and then computes the bearing loads. In this computation, it uses the live load reduction factor / multiple presence factor. In computing bearing and cap loads it assumes that the total longitudinal load is applied at 6 ft above the deck in the direction of stationing. By default the LF and BR load cases are reversible in RC-PIER so this load gets considered in both directions.

The total longitudinal load is distributed among all the bearings as bearing loads. The moment about the X axis is applied as cap load and the moment computed about Z axis is applied as a force couple in Y direction at exterior bearings. If there are two bearing lines, half of the force is distributed on bearings in each bearing line. For skewed bridges, RC-PIER resolves the load into X and Z components resulting in bearing loads in both X and Z directions of the pier.

In case of skewed bridges, RC-PIER calculates the appropriate force and resolves it into X and Z components appropriately. The moments generated about X axis are applied as cap load whereas the moments generated about Z axis is applied as couple on the exterior bearings. If the pier has two bearing lines then each line will resist half of the moment which is then divided by the distance between the first and the last bearing to generate the force in Y direction.

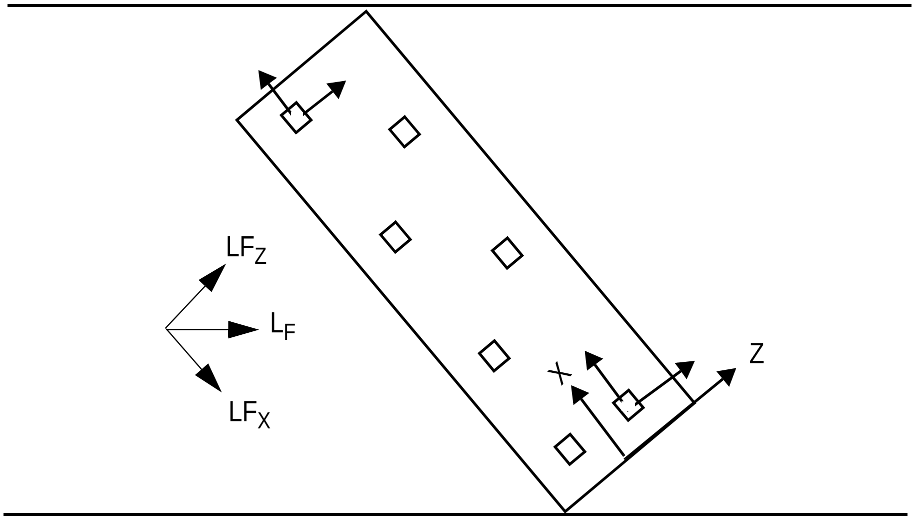


Figure TH-21 Auto Longitudinal/Braking Load Generation

Auto Centrifugal Load Generation

Centrifugal load can be generated from two locations in the program. First is the generation from Centrifugal load case dialog. Secondly it can be generated along with Live load generation. When centrifugal load is generated from its own load case, only one load case is generated by the program. When the centrifugal load is generated along with the live load generation, program generates same number of CF/CE cases as it generates live load cases. For this case, every CF/CE case is generated for same number of live load lanes loaded as in the live load case. For example, if program generates 5 live load cases, it will generate 5 CF/CE cases. If the first live load case generated is based on two lanes loaded simultaneously, then first CF/CE case will also have two lanes loaded.

As per AASHTO Standard Art. 3.10, the centrifugal load as a percent of live load is:

In US units:

$C = (6.68\ S^2)/R$

where:

C = the centrifugal force in percentage of Live Load, without impact.

S = the design speed in miles per hour.

R = the radius of curve in ft.

In SI units:

$C = (0.79\ S^2)/R$

where:

C = Centrifugal force in percentage of Live Load, without impact

S = Design speed in km/hr

R = Radius of curve in m

As per AASHTO LRFD Art. 3.6.3 and equation 3.6.3-1, the centrifugal force as a factor of live load is,

$C = 4/3\ v^2/gR$

where:

C = Centrifugal force as a factor of the total axle weight of design truck or tandem

V = highway design speed ft/sec (m/sec)

g = gravitational acceleration 32.2 ft/sec2 (9.8 m/sec2)

R = the radius of curve in ft. (m)

In this generation, all the axles of a truck are considered for generation even if the span is too short to accommodate the entire length of the truck.

Program also provides an option by which the total live load values for truck can be specified rather than selecting a truck from the library. When total truck load is specified, program uses the value input as total truck load per lane.

Once the percent (or factor) C is computed, program then determines the total centrifugal load and then computes the bearing loads. In computing bearing and cap loads it assumes that the total longitudinal load is applied at 6 ft above the deck in the direction of stationing. The load direction is as per user specified.

The curvature of the bridge alignment defines the direction of the centrifugal force of the bridge alignment. If the bridge curvature is such that the center of the curve is on the right side of the alignment looking upstation, as shown in Figure TH-22, user should specify to apply the load in +X direction. However, if the center of the curve is on the left side of the alignment looking upstation, as shown in Figure TH-23, user should specify to apply the load in -X direction.

The total centrifugal load is distributed among all the bearings as bearing loads. The moment about the X axis is applied at cap load and the moment computed about Z axis is applied as a force couple in Y direction at exterior bearings. If there are two bearing lines, half of the force is distributed on bearings in each bearing line. For skewed bridges, RC-PIER resolves the load into X and Z resulting in bearing loads in both X and Z directions of the pier.

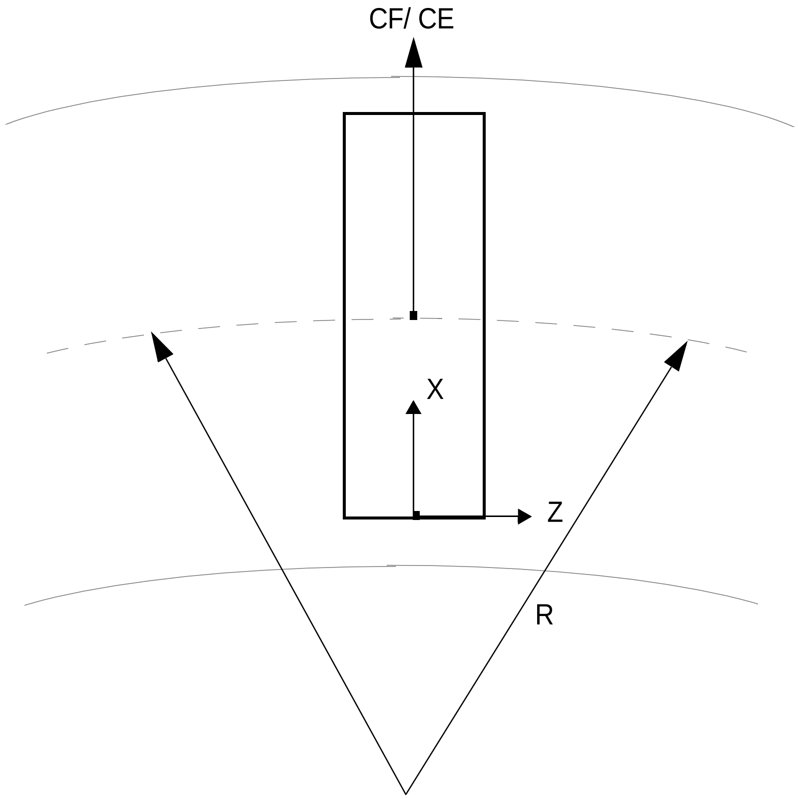


Figure TH-22 Centrifugal Load in +X Direction

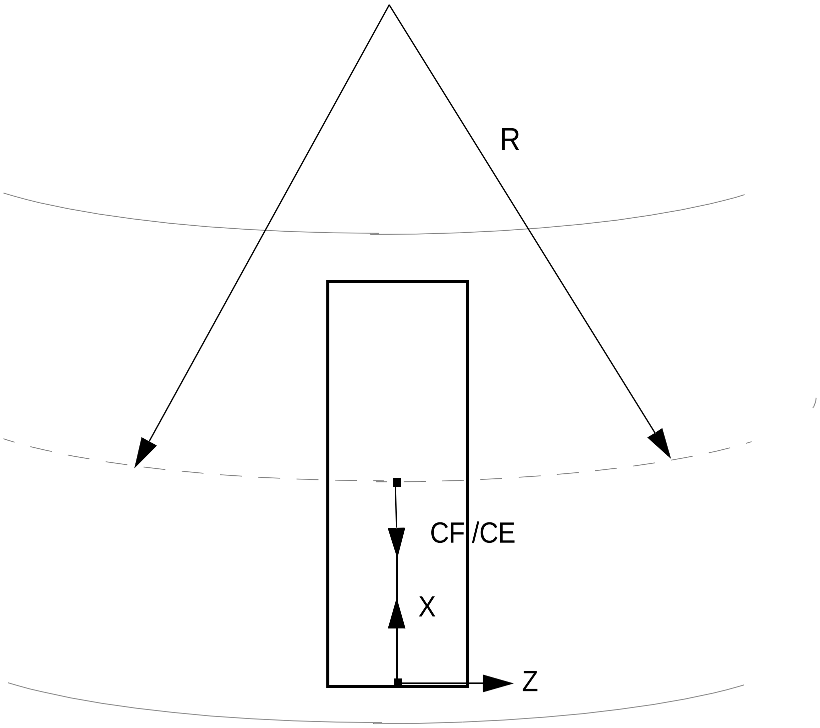


Figure TH-23 Centrifugal Load in -x Direction

Auto Wind Load Generation on Structure (LFD)

The wind angle is defined as the angle between the normal to the longitudinal direction of the bridge and wind direction, positive if anti-clockwise, measured from the bridge normal to the wind direction. The wind pressure on the superstructure depends on the wind angle and superstructure dimensions. By default, predefined values for wind pressure, in accordance with AASHTO specifications, are used. However, you can overwrite the default wind pressure and input custom values. The superstructure parameters must be defined before RC-PIER can automatically generate the wind load on structure.

The wind load is calculated in two steps:

1. Calculate the wind load acting on superstructure, and
2. Calculate the wind load acting on substructure.

To calculate the wind load acting on superstructure, program first calculates the superstructure area exposed to the wind load. This area is equal to the product of the total superstructure height and the average span length. Next, the wind pressures in the bridge longitudinal and transverse directions are multiplied with area to obtain the total forces, which are decomposed to the global X and Z directions if the bridge has a skew angle. Then, the total forces are averaged over the bearing points. The wind load also produces a moment due to the arm between the total forces and bearing points. This moment is assumed to be balanced by a force couple (in global Y-direction) acting on the external bearing points at left and right end of pier.

To calculate the wind load acting directly on the substructure, the projected area of pier cap and column are calculated first, depending on the wind direction and the bridge skew angle. For column shapes other than circular, RC-PIER uses the bounding rectangle of the section in the computation of the projected area. Taking a column as an example, the calculation of its projected area is illustrated as follows:

$$A_1 = b_z \cos(\theta_w - \theta_s) \times h$$

$$A_2 = b_x \sin(\theta_w - \theta_s) \times h$$

$$\text{Projected Area, } A = |A_1| + |A_2|$$

where:

b_z = column thickness, in global Z-direction

b_x = column width in global X-direction

h = column height exposed to wind pressure

θ_w = wind angle

θ_s = bridge skew angle

The projected area is multiplied by the wind pressure to obtain the total wind forces. The total wind force is subsequently resolved into global X and Z-directions, depending on the wind and bridge skew angle, as follows:

Total wind force, $F = P_w \times A$

where:

P_w= wind pressure

A= projected area

X-direction component, $F_x = F\cos(\theta_w - \theta_s)$

Z-direction component, $F_z = F\sin(\theta_w - \theta_s)$

The force acting on the cap in Z-direction is represented as a uniformly distributed load over the length of the cap, while the force acting in X-direction is applied as a concentrated load at the center of the cap. The length of the column subject to the wind load depends on the elevation above which the wind is applied (value entered by user on the *Auto Load Generation: Wind on Structure* screen).

For loads generated at zero degree angle wind, the user can include the load effects to Art. 3.15.3. If this option is selected, the program computes the total uplift by multiplying the wind pressure and the area of the bridge to the middle span on each side. This total force is applied at quarter point on the bridge. This will result in a moment on the bridge, which is equal to the force multiplied by a moment arm equal to 1/4 bridge width. The outer bearings resist this unbalanced moment and will result in equal force upward on one bearing and downward on the other. If there are two bearing lines present with other bearings having unequal distance, the average distance between the extreme bearings will be used to arrive at the bearing force.

Auto Wind Load Generation on Structure (LRFD)

The wind loads acting on the structure, for AASHTO LRFD, changes with the structural height, and exposure condition (bridge location). RC-PIER calculates the wind velocity first, using LRFD Eq. 3.8.1.1-1 as follows:

$$V_{DZ} = 2.5 V_o Ln\left(\frac{Z}{Z_o}\right)$$

where:

V_{DZ} = design wind velocity at design elevation

Z = height of structure, minimum of 30 ft

Z_o= friction length of upstream fetch

V_o= friction velocity

Z_o and V_o are as specified in LRFD Table 3.8.1.1-1 and depend on the bridge location. When calculating the wind velocity for superstructure, Z_{supstr}, RC-PIER uses the following:

Cap Elevation + Girder Height + Slab Depth + Railing Height

On the other hand, only cap elevation is used when the velocity of wind on pier cap and column are calculated.

After the calculation of wind velocity, RC-PIER determines the wind pressure, P_D, as follows:

$$P_D = P_B \frac{V_{DZ}^2}{10000}$$

where P_B = base wind pressure

For wind load applied directly to the substructure, RC-PIER takes P_B = 40 psf, and for wind loads from superstructure, uses LRFD Table 3.8.1.2.2-1. If $P_D < P_B$, RC-PIER always uses P_B.

For loads generated for wind at zero degree angle, the user can include the load effect due to Art. 3.8.2. This is similar to the load generation for LFD wind load generation according to Art. 3.15.3. Please read the description at the end of the Wind Load on Structure (LFD) section on page TH-24.

Auto Wind Load Generation on Live Load

As defined in the last section, the wind angle is the angle between the normal to the longitudinal direction of the bridge and wind direction, positive if anti-clockwise, measured from the normal of the bridge to the wind direction. The wind load on live load depends on the wind angle and live load length. It is calculated by multiplying the wind pressures in transverse and longitudinal directions with the live load lengths. Then, it is resolved into pier cap X and Z-directions (global coordinates) if the bridge is on skew.

The wind load acting on the live load is equally distributed among all bearings in the global X and Z-directions. The wind load also produces a moment due to the wind load acting above the bridge deck (e.g., 6 feet above the deck). This moment will result in a couple acting on the exterior bearing points at two ends of the pier and is applied as loads in global Y-direction.

The wind pressure on live load at 75 degrees is extrapolated from those for wind pressure angles from 0 to 60 degrees.

Auto Temperature Load Generation

Temperature load on the pier (from superstructure) is based on the amount of longitudinal movement in superstructure applied to the pier. The movement of a pier depends on the overall arrangement of piers, spans, bearing and skew in addition to temperature rise/fall and the coefficient of thermal expansion.

RC-PIER can do temperature load generation for two types of bearings. These are:

- Fixed bearings
- Expansion bearings.

For fixed bearings, RC-PIER only considers column stiffness to determine the thermal force.

Expansion bearings could be specified in the program:

- Elastomeric
- Sliding (plates)
- Rockers

Following table provides information as what items are considered in generation of thermal loads in the program.

Bearing	Bearing Stiffness	Column Stiffness	Friction Forces
Fixed	-	Δ_c	
Elastomeric	Δ_b	Δ_c	
Sliding	-	-	F_S
Rocker			F_R

For sliding bearings, RC-PIER computes the thermal force as a fraction of total dead load:

$F_s = \mu\ (DL)$

where:

F_s = Total thermal load on the pier

μ = Coefficient of friction between sliding parts in the bearing
Program defaults to a value of 0.06

DL = Total dead load on the pier

For rocker bearings, RC-PIER computes the thermal force which is a fraction of total dead load on the pier and also depends on the radius of the pin and the rocker.

$F_r = 0.25\ DL\ (r / R)$

where:

F_r = Total thermal load on the pier

DL = Total dead load on the pier

R = Radius of rocker

r = radius of rocker pin

In case of Elastomeric bearing pads, thermal movement will be resisted by the flexibility of the bearings and columns.

The total movement of the superstructure at the pier is,

$$\Delta_s = \Delta_b + \Delta_c$$

where:

Δ_s = Superstructure longitudinal movement at the top of the pier

Δ_b = Shear flexibility of elastomeric bearing pad

Δ_c = Bending flexibility of the pier column

The total deformation in superstructure based on change in temperature is,

$$\Delta_s = \alpha\ \Delta\ T\ L$$

where:

α = Coefficient of thermal expansion

Δ T = Change in temperature

L = Contributing length of superstructure

The movement of the column when a thermal force is applied is,

$$\Delta_c = T\ H^3 / (3\ E_c\ I_c)$$

where:

T = Thermal Force, K (N)

H = Distance from top of footing to top of cap, in (mm)

E_c = Modules of elasticity of concrete, ksi (MPa)

I_c = Inertia of pier columns in^4 (mm^4)

The movement of the column when a thermal force is applied is,

$$\Delta_b = T\ t / (n\ A\ G)$$

where:

T = Thermal Force, K (N)

t = total Elastomer thickness (without steel laminates), in (mm)

n = number of steel semi forced elastomeric bearing

A = bearing area of elastomeric bearing, in^2 (mm^2)

G = Shear modules of elastomer, ksi (MPa)

RC-PIER determines the thermal force on the pier using the above mentioned equations for movement of bearings and columns. Any contribution from footings/pile is ignored.

The point of zero movement, any hinges or construction joints should be considered while deciding about the contribution length for thermal movement.

For thermal generation, program either considers all bearings on a pier to be fixed or expansion. It considers the longitudinal movement of the bridge only which is the case for straight bridges or bridges with small skew. In case of large skew transverse flexibility may need to be considered which is not considered in the program. User should compute the loading manually for those or any other cases not covered and input directly. It does not check any unbalanced thermal force on the bridge. RC-PIER only generates the forces on the bearings and does not do any bearing design. User should check to see that bearing forces generated are not more than the capacity of the bearing.

Auto Earthquake Load (Seismic) Generation

To auto generate earthquake loads, RC-PIER uses the single mode spectral method. Bridge deck and columns are modeled as horizontal and vertical members respectively. All columns (vertical members) are assumed to be fixed at the base. At the abutments, the bridge is assumed to be hinged. You must define the cross-section area and moment of inertia of the superstructure. For columns, RC-PIER uses the total moment of inertia and area for all columns, as default values. However, you can manually modify these values. The pier height is the average height of all columns. Column modulus of elasticity is used for the whole structure.

In RC-PIER, the bridge longitudinal direction column member stiffness is computed as a cantilever member fixed at base ($3EI/L^3$). The transverse direction column stiffness is computed as a frame member ($12EI/L^3$). The program automatically computes self-weight of the superstructure, which is based on the first generated dead load case. If no such load case is present, then the program computes the dead load on the pier to the middle of the span on each side and load of cap and load of column to mid-height is added to it. This load is then divided by the average of two span lengths to arrive at the computed value of the superstructure unit weight. However, if the user changes this value manually, that value will be used for the generation of EQ loads.

An assumed uniform longitudinal load is applied to the bridge deck and static displacements, vs(x) are calculated, assuming only that the columns resist the load. Then, factors a, b, and g are computed. These factors are used to determine the period, T. The seismic response coefficient, Cs, is computed using the calculated period. Next, the intensity of longitudinal seismic loading is computed. The calculated force is evenly distributed among all piers (with no contribution to abutments). The pier force is then equally distributed among all the bearings for the pier under consideration.

For transverse loading, a similar approach is used as for the longitudinal direction. However, transverse displacements are computed with the bridge modeled as a three dimensional frame with line elements. Once the displacements are known, factors a, b, and g are computed. These factors are used to determine the period, T. Next, seismic response coefficient, Cs, is computed using the calculated period and intensity of parabolic transverse seismic loading. This loading is converted to concentrated loads applied at quarter points of each span and the transverse reaction on the pier is computed. This reaction is then distributed equally among all bearings for the pier.

By default, RC-PIER automatically generates four load cases using the above calculated loads on each bearing, where

- 100% force along Z+ with 30% force along X
- 100% force along Z- with 30% force along X
- 30% force along Z+ with 100% force along X
- 30% force along Z- with 100% force along X

Auto Vessel Collision Load Generation

RC-PIER will generate vessel collision, as specified in AASHTO LRFD, Art. 3.14. First, it calculates the vessel collision energy as follows:

$$KE = \frac{c_h w v^2}{29.2} \quad \text{(LRFD Eq. 3.14.7-1)}$$

where

KE= Vessel Collision Energy

w= Vessel displacement tonnage

c_h= Hydrodynamic mass coefficient

v= Vessel impact velocity

Second, RC-PIER calculates the barge bow damage length using the following equation:

$$a_B = 10.2\left(\sqrt{1 + \frac{KE}{5.672}} - 1\right) \quad \text{(LRFD Eq. 3.14.12-1)}$$

With different range of barge bow damage lengths, the barge collision force on pier is obtained as follows:

If $a_B < 0.34$	Then $P_B = 4112\ a_B$(LRFD Eq. 3.14.11-1)
If $a_B \geq 0.34$	Then $P_B = 1349 + 110\ a_B$(LRFD Eq. 3.14.11-2)

where P_B is the equivalent static barge impact force.

In RC-PIER, once the barge impact force is calculated, a pop-up message will appear showing the value of this force. Note that you need to manually apply the force on the columns.

Load Groups

RC-PIER provides all load groups as defined in AASHTO LFD Table 3.22.1A and AASHTO LRFD Table 3.4.1.1 and Table 3.4.1.2. You can use these load groups directly, or create your own load groups (e.g., you may need to create seismic load groups).

Each load in the load group is given as either unidirectional or reversible. A reversible load is where both positive and negative values are considered in the load combinations. A unidirection load only considers its positive value in the load combination. In AASHTO LFD, only W, WL, LF, and EQ are reversible loads. The rest of the loads are unidirectional. For AASHTO LRFD, BR, WS, WL, and EQ are reversible. Note that self-weight (SW) of structure is always associated with dead loads (DL). If you do not select a dead load, the self-weight will not be considered in the results of load combinations and envelopes.

Load Combinations

Load combinations are generated using the definitions of load groups specified in AASHTO LFD Art. 3.22 and LRFD Art. 3.4. RC-PIER uses two methods to generate the load combinations.

When program generates load combinations from load groups specified in library, it uses maximum, minimum factors, the dependency and the reversibility into account. Program creates separate combinations for all variations of maximum and minimum load factors which may result in more than one combinations. The reversible loads including generated wind loads when used in a combination generate two combinations. One with positive sign and the other with a negative sign. The actual load positions are not changed. This might be different than anticipated for wind cases with overturning load. If you would like wind overturning load treated differently, then you may add additional case to cover all scenarios.

The first method uses a comprehensive algorithm that provides all the possible combinations, to include so-called cross grouping. For example, assume a combination requires dead load, live load, and wind load. If you have one dead load (D1), two wind loads (W1, W2) and two wind loads on live load (WL1, WL2), then W1 will be grouped with WL1 and also with WL2. This results in comprehensive load combinations. To use this method, select the Use Cross Combinations check box on the *Analysis/Design Parameters* screen (see Fig. GO-36 on page GO-35).

The second method uses a dependency concept that distinguishes all loads into independent and dependent loads. Inclusion of an independent load in a particular load combination does not depend on the existence of any other load. Wind load is an example of an independent load. Inclusion of a dependent load in a particular load combination is predicted by the presence of a related independent load. For example, wind load on live load is a dependent load because it is dependent on the wind load. The following table lists the default dependent loads used in RC-PIER for AASHTO LFD and LRFD:

Default Dependent Loads	
LFD	**LRFD**
WL (depends on W)	WL (depends on WS)
CF (depends on L+In)	CE (depends on LL)
LF (depends on L+In)	BR (depends on LL)
	LS (depends on LL)
	CT (depends on LL)

User can view/modify the dependent load table on the A/D parameters screen. Please see page GO-39 on how to do this. This provides a great tool so that customized dependent/independent relationships can be specified.

The program checks to see that the same number of dependent load cases are provided as the load case on which it depends with the same indices. For example, if two wind load cases are provided and the user has selected to use the dependent load cases combinations option, the program allows either no wind on live cases or two wind on live cases. It no longer allows any other number of wind on live cases for the analysis. One further check is made to ensure that the load indices are the same. This means if the user has specified wind load on structure cases, W1 and W2 (in LFD), and wind on live load cases are needed, those must be WL1 and WL2. If the user has WL3 and WL3 along with W1 and W2, the program gives an error message and stops running.

Frame analysis results of the load combinations include the impact factor for the live load and load multipliers, while individual load cases provide results without either the impact factor or load multipliers.

Envelope Generation

For AASHTO LFD, RC-PIER generates two envelopes: service and strength. For LRFD, the program can generate a total of four envelopes and includes one of each of the following: strength, service, extreme event, and fatigue envelopes. For each envelope, RC-PIER selects the maximum and minimum values from all corresponding load groups. For example, the strength load group will include the maximum and minimum values among all strength load combinations.

Note RC-PIER does not consider the combinations for column design when generating the envelope, because the column combinations are only used for column design. In addition, column design uses the column combinations, not envelopes.

Analysis

Frame Analysis

RC-PIER uses a state-of-the-art three-dimensional frame analysis. The structure is modeled as straight lines using nodes and members.

Nodes and members are assigned physical properties to simulate the real structure. If there is a tapered member, the sizes of the start section and end section are averaged to obtain the average size for this member. The remaining properties, such as section area and moment of inertia for a tapered member, are computed based on the calculated average sizes.

For variable caps, the straight line passed through the mid-height of the smaller of the two ends and is parallel to the top of the cap surface. Analysis is done to the top of the footing excluding any footing depth.

Analysis is done to the top of the footing excluding footing thickness.

Truss Analysis

RC-PIER uses a two-dimensional truss analysis. To produce a stable truss structure, every truss member must be connected into triangles (STM only).

At least two supports must be provided. One is restrained in global X and Z direction. The second one, with a larger node number, is restrained only in global Y-direction. The truss analysis is only used to obtain the axial force in each member.

Strut-and-Tie Model

In AASHTO LRFD, you can use the strut-and-tie model (STM) as an alternative analysis tool. Note that in RC-PIER, the STM is only available for hammerhead piers and isolated pile caps. After the STM is constructed, the truss analysis is used. There are two options available to construct STM:

1. Manually input the X and Y coordinates for nodes and members.
2. Use the mouse to draw the model directly on the screen.

RC-PIER assumes that strut-and-tie locations will be the centroid of provided reinforcement.

Once the STM is constructed, these locations are well-defined; therefore, you cannot modify the distance between the edge of the cap to centroid of reinforcement (Bar Dist field). To get results for tension tie, you must use the auto design feature since it is assumed that the resistance of the concrete strut is adequate. You can manually add the reinforcement on the compressive side (concrete strut).

The required steel area for the tension is calculated as follows:

$$A_{s-req} = \frac{T}{\phi f_y}$$

where:

T= Tension in tie

ϕ= Resistance factor, taken as 0.9

In the case where the concrete strut cannot provide enough resistance, reinforcement on the compression side may be provided. The required compression steel is obtained as follows:

$$A_{s-req} = \frac{\frac{c}{\phi} - S_a A_{strut}}{f_y}$$

where:

c= Compression in the concrete strut

ϕ= Resistance factor, taken as 0.7

A_{strut}= Area of the strut, calculated as cap depth multiplied by strut width

S_a= Limiting stress

To calculate the allowable stress in the strut, refer to AASHTO LRFD Eq. 5.6.3.3.3-1 and Eq. 5.6.3.3.3.2.

When auto design is done, program computes required area of steel in horizontal and vertical ties. However it only computes the required area for vertical ties but does not design stirrups/spacing.

Factor of Reduced Moment of Inertia

There are many situations that would require the design of a cracked member. For example, if you design a column with its section 30% cracked, you will need to adjust the member section properties in order to simulate the cracking effect.

With this regard, RC-PIER allows you to reduce the moment of inertia of any members. When you define the cap and column geometry, specify the Factor of Reduced Moment of Inertia (MI). By default, this factor is one. However, for a cracked member, you can specify the value equal to or less than one. For instance, if you input 0.75 in the Factor of Reduced (MI) field, RC-PIER will use 75% of the moment of inertia of the original section without cracking in the frame analysis.

Service Load Analysis and Design (Working Stress)

RC-PIER can perform analysis and designs under service load (working stress design) according to AASHTO Standard. Service load design is only enabled if you exclusively select the service load group without any factored load group. The service load design provided by RC-PIER is in accordance with AASHTO LFD Art. 8.15.

When analysis is done for AASHTO Standard service groups only, program generates two sets of envelopes on analysis tab. Envelop N/O is the envelope which does not consider the overstress allowance. For all combinations in which overstress allowance is more than 100%, program divides the force/moment effects by the corresponding overstress factor and then compares and generates the envelope.

Design

Cap Design

For pier cap and strut designs, RC-PIER uses the envelopes for ultimate strength, service and fatigue, and the selected design code. For flexure design, the program considers the maximum positive and negative moments. The required steel area for each section is computed based on the maximum positive and negative moments. The effective steel area is obtained after adjusting the gross steel area taking into account development length. The calculated beam capacity ($\varnothing Mn$) is taken from the consideration of concrete material, and any available top and bottom bars.

Required steel as shown in the report is based on the actual moment and the minimum steel criteria for the appropriate code. In AASHTO Standard, this is based on Art. 8.17.1, which states that the minimum section capacity should be the smaller of 1.2 Mcr and 1.33 Mu. According to LRFD Art. 5.7.3.2, it should be 1.2 Mcr and 1.33 Mu. Further, the temperature and shrinkage criteria of Art. 5.10.8 should also be considered. RC-PIER computes temperature and shrinkage steel as per LRFD Art. 5.10.8.2 even when sections are deeper than 48 inches. The program first checks if the section is adequate for the applied moment from the critical combination. It then checks if the design moment is at least equal to the minimum moment. This has to be at least equal to the smaller of 1.33 Mu or 1.2 Mcr. If not, at least that capacity must be developed. If needed, revised steel is calculated. Then the program checks for temperate and shrinkage steel. As per that criteria, shrinkage steel is equal to 0.11*Ag/Fy. However, the code specifies to distribute this in two faces.The program internally checks separately for top and bottom steel. It checks if the top and bottom steel is at least equal to 0.5(0.11 Ag/Fy) individually. In certain cases, it is possible that there is no moment in a section. In such a case, zero moment might be reported for combination number 0. However, temperature and shrinkage steel requirements have to be checked. Therefore, the program in LRFD mode will show required steel based on temperature and shrinkage.

Using the auto design feature, all sections are considered. The steel requirement at each section is determined first by taking the section as single reinforced. However, if the section cannot be design as a single reinforced section, RC-PIER computes both tension and compression reinforcements requirements, resulting in a double reinforced design.

Geometrically, the cap beam is divided into spans that are considered to extend from each column centerline to next column centerline with supports at the column centerline. Cantilevers, if present, are considered separate spans. The reinforcement provided over each support is selected when it is at least equal to the required reinforcement in 1/4 of the span on each side of the support. The bars are extended beyond the 1/4 points in accordance with the development length, so the reinforcement is fully developed at 1/4 point. For moment in the middle section of the beam, the provided reinforcement is calculated as the largest requirement for the middle half section of the beam (from 1/4 to 3/4 in each span). The bars are then extended outwards to a length equal to the development length. If the span length is less than 11 ft (3.4 m), RC-PIER provides continuous reinforcing bars and does not compute separate detailing for reinforcement over the column and midspan.

The calculated area of steel is transformed into required rebar quantity, using the selected rebar sizes. When calculating the number of reinforcing bars, clear spacing of at least 1.5 times the bar diameter or 1.5 inch and specified side clear cover is maintained. If you are using the auto design feature, up to three layers of reinforcement for top and bottom can be created. If a section requires more than three layers, you must manually input the reinforcement, as follows.

To manually input a design (design check), input the appropriate reinforcing pattern and RC-PIER will compute the required reinforcement for all sections and the section capacity with provided reinforcement. RC-PIER uses the development length to compute the effective steel at each section at top and bottom and then calculates the capacity based on both top and bottom effective reinforcement.

Note that AASHTO LFD Art. 8.24.2 and 8.24.3 are not implemented in RC-PIER.

Program can do the design of cap either for cap moments at centerline of columns or at face of support. For round columns, RC-PIER converts the round column section to the equivalent square section. The face of the equivalent square will become the critical section. For rectangular chamfered, rectangular filleted, hexagonal, and octagonal columns RC-PIER determines the size of an equivalent rectangular section maintaining the overall section aspect ratio. When the face of support option is chosen, program ignores moments at all the check points between faces of supports including the centerline of the column values. Such designs mostly result in fewer number of bars at supports.

Shear and Torsion

Shear and torsion design or check is performed at each section for maximum shear force along the maximum torsion. This is calculated at the immediate left and right side of every section except at the beginning and end of span. Only the section to the right of the beginning of span is evaluated; only the section to the left of the end of span is evaluated.

In Auto Design, program tries to come up with stirrup spacing that satisfies the required transverse reinforcement Av/s (total transverse reinforcement area per unit length). In addition to Av/s, program also considers the maximum spacing allowed by the selected design code.

In AASHTO LRFD, there are two methods available, as specified in LRFD Art. 5.8.3: simplified and general. The simplified method uses β and θ, as specified in LRFD Art. 5.8.3.4., namely θ = 45 and β = 2.0. The general method computes β and θ, as specified in LRFD 5.8.3.4.2.

For shear and torsion calculations, the effective shear depth, d_v, is used, as specified in LRFD Art. 5.8.3.4. The value d_v is considered to be the maximum value obtained from the following three equations:

$$d_v = d_e - a/2$$

$$d_v = 0.72h$$

$$d_v = 0.9d_e$$

where:

d_e= distance from extreme compression fiber to center of tensile reinforcement

h= height of beam

a= depth of concrete rectangular fiber computed as follows:

$$= \frac{A_s f_y}{0.85 f_c' b}$$

where:

A_s= area of tension reinforcement

f_y= yield strength of reinforcement

f'_c= compression strength of concrete

b= width of member

Note that the effective steel area at each section is used to compute a.

For AASHTO LFD, design of shear is carried out according to AASHTO specifications. However, if torsion is present in the section along with shear, torsion design is based on ACI 318-95 specifications (as suggested by AASHTO LFD).

In AASHTO Standard, when the torsion is less than the limiting torsion, the maximum spacing is based on Art. 8.19.3. However, when torsion is more than the limiting value, program also checks the spacing as per ACI 11.6. In AASHTO LRFD, the maximum spacing is based on Art. 5.8.2.7.

Beam properties for torsion (A_{oh} and p_h) consider the stirrup size, specified or chosen for Auto Design. For the calculation of A_{oh}, the program uses the rebar clear cover for sides to be equal to the value specified on A/D parameters dialog for cap side cover. Computed Avs/s is the required steel for shear only, 2Ats/s is the required steel for two legs of torsion, and Av/s is the total computed steel due to shear and torsion at that section. When torsion is present at a section, the program also calculates longitudinal steel. For AASHTO Standard, the program reports the total longitudinal steel required for torsion (Alt). Users may reduce this amount by provided steel in excess of required steel already present at that section. For AASHTO LRFD, the program reports the additional required effective steel (Alx). This already accounts for excess effective steel present at that section, which is more than the required flexural steel.

Distribution of Flexural Reinforcement Check (Cracking Check)

Cracking check is performed only if at least one service load group is selected along with the load factor or strength load group,

Cracking Check

In AASHTO LFD, the distribution of the reinforcement is based on Art 8.16.4. For cracking check, program use the service moments without the consideration of overstress factor.

In AASHTO LRFD, program provides two options. User can choose to do the cracking check as per AASHTO LRFD 3rd Edition or as per AASHTO LRFD Interims of 2005 for 3rd Edition. The two approaches are significantly different. When design check is done as per AASHTO LRFD 3rd Edition, program follows the approach specified in Article 5.7.3.4. In this approach the stress in the provided rebar is computed based on service moments and then compared to stresses allowed at the service stage. Program checks the ratio of these two stresses. As long as the actual stress is less than allowed, design is deemed okay. If the stress exceeds the allowed limit, the results are flagged. When the option is selected to do the check as per the LRFD Interims of 2005 for 3rd Edition, the check is to determine the spacing of reinforcement which should not be exceeded by the provided reinforcement. For the provided spacing, program evaluates the provided number of bars at every section. It then computes the spacing considering the side cover specified in the A/D parameters on the Analysis tab. Then it compares to check if the provided spacing is adequate or not. If the provided spacing is less than the required spacing, it is considered adequate. In case the spacing is not adequate, program flags the results for inadequacy.

Fatigue Check

In LRFD, fatigue is only checked if the fatigue load groups is selected and is carried out as specified in LRFD Art. 5.5.3. For AASHTO LFD, it is only checked if at least one combination for service load is selected and performed in accordance with Art. 8.16.8.3, with one modification to f_{min}.

For LRFD, RC-PIER checks the stress range in the reinforcement for fatigue load combinations. If all load combinations produce tension in rebar, the stress range is equal to the largest stress value. However, if some combinations produce compression and others produce tension in the reinforcement, the stress range is obtained by adding the two absolute stresses together. The minimum stress level, f_{min}, is computed due to larger of the permanent loads and the permanent loads plus creep and shrinkage accompanied with minimum live load.

The calculation of stress range and f_{min} are the same for AASHTO LFD.

Column Design

RC-PIER uses all combinations generated during the analysis for column design. These combinations also include the values of β_D factor specified for the column design, in accordance with the footnote of AASHTO LFD Table 3.22.1.A. Three methods may be used for the column design:

1. Conventional Method without considering slenderness effect
2. Moment Magnification Method
3. P-Δ Analysis Method

Note if you select the Moment Magnification method (by selecting the Consider MM check box on the *Column* screen), you must further select if the column is braced or unbraced.

Moment Magnification Method

Moment magnification is an approximate evaluation of slenderness of a column. RC-PIER considers two types of slender columns: Unbraced and Braced. Note that RC-PIER does not determine the type of column slenderness (braced or unbraced) automatically, you must specify the type according to the circumstance of your project and relevant specifications.

For unbraced columns, RC-PIER will calculate the effective length factor, k, as follows:

$$\frac{G_a G_b \left(\frac{\pi}{k}\right)^2 - 36}{6(G_a + G_b)} = \frac{\frac{\pi}{k}}{\tan\frac{\pi}{k}} \quad \text{(LRFD Eq. C4.6.2.5-1)}$$

where subscripts “a” and “b” refer to the two ends of the column and “k” is the effective length factor for the column under consideration. The calculation of G is:

$$G = \frac{\Sigma\left(\frac{I_c}{L_c}\right)}{\Sigma\left(\frac{I_g}{L_g}\right)}$$

where:

Σ= summation of properties of components rigidly connected to an end of the column in the plane of flexure

I_c= Moment of inertia of column

L_c= Unbraced length of column

I_g= Moment of inertia of beam or other restraining member

L_g= Unsupported length of beam or other restraining member

The value of G for the end of column connecting to a footing depends on the degree of fixity in the foundation. The values (G_a) are as specified per LRFD Figure C4.6.2.5-1.

G_a	**Foundation Type**
1.5	Footing anchored on rock
3.0	Footing not anchored on rock
5.0	Footing on soil
1.0	Footing on multiple rows of end of bearing piles

When considering the restraining effect from cap beam, RC-PIER does not take into account the cantilever contribution. Note that the procedure mentioned above for calculating the effective length factor, k, is implemented in RC-PIER for both design codes, AASHTO Standard and LRFD.

To evaluate approximately the slenderness effect of the column, the factored moment is magnified as:

$$M_c = \delta_b M_{2b} + \delta_s M_{2s}$$

where δ_b and δ_s are the moment magnification factors, which are defined as the following:

$$\delta_b = \frac{C_m}{1 - \frac{P_u}{\phi P_c}} \geq 1.0$$

$$\delta_s = \frac{1}{1 - \frac{\Sigma P_u}{\phi \Sigma P_c}} \geq 1.0$$

where:

P_u = factored axial load

P_c = nominal axial load strength

$$P_c = \frac{\pi^2 EI}{(kl_u)^2}$$

where:

k = effective length factor

l_u = column height

EI = flexural stiffness of a compressive member

The calculation of M_{2b} and M_{2s} in RC-PIER is different from the AASHTO Specifications requirements.RC-PIER only considers the total moment (e.g., moments M_x and M_Z for bottom of the column) and does not distinguish the moments due to the gravity or lateral loads, as specified by AASHTO Specifications.

With this assumption, for braced columns, RC-PIER only calculates the moment magnification factor δ_b and multiples this factor with the total moment to obtain the magnified factored moment. For unbraced columns, it calculates δ_s and multiplies this factor with the total moment to obtain the magnified factored moment. In RC-PIER, EI is calculated by the following equation:

$$EI = \frac{\frac{E_c I_g}{2.5}}{1 + \beta_d}$$

where:

E_c= Concrete elastic modulus

I_g= Moment of inertia of gross concrete section

β_d= Ratio of maximum dead load moment to maximum total load moment

Note that the dead load moment includes the moment due to self weight. If the moment is zero or less than the value due to minimum eccentricity when calculating β_d, RC-PIER will multiply the minimum eccentricity (0.6 + 0.03h) with the axial force to obtain the moment.

According to AASHTO LFD and LRFD, C_m is calculated for members braced against sidesway as follows:

$$C_m = 0.6 + 0.4\left(\frac{M_{1b}}{M_{2b}}\right) \geq 0.4 \text{ (single curvature)}$$

$$C_m = 0.6 - 0.4\left(\frac{M_{1b}}{M_{2b}}\right) \geq 0.4 \text{ (double curvature)}$$

RC-PIER uses the smaller total end moment for M_{1b} and larger total end moment for M_{2b}. For a braced column with any transverse loads acting on the column, Cm = 1. In addition, for unbraced columns, C_m = 1. Same as the calculation of β_d, if the moment is zero or less than the value due to minimum eccentricity, RC-PIER uses the minimum eccentricity to obtain the moment.

Single curvature is due to the different sign of M_{1b} and M_{2b}, and double curvature results are from the same sign of M_{1b} and M_{2b}. If both M_{1b} and M_{2b} are zero, RC-PIER will use the single curvature equation and moments due to the minimum eccentricity to calculate C_m.

RC-PIER also takes into consideration a column with a drilled shaft or checking point. The connection point between the drilled shaft and column is treated the same as the column with a checking point, with respect to the moment magnification. The moment magnification factor at the checking point is interpolated from the moment magnification factors at the top and bottom of the column, as illustrated in Figure TH-24 on page TH-40.

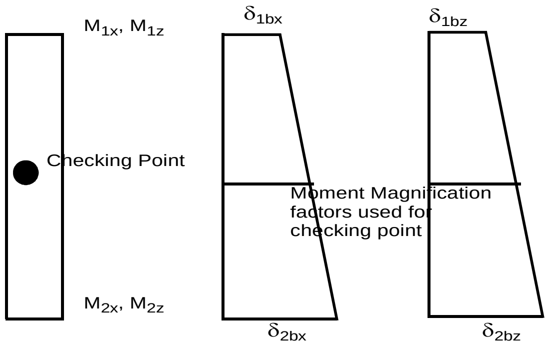


Figure TH-24 Moment Magnification Calculations for Column with Checking Point

If there is an intermediate strut, RC-PIER assumes the moment about global Z-direction is restrained due to the strut. As a result, RC-PIER calculates the moment magnification factor for the columns above the strut and below the strut separately for the moment with respect to Z-direction. The moment magnification factor in X-direction at the connection between the strut and column is interpolated from the moment magnification factors at the top and bottom of the column. This is illustrated in Figure TH-25 on page TH-40 where (a) Pier with an intermediate strut, (b) column is separated when calculates moment magnification for the moment in Z-direction at the connection between the column and strut, and (c) moment magnification factor in X-direction is interpolated at the connection between the column and strut.

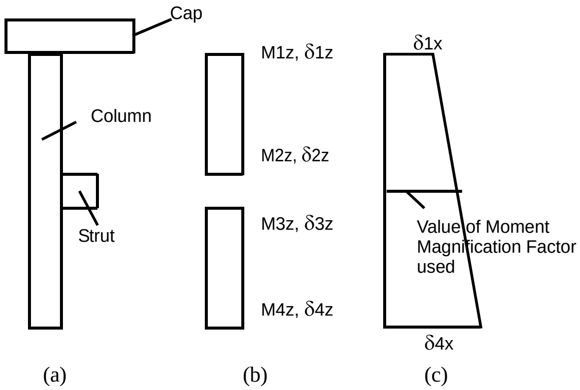


Figure TH-25 Moment Magnification Calculations for Column

P-Δ Analysis Method

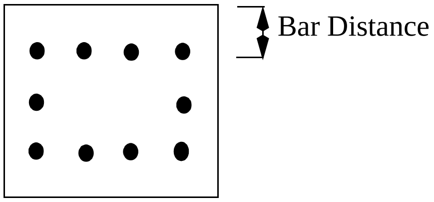
The P-Δ Analysis is performed by loading the modified frame with each of the column combinations. The modification of the frame is done by finding the disturbed frame model under the load. To do this, frame nodes are displaced by the value of the displacement from the previous load and loaded again with the load from the same load combination. This process is performed until the number of iterations reaches 5 or the difference of the displacement at the top of the column (column bottom is fixed) between the previous and present iterations is within 5%. Several implementation considerations are as follows:

1. The moments amplified by P-Δ are used only for column designs in the current version of RC-PIER. They are not applied to the cap and footing designs.
2. The impact factor of live load is applied to the final results.
3. If the cracking effect must be taken into account in column design with P-Δ analysis, you must specify the reduced moment of inertia for the column. RC-PIER does not automatically consider the cracking effect.

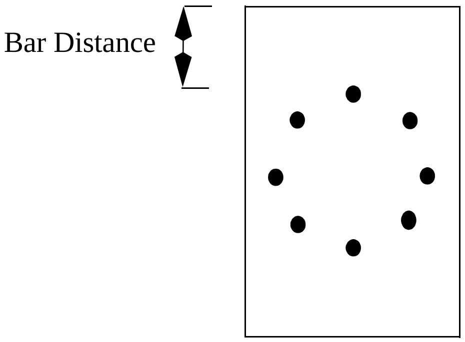
Column Rebar Patterns

RC-PIER allows four patterns for rebar in column cross-section. These are:

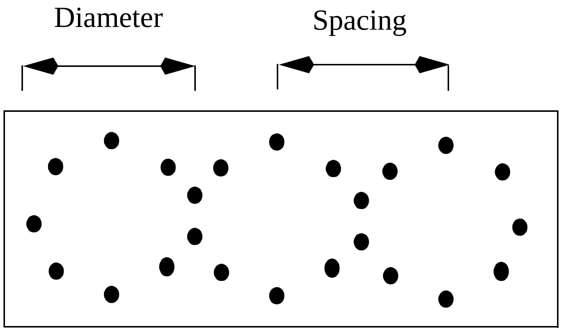
1. Rectangular Pattern
2. Circular Pattern
3. Intersecting Hoops Pattern
4. General Pattern



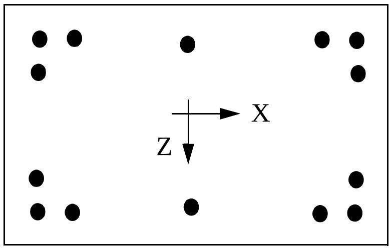
Rectangular Column with
Rectangular Rebar Pattern



Rectangular Column with
Circular Rebar Pattern



Rectangular Column with Intersecting Hoops
Rebar Pattern



Rectangular Column with
General Rebar Pattern

Figure TH-26 Rebar Patterns

For rectangular pattern, the bar distance is measured from the column overall dimensions (not considering the chamfers). For circular pattern specified bar distance is measured from the smaller of the two dimensions (width or depth).

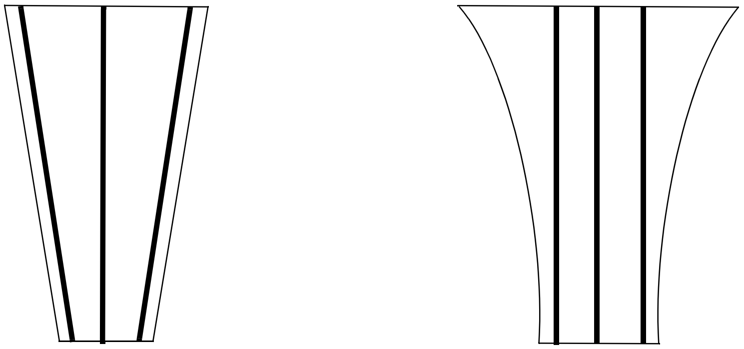
If intersecting hoops is selected as the rebar pattern, you must specify more than one hoop. The spacing between hoops should be greater than zero, but should be less than the hoop diameter. Number of bars must be an even number greater than 2x(NH-1) where NH is the number of hoops. RC-PIER first determines the radius of the circle where it will place the rebars by subtracting the hoop bar diameter and half the main bar diameter. Then it determines the location of bars at the intersection locations of the hoops. It then determines the number of rebar that could be placed in the interior and exterior loops at somewhat similar spacing. It then determines the location of each bar in X and Z coordinates with respect to the CG of the column intersection.

When general pattern is selected, specify the location of bars in terms of X and Z coordinates. Note that RC-PIER designs columns which are doubly symmetrical. Therefore the rebar pattern has to be doubly symmetrical too. That means if a bar is defined at location (X, Z) there should be other bars at (X, -Z), (-X, Z) and (-X, -Z) locations.

When auto design is done, RC-PIER generates a pattern for rectangular, rectangular chamfered and hex columns. It generates circular pattern for circular or octagonal columns.

Column Rebar Orientation

For sections which are tapered with linear or parabolic variation, rebar could either be placed vertically or parallel to the column faces with constant cover. When rectangular or circular rebar pattern is used, this rebar pattern is termed a “Face Parallel.” When general pattern or intersecting hoops is selected bars can only be vertical. When bars are oriented vertically, those should be placed such that at all heights, all bars lie within the section.



Column Variation Linear
Rebar Orientation Face Parallel

Column Variation Parabolic
Rebar Orientation Vertical

Figure TH-27 Rebar Orientation

Capacity Calculation

Only the axial force and moments in the global X and Z-direction are used for the column design. A moment interaction diagram is constructed for each combination. The capacity of a column is determined by the method described on page B-17. The critical combination is chosen from the computed minimum ratio of ($\phi M_n/M_u$) of all combinations. If a column is designed using the service load method, AASHTO LFD Art. 8.15.4 will be followed. When a column is designed for a hammerhead pier, the column section for the top of the column is the section at the bottom of the cap part.

Footing Design

Depending on the footing type selected, RC-PIER uses either the calculated maximum bearing pressure or maximum calculated pile reaction for the footing design. For flexure, one or two-way shear designs, RC-PIER allows the eccentricity between the footing centerline and column centerline. However, the eccentricity must be given in the global X-direction. For spread footing, uniform maximum pressure is applied at the bottom surface of the footing. Negative bearing pressure is not considered for spread footing design. For pile cap footing, the maximum pile reaction is applied at each pile location. Negative pile force is not considered for pile cap footing design.

The effect of self-weight and surcharge is not considered for the footing design, namely flexure and shear (one and two-way) designs. This means that the bearing pressure or pile reaction used in the footing design is without the contribution from the self-weight and surcharge.

When calculating either bearing pressure or pile reactions, the forces and moments at bottom of the column (top of footing) are used.

Figure TH-28 illustrates the auto design layout of footing reinforcement where Bar Dist (X-direction) is the distance from bottom of footing to centerline of reinforcement in X-direction (i.e., concrete clear cover + 1/2 reinforcement diameter) and Bar Dist (Z-direction) is the distance from bottom of footing in Z-direction (i.e., clear clover + diameter of X reinforcement + 1/2 reinforcement diameter in Z-direction).

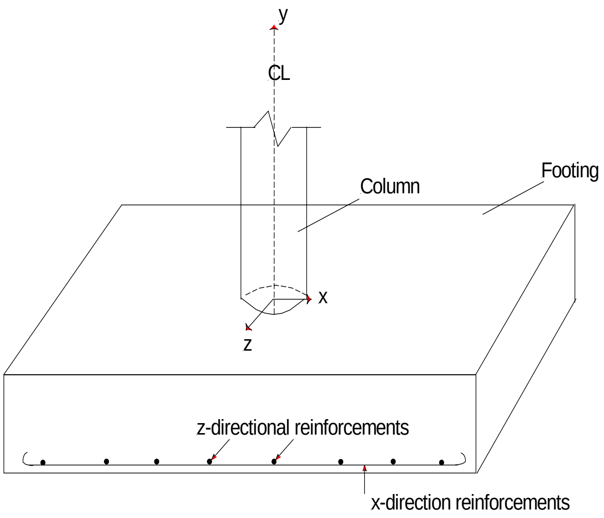


Figure TH-28 Schematic Diagram Showing Auto Design Layout of Footing Reinforcement

Bearing Pressure Calculation for Footing

For isolated spread footings, forces and moments obtained from each load combination, at the top of the footing, are applied at the centroid of the footing. The footing is treated as a member with infinite stiffness. The bearing pressure at each corner is calculated, as follows:

$$Q = \frac{P}{A} \pm \frac{(M_{xx})(z)}{I_{xx}} \pm \frac{(M_{zz})(x)}{Izz}$$

where:

P = Axial force acting on the centroid of the footing

M_{xx}, M_{zz} = Moment acting on centroid of footing about global X and Z-axes

I_{xx}, I_{zz} = Moment of inertia with respect to global X and Z-axes

x, z = Distance from the neutral axis of footing, along global X, Z axes

The self-weight of the footing and surcharge are added to the bearing pressure under service and factored load combinations, in addition to the pressure due to column forces and moments.

This same procedure is used to calculate the bearing pressure for combined spread and strap footings. For multi-column piers, the result of forces and moments, with respect to the centroid of the footing, is computed and the above equation is used to obtain the bearing pressure.

The pile cap is also assumed to be infinitely rigid. Forces and moments from the columns at the top of the footing are transferred to the centroid of the pile group. In the determination of pile reactions, the program first determines whether the piles are conlinear or not. If all piles are in a line and there is a moment acting perpendicular to the line of the piles, the program issues a warning message for unbalanced moment. For a general case where all piles are not in line, the program uses the following equation to determine the pile force.

$$R = \frac{P}{n} + \frac{\left(M_{zz}I_{xx} + M_{xx}I_{xz}\right)}{\left(I_{xx}I_{zz} - I_{xz}I_{xz}\right)}\left(x - x_{cg}\right) - \frac{\left(M_{xx}I_{zz} + M_{zz}I_{xz}\right)}{\left(I_{xx}I_{zz} - I_{xz}I_{xz}\right)}\left(y - y_{cg}\right)$$

where:

n = number of piles

I_{zz} = ΣX^2= summation of distances of piles in X-direction

I_{xx} = ΣZ^2= summation of distances of piles in Z-direction

I_{xz} = ΣXZ= summation of product of x and z coordinates of piles

The self-weight of the footing and surcharge are added to the pile reaction under service and factored load combinations, in addition to those due to column forces and moments.

Isolated Footing Design

Isolated footings are assumed to behave like beam cantilevers. The moment on the critical section of the footing is determined by crossing a vertical plane through the footing and computing the moment due to the bearing pressure or the pile reaction acting on one side of that vertical plane. For rectangular columns, the critical section for flexure is taken at the face of the column. For round columns, RC-PIER converts the round column section to the equivalent square section. The face of the equivalent square will become the critical section. For rectangular chamfered, rectangular filleted, hexagonal, and octagonal columns RC-PIER determines the size of an equivalent rectangular reaction maintaining the overall section aspect ratio.

Required steel as shown in the report is based on the actual moment and the minimum steel criteria for the appropriate code. In AASHTO Standard, this is based on Art. 8.17.1, which states that the minimum section capacity should be the smaller of 1.2 Mcr and 1.33 Mu. According to LRFD Art. 5.7.3.2, it should be 1.2 Mcr and 1.33 Mu. Further, the temperature and shrinkage criteria of Art. 5.10.8 should also be considered. RC-PIER computes temperature and shrinkage steel as per LRFD Art. 5.10.8.2 even when sections are deeper than 48 inches. The program first checks if the section is adequate for the computed moment at the face of columns. It then checks if the design moment is at least equal to the minimum moment. This has to be at least equal to the smaller of 1.33 Mu or 1.2 Mcr. If not, at least that capacity must be developed. If needed, revised steel is calculated. Then the program checks for temperate and shrinkage steel. As per that criteria, shrinkage steel is equal to 0.11*Ag/Fy. However, the code specifies to distribute this in two faces. For footing, total steel on all face need not be more than 0.0015 Ag. The program internally checks separately for top and bottom steel. It checks if the top and bottom steel is at least equal to 0.5(0.11 Ag/Fy) individually. For footings, it limits the reinforcement on each face equal to 1/2 (0.0015) Ag criteria is met. In certain cases, it is possible that there is no moment in a section. In such a case, zero moment might be reported for combination number 0. However, temperature and shrinkage steel requirements have to be checked. Therefore, the program in LRFD mode will show required steel based on temperature and shrinkage.

In the case of negative pile reaction in a pile, top reinforcement may be needed, which is not computed by the program at this time.

For one-way shear calculations, the critical section is located at the distance, d_v (for LRFD), or d (for LFD), both measured from the face of the column or equivalent square of the round column, as shown in Figure TH-29 (X-direction only). For shear capacity of the footing in LRFD mode, the program allows two approaches. By default, the program uses a simplified method as per Art. 5.8.3.4.1. However, the program also allows the use of the general method as per Art. 5.8.3.4.2. When using the general method, the program does two iterations to arrive at θ (theta) and β (beta) values to be used in shear capacity calculations.

If a pile perimeter is completely outside the critical section (pile A in Figure TH-29) then the full pile reaction is considered in the subsequent pile cap design. However, no pile reaction is taken into account in the case where the pile perimeter is inside the critical section (pile B in Figure TH-29). For any pile location between Location A and B (Figure TH-29) a linear interpolation of the pile reaction is performed to calculate the pile reaction.

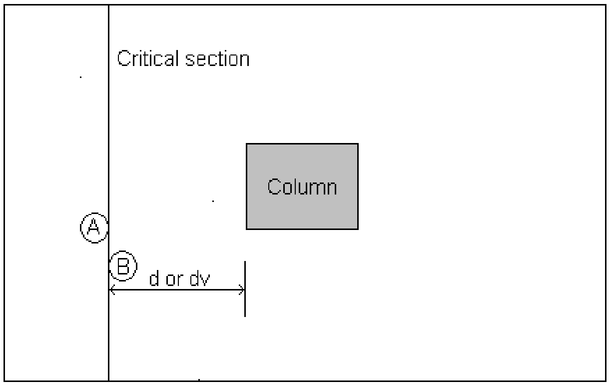


Figure TH-29 Determination of Critical Section of One-Way Shear for a Pile Cap (Equivalent square is used for a round pile)

For two-way shear designs, the critical perimeter is determined using the average d_v or d. The average d_v or d are calculated using the d_v or d in the global X or Z-direction. For a round column or pile, the circular profiles are used to determine the perimeter. If two round columns are close enough then their perimeters will merge and become one perimeter, as shown in Figure TH-30.

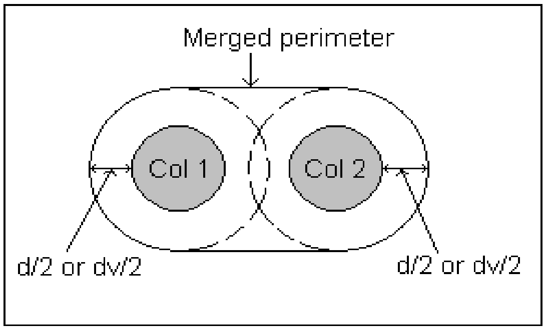


Figure TH-30 Situation Where Two Perimeters are Merged

Two-way shear is determined using the forces due to either the bearing pressure or pile reaction acting outside the critical perimeter.

When calculating the two-way shear for a pile and the pile is close to the edge of the pile cap, the perimeter might be cut by the edge. In this case, the two-way shear perimeter is taken by extending a tangent from the arc to the edge, as illustrated in Figure TH-31.

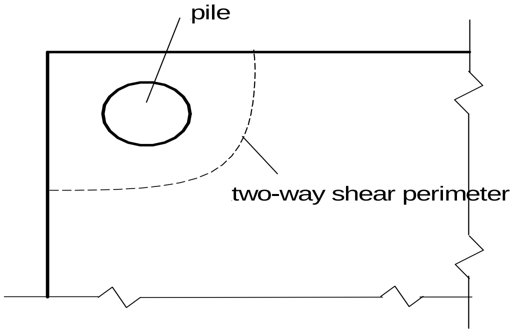


Figure TH-31 Situation Where Pile is Close to the Edge of Pile Cap

Strap Footing Design

A strap footing consists of two heads and one strap connecting the two heads together. The heads are designed as isolated footings. The maximum bearing pressure is calculated for each head, and applied as a uniform pressure on the bottom surface. The bottoms of the columns are assumed fixed. Note that each head may have different maximum bearing pressures. Figure TH-32 illustrates the uniform bearing pressure applied at the strap footing.

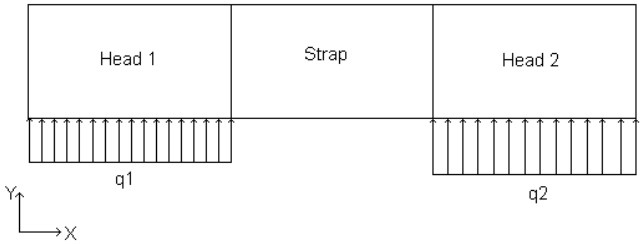


Figure TH-32 Analysis Model Used by RC-PIER for the Design of Heads in Strap Footing

The strap is designed as a beam. It is assumed that no bearing pressure and self-weight act on the strap. The moment and shear are from the bearing pressures acting on the heads adjacent to the strap. To calculate the moment and shear in the strap, RC-PIER assumes a trapezoidal distribution of the bearing pressure on the bottom of the heads. This distribution creates an eccentricity between the resultant of the trapezoidal bearing pressure and the centroid of the head. In turn, this eccentricity produces two moments at the left and right end of the strap. Only the larger of the two moments is used in the flexure design of the strap. Figure TH-33 shows the model used by RC-PIER for a strap design.

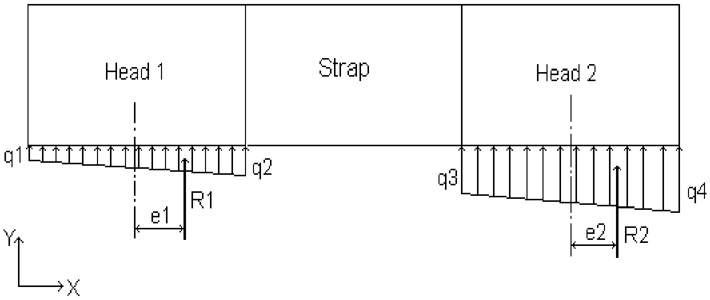


Figure TH-33 Analysis Model used by RC-PIER for Design of Strap Footing

The bearing pressure applied on each side of the head is averaged from the two corners on that side. For example, as shown in Figure TH-33, bearing pressure q_1 is the average value of two corners on the left side of head 1.

The shear in the strap is calculated using the largest of the two results, R_1 and R_2. The strap footing is considered to act as a cantilever in the global Z-direction.

Combined Footing Design

RC-PIER considers combined footings to act as a continuous beam in the global X-direction. For combined spread footings, a uniform bearing pressure taken as the maximum bearing pressure from four corners, is applied on the bottom of the footing. For a combined pile cap, the concentrated loads taken as the maximum pile reaction are applied at each pile location. The location of the columns is considered fixed. The moment and shear are calculated at the critical sections, using the model described above.

The combined footing is assumed to act as a cantilever in global Z-direction. The same bearing pressure or concentrated forces as those calculated in the X-direction are used, and all other calculations are computed in the same way.

Figure TH-34 shows a structural model used by RC-PIER for the design of a combined footing in the global X-direction. Note that a uniform bearing pressure is applied.

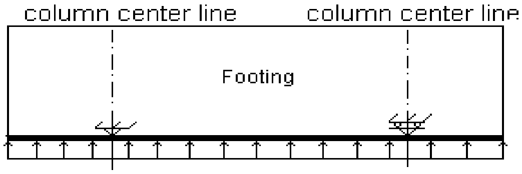


Figure TH-34 Structural Model used by RC-PIER for the Design of Combined Footing in Global X-Direction

A structural model for the design of a combined footing in the global Z-direction is shown in Figure TH-35.

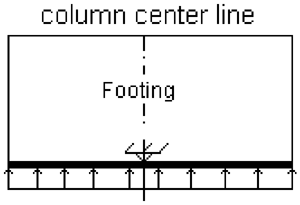


Figure TH-35 Structural Model used by RC-PIER for the Design of Combined Footing in Global Z-Direction

Overturning Check

Overturning can be checked for single column piers as well as for multiple column piers. Program can check and report overturning and stabilizing moments about X axis as well as about Z axis of the pier. In order to check the overturning, user must have completed the analysis, specified and designed one isolated spread footing below each column. If there are more than one footings under a column, program will not check overturning. It will check only when each column has only one footing. Overturning check can be done for service loads or for factored (strength/extreme event) load combinations. User can choose to check overturning for all combinations (service/factored) or for only those combinations which include wind or earthquake loads. Additionally user may ignore the effect of live loads from the combinations. The computations are based on applied loads on the structure and their moment arms. Program at this point does not use any analysis results. It only uses the generated combinations however. Even though the overturning is checked with reference to the bottom edge of the footing, program does not allow any lateral loads to be applied directly on the footing. The self weight of the structure (including footing weight) as well as the surcharge is considered in calculations.

Depending upon the pier geometry, applied loads, program computes the effect of all applied loads at the bottom of the exterior footing at extreme edges. If a load tends to overturn the pier, it will be added to the total overturning moment. However, if a load tends to have stabilizing effect, program will add its effect to total stabilizing moment. This is done for two cases in which overturning is checked about outer edges of the exterior footings. In case of single column pier those would be the two edges of the same footing. For the loads shown in Figure TH-36, the gravity load at bearing 1 tends to overturn the pier with respect to left edge of footing. This moment due to this load will be added to the overall overturning moment. All other gravity loads tend to stabilize the pier about left edge. Moment due to those will be considered in computing the total stabilizing moment. All the lateral loads in this figure turn to overturn the structure about footing left edge and will be considered in computation of overturning moment at this point. Similarly, program checks for overturning and stabilizing moments about X axis. In the report program also reports the total vertical load for the controlling combination as well as the total lateral load along with stabilizing and overturning moments and their ratio. For all the cases in which stabilizing moment is less than overturning, the computed ratio falls below 1.0. Program reports a flag for this case. However, in practice additional factors of safety are considered that need to be checked by the user for structure stability.

.

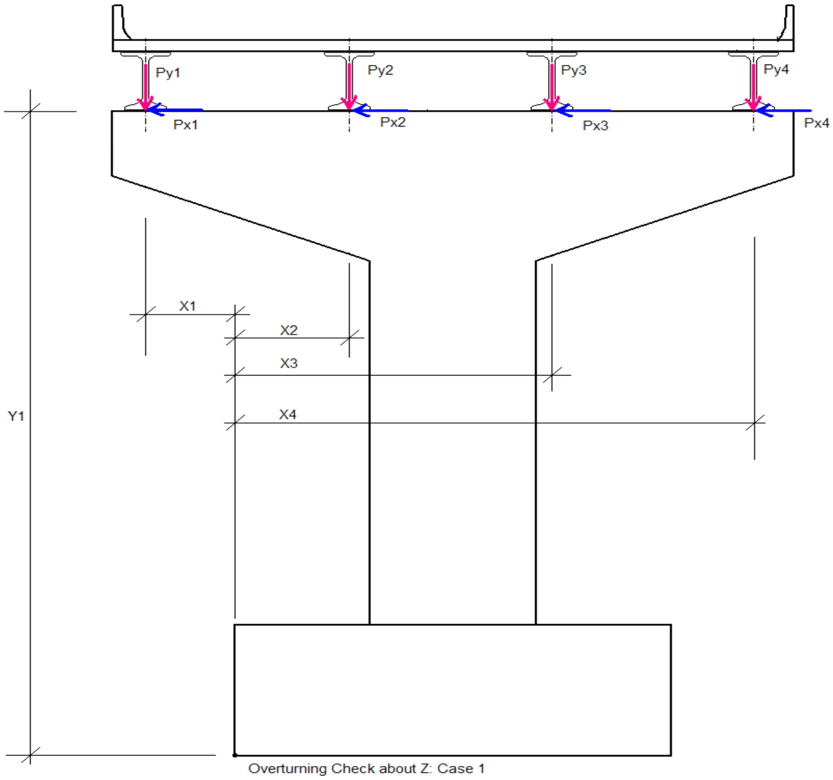


Figure TH-36 Overturning Check About Z

```
PROJECT DATA
============

Project     : Tutorial 3
User Job No.: Tutor3
State       : FL                             State Job No. : FL-003
Comments    : Hammer Head Pier
              Designed using Stut and Tie Method



PIER GEOMETRY
=============

Pier Type: Hammer Head

   Length(X) = 32.00 ft   Height max(Y) = 8.00 ft   Height min(Y) = 3.00 ft
   Bottom length(X) = 8.00 ft   Depth(Z) = 4.00 ft   Skew angle = 0.00    Reduction of I =
1.000

   Column Shape: Rectangular Non Tapered
   Bottom width(X) = 8.00 ft  Top width(X) = 8.00 ft  Depth(Z) = 4.00 ft  Height(Y) = 12.00 ft


STRUCTURE MODEL

================

FRAME Model:
                Member   Node   Hinge Check Pt Dist(ft) Memb length(ft)
                -------------------------------------------------------
   Column No. 1
                  1        1       -             0.00
                           2       -            18.50      18.50
   Cap
                  2        3       -             0.00
                           4       -             2.50       2.50
                  3        4       -             2.50
                           5       -             9.25       6.75
                  4        5       -             9.25
                           6       -            12.00       2.75
                  5        6       -            12.00
                           2       -            16.00       4.00
                  6        2       -            16.00
                           7       -            20.00       4.00
                  7        7       -            20.00
                           8       -            22.75       2.75
                  8        8       -            22.75
                           9       -            29.50       6.75
                  9        9       -            29.50
                          10       -            32.00       2.50

   Node coordinates:
      Number         X(ft)        Y(ft)          Node type
      ---------------------------------------------------------
         1          16.00         0.00        fixed at ground
         2          16.00        18.50     column-cap bearing
         3           0.00        18.50
         4           2.50        18.50                bearing
```

```
         5            9.25        18.50                bearing
         6           12.00        18.50
         7           20.00        18.50
         8           22.75        18.50                bearing
         9           29.50        18.50                bearing
        10           32.00        18.50
Strut and Tie Model Configuration for Hammer Head Cap:


   Strut Width =    0.40 ft

   Node Coordinates:

      Number      X ft       Z ft
      ----------------------------
         1       18.00       1.17   support
         2       22.75       2.30
         3       26.13       3.50
         4       29.50       4.70
         5       31.25       5.30
         6       31.25       7.20
         7       29.50       7.20
         8       26.13       7.20
         9       22.75       7.20
        10       18.00       7.20
        11       16.00       7.20
        12       14.00       7.20
        13        9.25       7.20
        14        5.88       7.20
        15        2.50       7.20
        16        0.75       7.20
        17        0.75       5.30
        18        2.50       4.70
        19        5.88       3.50
        20        9.25       2.30
        21       14.00       1.17   support


   Members:

      Number  Start Node   End Node
      -----------------------------
         1         1          2
         2         2          3
         3         3          4
         4         4          5
         5         5          6
         6         6          7
         7         7          8
         8         8          9
         9         9         10
        10        10         11
        11        11         12
        12        12         13
        13        13         14
        14        14         15
        15        15         16
        16        16         17
        17        17         18
        18        18         19
```

```
        19        19         20
        20        20         21
        21        21          1
        22        15         18
        23        14         19
        24        13         20
        25        12         21
        26        10          1
        27         9          2
        28         8          3
        29         7          4
        30        16         18
        31        15         19
        32        14         20
        33        13         21
        34        11         21
        35        11          1
        36         9          1
        37         8          2
        38         7          3
        39         6          4


SUPERSTRUCTURE INFO
===================

   Total number of spans: 2   Span number rear to current pier: 1
   Number of traffic lanes: 2

   Beam:  height = 36.00 in   section area = 369.00 in^2
   Barrier height = 36.00 in  Depth of slab = 8.00 in

            Span           Forward        Rear
      ---------------------------------------------
      Overall width        35.00 ft       35.00 ft
      Curbs  width         32.00 ft       32.00 ft
      Span Length          60.00 ft       60.00 ft


BEARING POINTS
==============

   Number of bearing lines: 1
      First bearing line      Eccentricity = 0.00 ft
                 Point      Distance ft
                 ----------------------
                   1            2.50
                   2            9.25
                   3           16.00
                   4           22.75
                   5           29.50
```

```
MATERIAL PROPERTIES
===================

                                  Cap         Column      Footing
                         ------------------------------------------
   Concrete Type                  normal       normal       normal
   Concrete Strength  (psi)      4000.00      4000.00      4000.00
   Concrete Density (lb/ft3)      150.00       150.00       150.00
   Concrete Modulus Ec  (ksi)    3834.30      3834.30      3834.30
   Steel Strength Fy  (ksi)        60.00        60.00        60.00


DESIGN PARAMETERS
=================

   AASHTO LRFD Code
      Resistance factors for reinf. concrete:        Multi presence factors for live load:
         Flexure and tension           0.90                         1  Lane       1.20
         Shear and torsion (normal)    0.90                         2  Lanes      1.00
                      (lightweight)    0.70                         3  Lanes      0.85
         Axial compression (ties)      0.75               more than 3  Lanes      0.65
         Axial compression (spiral)    0.75
         Compression in STM            0.70

      Eta factor:
                    Service  Fatigue  Strength  Extreme event
         ----------------------------------------------------
         Cap          1.00     1.00      1.00      1.00
         Column       1.00     1.00      1.00      1.00
         Footing      1.00     1.00      1.00      1.00

      Dynamic load allowance IM:
                     Truck     Lane    Fatigue
         --------------------------------------
         Cap          0.33     0.00      0.15
         Column       0.33     0.00      0.15
         Footing      0.00     0.00      0.00

               Crack control factor      Min clear cover
                     kip/ft                        in
         -----------------------------------------------
         Cap            170.00                   2.00
         Column         170.00                   2.00
         Footing        170.00                   3.00

      Degree of fixity in foundations for Moment Magnify Method:  R =  5.00
```

```
LOADS
=====

   Load Cases: 5


      Loadcase ID: DC1     Name:
      Multiplier =  1.000

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          1         Y       -82.48
           1          2         Y       -77.45
           1          3         Y       -77.45
           1          4         Y       -77.45
           1          5         Y       -82.48

      Auto generation details:

      Generated Dead Load
         Slab weight    =   150.00     pcf   Girder weight =   150.00     pcf
         Wearing weight =     Not included   Barrier load  =  1200.00     plf


      Loadcase ID: DW1     Name:
      Multiplier =  1.000

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          1         Y        -8.85
           1          2         Y        -8.10
           1          3         Y        -8.10
           1          4         Y        -8.10
           1          5         Y        -8.85

      Auto generation details:

      Generated Dead Load
         Slab weight    =     Not included   Girder weight =     Not included
         Wearing weight =   700.00     plf   Barrier load  =     Not included


      Loadcase ID: LL1     Name:
      Multiplier =  1.000

      Bearing loads:
         Line #    Bearing #   Dir.    Load,  kip
         -----------------------------------------
           1          1         Y       -47.35
           1          2         Y       -27.85
           1          3         Y         0.00
           1          4         Y         0.00
           1          5         Y         0.00
           1          1         Y       -29.21
           1          2         Y       -16.68
           1          3         Y        -0.19
           1          4         Y         0.00
           1          5         Y         0.00
```

```
Auto generation details:

Generated Live Load
      Library load:  Design Truck + Lane Load
      Number of loaded lanes = 1
          Total number of considered truck positions = 23


Loadcase ID: LL2     Name:
Multiplier =  1.000

Bearing loads:
   Line #    Bearing #   Dir.    Load,  kip
   -----------------------------------------
     1          1         Y         0.00
     1          2         Y         0.00
     1          3         Y         0.00
     1          4         Y       -27.85
     1          5         Y       -47.35
     1          1         Y         0.00
     1          2         Y         0.00
     1          3         Y        -0.19
     1          4         Y       -16.68
     1          5         Y       -29.21

Auto generation details:

Generated Live Load
      Library load:  Design Truck + Lane Load
      Number of loaded lanes = 1
          Total number of considered truck positions = 23


Loadcase ID: WS1     Name: Angle: 0
Multiplier =  1.000

Cap loads:
   Force(X) =    1.280 kip  Arm =   0.00 ft

Column loads:
   Col #      Type     Dir      Mag1        y1/L        Mag2       y2/L
   ----------------------------------------------------------------------
       1        UDL     X       0.160  k/ft  0.00       ----        0.65

Bearing loads:
   Line #    Bearing #   Dir.    Load,  kip
   -----------------------------------------
     1          1         X         4.00
     1          1         Y         2.47
     1          1         Z         0.00
     1          2         X         4.00
     1          2         Y         0.00
     1          2         Z         0.00
     1          3         X         4.00
     1          3         Y         0.00
     1          3         Z         0.00
     1          4         X         4.00
     1          4         Y         0.00
     1          4         Z         0.00
```

```
           1          5         X         4.00
           1          5         Y        -2.47
           1          5         Z         0.00

      Auto generation details:
      Generated Wind Load on Structure
         Angle of wind = 0.00 deg    Elevation above which wind load acts =   0.00 ft
         User defined wind pressure
         Wind pressure for superstructure:           Wind pressure for substructure:
              Transverse     50.000 psf                        Cap        40.000 psf
            Longitudinal      0.000 psf                     Column        40.000 psf
             Overturning   not considered


   Selected load groups:

      STRENGTH GROUP I
      STRENGTH GROUP III



LOAD COMBINATIONS - AASHTO LRFD

Comb #     1  (STR GP 1    ) = 1.00 (  1.25 DC1 + 1.50 DW1 + 1.75 LL1 )
Comb #     2  (STR GP 1    ) = 1.00 (  1.25 DC1 + 1.50 DW1 + 1.75 LL2 )
Comb #     3  (STR GP 1    ) = 1.00 (  1.25 DC1 + 0.65 DW1 + 1.75 LL1 )
Comb #     4  (STR GP 1    ) = 1.00 (  1.25 DC1 + 0.65 DW1 + 1.75 LL2 )
Comb #     5  (STR GP 1    ) = 1.00 (  0.90 DC1 + 1.50 DW1 + 1.75 LL1 )
Comb #     6  (STR GP 1    ) = 1.00 (  0.90 DC1 + 1.50 DW1 + 1.75 LL2 )
Comb #     7  (STR GP 1    ) = 1.00 (  0.90 DC1 + 0.65 DW1 + 1.75 LL1 )
Comb #     8  (STR GP 1    ) = 1.00 (  0.90 DC1 + 0.65 DW1 + 1.75 LL2 )
Comb #     9  (STR GP 3    ) = 1.00 (  1.25 DC1 + 1.50 DW1 + 1.40 WS1 )
Comb #    10  (STR GP 3    ) = 1.00 (  1.25 DC1 + 1.50 DW1 - 1.40 WS1 )
Comb #    11  (STR GP 3    ) = 1.00 (  1.25 DC1 + 0.65 DW1 + 1.40 WS1 )
Comb #    12  (STR GP 3    ) = 1.00 (  1.25 DC1 + 0.65 DW1 - 1.40 WS1 )
Comb #    13  (STR GP 3    ) = 1.00 (  0.90 DC1 + 1.50 DW1 + 1.40 WS1 )
Comb #    14  (STR GP 3    ) = 1.00 (  0.90 DC1 + 1.50 DW1 - 1.40 WS1 )
Comb #    15  (STR GP 3    ) = 1.00 (  0.90 DC1 + 0.65 DW1 + 1.40 WS1 )
Comb #    16  (STR GP 3    ) = 1.00 (  0.90 DC1 + 0.65 DW1 - 1.40 WS1 )


Load Combinations for Columns are the same
```

```
CAP DESIGN
==========

                          CAP DESIGN

Code: AASHTO LRFD
Units: US
                Strut and Tie Method Used for Analysis.

DESIGN PARAMETERS:
==================
            f'c = 4000.0  psi       fy = 60000.0  psi
            phi flex = 0.90         phi shear = 0.90
            Ec =  3834.3  ksi       Es = 29000.0  ksi
            crack control factor  z = 170.00  kips / in
            Concrete Type :  Normal Weight.


CAP GEOMETRY:
=============
       Hammer Head Cap :  Length(X) = 32.00  ft     Depth(Z) = 48.00  in


REINFORCEMENT:
==============

         Bar size   Quantity  Bar dist. As total    From      To       Hook
                                 in       in^2       ft       ft
------------------------------------------------------------------------------
TOP
          #  8         19       9.60    15.010     0.00     32.00      Both


   Stirrups size:  # 4


STRUT AND TIE DESIGN:
=====================
              User Defined Strut Width:    0.40  ft

 Top Tension Ties

 Member#       Force      Comb      As-req       As-prv
                kips                 in^2         in^2
----------------------------------------------------------
 ST    6         9.9        9         0.18        15.01
 ST    7       285.0        2         5.28        15.01
 ST    8       435.8        2         8.07        15.01
 ST    9       776.9        2        14.39        15.01
 ST   10       776.9        2        14.39        15.01
 ST   11       776.8        1        14.39        15.01
 ST   12       776.8        1        14.39        15.01
 ST   13       435.7        1         8.07        15.01
 ST   14       285.9        1         5.29        15.01
 ST   15         9.9       10         0.18        15.01
```

```
 Bottom Compression Struts

 Member#    Force     Comb    As-req      As-prv   Comp_c   Comp_allow
             kips              in^2        in^2       ksi        ksi
------------------------------------------------------------------------
 ST    1   -447.9       2       0.00        0.00     2.78      3.36
 ST    2   -302.5       2       0.00        0.00     1.88      3.23
 ST    3    -10.6      10       0.00        0.00     0.07      3.23
 ST    4     -0.1      10       0.00        0.00     0.00      3.24
 ST   17     -0.1       9       0.00        0.00     0.00      3.24
 ST   18    -10.6       9       0.00        0.00     0.07      3.23
 ST   19   -303.5       1       0.00        0.00     1.88      3.23
 ST   20   -447.9       1       0.00        0.00     2.78      3.36
 ST   21   -566.7       1       0.98 *      0.00     3.51 *    3.26


 Vertical Compression Struts

 Member#       Force      Comb       Comp_c     Comp_allow
                kips                    ksi           ksi
------------------------------------------------------------
 ST   25        -7.0        1         0.04          0.54
 ST   26        -7.0        1         0.04          0.54
 ST   30       -17.1       10         0.11          1.56
 ST   31      -409.3        1         2.54 *        2.02
 ST   32      -264.5        1         1.64 *        1.53
 ST   33      -551.2        1         3.42 *        1.76
 ST   34      -667.5        1         4.14 *        1.67
 ST   35      -667.5        2         4.14 *        1.67
 ST   36      -551.2        2         3.42 *        1.76
 ST   37      -265.4        2         1.65 *        1.54
 ST   38      -408.8        2         2.53 *        2.01
 ST   39       -17.1        9         0.11          1.56


 Vertical Tension Ties

 Member#       Force      Comb      Av-req       S max
                kips                 in^2           in
--------------------------------------------------------
 ST    5         7.0        9         0.13       12.00
 ST   16         7.0       10         0.13       12.00
 ST   22        17.6       10         0.33       12.00
 ST   23       210.9        1         3.91       12.00
 ST   24       223.0        1         4.13       12.00
 ST   27       223.0        2         4.13       12.00
 ST   28       211.5        2         3.92       12.00
 ST   29        17.6        9         0.33       12.00
 ST   34       563.9        8        10.44       12.00
 ST   35       563.9        7        10.44       12.00


ISOLATED FOOTING DESIGN

Code: AASHTO LRFD
Units: US
```

```
Geometry:
=========
   Name  : Stm Pile Cap
   Shape : Rectangular, Type : Pile/Shaft Cap

   Bf(X) = 20.00 ft, Hf(Z) = 10.00 ft, Thickness(Y) = 72.00 in

   Start at X = -10.00 ft from centerline of column.
   Columns located on the footing:
   Column No.  1  at x =   0.00 ft,  Rectangular  96.00 in x  48.00 in

   Ag = 200.00 ft^2, Ix = 54.00 ft^2, Iz = 294.00 ft^2

   Surcharge =   1.00 ksf

   Piles:   Circular      Size: 14.00 in   Capacity: 250.00 kips
```

```
Design Parameters:
==================     f'c =  4000.00  psi      fy = 60000.00  psi
                       phi flex = 0.90          phi shear = 0.90
                       Ec =    3834.3  ksi      Es =   29000.0 ksi
                       Crack control factor  z = 170.00 kips/in
                       Concrete Type :  Normal Weight.




Pile Reactions, Factored:
=========================

Pile Loc(X)   X       Z   |---------------- Column Loads ----------------| Pile Reac.
       ft     in      in   comb    Ovs      P, kips    Mxx, kft   Mzz, kft    kips
------------------------------------------------------------------------------------
  1  -7.00   36.0    36.0    1     ---     -990.88       0.00    2334.64      220.68
                             8     ---     -754.80       0.00   -2334.64       83.28
  2   0.00  120.0    36.0    1     ---     -990.88       0.00    2334.64      165.10
                            15     ---     -542.56       0.00    -702.61      115.28
  3   7.00  204.0    36.0    2     ---     -990.88       0.00   -2334.64      220.68
                             7     ---     -754.80       0.00    2334.64       83.28
  4  -7.00   36.0     0.0    1     ---     -990.88       0.00    2334.64      220.68
                             8     ---     -754.80       0.00   -2334.64       83.28
  5   0.00  120.0     0.0    1     ---     -990.88       0.00    2334.64      165.10
                            15     ---     -542.56       0.00    -702.61      115.28
  6   7.00  204.0     0.0    2     ---     -990.88       0.00   -2334.64      220.68
                             7     ---     -754.80       0.00    2334.64       83.28
  7  -7.00   36.0   -36.0    1     ---     -990.88       0.00    2334.64      220.68
                             8     ---     -754.80       0.00   -2334.64       83.28
  8   0.00  120.0   -36.0    1     ---     -990.88       0.00    2334.64      165.10
                            15     ---     -542.56       0.00    -702.61      115.28
  9   7.00  204.0   -36.0    2     ---     -990.88       0.00   -2334.64      220.68
                             7     ---     -754.80       0.00    2334.64       83.28


Note:

   Only max. force in piles is considered for design.
   Pile coordinates X and Z are from the most left edge of the footing.


Max. Pile Reaction Used in Design: (without selfweight and surcharge)
==================================

 Factored pile reaction  = 165.68 kips



Reinforcement Schedule:
=======================:

 Dir  Quantity  Size  Bar dist.  As total    Spacing    Hook
                         in        in^2         in
------------------------------------------------------------
  X      22     #  8    6.00        17.38      5.38     Both
  Z      44     #  8    6.00        34.76      5.42     Both
```

```
Tension Ties ( X direction )
========================================
Member#     Force    Comb   Asb_req    Asb_prov
             kips            in^2       in^2
------------------------------------------------
STx  6      298.23     2      7.92      17.38
STx  7      298.23     2      7.92      17.38


Tension Ties ( Z direction )
========================================
Member#     Force    Comb   Asb_req    Asb_prov
             kips            in^2       in^2
------------------------------------------------
STz  1       99.41     2     15.84      34.76
STz  2       99.41     2     15.84      34.76


Compression Struts ( X direction )
========================================
Member#     Force    Comb    Comp_c   Comp_allow
             kips             ksi        ksi
------------------------------------------------
STx  1      -99.41     1      0.25       3.40
STx  2     -579.66     1      1.44       2.86
STx  3     -579.66     1      1.44       2.86
STx  4     -318.27     1      0.79       2.51
STx  5     -318.27     1      0.79       2.51


Compression Struts ( Z direction )
========================================
Member#     Force    Comb    Comp_c   Comp_allow
             kips             ksi        ksi
------------------------------------------------
STz  3     -506.90     1      0.63       3.39
STz  5     -506.90     1      0.63       3.39
STz  6     -267.67     1      0.33       3.17
STz  7     -267.67     1      0.33       3.17


Cracking/Fatigue Check ( X direction )
========================================
Member#   fCrack   Crack    Comb  fFatigue   Fatigue  Comb
            ksi                      ksi
------------------------------------------------------------


Cracking/Fatigue Check ( Z direction )
========================================
Member#   fCrack   Crack    Comb  fFatigue   Fatigue  Comb
            ksi                      ksi
------------------------------------------------------------
```

Nomenclature and Printout Explanation

Appendix A

The following is a glossary of the terms that appear in the program.

Pier Geometry

Cap shape: straight	: Cap with constant section
Cap shape: tapered	: Cap with tapered segments at both ends
Cap shape: variable	: Cap with different sections along its length
Start elevation, cap	: Elevation at top surface of cap at left end
End elevation, cap	: Elevation at top surface of cap at right end
Cap depth	: Dimension of a cap in global Z-direction
Cap height	: Cap height in global Y-direction
Cap length	: Cap length in global X-direction
Skew angle	: Positive if measured anti-clockwise from normal to bridge traffic direction to the pier centerline
Reduction of I	: Factor of Reduced Moment of Inertia
Column shape: rectangular	: Column with rectangular section
Column shape: round	: Column with circular section
Column tapered in X-dir	: Rectangular column tapered in global X-direction
Column tapered in Z-dir	: Rectangular column tapered in global Z-direction
Column tapered in both X and Z-dir	: Rectangular column tapered in both global X and Z directions
Column top elevation	: Elevation at centerline of a cap where a column meets with it.

Structural Model

Member	: Member number
Node	: Node number
Hinge	: At a hinge, no restraining will be applied for rotation
Check Pt	: Checking points
Dist	: Distance of a node in the global coordinates
Length	: Length of a member

Superstructure Information

Beam height	: Beam (girder) height
Section area	: Section area of beam (girder)
Barrier height	: Height of barrier in superstructure
Depth of slab	: Slab depth
Number of spans	: Total number of spans in a bridge
Bridge overall width	: Width of a bridge in transverse direction
Bridge curb width	: Curb to curb distance of a bridge in transverse direction
Span length	: Length of a span in longitudinal direction
Span number rear to current pier	: Span rear to the pier under consideration

Bearing Points

Number of bearing lines	: Number of bearing lines on a pier
Eccentricity	: Eccentricity of bearing points with respect to centerline of pier cap

Material Properties

Concrete type	: Type of concrete used
Concrete strength	: 28-day compressive strength of concrete
Concrete density	: Unit weight of concrete
Concrete modulus	: Modulus of elasticity of 28-day strength concrete
Steel strength	: Steel yield strength

Design Parameters

Strength reduction factors	: ϕ factors used in design
Multi presence factor	: Factor for reduction in live load intensity due to more than one lane loaded
Impact factor	: Factor for increase of live load due to impact effect
Crack control factor	: z factor in calculation of distribution of flexural reinforcement
Min clear cover	: Concrete cover from edge of member to edge of stirrup
Degree of fixity in foundations	: Factor used in calculation of k for moment magnification for unbraced column

Loads

Slab weight : Unit weight of a slab
Girder weight : Unit weight of a girder
Wearing surface weight : Linear load for wearing surface
Barrier load : Linear load for a barrier
Multiplier : A factor multiplied to current loads
Bearing load : Load acting on bearing points
Cap load : Load acting on centerline of a cap
Arm : Distance between centerline of cap and point of load
Column load : Load acting on column
Dir : Direction of column load, either in global X, Y, or Z direction
Mag 1 : Magnitude of column load at starting point
y1/L : Location of column load at starting point, in terms of column length, measured from the fixity point of a column (i.e., bottom of the column)
Mag 2 : Magnitude of column load at ending point
y2/L : Location of column load at ending point, in terms of column length, measured from the fixity point of a column (i.e., bottom of the column)
Angle of wind : Angle of winds acting on a bridge, measured from the normal to a bridge longitudinal direction to the wind pressure direction. Positive if anti-clockwise.
Elevation above which wind load acting : Define the elevation, above which the wind load is applied on column
Live load length : Length of live load; default value is average span length
Library load : Live load designation selected from Live Load Library
Importance classification : Used to determine the Seismic Performance Category for seismic load generation
Unit weight of superstructure : Linear superstructure unit weight
Iy of superstructure : Moment of inertia of superstructure about global Y-axis
Iz of superstructure : Moment of inertia of superstructure about global Z-axis
Span number rear to current pier : Span number at back of pier under consideration
Avg. height : Average height of all columns for pier under consideration (0 if abutment)

Cap Design

Quantity : Number of steel bars used in design
Bar dist : Distance between center of steel bar to edge of cap
As total : Total steel area
From : Starting location of steel bar

To	Ending location of steel bar
Loc	Location of design section from start of current member
AbsLoc	Location of design section from left end of cap
H	Height of cap
Mmax	Maximum moment at design section
Mmin	Minimum moment at design section
Mn	Resisting moment, which has ϕ factor considered
Comb	Combination number
Asb-req	Required bottom steel area
Asb-prv	Provided bottom steel area
Asb-eff	Effective bottom steel area, after considering development length
Ast-req	Required top steel area
Ast-prv	Provided top steel area
Ast-eff	Effective top steel area, after considering development length
Vu	Factored shear force
Tu	Factored torsion
Phi*Vc	Shear capacity of concrete section, which has ϕ factor considered
Phi*Tcr	Torsion capacity of concrete section, which has ϕ factor considered
Av/s	Required stirrup area per unit length
Alx Alt	Additional longitudinal steel due to shear and torsion (in addition to the provided steel)
Smax	Maximum stirrup spacing
fs-t	Cracking/fatigue stress at top
fs-b	Cracking/fatigue stress at bottom
Ratio fs-t	Ratio between cracking/fatigue stress at top and allowable stress
Ratio fs-b	Ratio between cracking/fatigue stress at bottom and allowable stress
*	Warning flag if code requirement is not satisfied

Cap Design by STM

Force	Axial force in truss member
As-req	Required steel area
As-prv	Provided steel area
Comp_c	Factored compressive stress in strut
Comp_allow	Allowable compressive stress in strut
Av-req	Required area of stirrup

Smax	: Maximum stirrup spacing
*	: Warning flag if a code requirement is not satisfied

Column Design

Pu	: Factored axial load
Mux	: Factored moment about global X-direction
Muz	: Factored moment about global Z-direction
pMn	: Resisting moment, having ϕ factored considered
c	: Neutral axis location
Incl	: Inclined angle of neutral axis
pPn/Pu	: Ratio between the resisting axial capacity and factored force, set to one in RC-PIER
pMn/Mu	: Ratio between the resisting moment capacity and factored moment
k	: Effective length factor for compression members
Cm	: Factor relating the actual moment diagram to an equivalent uniform moment diagram
Beta	: Absolute value of ratio of maximum dead load moment to maximum total load moment
Delta B	: Moment magnification factor for members braced against sidesway
Delta S	: Moment magnification factor for members not braced against sidesway
pPc	: Critical Euler load
*	: Warning flag if code requirement is not satisfied

Footing Design

Ag	: Area of footing
Ix	: Moment of inertia with respect to global X-direction
Iz	: Moment of inertia with respect to global Z-direction
d	: Distance from extreme compression fiber to centroid of tension reinforcement
dv	: Effective shear depth used in LRFD shear calculations
Mmax	: Factored moment at critical section/checkpoints
Asb-prv	: Provided steel area at bottom
Asb-eff	: Effective steel area at bottom, after considering development length
Asb-req	: Required steel area at bottom
Ast-prv	: Provided steel area at top

Ast-eff : Effective steel area at top, after considering development length

Ast-req : Required steel area at top

fs-t : Cracking/fatigue stress at top

fs-b : Cracking/fatigue stress at bottom

Ratio fs-t : Ratio between cracking/fatigue stress at top and allowable stress

Ratio fs-b : Ratio between cracking/fatigue stress at bottom and allowable stress

Vu : Factored shear force at section

phi*Vc : Shear strength provided by concrete, having ϕ factored considered

Bo : Perimeter of critical section for two-way shear

Ao : Area within perimeter of critical section for two-way shear

Avg.d : Average distance from extreme compression fiber to centroid of tension reinforcement

Avg.dv : Average effective shear depth

* : Warning flag if code requirement is not satisfied

Hand Calculations for Selected Items

Tutorial 1: Two-Column Pier, Auto Load Generation (AASHTO LFD)

This example demonstrates the design of a two-column pier with rectangular pier cap beam, as shown in Figure TH-2. This example illustrates in detail the design of the cap at two different sections, the design of the column, spread footing and pile cap footing under left column. The design will be carried out in accordance with *AASHTO Standard Specifications for Highway Bridges*, Seventeenth Edition, 2003.

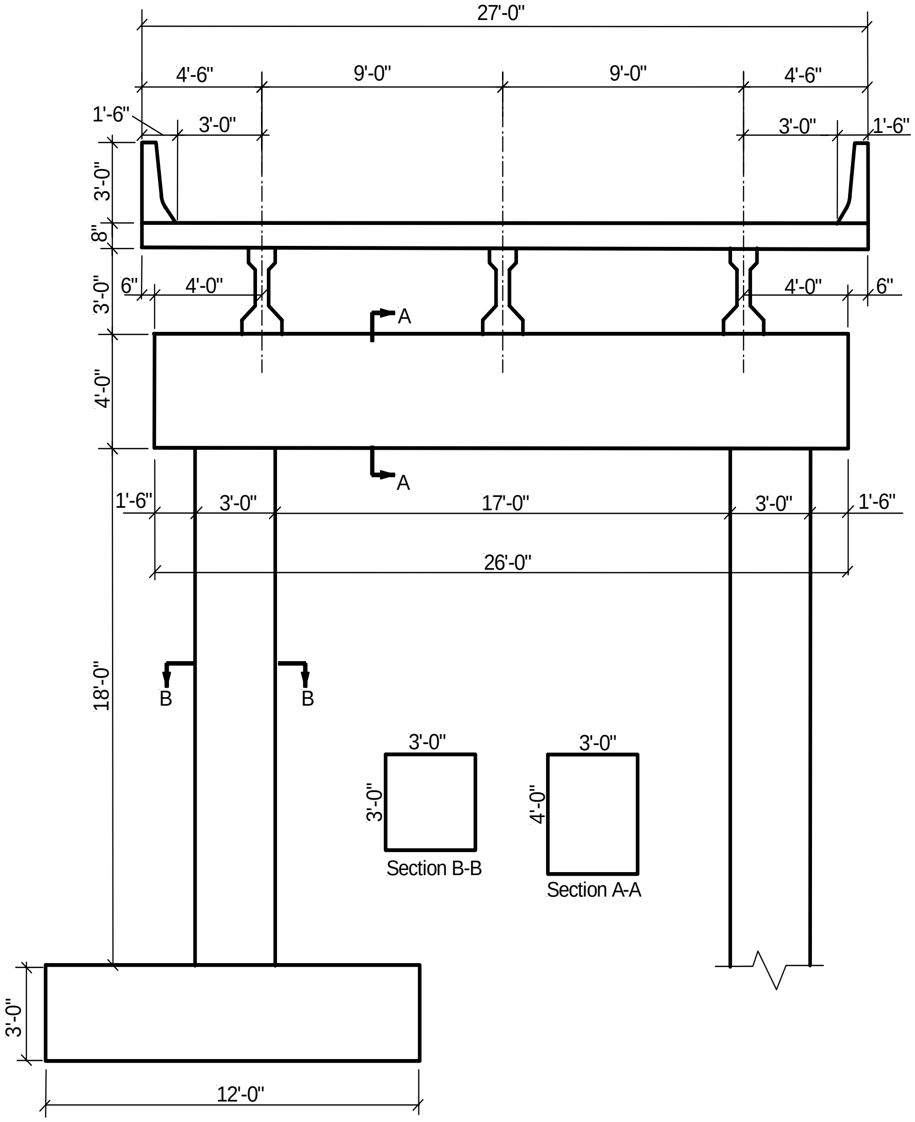
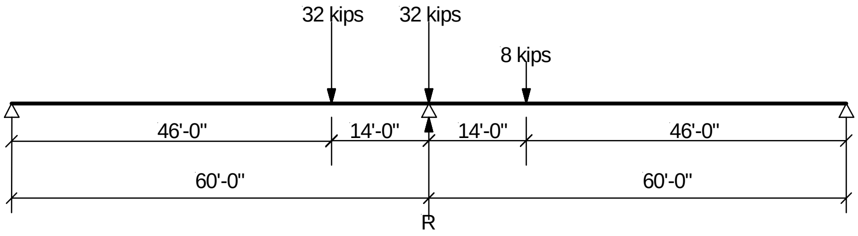
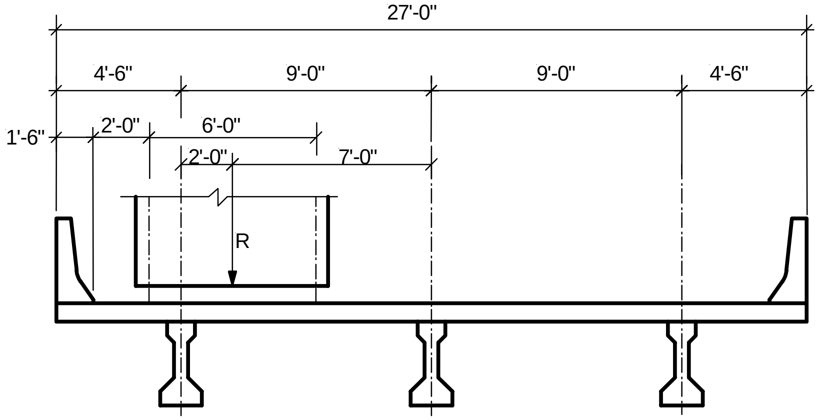


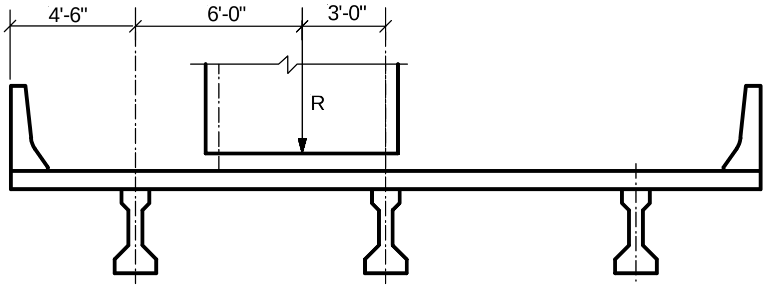
Figure B-1 Elevation of Bridge Pier with Cross-Sections for Cap and Column (Tutorial 1)



(a) Longitudinal View

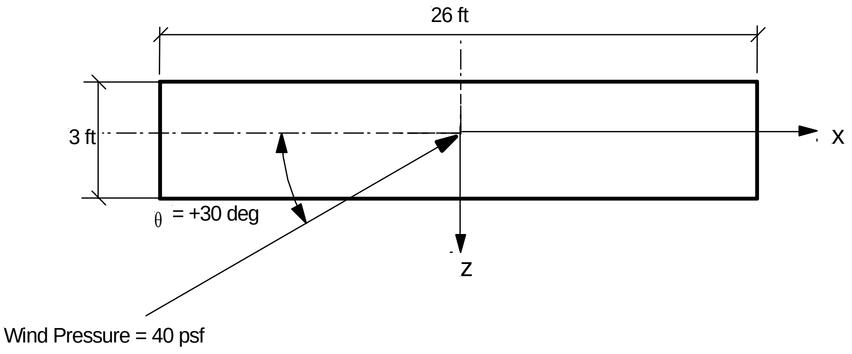


(b) Cases LL1 and LL4

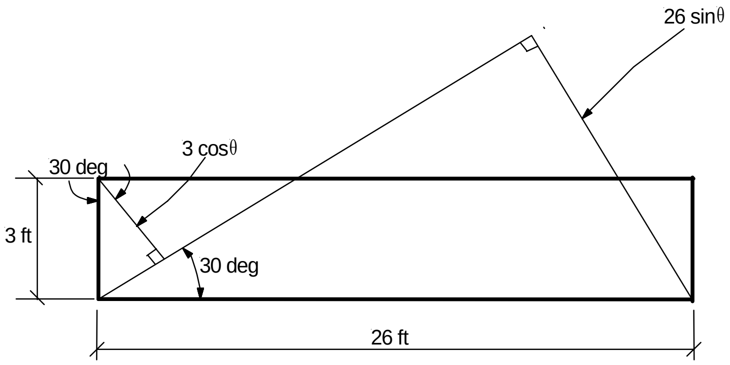


(c) Cases LL2 and LL3

Figure B-2 Live Load Truck Positions (Tutorial 1)



(a) Plan View of the Pier Cap where wind pressure acts on



(b) Calculation of projected area for wind pressure acts on a pier cap

Figure B-3 Wind Load on Substructure (Tutorial 1)

Problem Data

Concrete Strength

Cap	f’c = 4000 psi
Columns	f’c = 4000 psi
Footings	f’c = 4000 psi
Modulus of Elasticity	E_c = 3834 ksi

Concrete Density

Cap	ρ = 150 pcf
Columns	ρ = 150 pcf
Footings	ρ = 150 pcf

Steel Yield Strength

Cap f_y = 60 ksi
Columns f_y = 60 ksi
Footings f_y = 60 ksi
Modulus of Elasticity E_s = 29000 ksi

Concrete Type

Cap	Normal
Columns	Normal
Footings	Normal

Other Parameters

Crack Control Factor, z = 170 kips/in

Pier Configuration

Rectangular non-tapered multi-columns, with straight cap

Superstructure Parameters

Number of lanes = 2
(Note that, in this tutorial, the pier is only designed for 1 lane loaded)
Beam Height = 36 in
Beam Section Area = 369 in^2
Barrier Height = 36 in
Slab Depth = 8.0 in
Total number of spans = 2

Span Information

Bridge Overall Width, ft	Forward = 27 ft	Rear = 27 ft
Curb to Curb Distance, ft	Forward = 24 ft	Rear = 24 ft
Span Length, ft	Forward = 60 ft	Rear = 60 ft

Substructure Parameters

Cap: Length × Height × Depth = 26 ft × 48 in × 36 in
    Skew Angle = 0 degrees
    Start Elevation = 22 ft
    End Elevation = 22 ft
Factor of Reduced Moment of Inertia = 1.0 (non-cracked section)
Columns: Fixed at base
    Column Height = 20 ft
    Width × Depth = 36 in × 36 in
    Factor of Reduced Moment of Inertia = 1.0 (non-cracked section)
    Bearings = one line with no eccentricity

Loads

Dead Load

Self-weight = 150 pcf
Slab Girder = 150 pcf
Girder Weight = 150 pcf
Barrier Weight = 600 plf each side
Total Barrier Weight = 2 × 600 = 1200 plf
Future Wearing Surface Load = 20 psf
= 20 × 27 = 540 plf

Live Load

HS20 Truck

Wind on Structure

Direction of wind = +30°
Elevation above which wind acts = 0 ft
Trans. wind pressure on superstructure = 41 psf
Longitudinal pressure on superstructure = 12 psf
Wind pressure on cap = 40 psf
Wind pressure on column = 40 psf

Wind on Live Load

Wind Angle = +30°
Trans. wind pressure on live load = 82 plf
Longitudinal wind pressure on live load = 24 plf
Length of live load = 60 ft

Footing Surcharge

Footing $\sigma = 0.200$ ksf

Load Combinations

Service Group I

$Q = 1.00\ (1.00\ DL + 1.00\ (LL + I)_{1 \text{ or } 2 \text{ or } 3 \text{ or } 4})$

Service Group IA

$Q = 1.00\ (1.00\ DL + 2.00\ (LL + I)_{1 \text{ or } 2 \text{ or } 3 \text{ or } 4})$

Service Group III

$Q = 1.00\ (1.00\ DL + 1.00\ (LL + I)_{1 \text{ or } 2 \text{ or } 3 \text{ or } 4} \pm 0.30Wl \pm 1.00WLl)$

Service Group VI

$Q = 1.00\ (1.00\ DL + 1.00\ (LL + I)_{1 \text{ or } 2 \text{ or } 3 \text{ or } 4} \pm 0.30Wl \pm 1.00WL1)$

Load Factor Group I

$Q = 1.30\ (1.00\ DL + 1.67\ (LL + I)_{1 \text{ or } 2 \text{ or } 3 \text{ or } 4})$

Load Factor Group IA

$Q = 1.30\ (1.00\ DL + 2.20\ (LL + I)_{1 \text{ or } 2 \text{ or } 3 \text{ or } 4})$

Load Factor Group III. $Q = 1.30\ (1.00\ DL + 1.00\ (LL + I)_{1 \text{ or } 2 \text{ or } 3 \text{ or } 4} \pm 0.30Wl \pm 1.00WLl)$

Load Factor Group VI

$Q = 1.25\ (1.00\ DL + 1.00\ (LL + I)_{1 \text{ or } 2 \text{ or } 3 \text{ or } 4} \pm 0.30Wl \pm 1.00WLl)$

Load Generation

Dead Load from Superstructure

The tributary width of deck to each beam is 9 ft. The tributary span length to the pier is 30 ft rear and 30 ft forward (for simple span distribution). Thus, the total length is 60 ft.

Load Type	**Exterior Bearing Points (kips)**	**Interior Bearing Points (kips)**
Due to slab weight	(8/12)(0.150)(9)(60) = 54.00	(8/12)(0.150)(9)(60) = 54.00
Due to girder weight	(369/144)(0.150)(60) = 23.06	(369/144)(0.150)(60) = 23.06
Due to barrier weight	(1.2)(9/27)(60) = 24.00	(1.2)(9/27)(60) = 24.00
Due to wearing surface	(0.54)(9/27)(60) = 10.80	(0.54)(9/27)(60) = 10.80
Total	111.86	111.86

Dead Load

Bearing 1	P = -111.86 kips
Bearing 2	P = -111.86 kips
Bearing 3	P = -111.86 kips

Live Load: Due to HS20 Truck Only

Live load generation assumes only one lane loaded.

To get maximum reaction from the truck at the pier, position the truck such that the middle axle is over the pier centerline, first axle is at 14 ft from pier centerline in right span, and third axle is at 14 ft from pier centerline in left span, as shown in Figure TH-2a.

Therefore, total reaction at the pier is:

$$R = 32 + (8 \times 46)/60 + (32 \times 46)/60 = 62.67 \text{ kips (per lane)}$$

Cases LL1 and LL4

To get maximum reaction at exterior girders, position truck so the distance between the inner face of the barrier and the nearest wheel of the truck equals 2 ft, as shown in Figure TH-2b. Therefore, the weight of the truck will be distributed between the exterior and interior girders, as follows:

- Exterior girder takes 7/9 = 0.778 of truck reaction at the pier
- Interior girder takes 2/9 = 0.222 of truck reaction at the pier
- Bearing load from Exterior girder = 62.67 × 0.778 = 48.74 kips
- Bearing load from Interior girder = 62.67 × 0.222 = 13.93 kips

LL1:

Bearing 1	P = -48.74 kips
Bearing 2	P = -13.93 kips
Bearing 3	P = 0 kips

LL4:

Bearing 1	P = 0 kips
Bearing 2	P = -13.93 kips
Bearing 3	P = -48.74 kips

Cases LL2 and LL3

To get maximum reaction at interior girder, position truck so that one line of wheels lies exactly on centerline of interior girder, as shown in Figure TH-2c. Therefore, the weight of the truck will be distributed between the exterior and interior girder, as follows:

- Exterior girder takes 3/9 = 0.333 of truck reaction at the pier
- Interior girder takes 6/9 = 0.667 of truck reaction at the pier
- Bearing load from Exterior girder = 62.67 × 0.333 = 20.89 kips
- Bearing load from Interior girder = 62.67 × 0.667 = 41.78 kips

LL2:

Bearing 1	P = -20.89 kips
Bearing 2	P = -41.78 kips
Bearing 3	P = 0 kips

LL3:

Bearing 1	P = 0 kips
Bearing 2	P = -41.78 kips
Bearing 3	P = -20.89 kips

Wind Load

Assume wind load at an angle θ of +30 degrees, as shown in Figure TH-3a.

Transverse Wind Loads:

The transverse pressure is 41 psf, acting on the height of superstructure at the center of this area.

The total height of the superstructure is:

36 barrier + 8 deck slab + 36 beam = 80 in

Total transverse load is:

(41/1000) (80/12) (60) = 16.4 kips

Distribute the total load equally between the bearing points, and the transverse load is:

$F_x = 16.4/3 = +5.47$ kips/bearing

Since this load is applied at the center of the height of superstructure, it creates an overturning moment = (16.4)(80/12)/2 = 54.67 kips-ft at the bearing elevation.

This moment will be distributed to the two exterior girders, where

- One exterior girder (at bearing point #3) applies a downward force, that is, $-F_y$, on the pier cap, and
- Exterior girder on other side (at bearing point #1) applies upward force, $+F_y$.

This force, $F_y = 54.67/18 = +3.04$ kips at bearing 1 and -3.04 at bearing 3.

Longitudinal Wind Loads:

The longitudinal pressure, 12 psf, is acting on the height of superstructure (AASHTO Std. 3.15.1.2). Therefore, total longitudinal load is:

= (12/1000) (80/12) (60) = 4.8 kips

Distribute this force equally between the bearing points, and longitudinal force, F_z, is 4.8/3 = -1.6 kips/bearing.

The bearings are considered to act as a hinge about the X-axis of the pier and do not transmit any M_x moment to the substructure due to longitudinal loads. Therefore, the longitudinal wind load results only in F_Z force acting at the bearing elevation, which is the top surface of pier cap.

Loads from Superstructure:

For wind load with skew angle of 30 degrees, transverse pressure = 41 psf and longitudinal pressure = 12 psf (AASHTO Std. 3.15.2.1).

Bearing Loads:

Bearing 1

X	P = 5.47 kips
Y	P = 3.04 kips
Z	P = -1.6 kips

Bearing 2

X	P = 5.47 kips
Z	P = -1.6 kips

Bearing 3

X	P = 5.47 kips
Y	P = -3.04 kips
Z	P = -1.6 kips

Loads on Substructure:

According to AASHTO Std. 3.15.2.2, a 40 psf wind pressure acting on an angle of θ = +30 degrees, should be applied. Since the wind load is not normal to the exposed surface area, the wind force should be calculated based on the projected area of exposed surface.

Loads on Pier Cap:

Total projected length of pier cap (normal to the wind direction) is

= 26 sin(30) + 3 cos(30) = 15.598 ft

Total wind force = 0.04 × 4 (depth) × 15.598 = 2.496 kips

Therefore,

Transverse Load, F_x, is

= 2.496 cos(30) = +2.162 kips

Longitudinal Force, Fz, is

= 2.496 sin(30) = -1.248 kips = -1.248/26 = -0.048 kip/ft

Cap Loads

X-dir:P = 2.162 kips

Z-dir:ω = - 0.048 klf

Loads on the Column:

Total projected length of the column (normal to wind direction) is

= 3 sin(30) + 3 cos(30) = 4.098 ft

Total wind force = 0.04 × 4.098 = 0.164 kip/ft of column length

Therefore,

Transverse load, F_x = 0.164 cos(30) = +0.142 kip/ft

Longitudinal load, Fz = 0.164 sin(30) = -0.082 kip/ft of column length

Column load start location = $\frac{y_1}{L} = \frac{0.0\text{ft}}{20.0\text{ft}} = 0.0$

Column load end location = $\frac{y_2}{L} = \frac{18.0\text{ft}}{20.0\text{ft}} = 0.9$

Column Loads

Column 1

x-dir:	ω = 0.142 klf
z-dir:	ω = -0.082 klf

Column 2

x-dir:	ω = 0.142 klf
z-dir:	ω = -0.082 klf

All column loads starting at $\frac{y_1}{L} = 0.0$ and ending at $\frac{y_2}{L} = 0.9$

Wind Load on Moving Live Load:

With wind acting at an angle of θ = +30 degrees, the lateral wind in X-direction = 82 lb/ft and the longitudinal wind in Z-direction = -24 lb/ft. The wind load is applied to the tributary span length of the bridge (AASHTO Std. 3.15.2.1.2), i.e., 60 ft.

Total transverse force, F_x is

= +82(60)/1000 = +4.920 kips total
= 4.920/3 = +1.640 kips/bearing

The transverse load, Fx, acting 6ft above the deck, creates an overturning moment in longitudinal direction =

4.920(6 + 8/12 + 36/12) = 47.560 kip-ft.

Distributing this force between the exterior girders, creates an upward force on the left exterior girder and downward force on right exterior girder.

Fy = 47.560/18
= +2.642 kips at bearing 1
= -2.642 kips at bearing 3

Total longitudinal force applied at bearing elevation is:

Fz = -24(60)/1000
= -1.440 kips total
= -1.440/3
= -0.480 kips/bearing

Bearing Loads:

Bearing 1

X	P = 1.64 kips
Y	P = 2.642 kips
Z	P = -0.48 kips

Bearing 2

X	P = 1.64 kips
Z	P = -0.48 kips

Bearing 3

X	P = 1.64 kips
Y	P = -2.642 kips
Z	P = -0.48 kips

Impact Factor

Span Length = 60 ft

$$\text{Impact} = \frac{50}{L + 125} = \frac{50}{60 + 125} = 0.27$$ **[AASHTO Std. 3.8.2]**

Load Combination (Global Axis System)

Analysis Results

Service Group 1: Node 1:

Fy = 47.7 + 167.8+ 53.25*1.27
= 283.13 kips (283.1 by RC-PIER)

Fx = -1.771 + 13.17 + 1.979*1.27
= 17.45 kips (17.46 by RC-PIER)

Service Group 18: Node 1:

1.00(1.00D1 + 1.27L2 + 0.3W1 - 1.00WL1)

Fy = 47.7 + 167.8+ 40.73*1.27 + 0.3(-13.82) – (-5.14)
= 268.22 kips (268.3 by RC-PIER)

LFR Group 41: Node 1:

Fy = 1.3(47.7 + 167.8+ 53.25*1.27*1.67)
= 426.97 kips (427.0 by RC-PIER)

Section & Design Parameters

Width	b	=	36 in
Overall Depth	h	=	48 in
Minimum clear concrete cover		=	2.5 in
Longitudinal reinforcement		=	#8 bars
Trans. reinforcement (shear)		=	#4 bars
Effective Depth	d	=	48 - 2.5 - .5 - .5(1) = 44.5 in
Strength Reduction Factor	ϕ	=	0.9 for flexure
	ϕ	=	0.85 for shear

Design of Positive Moment Section At Center of Cap

For center of cap, see analysis results of Member 5 Node 7 and Member 6 Node 7.

Factored Moment M_u = 1326.8 kft (Envelope Strength)

Flexure Design

$M_u \leq \phi\, M_n$

$M_u = 1326.8$ kft

$$\text{Required } R_u = \frac{M_u}{\phi b d^2} = \frac{1326.8 \times 12 \times 1000}{0.9 \times 36 \times (44.5)^2} = 248.15 \text{ psi}$$

$$\rho = \frac{0.85 f_c'}{f_y}\left(1 - \sqrt{1 - \frac{2R_u}{0.85 f_c'}}\right) = \frac{0.85 \times 4000}{60000}\left(1 - \sqrt{1 - \frac{2 \times 248.15}{0.85 \times 4000}}\right)$$

$= 0.0043$

$$\rho_b = \frac{0.85\beta_1 f_c'}{f_y}\left(\frac{87000}{87000 + f_y}\right)$$ **[AASHTO Std. 8.16.4.2.3]**

$$= \frac{0.85 \times 0.85 \times 4000}{60000}\left(\frac{87000}{87000 + 60000}\right)$$

$= 0.0285$

$0.75\rho_b = 0.75 \times 0.0285 = 0.021$

$\rho < 0.75\rho_b$ **[AASHTO Std. 8.16.3.1]**

$A_s = 0.0043 \times 36 \times 44.5 = 6.89 \text{ in}^2$

Provided 10 - #8 bars = $7.90 \text{ in}^2 > 6.89 \text{ in}^2$ OK

Spacing of reinforcement = $\frac{36 - 2 \times 2.5 - 1}{9}$ = 3.33 in > 1.5db OK

[AASHTO Std. 8.21.1]

Clear spacing = 3.33 in - 1 in = 2.33 in > 1.5in OK

Check Moment of Resistance (AASHTO 8.16.3.2)

$$\rho = \frac{10 \times 0.79}{36 \times 44.5} = 0.00493$$

$M_r = \phi M_n$

$$= 0.9\left[7.9 \times 60 \times 44.5\left(1 - 0.6\frac{0.00493 \times 60}{4}\right)\right] \times \frac{1}{12}$$

= 1511.78 kft > 1326.8 kft

Check Minimum Reinforcement (AASHTO 8.17.1)

$\phi M_n \geq M_{n\text{-min}}$

where:

$M_{n\text{-min}}$ = smaller of 1.2 M_{cr} and 1.33 M_u

$M_{cr} = 7.5\sqrt{f'}c \times S_t$

S_t = section modulus for the extreme top fiber of non-composite concrete section

$$= \frac{I_g}{y_t}$$

where:

$$I_g = \frac{1}{12} \times 36 \times 48^3 = 331776 \text{ in}^4$$

$$y_t = \frac{48}{2} = 24 \text{ in}$$

$$S_t = \frac{331776}{24} = 13824 \text{ in}^3$$

$$M_{cr} = (7.5\sqrt{4000} \times 13824)\frac{1}{12000} = 546.4 \text{ kft}$$

$1.2M_{cr} = 1.2 \times 546.4 = 655.7$ kft ← controls

$1.33M_u = 1.33 \times 1326.8 = 1765$ kft

Mn-min = 655.7k-ft ≤ ϕM_n = 1511.78 kft

Design of Shear Section at Inside Face of Column 1

For maximum design values see the strength envelope results for Member 4 Node 2.

Factored Shear V_u = 435.0 kips (Fy max/min)

Factored Torsion T_u = 3.7 kft (Mx max/min)

Shear Design

$V_u \leq \phi V_n$

$V_n = V_c + V_s$

$$V_c = 2\sqrt{f'}cb_wd = 2\sqrt{4000} \times 36 \times 44.5 = 202.64 \text{ kips}$$

$\phi V_c = 0.85 \times 202.64 = 172.2$ kips

$$V_s = \frac{V_u - \phi V_c}{\phi} = \frac{435 - 172.2}{0.85} = 309.2 \text{ kips}$$

$$\frac{A_v}{s} = \frac{V_s}{fy \times d} = \frac{309.2}{60 \times 44.5} = 0.116 \text{ in}^2\text{/in} = 1.39 \text{ in}^2\text{/ft}$$

$$T_{lim} = \phi\sqrt{f'_c}\left(\frac{Acp^2}{Pcp}\right)$$ **[ACI 318-99 Art. 11.6.1]**

$Acp = bh = 36 \times 48 = 1728in^2$

$Pcp = 2(b + h) = 2(36 + 48) = 168$ in

$$T_{lim} = 185\sqrt{4000}\left(\frac{1728^2}{168}\right) \times \frac{1}{12000}$$

= 79.62 kft > 3.7 kft (Therefore, no torsional steel is needed.)

Check for minimum transverse reinforcement:

$$\left(\frac{A_v}{s}\right)_{min} = \frac{50b_w}{f_y} = \frac{50 \times 36}{60000} = 0.03 \text{ in}^2\text{/in}$$ **[AASHTO Std. 8.19.1.2]**

$= 0.03 \times 12 \text{ in}^2\text{/ft}$
$= 0.36 \text{ in}^2\text{/ft} < 1.39 \text{ in}^2\text{/ft}$

Maximum Spacing:

(i) $\frac{d}{2} = \frac{44.5}{2} = 22.25$ in
(ii) 24 in

Therefore, maximum spacing of 22.25 in governs.

Cracking and Fatigue Check at Midspan

Distribution of Flexural Reinforcement (AASHTO 8.16.8.4)

Service Moment	M_s	=	978.1 kft (Envelope Service)
Self Moment	M_{self}	=	58.2 kft (Self Load Case)
Dead Moment	M_d	=	495.2 kft (due to D1)
Live Moment, max	M_{Lmax}	=	167.2 kft (due to LL3)
Live Moment, min	M_{Lmin}	=	67.6 kft (due to LL4)

$$f_s = \frac{z}{(d_c A)^{1/3}} \le 0.6f_y$$ **[AASHTO Std. Eq. 8-6]**

$0.6 \text{ fy} = 0.6 \times 60 = 36$ ksi

Modular ratio, $n = \frac{29000}{3834} = 7.56$ **[AASHTO Std. 8.15.3.4]**

$$m = \frac{nA_s}{bd} = \frac{7.56 \times 7.9}{36 \times 44.5} = 0.0372$$

$$k = \sqrt{m^2 + 2m} - m$$

$$= \sqrt{(0.0372)^2 + 2 \times 0.0372} - 0.0372$$

$$= 0.24$$

$$j = +48\left(-\frac{k}{3}\right) = 1 - \frac{0.24}{3} = 0.$$

$$f_s = \frac{M_s}{A_s jd} = \frac{978.1 \times 12}{7.9 \times 0.92 \times 44.5} = 36.29 \text{ksi}$$

$$d_c = 2 + \frac{1}{2} = 2.5 \text{ in}$$

$$A = \frac{2 \times 2.5 \times 36}{10} = 18 \text{ in}^2$$

$$f_s = \frac{170}{(2.5 \times 18)^{1/3}} = 47.8 \text{ ksi} > 36 \text{ ksi, use } 36 \text{ ksi}$$

$$\text{Cracking Ratio} = \frac{36.29}{36} = 1.008$$

Fatigue Check (AASHTO 8.16.8.3)

The range between a maximum tensile stress and minimum stress in straight reinforcement caused by live load plus impact at service load shall not exceed:

$$f_f = 21 - 0.33 f_{min} + 8\left(\frac{r}{h}\right)$$

where:

$$\left(\frac{r}{h}\right) = 0.3$$

$$f_f = 23.4 - 0.33 f_{min}$$

Service moment stress due to dead load (permanent loads):

= 58.23 + 495.20 = 553.43 kft

Stress due to dead loads in bottom reinforcement:

$$f_s = \frac{553.43 \times 12}{7.9 \times 0.92 \times 44.5} = 20.53 \text{ ksi (tension)}$$

Maximum live load moment with impact:

$$= 167.20 \times 1.27 = 212.3 \text{ kft}$$

Service load stress due to maximum live load:

$$f_s = \frac{212.3 \times 12}{7.9 \times 0.92 \times 44.5} = 7.88 \text{ ksi (tension)}$$

Minimum live load moment with impact:

$$= 67.56 \times 1.27 = 85.80 \text{ kft}$$

Service load stress due to minimum live load moment:

$$f_s = \frac{85.80 \times 12}{7.9 \times 0.92 \times 44.5} = 3.18 \text{ ksi (tension)}$$

Fatigue Stress Range: 7.88 - 0 = 7.88 ksi

Minimum Stress Level:

$$f_f = 23.4 - 0.33 \times 23.71 = 15.58$$

$$\text{Fatigue Ratio} = \frac{7.88}{15.58} = 0.506$$

Column Design

Section Properties

Square Column:

Width of column = 36 in.
Depth of column = 36 in.
Height of column = 20 ft. (bottom of the column to the centerline of the cap)

Reinforcement

12 # 8 in X-direction

10 # 8 in Z-direction

Bar dist = 3.0 in.

Design Loads

At base of the column:

P_u = Factored axial load = 328.3 kips
M_{uz} = Transverse moment = 251.6 kft
M_{ux} = Longitudinal moment = 51.2 kft

At top of the column:

P_u = Factored axial load = 328.3 kips
M_{uz} = Transverse moment = 449.3 kft
M_{ux} = Longitudinal moment = 46.0 kft

Determine if slenderness needs to be considered

Consider the column braced in both X and Z directions.

Slenderness may be neglected if $\frac{kl}{r} < 34 - 12\left(\frac{M_1}{M_2}\right)$

where:

M_1 and M_2 are the smaller and larger end moments, respectively, and $\left(\frac{M_1}{M_2}\right)$ is positive for single curvature flexure.

$k_x = k_z = 1.0$

$r_x = r_z = 0.3(36) = 10.8$ in

Slenderness About The X-Axis

$$34 - 12\left(\frac{M_1}{M_2}\right) - 34 - 12\left(\frac{46}{51.2}\right) = 23.2$$

$$\frac{k_x l_x}{r_x} = \frac{1.0(2.0 \times 12)}{10.8} = 22.2 < 23.2$$

Therefore, neglect slenderness in the X-direction.

Slenderness About the Z-Axis

$$34 - 12\left(\frac{M_1}{M_2}\right) = 34 - 12\left(\frac{251.6}{449.3}\right) = 27.3$$

$$\frac{k_z l_z}{r_z} = \frac{1.0(240)}{10.8} = 22.2 < 27.3$$

Therefore, neglect slenderness in the Z-direction.

Interaction Diagram

The moment interaction diagram in RC-PIER as shown in Figure TH-5, is compared to that from PCACOL, as shown in Figure TH-4. The results for the bottom of the leftmost column is illustrated in Figure TH-5.

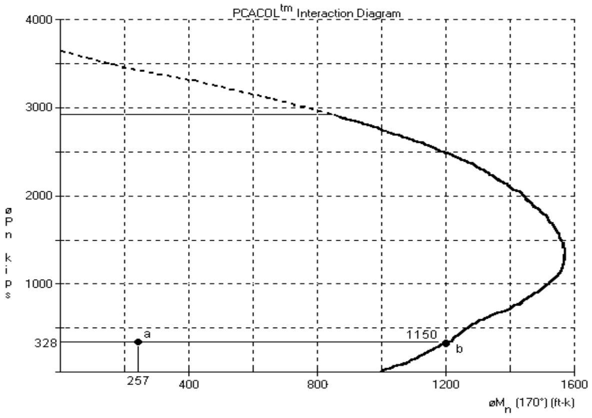


Figure B-4 PCACOL Interaction Diagram - Bottom Column

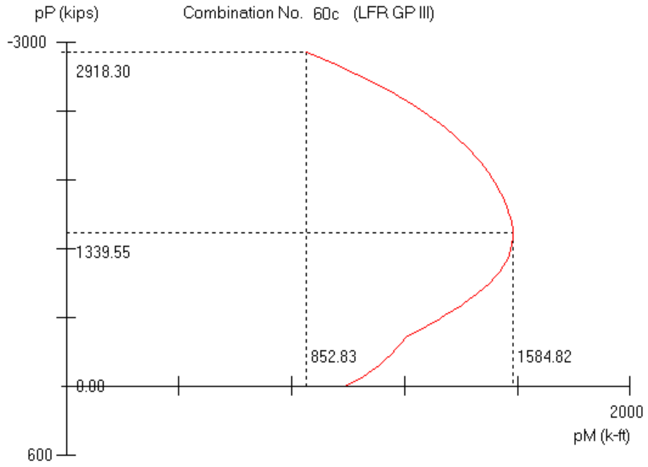


Figure B-5 Moment Interaction Diagram Column Component No. 1 at Bottom Location

Determine the Axial and Moment Capacity of Column

The results for the bottom of the leftmost column are used to show how to calculate the axial and moment capacity. First, locate the point representing the factored load and factored moment on the interaction diagram. Then, draw a horizontal line passing P_u = 331.9 kips, as shown in Figure TH-5, and a vertical line passing M, where

$$M_u = \sqrt{Mux^2 + Muz^2} = \sqrt{57.2^2 + 251.6^2} = 256.8 \quad kft$$

The intersection point (point a) is well within the boundary of the interaction diagram; therefore, the capacity of the column is adequate. Next, locate point b, as shown in Figure TH-4. Point b is the intersection of the horizontal line passing the factored load (Pu = 331.9 kips) with the moment interaction diagram. The corresponding load and moment values for point b are:

$\phi Pn = 328.3$ kips

$\phi Mn = 1150$ kft

Therefore,

$$\frac{\phi P_n}{P_u} = 1.0$$

$$\frac{\phi M_n}{M_u} = \frac{1150}{256.8} = 4.48$$

Check Minimum Area of Steel

$A_{s,min} > 0.01\ A_g$

$A_{s,min} > 0.01\ A_g = 0.01\ (36 \times 36) = 12.96\ in^2$

$A_{s,prov} = 18 - \#8 = 18(0.79\ in^2) = 14.22\ in^2 > A_{s,min}$ OK

Footing Design

Spread Footing

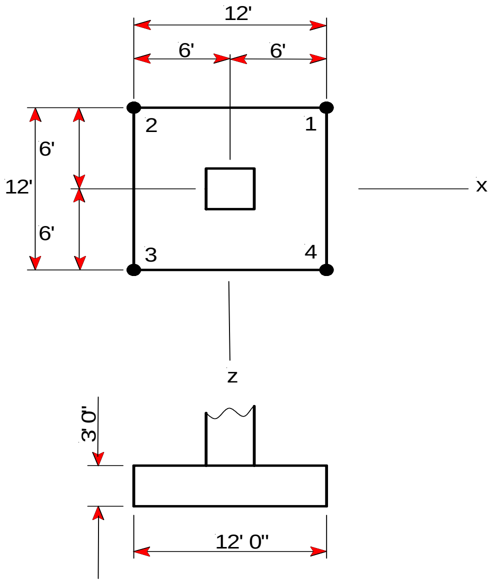


Figure B-6 Plan and Elevation of Isolated Spread Footing Design

Section Properties

A = Area of footing = (12)(12) = 144 ft^2

I_z = Moment of inertia $= \frac{1}{12}(12)(12)^3 = 1728\text{ ft}^4$

I_x = Moment of inertia $= \frac{1}{12}(12)(12)^3 = 1728\text{ ft}^4$

Determine Maximum Factored Soil Pressures

P_u = 432.44 kips

M_{uz} = 167.0 kft

M_{ux} = 0.00 kft

$$q_{1,4} = \frac{P_u}{A} - \frac{M_{uz}X}{I_z} = \frac{432.44}{144} - \frac{167.0(6)}{1728} + 1.30(0.20 + 0.45) = 3.27\text{ ksf}$$

$$q_{2,3} = \frac{P_u}{A} + \frac{M_{uz}X}{I_z} = \frac{432.44}{144} + \frac{167.0(6)}{1728} + 1.30(0.20 + 0.45) = 4.43\text{ ksf}$$

Max Soil Pressure for Design

$$q_{max} = \frac{P_u}{A} + \frac{M_{uz}x}{I_z} = \frac{432.44}{144} + \frac{167.0(6)}{172.8}$$

= 3.58 ksf

Flexure Design

X-Dir

Loc. = ± 1.5 ft.

d = 36 – 3 clr. – 0.5(1.0) = 32.5 in

$$M_u = \frac{wl^2}{2} = \frac{3.58x12(6 - 1.5)^2}{2} = 435\text{ kft}$$

Determine the Required Area of Steel

$$R_n = \frac{M_u}{\phi b d_s^2} = \frac{435(12000)}{0.9(144)(32.5)^2}$$

= 38.1 psi

$$\rho = \frac{0.85f_c'}{f_y}\left(1 - \sqrt{1 - \frac{2R_n}{0.85f_c'}}\right) = \frac{0.85(4)}{60}\left(1 - \sqrt{1 - \frac{2(38.1)}{0.85(4000)}}\right)$$

$= 0.0006386$

$A_{s,req} = \rho bd = 0.0006386(144)(32.5) = 2.99\ \text{in}^2$

Provide 10 - #8 bars ($A_s = 10 \times 0.79 = 7.90\ \text{in}^2$)

Check Moment of Resistance

$$\rho = \frac{10 \times 0.79}{12 \times 12 \times 32.5} = 0.001688$$

$M_r = \phi M_n$

$$= 0.9\left[7.9 \times 60 \times 32.5\left(1 - 0.6\frac{0.001688 \times 60}{4}\right)\right] \times \frac{1}{12}$$

$= 1137.8\ \text{kft} > 438\ \text{kft}$

Check Minimum Reinforcement

$\phi M_n \geq M_{n\text{-}min}$

where $M_{n\text{-}min}$ = smaller of 1.2 M_{cr} and 1.33 M_u

$$M_{cr} = 7.5\sqrt{f_c'} \times S_t$$

S_t = section modulus for the extreme top fiber of the non-composite concrete section

$$= \frac{I_g}{y_t}$$

where:

$$I_g = \frac{1}{12} \times 12 \times 12 \times 36^3 = 559872\ \text{in}^4$$

$y_t = 18\ \text{in}$

$$S_t = \frac{559872}{18} = 31104\ \text{in}^3$$

$$M_{cr} = \left(7.5\sqrt{4000} \times 31104\right)\frac{1}{12000} = 1229.5\ \text{kft}$$

$1.2\ M_{cr} = 1.2 \times 1229.5 = 1475.4\ \text{kft}$

$1.33\ M_u = 1.33 \times 435.0 = 578.6\ \text{kft}$

Since $1.33\ M_u < 1.2\ M_{cr}$

$M_{n\text{-}min} = 578.6\ \text{kft}$

$A_{s\text{-}min\text{-}req}$= 4.0 in^2 for $M_{n\text{-}min}$ = 578.6 kft

$A_{s\text{-}req}$= 4.00 $in^2 < 7.9\ in^2$ OK

Crack Control

X-Dir:

$$f_{sa} = \frac{z}{\sqrt[3]{d_c A}} \le 0.6 f_y$$ **[AASHTO Std. 8.16.8.4]**

where:

z = 170 k/in

d_c = 2.0 max. + 0.5(1.0) = 2.5 in

A = 2(2.5)(144 in/10 bars) = 72 in^2

$$f_{sa} = \frac{170}{\sqrt[3]{(2.5)(72)}} = 30\text{ ksi} < 0.6(60) = 36\text{ ksi}$$

Therefore, use f_{sa} = 30.1 ksi

$$f_s = \frac{M_s \times 12}{A_s jd}$$

$$n = \frac{E_s}{E_c} = \frac{29000}{3834.3} = 7.56$$ **[AASHTO Std. 8.15.3.4]**

$$m = \frac{nA_s}{bd} = \frac{7.56(7.9)}{(144)(32.5)} = 0.0128$$

$$k = \sqrt{m^2 + 2m} - m = \sqrt{0.0128^2 + 2(0.0128)} - 0.0128$$

$$j = 1 - \frac{k}{3} = 1 - \frac{0.148}{3} = 0.95$$

Service Load Moment

q = 3.35 – 0.2 – 0.45 = 2.7 ksf

$$M = \frac{wl^2}{2} = \frac{2.7 \times 12(6 - 1.5)^2}{2} = 328.05\text{ kft}$$

$$f_s = \frac{328.05(12)}{7.9(0.95)(32.5)} = 16.14\text{ ksi}$$

f_s = 16.14 < f_{sa} = 30.1 ksi

$$\text{Ratio} = \frac{16.14}{30.1} = 0.53 < 1.0$$ OK

Fatigue Check

$$f_f = 21 - 0.33f_{min} + 8.0\left(\frac{r}{h}\right)$$ **[AASHTO Std. 8.16.8.3]**

$= 21 - 0.33f_{min} + 8.0(0.3)$

$= 23.4 - 0.33f_{min}$

Maximum Soil Pressure Due to Live Load:

$P_{Lmax} = 53.25$ kips

$M_{Lmax} = 13.29$ kft

$$q = \frac{53.25}{144} + \frac{13.29(6)}{1728} = 0.416 \text{ ksf}$$

$$M_r = \frac{\omega l^2}{2} = \frac{(0.416 \times 12)(6 - 1.5)^2}{2} = 50.5 \text{ kft}$$

Minimum Soil Pressure Due to Live Load:

$P_{lmin} = 9.417$ kips

$M_{lmin} = 12.97$ kft

$$q = \frac{9.417}{144} - \frac{12.97(6)}{1728} = 0.020 \text{ ksf}$$

$$M_r = \frac{\omega l^2}{2} = \frac{(0.020 \times 12)(6 - 1.5)^2}{2} = 2.43 \text{ kft}$$

Minimum Permanent Load Moment:

$P_{dl} = 47.7 + 167.8 = 215.5$ kips

$M_{dl} = 11.76 + 87.45 = 99.21$ kft

$$q_1 = \frac{215.5}{144} + \frac{99.21(6)}{1728} = 1.84 \text{ ksf}$$

$$M_{dl} = \frac{\omega l^2}{2} = \frac{(1.84 \times 12)(6 - 1.5)^2}{2} = 223.7 \text{ kft}$$

X-Dir:

Service Load Stress Range Due to the Maximum Live Load:

$$f_s = \frac{M_s \times 12}{A_s jd} = \frac{50.5(12)}{7.9(0.95)(32.5)} = 2.48 \text{ ksi (tension)}$$

Service Load Stress Due to Minimum Live Load:

$$f_s = \frac{2.43(12)}{7.9(0.95)(32.5)} = 0.120 \text{ ksi (tension)}$$

Service Load Stress Due to Permanent Loads:

$$f_s = \frac{223.7(12)}{7.9(0.95)(32.5)} = 11.005 \text{ ksi (tension)}$$

Live Load Stress Range = 2.48 – 0 = 2.48 ksi

Minimum Stress Level:

f_{min} = 11.005 + 0.120 = 11.125 ksi

Allowable Fatigue Stress Range:

f_f = 23.4 – 0.33(11.125) = 19.73 ksi

$\text{Ratio} = \frac{2.48}{19.73} = 0.13 < 1.0$ OK

One Way Shear Design

X-Dir

[AASHTO Std. 8.16.6.6.1(a)]

$V_u \leq \phi V_n$ **[AASHTO Std. 8.16.6.1]**

d = 32.5 in (Critical section location from column centerline)

distance = 1.5 + 32.5/12 = 4.21 ft

q_{max} = 3.58 ksf

V_u at 4.21 from column centerline is

= 3.58 ksf × 12 × (6 – 4.21)
= 76.9 kips

$\phi V_c = \phi 2\sqrt{f'_c}bd$ **[AASHTO Std. 8.16.6.2]**

$$\phi V_c = 0.85(2)\sqrt{4000}(144)(31.5)/10^3 = 503.2 \text{ kips}$$

ϕV_c = 503.2 kips = $\phi V_n > V_u$ = 76.9 kips

Two Way Shear Design

[AASHTO Std. 8.16.6.6]

$$d_{avg} = \frac{d_1 + d_2}{2} = \frac{31.5 + 32.5}{2} = 32.0$$

$$b_o = \frac{4(36 + 32)}{12} = 22.67 \text{ ft}$$

$$A_o = (36 + 32)^2 / 144 = 32.11 \text{ ft}^2$$

$$V_u = 3.58 \text{ ksf} \times (144 - 32.11) = 401.4 \text{ kips}$$

$$\phi V_c = \phi 4\sqrt{f_c'}bd$$

$$\phi V_c = \frac{0.85(4)\sqrt{4000}(22.67 \times 12)(32)}{10^3} = 1871.9 \text{ kips}$$

$$\phi V_c = 1871.9 \text{ kips} > V_u = 401.4 \text{ kips}$$

Footing Design

Pile Cap Forces

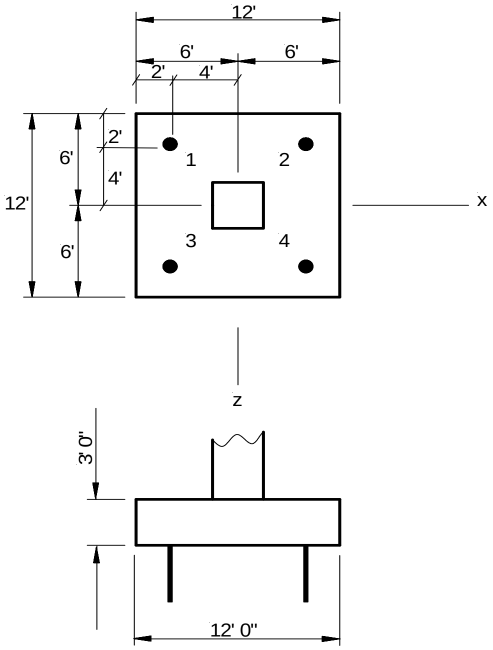


Figure B-7 Plan and Elevation of Isolated Pile Cap Footing Design

Pile Reactions

For symmetric pile pattern with pile group c.g. concentric with column,

$$P_p = \frac{P}{n} \pm \frac{M_z X_i}{\Sigma X_i^2} \pm \frac{M_x Z_i}{\Sigma Z_i^2}$$

Pile Reactions, Service

Service loads:

$$P = 321.99 + (0.2 + 0.45) \times 144 = 415.59 \text{ kips}$$

$$M_x = 0 \text{ kft}$$

$$M_z = 125.81 \text{ kft}$$

$$P_{1,3} = \frac{415.59}{4} + \frac{125.81 \times 4}{4(4)^2} =111.76 \text{ kips}$$

$$P_{2,4} = \frac{415.59}{4} - \frac{125.81 \times 4}{4(4)^2} = 96.03 \text{ kips}$$

Pile Reactions, Factored:

Factored loads:

$$P = 432.44 \text{ kips} + 1.3(0.2 + 0.45) \times 144 = 554.12 \text{ kips}$$

$$M_x = 0 \text{ kft}$$

$$M_z = 167.01 \text{ kft}$$

$$P_{1,3} = \frac{554.12}{4} + \frac{167.01 \times 4}{4(4)^2} = 148.97 \text{ kips}$$

$$P_{2,4} = \frac{554.12}{4} - \frac{167.01 \times 4}{4(4)^2} = 128.09 \text{ kips}$$

Tutorial 2: Multiple Column Pier (AASHTO LRFD)

This design example demonstrates the design of a multiple column pier with a tapered cap beam, as shown in Figure TH-8. This example demonstrates the design of the cap beam, columns, rectangular combined spread footing under columns 1 and 2, and a rectangular combined pile cap footing under columns 3 and 4. The design will be carried out in accordance with *AASHTO LRFD Bridge Design Specifications*, 3rd Edition, 2004.

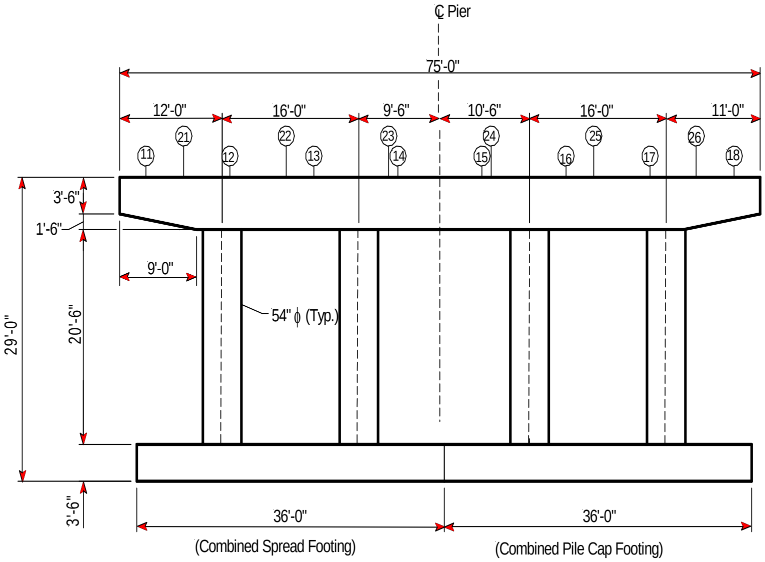
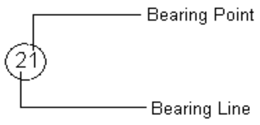


Figure B-8 Longitudinal Pier Elevation (Tutorial 2)

Legend:



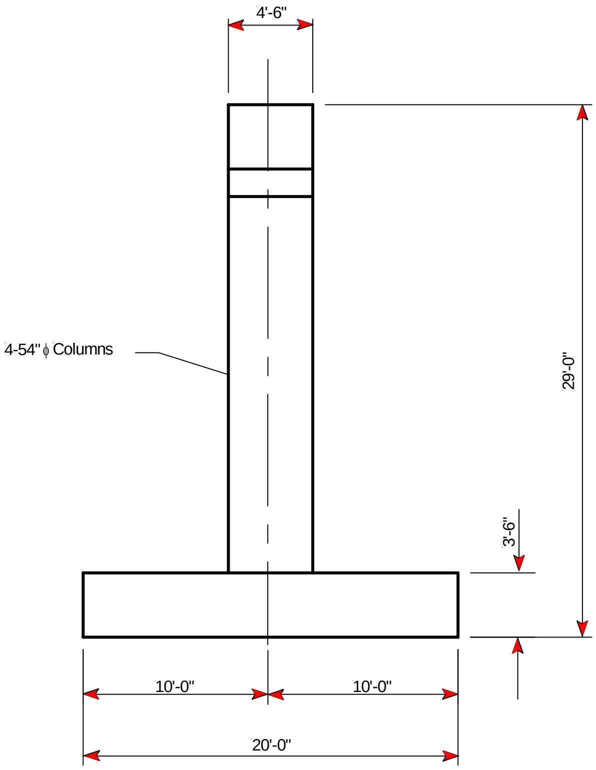


Figure B-9 Transverse Pier Elevation (Tutorial 2)

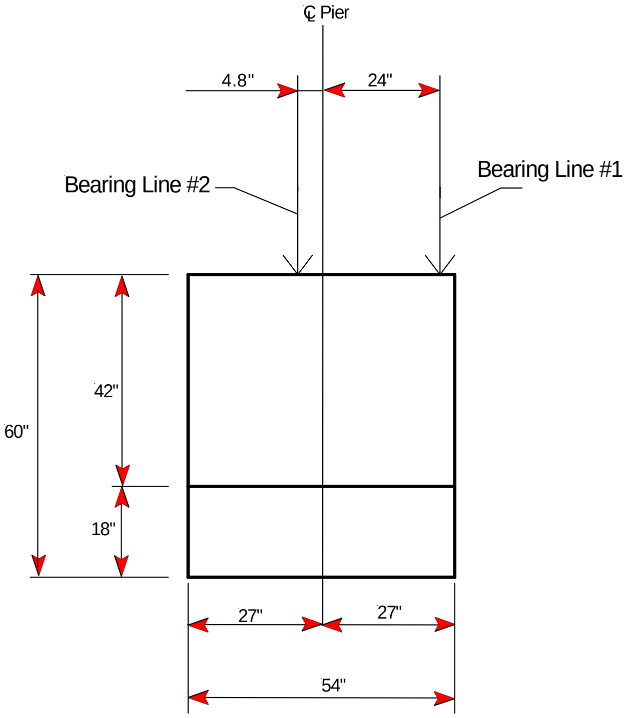


Figure B-10 Section of Cap Beam (Tutorial 2)

Materials

Concrete Strength

Cap	f’c = 4 ksi
Columns	f’c = 4 ksi
Footings	f’c = 4 ksi

Concrete Density

Cap	ρ	= 150 pcf
Columns	ρ	= 150 pcf
Footings	ρ	= 150 pcf

Steel Yield Strength

Cap	f_y	= 60 ksi
Columns	f_y	= 60 ksi
Footings	f_y	= 60 ksi

Concrete Type

Cap	Normal
Columns	Normal
Footings	Normal

Bearing Location

Bearing Line #1		**Bearing Line #2**	
Bearing Point	Distance from left edge of pier cap (ft)	Bearing Point	Distance from left edge of pier cap (ft)
1	3.06	1	7.50
2	12.90	2	19.50
3	22.74	3	31.50
4	32.58	4	43.50
5	42.42	5	55.50
6	52.26	6	67.50
7	62.10		
8	71.94		

Loads

Dead Load

Bearing Loads

Bearing Line 1 P = -300 kips (all bearings)
Bearing Line 2 P = -100 kips (all bearings)

Vehicular Live Load

Bearing Loads (LL1)

Line 1

Bearing 1 P = -20 kips

Bearing 2 P = -7 kips

Line 2

Bearing 1 P = -45 kips
Bearing 2 P = 6 kips

Bearing Loads (LL2)

Line 1

Bearing 7 P = -10 kips
Bearing 8 P = -29 kips

Line 2

Bearing 5 P = 4 kips
Bearing 6 P = -30 kips

Bearing Loads (LL3)

Line 1

Bearing 2 P = -43 kips
Bearing 3 P = -19 kips

Bearing Loads (LL4)

Line 1

Bearing 6 P = -19 kips
Bearing 7 P = -43 kips

Wind on Structure

Bearing Loads

Line 1

Bearing 1 P = 35 kips
Bearing 8 P = -35 kips

Line 2

Bearing 1 P = 42 kips
Bearing 6 P = -42 kips

Cap Loads

X dir: P = 40 kips

Z dir:

UDL $\omega = 0.4667$ klf

Creep

Creep in the Cap Beam $\varepsilon_{sh} = 0.0003$

Shrinkage

Shrinkage in the Cap Beam $\varepsilon_{sh} = -0.00027$

Footing Surcharge

Footing $\sigma = 0.200$ ksf

Load Combinations

Strength Group I

Q = 1.00 (1.25 max./0.90 min. DC + 1.35 EV + 1.75 (LL+I)
+ 0.50/1.20 (CR/SH))

Strength Group III

Q = 1.00 (1.25 max./0.90 min. DC + 1.35 EV + 1.40 WS
+ 0.50/1.20 (CR/SH))

Service Group I

Q = 1.00 (1.00 DC + 1.00 EV + 1.00 (LL+I) + 0.30 WS
+ 1.00/1.20 (CR/SH))

Service Group III

Q = 1.00 (1.00 DC + 1.00 EV + 0.80 (LL+I) + 1.00/1.20 (CR/SH))

Fatigue

Q= 1.00(0.75(LL+I))

Dynamic Allowance Factor

IM = 33%

Cap Beam Flexure Design for Maximum Positive Moment

At 42.4 ft (Abs.Loc.) from cap left end (inside span #3)

Service Moment	M_s	=	899.84 kft
Factored Moment	M_u	=	1145.5 kft
Self Moment	M_{self}	=	43.37 kft
Dead Moment	M_d	=	835.71 kft
Shrinkage Moment	M_{sh}	=	27.89
Creep Moment	M_{CR}	=	-30.99
Fatigue Moment max	M_{ftgmax}	=	8.72 kft
Fatigue Moment min	M_{ftgmin}	=	-17.39 kft

Section Geometry Properties

Width	b	=	54 in
Overall depth	h	=	60 in
Minimum bar distance		=	3.5 in
Longitudinal reinforcement		=	#11 bars
Trans. reinforcement (shear)		=	#4 bars
Effective Depth	d	=	60 – 3.5 = 56.5 in
Strength Reduction Factor	ϕ	=	0.9 for flexure

Flexure Design

$M_u \leq \phi\, M_n$

$M_u = 1145.5$ kft

$$\text{Required } R_u = \frac{M_u}{\phi b d^2} = \frac{1145.5 \times 12 \times 1000}{0.9 \times 54 \times (56.5)^2} = 88.60 \text{ psi}$$

$$\rho = \frac{0.85 f_c}{f_y} = \left(1 - \sqrt{1 - \frac{2R}{0.85 f_c{}'}}\right) = \frac{0.85 \times 4}{60}\left(1 - \sqrt{1 - \frac{2 \times 88.60}{0.85 \times 4000}}\right)$$

$= 0.0015$

$A_s = 0.0015 \times 54 \times 56.5 = 4.65 \text{ in}^2$

Provide 6 – #11 bars = 9.36 in^2

$$\text{Spacing of reinforcement} = \frac{54 - 2 \times 2 - 6 \times 1.41}{(6 - 1)} = 8.31 \text{ in}$$

Check Moment of Resistance

$$c = \frac{9.36 \times 60}{0.85 \times 0.85 \times 4 \times 54} = 3.6 \text{ in}$$

$a = 0.85 \times 3.6 = 3.06$ in

$$M_r = \phi M_n = 0.9\left[9.36 \times 60\left(56.5 - \frac{3.06}{2}\right)\right] \times \frac{1}{12}$$

$= 2315 \text{ kft} > 1145.5 \text{ kft}$

Check Minimum Reinforcement

$\phi M_n \geq M_{n\text{-min}}$

where:

$M_{n\text{-min}}$ = smaller of 1.2 M_{cr} and 1.33 M_u

$M_{cr} = 0.24\sqrt{f'}\, c \times S_t$

S_t = section modulus for the extreme top fiber of non-composite concrete section

$$= \frac{I_g}{y_t}$$

where:

$$I_g = \frac{1}{12} \times 54 \times 60^3 = 972{,}000 \text{ in}^4$$

$$y_t = \frac{60}{2} = 30 \text{ in}$$

$$S_t = \frac{972000}{30} = 32400 \text{ in}^3$$

$$M_{cr} = (0.24\sqrt{4} \times 32400)\frac{1}{12} = 1296 \text{ kft}$$

$1.2M_{cr} = 1.2 \times 1296 = 1555.2$ kft

$1.33M_u = 1.33 \times 1145.5 = 1523.5$ kft ← controls

Mn-min = 1523.5k-ft ≤ M_{cr} = 2315 kft

Check Maximum Reinforcement

0.42d = 0.42 x 56.5 = 23.73 in > 3.6 in

Distribution of Flexural Reinforcement

$$f_s = \frac{z}{(d_c A)^{1/3}} \le 0.6 f_y$$

$0.6\ f_y = 0.6 \times 60 = 36$ ksi

$$\text{Modular ratio } n = \frac{29000}{3834} = 7.56$$

$$m = \frac{nA_s}{bd} = \frac{7.56 \times 9.36}{54 \times 56.5} = 0.0232$$

$$k = \sqrt{m^2 + 2m} - m = \sqrt{(0.0232)^2 + 2 \times 0.0232} - 0.0232 = 0.193$$

$$j = 1 - \frac{k}{3} = 1 - \frac{0.193}{3} = 0.936$$

$$f_s = \frac{M_s}{A_s jd} = \frac{899.8 \times 12}{9.36 \times 0.936 \times 56.5} = 21.81 \text{ ksi}$$

$$d_c = 2 + \frac{1.41}{2} = 2.705 \text{ in.}$$

$$A = \frac{2 \times 2.705 \times 54}{6} = 48.69 \text{ in}^2$$

$$f_s = \frac{170}{(2.705 \times 48.69)^{1/3}} = 33.4 \text{ ksi} < 36 \text{ ksi, use } 33.4 \text{ ksi}$$

$$\text{Cracking Ratio} = \frac{21.81}{33.4} = 0.65$$

Design for Fatigue

The range between a maximum tensile stress and minimum stress in straight reinforcement caused by live load plus impact at service load shall not exceed:

$$f_f = 21 - 0.33f_{min} + 8\left(\frac{r}{h}\right)$$

where:

$$\left(\frac{r}{h}\right) = 0.3$$

f_f = 23.4 – 0.33 f_{min}

Service Moment due to permanent loads

= 43.37 + 835.71 = 879.08 kft

Service Moment due to permanent loads, creep, and shrinkage

= 43.37 + 835.71 + 27.89 – 30.99
= 875.98 kft

Service Moment due to permanent loads only governs.

Stress Due to Permanent Loads

$$= \frac{879.69 \times 12}{9.36 \times 0.936 \times 56.5} = 21.33 \text{ (tension)}$$

Service Stress in Bottom Reinforcement due to Maximum Fatigue Moment = 8.72 kft

$$f_1 = \frac{8.72 \times 12}{9.36 \times 0.936 \times 56.5} = 0.21 \text{ ksi (tension)}$$

Service Stress in Bottom Reinforcement due to Minimum Fatigue Moment = -17.39 kft

f_2 = -0.11 ksi (compression)

Fatigue Stress Range = 0.21 – (-0.11) = 0.32

Minimum Stress Level:

f_{min} = 21.33 + (-0.11) = 21.44

Allowable Fatigue Stress Range:

f_f = 23.4 – 0.33 × 21.44 = 16.32

$$\text{Fatigue Ratio} = \frac{0.32}{16.32} = 0.020$$

Cap Shear and Torsion Design (Outside Face of First Support)

12 ft from left end of the beam (inside span #1)

T_u = Maximum factored torsional moment

= 774.5 kft

V_u = Maximum factored shear force

= 696.2 kips

M_u = Maximum factored moment that occurs simultaneously with $\text{Max}T_{u \text{ or}}$ $\text{Max}V_u$

= 5056.37 kft

Section Properties

bv = width of beam = 54 in. (used for shear)

h = depth of beam = 60 in.

Longitudinal Reinforcement = #11 bars at 3.5 in

d_e = 60-3.5 = 56.5in

Resistance factor ϕ = 0.9 for shear and torsion

Check If Torsion May Be Neglected [LRFD Art. 5.8.2]

Torsion may be neglected if $T_u < T_{lim=}0.25\phi T_{cr}$

$$T_{cr} = 0.125\sqrt{f_c'}\frac{A_{cp}^2}{P_c}\sqrt{1+\frac{f_{pc}}{0.125\sqrt{f_c'}}}$$ **[LRFD Eq. 5.8.2.1-4]**

where:

ϕ = 0.9

f_c' = Specified strength of concrete, ksi

f_{pc} = Compressive stress in concrete after prestress losses have occurred, ksi

A_{cp} = total area enclosed by outside perimeter of concrete cross-section, in^2

= 54 × 60

= 3240 in^2

p_c = the length of the outside perimeter of the concrete section, in

= 2 × (54 + 60)

= 228 in

$$T_{cr} = \left[\frac{0.125\sqrt{4}\frac{3240^2}{228}\sqrt{1+\frac{0}{0.125\sqrt{4}}}}{12}\right] = 959.2 \text{ kft}$$

$\phi T_{cr} = 0.9(959.2) = 863.3$ kft

$T_u = 934.2$ kft $> T_{lim}$

where:

$T_{lim} = 0.25 \times 863.3 = 215.83$ kft

Therefore, torsion must be considered.

Determine Required Area of Stirrups for Torsion

$\phi T_n \geq T_u$

$$T_n = \frac{2A_\circ A_t f_y}{s}\cot\theta$$ **[LRFD Eq. 5.8.3.6.2-1]**

$A_o = 0.85A_{oh}$

$A_{oh} = [(60 - 2(3.5 - 0.5\ (1.41) - 0.5\ (0.625))) \times (54 - 2\ (2 + 0.5\ (0.625)))]$

$= 2718\ in^2$

$A_o = 0.85(2718) = 2310\ in^2$

$$\frac{A_t}{s} = \frac{T_u}{2\phi A_o f_y \cot\theta} = \frac{774.5(12000)}{2(0.9)(2310)(60000)(1.0)}$$

$= 0.037\ in^2/in/leg$

$= 0.45\ in^2/ft/leg$

$$\frac{2At}{S} = 0.89\ in^2/ft$$

Calculate Required Area of Stirrups for Shear

$$V_c = 0.0316\beta\sqrt{f_c'}b_v d_v$$ **[LRFD Eq. 5.8.3.3-3]**

where:

$$d_v = d_e - \frac{a}{2}$$

where:

$$a = \frac{A_s f_y}{0.85 f_c' b} = \frac{21.84(60)}{0.85(4)(54)} = 7.14\ in.$$

$$d_v = 56.5 - \frac{7.14}{2} = 52.93\ in$$

$$\phi V_c = (0.9)(0.0316)(2)\sqrt{4}(54)(52.93) = 325.2\ kips$$

$$V_{s,_{req}} = \frac{V_u}{\phi} - Vc = \frac{696.2}{0.9} - \frac{325.2}{0.9} = 412.2 \text{ kips}$$

$$\frac{A_v}{S} = \frac{V_s}{f_y d_v} = \frac{412.2}{60(52.93)} = 0.13 \text{ in}^2\text{/in=1.55 in}^2\text{/ft}$$

Determine Combined Shear and Torsion Stirrup Requirements

$$\frac{A_v}{S} + 2 \times \frac{A_t}{S} = 1.55 + 0.89 = 2.44 \text{ in}^2\text{/ft}$$

Check Minimum Stirrup Area

$$A_v = 0.0316\sqrt{f'_c}\frac{b_v S}{f_y}$$ **[LRFD Eq. 5.8.2.5-1]**

$$\frac{A_v}{S} = 0.0316\sqrt{4}\frac{54}{60} = 0.05688 \times 12 = 0.68 \text{ in}^2\text{/ft} < 2.44 \text{ in}^2\text{/ft}$$

Check Maximum Stirrup Spacing

$$0.1f'_c b_v d_v = 0.1(4)(54)(52.93) = 1143.3 \text{ kips} > V_u$$ **[LRFD Eq. 5.8.2.7-1]**

Therefore, $S \leq 0.8d_v \leq 24$ in

$S \leq 0.8(52.93) = 42.34$ in ≥ 24 in

Therefore, $S_{max} = 24$ in

Provided Stirrup Capacity

Provided, 4 Legged #5 @ 6" c/c

Total Stirrup Area Provided

$$\frac{Aprv}{S} = 2.48 \text{ in}^2\text{/ft}$$

Provided Stirrup Area for Shear

$$\left(\frac{Av}{S}\right)_{prv} = 2.48 - 0.89 = 1.59 \text{ in}^2$$

Provided Stirrup Shear Capacity

$$V_s = \left(\frac{Av}{S}\right) f_y \, dv$$

$= (1.59/_{12}) \times 60 \times 52.93$

= 420.79 kips

Calculate Longitudinal Reinforcement

$$A_s f_y + A_{ps} f_{ps} \geq \frac{M_u}{\phi d_v} + \frac{0.5N_u}{\phi} + \cot\theta \sqrt{\left(\frac{V_u}{\phi} - 0.5V_s - V_p\right)^2 + \left(\frac{0.45 p_h T_u}{2A_o \phi}\right)^2}$$

p_h = 2[(60 – 2 (3.5) + 2 × 0.5 × (1.41) + 2 × 0.5(0.625)) + (54 – 2(2) – 2 × 0.5(0.625))]
= 208.84 in

$$A_s f_y + 0 \geq \frac{5056.37(12)}{0.9(52.93)} + 0 + 1\sqrt{\left(\frac{696.2}{0.9} - 0.5(420.79) - 0\right)^2 + \left(\frac{0.45(208.84)(774.5 \times 12)}{2(2310)(0.9)}\right)^2}$$

$A_s f_y \geq 1875$ kip

Al = A_s = 31.25 in^2

Alx = Al – A_{pvr} = 31.25 – 21.84 = 9.41 in^2 for top

Column Design: Column #1 (Left Column)

Section Properties

dia = diameter of column = 54 in.

h = height of column = 23.75 ft

Reinforcement

19 – #10

Bar dist = 3.13 in

Loads

0 ft.(Bottom):

P_u = Factored axial load = 1149.4 kips
M_{uz} = Transverse moment = 525.7 kft
M_{ux} = Longitudinal moment = 1705.8 kft

23.8 ft. (Top):

P_u = Factored axial load = 1453.4 kips
M_{uz} = Transverse moment = 1150.2 kft
M_{ux} = Longitudinal moment = 1547.5 kft

Determine If Slenderness Needs To Be Considered [LRFD Art. 5.7.4.3]

Consider the column braced in both the X and Z directions.

Slenderness may be neglected if $\frac{kl}{r} < 34 - 12\left(\frac{M_1}{M_2}\right)$

where:

M_1 and M_2 are the smaller and larger end moments, respectively, and the term $\left(\frac{M_1}{M_2}\right)$ is positive for single curvature flexure.

$k_x = k_z = 1.0$

$r_x = r_z = 0.25(54) = 13.5$ in

$l_x = l_z = 23.75$ ft × 12= 285 in

X-Dir:

$$34 - 12\left(\frac{M_1}{M_2}\right) = 34 - 12\left(\frac{1547.5}{1705.8}\right) = 23.1$$

$$\frac{k_x l_x}{r_x} = \frac{1.0(285)}{13.5} = 21.1 < 23.1$$

Therefore, neglect slenderness in the X-direction.

Z-Dir:

$$34 - 12\left(\frac{M_1}{M_2}\right) = 34 - 12\left(\frac{525.7}{1150.2}\right) = 28.52$$

$$\frac{k_x l_x}{r_x} = \frac{1.0(285)}{13.5} = 21.1 < 28.52$$

Therefore, neglect slenderness in the Z-direction.

Interaction Diagram

Moment interaction diagram from RC-PIER is compared to that from PCACOL, as shown in Figure TH-11. The result for the bottom of the leftmost column from RC-PIER is shown in Figure TH-12.

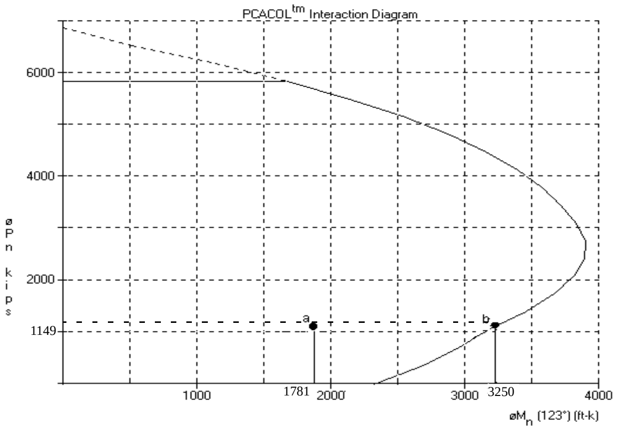


Figure B-11 PCACOL Interaction Diagram

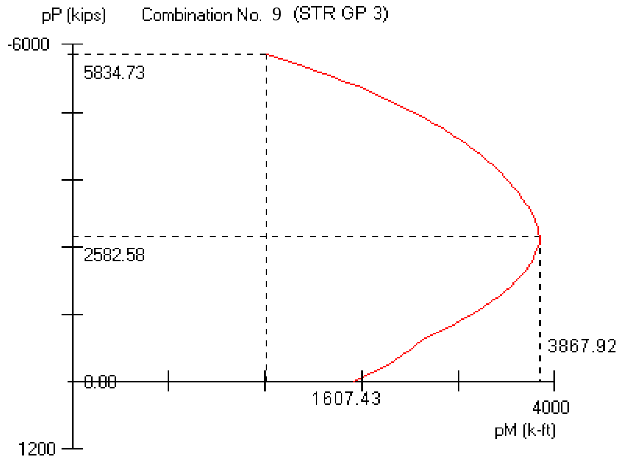


Figure B-12 Column Component #1 at Bottom Location

Determine the Axial and Moment Capacity of the Column

Using the results for the bottom of the leftmost column as an example, we will show you how to calculate the axial and moment capacity. The factored load P_u = 1149.4 kips and results of factored moment is

$$M_u = \sqrt{Mux^2 + Muz^2} = \sqrt{1705.8^2 + 525.7^2} = 1785\,kft$$

Now, draw a horizontal line passing P_u = 1149.4 kips and a vertical line passing M = 1785.7 kft, as shown in Figure TH-11. The intersection point (point a) is shown to be within the boundary of the interaction diagram; therefore, the capacity of the column is adequate. Next, locate point b, as shown in Figure TH-11. Point b is the intersection of the horizontal line passing the factored load (P = 1149.4 kips) with the moment interaction diagram. The corresponding load and moment values for point b are:

ϕP_n = 1149.4 kips

ϕM_n = 3250 kft

Therefore,

$$\frac{\phi P_n}{P_u} = 1.0$$

$$\frac{\phi M_n}{M_u} = \frac{3250}{1785} = 1.82 > 1.0$$ OK

Check Minimum Area of Steel

For non-prestressed components:

$$\frac{A_s f_y}{A_g f'_c} \geq 0.135$$ **[LRFD Eq. 5.7.4.2-3]**

$$\frac{A_s f_y}{A_g f'_c} = \frac{19(1.27)(60)}{(27)^2(3.14)(4)} = 0.158 > 0.135$$ OK

Footing Design - Spread

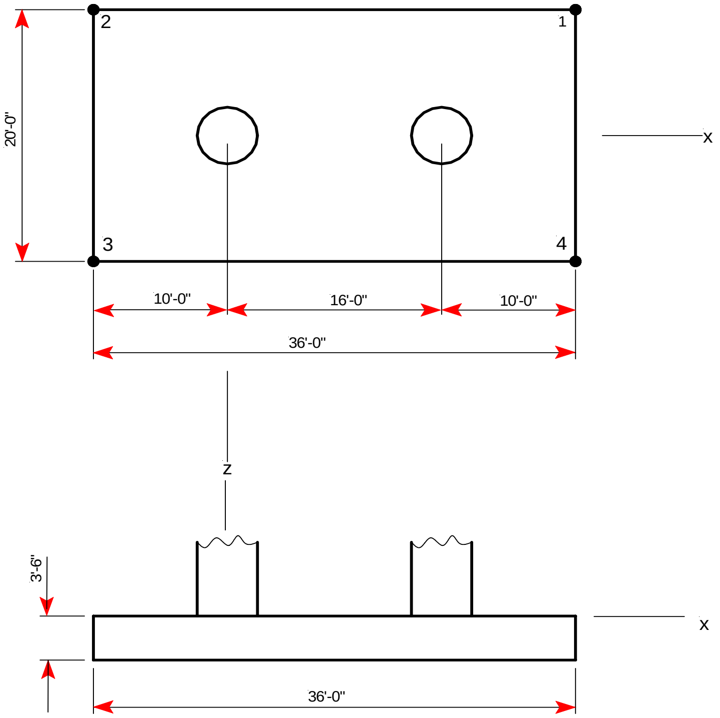


Figure B-13 Plan and Elevation of Combined Spread Footing (Tutorial 2)

Section Properties

A = Area of footing = (20)(36) = 720 ft^2

I_z = Moment of inertia = $\frac{1}{12}(20)(36)^3$ = 77760 ft^4

I_x = Moment of inertia = $\frac{1}{12}(36)(20)^3$ = 24000 ft^4

S_z = Section modulus = $\frac{77760}{18}$ = 4320 ft^3

S_x = Section modulus = $\frac{24000}{10}$ = 2400 ft^3

Loads

Self-weight = 0.15 kcf × 3.5 ft = 0.525 ksf

Surcharge = 0.20 ksf

Determine the Maximum Service Soil Pressure

Corner 3 for Load Combination 17:

Column #1:

P = Service axial load = 1174.30 kips
M_z = Service transverse moment = -288.53 kft
M_x = Service longitudinal moment = 1084.81 kft

Column #2:

P = Service axial load = 658.6 kips
M_z = Service transverse moment = 266.65 kft
M_x = Service longitudinal moment = 1112.56 kft

ΣP = 1174.30 + 658.6 = 1832.9 kips

ΣM_z = 1174.30 (8) – 658.6 (8) + 266.65 – 288.53 = 4103.72 kft

ΣM_x = 1084.81 + 1112.56 = 2197.37 kft

The maximum service soil pressure occurs at corner 3 is

$$q_3 = \frac{P}{A} + \frac{M_z}{S_z} + \frac{M_x}{S_x}$$

$$= \frac{1832.9}{720} + \frac{4103.72}{4320} + \frac{2197.37}{2400}$$

= 4.42 ksf

q_3 = 4.42 + 0.525 + 0.20 = 5.14 ksf

Determine the Maximum Factored Soil Pressure

Corner 3 for Load Combination 1:

Column #1:

P_u = Factored axial load = 1463.98 kips
M_{uz} = Factored transverse moment = -474.07 kft
M_{ux} = Factored longitudinal moment = 1439.85 kft

Column #2:

P_u = Factored axial load = 828.87 kips
M_{uz} = Factored transverse moment = 250.36 kft
M_{ux} = Factored longitudinal moment = 1470.47 kft

$\Sigma P_u = 1463.98 + 828.87 = 2292.85$ kips

$\Sigma M_{uz} = 1463.98\ (8) - 828.87\ (8) + -474.07 + 250.36 = 4857.17$ kft

$\Sigma M_{ux} = 1439.85 + 1470.47 = 2910.32$ kft

The maximum factored soil pressure occurs at corner 3 is

$$q_{3,fac} = \frac{P_u}{A} + \frac{M_{uz}}{S_z} + \frac{M_{ux}}{S_x}$$

$$= \frac{2292.85}{720} + \frac{4857.17}{4320} + \frac{2910.32}{2400}$$

$= 5.52$ ksf

$q_{3,fac} = 5.52 + 1.25(0.525) + 1.35(0.20) = 6.45$ ksf

Flexure Design

X-Dir:

$q_{3,fac}$ (used for design) = 5.52 ksf

Area of equivalent square = $0.25\pi(54)^2 = 2290\ \text{in}^2$

Side of square = $\sqrt{2290}$ = 47.85 in/12 = 3.99 ft

From two dimensional frame analysis maximum, M_u = 3979.4 kft

Determine the Required Area of Steel

d_e = 42 – 3 clr. – 0.5(1.0) = 38.5 in

$$R_n = \frac{M_u}{\phi b d_s^2}$$

$$= \frac{3979.4(12000)}{0.9(240)(38.5)^2}$$

= 149.15 psi

$$\rho = \frac{0.85f_c'}{f_y}\left(1 - \sqrt{1 - \frac{2R_n}{0.85f_c'}}\right)$$

$$= \frac{0.85(4)}{60}\left(1 - \sqrt{1 - \frac{2(149.15)}{0.85(4000)}}\right)$$

= 0.00254

$A_{s,req.} = \rho bd = 0.00254(240)(38.5) = 23.50\ in^2$

Provide 39 – #8 bars ($A_s = 39 \times 0.79 = 30.81\ in^2$)

Check Minimum Steel

$M_n \geq M_n$ -min

where: M_n -min = smaller of 1.2 Mcr and 1.33 M_u **[LRFD Eq. 5.7.3.3.2]**

$$Mcr = 0.24\sqrt{fc'}\,xS_t$$

St = Ig/Yt

$$Ig = bh^3/12 = \frac{20 \times 12 \times (42)^3}{12} = 1481760\ in^4$$

Yt = 21 in.

$$St = \frac{1481760}{21} = 70560\ in^3$$

$$Mcr = 0.24\sqrt{4}\ x\ 70560\ x\ \frac{1}{12} = 2822.4\ k\text{-}ft$$

1.2 Mcr = 3386.9 k-ft

1.33 M_u = 1.33 (3979.4) =5292.602 k-ft

M_n min = 3386.9 k-ft

$$Rn = \frac{M_u}{\phi bd^2} = \frac{3386.9}{0.9x20x12(38.5)^2} \quad 126.942 \text{ psi}$$ **[P-T.O]**

$$\rho = \frac{0.85f_{c'}}{f_y}\left(1 - \sqrt{1 - \frac{2R_n}{0.85f_c'}}\right)$$

$$= \frac{0.85(4)}{60}\left(1 - \sqrt{1 - \frac{2x126.942}{0.85x4000}}\right)$$

= 0.00216

A_s = 0.00216x20x12x(38.5) = 19.92 in^2

LRFD 5.7.3.3.2 requires to check temperature and shrinkage steel as per 5.10.8

As per LRFD 5.10.8.2

$$\geq \frac{0.11Ag}{f_y}$$ **[LRFD 5.10.8.2-1]**

Ag = 200 (42) = 10080 in^2

RC-PIER separately checks half of the reinforcement to be placed on each face.

$$\frac{0.11}{2}\left(\frac{Ag}{f_y}\right) = \frac{10080x0.11}{2} \quad x \quad \frac{1}{60} = 9.24 \text{ in}^2$$

Also for footing:

ΣA_b = 0.0015 Ag **[LRFD 5.10.8.2-2]**

RC-PIER separately checks half of this applies to each face.

$$\frac{0.0015Ag}{2} = \frac{0.0015(10080)}{2} = 7.56\text{in}^2$$

Therefore temperature and shrinkage steel = 7.56in^2

As prov > temperature and shrinkage steel. OK

Z-Dir :

$q_{3,fac}$ (used for design) = $q_{3,fac}$ – 1.35(0.20) – 1.25(0.525) = 5.52 ksf

Area of equivalent square = 0.25π(54)^2 = 2290 in^2

Side of square = $\sqrt{2290}$ = 47.85 in/12 = 3.99 ft

Distance from centerline column to face of equivalent square = 3.99/2 = 1.99 ft

$$M_u \text{ at } 1.99 = \frac{wl^2}{2} = \frac{(5.52 \times 36)(10 - 1.99)^2}{2} = 6374.94 \text{ kft}$$

Determine the Required Area of Steel

$d_e = 42 – 3 – 1 – 0.5(1.0) = 37.5$ in

$$R_n = \frac{M_u}{\phi b d_e^2} = \frac{6374.94(12000)}{0.9(432)(37.5)^2} = 139.91 \text{ psi}$$

$$\rho = \frac{0.85f_c'}{f_y}\left(1 - \sqrt{1 - \frac{2R_n}{0.85f_c'}}\right)$$

$$= \frac{0.85(4)}{60}\left(1 - \sqrt{1 - \frac{2x139.91}{0.85x4000}}\right)$$

= 0.002381

As required = ρ **bd**

= 0.002381(432)(37.5)

= 38.586 in^2

Provide 80 – #8 bars ($A_s = 80 \times 0.79 = 63.2$ in^2)

Check Minimum Steel

$\phi M_n \geq M_n$ min

where M_n min is smaller of 1.2 Mcr and 1.33 M_u **[LRFD 5.7.3.3.2]**

$$Ig = \frac{36(12) \times (42)^3}{12} = 2667168 \text{ in}^4$$

yt = 21 inch

st = 127008 in^3

$$M_{cr} = 0.24\sqrt{(4)} \times 127008x\frac{1}{12} = 5080.32 \text{ kft}$$

1.2 M_{cr} = 6096.38 kft

1.33 M_u =8478.67 kft

M_n min = 6096.38 kft

$$R_n = \frac{6096.38x12000}{0.9(36x12)(37.5^2)} = 133.80 \text{ psi}$$

$$\rho = \frac{0.85x4}{60}\left(1 - \sqrt{1 - \frac{2x133.8}{0.85x4000}}\right)$$

$\rho = 0.00228$

As = 0.00228 (36x12) (37.5) = 36.94 in^2

As (provided) >As (min) OK

LRFD 5.7.3.3.2 requires to check temperature and shrinkage steel as per 5.10.8. As per LRFD 5.10.8.2

$$\text{As prov} \geq \frac{0.11Ag}{f_y}$$ **[LRFD 5.10.8.2-1]**

Ag = 36x12x42=18144 in^2

RC-PIER separately checks half of the reinforcement to be placed on each face.

$$\frac{0.11}{2}\left(\frac{Ag}{f_y}\right) = 8.136 \text{ in}^2$$

Also for footings

$\Sigma A_b = 0.0015Ag$

$$\frac{0.0015}{2} = 13.608 \text{ in}^2$$

As (temp) = 8.136 in^2

As prov ≥ As (temp)

Crack Control

X-Dir (At inside face of first column):

$$f_{sa} = \frac{z}{\sqrt[3]{d_c A}} \leq 0.6f_y$$ **[LRFD Eq. 5.7.3.4-1]**

where:

z = 170 k/in

d_c = 2.0 max. + 0.5(1.0) = 2.5 in

A = 2(2.5)(240 in/39 bars) = 30.8 in^2/bar

$$f_{sa} = \frac{170}{\sqrt[3]{(2.5)(30.8)}} = 39.96 \text{ ksi} > 0.6(60) = 36 \text{ ksi}$$

Therefore, use f_{sa} = 36 ksi

$$f_s = \frac{M_s x12}{A_s jd}$$

$$n = \frac{E_s}{E_c} = \frac{29000}{3834.3} = 7.56$$

$$m = \frac{nA_s}{bd} = \frac{7.56(30.81)}{(240)(38.5)} = 0.025$$

$$k = \sqrt{m^2 + 2m} - m = \sqrt{0.025^2 + 2(0.025)} - 0.025 = 0.200$$

$$j = 1 - \frac{k}{3} = 1 - \frac{0.200}{3} = 0.933$$

Service Load Moment, $q_{3,service}$ (used in design)

= 5.14 – 0.2 – 0.525 = 4.42 ksf

From two dimensional frame analysis, $M_{service}$ at the face of the equivalent square = 3179.2 kft

$$f_s = \frac{3179.2(12)}{30.81(0.933)(38.5)} = 34.47 \text{ ksi}$$

f_s = 34.47 ksi < f_{sa} = 36.0 ksi

$$\text{ratio} = \frac{34.47}{36} = 0.96$$

Z-Dir (At face of column):

$$f_{sa} = \frac{z}{\sqrt[3]{d_c A}} \le 0.6 f_y$$ **[LRFD Eq. 5.7.3.4-1]**

where:

z = 170 k/in

d_c = 2.0 max. + 0.5(1.0) = 2.5 in

A = 2(2.5)(432 in/80 bars) = 27.0 in^2/bar

$$f_{sa} = \frac{170}{\sqrt[3]{(2.5)(27.0)}} = 41.75 \text{ ksi} > 0.6(60) = 36 \text{ ksi}$$

Therefore, use f_{sa} = 36 ksi

$$f_s = \frac{M_s x12}{A_s jd}$$

$$n = \frac{E_s}{E_c} = \frac{29000}{3834.3} = 7.56$$

$$m = \frac{nA_s}{bd} = \frac{7.56(63.2)}{(432)(37.5)} = 0.0295$$

$$k = \sqrt{m^2 + 2m} - m = \sqrt{0.0295^2 + 2(0.0295)} - 0.0295 = 0.215$$

$$j = 1 - \frac{k}{3} = 1 - \frac{0.215}{3} = 0.930$$

Service Load Moment, $q_{3,service}$ (used in design)

$= 5.14 – 0.2 – 0.525 = 4.42$ ksf

$$M_s = \frac{(4.42 \times 36)(10 - 1.99)^2}{2} = 5104.57 \text{ kft}$$

$$f_s = \frac{5104.57(12)}{63.2(0.930)(37.5)} = 27.79 \text{ ksi}$$

$f_s = 27.79$ ksi $< f_{sa} = 36$ ksi

$$\text{ratio} = \frac{27.79}{36} = 0.77$$

Fatigue Check

X-Dir:

The range between a maximum tensile stress and minimum stress in straight reinforcement caused by fatigue combination should not exceed.

$$f_f = 21.0 - 33f_{mm} + 8\left(\frac{r}{n}\right)$$

$$\text{where} \left(\frac{r}{n}\right) = 0.3$$

Maximum soil pressure due to fatigue combination occurs for combination #17.

Column #1

$P = 65.71$ kip

$M_z = -31.49$ kft

$M_x = 9.74$ kft

Column #2

P = -15.75 kip

M_z = 22.61 kft

M_x = 9.56 kft

Determine the Maximum Fatigue Soil Pressure

ΣP = 65.71 + (-15.75) = 49.96 kip

ΣM_z = 65.71 (8) – (-15.75)(8) – 31.49 + 22.61 = 642.8 kft

ΣM_x = 9.74 + 9.56 = 19.3 kft

Maximum Soil Pressure

$$q_{3,\text{ fatigue}} = \frac{P}{A} + \frac{M_z}{S_z} + \frac{M_x}{S_x} = \frac{49.96}{720} + \frac{642.8}{4320} + \frac{19.3}{2400} = 0.23 \text{ ksf}$$

The minimum soil pressure due to fatigue combination is negative which is not considered. Therefore, for f_{min}, use fatigue soil pressure = 0.

Determine the Soil Pressure due to Permanent Loads, Creep, and Shrinkage Loads

Column #1

P = 1052.12 kip

M_z = -273.36 kft

M_x = 1133.7 kft

Column #2

P = 688.9 kip

M_z = 178.2 kft

M_x = 1158.54 kft

ΣP = 1052.12 + 688.9 = 1740.9 kip

ΣM_z = 1052.12(8) – 688.9(8) – 273.36 + 178.2 = 2809.64 kft

ΣM_x = 1133.7 + 1158.54 = 2292.2 kft

Maximum Soil Pressure

$$q_3 = \frac{1740.9}{720} + \frac{2809.64}{4320} + \frac{2292.2}{2400} = 4.02 \text{ ksf}$$

Stress in Steel Due to Fatigue Load

q = 0.23 ksf

w = 0.23 × 20 = 4.28 kft

M = 163.00 kft

Stress in steel due to this moment:

$$f_s \quad = \quad \frac{163.00(12)}{30.81(0.933)(38.5)} \quad = 1.77 \text{ ksi}$$

Stress in Steel Due to Permanent Loads

$q \quad = 4.02 \text{ ksf}$

$w \quad = 4.0221 \times 20 = 80.44 \text{ kft}$

$M \quad = 2900.00 \text{ kft}$

Stress in steel due to this moment:

$$f_s \quad = \quad \frac{2900.00(12)}{30.81(0.933)(38.5)} \quad = 31.44 \text{ ksi}$$

$f_{range} = 1.77 - 0 = 1.77 \text{ ksi}$

$f_{min} = 31.44 + 0 = 31.67$

$f_f \quad = 23.4 - 0.33(f_{min})$

$\quad = 23.4 - 0.33(31.44)$

$\quad = 13.02 \text{ ksi}$

$$\text{Ratio} = \frac{1.77}{13.02} \quad = 0.14$$

Footing Design – Pile Cap

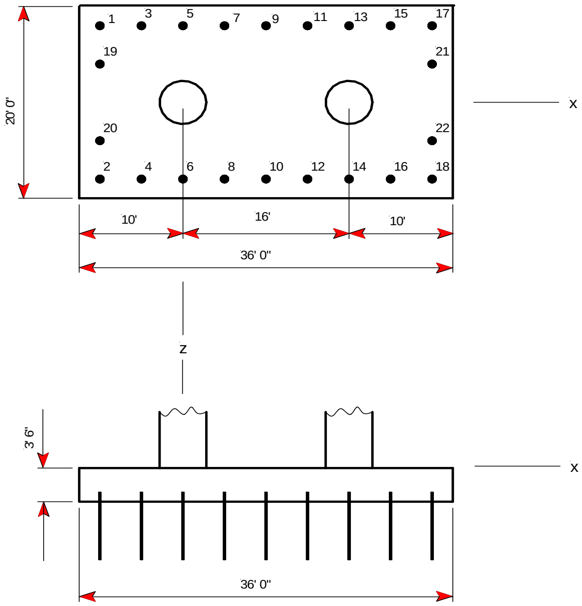


Figure B-14 Plan and Elevation of Combined Pile Footing (Example 2)

Geometry

n = number of piles = 22

Pile Reactions

$$P_p = \frac{P}{n} \pm \frac{M_z X_z}{\Sigma X_z^2} \pm \frac{M_x X_x}{\Sigma X_x^2}$$

$$\Sigma d_z^2 = 4(4)^2 + 4(8)^2 + 4(12)^2 + 8(16)^2 = 2944 \text{ ft}^2$$

$$\Sigma d_x^2 = 4(4)^2 + 18(8)^2 = 1216 \text{ ft}^2$$

Maximum Service Pile Reaction

Pile #18 for Load Combination 14:

Column #1:

P = Service axial load = 694.13 kips
M_z = Service transverse moment = -192.37 kft
M_x = Service longitudinal moment = 1232.46 kft

Column #2:

P = Service axial load = 1095.50 kips
M_z = Service transverse moment = 247.80 kft
M_x = Service longitudinal moment = 1209.42 kft

$$\Sigma P = 694.13 + 1095.50 + 0.15 \times 20 \times 36 \times \frac{42}{12}(1.0) + 0.2 \times 20 \times 36(1.0)$$

$$= 2311.63 \text{ kips}$$

$$\Sigma M_z = 694.13(8) - 1095.50(8) - 192.37 + 247.80 = -3155.53 \text{ kft}$$

$$\Sigma M_x = 1232.46 + 1209.42 = 2441.9 \text{ kft}$$

$$P_{18} = \frac{2311.63}{22} + \frac{3155.53 \times 16}{2944} + \frac{2441.9 \times 8}{1216} = 38.29 \text{ kips}$$

Maximum Factored Pile Reaction

Pile #18 for Load Combination 9:

Column #1:

P_u = Factored axial load = 865.62 kips
M_{uz} = Factored transverse moment = -380.55 kft
M_{ux} = Factored longitudinal moment = 1724.83 kft

Column #2:

P_u = Factored axial load = 1375.53 kips
M_{uz} = Factored transverse moment = +221.50 kft
M_{ux} = Factored longitudinal moment = 1710.96 kft

$$\Sigma P_u = 865.62 + 1375.53 + 0.15 \times 20 \times 36 \times \frac{42}{12} (1.25) + 0.2 \times 20 \times 36 (1.35)$$

$$= 2908.05.4 \text{ kips}$$

$$\Sigma M_{uz} = 865.62(8) - 1375.53(8) - 380.55 - 221.50 = -4238.33 \text{ kft}$$

$$\Sigma M_{ux} = 1724.83 + 1710.96 = 3435.8 \text{ kft}$$

$$P_{18} = \frac{2908.05}{22} + \frac{4238.33 \times 16}{2944} + \frac{3435.8 \times 8}{1216} = 177.82 \text{ kips}$$

Maximum Factored Pile Reaction Excluding Self-weight and Surcharge

$$P = 865.62 + 1357.53 = 2241.15 \text{ kips}$$

$$M_{uz} = 865.62(8) - 1375.53 \text{ x } (8) - 380.55 - 221.50 = -5022.46 \text{ kft}$$

$$M_{ux} = 1724.83 + 1710.96 = 3435.8 \text{ kft}$$

$$P_{18} = \frac{2241.15}{22} + \frac{5022.46 \times 16}{2944} + \frac{3435.8 \times 8}{1216} = 147.51 \text{ kips}$$

One Way Shear Design

X-Dir:

$$d_v = d_e - \frac{a}{2}$$

where:

d_e = 38.5

$$a = \frac{A_s f_y}{0.85 f_c' b} = \frac{30.81(60)}{0.85(4)(240)} = 2.265 \text{ in}$$

$$d_v = 38.5 - \frac{2.265}{2} = 37.37 \text{ in}$$

The distance from column centerline to distance d_v from the face of the equivalent square is

$$= 1.99 + \frac{37.37}{12} = 5.10 \text{ ft.}$$

V_u at 5.10 ft from column centerline = 4 × 147.51 = 590.04 kips

We will use Table 5.8.3.4.2-2 to determine θ and β because there is no transverse reinforcement in footing.

By iterations we determine $\theta = 53.83$ **[LRFD Eq. 5.8.3.4.2-2]**

$$\in \quad = \frac{\frac{1706.4 \times 12}{37.367} + 0.5 \times 590 \times 731}{29000 \text{ x } 30.81} = 0.0008456$$

Sxe = 37.367

θ = 53.83

β = 1.61

$$\phi V_c = \phi(0.0316)\beta\sqrt{f'_c}b_v d_v$$ **[LRFD Eq. 5.8.3.3-3]**

$$= 0.9(0.0316)(1.61)\sqrt{4}(240)(37.37) = 821.3 \text{ kips}$$

ϕV_c = 821.3 kips > V_u = 625.2 kips

Z-Dir:

$$d_v = d_e - \frac{a}{2}$$

where:

d_e = 37.5

$$a = \frac{A_s f_y}{0.85 f'_c b} = \frac{63.2(60)}{0.85(4)(432)} = 2.58 \text{ in}$$

$$d_v = 37.5 - \frac{2.580}{2} = 36.21 \text{ in}$$

The distance from column centerline to a distance d_v from the face of the equivalent square is

= 1.99 + 36.21/12 = 5.01 ft

V_u at 5.01 from column centerline = 9 × 147.51 = 1327.59 kips

We will use Table 5.8.3.4.2-2 to determine θ and β because there is no transverse reinforcement in footing/pile cap.

By iterations we determine θ = 54.64 **[LRFD Eq. 5.8.3.4.2-2]**

Final iterations calculations.

$$\in \; = \frac{\frac{Mu}{dv} + 0.5Nu + 0.5(Vu - Vp) \times \cot\theta \times A_{ps} \times F_{po}}{E_sA_s + E_pA_p}$$

$$= \frac{\frac{3967.6 \times 12}{36.21} + 0.5 \times 1327.6 \times 0.7096}{29000 \text{ x } 63.2} = 0.0009743$$

From table 5.8.3.4.2-2

θ = 54.64

β = 1.518

$$\phi V_c = \phi(0.0316)\beta\sqrt{f'_c}b_vd_v$$

$$= 0.9(0.0316)(1.518)\sqrt{4(}432)(36.21)$$

= 1350.65 kips

ϕV_c = 1350.65 kips > V_u = 1327.59 kips

Two Way Shear Design

d_v = $(d_{vx} + d_{vy})$ / 2 = (37.37 + 36.21) / 2 = 36.79 in

b_o = 0.5(2πd) = πd = π(54 + 36.79) / 12 = 23.77 ft

A_o = $0.25\pi d^2 = 0.25\pi(7.56)^2 = 44.89$ ft^2

From Frame Analysis:

V_u = 1622.6 kips

$$\phi V_c = \phi(0.0632)\beta\sqrt{f'_c}b_od_v$$ **[LRFD Eq. 5.13.3.6.3-3]**

$$\phi V_c = 0.9(0.0632)(2)\sqrt{4}(23.77 \times 12)(36.79) = 2387.6 \text{ kips}$$

ϕV_c = 2387.6 > V_u = 1622.6 kips

$$V_n = V_c + V_s \le 0.192\sqrt{f'_c}b_od_v$$ **[LRFD Eq. 5.13.3.6.3-2]**

$$V_n = \frac{2387.6}{0.9} = 2653\text{kips} \le 0.192\sqrt{4}(23.77 \times 12)(36.79)$$

= 4029.7 kips

Tutorial 3: Hammerhead Pier, Strut-and-Tie Model Design (AASHTO LRFD)

This example demonstrates the design of a hammerhead, as shown in Figure TH-15, by using the Strut and Tie Method (STM). The design will be carried out in accordance with *AASHTO LRFD Bridge Design Specifications*, 3rd Edition, 2004.

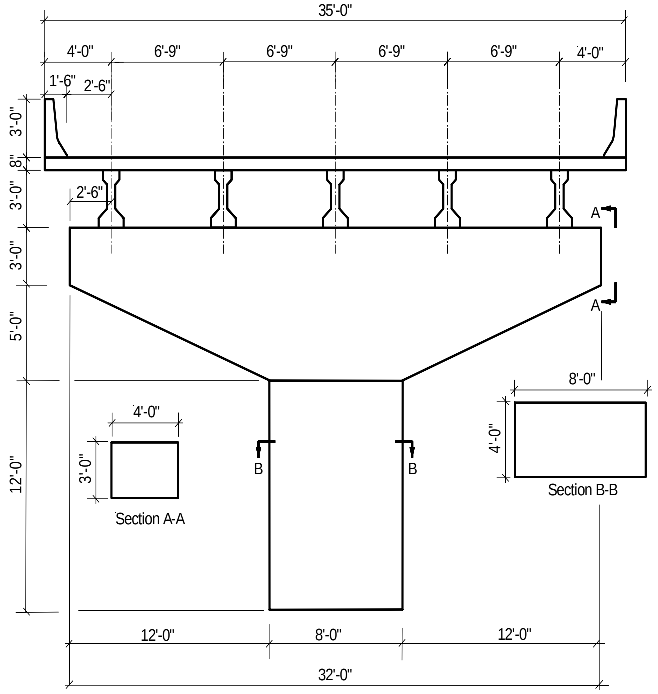


Figure B-15 Hammerhead Bridge Cross Section at the Pier

Materials

Concrete Strength

Cap	$f'_c = 4.0$ ksi
Columns	$f'_c = 4.0$ ksi
Footings	$f'_c = 4.0$ ksi
Modulus of Elasticity	$E_c = 3834.3$ ksi

Concrete Density

Cap	$\rho = 150$ pcf
Columns	$\rho = 150$ pcf
Footings	$\rho = 150$ pcf

Steel Yield Strength

Cap	$f_y = 60$ ksi
Columns	$f_y = 60$ ksi
Footings	$f_y = 60$ ksi

Modulus of Elasticity E_s = 29000 ksi

Other Parameters

Crack control factor z = 170 kips/in

Superstructure Parameters

Number of lanes = 2
Beam height = 36 in
Beam section area = 369 in^2
Barrier height = 36 in
Slab depth = 8.0 in
Total number of spans = 2
Wearing surface load = 20 psf
Barrier load = 600 plf each side

Span Information

Bridge overall width Forward = 35 ft Rear = 35 ft
Curb to curb distance Forward = 32 ft Rear = 32 ft
Span Length Forward = 60 ft Rear = 60 ft

Substructure Parameters

Dimensions as shown in Figure TH-15
Skew Angle = 0 degrees
Top elevation = 20 ft
Factor of reduced moment of inertia = 1.0 (non-cracked section)

Five bearing points, spaced at 6.75 ft, on one line with no eccentricity

Loads

Dead Loads

Self load = 150 pcf
Slab Weight = 150 pcf
Girder Weight = 150 pcf
Barrier Weight = 600 plf each side × 2 = 1200 plf
Wearing Surface = 200 psf × 35 ft = 700 plf

Live Load

HL-93 (a combination of HS-20 Truck with a lane load of 0.64 klf)

Design Truck + Design Lane Load

Loads to be considered: Dead load, Live load, Wind load on the structure, and Wind on live load.

Wind Load

Bridge Location = city
Direction of wind = 0^o
Elevation above which wind acts = 0 ft
Trans. wind pressure on superstructure = 50 psf
Longitudinal pressure on superstructure = 0 psf
Wind pressure on cap = 40 psf
Wind pressure on column = 40 psf

Load Generation

Dead Loads, DC

1. Self-weight of the pier cap:

   Total weight of the pier cap
   = [(3x32) + (0.5)(8+32)(5)](4)(0.150) = 117.6 kips
   Distribute the total weight equally between all nodes of the STM
   = 117.6/21 = 5.6 kips/node

2. Dead loads from the superstructure:

   The tributary span length of the pier is 30 ft rear and 30 ft forward = 60 ft total.

Load Type	**Exterior Bearing Points (kips)**	**Interior Bearing Points (kips)**
	Tributary width *= 4 + 0.5 × 6.75* = 7.375 ft	Tributary width = 6.75 ft
Slab Weight	(8/12)(0.150)(7.375)(60) = 44.250	(8/12)(0.150)(6.75)(60) = 40.500
Girder Weight	(369/144)(0.150)(60) = 23.063	(369/144)(0.150)(60) = 23.063
Barrier Weight	1.2(7.375/35)(60) = 15.171	1.2[6.75/35](60) = 13.886
Total DL	82.484	77.449

***NOTE**: In* RC-PIER, *all loads have negative signs, if they are downwards.*

Wearing Surface Loads, DW:

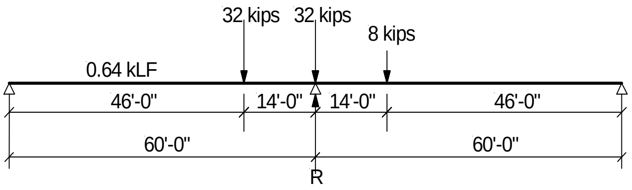
Wearing Surface, DW	$\left(\frac{700}{1000}\right)\left(\frac{7.375}{35}\right)60$ = 8.850	$\left(\frac{700}{1000}\right)\left(\frac{6.75}{35}\right)60$ = 8.100

Live Load, LL2:

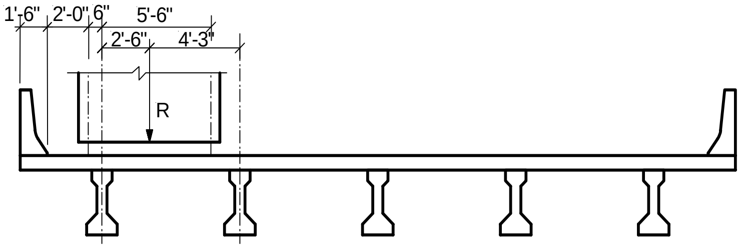
Auto live load generation is used. One design truck plus one design lane loaded is selected. Also, one lane loaded is selected. The auto live load generation provides, for this example, two live loads, where the calculation for the second live load (LL2) is shown in Figure TH-16.

To get maximum reaction from the truck at the pier, position the truck such that the middle axle is over the pier centerline, the first axle is at 14 ft from the pier centerline in the right span, and the third axle is at 14 ft from the pier centerline in the left span, as shown in Figure TH-16a. Thus,

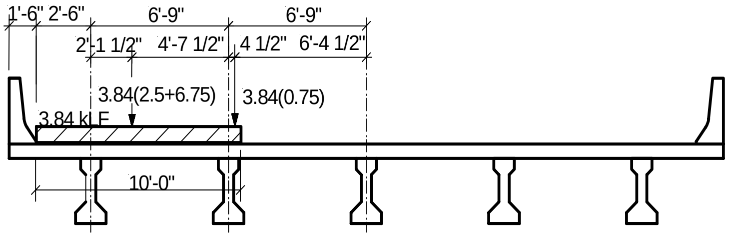
Reaction from the truck =32 + (8 × 46/60) + (32 × 46/60) = 62.7 kips
Reaction from the lane load = 0.64 klf × 60 ft = 38.4 kips



(a) Longitudinal View



(b) Truck Positioning



(c) Lane Load Positioning

Figure B-16 Live Load Positioning (a, b, c) - Tutorial 3

Cases LL2: Maximum Bearing Load at the Exterior Girder

To get maximum reaction at exterior girders, position the truck such that the distance between the inner face of the barrier and the nearest wheel of the truck = 2 ft, as shown in Figure TH-16b. Therefore,

Bearing load from the exterior girder = 62.7(4.25/6.75) = 39.5 kips
Bearing load from the first interior girder = 62.7(2.5/6.75) = 23.2 kips

To get reaction from the lane load, position the 10 ft wide lane such that it starts from the inner face of the barrier, as shown in Figure TH-16c. Distributing the total reaction of the lane load (38.4 kips) over the 10 ft wide lane results in 3.84 kip/ft.

Bearing load from the exterior girder
= (3.84)(2.5+6.75)(4.625/6.75)
= 24.3 kips
Bearing load from the first interior girder
= (3.84)(2.5 + 6.75)(2.125/6.75) + (3.84)(0.75)(6.375/6.75)
= 13.9 kips
Bearing load from the second interior girder
= (3.84)(0.75)(0.375/6.75)
= 0.16 kips

Taking into consideration, a multi presence factor of 1.2 for one loaded lane, bearing loads are:

Bearing	**Truck Load**	**Lane Load**
1	1.2 × 39.5 = 47.4	1.2 × 24.3 = 29.16
2	1.2 × 23.2 = 27.84	1.2 × 13.9 = 16.69
3	0	1.2 × 0.16 = 0.19
4	0	0
5	0	0

Impact factor, IM = 33% (applied to Truck only). Note the impact factor is only applied to the analysis results (moments and forces) of the load combinations and envelopes.

Wind Load on Structure, WS

Assume zero degree of the wind and bridge location is city.

Loads from the Superstructure:

The transverse pressure is acting on the height of the superstructure at the center of this area.

The superstructure has a total height = 36 barrier + 8 deck slab + 36 beam = 80 in.

$$\text{Total transverse load} = \left(\frac{50}{1000}\right) \times \left(\frac{80}{12}\right) \times 60 \quad = 20 \text{ kips}$$

Distribute the total load equally at the five bearing points, thus,

$$\text{Transverse load, } F_x = \frac{20}{5} \; = +4.0 \text{ kips/bearing}$$

Since this load is applied at the center of the height of the superstructure, it creates an overturning moment where

$$\frac{20\left(\frac{80}{12}\right)}{2} \; = 66.7 \text{ kips-ft at the bearing elevation.}$$

This moment will be distributed to the two exterior girders. Thus, one exterior girder (at bearing point #5) applies a downward force, i.e. $-F_y$, on the pier cap and the exterior girder on the other side (at bearing point #1) applies upward force, i.e. $+F_y$. This force, F_y is

$$\frac{66.7}{4 \times 6.75} \; = +2.47 \text{ kips at bearing 1} = -2.47 \text{ kips at bearing 5}$$

***NOTE**: 4 × 6.75 = 27 ft is the distance between bearing point #1 and bearing point #5.*

Loads on the Substructure

A 50 psf wind pressure should be applied to the pier cap.

Loads on the Pier Cap

Total projected area of the pier cap (normal to the wind direction) is = 8 × 4 = 32 sq ft

$$\text{Total wind force} = \frac{32 \times 50}{1000} = 1.6 \text{ kips}$$

Distribute this force equally between all of the nodes of the STM, and each node will have an applied X-force = 1.6 kips / 21 nodes = 0.076 kips

Loads on the Column

Assume that the column is subject to wind load from its bottom upwards.

$$\text{Total wind force} = \frac{40}{1000} \times 4 = 0.16 \text{ kip/ft of the column length}$$

Analysis and Load Combinations

Analysis

Figure TH-17 shows an example of a strut and tie model. Assume that the truss member thickness is 0.4 ft (4.8 in.) and the member width equals the pier cap width, 4ft (48 in.).

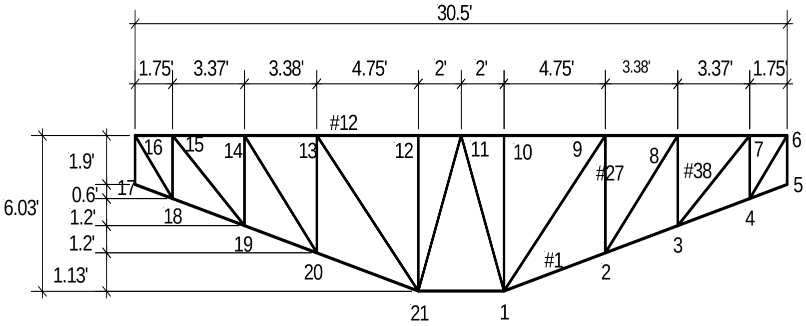


Figure B-17 Strut and Tie Model (STM) of Cap

Load Combination

Two limit states should be considered, Strength I and Strength III.

Strength I = 1.0[(1.25 or 0.9) DC + (1.5 or 0.65) DW + (1.75)LL]
Strength III = 1.0[(1.25 or 0.9) DC + (1.5 or 0.65) DW ± (1.4)WS]

Design of the Pier Cap

Top Tension Ties

Maximum tension force is at element # 12 = 776.8 kips

$$\text{Required area of mild steel} = T/(\phi f_y) = \frac{776.8\text{kips}}{0.9 \times 60\text{ksi}} = 14.4 \text{ in}^2.$$

Try 19 #8, provided area = 19 × 0.79 = 15.01 in^2.

$$\rho_{min} = 0.03 \times \frac{f_c'}{f_y} = 0.03\frac{4}{60} = 0.002$$

$A_{smin} = 0.002 \times 8 \times 4 \times 144 = 9.22 \text{ in}^2 < 15.01$ OK

Bottom Compression Struts

Try member #1 where the compression force in this member is 447.9 kips. The angle α is the one between member #1 and member #27.

Therefore,

$$\alpha = \tan\left(\frac{2.3 - 1.17}{6.75 - 2.0}\right) + 90 = 103.4$$

$$\varepsilon_1 = \varepsilon_s + (\varepsilon_s + 0.002) \cot^2 \alpha$$

$$\varepsilon_s = f_y/E_s = \frac{60}{29000} = 0.00207$$

Therefore, $\varepsilon_1 = 0.00230$ and the limiting compressive stress, f_{cu} (per LRFD Eq. 5.6.3.3.3-1) is:

$$f_{cu} = \frac{f_c}{0.8 + 170\varepsilon_1} = \frac{4.0}{0.8 + 170 \times 0.00230}$$

$$= 0.84(4.0)$$

$$= 3.360 \text{ ksi} < 0.85(4.0) = 3.4 \qquad \text{OK}$$

Applied compressive stress in the strut member #1 is

$$P/(\phi\, A_{strust}) = \frac{447.9}{0.7 \times 48 \times 4.8} = 2.78 \text{ ksi} < f_{cu} = 3.36 \qquad \text{OK}$$

Vertical Compression Struts

Try Member #38 where the force in this member is 408.8 kips.

$$\cot \alpha = \frac{7.2 - 3.5}{13.5 - 10.13} = 1.098,\ \alpha = 42.328 \text{ deg.}$$

$$\varepsilon_1 = \varepsilon_s + (\varepsilon_s + 0.002) \cot^2 \alpha$$

$$\varepsilon_s = f_y/E_s = \frac{60}{29000} = 0.00207$$

Therefore, $\varepsilon_1 = 0.00698$ and the limiting compressive stress, f_{cu} is

$$f_{cu} = \frac{f_c}{0.8 + 170\varepsilon_1} = \frac{4.0}{0.8 + 170 \times 0.00698}$$

$$= 0.503\ (4.0)$$

$$= 2.01 \text{ ksi} < 0.85(4.0) = 3.4 \text{ ksi}$$

Applied compressive stress in strut is

$$P/(\phi\, A_{strust}) = \frac{408.8}{0.7 \times 48 \times 4.8} = 2.53 \text{ ksi} > f_{cu} = 2.01 \qquad \text{NG}$$

Since actual stress in strut exceeds the allowable stress, additional steel (may be in terms of shear stirrups) is required.

Vertical Tension Ties

The tension force at element #27 is 223 kips.

Required area of mild steel = $T/(\phi fy) = \dfrac{223.0\text{kips}}{0.9 \times 60\text{ksi}} = 4.13\ \text{in}^2$

Design of the Pile Cap

Analysis

Figure TH-18 shows a plan view of the pile cap and Figure TH-19 and Figure TH-20 shows an example of the STM model in X and Z directions. For all members of the STM model, assume that the member thickness is 0.4 ft (4.8 in.) and the member width equals the pile cap width.

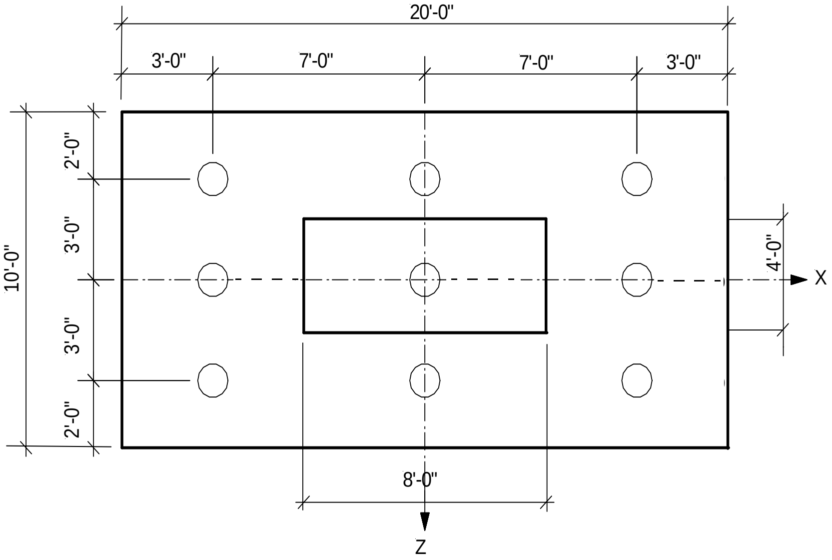


Figure B-18 Strut and Tie Model of Pile Cap - Plan View (Tutorial 3)

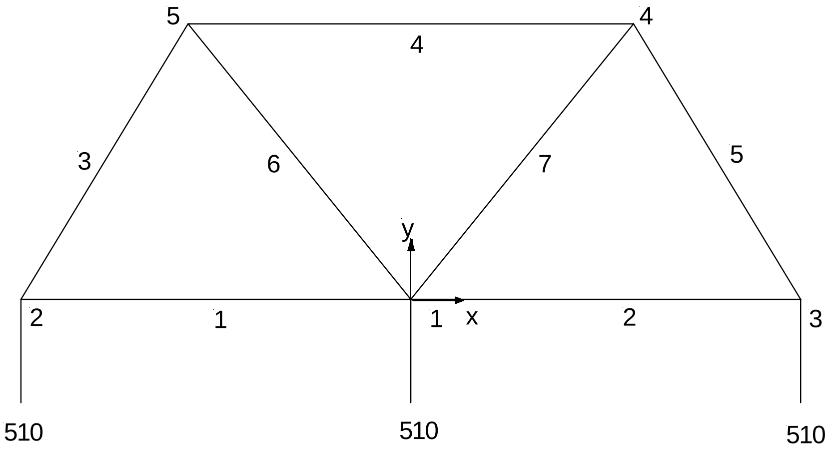


Figure B-19 Strut and Tie Model in Z-Direction (Tutorial 3)

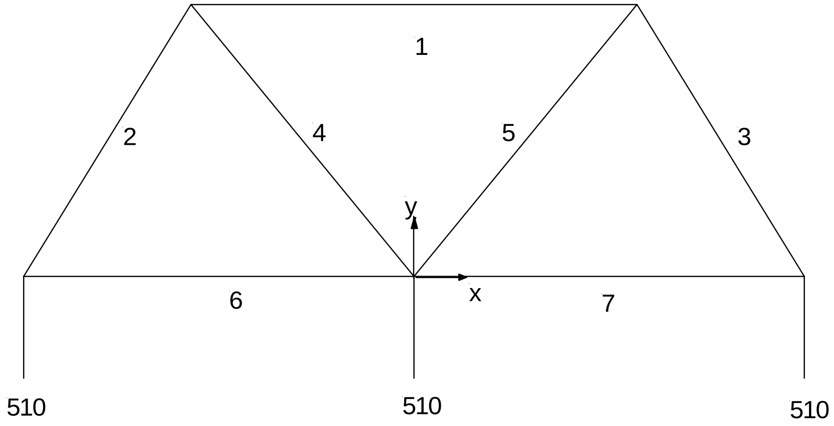


Figure B-20 Strut and Tie Model in X-Direction (Tutorial 3)

NOTE*: These calculations are based on load combination #1.*

Total Reaction from Load Combination #1

Vertical force = 990.88 kips, and moment about the Z-axis = 2334.6 kip-ft

Maximum reaction on piles on the left-hand side

$\Sigma z^2 = 3(7^2 + 0^2 + 7^2) = 294 \text{ ft}^2$

$P = 990.88 + 1.25\ (20 \times 10 \times 6 \times 0.15) + 1.35\ (20 \times 10 \times 1) = 1485.88 \text{ kips}$

$$\frac{P}{n} + \frac{M_x Z}{\Sigma Z^2} = \frac{1485.88}{9} + \frac{2334.6 \times 7}{294}$$

$= 165.09 + 55.6$

$= 220.68$ kips

Reaction on piles applied in a strut and tie model are those without the self-weight of the pile cap and surcharge.

$$\frac{P}{n} + \frac{M_x Z}{\Sigma Z^2} = \frac{990.88}{9} + \frac{2334.6 \times 7}{294} = 165.68 \text{ kips}$$

Since there are three rows of pile, 3 × 165.68 = 497.05 kips will be applied at pile location.

Design of the STM members

Bottom Tension Ties

X-Dir:

T = 298.23 kips

Required area of mild steel = $T/(\phi f_y) = \dfrac{298.23}{0.9 \times 60 \text{ksi}} = 5.52 \text{ in}^2$

Check minimum reinforcement

$$\rho_{min} = 0.03 \times \frac{f_c'}{f_y} = 0.03\frac{4}{60} = 0.002$$

$$A_{smin} = 0.002 \times 6 \times 10 \times 144 = 17.28 \text{ in}^2$$

Therefore, use minimum reinforcement.

Z-Dir:

T = 99.41 kips

Required area of mild steel = $T/(\phi f_y) = \frac{99.41\text{kips}}{0.9 \times 60\text{ksi}} = 1.84 \text{ in}^2$

Check minimum reinforcement

$A_{smin} = \rho_{min} \times 6 \times 20 \times 144 = 0.002 \times 6 \times 20 \times 144 = 34.6 \text{ in}^2$

Therefore, use minimum reinforcement.

Compression Struts

X-Dir:

Maximum compression force = 579.66 kips in member #2.

$$\cot\alpha = \frac{7 - 4}{5.5 - 0.5} = 0.6$$

$$\varepsilon_1 = \varepsilon_s + (\varepsilon_s + 0.002)\cot^2\alpha$$

$$\varepsilon_s = f_y/E_s = \frac{60}{29000} = 0.00207$$

Therefore, $\varepsilon_1 = 0.0035$ and the limiting compressive stress, f_{cu} is

$$f_{cu} = \frac{f_c}{0.8 + 170\varepsilon_1} = \frac{40}{0.8 + 170 \times 0.0035}$$

$$= 0.71(4.0)$$

$$= 2.88 \text{ ksi} < 0.85(4.0) = 3.4 \text{ ksi}$$

Applied compressive stress in the strut is

$$P/(\phi A_{strut}) = \frac{579.66}{0.7 \times 10 \times 12 \times 4.8} = 1.44 \text{ ksi} < f_{cu} \qquad \text{OK}$$

Z-Dir:

Maximum compression force = 506.90 kips in member #3

$$\cot\alpha = \frac{-2 + 3}{5.5 - 0.5} = 0.2$$

$$\varepsilon_1 = \varepsilon_s + (\varepsilon_s + 0.002)\cot^2\alpha$$

$$\varepsilon_s = f_y/E_s = \frac{60}{29000} = 0.00207$$

Therefore,

$$\varepsilon_1 = 0.00223$$

$$f_{cu} = \frac{f_c}{0.8 + 170\varepsilon_1} = \frac{40}{0.8 + 170 \times 0.00223}$$

$= 0.848(4.0)$

$= 3.39 < 0.85(4.0) = 3.4$ ksi OK

Applied compressive stress in the strut is

$$P/(\phi\ A_{strust}) = \frac{520.15}{0.7 \times 20 \times 12 \times 4.8} = 0.65 < f_{cu}$$

Example 1: Seismic Auto Load Generation (AASHTO LFD)

This example illustrates in detail the seismic load generation using the single mode spectral method for the bridge pier, as shown in Figure TH-21. The load generation will be carried out in accordance with *AASHTO Standard Specifications for Highway Bridges*, Seventeenth Edition, 2003. The pier geometry and material properties used in this example are the same as those used in Tutorial 1 (refer to input file seismic.rcp).

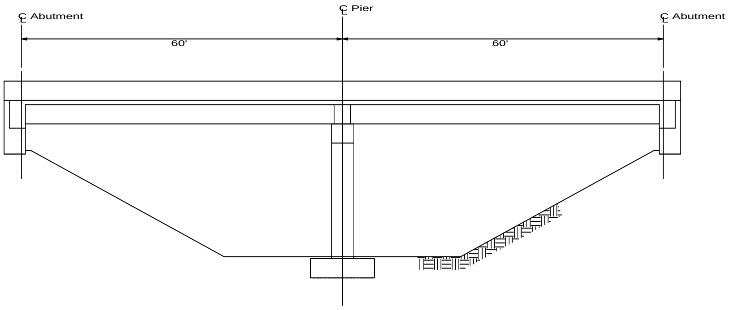


Figure B-21 Two Span Bridge

Materials

Concrete Strength

f'_c = 4000 psi

Modulus of Elasticity

E = 3834 ksi

Concrete Density

ρ = 150 pcf

Steel Yield Strength

f_y = 60 ksi

Concrete Type

Normal

Seismic Parameters

Acceleration Coeff = 0.4
Soil Profile = Type III
Response Modification Factor = 1

AASHTO Type II Girder

A = 369 in^2
I_x = 50980 in^4
I_y = 5332.5 in^4
$\bar{y}$ = 15.83 in

Barrier

Height = 3 ft

Avg. Width = 1'-4" = 1.33 ft

Area = 4 ft^2

$\bar{y}$ = 1.5 ft

$$I_x = \frac{1.333(3)^3}{12} = 3\text{ ft}^4$$

$$I_y = \frac{3(1.333)^3}{12} = 0.6\text{ ft}^4$$

Seismic Load Generation

Section Properties of the Superstructure

$$A = \text{Area} = 3\left(\frac{369}{144}\right) + 27\left(\frac{8}{12}\right) + 2 \times 4 = 33.7\text{ ft}^2\ (4853\text{ in}^2)$$

$$\bar{y} = \frac{3(369)(15.83) + (27 \times 12)(8)(40) + 2(4 \times 144)(62)}{3(369) + (27 \times 12)(8) + (2)(4)(144)}$$

$$= 39.7\text{ in}$$

I_y = Moment of inertia

$$= \frac{1}{12}(8)(27 \times 12)^3 + 3(5332.5) + (2)(369)(9 \times 12)^2$$

$$+ 2[(0.6 \times 12^4 + 4(12)^2 \times (12.83 \times 12)^2]$$

$$= 58854319\text{ in}^4\ (2838\text{ ft}^4)$$

I_x = Moment of inertia

$$= 3[50980 + 369(39.7 - 15.83)^2] + \frac{1}{12}(27 \times 12)(8)^3$$

$$+ (27 \times 12)(8)(40 - 39.7)^2 + 2[3(12)^4 + 4(12)^2(62 - 39.7)^2]$$

$$= 1495034\text{ in}^4\ (72.1\text{ ft}^4)$$

L = Length of structure = 120 ft.

Section Properties of the Substructure

$$A = \text{Area} = 2(3 \times 3) = 18\text{ ft}^2$$

$$I_z = \text{Moment of Inertia} = 2 \times \frac{1}{12}(3)(3)^3 = 13.5\text{ ft}^4\ (279936\text{ in}^4)$$

$$I_x = \text{Moment of Inertia} = 2 \times \frac{1}{12}(3)(3)^3 = 13.5\text{ ft}^4\ (279936\text{ in}^4)$$

H = Height of column = 20 ft.

Longitudinal Earthquake Loading

Calculate the Static Displacement, $v_s(x)$

Neglecting axial deformation in the deck and assuming that the deck behaves as a rigid member, the bridge may be idealized as a beam with roller supports at the abutments and a pier at midspan, as shown in Figure TH-22.

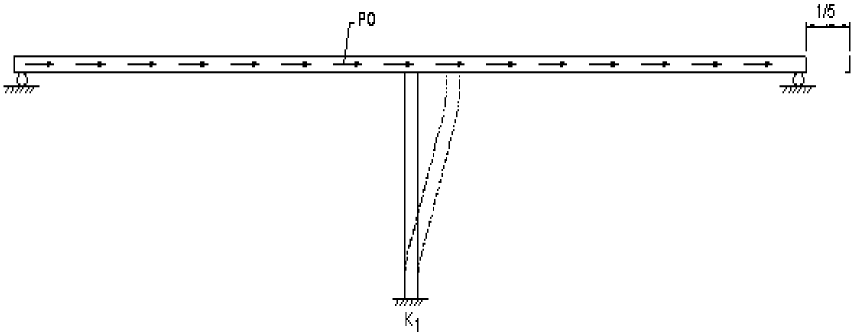


Figure B-22 Structural Idealization and Application of Assumed Uniform Loading for Longitudinal Mode of Vibration

The bridge is idealized so that the abutments do not contribute to the longitudinal stiffness. Applying an assumed uniform longitudinal loading of 1 kip/ft yields a constant displacement along the bridge. Assuming that the columns alone resist the longitudinal motion, the displacement is obtained by using a column stiffness of 12 EI/ H^3 in the longitudinal direction.

Using the column properties included in the section preceding this one, the stiffness for the pier is calculated as:

$$k \quad = \quad \frac{3EI}{H^3} = \frac{3(3834)(279936)}{(20 \times 12)^3} \times 12 \quad = 2795 \text{ k/ft}$$

$$vs \quad = \quad \frac{p_o 1}{k} = \frac{1.0(120)}{2795} = 0.04293 \text{ ft}$$

Calculate α, β, γ

$$Wt \quad = [33.7\ (120) + 4(3)(26) + 2(3 \times 3 \times 9)] \times 0.150 \text{ k/ft}^3 / 120$$
$$= 5.65 \text{ k/ft}$$

$$\alpha \quad = \quad \int_{Abut1}^{Abut2} v_s(x)dx \ = \ v_s 1 = 0.04293 \times 120 = 5.152 \text{ ft}^2$$

$$\beta \quad = \int_{Abut1}^{Abut2} w(x)v_s(x)dx \ = \ wv_s 1$$
$$= 5.65 \text{ k/ft} \times 0.04293 \times 120$$
$$= 29.109 \text{ k-ft}$$

$$\gamma \quad = \int_{Abut1}^{Abut2} w(x)v_s(x)^2 dx \; = \; wv_s^2 1$$

$= 5.65$ k/ft $\times (0.04293)^2 \times 120$

$= 1.2498$ k-ft^2

Calculate the Period, T

$$T \quad = \; 2\pi\sqrt{\frac{\gamma}{p_o g\alpha}} \; = \; 2\pi\sqrt{\frac{1.2498}{1.0(32.2)(5.152)}} = 0.5453 \text{ sec}$$

Calculate the Elastic Seismic Response Coefficient, C_s

$$C_s \quad = \quad \frac{1.2AS}{T^{2/3}}$$

where:

A = 0.4

S = 1.5

$$C_s \quad = \frac{1.2(0.4)(1.5)}{(0.5453)^{2/3}} \quad = 1.078 > 2.0A = 2.0(0.4) = 0.8$$

Therefore, $C_s = 0.8$

Calculate the Equivalent Static Earthquake Loading, $P_e(x)$

$$P_e(x) = \frac{\beta C_s w(x) v_s(x)}{\gamma} \; = \; \frac{7.277(0.8)(5.65)(0.01073)}{0.139} \; = 4.52 \text{ kft}$$

Calculate the Bearing Loads

$$\text{Force per bearing} = \frac{4.52 \times 120}{3 \text{ bearings}} \; = 180.8 \text{ kips}$$

Transverse Earthquake Loading

Calculate the Static Displacement, $v_S(x)$

Apply an assumed uniform transverse loading of 1 kip/ft to the bridge as shown in Figure TH-23. The resulting transverse displacements, vs(x), are tabulated at the span ¼ points and shown in the table on page A2-71. The displacements were determined by using frame analysis.

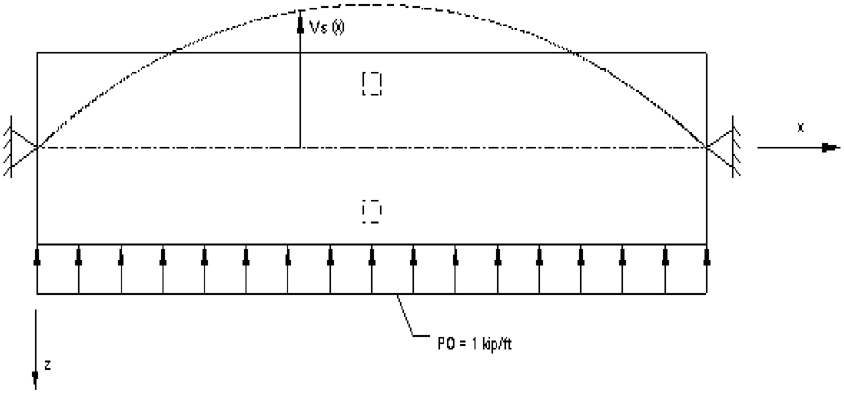


Figure B-23 Plan View of Two Span Bridge Subjected to Assumed Transverse Loading

Calculate the α, β, and γ factors by evaluating the integrals numerically

Wt = 5.65 k/ft

$$\alpha = \int_{\mathrm{Abut1}}^{\mathrm{Abut2}} v_s(x)dx \quad = 0.1055 \text{ ft}^2$$

$$\beta = \int_{\mathrm{Abut1}}^{\mathrm{Abut2}} w(x)v_s(x)dx \quad = 0.5961 \text{ kft}$$

$$\gamma = \int_{\mathrm{Abut1}}^{\mathrm{Abut2}} w(x)v_3(x)^2dx \quad = 0.00066 \text{ kft}^2$$

Calculate the Period, T

$$T = 2\pi\sqrt{\frac{\gamma}{p_o g\alpha}} = 2\pi\sqrt{\frac{0.00066}{1.0(32.2)(0.1055)}} \quad = 0.088 \text{ sec}$$

Calculate the Elastic Seismic Response Coefficient, Cs

$$C_s = \frac{1.2AS}{T^{2/3}}$$

where:

A = 0.4

S = 1.5

$$C_s = \frac{1.2(0.4)(1.5)}{(0.088)^{2/3}} = 3.65 > 2.0A = 2.0(0.4) = 0.8$$

Therefore, $C_s = 0.8$

Calculate the Equivalent Static Earthquake Loading, $P_e(x)$

$$P_e(x) = \frac{\beta C_s w(x) v_s(x)}{\gamma} = \frac{0.5961(0.8)(5.65)v_s(x)}{0.00066}$$

$$= 4080.3\ v_s(x)\ \text{kips/ft}^2$$

The following table shows the displacements and seismic loading intensity for transverse loading.

Displacements and Seismic Loading Intensity for Transverse Loading

Location	Displacements due to Uniform Transverse Loading $V_s(x)$, (ft)	Seismic Loading Intensity $P_e(x)$, (kips/ft)	Seismic Loads at Quarter Points, kips
Abutment 1	0	0	0
Span 1 1/4	0.000545	2.22	33.36
Span 1 1/2	0.000995	4.06	60.90
Span 1 3/4	0.001285	5.24	78.65
Pier 1	0.001384	5.65	84.71
Span 2 1/4	0.001285	5.24	78.65
Span 2 1/2	0.000995	4.06	60.90
Span 2 3/4	0.000545	2.22	33.36
Abutment 2	0	0	0

Calculate the Member Forces

From frame analysis, the transverse force on the pier = 66.2 kips

Force at each bearing = $\frac{66.2\ \text{kips}}{3\ \text{bearings}}$ = 22.07 kips/bearing

Example 2: Longitudinal Force Calculation for Pier in Tutor1

This example illustrates in detail the longitudinal load generation for the bridge pier, as shown in Figure TH-21. The load generation will be carried out in accordance with *AASHTO Standard Specifications for Highway Bridges*, Seventeenth Edition, 2002. The pier geometry and material properties used in this example are the same as those used in Tutorial 1.

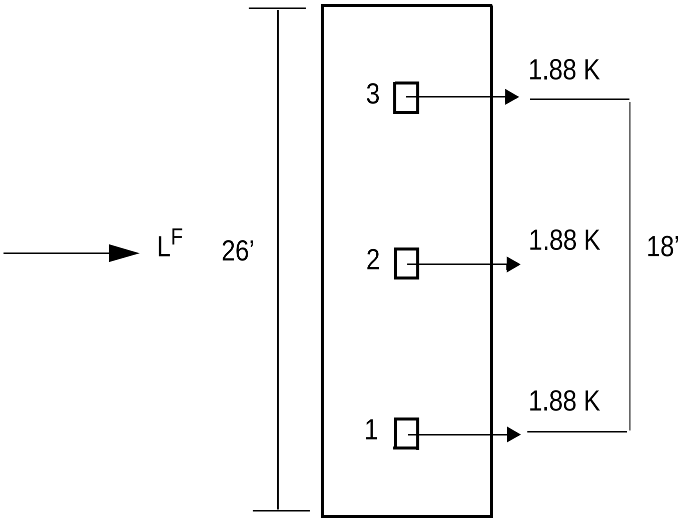


Figure B-24 Longitudinal Forces on Pier Bearings

Generating for H20/HS20 lane load:

Loading = 0.64 K/ft with 18k rider

Contributing length = 60 ft

Number of lanes = 2

Lane reduction factor for 2 lanes = 1

LF = 5% lane load x number of lanes x lane reduction factor

LF = 0.05 (0.64 x 60 + 18) x 2 x 1

= 2.82 x 2 x 1

= 5.64K

This force is applied 6 ft above the deck:

Arm = 6 + 36/12 + 8/12 = 9.667 ft

Load per bearing = 5.64/3 = 1.88k

Moment generated by LF and Arm:

Mx = 5.64 x 9.667 = 54.52

This moment is applied as Cap load as moment about X at the center of the CAP.

Example 3: Centrifugal Force Calculation for Pier in Tutor1

This example illustrates in detail the centrifugal load generation for the bridge pier, as shown in Figure TH-21. The load generation will be carried out in accordance with *AASHTO Standard Specifications for Highway Bridges*, Seventeenth Edition, 2002. The pier geometry and material properties used in this example are the same as those used in Tutorial 1.

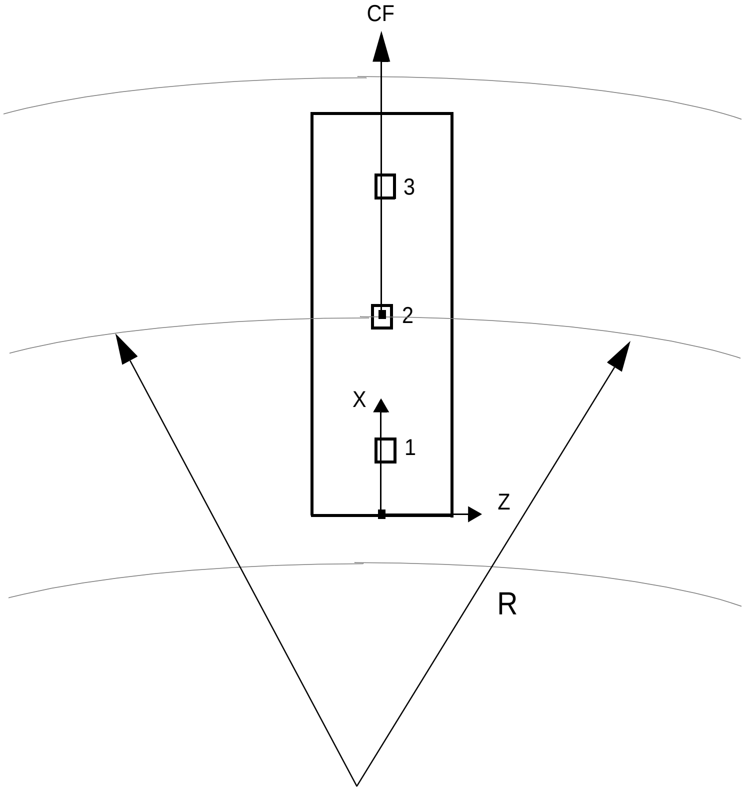


Figure B-25 Direction of Centrifugal Force on Pier Bearings

Considering HS20 Truck

Radius of curve = 1500 ft

Design Speed, S = 35 mph

Number of lanes = 2

Lane Reduction factor = 1

Considering the CF is applied in + X direction as shown in Figure TH-23.

$$C = \frac{6.68S^2}{R} = \frac{6.68(35)^2}{1500} = 5.455\% \text{ of LL}$$

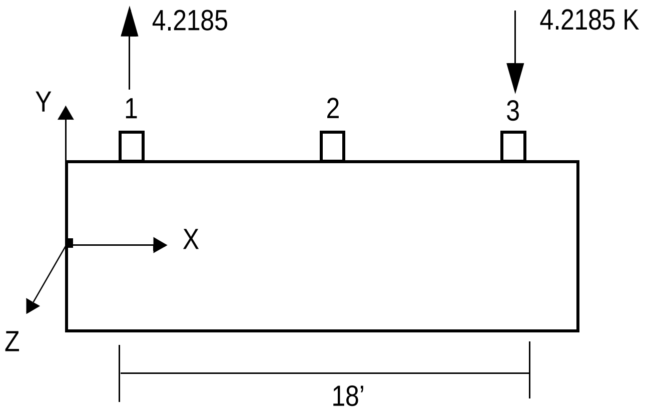


Figure B-26 Vertical Forces on Exterior Bearings

CF = 0.05455 x 72 * 2 x 1

= 7.855 Kip

Forces / bearing = 7.855/3 = 2.6184

CF is applied 6’ above the deck

Arm = 6 + 36/12 + 8/12 = 9.667’

CF generates moment about Z = axis, RC-PIER applies this moment as couple force in & direction.

M_z = 7.855 x 9.667 = 75.934

$$\text{Coupling force} = \frac{75.934}{18} = 4.2185 \text{ K}$$

References

1. American Association of State Highway Officials (AASHTO), *AASHTO LRFD Bridge Design Specifications*, Second Edition, Washington, D.C., 1998.
2. American Association of State Highway Officials (AASHTO), *AASHTO LRFD Bridge Design Specifications*, Interim Revisions, Washington, D.C., 2002.
3. American Association of State Highway Officials (AASHTO), *AASHTO LRFD Bridge Design Specifications*, Interim Revisions, Washington, D.C., 2003.
4. American Association of State Highway Officials (AASHTO), *AASHTO LRFD Bridge Design Specifications*, Third Edition, Washington, D.C., 2004.
5. American Association of State Highway Officials (AASHTO), *Standard Specifications for Highway Bridges*, Seventeenth Edition, Washington, D.C., 2003.
6. *PCI Bridge Design Manual*, 1st Phase, PCI, Chicago, October 1997.
7. American Concrete Institute, *Building Code Requirements for Structure Concrete (ACI 318-95) and Commentary (ACI318R-95)*, 1995.
8. Portland Cement Association, *Notes on ACI 318-95 Building Code Requirements for Structural Concrete with Design Applications*, 6th Edition, 1996.
9. Arthur H. Nilson and George Winter, *Design of Concrete Structure*, McGraw-Hill Inc., 11th Edition, 1991.
10. Joseph E. Bowles, *Foundation Analysis and Design,* McGraw-Hill Inc., 4th Edition, 1988.
11. S. U. Pillai and D. W. Kirk, *Reinforced Concrete Design*, McGraw-Hill Ryerson, 2nd Edition.

Index

IX