PERFORM-3D

Nonlinear Analysis and Performance
Assessment for 3D Structures

SHEAR WALL EXAMPLE

Prepared by
Graham Powell

JANUARY 2007

COMPUTERS & STRUCTURES INC.

COPYRIGHT

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA

tel: (510) 845-2177
fax: (510) 845-4096
e-mail: *info@csiberkeley.com*
web: *www.csiberkeley.com*

ACKNOWLEDGMENTS

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT, DOCUMENTATION AND TESTING OF PERFORM-3D. IN USING THE PROGRAM, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM. THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

THIS REPORT MAKES A NUMBER OF MODELING ASSUMPTIONS. GIVEN THE CURRENT STATE OF KNOWLEDGE, THE AUTHOR BELIEVES THAT THESE ARE REASONABLE ASSUMPTIONS. HOWEVER, THE MODELING METHODS USED IN THIS REPORT ARE SUGGESTIONS ONLY, NOT RECOMMENDATIONS. THE MODELING OF NONLINEAR SHEAR WALL STRUCTURES IS A COMPLEX TASK. THE MODELING ASSUMPTIONS ARE OPEN TO DEBATE, AND ARE SUBJECT TO CHANGE AS MORE INFORMATION BECOMES AVAILABLE.

PERFORM-3D Shear Wall Walk-Thru Example

Table of Contents

1 Description of Example Structure

1.1 Purpose

This example structure shows how PERFORM-3D might be used to model a shear wall structure. The example provides modeling suggestions only, not recommendations It is not promised that if you follow these suggestions you will have an appropriate model for practical purposes.

This example assumes that you are reasonably familiar with PERFORM-3D. If this is not the case, it is strongly recommended that you first study the steel frame "walk-through" example in the PERFORM-3D GETTING STARTED manual.

This example also assumes that you have an understanding of the PERFORM-3D Shear Wall and Frame elements, and how they can be used to model shear wall structures. Shear wall modeling is not a simple task. It is recommended that you obtain the DVD and seminar notes for the 2006 CSI Seminar on Performance based Design Using Nonlinear Analysis (available from CSI). The seminar includes a 90-minute session on modeling and performance assessment for shear walls, and the seminar notes include valuable additional material.

1.2 Number of Example Structures

To illustrate some aspects of design and behavior, the example consists of two structures, as follows.

(1) ***Original Structure***. This structure is similar to a design that might be produced using typical design procedures. The analyses show that this design does not have good performance.
(2) ***Revised Structure***. This structure is similar to a design that might be produced using displacement-based design and applying capacity design principles. It differs from the Original Structure in the amount and distribution of reinforcement. The analyses show that this design has better performance.

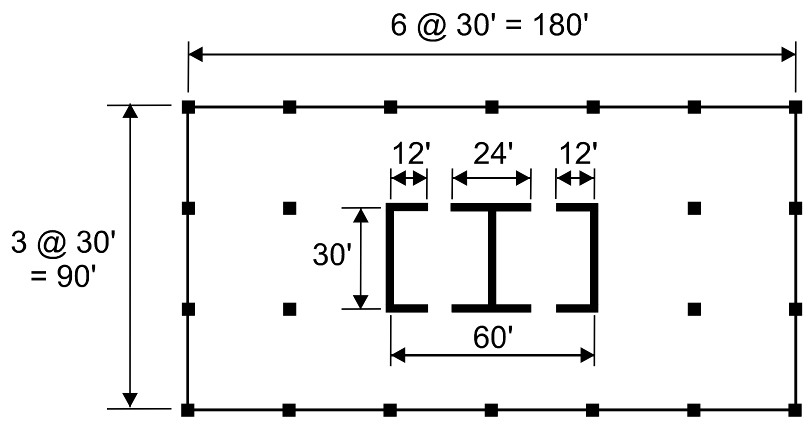
1.3 Installation

If you install PERFORM-3D Version 4.0.2 from the CD or DVD, the example structures and the required earthquake records are installed automatically.

Alternatively, you can install the examples directly. The examples are in the folders **ExampleCSI2A**, **ExampleCSI2B** and **ExampleCSI2C** on the CD or DVD, and the earthquake records are in the folder **WallExample**. To set up the examples, copy the ExampleCSI2A, ExampleCSI2Band ExampleCSI2C folders to the PERFORM-3D default Structures folder (or to some other folder if you prefer), and copy the WallExample folder to the PERFORM-3D Records folder.

1.4 Structure Geometry

The building plan is shown in Figure 1.1. Elevations of the shear core are shown in Figure 1.2.



All wall thicknesses = 18". All columns = 30" x 30".
Floor and roof slab thickness = 8".

Figure 1.1. Building Plan.

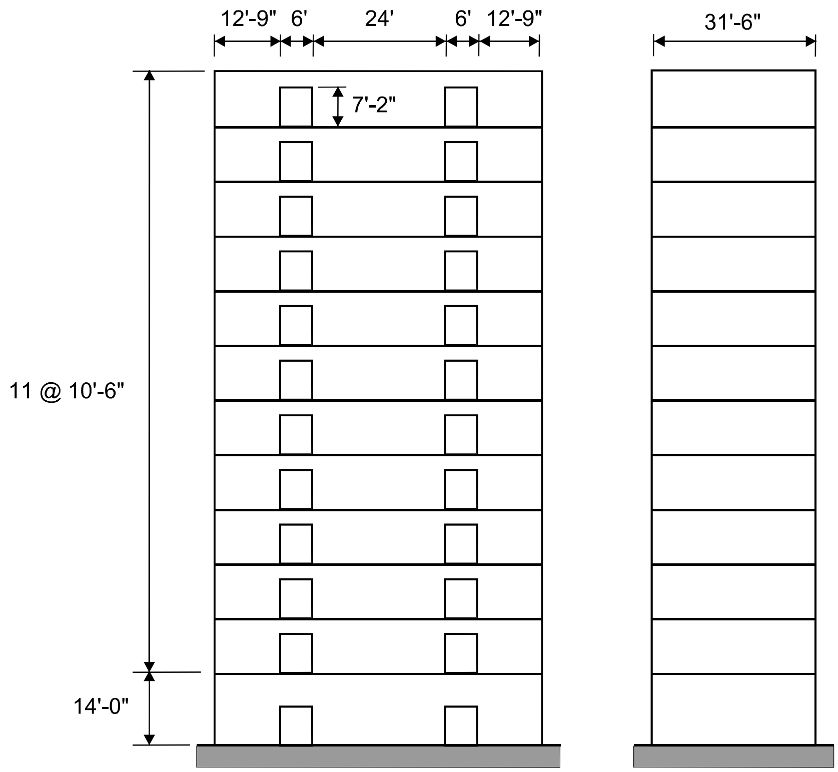


Figure 1.2. Shear Core Elevations

The emphasis is on the shear core. In the examples the core is assumed to resist the entire lateral load.

1.5 Gravity Loads

The dead loads are as follows.

(1) Floor and roof slabs = 100 psf.
(2) Gravity columns = 13 psf.
(3) Cladding, partitions, floor finish, etc. = 22 psf.
(4) Self weight of shear walls = 150 pcf. This is 370 kips for a clear story height of 10.5 feet, or about 23 psf when distributed over the floor area.

Hence, the floor dead load = 135 psf, and the total dead load, including wall self weight, = 158 psf.

The design live loads are as follows.

(1) Floor = 50 psf..
(2) Roof = 50 psf.

For earthquake analysis, 25% of the live load (12.5 psf) is assumed.

For simplicity in this example, the areas within the shear core are assumed to have the same dead and live loads as the rest of the floor.

1.6 Earthquake Loads

It is well known that different earthquakes, even with similar intensities, can cause substantially different responses. For that reason it is usual to run analyses for several ground motions that are representative of the building site.

The main purpose of this example is to illustrate modeling methods, not to assess the performance of an actual structure. However, performance assessment calculations are included in the example.

For the earthquake loads a set of six spectrum-matched ground motions is included in the **WallExample** folder. Three of these records are used for the analyses in this report. You can use the other records for additional analyses if you wish.

It may not be usual practice to use spectrum-matched motions. However, for this example it allows the results from dynamic analyses to be compared directly with those from static push-over analyses, since both are based on the same response spectrum. It also shows how the calculated dynamic response can vary from earthquake to earthquake even though the ground motions are nominally similar.

It must be emphasized that while the comparisons between dynamic analysis and push-over analysis is interesting, this report is by no means a rigorous study of the accuracy of push-over methods.

The response spectra, normalized to 1g maximum spectral acceleration, are shown in Figure 1.3, for all six motions.

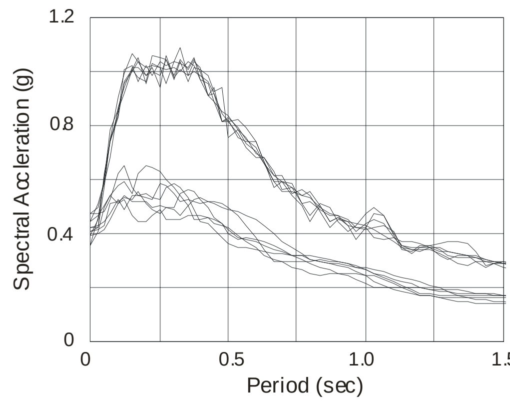


Figure 1.3. Response Spectra for 6 Ground Motions

Figure 1.3 shows the spectra for 5% and 20% damping. The spectra are closely similar for 5% damping (as they should be, since they are matched to this spectrum), but vary substantially for 20% damping. Figure 1.4 shows the mean spectra for 5%, 10%, 20%, 30% and 50% damping ratios.

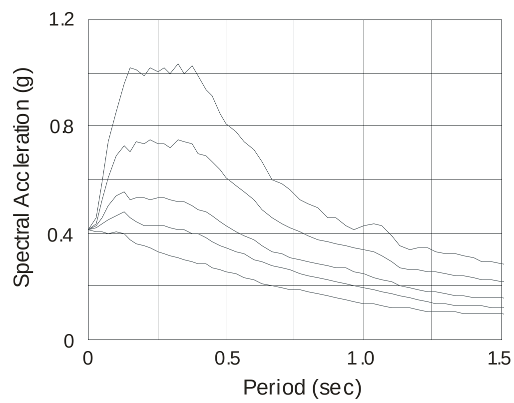


Figure 1.4. Response Spectrum for Push-Over Analysis

1.7 Performance Level and Intended Behavior

The structure is analyzed for the Collapse Prevention performance level. The desired behavior is as follows.

(1) The wall can hinge in bending at the base. The steel reinforcement can yield, but there should be little or no concrete crushing.
(2) The wall can crack in bending in the higher stories, but otherwise should remain essentially elastic (i.e., there should be little yielding of the reinforcement).
(3) The wall should remain essentially elastic in shear, including in the hinge region at the base.
(4) The coupling beams can yield. These beams are assumed to be controlled by shear.

1.8 Original Structure

The remaining sections in this chapter describe the reinforcement and other properties for the Original Structure. The performance of this structure is assessed in Chapter 3. Based on this assessment, the Revised Structure and its performance are considered in Chapter 4.

1.9 Wall Vertical Reinforcement

The vertical reinforcement at any story consists of a basic percentage plus additional reinforcement in the flange walls near the base of the structure.

The basic reinforcement extends over the full widths of the web and flange walls. Table 1.1 shows the basic reinforcement percentages. The additional reinforcement is added near the flange tips. Table 1.2 shows the additional flange reinforcement percentages, and the wall widths over which this reinforcement is added (measured from the flange tip).

The basic reinforcement has no confinement. The additional reinforcement has confining ties. The same steel properties are assumed for the basic and additional reinforcement.

TABLE 1.1
Wall Reinforcement

Story	Basic (unconfined) reinforcement		Additional (confined) reinforcement in flanges	
	Webs	Flanges	Area	Width (in)
1	0.75%	0.75%	0.3%	72
2				
3			0.3%	60
4	0.5%	0.5%		
5			0.3%	48
6				
7			0	0
8	0.35%	0.4%		
9				
10	0.25%	0.3%		
11				
12				

1.10 Wall Concrete Strength and Modulus

For the unconfined concrete, $f_c^{'}$ = 9000 psi, with a ductile limit strain of 0.0021. For the confined concrete, $f_c^{'}$ = 12000 psi, with a ductile limit strain of 0.0065. The elastic modulus in both cases is 5400 ksi.

1.11 Wall Shear Strength and Modulus

1.11.1 Shear Strength

It is not the purpose here to consider methods for calculating shear strength, or for designing the shear reinforcement. For the analyses, elastic shear behavior is assumed, and the shear stresses are calculated. For calculating the shear stresses, the effective section depth (i.e., effective wall width) is assumed to be 0.8 times the actual depth.

For the analyses, the shear strength in all stories is assumed to be 950 psi, corresponding to $10\sqrt{f_c^{'}}$ for the unconfined concrete. Shear strength D/C ratios are calculated at each story in the web and flange walls. The required shear reinforcement could be designed from these D/C ratios. For this example structure, only the D/C ratios are calculated, not the shear reinforcement. All D/C ratios must be smaller than 1.0, since $10\sqrt{f_c^{'}}$ is usually the maximum allowable shear stress.

The shear strength can be expected to be reduced in the hinge region (expected to be in the bottom story). Special design for shear would be needed in this region.

1.11.2 Shear Modulus

The shear modulus, G, for the material is chosen as follows. Young's modulus for the concrete is 5400 ksi. Assuming Poisson's ratio = 0.25, and using the usual formula for the shear modulus, G = 5400/2.5 = 2160 ksi. This value may be reasonable before any shear cracking occurs, but if there is significant cracking it is much too large. If this value is used, the shear strain at the shear strength of 0.95 ksi is 0.95/2160 = 0.00044, whereas in an actual concrete wall the shear strain at the point where the shear reinforcement yields is about 0.004, or roughly 10 times larger. When an elastic shear material is used, the effective shear modulus should be a secant value, somewhere between the uncracked modulus (equal to 2160 ksi in this case) and the secant modulus at yield of the reinforcement (equal to roughly 0.95/0.004 = 240 ksi in this case).

For this example a shear modulus G = 540 ksi is used (equal to one quarter of the 2160 ksi value). This modulus is used for both the walls (piers) and the coupling beams.

Since the wall is fairly slender (the height/width ratio for the web walls is 4.1), shear deformations may not be very significant, and hence inaccuracy in the shear modulus may not have much effect on the behavior.

1.12 Coupling Beam Strengths

The coupling beam strengths are assumed to be controlled by shear. As with shear in the walls, it is not the purpose here to consider methods for calculating shear strength, so the reinforcement details are not considered. For the purposes of assigning deformation capacities, the beams are assumed to be diagonally reinforced.

The shear strengths are shown in Table 1.2.

TABLE 1.2
Coupling Beam Shear Strengths

Stories	Shear Strength (k)
1	1500
2-7	750
8-9	575
10-12	425

For the coupling beam in story 1, using an effective beam depth of 0.8 times the overall depth, the shear strength corresponding to a stress of $10\sqrt{f_c^{'}}$ is (0.95ksi)(0.8)(82inches)(18inches) = 1121 kips, so diagonal reinforcement is required. For the coupling beams in the upper stories (40 inches deep) the shear strength for a stress of $10\sqrt{f_c^{'}}$ is 547 kips, so conventional shear reinforcement could be used for stories 10-12.

1.13 Columns and Floor Slab

For the analysis in this report, the shear core is assumed to resist all of the lateral load, and only this core is modeled, ignoring the stiffness and strength of the gravity columns. However, P-Δ effects in the columns are considered.

For analysis the floor slab is assumed to be a rigid diaphragm.

1.14 Floor Weights and Masses

The masses of the walls, floor slabs, columns, cladding and partitions are considered. The floor slab is modeled as a rigid diaphragm, and the masses are lumped at each floor.

With allowance for the shear core, the translational and rotational masses at each floor can be calculated approximately as follows.

Assume a rigid diaphragm 180 ft by 90 ft, with a uniform weight of W kips per square foot. The translational weight is (180)(90)(W) = 16200W kips. The rotational weight about the vertical axis is $(180)(90^3)W/12 + (90)(180^3)W/12 = 5.47\text{x}10^7W$ kip-ft^4.

For Dead Load, W is 158 psf for a 10.5 foot story. This corresponds to a translational weight of 2560 kips per floor and a rotational weight of $8.64\text{x}10^6$ kip-ft^4.

For 100% Live Load, W is 50 psf. The translational weight is 810 kips per floor and the rotational weight is $2.74\text{x}10^6$ kip-ft^4.

For DL + 0.25LL, the translational weight is 2760 kips per floor and the rotational weight is $9.33\text{x}10^6$ kip-ft^4.

These values are used at all floors and at the roof. A practical analysis might require a more accurate calculation.

2 Original Structure

2.1 Overview

This chapter walks you through the main features of the analysis model the loads, and the procedure for performance assessment.

2.2 Preliminaries

You must have installed PERFORM-3D, and you must have set up the example structures and the earthquake records, as described in the installation instructions.

As you take the tour, do not change any data. You can do this later if you wish (we suggest doing this on a copy of the example structure). If you do change data inadvertently, press the **Cancel** or **UnChange** button to undo the changes. If in doubt, exit PERFORM-3D, and when asked if you want to save the changes to the structure, choose "No".

Start PERFORM-3D. It helps if you have set up a desktop icon.

2.3 Open Structure

In the opening screen, press **Open an Existing Structure**. Then choose the **Default** option.

You should see **ExampleCSI2A** in the table. If you do not, this means that the example structures have not been set up in the default folder. If you have set up the examples in a different folder, use **Browse** to go to that folder. If you can not find the examples, press **Cancel** and then **Exit**.

When you see **ExampleCSI2A**, click on it to highlight it, then press **Open**. Alternatively, double click on it in the table.

You will get the OVERALL STRUCTURE screen. Press **OK**.

2.4 Nodes, Supports, Slaving

Press the toolbar button for the **Nodes** task.

The **Nodes** page shows the structure geometry. Right click on any node to show its coordinates. Use the view direction tools to look at plan and elevation views. Use the list of Frames in the toolbar to view different parts of the structure. See later for an explanation of the wall, beam and other elements.

Use the **Supports** page to see the support conditions. The supports at the base are all fixed. Foundation flexibility is not considered for this example.

Use the **Masses** page to see the mass patterns. These are explained in the next section.

Use the **Slaving** page to see the rigid floor constraints. There is a rigid diaphragm at each floor.

2.5 Masses

The masses are based on DL + 0.25LL. See Section 1.14 for the mass values.

Use the **Nodes** task and the **Masses** page. There are two mass patterns, with the same masses but different mass locations. Choose a pattern from the list to show the masses. The patterns are as follows.

(1) Masses are at the center of the structure, with no accidental torsion.

(2) Masses moved 9 feet along the H1 axis (= 5% of the 180 feet building dimension) and 4.5 feet along the H2 axis. This mass pattern can be used to account for accidental torsion. For an unsymmetrical structure it might be necessary to have more than one mass pattern of this type.

2.6 Material Properties

Press the toolbar button for the **Component Properties** task, then choose the **Materials** page.

2.6.1 Steel Material

In the list of inelastic material types choose **Inelastic Steel Material, Non-Buckling**. There is one material of this type, namely "All steel reinforcement", which is used for all steel fibers in the wall cross sections. Choose this material to see its properties. If you wish, press **Graph** to plot the stress-strain relationship.

Strain capacities are not specified for this material. As shown later, steel strains are used as D-C measures, but the D/C ratios are calculated using Strain Gage components and elements.

2.6.2 Concrete Materials

In the list of inelastic material types choose **Inelastic Concrete Material**. There are two main materials of this type, namely "Confined concrete" and "Unconfined concrete". Choose these materials to see their properties.

As with the steel material, strain capacities are not specified. Concrete strains are used as D-C measures, but the D/C ratios are calculated using Strain Gage components and elements.

The third concrete material is "Dummy material for steel areas", with a very small elastic modulus and large yield strains in tension and compression. The purpose of this material is explained later.

Some engineers treat the "shell" concrete, lying outside the reinforcement, as a material with a different set of material properties. In particular, the shell material is likely to be less ductile, and can spall off. This adds complexity, and may not be warranted given the many other approximations in the model. A simpler method is to ignore the shell concrete completely, and use a reduced wall thickness (for strength calculation, but not for weight).

For this example the shell concrete is not given special treatment.

2.6.3 Shear Material

As noted earlier, the walls are required to remain elastic in shear. In the list of elastic material types choose **Elastic Shear Material for a Wall**. There is one material of this type, namely "Elastic shear, G reduction factor = 4". This material is used for shear in all wall elements.

The material strength is 0.95 ksi, = $10\sqrt{f_c'}$ for the unconfined concrete. This stress is used to define the shear strength of the wall for strength D/C calculations.

A "Capacity Factor" of 0.8 is specified. This is to account for the effective section depth when shear stresses are calculated. When section areas are defined for shear stress calculation, the full section areas are specified. The capacity factor in effect reduces the section area. An alternative would be to specify reduced section areas, and a capacity factor of 1.0. The capacity factor is used because it is easier to change than the area of a shear section.

As noted earlier, a shear modulus G = 540 ksi is used.

2.7 Wall Elements

Before considering the fiber cross sections that make use of the steel and concrete materials, consider the wall elements.

The examples use PERFORM-3D Shear Wall elements, not General Wall elements. Shear Wall elements will usually be used for tall buildings. General Wall elements are intended mainly for squat walls, and are substantially more complex.

Press the toolbar button for the **Elements** task. There are nine element groups. For now consider only the first two, namely "Walls, basic steel" and "Walls, concrete + extra steel. The following are the main points for this section.

(1) In the Current Group list, choose these two element groups in turn. You will see that these two groups have the same elements. This means that there are two layers of elements, acting in parallel. The reason for this is covered later, when fiber sections and shear wall

compound components are considered. For this section this is not important.

(2) For each web wall there is one element across the 30-foot width. There is no need for more than one element. The wall is quite slender, and it is reasonable to assume that plane sections remain plane for bending. When this is the case, a single element across the width is sufficient, and it is a waste of computer time to use more elements.

(3) For each flange wall there is one element across the 12-foot width. Again, this is sufficient.

(4) In all stories except the bottom story there is one element over the story height. There is no need to use more than one element.

(5) In the bottom story there are either two or three elements over the story height. This requires some explanation, as follows.

For the view direction, push the button for the H2 Standard View. This shows the building elevation as viewed in the H2 direction.

In the center flange there are two elements over the story height in the bottom story. The upper element has a depth of 6.15 feet, which is 90% of the depth of the coupling beam. This is assumed to be the effective depth of the beam (this may be splitting hairs, and it would probably be OK to use the full beam depth of 6.83 feet). The lower element has a height of 7.85 feet, which is rather larger than the height of the door opening (7.17 feet).

The H2 Standard View also shows that the lowest coupling beams are modeled using horizontal beam elements (see later for details), and that they are connected to the adjacent walls by vertical "imbedded" beams (again, see later for details). In the other stories the coupling beams are still quite deep (40 inches). However, in those stories the beam elements are connected to the walls using horizontal "imbedded" beams.

In the outer flanges there are three elements over the story height in the bottom story. The uppermost element has a depth of 6.15 feet, which is 90% of the depth of the coupling beam. The lowest element has a height of 6.375 feet, which is the "hinge" length as considered later. The middle element covers the rest of the story height.

The lowest story height is 14 feet, which is the distance from the top of the floor slab to the top of the foundation beam or slab. This is usual practice, although it might be more accurate to use the distance from the mid-depth of the floor slab to some depth below the top of the foundation. This point is raised simply to emphasize that analysis models have approximations of many kinds, not only in the component properties but in the geometry.

The key point for this section is that there are single elements across the web and flange widths. This is important for setting up fiber sections, since the section widths are the same as the widths of the wall elements.

At the corners of the walls, the elements overlap as shown in Figure 2.1. Again, there are minor geometrical approximations.

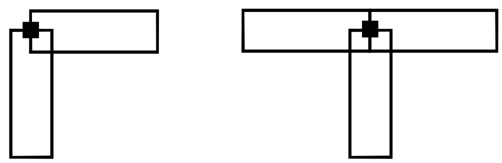


Figure 2.1. Nodes and Elements at Web-Flange Junctions

2.8 *Fiber Sections for Wall Elements*

2.8.1 Fixed Size and Auto Size Fiber Sections

There are two types of fiber section that can be specified, namely "Fixed Size" and "Auto Size". For a Fixed Size section the area and location must be specified for each fiber. This type of section is very general, but requires substantial care and effort. For an Auto Size section, only the numbers of steel and concrete fibers need to be specified, and PERFORM-3D calculates the fiber areas and locations, assuming equal size fibers. This is more restrictive, but requires less effort.

For this example, Auto Size sections are used to set up the fibers for the basic steel reinforcement, and Fixed Size sections are used for the additional reinforcement and the concrete. This is not the only way to specify the sections, but it can be convenient.

Auto Size sections are convenient for the basic reinforcement, because the reinforcement it is uniform across the full cross section width. Fixed Size sections are needed for the additional reinforcement, because it is present over only a part of the cross section width. Fixed Size sections are also usually better for concrete fibers because the fiber areas can be made smaller towards the outer edges of a wall, to allow concrete crushing to occur. If Auto Size sections are used for concrete, the fiber areas are equal, which may not be desirable.

2.8.2 Element (and Section) Widths

For a Fixed Size section, the section dimensions are determined by the fiber locations. Usually, when a Fixed Size section is used in a wall element, the section width is equal to the element width. However, this is not essential, and PERFORM-3D does not check. Hence, when you define and use Fixed Size sections, you must be careful to be consistent.

The situation is different for an Auto Size section. When an Auto Size section is defined, it does not have specified fiber locations or areas. Instead, when the section is used for a wall element, the section width is set equal to the element width, and the required fiber areas and locations are calculated by PERFORM-3D. If desired, the same Auto Size section can be used for elements of different widths (the elements will all have the same wall thickness, steel and concrete materials, steel percentage, and numbers of steel and concrete fibers).

For the example structure, a trick has been used to minimize the use of Fixed Size sections. The procedure is as follows. This is not necessarily a procedure that can be used in all cases, but it is convenient for this example and is worth considering for other structures.

(1) In the web walls, the steel percentages vary with height, as shown earlier in Table 1.1. In these walls there is only "basic" steel, which is uniform across the wall width. Auto Size sections are used to specify the steel fibers, one section for each different steel percentage. There are 4 sections of this type.

(2) The concrete fibers for the web walls could also be specified in these sections, but in the present version of PERFORM-3D this means that the concrete fibers would have equal areas. It is often preferable to have smaller fibers near the ends of the section, and

relatively larger fibers near the center of the section. The smaller fibers allow better modeling of concrete crushing near the ends of the cross section. Hence, a second fiber section, of Fixed Size type, has been set up to define the concrete fibers. There is one section of this type.

(3) In the flange walls, the steel percentages vary with height, as shown earlier in Table 1.1. In these walls there is both "basic" steel and "additional" steel. For this example, Auto Size sections are used to specify the steel fibers for the basic steel, one section for each different steel percentage. There are 4 of these sections. Separate Fixed Size sections are used for the "additional" steel fibers and the concrete fibers. There are 4 of these sections.

There are 13 different fiber sections, as considered below. See later for how these sections are used in Wall Compound Components and Shear Wall elements.

2.8.3 Auto Size Sections for Basic Reinforcement

In the **Component Properties** task, choose the **Cross Sections** page. Then choose the **Shear Wall, Inelastic Section** type. Choose sections in the name list to see their properties.

The first 8 sections in the list of sections are Auto Size sections, which define only the "basic" steel. The first 4 sections are for the web walls. These walls are 30 feet wide, and the sections have 8 steel fibers. The next 4 are for the flange walls. These walls are 12 feet wide (measured each side of the web for the I-section walls), and have 4 fibers.

There is an additional trick that is needed. The steel area is specified as a percentages of the concrete area, which means that a concrete material and a wall thickness is needed. Since these sections define steel fibers only, the concrete material is the "Dummy material for steel areas" that was considered earlier. This material has a very small modulus and remains elastic, so it adds negligible stiffness and does not have nonlinear events. There is only one concrete fiber,

2.8.4 Fixed Size Sections for Concrete and Additional Reinforcement

The 9th section is for concrete in the web walls, with no steel. The fiber areas are smaller towards the ends of the section.

The last 4 sections define fibers for the concrete and the additional reinforcement in the flange walls. The left edge of the section is at the mid-thickness of the web, and the right edge is at the flange tip. The additional steel is concentrated towards the flange tip, and the concrete fiber areas are smaller towards the ends of the section.

Note that for Fixed Size sections, the origin for the fiber coordinates is at the center of the cross section, not at one edge.

2.9 Shear Wall Compound Components

In the **Component Properties** task, choose the **Compound** page. Then choose the **Shear Wall Compound Components** type.

2.9.1 Use of Fiber Sections

There are 13 Shear Wall Compound Components, corresponding to the 13 fiber cross sections.

The first 8 components are based on the steel-only fiber sections. For each of these components the cross section for vertical axial/bending is the corresponding fiber section. For horizontal axial/bending and for shear the wall thicknesses, and hence the stiffnesses, are very small. The remaining 5 components are based on the fiber sections for the concrete and additional steel. These components have the "correct" thicknesses. You can see this by choosing components in the Name list.

The reason for using small thicknesses for the basic steel layer is that there are two layers of wall elements, the first to account for the basic steel, and the second to account for the concrete and additional steel. The horizontal and shear stiffnesses must be included in only one of those layers. In this example they are included in the components that account for the concrete. (It may be noted that it is essential not to "double-dip" on shear, but it does not matter much for horizontal axial/bending.)

2.9.2 Wall Self Weight

Choose a component from the name list, then choose the Self Weight page. For the sections that include concrete fibers, the wall thickness and concrete density are specified (8.68×10^{-5} kips per cubic inch = 150 pcf). This is used to set up self weight load cases, as considered later.

2.10 Wall Elements

2.10.1 Parallel Elements

As considered in the preceding section, there are two sets of fiber sections and compound components, one set for the basic steel and the second set for the concrete and additional steel. There are also two sets of elements, acting in parallel.

Press the toolbar button for the **Elements** task. There are nine element groups. For now consider only the first two, namely "Walls, basic steel" and "Walls, concrete + extra steel".

2.10.2 Element Orientations

Choose either of the above two element groups, and choose the **Orientations** page. This page is used to set up the local axes for wall elements. Axis 1 is normal to the element, Axis 2 is along the fiber directions, and Axis 3 is the transverse direction (in this direction the element behaves elastically). For all elements, the orientation option is "Axis 2 is parallel to edge IK". The axes are marked by short lines at the I node of each element. In other views the I-node is marked by a small dot, located a short distance along edge IK from the I-node

You can see the elements more clearly if you choose one of the Frame views from the list of frames on the toolbar. For example, choose the "South wall" frame, then press the S-F (Structure-Frame toggle) button to show only this frame.

Note that for the flange elements in the left (West) wall, Node I is on the left, adjacent to the web wall. Hence, the fiber coordinates for the flange elements are outwards from the web wall (measured from the section center). However, for the flange elements in the right (East) wall, Node I is on the right, again adjacent to the web wall. In both cases, therefore, the fiber coordinates for the flange elements are outwards from the web wall. This means that the same fiber sections and wall compound components can be used for both sets of flange walls. If Nodes I were at the left (West) edge for both sets of flange elements, different fiber sections and wall compound components would be needed for each flange. The key point is that you can save effort if you plan ahead when you choose the Node I-J-K-L sequence for the elements.

The Node I locations for the center wall flanges have similar locations, namely adjacent to the web wall for both flanges.

2.10.3 Element Properties

Choose either of the above two element groups, and choose the **Properties** page. This page is used to assign properties to elements. In the case of wall elements, each element must be assigned a corresponding compound component.

In the graphics panel, click on an element. The element color changes to red, and the data panel shows the compound component for that element. To see what makes up the compound component, press the **Show Properties** button. To show all elements that have the same properties, double-click on an element. All elements with the same properties are shown red. You may have to press the **Clear Selected Elements** button before you double-click.

2.10.4 Hinge Lengths

The intended behavior is that bending hinges can form near the base of the wall, but there should be no hinging in the upper stories.

Near the base of the wall it is important to use a "correct" hinge length (this is considered in some depth in the CSI Seminar). This length is important because it is the gage length for calculating rotations and strains. ASCE 41 specifies a hinge length equal to 0.5 times the wall width, but not more than the story height.

For load in the H2 direction, the wall width is 31.5 feet. Hence the hinge length is the smaller of 0.5 x 31.5 = 15.75 feet or the story height of 14 feet, and the story height governs.

For load in the H1 direction it is more complex, because the wall is coupled and has different widths in the outer and inner walls. There are basically two possible assumptions, as follows.

(1) If a wall structure has coupling beams that are very stiff and strong, so that the wall behaves like a solid wall, it is probably reasonable to base the hinge length on the total width of the wall.

(2) If the coupling beams are weak, so that the walls act substantially independently, it is more reasonable to use a different hinge length for each pier, based on the pier width.

With the second of these assumptions, the hinge length is smaller, which tends to be more conservative.

For this example structure, the second assumption is used. The hinge lengths are as follows.

(1) In the outer walls the pier width is 12.75 feet, corresponding to a hinge length of 6.375 feet. This height is used for this example.

(2) In the inner wall the pier width is 24 feet, corresponding to a hinge length of 12 feet. This is smaller than the story height, but larger than the height of the door opening (7.17 feet). Above the door opening there is a deep coupling beam. For this example it is assumed that the hinge does not extend into the coupling beam. As noted earlier, the effective depth of the coupling beam is assumed to be 90% of the overall depth, or 0.9 x 82 inches = 6.15 feet. Hence, the effective height for the door opening is 7.85 feet. This height is used for the hinge length.

2.10.5 Element Lengths (Heights) in Bottom Story

The length (height) of an element must not be larger than the corresponding hinge length.

In the flanges for the outer walls the length of the lowest element is 6.375 feet. A second element extends 1.475 feet to the bottom of the coupling beam, and a third element extends 6.15 feet to the top of the story. The elements in the web have the same heights. This means that there are three elements over the hinge length in the web walls.

In the flanges for the inner wall the length of the lowest element is 7.85 feet. A second element extends 6.15 feet to the top of the story. The elements in the web have the same heights, so there are two elements over the hinge length in the web.

2.10.6 Element Lengths in Upper Stories

For the remaining stories, cracking of the concrete is allowed, but there should be no significant yield of the steel, and hence no significant hinge rotation. In each story there is a single element over the story height. This means that the gage length for calculating steel strains is the story height. This also means that the coupling beams in the upper stories are assumed to be slender, as considered later.

2.10.7 Out-Of-Plane Behavior

In shear wall elements, the behavior for out-of-plane bending is assumed to be elastic. In the **Component Properties** task choose the **Cross Sects** page and the "Shear Wall, Inelastic Section" type.

Choose one of the "Steel only" sections from the Name list. For the Out-of-Plane Bending properties, the wall thickness and modulus are very small. This is done because the out-of-plane bending stiffness is included in the sections with concrete fibers. Choose one of the concrete sections in the Name list. For these sections the wall thickness is 18 inches and Young's modulus is assumed to be 1350 ksi, or one quarter of the modulus for concrete fibers in the fiber section, to allow for stiffness reduction when the concrete cracks.

Since the walls are much stiffer in-plane than out-of-plane, the stiffness contribution of the out-of-plane bending is only a small part of the total stiffness. However, for push-over analyses in which brittle strength loss is considered, particularly brittle strength loss in shear, it can happen that after strength loss has occurred, out-of-plane bending can contribute substantially to the stiffness and strength. For this reason, some engineers believe that the out-of-plane stiffness should be ignored. This can be done by specifying a small out-of-plane thickness for all wall sections. It is debatable whether this assumption is necessary, but it is conservative.

2.10.8 P-Δ Effects in Wall Elements

In the **Elements** task choose the **Group Data** page. Note that the option for Geometric Nonlinearity is "None", which means that P-Δ effects are not considered in the wall elements. P-Δ effects can be considered in wall elements if desired, but this is not done for this example, for a number of reasons.

The main reason is that P-Δ effects can be considered using "P-Δ column" elements, as considered later.

A second reason is that if the out-of-plane bending thickness were made small, as suggested in the preceding section, the buckling strength of the wall for out-of-plane bending would also be small. Hence, if P-Δ effects are considered in the wall elements, the analysis may predict local buckling failure of the wall. In this case there is no is no solution, since the structure theoretically can not support the gravity load, and the PERFORM-3D analysis will fail to converge. This will not happen in all cases. For example, if there is only one wall element over the height of each floor, there will be no local buckling. In general, however, local buckling could occur, so if the wall thickness is made artificially small for out-of-plane bending, P-Δ effects should not be specified in the wall elements. P-Δ effects can, however, always be considered using a P-Δ column.

A more subtle reason has to do with the way that shear stresses are calculated in the walls (using Structure Sections). This is considered later.

2.11 Coupling Beams

2.11.1 Wall vs. Beam Model

Coupling beams can be modeled using either wall elements or beam elements. In most cases, beam elements are better because they allow better control over the modeling assumptions.

The coupling beams in the example structure are quite deep (40 inches deep in the upper stories, 82 inches deep in the bottom story), and it may seem to be incorrect to model then using slender beam elements. However, the bending and shear stiffnesses and strengths for a deep beam can modeled using a beam element, provided the connection of the beam to the walls accounts for the beam depth (as considered below). Also, the behavior of coupling beams can be complex, and it is not necessarily accurate to model them using wall elements. The goal is to model the behavior of the beam and its interaction with the walls, not to model its geometry.

Figure 2.2 shows two alternative ways of modeling the coupling beam in the bottom story, one using beam elements and one using a wall element.

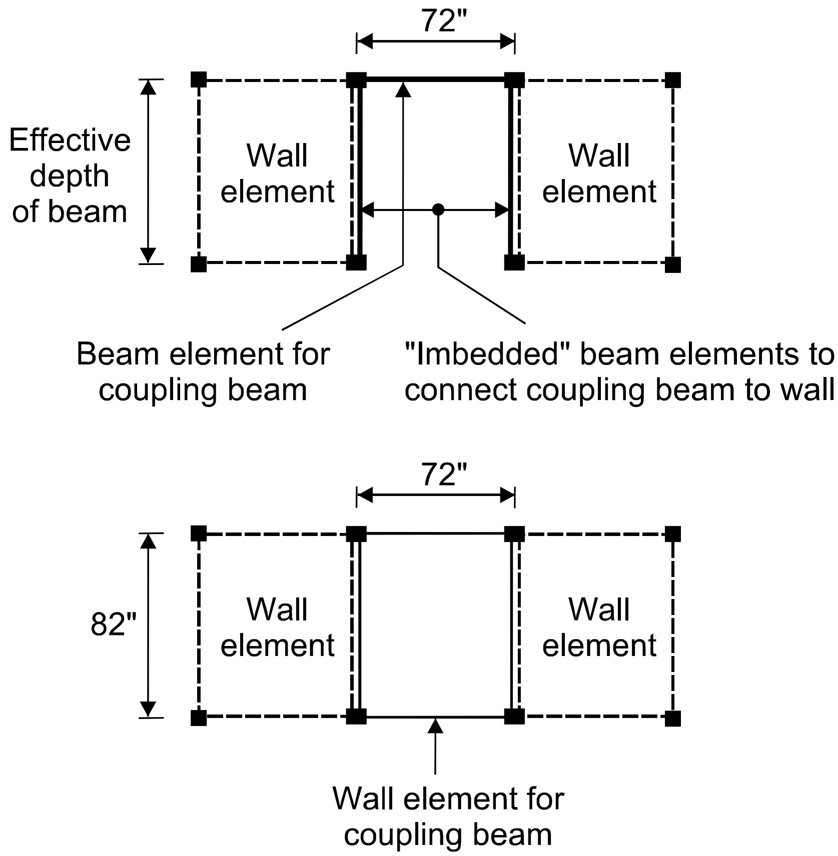


Figure 2.2. Alternative Models for Deep Coupling Beam

These two alternatives are considered in the next two sections. For the example structure, the beam element model has been used.

2.11.2 Deep Coupling Beam Using Beam Elements

For the beam model the assumptions for the example structure are as follows.

(1) The beam compound component consists of two elastic beam segments and a rigid-plastic shear hinge.

(2) The cross section of the beam is a rectangle, 18 inches wide by 82 inches deep. This ignores any composite action from the floor slab. However, it is likely that composite action affects only the bending stiffness, not the shear stiffness. For a beam with this span-to-depth ratio (72/82, = 0.88), the stiffness is controlled by shear. Increasing the bending stiffness to account for composite action with the slab has a negligible effect.

(3) The elastic modulus for bending is assumed to be one half of Young's modulus for the concrete (0.5 x 5400 = 2700 ksi). This could be any reasonable value.

(4) The shear modulus is assumed to be 540 ksi, the same as used for the walls. This is very approximate, but the elastic shear stiffness is probably not important. Most of the shear deformation is in the shear hinge, after the beam has yielded in shear. It may be noted that for a beam section in PERFORM-3D, the shear modulus, G, can not be specified directly. Instead G is calculated from E (= 2700 ksi) and ν, using $G = 0.5E/(1+\nu)$, or $\nu = 0.5(E/G-2)$. To get $G = 540$ ksi, it is necessary to specify $\nu = 1.5$.

(5) The coupling beam is located at the floor level. It can be argued that the beam axis is below this level. However, it does not have much effect, because of the way that the beam is connected to the walls. This is done using vertical "imbedded" beams, as shown in Figure 2.2. The shear force in the beam is transferred to the walls at the beam level. The axial force is unknown, because of the rigid floor diaphragm, but this force is also transferred to the walls at the floor level. The beam bending moment at the end of a beam is transferred to the walls as a tension-compression couple with a lever arm equal to the beam depth. These should be reasonable assumptions.

(6) The imbedded beams are very stiff in bending, but have negligible axial stiffness. As the walls bend, they extend of compress the imbedded beams. If these beams were axially stiff they would stiffen the wall locally. A deep coupling panel may stiffen the walls in an actual structure, but the amount of stiffening is difficult to assess. When the imbedded beams have negligible axial stiffness, the stiffening effect is assumed to be zero, which should be conservative.

This list of points shows that the modeling of coupling beams is not a simple task. The important point is to capture the essential aspects of beam behavior, in a way that is accurate enough for design. It is impossible to be "exact".

2.11.3 Deep Coupling Beam Using Wall Element

For the wall element model, the assumptions might be as follows.

(1) For a coupling beam with a span/depth ratio smaller than one, a single wall element is sufficient.
(2) For vertical and horizontal axial-bending behavior, the wall element is elastic (i.e. use an Elastic Material for Fiber Sections material and a Shear Wall, Elastic Section cross section).
(3) For vertical and horizontal axial-bending behavior, make the element very flexible, by specifying a small elastic modulus or a small cross section area, or both. If this is done the coupling beam element does not stiffen the adjacent wall elements in bending (see Point 6 in the above list for the beam element model). For horizontal behavior, the floor diaphragm transfers horizontal axial forces into the walls, similar to a beam element model.
(4) For shear behavior specify an Elastic Shear Material for a Wall, with an appropriate shear modulus (e.g., 540 ksi) and strength (e.g., 0.95 ksi). Then specify a Shear Wall Compound Component that uses this material.

Some alternative models using wall type elements are as follows.

(1) Use an Infill Panel, Shear Model component and a corresponding element. This component has only shear stiffness.
(2) Use a Shear Wall element with a fiber section for vertical behavior. Such an element can stiffen the adjacent walls as they bend.
(3) Use a Shear Wall element that is rotated 90 degrees so that the fiber section applies for horizontal behavior.
(4) Use a General Wall element, with fiber sections for vertical and horizontal behavior.

2.11.4 Slender Coupling Beam Using Beam Element

For a coupling beam that is more slender, a beam element can be used. If the beam strength is clearly governed by shear, the model can have elastic beam segments and a shear hinge, as above. If the beam is governed by bending, it should have moment hinges (or should use FEMA Beam components).

For a slender beam, the "imbedded" beams that connect the beam to the walls can be horizontal, rather than vertical.. This has the advantage that it reduces the number of wall elements.

In the example structure, horizontal imbedded beams have been used for the coupling beams in the upper stories. These coupling beams have 40 inch depths and span/depth ratios of 1.8, so they are not really slender. Even so, it is assumed that they can be treated as slender beams for the purposes of analysis.

2.11.5 Slender Coupling Beam Using Wall Elements

Slender coupling beams can also be modeled using wall elements.

It may require a number of wall elements along the span. If bending deformations are significant, it may be necessary to use inelastic fiber sections for horizontal behavior. Note, however, that if there is a rigid floor diaphragm, the neutral axis for bending is at that level. Also note that if a cross section has concrete fibers, and if those fibers crack, the coupling beam must extend. If there is a rigid diaphragm it can not extend at the floor level, Also, if the adjacent shear walls are stiff, they may restrain cracking, which increases the bending strength of the cross section.

Axial extension is a real effect in concrete beams, but it is not easy to model. If axial expansion is present in the real structure, extra care is needed in the modeling.

In general, it is not easy to capture beam bending behavior using wall elements.

2.11.6 Which Type of Model is Better?

A model using beam elements has the following advantages.

(1) A beam model can use shear hinges, moment hinges, elastic segments, fiber segments, etc. Hence, it is easier to control the behavior of a coupling beam if a beam model is used rather than a wall model.
(2) A beam model can be used equally well for both deep and slender coupling beams. It is more difficult to model slender beams using a wall model. The reason is that with a wall model the bending stiffness and strength depend entirely on the fiber properties and orientations, whereas in a beam model the bending behavior can be specified in a number of ways.

(3) With a beam model, if horizontal imbedded beams are used, there need to be only one wall element over the height of each story. With a wall model, the adjacent walls must have at least two elements over the story height.

The first choice should usually be a beam model.

2.11.7 Diagonally Reinforced Coupling Beams

Diagonally reinforced coupling beams behave differently from beams with conventional shear reinforcement. In particular, if the beam is to behave as intended, the tension diagonal must extend substantially (the concrete cracks and the diagonal steel yields), whereas the compression diagonal is stiffer (because of compression in the concrete) and has a smaller deformation. This means that the beam as a whole must extend (i.e., there must be axial growth). If this axial growth is restrained, by the floor slab or the adjacent walls, the behavior of the beam may not be the same as that observed in a test with no axial restraint.

This is a complex problem. Usual practice seems to be to assume that axial restraint does not have a substantial effect. For the example structure, the coupling beams are assumed to be controlled by shear, and inelastic shear deformation is modeled using shear hinges, which follows usual practice. When these hinges yield, there is no axial extension. This may not be an accurate model. Research is needed to decide this issue.

It may also be noted that if wall elements are used to model coupling beams, or if beam elements with fiber sections are used, cracking of the concrete fibers causes a shift in the neutral axis, which in turn causes axial growth of the beam. If this growth is restrained, cracking is suppressed and the beam is effectively stronger in bending. Some engineers believe that this can be a substantial effect. Research is also needed to decide this issue.

2.11.8 Deformation Capacities

The beams are controlled by shear, and are modeled using rigid-plastic shear hinges. The deformation capacities for the hinges are based on ASCE 41 for diagonally reinforced coupling beams.

For the Collapse Prevention performance level, the plastic shear rotation (shear strain) capacity is 0.03 radians. For the beam span of 72 inches, this correspond to a displacement across the shear hinge of (0.03)(72) = 2.16 inches.

2.11.9 Element Locations

In the **Elements** task, choose the "Coupling beams" element group. The coupling beam elements are shown in light blue. If you wish, use Frame views to show the elements more clearly. All coupling beams are in this group. They could be divided into smaller groups if desired. The elements are of Beam type.

The beam axes are located at the floor levels. The vertical coordinates of the nodes in the model correspond to the top of the floor slab. It might be more accurate to locate the nodes 4 inches lower, at the slab mid-thickness, but the difference is likely to be negligible.

2.11.10 Element Properties

In the **Elements** task, choose the "Coupling beams" element group and the **Properties** page.

Click on an element in the graphics panel, or double-click to select all elements with the same properties. Press the **Show Properties** button to show the Frame Compound Component for the element. Click on a component in the list, then press the **Show Properties button** to show the properties for that component. In particular, note the F-D relationship, strength loss, deformation capacities and cyclic degradation parameters for the shear hinge. Press the **Close** button to exit.

For the shear hinges the F-D relationship is trilinear. Yield is assumed to occur at 50% of the shear strength, and the U point deformation is 0.288 inches, corresponding to an average shear strain of 0.004 over the 72 inch beam length. The residual strength is 80% of the U point strength. Energy degradation factors are specified for the hysteresis loops. These are not necessarily the values that should be used for an actual structure.

2.11.11 Imbedded Beams

When beam elements are connected to wall elements, additional "imbedded" beams must be used to connect each coupling beam element to the wall elements. If imbedded beams are not used, the coupling beams are, in effect, pin-connected to the walls.

In the **Elements** task, choose the "Imbedded beams" element group and the **Properties** page. Press the **Show Properties** button to show the Frame Compound Component, etc.

For the horizontal imbedded beams, the stiffness is based on a rectangular section that is 20 times stiffer for vertical bending than the coupling beam section (the section width is 20 x 18 = 360 inches). For horizontal bending the stiffness is very large, which does not matter since there is a rigid floor diaphragm. If there were no rigid diaphragm, an appropriate value for this stiffness would be needed. The axial and torsional stiffnesses are set very small, to avoid stiffening the wall elements.

Similar properties are used for the vertical imbedded beams, except that the horizontal bending stiffness is made small to avoid stiffening the walls.

2.12 P-Δ Columns

In the **Elements** task, choose the "P-delta columns" element group. If you wish, use a Frame view to show the elements more clearly.

There are two P-Δ columns, one at the nominal center of gravity of the floors, and one offset 5% of the building dimensions along the H1 and H2 axes. The nodes for these columns are the same as those for the masses. It is not required to consider accidental torsion for P-Δ effects, but it can be argued that it is consistent to do so.

The sole purpose of a P-Δ column is to account for P-Δ effects. The elements are Simple Bar elements with Elastic Bar components. The bar stiffness is a reasonable value, not so small that the loads on the column cause large deflections, and not astronomically large. P-Δ effects are considered in these elements. At each floor the gravity loads on the floor are applied to the P-Δ column. At each floor, the nodes on the columns are included in the rigid floor slaving constraint.

In the example structure, P-Δ effects are considered only in the P-Δ column elements. As noted earlier, P-Δ effects are not considered in the wall elements. This has the advantage that the gravity load for an entire floor can be applied to the P-Δ column. If P-Δ effects were considered in the wall elements, the gravity load on the P-Δ column at each floor would have to be the total floor load minus the load applied to the walls.

2.13 Floor Loads on Wall

2.13.1 Methods for Applying Gravity Load to Wall

Gravity loads on the walls put them in compression. Vertical compression forces in a wall can have significant effects on the wall behavior, because they delay cracking of the concrete fibers. Hence, it is important to apply reasonably accurate gravity loads directly to the walls. In the actual structure, gravity loads on the wall consist of the following.

(1) The wall self weight. This is well defined and easy to model.

(2) Dead and live loads from the floor slabs. These depend on complex interactions between the walls and the slabs, and are of uncertain magnitude and distribution. They may also change significantly when the structure is loaded laterally and the slab bends.

It is possible to model the floor slab explicitly (and also the gravity columns), and to get the gravity loads on the wall by analysis. One way is to use slab finite elements, but this has the major disadvantage that it adds greatly to the complexity of the model and the computer time required to run nonlinear analyses. It is also not necessarily accurate. (It may appear to be accurate, but this is probably an illusion. Even an elaborate finite element mesh fails to capture many important aspects of behavior, such as cracking of the slab, long term creep, construction sequence effects, and complex interactions between the slab and the walls.)

For the example structure the floor slab is not modeled. Instead, an estimate is made of the gravity loads that the floor exerts on the walls, using tributary areas, and these loads are applied directly to the walls.

2.13.2 Loads Using Tributary Areas

At each floor there are 14 nodes that connect to wall elements. Figure 2.3 shows the assumed tributary areas. A nodal load pattern has been set up using these areas.

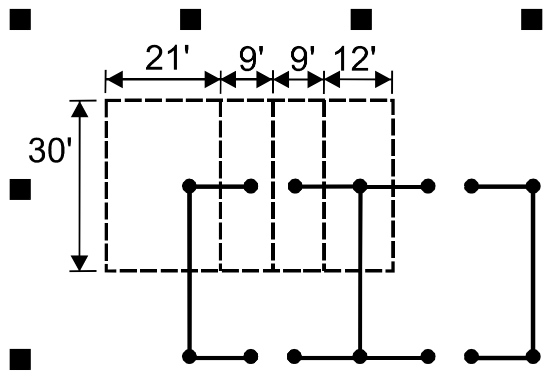


Figure 2.3. Tributary Areas for Floor Loads on Wall

Press the toolbar button for the **Load Patterns** task, and choose the **Nodal Loads** page.

A single load pattern has been set up, based on a load of a load of 1.0 ksf on the tributary areas shown in the figure. The loads are scaled for the actual load per square foot (135 psf DL plus 0.25 x 50 psf LL = 147.5 psf) when the load pattern is used in a gravity load case.

2.14 Rotation Gages and Rotation Capacities

2.14.1 Hinge Lengths and Element Heights

The hinge lengths (heights) at the base of the wall were considered earlier. For performance assessment, the main demand-capacity measure in ASCE 41 is hinge rotation, measured over the hinge length.

As considered earlier, the hinge length affects the element heights in the bottom story, as follows.

(1) In the flanges of the outer walls the hinge length is 6.375 feet. The lowest wall element has this height, and the rotation demand is calculated over the height of this element. The second element extends to the effective height of the door opening. The bottom story has three elements over its height.
(2) In the flanges of the inner walls the hinge length is the effective height of the door opening. The lowest element has this height, and the rotation demand is calculated over the height of this element. The second element extends to the top of the story. The bottom story has two elements over its height.
(3) The element heights in the web walls match those in the flange walls. The hinge length is equal to the story height, so the rotation demand is the sum of the rotations over three elements in the outer walls, and two elements in the inner wall.

In each case the rotation over the hinge length is calculated using Rotation Gage elements.

For stories above the base, hinges should not form. However, it is useful to check rotations in those stories. For this purpose, it is reasonable to calculate the rotation over each story, using Rotation Gage elements. More importantly, there should be little or no yield of the reinforcement in the higher stories. To check this, Strain Gage elements are used, as explained later.

2.14.2 Rotation Capacities

In ASCE 41, the rotation capacities (acceptance criteria) depend on the following.

(1) The distribution of vertical reinforcement. If the area of tension reinforcement is large, a wall cross section can be like an over-reinforced beam, and the rotation ductility is reduced.
(2) The shear force. If the shear force is large, the rotation ductility is reduced. (In addition, if the hinge rotation is large, the shear strength is reduced, but this is a separate issue.)
(3) Whether the boundaries of the wall are confined. Walls with confined edges have larger rotation ductilities.

The purpose of the example structure is to show how shear walls can be modeled, not to get into the details of deformation (or strength) capacities. For the example structure the following rotation capacities are used, based roughly on the ASCE 41 capacities for the Collapse Prevention performance level. If you are choosing rotation capacities for an actual structure, you should study the ASCE 41 criteria in detail.

(1) Web walls (for load in the H2 direction), stories 1 through 5 (with confinement in the flanges) : 0.010 radians for both positive and negative bending.
(2) Web walls stories 6 through 12 (no confinement in the flanges) : 0.006 radians for both positive and negative bending.
(3) Flanges in the outer walls (for load in the H1 direction), stories 1 through 5 : 0.009 radians when the flange tips are in compression (PERFORM-3D positive bending, since there is compression on the positive Axis 3 side), and 0.015 for negative bending (when the flange tips are in tension).
(4) Flanges in the outer walls, stories 6 through 12 : 0.003 radians for positive bending (flange tips are in compression), and 0.015 for negative bending.
(5) Flanges in the inner wall (for load in the H1 direction), stories 1 through 5 : 0.009 radians for both positive and negative bending. The flange tips are in compression in both cases, and the reinforcement in the web is in tension.
(6) Flanges in the inner wall, stories 6 through 12 : 0.003 radians for both positive and negative bending.

2.14.3 Rotation Gage Elements

In the current version of PERFORM-3D you can not specify rotation capacities directly for wall elements (you can specify them for fiber sections in beams and columns). For calculating hinge rotations in walls it is necessary to use Rotation Gage components and elements.

In the **Elements** task, choose the "Rotation gages" element group. If you wish, use a Frame view to show the elements more clearly.

In the upper stories, there is one rotation gage element for each wall element. This is also the case for the flange walls in the bottom story. In the web walls in the bottom story the rotation gages extend over three elements in the outer walls, and two elements in the inner wall.

Rotation gages have no strength or stiffness. Their sole purpose is the calculation of rotation demands and D/C ratios. To show the gage properties, choose the **Properties** page, and select an element in the graphics panel (or double-click to select all elements with the same properties). Press the **Show Properties** button to show the gage properties.

2.15 Strain Gages

2.15.1 Purpose

In the upper stories, hinges should not form. This means that there should be no substantial yield of the vertical reinforcement. To check this, strain gages are used.

Twelve strain gages are placed in each story, four at the corners of the wall and eight at the tips of the flanges. The steel yield stress is 70 ksi, and hence the yield strain is 70/29000 = 0.0024. A strain of 1.5 times this value is allowed (i.e., the tension strain capacity is 0.0036). This capacity is not specified by ASCE 41. For this example it is assumed that there is significant hinging if this capacity is exceeded.

The strain gages also have compression strain capacities. This is set to 0.0021 for unconfined concrete and 0.0065 for confined concrete, which are the strains at which the concrete begins to lose strength.

2.15.2 Strain Gage Elements in Upper Stories

In the **Elements** task, choose the "Strain gages, outside hinge region" element group. If you wish, use a Frame view to show the elements more clearly.

Strain gages have no strength or stiffness. Their sole purpose is to allow calculation of strain demands and D/C ratios. To show the gage properties, choose the **Properties** page, and select an element in the graphics panel (or double-click to select all elements with the same properties). Press the **Show Properties** button to show the gage properties. The tension strain capacities are all 0.0036, corresponding to 1.5 times the yield strain. For the compression capacities see the following section.

2.15.3 Strain Gage Elements in Bottom Story

Strain gages are also used to calculate strains in the bottom story hinge regions.

The tension strains in these gages are considered for interest only, not for calculating D/C ratios. The tension strain capacity is set to 1%. Hence, the numerical value of the tension strain D/C ratio is the tension strain in percent.

The compression strains in these gages are considered to see whether compression crushing of the concrete occurs.

In the **Elements** task, choose the "Strain gages, hinge region, flanges" or "Strain gages, hinge region, webs" element group. If you wish, use a Frame view to show the elements more clearly. To show the gage properties, choose the **Properties** page, and select an element in the graphics panel (or double-click to select all elements with the same properties). Press the **Show Properties** button to show the gage properties. The strain capacities in tension are all 1%. The capacities in compression depend on whether the concrete is confined or unconfined.

2.15.4 Commentary on Calculated Strains

It is important to note that the calculated strains are average values over the gage lengths, which are equal to the assumed hinge lengths. The calculated strains are sensitive to the gage length, and are by no means "exact". The calculated amount of crushing also depends on the fiber areas that are used in the fiber sections. If reasonable hinge lengths and fiber areas are used, compression strain D/C ratios should give a reasonable indication of whether there is crushing in the extreme fibers (ideally there should be little or no crushing). However, if crushing occurs and spreads over the cross section, the calculated compression strains are likely to give only a rough indication of the extent of the crushing.

2.15.5 Strain Calculation Using Monitored Fibers

Strain D/C ratios can also be calculated using "monitored" fibers.

In the **Component Properties** task, choose the **Cross Sects** page and the "Shear Wall, Inelastic Section" type. Choose any of the sections in the list, and note the Monitored Fibers part of the form. This provides an alternative method for calculating strains and strain D/C ratios. This method is not used for this example.

2.16 Structure Sections and Shear Strengths

2.16.1 Strength at Element and Structure Section Levels

The wall is required to remain essentially elastic in shear (there should be no yield of the shear reinforcement). Hence, the shear performance is checked using strength D/C ratios, not deformation.

Shear strength can be checked at the element level, or at the structure section level. This is illustrated in Figure 2.4.

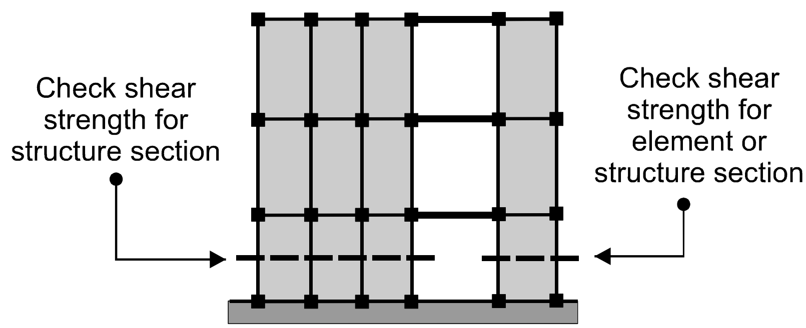


Figure 2.4. Shear at Section and Element Levels

In the left hand pier in Figure 2.4, there are several elements across the width of the wall. In this case, there may be shear stress concentrations, and larger shear stresses may be calculated for some elements than for others. For checking the shear strength, however, it is usual practice to consider the average shear stress in the section. If shear stress D/C ratios are calculated for each element, the maximum D/C ratio may be too conservative. In PERFORM-3D the average stress can be calculated by defining a Structure Section, and assigning a strength capacity to the section.

In the right hand pier in Figure 2.4, there is only one element across the width of the wall. In this case, the shear strength can be checked at the element level and/or using structure sections, where each section cuts only one element.

The example structure has three web walls. If there were no torsion on the structure, the shear strength in the H2 direction could be checked for the wall as a whole, using structure sections that include all three webs. However, because of accidental torsion the shear forces tend to be larger in the outer webs. Hence, each web wall should be checked separately. Since there is one element across the width of each wall, the shear strength can be checked at either the element or structure section level.

Shear in the H1 direction is more complicated. The strength could be checked for structure sections that cut through all three piers (i.e., all three flanges). However, it is probably better to check each pier separately. For the outer walls, each flange has one element across its width, so the strength can be checked at the element or structure section level. The center wall has two elements across the flange width, so it is better to use structure sections.

2.16.2 Advantages and Disadvantages

There are advantages and disadvantages to checking the shear strength at the element or structure section levels.

Checking at the structure section level has the following advantages and disadvantages.

(1) Appropriate structure sections must be specified. For a large structure there can be many sections. This is a disadvantage.

(2) If the structure sections are organized into section groups, shear force diagrams can be plotted over the height of the building, either as actual shear forces or as shear strength D/C ratios. This is a substantial advantage.

(3) When a structure section is specified, the area for calculating the shear stress must be input (for checking at the element level the area is obtained from the element width and the thickness specified for shear behavior). This is more work for a structure section, but it also gives more flexibility. For example, in the example structure the width of an element in a web wall is 30 feet, but the section depth is 31.5 feet. When a structure section is used, the area for shear stress calculation can be specified as 31.5 x 1.5 = 47.25 sq. ft. (a factor 0f 0.8 could also be applied, to account for effective depth, but as indicated earlier, this is done using by applying a capacity factor to the strength of the shear

material). The same adjustment can be made for checking at the element level, by specifying a thickness for shear behavior equal to 31.5/30 = 1.05 times the actual wall thickness, but this requires rather more planning.

Checking at the element level has the advantages that deflected shapes can be drawn with element colors that depend on the D/C ratio. This is useful, but each color covers a range of D/C ratios, whereas a plot of D/C ratios for a structure section group shows the value of the D/C ratio at each story.

For the example structure, shear strength is checked using structure sections only.

2.16.3 Effect of Axial Force on Shear Strength

PERFORM-3D allows the shear strength of a shear material to depend on the axial stress. This feature can be used for strength checking at both the element and structure section levels.

This feature is not used for the example structure. It might be important for designing the shear reinforcement, and for a practical structure you should consider including the axial force effect.

2.16.4 Structure Sections

Press the toolbar button for the **Structure Sections** task.

The **Define Sections** page shows the sections. For all except the last section, see Section 2.16.6, below for descriptions of the sections and their purpose.

The last section, "Base, for base shear, etc.", cuts through the wall elements and the P-Δ columns at the base of the structure. To see the cuts, choose an element group from the list. In the graphics panel the cut elements are shown red, and the cut locations are shown green.

2.16.5 Structure Section Shear Strengths

In the **Structure Sections** task, the **Strengths** page shows the sections for which shear strengths have been specified. A section strength is defined by associating it with a shear material, and specifying the section area.

2.16.6 Structure Section Groups

In the **Structure Sections** task, the **Groups** page shows section groups that have been set up for plotting shear force and shear D/C diagrams. The groups are as follows.

(1) West wall web, for shear strength. This group consists of sections through the web elements of the West wall. Each section cuts through two elements, one in the "Walls, basic steel" element group and one in the "Walls, concrete + extra steel" element group. There is one section at the bottom of each story, and an additional section just below the roof. For this last section, when the section was added to the group list, the "Change signs for M,V diagrams" box was checked. This is necessary to change the sign of the shear force on the section when the shear force diagram is plotted (otherwise the sign is incorrect, because forces on structure sections are forces exerted on the section by the cut elements).

(2) Center wall web, for shear strength. This group consists of sections through the web elements of the inner wall. Each section cuts through two elements, one in the "Walls, basic steel" element group and one in the "Walls, concrete + extra steel" element group.

(3) South-west flange, for shear strength. This group consists of sections through the south flange elements of the west wall. Each section cuts through two elements, one in the "Walls, basic steel" element group and one in the "Walls, concrete + extra steel" element group.

(4) South-center flange, for shear strength. This group consists of sections through the south flange elements of the inner wall. Each section cuts through four elements, two in the "Walls, basic steel" element group and two in the "Walls, concrete + extra steel" element group.

More sections and groups might be needed for a practical analysis, with sections through all webs and flanges, to make sure that the most critical locations are considered.

2.16.7 P-Δ Effects on Section Shear Forces

This is a subtle issue that is of interest only to theoreticians. It has little practical effect.

In the "Base, for base shear, etc." section, the P-Δ columns are included in the structure section. As a consequence, , when a static push-over analysis is run, the shear force on the section is equal to the applied horizontal load. If the P-Δ columns are not included in the structure section, the shear force on the section is not equal to the applied horizontal load (it is somewhat larger). This is illustrated in Figure 2.5.

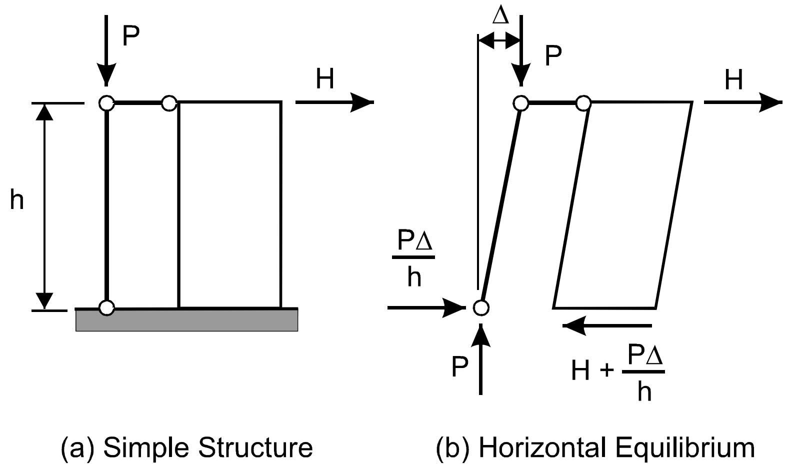


Figure 2.5. Shear Forces With P-Δ Effects

As shown in the figure, there is an effective shear force in the P-Δ column, equal to PΔ/h,. For equilibrium, the shear force on the structure section as a whole must equal the applied load, H. Hence, the effective shear force on the wall is H + PΔ/h. This is the force that must be used to calculate the shear stress on the wall section.

When a P-Δ column is used to account for P-Δ effects, the structure section shear forces calculated by PERFORM-3D are correct. If, however, the P-Δ effects are accounted for in the wall elements (i.e., in Figure 2.5 the P-Δ column is not present, and P-Δ effects are considered directly in the wall), the shear force on the structure section is only H. Hence, if P-Δ effects are considered in wall elements, and if the shear strength is checked using structure sections, the strength demand may be under-estimated.

In the example structure, only a small part of the total gravity load is applied to the walls, so even if P-Δ effects for these loads were considered in the walls rather than the P-Δ column, the change in shear on the structure sections would be small. Also, wall structures tend to be relatively stiff, and hence P-Δ effects tend to be relatively small. Hence, any error is likely to be negligible.

2.17 Limit States

2.17.1 Deformation

Press the toolbar button for the **Limit States** task. Choose the "Deformation" limit state type. There are 7 deformation limit states, as follows.

(1) *Bending rotation, from rotation gages*. This covers all Rotation Gage elements. The rotations of greatest interest are in the hinge regions in the bottom story.

(2) *Coupling beam shear deformation*. This covers the shear hinges in all coupling beam elements..

(3) *Tension strain, upper stories, 1.5 x yld*. This covers tension strain in all strain gages above the bottom story. This checks that there is no significant hinging outside the hinge regions.

(4) *Comprn strain, upper stories, crushing*. This covers compression strain in all strain gages above the bottom story. This checks that there is no significant concrete crushing outside the hinge regions.

(5) *Tension strain, hinge region, 1%*. This covers tension strain in all strain gages in the hinge regions. The D/C ratio for any gage is the tension strain in percent. This is mainly for interest, but also checks that the steel strains are not excessive.

(6) *Comprn strain, hinge, flanges, crushing*. This covers compression strain in the flange walls in the hinge regions. This checks that there is no significant concrete crushing in the flanges in the hinge regions.

(7) *Comprn strain, hinge, webs, crushing*. This covers compression strain in the web walls in the hinge regions. This checks that there is no significant concrete crushing in the flanges in the hinge regions.

2.17.2 Strength (at the Element Level)

In the **Limit States** task, choose the "Strength" limit state type. There are no limit states of this type. As noted earlier, shear strength could be checked at the element level, but it is generally better to do the check at the structure section level.

2.17.3 Drift

In the **Limit States** task, choose the "Drift" limit state type. There are two limit states of this type.

There are no limits on the drifts of the structure. However, it is convenient to check the drifts using limit states. Two drifts have been specified, namely H1 and H2 drifts at the roof. Each of these is used in a drift limit state, with 1% drift as a nominal drift capacity. The usage ratio for the limit state is the drift in percent.

It may be noted that it is usually a good idea to define several drifts, and to use all of them as "controlled" drifts for static push-over analyses. This is particularly important where there is brittle strength loss, and the roof drifts may not be well behaved. In this structure it is sufficient to control only the roof drifts.

2.17.4 Shear Strength at the Structure Section Level

Choose the **Struct Sectn** limit state type. There are 4 limit states of this type.

Each limit state corresponds to one of the Structure Section Groups. For example, the "West web wall, shear strength" limit state considers all of the structure sections in the "West wall web, for shear strength" group. The usage ratio for the limit state is the largest shear strength D/C ratio for all sections in the group.

2.18 Gravity Load Cases

2.18.1 Load Patterns

The gravity load patterns have been considered earlier. To review them, press the toolbar button for the **Load Patterns** task.

Choose the **Nodal Loads** page. There are three nodal load patterns, as follows.

(1) *Floor loads on walls, 1 ksf*. Floor loads acting on the walls, for 1 ksf load on the floor. This does not include the wall self weight.

(2) *Unit P-delta loads, center*. Unit (1 kip) loads on the P-Δ column at the center of the structure.

(2) *Unit P-delta loads, offset*. Unit (1 kip) loads on the P-Δ column that accounts for accidental mass eccentricity. When mass eccentricity is considered, it is consistent to include it in the P-Δ effects as well as the mass. This load pattern can be used to account for this, if desired. The effect is small, however.

These loads are scaled when they are used in load cases.

Choose the **Self Weight** page. There is one self weight load pattern, for the weight of the wall elements.

2.18.2 Load Cases

Press the toolbar buttons for the **Analysis** phase and the **Set up Load Cases** task. Choose the "Gravity" load case type.

For analyses of the example structure there is only one load case of this type, namely "DL + 0.25LL". This case uses three load patterns. The patterns are scaled to provide the required loads.

When gravity loads are applied to structures with fiber sections, it is possible for concrete cracking to occur (steel yield or concrete crushing should not occur). Concrete cracking is a nonlinear event, so it is often necessary to specify that the gravity load analysis is nonlinear. In this structure the behavior is linear, so linear analysis can be used.

If nonlinear analysis is needed, it is not necessary to specify a large number of load steps. One step is usually sufficient, and five steps is more than enough. The extra computer time required for nonlinear gravity analysis is usually small.

2.19 Dynamic Earthquake Load Cases

Usually, earthquake ground motions are chosen in pairs, one motion in the H1 direction and one in the H2 direction, and the motions are applied simultaneously (biaxial shaking). It is common to choose either three or seven pairs of related motions, usually based on actual earthquakes.

For this example there are six ground motions, all matched to the same response spectrum. The main purpose of the example is to illustrate modeling procedures, not to assess the performance of an actual structure. To keep the total length of the solution files reasonable, analyses are carried out for only three of the ground motions. Also, to allow simple comparison with static push-over methods, only single earthquake motions are applied, not pairs of motions (this is also the reason for using spectrum-matched ground motions).

The dynamic earthquake load cases that have been used are as follows.

(1) Motion 1 in the H1 direction.
(2) Motion 3 in the H1 direction.
(3) Motion 5 in the H1 direction.
(4) Motion 1 in the H2 direction.
(5) Motion 3 in the H2 direction.
(6) Motion 5 in the H2 direction.
(7) Motion 1 in the H1 di2ection, Sa = 2g.
(8) Motion 1 in the H2 direction, Sa = 2g.

For the raw earthquake records the peak Sa value in the response spectrum is 0.425g. For the first six load cases, Sa = 1.5g and the scale factor for is 3.53. For the cases with Sa = 2g, the scale factor is 4.71. The performance is assessed for the Sa = 1.5g motions. The Sa = 2g motions are used to see what happens when a stronger earthquake is applied.

In each case, 15 seconds of the earthquake is considered, using a time step of 0.02 seconds. Preliminary analyses showed that the maximum effects for all earthquakes occur in the first 15 seconds, and that the results with a time step of 0.02 seconds are similar to those with shorter time steps.

In the **Set up Load Cases** task, choose the "Dynamic Earthquake" load case type. In the list of Load Case Names, choose one of the cases to show the time step, the applied earthquake record, etc.

To see the ground accelerations for an earthquake, press the **Add/Review/Delete Earthquakes** button. In the "Use this section to review or delete an earthquake record" section, choose the "Wall Example" earthquake group, then choose one of the six files in the File Name list, and press the **Review** button. To exit, press the **Close** and **Return to Earthquake Load Case** buttons.

2.20 Push-Over Load Patterns and Load Cases

The main method of analysis for this example is dynamic earthquake analysis. However, it is interesting to run push-over analyses and to compare the results.

In PERFORM-3D, push-over loads can be based on horizontal nodal load patterns, on the inertia forces corresponding to the elastic mode shapes of the structure, or on inertia forces corresponding to user-specified displacement distributions over the building height. The last two of these methods have been used for this example.

In the **Set up Load Cases** task, choose the "Static Push-Over" load case type. There are 4 load cases of this type, with loads as follows.

(1) *H1 push, triangular*. A load distribution based on the masses and a linear displacement (or acceleration) variation over the building height, with loads in the H1 direction only.
(2) *H1 push, uniform*. A load distribution based on a constant displacement variation over the building height.
(3) *H2 push, triangular*.
(4) *H2 push, uniform*.

Choose one or more of these cases to see the load case details.

2.21 Analysis Parameters, Viscous Damping

In the **Analysis** phase, press the toolbar button for the **Run Analyses** task. This shows the ANALYSIS SERIES form.

For the example structure, three analysis series have been set up. A full set of analyses has been run in one of these series, to assess the performance of the structure and to compare dynamic and push-over analyses. A smaller number of analyses has been carried out in each of the other two series, to illustrate the effects of changing the series properties. If you wish, you can run additional analyses, in any of the analysis series.

The analysis series are as follows. Choose the series in the Series Name list to show the properties.

(1) Series 1. Mass at center of structure, 5% Rayleigh damping, include P-Δ effects.
(2) Series 2. Mass at center of structure, 5% "modal" damping, include P-Δ effects.
(3) Series 3. Eccentric mass, 5% Rayleigh damping, include P-Δ effects.

2.22 Commentary on "Accuracy"

One thing that is clear from this chapter is that many assumptions must be made to create a analysis model. In no way can the model be regarded as "accurate". However, the goal is not accuracy in the sense of precise calculation of how the actual building will behave in an earthquake. Rather, the goal is to obtain D/C ratios that are sufficiently accurate for making design decisions.

For strength- based design, the reason for using linear structural analysis is not that it is accurate, but that it gives strength D/C ratios that are sufficiently accurate for design. The same standard applies when nonlinear analysis is used for deformation based design. In both cases there are substantial uncertainties, which must be accounted for by factors of safety applied to the loads and/or the component capacities, and also through the use of capacity design and sound detailing practices.

It may appear that the uncertainties in creating a nonlinear model are greater than in creating a linear model. This is not correct. Unless a structure is designed to remain elastic in a strong earthquake, nonlinear behavior is present whether or not it is considered in the analysis. A linear model ignores nonlinear behavior. Hence, a linear model is of highly uncertain accuracy, and may be substantially inaccurate. A nonlinear analysis model is inherently better because it explicitly considers nonlinear behavior. Nonlinear modeling is difficult because nonlinear behavior is complex, and the designer must manage this complexity. Ignoring it does not make it go away.

3 Performance of Original Structure

3.1 Overview

The preceding chapter considered the analysis model and limit states. This chapter considers the performance assessment.

Only the dynamic earthquake analyses are used in this chapter. The results from dynamic earthquake and push-over are compared in a later chapter.

3.2 Load Sequence

Gravity load is applied first. The limit state usage ratios are all small for the gravity case. Dynamic earthquake and push-over loads are then added.

In the **Analysis** phase, press the toolbar button for the **Run Analyses** task, then choose the "Series 1" analysis series. Press the **OK** button to show the list of completed analyses in this series.

3.3 Dynamic Earthquake Analyses

3.3.1 Usage Ratio Envelopes

Press the toolbar buttons for the **Analysis** phase and the **Combinations and Envelopes** task. This task provides a summary of the performance assessment.

There is a total of 13 limit states. This is a small enough number that it is not necessary to set up limit state groups.

In the list of combinations choose "H1 earthquakes" combination and the Max-Max combination method. Click on the short red lines along the left side of the plot to show each limit state and the corresponding usage ratios for the three earthquakes.

The maximum values of all usage ratios are smaller than 1.0 except for the "South center flange, shear strength" limit state, for which the maximum usage ratio is 1.14. The corresponding earthquake is Motion 2.

Hence, the calculated shear stress demand in the flange exceeds $10\sqrt{f_c^{'}}$, and a design change is needed to decrease the shear stress demand.

In the list of combinations choose "H2 earthquakes" combination and the Max-Max combination method. Again, click on the short red lines along the left side of the plot to show each limit state and the corresponding usage ratios for the three earthquakes. The maximum usage ratios are larger than 1.0 for two limit states, , as follows.

(1) "Center wall web, shear strength" limit state, with a usage ratio of 1.05. The calculated shear stress demand in the web exceeds $10\sqrt{f_c^{'}}$, and a design change is needed to decrease the shear stress demand. The corresponding earthquake is Motion 1.
(2) "Tension strain, upper stories, 1.5 x yld" limit state, with a usage ratio of 2.36. This corresponds to a steel strain of 0.85%. This indicates that substantial hinging is occurring in one or more of the upper stories, and a design change is needed. The corresponding earthquake is Motion 1.

Other limit states of interest are as follows.

(1) Roof drift. The maximum drift is 0.53% for the H1 earthquakes, and 0.71% for the H2 earthquakes.
(2) Coupling beam shear deformations. The maximum usage ratio is 0.55, for the H1 earthquake.
(3) Tension strain in the hinge region. The maximum usage ratio is 0.67, corresponding to a maximum strain of 0.67%. This is smaller than the 0.85% maximum tension strain in the upper stories.

Note that there is substantial variation in the results from earthquake to earthquake, even though they all have similar elastic response spectra. This is not unexpected.

3.3.2 Shear Strength for H1 Earthquake

For earthquake Motion 3 in the H1 direction, the shear strength usage ratio for the in the south-center flange is 1.14.

To show the shear strength D/C ratios for all stories in this flange, do the following.

(1) Press the toolbar button for the **Moment and Shear Diagrams** task.
(2) Choose the **Section Group** page.
(3) In the list of structure section groups choose "South-center flange for shear strength".
(4) For the result type choose "Strength D/C ratio, Level 1".
(5) For the Plot Type choose "Envelopes only".
(6) In the Series list in the toolbar choose "Series 1", and in the name list choose "[1] + Motion 3, H1 direction".
(7) In the data panel, press the **Plot** button.

The plot shows the shear strength D/C ratios for all stories. In the plot, each vertical grid line corresponds to a structure section. Hence, the first grid line is for the section at the base, the second is for the section in the second story, etc.

The D/C ratio is 1.14 in the bottom story, and exceeds 1.0 in the first 3 stories. A D/C ratio of 1.0 corresponds to a shear stress demand of 950 psi. For this design, the wall must be reinforced in shear to provide sufficient strength capacity in each story. This might be difficult to do in the hinge region.

3.3.3 Shear Strength for H2 Earthquake

For earthquake Motion 1 in the H2 direction, the shear strength usage ratio for the in the web of the center wall is 1.05.

To plot the shear strength D/C ratios for all stories, follow a procedure similar to that in the preceding section. The ratio is 1.05 in the second story, and exceeds 1.0 in the first two stories. Again, the wall must be reinforced in shear to provide sufficient strength, and this might be difficult to do in the hinge region.

3.3.4 Hinging in Upper Stories

For Motion 1 in the H2 direction, the maximum tension strain in the upper stories is 2.36 times larger than the limit of 1.5 times the yield strain.

To see the distribution of tension strains, do the following.

(1) Press the toolbar button for the **Deflected Shapes** task.
(2) Choose the **Limit States** page.
(3) For the Limit States to be Considered choose "All deformations".
(4) For the Color Group for Usage ratios choose "2".
(5) In the Series list in the toolbar choose "Series 1", and in the name list choose "[1] + Motion 1, H2 direction".
(6) In the data panel, click in the right hand end of the "thumbnail" plot to move the yellow line to the end of the analysis.
(7) Press the **Plot** button.

The plot shows that the strain exceeds 1.5 time yield only in the sixth story. If you view the deflected shape in the H1 direction, and move the yellow line to a peak in the "thumbnail" plot (or if you animate the deflected shape), you will see that there is substantial "hinge" rotation in the sixth story.

Alternatively, use the **Combinations and Envelopes** task and the **Element Colors** page to show the element colors. This does not show the deflected shape, but is quicker.

To obtain the desired behavior, the vertical reinforcement would need to be increased in one or more of the upper stories.

3.3.5 Coupling Beam Deformations

For Motion 1 in the H1 direction, the maximum usage ratio for coupling beam shear deformation is 0.55.

To see the distribution of deformations, do the following.

(1) Press the toolbar button for the **Deflected Shapes** task.
(2) Choose the **Limit States** page.
(3) For the Limit States to be Considered choose "All deformations".
(4) For the Color Group for Usage ratios choose "2".

(5) In the Series list in the toolbar choose "Series 1", and in the name list choose "[1] + Motion 1, H1 direction".
(6) In the data panel, click in the right hand end of the "thumbnail" plot to move the yellow line to the end of the analysis.
(7) Press the **Plot** button.

3.3.6 Other Results

If you wish, use other PERFORM-3D tools to examine the results.

In particular, it is interesting is to use the **Moment and Shear Diagrams** task and the **Section Groups** page, and to choose the "Time history" plot type to plot shear force and shear strength D/C ratio diagrams. When these diagrams are animated, it is clear that higher mode effects have a substantial influence on the response. The higher mode effects show up as waves that propagate over the structure height.

4 Revised Structure

4.1 Revisions

The preceding chapter indicates that the vertical reinforcement in the original structure must be revised to avoid hinging in the upper stories, and that it mat be difficult to provide sufficient shear strength. The revised structure seeks to avoid these problems.

It is known that improved seismic performance can be obtained when displacement-based (rather than strength-based) design is used, and when capacity design principles are applied. In particular, the strength of the structure can be reduced without compromising performance. Among other things, reducing the bending strength for wall structures has the advantage that it reduces the shear strength demands, yet does not necessarily increase the structure drifts.

For the revised structure the vertical reinforcement is reduced substantially, as shown in Table 4.1.

TABLE 4.1
Revised Wall Reinforcement

Story	Basic (unconfined) reinforcement		Additional (confined) reinforcement in flanges	
	Webs	Flanges	Area	Width (in)
1	0.4%	0.4%	0.25%	72
2				
3				
4	0.33%	0.33%	0.25%	60
5				
6			0.25%	48
7				
8			0	0
9	0.25%	0.25%		
10				
11				
12				

A key point is that the maximum additional reinforcement is extended over three stories, rather than two. This is needed to satisfy capacity design principles (see Paulay and Priestley, "Seismic Design of Reinforced Concrete and Masonry Buildings", Wiley, 1992).

In addition, the coupling beam strengths are reduced as shown in Table 4.2.

TABLE 4.2
Revised Coupling Beam Shear Strengths

Stories	Shear Strength (k)
1	1500
2-7	500
8-9	385
10-12	285

The loads, component capacities, limit states, etc. are the same as the original structure.

For the properties of the revised structure see **ExampleCSI2B**, and follow similar steps to those in Chapter 2.

4.2 Performance for Dynamic Earthquake Analyses

The analyses are the same as for the original structure. For a summary of the performance, press the toolbar buttons for the **Analysis** phase and the **Combinations and Envelopes** task. To see the usage ratios, follow similar steps as in Chapter 3.

Tables 4.3 and 4.4 show the main results. The following are some key points.

(1) The tension strains in the upper stories are all smaller than 1.5 times the steel yield. The maximum strain is 0.31%, compared with 0.85% for the original structure.

(2) The maximum tension strain in the hinge region for the H2 earthquakes is 1.09%. This is 4.6 times the yield strain (i.e., a ductility ratio of about 4.6). If you plot deflected shapes, you

will see that it is dominated by hinging at the base, which is the desired behavior.

(3) The maximum roof drift is 0.54% in the H1 direction and 0.68% in the H2 direction. This compares with 0.53% and 0.72%, respectively, for the original structure.

(4) The shear stress demands are smaller than those in the original structure. This is to be expected because the wall is weaker in bending. For H1 earthquakes, the maximum usage ratio for the south-center flange is 0.95, compared with 1.13 for the original structure. For H2 earthquakes, the maximum usage ratio for the web of the center wall is 0.86, compared with 1.05 for the original structure.

(5) The maximum usage ratio for shear deformation in the coupling beams is 0.68. This compares with 0.55 for the original structure.

In summary, although the revised structure has substantially less reinforcement than the original structure, it has superior performance. In particular, it should be easier to design the shear reinforcement to satisfy the requirement of essentially elastic behavior in shear.

TABLE 4.3
Comparison of Original and Revised Structures
Dynamic Earthquake Loads, H1 Direction

Result or Limit State	Original	Revised
Roof drift.	0.53%	0.54%
South-center flange, shear strength. Limit state usage ratio.	1.13	0.95
South-west flange, shear strength. Limit state usage ratio.	0.95	0.82
Coupling Beam Deformation. Limit state usage ratio.	0.55	0.68
Steel tension strain, upper stories.	0.28%	0.28%
Tension strain, hinge region	0.50%	0.52%

TABLE 4.4
Comparison of Original and Revised Structures
Dynamic Earthquake Loads, H2 Direction

Result or Limit State	Original	Revised
Roof drift.	0.72%	0.68%
Center wall web, shear strength. Limit state usage ratio.	1.05	0.86
West wall web, shear strength. Limit state usage ratio.	0.92	0.77
Steel tension strain, upper stories.	0.85%	0.31%
Tension strain, hinge region.	0.66%	1.09%

5 Push-Over Analysis

5.1 Purpose

Push-over analysis can be useful for checking the analysis model and for examining the behavior of the structure. It can also be used to assess performance. When used in this way, push-over analysis is approximate, because it accounts for dynamic effects using static analysis. The approximation can be of particular concern when there are significant higher mode effects.

5.2 Load Distributions

For performance assessment using push-over analysis, ASCE 41 allows load patterns based on "triangular" and "uniform" load patterns. The usage ratio for performance assessment is the larger of the ratios from the two analyses.

5.3 Push-Over Curves and Base Shear Strengths

The revised structure has substantially less reinforcement than the original structure. This is reflected in the base shear strengths from push-over analysis. Table 5.1 compares the strengths.

TABLE 5.1
Base Shear Strengths From Push-Over Analyses
(proportion of gravity load, at 0.75% roof drift)

Direction	Load Pattern	Original Structure	Revised Structure
H1	Triangular	0.40	0.30
H1	Uniform	0.52	0.40
H2	Triangular	0.30	0.23
H2	Uniform	0.39	0.30

5.4 Target Displacements

Push-over analyses using the triangular and uniform patterns have been used to calculate target displacements for the original and revised structures, and hence to calculate usage ratios at these displacements. The ASCE 41 Coefficient Method (essentially the same as the FEMA 440 Coefficient Method) has been used, with the following parameters.

(1) A smoothed version of the 5% response spectrum (see Figure 1.4), with maximum spectral acceleration = 1.5g. This corresponds to the ground motions used for the dynamic earthquake analyses.
(2) Site Class B for coefficient C1.
(3) FEMA 440 Framing Type 1 (degrading) for Coefficient C2. This is required in ASCE 41.
(4) For calculating the base shear demand, assume that the deflected shape is the mode shape PERFORM-3D provides other options, but ASCE 41 assumes the mode shape.
(5) Calculate the effective period using the Rayleigh quotient. This is done because it is the most reliable method.

The calculated target displacements are shown in Table 5.2. If you wish you can use the **General Push-Over Plot** task to check these values.

TABLE 5.2
Target Displacements (Roof Drifts) from Push-Over Analyses

Direction	Load Pattern	Original Structure	Revised Structure
H1	Triangular	0.39%	0.40%
H1	Uniform	0.29%	0.29%
H2	Triangular	0.47%	0.44%
H2	Uniform	0.32%	0.33%

The target displacements are similar for the original and revised structures, even though the revised structure is substantially less strong. The sensitivity feature in the PERFORM-3D push-over analysis indicates that if the strength is changed and the stiffness is kept the same, the target drift changes only a small amount.

5.5 Usage Ratios

Tables 5.3 through 5.6 show some key results. In each table, the values for dynamic earthquake analysis are the average values from the three earthquakes, not the maximum. The reason is that push-over analysis methods are calibrated to match the average response over a large number of earthquakes, not the maximum. The values from the push-over analyses are the maxima for the triangular and uniform load distributions.

TABLE 5.3
Comparison of Dynamic and Push-Over Analyses
Original Structure, H1 Direction

Result or Limit State	Dynamic	Push-Over
Roof drift.	0.45%	0.39%
South-center flange, shear strength. Limit state usage ratio.	1.01	0.90
South-west flange, shear strength. Limit state usage ratio.	0.79	0.50
Coupling Beam Deformation. Limit state usage ratio.	0.45	0.30
Steel tension strain, upper stories.	0.23%	0.19%
Tension strain, hinge region	0.43%	0.31%

TABLE 5.4
Comparison of Dynamic and Push-Over Analyses
Original Structure, H2 Direction

Result or Limit State	Dynamic	Push-Over
Roof drift.	0.64%	0.47%
Center wall web, shear strength. Limit state usage ratio.	0.96	0.63
West wall web, shear strength. Limit state usage ratio.	0.83	0.56
Steel tension strain, upper stories.	0.75%	0.27%
Tension strain, hinge region.	0.52%	0.26%

TABLE 5.5
Comparison of Dynamic and Push-Over Analyses
Revised Structure, H1 Direction

Result or Limit State	Dynamic	Push-Over
Roof drift.	0.47%	0.40%
South-center flange, shear strength. Limit state usage ratio.	0.89	0.75
South-west flange, shear strength. Limit state usage ratio.	0.73	0.36
Coupling Beam Deformation. Limit state usage ratio.	0.62	0.43
Steel tension strain, upper stories.	0.24%	0.20%
Tension strain, hinge region	0.47%	0.35%

TABLE 5.6
Comparison of Dynamic and Push-Over Analyses
Revised Structure, H2 Direction

Result or Limit State	Dynamic	Push-Over
Roof drift.	0.63%	0.44%
Center wall web, shear strength. Limit state usage ratio.	0.84	0.60
West wall web, shear strength. Limit state usage ratio.	0.75	0.52
Steel tension strain, upper stories.	0.31%	0.20%
Tension strain, hinge region.	0.80%	0.36%

5.6 Comparison With Dynamic Analysis

The Tables show that the push-over analyses consistently underestimate the deformations and D/C ratios, often by substantial amounts.

For example, for the original structure Table 5.4 shows that push-over analysis seriously underestimates the tension strain in the upper stories. Hence, push-over analysis indicates that there is little or no hinging in the upper stories, whereas dynamic analysis indicates substantial hinging.

The reason appears to be that push-over analysis fails to capture the important effects of higher mode vibrations on the response (the effects of waves traveling up and down the structure).

This is by no means a conclusive study, but it suggests that push-over analysis should not be used for performance evaluation of structures of this type.

In this Chapter, only triangular and uniform load patterns have been used for the push-over analyses. Other patterns can be used, including those based on mode shapes. Also, "Modal Pushover Analysis" can be used, where push-over analyses are carried out for load patterns based on two or more mode shapes, and the results are combined, usually by the square-root-of-sum-of-squares method. The results are not shown, but the author has run push-over analyses of several types for the example structure. The analyses give similar results to those shown, and all substantially underestimate the response compared with the dynamic analyses.

6 Additional Analyses

6.1 Modal Damping

6.1.1 Purpose

The dynamic analyses in earlier chapters assume Rayleigh damping. PERFORM-3D also provides a "modal" damping option. Some analyses of shear walls have suggested that modal damping may not work well for shear wall structures.

For the revised structure, two analysis series have been set up with 5% modal damping (plus a small amount of Rayleigh damping). The first of these series uses only 9 mode shapes (from the total of 36 modes). This is "Series 2-9M". The second series considers all 36 mode shapes. This is "Series 2-36M". Six dynamic analyses have been run in each series, for Motions 1, 3 and 5 in each of the H1 and H2 directions.

This section compares the Rayleigh and modal cases, and considers the effect of using more modes for the modal case.

6.1.2 Results – Energy Balance

Open the revised structure (**ExampleCSI2B**). Press the toolbar buttons for the **Analysis** phase and the **Energy Balance** task.

In the Series list in the toolbar, choose "Series 1", and in the Case list choose "[1] + Motion 1, H1 direction". Press the **Plot** button to plot the energy balance. The total energy is 14050 kip-ft, and the dissipated inelastic energy is 36% of this total.

If you wish, go to the Element Groups page, and you will see that the coupling beams account for virtually all of the inelastic energy. You may note that the dissipated inelastic energy does not increase monotonically, which is theoretically incorrect. This occurs because there is stiffness degradation in the coupling beams, and PERFORM-3D calculates the inelastic energy correctly only when unloading occurs. See the PERFORM-3D User Guide for a more detailed explanation.

Series 1 uses Rayleigh damping. To compare with modal damping, choose "Series 2-9M", and "[1] + Motion 1, H1 direction", and press the **Plot** button. The total energy is 13421 kip-ft, and the dissipated inelastic energy is 37% of this total. Similarly, for "Series 2-36M" the total energy is 13510 kip-ft, and the dissipated inelastic energy is 33% of this total. Hence the energies for Rayleigh and modal damping are similar.

Repeat the process for "Series 1", "Series 2-9M" and "Series 2-36M", for the load case "[1] + Motion 1, H2 direction". Again, the total and inelastic energies are similar. For the H2 direction the inelastic energy is a much smaller percentage of the total than for the H1 direction, and is accounted for by yielding of the reinforcing steel.

Based on this comparison, the three damping cases give similar amounts and distributions of energy dissipation.

6.1.3 Results – Usage Ratios

In the **Analysis** phase, press the toolbar button for the **Combinations and Envelopes** task. This is not a normal use of this task, but it provides a convenient way to compare the results from different analyses.

In the list of combinations, choose the "Compare Damping H1" combination. This combination consists of the three analyses in the H1 direction for the Rayleigh case, three for the 36-mode case, and three for the 9-mode case. Press **Plot**, then use the plot to compare the results.

There are differences in the results, but they are not large. There is no consistent pattern in the differences.

Repeat the comparison for the "Compare Damping H2" combination. Again there are nine analyses, three for the Rayleigh case, three for the 36-mode case and three for the 9-mode case. There are now major differences in the results. The results for the Rayleigh and 36-mode cases are similar, but the 9-mode case gives much larger shear forces and much larger strain in the upper stories. If modal damping with 9 modes had been used for the performance evaluation, the result of the evaluation would have been very different.

It is not clear why the results for the 9-mode case are so different. It appears, however that the results for the 9-mode case are substantially in error. The results for the Rayleigh and 36-mode cases are more reasonable.

To see why, in the **Analysis** phase, press the toolbar button for the **Moment and Shear Diagrams** task and choose the **Section Group** page. In the Series list in the toolbar, choose "Series 1", and in the Case list choose "[1] + Motion 1, H2 direction". In the data panel choose the "West wall web, for shear strength" section group and the "Strength D/C ratio, Level 1" result type. Press the **Plot** button to plot, in effect, the shear force diagram.

The diagram looks reasonable. The D/C ratio (based on a shear strength of $10\sqrt{f_c^{'}}$) is 0.73 at the base and 0.15 at the roof. These are the results for the Rayleigh case.

Repeat for "Series 2-36M" and "[1] + Motion 1, H2 direction". In the data panel, choose the "West wall web, for shear strength" section group and the "Strength D/C ratio, Level 1" result type. Press the **Plot** button to plot the diagram.

Again the diagram looks reasonable. The D/C ratio is 0.82 at the base and 0.18 at the roof. These are the results for the case with 36-mode damping.

Finally, repeat for "Series 2-9M" and "[1] + Motion 1, H2 direction". In the data panel choose the "West wall web, for shear strength" section group and the "Strength D/C ratio, Level 1" result type. Press the **Plot** button to plot the diagram.

This diagram does not look reasonable. There is a large change in D/C ratio from the first story to the second, and the D/C ratio near the roof is about 0.5, which seems to be much too large.

To see why the shear force at the roof is so large, use the **Time History** task to plot absolute accelerations in the H2 direction at the roof for the above three analyses. For the Rayleigh and 36-mode cases the peak accelerations are respectively 1.03g and 1.28g, and the acceleration varies fairly smoothly. This seems reasonable. For the 9-mode case the peak acceleration exceeds 3g, and the variation is ragged, with high frequency oscillations. This does not seem reasonable.

Based on this limited study, it appears that for modal damping it is essential to use a large number of modes. Modal damping is attractive because it is conceptually similar to modal damping in linear analysis, and because it is easy to specify "5% modal damping". For Rayleigh damping it is more difficult to choose the damping parameters. However, care must be taken when using modal damping. For the analysis of practical structures it is probably wise to run preliminary analyses with both Rayleigh and modal damping, to check that they give similar results. It may also be wise to study the effect of changing the number of modes. It may be noted that it requires little or no additional computer time to consider more modes (although for a large structure the M000.txt file, which contains all of the mode shapes, may get large).

6.1.4 Note on Rayleigh Damping for Concrete Fibers

There is an aspect of Rayleigh damping that may not be well understood, and that can lead to excessive damping. This section considers that aspect, and explains the method used in PERFORM-3D to avoid the problem.

In Rayleigh damping, part of the damping matrix is proportional to the stiffness matrix (the rest is proportional to the mass matrix). In PERFORM-3D, the initial elastic stiffness matrix is used. This is often referred to as "βK0 damping". It is also possible to use the current tangent stiffness, which is often referred to as "βKT damping". PERFORM-3D uses only βK0 damping.

One problem that can arise concerns gap elements that are initially closed, and can open and close during a dynamic analysis. If βK0 damping is used for such elements, the initial stiffness, K0, is large, so βK0 is also large, and when the gap opens there can be a lot of viscous energy dissipation. Usually this should not happen in an actual structure. Hence, is usually wise to ignore βK0 damping in these elements. In PERFORM-3D this can be done by specifying a zero value for the "Scale Factor for Beta-K Damping" when the element group is created.

It may be noted that βKT damping also has a problem with gap elements. When a gap is open, KT is zero, so the element is undamped. However, when the gap closes, KT suddenly increases to a large value. Hence, the damping force can suddenly increase to a large value, sending a numerical "shock" through the analysis.

In a fiber section, concrete fibers are gap-type components that are initially stiff. Hence, if βK0 damping is used for fiber sections, there can be excessive viscous damping when the concrete cracks. To avoid this problem, when PERFORM-3D calculates the βK0 damping matrix for elements with fiber section, it uses only 15% of the fiber modulus for concrete fibers (i.e., in effect, PERFORM-3D automatically applies a βK0 scale factor of 0.15 to concrete fibers). This seems to be a reasonable number. No reduction is applied for steel fibers. If you wish, you can also specify a "Scale Factor for Beta-K Damping" for the element group.

6.2 Accidental Torsion

6.2.1 Purpose

The analyses in earlier chapters have all ignored accidental torsion. For the revised design, one of the analysis series (Series 3) uses eccentric masses. Otherwise the analysis series properties are the same as for Series 1. Eccentric P-Δ loads are not applied.

6.2.2 Results

Open the revised structure (**ExampleCSI2B**). Press the toolbar buttons for the **Analysis** phase and the **Combinations and Envelopes** task. As before, this is not a normal use of this task, but it provides a convenient way to compare the results from different analyses.

In the list of combinations, choose the "Compare Accidental Torsion" combination. This combination consists of the Motion 1 analyses, in the H2 direction, for the cases without and with accidental torsion. Press **Plot**, then use the plot to compare the results.

There are substantial differences in the results, with larger D/C ratios for the case with torsion. You must decide whether to include this effect in analyses of actual structures.

6.3 Stronger Earthquake

6.3.1 Purpose

As the analyses in this report show, there can be a great deal of variation in the analysis results. This is a powerful reason for applying Capacity Design principles. A major advantage of capacity design is that it can lead to more "forgiving" behavior, where changes or uncertainties in the structure properties and the ground motion (not to mention the modeling assumptions) do not lead to large changes in the behavior.

It may be noted that nonlinear analysis is not essential for capacity design, although the rules that have been established for capacity design using linear analysis were developed with the help of nonlinear analysis. It can be an advantage, however, to use nonlinear analysis, rather than to apply the "one size fits all" rules that apply when linear analysis is used.

To asses the "forgiveness" of the original and revised structures, they has been analyzed for an earthquake with larger ground accelerations than for the motions used in the other analyses. Analyses have been carried out only for the H2 direction ,for an earthquake that is 1.33 times as strong as the Motion 1 earthquake.

6.3.2 Results –Original Structure

Open the **ExampleCSI2A** structure. Press the toolbar buttons for the **Analysis** phase and the **Combinations and Envelopes** task. Again, this is not a normal use of this task, but it provides a convenient way to compare the analysis results.

In the list of combinations, choose the "Compare stronger earthquake" combination. Press **Plot**, then use the plot to compare the results.

The tension strain in the upper stories increases to 1.17%, about 1.4 times larger than the previous value of 0.85%. The tension strain in the hinge region increases by a factor of about 1.2, to 0.59%. This indicates that the weak point of the structure for bending is in the upper stories, not at the hinge. The shear strength demands increase by a factor of about 1.2.

6.3.3 Results –Revised Structure

Open the **ExampleCSI2B** structure. Press the toolbar buttons for the **Analysis** phase and the **Combinations and Envelopes** task.

In the list of combinations, choose the "Compare stronger earthquake" combination. Press **Plot**, then use the plot to compare the results.

With the stronger earthquake there is substantially more hinge rotation, with a calculated tension strain of 1.14% in the hinge region, about 1.5 times larger than the previous value of 0.75%, and equal to about 5 times the yield strain. The maximum tension strain in the upper stories also increases, to 0.53%.

Of greater concern is that the shear force demands increase substantially. For the web of the center wall, the shear strength D/C ratio increases to 1.10, about 1.33 times larger than the previous value of 0.83.

6.4 Elastic Upper Stories

6.4.1 Purpose

For the example structure, wall elements with fiber sections have been used in all stories. For a very tall building, computer time can be saved if some of the stories are modeled using wall elements with elastic sections. In the extreme case, only the bottom story, where hinging occurs, might be modeled using fiber sections, and all other stories might be modeled using elastic sections.

The structure **ExampleCSI2C** explores this type of modeling. In this structure, the bottom two stories are modeled using the fiber sections for the Revised Structure. The remaining stories are modeled using elements with elastic sections. The elastic modulus for these sections is assumed to be 0.5 times the nominal modulus for the concrete material (i.e., 2700 ksi).

Four push-over analyses have been run (triangular and uniform load patterns in the H1 and h2 directions) and six dynamic earthquake analyses (Motions 1, 3 and 5, in the H1 and H2 directions).

The purpose of the analyses is to assess whether it is reasonable to assume elastic sections.

6.4.2 Results – Push-Over Strength

Table 6.1 shows the push-over strengths for the two models. The strengths are virtually the same, indicating that the push-over strength is governed by the hinge region and/or the coupling beams.

TABLE 6.1
Base Shear Strengths From Push-Over Analyses for Structures with Fiber Sections and Elastic Section in Upper Stories
(proportion of gravity load, at 0.75% roof drift)

Direction	Load Pattern	Fiber Sections	Elastic Sections
H1	Triangular	0.30	0.31
H1	Uniform	0.40	0.40
H2	Triangular	0.23	0.24
H2	Uniform	0.30	0.30

6.4.3 Results – Dynamic Analysis

Tables 6.2 and 6.3 shows key results for the two models, for the H1 and H2 directions. These are obtained from the **Combinations and Envelopes** task.

For both the H1 and H2 directions, the model with elastic sections in the upper stories generally gives more conservative values for the response quantities than the model with fiber sections. The main exception is the limit state usage ratio for the coupling beams, for earthquakes in the H1 direction. This might be expected because in the fiber model there is shift of the neutral axis in the wall piers, which causes larger vertical displacements, and can cause larger coupling beam deformations.

This comparison suggests that it is reasonable to use elastic sections. However, this is not a definitive study. Before using a model with elastic sections for the design of an actual building, it would be wise to analyze a model with fiber sections in all stories, at the beginning of the study (to confirm that a model with elastic is sufficiently accurate), and/or at the end of the study (to confirm that the final design has satisfactory performance). It requires additional effort to set up models with both fiber and elastic sections, but for a large structure the savings in computer time could be substantial. The example structure takes little computer time, so the savings are small.

TABLE 6.2
Comparison of Dynamic Analyses for Structures with Fiber Sections and Elastic Section in Upper Stories
Revised Structure, H1 Direction, Max of 3 Earthquakes

Result or Limit State	Fiber Sections	Elastic Sections
Roof drift.	0.54%	0.49%
South-center flange, shear strength. Limit state usage ratio.	0.95	0.95
South-west flange, shear strength. Limit state usage ratio.	0.82	0.87
Coupling Beam Deformation. Limit state usage ratio.	0.68	0.59
Steel tension strain, upper stories.	0.28%	0.37%
Tension strain, hinge region	0.52%	0.58%

TABLE 6.3
Comparison of Dynamic Analyses for Structures with Fiber Sections and Elastic Section in Upper Stories
Revised Structure, H2 Direction, Max of 3 Earthquakes

Result or Limit State	Fiber Sections	Elastic Sections
Roof drift.	0.68	0.70%
Center wall web, shear strength. Limit state usage ratio.	0.86	1.07
West wall web, shear strength. Limit state usage ratio.	0.77	0.91
Steel tension strain, upper stories.	0.31%	0.40%
Tension strain, hinge region.	1.09%	1.36%

An additional comparison can be made using the **Energy Balance** task. The total energy is similar for both analyses, but for the analysis with elastic sections, the amount of dissipated inelastic energy is substantially larger than for the analyses with fiber sections (22% of the total vs. 9%). The energy dissipated by viscous damping is correspondingly smaller. It might be wise to tune the elastic model so that the amounts of inelastic energy dissipation are more similar.

Note that when elastic sections are used, the vertical reinforcement must be designed for the upper stories. This means that appropriate structure sections must be set up, to obtain design axial forces and bending moments. The structure sections in the example are for shear forces, and for the H2 direction they have cuts through the web elements only. Sections for design of the vertical reinforcement should also cut through the flanges.

6.5 Other Aspects

6.5.1 Gravity Framing

A key assumption for the analyses in this report is that the shear core resists all lateral load. This may be too conservative, and there may be significant differences in the response if the gravity columns are considered.

If the gravity columns are modeled, their interaction with the floor system should probably be considered. The bending stiffness of the floors could be modeled using elastic slab elements, but such a model would requires a lot of slab elements, and would add substantially to the computer time required for analysis. A simpler model would be to use beam elements along the column lines, with the effective width of the beam chosen to provide a reasonable estimate of the slab bending stiffness.

6.5.2 Bi-axial Earthquake Motions

The purpose of the analyses in this report has been to suggest modeling methods and to assess some aspects of behavior. For simplicity, the analyses assume earthquake motion in one direction only. It is usual practice to apply biaxial motions. Once appropriate pairs of motions have been chosen, this can easily be done. This report does not address the issue of selecting ground motions.

6.5.3 Sensitivity to Material Properties

In this report, only one set of material stiffness and strength values has been used. It would be useful to explore the effect of changing such properties as the yield strength of the reinforcement, shear modulus of the shear material, the support stiffness, etc.