# PERFORM
## COMPONENTS AND ELEMENTS

## FOR PERFORM-3D AND PERFORM-COLLAPSE

## VERSION 4
## AUGUST 2006

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*

# DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT, DOCUMENTATION AND TESTING OF PERFORM-3D AND PERFORM-COLLAPSE. IN USING THE PROGRAMS, 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 PROGRAMS. THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAMS AND MUST INDEPENDENTLY VERIFY THE RESULTS.

# ACKNOWLEDGMENTS

Special recognition is extended to Dr. Graham H. Powell, Professor Emeritus, University of California at Berkeley, who conceived and developed the PERFORM series of programs. Professor Powell's research, expertise and originality have resulted in many contributions to our profession, particularly in the modeling of structures for practical nonlinear analysis.

We extend thanks to the members of our profession for all of their valuable feedback that has contributed to enhancement of the products produced by CSI.

# PERFORM Components and Elements Version 4

## *Table of Contents*

# 1 PERFORM Hysteresis Loops

*Almost all of the PERFORM nonlinear components use the same action-deformation (F-D) relationship. The "backbone" YULRX relationship is described in detail in the User Guide, and the hysteresis loop for cyclic loading is described briefly. This chapter describes the hysteresis loop in more detail.*

## 1.1 Purpose

Hysteresis loops can have many different shapes, and the PERFORM model does not model all possible shapes. In practice, however, it is unlikely that you will have detailed knowledge about the behavior of the actual members in a structure, and the best you can do is make reasonable estimates of the hysteresis loop properties. If you are uncertain about such basic properties as the member strength (as you probably are) there may not be much point in worrying about details such as the amount of pinching in the hysteresis loop.

Figure 1.1 shows the type of action-deformation relationship and hysteresis loop that might be expected for a structural member.

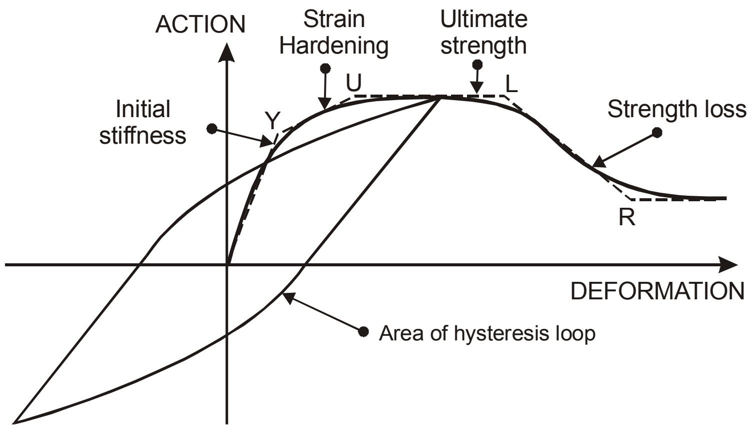



Figure 1.1 Main Aspects of Inelastic Behavior

The intent of the PERFORM action-deformation relationship, with points Y, U, L and R, is to capture the main aspects of the behavior, namely the initial stiffness, strain hardening, ultimate strength and strength loss, as shown in the figure. The main intent of the PERFORM hysteresis loop is to capture the dissipated energy (the area of the loop). This area is affected by stiffness degradation under cyclic loading.

This chapter describes the hysteresis loop model, for the following component types.

(1) Typical inelastic component with uniaxial force-deformation relationship.
(2) Concrete material for fiber cross sections and concrete struts.
(3) Buckling material for fiber cross sections and steel bar/tie/strut components.
(4) Components with added parallel stiffnesses.
(5) BRB component with "isotropic" hardening.
(6) Rubber type seismic isolator.
(7) Components with P-M-M and V-V interaction.

## *1.2 Typical Uniaxial Case*

### 1.2.1 Hysteresis Loops With No Stiffness Degradation

For the case with no stiffness (or energy) degradation, PERFORM assumes hysteresis loops as shown in Figures 1.2 and 1.3. Figure 1.2 shows the loop for elastic-perfectly-plastic (e-p-p) behavior, and Figure 1.3 shows the loop for trilinear behavior with deformation past the U point.

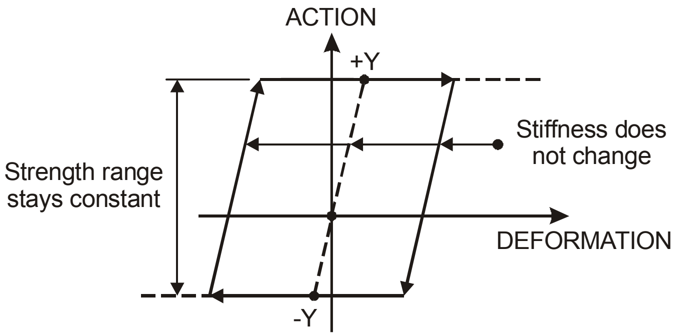



Figure 1.2 Non-Degrading Loop for E-P-P Behavior

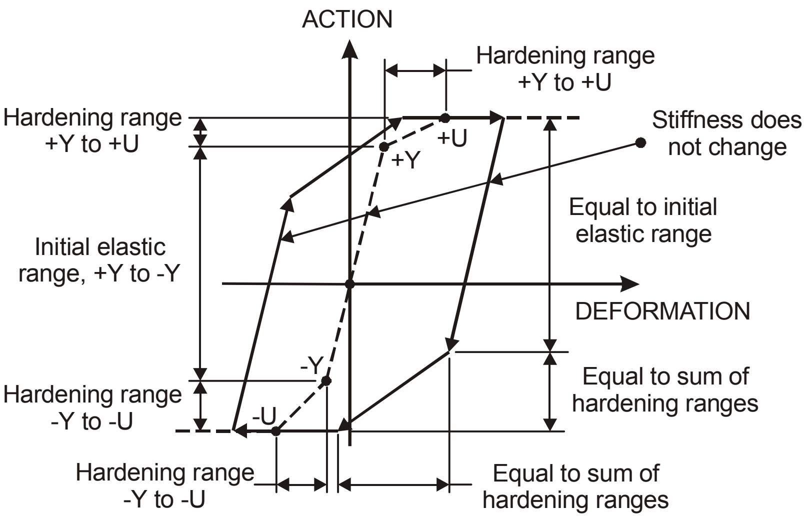



Figure 1.3 Non-Degrading Loop for Trilinear Behavior

### 1.2.2 Energy and Stiffness Degradation

If there is energy degradation (i.e., if the component properties include energy degradation factors), PERFORM adjusts the unloading and reloading stiffnesses to reduce the area under the loop. The method is simple for the elastic-perfectly-plastic (e-p-p) case, and rather more complicated for the trilinear case. The following sections consider different cases.

### 1.2.3 Degraded Loop, E-P-P Case

Figure 1.4 shows a loop for the e-p-p case with stiffness degradation.

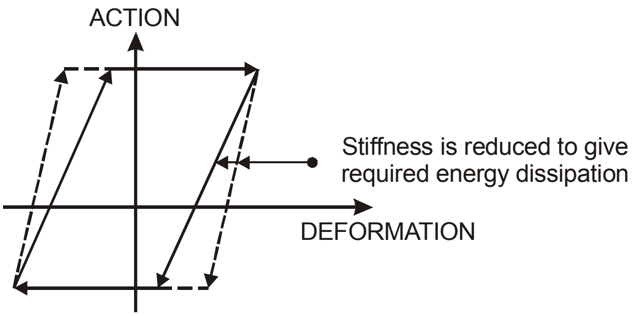



Figure 1.4 Degraded Loop for E-P-P Behavior

PERFORM uses the following method to set the loop properties.

(1) As part of the component properties, a relationship is specified between the maximum deformation of the component and the corresponding energy degradation factor. This factor is the area of the degraded hysteresis loop divided by the area of the non-degraded loop. See the User Guide for details of how energy degradation factors are specified.

(2) For the current state of the component, the maximum positive and negative deformations. These are the maximum deformations up to the current point in the analysis, not necessarily the deformations at the limits of the current deformation cycle.

(3) Let the positive and negative energy degradation factors at these deformations be, respectively, $e_{pos}$ and $e_{neg}$.

(4) The smaller of these values (more degradation) is $e_{\min}$ and the larger (less degradation) is $e_{\max}$.

(5) The energy degradation factor for the loop as a whole is $e = we_{\min} + (1-w)e_{\max}$, where $w$ is a weighting factor. In the current version of PERFORM, $w$ is always 1.0 (i.e., the amount of degradation is based on $e_{\min}$). A future version may allow you to specify the value of $w$.

(6) Degraded unloading and reloading stiffnesses are calculated to make the area of the degraded loop equal to $e$ times the area of the non-degraded loop.

### 1.2.4 Degraded Loop, Trilinear Case

The method for the trilinear case is illustrated in Figures 1.5 and 1.6.

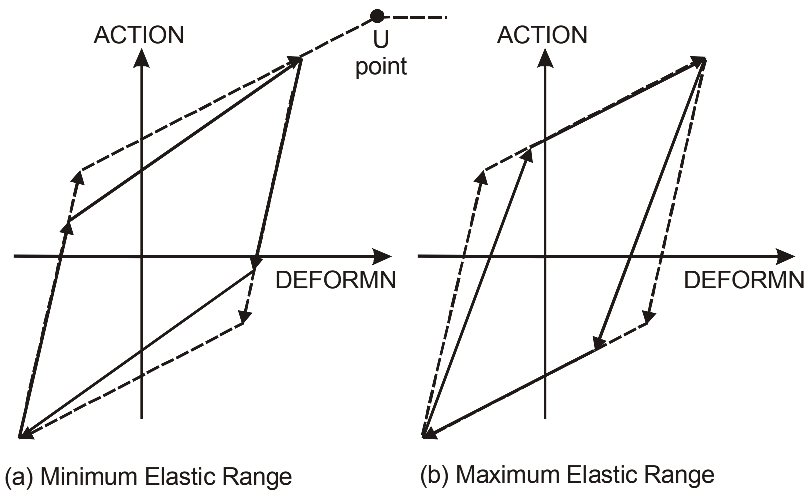



Figure 1.5 Extreme Cases, Before U Point

Figure 1.5 shows the case where the positive and negative deformations are both smaller than the U point deformation. The energy dissipation factor, *e*, is calculated as for the e-p-p case. The degraded elastic and hardening stiffnesses are then calculated to make the area of the degraded loop equal to *e* times the area of the non-degraded loop.

This figure shows two extreme shapes for the degraded loop. At one extreme, the elastic stiffness is equal to the non-degraded value. This gives a minimum elastic range and a maximum strain hardening range. At the other extreme, the hardening stiffness is equal to the non-degraded value. This gives a maximum elastic range and a minimum strain hardening range.

PERFORM allows you to control the elastic range, using the Unloading Stiffness Factor. A factor of 1.0 gives the behavior in Figure 1.5(a), with maximum unloading stiffness and minimum elastic range. A factor of minus 1.0 gives the behavior in Figure 1.5(b), with minimum unloading stiffness and maximum elastic range. The default is midway between these extremes.

Figure 1.6 shows the case where the positive and negative deformations are both larger than the U point deformation. The method is a combination of the methods in Figures 1.4 and 1.5.

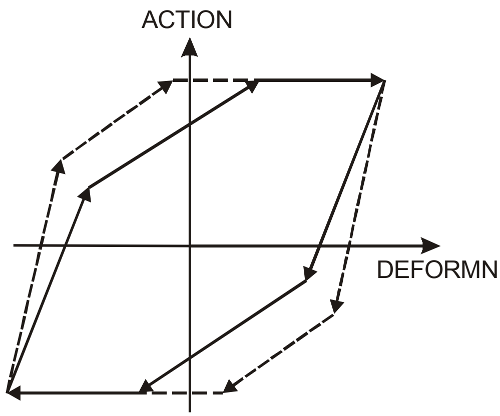



Figure 1.6  After U Point

### 1.2.5 Effect of Strength Loss

Figure 1.7 shows a hysteresis loop for deformation beyond the L point, causing strength loss. This figure is for the case with no stiffness degradation. It illustrates the effect of strength loss interaction

In the figure, strength loss occurs in the positive direction. In Case (a) this does not cause any strength loss in the negative direction. This corresponds to a value of zero for the Strength Loss Interaction Factor. Case (b) corresponds to a value of 1.0 for this factor, where strength loss in one direction causes the same amount of strength loss in the opposite direction (if the positive and negative strength losses are different, strength loss in one direction causes the same *proportional* strength loss in the opposite direction). A value of 0.5 for the strength loss interaction factor gives strength loss midway between Case (a) and Case (b).

For the case with stiffness degradation, the behavior is similar to that in Figure 1.7, except that the stiffness degrades as described in the preceding sections.

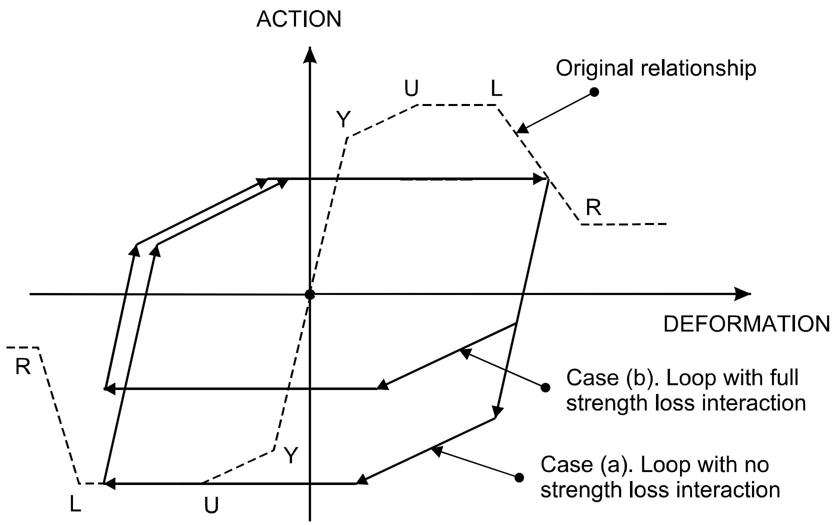



Figure 1.7 Non-Degrading Loop After Strength Loss.

## 1.3 Other Cases

### 1.3.1 Concrete Material

Figure 1.8 shows the PERFORM hysteresis model for a concrete fiber in compression. The unloading stiffness is always equal to the initial elastic stiffness. The model controls the dissipated energy by changing the reloading stiffness.

If the energy degradation factor is 1.0, reloading occurs as shown in Figure 1.8(a). This is the maximum amount of energy dissipation. If the energy degradation factor is less than 1.0, reloading occurs as shown in Figure 1.8(b). If the energy degradation factor is zero, the unloading and reloading lines are the same and there is no energy dissipation.

You can specify either finite or zero strength in tension. If you specify finite strength, unloading and reloading are as shown in Figure 1.9, with no cyclic energy dissipation.

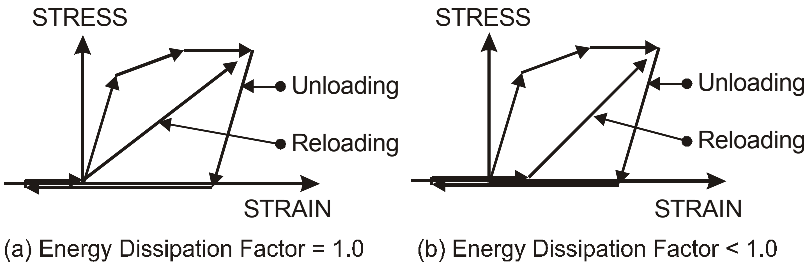



Figure 1.8 Concrete Material in Compression

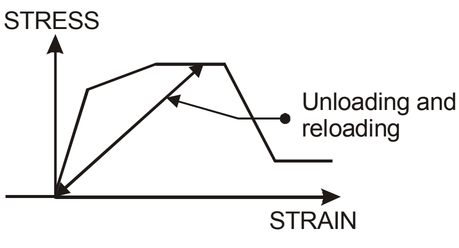



Figure 1.9 Concrete Material in Tension

PERFORM assumes that the behavior in tension and compression are independent. Hence, crushing in compression does not affect subsequent tension behavior, and cracking in tension does not affect subsequent compression behavior.

### 1.3.2 Tension-Only Material

The tension-only material is essentially the same as the concrete material, except that tension and compression are reversed.

### 1.3.3 Buckling Material

The Steel Material, Buckling Type has a hysteresis loop for axial stress vs. axial strain as shown in Figure 1.10. This material can be used in Steel Bar/Tie/Strut components, and hence in Simple Bar elements. It can also be used in fiber sections for beams and columns.

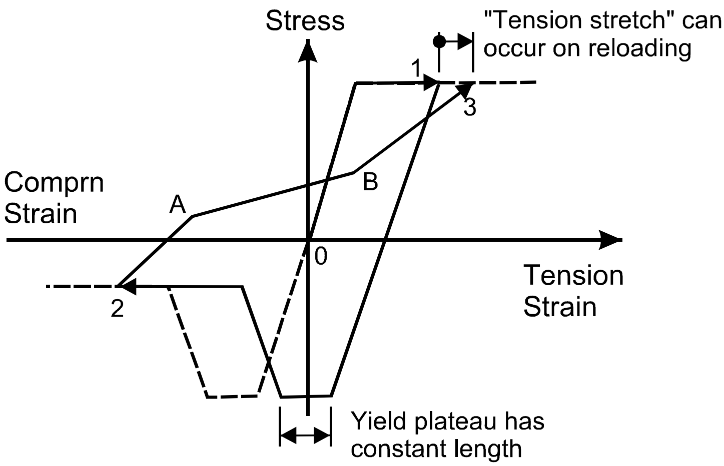



Figure 1.10 Buckling Hysteresis Loop

The dashed lines in Figure 1.10 show the basic stress-strain relationship. The solid line 0-1-2-A-B-3 shows a hysteresis loop with

(1) yield in tension, (2) buckling in compression and (3) reloading in tension.

The material has the following properties.

(1) The basic F-D relationship has the usual YULRX form. However, only e-p-p behavior is currently allowed, and strength loss in compression is required, not optional. Strength loss in tension is optional.

(2) In a typical inelastic component, the deformation at the L point is fixed. Hence, when a component is cycled inelastically the plateau between the Y and L points gets longer. In the buckling material this plateau has a fixed length. Under cyclic loading the L point moves.

(3) When the material buckles in compression then is reloaded in tension, the reloading line has three segments, controlled by points A and B in Figure 1.10. The locations of these points relative to the unloading and reloading points control the shape of the reloading line. These locations must be specified.

(4) When a bar buckles in compression, then is reloaded in tension, there is a tendency for it to increase in length. This occurs because the axial compression force is usually small after buckling, so there is not much P-M interaction, and the inelastic deformations are mainly bending. However, the tension forces are larger on reloading, so there is more P-M interaction, and the (reversed) flexural yielding tends to be accompanied by axial extension. The material has a “stretch factor” property that allows you to specify the amount of this “tension stretch”.

When you specify the properties for this material, it is a good idea to use the Plot Loops feature in the Component Properties task, to check that the hysteresis loop has the expected shape.

### 1.3.4 Component With Extra Parallel Stiffness

Some components can have an extra parallel stiffness, this stiffness is assumed to be elastic, as shown in Figure 1.11.

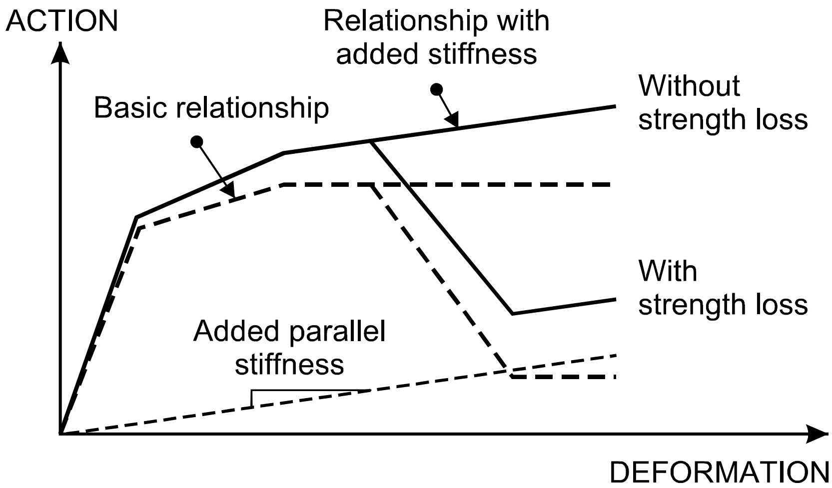



Figure 1.11 Additional Parallel Stiffness

The hysteresis loop for the basic relationship degrades in the usual way, and the added parallel stiffness stays constant.

### 1.3.5 BRB with Isotropic Hardening

The action-deformation relationship for a BRB component can be e-p-p or trilinear. There is no strength loss, and no stiffness degradation. However, to match the behavior observed in tests, the hysteresis loop can progressively grow in size This is sometimes referred to as "isotropic hardening". In effect, the component increases in strength under cyclic loading.

The PERFORM model is similar to that described in the paper *"Analytical and Experimental Studies on Buckling Restrained Braced Composite Frames" by L. A. Fahnestock, R. Sause and J. M. Ricles.*

The amount of strength increase can depend on either or both of the following.

(1) The maximum plastic deformation. You can specify a relationship between the maximum plastic (post-yield) deformation of the

element (in any cycle, not just the current cycle) and the amount of strength increase. It is important to note that the strength increase in compression depends on only the maximum deformation in tension, and vice versa.

(2) The accumulated plastic deformation. You can specify a second relationship between the accumulated plastic deformation (in all cycles up to the current cycle) and the amount of strength increase. The strength increase, whether in tension or compression, depends on the sum of the tension and compression plastic deformations. This is similar to a relationship between dissipated inelastic energy and the amount of strength increase.

You must specify two F-D relationships, one corresponding to monotonically increasing deformation, with no cyclic deformation, and a second (stronger) relationship corresponding to a fully hardened state after a maximum amount of cycling. You then specify how the F-D relationship transitions from the monotonic to the fully hardened relationship, in terms of (a) maximum deformation only, (b) accumulated deformation only, or (c) both. If you specify both, you must also specify a weighting factor that assigns relative weights to the maximum and accumulated deformations. For details see the **Component Properties** task and the input data form for the BRB component.

### 1.3.6 Rubber Type Seismic Isolator

An inelastic seismic isolator component can stiffen at large displacements. There is no stiffness degradation and no strength loss. The hysteresis loop is as shown in Figure 1.12.

As shown in the figure, the large displacement stiffening is modeled by adding an elastic stiffness with a gap. This stiffness does not affect the area under the loop, but stiffens it for both loading and unloading.

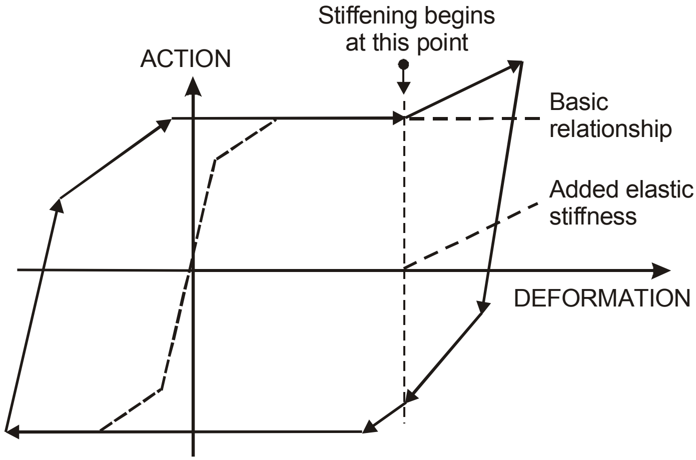



Figure 1.12 Rubber Type Isolator With Extra Stiffening

## 1.4 Components With Interaction (Multi-Axial Case)

### 1.4.1 Stiffness Degradation

For a discussion of multi-axial force-deformation relationships and non-degraded hysteresis loops see *Chapter 2, Plasticity Theory for P-M Interaction*. If you specify energy degradation factors, the hysteresis loops degrade.

For elastic-perfectly-plastic behavior there is only one yield surface, and only the elastic stiffness needs to be considered. Stiffness degradation is considered as shown in Figure 1.4 for the uniaxial case. The axial and bending stiffnesses are both reduced, in the same proportion.

For trilinear behavior there are two yield surfaces, and the elastic and hardening stiffnesses must both be considered. As shown in Figure 1.5 for the uniaxial case, there are two extreme options for stiffness degradation. In the case with the minimum elastic range, only the hardening stiffness changes. This case requires an extra linear segment (quadrilinear rather than trilinear behavior). In the case with the maximum elastic range, the hardening stiffness does not change, and the behavior stays trilinear.

For P-M-M or V-V interaction quadrilinear behavior requires an extra (third) yield surface. To avoid this complication, PERFORM keeps the hardening stiffness constant and reduces only the elastic stiffness. This is the assumption shown in Figure 1.5(b). Only two yield surfaces are needed, but the elastic deformation range and the elastic force range both increase. This means that the inner yield surface increases in size.

### 1.4.2 Behavior After Strength Loss

After strength loss occurs the behavior is assumed to become elastic-perfectly-plastic. The inner yield surface is no longer used, and the outer surface reduces in size.

## *1.5 A Warning on Strength Loss*

### 1.5.1 General

If you specify strength loss, you should be aware of the consequences and pitfalls. Some of these are considered in this section.

This section is also included in the Component Properties section of the User Guide.

### 1.5.2 Components in Parallel

Figure 1.13(a) shows a simple structure consisting of an elastic component in parallel with a yielding component that loses strength. Figure 1.13(b) shows the load-displacement relationship for the structure.

Figure 1.13(b) shows the structure behavior for two cases, as follows.

(1) If the strength loss is gradual, so that the stiffness $K_u$ is numerically smaller than the elastic stiffness $K_e$, the structure load-displacement relationship increases monotonically.

(2) However, if $K_u$ is numerically larger than $K_e$, the structure strength decreases. If $K_u$, is infinite, corresponding to sudden strength loss, the structure loses strength suddenly.

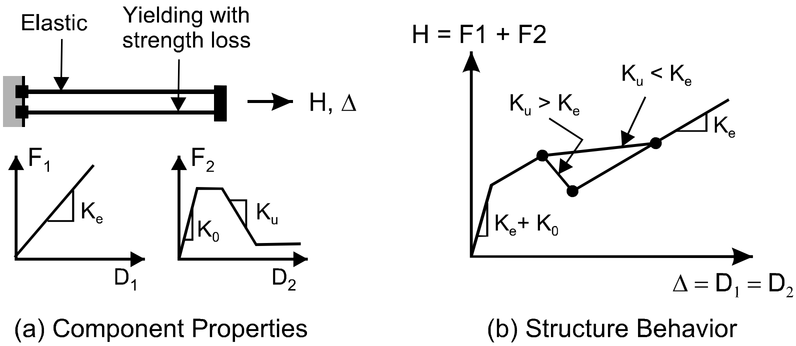



Figure 1.13 Effect of Strength Loss. Parallel Case.

### 1.5.3 Components in Series

In Figure 1.13 the components are in parallel, so that when the yielding component loses strength it transfers load to the elastic component. Figure 1.14 shows a structure with the components in series rather than in parallel.

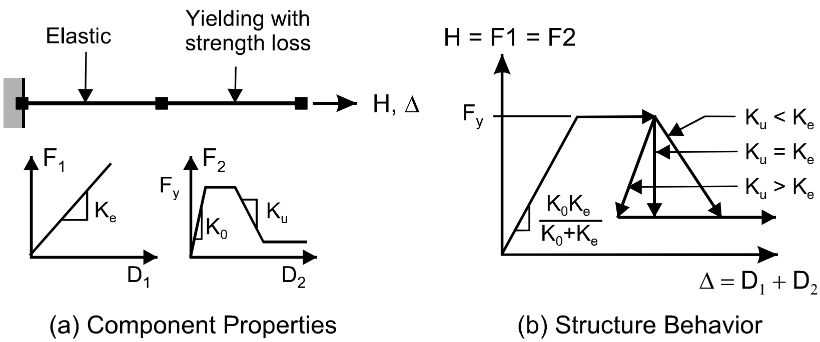



Figure 1.14 Effect of Strength Loss. Series Case.

In this case the structure load-displacement relationship always decreases as the yielding component loses strength. If $K_u$ is numerically larger than $K_e$, the load-displacement relationship reverses, as shown in

Figure 1.14(b). If the strength loss is sudden, $K_u$ is infinite and the relationship reverses with a stiffness equal to $K_e$.

### 1.5.4 Effect on Analysis Method

Nonlinear analysis models for a structure often have behavior that is a combination of the parallel and series cases. Experience indicates that PERFORM is able to obtain solutions reliably when there is strength loss, for both static and dynamic behavior. Dynamic analysis tends to be less sensitive than static, because sudden strength loss can be balanced by inertia and damping forces.

However, if reversal occurs, as in Figure 1.14(b) with $K_u > K_e$, the behavior can be very sensitive numerically. PERFORM is usually able to obtain a solution even for this type of behavior. However, this may not always be the case, and the analysis may fail to converge.

If reversal does occur, it will usually be local rather than global. This is a reason why it is always a good idea to use multiple “controlled drifts” in static push-over analysis. If only a single drift is controlled, it may be badly behaved, with reversal of the relationship between load and drift. If multiple drifts are controlled, it is unlikely that all of them will be badly behaved, even if there is local reversal.

Sudden strength loss is probably not a good idea. It rarely occurs in real structural components, and as a general rule should be avoided. Progressive strength loss as shown in Figure 1.15 is suggested as an alternative, with a strength at R equal to, say, 0.001 times the strength at U.

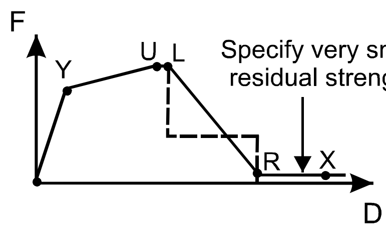



Figure 1.15 Suggested Alternative to Sudden Strength Loss

### 1.5.5 Hinge Strength Loss in a Compound Component

Although complete structures usually have behavior that is a combination of the parallel and series cases, there can be purely series behavior at the element level, specifically in PERFORM compound components.

If a frame compound component consists of elastic beam segments in series with plastic hinge components that can lose strength, the hinges and beam segments act in series, and the moment-rotation relationship for the compound component could reverse as in Figure 1.14(b). This does not happen when FEMA Beam or FEMA Column components are used, because each FEMA component is automatically divided into a rigid-plastic hinge component and an elastic beam component, and the value of $K_u$ for the hinge is chosen to give the specified $K_u$ for the beam as a whole. This means that $K_u$ for the hinge is numerically smaller than $K_u$ for the beam as a whole. However, if you use plastic hinge components directly, rather than FEMA components, you must specify the $K_u$ value for the hinge. If you make this value too large, the moment-rotation relationship for the beam as a whole can reverse. You should choose the value of $K_u$ for the hinge so that this does not happen ($K_u$ for the hinge should be numerically smaller than 6EI/L for the beam as a whole).

If $K_u$ is too large and the moment-rotation relationship for the beam as a whole reverses, PERFORM is usually able to obtain a solution. However, this may not always be the case, and the analysis may fail to converge. Behavior of this type also tends to increase the energy balance error.

# 2 Plasticity Theory For P-M Interaction

*Plasticity theory is often used to model column elements with P-M interaction. This chapter provides a simplified explanation of the essential features of plasticity theory. This chapter also shows that it can be reasonable to apply this theory to steel columns, but generally not to concrete columns.*

*PERFORM includes inelastic hinge components that have P-M interaction and are based on plasticity theory. Before using these components, you should be clear on the limitations of plasticity theory and hinge components.*

*This chapter also describes the yield surfaces that are used for P-M interaction in column elements.*

## 2.1 Yield of Metals

Figure 2.1(a) shows a piece of steel plate subjected to biaxial stress. Assume that the behavior is elastic-perfectly-plastic (e-p-p), and that the yield stress in simple (uniaxial) tension is $\sigma_Y$ . The uniaxial stress-strain relationship is shown in Figure 2.1(b).

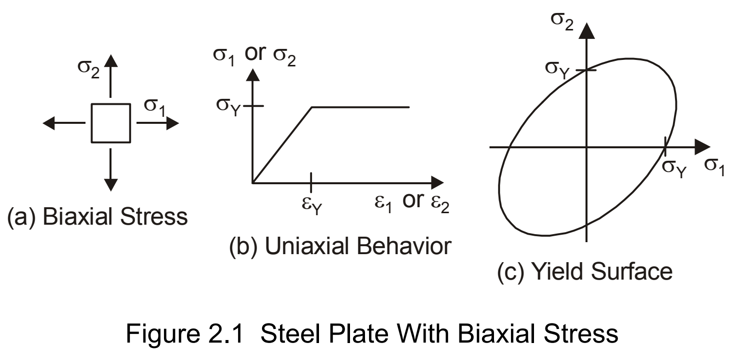



Figure 2.1 Steel Plate With Biaxial Stress

The well known von Mises theory says that for biaxial stress the material has a yield surface as shown in Figure 2.1(c). If the stress point is inside the yield surface the material is elastic. If the stress point is on the yield surface the material is yielded, and its behavior is elastic-plastic. This means that it is partly elastic and partly plastic, as explained below. Stress points outside the yield surface are not allowed.

The yield surface thus defines the strength of the material under biaxial stress. Plasticity theory defines the behavior of the material after it reaches the yield surface (i.e. after it yields). The ingredients of the theory are essentially as follows.

(1) As long as the stress point stays on the yield surface, the material stays in a yielded state. However, the stress point does not remain in one place. The stresses can change after yield, even though the material is e-p-p, which means that the stress point can move around the surface. The stress does not change after yield for an e-p-p material under for uniaxial stress, and hence biaxial stress is fundamentally different from uniaxial stress.

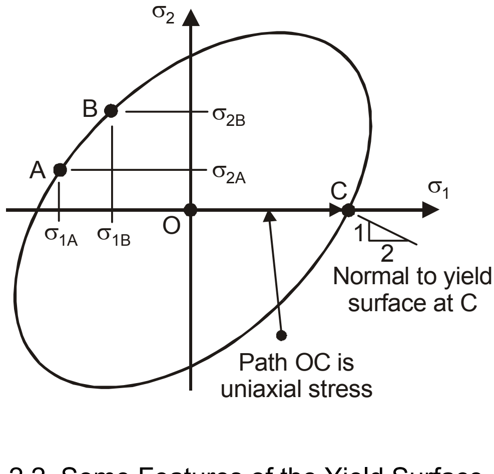



Figure 2.2 Some Features of the Yield Surface

(2) Figure 2.2 shows a yielded state, point A, defined by stresses $\sigma_{1A}$ and $\sigma_{2A}$. Suppose that strain increments $\Delta\varepsilon_1$ and $\Delta\varepsilon_2$ are imposed, causing the stresses to change to $\sigma_{1B}$ and $\sigma_{2B}$ at point B. Plasticity theory says that some of the strain increment is an elastic increment and the remainder is plastic flow. The elastic part of the strain causes the change in stress. The plastic part causes no change in stress. This is why the behavior is referred to as elastic-plastic. For yield of an e-p-p material under uniaxial stress there is no stress change after yield. Hence, all of the strain after yield is plastic strain.

(3) Plasticity theory also defines the direction of plastic flow. That is, it defines the ratio between the 1-axis and 2-axis components of the plastic strain. Essentially, the theory states that the direction of plastic flow is normal to the yield surface. For example, consider uniaxial stress along the 1-axis. As shown in Figure 2.2, the stress path is OC, and yield occurs at point C. After yield, the stress stays constant, and hence all subsequent strain is plastic. The normal to the yield surface at point C has 1-axis and 2-axis components in the ratio 2:1. Hence, the plastic strains are in this ratio, and the value of Poisson's ratio is 0.5 for plastic deformation. This agrees with experimental results.

These ingredients are sufficient to develop an analysis method for the yielding of steel. In particular, the theory can be extended from the e-p-p case to the case with strain hardening. There are many hardening theories. PERFORM uses the Mroz theory. For the case of trilinear behavior the Mroz theory is illustrated in Figure 2.3.

There are two yield surfaces, namely a Y surface (initial yield) and a larger U surface (ultimate strength). These surfaces both have the same shape. If the stress point is inside the Y surface the material is elastic. If the material is on the Y surface the material is elastic-plastic-strain-hardening. As the material hardens the Y surface moves, as indicated in the figure. When the stress point reaches the U surface, the material is elastic-plastic, as in the e-p-p case. Among other things, the Mroz theory specifies how the Y surface moves as the material strain hardens.

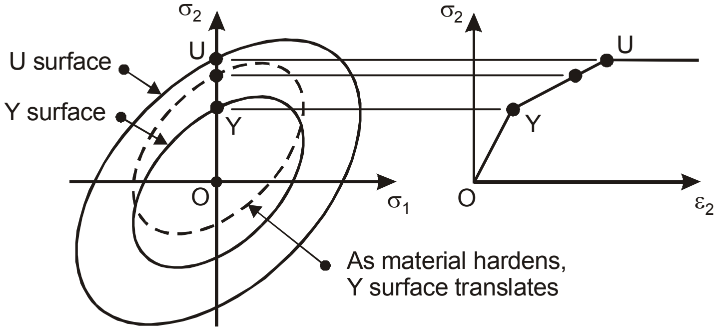



Figure 2.3 Trilinear Behavior With Mroz Theory

## 2.2 Extension to P-M Interaction

### 2.2.1 Concept

In a piece of steel under biaxial stress, the $\sigma_1$ and $\sigma_2$ stresses interact with each other. Plasticity theory models this interaction. By analogy, plasticity theory can be extended to P-M interaction in a column, where the axial force, P, and the bending moment, M, interact with each other. For the e-p-p case the yield surface is now the P-M strength interaction surface for the column cross section.

### 2.2.2 A Case Where The Analogy Works

Consider a short length of column with a cross section consisting of two steel fibers (in effect, an I section with one fiber for each flange and a web that can be ignored). This is shown in Figure 2.4(a). Each fiber is elastic-perfectly-plastic with area A and yield stress $\sigma_Y$.

A short length of the column is loaded with an axial force, P, and a bending moment, M, as shown. The axial force is applied at the reference axis for the column, which is the axis through the cross section centroid. This is important because it means that when the column is elastic there is no interaction between P and M. With the reference axis at the centroid, P alone causes axial strain but no curvature and M alone causes curvature but no axial strain (where axial strain is measured at the reference axis). If the reference axis is not at the centroid, P and M interact even before yield.

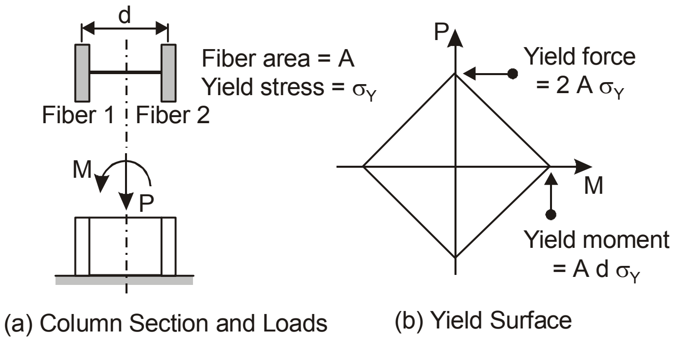



Figure 2.4 Simple Steel Column

Each fiber has only uniaxial stress, but the column has P-M interaction. It is easy to show that the P-M interaction surface is as shown in Figure 2.4(b). This is the yield surface for plasticity theory.

To see whether plasticity theory correctly predicts the behavior of the column, consider the behavior when the column is subjected to axial and bending effects. The loading and behavior are shown in Figure 2.5.

First, apply axial compression force equal to one half the yield force. The load path is O-A in Figure 2.5(a). Then hold this force constant and increase the moment. The load path is A-B. At Point B Fiber 1 yields in compression, while Fiber 2 remains elastic. The moment capacity has now been reached, and the moment-curvature relationship is e-p-p, as shown in Figure 2.5(b). However, when one fiber yields the neutral axis suddenly shifts from the center of the section to the unyielded fiber. Hence, any subsequent change in curvature is accompanied by a change in axial strain (always measured at the reference axis). This is shown in Figure 2.5(c).

The strains after yield are all plastic. That is, there is plastic bending of the cross section and plastic axial deformation. When the axial force is compression the plastic axial strain is compression, so that the column shortens as it yields in bending. If the column were in tension, it would extend as it yields in bending.

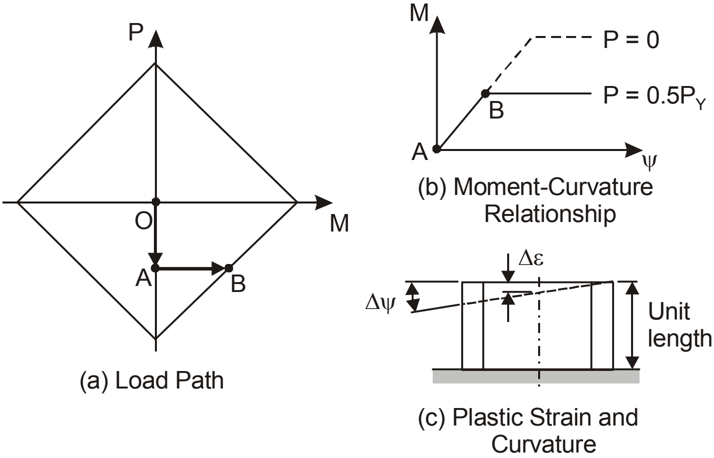



Figure 2.5 Behavior of Simple Steel Column

Figure 2.5(c) shows the changes in curvature, $\Delta\psi$, and axial strain, $\Delta\varepsilon$, after yield. The change in axial strain is $\Delta\varepsilon = 0.5d\Delta\psi$. This is the ratio that plasticity theory predicts, based on the normal to the yield surface. In this case, therefore, plasticity theory is correct.

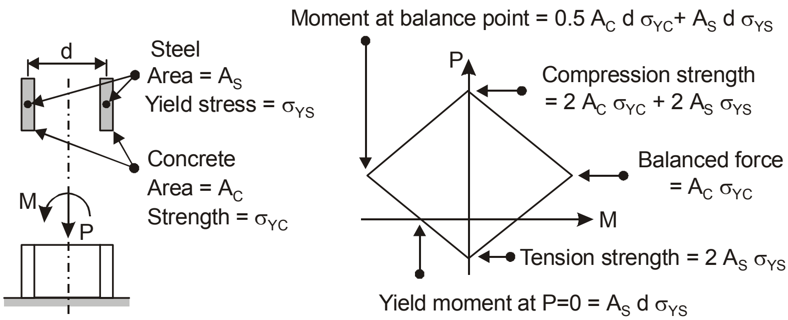
If the bending moment is reversed, keeping the axial force constant, Fiber 1 immediately unloads, and the cross section returns to an elastic state with the neutral axis at the center of the section. When the moment is fully reversed Fiber 2 yields in compression, while Fiber 1 remains elastic. This behavior is correctly predicted by plasticity theory. Hence, for this column the theory is also correct for cyclic load.

After yield in the opposite direction, the plastic axial strain is again compression. Hence, as the column is cycled plastically in bending it progressively shortens. After a number of cycles, the amount of shortening can be substantial.

This example is for a very simple cross section and for elastic-perfectly-plastic material. However, it indicates that plasticity theory can correctly account for P-M interaction. Analyses of more complex cross sections show that plasticity theory can make reasonably accurate predictions of cross section behavior. Hence, inelastic components based on plasticity theory can be used to model steel columns with P-M interaction, for both push-over and dynamic earthquake analyses.

### 2.2.3 A Case Where the Analogy Does Not Work So Well

Next, consider a simple reinforced concrete section, consisting of two concrete fibers and two steel fibers as shown in Figure 2.6(a).




(a) Column Section and Loads (b) Yield Surface

Figure 2.6 Simple Concrete Column

The steel fibers are elastic-perfectly-plastic. The concrete fibers are e-p-p in compression and have zero strength in tension. The P-M strength interaction surface for this section is shown in Figure 2.6(b). For plasticity theory, this is also the yield surface.

Consider the case with bending moment only, and zero axial force. The behavior is as follows.

(1) The concrete fiber on the tension side cracks immediately. Hence, the neutral axis shifts towards the compression side. This poses a problem for plasticity theory. Specifically, what bending and axial stiffnesses should be used for elastic behavior before the yield surface is reached?

(2) As the moment is increased there is both curvature and axial tension strain (measured at the reference axis). The relationship between curvature and axial strain depends on the shift of the neutral axis, which depends on the steel and concrete areas and moduli. In the plasticity theory there is no P-M interaction in the elastic range.

(3) When the moment reaches the yield moment the steel fiber on the tension side yields. The bending stiffness reduces to zero and the neutral axis shifts to the compression fiber. Plasticity theory captures this behavior.

(4) The moment remains constant as the curvature increases. The axial strain is tension. The relationship between axial strain and curvature is $\Delta\varepsilon = 0.5d\Delta\psi$ . Plasticity theory also captures this behavior.

Hence, plasticity theory correctly predicts the behavior at Steps (3) and (4), after the yield surface is reached, but the theory has problems in the elastic range.

Next cycle the bending moment from positive to negative, still with zero axial force. The behavior is as follows.

(5) When the bending moment is reduced the steel tension fiber immediately unloads and becomes elastic. Plasticity theory correctly predicts unloading.

(6) As the moment is decreased the curvature decreases and there is axial compression strain, which is opposite to Step (2). As before, plasticity theory does not capture this behavior.

(7) Immediately after the moment reaches zero the second concrete fiber cracks. Both concrete fibers are now cracked. The neutral axis moves to the center of the section, and the bending stiffness is the stiffness of the steel only. Plasticity theory assumes constant stiffnesses in the elastic range, and does not capture this behavior.

(8) When the moment reaches the strength of the steel fibers, both fibers yield. Plasticity theory does not capture this behavior.

(9) The steel fiber that previously yielded in tension is now yielding in compression. When the total strain in this fiber becomes zero the crack closes in the concrete fiber and it regains stiffness. The bending stiffness of the section increases and the neutral axis shifts. Plasticity theory does not capture this behavior.

(10) When the moment reaches the yield moment in the opposite direction the steel fiber on the tension side yields. The bending stiffness reduces to zero and the neutral axis shifts to the compression fiber. Plasticity theory does captures this behavior, but by now it is too late.

(11) The moment remains constant as the curvature increases, as in Step (4). The axial strain is tension. Plasticity theory does capture this behavior, but again it is too late.

In summary, plasticity theory does a mediocre job of modeling reinforced concrete for monotonically increasing loads, and a poor job for cyclic loads.

A major error for cyclic loads is that for axial forces below the balance point, plasticity theory predicts plastic strain in tension after the yield surface is reached, for both bending directions. Hence, under cyclic bending the theory predicts that the column will progressively increase in length. There can be axial growth in reinforced concrete members, but plasticity theory overestimates the amount for cyclic loading.

### 2.2.4 Are These Errors Fatal?

The major reason for considering interaction is to account for the effects of axial force on bending strength. Interaction between bending and axial deformations tends to be a secondary concern. In a typical column, the column will extend or shorten as it yields in bending, but the amount of axial deformation is probably not large. Given the many other complications and approximations in the modeling of inelastic behavior in columns, the fact that plasticity theory can overestimate the amount of axial deformation may not be very important.

This is a decision that you must make. If you use P-M-M hinges in a column, and if the extension of the column could have a significant effect on the behavior of the structure, you should examine the calculated axial extensions (for example using the Hysteresis Loops task) and satisfy yourself that these deformations are not large enough to affect the accuracy of the results for design purposes.

If you must calculate axial deformation effects more accurately, consider using fiber cross sections rather than P-M-M hinges. Fiber cross sections account for P-M-M interaction, but they use uniaxial stress-strain relationships and hysteresis loops for the fibers, and hence do not make use of plasticity theory.

One case where axial deformations are definitely important is for shear walls. If a shear wall is wide, as it cracks and yields there can be quite large axial extensions. P-M hinges may not be sufficiently accurate for modeling inelastic behavior in shear walls. This is why PERFORM uses only fiber cross sections for inelastic shear walls.

## 2.3 P-M-M Interaction

### 2.3.1 General

So far this chapter has considered only biaxial P-M interaction. For a column element in PERFORM there can be triaxial P-M-M interaction. The principles are exactly the same, the only difference being that the yield surface is 3D rather than a 2D. In plasticity theory there are major changes required to go from uniaxial plasticity to biaxial plasticity. There are no major changes in going from biaxial to triaxial, or higher.

PERFORM also uses plasticity theory for V-V shear interaction in shear hinges. Since the mechanism of inelastic shear in reinforced concrete is not plastic, plasticity theory really does not apply. However, it should give reasonable results for most practical purposes.

### 2.3.2 P-M-M Yield Surfaces

PERFORM uses a P-M-M yield surface that is similar to that described in the following pair of papers : *Nonlinear Analysis of Mixed Steel-Concrete Frames, Parts I and II, by S. El-Tawil and G. Deierlein, Journal of Structural Engineering, Vol. 126, No. 6, June 2001*. This yield surface requires only a few parameters to define its shape, yet gives you substantial control over the details of this shape.

When you specify the parameters for a yield surface in PERFORM you can plot the surface to see the effect of the parameters on its shape.

## Steel Yield Surface

Figure 2.7 shows the yield surface for a steel section.

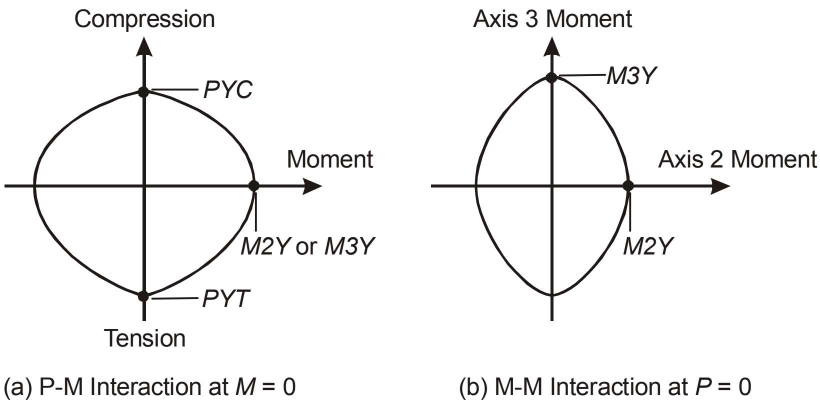



Figure 2.7 Steel Type P-M-M Yield Surface

The equations of the yield surface are essentially as follows.

In each P-M plane ($P$-$M_2$ and $P$-$M_3$) :

$$f_{PM} = \left(\frac{P}{P_{Y0}}\right)^{\alpha} + \left(\frac{M}{M_{Y0}}\right)^{\beta} \tag{2.1}$$

where $f_{PM}$ = yield function value, = 1.0 for yield, $P$ = axial force, $M$ = bending moment, $P_{Y0}$ = yield force at $M = 0$ , and $M_{Y0}$ = yield moment at $P = 0$.

Different values for the exponent $\alpha$ and the yield force $P_{Y0}$ can be specified for tension and compression. Different values for the exponent $\alpha$ can also be used in the P-$M_2$ and P-$M_3$ planes.

El Tawil and Deierlein use $\beta = 1$, which causes a sharp peak at $P = P_{Y0}$. PERFORM requires a value larger than 1.0 for $\beta$, with a suggested value of 1.1. This has little effect on the yield surface for smaller P values, yet avoids the sharp peak.

For any value of $P$, Equation (2.1) defines the $M$ values at which yield occurs, in both the P-$M_2$ and P-$M_3$ planes (put $f_{PM} = 1$ and solve for $M$). Call these values $M_{YP2}$ and $M_{YP3}$. The yield function in the $M_2$-$M_3$ plane is then:

$$f_{MM} = \left(\frac{M_2}{M_{YP2}}\right)^{\gamma} + \left(\frac{M_3}{M_{YP3}}\right)^{\gamma} \tag{2.2}$$

El Tawil and Deierlein suggest values for the exponents $\alpha$ and $\gamma$.

### Concrete Yield Surface

Figure 2.7 shows the yield surface for a concrete section.

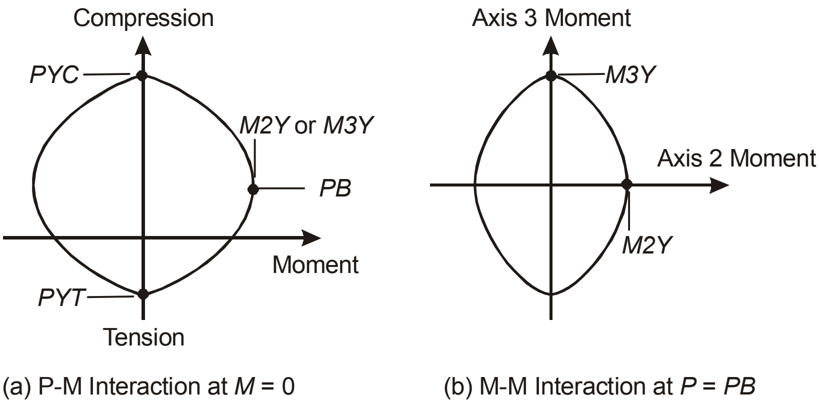



Figure 2.2 Concrete Type P-M-M Yield Surface

The equations of the yield surface are essentially as follows.

In each P-M plane:

$$f_{PM} = \left( \frac{P - P_B}{P_{Y0} - P_B} \right)^{\alpha} + \left( \frac{M}{M_{YB}} \right)^{\beta} \tag{2.3}$$

where $f_{PM}$ = yield function value, = 1.0 for yield, $P$ = axial force, $P_B$ = axial force at balance point (assumed to be the same in both *P-M* planes), $M$ = bending moment, $P_{Y0}$ = yield force at $M = 0$ , and $M_{YB}$ = yield moment at $P = P_B$ .

Different values for the exponent $\alpha$ and the yield force $P_{Y0}$ can be specified for tension and compression. Different values for the exponent $\alpha$ can also be used in the P-$M_2$ and P-$M_3$ planes.

El Tawil and Deierlein use $\beta = 1$, but PERFORM requires a value larger than 1.0.

For any value of $P$, Equation (2.3) defines the $M$ values at which yield occurs, in both the P-$M_2$ and P-$M_3$ planes (put $f_{PM} = 1$ and solve for $M$). The yield function in the $M_2$-$M_3$ plane is then given by Equation (2.2):

Again, El Tawil and Deierlein suggest values for the exponents $\alpha$ and $\gamma$.

### 2.3.3 Strain Hardening

For trilinear behavior PERFORM uses the Mroz hardening theory, as described earlier in this chapter.

### 2.3.4 Plastic Flow

PERFORM assumes plastic flow normal to the yield surface. Generally this means that as a P-M-M hinge yields in bending it also extends or shortens. As considered in this chapter, this may not be an accurate model of actual behavior. It may be noted that El Tawil and Deierlein assume no axial plastic deformation in the strain hardening range, and assume normal plastic flow only when the outer yield surface is reached. This has not been done in PERFORM, mainly because non-normal flow implies a non-symmetrical stiffness matrix, which can cause both theoretical and computational problems.

# 3 Fiber Sections and Segments

*A fiber cross section can have fibers of different types, usually steel and concrete. Fiber sections can be used for frame type elements and wall elements. This chapter describes the features of beam and column sections for frame type elements. For wall elements see later chapters.*

### 3.1.1 Fiber Sections

There are two types of fiber cross section for beam and column elements, namely the "Beam Inelastic Fiber Section" and the "Column Inelastic Fiber Section".

Beam sections use the fiber properties for axial force and in-plane bending only (usually vertical bending) and are elastic for out-of-plane bending (usually horizontal bending). Beam sections account for P-M interaction for in-plane bending.

Column sections use the fiber properties for bending about both axes, and account for P-M-M interaction.

Beam and column sections are both assumed to be elastic for shear and torsion. If you want to consider inelastic shear behavior you must use shear hinge components. PERFORM currently does not have components for inelastic torsion.

The following material types are currently allowed for the fibers : steel material, tension-only material, buckling material and concrete material. Other material types may be added in the future.

For a beam section you can specify up to 12 fibers, and for a column section up to 60. Resist the temptation to use the maximum number of fibers. This is usually not necessary, and can greatly increase the computation time. The goal should be to use the minimum number that gives reasonable results. It can be useful to set up a single-element structure (usually a vertical cantilever) and analyze it (using cyclic push-over analysis) for different numbers of fibers, to study the behavior and to find a reasonable number of fibers.

You can use a fiber cross section to model a connection that fractures. For the procedure see later in this section.

### 3.1.2 Fiber Segments in Frame Components

A Frame Compound Component can have a number of fiber segments, each of which is a segment of finite length with a uniform fiber cross section. Each fiber segment is defined by associating it with a fiber cross section and specifying the segment length.

If you have a short element, such as a short length of a pile, you may want to model it as a single fiber segment. In this case you must still use a Frame Compound Component, even though this component consists of only one fiber segment. One reason for this is that you may also want to add a shear hinge or strength section, which makes it a compound component.

### 3.1.3 Fiber Segment Behavior

The key aspect of a fiber segment is how it behaves when the fiber section becomes nonlinear, through yield of steel fibers and/or cracking and crushing of concrete fibers.

PERFORM determines the behavior of a fiber cross section by monitoring the behavior of all of its fibers. However, this is done at only one section in each fiber segment, namely the section at the midpoint of the segment. For example, if a fiber section is made up entirely of steel fibers (i.e., if it models a steel cross section), the fiber segment yields only when the combination of axial force and bending moment at the midpoint of the segment is large enough to cause a fiber to yield. The bending moment, and hence the fiber stresses, will usually be a maximum at one end of the segment, but only the stresses at the segment midpoint are considered.

One consequence of this is that when a fiber segment is located at the end of a beam, adjacent to a column face, the effective location of the plastic hinge in the segment is at the segment midpoint, not at the column face. Because of this, fiber segments should be fairly short, especially in parts of a beam or column element where the bending moment varies rapidly. If you were to use one fiber segment to model an entire beam, and if the beam had equal and opposite moments at its ends, the bending moment at the midpoint would be zero. Hence, the fiber section would never yield or crack (assuming zero axial force).

In finite element terms, a fiber segment is a beam-column finite element with a uniform cross section. In each plane of bending there are two bending deformation modes, namely a constant bending mode and a linear bending mode. The stiffness for the linear bending mode stays constant, based on the initial elastic stiffness of the cross section. The stiffness for the constant bending mode changes as the cross section becomes nonlinear. If there is P-M or P-M-M interaction (which is accounted for directly by the fiber section – there is no need for a P-M interaction surface), this interaction affects only the constant bending modes. The shear stiffness also stays constant, since this is associated only with the linear bending mode.

The behavior of a fiber segment is similar to that of a tributary length of beam with a hinge at its midpoint. Such a segment becomes nonlinear only when the hinge becomes nonlinear. Also, the stiffness of such a segment has two parts. If the bending moment is constant over the segment there is a bending moment at the hinge. Hence, if the hinge stiffness changes, the stiffness for constant bending changes. However, if the bending moment over the segment is linear, with equal and opposite moments at the ends, the moment at the hinge is zero. Hence the segment stiffness depends only on the elastic stiffness of the beam, and is not affected by the hinge (the hinge stiffness could be zero and the stiffness would still be the same as for a beam with no hinge). A fiber segment behaves in the same way.

### 3.1.4 Axial Growth

As a reinforced concrete beam is loaded and cracks, the neutral axis of its cross section moves towards the compression side, and hence the beam extends. If this extension is restrained, for example by adjacent columns, an axial compression force must develop in the beam (with corresponding shear forces in the columns). This axial force can substantially increase the strength of the beam. The effect may also have a significant effect on the behavior of the adjacent columns.

This effect is ignored in linear analysis, and also in nonlinear analysis using simple plastic hinge components. However, it is included when fiber sections are used. It is important that you be aware of this. It is a real effect, and can be present in actual structures.

Some engineers may prefer to ignore this effect. You can do this if you wish, by adding an axial release (using a PVM Release/Hinge component) to any Frame Compound Components that use fiber segments. Note that if you do this you are assuming that the axial force in the element is zero. This may be acceptable for a beam, but obviously should not be done for a column.

Be careful if you specify a rigid floor diaphragm, since this completely (and possibly artificially) restrains axial growth. If you assume a rigid floor diaphragm you should consider adding an axial release as described above.

### 3.1.5 Beta-K Damping

To account for "elastic" energy dissipation in dynamic analyses, PERFORM allows "$\alpha M+\beta K$" viscous damping. The physical significance of this damping is explained in Chapter 16 of the user guide.

For the $\beta K$ part of the damping, each element has, in parallel with it, a viscous damper element with a damping matrix $\beta K$, where (in most cases) K is the initial elastic stiffness of the element. For some elements (e.g. wall elements) K is modified to avoid potentially excessive viscous damping. For some elements (e.g. seismic isolators) K is zero.

As noted in the preceding section, when fiber segments are used in a frame type element, there can be significant axial growth. Since the initial axial stiffness of a frame element can be large, $\beta K$ damping can mean that there is a substantial axial viscous damper in parallel with the element. In some cases, this damper can develop substantial axial forces, and hence restrain the axial deformation. In our judgment this is undesirable. Hence, if a frame type element has one or more fiber segments, its $\beta K$ matrix is adjusted so that there is no axial damper (i.e., $\beta K$ damping exerts no axial forces on frame type elements).

If you use fiber segments for reinforced concrete members, the $\beta K$ matrix is based on the uncracked bending stiffness of the element. This can be large, since it assumes that concrete has the same stiffness in tension as in compression. Be sure to examine the energy balance to satisfy yourself that the amount of $\beta K$ energy dissipation in such elements is reasonable. When you specify element groups with such elements, you may want to specify a beta-K factor that is smaller than 1.

### 3.1.6 Demand-Capacity Measures

You can specify deformation capacities for fiber segments as strains and/or as rotations over the segment length. The rotation over the element length is similar to a hinge rotation.

Strain can be a useful demand-capacity measure if the bending moment gradient over the segment length is small, for example near mid-span in a beam. However, if the moment gradient is large, as near the end of a beam, the calculated strain demand will usually depend on the segment length, and will usually increase as the segment length is decreased. This is because inelastic deformations tend to concentrate in the end segments of the beam. As the end segment is made shorter, the calculated curvature in this segment usually gets larger, and hence the calculated strains also get larger. In this case it is better to use rotation over the segment length as the demand-capacity measure. This rotation is less sensitive to changes in the segment length than the curvature or strain.

If you have a number of short beam or column elements, the inelastic curvature (and effective hinge rotation) may be distributed over a number of elements. In this case the rotation over one element may be substantially smaller than the effective hinge rotation. For this situation you can use a beam type rotation gage element.

### 3.1.7 Strength Loss

If you specify strength loss for the concrete and/or steel materials in a fiber section, the ductile limit (L point) is defined in terms of strain. As noted in the preceding section, the calculated strain is sensitive to the fiber segment length. Hence, the overall deformation at which the L point is reached can also depend on the segment length. In general, the shorter the segment, the sooner strength loss will begin. This can make it difficult to choose an appropriate strain for the L point.

For a reinforced concrete cantilever column, Paulay and Priestley ("Seismic Design of Reinforced Concrete and Masonry Buildings", Wiley, 1992, p. 142) suggest that the member can be modeled as a short plastic zone at the end, with an essentially elastic member for the remainder of the member. They give the following formula for the plastic zone length:

$$L_p = 0.08L + 0.15 d_b f_y \quad \text{(ksi units)} \qquad (3.1a)$$

or

$$L_p = 0.08L + 0.022 d_b f_y \quad \text{(MPa units)} \qquad (3.1b)$$

where $L_p$ = plastic zone length, $L$ = length of cantilever (= distance to inflection point in a beam or column, usually one half of the member length), $d_b$ = reinforcing bar diameter, and $f_y$ = steel yield stress. For typical column proportions this equation gives roughly :

$$L_p = 0.5D \qquad (3.2)$$

where $D$ = member depth. For typical beam proportions the length is likely to be larger than $0.5D$.

For reinforced concrete, this length might be used as a reasonable length for a fiber segment in a reinforced concrete element.

In a Frame Compound Component that uses fiber segments, the simplest model is the plastic zone model. This has a fiber segment at each end, with lengths as above, and an elastic segment for the rest of the element. This model corresponds to that used by Paulay and Priestley. With this model, the strains that are calculated in the fiber segments should be reasonable for practical purposes. If strength loss is specified for, say, the concrete material, it should be reasonable to specify a strain at the L point that is equal to the strain at which the concrete begins to lose strength in a concrete test specimen (possibly accounting for the effects of confinement).

### 3.1.8 Bolted Connection Using Fiber Beam Section

A bolted connection that yields under axial force and/or bending moment can be modeled as a short segment with a fiber beam cross section. One application for this type of model is a bolted connection in progressive collapse analysis, where tension force due to catenary effects could cause the connection to yield or fail.

The following are some suggestions. This does not apply for shear on bolted connections. To model inelastic shear behavior you must use shear hinges.

(1) Use at least 2 fibers, and as many as one fiber per bolt, depending on the amount of detail that you require.
(2) Make the segment short, say 1 inch.
(3) Assign an arbitrary area to each fiber, say 1 sq. in.
(4) Define an Inelastic Steel Material (do this before starting the fiber beam section). Base the material strength on the bolt yield strength. If you use one fiber per bolt, if the strength per bolt is P kips, and if the fiber area is 1 sq. in., the material strength is P ksi. If you use 2 fibers and there are N bolts, the material strength is 0.5NP ksi (to give the correct tension strength).
(5) Estimate the material elastic modulus. To do this estimate the axial deformation of the connection at which the bolt yields. This does not have to be very accurate, since changes in its value probably do not affect the result much, but make a reasonable estimate. The yield strain is then this deformation divided by the segment length (for a 1 inch segment length the yield strain is numerically equal to the deformation at yield). Hence get the material elastic modulus.
(6) Estimate the L point strain for the material, at which strength loss begins. Estimate, or guess, this as a multiple of the yield strain. Again, the result may not be very sensitive to this value.
(7) Estimate the R point strain. The strength loss will not be sudden, and the R point strain is likely to be a lot larger than the L point strain. It is not a good idea to specify sudden strength loss, since it is usually unrealistic, it may cause numerical problems, and the sudden redistribution of deformation can cause strange behavior.

If you use one fiber per bolt, the connection will have bending stiffness. If you want to assume zero bending moment at the connection, insert a moment release (using a PMV release component) adjacent to the fiber segment. The moment in the segment will thus be essentially zero.

If you want to assume zero bending moment at the connection, do not do this by making the bending stiffness of the fiber section very small (by specifying very small distances between the fibers). The fiber segment is a beam segment, so it must carry shear as well as bending. In order for the effective shear deformation to be small, the initial bending EI (based on the fiber stiffnesses and locations) must be significant. The effective shear stiffness is about $12EI/L^3$, where L is the segment length. Since L is small (about 1 inch), this shear stiffness is large (as required), unless you make EI very small. If you specify very small coordinates for the beam fibers, in order to simulate a pinned connection that has a very small bending stiffness, you can also get a small effective shear stiffness, which would be a mistake.

You must also specify out-of-plane I and E for the fiber segment. Specify a reasonable value for the out-of-plane EI. Again, if you specify a very small value, the out-of-plane stiffness $12EI/L^3$ (the effective shear stiffness) may be small.

### 3.1.9 Fracturing Connection Using Fiber Beam Section

A connection that has fracturing flanges can be modeled in a similar manner. In this case the fracturing flanges are modeled as fibers, using one or more fibers per flange.

PERFORM does not have a material type that is specifically for modeling tensile fracture. We do not have specific recommendations on best way to model fracture, and you will have to experiment. The following are some suggestions.

(1) Use a tension-only material for tension, in parallel with a zero-tension concrete material for compression.
(2) Use a concrete material that has tension strength and is elastic in compression. This means that a flange will fracture in tension and will not yield in compression. This may not give the desired hysteresis loop for cyclic earthquake loading on a connection, but should be satisfactory for progressive collapse where there is unlikely to be cyclic loading.
(3) Use one of the above in parallel with an inelastic steel material to provide more energy dissipation under cyclic loading.

You will need to specify the strain at the L point, to get strength loss. This is the strain over the segment length (typically about 1 inch), and is not necessarily the same as the strain that might be measured in a test specimen. Usually you will be able to estimate the rotation across the connection at which fracture occurs. Convert this rotation to strain at the flange (essentially, strain in a yielding tension flange = rotation multiplied by distance between flanges, assuming pivoting about the compression flange).

To check the behavior you should set up a structure with a single element (usually a vertical cantilever) and analyze it for cyclic loading, to check that there is reasonable agreement with available experiments.

# 4 P-Δ and Large Displacement Effects

*This chapter reviews the different types of geometric nonlinearity, and summarizes the PERFORM options.*

*This chapter is also included as a section in the Elements chapter of the User Guide.*

## 4.1 General

P-Δ and large displacement effects can cause nonlinear behavior of elements and hence of complete structures. This is usually referred to as "geometric" nonlinearity. PERFORM-3D gives you options to include or ignore P-Δ effects. PERFORM-COLLAPSE also allows true large displacement effects.

## 4.2 P-Δ vs. True Large Displacements

In small displacements analysis there are two key assumptions, as follows.

(1) The geometric relationship between node displacements and element deformations is a linear relationship.
(2) The equilibrium equations can be formed in the undeformed position of the structure.

In reality neither of these assumptions is correct.

Mathematically, the first assumption is correct only in the limit as the displacements tend to zero. As the displacements of the nodes (or, more correctly, the rotations of the elements) increase, the relationship between node displacements and element deformations becomes progressively more nonlinear.

The second assumption is not correct for the simple reason that equilibrium must be satisfied in the deformed position. As the element rotations become progressively larger, this assumption becomes progressively less correct.

True large displacements analysis takes into account both types of nonlinearity. P-Δ analysis retains assumption (1), but considers equilibrium in the deformed position (actually it does not do this exactly, but this is not a critical point).

Figure 4.1 illustrates the difference for a simple bar.

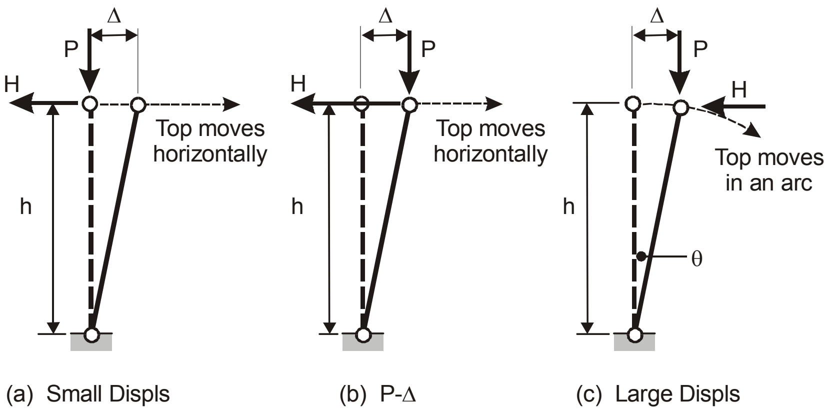



Figure 4.1 Geometric Nonlinearity

Assume for this example that the axial extension of the bar is negligible (assume that EA is very large). The three parts of the figure are as follows.

(a) ***Small displacements theory***. This theory says (i) that the top of the bar moves horizontally (this is small displacements geometry, which also predicts that the bar extension is zero), and (ii) that equilibrium can be considered in the undeformed position. Hence, the force H is zero for all values of Δ (take moments about the base of the bar).

(b) ***P-Δ theory***. This theory says (i) that the bar moves horizontally and the bar extension is zero (small displacements geometry), and

(ii) that equilibrium is considered in the deformed position. Hence $H = P\Delta/h$.

(c) ***Large displacements theory***. This theory says (i) that the top of the bar moves in an arc, so that it moves vertically as well as horizontally, so that the bar extension is indeed zero, and (ii) that equilibrium is considered in the deformed position. Hence $H=P\Delta/h\cos\theta$.

The difference between the value of H from P-Δ theory and from large displacements theory is small up to quite large rotations. For example for $\Delta/h = 0.05$ (a large drift for most structures), P-Δ theory gives $H = 0.05V$ and large displacements theory gives $H = 0.05006V$, which is a negligible difference. Also, for $\Delta/h = 0.05$ the vertical displacement in case (c) is 0.00125h. P-Δ theory predicts zero, which is not a significant error in most cases. Hence, for most structures it is accurate enough to use P-Δ theory.

Consider, however, the simple bar structure in Figure 4.2.

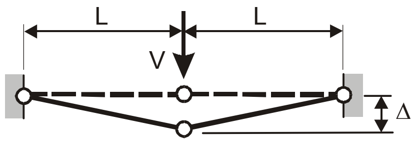



Figure 4.2  Case Where P-Δ Theory Is Not Accurate

For this structure, small displacements theory says that the structure has zero stiffness, since the theory predicts no extension of the bars as the deflection increases, and hence no axial force. Hence, force V is zero for all deflections.

If the initial force in the bars is zero, P-Δ theory also says that the force V must be zero, since the theory again predicts no extension of the bars.

Large displacements theory, however, predicts that the bars extend, and that there is a progressively increasing V force as the deflection increases.

If the initial force in the bars is P in tension, P-Δ theory says that this force stays constant, and that there is a linear relationship V = 2PΔ/L between vertical force and vertical displacement (apply equilibrium as in the preceding example). The stiffness 2P/L is the "geometric" or "initial stress" stiffness of the two bars. Large displacements theory correctly predicts a progressively increasing stiffness, with an initial stiffness equal to 2P/L.

In most building structures subjected to earthquake-type loads, the behavior is more closely analogous to Figure 4.1 than to Figure 4.2. P-Δ theory works well in this case, and has the advantage that it is both simpler to apply than large displacements theory, and requires less computation. The type of behavior in Figure 4.2 (catenary action) can occur in floor members in progressive collapse analysis. Note, however, that for catenary action to develop the members must be well anchored at both ends.

## 4.3 P-δ Effect

### 4.3.1 General

Figure 4.3(a) shows a cantilever column with vertical and horizontal loads.

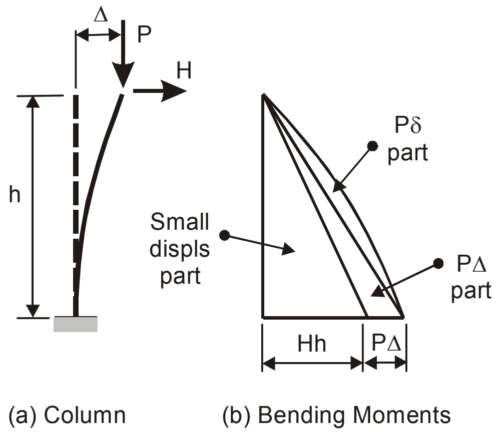



Figure 4.3 P-Δ and P-δ Effects

If the column remains elastic, it deforms as shown. Hence, considering equilibrium in the deformed position, the bending moment diagram is as shown in Figure 4.3(b) (not exactly, if we consider true large displacements, but to a high degree of accuracy).

The bending moment diagram has three parts, as follows.

(1) A small displacements part, with a moment Hh at the base. This is the moment from small displacements theory.
(2) A P-Δ part, with a moment PΔ at the base. This depends on the lateral displacement at the top of the column,
(3) A P-δ part. This depends on the bending of the column within its length.

Computationally, it is easy to account for the P-Δ part of the moment, since it depends on only the overall rotation of the column. It is more difficult to account for the P-δ part, since it depends on the bending deformation of the column (which in turn depends on the moments and on whether the column yields or remains elastic).

It is possible to account for the P-δ effect in structural analyses. However, it is important to be careful when considering this effect. In Figure 4.3, the column is elastic. Figure 4.4 shows the same column, but now it yields and forms a plastic hinge at the base.

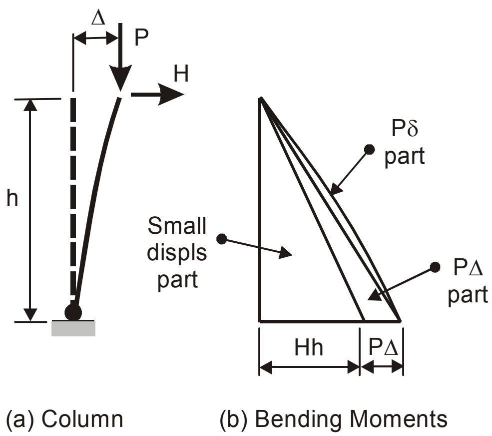



Figure 4.4 P-δ Effect When Column Yields

As the figure shows, for a given Δ the P-Δ moments are the same as before, but the P-δ moments are now much smaller. The P-δ theory must account for this. If the P-δ moments are calculated based on the elastic deformation of a column, these moments can be substantially in error.

### 4.3.2 Do You Need To Consider P-δ Effects?

If a column or brace forms plastic hinges only at its ends, it is unlikely in any practical case that the P-δ moments will be significant. If a column is stiff enough to attract substantial moments, its elastic bending deformations are usually so small that P-δ effects are insignificant. If a column is flexible enough that if could have significant P-δ effects, it usually will not attract much moment and its elastic bending deformations are again small. In most cases P-δ effects can be ignored.

This applies, however, only for columns or braces that yield only at their ends. P-δ effects can be substantial if a column or brace forms a plastic hinge within its length, since the deformations that contribute to the P-δ effect now include the inelastic as well as the elastic deformations.

Finally, note that if you divide a column member into, say, two elements, with a node in the middle of the member, the P-δ effect applies only within each element, and will almost certainly be very small. Any effects associated with displacement of the middle node are now P-Δ effects. This is one way to account for P-δ effects (i.e., add extra nodes and elements, and convert them to P-Δ effects).

### 4.3.3 Axial Shortening Due to Bending

As a column bends, the distance between its ends gets slightly shorter, because the distance along the curved column is slightly longer than the straight line between its ends. This is a geometric nonlinearity effect. It is the large displacements counterpart of the P-δ effect (i.e., P-δ theory considers P-δ moments, but ignores the shortening due to bending, whereas large displacements theory considers both). This shortening effect can be considered, but it adds a lot of complexity to the analysis.

If large displacements theory is used for column elements, strictly speaking it should include the overall effect on axial deformations, as considered for the simple bar examples, and also the axial shortening due to bending. It is not necessary, however, to consider both effects. We can choose to consider the "bar" effect and ignore the "bending" effect. This is equivalent to ignoring P-δ effects.

You can convert P-δ effects to P-Δ effects by adding nodes within a member length. Adding nodes also converts "bending" effects to "bar" effects.

## 4.4 Effect on Column Strength

Figure 4.4 also shows why P-Δ effects reduce the effective bending strength of a column. Let the moment capacity at the plastic hinge be M. This is a constant quantity – it is not affected by P-Δ effects. If we use small displacements theory, the plastic hinge forms when $M = Hh$, and the predicted horizontal strength of the column is $H = M/h$. If we consider P-Δ effects, the hinge forms when $M = Hh + P\Delta$, and the predicted strength is $H = (M - P\Delta)/h$.

## 4.5 PERFORM Options

### 4.5.1 PERFORM-3D

PERFORM-3D gives you the option of including or ignoring P-Δ effects.

For structures subjected to earthquake type loads, it is sufficient to consider only P-Δ effects. In the current version of PERFORM-3D you can consider P-Δ effects but not true large displacement effects.

### 4.5.2 PERFORM-COLLAPSE

PERFORM-COLLAPSE includes both P-Δ effects and true large displacement effects, Large displacement effects may be needed to model "catenary" effects in floor systems. It is not necessary to consider large displacement effects in columns or walls.

### 4.5.3 P-δ Effects

PERFORM does not currently consider P-δ effects (i.e. it does not consider geometric nonlinearity within the length of a column or brace element). Hence, if you use a single element to model a brace member, PERFORM will not model buckling of the brace within its length. You can, however, model this type of buckling by dividing a brace member into a number of short elements, and specifying that P-Δ effects are to be considered.

Buckling of this type can be sensitive to initial out-of straightness in the member, and you may have to make the member deliberately crooked in order to initiate buckling. If you want to consider this type of behavior, we recommend that you first test the member model in a small sub-assemblage, to make sure that you get the expected behavior.

A simpler alternative for a buckling bar is to use the buckling type steel material.

# 5 Simple Bar Element

*Simple Bar elements resist axial force only. You can specify bar elements of a variety of types, using different bar-type components.*

*This chapter reviews the components that can be used, and provides guidance on the use of Simple Bar elements.*

## 5.1 Bar-Type Components

### 5.1.1 Available Components

Simple Bar elements resist only axial force. Each element consists of one bar type component. The current version of PERFORM includes the following bar-type components.

Elastic components :
- (1) Linear elastic bar.
- (2) Nonlinear elastic gap-hook bar.
- (3) Nonlinear elastic bar with multi-linear force-extension relationship (up to 5 segments).

Inelastic components :
- (4) Inelastic bar. This has the standard PERFORM hysteresis loop.
- (5) Steel bar/tie/strut.
- (6) Concrete Strut.

Other components :
- (7) Strain gage. This is allowed for Simple Bar elements for historical reasons. It is better to use this component for Deformation Gage elements.

The required properties for each component are shown when you choose the **Component Properties** task. In most cases these properties are self-explanatory.

### 5.1.2 Deformation Measures

Some components use axial extension or shortening as the deformation measure, and others use axial strain. Be careful when you specify the stiffness properties.

For the Linear Elastic Bar and the Inelastic Bar components, the deformation measure is axial strain. Hence the initial stiffness is *EA*, where *E* = Young's modulus and *A* = bar area. The axial stiffness in terms of axial extension is calculated when the component is used in an element. This stiffness is *EA/L*, where *L* is the element length.

The deformation measure for the Gap-Hook Bar and Nonlinear Elastic Bar is axial extension.

The deformation measure for Steel Bar/Tie/Strut and Concrete Strut components is strain. These components make use of material properties. For a concrete strut component the material must be the Inelastic Concrete Material. For a steel bar/tie/strut component the material can be a Steel Material of Non-Buckling, Tension Only or Buckling type.

### 5.1.3 End Zones

In an actual bar member, the stiffness of the end connections may be larger or smaller than the stiffness in the body of the bar. If the connection stiffness has a significant effect on the bar as a whole, you may need to account for it by adjusting the component stiffness.

For steel bar/tie/strut components you can specify a rigid end zone length, as a proportion of the element length.

## *5.2 Bar Elements*

### 5.2.1 General

Simple Bar elements resist axial force only. Each element consists of one bar-type component. Depending on the component type the element is elastic or inelastic.

In an element group you can have bar elements with different component types.

Simple bar elements do not include buckling restrained brace or fluid damper elements. These are separate element types.

### 5.2.2 Geometric Nonlinearity

In PERFORM-3D you can include or ignore P-Δ effects. In PERFORM-COLLAPSE you can also consider true large displacement effects. Bar elements do not have P-δ effects.

### 5.2.3 Some Uses of Bar Elements

You can use linear elastic bar components to model members such as elastic truss bars and elastic braces in braced frames. You can also use these components to model support springs, but for elastic supports it is usually easier to use Support Spring elements.

You can use inelastic bar and/or steel bar/tie/strut components to model members such as yielding or buckling truss bars, energy absorbers such as ADAS devices, and yielding supports.

You can use gap-hook and nonlinear elastic bar components to model devices with gaps, tension-only bars, and supports that allow uplift in tension.

You can use steel bar/tie/strut and concrete strut components to set up strut-and-tie models for reinforced concrete structures, or account for thickness variations or reinforcement bands in walls.

An element that uses the strain gage component has no stiffness. You can use such elements to calculate the average extensional strain between any two nodes. However, it is better to use deformation gage elements.

### 5.2.4 Warnings on Elements for Supports and Gaps

If you use a bar element to model a support or a gap that closes, you can not make the element length zero. The minimum length is the minimum distance between nodes, with a default value of 6 inches or 15 centimeters.

If you make a bar element very short, and if you specify that P-Δ effects are to be considered, the P-Δ effect can be very large (since the P-Δ shear depends on the inverse of the element length). This is probably not what you intend when you model a support. Hence, be sure to specify no P-Δ effects.

Also, do not specify an astronomically large value, such as $10^{10}$, for the support stiffness, as this can cause numerical sensitivity in the analysis. It is common to assume that supports or gaps are rigid, but in reality they have significant flexibility. For example, a steel bar with a 6 inch (15 mm) length and a 20 square inch (2500 $mm^2$) area is extremely stiff, yet its stiffness, *EA/L*, is only about $10^5$ kips/inch or $2x10^4$ kN/mm. It is important to use realistic stiffnesses.

### 5.2.5 Initial State for Gap-Hook Bars

With a Gap-Hook Bar, if you specify that the tension (positive) gap is zero, this gap is initially closed. The bar thus has tension stiffness, but in compression the gap opens and the bar has no strength or stiffness. If you then make the compression gap very large, this is a tension-only bar.

If you specify that the compression (negative) gap is zero, this gap is initially closed. The bar thus has compression stiffness, but in tension the gap opens and the bar has no strength or stiffness. This can model a support that allows uplift. If the amount of uplift is unlimited, specify a large tension gap. If there is a tension stop, specify an appropriate gap and a tension stiffness.

If both gaps are nonzero the gap is initially open and the bar has no initial stiffness. At least one of the gaps must be nonzero.

Usually you should specify no P-Δ effects for this type of element.

## *5.3 Element Loads*

### 5.3.1 General

The only type of element load that can currently be specified for simple bar elements is Initial Strain or Extension. Other types may be added in the future.

### 5.3.2 Initial Strain

For a simple bar element you can specify, as an element load, an initial strain in the element. You can then use this load in a gravity load case to pre-stress the element.

You can use this type of load to simulate pre-stressing operations. To do this, calculate the amount that any bar element is stretched during the pre-stressing operation, and specify an initial strain that is opposite to this. For example, if a bar is 100 inches long and is stretched 0.5 inches during pre-stressing, specify an initial strain of minus 0.005. When you apply this initial strain in a gravity load case, the bar will shorten by 0.5 inches, simulating the pre-stressing operation.

### 5.3.3 Initial Extension

Alternatively you can specify the initial bar extension. In the above example the initial extension would be -0.5 inches.

# 6 Beam Element

*Beams and columns are both frame type elements that use frame compound components. Brace elements can also use be of frame type, or they can be of bar type (with no bending stiffness).*

*This chapter considers the relatively simple case of elements that have small axial forces and do not have significant biaxial bending. This includes most beam elements.*

*The next chapter builds on this chapter, and extends it to consider elements that have large axial forces and/or biaxial bending. This includes most column and brace elements.*

*It is not a simple task to model inelastic behavior in beams and columns, and there are not many guidelines that you can follow. It is important that you understand the different models and their limitations.*

## 6.1 Beam-Column Models

### 6.1.1 Modeling Goals

There are two main concerns for modeling a beam-column (or any structural member), as follows.

(1) *Force-Deformation Relationship*. A beam-column member exerts forces on the adjacent members and connections. A beam-column member also has deformations that contribute to the displacements of the complete structure. It is important to have a reasonably accurate force-deformation relationship, so that the forces and deformations are both calculated with reasonable accuracy.

(2) *Demand-Capacity Measures*. Forces and deformations are important for modeling the behavior of the structure, but demand-capacity ratios are needed to assess performance. It may be possible to assess performance using structure drifts or deflections, in which case it is not necessary to consider demand-capacity ratios at the element level. Usually, however, it is necessary to

consider member performance, which requires element demand-capacity ratios. It must be possible to calculate both demand and capacity values with sufficient accuracy for design purposes.

The main needs, therefore, are (a) sufficiently accurate F-D relationships, and (b) sufficiently accurate deformation capacities.

### 6.1.2 Beam vs. Column Models

When you set up a frame compound component in PERFORM, you can choose from a number of beam-type and column-type basic components. If you wish, you can mix beam-type and column-type basic components in a single compound component, but usually you will not. Usually you will model beam elements using beam-type basic components, and column and brace elements using column-type basic components. However, this is not required. For example, if you have a low-rise frame and the columns do not have significant biaxial bending, you could use beam-type components.

### 6.1.3 Emphasis in this Chapter

This main emphasis in this chapter is inelastic bending behavior in beam elements. However, this chapter also considers inelastic shear behavior using shear hinges, including the modeling of shear links in eccentrically braced frames.

## *6.2 Beam Element.*

### 6.2.1 Frame Compound Component

To define the properties for a beam element you must first define a frame compound component using one or more basic components. In principle you can use basic components of many different types in a single compound component. In most cases, however, you will use only a few basic component types. This chapter considers basic components of beam type. Basic components of column type are considered in the next chapter.

You can also include strength sections in frame compound components. Strength sections can also be of beam or column type.

### 6.2.2 Basic Components

PERFORM includes the following beam-type basic components. The first three of these are elastic, and the remainder are inelastic.

(1) Stiff end zone.

(2) P/V/M release or linear hinge. Use this component to model axial, shear, bending and/or torsional releases. If you wish you can specify stiffnesses, in which case the component acts as a linear hinge.

(3) Uniform elastic cross-section segment. This is a finite length segment, with a uniform cross section. For this type of segment you refer directly to a beam cross section.

(4) Uniform inelastic segment with a fiber cross section. For this type of segment you refer directly to a fiber cross section. The fibers can have material properties of a variety of types. You can use it to model beam segments and also end connections (using a very short segment length).

(5) FEMA beam, steel type. Use this component to model inelastic bending in steel beams, based on the FEMA 356 beam model. This is the simplest way to model a steel beam, but you should understand its limitations.

(6) FEMA beam, concrete type. Use this component to model inelastic bending in concrete beams, based on the FEMA 356 model. This is the simplest way to model a concrete beam, but again you should understand its limitations.

(7) Moment hinge, rotation model. This is a simple rotational hinge with rigid-plastic behavior. This hinge accounts for uniaxial bending only, with no P-M interaction.

(8) Moment hinge, curvature model. This is essentially the same as the rotation model, but it is based on the relationship between moment and curvature rather than moment and hinge rotation. This can make it easier to use in some cases. The difference between the rotation model and the curvature model is described later.

(9) Moment connection. This is a similar to a rotation hinge, but it has elastic-plastic instead of rigid-plastic behavior. You can use it model inelastic semi-rigid rotational connections.

(10) Uniaxial shear hinge, displacement model. This is a rigid-plastic shear hinge. The action is shear force and the deformation is shear displacement across the hinge. Use this component to model inelastic shear in steel or concrete beams.

(11) Uniaxial shear hinge, strain model. This is essentially the same as the displacement model, but it is based on the relationship between shear force and shear strain rather than shear force and hinge displacement. This can make it easier to use in some cases, in particular for shear links in eccentrically braced frames. The difference between the displacement model and the strain model is described later.

### 6.2.3 Strength Sections

PERFORM includes the following beam-type strength sections. You can use these to set up strength limit states.

(1) Uniaxial moment strength section.

(2) Uniaxial shear strength section.

(3) Axial force strength section. This considers axial force only.

### 6.2.4 Sign Convention

A beam element has local axes 1, 2 and 3, where Axis 1 is along the element and Axes 2 and 3 are normal to the element. Components such as cross sections and hinges use the same axes.

For a uniaxial moment hinge, inelastic bending is about Axis 3 (usually the strong axis of the member cross section). For a uniaxial shear hinge, inelastic shear is along Axis 2. Hence, uniaxial moment and shear are both in the 1-2 plane of the element.

The sign convention for axial force in a beam element is as follows.

(1) Axial force is tension positive.

(2) Bending moment about Axis 2 (usually the element weak axis) is positive for compression on the +3 side of the element. Beam elements are elastic for bending about this axis.
(3) Bending moment about Axis 3 is positive for compression on the +2 side of the element. Beam elements can be inelastic for bending about this axis.
(4) Shear force along Axis 2 is positive when the force at End I of the element is in the +2 direction. Beam elements can be inelastic for shear along this axis.
(5) Shear force along Axis 3 is positive when the force at End I of the element is in the +3 direction. Beam elements are elastic for shear along this axis.
(6) Torsional moment is positive when the moment at End I of the element is clockwise (right hand screw) about Axis 1. Beam elements are elastic for torsional moments.

This is illustrated in Figure 6.1.

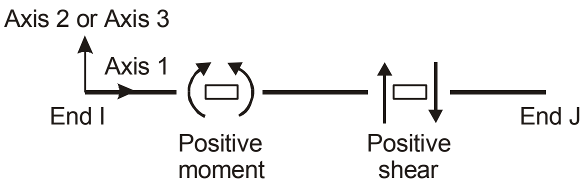



Figure 6.1 Sign Conventions

These sign conventions apply for input of basic component properties and for output of analysis results.

### 6.2.5 Model Types

There are a number of different ways to model inelastic beams in PERFORM. At one extreme are finite element models using fiber sections. At the other extreme are chord rotation models that consider the member as a whole, and essentially require that you specify only the relationship between end moment and end rotation. In between these extremes are a number of other models. The following models are considered in this chapter.

(1) Chord Rotation model. This model is the easiest to use, and it is probably the one that you will use in most cases. FEMA 356 gives specific guidelines for this model, for both steel and reinforced concrete members. However, you should understand its limitations. PERFORM implements this model using the FEMA steel beam and FEMA concrete beam components.

(2) Plastic Hinge model. This is probably the best model for beams where inelastic behavior is limited to specific locations in the beam, such as beams with "reduced beam sections". You may also need to use it if a beam can form hinges near midspan as well as at its ends. For the hinges you can use rotation hinge or curvature hinge components.

(3) Plastic Zone model. This type of model is often used for bridge members (mainly bridge columns), but you might want to use it for beams. For the plastic zone you can use hinge components or fiber section components.

(4) Detailed finite element model. You are likely to use this type of model only for particularly complex elements or to help calibrate simpler models. For this type of model you can use hinge components or fiber section components.

## *6.3 Plastic Hinges*

### 6.3.1 General

Rigid-plastic moment hinges can be used to model inelastic bending in all four of the model types. This section reviews the characteristics and limitations of plastic hinges.

### 6.3.2 Rigid-Plastic Hinge Concept

A moment plastic hinge is literally a hinge, closely analogous to a rusty hinge that rotates only after enough moment has been applied to overcome the friction between the hinge pin and the casing. Figure 6.2 shows a hinge and a possible moment-rotation relationship. The hinge is initially rigid, and begins to rotate at the yield moment.

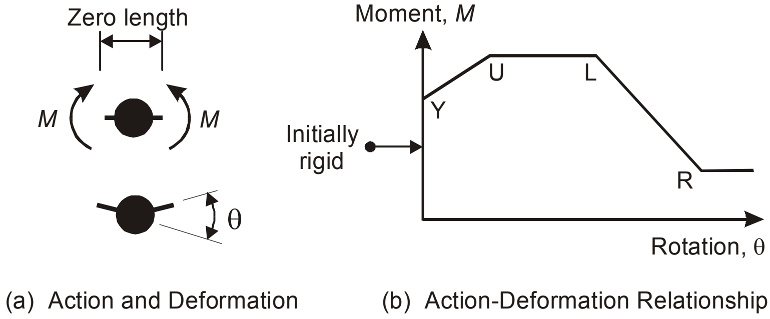



Figure 6.2 Plastic Hinge

### 6.3.3 Rotation and Curvature Hinges

PERFORM has "rotation" and "curvature" hinge components. A rotation hinge is a rigid-plastic hinge, exactly as in Figure 6.2. A curvature hinge is essentially the same, but in the action-deformation relationship the deformation is curvature rather than hinge rotation. Figure 6.3 shows the concept for a curvature hinge.

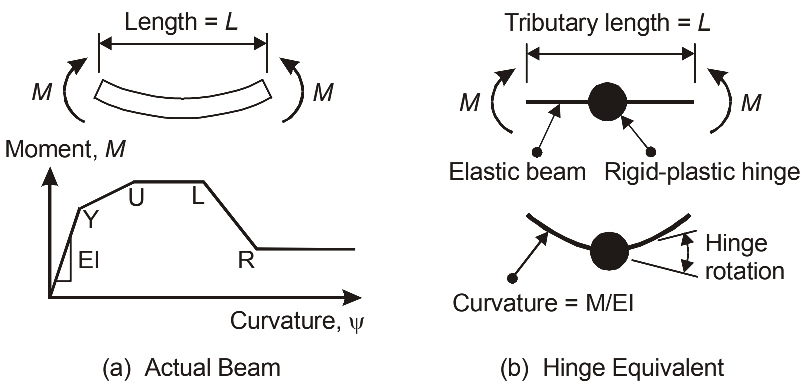



Figure 6.3 Curvature Hinge

Figure 6.3(a) shows a length of an inelastic beam, and its moment-curvature relationship. Figure 6.3(b) shows an equivalent length of beam using a rigid-plastic hinge component and an elastic beam component. The stiffness of the elastic beam component is the initial stiffness of the actual beam. The deformation of this beam accounts for the elastic part of the total deformation. The rigid-plastic hinge then accounts for the plastic part of the total deformation. The required properties for the rigid-plastic hinge are obtained by subtracting the elastic deformations from the total deformations, as shown in Figure 6.4.

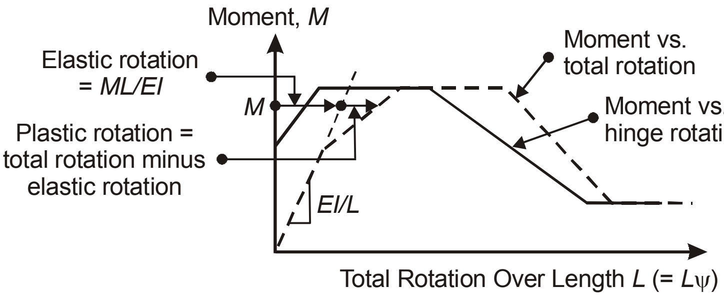



Figure 6.4  Required Properties for Rigid-Plastic Hinge

The dashed line in Figure 6.4 shows the relationship between the bending moment and the total rotation over the tributary length *L*. The total rotation is the beam curvature multiplied by *L*. For any bending moment, *M*, the rotation in the elastic beam is *ML/EI*, as shown in the figure. To give the same behavior as the actual beam, the plastic rotation in the hinge must be the total rotation minus the elastic rotation, as shown by the solid line. This is the moment-rotation relationship for the rigid-plastic hinge.

A curvature hinge requires a moment-curvature relationship and a tributary length. PERFORM uses the tributary length to convert the moment-curvature relationship to an equivalent moment-rotation relationship for the hinge. The elastic beam segment adjacent to the hinge accounts for the elastic curvature.

The advantage of a curvature hinge is that the hinge properties are independent of the tributary length. To change the tributary length for a hinge you only have to change the component length in the frame compound component. PERFORM then calculates the required properties for the equivalent rotation hinge You do not have to change the properties of the curvature hinge basic component. If you use a rotation hinge directly, and if you want to change the tributary length, you must change the properties of the rotation hinge basic component.

For a rotation hinge the deformation measure for demand-capacity calculations is hinge rotation. For a curvature hinge the deformation measure is curvature. For results output, and for calculating curvature demand-capacity ratios, PERFORM converts hinge rotations back to curvatures.

## *6.4 Fiber Segments*

### 6.4.1 General

Finite length segments using fiber cross sections can be used to model inelastic bending in some of the model types. This section explains the basic characteristics and limitations of fiber segments. See Chapter 3 for a more detailed explanation.

A fiber segment is a finite length of constant cross section in a frame compound component. A fiber segment is defined by associating it with a fiber cross section and specifying the segment length. The key aspect of a fiber segment is how it behaves when the fiber section becomes nonlinear, through yield of steel fibers and/or cracking and crushing of concrete fibers.

In physical terms, in a fiber segment, the cross section behavior is monitored at only one point, namely the midpoint of the segment. For example, if a fiber section is made up entirely of steel fibers (i.e., if it models a steel cross section), the fiber segment yields only when the combination of axial force and bending moment at the midpoint of the segment is large enough to cause a fiber to yield. The bending moment, and hence the fiber stresses, will usually be a maximum at one end of the segment, but only the stresses at the segment midpoint are considered. Because of this, fiber segments should be fairly short, especially in parts of a beam or column element where the bending moment varies rapidly. If you were to use one fiber segment to model an entire beam, and if the beam had equal and opposite moments at its ends, the bending moment at the midpoint would be zero. Hence, the fiber section would never yield or crack. Obviously this would not be a good model.

In finite element terms, a fiber segment is a beam-column finite element with a uniform cross section. In each plane of bending there are two bending deformation modes, namely a constant bending mode and a linear bending mode. The stiffness for the linear bending mode stays constant, based on the initial elastic stiffness of the cross section. The stiffness for the constant bending mode changes as the cross section becomes nonlinear. In a column element, if there is P-M-M interaction (which is accounted for directly by the fiber section – there is no need for a P-M-M interaction surface), this interaction affects only the constant bending modes. The shear stiffness also stays constant, since this is associated only with the linear bending mode.

The behavior of a fiber segment is similar to that of a tributary length of beam with a hinge at its midpoint, as in Figure 6.3. In that figure the segment becomes nonlinear only when the hinge, at the segment midpoint, becomes nonlinear. Also, the stiffness of the segment has two parts. If the bending moment is constant over the segment there is a bending moment at the hinge. Hence, if the hinge stiffness changes, the stiffness for constant bending changes. However, if the bending moment over the segment is linear, with equal and opposite moments at the ends, the moment at the hinge is zero. Hence this stiffness depends only on the elastic stiffness of the beam, and is not affected by the hinge (the hinge stiffness could be zero and the stiffness for linear bending be the same as for a beam with no hinge). A fiber segment behaves in exactly the same way.

## 6.5 Chord Rotation Model

The chord rotation model is the simplest model, with the most limitations. The basic model is shown in Figure 6.5. This is a symmetrical beam with equal and opposite end moments and no loads along the beam length.

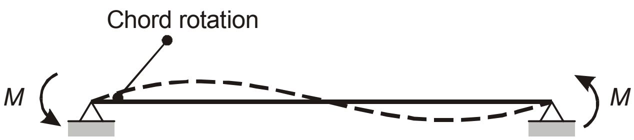



Figure 6.5 Chord Rotation Model

To use this model you must specify the nonlinear relationship between the end moment and end rotation. The end rotation is the rotation from the chord, which eliminates rigid body rotations. Note that the relationship between end moment and end rotation is not the same as the beam moment-curvature relationship.

An advantage of this model is that FEMA-356 gives specific properties, including end rotation capacities.

### 6.5.1 PERFORM Chord Rotation Model

Figure 6.6 shows a PERFORM Frame Compound Component for the chord rotation model.

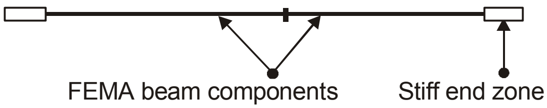



Figure 6.6 Basic Components for Chord Rotation Model

The key parts of this model are the FEMA beam components. These are finite length components with nonlinear properties. The model has two of these components, to allow for the case where the strengths are different at the two ends of the element.

Strictly speaking, the chord rotation model applies only to a symmetrical beam member, with equal strengths at both ends and an inflection point at midspan. PERFORM allows you to specify different strengths for the two components (i.e., unequal strengths at the member ends). You can also specify different lengths for the two components, to account for cases where the inflection point is not at midspan. If you wish you can also specify unequal positive and negative strengths for the components. Note, however, that when you specify different strengths at the two ends, or when you specify different positive and negative strengths, you may be stretching the limits of the chord rotation model.

### 6.5.2 Effect of End Zones on Chord Rotation

Figure 6.7 shows the deformed shape for an element with stiff end zones.


Figure 6.7 Definition of Chord Rotation

As shown in this figure, the chord rotation for the clear span between end zones is larger than for the element as a whole. In PERFORM, the rotation used for the chord rotation model is the rotation for the clear span.

### 6.5.3 Steel and Concrete Components

In FEMA-356 the properties for steel and concrete beams are presented in different ways. For steel beams the end rotation capacities are given as multiples of the yield rotation, whereas for concrete beams they are given as plastic rotations.

To account for this, PERFORM has separate FEMA components for steel and concrete beams. For a steel beam component you must specify the deformations at the U, L, R and X points as multiples of the deformation at the Y point. For a concrete beam component you must specify the deformations at the U, L, R and X points as plastic rotations. See later for the definition of plastic rotation. You must specify end rotation capacities the same way.

### 6.5.4 PERFORM Implementation

#### Components

In PERFORM the following basic components implement the chord rotation model for beams.

(1) FEMA beam, steel type.
(2) FEMA beam, concrete type.

PERFORM uses a chord rotation model as shown in Figure 6.6. It then converts this model to the model shown in Figure 6.8. Each FEMA beam component is actually two components, namely a plastic hinge and an elastic segment.

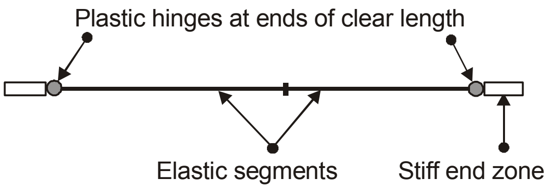



Figure 6.8 Implementation of Chord Rotation Model

If you draw a detailed deflected shape for this model (using the Moment and Shear Diagrams task), and if the end moment exceeds the yield moment, the deflected shape will show a rotation discontinuity at the beam end equal to the hinge rotation.

### Component Lengths

For this type of frame compound component you must have two FEMA beam components. If the element is symmetric, with equal strengths at both ends, the two components will have equal lengths and equal strengths. This corresponds to an inflection point at the element midpoint. If the element has different strengths at its ends, the components will have different strengths, and possibly should have different lengths (strictly speaking, the chord rotation model does not apply in this case, since it assumes symmetrical behavior).

When you specify a frame compound component you must estimate the inflection point location, and specify component lengths accordingly. If a beam element has different bending strengths for positive and negative bending, this creates a problem, since the inflection point is not fixed. The inflection point is also not fixed if there are substantial lateral loads on a beam element. This is an inherent weakness of the chord rotation model. You must use your best judgment in choosing the inflection point location.

You will usually assume that the inflection point is at the element midpoint. However, if you have a member with a moment release at one end, one component should theoretically have a length equal to the full clear span of the member, and the second should have a zero length. PERFORM requires that you specify two FEMA beam components, and does not allow a zero length. For this case, therefore, specify the length of one component to be 98% of the clear span, and the length of the other component to be 2% of the clear span. This is close enough for practical purposes.

### Hinge Properties

PERFORM calculates the properties of the plastic hinge components to give the required relationship between member end moment and end rotation (see the next section for details). If the member is symmetrical, with equal strengths at both ends the model is "exact". If the member is unsymmetrical, with unequal strengths at the ends, there is no "exact" solution. In this case PERFORM calculates the properties for each hinge using the length of the corresponding FEMA component.

## 6.5.5 Implementation Details : FEMA Steel Beam

For a FEMA steel beam component the steps in implementing the chord rotation model are as follows.

(1) The EI value for the elastic beam segment is the EI value for the FEMA component.

(2) The hinges at the element ends are curvature hinges. Note that the curvature hinge is not at the center of a tributary length. This is OK for this model.

(3) The initial stiffness in the hinge moment-curvature relationship is the EI value for the FEMA component.

(4) The shape of the moment-curvature relationship for the hinge is the same as the shape of the relationship between end moment and end rotation for the FEMA component. This means the following.

   (a) The Y, U, etc. moments for the hinge are the same as for the FEMA component.

(b) The U, L, etc. deformations in the hinge moment-curvature relationship are the same multiples of the Y deformation as in the relationship between end moment and end rotation.

(5) The tributary length of each hinge is 1/3 of the FEMA component length (1/6 of the clear length between end zones for a symmetrical element). With this tributary length, the relationship between end moment and end rotation for the combined hinge and elastic segment is the same as for the FEMA component (it is not difficult to prove this).

With these properties, the model in Figure 6.8 has the required relationship between end moment and chord rotation (exactly for a symmetrical element, as good as can be expected for an unsymmetrical element with different strengths at ends I and J).

### 6.5.6 Implementation Details : FEMA Concrete Beam

The model for a FEMA concrete beam is similar to that for a steel beam. However, rotation hinges are used, instead of curvature hinges. This is because FEMA-356 gives the properties for concrete beams and columns in terms of plastic end rotations rather than as multiples of the yield rotation. The key differences are as follows.

(1) The hinges at the element ends are rotation hinges, with rigid-plastic moment-rotation relationships.

(2) The moment-rotation relationship for each hinge is calculated by taking the moment-rotation relationship for the corresponding FEMA component and subtracting off the elastic rotations. The procedure is shown in Figure 6.9. This gives the relationship between end moment and plastic rotation for the FEMA beam component, which is the required moment-rotation relationship for the hinge.

As for the steel beam component, this gives the correct relationship between end moment and end rotation. The hinge rotations are the plastic end rotations of the element.

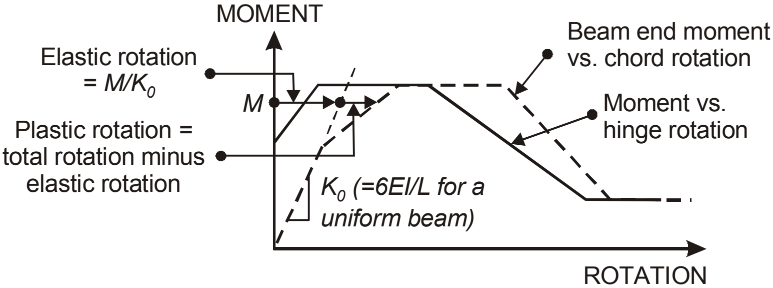



Figure 6.9 Hinge Moment-Rotation Relationship

### 6.5.7 Demands and Capacities

When you specify the deformation capacities for a FEMA beam component, these capacities are in terms of element end rotations. PERFORM calculates the hinge rotations, and converts these to element end rotations for the demand-capacity calculation.

## *6.6 Plastic Hinge Model*

### 6.6.1 Concept

The PERFORM implementation of the chord rotation model indirectly makes use of plastic hinges. You can also build models that make direct use of hinges. This gives you more flexibility in locating the hinges and assigning their properties.

For example, you might have substantial lateral load on a beam element, and it might be possible for a hinge to form near midspan as well as near the element ends. In this case you might use a model of the type shown in Figure 6.10.

You can not set up this model using FEMA beam components, because the chord rotation model assumes yielding only at the element ends. If you use FEMA components in a frame compound component, you can not also use moment hinge components.

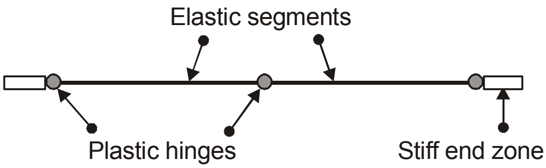



Figure 6.10 Beam Component with Plastic Hinges

A model such as that in Figure 6.10 is a "lumped plasticity" model, since the plastic deformations are concentrated in zero-length plastic hinges. In an actual member the plastic deformations are distributed over finite lengths ("distributed plasticity"). The chord rotation model is not strictly a lumped plasticity model, even though PERFORM uses plastic hinges to implement the model.

## 6.6.2 Plastic Hinge Model for Reduced Beam Section

Some beam members have regions that are deliberately made weaker than the body of the beam, so that they hinge in bending. This includes Reduced Beam Sections in steel beams, where the flange width is reduced over a length of the beam, beginning a short distance from the column face. Yielding is concentrated over a short length of beam, which approximates a zero-length plastic hinge. You can use a plastic hinge model for this type of beam.

Experimental results for reduced beam sections can be processed to give a relationship between the bending moment at the middle of the reduced section length and an equivalent plastic hinge rotation. These results can be used directly to specify the properties of a moment hinge component, including rotation capacities. A compound component for this case might be as shown in Figure 6.11.

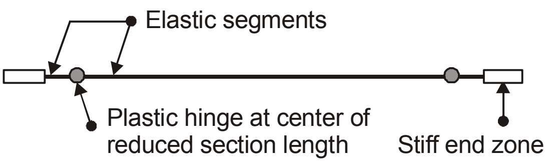



Figure 6.11 Model for Reduced Beam Section

Sometimes the experimental results for a reduced beam section are expressed in terms of a hinge located at the column face, rather than at the center of the reduced section length. In this case the hinge should be located at the column face. Be careful, because the hinge properties for the two cases can be significantly different.

### 6.6.3 Plastic Hinge Model With Several Hinges

In general you can use any number of hinge components in a frame compound component, at any locations along its length. For example, you may want to locate hinges at or near midspan as well as near the element ends, as shown in Figure 6.10.

For this type of model it may not be easy to determine the hinge properties. In an actual beam member the plastic deformations are distributed over finite lengths, whereas in the hinge model these deformations are lumped in zero-length hinges. Near the end of a beam, where the bending moment gradient is large, the length of the plastic zone may be quite short (of the order of the member depth). However, near midspan, where the moment gradient may be zero, the length of the plastic zone may be quite long. It may not be obvious how to choose the rotation properties if you use rotation hinges, or (more likely for this type of application) the tributary lengths if you use curvature hinges.

### 6.6.4 Plastic Hinge With Strength Loss

If you use a plastic hinge model, and if the hinges have strength loss, you must be careful not to specify a strength loss that is too rapid. This is considered in the main User Guide, Section 5, particularly Section 5.7.4.

This is not a concern with the chord rotation model. In this case the rate of strength loss is specified for the beam as a whole, PERFORM calculates the required hinge properties.

## 6.7 Plastic Zone Model

### 6.7.1 Concept

In a plastic hinge model the plastic deformation is concentrated in zero-length plastic hinges. In a plastic zone model the plastic deformation is distributed over finite length plastic zones, for example as shown in Figure 6.12.

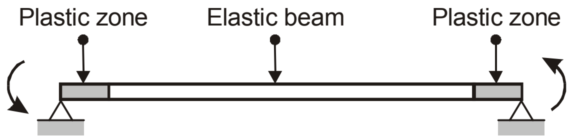



Figure 6.12 Plastic Zone Model

You can use this type of model in PERFORM, with the following restrictions.

(1) The plastic zones must be of fixed length. In an actual beam the yielded length usually changes as the load changes.

(2) You must model the plastic zone using either curvature hinges or fiber section segments.

### 6.7.2 Plastic Zone Length

The main difficulty with this model is the choice of the plastic zone length. For a practical model there are two key requirements, as follows.

(1) You must have a fairly simple way to determine the plastic zone length, preferably based on the dimensions of the beam member.

(2) The moment-curvature relationship that you specify for the plastic zone must be the same as the moment-curvature relationship for the actual beam. That is, the formula for the plastic zone length must be calibrated (preferably against experiment) so that you can use the nonlinear moment-curvature relationship of the actual

beam in the plastic zones, and the elastic stiffness of the beam for the rest of the element.

For a reinforced concrete cantilever column, Paulay and Priestley ("Seismic Design of Reinforced Concrete and Masonry Buildings", Wiley, 1992, p. 142) suggest the following plastic zone length:

$$L_p = 0.08L + 0.15 d_b f_y \quad \text{(ksi units)} \qquad (6.1a)$$

or

$$L_p = 0.08L + 0.022 d_b f_y \quad \text{(MPa units)} \qquad (6.1b)$$

where $L_p$ = plastic zone length, $L$ = length of cantilever (= distance to inflection point in a beam or column), $d_b$ = reinforcing bar diameter, and $f_y$ = *steel yield stress.* For typical beam and column proportions this equation gives about:

$$L_p = 0.5D \qquad (6.2)$$

where $D$ = member depth.

### 6.7.3 Implementation Using Curvature Hinges

Figure 6.13 shows a frame compound component for the plastic zone model, using curvature hinges.

For consistency, EI for the elastic beam components should equal the initial stiffness for the curvature hinge components.

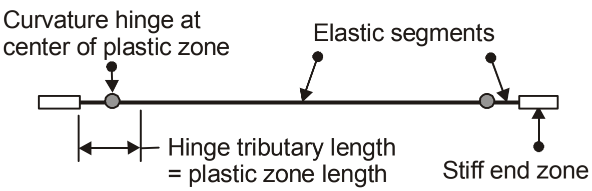



Figure 6.13 Plastic Zone Model With Hinges

### 6.7.4 Implementation Using Fiber Segments

In a model that uses fiber segments, each plastic zone is a fiber segment.

If you use concrete fiber segments, the beam element will want to extend as the concrete cracks. If you use a rigid floor constraint, you will prevent this extension. Hence, compression forces will develop in the beam, and the beam will be stiffer and stronger than in the case with unrestrained extension.

## *6.8 Detailed Finite Element Model*

### 6.8.1 General

In a detailed finite element model the beam member is divided into a number of elements along its length, for example as shown in Figure 6.14.

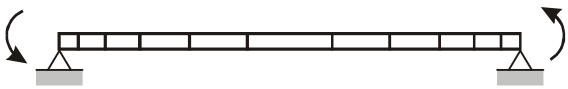


Figure 6.14 Detailed Finite Element Model

There are many different finite elements that might be used. In general they can be of low or high order (e.g., constant, linear or quadratic variation of curvature along the finite element length) and can be based on moment-curvature relationships or fiber stress-strain relationships. PERFORM provides two components that you can use for this type of model, namely curvature hinges and fiber segments.

### 6.8.2 PERFORM Models

Figure 6.15 shows a finite element model using curvature hinges.

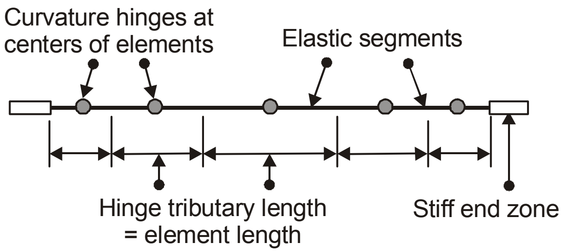



Figure 6.15 Finite Element Model With Hinges

This model has the following features.

(1) The stiffnesses for the elastic beam segments are equal to the elastic stiffnesses of the beam.
(2) The moment-curvature relationships for the moment hinges are the same as the moment-curvature relationship for the beam.
(3) The tributary lengths for the moment hinges are equal to the finite element lengths.

A model with fiber segments might look like Figure 6.14. Each element is a fiber segment. Inelastic behavior is monitored at the midpoint of each element.

In both cases, each element effectively has linear variation of elastic curvature over the element length and constant plastic curvature. This is a low order model, which is not necessarily a bad idea for inelastic analysis.

The current version of PERFORM imposes a limit of 12 basic components in a beam compound component. To get a fine finite element mesh you may have to divide a beam member into two or more elements.

### 6.8.3 A Fundamental Problem With FE Models

For an elastic structure, the goal of finite element analysis is to get a close approximation of the exact solution. In the case of a beam element, this means calculating accurate values for bending moments, shear forces and displacements. As a general rule, as the element mesh gets finer, the result gets more accurate.

For an inelastic linear structure, an additional goal is to calculate inelastic deformations that are sufficiently accurate for calculating demand/capacity ratios. The demand-capacity measure might be curvature or as fiber strain (which is closely related to curvature).

The problem is that as the mesh is made finer, the maximum calculated curvature (or strain) usually gets progressively larger. This is because beam theory for inelastic behavior predicts very large localized curvatures at the points of maximum bending moment, usually at the beam ends. Indeed, for an elastic-perfectly-plastic moment-curvature relationship, the maximum curvature after yield is theoretically infinite. Although beam theory predicts this, and finite element analysis seeks to give a solution that matches beam theory, this is not what happens in an actual beam. From a practical viewpoint the maximum curvature based on beam theory is a poor demand-capacity measure. Better measures are the beam end rotation (as in the chord rotation model), plastic hinge rotation (as in the hinge model) and average curvature over a plastic zone length (as in the plastic zone model).

Hence. If you use a finite element model, you must be careful to choose a practical demand-capacity measure. The maximum calculated curvature is not a good choice.

## *6.9 Shear Link Model*

### 6.9.1 Concept

Most inelastic beams are inelastic only in bending, and are essentially elastic in shear. A shear link in an eccentrically braced frame is the opposite. A shear link is expected to yield in shear, and to remain essentially elastic in bending.

Figure 6.16 shows a compound component for a shear link.

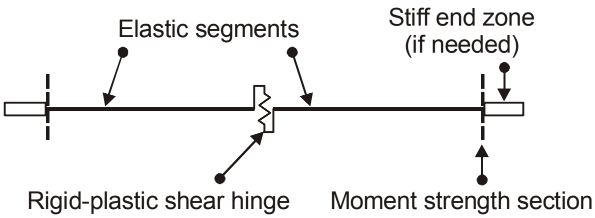



Figure 6.16 Model for a Shear Link

The key parts of this model are as follows.

(1) Two elastic beam segments that account for elastic axial, bending and shear deformations of the shear link.
(2) A shear hinge, which accounts for yield in shear. The properties of this hinge are described below.
(3) A pair of moment strength sections, to check that yield does not occur in bending.

If you draw a detailed deflected shape for this model (using the Moment and Shear Diagrams task), and if the shear hinge has yielded, you will see a displacement discontinuity at the hinge.

For inelastic shear you can use either a strain hinge or a displacement hinge, as explained in the following sections. A strain hinge is usually easier. It has the advantage that if you change the length of the shear link, the shear hinge properties are automatically updated. With a displacement hinge, if you change the shear link length you must re-calculate the hinge properties.

### 6.9.2 Hinge Properties Using a Plastic Strain Hinge

For a plastic strain hinge the action is shear force and the deformation is plastic shear strain. The action-deformation relationship is rigid-plastic. For the shear link as a whole you will have an elastic-plastic relationship between shear force and shear deformation. You must calculate rigid-plastic shear hinge properties that give the correct elastic-plastic behavior for the beam as a whole. The steps are as follows.

(1) You will know the shear area and shear modulus for the shear link. Hence you can calculate the elastic shear stiffness of its cross

section. This is GA', where G = shear modulus and A' = cross section shear area. Usually you will use an I-section beam, and the shear area will be the web area. Note that you should not use a section with a zero value for the shear area, since PERFORM will assume that elastic shear deformations are zero. For a shear link this could be a substantial error.

(2) Calculate the nonlinear elastic-plastic relationship between shear force and total shear strain for the shear link cross section. Typically this will have the same shape as the relationship between shear stress and shear strain for the material in the link.

(3) Subtract the elastic shear strain from this relationship, to get the relationship between shear force and plastic shear strain. This relationship will be initially rigid.

(4) Use the same subtraction process to convert total shear strain capacities into plastic strain capacities for the shear hinge.

(5) Specify a "shear hinge, plastic strain type" inelastic component with the above properties.

(6) When you use this basic component in the compound component for the shear link, specify its tributary length as 1.0 times the unassigned length (i.e., the full length between end zones).

### 6.9.3 Hinge Properties Using a Displacement Hinge

For a displacement hinge the action is shear force, the deformation is shear displacement across the hinge, and the action-deformation relationship is rigid-plastic. For the shear link as a whole you will have an elastic-plastic relationship between shear force and shear deformation. Strength section with the strain hinge, you must calculate rigid-plastic shear hinge properties that give the correct elastic-plastic behavior for the beam as a whole. The steps are as follows.

(1) You will know the shear area and shear modulus for the shear link. Hence you can calculate its elastic shear stiffness. This is GA'/L, where G = shear modulus, A' = cross section shear area and L = the length of the link. Usually you will use an I-section beam, and the shear area will be the web area. Note that if you use a section with a zero value for the shear area, PERFORM assumes that

elastic shear deformations are zero. For a shear link this could be a substantial error.

(2) Calculate the nonlinear elastic-plastic relationship between shear force and shear displacement for the shear link. This has the same shape as the relationship between shear stress and shear strain for the material in the link, or between shear force and shear strain for the cross section.

(3) Subtract the elastic shear displacements from this relationship, to get the relationship between shear force and plastic shear displacement. The procedure is the same as that for converting a curvature hinge to a rotation hinge, as shown earlier in Figure 6.4. However, the action is shear force rather than bending moment and the deformation is shear displacement over the shear link length rather than rotation over the hinge tributary length.

(4) Use the same subtraction process to convert shear strain capacities into deformation capacities for the shear hinge.

(5) Specify a "shear hinge, displacement type" inelastic component with the above properties.

(6) Use this basic component in the compound component for the shear link. Its length is automatically zero.

## *6.10 Element Loads*

PERFORM currently allows only gravity loads (in the –V or +V direction) for beam elements. The load can be any combination of the following types.

(1) Point load anywhere along the beam length.
(2) Uniformly distributed load over any part of the beam length.
(3) Linearly varying distributed load over any part of the beam length.

PERFORM currently does not allow initial strain load for beam elements. If this is required, you could model it using beam and simple bar elements in parallel. Make the axial stiffness (EA) of the beam element very small, and specify initial strain or initial extension load for the simple bar element.

## 6.11 Geometric Nonlinearity

In PERFORM-3D you can include or ignore P-Δ effects. Usually beam elements will be horizontal, and it is not necessary to consider P-Δ effects.

In PERFORM-COLLAPSE you can also consider true large displacement effects. This allows you to model "catenary" action.

You can not consider P-δ effects. If P-δ effects are important in a beam member, you must model these effects by dividing the member into a number of elements.

# 7 Column Element

*Beam, column and brace elements all use frame compound components. The preceding chapter considers frame compound components for the relatively simple case of beam elements. This chapter considers the more complex case of elements that have large axial forces and/or biaxial bending. This includes most column and brace elements.*

*The main difference from the preceding chapter is that column and brace elements can have substantial axial forces. Columns can also have biaxial bending, so that it is necessary to consider P-M-M interaction. For shear hinges there can also be biaxial shear, with V-V interaction.*

## 7.1 Components and Model Types

### 7.1.1 Basic Components.

PERFORM includes the following column-type basic components. The first three of these are elastic, and the remainder are inelastic.

(1) Stiff end zone.

(2) P/V/M release or linear hinge. Use this component to model axial, shear, bending and/or torsional releases. If you wish you can specify stiffnesses, in which case the component acts as a linear hinge.

(3) Uniform elastic X-section segment. This is a finite length segment, with a uniform cross section. For this type of segment you refer directly to a column cross section.

(4) Uniform inelastic segment with a fiber cross section. For this type of segment you refer directly to a fiber cross section.

(5) FEMA-356 type steel column. Use this component to model inelastic bending in steel columns, based on an interpretation of the

FEMA-356 model. This is the simplest way to model a steel column, but you should understand its limitations.

(6) FEMA-356 type concrete column. Use this component to model inelastic bending in concrete columns, based on an interpretation of the FEMA-356 model. This is the simplest way to model a concrete column, but you should understand its limitations.

(7) P-M-M steel hinge, rotation model. This is a rigid-plastic hinge with P-M-M interaction and a steel type yield surface.

(8) P-M-M concrete hinge, rotation model. This is a rigid-plastic hinge with P-M-M interaction and a reinforced concrete type yield surface.

(9) P-M-M steel hinge, curvature model. This is essentially the same as the steel rotation model, but it can be easier to use in some cases. The difference between the rotation model and the curvature model is the same as for beam hinges, as described in the preceding chapter.

(10) P-M-M concrete hinge, curvature model. This is essentially the same as the concrete rotation model, but it can be easier to use in some cases.

(11) Biaxial shear hinge, with V-V interaction. This is a rigid-plastic shear hinge that can yield in two directions.

### 7.1.2 Strength Sections

PERFORM includes the following column-type strength sections. You can use these to set up strength limit states.

(1) P-M-M steel strength section, with an interaction surface of steel type.

(2) P-M-M concrete strength section, with an interaction surface of reinforced concrete type.

(3) Biaxial shear strength section, with V-V interaction.

(4) Axial force strength section. This considers axial force only.

### 7.1.3 Sign Convention

The sign convention is the same as for a beam element. For a P-M-M hinge the bending moments are about Axes 2 and 3. For a V-V shear hinge the shear forces are along Axes 2 and 3.

### 7.1.4 Model Types

The same four model types can be used for columns as for beams, namely:

(1) Chord Rotation model.
(2) Plastic Hinge model.
(3) Plastic Zone model.
(4) Finite element model.

## *7.2 Hinges With P-M-M Interaction*

### 7.2.1 P-M-M Hinge

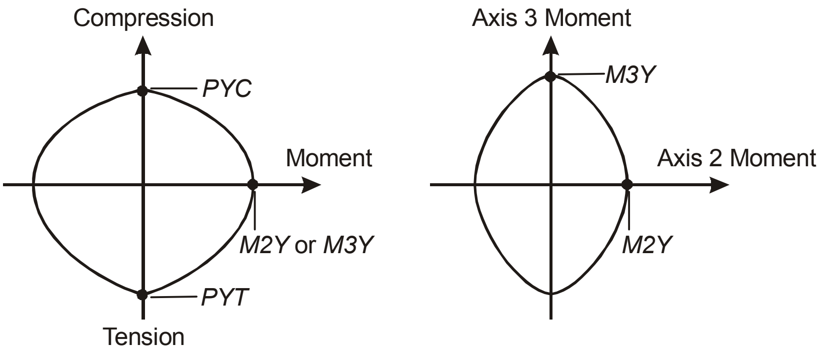
A rigid-plastic hinge with P-M-M interaction is conceptually similar to a uniaxial moment hinge. The major differences are as follows.

(1) A P-M-M hinge needs a yield (interaction) surface to define when yield occurs and what happens after yield. The surfaces used for steel and concrete columns are described later.
(2) When a hinge yields it deforms axially as well as rotationally. It is easy to visualize rotation at a zero-length moment hinge, but it not so easy to visualize axial deformation along a zero length axial hinge. Mathematically, however, the two concepts are the same. Both imply infinite strains.
(3) For a rotation hinge, bending deformation is rotation, and axial deformation is axial displacement across the hinge. For a curvature hinge, bending deformation is curvature and axial deformation is axial strain.

PERFORM uses plasticity theory to model P-M-M interaction. You should be familiar with the assumptions and limitations of this theory. For a physical explanation see Chapter 2, *Plasticity Theory for P-M Interaction.* That chapter shows that plasticity theory can give reasonable results for steel columns, but can be less accurate for concrete columns.

### 7.2.2 Steel Type P-M-M Interaction Surface

Figure 7.1 shows the shape of the yield surface that PERFORM uses for a P-M-M yield surface of steel type.




(a) P-M Interaction at $M = 0$ (b) M-M Interaction at $P = 0$

Figure 7.1 Steel Type P-M-M Yield Surface

The equations of this surface are given in *Chapter 2, Plasticity Theory for P-M Interaction.* When you specify the yield surface in PERFORM, you can plot it to show its shape. You can quickly vary its parameters and re-plot to explore their effects.

If the current P-M2-M3 point is inside the yield surface the hinge is elastic. When the P-M2-M3 point reaches the yield surface the hinge yields. For elastic-perfectly-plastic behavior the P-M2-M3 point stays on the yield surface. For trilinear behavior there is a Y surface that lies inside a larger U surface. Both surfaces must have the same shape. As the hinge strain hardens the Y surface translates, without changing size, until it reaches the U surface. This is kinematic hardening. The well known Mroz theory is used to define the motion of the Y surface.

### 7.2.3 Concrete Type P-M-M Interaction Surface

Figure 7.2 shows the yield surface that PERFORM uses for a P-M-M yield surface of concrete type.

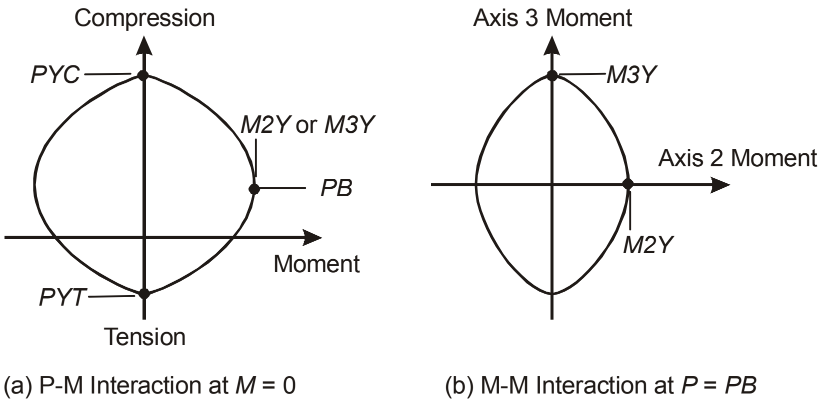



Figure 7.2 Concrete Type P-M-M Yield Surface

The equations of this surface are given in *Chapter 2, Plasticity Theory for P-M Interaction.* When you specify the yield surface in PERFORM, you can plot it to show its shape. You can quickly vary its parameters and re-plot to explore their effects.

### 7.2.4 Unsymmetrical Sections

In the current version of PERFORM the yield surfaces for P-M-M interaction apply only for symmetrical cross sections with equal positive and negative bending strengths. In principle it is possible to use yield surfaces for unsymmetrical sections, but the yield surface can become complex. PERFORM currently does not allow unsymmetrical yield surfaces. If you have a column with an unsymmetrical section and you must consider inelastic behavior, you must use a fiber cross section.

### 7.2.5 Tributary Lengths for Curvature Hinges

If you use a P-M-M curvature hinge in a frame compound component, you must specify a tributary length. PERFORM uses the same length for bending about both axes and for axial force. The moment-curvature and force-strain relationships for the curvature hinge are processed to obtain moment-rotation and force-displacement relationships for an equivalent rotation hinge, using the same procedure as for a uniaxial moment hinge.

### 7.2.6 Properties for Trilinear Behavior

Trilinear behavior with interaction is substantially more complex that elastic-perfectly plastic behavior. For e-p-p behavior there is only one yield surface, which does not move. For trilinear behavior there are two surfaces, and the inner one moves around. It is a substantially more complex task to specify the properties for a trilinear hinge than for an e-p-p hinge. Given the many uncertainties in modeling inelastic behavior in columns, it may be sufficiently accurate to assume e-p-p behavior.

If you specify trilinear behavior for a P-M-M hinge, you must specify U deformations for both bending and axial force. You can estimate the U point bending deformations in the same way as for a uniaxial moment hinge in a beam. It can be more difficult to estimate the U point axial deformation. This section suggests a method.

Consider a steel curvature hinge with trilinear relationships for moment vs. curvature and axial force vs. axial strain. The Y point corresponds to first significant yield. The yield moment, $M_Y$, and the yield axial force, $P_Y$, are given approximately by:

$$M_Y = \sigma_Y \frac{I}{c} \text{ and } P_Y = \sigma_Y A \tag{7.1}$$

where $\sigma_Y$ = yield stress, $A$ = cross section area, $I$ = cross section inertia, and $c$ = approximately distance to extreme fiber. The distance $c$ is roughly equal to the radius of gyration of the cross section. Hence, roughly:

$$\frac{M_Y}{P_Y} = \frac{I/c}{A} = \frac{Ar^2/c}{A} \cong r = \sqrt{\frac{I}{A}} = \sqrt{\frac{EI}{EA}} \tag{7.2}$$

That is, the ratio of yield moment to yield strength should be roughly equal to the square root of the ratio of initial bending stiffness to initial axial stiffness. This applies for both bending axes. PERFORM checks this, and issues a warning if the ratios are widely different.

By the same reasoning, the same equation should be satisfied by the strain hardening stiffnesses. Hence you can calculate a reasonable value for the axial strain at the U point as follows.

(1) Calculate $M_Y$ and $P_Y$, at the Y point.
(2) Decide on the U point curvature and bending moment.
(3) Hence calculate the hardening value of *EI*.
(4) Since the Y and U yield surfaces have the same shape, $P_U / P_Y$ must equal $M_U / M_Y$. This gives $P_U$.
(5) Hence calculate a reasonable value for the hardening value of *EA*.
(6) Hence get the U point axial strain.

You can apply the same method to get reasonable values for the hardening ratios and U point deformations for a rotation hinge.

### 7.2.7 Strength Loss

For the onset of strength loss (the L point) in P-M-M hinges, PERFORM uses bending deformations only (i.e., axial deformations are not considered).

When you specify the L point for strength loss you must specify L point bending deformations about both Axis 2 and Axis 3. The L point is reached when the following equation is satisfied.

$$\left(\frac{D2}{DL2}\right)^2 + \left(\frac{D3}{DL3}\right)^2 = 1 \qquad (7.3)$$

where *D2*, *D3* are the current bending deformations about Axes 2 and 3, and *DL2*, *DL3* are the L point deformations.

You must also specify the ratio between the L point strength and the R point strength. You can specify one ratio for bending moment and a different ratio for axial force. If you specify the same ratio for axial force as for bending, as the hinge loses strength the yield surface decreases in size without changing shape. If you specify different ratios, the yield surface reduces in size and changes shape.

### 7.2.8 X Point

When you specify the X point must specify X point bending deformations about both Axis 2 and Axis 3, and also an X point axial deformation. These bending and axial deformations are checked separately. The X point is reached when either deformation exceeds the corresponding X point deformation, whichever occurs first.

The X point in bending is reached when the following equation is satisfied.

$$\left(\frac{D2}{DX2}\right)^2 + \left(\frac{D3}{DX3}\right)^2 = 1 \tag{7.4}$$

where *D2*, *D3* are the current bending deformations about Axes 2 and 3, and *DX2*, *DX3* are the X point deformations.

### 7.2.9 Deformation Demand-Capacity Ratios

For deformation demand-capacity ratios in P-M-M hinges, PERFORM uses bending deformations only.

When you specify the deformation capacities, you can specify bending deformation capacities for up to 5 performance levels. The deformation demand-capacity ratio is calculated as follows.

$$\text{D/C Ratio} = \sqrt{\left(\frac{D2}{DC2}\right)^2 + \left(\frac{D3}{DC3}\right)^2} \tag{7.5}$$

where *D2*, *D3* are the current bending deformations about Axes 2 and 3, and *DC2*, *DC3* are the deformation capacities.

For steel P-M-M hinges you can specify that the deformation capacities depend on the axial force. For a concrete P-M-M hinge you can specify that the deformation capacities depend on both the axial force and the shear force.

## 7.3 Shear Hinges With V-V Interaction

### 7.3.1 Yield Surface

The equation of the yield surface for a shear hinge with interaction between shear forces V1 and V2 is:

$$\left(\frac{V2}{V2Y}\right)^{\alpha}+\left(\frac{V3}{V3Y}\right)^{\alpha}=1 \qquad (7.6)$$

where *V2* and *V3* are the current shear forces, and *V2Y* and *V3Y* are the yield values. The exponent $\alpha$ controls the shape of the surface.

### 7.3.2 Effect of Axial Force on Shear Strength

In reinforced concrete the shear strength can depend on the axial force. In the current version of Peform-3D you can not account for this effect. In Equation 7.6 the strengths *V2Y* and *V3Y* are constant values.

## 7.4 P-M-M and V-V Strength Sections

For P-M-M and V-V strength sections, PERFORM uses the same interaction surfaces as for P-M-M and V-V hinges.

In V-V strength sections, you can account for the effect of axial force on shear strength. You can also account for the effect of hinge rotation on shear strength. See the main user guide for details.

## 7.5 Chord Rotation Model

### 7.5.1 General

In PERFORM the following basic components implement the chord rotation model for beams.

(1) FEMA column, steel type.
(2) FEMA column, concrete type.

### 7.5.2 Implementation Details

The steps for implementing FEMA steel and concrete column components are essentially the same as for a beam, as described in the preceding chapter.

For a FEMA steel column component, the tributary lengths for the equivalent curvature hinges are assumed to be the same for axial effects as for bending. This requires some explanation.

If a column element has uniform axial strength over its length, and if it is subjected to pure axial force, it will yield axially over its full length. In this case, the effective hinge tributary length for axial effects is the full length of the element. It is not so clear what tributary length should be used when a column has non-uniform axial strength, or when there is both bending and axial force at the hinge. In fact, there is no single correct length.

In the PERFORM model, the same tributary length is used for axial forces as for bending moments (i.e., 1/3 of the FEMA component length for each hinge). Given all of the other approximations that are made when plasticity theory is used for P-M-M interaction, this should be a reasonable assumption. It is impractical to vary the length based on the axial force.

## *7.6 Other Models*

The concepts and procedures for the plastic hinge, plastic zone and finite element models are essentially same as for a beam.

## *7.7 Element Loads*

PERFORM allows only gravity loads (in the –V or +V direction) for column elements. For a vertical column this means only loads along the column, not transverse. To account for column weight it is usually easiest to use self weight loads, rather than element loads.

If needed, element loads can be any combination of the following types.

(1) Point load anywhere along the column length.

(2) Uniformly distributed load over any part of the column length.
(3) Linearly varying distributed load over any part of the column length.

## 7.8 Geometric Nonlinearity

In PERFORM-3D you can include or ignore P-Δ effects. You will usually include P-Δ effects for columns

In PERFORM-COLLAPSE you can also consider true large displacement effects. For collapse analysis it is not necessary to consider true large displacement effects for vertical columns, and P-Δ effects are sufficient.

You can not consider P-δ effects in a single element. In the unlikely event that P-δ effects are important in a column member, you must model these effects by dividing the member into a number of elements.

# 8 Connection Panel Zone Element

*This Chapter reviews the components that can be used to model panel zones in beam-to-column connections, and provides guidance on their use.*

## *8.1 Panel Zone Components*

### 8.1.1 Components

PERFORM includes the following panel zone components.

(1) Linear elastic panel zone.
(2) Inelastic panel zone.

The model is the same for both components, but the inelastic component has a nonlinear action-deformation relationship.

### 8.1.2 Model

Figure 8.1 shows a beam-to-column connection in a steel frame.

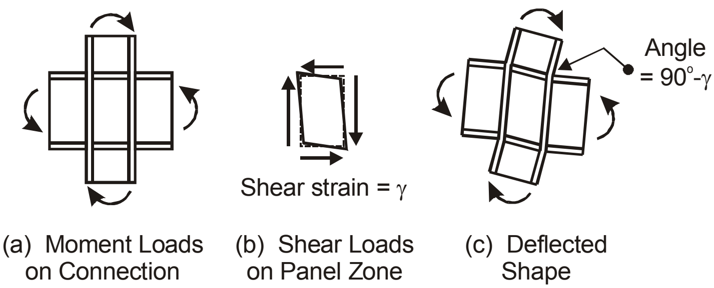



Figure 8.1 Distortion in Connection Panel Zone

As shown in the figure, when the connection transfers bending moment from the beams to the columns, the connection panel zone is subjected to shear stresses. These stresses are much larger than the shear stresses in the adjacent beams and columns. Because of the shear distortion in the panel zone, the beam and column cross sections rotate by different amounts, and the connection is not rigid. Also, the strength of the panel zone may be smaller than the strengths of the adjacent beams and columns, and the panel zone may yield first.

In a steel connection with thin column flanges, almost all of the panel zone stiffness and strength comes from the web part of the panel zone. For a steel connection with thicker column flanges the behavior is more complex, because the flanges resist bending. For a beam-to-column connection in a concrete frame the behavior is even more complex, because panel zone deformation is caused by diagonal cracking and bond slip rather than simple shear strain.

The model for a panel zone component is shown in Figure 8.2.

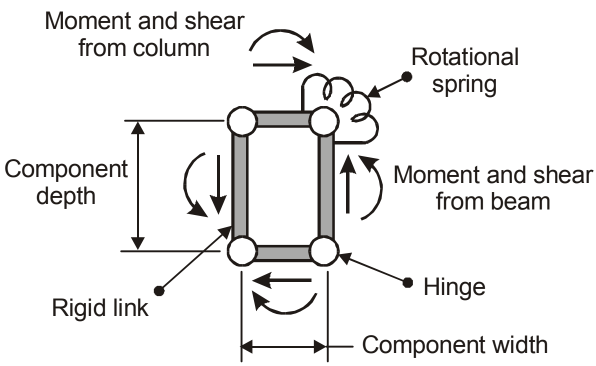



Figure 8.2 Model for Panel Zone Component

This is the Krawinkler model. It consists of four rigid links, hinged at the corners. The moments and shears from the columns and beams act on the rigid links. A rotational spring provides the connection strength and stiffness. This spring has a linear or nonlinear moment-rotation relationship.

For a steel connection you can specify the column and beam sizes, in which case PERFORM calculates some of the component properties, using the Krawinkler formulas as follows.

Initial stiffness of rotational spring = $0.95 d_b d_c t_p G$

Strength at Y point = $0.55 F_y t_p 0.95 d_b d_c$

Hardening stiffness = $1.04 b_{fc} t_{fc}^2 G$

Deformation at U point = 4 (Deformation at Y point)

where $d_b$ = beam depth, $d_c$ = column depth, $t_p$ = panel zone thickness, $b_{fc}$ = column flange width, $t_{fc}$ = column flange thickness, $G$ = shear modulus and $F_y$ = yield stress.

## 8.2 Panel Zone Elements

### 8.2.1 Connection of Nodes to Panel Zone Component

Each panel zone element consists of one panel zone component and has only one node (the node at the intersection of the beam and column axes). If you do not specify a panel zone element at a node, the beam-to-column connection is rigid. If you specify a panel zone element at a node, PERFORM sets up the model for the panel zone component.

The beam and column elements extend to the node, and do not stop at the edges of the panel zone. The ends of the beam and column elements are connected to the rigid bars of the panel zone component by rigid links. A model with beam and column elements is thus as shown in Figure 8.3.

As shown in Figure 8.3, the beam axis is not necessarily at mid-height of the panel zone. This is determined by the end zone lengths for the column elements.

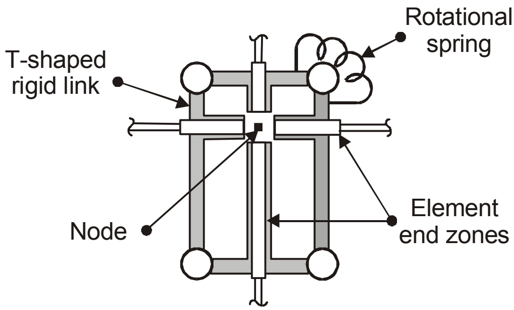



Figure 8.3 Model at a Panel Zone Node

### 8.2.2 Element Orientation

In a three-dimensional connection it is important to orient the panel zone in the correct plane. It is also important to connect the beams and columns to the correct panel zone edges (i.e., right side beam to right edge of panel zone, top column to top edge of panel zone, etc.).

A panel zone element has axes as shown in Figure 8.4. Axis 1 is normal to the plane of the panel zone, Axis 2 is along the column, and Axis 3 is along the beam. These axes define the left (L), right (R), top (T) and bottom (B) edges of the element. Axis 2 is usually vertical and Axis 3 is usually horizontal, in which case Axis 1 is also horizontal. However, in special cases these axes can be inclined.

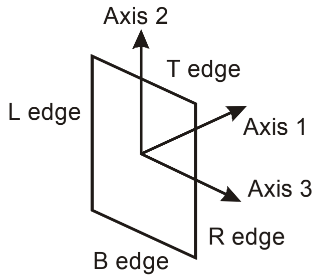



Figure 8.4 Panel Zone Axes and Edges

For each panel zone element you must specify an orientation, using the **Elements** task and the **Orientations** tab. The form has diagrams that illustrate the procedure.

Usually the "standard" orientation option will apply. With this option the panel zone orientation is determined by the column orientation. The column must be vertical, so that Axis 2 of the panel zone is also vertical. Axes 1 and 3 of the panel zone are therefore horizontal. In most cases Axis 3 of the panel zone is parallel to local Axis 3 of the column. The plane of the panel zone is thus the 1-3 plane of the column. For a steel I-section column, this is the plane of the web, and hence the panel zone is parallel to the column web. However, if you wish you can specify that Axis 3 of the panel zone is parallel to Axis 2 of the column. You may need to do this if you are using a panel zone element to model connection deformations in a reinforced concrete frame, and you want the panel zone to be in the 1-2 plane of the column rather than the 1-3 plane.

With the standard orientation option, if you change the orientation of the column element, the orientation of the panel zone is automatically changed. The panel zone must be vertical, but it can be inclined in plan (i.e., it is not necessarily parallel to the H1 or H2 axis).

If the standard orientation option does not apply, you must use the "nonstandard" option. In this case you must specify the directions of Axes 1, 2 and 3 explicitly. You must specify the plan angle from the global H1 axis to the element Axis 1, the tilt angle of Axis 1 from the horizontal plane, and the twist angle about Axis 1.

### 8.2.3 Beam and Column Connection to Panel Zone

As already noted, it is important to connect the beams and columns to the correct edges of the panel zone element (i.e., right side beam to right side of panel zone, top column to top side of panel zone, etc.). Figure 8.4 shows the left (L), right (R), top (T) and bottom (B) edges of the element.

If a beam connects to a node that has a panel zone element, the end of the beam must connect to either the L or R edge of the panel zone element. PERFORM determines this automatically for all beams that connect to panel zones, by comparing the direction of the beam axis with the direction of Axis 3 for the panel zone element. If End I of a beam element connects to a panel zone, and if the beam axis is essentially parallel to Axis 3 and in the same direction, then End I of the beam connects to edge R of the panel zone. If the beam axis is essentially parallel to Axis 3 but in the opposite direction, then End I of the beam connects to edge L of the panel zone. The opposite applies if End J of the beam connects to a panel zone. The angle between the beam axis and Axis 3 of the panel zone must not be larger than 20 degrees.

The panel zone edges for Ends I and J of a beam element are properties of the beam element. You can check this in the ECHO file. If a beam element connects to nodes that have panel zone elements, the panel zone edges for Ends I and J are shown in the echo of the beam element data (L = left edge, R = right edge, X = no connection, because the beam is not parallel to Axis 3 of the panel zone).

Similarly, if End I of a column connects to a node that has a panel zone element, and if the column axis is essentially parallel to Axis 2 of the panel zone, then End I of the column connects to edge T of the panel zone. If the column axis is essentially parallel to Axis 2 but in the opposite direction, then End I of the column connects to edge B of the panel zone. The opposite applies for End J.

A panel zone element must connect to beam or column elements on at least one of the L and R edges and at least one of the T and B edges. If this is not the case, PERFORM displays an error message. This is done when the structure data is checked, just before an analysis is carried out. The L, R, T and B connection assignments are also made at that time.

### 8.2.4 Number of Panel Zone Elements in a Connection

In the present version of PERFORM you can have only one panel zone element at any node.

## *8.3 P-Δ Effects and Element Loads*

Panel zone elements do not exert P-Δ effects. PERFORM does not include any element loads.

# 9 Shear Wall Element

*This chapter reviews the components and elements that can be used to model relatively slender shear walls, and provides guidance on their use.*

*It is not a simple task to model inelastic behavior in shear walls, and there are not many guidelines that you can follow. It is important that you understand the limitations of the models.*

## 9.1 Components and Elements

For each shear wall element you must assign a shear wall compound component. For each shear wall compound component you must specify shear properties and axial-bending properties. The steps are essentially as follows.

(1) Shear properties.

- (a) Define a shear material. This can be elastic or inelastic.
- (b) When you define the Shear Wall Compound Component, specify the shear material and a wall thickness.

(2) Axial-bending properties.

- (a) Define a fiber cross section for a shear wall. This can be an elastic section or an inelastic section.
- (b) When you define the Shear Wall Compound Component, specify the cross section.

You must also input data for the transverse stiffness of the wall (usually the horizontal direction) and for the out-of-plane bending stiffness. Both of these are assumed to be elastic.

If you specify inelastic shear you can set up limit states using the strain capacities of the shear material. If you specify elastic shear you can set up limit states using the shear strength of the shear material. You can also set up shear strength limit states for a cross section through a shear wall, where the cross section can cut through several elements.

If you specify elastic bending you can set up limit states using the stress capacities of the elastic material(s) in the section, at locations that you specify for "monitored" fibers. If you specify inelastic bending you can set up limit states using the strain capacities of the inelastic material(s) in the section, again at locations that you specify for "monitored" fibers.

You can also overlay shear wall elements with deformation gage elements, and hence set up limit states based on strain, rotation and/or shear deformation.

## *9.2 Elements*

### 9.2.1 Element Shape and Axes

A shear wall element does not have to be rectangular, but it must not be highly distorted. Each element must have clear longitudinal and transverse directions, where element Axis 2 is longitudinal and element Axis 3 is transverse. Usually Axis 2 will be vertical and Axis 3 horizontal, although this is not essential. Axis 1 is normal to the plane of the element.

Figure 9.1 shows some possible shapes for a shear wall element.

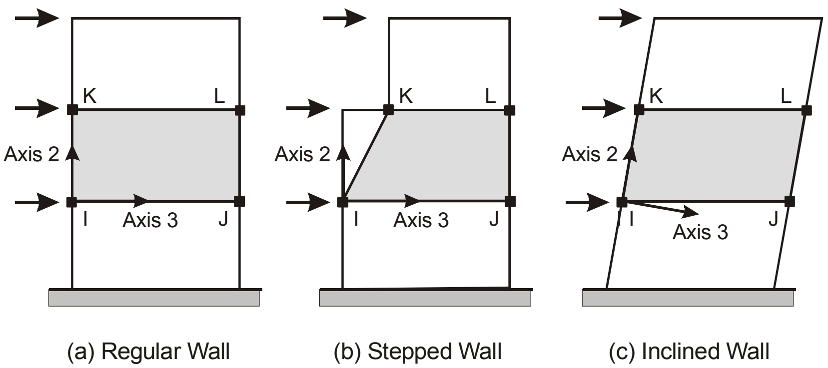
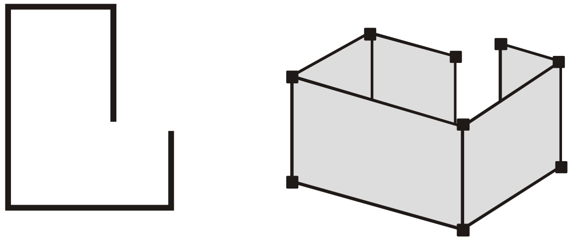



Figure 9.1 Shear Wall Elements

In each of these cases the shear wall element extends over a full story height. You can use more than one element in a story if you wish.

Figure 9.2 shows how shear wall elements might be used to model a three-dimensional wall. Shear Wall elements are intended mainly for modeling walls that are solid or essentially solid, or are "coupled" walls with regular openings. If you have a wall with irregular openings, it may be better to use General Wall elements.



(a) Plan (b) Elements

Figure 9.2 3D Wall

### 9.2.2 Element Properties and Behavior

The following are some key points.

(1) Each element connects 4 nodes and has 24 degrees of freedom.

(2) Longitudinal (usually vertical) in-plane behavior is more important than transverse (usually horizontal) behavior. In the longitudinal direction the element can be inelastic in bending and/or shear. Transverse in-plane behavior is secondary, and is assumed to be elastic. Out-of-plane bending is also secondary, and is assumed to be elastic.

(3) For the purposes of calculating the element stiffness, the cross section depth is assumed to be constant along the element length, based on the element width at its mid-height. When you define a cross section for a shear wall, you can choose "Fixed Size" or "Auto Size" for the cross section dimensions. If you choose "Fixed Size" you must make sure that the cross section depth corresponds

to the element width at it mid-height. If you choose "Auto Size", the cross section depth is automatically made equal to the element width at mid-height.

(4) If you specify elastic components for both bending and shear, the element is elastic. If you specify an inelastic shear material and an inelastic fiber cross section, the element is inelastic for both axial/bending effects and shear.

(5) In a fiber cross section, resist the temptation to use a large number of fibers. Given all of the other uncertainties, you can usually model a cross section sufficiently accurately using a fairly small number of steel and concrete fibers.

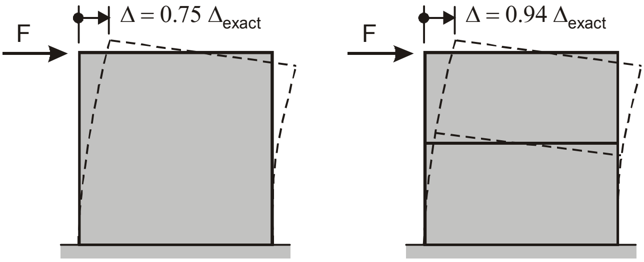
(6) The longitudinal axis of the element is at the midpoint of the cross section. For an "Auto Size" section this is also the elastic centroid of the cross section. For a "Fixed Size" section the midpoint of the section is not necessarily the elastic centroid.

As the fibers yield and/or crack in an inelastic fiber section, the effective centroidal axis shifts. However, the axial extension of an element is always calculated at the midpoint of the cross section (i.e., the longitudinal axis of the element does not shift).

(7) Axial strain, shear strain and curvature are assumed to be constant along the element length. Hence, a shear wall element is a lower order element than a typical beam element, where the curvature varies linearly. Figure 9.3 shows a consequence of this.

If you use a single element to model a one-story wall, the calculated elastic bending deflection is only 75% of the "exact" deflection from beam theory. The total deflection, consisting of bending plus shear, is more accurate. The total deflection is also more accurate for a story in a multistory wall with several elements. If you wish you can use two or more elements in a story. For two elements the calculated bending deflection is 94% of the "exact" deflection.

This is a concern only for wall with one or two stories. For taller walls it is sufficiently accurate to use one element per story. You may, however, have to use shorter elements in regions where the wall forms a hinge (see Section 9.4).




(a) Single Element (b) Two Elements

Figure 9.3 Error in Bending Stiffness

### 9.2.3 Sign Convention

The sign convention for axial force, shear force and bending moment is shown in Figure 9.4. Axial force is tension positive. Shear force is positive along Axis 3 at the IJ end. Bending moment is positive for compression on the +3 side.

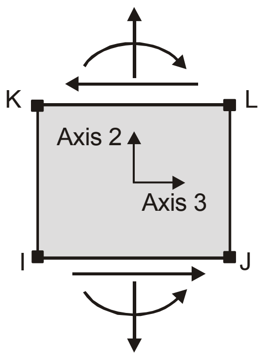



Figure 9.4 Sign Convention

### 9.2.4 Axial Extension Caused by Bending

A key aspect of the behavior for reinforced concrete shear walls is that as the concrete cracks in tension the neutral axis can shift towards the compression side. Hence there is interaction between axial and bending effects, and there is axial extension of the wall as it bends. This could have significant effects on adjacent parts of the structure.

If you specify elastic axial-bending behavior, the neutral axis does not shift and the effect of concrete cracking is not modeled. This may be a reasonable assumption for a steel shear wall or for a concrete wall that has sufficient axial force to prevent concrete cracking. However, if there is yield or cracking, shifting of the neutral axis can have a significant effect on the behavior.

### 9.2.5 Connecting a Beam to a Shear Wall

A shear wall element has no in-plane rotational stiffness at its nodes. Hence, if you connect a beam element to a shear wall, the connection is pinned. To specify a moment-resisting connections between a beam and a wall, you must imbed a beam element in the wall, as shown in Figure 9.5.

If you make the imbedded element very stiff in bending, the beam will be rigidly connected to the wall. This may not be accurate, since there may be local distortions where the beam connects to the wall, making the connection less than rigid. One way that you can account for this is to model the imbedded beam using a beam compound component as shown in Figure 9.5(c). Make the body of the beam very stiff in bending and choose the stiffness of the moment release component to model the stiffness of the beam-to-wall connection. If you wish, you can use an inelastic moment connection component rather than an elastic release component, so that the beam-to-wall connection is nonlinear.

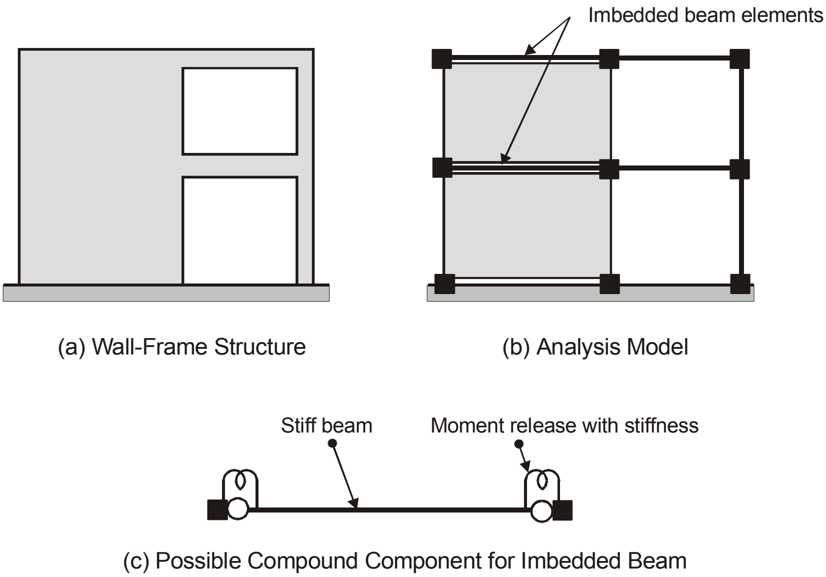



Figure 9.5 Beams Connecting to a Shear Wall

## *9.3 Limit States*

### 9.3.1 Strain Limit States

If you use an inelastic fiber cross section and/or inelastic shear material for a shear wall element, you can define strain capacities and hence strain (deformation) limit states.

It can be useful to use strain limit states, because they can show the most highly deformed elements in a color coded deflected shape. For performance assessment, however, it may be better to use deformation gages (see the next section).

When you define a fiber shear wall cross section, you must specify the "structural" fibers that define the structural behavior of the cross section. You can also specify "monitored" fibers. A monitored fiber is like a strain gage. At each step of an analysis, PERFORM calculates the strain at each monitored fiber. If you specify limit states that include strains in shear wall elements, PERFORM uses the strains to calculate demand-capacity ratios.

For each monitored fiber you must specify a material. The strain capacities for a fiber are the strain capacities for the corresponding material (a property of the material component). You will usually specify monitored fibers only at the extreme points of a cross section, where the strains have maximum values.

### 9.3.2 Deformation Gages

You can use Deformation Gage elements to monitor fiber strains, hinge rotations and shear deformations.

If strain is used as a demand-capacity measure, it may be better to use a strain gage than to use monitored fibers. The reason is that a strain gage can extend over several wall elements, and hence calculates the average strain over those elements. Monitored fibers give the strains in single elements. If there is localized strain concentration, it may be unnecessarily conservative to use single element strains.

Rotation and shear gages can also be used. These gages also have the advantage that they can extend over several elements, and hence calculate average rather than localized deformations. Also, FEMA 356 gives capacity values for rotations and shear deformations in walls.

### 9.3.3 Strength Limit States

If you use an elastic fiber cross section and/or elastic shear material for a shear wall element, you can define stress capacities and hence strength limit states. When you define an elastic fiber cross section you can specify monitored fibers. At each step of an analysis, PERFORM calculates the stress at each monitored fiber.

Because there can be stress concentrations in shear, it is usually better to consider the shear strength for a cross section through a wall, where the section cuts several elements, rather than the shear strength in individual elements. You can do this with the **Structure Sections** task, by specifying shear strengths for structure sections, and corresponding strength limit states.

## 9.4 Element Length in Hinge Region

### 9.4.1 Sensitivity of Calculated Strain

Consider a shear wall that is designed to hinge at its base, and to remain essentially elastic over the rest of it height. In this case it may be reasonable to use a shear wall element with an inelastic fiber section to model the wall in the hinge region, and elements with elastic fiber sections for the rest of the wall. Figure 9.6 shows two possible models for such a wall.

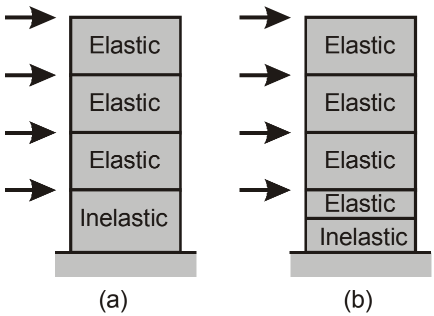



Figure 9.6 Wall With Hinge Region.

In Figure 9.6(a) a single nonlinear element is used in the bottom story. This model assumes that the hinge length is the full story height. In Figure 9.6(b) two elements are used in the bottom story, with a shorter inelastic element than in Figure 9.6(a). This model assumes that the hinge length is less than the story height.

If these two models are analyzed, the calculated strength for the model in Figure 9.6(a) will be larger than for that in Figure 9.6(b), because the lever arms between the loads and the midpoint of the inelastic element are shorter. For example, if each lateral load is H, the story heights are all h, and the inelastic element is one half of the story height, the calculated strength is H = M/8h for Figure 9.6(a) and H = M/9h for Figure 9.6(b), where M is the bending moment capacity of the inelastic cross section. This is a 12.5% difference.

The calculated strain demands are also different for the two models. The bending deformations in the wall are partly elastic and partly plastic. For the plastic part of the deflection there is a hinge at the midpoint of the inelastic element. For any given deflection at the roof, the hinge rotations are approximately the same for the two models. The shear wall element assumes constant curvature over the element height. Hence, the calculated strain depends on the hinge rotation divided by the element height. Hence, the calculated strain for Figure 9.6(a) is smaller than for Figure 9.6(b). If the inelastic element in Figure 9.6(b) is one half of the story height, the difference in calculated strains can approach 100%.

The calculated strain is thus sensitive to the assumed hinge length, and is much more sensitive than the calculated strength. For a given strain capacity, the calculated roof displacement capacity for the model in Figure 9.6(a) can approach twice the capacity for the model in Figure 9.6(b). It is important, therefore, that the "correct" hinge length be used.

### 9.4.2 Element Length

Paulay and Priestley ("Seismic Design of Reinforced Concrete and Masonry Buildings", Wiley, 1992) suggest the following hinge length for a wall.

$$L_p = 0.2D_w + 0.044h_e \tag{9.1}$$

where $L_p$ = hinge length, $D_w$ = depth of wall cross section and $h_e$ = effective wall height = height of a cantilever wall with a single load at the top and the same moment and shear at the hinge as in the actual wall. A larger shear (i.e., a larger bending moment gradient) gives a smaller hinge length.

FEMA 356, page 6-48, recommends a hinge length equal to the smaller of (a) one half the cross section depth and (b) the story height.

## *9.5 Element Loads*

PERFORM does not allow element loads for shear wall elements. However, wall weight can be considered using self weight loads.

## 9.6 Geometric Nonlinearity

You can include or ignore P-Δ effects. You can not consider true large displacement effects.

P-Δ effects are considered for both in-plane and out-of-plane (plate bending) effects. If you specify P-Δ effects, be careful not to specify a very small thickness for plate bending. You may want to ignore the shear forces resisted by plate bending, and consider only in-plane shear. You may be tempted to do this by specifying a small bending thickness. However, if you specify that P-Δ effects are to be considered, a small thickness means that the wall is likely to buckle.

Specify a realistic thickness. The amount of shear force resisted by plate bending is likely to be a very small part of the total shear force.

# 10 General Wall Element

*The General Wall element is intended for the analysis of complex reinforced concrete walls with irregular openings. This chapter provides a brief overview of wall behavior, and describes the features and behavior of the general wall element.*

*This chapter is in three main parts. Sections 10.1 through 10.7 describe the components and element. Sections 10.8 through 10.16 suggest methods for assigning bending and shear properties. Sections 10.17 through 10.19 suggest how to set up limit states to assess performance.*

*This chapter is concerned with use of the general wall element to assess the performance of complex wall structures. It does not address design issues. For design issues an excellent reference is* Seismic Design of Reinforced Concrete and Masonry Buildings *by T. Paulay and M.J.N. Priestley (Wiley, 1992).*

## *10.1 Wall Behavior*

### 10.1.1 Distinct Parts in a Wall

This section considers plane walls. Three-dimensional walls are considered in a later section.

Figure 10.1 shows a simplified wall structure, and identifies four distinct parts.

The left part of the wall is essentially a vertical cantilever. Desirable inelastic behavior is yielding of the vertical steel. Undesirable behavior is shear yield or vertical crushing of the concrete. In a squat wall, it is unlikely that vertical cantilevers will be slender, with well defined plastic hinges. Instead, inelastic bending behavior is likely to extend over much of the cantilever height. As a cantilever bends and the concrete cracks, the neutral axis of the cross section shifts towards the compression side. A possible effect of this shift is unsymmetrical behavior as shown as shown in Figure 10.2. The behavior under lateral load from the left could be substantially different from the behavior under load from the right.

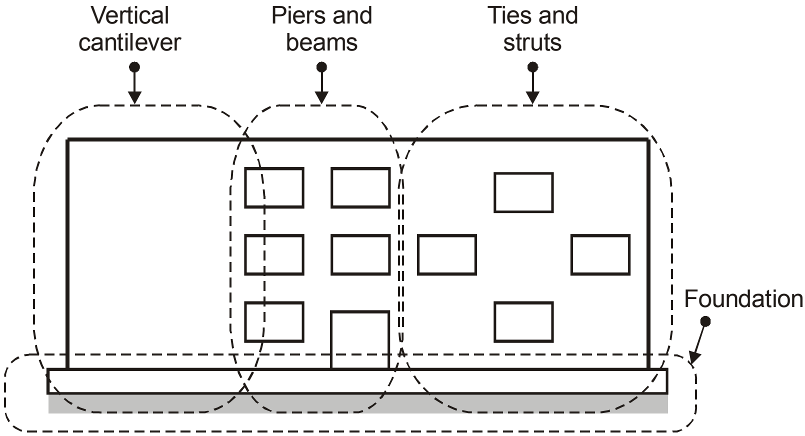



Figure 10.1 Wall Structure With Four Distinct Parts

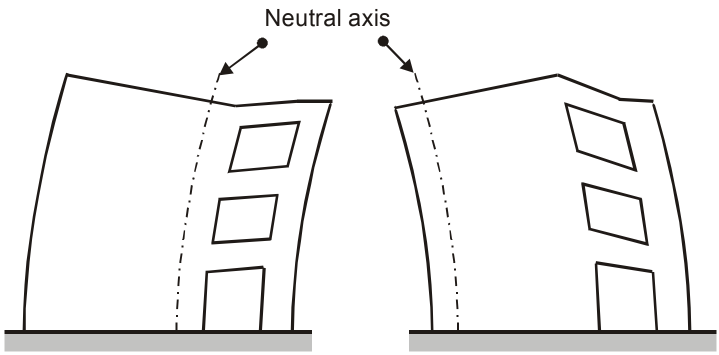



Figure 10.2 Unsymmetrical Bending Behavior

The central part of the wall in Figure 10.1 has well-defined vertical and horizontal wall segments, approximating a frame. In this chapter the vertical segments are termed "piers" and the horizontal segments are termed "beams". Piers are essentially short columns, and beams are essentially short beams. Desirable inelastic behavior is yield of the longitudinal steel. Undesirable behavior is shear yield, severe diagonal cracking, or concrete crushing. Short piers and beams may transfer load by diagonal compression action, rather than by bending and shear.  This part of the wall also has connection regions between the piers and beams. These are similar to connection panel zones in frames, but because they tend to be larger, they are not as heavily stressed.

The right part of the wall has staggered openings. This part can carry load by strut-and-tie action, for example as shown in Figure 10.3. Desirable inelastic behavior is yielding of the vertical ties. Undesirable behavior is yielding of the horizontal ties and crushing of the struts.

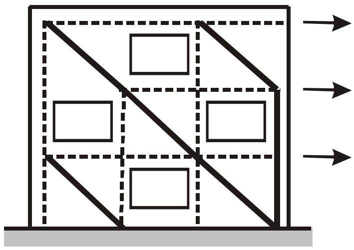


Figure 10.3 Strut and Tie Action

The wall must be tied into a foundation. Desirable behavior is for the foundation to be stiff and remain essentially elastic. Undesirable behavior is inelastic bending or shear deformation of the foundation, or sliding shear at the wall-foundation interface.

### 10.1.2 Modeling and Analysis Goals

There can be complex interactions among the components of a wall, involving vertical and horizontal bending, shear deformation, diagonal compression, and the deformations of concrete struts and steel ties. There can also be complex interactions between the wall and its foundation. An analysis model must capture the essential stiffnesses and strengths for the different types of behavior. Some of the requirements for the results of an analysis are as follows.

(1) For static push-over analysis the overall strength should be calculated correctly. The stiffnesses along the curve should be essentially accurate, since they affect the calculated period of vibration, and hence can affect the calculated base shear and drift demands.

(2) If a dynamic analysis is carried out, the cyclic behavior and energy dissipation should also be essentially correct.

(3) Meaningful deformation demand-capacity values and usage ratios should be calculated, for assessing performance.

(4) The demand-capacity values and deflected shape should show any concentrations of damage, for example large shear deformations in piers or beams.

Since modeling a wall can be a complex task, it is important to keep in mind that the goal is to get results that can be useful for design, not to get an exact simulation of the behavior. An exact simulation is probably not an achievable goal. However, an inelastic analysis that captures the essential aspects of behavior can be valuable for design purposes, even if it is approximate.

## *10.2 Main Features of General Wall Element*

### 10.2.1 Deformation Modes and Sign Convention

Each element has four nodes and 24 nodal displacements. Of the 24 displacements, eight are associated with in-plane deformations of the element, as shown in Figure 10.4(a). These are the most important deformation modes. There are also out-of-plane bending deformations, but these are of secondary importance.

The eight in-plane displacements correspond to five deformation modes and three rigid body modes, as shown in Figures 10.4(b) through 10.4(i). These figures also define positive deformations.

Note that the element has constant bending deformations (i.e., constant curvatures). Essentially this means that the curvatures are based on the bending moment at the middle of the element. A typical beam element allows a linear variation of curvature. Hence, the General Wall element is of lower order than a typical beam element.

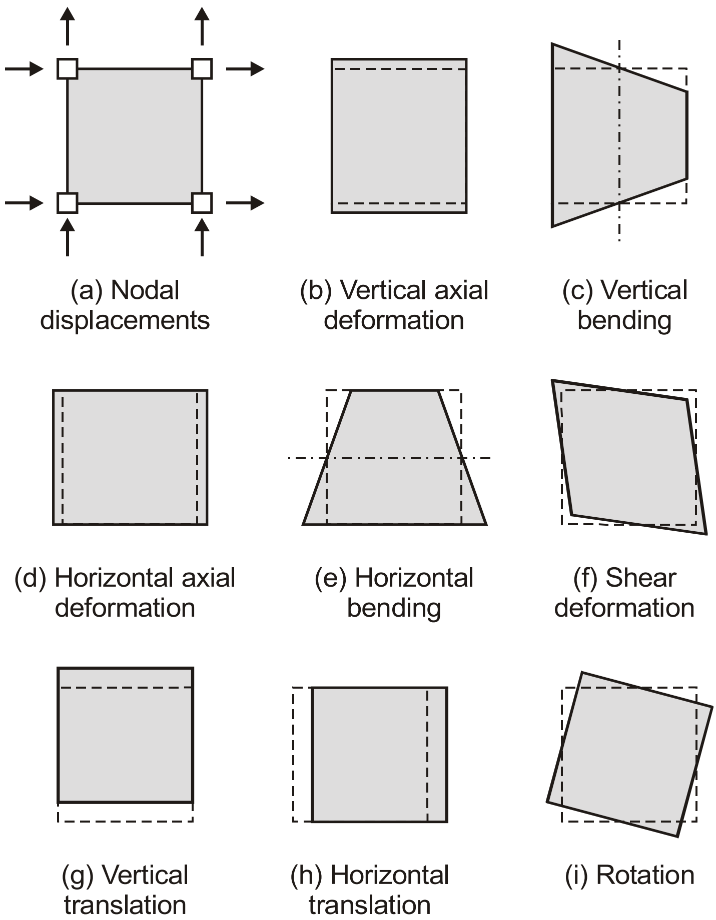


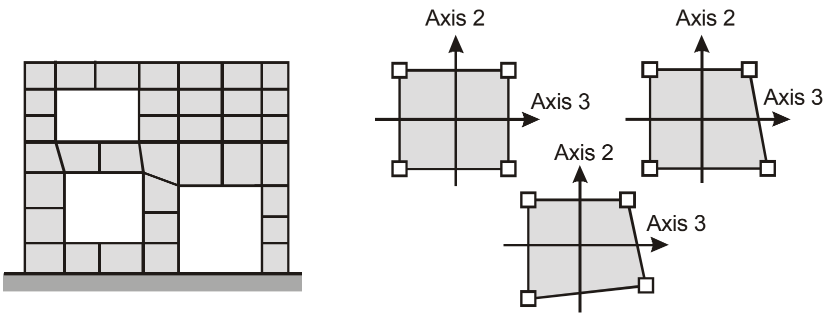

Figure 10.4 Element In-Plane Deformations

## 10.3 Element Axes and Shape

The general wall element is an "engineering" element intended for the specific purpose of modeling walls. It is not a general purpose finite element.

Figure 10.5(a) shows the type of element mesh that might be used for a wall (for a real analysis you might need a finer mesh). Where possible the elements should be rectangular. Some elements can be distorted to account for irregularities in the wall, but the amount of distortion must be small.

Figure 10.5(b) shows some possible element shapes. Each element must have clear "vertical" and "horizontal" directions. These directions will usually be vertical and horizontal, but this is not required. Element Axis 2 is along the vertical direction and element Axis 3 is along the horizontal direction. Axis 1 is normal to the plane of the element.




(a) Element Mesh (b) Element Shapes and Axes

Figure 10.5 Element Shapes

## 10.4 Bending, Shear and Diagonal Layers

### 10.4.1 Layers

To model bending, shear and diagonal compression behavior, an element consists of five layers, acting in parallel. These layers are shown in Figure 10.6.

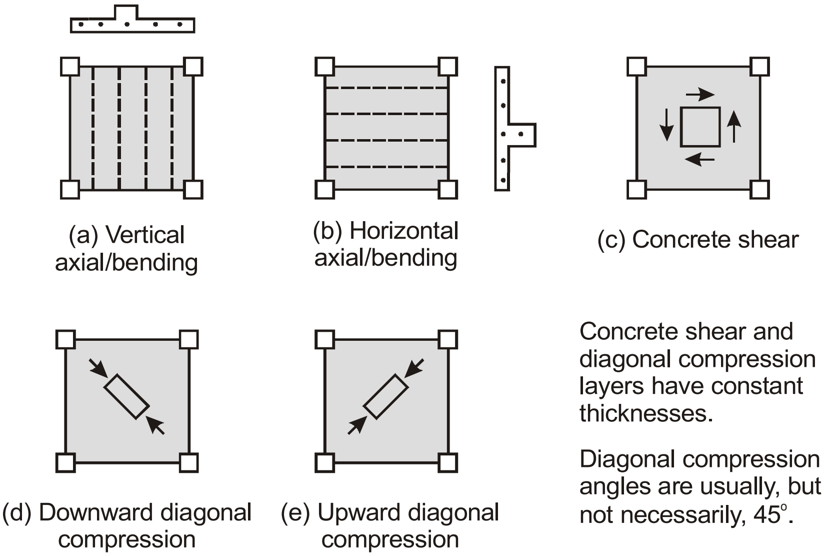



Figure 10.6 Parallel Layers

The layers are as follows.

(1) Axial-bending layer for the vertical axis, as shown in Figure 10.6(a). The cross section is a fiber section with steel and concrete fibers. This allows the neutral axis to shift as the concrete cracks. Note that the wall thickness is not necessarily constant.

(2) Axial-bending layer for the horizontal axis, as shown in Figure 10.6(b). This is also a fiber section.

(3) Conventional shear layer, as shown in Figure 10.6(c). This assumes constant shear stress and a uniform wall thickness. The shear properties for this layer are based on the contribution of the concrete to the shear strength, and it will be termed the "concrete shear" layer.

(4) Diagonal compression layer for downwards diagonal, as shown in Figure 10.6(d). This assumes constant diagonal compression stress and a uniform wall thickness. The slope of the diagonal will usually, but not necessarily, be 45 degrees. Through interaction with the axial-bending layers, this layer transmits shear and

accounts for the contribution of the reinforcing steel to the shear strength. The mechanism is explained later. This is a "diagonal shear" layer.

(5) Diagonal compression layer for upwards diagonal, as shown in Figure 10.6(e).

Each layer has different behavior. The layers interact because they are connected at the nodes. The combined behavior of all layers defines the behavior of the element.

## 10.5 Purpose of Diagonal Layers

The diagonal layers are intended mainly to model strut and tie action, as in Figure 10.3. For analysis of most walls, it is usual practice to ignore the diagonal layers (by specifying zero thickness), and to have only conventional shear.

One reason is that strut and tie behavior is complex, and there is a danger that a model that includes this behavior can over-estimate the shear strength of a wall. Also, FEMA 356 considers only conventional shear.

If you ignore the diagonal layers, the General Wall element is similar to the Shear Wall element, with the major difference that fiber sections are used for both horizontal and vertical cross sections.

**Unless you believe that diagonal strut action is important for your structure, and you are prepared to deal with the added complexity, we suggest that you ignore the diagonal layers.**

## 10.6 Element Behavior

### 10.6.1 Types of Behavior

The General Wall element seeks to model the following types of behavior.

(1) Axial force, bending and shear in vertical cantilevers.
(2) Axial force, bending and shear in piers and beams.
(3) Force transfer through struts and ties.

Vertical cantilevers, piers and beams are similar in that they all involve bending and shear. It is important to recognize that the bending and shear behavior of the components in a reinforced concrete or masonry wall can be substantially different from the behavior of the columns and beams in a slender frame. In this section, the behavior of the General Wall element is considered using a vertical cantilever as an example. Similar behavior will be present in piers and beams.

**If you do not plan to use diagonal compression layers, you can skip large parts of the following sections.**

### 10.6.2 A Major Approximation

In an actual wall the concrete is in a state of multi-axial stress. For example, there could be combined vertical compression, horizontal tension and shear. The inelastic behavior of a material under multi-axial stress is much more complex than its behavior under uniaxial stress. This is especially true for concrete. The General Wall element does not consider multi-axial stress. Instead it separates the various aspects of behavior into layers, with uniaxial stress in each layer. Some consequences of this are as follows.

(1) The axial-bending layers account for vertical and horizontal compression stresses in the concrete, and the diagonal layers account for diagonal compression stresses. In an actual wall these stresses interact directly. For example, instead of crushing vertically under a large vertical stress, the presence of a diagonal stress might cause the concrete to crush along an inclined direction, at a lower vertical stress than if the diagonal stress were not present. This type of effect is not considered in the element. The axial-bending layers interact with the diagonal layers, because they are connected at the element nodes, but this interaction is not the same as the interaction associated with multi-axial stresses.

(2) When concrete is subjected to combined compression and shear, the shear strength is increased, essentially because there is internal friction. The General Wall element does not account for frictional behavior. The shear strength in the concrete shear layer is assumed to be independent of any other stresses.

By separating the different modes of action into separate layers, the General Wall element makes it easier to model the behavior, but it ignores potentially important interactions. An alternative approach would be to model the concrete using a multi-axial inelastic model, which leads to a more general purpose finite element. However, this approach introduces major complications and approximations of its own, because of the complexity of modeling the crushing-cracking-shearing behavior of concrete under cyclically varying multi-axial stresses. PERFORM does not include an element of this type. The General Wall element is not an ideal solution, but it is offered as a reasonable compromise for practical analysis.

### 10.6.3 Concrete Shear vs. Diagonal Shear

The concrete shear and diagonal compression layers both resist shear, but by different mechanisms. As an example consider the cantilever shown in Figure 10.7.

The dimensions are shown in Figure 10.7(a). The height to width ratio is 2:1. This means that it is a fairly slender member for a squat wall. However, it is a useful example to illustrate the behavior.

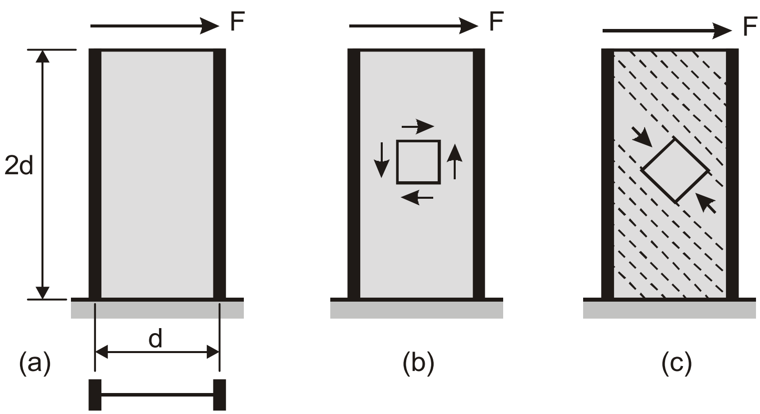



Figure 10.7 Example for Concrete and Diagonal Shear

The cross section consists of two thin flanges with a slender web. Assume that the flanges carry only axial forces, and resist all of the axial force and bending moment, and assume that the web carries all of the shear.

Consider two cases, one with conventional shear in the web and one with 45 degree diagonal compression. These are shown in Figures 10.7(b) and 10.7(c), respectively. Assume uniform shear stress in Figure 10.7(b), and uniform diagonal compression stress in Figure 10.7(c). In both cases the structure is statically determinate, and the forces acting on the web and the flanges can be found using equilibrium equations.

For the case with conventional shear, Figure 10.8 shows the flanges and web separated as free bodies, and shows the forces acting on these free bodies.

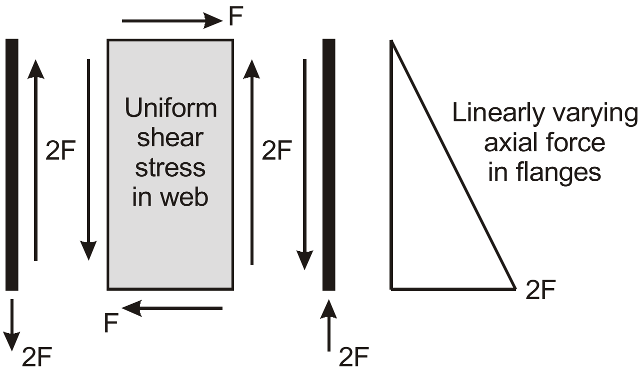



Figure 10.8  Forces for Conventional Shear Case

Figure 10.8 also shows the variation of axial force in the flanges (tension in the left flange, equal compression in the right flange).

For the case with diagonal compression, Figure 10.9 shows the same free bodies and the forces in the flanges.

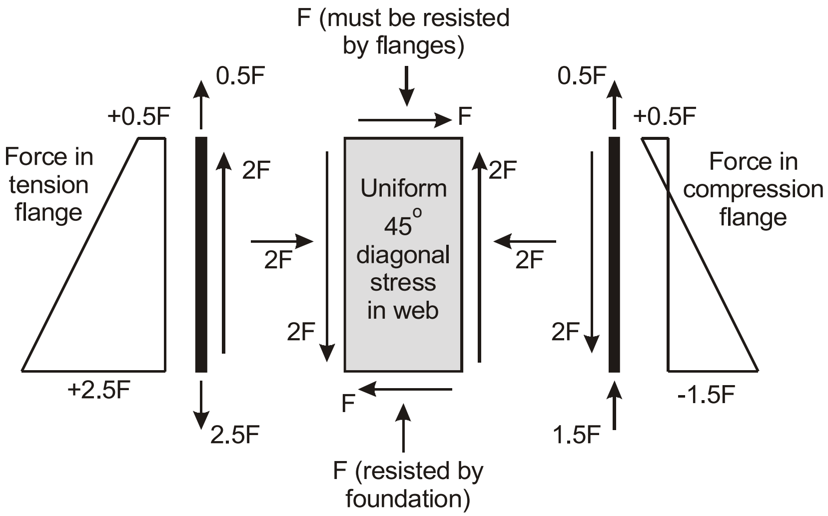



Figure 10.9 Forces for Diagonal Compression Case

This is more complex than the conventional shear case, and requires some explanation. The main cause of the difference is that the web not only has shear stresses but also has vertical and horizontal compression (confining) stresses. Figures 10.10 and 10.11 show why this is the case.

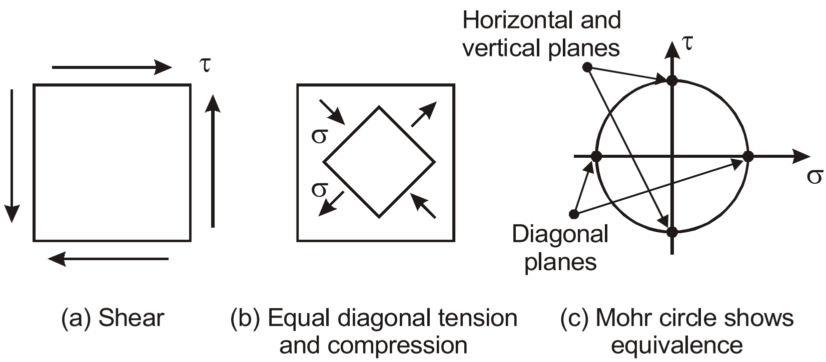



Figure 10.10 Equivalent Stresses for Conventional Shear

Figure 10.10 shows the shear stress and the equivalent diagonal stresses for the conventional shear case. As is well known, shear stress on the horizontal and vertical edges corresponds to equal tension and compression stresses along the diagonals.

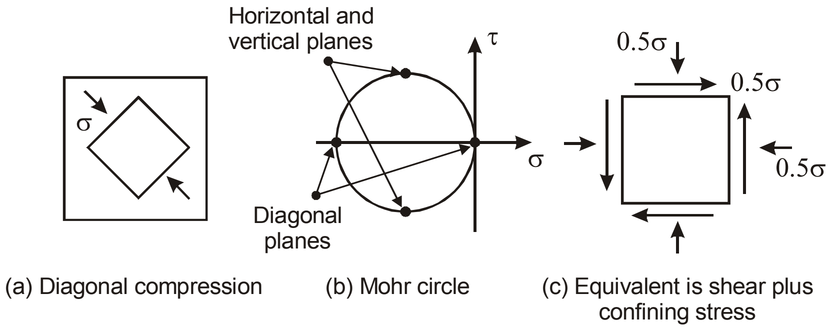



Figure 10.11  Equivalent Stresses for Diagonal Compression

Figure 10.11 shows similar diagrams for the diagonal compression case. In this case, a diagonal compression stress of σ is equivalent to a shear stress of 0.5σ plus horizontal and vertical compression (confining) stresses of 0.5σ.

A key point for the diagonal compression case is that unless the confining stress is present there can be no diagonal compression stress (i.e., if the confining stress, 0.5σ, is zero, the diagonal compression stress, σ, must also be zero). For the cantilever in Figure 10.9, the vertical part of the confining stress can be provided by a tension force of 0.5F in each flange (if we assume that the cross section at the cantilever end is constrained to remain plane). These tension forces explain the difference between the flange forces in Figure 10.9 and those in Figure 10.8. However, the horizontal part of the confining stress can not be provided by the flanges. Instead, in a reinforced concrete web it must be provided by horizontal steel reinforcement (shear stirrups) in the web. In effect, horizontal tension in these stirrups provides horizontal confining compression in the concrete, which is then able to resist diagonal compression stresses.

## 10.6.4 Finite Element Approximations

Figures 10.8 and 10.9 are "exact" solutions. Consider the same cases but with finite element approximations using General Wall elements. Use element properties as follows.

(1) For the vertical axial/bending layer the cross section must have fibers to represent the flanges. For each flange use a steel fiber and a concrete fiber, to represent a reinforced concrete section. The areas of the fibers and the material properties are not important for this discussion.

(2) For the horizontal axial-bending layer use steel fibers at the top and bottom of the cross section. These fibers model the shear stirrups.

(3) For the conventional shear case include a concrete shear layer, but make the thicknesses zero for the diagonal compression layers. For the diagonal compression case do the reverse. The thicknesses and material properties are not important.

For a model with two square elements, the results for the conventional shear case are shown in Figure 10.12.

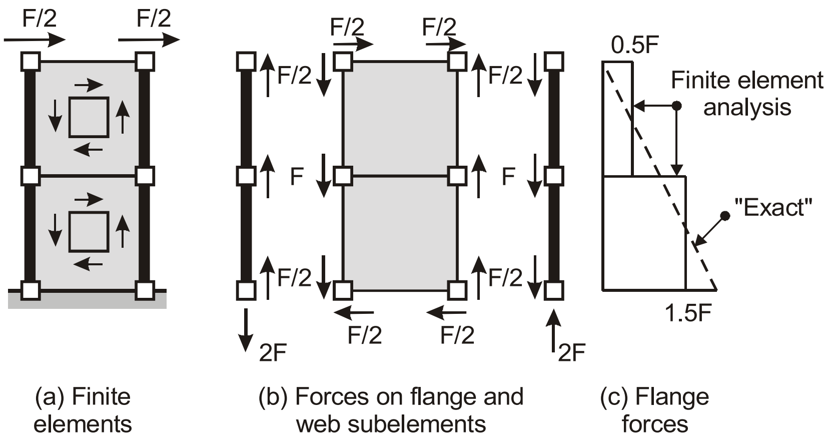



Figure 10.12 Conventional Shear Case With two Elements

The calculated web shear stress is the same as in the exact analysis. However, because the element assumes constant bending, the calculated flange forces are based on the bending moments at the element midpoint, and hence the analysis underestimates the maximum flange force compared with the exact solution. There are no axial forces or bending moments in the horizontal axial/bending layer (conventional shear does not rely on shear stirrups).

For a similar model, the results for the diagonal compression case are shown in Figure 10.13.

In this case the calculated diagonal compression stress is equal to twice the shear stress. This is consistent with Mohr's circle, and with the "exact" analysis. However, it is not necessarily correct in a practical sense because the diagonal compression in an actual cantilever might be larger near the middle of the web than near the corners, whereas the finite element assumes constant diagonal compression stress over the element. This is considered in more detail later.

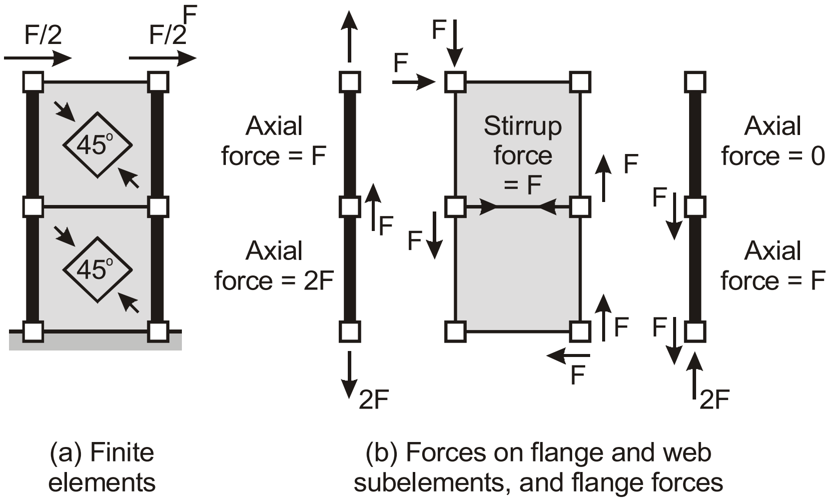



Figure 10.13 Diagonal Compression Case With Two Elements

As before, because the element assumes constant bending, the calculated flange forces are based on the bending moments at the element midpoints, and hence the analysis underestimates the maximum flange force compared with the exact solution.

The shear stirrup at the top of the cantilever has a tension force equal to 0.5F. The stirrups at the center each have tension forces equal to 0.5F, providing a confining force equal to F. The stirrup at the bottom has no extension or force, because of the supports.

Truss models are often used for reinforced concrete with diagonal compression. Figure 10.14 shows the axial forces in a truss structure with the same dimensions and loading as the example cantilever.

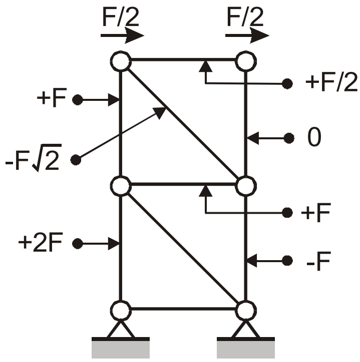



Figure 10.14 Axial Forces in an Equivalent Truss

The vertical and horizontal forces in this truss are the same as in Figure 10.13. The diagonal force is also statically equivalent to the diagonal compression stress in the finite element model. A model based on the General Wall element can thus give forces that are consistent with those from a truss model.

In a finite element analysis, as the mesh is refined the results usually agree more closely with the exact results. Figure 10.15 shows that this is the case for the finite element results when the number of elements in increased to four.

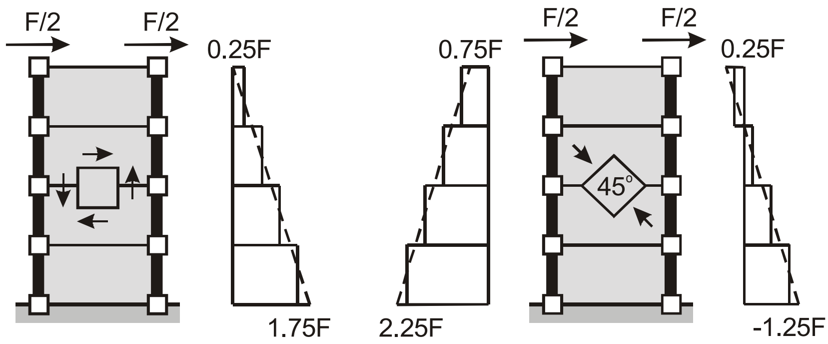



Figure 10.15  Results With Finer Element Meshes

Note that the maximum force in the tension flange is larger for the diagonal compression case than for the conventional shear case. Hence, for inelastic bending the flange will yield sooner in the diagonal compression case, and for the same cross section this case will predict a smaller bending strength.

## 10.6.5 Finite Element vs. Truss Models

The examples in this section have shown that in some cases a finite element model using the General Wall element gives the same forces as a truss model. This does not mean that the two models are equivalent in such cases. A truss model (in general, a strut and tie model) is valuable for considering the flow of forces in reinforced concrete structures, and hence is a valuable design tool. However, a truss model may not capture the stiffness properties of a structure, and it does not lend itself to specifying deformation limit states for assessing ductility demands. For example, before the forces predicted by a truss model can be developed, there may be excessive deformation demands in some parts of a wall. Hence, it may not be possible to develop the full strength predicted by a truss model. Also, since a truss model may not capture stiffnesses accurately, it may not be suitable for push-over or dynamic analyses, where the inertia forces depend on both mass and stiffness. A finite element model is better able to capture not only the overall stiffness but also the relative stiffnesses between different parts of the structure.

### 10.6.6 Shear Deformations

The preceding section shows that a model using the General Wall element can give correct results for simple cases. Up to this point, however, we have considered only forces, not deformations. These are considered in the next few sections. This section considers shear deformations, assuming that bending deformations are relatively small.

Figure 10.16 shows the example cantilever with shear deformations and negligible bending.

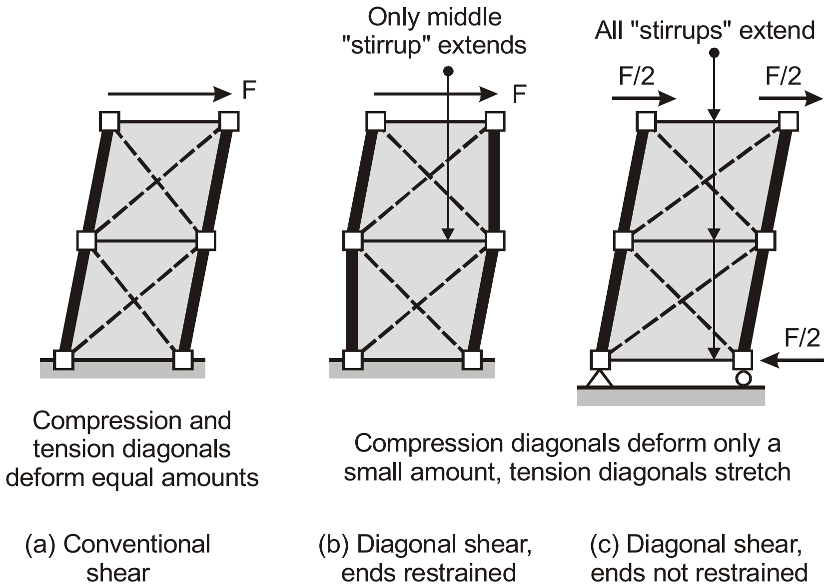



Figure 10.16 Shear Deformations

Figure 10.16(a) shows the case with only conventional shear. This case assumes that the web material has diagonal tension strength as well as compression strength, and that there are equal and opposite strains in the compression and tension diagonals. Horizontal stirrups are not needed, and if such stirrups are present they are do not have tension forces. In the General Wall element, the concrete shear layer provides shear behavior of this type.

Figures 10.16(b) and (c) show the case with only diagonal compression. In this case the tension diagonals crack. There must be horizontal stirrups, and there are tension forces in these stirrups. The stirrups will usually be flexible relative to the compression diagonals, so most of the effective shear deformation is due to stretching of the stirrups rather than compression of the diagonals. Figure 10.16(b) shows the case where the end cross sections are restrained, and only the center stirrup deforms. Figure 10.16(c) shows the case where the end cross sections are not restrained, and all of the stirrups deform. In this case the effective shear deformation is larger.

In the General Wall element, the diagonal compression layers, interacting with the horizontal axial/bending layer, provide shear behavior of this type. Figure 10.16 shows a vertical cantilever. For a member that spans horizontally, the stirrups are vertical, and the forces in these stirrups are caused by interaction between the diagonal compression layer and the vertical axial/bending layer. In general, diagonal compression forces cause tension forces in both the vertical and horizontal axial/bending layers.

### 10.6.7 Diagonal Angles Other Than 45°

In Figure 10.16(b) the diagonal compression is at a 45 degree angle, and as a result the axial force in the center stirrup is equal to the shear force. Hence, the shear strength provided by the shear stirrups is equal to the tension strength of the stirrups in a length equal to the cross section depth. The actual shear strength could, however, be larger than this, if the diagonal compression field were steeper. This is shown in Figure 10.17, for a finite element model and a truss model.

The two models give the same forces. With these forces it is not necessary to have stirrups within the cantilever length. However, the diagonal compression force is increased compared with the 45 degree case (for a given shear force it is 1.58 times larger).

For a typical long beam, it should be reasonable to assume a 45 degree compression field. However, for piers and beams in a squat wall it may be appropriate to consider other angles.

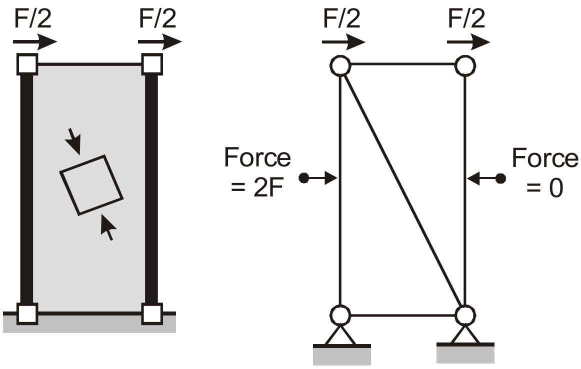



Figure 10.17 Steeper Diagonal Angle

## 10.6.8 Bending

The axial/bending layers account for bending behavior. These layers consist of steel and concrete fibers. The edges of the element are constrained to remain straight, which means that plane sections remain plane within a single element (see Figures 10.4(c) and 10.4(e)). The curvature is constant along the element length, and hence the strain in any fiber is constant.

An important aspect of bending behavior in reinforced concrete is that as the concrete cracks and the steel yields, the neutral axis shifts. Hence, bending and axial effects are coupled, with the following effects.

(1) If the neutral axis shifts, a bending moment causes not only curvature but also axial extension. One consequence of this was shown in Figure 10.2.

(2) If there is an axial compression force on a cross section, the bending strength is increased, because cracking of the concrete is delayed.

(3) If a member in bending is restrained axially (e.g. a beam between two stiff piers), it can develop axial compression forces as it attempts to grow axially. This can increase its bending strength, and hence can also increase the shear strength demand.

The General Wall element uses fiber cross section components to model the axial/bending behavior. A fiber cross section captures the above interaction effects. For example, in the example cantilever, when the concrete in the tension flange cracks, the neutral axis moves close to the compression flange, because that flange is now much stiffer than the tension flange.

### 10.6.9 Interaction Between Axial/Bending Layers

Since the vertical and horizontal axial/bending layers are at right angles to each other, there is no direct interaction between them. Axial and bending deformations in the vertical layer are associated only with vertical displacements at the nodes, and they do not cause any horizontal displacements. Similarly, axial and bending deformations in the horizontal layer are associated only with horizontal displacements at the nodes, and they do not cause any vertical displacements. In effect. Poisson's ratio is zero in each of these layers.

### 10.6.10 Interaction Between Axial/Bending and Concrete Shear Layers

For bending along the vertical axis, the concrete shear layer transmits forces to the vertical axial/bending layer, as shown earlier in Figures 10.9 and 10.12. This models typical beam action, with the limitation that the curvature is constant over the length of each element. For bending along the horizontal axis, the concrete shear layer transmits forces to the horizontal axial/bending layer in a similar way. In general, the concrete shear layer can interact with the axial/bending layers to model bending behavior in both the vertical and horizontal directions. This type of assumption for shear behavior is commonly made in the modeling of beams. Note that the concrete shear layer has shear stiffness only. It does not add any axial or bending strength or stiffness in the vertical and horizontal directions.

### 10.6.11 Interaction Between Axial/Bending and Diagonal Compression Layers

The diagonal compression layers interact with the axial/deformation layers in the following ways.

(1) Diagonal compression forces cause axial tension forces in the axial/bending layers, and hence cause effective shear deformations. This has been already considered, as shown in Figure 10.15. This

should be reasonable for modeling shear deformations in reinforced concrete members.

(2) Axial tension deformations in the axial/bending layers cause diagonal tension, and axial compression deformations in the axial/bending layers cause diagonal compression. This is considered in the next two sections.

### 10.6.12 Effect of Axial Extension on Diagonal Layers

Figure 10.18 shows two ways in which there can be extension of the axial/bending layer. Figure 10.18(a) shows simple vertical extension, and Figure 10.18(b) shows bending along the vertical axis. If the neutral axis shifts, as will usually be the case, bending causes axial extension.

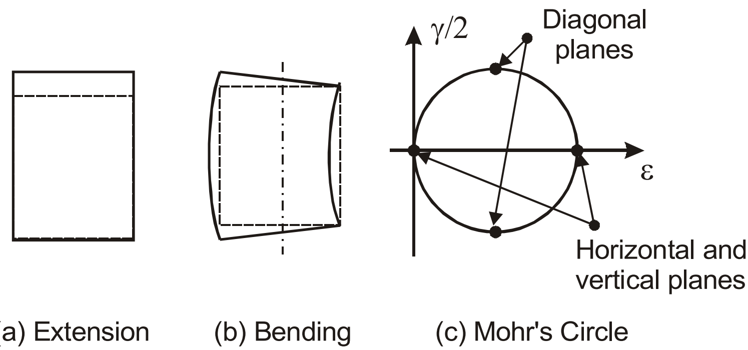



Figure 10.18 Effect of Vertical Extension on Diagonal Strain

Figure 10.18(c) shows Mohr's circle for strain for the case in Figure 10.18(a). For 45 degree diagonals an axial tension strain of ε, causes tension strains of 0.5ε along each diagonal direction. Since diagonal compression materials have no tension strength, the diagonals crack and the diagonal stresses are zero. The situation is the same for axial extension that is caused by bending and shift of the neutral axis. Within a single General Wall element the diagonal strains are constant. For the case in Figure 10.18(b) the diagonal strains depend on the vertical strain at the axis of the axial/bending layer.

Usually, if there is bending there will also be shear, and the situation is more complex. Consider an element that has no concrete shear layer, so that shear is carried only by the diagonal compression layer. If shear is added to Figure 10.18(b), the diagonal compression layers will offer no shear resistance until one of the diagonals goes into compression. This means that if the axial strain is ε, a shear strain equal to ε must be added before there is any shear resistance. Mohr's circle for strain at this point is shown in Figure 10.19.

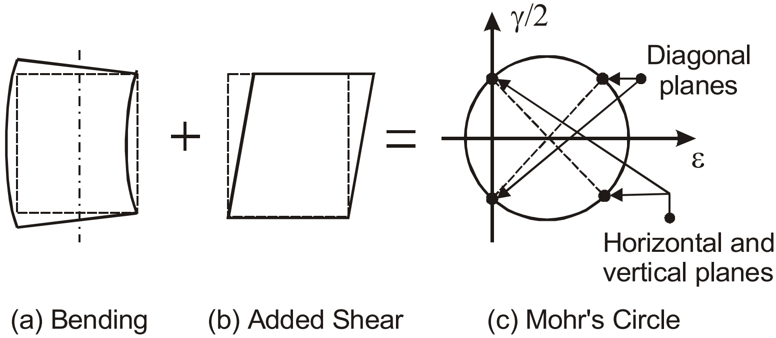



Figure 10.19  Mohr's Circle When Diagonal Gap Closes

The physical significance for reinforced concrete is shown in Figure 10.20.

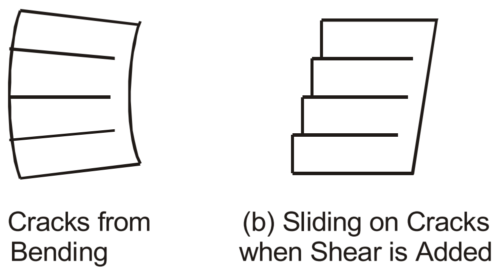



Figure 10.20  Behavior in Reinforced Concrete

As shown in Figure 10.20(a), the bending deformation causes cracking of the concrete. As shown in Figure 10.20(b), when the shear deformation is added there must be shear sliding displacements across the cracks. A real concrete beam would provide resistance to these displacements by a variety of mechanisms. In a General Wall element this type of resistance is modeled by the concrete shear layer. A General Wall with both concrete shear and diagonal compression layers would predict resistance based on the concrete shear layer alone until the gaps close in the diagonal compression layer, then resistance from both layers.

In an actual reinforced concrete member the bending and shear will usually be applied simultaneously, and the concrete will crack at an angle between zero and 45 degrees. In a General Wall there are separate cracks in the axial/bending and diagonal compression layers.

## 10.6.13 Effect of Axial Compression on Diagonal Layers

Next consider interaction between the diagonal compression layers and compression in the axial/bending layers.

First consider an element with no stiffness in the horizontal axial/bending layer, and hence zero horizontal force. Since there is no horizontal force, there is zero confining stress for diagonal compression, and hence the diagonal compression stress must be zero. When the element is deformed axially, the diagonal compression layers deform in such a way that the diagonals have zero strain. Figure 10.21 shows Mohr's circle for strain for this case.

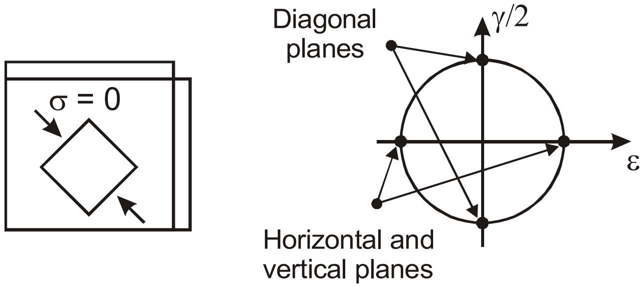



Figure 10.21 No Lateral Restraint : Implied Poisson's Ratio

For 45 degree diagonals this figure shows that the horizontal expansion strain is equal to the vertical compression strain. Physically this is not what we would expect, since it corresponds to a value of 1.0 for Poisson's ratio. A more reasonable ratio of horizontal expansion to vertical compression in plain concrete might be about 0.2. Hence, for this case the General Wall element does not give the expected behavior.

In practice there will never be zero horizontal stiffness, and hence the case in Figure 10.21 is not realistic. To explore this further, consider the other extreme, with rigid confinement in the horizontal direction, and hence zero horizontal strain. Figure 10.22 shows this case.

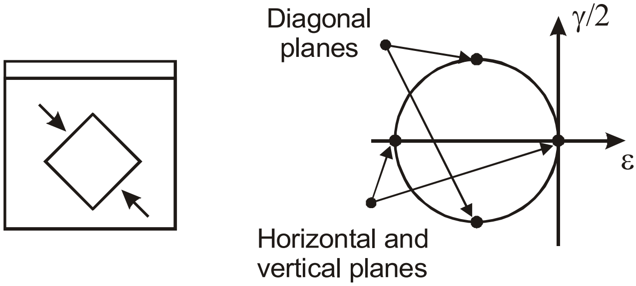



Figure 10.22 Full Lateral Restraint

From Mohr's circle for strain, the 45 degree diagonal strain is one half of the vertical compression strain. Hence, if the material is elastic, if the wall thickness is uniform, and if the same material and thickness are assumed for the vertical axial/bending layer and the two diagonal compression layers, the behavior is as follows.

(1) The horizontal stress is one half of the vertical stress.

(2) The vertical stiffness is 1.5 times larger than for the case with no horizontal stiffness.

Again, this is not the expected result. For a reasonable value of Poisson's ratio, and recognizing that the horizontal confinement is in the plane of the wall only, with essentially no confinement out-of-plane, we would expect a much smaller horizontal stress and a much smaller increase in vertical stiffness.

The reason for this behavior is that we are essentially counting the same concrete three times, once for the vertical axial/bending layer, and once for each diagonal compression layer. This is a weakness of the model for the General Wall element. However, the problem can essentially be avoided by specifying that the diagonal compression layers are inactive for gravity loads. The General Wall element provides this option, and as a general rule we suggest that you choose it. It is similar to assuming that diagonal braces in a braced building frame are inactive for gravity.

### 10.6.14 Material Properties

For the diagonal compression layers we suggest that you specify an elastic-perfectly-plastic Diagonal Compression Material, with a strength equal to the estimated strength of the concrete in the wall, and zero tension strength. Use this material for the diagonal compression layers in General Wall compound components.

Also, we suggest that when you define the diagonal compression layers you make them inactive for gravity loads, to avoid lateral growth when gravity loads are applied.

### 10.6.15 Compression Field Angle

In general we suggest a 45 degree angle for the compression field.

You may decide to use a different angle for piers and beams where you are sure that the shear force can be carried across the member entirely by diagonal compression, with no need for transverse shear reinforcement.

### 10.6.16 Concrete Crushing

Because of interaction between the axial/bending and diagonal compression layers, if there is a large shear force the diagonal compression layer must be confined by the axial/bending layers. This means that the diagonal compression layer pushes on the axial/bending layers. Consider the following case.

(1) Consider a vertical cantilever or a pier. In this case the vertical axial/bending layer resists axial force and bending moment, and the horizontal axial/bending layer models the shear reinforcement.

For a beam, the reverse is the case. Assume that the member has a vertical compression load.

(2) Diagonal stress in the diagonal compression layer causes tension forces on the cross sections in both axial/bending layers. In the horizontal axial/bending layer this puts the shear stirrups in tension, as expected. However, in the vertical axial/bending layer it reduces the vertical compression force on the section, and hence reduces the concrete compression stress. If there is a substantial shear force, and hence a substantial diagonal compression stress, the concrete compression stress in vertical axial/bending layer can be substantially reduced.

(3) In an actual section, the concrete is in both longitudinal and diagonal compression, and it crushes under the combined action. The concrete in the axial/bending and diagonal compression layers are separate, each with uniaxial stress. PERFORM calculates crushing in the concrete only if the concrete stress reaches the yield value in either the diagonal compression layer or in the vertical axial/bending layer. In effect, instead of combining the stresses, the General Wall element separates them, and hence does not capture the true behavior.

This is a weakness of the element for cases where concrete crushing can occur. In a squat wall it is unlikely that concrete crushing will control the performance. However, if it does, the General Wall element may not do a good job of modeling the crushing behavior.

## 10.7 βK Damping for Dynamic Analysis

For βK damping, K is usually the initial elastic stiffness. For a General Wall element, the initial stiffness for the axial/bending layers is calculated assuming that the concrete is not cracked. This uncracked stiffness is usually large relative to the cracked stiffness. Experience shows that if the uncracked stiffness is used for βK, there can be unrealistically large damping forces after cracking occurs, with heavy damping. For this reason, for the General Wall element, the stiffness K for βK damping in the axial/bending layers is based on the full areas of the steel fibers but only 15% of the areas of the concrete fibers. Note that this applies only to the stiffness used for βK damping. The initial stiffness for calculating elastic mode shapes and frequencies is still based on the uncracked section.

The initial stiffness for the diagonal compression layers is also calculated assuming that there is no diagonal cracking. Again, if βK for these layers were based on the initial stiffness, there could be excessive damping after one or both of the layers cracked. Hence, for the General Wall element, it is assumed that βK is zero for the diagonal compression layers (i.e., there is no viscous damping in these layers).

For the concrete shear layer, the usual procedure is used, and βK is based on the initial elastic stiffness of the layer.

If you run dynamic analyses using βK damping, be sure to look at the energy balance, and decide whether there are reasonable values for the relative amounts of energy dissipated through plastic and viscous behavior.

## 10.8 Analysis Model

### 10.8.1 General

This section makes suggestions for assigning properties to General Wall elements.

### 10.8.2 2D and 3D Walls

This section considers finite element meshes for plane walls. If a wall has a box shape in plan, or wide flanges, you should usually model each part of the wall in the same way that you would model a plane wall. A wall can have any shape in plan, including a curved shape.

If a flanged wall has only narrow flanges, you could model the flanges using concrete strut and steel tie elements, rather than wall elements.

### 10.8.3 Element Mesh

A squat wall will typically have relatively light reinforcement in the body of the wall, with additional vertical bars at the wall boundaries and around openings, and additional horizontal bars at the floor levels and around openings. This is shown diagrammatically in Figure 10.23.

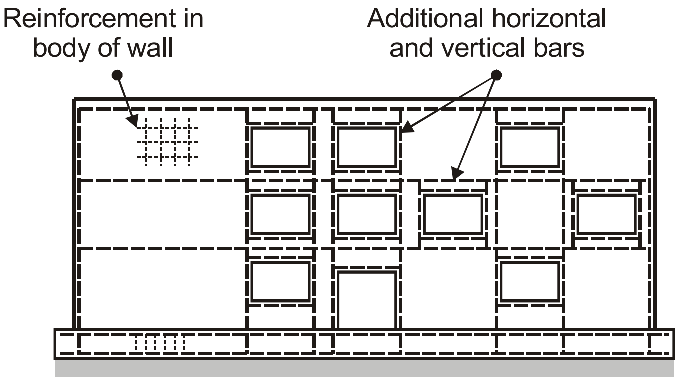



Figure 10.23 Typical Reinforcement

The first step in setting up an analysis model is to lay out an element mesh, as indicated in Figure 10.24.

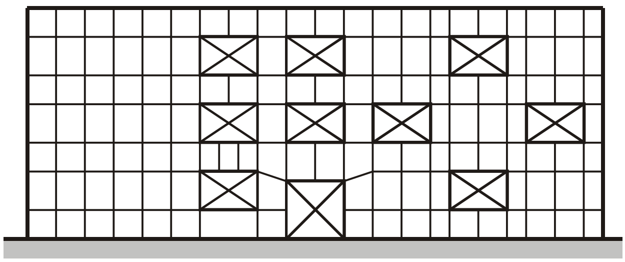


Figure 10.24 Element Mesh

The following are some points to consider.

(1) The elements should be essentially rectangular. Some distortion is allowed, for example as shown above the door opening in Figure 10.24.

(2) In the body of the wall choose vertical subdivisions to make the elements roughly square.

(3) You can change the mesh for piers and beams. For example, in Figure 10.24 the top two beams above the first column of windows have two elements each, whereas the lower beam has three elements.

For the General Wall elements you must specify properties for the axial/bending layers, the concrete shear layer, and the diagonal compression layers. The following sections make suggestions on how to do this.

## 10.8.4 Foundation

You may choose to fix the nodes at the base, assuming that the foundation is rigid. Alternatively you may choose to model the foundation. Figure 10.25 shows two possible models for the foundation.

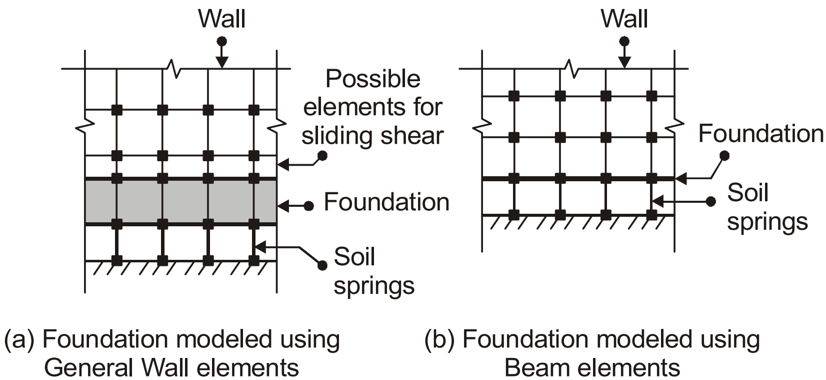



Figure 10.25 Possible Foundation Models

If the foundation can uplift, you can use gap-hook bar elements to model the soil. If there are piles, use stiffer springs or possibly model the piles as beams or columns. If you use General Wall elements to model a foundation, as in Figure 10.25(a), you must "imbed" the pile elements into the General Wall elements, by specifying a beam or column element between the top and bottom nodes of the foundation elements. This is essentially the same procedure as for a Shear Wall element. If you model the foundation using Beam elements, you can connect the pile elements to these element as in a typical frame.

If you suspect that there may be sliding shear at a construction joint, you may want to model it using a thin layer of General Wall elements, as indicated in Figure 10.25(a).

## 10.9 Fiber Sections for Axial/Bending Layers

### 10.9.1 General Considerations

For the axial/bending layers you must use fiber cross section components. The following are the behavior aspects to be modeled.

(1) Steel yield in tension.
(2) Concrete cracking.
(3) Possibly steel buckling in compression.
(4) Possibly concrete crushing in compression.

The following are the decisions that must be made.

(1) The steel material properties.
(2) The concrete material properties.
(3) The cross section dimensions.
(4) The number of fibers, and their areas and locations.

The following sections address these aspects.

### 10.9.2 Steel Material Properties

PERFORM allows a trilinear stress-strain relationship for steel materials. You can specify trilinear behavior if you wish, but given all of the uncertainties in the modeling and behavior of walls, we suggest that you assume elastic-perfectly-plastic (e-p-p) behavior.

To allow for uncertainty in the steel strength, you should probably consider at least two analysis models, with upper and lower estimates for the steel strength. One model should assume the upper strength for the flexural reinforcement (vertical steel in cantilevers and piers, horizontal in beams) and the lower strength for the shear reinforcement. This will tend to predict a larger bending strength capacity and a greater possibility of inelastic shear behavior. A second model could use a lower strength for the flexural reinforcement. This will tend to give a smaller bending strength capacity.

### 10.9.3 Concrete Material Properties

PERFORM allows a trilinear stress-strain relationship for concrete materials. You can specify trilinear behavior if you wish, but given all of the uncertainties in the modeling and behavior of walls, we suggest that you assume elastic-perfectly-plastic (e-p-p) behavior.

PERFORM allows you to specify a nonzero tension strength for concrete. You can specify a nonzero strength if you wish, but we suggest assuming zero strength.

### 10.9.4 Note on Cross Section Dimensions

The cross sections for the axial/bending layers will usually be inelastic fiber sections. However, they can also be elastic fiber sections. For example, if you use General Wall elements to model a foundation beam you might use an inelastic section for the horizontal (longitudinal) layer, to model bending of the foundation, but an elastic section for the vertical (transverse) layer. For this discussion assume inelastic fiber sections in both directions.

The steps required to specify the properties for General Wall elements are as follows.

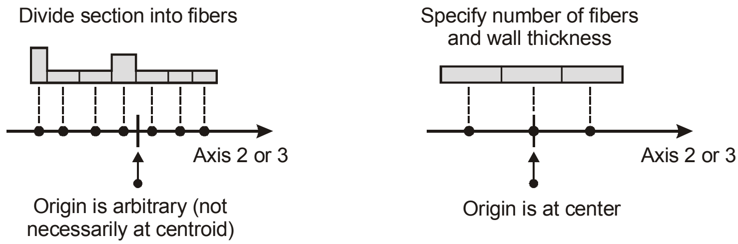
(1) Define properties for the steel material and concrete material components.

(2) Define the fiber cross section components, using these materials.

(3) Define General Wall compound components, assigning one fiber section to the vertical axial/bending layer and a second (possibly the same) fiber section to the horizontal axial/bending layer.

(4) Assign a General Wall compound components to each General Wall elements.

When you specify a fiber cross section, you have two options for the fiber areas and locations, namely "Fixed Size" and "Auto Size" as shown in Figure 10.26.




(a) "Fixed Size" Fiber Section. Specify area and coordinate for each fiber.

(b) "Auto Size" Fiber Section. Areas and coordinates depend on element width.

Figure 10.26 Fixed and Auto Options for Fiber Sections

The differences are as follows.

(1) In the "Fixed Size" option you specify actual areas and coordinates for the fibers. This has the advantage that you can consider any cross section geometry, including flanged sections. It has the disadvantage that you must be careful to make sure that the section dimensions are consistent with the size of the General Wall element. You must also define different Fiber Section and General Wall components for each different element width or depth. This is complicated for most walls, and you probably will not use this option.

(2) In the "Auto Size" option you specify a wall thickness and a number of fibers, and PERFORM calculates the fiber areas and coordinates, based on the size of the General Wall element. This has the advantage that the section dimensions are always consistent with the size of the element, and you do not need to define a different fiber cross section and general wall components for each different element width or depth. However, it has the disadvantage that you can consider only constant thickness walls with uniform reinforcement. You will probably use this option in most cases, and combine it with steel tie and concrete strut elements to account for variations in thickness and the amount of reinforcement. This is explained in the next section.

### 10.9.5 Use of Steel Tie and Concrete Strut Elements

Steel Tie and Concrete Strut elements are Bar elements. If you wish, you can use these elements to build a complete strut-and-tie model of a wall. More often, you will combine them with General Wall elements, for example as shown in Figure 10.27.

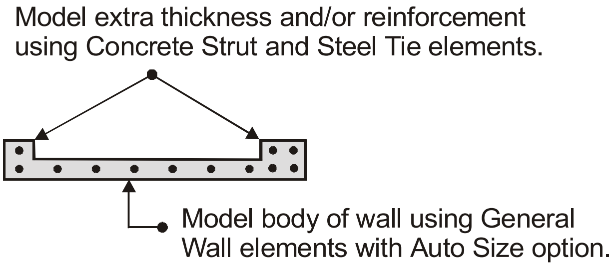



Figure 10.27 Model With Steel Tie and Concrete Strut Elements

The wall in Figure 10.27 has uniform thickness and reinforcing except for flanges at its ends. You can usually model the uniform thickness part with General Wall elements, and add Steel Tie and Concrete Strut elements to model the flanges. Examples are given in the General Wall Tutorial.

### 10.9.6 Flange Widths in a 2D Model

If a wall connects to perpendicular walls that act as flanges, you will usually model the flange using General Wall elements. However, if you wish you can model the wall as a 2D wall and use steel tie and concrete strut elements to model the flange. In this case you must estimate an effective flange width.

In most cases, the behavior of the wall when the concrete is in compression will not be sensitive to the assumed flange width. Usually a flange will provide a large concrete area, and when the wall cracks the neutral axis will move close to the flange. The neutral axis location, and hence the calculated behavior of the wall will not change much if the assumed flange width is changed. Also, if there is a flange it is unlikely that the concrete will crush, so the area of the flange is not critical.

However, when the flange is in tension the behavior can be sensitive to the assumed flange width, because it affects the tension steel area, and hence affects the bending strength of the wall.

The effective width of a flange depends on shear lag. For compression, Paulay and Priestley (page 369) recommend an effective flange width of 0.3 times the wall height (but not larger than the flange actual or tributary width) for a symmetrical flange. This means a width of 0.15 times the wall height for a flange on one side of the wall. This is a rather small effective width, corresponding to a "spread angle" of about 8.5 degrees. This recommendation is made because a relatively narrow width of the flange may be in compression after a number of deformation cycles, with other parts of the flange still cracked in tension.

For tension, Paulay and Priestley indicate that an effective width of 1.0 times the wall height has been used for a symmetrical flange, corresponding to a spread angle of 26.6 degrees. However, they refer to experiments that indicate a spread angle of as much as 45 degrees, corresponding to an effective width of 2.0 times the wall height for a symmetrical flange.

For calculating the bending stiffness of a wall for analysis, FEMA 356 (page 6-41) recommends a flange width of 0.2 times the wall height on each side of a wall. For calculating the bending strength, FEMA 356 (page 6-42) recommends a flange width of 0.1 times the wall height on each side, for both tension and compression. Since the bending behavior is not sensitive to the flange width in compression, the FEMA and Paulay-Priestley values for compression may not be significantly different. However, the bending strength capacity, and hence the shear strength demand, are both sensitive to the flange width in tension, and hence there are substantial differences between the FEMA and Paulay-Priestley values for tension.

You will have to use judgment on this issue. It may be wise to consider upper and lower bound estimates for the flange width.

### 10.9.7 Concrete Crushing on Axial/Bending Sections

Fiber cross sections can account for concrete crushing, but with one important caveat. In an actual cross section, as cracking spreads the concrete compression area continuously decreases, and the maximum concrete compression stress continuously increases. Also, after crushing begins in the extreme fiber, it progresses continuously into the cross section. In contrast, in a fiber section the changes in concrete compression area are discontinuous, with a reduction in area each time a concrete fiber cracks. Also, crushing occurs fiber by fiber, and hence progresses discontinuously into the cross section.

If the concrete crushing behavior is important, and if you want to model it accurately, you must estimate the depth over which the section will crush, and place a number of concrete fibers in this depth. This requires judgment, and we have no specific suggestions.

## *10.10 Fiber Sections for Different Parts of a Wall*

### 10.10.1 General

The preceding section considered fiber cross sections in general. This section makes suggestions on how to use fiber cross section components in the different parts of a wall, including vertical cantilevers, piers, beams, and strut-and-tie regions.

### 10.10.2 Fiber Sections For Vertical Cantilevers

For a vertical cantilever, the main axial/bending behavior is in the vertical direction. The horizontal direction is important mainly because it interacts with the diagonal compression layers to resist shear.

For vertical axial/bending behavior the total cross section of the cantilever will usually be divided into a number of elements. For example, in the left part of the wall in Figure 10.24 there are six elements across the width of the cantilever (you could probably use fewer than 6). The fiber cross sections for these six elements make up the cantilever cross section. You can provide a lot of fibers for the cross section as a whole with only a few fibers in each element. As noted earlier, in most cases you will probably use General Wall elements, with the "Auto Size" option to model the body of the wall, and use Steel Tie and Concrete Strut elements to account for additional reinforcement and thicker concrete areas. For the "Auto Size" fiber sections you may need only two concrete fibers and two steel fibers.

For the horizontal axial/bending layer in a vertical cantilever, the main concern is shear strength associated with diagonal compression. It is important to model the tension behavior of the reinforcement, but the concrete behavior should be less important. As for the vertical axial/bending layer, in most cases you will use General Wall elements with the "Auto Size" option plus Steel Tie and Concrete Strut elements to account for additional reinforcement and thicker concrete areas. For the "Auto" fiber sections you can use two concrete fibers and two steel fibers.

### 10.10.3 Fiber Sections for Piers and Beams

For a pier, the main axial/bending behavior is in the longitudinal (vertical) direction. If you do not consider diagonal compression layers, the transverse (horizontal) direction is less important. If you do consider diagonal compression layers, the transverse direction is important because it interacts with the diagonal compression layers to resist shear.

The same is true for a beam, except that the longitudinal direction is horizontal. A complication for a beam is that it may connect to a floor system within the height of the beam. This can stiffen and strengthen the beam in the longitudinal direction.

For longitudinal axial/bending behavior the main concerns are likely to be yielding of the steel and cracking of the concrete. It is unlikely that the performance will be controlled by crushing of the concrete. In most cases it should be possible to model the longitudinal concrete behavior adequately with five or six concrete fibers.

To model the longitudinal steel you can use either (a) steel fibers in a fiber section or (b) steel ties. This is illustrated in Figure 10.28.

Figures 10.28(b) and (c) show alternative models for the pier in Figure 10.28(a). The model in Figure 10.28(b) uses only General Wall elements. The longitudinal steel bars are modeled as fibers in a fiber section. With this model you must use the Standard option for the fiber areas and locations. The model in Figure 10.28(c) uses both General Wall and Steel Tie elements. The longitudinal steel bars are modeled as steel ties. In the longitudinal direction the fiber section needs to account only for the concrete, and the Auto option can be used for this section. However, because steel tie elements must connect nodes, either the nodes must be moved inwards, omitting some of the concrete, or the steel bars must moved outwards (or a combination of the two). For a fairly wide pier, or a fairly deep beam, the approximation should usually be acceptable.

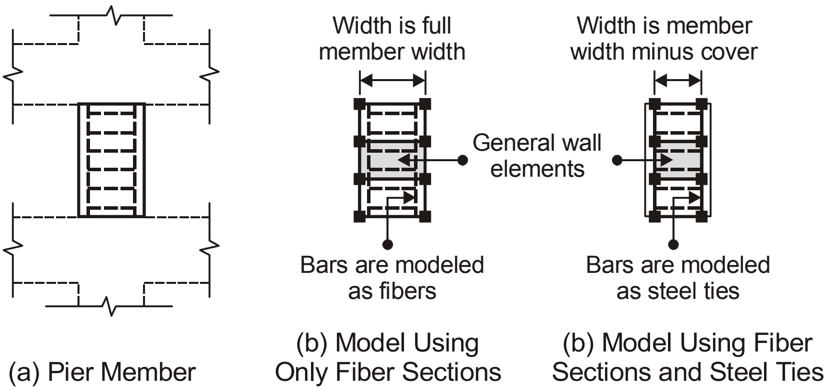



Figure 10.28 Alternative Models for a Pier

For transverse behavior the concern is yield of the ties or stirrups due to shear in the member. For the transverse direction we suggest a section with two steel fibers and two concrete fibers for each element. You can usually use the Auto option for the fiber section.

In walls that have strut-and-tie behavior, some piers may be essentially compression struts or tension ties. We suggest using the same models that you would use for piers that are essentially flexural members.

### 10.10.4 Fiber Sections for Strut-Tie Regions

In wall regions that have strut-tie behavior, there will usually need to be well defined bands of vertical and horizontal reinforcement. Vertical and horizontal ties and struts will develop in these bands. Inclined struts will develop in solid parts of the wall, but these are modeled by the diagonal compression layers. The reinforcement in the body of the wall can also be important in developing tie behavior, but its effect is distributed rather than concentrated.

In solid parts of the wall we suggest that you use vertical and horizontal axial/bending layers that are the same as you would use in vertical cantilevers. Although the modes of action can be substantially different for vertical cantilever and strut-tie regions, there is similar behavior in the axial/bending layers of the General Wall elements. If there are pier and beam members in strut-tie regions, model these like piers and beams in other parts of the wall.

## 10.11 Concrete Shear Layer for Vertical Cantilevers

### 10.11.1 Shear Stiffness

First consider shear stiffness. For modeling shear deformation in slender beams and columns, it is common practice to base the shear stiffness on the shear modulus of the material. If the shear modulus is $G$, and if the shear area of the member cross section is $A'$, then the shear stiffness for a member of length $L$ is $GA'/L$. For Poisson's ratio = 0.25, the shear modulus is $G = 0.4E$, where $E$ = Young's modulus. For a rectangular elastic cross section, $A'$ is 5/6 of the cross section area.

For the analysis of typical beams and columns this is a reasonable assumption for the shear stiffness. Shear deformations are typically small compared with bending deformations, so any error in the shear stiffness tends to have only a small effect on the overall behavior.

However, this is not necessarily the case for shear deformations in squat walls. Shear in reinforced concrete can be modeled as conventional shear and/or using a diagonal compression field. The shear stiffness $GA'/L$ is associated with conventional shear. First, however, consider the shear stiffness for the case with a diagonal compression field. Typically the body of a wall will have vertical and horizontal bars providing about 0.2% reinforcement area in each direction. Consider a square region in the body of a wall, as shown in Figure 10.29.

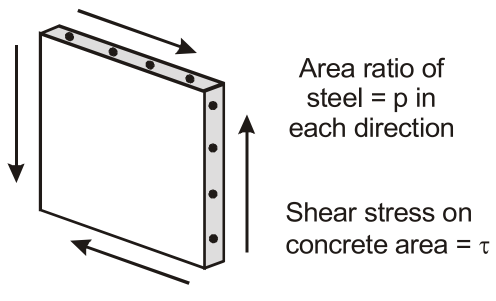



Figure 10.29 Shear in the Body of a Wall

If we consider only diagonal compression field behavior, the shear stiffness of this element can be calculated as follows.

(1) For small reinforcement percentages the diagonal concrete struts are much stiffer than the steel bars. Assume that the diagonal struts are rigid. Hence, essentially all of the shear deformation originates in the steel bars.
(2) Assume a strut angle of 45 degrees.
(3) Assume equal reinforcement proportions, *p*, in the vertical and horizontal directions.
(4) Assume elastic behavior, and hence calculate the elastic stiffness. We will consider strength later.
(5) When shear is applied, the vertical and horizontal reinforcing bars provide the confining stress. As already shown using Mohr's circle, for a shear stress equal to $\tau$, the confining stress must also be $\tau$.
(6) Since the steel area is *p* times the concrete area, a shear stress of $\tau$ in the concrete corresponds to a stress $\tau/p$ in the steel.
(7) Hence the horizontal and vertical steel strains are each $\tau/pE_s$, where $E_s$ = Young's modulus for steel.
(8) Since we have assumed a rigid diagonal, the strain in the compression diagonal is zero. Hence, from Mohr's circle for strain it follows that the shear strain is twice the horizontal or vertical strain. That is, $\gamma = 2\tau/pE_s$, where $\gamma$ is the shear strain.
(9) Hence the effective shear modulus is $G = \tau/\gamma = 0.5pE_s$.

For example, if $p = 0.002$ (i.e., 0.2%) and $E_s = 10E_c$, where $E_c$ = concrete modulus, $G = 0.01E_c$. This is 40 times smaller than the value $0.4E_c$ that might typically be used for beam or column analysis. Hence, the shear stiffness from diagonal compression action is much smaller than the shear stiffness that is commonly assumed for analysis.

In a General Wall element the shear stiffness is the sum of the conventional shear stiffness (in the concrete shear layer) and the stiffness provided by shear reinforcement (by interaction between the diagonal compression and axial/bending layers). At first sight, it might seem that for the analysis of walls we should base the stiffness of the concrete shear layer on conventional shear with *G* equal to about $0.4E_c$. This is almost certainly not a sound assumption, for the following reason.

In a steel beam, the shear stiffness and strength depend directly on the shear modulus of the material and its yield stress in shear. In a concrete beam, the concrete part of the shear strength is not based simply on the material properties. As noted by Paulay and Priestley: "Aggregate interlock along crack interfaces, dowel action of chord reinforcement, shear transfer by concrete in the flexural compression regions, arch action, and the tensile strength of un-cracked concrete are typical components of such complex mechanisms." The effective stiffness for such mechanisms is almost certainly much smaller than the stiffness based on the shear modulus of the concrete. This is especially true under cyclic loading, where these mechanisms are likely to degrade in both stiffness and strength.

In the General Wall element, the stiffness and strength of the concrete shear layer depend on the properties of a "shear material". When you define the shear modulus for this material we suggest that you must specify a value much smaller than $0.4E_c$. We suggest a specific value later in this section.

## 10.11.2 Shear Strength

It is also necessary to assign a strength to the concrete shear layer. This is the most difficult part of the problem.

For the design of shear walls, Paulay and Priestley (page 127) make the following recommendations for the contribution of the concrete to the shear strength, $v_c$, in structural walls.

(1) In all regions except potential plastic hinges :

$$v_c = 0.27\sqrt{f_c'} + \frac{P}{4A_g} \quad \text{(MPa)} \qquad (10.1a)$$

$$v_c = 3.3\sqrt{f_c'} + \frac{P}{4A_g} \quad \text{(psi)} \qquad (10.1b)$$

(2) In regions of plastic hinges :

$$v_c = 0.6\sqrt{P / A_g} \quad \text{(MPa)} \qquad (10.2a)$$

$$v_c = 7.2\sqrt{P / A_g} \quad \text{(psi)} \qquad (10.2b)$$

where the symbols have their usual meanings and $P$ is the axial force, compression positive.

For example, for $f_c^{'}$ = 3000 psi and $P / A_g$ = 400 psi, Equation 10.1b gives $v_c$ = 181 + 100 = 281 psi, and Equation 10.2b gives $v_c = 144$ psi. If $P$ is tension, the concrete in a plastic hinge region has no shear strength, and all shear resistance should be provided by shear reinforcement.

These strengths can be compared with the shear strength provided by the reinforcement and diagonal compression action. For the region of wall shown in Figure 10.29, with a steel ratio $p$ and steel strength $f_y$, the shear stress when the steel yields is $pf_y$. For $f_y$ = 40,000 psi and $p$ = 0.002, this stress is 80 psi.

The values given in Equations 10.1 and 10.2 both depend on the axial force, which can change during an earthquake. This is essentially a frictional effect, and in the current version of the General Wall element this effect is not taken into account. For the concrete shear layer you must specify a fixed shear strength that is independent of the axial force.

Also, Equations 10.1 and 10.2 are primarily for the design of relatively slender shear walls, where, in a well designed wall, there is a plastic hinge only at the base. In this case it is relatively easy to identify hinging and non-hinging regions, and hence to choose between Equation 10.1 and Equation 10.2. This is not so easy in a squat wall, where yield in bending can extend over most of the wall height.

### 10.11.3 Suggested Properties for Concrete Shear Layer

Based on the discussion in the preceding section, we make the following suggestions for the properties of the concrete shear layer.

(1) Consider the reinforcement in the body of the wall, and calculate the effective shear modulus for 45$^{o}$ diagonal compression action. As shown earlier, this is $G = 0.5\ pE_s$, where $p$ is the steel ratio and $E_s$ is Young's modulus for steel. Assume that the shear modulus for the Shear Material in the concrete shear layer is equal to two times this modulus, or $G = pE_s$.

This suggestion is based on the reasoning is that the actual shear stiffness for conventional shear in reinforced concrete is much less than that based on the shear modulus of concrete. A stiffness equal to twice the stiffness for diagonal compression action is suggested as a reasonable assumption. If you use both concrete shear and diagonal compression layers, the total shear stiffness in the body of the wall is then three times that for diagonal compression alone.

(2) Using your best judgment, choose a shear strength, considering Equations 10.1 and 10.2. For the value of $P / A_g$ use the stress under gravity load. Since a squat wall can crack over most of its height after a few earthquake cycles, we suggest that you err on the low side, using Equation 10.2 rather than Equation 10.1.

An alternative that you might consider is to calculate the effective shear strength provided by the shear reinforcement ($pf_y$, where $f_y$ is the steel yield stress) and assume that the shear strength for the concrete layer is equal to this value. If you use both concrete shear and diagonal compression layers , the total shear strength in the body of a wall is twice that provided by the shear reinforcement. This is a guess, but may be a reasonable value in a wall that is cracked after a few earthquake cycles.

(3) Define an elastic-perfectly plastic shear material with this shear modulus and shear strength. Use this material for the concrete shear layers in General Wall compound components.

At the time of writing, the General Wall element has not been calibrated against experiment or other analyses.

## *10.12 Concrete Shear Layer for Piers and Beams*

### 10.12.1 Shear Strength for Slender Frame Members

The concrete shear strength for walls was considered earlier. Since piers and beams might be regarded as essentially frame members rather than walls, it is useful to consider the shear strength for slender columns and beams.

For beams and columns, Paulay and Priestley (page 127) recommend the following values for the concrete contribution to the shear strength.

(1) In all regions except potential plastic hinges:

$$v_c = (0.85 + 120p)\sqrt{f_c^{'}} \leq 2.4\sqrt{f_c^{'}} \quad \text{(psi units)} \qquad (10.3)$$

This is smaller than the value given for walls in Equation 10.1.

(2) In the plastic hinge region of a beam, $v_c$ = 0 (i.e., all shear must be carried by shear reinforcement).

(3) In the plastic hinge region of a column in compression, $v_c$ is given essentially by:

$$v_c = 4(0.85 + 120p)\sqrt{\frac{P}{A_g}} \quad \text{(psi units)} \qquad (10.4)$$

where $p$ is the proportion of flexural reinforcement. For walls, $p$ is typically small, and Equation 10.4 gives roughly:

$$v_c = 5\sqrt{\frac{P}{A_g}} \quad \text{(psi units)} \qquad (10.5)$$

Equation 10.4 tends to give smaller values than Equation 10.2. For $P / A_g$ = 400 psi, Equation 10.4 gives $v_c$ = 100 psi.

(4) In the plastic hinge region of a column in tension, $v_c$ = 0.

Paulay and Priestley thus recommend rather smaller concrete shear strengths for frame members than for walls. Most piers and beams in squat walls will have small length-to-depth ratios, and hence will tend to be closer to wall elements than to frame elements. However, it is worthwhile to take Equations 10.3 and 10.4 into account.

### 10.12.2 Some Aspects of Pier and Beam Behavior

Even though piers and beams tend to be closer to walls than to frame members, they can have significantly different behavior from vertical cantilevers, as follows.

(1) If a short pier or beam yields in bending, essentially the whole length will be a "hinge region", especially after a few earthquake cycles. This poses a paradox for modeling. If a pier or beam does not yield in bending its shear strength can be a larger value based on Equation 10.1 or 10.3. However, if it has a larger shear strength it is more likely to yield in bending, in which case most of the member length is a hinge region and its shear strength is a smaller value based on Equation 10.2 or 10.4.

(2) Equations 10.1 and 10.3 are design equations that assume yield will not occur in shear. If yield occurs in shear in a pier or beam, diagonal cracks will develop and the subsequent shear capacity will be smaller than that given by Equation 10.1 or 10.3.

Hence, whether a member yields in bending or shear, it appears that for analysis purposes the concrete shear strength in piers and beams should be based on Equation 10.2 or 10.4, not Equation 10.1 or 10.3. The only time that Equation 10.1 or 10.3 applies is if the member does not yield in either bending or shear. This could be the case for a flexible pier in a wall where the drifts are controlled by stiffer elements, or in a beam that is deeper and stronger than adjacent piers. Such members will have relatively small deformations, and they will usually not be critical for performance assessment.

### 10.12.3 Suggested Properties for Shear Layer

For the concrete shear strength in piers and beams, we suggest that you use essentially the same procedure as for vertical cantilevers. We also suggest that you consider being rather more conservative for piers and beams, because of the possibility of rather larger bending and/or shear cracks in piers and beams than in vertical cantilever sections.

### 10.12.4 Number of Elements Along Length

An additional difference between a pier or beam and a vertical cantilever is that a pier or beam will usually be modeled with a lot fewer elements. Often there will be only one element through the depth of the member, and only a few elements along the length. That is, you will usually use relatively coarser element meshes for piers and beams than for vertical cantilevers. Since the calculated behavior can be sensitive to the element mesh, you must be careful not to use too coarse or too fine a mesh.

The main problem is when a pier or beam yields in bending rather than shear. If a member yields in shear, the shear force is constant along the member length, and the number of elements has little effect on the behavior. However, if the member yields in bending, the bending moment varies linearly over the length, and the calculated bending strength can depend a lot on the number of elements.

Figures 10.12 through 10.15 show some effects of mesh size. In Figure 10.12, with only conventional shear, the calculated maximum flange force is 75% of the theoretical maximum. Hence, the calculated strength for this mesh is 1.33 times larger than the expected strength. In Figure 10.13, with only diagonal compression, the calculated maximum flange force is equal to the theoretical maximum, and hence the calculated strength is equal to the expected value. For finer meshes, as shown in Figure 10.15, with only conventional shear the calculated strength is 1.14 times the expected strength, and with only diagonal compression the calculated strength is 0.89 times the expected strength (i.e., this model underestimates the strength).

The cantilever in this example corresponds to a slender pier with a length-depth ratio of 4, with four and eight elements along the length for the coarse and fine element meshes, respectively. Figure 10.30 shows some element meshes for a pier with a length-to-depth ratio of 2 (which is still a relatively slender pier).

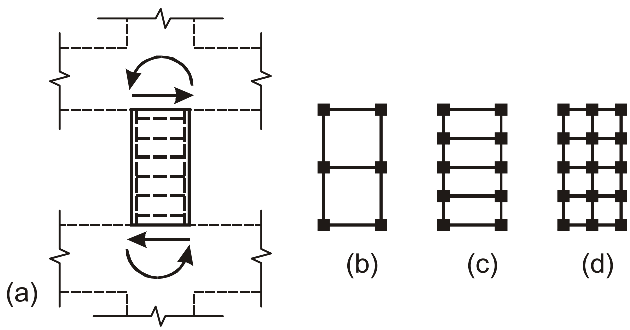



Figure 10.30  Element Meshes for a Pier

For the mesh in Figure 10.30(b), with two square elements along the pier length, the calculated strengths for the cases with only conventional shear and only diagonal compression are respectively 2.0 times and 1.0 times the expected strengths. The model with only diagonal compression gives the correct strength with square elements, since it models truss behavior. For the mesh in Figure 10.30(c), with four elements along the length, the calculated strengths are respectively 1.33 times and 0.8 times the expected strengths.

Hence, when bending governs, a model with only conventional shear tends to overestimate the strength, and a model with only diagonal compression tends to underestimate it. One possible solution is to use several elements along the length to account for the conventional shear effect, and to use square elements to account for the diagonal compression effect. One such mesh is shown in Figure 10.30(d). A simpler mesh that should give reasonable results in most cases is as follows.

(1) Assign a strength to the concrete shear layer that is essentially equal to the shear strength from diagonal compression action, as suggested earlier.

(2) Use one element through the member depth.

(3) Make the element length essentially one half of the member depth.

With this mesh the errors from conventional shear and diagonal compression will tend to cancel each other. If the piers and beams are not the major part of a wall, approximations in their bending strength may not have much effect on the overall behavior. If the piers and beams are the major part of a wall, you probably should use a finer mesh as in Figure 10.30(c).

This applies for piers and beams that have length-to-depth ratios larger than one. For a member with a length equal to its depth, shear is transmitted by diagonal compression across the member, with no bending (in general, very short elements are unlikely to yield in bending – they will either yield in shear or remain essentially elastic).

Remember that the mesh is important only if a pier or beam yields in bending. You can usually check this with a simple hand calculation. For example, consider the pier in Figure 10.31.

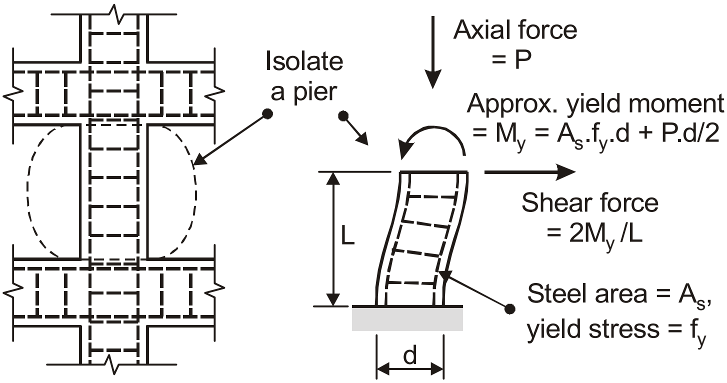



Figure 10.31 Forces on a Pier

For this pier, the bending moment required to yield the longitudinal steel is approximately:

$$M_y = A_s f_y d + 0.5Pd \qquad (10.6)$$

The corresponding shear force is $2M_y / L$. Hence, to ensure flexural yield the shear capacity, expressed as shear force divided by the gross concrete area, $A_g$ ,must be essentially

$$v_c + v_s = \frac{2M_y}{LA_c} = \frac{d}{L}\left(2\frac{A_s}{A_g} f_y + \frac{P}{A_g}\right) \qquad (10.7)$$

## *10.13 Brittle Strength Loss*

PERFORM allows you to specify ductile limits and strength losses for steel, concrete, shear and diagonal compression materials. Actual walls may exhibit strength loss, particularly in shear. Should you attempt to include this in the analysis model?

Given the many complications and uncertainties in the modeling of walls, we suggest that in most cases you do not consider strength loss in the analysis model.

## 10.14 Foundation Modeling

You may decide to assume that the nodes at the foundation level are fully restrained, or you may decide to model the foundation explicitly. Figure 10.25 indicated how a foundation might be modeled.

At this time we have no specific suggestions on the properties for foundation modeling.

## 10.15 Floor Diaphragms

As a general rule you should not specify rigid floor constraints in squat walls. The reason is that when you specify a rigid floor constraint at a floor level, in effect you are specifying a horizontal reinforcing band that is both infinitely stiff and infinitely strong, in both tension and compression. In a wall structure, the beams (and possibly other components) tend to grow horizontally as they bend, because of shift in the neutral axis. If you specify a rigid floor constraint you will suppress this expansion, causing axial compression in the beams. This can substantially, and artificially, increase their bending strengths.

## 10.16 Other Aspects

### 10.16.1 Connection Regions

Connection regions between piers and beams are similar to connection panel zones in frame structures. Figure 10.31(a) shows a connection region.

Considering only strut and tie behavior, the flow of forces through the connection is essentially as shown in Figure 10.31(b). Two key points are as follows.

(1) There is a diagonal compression strut in the connection region.

(2) The stresses in the longitudinal steel bars change from tension to compression in the joint region. Hence, there must be significant bond stresses.

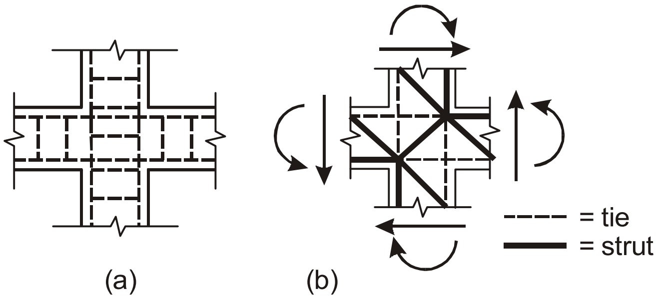



Figure 10.31 Connection Region

In principle, diagonal crushing and bond slip are both possible in a joint region. However, in most cases the joint region is large and the piers and spandrels have fairly small percentages of longitudinal reinforcement. Hence, the stresses in connection regions will usually not be large. Nevertheless you may need to check diagonal compression stresses and diagonal tension strains in joint regions.

We suggest that you model joint regions in the same way as vertical cantilevers, with the same stiffness and strength in the concrete shear layer.

## 10.16.2 Horizontal Distribution of Lateral Loads

For push-over analysis of frames, it is common to assume rigid floors and to apply the lateral loads on one side of the frame. The rigid floors then distribute the loads across the width of the frame. As noted earlier, for analysis of walls, you should not specify rigid floor constraints. Also, you should not apply the lateral load on only one edge of the wall.

If you apply the lateral load on one side of a wall, the load must be distributed across the wall as horizontal tension or compression forces in the elements of the analysis model. If you apply all of the load on the left edge, there will be substantial horizontal compression forces in the analysis model that are not present in the actual wall. If you apply all of the load on the right edge, there must be sufficient horizontal steel at the floor level to resist the entire force in tension, otherwise the steel will yield.

For narrow walls it may be acceptable to apply the force 50-50 on the left and right edges. In general, however, you should distribute the loads across the wall width, approximating the way that the loads in the real structure are transferred from the floors to the wall.

### 10.16.3 Horizontal Distribution of Mass

For the same reason that you should distribute lateral push-over loads across the width of a wall, you should also distribute the structure mass across the width. Note that this applies even if you are using only static push-over analysis to assess the performance, since the calculation of base shear demands makes use of the dynamic properties.

## *10.17 Deformation Measures*

### 10.17.1 Aspects to be Assessed

Limit states are useful for assessing one or more of the following aspects of performance.

(1) Bending in piers, beams and vertical cantilevers. Yield of the steel will usually be permissible. Crushing of the concrete and/or buckling of the steel will usually be undesirable.

(2) Shear in piers, beams and vertical cantilevers. Shear can be resisted by concrete shear and/or diagonal compression. Some inelastic shear behavior may be permissible.

(3) Tension in the ties of strut-and-tie regions. Inelastic behavior will usually be permissible in vertical ties, but is probably undesirable in horizontal ties.

(4) Compression in the struts of strut-and-tie regions. Inelastic behavior will usually be undesirable.

### 10.17.2 Available Deformation Measures

A General Wall compound component can have two Fiber Section components, a Shear Material component and a Diagonal Compression Material component. You can set up deformation limit states using any of the following deformations.

(1) Steel fiber tension strains in the longitudinal axial/bending layer. For a pier or a vertical cantilever this is the vertical axial/bending layer. For a beam it is the horizontal layer. Strains in excess of yield will usually be allowed.

(2) Steel fiber compression strains in the longitudinal axial/bending layer. Strains in excess of yield will usually not be allowed.

(3) Concrete fiber compression strains in the longitudinal axial/bending layer. Strains in excess of yield may be allowed.

(4) Steel fiber tension strains in the transverse axial/bending layer (i.e., strains in the shear reinforcement). Strains in excess of yield may be allowed.

(5) Shear strain in the concrete shear layer. Strains in excess of yield may be allowed.

(6) Diagonal compression strain in the diagonal compression layers. Strains in excess of yield will usually not be allowed.

(7) Diagonal tension strain in the diagonal compression layers. Cracking will occur, but should not be excessive.

You may also use Steel Tie and Concrete Strut components to model a wall. Hence, you can also use the following deformations.

(8) Tension and/or compression strain in Steel Tie components.

(9) Compression and/or tension strain in Concrete Strut components

Finally, you may also use Strain Gage components. These components have not been mentioned before in this tutorial, but as explained below they can useful for setting up limit states. With these components you can use the following deformations.

(10) Tension or compression strains in Strain Gage components.

### 10.17.3 Strain Gage Elements

One of the problems with finite element models of reinforced concrete is that the calculated steel strains can be sensitive to the element mesh. For example, refer back to Figures 10.12 and 10.15. In analyses using these element meshes, the lower left flange will yield first, and then the lower element will essentially act as a plastic hinge. In both figures, the strain in the yielding flange is its extension divided by the element height. For a given deflection at the top of the cantilever, the effective hinge rotations, and hence the flange extensions, are similar in both cases. However, the element height in Figure 10.15 is only one half that in Figure 10.12. Hence, the calculated strain for the mesh in Figure 10.15 will be roughly double that for the mesh in Figure 10.12. As the mesh gets progressively finer, the effective plastic hinge gets progressively shorter, and the calculated strain gets progressively larger.

A consequence of this is that the calculated strain in a steel fiber may not be a reliable deformation measure for assessing performance. The same is true of the calculated strain in the steel tie elements.

In a real wall we can expect that an effective plastic hinge will have a finite length, of the order of the cross section depth, as indicated in Figure 10.32(a).

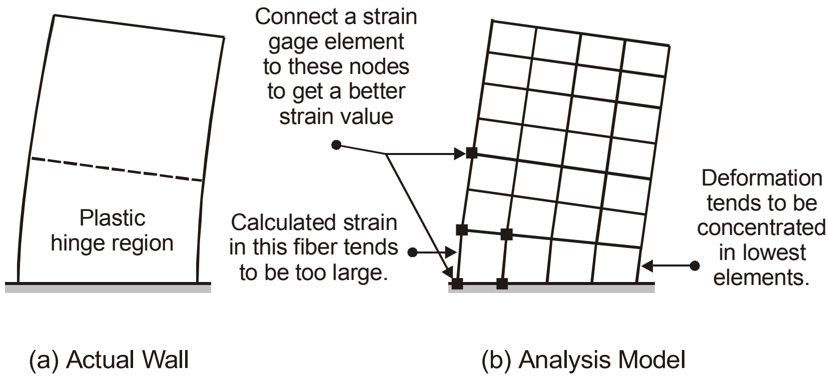



Figure 10.32 Strain Over Plastic Hinge Length

If there are several elements within the hinge length, as in Figure 10.32(b), the hinge rotation will tend to be concentrated in the lowest row of elements, and the calculated steel strain will tend to be much larger in the lowest elements than in those above. A more useful strain value for assessing performance is the average steel strain over the length of the hinge. This strain can be calculated by estimating the hinge length and using a strain gage element that extends over this length, as shown in Figure 10.32(b). The calculated strain in the strain gage an element is less sensitive to changes in the element mesh, and is a more useful measure of ductility demand and performance.

In order to use strain gage elements, you must estimate the effective hinge length. FEMA 356 recommends the smaller of (a) one half the cross section depth and (b) one half the story height.

You may note that a similar smoothing out of the calculated strains could be achieved by using Steel Tie elements and connecting these elements not to every node but only to the nodes at the ends of the effective plastic hinge. We do not recommend this, because it increases the band width of the structure stiffness matrix, and can significantly increase the computer time required for analysis.

It can also be useful to use Strain Gage elements in the transverse direction. In Figure 10.32(b), if the horizontal reinforcement yields at any level because of shear, the yield may be concentrated in one element at that level. A better strain measure can be obtained by using a Strain Gage element that extends over the depth of the cross section, connecting the left and right nodes

### 10.17.4 Other Deformation Gage Elements

You can also use wall rotation gage and shear strain gage elements to measure deformations and to set up corresponding limit states.

## *10.18 Deformation Limit States*

### 10.18.1 General

You can use limit states in three main ways, as follows.

(1) For limit points in capacity spectrum and target displacement plots. These enable you to make direct judgments on performance.

(2) For usage ratios in usage ratio plots. These also enable you to make direct judgments on performance.

(3) For color-coded deflected shape plots. These enable you to identify those elements that have the largest usage ratios, and hence the greatest amount of damage or the greatest strength demands.

As the preceding section shows, you can choose from a variety of deformation measures. Usually you will need to use only a few of these deformations, as outlined below.

## 10.18.2 Limit States for Vertical Cantilevers

For vertical cantilevers we suggest that you set up deformation limit states as follows. For each deformation type you may set up one or more limit states, each corresponding to a different performance level.

(1) If you use steel tie elements to model the main vertical reinforcement, set up one or more limit states based on tension strain in these elements. If you use fiber sections in General Wall elements to model the main vertical reinforcement, set up limit states based on tension strain in the vertical steel fibers for these elements. As noted earlier, tension strains in individual elements may not be a good measure of performance, because plastic strains tend to concentrate in a few elements. However, when you draw color-coded deflected shapes, the relative values of these strains can give a useful picture of the strain "hot spots".

(2) Set up limit states based on strain or rotation using deformation gage elements that extend over the estimated plastic hinge length in the vertical direction. For a complex wall you may need to run preliminary analyses to identify the locations that have the largest element strains, then add deformation gage elements in these locations.

(3) If you use steel tie elements to model horizontal reinforcement at the floor levels, set up limit states based on tension strain in these elements. If you use fiber sections in General Wall elements to model the horizontal reinforcement, set up limit states based on tension strain in the horizontal steel fibers for these elements. If you use steel ties, note that in order to have separate limit states for vertical and horizontal bars, you must put the vertical and horizontal Steel Tie elements in separate element groups. This is because a limit state applies to all elements in an element group.

(4) In the body of the wall you will probably use fiber sections (with the Auto option) to model the distributed vertical and horizontal reinforcement. To check for yield of the horizontal reinforcement, set up limit states based on tension strain in the horizontal fibers of general wall elements. These limit states can be useful in color coded deflected shapes.

(5) As with the vertical reinforcement, horizontal plastic strains may tend to concentrate in a few elements. Because of this, if you allow significant yield of the horizontal reinforcement, and if you want to use strain usage ratios to assess performance, you may also want to set up limit states based on horizontal strain gage elements.

(6) Vertical compression stresses in the concrete will usually be below yield, and you usually will not need to consider concrete compression strains for performance assessment. However, it can be useful to show vertical compression stresses in a color coded deflected shape. We suggest that yet up limit states based on vertical compression strain in general wall elements and vertical concrete strut elements, and use these strains as a measure of compression stress.

(7) Horizontal compression stresses in the concrete are likely to be less critical than vertical stresses. However, we suggest that yet up limit states based on horizontal compression strain in general wall elements and horizontal concrete strut elements, and use these strains as a measure of compression stress.

(8) Set up limit states based on shear strain in the concrete shear layer of general wall elements. Alternatively, or in addition, use wall type shear strain gage elements. Shear strain gages have the advantage that they can average out the shear strain over several elements.

(9) If you include diagonal layers, set up limit states for diagonal tension strain in General Wall elements. This can be a good measure of diagonal tension cracking and hence of shear deformation, to supplement the information from strain in the horizontal reinforcement.

(10) If you include diagonal layers, the diagonal compression stresses will usually be well below yield, and you usually will not need to consider diagonal compression strains for performance assessment.

However, in a color coded deflected shape, diagonal compression stresses can provide a useful picture of how forces are transmitted. We suggest that you set up limit states based on diagonal compression strain in General Wall elements, and use these strains as a measure of diagonal compression stress.

### 10.18.3 Limit States for Piers and Beams

For piers and beams we suggest that you set up deformation limit states as follows. For each deformation type you may set up one or more limit states, each corresponding to a different performance level.

(1) If you use steel tie elements to model the longitudinal reinforcement, set up one or more limit states based on tension strain in these elements. If you use fiber sections in general wall elements to model the longitudinal reinforcement, set up limit states based on tension strain in the steel fibers for these elements.

Unlike vertical cantilevers, you will usually not need to use strain gage elements. If you make the element lengths in piers and beams equal to essentially one half the member depth, this should be a reasonable value for the plastic hinge length. Hence, the strains in steel tie elements or the fibers of general wall elements should be reasonable measures of bending performance.

(3) In the transverse direction  you will probably use fiber sections (with the Auto option) to model the shear reinforcement. To check for yield of the transverse (shear) reinforcement, set up limit states based on tension strain in the transverse direction fibers of general wall elements.

(4) If you include diagonal layers , set up limit states for diagonal tension strain in general wall elements. This is a measure of diagonal tension cracking, and hence an additional measure of shear deformation to supplement the information from strain in the transverse reinforcement.

(5) Longitudinal compression stresses in the concrete will usually be below yield, and you usually will not need to consider concrete compression strains for performance assessment. However, it can be useful to show compression stresses in a color coded deflected shape. We suggest that yet up limit states based on concrete

compression strain in the longitudinal direction in general wall elements, and use these strains as a measure of compression stress.

(6) Diagonal compression stresses in piers and beams will usually be well below yield, and you will not need to consider diagonal compression strains for performance assessment. However, you might like to set up limit states based on diagonal compression strain in the General Wall elements in connection regions between piers and beams.

(7) You also might like to set up limit states based on diagonal tension strain in connection regions.

### 10.18.4 Limit States for Strut and Tie Regions

For the solid parts of strut and tie regions we suggest the same limit states as for vertical cantilevers. For piers and beams in strut and tie regions we suggest the same limit states as other piers and beams.

### 10.18.5 Monitored Fibers for Fiber Strains

A number of the above limit states are based on strains in the fibers of fiber sections. It is important to note that if you want to calculate strain demand-capacity ratios for a fiber section you must specify one or more monitored fibers for that section. If you specify only structural fibers, with no monitored fibers, strain demand-capacity ratios will not be calculated and the usage ratio will be zero.

### 10.18.6 Deformation Gages

As noted, you can use Deformation Gage elements to monitor fiber strains, hinge rotations and shear deformations.

If strain is used as a demand-capacity measure, it may be better to use a strain gage than to use monitored fibers. The reason is that a strain gage can extend over several wall elements, and hence calculate the average strain over those elements. Monitored fibers give the strains in single elements. If there is localized strain concentration, it may be unnecessarily conservative to use single element strains.

Rotation and shear gages can also extend over several elements, and hence calculate average rather than localized deformations.

## 10.19 Strength Limit States

If you wish, you can set up strength limit states for axial-bending behavior by using an elastic, rather than inelastic, fiber section, and using the stress capacities at the monitored fibers. This is the same as for a Shear Wall element.

In a Shear Wall element you can also specify that the shear material is elastic, and define strength limit states using the material shear strength. You can not do this in a General Wall element, since the shear material must be inelastic. If you expect that a wall will remain elastic in shear and you want to check the strength demand-capacity ratio in the concrete shear layer, you must do this using shear deformation, not shear strength. The procedure is as follows.

(1) Calculate the yield strain for the Shear Material.

(2) Specify this strain as the strain capacity for this component, for one of the performance levels.

(3) Specify a limit state that uses this strain. The usage ratio for the limit state will be the strength demand-capacity ratio for the material.

## 10.20 Element Loads

PERFORM does not allow element loads for general wall elements. However, wall weight can be considered using self weight loads.

## 10.21 Geometric Nonlinearity

You can include or ignore P-Δ effects. You can not consider true large displacement effects.

P-Δ effects are considered for both in-plane and out-of-plane (plate bending) effects. If you specify P-Δ effects, be careful not to specify a very small thickness for plate bending. You may want to ignore the shear forces resisted by plate bending, and consider only in-plane shear. You may be tempted to do this by specifying a small bending thickness. However, if you specify that P-Δ effects are to be considered, a small thickness means that the wall is likely to buckle.

Specify a realistic thickness. The amount of shear force resisted by plate bending is likely to be only a small part of the total shear force.

## 10.22 Conclusion

Modeling squat walls with irregular openings is a complex task. There are few, if any, established guidelines, and the process requires judgment The PERFORM General Wall element is suggested as a reasonable solution. Keep in mind that the goal is not to set up an "exact" model and do an "exact" simulation of the behavior, but to set up a model that is accurate enough for design purposes and gives useful information for performance assessment.

# 11 Infill Panel Element

*This chapter reviews the components and elements that can be used to model infill panels.*

*Infill panel elements can also be used to model deformable floor diaphragms for earthquake analysis.*

## 11.1 Infill Panel Components

PERFORM includes the following infill panel components.

(1) Linear elastic panel, shear model.
(2) Inelastic panel, shear model.
(3) Inelastic panel, diagonal strut model.

Figure 11.1 shows the shear model. The action for this model is the horizontal shear force and the deformation is the shear displacement over the height of the panel, as shown. This component has shear stiffness and strength only. It has zero stiffnesses for extension and bending, and zero out-of-plane stiffness. Hence, it should be used only if surrounded by frame-type elements.

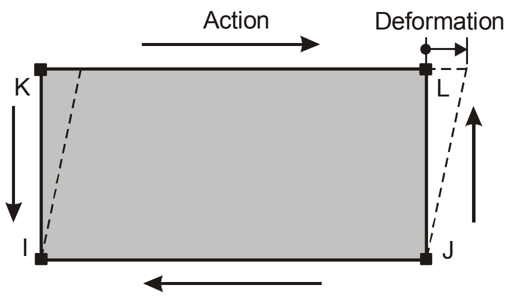



Figure 11.1 Infill Panel Shear Model

Infill panel elements using the shear model can be used for deformable floor diaphragms for PERFORM-3D. In this case, remember that the element has shear stiffness only, and must be surrounded by frame type elements or given equivalent support.

Figure 11.2 shows the diagonal strut model this model consists of two struts, each of which resists compression force only. The actions and deformations are the compression forces and compression deformations of the struts, as shown. This component has no other stiffnesses.

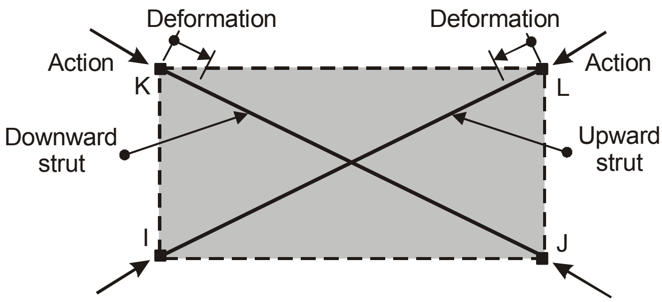



Figure 11.2  Infill Panel Diagonal Strut Model

Figure 11.3 shows the action-deformation relationship for a strut. The hysteresis loop for a strut is the same as for the inelastic concrete material.

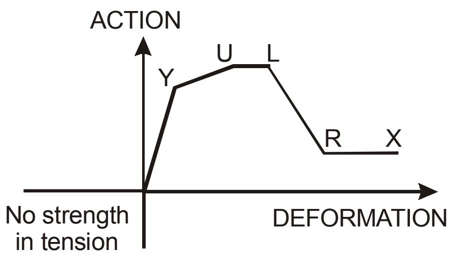



Figure 11.3  Action-Deformation Relationship for a Strut

## 11.2 Infill Panel Elements

Each infill panel element consists of one infill panel component. Elements are typically rectangular, but may have some taper or skew. If a shear model element is skewed, the shear force and displacement are normal to a line that connects the midpoints of the IJ and KL edges. Elements are typically vertical, but for special applications they could be horizontal or inclined.

For the diagonal strut model, the upward strut is a bar that connects the I and L nodes, and the downward strut is a bar that connects the K and J nodes, as shown in Figure 7.2.

## 11.3 Element Loads

PERFORM does not allow element loads for infill panel elements. However, panel weights can be considered using self weight loads.

## 11.4 P-Δ Effects

You can choose to include or ignore P-Δ effects. In-plane P-Δ effects are present only for the diagonal strut model (shear exerts no in-plane P-Δ effect). Each strut exerts a separate P-Δ effect, exactly like a simple bar. Both the shear model and the diagonal strut model exert out-of-plane P-Δ effects. The shear model exerts an out-of-plane effect if the element warps. This effect is usually small.

There is no option for true large displacement effects.

# 12 Viscous Bar Element

*This chapter reviews the components that can be used to model viscous bar (fluid damper) elements.*

## 12.1 Components

The current version of PERFORM requires that a viscous bar element consist of one each of the following components.

(1) Linear elastic bar.
(2) Fluid damper.

The deformation measure for the linear elastic bar component is axial strain, not axial extension. Hence the stiffness is *EA*, where *E* = Young's modulus and *A* = bar area. The axial stiffness in terms of axial extension is calculated when the component is used in an element.

The axial deformation measure for the fluid damper component is axial extension. Viscous damper components have no elastic stiffness. The relationship between axial force and axial deformation rate can be nonlinear, as shown in Figure 12.1.

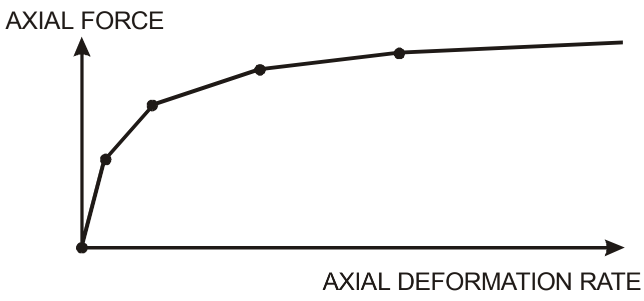



Figure 12.1 Fluid Damper Properties

You can specify up to 6 straight line segments. A linear damper has only one segment. Typically the tangent damping coefficient will decrease as the deformation rate increases, but this is not required. If desired, the properties can be different in tension and compression.

## 12.2 Viscous Bar Elements

Viscous bar elements resist axial force only. Each element must consist of one viscous bar compound component. Each compound component must consist of a fluid damper component in series with an elastic bar component, as shown in Figure 12.2.

Figure 12.2 Viscous Bar Element

## 12.3 P-Δ Effects and Element Loads

P-Δ effects can be specified. PERFORM does not include any element loads.

# 13 BRB Element

*This chapter reviews the component and element for modeling buckling restrained brace (BRB) members.*

## 13.1 BRB Basic Component

The Buckling Restrained Braces basic component is similar to other components in PERFORM. However, it some properties that are more complex than most other components. In particular:

(1) As actual BRB devices are deformed cyclically, tests show that they get progressively stronger. This is sometimes referred to as "isotropic hardening". The PERFORM BRB component allows you to specify this type of hardening. For details of the hysteresis loop see *Chapter 1, PERFORM Hysteresis Loop, Section 1.3.*

(2) For most inelastic components the deformation capacity is specified as the maximum allowable deformation. However, the deformation capacity of a BRB devices may be better defined in terms of accumulated deformation (the sum of tension and compression plastic excursions). The PERFORM BRB component allows you to specify both maximum and accumulated deformation capacities, and to set up corresponding limit states.

If you use the isotropic hardening feature, you may have to experiment with the hardening parameters to match known experimental results. The best way to do this is to use the **Plot Loops** feature in the **Component Properties** task.

## 13.2 BRB Compound Component

A BRB member in a frame structure typically consists of the BRB itself, a length of steel member that is stronger than the BRB, and stiff end regions consisting of gusset plates and the beam-column joint region. The BRB Compound Component allows you to model such a member as a single component, so that each BRB member, from joint center to joint center, can be modeled using a single element. A BRB Compound Component has three parts, as follows.

(1) A BRB basic component.
(2) An Elastic Bar basic component.
(3) A stiff end zone, with a specified length and a cross section area that is a multiple of the elastic bar area. This end zone accounts for the gusset plates, etc. at both ends of the member.

Note that a BRB Compound Component is a bar type component that resists axial force only, and is assumed to have zero bending and torsional stiffnesses.

## 13.3 P-Δ Effects and Element Loads

P-Δ effects can be specified. PERFORM does not include any element loads.

# 14 Rubber Type Seismic Isolator Element

*This chapter reviews the rubber type seismic isolator component and element.*

## 14.1 Isolator Component

### 14.1.1 Basic Action-Deformation Relationship

A rubber type isolator component has bearing and shear properties as shown in Figure 14.1.

The behavior in bearing is elastic, with different stiffnesses in tension and compression if needed. The behavior in shear is trilinear with optional stiffening at large displacements. There is no strength loss or stiffness degradation in shear.

The behavior in shear is independent of the bearing force.

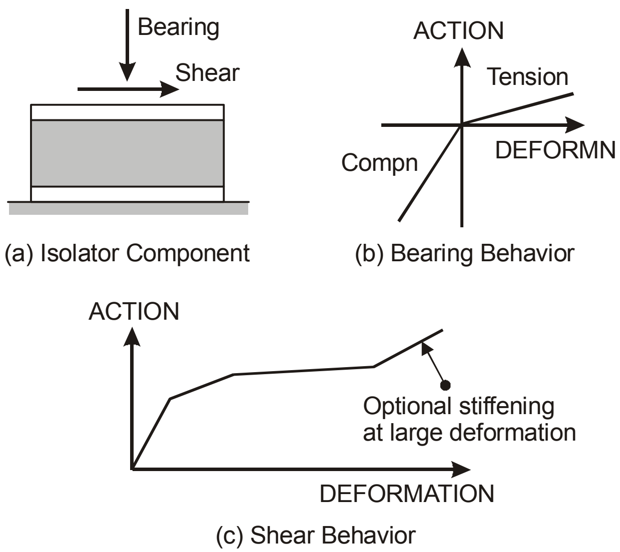



Figure 14.1 Seismic Isolator Properties

## 14.1.2 Isolator Axes

An isolator has three axes, as shown in Figure 14.2. Shear forces act along Axes 1 and 2. Axis 3 is along the bearing direction.

In most cases Axes 1, 2 and 3 will be parallel to the global H1, H2 and V axes, but the axes can be inclined if desired. This is considered later in this chapter.

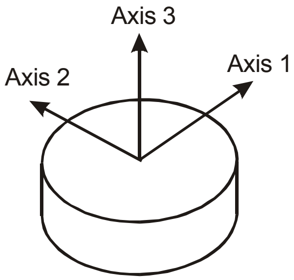



Figure 14.2 Isolator Axes

## 14.1.3 Biaxial Shear Behavior, Symmetrical Case

If a rubber type isolator component has symmetrical shear properties (i.e., the same properties for shear in any direction) it is assumed to be a typical rubber or lead-rubber isolator with a circular or square shape. For shear forces that are inclined to Axes 1 and 2 the effective shear force and shear deformation are given by:

$$F = \sqrt{F1^2 + F2^2} \tag{14.1}$$

and

$$D = \sqrt{D1^2 + D2^2} \tag{14.2}$$

where F and D are the effective shear forces and displacements, and F1, F2, D1, D2 are the components along Axes 1 and 2. This is the same as a circular yield surface. The relationship between F and D is as shown in Figure 14.1.

### 14.1.4 Biaxial Shear Behavior, Unsymmetrical Case

If a rubber type isolator component has unsymmetrical shear properties (i.e., different properties for shear along Axes 1 and 2) it is assumed to consist of two separate uniaxial devices, one along Axis 1 and one along Axis 2. The relationship for each device is as shown in Figure 14.1(c).

You must specify two action-deformation relationships, one for each device. There is no interaction between these devices.

### 14.1.5 Bearing Stiffness

Rubber–type isolators are very stiff in compression, but not rigid. It is not a good idea to specify an extremely large bearing stiffness (such as $10^{10}$), because it can lead to problems with numerical sensitivity. Always specify a realistic value for the stiffness.

### 14.1.6 Capacities

You can specify shear displacement capacities (for use in deformation limit states) and bearing force capacities (for use in strength limit states).

## *14.2 Isolator Elements*

### 14.2.1 General

Each element consists of one rubber type isolator component.

### 14.2.2 Element Orientation

This may seem a bit complex, but in almost all cases you can use the default orientation. You do, however, have a lot of flexibility for unusual cases.

Each seismic isolator element connects two nodes (Node I and Node J), and consists of one isolator component plus two rigid links. Figure 14.3 shows an isolator element.

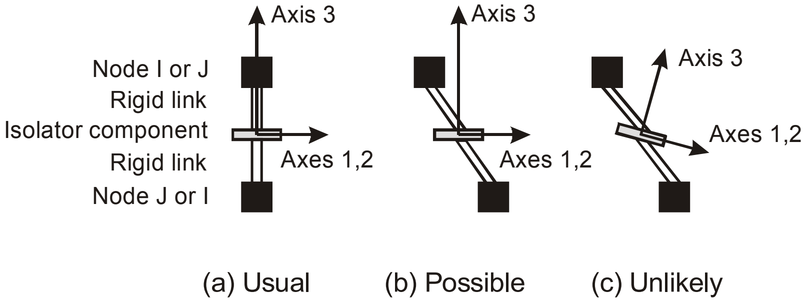



Figure 14.3 Isolator Element

Nodes I and J will usually be located at the axes of the beams above and below the isolator. Hence, the length I-J will usually be of the order of the isolator thickness plus the beam depth. The line I-J will usually be vertical, but this is not required.

An important point is that the orientation of Axes 1, 2 and 3 for the isolator component do not depend on the locations of nodes I and J. Instead, you must specify the orientations of Axes 1, 2 and 3 directly in the global H1, H2, V coordinate system.

In almost every case, Axis 3 will be vertical (i.e., the shear plane of the isolator will be horizontal) and Axes 1 and 2 will be parallel to the global H1 and H2 axes. In this case you do not need to specify any orientation angles (they are all zero).

If Axis 3 is vertical but Axes 2 and 3 are not parallel to H1 and H2, you must specify a rotation angle in plan from Axis H1 to Axis 2. In the rare cases where Axis 3 is not vertical (i.e., the shear plane is not horizontal), you must specify the orientations of Axes 1, 2 and 3 using a plan angle, a tilt angle, and in complex cases possibly a twist angle.

Figure 14.3(a) shows the most common case. In this case the line I-J between nodes I and J is vertical and the shear plane for the isolator is horizontal. Figure 14.3(b) shows a case where the shear plane is horizontal but the line joining nodes I and J is not vertical. Figure 14.3(c) shows a possible, but unlikely, case where the shear plane is inclined.

There is one additional consideration, which PERFORM takes care of automatically. There are to possibilities, as follows.

(1) If the line I-J has a positive projection on Axis 3 (i.e., if node I is below node J for the case with Axis 3 vertical), PERFORM connects node I to the bottom of the isolator and node J to the top. Then, when node I moves along Axis 3 relative to node J, the isolator bearing force is compression.
(2) Conversely, if the line I-J has a negative projection on Axis 3 (i.e., node I is above node J for the case with Axis 3 vertical), PERFORM connects node I to the top of the isolator and node J to the bottom. Then, when node I moves along Axis 3 relative to node J, the isolator bearing force is in tension.

This means that the line I-J can not be perpendicular to Axis 3 (if it is, the projection is zero and PERFORM cannot assign nodes I and J).

### 14.2.3 Bending and Torsion

The isolator component is placed at a specified point in the element, with the default location at the element midpoint. You can specify the location when you define the isolator component (not when you define isolator elements).

Since the isolator component has no stiffness in bending, the bending moments at the location of this component are zero. However, the bending moments are not zero at the nodes. For example, for the case in Figure 14.3(a), the bending moment about Axis 1 at the node is the shear force along Axis 2 multiplied by the rigid link length. There must be other elements that connect to the nodes and provide bending moment resistance, otherwise the isolator will be ineffective and the structure may be unstable.

The isolator component also has no torsional stiffness. Hence, the torsional moments in the element are zero.

## *14.3 P-Δ Effects*

### 14.3.1 General

Typically the shear force-deformation relationships for rubber type isolators are obtained from tests. The isolator is subjected to bearing force during the test, and hence the measured shear force-deformation relationship includes some geometric nonlinearity effects. The F-D relationship that is specified for the isolator should be based on tests, and hence should account for these effects.

There can, however, be significant P-Δ moments that are exerted on the members that connect to an isolator. These moments usually need to be considered when designing those members. This section considers these moments and how they can be calculated.

### 14.3.2 Dominant P-Δ Effect

Figure 14.4(a) shows, diagrammatically, a rubber type isolator.

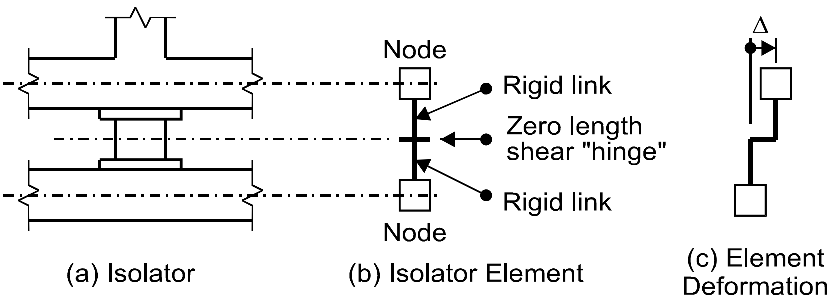



Figure 14.4 Isolator Model

The analysis model is shown in Figure 14.4(b). This element connects to two nodes and consists essentially of an inelastic shear hinge connected to the nodes by rigid links. Usually the upper node connects to the isolated superstructure and the lower node connects to the foundation, although this is not essential.

The most common mode of deformation for the element is shown in Figure 14.4(c). In this case the upper node moves horizontally relative to the lower node. Since the superstructure and foundation are both likely to be very stiff, there is usually little or no rotation of the nodes.

Figure 14.5 shows the end forces for an isolator element when the deformation in Figure 14.4(c) is imposed. These forces are the end forces that are in equilibrium with the element internal forces, which are the bearing and shear forces at the hinge. The bending moment at the hinge is zero.

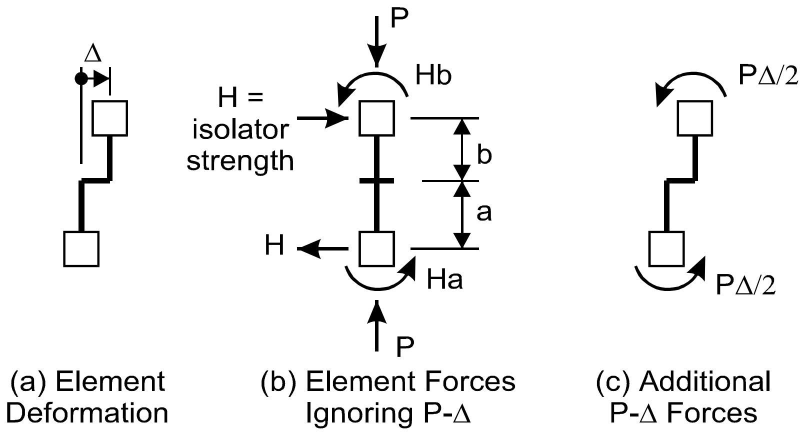



Figure 14.5 Element End Forces

Figure 14.5(a) shows the element deformation. The bearing force is P and the horizontal shear force is H.

Figure 14.5(b) shows the end forces when P-Δ effects are ignored. In this case Δ is assumed to be zero (i.e., negligibly small) for calculating the end forces. Hence, the bending moment at the top of the isolator element (which must be resisted by the superstructure) is the shear force multiplied by the length of the upper rigid link. Similarly, the bending moment at the bottom of the element (which must be resisted by the foundation) is the shear force multiplied by the length of the lower rigid link. These forces are for the case where Δ is increasing.

Figure 14.5(c) shows the additional end forces when P-Δ effects are considered. In this case Δ is not assumed to be negligibly small. Hence, there are additional bending moments at both the bottom and top of the element, each equal to the bearing force multiplied by one half of the horizontal deformation.

### 14.3.3 Other Deformations

Figure 14.5 shows only the element deformation when the node rotations are zero. If either the superstructure or the foundation has significant flexibility, there may be significant rotations at the nodes. In this case there may be deformations such as those shown in Figure 14.6, with additional P-Δ effects.

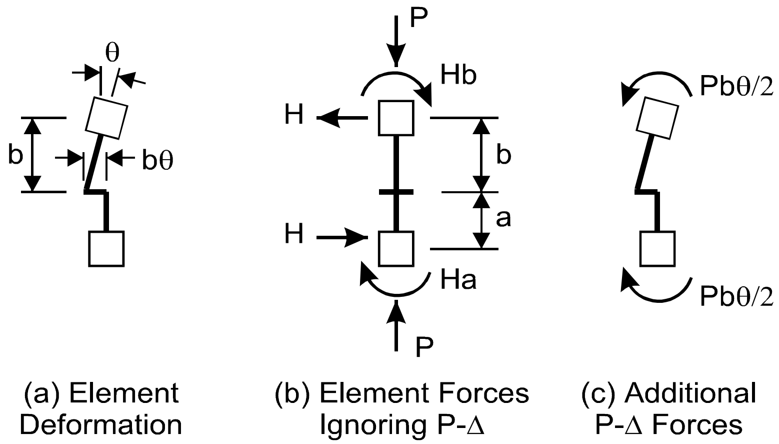



Figure 14.6 Another Possible Deformation for a Rubber Type Isolator Element

The PERFORM element takes deformations of this type into account. As a general rule, the points to which the isolator connects will be very stiff rotationally, so these deformations will usually be small, and the main P-Δ effect is that shown in Figure 14.5(c).

### 14.3.4 Node Numbering

For rubber type isolators it does not matter which element end is End I and which is End J, whether P-Δ effects are considered or ignored. That is, it does not matter if the element I-J direction is upwards or downwards.

### 14.3.5 Should P-Δ Effects be Considered?

If you ignore P-Δ effects in the analysis, the analysis model is simpler, but you must account for P-Δ moments on the members by separate calculations. If you consider P-Δ effects in the analysis, the P-Δ moments on the members are calculated directly by PERFORM, but the analysis model is more complex. You might like to try it both ways, and confirm that the two methods are consistent. Otherwise it is a matter of personal preference.

# 15 Friction Pendulum Isolator Element

*This chapter reviews the friction pendulum seismic isolator component and element.*

## 15.1 Friction-Pendulum Isolator Component

### 15.1.1 Bearing Behavior

Friction-pendulum isolators are very stiff in compression, but not rigid. It is not a good idea to specify an extremely large bearing stiffness (such as $10^{10}$), because it can lead to problems with numerical sensitivity. Always specify a realistic value for the stiffness.

Usually a friction pendulum component will be able to uplift in tension (although such uplift may not be permissible). This means a zero or small stiffness in tension.

If there is uplift, PERFORM currently assumes that there is zero friction. The "cylindrical rail" isolator is a variation on the friction pendulum concept that does act in uplift. It is possible to model this isolator using friction pendulum components, although it is admittedly rather complex to do so. For details see later in this chapter.

### 15.1.2 Action-Deformation Relationship in Shear

The shear behavior is as shown in Figure 15.1. The key aspects of this behavior are as follows.

(1) Before it slips the isolator has a stiffness K0. This may be large, but do not specify unrealistically high values. A real isolator can deform significantly before it begins to slip.

(2) The slip force depends on the bearing force in compression and the friction coefficient. If the bearing force is tension, the slip force is zero.

(3) The "hardening" stiffness is equal to the current bearing force divided by the radius of the slip surface.

(4) If desired, the friction coefficient can depend on the slip rate. You can specify up to five slip rates and corresponding friction coefficients. These define a multi-linear relationship between slip rate and friction coefficient.

(5) If the displacement exceeds the distance to the slip surface boundary, DS, the stiffness is increased, by adding a "boundary" stiffness, KS.

(6) In an actual friction-pendulum isolator, as the isolator slides horizontally it also moves vertically upwards, because the sliding surface is curved. In the PERFORM element this vertical displacement is not considered. Computationally the vertical displacement is a true large displacements effect. PERFORM uses P-Δ theory to account for surface curvature, not true large displacements theory. P-Δ theory gives the correct hardening stiffness, but assumes zero vertical displacement.

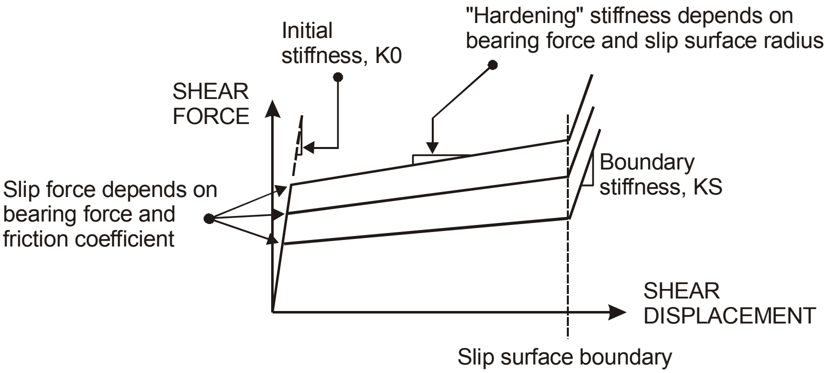



Figure 15.1 Behavior of Friction-Pendulum Isolator

## 15.1.3 Assumptions for Push-Over Analysis

For dynamic earthquake analysis, PERFORM accounts for changes in the bearing force, and adjusts the slip force and hardening stiffness accordingly. However, for push-over analysis PERFORM assumes that the bearing force is constant, equal to the bearing force at the end of the gravity load analysis. The slip force and hardening stiffness are thus constant for a push-over analysis. Note that it is essential to precede an earthquake or push-over analysis with a gravity analysis, otherwise the bearing force is zero, and hence the slip force is also zero.

### 15.1.4 Assumptions for Gravity Analysis

Special assumptions are also made for gravity load analysis. After the gravity load has been applied, it is usually assumed that the friction isolator elements will have vertical bearing forces but no horizontal shear forces. However, in a gravity load analysis it is possible that there will be horizontal displacements of the structure, and hence it is possible that there will be shear deformations in isolator elements. These deformations will be small, but when they are multiplied by the initial stiffness, K0, small deformations could correspond to significant shear forces. Hence, if the shear stiffnesses of friction pendulum isolators are assumed to be K0 for gravity load analysis, the shear forces in the isolators may be significant. (Note that since gravity load cases can have only vertical loads, the resultant of any horizontal forces in the isolators will be zero.)

The shear forces could be made zero by assuming zero shear stiffness for gravity load analysis. In this case, however, the complete structure would typically have zero stiffness in the horizontal direction, and hence be unstable.

PERFORM requires that you specify a K0 value to be used for gravity load analysis. Usually you will specify a value that is small enough to assure near zero shear forces in the isolators but large enough to avoid instability. In addition, regardless of what shear deformations and forces are calculated in the isolators for gravity load, PERFORM assumes that they are zero (i.e., for earthquake and push-over analyses, PERFORM initializes the isolators to zero shear force and deformation).

### 15.1.5 Biaxial Shear Behavior

For biaxial shear forces and deformations, a friction pendulum isolator has a slip surface that is analogous to a yield surface for an elastic-perfectly-plastic hinge with biaxial moments. The difference is that in a friction pendulum isolator the size of the slip surface is constantly changing, whereas the yield surface for an elastic-perfectly-plastic hinge has a fixed size.

For a symmetrical isolator, with the same radii along Axes 1 and 2, the yield surface is a circle. For an unsymmetrical isolator, with different radii. The yield surface is an ellipse. Also, the "hardening" stiffnesses are different along the two axes.

### 15.1.6 Boundary Behavior

The shear deformations of a friction pendulum isolator should usually not be large enough to reach the slip surface boundary, and you may choose to specify a zero value for the boundary stiffness, KS. However, you can specify a stiffness if you wish.

For a symmetrical isolator, PERFORM assumes that the slip surface boundary is circular, so that the distance to the boundary is DS in all directions. If the slip deformation in any direction exceeds DS, a boundary stiffness, KS, is added, along the normal to the circular boundary. There is no added stiffness along the tangent to the boundary. If the boundary is like a circular wall, this corresponds to zero friction between the isolator and the wall. If the isolator slides around the boundary, the direction of the boundary stiffness is adjusted so that the force is always normal to the boundary.

For an unsymmetrical isolator the boundary is assumed, to be a rectangle (not an ellipse). In this case the boundary stiffness is provided by two separate springs, one along Axis 1 of the isolator and one along Axis 2. Again. there is no friction.

### 15.1.7 Time Step for Dynamic Analysis

Because the slip force is affected by both the bearing force and the slip rate, friction pendulum isolators can be sensitive numerically. We recommend, therefore, that for dynamic analysis you use a smaller time step than you would use for a comparable structure with no isolators (we suggest one quarter as long). You should probably experiment with different time steps.

### 15.1.8 Capacities

You can specify shear displacement capacities (for use in deformation limit states) and bearing force capacities (for use in strength limit states).

## 15.2 Isolator Elements

### 15.2.1 Element Orientation

This may seem a bit complex, but in almost all cases you can use the default orientation. You do, however, have a lot of flexibility for unusual cases.

Each seismic isolator element connects two nodes (Node I and Node J), and consists of one isolator component plus two rigid links. Figure 15.2 shows an isolator element.

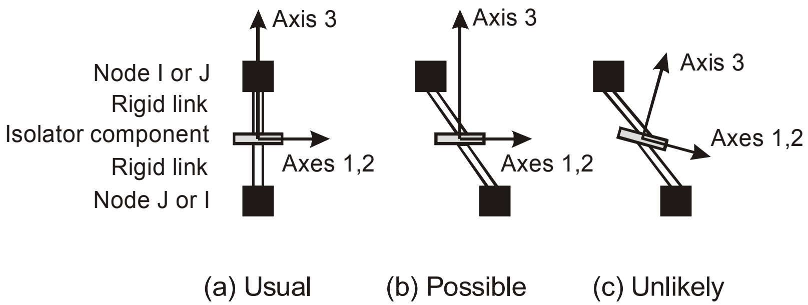



Figure 15.2 Isolator Element

Nodes I and J will usually be located at the axes of the beams above and below the isolator. Hence, the length I-J will usually be of the order of the isolator thickness plus the beam depth. The line I-J will usually be vertical, but this is not required.

An important point is that the orientation of Axes 1, 2 and 3 for the isolator component do not depend on the locations of nodes I and J. Instead, you must specify the orientations of Axes 1, 2 and 3 directly in the global H1, H2, V coordinate system.

In almost every case, Axis 3 will be vertical (i.e., the bearing surface of the isolator will be horizontal) and Axes 1 and 2 will be parallel to the global H1 and H2 axes. In this case you do not need to specify any orientation angles (they are all zero).

If Axis 3 is vertical but Axes 2 and 3 are not parallel to H1 and H2, you must specify a rotation angle in plan from Axis H1 to Axis 2. In the rare cases where Axis 3 is not vertical (i.e., the bearing surface is not horizontal), you must specify the orientations of Axes 1, 2 and 3 using a plan angle, a tilt angle, and in complex cases possibly a twist angle.

Figure 15.2(a) shows the most common case. In this case the line I-J between nodes I and J is vertical and the shear plane for the isolator is horizontal. Figure 15.2(b) shows a case where the bearing surface is horizontal but the line joining nodes I and J is not vertical. Figure 15.2(c) shows a possible, but unlikely, case where the bearing surface is inclined.

There is one additional consideration, which PERFORM takes care of automatically. There are to possibilities, as follows.

(1) If the line I-J has a positive projection on Axis 3 (i.e., if node I is below node J for the case with Axis 3 vertical), PERFORM connects node I to the bottom of the isolator and node J to the top. Then, when node I moves along Axis 3 relative to node J, the isolator bearing force is compression.
(2) Conversely, if the line I-J has a negative projection on Axis 3 (i.e., node I is above node J for the case with Axis 3 vertical), PERFORM connects node I to the top of the isolator and node J to the bottom. Then, when node I moves along Axis 3 relative to node J, the isolator bearing force is in tension.

This means that the line I-J can not be perpendicular to Axis 3 (if it is, the projection is zero and PERFORM cannot assign nodes I and J).

### 15.2.2 Bending and Torsion

The isolator component is placed at a specified point in the element, with the default location at the element midpoint. You can specify the location when you define the isolator component (not when you define isolator elements).

Since the isolator component has no stiffness in bending, the bending moments at the location of this component are zero. However, the bending moments are not zero at the nodes. For example, for the case in Figure 10.4(a), the bending moment about Axis 1 at the node is the shear force along Axis 2 multiplied by the rigid link length. There must be other elements that connect to the nodes and provide bending moment resistance, otherwise the isolator will be ineffective and the structure may be unstable.

The isolator component also has no torsional stiffness. Hence, the torsional moments in the element are zero.

## 15.3 P-Δ Effect in Friction Pendulum Isolator

### 15.3.1 General

There can be significant P-Δ moments that are exerted on the members that connect to an isolator. These moments usually need to be considered when designing those members. This section considers these moments and how they can be calculated.

### 15.3.2 Typical Isolator

Figure 15.3(a) shows, diagrammatically, a friction-pendulum isolator.

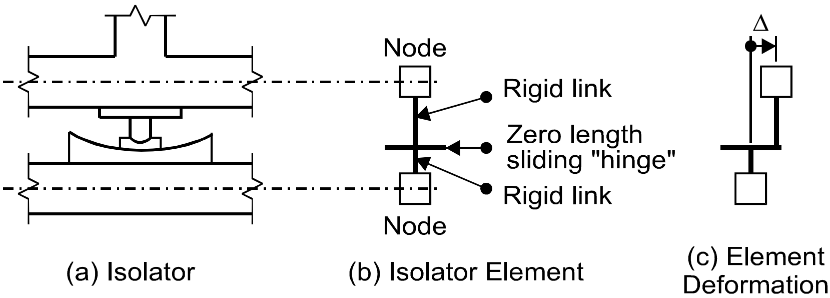



Figure 15.3 Friction-Pendulum Element

The analysis model (a friction-pendulum isolator element) is shown in Figure 15.3(b). This element connects to two nodes and consists of a sliding surface (in effect, a shear hinge) connected to the nodes by rigid links. Usually the sliding surface is horizontal, although this is not essential. Usually the upper node connects to the isolated superstructure and the lower node connects to the foundation, although again this is not essential.

The most common mode of deformation for the element is shown in Figure 15.3(c). In this case the upper node moves horizontally relative to the lower node. Since the superstructure and foundation are both likely to be very stiff, there is usually little or no rotation of the nodes.

Figure 15.4 shows the end forces for an isolator element when the deformation in Figure 15.3(c) is imposed. These forces are the end forces that are in equilibrium with the element internal forces, which for this element are the bearing and friction forces at the sliding surface. The bending moment is zero at this surface.

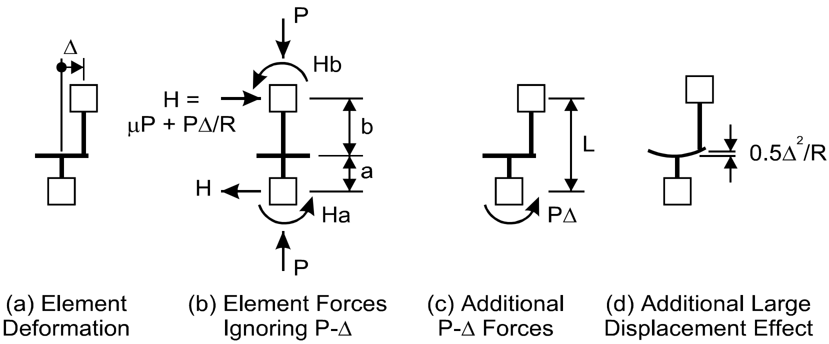



Figure 15.4 Element End Forces

Figure 15.4(a) shows the element deformation. The bearing force is P, the horizontal friction force is H, the friction coefficient is μ, and the radius of the slip surface is R.

Figure 15.4(b) shows the end forces when P-Δ effects are ignored (these are the forces if Δ is currently increasing – H is different if Δ is decreasing). Ignoring P-Δ effects is the same as assuming that Δ is zero (i.e., negligibly small) for the purposes of calculating the end forces. Hence, the bending moment at the top of the isolator element (which must be resisted by the superstructure) is the horizontal slip force multiplied by the length of the upper rigid link. Similarly, the bending moment at the bottom of the element (which must be resisted by the foundation) is the horizontal slip force multiplied by the length of the lower rigid link.

Figure 15.4(c) shows the additional end forces when P-Δ effects are considered. In this case Δ is not assumed to be negligibly small. Hence, there is an additional bending moment at the bottom of the element, equal to the bearing force multiplied by the horizontal deformation. This moment can be substantial. For example, if the coefficient of friction is 0.1, and Δ is equal to one half the element height, the P-Δ moment is about 10 times the moment due to the horizontal slip force.

### 15.3.3 True Large Displacements

Figure 15.4(d) shows an additional effect. As the isolator deforms, the slider moves upwards, as well as outwards, on the dish. It is this upwards movement that provides the post-slip “hardening” stiffness for the isolator. This is also a P-Δ effect. However, it is considered in the force-deformation relationship for the isolator, so it is accounted for even if P-Δ effects are not considered. However, the upwards displacement is not considered in the analysis model, whether or not P-Δ effects are considered. This is a large displacement effect. PERFORM considers only P-Δ effects for this element, not true large displacement effects.

### 15.3.4 Inverted Isolator

For the configuration in Figure 15.3, with the dish connected to the foundation, there is a P-Δ moment only on the foundation. However, if the isolator is inverted, with the dish attached to the superstructure at the top end of the element, there is a P-Δ moment only on the superstructure. This is shown in Figure 15.5.

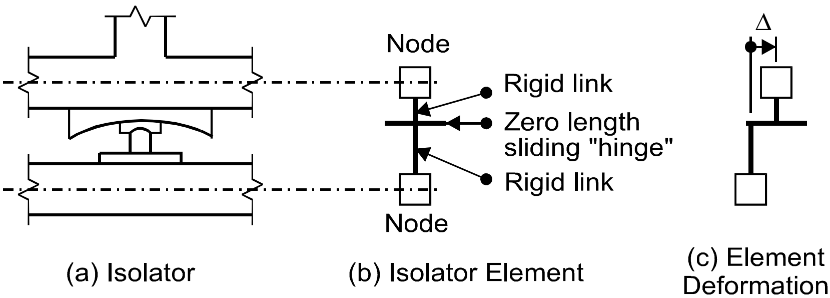



Figure 15.5 P-Δ Effect for Inverted Isolator

### 15.3.5 Other Deformations

Figures 15.3 and 15.4 show only element deformations when the node rotations are zero. If either the superstructure or the foundation has significant flexibility, there may be significant rotations at the nodes. In this case there may be deformations such as those shown in Figure 15.6, with additional P-Δ effects.

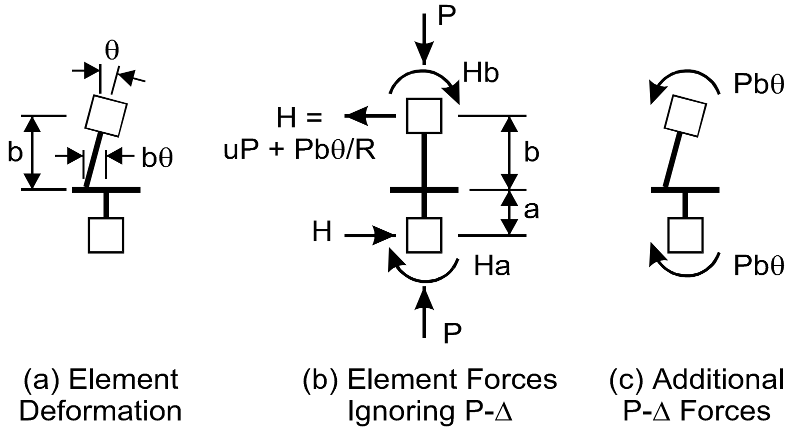



Figure 15.6 Possible Deformation for an Isolator Element

The PERFORM element takes deformations of this type into account. As a general rule, the points to which the isolator connects should be very stiff rotationally, otherwise the isolator may not operate correctly.

### 15.3.6 Node Numbering

If P-Δ effects are ignored, it does not matter which element end is End I and which is End J. That is, it does not matter if the element I-J direction is upwards or downwards. However, if P-Δ effects are considered the ordering of the I-J ends is very important. Specifically, the I end is always the dish end of the element. For the deformations shown in Figures 15.3 and 15.4 there is a P-Δ moment at End I, but not at End J.

Hence, for the usual configuration the element I end must be the lower end. For the inverted configuration the I end must be the upper end.

## *15.4 Should P-Δ Effects be Considered?*

If you ignore P-Δ effects in the analysis, the analysis model is simpler, but you must account for P-Δ moments on the members by separate calculations. If you consider P-Δ effects in the analysis, the P-Δ moments on the members are calculated directly by PERFORM, but the analysis model is more complex. You might like to try it both ways, and confirm that the two methods are consistent. Otherwise it is a matter of personal preference.

## *15.5 Cylindrical Rail Isolator*

### 15.5.1 Isolator Device

Figure 15.7 shows a “cylindrical rail” friction-pendulum isolator, which operates in tension (uplift) as well as compression.

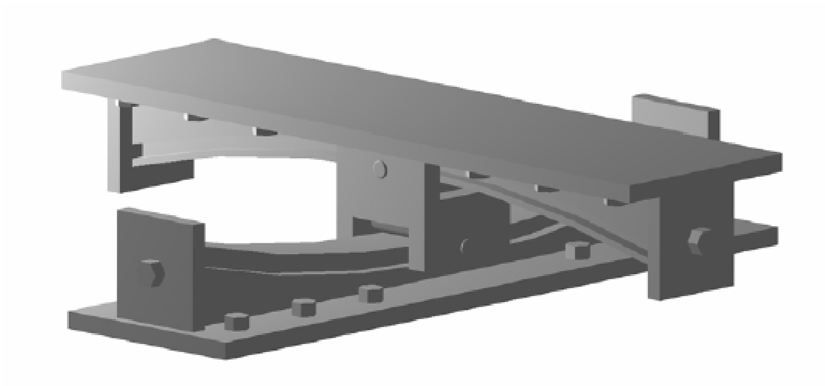


Figure 15.7 Cylindrical Rail Isolator

PERFORM does not yet have a component that will model this type of isolator directly, and in the current version it is necessary to use the existing friction pendulum component. This component is intended for modeling a typical isolator that has a spherical or ellipsoidal dish, and operates in compression only. It can, however, be used to model cylindrical rail bearings.

### 15.5.2 Isolator Model for Compression Only

For a cylindrical rail isolator that acts only in compression, the PERFORM model using the existing friction isolator component is shown in Figure 15.8.

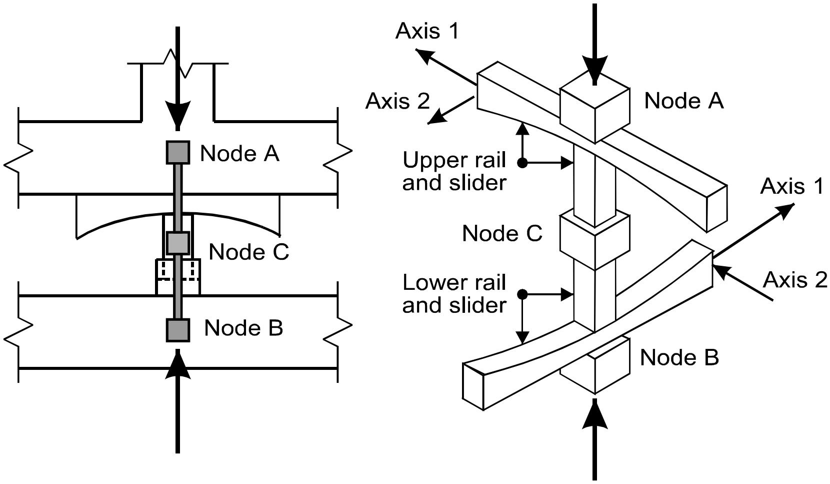



Figure 15.8 Model for Isolator in Compression

This model has two elements and three nodes. The upper element models the upper rail, and is oriented so that its Axis 1 is along the rail. The lower element models the lower rail, and is also oriented so that its Axis 1 is along the rail. In both elements, sliding can take place along Axis 1, but stops are used to prevent sliding along Axis 2. Nodes A and B connect to the superstructure and foundation, respectively. The displacements of Node C are the displacements of the slider, between the two rails.

Some details of the modeling procedure are as follows.

(1) Specify the upper element with End I at Node A (the rail end) and End J at Node C. Specify the lower element with End I at Node B (again at the rail end) and End J at Node C. This is in the **Elements** task.
(2) Orient the upper element so that its Axis 1 is along the upper rail, and orient the lower element so that its Axis 1 is along the lower rail. This is in the **Elements** task.
(3) Specify a Friction Pendulum Isolator component that is unsymmetrical. Specify the rail radius along Axis 1, and zero radius (i.e. no curvature) along Axis 2. Along Axis 2 specify a large KS stiffness and a small DS deformation, to model a stop along this axis. Do not specify an "astronomical" stiffness, as this can lead to numerical sensitivity problems. A stiffness KS = 1000 kips/inch or 200 KN/mm is very stiff. (The initial stiffness, K0, should also be a realistic value). A gap DS = 0.01 inches or 0.25 mm should be

reasonable (and is probably smaller than the slack in the actual device). This is in the **Component Properties** task.

(4) Assign the same component to both elements. This is in the **Elements** task.

(5) The elements provide no rotational stiffness for Node C. Hence, restrain all three rotations (about axes H1, H2 and V) at this node. This is in the **Nodes** task.

### 15.5.3 Isolator Model for Tension Only

Although this situation is unlikely to arise in practice, consider a cylindrical rail isolator that acts only in tension. This can be modeled using two friction isolator elements. However, since the actual isolator acts in tension and each isolator element acts only in compression, it is necessary to invert the elements so that they act in compression under uplift forces. The actual isolator and the analysis model are shown in Figure 15.9.

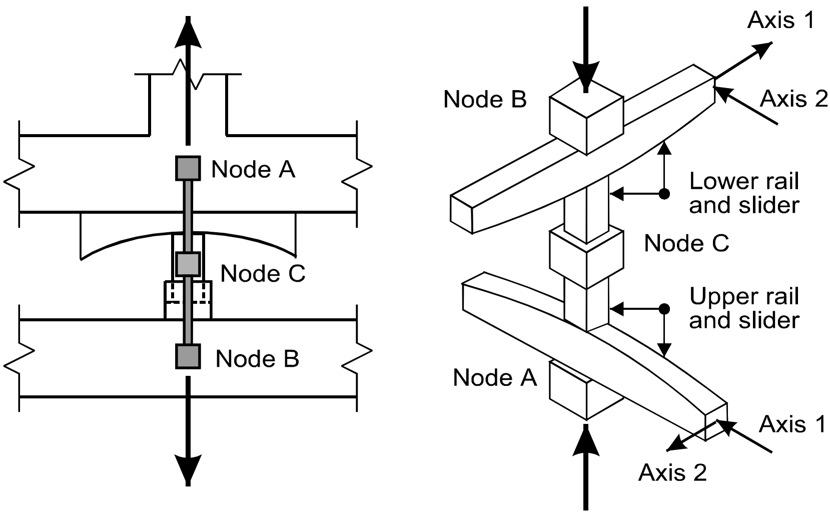



Figure 15.9 Model for Isolator in Tension

The node that connects to the superstructure is now the lower node, so that when the superstructure lifts up the elements go into compression. Also, the curvatures of the rails are now negative (when the actual isolator is in tension, there is a negative pendulum effect). Again, the elements allow sliding along Axis 1, and have stops along Axis 2.

Except for the fact that the nodes are not in the correct locations, this model captures the behavior of a cylindrical rail isolator in tension. As shown in the next section, it can be combined with the compression model to obtain a model that considers both compression and tension.

### 15.5.4 Isolator Model for Compression and Tension

An isolator that acts in both compression and tension can be modeled by combining the models in Figures 15.8 and 15.9. The resulting model has five nodes and four elements, as shown in Figure 15.10.

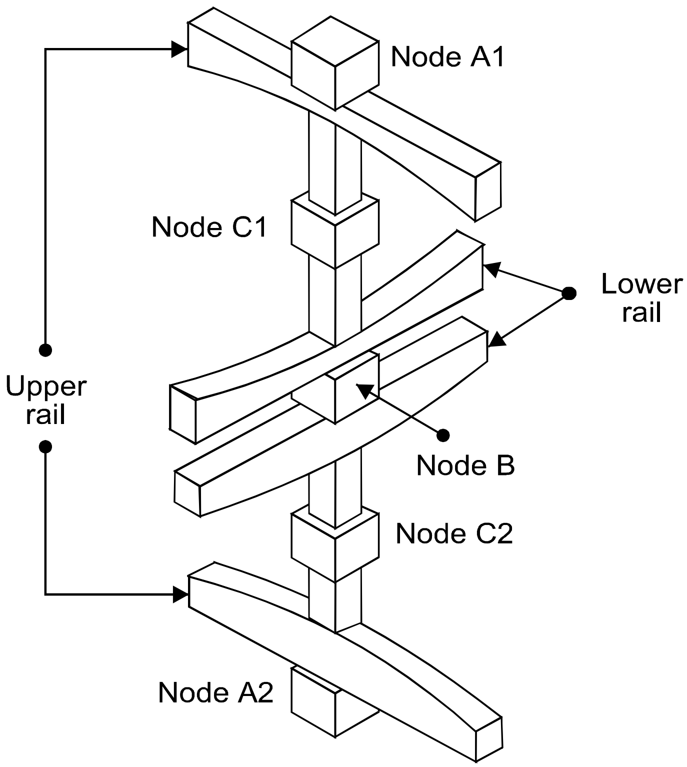



Figure 15.10 Model for Compression and Tension

In this model, the two upper elements act in compression but lift off in tension. When this happens the two lower elements become active. Two key points of this model are as follows.

(1) The nodes A1 and A2 are actually the same node, connected to the superstructure. This is achieved in the model by constraining these nodes to have identical displacements, which makes them effectively the same node.
(2) The nodes C1 and C2 are also the same node, representing the motion of the slider between the rails. These nodes are also constrained to have identical displacements.

The procedure for setting up this model is as follows.

(1) Define nodes for A1 and B at the superstructure and foundation locations. Define a node for A2 below B, so that lengths A1-B and A2-B are the same. Define nodes for C1 and C2 midway between the other nodes. This is in the **Nodes** task.
(2) Restrain the H1, H2 and V rotations at C1 and C2. This is in the **Nodes** task.
(3) Constrain all 6 displacements at A1 and A2 to be identical, effectively making them the same node. It is important to note that this requires an Equal Displacement constraint, not a Rigid Link constraint. One constraint of this type is required for each isolator in the structure. This is in the **Nodes** task.
(4) Constrain the H1, H2 and V displacements at C1 and C2 to be identical, effectively making them the same node for horizontal displacements. One constraint of this type is required for each isolator in the structure. This is in the **Nodes** task.
(5) Specify four friction pendulum elements, with End I at the rail end for each element, and End J at Node C1 or C2. This is in the **Elements** task. Since isolator elements are short, it is usually a good idea to put the compression and tension isolators in separate element groups. This makes it easier to select them graphically, allows you to see the dissipated energies for compression and tension separately in energy balance graphs, and enables you to set up separate limit states for compression and tension isolators.
(6) Orient the two elements for the upper rail so that Axis 1 is along the upper rail in the actual structure. Orient the two elements for the lower rail so that Axis 1 is along the lower rail. This is in the **Elements** task.
(7) For the compression elements, specify a Friction Pendulum Isolator component that is unsymmetrical. Specify the rail radius along Axis 1, and zero radius (i.e. no curvature) along Axis 2. Along Axis 2 specify a large KS stiffness and a small DS deformation, to model a stop along this axis. Do not specify an “astronomical” stiffness, as this can lead to numerical sensitivity problems. A stiffness KS = 1000 kips/inch or 200 KN/mm is very stiff. The initial stiffness, K0, should also be a realistic value. A gap DS = 0.01 inches or 0.25 mm should be

reasonable (and is probably smaller than the slack in the actual device). This is in the **Component Properties** task.

(8) For the tension elements, specify a second Friction Pendulum Isolator component. Specify a negative rail radius along Axis 1, and zero radius (i.e. no curvature) along Axis 2. Specify the same KS and DS values as for the compression components. The friction coefficients may be the same as for the compression components (this information must be provided by the manufacturer). If so, to specify this tension component you can use SaveAs to copy the compression component and change only the radius. This is in the **Component Properties** task.

(9) Assign the first (compression) component to the two compression elements, and the second (tension) compression component to the two tension (inverted) elements. This is in the **Elements** task.

### 15.5.5 P-Δ Effects

If you set up the model as described above, P-Δ effects will be considered, for both compression and uplift forces. You can choose to include or ignore these effects. If you include P-Δ effects, be sure to check these effects using a small structure, to satisfy yourself that the model is correct.

### 15.5.6 Integration Time Step and Energy Balance

Friction pendulum components are numerically sensitive. It is wise to use a shorter time step than usual for dynamic analysis. The energy balance error is also likely to be larger than for a typical structure. We believe an error smaller than 5% should be OK. If you get a larger error, reduce the time step.

# 16 Support Spring Element

*The support spring element models linear elastic supports. For inelastic or nonlinear elastic supports you must use bar elements.*

*A support spring element can have translational and/or rotational stiffnesses, can have up to a 6x6 stiffness matrix, and can be inclined to the H1-H2-V axes. Strength capacities and strength based limit states can be specified.*

*Support spring elements can be used to impose displacements at support points. This is done by making the elements very stiff and specifying that they have initial deformations (similar to thermal expansions). Imposed displacements can be included in any static Gravity load case.*

*This chapter describes the support spring component and element.*

## 16.1 Support Spring Component

Use the **Component Properties** task and the **Elastic** tab, and choose **Support Spring** component type.

Choose the deformations to be considered, and specify the spring stiffnesses. Diagonal stiffnesses must be positive. Off-diagonal stiffnesses are optional. If you specify off-diagonal stiffnesses, be careful that the stiffness matrix is correct, since PERFORM does not check the matrix for consistency. The element is linear elastic, and the stiffness matrix is always symmetrical.

Resist the temptation to specify “rigid” supports with astronomically large stiffnesses such as $10^{10}$. A steel bar that is 10 inches (25 cm) square and 10 inches (25 cm) long would be extraordinarily stiff in a real structure, but its stiffness is only about $3 \times 10^5$ kips/inch ($5 \times 10^4$ kN/mm). Astronomically large stiffnesses can cause numerical problems, especially if the support axes are rotated relative to the H1-H2-V axes, or if initial deformations are specified to impose support displacements.

Specify strength capacities if you wish. The strength capacities may be symmetrical (equal in the positive and negative directions) or unsymmetrical. If you specify strength capacities, you can define strength limit states.

## 16.2 Support Spring Element

Each support spring element connects to one node and consists of one support spring component. In effect, each element connects the supported node to a point that is fixed in space. Support spring elements have no P-Δ effects. They are plotted as small circles at the supported nodes.

Specify support spring elements in the usual way. The element local axes are Axes 1-2-3. These can be rotated from the global H1-H2-V axes if desired.

## 16.3 Viscous Damping

For dynamic analysis, if the support spring stiffness is very large and βK viscous damping is used, the damping forces can be large. If you use support spring elements to model the soil supports in a foundation, and if you specify realistic stiffnesses, it may be appropriate to include βK viscous damping. As a general rule, however, we suggest ignoring βK damping for support spring elements (when you set up the Element Group, specify zero for the “Scale Factor for Beta-K Damping”).

## 16.4 Element Loads : Initial Deformations

You can specify initial deformations in support spring elements, as element loads. Use the **Load Patterns** task and the **Element Loads** tab to define one or more static load patterns. You can then use these patterns in Gravity load cases to impose displacements on the supported node. A positive initial deformation for, say, Axis 1 translation, will move the supported node in the positive Axis 1 direction.

If the support spring is stiff relative to the structure, the displacement imposed on the node will be very close to the initial deformation (the displacement is the initial deformation minus the spring deformation, which is very small for a stiff spring). As noted earlier, resist the temptation to specify astronomically large stiffnesses.

## *16.5 P-Δ Effects*

Support spring elements do not exert P-Δ effects.

# 17 Deformation Gage Elements

*Deformation D/C ratios can be calculated for all inelastic elements. A possible problem is that there can be strain concentrations, causing large values of the D/C ratios for some elements. If the strain concentrations are localized, it can be unnecessarily conservative to use the D/C ratio for the "worst" element.*

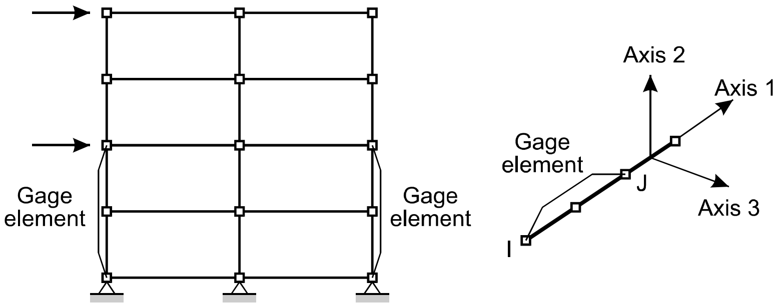
*Deformation gage elements allow you to calculate averaged deformations over a number of elements. Averaged values can be more representative for performance assessment.*

*PERFORM has deformation gage components and elements for axial strain, shear strain and rotation. These elements have no stiffness. Their sole purpose if to measure deformation. This chapter describes the components and elements.*

## 17.1 Purpose

In PERFORM you can specify deformation capacities for single components. In some cases, however, you may want to measure deformations over a number of elements, and specify corresponding limit states. Examples are as follows.

(1) In a wall structure you may want to calculate the average strain over a story height, where there are multiple elements over that height. You can do this with a Strain Gage element, as shown in Figure 17.1(a).
(2) If you have a series of short beam elements that use fiber cross sections, and if you use rotation as the deformation measure, you may want to calculate the rotation over a number of elements. You can do this with a Rotation Gage element, as shown in Figure 17.1(b).




(a) Strain gages over 2 wall elements

(b) Rotation gage over 2 beam elements

Figure 17.1  Strain and Beam Rotation Gages

Deformation gage elements allow you to calculate deformations of these and other types, and to set up corresponding deformation limit states. Deformation gage elements all have zero stiffness and strength. Their sole purpose is to measure deformations.

PERFORM currently has the following Deformation Gage components and elements.

(1) Axial strain gage, as in Figure 17.1(a).
(2) Beam type rotation gage, as in Figure 17.1(b).
(3) Wall type rotation gage, as considered later.
(4) Wall type shear strain gage, as considered later.

## 17.2 Axial Strain Gage

An axial strain gage component has tension and compression strain capacities. The strain demand is the average strain over the gage length.

## 17.3 Beam Rotation Gage

A Rotation Gage component has positive and negative rotation capacities. The rotation demand is the rotation at End J of the element minus the rotation at End I.

A Rotation Gage element has local axes 1, 2 and 3, as shown in Figure 17.1(b), exactly like a beam or column element. The rotation is about Axis 3, which is typically the strong axis for a beam element. A positive rotation for the element corresponds to compression on the +2 side of the element, similar to positive moment about Axis 3 for a beam or column element.

## *17.4 Wall Rotation Gage*

### 17.4.1 Wall Rotation

For shear walls, FEMA 356 specifies the bending deformation capacity as a plastic hinge rotation. Figure 17.2 shows how rotation is measured.

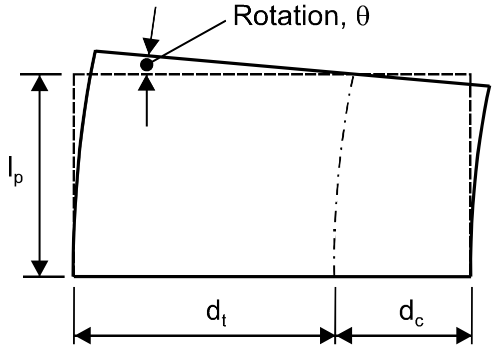



Figure 17.2 Rotation in a Shear Wall

In this figure the rotation, $\theta$, is the total rotation, equal to the plastic rotation, $\theta_p$, plus the yield rotation, $\theta_y$. FEMA 356 specifies the following formula for the yield rotation (FEMA 356 Equation 6-6):

$$\theta_y = \left(\frac{M_y}{E_c I}\right) l_p \qquad (17.1)$$

where $M_y$ = yield moment capacity, $E_c$ = concrete modulus, $I$ = moment of inertia of uncracked section, and $l_p$ = plastic hinge length. The plastic hinge length is the smaller of (a) one half the wall width and (b) the story height.

The value of $\theta_y$ from Equation (17.1) is usually small, and for practical purposes $\theta_p = \theta$ (see also FEMA 356 Figure 6-2).

### 17.4.2 Wall Rotation Gage Element

A wall rotation gage element is a 4-node that calculates the above type of rotation, for a single element or for a number of elements. The element can connect any 4 nodes, for example as shown in Figure 17.3.

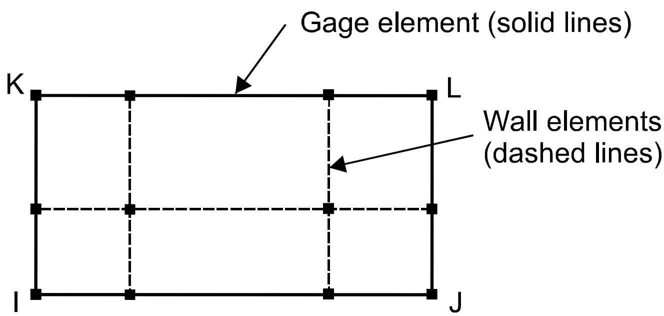



Figure 17.3 Four-Node Rotation Gage Element

The following are some key points.

(1) The element can correspond to a single wall element, or can extend over multiple elements as in Figure 17.3.
(2) The gage rotation is the rotation of edge KL minus the rotation of edge IJ. Edge rotation is positive clockwise in Figure 17.3. Hence, positive gage rotation corresponds to compression on edge JL.
(3) The element must be essentially plane, and should be close to rectangular.
(4) As with strain gage and beam type rotation gage elements, the wall rotation gage does not add any stiffness to the structure.

### 17.4.3 FEMA 356 Rotation Capacity

Shear wall rotation capacities are specified in FEMA 356 Table 6-18. These capacities depend on the shear force in the wall. The wall rotation gage currently has constant rotation capacities that do not depend on the shear force.

### 17.4.4 Rotation Using A Strain Gage

It is also possible to use one or more strain gage elements, and to convert hinge rotation to equivalent strain, using the relationship

$$\varepsilon = \frac{\theta}{l_p} d \qquad (17.2)$$

where $\theta$ = rotation, $l_p$ = plastic hinge length, and $d$ = distance from the neutral axis to the fiber where strain is measured (typically the extreme fiber).

If a strain gage is used to measure rotation, this will usually be a tension gage, at a distance $d_t$ from the neutral axis. Since the neutral axis shifts towards the compression side as a wall section cracks, $d_t$ is usually larger than one half of the wall width.

## *17.5 Wall Shear Strain Gage*

A shear strain gage element is similar to a distortion drift. The element can connect any 4 nodes, for example as shown in Figure 17.4.

The following are some key points.

(1) An element will usually extend over several wall elements, as indicated in Figure 17.4. However, like a distortion drift, a shear strain gage element can connect any four nodes. There do not have to be any wall elements.
(2) The element must be essentially plane, and should be rectangular or close to a rectangle.
(3) The gage deformation is the shear strain, as shown in Figure 17.4.
(4) The sign of the shear strain usually will not matter. However, if the sign is important, the convention is the same as for a distortion

drift. Positive shear strain is as shown in Figure 17.4. This is opposite to the sign convention for shear strain in a wall element.

(5) As with other deformation gage elements, a shear strain gage does not add any stiffness to the structure

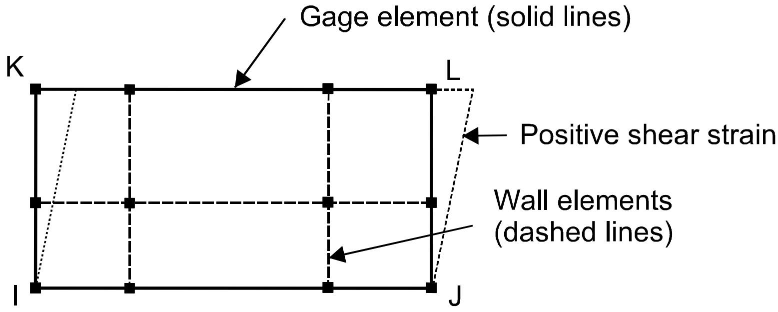



Figure 17.4 Four-Node Shear Strain Gage Element

## 17.6 Procedure

To specify gage components use the **Component Properties** task, to specify gage elements use the **Elements** task, and to calculate demand-capacity ratios, use the **Limit States** task in the usual way.

# 18 Elastic Slab/Shell Element

*The elastic slab/shell element is a 4-node element with membrane (in-plane) and plate bending (out-of-plane) stiffnesses. This element should not be used to model walls (use shear wall or general wall elements instead). It can be used to model deformable floor diaphragms, and to model shell structures such as tanks.*

*This chapter provides a brief description of the components and element.*

## 18.1 Purpose

The slab/shell element has membrane (in-plane) and plate bending (out-of-plane) stiffnesses. It can be used to model elastic walls, slabs and curved shells.

For PERFORM-3D the most likely application is to model deformable floor diaphragms. In this application the membrane behavior of the element accounts for in-plane effects in the diaphragm, and the plate bending behavior can be used for applying and distributing gravity loads. A floor diaphragm can be modeled using wall elements, but this requires four steps to define the element properties, namely material, cross section, wall compound component, and element. For a slab/shell element this is reduced to three steps, namely material, cross section, and element.

Note that a deformable floor diaphragm could also be modeled using infill panel elements.

For PERFORM-COLLAPSE the most likely application is to model floor slabs that can be assumed to be elastic. For slabs that can crack and yield, the inelastic layered slab/shell element can also be used. This element is described in a separate chapter. It is currently not available in PERFORM-3D.

## *18.2 Element Properties*

### 18.2.1 Element Geometry

A slab/shell element has four nodes and three local axes, as shown in Figure 18.1. As a general rule slab/shell elements should be rectangular or near rectangular. It is not essential that elements be plane (i.e., they can be warped quadrilaterals, with straight edges).

For a plane element, Axis 1 is normal to the element. For a warped element, Axis 1 is normal to the average plane for the element. Axes 2 and 3 are in the element plane (or average plane), along the principal axes of the material (see later). The procedure for orienting the axes is the same as for a wall element.

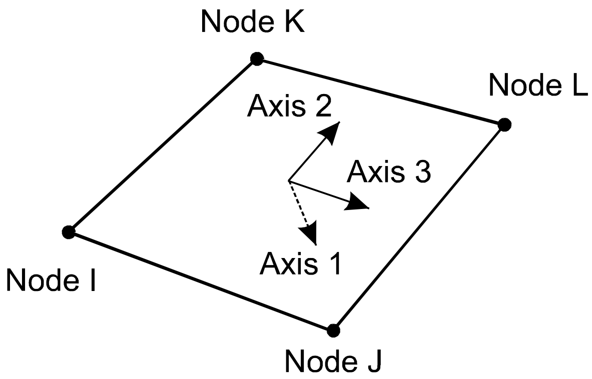



Figure 18.1 Element Nodes and Axes

Axes 1, 2 and 3 form a right-handed system. For a horizontal floor slab, if the node sequence I-J-L-K is counterclockwise in plan, Axis 1 is directed downwards as in Figure 18.1. If the I-J-L-K sequence is clockwise in plan, Axis 1 is directed upwards.

### 18.2.2 Material and Section Properties

The sequence for specifying element properties is as follows.

(1) Specify properties for one or more Elastic Materials for a Slab or Shell. This is the **Component Properties** task and the **Materials** page. The material can be isotropic or orthotropic.
(2) Specify properties for one or more Slab or Shell Elastic Sections. This is the **Component Properties** task and the **Cross Sects** page.
(3) Assign cross sections to slab/shell elements. This is the **Elements** task and the **Properties** page.

The properties of an isotropic material are defined by its Young's modulus and Poisson's ratio. The slab thickness is uniform over each element, but can be different for membrane and bending effects. The slab thickness will usually be the actual thickness.

The properties of an orthotropic material are defined by Young's modulus values along the principal directions of the material, a Poisson's ratio that couples these directions, and a shear modulus. An orthotropic material will usually mean a ribbed slab, rather than a solid slab with a true orthotropic material. The slab thickness is uniform over each element, but can be different for membrane and bending effects. For a ribbed slab the slab thickness must be an equivalent thickness that gives the required EA and EI values.

### 18.2.3 Beta-K Damping

Slab/shell elements currently have zero βK damping.

### 18.2.4 Axis 2, 3 Directions

Axes 2 and 3 are along the principal directions for the slab material, and are not necessarily parallel to the element edges. The results for slab/shell elements are forces and moments (per unit slab width) parallel to Axes 2 and 3. If force and moment strength capacities are used (see later), these are also parallel to Axes 2 and 3.

For an orthotropic ribbed slab, Axes 2 and 3 will usually be along the ribs. For a reinforced concrete slab, Axes 2 and 3 will usually be along the reinforcement directions, since the slab strengths will usually be known along those directions.

For an isotropic slab, the material properties are the same in all directions and the directions of Axes 2 and 3 may not be important. They will usually be parallel to one or more of the element edges.

To specify the directions of Axes 2 and 3, use the **Elements** task and the **Orientations** page.

### 18.2.5 Sign Convention

Force and moment results are in the Axis 1-2-3 coordinate system. Force and moment values are all per unit slab width.

Normal forces along Axes 2 and 3 are tension positive. Shear strain is positive as shown in Figure 18.2. Positive shear force causes positive shear stress and strain.

Bending moment along Axis 2 (about Axis 3) is positive when the bending stress on the positive Axis 1 side is compression. This is consistent with beam and column elements, where positive bending moment causes compression on the positive axis side of the element. Bending moment along Axis 3 (about Axis 2) is the same. Torsional moment is positive when the torsional shear stress on the positive Axis 1 side is negative.

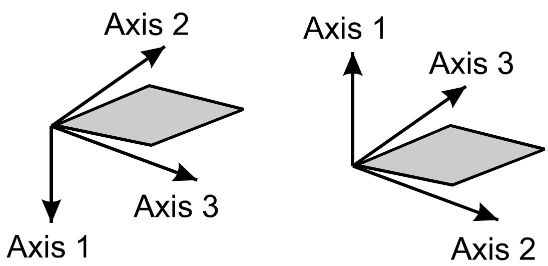



Figure 18.2 Positive Shear Strain

### 18.2.6 Thin Slab Assumption

It is currently assumed that bending shear deformations are small and can be ignored.

## *18.3 Element Loads and Geometric Nonlinearity*

PERFORM does not allow element loads for slab/shell elements. However, slab weights can be considered using self weight loads.

With the present version of this element you can not consider geometric nonlinearity.

## 18.4 Strength Limit States

You can specify strength capacities for bending and membrane effects, and use these capacities in strength limit states.

The following bending strengths are intended for use with reinforced concrete slabs. These are all bending moment per unit slab width.

(1) Positive and negative bending moment strengths for bending along element Axis 2 (i.e., bending moment about Axis 3).
(2) Positive and negative bending moment strengths for bending along element Axis 3 (i.e., bending moment about Axis 2).
(3) Positive and negative bending moment strengths for bending along an axis oriented 45 degrees to Axes 2 and 3.

The following bending strength is intended for use with uniform thickness steel plates.

(4) Strength for von Mises effective bending moment. This is calculated from the bending and torsional moments in the same way that the von Mises effective stress is calculated from normal and shear stresses.

The following membrane strengths are intended for use with reinforced concrete slabs. These are all forces per unit slab width.

(1) Tension and compression strengths along element Axis 2.
(2) Tension and compression strengths along element Axis 3.
(3) Membrane shear strength for shear along Axes 2 and 3.

The following membrane strength is intended for use with uniform thickness steel plates.

(4) Strength for von Mises effective force. This is calculated from the normal and shear forces in the same way that the von Mises effective stress is calculated from normal and shear stresses.

The D/C ratios are calculated at the element center (i.e., the demand values for the moments and forces are at the element center).

There are currently no strengths that account for moment-force interaction (i.e., for a plate with both large bending and large membrane effects).

## 18.5 Results Time Histories

You can plot results time histories for the following forces and moments. All are per unit slab width.

(1) Bending moment along Axis 2 (i.e., about Axis 3).
(2) Bending moment along Axis 3 (i.e., about Axis 2).
(3) Torsional moment.
(4) Membrane force along Axis 2.
(5) Membrane force along Axis 3.
(6) Membrane shear force.

## 18.6 Element Theory

### 18.6.1 General

This section describes the basis of the element theory, using a beam element as an example. The slab/shell element theory follows the same procedure, but is more complex. It will be documented in more detail in the future.

### 18.6.2 Outline of Element Theory

The element theory is stiffness based. However, instead of using geometric compatibility to establish the displacement transformation from node displacements to internal element deformations, equilibrium is used to establish the force transformation from internal actions to nodal forces. The displacement transformation is then obtained using the fact that the virtual displacements principle requires the displacement transformation to be the transpose of the force transformation. The result implies that displacement shape functions exist, but these functions are not explicitly defined. The implied shape functions do not satisfy displacement compatibility at the element boundaries, so convergence to the exact solution is not necessarily monotonic as the mesh is refined. However, analyses show that the element gives accurate results.

To illustrate the theory consider a beam element in bending, as shown in Figure 18.3.

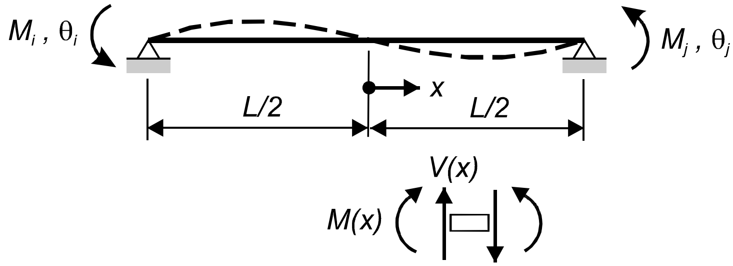



Figure 18.3 Beam Element to Illustrate Theory

The element has two rotation degrees of freedom, at the element ends, and two corresponding end moments. Consider two generalized internal forces as follows.

(1) Force $G_1$, such that the moment at any point in the beam is given by:

$$M(x) = G_1 \tag{18.1}$$

This is a constant moment mode.

(2) Force $G_2$, such that the moment at any point in the beam is given by:

$$M(x) = G_2 x \tag{18.2}$$

This is a linear moment mode.

From equilibrium, the following relationship can be established between the external nodal forces and the internal generalized forces:

$$\begin{bmatrix} M_i \\ M_j \end{bmatrix} = \begin{bmatrix} -1 & 0.5L \\ 1 & 0.5L \end{bmatrix} \begin{bmatrix} G_1 \\ G_2 \end{bmatrix} \tag{18.3}$$

For each generalized force there is a generalized stiffness. These stiffnesses can be established by equating energy as follows.

The relationship between a generalized force, $G$, and its corresponding generalized displacement, $g$, is:

$$G = K_g g \tag{18.4}$$

where $K_g$ is the generalized stiffness. The generalized displacements do not necessarily have obvious physical meanings. They are just the energy conjugates of the generalized force.

The strain energy associated with a force $G$ is:

$$SE = 0.5\frac{G^2}{K_g} \tag{18.5}$$

The strain energy for a uniform beam is also given by:

$$SE = \int 0.5\frac{M(x)^2}{EI}dx \tag{18.6}$$

where $EI$ has the usual meaning and the integration is over the element length. Using Equations (18.1) and (18.2), and equating the energies from Equations (18.5) and (18.6), gives the following generalized stiffness relationships.

$$G_1 = \frac{EI}{L}g_1 \tag{18.7a}$$

and

$$G_2 = \frac{12EI}{L^3}g_2 \tag{18.7b}$$

From the virtual displacement principle and Equation 18.3 it follows that the displacement transformation is given by:

$$\begin{bmatrix} g_1 \\ g_2 \end{bmatrix} = \begin{bmatrix} -1 & 1 \\ 0.5L & 0.5L \end{bmatrix} \begin{bmatrix} \theta_1 \\ \theta_2 \end{bmatrix} \tag{18.8}$$

Hence, combining Equations 18.3, 18.7 and 18.8:

$$\begin{bmatrix} M_i \\ M_j \end{bmatrix} = \begin{bmatrix} -1 & 0.5L \\ 1 & 0.5L \end{bmatrix} \begin{bmatrix} EI/L & 0 \\ 0 & 12EI/L^3 \end{bmatrix} \begin{bmatrix} -1 & 1 \\ 0.5L & 0.5L \end{bmatrix} \begin{bmatrix} \theta_1 \\ \theta_2 \end{bmatrix}$$

or:

$$\begin{bmatrix} M_i \\ M_j \end{bmatrix} = \begin{bmatrix} 4EI/L & 2EI/L \\ 2EI/L & 4EI/L \end{bmatrix} \begin{bmatrix} \theta_1 \\ \theta_2 \end{bmatrix} \tag{18.9}$$

This is the correct stiffness matrix, ignoring shear deformations. Shear deformations can be considered by observing that the generalized force G2 defines both a linear bending moment variation, given by Equation (18.2), and a constant shear force given by:

$$V(x) = G_2 \tag{18.10}$$

If shear flexibility is taken into account, Equation (18.6) becomes:

$$SE = \int 0.5 \left( \frac{M(x)^2}{EI} + \frac{V(x)^2}{GA'} \right) dx \tag{18.11}$$

where *GA'* has the usual meaning. Equating *SE* from Equations (18.5) and (18.11) gives the generalized stiffness with shear deformations. Hence the element stiffness matrix can be obtained. Again, this is the correct stiffness matrix. The method can also account for a cross section that varies over the element length, by changing equation (18.11) to make *EI* and *GA'* functions of *x*.

Shear deformations are not currently considered in the slab/shell element.

### 18.6.3 Slab/Shell Element

The slab/shell element uses essentially the preceding theory. The behavior is divided into membrane and plate bending parts, which requires that the element be plane. If the element is warped, the plane element stiffness matrix is modified to account for warping, by adding rigid link connections between the element corners and the nodes. The membrane and bending parts are coupled through the cross section stiffness matrix.

For the membrane part there are two in-plane displacements at each node, for a total of eight degrees of freedom. There are five generalized forces and three rigid body modes. The generalized forces define the following membrane force distributions.

(1) Normal force along the element A axis, constant over the element.
(2) Normal force along the element B axis, constant over the element.
(3) Membrane shear force, constant over the element.
(4) Normal force along the element A axis, constant along A and linear along B.
(5) Normal force along the element B axis, constant along B and linear along A.

For a rectangular element, axes A and B are parallel to the element edges. For an element that is not rectangular, axes A and B are "best fit" axes that are calculated by the program. The element A and B axes are not necessarily the same as the material 1 and 2 axes, although this will usually be the case. The integration to get the generalized stiffnesses is over the element area. For a non-rectangular element his involves some approximation.

For the bending part there are three displacements at each node (normal displacement and two rotations), for a total of twelve degrees of freedom. There are nine generalized forces and three rigid body modes. The generalized forces define the following moment distributions.

(1) Bending moment along the element A axis, constant over the element.
(2) Bending moment along the element B axis, constant over the element.
(3) Torsional moment, constant over the element.
(4) Bending moment along the element A axis, constant along A and linear along B.
(5) Bending moment along the element B axis, constant along A and linear along B.
(6) Torsional moment, constant along A and linear along B.
(7) Bending moment along the element A axis, constant along B and linear along A.
(8) Bending moment along the element B axis, constant along B and linear along A.
(9) Torsional moment, constant along B and linear along A.

The membrane and plate bending deformations account for only 20 of the 24 total node displacements. The remaining four displacements are the "drilling" rotations normal to the plane of the element. At the present time, small stiffnesses are assigned to these degrees of freedom.

## 18.7 Plate Bending Example

Figure 18.4 shows a very thin, uniform, elastic plate. Figure18.4(a) shows the dimensions and loads. The plate has uniform thickness, isotropic material, Young's modulus = 1.7472 x $10^7$ units, and Poisson's ratio = 0.3. The edges are either all simply supported or all clamped.

The plate has been analyzed using the PERFORM slab/shell element, with three regular meshes as shown in Figure 18.4(b) and one irregular mesh as shown in Figure 18.4(c).

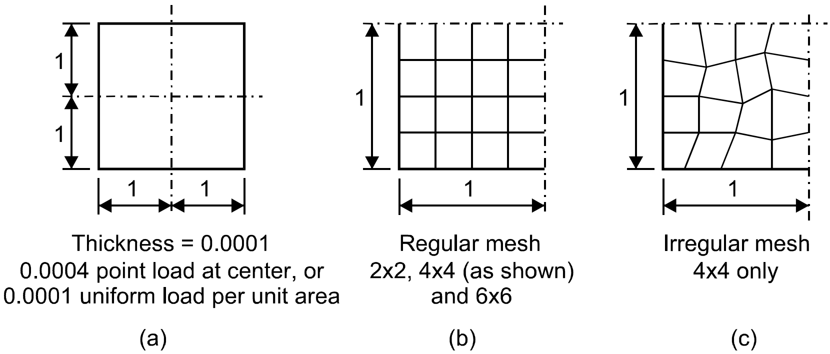



Figure 18.4 Elastic Plate Example

The calculated deflections at the center of the slab are shown in Table 18.1, as multiples of the exact deflections.

This example shows that the element gives accurate results for the deflections of a simple plate. However, if plate bending behavior is important in your application you should run examples to satisfy yourself that the element is sufficiently accurate for your needs.

**Table 18.1. Results for Elastic Plate Example**

| Support Type | Mesh | Deflection Calculated/Exact Point Load | Deflection Calculated/Exact Uniform Load |
|---|---|---|---|
| Simple | 2x2 | 1.123 | 1.001 |
| | 4x4 | 1.037 | 1.001 |
| | 6x6 | 1.018 | 1.001 |
| | 4x4 irregular | 1.008 | 0.973 |
| Clamped | 2x2 | 1.205 | 1.176 |
| | 4x4 | 1.067 | 1.049 |
| | 6x6 | 1.035 | 1.024 |
| | 4x4 irregular | 1.021 | 1.011 |

# 19 Inelastic Layered Slab/Shell Element

*The Inelastic Layered Slab/Shell element is intended for the analysis of reinforced concrete floor slabs in PERFORM-COLLAPSE. It is not available in PERFORM-3D.*

***The modeling of reinforced concrete slabs for collapse analysis is an extremely challenging task. As a general rule, it is neither necessary nor desirable to use this element to model slabs in steel floor systems, or most slab-on-beam concrete floors. These can be modeled more easily and efficiently using beam elements to model slab strips. The layered slab/shell element may be needed for flat plate floor systems.***

*This chapter describes the features and behavior of the element.*

## 19.1 Element Geometry

The slab/shell element is a 4-node element. Until more experience is gained, slab/shell elements should be rectangular or near rectangular.

It is not essential that elements be plane (i.e., they can be warped quadrilaterals, with straight edges). Elements in floor slabs will initially be plane, but they can become warped as the slab deflects. Elements in curved shells may be warped initially.

A layered slab/shell element has four nodes and three local axes, as described for the elastic slab/shell element.

## 19.2 Cross Section Properties

### 19.2.1 Inelastic Layered Cross Section

For a reinforced concrete slab/shell cross section you must divide the cross section into layers, and specify the location, thickness and material properties for each layer.

A set of steel reinforcing bars constitutes one layer, so a rectangular mesh of bars constitutes two layers. The steel in a layer has a uniaxial stress-strain relationship. The thickness of a steel layer is the steel area per unit slab width.

In a concrete layer, the concrete is in a state of plane stress. PERFORM uses a simplified theory to model the concrete, based on a lattice type of model. For details see Section 19.3. The thickness of a concrete layer in a solid slab is the actual layer thickness. A layer in a ribbed slab can have different thicknesses along Axes 2 and 3.

PERFORM permits a maximum of 7 concrete layers and 6 steel layers. For a slab with mesh reinforcement top and bottom you will need 4 layers for the steel reinforcement (one layer for the bars in each direction in the top surface, and the same in the bottom surface).

### 19.2.2 Other Properties

Plane sections are assumed to remain plane, and there is no bond slip between steel and concrete.

It is currently assumed that bending shear deformations are small and can be ignored. However, the element theory does not depend on this, and shear flexibility may be added in the future.

## *19.3 Material Properties*

### 19.3.1 Steel Layers

Steel layers are modeled using the non-buckling steel material.

### 19.3.2 Concrete Layers

A concrete layer is modeled in much the same way as the concrete in a general wall element. That is, the concrete material is modeled as a number of sub-layers, with uniaxial properties of concrete material or shear material type. This has the advantage of avoiding the complexity of a multi-axial material model for concrete. However, it also involves substantial approximations. On balance, we believe that it is a reasonable model for practical analysis, considering the complexity of concrete behavior and the uncertainty about the actual behavior of the concrete in a real structure.

For each concrete layer there are six sub-layers, as follows. They are illustrated in Figure 19.1.

(a) Tension-compression sub-layer along Axis 2. This is equivalent to the “vertical” layer in a general wall element.
(b) Tension-compression sub-layer along Axis 3. This is equivalent to the “horizontal” layer in a general wall element.
(c) Tension-compression sub-layer at 45 degrees to Axis 2. This is equivalent to one of the diagonal layers in a general wall element.
(d) Tension-compression sub-layer at -45 degrees to Axis 2. This is equivalent to the second diagonal layer in a general wall element.
(e) Shear sub-layer with shear stresses parallel to Axes 2 and 3. This is equivalent to the concrete shear layer in a general wall element.
(f) Shear sub-layer with shear stresses at 45 degrees to Axes 2 and 3. This layer has no counterpart in a general wall element.

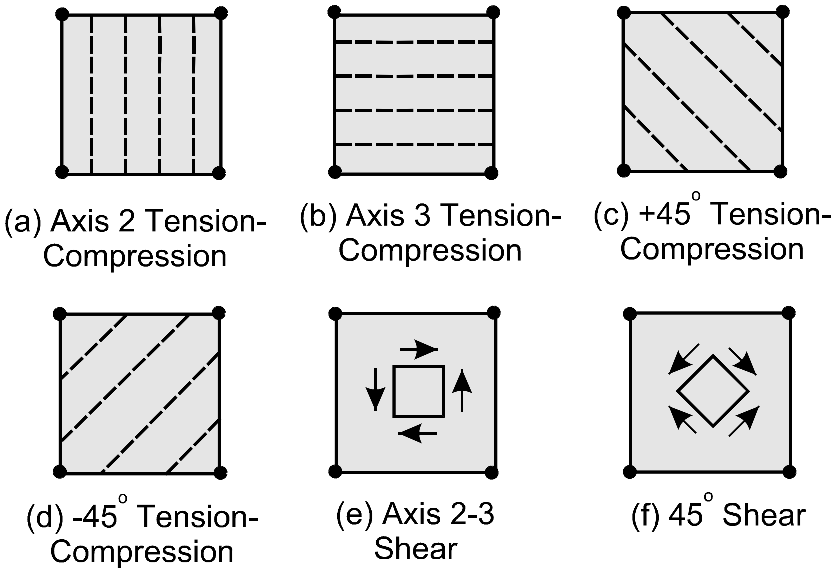



Figure 19.1 Concrete Material Sub-Layers

The tension-compression sub-layers have concrete material properties, with cracking, crushing and optional brittle strength loss, as for a concrete fiber in a shear wall. This is essentially a lattice model for concrete.

In tension, if the principal tension stress is parallel to a sub-layer, a crack can form in that sub-layer. If the principal stress orientation is between two sub-layer directions, cracks can form in both sub-layers, approximating a crack normal to the principal stress direction. Cracks can open and close if the stress is cyclic or as the principal tension stress changes direction. This is not an accurate model for concrete cracking, but it should be adequate for practical purposes.

In compression, if the principal stress is parallel to the fibers in a sub-layer, those fibers will resist the stress directly. The sub-layers at 45 degrees to the principal stress direction may also participate, depending on the amount of lateral confinement. If the principal stress is not parallel to the fibers in a sub-layer, the fibers that are most nearly parallel to the stress will be active. The behavior is roughly independent of the stress direction, with the greatest error when the stress is at 22.5 degrees. Some analyses that illustrate the behavior are presented in a later section.

The shear sub-layers have shear material properties, with an elastic-perfectly-plastic or trilinear relationship between shear stress and strain, and with optional brittle strength loss.

This concrete model is clearly approximate. Before you use Slab/Shell elements to model reinforced concrete slabs, you should compare the analysis results with available experimental results.

### 19.3.3 Membrane Shear Resistance in a Slab

This section considers membrane shear in the plane of the slab, not bending shear normal to the plane of the slab. Membrane shear is caused by shear forces in the plane of the slab and by torsional moments in the slab.

In elastic materials, and also in inelastic steel materials, it is usually assumed that shear stresses are carried by "conventional" shear as shown in Figure 19.1(e) and (f). The shear modulus for an elastic material is given by $G = E/2(1+\nu)$, and for steel the strength in shear is roughly 60% of the strength in tension. In reinforced concrete, however, the shear behavior is much more complex.

Shear resistance in reinforced concrete is provided by two mechanisms, as follows.

(1) Some of the shear is carried by truss action, involving interaction between concrete compression and steel tension. This is usually referred to as the shear carried by the shear reinforcement. Since we are concerned here with membrane shear, the "shear reinforcement" is the steel in the steel layers, which resists bending and other forces as well as shear. In a slab/shell element, resistance to this type of shear is developed by the four tension-compression sub-layers in each concrete layer, interacting with the steel layers to provide truss action.

(2) Additional shear is carried by the concrete. For analysis this is usually modeled as "conventional" shear, as in steel, although it is probably more of a frictional mechanism.

These two types of shear are considered in the following sections.

### 19.3.4 Shear Carried By Reinforcement

In a shear wall, the behavior can be essentially separated into axial behavior and shear behavior. For shear behavior the diagonal concrete "truss" members play an important role in the shear behavior, interacting with the vertical and horizontal reinforcement. These members are not necessarily at 45 degrees to the wall axis, but considering all of the approximations and complexities in modeling the inelastic behavior of reinforced concrete, it should be reasonable to assume a 45 degree angle.

If a Slab/Shell element were used to model a shear wall, with Axis 2 along the wall axis, the 45 degree sub-layers would provide the diagonal compression forces, in much the same way as the diagonal layers in a General Wall element. In an actual slab, however, there are no simple "axial" and "shear" directions. Instead, the principal stress and strain directions will vary from point to point over the slab, and possibly also from layer to layer. Truss action exists, and plays a major role in resisting both membrane shear and torsional moment, but the direction of the shear changes from point to point. The use of four tension-compression sub-layers is, we believe, a reasonable way to model this truss action. In some parts of a slab the tension-compression sub-layer at 45 degrees to Axis 2 might provide most or all of the compression member behavior, in some parts the sub-layer parallel to Axis 2 might do the job, and in some parts two sub-layers might be in compression. The greatest "error" will occur when the diagonal compression should be at 22.5 degrees to Axis 2, in which case the two closest sub-layers must do the job.

### 19.3.5 Shear Carried by Concrete

The shear carried by the concrete is modeled by the two shear sub-layers. These sub-layers assume "conventional" shear behavior, with a shear modulus and a yield stress in shear. Consider, first, the assumptions for one shear sub-layer, then consider the reason for having two sub-layers.

For a steel plate, the shear strength is roughly 0.6 times the tension strength, and the elastic shear modulus is given by $G = E/2(1+\nu)$, or about 0.4E. The stiffness and strength in shear are thus not too different from the stiffness and strength in tension. The yield strain in shear is also similar numerically to the yield strain in tension.

Concrete is different. The shear strength of concrete is only about 5% of the strength in compression. If we assume that $G = E/2(1+\nu)$ and put $\nu = 0.2$, the shear strain at yield is roughly 10% of the yield strain in compression. Intuitively this does not seem reasonable. Shear resistance in concrete is not developed by plastic yield, as in steel, but by complex mechanisms involving cracking, friction and aggregate interlock. We would expect "yield" in shear to be reached at much larger strains than 10% of the "yield" strain in compression.

This aspect is considered in some depth for the General Wall element. For that element a suggestion is made that the shear modulus for the concrete shear layer be made much smaller than that given by $G = E/2(1+\nu)$. If you use shear sub-layers in Slab/Shell elements, we suggest a similar value.

There are two concrete shear sub-layers in each concrete layer. The reason is that the principal shear stress direction is not known. If only one shear sub-layer is used, and it the principal shear direction is at 45 degrees to Axis 2, a single shear sub-layer oriented along Axes 2 and 3 contributes no shear strength or stiffness. The second shear sub-layer, at 45 degrees to Axis 2, is added to provide shear behavior that is more nearly invariant as the principal shear direction changes.

### 19.3.6 Implied Concrete Tension Strength

From Mohr's circle for stress, if the yield stress in shear of a shear sub-layer is $\tau_y$, this implies tension-compression strengths of $\tau_y$ at 45 degrees to the shear direction. Hence, the shear sub-layer Figure 19.1(e) has tension and compression strengths at 45 degrees to Axes 2 and 3, and the shear sub-layer Figure 19.1(f) has tension and compression strength along Axes 2 and 3. Because the concrete is stiff and strong in compression, the additional compression strength and stiffness from a shear sub-layer is small compared with the compression strengths and stiffnesses of the tension-compression sub-layers. However, the concrete has little or no strength and stiffness in tension, so the implied tension strength and stiffness from a shear sub-layer can have a substantial effect on the behavior.

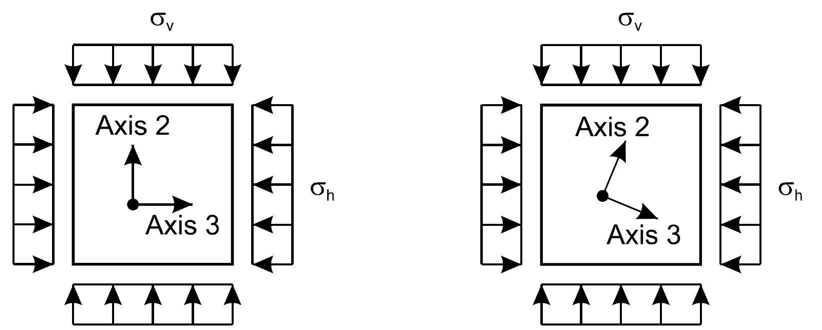
It is optional to use shear sub-layers, and you may choose not to specify them (by assigning a zero thickness). If you include shear sub-layers we strongly suggest that (a) you use small strengths and stiffnesses, and (b) that you specify brittle strength loss, so that when the shear strain (i.e., the diagonal tension strain) reaches a reasonable value, the shear strength drops to near zero. This will provide behavior of the type that is usually referred to as "tension stiffening".

## *19.4 Stress-Strain Behavior of a Concrete Layer*

### 19.4.1 Analyses With Different Stress Ratios

A concrete model with uniaxial sub-layers is simple computationally, and it has the advantage that it models truss-type action for shear. However, it is not necessarily accurate. This is considered in this section.

Figure 19.2 shows a slab/shell element with unit width, depth and thickness. The element is loaded in biaxial compression with no bending. The element has one concrete layer, with no steel layers. The concrete is assumed to be elastic-perfectly-plastic, with an elastic modulus $E_c$, a yield (crushing) strength $f_c$, and no tension strength. There are four tension-compression sub-layers, but no shear sub-layers.




(a) Stresses Along Axes 2,3 (b) Stresses at 22.5° to Axes 2,3

Figure 19.2 Element for Stress-Strain Study

The stiffness and strength of this element (i.e., of a single concrete layer) have been calculated for different ratios of horizontal stress to vertical stress. Although the stress-strain relationship for the concrete is elastic-perfectly-plastic, the calculated stress strain relationship can be trilinear. However, in most cases the behavior is essentially elastic-perfectly-plastic, with an initial modulus and a yield stress.

### 19.4.2 Results for Stresses Parallel to Axes 2 and 3

Table 19.1 shows the key results for the case in Figure 19.2(a), where the principal stresses are parallel to Axis 2. The same results are obtained if the principal stresses are at 45 degrees to Axis 2.

**Table 19.1. Behavior for Stresses Along Axis 2,3.**

| Ratio $\sigma_h/\sigma_v$ | Initial Modulus Ratio | Yield Stress Ratio |
|---|---|---|
| 0 | 1.00 | 1.00 |
| 0.125 | 1.14 | 1.14 |
| 0.25 | 1.33 | 1.33 |
| 0.375 | 1.50 | 1.60 |
| 0.5 | 1.59 | 2.00 |
| 0.75 | 1.77 | 2.00 |
| 1.0 | 1.99 | 2.00 |

For each ratio of horizontal stress (along Axis 3) to vertical stress (along Axis 2), this table shows (a) the calculated initial modulus as a multiple of the basic concrete modulus, and (b) the calculated yield stress as a multiple of the basic concrete yield stress.

The table shows that the effective elastic modulus in the vertical direction can be as much as two times the modulus of the basic concrete material. This is a larger increase than would be expected in actual concrete. For example, if Poisson’s ratio for concrete is 0.25, for a stress ratio of 1.0 the effective elastic modulus for an elastic material is 1.33 times the basic modulus. This difference may not be as serious as it might appear. In a concrete slab, most of the deformation is likely to result from cracking of the concrete and yielding of the steel. Compression strains in the concrete are likely to contribute a relatively small amount of the total deformation, so errors in these strains, whether due to inherent inaccuracies in the model or to uncertainty about the actual modulus, are likely to have little overall effect.

The table also shows that the effective yield stress increases under biaxial compression, and can be as much as two times the basic yield stress. This is larger than would be expected for actual concrete, but again the difference may not be as serious as it might appear. There are two reasons for this, as follows.

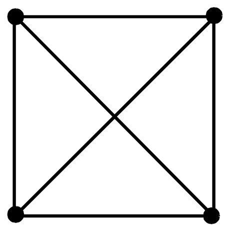
1. In a progressive collapse analysis, floor slabs are likely to be in membrane tension as they sag, and crushing of the concrete may not be an important part of the inelastic mechanism. This particularly the case because a slab is likely to be under-reinforced, so that the primary collapse mechanism is yielding of the reinforcement.

2. As the amount of bending in a slab is progressively increased, cracking spreads through the slab, and the depth of the compression zone progressively reduces. Concrete crushing commences when the depth of the compression zone is so small that the concrete stress exceeds the crushing stress. The depth of the compression zone also decreases in a layered slab element. However, whereas this is a continuous process in an actual slab, in a layered slab this is a discontinuous process, with a sudden decrease in the compression zone depth each time new layer cracks. If the thickness of the last layer is large (or if there is compression reinforcement) the concrete compression stress in the layer may never be large enough to cause crushing. This is a finite element modeling issue that is separate from the modeling of the material properties.

### 19.4.3 Cause of the Observed Behavior

The concrete model with sub-layers does not behave like a regular continuous material. Instead it behaves like a lattice or truss model. For a square element, a lattice model that is equivalent to the sub-layer model is shown in Figure 19.3. For understanding the behavior it can be useful to consider this model.

The sub-layer and lattice models have the same behavior for this example. In general the sub-layer model has the advantage that it is not restricted to square elements, and hence is more flexible than the lattice model.



Diagonal bar areas = vertical and horizontal bar areas x 1.414, to give same volume.

Figure 19.3 Equivalent Lattice Model

When the horizontal stress is zero, all vertical stress is carried by the Axis 2 sub-layer (or the Axis 2 bars in the lattice model), and the stiffness and strength are those for the basic concrete material. In this case, the stress and strain in the diagonal sub-layers are zero, the Axis 3 sub-layer is cracked, and the tension strain along Axis 3 is equal to the compression strain along Axis 2. This may seem to imply a Poisson ratio of 1.0, but note that the Axis 3 strain is cracking strain, and Poisson’s ratio does not apply to cracking strains. As shown later, Poisson’s ratio in cases with no cracking is 0.33. This is larger than the value of about 0.25 that is typically assume for concrete, but is not unreasonable.

As the ratio of horizontal stress to vertical stress increases, the diagonal sub-layers also carry load. As a result, the vertical stiffness and strength are both increased. For a stress ratio smaller than 0.5 the Axis 3 sub-layer still cracks, and only the Axis 2 sub-layer yields. The vertical stiffness and strength both increase. For a stress ratio larger than 0.5 the Axis 3 sub-layer does not crack, and the Axis 2 and diagonal sub-layers all yield. The vertical stiffness continues to increase, but the strength reaches a maximum of two times the basic concrete strength (the combined diagonal sub-layers have the same vertical strength as the Axis 2 sub-layer).

### 19.4.4 Poisson's Ratio

The implied value of Poisson's ratio can be calculated for the cases with no cracking. For an elastic material in a 2D state of stress the strains are given by:

$$\begin{bmatrix} \varepsilon_h \\ \varepsilon_v \end{bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\nu \\ -\nu & 1 \end{bmatrix} \begin{bmatrix} \sigma_h \\ \sigma_v \end{bmatrix}$$

Where the symbols have the usual meaning. For each stress ratio $\sigma_h / \sigma_v$ , the corresponding ratio $\varepsilon_h / \varepsilon_v$ in the elastic range can be obtained from the analysis, and hence the value of Poisson's ratio can be calculated. For stress ratios of 0.375, 0.5 and 0.75, the implied value of Poisson's ratio is essentially 0.33.

This is substantially larger than the value of about 0.25 that is typically assumed for concrete in the elastic range. However, the elastic concrete strains are small, and have little effect on the behavior of a concrete slab after it cracks and yields. Hence, although a high value of the elastic Poisson's ratio is not correct, it should not be a major flaw in the model (there are worse flaws).

### 19.4.5 Results for Stresses Inclined to Axes 2 and 3

The sub-layer model behaves the same way when the principal stresses are at 45 degrees to Axes 2 and 3 as when they are parallel to these axes. The greatest effect of the load direction occurs when the principal stresses are at 22.5 degrees to the axes. Table 19.2 shows the key results for the this case (Figure 19.2(b)). The corresponding values from Table 19.1 are shown in parentheses.

This table shows that the sub-layer model (and also the lattice model) does not have properties that are independent of the principal stress directions. Also, for small ratios of horizontal to vertical stress the model predicts cracking in all layers, with no stiffness or strength. However, this type of analysis, considering concrete only with no steel reinforcement, exaggerates this aspect of the model. When the model is used for concrete in slab elements, the effects of changing the sub-layer orientations are not as large. The next section illustrates this.

**Table 19.2. Behavior for Stresses at 22.5 Degrees to Axis 2.**

| Ratio $\sigma_h/\sigma_v$ | Initial Modulus Ratio | Yield Stress Ratio |
|---|---|---|
| 0 | 0 (1.00) | 0 (1.00) |
| 0.125 | 0 (1.14) | 0 (1.14) |
| 0.25 | 1.48 (1.33) | 1.73 (1.33) |
| 0.375 | 1.51 (1.50) | 1.77 (1.60) |
| 0.5 | 1.59 (1.59) | 1.81 (2.00) |
| 0.75 | 1.80 (1.77) | 1.90 (2.00) |
| 1.0 | 1.98 (1.99) | 2.00 (2.00) |

## *19.5 Behavior of a Concrete Slab*

### 19.5.1 Slab Example

Figure 19.4 shows a slab that is simply supported on all four edges (vertical support only), with a point load at the middle. The slab is modeled using an 8 x 8 mesh of slab/shell elements, as shown. The purpose of the analysis is to study the effects of changing the concrete sub-layer orientation, not to study the behavior of an actual slab.

The slab has 0.25% steel reinforcement in both directions, in the bottom of the slab only. The concrete has no tension strength, and the steel and concrete are both assumed to be elastic-perfectly-plastic. For the first set of analyses, there is no brittle strength loss in the concrete. This is considered later.

The ratio of initial steel stiffness to initial concrete stiffness is 8, and the ratio of steel strength to concrete compression strength is 10. The yield strain for the concrete in compression is 0.1%, and the yield strain for the steel in tension is 0.125%.

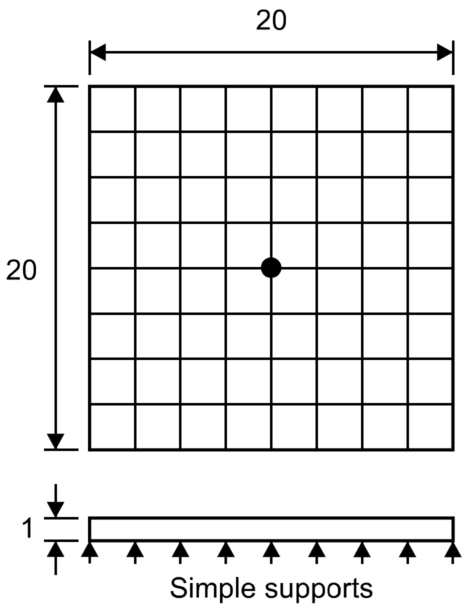



Figure 19.4 Concrete Slab Example

The slab cross section is modeled using 5 concrete layers, each 1/5 of the slab thickness. The thickness for the shear sub-layer is zero for all concrete layers (i.e., no shear strength or stiffness). There are 2 steel layers, at the middle of the lowest concrete layer.

The steel bars are oriented parallel to the slab edges. The slab has been analyzed with two different orientations for the concrete sub-layers, as follows.

(1) Concrete sub-layers oriented parallel to the slab edges (and also parallel to the steel reinforcement). This would be the usual orientation for a typical slab.
(2) Concrete sub-layers oriented at 22.5 degrees to the slab edges. This maximizes the difference in the behavior of the concrete layers.

Two sets of analyses have been carried out, the first ignoring large displacement effects and the second taking these effects into account.

### 19.5.2 Analysis Results

Table 19.3 shows calculated results for the slab analysis, for the case where large displacements are ignored.

**Table 19.3. Slab Results, Small Displacements Analysis**

| Deflection (multiple of slab thickness) | Load (multiple of load to yield steel for θ = 0). | | Max. steel tension strain (multiple of yield strain) | | Max. concrete comprn strain (multiple of yield strain) | |
|---|---|---|---|---|---|---|
| | θ=0 | θ=22.5 | θ=0 | θ=22.5 | θ=0 | θ=22.5 |
| 0.33 | 1.08 | 1.05 | 1.19 | 1.16 | 0.73 | 0.64 |
| 1.00 | 1.87 | 1.75 | 4.69 | 4.84 | 2.05 | 1.58 |
| 3.00 | 2.33 | 2.27 | 14.78 | 15.22 | 5.94 | 4.77 |

The results are shown for three deflections at the load point, namely 0.33, 1.0 and 3.0 times the slab thickness. The deflection of 0.33 times the slab thickness is a little larger than the deflection at first yield of the steel. The other results are expressed as the following ratios.

(1) The load is expressed as a multiple of the load that causes first yield of the steel for the case where the concrete sub-layers are parallel to the slab edges.
(2) The steel tension strain is a multiple of the steel yield strain.
(3) The concrete compression strain is a multiple of the concrete yield strain.

This table shows that the orientation of the concrete sub-layers has only a small effect on the calculated load-deflection relationship and steel strains. The sub-layer orientation does, however, have a significant effect on the calculated concrete strains. This is considered in a later section.

Table 19.4 shows the results when large displacements are considered.

This table again shows that the orientation of the concrete sub-layers has a small effect on the load-deflection relationship and the calculated steel strains, but a more significant effect on the calculated concrete strains.

**Table 19.4. Slab Results, Large Displacements Analysis**

| Deflection (multiple of slab thickness) | Load (multiple of load to yield steel for θ = 0). | | Max. steel tension strain (multiple of yield strain) | | Max. concrete comprn strain (multiple of yield strain) | |
|---|---|---|---|---|---|---|
| | θ=0 | θ=22.5 | θ=0 | θ=22.5 | θ=0 | θ=22.5 |
| 0.33 | 1.11 | 1.08 | 1.18 | 1.20 | 0.70 | 0.65 |
| 1.00 | 1.96 | 1.89 | 5.64 | 5.74 | 1.31 | 0.89 |
| 3.00 | 5.35 | 5.15 | 19.67 | 22.78 | 5.98 | 4.31 |

## 19.5.3 Calculated Concrete Strains

Tables 19.3 and 19.4 show calculated concrete strains. As shown, these strains depend on the orientation of the concrete sub-layers. However, they also depend on the way in which the slab cross section is divided into layers, and are likely to be inaccurate for this reason. This is considered in this section.

As an actual concrete slab (or beam) cracks, the neutral axis migrates progressively towards the compression side, and the depth of the compression stress block gets progressively smaller. At some point the concrete may crush at the extreme fiber, and progress continuously into the compression zone. Complete compression failure may occur. A layered model of a slab (or a fiber model of a beam) may not accurately capture this behavior.

In a layered model the stresses and strains in a layer are calculated at the layer mid-thickness, and are assumed to be constant over the depth of the layer. A layer is either completely in tension or completely in compression, and the depth of the stress block changes discontinuously rather than progressively. The neutral axis also tends to move discontinuously as each new layer cracks. As a consequence, the calculated concrete strains in a layered model are likely to be very approximate. The steel strains, however, are more accurate, because the distance from the neutral axis to the steel is larger than from the neutral axis to the compression concrete, and the steel strains are less sensitive to small changes in the neutral axis location.

Whether the concrete crushes also depends on the layer thicknesses. In the above slab example, each concrete layer is 0.2 times the slab thickness, and each steel layer is 0.025 times the slab thickness (based on 0.25% reinforcement area). Since the steel is 10 times stronger than the concrete, a steel layer is stronger than a concrete layer. Hence, if the lower four concrete layers crack, the outer layer can be expected to crush. This might not be the case if the outer concrete layer were thicker, or if the cross section had compression reinforcement. Also, as the number of concrete layers, or the layer thicknesses, are changed, the point at which the concrete crushes can change substantially. The calculated concrete strains are thus inherently unreliable, because they are sensitive to the neutral axis location. Predictions of whether the concrete will crush in bending are also unreliable, because they are sensitive to the layer thicknesses.

Fortunately, most slabs will be controlled by steel yield rather than concrete crushing, and the calculated behavior will be fairly insensitive to the concrete layer thicknesses. Also, the concrete strains will usually be small relative to the steel strains, so that inaccuracies in calculating the concrete strains will have relatively small effects on the load-deflection behavior of the slab or on the calculated steel strains.

However, if the bending behavior of a slab is indeed controlled by the strength of the concrete, analyses using the sub-layer concrete model probably will not be accurate.

### 19.5.4 Effect of Brittle Strength Loss

The strain at which strength loss begins in concrete is typically about 0.003, which is 3 times the concrete yield strain in this example. The calculated concrete strain exceeds this value in both Table 19.3 and Table 19.4. To study the effect of strength loss, the analyses have been repeated with modified concrete properties. Strength loss begins at 3 times the yield strain. The concrete strength then reduces linearly to 0.1 times its original strength at 5 times the yield strain, and remains constant for higher strains.

The effects on the calculated results are shown in Tables 19.5 and 19.6. Results are shown only for deflections equal to 3 times the slab thickness, since there is no strength loss for the deflections equal to 0.33 and 1.00 times the thickness.

Again, the orientation of the concrete sub-layers has a small effect on the load-deflection relationship and the calculated steel strains, and a somewhat larger effect on the calculated concrete strains. Overall, the effect of the concrete sub-layer orientation is small.

**Table 19.5. Results with Strength Loss, Small Displacements**

| Deflection (multiple of slab thickness) | Load (multiple of load to yield steel for θ = 0). | | Max. steel tension strain (multiple of yield strain | | Max. concrete comprn strain (multiple of yield strain | |
|---|---|---|---|---|---|---|
| | θ=0 | θ=22.5 | θ=0 | θ=22.5 | θ=0 | θ=22.5 |
| 3.00 | 1.77 | 1.75 | 14.47 | 14.54 | 8.31 | 8.17 |

**Table 19.6. Results with Strength Loss, Large Displacements**

| Deflection (multiple of slab thickness) | Load (multiple of load to yield steel for θ = 0). | | Max. steel tension strain (multiple of yield strain | | Max. concrete comprn strain (multiple of yield strain | |
|---|---|---|---|---|---|---|
| | θ=0 | θ=22.5 | θ=0 | θ=22.5 | θ=0 | θ=22.5 |
| 3.00 | 3.69 | 3.71 | 17.93 | 20.00 | 11.56 | 9.32 |

### 19.5.5 Summary

The sub-layer model for concrete clearly is not exact. However, given the many uncertainties about concrete properties and behavior, and the weaknesses in other concrete models, we believe it is a reasonable model for slabs in which the behavior is not controlled by concrete compression.

**Although we believe that the PERFORM slab/shell element is a reasonable element for practical application, we have not yet compared it against experimental results. Before you use this element for practical applications, you should check it against experimental results to confirm that its behavior is satisfactory.**

## *19.6 Element Theory*

The basic theory is the same as for the elastic slab/shell element, with differences to account for nonlinear behavior.

The behavior is divided into membrane and plate bending parts, which requires that the element be plane. If the element is warped, the plane element stiffness matrix is modified to account for warping, by adding rigid link connections between the element corners and the nodes. The membrane and bending parts are coupled through the cross section stiffness matrix.

For the membrane part there are two in-plane displacements at each node, for a total of eight degrees of freedom. There are five generalized forces and three rigid body modes. The generalized forces define the following membrane force distributions.

(1) Normal force along the element A axis, constant over the element.
(2) Normal force along the element B axis, constant over the element.
(3) Membrane shear force, constant over the element.
(4) Normal force along the element A axis, constant along A and linear along B.
(5) Normal force along the element B axis, constant along B and linear along A.

For a rectangular element, axes A and B are parallel to the element edges. For an element that is not rectangular, axes A and B are "best fit" axes that are calculated by the program. The element A and B axes are not necessarily the same as the material 1 and 2 axes, although this will usually be the case. The integration to get the generalized stiffnesses is over the element area. For a non-rectangular element his involves some approximation.

For the bending part there are three displacements at each node (normal displacement and two rotations), for a total of twelve degrees of freedom. There are nine generalized forces and three rigid body modes. The generalized forces define the following moment distributions.

(1) Bending moment along the element A axis, constant over the element.
(2) Bending moment along the element B axis, constant over the element.
(3) Torsional moment, constant over the element.
(4) Bending moment along the element A axis, constant along A and linear along B.
(5) Bending moment along the element B axis, constant along A and linear along B.
(6) Torsional moment, constant along A and linear along B.

(7) Bending moment along the element A axis, constant along B and linear along A.
(8) Bending moment along the element B axis, constant along B and linear along A.
(9) Torsional moment, constant along B and linear along A.

The membrane and plate bending deformations account for only 20 of the 24 total node displacements. The remaining four displacements are the "drilling" rotations normal to the plane of the element. At the present time, small stiffnesses are assigned to these degrees of freedom.

## 19.7 Element Loads and Geometric Nonlinearity

PERFORM does not allow element loads for slab/shell elements. However, slab weights can be considered using self weight loads.

With the present version of this element you can either (a) ignore geometric nonlinearity, or (b) consider true large displacements. There is currently no P-Δ option.

## 19.8 Deformation Limit States

### 19.8.1 Strain Limits

It is natural to use strain as the demand-capacity measure for slab/shell elements. This can be done, as described in this section. However, for the present version of PERFORM it may not be convenient to use strain. The reason is explained in this section. A more practical demand-capacity measure is rotation, as described in the next section.

To calculate strain D/C ratios you must do the following.

(1) Specify strain capacities for the steel and/or concrete materials.
(2) When you define a layered slab/shell cross section, select the layers for which strain D/C ratios are to be calculated.
(3) Set up one or more deformation limit states that specify the slab/shell elements and what type of strain to consider.

The problem with strain as a demand-capacity measure is that the calculated strain value (the strain demand) depends on the gage length over which the strain is calculated. Usually, the shorter the gage length the larger the calculated strain. This is especially true if there is brittle strength loss. When strain is calculated for an element, the gage length is essentially the size of the element. Hence, as the finite element mesh is refined, the calculated strain demand usually increases.

A similar problem arises when shear wall and general wall elements are used to model walls. For these elements, rather than calculate strain directly in the elements, it may be better to define strain gage elements and to calculate D/C ratios for these elements. The gage length for strain calculation can then be made independent of the element size.

In the present version of PERFORM, there is no strain gage element for use with slab/shell elements (the current strain gage element considers membrane strain only, not combined membrane and bending strain). Hence, if you use strain as a demand-capacity measure you must recognize that the element size defines the gage length, and base the strain capacities for steel and concrete materials on this gage length. Obviously this can be inconvenient. A more practical alternative is to use rotation as the demand-capacity measure, as considered in the next section.

### 19.8.2 Rotation Limits

Rotations can be used as demand-capacity measures for fiber segments in beam elements. Rotations can also be used for slab/shell elements.

As with a beam, if the bending moment gradient in a slab is large, as near a support, inelastic deformations will tend to concentrate in the end elements. In this case it may be better to use rotation over the element length as the demand-capacity measure. For any axis, this rotation is the average curvature multiplied by the element length along the axis. Rotation demand tends to be less sensitive to changes in the segment length than strain demand. Also, published capacity values tend to be in terms of rotation as the demand-capacity measure, rather than strain.

If you specify rotation capacities for slab elements, and limit states based on these capacities, the rotation D/C ratio is calculated at the element level. That is, the gage length for rotation calculation is the element size. It may be better to measure rotation on a longer gage length that extends over two or more elements. This will give larger rotation demand values, and hence, for the same capacity, larger D/C ratios, and hence is more conservative. To consider rotation over two or more elements, use Beam-Type Rotation Gage elements.

## 19.9 Connections Between Beams and Floor Slabs

Figure 19.5(a) shows, diagrammatically, a beam with a composite floor slab.

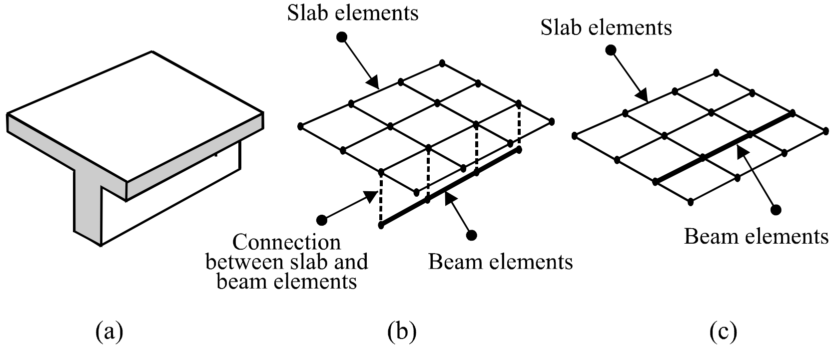



Figure 19.5. Composite Slab Models

If you model such a floor using slab elements and corresponding short beam elements, you must account for the eccentric connection between the beam and slab. There are two ways to do this, as follows.

The first method is shown in Figure 19.5(b). In this case the nodes for the slab elements are at the slab mid-thickness, and the nodes for the beam elements are at the beam centroid. This requires additional connection elements that link the beam and slab elements. These connection elements should be short beam elements. Do not make these elements extremely stiff, as this can cause numerical sensitivity problems. A reasonable size for these elements is about the same size as the beam elements (which provides an essentially rigid connection).

If you are modeling a steel beam with a composite slabs, and you want to model inelastic behavior in the shear connectors between the slab and beam, you must use this type of model. To model the shear connectors, put shear hinges in the short connection elements. In this case you can model the beam using hinge type elements, or you can use elements with fiber cross sections.

If you do not want to model shear connectors (i.e., if you can assume a rigid connection) it is simpler to use the second method, as shown in Figure 19.5(c). In this case you do not need the short connecting elements. However, you must use beam elements with fiber cross sections. The reason is that when you define a fiber cross section, the reference axis for the section does not have to be at the centroid of the section. Instead, it can be at any convenient location. For the model in Figure 19.5(c), you must define the reference axis for the beam to be at the slab mid-thickness location. There are two ways to do this, as follows.

(1) When you define the fiber beam cross section, measure the fiber y coordinates from the slab mid-thickness. In the usual case where Axis 2 for a beam element is vertically upwards, this means that the y coordinates will be negative.

(2) When you define the fiber beam cross section, measure the fiber y coordinates from the beam mid-depth (or any other convenient level). Then use the Other Properties tab and specify a value for Distance from Nodes to Beam Axis. When the beam section is used, this distance is added to each of the fiber y coordinates. In the usual case where Axis 2 for a beam element is vertically upwards, the distance will be negative. The advantage of this method is that it can be easier to change the beam location, since it requires a change in only this one distance, not in the coordinates of all fibers.