6 Methods of Analysis for Earthquake-Resistant Structures

6.1 Introduction ................................................................. 6-2
   Structural Analysis Procedures • Models of Structures
   Loads and Boundary Conditions • Notation

6.2 Equilibrium .................................................................... 6-6
   Node Equilibrium • Equilibrium of Basic System of Element
   Forces • Lower Bound Theorem of Plastic Analysis

6.3 Geometric Compatibility ................................................. 6-11
   Displacement–Deformation Relationship under Large
   Displacements • Linear Approximation of
   Displacement–Deformation Relations • Compatibility
   Relationship for Elements with Moment Releases

6.4 Equilibrium in Deformed Configuration ............................. 6-16

6.5 Principle of Virtual Work ................................................ 6-18
   Virtual Work Principles for the Structure • Virtual Work
   Principles for an Element • Upper Bound Theorem of Plastic
   Analysis • Example of Plastic Analysis

6.6 Force–Deformation Relationships .................................... 6-23
   Section Response • Material Models

6.7 Frame Elements ............................................................. 6-28
   Basic Relationships • Concentrated Plasticity Elements •
   Distributed Inelasticity Elements

6.8 Solution of Equilibrium Equations .................................. 6-36
   Newton–Raphson Iteration • Load Incrementation • Load
   Factor Control during Incrementation • Load Factor
   Control during Iteration • Structure State
   Determination • State Determination of Elements with
   Force Formulation • Nonlinear Solution of Section State
   Determination

6.9 Nonlinear Geometry and P-Δ Geometric Stiffness .............. 6-43
   Resisting Forces and Element Tangent Stiffness Matrix •
   Geometric Stiffness Matrix • P-Δ Geometric Stiffness

6.10 Examples of Nonlinear Static Analysis ............................... 6-47

6.11 Dynamic Analysis ......................................................... 6-52
   Free Vibration • Modal Analysis for Linear Response •
   Earthquake Excitation • Numerical Integration of
   Equations of Motion for Linear Response • Numerical
   Integration of Equations of Motion for Nonlinear Response

6.12 Applications of Linear and Nonlinear Dynamic
   Analysis ........................................................................... 6-57

6.13 Conclusions .................................................................... 6-61

6.14 Future Challenges .......................................................... 6-62
6.1 Introduction

This chapter presents structural analysis methods for designing earthquake-resistant structures. The focus is on structural models consisting of frame elements for modeling beam, column and brace members, which is the common type of modeling for buildings and bridges in current earthquake engineering practice. Structural analysis software often used in engineering design incorporates one or more of the methods of analysis presented in this chapter. The chapter discusses sources of nonlinear material and geometric behavior. It covers plastic analysis methods for collapse load and plastic deformation determination under the assumption of elastic-perfectly plastic material response. It briefly describes nonlinear hysteretic material models and uses these to derive the hysteretic response of sections under the interaction of biaxial moment and axial force. This chapter further discusses concentrated inelasticity frame elements, and compares two approaches for the derivation of the force-deformation relation of distributed inelasticity frame elements. The effect of nonlinear geometry is presented in the general form of the corotational formulation for frame elements under large displacements. Consistent approximations are introduced to arrive at simplified nonlinear geometry methods that suffice in many design situations, in particular, the so-called P-D geometric stiffness. The chapter concludes with a discussion of linear and nonlinear dynamic response under ground excitation. The key features of the analysis methods in this chapter are illustrated with examples of static and dynamic nonlinear response of components and structures. The chapter concludes with a few important observations and a discussion of future challenges for improving structural analysis procedures for earthquake-resistant design.

6.1.1 Structural Analysis Procedures

The analysis of a structural system to determine the deformations and forces induced by applied loads or ground excitation is an essential step in the design of a structure to resist earthquakes. A structural analysis procedure requires: (i) a model of the structure, (ii) a representation of the earthquake ground motion or the effects of the ground motion and (iii) a method of analysis for forming and solving the governing equations. There is a range of methods from a plastic analysis to a sophisticated nonlinear, dynamic analysis of a detailed structural model that can be used, depending on the purpose of the analysis in the design process. This chapter presents structural analysis procedures for earthquake-resistant design. The focus is on methods for structural models consisting of frame elements for modeling beam, column and brace members, which is the common type of modeling for buildings and bridges. Structural walls are often modeled with beam elements at the centerline and rigid joint offsets, even though this does not properly account for the uplift effect that may be important for this type of lateral load resisting system.

An important decision in a structural analysis is to assume whether the relationship between forces and displacements is linear or nonlinear. Linear analysis for static and dynamic loads has been used in structural design for decades. Nonlinear analysis methods are widely used, because emerging performance-based guidelines require representation of nonlinear behavior. There are two major sources of nonlinear behavior. The first is a nonlinear relationship between force and deformation resulting from material behavior such as ductile yielding, stiffness and strength degradation or brittle fracture. The second type of nonlinear behavior is caused by the inclusion of large displacements in the compatibility and equilibrium relationships. This chapter presents the nonlinear methods of analysis for both types of behavior. Linear methods are a special case.

An earthquake analysis generally includes gravity loads and a representation of the ground motion at the site of the structure. Earthquake ground motion induces the mass in a structure to accelerate, and the resulting response history can be computed by dynamic analysis methods. In many design procedures it is common to perform a dynamic analysis with a response spectrum representation of the ground motion expected at the site (Chopra, 2001). For response history analysis, several analyses with different ground motion histories of the earthquake hazard are generally required. (See Chapter 5 for more information about the definition of earthquake ground motion in a structural analysis.)
For many design procedures, however, it is common to use equivalent static loads that represent the effects of the earthquake on the structure. Traditional design procedures use a static linear analysis with a response modification coefficient to represent the effects of ductile, nonlinear behavior. In contrast, newer design procedures utilize a nonlinear static (pushover) analysis to determine the force–displacement relationship for the structure, and the inelastic deformation of its members.

After a structural model and earthquake loading are defined, an analysis method is needed to compute the response. The governing equations are formed using equilibrium, compatibility and force–deformation relationships for the elements and the structure, and are expressed in terms of unknown displacements (or degrees of freedom, referred to as DOFs in this chapter). To elucidate the theory and provide a compact mathematical representation, the fundamental relationships are expressed using matrix algebra. Since the governing equations may have a large number of degrees of freedom, they must be solved numerically using a computer-based analysis method. Nearly all structural analyses for earthquake-resistant design are performed using software that incorporates one or more of the analysis methods presented in this chapter. Modern software generally includes graphical features for visualizing the forces and deformations computed from an analysis. Before using any new structural analysis software, the engineer should conduct an independent verification to ensure that the software provides correct solutions.

The structural analysis procedures used in earthquake-resistant design are summarized in Table 6.1. Recent guidelines for seismic rehabilitation of buildings pioneered the requirements for dynamic and nonlinear analysis procedures, particularly FEMA 356 (FEMA, 2000a) and the predecessor FEMA 273 (FEMA, 1997). The ATC-40 guidelines for reinforced concrete buildings (ATC, 1996b) emphasize the use of a nonlinear static (pushover) analysis procedure to define the displacement capacity for buildings. The classification of analysis procedures in Table 6.1 is generally applicable to design regulations for new buildings, such as in the 2000 NEHRP recommended provisions (FEMA, 2001) and recent guidelines for steel moment frame buildings (FEMA, 2000b, c) and for bridges (ATC, 1996a). These provisions and guidelines are required for the selection of the analysis procedure depending on the seismic design category, performance level, structural characteristics (e.g., regularity or complexity), response characteristics (e.g., the fundamental vibration period and participation of higher vibration modes), amount of data available for developing a model and confidence limits (in a statistical sense) for performance evaluation. Design provisions for structures with seismic isolation systems and supplemental energy dissipation generally require a dynamic analysis procedure.

The analysis procedures in Table 6.1 are in order of increasingly rigorous representation of structural behavior, but also increasing requirements for modeling and complexity of the analysis. As described in Section 6.2, plastic analysis only requires the equilibrium relationships, and is useful for capacity design procedures (Paulay and Priestley, 1992). For a given load distribution and flexural strength of members, plastic analysis gives the collapse load and the location of plastic hinges in members. The linear static procedure has been a traditional structural analysis method for earthquake-resistant design (UBC, 1997),

<table>
<thead>
<tr>
<th>Category</th>
<th>Analysis Procedure</th>
<th>Force–Deformation Relationship</th>
<th>Displacements</th>
<th>Earthquake Load</th>
<th>Analysis Method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Equilibrium</td>
<td>Plastic Analysis Procedure</td>
<td>Rigid-plastic</td>
<td>Small</td>
<td>Equivalent lateral load</td>
<td>Equilibrium analysis</td>
</tr>
<tr>
<td>Linear</td>
<td>Linear Static Procedure</td>
<td>Linear</td>
<td>Small</td>
<td>Equivalent lateral load</td>
<td>Linear static analysis</td>
</tr>
<tr>
<td></td>
<td>Linear Dynamic Procedure I</td>
<td>Linear</td>
<td>Small</td>
<td>Response spectrum</td>
<td>Response spectrum analysis</td>
</tr>
<tr>
<td></td>
<td>Linear Dynamic Procedure II</td>
<td>Linear</td>
<td>Small</td>
<td>Ground motion history</td>
<td>Linear response history analysis</td>
</tr>
<tr>
<td>Nonlinear</td>
<td>Nonlinear Static Procedure</td>
<td>Nonlinear</td>
<td>Small or large</td>
<td>Equivalent lateral load</td>
<td>Nonlinear static analysis</td>
</tr>
<tr>
<td></td>
<td>Nonlinear Dynamic Procedure</td>
<td>Nonlinear</td>
<td>Small or large</td>
<td>Ground motion history</td>
<td>Nonlinear response history analysis</td>
</tr>
</tbody>
</table>
but it does not represent the nonlinear behavior or the dynamic response of a structure caused by an earthquake ground motion. The simplest dynamic analysis method is based on a linear model of the structure, which permits use of vibration properties (frequencies and mode shapes) and simplification of the solution with a modal representation of the dynamic response. An estimate of the maximum structural response can be obtained with response spectrum analysis, or the maximum can be computed by response history analysis with specific earthquake ground motion records. Linear dynamic analysis methods are covered in depth by Chopra (2001).

Increasingly, engineers are using, and design guidelines are requiring, nonlinear analysis in the design process, because a severe earthquake ground motion is expected to deform a structure into the inelastic range. Nonlinear analysis methods can provide the relationship between a lateral load representing the effect of the earthquake ground motion and the displacements of the structure and deformations of the members. The results are often presented as a pushover or capacity curve for the structure. More detailed response history of a structure (sometimes called the seismic demand) can be computed by nonlinear dynamic analysis methods, particularly the cyclic response, degradation and damage measures for the members.

For the earthquake analysis of many types of structures it is reasonable to assume that the foundation and soil are rigid compared to the structure itself and that the supports of the structure move in phase during an earthquake ground motion. Soil–structure interaction, as described in Chapter 4, modifies the input motion to a structure because of wave propagation and energy dissipation in the soil, however, this phenomenon is not discussed in this chapter. For two-dimensional analysis the ground motion is specified in the horizontal and vertical directions; the two horizontal and the vertical ground motion components are specified for three-dimensional analysis. The assumption of uniform ground motion may not be valid for long-span bridges because of wave passage effects, differential site response and incoherence of the ground motion.

6.1.2 Models of Structures

A structural analysis is performed on a model of the structure — not on the real structure — so the analysis can be no more accurate than the assumptions in the model. The model must represent the distribution and possible time variation of stiffness, strength, deformation capacity and mass of the structure with accuracy sufficient for the purpose of the analysis in the design process.

All structures are three dimensional, but it is important to decide whether to use a three-dimensional model or simpler two-dimensional models. The analysis methods are the same whether the model is two-dimensional or three-dimensional. Generally, two-dimensional models are acceptable for buildings with regular configuration and minimal torsion; otherwise, a three-dimensional model is necessary with a representation of the floor diaphragms as rigid or flexible components. Analysis of bridges is generally based on three-dimensional models, although nonlinear analysis is typically used for two-dimensional models of bridge piers (ATC, 1996a, Priestley et al., 1996).

A structural model of a frame consists of an assembly of frame elements connected at nodal points (or nodes) in a global coordinate system, as illustrated in Figure 6.1. The geometry of the structural model is described by the position of the nodes in a global coordinate system, denoted by X, Y and Z. In the graphic representations of the structural model, nodes have a small black square (see Figure 6.1).

Two nodes define a frame element, which may be either straight or curved. This chapter is limited to straight elements because a curved element can always be approximated by several straight elements at the expense of increased modeling effort and computational cost. The element geometry is established in a local coordinate system x, y, z (see Figure 6.1). As will be shown later in this chapter, the force–deformation relationship for the element is obtained from the integration of functions of x along the element axis between the nodes. These functions represent the section forces (such as shear, bending and axial forces), the corresponding section deformations and the relationship between section forces and deformations.

The element response can be completely described by the relation between the force vector \( \mathbf{p} \) and the displacement vector \( \mathbf{u} \). For three-dimensional (3d) elements, the force vector has 12 components: at each
Methods of Analysis for Earthquake-Resistant Structures

Before concluding this section on structural modeling with frame elements, it is worth noting that for many structural systems the joints connecting members may be substantial in size and have modes of deformation that affect the system behavior. In such cases the model needs to account for at least the finite size of the joint region. The topic of joint deformation is more advanced than the scope of this chapter but models for joints can be included in the analysis methods presented herein.

Finite element methods with continuum elements (Bathe, 1995) can provide a more refined distribution of stress and strain in solid, plate and shell models of structural components such as walls, diaphragms and joints, but with the exception of walls, this level of refinement is generally not warranted for earthquake-resistant design. This chapter does not discuss the details of finite element methods for continuum elements, although most of the solution methods are applicable for models that include continuum finite elements as well as frame elements.

6.1.3 Loads and Boundary Conditions

Loads are specified forces applied to elements or nodes. Gravity loads may be applied to elements or considered as nodal loads depending on the gravity load path. The vector of nodal loads for a structure is denoted by $P$, with six components of force at each node for 3d problems and three components for 2d problems. In contrast with nodal loads, element loads are included in the element force–deformation relationship as distributed loads $w(x)$ defined in the local coordinate system for the element.

Each node undergoes translations and rotations that can be combined into a displacement vector of three translations and three rotations at each node in the 3d case. The displacements of all nodes are collected into a single displacement vector $U$ for the entire model in which each component is a degree of freedom. We separate the set of all global DOFs into two subsets: the DOFs with unknown displacement values and the DOFs with specified displacement value. Each DOF in the model must be included in one of the two sets. The unknown displacements are called the free DOFs and are denoted by $U_f$. The second set of displacements corresponds to the restrained DOFs and is denoted by $U_r$. The restrained DOFs are generally assigned a value of zero to indicate a fixed displacement, but nonzero support displacement problems can be considered. The selection of restrained displacements at the supports is an important step in the structural modeling, and the supports of a model are commonly identified with the symbols shown in Figure 6.2 for typical 2d cases. The arrows in Figure 6.2 indicate the restrained DOFs, and thus the corresponding support reactions of each support type.
Since the displacements are partitioned into two sets, so is the nodal force vector, \( \mathbf{P} \). The nodal forces at the free DOFs of the model are specified as nodal loads, and are denoted by \( \mathbf{P}_f \). For earthquake analysis this would normally include only the gravity loads with all other nodal loads equal to zero. The forces at the restrained DOFs are the support reactions and are denoted by \( \mathbf{P}_d \). These can be evaluated once the equations for the free DOFs are solved.

### 6.1.4 Notation

A consistent notation assists in elucidating the fundamental structural analysis concepts. In general, uppercase symbols are matrices or vectors representing the structural system (Table 6.2a), whereas lowercase symbols represent quantities associated with individual elements (Table 6.2b). Vectors and matrices are written in boldface.

#### 6.2 Equilibrium

All structural analysis methods in Table 6.1 require satisfaction of equilibrium. This section presents the fundamental equilibrium relationships for nodes and elements in the model.

### 6.2.1 Node Equilibrium

With imaginary cuts around each node we separate \( n_n \) nodal-free bodies and \( n_e \) element-free bodies, as shown in Figure 6.3, in which \( n_n \) is the number of nodes and \( n_e \) is the number of elements in the model. Three force equilibrium equations and three moment equilibrium equations must be satisfied for each node.

**TABLE 6.2  Notation for Structural Analysis**

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a) Symbols for Structural (Global) System</td>
<td></td>
</tr>
<tr>
<td>Global coordinate system for structure</td>
<td>( X, Y, Z )</td>
</tr>
<tr>
<td>Displacements of structure at DOFs</td>
<td>( \mathbf{U} )</td>
</tr>
<tr>
<td>Applied loads to structure DOFs</td>
<td>( \mathbf{P} )</td>
</tr>
<tr>
<td>Resisting forces for structure at DOFs</td>
<td>( \mathbf{P}_r )</td>
</tr>
<tr>
<td>Structure equilibrium matrix for DOFs</td>
<td>( \mathbf{B} )</td>
</tr>
<tr>
<td>Structure compatibility matrix for DOFs</td>
<td>( \mathbf{A} )</td>
</tr>
<tr>
<td>Structural stiffness, mass and damping matrices</td>
<td>( \mathbf{K}, \mathbf{M}, \mathbf{C} )</td>
</tr>
<tr>
<td>(b) Symbols for Elements</td>
<td></td>
</tr>
<tr>
<td>Local coordinate system for element</td>
<td>( x, y, z )</td>
</tr>
<tr>
<td>Basic element deformations</td>
<td>( \mathbf{v} )</td>
</tr>
<tr>
<td>Element nodal displacements in local coordinate system and global coordinate system</td>
<td>( \mathbf{u}, \mathbf{U} )</td>
</tr>
<tr>
<td>Basic element forces</td>
<td>( \mathbf{q} )</td>
</tr>
<tr>
<td>Element nodal forces in local and global coordinate system</td>
<td>( \mathbf{p}, \mathbf{P} )</td>
</tr>
<tr>
<td>Basic element flexibility and stiffness matrices</td>
<td>( \mathbf{f}, \mathbf{k} )</td>
</tr>
</tbody>
</table>

Since the displacements are partitioned into two sets, so is the nodal force vector, \( \mathbf{P} \). The nodal forces at the free DOFs of the model are specified as nodal loads, and are denoted by \( \mathbf{P}_f \). For earthquake analysis this would normally include only the gravity loads with all other nodal loads equal to zero. The forces at the restrained DOFs are the support reactions and are denoted by \( \mathbf{P}_d \). These can be evaluated once the equations for the free DOFs are solved.
Methods of Analysis for Earthquake-Resistant Structures

node-free body for the 3d case and two force equilibrium and one moment equilibrium equations for a node in the 2d case. Using the 2d case for simplicity and examining the nodal equilibrium equations, it is apparent that the equilibrium equations involve the summation of element forces at a node when all element forces are expressed in the global coordinate system. An element is identified with a superscript in parentheses and the correspondence between element DOF and global DOF can be supplied by an array, known as an id-array with the number of entries equal to the number of element DOFs. With the relationship between element and global DOF for each element, the force vector for an element $\mathbf{p}_{(el)}$ can be mapped to the contribution of the element force vector to the global resisting force vector, represented symbolically as $\mathbf{p}_{(el)} \rightarrow \mathbf{P}_{r}^{(el)}$, in which $\mathbf{P}_{r}^{(el)}$ has nonzero terms corresponding to the forces in the element, as represented by the id-array. Using this relationship the nodal equations of static equilibrium are written as:

$$\mathbf{P} - \mathbf{P}_f = 0 \quad \text{or} \quad \begin{pmatrix} \mathbf{P}_f \\ \mathbf{P}_d \end{pmatrix} = \mathbf{0} \quad \text{with} \quad \mathbf{P}_f = \sum_{el} \mathbf{P}_{r}^{(el)} = \mathbf{A}_{el} \mathbf{p}_{(el)}^{(el)} (6.1)$$

Equation 6.1 states that the applied forces $\mathbf{P}$ consisting of forces at free DOFs $\mathbf{P}_f$ and forces at restrained DOFs $\mathbf{P}_d$ are in equilibrium with the resisting forces $\mathbf{P}_r$, which are the sum of the element contributions $\mathbf{p}_{(el)}^{(el)}$. The mapping of element DOFs to global DOFs followed by summation of the element contributions, as denoted by the symbol $\mathbf{A}_{el}$, indicates assembly of all element contributions, which is known as the direct assembly procedure.

Equation 6.1 assumes that the loading is applied so slowly that the resulting accelerations at the free DOFs can be neglected. If this is not the case, the equations of static equilibrium are extended using Newton's second law stating that

$$\mathbf{P} - \mathbf{P}_f = \mathbf{M} \ddot{\mathbf{U}} = \mathbf{0} \quad (6.2)$$

in which $\ddot{\mathbf{U}}$ is the acceleration with respect to a fixed frame of reference. Equation 6.2 is also known as the equation of motion for a structural model. It is worthwhile to state explicitly the dependence of the resisting forces on the displacement and displacement rate (velocity) vector.

$$\mathbf{P} = \dot{\mathbf{P}}_{(U, \dot{U})}$$

which allows for velocity-dependent resistance (viscous damping).
6.2.2 Equilibrium of Basic System of Element Forces

Turning attention to the free body for a 3d frame element, there are 12 unknown forces, with six at the imaginary cuts at each end of the element but only six independent equilibrium equations. Figures 6.4a and b show the end forces in the global and local coordinate systems for a 2d frame element, with the quantities in the local system denoted by a bar. A rotation matrix can be used to transform vectors between the two coordinate systems.

Because the element forces need to satisfy the equilibrium equations they are not independent. We select a subset of element forces and express the remainder in terms of the subset to assure that the equilibrium of the free body is satisfied. The independent element forces are called the basic forces, $q$. In the 2d case there are three basic forces, and in this chapter we select the axial force and the two end moments, as shown in Figure 6.4c. The three equilibrium equations for the element-free body are used to express the remaining element forces in terms of the basic forces.

The equilibrium equations can be satisfied in the undeformed configuration, if the displacements are small relative to the dimensions of the structure. However, if the displacements are large, the equilibrium needs to be satisfied in the deformed configuration. The latter leads to nonlinear geometric effects, which are presented after the geometric compatibility relationships are defined for large displacements. Equilibrium of an element-free body in the undeformed configuration is shown in Figure 6.4c and stated as follows:
If element loads are included, (6.3) is modified to include the effect of the element loads, where $\mathbf{b}$ is the vector of element forces due to the loads on the element. Figure 6.4d shows these forces for the case of uniform transverse and longitudinal distributed loads. The matrix $\mathbf{b}$ represents the equilibrium transformation matrix from the basic system to the complete system of element forces in local coordinates. We noted earlier that the transformation of the element end forces from the local to the global coordinate system involves a rotation transformation of the end forces at each end, expressed by $\mathbf{p} = \mathbf{b} \mathbf{p}^g$. Combining 6.3 with the rotation to the global coordinate system gives the element forces in the global system expressed in terms of the basic forces and element loads:

$$\mathbf{p} = \mathbf{b} \mathbf{q} + \mathbf{p}_e$$

(6.4)

The statement of element equilibrium can be extended to other cases, such as for rigid-end offsets to represent finite joint size. The element forces at the ends of the deformable portion of the element are denoted by $\mathbf{p}_e$, and the equilibrium of the rigid-end offsets is shown in Figure 6.5. The equilibrium relationship is $\mathbf{p} = \mathbf{b} \mathbf{p}_e$.
and noting that $p = b_{b}b_{q} + b_{r}b_{r}$, the element forces become $p = b_{b}b_{q} + b_{b}b_{r}$. In a compact notation the element equilibrium in the undeformed configuration can be stated as

$$p = b_{g}q + p_{w} \quad \text{with} \quad b_{g} = b_{b}b_{b} \quad \text{and} \quad p_{w} = b_{b}b_{r}b_{r}$$  \hspace{1cm} (6.5)

in which the equilibrium matrix $b_{g}$ transforms the basic forces to the element end forces in global coordinates and $p_{w}$ accounts for the effect of element loading. The equilibrium matrix $b_{g}$ is made up of at least the product $b_{b}b_{b}$. The transformation of the dependent element forces $b_{r}b_{r}$ in 6.4 to the global coordinate system includes at least the rotation matrix $b_{b}$. After substituting 6.5 into the assembly operation in 6.1, the global resisting force vector is

$$p_{g} = A^{(d)}b_{r}^{(d)}q^{(d)} + p_{w}^{(d)}$$  \hspace{1cm} (6.6)

For convenience of later discussion in this chapter, the basic forces of all elements are collected into a single vector $Q$. After noting that the assembly operation in 6.6 only affects the rows of $p$ and thus only the rows of $b_{g}$ and $p_{w}$, 6.6 can be written in the compact form,

$$p_{g} = BQ + p_{w}$$  \hspace{1cm} (6.7)

The structure equilibrium matrix $B$ results from the assembly of the rows of $b_{g}$ while the columns are collected from each element. Partitioning 6.7 into free and restrained DOFs, as indicated in 6.1, the equilibrium of the structure is represented by

$$
\begin{pmatrix}
    p_{f} \\
    p_{d}
\end{pmatrix} =
\begin{pmatrix}
    B_f \\
    B_d
\end{pmatrix}Q + p_{w}
$$  \hspace{1cm} (6.8)

Equation 6.8 is very significant in structural analysis because it must be satisfied for any frame element, made of linear or nonlinear material, under the limitation that equilibrium is satisfied in the undeformed configuration. The number of equilibrium equations in the free partition, $n_{f}$, is the number of rows in $B_{f}$, and the number of columns is the number of unknown basic forces, $n_{q}$. For a statically determinate structure, $n_{f}$ and $n_{q}$ are equal, and $B_{f}$ is square and invertible, if the structure has a stable equilibrium configuration. Hence, for a statically determinate structure, given the applied forces at the free DOFs, $p_{f}$, we can solve the first partition in 6.8 for the unknown basic forces $Q$. If the support reactions are desired, the second partition in 6.8 can be evaluated for $p_{d}$.

For a statically indeterminate structure the number of unknown basic element forces, $n_{q}$, is greater than the number of equilibrium equations, $n_{f}$, at the free DOFs. Thus, the size of the $B_{f}$ matrix gives the degree of static indeterminacy $NOS = n_{q} - n_{f}$. In statically indeterminate structures, the equilibrium equations must be satisfied, but they are not sufficient to give a unique solution.

6.2.3 Lower Bound Theorem of Plastic Analysis

Since structures designed to resist earthquakes are rarely statically determinate, the most significant application of the structural equilibrium equations in 6.8 is for plastic analysis (Livesley, 1975). Assuming perfectly plastic material response requires that the basic element forces satisfy the plastic condition $|q| \leq Q_{p}$, where $Q_{p}$ are the plastic capacities of the elements. The applied forces at the free DOFs in 6.8 are written as the product of a load factor $\lambda$ and a reference force vector $p_{w}$, that gives the distribution of the applied loads, such as the equivalent lateral earthquake loads. The lower bound theorem of plastic analysis states that the collapse load factor $\lambda_{c}$ is the largest load factor that satisfies the equilibrium equations in 6.8 and the plastic condition. We can write this as follows:
Considering that the plastic capacity may be different under positive basic forces than under negative ones, and collecting the unknowns of the problem $\lambda$ and $Q$ into a single vector, the equations in 6.9 are written in the compact form of a linear programming problem:

$$\begin{align*}
\lambda_c &= \max [1, \frac{\lambda}{Q}] \\
\text{for } \begin{bmatrix} P_{\text{ref}} & -B_i & \lambda \end{bmatrix} &= 0 \text{ and } \begin{bmatrix} 0 & I & \lambda \\ 0 & 1 & Q \end{bmatrix} \leq \begin{bmatrix} Q_p \end{bmatrix}
\end{align*}$$

(6.10)

$Q_p^+$ is the plastic capacity for a positive basic force and $Q_p^-$ is the capacity for a negative basic force, respectively (both in absolute value), and $I$ is the $nq \times nq$ identity matrix. The linear programming problem in 6.10 can be readily solved using the simplex method with widely available mathematical software packages such as Matlab or Mathcad.

The solution of the linear programming problem in 6.10 yields a unique collapse load factor $\lambda_c$ according to the lower bound theorem of plastic analysis. Even though the collapse load factor is unique, the basic forces at collapse $Q_c$ are unique only if a complete collapse mechanism forms. This requires that $\text{NOS}+1$ basic forces reach the plastic capacity (in other words, that $\text{NOS}+1$ plastic hinges form), where $\text{NOS}$ is the degree of static indeterminacy. This requirement derives from the equilibrium equations in 6.10, i.e., from the requirement that

$$\begin{bmatrix} P_{\text{ref}} & -B_i & \lambda \end{bmatrix} = 0$$

(6.11)

There are $nq+1$ unknowns and $n_f$ available equations of equilibrium in 6.11. Because $\text{NOS} = nq - n_f$, there are $\text{NOS}+1$ more unknowns than available equations of equilibrium. With the value at $\text{NOS}+1$ basic forces equal to the corresponding plastic capacity at collapse, there are as many unknowns in 6.11 as the number of equilibrium equations. A partial mechanism forms if fewer than $\text{NOS}+1$ basic forces reach the plastic capacity. In this case, there exist an infinite number of combinations of the remaining basic forces that satisfy the equilibrium equations in 6.11. A unique solution can only be obtained with an additional assumption about the force–deformation behavior of the elements before reaching the plastic capacity, which we will pursue later in this chapter.

### 6.3 Geometric Compatibility

The statement of geometric compatibility is analogous to the process of establishing the equilibrium equations of the structural model. Each node can undergo three translations and three rotations in the 3d case; for the 2d case each node undergoes two translations and one rotation. The displacements of all nodes in the model are collected in the displacement vector $U$. From the compatibility between elements and nodes, the displacements at the end of an element are equal to the corresponding DOF displacements at the nodes (see Figure 6.6). This correspondence between global and element DOFs is provided again by the $id$-array of each element, and the compatibility can be written symbolically as

$$u^{(el)} = U_{id}$$

(6.12)

Equation (6.12) indicates that an extraction from the global displacement vector $U$ of only the degrees of freedom corresponding to the entries in the $id$-array of the element $el$ takes place. The length of the vectors in 6.12 is, consequently, equal to the number of element DOFs.
As described in Section 6.1.3, the displacement vector for the structure is partitioned into free DOFs \( U_f \), which are unknown at the start of the analysis, and the displacements at the restrained DOFs \( U_d \), which are specified.

### 6.3.1 Displacement–Deformation Relationship under Large Displacements

Figure 6.7 shows a two-node frame element in the undeformed and deformed configuration under given end displacements. Since the rigid-body displacement of the element does not generate element deformations, Figure 6.7 shows three stages for representing the relationship between displacement and deformation. Figure 6.7a provides the overview of the entire process and identifies the rigid-body translation of the element. The relative displacements \( \Delta \mathbf{u}_i \) and \( \Delta \mathbf{u}_j \) in the local reference system are convenient for describing the relative position of node \( j \) with respect to node \( i \). This relative translation results in an extension \( \mathbf{v}_i \) of the element in Figure 6.7b. As long as the ends follow the rigid-body rotation of the element axes through the angle \( \beta \), no other deformation arises in Figure 6.7b, as illustrated by the rotation of the black squares representing the end nodes. Since the end rotations are independent DOFs, each end is subjected to an additional rotation past angle \( \beta \) and the element can deform, as shown in Figure 6.7c. The rotation of the end tangent to the deformed shape relative to the chord line for the element results from flexural deformation. There are two rotations caused by the flexural deformations, \( \mathbf{v}_1 \) and \( \mathbf{v}_2 \), in Figure 6.7c, one at each element end. Counterclockwise rotations measured from the chord to the tangent are positive, consistent with the sign convention for the end moments.

With the definition of element deformations, \( \mathbf{v} \), it is now possible to derive the relationship between the element deformations and the end displacements, \( \mathbf{u} \), in the local coordinate system. Using Figures 6.7b and 6.7c, the element deformations for the general case of large displacements and moderate deformations can be given as

As described in Section 6.1.3, the displacement vector for the structure is partitioned into free DOFs \( U_f \), which are unknown at the start of the analysis, and the displacements at the restrained DOFs \( U_d \), which are specified.

### 6.3.1 Displacement–Deformation Relationship under Large Displacements

Figure 6.7 shows a two-node frame element in the undeformed and deformed configuration under given end displacements. Since the rigid-body displacement of the element does not generate element deformations, Figure 6.7 shows three stages for representing the relationship between displacement and deformation. Figure 6.7a provides the overview of the entire process and identifies the rigid-body translation of the element. The relative displacements \( \Delta \mathbf{u}_i \) and \( \Delta \mathbf{u}_j \) in the local reference system are convenient for describing the relative position of node \( j \) with respect to node \( i \). This relative translation results in an extension \( \mathbf{v}_i \) of the element in Figure 6.7b. As long as the ends follow the rigid-body rotation of the element axes through the angle \( \beta \), no other deformation arises in Figure 6.7b, as illustrated by the rotation of the black squares representing the end nodes. Since the end rotations are independent DOFs, each end is subjected to an additional rotation past angle \( \beta \) and the element can deform, as shown in Figure 6.7c. The rotation of the end tangent to the deformed shape relative to the chord line for the element results from flexural deformation. There are two rotations caused by the flexural deformations, \( \mathbf{v}_1 \) and \( \mathbf{v}_2 \), in Figure 6.7c, one at each element end. Counterclockwise rotations measured from the chord to the tangent are positive, consistent with the sign convention for the end moments.

With the definition of element deformations, \( \mathbf{v} \), it is now possible to derive the relationship between the element deformations and the end displacements, \( \mathbf{u} \), in the local coordinate system. Using Figures 6.7b and 6.7c, the element deformations for the general case of large displacements and moderate deformations can be given as
In 6.13 we have used the definition of the Lagrange strain for the length change of the element. In a typical structural analysis for earthquake engineering, the engineering strain is a sufficient approximation, so the extension is \( v_1 = L_x - L \). The arctan function when expanded by a Taylor series about the point \( \Delta \bar{u}_x = 0 \), \( \Delta \bar{u}_y = 0 \) gives for the chord rotation:

\[
\beta = \arctan \left( \frac{\Delta \bar{u}_x}{L + \Delta \bar{u}_y} \right)
\]

Similarly, expansion of the first equation in 6.13 gives the change in length of the element:

\[
v_1 = \Delta \bar{u}_x + \frac{1}{2} \left( \frac{\Delta \bar{u}_x}{L} \right)^2 \Rightarrow
\]

Factoring \( L \) out on the right-hand side of the above expression gives

\[
v_1 = L \left[ \frac{\Delta \bar{u}_x}{L} + \frac{1}{2} \left( \frac{\Delta \bar{u}_x}{L} \right)^2 \right]
\]

In earthquake engineering applications the axial strain \( \frac{\Delta \bar{u}_x}{L} \) has values of order \( 10^{-3} \) to \( 10^{-2} \). In this case the first quadratic term on the right-hand side of 6.15 is of order \( 10^{-6} \) to \( 10^{-4} \) and can be neglected. The relative transverse displacement (commonly called the drift), \( \frac{\Delta \bar{u}_y}{L} \), is of order \( 10^{-2} \) to \( 10^{-1} \), so that the second quadratic term in 6.15 can be of the same order as the linear term. In such situations the element deformation should be approximated with

\[
v_1 = L \left[ \frac{\Delta \bar{u}_x}{L} + \frac{1}{2} \left( \frac{\Delta \bar{u}_x}{L} \right)^2 \right]
\]

The relation between the displacement components in the global and the local coordinate systems uses the rotation transformation for the translation components, whereas the rotations in the plane are not affected. The rotation angle for the transformation of the translation components corresponds to the angle of the undeformed element \( x \)-axis relative to the global \( X \)-axis (note that the positive element \( x \)-axis points from node \( i \) to node \( j \)). The translation components at each element end are transformed independently. The transformation of the end displacements is expressed symbolically by \( \bar{u} = a_u u \).
6.3.2 Linear Approximation of Displacement–Deformation Relation

In the cases where the relative transverse displacement of the element, \( \Delta \bar{y} \), is of order \( 10^{-2} \) or less, the quadratic term in 6.16 can be neglected and a linear compatibility relation remains \( v_i = \Delta \bar{y} \). In the expression for the angle \( \beta \), the term \( \Delta \bar{y} \) is very small relative to unity and can be neglected in any case. With these approximations, the compatibility relations between element deformations and end displacements in the local coordinate system become linear:

\[
\begin{align*}
v_1 &= \Delta \bar{y}_x = \bar{u}_x - \bar{u}_i, \\
v_2 &= \bar{u}_i - \frac{\Delta \bar{y}}{L} = \bar{u}_i - \frac{\bar{u}_x - \bar{u}_i}{L}, \\
v_3 &= \bar{u}_o - \frac{\Delta \bar{y}}{L} = \bar{u}_o - \frac{\bar{u}_o - \bar{u}_i}{L},
\end{align*}
\]

or

\[
v = \begin{bmatrix} v_1 \\ v_2 \\ v_3 \end{bmatrix} = \begin{bmatrix} -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & 1 & 1 & 0 & -1 & 0 \\ 0 & 1 & 0 & 0 & -1 & 1 \end{bmatrix} a \bar{u} = a \bar{u} \quad (6.17)
\]

Comparing 6.17 with 6.3, we note that the compatibility matrix \( a \) is the transpose of the equilibrium matrix \( b \). The same relation holds for matrices \( a_e \) and \( b_e \). If the element has rigid-end offsets, the compatibility matrix \( a \) between the displacements at the ends of the deformable portion of the element and those at the ends of the complete element is similarly the transpose of the equilibrium matrix \( b \).

The generality of this observation will be proven in the following section.

After combining the compatibility relationships, the element deformations can be expressed in terms of the element end displacements in the global coordinate system:

\[
v = a \bar{u} = a a_e u = a_a u
\]

(6.18)

The compatibility transformation matrix \( a_a \) is the transpose of the equilibrium matrix \( b_a \) in 6.5. With 6.12 the basic element deformations can now be expressed in terms of the global displacement vector components that correspond to the element as \( v = a a u = a_a \bar{U}_d \).

It is worthwhile to collect the element deformations into a single vector \( V \) for the structural model. In this process the element compatibility matrices can be combined with the aid of the \( sl \)-array to give the structural compatibility matrix \( A \). After this combination, the compatibility between the element deformations and the free and restrained DOFs at the nodes assumes the following form:

\[
V = A U = \begin{bmatrix} A_f & A_d \end{bmatrix} \begin{bmatrix} U_f \\ U_d \end{bmatrix}
\]

(6.19)

From the process of forming the structural compatibility matrix \( A \), we observe that it is the transpose of the equilibrium matrix \( B \) in 6.7 and 6.8. The same is obviously true for submatrices \( A_f \) and \( B_f \) and \( A_d \) and \( B_d \), respectively. In practice, only one of these matrices need be established and the other is obtained as its transpose. In the following we assume that the compatibility matrix \( A \) is assembled and the equilibrium matrix is obtained by transposition. Thus, we alternate our reference to the equilibrium matrix as \( B \) or \( A^T \), as circumstances require.
6.3.3 Compatibility Relationship for Elements with Moment Releases

It is worth looking in more detail into the case of a frame element with a moment release at one end because of later consideration of plastic hinges. A moment release at one end is achieved by introducing a hinge at the end. Referring to Figure 6.8 the element deformations $\mathbf{v}$ are measured from the chord to the tangent at the element side of a hinge at one end or the other. If an end moment is released to zero, the rotations at the ends are no longer independent, and we will show later that for a prismatic, linear elastic beam the deformation at a moment release is one half the value of the deformation at the opposite end, but of opposite sign. The total deformations $\mathbf{v}$ at the element ends are measured from the chord to the tangent at the node side of the hinge. These are the sum of the element deformations $\mathbf{v}$ and the hinge deformations $\mathbf{v}_h$. The following relationships for a moment release at end $i$ or end $j$ can be established from Figure 6.8:

For convenience, the geometric relationships for the two cases can be combined into a single relation by introducing a binary variable as moment release code for each end. Variable $mr$ assumes the value 0 when the end is continuous, and the value 1 when a release is present at the corresponding end. With this variable we can write the above transformation relations in a compact form,

$$
\begin{bmatrix}
    v_2 \\
    v_3
\end{bmatrix}
= \begin{bmatrix}
    1 - mr_i & -\frac{1}{2}(1 - mr_j)mr_i \\
    -\frac{1}{2}(1 - mr_i)mr_j & 1 - mr_j
\end{bmatrix}
\begin{bmatrix}
    v_2 \\
    v_3
\end{bmatrix}
$$

which now holds also for the case without moment releases, a moment release at either end and moment releases at both ends. After combining the flexural deformations with axial deformation, which is unaffected by the moment releases, the basic compatibility relationship is

$$
\mathbf{v} = a_h \mathbf{v} \quad \text{with} \quad a_h = \begin{bmatrix}
    1 & 0 & 0 \\
    0 & 1 - mr_i & -\frac{1}{2}(1 - mr_j)mr_i \\
    0 & -\frac{1}{2}(1 - mr_i)mr_j & 1 - mr_j
\end{bmatrix}
$$

(6.20)
A precise definition of the hinge rotation will be useful when plastic hinges are considered. The hinge rotations \( v_h \) are the difference between the deformations \( \tilde{v} \) and the element deformations \( v \). Consequently, hinge rotations are measured from the tangent at the element side of the hinge to the tangent at the node side. We can, therefore, write the hinge rotation as

\[
v_h = \tilde{v} - v = (1 - a_h) \tilde{v}
\]

where \( I \) is the identity matrix, and the total end deformations \( \tilde{v} \) are given by 6.18. In conclusion, the geometric compatibility relationship \( v = a_h L u \) covers all cases of frame elements with or without end moment releases. This result will be used in the subsequent discussion of plastic hinge rotations and in the development of the force–deformation relationship of elasto-plastic elements.

### 6.4 Equilibrium in Deformed Configuration

With the definition of the element compatibility relationship under large displacements we can now establish the equilibrium relationship for an element in the deformed configuration. Assuming that the basic forces act on the deformed element, \( q_1 \) acts always along the deformed element chord and it changes orientation as the element deforms. This approach is known as corotational formulation (Crisfield, 1990, 1991), but other names such as member-bound reference system or physical coordinates have also been used (Argyris, et al., 1979, Elias, 1986).

The element end forces in the reference frame of the deformed element can be expressed in terms of the basic forces by satisfying the equilibrium equations of the element-free body in the deformed configuration, as illustrated in Figure 6.9. These equations are identical to those in Figure 6.4c except for the fact that the deformed element length \( L_e \) is used in place of \( L \). Subsequently, all forces are transformed from the member-bound coordinate system to the local coordinate system by a rotation through angle \( \beta \), defined as the angle between the chord of the deformed element and the undeformed position (Figure 6.9). The equilibrium equations are

\[
\begin{pmatrix}
\vec{p}_1 \\
\vec{p}_2 \\
\vec{p}_3 \\
\vec{p}_4
\end{pmatrix}
= \begin{pmatrix}
\frac{L + \Delta L}{L_e} & -\frac{\Delta L}{L_e} & \frac{\Delta L}{L_e} \\
\frac{L}{L_e} & -\frac{L}{L_e} & \frac{L}{L_e} \\
\frac{L}{L_e} & 1 & 0 \\
\frac{L + \Delta L}{L_e} & \frac{\Delta L}{L_e} & \frac{\Delta L}{L_e}
\end{pmatrix}
\begin{pmatrix}
q_1 \\
q_2 \\
q_3 \\
q_4
\end{pmatrix}
= \begin{pmatrix}
b_1 \\
b_2 \\
b_3 \\
b_4
\end{pmatrix}
\]

(6.21)
This relationship is equivalent to 6.3 except that equilibrium is now satisfied in the deformed configuration. The subscript of the equilibrium matrix $b_n$ is a reminder that the equilibrium depends on the end displacements. For applications in earthquake engineering analysis, it is reasonable to approximate the equilibrium matrix in 6.21 by expanding the terms of $b_n$ in a Taylor series and including only terms that have a consistent order of magnitude. The expansion of the terms in the equilibrium matrix is

$$
\left( \frac{L + \Delta \bar{u}_n}{L_n} \right) = 1 - \left( \frac{\Delta \bar{u}_n}{L} \right)^2
$$

$$\Delta \bar{u}_n = \Delta \bar{u}_n \left[ 1 - \frac{\Delta \bar{u}_n}{L} - \frac{1}{2} \left( \frac{\Delta \bar{u}_n}{L} \right)^2 \right]
$$

$$\left( \frac{L + \Delta \bar{u}_n}{L_n^2} \right)^2 = 1 - \left[ 1 - \frac{\Delta \bar{u}_n}{L} - \left( \frac{\Delta \bar{u}_n}{L} \right)^2 \right]
$$

Recalling the order of magnitude of the terms $\Delta \bar{u}_n/L$ and $\Delta \bar{u}_n/L^2$ in comparison with unity, we conclude that we can neglect the factors in square brackets and the quadratic term in the first expression. With these approximations the equilibrium relations in 6.21 simplify to

$$\begin{align*}
\bar{p} &= \begin{bmatrix}
-1 & -\frac{\Delta \bar{u}_n}{L^2} & -\frac{\Delta \bar{u}_n}{L^2} \\
\Delta \bar{u}_n & 1 & 1 \\
0 & 0 & 0 \\
\frac{\Delta \bar{u}_n}{L^2} & -\frac{1}{L} & \frac{1}{L} \\
0 & 0 & 1
\end{bmatrix}
\begin{bmatrix}
q_1 \\
q_2 \\
q_3
\end{bmatrix} = b_n \begin{bmatrix} q_1 + q_3 \end{bmatrix} = b_n q
\end{align*}
$$

The relationship in 6.22 is a consistent representation of the effect of large displacements on the equilibrium of a frame element. For many models undergoing moderate deformations, it is reasonable to further simplify 6.22 by neglecting the contribution of the shear force $\frac{q_1 + q_3}{L}$ to the axial force, because of the small magnitude of shear relative to the axial force, and because the transverse deformation, $\frac{\Delta \bar{u}_n}{L}$, does not exceed 0.1 in most cases (about 10% drift of the element). With this approximation 6.22 simplifies further to

$$\begin{align*}
\bar{p} &= \begin{bmatrix}
-1 & 0 & 0 \\
\Delta \bar{u}_n & 1 & 1 \\
0 & 0 & 0 \\
\frac{\Delta \bar{u}_n}{L^2} & 0 & 0 \\
0 & 0 & 1
\end{bmatrix}
\begin{bmatrix}
q_1 \\
q_2 \\
q_3
\end{bmatrix} = b_{n1} q
\end{align*}
$$
The approximate equilibrium equation in 6.23 is often used in the so-called P-Δ analysis of structures. The transformation of the end forces \( \bar{p} \) from the local to the global coordinate system and any further equilibrium transformations addressed earlier are not affected by the equilibrium transformation of the basic system in Figure 6.9. Thus, the general case of element equilibrium can be expressed as

\[
p = b^T \delta q
\]

By selecting the transformation matrix \( b \) depending on the needs of the analysis, one can accommodate large displacements with \( b \) from 6.21, P-Δ geometry with \( b = b_{pa} \) from 6.23 and linear geometry under small displacements with \( b = b \) from 6.5.

### 6.5 Principle of Virtual Work

The virtual work principles are convenient representations of the equations of equilibrium and the conditions of geometric compatibility. With virtual work we can express the equilibrium and compatibility equations, which are vectorial, by a scalar work equation. These principles are derived by starting from either the equilibrium or compatibility equations of an element, converting these into integral form by multiplication with a virtual field, performing integration by parts that introduces the boundary terms and then summing up all element contributions. In the following we only state the final result of the derivation.

#### 6.5.1 Virtual Work Principles for the Structure

The principle of virtual displacements is equivalent to the satisfaction of the equations of equilibrium in the structure. It states that the work done by a set of virtual displacements on the external forces (external work) is equal to the work done by the compatible virtual deformations on the element forces (internal work), if the external forces are in equilibrium with the element forces, and vice versa. In the absence of element loads the principle of virtual displacements is

\[
\delta U^T P = \delta V^T Q
\]  

(6.24)

where the virtual displacements \( \delta U \) and deformations \( \delta V \) satisfy the compatibility requirement as \( \delta V = A \delta U \) in 6.19. Substituting this condition for the virtual displacements in 6.24 gives

\[
\delta U^T P = \delta V^T Q = \delta U^T A^T Q \quad \rightarrow \quad \delta U^T (P - A^T Q) = 0
\]  

(6.25)

If 6.25 holds for arbitrary virtual displacements \( \delta U \), then \( P - A^T Q = 0 \) and the converse is also true. Therefore the principle gives the equilibrium equations, \( P - A^T Q = 0 \) in 6.7, demonstrating that the equilibrium matrix is equal to the transpose of the compatibility matrix, \( B = A^T \).

The principle of virtual forces is equivalent to the satisfaction of the conditions of geometric compatibility. It states that, if the deformations are compatible with the displacements, then the complementary work of a set of virtual external forces on the displacements is equal to the work of the virtual internal forces that are in equilibrium with the external forces on the deformations. The reverse is also true. In compact form the principle of virtual forces is

\[
\delta P^T U = \delta Q^T V
\]  

(6.26)

Substituting the equilibrium requirement for the virtual forces \( \delta P = B \delta Q \), into 6.26 gives
which provides the geometric compatibility requirement in 6.19 along with the condition that $B^T = A$.

### 6.5.2 Virtual Work Principles for an Element

We now turn our attention to an individual frame element, see for example, Figures 6.4 and 6.7. The external work is done by the end forces on the corresponding displacements, whereas the internal work is done by the basic forces on the corresponding deformations. Without further derivation, we state this requirement as

$$\delta u^T \mathbf{p} = \delta v^T \mathbf{q}$$

(6.27)

The condition of geometric compatibility of the virtual displacements is $\delta v = a_i \delta u$, which upon substitution into 6.27 gives

$$\delta u^T \mathbf{p} = \delta v^T \mathbf{q} \quad \Rightarrow \quad \delta u^T (\mathbf{p} - a_i \mathbf{q}) = 0 \quad \Rightarrow \quad \mathbf{p} - a_i \mathbf{q} = 0$$

(6.28)

Comparing 6.28 with the element equilibrium equations in 6.5 shows that $b_i = a_i^T$.

We can generalize this fact by stating that if a set of displacements transforms from one system to another according to the relationship $v = a_i u$, then the forces corresponding to these displacements transform according to the contragradient relationship $p = a_i^T q$. Similarly, if a set of forces transforms from one system to another according to $p = b_i q$, then the corresponding displacements transform according to $v = b_i^T u$. Consequently, we only need to establish either the force or displacement transformation relationship from one coordinate system to another using the equilibrium or compatibility conditions, respectively.

Returning now to the virtual work statement for a single element, we can derive the internal work from the integral of the stress product with the corresponding virtual strains over the element volume $V$:

$$\delta v^T \mathbf{q} = \int_V \delta \mathbf{e}^T \mathbf{\sigma} dV$$

(6.29)

where $\mathbf{e}$ is a vector containing the components of the strain tensor and $\mathbf{\sigma}$ is a vector with the corresponding components of the stress tensor arranged in the same order. In many applications of nonlinear structural analysis, we limit ourselves to the internal work of the axial stress $\sigma_x$ and shear stress $\tau$ on the axial strain $\varepsilon_x$ and shear strain $\gamma$, respectively. In this case 6.29 reduces to

$$\delta v^T \mathbf{q} = \int_V (\delta \varepsilon_x \sigma_x + \delta \gamma \tau) dV$$

(6.30)

From the principle of complementary virtual work for the element we obtain the corresponding compatibility relationship for the element:

$$\delta \mathbf{q}^T \mathbf{v} = \int_V (\delta \varepsilon_x \sigma_x + \delta \gamma \tau) dV$$

These virtual work statements will be used in the next section for deriving the force–deformation relationships for linear and nonlinear elements.
6.5.3 Upper Bound Theorem of Plastic Analysis

We return to the problem of plastic analysis and complement the earlier development in Section 6.2.3 with the determination of the plastic collapse load factor by the upper bound theorem of plastic analysis. To state the latter it is necessary to formulate the external and internal plastic work past the point of maximum load. The assumption of perfect plasticity leads to the conclusion that only plastic deformation increments arise in the elements once a collapse mechanism has formed. The plastic deformation increments must satisfy the conditions of geometric compatibility for the free DOFs, which is written in a form similar to 6.19:

\[
\Delta V_p = A_f \Delta U_f
\]  

(6.31)

The external plastic work increment is \(\lambda P_{\text{ref}}^T \Delta U_f\). The internal work consists of the product of the plastic capacities of the elements and the plastic deformation increments. Because the plastic capacities have been defined as absolute values in 6.9, the plastic deformation increments are defined as \(\Delta V_p^+ = \Delta V_p\) if \(\Delta V_p \geq 0\), otherwise it is zero; \(\Delta V_p^- = -\Delta V_p\) if \(\Delta V_p \leq 0\), otherwise it is zero. With this definition the internal plastic work increment becomes \(Q_p^+ \Delta V_p^+ + Q_p^- \Delta V_p^-\) and 6.31 changes to \(\Delta V_p^+ - \Delta V_p^- = A_f \Delta U_f\), noting moreover that \(\Delta V_p^+ \geq 0\) and \(\Delta V_p^- \geq 0\).

The upper bound theorem of plastic analysis states that the collapse load factor \(\lambda_c\) is the smallest load factor satisfying the condition of incremental plastic work and geometric compatibility. This is stated in a compact form as

\[
\lambda_c = \min \lambda \quad \text{for} \quad \lambda P_{\text{ref}}^T \Delta U_f = Q_p^+ \Delta V_p^+ + Q_p^- \Delta V_p^- \quad \text{and} \quad \Delta V_p^+ \geq 0, \Delta V_p^- \geq 0
\]  

(6.32)

Equation 6.32 can be written in the standard form of linear programming by constraining \(P_{\text{ref}}^T \Delta U_f = 1\) and noting that the unknowns of the problem are now \(\Delta U\), \(\Delta V_p^+\), and \(\Delta V_p^-\) (Livesley, 1975). With these considerations, 6.32 is written in a compact form by collecting the unknowns in a single vector:

\[
\lambda_c = \min 0 \ Aug 0 \left\{ \begin{array}{c} \Delta U_f \cr \Delta V_p^+ \cr \Delta V_p^- \end{array} \right\} \text{ for } \left[ \begin{array}{ccc} P_{\text{ref}}^T & 0 & 0 \\ 0 & A_f & 1 \\ -A_f & -1 & -1 \end{array} \right] \left\{ \begin{array}{c} \Delta U_f \\ \Delta V_p^+ \\ \Delta V_p^- \end{array} \right\} = \left\{ \begin{array}{c} 1 \\ 0 \end{array} \right\}
\]  

(6.33)

Comparing 6.33 with 6.10 and recalling that \(A_f = B_j^T\), we conclude that the upper bound theorem of plastic analysis in 6.33 is the dual problem of linear programming to the lower bound theorem in 6.10, so that the solution of one satisfies the other. Consequently, the collapse load factor from either 6.10 or 6.33 is the unique solution of the plastic analysis problem. It is also interesting to observe from 6.31 that the columns of the compatibility matrix represent the independent collapse mechanisms of the structural model. The dependent collapse mechanisms can be obtained by linear combination of these columns.

6.5.4 Example of Plastic Analysis

The two-story frame in Figure 6.10a (Horne and Morris, 1982) illustrates the concepts of plastic analysis. With the assumption that axial deformations are negligibly small, there are ten free DOFs, as shown in Figure 6.10b. The plastic moment capacities of the columns and girders are enclosed in a circle in Figure 6.10a and the reference loading is also shown.
The first step consists in setting up the compatibility matrix $A_j$ relating the element deformations to the DOFs of the model. The model has eight elements with two deformations each, since axial deformations are neglected. Thus, the compatibility matrix has 16 rows and 10 columns. The deformed shapes for the vertical and horizontal translation DOFs and for one rotation DOF are shown in Figure 6.11.
Denoting the story height by \( h \) and the girder span by \( 2l \), the compatibility matrix is

\[
\begin{bmatrix}
0 & 0 & 0 & \frac{1}{h} & 0 & 0 & 0 & 0 & 0 \\
1 & 0 & 0 & \frac{1}{h} & 0 & 0 & 0 & 0 & 0 \\
1 & -\frac{1}{l} & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & -\frac{1}{l} & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{1}{l} & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{1}{l} & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{1}{h} & 1 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{1}{h} & 0 & 0 & 0 & 0 & 0 \\
1 & 0 & 0 & -\frac{1}{h} & 0 & 0 & 0 & 0 & \frac{1}{h} \\
0 & 0 & 0 & -\frac{1}{h} & 0 & 1 & 0 & 0 & \frac{1}{h} \\
0 & 0 & 0 & 0 & 0 & 1 & -\frac{1}{l} & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & -\frac{1}{l} & 1 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 1 \\
0 & 0 & 0 & -\frac{1}{h} & 0 & 0 & 0 & 0 & \frac{1}{h} \\
0 & 0 & 0 & -\frac{1}{h} & 1 & 0 & 0 & 0 & \frac{1}{h}
\end{bmatrix}
\]

The columns of the compatibility matrix represent the independent collapse mechanisms of the frame. Columns 1, 3, 5, 6, 8, 10 represent the joint mechanisms, columns 2 and 7 the beam mechanisms and columns 4 and 9 the story mechanisms. Figure 6.12 shows the beam mechanisms and the lower story mechanism in which a gray circle indicates a plastic hinge. Plastic rotations are measured from the tangent at the element side of the hinge to the tangent at the node side. Consequently, the lower girder rotation at the left end in Figure 6.12a is negative. The similarity of the shape for the mechanism with the corresponding deformed shape in Figure 6.11 is evident.

The equilibrium equations of the problem are \( \lambda P_{mi} = B_iQ = A_i^TQ \). We can elect to solve either the lower bound problem in 6.10 or the upper bound problem in 6.33. With either approach the collapse load of the frame is \( \lambda_c = 2.50 \). Only six plastic hinges form at collapse, as shown in Figure 6.13b, indicating a partial collapse mechanism since NOS = 6. Consequently, there are more unknown \( Q \) values than the available equations of equilibrium in 6.11, and the moment distribution in Figure 6.13a is not unique. In this regard, the double hinge at midspan of the upper girder counts as one.
6.6 Force–Deformation Relationships

The previous sections have developed the equilibrium and compatibility relationships for a frame element, which only depend on the element geometry and not on the constituent materials of the element. It is now necessary to relate the element basic forces to the corresponding deformations. To do so in a systematic manner we start with the consideration of the section response and show how it is established by integration of material response. Subsequently, we briefly describe a few material models used in the examples of this chapter.

6.6.1 Section Response

We return to the virtual work statement for an element in 6.30 and rewrite the integral over the element volume as integration over the section at a location $x$ followed by integration over the element length:
The strain and stress are functions of the position \( x \) along the element axis and the position within the cross section specified in local coordinates \( y \) and \( z \). The axial strain at point \( M \) in Figure 6.14 can be written as the product of two functions,

\[
\varepsilon_\alpha(x, y, z) = a_s(y, z) \varepsilon(x)
\]

where \( \varepsilon(x) \) are the section deformations and \( a_s(y, z) \) represents the strain distribution at section \( x \). When shear deformations are small, Bernoulli's assumption of plane sections remaining plane proves an excellent approximation. In this case the strain distribution at section \( x \) is

\[
a_s(y, z) = \begin{bmatrix} 1 & -y & z \end{bmatrix}
\]

and the section deformation vector, \( \varepsilon(x) \), consists of the axial strain at the coordinate origin \( \varepsilon_{\alpha} \), the curvature about the \( z \)-axis \( \kappa_z \), and the curvature about the \( y \)-axis \( \kappa_y \). Figure 6.14 shows an arbitrary section. The sign convention in 6.36 follows the right-hand rule for rotation and the definition of tensile strain as positive. The integration of the virtual work expression over the cross-section area \( A \) becomes

\[
\int_A \delta \varepsilon \sigma dA = \delta \varepsilon^T(x) \int_A a_s^T(y, z) \sigma dA = \delta \varepsilon^T(x) \int_A \begin{bmatrix} 1 & -y & z \end{bmatrix} \sigma dA = \delta \varepsilon^T(x) s(x)
\]

and the virtual work for the element in 6.34 can be written as

\[
\delta v^T q = \int_x \delta \varepsilon^T(x) s(x) dx
\]

The work terms corresponding to the section deformations \( \varepsilon(x) \) are the section forces \( s(x) \) defined according to

\[
s(x) = \int_A \begin{bmatrix} 1 & -y & z \end{bmatrix} \sigma dA = \begin{bmatrix} N(x) \\ M_y(x) \\ M_z(x) \end{bmatrix}
\]
The right-hand side of 6.38 is the standard definition for the axial force and bending moments about the \( z \)- and \( y \)-axes, respectively.

We define the section stiffness matrix \( k_s(x) \) as the partial derivative of the section forces \( s(x) \) with respect to section deformations \( \epsilon(x) \). With this definition and the rules of differentiation, the section stiffness is

\[
k_s(x) = \frac{\partial s(x)}{\partial \epsilon(x)} = \int_{A} \left[ \begin{array}{ccc} 1 & -y & z \\ -y & y^2 & -yz \\ z & -yz & z^2 \end{array} \right] dA = E \int_{A} \left[ \begin{array}{ccc} A & Q_z & Q_y \\ Q_z & I_z & I_{zy} \\ Q_y & I_{yz} & I_y \end{array} \right] dA \quad (6.39)
\]

Equation 6.39 uses 6.35 for the derivative of the axial strain with respect to \( \epsilon(x) \). The partial derivative of stress with respect to strain is the tangent modulus, \( E_t \), of the material stress–strain relationship.

For the special case of linear elastic material with modulus \( E \), 6.39 gives the standard definition for the section stiffness under axial and flexural behavior,

\[
k_s(x) = E \int_{A} \left[ \begin{array}{ccc} 1 & -y & z \\ -y & y^2 & -yz \\ z & -yz & z^2 \end{array} \right] dA = E \left[ \begin{array}{ccc} A & Q_z & Q_y \\ Q_z & I_z & I_{zy} \\ Q_y & I_{yz} & I_y \end{array} \right] \quad (6.40)
\]

in which \( Q \) denotes the first moment and \( I \) the second moment of area, respectively. For linear elastic material the section force–deformation relationship becomes

\[
s(x) = k_s(x)\epsilon(x) \quad (6.41)
\]

It is possible to select the origin at the centroidal axis of the section and the orientation of the \( y \)-\( z \) axes along the principal axes, which renders the off-diagonal terms of the section stiffness in 6.40 equal to zero. Although the centroidal axis is a useful reference point for a homogeneous section with linear elastic material, it is meaningless for the general case of a section with nonlinear materials.

In the general case of a nonlinear stress–strain relationship it is not possible to evaluate the integrals in 6.38 and 6.39 in closed form. Therefore, numerical integration is needed, which gives the value of an integral as a summation,

\[
\int_{A} g(y,z) dA \approx \sum_{i=1}^{nIP} w_i g(y_i,z_i) \quad (6.42)
\]

in which \( g(y,z) \) is the function to be integrated, \( nIP \) is the number of integration points and \( w_i \) is the weight at integration point \( i \). Different integration rules can be used in 6.42, but discontinuities associated with the stress distribution in 6.38 and the tangent modulus in 6.39, particularly under cyclic load reversals, favor low order integration schemes such as midpoint, trapezoidal or Simpson's rule. The accuracy of integration is improved with a larger number of integration points. The midpoint rule is the most common integration scheme in the application of nonlinear analysis in earthquake engineering and gives rise to the name layer model for \( y \)-integration or fiber model for \( y \)-\( z \) integration. It is worth noting that the midpoint rule exactly integrates linear polynomials. Consequently, the quadratic stiffness terms in 6.39 are not accurately represented even for the linear elastic case. For the typical number of integration points the error is, however, very small.

We conclude this section by summarizing that the cross-section response can be obtained by integration of the stress–strain response of the materials. A kinematic assumption about the strain distribution is
typically the starting point. In the general case of including all components of the strain tensor \( \varepsilon(x, y, z) \) we write

\[
\begin{align*}
\text{section kinematics or compatibility} & : \quad \varepsilon(x, y, z) = a_y(y, z) \varepsilon(x) \\
\text{section equilibrium} & : \quad s(x) = \int_A a_y^T(y, z) \sigma dA \\
\text{section stiffness} & : \quad k_s(x) = \int_A a_y^T(y, z) \frac{\partial \sigma}{\partial \varepsilon} a_y(y, z) dA
\end{align*}
\]

(6.43)

Definitions for section deformations and corresponding forces need to be generalized accordingly. \( \partial \sigma / \partial \varepsilon \) is, in general, a \( 6 \times 6 \) matrix representing the tangent material stiffness. The examples of this chapter are limited to uniaxial material response.

Before completing the discussion of force–deformation relationships, we mention that the section response can be directly defined in terms of explicit or implicit section force–deformation relations. Such an approach is based on the extension of plasticity theory to section force resultants and deformations (McGuire et al., 2000). While this approach is an excellent choice for homogeneous sections of metallic material, it is doubtful whether it constitutes a robust alternative to the integration of material response for sections composed of several materials. This is particularly the case under complex interactions of the constituent materials that may lead to softening response, as is the case for reinforced concrete sections.

### 6.6.2 Material Models

A key ingredient for establishing the section response with 6.43 is the constitutive model of the material. Under the assumptions in the preceding section, uniaxial material models suffice for many applications in earthquake engineering analysis. Thus, relatively complex uniaxial material relations can be deployed without difficulty. However, the high computational cost of integrating a complex material response, usually limits the selection to a few relatively simple hysteretic models.

For applications in performance-based earthquake engineering it is important to distinguish between path-dependent and path-independent material models. In a path-independent model the current stress is a function of current strain only. Thus, the material follows the same path whether it is loading or unloading. In static pushover analysis of structures this may be sufficient, if one can assume that limited unloading, if any, will take place. This is often the case. In cyclic static or dynamic analysis a path-dependent material model is required. In this the current stress depends on the current strain and several other variables, such as the strain history and internal variables, that describe the state of material damage. The more complex the path of loading–unloading of a material model, the higher the number of internal variables required in its description with a consequent increase in computational cost.

For the purposes of this chapter we limit ourselves to a brief description of three relatively simple material models which are used in the examples of this chapter: a bilinear elasto-plastic model with kinematic and isotropic hardening and a more involved version that includes the Bauschinger effect. Both models are suitable for describing the hysteretic behavior of steel. Finally, we briefly describe a simple hysteretic model for concrete.

Figure 6.15 shows a bilinear elasto-plastic model with kinematic and isotropic hardening. The isotropic hardening depends on the amount of plastic strain in the opposite stress direction, which is why it is only evident under compressive stress. The bilinear model is a good representation of the behavior of metals and is often used in earthquake analysis when the Bauschinger effect is not important for the simulations. It is computationally advantageous for its simplicity. When the Bauschinger effect is important, the stress–strain relation of Menegotto–Pinto in Figure 6.16 gives a very good representation of the material response (Menegotto and Pinto, 1973). It is particularly important for the computational economy of analysis of frame structures that the model expresses stress directly as function of strain. The
FIGURE 6.15  Hysteretic bilinear stress–strain relationship.

FIGURE 6.16  Hysteretic steel stress–strain relation by Menegotto-Pinto.

FIGURE 6.17  Hysteretic concrete stress–strain relation.
limitation of the model lies in its inability to reach the point of last unloading upon reloading in the same stress direction. This is evident in Figure 6.16. Details of the model implementation and parameter selection can be found elsewhere (Filippou et al., 1983).

Figure 6.17 shows a simple hysteretic model for concrete. The monotonic behavior in compression is represented by a parabolic ascending curve followed by a linear descending curve to a residual stress of between 10 and 20% of the compression strength. The slope of the descending curve can be adjusted to represent the effect of concrete confinement provided by transverse reinforcement (Scott et al., 1982). The unloading–reloading path is also linear with decreasing modulus that follows the observations of an extensive experimental study (Karsan and Jirsa, 1969). Several unloading–reloading cycles are shown in Figure 6.17. The model in Figure 6.17 does not account for the tensile strength of concrete and the effect of tension softening. On the other hand it is computationally very simple. More sophisticated models of concrete response under tensile stress are available at the expense of computational complexity (CEB, 1996). Their use is important when the precracked and preyield response of reinforced concrete structures is of particular interest.

6.7 Frame Elements

After having established equilibrium, compatibility and section force–deformation relationships, we now develop nonlinear frame elements with a range of applicability in structural analysis procedures for earthquake engineering. We discuss the advantages and limitations of each approach in the formulation of the frame elements. Emphasis is placed on a rigorous derivation of the element response and on the presentation of the element state determination process, which consists in computing the basic forces and the stiffness matrix that correspond to given element deformations.

6.7.1 Basic Relationships

The differential equations of equilibrium for a frame element in the undeformed configuration can be written, with reference to Figure 6.18, for axial and moment equilibrium, as

\[
\frac{\partial N}{\partial x} + w_x(x) = 0
\]

\[
\frac{\partial^2 M}{\partial x^2} - w_y(x) = 0
\]

in which \(w_x\) and \(w_y\) are the axial and transverse components of the distributed element load, respectively. An important characteristic of frame elements is that, under linear geometry, the differential
equations in 6.44 can be solved independently of the displacements and of the material response. In the absence of element loading the homogeneous solution of the differential equations in 6.44 gives a constant axial force and a linear bending moment distribution. We use the basic forces $q$ as boundary values of the problem to obtain the statement of equilibrium:

$$s(x) = \begin{pmatrix} N(x) \\ M(x) \end{pmatrix} = \begin{pmatrix} \frac{x}{L} - 1 + q_1 \frac{x}{L} \\ 0 \end{pmatrix} = \begin{pmatrix} 1 & 0 \\ 0 & \frac{x}{L} \end{pmatrix} \begin{pmatrix} q_1 \\ q_2 \end{pmatrix} = b(x)q \quad (6.45)$$

The matrix $b(x)$ represents the force-interpolation functions and can be regarded also as an equilibrium transformation matrix between section forces $s(x)$ and basic forces $q$. In the presence of element loads, the internal forces represent the particular solution of the differential equations in 6.44, which only need to satisfy homogeneous boundary conditions. For uniform element loads the axial force is a linear function and the bending moment is a quadratic function. Denoting the particular solution by $s_p(x)$, the equilibrium equations are

$$s(x) = b(x)q + s_p(x) \quad (6.46)$$

After setting up the equilibrium relations the geometric compatibility of the frame element can be established with the principle of virtual forces. The complementary virtual work is analogous to the virtual work principle in 6.37

$$\delta q^T \mathbf{v} = \int_0^L \delta s^T(x)e(x)dx \quad (6.47)$$

Using 6.46 for the equilibrium relation of the virtual force system, $\delta s(x) = b(x)\delta q$, and after substitution into 6.47 gives the compatibility statement as

$$\mathbf{v} = \int_0^L b^T(x)e(x)dx \quad (6.48)$$

the individual basic deformation quantities in 6.48 can be evaluated with the force-interpolation functions $b(x)$ from 6.45:

$$\mathbf{v}_1 = \int_0^L e_1(x)dx$$

$$\mathbf{v}_2 = \int_0^L \left( \frac{x}{L} - 1 \right) \kappa(x)dx$$

$$\mathbf{v}_3 = \int_0^L \frac{x}{L} \kappa(x)dx$$
It is important to emphasize that the compatibility relationships in 6.46, 6.48 and 6.49 hold true for any material response as long as the transverse displacements are sufficiently small that virtual force equilibrium can be satisfied in the undeformed configuration.

For the special case of linear elastic material response, the section relationship in 6.41 can be inverted to give the section deformations in terms of the section forces. We can generalize it by adding nonmechanical initial deformations $e_i(x)$, such as caused by temperature and shrinkage strains

$$e(x) = [k_i(x)]^{-1}s(x) + e_i(x) = f(x)s(x) + e_i(x)$$  \hspace{1cm} (6.50)

with $f_i(x)$ the section flexibility matrix. Substituting 6.50 into 6.48 and using 6.46 gives

$$v = \int b^T(x)f_i(x)b(x)d\xi = \int b^T(x)f_i(x)\{s(x) + e_i(x)\}dx = \int b^T(x)f_i(x)\{s(x) + e_i(x)\}dx = f q + v_w + v_0$$  \hspace{1cm} (6.51)

in which $f$ is the element flexibility matrix, $v_w$ are the deformations due to element loads and $v_0$ the deformations due to nonmechanical effects. In the general case of a tapered frame element with variable cross section, substitution of the force-interpolation functions from 6.45 gives

$$f = L \int_0^1 \begin{bmatrix} 1 & 0 & 0 \\ 0 & \frac{(1-\xi)(1-\xi)}{E(\xi)} & \frac{(1-\xi)^2}{E(\xi)} \\ 0 & \frac{(1-\xi)^2}{E(\xi)} & \frac{\xi^2}{E(\xi)} \end{bmatrix} \, d\xi$$  \hspace{1cm} (6.52)

with $\xi = x/L$. For general functions $EA(\xi)$ and $EI(\xi)$, 6.52 needs to be integrated numerically. Among the various schemes, Gauss or Gauss–Lobatto integration is preferred for the smallest number of function evaluations for a given accuracy level. Details of these integration schemes can be found in textbooks on finite element analysis (Bathe, 1995). In the specific case of a uniform prismatic frame element, 6.52 simplifies to

$$f = \begin{bmatrix} \frac{L}{EA} & 0 & 0 \\ 0 & \frac{L}{3EI} & -\frac{L}{6EI} \\ 0 & -\frac{L}{6EI} & \frac{L}{3EI} \end{bmatrix}$$  \hspace{1cm} (6.53)

For the case of linear elastic material, the deformation–force relation in 6.51 is inverted to give the force–deformation relation in the following form:

$$q = f^{-1}(v - v_w - v_0) = kv + q_w + q_0$$

where $k$ is the basic stiffness matrix as the inverse of the flexibility matrix, and $q_w$ and $q_0$ are the fixed-end forces under the element loads and the nonmechanical deformations, respectively. With $k = f^{-1}$ and $f$ from 6.53, the basic stiffness matrix of a prismatic frame element is
The use of moment releases at the ends of a frame element was presented in Section 6.3.3. For the case that the element has a moment release at end \( j \) and thus \( q_{j} = 0 \), 6.53 shows that \( v_{j} = -\frac{1}{2} v_{s} \), as already used in the compatibility relation of a prismatic, linear elastic frame element with a moment release in 6.20. For the frame element with a moment release at one or both ends, the compatibility relation in 6.20 and the contragradient property of force and displacement transformations give

\[
\vec{q} = a_{h}^{T} q = a_{h}^{T} (kv + q_{w} + q_{b}) = a_{h}^{T} k_{h} \vec{v} + a_{h}^{T} (q_{w} + q_{b}) = \vec{k} \vec{v} + \vec{q}_{w} + \vec{q}_{b}
\]

For the case that there is a moment release at end \( j \), with the stiffness matrix \( k \) from 6.54 this gives

\[
\begin{bmatrix}
1 & 0 & 0 \\
0 & 1 & 0 \\
0 & -\frac{1}{2} & 0 \\
\end{bmatrix}
= a_{h}^{T} k a_{h} = \begin{bmatrix}
\frac{EA}{L} & 0 & 0 \\
0 & \frac{3EI}{L} & 0 \\
0 & 0 & 0 \\
\end{bmatrix}
\]

### 6.7.2 Concentrated Plasticity Elements

#### 6.7.2.1 Truss or Brace Element

The simplest nonlinear element is the prismatic truss or brace element. In this case there is only one basic force \( q_{s} \), which is equal to the normal force in the truss element \( s_{l} = N \). The normal force is equal to the axial stress multiplied by the cross-sectional area \( A \). The axial stress is related to the axial strain by the material constitutive relation. Finally, the axial strain is related to the element deformation \( v_{s} \) by \( \varepsilon = v_{s} / L \). Thus, the element state determination is as follows: given \( v_{s} \), determine \( \varepsilon \); use the material constitutive relation to get the corresponding axial stress \( \sigma \); finally, compute the basic force with \( q_{s} = s_{l} = \sigma A \). The tangent stiffness matrix of the element can be readily obtained by the chain rule of differentiation

\[
k_{l} = \frac{\partial q_{s}}{\partial v_{s}} = A \frac{\partial \sigma}{\partial \varepsilon} \frac{\partial \varepsilon}{\partial v_{s}} = \frac{E}{L}
\]

where \( E \) is the tangent modulus of the material.

#### 6.7.2.2 Elastic Perfectly Plastic Beam

The next element of interest is a frame element with concentrated flexural hinges at the ends, where moments are assumed to be largest under the combination of gravity and lateral forces due to earthquake excitation. Although this is correct for columns, it is often an approximation for girders, where the combination of gravity and lateral forces, particularly in higher building floors, may lead to the formation of a plastic hinge away from the member ends. In such case, it is advisable to subdivide the member into two or more frame elements. The limitation that plastic hinges can only take place at specific locations
along the span is sufficiently accurate, particularly if one allows for plastic hinges to form at the outer quarter span points using three frame elements for the member.

The simplest way of accounting for the interaction between axial force and bending moment in the potential plastic hinge locations at the column ends is to use the axial forces from an elastic analysis under gravity loading to determine the plastic flexural capacity of the hinges. The girders are subjected to a small axial force so that the variation of this force during the analysis can be neglected. If significant overturning moments develop in the structure during the nonlinear pushover analysis under lateral forces, the axial force in the columns changes appreciably during the analysis and more sophisticated modeling of the axial force-bending moment interaction at the column plastic hinges is required.

The state determination of the frame element can be undertaken with an event-to-event strategy: given the element deformations \( v \), estimate the basic forces by using as tangent stiffness the elastic stiffness of the element \( k_e \), i.e., \( k_e = k_o \) and \( q = k_o v \). The end moments are compared with the corresponding plastic capacities \( q_p \). If these are not exceeded, then the basic force estimate is correct and the tangent stiffness is equal to the elastic stiffness. If the end moments exceed the corresponding plastic capacity, then the ratios \( q_1/q_{p1} \) and \( q_2/q_{p2} \) express the amount of overshoot. The first event factor \( \eta_1 \) is the inverse of the larger ratio:

\[
\eta_1 = \min \left( \frac{q_{p1}}{q_1}, \frac{q_{p2}}{q_2} \right)
\]

The initial deformations are scaled with this ratio, known as an event factor, the moment release code is set to unity for the end with the event factor, and a new tangent stiffness matrix \( k_{el} \) is formed. The new estimate of the basic forces is

\[
q = k_e \eta_1 v + k_o (1 - \eta_1) v
\]

If another hinge has not formed, then the basic force estimate is correct and the tangent stiffness of the element is \( k_{el} \). If a second hinge forms, then the last deformation increment is scaled by the new event factor \( \eta_1 \), the tangent stiffness is updated (which turns out to be zero in the presence of two hinges) and the end forces become

\[
q = k_e \eta_1 v + k_o (1 - \eta_1) \eta_2 v
\]

With this procedure a maximum of two iterations is required for convergence under monotonic loading. Even under cyclic loading the number of iterations does not exceed two, because the plastic hinges at the ends can be either open or closed. It is important to note, however, that cyclic loading requires that the process be conducted with deformation increments instead of total deformations and that the state of the hinges, the basic forces and the tangent stiffness matrix be saved from one iteration of the global equilibrium equations to the next.

### 6.7.2.3 Two-Component Parallel Model

The frame element with concentrated plastic hinges at the ends is straightforward in its implementation, but is also limited to elastic-perfectly plastic behavior. Thus, it can be a useful addition to the plastic analysis of Section 6.2.3 by providing the complete force–displacement relation of the structural model up to incipient collapse. For a more realistic representation of material behavior the inclusion of strain hardening is important. In such case the simple elasto-plastic frame element needs to be combined in parallel with a linear elastic frame element to form the two-component model (Clough et al., 1965).

The two-component model consists of an elasto-plastic element in parallel with a linear elastic element. The latter represents the strain hardening response of the frame member. The fact that the elements are in parallel implies that

\[
v = v_e = v_p \quad q = q_e + q_p \quad k = k_e + k_p
\]
where subscripts $e$ and $p$ denote the elastic component and the elasto-plastic component, respectively. The state determination process of the element is straightforward, because of the first relation in 6.55: given $\mathbf{v}$ the deformations of each component are known. For the elastic component the basic forces are $q_e = k_e \mathbf{v}_e$, and for the elasto-plastic component the state determination process of the preceding section determines the end forces. The same is true for the stiffness. Once the end forces and tangent stiffness of the elasto-plastic element are determined, then the last two equations in 6.55 yield the resisting forces and the stiffness of the entire element. This simplicity of the state determination process is characteristic of elements with an assumption about the deformation distribution.

The main difficulty with the two-component model is the calibration of the element parameters. Under uniform curvature the section stiffness of the linear elastic component can be set equal to the strain hardening section stiffness of the frame member. The initial stiffness of the elasto-plastic component can then be determined to make up the difference between the initial stiffness of the frame member and the strain hardening stiffness. Unfortunately, the case of uniform curvature is of little use in earthquake engineering analysis. Rather the calibration of the model parameters takes place under antisymmetric curvature with the point of inflection at member midspan. Another limitation of the two-component model is its inability to handle different plastic moment capacities under positive and negative curvature. For these reasons the two-component model has been superceded in earthquake engineering analysis by the one-component model (Giberson, 1967). It is worth discussing in some detail the formulation of this element, because it is characteristic of a class of elements that are based on an assumption about the internal force distribution. These elements play an important role in modern earthquake engineering analysis, because they represent exactly the force distribution in the member and result in a robust numerical implementation.

### 6.7.2.4 One-Component Series Model

The one-component model consists of a linear elastic element connected in series with a rigid-plastic linear hardening spring at each end. The conditions governing the response of the element are

$$ q = q_e = q_p \quad \mathbf{v} = \mathbf{v}_e + \mathbf{v}_p \quad f = f_e + f_p $$

It is convenient to establish these relations with a substructure approach to the statically determinate structure. Limiting attention to the flexural contribution, the force-interpolation functions for the basic forces $q_i$ and $q_j$ are

$$ \mathbf{b} = \begin{bmatrix} 1 & 1 & 0 & 0 \\ 0 & 0 & 1 & 1 \end{bmatrix} $$

and the composite flexibility matrix of the subelements is

$$ \mathbf{f} = \begin{bmatrix} f_i & 0 & 0 & 0 \\ 0 & \frac{L}{3EI} & \frac{L}{6EI} & 0 \\ 0 & -\frac{L}{6EI} & \frac{L}{3EI} & 0 \\ 0 & 0 & 0 & f_j \end{bmatrix} $$

where $f_i$ and $f_j$ are the spring flexibilities at ends $i$ and $j$, respectively. From the product $\mathbf{b}^T \mathbf{f} \mathbf{b}$ we obtain the last relation in 6.56 with $f_e$ given by the flexural terms in 6.53 and $f_p$ by the following expression

$$ f_p = \begin{bmatrix} f_i \\ 0 \\ 0 \\ f_j \end{bmatrix} $$
The flexibility coefficients $f_i$ and $f_p$ are zero for a moment less than the flexural plastic capacity and then assume a value equal to the inverse of the strain hardening stiffness. Because the two end springs are independent, their properties can also be specified independently. Moreover, it is possible to specify a different plastic capacity and a different strain hardening stiffness under a positive than under a negative curvature. This makes the one-component model more versatile than the two-component model. Another advantage of this model is its ability to account for the effect of element loads by the inclusion of $v_w$ from 6.51 so that $v = f q + v_w$ with $f$ from 6.56. The calibration of the model parameters takes place under antisymmetric curvature with the point of inflection at member midspan. This may not be a reasonable approximation for the case of a different plastic capacity under a positive than under a negative curvature.

It is important to discuss the process of element state determination for the one-component model, because it is characteristic of the class of elements that are based on an assumption about the internal force distribution. Because the element is implemented in a standard computer program that is based on the direct stiffness method of analysis, it is expected to return the resisting forces and current stiffness matrix for given element deformations $v$. In order to highlight the fact that these deformations are given we denote them with the symbol $\tilde{v}$. From the middle equation in 6.56 we have

$$\tilde{v} - v = 0 \quad \text{where} \quad v = v_e + v_p$$

Because the deformations of the elastic and plastic components of the one-component model are functions of the basic forces $q$ we formally write that

$$\tilde{v} - v(q) = 0$$

and note that we are dealing with a nonlinear system of equations, because of the rigid-plastic linear hardening component. The solution of the nonlinear system can be obtained with the Newton–Raphson algorithm, as will be discussed in more detail in the following section. We defer, therefore, the discussion of the state determination process of this class of elements until then. We note at this stage, however, that 6.57 implies an iterative process of state determination at the element level. An alternative approach that bypasses the element iteration is also possible.

### 6.7.3 Distributed Inelasticity Elements

The limitation of concentrated plasticity elements is that inelastic deformations take place at predetermined locations at the ends of the element. While this may be a reasonable assumption in lower floors of moment-resisting frames, it does not account for the possibility of inelastic deformations taking place within the element in the upper floors of the building model. Another, in many respects more serious limitation, is the fact that concentrated plasticity elements require calibration of their parameters against the response of an actual or ideal frame element under idealized loading conditions. This is necessary, because the response of concentrated plasticity elements derives from the moment–rotation relation of their components. In an actual frame element the end moment–rotation relation results from the integration of the section response. This can be achieved directly with elements of distributed inelasticity. In this case there are two approaches: the force formulation or the displacement formulation.

In the force formulation we make use of the fact that the internal forces $s(x)$ at a distance $x$ from end $i$ of a two-node frame element are given as the product of the force-interpolation functions $b(x)$ and the basic forces $q$ according to 6.45. In the presence of element loads we need to modify this relation according to 6.46. It is important to note that these relations hold for any material response, as long as the equilibrium can be satisfied in the undeformed configuration. The element deformations can then be established by the principle of virtual forces from 6.48. This implies that the section deformations $e(x)$ can be obtained from the section forces $s(x)$. In reality, this relation is not available, but its inverse is. Thus, $e(x)$ needs to be established from the solution of the nonlinear system of equations.
Methods of Analysis for Earthquake-Resistant Structures

\[ b(x)q + s(x) - s(e(x)) = 0 \]

For the solution of the nonlinear system of equations in 6.57 we also need to establish the change of the element deformations with \( q \). This change is reflected by the derivative of the expression in 6.48 with respect to \( q \). We obtain

\[
\frac{\partial v}{\partial q} = \frac{\partial}{\partial q} \int_\ell b^T(x)e(x)dx = \int_\ell b^T(x)\frac{\partial e(x)}{\partial s(x)} \frac{\partial s(x)}{\partial q} dx = \int_\ell b^T(x)f_s(x)b(x)dx
\]

where \( f_s(x) \) is the section flexibility, which can be obtained as the inverse of the section stiffness matrix in 6.39. The derivative of the section forces with respect to \( q \) is obtained from 6.46. We call the expression in 6.58 the tangent flexibility matrix \( b(x) \) of the element.

In the displacement formulation we assume that the axial and transverse displacements at distance \( x \) from the end node \( i \) are supplied by the product of suitable displacement interpolation functions with the element deformations \( v \). For a two-dimensional element we write

\[
u(x) = a(x)v = \begin{bmatrix} a_1(x) & 0 & 0 & v_1 \\ 0 & a_2(x) & a_3(x) & v_2 \\ & & & v_3 \end{bmatrix}
\]

The displacement interpolation functions correspond to the solution of the differential equation for a linear elastic, prismatic frame element. It is important to note that these functions thus only approximate the response of a frame element with distributed inelasticity. This has important ramifications under large inelastic deformations, as a later example will demonstrate. The section deformations at \( x \) can be obtained by application of the small deformation theory of beam kinematics: the axial strain \( \varepsilon_a \) at the reference axis is the first derivative of the axial displacement, and the curvature is the second derivative of the transverse displacement. We write formally

\[
e(x) = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & \frac{\partial}{\partial x} \end{bmatrix} u(x) = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} a_1(x) & 0 & 0 & v_1 \\ 0 & a_2(x) & a_3(x) & v_2 \\ & & & v_3 \end{bmatrix} = a^*(x)v
\]

The element forces \( q \) are established from the principle of virtual displacements in 6.37 using the variation of 6.59 for the virtual displacement field. We obtain

\[
q = \int_\ell a^*_s(x)s(x)dx
\]

The section forces \( s(x) \) in 6.60 are directly obtained from the section constitutive relation for given section deformations \( e(x) \). The latter, in turn, are directly obtained from the given element deformations \( v \) by 6.59. Thus, the path from the given element deformations \( v \) to the corresponding forces \( q \) is straightforward in the displacement formulation, which explains its appeal. This should not distract from the serious drawback of the method, which lies in the approximate nature of the displacement interpolation functions.

The change of element forces \( q \) with deformation is also required. This is obtained from the derivative of 6.60 with respect to \( v \), which yields
where $k_s(x)$ is the section stiffness matrix from 6.39, and the derivative of the section deformations with respect to $v$ is obtained from 6.59. The expression in 6.61 is the tangent stiffness matrix $k_t$ of the element.

The expressions in 6.48, 6.58, 6.60 and 6.61 for the state determination of the distributed inelasticity elements involve integrals over the element length. The evaluation of these integrals is accomplished numerically. In analogy with 6.42 the integrals are evaluated as

$$
\int g(x)dx = \sum_{i=1}^{n} w_i g(x_i)
$$

The most suitable numerical integration methods use the Gauss or the Gauss–Lobatto rule which optimize accuracy of smooth integrands for a given number of integration points (Bathe, 1995). The Gauss–Lobatto rule is particularly suitable when it is important to include the ends of the element in the evaluation. This is indeed the case in earthquake engineering applications, where the largest inelastic deformations quite often take place at the element ends. Four integration points suffice for the integrals in 6.48, 6.58, 6.60 and 6.61 as long as we are not interested in the effect of the midspan section. In the latter case, which is important in girders under significant gravity loads, five integration points are recommended. However, the inclusion of the effect of gravity loads can only be accommodated with the force formulation. By contrast, in the displacement formulation, the section forces in 6.60 are derived from the section deformations and do not include the effect of loads on the elements, such as due to gravity. The latter are included as external virtual work contribution by modification of 6.60 according to

$$
q + \int a^T(x)w(x)dx = \int a^T(x)s(x)dx
$$

We conclude from 6.62 that the initial or fixed-end forces of the displacement formulation are not different from those of a linear elastic, prismatic frame element. Moreover, in this case the element loads only affect the element forces and do not affect the internal force distribution. This is another serious limitation of the displacement formulation.

### 6.8 Solution of Equilibrium Equations

The equilibrium equations in 6.1 constitute the starting point for linear or nonlinear structural analysis methods. We focus our attention on the free DOFs of the model and write for the applied and resisting forces the system of equations:

$$
P_f - P_r = 0
$$

The subscript $f$ is dropped for brevity of notation, and the reactions at the restrained DOFs can be evaluated after the equations for the free DOFs are solved. In the general case the resisting forces are implicit functions of the displacements $U$ at the DOFs of the structural model and 6.63 becomes a set of nonlinear equations in the unknown displacements $U$

$$
P - P_r(U) = 0
$$
where we assume that the applied forces $P$ do not depend on the displacements $U$. Equation 6.64 applies to static analysis. For dynamic analysis it will be generalized with the inclusion of inertia effects in a later section.

### 6.8.1 Newton–Raphson Iteration

To develop a solution algorithm for the nonlinear system of equations in 6.64, the resisting forces are expanded in a Taylor series about an initial displacement vector $U_0$:

$$P_i(U) = P_i(U_0) + \frac{\partial P_i}{\partial U} \bigg|_{U_0} (U - U_0) + \frac{1}{2} \frac{\partial^2 P_i}{\partial U^2} \bigg|_{U_0} (U - U_0)^2 + ...$$  \hspace{1cm} (6.65)

Truncating the Taylor series after the linear term and substituting the expression of the resisting forces from 6.65 in 6.64 we obtain a linear system of equations for the unknown displacements $U$. Denoting the displacement increment by $\Delta U = U - U_0$, the linearized equilibrium relationship from 6.65 is

$$P - P_i(U_0) = \frac{\partial P}{\partial U} \bigg|_{U_0} \Delta U = K_{ii} \Delta U$$  \hspace{1cm} (6.67)

Equation 6.67 includes the tangent stiffness matrix, $K_{ii}$, for the free DOFs of the structural model as the derivative of the global resisting force vector with respect to the displacements. This derivative is evaluated at $U_0$ in 6.67. The term $(i, m)$ of the stiffness matrix represents the partial derivative of the resisting force at DOF $i$ with respect to the displacement at DOF $m$. The tangent stiffness matrix is obtained by direct assembly of the tangent stiffness matrices of the elements in the structural model after the latter have been transformed to the global coordinate system, as will be discussed later in this section.

The solution of 6.67 yields the displacement increment $\Delta U$. An improved estimate of the solution to the system of equilibrium equations in 6.64 can be obtained with $U_i = U_0 + \Delta U$. The repetition of this process will converge to the solution of the nonlinear set of equilibrium equations in 6.64 under certain conditions. This iterative procedure is known as Newton–Raphson algorithm. An important characteristic of the Newton-Raphson iteration is that the process converges with a quadratic rate to the solution. This can be observed by a slight modification of 6.65:

$$P_i(U) = P_i(U_0) + \frac{\partial P_i}{\partial U} \bigg|_{U_0} (U - U_0) + \frac{1}{2} \frac{\partial^2 P_i}{\partial U^2} \bigg|_{U_0} (U - U_0)^2$$  

$$P_i(U) = P_i(U_0) + \frac{\partial P_i}{\partial U} \bigg|_{U_0} (U_0 - U_0) + \frac{\partial P_i}{\partial U} \bigg|_{U_0} (U - U_0) + \frac{1}{2} \frac{\partial^2 P_i}{\partial U^2} \bigg|_{U_0} (U - U_0)^2$$  

$$0 = \frac{\partial P_i}{\partial U} \bigg|_{U_0} (U - U_0) + \frac{1}{2} \frac{\partial^2 P_i}{\partial U^2} \bigg|_{U_0} (U - U_0)^2$$

where the last equation results from the use of 6.67 for the first two terms of the second equation. By taking absolute values of the last expression we arrive at the desired result $|U - U| = c |U - U|$, where $c$ is a constant. This means that the error between the solution and iterate is less than the square of the error for the previous iterate. This characteristic is used in simulation studies to ascertain that the stiffness matrix is indeed tangent to the structural response, because the latter fact can have important ramifications for the convergence of the Newton–Raphson algorithm. The constant $c$ depends on the second derivative of the resisting force, or the change in the tangent stiffness. Large changes in stiffness result
in large constants and slower convergence. As will be covered in the following, the convergence can be improved by a load incrementation strategy.

The Newton–Raphson algorithm proceeds in the following steps:

1. At the start of iteration \( j \), compute the resisting forces \( P_{u-1} = P_f(U_{j-1}) \) and the tangent stiffness \( K_{u-1} \) for the displacements at the end of the previous iteration \( U_{j-1} \).
2. Solve the linearized system of equations \( P_r = P - P_{u-1} = K_{u-1} \Delta U_{j-1} \) for \( \Delta U_{j-1} \).
3. Update the displacement vector \( U_j = U_{j-1} + \Delta U_{j-1} \), advance the iteration index and return to step 1 repeating steps 1 to 3 until convergence. The convergence criterion will be discussed later in this section.

The convergence of the algorithm depends on the initial displacement to start the iteration, the characteristics of the tangent stiffness matrix \( K_t \), and the convergence criterion. It is important that the initial guess be close to the actual solution.

### 6.8.2 Load Incrementation

To improve the convergence of the Newton–Raphson algorithm for nonlinear structural analysis, it is necessary to incorporate a procedure for incrementing the load to limit the changes in the structural state for each load increment. Instead of applying the load in one step, the solution is divided into several steps, and it proceeds with load factor increments whose magnitude can be adjusted depending on the state of the structure. Within each load step the Newton–Raphson iteration process can be used to solve the equilibrium equations.

We assume that the applied loads are grouped in load patterns with independent histories. The simplest case is a single applied force pattern \( P_{ref} \), and this case has some important applications in the nonlinear static analysis of structures under equivalent lateral earthquake loads. The notation for the load incrementation procedure uses a superscript in parentheses to denote the load step number with index \( k \) while a subscript denotes the iteration number in each load step with index \( j \). The applied force vector at load step \( k \) can, therefore, be written as

\[
P^{(k)} = \lambda^{(k)} P_{ref}
\]

where the load factor \( \lambda^{(k)} \) results from a series of increments,

\[
\lambda^{(k)} = \lambda^{(k-1)} + \Delta \lambda^{(k)}
\]

so that the applied force vector at load step \( k \) can also be written as

\[
P^{(k)} = P^{(k-1)} + \Delta \lambda^{(k)} P_{ref}
\]

The equilibrium equations at load step \( k \) become

\[
P^{(k)} - P_r^{(k)} = P^{(k-1)} + \Delta \lambda^{(k)} P_{ref} - P_r^{(k)}
\]

and the starting displacement vector for satisfying the above equilibrium equations is \( U^{(k)} = U^{(k-1)} \), that is the state at the end of the previous load step. If the load factor increment is held constant during the equilibrium iterations, then the iteration process consists of the three steps presented earlier with the unbalanced force given by \( P_r^{(k)} = P^{(k-1)} + \Delta \lambda^{(k)} P_{ref} - P_r^{(k-1)} \). Only the resisting force vector is updated during the equilibrium iterations.

### 6.8.3 Load Factor Control during Incrementation

The load factor increment can be changed in the Newton–Raphson algorithm so that large increments are applied when the structure stiffness does not change much, and the increments are reduced when
the stiffness changes. An excellent parameter for this purpose is the current stiffness parameter (Bergan, 1978). This parameter is the scalar product of the reference force vector and the corresponding displacements caused by the forces. Denoting the displacements under the reference force vector with \( U \), the stiffness parameter is defined as

\[
S_p^{(0)} = P_{\text{ref}}^T U = P_{\text{ref}}^T K_{\text{ref}}^{-1} P_{\text{ref}}
\]

The initial value of the parameter \( S_p^{(0)} \) is computed with the initial tangent stiffness matrix. The following expression gives the change in the load factor for load step \( k \) (Bergan, 1978),

\[
\Delta \lambda^{(k)} = \Delta \lambda^{(0)} \begin{bmatrix} S_p^{(0)} \\ S_p^{(k-1)} \end{bmatrix} \begin{bmatrix} \gamma \\ 1 \end{bmatrix}
\]  

(6.68)

where \( S_p^{(k-1)} \) is the stiffness parameter at the end of the previous load step \( k-1 \), \( \Delta \lambda^{(0)} \) is the load factor increment for the first load step with \( k=1 \), commonly selected to be about 30% of the collapse load factor, and \( \gamma \) is a constant between 0.5 and 1.2, with 1.0 being a commonly used value. Figure 6.19 shows the load–displacement response of a structural model with an exponent value of \( \gamma = 1 \). The model is a truss structure with nonlinear material response. It is clear from the figure that with load factor adjustment during incrementation it is possible to approach the maximum strength of the model even though the tangent stiffness becomes nearly singular.

6.8.4 Load Factor Control during Iteration

Load incrementation allows the Newton-Raphson algorithm to approach the maximum strength, but it cannot compute the postpeak response. For tracing the postpeak response, the incrementation procedure needs to be further modified so as to control the load during the equilibrium iterations. In this case the applied force vector is updated during the equilibrium iterations of a load step. With a subscript denoting the iteration number, the unbalanced force is

\[
P_{\text{adj}}^{(k)} = P_{\text{ref}}^{(k)} - P_{\text{ref}}^{(k-1)} = P_{\text{adj}}^{(k)} + \Delta \lambda_{\text{adj}}^{(k)} P_{\text{adj}} - P_{\text{adj}}^{(k-1)}
\]  

(6.69)
During the first iteration, \( j = 1 \), the first term on the right-hand side of 6.69 is equal to the force vector at the end of the previous load step, \( P_{n-1} = P^{k-1} \). Since all superscripts in 6.69 refer to load step \( k \), these are not included in the following equations for brevity of notation. Using the Newton-Raphson algorithm, the following system of linear equations of equilibrium is solved at each iteration:

\[
P_{n}^{j} = K_{ij}^{j} \Delta U_{j}^{j}
\]

(6.70)

Substituting 6.69 into 6.70 gives

\[
P_{j-1} + \Delta \lambda_{j} P_{ref} - P_{n-1} = K_{ij}^{j} \Delta U_{j}^{j} = K_{ij}^{j} \left( \Delta U_{n-1} + \Delta \lambda_{j} U_{n-1} \right)
\]

(6.71)

so that

\[
P_{j-1} - P_{n-1} = K_{ij}^{j} \Delta U_{n-1}^{j}
\]

\[
P_{ref} = K_{ij}^{j} U_{n-1}^{j}
\]

With the decomposition of displacement increments into two terms on the right-hand side of 6.71, a separate condition can be introduced to determine the load factor increment during iteration \( j \). Several schemes have been proposed for the purpose and there is extensive literature on the subject (Clarke and Hancock, 1990, Crisfield, 1991). The schemes that prove particularly useful for the nonlinear static (pushover) analysis of structures with equivalent lateral earthquake loads involve load control during iterations under constant displacement at a single selected degree of freedom (sometimes referred to as the control node). In this case, we impose a condition on degree of freedom \( n \), such that

\[
\Delta U_{n}^{j} = \Delta \lambda_{j} U_{n-1}^{j} + \Delta U_{n-1}^{j} = 0
\]

to determine the load factor increment \( \Delta \lambda_{j} \). An alternative is to use the condition of constant external work to determine the load factor, which leads to the condition that

\[
\Delta W_{j}^{y} = \Delta U_{n}^{jT} \Delta \lambda_{j} P_{ref} = 0
\]

After substituting \( \Delta U_{j}^{y} \) from 6.71, the load factor is given by

\[
\Delta \lambda_{j} = \frac{\Delta U_{n}^{jT} P_{ref}}{U_{n-1}^{jT} P_{ref}}
\]

With load control during equilibrium iterations the steps of the Newton–Raphson algorithm for a single load step are:

1. Start with the structure state determination for the displacements at the end of the previous iteration \( j - 1 \) in load step \( k \) and determine the tangent stiffness matrix \( K_{ij}^{j} \) and resisting force vector \( P_{n-1}^{j} \). The applied force vector \( P_{n-1}^{j} \) is also known since \( P_{n-1}^{j} = \lambda_{j-1} P_{ref} \).
2. Compute the displacements \( U_{n-1}^{j} \) under the reference force vector and the residual displacement increments \( \Delta U_{n-1}^{j} \) with the equations following 6.71. Since this step involves the solution of a system of equations under different force vectors, the tangent stiffness need only be factored once followed by separate back substitutions for each system of equations.
3. Determine the load factor increment \( \Delta \lambda_{j} \) and the resulting displacement increments \( \Delta U_{j}^{j} \):

\[
\Delta U_{j}^{j} = \Delta \lambda_{j} U_{n-1}^{j} + \Delta U_{n-1}^{j}
\]

4. Update the displacements and the load factor:

\[
U_{j} = U_{j-1} + \Delta U_{j}
\]

\[
\lambda_{j} = \lambda_{j-1} + \Delta \lambda_{j}
\]
Update the iteration index and repeat steps 1 through 4 until convergence. Convergence is measured by the ratio of the relative work increment \( \Delta W_j \) in iteration \( j \) to the initial work increment at iteration \( j = 1 \) where

\[
\Delta W_j = \Delta U_j^T \left( P_{j-1} + \Delta \lambda_j P_{\text{int}} - P_{j-1} \right)
\]

Figure 6.20 shows the load–displacement response of the same truss structure as in Figure 6.19, for the case that the truss elements have a softening modulus equal to 10% of the initial modulus. The figure shows how the load control algorithm permits the tracing of the postpeak response.

### 6.8.5 Structure State Determination

A key step in the Newton–Raphson algorithm is the structure state determination, which involves the determination of the resisting force vector and structure stiffness matrix for given structural displacements and their increments. The tangent stiffness matrix of the structure is obtained by the partial derivative of the resisting forces with respect to the displacements at the global DOFs. Applying the chain rule of differentiation to 6.1 gives

\[
K_i = \frac{\partial P}{\partial U} = \frac{\partial}{\partial U} \sum_{el} P_{\text{el}}^{(d)} = A_{\text{el}}^{(d)} \frac{\partial P^{(d)}}{\partial u^{(d)}} \frac{\partial u^{(d)}}{\partial U}
\]

(6.72)

The assembly operation on the right-hand side involves row indexing and summation of the element contributions. Each row of the element resisting forces will produce as many columns as the number of element DOFs in vector \( u \) in the operation \( A_{\text{el}}^{(d)} \frac{\partial P^{(d)}}{\partial u^{(d)}} \). With 6.12 these columns will be postmultiplied by a matrix with a single nonzero term of unity in each row. This unit value is located at the column that corresponds to the global DOF number to which the element DOF in the corresponding row maps. Postmultiplication by this matrix amounts to mapping the column numbers of \( \frac{\partial P^{(d)}}{\partial u^{(d)}} \) to the column numbers corresponding to the global DOF numbers for the DOFs of this element. From 6.72 we conclude
that the global tangent stiffness matrix $K_i$ of the structure can be obtained by direct assembly (row indexing and summation) of the element stiffness coefficients in the global coordinate system, as long as the columns of the element stiffness matrix are mapped to the global DOF numbers corresponding to the element DOFs of the particular element. This is written in compact form as

$$K_i = \sum a^{(e)} k_i$$  \hspace{1cm} (6.73)

The assembly of the resisting force vector $P_i$ follows 6.1.

The partial derivative of the element forces with respect to the element displacements in the global reference system represents the element tangent stiffness matrix in the global coordinate system. The tangent stiffness matrix of the element can be obtained from 6.5 by the chain rule of differentiation,

$$k_i^{(d)} = \frac{\partial P}{\partial u} = \frac{\partial}{\partial u}(a^{(e)}q) = a^T \frac{\partial q}{\partial v} \frac{\partial v}{\partial u} = a^T k_s a_s$$  \hspace{1cm} (6.74)

where $k_i$ is the tangent stiffness of the basic element. Linear geometry is assumed in 6.74 so that the transformation matrix $a_s$ does not depend on the displacements $u$, and 6.18 can be used for the derivative of the element deformations with respect to $u$.

### 6.8.6 State Determination of Elements with Force Formulation

The Newton–Raphson algorithm can be used for the solution of the system of nonlinear equations in 6.57. We present it here because the state determination of this class of elements is not well known in the literature. For iteration $j$

$$\ddot{v} - v_{j-1} = \frac{\partial v}{\partial q_{j-1}} \Delta q_j = 0$$

Noting that the element stiffness is the inverse of the flexibility matrix, we can obtain the increment of the element forces $\Delta q_j$ according to

$$\Delta q_j = k_{j-1} \left( \ddot{v} - v_{j-1} \right)$$

and update the element forces to $q_j = q_{j-1} + \Delta q_j$. We solve for the section deformations and increments at the integration points from

$$b(x)q_j + s_{w_i}(x) - \left( s_{j-1}(x) + \frac{\partial q}{\partial e_{j-1}} \Delta e_{j-1}(x) \right) = 0$$

noting that the derivative of the section forces with respect to section deformations is the last section stiffness matrix $k_{j-1}(x)$. After determining $\Delta e_{j-1}(x)$, we update the section deformations to $e_{j}(x) = e_{j-1}(x) + \Delta e_{j-1}(x)$. Finally, we determine the current element deformations from the principle of virtual forces

$$v_j = \int b(x)e_j(x)dx$$

and return to the beginning of the algorithm with a new deformation residual $\ddot{v} - v_j$. 
The above process implies an iterative element state determination for each iteration of the solution of the global equilibrium equations (Ciampi and Carlesimo, 1986, Taucer et al., 1991). An alternative scheme has also been used with success whereby the element state determination consists of a single iteration that works in tandem with the global equilibrium equations (Neuenhofer and Filippou, 1997). In either case by setting \( j = 1 \) in the element state determination algorithm we conclude that we need to store the element forces \( q_{j=1} \) and the section deformations \( e_{j=1}(x) \) from a global iteration to the next.

### 6.8.7 Nonlinear Solution of Section State Determination

The section state determination forms part of the algorithm of the state determination of elements with distributed inelasticity, but is also important in its own right in the determination of moment–curvature and interaction diagrams of sections. Therefore, we briefly describe the process of section state determination for a couple of cases. In the simplest case the section deformations \( e \) are given. Using the assumption of section kinematics we determine the strain at the integration points of the cross section \( \varepsilon_i \) according to the first equation in 6.43 and use the material constitutive relation to obtain the corresponding stress and material stiffness. We then determine the section forces \( s \) and the section stiffness \( k \) by numerical evaluation of the integrals in 6.43. This is the process used at every integration point of the distributed inelasticity elements during state determination. The same process can be used to obtain the interaction diagram of a cross section, in which case the section deformations \( e \) are set so as to describe the limit state of the section.

In the determination of the moment–curvature diagram the problem is slightly different. In this case the curvatures are specified along with the axial force acting on the cross section. The corresponding axial strain is determined from the single available equilibrium equation, the axial force equilibrium. We write symbolically

\[
\tilde{N} - N(e, \kappa) = 0
\]

where the symbol over the axial force \( N \) denotes the given value. Because this is a scalar equation, different solution methods can be used. We show here the application of the Newton–Raphson solution. For this we write for the step of load incrementation

\[
\tilde{N} - \left( N_0 + \frac{\partial N}{\partial e} \Delta e + \frac{\partial N}{\partial \kappa} \Delta \kappa \right) = 0
\]

where \( N_0 \) denotes the axial force value of the last load step. We note also that the derivatives of the axial force with respect to the section deformations correspond to terms (1,1) and (1,2) of the section stiffness matrix \( k \). Because the curvature increment is specified, we solve the above equation for the initial increment of the axial strain \( \Delta e \) according to

\[
\frac{\partial N}{\partial e} \Delta e = \tilde{N} - \left( N_0 + \frac{\partial N}{\partial \kappa} \Delta \kappa \right)
\]

During subsequent iteration corrections the curvature increment is set equal to zero.

### 6.9 Nonlinear Geometry and P-Δ Geometric Stiffness

The element tangent stiffness matrix in Section 6.8.5 was derived on the assumption of linear geometry, in which case the element equilibrium equations are satisfied in the undeformed configuration and the
compatibility relation between element deformations and end displacements in the global reference system does not depend on the displacements. In the general case of nonlinear geometry, the element equilibrium equations need to be satisfied in the deformed configuration according to Section 6.4, and the compatibility relation between element deformations and end displacements in the global reference system becomes nonlinear on account of large displacements. For applications in earthquake engineering, approximations of the nonlinear equilibrium and compatibility relations are possible, as will be discussed in the following.

6.9.1 Resisting Forces and Element Tangent Stiffness Matrix

As discussed in Section 6.4, the equilibrium in the deformed configuration is expressed in the local coordinate system. Thus, the resisting forces $\mathbf{p}$ for given end displacements $\mathbf{u}$ in the local coordinate system are given by either 6.21, 6.22 or 6.23 depending on the desired accuracy in the geometrically nonlinear analysis. The compatibility transformation $\mathbf{u} = a^T \mathbf{u}$ is used to transform the end displacements from the global to the local coordinate system. After obtaining the element end forces $\mathbf{p} = a^T \mathbf{p}$ in the local coordinate system from either 6.21, 6.22 or 6.23, the rotation transformation $\mathbf{p} = a^T \mathbf{p}$ transforms the element resisting forces to the global coordinate system.

For the element stiffness matrix, we proceed in an analogous manner:

$$\frac{\partial \mathbf{p}}{\partial \mathbf{u}} = \frac{\partial}{\partial \mathbf{u}} (a^T \mathbf{p}) = a^T \frac{\partial \mathbf{p}}{\partial \mathbf{u}} \frac{\partial \mathbf{u}}{\partial \mathbf{u}} = a^T \mathbf{k} a,$$

(6.75)

There is nothing special in this transformation process relative to linear geometry. However, the tangent element stiffness matrix $\mathbf{k}$ in the local reference system requires additional consideration for geometrically nonlinear analysis. Using the resisting forces in 6.21, the chain rule of differentiation for the element stiffness matrix in the local reference system gives

$$\mathbf{k} = \frac{\partial \mathbf{p}}{\partial \mathbf{u}} = \frac{\partial}{\partial \mathbf{u}} (b^T q) = \frac{\partial b}{\partial \mathbf{u}} q + b a \frac{\partial q}{\partial \mathbf{u}} \frac{\partial q}{\partial \mathbf{u}}$$

(6.76)

The element stiffness matrix in the local coordinate system in 6.76 is composed of two contributions: the geometric stiffness $\mathbf{k}_g$, arising from the change of the equilibrium matrix with end displacements $\mathbf{u}$, and the material stiffness $\mathbf{k}_m$, which represents the transformation of the tangent basic stiffness $k_i, = \frac{\partial \mathbf{q}}{\partial \mathbf{v}}$ to the local coordinate system.

Using the deformation-large displacement relations in 6.13 and 6.14, the derivative of $\mathbf{v}$ relative to $\mathbf{u}$ is as follows:

$$\frac{\partial \mathbf{v}_1}{\partial \mathbf{u}} = \frac{L_n}{L} \frac{\partial L_n}{\partial \mathbf{u}} = \frac{L_n}{L} \frac{1}{L_n} \begin{pmatrix} - (L + \Delta \mathbf{u}) & - \Delta \mathbf{u} & 0 & L + \Delta \mathbf{u} & \Delta \mathbf{u} \\ \end{pmatrix}$$

$$\frac{\partial \mathbf{v}_2}{\partial \mathbf{u}} = (0 \ 0 \ 1 \ 0 \ 0) - \frac{1}{L_n} \begin{pmatrix} \Delta \mathbf{u} & - (L + \Delta \mathbf{u}) & 0 \ L + \Delta \mathbf{u} & \Delta \mathbf{u} \\ \end{pmatrix}$$

$$\frac{\partial \mathbf{v}_3}{\partial \mathbf{u}} = (0 \ 0 \ 0 \ 0 \ 1) - \frac{1}{L_n} \begin{pmatrix} \Delta \mathbf{u} & - (L + \Delta \mathbf{u}) & 0 \ L + \Delta \mathbf{u} & \Delta \mathbf{u} \\ \end{pmatrix}$$

By comparison of the above expressions with 6.21, it is clear that contragradience holds, so that

$$\frac{\partial \mathbf{v}}{\partial \mathbf{u}} = a \mathbf{v} = b^T \mathbf{v}$$
when the difference between $L_n$ and $L$ in $\frac{\partial v_i}{\partial \mathbf{u}}$ is neglected. Thus, 6.76 becomes

$$k = k_e + k_m = \frac{\partial b}{\partial \mathbf{u}} q + b_i k, b_i = \frac{\partial a_i^T}{\partial \mathbf{u}} q + a_i^T k, k, a_u \quad \text{with} \quad k_i = \frac{\partial q}{\partial v_i} \quad (6.77)$$

The form of 6.77 reinforces the earlier statement that the material stiffness is equal to the tangent basic stiffness transformed to the local coordinate system. Combining 6.77 with 6.75 gives the element stiffness matrix in the global coordinate system including the geometric and material stiffness contributions:

$$k^{el} = a^T q + a^T k, a_u = a^T \bar{k}, a, a^T q + a^T k, a_u$$

(6.78)

It is important to emphasize that 6.78 holds for all element types, as long as the element force-deformation relation is defined in the basic system. Of equal importance is the fact that by separating the geometric transformations from the actual element formulation in 6.78 different geometric theories can be implemented for the same element by selecting the form of the compatibility matrix $a_u$. Under linear geometry matrix $a_u$ simplifies to 6.17 and is displacement independent. Thus, the geometric stiffness is zero and the element stiffness matrix reduces to 6.74 with $a_u = a_a$, as defined in 6.18 without rigid end offsets.

### 6.9.2 Geometric Stiffness Matrix

There remains the task of determining the geometric stiffness $\bar{k}_g$ in 6.77 and 6.78 by taking the derivative $\frac{\partial b}{\partial \mathbf{u}}$. This operation is performed column by column noting that the derivative of the first column multiplies the axial force $q_1$ and supplies the contribution $\bar{k}_g$ to the geometric stiffness matrix, whereas the derivatives of the second and third columns multiply the end moments $q_2$ and $q_3$, respectively. These can be combined to give the contribution $\bar{k}_{g23}$ to the geometric stiffness matrix. From the derivative of the first column of $b_u$ the $\bar{k}_{g1}$ contribution is

$$\bar{k}_{g1} = \left[ \begin{array}{cccc} s^2 & -c_s & 0 & -s^2 \\ -c_s & c^2 & 0 & -c^2 \\ 0 & 0 & 0 & 0 \\ c_s & -c^2 & 0 & c^2 \\ 0 & 0 & 0 & 0 \end{array} \right] \frac{q_1}{L_u}$$

(6.79)

in which

$$c = \frac{L + \Delta \mathbf{u}}{L_u} \quad \text{and} \quad s = \frac{\Delta \mathbf{u}}{L_u}$$

From the second and third columns of $b_u$ we obtain

$$\bar{k}_{g23} = \left[ \begin{array}{cccc} -2sc & c^2 - s^2 & 0 & 2sc \\ c^2 - s^2 & 2cs & 0 & -c^2 + s^2 \\ 0 & 0 & 0 & 0 \\ 2sc & -c^2 + s^2 & 0 & c^2 - s^2 \\ -c^2 + s^2 & -2cs & 0 & c^2 - s^2 \\ 0 & 0 & 0 & 0 \end{array} \right] \frac{q_1 + q_3}{L_u} \frac{1}{L_u}$$

(6.80)
In the scalar factor for the matrix in 6.80, the deformed element length is used in the denominator because the first term represents the shear force in the deformed element configuration. With this identification a comparison of the two matrices in 6.79 and 6.80 is possible. In slender compression elements the shear force is often significantly less than the axial force, so that the contribution \( \tilde{K}_{p3} \) is in such cases significantly smaller than \( \tilde{K}_{p1} \) and can often be neglected.

### 6.9.3 P-\( \Delta \) Geometric Stiffness

We introduce now an approximation of nonlinear geometry that is often used in structural analysis for earthquake-resistant design. It is known by the name P-\( \Delta \) analysis, but this name is confusing for the following reason: when referring to a single member it is convenient to distinguish between the effect of axial force on the free body equilibrium of the entire member, the so-called P-\( \Delta \) effect and the effect of the axial force on the internal forces in the deformed configuration, the so-called P-\( \delta \) effect. Such distinction is, however, not relevant when the member is subdivided into several elements. In such case, even an element that only accounts for the P-\( \Delta \) effect can approximate the effect of the axial force on the internal forces of the member. The greater the number of elements used to model a compression member the more accurate the approximation of the internal forces.

To avoid confusion, we refer to the approximation of the exact geometric transformation by the name P-\( \Delta \) truss geometric stiffness. In this case we assume that the equilibrium matrix is given by \( \mathbf{b}_{p3} \) in 6.23 and the resulting geometric stiffness matrix is

\[
\tilde{K}_g = \tilde{K}_{p3} = \left( \frac{\partial}{\partial \mathbf{u}} \mathbf{b}_{p3} \right) \mathbf{q} = \frac{\mathbf{q}}{L} \begin{bmatrix}
0 & 0 & 0 & 0 & 0 \\
0 & 1 & 0 & 0 & -1 \\
0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 \\
0 & -1 & 0 & 0 & 1 \\
0 & 0 & 0 & 0 & 0
\end{bmatrix}
\]  \( (6.81) \)

Equations 6.81 and 6.79 are similar, leading to the observation that the former can be obtained from the latter by setting \( c = 1, \ s = 0 \) and \( L_n = L \). A difficulty arises, however, in using a consistent deformation–displacement relation for \( \partial \mathbf{v} / \partial \mathbf{u} \). We do not pursue this subject further here and note that commonly \( \mathbf{a}_u \) is set equal to the linear compatibility matrix \( \mathbf{a} \) in 6.77. With these assumptions, the element stiffness in the global coordinate system is

\[
\mathbf{k}^{(d)} = \mathbf{a}^T \tilde{K}_{p3} \mathbf{a} + \mathbf{a}^T \tilde{K}_g \mathbf{a}
\]  \( (6.82) \)

Although simple, the element stiffness matrix in 6.82 deviates quickly from the tangent stiffness to the element–force deformation relation, thus resulting in poor convergence behavior even under moderate displacements.

The geometrically nonlinear behavior with the exact transformation in 6.77 and the matrices in 6.79 and 6.80, or, with the approximate geometrically nonlinear behavior in 6.82 can be improved as necessary by subdividing the compressed member into smaller elements. In such case, the deformations of the actual member relative to the chord of the elements can be made as small as necessary, so that even a very simple basic force–deformation relation can yield excellent results. Thus, the trade-off is between a smaller number of elements with more accurate force–deformation relation and a large number of very simple elements. The decision as to which approach to follow depends on the element library of the computer software used for the structural analysis. The power of the corotational approach lies in its ability to represent accurately nonlinear geometry under large displacements with simple basic force–deformation relations. We will illustrate this in the examples of the following section.
6.10 Examples of Nonlinear Static Analysis

The examples in this section provide a brief overview of the type of problems that are encountered in structural analysis for earthquake engineering.

The response of sections made of one or several materials is of interest in the assessment of local response. Furthermore, it is an important ingredient in the determination of hysteretic element response. The selection of the number of integration points for sufficient accuracy is of particular interest. Figure 6.21 shows the axial force-bending moment interaction diagram of a rectangular section with bilinear elastic-perfectly plastic material. Because the results are normalized with respect to the plastic axial and flexural capacity, $N_p$ and $M_p$, respectively, the dimensions and material properties are not relevant. Figure 6.21 shows clearly that the numerical solution with eight (8) midpoint evaluations for the integrals of 6.43 gives excellent accuracy. The same is true for the monotonic moment–curvature relation of the same section under three different axial load levels of 0, 30 and 60% of the plastic axial capacity $N_p$ in Figure 6.22. In this example the closed-form solution is available and the comparison shows that a few midpoint evaluations suffice for the accurate representation of the section response. The section stiffness in 6.43 involves quadratic terms of the coordinates and is, therefore, more sensitive to the number of integration points. Usually 10 to 15 midpoint evaluations suffice for the purpose. Under biaxial loading 25 ($5 \times 5$) to 64 ($8 \times 8$) fibers yield excellent accuracy. For wide flange sections three layers in each flange and four layers in the web are recommended. In a reinforced concrete section the hysteretic response is dominated by the behavior of reinforcing steel. Thus, it is important to represent the area and distribution of reinforcement relatively well and then use 16 ($4 \times 4$) or 25 ($5 \times 5$) fibers for the concrete. A larger number of fibers may be necessary for distinguishing between cover concrete and core concrete confined by transverse reinforcement.

Figure 6.23 shows a three-story steel frame under the action of vertical and horizontal forces. The material is assumed to be elastic-perfectly plastic. The vertical and horizontal forces are collected into the same reference force vector, so that both are incremented in the following nonlinear static pushover analyses. In typical analyses the vertical forces due to gravity are kept constant, while the lateral forces are incremented to collapse. Figure 6.24 shows the relation between load factor and top story horizontal
displacement. The results are obtained with the elastic-perfectly plastic element of Section 6.7.2.2. Figure 6.24 shows the load-displacement response for two cases:

- In the first case the plastic flexural capacity does not account for the effect of the axial force
- In the second case a linear elastic analysis under the application of the vertical forces yields the axial forces in the columns. These are used to reduce the plastic flexural capacity of the members according to the LRFD specification. This approach does not account for the change in axial force in the columns on account of the overturning moments due to the lateral forces. Nonetheless, it gives a relatively accurate estimate of the collapse load factor of the frame, as will be shown later.

It is interesting to observe that while the collapse load factor is not very different between the two cases, a collapse mechanism forms at a significantly smaller horizontal displacement in the case in which the effect of the axial force on the plastic flexural capacity is accounted for. The cause for this is apparent from Figures 6.24b and c, which show the collapse mechanism for the two
cases. In the first case an almost complete collapse mechanism forms with nine plastic hinges, as shown in Figure 6.24b. In the second case a partial first story collapse mechanism forms, as shown in Figure 6.24c. Clearly, the difference in collapse mechanism of the two cases significantly affects the plastic rotations of the first-story columns for a given horizontal displacement.

Figure 6.25 shows the load factor–top story displacement response of the same frame for the case that a distributed inelasticity element with layer section is used. In this example the force formulation of Section 6.7.3 is used with five control sections. Each section is discretized into 20 layers, 5 in each flange and 10 in the web. Figure 6.25 shows the results of two analyses: in the first case the geometry is linear, while in the second the P-Δ geometric stiffness of Section 6.9.3 is included in the element formulation.

The comparison of the response in Figure 6.25 with that in Figure 6.24a, shows that the axial force variation on account of the overturning moments results in a reduction of the collapse load factor. With a layer discretization of the cross section this effect is automatically accounted for. From the response in Figure 6.25 we conclude that the effect of the P-Δ geometric stiffness becomes noticeable for values of...
the top story horizontal displacement larger than 0.35 ft, which amounts to an average story drift of 1%. Clearly, it is essential to include this effect in the pushover analysis of frame structures. Finally, Figure 6.25 shows that the load control measures that were discussed in Sections 6.8.3 and 6.8.4 permit the tracing of the load-displacement response past the point of peak strength. This is true for the case of softening response and for the case of linear geometry where the load factor remains practically constant after attaining the maximum value. Figures 6.25b and c show the distributions of section deformations and section forces, respectively, at maximum displacement under linear geometry. In the force formulation the distribution of section forces is always exact, as reflected by the constant axial force and linear bending moment distributions in Figure 6.25c. The section deformations in Figure 6.25b show that large inelastic strains take place at the top and bottom end sections of the first-story columns. The accuracy of the inelastic strain estimate depends on the integration weight of the end sections in the element response. In this respect four or five integration points yield results of comparable accuracy to proposals of plastic hinge length estimation. Figure 6.25b shows that an element with inelastic zones of finite length
at the ends and an elastic core is an excellent compromise between concentrated and distributed inelasticity elements for modeling the inelastic response of columns. The distributed inelasticity elements are particularly suitable for the representation of the inelastic response of girders with significant influence of gravity loads.

Figure 6.26a to c shows the response of the same frame under linear geometry with elements based on the displacement formulation. With only one element per member, this model overestimates the collapse load factor by almost 50%, as shown in Figure 6.26a. The cause of this discrepancy is apparent in Figures 6.26b and c, which show the distribution of section deformations and section forces at maximum displacement, respectively. In the basic displacement formulation a constant axial strain and linear curvature distribution is assumed, as shown in Figure 6.26b. The corresponding axial force and bending moment at each section need to satisfy the material response, while equilibrium is not satisfied in a strict sense, but only for the element. This results in the rather unusual axial force and bending moment distributions of Figure 6.26c. To improve the accuracy of the results members with yielding
should be subdivided into several smaller elements, thus increasing significantly the computational cost. Alternatively, higher order elements with internal nodes can be used permitting higher order polynomials for the displacement interpolation functions. Neither approach, however, is completely successful, particularly under cyclic loading conditions, and the force formulation is preferable for inelastic frame elements with distributed inelasticity.

### 6.11 Dynamic Analysis

The equations of motion at the structural DOFs according to 6.2 are

\[ P(t) - P_i(U, \dot{U}) = M \ddot{U} \]

(6.83)

where a dot denotes differentiation with respect to time and we have explicitly noted the variation of applied forces \( P(t) \) with time. \( M \) is the mass matrix of the structure, \( \ddot{U} \) is the total acceleration in a fixed reference frame, and the resisting forces \( P_i \) in general depend on the displacement and velocity at the global DOFs. If the resisting forces are simply linear functions of velocity and displacement, we can simplify 6.83 to the following

\[ P(t) - C U - K U = M \ddot{U} \]

(6.84)

where \( C \) is the viscous damping matrix for the free DOFs of the model.

#### 6.11.1 Free Vibration

Setting the forcing function \( P(t) \) equal to zero in 6.84 and assuming that the viscous damping is zero and that the fixed reference frame is the base of the structure, hence \( \ddot{U} = \ddot{U} \), give the free vibration problem:

\[ M \ddot{U} + K U = 0 \]

(6.85)

The solution of 6.85 can be expressed in terms of the vibration mode shapes \( \Phi \) and natural vibration frequencies \( \omega_i \), as defined by the eigenvalue problem:

\[ K \Phi = \omega^2 M \Phi \]

The number of pairs of \( \omega_i, \Phi \) that satisfies the eigenvalue problem is equal to the number of free DOFs, although in practice much fewer modes are necessary to represent the dynamic response under earthquake excitation. The eigenfrequencies can be collected in a diagonal matrix and the eigenmodes in a matrix of the form

\[ \Omega = \begin{bmatrix} \omega_1 & 0 & 0 \\ 0 & \ddots & 0 \\ 0 & 0 & \omega_n \end{bmatrix} \]

\[ \Phi = \begin{bmatrix} \Phi_1 & \cdots & \Phi_n \end{bmatrix} \]

These satisfy the equation

\[ K \Phi = M \Phi \Omega^2 \]

(6.86)

The key characteristic of the mode shapes is that they are orthogonal with respect to the mass and stiffness matrices. Premultiplying both sides of 6.86 by \( \Phi^T \) gives the following relationship:
both sides of which are diagonal matrices because of orthogonality (Chopra, 2001). Since the modes can be scaled arbitrarily we select a scaling such that $\Phi^T M \Phi = I$, where $I$ is the identity matrix. This is known as orthonormality property of the vibration modes. With this scaling of the eigenmodes we obtain from 6.87 that

$$\Omega^2 = \Phi^T K \Phi$$

(6.88)

### 6.11.2 Modal Analysis for Linear Response

Returning to the solution of the free vibration problem in 6.85 with initial conditions on the displacement, $U_0$, and velocity, $\dot{U}_0$, the displacement vector can be represented as summation of contributions of the vibration modes:

$$U = \Phi^T Y$$

(6.89)

in which $Y=Y(t)$ are called the generalized coordinates. Usually much fewer generalized coordinates are needed compared with the number of free DOFs. After substituting 6.89 into 6.85 and premultiplying the equation by $\Phi^T$ we obtain

$$\Phi^T M \Phi \ddot{Y} + \Phi^T K \Phi Y = 0$$

which on account of 6.88 and the orthogonality of the mode shapes gives

$$\ddot{Y} + \Omega^2 Y = 0$$

(6.90)

6.90 are uncoupled second order initial value problems with the following solution for mode $k$:

$$Y_k(t) = Y_{0k} \cos(\omega_k t) + \frac{\dot{Y}_{0k}}{\omega_k} \sin(\omega_k t)$$

Using 6.89 the initial values for the generalized coordinates can be expressed in terms of the initial conditions by premultiplying both sides of the equation with $\Phi^T M$:

$$\Phi^T M U_0 = \Phi^T M \Phi Y_0 = Y_0 \Rightarrow Y_0 = \Phi^T M U_0$$

$$\Phi^T M \dot{U}_0 = \Phi^T M \Phi \dot{Y}_0 = \dot{Y}_0 \Rightarrow \dot{Y}_0 = \Phi^T M \dot{U}_0$$

For the more interesting forced vibration case with damping in 6.84 we obtain

$$\Phi^T M \Phi \ddot{Y} + \Phi^T C \Phi \dot{Y} + \Phi^T K \Phi Y = \Phi^T P(t)$$

which on account of the orthogonality properties of the vibration modes gives

$$\ddot{Y} + [\Phi^T C \Phi] \dot{Y} + \Omega^2 Y = \Phi^T P(t)$$

(6.91)

For a general damping matrix all modes are coupled through the damping terms in 6.91. However, since damping is generally assumed, it is reasonable to use the so-called Rayleigh damping and express the
damping matrix in terms of the mass and stiffness matrix: $C = \alpha_0 M + \alpha_1 K$. Since the mode shapes are orthogonal to $M$ and $K$, they are also orthogonal to this specific form of the damping matrix. Substitution of Rayleigh damping into (6.91) gives

$$\ddot{Y} + (\alpha_0 \hat{\omega}_k + \alpha_1 \hat{\omega}_1) \dot{Y} + \Omega^2 Y = \Phi^T P(t) \tag{6.92}$$

The Rayleigh damping coefficients $\alpha_0$ and $\alpha_1$ are selected to match the desired damping ratio for two modes, oftentimes the two lowest, but not always. Calling these modes $k$ and $m$ we can write

$$\omega_k + \alpha_0 \omega_k^2 = 2\zeta_k \omega_k$$

with

$$\omega_m + \alpha_1 \omega_m^2 = 2\zeta_m \omega_m \tag{6.93}$$

With given damping ratios $\zeta_k$ and $\zeta_m$ we can solve the two equations in 6.93 for $\alpha_0$ and $\alpha_1$ (Chopra, 2001). The damping ratio for another mode is given by

$$\zeta_n = \frac{1}{2} \left( \frac{\alpha_0 + \alpha_1 \omega_n^2}{\omega_n^2} \right)$$

### 6.11.3 Earthquake Excitation

In the case of earthquake excitation the support DOFs are assumed to move together through a specified ground acceleration history, $\ddot{U}_g(t)$, in the global coordinate system, which generally has two components for 2d problems (a horizontal and a vertical acceleration) and three components for 3d problems (two horizontal accelerations and one vertical acceleration). The key step is to define the acceleration with respect to the fixed reference frame as the sum of the acceleration of the support DOFs and the additional acceleration of the free DOFs relative to the supports, expressed as follows:

$$\ddot{U} = \ddot{U} + \ddot{R}_{t} \dot{U}(t) \tag{6.94}$$

The number of columns in $R$ is equal to the number of specified support acceleration components. For each component, the column of $R$ corresponds to the displacements of the free DOFs due to a unit displacement of the corresponding support. If all supports move as a rigid body, then $R$ represents the rigid-body displacement of the entire structure. For linear systems, this procedure can be generalized to include different motions at the supports, in which case $R$ represents the displacements of the free DOFs due to unit displacement of each support and is obtained by solving the static support displacement problem (Clough and Penzien, 1993).

Under the assumption of no applied nodal loads, the substitution of 6.94 into 6.83 or 6.84 gives the equations of motion due to earthquake excitation for the nonlinear and linear models, respectively:

$$M\ddot{U} + P(U, \dot{U}) = -MR\ddot{U} \tag{6.95}$$

### 6.11.4 Numerical Integration of Equations of Motion for Linear Response

Because of the difficulty of solving the linear differential equation for arbitrary variation of forcing functions as a function of time (earthquake excitation is a particularly complex case in point), it is necessary to use a numerical method to solve 6.95. In the case of linear elastic response of the structural
system we use modal analysis and thus integrate numerically \( m \) single DOF differential equations of the form

\[
\ddot{Y}_i + (\alpha_c + \alpha_m \omega_n^2) \dot{Y}_i + \omega_n^2 Y_i = P_i(t)
\]

where \( P_i(t) = -\{\Phi^T MR\}_{i} \dot{U}_g(t) \). In general, the number of modes should be selected based on the frequency content and spatial participation of the modes as indicated by the participating mass (Chopra, 2001).

In the numerical solution of differential equations the acceleration, velocity and displacement at time \( t + \Delta t \) are defined in terms of the acceleration, velocity and displacement at time \( t \). To keep the notation short we use subscript \( i+1 \) for time \( t+\Delta t \) and subscript \( i \) for time \( t \), respectively. Because the equations are the same whether we are dealing with a single DOF or a multi-DOF system we use the latter in the following presentation for generality.

Newmark introduced one of the most widely used methods of numerical integration in earthquake engineering (Newmark, 1959). It uses the following relations between displacement, velocity and acceleration at time steps \( i \) and \( i+1 \):

\[
\dot{U}_{i+1} = \dot{U}_i + (1-\gamma)\Delta t \ddot{U}_i + \gamma \Delta t \ddot{U}_{i+1}
\]

\[
U_{i+1} = U_i + \Delta t \dot{U}_i + \left(\frac{1}{2} - \beta \right) \Delta t^2 \ddot{U}_i + \beta \Delta t^2 \ddot{U}_{i+1}
\] (6.96)

where subscript \( f \) for the free DOFs has been dropped for convenience. With the second equation in 6.96, \( \ddot{U}_{i+1} \) can be expressed in terms of the displacement at time step \( i+1 \) and the response at the previous time step:

\[
\ddot{U}_{i+1} = \frac{1}{\beta \Delta t^2} (U_{i+1} - U_i) - \frac{1}{\beta \Delta t} \ddot{U}_i - \left(\frac{1}{2\beta} - 1\right) \dddot{U}_i
\] (6.97)

Substituting 6.97 into the first equation in 6.96 gives the velocity, \( \dot{U}_{i+1} \):

\[
\dot{U}_{i+1} = \dot{U}_i + (1-\gamma)\Delta t \ddot{U}_i + \frac{\gamma}{\beta \Delta t} (U_{i+1} - U_i) - \frac{\gamma}{\beta} \ddot{U}_i - \left(\frac{1}{2\beta} - 1\right) \gamma \Delta t \dddot{U}_i
\]

\[
= \frac{\gamma}{\beta \Delta t} (U_{i+1} - U_i) + \left(1 - \frac{\gamma}{\beta}\right) \dot{U}_i + \left(1 - \frac{\gamma}{2\beta}\right) \Delta t \dddot{U}_i
\] (6.98)

We introduce now the following constants for a given time step \( \Delta t \):

\[
C_0 = \frac{1}{\beta \Delta t^2}, \quad C_1 = \frac{1}{\beta \Delta t}, \quad C_2 = \frac{\gamma}{\beta \Delta t}, \quad C_3 = \left(\frac{1}{2\beta} - 1\right), \quad C_4 = \left(\frac{\gamma}{\beta} - 1\right), \quad C_5 = \left(\frac{\gamma}{2\beta} - 1\right) \Delta t
\]

and rewrite 6.97 and 6.98 in a more compact form:

\[
\dot{U}_{i+1} = C_0 (U_{i+1} - U_i) - C_1 \dot{U}_i - C_2 \dddot{U}_i
\] (6.99)

\[
U_{i+1} = C_1 (U_{i+1} - U_i) - C_4 \dot{U}_i - C_5 \dddot{U}_i
\] (6.100)
We use 6.97 and 6.98 in two ways: first we substitute the velocity and acceleration in the equations of motion 6.84 at time \( t + \Delta t \), i.e.,

\[
P_{i+1} - \mathbf{C}\ddot{\mathbf{U}}_{i+1} - \mathbf{K}\mathbf{U}_{i+1} = \mathbf{M}\dddot{\mathbf{U}}_{i+1}
\]  

(6.101)

and obtain a system of equations for the unknown displacement at time \( t + \Delta t \) :

\[
P_{i+1} - \mathbf{C}\left[\mathbf{C}_m(\mathbf{U}_{i+1} - \mathbf{U}_i) - \mathbf{C}_a\dot{\mathbf{U}}_i - \mathbf{C}_i\ddot{\mathbf{U}}_i\right] - \mathbf{K}\mathbf{U}_{i+1} = \mathbf{M}\left[\mathbf{C}_m(\mathbf{U}_{i+1} - \mathbf{U}_i) - \mathbf{C}_a\dot{\mathbf{U}}_i - \mathbf{C}_i\ddot{\mathbf{U}}_i\right]
\]

After collecting terms for the unknown displacement at time \( t + \Delta t \) we get

\[
(C_m\mathbf{M} + \mathbf{C}_m\mathbf{C} + \mathbf{K})\mathbf{U}_{i+1} = P_{i+1} + \mathbf{U}_i(C_m\mathbf{M} + \mathbf{C}_m\mathbf{C}) + \left(C_a\dot{\mathbf{U}}_i + C_i\ddot{\mathbf{U}}_i\right)\mathbf{M} + \left(C_a\dot{\mathbf{U}}_i + C_i\ddot{\mathbf{U}}_i\right)\mathbf{C}
\]

or, in short, a system of linear equations:

\[
\mathbf{K}_{\text{eff}}\mathbf{U}_{i+1} = \mathbf{P}_{\text{eff}}
\]  

(6.102)

Solving for the displacements at time \( t + \Delta t \) from 6.102, 6.97 and 6.98 gives the velocities and accelerations at the free DOFs at time \( t + \Delta t \), thus advancing the solution of the equation of motion by one time increment. Repeating this process for the necessary number of time steps gives the solution as a function of time.

Because a solution of simultaneous equations is required in 6.102, Newmark’s numerical solution belongs to the class of implicit methods. The numerical stability and accuracy of such methods are discussed elsewhere (Hughes, 2000).

### 6.11.5 Numerical Integration of Equations of Motion for Nonlinear Response

Newmark’s time integration algorithm can now be used to solve the equations of motion for a general nonlinear model of a structure. In this case 6.101 is written as follows for time \( t + \Delta t \)

\[
P_{i+1} - P_i(\ddot{\mathbf{U}}_{i+1}, \mathbf{U}_{i+1}) = \mathbf{M}\dddot{\mathbf{U}}_{i+1}
\]

Assuming that the resisting forces are linearly dependent on the velocity, the equations of motion become

\[
P_{i+1} - \mathbf{C}\ddot{\mathbf{U}}_{i+1} - P_i(\mathbf{U}_{i+1}) = \mathbf{M}\dddot{\mathbf{U}}_{i+1}
\]  

(6.103)

Since 6.103 is a nonlinear system of equations, the Newton–Raphson algorithm is needed to solve it. In analogy with the static case, it is necessary to obtain the derivative of 6.103 with respect to the unknown displacements at time \( t + \Delta t \). The velocities and accelerations are expressed in terms of these displacements using the result from Newmark’s method. Thus, the chain rule of differentiation on 6.103 gives the effective stiffness matrix at time step \( i+1 \):

\[
\mathbf{K}_{\text{eff}} = \frac{\partial}{\partial \mathbf{U}_{i+1}} \left[ \mathbf{M}\dddot{\mathbf{U}}_{i+1} + \mathbf{C}\ddot{\mathbf{U}}_{i+1} + P_i(\mathbf{U}_{i+1}) \right] - \frac{\partial}{\partial \mathbf{U}_{i+1}} P_{i+1}
\]

\[
\mathbf{K}_{\text{eff}} = \frac{\partial}{\partial \dddot{\mathbf{U}}_{i+1}} \left[ \mathbf{M}\dddot{\mathbf{U}}_{i+1} + \mathbf{C}\ddot{\mathbf{U}}_{i+1} + P_i(\mathbf{U}_{i+1}) \right] - \frac{\partial}{\partial \mathbf{U}_{i+1}} P_{i+1}
\]  

(6.104)
where we note that the applied forces $P_{i_1}$ do not depend on the displacements $U_{i_1}$. After substituting 6.99 and 6.100 in 6.104, we conclude that the effective stiffness is similar to the linear case except for the fact that the tangent stiffness matrix is used,

$$K_{eff} = C_0 M + C_2 C + K_{i_1}$$

where $K_{i_1} = \frac{\partial P(U_{i_1})}{\partial U_{i_1}}$ in accordance with the tangent stiffness definition in 6.72. The force unbalance vector of the equation in 6.103 is also needed. It expresses the amount of equilibrium error under inclusion of the mass and damping terms. It is given by

$$P_u = P_{i_1} - P(U_{i_1}) - M\ddot{U}_{i_1} - C\dot{U}_{i_1}$$ \hspace{1cm} (6.105)

Substituting the expressions in 6.99 and 6.100 for the acceleration and velocity at time $t + \Delta t$ in a slightly modified form in 6.105 gives the unbalanced force vector as

$$P_u = P_{i_1} - P(U_{i_1}) - M\ddot{U}_{i_1} - C\dot{U}_{i_1}$$ \hspace{1cm} (6.106)

where $\Delta U_{i_1} = U_{i_1} - U_{i_1}$. Note that during Newton–Raphson iterations $U_{i_1}$ is updated during each iteration, while $U_{i_0}$, of course, remains constant and equal to the displacement values at the previously converged time step. With this in mind we collect terms in 6.106 as follows

$$P_u = P_{i_1} + M\begin{bmatrix} C_1 \ddot{U}_{i_1} + C_2 \dot{U}_{i_1} \end{bmatrix} + C\begin{bmatrix} C_4 \ddot{U}_{i_1} + C_5 \dot{U}_{i_1} \end{bmatrix} - P(U_{i_1}) - C_0 M\Delta U_{i_1} - C_2 C\Delta U_{i_1}$$ \hspace{1cm} (6.107)

The first half of the right-hand side in 6.107, i.e., $P_{i_1} + M\begin{bmatrix} C_1 \ddot{U}_{i_1} + C_2 \dot{U}_{i_1} \end{bmatrix} + C\begin{bmatrix} C_4 \ddot{U}_{i_1} + C_5 \dot{U}_{i_1} \end{bmatrix}$ does not change during a time step and can be considered the effective applied force. The second half, namely $-P(U_{i_1}) - C_0 M\Delta U_{i_1} - C_2 C\Delta U_{i_1}$ needs to be updated with every new estimate of the displacements $U_{i_1}$ during equilibrium iterations and can be regarded as the effective resisting force vector.

### 6.12 Applications of Linear and Nonlinear Dynamic Analysis

As an example of the structural analysis methods presented in this chapter, this section presents the nonlinear dynamic analysis of a 20-story moment-resisting steel frame building. The example building, designed for the seismic hazard in Los Angeles, has been used in the SAC studies to assess the performance of steel moment-resisting frame buildings (Gupta and Krawinkler, 1999). The building has 20 stories above ground level and two basement levels. The total height above the ground is 265 ft and the story height is 13 ft except for the ground level story of 18 ft height. The North-South frames consist of five bays with perimeter box columns of $15 \times 15$ with various thicknesses and interior columns with wide flange sections varying from W24 $\times$ 335 to W24 $\times$ 84. The beams consist of various wide flange section members ranging between W30 $\times$ 108 and W21 $\times$ 50.

The structural model of the NS frame, shown in Figure 6.27, consists of two-node frame elements connected at nodes representing the joints. Centerline dimensions are used and the joints are assumed to be rigid. The base of the columns is hinged and the perimeter basement columns are constrained in the horizontal direction at the ground level to represent the embedment of the basement, although a more refined model could include soil–foundation–structure interaction effects. The frame elements represent distributed inelasticity with five control sections. Each element has section properties with a discretization of typically 60 layers (fibers), although as described in Section 6.10 a smaller number can
The material model for the steel is a bilinear plasticity model with 2% strain hardening ratio. The structural model has 585 free DOFs for the translational and rotational components at the nodes.

The mass of the building is represented by lumped masses at the nodes of the model. The gravity resisting frames in the building are not included in the model, but they contribute substantial P-D effects to the moment-resisting frame. To account for the destabilizing effect of the gravity loads on the gravity resisting frames, the loads are collected to an additional column member that is attached to the moment-resisting frame by truss members. This is commonly known as leaning column approach. The geometric compatibility transformation for the beams and columns, including the leaning column, uses the P-Δ transformation presented in Section 6.9.3.

The lower vibration mode shapes and periods, using the stiffness matrix of the building under linear elastic behavior, are shown in Figure 6.28. For dynamic analysis, Rayleigh damping is assumed based on a damping ratio of 0.02 in the first two vibration modes.

The horizontal ground motion record used in this example analysis is obtained from the simulation of a fault rupture and resulting wave propagation in a 10 km × 10 km region (Bao et al., 1996). The location of the station is in the forward rupture directivity region and is about 1 km from the surface.
projection of the fault. The simulated ground motion has a pulse with large peak ground acceleration of 2 g. Although such large peak ground accelerations have not been recorded to date under this condition, the simulated record has the large pulse that is characteristic of near-source ground motion and it may be considered a very severe case for the expected ground motion. The purpose of using the large simulated record is to investigate the inelastic behavior of the frame in an extreme event. The ground motion acceleration record is shown in Figure 6.27b, and it is applied as horizontal free-field acceleration at the base of the model. For the nonlinear dynamic analysis, the Newmark time integration method is used with 1800 time steps of $\Delta t = 0.005$ sec. The equilibrium equations at each time step are solved using the Newton–Raphson algorithm, and typically three or four iterations were required for convergence. On a desktop computer the analysis took three to five minutes, demonstrating the computational efficiency of the nonlinear dynamic analysis methods.

The history of displacement at five floors is shown in Figure 6.29. The propagation of the ground motion pulse over the height of the building can be clearly seen. The maximum horizontal roof displacement of 38 in. results in residual deformation at the end of the 9-sec response history. The envelopes of maximum horizontal floor displacement and drift are shown in Figure 6.30. The largest story drift occurs in the first floor, with a drift ratio of 0.026, and the upper floors.
The location and maximum values of the plastic hinge rotations in the beams and columns are shown in Figure 6.31. Most of the hinges form at the end of the beams, with the largest rotation, reaching 0.030 rad, in floors 3 to 5, and somewhat smaller plastic hinging in the upper four floors. Plastic hinges form at the bases of the tall columns at the ground level, as would be expected in a sway mechanism, but limited column hinging of approximately 0.01 rad occurs at perimeter and interior columns also in floors 3 to 5. In addition there is limited plastic hinging in upper floor columns (16 to 18). The deformed shape in Figure 6.31 shows the residual displacements of the frame after the earthquake ends, with the shape magnified by a scale factor of 20 for plotting.

The modeling and computation for this example were done with OpenSees — Open System for Earthquake Engineering Simulation (http://opensees.berkeley.edu). OpenSees is an open-source software
Methods of Analysis for Earthquake-Resistant Structures

6.13 Conclusions

The objective of this chapter has been to present the methods of structural analysis in a manner that unifies static and dynamic analysis for linear and nonlinear models. The emphasis has been on providing a consistent approach for satisfying the equations of equilibrium, compatibility and force–deformation. The methods presented in this chapter encompass the major structural analysis procedures used in earthquake-resistant design, and they recognize the increasing importance of nonlinear analysis procedures. The presentation has been limited to frame elements, although the methods can be extended to include joints, walls, diaphragms and foundation components.

Plastic analysis methods are very useful in the capacity design of structures but are limited to elastic-perfectly plastic behavior. Concentrated plasticity beam models are computationally simple and are capable of accounting for the effects of axial force-shear-bending moment interaction. These models, however, require calibration under idealized assumptions about either the force or the deformation distribution within the member. Furthermore, they require that the location of inelastic deformations be specified a priori and, typically, do not include the effect of distributed element loading. Distributed inelasticity frame elements do not suffer from the limitations of calibration and a priori specification of the location of inelastic deformation, but are computationally more demanding. In the displacement formulation the assumed displacement interpolation functions are not a satisfactory approximation of the deformation distribution in the member, unless the latter is subdivided into several elements. Adaptive mesh refinement methods have been proposed for the purpose. The force formulation offers the advantage that the force-interpolation functions are exact under the assumption of linear geometry. Consequently, a single element with several integration points (control sections) suffices for the representation of the inelastic behavior of the member. Moreover, the force formulation accounts directly for the effect of distributed element loads in girders, which can cause inelastic deformations to arise within the member span, instead of the member ends. Four integration points are recommended for the typical case without element loads, while five integration points should be used in the presence of element loading. By integrating the material response over the control sections with the so-called layer or fiber section models, it is possible to directly account for the interaction of axial force and bending moment. Simple uniaxial normal stress–strain models suffice for the purpose. Studies show that a few layers or fibers suffice to yield an excellent representation of the hysteretic response of the section. Under uniaxial bending eight to ten layers are usually sufficient for rectangular sections. For wide flange sections three layers in each flange and four layers in the web are recommended. Under biaxial loading 25 (5 × 5) to 64 (8 × 8) fibers yield excellent accuracy. In a reinforced concrete section the hysteretic response is dominated by the behavior of reinforcing steel. Thus, it is important to represent the area and distribution of reinforcement relatively well and then use 16 (4 × 4) or 25 (5 × 5) fibers for the concrete. A larger number of fibers may be necessary for distinguishing between cover concrete and core concrete confined by transverse reinforcement.

Under nonlinear geometry the most significant contribution arises from the rigid-body displacements of the frame element. It is possible to isolate this effect with the corotational formulation, in which the element response is defined in the basic system without rigid-body modes. The end forces of the basic system are then transformed exactly to the local coordinate system of the undeformed element. In this case the tangent stiffness matrix of the element is made up of two contributions: the transformation of the material stiffness from the basic to the local system and the geometric stiffness matrix. Consistent approximation of the displacement terms in the equilibrium equations and in the deformation–displacement relations leads to approximate theories of nonlinear geometry, such as the P-Δ geometric stiffness. The advantage of the presented approach is that one element type can accommodate several nonlinear geometric transformations. The effect of nonlinear geometry should be included in the nonlinear static
Earthquake Engineering: From Engineering Seismology to Performance-Based Engineering

(pushover) analysis of buildings for average relative story drifts in excess of 1%. When the relative story drift varies considerably over the height of the structure, it is important to include the effect of nonlinear geometry when the maximum inter-story drift exceeds 2%.

The strength softening response of structural systems under nonlinear material and geometry can be traced with load control strategies. Among these the load control strategy under constant displacement at a particular degree of freedom proves very useful in the nonlinear static (pushover) analysis of buildings, when the horizontal translation at a floor is representative of the response of the entire building, or, a single soft story collapse mechanism forms. In more complex cases the load control strategy under constant external work is an excellent alternative.

Most structural analysis is performed using computer software that implements one or more of the analysis methods described in this chapter. When using computer software for analysis, the engineer must confirm that the assumptions and limitations of the models are appropriate for the structural analysis problem under consideration.

6.14 Future Challenges

Nonlinear structural analysis is becoming more important in earthquake-resistant design, particularly with the development of performance-based earthquake engineering, which requires more detailed information about the displacements, drifts and inelastic deformation of a structure than traditional design procedures. Nevertheless, many challenges remain in the field of structural analysis to meet the goal of providing predictive simulations of the performance of a structure under earthquake excitation. The challenges encompass the needs for research in analysis and simulation, improved technology for structural analysis software and education of students and design professionals in structural analysis advances.

The Pacific Earthquake Engineering Research Center has undertaken the development of the Open System for Earthquake Engineering Simulation (OpenSees) to address these challenges. The OpenSees software is called a framework because it is an integrated set of software components used to build simulation applications for structural and geotechnical engineering problems. OpenSees is not a “code,” by the usual definition of a program, to solve a specific class of problems. Rather it involves a set of classes and objects that represent models, perform computations for solving the governing equations and provide access to databases for the processing of results. At its most fundamental level, OpenSees can be viewed as a set of objects that are accessed through a defined application program interface (API). The framework was designed using object-oriented principles, and is implemented in C++, a widely used object-oriented programming language. The development of OpenSees is open-source, meaning that all versions of the program, documentation, examples, are available on the website (http://openseis.berkeley.edu) for researchers, professionals and students interested in using and contributing to the software.

PEER's OpenSees research and development addresses three major future challenges. The first challenge is to improve the models of structural behavior of components, and particularly the representation of damage under cyclic loading. Although the computational methods for analysis have become more sophisticated in the past decade, the models used in many nonlinear analyses consist of very simple elements. Simple nonlinear models, such as the lumped plasticity models described in Section 6.7.2, provide an indication of the nonlinear behavior of a structure, but they do not include several important aspects of behavior that can have an appreciable effect on performance. For example, the interaction between flexure and shear, particularly in reinforced concrete members, is poorly understood and current models only approximately attempt to capture this phenomenon, if it is included at all. Beyond the component models, system models of structures are generally very approximate. Quite often these models are two dimensional with the approximation of three-dimensional effects. In particular, models for slabs and diaphragms are rarely used in earthquake analysis, and structural walls are represented with beam-column elements. The simple system models can only provide an approximate assessment of the failure sequence, particularly for structures with components of limited ductility and brittle behavior, which
then requires significant judgment and interpretation on the part of the engineer about the performance of the system. Another important system aspect rarely considered in an analysis is the interaction between the structure, foundation components and soil during an earthquake. In many cases, soil–foundation–structure interaction can affect the response and it should be included in the model and analysis of the system. As with all models, there are great challenges in validating the models using experimental and field-observation data and characterizing the sensitivity of the modeled response in terms of the uncertainty in identifying the parameters of the models. Each of these issues has been a subject of research in PEER and new models and approaches, particularly for soil–structure–foundation interaction, have been incorporated into OpenSees.

A second major area of challenge is the observation that the improvements in structural analysis methods and software have not kept pace with the rapid improvement in computing over the past decade. It is common for an engineer to perform a nonlinear analysis of a structure on a desktop computer today. However, it is often with software that uses simple models because the rate of innovation in the software has not been as rapid as the hardware technology that produced the high-performance computer on the desktop. There are tremendous opportunities with new technology for major improvements in structural analysis for earthquake engineering. Considering hardware, there will be increasing computational power on not only individual computers, but also on networks of computers connected together in a design office or remote computational centers that will allow for parallel computation transparent to the user. This computational power will allow routine analysis of complete three-dimensional models. Perhaps even more important are the challenges that must be met to develop the analysis software of the future. New advances in software engineering of modularity and open standards hold promise in the earthquake engineering field for advances in software development to support analysis and design applications. In addition to modeling and computational aspects, modular and open software can provide improved facilities for the visualization of structural behavior, linkages to databases for experimental data and validation studies and design databases. Software can provide support for engineers to collaborate, not only on analysis, but also on integrating the analysis with the design process. The OpenSees framework addresses these shortcomings by providing well-defined interfaces to software components for modeling and analysis, and also software tools for equation solvers, visualization, databases and distributed network communication.

The third challenge is educating future and current earthquake engineers in modern methods of structural analysis and application to earthquake-resistant design, and also implementation in modern computational environments, such as OpenSees. This chapter has presented the fundamentals of analysis in a way that can be integrated into undergraduate and graduate curricula, serve as a framework for future advances and provide the necessary background for engineers to use nonlinear analysis methods with confidence. It is hoped that the consistent exposition of the fundamentals and examples of structural analysis applications is a step toward the goal of improving the education of engineers on this important subject.

References


Pau


**Glossary**

**basic element forces** — set of independent element forces in equilibrium equations of element free body

**concentrated or lumped inelasticity** — inelastic deformations may arise at specific locations along the element axis, typically at the element ends

**corotational formulation** — element force-deformation relation is set up in a reference system that moves with the element as it deforms

**distributed inelasticity** — inelastic deformations may arise anywhere along element axis

**element deformations** — relative element end displacements excluding rigid body modes

**layer or fiber section** — integration of material response over the cross section in one or two dimensions by midpoint rule

**load factor control** — relations for load factor adjustment during load incrementation and/or equilibrium iterations

**local coordinate system** — orthogonal Cartesian coordinate system with x-axis coinciding with the line connecting the end nodes of the element

**modal analysis** — decomposition of linear dynamic response in eigenvector contributions

**nonlinear geometry** — large displacement compatibility relations and equilibrium in the deformed configuration

**nonlinear response** — nonlinear relation between displacements at global degrees of freedom and corresponding resisting forces

**P-Δ geometric stiffness** — small displacement compatibility relations and equilibrium in the deformed configuration for axial force effect only

**push-over analysis** — step-by-step nonlinear analysis to collapse under constant gravity loads and a reference lateral force vector with gradually increasing load factor

**section deformations** — deformation measures of infinitesimal slice of frame element

**section forces** — resultant forces at section of frame element

**structural model** — collection of points (structural nodes) in space interconnected by structural elements