

SAP2000[®]

**Linear and Nonlinear
Static and Dynamic
Analysis and Design
of
Three-Dimensional Structures**

BASIC ANALYSIS REFERENCE MANUAL



Computers and Structures, Inc.
Berkeley, California, USA

Version 9.0
August 2004

COPYRIGHT

The computer program SAP2000 and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of the program or reproduction of the documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

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

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

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

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

ACKNOWLEDGMENT

Thanks are due to all of the numerous structural engineers, who over the years have given valuable feedback that has contributed toward the enhancement of this product to its current state.

Special recognition is due Dr. Edward L. Wilson, Professor Emeritus, University of California at Berkeley, who was responsible for the conception and development of the original SAP series of programs and whose continued originality has produced many unique concepts that have been implemented in this version.

Table of Contents

Chapter I	Introduction	I
	About This Manual	1
	Topics	2
	Typographic Conventions	2
	Bibliographic References	3
Chapter II	Objects and Elements	5
Chapter III	Coordinate Systems	7
	Overview	7
	Global Coordinate System	8
	Upward and Horizontal Directions	8
	Local Coordinate Systems	9
Chapter IV	The Frame Element	II
	Overview	12
	Joint Connectivity	13
	Joint Offsets	13
	Degrees of Freedom	14
	Local Coordinate System	14
	Longitudinal Axis 1	15
	Default Orientation	15
	Coordinate Angle	15
	Section Properties	17
	Local Coordinate System	17

	Material Properties	17
	Geometric Properties and Section Stiffnesses	17
	Shape Type	18
	Section Property Database Files	20
	Insertion Point	22
	End Offsets	24
	Clear Length	24
	Effect upon Internal Force Output	24
	Effect upon End Releases	25
	End Releases	25
	Unstable End Releases	26
	Effect of End Offsets	27
	Mass	27
	Self-Weight Load	27
	Concentrated Span Load	28
	Distributed Span Load	28
	Loaded Length	29
	Load Intensity	30
	Internal Force Output	32
	Effect of End Offsets	34
Chapter V	The Shell Element	35
	Overview	36
	Joint Connectivity	37
	Degrees of Freedom	40
	Local Coordinate System	40
	Normal Axis 3	41
	Default Orientation	41
	Coordinate Angle	41
	Section Properties	42
	Section Type	43
	Thickness Formulation	43
	Material Properties	44
	Thickness	44
	Mass	45
	Self-Weight Load	45
	Uniform Load	45
	Internal Force and Stress Output	46
Chapter VI	Joints and Degrees of Freedom	49

	Overview	50
	Modeling Considerations	51
	Local Coordinate System	52
	Degrees of Freedom	52
	Available and Unavailable Degrees of Freedom	53
	Restrained Degrees of Freedom	54
	Constrained Degrees of Freedom	54
	Active Degrees of Freedom	54
	Null Degrees of Freedom	55
	Restraints and Reactions	55
	Springs	57
	Masses	58
	Force Load	59
	Ground Displacement Load	59
	Restraint Displacements	61
	Spring Displacements	61
Chapter VII	Joint Constraints	65
	Overview	65
	Diaphragm Constraint	66
	Joint Connectivity	66
	Plane Definition	67
	Local Coordinate System	68
	Constraint Equations	68
Chapter VIII	Static and Dynamic Analysis	69
	Overview	70
	Loads	70
	Load Cases	70
	Acceleration Loads	71
	Analysis Cases	71
	Linear Static Analysis	72
	Modal Analysis	73
	Eigenvector Analysis	73
	Ritz-vector Analysis	74
	Modal Analysis Results	75
	Response-Spectrum Analysis	77
	Local Coordinate System	78
	Response-Spectrum Functions	79
	Response-Spectrum Curve	79

Modal Combination	81
Directional Combination	82
Response-Spectrum Analysis Results	84

Chapter IX	Bibliography	87
-------------------	---------------------	-----------

Introduction

SAP2000 is the latest and most powerful version of the well-known SAP series of structural analysis programs.

About This Manual

This manual describes the basic and most commonly used modeling and analysis features offered by the SAP2000 structural analysis program. It is imperative that you read this manual and understand the assumptions and procedures used by the program before attempting to create a model or perform an analysis.

The complete set of modeling and analysis features is described in the *SAP2000, ETABS, and SAFE Analysis Reference Manual*.

As background material, you should first read chapter “The Structural Model” in the *SAP2000 Getting Started* manual earlier in this volume. It describes the overall features of a SAP2000 model. The present manual (*Basic Analysis Reference*) will provide more detail on some of the elements, properties, loads, and analysis types.

Topics

Each chapter of this manual is divided into topics and subtopics. Most chapters begin with a list of topics covered. Following the list of topics is an Overview which provides a summary of the chapter.

Typographic Conventions

Throughout this manual the following typographic conventions are used.

Bold for Definitions

Bold roman type (e.g., **example**) is used whenever a new term or concept is defined. For example:

The **global coordinate system** is a three-dimensional, right-handed, rectangular coordinate system.

This sentence begins the definition of the global coordinate system.

Bold for Variable Data

Bold roman type (e.g., **example**) is used to represent variable data items for which you must specify values when defining a structural model and its analysis. For example:

The Frame element coordinate angle, **ang**, is used to define element orientations that are different from the default orientation.

Thus you will need to supply a numeric value for the variable **ang** if it is different from its default value of zero.

Italics for Mathematical Variables

Normal italic type (e.g., *example*) is used for scalar mathematical variables, and bold italic type (e.g., ***example***) is used for vectors and matrices. If a variable data item is used in an equation, bold roman type is used as discussed above. For example:

$$0 \quad \mathbf{da} < \mathbf{db} \quad L$$

Here **da** and **db** are variables that you specify, and *L* is a length calculated by the program.

Italics for Emphasis

Normal italic type (e.g., *example*) is used to emphasize an important point, or for the title of a book, manual, or journal.

All Capitals for Literal Data

All capital type (e.g., EXAMPLE) is used to represent data that you type at the keyboard exactly as it is shown, except that you may actually type lower-case if you prefer. For example:

SAP2000

indicates that you type “SAP2000” or “sap2000” at the keyboard.

Capitalized Names

Capitalized names (e.g., Example) are used for certain parts of the model and its analysis which have special meaning to SAP2000. Some examples:

Frame element

Diaphragm Constraint

Frame Section

Load Pattern

Common entities, such as “joint” or “element” are not capitalized.

Bibliographic References

References are indicated throughout this manual by giving the name of the author(s) and the date of publication, using parentheses. For example:

See Wilson and Tetsuji (1983).

It has been demonstrated (Wilson, Yuan, and Dickens, 1982) that ...

All bibliographic references are listed in alphabetical order in Chapter “Bibliography” (page 87).

Objects and Elements

The physical structural members in a SAP2000 model are represented by objects. Using the graphical user interface, you “draw” the geometry of an object, then “assign” properties and loads to the object to completely define the model of the physical member.

The following object types are available, listed in order of geometrical dimension:

- **Point objects**, of two types:
 - **Joint objects:** These are automatically created at the corners or ends of all other types of objects below, and they can be explicitly added to model supports or other localized behavior.
 - **Grounded (one-joint) support objects:** Used to model special support behavior such as isolators, dampers, gaps, multilinear springs, and more. These are not covered in this manual
- **Line objects**, of two types
 - **Frame/cable objects:** Used to model beams, columns, braces, trusses, and/or cable members
 - **Connecting (two-joint) link objects:** Used to model special member behavior such as isolators, dampers, gaps, multilinear springs, and more. Un-

like frame/cable objects, connecting link objects can have zero length. These are not covered in this manual.

- **Area objects:** Used to model walls, floors, and other thin-walled members, as well as two-dimensional solids (plane stress, plane strain, and axisymmetric solids). Only shell-type area objects are covered in this manual
- **Solid objects:** Used to model three-dimensional solids. These are not covered in this manual.

As a general rule, the geometry of the object should correspond to that of the physical member. This simplifies the visualization of the model and helps with the design process.

If you have experience using traditional finite element programs, including earlier versions of SAP2000, you are probably used to meshing physical models into smaller finite elements for analysis purposes. Object-based modeling largely eliminates the need for doing this.

For users who are new to finite-element modeling, the object-based concept should seem perfectly natural.

When you run an analysis, SAP2000 automatically converts your object-based model into an element-based model that is used for analysis. This element-based model is called the analysis model, and it consists of traditional finite elements and joints (nodes). Results of the analysis are reported back on the object-based model.

You have control over how the meshing is performed, such as the degree of refinement, and how to handle the connections between intersecting objects. You also have the option to manually mesh the model, resulting in a one-to-one correspondence between objects and elements.

In this manual, the term “element” will be used more often than “object”, since what is described herein is the finite-element analysis portion of the program that operates on the element-based analysis model. However, it should be clear that the properties described here for elements are actually assigned in the interface to the objects, and the conversion to analysis elements is automatic.

Coordinate Systems

Each structure may use many different coordinate systems to describe the location of points and the directions of loads, displacement, internal forces, and stresses. Understanding these different coordinate systems is crucial to being able to properly define the model and interpret the results.

Topics

- Overview
- Global Coordinate System
- Upward and Horizontal Directions
- Local Coordinate Systems

Overview

Coordinate systems are used to locate different parts of the structural model and to define the directions of loads, displacements, internal forces, and stresses.

All coordinate systems in the model are defined with respect to a single, global X-Y-Z coordinate system. Each part of the model (joint, element, or constraint) has its own local 1-2-3 coordinate system. In addition, you may create alternate coordi-

nate systems that are used to define locations and directions. All coordinate systems are three-dimensional, right-handed, rectangular (Cartesian) systems.

SAP2000 always assumes that Z is the vertical axis, with +Z being upward. The upward direction is used to help define local coordinate systems, although local coordinate systems themselves do not have an upward direction.

For more information and additional features, see Chapter “Coordinate Systems” in the *SAP2000, ETABS, and SAFE Analysis Reference Manual* and the Help Menu in the SAP2000 graphical user interface.

Global Coordinate System

The **global coordinate system** is a three-dimensional, right-handed, rectangular coordinate system. The three axes, denoted X, Y, and Z, are mutually perpendicular and satisfy the right-hand rule. The location and orientation of the global system are arbitrary.

Locations in the global coordinate system can be specified using the variables **x**, **y**, and **z**. A vector in the global coordinate system can be specified by giving the locations of two points, a pair of angles, or by specifying a coordinate direction. Coordinate directions are indicated using the values X, Y, and Z. For example, +X defines a vector parallel to and directed along the positive X axis. The sign is required.

All other coordinate systems in the model are defined with respect to the global coordinate system.

Upward and Horizontal Directions

SAP2000 always assumes that Z is the vertical axis, with +Z being upward. Local coordinate systems for joints, elements, and ground-acceleration loading are defined with respect to this upward direction. Self-weight loading always acts downward, in the $-Z$ direction.

The X-Y plane is horizontal. The primary horizontal direction is +X. Angles in the horizontal plane are measured from the positive half of the X axis, with positive angles appearing counter-clockwise when you are looking down at the X-Y plane.

Local Coordinate Systems

Each part (joint, element, or constraint) of the structural model has its own local coordinate system used to define the properties, loads, and response for that part. The axes of the local coordinate systems are denoted 1, 2, and 3. In general, the local coordinate systems may vary from joint to joint, element to element, and constraint to constraint.

There is no preferred upward direction for a local coordinate system. However, the joint and element local coordinate systems are defined with respect to the global upward direction, +Z.

The joint local 1-2-3 coordinate system is normally the same as the global X-Y-Z coordinate system.

For the Frame and Shell elements, one of the element local axes is determined by the geometry of the individual element. You may define the orientation of the remaining two axes by specifying a single angle of rotation.

The local coordinate system for a Diaphragm Constraint is normally determined automatically from the geometry or mass distribution of the constraint. Optionally, you may specify one global axis that determines the plane of a Diaphragm Constraint; the remaining two axes are determined automatically.

For more information:

- See Topic “Local Coordinate System” (page 14) in Chapter “The Frame Element.”
- See Topic “Local Coordinate System” (page 40) in Chapter “The Shell Element.”
- See Topic “Local Coordinate System” (page 52) in Chapter “Joints and Degrees of Freedom.”
- See Topic “Diaphragm Constraint” (page 66) in Chapter “Joint Constraints.”

The Frame Element

The **Frame** element is used to model beam-column and truss behavior in planar and three-dimensional structures. The frame element can also be used to model cable behavior when nonlinear properties are added (e.g., tension only, large deflections). Although everything described in this chapter can apply to cables, cable-specific behavior is not discussed.

Topics

- Overview
- Joint Connectivity
- Degrees of Freedom
- Local Coordinate System
- Section Properties
- Insertion Point
- End Offsets
- End Releases
- Mass
- Self-Weight Load
- Concentrated Span Load

- Distributed Span Load
- Internal Force Output

Overview

The Frame element uses a general, three-dimensional, beam-column formulation which includes the effects of biaxial bending, torsion, axial deformation, and biaxial shear deformations. See Bathe and Wilson (1976).

Structures that can be modeled with this element include:

- Three-dimensional frames
- Three-dimensional trusses
- Planar frames
- Planar grillages
- Planar trusses
- Cables

A Frame element is modeled as a straight line connecting two points. In the graphical user interface, you can divide curved objects into multiple straight objects, subject to your specification.

Each element has its own local coordinate system for defining section properties and loads, and for interpreting output.

Each Frame element may be loaded by self-weight, multiple concentrated loads, and multiple distributed loads.

Insertion points and end offsets are available to account for the finite size of beam and column intersections. End releases are also available to model different fixity conditions at the ends of the element.

Element internal forces are produced at the ends of each element and at a user-specified number of equally-spaced output stations along the length of the element.

Cable behavior is modeled using the frame element and adding the appropriate features. You can release the moments at the ends of the elements, although we recommend that you retain small, realistic bending stiffness instead. You can also add nonlinear behavior as needed, such as the no-compression property, tension stiffening (p -delta effects), and large deflections. These features require nonlinear analysis, and are not covered in this manual.

For more information and additional features, see Chapter “The Frame Element” in the *SAP2000, ETABS, and SAFE Analysis Reference Manual*.

Joint Connectivity

A Frame element is represented by a straight line connecting two joints, **i** and **j**, unless modified by joint offsets as described below. The two joints must not share the same location in space. The two ends of the element are denoted end I and end J, respectively.

By default, the neutral axis of the element runs along the line connecting the two joints. However, you can change this using the insertion point, as described in Topic “Insertion Point” (page 22).

Joint Offsets

Sometimes the axis of the element cannot be conveniently specified by joints that connect to other elements in the structure. You have the option to specify **joint offsets** independently at each end of the element. These are given as the three distance components (X, Y, and Z) parallel to the global axes, measured from the joint to the end of the element (at the insertion point.)

The two locations given by the coordinates of joints **i** and **j**, plus the corresponding joint offsets, define the axis of the element. These two locations must not be coincident. It is generally recommended that the offsets be perpendicular to the axis of the element, although this is not required.

Offsets along the axis of the element are usually specified using end offsets rather than joint offsets. See topic “End Offsets” (page 24). End offsets are part of the length of the element, have element properties and loads, and may or may not be rigid. Joint offsets are external to the element, and do not have any mass or loads. Internally the program creates a fully rigid constraint along the joints offsets.

Joint offsets are specified along with the cardinal point as part of the insertion point assignment, even though they are independent features.

For more information:

- See Topic “Insertion Point” (page 22) in this chapter.
- See Topic “End Offsets” (page 24) in this chapter.

Degrees of Freedom

The Frame element activates all six degrees of freedom at both of its connected joints. If you want to model truss or cable elements that do not transmit moments at the ends, you may either:

- Set the geometric Section properties **j**, **i33**, and **i22** all to zero (**a** is non-zero; **as2** and **as3** are arbitrary), or
- Release both bending rotations, R2 and R3, at both ends and release the torsional rotation, R1, at either end

For more information:

- See Topic “Degrees of Freedom” (page 52) in Chapter “Joints and Degrees of Freedom.”
- See Topic “Section Properties” (page 17) in this chapter.
- See Topic “End Releases” (page 25) in this chapter.

Local Coordinate System

Each Frame element has its own **element local coordinate system** used to define section properties, loads and output. The axes of this local system are denoted 1, 2 and 3. The first axis is directed along the length of the element; the remaining two axes lie in the plane perpendicular to the element with an orientation that you specify.

It is important that you clearly understand the definition of the element local 1-2-3 coordinate system and its relationship to the global X-Y-Z coordinate system. Both systems are right-handed coordinate systems. It is up to you to define local systems which simplify data input and interpretation of results.

In most structures the definition of the element local coordinate system is extremely simple using the **default orientation** and the **Frame element coordinate angle**. Additional methods are available.

For more information:

- See Chapter “Coordinate Systems” (page 7) for a description of the concepts and terminology used in this topic.
- See Topic “Advanced Local Coordinate System” in Chapter “The Frame Element” in the *SAP2000, ETABS, and SAFE Analysis Reference Manual*.

- See Topic “Joint Offsets” (page 13) in this chapter.

Longitudinal Axis I

Local axis 1 is always the longitudinal axis of the element, the positive direction being directed from end I to end J.

Specifically, end I is joint **i** plus its joint offsets (if any), and end J is joint **j** plus its joint offsets (if any.) The axis is determined independently of the cardinal point; see Topic “Insertion Point” (page 22.)

Default Orientation

The default orientation of the local 2 and 3 axes is determined by the relationship between the local 1 axis and the global Z axis:

- The local 1-2 plane is taken to be vertical, i.e., parallel to the Z axis
- The local 2 axis is taken to have an upward (+Z) sense unless the element is vertical, in which case the local 2 axis is taken to be horizontal along the global +X direction
- The local 3 axis is always horizontal, i.e., it lies in the X-Y plane

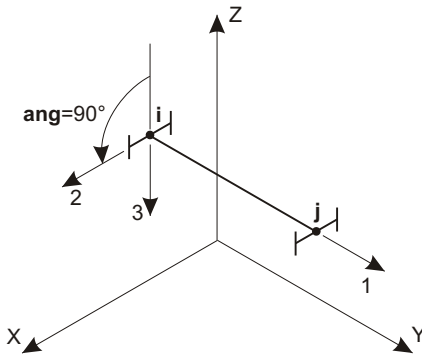
An element is considered to be vertical if the sine of the angle between the local 1 axis and the Z axis is less than 10^{-3} .

The local 2 axis makes the same angle with the vertical axis as the local 1 axis makes with the horizontal plane. This means that the local 2 axis points vertically upward for horizontal elements.

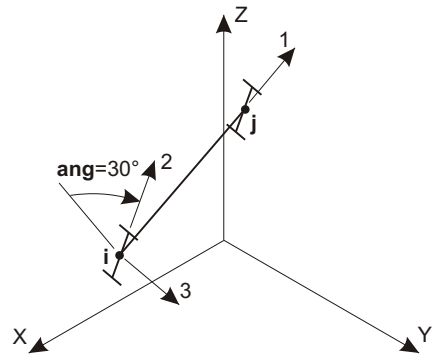
Coordinate Angle

The Frame element coordinate angle, **ang**, is used to define element orientations that are different from the default orientation. It is the angle through which the local 2 and 3 axes are rotated about the positive local 1 axis from the default orientation. The rotation for a positive value of **ang** appears counter-clockwise when the local +1 axis is pointing toward you.

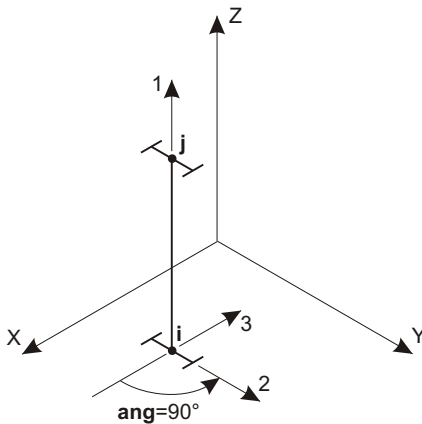
For vertical elements, **ang** is the angle between the local 2 axis and the horizontal +X axis. Otherwise, **ang** is the angle between the local 2 axis and the vertical plane containing the local 1 axis. See Figure 1 (page 16) for examples.



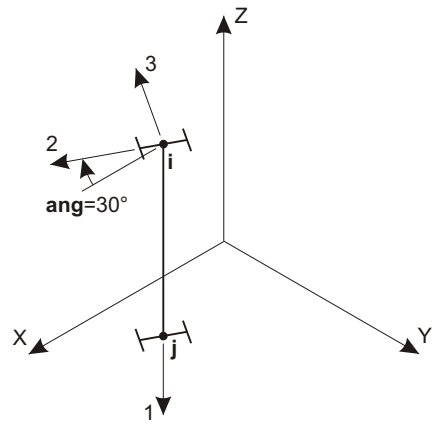
Local 1 Axis is Parallel to +Y Axis
Local 2 Axis is Rotated 90° from Z-1 Plane



Local 1 Axis is Not Parallel to X, Y, or Z Axes
Local 2 Axis is Rotated 30° from Z-1 Plane



Local 1 Axis is Parallel to +Z Axis
Local 2 Axis is Rotated 90° from X-1 Plane



Local 1 Axis is Parallel to -Z Axis
Local 2 Axis is Rotated 30° from X-1 Plane

Figure 1

The Frame Element Coordinate Angle with Respect to the Default Orientation

Section Properties

A **Frame Section** is a set of material and geometric properties that describe the cross-section of one or more Frame elements. Sections are defined independently of the Frame elements, and are assigned to the elements.

Local Coordinate System

Section properties are defined with respect to the local coordinate system of a Frame element as follows:

- The 1 direction is along the axis of the element. It is normal to the Section and goes through the intersection of the two neutral axes of the Section.
- The 2 and 3 directions define the plane of the Section. Usually the 2 direction is taken along the major dimension (depth) of the Section, and the 3 direction along its minor dimension (width), but this is not required.

See Topic “Local Coordinate System” (page 14) in this chapter for more information.

Material Properties

The material properties for the Section are specified by reference to a previously-defined Material. The material properties used by the Section are:

- The modulus of elasticity, **e1**, for axial stiffness and bending stiffness;
- The shear modulus, **g12**, for torsional stiffness and transverse shear stiffness; this is computed from **e1** and the Poisson's ratio, **u12**
- The mass density (per unit of volume), **m**, for computing element mass;
- The weight density (per unit of volume), **w**, for computing Self-Weight Load.
- The design-type indicator, **ides**, that indicates whether elements using this Section should be designed as steel, concrete, aluminum, cold-formed steel, or none (no design).

Geometric Properties and Section Stiffnesses

Six basic geometric properties are used, together with the material properties, to generate the stiffnesses of the Section. These are:

- The cross-sectional area, **a**. The axial stiffness of the Section is given by **a e1**;

- The moment of inertia, **i33**, about the 3 axis for bending in the 1-2 plane, and the moment of inertia, **i22**, about the 2 axis for bending in the 1-3 plane. The corresponding bending stiffnesses of the Section are given by **i33 e1** and **i22 e1**;
- The torsional constant, **j**. The torsional stiffness of the Section is given by **j g12**. *Note that the torsional constant is not the same as the polar moment of inertia, except for circular shapes.* See Roark and Young (1975) or Cook and Young (1985) for more information.
- The shear areas, **as2** and **as3**, for transverse shear in the 1-2 and 1-3 planes, respectively. The corresponding transverse shear stiffnesses of the Section are given by **as2 g12** and **as3 g12**. Formulae for calculating the shear areas of typical sections are given in Figure 2 (page 19).

Setting **a**, **j**, **i33**, or **i22** to zero causes the corresponding section stiffness to be zero. For example, a truss member can be modeled by setting **j = i33 = i22 = 0**, and a planar frame member in the 1-2 plane can be modeled by setting **j = i22 = 0**.

Setting **as2** or **as3** to zero causes the corresponding transverse shear *deformation* to be zero. In effect, a zero shear area is interpreted as being infinite. The transverse shear stiffness is ignored if the corresponding bending stiffness is zero.

Shape Type

For each Section, the six geometric properties (**a**, **j**, **i33**, **i22**, **as2** and **as3**) may be specified directly, computed from specified Section dimensions, or read from a specified property database file. This is determined by the shape type, **sh**, specified by the user:

- If **sh=G** (general section), the six geometric properties must be explicitly specified
- If **sh=R, P, B, I, C, T, L, or 2L**, the six geometric properties are automatically calculated from specified Section dimensions as described in “Automatic Section Property Calculation” below.
- If **sh** is any other value (e.g., W27X94 or 2L4X3X1/4), the six geometric properties are obtained from a specified property database file. See “Section Property Database Files” below.

The six geometric Section properties can be automatically calculated from specified dimensions for the simple shapes shown in Figure 3 (page 21). The required dimensions for each shape are shown in the figure.

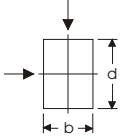
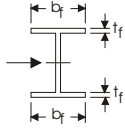
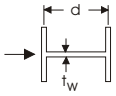
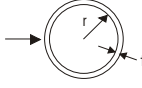
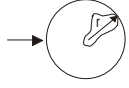
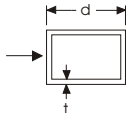
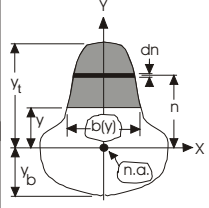
Section	Description	Effective Shear Area
	Rectangular Section Shear Forces parallel to the b or d directions	$\frac{5}{6} bd$
	Wide Flange Section Shear Forces parallel to flange	$\frac{5}{3} t_f b_f$
	Wide Flange Section Shear Forces parallel to web	$t_w d$
	Thin Walled Circular Tube Section Shear Forces from any direction	$\pi r t$
	Solid Circular Section Shear Forces from any direction	$0.9 \pi r^2$
	Thin Walled Rectangular Tube Section Shear Forces parallel to d-direction	$2 t d$
	General Section Shear Forces parallel to Y-direction $I_x =$ moment of inertia of section about X-X $Q(Y) = \int_y^{y_t} n b(n) dn$	$\frac{I_x^2}{\int_{y_b}^{y_t} \frac{Q^2(y)}{b(y)} dy}$

Figure 2
Shear Area Formulae

Note that the dimension **t3** is the depth of the Section in the 2 direction and contributes primarily to **i33**.

Automatic Section property calculation is available for the following shape types:

- **sh=R**: Rectangular Section
- **sh=P**: Pipe Section, or Solid Circular Section if **tw=0** (or not specified)
- **sh=B**: Box Section
- **sh=I**: I-Section
- **sh=C**: Channel Section
- **sh=T**: T-Section
- **sh=L**: Angle Section
- **sh=2L**: Double-angle Section

Section Property Database Files

Geometric Section properties may be obtained from one or more Section property database files. Several database files are supplied with SAP2000, including:

- **AISC.pro** and **AISC3.pro**: American Institute of Steel Construction shapes
- **AA6061-T6.pro**: Aluminum Association shapes
- **CISC.pro**: Canadian Institute of Steel Construction shapes
- **SECTIONS8.pro**: This is just a copy of **AISC3.pro**.

Additional files are provided containing standard shapes for other countries.

You may create your own property database files using the program **PROPER**, which is available upon request from Computers and Structures, Inc.

The geometric properties are stored in the length units specified when the database file was created. These are automatically converted to the units being used by SAP2000.

Each shape type stored in a database file may be referenced by one or two different labels. For example, the W 36x300 shape type in file **AISC.PRO** may be referenced either by label “W36X300” or by label “W920X446”. Shape types stored in **CISC.PRO** may only be referenced by a single label.

You may select one database file to be used when defining a given Frame Section. The database file in use can be changed at any time when defining Sections. If no database filename is specified, the default file **SECTIONS8.PRO** is used. You may copy any property database file to **SECTIONS8.PRO**.

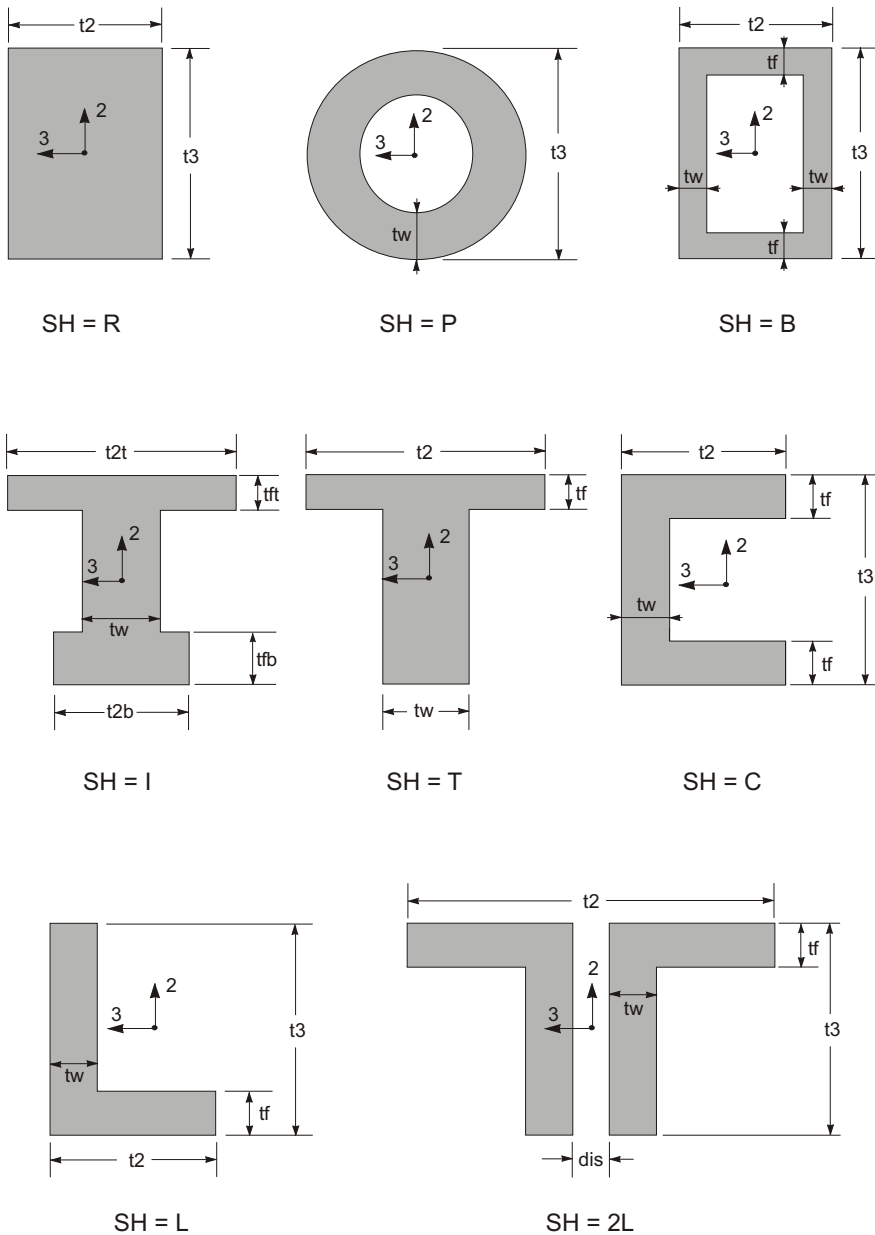
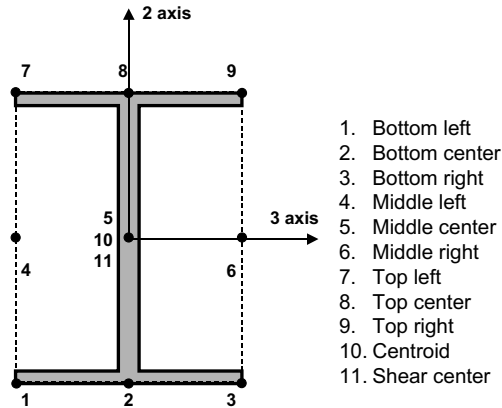


Figure 3
Automatic Section Property Calculation

All Section property database files, including file SECTIONS8.PRO, must be located either in the directory that contains the data file, or in the directory that con-



1. Bottom left
2. Bottom center
3. Bottom right
4. Middle left
5. Middle center
6. Middle right
7. Top left
8. Top center
9. Top right
10. Centroid
11. Shear center

Note: For doubly symmetric members such as this one, cardinal points 5, 10, and 11 are the same.

Figure 4
Frame Cardinal Points

tains the SAP2000 program files. If a specified database file is present in both directories, the program will use the file in the data-file directory.

Insertion Point

By default the local 1 axis of the element runs along the neutral axis of the section, i.e., at the centroid of the section. It is often convenient to specify another location on the section, such as the top of a beam or an outside corner of a column. This location is called the **cardinal point** of the section.

The available cardinal point choices are shown in Figure 4 (page 22). The default location is point 10.

Joint offsets are specified along with the cardinal point as part of the insertion point assignment, even though they are independent features. Joint offsets are used first to calculate the element axis and therefore the local coordinate system, then the cardinal point is located in the resulting local 2-3 plane.

This feature is useful, as an example, for modeling beams and columns when the beams do not frame into the center of the column. Figure 5 (page 22) shows an elevation and plan view of a common framing arrangement where the exterior beams

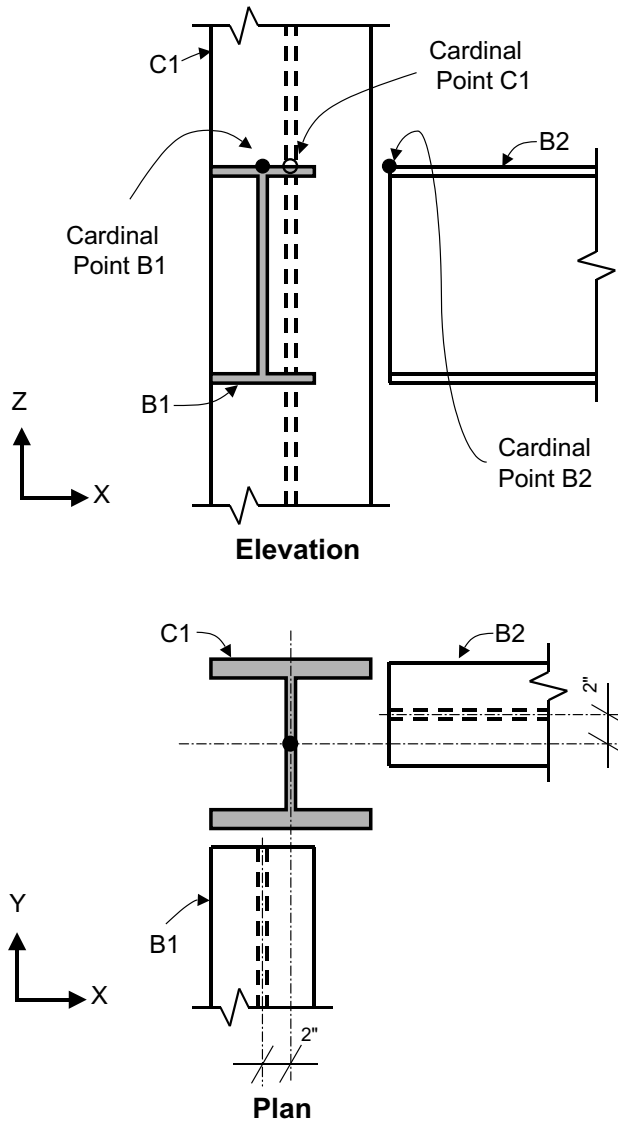


Figure 5
Example Showing Joint Offsets and Cardinal Points

are offset from the column center lines to be flush with the exterior of the building. Also shown in this figure are the cardinal points for each member and the joint offset dimensions.

End Offsets

Frame elements are modeled as line elements connected at points (joints). However, actual structural members have finite cross-sectional dimensions. When two elements, such as a beam and column, are connected at a joint there is some overlap of the cross sections. In many structures the dimensions of the members are large and the length of the overlap can be a significant fraction of the total length of a connecting element.

You may specify two **end offsets** for each element using parameters **ioff** and **joff** corresponding to ends I and J, respectively. End offset **ioff** is the length of overlap for a given element with other connecting elements at joint **i**. It is the distance from the joint to the face of the connection for the given element. A similar definition applies to end offset **joff** at joint **j**. See Figure 6 (page 25).

End offsets can be automatically calculated by the SAP2000 graphical user interface for selected elements based on the maximum Section dimensions of all other elements that connect to that element at a common joint.

Clear Length

The **clear length**, denoted L_c , is defined to be the length between the end offsets (support faces) as:

$$L_c = L - (\text{ioff} + \text{joff})$$

where L is the total element length. See Figure 6 (page 25).

If end offsets are specified such that the clear length is less than 1% of the total element length, the program will issue a warning and reduce the end offsets proportionately so that the clear length is equal to 1% of the total length. Normally the end offsets should be a much smaller proportion of the total length.

Effect upon Internal Force Output

All internal forces and moments are output at the faces of the supports and at other equally-spaced points within the clear length of the element. No output is produced within the end offset, which includes the joint.

See Topic “Internal Force Output” (page 32) in this chapter for more information.

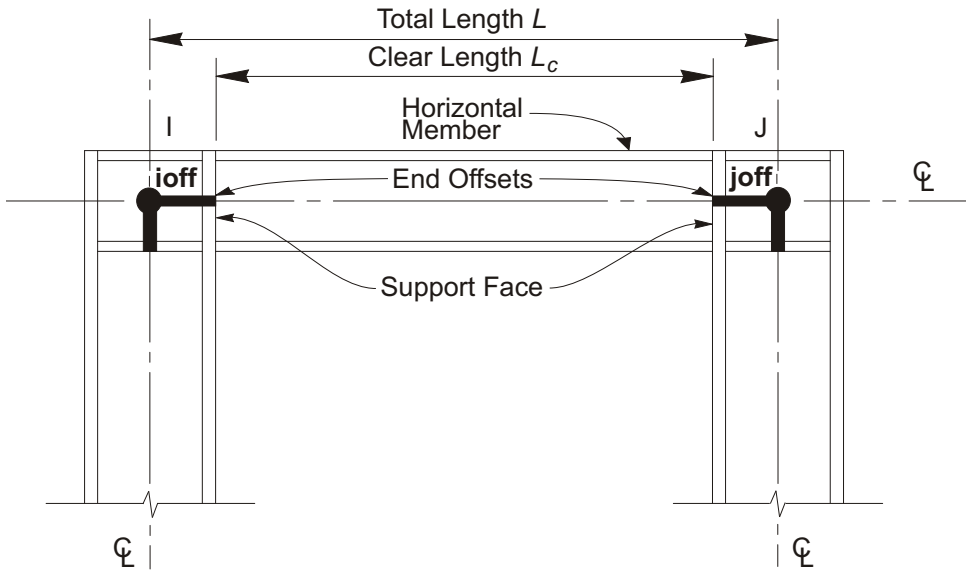


Figure 6
Frame Element End Offsets

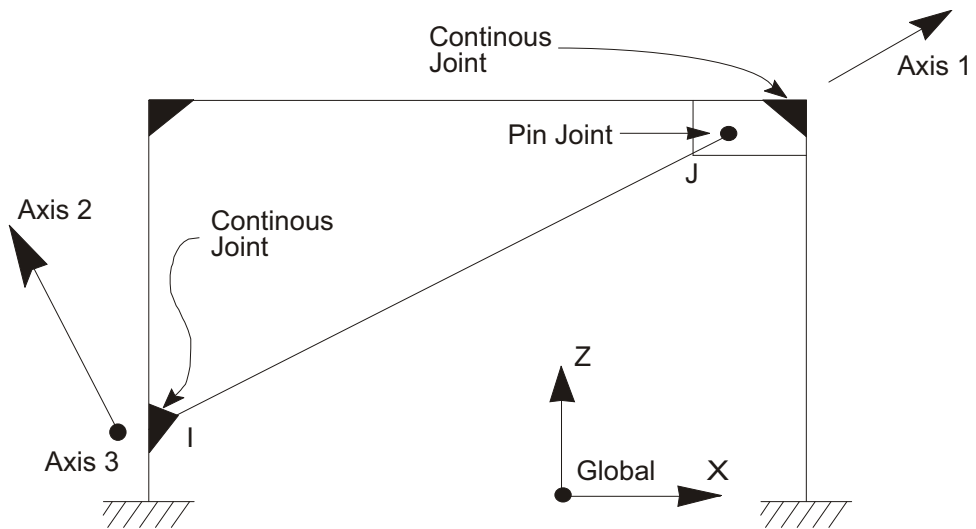
Effect upon End Releases

End releases are always assumed to be at the support faces, i.e., at the ends of the clear length of the element. If a moment or shear release is specified in either bending plane at either end of the element, the end offset is assumed to be rigid for bending and shear *in that plane at that end*.

See Topic “End Releases” (page 25) in this chapter for more information.

End Releases

Normally, the three translational and three rotational degrees of freedom at each end of the Frame element are continuous with those of the joint, and hence with those of all other elements connected to that joint. However, it is possible to release (disconnect) one or more of the element degrees of freedom from the joint when it is known that the corresponding element force or moment is zero. The releases are always specified in the element local coordinate system, and do not affect any other element connected to the joint.



For diagonal element: R3 is released at end J

Figure 7
Frame Element End Releases

In the example shown in Figure 7 (page 26), the diagonal element has a moment connection at End I and a pin connection at End J. The other two elements connecting to the joint at End J are continuous. Therefore, in order to model the pin condition the rotation R3 at End J of the diagonal element should be released. This assures that the moment is zero at the pin in the diagonal element.

Unstable End Releases

Any combination of end releases may be specified for a Frame element provided that the element remains stable; this assures that all load applied to the element is transferred to the rest of the structure. The following sets of releases are unstable, either alone or in combination, and are not permitted:

- Releasing U1 at both ends
- Releasing U2 at both ends
- Releasing U3 at both ends
- Releasing R1 at both ends
- Releasing R2 at both ends *and* U3 at either end

- Releasing R3 at both ends *and* U2 at either end

Effect of End Offsets

End releases are always applied at the support faces, i.e., at the ends of the element clear length. The presence of a moment or shear release will cause the end offset to be rigid in the corresponding bending plane at the corresponding end of the element.

See Topic “End Offsets” (page 24) in this chapter for more information.

Mass

In a dynamic analysis, the mass of the structure is used to compute inertial forces. The mass contributed by the Frame element is lumped at the joints **i** and **j**. *No inertial effects are considered within the element itself.*

The total mass of the element is equal to the integral along the length of the mass density, **m**, multiplied by the cross-sectional area, **a**.

The total mass is apportioned to the two joints in the same way a similarly-distributed transverse load would cause reactions at the ends of a simply-supported beam. The effects of end releases are ignored when apportioning mass. The total mass is applied to each of the three translational degrees of freedom: UX, UY, and UZ. No mass moments of inertia are computed for the rotational degrees of freedom.

For more information:

- See Topic “Section Properties” (page 17) in this chapter for the definitions of **m** and **a**.
- See Chapter “Static and Dynamic Analysis” (page 69).

Self-Weight Load

Self-Weight Load can be applied in any Load Case to activate the self-weight of all elements in the model. For a Frame element, the self-weight is a force that is distributed along the length of the element. The magnitude of the self-weight is equal to the weight density, **w**, multiplied by the cross-sectional area, **a**.

Self-weight always acts downward, in the global $-Z$ direction. The self-weight may be scaled by a single factor that applies to the whole structure.

For more information:

- See Topic “Section Properties” (page 17) in this chapter for the definitions of \mathbf{w} and \mathbf{a} .
- See Chapter “Static and Dynamic Analysis” (page 69).

Concentrated Span Load

The Concentrated Span Load is used to apply concentrated forces and moments at arbitrary locations on Frame elements. The direction of loading may be specified in the global coordinate system or in the element local coordinate system.

The location of the load may be specified in one of the following ways:

- Specifying a relative distance, \mathbf{rd} , measured from joint \mathbf{i} . This must satisfy $0 \leq \mathbf{rd} \leq 1$. The relative distance is the fraction of element length;
- Specifying an absolute distance, \mathbf{d} , measured from joint \mathbf{i} . This must satisfy $0 \leq \mathbf{d} \leq L$, where L is the element length.

Any number of concentrated loads may be applied to each element. Loads given in global coordinates are transformed to the element local coordinate system. See Figure 8 (page 29). Multiple loads that are applied at the same location are added together.

See Chapter “Static and Dynamic Analysis” (page 69) for more information.

Distributed Span Load

The Distributed Span Load is used to apply distributed forces and moments on Frame elements. The load intensity may be uniform or trapezoidal. The direction of loading may be specified in the global coordinate system or in the element local coordinate system.

See Chapter “Static and Dynamic Analysis” (page 69) for more information.

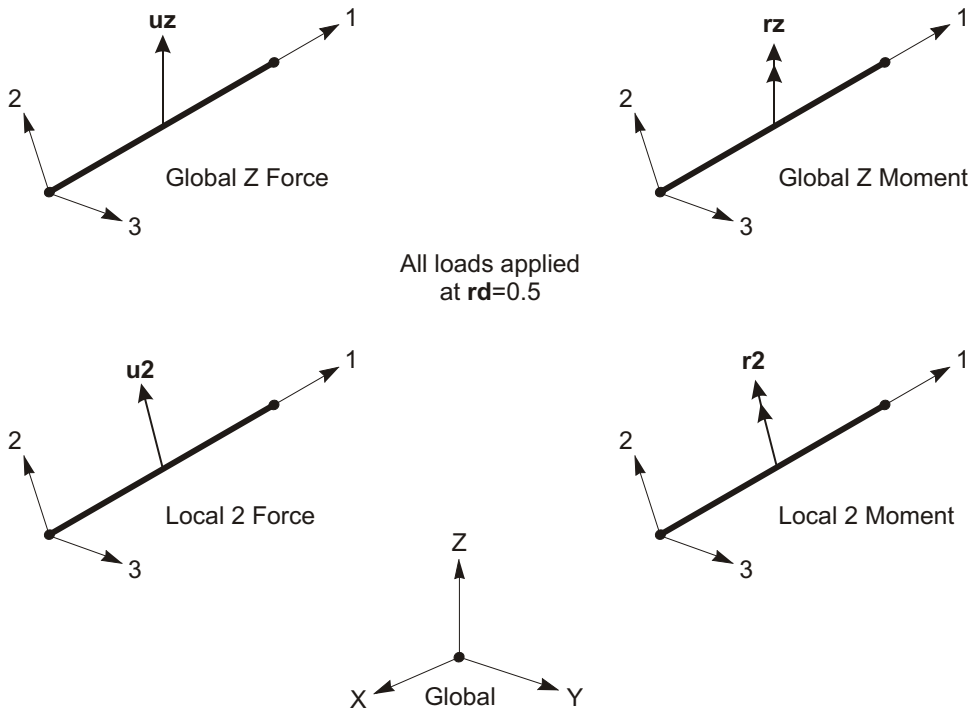


Figure 8
Examples of the Definition of Concentrated Span Loads

Loaded Length

Loads may apply to full or partial element lengths. Multiple loads may be applied to a single element. The loaded lengths may overlap, in which case the applied loads are additive.

A loaded length may be specified in one of the following ways:

- Specifying two relative distances, **rda** and **rdb**, measured from joint **i**. They must satisfy $0 \leq \mathbf{rda} \leq \mathbf{rdb} \leq 1$. The relative distance is the fraction of element length;
- Specifying two absolute distances, **da** and **db**, measured from joint **i**. They must satisfy $0 \leq \mathbf{da} \leq \mathbf{db} \leq L$, where L is the element length;
- Specifying no distances, which indicates the full length of the element.

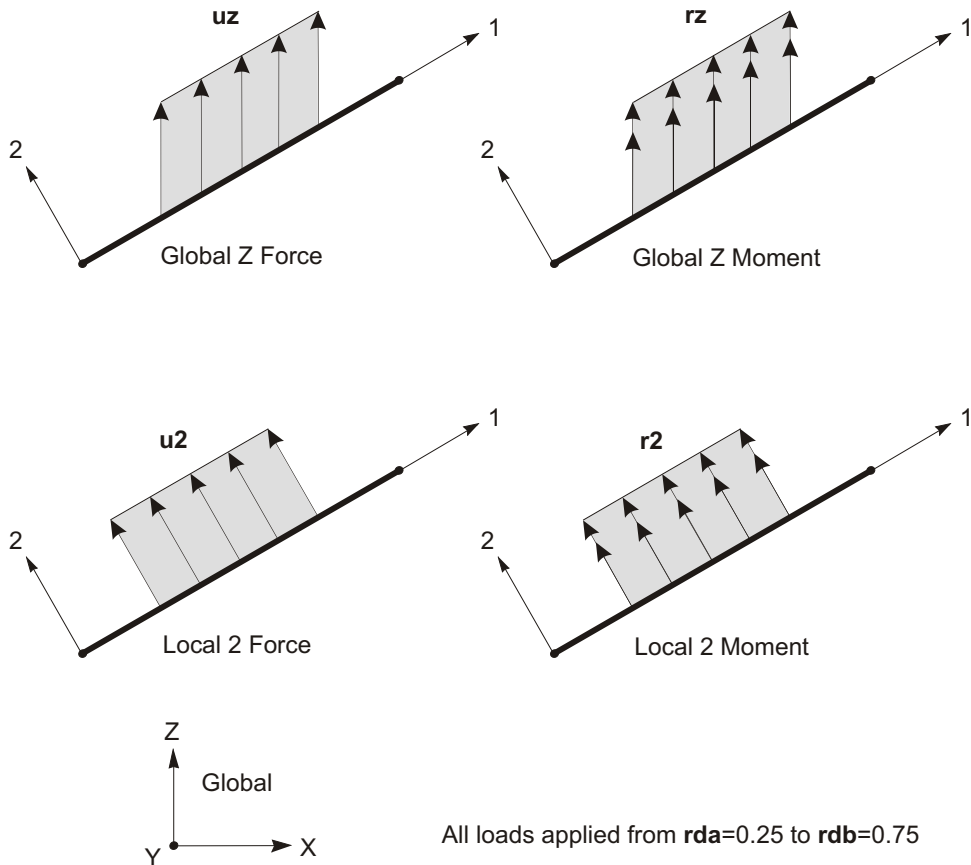


Figure 9
Examples of the Definition of Distributed Span Loads

Load Intensity

The load intensity is a force or moment per unit of length. For each force or moment component to be applied, a single load value may be given if the load is uniformly distributed. Two load values are needed if the load intensity varies linearly over its range of application (a trapezoidal load).

See Figure 9 (page 30) and Figure 10 (page 31).

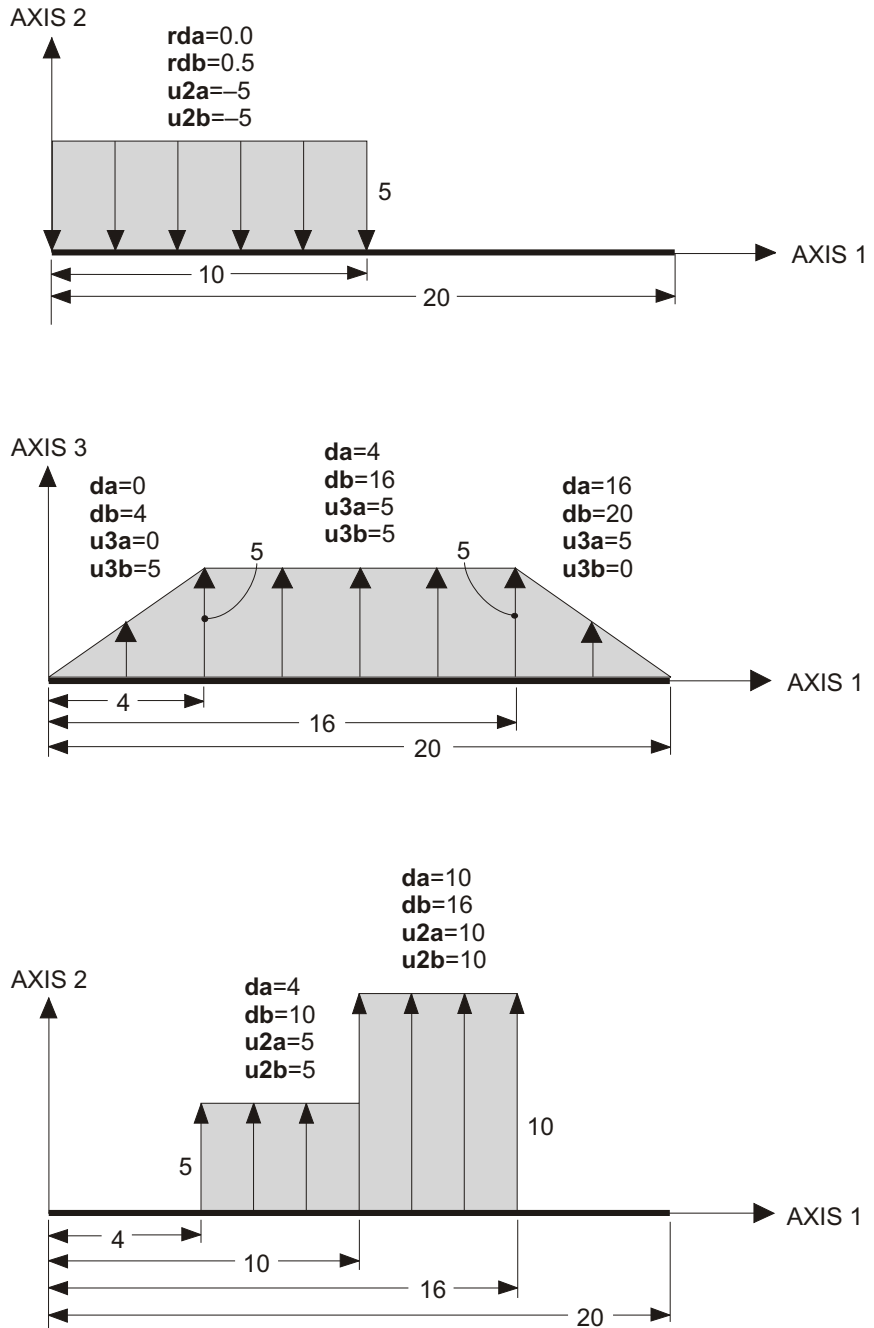


Figure 10
Examples of Distributed Span Loads

Internal Force Output

The **Frame element internal forces** are the forces and moments that result from integrating the stresses over an element cross section. These internal forces are:

- P, the axial force
- V2, the shear force in the 1-2 plane
- V3, the shear force in the 1-3 plane
- T, the axial torque
- M2, the bending moment in the 1-3 plane (about the 2 axis)
- M3, the bending moment in the 1-2 plane (about the 3 axis)

These internal forces and moments are present at every cross section along the length of the element.

The sign convention is illustrated in Figure 11 (page 33). Positive internal forces and axial torque acting on a positive 1 face are oriented in the positive direction of the element local coordinate axes. Positive internal forces and axial torque acting on a negative face are oriented in the negative direction of the element local coordinate axes. A positive 1 face is one whose outward normal (pointing away from element) is in the positive local 1 direction.

Positive bending moments cause compression at the positive 2 and 3 faces and tension at the negative 2 and 3 faces. The positive 2 and 3 faces are those faces in the positive local 2 and 3 directions, respectively, from the neutral axis.

The internal forces and moments are computed at equally-spaced output points along the length of the element. The **nseg** parameter specifies the number of equal segments (or spaces) along the length of the element between the output points. For the default value of “2”, output is produced at the two ends and at the midpoint of the element. See “Effect of End Offsets” below.

The Frame element internal forces are computed for all Analysis Cases: Loads, Modes, and Specs.

It is important to note that the Response Spectrum results are always positive, and that the correspondence between different values has been lost.

See Chapter “Static and Dynamic Analysis” (page 69) for more information.

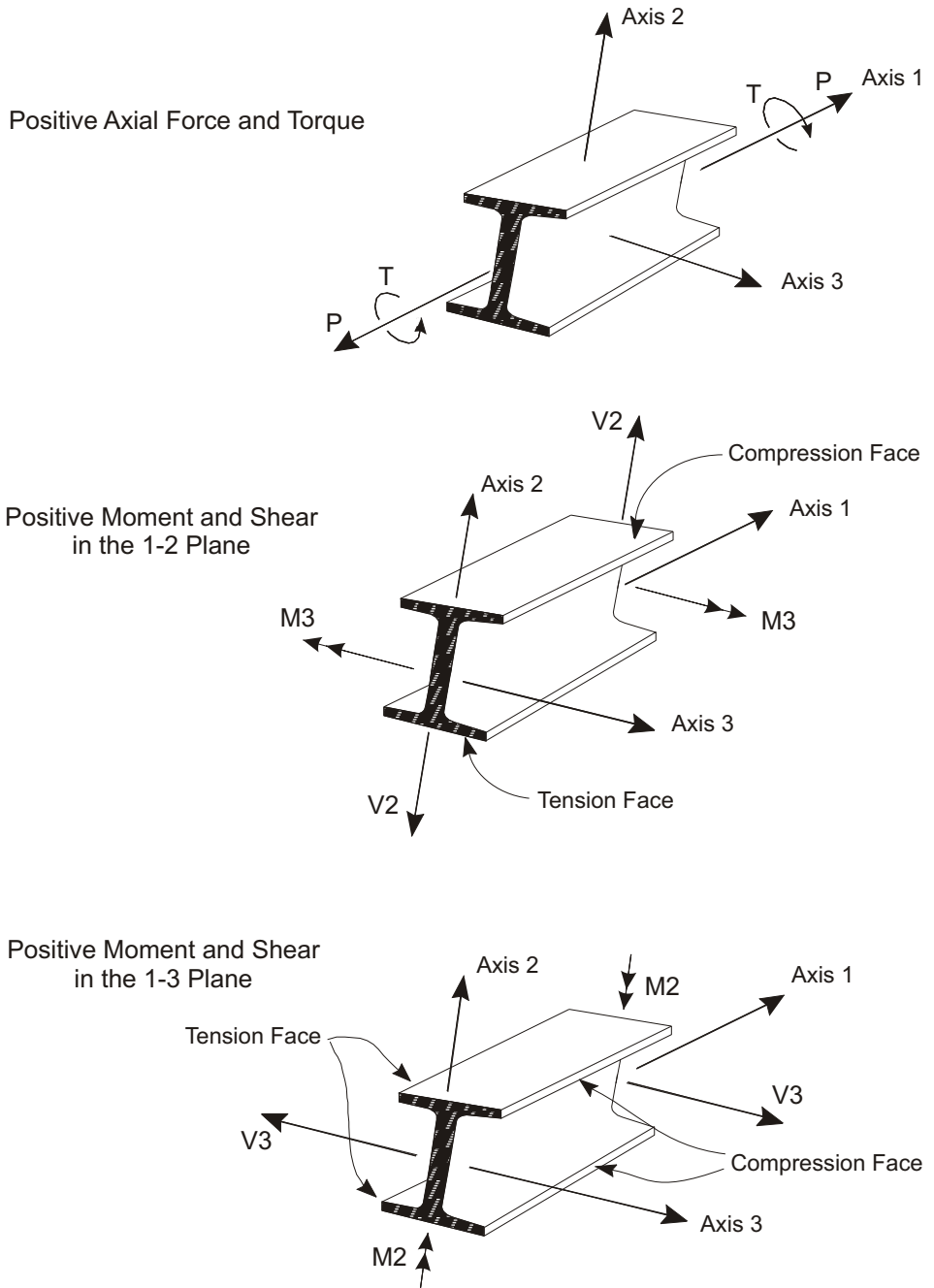


Figure 11
Frame Element Internal Forces and Moments

Effect of End Offsets

When end offsets are present, internal forces and moments are output at the faces of the supports and at **nseg** - 1 equally-spaced points within the clear length of the element. No output is produced within the length of the end offset, which includes the joint. Output will only be produced at joints **i** or **j** when the corresponding end offset is zero.

See Topic “End Offsets” (page 24) in this chapter for more information.

The Shell Element

The **Shell** element is used to model shell, membrane, and plate behavior in planar and three-dimensional structures. The shell element/object is one type of area object. Depending on the type of section properties you assign to an area, the object could also be used to model plane stress/strain and axisymmetric solid behavior; these types of behavior are not considered in this manual.

Topics

- Overview
- Joint Connectivity
- Degrees of Freedom
- Local Coordinate System
- Section Properties
- Mass
- Self-Weight Load
- Uniform Load
- Internal Force and Stress Output

Overview

The Shell element is a three- or four-node formulation that combines separate membrane and plate-bending behavior. The four-joint element does not have to be planar.

The membrane behavior uses an isoparametric formulation that includes translational in-plane stiffness components and a rotational stiffness component in the direction normal to the plane of the element. See Taylor and Simo (1985) and Ibrahimbegovic and Wilson (1991).

The plate bending behavior includes two-way, out-of-plane, plate rotational stiffness components and a translational stiffness component in the direction normal to the plane of the element. By default, a thin-plate (Kirchhoff) formulation is used that neglects transverse shearing deformation. Optionally, you may choose a thick-plate (Mindlin/Reissner) formulation which includes the effects of transverse shearing deformation.

Structures that can be modeled with this element include:

- Three-dimensional shells, such as tanks and domes
- Plate structures, such as floor slabs
- Membrane structures, such as shear walls

For each Shell element in the structure, you can choose to model pure membrane, pure plate, or full shell behavior. It is generally recommended that you use the full shell behavior unless the entire structure is planar and is adequately restrained.

Each Shell element has its own local coordinate system for defining Material properties and loads, and for interpreting output. Each element may be loaded by gravity or uniform load in any direction.

A variable, four-to-eight-point numerical integration formulation is used for the Shell stiffness. Stresses and internal forces and moments, in the element local coordinate system, are evaluated at the 2-by-2 Gauss integration points and extrapolated to the joints of the element. An approximate error in the element stresses or internal forces can be estimated from the difference in values calculated from different elements attached to a common joint. This will give an indication of the accuracy of a given finite-element approximation and can then be used as the basis for the selection of a new and more accurate finite element mesh.

For more information and additional features, see Chapter “The Shell Element” in the *SAP2000, ETABS, and SAFE Analysis Reference Manual*.

Joint Connectivity

Each Shell element may have either of the following shapes, as shown in Figure 12 (page 38):

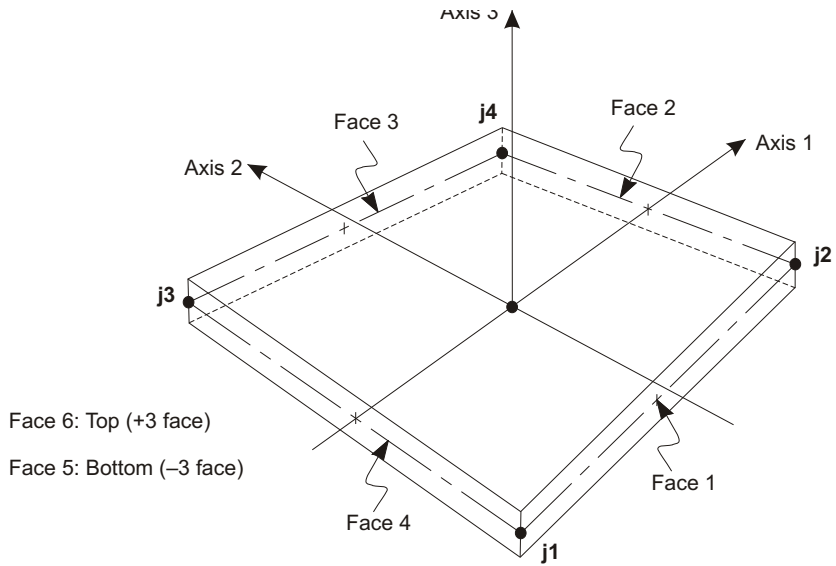
- Quadrilateral, defined by the four joints **j1**, **j2**, **j3**, and **j4**.
- Triangular, defined by the three joints **j1**, **j2**, and **j3**.

The quadrilateral formulation is the more accurate of the two. The triangular element is recommended for transitions only. The stiffness formulation of the three-node element is reasonable; however, its stress recovery is poor. The use of the quadrilateral element for meshing various geometries and transitions is illustrated in Figure 13 (page 39).

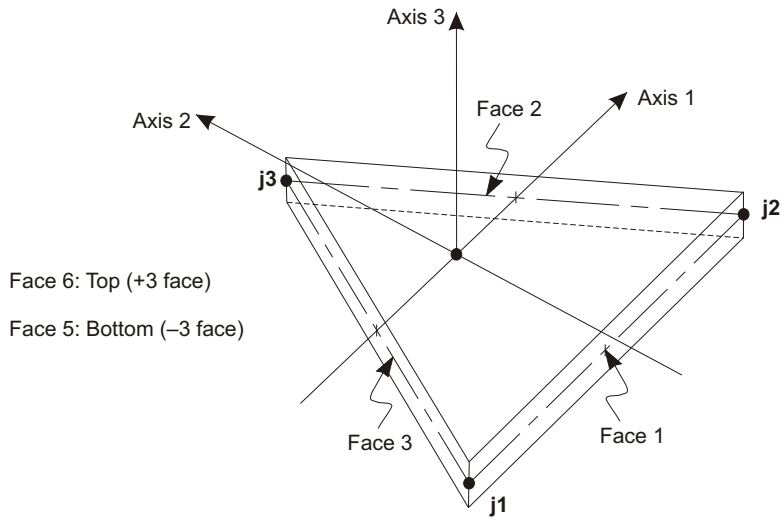
The locations of the joints should be chosen to meet the following geometric conditions:

- The inside angle at each corner must be less than 180° . Best results for the quadrilateral will be obtained when these angles are near 90° , or at least in the range of 45° to 135° .
- The aspect ratio of an element should not be too large. For the triangle, this is the ratio of the longest side to the shortest side. For the quadrilateral, this is the ratio of the longer distance between the midpoints of opposite sides to the shorter such distance. Best results are obtained for aspect ratios near unity, or at least less than four. The aspect ratio should not exceed ten.
- For the quadrilateral, the four joints need not be coplanar. A small amount of twist in the element is accounted for by the program. The angle between the normals at the corners gives a measure of the degree of twist. The normal at a corner is perpendicular to the two sides that meet at the corner. Best results are obtained if the largest angle between any pair of corners is less than 30° . This angle should not exceed 45° .

These conditions can usually be met with adequate mesh refinement.

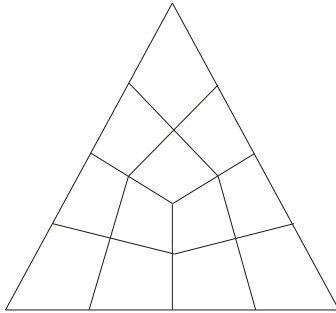


Four-node Quadrilateral Shell Element

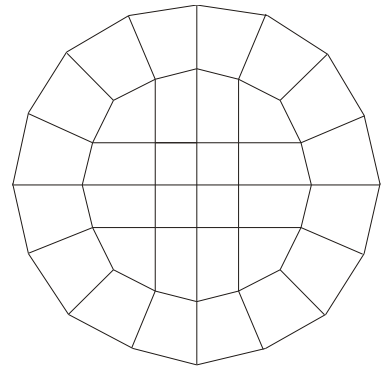


Three-node Triangular Shell Element

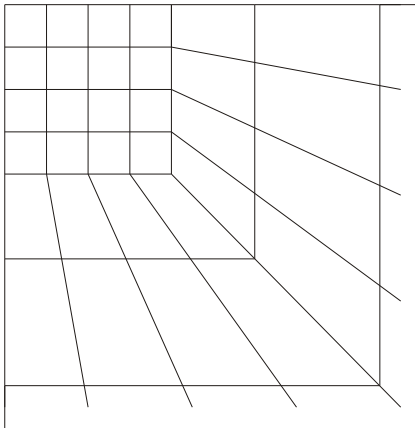
Figure 12
Shell Element Joint Connectivity and Face Definitions



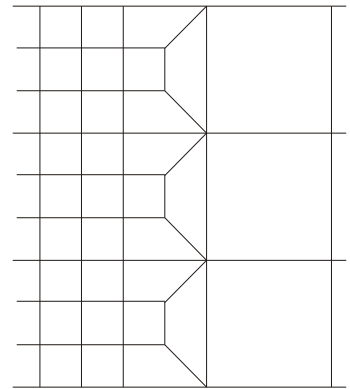
Triangular Region



Circular Region



Infinite Region



Mesh Transition

Figure 13
Mesh Examples Using the Quadrilateral Shell Element

Degrees of Freedom

The Shell element always activates all six degrees of freedom at each of its connected joints. When the element is used as a pure membrane, you must ensure that restraints or other supports are provided to the degrees of freedom for normal translation and bending rotations. When the element is used as a pure plate, you must ensure that restraints or other supports are provided to the degrees of freedom for in-plane translations and the rotation about the normal.

The use of the full shell behavior (membrane plus plate) is recommended for all three-dimensional structures.

See Topic “Degrees of Freedom” (page 52) in Chapter “Joints and Degrees of Freedom” for more information.

Local Coordinate System

Each Shell element has its own **element local coordinate system** used to define Material properties, loads and output. The axes of this local system are denoted 1, 2 and 3. The first two axes lie in the plane of the element with an orientation that you specify; the third axis is normal.

It is important that you clearly understand the definition of the element local 1-2-3 coordinate system and its relationship to the global X-Y-Z coordinate system. Both systems are right-handed coordinate systems. It is up to you to define local systems which simplify data input and interpretation of results.

In most structures the definition of the element local coordinate system is extremely simple using the **default orientation** and the **Shell element coordinate angle**. Additional methods are available.

For more information:

- See Chapter “Coordinate Systems” (page 7) for a description of the concepts and terminology used in this topic.
- See Topic “Advanced Local Coordinate System” in Chapter “The Shell Element” in the *SAP2000, ETABS, and SAFE Analysis Reference Manual*.

Normal Axis 3

Local axis 3 is always normal to the plane of the Shell element. This axis is directed toward you when the path **j1-j2-j3** appears counter-clockwise. For quadrilateral elements, the element plane is defined by the vectors that connect the midpoints of the two pairs of opposite sides.

Default Orientation

The default orientation of the local 1 and 2 axes is determined by the relationship between the local 3 axis and the global Z axis:

- The local 3-2 plane is taken to be vertical, i.e., parallel to the Z axis
- The local 2 axis is taken to have an upward (+Z) sense unless the element is horizontal, in which case the local 2 axis is taken to be horizontal along the global +Y direction
- The local 1 axis is always horizontal, i.e., it lies in the X-Y plane

The element is considered to be horizontal if the sine of the angle between the local 3 axis and the Z axis is less than 10^{-3} .

The local 2 axis makes the same angle with the vertical axis as the local 3 axis makes with the horizontal plane. This means that the local 2 axis points vertically upward for vertical elements.

Coordinate Angle

The Shell element coordinate angle, **ang**, is used to define element orientations that are different from the default orientation. It is the angle through which the local 1 and 2 axes are rotated about the positive local 3 axis from the default orientation. The rotation for a positive value of **ang** appears counter-clockwise when the local +3 axis is pointing toward you.

For horizontal elements, **ang** is the angle between the local 2 axis and the horizontal +Y axis. Otherwise, **ang** is the angle between the local 2 axis and the vertical plane containing the local 3 axis. See Figure 14 (page 42) for examples.

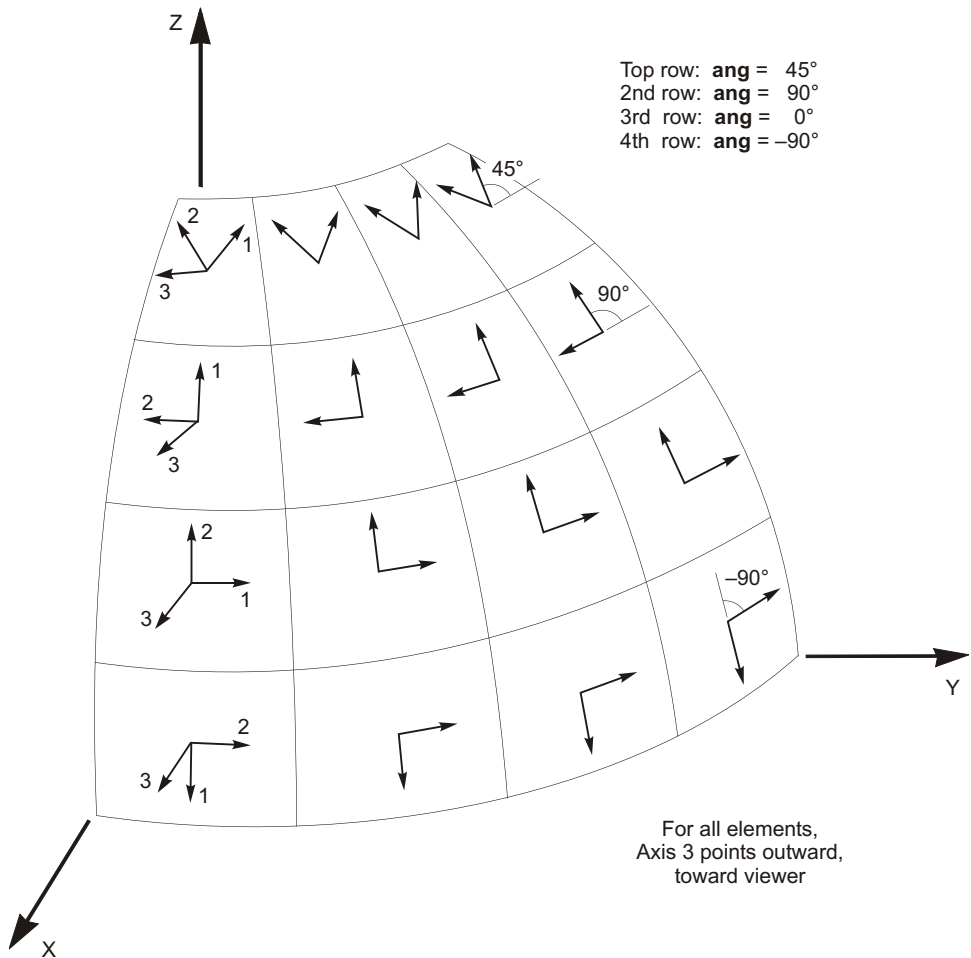


Figure 14
The Shell Element Coordinate Angle with Respect to the Default Orientation

Section Properties

A **Shell Section** is a set of material and geometric properties that describe the cross-section of one or more Shell elements. Sections are defined independently of the Shell elements, and are assigned to the area objects.

Section Type

When defining an area section, you have a choice of three basic element types:

- Shell– the subject of this chapter, with translational and rotational degrees of freedom, capable of supporting forces and moments
- Plane (stress or strain) – a two-dimensional solid, with translational degrees of freedom, capable of supporting forces but not moments. This element is not covered in this manual.
- A solid – axisymmetric solid, with translational degrees of freedom, capable of supporting forces but not moments. This element is not covered in this manual.

For shell sections, you may choose one of the following sub-types of behavior:

- Membrane – pure membrane behavior; only the in-plane forces and the normal (drilling) moment can be supported
- Plate – pure plate behavior; only the bending moments and the transverse force can be supported
- Shell – full shell behavior, a combination of membrane and plate behavior; all forces and moments can be supported

It is generally recommended that you use the full shell behavior unless the entire structure is planar and is adequately restrained.

Thickness Formulation

Two thickness formulations are available, which determine whether or not transverse shearing deformations are included in the plate-bending behavior of a plate or shell element:

- The thick-plate (Mindlin/Reissner) formulation, which includes the effects of transverse shear deformation
- The thin-plate (Kirchhoff) formulation, which neglects transverse shearing deformation

Shearing deformations tend to be important when the thickness is greater than about one-tenth to one-fifth of the span. They can also be quite significant in the vicinity of bending-stress concentrations, such as near sudden changes in thickness or support conditions, and near holes or re-entrant corners.

Even for thin-plate bending problems where shearing deformations are truly negligible, the thick-plate formulation tends to be more accurate, although somewhat

stiffer, than the thin-plate formulation. However, the accuracy of the thick-plate formulation is more sensitive to large aspect ratios and mesh distortion than is the thin-plate formulation.

It is generally recommended that you use the thick-plate formulation unless you are using a distorted mesh and you know that shearing deformations will be small, or unless you are trying to match a theoretical thin-plate solution.

The thickness formulation has no effect upon membrane behavior, only upon plate-bending behavior.

Material Properties

The material properties for each Section are specified by reference to a previously-defined Material. The material properties used by the Shell Section are:

- The modulus of elasticity, **e1**, and Poisson's ratio, **u12**, to compute the membrane and plate-bending stiffness;
- The mass density (per unit volume), **m**, for computing element mass;
- The weight density (per unit volume), **w**, for computing Self-Weight Load.

Orthotropic properties are also available, as discussed in the complete *SAP2000*, *ETABS*, and *SAFE Analysis Reference Manual*.

Thickness

Each Shell Section has a constant membrane thickness and a constant bending thickness. The membrane thickness, **th**, is used for calculating:

- The membrane stiffness for full-shell and pure-membrane Sections
- The element volume for the element self-weight and mass calculations

The bending thickness, **thb**, is use for calculating:

- The plate-bending stiffness for full-shell and pure-plate Sections

Normally these two thicknesses are the same. However, for some applications, such as modeling corrugated surfaces, the membrane and plate-bending behavior cannot be adequately represented by a homogeneous material of a single thickness. For this purpose, you may specify a value of **thb** that is different from **th**.

Mass

In a dynamic analysis, the mass of the structure is used to compute inertial forces. The mass contributed by the Shell element is lumped at the element joints. *No inertial effects are considered within the element itself.*

The total mass of the element is equal to the integral over the plane of the element of the mass density, **m**, multiplied by the thickness, **th**. The total mass is apportioned to the joints in a manner that is proportional to the diagonal terms of the consistent mass matrix. See Cook, Malkus, and Plesha (1989) for more information. The total mass is applied to each of the three translational degrees of freedom: UX, UY, and UZ. No mass moments of inertia are computed for the rotational degrees of freedom.

For more information:

- See Subtopic “Thickness” (page 44) in this chapter for the definition of **th**.
- See Chapter “Static and Dynamic Analysis” (page 69).

Self-Weight Load

Self-Weight Load can be applied in any Load Case to activate the self-weight of all elements in the model. For a Shell element, the self-weight is a force that is uniformly distributed over the plane of the element. The magnitude of the self-weight is equal to the weight density, **w**, multiplied by the thickness, **th**.

Self-weight always acts downward, in the global $-Z$ direction. The self-weight may be scaled by a single factor that applies to the whole structure.

For more information:

- See Topic “Section Properties” (page 42) in this chapter for the definitions of **w** and **th**.
- See Chapter “Static and Dynamic Analysis” (page 69).

Uniform Load

Uniform Load is used to apply uniformly distributed forces to the midsurfaces of the Shell elements. The direction of the loading may be specified in the global coordinate system or in the element local coordinate system.

Load intensities are given as forces per unit area. Load intensities specified in different coordinate systems are converted to the element local coordinate system and added together. The total force acting on the element in each local direction is given by the total load intensity in that direction multiplied by the area of the midsurface. This force is apportioned to the joints of the element.

See Chapter “Static and Dynamic Analysis” (page 69) for more information.

Internal Force and Stress Output

The **Shell element stresses** are the forces-per-unit-area that act *within* the volume of the element to resist the loading. These stresses are:

- In-plane direct stresses: S11 and S22
- In-plane shear stress: S12
- Transverse shear stresses: S13 and S23
- Transverse direct stress: S33 (always assumed to be zero)

The three in-plane stresses are assumed to be constant or to vary linearly through the element thickness.

The two transverse shear stresses are assumed to be constant through the thickness. The actual shear stress distribution is parabolic, being zero at the top and bottom surfaces and taking a maximum or minimum value at the midsurface of the element.

The **Shell element internal forces** (also called **stress resultants**) are the forces and moments that result from integrating the stresses over the element thickness. These internal forces are:

- Membrane direct forces: F11 and F22
- Membrane shear force: F12
- Plate bending moments: M11 and M22
- Plate twisting moment: M12
- Plate transverse shear forces: V13 and V23

It is very important to note that these stress resultants are forces and moments *per unit of in-plane length*. They are present at every point on the midsurface of the element.

The sign conventions for the stresses and internal forces are illustrated in Figure 15 (page 47). Stresses acting on a positive face are oriented in the positive direction of

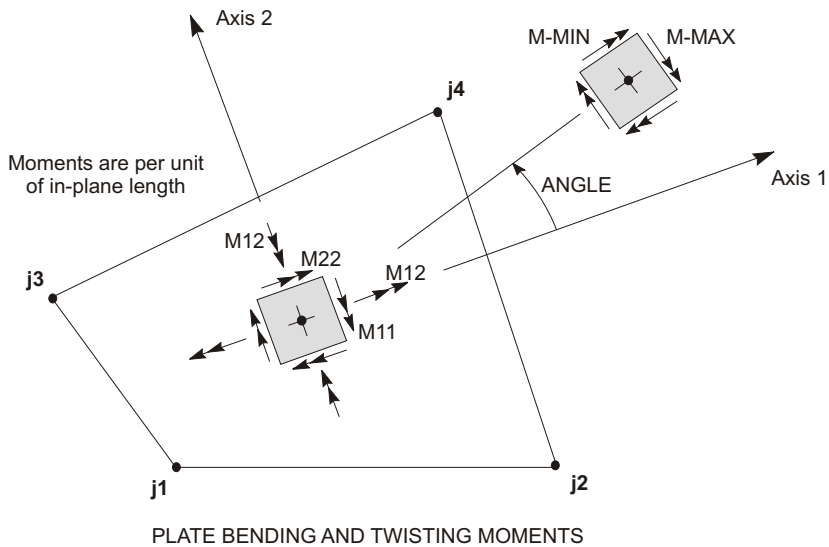
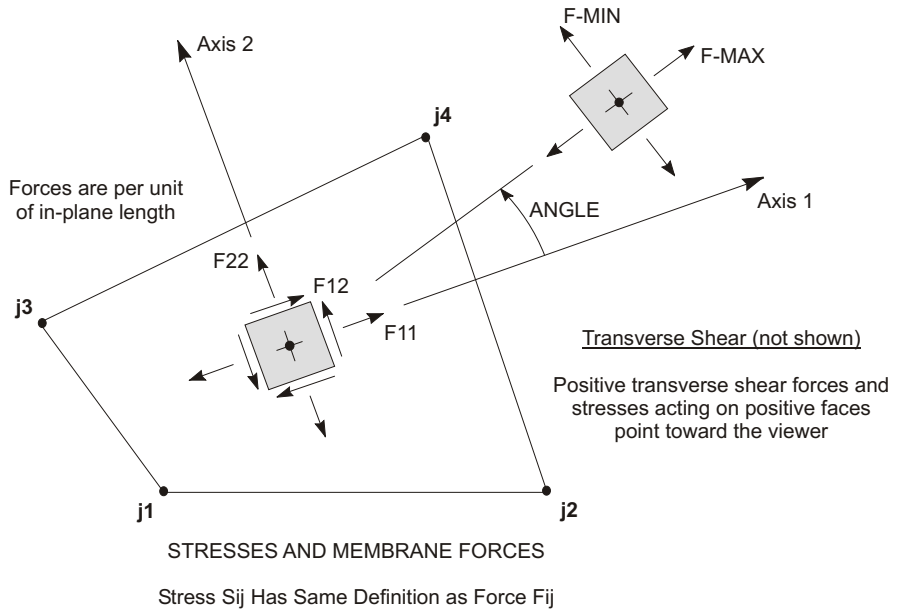


Figure 15
Shell Element Stresses and Internal Forces

the element local coordinate axes. Stresses acting on a negative face are oriented in the negative direction of the element local coordinate axes. A positive face is one

whose outward normal (pointing away from element) is in the positive local 1 or 2 direction.

Positive internal forces correspond to a state of positive stress that is constant through the thickness. Positive internal moments correspond to a state of stress that varies linearly through the thickness and is positive at the bottom.

The stresses and internal forces are evaluated at the standard 2-by-2 Gauss integration points of the element and extrapolated to the joints. Although they are reported at the joints, the stresses and internal forces exist throughout the element. See Cook, Malkus, and Plesha (1989) for more information.

The Shell element stresses and internal forces are computed for all Analysis Cases: Loads, Modes, and Specs.

Principal values and the associated principal directions are also computed for the Loads and Modes. The angle given is measured counter-clockwise (when viewed from the top) from the local 1 axis to the direction of the maximum principal value.

It is important to note that the Response Spectrum results are always positive, and that the correspondence between different values has been lost.

See Chapter “Static and Dynamic Analysis” (page 69) for more information.

Joints and Degrees of Freedom

The **joints** play a fundamental role in the analysis of any structure. Joints are the points of connection between the elements, and they are the primary locations in the structure at which the displacements are known or are to be determined. The displacement components (translations and rotations) at the joints are called the **degrees of freedom**.

Topics

- Overview
- Modeling Considerations
- Local Coordinate System
- Degrees of Freedom
- Restraints and Reactions
- Springs
- Masses
- Force Load
- Ground Displacement Load

Overview

Joints, also known as **nodal points** or **nodes**, are a fundamental part of every structural model. Joints perform a variety of functions:

- All elements are connected to the structure (and hence to each other) at the joints
- The structure is supported at the joints using restraints and/or springs
- Rigid-body behavior and symmetry conditions can be specified using Constraints that apply to the joints
- Concentrated loads may be applied at the joints
- Lumped (concentrated) masses and rotational inertia may be placed at the joints
- All loads and masses applied to the elements are actually transferred to the joints
- Joints are the primary locations in the structure at which the displacements are known (the supports) or are to be determined

All of these functions are discussed in this chapter except for the Constraints, which are described in Chapter “Joint Constraints” (page 65).

Joints in the analysis model correspond to point objects in the structural-object model. Using the SAP2000 graphical interface, joints (points) are automatically created at the ends of each frame/cable object and at the corners of each shell object. Joints may also be defined independently of any element.

Automatic meshing of frame/cable and shell objects will create additional joints corresponding to any frame/cable and shell elements that are created.

Joints may themselves be considered as elements. Each joint may have its own local coordinate system for defining the degrees of freedom, restraints, joint properties, and loads; and for interpreting joint output. In most cases, however, the global X-Y-Z coordinate system is used as the local coordinate system for all joints in the model.

There are six displacement degrees of freedom at every joint — three translations and three rotations. These displacement components are aligned along the local coordinate system of each joint.

Joints may be loaded directly by concentrated loads or indirectly by ground displacements acting through restraints or spring supports.

Displacements (translations and rotations) are produced at every joint. The external and internal forces and moments acting on each joint are also produced.

For more information and additional features:

- See Chapter “Joint Coordinates” in the *SAP2000, ETABS, and SAFE Analysis Reference Manual*.
- See Chapter “Joints and Degrees of Freedom” in the *SAP2000, ETABS, and SAFE Analysis Reference Manual*.
- See Chapter “Constraints and Welds” in the *SAP2000, ETABS, and SAFE Analysis Reference Manual*.

Modeling Considerations

The location of the joints and elements is critical in determining the accuracy of the structural model. Some of the factors that you need to consider when defining the elements (and hence joints) for the structure are:

- The number of elements should be sufficient to describe the geometry of the structure. For straight lines and edges, one element is adequate. For curves and curved surfaces, one element should be used for every arc of 15° or less.
- Element boundaries, and hence joints, should be located at points, lines, and surfaces of discontinuity:
 - Structural boundaries, e.g., corners and edges
 - Changes in material properties
 - Changes in thickness and other geometric properties
 - Support points (restraints and springs)
 - Points of application of concentrated loads, except that Frame elements may have concentrated loads applied within their spans
- In regions having large stress gradients, i.e., where the stresses are changing rapidly, a shell element mesh should be refined using small elements and closely-spaced joints. This may require changing the mesh after one or more preliminary analyses, which can be done by modifying the automated-mesh parameters for an area object.
- More than one element should be used to model the length of any span for which dynamic behavior is important. This is required because the mass is always lumped at the joints, even if it is contributed by the elements.

Local Coordinate System

Each joint has its own **joint local coordinate system** used to define the degrees of freedom, restraints, properties, and loads at the joint; and for interpreting joint output. The axes of the joint local coordinate system are denoted 1, 2, and 3. By default these axes are identical to the global X, Y, and Z axes, respectively. Both systems are right-handed coordinate systems.

The default local coordinate system is adequate for most situations. However, for certain modeling purposes it may be useful to use different local coordinate systems at some or all of the joints. This is described in Chapter “Joint Degrees of Freedom” in the *SAP2000, ETABS, and SAFE Analysis Reference Manual*.

For more information see Chapter “Coordinate Systems” (page 7).

Degrees of Freedom

The deflection of the structural model is governed by the displacements of the joints. Every joint of the structural model may have up to six displacement components:

- The joint may translate along its three local axes. These translations are denoted U1, U2, and U3.
- The joint may rotate about its three local axes. These rotations are denoted R1, R2, and R3.

These six displacement components are known as the **degrees of freedom** of the joint. The joint local degrees of freedom are illustrated in Figure 16 (page 53).

In addition to the regular joints defined as part of your structural model, the program automatically creates master joints that govern the behavior of any Constraints that you may have defined. Each master joint has the same six degrees of freedom as do the regular joints. See Chapter “Joint Constraints” (page 65) for more information.

Each degree of freedom in the structural model must be one of the following types:

- Active — the displacement is computed during the analysis
- Restrained — the displacement is specified, and the corresponding reaction is computed during the analysis

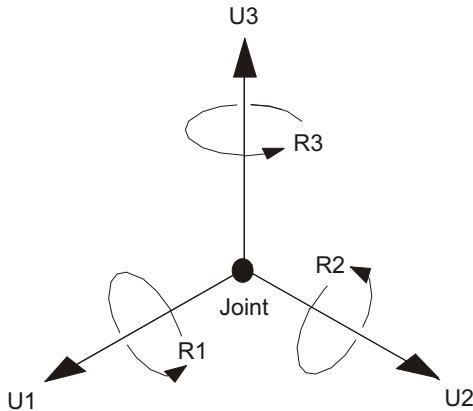


Figure 16

The Six Displacement Degrees of Freedom in the Joint Local Coordinate System

- **Constrained** — the displacement is determined from the displacements at other degrees of freedom
- **Null** — the displacement does not affect the structure and is ignored by the analysis
- **Unavailable** — the displacement has been explicitly excluded from the analysis

These different types of degrees of freedom are described in the following subtopics.

Available and Unavailable Degrees of Freedom

You may explicitly specify the set of global degrees of freedom that are available to every joint in the structural model. By default, all six degrees of freedom are available to every joint. This default should generally be used for all three-dimensional structures.

For certain planar structures, however, you may wish to restrict the **available degrees of freedom**. For example, in the X-Z plane: a planar truss needs only UX and UZ; a planar frame needs only UX, UZ, and RY; and a planar grid or flat plate needs only UY, RX, and RZ.

The degrees of freedom that are not specified as being available are called **unavailable degrees of freedom**. Any stiffness, loads, mass, restraints, or constraints that are applied to the unavailable degrees of freedom are ignored by the analysis.

Available degrees of freedom may be restrained, constrained, active, or null.

Restrained Degrees of Freedom

If the displacement of a joint along any one of its available degrees of freedom is known, such as at a support point, that degree of freedom is **restrained**. The known value of the displacement may be zero or non-zero, and may be different in different Load Cases. The force along the restrained degree of freedom that is required to impose the specified restraint displacement is called the reaction, and is determined by the analysis.

Unavailable degrees of freedom are essentially restrained. However, they are excluded from the analysis and no reactions are computed, even if they are non-zero.

See Topic “Restraints and Reactions” (page 55) in this chapter for more information.

Constrained Degrees of Freedom

Any joint that is part of a constraint may have one or more of its available degrees of freedom **constrained**. The program automatically creates a master joint to govern the behavior of each constraint. The displacement of a constrained degree of freedom is then computed as a linear combination of the displacements along the degrees of freedom at the corresponding master joint.

If a constrained degree of freedom is also restrained, the restraint will apply to the whole set of constrained joints.

See Chapter “Joint Constraints” (page 65) for more information.

Active Degrees of Freedom

All available degrees of freedom that are neither constrained nor restrained must be either active or null. The program will automatically determine the **active degrees of freedom** as follows:

- If any load or stiffness is applied along *any* translational degree of freedom at a joint, then all available translational degrees of freedom at that joint are made active unless they are constrained or restrained.

- If any load or stiffness is applied along *any* rotational degree of freedom at a joint, then all available rotational degrees of freedom at that joint are made active unless they are constrained or restrained.
- All degrees of freedom at a master joint that govern constrained degrees of freedom are made active.

A joint that is connected to any frame or shell element will have all of its available degrees of freedom activated. An exception is a Frame element with only truss-type stiffness, which will not activate rotational degrees of freedom.

Every active degree of freedom has an associated equation to be solved. If there are N active degrees of freedom in the structure, there are N equations in the system, and the structural stiffness matrix is said to be of order N . The amount of computational effort required to perform the analysis increases with N .

The load acting along each active degree of freedom is known (it may be zero). The corresponding displacement will be determined by the analysis.

If there are active degrees of freedom in the system at which the stiffness is known to be zero, such as the out-of-plane translation in a planar-frame, these must either be restrained or made unavailable. Otherwise, the structure is unstable and the solution of the static equations will complain.

For more information:

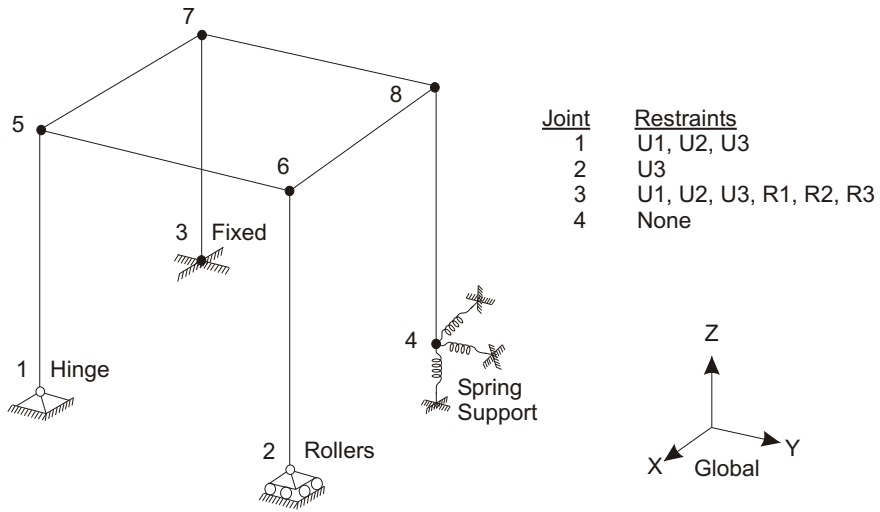
- See Topic “Degrees of Freedom” (page 14) in Chapter “The Frame Element.”
- See Topic “Degrees of Freedom” (page 40) in Chapter “The Shell Element.”

Null Degrees of Freedom

The available degrees of freedom that are not restrained, constrained, or active, are called the null degrees of freedom. Because they have no load or stiffness, their displacements and reactions are zero, and they have no effect on the rest of the structure. The program automatically excludes them from the analysis.

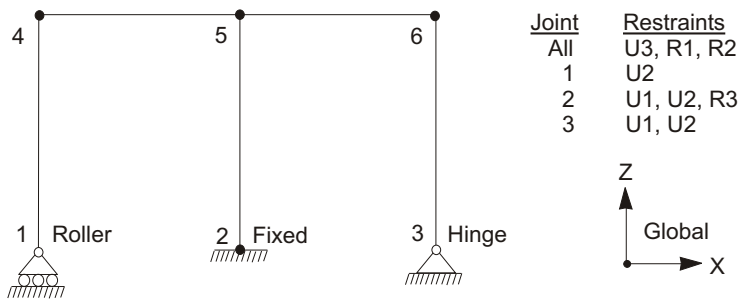
Restraints and Reactions

If the displacement of a joint along any of its degrees of freedom has a known value, either zero (e.g., at support points) or non-zero (e.g., due to support settlement), a **restraint** must be applied to that degree of freedom. The known value of the displacement may differ from one Load Case to the next, but the degree of freedom is



3-D Frame Structure

Notes: Joints are indicated with dots:
 • Solid dots indicate moment continuity
 ○ Open dots indicate hinges
 All joint local 1-2-3 coordinate systems are identical to the global X-Y-Z coordinate system



2-D Frame Structure, X-Z plane

Figure 17
Examples of Restraints

restrained for all Load Cases. In other words, it is not possible to have the displacement known in one Load Case and unknown (unrestrained) in another Load Case.

Restraints should also be applied to available degrees of freedom in the system at which the stiffness is known to be zero, such as the out-of-plane translation and in-plane rotations of a planar-frame. Otherwise, the structure is unstable and the solution of the static equations will complain.

The force or moment along the degree of freedom that is required to enforce the restraint is called the **reaction**, and it is determined by the analysis. The reaction may differ from one Load Case to the next. The reaction includes the forces (or moments) from all elements connected to the restrained degree of freedom, as well as all loads applied to the degree of freedom.

Examples of Restraints are shown in Figure 17 (page 56).

For more information:

- See Topic “Degrees of Freedom” (page 52) in this chapter.
- See Topic “Ground Displacement Load” (page 59) in this chapter.

Springs

Any of the six degrees of freedom at any of the joints in the structure can have translational or rotational spring support conditions. These springs elastically connect the joint to the ground. Spring supports along restrained degrees of freedom do not contribute to the stiffness of the structure.

The spring forces that act on a joint are related to the displacements of that joint by a 6x6 symmetric matrix of spring stiffness coefficients. These forces tend to oppose the displacements. Spring stiffness coefficients are specified in the joint local coordinate system. The spring forces and moments F_1, F_2, F_3, M_1, M_2 and M_3 at a joint are given by:

$$\begin{array}{rccccccc}
 F_1 & \mathbf{u1} & \mathbf{0} & \mathbf{0} & \mathbf{0} & \mathbf{0} & u_1 \\
 F_2 & & \mathbf{u2} & \mathbf{0} & \mathbf{0} & \mathbf{0} & u_2 \\
 F_3 & & & \mathbf{u3} & \mathbf{0} & \mathbf{0} & u_3 \\
 M_1 & & & & \mathbf{r1} & \mathbf{0} & r_1 \\
 M_2 & & \mathbf{sym.} & & & \mathbf{r2} & r_2 \\
 M_3 & & & & & & \mathbf{r3} & r_3
 \end{array} \quad (\text{Eqn. 1})$$

where u_1, u_2, u_3, r_1, r_2 and r_3 are the joint displacements and rotations, and the terms $\mathbf{u1}, \mathbf{u2}, \mathbf{u3}, \mathbf{r1}, \mathbf{r2}$, and $\mathbf{r3}$ are the specified spring stiffness coefficients.

The displacement of the grounded end of the spring may be specified to be zero or non-zero (e.g., due to support settlement). This **ground displacement** may vary from one Load Case to the next.

For more information:

- See Topic “Degrees of Freedom” (page 52) in this chapter.
- See Topic “Ground Displacement Load” (page 59) in this chapter.

Masses

In a dynamic analysis, the mass of the structure is used to compute inertial forces. Normally, the mass is obtained from the elements using the mass density of the material and the volume of the element. This automatically produces lumped (uncoupled) masses at the joints. The element mass values are equal for each of the three translational degrees of freedom. No mass moments of inertia are produced for the rotational degrees of freedom. This approach is adequate for most analyses.

It is often necessary to place additional concentrated masses and/or mass moments of inertia at the joints. These can be applied to any of the six degrees of freedom at any of the joints in the structure.

For computational efficiency and solution accuracy, SAP2000 always uses lumped masses. This means that there is no mass coupling between degrees of freedom at a joint or between different joints. These uncoupled masses are always referred to the local coordinate system of each joint. Mass values along restrained degrees of freedom are ignored.

Inertial forces acting on the joints are related to the accelerations at the joints by a 6x6 matrix of mass values. These forces tend to oppose the accelerations. In a joint local coordinate system, the inertia forces and moments F_1, F_2, F_3, M_1, M_2 and M_3 at a joint are given by:

$$\begin{array}{rccccccc}
 F_1 & \mathbf{u1} & \mathbf{0} & \mathbf{0} & \mathbf{0} & \mathbf{0} & \mathbf{0} & \ddot{u}_1 \\
 F_2 & & \mathbf{u2} & \mathbf{0} & \mathbf{0} & \mathbf{0} & \mathbf{0} & \ddot{u}_2 \\
 F_3 & & & \mathbf{u3} & \mathbf{0} & \mathbf{0} & \mathbf{0} & \ddot{u}_3 \\
 M_1 & & & & \mathbf{r1} & \mathbf{0} & \mathbf{0} & \ddot{r}_1 \\
 M_2 & & \mathbf{sym.} & & & \mathbf{r2} & \mathbf{0} & \ddot{r}_2 \\
 M_3 & & & & & & \mathbf{r3} & \ddot{r}_3
 \end{array}$$

where $\ddot{u}_1, \ddot{u}_2, \ddot{u}_3, \ddot{r}_1, \ddot{r}_2$ and \ddot{r}_3 are the translational and rotational accelerations at the joint, and the terms **u1**, **u2**, **u3**, **r1**, **r2**, and **r3** are the specified mass values.

Mass values must be given in consistent mass units (W/g) and mass moments of inertia must be in WL^2/g units. Here W is weight, L is length, and g is the acceleration due to gravity. The net mass values at each joint in the structure should be zero or positive.

See Figure 18 (page 60) for mass moment of inertia formulations for various planar configurations.

For more information:

- See Topic “Degrees of Freedom” (page 52) in this chapter.
- See Chapter “Static and Dynamic Analysis” (page 69).

Force Load

Force load is used to apply concentrated forces and moments at the joints. Values are specified in global coordinates as shown in Figure 19 (page 61). The force load may vary from one Load Case to the next.

Forces and moments applied along restrained degrees of freedom add to the corresponding reaction, but do not otherwise affect the structure.

For more information, see Topic “Degrees of Freedom” (page 52) in this chapter.

Ground Displacement Load

The ground displacement load is used to apply specified displacements (translations and rotations) at the *grounded end* of joint restraints and spring supports. Displacement values are specified in global coordinates as shown in Figure 19 (page 61). These values are converted to joint local coordinates before being applied to the joint through the restraints and springs.

Restraints may be considered as *rigid* connections between the joint degrees of freedom and the ground. Springs may be considered as *flexible* connections between the joint degrees of freedom and the ground.

It is very important to understand that ground displacement load applies to the ground, and does not affect the structure unless the structure is supported by restraints or springs in the direction of loading!

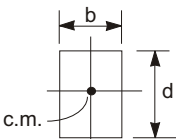
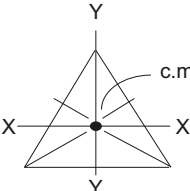
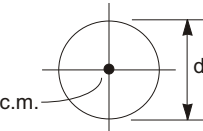
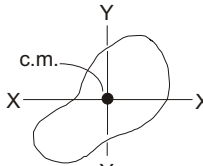
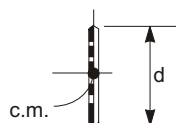
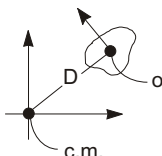
Shape in plan	Mass Moment of Inertia about vertical axis (normal to paper) through center of mass	Formula
	<p>Rectangular diaphragm: Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g)</p>	$MMI_{cm} = \frac{M(b^2+d^2)}{12}$
	<p>Triangular diaphragm: Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g)</p>	<p>Use general diaphragm formula</p>
	<p>Circular diaphragm: Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g)</p>	$MMI_{cm} = \frac{Md^2}{8}$
	<p>General diaphragm: Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g) Area of diaphragm = A Moment of inertia of area about X-X = I_x Moment of inertia of area about Y-Y = I_y</p>	$MMI_{cm} = \frac{M(I_x+I_y)}{A}$
	<p>Line mass: Uniformly distributed mass per unit length Total mass of line = M (or w/g)</p>	$MMI_{cm} = \frac{Md^2}{12}$
	<p>Axis transformation for a mass: If mass is a point mass, $MMI_o = 0$</p>	$MMI_{cm} = MMI_o + MD^2$

Figure 18
Formulae for Mass Moments of Inertia

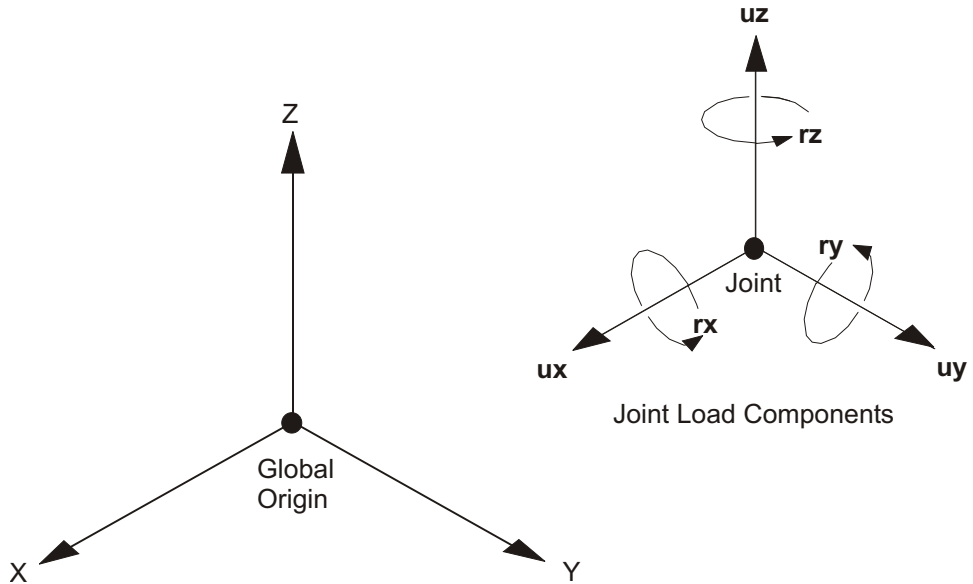


Figure 19
Specified Values for Force Load and Ground Displacement Load

Restraint Displacements

If a particular joint degree of freedom is restrained, the displacement of the joint is equal to the ground displacement along that local degree of freedom. This applies regardless of whether or not springs are present.

The ground displacement, and hence the joint displacement, may vary from one Load Case to the next. If no ground displacement load is specified for a restrained degree of freedom, the joint displacement is zero for that Load Case.

Components of ground displacement that are not along restrained degrees of freedom do not load the structure (except possibly through springs). An example of this is illustrated in Figure 20 (page 62).

Spring Displacements

The ground displacements at a joint are multiplied by the spring stiffness coefficients to obtain effective forces and moments that are applied to the joint. Spring displacements applied in a direction with no spring stiffness result in zero applied

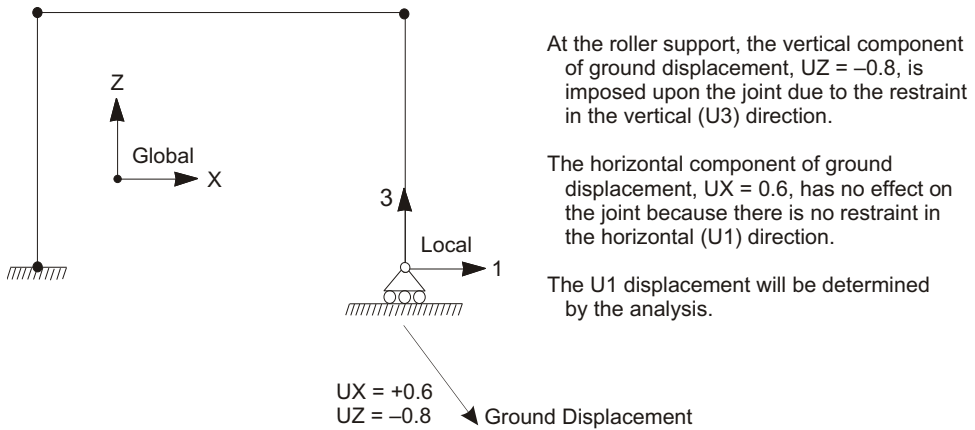


Figure 20
Ground Displacement at Restrained and Unrestrained Degrees of Freedom

load. The ground displacement, and hence the applied forces and moments, may vary from one Load Case to the next.

In a joint local coordinate system, the applied forces and moments F_1, F_2, F_3, M_1, M_2 and M_3 at a joint due to ground displacements are given by:

$$\begin{array}{rcl}
 F_1 & \mathbf{u1} & \mathbf{0} \quad \mathbf{0} \quad \mathbf{0} \quad \mathbf{0} \quad \mathbf{0} \quad u_{g1} \\
 F_2 & & \mathbf{u2} \quad \mathbf{0} \quad \mathbf{0} \quad \mathbf{0} \quad \mathbf{0} \quad u_{g2} \\
 F_3 & & & \mathbf{u3} \quad \mathbf{0} \quad \mathbf{0} \quad \mathbf{0} \quad u_{g3} \\
 M_1 & & & & \mathbf{r1} \quad \mathbf{0} \quad \mathbf{0} \quad r_{g1} \\
 M_2 & & \mathbf{sym.} & & \mathbf{r2} \quad \mathbf{0} \quad r_{g2} \\
 M_3 & & & & & \mathbf{r3} \quad r_{g3}
 \end{array} \quad (\text{Eqn. 2})$$

where $u_{g1}, u_{g2}, u_{g3}, r_{g1}, r_{g2}$ and r_{g3} are the ground displacements and rotations, and the terms $\mathbf{u1}, \mathbf{u2}, \mathbf{u3}, \mathbf{r1}, \mathbf{r2}$, and $\mathbf{r3}$ are the specified spring stiffness coefficients.

The net spring forces and moments acting on the joint are the sum of the forces and moments given in Equations (1) and (2); note that these are of opposite sign. At a restrained degree of freedom, the joint displacement is equal to the ground displacement, and hence the net spring force is zero.

For more information:

- See Topic “Restraints and Reactions” (page 55) in this chapter.

- See Topic “Springs” (page 57) in this chapter.

Joint Constraints

Constraints are used to enforce certain types of rigid-body behavior, to connect together different parts of the model, and to impose certain types of symmetry conditions.

Topics

- Overview
- Diaphragm Constraint

Overview

A constraint consists of a set of two or more **constrained joints**. The displacements of each pair of joints in the constraint are related by constraint equations. The types of behavior that can be enforced by constraints are:

- Rigid-body behavior, in which the constrained joints translate and rotate together as if connected by rigid links. The types of rigid behavior that can be modeled are:
 - Rigid Body: fully rigid for all displacements
 - Rigid Diaphragm: rigid for membrane behavior in a plane

- Rigid Plate: rigid for plate bending in a plane
- Rigid Rod: rigid for extension along an axis
- Rigid Beam: rigid for beam bending on an axis
- Equal-displacement behavior, in which the translations and rotations are equal at the constrained joints
- Symmetry and anti-symmetry conditions

The use of constraints reduces the number of equations in the system to be solved and will usually result in increased computational efficiency.

Only the diaphragm constraint is described in the chapter, since it is the most commonly used type of constraint.

For more information and additional features see Chapter “Constraints and Welds” in the *SAP2000, ETABS, and SAFE Analysis Reference Manual*.

Diaphragm Constraint

A diaphragm constraint causes all of its constrained joints to move together as a planar diaphragm that is rigid against membrane deformation. Effectively, all constrained joints are connected to each other by links that are rigid in the plane, but do not affect out-of-plane (plate) deformation.

This constraint can be used to:

- Model concrete floors (or concrete-filled decks) in building structures, which typically have very high in-plane stiffness
- Model diaphragms in bridge superstructures

The use of the diaphragm constraint for building structures eliminates the numerical-accuracy problems created when the large in-plane stiffness of a floor diaphragm is modeled with membrane elements. It is also very useful in the lateral (horizontal) dynamic analysis of buildings, as it results in a significant reduction in the size of the eigenvalue problem to be solved. See Figure 21 (page 67) for an illustration of a floor diaphragm.

Joint Connectivity

Each Diaphragm Constraint connects a set of two or more joints together. The joints may have any arbitrary location in space, but for best results all joints should

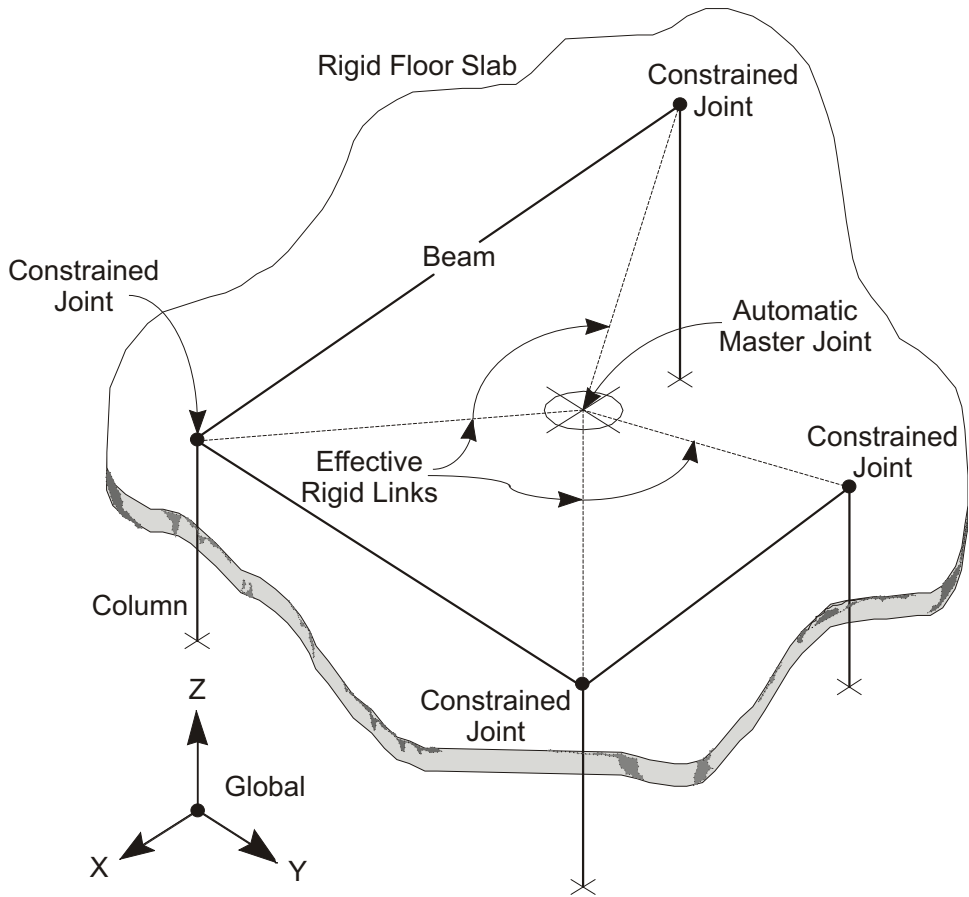


Figure 21

Use of the Diaphragm Constraint to Model a Rigid Floor Slab

lie in the plane of the constraint. Otherwise, bending moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure.

Plane Definition

The constraint equations for each diaphragm constraint are written with respect to a particular plane. The location of the plane is not important, only its orientation.

By default, the plane is determined automatically by the program from the spatial distribution of the constrained joints. If no unique direction can be found, the hori-

zontal (X-Y) plane is assumed; this can occur if the joints are coincident or collinear, or if the spatial distribution is more nearly three-dimensional than planar.

You may override automatic plane selection by specifying the global axis (X, Y, or Z) that is normal to the plane of the constraint. This may be useful, for example, to specify a horizontal plane for a floor with a small step in it.

Local Coordinate System

Each diaphragm constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. Local axis 3 is always normal to the plane of the constraint. The program automatically arbitrarily chooses the orientation of axes 1 and 2 in the plane. The actual orientation of the planar axes is not important since only the normal direction affects the constraint equations.

Constraint Equations

The constraint equations relate the displacements at any two constrained joints (subscripts i and j) in a diaphragm constraint. These equations are expressed in terms of in-plane translations (u_1 and u_2), the rotation (r_3) about the normal, and the in-plane coordinates (x_1 and x_2), all taken in the constraint local coordinate system:

$$u_{1j} = u_{1i} - r_{3i} \ x_2$$

$$u_{2j} = u_{2i} + r_{3i} \ x_1$$

$$r_{3i} = r_{3j}$$

where $x_1 = x_{1j} - x_{1i}$ and $x_2 = x_{2j} - x_{2i}$.

Static and Dynamic Analysis

Static and dynamic analyses are used to determine the response of the structure to various types of loading. This chapter describes the basic types of analysis available for SAP2000.

Topics

- Overview
- Loads
- Analysis Cases
- Linear Static Analysis
- Eigenvector Analysis
- Ritz-vector Analysis
- Modal Analysis Results
- Response-Spectrum Analysis
- Response-Spectrum Analysis Results

Overview

Many different types of analysis are available using program SAP2000. These include:

- Linear static analysis
- Modal analysis for vibration modes, using eigenvectors or Ritz vectors
- Response-spectrum analysis for seismic response
- Other types of linear and nonlinear, static and dynamic analysis that are beyond the scope of this introductory manual

These different types of analyses can all be defined in the same model, and the results combined for output.

For more information and additional features see Chapter “Analysis Cases” in the *SAP2000, ETABS, and SAFE Analysis Reference Manual*.

Loads

Loads represent actions upon the structure, such as force, pressure, support displacement, thermal effects, ground acceleration, and others. You may define named Load Cases containing any mixture of loads on the objects. The program automatically computes built-in ground acceleration loads.

In order to calculate any response of the structure due to the Load Cases, you must define and run analysis cases which specify how the Load Cases are to be applied (e.g., statically, dynamically, etc.) and how the structure is to be analyzed (e.g., linearly, nonlinearly, etc.) The same Load Case can be applied differently in different analysis cases.

By default, the program creates a linear static analysis case corresponding to each load case that you define.

Load Cases

You can define as many named Load Cases as you like. Typically you would have separate Load Cases for dead load, live load, wind load, snow load, thermal load, and so on. Loads that need to vary independently, either for design purposes or because of how they are applied to the structure, should be defined as separate Load Cases.

After defining a Load Case name, you must assign specific load values to the objects as part of that Load Case. Each Load Case may include:

- Self-Weight Loads on Frame and/or Shell elements
- Concentrated and Distributed Span Loads on Frame elements
- Uniform Loads on Shell elements
- Force and/or Ground Displacement Loads on Joints
- Other types of loads described in the *SAP2000*, *ETABS*, and *SAFE Analysis Reference Manual*

Each object can be subjected to multiple Load Cases.

Acceleration Loads

The program automatically computes three Acceleration Loads that act on the structure due to unit translational accelerations in each of the three global directions. They are determined by d'Alembert's principal, and are denoted m_x , m_y , and m_z . These loads are used for applying ground accelerations in response-spectrum analyses, and are used as starting load vectors for Ritz-vector analysis.

These loads are computed for each joint and element and summed over the whole structure. The Acceleration Loads for the joints are simply equal to the negative of the joint translational masses in the joint local coordinate system. These loads are transformed to the global coordinate system.

The Acceleration Loads for all elements are the same in each direction and are equal to the negative of the element mass. No coordinate transformations are necessary.

The Acceleration Loads can be transformed into any coordinate system. In the global coordinate system, the Acceleration Loads along the positive X, Y, and Z axes are denoted UX, UY, and UZ, respectively. In a local coordinate system defined for a response-spectrum analysis, the Acceleration Loads along the positive local 1, 2, and 3 axes are denoted U1, U2, and U3, respectively.

Analysis Cases

Each different analysis performed is called an **analysis case**. You assign a label to each analysis case as part of its definition. These labels can be used to create additional combinations and to control output.

The basic types of analysis cases are:

- Linear static analysis
- Modal analysis
- Response-spectrum analysis

You may define any number of each different type of analysis case for a single model. Other types of analysis cases are also available.

By default, the program creates a linear static analysis case for each load case that you define, as well as a single modal analysis case for the first few eigen-modes of the structure.

Linear combinations and envelopes of the various analysis cases are available through the SAP2000 graphical interface.

For more information:

- See Topic “Eigenvector Analysis” (page 73) in this chapter.
- See Topic “Ritz-vector Analysis” (page 74) in this chapter.
- See Topic “Response-Spectrum Analysis” (page 77) in this chapter.

Linear Static Analysis

The static analysis of a structure involves the solution of the system of linear equations represented by:

$$\mathbf{K} \mathbf{u} = \mathbf{r}$$

where \mathbf{K} is the stiffness matrix, \mathbf{r} is the vector of applied loads, and \mathbf{u} is the vector of resulting displacements. See Bathe and Wilson (1976).

For each linear static Analysis Case that you define, you may specify a linear combination of one or more Load Cases and/or Acceleration Loads to be applied in vector \mathbf{r} . Most commonly, however, you will want to solve a single a Loads Case in a each linear static Analysis Case, and combine the results later.

Modal Analysis

You may define as many modal analysis cases as you wish, although for most purposes one case is enough. For each modal analysis case, you may choose either eigenvector or Ritz-vector analysis.

Eigenvector Analysis

Eigenvector analysis determines the undamped free-vibration mode shapes and frequencies of the system. These natural Modes provide an excellent insight into the behavior of the structure. They can also be used as the basis for response-spectrum analyses, although Ritz vectors are recommended for this purpose.

Eigenvector analysis involves the solution of the generalized eigenvalue problem:

$$[\mathbf{K} - \omega^2 \mathbf{M}] \mathbf{0}$$

where \mathbf{K} is the stiffness matrix, \mathbf{M} is the diagonal mass matrix, ω^2 is the diagonal matrix of eigenvalues, and $\mathbf{0}$ is the matrix of corresponding eigenvectors (mode shapes).

Each eigenvalue-eigenvector pair is called a natural Vibration Mode of the structure. The Modes are identified by numbers from 1 to n in the order in which the modes are found by the program.

The eigenvalue is the square of the circular frequency, ω , for that Mode. The cyclic frequency, f , and period, T , of the Mode are related to ω by:

$$T = \frac{1}{f} \quad \text{and} \quad f = \frac{\omega}{2\pi}$$

You may specify the number of Modes, \mathbf{n} , to be found. The program will seek the \mathbf{n} lowest-frequency (longest-period) Modes.

The number of Modes actually found, n , is limited by:

- The number of Modes requested, \mathbf{n}
- The number of mass degrees of freedom in the model

A mass degree of freedom is any *active* degree of freedom that possesses translational mass or rotational mass moment of inertia. The mass may have been assigned directly to the joint or may come from connected elements.

Only the Modes that are actually found will be available for any subsequent response-spectrum analysis processing.

See Topic “Degrees of Freedom” (page 52) in Chapter “Joints and Degrees of Freedom.”

Ritz-vector Analysis

Research has indicated that the natural free-vibration mode shapes are not the best basis for a mode-superposition analysis of structures subjected to dynamic loads. It has been demonstrated (Wilson, Yuan, and Dickens, 1982) that dynamic analyses based on a special set of load-dependent Ritz vectors yield more accurate results than the use of the same number of natural mode shapes.

The reason the Ritz vectors yield excellent results is that they are generated by taking into account the spatial distribution of the dynamic loading, whereas the direct use of the natural mode shapes neglects this very important information.

The spatial distribution of the dynamic load vector serves as a **starting load vector** to initiate the procedure. The first Ritz vector is the static displacement vector corresponding to the starting load vector. The remaining vectors are generated from a recurrence relationship in which the mass matrix is multiplied by the previously obtained Ritz vector and used as the load vector for the next static solution. Each static solution is called a **generation cycle**.

When the dynamic load is made up of several independent spatial distributions, each of these may serve as a starting load vector to generate a set of Ritz vectors. Each generation cycle creates as many Ritz vectors as there are starting load vectors. If a generated Ritz vector is redundant or does not excite any mass degrees of freedom, it is discarded and the corresponding starting load vector is removed from all subsequent generation cycles.

For seismic analysis, including response-spectrum analysis, you should use the three acceleration loads as the starting load vectors. This produces better response-spectrum results than using the same number of eigen Modes.

Standard eigensolution techniques are used to orthogonalize the set of generated Ritz vectors, resulting in a final set of Ritz-vector Modes. Each Ritz-vector Mode consists of a mode shape and frequency. The full set of Ritz-vector Modes can be used as a basis to represent the dynamic displacement of the structure.

Once the stiffness matrix is triangularized it is only necessary to statically solve for one load vector for each Ritz vector required. This results in an extremely efficient

algorithm. Also, the method automatically includes the advantages of the proven numerical techniques of static condensation, Guyan reduction, and static correction due to higher-mode truncation.

The algorithm is detailed in Wilson (1985).

When a sufficient number of Ritz-vector Modes have been found, some of them may closely approximate natural mode shapes and frequencies. In general, however, Ritz-vector Modes do not represent the intrinsic characteristics of the structure in the same way the natural eigen-Modes do. The Ritz-vector Modes are biased by the starting load vectors.

You may specify the total number of Modes, n , to be found. The total number of Modes actually found, n , is limited by:

- The number of Modes requested, n
- The number of mass degrees of freedom present in the model
- The number of natural free-vibration modes that are excited by the starting load vectors (some additional natural modes may creep in due to numerical noise)

A mass degree of freedom is any *active* degree of freedom that possesses translational mass or rotational mass moment of inertia. The mass may have been assigned directly to the joint or may come from connected elements.

Only the Modes that are actually found will be available for any subsequent response-spectrum analysis processing.

See Topic “Degrees of Freedom” (page 52) in Chapter “Joint Degrees of Freedom.”

Modal Analysis Results

Each modal analysis cases results in a set of modes. Each mode consists of a mode shape (normalized deflected shape) and a set of modal properties. These are available for display and printing from the SAP2000 graphical interface. This information is the same regardless of whether you use eigenvector or Ritz-vector analysis, and is described in the following subtopics.

Periods and Frequencies

The following time-properties are given for each Mode:

- Period, T , in units of time
- Cyclic frequency, f , in units of cycles per time; this is the inverse of T

- Circular frequency, ω , in units of radians per time; $\omega = 2\pi f$
- Eigenvalue, ω^2 , in units of radians-per-time squared

Modal Stiffness and Mass

For each mode shape, only the relative deflection values are important. The overall scaling is arbitrary. In SAP2000, the modes shapes are each normalized, or scaled, with respect to the mass matrix such that:

$$\mathbf{m}_n^T \mathbf{M} \mathbf{m}_n = 1$$

This quantity is called the modal mass. Similarly, the modal stiffness is defined as:

$$\mathbf{m}_n^T \mathbf{K} \mathbf{m}_n$$

Regardless of how the modes are scaled, the ratio of modal stiffness to modal mass always gives the modal eigenvalue:

$$\frac{\mathbf{m}_n^T \mathbf{K} \mathbf{m}_n}{\mathbf{m}_n^T \mathbf{M} \mathbf{m}_n} = \omega_n^2$$

Participation Factors

The modal participation factors are the dot products of the three Acceleration Loads with the modes shapes. The participation factors for Mode n corresponding to Acceleration Loads in the global X, Y, and Z directions are given by:

$$f_{xn} = \mathbf{m}_n^T \mathbf{m}_x$$

$$f_{yn} = \mathbf{m}_n^T \mathbf{m}_y$$

$$f_{zn} = \mathbf{m}_n^T \mathbf{m}_z$$

where \mathbf{m}_n is the mode shape and \mathbf{m}_x , \mathbf{m}_y , and, \mathbf{m}_z are the unit Acceleration Loads. These factors are the generalized loads acting on the Mode due to each of the Acceleration Loads. They are referred to the global coordinate system.

These participation factors indicate how strongly each mode is excited by the respective acceleration loads.

Participating Mass Ratios

The participating mass ratio for a Mode provides a relative measure of how important the Mode is for computing the response to the Acceleration Loads in each of the three global directions. Thus it is useful for determining the accuracy of response-spectrum analyses.

The participating mass ratios for Mode n corresponding to Acceleration Loads in the global X, Y, and Z directions are given by:

$$P_{xn} = \frac{(f_{xn})^2}{M_x}$$

$$P_{yn} = \frac{(f_{yn})^2}{M_y}$$

$$P_{zn} = \frac{(f_{zn})^2}{M_z}$$

where f_{xn} , f_{yn} , and f_{zn} are the participation factors defined in the previous subtopic; and M_x , M_y , and M_z are the total unrestrained masses acting in the X, Y, and Z directions. The participating mass ratios are expressed as percentages.

The cumulative sums of the participating mass ratios for all Modes up to Mode n are printed with the individual values for Mode n . This provides a simple measure of how many Modes are required to achieve a given level of accuracy for ground-acceleration loading.

If all eigen-Modes of the structure are present, the participating mass ratio for each of the three Acceleration Loads should generally be 100%. However, this may not be the case in the presence of certain types of Constraints where symmetry conditions prevent some of the mass from responding to translational accelerations.

Response-Spectrum Analysis

The dynamic equilibrium equations associated with the response of a structure to ground motion are given by:

$$\mathbf{K} \mathbf{u}(t) + \mathbf{C} \dot{\mathbf{u}}(t) + \mathbf{M} \ddot{\mathbf{u}}(t) = \mathbf{m}_x \ddot{u}_{gx}(t) + \mathbf{m}_y \ddot{u}_{gy}(t) + \mathbf{m}_z \ddot{u}_{gz}(t)$$

where \mathbf{K} is the stiffness matrix; \mathbf{C} is the proportional damping matrix; \mathbf{M} is the diagonal mass matrix; \mathbf{u} , $\dot{\mathbf{u}}$, and $\ddot{\mathbf{u}}$ are the relative displacements, velocities, and accel-

erations with respect to the ground; m_x , m_y , and m_z are the unit Acceleration Loads; and \ddot{u}_{gx} , \ddot{u}_{gy} , and \ddot{u}_{gz} are the components of uniform ground acceleration.

Response-spectrum analysis seeks the likely maximum response to these equations rather than the full time history. The earthquake ground acceleration in each direction is given as a digitized response-spectrum curve of pseudo-spectral acceleration response versus period of the structure.

Even though accelerations may be specified in three directions, only a single, positive result is produced for each response quantity. The response quantities include displacements, forces, and stresses. Each computed result represents a statistical measure of the likely maximum magnitude for that response quantity. The actual response can be expected to vary within a range from this positive value to its negative.

No correspondence between two different response quantities is available. No information is available as to when this extreme value occurs during the seismic loading, or as to what the values of other response quantities are at that time.

Response-spectrum analysis is performed using mode superposition (Wilson and Button, 1982). Modes may have been computed using eigenvector analysis or Ritz-vector analysis. Ritz vectors are recommended since they give more accurate results for the same number of Modes.

Any number of response-spectrum analyses can be defined in a single model. You assign a unique label to each response-spectrum analysis case. Each case can differ in the acceleration spectra applied, the modal analysis case used to generate the modes, and in the way that results are combined.

The following subtopics describe in more detail the parameters that you use to define each case.

Local Coordinate System

Each response-spectrum case has its own **response-spectrum local coordinate system** used to define the directions of ground acceleration loading. The axes of this local system are denoted 1, 2, and 3. By default these correspond to the global X, Y, and Z directions, respectively.

You may change the orientation of the local coordinate system by specifying a coordinate angle, **ang** (the default is zero). The local 3 axis is always the same as the vertical global Z axis. The local 1 and 2 axes coincide with the X and Y axes if angle **ang** is zero. Otherwise, **ang** is the angle in the horizontal plane from the global X

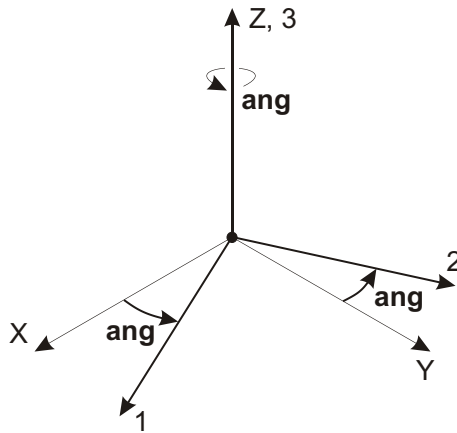


Figure 22
Definition of Response Spectrum Local Coordinate System

axis to the local 1 axis, measured counterclockwise when viewed from above. This is illustrated in Figure 22 (page 79).

Response-Spectrum Functions

A Response-spectrum Function is a series of digitized pairs of structural-period and corresponding pseudo-spectral acceleration values. You may define any number of Functions, assigning each one a unique label. You may scale the acceleration values whenever the Function is used.

Specify the pairs of period and acceleration values as:

$$t_0, f_0, t_1, f_1, t_2, f_2, \dots, t_n, f_n$$

where $n + 1$ is the number of pairs of values given. All values for the period and acceleration must be zero or positive. These pairs must be specified in order of increasing period.

Response-Spectrum Curve

The response-spectrum curve for a given direction is defined by digitized points of pseudo-spectral acceleration response versus period of the structure. The shape of the curve is given by specifying the name of a Response-spectrum Function.

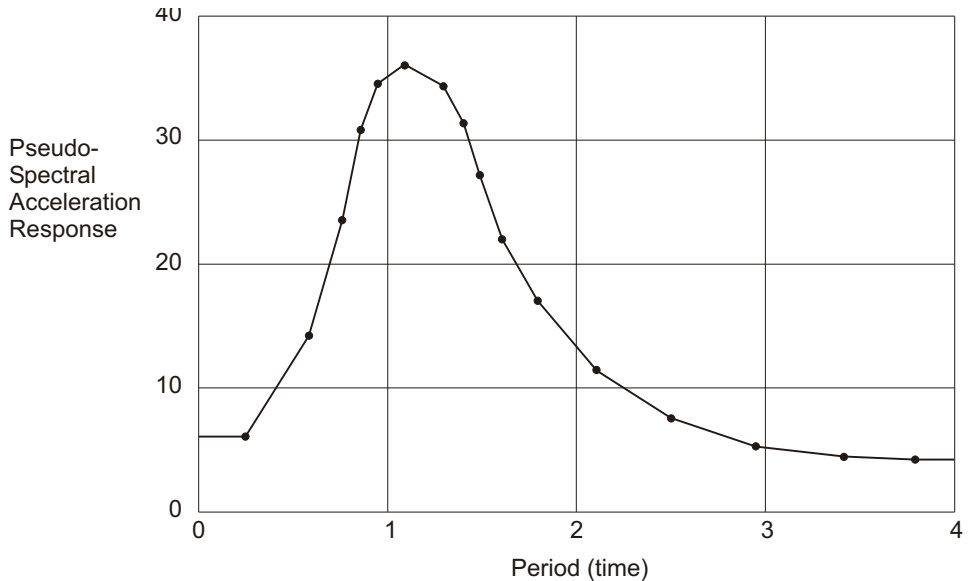


Figure 23
Digitized Response-Spectrum Curve

If no Function is specified, a constant function of unit acceleration value for all structural periods is assumed.

The pseudo spectral acceleration response of the Function may be scaled by the factor **sf**. After scaling, the acceleration values must be in consistent units. See Figure 23 (page 80).

The response-spectrum curve chosen should reflect the damping that is present in the structure being modeled. Note that the damping is inherent in the response-spectrum curve itself. It is not affected by the damping ratio, **damp**, used for the CQC or GMC method of modal combination, although normally these two damping values should be the same.

If the response-spectrum curve is not defined over a period range large enough to cover the modes calculated in the modal analysis case, the curve is extended to larger and smaller periods using a constant acceleration equal to the value at the nearest defined period.

Modal Combination

For a given direction of acceleration, the maximum displacements, forces, and stresses are computed throughout the structure for each of the Vibration Modes. These modal values for a given response quantity are combined to produce a single, positive result for the given direction of acceleration using one of the following methods. Use the parameter **modc** to specify which method to use.

CQC Method

Specify **modc** = CQC to combine the modal results by the Complete Quadratic Combination technique described by Wilson, Der Kiureghian, and Bayo (1981). This is the default method of modal combination.

The CQC method takes into account the statistical coupling between closely-spaced Modes caused by modal damping. You may specify a CQC damping ratio, **damp**, measured as a fraction of critical damping: $0 < \text{damp} < 1$. This should reflect the damping that is present in the structure being modeled. Note that the value of **damp** does not affect the response-spectrum curve, which is developed independently for an assumed value of structural damping. Normally these two damping values should be the same.

If the damping is zero, this method degenerates to the SRSS method.

GMC Method

Specify **modc** = GMC to combine the modal results by the General Modal Combination technique. This is the same as the complete modal combination procedure described by Equation 3.31 in Gupta (1990). The GMC method takes into account the statistical coupling between closely-spaced Modes similarly to the CQC method, but also includes the correlation between modes with rigid-response content.

As with the CQC method, you may specify a GMC damping ratio, **damp**, such that $0 < \text{damp} < 1$. Greater damping increases the coupling between closely-spaced modes.

In addition, the GMC method requires you to specify two frequencies, **f1** and **f2**, which define the rigid-response content of the ground motion. These must satisfy $0 < \text{f1} < \text{f2}$. The rigid-response parts of all modes are assumed to be perfectly correlated.

The GMC method assumes no rigid response below frequency **f1**, full rigid response above frequency **f2**, and an interpolated amount of rigid response for frequencies between **f1** and **f2**.

Frequencies **f1** and **f2** are properties of the seismic input, not of the structure. Gupta defines **f1** as:

$$\mathbf{f1} = \frac{S_{A_{\max}}}{2 S_{v_{\max}}}$$

where $S_{A_{\max}}$ is the maximum spectral acceleration and $S_{v_{\max}}$ is the maximum spectral velocity for the ground motion considered. The default value for **f1** is unity.

Gupta defines **f2** as:

$$\mathbf{f2} = \frac{1}{3} \mathbf{f1} - \frac{2}{3} f_r$$

where f_r is the rigid frequency of the seismic input, i.e., that frequency above which the spectral acceleration is essentially constant and equal to the value at zero period (infinite frequency).

Others have defined **f2** as:

$$\mathbf{f2} = f_r$$

The default value for **f2** is zero, indicating infinite frequency. For the default value of **f2**, the GMC method gives results similar to the CQC method.

SRSS Method

Specify **modc** = SRSS to combine the modal results by taking the square root of the sum of their squares. This method does not take into account any coupling of Modes as do the CQC and GMC methods.

Absolute Sum Method

Specify **modc** = ABS to combine the modal results by taking the sum of their absolute values. This method is usually over-conservative.

Directional Combination

For each displacement, force, or stress quantity in the structure, modal combination produces a single, positive result for each direction of acceleration. These direc-

tional values for a given response quantity are combined to produce a single, positive result. Use the directional combination scale factor, **dirf**, to specify which method to use.

SRSS Method

Specify **dirf** = 0 to combine the directional results by taking the square root of the sum of their squares. This method is invariant with respect to coordinate system, i.e., the results do not depend upon your choice of coordinate system when the given response-spectrum curves are the same. This is the recommended method for directional combination, and is the default.

Absolute Sum Method

Specify **dirf** = 1 to combine the directional results by taking the sum of their absolute values. This method is usually over-conservative.

Scaled Absolute Sum Method

Specify $0 < \mathbf{dirf} < 1$ to combine the directional results by the scaled absolute sum method. Here, the directional results are combined by taking the maximum, over all directions, of the sum of the absolute values of the response in one direction plus **dirf** times the response in the other directions.

For example, if **dirf** = 0.3, the spectral response, R , for a given displacement, force, or stress would be:

$$R = \max(\bar{R}_1, \bar{R}_2, \bar{R}_3)$$

where:

$$\bar{R}_1 = R_1 + 0.3(R_2 + R_3)$$

$$\bar{R}_2 = R_2 + 0.3(R_1 + R_3)$$

$$\bar{R}_3 = R_3 + 0.3(R_1 + R_2)$$

and R_1 , R_2 , and R_3 are the modal-combination values for each direction.

The results obtained by this method will vary depending upon the coordinate system you choose. Results obtained using **dirf** = 0.3 are comparable to the SRSS method (for equal input spectra in each direction), but may be as much as 8% un-conservative or 4% over-conservative, depending upon the coordinate system. Larger values of **dirf** tend to produce more conservative results.

Response-Spectrum Analysis Results

Certain information about each response-spectrum analysis case is available for printing from the SAP2000 graphical interface. This information is described in the following subtopics.

Damping and Accelerations

The modal damping and the ground accelerations acting in each direction are given for every Mode. The damping value printed for each Mode is just the specified CQC or GMC damping ratio, **damp**.

The accelerations printed for each Mode are the actual values as interpolated at the modal period from the response-spectrum curves after scaling by the specified value of **sf**. The accelerations are always referred to the local axes of the response-spectrum analysis. They are identified in the output as U1, U2, and U3.

Modal Amplitudes

The response-spectrum modal amplitudes give the multipliers of the mode shapes that contribute to the displaced shape of the structure for each direction of Acceleration Load. For a given Mode and a given direction of acceleration, this is the product of the modal participation factor and the response-spectrum acceleration, divided by the eigenvalue, ω^2 , of the Mode.

The acceleration directions are always referred to the local axes of the response-spectrum analysis. They are identified in the output as U1, U2, and U3.

For more information:

- See the previous Subtopic “Damping and Acceleration” for the definition of the response-spectrum accelerations.
- See Topic “Modal Analysis Results” (page 75) in this chapter for the definition of the modal participation factors and the eigenvalues.

Base Reactions

The base reactions are the total forces and moments about the global origin required of the supports (restraints and springs) to resist the inertia forces due to response-spectrum loading. These are printed for each individual Mode after performing only directional combination, and then summed for all Modes after performing modal combination and directional combination.

The reaction forces and moments are always referred to the local axes of the response-spectrum analysis. They are identified in the output as F1, F2, F3, M1, M2, and M3.

Chapter IX

Bibliography

AASHTO, 1992

Standard Specifications for Highways Bridges, 15th Edition, The American Association of State Highway and Transportation Officials, Inc., Washington, D.C.

AASHTO, 2004

LRFD Bridge Design Specifications, 3rd Edition, The American Association of State Highway and Transportation Officials, Inc., Washington, D.C.

ACI, 2002

Building Code Requirements for Structural Concrete (ACI 318-02) and Commentary (ACI 318R-02), American Concrete Institute, Farmington Hills, Mich.

AISC, 1999

Load & Resistance Factor Design for Structural Steel Buildings, American Institute of Steel Construction, Chicago, Ill.

K. J. Bathe, 1982

Finite Element Procedures in Engineering Analysis, Prentice-Hall, Englewood Cliffs, N.J.

K. J. Bathe and E. L. Wilson, 1976

Numerical Methods in Finite Element Analysis, Prentice-Hall, Englewood Cliffs, N.J.

K. J. Bathe, E. L. Wilson, and F. E. Peterson, 1974

SAP IV — A Structural Analysis Program for Static and Dynamic Response of Linear Systems, Report No. EERC 73-11, Earthquake Engineering Research Center, University of California, Berkeley.

J. L. Batoz and M. B. Tahar, 1982

“Evaluation of a New Quadrilateral Thin Plate Bending Element,” *International Journal for Numerical Methods in Engineering*, Vol. 18, pp. 1655–1677.

Caltrans, 1995

Bridge Design Specifications Manual, as amended to December 31, 1995, State of California, Department of Transportation, Sacramento, Calif.

R. D. Cook, D. S. Malkus, and M. E. Plesha, 1989

Concepts and Applications of Finite Element Analysis, 3rd Edition, John Wiley & Sons, New York, N.Y.

R. D. Cook and W. C. Young, 1985

Advanced Mechanics of Materials, Macmillan, New York, N.Y.

A. K. Gupta, 1990

Response Spectrum Method in Seismic Analysis and Design of Structures, Blackwell Scientific Publications, Cambridge, Mass.

J. P. Hollings and E. L. Wilson, 1977

3–9 Node Isoparametric Planar or Axisymmetric Finite Element, Report No. UC SEM 78-3, Division of Structural Engineering and Structural Mechanics, University of California, Berkeley.

A. Ibrahimbegovic and E. L. Wilson, 1989

“Simple Numerical Algorithms for the Mode Superposition Analysis of Linear Structural Systems with Nonproportional Damping,” *Computers and Structures*, Vol. 33, No. 2, pp. 523–531.

A. Ibrahimbegovic and E. L. Wilson, 1991

“A Unified Formulation for Triangular and Quadrilateral Flat Shell Finite Elements with Six Nodal Degrees of Freedom,” *Communications in Applied Numerical Methods*, Vol. 7, pp. 1–9.

L. E. Malvern, 1969

Introduction to the Mechanics of a Continuous Medium, Prentice-Hall, Englewood Cliffs, N.J.

S. Nagarajaiah, A. M. Reinhorn, and M. C. Constantinou, 1991

3D-Basis: Nonlinear Dynamic Analysis of Three-Dimensional Base Isolated Structures: Part II, Technical Report NCEER-91-0005, National Center for Earthquake Engineering Research, State University of New York at Buffalo, Buffalo, N. Y.

Y. J. Park, Y. K. Wen, and A. H-S. Ang, 1986

“Random Vibration of Hysteretic Systems under Bi-Directional Ground Motions,” *Earthquake Engineering and Structural Dynamics*, Vol. 14.

R. J. Roark and W. C. Young, 1975

Formulas for Stress and Strain. 5th Edition, McGraw-Hill, New York, N.Y.

R. L. Taylor and J. C. Simo, 1985

“Bending and Membrane Elements for Analysis of Thick and Thin Shells,” *Proceedings of the NUMETA 1985 Conference*, Swansea, Wales.

K. Terzaghi and R. B. Peck, 1967

Soil Mechanics in Engineering Practice, 2nd Edition, John Wiley & Sons, New York, N.Y.

S. Timoshenko and S. Woinowsky-Krieger, 1959

Theory of Plates and Shells, 2nd Edition, McGraw-Hill, New York, N.Y.

Y. K. Wen, 1976

“Method for Random Vibration of Hysteretic Systems,” *Journal of the Engineering Mechanics Division*, ASCE, Vol. 102, No. EM2.

D. W. White and J. F. Hajjar, 1991

“Application of Second-Order Elastic Analysis in LRFD: Research to Practice,” *Engineering Journal*, ACI, Vol. 28, No. 4, pp. 133–148.

E. L. Wilson, 1970

SAP — A General Structural Analysis Program, Report No. UC SESM 70-20, Structural Engineering Laboratory, University of California, Berkeley.

E. L. Wilson, 1972

SOLID SAP — A Static Analysis Program for Three Dimensional Solid Structures, Report No. UC SESM 71-19, Structural Engineering Laboratory, University of California, Berkeley.

E. L. Wilson, 1985

“A New Method of Dynamic Analysis for Linear and Non-Linear Systems,” *Finite Elements in Analysis and Design*, Vol. 1, pp. 21–23.

E. L. Wilson, 1993

“An Efficient Computational Method for the Base Isolation and Energy Dissipation Analysis of Structural Systems,” ATC17-1, *Proceedings of the Seminar on Seismic Isolation, Passive Energy Dissipation, and Active Control*, Applied Technology Council, Redwood City, Calif.

E. L. Wilson and M. R. Button, 1982

“Three Dimensional Dynamic Analysis for Multicomponent Earthquake Spectra,” *Earthquake Engineering and Structural Dynamics*, Vol. 10.

E. L. Wilson, A. Der Kiureghian, and E. P. Bayo, 1981

“A Replacement for the SRSS Method in Seismic Analysis,” *Earthquake Engineering and Structural Dynamics*, Vol. 9.

E. L. Wilson and I. J. Tetsuji, 1983

“An Eigensolution Strategy for Large Systems,” *Computers and Structures*, Vol. 16.

E. L. Wilson, M. W. Yuan, and J. M. Dickens, 1982

“Dynamic Analysis by Direct Superposition of Ritz Vectors,” *Earthquake Engineering and Structural Dynamics*, Vol. 10, pp. 813–823.

E. L. Wilson, 1998

The Three Dimensional Dynamic Analysis of Structures With Emphasis on Earthquake Engineering, 2nd Edition, Computers and Structures Inc., Berkeley, California.

V. Zayas and S. Low, 1990

“A Simple Pendulum Technique for Achieving Seismic Isolation,” *Earthquake Spectra*, Vol. 6, No. 2.

O. C. Zienkiewicz and R. L. Taylor, 1989

The Finite Element Method, 4th Edition, Vol. 1, McGraw-Hill, London.

O. C. Zienkiewicz and R. L. Taylor, 1991

The Finite Element Method, 4th Edition, Vol. 2, McGraw-Hill, London.

Index

B

Beam

see Frame element

C

Column

see Frame element

Coordinate directions, 8

Coordinate systems, 7

Diaphragm constraint, 9

Frame element, 9,14

global, 8

horizontal directions, 8

local, 9

overview, 7

Shell element, 9

upward direction, 8

vertical direction, 8

X-Y-Z, 8

F

Frame element, 11

clear length, 24

connections, 24

connectivity, 13

coordinate angle, 15

coordinate system, 14

default orientation, 15

degrees of freedom, 14

end I, 13

end J, 13

end offsets, 24,27

end releases, 25

internal forces, 24

joints, 13

longitudinal axis, 15

mass, 27

overview, 12

section properties

see Frame section

support faces, 24

truss behavior, 14

vertical, 15

Frame section, 17

angle section, 18

area, 17

box section, 18

channel section, 18

database file, 20

database section, 20

double-angle section, 18

general section, 18

I section, 18
local coordinate system, 17
material properties, 17
moment of inertia, 17
pipe section, 18
rectangular section, 18
shape, 18
shear area, 17
shear deformation, 17
solid circular section, 18
T section, 18
torsional constant, 17

T

Truss

see Frame element

Typographical conventions, 2