SAP2000®

Integrated
Finite Element Analysis
and
Design of Structures

ANALYSIS REFERENCE



Computers and Structures, Inc.
Berkeley, California, USA

Version 7.0
Revised October 1998

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*

© Copyright Computers and Structures, Inc., 1978–1998.
The CSI Logo is a registered trademark of Computers and Structures, Inc.
SAP2000 is a registered trademark of Computers and Structures, Inc.
Windows is a registered trademark of Microsoft Corporation.

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 **1**

SAP2000 Analysis Features 1
Structural Analysis and Design 2
About This Manual 3
Topics 3
Typographical Conventions 4
Bold for Definitions 4
Bold for Variable Data 4
Italics for Mathematical Variables 4
Italics for Emphasis 4
All Capitals for Literal Data 5
Capitalized Names 5
Bibliographic References 5

Chapter II Labels, Arrays, and Generation **7**

Overview 8
Labels 8
Label Increments 9
Regular Arrays 10
Single Label 11
One-dimensional Regular Arrays 11
Two-dimensional Regular Arrays 12
Three-dimensional Regular Arrays 13
Generation 14
Joints 14
Elements, Constraints, and Welds 15

Deletion 18
Assignment 19

Chapter III Coordinate Systems 21

Overview. 22
Global Coordinate System. 22
Upward and Horizontal Directions 23
Defining Coordinate Systems 23
Vector Cross Product 23
Defining the Three Axes Using Two Vectors 24
Local Coordinate Systems. 24
Alternate Coordinate Systems 26
Cylindrical and Spherical Coordinates 28

Chapter IV Joint Coordinates 31

Overview. 32
Joint Definition 32
One-dimensional Joint Generation 33
One-dimensional Joint Array Specification 33
One-dimensional Joint Definition 33
One-dimensional Linear Generation 34
One-dimensional Cylindrical Generation. 34
Two-dimensional Joint Generation 36
Two-dimensional Joint Array Specification 37
Two-dimensional Joint Definition 37
Two-dimensional Linear Generation 38
Two-dimensional Frontal Generation. 38
Two-dimensional Edge Generation. 40
Three-dimensional Joint Generation. 43
Three-dimensional Joint Array Specification. 43
Three-dimensional Joint Definition. 44
Three-dimensional Linear Generation 45
Three-dimensional Frontal Generation 46
Three-dimensional Edge Generation 46
Variable Joint Spacing. 47
One Dimension 48
Two Dimensions 48
Three Dimensions. 49
Joint Definition in Polar Coordinates 50
Cylindrical Coordinates. 50

Spherical Coordinates 52
Variable Joint Spacing in Polar Coordinates 53
Joint Definition in Alternate Coordinate Systems 54

Chapter V Joint Patterns 57

Overview 58
Pattern Definition 59
Gradient or Hydrostatic Specification 59
One-dimensional Pattern-value Generation 60
Joint Array Specification 61
One-dimensional Addition 62
One-dimensional Gradient Addition 62
One-dimensional Linear Mapping 63
Two-dimensional Pattern-value Generation 63
Joint Array Specification 64
Two-dimensional Addition 64
Two-dimensional Gradient Addition 65
Two-dimensional Linear Mapping 65
Two-dimensional Frontal Mapping 66
Two-dimensional Edge Mapping 66
Three-dimensional Pattern-value Generation 67
Joint Array Specification 67
Three-dimensional Addition 68
Three-dimensional Gradient Addition 69
Three-dimensional Linear Mapping 69
Three-dimensional Frontal Mapping 70
Three-dimensional Edge Mapping 71
Variable Pattern-value Increments 72
One Dimension 73
Two Dimensions 73
Three Dimensions 74

Chapter VI Joints and Degrees of Freedom 75

Overview 76
Modeling Considerations 77
Local Coordinate System 78
Advanced Local Coordinate System 79
Reference Vectors 79
Defining the Axis Reference Vector 80
Defining the Plane Reference Vector 80
Determining the Local Axes from the Reference Vectors 81

Joint Coordinate Angles 82
Degrees of Freedom 83
Available and Unavailable Degrees of Freedom 85
Restrained Degrees of Freedom 86
Constrained Degrees of Freedom 86
Active Degrees of Freedom 86
Null Degrees of Freedom 88
Restraints and Reactions 88
Springs 89
Masses 91
Force Load 93
Restraint Displacement Load 93
Spring Displacement Load 95
Degree of Freedom Output 97
Joint Mass Output 98
Displacement and Acceleration Output 100
Force Output 102
Applied Loads 103
Inertial Loads 104
Stiffness Forces 106
Spring Forces 106
Nllink Forces 106
Restraint Forces (Reactions) 106
Constraint Forces 107
P-Delta Moments 107
Global Force Balance Output 107

Chapter VII Constraints and Welds 113

Overview 114
Body Constraint 115
Joint Connectivity 115
Local Coordinate System 115
Constraint Equations 115
Plane Definition 116
Diaphragm Constraint 117
Joint Connectivity 117
Local Coordinate System 117
Constraint Equations 118
Plate Constraint 119
Joint Connectivity 119
Local Coordinate System 119

Constraint Equations . . . 119
Axis Definition . . . 120
Rod Constraint . . . 120
Joint Connectivity . . . 121
Local Coordinate System . . . 122
Constraint Equations . . . 122
Beam Constraint . . . 122
Joint Connectivity . . . 123
Local Coordinate System . . . 123
Constraint Equations . . . 123
Equal Constraint . . . 124
Joint Connectivity . . . 124
Local Coordinate System . . . 125
Selected Degrees of Freedom . . . 125
Constraint Equations . . . 125
Local Constraint . . . 127
Joint Connectivity . . . 127
No Local Coordinate System . . . 128
Selected Degrees of Freedom . . . 128
Constraint Equations . . . 128
Welds . . . 131
Automatic Master Joints . . . 132
Stiffness, Mass, and Loads . . . 132
Local Coordinate Systems . . . 133
Constraint Output . . . 133

Chapter VIII Material Properties 135

Overview . . . 136
Local Coordinate System . . . 136
Stresses and Strains . . . 137
Isotropic Materials . . . 138
Orthotropic Materials . . . 139
Anisotropic Materials . . . 140
Temperature-Dependent Properties . . . 141
Element Material Temperature . . . 142
Mass Density . . . 142
Weight Density . . . 143
Design-Type Indicator . . . 143

Chapter IX The Frame Element 145

Overview 146
Joint Connectivity 147
Degrees of Freedom 147
Local Coordinate System. 148
Longitudinal Axis 1 148
Default Orientation 148
Coordinate Angle 149
Advanced Local Coordinate System 149
Reference Vector 150
Determining Transverse Axes 2 and 3 153
Section Properties 153
Local Coordinate System 154
Material Properties 154
Geometric Properties and Section Stiffnesses. 154
Shape Type 155
Automatic Section Property Calculation 157
Section Property Database Files. 157
Additional Mass and Weight 159
Non-prismatic Sections 159
End Offsets 162
Clear Length. 163
Rigid-end Factor. 164
Effect upon Non-prismatic Elements 164
Effect upon Internal Force Output. 165
Effect upon End Releases 165
End Releases 165
Unstable End Releases 165
Effect of End Offsets 166
Effect upon Prestress Load 167
Mass 167
Self-Weight Load. 168
Gravity Load 168
Concentrated Span Load 169
Distributed Span Load 169
Loaded Length. 169
Load Intensity 170
Projected Loads 173
Temperature Load 173
Prestress Load 174
Prestressing Cables 174

Prestress Load 174
Effect upon P-Delta Analysis 175
Internal Force Output. 176
Effect of End Offsets 177
Internal Forces in the Output File 179
Joint Force Output 179

Chapter X The Shell Element 181

Overview 182
Joint Connectivity 183
Degrees of Freedom 185
Local Coordinate System. 186
Normal Axis 3 186
Default Orientation 187
Element Coordinate Angle 187
Advanced Local Coordinate System 187
Reference Vector 189
Determining Tangential Axes 1 and 2. 190
Section Properties 191
Section Type. 191
Thickness Formulation 191
Material Properties 192
Material Angle. 193
Thickness 194
Mass 194
Self-Weight Load. 195
Gravity Load 195
Uniform Load 195
Surface Pressure Load 196
Temperature Load 197
Internal Force and Stress Output 198
Joint Force Output 203

Chapter XI The Plane Element 205

Overview 206
Joint Connectivity 206
Degrees of Freedom 208
Local Coordinate System. 210
Stresses and Strains. 211
Material Properties 211

Material Angle 212
Thickness 212
Mass 213
Self-Weight Load 213
Gravity Load 214
Surface Pressure Load 214
Pore Pressure Load 214
Temperature Load 215
Stress Output 215
Joint Force Output 216

Chapter XII The Asolid Element 219

Overview 220
Joint Connectivity 221
Degrees of Freedom 224
Local Coordinate System 224
Stresses and Strains 224
Material Properties 225
Material Angle 226
Arc and Thickness 227
Mass 227
Self-Weight Load 228
Gravity Load 228
Surface Pressure Load 229
Pore Pressure Load 229
Temperature Load 229
Rotate Load 230
Stress Output 230
Joint Force Output 231

Chapter XIII The Solid Element 233

Overview 234
Joint Connectivity 234
Degrees of Freedom 236
Local Coordinate System 236
Stresses and Strains 236
Material Properties 236
Material Angles 237

Incompatible Bending Modes 237
Mass 238
Self-Weight Load. 239
Gravity Load 239
Surface Pressure Load 239
Pore Pressure Load 240
Temperature Load 240
Stress Output 240
Joint Force Output 241

Chapter XIV The Nllink Element 243

Overview 244
Joint Connectivity 245
Zero-Length Elements 245
Degrees of Freedom 246
Local Coordinate System. 246
Longitudinal Axis 1 247
Default Orientation 247
Coordinate Angle 247
Advanced Local Coordinate System 249
Axis Reference Vector 249
Plane Reference Vector 250
Determining Transverse Axes 2 and 3 251
Internal Deformations 253
Nlprop Properties. 254
Local Coordinate System 255
Internal Nonlinear Springs 255
Spring Force-Deformation Relationships 256
Element Internal Forces 257
Linear Force-Deformation Relationships 258
Linear Effective Stiffness 259
Linear Effective Damping 261
Nonlinear Properties. 262
Nonlinear Deformation Loads 271
Mass 272
Self-Weight Load. 274
Gravity Load 274
Internal Force and Deformation Output 275
Joint Force Output 277

Chapter XV Load Cases 279

Overview . . . 280
Load Cases for Analysis . . . 281
Defining Load Cases . . . 281
Coordinate Systems and Load Components . . . 282
Force Load . . . 282
Restraint Displacement Load . . . 283
Spring Displacement Load . . . 283
Self-Weight Load . . . 283
Gravity Load . . . 284
Concentrated Span Load . . . 284
Distributed Span Load . . . 285
Prestress Load . . . 285
Uniform Load . . . 285
Surface Pressure Load . . . 286
Pore Pressure Load . . . 286
Temperature Load . . . 287
Reference Temperature . . . 288
Rotate Load . . . 289

Chapter XVI Static and Dynamic Analysis 291

Overview . . . 292
Analysis Cases . . . 292
Static Analysis . . . 293
Harmonic Steady-State Analysis . . . 294
Acceleration Loads . . . 295
Eigenvector Analysis . . . 296
- Number of Modes . . . 297
- Frequency Range . . . 297
- Convergence Tolerance . . . 298

Ritz-vector Analysis . . . 299
- Number of Modes . . . 300
- Starting Load Vectors . . . 301
- Number of Generation Cycles . . . 302

Modal Analysis Output . . . 303
- Periods and Frequencies . . . 303
- Participation Factors . . . 304
- Participating Mass Ratios . . . 304
- Static and Dynamic Load Participation Ratios . . . 306

Functions . . . 309
Response-Spectrum Analysis . . . 310
Local Coordinate System . . . 311
Response-Spectrum Curve . . . 311
Modal Combination . . . 313
Directional Combination . . . 315
Response-Spectrum Analysis Output . . . 317
Damping and Accelerations . . . 317
Modal Amplitudes . . . 317
Modal Correlation Factors . . . 319
Base Reactions . . . 319
Time-History Analysis . . . 319
Loading . . . 320
Mode Superposition . . . 323
Modal Damping . . . 324
Time Steps . . . 325
Initial Conditions . . . 325
Analysis Results . . . 327
Nonlinear Time-History Analysis . . . 328
Nllink Effective Stiffness . . . 328
Mode Superposition . . . 329
Modal Damping . . . 330
Iterative Solution . . . 330
Static Period . . . 333
Combos . . . 333

Chapter XVII P-Delta Analysis 337

Overview . . . 338
Geometric Nonlinearity . . . 339
The P-Delta Effect . . . 340
Equilibrium Equations . . . 343
P-Delta Axial Forces . . . 343
Directly Specified Axial Forces . . . 344
P-Delta Load Combination . . . 345
Iterative Analysis . . . 345
Convergence Criterion . . . 346
Maximum Number of Iterations . . . 346
Convergence Failure . . . 347
Frame Element . . . 347
Small Deflections . . . 347
Cubic Deflected Shape . . . 347

Computed P-Delta Axial Forces 348
Prestress 349
Effect upon Other Analyses 349
Dynamic Analyses 349
Harmonic Steady-State Analysis 350
Bridge Moving-Load Analysis 350
Buckling 350
Detection of Buckling 351
Estimating the Buckling Load 351
Local Buckling 351
Practical Application 352
Preliminary Linear Analysis 352
Building Structures 352
Cable Structures 353
Guyed Towers 355

Chapter XVIII Bridge Analysis 357

Overview 358
Modeling the Bridge Structure 359
Frame Elements 359
Supports 360
Bearings and Expansion Joints 361
Other Element Types 361
Roadways and Lanes 363
Roadways 363
Lanes 363
Eccentricities 364
Modeling Guidelines 364
Examples 365
Spatial Resolution 367
Load and Output Points 367
Resolution 369
Modeling Guidelines 369
Influence Lines 370
Vehicles 372
Direction of Loads 372
Application of Loads 372
General Vehicle 373
Standard Vehicles 377
Vehicle Classes 383
Moving Load Cases 383

Example 1 — AASHTO HS Loading 384
Example 2 — AASHTO HL Loading 386
Example 3 — Caltrans Permit Loading 387
Example 4 — Restricted Caltrans Permit Loading 389
Influence Line Tolerance 391
Exact and Quick Response Calculation 391
Moving Load Response Control 392
Correspondence 392
Computational Considerations 393

Chapter XIX The Output Files 395

Overview 396
The Input Echo (.EKO) File 396
The Analysis Log (.LOG) File 397
The Results Output (.OUT) File 399
Element Joint Force Output 402
Joint and Element Output Control 405
Joints 412
Frame and Nllink Elements 412
Shell Elements 413
Plane, Asolid, and Solid Elements 413
Pagination Control 414
Pagination by Lines 414
Pagination by Sections 414

Chapter XX References 415

Chapter I

Introduction

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

Basic Topics for All Users

- SAP2000 Analysis Features
- Structural Analysis and Design
- About This Manual
- Topics
- Typographical Conventions
- Bibliographic References

SAP2000 Analysis Features

The SAP2000 structural analysis program offers the following features:

- Static and dynamic analysis
- Linear and nonlinear analysis
- Dynamic seismic analysis and static pushover analysis

- Vehicle live-load analysis for bridges
- P-Delta analysis
- Frame and shell structural elements, including beam-column, truss, membrane, and plate behavior
- Two- and three-dimensional and axisymmetric solid elements
- Nonlinear link and spring elements
- Multiple coordinate systems
- Many types of constraints
- A wide variety of loading options
- Alpha-numeric labels
- Large capacity
- Highly efficient and stable solution algorithms

These features, and many more, make SAP2000 the state-of-the-art in structural analysis programs.

Structural Analysis and Design

The following general steps are required to analyze and design a structure using SAP2000:

1. Create or modify a model that numerically defines the geometry, properties, loading, and analysis parameters for the structure
2. Perform an analysis of the model
3. Review the results of the analysis
4. Check the design of the structure

This is usually an iterative process that may involve many cycles of the above sequence of steps. All of these steps can be performed seamlessly using the SAP2000 graphical user interface.

A few advanced analysis options are not yet available through the SAP2000 graphical user interface. To access these features, you must edit an input data text file. This file can then be brought into the graphical user interface for analysis, display, and design. However, most users will find the SAP2000 graphical user interface sufficient for their needs.

About This Manual

This manual describes the theoretical concepts behind the modeling and analysis features offered by the SAP2000 structural analysis program. The focus of this manual is on the analysis portion of the program. It is imperative that you read this manual and understand the assumptions and procedures used by the program before attempting to use the analysis features.

The graphical user interface and the design modules are described in separate manuals. Static Pushover Analysis capabilities are descibed in the *SAP2000 Detailed Tutorial including Pushover Analysis* and in the on-line help feature of the graphical user interface.

See the *SAP2000 Getting Started* manual for a listing of all the manuals supplied with the program.

Topics

Each chapter of this manual is divided into topics and subtopics. All chapters begin with a list of topics covered. These are divided into two groups:

- Basic topics — recommended reading for all users
- Advanced topics — for users with specialized needs, and for all users as they become more familiar with the program.

Following the list of topics is an Overview which provides a summary of the chapter. Reading the Overview for every chapter will acquaint you with the full scope of the program.

The *SAP2000 Basic Analysis Reference* is a condensation of the basic topics covered in the present manual.

Typographical 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:

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

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 Case

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 “References” (page 415).

Chapter II

Labels, Arrays, and Generation

Labels are names that you assign to the various entities — such as joints, elements, materials, and loads — that define a structural model and its analysis. A regular array is group of labels that increment (differ from each other) in a regular fashion. Generation allows you to define large numbers of joints, elements, constraints, or welds using arrays and simple data specifications.

Basic Topics for All Users

- Overview
- Labels
- Label Increments
- Regular Arrays
- Generation
- Deletion
- Assignment

Overview

Every item that you define as part of the structural model or its analysis has an alpha-numeric label.

Certain types of entities, which are typically defined in large quantities, may be generated using simple data specifications. The generatable entities are:

- Joints
- Elements
- Constraints
- Welds

The remaining types of labeled entities used by the program must each be explicitly defined and cannot be generated:

- Alternate Coordinate Systems
- Patterns
- Materials, Frame Sections, Shell Sections, and Nlprops
- Loads
- Functions
- Specs and Histories
- Lanes, Vehicles, Classes, and Moving Loads
- Combos

A group of generatable entities may be referred to using a regular array, which is specified by giving the first and last label and the label increment for each of one, two, or three dimensions. Arrays may be used to generate and delete joints, elements, Constraints, and Welds; and to assign loads and properties to these entities.

Labels

Labels are names that you assign to the various entities that make up a structural model and its analysis. Such entities include joints, elements, Constraints, Materials, Loads, and analyses. A few entities, such as the Modes, are numbered automatically by the program.

Labels are alpha-numeric according to the following rules:

- They may contain from one to seven letters and/or numbers
- No spaces or other characters are permitted
- Uppercase and lowercase letters are treated the same, e.g., “A3” and “a3” are the same
- A label may not contain all zeroes
- Leading zeroes are ignored, e.g., “0A3” is the same as “A3”

Some examples of valid labels are:

```
      1
      2
    A03
    33B
123x567
 Corner
  STEEL
```

Some examples of invalid labels are:

```
      0
    000
   A_03
    33%
123+567
  -5001
ABC 123
```

The same label may be used for different types of entities without any conflict. For example, you may have a joint, a Frame element, a Material, and a Load Case, each with the label “1”.

Label Increments

For the purpose of generation, it is convenient to define a group of entities having labels that increment in a regular fashion. Some examples are:

```
1, 2, 3, 4 ...
A00, A05, A10, A15 ...
8TH, 9TH, 10TH, 11TH ...
1X1, 2X2, 3X3, 4X4 ...
9000, 8000, 7000, 6000 ...
```

The **label increment** is the constant difference between successive labels in such a series. The following rules apply to label increments:

- Only the numeric parts of a label may increment; the alphabetic parts of the labels must be unchanged
- Leading blanks are considered to be numeric and may increment
- The increment is always specified as a number, with zeroes at each position where a letter appears
- Increments may be negative

Thus the increments for the above examples are:

```
    1
    5
  100
  101
-1000
```

A series may continue to increment until one of the following occurs:

- A numeric part carries over into an alphabetic part
- A zero or negative number is generated
- Seven characters is exceeded

Thus the limiting labels for the above examples are:

```
9999999
    A95
99999TH
    9X9
   1000
```

Regular Arrays

The Regular Arrays described in this topic are used only in the input data text file. Skip this topic if you are preparing your model using the SAP2000 graphical user interface.

A **regular array**, or simply an **array**, is set of labels that increment in a regular fashion. The labels in an array must correspond to entities of the same type, such as joints or Frame elements. Regular arrays are used to refer to large numbers of items for the purposes of generation, deletion, and assignment.

The labels in a regular array may increment separately in one, two, or three dimensions. The **dimension** of an array refers to the number of fixed increment values that are used to describe the set of labels.

The main advantage of regular arrays is that they are easily described with just a few parameters. SAP2000 uses a simple, standardized format for specifying arrays that makes use of the following parameters:

- A starting label
- For each dimension, an ending label and a label increment

These concepts will be made clearer in the following subtopics.

Single Label

For the sake of generality, a single label may be considered to be a zero-dimensional regular array. It has a starting label, but no ending labels or increments.

One-dimensional Regular Arrays

A one-dimensional regular array is a set of labels that differ, one from the next, by a single increment value. For example, consider the following set of labels:

```
1 3 5 7 9 11 13
```

This set of labels can be specified by giving the starting label, "1", the ending label "13", and a label increment, "2".

In general, the following information is required to specify a one-dimensional regular array:

- The label, **a0**, at the start of the array
- The label, **a1**, at the end of the array
- The label increment, **ai1**, between any pair of successive labels

To say it in words, the array "goes from **a0** to **a1** by **ai1**."

The number of labels, n_1, in the array is given by:

$$n_1 = \frac{\mathbf{a1} - \mathbf{a0}}{\mathbf{ai1}} + 1$$

You must choose the starting and ending labels and the label increment such that n_1 is a whole, positive number. For example, an array that goes from 1 to 12 by 5 is not valid.

If the ending label is less than the starting label, the increment must be negative.

Two-dimensional Regular Arrays

A two-dimensional regular array is a set of labels that differ, one from the next, by two different increment values in two different directions. For example, consider the following set of labels:

```
101 103 105 107 109
201 203 205 207 209
301 303 305 307 309
```

This set of labels increments by "2" in the horizontal direction and by "100" in the vertical direction. It can be specified by giving the starting label "101" in one corner, the ending labels "109" and "301" at the two adjacent corners, and the two increments.

In general, the following information is required to specify a two-dimensional regular array:

- The label, **a0**, at the starting corner of the array
- The label, **a1**, at one ending corner of the array
- The label, **a2**, at the other ending corner of the array
- The label increment, **ai1**, between any pair of successive labels along side **a0**-**a1**
- The label increment, **ai2**, between any pair of successive labels along side **a0**-**a2**

You may choose any corner to be the starting corner. The two ending corners must be adjacent to the starting corner. Two corners are adjacent if they are the two ends of one side of the array.

To say it in words, the array "goes from **a0** to **a1** by **ai1** and to **a2** by **ai2**."

The number of labels, n_1 and n_2, along the two dimensions of the array are given by:

$$n_1 = \frac{\mathbf{a1} - \mathbf{a0}}{\mathbf{ai1}} + 1 \quad \text{and} \quad n_2 = \frac{\mathbf{a2} - \mathbf{a0}}{\mathbf{ai2}} + 1$$

You must choose the starting and ending labels and the label increments such that n_1 and n_2 are a whole, positive numbers. The total number of labels in the array is given by $n_1 n_2$.

The physical locations of the labeled items (joints, elements, etc.) do not have to correspond in any way to the layout of the array of the labels. For example, suppose six joints labeled "JOINT1" to "JOINT6" physically lie along a straight line. They may be identified by the following one-dimensional array from JOINT1 to JOINT6 by 1:

```
JOINT1 JOINT2 JOINT3 JOINT4 JOINT5 JOINT6
```

or by the following two-dimensional array from JOINT1 to JOINT3 by 1 and to JOINT4 by 3:

```
JOINT1 JOINT2 JOINT3
JOINT4 JOINT5 JOINT6
```

Three-dimensional Regular Arrays

A three-dimensional regular array is a set of labels that differ, one from the next, by three different increment values in three different directions. For example, consider the following set of labels:

```
111 112
121 122   211 212
131 132   221 222   311 312
          231 232   321 322   411 412
                    331 332   421 422
                              431 432
```

This set of labels increments by "1" in the horizontal direction, by "10" in the vertical direction, and by "100" in the third "out-of-plane" direction. It can be specified by giving the starting label "111" in one corner, the ending labels "112", "131", and "411" at the three adjacent corners, and the three increments.

In general, the following information is required to specify a three-dimensional regular array:

- The label, **a0**, at the starting corner of the array
- The label, **a1**, at one ending corner of the array
- The label, **a2**, at another ending corner of the array
- The label, **a3**, at the third ending corner of the array

- The label increment, **ai1**, between any pair of successive labels along side **a0-a1**
- The label increment, **ai2**, between any pair of successive labels along side **a0-a2**
- The label increment, **ai3**, between any pair of successive labels along side **a0-a3**

You may choose any corner to be the starting corner. The three ending corners must be adjacent to the starting corner. Two corners are said to be adjacent if they are on the same side (edge) of the array.

To say it in words, the array "goes from **a0** to **a1** by **ai1**, to **a2** by **ai2**, and to **a3** by **ai3**."

The number of labels, n_1, n_2, and n_3, along the three dimensions of the array are given by:

$$n_1 = \frac{\mathbf{a1} - \mathbf{a0}}{\mathbf{ai1}} + 1, \quad n_2 = \frac{\mathbf{a2} - \mathbf{a0}}{\mathbf{ai2}} + 1, \quad \text{and} \quad n_3 = \frac{\mathbf{a3} - \mathbf{a0}}{\mathbf{ai3}} + 1$$

You must choose the starting and ending labels and the label increments such that n_1, n_2, and n_3 are a whole, positive numbers. The total number of labels in the array is given by $n_1\, n_2\, n_3$.

Generation

Generation as described in this topic is used only in the input data text file. Skip this topic if you are preparing your model using the SAP2000 graphical user interface.

Generation is used to define a regular array of joints, elements, Constraints, or Welds.

Joints

Two distinct methods are available for generating joints:

- All joints in an array are simultaneously defined from specified data
- Joints at the corners or along the edges of an array are first defined; the remaining joints are generated with respect to these previously-defined joints

Complete details about joint generation are given in Chapter “Joint Coordinates” (page 31).

Elements, Constraints, and Welds

Elements, Constraints, and Welds are identical for the purposes of generation. Each element, Constraint, or Weld is defined by:

- A label
- A set of connected joints
- Various other properties and parameters

Elements, Constraints, and Welds will all be referred to as “elements” for the remainder of this discussion.

Generation defines a regular array of elements based on the definition of the starting element. The starting element is the element having the starting label in the array.

The starting element must have been previously defined or generated, and is unchanged by the generation. All other elements in the array are created if they do not exist, or are redefined if they already exist.

Each generated element has the same properties and parameters as the starting element. Only the element label and the connected joints differ.

The joint labels differ by joint increments that you specify. There is one joint increment for each dimension of the element array, except for the Frame and Nllink elements which permit two joint increments for each dimension of the array. This is described in more detail in the following.

One-dimensional Generation

The following information is required to specify one-dimensional element generation:

- The starting element label: **e0**
- The ending element label: **e1**
- The element label increment: **ei1**
- The joint label increment: **ji1**
- For the Frame and Nllink elements, the joint label increment: **ii1**

Element **e0** must already be defined. Suppose it is connected to joints **j1**, **j2**, **j3**, ..., **jn**. Then generated element **e0**+**ei1** will be connected to joints **j1**+**ji1**, **j2**+**ji1**, **j3**+**ji1**, ..., **jn**+**ji1**, and so on for the rest of the generated elements. Thus all joint labels for a generated element differ from those of the starting element by the same amount.

For a Frame or Nllink element, suppose that element **e0** is connected to joints **i** and **j**. Then generated element **e0**+**ei1** will be connected to joints **i1**+**ii1** and **j**+**ji1**, and so on for the rest of the generated elements.

The default for the joint label increment **ji1** is the element label increment **ei1**. The default for the joint label increment **ii1** is the joint label increment **ji1**. Thus it is often convenient to assign element labels that are consistent with the joint labels.

Two-dimensional Generation

The following information is required to specify two-dimensional element generation:

- The starting element label: **e0**
- The ending element labels: **e1** and **e2**
- The element label increments: **ei1** and **ei2**
- The joint label increments: **ji1** and **ji2**
- For the Frame and Nllink elements, the joint label increments: **ii1** and **ii2**

Element **e0** must already be defined. Suppose it is connected to joints **j1**, **j2**, **j3**, ..., **jn**. Then generated element **e0**+**ei1** will be connected to joints **j1**+**ji1**, **j2**+**ji1**, **j3**+**ji1**, ..., **jn**+**ji1**, generated element **e0**+**ei2** will be connected to joints **j1**+**ji2**, **j2**+**ji2**, **j3**+**ji2**, ..., **jn**+**ji2**, and so on for the rest of the generated elements. Thus all joint labels for a generated element differ from those of the starting element by the same amount. See Figure 1 (page 17) and Figure 2 (page 17) for examples.

For a Frame or Nllink element, suppose that element **e0** is connected to joints **i** and **j**. Then generated element **e0**+**ei1** will be connected to joints **i1**+**ii1** and **j**+**ji1**, generated element **e0**+**ei2** will be connected to joints **i1**+**ii2** and **j**+**ji2**, and so on for the rest of the generated elements.

The default for the joint label increments **ji1** and **ji2** are the element label increments **ei1** and **ei2**, respectively. The default for the joint label increments **ii1** and **ii2** are the joint label increments **ji1** and **ji2**, respectively. Thus it is often convenient to assign element labels that are consistent with the joint labels.

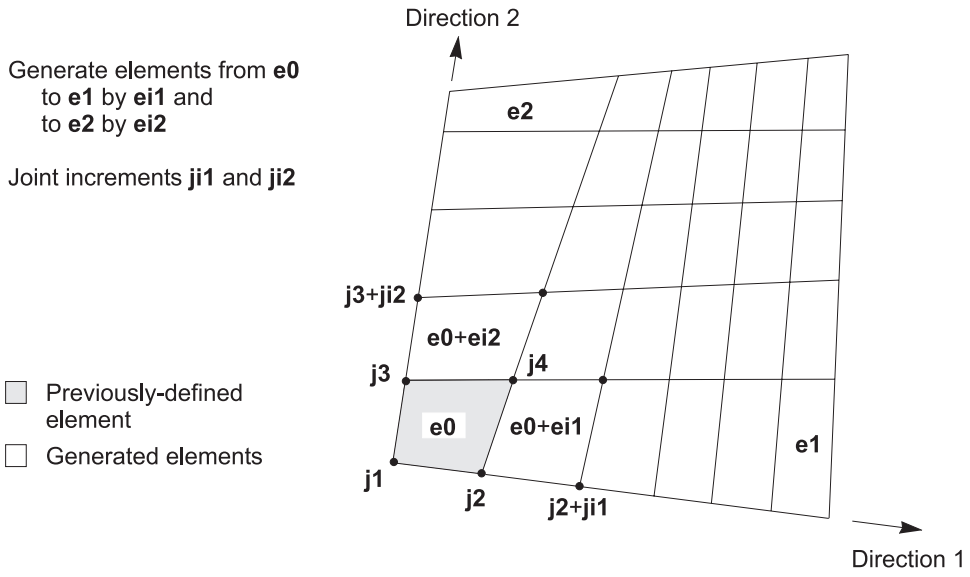


Figure 1
Two-dimensional Generation of Shell elements

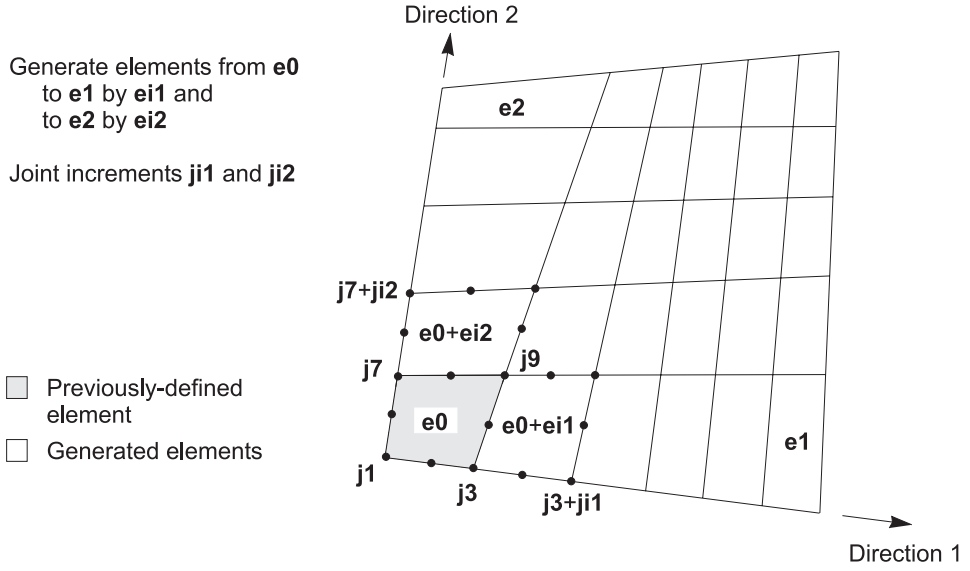


Figure 2
Two-dimensional Generation of Plane or Asolid Elements

Three-dimensional Generation

The following information is required to specify three-dimensional element generation:

- The starting element label: **e0**
- The ending element labels: **e1**, **e2**, and **e3**
- The element label increments: **ei1**, **ei2**, and **ei3**
- The joint label increments: **ji1**, **ji2**, and **ji3**
- For the Frame and Nllink elements, the joint label increments: **ii1**, **ii2**, and **ii3**

Element **e0** must already be defined. Suppose it is connected to joints **j1**, **j2**, **j3**, ..., **jn**. Then generated element **e0**+**ei1** will be connected to joints **j1**+**ji1**, **j2**+**ji1**, **j3**+**ji1**, ..., **jn**+**ji1**, generated element **e0**+**ei2** will be connected to joints **j1**+**ji2**, **j2**+**ji2**, **j3**+**ji2**, ..., **jn**+**ji2**, generated element **e0**+**ei3** will be connected to joints **j1**+**ji3**, **j2**+**ji3**, **j3**+**ji3**, ..., **jn**+**ji3**, and so on for the rest of the generated elements. Thus all joint labels for a generated element differ from those of the starting element by the same amount. See Figure 3 (page 19) for an example.

For a Frame or Nllink element, suppose that element **e0** is connected to joints **i** and **j**. Then generated element **e0**+**ei1** will be connected to joints **i1**+**ii1** and **j**+**ji1**, generated element **e0**+**ei2** will be connected to joints **i1**+**ii2** and **j**+**ji2**, generated element **e0**+**ei3** will be connected to joints **i1**+**ii3** and **j**+**ji3**, and so on for the rest of the generated elements.

The default for the joint label increments **ji1**, **ji2**, and **ji3** are the element label increments **ei1**, **ei2**, and **ei3**, respectively. The default for the joint label increments **ii1**, **ii2**, and **ii3** are the joint label increments **ji1**, **ji2**, and **ji3**, respectively. Thus it is often convenient to assign element labels that are consistent with the joint labels.

Deletion

Deletion as described in this topic is used only in the input data text file. Skip this topic if you are preparing your model using the SAP2000 graphical user interface.

Deletion is used to eliminate a regular array of previously-defined elements, Constraints, or Welds from the model. You can use a combination of generation and deletion to efficiently model a structure that has gaps or holes.

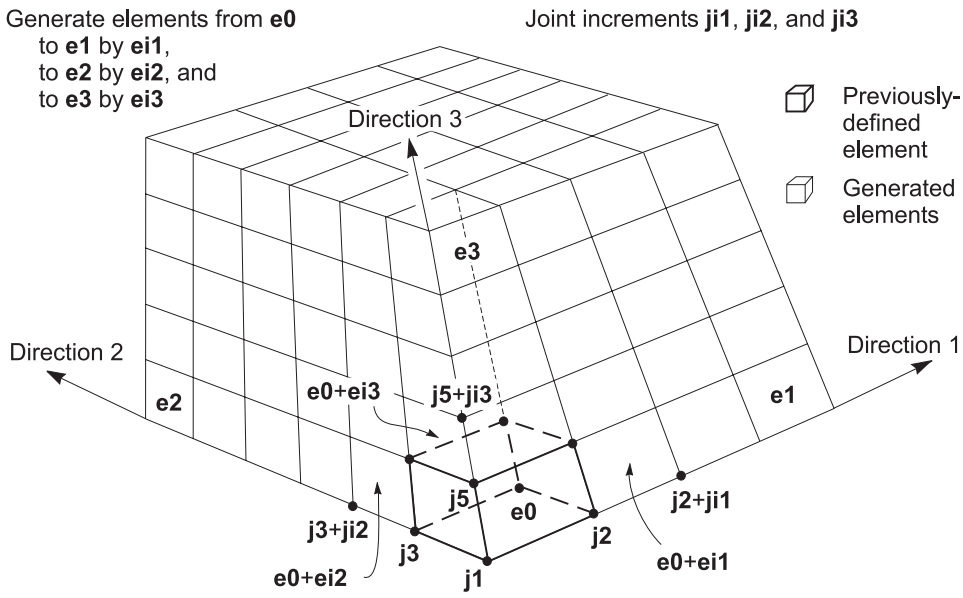


Figure 3
Three-dimensional Generation of Solid Elements

Joints cannot be deleted once they have been defined. However, the program will automatically ignore any unloaded joint that is not connected to an element or Constraint.

Assignment

Assignment as described in this topic is used only in the .S2K input data text file. Skip this topic if you are preparing your model using the SAP2000 graphical user interface.

Assignment is used to define loads and properties for regular arrays of joints or elements. The joints or elements must have been previously defined.

Three types of assignment may be available, depending upon the load or property being assigned:

- Addition: The specified load or property values are added to the current values for each joint or element in the array

- Replacement: The specified load or property values replace the current values for each joint or element in the array
- Removal: The specified type of load or property is removed from (set to zero for) each joint or element in the array

Loads are applied to all joints and elements by assignment.

All joint properties are defined by assignment. A few element properties are defined by assignment, but most properties are given when the elements are explicitly defined or generated.

Chapter III

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.

Basic Topics for All Users

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

Advanced Topics

- Alternate Coordinate Systems
- Cylindrical and Spherical Coordinates

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 coordinate system. Each part of the model (joint, element, or constraint) has its own local coordinate system. In addition, you may create alternate coordinate systems that are used to define locations and directions.

All coordinate systems are three-dimensional, right-handed, rectangular (Cartesian) systems. Vector cross products are used to define the local and alternate coordinate systems with respect to the global system.

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.

The locations of points in a coordinate system may be specified using rectangular, cylindrical, or spherical coordinates. Likewise, directions in a coordinate system may be specified using rectangular, cylindrical, or spherical coordinate directions at a point.

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. The Z direction is normally upward, but this is not required.

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 ultimately defined with respect to the global coordinate system, either directly or indirectly. Likewise, all joint coordinates are ultimately converted to global X, Y, and Z coordinates, regardless of how they were specified.

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 counterclockwise when you are looking down at the X-Y plane.

The upward and horizontal directions apply to the global coordinate system and all alternate coordinate systems.

Defining Coordinate Systems

Each coordinate system to be defined must have an origin and a set of three, mutually-perpendicular axes that satisfy the right-hand rule.

The origin is defined by simply specifying three coordinates in the global coordinate system.

The axes are defined as vectors using the concepts of vector algebra. A fundamental knowledge of the **vector cross product** operation is very helpful in clearly understanding how coordinate system axes are defined.

Vector Cross Product

A vector may be defined by two points. It has length, direction, and location in space. For the purposes of defining coordinate axes, only the direction is important. Hence any two vectors that are parallel and have the same sense (i.e., pointing the same way) may be considered to be the same vector.

Any two vectors, $\boldsymbol{V}_i$ and $\boldsymbol{V}_j$, that are not parallel to each other define a plane that is parallel to them both. The location of this plane is not important here, only its orientation. The cross product of $\boldsymbol{V}_i$ and $\boldsymbol{V}_j$ defines a third vector, $\boldsymbol{V}_k$, that is perpendicular to them both, and hence normal to the plane. The cross product is written as:

$$\boldsymbol{V_k} = \boldsymbol{V_i} \times \boldsymbol{V_j}$$

The length of $\boldsymbol{V}_k$ is not important here. The side of the $\boldsymbol{V}_i$-$\boldsymbol{V}_j$ plane to which $\boldsymbol{V}_k$ points is determined by the right-hand rule: The vector $\boldsymbol{V}_k$ points toward you if the acute angle (less than 180°) from $\boldsymbol{V}_i$ to $\boldsymbol{V}_j$ appears counterclockwise.

Thus the sign of the cross product depends upon the order of the operands:

$$\boldsymbol{V_j} \times \boldsymbol{V_i} = -\boldsymbol{V_i} \times \boldsymbol{V_j}$$

Defining the Three Axes Using Two Vectors

A right-handed coordinate system R-S-T can be represented by the three mutually-perpendicular vectors $\boldsymbol{V}_r$, $\boldsymbol{V}_s$, and $\boldsymbol{V}_t$, respectively, that satisfy the relationship:

$$\boldsymbol{V_t} = \boldsymbol{V_r} \times \boldsymbol{V_s}$$

This coordinate system can be defined by specifying two non-parallel vectors:

- An axis reference vector, $\boldsymbol{V_a}$, that is parallel to axis R
- A plane reference vector, $\boldsymbol{V_p}$, that is parallel to plane R-S, and points toward the positive-S side of the R axis

The axes are then defined as:

$$\boldsymbol{V_r} = \boldsymbol{V_a}$$

$$\boldsymbol{V_t} = \boldsymbol{V_r} \times \boldsymbol{V_p}$$

$$\boldsymbol{V_s} = \boldsymbol{V_t} \times \boldsymbol{V_r}$$

Note that $\boldsymbol{V}_p$ can be any convenient vector parallel to the R-S plane; it does not have to be parallel to the S axis. This is illustrated in Figure 4 (page 25).

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 upward +Z direction is used to define the default joint and element local coordinate systems with respect to the global or any alternate coordinate system.

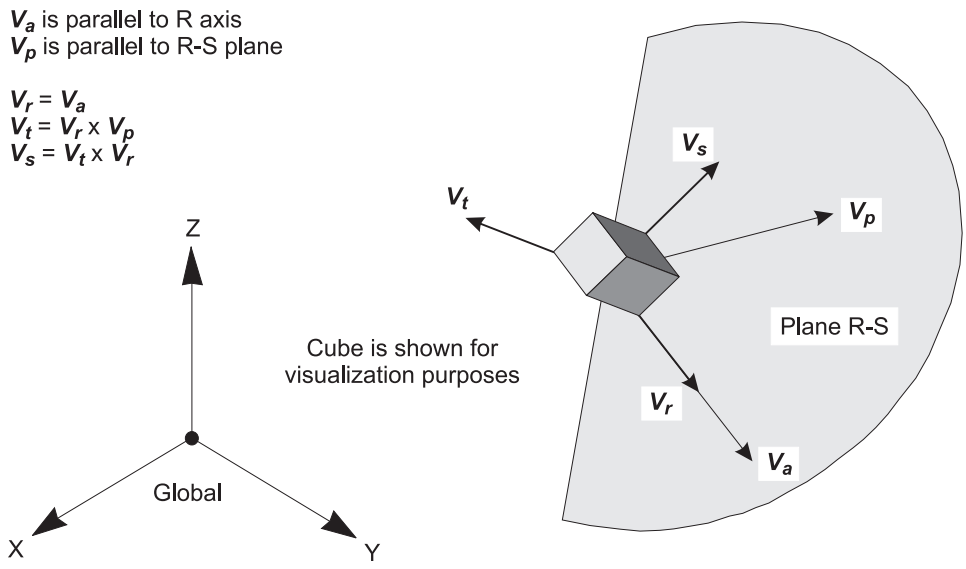


Figure 4
Determining an R-S-T Coordinate System from Reference Vectors V_a and V_p

The joint local 1-2-3 coordinate system is normally the same as the global X-Y-Z coordinate system. However, you may define any arbitrary orientation for a joint local coordinate system by specifying two reference vectors and/or three angles of rotation.

For the Frame, Shell, and Nllink 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 reference vector and/or a single angle of rotation.

The element local coordinate systems for the Plane and Asolid elements are always aligned with the global coordinate axes. The definition varies according to which global plane is parallel to the element.

The Solid element local 1-2-3 coordinate system is always the same as the global X-Y-Z coordinate system.

The local coordinate system for a Body, Diaphragm, Plate, Beam, or Rod Constraint is normally determined automatically from the geometry or mass distribution of the constraint. Optionally, you may specify one local axis for any Dia-

phragm, Plate, Beam, or Rod Constraint (but not for the Body Constraint); the remaining two axes are determined automatically.

The local coordinate system for an Equal Constraint may be arbitrarily specified; by default it is the global coordinate system. The Local Constraint does not have its own local coordinate system.

For more information:

- See Topic “Local Coordinate System” (page 78) in Chapter “Joints and Degrees of Freedom.”
- See Topic “Local Coordinate System” (page 148) in Chapter “The Frame Element.”
- See Topic “Local Coordinate System” (page 186) in Chapter “The Shell Element.”
- See Topic “Local Coordinate System” (page 210) in Chapter “The Plane Element.”
- See Topic “Local Coordinate System” (page 224) in Chapter “The Asolid Element.”
- See Topic “Local Coordinate System” (page 236) in Chapter “The Solid Element.”
- See Topic “Local Coordinate System” (page 245) in Chapter “The Nllink Element.”
- See Chapter “Constraints and Welds (page 113).”

Alternate Coordinate Systems

You may define **alternate coordinate systems** that can be used for locating the joints; for defining local coordinate systems for joints, elements, and constraints; and as a reference for defining other properties and loads. The axes of the alternate coordinate systems are denoted X, Y, and Z.

The global coordinate system and all alternate systems are called **fixed coordinate systems**, since they apply to the whole structural model, not just to individual parts as do the local coordinate systems. Each fixed coordinate system may be used in rectangular, cylindrical or spherical form.

The definition of the upward and horizontal directions for each alternate coordinate system is the same as for the global coordinate system.

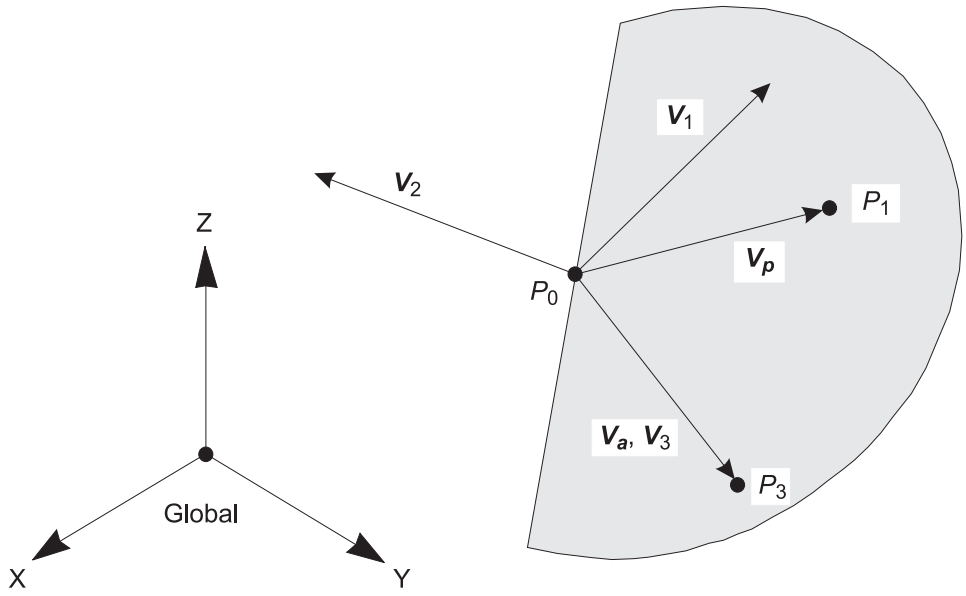


Figure 5
Definition of Alternate Coordinate System Using Three Points

Each alternate coordinate system is defined by specifying the location of three points in the global coordinate system:

- Point P_0 at the origin of the new system
- Point P_3 anywhere on the +Z half of the new Z axis
- Point P_1 anywhere on the +X half of the new Z-X plane

An axis reference vector, $\boldsymbol{V_a}$, is defined from point P_0 to point P_3, and a plane reference vector $\boldsymbol{V_p}$ is defined from point P_0 to point P_1. The new, positive X, Y and Z axes then have the directions of $\boldsymbol{V}_1$, $\boldsymbol{V}_2$, and $\boldsymbol{V}_3$, respectively, defined as:

$$\boldsymbol{V}_3 = \boldsymbol{V_a}$$

$$\boldsymbol{V}_2 = \boldsymbol{V}_3 \times \boldsymbol{V_p}$$

$$\boldsymbol{V}_1 = \boldsymbol{V}_2 \times \boldsymbol{V}_3$$

This is illustrated in Figure 5 (page 27).

Cylindrical and Spherical Coordinates

The location of points in the global or an alternate coordinate system may be specified using polar coordinates instead of rectangular X-Y-Z coordinates. Polar coordinates include cylindrical CR-CA-CZ coordinates and spherical SB-SA-SR coordinates. See Figure 6 (page 29) for the definition of the polar coordinate systems. Polar coordinate systems are always defined with respect to a rectangular X-Y-Z system.

The coordinates CR, CZ, and SR are lineal and are specified in length units. The coordinates CA, SB, and SA are angular and are specified in degrees.

Locations are specified in *cylindrical* coordinates using the variables **cr**, **ca**, and **cz**. These are related to the rectangular coordinates as:

$$\mathbf{cr} = \sqrt{\mathbf{x}^2 + \mathbf{y}^2}$$

$$\mathbf{ca} = \tan^{-1}\frac{\mathbf{y}}{\mathbf{x}}$$

$$\mathbf{cz} = \mathbf{z}$$

Locations are specified in *spherical* coordinates using the variables **sb**, **sa**, and **sr**. These are related to the rectangular coordinates as:

$$\mathbf{sb} = \tan^{-1}\frac{\sqrt{\mathbf{x}^2 + \mathbf{y}^2}}{\mathbf{z}}$$

$$\mathbf{sa} = \tan^{-1}\frac{\mathbf{y}}{\mathbf{x}}$$

$$\mathbf{sr} = \sqrt{\mathbf{x}^2 + \mathbf{y}^2 + \mathbf{z}^2}$$

A vector in a fixed coordinate system can be specified by giving the locations of two points or by specifying a coordinate direction at a single point *P*. Coordinate directions are tangential to the coordinate curves at point *P*. A positive coordinate direction indicates the direction of increasing coordinate value at that point.

Cylindrical coordinate directions are indicated using the values ±CR, ±CA, and ±CZ. Spherical coordinate directions are indicated using the values ±SB, ±SA, and ±SR. The sign is required. See Figure 6 (page 29).

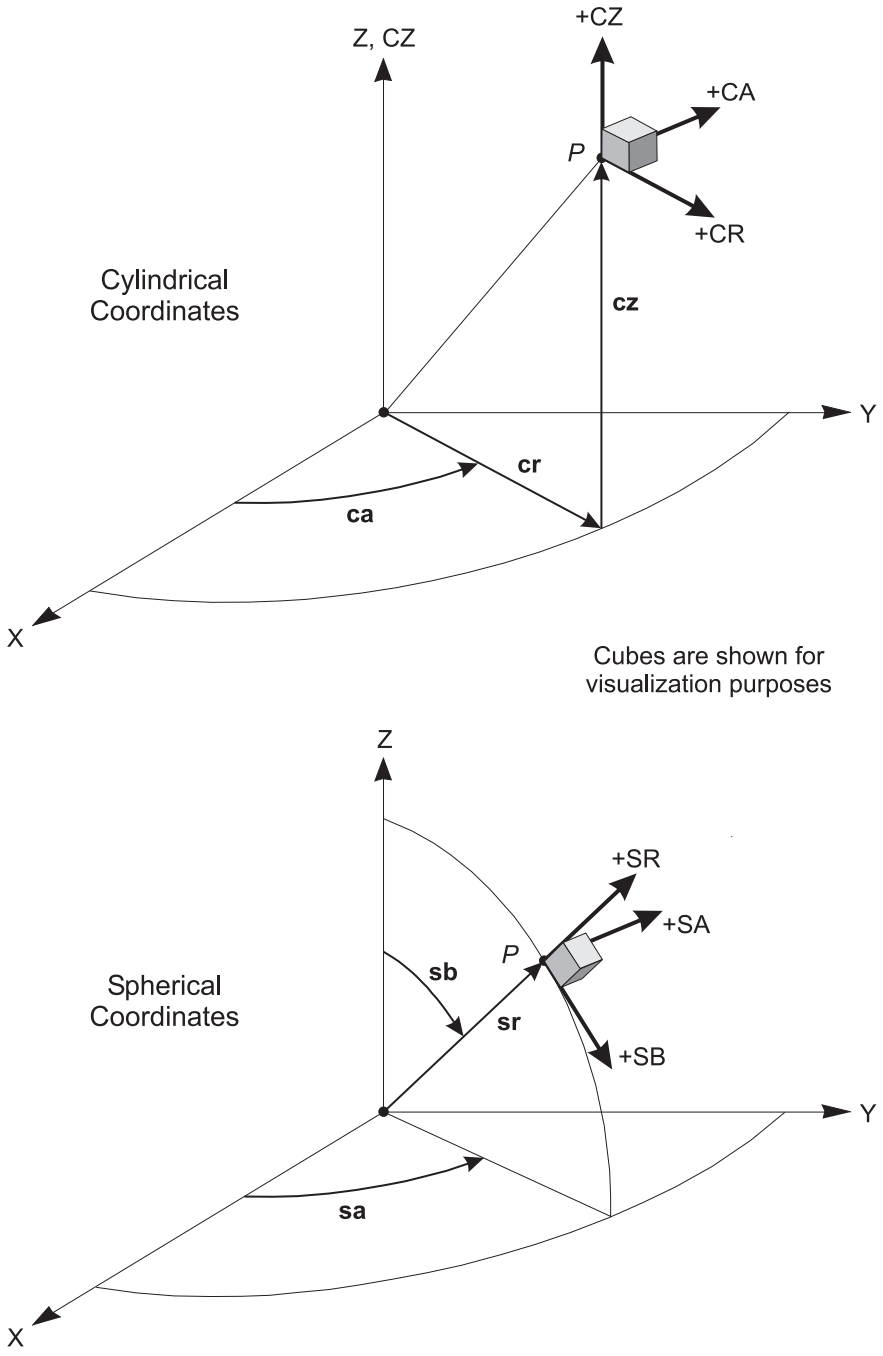


Figure 6
Cylindrical and Spherical Coordinates and Coordinate Directions

The cylindrical and spherical coordinate directions are not constant but vary with angular position. The coordinate directions do not change with the lineal coordinates. For example, +SR defines a vector directed from the origin to point *P*.

Note that the coordinates Z and CZ are identical, as are the corresponding coordinate directions. Similarly, the coordinates CA and SA and their corresponding coordinate directions are identical.

Chapter IV

Joint Coordinates

This chapter describes the definition of the joints and their use in defining the geometry of the structure. This chapter is only of interest if you are preparing your model using the input data text file. See Chapter “Joints and Degrees of Freedom” (page 75) for more general information about the joints.

Basic Topics for All Users

- Overview

Advanced Topics

- Joint Definition
- One-dimensional Joint Generation
- Two-dimensional Joint Generation
- Three-dimensional Joint Generation
- Variable Joint Spacing
- Joint Definition in Polar Coordinates
- Joint Definition in Alternate Coordinate Systems

Overview

Joints, also known as **nodal points** or **nodes**, are a fundamental part of every structural model. Joints perform a variety of functions, which are discussed in Chapter "Joints and Degrees of Freedom" (page 75).

The method used to define the structural model affects how joints are created:

- Using the SAP2000 graphical interface — joints are automatically created at the ends of each Frame or Nllink element and at the corners of each Shell element; additional joints may also be defined independently of any element
- Using the input data text file — joint locations must be explicitly defined in order to describe the geometry of the structure; these joints are then connected by elements to build the structure

The use of joints to define the geometry of the structure is discussed in this chapter. If you are using the SAP2000 graphical user interface, you may skip the rest of this chapter.

A variety of methods are available in the input data text file to define the layout of the joints, and hence the geometry of the structure. Joints may be located in arbitrary coordinate systems using rectangular, cylindrical, or spherical coordinates. Large arrays of joints may be generated in one, two, and three dimensions after defining a smaller number of joints at the corners or along the edges of the region.

Joint Definition

Joint definition as described in this topic is used only in the input data text file. You may skip this topic if you are preparing your model using the SAP2000 graphical user interface.

A joint is defined by specifying its label, **j**, and three spatial coordinates, **x**, **y**, **z**, that locate the joint in space. You may define a joint individually, or use a generation operation that defines many joints on a line (or curve), a surface, or throughout a three-dimensional region.

You may define extraneous joints in the system, for geometric reference, that do not attach to any elements. The program will automatically restrain them.

One-dimensional Joint Generation

Joint generation as described in this topic is used only in the input data text file. You may skip this topic if you are preparing your model using the SAP2000 graphical user interface.

It is often useful to define a large number of equally-spaced joints along a straight line or curve. This is easily done with the generation options in SAP2000 if the labels of the joints increment in a uniform fashion along the line or curve. Such a set of joints is called a one-dimensional joint array.

Each generation option requires the specification of the labels for the joint array. This is described in Subtopic "One-dimensional Joint Array Specification" below. The additional information that is required to define the location of the joints is described in the subtopics that follow.

See Subtopic "One-dimensional Regular Arrays" (page 11) in Chapter "Labels, Arrays, and Generation" for more information.

One-dimensional Joint Array Specification

The following information is required to specify a one-dimensional array of joints:

- The label, **j0**, of the joint at the start of the array
- The label, **j1**, of the joint at the end of the array
- The label increment (constant difference), **ji1**, between any pair of successive joints along the line

The number of spaces, n_1, between the joints along the line is then given by:

$$n_1 = \frac{\mathbf{j1} - \mathbf{j0}}{\mathbf{ji1}}$$

The labels and label increment must be chosen such that n_1 is a whole, positive number. The total number of joints in the array, including the ends, is n_1+1.

One-dimensional Joint Definition

A set of equally-spaced joints along a straight line can be directly defined by specifying:

- The information in Subtopic “One-dimensional Joint Array Specification” above
- The three spatial coordinates, (**x0**, **y0**, **z0**), for joint **j0**
- The three spatial coordinates, (**x1**, **y1**, **z1**), for joint **j1**

The spatial coordinates of the intermediate joints are determined by linear interpolation such that the constant coordinate differences, Δx_1, Δy_1, and Δz_1, between any pair of successive joints are given by:

$$\Delta x_1 = \frac{\mathbf{x1} - \mathbf{x0}}{n_1}, \quad \Delta y_1 = \frac{\mathbf{y1} - \mathbf{y0}}{n_1}, \quad \text{and} \quad \Delta z_1 = \frac{\mathbf{z}_1 - \mathbf{z0}}{n_1}$$

See Figure 7 (page 35) for an example of one-dimensional joint definition.

One-dimensional Linear Generation

One-dimensional linear generation is the same as one-dimensional joint array definition, except that the joints at the two ends must have previously been defined:

1. Joint **j0**. The spatial coordinates, (**x0**, **y0**, **z0**), for this joint are known
2. Joint **j1**. The spatial coordinates, (**x1**, **y1**, **z1**), for this joint are known

It is then only necessary to provide the information in Subtopic “One-dimensional Joint Array Specification” above.

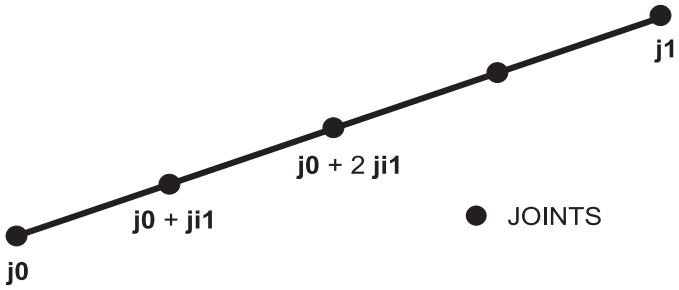
The spatial coordinates of the intermediate joints are determined by linear interpolation such that the constant coordinate differences, Δx_1, Δy_1, and Δz_1, between any pair of successive joints are given by:

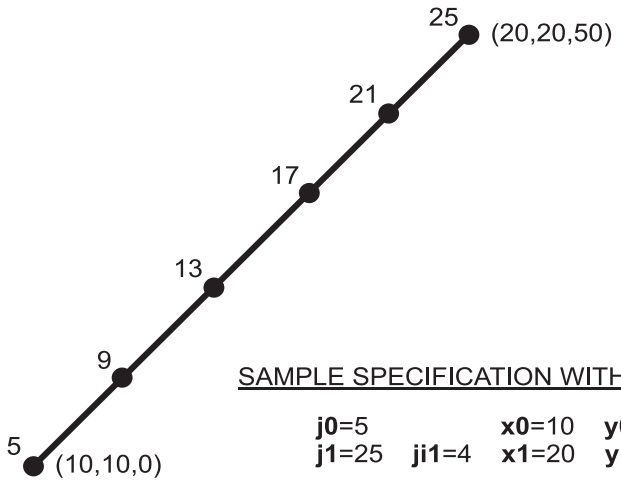
$$\Delta x_1 = \frac{\mathbf{x1} - \mathbf{x0}}{n_1}, \quad \Delta y_1 = \frac{\mathbf{y1} - \mathbf{y0}}{n_1}, \quad \text{and} \quad \Delta z_1 = \frac{\mathbf{z1} - \mathbf{z0}}{n_1}$$

One-dimensional Cylindrical Generation

One-dimensional cylindrical generation can be used to generate joints along circular, helical, and spiral curves. Before using cylindrical generation, the starting joint **j0** must have been previously defined; the spatial coordinates, (**x0**, **y0**, **z0**), for this joint are known. The following additional information must be specified:

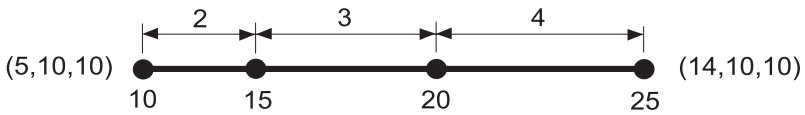
1. The information in Subtopic “One-dimensional Joint Array Specification” above.





SAMPLE SPECIFICATION WITH EQUAL SPACES:

j0=5 **x0**=10 **y0**=10 **z0**=0
j1=25 **ji1**=4 **x1**=20 **y1**=20 **z1**=50



SAMPLE SPECIFICATION WITH UNEQUAL SPACES:

j0=10 **x0**=5 **y0**=10 **z0**=10
j1=25 **ji1**=5 **x1**=14 **y1**=10 **z1**=10 **ratio1**=2

Figure 7
One-dimensional Joint Array Definition

2. A pair of previously-defined joints, **axveca** and **axvecb**, that define the axis about which the generation is performed. The positive direction of this axis is from **axveca** to **axvecb**. Joint **j0** must not lie on this axis.

3. The angle increment, **da**, between successive joints in the array. The positive direction of **da** is given by the right-hand rule with respect to the axis of generation.

4. For spiral generation, the radius increment, **dr**, between successive joints in the array. The positive direction of **dr** is away from the axis of generation.

5. For helical generation, the axial increment, **dz**, between successive joints in the array. The positive direction of **dz** is along the positive direction of the axis of generation.

The spatial coordinates of each generated joint are obtained from the previous joint in the array by moving a distance **dz** parallel to the axis of generation, moving a distance **dr** away from the axis, and moving through an angle **da** about the axis.

A circular arc is defined if **dr** and **dz** are both zero. A spiral is defined if **dr** is nonzero and **dz** is zero. A helix is defined if **dz** is nonzero; it is a cylindrical helix if **dr** is zero, and a conical helix otherwise. If **da** is zero, a straight line is defined.

Two-dimensional Joint Generation

Joint generation as described in this topic is used only in the input data text file. You may skip this topic if you are preparing your model using the SAP2000 graphical user interface.

Two-dimensional joint generation can be used to define large numbers of joints over four-sided regions on flat or curved surfaces. All that is required is that the labels of the joints increment in a uniform fashion in two separate directions. Such a set of joints is called a two-dimensional joint array.

A variety of generation options is available, differing primarily in the amount of information that must be specified and therefore the complexity of the shapes that can be described. Each generation option requires the specification of the labels for the joint array. This is described in Subtopic "Two-dimensional Joint Array Specification" below. The additional information that is required to define the location of the joints is described in the subtopics that follow.

See Subtopic “Two-dimensional Regular Arrays” (page 12) in Chapter “Labels, Arrays, and Generation” for more information.

Two-dimensional Joint Array Specification

The following information is required to specify a two-dimensional array of joints:

- The label, **j0**, of the joint at one corner
- The label, **j1**, of the joint at the end of one side that contains joint **j0**
- The label, **j2**, of the joint at the end of the other side that contains joint **j0**
- The label increment (constant difference), **ji1**, between any pair of successive joints along side **j0**-**j1**
- The label increment (constant difference), **ji2**, between any pair of successive joints along side **j0**-**j2**

The numbers of spaces, n_1 and n_2, between the joints along directions “parallel” to sides **j0**-**j1** and **j0**-**j2**, respectively, are then given by:

$$n_1 = \frac{\mathbf{j1} - \mathbf{j0}}{\mathbf{ji1}} \quad \text{and} \quad n_2 = \frac{\mathbf{j2} - \mathbf{j0}}{\mathbf{ji2}}$$

The labels and label increments must be chosen such that n_1 and n_2 are whole, positive numbers. The total number of joints in the array is $(n_1 + 1)(n_2 + 1)$.

Two-dimensional Joint Definition

A set of equally-spaced joints on a rectangular or parallelogram-shaped region of a flat surface can be defined by specifying:

- The information in Subtopic “Two-dimensional Joint Array Specification” above
- The three spatial coordinates, (**x0**, **y0**, **z0**), for joint **j0**
- The three spatial coordinates, (**x1**, **y1**, **z1**), for joint **j1**
- The three spatial coordinates, (**x2**, **y2**, **z2**), for joint **j2**

The spatial coordinates of the remaining joints are determined by linear interpolation and extrapolation. The constant coordinate differences, Δx_1, Δy_1, and Δz_1, between any pair of joints whose labels differ by **ji1** are given by:

$$\Delta x_1 = \frac{\mathbf{x1 - x0}}{n_1}, \quad \Delta y_1 = \frac{\mathbf{y1 - y0}}{n_1}, \quad \text{and} \quad \Delta z_1 = \frac{\mathbf{z1 - z0}}{n_1}$$

Similarly, the constant coordinate differences, Δx_2, Δy_2, and Δz_2, between any pair of joints whose labels differ by **ji2** are given by:

$$\Delta x_2 = \frac{\mathbf{x2 - x0}}{n_2}, \quad \Delta y_2 = \frac{\mathbf{y2 - y0}}{n_2}, \quad \text{and} \quad \Delta z_2 = \frac{\mathbf{z2 - z0}}{n_2}$$

Two-dimensional Linear Generation

Two-dimensional linear generation creates a set of joints on a straight-sided quadrilateral surface determined by four previously-defined joints at the corners. If the four corner joints do not lie in a plane, the surface is called a hyperbolic-paraboloid, commonly known as a "hypar" (Timoshenko and Woinowsky-Krieger, 1959). The generated joints are equally spaced along each of the four sides of the surface.

Before using two-dimensional linear generation, the four corner joints must have already been defined:

1. Joint **j0**
2. Joint **j1**
3. Joint **j2**
4. Joint **j1**+**j2**–**j0** at the corner opposite joint **j0**

It is then only necessary to provide the information in Subtopic "Two-dimensional Joint Array Specification" above.

The spatial coordinates of the remaining joints are determined by standard bilinear interpolation (Cook, Malkus, and Plesha, 1989). Any subset of the generated joints having labels that increment uniformly by **ji1** or **ji2** will lie on a straight line and will be equally spaced.

See Figure 8 (page 39) for an example.

Two-dimensional Frontal Generation

Two-dimensional frontal generation creates a set of joints on a four-sided surface determined by previously-defined joints along two adjacent sides. These

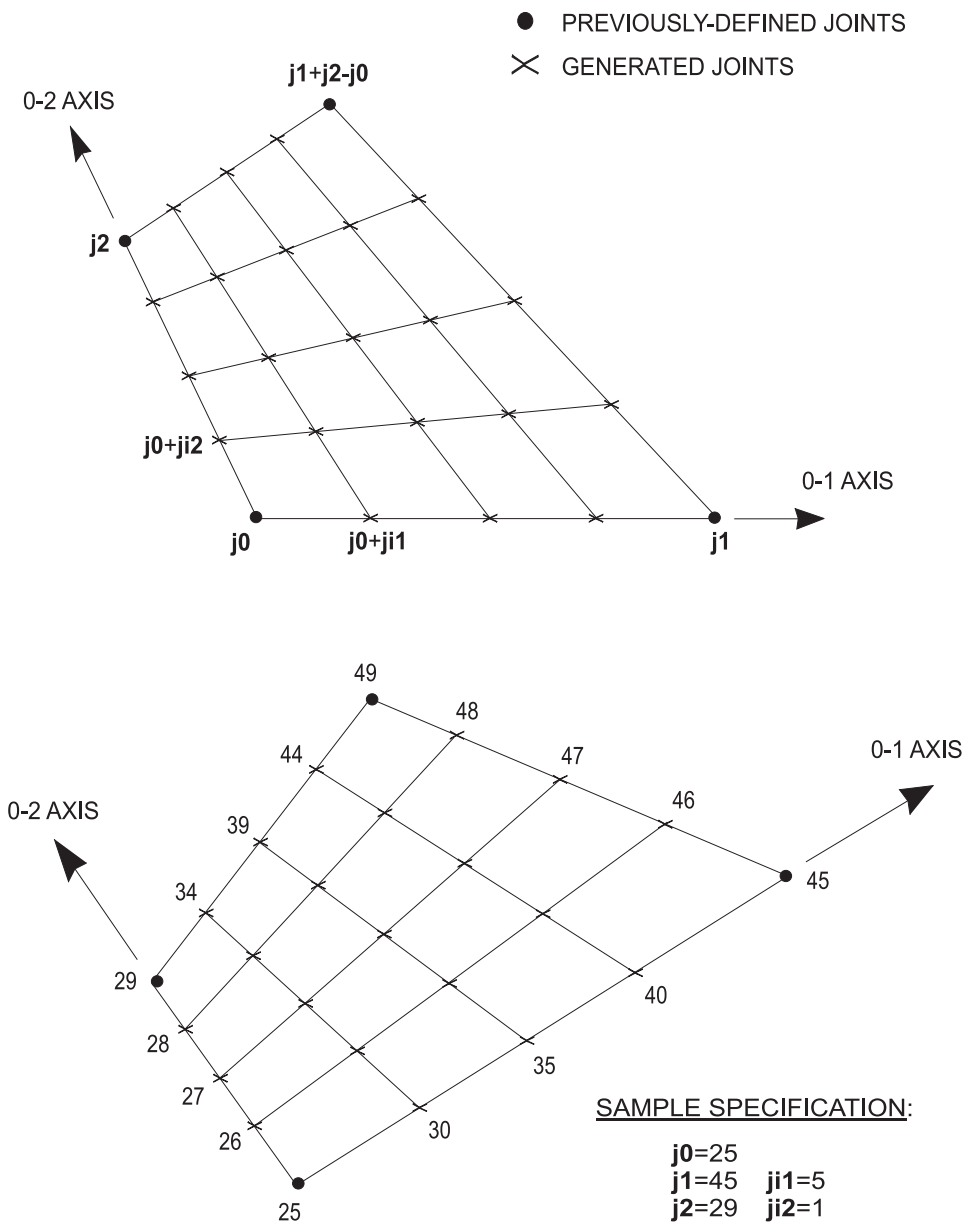


Figure 8
Two-dimensional Linear Generation

previously-defined sides may have any shape, and the joint spacing may be uniform or nonuniform.

The generated joints lie along lines or curves that are parallel to the two defining sides. The spacing between the lines or curves is determined by the joints on the two sides.

Before using two-dimensional frontal generation, all joints on the two adjacent sides must have already been defined:

1. Joints **j0** to **j1** by increment **ji1**
2. Joints **j0** to **j2** by increment **ji2**

It is then only necessary to provide the information in Subtopic "Two-dimensional Joint Array Specification" above.

See Figure 9 (page 41) for an example of frontal generation used for a planar building frame.

Two-dimensional Edge Generation

Two-dimensional edge generation creates a set of joints over a complex, four-sided surface determined by previously-defined joints along all four sides. The previously-defined sides may have any shape, and the joint spacing may be uniform or nonuniform.

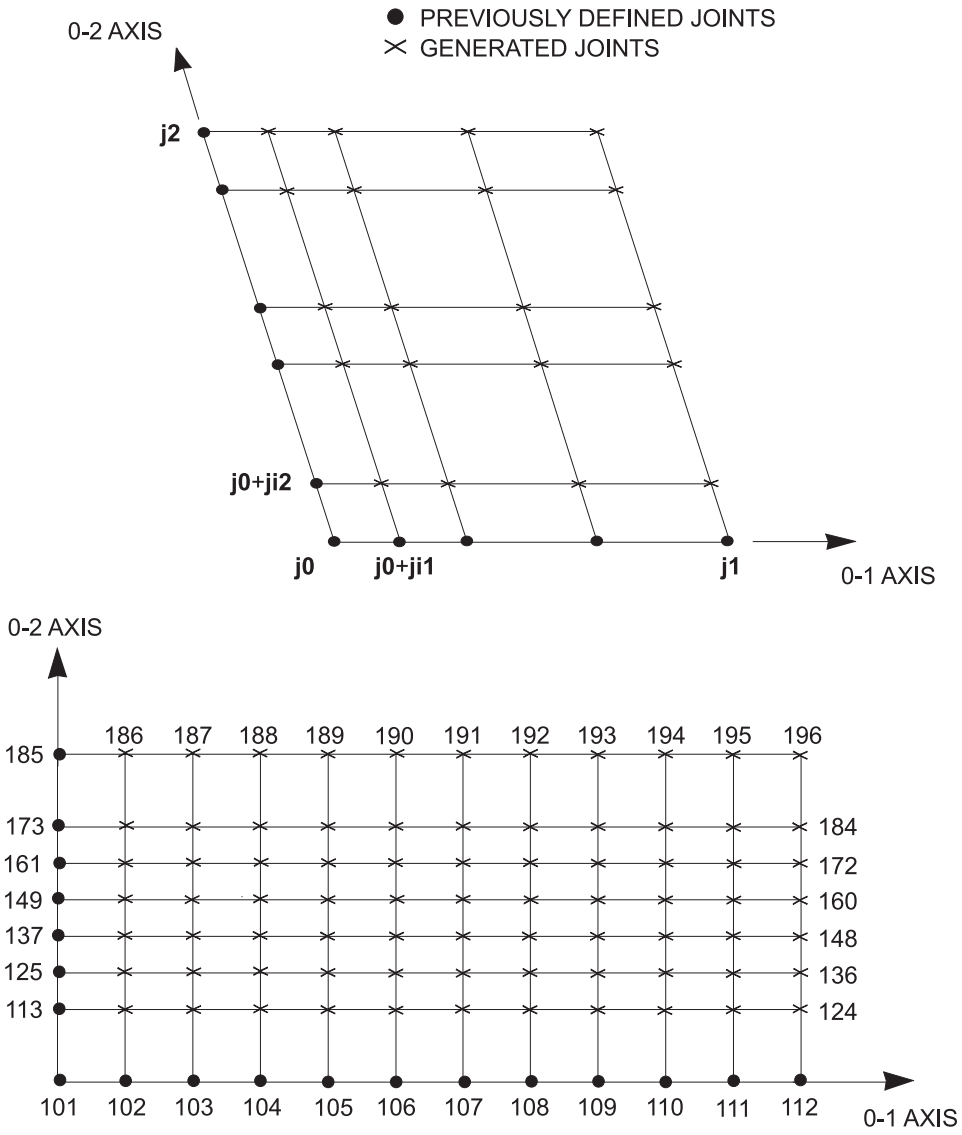
The joints in the interior of the surface are generated using serendipity polynomial interpolation (Cook, Malkus, and Plesha, 1989).

Before using two-dimensional edge generation, all joints on the four sides must have already been defined:

1. Joints **j0** to **j1** by increment **ji1**
2. Joints **j0** to **j2** by increment **ji2**
3. Joints **j2** to **j1**+**j2**–**j0** by increment **ji1**
4. Joints **j1** to **j1**+**j2**–**j0** by increment **ji2**

It is then only necessary to provide the information in Subtopic "Two-dimensional Joint Array Specification" above.

See Figure 10 (page 42) for an example of two-dimensional edge generation.



SAMPLE SPECIFICATION:

j0=101
j1=112 **ji1**=1
j2=185 **ji2**=12

Figure 9
Two-dimensional Frontal Generation

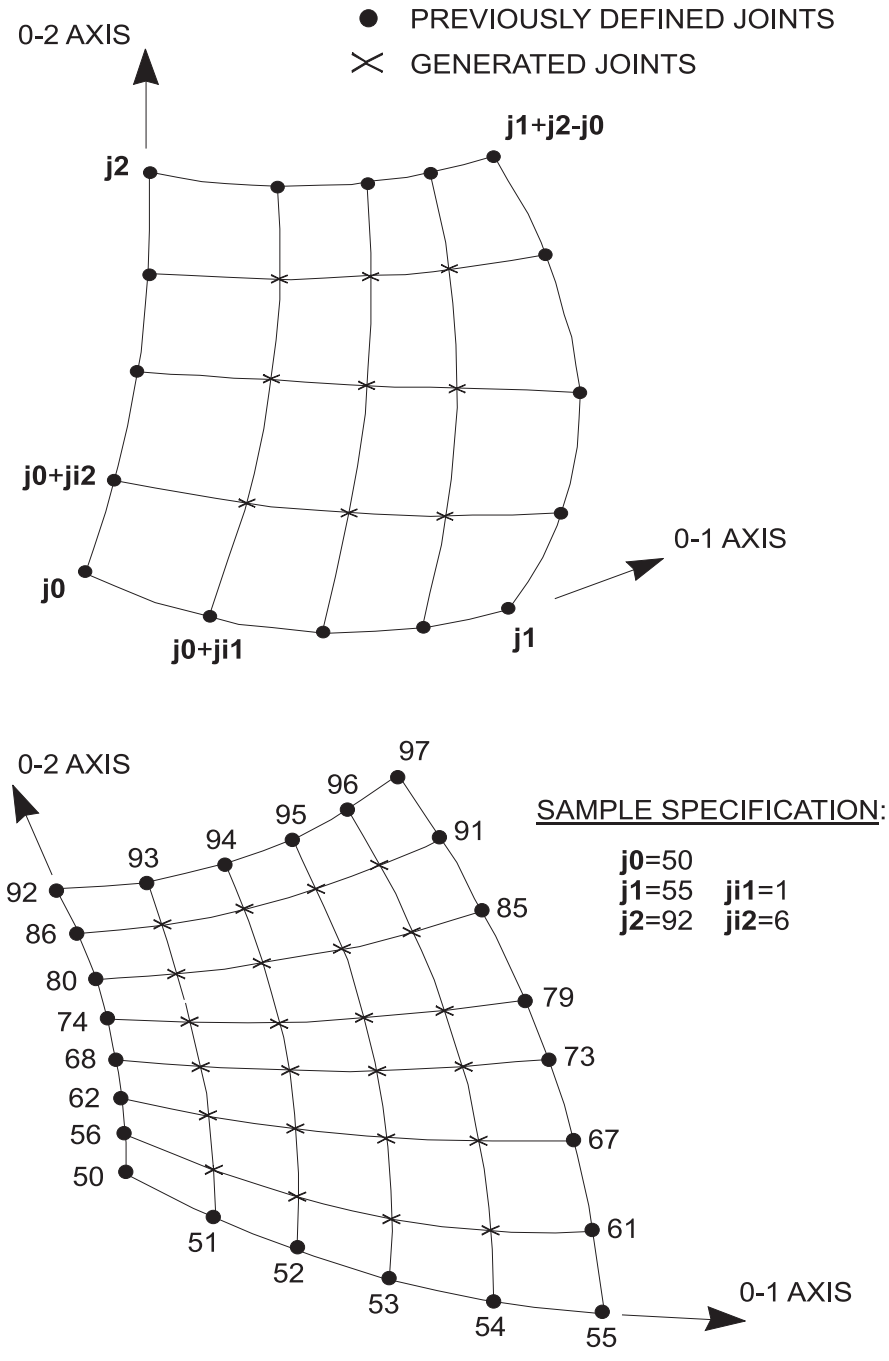


Figure 10
Two-dimensional Edge Generation

Three-dimensional Joint Generation

Joint generation as described in this topic is used only in the input data text file. You may skip this topic if you are preparing your model using the SAP2000 graphical user interface.

Three-dimensional joint generation can be used to define large numbers of joints over three-dimensional regions enclosed by hexahedra, i.e., having six faces. All that is required is that the labels of the joints increment in a uniform fashion in three separate directions. Such a set of joints is called a three-dimensional joint array.

A variety of generation options is available, differing primarily in the amount of information that must be specified and therefore the complexity of the shapes that can be described. Each generation option requires the specification of the labels for the joint array. This is described in Subtopic "Three-dimensional Joint Array Specification" below. The additional information that is required to define the location of the joints is described in the subtopics that follow.

Three-dimensional joint generation is a generalization of two-dimensional joint generation.

For more information:

- See Topic "Two-dimensional Joint Generation" (page 36) in this chapter.
- See Subtopic "Three-dimensional Regular Arrays" (page 13) in Chapter "Labels, Arrays, and Generation."

Three-dimensional Joint Array Specification

The following information is required to specify a three-dimensional array of joints:

- The label, **j0**, of the joint at one corner
- The label, **j1**, of the joint at the end of one side that contains joint **j0**
- The label, **j2**, of the joint at the end of another side that contains joint **j0**
- The label, **j3**, of the joint at the end of the third side that contains joint **j0**
- The label increment (constant difference), **ji1**, between any pair of successive joints along side **j0**-**j1**
- The label increment (constant difference), **ji2**, between any pair of successive joints along side **j0**-**j2**

- The label increment (constant difference), **ji3**, between any pair of successive joints along side **j0**-**j3**

The numbers of spaces, n_1, n_2, and n_3, between the joints along directions "parallel" to sides **j0**-**j1**, **j0**-**j2**, and **j0**-**j3**, respectively, are then given by:

$$n_1 = \frac{\mathbf{j1} - \mathbf{j0}}{\mathbf{ji1}}, \quad n_2 = \frac{\mathbf{j2} - \mathbf{j0}}{\mathbf{ji2}}, \quad \text{and} \quad n_3 = \frac{\mathbf{j3} - \mathbf{j0}}{\mathbf{ji3}}$$

The labels and label increments must be chosen such that n_1, n_2, and n_3 are whole, positive numbers. The total number of joints in the array is $(n_1 + 1)(n_2 + 1)(n_3 + 1)$.

Three-dimensional Joint Definition

A set of equally-spaced joints within a parallelepiped (the six faces are parallelograms) can be defined by specifying:

- The information in Subtopic "Three-dimensional Joint Array Specification" above
- The three spatial coordinates, (**x0**, **y0**, **z0**), for joint **j0**
- The three spatial coordinates, (**x1**, **y1**, **z1**), for joint **j1**
- The three spatial coordinates, (**x2**, **y2**, **z2**), for joint **j2**
- The three spatial coordinates, (**x3**, **y3**, **z3**), for joint **j3**

The spatial coordinates of the remaining joints are determined by linear interpolation and extrapolation. The constant coordinate differences, Δx_1, Δy_1, and Δz_1, between any pair of joints whose labels differ by **ji1** are given by:

$$\Delta x_1 = \frac{\mathbf{x1} - \mathbf{x0}}{n_1}, \quad \Delta y_1 = \frac{\mathbf{y1} - \mathbf{y0}}{n_1}, \quad \text{and} \quad \Delta z_1 = \frac{\mathbf{z1} - \mathbf{z0}}{n_1}$$

Similarly, the constant coordinate differences, Δx_2, Δy_2, and Δz_2, between any pair of joints whose labels differ by **ji2** are given by:

$$\Delta x_2 = \frac{\mathbf{x2} - \mathbf{x0}}{n_2}, \quad \Delta y_2 = \frac{\mathbf{y2} - \mathbf{y0}}{n_2}, \quad \text{and} \quad \Delta z_2 = \frac{\mathbf{z2} - \mathbf{z0}}{n_2}$$

Finally, the constant coordinate differences, Δx_3, Δy_3, and Δz_3, between any pair of joints whose labels differ by **ji3** are given by:

$$\Delta x_3 = \frac{\mathbf{x3 - x0}}{n_3}, \quad \Delta y_3 = \frac{\mathbf{y3 - y0}}{n_3}, \quad \text{and} \quad \Delta z_3 = \frac{\mathbf{z3 - z0}}{n_3}$$

Three-dimensional Linear Generation

Three-dimensional linear generation creates a set of joints over a three-dimensional region enclosed by six quadrilateral faces. Each of the twelve edges is a straight line. The shape of the region is determined by eight previously-defined joints at the corners.

If the four corner joints of a face do not lie in a plane, the face is a surface called a hyperbolic-paraboloid, commonly known as a "hypar" (Timoshenko and Woinowsky-Krieger, 1959). The generated joints are equally spaced along each of the twelve edges of the region.

Before using three-dimensional linear generation, the eight corner joints must have already been defined:

1. Joint **j0**
2. Joint **j1**
3. Joint **j2**
4. Joint **j3**
5. Joint **j1**+**j2**–**j0** at the corner opposite joint **j3**
6. Joint **j1**+**j3**–**j0** at the corner opposite joint **j2**
7. Joint **j2**+**j3**–**j0** at the corner opposite joint **j1**
8. Joint **j1**+**j2**+**j3**–**j0**–**j0** at the corner opposite joint **j0**

It is then only necessary to provide the information in Subtopic "Three-dimensional Joint Array Specification" above.

The spatial coordinates of the remaining joints are determined by standard trilinear interpolation (Cook, Malkus, and Plesha, 1989). Any subset of the generated joints having labels that increment uniformly by **ji1**, **ji2**, or **ji3** will lie on a straight line and will be equally spaced.

Three-dimensional Frontal Generation

Three-dimensional frontal generation creates a set of joints over a three-dimensional region determined by previously-defined joints along three adjacent sides. These previously-defined sides may have any shape, and the joint spacing may be uniform or nonuniform.

The generated joints lie along lines or curves that are parallel to the three defining sides. The spacing between the lines or curves is determined by the joints on the three sides.

Before using three-dimensional frontal generation, all joints on the three adjacent sides must have already been defined:

1. Joints **j0** to **j1** by increment **ji1**
2. Joints **j0** to **j2** by increment **ji2**
3. Joints **j0** to **j3** by increment **ji3**

It is then only necessary to provide the information in Subtopic "Three-dimensional Joint Array Specification" above.

Three-dimensional Edge Generation

Three-dimensional edge generation creates a set of joints over a complex, six-faced volume determined by previously-defined joints along all twelve edges. The previously-defined edges may have any shape, and the joint spacing may be uniform or nonuniform.

The joints in the interior of the volume and the faces are generated using serendipity polynomial interpolation (Zienkiewicz and Taylor, 1989).

Before using three-dimensional frontal generation, all joints on the twelve edges must have already been defined:

1. Joints **j0** to **j1** by increment **ji1**
2. Joints **j0** to **j2** by increment **ji2**
3. Joints **j0** to **j3** by increment **ji3**
4. Joints **j1** to **j1**+**j2**–**j0** by increment **ji2**
5. Joints **j1** to **j1**+**j3**–**j0** by increment **ji3**

6. Joints **j2** to **j1**+**j2**–**j0** by increment **ji1**
7. Joints **j2** to **j2**+**j3**–**j0** by increment **ji3**
8. Joints **j3** to **j1**+**j3**–**j0** by increment **ji1**
9. Joints **j3** to **j2**+**j3**–**j0** by increment **ji2**
10. Joints **j1**+**j2**–**j0** to **j1**+**j2**+**j3**–**j0**–**j0** by increment **ji3**
11. Joints **j1**+**j3**–**j0** to **j1**+**j2**+**j3**–**j0**–**j0** by increment **ji2**
12. Joints **j2**+**j3**–**j0** to **j1**+**j2**+**j3**–**j0**–**j0** by increment **ji1**

It is then only necessary to provide the information in Subtopic "Two-dimensional Joint Array Specification" above.

Variable Joint Spacing

Variable joint spacing as described in this topic is used only in the input data text file. You may skip this topic if you are preparing your model using the SAP2000 graphical user interface.

Joints created by array definition or linear generation normally have uniform spacing along any straight line, provided that the joint labels differ by one of the specified label increments.

Sometimes, however, it is desirable to have the spacing between the joints vary gradually along these straight lines. This is often the case in finite element meshes where smaller elements are needed near a region of interest, such as at a stress concentration, and larger elements are to be used farther away for computational efficiency. Such a mesh is called a graded mesh.

An option is available to specify how the joint spacing is to vary along each generation dimension. The joint spacing changes by arithmetic progression so that the *difference* between successive joint spacings is constant.

This option is only available for array definition and linear generation. For frontal generation and edge generation, joint spacing is determined by the locations of the joints on the previously-defined edges.

This option may also be used for array definition in polar coordinates. In this case, it is the polar-coordinate differences, rather than the absolute joint spacing, that varies along the generation dimensions.

For more information:

- See Subtopic “One-dimensional Joint Definition” (page 33) in this chapter.
- See Subtopic “One-dimensional Linear Generation” (page 34) in this chapter.
- See Subtopic “Two-dimensional Joint Definition” (page 37) in this chapter.
- See Subtopic “Two-dimensional Linear Generation” (page 38) in this chapter.
- See Subtopic “Three-dimensional Joint Definition” (page 44) in this chapter.
- See Subtopic “Three-dimensional Linear Generation” (page 45) in this chapter.
- See Topic “Joint Definition in Polar Coordinates” (page 50) in this chapter.

One Dimension

The parameter **ratio1** can be used to specify how the joint spacing varies along the line of generation. It gives the ratio of the last spacing, d_{1_n}, between joints **j1**–**ji1** and **j1**, and the first spacing, d_{1_1}, between joints **j0** and **j0**+**ji1**:

$$\textbf{ratio1} = \frac{d_{1_n}}{d_{1_1}}$$

If the value of **ratio1** is greater than unity, then the distance between joints increases along the length. Connected elements would be smaller at the start of the line, and larger at the end of the line. If the value of **ratio1** is less than unity, then the reverse would be true. The default value of unity indicates uniform joint spacing.

See Figure 7 (page 35) for an example.

Two Dimensions

The two parameters **ratio1** and **ratio2** can be used to specify how the joint spacing varies along two generation dimensions. Parameter **ratio1** is defined above. Parameter **ratio2** is defined similarly: it gives the ratio of the last spacing, d_{2_n}, between joints **j2**–**ji2** and **j2**, and the first spacing, d_{2_1}, between joints **j0** and **j0**+**ji2**:

$$\textbf{ratio2} = \frac{d_{2_n}}{d_{2_1}}$$

If the value of **ratio1** is greater than unity, then the distance between joints increases along any straight line where the joint labels increment by **ji1**. Connected

elements would be smaller toward side **j0**-**j2**. If the value of **ratio1** is less than unity, then the reverse would be true. The default value of unity indicates uniform joint spacing.

Similarly, if the value of **ratio2** is greater than unity, then the distance between joints increases along any straight line where the joint labels increment by **ji2**. Connected elements would be smaller toward side **j0**-**j1**. If the value of **ratio2** is less than unity, then the reverse would be true. The default value of unity indicates uniform joint spacing.

Three Dimensions

The three parameters **ratio1**, **ratio2** and **ratio3** can be used to specify how the joint spacing varies along three generation dimensions. Parameters **ratio1** and **ratio2** are defined above. Parameter **ratio3** is defined similarly: it gives the ratio of the last spacing, d_{3_n}, between joints **j3**–**ji3** and **j3**, and the first spacing, d_{3_1}, between joints **j0** and **j0**+**ji3**:

$$\textbf{ratio3} = \frac{d_{3_n}}{d_{3_1}}$$

If the value of **ratio1** is greater than unity, then the distance between joints increases along any straight line where the joint labels increment by **ji1**. Connected elements would be smaller toward face **j0**-**j2**-**j3**. If the value of **ratio1** is less than unity, then the reverse would be true. The default value of unity indicates uniform joint spacing.

Similarly, if the value of **ratio2** is greater than unity, then the distance between joints increases along any straight line where the joint labels increment by **ji2**. Connected elements would be smaller toward face **j0**-**j1**-**j3**. If the value of **ratio2** is less than unity, then the reverse would be true. The default value of unity indicates uniform joint spacing.

Finally, if the value of **ratio3** is greater than unity, then the distance between joints increases along any straight line where the joint labels increment by **ji3**. Connected elements would be smaller toward face **j0**-**j1**-**j2**. If the value of **ratio3** is less than unity, then the reverse would be true. The default value of unity indicates uniform joint spacing.

Joint Definition in Polar Coordinates

Joint definition in polar coordinates as described in this topic is used only in the input data text file. Cylindrical coordinates are available in the SAP2000 graphical user interface through the use of cylindrical grids. You may skip this topic if you are preparing your model using the SAP2000 graphical user interface.

The location of the joints may be specified using polar coordinates instead of rectangular X-Y-Z coordinates. Polar coordinates include cylindrical CR-CA-CZ coordinates and spherical SB-SA-SR coordinates. See Figure 11 (page 51) for the definition of the polar coordinate systems. Polar coordinate systems are always defined with respect to a rectangular X-Y-Z system.

Polar coordinates may be used for the definition of a single joint or for the definition of joint arrays in one, two, or three dimensions. When defining a joint array, all specified coordinates must be of the same type: rectangular, cylindrical, or spherical.

Linear generation, frontal generation, and edge generation are always performed in rectangular coordinates, even if polar coordinates were used to locate the previously-defined joints.

The location of each joint is converted to rectangular coordinates in the global coordinate system for printing in the .EKO output file.

For more information:

- See Chapter “Coordinate Systems” (page 21).
- See Topic “Joint Definition” (page 32) in this chapter.
- See Subtopic “One-dimensional Joint Definition” (page 33) in this chapter.
- See Subtopic “Two-dimensional Joint Definition” (page 37) in this chapter.
- See Subtopic “Three-dimensional Joint Definition” (page 44) in this chapter.
- See Topic “Joint Definition in Alternate Coordinate Systems” (page 54) in this chapter.

Cylindrical Coordinates

The following coordinate values must be specified when defining joints in cylindrical coordinates:

- For a single joint, **j**, the coordinates (**cr**, **ca**, **cz**)

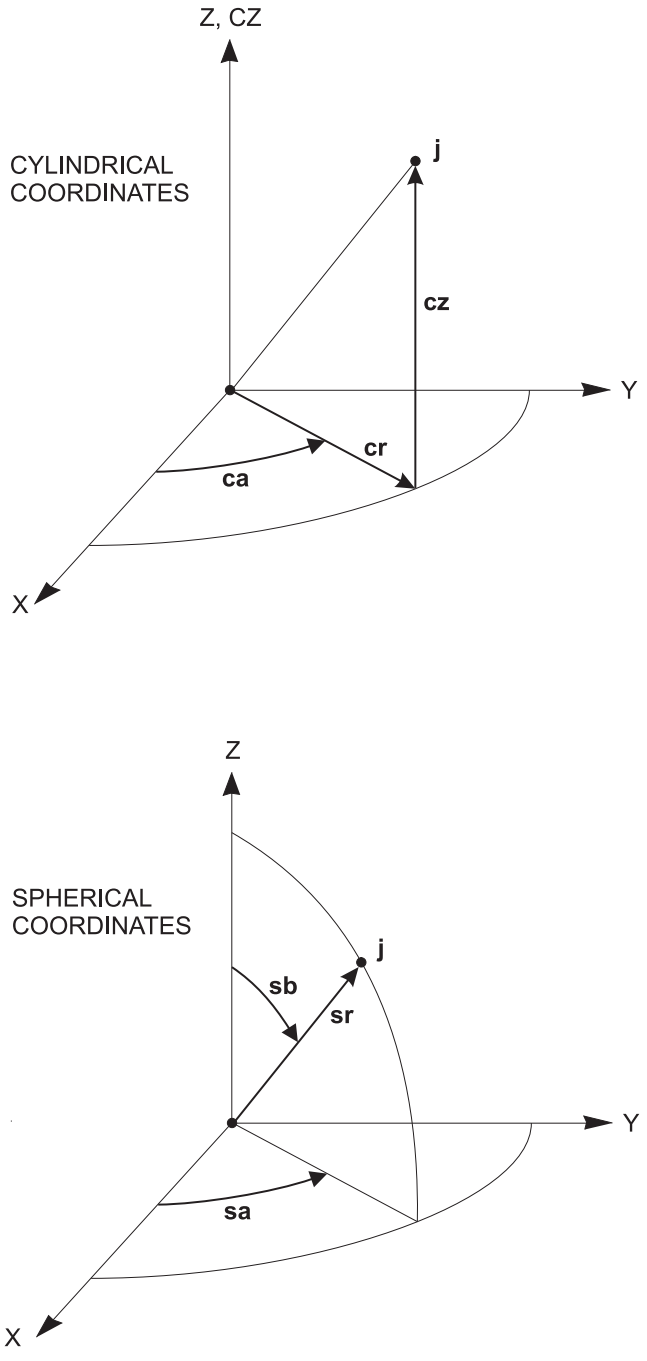


Figure 11
Locating a Joint Using Cylindrical or Spherical Coordinates

- For one-, two-, or three-dimensional joint definition, the coordinates (**cr0**, **ca0**, **cz0**) of joint **j0** and coordinates (**cr1**, **ca1**, **cz1**) of joint **j1**
- For two- or three-dimensional joint definition, the coordinates (**cr2**, **ca2**, **cz2**) of joint **j2**
- For three-dimensional joint definition, the coordinates (**cr3**, **ca3**, **cz3**) of joint **j3**

The spatial coordinates of the remaining joints are determined by linear interpolation and extrapolation of the cylindrical coordinate values. The constant coordinate differences, Δcr_1, Δca_1, and Δcz_1, between any pair of joints whose labels differ by **ji1** are given by:

$$\Delta cr_1 = \frac{\mathbf{cr1} - \mathbf{cr0}}{n_1}, \quad \Delta ca_1 = \frac{\mathbf{ca1} - \mathbf{ca0}}{n_1}, \quad \text{and} \quad \Delta cz_1 = \frac{\mathbf{cz1} - \mathbf{cz0}}{n_1}$$

Similar relationships exist for the constant coordinate differences between pairs of joints whose labels differ by **ji2** or **ji3** for two- and three-dimensional joint arrays.

See Figure 12 (page 53) for an example of one-dimensional joint array definition.

Spherical Coordinates

The following coordinate values must be specified when defining joints in spherical coordinates:

- For a single joint, **j**, the coordinates (**sb**, **sa**, **sr**)
- For one-, two-, or three-dimensional joint definition, the coordinates (**sb0**, **sa0**, **sr0**) of joint **j0** and coordinates (**sb1**, **sa1**, **sr1**) of joint **j1**
- For two- or three-dimensional joint definition, the coordinates (**sb2**, **sa2**, **sr2**) of joint **j2**
- For three-dimensional joint definition, the coordinates (**sb3**, **sa3**, **sr3**) of joint **j3**

The spatial coordinates of the remaining joints are determined by linear interpolation and extrapolation of the spherical coordinate values. The constant coordinate differences, Δsb_1, Δsa_1, and Δsr_1, between any pair of joints whose labels differ by **ji1** are given by:

$$\Delta sb_1 = \frac{\mathbf{sb1} - \mathbf{sb0}}{n_1}, \quad \Delta sa_1 = \frac{\mathbf{sa1} - \mathbf{sa0}}{n_1}, \quad \text{and} \quad \Delta sr_1 = \frac{\mathbf{sr1} - \mathbf{sr0}}{n_1}$$

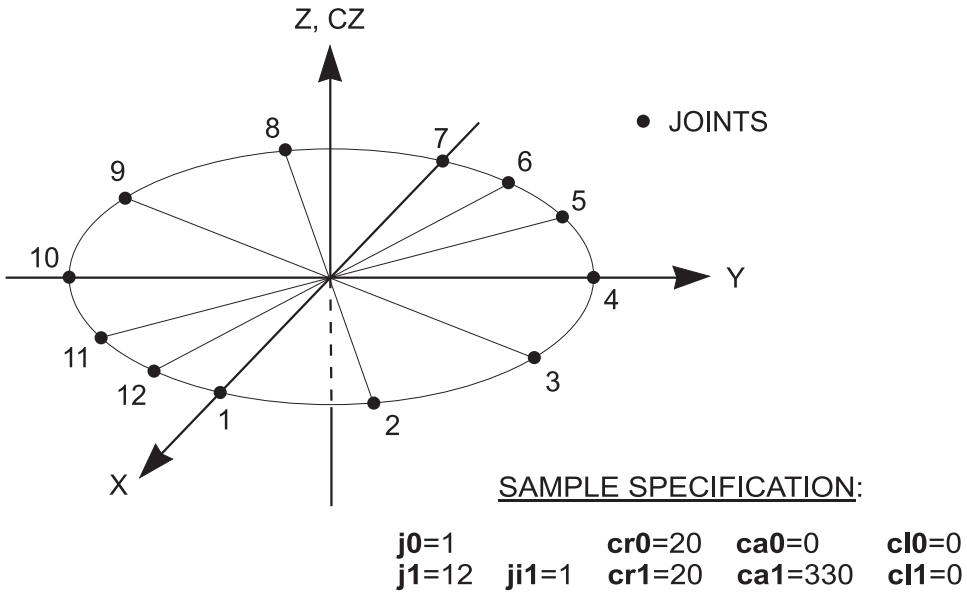


Figure 12
One-dimensional Joint Array Definition in Cylindrical Coordinates

Similar relationships exist for the constant coordinate differences between pairs of joints whose labels differ by **ji2** or **ji3** for two- and three-dimensional joint arrays.

Variable Joint Spacing in Polar Coordinates

The coordinate differences are normally constant between joints generated using one-, two-, three-dimensional joint array definition in polar coordinates. More specifically, equal cylindrical or spherical coordinate differences exist between the joints along any given path where the joint labels increment by **ji1**, **ji2**, or **ji3**. This does not correspond to constant spacing between the joints along this path unless only a single coordinate value is changing.

It is sometimes desirable to create a graded mesh where smaller elements are defined in one portion of the region. The parameters **ratio1**, **ratio2**, and **ratio3** can be specified for this purpose when using one-, two-, or three-dimensional joint array definition in polar coordinates.

In cylindrical coordinates, parameter **ratio1** gives the ratio of the coordinate differences, Δcr_{1_n}, Δca_{1_n}, and Δcz_{1_n}, between joints **j1**–**ji1** and **j1**, and the coordinate differences, Δcr_{1_1}, Δca_{1_1}, and Δcz_{1_1}, between joints **j0** and **j0**+**ji1**:

$$\textbf{ratio1} = \frac{\Delta cr_{1_n}}{\Delta cr_{1_1}} = \frac{\Delta ca_{1_n}}{\Delta ca_{1_1}} = \frac{\Delta cz_{1_n}}{\Delta cz_{1_1}}$$

In spherical coordinates the coordinate differences are Δsb_{1_n}, Δsa_{1_n}, and Δsr_{1_n} and Δsb_{1_1}, Δsa_{1_1}, and Δsr_{1_1}, respectively, and:

$$\textbf{ratio1} = \frac{\Delta sb_{1_n}}{\Delta sb_{1_1}} = \frac{\Delta sa_{1_n}}{\Delta sa_{1_1}} = \frac{\Delta sr_{1_n}}{\Delta sr_{1_1}}$$

Similar definitions apply for **ratio2** and **ratio3** for joints whose labels increment by **ji2** and **ji3**, respectively, in two- and three-dimensional joint arrays.

The coordinate differences in each case change by arithmetic progression so that the *difference* between successive coordinate differences is constant.

See Topic “Variable Joint Spacing” (page 47) in this chapter for more information.

Joint Definition in Alternate Coordinate Systems

Joint coordinates are normally given in the global coordinate system. However, it is sometimes convenient to define different parts of a structure in separate coordinate systems. Each coordinate system has its own origin and orientation of the axes. For example, you might use a separate coordinate system for each wing of a building or each bent of a bridge.

The parameter **csys** is used to specify which coordinate system is being used to define joint coordinates:

- If **csys**=0, the global coordinate system is used; this is the default
- Otherwise, **csys** is the label of an Alternate Coordinate System

A different value of **csys** may be used for each joint or set of joints. The coordinates of a joint may be specified in rectangular, cylindrical, or spherical coordinates with respect to coordinate system **csys**.

The choice of the coordinate system affects only the definition of individual joints and joint arrays in one, two, or three dimensions. Linear generation, frontal genera-

tion, and edge generation are always performed in rectangular coordinates in the global coordinate system.

The location of each joint is converted to rectangular coordinates in the global coordinate system for printing in the .EKO output file.

It is important to understand that the use of an Alternate Coordinate System to locate a joint has no effect upon the local coordinate system defined for that joint.

For more information:

- See Chapter “Coordinate Systems” (page 21).
- See Topic “Joint Definition” (page 32) in this chapter.
- See Subtopic “One-dimensional Joint Definition” (page 33) in this chapter.
- See Subtopic “Two-dimensional Joint Definition” (page 37) in this chapter.
- See Subtopic “Three-dimensional Joint Definition” (page 44) in this chapter.
- See Topic “Joint Definition in Polar Coordinates” (page 50) in this chapter.

Chapter V

Joint Patterns

Joint **Patterns** can be used to describe complicated spatial distributions of loads and properties over the structure. Patterns are optional and are not required for simple problems.

This chapter describes the definition of the joint Patterns. Their use is described in Chapter “Joints and Degrees of Freedom” (page 75) and Chapter “Load Cases” (page 279).

Advanced Topics

- Overview
- Pattern Definition
- Gradient or Hydrostatic Specification
- One-dimensional Pattern-value Generation
- Two-dimensional Pattern-value Generation
- Three-dimensional Pattern-value Generation
- Variable Pattern-value Increments

Overview

A joint Pattern consists of a set of scalar numeric values, one value for each joint of the structure. A Pattern can be used to describe how certain loads and properties vary over the structure:

- Joint spring and mass properties
- Joint force and displacement loads
- Pressure-type loads
- Temperature-type loads

Patterns are most effective for describing complicated spatial distributions of numeric values. Their use is optional and is not required for simple problems.

Joint Patterns do not affect the location of the joints, or have any other effect upon the structure except as utilized in the specification of loads and properties.

Pattern values are assigned to previously-defined joints. The specification of Pattern values is very similar to the definition of the joint coordinates, except that only a single value, **v**, is assigned to a joint in a Pattern, whereas three coordinate values, (**x**, **y**, **z**), are assigned during joint definition. You may define as many Patterns as you need, each having a single value at every joint.

The generation options available for specifying joint Pattern values correspond exactly to the generation options for joint coordinates. If the given Pattern values for a generation specification are proportional to the **x**, **y**, or **z** coordinates of the joints, then all generated values will also be proportional to **x**, **y**, or **z**, respectively. This is guaranteed by the properties of isoparametric interpolation (Cook, Malkus, and Plesha, 1989).

Pattern values may also be specified that vary linearly in a given gradient direction from zero at a given datum point. An option is available to permit only positive or only negative values to be defined. This is useful for defining hydrostatic pressure distributions.

Since Pattern values are scalar quantities, they are independent of any coordinate system.

For more information:

- See Chapter “Joints and Degrees of Freedom” (page 75).
- See Chapter “Load Cases” (page 279).

Pattern Definition

For each Pattern to be defined, you must assign a label that will be referenced whenever you later use the Pattern to describe the variation of a load or property.

Each Pattern starts with a numeric value of zero at every joint. You then define the numeric values at the joints using a combination of addition, resetting, and mapping specifications as needed:

- Each **addition specification** adds numeric values to one or more joints. The assigned values are *algebraically added* to the values that are already present at the joints.
- Each **resetting specification** resets the numeric values to zero at one or more joints.
- Each **mapping specification** interpolates or extrapolates numeric values from certain joints in an array to the remaining joints in the array. Mapping is *not additive*, but overwrites previous numeric values at the joints.

These specifications are applied separately to each Pattern being defined.

Addition and resetting specifications may be applied to individual joints or to arrays of joints. Mapping specifications apply only to joint arrays. The application of these specifications to arrays of one, two, and three dimensions is described in the remainder of this chapter.

Gradient or Hydrostatic Specification

You may define Pattern values for a given array of joints that vary linearly along a specified gradient direction from zero value at a specified datum point. You have the option of allowing only negative values, only positive values, or both to be generated this way. For example, only positive values would be desired when defining a Pattern for hydrostatic pressure.

Any Pattern values defined by a gradient specification are algebraically additive to the values already present at the joints of the given array.

The following parameters are needed for a gradient specification:

- A fixed coordinate system, **csys** (the default is zero, indicating the global coordinate system)
- The components of the gradient, **vx**, **vy**, and **vz**, in coordinate system **csys**

- The coordinates of the zero datum point, **x**, **y**, and **z**, in coordinate system **csys**
- The parameter **setzero** which can take on values:
  - NEG, indicating to set negative values to zero
  - POS, indicating to set positive values to zero
  - NO, indicating to set no values to zero (this is the default)

The component **vx** indicates, for example, how much the Pattern value changes per unit of distance parallel to the X axis in coordinate system **csys**.

The Pattern value, v_j, defined for a joint j that has coordinates (x_j, y_j, z_j) is given by:

$$v_j = \mathbf{vx}\,(x_j - \mathbf{x}) + \mathbf{vy}\,(y_j - \mathbf{y}) + \mathbf{vz}\,(z_j - \mathbf{z}) \qquad \text{(Eqn. 1)}$$

if $v_j < 0$ and **setzero** = NEG, then $v_j = 0$

if $v_j > 0$ and **setzero** = POS, then $v_j = 0$

In most cases, the gradient will be parallel to one of the coordinate axes, and only one term in the above equation is needed.

For example, consider a hydrostatic pressure distribution caused by water impounded behind a dam as shown in Figure 13 (page 61). The Z direction is up in the global coordinate system. The pressure gradient is simply given by the fluid weight density acting in the downward direction. Therefore, **vx** = 0, **vy** = 0 , and **vz** = –62.4 lb/ft^3 or –9810 N/m^3. The zero-pressure datum can be any point on the free surface of the water. Thus **z** should be set to the elevation of the free surface in feet or meters, as appropriate. For hydrostatic pressure, the parameter **setzero** should be specified as NEG so that any joints above the free surface will be assigned a zero value for pressure.

One-dimensional Pattern-value Generation

Pattern-value generation as described in this topic is used only in the input data text file. You may skip this topic if you are preparing your model using the SAP2000 graphical user interface.

Pattern values can be generated for a one-dimensional array of joints, i.e., a set of joints whose labels increment uniformly along a line or curve. The generation options include addition, resetting, and linear mapping.

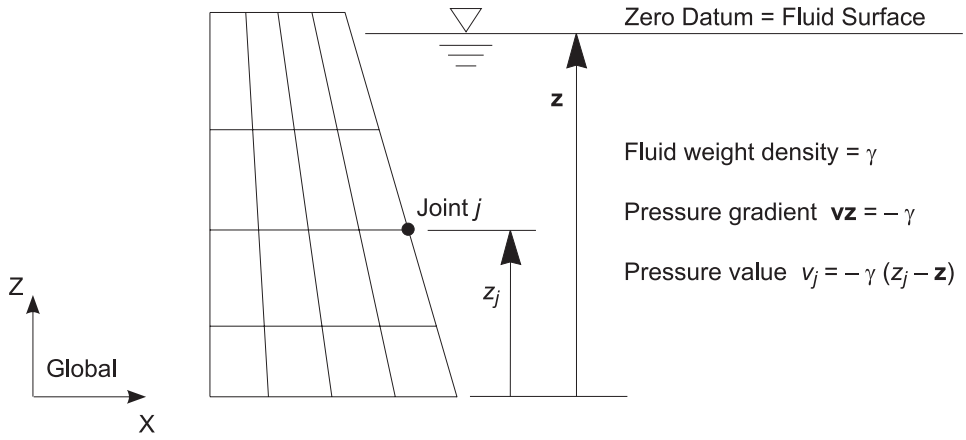


Figure 13
Example of a Hydrostatic Pressure Pattern

Each generation option requires the specification of a joint array for which the Pattern values are to be generated, as described below. No other information is required for a resetting specification: the Pattern values for all joints in the array are set to zero. The additional information that is required for addition and linear-mapping specifications is described in the subtopics below.

See Subtopic “One-dimensional Regular Arrays” (page 11) in Chapter “Labels, Arrays, and Generation” for more information.

Joint Array Specification

Every one-dimensional generation specification requires the following information to describe the array of joints to which Pattern values are being assigned:

- The label, **j0**, of the joint at the start of the line or curve
- The label, **j1**, of the joint at the end of the line or curve
- The label increment (constant difference), **ji1**, between any pair of successive joints along the line or curve

The number of spaces, n_1, between the joints along the line or curve is then given by:

$$n_1 = \frac{\mathbf{j1 - j0}}{\mathbf{ji1}}$$

The labels and label increment must be chosen such that n_1 is a whole, positive number. The total number of joints in the array, including the ends, is n_1+1.

One-dimensional Addition

A set of uniformly incrementing Pattern values can be added to the joints along a line or curve by specifying:

- The information in Subtopic "Joint Array Specification" above
- The Pattern value, **v0**, to be added to joint **j0** at the start of the line or curve
- The Pattern value, **v1**, to be added to joint **j1** at the end of the line or curve

The Pattern values added to the intermediate joints are determined by linear interpolation from these values such that the constant difference, Δv_1, between the values added to any pair of successive joints is given by:

$$\Delta v_1 = \frac{\mathbf{v1 - v0}}{n_1}$$

The computed values are *algebraically added* to the values that are already present at the joints.

One-dimensional Gradient Addition

A set of Pattern values determined from a constant gradient can be added to the joints along a line or curve by specifying:

- The information in Subtopic "Joint Array Specification" above
- The information in Topic "Gradient or Hydrostatic Specification"

The Pattern values added to the joints are determined by Equation 1 in Topic "Gradient or Hydrostatic Specification" (page 59) in this chapter.

The computed values are *algebraically added* to the values that are already present at the joints.

One-dimensional Linear Mapping

One-dimensional linear mapping interpolates (maps) Pattern values from existing values at the ends of a line or curve. Before using this option, the Pattern values at the following joints must have already been defined (or allowed to default to zero):

1. Joint **j0** at the start of the line or curve
2. Joint **j1** at the end of the line or curve

It is then only necessary to provide the information in Subtopic "Joint Array Specification" above.

The Pattern values assigned to the intermediate joints are determined by linear interpolation such that the constant value difference, Δv_1, between any pair of successive joints is given by:

$$\Delta v_1 = \frac{\mathbf{v1} - \mathbf{v0}}{n_1}$$

The assigned values are *not additive*, but instead they replace (overwrite) the values present at the joints.

Two-dimensional Pattern-value Generation

Pattern-value generation as described in this topic is used only in the input data text file. You may skip this topic if you are preparing your model using the SAP2000 graphical user interface.

Pattern values can be generated for a two-dimensional array of joints, i.e., a set of joints whose labels increment in a uniform fashion in two separate directions. This will typically correspond to a four-sided region on a flat or curved surface. The generation options include addition, resetting, and several types of mapping.

Each generation option requires the specification of a joint array for which the Pattern values are to be generated. This is described in Subtopic "Joint Array Specification" below. No other information is required for a resetting specification: the Pattern values for all joints in the array are set to zero. The additional information that is required for addition and mapping specifications is described in the remaining subtopics below.

See Subtopic "Two-dimensional Regular Arrays" (page 12) in Chapter "Labels, Arrays, and Generation" for more information.

Joint Array Specification

Each two-dimensional generation specification requires the following information to describe the array of joints to which Pattern values are being assigned:

- The label, **j0**, of the joint at one corner
- The label, **j1**, of the joint at the end of one side that contains joint **j0**
- The label, **j2**, of the joint at the end of the other side that contains joint **j0**
- The label increment (constant difference), **ji1**, between any pair of successive joints along side **j0**-**j1**
- The label increment (constant difference), **ji2**, between any pair of successive joints along side **j0**-**j2**

The numbers of spaces, n_1 and n_2, between the joints along directions "parallel" to sides **j0**-**j1** and **j0**-**j2**, respectively, are then given by:

$$n_1 = \frac{\mathbf{j1} - \mathbf{j0}}{\mathbf{ji1}} \quad \text{and} \quad n_2 = \frac{\mathbf{j2} - \mathbf{j0}}{\mathbf{ji2}}$$

The labels and label increments must be chosen such that n_1 and n_2 are whole, positive numbers. The total number of joints in the array is $(n_1 + 1)(n_2 + 1)$.

Two-dimensional Addition

A set of uniformly incrementing Pattern values can be added to the joints in a two-dimensional array by specifying:

- The information in Subtopic "Joint Array Specification" above
- The Pattern value, **v0**, to be added to joint **j0**
- The Pattern value, **v1**, to be added to joint **j1**
- The Pattern value, **v2**, to be added to joint **j2**

The Pattern values added to the remaining joints are determined by linear interpolation and extrapolation from these values. The constant difference, Δv_1, between the values added to any pair of joints whose labels differ by **ji1** is given by:

$$\Delta v_1 = \frac{\mathbf{v1} - \mathbf{v0}}{n_1}$$

Similarly, the constant difference, Δv_2, between the values added to any pair of joints whose labels differ by **ji2** is given by:

$$\Delta v_2 = \frac{\mathbf{v2 - v0}}{n_2}$$

The computed values are *algebraically added* to the values that are already present at the joints.

Two-dimensional Gradient Addition

A set of Pattern values determined from a constant gradient can be added to the joints in a two-dimensional array by specifying:

- The information in Subtopic "Joint Array Specification" above
- The information in Topic "Gradient or Hydrostatic Specification"

The Pattern values added to the joints are determined by Equation 1 in Topic "Gradient or Hydrostatic Specification" (page 59) in this chapter.

The computed values are *algebraically added* to the values that are already present at the joints.

Two-dimensional Linear Mapping

Two-dimensional linear mapping interpolates (maps) a set of Pattern values over a four-sided region from existing values at the four corners. Before using this option, the Pattern values at the corner joints must have already been defined (or allowed to default to zero):

1. Joint **j0**
2. Joint **j1**
3. Joint **j2**
4. Joint **j1**+**j2**–**j0** at the corner opposite joint **j0**

It is then only necessary to provide the information in Subtopic "Joint Array Specification" above.

The Pattern values assigned to the remaining joints are determined by standard bilinear interpolation. Any subset of the joints having labels that increment uniformly by **ji1** or **ji2** will have a constant difference between the generated Pattern values.

The assigned values are *not additive*, but instead they replace (overwrite) the values present at the joints.

Two-dimensional Frontal Mapping

Two-dimensional frontal mapping extrapolates (maps) a set of Pattern values over a four-sided region from existing values along two adjacent sides. Before using this option, the Pattern values at all joints on the two adjacent sides must have already been defined (or allowed to default to zero):

1. Joints **j0** to **j1** by increment **ji1** along one side
2. Joints **j0** to **j2** by increment **ji2** along the adjacent side

It is then only necessary to provide the information in Subtopic "Joint Array Specification" above.

The Pattern values assigned to the remaining joints are determined so that:

- Any subset of the joints having labels that increment uniformly by **ji1** will have differences between the generated Pattern values that are the same as for the corresponding joints along side **j0**-**j1**
- Any subset of the joints having labels that increment uniformly by **ji2** will have differences between the generated Pattern values that are the same as for the corresponding joints along side **j0**-**j2**

The assigned values are *not additive*, but instead they replace (overwrite) the values present at the joints.

Two-dimensional Edge Mapping

Two-dimensional edge mapping interpolates (maps) a set of Pattern values over a four-sided region from existing values along all four sides. Before using this option, the Pattern values at all joints on the four sides must have already been defined (or allowed to default to zero):

1. Joints **j0** to **j1** by increment **ji1**
2. Joints **j0** to **j2** by increment **ji2**
3. Joints **j2** to **j1**+**j2**–**j0** by increment **ji1**
4. Joints **j1** to **j1**+**j2**–**j0** by increment **ji2**

It is then only necessary to provide the information in Subtopic "Joint Array Specification" above.

The Pattern values assigned to the remaining joints are determined by serendipity polynomial interpolation (Cook, Malkus, and Plesha, 1989).

The assigned values are *not additive*, but instead they replace (overwrite) the values present at the joints.

Three-dimensional Pattern-value Generation

Pattern-value generation as described in this topic is used only in the input data text file. You may skip this topic if you are preparing your model using the SAP2000 graphical user interface.

Pattern values can be generated for a three-dimensional array of joints, i.e., a set of joints whose labels increment in a uniform fashion in three separate directions. This will typically correspond to a three-dimensional region enclosed by a hexahedron, i.e., having six faces. The generation options include addition, resetting, and several types of mapping.

Each generation option requires the specification of a joint array for which the Pattern values are to be generated. This is described in Subtopic "Joint Array Specification" below. No other information is required for a resetting specification: the Pattern values for all joints in the array are set to zero. The additional information that is required for addition and mapping specifications is described in the remaining subtopics below.

Three-dimensional Pattern-value generation is a generalization of two-dimensional Pattern-value generation.

For more information:

- See Topic "Two-dimensional Pattern-value Generation" (page 63) in this chapter.
- See Subtopic "Three-dimensional Regular Arrays" (page 13) in Chapter "Labels, Arrays, and Generation."

Joint Array Specification

Each three-dimensional generation specification requires the following information to describe the array of joints to which Pattern values are being assigned:

- The label, **j0**, of the joint at one corner
- The label, **j1**, of the joint at the end of one side that contains joint **j0**
- The label, **j2**, of the joint at the end of another side that contains joint **j0**
- The label, **j3**, of the joint at the end of the third side that contains joint **j0**
- The label increment (constant difference), **ji1**, between any pair of successive joints along side **j0**-**j1**
- The label increment (constant difference), **ji2**, between any pair of successive joints along side **j0**-**j2**
- The label increment (constant difference), **ji3**, between any pair of successive joints along side **j0**-**j3**

The numbers of spaces, n_1, n_2, and n_3, between the joints along directions "parallel" to sides **j0**-**j1**, **j0**-**j2**, and **j0**-**j3**, respectively, are then given by:

$$n_1 = \frac{\mathbf{j1 - j0}}{\mathbf{ji1}}, \quad n_2 = \frac{\mathbf{j2 - j0}}{\mathbf{ji2}}, \quad \text{and} \quad n_3 = \frac{\mathbf{j3 - j0}}{\mathbf{ji3}}$$

The labels and label increments must be chosen such that n_1, n_2, and n_3 are whole, positive numbers. The total number of joints in the array is $(n_1 + 1)(n_2 + 1)(n_3 + 1)$.

Three-dimensional Addition

A set of uniformly incrementing Pattern values can be added to the joints in a three-dimensional array by specifying:

- The information in Subtopic "Joint Array Specification" above
- The Pattern value, **v0**, to be added to joint **j0**
- The Pattern value, **v1**, to be added to joint **j1**
- The Pattern value, **v2**, to be added to joint **j2**
- The Pattern value, **v3**, to be added to joint **j3**

The Pattern values added to the remaining joints are determined by linear interpolation and extrapolation from these values. The constant difference, Δv_1, between the values added to any pair of joints whose labels differ by **ji1** is given by:

$$\Delta v_1 = \frac{\mathbf{v1 - v0}}{n_1}$$

Similarly, the constant difference, Δv_2, between the values added to any pair of joints whose labels differ by **ji2** is given by:

$$\Delta v_2 = \frac{\mathbf{v2} - \mathbf{v0}}{n_2}$$

Finally, the constant difference, Δv_3, between the values added to any pair of joints whose labels differ by **ji3** is given by:

$$\Delta v_3 = \frac{\mathbf{v3} - \mathbf{v0}}{n_3}$$

The computed values are *algebraically added* to the values that are already present at the joints.

Three-dimensional Gradient Addition

A set of Pattern values determined from a constant gradient can be added to the joints in a three-dimensional array by specifying:

- The information in Subtopic “Joint Array Specification” above
- The information in Topic “Gradient or Hydrostatic Specification”

The Pattern values added to the joints are determined by Equation 1 in Topic “Gradient or Hydrostatic Specification” (page 59) in this chapter.

The computed values are *algebraically added* to the values that are already present at the joints.

Three-dimensional Linear Mapping

Three-dimensional linear mapping interpolates (maps) a set of Pattern values over a hexahedral (six-faced) region from existing values at the eight corners. Before using this option, the Pattern values at the corner joints must have already been defined (or allowed to default to zero):

1. Joint **j0**
2. Joint **j1**
3. Joint **j2**
4. Joint **j3**

5. Joint **j1**+**j2**–**j0** at the corner opposite joint **j3**
6. Joint **j1**+**j3**–**j0** at the corner opposite joint **j2**
7. Joint **j2**+**j3**–**j0** at the corner opposite joint **j1**
8. Joint **j1**+**j2**+**j3**–**j0**–**j0** at the corner opposite joint **j0**

It is then only necessary to provide the information in Subtopic "Joint Array Specification" above.

The Pattern values assigned to the remaining joints are determined by standard trilinear interpolation. Any subset of the joints having labels that increment uniformly by **ji1**, **ji2**, or **ji3** will have a constant difference between the generated Pattern values.

The assigned values are *not additive*, but instead they replace (overwrite) the values present at the joints.

Three-dimensional Frontal Mapping

Three-dimensional frontal mapping extrapolates (maps) a set of Pattern values over a hexahedral (six-faced) region from existing values along three adjacent edges. Before using this option, the Pattern values at all joints on the three adjacent edges must have already been defined (or allowed to default to zero):

1. Joints **j0** to **j1** by increment **ji1** along one edge
2. Joints **j0** to **j2** by increment **ji2** along an adjacent edge
3. Joints **j0** to **j3** by increment **ji3** along the other adjacent edge

It is then only necessary to provide the information in Subtopic "Joint Array Specification" above.

The Pattern values assigned to the remaining joints are determined so that:

- Any subset of the joints having labels that increment uniformly by **ji1** will have differences between the generated Pattern values that are the same as for the corresponding joints along edge **j0**-**j1**
- Any subset of the joints having labels that increment uniformly by **ji2** will have differences between the generated Pattern values that are the same as for the corresponding joints along edge **j0**-**j2**

- Any subset of the joints having labels that increment uniformly by **ji3** will have differences between the generated Pattern values that are the same as for the corresponding joints along edge **j0-j3**

The assigned values are *not additive*, but instead they replace (overwrite) the values present at the joints.

Three-dimensional Edge Mapping

Three-dimensional edge mapping interpolates (maps) a set of Pattern values over a hexahedral (six-faced) region from existing values along all twelve edges. Before using this option, the Pattern values at all joints on the twelve edges must have already been defined (or allowed to default to zero):

1. Joints **j0** to **j1** by increment **ji1**
2. Joints **j0** to **j2** by increment **ji2**
3. Joints **j0** to **j3** by increment **ji3**
4. Joints **j1** to **j1**+**j2**–**j0** by increment **ji2**
5. Joints **j1** to **j1**+**j3**–**j0** by increment **ji3**
6. Joints **j2** to **j1**+**j2**–**j0** by increment **ji1**
7. Joints **j2** to **j2**+**j3**–**j0** by increment **ji3**
8. Joints **j3** to **j1**+**j3**–**j0** by increment **ji1**
9. Joints **j3** to **j2**+**j3**–**j0** by increment **ji2**
10. Joints **j1**+**j2**–**j0** to **j1**+**j2**+**j3**–**j0**–**j0** by increment **ji3**
11. Joints **j1**+**j3**–**j0** to **j1**+**j2**+**j3**–**j0**–**j0** by increment **ji2**
12. Joints **j2**+**j3**–**j0** to **j1**+**j2**+**j3**–**j0**–**j0** by increment **ji1**

It is then only necessary to provide the information in Subtopic “Joint Array Specification” above.

The Pattern values assigned to the remaining joints are determined by serendipity polynomial interpolation (Zienkiewicz and Taylor, 1989).

The assigned values are *not additive*, but instead they replace (overwrite) the values present at the joints.

Variable Pattern-value Increments

Variable pattern-value increments as described in this topic are used only in the input data text file. You may skip this topic if you are preparing your model using the SAP2000 graphical user interface.

Pattern values generated by an addition specification or a linear-mapping specification normally have a constant increment (difference) between successive joints along any line or curve in a joint array, provided that the joint labels differ by one of the specified label increments.

Sometimes, however, it is desirable to have the increment in Pattern values vary gradually along these lines or curves. The principal situation where this is useful is when you want the Pattern values to be proportional to the *x*, *y*, or *z* coordinates of the joints and the joint spacing is not constant, as in a graded mesh.

The addition and linear-mapping specifications for Pattern values permit the use of the parameters **ratio1**, **ratio2**, and **ratio3** to control the variation of generated Pattern values. The effect of these parameters is analogous to their effect upon joint spacing in joint array definition and joint linear mapping.

To generate Pattern values proportional to the **x**, **y**, or **z** coordinates of joints that were created using joint array definition:

- Use an addition specification for the same joint array
- Specify pattern values that are proportional to the **x**, **y**, or **z** coordinates of the joints, as desired
- Specify the same values of the parameters **ratio1**, **ratio2**, and **ratio3** as used for the joint array definition

To generate Pattern values proportional to the **x**, **y**, or **z** coordinates of joints that were created using joint linear generation:

- Use a linear-mapping specification for the same joint array
- Make sure that the Pattern values at the corners have already been defined to be proportional to the **x**, **y**, or **z** coordinates of the joints, as desired
- Specify the same values of the parameters **ratio1**, **ratio2**, and **ratio3** as used for the joint linear generation

The increments in Pattern values change by arithmetic progression so that the *difference* between successive increments is constant.

This option is only available for addition and linear-mapping specifications. For frontal-mapping and edge-mapping specifications, the increments in Pattern values is determined from the values at the joints along the previously-defined edges.

The **ratio** parameters are described in more detail in the subtopics below.

For more information:

- See Topic "Variable Joint Spacing" (page 47) in Chapter "Joint Coordinates."
- See Subtopic "One-dimensional Addition" (page 62) in this chapter.
- See Subtopic "One-dimensional Linear Mapping" (page 63) in this chapter.
- See Subtopic "Two-dimensional Addition" (page 64) in this chapter.
- See Subtopic "Two-dimensional Linear Mapping" (page 65) in this chapter.
- See Subtopic "Three-dimensional Addition" (page 68) in this chapter.
- See Subtopic "Three-dimensional Linear Mapping" (page 69) in this chapter.

One Dimension

The parameter **ratio1** can be used to specify how the Pattern-value increment varies along the line of generation. It gives the ratio of the last increment, Δv_{1_n}, between the values at joints **j1**–**ji1** and **j1**, and the first increment, Δv_{1_1}, between the values at joints **j0** and **j0**+**ji1**:

$$\textbf{ratio1} = \frac{\Delta v_{1_n}}{\Delta v_{1_1}}$$

If the value of **ratio1** is greater than unity, then the increment increases along the length. If the value of **ratio1** is less than unity, then the reverse would be true. The default value of unity indicates a uniform Pattern-value increment.

Two Dimensions

The two parameters **ratio1** and **ratio2** can be used to specify how the Pattern-value increment varies along two generation dimensions. Parameter **ratio1** is defined above. Parameter **ratio2** is defined similarly: it gives the ratio of the last increment, Δv_{2_n}, between the values at joints **j2**–**ji2** and **j2**, and the first increment, Δv_{2_1}, between the values at joints **j0** and **j0**+**ji2**:

$$\textbf{ratio2} = \frac{\Delta v_{2_n}}{\Delta v_{2_1}}$$

If the value of **ratio1** is greater than unity, then the increment increases away from side **j0**-**j2** along any line or curve where the joint labels increment by **ji1**. If the value of **ratio1** is less than unity, then the reverse would be true. The default value of unity indicates a uniform Pattern-value increment.

Similarly, if the value of **ratio2** is greater than unity, then the increment increases away from side **j0**-**j1** along any line or curve where the joint labels increment by **ji2**. If the value of **ratio2** is less than unity, then the reverse would be true. The default value of unity indicates a uniform Pattern-value increment.

Three Dimensions

The three parameters **ratio1**, **ratio2** and **ratio3** can be used to specify how the Pattern-value increment varies along three generation dimensions. Parameters **ratio1** and **ratio2** are defined above. Parameter **ratio3** is defined similarly: it gives the ratio of the last increment, Δv_{3_n}, between the values at joints **j3**–**ji3** and **j3**, and the first increment, Δv_{3_1}, between the values at joints **j0** and **j0**+**ji3**:

$$\textbf{ratio3} = \frac{\Delta v_{3_n}}{\Delta v_{3_1}}$$

If the value of **ratio1** is greater than unity, then the increment increases away from face **j0**-**j2**-**j3** along any line or curve where the joint labels increment by **ji1**. If the value of **ratio1** is less than unity, then the reverse would be true. The default value of unity indicates a uniform Pattern-value increment.

Similarly, if the value of **ratio2** is greater than unity, then the increment increases away from face **j0**-**j1**-**j3** along any line or curve where the joint labels increment by **ji2**. If the value of **ratio2** is less than unity, then the reverse would be true. The default value of unity indicates a uniform Pattern-value increment.

Finally, if the value of **ratio3** is greater than unity, then the increment increases away from face **j0**-**j1**-**j2** along any line or curve where the joint labels increment by **ji3**. If the value of **ratio3** is less than unity, then the reverse would be true. The default value of unity indicates a uniform Pattern-value increment.

Chapter VI

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**.

This chapter describes joint properties, degrees of freedom, loads, and output. Chapter “Joint Coordinates” (page 31) describes the use of joints to define the geometry of the structure. Additional information about joints and degrees of freedom is given in Chapter “Constraints and Welds” (page 113).

Basic Topics for All Users

- Overview
- Modeling Considerations
- Local Coordinate System
- Degrees of Freedom
- Restraints and Reactions
- Springs
- Masses

- Force Load
- Restraint Displacement Load
- Spring Displacement Load
- Degree of Freedom Output
- Joint Mass Output
- Displacement and Acceleration Output
- Force Output
- Global Force Balance Output

Advanced Topics

- Advanced Local Coordinate System

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 “Constraints and Welds” (page 113).

The method used to define the structural model affects how joints are created:

- Using the SAP2000 graphical interface — joints are automatically created at the ends of each Frame or Nllink element and at the corners of each Shell element; additional joints may also be defined independently of any element
- Using the input data text file — joint locations must be explicitly defined in order to describe the geometry of the structure; these joints are then connected by elements to build the structure

The use of joints to define the geometry of the structure is discussed in Chapter "Joint Coordinates" (page 31).

Joints act independently of each other unless connected by elements. Frame and Nllink elements connect to the joints at the element ends. The two- and three-dimensional elements connect to the joints at the element corners, and possibly at other points on the element.

The joints may themselves be considered as elements with no connectivity. 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 though 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:

- See Chapter "Joint Coordinates" (page 31).
- See Chapter "Constraints and Welds" (page 113).

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 the 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.
- More that 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 the next topic.

For more information:

- See Topic “Upward and Horizontal Directions” (page 23) in Chapter “Coordinate Systems.”
- See Topic “Advanced Local Coordinate System” (page 79) in this chapter.

Advanced Local Coordinate System

By default, the joint local 1-2-3 coordinate system is identical to the global X-Y-Z coordinate system, as described in the previous topic. However, it may be necessary to use different local coordinate systems at some or all joints in the following cases:

- Skewed Restraints (supports) are present
- Constraints are used to impose rotational symmetry
- Constraints are used to impose symmetry about a plane that is not parallel to a global coordinate plane
- The principal axes for the joint mass (translational or rotational) are not aligned with the global axes
- Joint displacement and force output is desired in another coordinate system

Joint local coordinate systems need only be defined for the affected joints. The global system is used for all joints for which no local coordinate system is explicitly specified.

A variety of methods are available to define a joint local coordinate system. These may be used separately or together. Local coordinate axes may be defined to be parallel to arbitrary coordinate directions in an arbitrary coordinate system or to vectors between pairs of joints. In addition, the joint local coordinate system may be specified by a set of three joint coordinate angles. These methods are described in the subtopics that follow.

For more information:

- See Chapter “Coordinate Systems” (page 21).
- See Topic “Local Coordinate System” (page 78) in this chapter.

Reference Vectors

To define a joint local coordinate system you must specify two **reference vectors** that are parallel to one of the joint local coordinate planes. The **axis reference vector**, $\mathbf{V_a}$, must be parallel to one of the local axes (i = 1, 2, or 3) in this plane and have a positive projection upon that axis. The **plane reference vector**, $\mathbf{V_p}$, must have a positive projection upon the other local axis (j = 1, 2, or 3, but $i \neq j$) in this plane, but need not be parallel to that axis. Having a positive projection means that the posi-

tive direction of the reference vector must make an angle of less than 90° with the positive direction of the local axis.

Together, the two reference vectors define a local axis, *i*, and a local plane, *i-j*. From this, the program can determine the third local axis, *k*, using vector algebra.

For example, you could choose the axis reference vector parallel to local axis 1 and the plane reference vector parallel to the local 1-2 plane ($i = 1, j = 2$). Alternatively, you could choose the axis reference vector parallel to local axis 3 and the plane reference vector parallel to the local 3-2 plane ($i = 3, j = 2$). You may choose the plane that is most convenient to define using the parameter **local**, which may take on the values 12, 13, 21, 23, 31, or 32. The two digits correspond to *i* and *j*, respectively. The default is value is 31.

Defining the Axis Reference Vector

To define the axis reference vector for joint **j**, you must first specify or use the default values for:

- A coordinate direction **axdir** (the default is +Z)
- A fixed coordinate system **csys** (the default is zero, indicating the global coordinate system)

You may optionally specify:

- A pair of joints, **axveca** and **axvecb** (the default for each is zero, indicating joint **j** itself). If both are zero, this option is not used.

For each joint, the axis reference vector is determined as follows:

1. A vector is found from joint **axveca** to joint **axvecb**. If this vector is of finite length, it is used as the reference vector $\mathbf{V}_a$
2. Otherwise, the coordinate direction **axdir** is evaluated at joint **j** in fixed coordinate system **csys**, and is used as the reference vector $\mathbf{V}_a$

Defining the Plane Reference Vector

To define the plane reference vector for joint **j**, you must first specify or use the default values for:

- A primary coordinate direction **pldirp** (the default is +X)

- A secondary coordinate direction **pldirs** (the default is +Y). Directions **pldirs** and **pldirp** should not be parallel to each other unless you are sure that they are not parallel to local axis 1
- A fixed coordinate system **csys** (the default is zero, indicating the global coordinate system). This will be the same coordinate system that was used to define the axis reference vector, as described above

You may optionally specify:

- A pair of joints, **plveca** and **plvecb** (the default for each is zero, indicating joint **j** itself). If both are zero, this option is not used.

For each joint, the plane reference vector is determined as follows:

1. A vector is found from joint **plveca** to joint **plvecb**. If this vector is of finite length and is not parallel to local axis *i*, it is used as the reference vector $\mathbf{V}_p$
2. Otherwise, the primary coordinate direction **pldirp** is evaluated at joint **j** in fixed coordinate system **csys**. If this direction is not parallel to local axis *i*, it is used as the reference vector $\mathbf{V}_p$
3. Otherwise, the secondary coordinate direction **pldirs** is evaluated at joint **j** in fixed coordinate system **csys**. If this direction is not parallel to local axis *i*, it is used as the reference vector $\mathbf{V}_p$
4. Otherwise, the method fails and the analysis terminates. This will never happen if **pldirp** is not parallel to **pldirs**

A vector is considered to be parallel to local axis *i* if the sine of the angle between them is less than 10^{-3}.

Determining the Local Axes from the Reference Vectors

The program uses vector cross products to determine the local axes from the reference vectors. The three axes are represented by the three unit vectors $\mathbf{V}_1$, $\mathbf{V}_2$ and $\mathbf{V}_3$, respectively. The vectors satisfy the cross-product relationship:

$$\mathbf{V}_1 = \mathbf{V}_2 \times \mathbf{V}_3$$

The local axis $\mathbf{V_i}$ is given by the vector $\mathbf{V_a}$ after it has been normalized to unit length.

The remaining two axes, $\mathbf{V_j}$ and $\mathbf{V_k}$, are defined as follows:

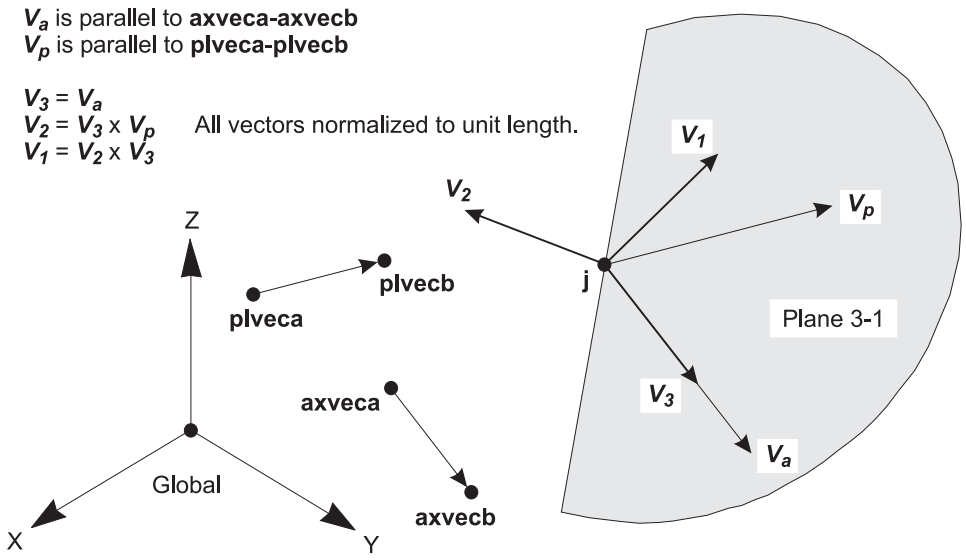


Figure 14
Example of the Determination of the Joint Local Coordinate System Using Reference Vectors for ***local****=31*

- If *i* and *j* permute in a positive sense, i.e., **local** = 12, 23, or 31, then:

$$\mathbf{V}_k = \mathbf{V}_i \times \mathbf{V}_p \quad \text{and}$$
$$\mathbf{V}_j = \mathbf{V}_k \times \mathbf{V}_i$$

- If *i* and *j* permute in a negative sense, i.e., **local** = 21, 32, or 13, then:

$$\mathbf{V}_k = \mathbf{V}_p \times \mathbf{V}_i \quad \text{and}$$
$$\mathbf{V}_j = \mathbf{V}_i \times \mathbf{V}_k$$

An example showing the determination of the joint local coordinate system using reference vectors is given in Figure 14 (page 82).

Joint Coordinate Angles

The joint local coordinate axes determined from the reference vectors may be further modified by the use of three **joint coordinate angles**, denoted **a**, **b**, and **c**. In the case where the default reference vectors are used, the joint coordinate angles define the orientation of the joint local coordinate system with respect to the global axes.

The joint coordinate angles specify rotations of the local coordinate system about its own current axes. The resulting orientation of the joint local coordinate system is obtained according to the following procedure:

1. The local system is first rotated about its +3 axis by angle **a**
2. The local system is next rotated about its resulting +2 axis by angle **b**
3. The local system is lastly rotated about its resulting +1 axis by angle **c**

The order in which the rotations are performed is important. The use of coordinate angles to orient the joint local coordinate system with respect to the global system is shown in Figure 15 (page 84).

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. In the usual case where the joint local coordinate system is parallel to the global system, the degrees of freedom may also be identified as UX, UY, UZ, RX, RY and RZ, according to which global axes are parallel to which local axes. The joint local degrees of freedom are illustrated in Figure 16 (page 85).

In addition to the regular joints that you explicitly define as part of your structural model, the program automatically creates master joints that govern the behavior of any Constraints and Welds that you may have defined. Each master joint has the same six degrees of freedom as do the regular joints. See Chapter “Constraints and Welds” (page 113) 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

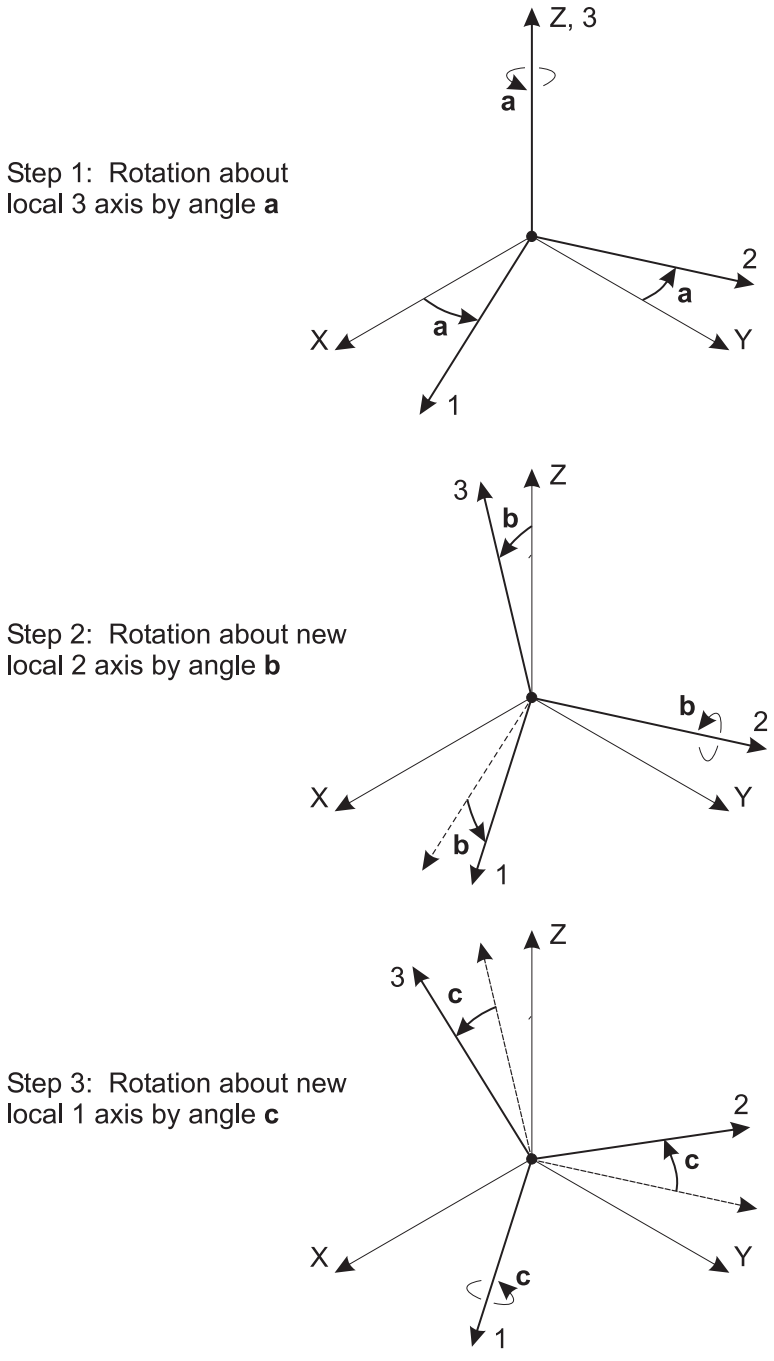


Figure 15
Use of Joint Coordinate Angles to Orient the Joint Local Coordinate System

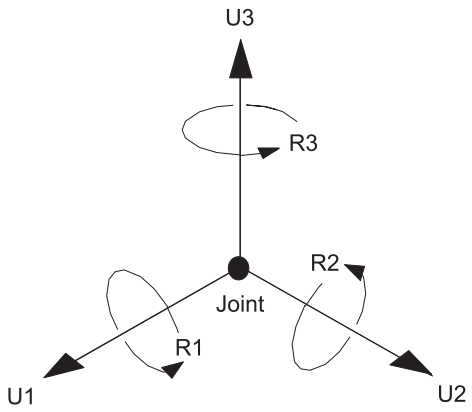


Figure 16
The Six Displacement Degrees of Freedom in the Joint Local Coordinate System

- 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 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-Y plane: a planar truss needs only UX and UY; a planar frame needs only UX, UY, and RZ; and a planar grid or flat plate needs only UZ, RX, and RY.

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.

The available degrees of freedom are always referred to the global coordinate system, and they are the same for every joint in the model. If any joint local coordinate systems are used, they must not couple available degrees of freedom with the unavailable degrees of freedom at any joint. For example, if the available degrees of freedom are UX, UY, and RZ, then all joint local coordinate systems must have one local axis parallel to the global Z axis.

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 88) in this chapter for more information.

Constrained Degrees of Freedom

Any joint that is part of a Constraint or Weld 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, and a master joint to govern the behavior of each set of joints that are connected together by a Weld. 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.

A degree of freedom *may not* be both constrained and restrained.

See Chapter "Constraints and Welds" (page 113) 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 element or to a translational spring will have all of its translational degrees of freedom activated. A joint that is connected to a Frame, Shell, or Nllink element, or to any rotational spring will have all of its rotational 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 fail.

For more information:

- See Topic “Springs” (page 89) in this chapter.
- See Topic “Degrees of Freedom” (page 147) in Chapter “The Frame Element.”
- See Topic “Degrees of Freedom” (page 185) in Chapter “The Shell Element.”
- See Topic “Degrees of Freedom” (page 208) in Chapter “The Plane Element.”
- See Topic “Degrees of Freedom” (page 224) in Chapter “The Asolid Element.”
- See Topic “Degrees of Freedom” (page 236) in Chapter “The Solid Element.”
- See Topic “Degrees of Freedom” (page 245) in Chapter “The Nllink 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.

Joints that have no elements connected to them typically have all six degrees of freedom null. Joints that have only solid-type elements (Plane, Asolid, and Solid) connected to them typically have the three rotational degrees of freedom null.

Restraints and Reactions

If the displacement of a joint along any of its available 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 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 fail.

Restraints are always applied to the joint local degrees of freedom U1, U2, U3, R1, R2, and R3. However, it is permissible to specify restraints in terms of the global degrees of freedom UX, UY, UZ, RX, RY, and RZ. If a local degree of freedom is found at the joint that is parallel to the specified global degree of freedom, the restraint is applied to that local degree of freedom. If no parallel local degree of freedom is found, *then the specified restraint is not applied*.

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 and springs connected to the restrained degree of freedom, as well as all loads applied to the degree of freedom.

A restrained degree of freedom may not be constrained. If a restraint is applied to an unavailable degree of freedom, it is ignored and no reaction is computed.

Examples of Restraints are shown in Figure 17 (page 90).

For more information:

- See Topic “Degrees of Freedom” (page 83) in this chapter.
- See Topic “Restraint Displacement Load” (page 93) 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.

Springs may be specified that couple the degrees of freedom at a joint. 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.

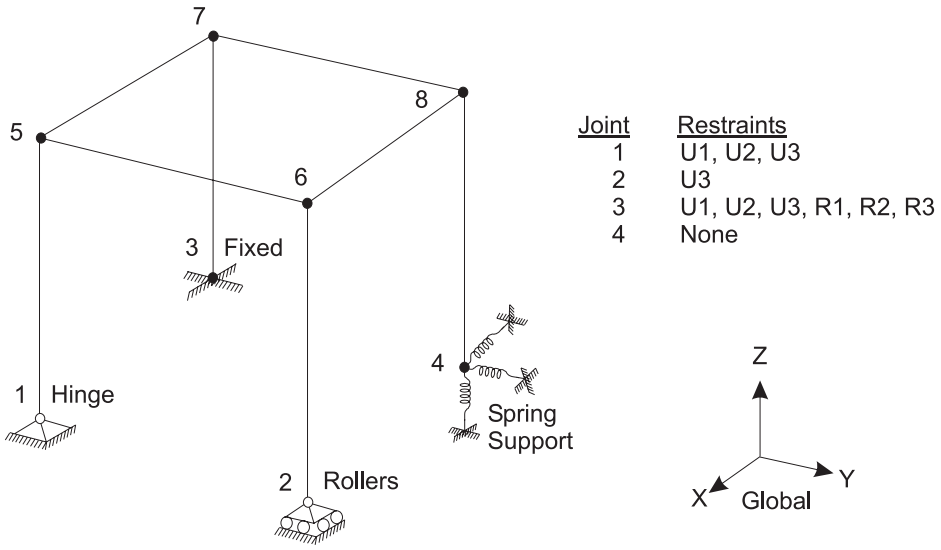
Spring stiffness coefficients may be specified in the global coordinate system, an Alternate Coordinate System, or the joint local coordinate system.

In a 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:

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ M_1 \\ M_2 \\ M_3 \end{Bmatrix} = - \begin{bmatrix} \mathbf{u1} & \mathbf{u1u2} & \mathbf{u1u3} & \mathbf{u1r1} & \mathbf{u1r2} & \mathbf{u1r3} \\ & \mathbf{u2} & \mathbf{u2u3} & \mathbf{u2r1} & \mathbf{u2r2} & \mathbf{u2r3} \\ & & \mathbf{u3} & \mathbf{u3r1} & \mathbf{u3r2} & \mathbf{u3r3} \\ & & & \mathbf{r1} & \mathbf{r1r2} & \mathbf{r1r3} \\ & \mathbf{sym.} & & & \mathbf{r2} & \mathbf{r2r3} \\ & & & & & \mathbf{r3} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ r_1 \\ r_2 \\ r_3 \end{Bmatrix} \qquad \text{(Eqn. 1)}$$

where u_1, u_2, u_3, r_1, r_2 and r_3 are the joint displacements and rotations, and the terms **u1**, **u1u2**, **u2**, ... are the specified spring stiffness coefficients.

In any fixed coordinate system, the spring forces and moments F_x, F_y, F_z, M_x, M_y and M_z at a joint are given by:

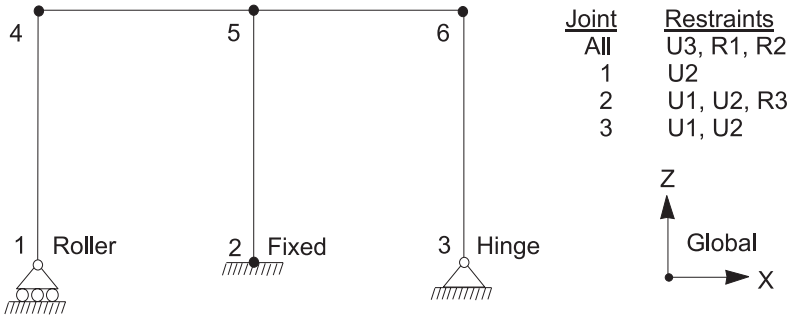


Joint	Restraints
1	U1, U2, U3
2	U3
3	U1, U2, U3, R1, R2, R3
4	None

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



Joint	Restraints
All	U3, R1, R2
1	U2
2	U1, U2, R3
3	U1, U2

2-D Frame Structure, X-Z plane

Figure 17
Examples of Restraints

$$\begin{Bmatrix} F_x \\ F_y \\ F_z \\ M_x \\ M_y \\ M_z \end{Bmatrix} = - \begin{bmatrix} \mathbf{ux} & \mathbf{uxuy} & \mathbf{uxuz} & \mathbf{uxrx} & \mathbf{uxry} & \mathbf{uxrz} \\ & \mathbf{uy} & \mathbf{uyuz} & \mathbf{uyrx} & \mathbf{uyry} & \mathbf{uyrz} \\ & & \mathbf{uz} & \mathbf{uzrx} & \mathbf{uzry} & \mathbf{uzrz} \\ & & & \mathbf{rx} & \mathbf{rxry} & \mathbf{rxrz} \\ & \mathbf{sym.} & & & \mathbf{ry} & \mathbf{ryrz} \\ & & & & & \mathbf{rz} \end{bmatrix} \begin{Bmatrix} u_x \\ u_y \\ u_z \\ r_x \\ r_y \\ r_z \end{Bmatrix}$$

where u_x, u_y, u_z, r_x, r_y and r_z are the joint displacements and rotations, and the terms **ux**, **uxuy**, **uy**, ... are the specified spring stiffness coefficients.

For springs that do not couple the degrees of freedom in a particular coordinate system, only the six diagonal terms need to be specified since the off-diagonal terms are all zero. When coupling is present, all 21 coefficients in the upper triangle of the matrix must be given; the other 15 terms are then known by symmetry.

If the springs at a joint are specified in more than one coordinate system, standard coordinate transformation techniques are used to convert the 6x6 spring stiffness matrices to the joint local coordinate system, and the resulting stiffness matrices are then added together on a term-by-term basis. The final spring stiffness matrix at each joint in the structure should have a determinant that is zero or positive. Otherwise the springs may cause the structure to be unstable.

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 **spring displacement** may vary from one Load Case to the next.

For more information:

- See Topic “Degrees of Freedom” (page 83) in this chapter.
- See Topic “Spring Displacement Load” (page 95) 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:

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

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.

Uncoupled joint masses may instead be specified in the global coordinate system, in which case they are transformed to the joint local coordinate system. Coupling terms will be generated during this transformation in the following situation:

- The joint local coordinate system directions are not parallel to global coordinate directions, and
- The three translational masses or the three rotational mass moments of inertia are not equal at a joint.

These coupling terms will be discarded by the program, resulting in some loss of accuracy. For this reason, it is recommended that you choose joint local coordinate systems that are aligned with the principal directions of translational or rotational mass at a joint, and then specify mass values in these joint local coordinates.

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 94) for mass moment of inertia formulations for various planar configurations.

For more information:

- See Topic “Degrees of Freedom” (page 83) in this chapter.
- See Chapter “Static and Dynamic Analysis” (page 291).

Force Load

The Force Load is used to apply concentrated forces and moments at the joints. Values may be specified in a fixed coordinate system (global or alternate coordinates) or the joint local coordinate system. All forces and moments at a joint are transformed to the joint local coordinate system and added together. The specified values are shown in Figure 19 (page 95).

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 83) in this chapter.
- See Chapter “Load Cases” (page 279).

Restraint Displacement Load

The Restraint Displacement Load is used to apply specified displacements (translations and rotations) at the *grounded end* of joint restraints. Restraint displacements may be specified in a fixed coordinate system (global or alternate coordinates) or the joint local coordinate system. The specified values are shown in Figure 19 (page 95). All displacements at a joint are transformed to the joint local coordinate system and added together.

Restraints may be considered as rigid connections between the joint degrees of freedom and the ground, and are always defined in the joint local coordinate system. If a particular joint degree of freedom is restrained, the displacement of the joint is equal to the restraint displacement along that local degree of freedom. Com-

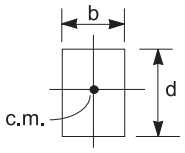
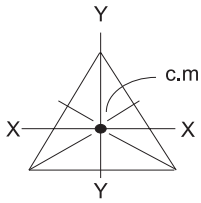
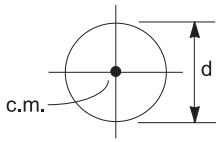
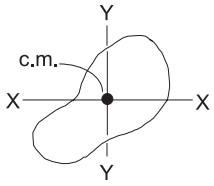
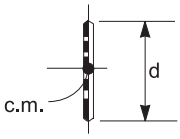
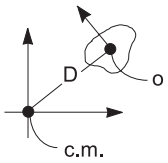
Shape in plan	Mass Moment of Inertia about vertical axis (normal to paper) through center of mass	Formula
	Rectangular diaphragm: Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g)	$MMI_{cm} = \frac{M(b^2+d^2)}{12}$
	Triangular diaphragm: Uniformly distributed mass per unit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

Figure 18
Formulae for Mass Moments of Inertia

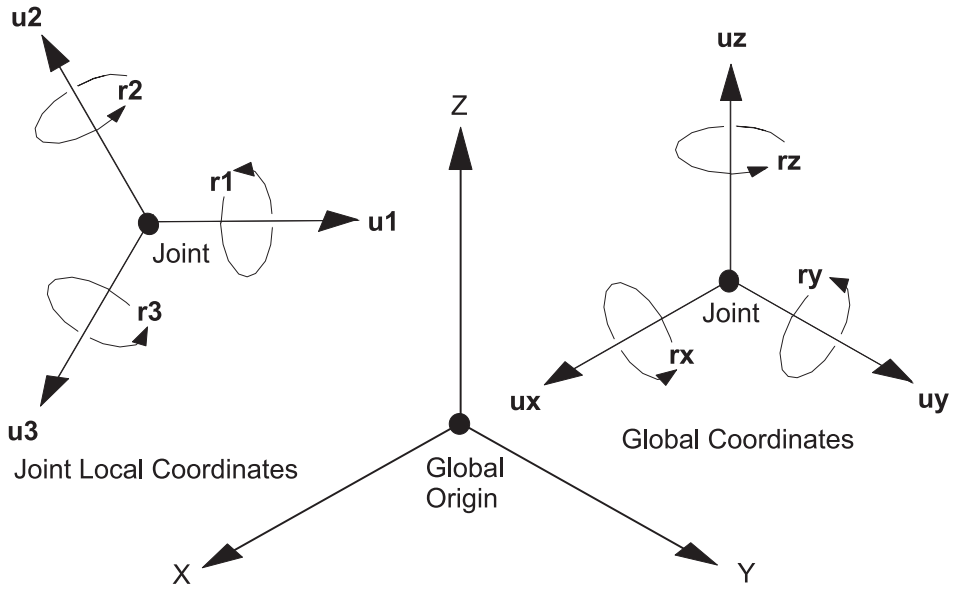


Figure 19
Specified Values for Force Load, Restraint Displacement Load, and Spring Displacement Load

ponents of restraint displacement that are not along restrained degrees of freedom do not load the structure. An example of this is illustrated in Figure 20 (page 96).

For more information:

- See Topic “Restraints and Reactions” (page 88) in this chapter.
- See Chapter “Load Cases” (page 279).

Spring Displacement Load

The Spring Displacement Load is used to apply specified displacements (translations and rotations) at the *grounded end* of joint spring supports. Spring displacements may be specified in a fixed coordinate system (global or alternate coordinates) or the joint local coordinate system. The specified values are shown in Figure 19 (page 95).

All displacements at a joint are transformed to the joint local coordinate system and added together. The resulting displacements are multiplied by the spring stiffness

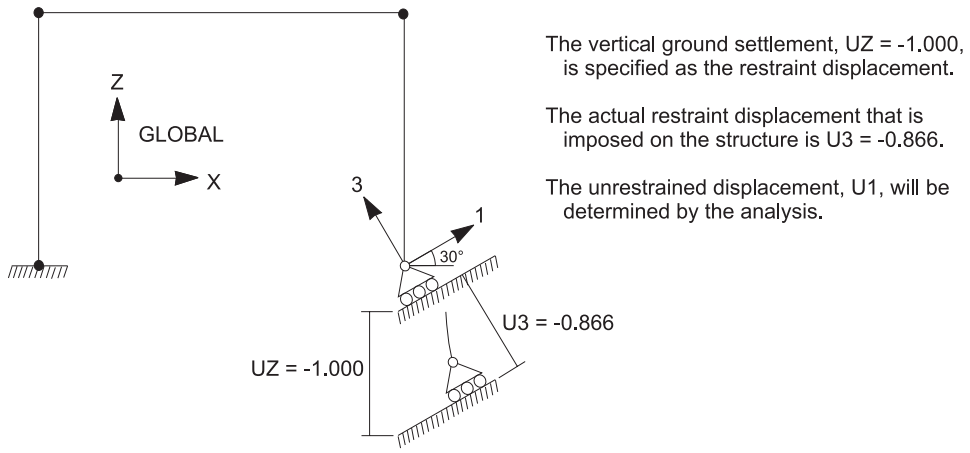


Figure 20
Example of Restraint Displacement Not Aligned with Local Degrees of Freedom

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 load.

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

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ M_1 \\ M_2 \\ M_3 \end{Bmatrix} = \begin{bmatrix} \mathbf{u1} & \mathbf{u1u2} & \mathbf{u1u3} & \mathbf{u1r1} & \mathbf{u1r2} & \mathbf{u1r3} \\ & \mathbf{u2} & \mathbf{u2u3} & \mathbf{u2r1} & \mathbf{u2r2} & \mathbf{u2r3} \\ & & \mathbf{u3} & \mathbf{u3r1} & \mathbf{u3r2} & \mathbf{u3r3} \\ & & & \mathbf{r1} & \mathbf{r1r2} & \mathbf{r1r3} \\ & \mathbf{sym.} & & & \mathbf{r2} & \mathbf{r2r3} \\ & & & & & \mathbf{r3} \end{bmatrix} \begin{Bmatrix} u_{s1} \\ u_{s2} \\ u_{s3} \\ r_{s1} \\ r_{s2} \\ r_{s3} \end{Bmatrix} \qquad \text{(Eqn. 2)}$$

where u_{s1}, u_{s2}, u_{s3}, r_{s1}, r_{s2} and r_{s3} are the spring displacements and rotations, and the terms **u1**, **u1u2**, **u2**, ... 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, a restraint displacement and a spring displacement may both be specified. The restraint displacement will control the displacement at the joint, and the net spring forces and moments acting on the joint are still

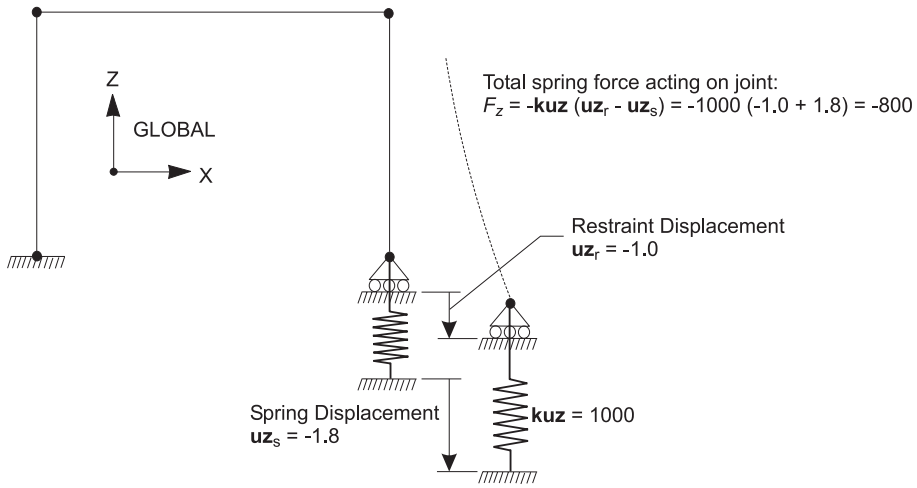


Figure 21
Interaction Between Restraint Displacements and Spring Displacements

given by the sum of the forces and moments of Equations (1) and (2). An example of this is illustrated in Figure 21 (page 97).

For more information:

- See Topic “Springs” (page 89) in this chapter.
- See Chapter “Load Cases” (page 279).

Degree of Freedom Output

A table of the types of degrees of freedom present at every joint in the model is printed in the output file under the heading:

```
DISPLACEMENT DEGREES OF FREEDOM
```

The degrees of freedom are listed for all of the regular joints, as well as for the master joints created automatically by the program. For Constraints, the master joints are identified by the labels of their corresponding Constraints. For Welds, the master joint for each set of joints that are welded together is identified by the label of one of the welded joints. Joints are printed in alpha-numeric order of the labels.

The type of each of the six degrees of freedom at a joint is identified by the following symbols:

(A)	Active degree of freedom
(-)	Restrained degree of freedom
(+)	Constrained degree of freedom
( )	Null or unavailable degree of freedom

The degrees of freedom are always referred to the local axes of the joint. They are identified in the output as U1, U2, U3, R1, R2, and R3 for all joints. However, if all regular joints use the global coordinate system as the local system (the usual situation), then the degrees of freedom for the regular joints are identified as UX, UY, UZ, RX, RY, and RZ.

The types of degrees of freedom are a property of the structure and are independent of the Analysis Cases.

Typical degree-of-freedom output is shown in Figure 22 (page 99).

For more information:

- See Topic “Degrees of Freedom” (page 83) in this chapter.
- See Chapter “The Output Files” (page 395).

Joint Mass Output

A table of the assembled masses at every joint in the model is printed in the output file under the heading:

```
ASSEMBLED JOINT MASSES
```

The mass at a given joint includes the mass assigned directly to that joint as well as a portion of the mass from each element connected to that joint. Mass at restrained degrees of freedom is set to zero. All mass assigned to the elements is apportioned to the connected joints, so that this table represents the total *unrestrained* mass of the structure.

Joints are printed in alpha-numeric order of the labels. Joints that have no unrestrained mass are not printed.

The masses are always referred to the local axes of the joint. They are identified in the output as U1, U2, U3, R1, R2, and R3. However, if all joints use the global coor-

```
D I S P L A C E M E N T   D E G R E E S   O F   F R E E D O M

       (A) = Active DOF, equilibrium equation
       (-) = Restrained DOF, reaction computed
       (+) = Constrained DOF
       ( ) = Null DOF

 JOINTS              UX  UY  UZ  RX  RY  RZ
      1 TO       2    -   +   +
      3               +   +   +
      4               +   +   +   +   -   -
      5               -   +   +   +   -   +
      6 TO       7    -   +   +
      8               +   +   +
      9               +   +   +   +   -   -
     10               A   +   +   +   A   +
     11 TO      12    -   +   +
     13               +   +   +
     14               +   +   +   +   -   -
     15               -   +   +   +   -   +
     16 TO      17    -   +   +
     18               +   +   +
     19               +   +   +   +   -   -
     20               -   +   +   +   -   +
     21 TO      22    -   +   +
     23               +   +   +
     24               +   +   +   +   -   -
     25               A   +   +   +   A   +
     26 TO      27    -   +   +
     28               +   +   +
     29               +   +   +   +   -   -
     30               -   +   +   +   -   +
     31 TO      34    +   +   +   +   +   +
     35 TO      36    -   -   -   -   -   -

 CONSTRAINTS         U1  U2  U3  R1  R2  R3
  PLATE                      A   A   A
  SYM01 TO   SYM02       A   A
  SYM03              A   A   A
  SYM04              A   A   A   A
  SYM05                      A
  SYM06 TO   SYM07       A   A
  SYM08              A   A   A
  SYM09              A   A   A   A
  SYM10                      A
  SYM11 TO   SYM12       A   A
  SYM13              A   A   A
  SYM14              A   A   A   A
  SYM15                      A
  SYM31 TO   SYM33   A   A   A   A   A   A
```

Figure 22
Degree of Freedom Table in File EXAMPLE.OUT

dinate system as the local system (the usual situation), then the masses are identified as UX, UY, UZ, RX, RY, and RZ.

The total unrestrained masses of the structure, M_x, M_y, and M_z, acting in the global X, Y, and Z directions, are printed in the output file under the heading:

```
TOTAL UNRESTRAINED MASS AND LOCATION
```

These three masses may differ even if the three translational masses assigned to each joint are equal, since the restraints for the three translational degrees of freedom at a joint need not be the same.

The locations of the centers of mass for M_x, M_y, and M_z are given with respect to the global origin. These can be used together with the mass values to determine the moments caused by the Acceleration Loads.

For more information:

- See Topic “Masses” (page 91) in this chapter.
- See Chapter “Static and Dynamic Analysis” (page 291).
- See Chapter “The Output Files” (page 395).

Displacement and Acceleration Output

Joint displacement output is available for the following types of Analysis Cases: Loads, Modes, Specs, Histories, Moving Loads, and Combos. History results are the maximum and minimum values attained during the analysis, and are only available if specifically requested in the definition of the individual History cases.

Joint translations and rotations are printed in the output file under the heading:

```
JOINT DISPLACEMENTS
```

The displacements at all joints are printed together for each Analysis Case. Joints are presented in alpha-numeric order. The displacements are given only for the available degrees of freedom. No results are printed for any joint that has only null or unavailable degrees of freedom.

The displacements are always referred to the local axes of the joint. They are identified in the output as U1, U2, U3, R1, R2, and R3. However, if all joints use the global coordinate system as the local system (the usual situation), then the displacements are identified as UX, UY, UZ, RX, RY, and RZ.

Typical joint displacement output is shown in Figure 23 (page 101).

```
J O I N T   D I S P L A C E M E N T S

TRANSLATIONS AND ROTATIONS, IN GLOBAL COORDINATES

LOAD       1 ------------------

  JOINT          UX          UY          UZ          RX          RY          RZ
      1     .000000   -0.008161    0.076681     .000000     .000000     .000000
      2     .000000   -0.007954    0.060396     .000000     .000000     .000000
      3    0.000108   -0.007493    0.044780     .000000     .000000     .000000
      4    0.000103   -0.006662    0.030495    -0.001115    .000000     .000000
      5     .000000   -0.005493    0.018238    -0.000915    .000000    4.68E-35
      6     .000000    1.45E-17    0.076676     .000000     .000000     .000000
      7     .000000    2.50E-17    0.060375     .000000     .000000     .000000
      8    3.09E-18    2.77E-17    0.044741     .000000     .000000     .000000
      9    2.26E-18    1.92E-17    0.030437    -0.001115    .000000     .000000
     10   -6.57E-19    1.62E-17    0.018137    -0.000915   -2.79E-05    4.68E-35
     11     .000000    0.008161    0.076681     .000000     .000000     .000000
     12     .000000    0.007954    0.060396     .000000     .000000     .000000
     13   -0.000108    0.007493    0.044780     .000000     .000000     .000000
     14   -0.000103    0.006662    0.030495    -0.001115    .000000     .000000

COMB    BOTH ------------------ MAX

  JOINT          UX          UY          UZ          RX          RY          RZ
      1     .000000   -0.007855    0.079786     .000000     .000000     .000000
      2     .000000   -0.007650    0.062892     .000000     .000000     .000000
      3    0.000111   -0.007198    0.046676     .000000     .000000     .000000
      4    0.000106   -0.006389    0.031820    -0.001070    .000000     .000000
      5     .000000   -0.005258    0.019050    -0.000876    .000000    2.65E-11
      6     .000000    3.16E-05    0.079780     .000000     .000000     .000000
      7     .000000    3.08E-05    0.062871     .000000     .000000     .000000
      8    5.36E-07    2.87E-05    0.046637     .000000     .000000     .000000
      9    4.92E-07    2.46E-05    0.031760    -0.001070    .000000     .000000
     10    7.20E-07    1.97E-05    0.018946    -0.000876   -2.69E-05    2.65E-11
     11     .000000    0.008467    0.079786     .000000     .000000     .000000
     12     .000000    0.008257    0.062892     .000000     .000000     .000000
     13   -0.000105    0.007789    0.046676     .000000     .000000     .000000
     14   -0.000100    0.006936    0.031820    -0.001070    .000000     .000000

COMB    BOTH ------------------ MIN

  JOINT          UX          UY          UZ          RX          RY          RZ
      1     .000000   -0.008467    0.073577     .000000     .000000     .000000
      2     .000000   -0.008257    0.057900     .000000     .000000     .000000
      3    0.000105   -0.007789    0.042884     .000000     .000000     .000000
      4    0.000100   -0.006936    0.029171    -0.001161    .000000     .000000
      5     .000000   -0.005728    0.017425    -0.000955    .000000   -2.65E-11
      6     .000000   -3.16E-05    0.073571     .000000     .000000     .000000
      7     .000000   -3.08E-05    0.057879     .000000     .000000     .000000
      8   -5.36E-07   -2.87E-05    0.042846     .000000     .000000     .000000
      9   -4.92E-07   -2.46E-05    0.029114    -0.001161    .000000     .000000
     10   -7.20E-07   -1.97E-05    0.017329    -0.000955   -2.89E-05   -2.65E-11
     11     .000000    0.007855    0.073577     .000000     .000000     .000000
     12     .000000    0.007650    0.057900     .000000     .000000     .000000
     13   -0.000111    0.007198    0.042884     .000000     .000000     .000000
     14   -0.000106    0.006389    0.029171    -0.001161    .000000     .000000
```

...Output Truncated for Brevity...

Figure 23
Joint Displacements in the Output File

By default, no displacements are printed. You may specify which Analysis Cases are to be printed. The joint displacements can always be displayed using the SAP2000 graphical interface even if no results have been selected for printing.

For Response-spectrum cases *only*, acceleration output will also be provided at any joint for which joint displacements are requested. Joint translational and rotational accelerations are printed in the output file under the heading:

```
JOINT ACCELERATIONS
```

The format of the joint acceleration output is identical to that for joint displacements.

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

For more information:

- See Topic “Degrees of Freedom” (page 83) in this chapter.
- See Chapter “Load Cases” (page 279).
- See Chapter “Static and Dynamic Analysis” (page 291).
- See Chapter “The Output Files” (page 395).

Force Output

Joint force output is available for the following types of Analysis Cases: Loads, Modes, Specs, Histories (restricted), Moving Loads (restricted), and Combos. History results are the maximum and minimum values attained during the analysis, and are only available if specifically requested in the definition of the History case.

The forces and moments that act on the joints come from a variety of sources:

- Applied loads
- Inertial loads
- Stiffness forces
- Spring forces
- Nllink forces
- Restraint Forces (Reactions)
- Constraint forces
- P-delta forces

For Histories, Moving Loads, and Combos that contain Histories or Moving Loads, only spring forces and reactions are available.

The forces and moments from each different source are computed and printed separately in the output file, as discussed in the subtopics below.

The forces and moments are always referred to the local axes of the joint. They are identified in the output as F1, F2, F3, M1, M2, and M3. However, if all joints use the global coordinate system as the local system (the usual situation), then the forces and moments are identified as FX, FY, FZ, MX, MY, and MZ.

The values printed are always the forces and moments that *act on* the joints. Thus a positive value of joint force or moment tends to cause a positive value of joint translation or rotation along the corresponding degree of freedom.

The forces and moments for all joints are printed together for each Analysis Case. Joints are presented in alpha-numeric order. Values are given only for the available degrees of freedom. No results are printed for any joint that has only null or unavailable degrees of freedom.

By default, no force output is produced. You may specify which Analysis Cases are to be printed. Certain joint forces and moments can be displayed using the SAP2000 graphical interface even if no results have been selected for printing.

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

For more information:

- See Topic “Degrees of Freedom” (page 83) in this chapter.
- See Chapter “Load Cases” (page 279).
- See Chapter “Static and Dynamic Analysis” (page 291).
- See Chapter “P-Delta Analysis” (page 337).
- See Chapter “The Output Files” (page 395).

Applied Loads

The applied loads include all specified loads that act directly on the joints and those that act on elements connected to the joints. These specified loads include Force, Self-Weight, Gravity, Temperature, Concentrated Span, Distributed Span, Prestress, Uniform, Surface Pressure, Pore Pressure, and Rotate Loads. The effects

of the Restraint Displacement and Spring Displacement Loads are not included here.

Joint forces and moments due to applied loads are printed in the output file under the heading:

```
APPLIED LOADS
```

Results are given only for joints that have at least one non-zero force or moment value. The applied loads are always zero for Mode and Spec Analysis Cases.

Typical applied-load output is shown in Figure 24 (page 105).

Inertial Loads

The inertial loads are the effective, or D'Alembert, forces and moments which resist acceleration. For a Vibration Mode of frequency ω, the acceleration for translational degree of freedom *i* at a joint is given by $-\omega^2 \ u_i$, where u_i is the translation. Hence the inertial force F_i for that degree of freedom is given by:

$$F_i = -\omega^2 \ m_{u_i} \ u_i$$

where m_{u_i} is the mass for degree of freedom *i* at the joint. Similarly, the inertial moment M_i for rotational degree of freedom *i* at a joint is given by:

$$M_i = -\omega^2 \ m_{r_i} \ r_i$$

where r_i is the rotation and m_{r_i} is the mass moment of inertia for degree of freedom *i* at the joint.

The displacement at a restrained degree of freedom is always zero in a Vibration Mode, hence the inertial loads are always zero for these degrees of freedom.

The inertial loads for Spec and Combo Analysis Cases are just the appropriate combinations of the inertial loads for the contributing Modes. The inertial loads for the Load Cases are always zero.

Joint forces and moments due to inertial loads are printed in the output file under the heading:

```
INERTIAL LOADS
```

```
 A P P L I E D   L O A D S

FORCES AND MOMENTS ACTING ON JOINTS, IN GLOBAL COORDINATES

LOAD       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      1     .000000     .000000  167.000000     .000000     .000000     .000000
      6     .000000     .000000  666.000000     .000000     .000000     .000000
     11     .000000     .000000  167.000000     .000000     .000000     .000000
     16     .000000     .000000  167.000000     .000000     .000000     .000000
     21     .000000     .000000  666.000000     .000000     .000000     .000000
     26     .000000     .000000  167.000000     .000000     .000000     .000000

COMB  STATIC ------------------ MAX

  JOINT          FX          FY          FZ          MX          MY          MZ
      1     .000000     .000000  167.000000     .000000     .000000     .000000
      6     .000000     .000000  666.000000     .000000     .000000     .000000
     11     .000000     .000000  167.000000     .000000     .000000     .000000
     16     .000000     .000000  167.000000     .000000     .000000     .000000
     21     .000000     .000000  666.000000     .000000     .000000     .000000
     26     .000000     .000000  167.000000     .000000     .000000     .000000

COMB  STATIC ------------------ MIN

  JOINT          FX          FY          FZ          MX          MY          MZ
      1     .000000     .000000  167.000000     .000000     .000000     .000000
      6     .000000     .000000  666.000000     .000000     .000000     .000000
     11     .000000     .000000  167.000000     .000000     .000000     .000000
     16     .000000     .000000  167.000000     .000000     .000000     .000000
     21     .000000     .000000  666.000000     .000000     .000000     .000000
     26     .000000     .000000  167.000000     .000000     .000000     .000000

COMB    BOTH ------------------ MAX

  JOINT          FX          FY          FZ          MX          MY          MZ
      1     .000000     .000000  167.000000     .000000     .000000     .000000
      6     .000000     .000000  666.000000     .000000     .000000     .000000
     11     .000000     .000000  167.000000     .000000     .000000     .000000
     16     .000000     .000000  167.000000     .000000     .000000     .000000
     21     .000000     .000000  666.000000     .000000     .000000     .000000
     26     .000000     .000000  167.000000     .000000     .000000     .000000

COMB    BOTH ------------------ MIN

  JOINT          FX          FY          FZ          MX          MY          MZ
      1     .000000     .000000  167.000000     .000000     .000000     .000000
      6     .000000     .000000  666.000000     .000000     .000000     .000000
     11     .000000     .000000  167.000000     .000000     .000000     .000000
     16     .000000     .000000  167.000000     .000000     .000000     .000000
     21     .000000     .000000  666.000000     .000000     .000000     .000000
     26     .000000     .000000  167.000000     .000000     .000000     .000000
```

Figure 24
Applied Loads in the Output File

Results are given only for joints that have non-zero, unrestrained mass or mass moments of inertia.

Stiffness Forces

The stiffness forces are the elastic forces contributed by any Frame, Shell, Plane, Asolid, and Solid elements that are connected to a joint. These forces are obtained as the product of the element stiffness matrices and the displacements at the connected joints. The stiffness forces are not printed in the output file, but are used to compute the constraint forces and reactions.

Spring Forces

The spring forces are the product of the spring stiffness coefficients and the joint displacements as given by Equation (1). If spring displacements are present, these produce additional forces as given by Equation (2).

Spring forces are printed in the output file under the heading:

```
SPRING FORCES
```

Results are given only for joints that have non-zero spring stiffness coefficients.

Nllink Forces

The Nllink forces are the elastic forces contributed by any Nllink elements that are connected to a joint. These forces are obtained as the product of the Nllink element stiffness matrices and the displacements at the connected joints. The Nllink stiffness matrices are based upon the effective-stiffness parameters. For two-joint elements, the forces and moments should be self-equilibrating. For single-joint elements, the forces and moments are supporting the structure.

Nllink forces are printed in the output file under the heading:

```
NLLINK FORCES
```

Results are given only for joints that are connected to Nllink elements.

Restraint Forces (Reactions)

The restraint forces are the forces and moments that are required to enforce the restraints. These reactions are computed as the sum of the applied loads, stiffness forces, spring forces, and Nllink forces acting on the restrained degrees of freedom.

Restraint forces are printed in the output file under the heading:

```
RESTRAINT FORCES (REACTIONS)
```

Results are given only for joints that possess restrained degrees of freedom.

Typical reaction output is shown in Figure 25 (page 108).

Constraint Forces

The constraint forces are the forces and moments that are required to enforce the constraints. These are computed as the sum of the applied loads, inertial loads, stiffness forces, spring forces, and Nllink forces acting on the constrained degrees of freedom. These forces and moments should be self-equilibrating over the entire structure if the constraints are properly defined. See the next Subtopic, "Global Force Balance Output," for more information.

Constraint forces are printed in the output file under the heading:

```
CONSTRAINT FORCES
```

Results are given only for joints that possess constrained degrees of freedom.

P-Delta Moments

The P-delta moments are present only when a P-Delta analysis has been performed. For each Frame element, the P-Delta moments are computed as the vector cross product of the acting P-Delta force in the element and the difference between the displacements at the two ends of the element. These individual P-Delta moments are not assigned to specific joints or printed in the output file, but are summed and printed as part of the global force balance.

Global Force Balance Output

For each Analysis Case, the sum of all joint forces and moments acting on the structure should be in equilibrium. The program computes and prints a global force balance for the following types of Analysis Cases: Loads, Modes, Specs, and Combos; no global force balance is computed for Histories, Moving Loads, or Combos that contain Histories or Moving Loads. You should always review the global force balance as a check on the validity of the structural model and the analysis.

```
R E S T R A I N T   F O R C E S   ( R E A C T I O N S )

FORCES AND MOMENTS ACTING ON JOINTS, IN GLOBAL COORDINATES

LOAD       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
     35    7.35E-13   -9.73E-11   -1000.000   84000.000    2026.122    7.47E-12
     36   -7.35E-13   -9.73E-11   -1000.000   84000.000   -2026.122   -7.47E-12

SPEC    SPEC ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
     35    0.972230  126.377896   68.323489    3978.697   89.135262    9.788509
     36    0.972230  126.377896   68.323489    3978.697   89.135262    9.788509

COMB  STATIC ------------------ MAX

  JOINT          FX          FY          FZ          MX          MY          MZ
     35    7.35E-13   -9.73E-11   -1000.000   84000.000    2026.122    7.47E-12
     36   -7.35E-13   -9.73E-11   -1000.000   84000.000   -2026.122   -7.47E-12

COMB  STATIC ------------------ MIN

  JOINT          FX          FY          FZ          MX          MY          MZ
     35    7.35E-13   -9.73E-11   -1000.000   84000.000    2026.122    7.47E-12
     36   -7.35E-13   -9.73E-11   -1000.000   84000.000   -2026.122   -7.47E-12

COMB DYNAMIC ------------------ MAX

  JOINT          FX          FY          FZ          MX          MY          MZ
     35    0.972230  126.377896   68.323489    3978.697   89.135262    9.788509
     36    0.972230  126.377896   68.323489    3978.697   89.135262    9.788509

COMB DYNAMIC ------------------ MIN

  JOINT          FX          FY          FZ          MX          MY          MZ
     35   -0.972230 -126.377896  -68.323489   -3978.697  -89.135262   -9.788509
     36   -0.972230 -126.377896  -68.323489   -3978.697  -89.135262   -9.788509

COMB    BOTH ------------------ MAX

  JOINT          FX          FY          FZ          MX          MY          MZ
     35    0.972230  126.377896 -931.676511   87978.697    2115.257    9.788509
     36    0.972230  126.377896 -931.676511   87978.697   -1936.987    9.788509

COMB    BOTH ------------------ MIN

  JOINT          FX          FY          FZ          MX          MY          MZ
     35   -0.972230 -126.377896   -1068.323   80021.303    1936.987   -9.788509
     36   -0.972230 -126.377896   -1068.323   80021.303   -2115.257   -9.788509
```

...Output has been Edited for Brevity

Figure 25
Restraint Forces (Reactions) in the Output File

For each Analysis Case, the program computes the resultants at the global origin for all joint forces and moments acting on the structure. Separate resultants are computed for each type of joint force:

- Applied loads
- Inertial loads
- Spring forces
- Nllink forces
- Restraint Forces (Reactions)
- Constraint forces
- P-Delta forces

Each force resultant is computed as the sum of the forces acting on all joints in the structure. Each moment resultant is computed as the sum of the moments acting on all joints in the structure, plus the moments about the origin of the forces acting on all joints in the structure. This results in three force and three moment components, all referred to the global coordinate system.

The resultants are also computed for the total of all these different forces and moments acting on the structure. These total resultants should be zero if the structure is in exact equilibrium. Due to the approximate nature of computer arithmetic, the totals may not be exactly zero. However, their values relative to the magnitude of the contributing forces gives a measure of the accuracy and stability of the solution.

The resultant forces and moments are always printed in the output file for all Analysis Cases under the heading:

```
GLOBAL FORCE BALANCE
```

Typical output is shown in Figure 26 (page 110).

In addition, relative equilibrium errors are printed in the log file under the heading:

```
GLOBAL FORCE BALANCE RELATIVE ERRORS
```

Here the total force and moment components are expressed as a percentage of the maximum possible equilibrium error. The maximum possible error is computed as follows:

- For each component (FX, FY, FZ, MX, MY, and MZ), the absolute values of the resultants for applied loads, inertial loads, spring forces, Nllink forces, reactions, constraint forces, and P-Delta forces are summed.
- The maximum of the absolute sums for FX, FY, and FZ is determined.

```
G L O B A L   F O R C E   B A L A N C E

TOTAL FORCE AND MOMENT AT THE ORIGIN, IN GLOBAL COORDINATES

LOAD       1 ------------------

                FX          FY          FZ          MX          MY          MZ
APPLIED     .000000     .000000    2000.000     .000000  -12000.000     .000000
INERTIA     .000000     .000000     .000000     .000000     .000000     .000000
REACTNS     .000000   -1.95E-10   -2000.000    1.13E-08   12000.000   -1.17E-09
CONSTRS    3.04E-11   -2.46E-11   -9.85E-10    1.86E-08    7.33E-09   -1.21E-09

  TOTAL    3.04E-11   -2.19E-10    2.32E-10    3.00E-08    3.22E-11   -2.38E-09

MODE       1 ------------------

                FX          FY          FZ          MX          MY          MZ
APPLIED     .000000     .000000     .000000     .000000     .000000     .000000
INERTIA    -4.36E-07   -1.18E-06  -49672.588 -1.1135E+06  298035.531    1.25E-05
REACTNS     .000000    0.001497   49672.588  1.1135E+06 -298035.530    0.008985
CONSTRS    0.000439   -0.001496    7.42E-05    0.014819    0.002190   -0.030060

  TOTAL    0.000439   -3.92E-09    4.53E-09   -4.38E-07    0.002633   -0.021063

SPEC    SPEC ------------------

                FX          FY          FZ          MX          MY          MZ
APPLIED     .000000     .000000     .000000     .000000     .000000     .000000
INERTIA    0.000784  252.768070  136.646868    4505.715  819.881682    1516.622
REACTNS     .000000  252.755793  136.646979    4505.613  819.881873    1516.535
CONSTRS    0.006039    0.016173    0.004416    0.444113    0.030539    0.373549

  TOTAL    0.005267    1.03E-11    1.17E-11    1.14E-09    0.031604    0.252835

COMB    BOTH ------------------ MAX

                FX          FY          FZ          MX          MY          MZ
APPLIED     .000000     .000000    2000.000     .000000  -12000.000     .000000
INERTIA    0.000784  252.768070  136.646868    4505.715  819.881682    1516.622
REACTNS     .000000  252.755793   -1863.353    4505.613   12819.882    1516.535
CONSTRS    0.006039    0.016173    0.004416    0.444113    0.030539    0.373549

  TOTAL    0.005267   -2.09E-10    2.43E-10    3.11E-08    0.031604    0.252835

COMB    BOTH ------------------ MIN

                FX          FY          FZ          MX          MY          MZ
APPLIED     .000000     .000000    2000.000     .000000  -12000.000     .000000
INERTIA   -0.000784 -252.768070 -136.646868   -4505.715 -819.881682   -1516.622
REACTNS     .000000 -252.755793   -2136.647   -4505.613   11180.118   -1516.535
CONSTRS   -0.006039   -0.016173   -0.004416   -0.444113   -0.030539   -0.373549

  TOTAL   -0.005267   -2.29E-10    2.20E-10    2.88E-08   -0.031604   -0.252835
```

...Output has been Edited for Brevity

Figure 26
Global Force Balance in the Output File

```
G L O B A L   F O R C E   B A L A N C E   R E L A T I V E   E R R O R S

PERCENT FORCE AND MOMENT ERROR AT THE ORIGIN, IN GLOBAL COORDINATES

   LOAD          FX          FY          FZ          MX          MY          MZ
      1    7.60E-13    5.48E-12    5.79E-12    2.20E-11    2.36E-14    1.75E-12

   MODE          FX          FY          FZ          MX          MY          MZ
      1    4.42E-07    3.94E-12    4.56E-12    1.30E-11    7.79E-08    6.23E-07
      2    7.22E-05    2.08E-13    3.59E-14    1.42E-12    1.28E-05    0.000102
      3    0.000232    3.36E-13    1.62E-14    1.23E-13    0.000139    0.001113
      4    0.007859    3.36E-14    9.51E-15    9.00E-14    0.001394    0.011150

   SPEC          FX          FY          FZ          MX          MY          MZ
   SPEC    0.001042    2.03E-12    2.32E-12    1.22E-11    0.000340    0.002719

COMB MAX/MIN     FX          FY          FZ          MX          MY          MZ
 STATIC    7.60E-13    5.48E-12    5.79E-12    2.20E-11    2.36E-14    1.75E-12
 STATIC    7.60E-13    5.48E-12    5.79E-12    2.20E-11    2.36E-14    1.75E-12
DYNAMIC    0.001042    2.03E-12    2.32E-12    1.22E-11    0.000340    0.002719
DYNAMIC    0.001042    2.03E-12    2.32E-12    1.22E-11    0.000340    0.002719
   BOTH    0.000132    5.22E-12    6.09E-12    2.29E-11    2.32E-05    0.000186
   BOTH    0.000123    5.37E-12    5.15E-12    1.98E-11    2.17E-05    0.000174
```

Figure 27
Global Force Balance Relative Errors in the Log File

- The maximum of the absolute sums for MX, MY, and MZ is determined.
- The maximum possible error for the force components is the maximum of the absolute force sums, or the maximum of the absolute moment sums divided by the average moment arm for the structure, whichever is larger.
- The maximum possible error for the moment components is the maximum of the absolute moment sums, or the maximum of the absolute force sums multiplied by the average moment arm for the structure, whichever is larger.

This definition, while complicated, assures that only numerically meaningful equilibrium errors are indicated as such.

Typical output is shown in Figure 27 (page 111).

It is strongly recommended that you always review the global force balance in the output file and the relative equilibrium errors in the log file. These results can alert you to potential problems with the structural model or the analysis. In particular, you should look for the following:

- The accuracy of the computer calculations is on the order of 10^{-15}, or 10^{-13} percent. If the relative force or moment error for the Load Cases is significantly

larger than this, it may indicate that the structure is unstable or that the stiffness matrix is ill-conditioned.

- You can expect somewhat larger equilibrium errors for Vibration Modes than for Load Cases. For Eigen Modes, the errors generally reflect the convergence tolerance and can be reduced by using a smaller tolerance. For Ritz Modes, the errors indicate the fact that the Ritz modes are not the true Eigen Modes of the structure. These errors can generally be reduced by requesting more modes. Some Modes with large errors may be orthogonal to the Ritz starting vectors and have no effect upon response-spectrum and time-history analyses.
- The constraint forces should be self-equilibrating, i.e., the resultant constraint forces and moments should be essentially zero. If the values are significantly different from zero (compared to the other resultants), the constraints may be poorly defined. An example of this is a Diaphragm constraint whose joints are not all in the same plane: in-plane forces in the Diaphragm can cause moments that are not captured by the structural model. Such errors in the constraints affect the validity of the model even though the overall equilibrium of the structure may be satisfied.

See Topic “Force Output” (page 102) in this chapter for more information.

Chapter VII

Constraints and Welds

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. **Welds** are used to generate a set of constraints that connect together different parts of the model.

Basic Topics for All Users

- Overview
- Body Constraint
- Plane Definition
- Diaphragm Constraint
- Plate Constraint
- Axis Definition
- Rod Constraint
- Beam Constraint
- Equal Constraint
- Welds

Advanced Topics

- Local Constraint
- Automatic Master Joints
- Constraint Output

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.

Most constraint types must be defined with respect to some fixed coordinate system. The coordinate system may be the global coordinate system or an alternate coordinate system, or it may be automatically determined from the locations of the constrained joints. The Local Constraint does not use a fixed coordinate system, but references each joint using its own joint local coordinate system.

Welds are used to connect together different parts of the model that were defined separately. Each Weld consists of a set of joints that *may* be joined. The program searches for joints in each Weld that share the same location in space and constrains them to act as a single joint.

Body Constraint

A Body Constraint causes all of its constrained joints to move together as a three-dimensional rigid body. Effectively, all constrained joints are connected to each other by rigid links and cannot displace relative to each other.

This Constraint can be used to:

- Model rigid connections, such as where several beams and/or columns frame together
- Connect together different parts of the structural model that were defined using separate meshes
- Connect Frame elements that are acting as eccentric stiffeners to Shell elements

Welds can be used to automatically generate Body Constraints for the purpose of connecting *coincident joints*.

See Topic “Welds” (page 131) in this chapter for more information.

Joint Connectivity

Each Body Constraint connects a set of two or more joints together. The joints may have any arbitrary location in space.

Local Coordinate System

Each Body Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. These correspond to the X, Y, and Z axes of the global coordinate system, respectively. The actual orientation of the local axes is not important since the constraint equations are independent of the coordinate system.

Constraint Equations

The constraint equations relate the displacements at any two constrained joints (subscripts *i* and *j*) in a Body Constraint. These equations are expressed in terms of the translations (u_1, u_2, and u_3), the rotations (r_1, r_2, and r_3), and the coordinates (x_1, x_2, and x_3), all taken in the Constraint local coordinate system:

$$u_{1j} = u_{1i} + r_{2i}\,\Delta x_3 - r_{3i}\,\Delta x_2$$

$$u_{2j} = u_{2i} + r_{3i}\,\Delta x_1 - r_{1i}\,\Delta x_3$$

$$u_{3j} = u_{3i} + r_{1i}\,\Delta x_2 - r_{2i}\,\Delta x_1$$

$$r_{1i} = r_{1j}$$

$$r_{2i} = r_{2j}$$

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

where $\Delta x_1 = x_{1j} - x_{1i}$, $\Delta x_2 = x_{2j} - x_{2i}$, and $\Delta x_3 = x_{3j} - x_{3i}$.

Plane Definition

The constraint equations for each Diaphragm or Plate 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 as follows:

- The centroid of the constrained joints is determined
- The second moments of the locations of all of the constrained joints about the centroid are determined
- The principal values and directions of these second moments are found
- The direction of the smallest principal second moment is taken as the normal to the constraint plane; if all constrained joints lie in a unique plane, this smallest principal moment will be zero
- If no unique direction can be found, a horizontal (X-Y) plane is assumed in coordinate system **csys**; this situation 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 following:

- **csys**: A fixed coordinate system (the default is zero, indicating the global coordinate system)
- **axis**: The axis (X, Y, or Z) normal to the plane of the constraint, taken in coordinate system **csys**.

This may be useful, for example, to specify a horizontal plane for a floor with a small step in it.

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 28 (page 118) 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 lie in the plane of the constraint. Otherwise, bending moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

For more information, see:

- Topic “Force Output” (page 102) in Chapter “Joints and Degrees of Freedom.”
- Topic “Global Force Balance Output” (page 107) in Chapter “Joints and Degrees of Freedom.”

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 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.

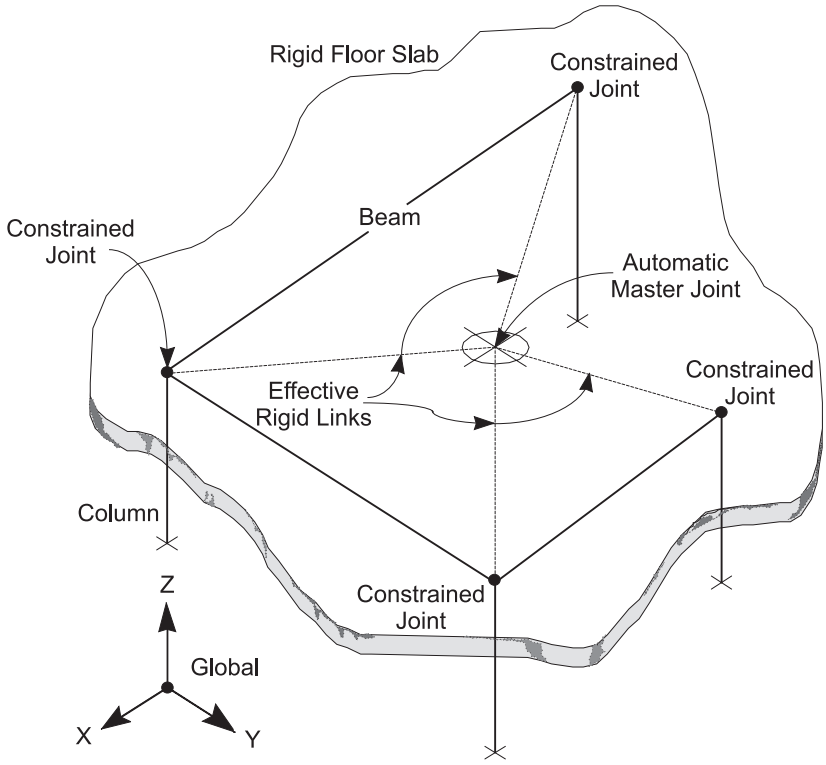


Figure 28
Use of the Diaphragm Constraint to Model a Rigid Floor Slab

For more information, see Topic "Plane Definition" (page 116) in this chapter.

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:

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

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

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

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

Plate Constraint

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

This Constraint can be used to:

- Connect structural-type elements (Frame and Shell) to solid-type elements (Plane and Solid); the rotation in the structural element can be converted to a pair of equal and opposite translations in the solid element by the Constraint
- Enforce the assumption that “plane sections remain plane” in detailed models of beam bending

Joint Connectivity

Each Plate Constraint connects a set of two or more joints together. The joints may have any arbitrary location in space. Unlike the Diaphragm Constraint, equilibrium is not affected by whether or not all joints lie in the plane of the Plate Constraint.

Local Coordinate System

Each Plate 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 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.

For more information, see Topic “Plane Definition” (page 116) in this chapter.

Constraint Equations

The constraint equations relate the displacements at any two constrained joints (subscripts *i* and *j*) in a Plate Constraint. These equations are expressed in terms of the out-of-plane translation (u_3), the bending rotations (r_1 and r_2), and the in-plane coordinates (x_1 and x_2), all taken in the Constraint local coordinate system:

$$u_{3j} = u_{3i} + r_{1i}\,\Delta x_2 - r_{2i}\,\Delta x_1$$

$$r_{1i} = r_{1j}$$

$$r_{2i} = r_{2j}$$

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

Axis Definition

The constraint equations for each Rod or Beam Constraint are written with respect to a particular axis. The location of the axis is not important, only its orientation.

By default, the axis is determined automatically by the program from the spatial distribution of the constrained joints as follows:

- The centroid of the constrained joints is determined
- The second moments of the locations of all of the constrained joints about the centroid are determined
- The principal values and directions of these second moments are found
- The direction of the largest principal second moment is taken as the axis of the constraint; if all constrained joints lie on a unique axis, the two smallest principal moments will be zero
- If no unique direction can be found, a vertical (Z) axis is assumed in coordinate system **csys**; this situation can occur if the joints are coincident, or if the spatial distribution is more nearly planar or three-dimensional than linear.

You may override automatic axis selection by specifying the following:

- **csys**: A fixed coordinate system (the default is zero, indicating the global coordinate system)
- **axis**: The axis (X, Y, or Z) of the constraint, taken in coordinate system **csys**.

This may be useful, for example, to specify a vertical axis for a column with a small offset in it.

Rod Constraint

A Rod Constraint causes all of its constrained joints to move together as a straight rod that is rigid against axial deformation. Effectively, all constrained joints main-

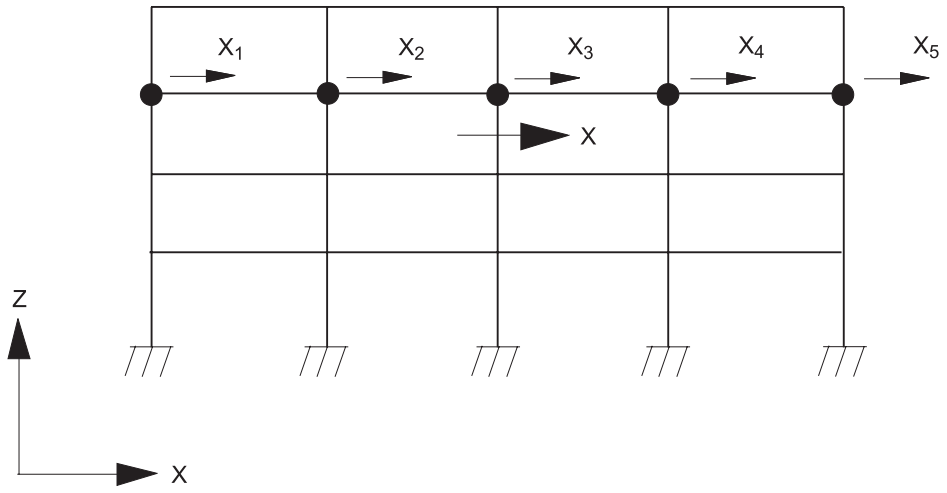


Figure 29
Use of the Rod Constraint to Model Axially Rigid Beams

tain a fixed distance from each other in the direction parallel to the axis of the rod, but translations normal to the axis and all rotations are unaffected.

This Constraint can be used to:

- Prevent axial deformation in Frame elements
- Model rigid truss-like links

An example of the use of the Rod Constraint is in the analysis of the two-dimensional frame shown in Figure 29 (page 121). If the axial deformations in the beams are negligible, a single Rod Constraint could be defined containing the five joints. Instead of five equations, the program would use a single equation to define the X-displacement of the whole floor. However, it should be noted that this will result in the axial forces of the beams being output as zero, as the Constraint will cause the ends of the beams to translate together in the X-direction. Interpretations of such results associated with the use of Constraints should be clearly understood.

Joint Connectivity

Each Rod 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 lie on the axis of the constraint. Otherwise, bending moments may be generated that are re-

strained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

For more information, see:

- Topic "Force Output" (page 102) in Chapter "Joints and Degrees of Freedom."
- Topic "Global Force Balance Output" (page 107) in Chapter "Joints and Degrees of Freedom."

Local Coordinate System

Each Rod Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. Local axis 1 is always the axis of the constraint. The program arbitrarily chooses the orientation of the transverse axes 2 and 3. The actual orientation of the transverse axes is not important since only the axial direction affects the constraint equations.

For more information, see Topic "Axis Definition" (page 120) in this chapter.

Constraint Equations

The constraint equations relate the displacements at any two constrained joints (subscripts *i* and *j*) in a Rod Constraint. These equations are expressed only in terms of the axial translation (u_1):

$$u_{1j} = u_{1i}$$

Beam Constraint

A Beam Constraint causes all of its constrained joints to move together as a straight beam that is rigid against bending deformation. Effectively, all constrained joints are connected to each other by links that are rigid for off-axis bending, but do not affect translation along or rotation about the axis.

This Constraint can be used to:

- Connect structural-type elements (Frame and Shell) to solid-type elements (Plane and Solid); the rotation in the structural element can be converted to a pair of equal and opposite translations in the solid element by the Constraint
- Prevent bending deformation in Frame elements

Joint Connectivity

Each Beam 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 lie on the axis of the constraint. Otherwise, torsional moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

For more information, see:

- Topic “Force Output” (page 102) in Chapter “Joints and Degrees of Freedom”
- Topic “Global Force Balance Output” (page 107) in Chapter “Joints and Degrees of Freedom”

Local Coordinate System

Each Beam Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. Local axis 1 is always the axis of the constraint. The program arbitrarily chooses the orientation of the transverse axes 2 and 3. The actual orientation of the transverse axes is not important since only the axial direction affects the constraint equations.

For more information, see Topic “Axis Definition” (page 120) in this chapter.

Constraint Equations

The constraint equations relate the displacements at any two constrained joints (subscripts *i* and *j*) in a Beam Constraint. These equations are expressed in terms of the transverse translations (u_2 and u_3), the transverse rotations (r_2 and r_3), and the axial coordinate (x_1), all taken in the Constraint local coordinate system:

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

$$u_{3j} = u_{3i} - r_{2i}\,\Delta x_1$$

$$r_{2i} = r_{2j}$$

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

where $\Delta x_1 = x_{1j} - x_{1i}$.

Equal Constraint

An Equal Constraint causes all of its constrained joints to move together with the same (or opposite) displacements for each selected degree of freedom, taken in the constraint local coordinate system. The other degrees of freedom are unaffected.

The Equal Constraint differs from the rigid-body types of Constraints in that there is *no coupling* between the rotations and the translations.

This Constraint can be used to:

- Model symmetry and anti-symmetry conditions with respect to a plane
- Partially connect together different parts of the structural model, such as at expansion joints and hinges

For fully connecting meshes, it is better to use the Body Constraint when the constrained joints are not in exactly the same location.

Joint Connectivity

Each Equal Constraint connects a set of two or more joints together. For a given Constraint, if *any* of the selected degrees of freedom are negative (i.e., opposite), only two constrained joints are allowed for that Constraint. Otherwise any number of constrained joints are permitted.

The joints may have any arbitrary location in space, but for best results all joints should share the same location in space if used for connecting meshes. Otherwise, moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

Such restrained moments may also be generated when Equal Constraints are used for symmetry purposes. They are necessary to enforce the desired symmetry or anti-symmetry of the displacements when the applied loads are not correspondingly symmetric or anti-symmetric.

For more information, see:

- Topic “Force Output” (page 102) in Chapter “Joints and Degrees of Freedom.”
- Topic “Global Force Balance Output” (page 107) in Chapter “Joints and Degrees of Freedom.”

Local Coordinate System

Each Equal Constraint uses a fixed coordinate system, **csys**, that you specify. The default for **csys** is zero, indicating the global coordinate system. The axes of the fixed coordinate system are denoted X, Y, and Z.

Selected Degrees of Freedom

For each Equal Constraint you may specify a list, **cdofs**, of up to six degrees of freedom in coordinate system **csys** that are to be constrained. The degrees of freedom are indicated as UX, UY, UZ, RX, RY, and RZ. A negative sign indicates a degree of freedom that is constrained to be opposite, e.g., –UX.

Constraint Equations

The constraint equations relate the displacements at any two constrained joints (subscripts *i* and *j*) in an Equal Constraint. These equations are expressed in terms of the translations (u_x, u_y, and u_z) and the rotations (r_x, r_y, and r_z), all taken in fixed coordinate system **csys**. The equations used depend upon the selected degrees of freedom and their signs. Some important cases are described next.

Symmetry About a Plane

For a structure that is symmetric about a plane, symmetric loading causes symmetric displacements as follows:

- Forces and displacements parallel to the plane of symmetry are equal
- Forces and displacements normal to the plane of symmetry are opposite
- Moments and rotations parallel to the plane of symmetry are opposite
- Moments and rotations normal to the plane of symmetry are equal

As an example, consider a structure that is symmetric with respect to a plane normal to the X axis and subjected to symmetric loading. A separate Equal Constraint must be defined for each pair of joints that is symmetrically located with respect to the plane. The degrees of freedom that would be specified for these Constraints are: –UX, UY, UZ, RX, –RY, and –RZ. The corresponding constraint equations are:

$$u_{xj} = -u_{xi}$$

$$u_{yj} = u_{yi}$$

$$u_{zj} = u_{zi}$$

$$r_{xi} = r_{xj}$$

$$r_{yi} = -r_{yj}$$

$$r_{zi} = -r_{zj}$$

Any joints on the plane of symmetry should not be constrained, but instead have their UX, RY, and RZ degrees of freedom restrained.

Anti-symmetry About a Plane

For a structure that is symmetric about a plane, anti-symmetric loading causes anti-symmetric displacements. All degrees of freedom that are equal when symmetric are opposite when anti-symmetric, and all degrees of freedom that are opposite when symmetric are equal when anti-symmetric. Thus the specification of the anti-symmetric degrees of freedom simply uses the opposite signs from the symmetric case.

Consider the example above of a structure that is symmetric with respect to a plane normal to the X axis, but now subjected to anti-symmetric loading. A separate Equal Constraint must be defined for each pair of joints that is symmetrically located with respect to the plane. The degrees of freedom that would be specified for these Constraints are: UX, –UY, –UZ, –RX, RY, and RZ. The signs of the constraint equations are corresponding changed from the symmetric case.

Partial Connection

When joints are being connected, all specified degrees of freedom are positive. For example, consider an idealized hinge connection of eight space-truss members. Only displacements are continuous across the hinge, not rotations. Each truss member is connected to a separate joint (node) at the connection. One Equal Constraint is defined for the eight constrained joints. The degrees of freedom that would be specified for this Constraint are: UX, UY, and UZ. The corresponding constraint equations are:

$$u_{xj} = u_{xi}$$

$$u_{yj} = u_{yi}$$

$$u_{zj} = u_{zi}$$

The eight joints should be coincident or the axes of the truss members should all intersect at the same point. Otherwise, moments may be generated that are unrealistically restrained by the Constraint.

Local Constraint

A Local Constraint causes all of its constrained joints to move together with the same (or opposite) displacements for each selected degree of freedom, taken in the separate joint local coordinate systems. The other degrees of freedom are unaffected.

The Local Constraint differs from the rigid-body types of Constraints in that there is *no coupling* between the rotations and the translations. The Local Constraint is the same as the Equal Constraint if all constrained joints have the same local coordinate system.

This Constraint can be used to:

- Model symmetry conditions with respect to a line or a point
- Model displacements constrained by mechanisms

The behavior of this Constraint is dependent upon the choice of the local coordinate systems of the constrained joints.

Joint Connectivity

Each Local Constraint connects a set of two or more joints together. If *any* of the selected degrees of freedom for a given Constraint are negative (i.e., opposite) only two constrained joints are allowed for that Constraint. Otherwise any number of constrained joints are permitted.

The joints may have any arbitrary location in space. If the joints do not share the same location in space, moments may be generated that are restrained by the Constraint. If this happens, the constraint forces printed in the output file may not be in equilibrium. These moments are necessary to enforce the desired symmetry of the displacements when the applied loads are not symmetric, or may represent the constraining action of a mechanism.

For more information, see:

- Topic “Force Output” (page 102) in Chapter “Joints and Degrees of Freedom.”

- Topic “Global Force Balance Output” (page 107) in Chapter “Joints and Degrees of Freedom.”

No Local Coordinate System

A Local Constraint does not have its own local coordinate system. The constraint equations are written in terms of constrained joint local coordinate systems, which may differ. The axes of these coordinate systems are denoted 1, 2, and 3.

Selected Degrees of Freedom

For each Local Constraint you may specify a list, **ldofs**, of up to six degrees of freedom in the joint local coordinate systems that are to be constrained. The degrees of freedom are indicated as U1, U2, U3, R1, R2, and R3. A negative sign indicates a degree of freedom that is constrained to be opposite, e.g., –U1.

Constraint Equations

The constraint equations relate the displacements at any two constrained joints (subscripts *i* and *j*) in a Local Constraint. These equations are expressed in terms of the translations (u_1, u_2, and u_3) and the rotations (r_1, r_2, and r_3), all taken in joint local coordinate systems. The equations used depend upon the selected degrees of freedom and their signs. Some important cases are described next.

Axisymmetry

Axisymmetry is a type of symmetry about a line. It is best described in terms of a cylindrical coordinate system having its Z axis on the line of symmetry. The structure, loading, and displacements are each said to be axisymmetric about a line if they do not vary with angular position around the line, i.e., they are independent of the angular coordinate CA.

To enforce axisymmetry using the Local Constraint:

- Model any cylindrical sector of the structure using any axisymmetric mesh of joints and elements
- Assign each joint a local coordinate system such that local axes 1, 2, and 3 correspond to the coordinate directions +CR, +CA, and +CZ, respectively
- For each axisymmetric set of joints (i.e., having the same coordinates CR and CZ, but different CA), define a Local Constraint using all six degrees of freedom: U1, U2, U3, R1, R2, and R3

- Restrain joints that lie on the line of symmetry so that, at most, only axial translations (U3) and rotations (R3) are permitted

The corresponding constraint equations are:

$u_{1j} = u_{1i}$

$u_{2j} = u_{2i}$

$u_{3j} = u_{3i}$

$r_{1i} = r_{1j}$

$r_{2i} = r_{2j}$

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

The numeric subscripts refer to the corresponding joint local coordinate systems.

Cyclic symmetry

Cyclic symmetry is another type of symmetry about a line. It is best described in terms of a cylindrical coordinate system having its Z axis on the line of symmetry. The structure, loading, and displacements are each said to be cyclically symmetric about a line if they vary with angular position in a repeated (periodic) fashion.

To enforce cyclic symmetry using the Local Constraint:

- Model any number of adjacent, representative, cylindrical sectors of the structure; denote the size of a single sector by the angle θ
- Assign each joint a local coordinate system such that local axes 1, 2, and 3 correspond to the coordinate directions +CR, +CA, and +CZ, respectively
- For each cyclically symmetric set of joints (i.e., having the same coordinates CR and CZ, but with coordinate CA differing by multiples of θ), define a Local Constraint using all six degrees of freedom: U1, U2, U3, R1, R2, and R3.
- Restrain joints that lie on the line of symmetry so that, at most, only axial translations (U3) and rotations (R3) are permitted

The corresponding constraint equations are:

$u_{1j} = u_{1i}$

$u_{2j} = u_{2i}$

$u_{3j} = u_{3i}$

$$r_{1i} = r_{1j}$$

$$r_{2i} = r_{2j}$$

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

The numeric subscripts refer to the corresponding joint local coordinate systems.

For example, suppose a structure is composed of six identical 60° sectors, identically loaded. If two adjacent sectors were modeled, each Local Constraint would apply to a set of two joints, except that three joints would be constrained on the symmetry planes at 0°, 60°, and 120°.

If a single sector is modeled, only joints on the symmetry planes need to be constrained.

Symmetry About a Point

Symmetry about a point is best described in terms of a spherical coordinate system having its Z axis on the line of symmetry. The structure, loading, and displacements are each said to be symmetric about a point if they do not vary with angular position about the point, i.e., they are independent of the angular coordinates SB and SA. Radial translation is the only displacement component that is permissible.

To enforce symmetry about a point using the Local Constraint:

- Model any spherical sector of the structure using any symmetric mesh of joints and elements
- Assign each joint a local coordinate system such that local axes 1, 2, and 3 correspond to the coordinate directions +SB, +SA, and +SR, respectively
- For each symmetric set of joints (i.e., having the same coordinate SR, but different coordinates SB and SA), define a Local Constraint using only degree of freedom U3
- For all joints, restrain the degrees of freedom U1, U2, R1, R2, and R3
- Fully restrain any joints that lie at the point of symmetry

The corresponding constraint equations are:

$$u_{3j} = u_{3i}$$

The numeric subscripts refer to the corresponding joint local coordinate systems.

It is also possible to define a case for symmetry about a point that is similar to cyclic symmetry around a line, e.g., where each octant of the structure is identical.

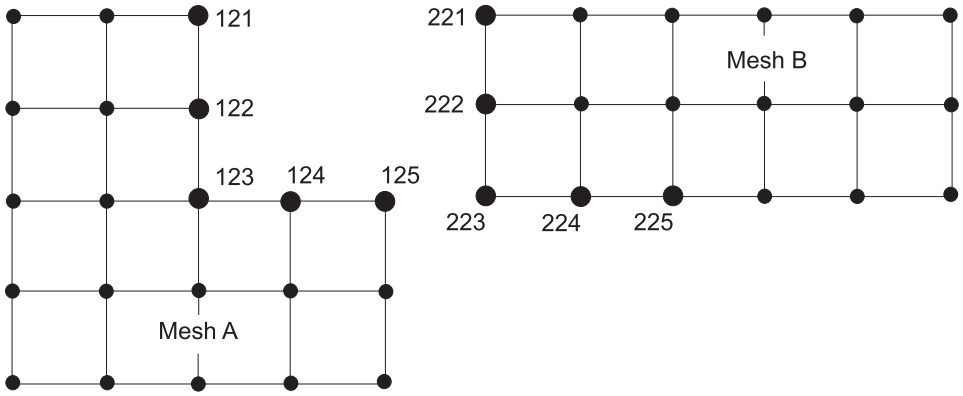


Figure 30
Use of a Weld to Connect Separate Meshes at Coincident Joints

Welds

A Weld can be used to connect together different parts of the structural model that were defined using separate meshes. A Weld is not a single Constraint, but rather is a set of joints from which the program will automatically generate multiple Body Constraints to connect together coincident joints.

Joints are considered to be coincident if the distance between them is less than or equal to a tolerance, **tol**, that you specify. Setting the tolerance to zero is permissible but is not recommended.

One or more Welds may be defined, each with its own tolerance. Only the joints within each Weld will be checked for coincidence with each other. In the most common case, a single Weld is defined that contains all joints in the model; all coincident groups of joints will be welded. However, in situations where structural discontinuity is desired, it may be necessary to prevent the welding of some coincident joints. This may be facilitated by the use of multiple Welds.

Figure 30 (page 131) shows a model developed as two separate meshes, A and B. Joints 121 through 125 are associated with mesh A, and Joints 221 through 225 are associated with mesh B. Joints 121 through 125 share the same location in space as Joints 221 through 225, respectively. These are the interfacing joints between the two meshes. To connect these two meshes, a single Weld can be defined containing all joints, or just joints 121 through 125 and 221 through 225. The program would

generate five Body Constraints, each containing two joints, resulting in an integrated model.

It is permissible to include the same joint in more than one Weld. This could result in the joints in different Welds being constrained together if they are coincident with the common joint. For example, suppose that Weld 1 contained joints 1,2, and 3, Weld 2 contained joints 3, 4, and 5. If joints 1, 3, and 5 were coincident, joints 1 and 3 would be constrained by Weld 1, and joints 3 and 5 would be constrained by Weld 2. The program would create a single Body Constraint containing joints 1, 3, and 5. One the other hand, if Weld 2 did not contain joint 3, the program would only generate a Body Constraint containing joint 1 and 3 from Weld 1; joint 5 would not be constrained.

For more information, see Topic “Body Constraint” (page 115) in this chapter.

Automatic Master Joints

The program automatically creates an internal **master joint** for each explicit Constraint, and a master joint for each internal Body Constraint that is generated by a Weld. Each master joint governs the behavior of the corresponding constrained joints. The displacement at a constrained degree of freedom is computed as a linear combination of the displacements of the master joint.

See Topic “Degrees of Freedom” (page 83) in Chapter “Joints and Degrees of Freedom” for more information.

Stiffness, Mass, and Loads

Joint local coordinate systems, springs, masses, and loads may all be applied to constrained joints. Elements may also be connected to constrained joints. The joint and element stiffnesses, masses and loads from the constrained degrees of freedom are be automatically transferred to the master joint in a consistent fashion.

The translational stiffness at the master joint is the sum of the translational stiffnesses at the constrained joints. The same is true for translational masses and loads.

The rotational stiffness at a master joint is the sum of the rotational stiffnesses at the constrained degrees of freedom, plus the second moment of the translational stiffnesses at the constrained joints for the Body, Diaphragm, Plate, and Beam Constraints. The same is true for rotational masses and loads, except that only the first moment of the translational loads is used. The moments of the translational stiff-

nesses, masses, and loads are taken about the center of mass of the constrained joints. If the joints have no mass, the centroid is used.

Local Coordinate Systems

Each master joint has *two* local coordinate systems: one for the translational degrees of freedom, and one for the rotational degrees of freedom. The axes of each local system are denoted 1, 2, and 3. For the Local Constraint, these axes correspond to the local axes of the constrained joints. For other types of Constraints, these axes are chosen to be the principal directions of the translational and rotational masses of the master joint. Using the principal directions eliminates coupling between the mass components in the master-joint local coordinate system.

For a Diaphragm or Plate Constraint, the local 3 axes of the master joint are always normal to the plane of the Constraint. For a Beam or Rod Constraint, the local 1 axes of the master joint are always parallel to the axis of the Constraint.

Constraint Output

For each Body, Diaphragm, Plate, Rod, and Beam Constraint having more than two constrained joints, the following information about the Constraint and its master joint is printed in the output file:

- The translational and rotational local coordinate systems for the master joint
- The total mass and mass moments of inertia for the Constraint that have been applied to the master joint
- The center of mass for each of the three translational masses

The degrees of freedom are indicated as U1, U2, U3, R1, R2, and R3. These are referred to the two local coordinate systems of the master joint.

Typical constraint output is shown in Figure 31 (page 134).

```
C O N S T R A I N T   C O O R D I N A T E S   A N D   M A S S E S

CONS       1 ================== TYPE = DIAPH, NORMAL DIRECTION = U3

                    LOCAL COORDINATE SYSTEM FOR CONSTRAINT MASTER
 GLOBAL          U1          U2          U3          R1          R2          R3
    X      1.000000     .000000     .000000    1.000000     .000000     .000000
    Y       .000000    1.000000     .000000     .000000    1.000000     .000000
    Z       .000000     .000000    1.000000     .000000     .000000    1.000000

                    TRANSLATIONAL MASS AND MASS MOMENTS OF INERTIA
               U1          U2          U3          R1          R2          R3
          3.880000    3.880000     .000000     .000000     .000000    1011.000

                    CENTER OF MASS
 GLOBAL          U1          U2          U3
    X     25.000000   25.000000     .000000
    Y     12.500000   12.500000     .000000
    Z     12.500000   12.500000     .000000

CONS       2 ================== TYPE = DIAPH, NORMAL DIRECTION = U3

                    LOCAL COORDINATE SYSTEM FOR CONSTRAINT MASTER
 GLOBAL          U1          U2          U3          R1          R2          R3
    X      1.000000     .000000     .000000    1.000000     .000000     .000000
    Y       .000000    1.000000     .000000     .000000    1.000000     .000000
    Z       .000000     .000000    1.000000     .000000     .000000    1.000000

                    TRANSLATIONAL MASS AND MASS MOMENTS OF INERTIA
               U1          U2          U3          R1          R2          R3
          3.880000    3.880000     .000000     .000000     .000000    1011.000

                    CENTER OF MASS
 GLOBAL          U1          U2          U3
    X     25.000000   25.000000     .000000
    Y     12.500000   12.500000     .000000
    Z     25.000000   25.000000     .000000
```

...Output Edited for Brevity...

Figure 31
Constraint Coordinates and Masses in the Output File

Chapter VIII

Material Properties

The Materials are used to define the mechanical, thermal, and density properties used by the Frame, Shell, Plane, Asolid, and Solid elements.

Basic Topics for All Users

- Overview
- Local Coordinate System
- Stresses and Strains
- Isotropic Materials
- Mass Density
- Weight Density
- Design-Type Indicator

Advanced Topics

- Orthotropic Materials
- Anisotropic Materials
- Temperature-Dependent Materials
- Element Material Temperature

Overview

The Material properties are always linear elastic. They may be defined as being isotropic, orthotropic or anisotropic. How the properties are actually utilized depends on the element type. Each Material that you define may be used by more than one element or element type. For the Frame element, the Materials are referenced indirectly through the Section properties.

All material properties, except mass and weight density, may be temperature dependent. Properties are given at a series of specified temperatures. Properties at other temperatures are obtained by linear interpolation.

For a given execution of the program, the properties used by an element are assumed to be constant regardless of any temperature changes experienced by the structure. Each element may be assigned a **material temperature** that determines the material properties used for the analysis.

Local Coordinate System

Each Material has its own **Material local coordinate system** used to define the elastic and thermal properties. This system is significant only for orthotropic and anisotropic materials. Isotropic materials are independent of any particular coordinate system.

The axes of the Material local coordinate system are denoted 1, 2, and 3. By default, the Material coordinate system is aligned with the local coordinate system for each element. However, you may specify a set of one or more **material angles** that rotate the Material coordinate system with respect to the element system for those elements that permit orthotropic or anisotropic properties.

For more information:

- See Topic “Material Angle” (page 212) in Chapter “The Plane Element.”
- See Topic “Material Angle” (page 226) in Chapter “The Asolid Element.”
- See Topic “Material Angles” (page 237) in Chapter “The Solid Element.”

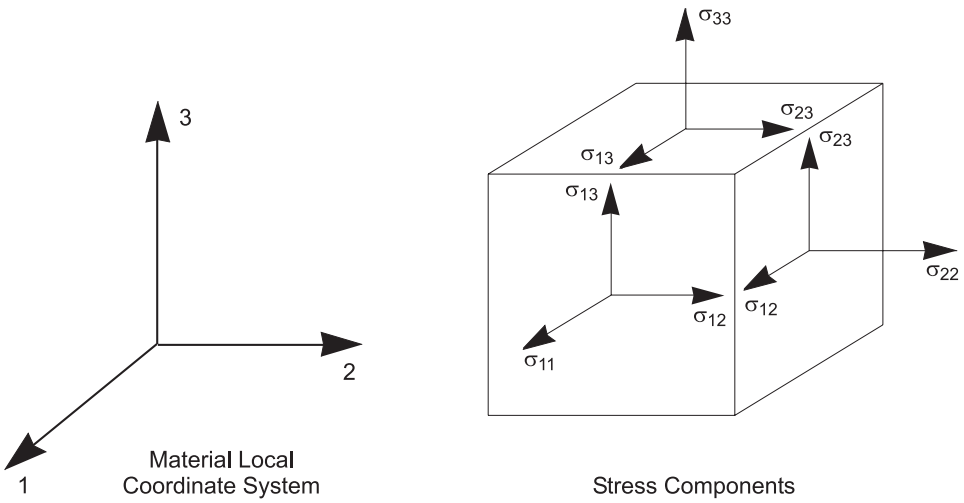


Figure 32
Definition of Stress Components in the Material Local Coordinate System

Stresses and Strains

The elastic mechanical properties relate the behavior of the stresses and strains within the Material. The stresses are defined as forces per unit area acting on an elemental cube aligned with the material axes as shown in Figure 32 (page 137). The stresses σ_{11}, σ_{22}, and σ_{33} are called the direct stresses and tend to cause length change, while σ_{12}, σ_{13}, and σ_{23} are called the shear stresses and tend to cause angle change.

Not all stress components exist in every element type. For example, the stresses σ_{22}, σ_{33}, and σ_{23} are assumed to be zero in the Frame element, and stress σ_{33} is taken to be zero in the Shell element.

The direct strains ε_{11}, ε_{22}, and ε_{33} measure the change in length along the Material local 1, 2, and 3 axes, respectively, and are defined as:

$$\varepsilon_{11} = \frac{du_1}{dx_1}$$

$$\varepsilon_{22} = \frac{du_2}{dx_2}$$

$$\varepsilon_{33} = \frac{du_3}{dx_3}$$

where u_1, u_2, and u_3 are the displacements and x_1, x_2, and x_3 are the coordinates in the Material 1, 2, and 3 directions, respectively.

The engineering shear strains γ_{12}, γ_{13}, and γ_{23}, measure the change in angle in the Material local 1-2, 1-3, and 2-3 planes, respectively, and are defined as:

$$\gamma_{12} = \frac{du_1}{dx_2} + \frac{du_2}{dx_1}$$

$$\gamma_{13} = \frac{du_1}{dx_3} + \frac{du_3}{dx_1}$$

$$\gamma_{23} = \frac{du_2}{dx_3} + \frac{du_3}{dx_2}$$

Note that the engineering shear strains are equal to twice the tensorial shear strains ε_{12}, ε_{13}, and ε_{23}, respectively.

Strains can also be caused by a temperature change, ΔT, from a zero-stress reference temperature. No stresses are caused by a temperature change unless the induced thermal strains are restrained.

See Cook, Malkus, and Plesha (1989), or any textbook on elementary mechanics.

Isotropic Materials

The behavior of an isotropic material is independent of the direction of loading or the orientation of the material. In addition, shearing behavior is uncoupled from extensional behavior and is not affected by temperature change. Isotropic behavior is usually assumed for steel and concrete, although this is not always the case.

The isotropic mechanical and thermal properties relate strain to stress and temperature change as follows:

$$\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{Bmatrix} = \begin{bmatrix} \frac{1}{\mathbf{e1}} & \frac{-\mathbf{u12}}{\mathbf{e1}} & \frac{-\mathbf{u12}}{\mathbf{e1}} & 0 & 0 & 0 \\ & \frac{1}{\mathbf{e1}} & \frac{-\mathbf{u12}}{\mathbf{e1}} & 0 & 0 & 0 \\ & & \frac{1}{\mathbf{e1}} & 0 & 0 & 0 \\ & & & \frac{1}{\mathbf{g12}} & 0 & 0 \\ & \text{sym.} & & & \frac{1}{\mathbf{g12}} & 0 \\ & & & & & \frac{1}{\mathbf{g12}} \end{bmatrix} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix} + \begin{Bmatrix} \mathbf{a1} \\ \mathbf{a1} \\ \mathbf{a1} \\ 0 \\ 0 \\ 0 \end{Bmatrix} \Delta T \qquad \text{(Eqn. 1)}$$

where **e1** is Young's modulus of elasticity, **u12** is Poisson's ratio, **g12** is the shear modulus, and **a1** is the coefficient of thermal expansion. This relationship holds regardless of the orientation of the Material local 1, 2, and 3 axes.

The shear modulus is not directly specified, but instead is defined in terms of Young's modulus and Poisson's ratio as:

$$\mathbf{g12} = \frac{\mathbf{e1}}{2(1+\mathbf{u12})}$$

Note that Young's modulus must be positive, and Poisson's ratio must satisfy the condition:

$$-1 < \mathbf{u12} < \frac{1}{2}$$

Orthotropic Materials

The behavior of an orthotropic material can be different in each of the three local coordinate directions. However, like an isotropic material, shearing behavior is uncoupled from extensional behavior and is not affected by temperature change.

The orthotropic mechanical and thermal properties relate strain to stress and temperature change as follows:

$$\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{Bmatrix} = \begin{bmatrix} \frac{1}{\mathbf{e1}} & \frac{-\mathbf{u12}}{\mathbf{e2}} & \frac{-\mathbf{u13}}{\mathbf{e3}} & 0 & 0 & 0 \\ & \frac{1}{\mathbf{e2}} & \frac{-\mathbf{u23}}{\mathbf{e3}} & 0 & 0 & 0 \\ & & \frac{1}{\mathbf{e3}} & 0 & 0 & 0 \\ & & & \frac{1}{\mathbf{g12}} & 0 & 0 \\ & \text{sym.} & & & \frac{1}{\mathbf{g13}} & 0 \\ & & & & & \frac{1}{\mathbf{g23}} \end{bmatrix} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix} + \begin{Bmatrix} \mathbf{a1} \\ \mathbf{a2} \\ \mathbf{a3} \\ 0 \\ 0 \\ 0 \end{Bmatrix} \Delta T \qquad \text{(Eqn. 2)}$$

where **e1**, **e2**, and **e3** are the moduli of elasticity; **u12**, **u13**, and **u23** are the Poisson's ratios; **g12**, **g13**, and **g23** are the shear moduli; and **a1**, **a2**, and **a3** are the coefficients of thermal expansion.

Note that the elastic moduli and the shear moduli must be positive. The Poisson's ratios may take on any values provided that the upper-left 3x3 portion of the stress-strain matrix is positive-definite (i.e., has a positive determinant.)

Anisotropic Materials

The behavior of an anisotropic material can be different in each of the three local coordinate directions. In addition, shearing behavior can be fully coupled with extensional behavior and can be affected by temperature change.

The anisotropic mechanical and thermal properties relate strain to stress and temperature change as follows:

$$\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{Bmatrix} = \begin{bmatrix} \frac{1}{\mathbf{e1}} & \frac{\mathbf{-u12}}{\mathbf{e2}} & \frac{\mathbf{-u13}}{\mathbf{e3}} & \frac{\mathbf{-u14}}{\mathbf{g12}} & \frac{\mathbf{-u15}}{\mathbf{g13}} & \frac{\mathbf{-u16}}{\mathbf{g23}} \\ & \frac{1}{\mathbf{e2}} & \frac{\mathbf{-u23}}{\mathbf{e3}} & \frac{\mathbf{-u24}}{\mathbf{g12}} & \frac{\mathbf{-u25}}{\mathbf{g13}} & \frac{\mathbf{-u26}}{\mathbf{g23}} \\ & & \frac{1}{\mathbf{e3}} & \frac{\mathbf{-u34}}{\mathbf{g12}} & \frac{\mathbf{-u35}}{\mathbf{g13}} & \frac{\mathbf{-u36}}{\mathbf{g23}} \\ & & & \frac{1}{\mathbf{g12}} & \frac{\mathbf{-u45}}{\mathbf{g13}} & \frac{\mathbf{-u46}}{\mathbf{g23}} \\ & & & & \frac{1}{\mathbf{g13}} & \frac{\mathbf{-u56}}{\mathbf{g23}} \\ & \text{sym.} & & & & \frac{1}{\mathbf{g23}} \end{bmatrix} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix} + \begin{Bmatrix} \mathbf{a1} \\ \mathbf{a2} \\ \mathbf{a3} \\ \mathbf{a12} \\ \mathbf{a13} \\ \mathbf{a23} \end{Bmatrix} \Delta T \qquad \text{(Eqn. 3)}$$

where **e1**, **e2**, and **e3** are the moduli of elasticity; **u12**, **u13**, and **u23** are the standard Poisson's ratios; **u14**, **u24**..., **u56** are the shear and coupling Poisson's ratios; **g12**, **g13**, and **g23** are the shear moduli; **a1**, **a2**, and **a3** are the coefficients of thermal expansion; and **a12**, **a13**, and **a23** are the coefficients of thermal shear.

Note that the elastic moduli and the shear moduli must be positive. The Poisson's ratios must be chosen so that the 6x6 stress-strain matrix is positive definite. This means that the determinant of the matrix must be positive.

These material properties can be evaluated directly from laboratory experiments. Each column of the elasticity matrix represents the six measured strains due to the application of the appropriate unit stress. The six thermal coefficients are the measured strains due to a unit temperature change.

Temperature-Dependent Properties

All of the mechanical and thermal properties given in Equations 1 to 3 may depend upon temperature. These properties are given at a series of specified material temperatures **t**. Properties at other temperatures are obtained by linear interpolation between the two nearest specified temperatures. Properties at temperatures outside the specified range use the properties at the nearest specified temperature. See Figure 33 (page 142) for examples.

If the Material properties are independent of temperature, you need only specify them at a single, arbitrary temperature.

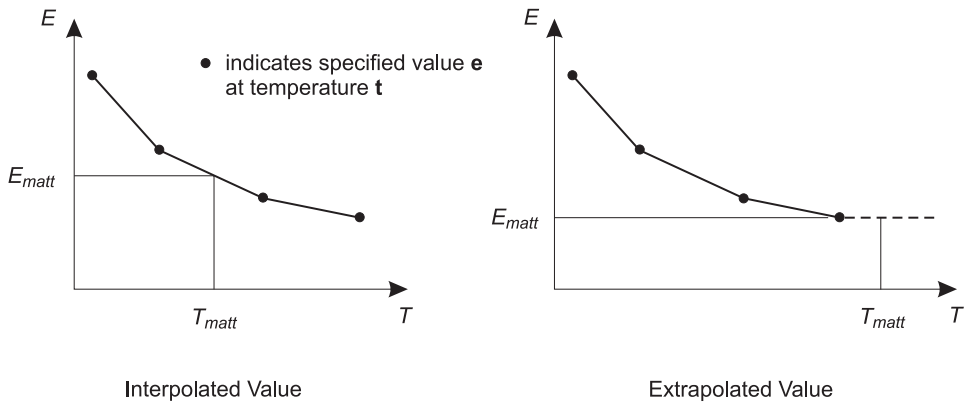


Figure 33
Determination of Property E_{matt} at Temperature T_{matt} from Function E(T)

Element Material Temperature

You can assign each element an **element material temperature**. This is the temperature at which temperature-dependent material properties are evaluated for the element. The properties at this fixed temperature are used for all analyses regardless of any temperature changes experienced by the element during loading. Thus the material properties are independent of the reference temperature and the load temperatures.

The element material temperature may be uniform over an element or interpolated from values given at the joints. In the latter case, a uniform material temperature is used that is the average of the joint values. The default material temperature for any element is zero.

The properties for a temperature-independent material are constant regardless of the element material temperatures specified.

Mass Density

For each Material you may specify a mass density, **m**, that is used for calculating the mass of the element. The total mass of the element is the product of the mass density (mass per unit volume) and the volume of the element. This mass is apportioned to each joint of the element. The same mass is applied along of the three

translational degrees of freedom. No rotational mass moments of inertia are computed.

Consistent mass units must be used. Typically the mass density is the same as the weight density divided by the acceleration due to gravity, but this is not required.

The mass density property is independent of temperature.

For more information:

- See Topic “Mass” (page 167) in Chapter “The Frame Element.”
- See Topic “Mass” (page 194) in Chapter “The Shell Element.”
- See Topic “Mass” (page 213) in Chapter “The Plane Element.”
- See Topic “Mass” (page 227) in Chapter “The Asolid Element.”
- See Topic “Mass” (page 238) in Chapter “The Solid Element.”

Weight Density

For each Material you may specify a weight density, **w**, that is used for calculating the self-weight of the element. The total weight of the element is the product of the weight density (weight per unit volume) and the volume of the element. This weight is apportioned to each joint of the element. Self-weight is activated using Self-weight Load and Gravity Load.

The weight density property is independent of temperature.

For more information:

- See Topic “Self-Weight Load” (page 283) in Chapter “Load Cases.”
- See Topic “Gravity Load” (page 284) in Chapter “Load Cases.”

Design-Type Indicator

You may specify a **design-type** indicator for each Material that indicates how it is to be treated for design by the SAP2000 graphical interface. The three options for this indicator, **ides**, are:

- Steel: Frame elements made of this material will be designed according to steel design codes

- Concrete: Frame elements made of this material will be designed according to concrete design codes
- Neither: Frame elements made of this material will not be designed

Chapter IX

The Frame Element

The **Frame** element is used to model beam-column and truss behavior in planar and three-dimensional structures.

Basic Topics for All Users

- Overview
- Joint Connectivity
- Degrees of Freedom
- Local Coordinate System
- Section Properties
- End Offsets
- End Releases
- Mass
- Self-Weight Load
- Concentrated Span Load
- Distributed Span Load
- Internal Force Output

Advanced Topics

- Advanced Local Coordinate System
- Non-prismatic Elements
- Gravity Load
- Temperature Load
- Prestress Load
- Joint 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

A Frame element is modeled as a straight line connecting two joints. Each element has its own local coordinate system for defining section properties and loads, and for interpreting output.

The element may be prismatic or non-prismatic. The non-prismatic formulation allows the element length to be divided into any number of segments over which properties may vary. The variation of the bending stiffness may be linear, parabolic, or cubic over each segment of length. The axial, shear, torsional, mass, and weight properties all vary linearly over each segment.

End offsets are available to account for the finite size of beam and column intersections. The end offsets may be made partially or fully rigid to model the stiffening effect that can occur when the ends of an element are embedded in beam and column intersections. End releases are also available to model different fixity conditions at the ends of the element.

Each Frame element may be loaded by gravity (in any direction), multiple concentrated loads, multiple distributed loads, loads due to prestressing cables, and loads due to temperature change.

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.

Joint Connectivity

A Frame element is represented by a straight line connecting two joints, **i** and **j**. 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.

Degrees of Freedom

The Frame element normally activates all six degrees of freedom at both of its connected joints.

However, the three rotational degrees of freedom will not be activated at a joint if the element does not provide any moment stiffness or moment load to that joint. This can occur under either of the following conditions:

- The end offset at that end is zero, and the geometric Section properties **j**, **i33**, and **i22** are all zero (**a** is non-zero; **as2** and **as3** are arbitrary), or
- The end offset at that end is zero; both bending rotations, R2 and R3, are released at that end; and the torsional rotation, R1, is released at either end

When these conditions apply to *both* ends, the element behaves as a truss member.

For more information:

- See Topic “Degrees of Freedom” (page 83) in Chapter “Joints and Degrees of Freedom.”
- See Topic “Section Properties” (page 153) in this chapter.
- See Topic “End Offsets” (page 162) in this chapter.
- See Topic “End Releases” (page 165) 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. The methods provided, however, provide sufficient power and flexibility to describe the orientation of Frame elements in the most complicated situations.

The simplest method, using the **default orientation** and the **Frame element coordinate angle**, is described in this topic. Additional methods for defining the Frame element local coordinate system are described in the next topic.

For more information:

- See Chapter “Coordinate Systems” (page 21) for a description of the concepts and terminology used in this topic.
- See Topic “Advanced Local Coordinate System” (page 149) in this chapter.

Longitudinal Axis 1

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

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 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 counterclockwise 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 34 (page 150) for examples.

Advanced Local Coordinate System

By default, the element local coordinate system is defined using the element coordinate angle measured with respect to the global +Z and +X directions, as described in the previous topic. In certain modeling situations it may be useful to have more control over the specification of the local coordinate system.

This topic describes how to define the orientation of the transverse local 2 and 3 axes with respect to an arbitrary reference vector when the element coordinate angle, **ang**, is zero. If **ang** is different from zero, it is the angle through which the local 2 and 3 axes are rotated about the positive local 1 axis from the orientation determined by the reference vector. The local 1 axis is always directed from end I to end J of the element.

For more information:

- See Chapter “Coordinate Systems” (page 21) for a description of the concepts and terminology used in this topic.
- See Topic “Local Coordinate System” (page 148) in this chapter.

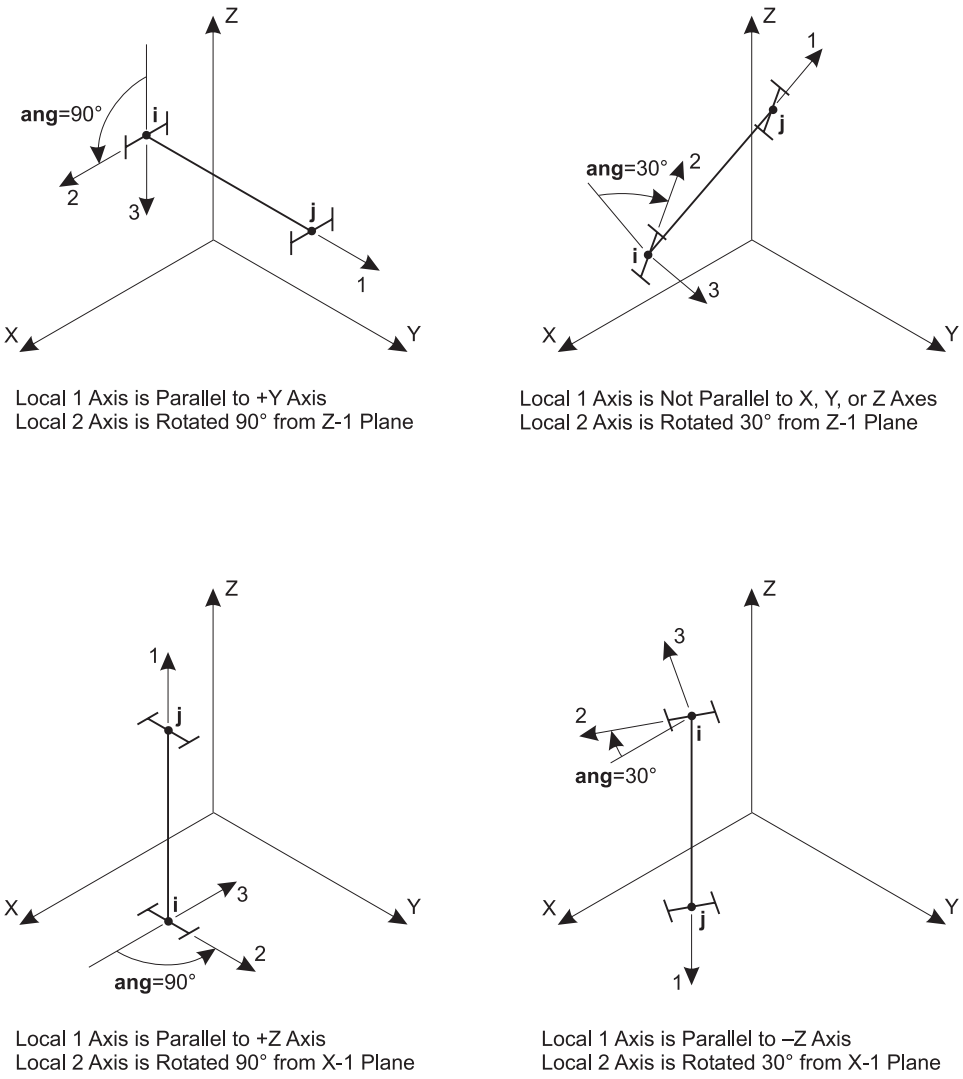


Figure 34
The Frame Element Coordinate Angle with Respect to the Default Orientation

Reference Vector

To define the transverse local axes 2 and 3, you specify a **reference vector** that is parallel to the desired 1-2 or 1-3 plane. The reference vector must have a positive projection upon the corresponding transverse local axis (2 or 3, respectively). This

means that the positive direction of the reference vector must make an angle of less than 90° with the positive direction of the desired transverse axis.

To define the reference vector, you must first specify or use the default values for:

- A primary coordinate direction **pldirp** (the default is +Z)
- A secondary coordinate direction **pldirs** (the default is +X). Directions **pldirs** and **pldirp** should not be parallel to each other unless you are sure that they are not parallel to local axis 1
- A fixed coordinate system **csys** (the default is zero, indicating the global coordinate system)
- The local plane, **local**, to be determined by the reference vector (the default is 12, indicating plane 1-2)

You may optionally specify:

- A pair of joints, **plveca** and **plvecb** (the default for each is zero, indicating the center of the element). If both are zero, this option is not used

For each element, the reference vector is determined as follows:

1. A vector is found from joint **plveca** to joint **plvecb**. If this vector is of finite length and is not parallel to local axis 1, it is used as the reference vector $\mathbf{V}_p$
2. Otherwise, the primary coordinate direction **pldirp** is evaluated at the center of the element in fixed coordinate system **csys**. If this direction is not parallel to local axis 1, it is used as the reference vector $\mathbf{V}_p$
3. Otherwise, the secondary coordinate direction **pldirs** is evaluated at the center of the element in fixed coordinate system **csys**. If this direction is not parallel to local axis 1, it is used as the reference vector $\mathbf{V}_p$
4. Otherwise, the method fails and the analysis terminates. This will never happen if **pldirp** is not parallel to **pldirs**

A vector is considered to be parallel to local axis 1 if the sine of the angle between them is less than 10^{-3}.

The use of the Frame element coordinate angle in conjunction with coordinate directions that define the reference vector is illustrated in Figure 35 (page 152). The use of joints to define the reference vector is shown in Figure 36 (page 152).

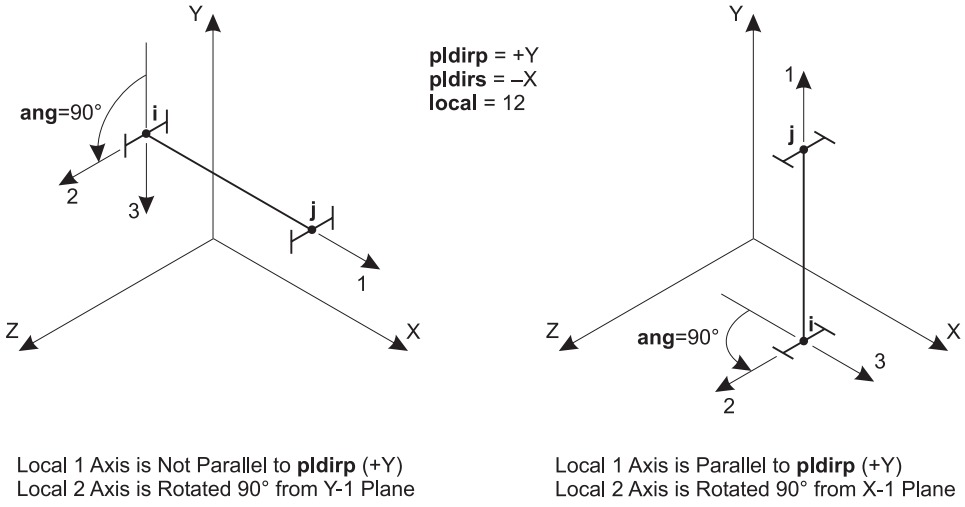


Figure 35
The Frame Element Coordinate Angle with Respect to Coordinate Directions

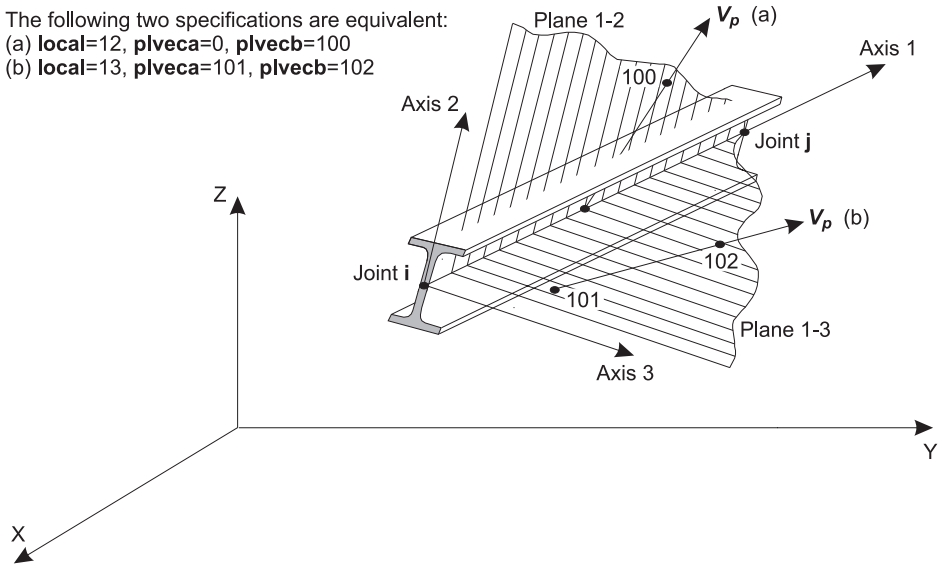


Figure 36
Using Joints to Define the Frame Element Local Coordinate System

Determining Transverse Axes 2 and 3

The program uses vector cross products to determine the transverse axes 2 and 3 once the reference vector has been specified. The three axes are represented by the three unit vectors $\mathbf{V}_1$, $\mathbf{V}_2$ and $\mathbf{V}_3$, respectively. The vectors satisfy the cross-product relationship:

$$\mathbf{V}_1 = \mathbf{V}_2 \times \mathbf{V}_3$$

The transverse axes 2 and 3 are defined as follows:

- If the reference vector is parallel to the 1-2 plane, then:

  $$\mathbf{V}_3 = \mathbf{V}_1 \times \mathbf{V}_p \quad \text{and}$$
  $$\mathbf{V}_2 = \mathbf{V}_3 \times \mathbf{V}_1$$

- If the reference vector is parallel to the 1-3 plane, then:

  $$\mathbf{V}_2 = \mathbf{V}_p \times \mathbf{V}_1 \quad \text{and}$$
  $$\mathbf{V}_3 = \mathbf{V}_1 \times \mathbf{V}_2$$

In the common case where the reference vector is perpendicular to axis $\mathbf{V}_1$, the transverse axis in the selected plane will be equal to $\mathbf{V}_p$.

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.

Section properties are of two basic types:

- Prismatic — all properties are constant along the full element length
- Non-prismatic — the properties may vary along the element length

Non-prismatic Sections are defined by referring to two or more previously defined prismatic Sections.

All of the following subtopics, except the last, describe the definition of prismatic Sections. The last subtopic, “Non-prismatic Sections”, describes how prismatic Sections are used to define non-prismatic Sections.

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 are parallel to the neutral axes 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 148) in this chapter for more information.

Material Properties

The material properties for the Section are specified by reference to a previously-defined Material. Isotropic material properties are used, even if the Material selected was defined as orthotropic or anisotropic. 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
- The coefficient of thermal expansion, **a1**, for axial expansion and thermal bending strain
- The mass density, **m**, for computing element mass
- The weight density, **w**, for computing Self-Weight and Gravity Loads

The material properties **e1**, **g12**, and **a1** are all obtained at the material temperature of each individual Frame element, and hence may not be unique for a given Section.

See Chapter "Material Properties" (page 135) for more information.

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 $\mathbf{a} \cdot \mathbf{e1}$;

- 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 37 (page 156).

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.

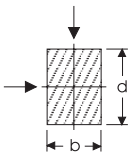
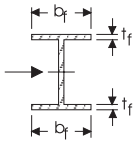
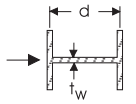
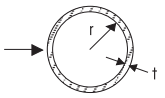
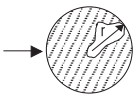
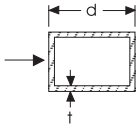
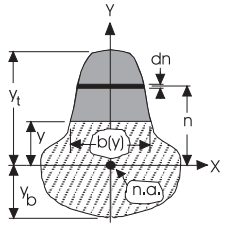
Section	Description	Effective Shear Area
	Rectangular Section Shear Forces parallel to the b or d directions	$5/6\ bd$
	Wide Flange Section Shear Forces parallel to flange	$5/3\ t_f\ b_f$
	Wide Flange Section Shear Forces parallel to web	$t_w\ d$
	Thin Walled Circular Tube Section Shear Forces from any 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 $Q(Y) = \int_{y}^{y_t} n\ b(n)\ dn$	$\dfrac{I_x^2}{\int_{y_b}^{y_t} \dfrac{Q^2(y)}{b(y)}\ dy}$

Figure 37
Shear Area Formulae

Automatic Section Property Calculation

The six geometric Section properties can be automatically calculated from specified dimensions for the simple shapes shown in Figure 38 (page 158). The required dimensions for each shape are shown in the figure.

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. Three database files are supplied with SAP2000:

- AISC.PRO: American Institute of Steel Construction shapes
- CISC.PRO: Canadian Institute of Steel Construction shapes
- SECTIONS.PRO: This is just a copy of AISC.PRO.

Additional property database files may be created 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 by SAP2000 to the units used in the input data file.

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.

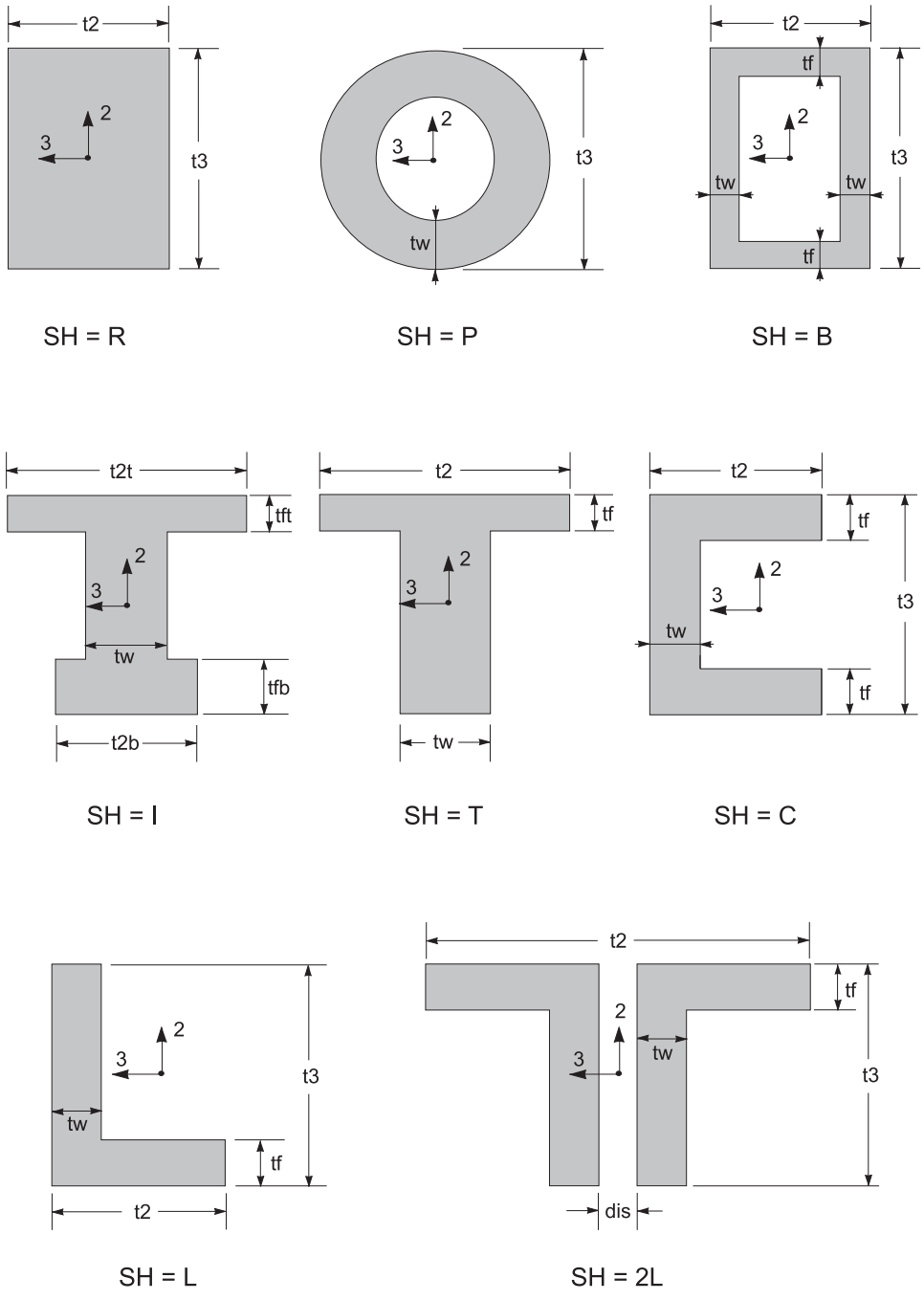


Figure 38
Automatic Section Property Calculation

The shape type labels available for a given database file are stored in an associated label file with extension ".LBL". For examples, the labels for database file AISC.PRO are stored in file AISC.LBL. The label file is a text file that may be printed or viewed with a text editor. Each line in the label file shows the one or two labels corresponding to a single shape type stored in the database file.

You may select one database file to be used when defining a given Frame Section. If shape type **sh** cannot be found in the database file, an error results. The database file in use can be changed at any time when defining Sections. If no database filename is specified, the default file SECTIONS.PRO is used. You may copy any property database file to SECTIONS.PRO.

All Section property database files, including file SECTIONS.PRO, must be located either in the directory that contains the input data file, or in the directory that contains the SAP2000 executable files. If a specified database file is present in both directories, the program will use the file in the input-data-file directory.

Additional Mass and Weight

You may specify mass and/or weight for a Section that acts *in addition to* the mass and weight of the material. The additional mass and weight are specified *per unit of length* using the parameters **mpl** and **wpl**, respectively. They could be used, for example, to represent the effects of nonstructural material that is attached to a Frame element.

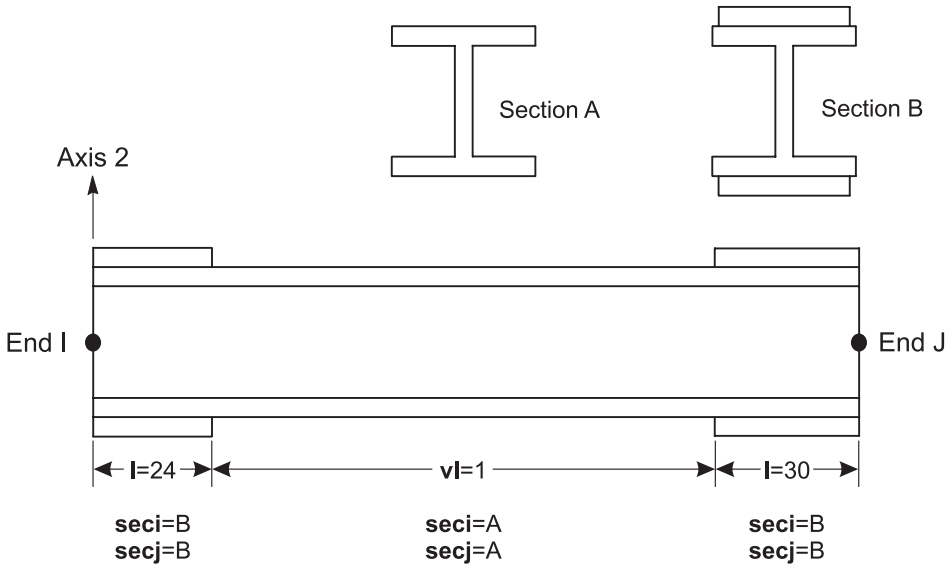
The additional mass and weight act regardless of the cross-sectional area of the Section. The default values for **mpl** and **wpl** are zero for all shape types.

Non-prismatic Sections

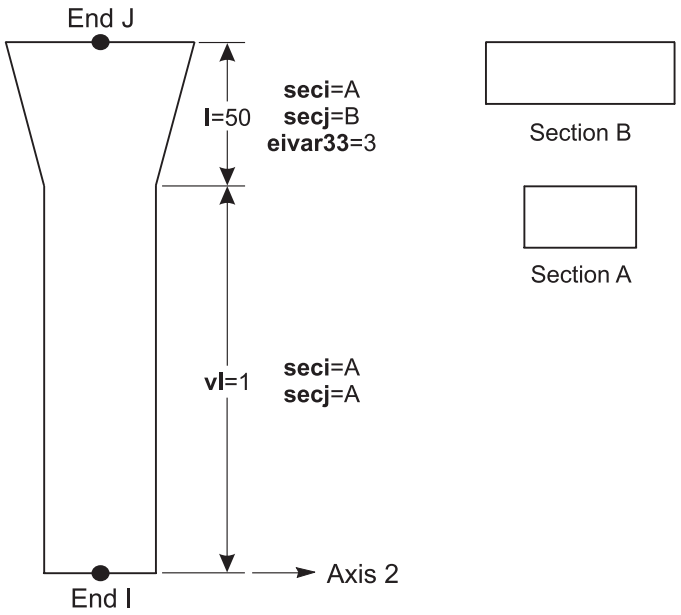
Non-prismatic Sections may be defined for which the properties vary along the element length. You may specify that the element length be divided into any number of segments; these do not need to be of equal length. Most common situations can be modeled using from one to five segments.

The variation of the bending stiffnesses may be linear, parabolic, or cubic over each segment of length. The axial, shear, torsional, mass, and weight properties all vary linearly over each segment. Section properties may change discontinuously from one segment to the next.

See Figure 39 (page 160) for examples of non-prismatic Sections.



Steel Beam with Cover Plates at Ends



Concrete Column with Flare at Top

Figure 39
Examples of Non-prismatic Sections

Segment Lengths

The length of a non-prismatic segment may be specified as either a variable length, **vl**, or an absolute length, **l**. The default is **vl** = 1.

When a non-prismatic Section is assigned to an element, the actual lengths of each segment for that element are determined as follows:

- The clear length of the element, L_c, is first calculated as the total length minus the end offsets:

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

  See Topic “End Offsets” (page 162) in this chapter for more information.
- If the sum of the absolute lengths of the segments exceeds the clear length, they are scaled down proportionately so that the sum equals the clear length. Otherwise the absolute lengths are used as specified.
- The remaining length (the clear length minus the sum of the absolute lengths) is divided among the segments having variable lengths in the same proportion as the specified lengths. For example, for two segments with **vl** = 1 and **vl** = 2, one third of the remaining length would go to the first segment, and two thirds to the second segment.

Starting and Ending Sections

The properties for a segment are defined by specifying:

- The label, **seci**, of a previously defined prismatic Section that defines the properties at the start of the segment, i.e., at the end closest to joint **i**.
- The label, **secj**, of a previously defined prismatic Section that defines the properties at the end of the segment, i.e., at the end closest to joint **j**. The starting and ending Sections may be the same if the properties are constant over the length of the segment.

The Material would normally be the same for both the starting and ending Sections and only the geometric properties would differ, but this is not required.

Variation of Properties

Non-prismatic Section properties are interpolated along the length of each segment from the values at the two ends.

The variation of the bending stiffnesses, **i33·e1** and **i22·e1**, are defined by specifying the parameters **eivar33** and **eivar22**, respectively. Assign values of 1, 2, or 3 to

these parameters to indicate variation along the length that is linear, parabolic, or cubic, respectively.

Specifically, the **eivar33**-th root of the bending stiffness in the 1-2 plane:

$$\sqrt[\mathbf{eivar33}]{\mathbf{i33} \cdot \mathbf{e1}}$$

varies linearly along the length. This usually corresponds to a linear variation in one of the Section dimensions. For example, referring to Figure 38 (page 158): a linear variation in **t2** for the rectangular shape would require **eivar33**=1, a linear variation in **t3** for the rectangular shape would require **eivar33**=3, and a linear variation in **t3** for the I-shape would require **eivar33**=2.

The interpolation of the bending stiffness in the 1-2 plane, **i22** · **e1**, is defined in the same manner by the parameter **eivar22**.

The remaining properties are assumed to vary linearly between the ends of each segment:

- Stiffnesses: **a** · **e1**, **j** · **g12**, **as2** · **g12**, and **as3** · **g12**
- Mass: **a**·**m** + **mpl**
- Weight: **a**·**w** + **wpl**

If a shear area is zero at either end, it is taken to be zero along the full segment, thus eliminating all shear deformation in the corresponding bending plane for that segment.

Effect upon End Offsets

Properties vary only along the clear length of the element. Section properties within end offset **ioff** are constant using the starting Section of the first segment. Section properties within end offset **joff** are constant using the ending Section of the last segment.

See Topic “End Offsets” (page 162) in this chapter for more information.

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

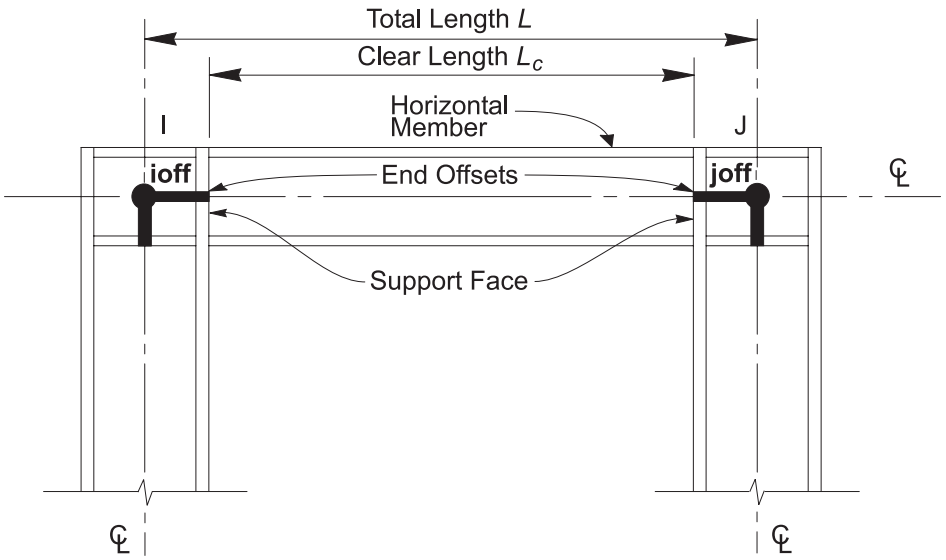


Figure 40
Frame Element End Offsets

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 40 (page 163).

End offsets are automatically calculated by the SAP2000 graphical interface for each element 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:

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

where L is the total element length. See Figure 40 (page 163).

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.

Rigid-end Factor

An analysis based upon the centerline-to-centerline (joint-to-joint) geometry of Frame elements may overestimate deflections in some structures. This is due to the stiffening effect caused by overlapping cross sections at a connection. It is more likely to be significant in concrete than in steel structures.

You may specify a **rigid-end factor** for each element using parameter **rigid**, which gives the fraction of each end offset that is assumed to be rigid for bending and shear deformation. The length **rigid**·**ioff**, starting from joint **i**, is assumed to be rigid. Similarly, the length **rigid**·**joff** is rigid at joint **j**. The flexible length L_f of the element is given by:

$$L_f = L - \mathbf{rigid}\,(\,\mathbf{ioff} + \mathbf{joff}\,)$$

The rigid-zone offsets never affect axial and torsional deformation. The full element length is assumed to be flexible for these deformations.

The default value for **rigid** is zero. The maximum value of unity would indicate that the end offsets are fully rigid. You must use engineering judgment to select the appropriate value for this parameter. It will depend upon the geometry of the connection, and may be different for the different elements that frame into the connection. Typically the value for **rigid** would not exceed about 0.5.

Effect upon Non-prismatic Elements

At each end of a non-prismatic element, the Section properties are assumed to be constant within the length of the end offset. Section properties vary only along the clear length of the element between support faces. This is not affected by the value of the rigid-end factor, **rigid**.

See Subtopic “Non-prismatic Sections” (page 159) in this chapter for more information.

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. This is not affected by the value of the rigid-end factor, **rigid**.

See Topic "Internal Force Output" (page 176) in this chapter for more information.

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* (i.e., it acts as if **rigid** = 1). This does not affect the values of the rigid-end factor at the other end or in the other bending plane.

See Topic "End Releases" (page 165) 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.

In the example shown in Figure 41 (page 166), 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.

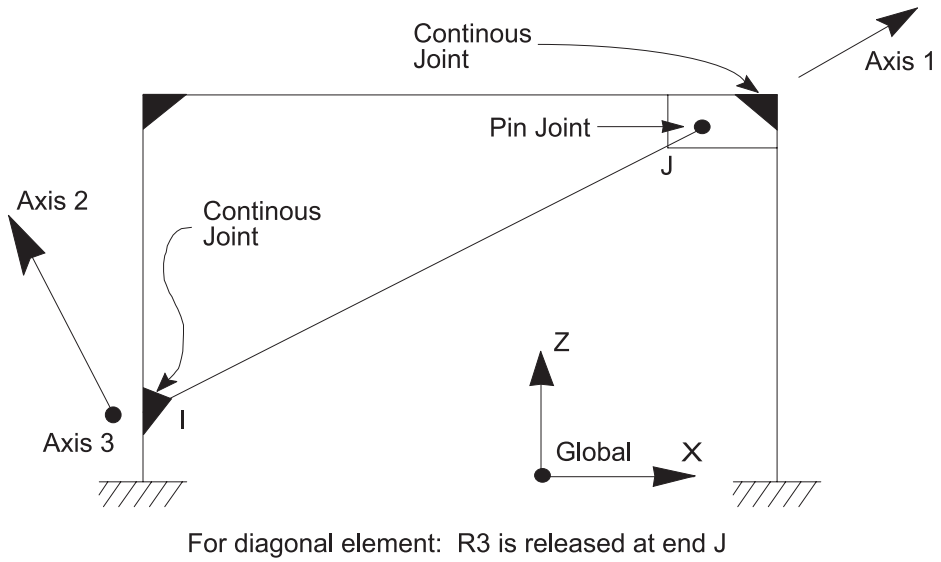


Figure 41
Frame Element End Releases

- 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 162) in this chapter for more information.

Effect upon Prestress Load

Certain end releases (e.g., axial) are not compatible with the presence of prestressing cables, even though the program will permit both to be present. End releases affect only the portion of the prestress load that is uniformly distributed over the element, not the loads that are applied to the joints.

See Topic “Prestress Load” (page 174) 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**, plus the additional mass per unit length, **mpl**.

For non-prismatic elements, the mass varies linearly over each non-prismatic segment of the element, and is constant within the end offsets.

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 “Mass Density” (page 142) in Chapter “Material Properties.”
- See Topic “Section Properties” (page 153) in this chapter for the definition of **a** and **mpl**.
- See Subtopic “Non-prismatic Sections” (page 159) in this chapter.
- See Topic “End Offsets” (page 162) in this chapter.
- See Chapter “Static and Dynamic Analysis” (page 291).

Self-Weight Load

Self-Weight Load activates 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**, plus the additional weight per unit length, **wpl**.

For non-prismatic elements, the self-weight varies linearly over each non-prismatic segment of the element, and is constant within the end offsets.

Self-Weight Load always acts downward, in the global –Z direction. You may scale the self-weight by a single scale factor that applies equally to all elements in the structure.

For more information:

- See Topic “Weight Density” (page 143) in Chapter “Material Properties” for the definition of **w**.
- See Topic “Section Properties” (page 153) in this chapter for the definition of **a** and **wpl**..
- See Subtopic “Non-prismatic Sections” (page 159) in this chapter.
- See Topic “End Offsets” (page 162) in this chapter.
- See Topic “Self-Weight Load” (page 283) in Chapter “Load Cases.”

Gravity Load

Gravity Load can be applied to each Frame element to activate the self-weight of the element. Using Gravity Load, the self-weight can be scaled and applied in any direction. Different scale factors and directions can be applied to each element.

If all elements are to be loaded equally and in the downward direction, it is more convenient to use Self-Weight Load.

For more information:

- See Topic “Self-Weight Load” (page 168) in this chapter for the definition of self-weight for the Frame element.
- See Topic “Gravity Load” (page 284) in Chapter “Load Cases.”

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 a fixed coordinate system (global or alternate coordinates) 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, **rd**, measured from joint **i**. This must satisfy $0 \le \mathbf{rd} \le 1$. The relative distance is the fraction of element length;
- Specifying an absolute distance, **d**, measured from joint **i**. This must satisfy $0 \le \mathbf{d} \le L$, where L is the element length.

Any number of concentrated loads may be applied to each element. Loads given in fixed coordinates are transformed to the element local coordinate system. See Figure 42 (page 170). Multiple loads that are applied at the same location are added together.

See Chapter “Load Cases” (page 279) 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 a fixed coordinate system (global or alternate coordinates) or in the element local coordinate system.

See Chapter “Load Cases” (page 279) for more information.

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 $0 \le \mathbf{rda} < \mathbf{rdb} \le 1$. The relative distance is the fraction of element length;

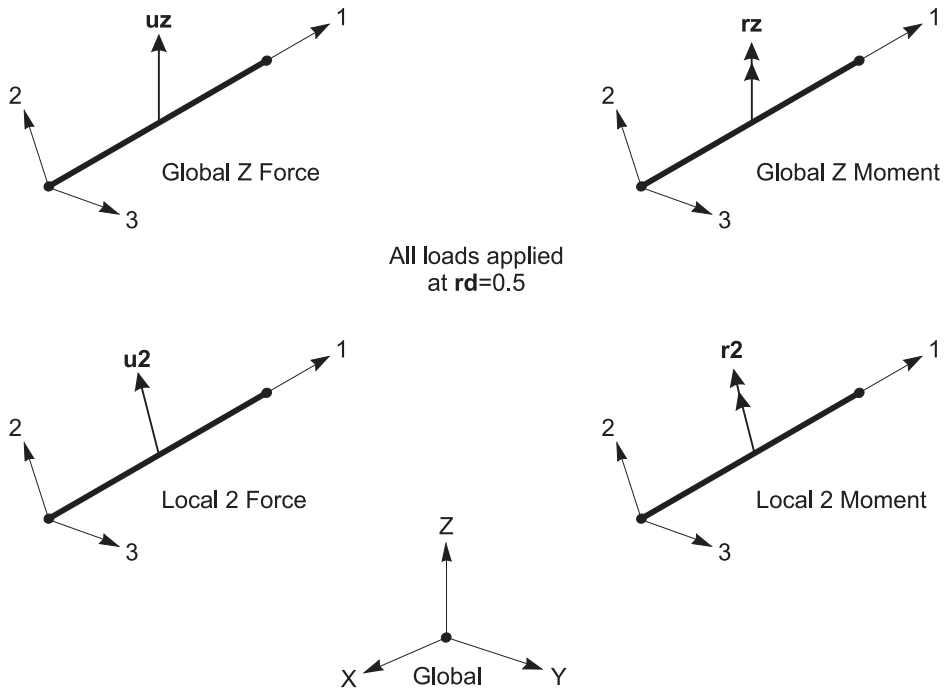


Figure 42
Examples of the Definition of Concentrated Span Loads

- Specifying two absolute distances, **da** and **db**, measured from joint **i**. They must satisfy $0 \le \mathbf{da} < \mathbf{db} \le L$, where L is the element length;
- Specifying no distances, which indicates the full length of the element.

Load Intensity

The load intensity is a force or moment per unit of length. Except for the case of projected loads described below, the intensity is measured per unit of *element* 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 43 (page 171) and Figure 44 (page 172).

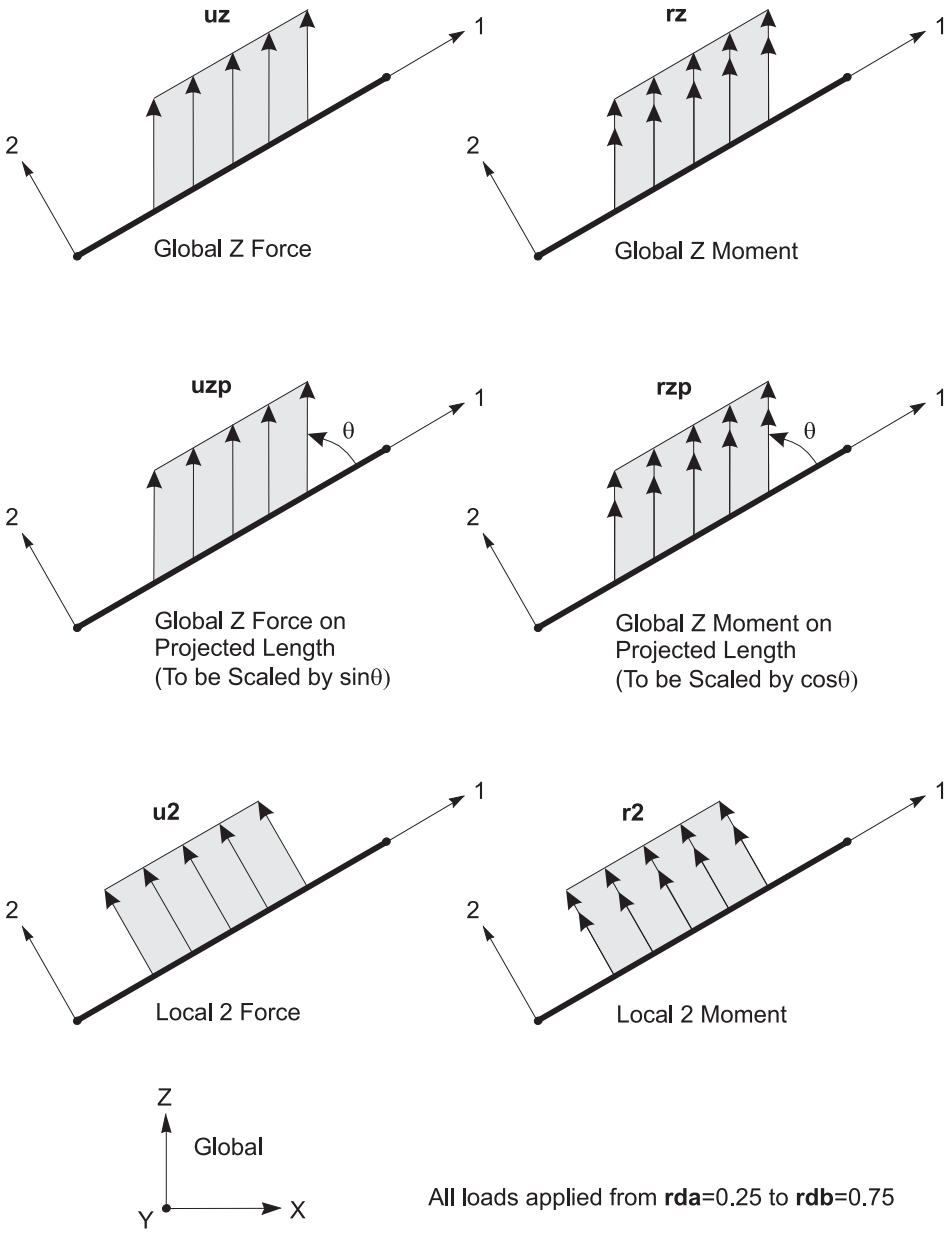


Figure 43
Examples of the Definition of Distributed Span Loads

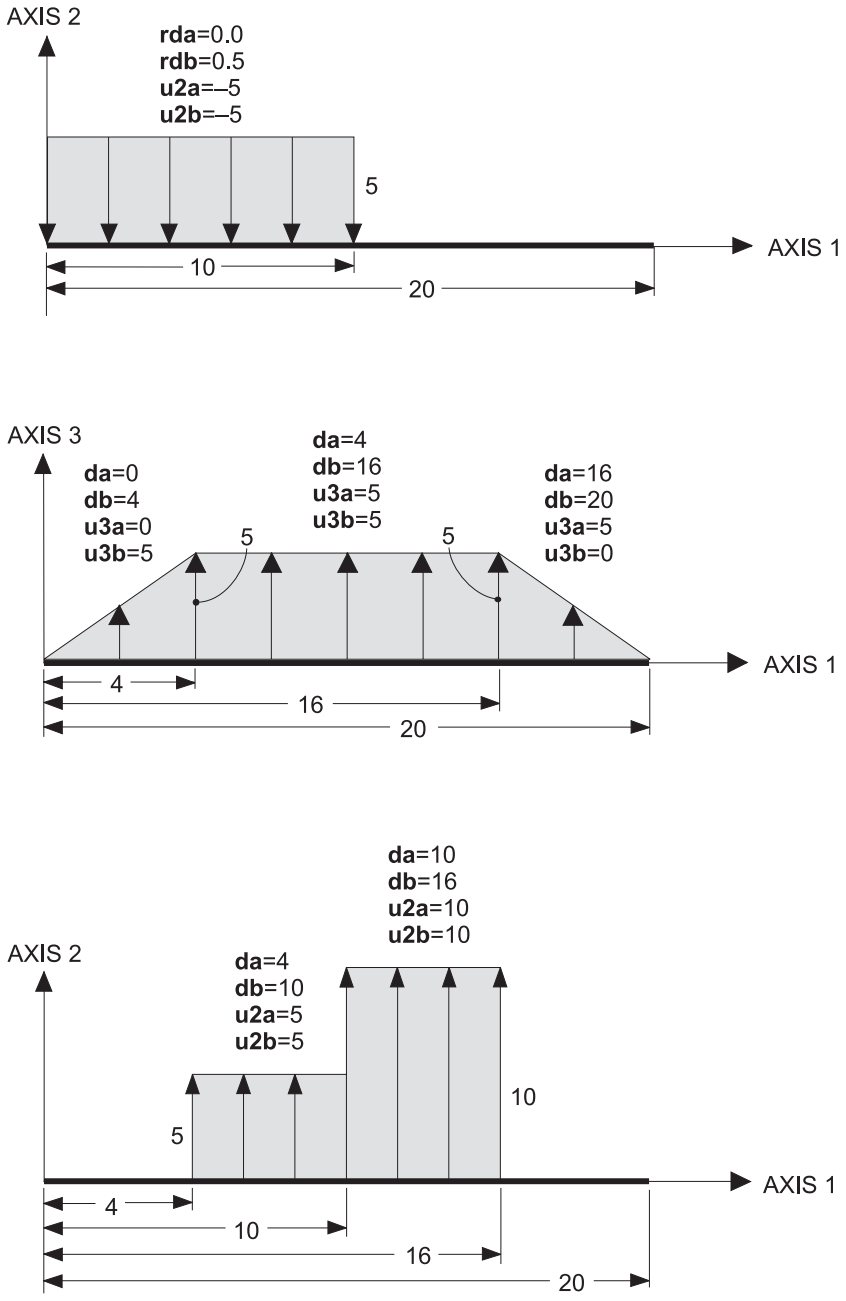


Figure 44
Examples of Distributed Span Loads

Projected Loads

A distributed snow or wind load produces a load intensity (force per unit of element length) that is proportional to the sine of the angle between the element and the direction of loading. This is equivalent to using a fixed load intensity that is measured per unit of *projected* element length. The fixed intensity would be based upon the depth of snow or the wind speed; the projected element length is measured in a plane perpendicular to the direction of loading.

Distributed Span Loads may be specified as acting upon the projected length. The program handles this by reducing the load intensity according to the angle, θ, between the element local 1 axis and the direction of loading. Projected force loads are scaled by $\sin\theta$, and projected moment loads are scaled by $\cos\theta$. The reduced load intensities are then applied per unit of element length.

The scaling of the moment loads is based upon the assumption that the moment is caused by a force acting upon the projected element length. The resulting moment is always perpendicular to the force, thus accounting for the use of the cosine instead of the sine of the angle. The specified intensity of the moment should be computed as the product of the force intensity and the *perpendicular* distance from the element to the force. The appropriate sign of the moment must be given.

Temperature Load

The Temperature Load creates thermal strain in the Frame element. This strain is given by the product of the Material coefficient of thermal expansion and the temperature change of the element. The temperature change is measured from the element Reference Temperature to the element Load Temperature.

Three independent Load Temperature fields may be specified:

- Temperature, **t**, which is constant over the cross section and produces axial strains
- Temperature gradient, **t2**, which is linear in the local 2 direction and produces bending strains in the 1-2 plane
- Temperature gradient, **t3**, which is linear in the local 3 direction and produces bending strains in the 1-3 plane

Temperature gradients are specified as the *change* in temperature per unit length. The temperature gradients are positive if the temperature increases (linearly) in the

positive direction of the element local axis. The gradient temperatures are zero at the neutral axes, hence no axial strain is induced.

Each of the three Load Temperature fields may be constant along the element length or interpolated from values given at the joints.

The Reference Temperature gradients are always taken to be zero, hence the temperature changes that produce the bending strain are equal to the Load Temperature gradients.

See Chapter “Load Cases” (page 279) for more information.

Prestress Load

Any of the Frame elements in the model can be subjected to loading produced by one or more Prestressing cables. This load always acts in the local 1-2 plane of the element.

See Chapter “Load Cases” (page 279) for more information.

Prestressing Cables

Any number of Prestressing cables may act on a single element. Each cable is subject to the following assumptions and specifications:

- The tension, **t**, is assumed to be constant along the length and does not change with element deformation.
- The drape configuration is assumed to be parabolic. It is specified by giving three dimensions measured from the local 1 axis , as shown in Figure 45 (page 175):
  - The drapes at the two ends, **di** and **dj**, measured in the positive 2 direction
  - The drape at the center, **dc**, measured in the negative 2 direction
- The cable is assumed to act in a narrow duct within the element, so that the transverse deflection of the Frame element and the Prestressing cable are the same. This affects the P-delta force in the element.

Prestress Load

Each Prestressing cable produces a set of self-equilibrating forces and moments that are proportional to the cable tension, **t**:

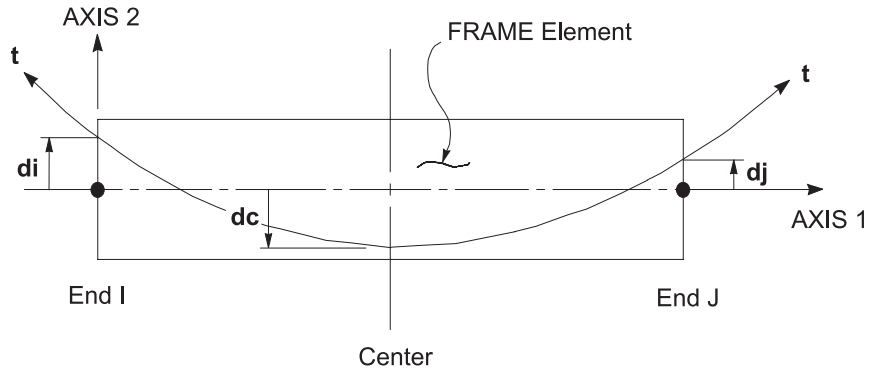


Figure 45
Prestressing Cable Profile

- Tensile forces acting on joints **i** and **j**;
- Moments acting on joints **i** and **j** that are proportional to the drapes **di** and **dj**, respectively;
- Shear forces acting on joints **i** and **j** that are proportional to the slopes of the cable at ends **i** and **j**, respectively;
- A uniform, distributed load acting on the element that is proportional to the curvature of the cable.

The sum of these forces and moments for all Prestressing cables acting on a Frame element forms the unscaled prestress load for that element. This load has no effect upon the structure until scaled and applied using the Prestress Load specification. All cables acting on a given element are applied simultaneously and scaled by the same factor.

Only the uniform load acting on the element is affected by end releases and other properties of the element. The forces and moments acting on joints **i** and **j** are independent of all element properties.

See Topic “End Releases” (page 165) in this chapter for more information.

Effect upon P-Delta Analysis

It is assumed that the transverse deflections of a Frame element and its Prestressing cables are the same. This means that the net P-Delta axial force acting on a Frame

element is the sum of the axial forces in the element and all of its cables, taken in the P-Delta load combination.

The tension in a Prestressing cable tends to stiffen a Frame element against transverse deflection when a P-Delta analysis is performed. If the element is allowed to deform axially the cable will also produce compression in the element, reducing the net axial force in the element. Thus the P-Delta axial force due to a Prestressing cable alone will vary from zero to the scaled tension force in the cable, depending upon the degree of axial restraint. The net effect will always be some degree of stiffening (zero or more); it will never cause buckling.

If the actual Prestressing cables are not constrained to deflect transversely with the Frame element, special modeling procedures are needed when P-Delta effects are important. Each such Frame element is replaced by two elements connected to the same joints:

- The first element represents the beam in the absence of prestressing. It is given the real Material and Section properties and is assigned all Loads except the Prestress Load.
- The second element represents the Prestressing cables. It is given arbitrary Material properties and all geometric Section properties are set to zero. The Prestressing cables and the Prestress Load are assigned to this element.

This model can create a negative P-Delta axial force in the first element due to prestressing alone, and may cause buckling of that element.

See Chapter “P-Delta Analysis” (page 337) for the definition of P-Delta axial force and the P-delta load combination for more information.

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 46 (page 178). 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, Specs, Histories, Moving Loads, and Combos. History results are the maximum and minimum values attained during the analysis, and are only available if specifically requested in the definition of the individual History cases. These can be printed in the output file and displayed using the SAP2000 graphical interface. See Subtopic "Internal Forces in the Output File" below.

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

For more information:

- See Chapter "Load Cases" (page 279).
- See Chapter "Static and Dynamic Analysis" (page 291).
- See Chapter "Bridge Analysis" (page 357).
- See Chapter "The Output Files" (page 395).

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

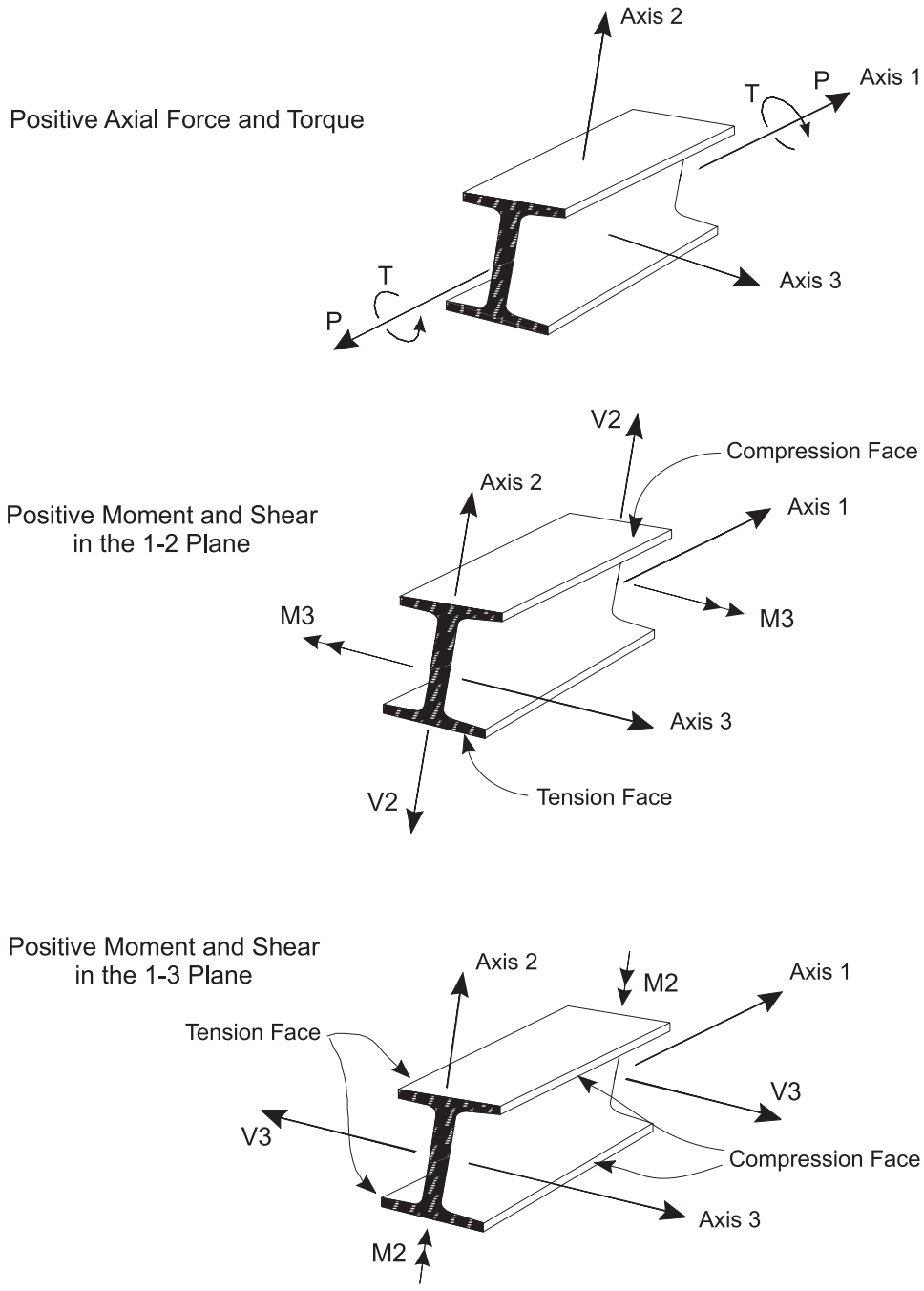


Figure 46
Frame Element Internal Forces and Moments

joint. Output will only be produced at joints **i** or **j** when the corresponding end offset is zero.

See Topic “End Offsets” (page 162) in this chapter for more information.

Internal Forces in the Output File

Frame element internal forces and moments are printed in the output file under the heading:

```
FRAME ELEMENT INTERNAL FORCES
```

All internal forces for a single element are printed together. Elements are presented in alpha-numeric order. For each Analysis Case internal forces are printed at all output points. The locations of the output points are identified by the relative distance of the points from joint **i**. The relative distance is a fraction of the element length (between zero and one.)

Typical Frame element output is shown in Figure 47 (page 180).

By default, no internal forces are printed. You may specify which Analysis Cases are to be printed. The internal forces can always be displayed using the SAP2000 graphical interface even if no results have been selected for printing.

See Chapter “The Output Files” (page 395) for more information.

Joint Force Output

The **Frame element joint forces** are concentrated forces and moments acting at the joints of the element that represent the effect of the rest of the structure upon the element.

Element joint forces must not be confused with internal forces and moments which, like stresses, act *throughout* the element, not just at the joints.

See Topic “Element Joint Force Output” (page 402) in Chapter “The Output Files” for more information.

```
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S

ELEM       1 ================== LENGTH =    12.000000

LOAD       1 ------------------

REL DIST          P          V2          V3          T          M2          M3
0.00000   -9.73E-11 -1.00000E+3    7.35E-13  2.02612E+3   -1.35E-12  7.20000E+4
0.50000   -9.73E-11 -1.00000E+3    7.35E-13  2.02612E+3    3.06E-12  7.80000E+4
1.00000   -9.73E-11 -1.00000E+3    7.35E-13  2.02612E+3    7.47E-12  8.40000E+4

SPEC    SPEC ------------------

REL DIST          P          V2          V3          T          M2          M3
0.00000  126.377896   68.323489    0.972230   89.135262    1.878252  3.19034E+3
0.50000  126.377896   68.323489    0.972230   89.135262    3.955129  3.58275E+3
1.00000  126.377896   68.323489    0.972230   89.135262    9.788509  3.97870E+3

COMB    BOTH ------------------  MAX

REL DIST          P          V2          V3          T          M2          M3
0.00000  126.377896 -931.676511    0.972230  2.11526E+3    1.878252  7.51903E+4
0.50000  126.377896 -931.676511    0.972230  2.11526E+3    3.955129  8.15828E+4
1.00000  126.377896 -931.676511    0.972230  2.11526E+3    9.788509  8.79787E+4

COMB    BOTH ------------------  MIN

REL DIST          P          V2          V3          T          M2          M3
0.00000 -126.377896 -1.06832E+3   -0.972230  1.93699E+3   -1.878252  6.88097E+4
0.50000 -126.377896 -1.06832E+3   -0.972230  1.93699E+3   -3.955129  7.44172E+4
1.00000 -126.377896 -1.06832E+3   -0.972230  1.93699E+3   -9.788509  8.00213E+4

ELEM       2 ================== LENGTH =    12.000000

LOAD       1 ------------------

REL DIST          P          V2          V3          T          M2          M3
0.00000   -9.73E-11 -1.00000E+3   -7.35E-13 -2.02612E+3    1.35E-12  7.20000E+4
0.50000   -9.73E-11 -1.00000E+3   -7.35E-13 -2.02612E+3   -3.06E-12  7.80000E+4
1.00000   -9.73E-11 -1.00000E+3   -7.35E-13 -2.02612E+3   -7.47E-12  8.40000E+4

SPEC    SPEC ------------------

REL DIST          P          V2          V3          T          M2          M3
0.00000  126.377896   68.323489    0.972230   89.135262    1.878252  3.19034E+3
0.50000  126.377896   68.323489    0.972230   89.135262    3.955129  3.58275E+3
1.00000  126.377896   68.323489    0.972230   89.135262    9.788509  3.97870E+3
```

...Output Edited for Brevity...

Figure 47
Frame Element Internal Forces in the Output File

Chapter X

The Shell Element

The **Shell** element is used to model shell, membrane, and plate behavior in planar and three-dimensional structures.

Basic Topics for All Users

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

Advanced Topics

- Advanced Local Coordinate System
- Gravity Load

- Surface Pressure Load
- Temperature Load
- Joint Force 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 thick-plate (Mindlin/Reissner) formulation is used which includes the effects of transverse shearing deformation. Optionally, you may choose a thin-plate (Kirchhoff) formulation that neglects 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. Temperature-dependent, orthotropic material properties are allowed. Each element may be loaded by gravity and uniform loads in any direction; surface pressure on the top, bottom, and side faces; and loads due to temperature change.

An 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.

Joint Connectivity

Each Shell element may have either of the following shapes, as shown in Figure 48 (page 184):

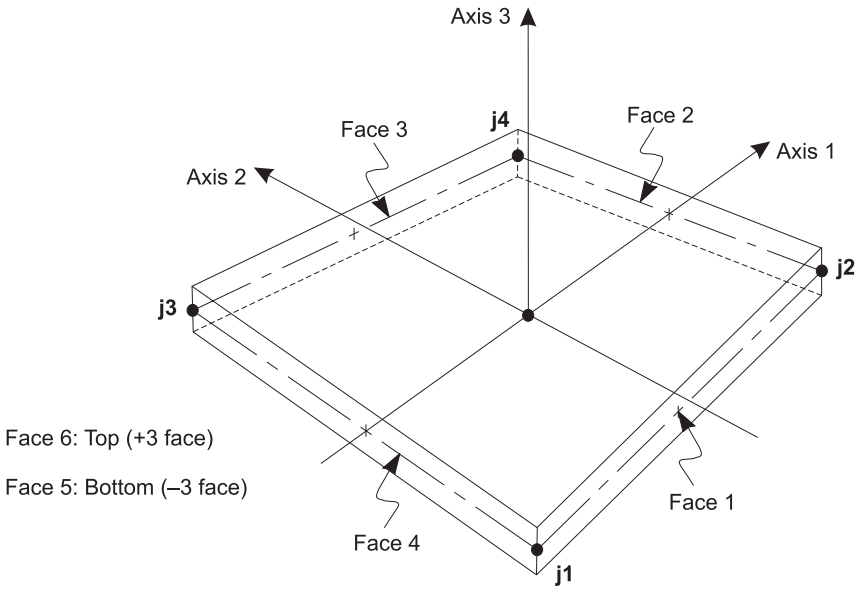
- Quadrilateral, defined by the four joints **j1**, **j2**, **j3**, and **j4**.
- Triangular, defined by the three joints **j1**, **j2**, and **j3**.

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 49 (page 185).

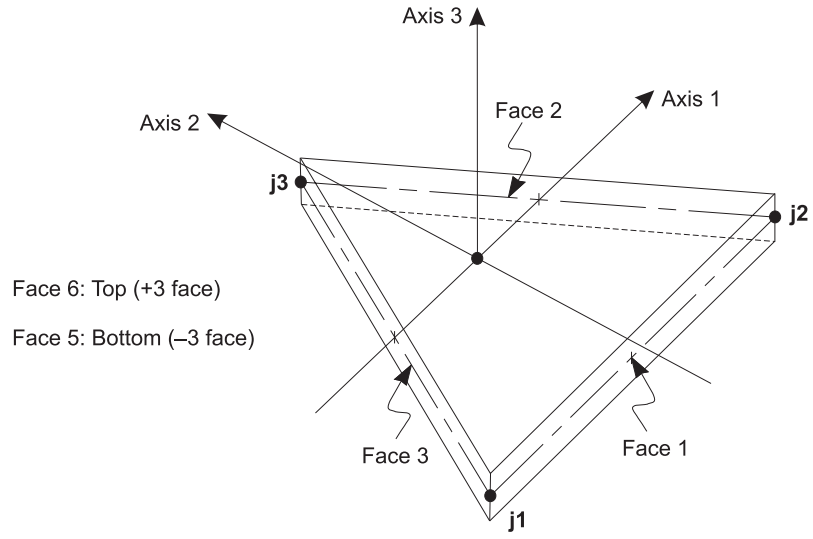
The locations of the joints should be chosen to meet the following geometric conditions:

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



Four-node Quadrilateral Shell Element



Three-node Triangular Shell Element

Figure 48
Shell Element Joint Connectivity and Face Definitions

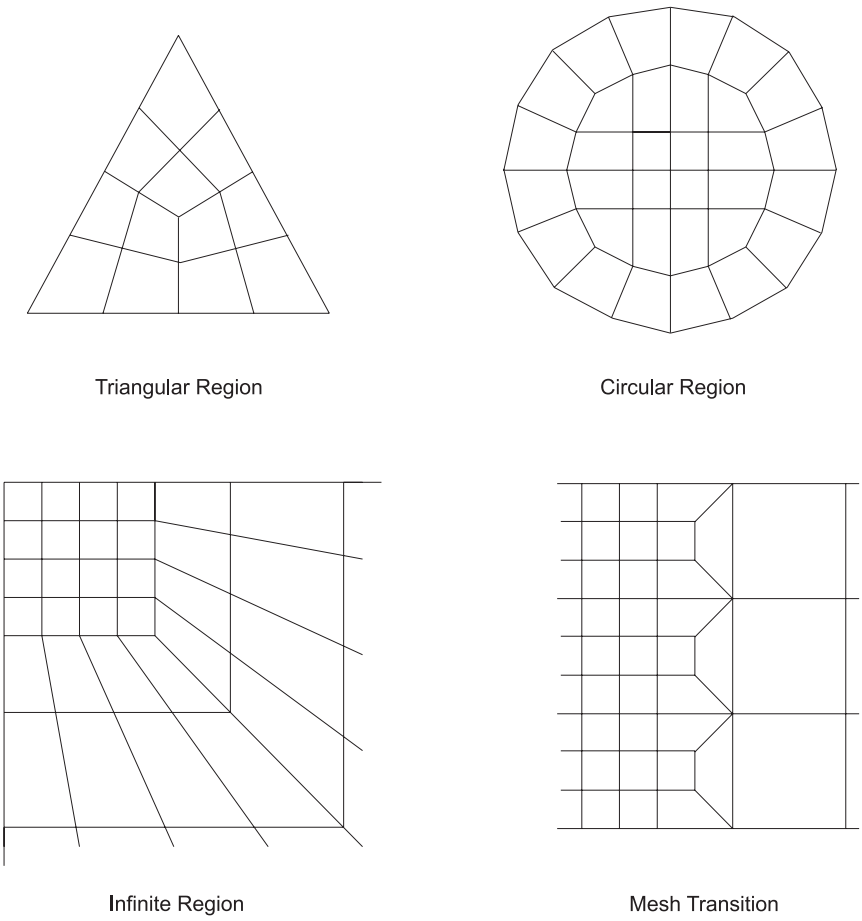


Figure 49
Mesh Examples Using the Quadrilateral Shell Element

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 83) 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. The methods provided, however, provide sufficient power and flexibility to describe the orientation of Shell elements in the most complicated situations.

The simplest method, using the **default orientation** and the **Shell element coordinate angle**, is described in this topic. Additional methods for defining the Shell element local coordinate system are described in the next topic.

For more information:

- See Chapter “Coordinate Systems” (page 21) for a description of the concepts and terminology used in this topic.
- See Topic “Advanced Local Coordinate System” (page 187) in this chapter.

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 counterclockwise. 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 along the global +Y direction
- The local 1 axis is 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.

Element 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 counterclockwise 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 50 (page 188) for examples.

Advanced Local Coordinate System

By default, the element local coordinate system is defined using the element coordinate angle measured with respect to the global +Z and +Y directions, as described in the previous topic. In certain modeling situations it may be useful to have more control over the specification of the local coordinate system.

This topic describes how to define the orientation of the tangential local 1 and 2 axes, with respect to an arbitrary reference vector when the element coordinate angle, **ang**, is zero. If **ang** is different from zero, it is the angle through which the local 1 and 2 axes are rotated about the positive local 3 axis from the orientation deter-

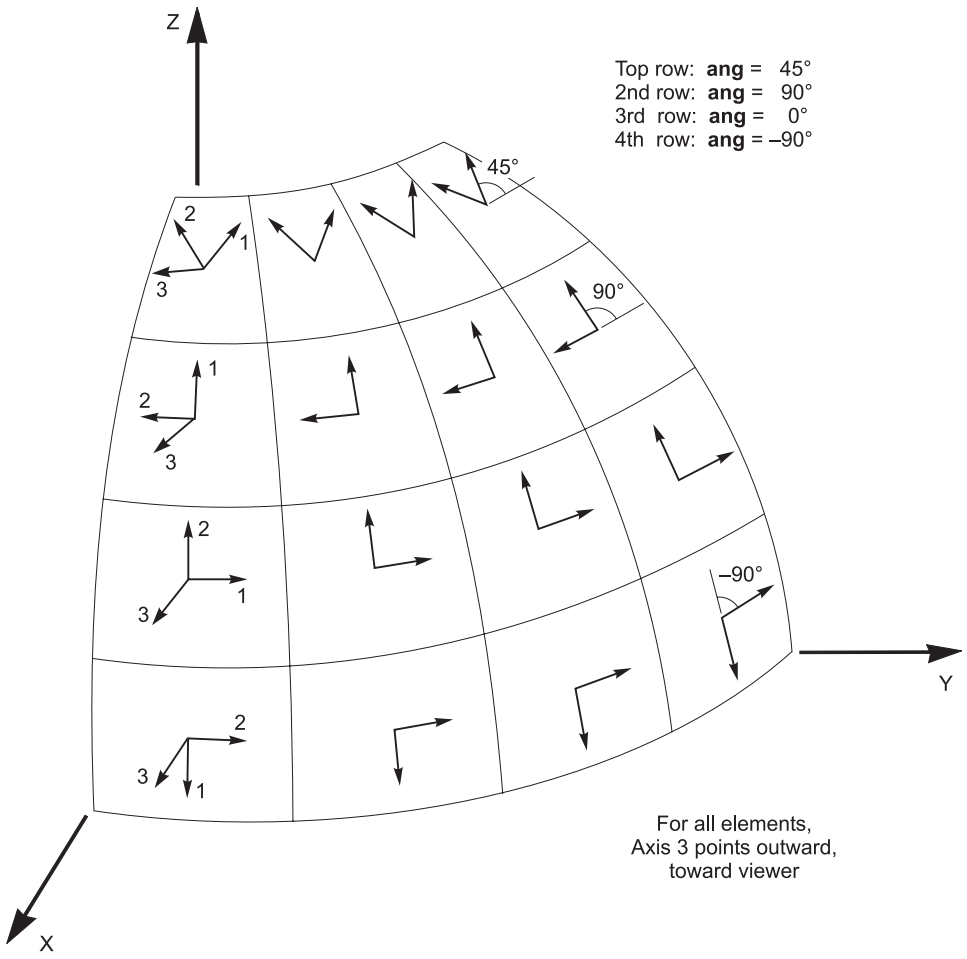


Figure 50
The Shell Element Coordinate Angle with Respect to the Default Orientation

mined by the reference vector. The local 3 axis is always normal to the plane of the element.

For more information:

- See Chapter “Coordinate Systems” (page 21) for a description of the concepts and terminology used in this topic.
- See Topic “Local Coordinate System” (page 186) in this chapter.

Reference Vector

To define the tangential local axes, you specify a **reference vector** that is parallel to the desired 3-1 or 3-2 plane. The reference vector must have a positive projection upon the corresponding tangential local axis (1 or 2, respectively). This means that the positive direction of the reference vector must make an angle of less than 90° with the positive direction of the desired tangential axis.

To define the reference vector, you must first specify or use the default values for:

- A primary coordinate direction **pldirp** (the default is +Z)
- A secondary coordinate direction **pldirs** (the default is +Y). Directions **pldirs** and **pldirp** should not be parallel to each other unless you are sure that they are not parallel to local axis 3
- A fixed coordinate system **csys** (the default is zero, indicating the global coordinate system)
- The local plane, **local**, to be determined by the reference vector (the default is 32, indicating plane 3-2)

You may optionally specify:

- A pair of joints, **plveca** and **plvecb** (the default for each is zero, indicating the center of the element). If both are zero, this option is not used

For each element, the reference vector is determined as follows:

1. A vector is found from joint **plveca** to joint **plvecb**. If this vector is of finite length and is not parallel to local axis 3, it is used as the reference vector $\mathbf{V}_p$
2. Otherwise, the primary coordinate direction **pldirp** is evaluated at the center of the element in fixed coordinate system **csys**. If this direction is not parallel to local axis 3, it is used as the reference vector $\mathbf{V}_p$
3. Otherwise, the secondary coordinate direction **pldirs** is evaluated at the center of the element in fixed coordinate system **csys**. If this direction is not parallel to local axis 3, it is used as the reference vector $\mathbf{V}_p$
4. Otherwise, the method fails and the analysis terminates. This will never happen if **pldirp** is not parallel to **pldirs**

A vector is considered to be parallel to local axis 3 if the sine of the angle between them is less than 10^{-3}.

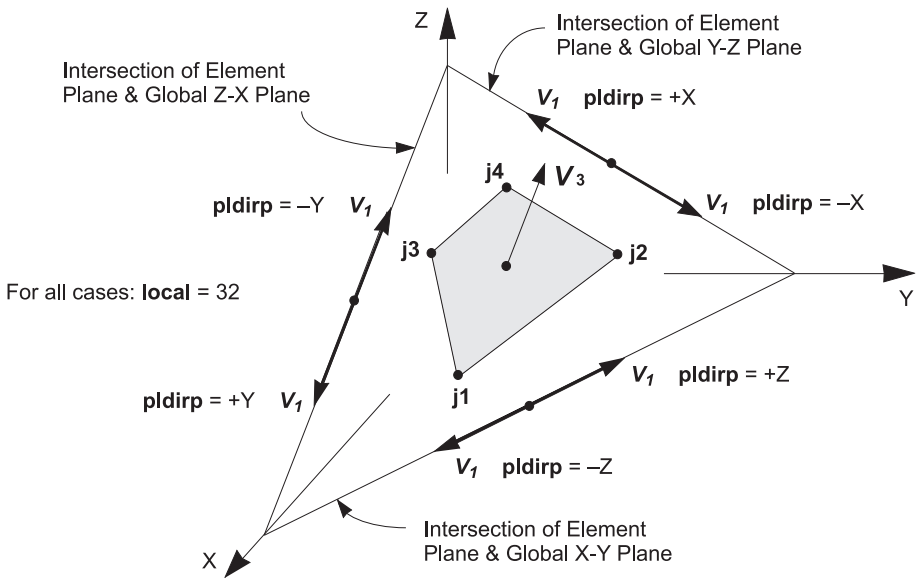


Figure 51
Shell Element Local Coordinate System Using Coordinate Directions

The use of the coordinate direction method is illustrated in Figure 51 (page 190) for the case where **local** = 32.

A special option is available for backward compatibility with previous versions of the program. If **pldirp** is set to zero, the reference vector $\mathbf{V}_p$ is directed from the midpoint of side **j1-j3** to the midpoint of side **j2-j4** (or side **j2-j3** for the triangle). This is illustrated in Figure 48 (page 184), where the reference vector would be identical to local axis 1. With this option, the orientation of the tangential local axes is very dependent upon the mesh used.

Determining Tangential Axes 1 and 2

The program uses vector cross products to determine the tangential axes 1 and 2 once the reference vector has been specified. The three axes are represented by the three unit vectors $\mathbf{V}_1$, $\mathbf{V}_2$ and $\mathbf{V}_3$, respectively. The vectors satisfy the cross-product relationship:

$$\mathbf{V}_1 = \mathbf{V}_2 \times \mathbf{V}_3$$

The tangential axes 1 and 2 are defined as follows:

- If the reference vector is parallel to the 3-1 plane, then:

$$\mathbf{V}_2 = \mathbf{V}_3 \times \mathbf{V}_p \quad \text{and}$$
$$\mathbf{V}_1 = \mathbf{V}_2 \times \mathbf{V}_3$$

- If the reference vector is parallel to the 3-2 plane, then:

$$\mathbf{V}_1 = \mathbf{V}_p \times \mathbf{V}_3 \quad \text{and}$$
$$\mathbf{V}_2 = \mathbf{V}_3 \times \mathbf{V}_1$$

In the common case where the reference vector is parallel to the plane of the element, the tangential axis in the selected local plane will be equal to $\mathbf{V}_p$.

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 referenced during the definition of the elements.

Section Type

The Section type, specified by the parameter **type**, determines the type of behavior modeled by the corresponding Shell elements:

- **type** = MEMBR: Pure membrane behavior; only the in-plane forces and the normal (drilling) moment can be supported
- **type** = PLATE: Pure plate behavior; only the bending moments and the transverse force can be supported
- **type** = 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

The thickness formulation, specified by the parameter **thicktype**, determines whether or not transverse shearing deformations are included in the plate-bending behavior of a plate or shell element:

- **thicktype** = THICK: A thick-plate (Mindlin/Reissner) formulation is used which includes the effects of transverse shear deformation
- **thicktype** = THIN: A thin-plate (Kirchhoff) formulation is used that 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 default 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. Orthotropic properties are used, even if the Material selected was defined as anisotropic. The material properties used by the Shell Section are:

- The moduli of elasticity, **e1**, **e2**, and **e3**
- The shear modulus, **g12**, **g13**, and **g23**
- The Poisson's ratios, **u12**, **u13**, and **u23**
- The coefficients of thermal expansion, **a1** and **a2**
- The mass density, **m**, for computing element mass
- The weight density, **w**, for computing Self-Weight and Gravity Loads

The properties **e3**, **u13**, and **u23** are condensed out of the matrial matrix by assuming a state of plane stress in the element. The resulting, modified values of **e1**, **e2**, **g12**, and **u12** are used to compute the membrane and plate-bending stiffnesses.

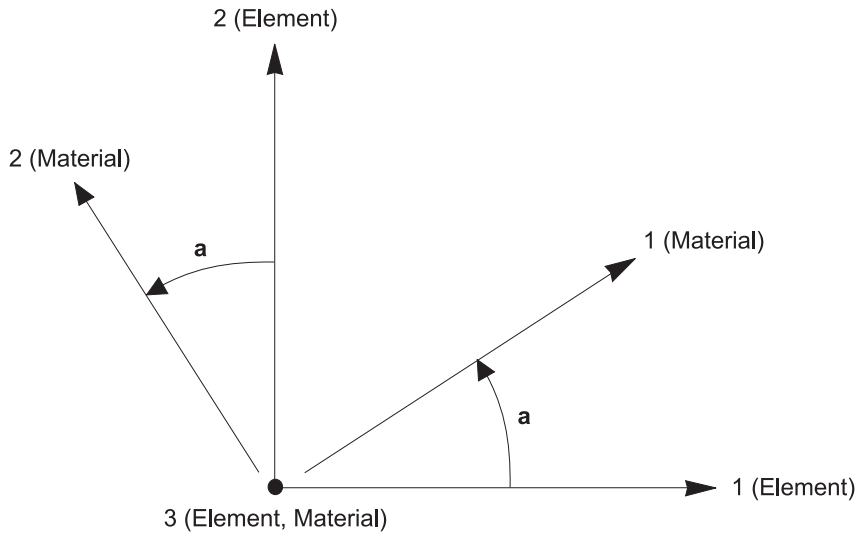


Figure 52
Shell Element Material Angle

The shear moduli, **g13** and **g23**, are used to compute the transverse shearing stiffness if the thick-plate formulation is used. The coefficients of thermal expansion, **a1** and **a2**, are used for membrane expansion and thermal bending strain.

All material properties (except the densities) are obtained at the material temperature of each individual element.

See Chapter “Material Properties” (page 135) for more information.

Material Angle

The material local coordinate system and the element (Shell Section) local coordinate system need not be the same. The local 3 directions always coincide for the two systems, but the material 1 axis and the element 1 axis may differ by the angle **a** as shown in Figure 52 (page 193). This angle has no effect for isotropic material properties since they are independent of orientation.

See Topic “Local Coordinate System” (page 136) in Chapter “Material Properties” for more information.

Thickness

Each 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 and transverse-shearing stiffnesses for full-shell and pure-plate Sections

Normally these two thicknesses are the same and you only need to specify **th**. 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 Topic “Mass Density” (page 142) in Chapter “Material Properties”.
- See Subtopic “Thickness” (page 194) in this chapter for the definition of **th**.
- See Chapter “Static and Dynamic Analysis” (page 291).

Self-Weight Load

Self-Weight Load activates 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 Load always acts downward, in the global –Z direction. You may scale the self-weight by a single scale factor that applies equally to all elements in the structure.

For more information:

- See Topic “Weight Density” (page 143) in Chapter “Material Properties” for the definition of **w**.
- See Subtopic “Thickness” (page 194) in this chapter for the definition of **th**.
- See Topic “Self-Weight Load” (page 283) in Chapter “Load Cases.”

Gravity Load

Gravity Load can be applied to each Shell element to activate the self-weight of the element. Using Gravity Load, the self-weight can be scaled and applied in any direction. Different scale factors and directions can be applied to each element.

If all elements are to be loaded equally and in the downward direction, it is more convenient to use Self-Weight Load.

For more information:

- See Topic “Self-Weight Load” (page 195) in this chapter for the definition of self-weight for the Shell element.
- See Topic “Gravity Load” (page 284) in Chapter “Load Cases.”

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 a fixed coordinate system (global or Alternate Coordinates) or in the element local coordinate system.

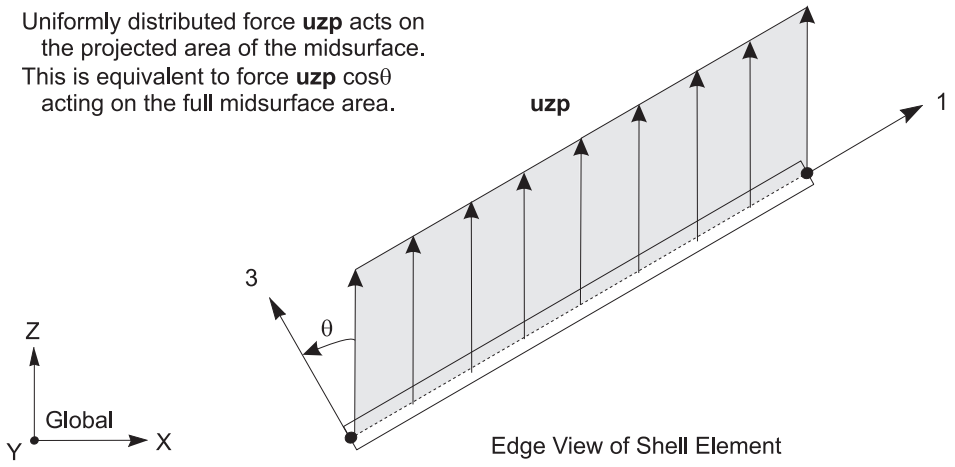


Figure 53
Example of Uniform Load Acting on the Projected Area of the Midsurface

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.

Forces given in fixed coordinates can optionally be specified to act on the projected area of the midsurface, i.e., the area that can be seen along the direction of loading. The specified load intensity is automatically multiplied by the cosine of the angle between the direction of loading and the normal to the element (the local 3 direction). This can be used, for example, to apply distributed snow or wind loads. See Figure 53 (page 196).

See Chapter “Load Cases” (page 279) for more information.

Surface Pressure Load

The Surface Pressure Load is used to apply external pressure loads upon any of the six faces of the Shell element. The definition of these faces is shown in Figure 48 (page 184). Surface pressure always acts normal to the face. Positive pressures are directed toward the interior of the element.

The pressure may be constant over a face or interpolated from values given at the joints. The values given at the joints are obtained from Joint Patterns, and need not be the same for the different faces. Joint Patterns can be used to easily apply hydrostatic pressures.

The bottom and top faces are denoted Faces 5 and 6, respectively. The top face is the one visible when the +3 axis is directed toward you and the path **j1**-**j2**-**j3** appears counterclockwise. The pressure acting on the bottom or top face is integrated over the plane of the element and apportioned to the corner joints..

The sides of the element are denoted Faces 1 to 4 (1 to 3 for the triangle), counting counterclockwise from side **j1**-**j2** when viewed from the top. The pressure acting on a side is multiplied by the thickness, **th**, integrated along the length of the side, and apportioned to the two joints on that side.

For more information:

- See Topic “Thickness” (page 194) in this chapter for the definition of **th**.
- See Chapter “Load Cases” (page 279).

Temperature Load

The Temperature Load creates thermal strain in the Shell element. This strain is given by the product of the Material coefficient of thermal expansion and the temperature change of the element. The temperature change is measured from the element Reference Temperature to the element Load Temperature.

Two independent Load Temperature fields may be specified:

- Temperature, **t**, which is constant through the thickness and produces membrane strains
- Temperature gradient, **t3**, which is linear in the thickness direction and produces bending strains

The temperature gradient is specified as the *change* in temperature per unit length. The temperature gradient is positive if the temperature increases (linearly) in the positive direction of the element local 3 axis. The gradient temperature is zero at the midsurface, hence no membrane strain is induced.

Each of the two Load Temperature fields may be constant over the plane of the element or interpolated from values given at the joints.

The Reference Temperature gradient is always taken to be zero, hence the temperature change that produces the bending strain is equal to the Load Temperature gradient.

See Chapter “Load Cases” (page 279) for more information.

Internal Force and Stress Output

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:

$$F_{11} = \int_{-\mathbf{th}/2}^{+\mathbf{th}/2} \sigma_{11}\, dx_3 \qquad \text{(Eqns. 1)}$$

$$F_{22} = \int_{-\mathbf{th}/2}^{+\mathbf{th}/2} \sigma_{22}\, dx_3$$

- Membrane shear force:

$$F_{12} = \int_{-\mathbf{th}/2}^{+\mathbf{th}/2} \sigma_{12}\, dx_3$$

- Plate bending moments:

$$M_{11} = -\int_{-\mathbf{thb}/2}^{+\mathbf{thb}/2} t\, \sigma_{11}\, dx_3$$

$$M_{22} = -\int_{-\mathbf{thb}/2}^{+\mathbf{thb}/2} t\, \sigma_{22}\, dx_3$$

- Plate twisting moment:

$$M_{12} = -\int_{-\mathbf{thb}/2}^{+\mathbf{thb}/2} t\, \sigma_{12}\, dx_3$$

- Plate transverse shear forces:

$$V_{13} = \int_{-\mathbf{thb}/2}^{+\mathbf{thb}/2} \sigma_{13}\, dx_3$$

$$V_{23} = \int_{-\mathbf{thb}/2}^{+\mathbf{thb}/2} \sigma_{23}\, dx_3$$

where x_3 represents the thickness coordinate measured from the midsurface of the element.

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 transverse shear forces are computed from the moments using the equilibrium equations:

$$V_{13} = -\frac{dM_{11}}{dx_1} - \frac{dM_{12}}{dx_2}$$

$$V_{23} = -\frac{dM_{12}}{dx_1} - \frac{dM_{22}}{dx_2}$$

where x_1 and x_2 are in-plane coordinates parallel to the local 1 and 2 axes.

The sign conventions for the stresses and internal forces are illustrated in Figure 54 (page 200). Stresses acting on a positive face are oriented in the positive direction of 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. Thus:

(Eqns. 2)

$$\sigma_{11} = \frac{F_{11}}{\mathbf{th}} - \frac{12\,M_{11}}{\mathbf{thb}^3}\,x_3$$

$$\sigma_{22} = \frac{F_{22}}{\mathbf{th}} - \frac{12\,M_{22}}{\mathbf{thb}^3}\,x_3$$

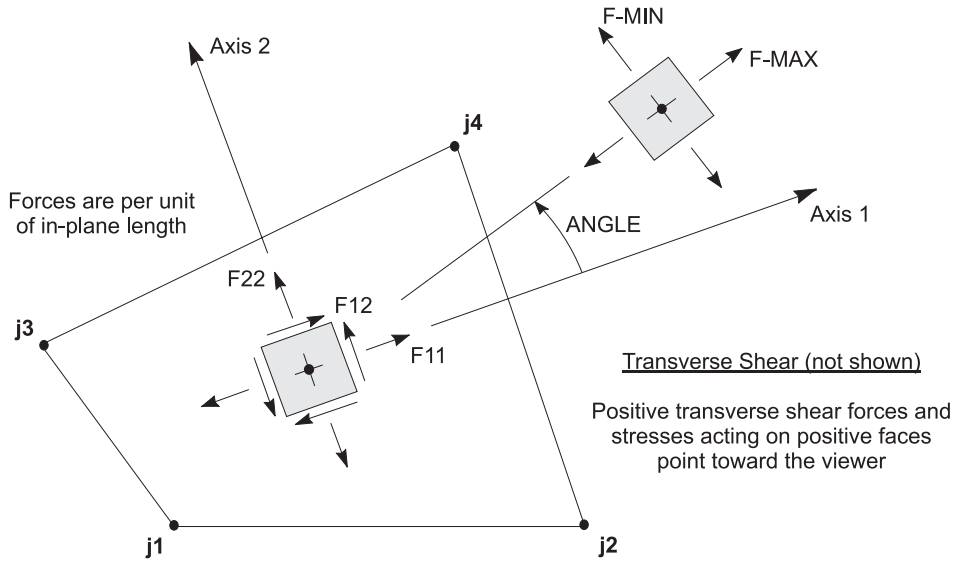
$$\sigma_{12} = \frac{F_{12}}{\mathbf{th}} - \frac{12\,M_{12}}{\mathbf{thb}^3}\,x_3$$

$$\sigma_{13} = \frac{V_{13}}{\mathbf{thb}}$$

$$\sigma_{23} = \frac{V_{23}}{\mathbf{thb}}$$

$$\sigma_{33} = 0$$

The transverse shear stresses given here are average values. 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.



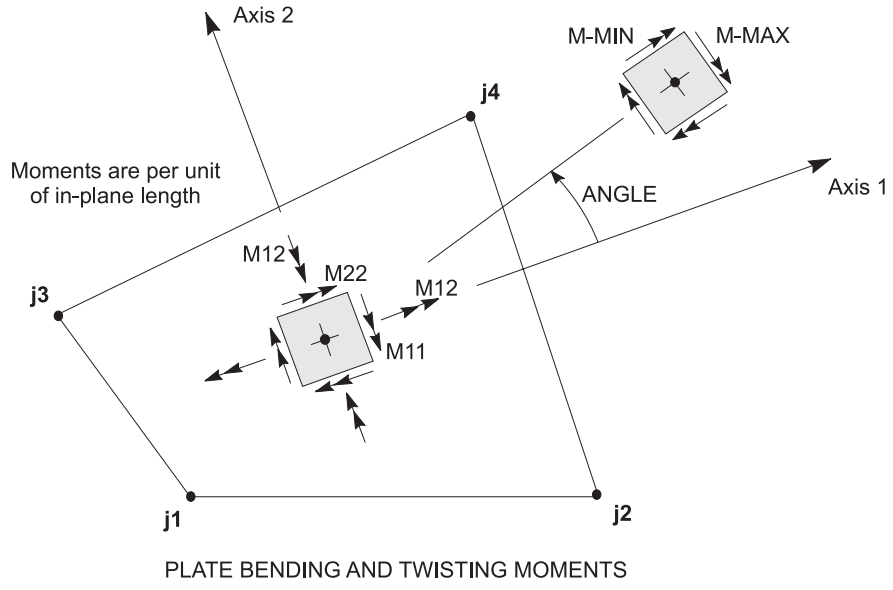


Figure 54
Shell Element Stresses and Internal Forces

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 the following types of Analysis Cases: Loads, Modes, Specs, Histories, and Combos. History results are the maximum and minimum values attained during the analysis, and are only available if specifically requested in the definition of the individual History cases. No results are available for Moving Loads or Combos that contain Moving Loads.

Principal values and the associated principal directions are also computed for the Loads and Modes. The angle given is measured counterclockwise (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.

Shell element stresses and internal forces are printed in the output file under the heading:

```
SHELL ELEMENT INTERNAL FORCES & STRESSES
```

All results for a single element are printed together. Elements are presented in alpha-numeric order. For each Analysis Case the internal forces are printed at the element joints, followed by the stresses at the joints.

The internal forces and stresses can be interpolated over the whole element from the values printed at the joints.

The internal forces are labeled F11, F22, F12, M11, M22, M12, V13, and V23, corresponding to the terms defined in Equations 1 above. The stresses are labeled S11, S22, S12, S13, and S23, corresponding to the terms defined in Equations 2 above. Stress values at the top and bottom surface are indicated.

Typical Shell element output is shown in Figure 55 (page 202).

By default, no internal forces or stresses are printed. You may specify which Analysis Cases are to be printed. The internal forces and stresses can always be displayed using the SAP2000 graphical interface even if no results have been selected for printing.

For more information:

- See Topic “Stresses and Strains” (page 137) in Chapter “Material Properties.”

```
S H E L L   E L E M E N T   I N T E R N A L   F O R C E S   &   S T R E S S E S

ELEM       1 ================== TYPE = SHELL

LOAD       1 ------------------

  JOINT         F11         F22         F12       F-MAX       F-MIN       ANGLE
     10   -1.63E-11   -1.22E-13    1.38E-13   -1.20E-13   -1.63E-11   89.511243
     31   -1.62E-11    1.53E-13    1.38E-13    1.54E-13   -1.62E-11   89.517398
     25   -1.63E-11   -1.22E-13   -1.38E-13   -1.21E-13   -1.63E-11  -89.511244
     32   -1.62E-11    1.53E-13   -1.38E-13    1.54E-13   -1.62E-11  -89.517399

  JOINT         M11         M22         M12       M-MAX       M-MIN       ANGLE
     10   -8000.000   10.370186 -117.222361   12.085230   -8001.715  -89.161783
     31  -10000.000   96.481989 -117.222361   97.842783  -10001.361  -89.334903
     25   -8000.010   10.370617  117.221187   12.085625   -8001.725   89.161792
     32   -9999.990   96.481558  117.221187   97.842326  -10001.351   89.334909

  JOINT         V13         V23                   V-MAX                   ANGLE
     10 -147.129704    3.59E-05              147.129704              179.999986
     31 -147.129704   -3.59E-05              147.129704             -179.999986
     25 -147.128035    3.59E-05              147.128035              179.999986
     32 -147.128035   -3.59E-05              147.128035             -179.999986

  JOINT      S11-TOP     S22-TOP     S12-TOP   S-TOP-MAX   S-TOP-MIN       ANGLE
     10   333.333333   -0.432091    4.884265  333.404794   -0.503551    0.838217
     31   416.666667   -4.020083    4.884265  416.723366   -4.076783    0.665097
     25   333.333751   -0.432109   -4.884216  333.405209   -0.503568   -0.838208
     32   416.666249   -4.020065   -4.884216  416.722948   -4.076764   -0.665091

  JOINT      S11-BOT     S22-BOT     S12-BOT   S-BOT-MAX   S-BOT-MIN       ANGLE
     10 -333.333333    0.432091   -4.884265    0.503551 -333.404794  -89.161783
     31 -416.666667    4.020083   -4.884265    4.076783 -416.723366  -89.334903
     25 -333.333751    0.432109    4.884216    0.503568 -333.405209   89.161792
     32 -416.666249    4.020065    4.884216    4.076764 -416.722948   89.334909

  JOINT      S13-AVG     S23-AVG               S-AVG-MAX                   ANGLE
     10   -12.260809    2.99E-06              12.260809              179.999986
     31   -12.260809   -2.99E-06              12.260809             -179.999986
     25   -12.260670    2.99E-06              12.260670              179.999986
     32   -12.260670   -2.99E-06              12.260670             -179.999986

ELEM       3 ================== TYPE = MEMBRANE

LOAD       1 ------------------

  JOINT         F11         F22         F12       F-MAX       F-MIN       ANGLE
      4    1824.191   279.849372  -79.513613    1828.274   275.766246   -2.939630
      5    1790.814   146.341527  -43.335783    1791.955   145.200317   -1.508485
      9  -46.975988 -187.937212 -104.469157    8.564515 -243.477715  -27.997121
     10  -80.351616 -321.443470  -68.289377  -62.352389 -339.442697  -14.765796

  JOINT      S11-TOP     S22-TOP     S12-TOP   S-TOP-MAX   S-TOP-MIN       ANGLE
      4   304.031827    46.641562  -13.252269  304.712348   45.961041   -2.939630
      5   298.469000    24.390255   -7.222631  298.659202   24.200053   -1.508485
      9    -7.829331   -31.322869  -17.411526    1.427419  -40.579619  -27.997121
     10   -13.391936   -53.573912  -11.381563  -10.392065  -56.573783  -14.765796
```

...Output Edited for Brevity...

Figure 55

Shell Element Internal Forces and Stresses in the Output File

- See Subtopic “Thickness” (page 194) in this chapter for the definition of **th** and **thb**.
- See Chapter “Load Cases” (page 279).
- See Chapter “Static and Dynamic Analysis” (page 291).
- See Chapter “The Output Files” (page 395).

Joint Force Output

The **Shell element joint forces** are concentrated forces and moments acting at the joints of the element that represent the effect of the rest of the structure upon the element.

Element joint forces must not be confused with internal forces and moments which, like stresses, act *throughout* the element, not just at the joints. Furthermore, the internal forces are defined on a per-unit-of-length basis. Element joints forces are concentrated values of force and moment.

See Topic “Element Joint Force Output” (page 402) in Chapter “The Output Files” for more information.

Chapter XI

The Plane Element

The **Plane** element is used to model plane-stress and plane-strain behavior in two-dimensional solids.

Advanced Topics

- Overview
- Joint Connectivity
- Degrees of Freedom
- Local Coordinate System
- Stresses and Strains
- Material Properties
- Material Angle
- Thickness
- Mass
- Self-Weight Load
- Gravity Load
- Surface Pressure Load
- Pore Pressure Load

- Temperature Load
- Stress Output
- Joint Force Output

Overview

The Plane element is a three- to nine-node element for modeling two-dimensional solids of uniform thickness. It is based upon an isoparametric formulation (Hollings and Wilson, 1977). The element must be planar and must always exist parallel to one of the global principal planes (i.e., parallel to the X-Y, Y-Z, or Z-X planes).

Structures that can be modeled with this element include:

- Thin, planar structures in a state of plane stress
- Long, prismatic structures in a state of plane strain

The stresses and strains are assumed not to vary in the thickness direction.

Each Plane element has its own local coordinate system that is aligned with the global system. The local coordinate system is used for defining Material properties and loads, and for interpreting output.

Temperature-dependent, orthotropic material properties are allowed. Each element may be loaded by gravity (in any direction); surface pressure on the side faces; pore pressure within the element; and loads due to temperature change.

An eight-point numerical integration scheme is used for the Plane. Stresses in the element local coordinate system are evaluated at the integration points and extrapolated to the joints of the element. An approximate error in the stresses 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 the finite element approximation and can then be used as the basis for the selection of a new and more accurate finite element mesh.

The nine-node option is the recommended formulation.

Joint Connectivity

Each Plane element may have either of the following shapes, as shown in Figure 56 (page 207):

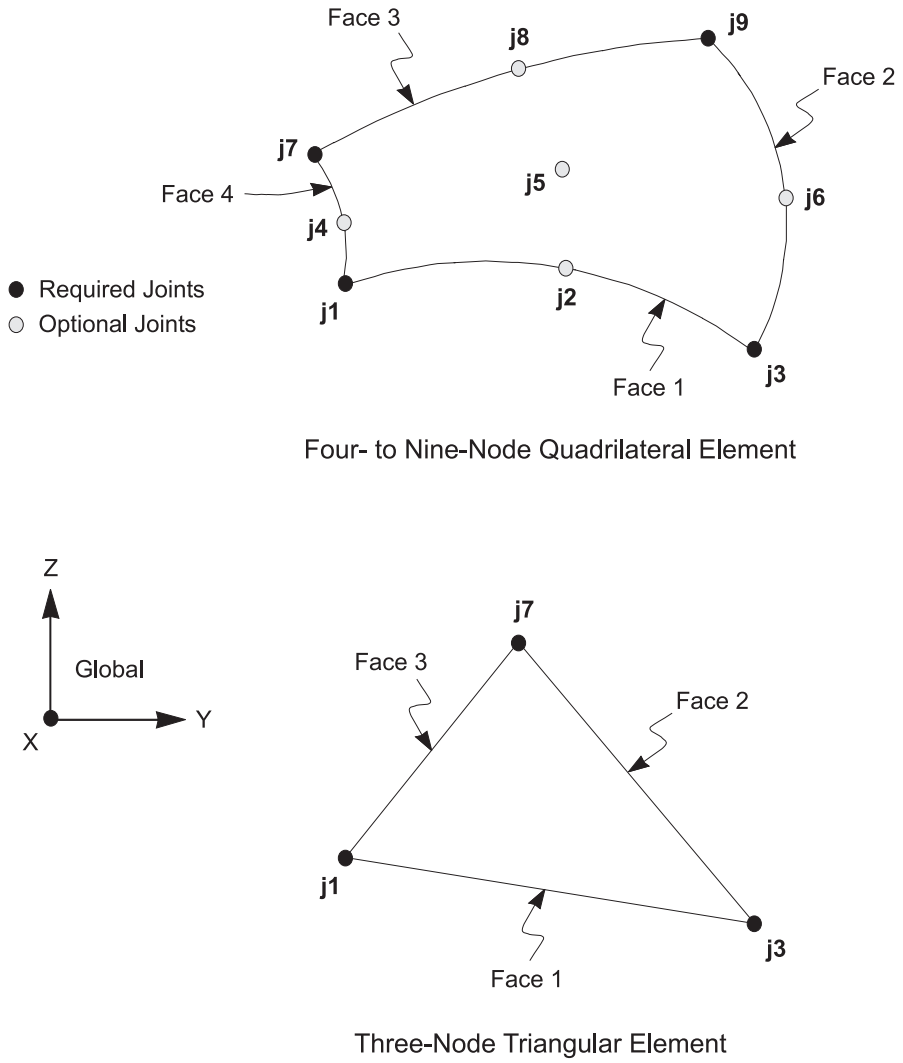


Figure 56
Plane Element Joint Connectivity and Face Definitions

- Quadrilateral, defined by the four corner joints **j1**, **j3**, **j7**, and **j9**, plus the optional midside joints **j2**, **j4**, **j6**, and **j8**, and the optional center joint **j5**.
- Triangular, defined by the three joints **j1**, **j3**, and **j7** only.

All joints for a given element must lie in a single plane that is parallel to one of the principal global planes: X-Y, Y-Z, or Z-X. The path **j1**-**j3**-**j7** *must* appear counterclockwise when viewed from the positive global coordinate direction that is perpendicular to the plane of the element, i.e., when viewed from +Z, +X, or +Y, respectively.

For the best accuracy, the use of the nine-node quadrilateral is recommended. However, the variable node option of the element is very useful for generating transition interfaces between coarse and fine finite element meshes. See Figure 57 (page 209) for examples.

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.
- If a midside joint is present, it should be near the center of the side, or at least between the quarter points. Best results are obtained if the side is straight. If the side is curved, the distance from the midside joint to the straight line between the corner joints should be no larger than half the distance between the corner joints.
- The position of the center joint with respect to each pair of opposite midside joints is subject to the same rules as is the position of a midside joint with respect to the corner joints.

Degrees of Freedom

The Plane element activates the three translational degrees of freedom at each of its connected joints. Rotational degrees of freedom are not activated.

This element contributes stiffness only to the degrees of freedom in the plane of the element. It is necessary to provide restraints or other supports for the translational degrees of freedom that are normal to this plane; otherwise, the structure will be unstable.

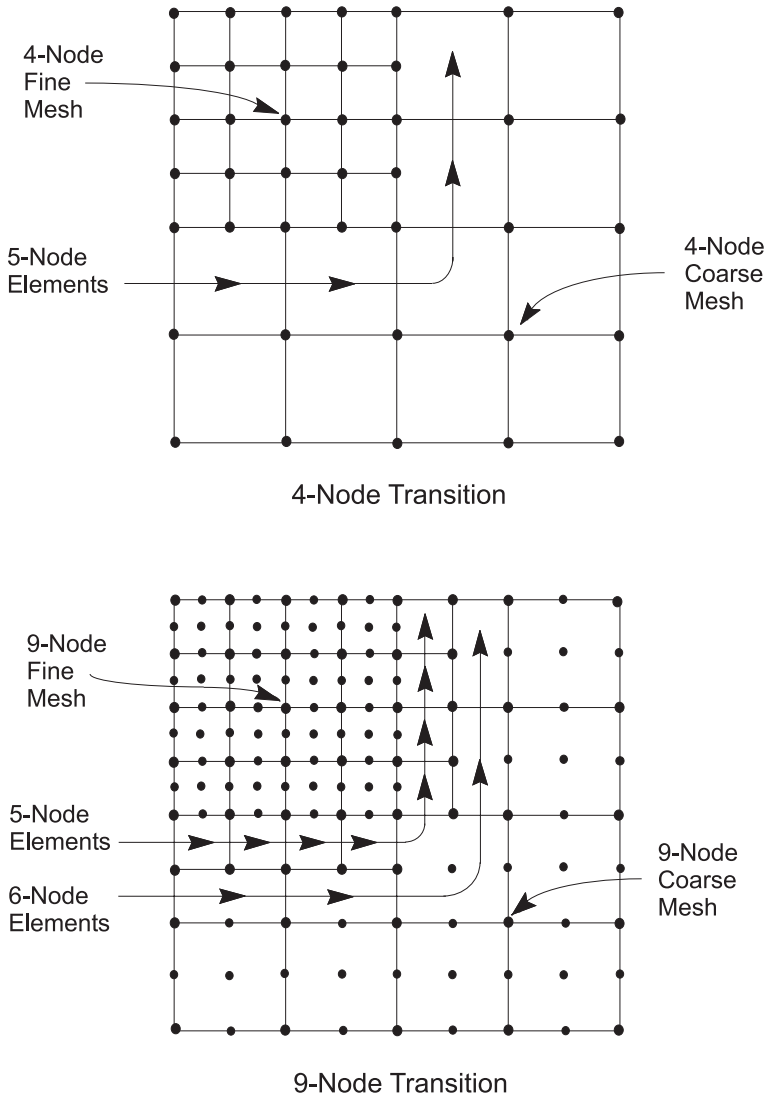


Figure 57
Examples of Plane Element Mesh Transitions

See Topic “Degrees of Freedom” (page 83) in Chapter “Joints and Degrees of Freedom” for more information.

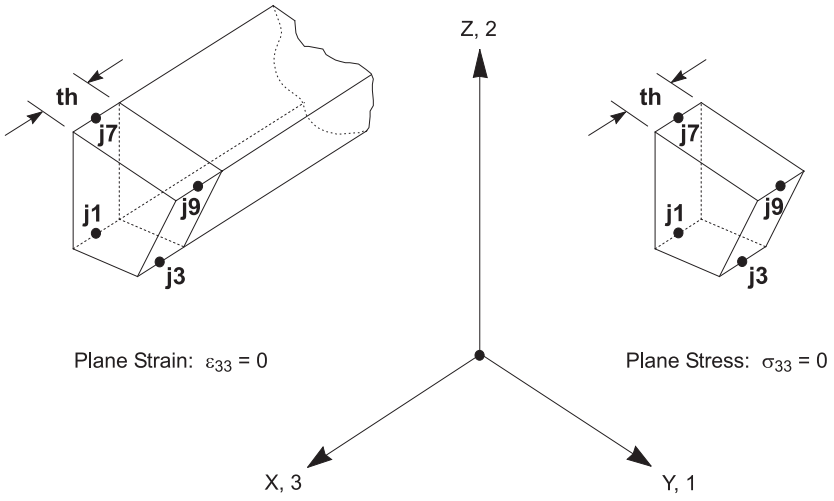


Figure 58
Plane Element Local Coordinate System and Thickness Definition

Local Coordinate System

Each Plane 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. These axes are always parallel to the axes of the global coordinate system, with the 3 direction being perpendicular to the plane of the element. The local axes all have the same positive sense as do the corresponding global axes. Thus:

- For an element parallel to the global X-Y plane, the local 1-2-3 system is the same as the global X-Y-Z system
- For an element parallel to the global Y-Z plane, the local 1-2-3 system is the same as the global Y-Z-X system
- For an element parallel to the global Z-X plane, the local 1-2-3 system is the same as the global Z-X-Y system

The local coordinate system for elements parallel to the Y-Z plane is shown in Figure 58 (page 210).

Stresses and Strains

The Plane element models the mid-plane of a structure having uniform thickness, and whose stresses and strains do not vary in the thickness direction.

Transverse shear stresses (σ_{12}, σ_{13}) and shear strains (γ_{12}, γ_{13}) are assumed to be zero. Displacements in the thickness (local 3) direction have no effect on the element.

Plane-stress is appropriate for structures that are *thin* compared to their planar dimensions. The thickness normal stress (σ_{33}) is assumed to be zero. The thickness normal strain (ε_{33}) may not be zero due to Poisson effects.

Plane-strain is appropriate for structures that are *thick* compared to their planar dimensions. The thickness normal strain (ε_{33}) is assumed to be zero. The thickness normal stress (σ_{33}) may not be zero due to Poisson effects.

See Topic “Stresses and Strains” (page 137) in Chapter “Material Properties” for more information.

Material Properties

The material properties for each Plane element are specified by reference to a previously-defined Material. Orthotropic properties are used, even if the Material selected was defined as anisotropic. The material properties used by the Plane element are:

- The moduli of elasticity, **e1**, **e2**, and **e3**
- The shear modulus, **g12**
- The Poisson’s ratios, **u12**, **u13** and **u23**
- The coefficients of thermal expansion, **a1**, **a2**, and **a3**
- The mass density, **m**, for computing element mass
- The weight density, **w**, for computing Self-Weight and Gravity Loads

The properties **e3**, **u13**, **u23**, and **a3** are not used for plane stress. They are used to compute the thickness-normal stress (σ_{33}) in plane strain.

All material properties (except the densities) are obtained at the material temperature of each individual element.

See Chapter “Material Properties” (page 135) for more information.

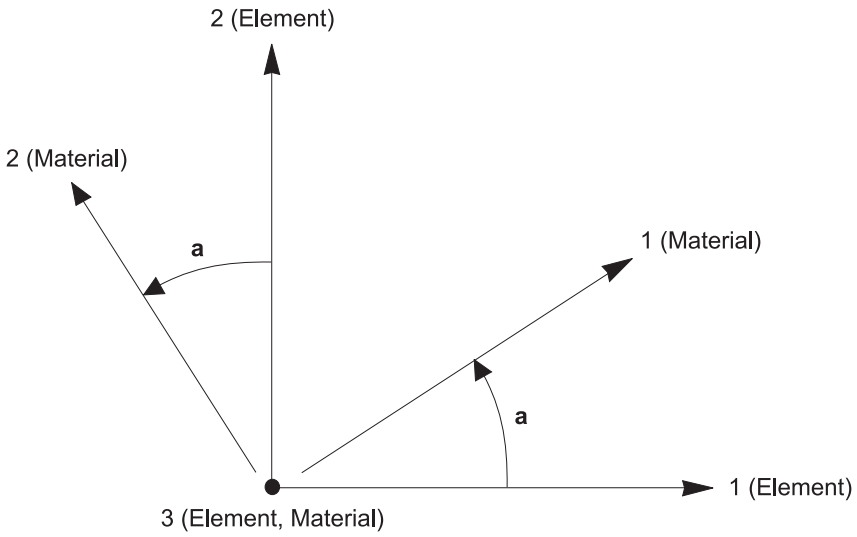


Figure 59
Plane Element Material Angle

Material Angle

The material local coordinate system and the element local coordinate system need not be the same. The local 3 directions always coincide for the two systems, but the material 1 axis and the element 1 axis may differ by the angle **a** as shown in Figure 59 (page 212). This angle has no effect for isotropic material properties since they are independent of orientation.

See Topic "Local Coordinate System" (page 136) in Chapter "Material Properties" for more information.

Thickness

Each element has a uniform thickness, **th**. This may be the actual thickness, particularly for plane-stress elements; or it may be a representative portion, such as a unit thickness of an infinitely-thick plane-strain element. See Figure 58 (page 210).

The element thickness is used for calculating the element stiffness, mass, and loads. Hence, joint forces computed from the element are proportional to this thickness.

Mass

In a dynamic analysis, the mass of the structure is used to compute inertial forces. The mass contributed by the Plane 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) even though the element contributes stiffness to only two of these degrees of freedom.

For more information:

- See Topic “Mass Density” (page 142) in Chapter “Material Properties.”
- See Chapter “Static and Dynamic Analysis” (page 291).

Self-Weight Load

Self-Weight Load activates the self-weight of all elements in the model. For a Plane 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 Load always acts downward, in the global –Z direction. You may scale the self-weight by a single scale factor that applies equally to all elements in the structure.

For more information:

- See Topic “Weight Density” (page 143) in Chapter “Material Properties” for the definition of **w**.
- See Topic “Thickness” (page 212) in this chapter for the definition of **th**.
- See Topic “Self-Weight Load” (page 283) in Chapter “Load Cases.”

Gravity Load

Gravity Load can be applied to each Plane element to activate the self-weight of the element. Using Gravity Load, the self-weight can be scaled and applied in any direction. Different scale factors and directions can be applied to each element.

If all elements are to be loaded equally and in the downward direction, it is more convenient to use Self-Weight Load.

For more information:

- See Topic “Self-Weight Load” (page 213) in this chapter for the definition of self-weight for the Plane element.
- See Topic “Gravity Load” (page 284) in Chapter “Load Cases.”

Surface Pressure Load

The Surface Pressure Load is used to apply external pressure loads upon any of the three or four side faces of the Plane element. The definition of these faces is shown in Figure 56 (page 207). Surface pressure always acts normal to the face. Positive pressures are directed toward the interior of the element.

The pressure may be constant over a face or interpolated from values given at the joints. The values given at the joints are obtained from Joint Patterns, and need not be the same for the different faces. Joint Patterns can be used to easily apply hydrostatic pressures.

The pressure acting on a side is multiplied by the thickness, **th**, integrated along the length of the side, and apportioned to the two or three joints on that side.

See Chapter “Load Cases” (page 279) for more information.

Pore Pressure Load

The Pore Pressure Load is used to model the drag and buoyancy effects of a fluid *within* a solid medium, such as the effect of water upon the solid skeleton of a soil.

Scalar fluid-pressure values are given at the element joints by Joint Patterns, and interpolated over the element. The total force acting on the element is the integral of the gradient of this pressure field over the plane of the element, multiplied by the thickness, **th**. This force is apportioned to each of the joints of the element. The

forces are typically directed from regions of high pressure toward regions of low pressure.

See Chapter “Load Cases” (page 279) for more information.

Temperature Load

The Temperature Load creates thermal strain in the Plane element. This strain is given by the product of the Material coefficient of thermal expansion and the temperature change of the element. The temperature change is measured from the element Reference Temperature to the element Load Temperature. Temperature changes are assumed to be constant through the element thickness.

See Chapter “Load Cases” (page 279) for more information.

Stress Output

The Plane element stresses are evaluated at the standard 2-by-2 Gauss integration points of the element and extrapolated to the joints. See Cook, Malkus, and Plesha (1989) for more information.

The stresses are computed for the following types of Analysis Cases: Loads, Modes, Specs, Histories, and Combos. History results are the maximum and minimum values attained during the analysis, and are only available if specifically requested in the definition of the individual History cases. No results are available for Moving Loads or Combos that contain Moving Loads.

Principal values and their associated principal directions in the element local 1-2 plane are also computed for the Loads and Modes. The angle given is measured counterclockwise (when viewed from the +3 direction) 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.

Plane element stresses are printed in the output file under the heading:

```
PLANE ELEMENT STRESSES
```

All results for a single element are printed together. Elements are presented in alpha-numeric order. For each Analysis Case the stresses are printed at all element joints.

The stresses are labeled S11, S22, S33, and S12 corresponding to σ_{11}, σ_{22}, σ_{33}, and σ_{12}, respectively, in the element local coordinate system. Stresses σ_{12} and σ_{13} are always zero and are not printed.

Typical Plane element output is shown in Figure 60 (page 217).

By default, no stresses are printed. You may specify which Analysis Cases are to be printed. The stresses can always be displayed using the SAP2000 graphical interface even if no results have been selected for printing.

For more information:

- See Chapter “Load Cases” (page 279).
- See Chapter “Static and Dynamic Analysis” (page 291).
- See Chapter “The Output Files” (page 395).

Joint Force Output

The **Plane element joint forces** are concentrated forces acting at the joints of the element that represent the effect of the rest of the structure upon the element.

See Topic “Element Joint Force Output” (page 402) in Chapter “The Output Files” for more information.

```
P L A N E   E L E M E N T   S T R E S S E S

ELEM       1 ================== TYPE = PLANE STRESS

LOAD       1 ------------------

  JOINT         S11         S22         S33         S12
      1   20.255067    0.465601     .000000  -13.888889
      2   83.333333   -0.409666     .000000  -13.888889
      3  146.411600   -1.284933     .000000  -13.888889
      6    4.09E-12   -4.70E-13     .000000  -13.888889
      7    2.35E-12    2.63E-12     .000000  -13.888889
      8    6.06E-13    5.72E-12     .000000  -13.888889
     11  -20.255067   -0.465601     .000000  -13.888889
     12  -83.333333    0.409666     .000000  -13.888889
     13 -146.411600    1.284933     .000000  -13.888889

  JOINT       S-MAX       S-MIN                ANGLE
      1   27.413397   -6.692729              -27.266570
      2   85.576726   -2.653058               -9.175411
      3  147.706315   -2.579648               -5.325693
      6   13.888889  -13.888889              -45.000000
      7   13.888889  -13.888889              -45.000000
      8   13.888889  -13.888889              -45.000000
     11    6.692729  -27.413397              -62.733430
     12    2.653058  -85.576726              -80.824589
     13    2.579648 -147.706315              -84.674307

ELEM       2 ================== TYPE = PLANE STRESS

LOAD       1 ------------------

  JOINT         S11         S22         S33         S12
     16   20.255067    0.465601     .000000  -13.888889
     17   83.333333   -0.409666     .000000  -13.888889
     18  146.411600   -1.284933     .000000  -13.888889
     21    4.09E-12   -4.70E-13     .000000  -13.888889
     22    2.35E-12    2.63E-12     .000000  -13.888889
     23    6.06E-13    5.72E-12     .000000  -13.888889
     26  -20.255067   -0.465601     .000000  -13.888889
     27  -83.333333    0.409666     .000000  -13.888889
     28 -146.411600    1.284933     .000000  -13.888889

  JOINT       S-MAX       S-MIN                ANGLE
     16   27.413397   -6.692729              -27.266570
     17   85.576726   -2.653058               -9.175411
     18  147.706315   -2.579648               -5.325693
     21   13.888889  -13.888889              -45.000000
     22   13.888889  -13.888889              -45.000000
     23   13.888889  -13.888889              -45.000000
     26    6.692729  -27.413397              -62.733430
     27    2.653058  -85.576726              -80.824589
     28    2.579648 -147.706315              -84.674307
```

...Output Edited for Brevity...

Figure 60
Plane Element Stresses in the Output File

Chapter XII

The Asolid Element

The **Asolid** element is used to model axisymmetric solids under axisymmetric loading.

Advanced Topics

- Overview
- Joint Connectivity
- Degrees of Freedom
- Local Coordinate System
- Stresses and Strains
- Material Properties
- Material Angle
- Arc and Thickness
- Mass
- Self-Weight Load
- Gravity Load
- Surface Pressure Load
- Pore Pressure Load

- Temperature Load
- Rotate Load
- Stress Output
- Joint Force Output

Overview

The Asolid element is a three- to nine-node element for modeling axisymmetric structures under axisymmetric loading. It is based upon an isoparametric formulation (Hollings and Wilson, 1977).

The element models a representative two-dimensional cross section of the three-dimensional axisymmetric solid. The axis of symmetry must be one of the global axes, and the element must exist in one of the global principal planes (in the X-Y, Y-Z, or Z-X planes).

The geometry, loading, displacements, stresses, and strains are assumed not to vary in the circumferential direction. Any displacements that occur in the circumferential direction do not affect the element..

Each Asolid element has its own local coordinate system that is aligned with the global system. The local coordinate system is used for defining Material properties and loads, and for interpreting output.

Temperature-dependent, orthotropic material properties are allowed. Each element may be loaded by gravity (in any direction); centrifugal force; surface pressure on the side faces; pore pressure within the element; and loads due to temperature change.

An eight-point numerical integration scheme is used for the Asolid. Stresses in the element local coordinate system are evaluated at the integration points and extrapolated to the joints of the element. An approximate error in the stresses 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 the finite element approximation and can then be used as the basis for the selection of a new and more accurate finite element mesh.

The nine-node option is the recommended formulation.

Joint Connectivity

All joints for a given element must lie in one of the principal global planes. The axis of symmetry and radial direction are as follows:

Element Plane	Radial Direction Local 1	Axis of Symmetry Local 2	Circumferential Direction Local 3
X-Y	X	Y	Z
Y-Z	Y	Z	X
Z-X	Z	X	Y

The radial coordinate must not be negative for any of the element's joints.

Each Asolid element may have either of the following shapes, as shown in Figure 61 (page 222):

- Quadrilateral, defined by the four corner joints **j1**, **j3**, **j7**, and **j9**, plus the optional midside joints **j2**, **j4**, **j6**, and **j8**, and the optional center joint **j5**.
- Triangular, defined by the three joints **j1**, **j3**, and **j7** only.

The path **j1**-**j3**-**j7** *must* appear counterclockwise when viewed from the positive global coordinate direction that is perpendicular to the plane of the element, i.e., when viewed from +Z, +X, or +Y, respectively, corresponding to the table above.

For the best accuracy, the use of the nine-node quadrilateral is recommended. However, the variable node option of the element is very useful for generating transition interfaces between coarse and fine finite element meshes. See Figure 62 (page 223) for examples.

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

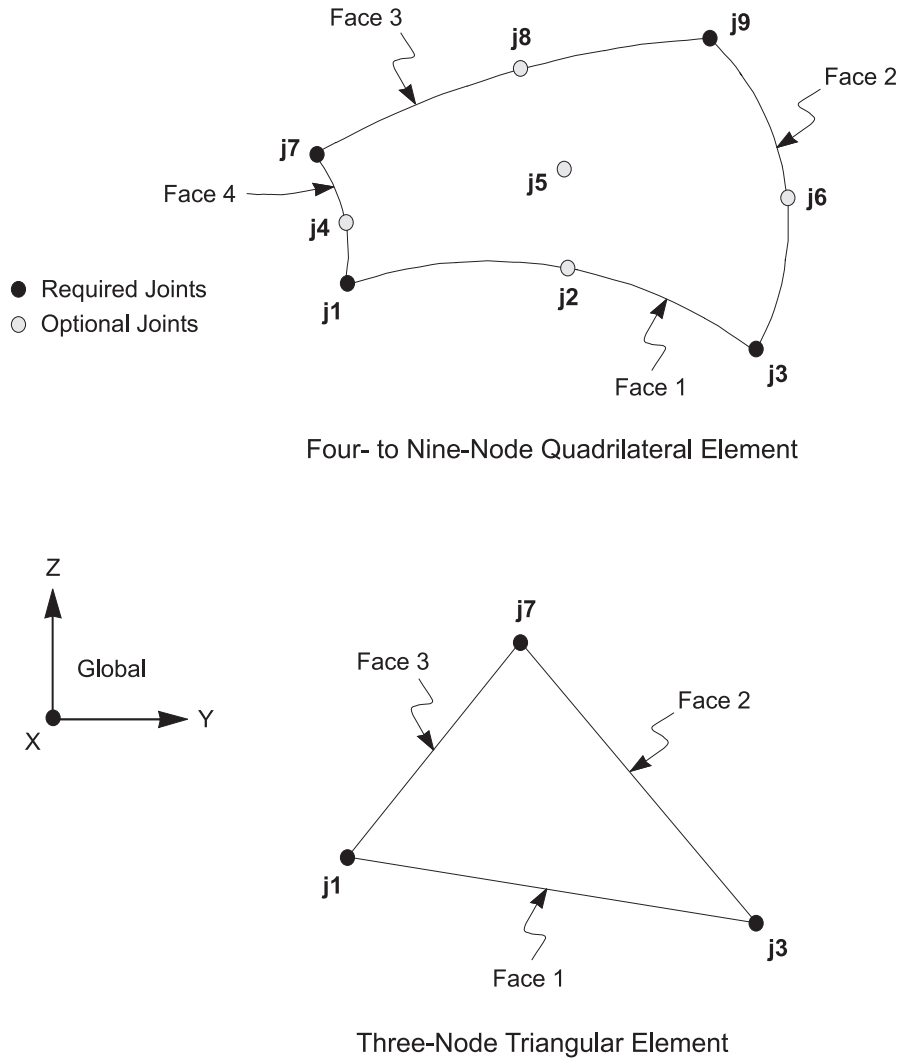


Figure 61
Asolid Element Joint Connectivity and Face Definitions

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.

- If a midside joint is present, it should be near the center of the side, or at least between the quarter points. Best results are obtained if the side is straight. If the side is curved, the distance from the midside joint to the straight line between

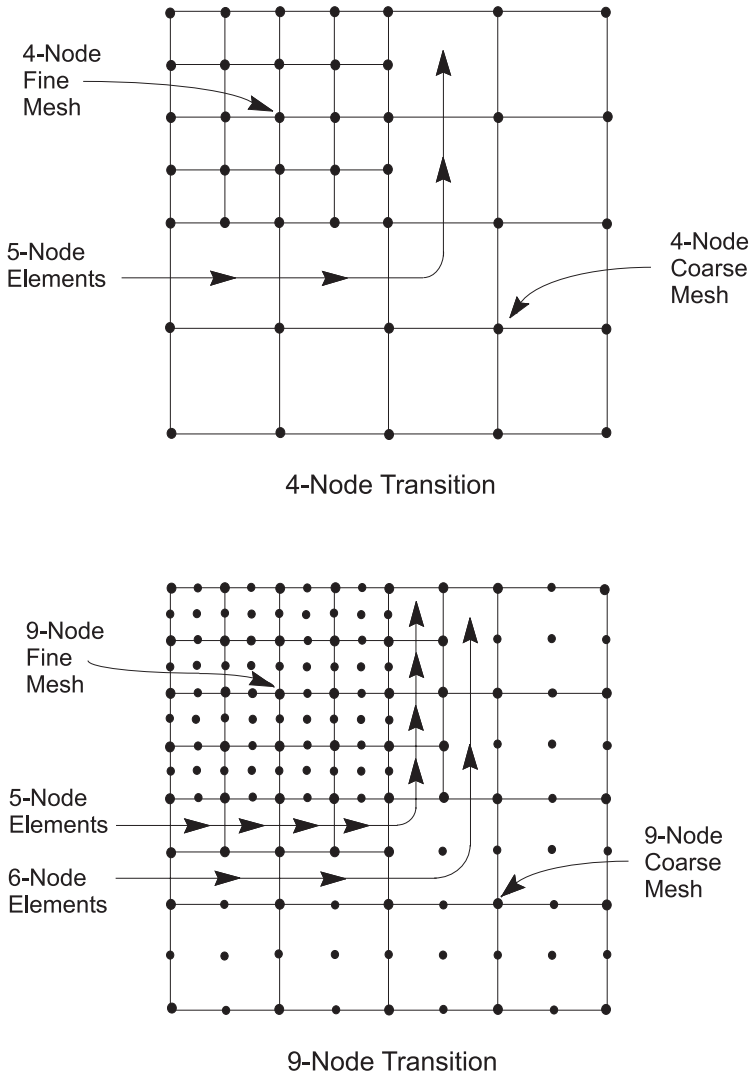


Figure 62
Examples of Asolid Element Mesh Transitions

the corner joints should be no larger than half the distance between the corner joints.

- The position of the center joint with respect to each pair of opposite midside joints is subject to the same rules as is the position of a midside joint with respect to the corner joints.

Asolid elements should not be connected to joints that are connected to other types of elements unless special provisions are made to enforce axisymmetric conditions upon these joints.

Degrees of Freedom

The Asolid element activates the three translational degrees of freedom at each of its connected joints. Rotational degrees of freedom are not activated.

This element contributes stiffness only to the degrees of freedom in the plane of the element. It is necessary to provide restraints or other supports for the translational degrees of freedom that are normal to this plane; otherwise, the structure will be unstable.

See Topic "Degrees of Freedom" (page 83) in Chapter "Joints and Degrees of Freedom" for more information.

Local Coordinate System

Each Asolid 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. These axes are always parallel to the axes of the global coordinate system with the same positive sense.

Local axis 1 is parallel to the radial direction, axis 2 is parallel to the axis of symmetry, and axis 3 is tangent to the circumferential direction of the axisymmetric solid. See the table in Topic "Joint Connectivity" (page 221). The local coordinate system for elements parallel to the Y-Z plane is shown in Figure 63 (page 225).

Stresses and Strains

The Asolid element models the mid-plane of a representative sector of an axisymmetric structure whose stresses and strains do not vary in the circumferential direction.

Circumferential shear stresses (σ_{12}, σ_{13}) and shear strains (γ_{12}, γ_{13}) are assumed to be zero. Displacements in the circumferential (local 3) direction have no effect on the element.

Displacements in the radial (local 1) direction cause circumferential normal strains:

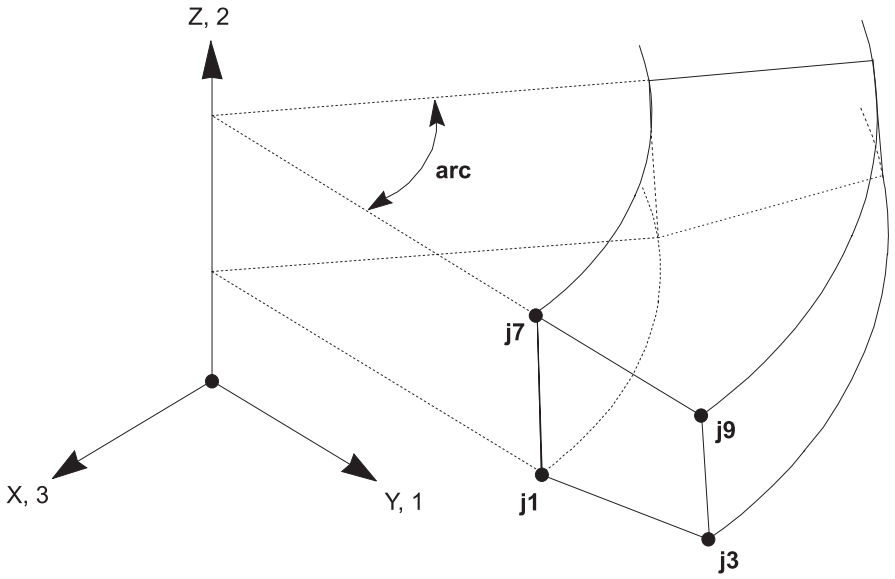


Figure 63
Asolid Element Local Coordinate System and Arc Definition

$$\varepsilon_{33} = \frac{u_1}{r}$$

where u_1 is the radial displacement, and r is the radius at the point in question.

The circumferential normal stress (σ_{33}) is computed as usual from the three normal strains.

See Topic “Stresses and Strains” (page 137) in Chapter “Material Properties” for more information.

Material Properties

The material properties for each Asolid element are specified by reference to a previously-defined Material. Orthotropic properties are used, even if the Material selected was defined as anisotropic. The material properties used by the Asolid element are:

- The moduli of elasticity, **e1**, **e2**, and **e3**
- The shear modulus, **g12**

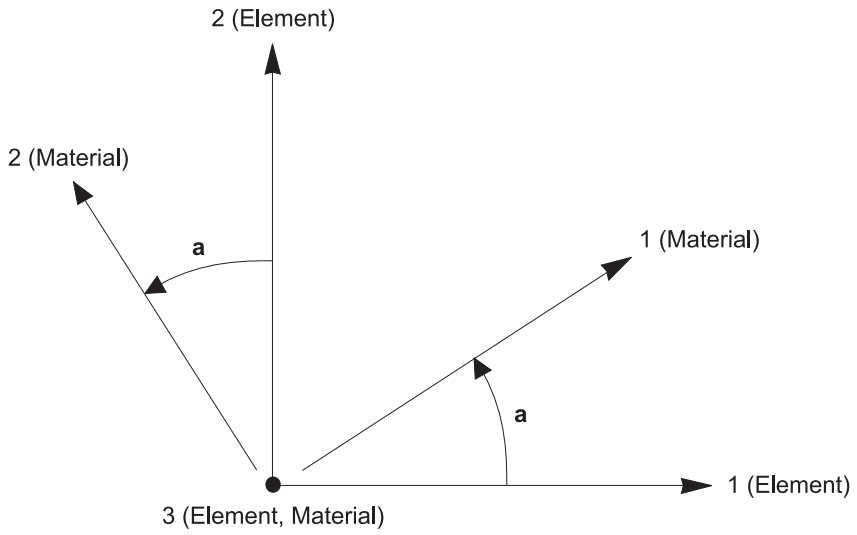


Figure 64
Asolid Element Material Angle

- The Poisson's ratios, **u12**, **u13** and **u23**
- The coefficients of thermal expansion, **a1**, **a2**, and **a3**
- The mass density, **m**, for computing element mass
- The weight density, **w**, for computing Self-Weight and Gravity Loads

All material properties (except the densities) are obtained at the material temperature of each individual element.

See Chapter "Material Properties" (page 135) for more information.

Material Angle

The material local coordinate system and the element local coordinate system need not be the same. The local 3 directions always coincide for the two systems, but the material 1 axis and the element 1 axis may differ by the angle **a** as shown in Figure 64 (page 226). This angle has no effect for isotropic material properties since they are independent of orientation.

See Topic “Local Coordinate System” (page 136) in Chapter “Material Properties” for more information.

Arc and Thickness

The Asolid element represents a solid that is created by rotating the element’s planar shape through 360° about the axis of symmetry. However, the analysis considers only a representative sector of the solid. You can specify the size of the sector, in degrees, using the parameter **arc**. For example, **arc**=360 models the full structure, and **arc**=90 models one quarter of it. See Figure 63 (page 225). Setting **arc**=0, the default, models a one-radian sector. One radian is the same as 180°/π, or approximately 57.3°.

The element “thickness” (circumferential extent), *h*, increases with the radial distance, *r*, from the axis of symmetry:

$$h = \frac{\pi \cdot \mathbf{arc}}{180} r$$

Clearly the thickness varies over the plane of the element.

The element thickness is used for calculating the element stiffness, mass, and loads. Hence, joint forces computed from the element are proportional to **arc**.

Mass

In a dynamic analysis, the mass of the structure is used to compute inertial forces. The mass contributed by the Asolid 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 product of the mass density, **m**, multiplied by the thickness, *h*. 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) even though the element contributes stiffness to only two of these degrees of freedom.

For more information:

- See Topic “Mass Density” (page 142) in Chapter “Material Properties.”

- See Chapter “Static and Dynamic Analysis” (page 291).

Self-Weight Load

Self-Weight Load activates the self-weight of all elements in the model. For an Asolid element, the self-weight is a force that is distributed over the plane of the element. The magnitude of the self-weight is equal to the weight density, **w**, multiplied by the thickness, *h*.

Self-Weight Load always acts downward, in the global –Z direction. If the downward direction corresponds to the radial or circumferential direction of an Asolid element, the Self-Weight Load for that element will be zero, since self-weight acting in these directions is not axisymmetric. Non-zero Self-Weight Load will only exist for elements whose axial (local 2) direction is vertical.

You may scale the self-weight by a single scale factor that applies equally to all elements in the structure.

For more information:

- See Topic “Weight Density” (page 143) in Chapter “Material Properties” for the definition of **w**.
- See Topic “Arc and Thickness” (page 227) in this chapter for the definition of *h*.
- See Topic “Self-Weight Load” (page 283) in Chapter “Load Cases.”

Gravity Load

Gravity Load can be applied to each Asolid element to activate the self-weight of the element. Using Gravity Load, the self-weight can be scaled and applied in any direction. Different scale factors and directions can be applied to each element. However, only the components of Gravity load acting in the axial direction of an Asolid element will be non-zero. Components in the radial or circumferential direction will be set to zero, since gravity acting in these directions is not axisymmetric.

If all elements are to be loaded equally and in the downward direction, it is more convenient to use Self-Weight Load.

For more information:

- See Topic “Self-Weight Load” (page 228) in this chapter for the definition of self-weight for the Asolid element.
- See Topic “Gravity Load” (page 284) in Chapter “Load Cases.”

Surface Pressure Load

The Surface Pressure Load is used to apply external pressure loads upon any of the three or four side faces of the Asolid element. The definition of these faces is shown in Figure 61 (page 222). Surface pressure always acts normal to the face. Positive pressures are directed toward the interior of the element.

The pressure may be constant over a face or interpolated from values given at the joints. The values given at the joints are obtained from Joint Patterns, and need not be the same for the different faces. Joint Patterns can be used to easily apply hydrostatic pressures.

The pressure acting on a side is multiplied by the thickness, *h*, integrated along the length of the side, and apportioned to the two or three joints on that side.

See Chapter “Load Cases” (page 279) for more information.

Pore Pressure Load

The Pore Pressure Load is used to model the drag and buoyancy effects of a fluid *within* a solid medium, such as the effect of water upon the solid skeleton of a soil.

Scalar fluid-pressure values are given at the element joints by Joint Patterns, and interpolated over the element. The total force acting on the element is the integral of the gradient of this pressure field, multiplied by the thickness *h*, over the plane of the element. This force is apportioned to each of the joints of the element. The forces are typically directed from regions of high pressure toward regions of low pressure.

See Chapter “Load Cases” (page 279) for more information.

Temperature Load

The Temperature Load creates thermal strain in the Asolid element. This strain is given by the product of the Material coefficient of thermal expansion and the temperature change of the element. The temperature change is measured from the ele-

ment Reference Temperature to the element Load Temperature. Temperature changes are assumed to be constant through the element thickness.

See Chapter “Load Cases” (page 279) for more information.

Rotate Load

Rotate Load is used to apply centrifugal force to Asolid elements. Each element is assumed to rotate about its own axis of symmetry at a constant angular velocity.

The angular velocity creates a load on the element that is proportional to its mass, its distance from the axis of rotation, and the square of the angular velocity. This load acts in the positive radial direction, and is apportioned to each joint of the element. No Rotate Load will be produced by an element with zero mass density.

Since Rotate Loads assume a constant rate of rotation, it does not make sense to use a Load Case that contains Rotate Load in a time-history analysis unless that Load Case is applied quasi-statically (i.e., with a *very slow* time variation).

For more information:

- See Topic “Mass Density” (page 142) in Chapter “Material Properties.”
- See Chapter “Load Cases” (page 279).

Stress Output

The Asolid element stresses are evaluated at the standard 2-by-2 Gauss integration points of the element and extrapolated to the joints. See Cook, Malkus, and Plesha (1989) for more information.

The stresses are computed for the following types of Analysis Cases: Loads, Modes, Specs, Histories, and Combos. History results are the maximum and minimum values attained during the analysis, and are only available if specifically requested in the definition of the individual History cases. No results are available for Moving Loads or Combos that contain Moving Loads.

Principal values and their associated principal directions in the element local 1-2 plane are also computed for the Loads and Modes. The angle given is measured counterclockwise (when viewed from the +3 direction) 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.

Asolid element stresses are printed in the output file under the heading:

```
ASOLID ELEMENT STRESSES
```

All results for a single element are printed together. Elements are presented in alpha-numeric order. For each Analysis Case the stresses are printed at all element joints.

The stresses are labeled S11, S22, S33, and S12 corresponding to σ_{11}, σ_{22}, σ_{33}, and σ_{12}, respectively, in the element local coordinate system. Stresses σ_{12} and σ_{13} are always zero and are not printed.

Typical Asolid element output is shown in Figure 65 (page 232).

By default, no stresses are printed. You may specify which Analysis Cases are to be printed. The stresses can always be displayed using the SAP2000 graphical interface even if no results have been selected for printing.

For more information:

- See Chapter “Load Cases” (page 279).
- See Chapter “Static and Dynamic Analysis” (page 291).
- See Chapter “The Output Files” (page 395).

Joint Force Output

The **Asolid element joint forces** are concentrated forces acting at the joints of the element that represent the effect of the rest of the structure upon the element.

See Topic “Element Joint Force Output” (page 402) in Chapter “The Output Files” for more information.

```
A S O L I D   E L E M E N T   S T R E S S E S

ELEM       1 ==================

LOAD INTPRES ------------------

  JOINT         S11         S22         S33         S12
      1   -0.939522    0.157629    1.255412    0.000920
      2   -0.836460    0.114523    1.065966    0.000829
      3   -0.733398    0.071418    0.876520    0.000739
     12   -0.939522    0.157629    1.255412   -2.89E-13
     13   -0.836460    0.114523    1.065966   -2.32E-13
     14   -0.733398    0.071418    0.876520   -1.76E-13
     23   -0.939522    0.157629    1.255412   -0.000920
     24   -0.836460    0.114523    1.065966   -0.000829
     25   -0.733398    0.071418    0.876520   -0.000739

  JOINT       S-MAX       S-MIN                   ANGLE
      1    0.157630   -0.939523                89.951947
      2    0.114524   -0.836461                89.950033
      3    0.071418   -0.733399                89.947423
     12    0.157629   -0.939522               -90.000000
     13    0.114523   -0.836460               -90.000000
     14    0.071418   -0.733398               -90.000000
     23    0.157630   -0.939523               -89.951947
     24    0.114524   -0.836461               -89.950033
     25    0.071418   -0.733399               -89.947423

LOAD LOGTEMP ------------------

  JOINT         S11         S22         S33         S12
      1   -9.421631 -172.393810 -136.017533   -0.187845
      2   -9.169759 -150.661663 -106.777127   -0.172932
      3   -8.917887 -128.929517  -77.536722   -0.158019
     12   -9.421631 -172.393810 -136.017533   -1.28E-11
     13   -9.169759 -150.661663 -106.777127   -9.82E-12
     14   -8.917887 -128.929517  -77.536722   -6.79E-12
     23   -9.421631 -172.393810 -136.017533    0.187845
     24   -9.169759 -150.661663 -106.777127    0.172932
     25   -8.917887 -128.929517  -77.536722    0.158019

ELEM       2 ==================

LOAD INTPRES ------------------

  JOINT         S11         S22         S33         S12
      3   -0.634081    0.168448    0.971652    0.000681
      4   -0.561526    0.117673    0.797344    0.000613
      5   -0.488971    0.066898    0.623036    0.000545
     14   -0.634081    0.168448    0.971652   -1.46E-13
     15   -0.561526    0.117673    0.797344   -2.06E-14
     16   -0.488971    0.066898    0.623036    1.05E-13
     25   -0.634081    0.168448    0.971652   -0.000681
     26   -0.561526    0.117673    0.797344   -0.000613
     27   -0.488971    0.066898    0.623036   -0.000545
```

...Output Edited for Brevity...

Figure 65
Asolid Element Stresses in the Output File

Chapter XIII

The Solid Element

The **Solid** element is used to model three-dimensional solid structures.

Advanced Topics

- Overview
- Joint Connectivity
- Degrees of Freedom
- Local Coordinate System
- Stresses and Strains
- Material Properties
- Material Angles
- Incompatible Bending Modes
- Mass
- Self-Weight Load
- Gravity Load
- Surface Pressure Load
- Pore Pressure Load
- Temperature Load

- Stress Output
- Joint Force Output

Overview

The Solid element is an eight-node element for modeling three-dimensional structures and solids. It is based upon an isoparametric formulation that includes nine optional incompatible bending modes.

The incompatible bending modes significantly improve the bending behavior of the element if the element geometry is of a rectangular form. Improved behavior is exhibited even with non-rectangular geometry.

The local coordinate system for each Solid element is identical to the global system. The local coordinate system is used for defining Material properties and loads, and for interpreting output.

Temperature-dependent, anisotropic material properties are allowed. Each element may be loaded by gravity (in any direction); surface pressure on the faces; pore pressure within the element; and loads due to temperature change.

An 2 x 2 x 2 numerical integration scheme is used for the Solid. Stresses in the element local coordinate system are evaluated at the integration points and extrapolated to the joints of the element. An approximate error in the stresses 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 the finite element approximation and can then be used as the basis for the selection of a new and more accurate finite element mesh.

Joint Connectivity

Each Solid element has six quadrilateral faces, with a joint located at each of the eight corners as shown in Figure 66 (page 235). It is important to note the relative position of the eight joints: the paths **j1**-**j2**-**j3** and **j5**-**j6**-**j7** should appear counterclockwise when viewed along the direction from **j5** to **j1**. Mathematically stated, the three vectors:

- $\mathbf{V}_{12}$, from joints **j1** to **j2**,
- $\mathbf{V}_{13}$, from joints **j1** to **j3**,

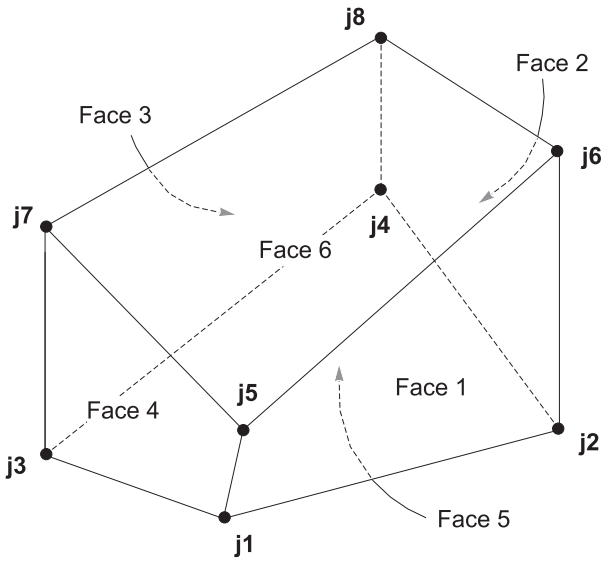


Figure 66
Solid Element Joint Connectivity and Face Definitions

- $\mathbf{V}_{15}$, from joints **j1** to **j5**,

must form a positive triple product, that is:

$$(\mathbf{V}_{12} \times \mathbf{V}_{13}) \cdot \mathbf{V}_{15} > 0$$

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

- The inside angle at each corner of the faces must be less than 180°. Best results 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. This is the ratio of the longest dimension of the element to its shortest dimension. Best results are obtained for aspect ratios near unity, or at least less than four. The aspect ratio should not exceed ten.

These conditions can usually be met with adequate mesh refinement.

Degrees of Freedom

The Solid element activates the three translational degrees of freedom at each of its connected joints. Rotational degrees of freedom are not activated. This element contributes stiffness to all of these translational degrees of freedom.

See Topic "Degrees of Freedom" (page 83) in Chapter "Joints and Degrees of Freedom" for more information.

Local Coordinate System

Each Solid 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. These axes *always* correspond with the global coordinate axes X, Y and Z, respectively, regardless of the orientation of the element.

Stresses and Strains

The Solid element models a general state of stress and strain in a three-dimensional solid. All six stress and strain components are active for this element.

See Topic "Stresses and Strains" (page 137) in Chapter "Material Properties" for more information.

Material Properties

The material properties for each Solid element are specified by reference to a previously-defined Material. Fully anisotropic material properties are used. The material properties used by the Solid element are:

- The moduli of elasticity, **e1**, **e2**, and **e3**
- The shear moduli, **g12**, **g13**, and **g23**
- All of the Poisson's ratios, **u12**, **u13**, **u23**, ..., **u56**
- The coefficients of thermal expansion, **a1**, **a2**, **a3**, **a12**, **a13**, and **a23**
- The mass density, **m**, used for computing element mass
- The weight density, **w**, used for computing Self-Weight and Gravity Loads

All material properties (except the densities) are obtained at the material temperature of each individual element.

See Chapter “Material Properties” (page 135) for more information.

Material Angles

The material local coordinate system and the element local coordinate system need not be the same. The material coordinate system is oriented with respect to the element coordinate system using the three angles **a**, **b**, and **c** according to the following procedure:

- The material system is first aligned with the element system;
- The material system is then rotated about its +3 axis by angle **a**;
- The material system is next rotated about the resulting +2 axis by angle **b**;
- The material system is lastly rotated about the resulting +1 axis by angle **c**;

This is shown in Figure 67 (page 238). These angles have no effect for isotropic material properties since they are independent of orientation.

See Topic “Local Coordinate System” (page 136) in Chapter “Material Properties” for more information.

Incompatible Bending Modes

By default each Solid element includes nine incompatible bending modes in its stiffness formulation. These incompatible bending modes significantly improve the bending behavior of the element if the element geometry is of a rectangular form. Improved behavior is exhibited even with non-rectangular geometry.

If an element is severely distorted, the inclusion of the incompatible modes should be suppressed. The element then uses the standard isoparametric formulation. Incompatible bending modes may also be suppressed in cases where bending is not important, such as in typical geotechnical problems.

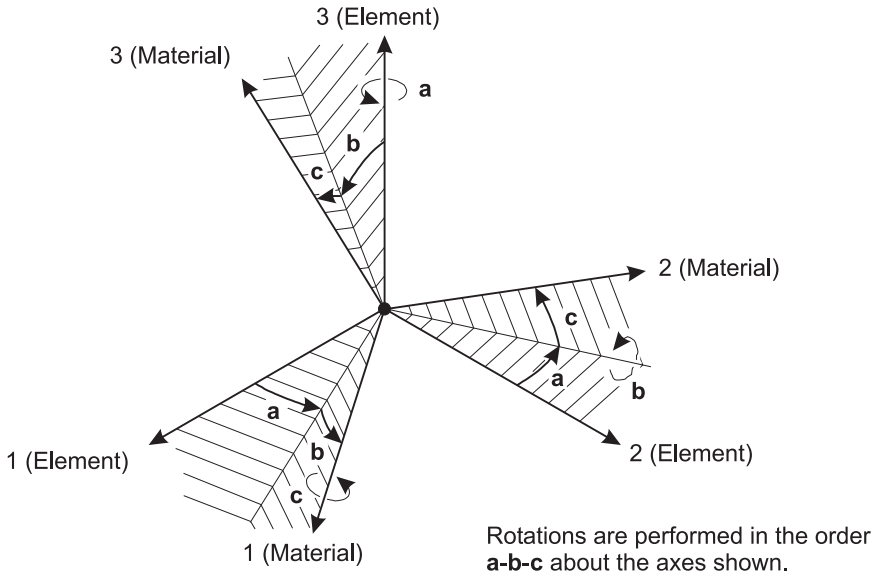


Figure 67
Solid Element Material Angles

Mass

In a dynamic analysis, the mass of the structure is used to compute inertial forces. The mass contributed by the Solid 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 of the mass density, **m**, over the volume of the element. 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).

For more information:

- See Topic “Mass Density” (page 142) in Chapter “Material Properties.”
- See Chapter “Static and Dynamic Analysis” (page 291).

Self-Weight Load

Self-Weight Load activates the self-weight of all elements in the model. For a Solid element, the self-weight is a force that is uniformly distributed over the volume of the element. The magnitude of the self-weight is equal to the weight density, **w**.

Self-Weight Load always acts downward, in the global –Z direction. You may scale the self-weight by a single scale factor that applies equally to all elements in the structure.

For more information:

- See Topic “Weight Density” (page 143) in Chapter “Material Properties” for the definition of **w**.
- See Topic “Self-Weight Load” (page 283) in Chapter “Load Cases.”

Gravity Load

Gravity Load can be applied to each Solid element to activate the self-weight of the element. Using Gravity Load, the self-weight can be scaled and applied in any direction. Different scale factors and directions can be applied to each element.

If all elements are to be loaded equally and in the downward direction, it is more convenient to use Self-Weight Load.

For more information:

- See Topic “Self-Weight Load” (page 239) in this chapter for the definition of self-weight for the Solid element.
- See Topic “Gravity Load” (page 284) in Chapter “Load Cases.”

Surface Pressure Load

The Surface Pressure Load is used to apply external pressure loads upon any of the six faces of the Solid element. The definition of these faces is shown in Figure 66 (page 235). Surface pressure always acts normal to the face. Positive pressures are directed toward the interior of the element.

The pressure may be constant over a face or interpolated from values given at the joints. The values given at the joints are obtained from Joint Patterns, and need not

be the same for the different faces. Joint Patterns can be used to easily apply hydrostatic pressures.

The pressure acting on a given face is integrated over the area of that face, and the resulting force is apportioned to the four corner joints of the face.

See Chapter "Load Cases" (page 279) for more information.

Pore Pressure Load

The Pore Pressure Load is used to model the drag and buoyancy effects of a fluid *within* a solid medium, such as the effect of water upon the solid skeleton of a soil.

Scalar fluid-pressure values are given at the element joints by Joint Patterns, and interpolated over the element. The total force acting on the element is the integral of the gradient of this pressure field over the volume of the element. This force is apportioned to each of the joints of the element. The forces are typically directed from regions of high pressure toward regions of low pressure.

See Chapter "Load Cases" (page 279) for more information.

Temperature Load

The Temperature Load creates thermal strain in the Solid element. This strain is given by the product of the Material coefficient of thermal expansion and the temperature change of the element. The temperature change is measured from the element Reference Temperature to the element Load Temperature. Temperature changes are assumed to be constant through the element thickness.

See Chapter "Load Cases" (page 279) for more information.

Stress Output

The Solid element stresses are evaluated at the standard 2 x 2 x 2 Gauss integration points of the element and extrapolated to the joints. See Cook, Malkus, and Plesha (1989) for more information.

The stresses are computed for the following types of Analysis Cases: Loads, Modes, Specs, Histories, and Combos. History results are the maximum and minimum values attained during the analysis, and are only available if specifically re-

quested in the definition of the individual History cases. No results are available for Moving Loads or Combos that contain Moving Loads.

Principal values and their associated principal directions in the element local coordinate system are also computed for the Loads and Modes. Three direction cosines each are given for the directions of the maximum and minimum principal stresses. The direction of the middle principal stress is perpendicular to the maximum and minimum principal directions.

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

Solid element stresses are printed in the output file under the heading:

```
SOLID ELEMENT STRESSES
```

All results for a single element are printed together. Elements are presented in alpha-numeric order. For each Analysis Case the stresses are printed at all element joints.

The stresses are labeled S11, S22, S33, S12, S13, and S23 corresponding to σ_{11}, σ_{22}, σ_{33}, σ_{12}, σ_{13}, and σ_{23}, respectively, in the element local coordinate system.

Typical Solid element output is shown in Figure 68 (page 242).

By default, no stresses are printed. You may specify which Analysis Cases are to be printed. The stresses can always be displayed using the SAP2000 graphical interface even if no results have been selected for printing.

For more information:

- See Chapter “Load Cases” (page 279).
- See Chapter “Static and Dynamic Analysis” (page 291).
- See Chapter “The Output Files” (page 395).

Joint Force Output

The **Solid element joint forces** are concentrated forces acting at the joints of the element that represent the effect of the rest of the structure upon the element.

See Topic “Element Joint Force Output” (page 402) in Chapter “The Output Files” for more information.

```
S O L I D   E L E M E N T   S T R E S S E S

ELEM       1 ==================

LOAD       1 ------------------

  JOINT         S11         S22         S33         S12         S13         S23
      3   -0.364752  207.803651    6.559145   -0.247394   -0.494788  -13.888889
     18   -0.364752  207.803651    6.559145    0.247394    0.494788  -13.888889
      4   -0.199823  208.298440   -3.171692   -0.247394    0.494788  -13.888889
     19   -0.199823  208.298440   -3.171692    0.247394   -0.494788  -13.888889
      8    1.966476    0.811970    7.053933    0.247394   -0.494788  -13.888889
     23    1.966476    0.811970    7.053933   -0.247394    0.494788  -13.888889
      9   -0.837325    0.317181   -3.666480    0.247394    0.494788  -13.888889
     24   -0.837325    0.317181   -3.666480   -0.247394   -0.494788  -13.888889

  JOINT       S-MAX       S-MID       S-MIN MAX-1 MAX-2 MAX-3 MIN-1 MIN-2 MIN-3
      3  208.757887    5.648477   -0.408320 -.001  .998 -.069  .996  .007  .084
     18  208.757887    5.648477   -0.408320  .001  .998 -.069  .996 -.007 -.084
      4  209.207102   -0.142270   -4.137907 -.001  .998 -.065 -.120  .065  .991
     19  209.207102   -0.142270   -4.137907  .001  .998 -.065  .120  .065  .991
      8   18.186219    1.949535  -10.303376 -.033 -.624  .780  .009  .781  .625
     23   18.186219    1.949535  -10.303376  .033 -.624  .780 -.009  .781  .625
      9   12.357765   -0.819458  -15.724930 -.010  .755 -.655 -.036  .654  .755
     24   12.357765   -0.819458  -15.724930  .010  .755 -.655  .036  .654  .755

ELEM       2 ==================

LOAD       1 ------------------

  JOINT         S11         S22         S33         S12         S13         S23
      8   -1.966476   -0.811970   -7.053933   -0.247394   -0.494788  -13.888889
     23   -1.966476   -0.811970   -7.053933    0.247394    0.494788  -13.888889
      9    0.837325   -0.317181    3.666480   -0.247394    0.494788  -13.888889
     24    0.837325   -0.317181    3.666480    0.247394   -0.494788  -13.888889
     13    0.364752 -207.803651   -6.559145    0.247394   -0.494788  -13.888889
     28    0.364752 -207.803651   -6.559145   -0.247394    0.494788  -13.888889
     14    0.199823 -208.298440    3.171692    0.247394    0.494788  -13.888889
     29    0.199823 -208.298440    3.171692   -0.247394   -0.494788  -13.888889

  JOINT       S-MAX       S-MID       S-MIN MAX-1 MAX-2 MAX-3 MIN-1 MIN-2 MIN-3
      8   10.303376   -1.949535  -18.186219  .009  .781 -.625  .033  .624  .780
     23   10.303376   -1.949535  -18.186219 -.009  .781 -.625 -.033  .624  .780
      9   15.724930    0.819458  -12.357765  .036 -.654  .755 -.010  .755  .655
     24   15.724930    0.819458  -12.357765 -.036 -.654  .755  .010  .755  .655
     13    0.408320   -5.648477 -208.757887  .996  .007 -.084 -.001  .998  .069
     28    0.408320   -5.648477 -208.757887  .996 -.007  .084  .001  .998  .069
     14    4.137907    0.142270 -209.207102  .120 -.065  .991 -.001  .998  .065
     29    4.137907    0.142270 -209.207102 -.120 -.065  .991  .001  .998  .065
```

...Output Edited for Brevity...

Figure 68
Solid Element Stresses in the Output File

Chapter XIV

The Nllink Element

The **Nllink** element is used to model local structural nonlinearities such as gaps, dampers, isolators, and the like. Nonlinear behavior is exhibited only during nonlinear time-history analyses. For all other analyses, the Nllink element behaves linearly.

Advanced Topics

- Overview
- Joint Connectivity
- Zero-Length Elements
- Degrees of Freedom
- Local Coordinate System
- Advanced Local Coordinate System
- Internal Deformations
- Nlprop Properties
- Nonlinear Deformation Loads
- Mass
- Self-Weight Load

- Gravity Load
- Internal Force and Deformation Output
- Joint Force Output

Overview

The **Nllink** element is used to model local structural nonlinearities. *Nonlinear behavior is only exhibited during nonlinear time-history analyses. For all other analyses, the Nllink element behaves linearly.*

Each Nllink element may be either a one-joint grounded spring or a two-joint link. Properties for either type of element are defined in the same way.

Each element is assumed to be composed of six separate "springs," one for each of six deformational degrees-of freedom (axial, shear, torsion, and pure bending). Each of these springs possesses a dual set of properties:

- Linear effective-stiffness and effective-damping properties used for all linear analyses
- An optional nonlinear force-deformation relationship used only for nonlinear time-history analyses

If the optional nonlinear properties are not specified for a given degree of freedom, the linear stiffness (but not damping) properties are used for nonlinear time-history analyses.

The linear effective damping property is only used for response-spectrum analyses and linear time-history analyses.

The nonlinear force-deformation relationships of these springs may be coupled or uncoupled, depending on the type of behavior modeled.

A set of properties for all six degrees of freedom is called an Nlprop. Each Nlprop consists of mass, weight, and up to six linear and nonlinear force-deformation relationships that may be used by one or more Nllink elements.

The types of nonlinear behavior that can be modeled with this element include:

- Viscoelastic damping
- Gap (compression only) and hook (tension only)
- Uniaxial plasticity

- Biaxial-plasticity base isolator
- Friction-pendulum base isolator

Each element has its own local coordinate system for defining the force-deformation properties and for interpreting output.

Each Nllink element may be loaded by gravity (in any direction).

Available output includes the deformation across the element, and the internal forces at the joints of the element.

Joint Connectivity

Each Nllink element may take one of the following two configurations:

- A link connecting two joints, **i** and **j**; it is permissible for the two joints to share the same location in space
- A spring connecting a single joint, **j**, to ground

Zero-Length Elements

The following types of Nllink elements are considered to be of **zero length**:

- Single-joint elements
- Two-joint elements with the distance from joint **i** to joint **j** being less than or equal to the zero-length tolerance, **zero**, that you specify.

The default value for **zero** is 10^{-3}. The purpose of this tolerance is to account for numerical round-off in the specification and generation of joint coordinates. For example, if joint coordinates are specified to the nearest millimeter, then a possible value for **zero** might be 2 or 3 mm.

Two-joint elements having a length greater than the tolerance **zero** are considered to be of finite length. Whether an element is of zero length or finite length affects the definition of the element local coordinate system.

Degrees of Freedom

The Nllink element always activates all six degrees of freedom at its one or two connected joints. To which joint degrees of freedom the element contributes stiffness depends upon the properties you assign to the element. You must ensure that restraints or other supports are provided to those joint degrees of freedom that receive no stiffness.

For more information:

- See Topic “Degrees of Freedom” (page 83) in Chapter “Joints and Degrees of Freedom.”
- See Topic “Nlprop Properties” (page 254) in this chapter.

Local Coordinate System

Each Nllink element has its own **element local coordinate system** used to define force-deformation properties 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 and corresponds to extensional deformation. The remaining two axes lie in the plane perpendicular to the element and have an orientation that you specify; these directions correspond to shear deformation.

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. The methods provided, however, provide sufficient power and flexibility to describe the orientation of Nllink elements in the most complicated situations.

The simplest method, using the **default orientation** and the **Nllink element coordinate angle**, is described in this topic. Additional methods for defining the Nllink element local coordinate system are described in the next topic.

For more information:

- See Chapter “Coordinate Systems” (page 21) for a description of the concepts and terminology used in this topic.
- See Topic “Advanced Local Coordinate System” (page 249) in this chapter.

Longitudinal Axis 1

Local axis 1 is the longitudinal axis of the element, corresponding to extensional deformation. This axis is determined as follows:

- For elements of finite length this axis is automatically defined as the direction from joint **i** to joint **j**
- For zero-length elements the local 1 axis defaults to the +Z global coordinate direction (upward)

For the definition of zero-length elements, see Topic "Zero-Length Elements" (page 245) in this chapter.

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 procedure used here is identical to that for the Frame element:

- 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 Nllink 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 counterclockwise when the local +1 axis is pointing toward you. The procedure used here is identical to that for the Frame element.

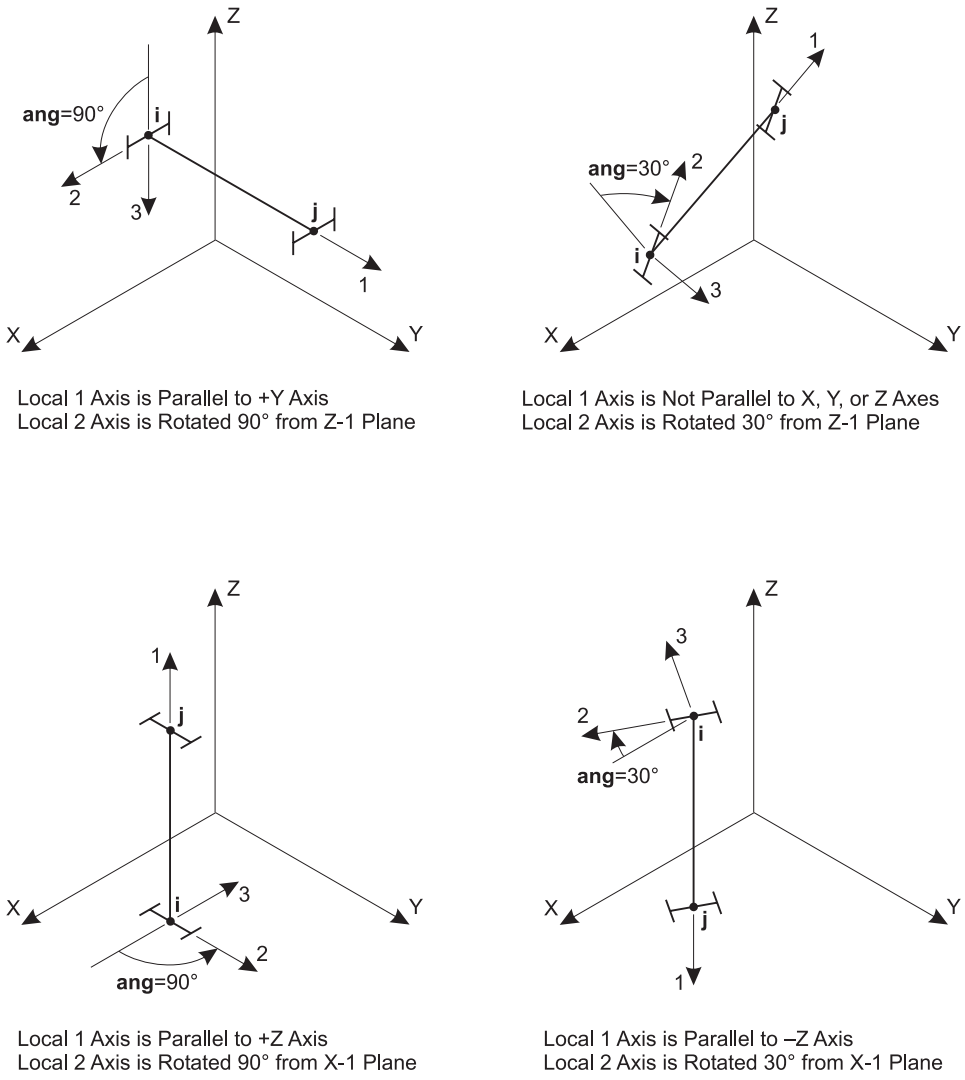


Figure 69
The Nllink Element Coordinate Angle with Respect to the Default Orientation

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 69 (page 248) for examples.

Advanced Local Coordinate System

By default, the element local coordinate system is defined using the element coordinate angle measured with respect to the global +Z and +X directions, as described in the previous topic. In certain modeling situations it may be useful to have more control over the specification of the local coordinate system.

This topic describes how to define the orientation of the transverse local 2 and 3 axes with respect to an arbitrary reference vector when the element coordinate angle, **ang**, is zero. If **ang** is different from zero, it is the angle through which the local 2 and 3 axes are rotated about the positive local 1 axis from the orientation determined by the reference vector.

This topic also describes how to change the orientation of the local 1 axis from the default global +Z direction for zero-length elements. The local 1 axis is *always* directed from joint **i** to joint **j** for elements of *finite length*.

For more information:

- See Chapter “Coordinate Systems” (page 21) for a description of the concepts and terminology used in this topic.
- See Topic “Local Coordinate System” (page 245) in this chapter.

Axis Reference Vector

To define the local 1 axis for *zero-length* elements, you specify an **axis reference vector** that is parallel to and has the same positive sense as the desired local 1 axis. The axis reference vector has no effect upon finite-length elements.

To define the axis reference vector, you must first specify or use the default values for:

- A coordinate direction **axdir** (the default is +Z)
- A fixed coordinate system **csys** (the default is zero, indicating the global coordinate system). This will be the same coordinate system that is used to define the plane reference vector, as described below

You may optionally specify:

- A pair of joints, **axveca** and **axvecb** (the default for each is zero, indicating the center of the element). If both are zero, this option is not used

For each element, the axis reference vector is determined as follows:

1. A vector is found from joint **axveca** to joint **axvecb**. If this vector is of finite length, it is used as the reference vector $\mathbf{V}_a$
2. Otherwise, the coordinate direction **axdir** is evaluated at the center of the element in fixed coordinate system **csys**, and is used as the reference vector $\mathbf{V}_a$

The center of a zero-length element is taken to be at joint **j**.

The local 1 axis is given by the vector $\mathbf{V_a}$ after it has been normalized to unit length.

Plane Reference Vector

To define the transverse local axes 2 and 3, you specify a **plane reference vector** that is parallel to the desired 1-2 or 1-3 plane. The procedure used here is identical to that for the Frame element.

The reference vector must have a positive projection upon the corresponding transverse local axis (2 or 3, respectively). This means that the positive direction of the reference vector must make an angle of less than 90° with the positive direction of the desired transverse axis.

To define the reference vector, you must first specify or use the default values for:

- A primary coordinate direction **pldirp** (the default is +Z)
- A secondary coordinate direction **pldirs** (the default is +X). Directions **pldirs** and **pldirp** should not be parallel to each other unless you are sure that they are not parallel to local axis 1
- A fixed coordinate system **csys** (the default is zero, indicating the global coordinate system). This will be the same coordinate system that was used to define the axis reference vector, as described above
- The local plane, **local**, to be determined by the reference vector (the default is 12, indicating plane 1-2)

You may optionally specify:

- A pair of joints, **plveca** and **plvecb** (the default for each is zero, indicating the center of the element). If both are zero, this option is not used

For each element, the reference vector is determined as follows:

1. A vector is found from joint **plveca** to joint **plvecb**. If this vector is of finite length and is not parallel to local axis 1, it is used as the reference vector $\boldsymbol{V}_p$.

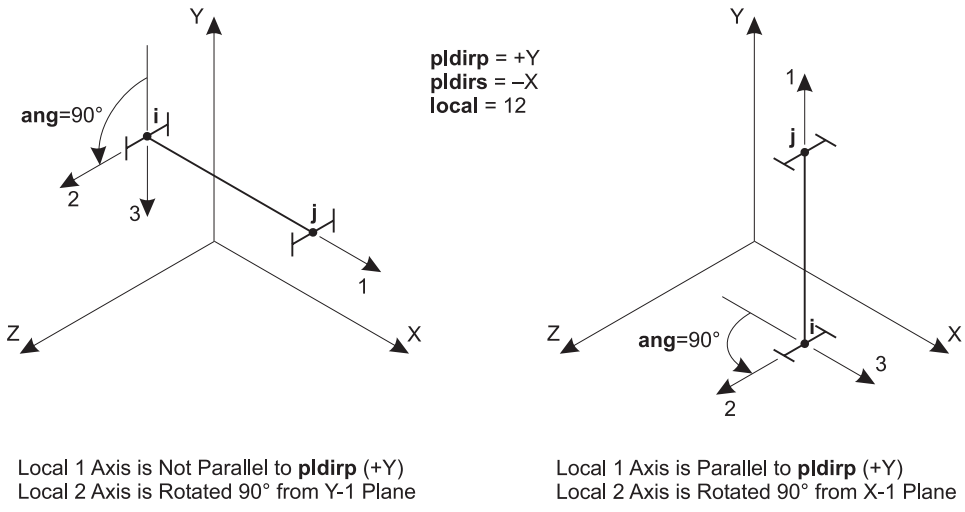


Figure 70
The Nllink Element Coordinate Angle with Respect to Coordinate Directions

2. Otherwise, the primary coordinate direction **pldirp** is evaluated at the center of the element in fixed coordinate system **csys**. If this direction is not parallel to local axis 1, it is used as the reference vector $\boldsymbol{V_p}$.

3. Otherwise, the secondary coordinate direction **pldirs** is evaluated at the center of the element in fixed coordinate system **csys**. If this direction is not parallel to local axis 1, it is used as the reference vector $\boldsymbol{V_p}$.

4. Otherwise, the method fails and the analysis terminates. This will never happen if **pldirp** is not parallel to **pldirs**

A vector is considered to be parallel to local axis 1 if the sine of the angle between them is less than 10^{-3}.

The use of the Nllink element coordinate angle in conjunction with coordinate directions that define the reference vector is illustrated in Figure 70 (page 251). The use of joints to define the reference vector is shown in Figure 71 (page 252).

Determining Transverse Axes 2 and 3

The program uses vector cross products to determine the transverse axes 2 and 3 once the reference vector has been specified. The three axes are represented by the

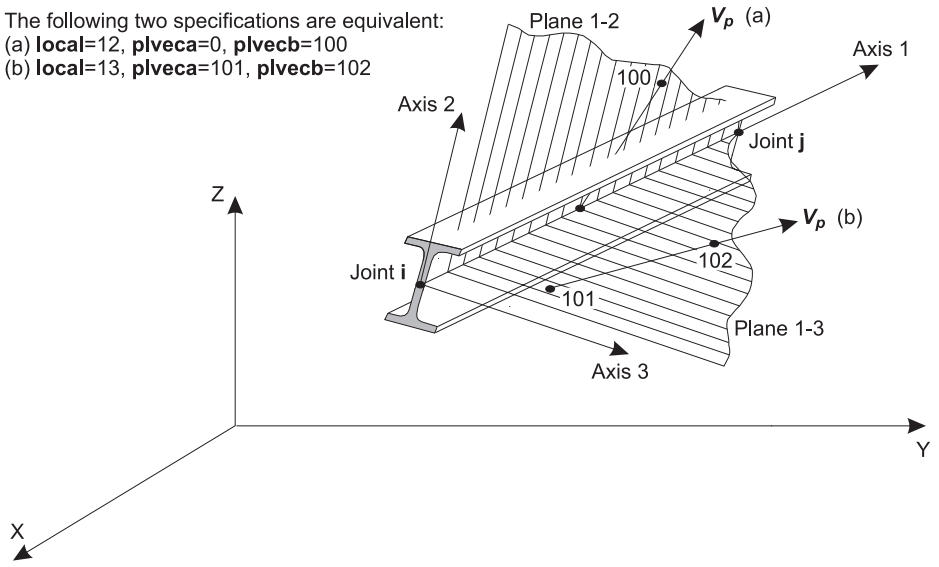


Figure 71
Using Joints to Define the Nllink Element Local Coordinate System

three unit vectors $\mathbf{V}_1$, $\mathbf{V}_2$ and $\mathbf{V}_3$, respectively. The vectors satisfy the cross-product relationship:

$$\mathbf{V}_1 = \mathbf{V}_2 \times \mathbf{V}_3$$

The transverse axes 2 and 3 are defined as follows:

- If the reference vector is parallel to the 1-2 plane, then:

  $$\mathbf{V}_3 = \mathbf{V}_1 \times \mathbf{V}_p \quad \text{and}$$
  $$\mathbf{V}_2 = \mathbf{V}_3 \times \mathbf{V}_1$$

- If the reference vector is parallel to the 1-3 plane, then:

  $$\mathbf{V}_2 = \mathbf{V}_p \times \mathbf{V}_1 \quad \text{and}$$
  $$\mathbf{V}_3 = \mathbf{V}_1 \times \mathbf{V}_2$$

In the common case where the reference vector is perpendicular to axis $\mathbf{V}_1$, the transverse axis in the selected plane will be equal to $\boldsymbol{V}_p$.

Internal Deformations

Six independent **internal deformations** are defined for the Nllink element. These are calculated from the relative displacements of joint **j** with respect to:

- Joint **i** for a two-joint element
- The ground for a single-joint element

For *two-joint link* elements the internal deformations are defined as:

- Axial: $d_{u1} = u_{1\mathbf{j}} - u_{1\mathbf{i}}$
- Shear in the 1-2 plane: $d_{u2} = u_{2\mathbf{j}} - u_{2\mathbf{i}} - \mathbf{dj2}\ r_{3\mathbf{j}} - (L - \mathbf{dj2})\ r_{3\mathbf{i}}$
- Shear in the 1-3 plane: $d_{u3} = u_{3\mathbf{j}} - u_{3\mathbf{i}} + \mathbf{dj3}\ r_{2\mathbf{j}} + (L - \mathbf{dj3})\ r_{2\mathbf{i}}$
- Torsion: $d_{r1} = r_{1\mathbf{j}} - r_{1\mathbf{i}}$
- Pure bending in the 1-3 plane: $d_{r2} = r_{2\mathbf{i}} - r_{2\mathbf{j}}$
- Pure bending in the 1-2 plane: $d_{r3} = r_{3\mathbf{j}} - r_{3\mathbf{i}}$

where:

- $u_{1\mathbf{i}}$, $u_{2\mathbf{i}}$, $u_{3\mathbf{i}}$, $r_{1\mathbf{i}}$, $r_{2\mathbf{i}}$, and $r_{3\mathbf{i}}$ are the translations and rotations at joint **i**
- $u_{1\mathbf{j}}$, $u_{2\mathbf{j}}$, $u_{3\mathbf{j}}$, $r_{1\mathbf{j}}$, $r_{2\mathbf{j}}$, and $r_{3\mathbf{j}}$ are the translations and rotations at joint **j**
- **dj2** is the distance you specify from joint **j** to the location where the shear deformation d_{u2} is measured (the default is zero)
- **dj3** is the distance you specify from joint **j** to the location where the shear deformation d_{u3} is measured (the default is zero)
- L is the length of the element

All translations, rotations, and deformations are expressed in terms of the element local coordinate system.

Note that shear deformation can be caused by rotations as well as translations. These definitions ensure that all deformations will be zero under rigid-body motions of the element.

It is important to note that the negatives of the rotations r_{2i} and r_{2j} have been used for the definition of shear and bending deformations in the 1-3 plane. This provides consistent definitions for shear and moment in both the Nllink and Frame elements.

Three of these internal deformations are illustrated in Figure 72 (page 254).

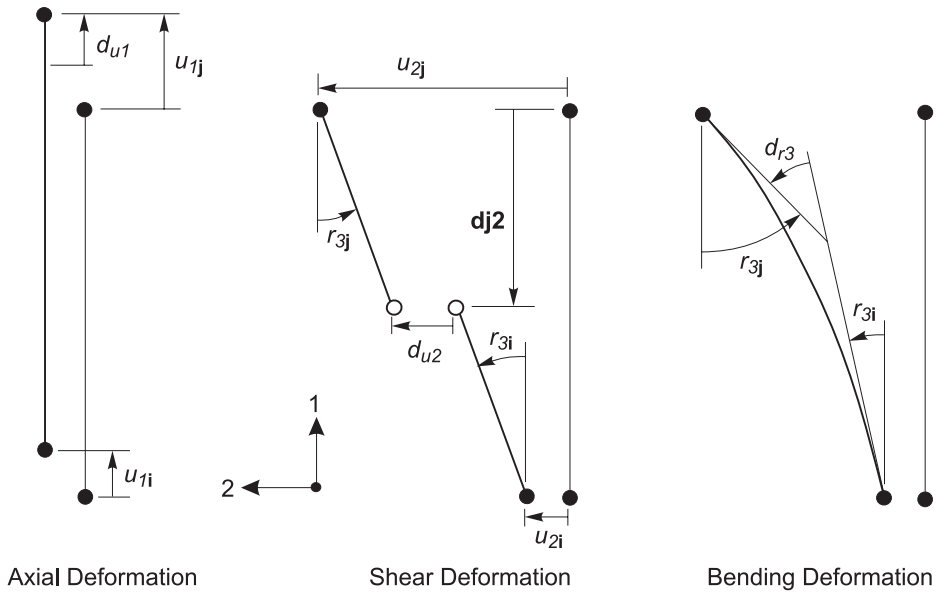


Figure 72
Internal Deformations for a Two-Joint Nllink Element

For *one-joint grounded-spring* elements the internal deformations are the same as above, except that the translations and rotations at joint **i** are taken to be zero:

- Axial: $d_{u1} = u_{1\mathbf{j}}$
- Shear in the 1-2 plane: $d_{u2} = u_{2\mathbf{j}} - \mathbf{dj2}\ r_{3\mathbf{j}}$
- Shear in the 1-3 plane: $d_{u3} = u_{3\mathbf{j}} + \mathbf{dj3}\ r_{2\mathbf{j}}$
- Torsion: $d_{r1} = r_{1\mathbf{j}}$
- Pure bending in the 1-3 plane: $d_{r2} = -r_{2\mathbf{j}}$
- Pure bending in the 1-2 plane: $d_{r3} = r_{3\mathbf{j}}$

Nlprop Properties

An Nlprop is a set of structural properties that can be used to define the behavior of one or more Nllink elements. Nlprops are defined independently of the Nllink elements and are referenced during the definition of the elements.

Each Nlprop specifies the optional nonlinear force-deformation relationships for the six internal deformations. These nonlinear properties are used *only* during a nonlinear time-history analysis.

Effective-stiffness and effective-damping properties may also be specified. These properties are used for all *linear* analyses, which include: static analysis, P-delta analysis, modal analysis, moving-load analysis, response-spectrum analysis, harmonic steady-state analysis, and linear or periodic time-history analysis.

The effective stiffness is also used during nonlinear time-history analyses for all degrees of freedom for which nonlinear properties are not specified. The effective damping is *never* used for nonlinear time-history analysis.

Mass and weight properties may also be specified.

Local Coordinate System

Nlprop properties are defined with respect to the local coordinate system of the Nllink element. The local 1 axis is the longitudinal direction of the element and corresponds to extensional and torsional deformations. The local 2 and 3 directions correspond to shear and bending deformations.

See Topic “Local Coordinate System” (page 245) in this chapter.

Internal Nonlinear Springs

Each Nlprop is assumed to be composed of six internal nonlinear “springs,” one for each of six internal deformations. Each “spring” may actually consist of several components, including springs and dashpots. The force-deformation relationships of these springs may be coupled or independent of each other.

Figure 73 (page 256) shows the springs for three of the deformations: axial, shear in the 1-2 plane, and pure-bending in the 1-2 plane. It is important to note that the shear spring is located a distance **dj2** from joint **j**. All shear deformation is assumed to occur in this spring; the links connecting this spring to the joints (or ground) are rigid in shear. Deformation of the shear spring can be caused by rotations as well as translations at the joints. The force in this spring will produce a linearly-varying moment along the length. This moment is taken to be zero at the shear spring, which acts as a moment hinge. The moment due to shear is independent of, and additive to, the constant moment in the element due to the pure-bending spring.

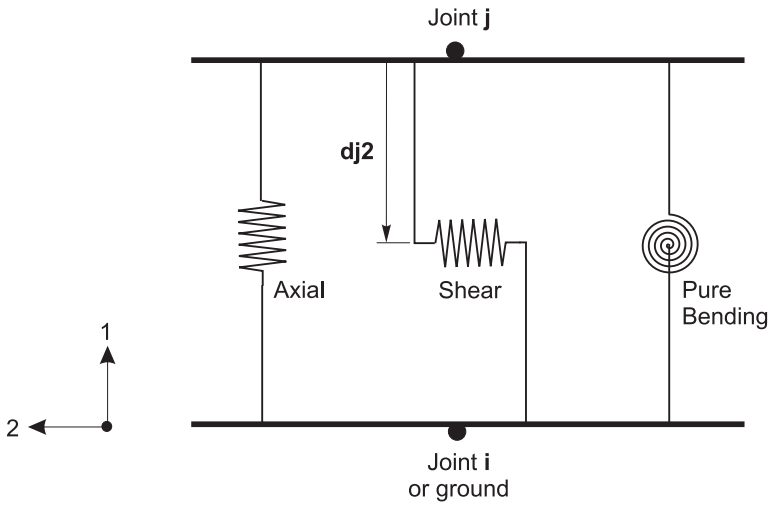


Figure 73
Three of the Six Independent Nonlinear Springs in an Nllink Element

The other three springs that are not shown are for torsion, shear in the 1-3 plane, and pure-bending in the 1-3 plane. The shear spring is located a distance **dj3** from joint **j**.

The values of **dj2** and **dj3** may be different, although normally they would be the same for the friction-pendulum (Isolator2) element.

Spring Force-Deformation Relationships

There are six force-deformation relationships that govern the behavior of the element, one for each of the internal springs:

- Axial: f_{u1} vs. d_{u1}
- Shear: f_{u2} vs. d_{u2} , f_{u3} vs. d_{u3}
- Torsional: f_{r1} vs. d_{r1}
- Pure bending: f_{r2} vs. d_{r2} , f_{r3} vs. d_{r3}

where f_{u1}, f_{u2}, and f_{u3} are the internal-spring forces; and f_{r1}, f_{r2}, and f_{r3} are the internal-spring moments.

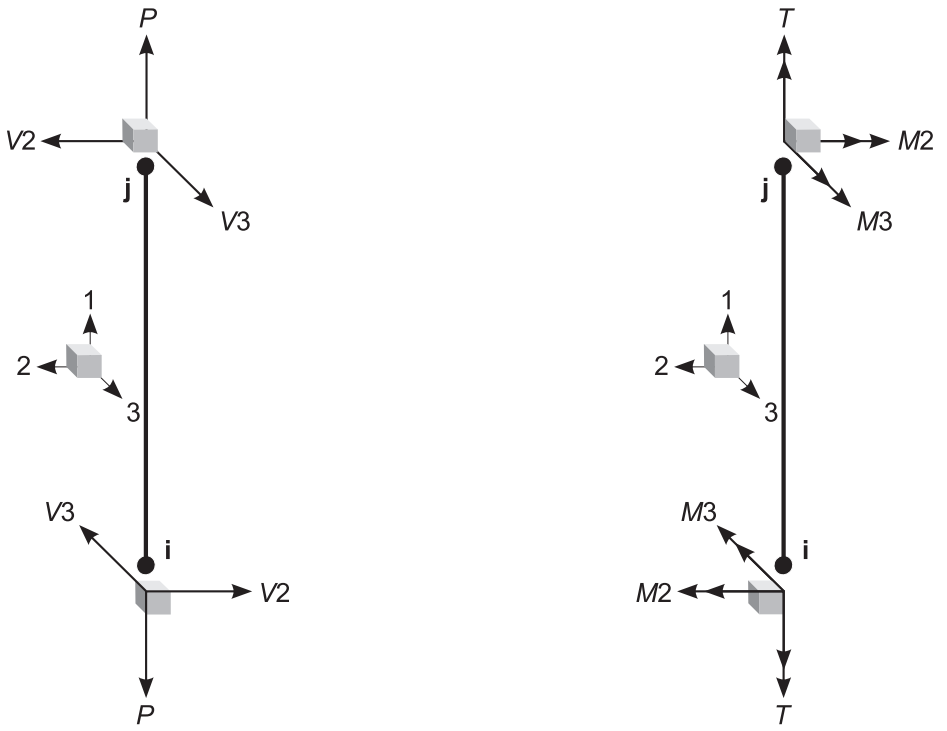


Figure 74
Nllink Element Internal Forces and Moments, Shown Acting at the Joints

Each of these relationships may be zero, linear only, or linear/nonlinear for a given Nlprop. These relationships may be independent or coupled. The forces and moments may be related to the deformation rates (velocities) as well as to the deformations.

Element Internal Forces

The Nllink element internal forces, $P, V2, V3$, and the internal moments, $T, M2, M3$, have the same meaning as for the Frame element. These are illustrated in Figure 74 (page 257). These can be defined in terms of the spring forces and moments as:

- Axial: $P = f_{u1}$
- Shear in the 1-2 plane: $V2 = f_{u2}$, $M3_s = (d - \mathbf{dj2})\, f_{u2}$
- Shear in the 1-3 plane: $V3 = f_{u3}$, $M2_s = (d - \mathbf{dj3})\, f_{u3}$

- Torsion: $T = f_{r1}$
- Pure bending in the 1-3 plane: $M2_b = f_{r2}$
- Pure bending in the 1-2 plane: $M3_b = f_{r3}$

where d is the distance from joint **j**. The total bending-moment resultants $M2$ and $M3$ composed of shear and pure-bending parts:

$$M2 = M2_s + M2_b$$
$$M3 = M3_s + M3_b$$

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

See Topic “Internal Force Output” (page 176) in Chapter “The Frame Element.”

Linear Force-Deformation Relationships

If each of the internal springs behaves linearly, the spring force-deformation relationships can be expressed in matrix form as:

$$\begin{Bmatrix} f_{u1} \\ f_{u2} \\ f_{u3} \\ f_{r1} \\ f_{r2} \\ f_{r3} \end{Bmatrix} = \begin{bmatrix} k_{u1} & 0 & 0 & 0 & 0 & 0 \\ & k_{u2} & 0 & 0 & 0 & 0 \\ & & k_{u3} & 0 & 0 & 0 \\ & & & k_{r1} & 0 & 0 \\ & sym. & & & k_{r2} & 0 \\ & & & & & k_{r3} \end{bmatrix} \begin{Bmatrix} d_{u1} \\ d_{u2} \\ d_{u3} \\ d_{r1} \\ d_{r2} \\ d_{r3} \end{Bmatrix} \qquad \text{(Eqn. 1)}$$

where k_{u1}, k_{u2}, k_{u3}, k_{r1}, k_{r2}, and k_{r3} are the linear stiffness coefficients of the internal springs.

This can be recast in terms of the element internal forces and displacements at joint **j** for a one-joint element as:

(Eqn. 2)

$$\begin{Bmatrix} P \\ V2 \\ V3 \\ T \\ M2 \\ M3 \end{Bmatrix}_{\mathbf{j}} = \begin{bmatrix} k_{u1} & 0 & 0 & 0 & 0 & 0 \\ & k_{u2} & 0 & 0 & 0 & -\mathbf{dj2}\, k_{u2} \\ & & k_{u3} & 0 & -\mathbf{dj3}\, k_{u3} & 0 \\ & & & k_{r1} & 0 & 0 \\ & sym. & & & k_{r2} + \mathbf{dj3}^2\, k_{u3} & 0 \\ & & & & & k_{r3} + \mathbf{dj2}^2\, k_{u2} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ r_1 \\ -r_2 \\ r_3 \end{Bmatrix}_{\mathbf{j}}$$

This relationship also holds for a two-joint element if all displacements at joint **i** are zero.

Similar relationships hold for linear damping behavior, except that the stiffness terms are replaced with damping coefficients, and the displacements are replaced with the corresponding velocities.

Consider an example where the equivalent shear and bending springs are to be computed for a prismatic beam with a section bending stiffness of *EI* in the 1-2 plane. The stiffness matrix at joint **j** for the 1-2 bending plane is:

$$\begin{Bmatrix} V2 \\ M3 \end{Bmatrix}_{\mathbf{j}} = \frac{EI}{L^3} \begin{bmatrix} 12 & -6L \\ -6L & 4L^2 \end{bmatrix} \begin{Bmatrix} u_2 \\ r_3 \end{Bmatrix}_{\mathbf{j}}$$

From this it can be determined that the equivalent shear spring has a stiffness of $k_{u2} = 12\frac{EI}{L^3}$ located at $\mathbf{dj2} = \frac{L}{2}$, and the equivalent pure-bending spring has a stiffness of $k_{r3} = \frac{EI}{L}$.

For an element that possesses a true moment hinge in the 1-2 bending plane, the pure-bending stiffness is zero, and **dj2** is the distance to the hinge. See Figure 75 (page 260).

Linear Effective Stiffness

For each Nlprop you may specify six linear effective-stiffness coefficients, **ke**, one for each of the internal springs.

The **linear effective stiffness** represents the total elastic stiffness for the Nllink element that is used for *all linear analyses*: static analysis, P-delta analysis, modal analysis, moving-load analysis, response-spectrum analysis, harmonic steady-state

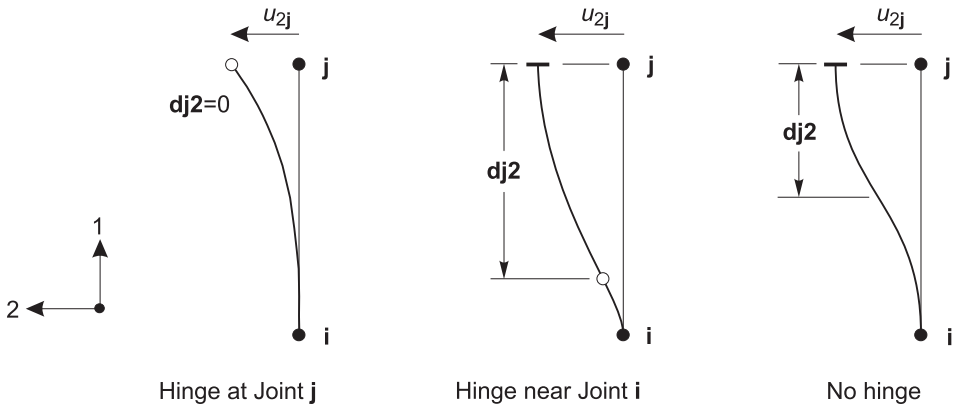


Figure 75
Location of Shear Spring at a Moment Hinge or Point of Inflection

analysis, and linear or periodic time-history analysis. The actual nonlinear properties are ignored for these types of analysis.

The linear effective stiffness is also used for all linear degrees of freedom during a nonlinear time-history analysis.

The effective force-deformation relationships for the Nlprops are given by Equation 1 above with the appropriate values of **ke** substituted for k_{u1}, k_{u2}, k_{u3}, k_{r1}, k_{r2}, and k_{r3}.

The effective stiffness properties are not directly used for nonlinear degrees of freedom during nonlinear time-history analysis. However, these analyses do make use of the vibration modes that are computed based on the effective stiffness. During time integration the behavior of these modes is modified so that the structural response reflects the actual stiffness and other nonlinear parameters specified. The rate of convergence of the nonlinear iteration may be improved by changing the effective stiffness.

Following are some guidelines for selecting the linear effective stiffness. You should deviate from these as necessary to achieve your modeling and analysis goals. In particular, you should consider whether you are more interested in the results to be obtained from linear analyses, or in obtaining modes that are used as the basis for nonlinear time-history analyses.

- When carrying out analyses based on the UBC ‘94 code, the effective stiffness should usually be the code-defined maximum effective stiffness

- For Gap and Hook elements the effective stiffness should usually be zero or **k**, depending on whether the element is likely to be open or closed, respectively, in normal service
- For Damper elements, the effective stiffness should usually be zero
- For other elements, the stiffness should be between zero and **k**
- If you have chosen an artificially large value for **k**, be sure to use a much smaller value for **ke** to help avoid numerical problems in nonlinear time-history analyses

In the above, **k** is the nonlinear stiffness property for a given degree of freedom. See subtopic "Nonlinear Properties" below.

For more information, see Topic "Nonlinear Time-History Analysis" (page 328) in Chapter "Static and Dynamic Analysis."

Linear Effective Damping

For each Nlprop you may specify six linear effective-damping coefficients, **ce**, one for each of the internal springs. By default, each coefficient **ce** is equal to zero.

The **linear effective damping** represents the total viscous damping for the Nllink element that is used for response-spectrum analyses, and for linear and periodic time-history analyses. The actual nonlinear properties are ignored for these types of analysis. Effective damping can be used to represent energy dissipation due to nonlinear damping, plasticity, or friction.

The effective force/deformation-rate relationships for the Nlprops are given by Equation 1 above with the appropriate values of **ce** substituted for k_{u1}, k_{u2}, k_{u3}, k_{r1}, k_{r2}, and k_{r3}, and deformation rates substituted for the corresponding deformations.

The effective damping values are converted to modal damping ratios assuming proportional damping, i.e., the modal cross-coupling damping terms are ignored. These effective modal-damping values are added to any other modal damping that you specify directly. The program will not permit the total damping ratio for any mode to exceed 99.995%.

Important Note: Modal cross-coupling damping terms can be very significant for some structures. A linear analysis based on effective-damping properties may grossly overestimate the amount of damping present in the structure. Nonlinear time-history analysis is strongly recommended to determine the effect of added energy dissipation devices.

Nonlinear time-history analysis does not use the effective damping values since it accounts for energy dissipation in the elements directly, and correctly accounts for the effects of modal cross-coupling.

Nonlinear Properties

The **nonlinear properties** for each Nlprop must be of one of the six types described below. The type determines which degrees of freedom may be nonlinear and the kinds of nonlinear force-deformation relationships available for those degrees of freedom.

Every degree of freedom may have linear effective-stiffness and effective-damping properties specified, as described above in Subtopics "Linear Effective Stiffness" and "Linear Effective Damping."

During *nonlinear time-history analysis*, the nonlinear force-deformation relationships are used at all degrees of freedom for which nonlinear properties were specified. For all other degrees of freedom, the linear effective stiffnesses are used during a nonlinear time-history analysis.

Nonlinear properties are *not* used for any other type of analysis. Linear effective stiffnesses are used for *all* degrees of freedom for all linear analyses.

Each nonlinear force-deformation relationship includes a stiffness coefficient, **k**. This represents the linear stiffness when the nonlinear effect is negligible, e.g., for rapid loading of the Damper; for a closed Gap or Hook; or in the absence of yielding or slipping for the Plastic1, Isolator1, or Isolator2 properties. If **k** is zero, no nonlinear force can be generated for that degree of freedom, with the exception of the pendulum force in the Isolator2 property.

IMPORTANT! You may sometimes be tempted to specify very large values for **k**, particularly for Damper, Gap, and Hook properties. Resist this temptation! If you want to limit elastic deformations in a particular internal spring, it is usually sufficient to use a value of **k** that is from 10^2 to 10^4 times as large as the corresponding stiffness in any connected elements. Larger values of **k** may cause numerical difficulties during solution. See the additional discussion for the Damper property below.

Damper Property

For each deformational degree of freedom you may specify independent damping properties. The damping properties are based on the Maxwell model of viscoelas-

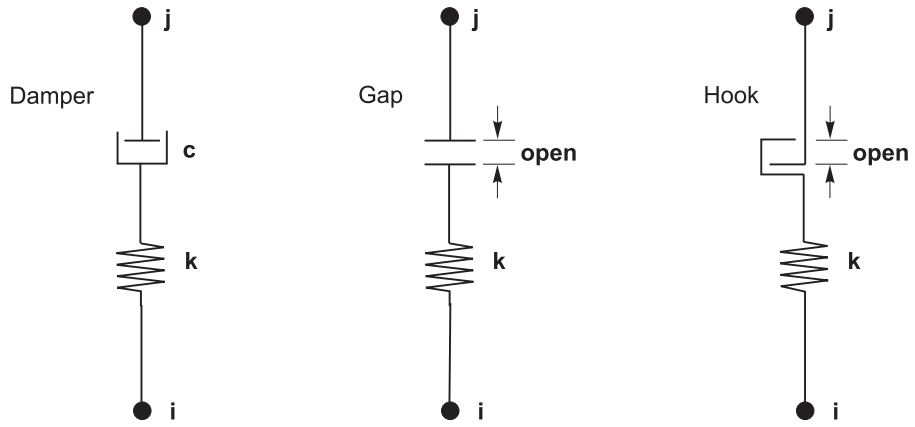


Figure 76
Damper, Gap, and Hook Property Types, Shown for Axial Deformations

ticity (Malvern, 1969) having a nonlinear damper in series with a spring. See Figure 76 (page 263).

If you do not specify nonlinear properties for a degree of freedom, that degree of freedom is linear using the effective stiffness, which may be zero.

The nonlinear force-deformation relationship is given by:

$$f = \mathbf{k}\, d_k = \mathbf{c}\, \dot{d}_c^{\,\mathbf{cexp}}$$

where **k** is the spring constant, **c** is the damping coefficient, **cexp** is the damping exponent, d_k is the deformation across the spring, and $\dot{d}_c$ is the deformation rate across the damper. The damping exponent must be positive; the practical range is between 0.2 and 2.0. The spring and damping deformations sum to the total internal deformation:

$$d = d_k + d_c$$

If pure damping behavior is desired, the effect of the spring can be made negligible by making it sufficiently stiff. The spring stiffness should be large enough so that the characteristic time of the spring-dashpot system, given by $\tau = \mathbf{c} / \mathbf{k}$ (when **cexp** = l), is an order of magnitude smaller than the size of the load steps. The load steps are the time intervals over which the load is changing. The stiffness should not be made excessively large or else numerical sensitivity may result.

Gap Property

For each deformational degree of freedom you may specify independent gap ("compression-only") properties. See Figure 76 (page 263).

All internal deformations are independent. The opening or closing of a gap for one deformation does not affect the behavior of the other deformations.

If you do not specify nonlinear properties for a degree of freedom, that degree of freedom is linear using the effective stiffness, which may be zero.

The nonlinear force-deformation relationship is given by:

$$f = \begin{cases} \mathbf{k}\,(d + \mathbf{open}) & \text{if } d + \mathbf{open} < 0 \\ 0 & \text{otherwise} \end{cases}$$

where **k** is the spring constant, and **open** is the initial gap opening, which must be zero or positive.

Hook Property

For each deformational degree of freedom you may specify independent hook ("tension-only") properties. See Figure 76 (page 263).

All internal deformations are independent. The opening or closing of a hook for one deformation does not affect the behavior of the other deformations.

If you do not specify nonlinear properties for a degree of freedom, that degree of freedom is linear using the effective stiffness, which may be zero.

The nonlinear force-deformation relationship is given by:

$$f = \begin{cases} \mathbf{k}\,(d - \mathbf{open}) & \text{if } d - \mathbf{open} > 0 \\ 0 & \text{otherwise} \end{cases}$$

where **k** is the spring constant, and **open** is the initial hook opening, which must be zero or positive.

Plastic1 Property

For each deformational degree of freedom you may specify independent uniaxial-plasticity properties. The plasticity model is based on the hysteretic behavior proposed by Wen (1976). See Figure 77 (page 265).

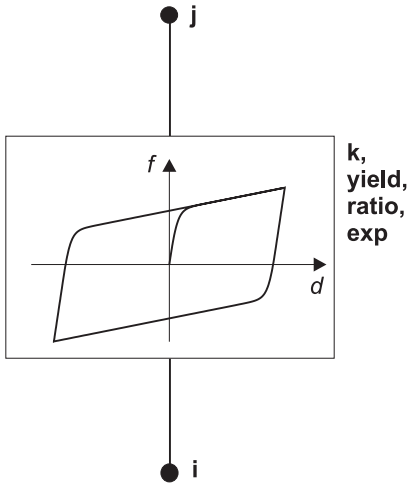


Figure 77
Plastic1 Property Type for Uniaxial Deformation

All internal deformations are independent. The yielding at one degree of freedom does not affect the behavior of the other deformations.

If you do not specify nonlinear properties for a degree of freedom, that degree of freedom is linear using the effective stiffness, which may be zero.

The nonlinear force-deformation relationship is given by:

$$f = \textbf{ratio}\ \textbf{k}\ d + (1 - \textbf{ratio})\ \textbf{yield}\ z$$

where **k** is the elastic spring constant, **yield** is the yield force, **ratio** is the specified ratio of post-yield stiffness to elastic stiffness (**k**), and z is an internal hysteretic variable. This variable has a range of $|z| \le 1$, with the yield surface represented by $|z| = 1$. The initial value of z is zero, and it evolves according to the differential equation:

$$\dot{z} = \frac{\textbf{k}}{\textbf{yield}} \begin{cases} \dot{d}\,(1 - |z|^{\textbf{exp}}) & \text{if } \dot{d}\,z > 0 \\ \dot{d} & \text{otherwise} \end{cases}$$

where **exp** is an exponent greater than or equal to unity. Larger values of this exponent increases the sharpness of yielding as shown in Figure 78 (page 266). The practical limit for **exp** is about 20. The equation for $\dot{z}$ is equivalent to Wen's model with $A = 1$ and $\alpha = \beta = 0.5$.

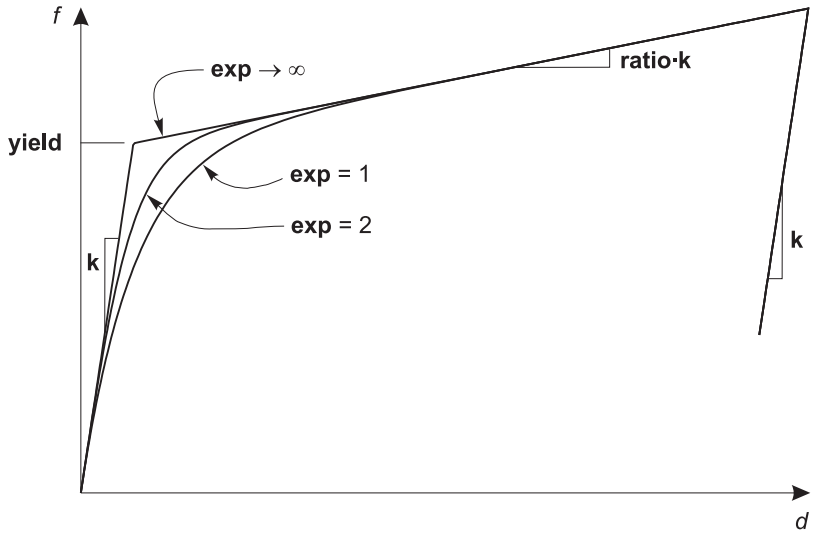


Figure 78
Definition of Parameters for Plastic1 Property

Isolator1 Property

This is a biaxial hysteretic isolator that has coupled plasticity properties for the two shear deformations, and linear effective-stiffness properties for the remaining four deformations. The plasticity model is based on the hysteretic behavior proposed by Wen (1976), and Park, Wen and Ang (1986), and recommended for base-isolation analysis by Nagarajaiah, Reinhorn and Constantinou (1991). See Figure 79 (page 267).

For each *shear* deformation degree of freedom you may independently specify either linear or nonlinear behavior:

- If both shear degrees of freedom are nonlinear, the coupled force-deformation relationship is given by:

$$f_{u2} = \textbf{ratio2}\ \textbf{k2}\ d_{u2} + (1 - \textbf{ratio2})\ \textbf{yield2}\ z_2$$
$$f_{u3} = \textbf{ratio3}\ \textbf{k3}\ d_{u3} + (1 - \textbf{ratio3})\ \textbf{yield3}\ z_3$$

  where **k2** and **k3** are the elastic spring constants, **yield2** and **yield3** are the yield forces, **ratio2** and **ratio3** are the ratios of post-yield stiffnesses to elastic stiffnesses (**k2** and **k3**), and z_2 and z_3 are internal hysteretic variables. These vari-

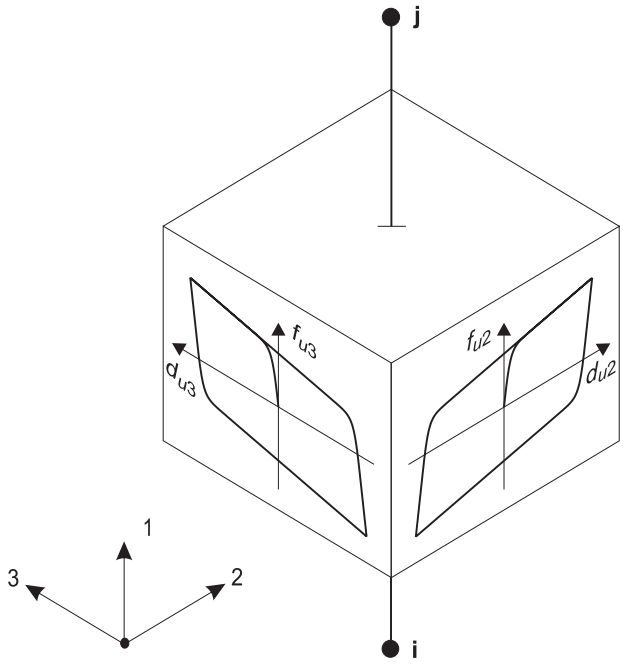


Figure 79
Isolator1 Property for Biaxial Shear Deformation

ables have a range of $\sqrt{{z_2}^2 + {z_3}^2} \le 1$, with the yield surface represented by $\sqrt{{z_2}^2 + {z_3}^2} = 1$. The initial values of z_2 and z_3 are zero, and they evolve according to the differential equations:

$$\begin{Bmatrix} \dot{z}_2 \\ \dot{z}_3 \end{Bmatrix} = \begin{bmatrix} 1 - a_2 \, {z_2}^2 & -a_3 \, z_2 \, z_3 \\ -a_2 \, z_2 \, z_3 & 1 - a_3 \, {z_3}^2 \end{bmatrix} \begin{Bmatrix} \dfrac{\mathbf{k2}}{\mathbf{yield2}} \, \dot{d}_{u2} \\ \dfrac{\mathbf{k3}}{\mathbf{yield3}} \, \dot{d}_{u3} \end{Bmatrix}$$

Where:

$$a_2 = \begin{cases} 1 & \text{if } \dot{d}_{u2} \, z_2 > 0 \\ 0 & \text{otherwise} \end{cases}$$

$$a_3 = \begin{cases} 1 & \text{if } \dot{d}_{u3}\, z_3 > 0 \\ 0 & \text{otherwise} \end{cases}$$

These equations are equivalent to those of Park, Wen and Ang (1986) with $A = 1$ and $\beta = \gamma = 0.5$.

- If only one shear degree of freedom is nonlinear, the above equations reduce to the uniaxial plasticity behavior of the Plastic1 property with **exp** = 2 for that degree of freedom.

A linear spring relationship applies to the axial deformation, the three moment deformations, and to any shear deformation without nonlinear properties. All linear degrees of freedom use the corresponding effective stiffness, which may be zero.

Isolator2 Property

This is a biaxial friction-pendulum isolator that has coupled friction properties for the two shear deformations, post-slip stiffness in the shear directions due the pendulum radii of the slipping surfaces, gap behavior in the axial direction, and linear effective-stiffness properties for the three moment deformations. See Figure 80 (page 269).

The friction model is based on the hysteretic behavior proposed by Wen (1976), and Park, Wen and Ang (1986), and recommended for base-isolation analysis by Nagarajaiah, Reinhorn and Constantinou (1991). The pendulum behavior is as recommended by Zayas and Low (1990).

The friction forces and pendulum forces are directly proportional to the compressive axial force in the element. The element cannot carry axial tension.

The axial force, *P*, is *always* nonlinear, and is given by:

$$P \equiv f_{u1} = \begin{cases} \mathbf{k1}\, d_{u1} & \text{if } d_{u1} < 0 \\ 0 & \text{otherwise} \end{cases}$$

Stiffness **k1** must be positive in order to generate nonlinear shear force in the element.

For each *shear* deformation degree of freedom you may independently specify either linear or nonlinear behavior:

- If both shear degrees of freedom are nonlinear, the friction and pendulum effects for each shear deformation act in parallel:

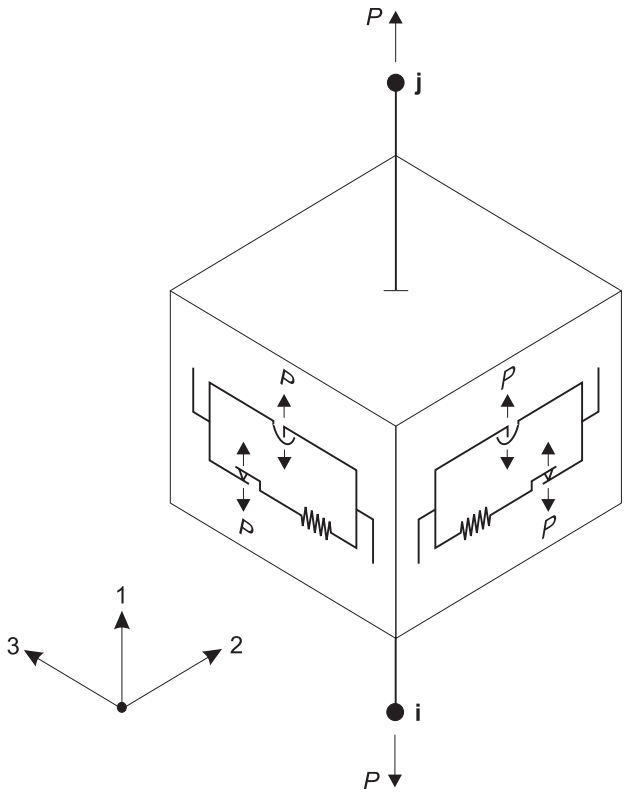


Figure 80
Isolator2 Property for Friction-Pendulum Shear Behavior

$$f_{u2} = f_{u2f} + f_{u2p}$$
$$f_{u3} = f_{u3f} + f_{u3p}$$

The frictional force-deformation relationships are given by:

$$f_{u2f} = -P\,\mu_2\,z_2$$
$$f_{u3f} = -P\,\mu_3\,z_3$$

where μ_2 and μ_3 are friction coefficients, and z_2 and z_3 are internal hysteretic variables. The friction coefficients are velocity-dependent according to:

$$\mu_2 = \mathbf{fast2} - (\mathbf{fast2} - \mathbf{slow2})\,e^{-r\,v}$$
$$\mu_3 = \mathbf{fast3} - (\mathbf{fast3} - \mathbf{slow3})\,e^{-r\,v}$$

where **slow2** and **slow3** are the friction coefficients at zero velocity, **fast2** and **fast3** are the friction coefficients at fast velocities, v is the resultant velocity of sliding:

$$v = \sqrt{\dot{d}_{u2}{}^2 + \dot{d}_{u3}{}^2}$$

r is an effective inverse velocity given by:

$$r = \frac{\mathbf{rate2}\ \dot{d}_{u2}{}^2 + \mathbf{rate3}\ \dot{d}_{u3}{}^2}{v^2}$$

and **rate2** and **rate3** are the inverses of characteristic sliding velocities. For a Teflon-steel interface the coefficient of friction normally increases with sliding velocity (Nagarajaiah, Reinhorn, and Constantinou, 1991).

The internal hysteretic variables have a range of $\sqrt{z_2{}^2 + z_3{}^2} \le 1$, with the yield surface represented by $\sqrt{z_2{}^2 + z_3{}^2} = 1$. The initial values of z_2 and z_3 are zero, and they evolve according to the differential equations:

$$\begin{Bmatrix} \dot{z}_2 \\ \dot{z}_3 \end{Bmatrix} = \begin{bmatrix} 1 - a_2\ z_2{}^2 & -a_3\ z_2\ z_3 \\ -a_2\ z_2\ z_3 & 1 - a_3\ z_3{}^2 \end{bmatrix} \begin{Bmatrix} \dfrac{\mathbf{k2}}{P\,\mu_2}\ \dot{d}_{u2} \\ \dfrac{\mathbf{k3}}{P\,\mu_3}\ \dot{d}_{u3} \end{Bmatrix}$$

where **k2** and **k3** are the elastic shear stiffnesses of the slider in the absence of sliding, and

$$a_2 = \begin{cases} 1 & \text{if } \dot{d}_{u2}\ z_2 > 0 \\ 0 & \text{otherwise} \end{cases}$$

$$a_3 = \begin{cases} 1 & \text{if } \dot{d}_{u3}\ z_3 > 0 \\ 0 & \text{otherwise} \end{cases}$$

These equations are equivalent to those of Park, Wen and Ang (1986) with $A = 1$ and $\beta = \gamma = 0.5$.

This friction model permits some sliding at all non-zero levels of shear force; the amount of sliding becomes much larger as the shear force approaches the "yield" value of $P\,\mu$. Sliding at lower values of shear force can be minimized by using larger values of the elastic shear stiffnesses.

The pendulum force-deformation relationships are given by:

$$f_{u2p} = -P\,\frac{d_{u2}}{\textbf{radius2}}$$

$$f_{u3p} = -P\,\frac{d_{u3}}{\textbf{radius3}}$$

A zero radius indicates a flat surface, and the corresponding shear force is zero. Normally the radii in the two shear directions will be equal (spherical surface), or one radius will be zero (cylindrical surface). However, it is permitted to specify unequal non-zero radii.

- If only one shear degree of freedom is nonlinear, the above frictional equations reduce to:

$$f_f = -P\,\mu\,z$$

$$\mu = \textbf{fast} - (\textbf{fast} - \textbf{slow})\,e^{-\textbf{rate}\,\dot{d}}$$

$$\dot{z} = \frac{\textbf{k}}{P\,\mu}\begin{cases}\dot{d}\,(1-z^2) & \text{if } \dot{d}\,z > 0\\ \dot{d} & \text{otherwise}\end{cases}$$

The above pendulum equation is unchanged for the nonlinear degree of freedom.

A linear spring relationship applies to the three moment deformations, and to any shear deformation without nonlinear properties. All linear degrees of freedom use the corresponding effective stiffness, which may be zero. The axial degree of freedom is always nonlinear for nonlinear time-history analyses.

Nonlinear Deformation Loads

A **nonlinear deformation load** is a set of forces and/or moments on the structure that activates a nonlinear internal deformation of an Nllink element. A nonlinear internal deformation is an Nllink internal deformation for which nonlinear properties have been specified.

Nonlinear deformation loads are used as starting load vectors for Ritz-vector analysis. Their purpose is to generate Modes that can adequately represent nonlinear behavior when performing nonlinear time-history analyses. A separate nonlinear deformation load should be used for each nonlinear internal deformation of each Nllink element.

When requesting a Ritz-vector analysis, you may specify that the program use built-in nonlinear deformation loads, or you may define your own Load Cases for this purpose. In the latter case you may need up to six of these Load Cases per Nllink element in the model.

The built-in nonlinear deformation loads for a single two-joint Nllink element are shown in Figure 81 (page 273). Each set of forces and/or moments is self-equilibrating. This tends to localize the effect of the load, usually resulting in a better set of Ritz-vectors. For a single-joint element, only the forces and/or moments acting on joint **j** are needed.

It is *strongly recommended* that mass or mass moment of inertia be present at each degree of freedom that is acted upon by a force or moment from a nonlinear deformation load. This is needed to generate the appropriate Ritz vectors.

For more information:

- See Topic “Internal Deformations” (page 253) in this chapter.
- See Topic “Nlprop Properties” (page 254) in this chapter.
- See Topic “Mass” (page 272) in this chapter.
- See Topic “Ritz-Vector Analysis” (page 299) in Chapter “Static and Dynamic Analysis.”
- See Topic “Nonlinear Time-History Analysis” (page 328) in Chapter “Static and Dynamic Analysis.”

Mass

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

For each Nlprop, you may specify a total translational mass, **m**. Half of the mass is assigned to the three translational degrees of freedom at each of the element’s one or two joints. For single-joint elements, half of the mass is assumed to be grounded.

You may additionally specify total rotational mass moments of inertia, **mr1**, **mr2**, and **mr3**, about the three local axes of each element. Half of each mass moment of inertia is assigned to each of the element’s one or two joints. For single-joint elements, half of each mass moment of inertia is assumed to be grounded.

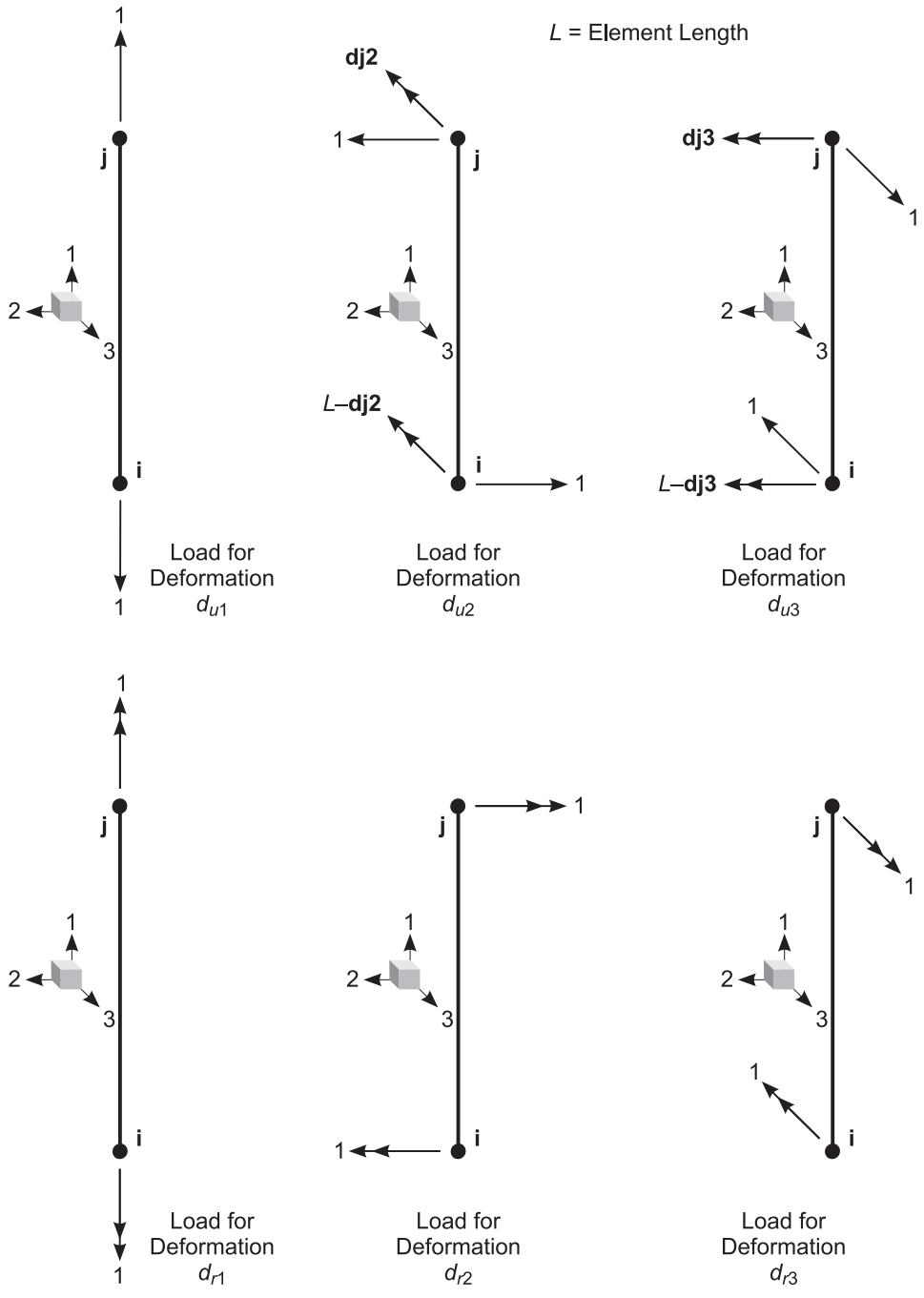


Figure 81
Built-in Nonlinear Deformation Loads for a Two-joint Nllink Element

The rotational inertias are defined in the element local coordinate system, but will be transformed by the program to the local coordinate systems for joint **i** and **j**. If the three inertias are not equal and element local axes are not parallel to the joint local axes, then cross-coupling inertia terms will be generated during this transformation. These will be discarded by the program, resulting in some error.

It is *strongly recommended* that there be mass corresponding to each nonlinear deformation load in order to generate appropriate Ritz vectors for nonlinear time-history analysis. Note that rotational inertia is needed as well as translational mass for nonlinear shear deformations if either the element length or **dj** is non-zero.

For more information:

- See Chapter "Static and Dynamic Analysis" (page 291).
- See Topic "Nonlinear Deformation Loads" (page 271) in this chapter.

Self-Weight Load

Self-Weight Load activates the self-weight of all elements in the model. For each Nlprop, a total self-weight, **w**, may be defined. Half of this weight is assigned to each joint of each Nllink element using that Nlprop. For single-joint elements, half of the weight is assumed to be grounded.

Self-Weight Load always acts downward, in the global –Z direction. You may scale the self-weight by a single scale factor that applies equally to all elements in the structure.

See Topic "Self-Weight Load" (page 283) in Chapter "Load Cases" for more information.

Gravity Load

Gravity Load can be applied to each Nllink element to activate the self-weight of the element. Using Gravity Load, the self-weight can be scaled and applied in any direction. Different scale factors and directions can be applied to each element.

If all elements are to be loaded equally and in the downward direction, it is more convenient to use Self-Weight Load.

For more information:

- See Topic “Self-Weight Load” (page 274) in this chapter for the definition of self-weight for the Nllink element.
- See Topic “Gravity Load” (page 284) in Chapter “Load Cases.”

Internal Force and Deformation Output

Nllink element internal forces and deformations are computed for the following types of Analysis Cases: Loads, Modes, Specs, Histories, and Combos. History results are the maximum and minimum values attained during the analysis, and are only available if specifically requested in the definition of the individual History cases. No results are available for Moving Loads or Combos that contain Moving Loads.

Results for linear analyses are based upon the linear effective-stiffness and effective-damping properties and do not include any nonlinear effects. Only the results for nonlinear Histories include nonlinear behavior.

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

The element internal forces were defined in Subtopic “Element Internal Forces” (page 257) of this chapter. The internal deformations were defined in Topic “Internal Deformations” (page 253) of this chapter.

Nllink element internal forces and deformations are printed in the output file under the heading:

```
NLLINK ELEMENT INTERNAL FORCES & DEFORMS
```

All results for a single element are printed together. Elements are presented in alpha-numeric order. For each Analysis Case internal forces are printed at the element joints, followed by the internal deformations.

The element internal forces are labeled P, V2, V3, T, M2, and M3 in the output. The internal deformations are labeled U1, U2, U3, R1, R2, and R3 in the output, corresponding to the values of d_{u1}, d_{u2}, d_{u3}, d_{r1}, d_{r2}, and d_{r3}.

Typical Nllink element output is shown in Figure 82 (page 276).

By default, no internal forces or deformations are printed. You may specify which Analysis Cases are to be printed. The internal forces and deformations can always be displayed using the SAP2000 graphical interface even if no results have been selected for printing.

```
N L L I N K   E L E M E N T   I N T E R N A L   F O R C E S   &   D E F O R M S

ELEM      N1 ================== TYPE = TWO-JOINT LINK

MODE       1 ------------------

  JOINT           P          V2          V3          T          M2          M3
      8     .000000   -7339.063     .000000     .000000     .000000     .000000
     15     .000000   -7339.063     .000000     .000000     .000000     .000000

  JOINT          U1          U2          U3          R1          R2          R3
     15    5.70E-16   -0.733906     .000000     .000000     .000000   -0.000139

MODE       2 ------------------

  JOINT           P          V2          V3          T          M2          M3
      8     .000000  -15370.177     .000000     .000000     .000000     .000000
     15     .000000  -15370.177     .000000     .000000     .000000     .000000

  JOINT          U1          U2          U3          R1          R2          R3
     15    9.20E-16   -1.537018     .000000     .000000     .000000    0.000489

ELEM      N2 ================== TYPE = TWO-JOINT LINK

MODE       1 ------------------

  JOINT           P          V2          V3          T          M2          M3
      5     .000000   -5608.591     .000000     .000000     .000000     .000000
     14     .000000   -5608.591     .000000     .000000     .000000     .000000

  JOINT          U1          U2          U3          R1          R2          R3
     14    1.36E-16   -0.560859     .000000     .000000     .000000   -0.000184

MODE       2 ------------------

  JOINT           P          V2          V3          T          M2          M3
      5     .000000   11274.612     .000000     .000000     .000000     .000000
     14     .000000   11274.612     .000000     .000000     .000000     .000000

  JOINT          U1          U2          U3          R1          R2          R3
     14   -2.50E-17    1.127461     .000000     .000000     .000000    0.000779
```

...Output Edited for Brevity...

Figure 82

Nllink Element Internal Forces and Deformations in the Output File

For more information:

- See Chapter “Load Cases” (page 279).
- See Chapter “Static and Dynamic Analysis” (page 291).
- See Chapter “The Output Files” (page 395).

Joint Force Output

The **Nllink element joint forces** are concentrated forces and moments acting at the joints of the element that represent the effect of the rest of the structure upon the element.

The Nllink element joint forces do not include any nonlinear effects.

Element joint forces must not be confused with internal forces and moments which, like stresses, act *throughout* the element, not just at the joints.

See Topic “Element Joint Force Output” (page 402) in Chapter “The Output Files” for more information.

Chapter XV

Load Cases

A **Load Case** is a specified spatial distribution of forces, displacements, temperatures, and other effects that act upon the structure. The Load Cases form the basis for most of the loading used by the different types of analyses that can be performed by the program.

Basic Topics for All Users

- Overview
- Load Cases for Analysis
- Defining Load Cases
- Coordinate Systems and Load Components
- Force Load
- Restraint Displacement Load
- Spring Displacement Load
- Self-Weight Load
- Concentrated Span Load
- Distributed Span Load
- Uniform Load

Advanced Topics

- Gravity Load
- Prestress Load
- Surface Pressure Load
- Pore Pressure Load
- Temperature Load
- Reference Temperature
- Rotate Load

Overview

Each Load Case may consist of an arbitrary combination of the available load types:

- Concentrated forces and moments acting at the joints
- Displacements of the grounded ends of restraints at the joints
- Displacements of the grounded ends of springs at the joints
- Self-weight and/or gravity acting on all element types
- Concentrated or distributed forces and moments acting on the Frame elements
- Distributed forces acting on the Shell elements
- Surface pressure acting on the Shell, Plane, Asolid, and Solid elements
- Pore pressure acting on the Plane, Asolid, and Solid elements
- Thermal expansion acting on the Frame, Shell, Plane, Asolid, and Solid elements
- Prestress cable forces acting on Frame elements
- Centrifugal forces acting on Asolid elements

For practical purposes, it is usually most convenient to restrict each Load Case to a single type of load, using Combos and Histories to create more complicated combinations. Any number of Load Cases may be specified.

Load Cases for Analysis

The Load Cases are the basic loadings used by the different analysis types as follows:

- Static analysis — The program automatically computes the linear response to each Load Case unless steady-state analysis has been requested.
- Steady-state analysis — Each Load Case represents a spatial load distribution applied at a harmonic frequency, **cyc**. The program automatically computes the linear response to each Load Case.
- P-delta analysis — A specified linear combination of the Load Cases is applied to the structure to determine the P-delta effect.
- Vibration-mode analysis — Selected Load Cases may be used to bias the Ritz vectors obtained. Load Cases do not affect eigenvectors.
- Time-history analysis — The Load Cases may be multiplied by time-dependent functions and combined to form a complete load that varies with time and space.

For more information:

- See Chapter “Static and Dynamic Analysis” (page 291).
- See Chapter “P-Delta Analysis” (page 337).

Defining Load Cases

You specify a unique label, **name**, for each Load Case to be defined. Within each Load Case, any number of joints or elements may be loaded by any number of different load types.

Loads may be applied to the joints and elements, individually or in arrays. Multiple applications of the same type of load to a joint or element are additive. See the individual topics that follow for how each type of load adds.

Loads of a given type may be removed from the joints and elements, individually or in arrays. Subsequent applications of that same type of load to the joints or elements are additive from zero.

Coordinate Systems and Load Components

Certain types of loads, such as temperature and pressure, are scalars that are independent of any coordinate system. Forces and displacements, however, are vectors whose components depend upon the coordinate system in which they are specified.

Vector loads may be specified with respect to any fixed coordinate system. The fixed coordinate system to be used is specified as **csys**. If **csys** is zero (the default), the global system is used. Otherwise **csys** refers to an Alternate Coordinate System.

The X, Y, and Z components of a force or translation in a fixed coordinate system are specified as **ux**, **uy**, and **uz**, respectively. The X, Y, and Z components of a moment or rotation are specified as **rx**, **ry**, and **rz**, respectively.

Most vector loads may also be specified with respect to joint and element local coordinate systems. Unlike fixed coordinate systems, the local coordinate systems may vary from joint to joint and element to element.

The 1, 2, and 3 components of a force or translation in a local coordinate system are specified as **u1**, **u2**, and **u3**, respectively. The 1, 2, and 3 components of a moment or rotation are specified as **r1**, **r2**, and **r3**, respectively.

You may use a different coordinate system, as convenient, for each application of a given type of load to a particular joint or element. The program will convert all these loads to a single coordinate system and add them together to get the total load.

See Chapter “Coordinate Systems” (page 21) for more information.

Force Load

Force Load applies concentrated forces and moments to the joints. You may specify components **ux**, **uy**, **uz**, **rx**, **ry**, and **rz** in any fixed coordinate system **csys**, and components **u1**, **u2**, **u3**, **r1**, **r2**, and **r3** in the joint local coordinate system. Joint Patterns can be used to specify the distribution of component values. Force values are additive after being converted to the joint local coordinate system.

See Topic “Force Load” (page 93) in Chapter “Joints and Degrees of Freedom” for more information.

Restraint Displacement Load

Restraint Displacement Load applies specified ground displacements (translations and rotations) along the restrained degrees of freedom at the joints. You may specify components **ux**, **uy**, **uz**, **rx**, **ry**, and **rz** in any fixed coordinate system **csys**, and components **u1**, **u2**, **u3**, **r1**, **r2**, and **r3** in the joint local coordinate system. Joint Patterns can be used to specify the distribution of component values. Displacement values are additive after being converted to the joint local coordinate system.

See Topic “Restraint Displacement Load” (page 93) in Chapter “Joints and Degrees of Freedom” for more information.

Spring Displacement Load

Spring Displacement Load applies specified displacements (translations and rotations) at the grounded end of spring supports at the joints. You may specify components **ux**, **uy**, **uz**, **rx**, **ry**, and **rz** in any fixed coordinate system **csys**, and components **u1**, **u2**, **u3**, **r1**, **r2**, and **r3** in the joint local coordinate system. Joint Patterns can be used to specify the distribution of component values. Displacement values are additive after being converted to the joint local coordinate system.

See Topic “Spring Displacement Load” (page 95) in Chapter “Joints and Degrees of Freedom” for more information.

Self-Weight Load

Self-Weight Load activates the self-weight of all elements in the model. Self-weight always acts downward, in the global –Z direction. You may scale the self-weight by a single scale factor that applies to the whole structure. No Self-Weight Load can be produced by an element with zero weight.

For more information:

- See Topic “Upward and Horizontal Directions” (page 23) in Chapter “Coordinate Systems.”
- See Topic “Self-Weight Load” (page 168) in Chapter “The Frame Element.”
- See Topic “Self-Weight Load” (page 195) in Chapter “The Shell Element.”
- See Topic “Self-Weight Load” (page 213) in Chapter “The Plane Element.”
- See Topic “Self-Weight Load” (page 228) in Chapter “The Asolid Element.”

- See Topic “Self-Weight Load” (page 239) in Chapter “The Solid Element.”
- See Topic “Self-Weight Load” (page 274) in Chapter “The Nllink Element.”

Gravity Load

Gravity Load activates the self-weight of the Frame, Shell, Plane, Asolid, Solid, and Nllink elements. For each element to be loaded, you may specify the gravitational multipliers **ux**, **uy**, and **uz** in any fixed coordinate system **csys**. Multiplier values are additive after being converted to the global coordinate system.

Each element produces a Gravity Load, having three components in system **csys**, equal to its self-weight multiplied by the factors **ux**, **uy**, and **uz**. This load is apportioned to each joint of the element. For example, if **uz** = –2, twice the self-weight is applied to the structure acting in the negative Z direction of system **csys**. No Gravity Load can be produced by an element with zero weight.

The difference between Self-Weight Load and Gravity Load is:

- Self-Weight Load acts equally on all elements of the structure and always in the global –Z direction
- Gravity Load may have a different magnitude and direction for each element in the structure

Both loads are proportional to the self-weight of the individual elements.

For more information:

- See Topic “Gravity Load” (page 168) in Chapter “The Frame Element.”
- See Topic “Gravity Load” (page 195) in Chapter “The Shell Element.”
- See Topic “Gravity Load” (page 214) in Chapter “The Plane Element.”
- See Topic “Gravity Load” (page 228) in Chapter “The Asolid Element.”
- See Topic “Gravity Load” (page 239) in Chapter “The Solid Element.”
- See Topic “Gravity Load” (page 274) in Chapter “The Nllink Element.”

Concentrated Span Load

Concentrated Span Load applies concentrated forces and moments at arbitrary locations on Frame elements. You may specify components **ux**, **uy**, **uz**, **rx**, **ry**, and **rz** in any fixed coordinate system **csys**, and components **u1**, **u2**, **u3**, **r1**, **r2**, and **r3** in

the Frame element local coordinate system. Force values are additive after being converted to the Frame element local coordinate system.

See Topic “Concentrated Span Load” (page 169) in Chapter “The Frame Element” for more information.

Distributed Span Load

Distributed Span Load applies distributed forces and moments at arbitrary locations on Frame elements. You may specify components **ux**, **uy**, **uz**, **rx**, **ry**, and **rz** in any fixed coordinate system **csys**, and components **u1**, **u2**, **u3**, **r1**, **r2**, and **r3** in the Frame element local coordinate system. Force values are additive after being converted to the Frame element local coordinate system.

See Topic “Distributed Span Load” (page 169) in Chapter “The Frame Element” for more information.

Prestress Load

Prestress Load applies the forces and moments caused by prestressing cables in Frame elements. For each element you may specify a scale factor, **p**, that multiplies the effect of all cables that act on that element. The scale factors are additive.

See Topic “Prestress Load” (page 174) in Chapter “The Frame Element” for more information.

Uniform Load

Uniform Load applies uniformly distributed forces to the midsurface of Shell elements. You may specify components **ux**, **uy**, and **uz** in any fixed coordinate system **csys**, and components **u1**, **u2**, and **u3** in the element local coordinate system. Force values are additive after being converted to the element local coordinate system.

See Topic “Uniform Load” (page 195) in Chapter “The Shell Element” for more information.

Surface Pressure Load

Surface Pressure Load applies an external pressure to any of the outer faces of the Shell, Plane, Asolid, and Solid elements. The load on each face of an element is specified independently.

You may specify pressures, **p**, that are uniform over an element face or interpolated from pressure values given by Joint Patterns. Hydrostatic pressure fields may easily be specified using Joint Patterns. Pressure values are additive.

For more information:

- See Topic “Surface Pressure Load” (page 196) in Chapter “The Shell Element.”
- See Topic “Surface Pressure Load” (page 214) in Chapter “The Plane Element.”
- See Topic “Surface Pressure Load” (page 229) in Chapter “The Asolid Element.”
- See Topic “Surface Pressure Load” (page 239) in Chapter “The Solid Element.”

Pore Pressure Load

Pore Pressure Load models the drag and buoyancy effects of a fluid *within* a solid medium, such as the effect of water upon the solid skeleton of a soil. Pore Pressure Load may be used with Shell, Asolid, and Solid elements.

Scalar fluid-pressure values are given at the element joints by Joint Patterns, and interpolated over the element. These pressure values may typically be obtained by flow-net analysis, such as illustrated in Figure 83 (page 287). Hydrostatic pressure fields may easily be specified using Joint Patterns. Pressure values are additive.

The total force acting on the element is the integral of the gradient of this pressure field over the volume of the element. This force is apportioned to each of the joints of the element. The forces are typically directed from regions of high pressure toward regions of low pressure.

Note that although pressures are specified, it is the *pressure gradient* over an element that causes the load. Thus a uniform pressure field over an element will cause no load. Pressure differences *between* elements also cause no load. For this reason, it is important that the pore-pressure field be continuous over the structure.

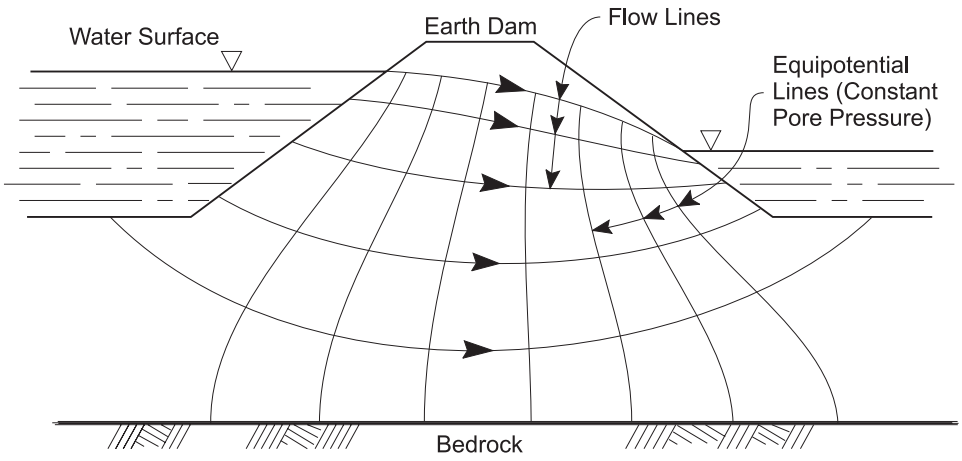


Figure 83
Flow-net Analysis of an Earth Dam to Obtain Pore Pressures

The displacements, stresses, and reactions due to Pore Pressure Load represent the response of the solid medium, not that of the combined fluid and solid structure. In the case of soil, the stresses obtained are the usual "effective stresses" of soil mechanics (Terzaghi and Peck, 1967). Note, however, that the *total* soil weight and mass density should be used for the material properties.

For more information:

- See Topic "Pore Pressure Load" (page 214) in Chapter "The Plane Element."
- See Topic "Pore Pressure Load" (page 229) in Chapter "The Asolid Element."
- See Topic "Pore Pressure Load" (page 240) in Chapter "The Solid Element."

Temperature Load

Temperature Load creates thermal strains in the Frame, Shell, Plane, Asolid, and Solid elements. These strains are given by the product of the Material coefficients of thermal expansion and the *temperature change* of the element. The temperature change is measured from the element Reference Temperature to the element Load Temperature.

The Load Temperature may be different for each Load Case. You may specify Load temperatures, **t**, that are uniform over an element or that are interpolated from

values given by Joint Patterns. Temperature values are additive and add from zero, not from the Reference Temperature.

In any Load Case where the Load Temperature is not specified for a given element, it is assumed to be equal to the Reference Temperature, and hence causes no load. Note, however, that when Load Temperatures are specified they add from zero, not from the Reference Temperature.

Load Temperature gradients may also be specified in the two transverse directions of the Frame element, and in the thickness direction of the Shell element. These gradients induce bending strains in the elements. Temperature gradients are specified as the *change* in temperature per unit length. The Reference Temperature gradients are always assumed to be zero.

The Load Temperature gradients may be different for each Load Case. You may specify temperature gradients, **t2** and/or **t3**, that are uniform over an element or that are interpolated from values given by Joint Patterns. Temperature gradient values are additive.

For more information:

- See Topic “Temperature Load” (page 173) in Chapter “The Frame Element.”
- See Topic “Temperature Load” (page 197) in Chapter “The Shell Element.”
- See Topic “Temperature Load” (page 215) in Chapter “The Plane Element.”
- See Topic “Temperature Load” (page 229) in Chapter “The Asolid Element.”
- See Topic “Temperature Load” (page 240) in Chapter “The Solid Element.”

Reference Temperature

Each Frame, Shell, Plane, Asolid, and Solid element has a single Reference Temperature field that is used for all Load Cases. This is the temperature at which the unloaded element is assumed to be stress-free. The Reference Temperature is used as part of the Temperature Load.

You may specify Reference Temperatures, **t**, that are uniform over an element or that are interpolated from values given by Joint Patterns. Temperature values are additive.

If no Reference Temperature is given for an element, a value of zero is assumed.

The Reference Temperature gradients for the Frame and Shell elements are always assumed to be zero.

Rotate Load

Rotate Load applies centrifugal force to Asolid elements. You may specify an angular velocity, **r**, for each element. The centrifugal force is proportional to the square of the angular velocity. The angular velocities are additive. The load on the element is computed from the total angular velocity.

See Topic “Rotate Load” (page 230) in Chapter “The Asolid Element.”

Chapter **XVI**

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. See Chapter “P-Delta Analysis” (page 337) for information on that subject. See Chapter “Bridge Analysis” (page 357) for information on moving-load analysis.

Basic Topics for All Users

- Overview
- Analysis Cases
- Static Analysis
- Acceleration Loads
- Eigenvector Analysis
- Ritz-vector Analysis
- Modal Analysis Output
- Functions
- Response-Spectrum Analysis
- Response-Spectrum Analysis Output

Advanced Topics

- Harmonic Steady-State Analysis
- Time-History Analysis
- Nonlinear Time-History Analysis
- Combos

Overview

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

- Static analysis
- P-delta analysis
- Modal analysis for vibration modes, using eigenvectors or Ritz vectors
- Response-spectrum analysis for seismic response
- Time-history dynamic response analysis: linear, nonlinear, and periodic
- Moving-load analysis for bridge vehicle live loads
- Harmonic steady-state analysis

These different types of analyses can all be performed in the same execution of the program, and the results combined for output, with the following exceptions:

- Modal analysis is required in order to perform response-spectrum and/or time-history analyses
- Only one type of modal analysis may be performed in a single run: eigenvalue analysis or Ritz-vector analysis
- When harmonic steady-state analysis is performed, no other analysis type is permitted

P-delta analysis, when performed, affects the results of all other analyses performed in the same execution of the program.

Analysis Cases

Each different analysis performed or combination of analysis results 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 different types of analysis cases are:

- **Load Case**, or simply **Load** — a basic spatial distribution of loading, and the corresponding result of a static analysis or a harmonic steady-state analysis
- **Mode** — an eigenvector or Ritz-vector, and the corresponding frequency, resulting from vibration-mode analysis
- **Specs** — the basic result of a response-spectrum analysis
- **History** — the basic result of a time-history analysis
- **Moving Load** — the basic result of a moving-load analysis
- **Combo** — a combination or envelope of Loads, Modes, Specs, Histories, Moving Loads, and/or other Combos

You may define any number of each different type of analysis case to be computed in a single execution of the program. For example, you could request lots of Loads, many Modes, several Specs, multiple Moving Loads, a handful of Histories, and a couple of Combos.

P-delta analysis is not considered to be an analysis case. When requested, a single P-delta analysis is performed. It effectively modifies the characteristics of the structure, and thus affects the results of all the analysis cases.

For more information:

- See Chapter “Load Cases” (page 279).
- See Topic “Harmonic Steady State Analysis” (page 294) in this chapter.
- See Topic “Eigenvector Analysis” (page 296) in this chapter.
- See Topic “Ritz-vector Analysis” (page 299) in this chapter.
- See Topic “Response-Spectrum Analysis” (page 310) in this chapter.
- See Topic “Time History Analysis” (page 319) in this chapter.
- See Topic “Moving Load Cases” (page 383) in Chapter “Bridge Analysis.”
- See Topic “Combos” (page 333) in this chapter.
- See Chapter “P-Delta Analysis” (page 337).

Static Analysis

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

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

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

For each Load Case that you define, the program automatically creates the load vector $\boldsymbol{r}$ and solves for the static displacements $\boldsymbol{u}$.

The Load Cases do not yield static results when steady-state analysis is requested or when a frequency shift has been used for eigenvalue analysis.

See Chapter "Load Cases" (page 279) for more information.

Harmonic Steady-State Analysis

A very common type of loading is of the form $\boldsymbol{r}(t) = \boldsymbol{p}\cos(\omega t)$, where ω is the circular frequency of the excitation, so that $\boldsymbol{r}$ varies with respect to time; however, the spatial distribution of load $\boldsymbol{p}$ does not vary as a function of time. For the case of zero damping, the equilibrium equations for the structural system are of the following form:

$$\boldsymbol{K}\ \boldsymbol{u}(t) + \boldsymbol{M}\ \ddot{\boldsymbol{u}}(t) = \boldsymbol{r}(t) = \boldsymbol{p}\cos(\omega t)$$

where $\boldsymbol{K}$ is the stiffness matrix and $\boldsymbol{M}$ is the diagonal mass matrix. The steady-state solution of this equation requires that the joint displacements $\boldsymbol{u}$ and accelerations $\ddot{\mathbf{u}}$ are of the following form:

$$\boldsymbol{u}(t) = \boldsymbol{a}\cos(\omega t)$$

$$\ddot{\boldsymbol{u}}(t) = -\omega^2\ \boldsymbol{a}\cos(\omega t)$$

Therefore, the response amplitude $\boldsymbol{a}$ is given by the solution of the following set of linear equations:

$$[\ \boldsymbol{K} - \omega^2\ \boldsymbol{M}\ ]\ \boldsymbol{a} = \boldsymbol{p}$$

It is of interest to note that the solution for static loads is nothing more than a solution of this equation with zero frequency.

For each Load Case that you define, the program automatically creates the spatial load vector $\boldsymbol{p}$ and solves for the steady-state displacement amplitudes $\boldsymbol{a}$. The Load Case may be composed of the same types of loading used for static analysis, except that the constant-velocity Rotate Load for the Asolid element would not be meaningful here.

The displacements printed by the program are amplitudes ***a*** (the maximum displacements) which vary as $\cos(\omega t)$. All resulting forces and stresses are also maximum values, which vary as $\cos(\omega t)$.

The limitations of the harmonic steady-state analysis option are:

- The structural damping is assumed to be zero.
- The structure may be analyzed for any number of spatial distributions of steady-state loads. However, the excitation frequency of *all* the loads in any one execution of the program must be the same.
- If the excitation frequency corresponds to a natural frequency of the structure, the system will go into resonance, resulting in an infinite response. In such cases the solution of the system will fail as the matrix $[\mathbf{K} - \omega^2 \mathbf{M}]$ will be singular. The solver will report that the structure is unstable.

Harmonic steady-state analysis is requested by specifying a positive value for the cyclic frequency **cyc**. The circular frequency $\omega = 2\pi$ **cyc**. Setting **cyc** to zero (the default) enables static analysis.

P-delta, response-spectrum, time-history, and moving-load analyses may not be performed when the program is in harmonic steady-state analysis mode.

See Chapter “Load Cases” (page 279) for more information.

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 $\boldsymbol{m}_x$, $\boldsymbol{m}_y$, and $\boldsymbol{m}_z$. These loads are used for applying ground accelerations in response-spectrum and time-history analyses, and can be 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 except for the Asolid are the same in each direction and are equal to the negative of the element mass. No coordinate transformations are necessary.

For the Asolid element, the Acceleration Load in the global direction corresponding to the axial (local 2) direction is equal to the negative of the element mass. The Acceleration Loads in the radial and circumferential directions are zero, since translations in the corresponding global directions are not axisymmetric.

The Acceleration Loads can be transformed into any coordinate system. In a fixed coordinate system (global or Alternate), 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 or time-history analysis, the Acceleration Loads along the positive local 1, 2, and 3 axes are denoted U1, U2, and U3, respectively.

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 or time-history analyses, although Ritz vectors are recommended for this purpose.

Eigenvector analysis involves the solution of the generalized eigenvalue problem:

$$[\boldsymbol{K} - \Omega^2 \boldsymbol{M}]\Phi = \mathbf{0}$$

where $\boldsymbol{K}$ is the stiffness matrix, $\boldsymbol{M}$ is the diagonal mass matrix, Ω^2 is the diagonal matrix of eigenvalues, and Φ 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 (unless a frequency shift is used, see below). 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 to be found, a convergence tolerance, and the frequency range of interest. These parameters are described in the following subtopics.

Number of Modes

You may specify the number of Modes, **n**, to be found. The program will seek the **n** lowest-frequency (longest-period) Modes. If a non-zero frequency shift has been specified, the program will seek the **n** Modes with frequencies closest to **shift**.

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

- The number of Modes requested, **n**
- The number of Modes present in the specified frequency range; see below
- 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 or time-history analysis processing.

See Topic “Degrees of Freedom” (page 83) in Chapter “Joints and Degrees of Freedom.”

Frequency Range

You may specify a restricted frequency range in which to seek the Vibration Modes by using the parameters:

- **shift**: The center of the cyclic frequency range, known as the shift frequency
- **cut**: The radius of the cyclic frequency range, known as the cutoff frequency

The program will only seek Modes with frequencies f that satisfy:

$$|f - \mathbf{shift}| \le \mathbf{cut}$$

The default value of $\mathbf{cut} = 0$ does not restrict the frequency range of the Modes.

Modes are found in order of increasing distance of frequency from the shift. This continues until the cutoff is reached, the requested number of Modes is found, or the number of mass degrees of freedom is reached.

A stable structure will possess all positive natural frequencies. When performing a seismic analysis and most other dynamic analyses, the lower-frequency modes are usually of most interest. It is then appropriate to use a shift of zero, resulting in the lowest-frequency modes of the structure being calculated. If the shift is not zero,

response-spectrum and time-history analyses may be performed; however, static, moving-load, and p-delta analyses are not allowed.

If the dynamic loading is known to be of high frequency, such as that caused by vibrating machinery, it may be most efficient to use a positive shift near the center of the frequency range of the loading.

A structure that is unstable when unloaded will have some modes with zero frequency. These modes may correspond to rigid-body motion of an inadequately supported structure, or to mechanisms that may be present within the structure. It is not possible to compute the static response of such a structure. However, by using a small negative shift, the lowest-frequency vibration modes of the structure, including the zero-frequency instability modes, can be found. This does require some mass to be present that is activated by each instability mode.

A structure that has buckled under P-Delta load will have some modes with zero or negative frequency. If the program detects a negative-frequency mode it will stop the analysis since the results will be meaningless. During equation solution, the number of frequencies less than the shift is determined and printed in the log file. If any frequencies exist that are less than a zero or negative shift, the analysis is halted. If a positive shift is used, the program must perform an eigen-analysis to find negative frequencies. These may not be found if the cutoff radius is too small, or if not enough modes were requested. In order to avoid this problem whenever a positive shift is used with a P-Delta analysis, it is recommended that a preliminary P-Delta analysis be performed using a zero shift.

When using a frequency shift, the stiffness matrix is modified by subtracting from it the mass matrix multiplied by $\omega_0{}^2$, where $\omega_0 = 2\pi$ **shift**. If the shift is very near a natural frequency of the structure, the solution becomes unstable and will be halted during equation solution.

The circular frequency, ω, of a Vibration Mode is determined from the shifted eigenvalue, μ, as:

$$\omega = \sqrt{\mu + \omega_0{}^2}$$

Convergence Tolerance

SAP2000 solves for the eigenvalue-eigenvectors pairs using an accelerated subspace iteration algorithm. During the solution phase, the program prints the approximate eigenvalues after each iteration. As the eigenvectors converge they are

removed from the subspace and new approximate vectors are introduced. For details of the algorithm, see Wilson and Tetsuji (1983).

You may specify the relative convergence tolerance, **tol**, to control the solution; the default value is **tol** = 10^{-5}. The iteration for a particular Mode will continue until the relative change in the eigenvalue between successive iterations is less than $2 \cdot$ **tol**, that is until:

$$\frac{1}{2}\left|\frac{\mu_{i+1} - \mu_i}{\mu_{i+1}}\right| \leq \textbf{tol}$$

where μ is the eigenvalue relative to the frequency shift, and i and $i+1$ denote successive iteration numbers.

In the usual case where the frequency shift is zero, the test for convergence becomes approximately the same as:

$$\left|\frac{T_{i+1} - T_i}{T_{i+1}}\right| \leq \textbf{tol} \quad \text{or} \quad \left|\frac{f_{i+1} - f_i}{f_{i+1}}\right| \leq \textbf{tol}$$

provided that the difference between the two iterations is small.

Note that the error in the eigenvectors will generally be larger than the error in the eigenvalues. The relative error in the global force balance for a given Mode gives a measure of the error in the eigenvector. This error can usually be reduced by using a smaller value of **tol**, at the expense of more computation time.

See Topic “Global Force Balance Output” (page 107) 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.

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 Modes do. The Ritz-vector modes are biased by the starting load vectors.

You may specify the number of Modes to be found, the starting load vectors to be used, and the number of generation cycles to be performed for each starting load vector. These parameters are described in the following subtopics.

Number of Modes

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 or time-history analysis processing.

See Topic "Degrees of Freedom" (page 83) in Chapter "Joints and Degrees of Freedom."

Starting Load Vectors

You may specify any number of starting load vectors. Each starting load vector may be one of the following:

- An Acceleration Load in the global X, Y, or Z direction
- A Load Case
- A built-in nonlinear deformation load, as described below

For response-spectrum analysis, only the Acceleration Loads are needed. For time-history analysis, one starting load vector is needed for each Load Case or Acceleration Load that is used in any History.

If nonlinear time-history analysis is to be performed, an additional starting load vector is needed for *each* independent nonlinear deformation. You may specify that the program use the built-in nonlinear deformation loads, or you may define your own Load Cases for this purpose. See Topic "Nonlinear Deformation Loads" (page 271) in Chapter "The Nllink Element" for more information.

If you define your own starting load vectors, do the following for *each* nonlinear deformation:

- Explicitly define a Load Case that consists of a set of self-equilibrating forces that activates the desired nonlinear deformation
- Specify that Load Case as a starting load vector

The number of such Load Cases required is equal to the number of independent nonlinear deformations in the model.

If several Nllink elements act together, you may be able to use fewer starting load vectors. For example, suppose the horizontal motion of several base isolators are coupled with a diaphragm. Only three starting load vectors acting on the diaphragm are required: two perpendicular horizontal loads and one moment about the vertical axis. Independent Load Cases may still be required to represent any vertical motions or rotations about the horizontal axes for these isolators.

It is *strongly recommended* that mass (or mass moment of inertia) be present at every degree of freedom that is loaded by a starting load vector. This is automatic for Acceleration Loads, since the load is caused by mass. If a Load Case or nonlinear deformation load acts on a non-mass degree of freedom, the program issues a warning. Such starting load vectors may generate inaccurate Ritz vectors, or even no Ritz vectors at all.

Generally, the more starting load vectors used, the more Ritz vectors must be requested to cover the same frequency range. Thus including unnecessary starting load vectors is not recommended.

In each generation cycle, Ritz vectors are found in the order in which the starting load vectors are specified. In the last generation cycle, only as many Ritz vectors will be found as required to reach the total number of Modes, **n**. For this reason, the most important starting load vectors should be specified first, especially if the number of starting load vectors is not much smaller than the total number of Modes.

If no starting load vectors are explicitly specified, the three Acceleration Loads are used by default. This is adequate for response-spectrum analysis and linear seismic time-history analysis with no other loads applied.

For more information:

- See Topic “Nonlinear Time-History Analysis” (page 328) in this chapter.
- See Chapter “Load Cases” (page 279).

Number of Generation Cycles

You may specify the maximum number of generation cycles, **ncyc**, to be performed for each starting load vector. This enables you to obtain more Ritz vectors for some starting load vectors than others. By default, the number of generation cycles performed for each starting load vector is unlimited, i.e., until the total number, **n**, of requested Ritz vectors have been found.

As an example, suppose that two linear time-history analyses are to be performed:

(1) Gravity load is applied quasi-statically to the structure using Load Cases DL and LL

(2) Seismic load is applied in all three global directions starting from the conditions at the end of the first History

The starting load vectors required are the three Acceleration Loads and Load Cases DL and LL. The first generation cycle creates the static solution for each starting load vector. This is all that is required for Load Cases DL and LL in the first History, hence for these starting load vectors **ncyc** = 1 should be specified. Additional Modes may be required to represent the dynamic response to the seismic loading, hence an unlimited number of cycles should be specified for these starting load vectors. If 12 Modes are requested (**n** = 12), there will be one each for DL and LL, three each for two of the Acceleration Loads, and four for the Acceleration Load that was specified first as a starting load vector.

Starting load vectors corresponding to nonlinear deformation loads may often need only a limited number of generation cycles. Many of these loads affect only a small local region and excite only high-frequency natural modes that may respond quasi-statically to typical seismic excitation. If this is the case, you may be able to specify **ncyc** = 1 or 2 for these starting load vectors. More cycles may be required if you are particularly interested in the dynamic behavior in the local region.

You must use your own engineering judgment to determine the number of Ritz vectors to be generated for each starting load vector. No simple rule can apply to all cases.

Modal Analysis Output

Various properties of the Vibration Modes are printed in the output file. This information is the same regardless of whether you use eigenvector or Ritz-vector analysis, and is described in the following subtopics.

Typical modal-analysis output is shown in Figure 84 (page 305).

Periods and Frequencies

The following time-properties are printed 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

These can be found in the output file under the heading:

```
MODAL PERIODS AND FREQUENCIES
```

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:

$$f_{xn} = \varphi_n{}^{\mathrm{T}}\, \boldsymbol{m}_x$$

$$f_{yn} = \varphi_n{}^{\mathrm{T}}\, \boldsymbol{m}_y$$

$$f_{zn} = \varphi_n{}^{\mathrm{T}}\, \boldsymbol{m}_z$$

where φ_n is the mode shape and $\boldsymbol{m}_x$, $\boldsymbol{m}_y$, and, $\boldsymbol{m}_z$ 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 and can be found in the output file under the heading:

```
MODAL PARTICIPATION FACTORS
```

These values are called “factors” because they are related to the mode shape and to a unit acceleration. The modes shapes are each normalized, or scaled, with respect to the mass matrix such that:

$$\varphi_n{}^{\mathrm{T}}\, \boldsymbol{M}\, \varphi_n = 1$$

The actual magnitudes and signs of the participation factors are not important. What is important is the relative values of the three factors for a given Mode.

Participating Mass Ratios

The participating mass ratio for a Mode provides a 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 and seismic time-history analyses. The participating mass ratio

```
M O D A L   P E R I O D S   A N D   F R E Q U E N C I E S

   MODE      PERIOD   FREQUENCY   FREQUENCY  EIGENVALUE
             (TIME)  (CYC/TIME)  (RAD/TIME)  (RAD/TIME)**2

      1    2.425536    0.412280    2.590431    6.710334
      2    2.407723    0.415330    2.609596    6.809991
      3    2.134032    0.468596    2.944279    8.668776
      4    0.233621    4.280440   26.894795  723.329999
      5    0.229738    4.352780   27.349324  747.985515
      6    0.168388    5.938672   37.313777    1392.318
      7    0.131499    7.604615   47.781207    2283.044
      8    0.115861    8.631016   54.230275    2940.923

M O D A L   P A R T I C I P A T I O N   F A C T O R S

FOR UNIT ACCELERATION LOADS IN GLOBAL COORDINATES

   MODE      PERIOD          UX          UY          UZ
      1    2.425536    1.391631    1.732188   -7.71E-05
      2    2.407723   -1.781824    1.426170   -0.000268
      3    2.134032   -0.312429   -0.418085    0.000114
      4    0.233621    0.027199   -0.000259   -0.017982
      5    0.229738    8.72E-05    0.025523    0.056865
      6    0.168388   -6.46E-05   -0.000624    2.172956
      7    0.131499   -0.000283    0.002020    0.072018
      8    0.115861    0.000830    0.000853    0.373158

M O D A L   P A R T I C I P A T I N G   M A S S   R A T I O S

   MODE      PERIOD    INDIVIDUAL MODE (PERCENT)     CUMULATIVE SUM (PERCENT)
                           UX       UY       UZ         UX       UY       UZ

      1    2.425536    37.1724  57.5919   0.0000    37.1724  57.5919   0.0000
      2    2.407723    60.9398  39.0404   0.0000    98.1122  96.6323   0.0000
      3    2.134032     1.8736   3.3551   0.0000    99.9858  99.9874   0.0000
      4    0.233621     0.0142   0.0000   0.0062   100.0000  99.9874   0.0062
      5    0.229738     0.0000   0.0125   0.0621   100.0000  99.9999   0.0683
      6    0.168388     0.0000   0.0000  90.6304   100.0000  99.9999  90.6986
      7    0.131499     0.0000   0.0001   0.0996   100.0000 100.0000  90.7982
      8    0.115861     0.0000   0.0000   2.6727   100.0000 100.0000  93.4709

M O D A L   L O A D   P A R T I C I P A T I O N   R A T I O S

LOAD, ACC, OR NLLINK/DEF      STATIC    DYNAMIC
       (TYPE)      (NAME)   (PERCENT)  (PERCENT)

        LOAD     GRAVITY    100.0000 -> 93.4709<- EXCLUDES LOAD ON NON-MASS DOF
         ACC          UX    100.0000   100.0000
         ACC          UY    100.0000   100.0000
         ACC          UZ     99.0042    93.4709
  NLLINK/DEF        1/U1    100.0000    31.6238
  NLLINK/DEF        1/U2    100.0000    67.2915
  NLLINK/DEF        1/U3    100.0000    63.9055
  NLLINK/DEF        2/U1    100.0000    37.9576
  NLLINK/DEF        2/U2    100.0000    67.2915
  NLLINK/DEF        2/U3    100.0000    89.9700
  NLLINK/DEF        3/U1      5.3432     0.8109
  NLLINK/DEF        3/U2    100.0000    67.2915
  NLLINK/DEF        3/U3    100.0000    57.5064
```

Figure 84
Modal Analysis Output in the Output File

provides *no information* about the accuracy of time-history analyses subjected to other loads.

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

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

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

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

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 and printed in the output file under the heading:

```
MODAL PARTICIPATING MASS RATIOS
```

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 Asolid elements or certain types of Constraints where symmetry conditions prevent some of the mass from responding to translational accelerations.

Static and Dynamic Load Participation Ratios

The static and dynamic load participation ratios provide a measure of how adequate the calculated modes are for representing the response to time-history analyses. These two measures are printed in the output file for each of the following spatial load vectors:

- The three unit Acceleration Loads
- All static Load Cases

- All nonlinear deformation loads

The first two represent spatial loads that you can explicitly specify in a time-history analysis, whereas the last represents loads that can act implicitly in a *nonlinear* time-history analysis.

The load participation ratios, expressed as percentages, are printed in the output file under the heading:

```
MODAL LOAD PARTICIPATION RATIOS
```

For more information:

- See Topic "Nonlinear Deformation Loads" (page 271) in Chapter "The Nllink Element."
- See Chapter "Load Cases" (page 279).
- See Topic "Acceleration Loads" (page 295) in this chapter.
- See Topic "Time-History Analysis" (page 319) in this chapter.
- See Topic "Nonlinear Time-History Analysis" (page 328) in this chapter.

Static Load Participation Ratio

The static load participation ratio measures how well the calculated modes can represent the response to a given static load. This measure was first presented by Wilson (1997). For a given spatial load vector ***p***, the participation factor for Mode *n* is given by

$$f_n = \varphi_n{}^{\mathrm{T}} \boldsymbol{p}$$

where φ_n is the mode shape (vector) of Mode *n*. This factor is the generalized load acting on the Mode due to load ***p***. Note that f_n is just the usual participation factor when ***p*** is one of the unit acceleration loads.

The static participation ratio for this mode is given by

$$r_n^S = \frac{\left(\frac{f_n}{\omega_n}\right)^2}{\boldsymbol{u}^{\mathrm{T}} \boldsymbol{p}}$$

where ***u*** is the static solution given by $\boldsymbol{Ku} = \boldsymbol{p}$. This ratio gives the fraction of the total strain energy in the exact static solution that is contained in Mode *n*. Note that the denominator can also be represented as $\boldsymbol{u}^{\mathrm{T}} \boldsymbol{Ku}$.

Finally, the cumulative sum of the static participation ratios for all the calculated modes is printed in the output file:

$$R^S = \sum_{n=1}^{N} r_n^S = \frac{\sum_{n=1}^{N} \left(\frac{\varphi_n{}^{\mathrm{T}} \boldsymbol{p}}{\omega_n} \right)^2}{\boldsymbol{u}^{\mathrm{T}} \boldsymbol{p}}$$

where N is the number of modes found. This value gives the fraction of the total strain energy in the exact static solution that is captured by the N modes.

When solving for static solutions using quasi-static time-history analysis, the value of R^S should be close to 100% for any applied static Loads, and also for all nonlinear deformation loads if the analysis is nonlinear.

Note that when Ritz-vectors are used, the value of R^S will always be 100% for all starting load vectors. This may not be true when eigenvectors are used. In fact, even using all possible eigenvectors will not give 100% static participation if load $\boldsymbol{p}$ acts on any massless degrees-of-freedom.

Dynamic Load Participation Ratio

The dynamic load participation ratio measures how well the calculated modes can represent the response to a given dynamic load. This measure was developed for SAP2000, and it is an extension of the concept of participating mass ratios. It is assumed that the load acts only on degrees of freedom with mass. Any portion of load vector $\boldsymbol{p}$ that acts on massless degrees of freedom cannot be represented by this measure and is ignored in the following discussion.

For a given spatial load vector $\boldsymbol{p}$, the participation factor for Mode n is given by

$$f_n = \varphi_n{}^{\mathrm{T}} \boldsymbol{p}$$

where φ_n is the mode shape for Mode n. Note that f_n is just the usual participation factor when $\boldsymbol{p}$ is one of the unit acceleration loads.

The dynamic participation ratio for this mode is given by

$$r_n^D = \frac{(f_n)^2}{\boldsymbol{a}^{\mathrm{T}} \boldsymbol{p}}$$

where $\boldsymbol{a}$ is the acceleration given by $\boldsymbol{Ma} = \boldsymbol{p}$. The acceleration $\boldsymbol{a}$ is easy to calculate since $\boldsymbol{M}$ is diagonal. The values of $\boldsymbol{a}$ and $\boldsymbol{p}$ are taken to be zero at all massless degrees of freedom. Note that the denominator can also be represented as $\boldsymbol{a}^{\mathrm{T}}\boldsymbol{Ma}$.

Finally, the cumulative sum of the dynamic participation ratios for all the calculated modes is printed in the output file:

$$R^D = \sum_{n=1}^{N} r_n^D = \frac{\sum_{n=1}^{N} \left(\varphi_n{}^{\mathrm{T}} \boldsymbol{p} \right)^2}{\boldsymbol{a}^{\mathrm{T}} \boldsymbol{p}}$$

where N is the number of modes found. When $\boldsymbol{p}$ is one of the unit acceleration loads, r^D is the usual mass participation ratio, and R^D is the usual cumulative mass participation ratio.

When R^D is 100%, the calculated modes should be capable of exactly representing the solution to any time-varying application of spatial load $\boldsymbol{p}$. If R^D is less than 100%, the accuracy of the solution will depend upon the frequency content of the time-function multiplying load $\boldsymbol{p}$. Normally it is the high frequency response that is not captured when R^D is less than 100%.

The dynamic load participation ratio only measures how the modes capture the spatial characterisitcs of $\boldsymbol{p}$, not its temporal characteristics. For this reason, R^D serves only as a qualitative guide as to whether enough modes have been computed. You must still examine the response to each different dynamic loading with varying number of modes to see if enough modes have been used.

Functions

A Function is a series of digitized abscissa-ordinate pairs that may represent:

- Pseudo-spectral acceleration vs. period for response-spectrum analysis, or
- Load vs. time for time-history analysis

You may define any number of Functions, assigning each one a unique label. You may scale the abscissa and/or ordinate values whenever the Function is used.

The abscissa of a Function is always time or period (which has time units). The abscissa-ordinate pairs must be specified in order of increasing time value.

If the increment between time values (abscissas) is constant and the Function starts at time zero, you need only specify the time increment, **dt**, and the successive function values (ordinates) starting at time zero. The function values are specified as:

f0, **f1**, **f2**, ..., **fn**

corresponding to times:

0, **dt**, 2 **dt**, ..., *n* **dt**

where *n* + 1 is the number of values given.

If the time increment is not constant or the Function does not start at time zero, you must specify the pairs of time and function value as:

t0, **f0**, **t1**, **f1**, **t2**, **f2**, ..., **tn**, **fn**

where *n* + 1 is the number of pairs of values given.

Response-Spectrum Analysis

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

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

where $\boldsymbol{K}$ is the stiffness matrix; $\boldsymbol{C}$ is the proportional damping matrix; $\boldsymbol{M}$ is the diagonal mass matrix; $\boldsymbol{u}$, $\dot{\boldsymbol{u}}$, and $\ddot{\boldsymbol{u}}$ are the relative displacements, velocities, and accelerations with respect to the ground; $\boldsymbol{m}_x$, $\boldsymbol{m}_y$, and $\boldsymbol{m}_z$ 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 performed in a single execution of the program. Each analysis case is called a **Spec**, to which you assign a unique label. Each Spec can differ in the acceleration spectra applied and in the way that results are combined. Results from each Spec can be output directly or used in Combos.

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

Local Coordinate System

Each Spec 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 fixed coordinate system **csys** (the default is zero, indicating the global coordinate system)
- A coordinate angle, **ang** (the default is zero)

The local 3 axis is always the same as the Z axis of coordinate system **csys**. The local 1 and 2 axes coincide with the X and Y axes of **csys** if angle **ang** is zero. Otherwise, **ang** is the angle from the X axis to the local 1 axis, measured counterclockwise when the +Z axis is pointing toward you. This is illustrated in Figure 85 (page 312).

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

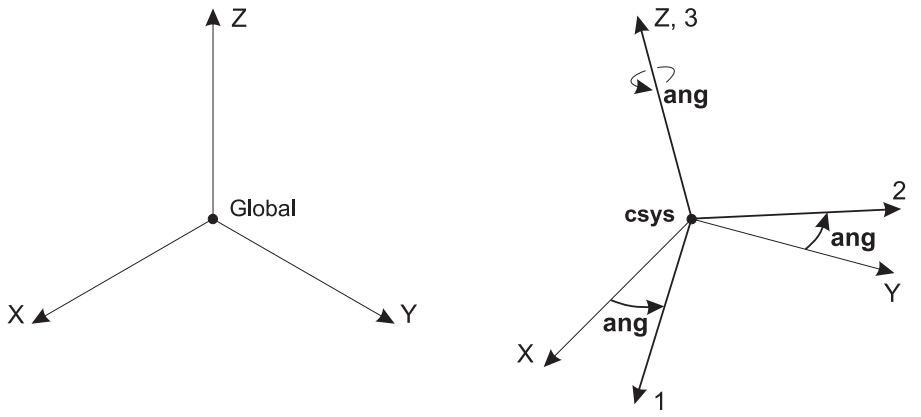


Figure 85
Definition of Response Spectrum Local Coordinate System

the curve is given by specifying the name of a Function. All values for the abscissa and ordinate of this Function must be zero or positive.

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

You may specify a scale factor **sf** to multiply the ordinate (pseudo spectral acceleration response) of the function. This is often needed to convert values given in terms of the acceleration due to gravity to units consistent with the rest of the model. See Figure 86 (page 313).

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 Vibration Modes of the structure, the curve is extended to larger and smaller periods using a constant acceleration equal to the value at the nearest defined period.

See Topic “Functions” (page 309) in this chapter for more information.

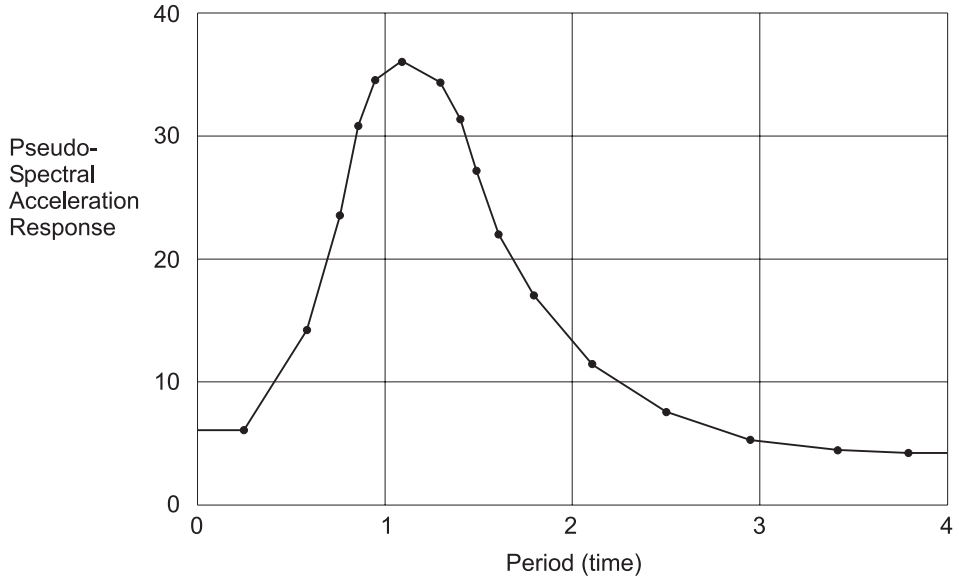


Figure 86
Digitized Response-Spectrum Curve

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. Increasing the modal damping increases the coupling between closely-spaced modes. If the damping is zero for all Modes, this method degenerates to the SRSS method.

Modal damping comes from two sources:

- You may specify a CQC damping ratio, **damp**, measured as a fraction of critical damping: $0 \le$ **damp** < 1. This damping equally affects all Modes, and 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.
- You may specify non-zero effective-damping coefficients for any Nllink elements in the model. These effective damping values are converted to modal damping ratios by assuming proportional damping, i.e., by ignoring any modal cross-coupling terms. These effective modal-damping values will generally be different for each Mode, depending upon how much deformation each Mode causes in the Nllink elements. Only effective damping, not actual damping, in the Nllink elements affects the CQC results.

The total damping ratio for each Mode is the sum of these two sources. The program will automatically make sure that the total is less than one.

GMC Method

Specify **modc** = GMC to combine the modal results by the General Modal Combination technique. This method 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.

Increasing the modal damping increases the coupling between closely-spaced modes. As with the CQC method, modal damping comes from the two sources:

- You may specify a GMC damping ratio, **damp**, such that $0 \le$ **damp** < 1. This damping equally affects all Modes.
- Additional modal damping is computed from the effective-damping coefficients for any Nllink elements in the model

See the CQC method above for more information on modal damping.

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 $0 <$ **f1** $<$ **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:

$$\mathbf{f1} = \frac{S_{A\max}}{2\pi\, S_{V\max}}$$

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 directional values for a given response quantity are combined to produce a single, posi-

tive 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 < **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:

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

where:

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

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

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

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% unconservative or 4% over-conservative, depending upon the coordinate system. Larger values of **dirf** tend to produce more conservative results.

Response-Spectrum Analysis Output

Certain information is printed in the output file for each response-spectrum analysis. This information is described in the following subtopics.

Typical response-spectrum analysis output is shown in Figure 87 (page 318).

Damping and Accelerations

The modal damping and the ground accelerations acting in each direction are printed for every Mode under the heading:

```
RESPONSE SPECTRUM ACCELERATIONS
```

The damping value printed for each Mode is the sum of the specified CQC or GMC damping ratio, **damp**, plus the modal damping contributed by effective damping in the Nllink elements, if any.

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 values of **sf** and **tf**. 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. 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. These values are printed in the output file under the heading:

```
RESPONSE SPECTRUM MODAL AMPLITUDES
```

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 Output” (page 303) in this chapter for the definition of the modal participation factors and the eigenvalues.

```
R E S P O N S E   S P E C T R U M   A C C E L E R A T I O N S

IN RESPONSE-SPECTRUM LOCAL COORDINATES

SPEC       1 ------------------

   MODE      PERIOD  DAMP-RATIO          U1          U2          U3
      1    0.414601    0.050000   29.443875   29.443875     .000000
      2    0.375298    0.050000   28.919579   28.919579     .000000
      3    0.243572    0.050000   26.554969   26.554969     .000000

R E S P O N S E   S P E C T R U M   M O D A L   A M P L I T U D E S

IN RESPONSE-SPECTRUM LOCAL COORDINATES

SPEC       1 ------------------

   MODE      PERIOD          U1          U2          U3
      1    0.414601   -0.168018   -0.286686     .000000
      2    0.375298   -0.231232    0.135573     .000000
      3    0.243572    0.010526   -0.006012     .000000

R E S P O N S E   S P E C T R U M   M O D A L   C O R R E L A T I O N S

PARTIAL MATRIX SHOWING CORRELATION FACTORS BETWEEN NEARBY MODES

SPEC       1 ------------------

 MODE I      PERIOD     I   I+1   I+2   I+3   I+4   I+5   I+6   I+7   I+8   I+9
      1    0.414601 1.000 0.501 0.032
      2    0.375298 1.000 0.049
      3    0.243572 1.000

R E S P O N S E   S P E C T R U M   B A S E   R E A C T I O N S

IN RESPONSE-SPECTRUM LOCAL COORDINATES

SPEC       1 ------------------

   FOR EACH MODE, DUE TO SPECTRAL ACCELERATION IN DIRECTION U1:

   MODE          F1          F2          F3          M1          M2          M3
      1   50.572403   86.290877     .000000   -1830.315    1072.735    2492.263
      2  145.250875  -85.161739     .000000    1793.046    3058.354   -2132.818
      3    1.847642   -1.055259     .000000   20.956121   36.678180  -26.348178

   FOR EACH MODE, DUE TO SPECTRAL ACCELERATION IN DIRECTION U2:

   MODE          F1          F2          F3          M1          M2          M3
      1   86.290877  147.236736     .000000   -3123.037    1830.390    4252.508
      2  -85.161739   49.931002     .000000   -1051.277   -1793.137    1250.488
      3   -1.055259    0.602698     .000000  -11.968838  -20.948303   15.048447

   COMBINED FOR ALL MODES AND ALL DIRECTIONS OF SPECTRAL ACCELERATION:

                F1          F2          F3          M1          M2          M3
   SPEC  195.935537  197.196505     .000000    4174.970    4133.414    5515.265
```

Figure 87
Response-Spectrum Analysis Output in the Output File

Modal Correlation Factors

When the type of modal combination is either CQC or GMC, a partial modal correlation matrix is printed out. This matrix shows the coupling assumed between closely-spaced modes. The correlation factors are always between zero and one. The correlation factors coupling each mode with the next nine higher modes are printed in the output file under the heading:

```
RESPONSE SPECTRUM MODAL CORRELATIONS
```

The correlation matrix is symmetric.

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 in the output file under the heading:

```
RESPONSE SPECTRUM BASE REACTIONS
```

These are printed separately for each individual Mode and each direction of loading without any combination. The total response-spectrum reactions are then printed 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.

Time-History Analysis

Time-history analysis is used to determine the dynamic response of a structure to arbitrary loading. The dynamic equilibrium equations to be solved are given by:

$$\boldsymbol{K}\ \boldsymbol{u}(t) + \boldsymbol{C}\ \dot{\boldsymbol{u}}(t) + \boldsymbol{M}\ \ddot{\boldsymbol{u}}(t) = \boldsymbol{r}(t)$$

where $\boldsymbol{K}$ is the stiffness matrix; $\boldsymbol{C}$ is the proportional damping matrix; $\boldsymbol{M}$ is the diagonal mass matrix; $\boldsymbol{u}$, $\dot{\boldsymbol{u}}$, and $\ddot{\boldsymbol{u}}$ are the relative displacements, velocities, and accelerations with respect to the ground; and $\boldsymbol{r}$ is the applied load.

Any number of time-history analyses can be performed in a single execution of the program. Each analysis case is called a **History**, to which you assign a unique label. Each History can differ in the load applied and in the type of analysis to be performed.

Three types of time-history analysis are available:

- Linear transient: The structure starts with zero initial conditions or with the conditions at the end of a previous linear transient History that you specify. All elements are assumed to behave linearly for the duration of the analysis.
- Periodic: The initial conditions are adjusted to be equal to the those at the end of the period of analysis. All elements are assumed to behave linearly for the duration of the analysis.
- Nonlinear transient: The structure starts with zero initial conditions or with the conditions at the end of a previous nonlinear transient History that you specify. The Nllink elements may exhibit nonlinear behavior during the analysis. All other elements behave linearly.

P-Delta analysis, if requested, is performed before any time-history analyses.

This topic describes features that are common to all three types of analysis. Additional features applying only to nonlinear transient analysis are described in the Topic "Nonlinear Time-History Analysis" (page 328) in this chapter.

Loading

The load, ***r(t)***, applied in a given History may be an arbitrary function of space and time. It can be written as a finite sum of spatial load vectors, $\boldsymbol{p}_i$, multiplied by time functions, $f_i(t)$, as:

$$\boldsymbol{r}(t) = \sum_i f_i(t)\ \boldsymbol{p}_i \qquad \text{(Eqn. 1)}$$

The program uses Load Cases and/or Acceleration Loads to represent the spatial load vectors. The time functions can be arbitrary functions of time or periodic functions such as those produced by wind or sea wave loading.

If Acceleration Loads are used, the displacements, velocities, and accelerations are all measured relative to the ground. The time functions associated with the Acceleration Loads $\boldsymbol{m}_x$, $\boldsymbol{m}_y$, and $\boldsymbol{m}_z$ are the corresponding components of uniform ground acceleration, $\ddot{u}_{gx}$, $\ddot{u}_{gy}$, and $\ddot{u}_{gz}$.

Defining the Spatial Load Vectors

To define the spatial load vector, $\boldsymbol{p}_i$, for a single term of the loading sum of Equation 1, you may specify either:

- The label of a Load Case using the parameter **load**, or

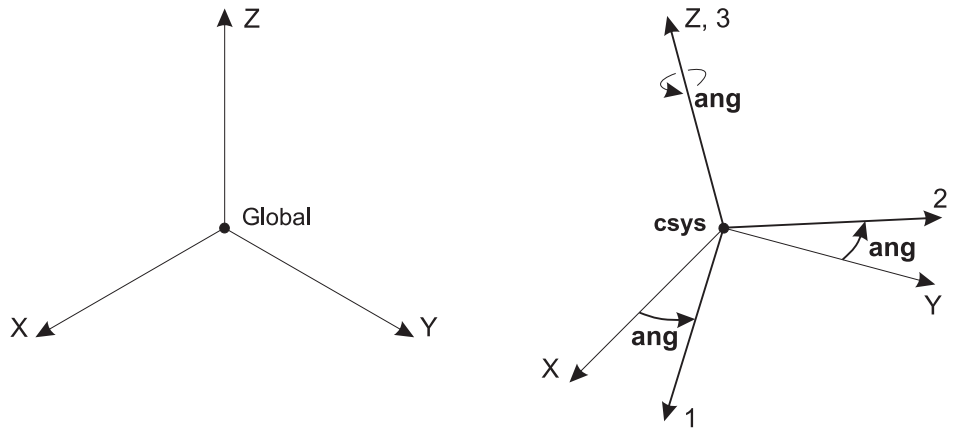


Figure 88
Definition of History Acceleration Local Coordinate System

- An Acceleration Load using the parameters **csys**, **ang**, and **acc**, where:
  - **csys** is a fixed coordinate system (the default is zero, indicating the global coordinate system)
  - **ang** is a coordinate angle (the default is zero)
  - **acc** is the Acceleration Load (U1, U2, or U3) in the acceleration local coordinate system as defined below

Each Acceleration Load in the loading sum may have its own acceleration local coordinate system with local axes denoted 1, 2, and 3. The local 3 axis is always the same as the Z axis of coordinate system **csys**. The local 1 and 2 axes coincide with the X and Y axes of **csys** if angle **ang** is zero. Otherwise, **ang** is the angle from the X axis to the local 1 axis, measured counterclockwise when the +Z axis is pointing toward you. This is illustrated in Figure 88 (page 321).

The response-spectrum local axes are always referred to as 1, 2, and 3. The global Acceleration Loads $\boldsymbol{m}_x$, $\boldsymbol{m}_y$, and $\boldsymbol{m}_z$ are transformed to the local coordinate system for loading.

It is generally recommended, but not required, that the same coordinate system be used for all Acceleration Loads applied in a given History.

Load Cases and Acceleration Loads may be mixed in the loading sum.

For more information:

- See Chapter "Load Cases" (page 279).
- See Topic "Acceleration Loads" (page 295) in this chapter.

Defining the Time Functions

To define the time function, $f_i(t)$, for a single term of the loading sum of Equation 1, you may specify:

- The label of a Function, using the parameter **func**, that defines the shape of the time variation (the default is zero, indicating the built-in ramp function defined below)
- A scale factor, **sf**, that multiplies the ordinate values of the Function (the default is unity)
- A time-scale factor, **tf**, that multiplies the time (abscissa) values of the Function (the default is unity)
- An arrival time, **at**, when the Function begins to act on the structure (the default is zero)

The time function, $f_i(t)$, is related to the specified Function, **func(t)**, by:

$$f_i(t) = \mathbf{sf} \cdot \mathbf{func(t)}$$

The analysis time, t, is related to the time scale, **t**, of the specified Function by:

$$t = \mathbf{at} + \mathbf{tf} \cdot \mathbf{t}$$

If the arrival time is positive, the application of Function **func** is delayed until after the start of the analysis. If the arrival time is negative, that portion of Function **func** occurring before $\mathbf{t} = -\mathbf{at} / \mathbf{tf}$ is ignored.

For a Function **func** defined from initial time $\mathbf{t}_0$ to final time $\mathbf{t}_n$, the value of the Function for all time $\mathbf{t} < \mathbf{t}_0$ is taken as zero, and the value of the Function for all time $\mathbf{t} > \mathbf{t}_n$ is held constant at $\mathbf{f}_n$, the value at $\mathbf{t}_n$.

If no Function is specified, or **func** = 0, the built-in ramp function is used. This function increases linearly from zero at $\mathbf{t} = 0$ to unity at $\mathbf{t} = 1$ and for all time thereafter. When combined with the scaling parameters, this defines a function that increases linearly from zero at t = **at** to a value of **sf** at t = **at** + **tf** and for all time thereafter, as illustrated in Figure 88 (page 323). This function is most commonly used to gradually apply static loads, but can also be used to build up triangular pulses and more complicated functions.

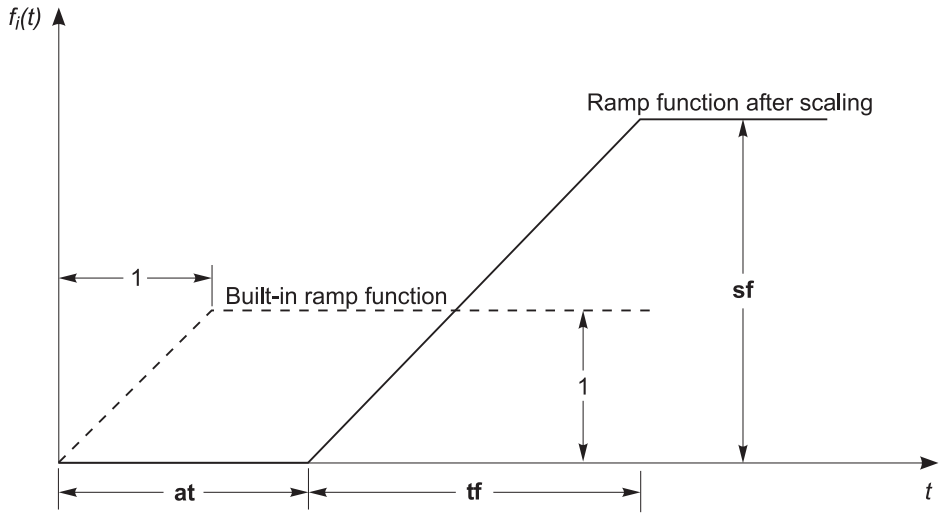


Figure 89
Built-in Ramp Function before and after Scaling

See Topic “Functions” (page 309) in this chapter for more information.

Mode Superposition

The standard mode-superposition method of response analysis is used by the program to solve the dynamic equilibrium equations of motion for the complete structure. The Modes used can be the undamped free-vibration Modes (eigenvectors) or the load-dependent Ritz-vector Modes.

If all of the spatial load vectors, $\boldsymbol{p}_i$, are used as starting load vectors for Ritz-vector analysis, then the Ritz vectors will always produce more accurate results than if the same number of eigenvectors is used. Since the Ritz-vector algorithm is faster than the eigenvector algorithm, the former is recommended for time-history analyses.

It is up to you to determine if the Modes calculated by the program are adequate to represent the time-history response to the applied load. You should check:

- That enough Modes have been computed
- That the Modes cover an adequate frequency range
- That the participating mass ratios are adequate if Acceleration Loads are applied

- That the modes shapes adequately represent all desired deformations
- That the global force balance for each Mode is acceptable

Special considerations must be made for the Modes used when performing nonlinear analysis. See Topic “Nonlinear Time-History Analysis” (page 328) in this chapter for more information.

For more information:

- See Topic “Eigenvector Analysis” (page 296) in this chapter.
- See Topic “Ritz-vector Analysis” (page 299) in this chapter.
- See Subtopic “Participating Mass Ratios” (page 304) in this chapter
- See Topic “Global Force Balance Output” (page 107) in Chapter “Joints and Degrees of Freedom.”

Modal Damping

The damping in the structure is modeled using modal damping, also known as proportional or classical damping. The damping in each Mode may be specified using the parameter **damp**, which is measured as a fraction of critical damping. The damping value for each Mode must satisfy:

$$0 \le \mathbf{damp} < 1$$

For a nonlinear transient analysis, this is the only source of modal damping. See Topic “Nonlinear Time-History Analysis” (page 328) for important considerations about using modal damping with nonlinear transient analysis.

For linear transient and periodic analyses, additional modal damping may come from any non-zero effective-damping coefficients that may be specified for Nllink elements in the model. These effective damping values are converted to modal damping ratios assuming proportional damping, i.e., by ignoring any modal cross-coupling terms. These effective modal-damping values will generally be different for each Mode, depending upon how much deformation each Mode causes in the Nllink elements.

Only effective damping, not the actual damping in the Nllink elements, affects the linear time-history results. The total damping ratio for each Mode is the sum of these two sources. The program automatically makes sure that the total is less than one.

Time Steps

Time-history analysis is performed at discrete time steps. You may specify the number of time steps with parameter **nstep** and the size of the time steps with parameter **dt**.

The time span over which the analysis is carried out is given by **nstep**·**dt**. For periodic analysis, the period of the cyclic loading function is assumed to be equal to this time span.

Responses are calculated at the end of each **dt** time increment, resulting in **nstep**+1 values for each output response quantity.

For linear transient and periodic analysis, closed-form integration of the modal equations is used to compute the response, assuming linear variation of the time functions, $f_i(t)$, between the input data time points. Therefore, numerical instability problems are never encountered, and the time increment may be any sampling value that is deemed fine enough to capture the maximum response values. One-tenth of the time period of the highest mode is usually recommended; however, a larger value may give an equally accurate sampling if the contribution of the higher modes is small.

For nonlinear transient analysis, closed-form integration is again used to compute the response, with the forces from the nonlinear Nllink elements applied to the right-hand side of the dynamic modal equilibrium equations. Since these forces depend upon the dynamic response, the solution is iterated at each time step until the nonlinear forces converge. The accuracy of the results may depend upon the size of the time step, improving with decreasing **dt**. See Topic “Nonlinear Time-History Analysis” (page 328) in this chapter for more information.

Initial Conditions

The initial conditions describe the state of the structure at the beginning of a History. These include:

- Displacements and velocities
- Internal forces and stresses
- Internal state variables for the Nllink elements, if any
- Energy values for the structure

The accelerations are not considered initial conditions, but are computed from the equilibrium equation.

For each linear and nonlinear transient analysis to be performed, you may use the parameter **prev** to specify the initial conditions to be used. It is not applicable to periodic analyses.

If **prev** = 0 (the default), zero initial conditions are assumed for all displacements, velocities, energies, and the internal state variables. The initial acceleration of the structure is computed by equilibrium using the applied loads and/or ground accelerations at time $t = 0$.

Otherwise **prev** must be the label of a previous History of the same type (linear or nonlinear transient) as the current History. The initial conditions of the current History are taken to be the conditions that existed at the end of History **prev**. This includes the displacements, velocities, energies, and the internal state variables. In addition, the applied loads and/or ground accelerations that existed at the end of History **prev** are held constant and are added to the loads and/or ground accelerations specified for the current History. The initial acceleration of the structure is computed from equilibrium using the total applied loads and/or ground accelerations at time $t = 0$.

For example, suppose three linear transient Histories are defined for a single execution of the program by the following data:

(1) **name**=GRAV, **nstep**=40, **dt**=1, **prev**=0, **damp**=0.99 for all Modes, and loading comes from the following two Load Cases:

- **load**=DL, **func**=0, **tf**=10
- **load**=LL, **func**=0, **tf**=10

(2) **name**=LPRIET, **nstep**=2000, **dt**=0.02, **prev**=GRAV, **damp**=0.05 for all Modes, and loading comes from the following two Acceleration Loads:

- **acc**=U1, **func**=LPRIET1, **sf**=386.4
- **acc**=U2, **func**=LPRIET2, **sf**=386.4

(3) **name**=ELCENT, **nstep**=2000, **dt**=0.02, **prev**=GRAV, **damp**=0.05 for all Modes, and loading comes from the following two Acceleration Loads:

- **acc**=U1, **func**=ELCENT1, **sf**=386.4
- **acc**=U2, **func**=ELCENT2, **sf**=386.4

The first History, GRAV, starts with zero initial conditions (it must, since there is no previous History). Dead load and live load are applied slowly using a ramp function over a length of 10 seconds and held for an additional 30 seconds. The length of

the ramp function was chosen to be much longer than the longest period of the structure. A large damping value is used to prevent oscillation. This is especially important for nonlinear analysis where oscillation may cause unrealistic hysteric behavior.

The second and third Histories, LPRIET and ELCENT each start from History GRAV and apply ground accelerations to the structure with 5% damping. Gravity loads are still acting on the structure throughout Histories LPRIET and ELCENT since they were present at the end of History GRAV.

This approach is particularly useful for nonlinear analysis where the behavior of certain Nllink elements, especially the Gap, Hook and Isolat2 types, is strongly dependent on the *total* force or displacement acting on the elements.

Analysis Results

Two types of results are available from a time-history analysis: traces and envelopes. These are described below.

Traces

Time-history traces are plots of response quantities as functions of time. Using the SAP2000 graphical user interface, you may plot one or more response quantities against time, or any two response quantities against each other. Time-history traces are always available after an analysis. The following type of response quantities may be plotted:

- Joint displacements, velocities, and accelerations
- Joint reactions and spring forces
- Frame, Shell, and Nllink element internal forces
- Shell, Plane, Asolid, and Solid element stresses
- Base reactions for ground-acceleration loading
- Solution energies: input, kinetic, potential, modal damping, nonlinear damping, and nonlinear hysteretic

Envelopes

Time-history envelopes are the maximum and minimum values attained during a time-history analysis for each response quantity. Envelope values may be printed in the output file, used in Combos, and displayed and printed in the SAP2000 graphical user interface.

For each time-history case, you may specify whether or not envelope values are to be computed. By default, envelope values are not computed and thus they cannot be printed, displayed, or used in Combos. If you request envelopes for a given History, values are computed for all of the following quantities:

- Joint displacements
- Joint reactions and spring forces
- Frame, Shell, and Nllink element internal forces
- Shell, Plane, Asolid, and Solid element stresses

Nonlinear Time-History Analysis

The method of nonlinear time-history analysis used in SAP2000 is an extension of the Fast Nonlinear Analysis (FNA) method developed by Wilson (Ibrahimbegovic and Wilson, 1989; Wilson, 1993). The method is extremely efficient and is designed to be used for structural systems which are primarily linear elastic, but which have a limited number of predefined nonlinear elements. In SAP2000, all nonlinearity is restricted to the Nllink elements. A short description of the method follows.

The dynamic equilibrium equations of a linear elastic structure with predefined nonlinear elements subjected to an arbitrary load can be written as:

$$\boldsymbol{K}_L\ \boldsymbol{u}(t) + \boldsymbol{C}\ \dot{\boldsymbol{u}}(t) + \boldsymbol{M}\ \ddot{\boldsymbol{u}}(t) + \boldsymbol{r}_N(t) = \boldsymbol{r}(t)$$

where $\boldsymbol{K}_L$ is the stiffness matrix for the linear elastic elements (all elements except the Nllinks); $\boldsymbol{C}$ is the proportional damping matrix; $\boldsymbol{M}$ is the diagonal mass matrix; $\boldsymbol{r}_N$ is the vector of forces from the nonlinear degrees of freedom in the Nllink elements; $\boldsymbol{u}$, $\dot{\boldsymbol{u}}$, and $\ddot{\boldsymbol{u}}$ are the relative displacements, velocities, and accelerations with respect to the ground; and $\boldsymbol{r}$ is the vector of applied loads. See Topic "Time-History Analysis" (page 319) in this chapter for the definition of $\boldsymbol{r}$.

Nllink Effective Stiffness

For the purposes of analysis, a linear **effective stiffness** is defined for each degree of freedom of the nonlinear elements. The effective stiffness at nonlinear degrees of freedom is arbitrary, but generally varies between zero and the maximum nonlinear stiffness of that degree of freedom.

The equilibrium equation can then be rewritten as:

$$\boldsymbol{K}\ \boldsymbol{u}(t)+\boldsymbol{C}\ \dot{\boldsymbol{u}}(t)+\boldsymbol{M}\ \ddot{\boldsymbol{u}}(t)=\boldsymbol{r}(t)-[\ \boldsymbol{r}_N(t)-\boldsymbol{K}_N\ \boldsymbol{u}(t)\ ]$$

where $\boldsymbol{K}=\boldsymbol{K}_L+\boldsymbol{K}_N$, with $\boldsymbol{K}_L$ being the stiffness of all the linear elements and for the linear degrees of freedom of the Nllink elements, and $\boldsymbol{K}_N$ being the linear effective-stiffness matrix for all of the nonlinear degrees of freedom.

See Chapter “The Nllink Element” (page 243) for more information.

Mode Superposition

Modal analysis is performed using the full stiffness matrix, ***K***, and the mass matrix, ***M***. It is *strongly recommended* that the Ritz-vector method be used to perform the modal analysis.

Using standard techniques, the equilibrium equation can be written in modal form as:

$$\Omega^2\ \boldsymbol{a}(t)+\Lambda\ \dot{\boldsymbol{a}}(t)+\boldsymbol{I}\ \ddot{\boldsymbol{a}}(t)=\boldsymbol{q}(t)-\boldsymbol{q}_N(t)$$

where Ω^2 is the diagonal matrix of squared structural frequencies given by:

$$\Omega^2=\Phi^{\mathrm{T}}\ \boldsymbol{K}\ \Phi$$

Λ is the modal damping matrix which is assumed to be diagonal:

$$\Lambda=\Phi^{\mathrm{T}}\ \boldsymbol{C}\ \Phi$$

I is the identity matrix which satisfies:

$$\boldsymbol{I}=\Phi^{\mathrm{T}}\ \boldsymbol{M}\ \Phi$$

$\boldsymbol{q}(t)$ is the vector of modal applied loads:

$$\boldsymbol{q}(t)=\Phi^{\mathrm{T}}\ \boldsymbol{r}(t)$$

$\boldsymbol{q}_N(t)$ is the vector of modal forces from the nonlinear elements:

$$\boldsymbol{q}_N(t)=\Phi^{\mathrm{T}}\ [\ \boldsymbol{r}_N(t)-\boldsymbol{K}_N\ \boldsymbol{u}(t)\ ]$$

$\boldsymbol{a}(t)$ is the vector of modal displacement amplitudes such that:

$$\boldsymbol{u}(t)=\Phi\ \boldsymbol{a}(t)$$

and Φ is the matrix of mode shapes.

It should be noted that, unlike linear dynamic analysis, the above modal equations are coupled. In general the nonlinear forces, $\boldsymbol{q}_N(t)$, will couple the modes since they are functions of the modal displacements, $\boldsymbol{a}(t)$.

It is important to recognize that the solution to these modal equations is dependent on being able to adequately represent the nonlinear forces by the modal forces, $\boldsymbol{q}_N(t)$. This is not automatic, but requires the following special considerations:

- Mass and/or mass moments of inertia should be present at all nonlinear degrees of freedom.
- The Ritz-vector method should be used to determine the Modes, unless *all* possible structural Modes are found using eigenvector analysis
- The Ritz starting load vectors should include a nonlinear deformation load for each independent nonlinear degree of freedom
- A sufficient number of Ritz-vectors should be sought to capture the deformation in the nonlinear elements completely

For more information:

- See Topic “Ritz-vector Analysis” (page 299) in this chapter.
- See Chapter “The Nllink Element” (page 243).

Modal Damping

It is also important to note that the assumption of proportional modal damping is being made with respect to the total stiffness matrix, $\boldsymbol{K}$, which includes the effective stiffness from the nonlinear elements. If non-zero modal damping is to be used, then the effective stiffness specified for these elements is important. The effective stiffness should be selected such that the modes for which these damping values are specified are realistic.

In general it is recommended that either the initial stiffness of the element be used as the effective stiffness or the secant stiffness obtained from tests at the expected value of the maximum displacement be used. Initially open gap and hook elements and all damper elements should generally be specified with zero effective stiffness.

Iterative Solution

The nonlinear modal equations are solved iteratively in each time step. The program assumes that the right-hand sides of the equations vary linearly during a time step, and uses exact, closed-form integration to solve these equations in each itera-

tion. The iterations are carried out until the solution converges. If convergence cannot be achieved, the program divides the time step into smaller substeps and tries again.

Several parameters are available for you to control the iteration process. In general, the use of the default values is recommended since this will solve most problems. If convergence cannot be achieved, inaccurate results are obtained, or the solution takes too long, changing these control parameters may help. However, you should first check that reasonable loads and properties have been specified, and that appropriate Modes have been obtained, preferably using the Ritz vector method.

The parameters that are available to control iteration and substepping are:

- The relative force convergence tolerance, **ftol**
- The relative energy convergence tolerance, **etol**
- The maximum allowed substep size, **dtmax**
- The minimum allowed substep size, **dtmin**
- The maximum number of force iterations permitted for small substeps, **itmax**
- The maximum number of force iterations permitted for large substeps, **itmin**
- The convergence factor, **cf**

These parameters are used in the iteration and substepping algorithm as described in the following.

Force Convergence Check

Each time step of length **dt** is divided into substeps as needed to achieve convergence. In each substep, the solution is iterated until the change in the right-hand side of the modal equations, expressed as a fraction of the right-hand side, becomes less than the force tolerance, **ftol**. If this does not occur within the permitted number of iterations, the substep size is halved and the iteration is tried again.

The default value for **ftol** is 10^{-5}. It must satisfy **ftol** > 0.

Energy Convergence Check

If force convergence occurs within the permitted number of iterations, the work done by the nonlinear forces is compared with the work done by all the other force terms in the modal equilibrium equations. If the difference, expressed as a fraction of the total work done, is greater than the energy tolerance, **etol**, the substep size is halved and the iteration is tried again.

This energy check essentially measures how close to linear is the variation of the nonlinear force over the time step. It is particularly useful for detecting sudden changes in nonlinear behavior, such as the opening and closing of gaps or the onset of yielding and slipping. Setting **etol** greater than unity turns off this energy check.

The default value for **etol** is 10^{-5}. It must satisfy **etol** > 0.

Maximum and Minimum Substep Sizes

If the substep meets both the force and energy convergence criteria, the results of the substep are accepted, and the next substep is attempted using twice the previous substep length. The substep size is never increased beyond **dtmax**.

When the substep size is halved because of failure to meet either the force or energy convergence criteria, the resulting substep size will never be set less than **dtmin**. If the failed substep size is already **dtmin**, the results for the remaining time steps in the current History are set to zero and a warning message is issued.

The default value for **dtmax** is **dt**. The default value for **dtmin** is **dtmax**$\cdot 10^{-9}$. They must satisfy $0 <$ **dtmin** $\le$ **dtmax** $\le$ **dt**.

Maximum Number of Iterations

The maximum number of iterations permitted for force iteration varies between **itmin** and **itmax**. The actual number permitted for a given substep is chosen automatically by the program to achieve a balance between iteration and substepping. The number of iterations permitted tends to be larger for smaller substeps.

The default values for **itmin** and **itmax** are 2 and 100, respectively. They must satisfy $2 \le$ **itmin** $\le$ **itmax**.

Convergence Factor

Under-relaxation of the force iteration may be used by setting the convergence factor, **cf**, to a value less than unity. Smaller values increase the stability of the iteration, but require more iterations to achieve convergence. This is generally only needed when Damper-type elements are present with nonlinear damping exponents. Specifying **cf** to be greater than unity may reduce the number of iterations required for certain types of problems, but may cause instability in the iteration and is not recommended.

The default value for **cf** is 1. It must satisfy **cf** > 0.

Static Period

Normally all modes are treated as being dynamic. In each time step, the response of a dynamic mode has two parts:

- Forced response, which is directly proportional to the modal load
- Transient response, which is oscillatory, and which depends on the displacements and velocities of the structure at the beginning of the time step

You may optionally specify that high-frequency (short period) modes be treated as static, so that they follow the load without any transient response. This is done by specifying a static period, **tstat**, such that all modes with periods less than **tstat** are considered to be static modes. The default for **tstat** is zero, meaning that all modes are considered to be dynamic.

Although **tstat** can be used for any nonlinear time-history analysis, it is of most use for quasistatic analyses. If the default iteration parameters do not work for such an analysis, you may try using the following parameters as a starting point:

- **tstat** greater than the longest period of the structure
- **itmax** = **itmin** ≥ 1000
- **dtmax** = **dtmin** = **dt**
- **ftol** ≤ 10^{-6}
- **cf** = 0.1

This causes all modes to be treated as static, and uses iteration rather than substepping to find a solution. The choice of parameters to achieve convergence is very problem dependent, and you should experiment to find the best values to use for each different model.

Combos

A Combo is a combination of the results from Loads, Modes, Specs, Histories, Moving Loads, and/or previously-defined Combos. Combo results include all displacements and forces at the joints and internal forces or stresses in the elements.

You may specify any number of Combos. To each one of these you assign a unique label.

Each Combo produces a pair of values for each response quantity: a maximum and a minimum. These two values may be equal for certain type of Combos, as discussed below.

Each contributing analysis case supplies one or two values to the Combo for each response quantity:

- Loads, Modes, and additive Combos of Loads and Modes provide a single value. For the purposes of defining the Combos below, this single value can also be considered to be two equal values
- All other analysis cases (including Combos) provide two values: a maximum and minimum. For some types of Combos, both values are used. For other types of Combos, only the value with the larger absolute value is used.

For contributing response-spectrum analyses, the maximum value used is the positive Spec value, and the minimum value is just the negative of the maximum.

For contributing time-history analyses, the values used are the maximum and minimum values attained at any time during the analysis. *These values are only available for a given History if you specified that envelopes be computed for that History. Otherwise, zero values will be used for that History!* See Subtopic "Analysis Results" (page 327) under Topic "Time-History Analysis" for more information.

For contributing moving-load analyses, the values used are the maximum and minimum values obtained for any vehicle loading of the lanes permitted by the parameters of the analysis.

Each contributing analysis case is multiplied by a scale factor, **sf**, before being included in a Combo.

Four types of Combos are available. For each individual response quantity (force, stress, or displacement component) the two Combo values are calculated as follows:

- Additive type: The Combo maximum is an algebraic linear combination of the maximum values for each of the contributing cases. Similarly, Combo minimum is an algebraic linear combination of the minimum values for each of the contributing cases.
- Absolute type: The Combo maximum is the sum of the larger absolute values for each of the contributing cases. The Combo minimum is the negative of the Combo maximum.

- SRSS type: The Combo maximum is the square root of the sum of the squares of the larger absolute values for each of the contributing cases. The Combo minimum is the negative of the Combo maximum.
- Envelope type: The Combo maximum is the maximum of all of the maximum values for each of the contributing cases. Similarly, Combo minimum is the minimum of all of the minimum values for each of the contributing cases.

Only additive Combos of single-valued analysis cases produce a single-valued result, i.e., the maximum and minimum values are equal. Thus only Combos of Loads, Modes, and other single-valued Combos are single-valued. All other Combos will generally have different maximum and minimum values.

For example, suppose that the values, after scaling, for the displacement at a particular joint are 3.5 for Load Case LL and are 2.0 for Spec QUAKE. Suppose that these two cases have been included in an additive-type Combo called COMB1 and an envelope-type Combo called COMB2. The results for the displacement at the joint are computed as follows:

- COMB1: The maximum is 3.5 + 2.0 = 5.5, and the minimum is 3.5 – 2.0 = 1.5
- COMB2: The maximum is max (3.5, 2.0) = 3.5, and the minimum is min (3.5, –2.0) = –2.0

As another example, suppose that Load Cases GRAV, WINDX and WINDY are gravity load and two perpendicular, transverse wind loads, respectively; and that a response-spectrum analysis named EQ has been performed. The following four Combos could be defined:

- WIND: An SRSS-type Combo of the two wind loads, WINDX and WINDY. The maximum and minimum results produced for each response quantity are equal and opposite
- GRAVEQ: An additive-type Combo of the gravity load, GRAV, and the response-spectrum results, EQ. The Combo automatically accounts for the positive and negative senses of the earthquake load
- GRAVWIN: An additive-type Combo of the gravity load, GRAV, and the wind load given by Combo WIND, which already accounts for the positive and negative senses of the load
- SEVERE: An envelope-type Combo that produces the worst case of the two additive Combos, GRAVEQ and GRAVWIN

Suppose that the values of axial force in a frame element, after scaling, are 10, 5, 3, and 7 for cases GRAV, WINDX, WINDY, and EQ, respectively. The following results for axial force are obtained for the Combos above:

- WIND: maximum $= \sqrt{5^2 + 3^2} = 5.8$, minimum $= -5.8$
- GRAVEQ: maximum $= 10 + 7 = 17$, minimum $= 10 - 7 = 3$
- GRAVWIN: maximum $= 10 + 5.8 = 15.8$, minimum $= 10 - 5.8 = 4.2$
- SEVERE: maximum $= \max(17, 15.8) = 17$, minimum $= \min(3, 4.2) = 3$

As you can see, using Combos of Combos gives you considerable power and flexibility in how you can combine the results of the various analysis cases.

Moving Load Cases should not normally be added together, in order to avoid multiple loading of the lanes. Additive combinations of Moving Loads should only be defined within the Moving Load Case itself. Therefore, it is recommended that only a single Moving Load be included in any additive-, absolute-, or SRSS-type Combo, whether referenced directly as a Moving Load or indirectly through another Combo. Multiple Moving Loads may be included in any envelope-type Combo, since they are not added.

When Combos are used for design, they may be treated somewhat differently than has been described here for output purposes. For example, every time step in a History may be considered under certain circumstances. Similarly, corresponding response quantities at the same location in a Moving Load case may be used for design purposes. See the *SAP2000 Steel Design Manual* and the *SAP2000 Concrete Design Manual* for more information.

Chapter XVII

P-Delta Analysis

The P-Delta analysis option causes the transverse bending stiffness of all Frame elements to be modified to account for the P-Delta effect under large axial force. This affects the results of all static, dynamic, and bridge analyses performed in the same execution of the program. See Chapters “Static and Dynamic Analysis” (page 291) and “Bridge Analysis” (page 357) for information on these other types of analysis.

Advanced Topics

- Overview
- Geometric Nonlinearity
- The P-Delta Effect
- Equilibrium Equations
- P-Delta Axial Forces
- Iterative Analysis
- Frame Element
- Effect upon Other Analyses
- Buckling
- Practical Application

Overview

The P-Delta analysis option accounts for the effect of a large axial load upon the transverse bending behavior of Frame elements. Axial compression reduces the flexural stiffness of Frame elements, and axial tension stiffens them. This is a type of geometric nonlinearity known as the P-Delta effect. P-delta analysis does not include large-strain or large-rotation effects.

This option is particularly useful for considering the effect of gravity loads upon the lateral stiffness of building structures, as required by certain design codes (ACI 1995; AISC 1994). It can also be used for the analysis of some cable structures, such as suspension bridges, cable-stayed bridges, and guyed towers. Other applications are possible.

P-delta analysis, when requested, is performed before all static, dynamic, and bridge analyses in the same execution of the program. The axial forces in the Frame elements that cause the P-delta effect may be:

- directly specified, and/or
- computed from a specified set of applied static loads

The P-Delta effect essentially modifies the characteristics of the structure, affecting the results of all other analyses performed, including: static Loads, Modes, Specs, bridge influence lines, Moving Loads, Combos, and Histories.

The P-Delta analysis option provides considerable power and flexibility for analyzing the P-Delta effect in buildings, bridges and other structures. To make the most effective and efficient use of this capability, you should fully understand the assumptions and restrictions that underlie the implementation of this feature in SAP2000. Some of the more important assumptions include:

- The P-Delta effect is analyzed only in Frame elements. Other types of elements may still be present in the model.
- Only the large-stress effect of an axial force upon transverse bending and shear deformation is considered.
- All deflections, strains and rotations are assumed to be small.
- The transverse deflected shape of a Frame element is assumed to be cubic in bending and linear in shear between the reduced rigid zone offsets.
- The P-Delta axial forces are assumed to be constant along the element length.

P-Delta analysis under applied loads is iterative in nature, and may considerably increase computation time. Including a P-Delta analysis may make interpretation of

the results more difficult. It is *strongly recommended* that you perform a preliminary linear analysis to check your model for correctness before using the P-Delta analysis option.

Geometric Nonlinearity

When the load acting on a structure and the resulting deflections are small enough, the load-deflection relationship for the structure is linear. For the most part, SAP2000 analyses assume such linear behavior. This permits the program to form the equilibrium equations using the original (undeformed) geometry of the structure. Strictly speaking, the equilibrium equations should actually refer to the geometry of the structure after deformation.

The linear equilibrium equations are independent of the applied load and the resulting deflection. Thus the results of different static and/or dynamic loads can be superposed (scaled and added), resulting in great computational efficiency.

If the load on the structure and/or the resulting deflections are large, then the load-deflection behavior may become nonlinear. Several causes of this nonlinear behavior can be identified:

- **Large-stress effect:** when large stresses (or forces and moments) are present within a structure, equilibrium equations written for the original and the deformed geometries may differ significantly, even if the deformations are very small. The P-Delta effect is an example of this large stress effect.
- **Large-displacement effect:** when a structure undergoes large deformation (in particular, large strains and rotations), the usual engineering stress and strain measures no longer apply, and the equilibrium equations must be written for the deformed geometry. This is true even if the stresses are small.
- **Material nonlinearity:** when a material is strained beyond its proportional limit, the stress-strain relationship is no longer linear. Plastic materials strained beyond the yield point may exhibit history-dependent behavior. Material nonlinearity may affect the load-deflection behavior of a structure even when the equilibrium equations for the original geometry are still valid.

The large-stress and large-displacement effects are both termed geometric (or kinematic) nonlinearity, as distinguished from material nonlinearity. Kinematic nonlinearity may also be referred to as second-order geometric effects.

Other sources of nonlinearity are also possible, including nonlinear loads, boundary conditions and constraints.

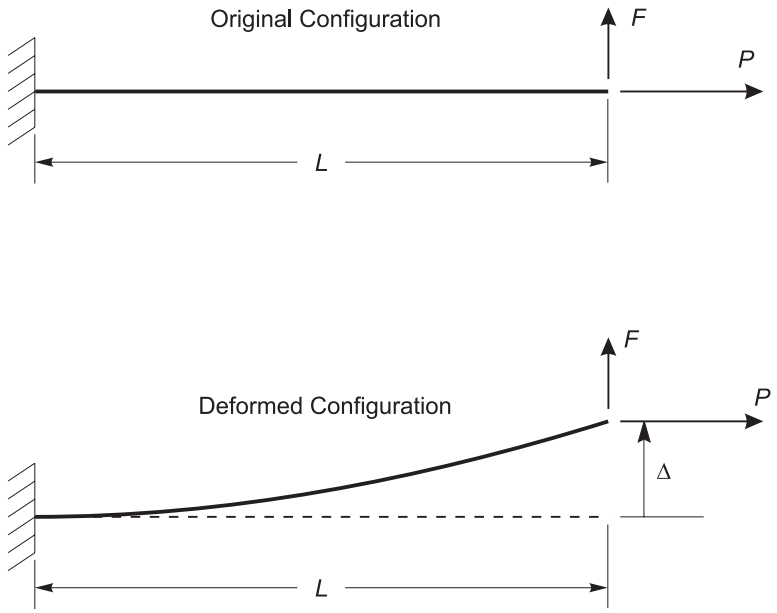


Figure 90
Geometry for Cantilever Beam Example

The P-Delta analysis option can be used to analyze the P-Delta effect, one type of large-stress effect. Other types of large-stress effects are not usually important for structures, and are not considered by the program.

The P-Delta Effect

The **P-Delta effect** refers specifically to the nonlinear geometric effect of a large tensile or compressive direct stress upon transverse bending and shear behavior. A compressive stress tends to make a structural member more flexible in transverse bending and shear, whereas a tensile stress tends to stiffen the member against transverse deformation.

The basic concepts behind the P-Delta effect are illustrated in the following example. Consider a cantilever beam subject to an axial load *P* and a transverse tip load *F* as shown in Figure 90 (page 340). The internal axial force throughout the member is also equal to *P*.

If equilibrium is examined in the original configuration (using the undeformed geometry), the moment at the base is $M = FL$, and decreases linearly to zero at the loaded end. If, instead, equilibrium is considered in the deformed configuration, there is an additional moment caused by the axial force *P* acting on the transverse tip displacement, Δ. The moment no longer varies linearly along the length; the variation depends instead upon the deformed shape. The moment at the base is now $M = FL - P\Delta$. The moment diagrams for various cases are shown in Figure 91 (page 342).

Note that only the transverse deflection is considered in the deformed configuration. Any change in moment due to a change in length of the member is neglected here.

If the beam is in tension, the moment at the base and throughout the member is reduced, hence the transverse bending deflection, Δ, is also reduced. Thus the member is effectively stiffer against the transverse load *F*.

Conversely, if the beam is in compression, the moment throughout the member, and hence the transverse bending deflection, Δ, are now increased. The member is effectively more flexible against the load *F*.

If the compressive force is large enough, the transverse stiffness goes to zero and hence the deflection Δ tends to infinity; the structure is said to have buckled. The theoretical value of *P* at which this occurs is called the Euler buckling load for the beam; it is denoted by P_{cr} and is given by the formula

$$P_{cr} = -\frac{\pi^2 EI}{4L^2}$$

where *EI* is the bending stiffness of the beam section.

The exact P-Delta effect of the axial load upon the transverse deflection and stiffness is a rather complicated function of the ratio of the force *P* to the buckling load P_{cr}. The true deflected shape of the beam, and hence the effect upon the moment diagram, is described by cubic functions under zero axial load, hyperbolic functions under tension, and trigonometric functions under compression.

The P-Delta effect can be present in any other beam configuration, such as simply-supported, fixed-fixed, etc. The P-Delta effect may apply locally to individual members, or globally to the structural system as a whole.

The key feature is that a large axial force, acting upon a small transverse deflection, produces a significant moment that affects the behavior of the member or structure.

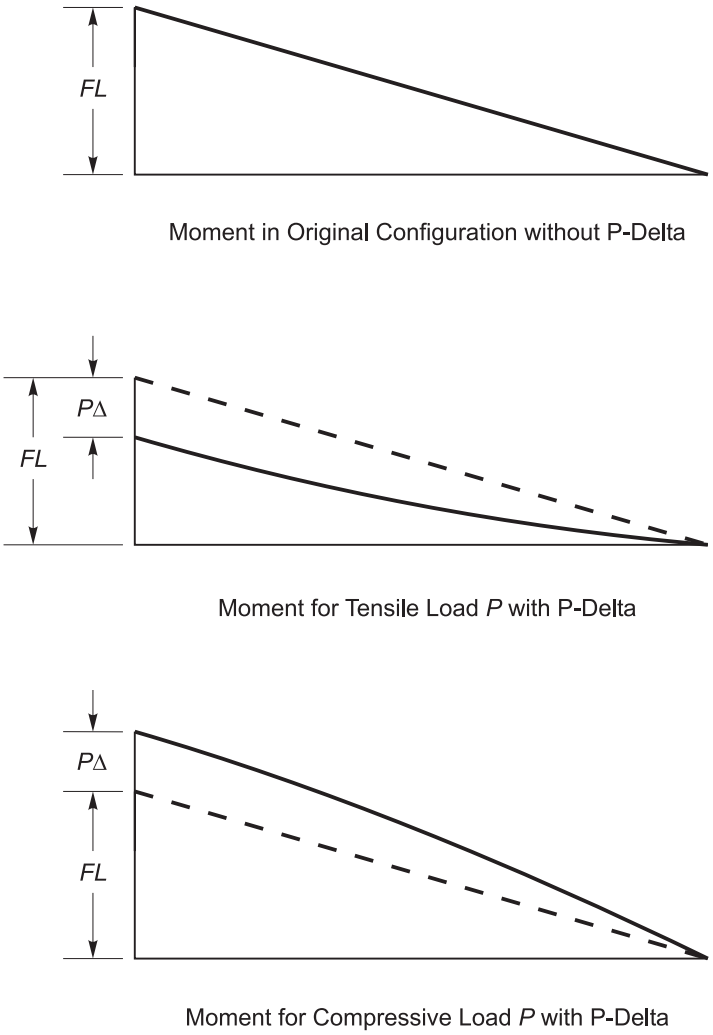


Figure 91
Moment Diagrams for Cantilever Beam Examples

If the deflection is small, then the moment produced is proportional to the deflection.

Equilibrium Equations

The equilibrium equations are manifested in SAP2000 in two ways:

- The element stiffness matrices, which are assembled (combined) to form the overall structure stiffness matrix.
- The element stress-displacement relations, which are used to determine the internal element stresses or forces for output.

Only Frame element stiffness and stress-transformation matrices are modified to include the P-Delta effect.

The P-Delta option may be used with models containing any or all types of elements. For Shell, Plane, Asolid, Solid, and Nllink elements the linear elastic stiffnesses will always be used, and the element stresses will not reflect the P-Delta effect within those elements.

Frame elements should be used to model those portions of any structure that may carry the significant P-Delta loads.

P-Delta Axial Forces

SAP2000 modifies the equilibrium equations to include the P-Delta effect due to a *single* set of axial forces acting in the Frame elements of the model. These modified equations are then used for *all* subsequent analyses that are applied to the structure. The results for these different analyses may then be superposed (except for nonlinear time-history analysis).

These axial forces that create the P-Delta effect may come from two sources:

- You may directly specify the axial forces in the Frame elements
- You may specify a combination of applied static loads from which the axial forces are computed by an iterative analysis

The P-Delta axial forces from these two sources are added together.

The total P-Delta axial force in each Frame element is *only* used to modify the equilibrium equations. It is printed in the output file for each element. It *is not* included in the axial force results that are reported for the various analysis cases in the output file or by the SAP2000 graphical interface.

Directly Specified Axial Forces

Direct specification of the P-delta forces can be used for any structure, but is most appropriate for cable structures where the tensions are large and well-known, and where large deformations of the structure are required to generate these tensions. For other situations, it is generally better to allow the program to compute the axial forces from applied loads.

No iterative analysis is required to include the effect of directly specified P-Delta axial forces.

You may directly specify the P-Delta axial force acting in any Frame element using the following parameters:

- The P-Delta axial force, **p**
- A fixed coordinate system, **csys** (the default is zero, indicating the global coordinate system)
- The projection, **px**, of the P-Delta axial force upon the X axis of **csys**
- The projection, **py**, of the P-Delta axial force upon the Y axis of **csys**
- The projection, **pz**, of the P-Delta axial force upon the Z axis of **csys**

Normally only one of the parameters **p**, **px**, **py**, or **pz** should be given for each Frame element. If you do choose to specify more than one value, they are additive:

$$P_0 = \mathbf{p} + \frac{\mathbf{px}}{c_x} + \frac{\mathbf{py}}{c_y} + \frac{\mathbf{pz}}{c_z}$$

where P_0 is the P-Delta axial force, and c_x, c_y, and c_z are the cosines of the angles between the local 1 axis of the Frame element and the X, Y, and Z axes of coordinate system **csys**, respectively. To avoid division by zero, you may not specify the projection upon any axis of **csys** that is perpendicular to the local 1 axis of the element.

The use of the P-delta axial force projections is convenient, for example, when specifying the tension in the main cable of a suspension bridge, since the horizontal component of the tension is usually the same for all elements.

It is important when directly specifying P-Delta axial forces that you include all significant forces in the structure. The program does not check for equilibrium of the specified P-Delta axial forces. In a suspension bridge, for example, the cable tension is supported at the anchorages, and it is usually sufficient to consider the P-Delta effect only in the main cable (and possibly the towers). On the other hand, the

cable tension in a cable-stayed bridge is taken up by the deck and tower, and it is usually necessary to consider the P-Delta effect in all three components.

P-Delta Load Combination

In a complicated structure, especially one that is redundant, the P-Delta axial forces in the Frame elements may not be known *a priori*. This is typically the case in building structures where the P-Delta effect is due primarily to gravity load.

For these situations, the program can compute the P-Delta axial forces from a single static load combination that you define. This **P-Delta load combination** is an arbitrary combination of the usual static Load Cases. Each Load Case may be scaled before it is added into this load combination. These Load Cases may include any type of loading.

The P-Delta load combination is independent of any Combos that may have been defined. Forces from response-spectrum, time-history and bridge moving-load analyses cannot be included in this load combination.

An iterative analysis is required to determine the P-Delta axial forces in the Frame elements due to the specified P-Delta Load Combinations. It is important that the strains and rotations remain small during this analysis; otherwise, meaningful results may not be obtained.

For more information:

- See Topic “Iterative Analysis” (page 345) in this chapter.
- See Chapter “Load Cases” (page 279).

Iterative Analysis

When a P-Delta load combination is specified, an iterative analysis is required to determine the P-Delta axial forces in the Frame elements. A preliminary analysis is performed to estimate the axial forces throughout the structure. The equilibrium equations are then re-formed and re-solved taking these axial forces into account. This second analysis may produce different axial forces in the members if the modified stiffness causes a force redistribution. Additional iterations, each re-forming and re-solving the equilibrium equations, may be required, until the axial forces and the structural deflections converge, that is, until they do not significantly change from one iteration to the next.

Such an iterative type of solution is typical for nonlinear problems. The type of iteration described above is called "direct iteration" (Bathe 1982; Cook, Malkus, and Plesha 1989; Zienkiewicz and Taylor 1991). Because each iteration requires forming and solving the equilibrium equations, the total solution time may be considerably longer than that required for a linear problem.

Directly specified P-Delta axial forces, if present, are included in every iteration. They are added to the computed P-Delta axial forces. Only the directly specified axial forces are present for the initial iteration.

The final equilibrium equations obtained after convergence of the iteration process will be valid for any loading situation that does not change the axial forces in the members. The results for different loads can then be superposed as for a linear structure.

In general, however, the axial forces throughout a structure may depend upon the applied loads. In such a case, a separate iterative analysis may be required for each set of applied loads, and the results of these analyses should not be superposed.

Convergence Criterion

You may specify a relative displacement tolerance, **told**, which measures convergence. The default value is 10^{-3}. If the relative change in displacement from one iteration to the next is less than the tolerance, then no further iterations are performed. The relative change in displacement is defined as the ratio of the maximum change in displacement to the largest displacement in either iteration. Note that rotations and translations are treated equally.

Maximum Number of Iterations

You may specify the maximum number of iterations, **itmax**, that the program is allowed to perform. This is used to prevent excessive computational time, since each iteration requires about as much computational effort as a linear static analysis.

The initial iteration is termed the **zero-th** iteration. It is a standard linear analysis that is always performed whether or not a P-Delta load combination is specified. Parameter **itmax** limits the number of *additional* iterations performed that correct for the effect of the P-Delta load combination. Setting **itmax** to zero is the same as not specifying the P-Delta load combination.

Setting **told** to zero will force all **itmax** additional iterations to be performed.

Convergence Failure

If convergence has *not* been obtained after the maximum number of iterations has been performed, then the results of the analysis may be meaningless, and they should be viewed with great skepticism. Failure to converge may be due to several causes:

- Too few iterations were permitted. A reasonable number is usually 2 to 5, although more may be required, depending on the particular problem at hand.
- A convergence tolerance that is too small is used. A reasonable value depends on the particular problem. Beware, however, that using a value that is too large may result in convergence to meaningless results.
- The structure is near buckling. The structure should be stiffened against buckling, or the magnitude of the P-Delta load combination reduced.

Frame Element

The implementation of the P-Delta effect in the Frame element is described in the following subtopics.

Small Deflections

All deflections are assumed to be small. In particular, the magnitude of all strains and rotations are assumed to be very much less than unity.

The P-Delta effect does not capture any change in *axial* stiffness of the member, such as might occur after buckling. This would require consideration of large-displacement effects.

Cubic Deflected Shape

The P-Delta effect is integrated along the length of each Frame element, taking into account the deflection within the element. For this purpose the transverse deflected shape is assumed to be cubic for bending and linear for shear between the rigid ends of the element. The length of the rigid ends is the product of the rigid-end factor and the end offsets, and is usually zero. See Topic “End Offsets” (page 162) in Chapter “The Frame Element” for more information.

The true deflected shape may differ somewhat from this assumed cubic/linear deflection in the following situations:

- The element has non-prismatic Section properties. In this case the P-Delta deflected shape is computed as if the element were prismatic using the average of the properties over the length of the element
- Loads are acting along the length of the element. In this case the P-Delta deflected shape is computed using the equivalent fixed-end forces applied to the ends of the element.
- A large P-force is acting on the element. The true deflected shape is actually described by trigonometric functions under large compression, and by hyperbolic functions under large tension.

The assumed cubic shape is usually a good approximation to these shapes except under a compressive P-force near the buckling load with certain end restraints. Excellent results, however, can be obtained by dividing any structural member into two or more Frame elements. See the *SAP2000 Verification Manual* for more detail.

Computed P-Delta Axial Forces

The following discussion applies only to P-Delta axial forces computed from the P-Delta load combination. Directly specified P-Delta axial forces are not affected by any of the following considerations.

The P-Delta axial force in each Frame element is determined from the axial displacements computed in the previous iteration. For meaningful results, it is important to use realistic values for the axial stiffness of these elements. The axial stiffness is determined from the Section properties that define the cross-sectional area and the modulus of elasticity. Using values that are too small may underestimate the P-Delta effect. Using values that are too large may make the P-Delta force in the element very sensitive to the iteration process.

Elements that have an axial force release, or that are constrained against axial deformation by a Constraint, will have a zero P-Delta axial force and hence no P-Delta effect.

The P-Delta axial force also includes loads that act within the element itself. These may include Self-Weight and Gravity Loads, Concentrated and Distributed Span Loads, Prestress Load, and Temperature Load.

The P-Delta axial force is assumed to be constant over the length of each Frame element. If the P-Delta load combination includes loads that cause the axial force to vary, then the average axial force is used for computing the P-Delta effect. If the difference in axial force between the two ends of an element is small compared to

the average axial force, then this approximation is usually reasonable. This would normally be the case for the columns in a building structure. If the difference is large, then the element should be divided into many smaller Frame elements wherever the P-Delta effect is important. An example of the latter case could be a flagpole under self-weight.

For more information:

- See Topic “Section Properties” (page 153) in Chapter “The Frame Element.”
- See Topic “End Releases” (page 165) in Chapter “The Frame Element.”
- See Chapter “Constraints and Welds” (page 113).

Prestress

When Prestress Load is included in the P-Delta load combination, the combined tension in the prestressing cables tends to stiffen the Frame elements against transverse deflections. This is true regardless of any axial end releases. Axial compression of the Frame element due to Prestress Load may reduce this stiffening effect, perhaps to zero.

See Topic “Prestress Load” (page 174) in Chapter “The Frame Element” for more information.

Effect upon Other Analyses

The P-Delta effect essentially modifies the characteristics of the structure, affecting the results of all other analyses performed, including: static Loads, Modes, Specs, Histories, bridge influence lines, Moving Loads, and Combos. Because the same constant stiffness matrix is used, these analyses are all linear, resulting in great computational efficiency, and permitting superposition of the results (except for nonlinear time-history analysis).

Dynamic Analyses

Eigenvectors and Ritz-vectors are obtained using the stiffness matrix as modified for the P-Delta effect. Compressive P-Delta axial forces tend to soften the structure, lengthening the periods of vibration. Tensile axial forces tend to shorten the periods. In structures where the P-Delta axial forces are both compressive and tensile, the effect on the periods is not easily predicted.

Response-spectrum analyses and time-history analyses are performed using these eigenvectors or Ritz vectors as a basis. Thus the static P-Delta effect is automatically included. Any additional P-Delta effect due to dynamic axial forces is not accounted for, since this would require many separate, nonlinear P-Delta analyses taking great computational effort.

See Chapter “Static and Dynamic Analysis” (page 291) for more information.

Harmonic Steady-State Analysis

Direct specification of the P-Delta axial forces may be used with harmonic steady-state analysis. These forces are assumed to be constant during the period of loading.

You should not specify a P-Delta load combination since this load would be harmonically varying, causing a varying P-Delta effect that is not accounted for by the program.

See Topic “Harmonic Steady-State Analysis” (page 294) in Chapter “Static and Dynamic Analysis” for more information

Bridge Moving-Load Analysis

Bridge influence lines and moving-load response are computed using the stiffness matrix as modified for the P-Delta effect. The additional P-Delta effect due to vehicle moving loads is not accounted for, since this would require many separate nonlinear P-Delta analyses taking great computational effort.

If the axial forces due to vehicle moving loads are significant in comparison to the dead-load axial forces, it may be necessary to specify P-Delta axial forces that include some average of the moving-load forces. This may require a prior SAP2000 Moving-Load analysis to determine these values.

See Chapter “Bridge Analysis” (page 357) for more information.

Buckling

If compressive P-Delta axial forces are present and are large enough, the structure may buckle. Local buckling of individual members or global buckling of the whole structure are possible. The program makes no distinction between local and global buckling.

If the program detects that buckling has occurred, the analysis is terminated and no results are produced. This is because the analysis of a structure that has buckled requires consideration of large-displacement effects that are not modeled by SAP2000.

Detection of Buckling

Buckling may be detected during any iteration at one of two possible stages:

- During the formation of the stiffness matrix: if a sufficiently large, compressive P-Delta axial force is present in a Frame element, a negative value may be created on the diagonal of the stiffness matrix.
- During the solution of the equilibrium equations: a zero or negative value may be produced on the diagonal of the stiffness matrix.

In either case, the analysis will be terminated immediately.

Estimating the Buckling Load

SAP2000 does not provide a direct method of determining the buckling load of a structure. It may be estimated, however, by performing a series of runs, each time increasing the magnitude of the P-Delta load combination, until buckling is detected. It is important to keep unchanged the *relative* contributions from each Load Case to the P-load, increasing all Load Case scale factors by the same amount between runs.

It is important to understand that there is no single buckling load for a structure. Rather, there is a different buckling load corresponding to each spatial distribution of loads. If buckling of the structure is a concern under various loading situations, the buckling load should be estimated separately for each situation, as described above, by starting with different P-Delta load combinations.

Local Buckling

The buckling of individual members within a structure can be detected provided that they are adequately modeled. For some support conditions, a single Frame element may adequately represent a structural member. However, for best results for all support conditions, two Frame elements should be used to represent any structural member for which buckling may be a problem. The improvement obtained by using more than two elements does not usually warrant the additional effort.

Practical Application

This section provides some additional guidelines regarding practical use of the P-Delta analysis option. See also the *SAP2000 Verification Manual* for example problems.

Preliminary Linear Analysis

After the structural model is developed, including the basic Load Cases, it is *strongly recommended* that a linear, static analysis be performed before running any P-Delta analyses. The results of this preliminary static analysis should be used to check the model for correctness before incorporating the complexity of the nonlinear P-Delta effect.

Examination of the deflected shape and the element forces and stresses will help detect errors in geometry, element connectivity, material properties, restraints and constraints.

It is useful to define a Combo that is equivalent to the P-Delta load combination. The Frame element axial force distribution due to this Combo should be examined to make sure that the axial stiffnesses are reasonable, and that the axial force does not vary too much within any single Frame element.

Building Structures

For most building structures, especially tall buildings, the P-Delta effect of most concern occurs in the columns due to gravity load, including dead and live load. The column axial forces are compressive, making the structure more flexible against lateral loads.

Building codes (ACI 1995; AISC 1994) normally recognize two types of P-Delta effects: the first due to the overall sway of the structure and the second due to the deformation of the member between its ends. The former effect is often significant; it can be accounted for fairly accurately by considering the total vertical load at a story level, which is due to gravity loads and is unaffected by any lateral loads. The latter effect is significant only in very slender columns or columns bent in single curvature (not the usual case); this requires consideration of axial forces in the members due to both gravity and lateral loads.

SAP2000 can analyze both of these P-Delta effects. However, it is recommended that the former effect be accounted for in the SAP2000 analysis, and the latter effect be accounted for in design by using the applicable building-code moment-

magnification factors (White and Hajjar 1991). This is how the SAP2000 design processors for steel frames and concrete frames are set up.

The P-Delta effect due to the sway of the structure can be accounted for accurately and efficiently, even if each column is modeled by a single Frame element, by using the factored dead and live loads as the P-Delta load combination. The iterative P-Delta analysis should converge rapidly, usually requiring only a single iteration.

As an example, suppose that the building code requires the following load combinations to be considered for design:

(1) 1.4 dead load

(2) 1.2 dead load + 1.6 live load

(3) 1.2 dead load + 0.5 live load + 1.3 wind load

(4) 1.2 dead load + 0.5 live load – 1.3 wind load

(5) 0.9 dead load + 1.3 wind load

(6) 0.9 dead load + 1.3 wind load

For this case, the P-Delta effect due to overall sway of the structure can usually be accounted for, conservatively, by specifying the P-Delta load combination to be 1.2 times the dead load plus 0.5 times the live load. This will accurately account for this effect in load combinations 3 and 4 above, and will conservatively account for this effect in load combinations 5 and 6. This P-Delta effect is not generally important in load combinations 1 and 2 since there is no lateral load.

The P-Delta effect due to the deformation of the member between its ends can be accurately analyzed only when separate SAP2000 runs are made for each load combination above, with the P-Delta load combination specified equal to the combination being analyzed. Six runs would be needed for the example above. Also, at least two Frame elements per column should be used. Again, it is recommended that this effect be accounted for instead by using the SAP2000 design processors.

Cable Structures

The P-Delta effect can be a very important contributor to the stiffness of suspension bridges, cable-stayed bridges, and other cable structures. The lateral stiffness of cables is due almost entirely to tension, since they are very flexible in bending when unstressed.

The easiest way to model this is by directly specifying the P-Delta axial forces if they are known. When this is not possible, a P-Delta load combination can be used provided that the cable geometry does not change too much upon loading.

In many cable structures, the tension in the cables is due primarily to gravity load, and it is relatively unaffected by other loads. If this is the case, it is appropriate to define the P-Delta load combination to be a realistic combination of the dead load and live load. It is important to use realistic values for the P-Delta load combination, since the lateral stiffness of the cables is approximately proportional to the P-Delta axial forces.

Frame elements are used to model cables. A single element is sufficient between points of concentrated load. Additional elements may be needed if significant distributed loads, including self weight, act upon the cable. Concentrated loads should only be applied at joints, not as Concentrated Span loads, since cables "kink" at such loads.

Each Frame cable element should be given a small, realistic bending stiffness. Otherwise the structure may be unstable in the zero-th iteration before the tensile P-Delta axial forces can provide lateral stiffness. For the same reason, moment end-releases should generally not be used for cable elements.

The geometry of a loaded cable is strongly dependent upon the type of loading applied. Because SAP2000 only considers small deflections, it is important to define the cable geometry (joint coordinates) to be close to what is expected after the structure is loaded. It may be necessary to correct the geometry after one or more preliminary runs that determine the shape of the cable under the P-Delta load combination. If the stretching or rotation of the cable is large (say more than a few hundredths of a percent) it may not be possible to obtain meaningful results with a P-Delta load combination.

The P-Delta effect only affects transverse stiffness, not axial stiffness. Therefore, Frame elements representing a cable can carry compression as well as tension; this type of behavior is generally unrealistic. You should review the analysis results to make sure that this does not occur.

Because convergence tends to be slower for stiffening than softening structures, the nonlinear P-Delta analysis may require many iterations. Five to ten iterations would not be unusual.

Guyed Towers

In guyed towers and similar structures, the cables are under a large tension produced by mechanical methods that shorten the length of the cables. These structures can be analyzed by the same methods discussed above for cabled bridges.

A Temperature load causing a decrease in the temperature of the cables can be used to produce the requisite shortening. The P-Delta load combination should include this load, and may also include other loads that cause significant axial force in the cables, such as gravity and wind loads. Several analyses may be required to determine the magnitude of the temperature change needed to produce the desired amount of cable tension.

Alternatively, the P-Delta axial force can be directly specified in the cables and in the tower.

Chapter XVIII

Bridge Analysis

Bridge Analysis can be used to compute influence lines for traffic lanes on bridge structures and to analyze these structures for the response due to vehicle live loads.

Advanced Topics

- Overview
- Modeling the Bridge Structure
- Roadways and Lanes
- Spatial Resolution
- Influence Lines
- Vehicles
- Vehicle Classes
- Moving Load Cases
- Influence Line Tolerance
- Exact and Quick Response Calculation
- Moving Load Response Control
- Correspondence
- Computational Considerations

Overview

Bridge Analysis can be used to determine the response of bridge structures due to the weight of Vehicle live loads. Considerable power and flexibility is provided for determining the maximum and minimum displacements and forces due to multiple-lane loads on complex structures, such as highway interchanges. The effects of Vehicle live loads can be combined with static and dynamic loads, and envelopes of the response can be computed.

The bridge to be analyzed is modeled with Frame elements representing the superstructure, substructure and other components of interest. Displacements, reactions, spring forces, and Frame-element internal forces can be determined due to the influence of Vehicle live loads. Other element types (Shell, Plane, Asolid, Solid, and Nllink) may be used; they contribute to the stiffness of the structure, but they are not analyzed for the effect of Vehicle load.

Lanes are defined on the superstructure that represent where the live loads can act. These Lanes need not be parallel nor of the same length, so that complex traffic patterns may be considered. The program computes conventional influence lines for all response quantities due to the loading of each Lane. These influence lines may be displayed using the SAP2000 graphical interface.

You may select Vehicle live loads from a set of standard highway and railway Vehicles, or you may create your own Vehicle live loads. Vehicles move in both directions along each Lane of the bridge. Vehicles are automatically located at such positions along the length of the Lanes to produce the maximum and minimum response quantities throughout the structure. Each Vehicle live load may be allowed to act on every lane or be restricted to certain lanes. The program can automatically find the maximum and minimum response quantities throughout the structure due to placement of different Vehicles in different Lanes.

For each maximum or minimum extreme response quantity, the corresponding values for the other components of response can also be computed.

In summary, the procedure to perform a Bridge Analysis is to:

- Model the structural behavior of the bridge with Frame elements
- Define traffic Lanes describing where the Vehicle live loads act
- Define the different Vehicle live loads that may act on the bridge
- Define Vehicle Classes (groups) containing one or more Vehicles that must be considered interchangeably

- Define Moving Load cases that assign Vehicle Classes to act on the traffic Lanes in various combinations
- Specify for which joints and Frame elements the Moving Load response is to be calculated

The most extreme (maximum and minimum) displacements, reactions, spring forces, and Frame element internal forces are automatically computed for each Moving Load case defined.

Modeling the Bridge Structure

You should model the bridge structure primarily with Frame elements as described in the following.

Frame Elements

In simple cases you may define a "two-dimensional" model with longitudinal elements representing the superstructure and roadway, and vertical elements representing the piers and supports. For curved bridge structures these Frame elements need not exist in a single plane. Elements directed in the third, transverse direction may also be used for modeling the bents and other features. Figure 92 (page 360) shows an example of a bridge model.

Specify appropriate Section properties to represent the total effective-stiffness properties of the superstructure and substructure members. These elements should be placed along the neutral axis of the members they represent.

The results of the Bridge Analysis will report the Frame element internal forces and moments which can then be used to design the actual sections. Moving-load response will only be calculated for those elements you specifically request.

For more information:

- See Topic "Local Coordinate System" (page 148) in Chapter "The Frame Element."
- See Topic "Section Properties" (page 153) in Chapter "The Frame Element."
- See Topic "Vehicles" (page 372) in this chapter.
- See Topic "Moving Load Response Control" (page 392) in this chapter.

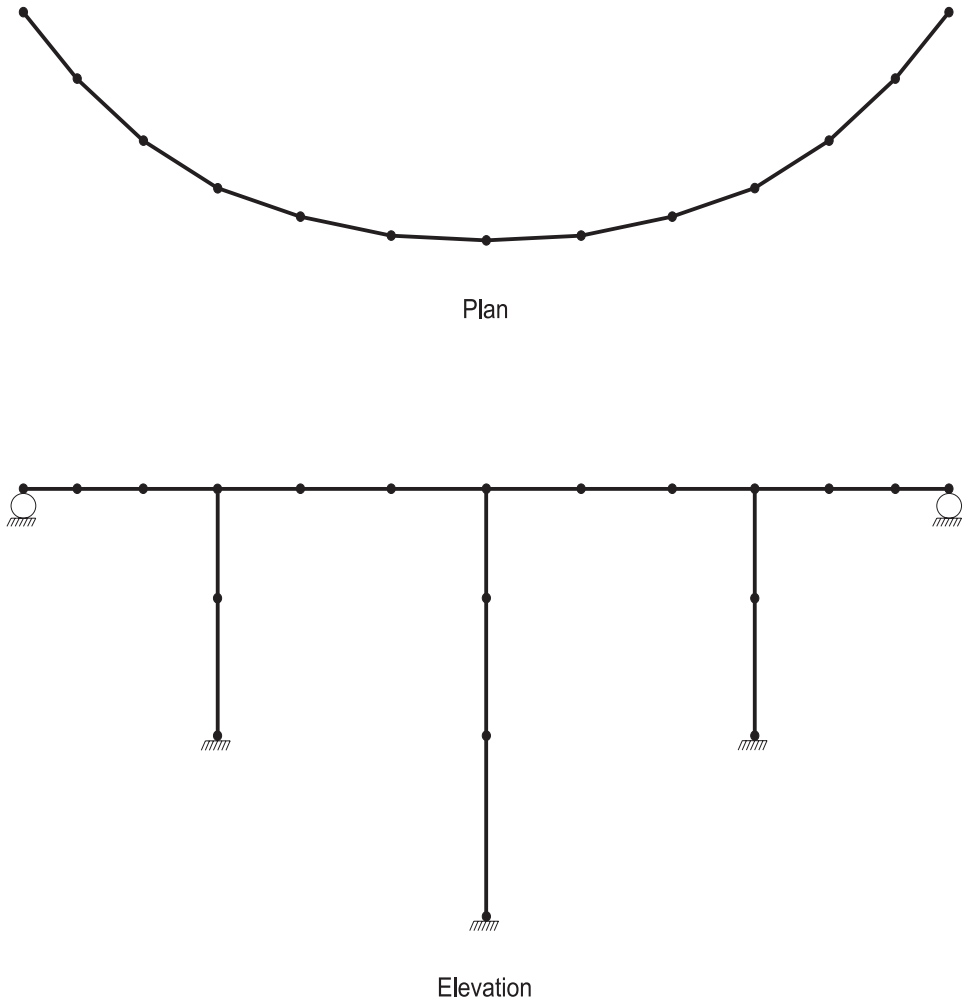


Figure 92
Frame Element Model of a Bridge Structure

Supports

Supports can be modeled using either springs or restraints. Moving-load response will only be calculated for those springs or restraints you specifically request.

See Topic “Moving Load Response Control” (page 392) in this chapter for more information.

Bearings and Expansion Joints

Effective modeling of support conditions at bearings and expansion joints requires careful consideration of the continuity of each translational and rotational component of displacement. Continuous components require that the corresponding degrees-of-freedom remain connected across the bearing or expansion joint; degrees-of-freedom representing discontinuous components must be disconnected. You can achieve this by two principal methods:

(1) Attaching elements to separate joints at the same location (which automatically disconnects all degrees-of-freedom between the elements) and constraining together the connected degrees-of-freedom using an Equal or Local Constraint, or

(2) Attaching several elements to a common joint (which automatically connects the degrees-of-freedom between the elements) and using Frame element end releases to free the unconnected degrees-of-freedom.

Both methods are acceptable for static analysis. For dynamic analysis, method (1) is recommended since method (2) does not properly distribute the mass on either side of the joint.

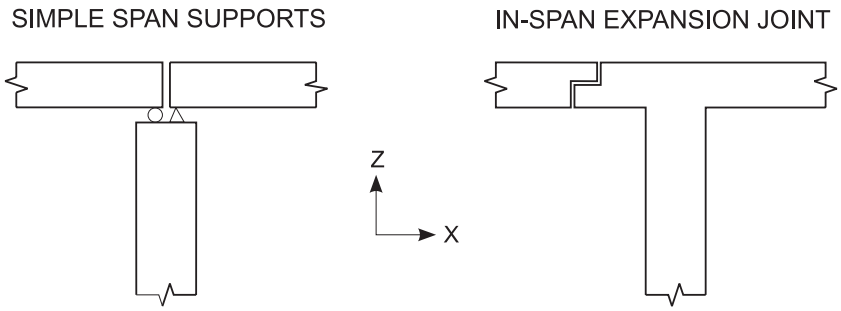
Typically the vertical and transverse translations and the torsional rotation would be connected, while the longitudinal translations and the bending and in-plane rotations would be disconnected. However, the appropriate use of constrained or released degrees-of-freedom depends on the details of each individual bearing or joint. See Figure 93 (page 362) for examples.

For more information:

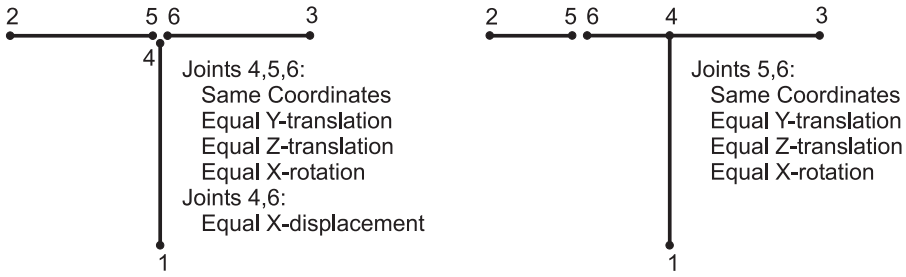
- See Topic “End Releases” (page 165) in Chapter “The Frame Element.”
- See Topics “Equal Constraint” (page 123) and “Local Constraint” (page 127) in Chapter “Constraints and Welds.”

Other Element Types

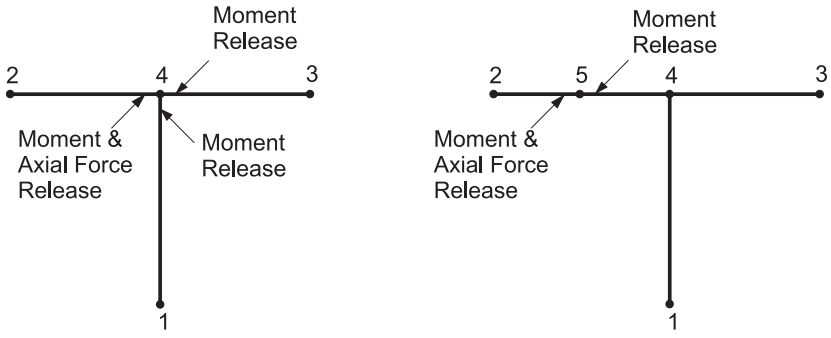
Shell, Plane, Asolid, Solid, and Nllink elements should not generally be used in models subjected to Vehicle loads. If you do use these types of elements, you should do so with caution and with complete understanding of the following implications:



(a) Bridge Decks and Piers



(b) Use of Constraints on Separate Joints at Common Locations



(c) Use of Common Joints and Element End Releases

Figure 93
Modeling of Bearings and Expansion Joints

- Vehicle live loads can only be applied to Frame elements. Thus live loads cannot be represented as acting directly on bridge decks modeled with Shell or other element types.
- All elements present in the structure contribute to the stiffness and may carry part of the load. However, element internal forces (stresses) due to Vehicle live loads are computed only for Frame elements. Therefore, the presence of other element types may result in an underestimate of the internal forces in Frame elements if these are intended to represent the complete behavior of the substructure or superstructure. The corresponding response in the other element types will not be reported. This approach may be unconservative for all element types.

Roadways and Lanes

The Vehicle live loads are considered to act in traffic Lanes transversely spaced across the bridge roadway. These Lanes are supported by Frame elements representing the bridge deck. The number of Lanes and their transverse spacing can be chosen to satisfy the appropriate design-code requirements. For simple bridges with a single roadway, the Lanes will usually be parallel and evenly spaced, and will run the full length of the bridge structure.

For complex structures, such as interchanges, multiple roadways may be considered; these roadways can merge and split. Lanes need not be parallel nor be of the same length. The number of Lanes across the roadway may vary along the length to accommodate merges.

Roadways

Typically each roadway is modeled with a single string (or chain) of Frame elements running along the length of the roadway. These elements should possess Section properties representing the full width and depth of the bridge deck. They are modeled as a normal part of the overall structure and are not explicitly identified as being roadway elements.

Lanes

A traffic **Lane** on a roadway has its length represented by a consecutive set of some or all of the roadway elements. The transverse position of the Lane center line is specified by its eccentricity relative to the roadway elements. Each Lane across the roadway width will usually refer to the same set of roadway elements, but will typi-

cally have a different eccentricity. The eccentricity for a given Lane may also vary along the length.

A Lane is thus defined by listing, in sequence, the labels of a chain of Frame elements that already exist as part of the structure. Each Lane is said to "run" in a particular direction, namely from the first element in the listed sequence to the second element, and so on, to the last element. This direction may be the same or different for different Lanes using the same roadway elements, depending on the order in which each Lane is defined. It is independent of the direction that traffic travels.

Eccentricities

The sign of a Lane **eccentricity** is defined as follows: in an elevation view of the bridge where the Lane runs from left to right, Lanes located behind the roadway elements have positive eccentricity. Alternatively, to a driver traveling on the roadway in the direction that the Lane runs, a Lane to the left of the roadway elements has a positive eccentricity.

The use of eccentricities is primarily important for the determination of axial torsion in the bridge deck and transverse bending in the substructure; secondary effects may also be found in more complex structures. Although the modeling of lane eccentricities is generally realistic and advantageous, some savings in computation time, memory requirements, and disk storage space can be realized by using zero eccentricities for all elements in all Lanes.

Modeling Guidelines

Although roadway elements are not explicitly defined as such, they can be identified as those Frame elements in the structure that are referred to by one or more Lane definitions. Since the Vehicle live loads will be applied to the roadway Frame elements through the use of the Lanes, the modeling of roadway elements should adhere to the following guidelines:

- They should be located at the neutral axis of the bridge deck
- They should be parallel to the direction of traffic, or approximately so
- They should form one or more (nearly) contiguous chains of elements. To be contiguous, the end of one element should be located at the beginning of the next element in the chain. The two elements may be attached to the same joint, or to two different joints at the same location. If they are not contiguous, the gap between adjacent elements should be small, especially in the longitudinal direction; gaps in the transverse and vertical direction are not usually significant

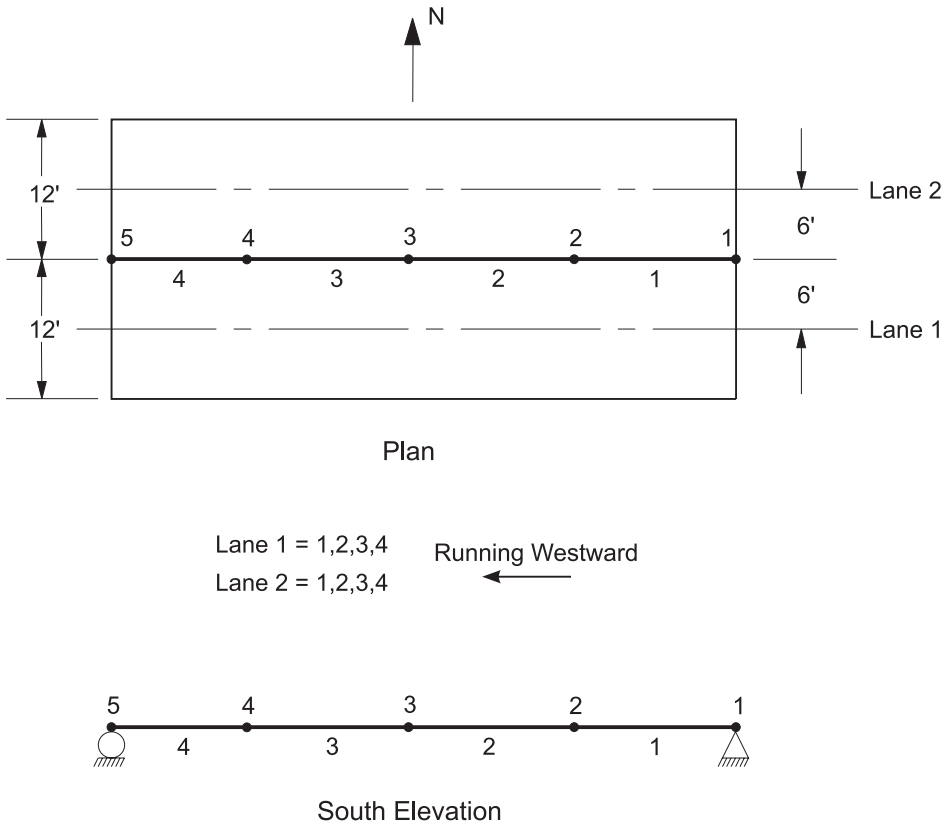


Figure 94
Definition of Lanes for a Simple Bridge

- They must not be vertical

Each Lane should be a consecutive set of some or all of the roadway elements, chosen to form a (nearly) contiguous chain or path

Examples

Figure 93 (page 365) shows a simple 24 ft wide bridge carrying two opposing 12 ft traffic Lanes. The roadway, and also each traffic Lane, are modeled by four Frame elements (1, 2, 3, 4) running along the center line of the bridge from east to west.

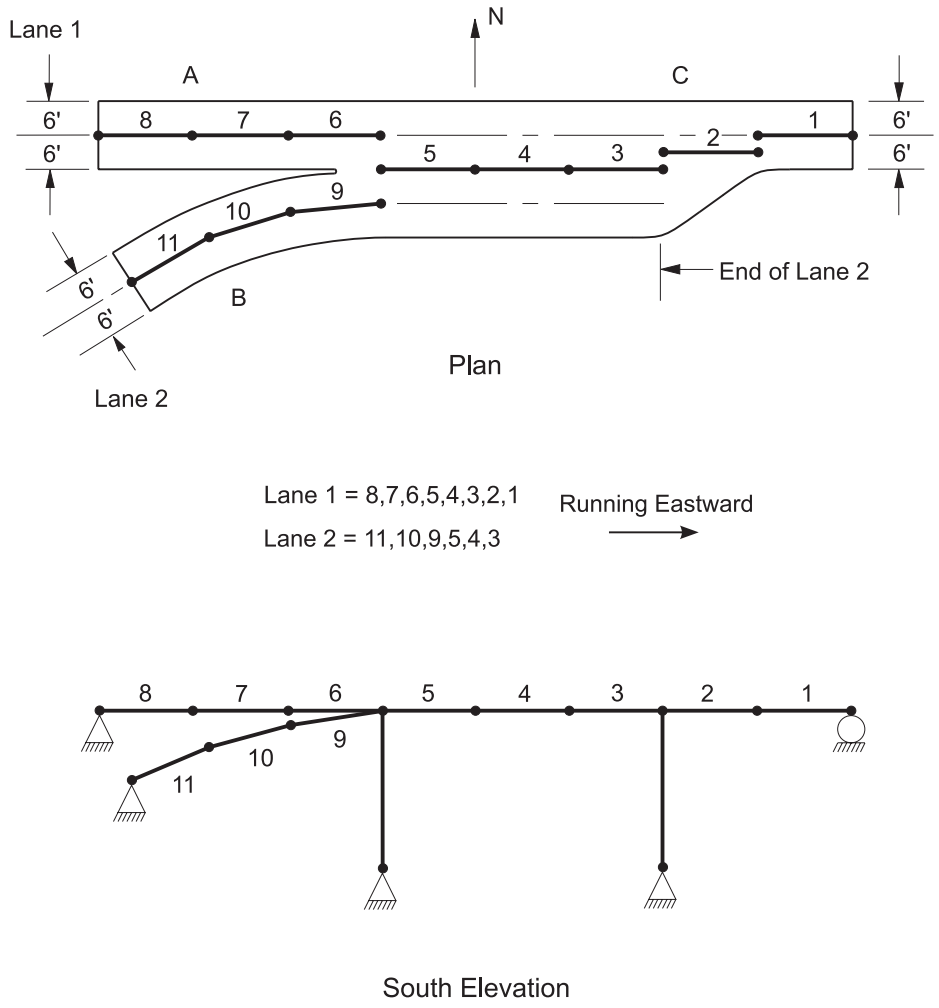


Figure 95
Definition of Lanes for a Two-Roadway Merge

The eccentricities are constant at +6 ft and –6 ft for the eastbound and westbound Lanes, respectively.

A second example showing a simple portion of an interchange is presented in Figure 94 (page 366). Here two 12 ft wide roadways (A and B) merge into a single 24 ft wide roadway (C), which then tapers down to a 12 ft width. Each roadway is

modeled with a single chain of elements. Elements representing the substructure and other structural members are not shown.

Two 12 ft wide traffic Lanes are defined: the first runs the full length of roadways A and C; the second runs the full length of roadway B and the 24 ft wide portion of roadway C. The chain of elements defining the first Lane is: 8, 7, 6, 5, 4, 3, 2, 1; the chain defining the second Lane is: 11, 10, 9, 5, 4, 3. Both Lanes run eastward. The eccentricities at the centers of the elements are zero except for: +3 ft for element 2 and +6 ft for elements 3, 4 and 5 in the first Lane; and –6 ft for elements 3, 4 and 5 in the second Lane.

A significant transverse gap exists between element 5 and elements 6 and 9 in Lanes 1 and 2, respectively. Significant transverse gaps also exist on either side of element 2, which is taken to be parallel to the direction of traffic in order to obtain the most meaningful definitions for moments and torque. Since no *longitudinal* gaps exists, the Vehicle loads will be modeled adequately. However, appropriate stiffness connections must still be made to tie the roadways together at the gaps. This can be done using elements or rigid Body Constraints.

See Topic “Body Constraints” (page 115) in Chapter “Constraints and Welds” for more information.

Spatial Resolution

The accuracy of the Bridge Analysis is determined by the spatial resolution (the number of load and response points) of the Lanes.

Load and Output Points

The program applies the Vehicle loads at a finite set of fixed **load points** along the traffic Lanes. Likewise, Frame element internal forces are computed and output at fixed **output points** along all Frame elements. The accuracy of a Vehicle live-load analysis is dependent upon the resolution, i.e., the number of load points and output points used. Increasing the resolution increases the likelihood of obtaining accurate values and locations for the maximum and minimum displacements and forces in the structure; it also significantly increases computation time, memory, and disk storage requirements.

The number of load and output points for each Frame element is determined by the number, **nseg**, of output segments specified for that element:

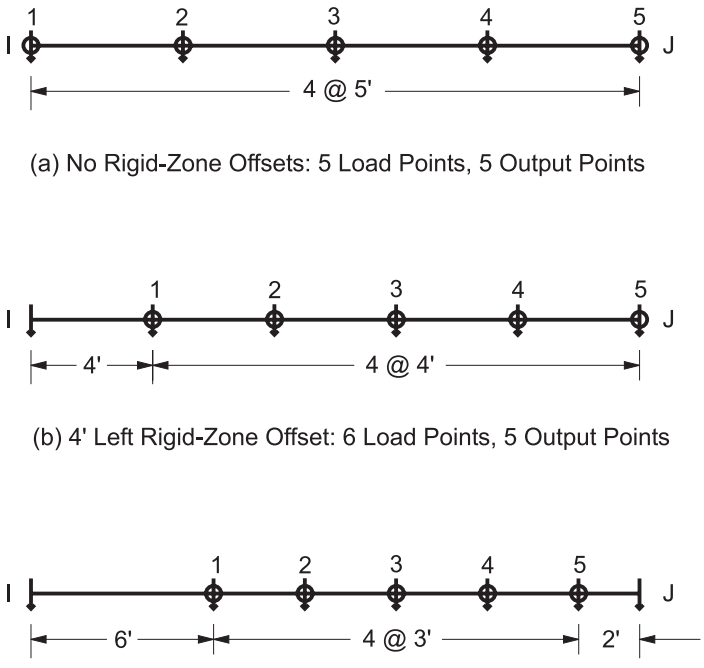


Figure 96
*Load and Output Points for a Single Frame Element with **nseg** = 4*

- The output points for each element are the **nseg**+1 equally spaced points along the *clear length* of the element. The clear length is that length between the rigid zone offsets at either end of the element, if any.
- The load points for a Lane element are the same as the output points, plus an additional point at each joint of the element wherever there is a non-zero rigid zone offset. Thus the number of load points will be from **nseg**+1 to **nseg**+3. See Figure 95 (page 368) for an example. Only Lane elements possess load points.

Displacements, reactions, and spring-forces are only available at the joints. The accuracy of these results may still be dependent upon the number of load points.

See Topic “Internal Force Output” (page 176) in Chapter “The Frame Element” for more information.

Resolution

The resolution of a live-load analysis may be increased by increasing the number of Frame elements, the number of output segments for each Frame element, or both.

Increasing the number of output segments, **nseg**, is the simplest way to increase the resolution; it is also the most computationally efficient. See Topic “Computational Considerations” (page 393) in this chapter for more information.

Increasing the number of Frame elements as a way to increase resolution is not generally recommended. However, other factors may govern the number of Frame elements that need to be used in a given region, such as:

- Curved roadways: more than one element may be needed between supports, diaphragms, or cross-braces. You should experiment to determine the number of elements required to adequately represent the stiffness and loading of the curved roadway.
- Dynamic analyses: element masses are lumped at the joints, hence a sufficient number of Frame elements must be defined to represent the governing vibration modes of the structure
- Non-constant Lane eccentricities

For example, a simple span represented by a single element with **nseg**=8 has the same effective number of load and response points as four elements with **nseg**=2. This is true because load and response points at the interior joints are duplicated in the latter case. Both meshes have the same resolution for live-load analysis and will produce the same results. The former mesh is easier to define and will be somewhat more efficient. However, the latter mesh is superior when dynamics are to be considered.

Modeling Guidelines

A suggested approach for achieving adequate resolution for live-load analysis is as follows:

- Devise a SAP2000 bridge model that uses an appropriate number of Frame elements to capture significant structural behavior and inertia properties, and to properly represent the traffic lanes and spans

- Perform a preliminary analysis using a moderate number of output segments (say **nseg**=2 for all elements) to assess the correctness of the model and to check for adequate local resolution
- Correct the model as required, adding more Frame elements if necessary
- Perform another analysis using the corrected model with an increased number of output segments, **nseg**, where needed
- Repeat the previous steps as necessary

Checking the model for adequate resolution should ideally be done using the SAP2000 graphical interface to examine the influence lines and the Moving Load response. The influence lines will give a more critical view of the resolution of the model, although the Moving Load results are of more practical interest.

Influence Lines

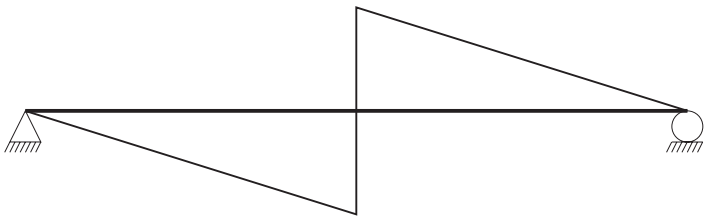
SAP2000 automatically computes influence lines for the following response quantities:

- Frame element internal forces at the output points
- Joint displacements
- Reactions
- Spring forces

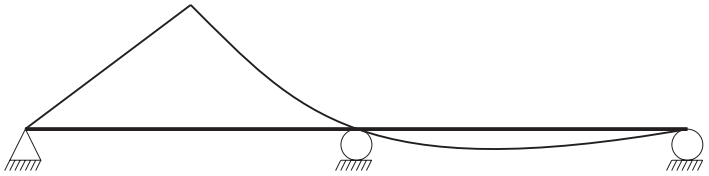
For each response quantity in the structure, there is one influence line for each traffic Lane.

An influence line can be viewed as a curve of influence values plotted at the load points along a traffic Lane. For a given response quantity at a given location in the structure, the influence value at a load point is the value of that response quantity due to a unit concentrated downward force acting at that load point. The influence line thus shows the influence upon the given response quantity of a unit force moving along the traffic lane. Figure 96 (page 371) shows some simple examples of influence lines.

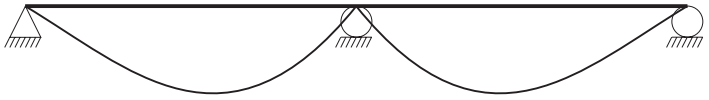
Influence lines may exhibit discontinuities (jumps) at the output point when it is located at a load point on the traffic lane. Discontinuities may also occur where the structure itself is not continuous (e.g., expansion joints).



(a) Influence Line for Vertical Shear at Center
of a Simple Span



(b) Influence Line for Moment at Center of Left Span
of Two Continuous Spans



(c) Influence Line for Moment at Center Support
of Two Continuous Spans

Figure 97
Examples of Influence Lines for One-Span and Two-Span Beams

SAP2000 uses influence lines to compute the response to vehicle live loads. Influence lines are also of interest in their own right for understanding the sensitivity of various response quantities to traffic loads.

Influence lines can be displayed using the SAP2000 graphical user interface. They are plotted along the Lane elements with the influence values plotted in the vertical direction. A positive influence value due to gravity load is plotted upward. Influence values are linearly interpolated between the known values at the load points. Influence values may also be written to a text file from the graphical interface.

Influence lines are available after any analysis for which traffic Lanes were defined. It is not necessary to define Vehicles, Classes, Moving Load cases, or response control in order to get influence lines.

Vehicles

Any number of **Vehicle live loads**, or simply **Vehicles**, may be defined to act on the traffic Lanes. You may use standard types of Vehicles known to the program, or design your own using the general Vehicle specification.

Direction of Loads

All vehicle live loads represent weight and are assumed to act downward, in the –Z global coordinate direction.

See “Upward and Horizontal Directions” (page 23) in Chapter “Coordinate Systems.”

Application of Loads

Each Vehicle consists of one or more concentrated (point) and/or uniform line loads. These act on the center line of the lane, i.e., along lines parallel to the Lane elements, horizontally offset from the Lane elements by the lane eccentricity.

Each concentrated or uniform load is considered to represent a range of values from zero up to a specified maximum. When computing a response quantity (force or displacement) the maximum value of load is used where it increases the severity of the response, and zero is used where the load would have a relieving effect. Thus the specified load values for a given Vehicle may not always be applied proportionally. This is a conservative approach that accounts for Vehicles that are not fully loaded.

The maximum and minimum values of a response quantity are computed using the corresponding influence line. Concentrated loads are multiplied by the influence value at the point of application to obtain the corresponding response; uniform

loads are multiplied by the influence values and integrated over the length of application.

Loads acting in regions of positive influence value only add to the maximum response; they never affect the minimum response. Similarly, loads acting in regions of negative influence value only subtract from the minimum response. Thus the maximum response is always positive (or zero); the minimum response is always negative (or zero).

By way of example, consider the influence line for the moment at the center of the left span shown in Figure 97(b) (page 371). Any concentrated load or portion of a uniform load that acts on the left span would contribute only to the positive maximum value of the moment response. Loads acting on the right span would not decrease this maximum, but would contribute to the negative minimum value of this moment response.

General Vehicle

The general Vehicle may represent an actual vehicle or a notional vehicle used by a design code. Most trucks and trains can be modeled by the SAP2000 general Vehicle.

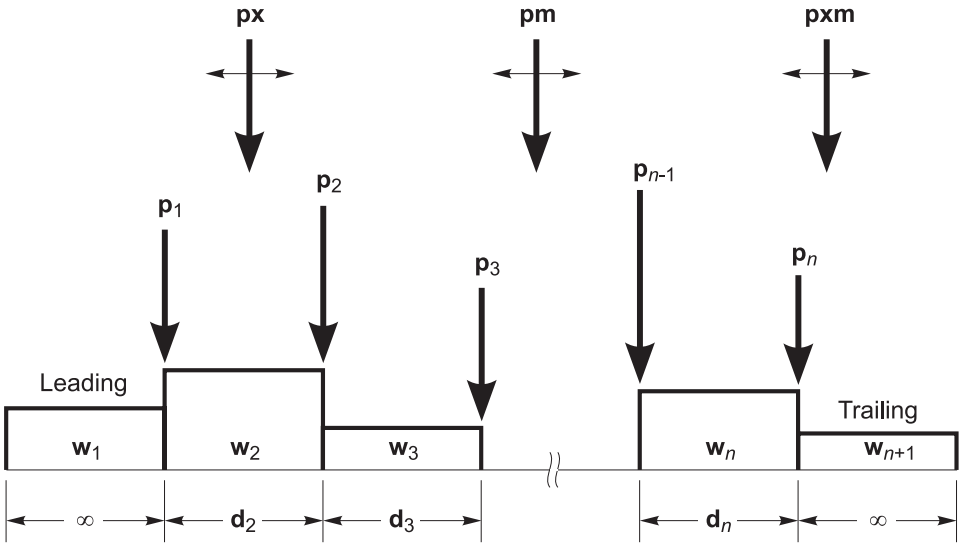
The general Vehicle consists of *n* axles with specified distances between them. Concentrated loads may exist at the axles. Uniform loads may exist between pairs of axles, in front of the first axle, and behind the last axle. The distance between any one pair of axles may vary over a specified range; the other distances are fixed. The leading and trailing uniform loads are of infinite extent. Additional "floating" concentrated loads may be specified that are independent of the position of the axles.

Because the applied loads never decrease the severity of the computed response, the effect of a shorter Vehicle is captured by a longer Vehicle that includes the same loads and spacings as the longer Vehicle. Only the longer Vehicle need be considered in such cases.

Specification

To define a Vehicle, you may specify:

- *n*–1 positive distances, **d**, between the pairs of axles; one inter-axle distance may be specified as a range from **dmin** to **dmax**, where $0 < \mathbf{dmin} \le \mathbf{dmax}$, and **dmax** = 0 is used to represent a maximum distance of infinity
- *n* concentrated loads, **p**, at the axles



Notes:

(1) All loads are point loads or uniform line loads acting on the Lane center line

(2) Any of the point loads or uniform line loads may be zero

(3) The number of axles, *n*, may be zero or more

(4) One of the inter-axle spacings, $\mathbf{d}_2$ through $\mathbf{d}_n$, may vary over a specified range

(5) The locations of loads **px**, **pm**, and **pxm** are arbitrary

Figure 98
General Vehicle Definition

- *n*+1 uniform loads, **w**: the leading load, the inter-axle loads, and the trailing load
- Floating concentrated loads; either:
  - A single floating load, **px**, for all response quantities, or
  - A pair of floating loads:
    * Load **pm** for span moments in the Lane elements. This load receives special treatment for span moments over the supports, as described below, and

    - Load **pxm** for all response quantities except span moments in the Lane elements

- Whether or not this Vehicle is to be used for:
  - "Negative" span moments over the supports in the Lane elements
  - Vertical forces in interior piers and/or interior supports
  - Response quantities other than the two types above

The number of axles, *n*, may be zero, in which case only a single uniform load and the floating concentrated loads can be specified.

These parameters are illustrated in Figure 98 (page 374). Specific examples are given in the next subtopic using the standard Vehicles. Additional detail is provided in the following.

Moving the Vehicle

When a Vehicle is applied to a traffic Lane, the axles are moved along the length of the lane to where the maximum and minimum values are produced for every response quantity in every element. Usually this location will be different for each response quantity. For asymmetric (front to back) Vehicles, both directions of travel are considered.

Vehicle Response Categories

In order to satisfy certain requirements of the AASHTO HL design vehicular live load (AASHTO, 1996 b), the available response quantities are divided into the following categories:

(1) "Negative" span moments over the supports in Lane elements only. A "negative" span moment is defined as that moment which causes tension in the upward-most face of a Lane element:
   - Negative M3 if the local +2 axis is most upward (the default)
   - Positive M3 if the local –2 axis is most upward
   - Negative M2 if the local +3 axis is most upward
   - Positive M2 if the local –3 axis is most upward

   SAP2000 considers all "negative" span moments in the Lane elements to be in this category without regard for the location of the supports (piers).

(2) Reactions at interior supports (piers). This takes two forms:

- A compressive axial force in vertical Frame elements, where vertical is defined here as being within 15° of the Z axis
- The most upward local component of reactions and spring forces. For example:
  - Positive F3 if the joint local +3 axis is most upward (the default)
  - Negative F2 if the joint local –2 axis is most upward

The program automatically determines if these force components correspond to an interior support from the shape of the influence line. If the absolute minimum influence value does not occur at either end of the influence line, the support is considered to be interior.

(3) All other response quantities not included in (1) and (2)

You may specify for each Vehicle whether or not to calculate each of these three categories of response using the parameters **supmom**, **intsup**, and **other**, respectively. Each of these parameters may take either value Y (yes) or N (no). The default is Y for all three. This enables you to define different vehicles for the different categories of response. You must be sure that each category of response is captured by at least one Vehicle in each Lane when defining the Moving Load cases.

Floating Concentrated Loads

The floating concentrated loads (**px**, **pm**, and **pxm**) are placed at the point of maximum positive influence value (if any) and the point of minimum negative influence value (if any).

You may specify either **px**, or the pair **pm** and **pxm**.

Floating load **px** is used equally for all response quantities.

Floating load **pm** is only used for span moments in the Lane elements. A span moments is defined as the moment in the most vertical local plane of a Lane element:

- M3 if the local 2 axis is most vertical (the default)
- M2 if the local 3 axis is most vertical

For “negative” span moments, as defined in category (1) under “Vehicle Response Categories” above, two floating concentrated loads of magnitude **pm** are located in two different spans. This is used to meet the requirements of the AASHTO HS Lane Load (AASHTO, 1996 a) for negative moments in continuous spans. The program automatically determines the spans from the shape of the influence line by using the

two most negative influence values (if any) that are separated by at least one local maximum. A single value of **pm** is used for "positive" span moments.

Floating load **pxm** is used for all response quantities except the span moments in the Lane elements.

Standard Vehicles

The following standard vehicle types are available in SAP2000 to represent vehicular live loads specified in various design codes. The type of vehicle is specified using the parameter **type**.

Hn-44 and HSn-44

Vehicles specified with **type** = H**n**-44 and **type** = HS**n**-44 represent the AASHTO standard H and HS Truck Loads, respectively. The **n** in the **type** is an integer scale factor that specifies the nominal weight of the Vehicle in tons. Thus H15-44 is a nominal 15 ton H Truck Load, and HS20-44 is a nominal 20 ton HS Truck Load. These Vehicles are illustrated in Figure 98 (page 378).

The effect of an H Vehicle is included in an HS Vehicle of the same nominal weight. If you are designing for both H and HS Vehicles, only the HS Vehicle is needed.

Hn-44L and HSn-44L

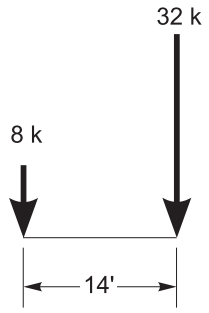
Vehicles specified with **type** = H**n**-44L and **type** = HS**n**-44L represent the AASHTO standard H and HS Lane Loads, respectively. The **n** in the **type** is an integer scale factor that specifies the nominal weight of the Vehicle in tons. Thus H15-44 is a nominal 15 ton H Lane Load, and HS20-44 is a nominal 20 ton HS Lane Load. These Vehicles are illustrated in Figure 98 (page 378). The H**n**-44L and HS**n**-44L Vehicles are identical.

AML

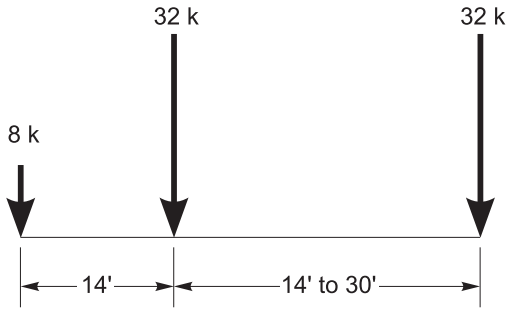
Vehicles specified with **type** = AML represent the AASHTO standard Alternate Military Load. This Vehicle consists of two 24 kip axles spaced 4 ft apart.

HL-93K, HL-93M and HL-93S

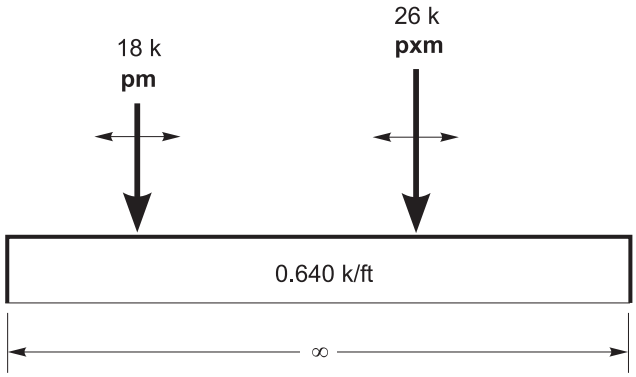
Vehicles specified with **type** = HL-93K represent the AASHTO standard HL-93 Load consisting of the code-specified design truck and the design lane load.



H20-44 Truck Load

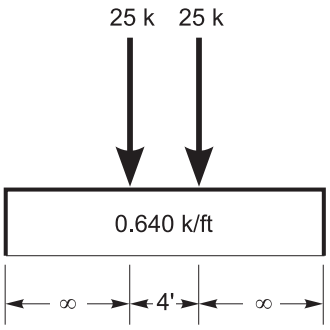


HS20-44 Truck Load



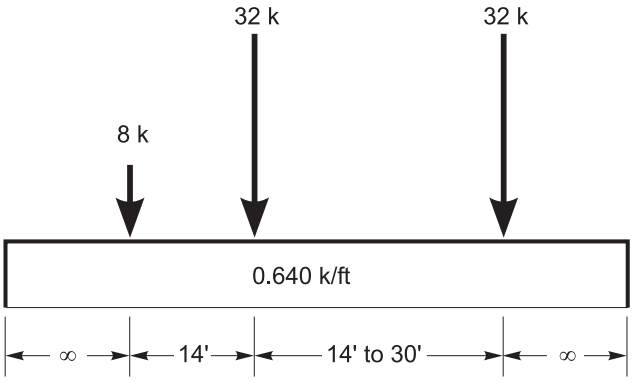
H20-44L and HS20-44L Lane Loads

Figure 99
AASHTO Standard H and HS Vehicles

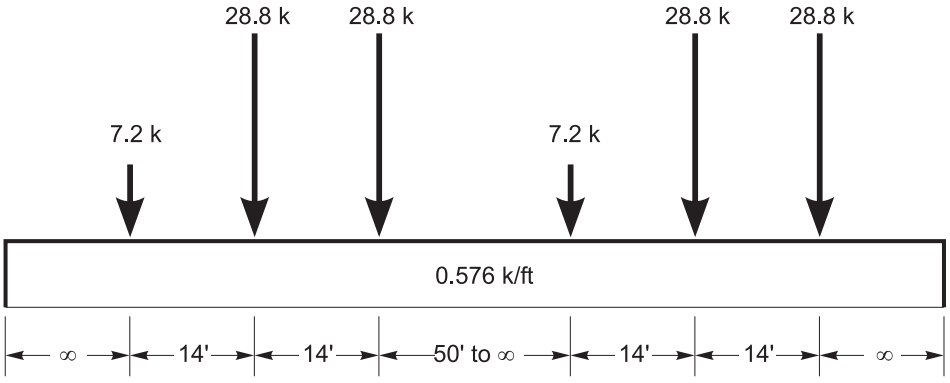


HL-93K Tandem and Lane Load

Note: All point loads will be increased by the dynamic load allowance, **im**, expressed as a percentage



HL-93M Truck and Lane Load



HL-93S Truck and Lane Load for Negative Moment and Reactions at Interior Piers

Figure 100
AASHTO Standard HL Vehicles

Vehicles specified with **type** = HL-93M represent the AASHTO standard HL-93 Load consisting of the code-specified design tandem and the design lane load.

Vehicles specified with **type** = HL-93S represent the AASHTO standard HL-93 Load consisting of two code-specified design trucks and the design lane load, all scaled by 90%. The axle spacing for each truck is fixed at 14 ft. The spacing between the rear axle of the lead truck and the lead axle of the rear truck varies from 50 ft to the length of the Lane. This vehicle is only used for negative moment over supports and reactions at interior piers, so **supmom** = Y, **intsup** = Y, and **other** = N.

A dynamic load allowance may be specified for each Vehicle using the parameter **im**. This is the additive percentage by which the concentrated truck or tandem axle loads will be increased. The uniform lane load is not affected. Thus if **im** = 33, all concentrated axle loads for the vehicle will be multiplied by the factor 1.33.

These Vehicles are illustrated in Figure 100 (page 379) for **im** = 0.

P5, P7, P9, P11, and P13

Vehicles specified with **type** = P5, **type** = P7, **type** = P9, **type** = P11, and **type** = P13 represent the Caltrans standard Permit Loads. These Vehicles are illustrated in Figure 100 (page 381).

The effect of a shorter Caltrans Permit Load is included in any of the longer Permit Loads. If you are designing for all of these permit loads, only the P13 Vehicle is needed.

Cooper E 80

Vehicles specified with **type** = COOPERE80 represent the AREA standard Cooper E 80 train load. This Vehicle is illustrated in Figure 101 (page 382).

UICn

Vehicles specified with **type** = UIC**n** represent the European UIC (or British RU) train load. The **n** in the **type** is an integer scale factor that specifies magnitude of the uniform load in kN/m. Thus UIC80 is the full UIC load with an 80 kN/m uniform load, and UIC60 is the UIC load with an 60 kN/m uniform load. The concentrated loads are not affected by **n**.

This Vehicle is illustrated in Figure 101 (page 382).

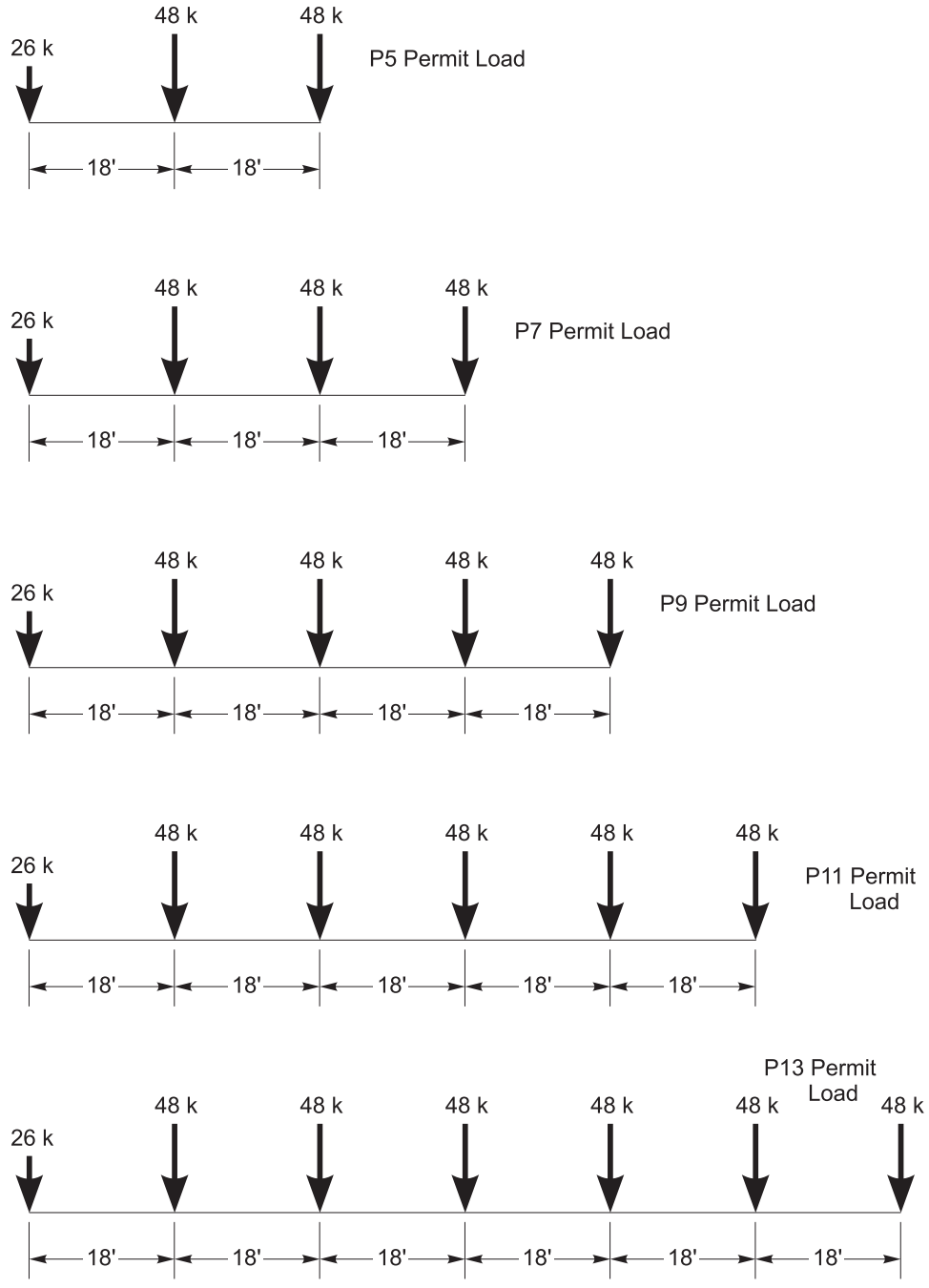
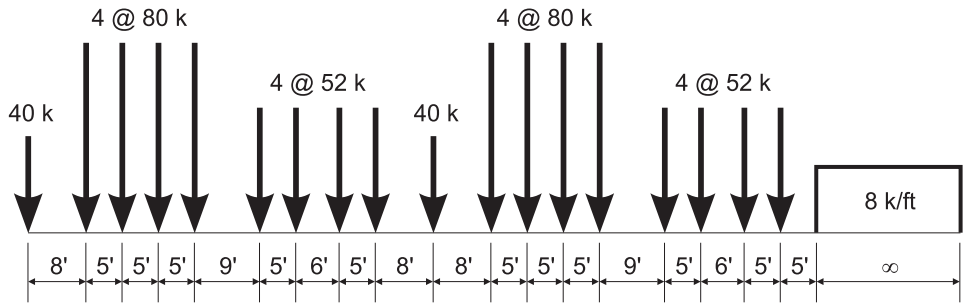
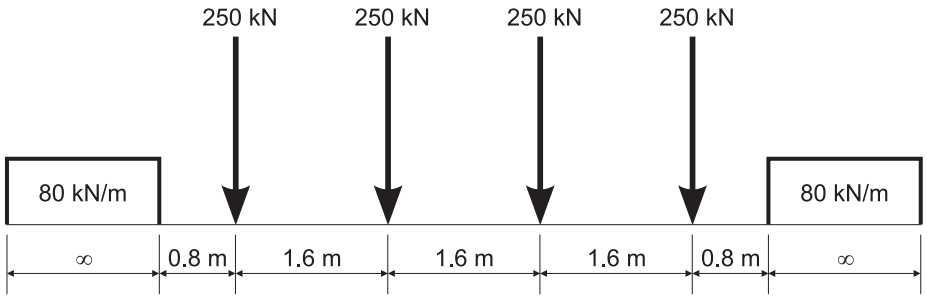


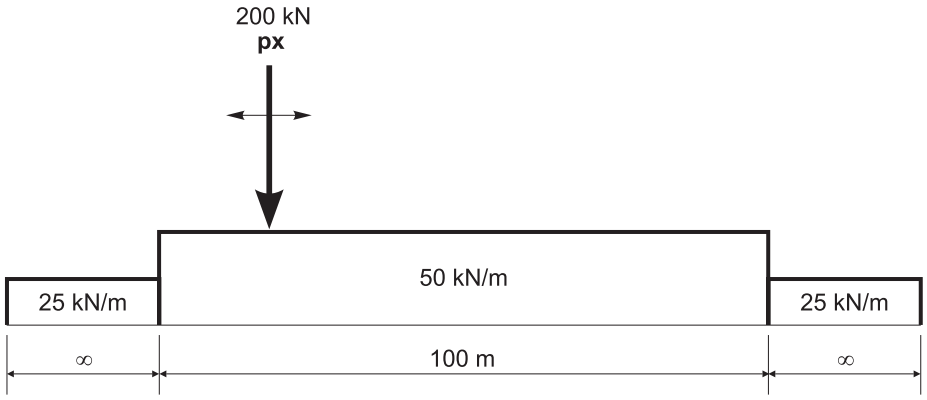
Figure 101
Caltrans Standard Permit Vehicles



Cooper E 80 Train Load



UIC80 Train Load



RL Train Load

Figure 102
Standard Train Vehicles

RL

Vehicles specified with **type** = RL represent the British RL train load. This Vehicle is illustrated in Figure 102 (page 382).

Vehicle Classes

The designer is often interested in the maximum and minimum response of the bridge to the most extreme of several types of Vehicles rather than the effect of the individual Vehicles. For this purpose, Vehicle Classes are defined that may include any number of individual Vehicles. The maximum and minimum force and displacement response quantities for a Vehicle Class will be the maximum and minimum values obtained for any individual Vehicle in that Class. Only one Vehicle ever acts at a time.

All Vehicle loads are applied to the traffic Lanes through the use of Vehicle Classes. If it is desired to apply an individual Vehicle load, you must define a Vehicle Class that contains only that single Vehicle.

For example, the you may need to consider the most severe of a Truck Load and the corresponding Lane Load, say the HS20-44 and HS20-44L loads. A Vehicle Class can be defined to contain these two Vehicles. Additional Vehicles, such as the Alternate Military Load type AML, could be included in the Class as appropriate. Different members of the Class may cause the most severe response at different locations in the structure.

For HL-93 loading, you would first define three Vehicles, one each of the standard types HL-93K, HL-93M, and HL-93S. You then could define a single Vehicle Class containing all three Vehicles.

Moving Load Cases

The final step in the definition of the vehicle live loading is the application of the Vehicle Classes to the traffic Lanes. This is done by creating independent **Moving Load cases**. Each Moving Load case consists of a set of assignments that specify how the Classes are assigned to the Lanes.

Each **assignment** in a Moving Load case requires the following data:

- A Vehicle Class, **class**
- A scale factor, **sf**, multiplying the effect of **class** (the default is unity)

- A list, **lanes**, of one or more Lanes in which **class** may act (the default is all Lanes)
- The minimum number, **lmin**, of Lanes **lanes** in which **class** must act (the default is zero)
- The maximum number, **lmax**, of Lanes **lanes** in which **class** may act (the default is all of **lanes**)

The program looks at all of the assignments in a Moving Load case, and tries every possible permutation of loading the traffic Lanes with Vehicle Classes that is permitted by the assignments. No Lane is ever loaded by more than one Class at a time.

You may specify multiple-lane scale factors, **rf1**, **rf2**, **rf3**, ..., for each Moving Load case that multiply the effect of each permutation depending upon the number of loaded Lanes. For example, the effect of a permutation that loads two Lanes is multiplied by **rf2**.

The maximum and minimum response quantities for a Moving Load case will be the maximum and minimum values obtained for any permutation permitted by the assignments. Usually the permutation producing the most severe response will be different for different response quantities.

The results for a Moving Load case include the response due to loads ranging from zero up to the specified Vehicle live loads. As a result, the envelope always includes zero response: the maximum will never be negative and the minimum will never be positive. Thus there is no need to consider smaller vehicles that differ only by a scale factor. For example, the range of response for an HS20-44 vehicle includes the response for an HS15-44 vehicle.

The concepts of assignment can be clarified with the help of the following examples.

Example 1 — AASHTO HS Loading

Consider a four-lane bridge designed to carry AASHTO HS20-44 Truck and Lane Loads, and the Alternate Military Load (AASHTO, 1996 a). Suppose that it is required that the number of Lanes loaded be that which produces the most severe response in every member. Only one of the three Vehicle loads is allowed per lane. Load intensities may be reduced by 10% and 25% when three or four Lanes are loaded, respectively.

Generally, loading all of the Lanes will produce the most severe moments and shears along the span and axial forces in the piers. However, the most severe torsion

of the bridge deck and transverse bending of the piers will usually be produced by loading only those Lanes possessing eccentricities of the same sign.

Assume that the bridge structure and traffic Lanes have been defined. Three Vehicles are defined:

- **name** = HSK, **type** = HS20-44
- **name** = HSL, **type** = HS20-44L
- **name** = AML, **type** = AML

where **name** is an arbitrary label assigned to each Vehicle. The three Vehicles are assigned to a single Vehicle Class, with an arbitrary label of **name** = HS, so that the most severe of these three Vehicle loads will be used for every situation.

A single Moving Load case is then defined that seeks the maximum and minimum responses throughout the structure for the most severe of loading all four Lanes, any three Lanes, any two Lanes or any single Lane. This can be accomplished using a single assignment. The parameters for the assignment are:

- **class** = HS
- **sf** = 1
- **lanes** = 1, 2, 3, 4
- **lmin** = 1
- **lmax** = 4

The scale factors for the loading of multiple Lanes in the set of assignments are **rf1** = 1, **rf2** = 1, **rf3** = 0.9, and **rf4** = 0.75.

There are fifteen possible permutations assigning the single Vehicle Class HS to any one, two, three, or four Lanes. These are presented in the following table:

Permutation	Lane 1	Lane 2	Lane 3	Lane 4	Scale Factor
1	HS				1.00
2		HS			1.00
3			HS		1.00
4				HS	1.00
5	HS	HS			1.00

Permutation	Lane 1	Lane 2	Lane 3	Lane 4	Scale Factor
6		HS	HS		1.00
7			HS	HS	1.00
8	HS			HS	1.00
9	HS		HS		1.00
10		HS		HS	1.00
11	HS	HS	HS		0.90
12	HS	HS		HS	0.90
13	HS		HS	HS	0.90
14		HS	HS	HS	0.90
15	HS	HS	HS	HS	0.75

An “HS” in a Lane column of this table indicates application of Class HS; a blank indicates that the Lane is unloaded. The scale factor for each permutation is determined by the number of Lanes loaded.

Example 2 — AASHTO HL Loading

Consider a four-lane bridge designed to carry AASHTO HL-93 loading (AASHTO, 1996 b). The approach is the same as used for AASHTO HS loading in the previous example. Only the multiple-lane scale factors and the Vehicles differ.

Three Vehicles are defined:

- **name** = HLK, **type** = HL-93K
- **name** = HLM, **type** = HL-93M
- **name** = HLS, **type** = HL-93S

where **name** is an arbitrary label assigned to each Vehicle.

The three Vehicles are assigned to a single Vehicle Class, with an arbitrary label of **name** = HL, so that the most severe of these three Vehicle loads will be used for every situation. By definition of the standard Vehicle type HL-93S, Vehicle HLS will only be used when computing negative moments over supports or the reaction

at interior piers. The other two Vehicles will be considered for all response quantities.

A single Moving Load case is then defined that is identical to that of the previous example, except that **class** = HL, and the scale factors for multiple Lanes are **rf1** = 1.2, **rf2** = 1, **rf3** = 0.85, and **rf4** = 0.65.

There are again fifteen possible permutations assigning the single Vehicle Class HL to any one, two, three, or four Lanes. These are similar to the permutations of the previous example, with the scale factors changed as appropriate.

Example 3 — Caltrans Permit Loading

Consider the four-lane bridge of the previous examples now subject to Caltrans Combination Group I_{PW} (Caltrans, 1995). Here the permit load(s) are to be used alone in a single traffic Lane, or in combination with one HS or Alternate Military Load in a separate traffic lane, depending upon which is more severe.

Four Vehicles are defined:

- **name** = HSK, **type** = HS20-44
- **name** = HSL, **type** = HS20-44L
- **name** = AML, **type** = AML
- **name** = P13, **type** = P13

where **name** is an arbitrary label assigned to each Vehicle.

The first three Vehicles are assigned to a Vehicle Class that is given the label **name** = HS, as in Example 1. The last Vehicle is assigned as the only member of a Vehicle Class that is given the label **name** = P13. Note that the effects of SAP2000 Vehicle types P5, P7, P9, and P11 are captured by Vehicle type P13.

Combination Group I_{PW} is then represented as a single Moving Load case consisting of the assignment of Class P13 to any single Lane with or without Class HS being assigned to any other single Lane. This can be accomplished using two assignments. A scale factor of unity is used regardless of the number of loaded Lanes.

The first assignment assigns Class P13 to any single Lane:

- **class** = P13
- **sf** = 1
- **lanes** = 1, 2, 3, 4

- **lmin** = 1
- **lmax** = 1

The second assignment assigns Class HS to any single Lane, or to no Lane at all:

- **class** = HS
- **sf** = 1
- **lanes** = 1, 2, 3, 4
- **lmin** = 0
- **lmax** = 1

There are sixteen possible permutations for these two assignments such that no Lane is loaded by more than one Class at a time. These are presented in the following table:

Permutation	Lane 1	Lane 2	Lane 3	Lane 4	Scale Factor
1	P				1.00
2	P	HS			1.00
3	P		HS		1.00
4	P			HS	1.00
5	HS	P			1.00
6		P			1.00
7		P	HS		1.00
8		P		HS	1.00
9	HS		P		1.00
10		HS	P		1.00
11			P		1.00
12			P	HS	1.00
13	HS			P	1.00
14		HS		P	1.00

Permutation	Lane 1	Lane 2	Lane 3	Lane 4	Scale Factor
15			HS	P	1.00
16				P	1.00

Example 4 — Restricted Caltrans Permit Loading

Consider the four-Lane bridge and the Caltrans permit loading of Example 3, but subject to the following restrictions:

- The permit Vehicle is only allowed in Lane 1 or Lane 4
- The Lane adjacent to the Lane occupied by the permit Vehicle must be empty

Two Moving Load cases are required, each containing two assignments. A scale factor of unity is used regardless of the number of loaded Lanes.

The first Moving Load case considers the case where the permit Vehicle occupies Lane 1. The first assignment assigns Class P13 to Lane 1

- **class** = P13
- **sf** = 1
- **lanes** = 1
- **lmin** = 1
- **lmax** = 1

The second assignment assigns Class HS to either Lane 3 or 4, or to no Lane at all:

- **class** = HS
- **sf** = 1
- **lanes** = 3, 4
- **lmin** = 0
- **lmax** = 1

These assignments permits the following three permutations:

Permutation	Lane 1	Lane 2	Lane 3	Lane 4	Scale Factor
1	P				1.00
2	P		HS		1.00
3	P			HS	1.00

Similarly, the second Moving Load case considers the case where the permit Vehicle occupies Lane 4. The first assignment assigns Class P13 to Lane 4

- **class** = P13
- **sf** = 1
- **lanes** = 4
- **lmin** = 1
- **lmax** = 1

The second assignment assigns Class HS to either Lane 1 or 2, or to no Lane at all:

- **class** = HS
- **sf** = 1
- **lanes** = 1, 2
- **lmin** = 0
- **lmax** = 1

These assignments permits the following three permutations:

Permutation	Lane 1	Lane 2	Lane 3	Lane 4	Scale Factor
1				P	1.00
2	HS			P	1.00
3		HS		P	1.00

An envelope-type Combo that includes only these two Moving Load cases would produce the most severe response for the six permutations above.

See Topic “Combos” (page 333) in Chapter “Static and Dynamic Analysis” for more information.

Influence Line Tolerance

SAP2000 simplifies the influence lines used for response calculation in order to increase efficiency. A relative tolerance is used to reduce the number of load points by removing those that are approximately duplicated or that can be approximately linearly-interpolated. The default value of this tolerance permits response errors on the order of 0.01%. Setting the tolerance to zero will provide exact results to within the resolution of the analysis.

Exact and Quick Response Calculation

For the purpose of moving a Vehicle along a lane, each axle is placed on every load point in turn. When another axle falls between two load points, the effect of that axle is determined by linear interpolation of the influence values. The effect of uniform loads is computed by integrating the linearly-interpolated segments of the influence line. This method is exact to within the resolution of the analysis, but is computationally intensive if there are many load points.

A "Quick" method is available which may be much faster than the usual "Exact" method, but it may also be less accurate. The Quick method approximates the influence line by using a limited number of load points in each "span." For purposes of this discussion, a span is considered to be a region where the influence line is all positive or all negative.

The degree of approximation to be used is specified by the parameter **quick**, which may be any non-negative integer. The default value is **quick** = 0, which indicates to use the full influence line, i.e., the Exact method.

Positive values indicate increasing degrees of refinement for the Quick method. For **quick** = 1, the influence line is simplified by using only the maximum or minimum value in each span, plus the zero points at each end of the span. For **quick** = 2, an additional load point is used on either side of the maximum/minimum. Higher degrees of refinement use additional load points. The number of points used in a span can be as many as $2^{\mathbf{quick}}+1$, but not more than the number of load points available in the span for the Exact method.

It is strongly recommended that **quick** = 0 be used for all final analyses. For preliminary analyses, **quick** = 1, 2, or 3 is usually adequate, with **quick** = 2 often providing a good balance between speed and accuracy. The effect of parameter **quick** upon speed and accuracy is problem-dependent, and you should experiment to determine the best value to use for each different model.

Moving Load Response Control

By default, *no Moving Load response is calculated for any joint or element*, since this calculation is computationally intensive. You must explicitly request the Moving Load response that you want calculated.

For each joint, you may explicitly request the following types of results to be calculated:

- Displacements
- Reactions, and/or
- Spring forces

For each Frame element, you may explicitly request that the internal forces be calculated.

Each of the selected joint or Frame-element response quantities is calculated for all Moving Load cases.

If the displacements, reactions, spring forces, or internal forces are not calculated for a given joint or Frame element, no Moving Load response can be printed or plotted for that joint or element. Likewise, no response can be printed or plotted for any Combo that contains a Moving Load case.

Correspondence

For each maximum or minimum Frame-element response quantity computed, the corresponding values for the other five internal force and moment components may be determined. For example, the shear, moment, and torque that occur at the same time as the maximum axial force in a Frame element may be computed.

These corresponding response quantities are only used for steel and concrete design in the SAP2000 graphical user interface. They cannot be printed or displayed. When Moving Load cases are printed or displayed, the extreme values of each response quantity are given without correspondence.

By default, no corresponding quantities are computed for the Frame elements, since this significantly increases the computation time for moving-load response. You may specify that correspondence is to be calculated, in which case it will be done for all Frame elements for which moving-load response is requested, and for all Moving Load cases.

Computational Considerations

The computation of influence lines requires a moderate amount of computer time and a large amount of disk storage compared with the execution of other typical SAP2000 analyses. The amount of computer time is approximately proportional to N^2L, where N is the number of structure degrees-of-freedom, and L is the number of load points. The amount of disk storage required is approximately proportional to NL.

Increasing the resolution of the analysis by increasing the number of Frame elements (holding **nseg** constant) causes approximately proportional increases in N and L, and hence increases computation time by about L^3 and storage space by about L^2. Holding the number of elements constant and increasing **nseg** instead increases computation time and storage space each by approximately L. Clearly the latter method is more efficient.

If all traffic Lanes have zero eccentricities everywhere, computation time and storage space for the influence lines are cut in half.

The computation of Moving Load response may require a large amount of computer time compared with the execution of other typical SAP2000 analyses. The amount of disk storage needed (beyond the influence lines) is small.

The computation time for Moving Load response is proportional to the number of response points. To obtain the same effective increase in resolution, increasing the values of **nseg** produces fewer additional response points than does increasing the number of elements, since the latter approach introduces duplicate response points at the joints. Hence changing **nseg** is more efficient. Considerable savings in computation time can also be realized by restricting Moving Load calculations to only those joints and Frame elements of significant interest.

The computation time for Moving Load response is also directly proportional to the number of Lanes. It is not, however, sensitive to whether or not Lane eccentricities are present.

For each Vehicle load, the computation time is approximately proportional to the square of the number of axles. It is also proportional to L', the effective number of load points. Larger values of the truck influence tolerance tend to produce smaller values of L' compared to L. The value of L' will be different for each response quantity; it tends to be smaller for structures with simple spans than with continuous spans. The value of L' is not sensitive to whether increases in resolution are obtained by increasing the number of elements or the number of output segments.

Chapter XIX

The Output Files

Three output text files are produced during a SAP2000 analysis: the input echo file, the analysis log file, and the results output file. Additional output may be obtained using the SAP2000 graphical user interface.

Basic Topics for All Users

- Overview
- The Input Echo (.EKO) File
- The Analysis Log (.LOG) File
- The Results Output (.OUT) File

Advanced Topics

- Element Joint Force Output
- Joint and Element Output Control
- Pagination Control

Overview

An analysis performed by SAP2000 produces three output text files. These are:

- The input echo file, having file extension .EKO; this is a tabulation of the input data text file as read and checked by the analysis
- The analysis log file, having file extension .LOG; this contains analysis statistics, warnings, and error messages
- The results output file, having file extension .OUT; this contains displacements, forces, stresses and other results of the analysis

When importing a SAP90 or SAP2000 input data text file, only the input echo file is produced.

For an input data file named EXAMPLE.SDB or EXAMPLE.S2K, these files would be called EXAMPLE.EKO, EXAMPLE.LOG, and EXAMPLE.OUT, respectively.

These are ASCII text files with an 80-character wide output format. Smaller files may be viewed with a text editor such as Windows Notepad, or any other editor or word-processing program in text mode. For larger problems, you may need to use a large-capacity editor or word-processing program. You may use any standard Windows printer to print these files.

The Input Echo (.EKO) File

The **input echo file**, more simply called the **echo file**, is created during the data-checking phase of the program. It contains a tabulation of the input data file as read and interpreted by SAP2000, as well as the details of any warnings or errors detected during this phase. The contents of this file are self-explanatory.

It is recommended that you carefully review the contents of the echo file for errors, warnings, and the correctness of the model.

If errors are present, the analysis phase of the program will not be executed. The model must be corrected and the program executed again.

If only warnings are present, the analysis phase may proceed. You should check these warnings carefully before reviewing the results of the analysis.

All data that is used by the analysis phase is presented in the input echo file. As such, this file forms a record of the analysis being performed.

The Analysis Log (.LOG) File

The **analysis log file**, more simply called the **log file**, is created during the analysis phase of the program. It contains analysis statistics and results summaries, as well as the details of any warnings or errors detected during this phase.

It is *strongly recommended* that you carefully review the contents of the log file for errors, warnings, and the results summaries, including the global force balance relative errors.

If *errors* are present, the analysis phase of the program will have terminated before completion. If only *warnings* are present, the analysis phase should have completed. In either case, you should check these errors and warnings carefully before examining the results of the analysis.

The analysis phase is divided into many sub-phases. The time and date at the beginning and end of each of these sub-phases is reported. The date is given in year/month/day format, and the time is given in hour:minute:second format.

The following additional information is provided in the analysis log file:

- The amount of memory allocated for data storage, in bytes
- The number of elements formed
- The number of Welds and Constraints formed
- The number of equations formed, the size of the stiffness matrix, and the number of loads solved
- The number of natural frequencies of the model below frequency **cyc** if harmonic steady-state analysis is performed; see Topic “Harmonic Steady-State Analysis” (page 294) in Chapter “Static and Dynamic Analysis” for more information
- The number of natural frequencies of the model below frequency **shift** if eigenvector analysis is performed with a frequency shift; see Topic “Eigenvector Analysis” (page 296) in Chapter “Static and Dynamic Analysis” for more information
- For modal analysis, the control parameters and the number of Modes found
- For eigenvector analysis, the number of iterations performed
- For Ritz-vector analysis, the number of generation cycles performed and the reasons for discarding the starting load vectors (if any)
- For response-spectrum analysis, the number of analyses performed

```
------------------------------------------------------ 1996/02/01 16:36:51.06

MEMORY AVAILABLE FOR DATA (BYTES)         =     2000000

------------------------------------------------------ 1996/02/01 16:36:51.17

J O I N T   E L E M E N T   F O R M A T I O N   P H A S E

NUMBER OF JOINT  ELEMENTS FORMED          =          14
NUMBER OF SPRING ELEMENTS FORMED          =           0

------------------------------------------------------ 1996/02/01 16:36:51.66

F R A M E   E L E M E N T   F O R M A T I O N   P H A S E

NUMBER OF FRAME  ELEMENTS FORMED          =          35

------------------------------------------------------ 1996/02/01 16:36:53.75

E Q U A T I O N   F O R M A T I O N   A N D   S O L U T I O N   P H A S E

TOTAL NUMBER OF EQUILIBRIUM EQUATIONS     =          49
APPROXIMATE "EFFECTIVE" BAND WIDTH        =           8

NUMBER OF EQUATION STORAGE BLOCKS         =           1
MAXIMUM BLOCK SIZE (NUMBER OF TERMS)      =         418
SIZE OF STIFFNESS FILE (BYTES)            =        3556

NUMBER OF EQUATIONS TO REDUCE             =          49
NUMBER OF LOAD CONDITIONS TO SOLVE        =           2

------------------------------------------------------ 1996/02/01 16:36:58.09

J O I N T   O U T P U T   P H A S E

G L O B A L   F O R C E   B A L A N C E   R E L A T I V E   E R R O R S

PERCENT FORCE AND MOMENT ERROR AT THE ORIGIN, IN GLOBAL COORDINATES

   LOAD          FX          FY          FZ          MX          MY          MZ
      1     .000000    7.38E-17    4.72E-14    6.62E-15     .000000     .000000
      2     .000000    9.97E-16    7.98E-15    2.50E-14     .000000     .000000

   COMB          FX          FY          FZ          MX          MY          MZ
      1     .000000    7.38E-17    4.84E-14    1.67E-14     .000000     .000000

------------------------------------------------------ 1996/02/01 16:37:02.54

F R A M E   E L E M E N T   O U T P U T   P H A S E

NUMBER OF FRAME  ELEMENTS SAVED           =          35
NUMBER OF FRAME  ELEMENTS PRINTED         =          35

------------------------------------------------------ 1996/02/01 16:37:04.62
```

...Output Edited for Brevity...

Figure 103
Typical Contents of the Analysis Log (.LOG) File

- For each time-history analysis, the control parameters and the number of time steps completed
- For each nonlinear time-history analysis, the number of substeps and iterations completed and other statistics
- For moving-load analysis, the control parameters and solution statistics
- The global force balance relative errors; see Topic “Global Force Balance Output” (page 107) in Chapter “Joints and Degrees of Freedom” for more information
- The number of elements processed for output

Typical output is shown in Figure 103 (page 398).

The Results Output (.OUT) File

The **results output file**, more simply called the **output file**, is created during the analysis phase of the program. The results given for the joints and elements are subject to your control. See Topic “Joint and Element Output Control” (page 405) in this chapter for more information. The remaining results are always given by the program when appropriate.

It is recommended that you carefully review the input echo (.EKO) file and the analysis log (.LOG) file for errors, warnings, and correctness of the model before looking at the results output file.

The output file does not contain any warnings or error messages.

The results in the output file are divided into various sections, each with a heading that describes the type of results that follow. These sections are described below in the same order that they appear in the output file:

- The following output section is present if there are any Body, Diaphragm, Plate, Rod, and Beam Constraints that constrain more than two joints:

  ```
  CONSTRAINT COORDINATES AND MASSES
  ```

  See Topic “Constraint Output” (page 133) in Chapter “Constraints and Welds.”

- The following output section is always present:

  ```
  DISPLACEMENT DEGREES OF FREEDOM
  ```

See Topic “Degree of Freedom Output” (page 97) in Chapter “Joints and Degrees of Freedom.”

- The following output sections are present if the structure has any mass:

  ```
  ASSEMBLED JOINT MASSES
  TOTAL UNRESTRAINED MASS AND LOCATION
  ```

  See Topic “Joint Mass Output” (page 98) in Chapter “Joints and Degrees of Freedom.”

- The following output sections are present if an eigenvector or Ritz-vector modal analysis has been performed:

  ```
  MODAL PERIODS AND FREQUENCIES
  MODAL PARTICIPATION FACTORS
  MODAL PARTICIPATING MASS RATIOS
  MODAL LOAD PARTICIPATION RATIOS
  ```

  See Topic “Modal Analysis Output” (page 303) in Chapter “Static and Dynamic Analysis.”

- The following output sections are present if response-spectrum analyses have been performed:

  ```
  RESPONSE SPECTRUM ACCELERATIONS
  RESPONSE SPECTRUM MODAL AMPLITUDES
  RESPONSE SPECTRUM MODAL CORRELATIONS
  RESPONSE SPECTRUM BASE REACTIONS
  ```

  See Topic “Response-Spectrum Analysis Output” (page 317) in Chapter “Static and Dynamic Analysis.”

- The following output section is present whenever joint displacements are requested:

  ```
  JOINT DISPLACEMENTS
  ```

  See Topic “Joint Displacement Output” (page 100) in Chapter “Joints and Degrees of Freedom.”

- The following output sections are present whenever joint applied loads are requested and the corresponding loads exist:

  ```
  APPLIED LOADS
  INERTIAL LOADS
  ```

  See Topic “Force Output” (page 102) in Chapter “Joints and Degrees of Freedom.”

- The following output sections are present whenever joint reactions are requested and the corresponding types of forces exist:

  ```
  SPRING FORCES
  NLLINK FORCES
  RESTRAINT FORCES (REACTIONS)
  CONSTRAINT FORCES
  ```

  See Topic “Force Output” (page 102) in Chapter “Joints and Degrees of Freedom.”

- The following output section is always present:

  ```
  GLOBAL FORCE BALANCE
  ```

  See Topic “Global Force Balance Output” (page 107) in Chapter “Joints and Degrees of Freedom.”

- The following output sections are present whenever joint forces are requested for the corresponding type of element:

  ```
  FRAME  ELEMENT JOINT FORCES
  SHELL  ELEMENT JOINT FORCES
  PLANE  ELEMENT JOINT FORCES
  ASOLID ELEMENT JOINT FORCES
  SOLID  ELEMENT JOINT FORCES
  NLLINK ELEMENT JOINT FORCES
  ```

  See Topic “Joint Force Output” (page 402) in this chapter.

  See Topic “Joint Force Output” (page 179) in Chapter “The Frame Element.”

  See Topic “Joint Force Output” (page 203) in Chapter “The Shell Element.”

  See Topic “Joint Force Output” (page 216) in Chapter “The Plane Element.”

See Topic “Joint Force Output” (page 231) in Chapter “The Asolid Element.”

See Topic “Joint Force Output” (page 241) in Chapter “The Solid Element.”

See Topic “Joint Force Output” (page 277) in Chapter “The Nllink Element.”

- The following output sections are present whenever internal forces or stresses are requested for the corresponding type of element:

  ```
  FRAME ELEMENT INTERNAL FORCES
  SHELL ELEMENT INTERNAL FORCES & STRESSES
  PLANE ELEMENT STRESSES
  ASOLID ELEMENT STRESSES
  SOLID ELEMENT STRESSES
  NLLINK ELEMENT INTERNAL FORCES & DEFORMS
  ```

  See Topic “Internal Force Output” (page 176) in Chapter “The Frame Element.”

  See Topic “Internal Force and Stress Output” (page 198) in Chapter “The Shell Element.”

  See Topic “Stress Output” (page 215) in Chapter “The Plane Element.”

  See Topic “Stress Output” (page 230) in Chapter “The Asolid Element.”

  See Topic “Stress Output” (page 240) in Chapter “The Solid Element.”

  See Topic “Internal Force and Deformation Output” (page 275) in Chapter “The Nllink Element.”

Typical output is shown in the respective topics referenced above.

Element Joint Force Output

The **element joint forces** are concentrated forces and moments acting at the joints of the element that represent the effect of the rest of the structure upon the element and that cause the deformation of the element. The moments will always be zero for the solid-type elements: Plane, Asolid, and Solid.

A positive value of force or moment tends to cause a positive value of translation or rotation of the element along the corresponding joint degree of freedom. See Figure 104 (page 403).

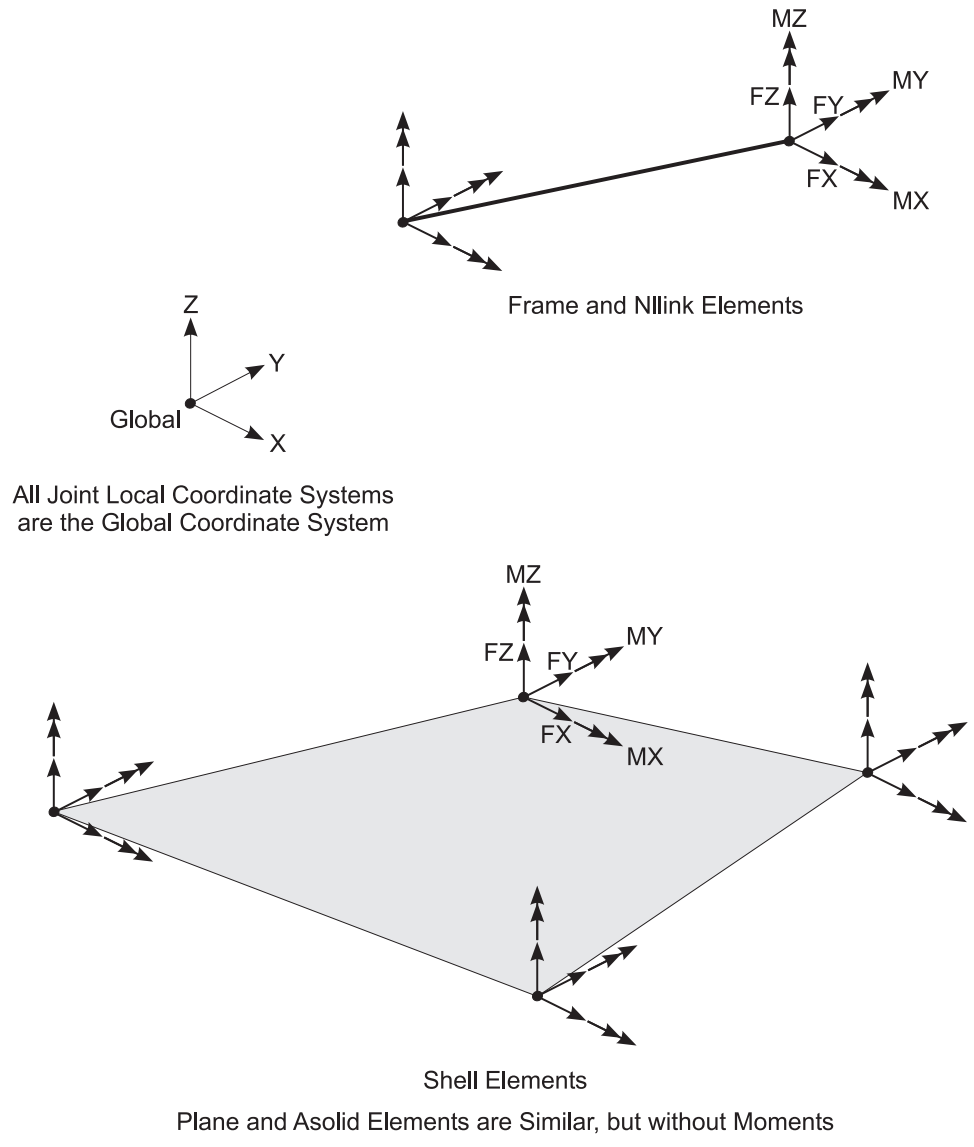


Figure 104
Element Joint Forces when All Joints Have the Global Coordinate System for Their Local Coordinate Systems

Element joint forces must not be confused with internal forces and moments which, like stresses, act *within* the volume of the element.

For a given element, the vector of element joint forces, $\boldsymbol{f}$, is computed as:

$$\boldsymbol{f} = \boldsymbol{K}\ \boldsymbol{u} - \boldsymbol{r}$$

where $\boldsymbol{K}$ is the element stiffness matrix, $\boldsymbol{u}$ is the vector of element joint displacements, and $\boldsymbol{r}$ is the vector of element applied loads as apportioned to the joints. The applied loads may include Force, Self-Weight, Gravity, Concentrated Span, Distributed Span, Uniform, Surface Pressure, Pore Pressure, Temperature, Prestress, and Rotate Loads, as appropriate, that act directly on the given element.

The element joint forces are always referred to the local axes of the individual joints. They are identified in the output as F1, F2, F3, M1, M2, and M3. However, if all joints use the global coordinate system as the local system (the usual situation), then the forces and moments are identified as FX, FY, FZ, MX, MY, and MZ.

Element joint forces and moments for the different types of elements are printed in the output file under the headings:

```
FRAME  ELEMENT JOINT FORCES

SHELL  ELEMENT JOINT FORCES

PLANE  ELEMENT JOINT FORCES

ASOLID ELEMENT JOINT FORCES

SOLID  ELEMENT JOINT FORCES

NLLINK ELEMENT JOINT FORCES
```

All joint forces for a single element are printed together. Elements of a given type are presented in alpha-numeric order. For each Analysis Case the joint forces are printed at all element joints.

Typical element joint force output is shown for the Frame element in Figure 105 (page 406), the Shell element in Figure 106 (page 407), the Plane element in Figure 107 (page 408), the Asolid element in Figure 108 (page 409), the Solid element in Figure 109 (page 410), and the Nllink element in Figure 110 (page 411).

By default, no element joint forces are printed. You may specify which Analysis Cases are to be printed for each element. Available Analysis Cases include Loads,

Modes, Specs, and Combos, but not Histories, Moving Loads, or Combos that contain Histories or Moving Loads.

The element joint forces are available through the SAP2000 graphical user interface using the "Group Joint Force Sum." This sum is the resultant of the forces and moments acting on one or more joints of one or more elements.

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

For more information:

- See Topic "Joint and Element Output Control" (page 405) in this chapter
- Chapter "The Frame Element" (page 145)
- Chapter "The Shell Element" (page 181)
- Chapter "The Plane Element" (page 205)
- Chapter "The Asolid Element" (page 219)
- Chapter "The Solid Element" (page 233)
- Chapter "The Nllink Element" (page 243)

Joint and Element Output Control

By default, no joint and element results are written to the results output (.OUT) file.

You may select the types of analysis results to be printed for each of the following analysis cases:

- Load Cases
- Modes
- Specs
- Histories
- Moving Loads
- Combos

The printed History results are just the maximum and minimum values of each response quantity attained during the History. For the time-variation of the response, use the SAP2000 graphical user interface.

```
F R A M E    E L E M E N T  J O I N T  F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM       1 ==================

LOAD       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
     33   -7.35E-13    9.73E-11    1000.000  -72000.000   -2026.122    1.35E-12
     35    7.35E-13   -9.73E-11   -1000.000   84000.000    2026.122    7.47E-12

SPEC    SPEC ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
     33    0.972230  126.377896   68.323489    3190.341   89.135262    1.878252
     35    0.972230  126.377896   68.323489    3978.697   89.135262    9.788509

COMB    BOTH ------------------ MAX

  JOINT          FX          FY          FZ          MX          MY          MZ
     33    0.972230  126.377896    1068.323  -68809.659   -1936.987    1.878252
     35    0.972230  126.377896 -931.676511   87978.697    2115.257    9.788509

COMB    BOTH ------------------ MIN

  JOINT          FX          FY          FZ          MX          MY          MZ
     33   -0.972230 -126.377896  931.676511  -75190.341   -2115.257   -1.878252
     35   -0.972230 -126.377896   -1068.323   80021.303    1936.987   -9.788509

ELEM       2 ==================

LOAD       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
     34    7.35E-13    9.73E-11    1000.000  -72000.000    2026.122   -1.35E-12
     36   -7.35E-13   -9.73E-11   -1000.000   84000.000   -2026.122   -7.47E-12

SPEC    SPEC ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
     34    0.972230  126.377896   68.323489    3190.341   89.135262    1.878252
     36    0.972230  126.377896   68.323489    3978.697   89.135262    9.788509

COMB    BOTH ------------------ MAX

  JOINT          FX          FY          FZ          MX          MY          MZ
     34    0.972230  126.377896    1068.323  -68809.659    2115.257    1.878252
     36    0.972230  126.377896 -931.676511   87978.697   -1936.987    9.788509

COMB    BOTH ------------------ MIN

  JOINT          FX          FY          FZ          MX          MY          MZ
     34   -0.972230 -126.377896  931.676511  -75190.341    1936.987   -1.878252
     36   -0.972230 -126.377896   -1068.323   80021.303   -2115.257   -9.788509
```

...Output Edited for Brevity...

Figure 105
Frame Element Joint Forces in the Output File

```
S H E L L    E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM       1 ==================

LOAD       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
     10    6.28E-27    9.73E-11    1000.000  -48000.000    9.92E-11    1.31E-12
     31   -1.86E-13   -9.73E-11   -1000.000   60000.000 -641.113050    1.32E-12
     25    1.49E-27    9.73E-11    1000.000  -48000.000   -2.86E-10   -1.31E-12
     32    1.86E-13   -9.73E-11   -1000.000   60000.000  641.113050   -1.32E-12

SPEC    SPEC ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
     10    0.485709  107.747660   61.665399    1718.664    6.42E-13    1.818576
     31    1.320783  107.748554   61.665399    2429.634   33.426016    1.835782
     25    0.484206  107.746818   61.665399    1718.664    1.79E-12    1.815794
     32    1.322272  107.745924   61.665399    2429.634   33.426016    1.831400

COMB    BOTH ------------------ MAX

  JOINT          FX          FY          FZ          MX          MY          MZ
     10    0.485709  107.747660    1061.665  -46281.336    9.99E-11    1.818576
     31    1.320783  107.748554 -938.334601   62429.634 -607.687034    1.835782
     25    0.484206  107.746818    1061.665  -46281.336   -2.85E-10    1.815794
     32    1.322272  107.745924 -938.334601   62429.634  674.539066    1.831400

COMB    BOTH ------------------ MIN

  JOINT          FX          FY          FZ          MX          MY          MZ
     10   -0.485709 -107.747660  938.334601  -49718.664    9.86E-11   -1.818576
     31   -1.320783 -107.748554   -1061.665   57570.366 -674.539066   -1.835782
     25   -0.484206 -107.746818  938.334601  -49718.664   -2.88E-10   -1.815794
     32   -1.322272 -107.745924   -1061.665   57570.366  607.687034   -1.831400

ELEM       2 ==================

LOAD       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
     31    1.86E-13    9.73E-11    1000.000  -60000.000  641.113050   -1.32E-12
     33    7.35E-13   -9.73E-11   -1000.000   72000.000    2026.122   -1.35E-12
     32   -1.86E-13    9.73E-11    1000.000  -60000.000 -641.113050    1.32E-12
     34   -7.35E-13   -9.73E-11   -1000.000   72000.000   -2026.122    1.35E-12

SPEC    SPEC ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
     31    0.584210  120.062949   66.239113    2429.634   33.426016    1.833578
     33    0.235931  120.062949   66.239113    3190.341   89.135262    1.878252
     32    0.584210  120.062949   66.239113    2429.634   33.426016    1.833578
     34    0.235931  120.062949   66.239113    3190.341   89.135262    1.878252
```

...Output Edited for Brevity...

Figure 106
Shell Element Joint Forces in the Output File

```
P L A N E    E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM       1 ==================

LOAD       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      1     .000000   -4.04E-11  167.000000     .000000     .000000     .000000
      2     .000000    6.17E-11   -1.03E-10     .000000     .000000     .000000
      3    0.000108    2000.000 -127.672098     .000000     .000000     .000000
      6     .000000   -1.08E-10  666.000000     .000000     .000000     .000000
      7     .000000    6.90E-11    3.92E-10     .000000     .000000     .000000
      8    3.09E-18    4.15E-11 -744.655804     .000000     .000000     .000000
     11     .000000   -1.78E-11  167.000000     .000000     .000000     .000000
     12     .000000    1.43E-11   -1.35E-10     .000000     .000000     .000000
     13   -0.000108   -2000.000 -127.672098     .000000     .000000     .000000

SPEC    SPEC ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      1     .000000    2.095103    1.960365     .000000     .000000     .000000
      2     .000000    6.736706    4.555885     .000000     .000000     .000000
      3    3.00E-06   45.955743    3.475362     .000000     .000000     .000000
      6     .000000    6.792403    6.514757     .000000     .000000     .000000
      7     .000000   21.085504   14.519078     .000000     .000000     .000000
      8    5.36E-07   24.347802   26.751296     .000000     .000000     .000000
     11     .000000    2.095289    1.960390     .000000     .000000     .000000
     12     .000000    6.737990    4.555878     .000000     .000000     .000000
     13    3.00E-06   45.955810    3.475244     .000000     .000000     .000000

ELEM       2 ==================

LOAD       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
     16     .000000   -4.04E-11  167.000000     .000000     .000000     .000000
     17     .000000    6.17E-11   -1.03E-10     .000000     .000000     .000000
     18   -0.000108    2000.000 -127.672098     .000000     .000000     .000000
     21     .000000   -1.08E-10  666.000000     .000000     .000000     .000000
     22     .000000    6.90E-11    3.92E-10     .000000     .000000     .000000
     23   -3.09E-18    4.15E-11 -744.655804     .000000     .000000     .000000
     26     .000000   -1.78E-11  167.000000     .000000     .000000     .000000
     27     .000000    1.43E-11   -1.35E-10     .000000     .000000     .000000
     28    0.000108   -2000.000 -127.672098     .000000     .000000     .000000
```

...Output Edited for Brevity...

Figure 107
Plane Element Joint Forces in the Output File

```
A S O L I D   E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM       1 ==================

LOAD INTPRES ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      1    2.93E-14   -0.060041     .000000     .000000     .000000     .000000
      2   -4.98E-15    0.084594     .000000     .000000     .000000     .000000
      3   -0.035999   -0.056720     .000000     .000000     .000000     .000000
     12   -5.95E-14   -9.15E-15     .000000     .000000     .000000     .000000
     13   -1.96E-13   -9.98E-14     .000000     .000000     .000000     .000000
     14   -0.143192    1.38E-14     .000000     .000000     .000000     .000000
     23   -1.69E-14    0.060041     .000000     .000000     .000000     .000000
     24   -4.04E-14   -0.084594     .000000     .000000     .000000     .000000
     25   -0.035999    0.056720     .000000     .000000     .000000     .000000

LOAD LOGTEMP ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      1    4.55E-13   13.546107     .000000     .000000     .000000     .000000
      2    1.36E-12   16.192718     .000000     .000000     .000000     .000000
      3   -0.746624   12.833193     .000000     .000000     .000000     .000000
     12   -3.64E-12    3.47E-13     .000000     .000000     .000000     .000000
     13   -3.75E-12   -5.54E-13     .000000     .000000     .000000     .000000
     14   -3.173884    3.53E-12     .000000     .000000     .000000     .000000
     23   -1.36E-12  -13.546107     .000000     .000000     .000000     .000000
     24    1.34E-12  -16.192718     .000000     .000000     .000000     .000000
     25   -0.746624  -12.833193     .000000     .000000     .000000     .000000

ELEM       2 ==================

LOAD INTPRES ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      3    0.035999   -0.099947     .000000     .000000     .000000     .000000
      4   -3.80E-14    0.138916     .000000     .000000     .000000     .000000
      5   -0.028017   -0.093595     .000000     .000000     .000000     .000000
     14    0.143192   -3.29E-14     .000000     .000000     .000000     .000000
     15   -1.44E-13    8.16E-14     .000000     .000000     .000000     .000000
     16   -0.110260    1.34E-13     .000000     .000000     .000000     .000000
     25    0.035999    0.099947     .000000     .000000     .000000     .000000
     26   -9.38E-15   -0.138916     .000000     .000000     .000000     .000000
     27   -0.028017    0.093595     .000000     .000000     .000000     .000000
```

...Output Edited for Brevity...

Figure 108
Asolid Element Joint Forces in the Output File

```
S O L I D    E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM       1 ==================

LOAD       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      3    5.27E-12   -2000.000  127.672098     .000000     .000000     .000000
     18    3.77E-12   -2000.000  127.672098     .000000     .000000     .000000
      4    2.19E-11    3000.000 -249.620408     .000000     .000000     .000000
     19   -2.39E-12    3000.000 -249.620408     .000000     .000000     .000000
      8  -12.997792   -1755.081  372.327902     .000000     .000000     .000000
     23   12.997792   -1755.081  372.327902     .000000     .000000     .000000
      9    2.835433  755.081180 -250.379592     .000000     .000000     .000000
     24   -2.835433  755.081180 -250.379592     .000000     .000000     .000000

SPEC    SPEC ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      3    0.109772   47.939171    5.993277     .000000     .000000     .000000
     18    0.109772   47.939171    5.993277     .000000     .000000     .000000
      4    0.141037   91.193957   11.462987     .000000     .000000     .000000
     19    0.141037   91.193957   11.462987     .000000     .000000     .000000
      8    0.570985   59.100845   16.591391     .000000     .000000     .000000
     23    0.570985   59.100845   16.591391     .000000     .000000     .000000
      9    0.174693   20.588032   11.247946     .000000     .000000     .000000
     24    0.174693   20.588032   11.247946     .000000     .000000     .000000

ELEM       2 ==================

LOAD       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      8   12.997792    1755.081  372.327902     .000000     .000000     .000000
     23  -12.997792    1755.081  372.327902     .000000     .000000     .000000
      9   -2.835433 -755.081180 -250.379592     .000000     .000000     .000000
     24    2.835433 -755.081180 -250.379592     .000000     .000000     .000000
     13   -1.28E-11    2000.000  127.672098     .000000     .000000     .000000
     28    2.42E-11    2000.000  127.672098     .000000     .000000     .000000
     14   -1.25E-11   -3000.000 -249.620408     .000000     .000000     .000000
     29   -2.08E-11   -3000.000 -249.620408     .000000     .000000     .000000
```

...Output Edited for Brevity...

Figure 109
Solid Element Joint Forces in the Output File

```
N L L I N K   E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM      N1 ==================

MODE       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      8    7339.063     .000000     .000000     .000000     .000000     .000000
     15   -7339.063     .000000     .000000     .000000     .000000     .000000

MODE       2 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      8   15370.177     .000000     .000000     .000000     .000000     .000000
     15  -15370.177     .000000     .000000     .000000     .000000     .000000

ELEM      N2 ==================

MODE       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      5    5608.591     .000000     .000000     .000000     .000000     .000000
     14   -5608.591     .000000     .000000     .000000     .000000     .000000

MODE       2 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      5  -11274.612     .000000     .000000     .000000     .000000     .000000
     14   11274.612     .000000     .000000     .000000     .000000     .000000

ELEM      N3 ==================

MODE       1 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      2    2499.958     .000000     .000000     .000000     .000000     .000000
     13   -2499.958     .000000     .000000     .000000     .000000     .000000

MODE       2 ------------------

  JOINT          FX          FY          FZ          MX          MY          MZ
      2  -17122.498     .000000     .000000     .000000     .000000     .000000
     13   17122.498     .000000     .000000     .000000     .000000     .000000
```

...Output Edited for Brevity...

Figure 110
Nllink Element Joint Forces in the Output File

The results for Moving Loads and Combos that contain Moving Loads may not be available for all joints and elements. When Moving Load results are printed, the extreme values of each individual response quantity are always given without correspondence, even if corresponding values were calculated.

The selection of analysis results for the output file has no affect on the availability of results in the SAP2000 graphical user interface.

See Topic “Analysis Cases” (page 292) in Chapter “Static and Dynamic Analysis” for more information.

Joints

For each analysis case, you may select the types of joint response to be output using the parameter **jtypes**, which can take on one or more of the following values:

- DISP: Displacements
- APPL: Applied and inertial loads
- REAC: Restraint, constraint, spring, and Nllink Forces

The selected results are printed for all applicable joints.

For more information:

- See Topic “Displacement Output” (page 100) in Chapter “Joints and Degrees of Freedom.”
- See Topic “Force Output” (page 102) in Chapter “Joints and Degrees of Freedom.”

Frame and Nllink Elements

For each analysis case, you may select the types of Frame and/or Nllink element response to be output using the parameter **frtypes**, which can take on one or more of the following values:

- FORCE: Internal forces (and deformations for the Nllink element)
- JOINTF: Element joint forces

The selected results are printed for all Frame and/or Nllink elements.

For more information:

- See Topic “Internal Force Output” (page 176) in Chapter “The Frame Element.”
- See Topic “Internal Force and Deformation Output” (page 275) in Chapter “The Nllink Element.”
- See Topic “Element Joint Force Output” (page 402) in this chapter.

Shell Elements

For each analysis case, you may select the types of Shell element response to be output using the parameter **shtypes**, which can take on one or more of the following values:

- FORCE: Internal forces
- STRESS: Stresses
- JOINTF: Element joint forces

The selected results are printed for all Shell elements.

For more information:

- See Topic “Internal Force and Stress Output” (page 198) in Chapter “The Shell Element.”
- See Topic “Element Joint Force Output” (page 402) in this chapter.

Plane, Asolid, and Solid Elements

For each analysis case, you may select the types of Plane, Asolid, and/or Solid element response to be output using the parameter **pltypes**, which can take on one or more of the following values:

- STRESS: Stresses
- JOINTF: Element joint forces

The selected results are printed for all Plane, Asolid, and/or Solid elements.

For more information:

- See Topic “Stress Output” (page 215) in Chapter “The Plane Element.”
- See Topic “Stress Output” (page 230) in Chapter “The Asolid Element.”
- See Topic “Stress Output” (page 240) in Chapter “The Solid Element.”

- See Topic “Element Joint Force Output” (page 402) in this chapter.

Pagination Control

Pagination control as described in this topic is available only by using the input data text file. You may skip this topic if you are preparing your model using the SAP2000 graphical user interface.

The input echo (.EKO) and the results output (.OUT) files are both paginated. A page header is printed at the top of each page that includes the company name, the problem title, the filename, and the page number. A sample page header is shown below:

```
PROGRAM SAP2000 - VERSION N6.10                          FILE:FRAME.OUT
NONLINEAR VERSION                                                PAGE
YOUR PROBLEM TITLE GOES HERE                                        1
```

The analysis log (.LOG) file is not paginated. A single page header appears at the beginning of the file.

You may use the parameters **page** and **lines** to control pagination as described below.

Pagination by Lines

If you want a header at the top of every physical printed page, specify **page** = LINES and specify the maximum number of lines per page using the parameter **lines**. The default value for **lines** is 59, which is appropriate for many laser printers. A new page will be started at each new section heading and each time the specified number of lines per page is reached.

Pagination by lines is most appropriate for printed output files.

Pagination by Sections

For easier viewing of the output files with an editor or word-processor, specify **page**=SECTIONS. A new page will be started only at each new section heading. The prevents page headers from interrupting the various output sections. The parameter **lines** is ignored for pagination by sections.

Chapter **XX**

References

AASHTO, 1996 a

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

AASHTO, 1996 b

LRFD Bridge Design Specifications, Customary U.S. Units, 1996 Interim Revisions, The American Association of State Highway and Transportation Officials, Inc., Washington, D.C.

ACI, 1995

Building Code Requirements for Structural Concrete (ACI 318-95) and Commentary (ACI 318R-95), American Concrete Institute, Farmington Hills, Mich.

AISC, 1994

Manual of Steel Construction, Load & Resistance Factor Design, 2nd Edition, 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 SESM 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 NUMEETA 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, 1997

Three Dimensional Dynamic Analysis of Structures with Emphasis on Earthquake Engineering, Computers and Structures, Inc., Berkeley, 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.

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.