



**User Guide**  
**PERFORM-3D™**  
Nonlinear Analysis and  
Performance Assessment for  
3D Structures

# Copyright

Copyright © Computers and Structures, Inc., 2006-2011  
All rights reserved.

The CSi Logo is a registered trademark of Computers and Structures, Inc. PERFORM is a registered trademark of Computers and Structures, Inc. Windows is a registered trademark of Microsoft Corporation

The computer program PERFORM-3D and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of these programs or reproduction of documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

No part of this publication may be reproduced or distributed in any form or by any means, or stored in a database or retrieval system, without the prior explicit written permission of the publisher.

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

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

Phone: (510) 649-2200  
FAX: (510) 649-2299  
e-mail: [info@csiberkeley.com](mailto:info@csiberkeley.com) (for general questions)  
e-mail: [support@csiberkeley.com](mailto:support@csiberkeley.com) (for technical support questions)  
web: [www.csiberkeley.com](http://www.csiberkeley.com)

# Disclaimer

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

# Acknowledgments

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

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

PERFORM-3D uses a node renumbering scheme developed by Professor Marc Hoit (see Hoit, M.I., Stoker, D. and Consolazio, G. "Neural Networks for Equation Renumbering" Journal of Computers and Structures, Vol. 52, No. 5, 1994, pp. 1011-1021). We thank Professor Hoit for allowing us to use his computer code.

# PERFORM-3D User Guide

## Version 5

### Contents

---

<b>1</b>	<b>Overview</b>	<b>1-1</b>
1.1	<i>Screen Layout</i>	1-1
1.2	<i>Structures</i>	1-2
1.3	<i>Phases</i>	1-3
1.4	<i>Modeling Phase</i>	1-3
1.5	<i>Analysis Phase</i>	1-5
1.6	<i>View Direction and Perspective</i>	1-8
1.7	<i>Reports</i>	1-9
1.7.1	Printed Reports	1-9
1.7.2	Saving Results to a File	1-9
1.7.3	ECHO File	1-9
1.7.4	Direct Access to Results Files	1-10
1.8	<i>Reporting Errors</i>	1-10
1.9	<i>Some Things to Keep in Mind</i>	1-10
<b>2</b>	<b>Structures, Folders and Files</b>	<b>2-1</b>
2.1	<i>List of Folders</i>	2-1
2.2	<i>Saving and Opening a Structure</i>	2-2
2.2.1	Saving a Structure	2-2
2.2.2	Opening a Structure	2-4
2.3	<i>Location of ECHO file</i>	2-5
2.4	<i>Moving or Copying a Project or Structure</i>	2-6
2.5	<i>Other Folders</i>	2-7
2.5.1	Records Folder	2-7
2.5.2	RecordsF Folder	2-7
2.5.3	Spectra Folder	2-8
2.5.4	User Folder	2-9
2.6	<i>Do Folder Locations Matter?</i>	2-10

<b>3</b>	<b>Nodes</b>	<b>3-1</b>
	3.1 <i>Methods for Specifying Nodes</i>	3-1
	3.2 <i>Zooming</i>	3-3
	3.3 <i>Closely Spaced Nodes</i>	3-3
	3.4 <i>Supports</i>	3-3
	3.5 <i>Masses</i>	3-4
	3.6 <i>Slaving Constraints</i>	3-5
	3.6.1 <i>General</i>	3-5
	3.6.2 <i>Rigid Floor Constraint</i>	3-5
	3.6.3 <i>Full Rigid Link Constraint</i>	3-6
	3.6.4 <i>Eccentric Connection Constraint</i>	3-6
	3.6.5 <i>Equal Displacement Constraint</i>	3-8
<b>4</b>	<b>Frames</b>	<b>4-1</b>
	4.1 <i>Plotting of Nodes and Elements</i>	4-1
	4.2 <i>Defining a Frame</i>	4-2
	4.3 <i>Duplicating a Frame</i>	4-3
<b>5</b>	<b>Component Properties</b>	<b>5-1</b>
	5.1 <i>Elements and Components</i>	5-1
	5.2 <i>Procedure for Specifying Components</i>	5-2
	5.2.1 <i>Scope</i>	5-2
	5.2.2 <i>Component Types and Names</i>	5-3
	5.2.3 <i>Materials</i>	5-4
	5.2.4 <i>Beam and Column Cross Sections</i>	5-4
	5.2.5 <i>Fiber Cross Sections for Walls</i>	5-5
	5.2.6 <i>Fiber Cross Sections for Frame Elements</i>	5-7
	5.2.7 <i>Basic Structural Components</i>	5-7
	5.2.8 <i>Plot Hysteresis Loops</i>	5-9
	5.2.9 <i>Strength Sections</i>	5-10
	5.2.10 <i>Frame Compound Components</i>	5-10
	5.2.11 <i>Shear Wall Compound Components</i>	5-11
	5.2.12 <i>General Wall Compound Components</i>	5-12
	5.2.13 <i>Viscous Bar and BRB Compound Components</i>	5-12
	5.3 <i>Managing Component Properties</i>	5-13
	5.3.1 <i>General</i>	5-13

5.3.2	Filter	5-13
5.3.3	Purge	5-13
5.3.4	Use Cross Section Strengths	5-14
5.3.5	Save As	5-14
5.3.6	Copy-Paste	5-14
5.4	<i>Printed Reports</i>	5-15
5.5	<i>F-D Relationship</i>	5-15
5.5.1	Actions and Deformations	5-15
5.5.2	Actual Components	5-15
5.5.3	PERFORM F-D Relationship	5-16
5.5.4	E-P-P and Bilinear Relationships	5-17
5.5.5	Additional Parallel Stiffness	5-17
5.6	<i>Strength Loss</i>	5-18
5.6.1	General	5-18
5.6.2	Total Strength Loss at X Point	5-19
5.6.3	Strength Loss Interaction	5-20
5.7	<i>A Warning on Strength Loss</i>	5-20
5.7.1	General	5-20
5.7.2	Components in Parallel	5-21
5.7.3	Components in Series	5-21
5.7.4	Effect on Analysis Method	5-22
5.7.5	Hinge Strength Loss in a Compound Component	5-23
5.8	<i>Column Ductile Limit Depends on Axial Force</i>	5-24
5.8.1	Effect to be Modeled	5-24
5.8.2	FEMA 356 Requirements	5-25
5.8.3	PERFORM Implementation of Variable Ductile Limit	5-27
5.8.4	Procedure	5-28
5.9	<i>Deformation and Strength Capacities</i>	5-29
5.9.1	General	5-29
5.9.2	Demand and Capacity	5-29
5.9.3	Deformation Capacities	5-30
5.9.4	Drift Capacities	5-30
5.9.5	Strength Capacities	5-31
5.9.6	Overview of Limit States and Usage Ratios	5-31
5.9.7	Note on Maximum Demand-Capacity Ratio	5-32
5.10	<i>Beam or Column Shear Strength Depends on Hinge Rotation</i>	5-33

5.10.1	General	5-33
5.10.2	PERFORM Implementation	5-33
5.10.3	Procedure for Strength Section Properties	5-34
5.10.4	Procedure for Frame Compound Component	5-35
<b>5.11</b>	<b><i>Cyclic Degradation</i></b>	<b>5-36</b>
5.11.1	Backbone Relationship	5-36
5.11.2	Hysteresis Loops	5-37
5.11.3	Dynamic Analysis	5-38
5.11.4	Procedure for Degradation Factors	5-38
5.11.5	Push-Over Analysis	5-39
<b>5.12</b>	<b><i>Control of Hysteresis Loop Shape</i></b>	<b>5-40</b>
5.12.1	Alternative Loops	5-40
5.12.2	E-P-P Relationship	5-41
5.12.3	Trilinear Relationship After U Point	5-42
5.12.4	Procedure for Unloading Stiffness Factor	5-43
<b>5.13</b>	<b><i>Cross Sections</i></b>	<b>5-43</b>
5.13.1	The Role of Cross Sections	5-43
5.13.2	Beam and Column Sections	5-44
5.13.3	Fiber Cross Sections for Beams and Columns	5-45
5.13.4	Fiber Cross Sections for Walls	5-45
5.13.5	Other Sections	5-46
<b>5.14</b>	<b><i>Using Cross Section Dimensions</i></b>	<b>5-46</b>
5.14.1	General	5-46
5.14.2	Length of Uniform X-Section Segment	5-46
5.14.3	Tributary Length for a Curvature Hinge	5-47
<b>5.15</b>	<b><i>"Auto" Components</i></b>	<b>5-48</b>
5.15.1	Auto End Zones for Beams and Columns	5-48
5.15.2	Auto Panel Zones in Steel Frames	5-48
5.15.3	Panel Zone Limit States	5-50
<b>5.16</b>	<b><i>Upper and Lower Bounds</i></b>	<b>5-50</b>
5.16.1	Overview	5-50
5.16.2	Procedure for Specifying Upper/Lower Bounds	5-51
5.16.3	Procedure for Using Upper/Lower Bounds	5-52
<b>6</b>	<b>Elements</b>	<b>6-1</b>
<b>6.1</b>	<b><i>Element Types</i></b>	<b>6-1</b>
6.1.1	General	6-1
6.1.2	Simple Bar Elements	6-1



6.1.3	Beam Elements	6-1
6.1.4	Column Elements	6-2
6.1.5	Brace/Other Frame Elements	6-3
6.1.6	Shear Wall Elements	6-4
6.1.7	General Wall Elements	6-5
6.1.8	Infill Panel Elements	6-5
6.1.9	Connection Panel Zone Elements	6-5
6.1.10	BRB Elements	6-5
6.1.11	Viscous Bar Elements	6-6
6.1.12	Seismic Isolator Elements	6-6
6.1.13	Slab/Shell Elements	6-6
6.1.14	Support Spring Elements	6-7
6.1.15	Deformation Gage Elements	6-7
6.2	<i>Specifying Elements</i>	6-8
6.2.1	Element Groups	6-8
6.2.2	Steps	6-9
6.2.3	Start a New Element Group	6-9
6.2.4	Add Elements	6-10
6.2.5	Parallel Elements	6-10
6.2.6	Delete Elements	6-10
6.3	<i>Element Orientations</i>	6-11
6.4	<i>Element Properties</i>	6-12
6.5	<i>Moving Elements Between Groups</i>	6-13
6.6	<i>P-<math>\Delta</math>, P-<math>\delta</math> and Large Displacement Effects</i>	6-13
6.6.1	General	6-13
6.6.2	P- $\Delta$ vs. True Large Displacements	6-14
6.6.3	P- $\delta$ Effect	6-17
6.6.4	Do You Need To Consider P- $\delta$ Effects?	6-19
6.6.5	Axial Shortening Due to Bending	6-19
6.6.6	PERFORM Options	6-20
<b>7</b>	<b>Load Patterns</b>	<b>7-1</b>
7.1	<i>Pattern Types and Restrictions</i>	7-1
7.2	<i>Nodal Load Patterns</i>	7-2
7.3	<i>Element Load Patterns</i>	7-2
7.4	<i>Self Weight Load Patterns</i>	7-4

<b>8</b>	<b>Drifts and Deflections</b>	<b>8-1</b>
8.1	<i>Drifts</i>	8-1
8.1.1	Simple Drift	8-1
8.1.2	Distortion Drift	8-1
8.1.3	Procedure for Specifying a Drift	8-2
8.1.4	Reference Drift	8-2
8.2	<i>Deflections</i>	8-3
<b>9</b>	<b>Structure Sections</b>	<b>9-1</b>
9.1	<i>Concept</i>	9-1
9.2	<i>Specifying Element Cuts</i>	9-3
9.3	<i>Shear Strength for a Wall Section</i>	9-4
9.4	<i>Section Groups and Strength Limit States</i>	9-5
9.5	<i>Moment, Shear and D/C Ratio Diagrams</i>	9-6
9.5.1	Procedure	9-6
9.5.2	Save D/C Values to a File	9-6
9.5.3	Signs and Sign Reversal	9-7
<b>10</b>	<b>Limit States and Usage Ratios</b>	<b>10-1</b>
10.1	<i>Limit State Types</i>	10-1
10.2	<i>Deformation Limit States</i>	10-2
10.3	<i>Strength Limit States</i>	10-4
10.4	<i>Drift Limit States</i>	10-4
10.5	<i>Deflection Limit States</i>	10-5
10.6	<i>Structure Section Limit States</i>	10-5
10.7	<i>Limit State Groups</i>	10-5
<b>11</b>	<b>Inactive Elements</b>	<b>11-1</b>
11.1	<i>Inactive Elements</i>	11-1
11.2	<i>Procedure</i>	11-1
<b>12</b>	<b>Gravity Load Cases</b>	<b>12-1</b>
12.1	<i>Gravity Loads</i>	12-1

12.2	<i>Procedure</i>	12-2
12.2.1	Steps	12-2
12.2.2	Number of Load Steps	12-2
12.2.3	Maximum Number of Events	12-3
12.2.4	Initial Step to First Event	12-3
12.2.5	Analysis Limit State	12-4
<b>13</b>	<b>Static Push-Over Load Cases</b>	<b>13-1</b>
13.1	<i>Push-Over Loads</i>	13-1
13.1.1	Load Distribution and Magnitude	13-1
13.1.2	Choice of Load Distribution	13-1
13.1.3	Load Distribution Based on Mode Shapes	13-2
13.1.4	Linear Analysis Option	13-3
13.2	<i>Procedure</i>	13-3
13.2.1	Steps	13-3
13.2.2	Reference Drift	13-5
13.2.3	Number of Steps	13-5
13.2.4	Maximum Number of Events	13-6
13.2.5	Initial Step to First Event	13-7
13.2.6	Analysis Limit State	13-7
13.3	<i>Loads Based on Mode Shapes</i>	13-8
13.3.1	Symmetrical Structure	13-8
13.3.2	Unsymmetrical Structure or Diagonal Push-Over	13-9
13.3.3	Other Key Points	13-10
<b>14</b>	<b>Dynamic Earthquake Load Cases</b>	<b>14-1</b>
14.1	<i>Earthquake Loads</i>	14-1
14.2	<i>Procedure</i>	14-1
14.2.1	Steps	14-1
14.2.2	Total Time	14-2
14.2.3	Time Step	14-3
14.2.4	Maximum Number of Events	14-4
14.2.5	Save Interval	14-4
14.2.6	Analysis Limit State	14-4
<b>15</b>	<b>Earthquake Records</b>	<b>15-1</b>
15.1	<i>Organization</i>	15-1
15.1.1	Overall Steps	15-1
15.1.2	Organization of Earthquake Records	15-1

15.1.3	Copying Earthquake Records	15-3
15.2	<i>Setting Up a New Earthquake Record</i>	15-3
15.2.1	Text File Format	15-3
15.2.2	Adding A New Record	15-4
15.3	<i>Reviewing Existing Records</i>	15-5
<b>16</b>	<b>Dynamic Force Load Cases and Analysis</b>	<b>16-1</b>
16.1	<i>Purpose and Overall Procedure</i>	16-1
16.1.1	Purpose	16-1
16.1.2	Overall Procedure	16-1
16.2	<i>Organization of Dynamic Force Records</i>	16-2
16.2.1	RecordsF Folder	16-2
16.2.2	Steps for Setting Up Dynamic Force Records	16-3
16.2.3	Copying Dynamic Force Records	16-3
16.3	<i>Setting Up a New Dynamic Force Record</i>	16-4
16.3.1	Text File Format	16-4
16.3.2	Adding a New Record	16-4
16.4	<i>Reviewing Existing Records</i>	16-5
16.5	<i>Dynamic Force Load Cases</i>	16-6
16.5.1	Load Cases	16-6
16.6	<i>Multi-Support Earthquake Analysis</i>	16-7
16.6.1	Purpose	16-7
16.6.2	Analysis Method	16-7
16.6.3	Imposed Dynamic Support Displacements	16-8
16.6.4	Ground Displacements	16-9
16.6.5	Ground Displacement Records	16-10
16.6.6	Alpha-M Damping	16-12
16.6.7	Results Interpretation	16-12
<b>17</b>	<b>Analysis Series</b>	<b>17-1</b>
17.1	<i>Analysis Series Concept</i>	17-1
17.1.1	Concept	17-1
17.1.2	Procedure	17-1
17.2	<i>Basic Parameters and Masses</i>	17-3
17.2.1	Event Overshoot Factor	17-4
17.3	<i>Damping</i>	17-6

17.3.1	General	17-6
17.3.2	Procedure for Modal Damping	17-7
17.3.3	Procedure for Rayleigh Damping	17-7
17.4	<i>Procedure for Using Upper/Lower Bounds</i>	17-7
17.5	<i>Quick'n'Dirty</i>	17-10
<b>18</b>	<b>"Elastic" Viscous Damping</b>	<b>18-1</b>
18.1	<i>Modal Damping</i>	18-1
18.1.1	Damping Matrix	18-1
18.1.2	Physical Interpretation	18-2
18.2	<i>Rayleigh Damping</i>	18-3
18.2.1	Rayleigh Damping Model	18-3
18.2.2	Implied Modal Damping Ratio for Linear Analysis	18-4
18.2.3	Implied Damping for Nonlinear Analysis	18-6
18.2.4	PERFORM Implementation	18-7
18.2.5	Procedure for Specifying Rayleigh Damping	18-8
18.2.6	Special Assumption for Concrete Fibers	18-8
18.2.7	Alternative Procedure for Base Isolation	18-9
18.3	<i>A Case Where Modal vs. Rayleigh Damping May Be Substantially Different : Coupled Shear Walls</i>	18-11
18.4	<i>Combined Modal and Rayleigh Damping</i>	18-12
<b>19</b>	<b>General Load Sequence</b>	<b>19-1</b>
1.1	<i>Standard and General Load Sequences</i>	19-1
1.2	<i>Points to Note for General Sequence</i>	19-2
19.1.1	General Push-Over and Target Displacement Plots	19-2
19.1.2	Unload Push-Over Load Case Type	19-2
19.1.3	Analysis Limit State	19-3
19.1.4	Controlled Drifts	19-3
19.1.5	Removing Gravity Loads	19-4
19.1.6	Return To Static State After Dynamic Analysis	19-4
19.1.7	Graphs That Cover Several Analyses	19-5
19.1.8	Inactive Elements	19-5

<b>20</b>	<b>Running Analyses</b>	<b>20-1</b>
	20.1 <i>Steps for Running Analyses</i>	20-1
	20.2 <i>Adding and Deleting Analyses</i>	20-2
<b>21</b>	<b>Mode Shapes</b>	<b>21-1</b>
	21.1 <i>Plotting Mode Shapes</i>	21-1
	21.2 <i>Using Mode Shapes for Push-Over Analysis</i>	21-2
	21.3 <i>Vertical Masses</i>	21-2
<b>22</b>	<b>Response Spectrum Analysis</b>	<b>22-1</b>
	22.1 <i>Response Spectrum Load Cases</i>	22-1
	22.1.1 Spectra	22-1
	22.1.2 Response Spectrum Load Cases	22-2
	22.2 <i>Response Spectrum Analyses</i>	22-3
	22.2.1 Run Analysis	22-3
	22.2.2 Analysis Results	22-4
	22.2.3 Lateral Load Patterns from Analysis Results	22-4
<b>23</b>	<b>Energy Balance</b>	<b>23-1</b>
	23.1 <i>Types of Energy</i>	23-1
	23.1.1 Dynamic Analysis	23-1
	23.1.2 Static Analysis	23-2
	23.1.3 Calculation of Inelastic and Strain Energies	23-2
	23.2 <i>Energy Plots</i>	23-5
	23.2.1 Complete Structure	23-5
	23.2.2 Inelastic Energies in Element Groups	23-5
	23.2.3 $\beta$ -K Energies in Element Groups	23-5
	23.2.4 Energy Error	23-6
	23.3 <i>Approximate Damping Ratio</i>	23-6
	23.3.1 Overview	23-6
	23.3.2 Theory	23-6
	23.3.3 Procedure	23-8
<b>24</b>	<b>Deflected Shape Plots</b>	<b>24-1</b>
	24.1 <i>Deflected Shape</i>	24-1
	24.2 <i>Procedure</i>	24-2

24.2.1	Deflected Shape Only	24-2
24.2.2	Deflected Shape with Usage Ratio Coloring	24-3
24.2.3	Some Points on Components vs. Entire Element	24-4
24.2.4	Other Aspects	24-4
24.2.5	Save Usage Ratios to a File	24-5
<b>25</b>	<b>Time History Plots</b>	<b>25-1</b>
25.1	<i>Plot History for a Single Node</i>	25-1
25.1.1	Options	25-1
25.1.2	Procedure	25-2
25.1.3	Save Plotted History to a File	25-2
25.2	<i>Save Histories for Multiple Nodes</i>	25-2
25.3	<i>Plot History for a Single Element</i>	25-3
25.3.1	Procedure	25-3
25.3.2	Save Plotted History to a File	25-4
25.4	<i>Save Histories for Multiple Elements</i>	25-5
25.5	<i>Drift or Deflection Histories</i>	25-6
25.6	<i>Structure Section Force Histories</i>	25-6
25.7	<i>Plot Section Results for Multiple Load Cases</i>	25-7
25.7.1	Purpose	25-7
25.7.2	Procedure	25-7
<b>26</b>	<b>Hysteresis Loop Plots</b>	<b>26-1</b>
26.1	<i>Purpose</i>	26-1
26.2	<i>Procedure</i>	26-1
26.3	<i>Cut-Off Corners</i>	26-3
26.4	<i>Save Loops to a File</i>	26-3
<b>27</b>	<b>Moment and Shear Diagrams</b>	<b>27-1</b>
27.1	<i>Detailed Results for a Single Element</i>	27-1
27.1.1	Element Selection	27-1
27.1.2	Element Loads Option	27-2
27.1.3	Moment and Shear Envelopes Option	27-2
27.1.4	Moment and Shear Histories Option	27-2
27.1.5	Deflected Shapes Option	27-3

27.2	<i>Moment and Shear Diagrams for a Line of Elements</i>	27-4
27.3	<i>Moment and Shear Diagrams Based on Structure Sections</i>	27-5
<b>28</b>	<b>General Push-Over Plots</b>	<b>28-1</b>
28.1	<i>Overview</i>	28-1
28.1.1	Available Methods	28-1
28.1.2	Differences Among Methods	28-1
28.1.3	Some Key Points	28-6
28.1.4	Main Steps	28-8
28.2	<i>Procedure for Defining Response Spectra</i>	28-10
28.2.1	Spectrum Types	28-10
28.2.2	Add or Edit a Spectrum	28-10
28.2.3	Copy Spectra from a Different Structure	28-11
28.3	<i>Procedure for Capacity and Demand Curves</i>	28-11
28.3.1	Choose Task	28-11
28.3.2	Plot Capacity Curve	28-11
28.3.3	$S_a$ vs. $S_d$ Option	28-12
28.3.4	Spectral Acceleration Capacity	28-12
28.3.5	Define Trial Points	28-14
28.3.6	Period Calculation	28-15
28.3.7	Choose Push-Over Method and Response Spectrum	28-20
28.3.8	Plot Demand Curve	28-22
28.3.9	Capacity Curve Scaling for Sensitivity Studies	28-23
28.4	<i>Performance Evaluation</i>	28-23
28.5	<i>Consistency</i>	28-24
28.5.1	General	28-24
28.5.2	Calculation of Spectral Displacement	28-24
28.5.3	Suggested Methods	28-26
28.5.4	A Reason for Using $S_a$ vs. $S_d$ Option	28-26
28.6	<i>Energy Dissipation and Damping</i>	28-27
28.6.1	General	28-27
28.6.2	Stiffness and Degradation and Energy Ratio	28-27
28.6.3	Equivalent Damping Ratio	28-28
28.6.4	Degradation in Push-Over Methods	28-29
28.6.5	PERFORM Method for Energy Ratio	28-30
28.6.6	Extra Damping from Fluid Dampers	28-30



28.6.7	Strength Loss and Degradation	28-32
28.7	<i>Theory for Relationship Between Sa and H</i>	28-32
28.7.1	General	28-32
28.7.2	Special Case With Modal Load and Displaced Shape	28-33
28.7.3	General Case Where Load Matches Displaced Shape	28-36
28.7.4	Case Where Load Does Not Match Displaced Shape	28-36
<b>29</b>	<b>Target Displacement Plot</b>	<b>29-1</b>
29.1	<i>Target Displacement Method</i>	29-1
29.1.1	Concept	29-1
29.1.2	Target Drift	29-2
29.1.3	Spectral Drift	29-3
29.1.4	Factor $C_0$	29-3
29.1.5	Factor $C_1$	29-3
29.1.6	Factor $C_2$	29-4
29.1.7	Factor $C_3$	29-4
29.2	<i>PERFORM Implementation</i>	29-4
<b>30</b>	<b>Usage Ratio Graphs</b>	<b>30-1</b>
30.1	<i>Usage Ratios</i>	30-1
30.2	<i>Procedure</i>	30-2
<b>31</b>	<b>Load Case Combinations and Envelopes.</b>	<b>31-1</b>
31.1	<i>Overview</i>	31-1
31.1.1	Main Steps	31-1
31.1.2	Load Case Combinations	31-2
31.1.3	Combination Methods	31-3
31.2	<i>Procedure</i>	31-4
31.2.1	Load Combinations	31-4
31.2.2	Usage Ratios for Structure	31-5
31.2.3	Usage Ratios for Elements	31-6
<b>32</b>	<b>Importing/Exporting Structure Data</b>	<b>32-1</b>
32.1	<i>Importing Structure Data</i>	32-1
32.1.1	Text Files	32-1

32.1.2	File for Elements (with Optional Nodes)	32-2
32.1.3	File for Nodes	32-3
32.1.4	File for Nodal Masses	32-4
32.1.5	File for Nodal Loads	32-5
32.2	<i>Note on Element Import</i>	32-6
32.3	<i>Import and Export Procedures</i>	32-8
32.3.1	Import	32-8
32.3.2	Export	32-9
32.4	<i>Component Properties Export/Import</i>	32-10
32.4.1	General	32-10
32.4.2	Export/Import Steps	32-10
<b>33</b>	<b>Procedure For Reporting Errors</b>	<b>33-1</b>
33.1	<i>Reporting Errors</i>	33-1
33.2	<i>Procedure For Sending Files</i>	33-1
33.2.1	Structure Data	33-1
33.2.2	Send As An Email Attachment	33-2
33.2.3	Upload to FTP Site	33-2

# 1 Overview

---

*PERFORM-3D is a typical Windows program. You can move around the program by clicking on buttons and folder tabs; select items by clicking on a drawing of the structure or by choosing from drop-down lists; and enter data by typing into text boxes. If you hover the mouse pointer over a button, a tool tip that describes its purpose will pop up. Some text boxes also have tool tips. In this version of PERFORM-3D there is no Help button to call up context-sensitive help.*

*This chapter provides an overview of the program and covers some things you should know before you begin.*

## 1.1 Screen Layout

---

Figure 1.1 shows the screen layout for most tasks. The minimum required screen resolution is 1024 x768 pixels. Figure 1.1 is the layout at this resolution. If the resolution is finer the size of the graphics panel is increased.

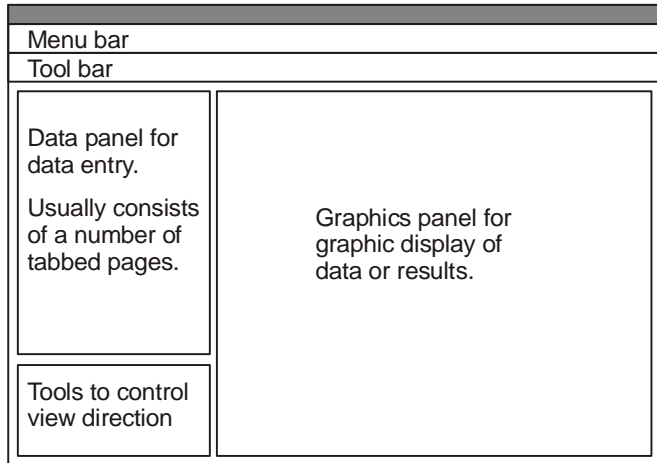


Figure 1.1 Screen Layout For Most Tasks

Figure 1.2 shows the screen layout for specifying the properties of linear and nonlinear components. This screen has diagrams to explain the required properties.

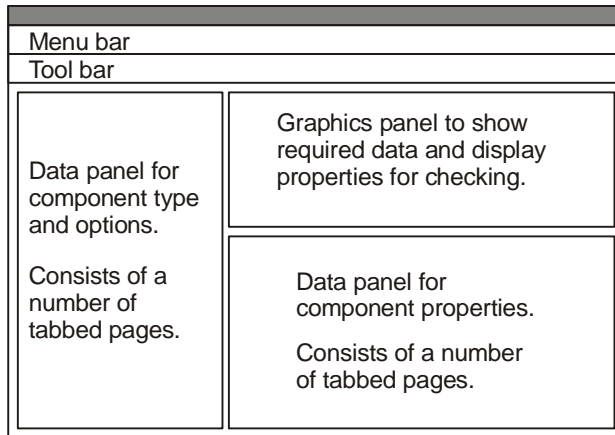


Figure 1.2 Screen Layout For Component Properties

There are a number of other screens, most of them self-explanatory. The best way to learn the program is to work through a tutorial, and also to experiment with the program features.

## 1.2 Structures

---

You can set up any number of structures (analysis models). Each structure must have a unique name. For an explanation of how to manage structures and data files, see *Chapter 2, Structures, Folders and Files*.

When you start PERFORM-3D you have the option of starting a new structure or opening an existing structure. After you make this choice

you can press buttons on the toolbar to move around the program. If you prefer, you can use the menus rather than the toolbar buttons.

The toolbar buttons are as follows.



Start a new structure from scratch.



Open an existing structure.



Save any changes you have made to the current structure.



Save the current structure as a new structure.



Delete the current structure.

If you are a beginning user, we suggest that you start by opening one of the example structures and playing with it. If you make changes to an example, be careful not to save these changes. If you want to make changes to an example, first save it as a new structure, then make the changes.

### **1.3 Phases**

---

There are two phases, namely Modeling and Analysis. The toolbar buttons for phases are as follows.



Modeling phase (build the structure).



Analysis phase (calculate and process the analysis results).

Each phase has a number of tasks, as described in the following sections.

### **1.4 Modeling Phase**

---

The toolbar buttons for tasks in the modeling phase are as follows.



Umbrella task. Specify overall information for the structure. After you press the toolbar button the required information is self-explanatory.



Nodes task. Specify node data, including coordinates, support conditions, slaving constraints (e.g. rigid floor diaphragms) and structure masses. This is a relatively simple task, and the required information is largely self-explanatory. For some key points see *Chapter 3, Nodes*.



Component Properties task. Specify properties for nonlinear and linear components. This is the most critical and complex task. For an explanation see *Chapter 5, Component Properties*.



Elements task. Specify element data, including element types, locations and properties. For an explanation see *Chapter 6, Elements*.



Frames task. The plotted geometry for a 3D structure can be complex. You can (and usually must) simplify the plot by showing only selected parts of the structure. A "frame" is any part of the complete structure. It can be a plane frame, a floor, or any other part of the structure that you would like to plot separately from the structure as a whole. You can quickly set up, modify and delete frames, and then toggle between a full structure view and a view showing only a single frame. For an explanation see *Chapter 4, Frames*.



Load Patterns task. Specify load patterns for node, element and/or self weight loads. For this version of the program there is only a limited choice for element loads. For an explanation see *Chapter 7, Load Patterns*. In the analysis phase you can combine load patterns to form load cases. See *Chapter 12, Gravity Load Cases*, and *Chapter 13, Static Push-Over Load Cases*.



Import Structure Data task. Import nodes, masses, elements and/or loads from a text file. If you have built an analysis model in another computer program, and if you can set up the

data for this model in a comma-delimited text file, you can use this task to import much of the structure into PERFORM-3D. For an explanation see *Chapter 32, Importing/Exporting Structure Data*.



Drifts and Deflections task. Specify drifts and deflections. Horizontal drifts are valuable measures of structure deformation, and you will typically specify a number of them. For long span structures you may also use vertical deflections as deformation measures. For an explanation see *Chapter 8, Drifts and Deflections*.



Structure Sections task. Specify structure sections. If the structure has a number of lateral load resisting systems, you may want to know how the total load is distributed among them. You can do this by specifying sections through parts of the structure and examining the forces on these sections. For an explanation see *Chapter 9, Structure Sections*.



Limit States task. Specify limit states. This is an extremely important task. If you make effective use of limit states you can greatly simplify the decision making process. For an explanation see *Chapter 10, Limit States and Usage Ratios*.



Inactive Elements task. Specify elements that are inactive for gravity loads. For an explanation see *Chapter 11, Inactive Elements*.

You do not have to perform these tasks in any particular sequence. For example, you can specify some nodes, then some elements, then more nodes, then some component properties, etc.

## 1.5 Analysis Phase

---

The tasks in the analysis phase can be divided into Structural Analysis tasks, Behavior Assessment tasks and Demand-Capacity tasks. The structural analysis tasks are for defining load cases and running structural analyses. The behavior assessment tasks allow you to examine and check the behavior of the analysis model. The demand-

capacity tasks allow you to calculate demand-capacity ratios, and hence make decisions about the performance of the structure.

The toolbar buttons for the structural analysis tasks are as follows.



Load Cases task. Specify gravity, push-over, earthquake and other load cases. For explanations see *Chapter 12, Gravity Load Cases*, *Chapter 13, Push-Over Load Cases*, *Chapter 14, Earthquake Load Cases* and *Chapter 16, Dynamic Force Load Cases and Analyses*.



Run Analyses task. Run static and dynamic analyses. For an explanation see *Chapter 17, Analysis Series*, *Chapter 19, General Load Sequence* and *Chapter 20, Running Structural Analyses*.

The toolbar buttons for the behavior assessment tasks are as follows.



Mode Properties task. Examine the calculated mode periods and mode shapes, and examine the results of response spectrum analyses.. For an explanation see *Chapter 21, Mode Shapes*. For response spectrum analysis see *Chapter 22, Response Spectrum Analysis*.



Energy Balance task. Plot graphs that show how much energy of different types is dissipated by the structure. For an explanation see *Chapter 23, Energy Balance*.



Limit State Groups task. Specify limit state groups. In the modeling phase you may have specified a large number of limit states. This task allows you to organize the limit states into related groups, and hence simplify the decision-making process. For an explanation see *Chapter 10, Limit States and Usage Ratios*.



Deflected Shape task. Plot deflected shapes, either static or animated. As an option, color the elements based on their usage ratios. This can be extremely useful for assessing the behavior of a structure. For an explanation see *Chapter 24, Deflected Shape Plots*.





Time History task. For dynamic analyses, plot time history graphs of a variety of response quantities, including nodal displacements, velocities and accelerations, element responses of a variety of types, drifts, and cross section forces. For static push-over analyses, plot graphs of similar response quantities against structure drift. For an explanation see *Chapter 25, Time History Plots*.



Hysteresis Loop task. For dynamic analyses, plot hysteresis loops for inelastic components (e.g. plot loops of bending moment at a plastic hinge versus hinge rotation). For an explanation see *Chapter 26, Hysteresis Loop Plots*.



Moment and Shear Diagrams task. For beams, columns and walls, plot bending moment and shear force diagrams. For beams and columns draw detailed deflected shapes. For an explanation see *Chapter 27, Moment and Shear Diagrams*.

The toolbar buttons for the demand-capacity tasks are as follows.



General Push-Over task. Plot capacity and demand curves, and hence assess the structural performance. This task covers a number of different push-over methods. For an explanation see *Chapter 28, General Push-Over Plot*.



Target Displacement task. Plot the capacity curve and calculate the target displacement for push-over analysis using the FEMA 356 Coefficient (Target Displacement) method. This task has been superseded by the preceding General Push-Over task, but has been kept for historical reasons. For an explanation see *Chapter 29, Target Displacement Plot*.



Usage Ratio task. For any analysis and limit states, plot graphs of usage ratio versus time, drift or load factor. For an explanation see *Chapter 30, Usage Ratio Plots*.



Combinations and Envelopes task. Combine the results from a number of analyses, and calculate usage ratios based on maximum or mean values, using a variety of combination methods. If desired, color the elements based on their usage ratios. See *Chapter 31, Load Case Combinations and Envelopes*.

## 1.6 View Direction and Perspective

The tools for controlling the view direction are at the bottom left of the screen. These tools are shown in Figure 1.3.

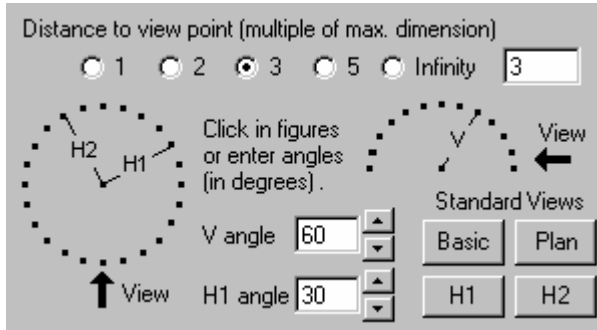


Figure 1.3 Tools for View Direction and Perspective

You can view the structure from any direction in plan, and at any vertical angle from directly downwards to directly upwards. To change the view direction, either click in the diagrams to set the directions of the axes, or enter angles in the text boxes. To change the viewing distance for the perspective projection, choose one of the standard distances or enter a distance in the text box. Then press **OK** to change the view, or **Cancel** to retain the current view.

The "Basic" view is the default. If you have changed the view direction, press the "Basic" button to return to this view. Press the "Plan", "H1" or "H2" button to get a plan or elevation view, with no perspective (the view distance is infinity). You can add perspective to plan and elevation views by choosing a view distance.

## 1.7 Reports

---

### 1.7.1 Printed Reports

You can obtain printed reports in both the Modeling and Analysis phases. For each task there is a button with a printer symbol. For most tasks in the Modeling phase this button is near the right end of the toolbar. For a few tasks it is in body of the form. To get a hard copy of the data in the current form, press the printer button. If the button is disabled you can not print the data. Either the task is not yet complete, or there is no print option for the current data.

You can change the paper size and orientation by choosing **Printer setup** from the **File** menu. Some reports are printed only in portrait orientation.

### 1.7.2 Saving Results to a File

In the Analysis phase you can save results to a text file for several of the tasks. For example, you can save time history results in the Time History task, and you can save usage ratios in the Usage Ratio task. You can then process the results using a spreadsheet program.

There is a file button near the right end of the toolbar. If this button is enabled (it will be green) you can save the current results to a file. You will be asked to give a name and other information for the file. The file will have header lines that explain its contents.

### 1.7.3 ECHO File

If you want a printed report that covers the entire Modeling phase (i.e., a report that describes the entire analysis model), you must go through each of the modeling tasks, print one or more reports for each task, and organize the separate parts into a single report. This gives you a lot of flexibility in setting up a report, but it can take a fair amount of work. As an alternative you can use the PERFORM-3D ECHO file.

As PERFORM-3D processes the structure data, it writes a record of the data to the ECHO file. This is a text file. It is not as easy to interpret as the printed reports, but it provides a full description of the structure and loading. If you wish you can examine this file (e.g., using the Windows WordPad utility program) and also print it out.

For the location of the ECHO file, see *Chapter 2, Structures, Folders and Files*.

### 1.7.4 Direct Access to Results Files

PERFORM-3D provides several tools for processing analysis results, and also allows you to save most results to text files for processing by a spreadsheet program. However, you may have special processing needs that are not included in these processing options.

If this is the case, it is possible for you to access the PERFORM-3D results files directly, assuming that you are able to write your own computer code to read the files and process the results. The results files are mostly binary direct access files (simple byte streams). In order to use these files you must know their record lengths and the data contained in each record. If you need this information, please contact us.

## 1.8 Reporting Errors

---

We do our best to keep PERFORM-3D reliable and free of errors, but we are not always successful. If you find an error in the program, please report it to us. For the procedure see *Chapter 33, Reporting Errors*.

## 1.9 Some Things to Keep in Mind

---

PERFORM-3D provides you with tools to build nonlinear analysis models, analyze them, and process the results to help you make design decisions. You will find that you do not need to know much about the details of nonlinear analysis, because PERFORM-3D largely automates the analysis process. However, nonlinear behavior is more complex than linear behavior, and deformation-based design is different from strength-based design. To use the program effectively you must understand nonlinear structural behavior and performance-based design.

Keep in mind that the goal of structural analysis is not to get an accurate simulation of a structure's behavior, but to get information to help in making design decisions. Once an analysis model has been set up, it is easy to be seduced into believing that the model is an exact representation of the real structure. It probably is not, and it does not need to be. The model must be sufficiently accurate to provide useful design information, but the analysis results are almost certainly approximate, no matter how sophisticated the analysis model.

Also keep in mind that analysis is not an end in itself, but merely a tool to help in decision making. It can be entertaining to plot deflected shapes, time histories of node and element response, and hysteresis loops for nonlinear components. This is not a trivial activity, since such plots are certainly useful for checking the behavior of the analysis model. Ultimately, however, decision making must be based on limit states and demand-capacity ratios.



## 2 Structures, Folders and Files

---

*A PERFORM-3D analysis model is a "structure". For each structure PERFORM-3D sets up a number of files. Since you may create and analyze a large number of structures, you should know a few things about how the files are organized. This chapter gives an overview. It is assumed that you are familiar with Windows folders (directories) and files.*

### 2.1 List of Folders

---

PERFORM-3D makes use of a number of "standard" folders, for storing the executable program, the user manuals, ground acceleration records, and other things. For Version 4, all default folders were located in the Microsoft Windows ProgramFiles folder. Starting with Version 5, several of the standard folders have been relocated because Microsoft has added security features to Windows, which generally prevent the original folder locations from being used. See the Important Note in Section 2.2.1 *Saving a Structure* with respect to the relocation of existing files when changing from Version 4 to Version 5 or later.

The standard folders include the following.

- (1) *Program* folder. This folder contains the PERFORM-3D executable program, plus several files that are used by the program. PERFORM-3D installs, by default, in the following Windows folder (directory) :

*C:\ProgramFiles\Computers and  
Structures\PERFORM\PERFORM-3D\Program*

You may have chosen some other folder when you installed the program. Call this the *Program Folder*.

- (2) *Manuals* folder. This folder contains the PERFORM-3D user documentation. This folder is stored as a subfolder in the *PERFORM-3D Folder*.

- (3) *Key Driver* folder. This folder contains drivers for the PERFORM-3D security hardware. This folder is stored as a subfolder in the *PERFORM-3D Folder*.
- (4) *Structures* folder. This folder is the default location for PERFORM-3D structures. See Section 2.2.1 for a description of its use and location.
- (5) *Records* folder. This folder contains the ground acceleration records that are available to PERFORM. See Section 2.5.1 for an explanation of its use and default location.
- (6) *RecordsF* folder. This folder contains the dynamic force records that are available to PERFORM. See Section 2.5.2 for an explanation of its use and default location.
- (7) *Spectra* folder. This folder contains the response spectra that are available to PERFORM. See Section 2.5.3 for an explanation of its use and default location.
- (8) *User* folder. You can use this as the default folder for exporting and importing data. Alternatively you can specify a folder in a different location. Usually you will specify a different location. See Section 2.5.4 for more information about its use and an explanation of its default location.

## **2.2 Saving and Opening a Structure**

---

### 2.2.1 Saving a Structure

When you first save a new structure, you have the option of putting it in the *Default Structures Folder* (choose the "Default" option) or in a *User Structures Folder* (choose the "User defined" option). For the user defined option you can type in the path to the folder or (more likely) browse to find the folder. If the folder does not yet exist, it will be created when you save the structure (see Section 2.5.4 for more information about its use and an explanation of its default location).

For Version 4, the *Default Structures Folder* set up during installation is located here:



*C:\ProgramFiles\Computers and Structures\  
PERFORM\PERFORM-3D\Structures*

Starting with Version 5, the location depends on Windows, which can be a bit quirky, so the *Default Structures Folder* is set up here:

For computers using Windows 7 or Vista:

*C:\Users\((UserNameAtInstallation)\My Documents\  
Computers and Structures\PERFORM*

For computers using Windows XP:

*C:\Documents and Settings\((UserNameAtInstallation)\  
My Documents\Computers and Structures\PERFORM*

where *(UserNameAtInstallation)* is the logged-on user at the time that PERFORM-3D was installed. For a computer that has several users, you may wish to set up a special user for the installation, and to log in as that user when running PERFORM-3D.

Unless you specify otherwise, when you save a structure it will be placed in the *Default Structures Folder*. When you re-save a structure that has already been saved, it is saved automatically in its folder. You can use **Save As** to save a structure as a new structure with a different name. Do not try to achieve the same result by copying and renaming the folder for a structure. This will not work.

For each structure in the Default Structures Folder there will be a subfolder that has the same name as the structure. For example, if you have a structure named "Structure-1", it will be in the folder :

Version 4

*C:\ProgramFiles\Computers and Structures\PERFORM\  
PERFORM-3D\Structures\Structure-1*

or

Version 5 or later and Windows or Vista

*C:\Users\((UserNameAtInstallation)\My Documents\  
Computers and Structures\PERFORM\Structures\Structure-1*

or

Version 5 or later and Windows XP

*C:\Documents and Settings\(*UserNameAtInstallation*)\  
My Documents\Computers and Structures\  
PERFORM\Structures\Structure-1*

If you set up a lot of structures, the Default Structures Folder can get crowded. To avoid this you have the option of setting up your own *User Structures Folders*. For example, if you have a number of structures in a project name Project A, you might set up a User Structures Folder *Project-A*. The path to this folder, using Version 5 or later and Windows 7 or Vista might be :

*C:\Users\(*UserNameAtInstallation*)\My Documents\  
Computers and Structures\PERFORM\Project-A*

However, you can choose any path you wish.

**IMPORTANT NOTE:** If you are replacing Version 4 with Version 5 or later, you will have existing *Structure/Default Structures Folder* containing subfolders and files, as well as *User, Records, RecordF*, and *Spectra* folders (see Section 2.5), that you probably want to keep. These folders are not moved to the new locations during the install, and they can not be used by Version 5 or later. You must move the folders to the new locations.

To do this you will presumably use Windows Explorer. You must know the locations of the existing files and the new locations as explained in Sections 2.2.1 and 2.5. You may need Administrator status to move the folders.

It goes without saying that you should copy, rather than move, the folders, and that you should delete the existing folders only after you are sure that they have been copied correctly.

## 2.2.2 Opening a Structure

When you open an existing structure you again have the "Default" and "User" options. If you choose "Default", the structures in the Default Structures Folder are listed in a table. You can sort this table by date or by name. To open a structure, click on it in the table and press the Open button (or alternatively double click on it in the table). If you choose "User" you can type in the folder name or find it by browsing. When

you select a User Structures Folder, the structures in it will be listed in the same way.

There is also a third option when you open an existing structure, namely the "Recent" option. If you choose this option the table will contain up to the 10 most recently opened structures. This is the option that you are likely to use most often.

## 2.3 Location of ECHO file

---

The file ECHO.txt contains a record of the structure data, loading and analyses. The location of this file is as follows.

Suppose that you have set up a structure "Structure-1" The Structures Folder will contain a folder with this name. If you use the Default Structures Folder, the path is :

Version 4

*C:\ProgramFiles\Computers and Structures\PERFORM\  
PERFORM-3D\Structures\Structure-1*

or

Version 5 or later and Windows or Vista

*C:\Users\{(UserNameAtInstallation)\My Documents\  
Computers and Structures\PERFORM\Structures\Structure-1*

or

Version 5 or later and Windows XP

*C:\Documents and Settings\{(UserNameAtInstallation)\  
My Documents\Computers and Structures\  
PERFORM\Structures\Structure-1*

The folder *Structure-1* has a number of files that define the structure, plus a subfolder for each Analysis Series (see *Chapter 17, Analysis Series*). For example, if you have an Analysis Series "Series-A" and you use the Default Structures Folder, the folder for the Analysis Series is :

Version 4

*C:\ProgramFiles\Computers and Structures\PERFORM\  
PERFORM-3D\Structures\Structure-1\Series-A*

or

Version 5 or later and Windows or Vista

*C:\Users\(*UserNameAtInstallation*)\My Documents\  
Computers and Structures\PERFORM\Structures\  
Structure-1\Series-A*

or

Version 5 or later and Windows XP

*C:\Documents and Settings\(*UserNameAtInstallation*)\  
My Documents\Computers and Structures\  
PERFORM\Structures\Structure-1\Series-A*

This folder contains a number (usually a lot) of files. Never delete, rename or otherwise change any of these files. One of the files is the ECHO.txt file for the Analysis Series. Also see the Important Note in Section 2.2.1 *Saving a Structure* with respect to the relocation of existing files when changing from Version 4 to Version 5.

## **2.4 Moving or Copying a Project or Structure**

---

To move an entire project or a single structure, move the corresponding folder using Windows Explorer.

To copy a project or a structure from one computer to another, copy the corresponding folder. If you do this, and if you are running dynamic analyses, you must make sure that the earthquake acceleration records are available on the second computer. For the procedure see *Chapter 15, Earthquake Records*.

You can rename the folder for a project. However, do not rename the folder for a structure. The folder name for a structure must be the same as the structure name.

Also see the Important Note in Section 2.2.1 *Saving a Structure* with respect to the relocation of existing files when changing from Version 4 to Version 5 or later.

## 2.5 Other Folders

---

### 2.5.1 Records Folder

The first time that you execute PERFORM-3D it creates a *Records* folder to hold ground acceleration records for dynamic earthquake analysis. All acceleration records are stored in this folder. For the procedure see *Chapter 15, Earthquake Records*.

For Version 4, if you installed the program in the default location, the Records folder is:

*C:\ProgramFiles\Computers and Structures\PERFORM\Records*

Starting with Version 5, the *Records* folder location depends on Windows, which can be a bit quirky, and is probably here:

For computers using Windows 7 or Vista:

*C:\Users\((UserNameAtInstallation))\My Documents\Computers and Structures\PERFORM*

For computers using Windows XP:

*C:\Documents and Settings\((UserNameAtInstallation))\My Documents\Computers and Structures\PERFORM*

where *(UserNameAtInstallation)* is the logged-on user at the time that PERFORM-3D was installed. For a computer that has several users, you may wish to set up a special user for the installation, and to log in as that user when running PERFORM-3D. See Section 2.2.1 for more information, including the Important Note concerning relocation of files when upgrading from Version 4 to 5 or later.

### 2.5.2 RecordsF Folder

The first time that you execute PERFORM-3D it creates a *RecordsF* folder to hold force time-history records for dynamic force analysis. All force time history records are stored in this folder. For the procedure see *Chapter 16, Dynamic Force Load Cases and Analyses, Section 16.3*. You cannot change the location of this folder.

For Versions 4, if you installed the program in the default location, the *RecordsF* folder is:

*C:\ProgramFiles\Computers and Structures\PERFORM\RecordsF*

Starting with Version 5, the *RecordsF* folder location depends on Windows, which can be a bit quirky, and is probably here:

For computers using Windows 7 or Vista:

*C:\Users\{UserNameAtInstallation}\My Documents\Computers and Structures\PERFORM*

For computers using Windows XP:

*C:\Documents and Settings\{UserNameAtInstallation}\My Documents\Computers and Structures\PERFORM*

where (*UserNameAtInstallation*) is the logged-on user at the time that PERFORM-3D was installed. For a computer that has several users, you may wish to set up a special user for the installation, and to log in as that user when running PERFORM-3D. See Section 2.2.1 for more information, including the Important Note concerning relocation of files when upgrading from Version 4 to 5 or later.

### 2.5.3 Spectra Folder

The first time that you execute PERFORM-3D it creates a *Spectra* folder to hold response spectra for response spectrum analyses. Response spectra are stored in this folder. For the procedure see *Chapter 22, Response Spectrum Analysis, Section 22.1*.

For Version 4, if you installed the program in the default location, the *Spectra* folder is:

*C:\ProgramFiles\Computers and Structures\PERFORM\Spectra*

Starting with Version 5, the *Spectra* folder location depends on Windows, which can be a bit quirky, and is probably here:

For computers using Windows 7 or Vista:

*C:\Users\((UserNameAtInstallation))\My Documents\Computers and Structures\PERFORM*

For computers using Windows XP:

*C:\Documents and Settings\((UserNameAtInstallation))\My Documents\Computers and Structures\PERFORM*

where *((UserNameAtInstallation))* is the logged-on user at the time that PERFORM-3D was installed. For a computer that has several users, you may wish to set up a special user for the installation, and to log in as that user when running PERFORM-3D. See Section 2.2.1 for more information, including the Important Note concerning relocation of files when upgrading from Version 4 to 5 or later.

## 2.5.4 User Folder

The first time that you execute PERFORM-3D it creates a *User* folder. This folder is used for certain operations as follows.

- (1) For some analysis tasks you can save the results in one or more text files that you can then process using a spreadsheet program. The default location for these text files is the *User* folder.
- (2) To set up a new earthquake ground acceleration record you must have the record in a text file, and process this file using PERFORM-3D. For the procedure see *Chapter 15, Earthquake Records*. The process is a little simpler if you first copy the text file to the *User* folder.

For Versions 4, if you installed the program in the default location, the *User* folder is :

*C:\ProgramFiles\Computers and Structures\PERFORM\PERFORM-3D\User*

Starting with Version 5, the *User* folder location depends on Windows, which can be a bit quirky, and is probably here:

For computers using Windows 7 or Vista:

*C:\Users\((UserNameAtInstallation))\My Documents\Computers and Structures\PERFORM*

For computers using Windows XP:

*C:\Documents and Settings\(*UserNameAtInstallation*)\My Documents\Computers and Structures\PERFORM*

where (*UserNameAtInstallation*) is the logged-on user at the time that PERFORM-3D was installed. For a computer that has several users, you may wish to set up a special user for the installation, and to log in as that user when running PERFORM-3D. See Section 2.2.1 for more information, including the Important Note concerning relocation of files when upgrading from Version 4 to 5 or later.

## **2.6 Do Folder Locations Matter?**

---

For most users of PERFORM-3D, the locations of the standard folders are not important. The folders that have been relocated are used by PERFORM-3D, but most users will rarely need to read, write or copy any files to those folders.

You may, however, need to use the folders for the following reasons.

- (1) You may need to copy subfolders and files to the *Records* folder. Earthquake records can be added to this folder in two ways. The first is to use PERFORM-3D to import a record by processing a text file of the record. This allows you to check that the record is imported correctly. To do this you do not need to know the location of the *Records* folder. The second is to copy a set of records from one computer to another. This saves time by avoiding the import operation. To do this you will use Windows Explorer, and you must know the location of the *Records* folder.
- (2) You may need to do the same for the *RecordsF* and/or *Spectra* folders.
- (3) If you save a structure in the *Structures* folder (which is usually not recommended), you may want to open the ECHO.txt or M000.txt files (see Section 2.3), using a utility such as Notebook. To do this you must know the location of the *Structures* folder.



- (4) If you use PERFORM-3D to write analysis results to the *User* folder (as text files), you must know the location of this folder to use the files. The *User* folder is the default for writing such files. It is usually better to specify a different folder.
- (5) The user documentation files are installed in the *Manuals* folder.

Also, see Section 2.2.1 for an Important Note concerning relocation of files when upgrading from Version 4 to 5 or later.



## 3 Nodes

---

*A PERFORM-3D analysis model consists of nodes connected by elements. You can specify nodes and elements in any order (e.g., some nodes, then some elements, then more nodes, etc.) This chapter describes how to specify nodes and node-related data.*

### 3.1 Methods for Specifying Nodes

---

To specify nodes press the toolbar buttons for the **Modeling** phase and the **Nodes** task.

The axes of the 3D coordinate system are H1, H2 (both horizontal) and V (vertical upwards). These axes form a right-handed coordinate system. Nodes are identified by their coordinates. Nodes are not numbered.

To show or hide the node coordinates in a structure plot, press the "H,V" button on the toolbar. When the coordinates are shown, the order at each node, from top number to bottom number, is V, H2, H1.

If you show the coordinates, the structure plot can be crowded. As an alternative, to show the coordinates of a single node, right click on the node. To hide the coordinates. Left click anywhere, or right click another node.

To add, move and/or delete nodes, choose the **Nodes** tab in the data panel. You can then choose from several methods for specifying nodes.

For some methods you must select one or more existing nodes. There are two ways to do this, as follows.

- (1) Click on a single node in the graphics panel. The selected node changes color (to green or red, depending on the method).
- (2) Click, hold and drag a rectangle to enclose one or more nodes (a "box" or "fence" select). This may not work for some methods.

In some cases, clicking on a selected node deselects it. To clear all selected nodes, press the **Clear** button.

After you have entered the required data in the data panel, press the **Test** button to show the effect. Any additions or changes are shown in yellow. You then have the choice of pressing **OK** (which accepts the new or changed nodes) or **Undo**, which undoes the operation.

The available methods for specifying nodes are as follows.

- (1) *Create a regular grid of nodes.* Choose the **Grid** tab. It is usually convenient to begin with this method, then use other methods to manipulate the grid. You can create a 3D grid, but if you do you may initially get a jumble of nodes in the structure plot. If you create a 3D grid it is may be useful to view the structure in plan, so that the node layout makes sense. You should then set up a number of frames (see *Chapter 4, Frames*), so that when you define elements you can work in a frame-only view and show only the nodes in one frame. An alternative is to create a 2D grid, add elements to it, then use the **Duplicate** tab (see below) to duplicate that grid. You can also use the **Frames** task to duplicate nodes and elements.
- (2) *Move one or more nodes.* Choose the **Move** tab. There are three options for moving nodes, namely **Translate**, **Tilt** and **Rotate**. If you have already specified elements, the elements are also moved.
- (3) *Delete one or more nodes.* Choose the **Delete** tab. If you have already specified elements, any elements connected to the deleted nodes are deleted.
- (4) *Add new nodes one at a time.* Choose the **Single** tab. There are three options for adding single nodes, namely **Total H,V**, coordinates, **H,V Offsets** from an existing node, and **Polar Coords** using an existing node as the origin.
- (5) *Interpolate one or more nodes along a straight line between two existing nodes.* Choose the **Interpolate** tab. You can also extrapolate.
- (6) *Duplicate a line or block of nodes.* Choose the **Duplicate** tab. This operation duplicates nodes only. To duplicate both nodes and elements see the **Duplicate** operation in the **Frames** task.

## 3.2 Zooming

---

You can zoom in by pressing the zoom-in button on the toolbar. After pressing this button, click in the graphics panel, hold, and drag a rectangle covering the region you want to view. You can zoom in multiple times, to get progressively more detailed views.

To zoom out, press the zoom-out button on the toolbar. Pressing this button returns to the un-zoomed view.

## 3.3 Closely Spaced Nodes

---

You can not have nodes with identical coordinates. In the **Umbrella** task you can specify a minimum node spacing, with a default of 6 inches or 15 centimeters. If you attempt to specify a new node that is closer than this spacing to an existing node, in the H1, H2 or V direction, this is an error.

You can change the minimum spacing but it can not be zero. This means that you can not have zero length elements. If you make the minimum spacing too small, you may not be able to distinguish closely spaced nodes.

If you specify some nodes then change the minimum spacing, the new minimum applies only to any new or changed nodes.

## 3.4 Supports

---

To specify rigidly supported (restrained) nodes choose the **Supports** tab, set the support type, select nodes singly or using a box select, and use the **Test**, **Clear**, **OK** and **Undo** buttons in the usual way. You can add, change and delete supports at any time.

When you are in the **Supports** page, support symbols are always shown. When you are in other pages you can show or hide the support symbols by pressing the support symbol button in the toolbar.

You do not need to specify supports for nodal displacements that are completely unrestrained. For example, if your structure is a truss you do not have to restrain the rotational displacements. PERFORM-3D does this automatically. There will be a warning message in the ECHO file.

### 3.5 Masses

---

To specify masses choose the **Masses** tab. In the current version of PERFORM-3D you must specify all masses as nodal properties. There is no provision for element masses.

You are not restricted to a single mass distribution. Instead, you can specify up to 6 mass patterns, each identified by a name. For example, you might specify a mass pattern for dead load masses and a second pattern for live load masses. Then, in the **Analysis** phase, you can specify the masses for the analysis as a linear combination of the mass patterns (e.g., 1.0 times dead load mass plus 0.25 times live load mass). For a more detailed discussion see *Chapter 17, Analysis Series*.

To start a new mass pattern press the **New** button, specify the pattern name, then specify the masses, select nodes singly or using a box select, and use the **Test**, **Clear**, **OK** and **Undo** buttons in the usual way.

To change the masses for an existing pattern, choose the pattern name from the list and specify changes as needed. You can also delete any mass pattern.

If you have a rigid floor diaphragm (with a rigid floor constraint – see the next section) you can specify the floor mass in either of two ways, as follows.

- (1) Calculate the center of mass for the floor, place a node at this point (be sure to include it in the rigid floor constraint), and put the translational mass and rotational inertia for the entire floor at the point (be sure to include the rotational inertia). This can be the best method if you want to consider different mass locations, to account for "accidental" mass eccentricity. If you define a series of nodes that have masses but are not connected to any elements in the

structure, you can change the mass locations by moving these nodes.

- (2) Do not calculate the center of mass. Instead, specify a mass at each beam-column intersection in the floor, using tributary areas. When you specify a rigid floor constraint, PERFORM-3D will calculate the floor rotational inertia and the center of mass, and place the floor mass at this center. You probably do not need to calculate the tributary rotational inertias at the individual mass points, since in most cases they will be small. However, you can do so if you wish.

## 3.6 Slaving Constraints

---

### 3.6.1 General

Each constraint constitutes a Constraint Set, and must affect at least two nodes. As an example, if you have a three story frame and use rigid floor slaving, you will have three constraint sets, one for each floor or roof level. If you have four column lines along each of the H1 and H2 directions, each constraint set will affect 16 nodes.

To start a new constraint set press the **New** button, specify the constraint set name, and choose the constraint type. You can either add or delete nodes. Select the affected nodes, singly or using a box select, and use the **Test**, **Clear**, **OK** and **Undo** buttons in the usual way. The order in which you select the nodes does not matter. It is not necessary to identify a "master" node and a number of "slave" nodes.

### 3.6.2 Rigid Floor Constraint

In a rigid floor constraint set, all nodes must have the same V coordinate (i.e. the floor must be level). The constraint makes the H1 and H2 translations and the V rotation (rotation about the V axis) the same at all nodes in the set. The V translation and the H1 and H2 rotations are not affected.

Note that all of the beam elements in a rigid floor will have zero axial deformation. Hence if you use a typical beam model the beams will all have zero axial force. However, if you use a beam model that has fiber

sections or concrete type P-M-M hinges, the beam element may want to extend. Since you are restraining this extension, there may be compression forces in the beams.

You can not include a node in a rigid floor constraint if the node has an H1 or H2 translational support or a V rotational support. Conversely, you can not add an H1 or H2 translational support, or a V rotational support, to a node that is part of a rigid floor constraint.

### 3.6.3 Full Rigid Link Constraint

In a rigid link constraint set, a rigid link is added that connects all of the nodes in the set. The H1, H2 and V translations and rotations are all affected. If you were to specify a rigid link constraint for the nodes at a floor level, the floor would be rigid both axially and in bending.

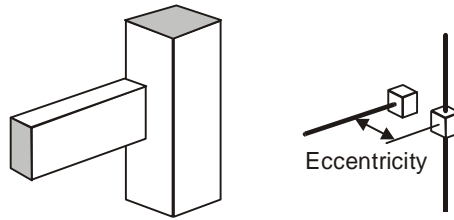
You can not include a node in a rigid link constraint if the node has a support of any kind. Correspondingly, you can not add a support to a node that is part of a rigid link constraint.

You will rarely need to use this constraint type. If you do, be careful with links that are not horizontal. In effect, the constraint links are rigid elements, and they will generally have axial forces, bending moments and shear forces. These forces are not calculated. If a rigid link is vertical or has a vertical component, and if it has an axial force, it may exert a significant P- $\Delta$  effect. This effect is NOT accounted for in the analysis, and it could introduce significant errors.

### 3.6.4 Eccentric Connection Constraint

In some frame structures, the beams may be connected eccentrically to the columns, for example as shown in Figure 3.1.

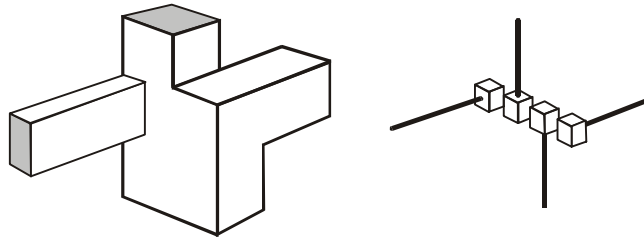




(a) Beam and Column      (b) Analysis Model

Figure 3.1 Eccentric Connection of Beam to Column

Figure 3.2 shows a more complex example, where both the beams and columns have eccentricities.



(a) Beams and Columns      (b) Analysis Model

Figure 3.2 More Complex Eccentric Connection

If you want to take this type of eccentricity into account, you can model the connection in three different ways as follows.

- (1) Connect the nodes by short, stiff elements. This requires that you specify the extra elements and element properties, and is usually not a very convenient method.
- (2) Connect the nodes using a rigid link constraint. You can use this constraint only if the nodes are not part of a rigid floor constraint, since any displacement can be a part of only one constraint. Rigid link and rigid floor constraints both constrain the H1 and H2 translations and the V rotation.

- (3) Connect the nodes using an eccentric connection constraint. This constraint is specifically intended for the case where the nodes are already part of a rigid floor constraint. It constrains the V translation and the H1 and H2 rotations. You should use this constraint only if the nodes are already part of a rigid floor constraint.

Note that you must have a separate constraint set for each eccentric connection.

### 3.6.5 Equal Displacement Constraint

With an equal displacement constraint you can specify that a number of nodes (usually only two nodes) have equal displacements for any of the H1, H2 and V translations and rotations.

As with all constraints, you should use this type carefully, otherwise you can introduce spurious external forces and violate the equilibrium requirements for the structure. If you were to impose equal V displacements on two nodes that have identical coordinates, this would introduce no equilibrium errors, for two reasons. First, since the H coordinates are equal, any vertical forces applied on the nodes by the constraint are equal and opposite in all respects. Second, since the V coordinates are equal, there are no P- $\Delta$  effects in the constraint. However, if the H coordinates of the two points are not the same, the forces applied on the nodes by the constraint are equal and opposite in magnitude and direction but not in location, so the constraint imposes an external moment on the structure. Also, if the V coordinates of the two points are not the same, and if you are considering P- $\Delta$  effects in the analysis, there can be implied P- $\Delta$  forces but these are not considered by PERFORM-3D.

In PERFORM-3D you can not have nodes with identical coordinates. If you have two points with identical coordinates, you must use nodes that are separated by a small distance. If you create two such nodes and separate them by the minimum allowed distance, it will usually be OK to impose equal displacements, since any error is small. In all other cases, be sure that you understand the physical significance of the constraint.

## 4 Frames

---

*The plotted geometry for a 3D structure can be complex. You can (and usually must) simplify the plot by showing only selected parts of the structure. A "frame" is any part of the complete structure. It can be a plane frame, a floor, or any other part of the structure that you would like to operate on separately from the complete structure.*

*You can quickly set up, modify and delete frames, and then toggle between a full structure view and a view showing only a single frame. This chapter explains the procedure.*

### 4.1 Plotting of Nodes and Elements

---

In PERFORM-3D, the analysis mode for a structure is a node-element model. A plot of the structure usually shows both nodes and elements. Nodes are shown as small squares and elements as lines (for 2-node elements) and quadrilaterals (for 4-node and 1-node elements). Elements are usually drawn shrunk (i.e., with gaps between the element ends or corners and the nodes). PERFORM-3D draws only a "wire frame" plot of a structure. You can not show the elements as solid bodies.

Plots of large structures can obviously have many lines for elements. To help clarify the plots, the elements are plotted in different colors. For example, elements must be divided into groups (see later). In a plot of the structure, the elements in the current group are shown light blue to make them stand out.

For relatively simple structures, the use of color combined with a perspective projection can be sufficient to make the plot clear. For complex structures, however, this is usually not enough. To simplify the plots when you work with complex structures you should make liberal use of "frames". A frame is any part of the complete structure. It can be a plane frame, a floor, an elevator core, or any other part of the structure that you would like to plot separately from the complete structure. You can quickly define, modify and delete frames, and then

toggle between a "structure" view that shows the complete structure and a "frame" view that shows only a single frame.

In a structure view, the elements in the current frame are highlighted to help them stand out. Any elements that are in both the current frame and the current element group are plotted light blue. In a frame-only view, only the nodes and elements in the current frame are plotted. You can add elements, specify loads, define structure sections, etc. in either a structure view or a frame view.

## 4.2 Defining a Frame

---

To define a new frame or modify an existing frame, press the toolbar buttons for the **Modeling** phase and the **Frames** task. The steps for adding a new frame are as follows.

- (1) Press the **New** button.
- (2) Input a name for the frame, then press **OK**.
- (3) Choose the **Add Nodes** tab.
- (4) In the graphics panel, select the nodes to be added to the frame, using either single node or box selection. If you are defining a vertical frame it is usually best to change to a plan view, so that you can select the nodes with a box select. If you are defining a horizontal floor it is usually best to change to an elevation view.
- (5) If the selected nodes are OK, press **OK** to add them to the frame. If not, press **Clear**. Any elements that are connected to nodes in the frame are also a part of the frame, including any elements that you specify later.
- (6) Repeat Steps (4) and (5) until you have selected all the desired nodes.

To delete nodes from a frame, choose the **Delete Nodes** tab. You can return to a previously defined frame at any time to add or delete nodes. To delete a complete frame press the delete (**X**) button next to the **New** button.

To toggle between structure and frame-only views, press the **S-F** (Structure-Frame) button. When you are in the **Frames** task, this button and the list of frames is in the data panel. After you exit this task, the list of frames and the **S-F** button is at the right end of the toolbar. You

can choose a frame from the list and toggle between structure and frame views at any time

### **4.3 Duplicating a Frame**

---

If you have a number of similar frames in a structure, you can save time by defining one frame then duplicating it. When you duplicate a frame you duplicate both the nodes and the elements in the frame (if you use the Duplicate option for nodes, you duplicate only the nodes).

If you have similar frames, you may duplicate a frame after you have specified the elements but before you have assigned properties to the elements. If you have frames that are more nearly identical, you can duplicate a frame after you have assigned properties to the elements.

When you duplicate a frame you can specify H1, H2 and/or V distances between the original frame and the duplicate, which means that the new frame is parallel to the original. If you also want the new frame to be rotated, you must first create it parallel to the original frame, then use the **Move** operation in the **Nodes** task to rotate the new frame.

To duplicate a frame, press the toolbar buttons for the **Modeling** phase and the **Frames** task, and choose a frame from the frame list. The steps for duplicating the frame are as follows.

- (1) Choose the **Duplicate** tab.
- (2) Specify the name for the duplicate frame.
- (3) Specify the H1, H2 and V translations from the current frame.
- (4) Press **Test** to show the new frame.
- (5) Press either **OK** or **Undo**.

In the current frame the elements will belong to element groups. Corresponding elements in the new frame belong to the same element groups.



## 5 Component Properties

---

*Elements are made up of components. A simple element, such as a truss bar, may consist of a single component. A more complex element, such as a column, can consist of a number of components of different types. Before you can assign properties to elements, you must first specify properties for components.*

*This is the most complex task for nonlinear modeling, and usually the task that will require most of your time. This chapter covers the essential concepts, and the procedures for specifying component properties. For more details, see the document "PERFORM Components and Elements".*

### 5.1 Elements and Components

---

PERFORM-3D has several element types (bar, beam, column, wall, seismic isolator, etc.). To specify the properties for an element you must first specify the properties of one or more components that make up the element.

There are several component types, as follows.

- (1) **Materials.** These include steel, concrete and shear materials.
- (2) **Cross sections.** These include beam, column and wall sections. The properties of some cross sections can depend on the properties of materials.
- (3) **Basic structural components.** These include bars, plastic hinges, connection panel zones, seismic isolators and many other components. The properties of some basic structural components can depend on the properties of cross sections and/or materials.

Basic components are divided into inelastic and elastic components. Inelastic components can yield and dissipate energy. Elastic components do not dissipate energy. Elastic components are mostly linear, but can be nonlinear (e.g., a gap-hook bar). Inelastic components are always nonlinear, and are much more

complex than elastic components. A fluid damper dissipates energy and is classified as inelastic, although its behavior is different from that of other inelastic components.

- (4) **Strength sections.** These are not structural components. Their purpose is to allow you to calculate strength demand-capacity ratios at interior points in beam and column elements.
- (5) **Compound components.** A compound component is made up of a number of cross section, basic and/or strength section components. For example, a compound component for a beam might consist of an elastic end zone component, a plastic hinge component, an elastic segment with a uniform cross section, a second plastic hinge, and a final end zone. It might also have shear strength sections at the plastic hinge locations, to check that the shear strength is not exceeded.

Some elements consist of one compound component. These include frame type elements (beam, column, diagonal brace), and also shear wall, general wall, buckling restrained brace and viscous bar elements. Other elements consist of one basic component.

## ***5.2 Procedure for Specifying Components***

---

### **5.2.1 Scope**

This section describes the basic steps for specifying components.

Nonlinear components can have complex properties, and the forms for input of component properties may appear to be complex. You do not, however, have to specify values for all possible properties. For example, you may choose to specify elastic-perfectly-plastic behavior rather than the more complex trilinear behavior; you may not need to consider brittle strength loss; and unless you are an advanced user you will not need to specify upper/lower bounds. It is usually a good idea to start with relatively simple component properties, and to add more complex properties later, as the analysis proceeds.

If you are a new user, you might like to experiment with the component properties task before starting a complete structure. The forms for input



of component properties contain a lot of information, so be sure to look at them carefully.

For more information on component properties see the following sections.

- (1) If you are not familiar with the PERFORM action -deformation (F-D) relationship, see the brief description in Section 5.5. For more detailed information see the document "*PERFORM Components and Elements*".
- (2) For brittle strength loss, see Sections 5.6, 5.7 and 5.8.
- (3) For deformation and strength capacities, and an overview how they are used for limit states and usage ratios, see Section 5.9.
- (4) For cyclic degradation and hysteresis loops see Sections 5.10, 5.11 and 5.12.
- (5) For the role of cross sections and how to use them see Sections 5.13 and 5.14.
- (6) For "auto" end zone and connection panel zone components, see Sections 5.15.
- (7) For upper and lower bounds on strength and stiffness see Section 5.16.

## 5.2.2 Component Types and Names

To specify or change component properties, press the toolbar buttons for the **Modeling** phase and the **Components** task.

You can choose from the following component types : **Inelastic** (inelastic basic components), **Elastic** (elastic basic components), **Cross Sects** (cross sections), **Material** (elastic and inelastic materials) **Strength Sects** (strength sections) and **Compound** (compound components).

For each component that you define, you must assign an identifying name (max. 40 characters). For a large structure you can have many components, including compound components that consist of several lower level components. Be sure to plan component names carefully, so that you can easily identify them.

See Section 5.3 for some methods of managing component properties.

### 5.2.3 Materials

To specify a material the steps are the same as for a basic structural component, as described later.

### 5.2.4 Beam and Column Cross Sections

To specify a beam or column cross section the steps are as follows.

- (1) Choose the **Cross Sects** tab.
- (2) Choose the cross section type from the **Type** list. Take care whether you choose a beam type or column type section. You can use either type of cross section in a frame compound component, but you must use beam sections for beam type plastic hinge components, and column sections for column type hinge components.
- (3) Press the **New** button to start a new cross section.
- (4) Enter the component name and choose the units, then press **OK**. Plan the names carefully (although you can change the name later if you wish). Be sure to choose the correct units.
- (5) Choose the appropriate Symmetry option (symmetric or non-symmetric) if it is available.
- (6) Enter the section properties. The Section Stiffness properties are required (they are obtained automatically for a standard section). The properties in the **Inelastic Strength** tab are optional. If you specify some or all of these properties, they are used when you associate the cross section with an inelastic basic component such as a plastic hinge. The properties in the **Elastic Strength** tab are also optional. If you specify some or all of these properties, they are used if you associate the cross section with a strength section.
- (7) Press the **Check** button to check the data. You will be told if there are any errors. Keep an eye on the Status box.
- (8) When the data is correct press the **Save** button to save the data. Again, keep an eye on the Status box.
- (9) To cancel any changes press the **UnChange** button. This restores the last saved data.

To review or change an existing cross section, choose the section type from the **Type** list and the name from the **Name** list. If you make changes, keep an eye on the **Status** box.

You can press the **Save As** button to save the current cross section as a new section with a different name. It can be a time saver to do this if you have cross sections that are similar. Set up and save one section, then use **Save As** to initialize each similar section, and make just the needed changes. Be sure to press **Save** after you have made the changes.

You can press the **Delete** button to delete the current cross section provided that you have not yet saved it. Do this if you have started a new section and find that you do not want to complete it.

You can delete a saved cross section by pressing the delete (**X**) button (next to the **New** button). If a cross section is used by a basic component or a compound component, you will not be able to delete it.

You can change the name of a saved cross section by pressing the **Rename** button.

## 5.2.5 Fiber Cross Sections for Walls

Cross sections for shear walls and general walls are always fiber sections. There are four cross sections of this type, as follows, all of them requiring similar data.

- (1) Shear wall, inelastic fiber section.
- (2) Shear wall, elastic fiber section.
- (3) General wall, inelastic fiber section.
- (4) General wall, elastic fiber section.

For each cross section there are two options for specifying the fibers, as follows.

- (1) **Fixed Size.** You must specify the area and coordinate location for each fiber in the cross section. The width of the cross section is fixed, based on the maximum and minimum fiber coordinates. When you assign this cross section to an element, you must make sure that the element width is consistent with the cross section width. With this type of section you can account for thickness variations in the wall. You can also have different reinforcement areas in different part of the section.

- (2) **Auto Size.** You must specify the wall thickness and the number of fibers. The width of the cross section is not fixed. When you assign the cross section to an element, PERFORM-3D makes the cross section width equal to the element width, and hence calculates the fiber areas and coordinates. With this type of section you can not vary the wall thickness over the element width, and the reinforcement percentage must be constant. You can, however, account for changes in thickness and additional reinforcement, using additional Concrete Strut and Steel Tie elements.

The steps in defining a fiber wall section are as follows.

- (1) Choose the **Cross Sects** tab.
- (2) Choose the cross section type from the **Type** list.
- (3) Press the **New** button to start a new cross section.
- (4) Enter the section name and choose the units. Plan the names carefully (although you can change the name later if you wish). Be sure to choose the correct units.
- (5) Choose the "Fixed Size" or "Auto Size" option.
- (6) Press **OK**.
- (7) Enter the data for structural fibers. Also enter the data for out-of-plane bending. Use the **Add**, **Insert**, **Replace** and **Delete** buttons to add or modify fibers.
- (8) If you wish, enter data for monitored fibers. This is usually a good idea, since it allows you to define limit states based on strain (for an inelastic section) or stress (for an elastic section). You can also monitor deformations of wall elements using separate Deformation Gage components (these are elastic basic components).
- (9) Press the **Check** button to check the data. You will be told of any errors. Keep an eye on the Status box.
- (10) When the data is correct press the **Save** button to save the data. Again, keep an eye on the Status box.
- (11) To cancel any changes press the **UnChange** button. This restores the last saved data.

To review or change an existing cross section, choose the section type from the **Type** list and the name from the **Name** list. If you make changes, keep an eye on the **Status** box.

You can press the **Save As** button at any time to save the current cross section as a new section with a different name. It can be a time saver to do this if you have cross sections that are similar. Set up and save one

section, then use **Save As** to initialize each similar section, and make just the needed changes. Be sure to press **Save** after you have made the changes.

You can press the **Delete** button to delete the current cross section provided that you have not yet saved it. Do this if you have started a new section and find that you do not want to complete it.

You can delete a saved cross section by pressing the delete (**X**) button (next to the **New** button). If a cross section is used by a shear wall or general wall compound component, you will not be able to delete it.

You can change the name of a saved cross section by pressing the **Rename** button.

## 5.2.6 Fiber Cross Sections for Frame Elements

There are two cross sections of this type, namely Inelastic Fiber Section for a Beam and Inelastic Fiber Section for a Column. The data is similar to that for a fiber wall section, with the main difference that for a column section you must specify fiber locations along both Axis 2 and Axis 3 of the cross section.

Please be careful using fiber column cross sections. You may be tempted to specify cross sections with large numbers of fibers. If you do, and if you use these cross sections in a large number of elements, it can take a lot of computer time to run the analysis. This cross section is intended mainly for applications such a bridge piers, where there are relatively few pier elements. It is not intended for typical columns in buildings, where there may be many column elements.

If you use this type of section, we recommend that you analyze single element examples to determine the smallest number of fibers that you can use and still get reasonable results. You are likely to find that you do not need a large number of fibers.

## 5.2.7 Basic Structural Components

To specify an inelastic or elastic basic structural component the steps are as follows. The steps for a material are the same, and the steps for a strength section are very similar.

- (1) Choose the **Inelastic** or **Elastic** tab.
- (2) Choose the component type from the **Type** list.
- (3) Press the **New** button to start a new component.
- (4) Enter the component name and choose the units, then press **OK**. Plan the names carefully (although you can change the name later if you wish). Be sure to choose the correct units.
- (5) Choose the options (symmetry, whether the component uses a cross section, whether there is strength loss, etc.). You can change these later if you wish.
- (6) Enter the component properties. You will usually have to fill in data on a number of tabbed pages in the form. Diagrams in the form explain the required data. If you choose "Yes" for the "Use Cross Section" option, you must select a cross section type and name from the lists. If you do this, the properties that are obtained from the cross section are shown green. You can not change these properties. If you have specified cross section strengths that can be used by the component, these strengths will be shown green. If you have not specified such cross section strengths, the strength text boxes will be white and you must enter the properties.
- (6) Press the **Check** button to check and (in most cases) plot the data so that you can review it graphically. You will be told of any errors. Keep an eye on the Status box.
- (7) When the data is correct press the **Save** button to save the data. Again, keep an eye on the Status box. After you have saved the data, you can plot hysteresis loops for most inelastic components. This can be valuable for checking the component behavior. See the next section for the procedure.
- (8) To cancel any changes press the **UnChange** button. This restores the last saved data.

To review or change an existing component, choose the component type from the **Type** list and the component name from the **Name** list. If you make changes, keep an eye on the **Status** box.

You can press the **Save As** button at any time to save the current component as a new component with a different name. It can be a time saver to do this if you have components that are similar. Set up and save one component, then use **Save As** to initialize each similar component, and make just the needed changes. Be sure to press **Save** after you have made the changes.

You can press the **Delete** button to delete the current component provided that you have not yet saved it. Do this if you have started a new component and find that you do not want to complete it.

You can delete a saved component by pressing the delete (**X**) button (next to the **New** button). If a basic component is used in a compound component, you will not be able to delete it. If a basic component is assigned directly to an element, you will be able to delete the component, but you will need to assign a new component to the element.

You can change the name of a saved component by pressing the **Rename** button.

### 5.2.8 Plot Hysteresis Loops

When you check the properties for an inelastic component or an inelastic material, PERFORM plots the force-deformation relationship. This provides a useful graphic view of the properties. However, the graph does not show what happens to a component when it is deformed cyclically. If you wish, you can plot hysteresis loops for cyclic deformations. This can give you a much better idea of how the component behaves.

The procedure is as follows.

- (1) If you are defining a new component or changing an existing one, specify the component properties in the usual way, and press the **Check** and **Save** buttons. If you are reviewing an existing component, just go to that component to display its properties.
- (2) Press the **Graph** button to draw the force-deformation relationship. When the relationship is plotted you will see a **Plot Loops** button (in the same location as the **Save** button). Press the **Plot Loops** button. Note that this button is present only when the force-deformation relationship is plotted. Some components do not have this button.
- (3) A form with spaces for deformation ranges will be shown. Enter as many deformation ranges as you wish, and check the ones that you want to plot.
- (4) Press **Plot** to plot the loops. Uncheck or check deformation ranges to add or delete them in the plot. Press **Print** to print the current loops, or **Save** to save the loops to a text file. Press **Close** to exit.

### 5.2.9 Strength Sections

To specify a strength section the steps are very similar to those for a basic structural component.

### 5.2.10 Frame Compound Components

Frame compound components are used for beams, columns, braces, and other elements of frame type. To specify a frame compound component the steps are as follows.

- (1) Choose the **Compound** tab and choose the Frame Compound Component type from the Type list. Figure 5.1 shows the form.
- (2) Press the **New** button to start a new component, and enter the component name in the usual way.
- (3) The compound component is made up of a number of basic components. Use the COMPONENT TO BE ADDED section to add components. For each basic component choose the type and name from the lists, choose the length type and specify the length value. Then position the basic component along the compound component using the **Add, Insert, Replace** and **Delete** buttons.
- (4) Do essentially the same for the strength sections, if any.
- (5) Press the **Check** button to check and plot the data so that you can review it graphically. In the COMPONENT LIST, if you click on the row for any basic component, the corresponding component in the graphical view will be shown yellow. This applies also for strength sections.
- (6) If you want to change a component, the best way is to double click on it in the COMPONENT LIST. Its properties will be set up in the COMPONENT TO BE ADDED section. Edit these properties, then press **Replace**. This sequence (double click, change the data, press **Replace**) can save time when you need to change a component in the list.
- (6) When the data is correct press the **Save** button. Keep an eye on the Status box.
- (7) To cancel any changes press the **UnChange** button. This restores the last saved data.

You can return to examine or change a compound component at any time. If you make changes, keep an eye on the Status box, and be sure to press **Save** to save the component.





For a shear wall compound component the overall steps are similar to those for a frame compound component, but the form is very different. For the primary bending behavior you must choose a fiber wall cross section. For shear you must choose a shear material and specify an effective wall thickness. For transverse behavior you must specify an effective thickness and an elastic modulus. For out-of-plane bending you must specify an effective plate bending thickness and a modulus.

### 5.2.12 General Wall Compound Components

Whereas a shear wall element has a primary axial-bending direction (usually the vertical direction), a general wall may not have a primary direction, and may be just as likely to have inelastic axial-bending behavior horizontally as vertically. Also, a general wall element can resist shear in two ways, namely by conventional shear ("concrete shear" in a general wall element) and by diagonal compression action ("diagonal shear").

A general wall element can bend out of plane. As with shear wall elements, this is a secondary mode of behavior, and is assumed to be elastic.

The steps and the form for a general wall component are similar to those for a shear wall component. However, you must choose two fiber cross sections, one each for the vertical direction and horizontal directions. You must also specify material and thicknesses for both concrete shear and diagonal shear. If you wish you can ignore one of the shear types by specifying zero thickness. For out-of-plane bending you must specify an effective plate bending thickness and a modulus.

The general wall compound element is a complex element. Before you sue it you should study its description in the document "*PERFORM Components and Elements*".

### 5.2.13 Viscous Bar and BRB Compound Components

The overall steps for viscous bar and BRB (buckling restrained brace) compound components are the similar to those for other compound components, and the form is self-explanatory. A viscous bar compound component uses fluid damper and elastic bar basic components. A BRB compound component uses BRB and elastic bar basic components.

## 5.3 Managing Component Properties

---

### 5.3.1 General

In a large structure there can be a lot of components. This section describes some tools that can be useful for managing the components and their properties.

### 5.3.2 Filter

In PERFORM, components are selected from drop-down lists. If there are many components of a given type, the drop-down list can be long, and finding a particular component can involve searching. Also, the components are listed in the order that they are defined, so that related components may not be adjacent to each other in the list. You can use the “filter” feature to shorten the list.

In the **Component Properties** task, below the list of component names there is a “Filter” button as shown in Figure 5.2.

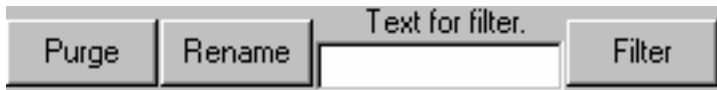


Figure 5.2 Purge, Rename and Filter Buttons

To filter the list of component names, type a text string into the “Text for filter” box, then press the **Filter** button. Only component names that include the text string will be listed. The text string is case sensitive.

To use this feature effectively you must use distinctive text strings for component names. If you wish you can press the **Rename** button to change component names.

### 5.3.3 Purge

A list of component names may contain components that are not used in any higher level components (such as a compound component) or in any elements. To purge such components from the list, press the **Purge** button (Figure 5.2).

### 5.3.4 Use Cross Section Strengths

If an inelastic basic component depends on a cross section, it is usually a good idea to specify the strength data with the cross section. The inelastic component then gets its basic strength from the cross section, and if you change the cross section the inelastic component strength also changes.

### 5.3.5 Save As

Rather than start each new component by pressing the **New** button, use the **Save As** button and edit the properties.

### 5.3.6 Copy-Paste

In the **Component Properties** task, most component property pages have **Copy** and **Paste** buttons as shown in Figure 5.3.



Figure 5.3 Copy-Paste Buttons

You can use these buttons to copy properties from one component to another, as follows.

- (1) Set up the properties for a component. Go to one of the pages that has Copy-Paste buttons.
- (2) Press the **Copy** button. The name of the component will be shown in the box between the buttons, as a reminder, and also to show that properties for that page can be pasted.
- (3) Choose another component of the same type, and go to the same page. Press the **Paste** button. All of the properties on that page from the copied component will be pasted into the current component.

You can press **Copy** in multiple pages if you wish, then **Paste** to multiple pages (one page at a time). The main restriction is that all of the properties on a page are copied. You can not copy selected properties.

## **5.4 Printed Reports**

---

To get a printed report for any component, press the button with the printer symbol. In the current version of PERFORM-3D you can print the properties for only the current component. If you want to print the properties for all components, you must print each one in turn.

The component properties are also included in the ECHO file.

## **5.5 F-D Relationship**

---

### **5.5.1 Actions and Deformations**

Each material and each basic structural component has one or more actions (or forces) and corresponding deformations. For example, for a simple material the action is stress and the deformation is strain, and for a simple plastic hinge the action is bending moment and the deformation is hinge rotation. The relationship between the two is the F-D relationship.

This section is concerned with the nonlinear F-D relationship for inelastic components, and the PERFORM-3D procedure for modeling this relationship. Elastic components are simpler, and do not need any general explanation.

### **5.5.2 Actual Components**

For the components in a real structure there will usually be a great deal of uncertainty about the actual behavior. In most cases the best we can do is capture those aspects of nonlinear behavior that have a substantial influence on the result of the analysis. Common sense suggests that if minor differences in the nonlinear behavior of the components have major effects on the analysis results, then the design is too sensitive, and we will have difficulty getting analysis results that are reliable enough for design purposes. Simplicity is also important. Structural design is a complex task, and it can be made impossibly complex if we have to use elaborate analysis models.

### 5.5.3 PERFORM F-D Relationship

Most of the inelastic components in PERFORM-3D have the same form for the F-D relationship. This is a trilinear relationship with optional strength loss, as shown in Figure 5.4.

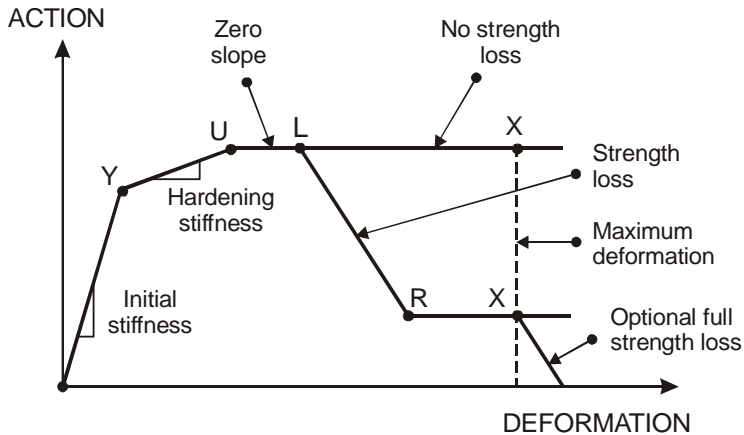


Figure 5.4 PERFORM Action-Deformation Relationship

The key points in the relationship are as follows.

- (1) **Y Point.** This is the first yield point, where significant nonlinear behavior begins.
- (2) **U Point.** This is the ultimate strength point, where the maximum strength is reached.
- (3) **L Point.** This is the ductile limit point, where significant strength loss begins.
- (4) **R Point.** This is the residual strength point, where the minimum residual strength is reached.
- (5) **X Point.** This is usually at a deformation that is so large that there is no point in continuing the analysis. You can continue an analysis beyond this point if you wish, but usually you will stop the analysis if any component is deformed beyond its X point. For some components you can specify that the strength reduces to zero at the X point. In this case the analysis does not stop.

For most components the relationships can be different for positive and negative deformations (i.e., the component can be unsymmetrical).

### 5.5.4 E-P-P and Bilinear Relationships

For many components an elastic-perfectly plastic (e-p-p) relationship, rather than a trilinear relationship may be adequate. In this case the Y and U points are the same.

To get a bilinear relationship with no strength loss, specify a large value for the deformation at the U point. To get a bilinear relationship with strength loss, specify a U point deformation that is slightly smaller than the L point deformation.

### 5.5.5 Additional Parallel Stiffness

Some components may continue to strain harden without reaching an ultimate load. PERFORM-3D allows this for some components, by allowing you to specify an additional parallel stiffness as shown in Figure 5.5.

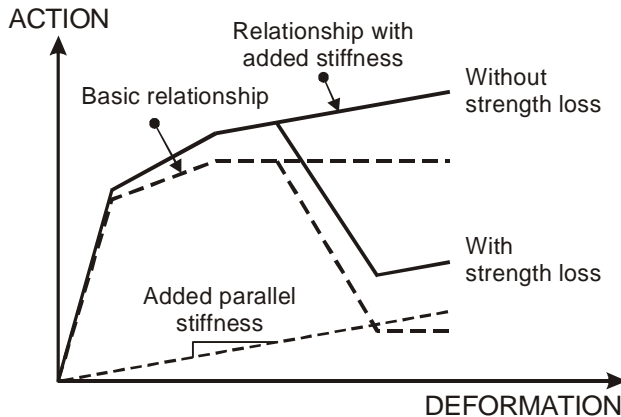


Figure 5.5 Additional Parallel Stiffness

Note that the parallel stiffness is added to all segments of the F-D relationship, including the initial elastic stiffness. You may have to adjust the initial elastic stiffness and the component actions to allow for this.

## 5.6 Strength Loss

### 5.6.1 General

In a structural component, "brittle" strength loss can be caused by a number of effects, including tensile fracture, concrete crushing, concrete shear failure, and buckling. When a component loses strength, the lost strength is redistributed to adjacent components (if this is possible), and the resulting behavior can be complex. The behavior can also be sensitive to small changes in the loads and/or the component properties.

In PERFORM, strength loss is optional. As a general rule, you should specify it only if it is essential.

Usually it is not permissible to deform an inelastic component beyond the L point (i.e., the deformation capacity is usually smaller than the L point deformation). For example, the FEMA 356 criteria generally allow deformation beyond the L point only for certain secondary members at the collapse prevention performance level. Hence, what happens after the L point is reached may be academic, and there may be no point in doing the calculations. For an exception to this rule, see Section 5.8.

Figure 5.6 shows the action-deformation relationships for FEMA 356 ( $Q-\Delta$  relationship) and PERFORM (F-D relationship).

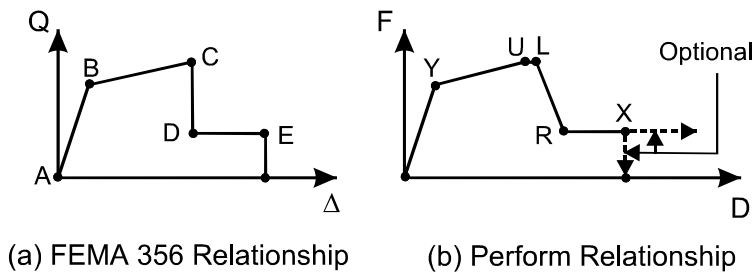


Figure 5.6 Strength Loss



In the FEMA 356 relationship there is sudden strength loss at Point C, and total strength loss at Point E. In the PERFORM relationship, strength loss begins at Point L and can be sudden or gradual. It is likely that strength loss in an actual structure will be gradual, and hence sudden strength loss is not realistic. As a general rule you should avoid specifying sudden strength loss. See the Section 5.7 for some problems that can occur when there is sudden or rapid strength loss.

In the FEMA 356 relationship there is total strength loss at Point E. In the PERFORM relationship total strength loss at Point X is optional, as explained in the next section.

### 5.6.2 Total Strength Loss at X Point

In the PERFORM relationship there is no automatic strength loss corresponding to FEMA 356 Point E. The PERFORM relationship has Point X, but this is usually intended to be a maximum deformation, beyond which the component is so badly deformed that there is no point in continuing the analysis. For some components, however, there is an option to reduce the strength to zero at this point. At the time of writing this is limited to the following components.

- (1) FEMA beam, steel type or concrete type.
- (2) Moment hinge.
- (3) Inelastic steel material (non-buckling).
- (4) Inelastic shear material.

For the FEMA beam and moment hinge components, the moment strength of the component reduces to zero at Point X. However, the shear strength is not affected. For these components the effect of total strength loss is the same as inserting a moment release.

If this option is used for an inelastic shear material (in a wall element), the shear strength and stiffness reduce to zero at Point X.

The procedure for specifying total strength loss is as follows.

Choose a component type, and go to the **Strength Loss** page. If the option for total strength loss is available for the component type, there will be a **Total Strength Loss at Point X** frame. To get total strength loss, choose the “Yes” option.

If you specify total strength loss at the X point, and if the component has an added parallel stiffness (see Figure 5.5), this parallel stiffness is still present. Hence, the strength does not actually reduce to zero.

### 5.6.3 Strength Loss Interaction

If a component loses strength in one direction, the strength in the other direction may or may not be affected. For example, if a concrete beam loses strength for negative bending because the concrete crushes, the strength for positive bending may not be affected. For other components this may not be the case. For example, if a component fractures in tension it may be best to assume that the strength in compression is also reduced.

You can account for this type of interaction by specifying a value for the **strength loss interaction factor**. A value of 0 means that strength loss in one direction causes no strength loss in the opposite direction. A value of 1 means that strength loss in one direction causes the same amount of strength loss in the opposite direction (if the positive and negative strength losses are different, strength loss in one direction causes the same proportional strength loss in the opposite direction). A value of 0.5 means that strength loss in one direction causes one half that loss (or one half of the corresponding proportion) in the opposite direction.

The procedure for specifying the interaction factor is as follows.

Choose a component type, and go to the **Strength Loss** page. If strength loss interaction is available for the component type, there will be a **Strength Loss Interaction** frame. Specify the interaction factor in this frame. The default factor is zero.

To see the effect of this factor, it can be useful to plot hysteresis loops, as described earlier in this chapter.

## 5.7 A Warning on Strength Loss

---

### 5.7.1 General

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

### 5.7.2 Components in Parallel

Figure 5.7(a) shows a simple structure consisting of an elastic component in parallel with a yielding component that loses strength. Figure 5.7(b) shows the load-displacement relationship for the structure.

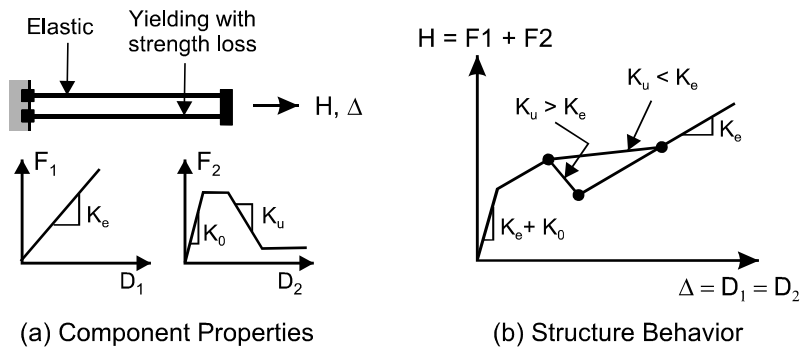


Figure 5.7 Effect of Strength Loss. Parallel Case.

Figure 5.7(b) shows the structure behavior for two cases, as follows.

- (1) If the strength loss is gradual, so that the stiffness  $K_u$  is numerically smaller than the elastic stiffness  $K_e$ , the structure load-displacement relationship increases monotonically.
- (2) However, if  $K_u$  is numerically larger than  $K_e$ , the structure strength decreases. If  $K_u$  is infinite, corresponding to sudden strength loss, the structure loses strength suddenly.

### 5.7.3 Components in Series

In Figure 5.7 the components are in parallel, so that when the yielding component loses strength it transfers load to the elastic component. Figure 5.8 shows a structure with the components in series rather than in parallel.

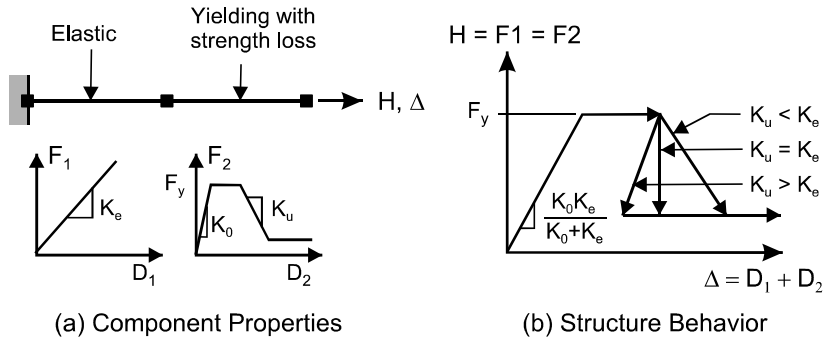


Figure 5.8 Effect of Strength Loss. Series Case.

In this case the structure load-displacement relationship always decreases as the yielding component loses strength. If  $K_u$  is numerically larger than  $K_e$ , the load-displacement relationship reverses, as shown in Figure 5.8(b). If the strength loss is sudden,  $K_u$  is infinite and the relationship reverses with a stiffness equal to  $K_e$ .

#### 5.7.4 Effect on Analysis Method

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

However, if reversal occurs, as in Figure 5.8(b) with  $K_u > K_e$ , the behavior can be very sensitive numerically. PERFORM is usually able to obtain a solution even for this type of behavior. However, this may not always be the case, and the analysis may fail to converge.

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

Sudden strength loss is probably not a good idea. It rarely occurs in real structural components, and as a general rule should be avoided.

Progressive strength loss as shown in Figure 5.9 is suggested as an alternative, with a strength at R equal to, say, 0.001 times the strength at U.

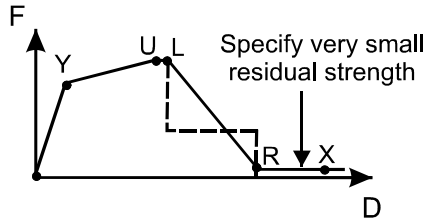


Figure 5.9 Suggested Alternative to Sudden Strength Loss

### 5.7.5 Hinge Strength Loss in a Compound Component

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

If a frame compound component consists of elastic beam segments in series with plastic hinge components that can lose strength, the hinges and beam segments act in series, and the moment-rotation relationship for the compound component could reverse as in Figure 5.8(b). This does not happen when FEMA Beam or FEMA Column components are used, because each FEMA component is automatically divided into a rigid-plastic hinge component and an elastic beam component, and the value of  $K_u$  for the hinge is chosen to give the specified  $K_u$  for the beam as a whole. This means that  $K_u$  for the hinge is numerically smaller than  $K_u$  for the beam as a whole. However, if you use plastic hinge components directly, rather than FEMA components, you must specify the  $K_u$  value for the hinge. If you make this value too large, the moment-rotation relationship for the beam as a whole can reverse. You should choose the value of  $K_u$  for the hinge so that this does not happen ( $K_u$  for the hinge should be numerically smaller than  $6EI/L$  for the beam as a whole).

If  $K_u$  is too large and the moment-rotation relationship for the beam as a whole reverses, PERFORM is usually able to obtain a solution. However, this may not always be the case, and the analysis may fail to

converge. Behavior of this type also tends to increase the energy balance error.

## 5.8 Column Ductile Limit Depends on Axial Force

### 5.8.1 Effect to be Modeled

If a column has a large axial force and is cycled inelastically, its ductile limit can be expected to be smaller than for the same column with a smaller axial force. This is illustrated in Figure 5.10.

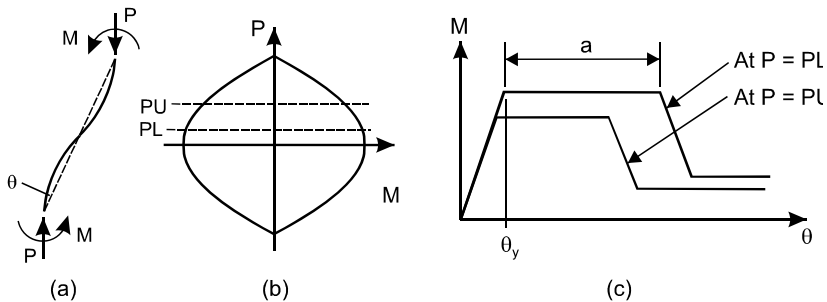


Figure 5.10 Effect of Axial Force on Ductile Limit

Figure 5.10(a) shows a steel column with equal end moments,  $M$ , and a chord rotation  $\theta$ . Figure 5.10(b) shows the P-M interaction surface, and two compression forces,  $PL$  and  $PU$ . Figure 5.10(c) shows an elastically-perfectly-plastic moment-rotation relationship with strength loss at the ductile limit. The yield moment is smaller at  $PU$  than at  $PL$ , because of the shape of the interaction surface. The ductile limit is smaller at  $PU$  than at  $PL$ , because the column is less ductile at the larger compression force.

As noted earlier, if the deformation capacity is smaller than the ductile limit, then there may be no need to model strength loss, since any column with strength loss will have a deformation  $D/C$  ratio larger than 1.0, and hence will not satisfy the performance requirement. However, FEMA 356 Section 3.4.3.2.1 states the following.

*“Primary and secondary component demands shall be within the acceptance criteria for secondary components at the selected Structural Performance Level.”*

This section applies to the Nonlinear Static Procedure (NSP), where demand/capacity ratios for component deformations are evaluated at the estimated displacement demand (target displacement).

The FEMA 356 commentary on this section states the following.

*“When all components are explicitly modeled with full backbone curves, the NSP (Nonlinear Static Procedure) can be used to evaluate the full contribution of all components to the lateral force resistance of the structure as they degrade to residual strength values. When degradation is explicitly evaluated in the NSP, components can be relied upon for lateral-force resistance out to the secondary component limits of response.”*

Some engineers interpret this to mean that the deformation capacities (acceptance criteria) for secondary members can be used for steel columns. These capacities are larger than for primary members, and also larger than the ductile limit deformation. If these capacities are used, a steel column can be deformed beyond its ductile limit, and hence strength loss must be modeled in the analysis, including the effect of axial force on the ductile limit.

### 5.8.2 FEMA 356 Requirements

FEMA 356 Table 5-6 gives values for the plastic rotation,  $a$ , in Figure 5.10(c). For a compact section the FEMA 356 equations are as follows.

$$\text{For } P/P_{CL} < 0.20, \quad a = 9\theta_y \quad (5.1a)$$

$$\text{For } 0.20 < P/P_{CL} < 0.50, \quad a = 11 \left( 1 - 1.7 \frac{P}{P_{CL}} \right) \theta_y \quad (5.1b)$$

where  $P$  = axial compression force,  $P_{CL}$  = compression strength of the column, and  $\theta_y$  = yield rotation. The yield rotation is given by

$$\theta_y = ZF_y \frac{L}{6EI} \left( 1 - \frac{P}{AF_y} \right) = \theta_{y0} \left( 1 - \frac{P}{AF_y} \right) \quad (5.2)$$

where  $F_y$  = material yield stress and  $Z$ ,  $A$ ,  $E$ ,  $I$  and  $L$  have their usual meanings, and  $\theta_{y0}$  = yield rotation when  $P = 0$ .

Note the following points.

- (1) In Equation 5.1b the axial strength is  $P_{CL}$ , which includes the effect of slenderness ratio and hence is a property of the member as a whole, whereas in Equation 5.2 the axial strength is  $AF_y$ , which is a property of the cross section only.
- (2) From Equations 5.1 and 5.2, the rotation at the ductile limit varies with axial compression force essentially as shown in Figure 5.11a. The ductile limit rotations in this figure are as follows, where the approximate values are calculated assuming that  $P_{CL} \approx AF_y$ .

$$\theta_0 = 9\theta_{y0} \quad (5.3a)$$

$$\theta_1 = 9\theta_{y0} \left( 1 - \frac{0.2P_{CL}}{AF_y} \right) \approx 7.2\theta_{y0} \quad (5.3b)$$

$$\theta_2 = 11\theta_{y0} \left( 1 - \frac{0.2P_{CL}}{AF_y} \right) \left( 1 - 1.7 \frac{0.2P_{CL}}{P_{CL}} \right) \approx 5.8\theta_{y0} \quad (5.3c)$$

$$\theta_3 = 11\theta_{y0} \left( 1 - \frac{0.5P_{CL}}{AF_y} \right) \left( 1 - 1.7 \frac{0.5P_{CL}}{P_{CL}} \right) \approx 0.8\theta_{y0} \quad (5.3d)$$



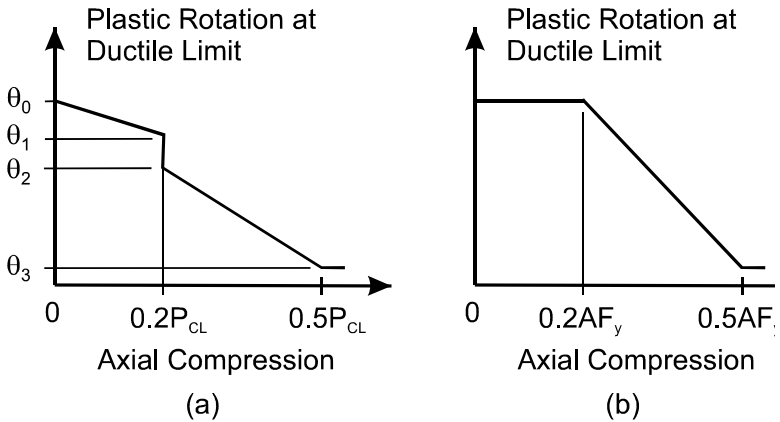


Figure 5.11 Rotation at Ductile Limit

This can be simplified with no practical loss of accuracy, as shown in Figure 5.11(b). In this figure the reduction in the ductile limit rotation is based on  $AF_y$  rather than  $P_{CL}$ , with effective length effects accounted for by considering P- $\Delta$  effects in the analysis. Also, all rotations are multiples of the yield rotation at zero axial force, which is constant, not a yield rotation that varies with the axial force.

PERFORM allows the ductile limit to vary only as shown in Figure 5.11(b). If FEMA 356 is used, the curve in Figure 5.11(a) must be approximated.

It may be noted that the FEMA 356 equations for rotation capacities in columns are similar to the above equations, and they can also be simplified. PERFORM allows the rotation capacities to vary with axial force, but using only the simplified form.

### 5.8.3 PERFORM Implementation of Variable Ductile Limit

In PERFORM, the effect of axial force on the ductile limit can currently be specified only for the following components.

- (1) FEMA Column, Steel Type.
- (2) PMM Hinge, Steel Rotation Type.
- (3) PMM Hinge, Steel Curvature Type.

In PERFORM, the equivalent of the FEMA 356 rotation “a” is the rotation, DL, at the ductile limit, Point L. This is different for a FEMA column component and a PMM hinge component as follows.

- (1) For a FEMA Column, Steel Type component the deformation D is the total end rotation expressed as a multiple of the yield rotation. Hence,  $DL = 1 + a/\theta_y$ .
- (2) For a PMM Hinge, Steel Rotation Type component the deformation D is the hinge plastic rotation. Hence  $DL = a$ .
- (3) For a PMM Hinge, Steel Curvature Type component the deformation D is the total curvature. Hence  $DL = \psi_y + a$ , where

$$\psi_y = \frac{ZF_y}{EI} \left( 1 - \frac{P}{AF_y} \right) \quad (5.4)$$

During each analysis (static or dynamic), the rotation and axial force for each component are monitored at each step. Based on the axial force, the ductile limit rotation, DL, is calculated. If the rotation demand exceeds DL, the ductile limit has been reached, and the component begins to lose strength. After this point, the rotation DL is “frozen” (kept constant), and monitoring of the axial force ceases.

#### 5.8.4 Procedure

The procedure for specifying the component properties is the same for FEMA Column, Steel Type components and P-M2-M3 Hinge, Steel Type components.

In the **Strength Loss** page you have the option of specifying that strength loss depends or does not depend on the axial compression force. If you choose the “No” option the strength loss properties are fixed, and you specify only one set of properties. If you choose “Yes” you must specify two sets of properties, one set at an upper compression force, PU, and a second set at a lower compression force, PL. Typically, PU will be  $0.5P_{CL}$  (or  $0.5AF_y$  in the simplified form) and PL will be  $0.2P_{CL}$  (or  $0.2AF_y$ ). For axial forces between PU and PL, PERFORM interpolates linearly.

As noted in the preceding section, at each step of an analysis (static or dynamic), PERFORM calculates the DL value based on the current axial force, until DL is reached. When DL is reached, the strength loss properties are “frozen”, and do not change in later analysis steps.

## **5.9 Deformation and Strength Capacities**

---

### 5.9.1 General

PERFORM-3D is a tool for both nonlinear analysis and performance based structural design.

To use the performance based design capabilities, you must specify deformation capacities for inelastic components and/or strength capacities for elastic components. You can then define limit states to make use of these capacities, and process the analysis results to give demand-capacity ratios (or limit state usage ratios). This section gives an overview of the procedure.

### 5.9.2 Demand and Capacity

Structural design decisions are made by comparing demand and capacity. To do this we need (a) demand-capacity measures (or engineering demand parameters), (b) demand values and (c) capacity values.

Demand-capacity measures can be classified as deformation measures (e.g., plastic hinge rotation) and strength measures (e.g., bending moment).

Strength measures have been used for many years, and methods for obtaining demands (usually linear structural analysis) and capacities (usually design code equations) are well established.

Some deformation measures, such as drift and deflection, have also been used for many years. However, engineers have less experience with deformation measures such as plastic hinge rotation. On the demand side, to obtain rotation demands for a plastic hinge we must use nonlinear structural analysis. On the capacity side, we do not yet have definitive design code equations for rotation capacities (although

FEMA 356 will become a design code). Hence, the calculation of deformation demand-capacity ratios for deformation-based design is more complex, and requires greater judgment than the calculation of strength demand-capacity ratios for conventional strength-based design.

PERFORM-3D has extensive capabilities for calculating demand-capacity ratios, considering both deformation and strength. You can save a lot of time if you make effective use of these capabilities.

### 5.9.3 Deformation Capacities

If a component is allowed to be inelastic, its structural adequacy is determined by its ductility rather than its strength. For all inelastic components in PERFORM-3D you can specify deformation capacities for up to 5 performance levels, identified as Level 1 through Level 5. Often there will be 3 levels, with Level 1 corresponding to Immediate Occupancy, Level 2 to Life Safety, and Level 3 to Collapse Prevention. However, this is not required.

Most components have constant deformation capacities, but for some components the capacity can be affected by interaction. For example, in a reinforced concrete beam a large shear force can reduce the bending ductility. Hence it may be necessary to specify the rotation capacity of a plastic hinge component as a function of the shear force at the hinge. In a number of components, PERFORM-3D allows you to account for interaction of this type when you specify capacities.

### 5.9.4 Drift Capacities

If you specify deformation capacities at the component level you can get detailed information about the deformation distribution in the structure, and hence identify "hot spots" that have the largest demand-capacity ratios. You may decide, however, that this is not needed, and that you can assure satisfactory performance by considering only structure drifts, not individual element deformations. You can do this by specifying one or more drifts, and assigning drift capacities (use the **Drifts and Deflections** task to define drifts, and the **Limit States** task to specify drift capacities). Most often you will use both drifts and component deformations as demand-capacity measures.

### 5.9.5 Strength Capacities

If a component is required to remain essentially elastic, its structural adequacy is determined by its strength. If the strength demand exceeds the strength capacity, then the component does not remain essentially elastic.

For most elastic components in PERFORM-3D you can specify strength capacities. You can then calculate strength demand-capacity ratios for any elements that use these components.

The procedure is different for Frame Compound Components. For these components, strength is checked using Strength Section components. In a compound component for, say, a beam element, you can place strength sections for bending and/or shear at a number of points along the element. For example, you might assume that plastic hinges form in the beam only at the column faces, and that the beam remains essentially elastic in shear. Hence, the only inelastic structural components would be plastic hinge components, placed at the column faces. You could then place shear strength sections at the column faces, and one or more moment strength sections near midspan. If the demand-capacity ratios for the strength sections are less than one, the behavior assumptions for the beam are correct. If not, there may be shear failure near the beam ends or moment hinging near midspan.

As with deformation capacities, you can specify strength capacity values for up to 5 performance levels. However, these values are now capacity factors. You can treat them as capacity reduction factors applied to a nominal strength ( $\phi$  factors in conventional design) or as factors to account for ductility ( $m$  factors in FEMA 356). You first specify a nominal strength, then specify up to 5 capacity factors, corresponding to different performance levels.

### 5.9.6 Overview of Limit States and Usage Ratios

If you have a large structure you can have a large number of component capacities and hence a large number of demand-capacity ratios. Limit states allow you to organize the demand-capacity ratios into manageable packages.

For example, you might create a structure with a number of drifts, a number of plastic hinges with rotation capacities for three performance

levels, and a number of shear strength sections with strength capacities for one performance level. For this structure you might set up the following limit states.

- (1) Drift limit state for Immediate Occupancy (IO), considering all drifts.
- (2) Drift limit state for Life Safety (LS), considering all drifts.
- (3) Drift limit state for Collapse Prevention (CP), considering all drifts.
- (4) Hinge rotation limits state for IO, considering all beams and using the Level 1 deformation capacities.
- (5) Hinge rotation limits state for LS, considering all beams and using the Level 2 deformation capacities.
- (6) Hinge rotation limits state for CP, considering all beams and using the Level 3 deformation capacities.
- (7) Hinge rotation limits state for IO, considering all columns and using the Level 1 deformation capacities.
- (8) Hinge rotation limits state for LS, considering all columns and using the Level 2 deformation capacities.
- (9) Hinge rotation limits state for CP, considering all columns and using the Level 3 deformation capacities.
- (10) Shear strength limit state, considering all beams.

During each structural analysis, PERFORM-3D calculates all needed demand-capacity ratios, and expresses the results as usage ratios for the limit states. For example, for Limit State (4) in the above list, PERFORM-3D calculates the demand-capacity ratios for all plastic hinge components in all beams, and saves the maximum value at each step of the analysis. This maximum value is the usage ratio for the limit state.

For details on setting up Limit States see *Chapter 10, Limit States and Usage Ratios*.

### 5.9.7 Note on Maximum Demand-Capacity Ratio

In the Analysis phase, the maximum value for any demand-capacity ratio is 30. In the unlikely event that a calculated ratio exceeds 30, a value of 30 is used. If you are not sure what to specify for a deformation or strength capacity, you may be tempted to specify an arbitrary value and to see what happens. If you specify too small a value, you may hit the 30 limit, and you will not get useful information. To avoid unrealistic demand-capacity ratios, you should take care to

specify reasonably realistic values for deformation and strength capacities.

## ***5.10 Beam or Column Shear Strength Depends on Hinge Rotation***

---

### 5.10.1 General

In a reinforced concrete beam or column, the shear strength is usually smaller in plastic hinge regions than outside those regions. If the behavior in shear is required to remain essentially elastic, the shear strength can be checked using a shear force strength section.

Usually the shear strength will be a fixed value. However, a shear force strength section can be linked to an adjacent flexural plastic hinge component, and the shear strength can then vary, depending on the rotation of the hinge.

This feature can be used for Shear Force Strength Section and V2-V3 Shear Strength Section components. It does not apply to inelastic shear hinges.

### 5.10.2 PERFORM Implementation

To use this feature, properties must be specified in two places, as follows. Details are considered in the next section.

- (1) In the strength section properties. See the **Rotation Effect** page. The properties on this page specify how the shear strength of the strength section component is affected by hinge rotation. The specific hinge is not considered at this stage, only the effect of rotation.
- (2) In the properties of any Frame Member Compound Component that uses the strength section. See the **Strength Sections** page. When you use a shear strength section, you can specify which rotational hinge component affects its shear strength. The strength section and the hinge will usually have the same location, but PERFORM does not check this.

At each step in an analysis, PERFORM checks whether a shear strength section is affected by the rotation of a hinge component. If so, PERFORM gets the maximum rotation of the hinge up to the current step (positive or negative rotation, whichever is larger), uses this rotation to calculate the strength capacity of the shear strength section, and hence calculates the shear strength Demand/Capacity ratio for the strength section.

### 5.10.3 Procedure for Strength Section Properties

Consider a V2-V3 Shear Strength Section, for a column. A Shear Force Strength Section, for a beam, is similar.

The steps for specifying the strength section properties are as follows.

- (1) In the **Component Properties** task go to the **Strength Sections** page and the **V2-V3 Shear Strength Section** or **Shear Force Strength Section** type. Either start a new strength section or choose an existing one.
- (2) Go to the **Rotation Effect** page.
- (3) Choose the Option for Rotation Effect. The shear strength of a concrete section will usually have contributions from the concrete and the shear reinforcement. Choose whether hinge rotation affects only the concrete contribution (which is usually the case) or the total shear strength.
- (4) Figure 5.12 shows how the shear strength varies with hinge rotation. Specify the rotations  $r_0$ ,  $r_1$  and  $r_2$  (in radians) and the strength reduction factors  $s_1$  and  $s_2$ .
- (5) Specify the Axis 2 Rotation Factor. The default is 1.0, which means that the effective rotation of a P-M-M hinge in a column is the resultant rotation. If a given rotation about Axis 2 of a column causes a smaller shear strength reduction than the same rotation about Axis 3, specify an Axis 2 Rotation Factor larger than 1.0. The effective rotation is the resultant of (a) the Axis 3 rotation and (b) the Axis 2 rotation divided by the Rotation Factor. If an Axis 2 rotation causes a larger shear strength reduction, specify a Rotation Factor smaller than 1.0. This factor is not needed for a shear strength section in a beam.
- (6) Save the properties in the usual way.



The above steps and Figure 5.12 refer to hinge rotation. As noted in the next section, the rotation is not limited to the rotation of a plastic hinge component.

Shear Strength Reduction Factor

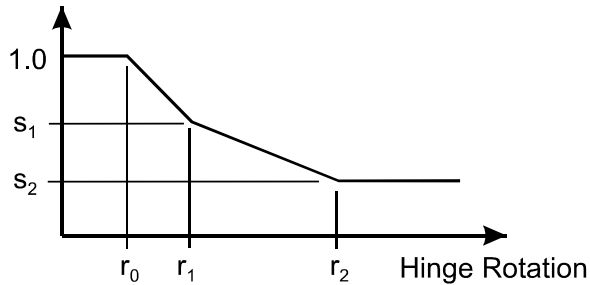


Figure 5.12 Effect of Rotation on Shear Strength

#### 5.10.4 Procedure for Frame Compound Component

The steps for associating a shear strength section with a hinge or other rotational component are as follows.

- (1) In the **Component Properties** task go to the **Compound** page and the **Frame Member Compound Component** type. Either start a new component or choose an existing one.
- (2) Specify the basic components in the usual way. Some of these components will be of rotational type. These can include FEMA beams, FEMA columns, steel or concrete moment hinges, steel or concrete PMM hinges, and also fiber beam or column segments. Any of these can be associated with shear strength sections, not just hinge components.
- (3) Go to the **Strength Sections** page. If you add a shear strength section, in addition to the usual name and location properties, you can specify a range of basic components for calculating rotation. For example, if the second basic component is a PMM hinge, and if you specify a range from 2 to 2, the rotation of the hinge will be used. The shear strength section in this case should be at or close to the PMM hinge component, but PERFORM does not check this. As a second example, if the second and third components are fiber segments, and if you specify a range from 2 to 3, the sum of the rotations across the segments will be used. You can specify any

range, covering components of any type or types. The rotation is the sum of the rotations over all of the components in the range.

Be sure that you define one or more Strength Limit States that use the shear strength sections.

If you specify a compound component and later edit it by changing its basic components, be sure to check that the rotational component ranges for any strength sections are still correct, since the rotational components are identified by sequence number, and these numbers may have changed.

## **5.11 Cyclic Degradation**

---

### **5.11.1 Backbone Relationship**

Under cyclic loading, most structural components will undergo stiffness and strength degradation. Note that strength degradation caused by cyclic loading is different from "brittle" strength loss at the ductile limit.

As a general rule the degradation will be progressive. In effect, a component will have an initial force-deformation relationship that applies for the first loading cycle, or for "monotonic" loading where there is no cycling. As the number of cycle increases, the effective force-deformation relationship will degrade, with smaller stiffness, strength, energy dissipation and/or ductility.

Ideally the force-deformation relationship would account for this progressive degradation, and it is possible that this will be done in future nonlinear analyses. At present, however, it is accepted practice to use a single force-deformation relationship that accounts implicitly for cyclic loading, assuming a certain amount of cycling. This is usually referred to as the "backbone" relationship.

The F-D relationship for analyses in PERFORM is the backbone relationship. FEMA 356 suggests backbone relationships for components of a variety of types, and also provides guidelines for constructing this relationship from experimental results (see FEMA 356 Section 2.8).

The backbone relationship accounts implicitly for cyclic degradation on the effective strength and ductility. It does not, however, account for energy degradation in the hysteresis loop. This is considered in the next section.

### 5.11.2 Hysteresis Loops

Figure 5.13 shows two hysteresis loops. In one loop the stiffness degrades and in the other loop it does not. The amount of energy that is dissipated (the area of the loop) is smaller for the degraded loop. The amount of energy degradation is related to the amount of stiffness degradation.

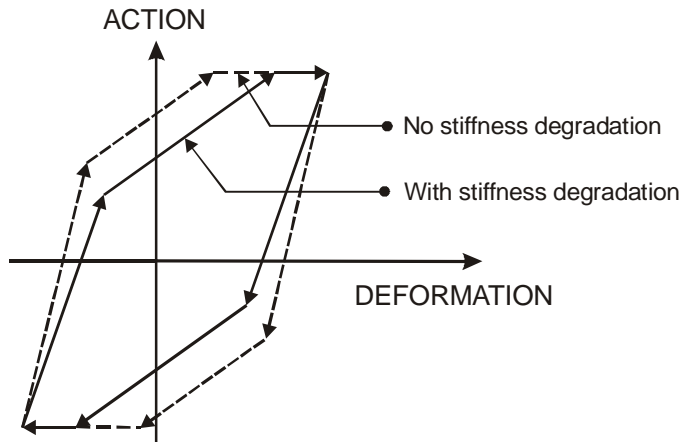


Figure 5.13 Hysteresis Loop With Stiffness Degradation

Since the seismic response of a structure can be sensitive to the amount of energy dissipation, it may be important to account for stiffness degradation in the hysteresis loop. Different methods are needed for dynamic analysis and static push-over analysis, as considered in the following sections.

For more details see the document "*PERFORM Components and Elements*".

### 5.11.3 Dynamic Analysis

For step-by-step dynamic analysis, if stiffness and energy degradation is important it must be accounted for directly, by changing the shape of the hysteresis loop as indicated in Figure 5.13.

In PERFORM-3D you can do this by specifying Energy Degradation Factors for inelastic components. In Figure 5.13 the Energy Degradation Factor is the ratio between the area of the degraded hysteresis loop and the area and the non-degraded loop. For a typical component, this area ratio is 1.0 for small deformation cycles (no degradation) and gets progressively smaller as the maximum deformation increases (increasing degradation).

In PERFORM, energy degradation is an option. If you choose "yes" for this option you must specify the relationship between the maximum deformation of the component and the energy degradation factor. During an analysis, for each inelastic component and at each time there is an unloading event for that component (i.e., each time a new hysteresis loop begins) PERFORM determines the maximum deformation, considering all cycles up to that point, not just the current cycle. PERFORM then calculates the energy degradation factor for that deformation, and adjusts the component stiffnesses to give this area ratio for the new loop.

A good way to see the effect of degradation is to create a component with energy degradation factors, and use the Plot Loops feature to draw hysteresis loops. You can also do this if you are using component properties that are based on experimental results, and you want to calibrate the degrading behavior.

### 5.11.4 Procedure for Degradation Factors

Use the **Component Properties** task and the **Inelastic** component page. Choose a component type, and go to the **Cyclic Degradation** page.

You have two options for specifying the energy degradation factors, as follows.

- (1) If the component F-D relationship is trilinear with strength loss you can specify the degradation factor at each of the Y, U, L, R

and X points. This is the "YULRX" option. You can also use this option for e-p-p behavior with strength loss (specify the same factor for the Y and U points).

- (2) You can specify degradation factors at the Y point, the X point, and at 3 intermediate points. This is the "YX+3" option. If there is no strength loss, this is the only available option.

### 5.11.5 Push-Over Analysis

In a static push-over analysis there are no cyclic loads or hysteresis loops, so energy degradation must be accounted for indirectly. In most cases you will use the degradation method that is built into the push-over method. For an explanation of push-over methods, including how they account for energy degradation, see *Chapter 28, General Push-Over Plots*.

An alternative, which can be used only with the Capacity Spectrum push-over method, is to use the PERFORM energy degradation factors. If you choose this alternative, PERFORM uses the following procedure.

- (1) At each step of the push-over analysis, and for each inelastic component, get the maximum deformation experienced by the component, and hence calculate the energy degradation factor.
- (2) For each component estimate the dissipated energy for the non-degraded loop. Note that the area of this loop is estimated, not explicitly calculated, since there is no actual loop. Given the current deformation for the component, PERFORM assumes a loop that has the same deformation in the opposite direction.
- (3) For each component, multiply the non-degraded energy by the degradation factor.
- (4) Sum these dissipated energies over all components.
- (5) Calculate the effective push-over strain energy in the structure. If the push-over forces are  $\{H\}$  and the corresponding nodal displacements are  $\{\Delta\}$ , this energy is  $0.5\{H\}^T\{\Delta\}$ .
- (6) Given the dissipated and strain energies, calculate the equivalent damping ratio = (dissipated energy)/(4 $\pi$  (strain energy)). This equation assumes that dissipated energy from an inelastic hysteresis loop has the same damping effect as the same amount of energy dissipated by viscous damping.

- (7) Use this damping ratio to calculate the spectral acceleration demand in the push-over analysis.

Since this method takes into account the properties of each specific structure, it may have an advantage over the "one size fits all" formulas in the capacity spectrum method.

## **5.12 Control of Hysteresis Loop Shape**

---

### **5.12.1 Alternative Loops**

Given an energy degradation factor, the unloading and reloading stiffnesses in a hysteresis loop can be chosen to give the required area ratio. There are, however, many ways in which the stiffnesses can be adjusted. PERFORM gives you some control over the way that this is done.

Figure 5.14 shows possible hysteresis loops for a component with a bilinear force-deformation relationship, all with the same amount of energy degradation.

In this figure the loop with no energy degradation is shown dashed, and three alternatives are shown for degraded loops. The degraded loops all have the same area, and hence the same ratio of degraded to non-degraded energy (about 0.55 in the figure, where 1.0 = no degradation). However, the loops have very different shapes. The differences are as follows,

- (1) In Loop A the unloading stiffness is the same as in the non-degrading case. The unloading elastic range is reduced to provide the required energy degradation.
- (2) In Loop B the elastic range on unloading is a maximum. The unloading stiffness is reduced to provide the required energy degradation.
- (3) Loop C is midway between Loops A and B.

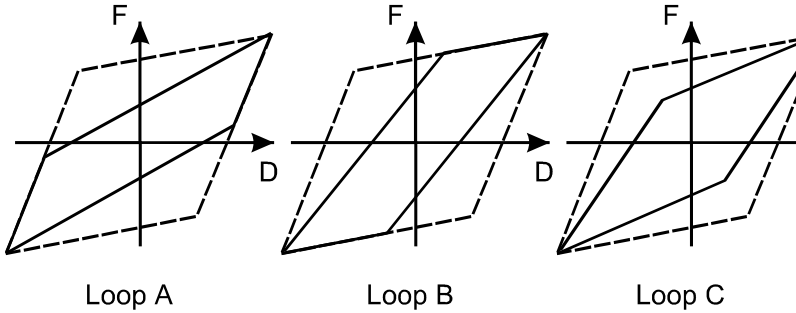


Figure 5.14 Loops with Same Energy Degradation

In PERFORM, the default option corresponds to Loop C. If you wish, you can specify an "unloading stiffness factor" that controls the loop shape. A value of zero for this factor (the default) corresponds to Loop C, a value of +1 corresponds to Loop A (maximum stiffness, smallest elastic range), and a value of -1 corresponds to Loop B (minimum stiffness, largest elastic range).

### 5.12.2 E-P-P Relationship

Figure 5.14 shows hysteresis loops for the bilinear case. These loops do not apply for an elastic-perfectly-plastic F-D relationship. For an e-p-p relationship the hysteresis loop degrades as shown in Figure 5.15.

This is the same as Loop B in Figure 5.14. The Loop A or Loop C type of relationship can not be obtained for a truly e-p-p component. If you specify a truly e-p-p F-D relationship, the unloading stiffness factor is not used.

You can, however, get loops of Types A and C if you specify a trilinear F-D relationship that is nearly e-p-p, with a small hardening stiffness.

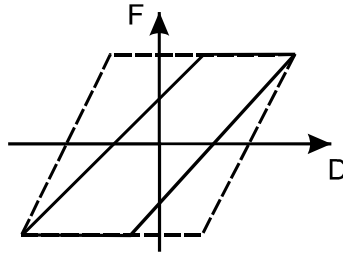
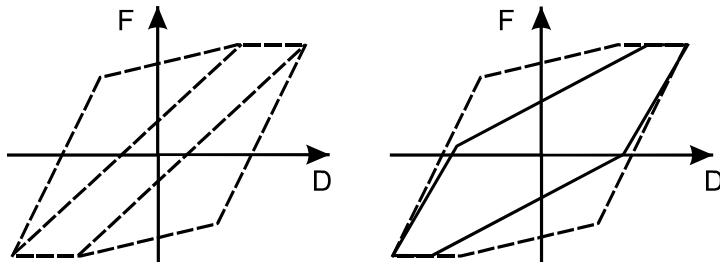


Figure 5.15 E-P-P Relationship

### 5.12.3 Trilinear Relationship After U Point

The loops in Figure 5.14 apply for a trilinear F-D relationship before the U point is reached. If a trilinear component is deformed past the U point, the hysteresis loop degrades as shown in Figure 5.16.



(a) E-P-P and Bilinear Parts

(b) Degraded Loop

Figure 5.16 Trilinear Relationship Beyond U Point

Figure 5.16(a) shows the non-degraded loop. This is divided into a parallelogram and a pair of triangles, as shown. The parallelogram is degraded as for the E-P-P component in Figure 5.15. The two triangles make up a bilinear loop as in Figure 5.14, which is degraded using the unloading stiffness factor. The result for the case of Loop A (unloading stiffness factor = +1) is shown in Figure 5.16(b).



### 5.12.4 Procedure for Unloading Stiffness Factor

Use the **Component Properties** task and the **Inelastic** component page. Choose a component type, and go to the **Cyclic Degradation** page. If the option to control the loop shape is available for the component type, there will be an **Unloading Behavior** frame.

In this frame, specify the value for the Unloading Stiffness Factor. You must also specify energy degradation factors, using either the YULRX or YU+3 option.

After you have specified the component properties and pressed **Save**, it is a good idea to press the **Plot Loops** button, and to plot hysteresis loops to check the cyclic behavior. If the **Plot Loops** button is not shown, first press **Graph** to plot the component properties.

This feature is currently limited to the following components.

- (1) FEMA beam, steel type or concrete type.
- (2) Moment hinge.
- (3) Inelastic steel material.
- (4) Inelastic shear material.

## 5.13 Cross Sections

---

### 5.13.1 The Role of Cross Sections

In a computer program for linear analysis you usually specify the properties of an element by assigning it a cross section and an elastic material. In most cases the element properties are then completely defined. It is not so simple for nonlinear analysis, because more properties are needed. Linear analysis needs only stiffness properties, whereas nonlinear analysis needs both stiffness properties and strength properties. Nevertheless, PERFORM-3D uses cross sections in essentially the same way as a linear analysis program.

If an element is based on a cross section and consists of one basic component (e.g., a panel zone element), it typically requires three steps to assign the element properties, as follows.

- (1) Define cross sections.

- (2) Define basic components, using cross section properties.
- (3) Assign basic components to element.

If a beam or column element consists of a compound component it could require up to six steps to assign the element properties. For example, if a beam element has plastic hinges, the steps might be as follows.

- (1) Define beam cross sections, with stiffness and strength properties.
- (2) Define plastic hinge components, using cross section properties.
- (3) Define shear strength sections, using cross section properties.
- (4) Define compound components, using plastic hinge components (for hinges), beam cross sections (for beam segments between hinges) and strength sections (to check shear strength).
- (5) Assign compound components to elements.

As another example, if a beam element uses fiber cross sections the steps might be as follows.

- (1) Define materials.
- (2) Define fiber cross sections, using material properties.
- (3) Define non-fiber beam cross sections.
- (4) Define shear strength sections, using non-fiber cross section properties.
- (5) Define compound components, using fiber cross sections (for hinge regions), non-fiber cross sections (for beam segments between hinge regions) and strength sections (to check shear strength).
- (6) Assign compound components to elements.

The process is thus more complex than for a linear analysis, and requires more careful planning.

### 5.13.2 Beam and Column Sections

In a linear analysis you may not need to distinguish between cross sections that are used for beam elements and those that are used for column elements. In PERFORM-3D it is necessary to make this distinction, because the strength properties required for a column are more complex than those required for a beam. For example, a beam element usually has a small axial force, and is bent about only one axis. Hence, in defining the bending strength for a beam section it is

sufficient to consider uniaxial bending with no P-M interaction. However, a column element can have large axial forces and is usually bent about both axes. Hence, in defining the bending strength for a column section it is necessary to consider biaxial bending and P-M-M interaction.

For any beam or column section, you must define specify the stiffness properties, but it is optional to specify the strength properties. If you specify only the stiffnesses, there is no difference between a beam and column section. However, it is usually a good idea to specify cross section strengths, because these strengths can be used to define the strengths of other components. For example, if you specify a P-M-M yield surface for a column cross section, and if you create a P-M-M plastic hinge component that uses the cross section, the hinge component gets its yield surface from the cross section.

For steel beam and column sections you can specify the cross section properties directly, or you can make use of standard steel sections. Details of the procedure are presented later.

### 5.13.3 Fiber Cross Sections for Beams and Columns

In a fiber section for a beam or column, the cross section shape is divided into a number of fibers, and material properties are assigned to each fiber. Several different materials can be used, including concrete, non-buckling steel, and buckling steel. For beams, short segments with fiber sections can be used to model hinge regions and also connections, including connections with fracture. For columns, fiber sections can be used to model hinges with P-M-M interaction that do not rely on a yield surface and plasticity theory. It is, however, substantially more work to use a fiber section than a standard beam or column section, and the computer time required for analysis may be larger.

### 5.13.4 Fiber Cross Sections for Walls

Shear wall and general wall compound components have stiffness and strength in shear and in combined axial-bending. For the shear properties you must first specify a shear material, then specify an effective wall thickness for shear. For axial-bending you must define a cross section. This is a "fiber" cross section, usually made up of steel and concrete fibers. Essentially, you must define steel and concrete material properties, then specify the area, location and material for each fiber in the cross section.

### 5.13.5 Other Sections

Other types of cross section may be used, including sections for elastic membrane and elastic shell elements.

## ***5.14 Using Cross Section Dimensions***

---

### 5.14.1 General

When you create a beam or column cross section, you can specify its breadth (B) and depth (D) (for standard sections the dimensions are obtained automatically). When you create certain other components (e.g., plastic hinges) you can associate the component with a cross section, and the component then inherits the dimensions from the cross section. You can use these dimensions in the following ways.

- (1) To assign lengths to basic components in a compound component.
- (2) To determine the properties of connection panel zones.
- (3) To assign lengths to stiff end zones for beam and column elements.

The first of these uses is considered in this section. Panel zones and stiff end zones are considered in later sections.

### 5.14.2 Length of Uniform X-Section Segment

Frame compound components will usually consist of segments with uniform cross sections, separated by components such as hinges. You specify the beam or column cross section for each segment, and this defines the segment stiffness. If you wish, you can also use the cross section dimensions to specify the segment length.

For example, consider a beam member with a Reduced Beam Section (RBS) located at some distance from the column face. This distance is often specified as a multiple of the beam flange width. Part of a possible frame compound component for this member is shown in Figure 5.17.

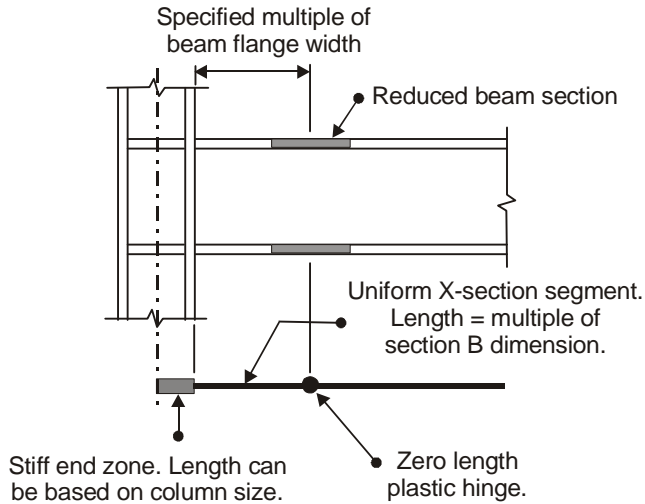


Figure 5.17 Model Using Component Dimensions

The beam member will be modeled using a frame compound component. At each end of the compound component there is (a) an end zone that extends from the end node to the column face, (b) a uniform X-section segment that extends to the center of the RBS, and (c) a zero-length plastic hinge to represent the RBS. The flange width in the uniform X-section segment is the B dimension for the cross section. One of the options for the length of a uniform X-section segment is to specify a fixed length, which you could calculate as a multiple of the cross section breadth. If you choose this option, and if some time later you change the size of the cross section, you must also change the length for the segment. This requires changes in two places, and can make it difficult to maintain consistency. Another option for the length of a uniform X-section segment is to specify the length as a multiple of the cross section breadth. If you choose this option, and if some time later you change the size of the cross section, the segment length is automatically updated. Hence, by using the cross section dimensions you can save substantial effort.

### 5.14.3 Tributary Length for a Curvature Hinge

In PERFORM-3D a plastic hinge can be of "rotation" type, in which case it automatically has zero length, or of "curvature" type, in which

case you must specify a tributary length. In some cases the tributary length may be a multiple of the beam or column depth,  $D$ . As in the preceding section, if you associate the plastic hinge component with a cross section, the depth  $D$  becomes a property of the hinge component. You can then specify the tributary length of the hinge as a multiple of the  $D$  dimension. If you change the cross section, the tributary length is automatically updated.

Note that in this case you probably will get not only the hinge component dimensions from the cross section but also some of the hinge component strength data. If you change the cross section, the strength data is also updated.

### **5.15 "Auto" Components**

---

#### 5.15.1 Auto End Zones for Beams and Columns

In PERFORM-3D, end zones are elastic components that are stiff but not rigid. They can be used at the ends of frame compound components.

In most cases you will not need to define any end zone components (although you can if you wish). The reason is that PERFORM-3D automatically sets up a default end zone component that can be used for beam and column elements. This component has a stiffness that is 10 times larger than the body of the component, and an "auto" length that is obtained from the dimensions of the adjacent beams and columns. Default end zones in frame compound components that are used for beam elements get their lengths from the depths of the adjacent columns, and default end zones in components that are used for column elements get their lengths from the depths of the adjacent beams. For brace/other frame elements (e.g., actual bracing members), you can not use default end zones.

#### 5.15.2 Auto Panel Zones in Steel Frames

The properties of a connection panel zone in a steel frame depend on the properties of the connected beams and columns. When you specify a "regular" panel zone component in a steel frame, you have the option of specifying cross sections for the beam and column (these must be I-shaped sections). If you use this option, PERFORM-3D uses the cross

section properties to calculate the panel zone stiffness and strength. If you change a cross section, the panel zone properties are automatically updated.

You must, however, be careful. PERFORM-3D uses the cross sections that you specify with the properties of the panel zone component, not the cross sections of the connected beams and columns. At each beam-to-column connection you must be careful to specify the same cross sections for the panel zone as for the for the beam and column. If there are several beam and column sizes in the structure, you will also need several panel zone components, one for each different combination of beam and column size.

An alternative is to use the "auto" panel zone component. With this component, PERFORM gets the beam and column sizes at each connection, and creates a different panel zone component for each different combination. This can greatly reduce the effort required to specify panel zones, at the expense of some loss of control over the process.

The steps for specifying an Auto Panel Zone component are as follows.

- (1) In the **Component Properties** task go to the **Inelastic** page and the **Auto Connection Panel Zone** component type. Either start a new component or choose an existing one.
- (2) Specify the component properties in much the same way as for other components. In the **Formulas** page you can specify parameters that control the stiffness and strength. Only the basic F-D relationship is calculated automatically. You must still specify strength loss, deformation capacities and cyclic degradation as for a standard panel zone component.
- (3) Use **Save** and **SaveAs** in the usual way.

If all panel zones in a structure have the same column yield stress, the same doubler plate thickness and yield stress, and the same data for strength loss, deformation capacities and cyclic degradation, you will need only one Auto Panel Zone component.

The F-D relationship is not drawn for this component type (if you press **Graph**, nothing happens), because the F-D relationship is not yet known. This means that you can not immediately check that the

component properties are correct, and that you have to exercise more care. You can (and should) check the properties as follows.

- (1) Run one or more analyses, in order to create an ECHO.txt file.
- (2) Open the ECHO.txt file. Go to the section for inelastic panel zone component properties (search for "IZ1A").
- (3) If you have any "regular" panel zone components, their properties are listed first. A set of properties is then listed for each different beam-and-column combination where auto panel zone components are used. The name for each set of properties consists of the number of the column section (search for "COLM" in ECHO.txt to see the section details), followed by the depth of the beam section, and followed by the name of the Auto Panel Zone component. If this name is long it may be truncated, since a component name is limited to 40 characters.
- (4) Check that the properties are correct.
- (5) After you have run some analyses. use the Time History and/or Hysteresis Loops tasks to plot results for typical panel zone elements, to check that the behavior is correct.

### 5.15.3 Panel Zone Limit States

For defining limit states, non-auto and auto panel zone components are different components. Be sure to specify the correct component type when you define limit states. If you use both auto and non-auto components in a structure, you can have separate limit states for each type if you wish.

## ***5.16 Upper and Lower Bounds***

---

### 5.16.1 Overview

There can be substantial uncertainty about the strengths and stiffnesses of components and elements, and you may want to run analyses with different strength and/or stiffness values to assess sensitivity. One way to do this is to set up different analysis models with different strengths and stiffnesses. An easier way, in most cases, is to specify upper and lower bounds. You can then vary strength and stiffness without setting up multiple analysis models.



Upper and lower bounds can be specified for certain component properties. Inelastic components and materials can have bounds on both strength and stiffness. Elastic components and materials can have bounds on stiffness. Strength sections can have bounds on strength.

Upper and lower bounds can be specified for most, but not all, components.

There are two steps in the process, as follows.

- (1) When you specify component properties, you have the option of specifying upper and lower bounds. For a typical inelastic component, upper and lower bounds can be specified as multiples of the nominal strength and stiffness. The multiple for upper bound values must be between 1.0 and 2.0 (with a few exceptions). The multiple for lower bound values must be between 0.5 and 1.0.
- (2) When you define a new Analysis Series (see *Chapter 17, Analysis Series*), you can specify that strengths and stiffnesses other than the nominal values are to be used. For any inelastic component you can specify U/L Bound Ratios for strength and stiffness. A zero ratio means that the nominal values are to be used. A ratio of 1.0 means the upper bound values, and a ratio of -1.0 means the lower bound values. A ratio between -1.0 and 1.0 means interpolation between the upper and lower bounds.

The default is to use the nominal values.

### 5.16.2 Procedure for Specifying Upper/Lower Bounds

Use the **Component Properties** task. If a component allows upper and lower bounds, there will be an **Upper/Lower Bounds** option (“Yes” or “No”) and an **Upper/Lower Bounds** properties page.

A relatively simple component such as an inelastic bar allows upper and lower bound values for axial strength and axial stiffness. A more complex component such as a P-M-M hinge has separate strength bounds for bending and axial force.

To specify bounds, choose the “Yes” option for Upper/Lower Bounds, then go to the **Upper/Lower Bounds** page. In addition to the upper and lower bound values, there are options that define the effects of the bounds on the following parameters.

- (1) Deformation at U Point, DU. This applies for trilinear F-D relationships only. The options are (a) keep DU constant as the strength and/or stiffness are varied, (b) keep the ratio DU/DY constant, and (c) keep the deformation DU-DY constant.
- (2) Deformation at L Point, DL. This applies only for cases with strength loss. The options are (a) constant DL, (b) constant DL/DY, (c) constant DL-DY, (d) constant DL/DU (for trilinear relationships only) and (e) constant DL-DU.
- (3) Deformation at X Point, DX. The options are the same as for DL.
- (4) Deformation capacities. The options are the same as for DL.

To show the effects of the bounds you can specify U/L Bound Ratios in the **Upper/Lower Bounds** page. These are the same as the ratios that can be specified for an Analysis Series (i.e., values between -1 and 1). To see the effect, specify U/L Bound Ratios for strength and/or stiffness, and check the **Show Effect** box. Then, when you press **Check** or **Graph**, the plotted properties show the U/L bound effects. If you plan to use U/L bounds, be sure to use this feature to check that the scaled F-D relationships and other properties are as expected.

Upper and lower bounds are not considered if you use the **Plot Loops** feature. For this feature, only the nominal values are used.

### 5.16.3 Procedure for Using Upper/Lower Bounds

This section has described how to specify Upper/Lower bounds as component properties. The procedure for using these bounds is described in *Chapter 17, Analysis Series, Section 17.4*.

## 6 Elements

---

*This chapter describes how to specify elements. There are three main parts to the data for an element, namely overall location, local axis orientation and component properties. If you wish you can specify element locations before you specify component properties. However, you will not be able to assign properties to elements until you have specified the required components.*

### 6.1 Element Types

---

#### 6.1.1 General

This section presents brief descriptions of the available element types. For more detailed descriptions see the document *PERFORM Components and Elements*.

#### 6.1.2 Simple Bar Elements

Use Simple Bar elements to model members that resist axial force only. Each element connects two nodes and consists of one basic component. Bar elements do not need local axes.

You can use a number of different basic components for bar elements, including simple elastic bars, inelastic bars, elastic gap-hook bars, nonlinear elastic bars, concrete strut components that use concrete materials, and steel bar/tie/strut components that use non-buckling or buckling steel materials.

You can specify initial strains in bar elements (essentially the same as thermal expansion effects). This can be useful for modeling bars that have initial stresses from prestressing or other causes.

#### 6.1.3 Beam Elements

Use Beam elements for members that would typically be identified as beams or girders.

Beams will usually have relatively small axial forces, so that it is usually not necessary to account for P-M interaction. Also beams will usually be bent in only one plane, about the cross section strong axis. Hence it is usually not necessary to consider biaxial bending or biaxial shear. However, you can consider P-M interaction, biaxial bending and/or biaxial shear if you wish (i.e., you can model a beam element using column type components).

Each beam element connects two nodes and consists of one frame compound component. In most cases you can use a single beam element to model a complete beam member, from column centerline to column centerline. The exception is if a member such as a diagonal brace connects to the beam within its span. In this case you must locate a node at the connection point and divide the beam into two or more elements.

You must assign local axis orientations to beam elements. Local Axis 1 is along the element, from element End I to element End J. Local Axes 2 and 3 are normal to Axis 1, with Axes 1, 2 and 3 forming a right handed coordinate system. Bending in the 1-2 plane (i.e., bending about Axis 3) is usually strong axis bending. Bending in the 1-3 plane (about Axis 2) is usually weak axis bending. Axis 2 is usually directed vertically upwards, and Axis 3 is usually horizontal. However, the axes may be inclined.

When you add beam elements to a structure, be sure to use a consistent method for choosing Ends I and J. If the beams are essentially along the H1 and H2 directions, we suggest that you always direct I-J along the corresponding axis.

You can specify distributed and/or concentrated loads along the element length (these are element loads, as distinct from nodal loads). In the current version of PERFORM-3D, only gravity loads are allowed (usually loads in the negative V direction).

### 6.1.4 Column Elements

Use Column elements for members that would typically be identified as columns.

Columns will usually have relatively large axial forces, so that it is necessary to account for P-M interaction. Also columns will usually be

bent in about both cross section axes, so that it is necessary to consider biaxial bending and biaxial shear. However, if a column has small axial forces and is bent mainly about one axis, you can model it using beam type components.

Each column element connects two nodes and consists of one Frame Type compound component. In most cases you can use a single column element to model a complete column member, from beam axis to beam axis. The exception is if a member such as a diagonal brace connects to the column within its height. In this case you must locate a node at the connection point and divide the column into two or more elements.

You must assign local axis orientations to column elements. Local Axis 1 is along the element, from element End I to element End J, and is usually vertical. We suggest that you consistently make End I the lower end and End J the upper end. Local Axes 2 and 3 are normal to Axis 1 and are usually horizontal. You must specify the orientations in plan of Axes 2 and 3.

You can specify distributed and/or concentrated loads along the element length. However, since only gravity loads are allowed (usually loads in the negative V direction) you probably will not do this for vertical column elements.

### 6.1.5 Brace/Other Frame Elements

Use Brace/Other Frame elements for members such as diagonal braces that are not clearly beams or columns, and can not be modeled as bar elements that resist only axial force.

You could use brace/other frame elements for beam and column members. However, if you want to use default end zones or panel zone elements, you must model beam and column members using beam and column elements. The reason is that PERFORM-3D uses beam elements to get the "auto" end zone lengths for columns and the depths for panel zones, and similarly uses column elements to get end zone lengths for beams and widths for panel zones. If you do not use beam and column elements, you must specify end zone lengths and panel zone dimensions explicitly.

You can model diagonal bracing members using simple bar elements or brace/other frame elements. If you can ignore the bending stiffness of a

bracing member, the simplest approach is to use a simple bar element. If you must consider the bending stiffness you must use a brace/other frame element. Each element consists of one frame compound component. You can use beam type basic components if you do not need to account for P-M interaction. You must use column type basic components if P-M interaction is important.

Do not model bracing members using beam or column elements. As noted earlier, PERFORM-3D bases the lengths of default end zones for columns on the dimensions of the adjacent beams, and the lengths of default end zone for beams on the dimensions of the adjacent columns. If you use beam or column elements for bracing members, PERFORM-3D may get confused and use the wrong dimensions.

As with beams and columns, you must assign local axis orientations to brace/other frame elements. Local Axis 1 is along the element, from element End I to element End J. Axis 2 will usually be in the vertical plane, and Axis 3 will usually be horizontal.

You can specify distributed and/or concentrated gravity loads along the element length.

### 6.1.6 Shear Wall Elements

Use Shear Wall elements to model most shear walls and shear cores, including coupled walls.

Shear wall elements act essentially as beams, with bending, axial and shear deformations. They can be elastic or inelastic for shear, and elastic or inelastic for bending, using fiber cross sections. Each element connects four nodes and consists of one Shear Wall compound component.

You must assign local axis orientations to shear wall elements. Local Axis 2 is usually vertical, along one element edge. Local Axis 3 is usually horizontal, also along an element edge (elements will usually be rectangular). Axis 1 is normal to the plane of the element (all four nodes for an element must be in one plane).

### 6.1.7 General Wall Elements

Use General Wall elements to model squat walls with irregular openings.

General wall elements are nonlinear finite elements specifically intended for modeling reinforced concrete walls that have irregular openings, and hence have complex bending and shear behavior. Each element connects four nodes and consists of one General Wall compound component.

You must assign local axis orientations to general wall elements. Local Axis 2 is usually vertical, along one element edge. Local Axis 3 is usually horizontal, also along an element edge (elements will usually be rectangular). Axis 1 is normal to the plane of the element.

### 6.1.8 Infill Panel Elements

Use Infill Panel elements to model masonry type infill in frames. Infill panel elements can be elastic or inelastic. Each element connects four nodes and consists of one basic component. You can use basic components with shear type or diagonal strut type behavior for infill panel elements.

Infill panel elements do not need local axes.

### 6.1.9 Connection Panel Zone Elements

Use Panel Zone elements to model shear deformations in beam-to-column connections. Panel zone elements can be elastic or inelastic. Each element connects to only one node (at the beam-column connection) and consists of one basic component.

You must assign local axis orientations to panel zone elements. Local Axis 1 is normal to the plane of the panel zone, Axis 2 is along the column, and Axis 3 is along the beam (assuming the usual case of beams and columns at right angles).

### 6.1.10 BRB Elements

Use BRB element to model buckling restrained brace members. BRB elements resist axial force only. Each element connects two nodes and

consists of one compound component. This component consists of an inelastic BRB component in series with an elastic bar component and an optional rigid end zone.

BRB elements do not need local axes.

### 6.1.11 Viscous Bar Elements

Use Viscous Bar elements to model fluid dampers. Viscous bar elements resist axial force only. Each element connects two nodes and consists of one compound component. This component consists of a fluid damper component in series with an elastic bar component.

Viscous bar elements do not need local axes.

### 6.1.12 Seismic Isolator Elements

Use Seismic Isolator elements to model isolators of rubber and friction pendulum type. You can also use rubber-type isolator elements to model other energy absorbing devices, such as the ADAS device (alternatively you could use inelastic bar elements).

Seismic isolator elements resist bearing and shear forces. In bearing the element is elastic, with an option for different tension and compression stiffnesses. In shear the element is inelastic. Each element connects two nodes and consists of one basic component.

You must assign local axis orientations to seismic isolator elements. Local Axis 3 is in the bearing direction, usually vertical. Axes 1 and 2 are in the shear directions, usually horizontal

### 6.1.13 Slab/Shell Elements

The slab/shell element is an elastic 4-node element with membrane (in-plane) and plate bending (out-of-plane) stiffnesses. It can be used to model slabs and curved shells.

For PERFORM-3D the most likely application is to model deformable floor diaphragms. In this application the membrane behavior of the element accounts for in-plane effects in the diaphragm, and the plate bending behavior can be used for applying and distributing gravity loads. A floor diaphragm can be modeled using wall elements, but this requires four steps



to define the element properties, namely material, cross section, wall compound component, and element. For a slab/shell element this is reduced to three steps, namely material, cross section, and element.

This element is a special case of the inelastic slab/shell element that is available in PERFORM-COLLAPSE. That element is intended primarily for modeling two-way slab floors. It has two cross section options, namely elastic (as in the PERFORM-3D element) and reinforced concrete (using a layered model similar to a fiber model for a beam section). The reinforced concrete version of this element is not currently available in PERFORM-3D.

You must assign local axis orientations to slab/shell elements. Axis 1 is normal to the plane of the element, and Axes 2 and 3 are in the element plane, similar to a wall element.

#### 6.1.14 Support Spring Elements

Each support spring element connects to one node and consists of one support spring component. In effect, each element connects the supported node to a point that is fixed in space. Support spring elements have no P- $\Delta$  effects. They are plotted as small circles at the supported nodes.

Support spring elements can have translational and/or rotational stiffnesses, with up to a 6x6 stiffness matrix. Strength capacities and strength based limit states can be specified.

You must assign local axes to support spring elements. These axes can be inclined to the H1-H2-V axes if desired.

Support spring elements can be used to impose displacements at support points. This is done by making the elements very stiff and specifying that they have initial deformations (similar to thermal expansions).

#### 6.1.15 Deformation Gage Elements

PERFORM allows you to specify deformation capacities for single components. In some cases, however, you may want to measure deformations over a number of elements, and specify corresponding limit states. For example, in a wall structure you may want to calculate the average strain over a story height, where there are multiple

elements over that height. You can do this with a Strain Gage element that extends over the story height.

Deformation gage elements do not add any stiffness to the structure. They act literally like strain gages. You can specify gage elements of the following types.

- (1) 2-node extensional strain gage.
- (2) 2-node rotation gage of beam type.
- (3) 4-node rotation gage for walls.
- (4) 4-node shear strain gage for walls.

Only the beam type rotation gage requires local axes. These are the same as for a beam element.

## **6.2 Specifying Elements**

---

### 6.2.1 Element Groups

Elements must be organized into Element Groups. The elements in any group must all be of the same type. However, the elements of any type need not all be in one group.

In a small frame you may need only two element groups, one containing all beam elements and one containing all column elements. In a larger frame you are likely to use more groups. For example, you might put the exterior columns in one group and the interior columns in another. One reason is that deformation and strength limit states are based on element groups, and you may want to have the exterior and interior columns in different limit states. If you have a structure with a variety of element types, you will need at least one element group for each element type.

Before you begin specifying a structure, spend some time planning the element groups.

If you need to, you can combine or split element groups after they have been created.

### 6.2.2 Steps

To specify elements press the toolbar button for the **Modeling** phase and the **Elements** task. The overall steps are as follows.

- (1) Create an element group. Each group is identified by a name.
- (2) Add elements to the group.
- (3) Assign orientations and properties to the elements.

You can edit an existing element group at any time, to add elements, delete elements or change element properties. You can also delete entire groups.

### 6.2.3 Start a New Element Group

To start a new group press the **New** button, choose the element type, input the group name, and specify whether you want to consider or ignore geometric nonlinearity.

Choose "None" for geometric nonlinearity effects if you are sure that the elements in the group contribute no significant geometric nonlinearity effects. This will often be the case for beam elements, since they are horizontal and typically have small axial forces. If in doubt choose "P-delta". For some elements the option is automatically "None". The "Large displacements" option is currently disabled for all elements (this option can be chosen in PERFORM-COLLAPSE).

You also have the option of specifying a scale factor for " $\beta K$ " viscous damping. For an explanation of the available damping models see *Chapter 18, Elastic Viscous Damping*. If you wish, you can increase or decrease the  $\beta K$  damping for an element group by specifying a scale factor that is larger or smaller than 1.0. If you believe that the calculated  $\beta$  value will overestimate the energy dissipation for the elements in a group, specify a factor smaller than 1.0 for that group. If you believe that the calculated  $\beta$  value will underestimate the energy dissipation, specify a factor larger than 1.0. If you want to ignore  $\beta K$  damping for an element group, specify a factor of 0.

### 6.2.4 Add Elements

To add elements choose the **Add Elements** tab. To add one or more elements, select a set of nodes and press the **Test** button (use the **Test**, **Clear**, **OK** and **Undo** buttons in the usual way).

For elements that connect two nodes you have the choice of *Sequence*, *Series*, *Straight Line* or *Grid* generation. There are diagrams that illustrate these options. You can save time by using the appropriate generation option.

For elements that connect four nodes you have the choice of *Sequence*, *Column*, *Row* or *Grid* generation. There are diagrams that illustrate these options.

As noted earlier, take care with the element I and J ends. Each 2-node element has an I end and a J end. When the elements are plotted the I ends are marked with a small dot. Each 4-node element has I, J, K and L corners. The I end of edge IJ is marked by a dot.

### 6.2.5 Parallel Elements

There may be occasions where you want to have two or more elements in parallel (i.e., connecting the same nodes). You can not do this within an element group. You can have parallel elements, but they must be in different element groups.

### 6.2.6 Delete Elements

To delete elements choose the **Delete Elems** tab. You can select elements one at a time by clicking on an element or, for a 4-node element, an element edge. Alternatively you can select multiple elements with a box select, by dragging a rectangle that contains the middle parts of the elements to be selected.

Selected elements will change color. If you select an element a second time, it is deselected. Use the **Test**, **Clear**, **OK** and **Undo** buttons in the usual way.

### 6.3 *Element Orientations*

---

To assign local axis orientations to elements choose the **Orientations** tab. There are diagrams to illustrate how Axes 1, 2 and 3 are defined. The steps are as follows.

- (1) Choose the element group from the Current Group list. Elements in the current group are light blue. Elements that already have orientations assigned have axis symbols (see below for explanation of these symbols).
- (2) Select one or more elements. You can select elements in the following ways. Selected elements are red.
  - (a) Select singly or using a box select.
  - (b) Double click on an element. This selects all elements that have the same orientation data.
  - (c) Double click in blank space. This selects all elements that do not yet have assigned orientations.

If all of the selected elements have the same orientation data, the data in the **Orientations** tab will be set. You can use this feature to review the orientation of an element. If the selected elements have no orientations, or if they have different orientations, the data in the **Orientations** tab is not set.

- (3) Choose the orientation type, and input any needed data.
- (4) Press **Test** to show the new orientations, then either **OK** or **Undo**.

Diagrams in the form illustrate the elements axes (Axes 1, 2 and 3) and explain how the orientations are assigned.

For panel zone elements you have two options for assigning the orientations, namely "Standard" and "Nonstandard". If you choose the Standard option, the orientation of a panel zone element depends on the orientation of the adjacent column elements. This applies only to cases where the columns are I sections, and only if there is no ambiguity. If you choose the Nonstandard option you must input the orientation data directly. If you assign "auto" panel zone components to any elements, you must use the Standard option for those elements.

The plotted orientation symbols have the following meanings.

- (1) For a frame type element the symbol is plotted near the element End I. Axis 1 is along the element from End I to End J. The orientation symbol is a pair of lines along Axes 2 and 3, with a dot on the positive Axis 2 side.
- (2) For a panel zone element Axis 1 is normal to the element, and has a dot at its end. Axes 2 and 3 are in the plane of the element. Axis 2 has a dot at its end.
- (3) For a shear wall or general wall element the symbol is plotted at Node I. Axis 1 is normal to the element, and has a dot at its end. Axes 2 and 3 are in the plane of the element. Axis 2 has a dot at its end.
- (4) For a seismic isolator element, Axis 3 is the bearing direction and Axes 1 and 2 are the shear directions. The 1-2 plane is shown as a rectangle. The axes have dots at their ends.
- (5) For a support spring element, Axes 1 and 2 are usually the structure H1 and H2 axes, and Axis 3 is usually the V axis. However, the axes can be rotated if desired. The 1-2 plane is shown as a rectangle. The axes have dots at their ends.
- (6) For a beam-type rotation gage element, the axes are the same as for a frame type element.

## **6.4 Element Properties**

---

To assign properties to elements choose the **Properties** tab. You must have specified the properties for all needed components. The steps are as follows.

- (1) Select one or more elements. You can select elements in the following ways. Selected elements are red.
  - (a) Select element singly or using a box select. The properties that are currently assigned to the selected elements, if any, will be shown in the **Component Currently Assigned** part of the form.
  - (b) Double-click on an element. This selects that element plus all other elements in the group that have the same properties. The

currently assigned properties will be shown in the **Component Currently Assigned** part of the form.

- (c) Double click in blank space. This selects all elements in the group that do not yet have assigned properties.
- (2) If you want to assign a component to the selected elements, or change the currently assigned component, press the **Assign Component** button. The properties that you can assign will be shown in the **Component to be Assigned** part of the form. Select the component, then press **Assign**.

The **Properties** tab has a **Show Properties** button. If you press this button, you can review the properties of the current component. You can not change the properties in this mode. To change the properties you must use the Component Properties task.

## ***6.5 Moving Elements Between Groups***

---

To move elements from one group to another, choose the **Change Group** tab. You can then move elements from the current group to another group. If you want to split an element group, first create one or more new groups, with no elements. Then use Change Group to move elements into the new groups. You can only move elements that have no element loads.

You can also change the name of any element group, by pressing **Rename** in the **Group Data** tab.

## ***6.6 P- $\Delta$ , P- $\delta$ and Large Displacement Effects***

---

### **6.6.1 General**

P- $\Delta$ , P- $\delta$  and large displacement effects can cause nonlinear behavior of elements and hence of complete structures. This is usually referred to as "geometric" nonlinearity. PERFORM-3D gives you options to include or ignore geometric nonlinearity.

This section reviews the different types of geometric nonlinearity, and summarizes the PERFORM-3D options.

### 6.6.2 P- $\Delta$ vs. True Large Displacements

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

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

In reality neither of these assumptions is correct.

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

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

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

Figure 6.1 illustrates the difference for a simple bar.



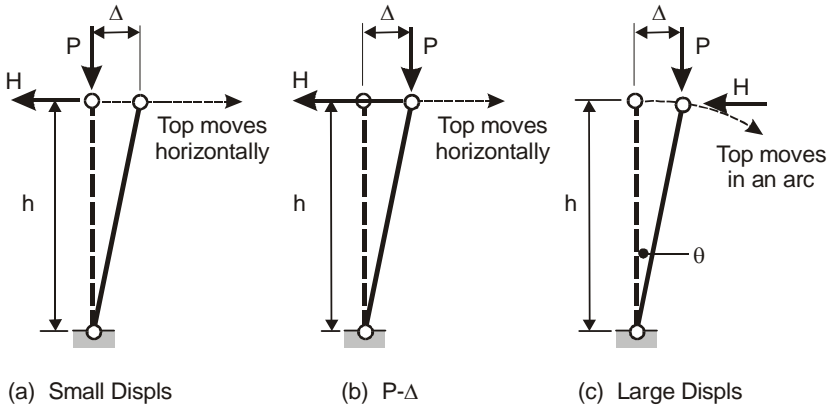


Figure 6.1 Geometric Nonlinearity

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

- (a) **Small displacements theory.** This theory says (i) that the top of the bar moves horizontally (this is small displacements geometry, which also predicts that the bar extension is zero), and (ii) that equilibrium can be considered in the undeformed position. Hence, the force  $H$  is zero for all values of  $\Delta$  (take moments about the base of the bar).
- (b) **P- $\Delta$  theory.** This theory says (i) that the bar moves horizontally and the bar extension is zero (small displacements geometry), and (ii) that equilibrium is considered in the deformed position. Hence  $H = P\Delta/h$ .
- (c) **Large displacements theory.** This theory says (i) that the top of the bar moves in an arc, so that it moves vertically as well as horizontally, and the bar extension is indeed zero, and (ii) that equilibrium is considered in the deformed position. Hence  $H = P\Delta/h\cos\theta$ .

The difference between the value of  $H$  from P- $\Delta$  theory and from large displacements theory is small up to quite large rotations. For example for  $\Delta/h = 0.05$  (a large drift for most structures), P- $\Delta$  theory gives  $H =$

0.05V and large displacements theory gives  $H = 0.05006V$ , which is a negligible difference. Also, for  $\Delta/h = 0.05$  the vertical displacement in case (c) is  $0.00125h$ . P- $\Delta$  theory predicts zero, which is not a significant error in most cases. Hence, for most structures it is accurate enough to use P- $\Delta$  theory.

Consider, however, the simple bar structure in Figure 6.2.

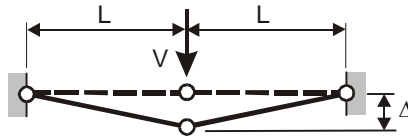


Figure 6.2 Case Where P- $\Delta$  Theory Is Not Accurate

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

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

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

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

In most building structures subjected to earthquake-type loads, the behavior is more closely analogous to Figure 6.1 than to Figure 6.2. P- $\Delta$  theory works well in this case, and has the advantage that it is both simpler to apply than large displacements theory, and requires less computation.

### 6.6.3 P- $\delta$ Effect

Figure 6.3(a) shows a cantilever column with vertical and horizontal loads.

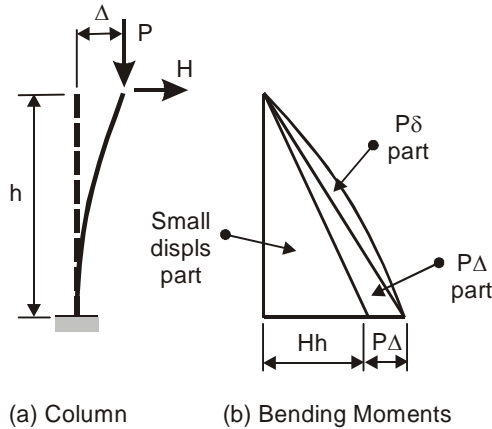


Figure 6.3 P- $\Delta$  and P- $\delta$  Effects

If the column remains elastic, it deforms as shown. Hence, considering equilibrium in the deformed position, the bending moment diagram is as shown in Figure 6.3(b) (not exactly, if we consider true large displacements, but to a high degree of accuracy).

The bending moment diagram has three parts, as follows.

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

Computationally, it is easy to account for the P- $\Delta$  part of the moment, since it depends on only the overall rotation of the column. It is more difficult to account for the P- $\delta$  part, since it depends on the bending

deformation of the column (which in turn depends on the moments and on whether the column yields or remains elastic).

It is possible to account for the  $P-\delta$  effect in structural analyses. However, it is important to be careful when considering this effect. In Figure 6.3, the column is elastic. Figure 6.4 shows the same column, but now it yields and forms a plastic hinge at the base.

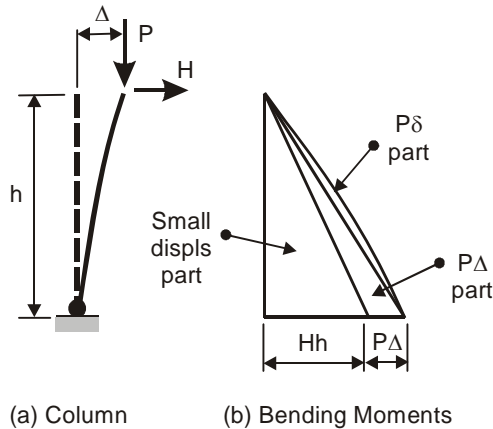


Figure 6.4  $P-\delta$  Effect When Column Yields

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

Figure 6.4 also shows why  $P-\Delta$  effects reduce the strength of a column. Let the moment capacity at the plastic hinge be  $M$ . This is a constant quantity – it is not affected by  $P-\Delta$  effects. If we use small displacements theory, the plastic hinge forms when  $M = Hh$ , and the predicted horizontal strength of the column is  $H = M/h$ . If we consider  $P-\Delta$  effects, the hinge forms when  $M = Hh + P\Delta$ , and the predicted strength is  $H = (M - P\Delta)/h$ .

#### 6.6.4 Do You Need To Consider P- $\delta$ Effects?

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

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

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

#### 6.6.5 Axial Shortening Due to Bending

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

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

You can convert P- $\delta$  effects to P- $\Delta$  effects by adding nodes within a member length. Adding nodes also converts "bending" effects to "bar" effects.

### 6.6.6 PERFORM Options

PERFORM-3D gives you the option of including or ignoring geometric nonlinearity.

As noted, for structures subjected to earthquake type loads, it is sufficient to consider only P- $\Delta$  effects. In the present version of PERFORM-3D you can consider P- $\Delta$  effects but not true large displacement effects. This option is included in PERFORM-COLLAPSE, where true large displacement effects may be needed for accurate analysis of "catenary" effects in floor systems.

PERFORM-3D does not currently consider P- $\delta$  effects (i.e. it does not consider geometric nonlinearity within the length of a column or brace element). Hence, if you use a single element to model a brace member, PERFORM-3D will not model buckling of the brace within its length. You can, however, model this type of buckling by dividing a brace member into a number of short elements, and specifying that P- $\Delta$  effects are to be considered. Buckling of this type can be sensitive to initial out-of-straightness in the member, and you may have to make the member deliberately crooked in order to initiate buckling. If you want to consider this type of behavior, we recommend that you first test the member model in a small sub-assembly, to make sure that you get the expected behavior.

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

## 7 Load Patterns

---

*Load definition involves two steps. In the Modeling phase you set up load patterns. In the Analysis phase you set up load cases, by combining load patterns and adding other data. This chapter describes how to set up load patterns.*

### 7.1 Pattern Types and Restrictions

---

Load patterns are used to construct load cases for static analyses. There are three load pattern types, namely nodal load patterns, element load patterns and self weight patterns.

Nodal load patterns are straightforward, with H, V and/or R loads applied directly on nodes.

Element load patterns are more complex. In principle an element load pattern can include a variety of loads, including distributed loads, point loads and thermal expansion effects, applied on elements of a variety of types. PERFORM-3D currently includes only a few element load types, including gravity (vertical) loads on beam and column elements, and thermal expansion effects in bar elements.

Self weight patterns automatically calculate gravity nodal loads using component weights and element lengths or areas.

Nodal load patterns are usually used for either gravity load cases or push-over load cases. Gravity loads usually have V loads only, and push-over loads usually have H loads only. However, PERFORM-3D does not prevent you from specifying nodal load patterns that include both H and V loads.

Nodal load patterns are also used for Dynamic Force load cases.

## 7.2 Nodal Load Patterns

---

To specify a nodal load pattern press the toolbar buttons for the **Modeling** phase and the **Load Patterns** task, then choose the **Nodal Loads** tab.

Nodal load patterns are identified by name. To start a new pattern press the **New** button and follow the instructions. Select nodes and use the **Test**, **Cancel**, **OK** and **Undo** buttons in the usual way.

You can review, add, delete or change nodal loads at any time. You can also delete complete load patterns.

## 7.3 Element Load Patterns

---

To specify an element load pattern press the toolbar buttons for the **Modeling** phase and the **Load Patterns** task, then choose the **Element Loads** tab.

Element load patterns are identified by name. To start a new pattern press the **New** button and input the pattern name. The steps for specifying element loads are then as follows. These steps are rather complicated, so proceed carefully. It can be a good idea to use Frames to show only parts of the structure.

- (1) Choose an element group from the list.
- (2) Within the element group set up element subgroups, where all elements in a subgroup have identical element loads. At one extreme, if all of the elements in the element group have identical loads there is only one subgroup. At the other extreme, if every element has a different load each subgroup consists of only one element. You will usually have a number of subgroups, each with a number of elements. To define a subgroup choose the **Loaded Elems** tab and the **Define new subgroup** option. Then define the subgroup as follows.
  - (a) The elements that are not yet assigned to subgroups are shown light blue. Select the elements for the subgroup from these



elements. As you select elements for the current subgroup they will change to green. To deselect an element, select it a second time. You can use single element or "box" selects.

- (b) When the subgroup is complete, press **Done**.

This completes the subgroup. You can now either define another subgroup, by repeating Steps (a) and (b), or you can specify element loads and return to add more subgroups later. If you are ready to specify element loads, go to Step (3). If you wish you can change any subgroup, by adding or deleting elements, as noted later.

- (3) Add element loads. To do this choose the **Add Loads** tab. Define the loads as follows.
- (a) Choose the subgroup number from the list. The elements in the subgroup will be shown green. The "No. of elems" box shows the number of elements in the subgroup, and the "No. of loads" box shows the number of load components currently defined for this subgroup. If there are any loads they will be plotted in a floating window. If this window obscures the model, drag it out of the way (click and hold inside the window, drag, then release).
- (b) Choose the load type from the list. A diagram in the form will show the required data.
- (c) Enter the required data, and use the **Test**, **Cancel**, **OK** and **Undo** buttons in the usual way.

Repeat Steps (a), (b) and (c) for as many load components as you want to apply. You can change subgroups by choosing the subgroup number from the list. You can define new subgroups by returning to the **Loaded Elems** tab.

Repeat Steps (1), (2) and (3) for as many element groups as you want to include in the load pattern.

To review, add, change or delete a subgroup, choose the **Loaded Elem** tab and the **View/Edit existing subgroups** option, and follow the instructions.

To delete load components, choose the **Delete Loads** tab, and follow the instructions.

## **7.4 Self Weight Load Patterns**

---

To specify a nodal load pattern press the toolbar buttons for the **Modeling** phase and the **Load Patterns** task, then choose the **Self Weight** tab.

Self weight load patterns are identified by name. To start a new pattern press the **New** button and follow the instructions.

To specify a self weight load pattern you select the elements that are to be included in the pattern. You can add as many elements as you wish, from any number of element groups, provided that the elements are of a type that allow self weight loads.

To add elements choose the **Add Elements** tab and follow the instructions. To delete elements choose the **Delete Elements** tab.

The self weight for a 2-node element is applied as nodal loads at its I and J end, 50% at each end. The self weight for a 4-node element is applied as nodal loads at its I, J, K and L corners. For a rectangular element 25% of the weight is applied at each corner. For an element that is not rectangular the weight is distributed among the nodes in a reasonable way.

You can show the calculated loads at any time by pressing **Plot Loads**. These loads are based on the current self weights of the components. If you change these weights, the calculated loads will change. For analysis of the structure, the loads are calculated using the component weights at the time the analysis is run.

## 8 Drifts and Deflections

---

*Horizontal drifts are useful measures of deformation for lateral loads. Vertical deflections can be useful measures of deformation for long span structures. You must specify at least one drift, as a Reference Drift for the Analysis phase. This chapter explains how to specify drifts and deflections.*

### 8.1 Drifts

---

#### 8.1.1 Simple Drift

A simple drift is the horizontal displacement of one node relative to a second node that is lower down in the structure, divided by the height between the nodes (i.e., in PERFORM, a "drift" is a dimensionless drift ratio). If the upper node is the roof of a building and the lower node is at ground level, the drift value is the roof displacement divided by the building height. This is the average drift over all stories. If the upper node is at one floor and the lower node is at the floor below, the drift is for one story of the building.

Note that a drift value of, say, 0.02 is a 2% drift.

#### 8.1.2 Distortion Drift

In some frame structures the effective drift can be different in different bays of the frame. This is shown in Figure 8.1.

In this case the maximum shear deformation in a story can be substantially larger than the simple drift. You can account for this using distortion drifts, each of which is defined by four nodes. The drift magnitude is the shear deformation, in radians. Again, this is dimensionless, and a value of, say, 0.02 corresponds to a drift of 2%.

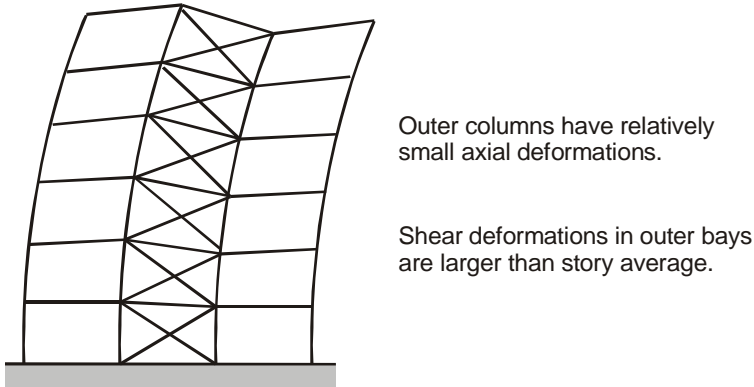


Figure 8.1 Effective Drift Exceeds Story Drift

### 8.1.3 Procedure for Specifying a Drift

To define one or more drifts press the toolbar buttons for the **Modeling** phase and the **Drifts and Deflections** task, then choose the **Drifts** tab.

Drifts are identified by name, with a maximum of 20 characters. To define a new drift press the **New** button and enter the drift name. Then choose either the **Simple Drift** or **Distortion Drift** option. For a simple drift choose the drift direction, select the upper and lower nodes as indicated in the form, and press **Test** to show the drift. For a distortion drift, select the four nodes in the sequence shown in the form, and press **Test** to show the drift. Then use the **OK**, **Undo** and **Clear** buttons in the usual way.

Once you have defined a drift you can not change it. You must delete it and replace it with a new one.

### 8.1.4 Reference Drift

You must define at least one drift, to use as the Reference Drift in the Analysis phase. This must be a simple drift, and will usually be a drift with the upper node at the roof level and the lower node at ground level. The drift value is then the average drift for all stories in the structure, and the roof displacement is the drift value multiplied by the structure height. You will usually (and you usually should) define other

drifts, possibly one for each story, to use in drift limit states and to serve as "controlled" drifts for static push-over analysis.

## **8.2 Deflections**

---

To define one or more deflections press the toolbar buttons for the **Modeling** phase and the **Drifts and Deflections** task, then choose the **Deflections** tab.

A deflection is the downwards V displacement of one node (the deflected node) relative to a second node (the reference node). The reference node is not necessarily at a support. Deflections have the units of length.

Deflections are identified by name, with a maximum of 20 characters. To define a new deflection press the **New** button and enter the deflection name. Then select the nodes, and press **OK** to save the deflection.

Once you have defined a deflection you can not change it. You must delete it and replace it with a new one.



## 9 Structure Sections

---

*A Structure Section is a cut through all or part of a structure. Structure Sections have a number of uses.*

*The forces on structure sections can be useful for checking the behavior of a structure. For example, if a structure has more than one lateral load resisting system, you may want to know how the total load is distributed among them. You can do this by specifying sections through parts of the structure and comparing the forces on these sections.*

*In tall shear wall structures it is common to allow inelastic behavior in bending, but to require that the wall remain essentially elastic in shear. The shear strength in a wall can be checked either (a) in individual wall elements, or (b) for a wall cross section that extends over several elements. Because there can be stress concentrations, it may be better to check the strength for the wall section as a whole, rather than for individual elements. You can do this by defining structure sections through walls and assigning shear strengths to these sections.*

*You can plot diagrams of bending moment and shear force over the height of a tall shear wall, by cutting sections at different levels over the height and using the forces on these sections.*

*This Chapter explains the procedures.*

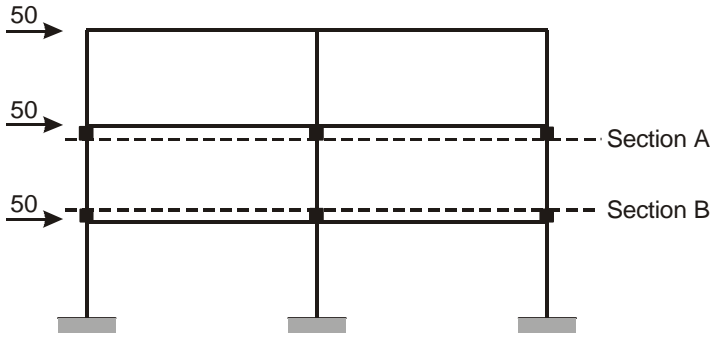
### 9.1 Concept

---

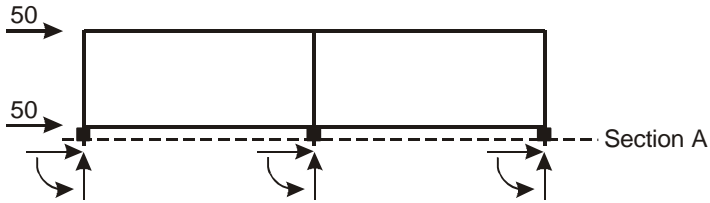
A structure section is defined by cutting through a number of elements. You can cut elements only at their ends. To define a structure section you must specify the cut elements and the cut location for each element (whether at the I or J end for a 2-node element, and whether at the IJ, KL, IK or JL edge for a 4-node element).

Figure 9.1(a) shows two alternative sections through a story of a frame. For Section A the element cut ends (shown by rectangles) are at the top

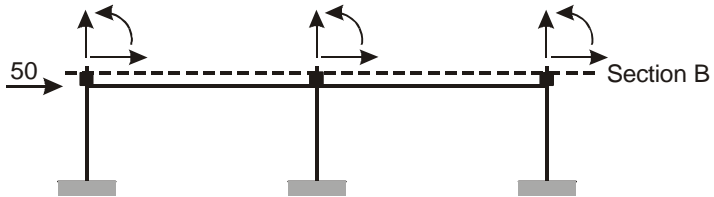
of the story, whereas for Section B the element cut ends are at the bottom of the story.



(a) Alternative Structure Sections in a Story



(b) Forces on Section A



(c) Forces on Section B

Figure 9.1 Forces on a Structure Section

To get the section forces the cut elements are removed, exposing the cut ends. The cut elements exert forces and moments on the section as shown in Figures 9.1(b) and (c). The H1, H2 and V forces and moments on the section are the resultants of the element H1, H2 and V forces and moments.



Structure sections are usually plane, but this is not essential, and PERFORM-3D does not check. Sections will most often be horizontal, and you will usually be interested in the horizontal (shear) forces. For the loads shown in Figure 9.1, the H force on section A is minus 100 units, and the H force on Section B is plus 100 units. The V force is zero for both sections.

## 9.2 Specifying Element Cuts

---

To specify a structure section press the toolbar buttons for the **Modeling** phase and the **Structure Sections** task. Choose the **Define Sections** page.

Structure sections are identified by name. To start a new section press the **New** button and input the section name. The steps for specifying the cut elements are then as follows.

- (1) Choose an element group from the list. Uncut elements in the group are shown white. Cut elements are shown red, with the cut ends marked by rectangles.
- (2) Choose the **Add Cuts** tab.
- (3) Choose the **End I** or **End J** option, depending on the desired cut location. For a 4-node element End I means at the element IJ edge, and End J means at the element KL edge.
- (4) Select the new elements to be cut, using single element or box select. The selected elements are yellow and the cut ends are marked by green squares.
- (5) Press **OK** to add the new cuts to the section. Press **Cancel** to clear the selected elements.
- (6) Repeat from Step (1) for as many element groups as needed.

To delete element cuts, choose the **Delete Cuts** tab, select the cut elements to be deleted from the section, and press **OK**.

You can change an existing structure section at any time, adding or deleting element cuts. You can also delete complete structure sections.

### 9.3 Shear Strength for a Wall Section

---

Figure 9.2 shows a plan view of a shear wall core.

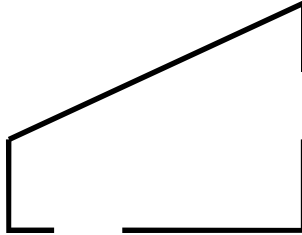


Figure 9.2 Plan of a Shear Core

Suppose it is required that the wall remain essentially elastic in shear. The wall has four sides, usually with a number of wall elements along each side. You may want to check the shear strength for each side separately. To do this the steps are as follows.

- (1) Calculate the shear strength of the wall, considering contributions from the concrete and the steel reinforcement in the usual way. The strength can be the strength of the entire wall, the strength per unit length of the wall, or the strength per unit area. There will probably be several different strengths, as the reinforcement percentage varies over the height of the wall.
- (2) Using the **Component Properties** task and the **Materials** tab, define one “Elastic Shear Material for Walls” material for each different strength. You will use these materials to define the properties of the Shear Wall Compound Components as well as the structure section strengths. If you wish you can specify that the shear strength depends on the axial stress.

If you have calculated the shear strength as the strength per unit area, the material strength is in terms of stress. The wall strength in this case is the material strength multiplied by the effective wall area, where the area is specified in Step (4). If you have calculated the shear strength as the strength per unit length of wall, the wall “area” in Step (4) is the effective wall length. If you have calculated the shear strength as the strength of the entire wall, the wall “area” in Step (4) is 1.0.

- (3) Using the **Structure Sections** task and the **Define Sections** tab, define one Structure Section for each side of the wall, at each story where you want to check the shear strength (usually all stories). You may also define a structure section for the entire wall section at each story. You probably will not specify strengths for these entire wall sections, but they can be useful for plotting moment and shear diagrams for the wall.
- (4) Using the Structure Sections task and the Strengths tab, specify a shear strength for each section at which you want to check the strength. Do this by choosing the section and associating it with an Elastic Shear Material.

You must also specify the section “area”. As noted in Step (2) this may be the actual area, the length of shear wall in the section, or a unit value. The section shear strength is the material strength multiplied by the section area. Be careful to use consistent units.

This completes the structure section definition and the strength specification. The next section explains how to set up shear strength limit states.

#### ***9.4 Section Groups and Strength Limit States***

---

Strength limit states for structure sections are specified not in terms of individual sections but in terms of Structure Section Groups (in the same way that limit states for elements are in terms of Element Groups). The steps are as follows.

- (1) Using the **Structure Sections** task and the **Groups** tab, arrange the structure sections in groups. You can put all of the sections in one group if you wish, but it may be more useful to use a number of groups. For example, for the shear core in Figure 9.2 you may have specified structure sections at several stories (possibly all stories), with sections for each side of the wall and for the entire wall at each story. In this case you would probably set up five Structure Section Groups, one for each side of the wall and one for the entire wall. When you set up structure section groups, also consider whether you want to plot bending moment and shear force diagrams, as described in a later section.

- (2) Using the **Limit States** task (see Chapter 10), define one or more limit states of **Struct Sectn** type. The concept is similar to that for element strength limit states that are based on element groups.

Usage ratios are calculated for all Struct Sectn limit states. In Usage Ratio plots, the usage ratios are shown in the same way as for limit states of other types.

## **9.5 Moment, Shear and D/C Ratio Diagrams**

---

### 9.5.1 Procedure

It can be useful to plot bending moment and shear force diagrams for shear walls. Diagrams showing the variation of shear strength D/C ratio over the height of a shear wall can also be useful, because the usage ratio for a shear strength limit state gives the D/C ratio for only the worst structure section covered by the limit state. If you plot D/C variations over the height of a wall you can see which sections are most highly stressed.

The procedure is as follows.

- (1) Use the **Moment and Shear Diagrams** task (see Chapter 27) and the **Section Group** tab.
- (2) Choose the result type and whether to plot envelopes only or time histories. Press **Plot** to begin the plot. Time histories can be animated.

To get a sensible plot, the structure section group must be a sensible group, such as a series of horizontal sections over the height of a wall. The group should not contain multiple sections at any level, otherwise the plot will be strange.

This feature is most likely to be used for horizontal sections in shear walls, but it can be used for other purposes. For example, you can cut horizontal sections through a frame structure and plot story shear and overturning moment envelopes and time histories. You can also cut vertical sections through long span structures such as bridges.

Currently the moment and shear diagrams are plotted on a horizontal axis, even for a group of sections along a vertical line.

### 9.5.2 Save D/C Values to a File

After you have plotted envelope values for D/C ratios, you can save these values to a text file. The “save these results to a file” button on

the toolbar will be activated. To save the D/C ratios, press this button and follow the instructions.

Currently you can save the results only for envelope values and only for D/C ratios.

The most likely application is when you have run analyses for a number of earthquakes and you want to get the maximum or average D/C ratio for each section. You can save a text file for each analysis (and possibly a number of structure section groups), then process the results using a spreadsheet program.

### 9.5.3 Signs and Sign Reversal

For a structure section, the section forces are *on the section after the cut elements are removed*. If you create a horizontal section by cutting the elements at the bottom of a story, the elements above the cut are removed. Hence, a compression force on the section corresponds to a downwards force. However, if you cut the elements at the top of a story, the elements below the cut are removed, and a compression force on the section is now an upwards force.

Usually, a group of structure sections will all be of the same type, with all cuts at the story bottom or all at the story top. Sometimes, however, you may want to put sections of different types in a group. For example, you may cut at the bottom of each story until you reach the roof, then make the final cut at the top of the story. To get correct moment and shear diagrams in this case you must change the signs of the forces on this last section. To do this, when you add this section to the structure section group, check the “Change signs for M,V diagrams” box.

If you forget to check this box, delete the section from the group, and re-add it with the box checked.



## 10 Limit States and Usage Ratios

---

*A nonlinear analysis can produce a huge volume of analysis results. If you make effective use of limit states you can distill the results down to a few usage ratios that can tell you whether the structure satisfies the performance requirements.*

*Capacities, limit states and usage ratios for components are described in Chapter 5, Component Properties. Strength capacities for structure sections are described in Chapter 9, Structure Sections. This chapter explains how to define limit states and how usage ratios are calculated.*

### 10.1 Limit State Types

---

You can specify limit states of the following types.

- (1) Deformation limit states, based on deformation capacities for inelastic components.
- (2) Strength limit states, based on strength capacities for elastic components and strength sections.
- (3) Drift limit states, based on drifts.
- (4) Deflection limit states, based on deflections.
- (5) Shear strength limit states for structure sections.

Limit states are identified by name. To start a new limit state, press the toolbar buttons for the **Modeling** phase and the **Limit States** task, then choose the limit state type from the list, press the **New** button, input the limit state name, and define the limit state conditions as described in the following sections.

Keep an eye on the **Status** box. The **Save**, **Save As**, **Delete** and **UnChange** buttons operate in the same way as for component properties.

You can edit an existing limit state at any time. You can also delete any limit state.

## 10.2 Deformation Limit States

---

A deformation limit state consists of a number of deformation conditions. A deformation condition is defined by the following.

**Element group.** Choose this from the Element Group list.

The condition applies to all elements in the group. This is one reason why you might divide the elements of a given type into several groups.

**Component type.** Choose this from the Component Type list.

By default the list contains only those component types for which at least one component has been specified. If there are no components of a particular type, that type is not listed. You may, however, want to list all possible component types that could be used by the element group. To do this, check the **List all possible types** box. You might do this because you plan to add components later.

Some elements consist of one basic component (e.g., simple bar elements). If so, different elements in the group may have different component types (e.g., some bar elements may be assigned elastic bar components, and some may be assigned inelastic bar components). The limit state condition applies only to those elements that have the specified component type. If you choose a component type and none of the elements in the group has a component of this type, the condition does not apply to any elements, and the demand-capacity ratio for the condition will be zero. PERFORM-3D does not check this, so be careful when you specify limit state conditions.

Other elements consist of compound components (e.g., beam elements). In this case an element can have more than one basic component of a given type (e.g., a beam element can have multiple plastic hinge components). The limit state condition applies to all components of the chosen type. For example, if a beam element has plastic hinge components at its I and J ends, the demand-capacity ratio for the condition is based on the larger of the ratios for these two hinges.



As before, if you choose a component type and none of the elements in the group has a component of this type, the condition does not apply to any elements. Again, PERFORM-3D does not check this.

***Deformation type.*** Select this from the Deformation Type list.

A component typically has only one deformation, but this deformation can be positive or negative. Usually you will choose to consider the deformation regardless of its sign. However, you can choose to consider only positive deformations or only negative deformations. For some components you will have more choices.

***Performance level.*** Select this from the Performance Level list.

The list is always 1 through 5, even if you have defined component capacities at fewer than 5 levels. If you choose, say, Level 4 and none of the components has capacities for that level, the condition will not apply to any elements. PERFORM-3D does not check this. (The reason is that you can edit the component properties at any time, and PERFORM-3D would have to check for consistency each time you make a change. Also, you might want to define a limit state before you specify the component properties.)

***Limit on the demand-capacity ratio.*** Input this value in the box.

Usually this value will be 1.0 (i.e., the usage ratio is 1.0 when the demand equals the capacity for the chosen performance level). You may, however, specify some other value. For example, suppose that when you defined the component properties you specified a deformation capacity of 0.01. You then decide that for this limit state it is justified to increase this capacity by 20%, to 0.012. You can do this by specifying a limit of 1.2 on the demand-capacity ratio for this condition.

At any step in a structural analysis the demand-capacity ratio for a component is the calculated demand divided by the capacity at the specified performance level. The usage ratio for a limit state condition is the demand-capacity ratio for the "worst" component, divided by the limit on the demand-capacity ratio (e.g., divided by 1.2 in the above example). The usage ratio for the limit state as a whole is the largest usage ratio for the "worst" condition.

To add a condition to a limit state, specify the above properties for the condition, then press the **Add** button. You can also **Insert**, **Replace** and **Delete** conditions from a limit state. When you have specified all conditions, press the **Save** button to save the limit state.

You can return to this task to edit an existing limit state at any time. Be sure to keep an eye on the **Status** box, and to press **Save** to save any changes.

### **10.3 Strength Limit States**

---

A strength limit state consists of a number of strength conditions. A strength condition is defined by essentially the same properties as a deformation condition, with the following differences.

- (1) The component may be a basic component or a strength section.
- (2) Instead of the component deformation type you choose the strength type.
- (3) The strength capacity is the nominal strength multiplied by the capacity factor for the chosen performance level.

To add a condition to a limit state, specify the properties for the condition, then press the **Add** button. You can also **Insert**, **Replace** and **Delete** conditions from the limit state. When you have specified all conditions, press the **Save** button.

You can return to edit an existing limit state at any time.

### **10.4 Drift Limit States**

---

A drift limit state is defined by the following.

- (1) A positive drift limit.
- (2) A negative drift limit. Usually the positive and negative limits are the same.
- (3) A set of drifts. This set can include all of the drifts or only selected drifts. Each drift in the set constitutes a drift condition.

Specify the positive and negative limits, and select the **All drifts** or **Highlighted drifts only** option. For the highlighted option, click the drifts that you want to include. Then press the **Save** button.

## **10.5 Deflection Limit States**

---

A deflection limit state is like a drift limit state, except with deflection limits and a set of deflections. Also, the deflection limits have units of length, whereas drifts are dimensionless.

## **10.6 Structure Section Limit States**

---

A structure section limit state is similar to a strength limit state, with the following differences.

- (1) Instead of an element group, choose a structure section group. You must have defined some structure section groups in the Structure Sections task.
- (2) Instead of a component type and a strength type, choose only a strength type. The only strength type currently available is shear strength. You must have defined some structure section shear strengths in the Structure Sections task.

To add a condition to a limit state, specify the properties for the condition, then press the **Add** button. You can also **Insert**, **Replace** and **Delete** conditions from the limit state. When you have specified all conditions, press the **Save** button.

You can return to edit an existing limit state at any time.

## **10.7 Limit State Groups**

---

You must define limit states in the Modeling phase, since the usage ratios are calculated during the structural analysis, not as a post-processing operation. For a large structure you can have many limit

states with different types and performance levels, and hence a large number of usage ratios. In the Analysis phase you can organize the limit states into related groups, to decrease the number of usage ratios and simplify the decision making process.

Often you will organize by performance level. For example, if you have limit states for the Immediate Occupancy, Life Safety and Collapse Prevention levels you might place all IO states in one group, all LS states in a second group, and all CP states in a third group. For any load case you can then quickly determine when the IO performance level is exceeded, by considering only the IO group. The usage ratio for the group is the largest usage ratio for any limit state in the group. The results have thus been reduced to a single usage ratio that indicates whether the structure satisfies all of the IO design criteria. You might also set up a limit state group containing all IO limit states that affect beam elements, and another group containing all IO limit states that affect column elements. You can then quickly see whether the beams or columns are more critical for the IO limit state.

Limit state groups are identified by name. To start a new group, press the toolbar buttons for the **Analysis** phase and the **Limit State Groups** task, press the **New** button, and input the group name. Then choose the limit states for the group using the type and name lists and the **Add**, **Insert**, **Delete** and **Clear** buttons. When you have chosen the limit states, press the **Save** button to save the group.

You can edit an existing group at any time. When you do this, keep an eye on the **Status** box. You can also delete an existing group.

If you change a limit state, this is a change in the analysis model, and you must re-run the analyses. If you add, delete or change a limit state group, you do not have to re-run the analyses.

# 11 Inactive Elements

---

*PERFORM-3D allows you to specify that some elements are inactive for gravity load. This chapter explains the procedure.*

## 11.1 Inactive Elements

---

For gravity load analysis it is assumed, in effect, that the structure is built in a weightless environment and the loads are then applied. Consequently, some members that have small or zero gravity forces in the actual structure (e.g., diagonal braces) may have substantial forces in the analysis model. This may not be a serious error for linear analysis, but it can lead to incorrect behavior for nonlinear analysis. For example, a gravity load analysis may calculate significant compression forces in the elements used to model diagonal braces. These braces may then yield prematurely in subsequent push-over or dynamic earthquake analyses.

It may be more reasonable to assume that such elements have zero forces under gravity load. You can make this assumption by making the elements inactive for gravity load. PERFORM-3D deletes inactive elements at the beginning of the gravity load analysis, and restores them immediately afterwards.

## 11.2 Procedure

---

The only element types that you can make inactive are Simple Bar, Brace, Infill Panel and BRB elements.

To specify inactive elements press the toolbar buttons for the **Modeling** phase and the **Inactive Elements** task, then follow the instructions. Be careful to ensure that the structure without the inactive elements is stable, otherwise the gravity load may cause collapse.



## 12 Gravity Load Cases

---

*You can define and save any number of gravity load cases. You can then use these load cases to run structural analyses.*

*This chapter describes how to set up gravity load cases. For an explanation of how to use these load cases in structural analyses see Chapter 20, Running Analyses.*

### 12.1 Gravity Loads

---

A gravity load case is a combination of nodal, element and/or self weight load patterns. You should usually apply only vertical loads, usually in the downwards (minus V) direction. PERFORM will warn you if a gravity load case has horizontal loads.

To specify a load case you must choose the load patterns and specify a scale factor for each pattern. You must also specify whether the analysis is to be linear or nonlinear.

In most cases you will expect the structure to remain linear and elastic under gravity loads, and hence you will usually specify linear analysis. Some structures can have nonlinear behavior under gravity load even though they remain essentially elastic. For example, a gap may close or a concrete fiber in a shear wall may crack. In this case you must specify nonlinear analysis. If you specify linear analysis and there is nonlinear behavior, you will get an error message during the analysis. If this happens, change the analysis option to nonlinear and re-run the analysis.

Usually you will apply loads that represent actual gravity loads on the structure. However, if you wish you can apply gravity loads that deform the structure well beyond yield, for example to assess the gravity load capacity of a structure. In this case you must choose nonlinear analysis, and specify a load that exceeds the strength of the structure.

## 12.2 Procedure

---

### 12.2.1 Steps

To start a new gravity load case press the toolbar buttons for the **Analysis** phase and the **Load Cases** task. In the Load Cases form choose Gravity as the Load Case Type, press the **New** button, and enter the load case name. Keep an eye on the **Status** box.

The steps are as follows.

Choose the option (linear or nonlinear) for the analysis method.

- (1) If you have chosen nonlinear analysis complete the Control Information for Nonlinear Analysis. You must specify the following. For more details on each of these items see later in this section.
  - (a) Number of equal load steps. See Section 12.2.2.
  - (b) Maximum number of nonlinear events allowed in any step. See Section 12.2.3.
  - (c) Whether to use equal steps throughout, or to have an initial step to the first event then equal steps after that. See Section 12.2.4.
- (2) Specify the load patterns and scale factors. Use the **Add**, **Insert**, **Replace** and **Delete** buttons in the usual way to build up a load pattern list.
- (3) When the load case is complete press the **Save** button to save it. Use the **Delete** button if you want to delete a load case before saving it.

You can edit a saved load case at any time. If you do, keep an eye on the **Status** box. Use the **Save**, **Save As** and **UnChange** buttons in the usual way. You can also delete saved load cases.

### 12.2.2 Number of Load Steps

For linear analysis there is always only one load step. For nonlinear analyses you will usually apply actual gravity loads, and you can specify just one load step. If you want to increase the gravity load until



the structure collapses, a reasonable number of load steps is about 50. The analysis results are saved at the end of each step. For plotting analysis results, 50 steps gives a reasonable number of points along a graph for a collapse load analysis.

### 12.2.3 Maximum Number of Events

PERFORM-3D uses an event-to-event solution strategy, where the structure properties are re-formed each time there is a nonlinear event (a change in stiffness). The program automatically divides each load step into a number of substeps, with a new substep for each event. The analysis stops if the number of events (substeps) in any load step exceeds the specified maximum.

If there is a very large number of events in any load step, this could indicate that the analysis has become numerically unstable and that PERFORM-3D can not find a solution (experience suggests that this is unlikely, but it could happen). In this case you would like the analysis to stop, otherwise it might waste a lot of computer time. On the other hand, if you have a large structure with a lot of nonlinear components you could have a large number of real events in a load step. In this case you do not want the analysis to stop prematurely.

For a structure of moderate size a reasonable maximum for the number of events in any load step is about 200. It is unlikely that there will be that many real events in any load step, so it is unlikely that the analysis will stop prematurely, and if the analysis becomes unstable you will not waste a lot of computer time before the analysis stops.

If a gravity load analysis stops because it reaches the maximum number of events in any load step, this is an abnormal exit. Since the specified load has not been applied, the load case can not be used as a preceding load case for subsequent push-over or dynamic earthquake load cases. You can still process the analysis results, provided at least one load step has been completed.

### 12.2.4 Initial Step to First Event

In a typical structure, first yield does not occur until a substantial part of the collapse load has been applied. In this case it is more efficient to go to first yield in one initial step, then apply equal load increments

until collapse, rather than have several linear steps before first yield. Usually, therefore, you can choose the **Yes** option.

There may, however, be cases where it is better to use equal steps throughout. For example if there are large  $P-\Delta$  effects there can be substantial nonlinear behavior before first yield is reached. Since the analysis assumes linear behavior in the initial step, if you choose the **Yes** option the calculated behavior at first yield can be incorrect.

In most practical cases the  $P-\Delta$  effect at first yield is small. Nevertheless, for analyses that include  $P-\Delta$  effects you may prefer to choose the **No** option to be safe.

If you switch from one option to the other, note that the load step size will change. For example, if the load at first yield is, say, 40% of the total gravity load (i.e., if the load factor at first yield is 0.4), and if you specify 50 steps with the **Yes** option, the first 40% of the load is applied in one step, and the last 60% is applied in 50 equal steps. If you choose the **No** option with the same 50 steps, it takes 20 steps to reach first yield, with only 30 steps for the last 60% of the load.

### 12.2.5 Analysis Limit State

In a nonlinear analysis a point may be reached where the structure is so severely deformed that there is no more useful information to be obtained, and hence there is no reason to continue the analysis. You can stop the analysis by defining a limit state that corresponds to such a severe deformation. This is the *Analysis Limit State*.

To specify the analysis limit state, choose the limit state type and name from the lists.

If you do not specify an analysis limit state, a default state is used. For gravity loads the default is to stop the analysis if the deformation for any inelastic component exceeds its U point deformation. This is based on the assumption that actual gravity loads should never be large enough to load any member beyond its ultimate strength.

For gravity load analysis you will almost always apply the actual gravity load, and expect the structure to remain essentially linear. If there is substantial yielding you have probably made an error and applied too much load. Hence, if a gravity load analysis stops because

the analysis limit state is reached, this is regarded as an abnormal exit. The load case can not be used as the preceding load case for subsequent push-over or dynamic earthquake cases. You can still process the analysis results, provided at least one load step has been completed.



## 13 Static Push-Over Load Cases

---

*You can define and save any number of static push-over load cases. You can then use these load cases to run push-over analyses.*

*This chapter describes how to set up push-over load cases. For an explanation of how to use these load cases in structural analyses see Chapter 20, Running Analyses.*

### 13.1 Push-Over Loads

---

#### 13.1.1 Load Distribution and Magnitude

To perform a static push-over analysis you must specify the distribution of horizontal loads over the structure height. The current version of PERFORM-3D allows fixed distributions only (i.e., you can not consider "adaptive" load distributions that depend on the deflected shape). As explained later you can specify load distributions using nodal load patterns, or you can specify loads based on the structure mass and a deflected shape.

When you specify a push-over load case you need to be concerned only with the load distribution and direction. You do not have to worry about the load magnitude. For the load steps in a push-over analysis the structure is deformed in equal drift increments, and in each step PERFORM-3D calculates the required load increment. The load increment per step typically gets smaller as the structure yields, and can be negative if the structure loses strength or becomes unstable because of P- $\Delta$  effects.

#### 13.1.2 Choice of Load Distribution

One of the most difficult issues for push-over analysis is choosing the push-over load distribution. During an actual earthquake, the effective loads on a structure change continuously in magnitude, distribution and direction. The distribution of story shears over the height of a building can thus change substantially with time, especially for taller buildings where higher modes of vibration can have significant effects. In a static push-over analysis the distribution and direction of the loads are fixed,

and only the magnitude varies. Hence, the distribution of story shears stays constant. To account for different story shear distributions it is necessary to consider a number of different push-over load distributions.

One option in FEMA 356 is to use uniform and triangular distributions over the building height. Note that a uniform distribution usually corresponds to a uniform acceleration over the building height, so that the load at any floor level is proportional to the mass at the floor. Similarly, a triangular distribution usually corresponds to a linearly increasing acceleration over the building height.

PERFORM has the following options for push-over loads.

- (1) Load distributions based on nodal load patterns.
- (2) Load distributions based on masses and a specified displacement (acceleration) variation over the structure height. Examples are linear and triangular variations.
- (3) Load distribution based on masses and the mode shapes of the structure. For more details on this option see the next section.

### 13.1.3 Load Distribution Based on Mode Shapes

Load distributions can be based on the structure mode shapes. For the purposes of discussion consider a 2D structure. The procedure can be extended to 3D structures as shown later in this chapter.

For a low-rise structure that is dominated by its first mode response, a load distribution based on the first mode may be reasonable. For a structure with significant second mode response a possible approach is to specify three load distributions as follows (requiring three push-over analyses).

- (1) *Mode 1*. Use a force distribution based on mode 1.
- (2) *Mode 1 + Mode 2*. Use force distributions based on modes 1 and 2, and set the sign of the mode 2 distribution so that the mode 1 and 2 base shears are in the same direction. This tends to give larger story shears in the lower stories than for mode 1 alone, and smaller shears in the upper stories.

- (3) *Mode 1 - Mode 2*. Use force distributions based on modes 1 and 2, with the mode 1 and 2 base shears in opposite directions. This tends to give smaller story shears in the lower stories than for mode 1 alone, and larger shears in the upper stories.

PERFORM-3D allows you to specify load distributions of this type. The procedure is described later in this chapter.

### 13.1.4 Linear Analysis Option

In the current version of PERFORM-3D you can not specify a linear push-over analysis (the option has been disabled).

If you want to apply a specified lateral load, you can do so using a Gravity load. In this case the analysis can be linear.

## **13.2 Procedure**

---

### 13.2.1 Steps

To start a new push-over load case press the toolbar buttons for the **Analysis** phase and the **Load Cases** task. In the Load Cases form choose Static Push-Over as the Load Case Type, press the **New** button, and enter the load case name. Keep an eye on the **Status** box.

The steps are as follows.

- (1) Choose the option for the analysis method. In the current version this must be Nonlinear Analysis.
- (2) Complete the Control Information for Nonlinear Analysis. You must specify the following. For more details on each of these items see later in this section.
  - (a) Number of load steps. See Section 13.2.3.
  - (b) Maximum number of nonlinear events allowed in any step. See Section 13.2.4.
  - (c) Whether to use equal steps throughout, or to have an initial step to the first event then equal steps after that. See Section 13.2.5.
  - (d) The maximum allowable drift. See Section 13.2.3.

- (e) A limit state to stop the analysis. See Section 13.2.6.
  - (f) The reference drift. See Section 13.2.2.
- (3) Choose the Load Type option (based on nodal load patterns, mode shapes or a displacement pattern).
- (4a) If the load type is based on nodal load patterns, specify the load patterns and scale factors. To push the structure in, say, the +H1 direction specify scale factors that give +H1 loads. To push in the -H1 direction specify scale factors to give -H1 loads. For push-over along an inclined direction, specify load patterns and scale factors to give forces in the required direction. Use the **Add**, **Insert**, **Replace** and **Delete** buttons in the usual way to build up a load pattern list. Only horizontal loads are allowed for push-over loads, and hence you can use only load patterns that have no vertical loads.
- (4b) If the load case type is based on a displacement pattern, specify the displacement pattern over the structure height. For a displacement pattern choose the base and roof levels of the structure, and from zero to 4 intermediate levels, for a total of up to 6 levels. Specify the V coordinate and the H1 and H2 displacements at each level. For a uniform pattern specify the same H displacements at the base and roof levels. For a linear pattern, specify zero H displacement at the base and unit H displacement (or some other non-zero value) at the roof. The V coordinates must be in the structure coordinate system and the structure length unit. The H displacements are relative values only, and do not have units. For each mass in the structure, the H1 and H2 loads are proportional to the mass multiplied by its H1 and H2 displacements.
- (4c) If the load case type is based on mode shapes, specify the push-over direction, the modes to be used, a scale factor for each mode, and a base shear direction for each mode. For an explanation see Section 13.3. Use the **Add**, **Insert**, **Replace** and **Delete** buttons in the usual way to build up a modal load list. Only horizontal translational masses are considered for push-over loads. Vertical translational masses are ignored, and rotational masses do not affect push-over loads.
- (5) Choose the controlled drifts. For an explanation see Section 13.2.3. The default is to use all drifts as controlled drifts. To over-ride this



default choose the **Controlled Drifts** tab and choose the controlled drifts. As a general rule you should use the default. It is not a good idea to run an analysis with only a single controlled drift, especially if there is strength loss.

- (6) When the load case is complete press the **Save** button to save it. Use the **Delete** button if you want to delete a load case before saving it.

You can edit a saved load case at any time. If you do, keep an eye on the **Status** box. Use the **Save**, **Save As** and **UnChange** buttons in the usual way. You can also delete saved load cases.

### 13.2.2 Reference Drift

In push-over plots and other graphs for the results of push-over analyses, the horizontal axis in the plot is reference drift.

FEMA 356 uses the horizontal displacement at the roof as a key measure of building deformation. In PERFORM-3D the equivalent measure is the drift of the roof relative to a supported point at the base. As a general rule you should use this drift as the reference drift. For a Target Displacement Plot the reference drift must be the roof drift.

### 13.2.3 Number of Steps

A reasonable number of steps for a push-over analysis is about 50. The analysis results are saved at the end of each step, and 50 steps gives a sufficient number of points for plotting push-over curves. It is not a good idea to use a large number of steps, such as 500. Not only does this waste computer time, but the analysis can be more sensitive if the load increments are very small.

In Step 2(d) you specify a maximum allowable drift, and in Step 5 you choose the controlled drifts. The allowable drift per load step is the maximum allowable drift divided by the number of steps. In any step the load increment is chosen so that the largest increment for any of the controlled drifts is equal to the allowable drift per step (actually, there is a tolerance and the largest drift increment is not exactly equal to the drift per step). The analysis stops when the largest of the controlled drifts exceeds the maximum allowable drift.

This procedure prevents large increases in any drift in a time step, which could cause inaccuracies or other problems. This is why it is advisable to include all drifts in the set of controlled drifts. If you control the analysis using only the roof drift (i.e., if you choose only one controlled drift), and if the structure has a weak story, it is possible for the (uncontrolled) drift in the weak story to increase rapidly even though the (controlled) roof drift increases slowly (or decreases). This can cause inaccuracies in the solution process.

If you choose only one controlled drift, and if this is the same as the reference drift, then the reference drift will have equal increments in each analysis step. However, if you choose a number of controlled drifts the reference drift generally will not increase in equal increments. This is not a problem – in fact it is often desirable. For example, if the reference drift is the roof drift, and if the structure has a weak story, you do not want the reference drift to have uniform increments. The reference drift should increase more slowly as the weak story yields, and may even decrease in some steps if the weak story loses strength and load is redistributed through the structure.

The reference drift at the end of the analysis may be smaller than the maximum allowable drift, and the actual number of analysis steps may be larger than the specified number. This is because the reference drift is not necessarily the largest drift, and because the controlling drift is not necessarily the same drift in all analysis steps.

As a general rule, we suggest that you use the roof drift as the reference drift, that you use a number of controlled drifts, and that you specify about 50 load steps with a maximum drift that is rather larger than the maximum that you actually want to consider.

### 13.2.4 Maximum Number of Events

PERFORM-3D uses an event-to-event solution strategy, where the structure properties are re-formed each time there is a nonlinear event (a change in stiffness). The program automatically divides each analysis step into a number of substeps, with a new substep for each event. The analysis stops if the number of events (substeps) in any step exceeds the specified maximum.

If there is a very large number of events in any analysis step, this could indicate that the analysis has become numerically unstable and that

PERFORM-3D can not find a solution (experience suggests that this is unlikely, but it could happen). In this case you would like the analysis to stop, otherwise it might waste a lot of computer time. On the other hand, if you have a large structure with a lot of nonlinear components you could have a large number of real events in a step. In this case you do not want the analysis to stop prematurely.

For a moderately large structure a reasonable maximum for the number of events in any analysis step is about 1000. It is unlikely that there will be that many real events in any analysis step, so it is unlikely that the analysis will stop prematurely, and if the analysis becomes unstable you will not waste a lot of computer time before the analysis stops.

If a push-over analysis stops because it reaches the maximum number of events in any load step, this is an abnormal exit. However, the results up to this point will usually be OK.

### 13.2.5 Initial Step to First Event

Often, first yield will not occur until there is a substantial amount of structure drift. In this case it is more efficient to go to first yield in one initial step, then apply equal drift increments until the maximum allowable drift is reached, rather than have several linear steps before first yield. Usually, therefore, you can choose the **Yes** option.

### 13.2.6 Analysis Limit State

In a push-over analysis, a point will usually be reached where the structure is so severely deformed that there is no more useful information to be obtained, and hence no reason to continue the analysis. You can stop the analysis by defining a limit state that corresponds to such a severe deformation. This is the *Analysis Limit State*.

To specify the analysis limit state, choose the limit state type and name from the lists.

If you do not specify an analysis limit state, a default state is used. For push-over analysis the default is to stop the analysis if the deformation for any inelastic component exceeds its X point deformation. This is based on the assumption that the analysis is no longer meaningful if

any element is deformed past its maximum allowable deformation. As a general rule, this is the best choice for the analysis limit state.

### ***13.3 Loads Based on Mode Shapes***

---

#### **13.3.1 Symmetrical Structure**

If you choose loads based on mode shapes, and if you use more than one mode shape, you must specify a scale factor for each mode. Only the relative values of the scale factors are important, since the concern is the load distribution over the building height, not the load magnitude.

Consider first the case of a symmetrical 3D structure with mode shapes that are clearly oriented along the H1 and H2 directions. The following procedure is suggested.

- (1) Get the mode shapes and periods. This usually means that you must run an analysis to calculate the periods before you can define the push-over load case.
- (2) Examine the mode shapes and determine, for each of the H1 and H2 directions, which is the "first" mode and which is the "second" mode. For example, modes 1 and 4 might be the first and second H1 modes, and modes 2 and 5 might be the first and second H2 modes. You may also consider the "third" modes. Ignore torsional modes.
- (3) Choose a design response spectrum. This is a separate task, and you can not do it in PERFORM-3D. Only the shape of the spectrum is important, not the intensity.
- (4) For each mode, get the mode period and hence get the spectral acceleration from the response spectrum. This is the scale factor for the mode.
- (5) Decide on a push-over direction, say H2. Specify the angle to this direction (= 90 degrees for H2).
- (6) Specify the mode number for the "first" mode in the push-over direction (= 2) and the scale factor for this mode from Step (4). Choose "+D" for the base shear direction. Since the structure in

this example is symmetrical, the results will be the same for +D and -D push-over.

- (7) Specify the mode number for the "second" mode in the push-over direction (= 5) and the scale factor for this mode from Step (4). If you want the base shears for modes 1 and 2 to be in the same direction choose "+D" for the direction, else choose "-D".
- (8) If you wish, repeat Step(7) for the "third" mode.

This defines the load distribution and direction for this load case.

You may ask why it is necessary to specify the angle to the push-over direction. For the symmetrical structure in this example, PERFORM-3D could figure out the direction, since both mode 1 and mode 2 have base shears only along H2. Also, since the structure is symmetrical it is necessary to consider push-over only in the +H2 direction. In general, however, it is not that simple. This is considered in the next section.

### 13.3.2 Unsymmetrical Structure or Diagonal Push-Over

There are three reasons why the push-over direction must be specified, as follows.

- (1) For a general structure the base shears for the "first" and "second" modes may not be in exactly the same direction. Hence, the push-over direction can not necessarily be inferred from the modal base shear direction.
- (2) You may want to push the structure in a diagonal direction that is not related to a base shear direction for any particular mode. Again, the push-over direction can not be inferred from the modal base shear directions.
- (3) An unsymmetrical structure may have different behavior for positive and negative push-over. You must define the positive push-over direction.

In the general case it is also more complex to calculate the push-over forces. In the simple example, where the push-over direction is H2 and the modes have base shears only in this direction, the load for each mass is the product of the following values.

- (1) The H2 mass.

- (2) The H2 displacement of the mass for unit modal amplitude.
- (3) The modal (mass) participation factor of the mode for ground acceleration in the H2 direction.
- (4) The spectral acceleration value from the response spectrum.

In the general case there are loads in both the H1 and H2 directions. For each mass the H1 force is the product of the following values.

- (1) The H1 mass.
- (2) The H1 displacement of the mass for unit modal amplitude.
- (3) The modal (or mass) participation factor of the mode for ground acceleration in the specified push-over direction. In the simple example this is zero.
- (4) The spectral acceleration value from the response spectrum.

Similarly, for each mass the H2 force is the product of the following values.

- (1) The H2 mass.
- (2) The H2 displacement of the mass for unit modal amplitude.
- (3) The participation factor of the mode for ground acceleration in the push-over direction.
- (4) The spectral acceleration value from the response spectrum.

In general there will be both H1 and H2 loads at each mass point. For equal H1 and H2 masses (which is required in PERFORM-3D), the resultant force at any mass point is along the push-over direction. Hence the resultant base shear is also along the push-over direction.

### 13.3.3 Other Key Points

The following are two other key points.

- (1) Be sure to choose a reference drift that is along the push-over direction. In the above simple example with H2 forces, if you choose a reference drift that is in the H1 direction the reference drift will be zero.
- (2) Be sure that at least one of the controlled drifts is along the push-over direction. If not, the specified load will not cause any drifts and the analysis will stop. It is usually wise to have several controlled drifts, along both directions.







## 14 Dynamic Earthquake Load Cases

---

*You can define and save any number of dynamic earthquake load cases. You can then use these load cases to run structural analyses.*

*This chapter describes how to set up earthquake load cases. For an explanation of how to use these load cases in structural analyses see Chapter 20, Running Analyses.*

### 14.1 Earthquake Loads

---

For dynamic step-by-step analysis the earthquake load is defined by H1 and/or H2 and/or V ground acceleration records (ground accelerograms). Experience has shown that nonlinear analysis results can be sensitive to relatively small changes in the ground motion. Hence, it is usually necessary to run analyses for a number of records. It is not usually necessary to consider vertical ground accelerations.

### 14.2 Procedure

---

#### 14.2.1 Steps

To start a new earthquake load case press the toolbar buttons for the **Analysis** phase and the **Load Cases** task. In the Load Cases form choose Dynamic Earthquake as the Load Case Type, press the **New** button, and enter the load case name. Keep an eye on the **Status** box.

The steps are as follows.

- (1) If you have not already set up the needed earthquake records (ground accelerograms) press the **Add/Review/Delete Earthquakes** button. See *Chapter 15, Earthquake Records* for instructions on how to set up earthquake records.
- (2) Complete the Control Information for Dynamic Analysis. You must specify the following. For more details on these items see later in this section.

- (a) Total time. See Section 14.2.2.
  - (b) Time step. See Section 14.2.3.
  - (c) Maximum number of nonlinear events allowed in any step. See Section 14.2.4.
  - (c) Interval for saving results. See Section 14.2.5.
  - (d) A limit state to stop the analysis. See Section 14.2.6.
  - (e) The reference drift. For processing the analysis results, PERFORM-3D uses the reference drift as an overall measure of the structure response. As a general rule you should use the drift at the roof as the reference drift.
- (3) Specify the angle from the structure H1 axis to the earthquake Q1 direction. This allows you to apply an earthquake in any horizontal direction.
- (4) Choose the Q1 Earthquake, if any, from the list of earthquake records. The peak acceleration and duration for the record will be shown in the boxes. Also specify acceleration and time scale factors. The accelerations and times in the earthquake record are multiplied by these factors. The time factor should usually be 1.0. You can plot the earthquake record before you specify the scale factors (See *Chapter 15, Earthquake Records*), but to plot it after scaling you must run an analysis and use the Time History task. Hence, take care with the scale factors.
- (5) Similarly choose the Q2 and V Earthquakes, if any. The default is none.
- (6) Press the **Save** button to save the load case. Use the **Delete** button if you want to delete a load case before saving it.

You can edit a saved load case at any time. If you do, keep an eye on the **Status** box. Use the **Save**, **Save As** and **UnChange** buttons in the usual way. You can also delete saved load cases.

### 14.2.2 Total Time

The total time is the duration of the earthquake for the analysis. This may be longer or shorter than the length of the earthquake record. If it is longer the record is padded with zeros. If it is shorter only a part of the record is used.

### 14.2.3 Time Step

The analysis is performed using step-by-step integration through time, using the constant average acceleration method (also known as the trapezoidal rule or the Newmark  $\beta = 1/4$  method). You must specify the integration time step. The number of steps is the total time divided by the time step, unless the analysis terminates before the end of the earthquake.

Since dynamic earthquake analyses can be time consuming, it is natural to want to use as large a time step as possible. There are two things to keep in mind when choosing a time step, as follows.

- (1) You must accurately capture the structure response. For a linear single degree-of-freedom system, the constant average acceleration method gives sufficiently accurate results for most practical purposes if the time step is smaller than about 1/12 th of the structure period. For a linear multi-degree-of-freedom structure, one method for choosing the time step is to identify the highest mode that contributes significantly to the response, and use a time step equal to 1/12th of the period for that mode. Since most structures increase in period as they yield, a time step based on linear behavior should usually be short enough for calculating inelastic response.
- (2) You must accurately capture the ground motion. For example, if the earthquake record is at 0.02 second intervals, you should not use a time step larger than 0.02 seconds, even for a very long period structure, otherwise you risk distorting the ground motion.

You may need to experiment by progressively reducing the time step until there are no significant changes in the analysis results.

Note that you may not save much time by increasing the time step. PERFORM-3D uses an event-to-event solution strategy, where the structure properties are re-formed each time there is a nonlinear event (a change in stiffness). The program automatically divides each time step into a number of substeps, with a new substep for each event. Increasing the time step has no effect on the total number of substeps, and hence may not have much effect on the total computation time.

#### 14.2.4 Maximum Number of Events

As noted, PERFORM-3D uses an event-to-event solution strategy. The analysis stops if the number of events (substeps) in any step exceeds the specified maximum.

If there is a very large number of events in any time step, this could indicate that the analysis has become numerically unstable and that PERFORM-3D can not find a solution (experience suggests that this is unlikely, but it could happen). In this case you would like the analysis to stop, otherwise it might waste a lot of computer time. On the other hand, if you have a large structure with a lot of nonlinear components you could have a large number of real events in a step. In this case you do not want the analysis to stop prematurely.

For a structure of moderate size, a reasonable maximum for the number of events in any time step is about 200. It is unlikely that there will be that many real events in any time step, so it is unlikely that the analysis will stop prematurely, and if the analysis becomes unstable you will not waste a lot of computer time before the analysis stops.

If an analysis stops for this reason it is an abnormal exit, but the results up to that point will usually be OK.

#### 14.2.5 Save Interval

By default the analysis results are saved every time step. If you wish, you can reduce the lengths of the results files by saving the results every  $N$  steps, where  $N > 1$ .

If you do not save the results every step, you may lose detail in time history results plots. This does not affect the values of demand-capacity ratios and usage ratios. These are calculated at every step (actually, every substep).

#### 14.2.6 Analysis Limit State

During the analysis a point may be reached where the structure is so severely deformed that there is no more useful information to be obtained, and hence no reason to continue the analysis. You can stop the analysis by defining a limit state that corresponds to such a severe deformation. This is the *Analysis Limit State*.

To specify the analysis limit state, choose the limit state type and name from the lists.

If you do not specify an analysis limit state, a default state is used. For dynamic earthquake analysis the default is to stop the analysis if the deformation for any inelastic component exceeds its X point deformation. This is based on the assumption that the analysis is no longer meaningful if any element is deformed past its maximum allowable deformation. As a general rule, this is the best choice for the analysis limit state.



## 15 Earthquake Records

---

*You can set up and store any number of earthquake records. You can then use any record to define an earthquake load case. This chapter describes how to set up earthquake records.*

### 15.1 Organization

---

#### 15.1.1 Overall Steps

The overall steps for setting up a new earthquake record are as follows.

- (1) Obtain a text (formatted) file containing the earthquake ground accelerations. This file can have a variety of formats, as explained later in this chapter.
- (2) Decide on a group and name for the earthquake record. This is explained later.
- (3) Convert the ground accelerations file into an earthquake record. This is explained later.

You can then use the record in any earthquake load case, for the current structure or any other structure.

#### 15.1.2 Organization of Earthquake Records

PERFORM-3D is installed, by default, in the following folder.

*C:\ProgramFiles\Computers and Structures\PERFORM\PERFORM-3D\Program*

During installation you may have specified a different folder.

The first time that you execute PERFORM-3D it creates a *Records* folder (if the program is distributed with example earthquake records, the Records folder is created when the program is installed). If you installed the program in the default location, the Records folder is :

*C:\ProgramFiles\Computers and Structures\PERFORM\Records*

Note that the parent folder is *PERFORM*, not *PERFORM-3D*. This is done so that the earthquake records are available to other programs in the *PERFORM* series.

All earthquake records are (and must be) placed in the Records folder. If they are not, *PERFORM-3D* will not be able to find them.

Within the Records folder you can organize the earthquake records into related groups, where the records in any group are in a subfolder of the Records folder. Since you can have a large number of records, it is a good idea to organize them in groups.

Each time you add a new earthquake record you must specify the group name (max. 12 characters). If you specify a new group, *PERFORM-3D* creates a record group folder of the same name, as a subfolder of the Records folder. For example, if you name the group "Group-A", the default location of the folder for this group is :

*C:\ProgramFiles\Computers and Structures\PERFORM\Records\Group-A*

Each time you add an earthquake record to a group you must specify a file name (max. 12 characters) and a record name (max. 40 characters). The earthquake record is stored as a file in the folder for the group, with the specified file name. For example, if you name the earthquake record "Eqke-A" and put it in Group-A, the default location of the file is :

*C:\ProgramFiles\Computers and Structures\PERFORM\Records\Group-A\Eqke-A*

When you use an earthquake record in an earthquake load case you must identify the record by its group name and record name (see the preceding chapter). You do not use the file name for this purpose. The file name is important only if you need to copy earthquake records from one computer to another.

Once you have set up a record group folder and the files for a number of earthquake records in the folder, do not change the folder or file names. If you do, *PERFORM-3D* may not be able to find the earthquake record.



### 15.1.3 Copying Earthquake Records

If you have PERFORM-3D installed on a number of computers, you may wish to set up a number of earthquake records on one computer, using the procedure described in the next section, then copy these records to another computer, rather than go through the set-up procedure again.

You can copy any of the following. If the *Records* folder does not exist on the second computer, you may need to create it.

- (1) The complete *Records* folder.
- (2) The folder for any record group.
- (3) The file for any record.

## ***15.2 Setting Up a New Earthquake Record***

---

### 15.2.1 Text File Format

The ground accelerations file must be a text file. It can have a number of header lines before the accelerations begin. The rest of the file must then contain one of the following.

- (1) Accelerations only. There must be a fixed number of values per line at a constant time interval.
- (2) Time-Acceleration pairs. There must be a fixed number of pairs per line.
- (3) Acceleration-Time pairs.
- (4) Acceleration-Velocity-Displacement sets. There must be a fixed number of sets per line at a constant time interval.
- (5) Time-Acceleration-Velocity-Displacement sets. There must be a fixed number of sets per line.
- (6) Acceleration-Velocity-Displacement-Time sets.

The values in any line can be separated by spaces or commas.

The file can be on a diskette or CD, or it can be on your hard drive.

### 15.2.2 Adding A New Record

To read a ground accelerations text file and set up a corresponding earthquake record, press the toolbar buttons for the **Analysis** phase and the **Load Cases** task, choose Dynamic Earthquake for the Load Case Type, and press the **Add/Review/Delete Earthquakes** button. You can then add one or more new earthquake records, review (plot) any existing record, and delete any existing record.

The steps to add a new record are as follows.

- (1) Enter the name of the ground accelerations text file. If you have copied this file to the *User* folder you need to enter only the file name. Otherwise you must give the complete path to the file. The file can be on a diskette or CD if you wish.
- (2) Choose the file contents type from the list.
- (3) If the contents type is Accelerations only or Accn-Veloc-Displ sets, specify the time interval and whether the first value or set of values is at time step 0 or time step 1. If in doubt specify time step 1, and the initial value will be set to zero.
- (4) Choose the acceleration unit from the list.
- (5) Specify the duration to be used for the resulting earthquake record. This may be longer or shorter than the maximum duration in the file. If it is longer the resulting earthquake record is padded with zeros. If it is shorter only a part of the file is read.
- (6) Specify the number of acceleration values per line (or the number of acceleration-velocity-displacement sets).
- (7) Specify the number of header lines to be skipped before the beginning of the acceleration data.
- (8) Specify whether the values are separated by spaces or commas.
- (9) If the record is to be placed in an existing record group, choose the name of the group from the list. To create a new group, press the **New** button and specify the group name.

- (10) Specify a file name and a record name for the earthquake record. Choose the record name carefully because you must use it to identify the record when you review it or use it in a load case.
- (11) To read the file and set up the earthquake record press the **Read File** button. If the file is read successfully the earthquake record will be plotted. If the record is OK press **Save** to save the record, else press **Cancel**, revise the above data, and try again.

When you have set up all of the needed records, press **Return to Earthquake Load Case**.

### ***15.3 Reviewing Existing Records***

---

To review any record go to the Load Cases task, choose Dynamic Earthquake for the Load Case Type, press the **Add/Review/Delete Earthquakes** button, and choose the group and record names from the lists in the Review/Delete Earthquakes section. To review the record press **Review**. To delete it press **Delete**.



## 16 Dynamic Force Load Cases and Analysis

---

*This chapter describes the procedures for setting up Dynamic Force Load Cases. Dynamic force analysis calculates the response of the structure to a dynamically varying force, such as wind force or blast load.*

*This chapter also shows how dynamic force analysis can be used for earthquake analyses of long-span structures with multi-support excitation, where different support points have different ground motions. This is, however, an indirect method and it is an interim solution. A better and more direct method is to impose dynamic displacements using support spring elements with initial deformations. This option is currently available only for static imposed displacements. It will be extended to dynamic imposed displacements in a future version.*

### 16.1 Purpose and Overall Procedure

---

#### 16.1.1 Purpose

A dynamic force analysis calculates the response of the structure to a dynamically varying force, such as wind force or blast load. This type of analysis can also be used for dynamic earthquake analysis where different supports for the structure have different ground motions (i.e., for earthquake analysis with multi-support excitation).

#### 16.1.2 Overall Procedure

A dynamic force load has two parts, as follows.

- (1) A nodal load pattern. This is defined in exactly the same way as a nodal load pattern for gravity or static push-over analysis. It defines the spatial distribution of the load, as nodal loads of specified magnitudes.
- (2) A dynamic force record. This defines the time variation of the loads. A dynamic force record is actually a time history of load factors that are to be applied to the nodal load pattern. It is set up in much the same way as an earthquake ground acceleration record. However, it defines a load factor history rather than a ground acceleration history.

The overall steps for setting up a dynamic force load case are as follows.

- (1) Define the required nodal load patterns.
- (2) Set up the required dynamic force records.
- (3) Define one or more dynamic force load cases. In any load case you can apply up to 40 load patterns, each with a corresponding force record. Hence, if you use a dynamic force load case for an earthquake analysis with multi-support excitation, you can apply up to 40 different motions.

## **16.2 Organization of Dynamic Force Records**

---

### 16.2.1 RecordsF Folder

PERFORM-3D is installed, by default, in the following folder.

*C:\ProgramFiles\Computers and Structures\PERFORM\PERFORM-3D\Program*

During installation you may have specified a different folder.

The first time that you execute PERFORM-3D it creates a *RecordsF* folder (if the program is distributed with example dynamic force records, the *RecordsF* folder is created when the program is installed). If you installed the program in the default location, the *RecordsF* folder is:

*C:\ProgramFiles\Computers and Structures\PERFORM\RecordsF*

Note that the parent folder is *PERFORM*, not *PERFORM-3D*. This is done so that the dynamic force records are available to other programs in the PERFORM series.

All dynamic force records must be placed in the *RecordsF* folder. If they are not, PERFORM-3D will not be able to find them.

Within the *RecordsF* folder you can organize the records into related groups, where the records in any group are in a subfolder of the *RecordsF* folder. If you have a large number of records, it is a good idea to organize them in groups.

Each time you add a new dynamic force record you must specify the group name (max. 12 characters). If you specify a new group, PERFORM-3D creates a folder for the group, as a subfolder of the *RecordsF* folder. For

example, if you name the group "Group-A", the default location of the folder for this group is:

*C:\ProgramFiles\Computers and Structures\PERFORM\RecordsF\Group-A*

Each time you add a dynamic force record to a group you must specify a file name (max. 12 characters) and a record name (max. 40 characters). The record is stored as a file in the folder for the group, with the specified file name. For example, if you name the record "Wind-A" and put it in Group-A, the default location of the file is:

*C:\ProgramFiles\Computers and Structures\PERFORM\RecordsF\Group-A\Wind-A*

When you use a dynamic force record in an dynamic force load case you must identify the record by its group name and record name. You do not use the file name for this purpose. The file name is important only if you need to copy records from one computer to another.

Once you have set up a record group folder and the files for a number of records in the folder, do not change the folder or file names. If you do, PERFORM-3D may not be able to find the records.

### 16.2.2 Steps for Setting Up Dynamic Force Records

The steps for setting up a new dynamic force record are as follows.

- (1) Set up a text (formatted) file containing the record. This file can have a variety of formats, as explained later in this chapter.
- (2) Decide on a group and name for the record, as explained in the preceding section.
- (3) Convert the text file into a PERFORM force record file. This is explained later.

You can then use the record in any dynamic force load case, for the current structure or any other structure, and for PERFORM-COLLAPSE as well as PERFORM-3D.

### 16.2.3 Copying Dynamic Force Records

If you have PERFORM-3D installed on a number of computers, you may wish to set up a number of dynamic force records on one computer, then copy these records to another computer, rather than go through the set-up procedure again.

You can copy any of the following. .

- (1) The complete *RecordsF* folder.
- (2) The folder for any record group.
- (3) The file for any record.

## ***16.3 Setting Up a New Dynamic Force Record***

---

### 16.3.1 Text File Format

The dynamic force file must be a text file. It can have a number of header lines before the force values (load factors) begin. The rest of the file must then contain one of the following.

- (1) Forces only. There must be a fixed number of values per line at a constant time interval.
- (2) Time-Force pairs. There must be a fixed number of pairs per line.
- (3) Force-Time pairs. There must be a fixed number of pairs per line.

The values in any line can be separated by spaces or commas, but not by tabs.

The file can be on a diskette or CD, or it can be on your hard drive.

### 16.3.2 Adding a New Record

To read a dynamic force text file and set up a corresponding dynamic force record, press the toolbar buttons for the **Analysis** phase and the **Load Cases** task, choose **Dynamic Force** for the Load Case Type, and press the **Add/Review/Delete Force Records** button. You can add one or more new records, review (plot) an existing record, and/or delete an existing record.

The steps to add a new record are as follows.

- (1) Enter the name of the dynamic force text file. If you have copied this file to the *User* folder you need to enter only the file name. Otherwise you must give the complete path to the file. You can browse to find the file (press the **Browse** button). The file can be on a diskette or CD if you wish.
- (2) Choose the file contents type from the list (Forces Only, Time-Force pairs, or Force-Time pairs).



- (3) If the contents type is Forces Only, specify the time interval.
- (4) Specify the duration to be used for the imported record. This may be longer or shorter than the maximum duration in the file. If it is longer the imported record is padded with zeros. If it is shorter only a part of the file is read.
- (5) Specify the number of force values per line.
- (6) Specify the number of header lines to be skipped before the beginning of the force data.
- (7) Specify whether the values are separated by spaces or commas.
- (8) If the record is to be placed in an existing record group, choose the name of the group from the list. To create a new group, press the **New** button and specify the group name.
- (9) Specify a file name and a record name. Choose the record name carefully because you must use it to identify the record when you review it or use it in a load case.
- (10) To read the text file and set up the record press the **Check** button. If the file is read successfully the record will be plotted. If the record is OK press **Save** to save the record, else press **Cancel**, revise the above data, and try again.

When you have set up all of the needed records, press **Return to Dynamic Force Load Case**.

Note that you do not specify the force unit. The reason is that a dynamic force record is a series of load factors, not actual forces. When you specify a dynamic force load case you must associate each record with a nodal load pattern. The actual forces at any time in the record are the load pattern forces multiplied by the current dynamic force (load factor). The time unit is always seconds.

## **16.4 Reviewing Existing Records**

---

To review a record go to the Load Cases task, choose Dynamic Force for the Load Case Type, press the **Add/Review/Delete Force Records** button,

and choose the group and file names from the lists in the Review or Delete section. To review the record press **Review**. To delete it press **Delete**.

## 16.5 Dynamic Force Load Cases

---

### 16.5.1 Load Cases

To start a new dynamic force load case press the toolbar buttons for the **Analysis** phase and the **Load Cases** task. In the Load Cases form choose **Dynamic Force** for the Load Case Type, press the **New** button, and enter the load case name. Keep an eye on the **Status** box.

The steps are as follows.

- (1) If you have not already set up the needed dynamic force records, press the **Add/Review/Delete Force Records** button and add one or more records as described earlier.
- (2) Specify the Control Information for Dynamic Analysis. This is essentially the same as for a Dynamic Earthquake load case (*see Chapter 14, Dynamic Earthquake Load Cases, Section 14.2*). For the Reference Drift or Reference Deflection, choose a drift or deflection that is excited by the loading.
- (3) Specify the load list, up to a maximum of 40 loads. For each load specify a combination of a Nodal Load Pattern and a Dynamic Force Record. You can also specify a scale factor and a time delay. If the time delay is not zero, the dynamic force record is delayed by that amount until it begins to be applied (in effect, zero load factors are added to the beginning of the record). Create the list using Add, Insert, Replace and Delete in the usual way.
- (4) Press the **Save** button to save the load case. Use the **Delete** button if you want to delete a load case before saving it.

You can edit a saved load case at any time. If you do, keep an eye on the **Status** box. Use the **Save**, **Save As** and **UnChange** buttons in the usual way. You can also delete saved load cases.

## 16.6 Multi-Support Earthquake Analysis

---

*Note: We believe that the method described in this section is reasonable for this type of analysis. However, we are continuing to study the details and to make improvements. If you plan to use this method, we recommend that you contact us to get the latest developments.*

### 16.6.1 Purpose

During an earthquake the ground displaces dynamically. The ground displacements, velocities and accelerations can all be important in determining the response of a structure.

When a structure and the adjacent ground are instrumented, the accelerations are measured directly, and the velocities and displacements are usually obtained by integration. For most building structures, it is reasonable to assume that the ground motions are the same at all support points. For long structures such as bridges, however, different support points may have substantially different ground motions. This will be referred to here as “multi-support excitation”.

Using PERFORM-3D dynamic force analysis, it is possible to run analyses for multi-support excitation. This section describes the procedure. The method is indirect, and it has some weaknesses as noted in this section. It is provided as a temporary method. A more direct method, based on dynamic imposed support displacements, may be added in a future version.

### 16.6.2 Analysis Method

For earthquake analysis where all supports have the same ground motions, it is usual to use the earthquake ground acceleration record to calculate time histories of the effective inertia forces on the structure. The analysis is then carried out for these forces. In this type of analysis the displacements at the support points are zero, and hence the calculated displacements, velocities and accelerations at the nodes are relative to the ground. To get the absolute accelerations at any node (which are needed to calculate the “floor” response spectrum), the ground accelerations must be added to the calculated relative accelerations.

For linear structural analysis with multi-support excitation, it is usual to use a similar analysis method. In this method the ground accelerations at the support points are used to calculate effective inertia forces, and an analysis is carried out for these forces. This analysis assumes fixed supports, so the displacements at the support points are zero. This is not correct, since different supports can have different displacements, and hence the support

points can displace relative to each other. This effect is accounted for by a second analysis in which the relative support displacements are considered. The results of the two analyses are added together. This approach can be used for linear time history analysis and also for response spectrum analysis.

The time history version of this method can be extended to nonlinear time history analysis. However, it is complex. A simpler method is to impose dynamic support displacements on the structure, where the displacement time history at any support is the earthquake ground displacement record. This method is used in PERFORM-3D.

In this method, ground displacements are applied in the analysis, not ground accelerations. One consequence of this is that the calculated node displacements during the analysis are absolute displacements, not displacements relative to the ground. Also, the ground accelerations follow from the ground displacements by double differentiation, which can lead to a loss of accuracy. These aspects are considered later in this section.

### 16.6.3 Imposed Dynamic Support Displacements

PERFORM-3D currently does not provide the ability to apply dynamic support displacements directly (this will be added in a future version). Support displacements can, however, be imposed as follows.

- (1) At each node where a ground displacement is to be imposed, add a Support Spring element. This element must be very stiff relative to the structure stiffness, but avoid astronomically large stiffnesses. Put these elements in one element group (this is not essential but it is usually a good idea), and specify zero for the Scale Factor for Beta-K Damping.

It is unlikely that you will want to impose rotational displacements. Hence, you probably do not need rotational support springs. It is usually easier to specify translational spring stiffnesses only, and to provide rotational support using rigid restraints.

- (2) For each different ground displacement record, set up a nodal load pattern. In each pattern, specify a large load at each affected node, in the direction of the required displacement. If the stiffness of the support spring element along a displacement direction is  $K$ , specify a load with a magnitude equal to  $K$ . This will cause a unit displacement at the node (actually slightly less, but if  $K$  is large enough the difference is negligible).

- (3) For each different ground displacement record set up a Dynamic Force Record that defines the ground displacement time history. See the next section for a method to get ground displacement records. Read the records into PERFORM as described earlier in this Chapter.

Each dynamic force record is actually a time history of load factors that are to be applied to a load pattern. If the forces in the load pattern cause unit displacements, a unit load factor in the dynamic force record corresponds to a unit displacement. Hence, the values in any force record are numerically equal to the ground displacements for that record.

- (4) Define one or more dynamic force load cases as described earlier in this Chapter. For each imposed displacement there will be a load pattern and an associated dynamic force record. If you wish you can specify a scale factor to adjust the displacement magnitude. If the load pattern causes a unit displacement, this factor will be 1.0.
- (5) Run analyses with the load cases.

#### 16.6.4 Ground Displacements

Displacement records are usually not measured directly, but are obtained from the ground acceleration records by double integration through time. A displacement record obtained by double integration can be sensitive to small errors in the acceleration record, and can show drift over the duration of the record. That is, even if the actual ground displacement at the end of the earthquake is zero, the displacement obtained from the acceleration record can show a non-zero final value. It is usual (and usually necessary) to correct for this. It is typically assumed that the displacement error varies quadratically during the earthquake. Hence, the displacement record is typically corrected to a base line that varies quadratically over the duration of the record. After this correction the acceleration and displacement records are no longer completely consistent (unless they are also corrected), but any inconsistency is small.

Commonly, earthquake acceleration records are discretized at time intervals of 0.01 or 0.02 seconds, which means that information on high frequency ground accelerations is not captured accurately. Time history analyses in PERFORM-3D use the Constant Average Acceleration (CAA) Method of step-by step integration (also known as the Trapezoidal Rule or the Newmark  $\beta = 1/4$  Method). This method assumes that the acceleration is constant in each time step, equal to the average of the values at the beginning and end of the step. Hence the velocity is assumed to vary linearly in a time step. Theoretically the displacement varies quadratically,

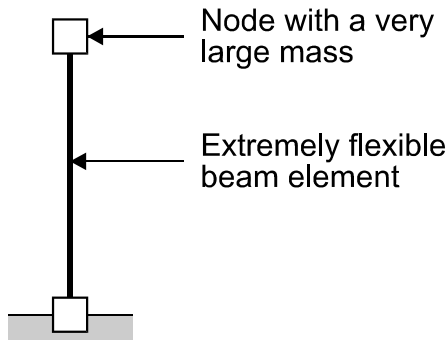
but if there are events in a step they are calculated assuming a linear displacement variation.

The CAA method is the simplest integration method, and arguably the best for nonlinear analysis. Like all integration methods it is approximate. In particular, it can calculate inaccurate response for vibration modes with periods that are shorter than about 12 times the time step. For a time step of 0.02 seconds this means that the calculated response for any mode with a period shorter than about 0.25 seconds can be inaccurate. For stiff structures a time step shorter than 0.02 seconds should be used. Note, however, that earthquake acceleration records are usually discretized at 0.02 second intervals, and that when a time step shorter than 0.02 seconds is used, the accelerations are interpolated linearly. This is not strictly correct, so the discretization of the acceleration record can cause inaccuracy for short period structures.

### 16.6.5 Ground Displacement Records

If an earthquake ground displacement record is available it will typically be discretized at 0.01 or 0.02 second time intervals. For time history analysis it is usually assumed that the displacements vary linearly in each interval. This means that the implied ground accelerations (which, in effect, are now obtained by double differentiation of the displacement record) can be substantially inaccurate for high frequency vibrations. Fortunately, structures that have multi-support excitations are always large structures such as bridges, where the important vibrations tend to have longer periods.

If you have only a ground acceleration record, you must double integrate it to get the ground displacement record. You can do this directly, for example using a spreadsheet program. Alternatively you can use a simple PERFORM structure that models a displacement gage. This structure is shown in Figure 16.1.



### Figure 16.1 Displacement Gage for Converting Ground Accelerations to Ground Displacements

To get the ground displacements you can analyze this structure using a dynamic earthquake load case, and plot the relative displacement of the mass. Since the mass is very large it does not move in space, so its displacement relative to the ground is minus the ground displacement. Save the displacement time history to a text file, and if necessary apply a baseline correction using a spreadsheet program. Then read the time history into PERFORM-3D as explained earlier in this Chapter.

If you double integrate an acceleration record, you must choose an integration method and an integration time step. If you analyze the structure in Figure 16.1, the integration method is the CAA method in PERFORM, but you must still choose a time step. The following method is suggested.

- (1) Decide on the time step that you will use for the dynamic force analyses of the structure (say 0.02 seconds).

The ground acceleration record will usually be at constant time intervals, often 0.02 seconds. Since structures that have multi-support excitation will tend to be larger structures with longer periods, the time step will usually be this value. As a general rule, do not use a time step that is larger than this value, since it may not capture the ground motion accurately. If you use a time step that is smaller, make it a whole number subdivision (e.g., for an acceleration record at 0.02 second intervals, use time steps such as 0.02, 0.01, 0.005, 0.002 seconds).

- (2) Use this time step to analyze the displacement gage in Figure 16.1, and output the displacements at every time step. This is the displacement record.

This method is suggested because it should be the most consistent. For the case where all supports move in phase, this method should give the same ground motions on the structure whether you run (a) a dynamic force analysis with the calculated ground displacement record or (b) a dynamic earthquake analysis using the ground acceleration record.

### 16.6.6 Alpha-M Damping

“Alpha-M” damping implies that there is a linear viscous damper that connects each mass to a fixed point. For dynamic earthquake analysis, these fixed points are fixed to the ground, and the deformation rates of the viscous dampers depend on the velocities of the nodes relative to the ground. For a dynamic force analysis, however, the points are fixed in space, while the ground moves, and the deformation rates of the viscous dampers depend on the absolute velocities of the nodes. Hence, Alpha-M damping has a different physical meaning for the two types of analysis. “Beta-K” viscous damping depends on element deformations (i.e., relative displacements), so this damping is not affected.

It is not clear whether this has much effect. In PERFORM-3D, one way to assess the effect of viscous damping is to plot the energy balance and examine the amount of energy dissipated by viscous damping.

Unfortunately, a weakness of using force time-history analysis for earthquake excitation is that large deformations are imposed on the stiff support spring elements, and hence the strain energy in these elements is very large. When you plot an energy balance graph, you will find that virtually all of the energy is strain energy in the support springs, and you can not tell how much energy is dissipated by viscous damping. You can still see the relative amounts of energy dissipated by inelastic hysteresis in the element groups. Also, you can get the amounts of  $\alpha M$  and  $\beta K$  damping from the ECHO.txt file (from the printed energy balance at the end of each analysis). However, this is not as convenient as using an energy balance graph.

### 16.6.7 Results Interpretation

The results interpretation is similar to that for a dynamic earthquake analysis. However, please note the following two aspects.

- (1) As noted in the preceding section, you can not get an overall energy balance, because the strain energy in the support springs is much larger than the other energies.
- (2) If you plot a time history of displacement at a support point, you will find that it closely matches the input displacement (assuming the support spring is sufficiently stiff). However, if you plot a velocity time history you may find that it is more jagged than expected, and if you plot an acceleration time history you may find that it is very jagged. The acceleration time history should, of course, match the original ground acceleration record. You may find that it does this fairly well for the first second or two of the response, but then



becomes very jagged with large oscillating values. We think that this is numerical precision problem, and we are looking into improvements. We do not think that it affects the overall results.

Note that node displacements, velocities and accelerations are absolute values. However, drifts are still relative.



## 17 Analysis Series

---

*When you have set up an analysis model and some load cases you are ready to run structural analyses. Before you do this you must set up one or more Analysis Series. This chapter explains the procedure.*

### 17.1 Analysis Series Concept

---

#### 17.1.1 Concept

At this stage you have set up the structure and the load cases. An "Analysis Series" allows you to specify certain other parameters before you run any analyses. Each analysis series corresponds to a different analysis model, so that you can analyze several different models without creating a complete new structure. For example, you may want to run one set of analyses ignoring P- $\Delta$  effects and a second set considering these effects. To do this, you can set up one analysis series with P- $\Delta$  effects turned on, and a second series with these effects turned off. You can also specify different masses, damping and certain other parameters.

You can have as many analysis series as you like. You will usually run several analyses for each series. For the steps in running analyses, see *Chapter 20, Running Analyses*.

#### 17.1.2 Procedure

Figure 17.1 shows the form for specifying Analysis Series parameters.

Figure 17.1 Form for Analysis Series Parameters

The steps for starting a new analysis series are as follows.

- (1) Press the toolbar buttons for the **Analysis** phase and the **Run Analyses** task.
- (2) Choose the **Start a new analysis series** option. The other options are to continue or delete an existing series, as explained later.
- (3) Input the series name (choose this carefully) and description.

You can specify the following parameters.

- (1) Type of load sequence (Standard or General).
- (2) Whether or not P- $\Delta$  effects are to be considered.
- (3) Masses.
- (4) Number of mode shapes to be calculated.
- (5) Modal damping.
- (6) Rayleigh ( $\alpha M + \beta K$ ) damping.
- (7) Upper/lower bound ratios.
- (8) Certain options for simplified analysis.

See *Chapter 18, Elastic Viscous Damping* for an explanation of Modal and Rayleigh damping. See the later sections in this chapter for explanations of the other parameters.

After you have specified the properties for the Analysis Series, press **OK** to save the properties and move to the **Run Analyses** form. See *Chapter 20, Running Analyses*.

You can add analyses to an existing series, or change the parameters for an existing series, using the **Continue or change an existing series** option. If you change the parameters for an existing series, the analyses in that series are no longer valid, and they will be deleted (you will be warned before this happens). If you make changes to the parameters and want to cancel those changes, press the **Unchange** button.

You can delete an existing series, using the **Delete an existing series** option. You can delete just the analyses in any series, or you can delete the series completely.

## ***17.2 Basic Parameters and Masses***

---

To specify the masses and some basic parameters, use the **Basic + Masses** page and do the following.

- (1) Choose whether or not P- $\Delta$  effects are to be considered. If you choose “Yes”, P- $\Delta$  effects are considered, for those element groups that have P- $\Delta$  effects. If you choose “No”, P- $\Delta$  effects are ignored. When you are running analyses for a new structure, always choose “No” until you are sure the analysis model is OK. Then choose “Yes” to add P- $\Delta$  effects.

- (2) Choose an event overshoot factor. This is explained later in this chapter. For a large structure you can save computer time by specifying a value larger than the default value of 1.0.
- (3) Choose whether you want a Standard or General load sequence. In almost all cases you can use the Standard sequence. The General sequence is for advanced users with special needs. For an explanation of the General sequence, see *Chapter 19, General Load Sequence*.
- (4) Specify scale factors for the mass patterns. If you have not specified any mass patterns, or if you specify zero scale factors for all patterns, the analysis model has no mass. Hence, you can apply gravity and static push-over load cases but not dynamic earthquake loads. Also, you can not use the General Push-Over Plot or Target Displacement Push-Over Plot task to assess performance. In the early stages of an investigation you may have an analysis series with no mass, but in order to get useful results you will need to specify masses. For a new analysis model it is can be a good idea to specify zero mass and run only static analyses until you are sure the model is correct.
- (4) Specify the number of mode shapes that are to be calculated. If you use only Rayleigh damping, you will need only a few mode shapes. If you use modal damping you will usually need more shapes.

### 17.2.1 Event Overshoot Factor

PERFORM-3D uses an event-to-event solution strategy. In this strategy the program determines when a significant change occurs in the structure stiffness (an "event"). If an event occurs within a load step or a time step, the step is subdivided, and the structure stiffness is modified.

The event-to-event strategy is more reliable than other strategies. It has the disadvantage, however, that for a large structure the number of possible events increases, and the time taken to modify the stiffness after an event also increases. Hence, the computation time increases rapidly as the size of the structure increases. With modern high speed computers, experience has shown that this is not a serious problem, and

that PERFORM-3D is efficient computationally even for very large structures. Nevertheless, it can be useful to use the Event Overshoot Factor to reduce the computation time. This factor works as follows.

If PERFORM-3D were to consider each stiffness change exactly as it occurs, the number of events could be very large. To reduce the number of events, PERFORM-3D uses an event overshoot tolerance, so that a yield event does not occur until the calculated force exceeds the yield force by a small amount (the overshoot tolerance). If there are closely spaced stiffness changes, a number of these changes can occur within this tolerance. If this happens, these stiffness changes are treated as a single event, and the total number of events is reduced.

The disadvantage of using an overshoot tolerance is that the overshoot causes an equilibrium error (unbalanced forces) that must be corrected. If there is an equilibrium error at the end of any substep, PERFORM-3D corrects for it in the following substeps, by applying the unbalanced forces as extra loads. Any unbalanced forces are thus temporary, and usually have little effect on the calculated response. Equilibrium errors also lead to energy errors, where the work done by the external forces on the structure is different from the internal work (which is partly dissipated and partly stored as recoverable strain or kinetic energy). PERFORM-3D calculates an energy balance, which is shown in the ECHO file. If the difference between external and internal work is larger than a few percent, this indicates that the equilibrium errors could be significant. In PERFORM-3D the energy balance is usually very close, although the energy error may be larger if there is brittle strength loss.

For elements that yield, PERFORM-3D uses a default overshoot tolerance of 1%. This means that any unbalanced forces are relatively small. It also means that there is a relatively small chance that a number of stiffness changes will occur within a single event. For structures of moderate size, it is usually a good idea to use the default tolerance. For very large structures, however, it is usually acceptable to use a larger tolerance, and hence reduce the number of events. The reason is that if an element yields and there are unbalanced forces, these forces are transferred to the other elements in the structure. In a small structure, there are relatively few elements to absorb the unbalance, so it is best to keep the unbalance as a small percentage of the strength of a single element. In a large structure, however, there are more elements to absorb the unbalance, and this unbalance can safely be a larger

percentage of the strength of a single element. A larger unbalance means a larger chance that a number of stiffness changes will occur within a single event, so the computation time can be significantly reduced.

When you set up a new Analysis Series, you can choose an event overshoot factor. If you choose a factor of, say, 5, you are essentially specifying a 5% overshoot tolerance for a yielding element. For a very large structure you should be able to use a factor of 10 or even higher. You should experiment to make sure that a larger tolerance does not cause significant differences in the results, and for each analysis you should check the energy balance in the ECHO file.

In a push-over analysis, if there are any unbalanced loads at the end of a load step (as distinct from a substep), PERFORM-3D iterates to eliminate this unbalance before continuing with the next load step. Hence, a load step typically consists of a number of "advancing" substeps, where the structure drift progressively increases, followed by a few "correcting" substeps, where the unbalanced forces are corrected at essentially constant drift. If you specify an event overshoot factor of 10 or larger, the correcting substeps are omitted, and any unbalanced forces are carried over to the next load step.

Correcting steps are also used in nonlinear gravity load analyses. They are not used in dynamic earthquake analyses (any unbalanced loads at the end of one time step are carried over to the next time step).

## **17.3 Damping**

---

### **17.3.1 General**

See *Chapter 18, "Elastic" Viscous Damping* for an explanation of Modal and Rayleigh damping.

For most structures we suggest (a) using modal damping, with a number of modes roughly the same as you would use for a linear modal analysis of the same structure, and (b) using a small amount of Rayleigh damping to avoid potential problems with undamped degrees of freedom. This is explained in Chapter 18, Section 18.4.



### 17.3.2 Procedure for Modal Damping

Use the **Modal Damping** page. The options are (a) no modal damping, (b) the same damping ratio for all modes, and (c) damping ratios that depend on the mode period.

If you choose the same damping ratio for all modes, specify this ratio. This ratio is used for only the modes for which you have calculated mode shapes. Any higher modes are not damped.

If you choose damping ratios that depend on period, specify from 2 to 6 periods and corresponding damping ratios. The damping ratio for each mode is determined by interpolation. As before, there is damping only in the modes for which you have calculated mode shapes.

### 17.3.3 Procedure for Rayleigh Damping

Use the **Rayleigh Damping** page.

Use the **Basic Values** page to specify how the damping ratio varies with period. See Chapter 18, Section 18.2.4 for an explanation.

Use the **Beta-K Options** page to specify different amounts of  $\beta K$  damping for different element groups. The default is to use the factors specified with the element groups.

If you have a structure with seismic isolation, you may want to use the **Alpha-M Options** page. See Chapter 18, Section 18.2.6 for an explanation. However, experience indicates that there is little difference between the Standard and Base Isolation options. Also, for a base isolated structure it is probably better to use modal damping.

## ***17.4 Procedure for Using Upper/Lower Bounds***

---

For specifying and using upper/lower bounds as component properties, see *Chapter 5, Component Properties, Section 5.15*.

To use these bounds, use **U/L Bounds** page. Figure 17.2 shows this page.

ANALYSIS SERIES

**Structure is OK**

The structure is checked automatically when you start a new analysis series. If you wish, you can check it beforehand by pressing this button.

**TYPE OF OPERATION**

Start a new analysis series  
 Continue or change an existing series  
 Delete an existing series

---

**CONTINUE OR CHANGE AN EXISTING SERIES**

Series name:   Number of analyses =

Description:

Change analysis series properties below if desired.  
Press OK to save properties and continue.

---

Basic + Masses
Modal Damping
Rayleigh Damping
**U/L Bounds**

Choose component group and type. The table will list the components that have U/L Bounds, and show their current U/L Bound Ratios. Specify new U/L Bound Ratios as needed. Click on one or more components to select them, or check Select All. Press Apply to apply to the selected components.

Group:   Type:

Text for filter:

Current U/L Bound Ratios (blank = 0 = nominal)

Component Name	P	M	V	K
Roof, MR frame, W24x84		0.4		0.25
Floor, MR frame, W27x94				

**New U/L Bound Ratios**

0 = nominal. 1 = upper. -1 = lower.

Axial (P) Strength  
 Moment (M) Strength  
 Shear (V) Strength  
 Stiffness (K)

Select All

Figure 17.2 Form for U/L Bound Ratios

When you set up a new Analysis Series, you have the option of varying the strengths and/or stiffnesses of any or all components. The default is to use nominal strength and stiffness values for all components. If you want to use larger or smaller values for any component, you must specify U/L Bound Ratios for that component. These ratios have the following meaning.

- (1) If the ratio = 0, use the nominal value for the corresponding strength or stiffness.
- (2) If the ratio = 1.0, use the upper bound value.
- (3) If the ratio = -1.0, use the lower bound value.

- (4) If the ratio is between 0 and 1.0, interpolate between the nominal and upper bound values.
- (5) If the ratio is between 0 and -1.0, interpolate between the nominal and lower bound values.

In general you can specify U/L Bound Ratios for axial strength (P), bending strength (M), shear strength (V) and stiffness (K). However, most components have upper/lower bound values for only some of these quantities. For example, a FEMA Beam, Steel Type component has bounds for moment strength, shear strength and (bending) stiffness. This component has only moment strength, but shear strength is included because the rotation capacity can depend on shear force.

The steps for specifying U/L Bound Ratios are as follows.

- (1) Choose a component group (Inelastic, Elastic, Material or Strength Section) from the Group list.
- (2) Choose a component type (e.g., Inelastic Bar) from the Type list. All components of that type will be listed in the table. You can reduce the length of the list by typing a text string in the “Text for filter” box and pressing the **Filter** button. Only component names that include the text string will be listed.
- (3) Specify U/L Bound Ratios.
- (4) Click individual components in the table, to highlight them. Click a second time to remove a highlight. Alternatively, check **Select All** to select all components in the table.
- (5) Press **Apply**. The specified ratios will be shown in the table, and the table will also be saved. You must press **Apply** for changes to take effect.
- (6) Repeat as necessary for all components that have U/L Bound Ratios.

You can edit the tables at any time. If analyses have been run in an Analysis Series and you change any U/L Bound Ratios, the analyses must be deleted, since the structure has changed.

## **17.5 Quick'n'Dirty**

---

It can often be useful to run a simplified analysis, to save time or to check an analysis model. The "Quick'n'Dirty" page allows you to run simplified analyses without changing the analysis model.

At present there is only one option on this page, namely ignoring strength loss. If you have components that have strength loss, it can be useful to run preliminary analyses without strength loss, to check the analysis model. We strongly recommend that you do this in all cases with strength loss, until you are sure that the model is correct. Then start a new Analysis Series where strength loss effects are included. (We also recommend that you turn off P- $\Delta$  effects until you are sure that the model is correct.)

For this option, cyclic degradation of stiffness and energy is also ignored.

## 18 "Elastic" Viscous Damping

---

*A structure that is essentially elastic dissipates energy by various mechanisms. This is "elastic" energy dissipation, which is usually modeled using viscous damping. A common assumption for linear structural analysis is 5% of critical viscous damping in each natural mode of vibration. Since the actual damping mechanism is almost certainly not viscous, this is an approximation.*

*After a structure yields it dissipates additional energy through inelastic behavior. In a nonlinear dynamic analysis this is modeled directly. The total energy dissipation is the sum of the elastic and inelastic dissipation.*

*In nonlinear dynamic analysis it is usual practice to assume viscous damping to account for the "elastic" energy dissipation. PERFORM allows two types of viscous damping, namely Modal and Rayleigh damping. This Chapter describes the assumptions and theory.*

### 18.1 Modal Damping

---

#### 18.1.1 Damping Matrix

When modal damping is specified, PERFORM uses a damping matrix based on the modes for which mode shapes have been calculated. You can specify that mode shapes are to be calculated for up to 50 modes, but it is usually not necessary to consider that many. As a general rule it should be sufficient to consider as many modes as you would for a linear analysis of the same structure.

PERFORM uses the (elastic) mode shapes and periods, and calculates the implied damping matrix using structural dynamics principles. This matrix is given by

$$\underline{C} = \sum_{n=1}^{n=N} \frac{4\pi}{T_n} \xi_n \frac{(\underline{M}\phi_n)(\underline{M}\phi_n)^T}{\phi_n^T \underline{M} \phi_n} \quad (18.1)$$

where  $N$  = number of damped modes,  $T_n$  = mode period,  $\xi_n$  = mode damping ratio,  $\underline{M}$  = mass matrix and  $\underline{\phi}_n$  = mode shape.

As noted later, when you use modal damping, we recommend that you add a small amount of Rayleigh damping.

If you do not understand the following paragraph, don't worry. The important thing is that modal damping can cause an increase in the energy balance error. Experience indicates that this effect is minor.

For solution of the equilibrium equations at each step of an analysis, the structure stiffness matrix is banded. However, the damping matrix is not banded. All terms in the damping matrix that lie within the stiffness matrix band are included in the stiffness matrix (they are "left-hand-sided"). The remaining coefficients are not included in the stiffness matrix, but are considered when the resisting force, and hence the equilibrium unbalance, is calculated at the end of the step (they are "right-hand-sided"). Because of this, the damping forces cause some equilibrium unbalance. This unbalance is added to any unbalances that result from nonlinear behavior. This tends to increase the energy balance error. Experience indicates that the effect is usually minor.

### 18.1.2 Physical Interpretation

It is useful to consider the physical meaning of modal damping, and to understand the difference between modal damping in linear and nonlinear structures.

As a linear structure deforms under dynamic earthquake load, its deformed shape continually changes. At any instant of time this shape can be decomposed into mode shapes, each mode shape vibrating independently in its natural frequency. If modal damping is assumed, each mode is independently damped. Decades of experience have shown that it is reasonable to assume modal damping for linear analysis.

In principle this could be extended to a nonlinear structure. If the behavior of the structure is linear between well defined nonlinear "events", such as plastic hinge formation, the mode shapes could be recalculated at each event (i.e., each time the structure stiffness changes), and hence modal damping could be assumed. It is

prohibitively expensive, however, to recalculate the mode shapes (and hence, in effect, the damping matrix) at every event. Hence, when modal damping is used for a nonlinear structure, it is assumed that the damping matrix remains constant. At any instant of time the deformed shape of the structure still contains contributions from the elastic mode shapes. However, unlike the linear case, the effective periods of vibration for these shapes are not the linear periods, the shapes are generally not independent (uncoupled), and the deformed shape contains shapes other than the linear mode shapes. This has the following effects.

- (1) The mode shapes are still damped, but since the effective period may have changed (probably increased) while the effective damping coefficient is unchanged, the amount of damping, expressed as a proportion of critical damping, generally changes.
- (2) The only components of the deformed shape that are damped are those that correspond to the linear mode shapes. All other deformations are undamped.

This does not invalidate a damping model based on linear mode shapes. It just indicates that a physical interpretation is difficult. This type of model is probably the best that is currently available (with the addition of a small amount of Rayleigh damping, as noted later).

The next section describes the Rayleigh damping model in some depth. A later section makes suggestions for a reasonable damping model that combines modal damping with a small amount of Rayleigh damping.

## ***18.2 Rayleigh Damping***

---

### **18.2.1 Rayleigh Damping Model**

Rayleigh damping assumes that the structure has a damping matrix,  $\underline{C}$ , given by:

$$\underline{C} = \alpha \underline{M} + \beta \underline{K} \quad (18.2)$$

where  $\underline{M}$  is the structure mass matrix,  $\underline{K}$  is the initial elastic stiffness matrix, and  $\alpha$  and  $\beta$  are multiplying factors.

The physical meaning of  $\alpha M + \beta K$  damping is illustrated in Figure 18.1.

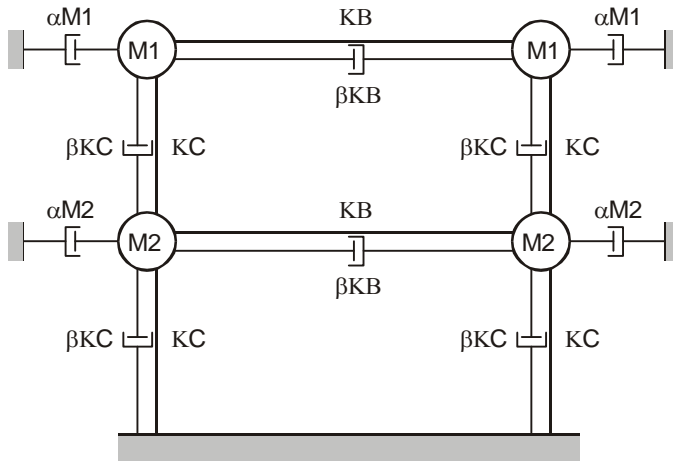


Figure 18.1 Physical Meaning of  $\alpha M + \beta K$  Damping

The figure shows a frame with beam stiffnesses  $K_B$ , column stiffnesses  $K_C$ , and horizontal masses  $M_1$  and  $M_2$ . The physical meaning of  $\alpha M$  damping is that viscous dampers with damping coefficients  $\alpha M_1$  and  $\alpha M_2$  connect the masses to external fixed points. The physical meaning of  $\beta K$  damping is that internal viscous dampers with damping coefficients  $\beta K_B$  and  $\beta K_C$  act in parallel with the elements. For a truss bar element the  $\beta K$  damper is a simple axial damper. For a frame element the concept is the same, but physically the damper is a velocity-dependent bending element.

### 18.2.2 Implied Modal Damping Ratio for Linear Analysis

For linear analysis, modal damping is almost always used. However, Rayleigh damping can be used as an alternative. It can be shown that  $\alpha M$  damping corresponds to more damping in the lower (longer period) modes and less damping in the higher (shorter period) modes. The relationship between damping ratio and mode period is



$$\xi_i = \alpha \frac{T_i}{4\pi} \quad (18.3)$$

where  $T_i$  = period of mode  $i$  and  $\xi_i$  = proportion of critical damping in this mode.

It can also be shown that  $\beta K$  damping corresponds to less damping in the lower modes and more damping in the higher modes, with the relationship:

$$\xi_i = \beta \frac{\pi}{T_i} \quad (18.4)$$

For linear analysis, it can also be shown that Rayleigh damping does not cause coupling between mode shapes (i.e., if a structure is deformed in a mode shape, the damping forces affect only that mode shape, and have no effects on any other mode shapes). Modal damping does the same, of course. With modal damping, however, the damping ratio can be specified independently for each mode, and is usually specified to be the same in all modes. With Rayleigh damping this is not the case. There are only two independent parameters for Rayleigh damping, namely the factors  $\alpha$  and  $\beta$ . Hence, the damping ratio can be specified for only two modes. For the other modes the damping ratio varies with mode period as shown in Figure 18.2

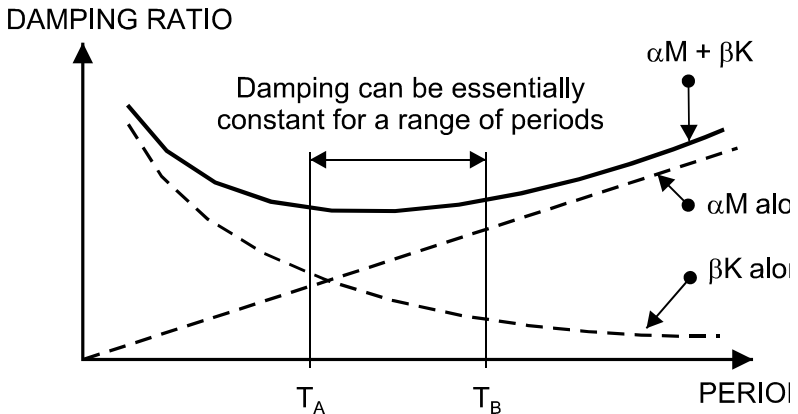


Figure 18.2 Variation of Damping Ratio with Period for Linear Analysis

It is not possible to get the same damping ratio in all modes. However, by choosing appropriate values for  $\alpha$  and  $\beta$  it is possible to have essentially constant damping over a significant range of periods, as indicated in Figure 18.2.

When this type of damping is used for linear dynamic analysis, the longer period,  $T_B$ , is usually the first mode period (or close to it) and the shorter period,  $T_A$ , corresponds to some higher mode. For example, if  $\alpha$  and  $\beta$  are chosen so that the damping is 5% at  $T_B = 0.9T_1$ , where  $T_1$  is the first mode period, and so that the damping is also 5% at  $T_A = 0.25T_1$ , then the damping is close to 5% over a range of periods from  $0.2T_1$  to  $T_1$ . This might cover the most important modes. Higher modes are more heavily damped.

### 18.2.3 Implied Damping for Nonlinear Analysis

As with modal damping, it is possible in principle to recalculate the mode shapes and periods at each nonlinear event, and hence to recalculate the damping matrix. The implied modal damping ratios would then be as in the preceding section. In practice, however, this is too expensive computationally, and the damping matrix is kept constant, based on Equation 18.2 with  $\underline{K}$  equal to the elastic stiffness of the structure.

The physical interpretation is similar to that for modal damping, with the following differences.

- (1) All mode shapes are damped, not just the lower modes.
- (2) Almost all structures have more degrees of freedom than mode shapes (roughly 6 times the number of nodes for the total number of DOFs, versus 2 or 3 times the number of masses for the number of mode shapes). The deformed shapes that do not correspond to mode shapes have zero mass and hence zero period. This does not, however, mean infinite damping, because the dynamic analysis is carried out step-by-step with a finite time step, and the response in any mode with a period smaller than about 10 times the time step

is inaccurate. There is, however, high effective damping in the short (and zero) period modes.

Hence, Rayleigh damping, with  $T_A$  and  $T_B$  chosen as in the preceding section, is similar to modal damping, but with more damping in the higher modes shapes.

#### 18.2.4 PERFORM Implementation

In PERFORM-3D you do not specify values for  $\alpha$  and  $\beta$  directly. Instead you must specify two period ratios,  $T_A/T_1$  and  $T_B/T_1$ , and the damping percentages at these ratios (usually the same percentage at both). PERFORM-3D then calculates the required values of  $\alpha$  and  $\beta$ .

There are two points of view on how the periods  $T_A/T_1$  and  $T_B/T_1$  should be chosen, as follows.

- (1) Use the same method as might be used for linear analysis. That is, specify  $T_A/T_1 =$  about 0.25 and  $T_B/T_1 =$  about 0.9.
- (2) Observe that as a structure yields it usually softens, and it hence its effective vibration periods usually increase. Period is inversely proportional to the square root of stiffness (based on secant stiffness). Roughly, therefore, if the stiffness decreases by a factor of 4 the period increases by a factor of 2. In terms of ductility ratio, a stiffness decrease of 4 corresponds roughly to a ductility ratio of 4. Hence, a ductility ratio of 4 corresponds roughly to a period increase of 2 (or, in general, a ductility of  $N$  corresponds roughly to a period increase of  $\sqrt{N}$ ).

Suppose that you have a structure where the response is dominated by the first mode, and that you estimate a ductility ratio of 6. You can expect that the first mode period will increase, and will vary in the range from the elastic value to about 2.5 times that value. Hence, you might like to keep the elastic damping essentially constant over that range of periods. You can do this by specifying  $T_A/T_1 =$  about 1.2 and at  $T_B/T_1 =$  about 2.2.

The first of these two methods gives a smaller value for  $\beta$  and a larger value for  $\alpha$  than the second method. It is not possible to say which

method is better. Intuitively, the first method, with a smaller value for  $\beta$  sounds preferable, and if you use only Rayleigh damping, we suggest that you use that method.

### 18.2.5 Procedure for Specifying Rayleigh Damping

Use the **Run Analyses** task, the **Analysis Series** form, and the **Rayleigh Damping** and **Basic Values** pages. The steps are as follows.

- (1) Input the period ratios  $T_A/T_1$  and  $T_B/T_1$ , and the damping percentage at each ratio. If you do not intend to run any dynamic analyses, you can specify zero damping. To do this leave the boxes blank.
- (2) Press the **Draw Graph** button to plot the variation in damping ratio. If you have left the boxes blank the damping will be zero.
- (3) If the result is not as expected, press the **Close Graph** button and try again.

### 18.2.6 Special Assumption for Concrete Fibers

If a structure has components that use concrete fibers (Fiber Section and/or Concrete Strut components), these fibers are stiff in compression and after cracking have no stiffness in tension.

For  $\beta K$  viscous damping, the stiffness  $K$  is the initial stiffness of the structure. This is calculated assuming that all concrete fibers are in compression (i.e., all concrete fibers are assumed to be initially uncracked). The same value of  $\beta K$  is used after the concrete cracks (i.e.,  $\beta K$  is constant). Experience indicates that this can lead to unrealistically large amounts of viscous damping in concrete fibers after cracking occurs.

This can occur, for example, when inelastic fiber cross sections are used in shear wall elements. When a fiber cross section cracks, the neutral axis shifts, and as the wall bends it also deforms axially. If the axial  $\beta K$  damping stiffness is large, this can lead to unrealistically large axial viscous damping forces in the wall. It can also stiffen the wall excessively, and lead to unrealistically large amounts of viscous energy dissipation.

To avoid this problem, PERFORM-3D makes a special assumption when it calculates  $\beta K$  stiffnesses for elements that use concrete fibers. These include shear wall and general wall elements that use fiber wall cross sections, frame type elements that use fiber cross sections, and bar elements that use concrete strut components. For these elements, the stiffness,  $K$ , used for the element  $\beta K$  stiffness is based on only 15% of the concrete material area. In other words, in components where concrete cracking can occur, the concrete is considered to be only 15% effective when the  $\beta K$  damping stiffness is calculated. Steel fiber areas are not reduced, and hence steel fibers are assumed to be 100% effective for the  $\beta K$  damping stiffness.

### 18.2.7 Alternative Procedure for Base Isolation

The procedure described in this section is available as a PERFORM-3D option. Experience suggests that choosing this option has little effect on the analysis results. Hence, we suggest that you do not bother.

If you specify  $\alpha M + \beta K$  damping for a structure with seismic base isolation, the physical meaning of the model is as shown in Figure 18.3(a). In this case the  $\alpha M$  dampers connect the structure masses to the ground level, below the isolation level. This may not be what you intend, since it can overestimate the amount of damping. An alternative model is shown in Figure 18.3(b). In this case the  $\alpha M$  dampers connect the structure masses to the level immediately above the seismic isolators, so that the  $\alpha M$  part of the damping applies only to the isolated part of the structure.

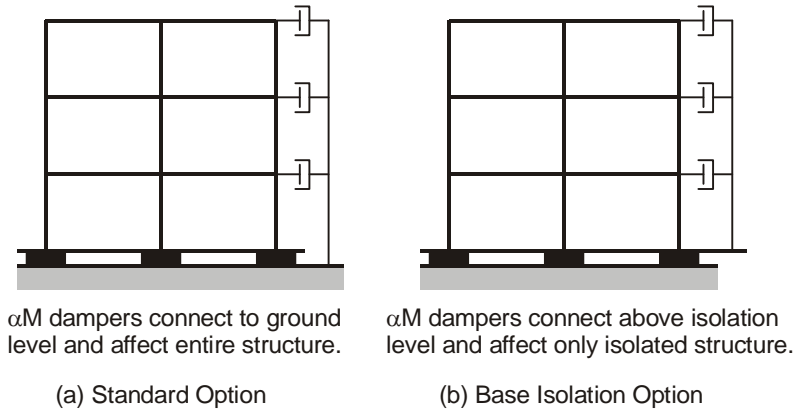


Figure 18.3 Alternative  $\alpha M$  Damping Models

PERFORM-3D allows you to choose either the "Standard" or the "Base Isolation" option. As noted earlier, experience suggest that there is little difference between these options.

The following are some points on the Base Isolation option.

- (1) Only the H masses are connected as shown in Figure 18.3(b). If there are any V or R masses, the  $\alpha M$  dampers for these masses are connected to the ground.
- (2) The  $\beta K$  damping model does not change.
- (3) You must have a rigid floor constraint at the level immediately above the isolators. The  $\alpha M$  dampers connect to this level.

The steps for specifying this option are as follows.

- (1) Before you begin the analysis series you must calculate the first mode period for the non-isolated structure. In effect this is the first mode period for the structure with rigid seismic isolators. You can do this using PERFORM-3D, by changing the isolator properties to make them very stiff before yield. However, you will probably have done the calculation earlier, using a linear analysis program such as SAP or ETABS. Call this period  $T_{1R}$ , where "R" indicates rigid isolators.

- (2) You must also calculate the elastic first mode period for the structure with elastic isolators (i.e., with the isolators assigned their pre-yield stiffnesses). Again, you can use PERFORM-3D but you will probably have done the calculation earlier. Call this period  $T_{1E}$ , where "E" indicates elastic isolators.
- (3) In the **Run Analyses** task and the **Analysis Series** form, choose the **Rayleigh Damping** and **Alpha-M Options** pages.
- (4) Choose the Base Isolation damping option, not the Standard option.
- (5) Choose the level immediately above the base isolators from the list of rigid floor constraints. You must already have defined the required constraint.
- (6) Specify the ratio  $T_{1E} / T_{1R}$ . This ratio will be larger than 1.0.

When you specify the period ratios  $T_A / T_1$  and  $T_B / T_1$ , the standard option calculates the  $\alpha$  and  $\beta$  values using the periods  $T_{1E}$  (with elastic isolators). If you choose the base isolation option, PERFORM adjusts these values so that they are based on  $T_{1R}$  (with a fixed base).

### ***18.3 A Case Where Modal vs. Rayleigh Damping May Be Substantially Different : Coupled Shear Walls***

---

There is an important case where modal and Rayleigh damping may give substantially different results. This is the case of a shear wall structure with stiff coupling panels.

When  $\beta K$  damping is used, the  $\beta K$  damping coefficient for a coupling panel is based on the initial elastic bending and shear stiffnesses of the panel, which can be large. After the panel yields (often in shear) the ductility ratio can be large. Since the  $\beta K$  damping coefficients remain constant, the  $\beta K$  energy dissipation might be overestimated, and the deformations of the panel might be underestimated.

In the **Energy Balance** task you can examine the dissipated  $\beta K$  energy in each element group. We suggest that you always do this. In the case of a shear wall, if the amount of energy seems to be unreasonable for the coupling panel elements, you may wish to specify a reduced Beta-K damping scale factor (possibly zero) for the corresponding element group(s).

This potential problem may not be present with modal damping. When modal damping is used, in effect a viscous damper "element" is added for each mode shape. When the structure deforms, each of these elements dissipates energy based on the amount of the mode shape that is present in the total deformed shape (and on the current deformation rate for this mode). In the case of coupled walls, it is likely that none of the elastic mode shapes involves large deformations of the coupling panels relative to the rest of the wall. Hence, when the panels yield, modal damping should not contribute a lot of added dissipation.

We must emphasize that this is conjecture. We have not run comparative analyses to compare the effects of the different kinds of damping for this type of structure.

### ***18.4 Combined Modal and Rayleigh Damping***

---

You can combine Modal and Rayleigh damping if you wish. One reason why you might do this is as follows. We suggest that you consider using this method.

As noted earlier, if you use only modal damping, the only displaced shapes that have damping are those that correspond to the (elastic) mode shapes. The total number of degrees of freedom in the structure will always be larger than the number of modes, so if only modal damping is used there will be many displaced shapes that are undamped. We recommend that you add a small amount of  $\beta K$  damping to make sure that these "higher mode" displacements are damped.

To do this, get the period of the highest calculated mode, and specify  $\beta K$  damping (with no  $\alpha M$  damping) so that the damping ratio at this period is small, say 0.2%. This will ensure that "higher mode"



displaced shapes have some damping, but that Rayleigh damping in the lower modes is negligible.



## 19 General Load Sequence

---

*PERFORM-3D is intended mainly as a tool for performance based design. However, it can also be used for general purpose nonlinear analysis. For performance based design it is sufficient to use the "Standard" load sequence option that allows gravity load analysis followed by either push-over analysis or dynamic analysis. For more general nonlinear analysis you may want to use the "General" option that allows arbitrary load sequences.*

*This chapter describes the difference between the two options, and gives guidance on how to use the General option.*

### 1.1 Standard and General Load Sequences

---

When you begin a new Analysis Series, you have the option of choosing the "Standard" or "General" load sequence for that series.

The default option is the Standard sequence. This is the sequence that you will almost always use if the purpose of the analysis is to assess structural performance. With this option you can apply gravity load, then add static push-over or dynamic earthquake load (earthquake or dynamic force). You can consider any number of gravity, push-over, dynamic earthquake and dynamic force load cases, but the only load sequences that are allowed are (1) gravity followed by static push-over, (2) gravity followed by dynamic load, (3) static push-over applied to the unloaded structure (unlikely), and (4) dynamic load applied to the unloaded structure (also unlikely).

If you wish, you can apply more than one gravity load case in a load sequence. For example, if some elements are inactive for gravity load, you may want to remove them for dead load, and restore them for live load (for the removal procedure see *Chapter 11, Inactive Elements*). You can do this by applying gravity dead load as the first analysis, followed by gravity live load as the second analysis. If you specify any inactive elements, they are removed only for the first gravity load case in any load sequence, then restored for the second load case in the sequence, whether it is a gravity case, a push-over case or a dynamic earthquake case.

If you choose the General sequence, you can apply loads in any sequence. This provides you with a great deal of flexibility for performing nonlinear analyses. However, it also requires that you exercise a much greater degree of care and skill.

## **1.2 Points to Note for General Sequence**

---

The following are important point on the use of the general load sequence option.

### **19.1.1 General Push-Over and Target Displacement Plots**

You can not use the General Push-Over or Target Displacement tasks if you use the general sequence. However, you can still use the Usage Ratio task. Note that a usage ratio can never decrease. Hence, for some analyses the usage ratios may stay constant.

### **19.1.2 Unload Push-Over Load Case Type**

There is a load case type, namely Unload Push-Over, that is not allowed for the Standard sequence. This allows you to remove push-over loads on the structure. You can choose this type when you define load cases.

When you perform a static push-over analysis, you specify the maximum drift, not the maximum load, and the program calculates the amount of load that is needed. Hence, if you want to remove the push-over loads, and return to a state with only gravity loads, you generally will not know how much load to remove. The Unload Push-Over load case type removes push-over loads automatically.

Usually you will specify that all push-over loads are to be removed (this is the "Unload all push-over cases" option). However, if you want to specify that only the loads for a single push-over case are to be removed, you can do so (this is the "Unload only one case" option). Be careful using the latter option. Suppose, for example, that you apply the following load sequence.

- (1) Push to plus 1% drift. This requires that you apply, say, a load case "Push plus 1%".
- (2) Push to minus 1% drift. This requires that you apply, say, a load case "Push minus 2%".

- (3) Push to zero drift. This requires that you apply "Push plus 1%" a second time.
- (4) The last push-over load case was "Push plus 1%". Hence, you may try to remove the push-over loads by using an Unload Push-Over load case with the unload single case option, and unloading the "Push plus 1%" case.

If you do this, at Step (4) you will be adding minus the loads applied in Steps (1) and (3) (i.e., you will be unloading all "Push plus 1%" loads), but you will leave in place the loads applied in Step (2). This is probably not what you intend. Also, the structure may not be able to support these loads, and the analysis may not converge. The analysis for Unload Push-Over load cases is load controlled, not displacement controlled, and if the load is larger than the collapse load there is no solution.

In the above example, in Step (4) you usually should use an Unload Push-Over load case with the "unload all" option. This will remove all previous push-over loads.

For Unload Push-Over load cases we suggest that you specify 5 load steps and about 500 maximum events (there will usually be only a few unloading events, but specify a large number to be safe).

### 19.1.3 Analysis Limit State

When you run a push-over analysis, it may stop because the maximum specified drift is reached, or because the Analysis Limit State is reached. The default Analysis Limit State for push-over analysis is reaching the X point in any inelastic component.

Be careful if a push-over analysis stops because it has reached its Analysis Limit State. If you specify a following analysis (either another push-over case or an unloading case), and if you specify the same Analysis Limit State, the new analysis will stop after the first load step, because it is already past its Analysis Limit State. To avoid this you must specify a more lenient Analysis Limit State.

### 19.1.4 Controlled Drifts

When you perform a static push-over analysis, you must specify a set of controlled drifts and a maximum drift value. This is the change in drift, not the total drift. The analysis stops when the change in any of the controlled drifts reaches the maximum value. You will also have a

reference drift, which is used for plotting graphs. The reference drift is not necessarily the largest drift. Hence, if you specify a maximum drift of, say, 0.02 (2%), when the analysis stops the change in the reference drift may be smaller than 2%.

If you know that the structure is well behaved, it is OK to specify only one controlled drift, and to make it the same as the reference drift. In general, however, it is a good idea to use several controlled drifts. For example, if you control only the drift at the roof, and if the structure develops a brittle weak story, when the weak story fails the roof drift may have to decrease as the forces are redistributed. If you control only the roof drift, that drift is not allowed to decrease, and there may be very large deformations in the weak story as the structure tries to redistribute the forces (also, the analysis may not be able to converge).

There is a class of problem where you may want to use only a single controlled drift. This is if you want to run a cyclic push-over analysis, to calculate hysteresis loops for a complete structure or (more likely) a sub-assembly. In this case you will usually want to cycle between specified values of the reference drift. You must specify only one controlled drift, and make it the same as the reference drift. Suppose that you specify a maximum drift of 2%. Assuming that the analysis limit state is not reached, you will find that the analysis stops when the reference drift is slightly larger than 2%. The difference will usually be small, but if you must push to a precise drift you may have to try different maximum drift values to get the exact actual value that you want.

### 19.1.5 Removing Gravity Loads

If you want to remove gravity loads you must set up one or more gravity load cases that are the same as the applied load cases but have opposite signs, then add these load cases. Since unloading events (nonlinear behavior) may occur, you should specify nonlinear analysis. We suggest applying the load in 5 steps.

### 19.1.6 Return To Static State After Dynamic Analysis

At the end of a dynamic earthquake analysis the structure will still be vibrating. If you want to add more loads (e.g., static push-over loads, or another dynamic earthquake), you must first return the structure to a static state. PERFORM-3D does not have an automatic way to do this, and you must do the following.

- (1) Define a dummy gravity load case with the nonlinear analysis option and a very small load.
- (2) Apply this load case after the dynamic earthquake. This will return the structure to a static state without significantly changing the gravity loads. We suggest that you specify 5 load steps and about 500 maximum events.

If you plan to apply loads after a dynamic earthquake analysis, it is a good idea to run the dynamic earthquake analysis well beyond the time that the strong shaking ends, so that the structure is nearly at rest at the end of the analysis. Then, when you apply the dummy gravity load there are only small changes in the state of the structure.

### 19.1.7 Graphs That Cover Several Analyses

When you plot graphs in PERFORM-3D (e.g., a graph of cross section force vs. reference drift, using the Time History task), each graph will cover only one analysis. If you want to plot a graph that extends over several analyses (e.g., a graph showing the hysteresis loop for cross section force vs. reference drift), you must press the **Save to File** button for each analysis, then combine the files in a spreadsheet program, and use the spreadsheet program to draw the combined graph.

This is not necessary for structure section time histories. See *Chapter 25, Time History Plots, Section 25.7*.

### 19.1.8 Inactive Elements

If you specify inactive elements, these elements are removed only for gravity analyses that begin with the unloaded state. The elements are restored at the beginning of the second analysis in the sequence, even if this is a gravity analysis.

If the first analysis in any sequence (i.e., the analysis that begins with the unloaded state) is not a gravity analysis, elements are not removed.





## 20 Running Analyses

### 20.1 Steps for Running Analyses

The form for running analyses is shown in Figure 20.1.

**SET UP AND RUN ANALYSES**

New Analyses to be Run

**ANALYSIS TO BE ADDED**

Load Case Type: Dynamic Earthquake

Load Case Name: Artificial biaxial

Preceding Analysis Number (0 = unloaded): 1

Add Insert Delete

Set up as many analyses as you wish.  
Press GO to run the analyses.

GO Don't Go

**ANALYSIS LIST**  
Click to highlight for Insert or Delete.

No.	Load Type	[Preceding Analysis No.] + Load Case Name
5	Dynamic Earthquake	[1] + Artificial biaxial

Previous Analyses in this Series

Analysis Series Name = Series 1

No. of mode shapes = 6 No. of analyses = 4

For more details on any analysis, click to highlight, then press Details. Details

No.	Load Type	[Preceding Analysis No.] + Load Case Name	Status
1	Gravity	[0] + DL+0.25LL, floors 92.5 psf, roof 30 psf	All load applied
2	Push-over	[1] + H1 uniform push to 4%	Reached a component X point at step 42
3	Push-over	[1] + H2 uniform push to 4%	Reached a component X point at step 38
4	Push-over	[1] + H1 negative uniform push to 4%	Reached a component X point at step 42

Figure 20.1 Form for Running Analyses

You can run as many analyses as you wish in a single analysis session, and you can add analyses to any analysis series at any time. The steps for adding analyses and running them are as follows.

- (1) Choose the load case type from the Load Case Type list.
- (2) Choose the load case name from the Load Case Name list.
- (3) Choose the number of the preceding analysis. For a Standard load sequence you will usually apply gravity load first, then add push-over or dynamic earthquake load. The Preceding Analysis list contains only the analysis numbers for preceding gravity load analyses. The analyses are numbered in the order that they are

performed. For a General load sequence, all previous analyses are listed. For either sequence, make sure you choose the correct analysis.

- (4) Use the **Add**, **Insert** and **Delete** buttons in the usual way to build up an analysis list.
- (5) When you have specified the required analyses, press **GO** to run the analyses. Press **Don't Go** to return to the Analyses Series form.

If you have specified masses for the current analysis series, a mode shape analysis will automatically be set up. You can run this analysis alone if you wish, without adding any other analyses to the list.

During the analyses a window is displayed showing the analysis progress. For a new analysis series PERFORM-3D first sets up data for the analysis model, then calculates the mode shapes, then runs the analyses.

After the analyses have been run, they are added to the list of completed analyses. This list shows the status of each analysis, in particular whether the full load was applied or the analysis terminated early. You can get additional details on the status of any analysis by highlighting it, then pressing the **Details** button.

## ***20.2 Adding and Deleting Analyses***

---

You can add new analyses to an analysis series at any time. You can also delete a complete analysis series. You can not delete individual analyses from a series.

If you delete an analysis series, you have the option of deleting that series completely, or deleting any analyses in the series but keeping the analysis series settings (masses, damping, etc.). This second option is the default, and this is the option that you will most commonly use.

If you make changes to the analysis model, you must delete all analyses in all analysis series before making the change, since the analysis results are no longer consistent with the changed model. Unfortunately this is necessary even if the change is a minor one, since it is the only

way to ensure consistency. When you make a change to the model, you will be asked if you want to delete the existing analysis series (the analysis series settings are not deleted), and if you answer "Yes", PERFORM-3D will delete them. If you add or change a Limit State, Drift, Deflection or Structure Section, this is a change in the model, since PERFORM-3D processes limit states, drifts, etc. during the analysis. If you add or change a Frame or a Limit State Group, this is not a change in the model.



## 21 Mode Shapes

---

*Mode shapes and periods can be useful for checking the behavior of the structure, and for comparing a PERFORM-3D model with a linear analysis model. You can also use mode shapes for static push-over analysis. If you do, you must be careful to choose the "correct" mode shape(s).*

*With the Mode Analysis Results task you can plot and animate mode shapes, and get other mode shape properties. This chapter explains the procedure.*

### 21.1 Plotting Mode Shapes

---

If you specify masses when you define an Analysis Series, one or more mode shapes and periods are calculated. These mode shapes are for the structure in its initial unloaded, elastic state. With the present version of PERFORM-3D you can calculate mode shapes for only this state.

The steps for plotting a mode shape are as follows.

- (1) Choose the **Modal Analysis Results** task and the **Modes** page.
- (2) Specify the displacement scale factor, as the ratio of the maximum displacement for the mode to the maximum dimension of the structure. The default is 0.1, which is usually about right.
- (3) Choose the mode number from the list.
- (4) Press **Plot** to plot the mode shape, then **Animate** to animate it.

When you choose a mode, certain of its properties are displayed. These are as follows.

- (1) The mode period.
- (2) The effective mass factors for the H1, H2 and V directions for this mode.
- (3) The H1, H2 and V effective mass factors summed for all modes up to this mode.
- (4) The principal horizontal direction for this mode, and the corresponding effective mass (or base shear) factor. The principal direction is the direction along which the horizontal effective mass

is a maximum. For a symmetrical structure the principal directions for the dominant modes will be along the axes of symmetry of the structure. For a purely torsional mode the base shear factor will be zero.

## **21.2 Using Mode Shapes for Push-Over Analysis**

---

You can use mode shapes for push-over analyses in the following two ways.

- (1) To specify lateral load distributions. You can use a single mode or combine a number of modes. For details see *Chapter 13, Static Push-Over Load Cases*.
- (2) To use as the deflected shape (effective mode shape) for push-over plots. For details see *Chapter 28, General Push-Over Plots* or *Chapter 29, Target Displacement Plot*.

Mode shapes are not used for dynamic earthquake analyses.

## **21.3 Vertical Masses**

---

PERFORM-3D allows you to specify vertical (V) masses. As a general rule you should not do so for building structures. For the assessment of the seismic performance of a building, the horizontal resistance is usually of greatest concern, and vertical inertia effects are less important.

The behavior of the structure is also less sensitive to the distribution of mass when there is only horizontal (H) mass. If you specify vertical mass, you must be careful how you locate the mass points. For example, if you have rigid floor diaphragms you can lump the H masses at the mass centroids of the floors, but you probably should not do this for V mass. More likely you will be tempted to lump the V masses at the beam-column connections. If you do this, because of the large axial stiffnesses of the columns there can be high frequency vertical vibrations that cause rapid, and relatively small, variations in the column axial forces. These force variation are unlikely to have

much effect on the overall behavior of the structure, but if there is inelastic P-M-M interaction, or if the structure has friction-pendulum isolators, they can cause the nonlinear analysis to be substantially less efficient, and possibly to be inaccurate. To get an accurate analysis you may have to choose a very small time step.

For long span structures such as bridges, vertical inertia effects can be significant. In this case, be sure to distribute the V masses in a reasonable way, and consider the vertical vibration periods when selecting the time step for a dynamic earthquake analysis.

If you specify V masses, and if you use mode shapes to define load patterns for static push-over analysis, the V masses are ignored. Static push-over loads are horizontal only.





## 22 Response Spectrum Analysis

---

*This is not a full-featured response spectrum analysis. Rather, it is intended to help in making decisions based on certain sections of FEMA 356 that refer to analysis results of response spectrum type.*

*This chapter describes the procedure and limitations.*

### 22.1 Response Spectrum Load Cases

---

#### 22.1.1 Spectra

Before you can run response spectrum analyses you must define one or more acceleration response spectra. You must first set up one or more text files containing the spectra, then read these spectra into PERFORM. The procedure is similar to that for defining earthquake acceleration records. The spectra are stored in a **Spectra** folder that is similar to the **Records** folder for earthquake records.

The text file for a response spectrum must be a comma-delimited or space-delimited file containing the following.

- (1) Any number of header lines. These are skipped during the reading process.
- (2) One line containing the damping ratios (maximum 6 ratios). When the file is read, the number of damping ratios for the spectrum set is obtained from the number of items in this line.
- (3) One line for each period, in order of increasing period. Each line must contain the period followed one spectral acceleration value for each damping ratio. The number of lines determines the number of periods. ***There must be a line break (carriage return) after the last line, but do not add blank lines.***

Each file must contain only one spectrum (actually a spectrum set, for different damping ratios).

To read a spectrum text file and add a new spectrum to the **Spectra** folder, press the toolbar buttons for the **Analysis** phase and the **Load Cases** task, choose Response Spectrum for the Load Case Type, and

press the **Add/Review/Delete Spectra** button. You can then add one or more new spectra, review (plot) any existing spectrum, and delete any existing spectrum.

The steps to add a new spectrum are as follows.

- (1) Enter the name of the spectrum text file. If you have copied this file to the *User* folder you need to enter only the file name. Otherwise you must give the complete path to the file. The file can be on a diskette or CD if you wish.
- (2) Choose the acceleration unit from the list.
- (3) Specify the number of header lines to be skipped before the beginning of the data.
- (4) Specify whether the values are separated by spaces or commas.
- (5) If the spectrum is to be placed in an existing spectrum group, choose the name of the group from the list. To create a new group, press the **New** button and specify the group name.
- (6) Specify a file name and a spectrum name for the spectrum. Choose the spectrum name carefully because you must use it to identify the spectrum when you review it or use it in a load case.
- (7) Press the **Check** button to read the file and set up the spectrum. If the file is read successfully the spectrum will be plotted. If the spectrum is OK press **Save** to save the spectrum, else press **Cancel**, revise the above data, and try again.

When you have set up all of the needed spectra, press **Return to Spectrum Load Case**.

### 22.1.2 Response Spectrum Load Cases

To start a new response spectrum load case press the toolbar buttons for the **Analysis** phase and the **Load Cases** task. In the Load Cases form choose Response Spectrum as the Load Case Type, press the **New** button, and enter the load case name. Keep an eye on the **Status** box.

The steps are as follows.

- (1) If you have not already set up the needed spectra, press the **Add/Review/Delete Spectra** button and add spectra as described in the preceding section.
- (2) Specify the angle from the structure H1 axis to the earthquake Q1 direction. This allows you to apply an earthquake in any horizontal direction.
- (3) Specify the number of modes to be used. If this is larger than the number of modes actually calculated (when the analysis is run), the smaller number will be used.
- (4) Choose the Q1 Spectrum, if any, from the list of spectra. Also choose the damping ratio (i.e., the particular spectrum that is to be used if there are spectra for more than one damping ratio), and specify a spectrum scale factor if desired.
- (5) Repeat for the Q2 and V Spectra, if any. The default is none.
- (6) Press the **Save** button to save the load case. Use the **Delete** button if you want to delete a load case before saving it.

You can edit a saved load case at any time. If you do, keep an eye on the **Status** box. Use the **Save**, **Save As** and **UnChange** buttons in the usual way. You can also delete saved load cases.

## ***22.2 Response Spectrum Analyses***

---

### **22.2.1 Run Analysis**

To run a response spectrum analysis, go to the Set Up And Run Analyses form, choose Response Spectrum for the Load Case Type and choose the load case name from the Load Case Name list. You must have specified some masses, so that mode shapes will be or have been calculated. If there are no mode shapes, there will be no load cases in the Load Case Name list. The Preceding Analysis Number is always zero for this load case type.

Use the **Add**, **Insert** and **Delete** buttons in the usual way to build up an analysis list. You can mix analyses of different types. Press the **Go** button to run the analyses.

### 22.2.2 Analysis Results

To see the analysis results choose the **Modal Analysis Results** task, then chose the **Nodes**, **Drifts** or **Sections** page and follow the instructions. You can show the analysis results and/or and save the results to a text file. You can not get element results.

### 22.2.3 Lateral Load Patterns from Analysis Results

For static push-over analysis, you may want to set up a lateral load pattern where the distribution of story shears is the same as in a response spectrum analysis. This is not necessarily a good choice for a load pattern, but you can set it up as follows.

- (1) Cut a structure section through all elements in each story of the structure. To be consistent, cut at the bottom of each story. You must cut a section for every story.
- (2) In the **Modal Analysis Results** task choose the **Sections** tab and then the **Save** tab.
- (3) Select the story sections in the list, press the **Save** button, and follow the instructions to set up a text file with the structure section forces. These are SRSS values, so they do not have signs.
- (4) Using spreadsheet program, process the file to get a lateral load pattern that that gives the same story shears.
- (5) Enter this pattern into PERFORM as a nodal load pattern.

## 23 Energy Balance

---

*The response of a structure to an earthquake can depend on the amount of energy that the structure is able to dissipate. In the analysis of elastic structures, it is usually assumed that energy is dissipated by viscous damping (which is a modeling approximation, except for structures with actual viscous dampers). In the analysis of inelastic structures, it is still usually assumed that there is viscous damping, and in addition energy is dissipated by inelastic effects (yielding, friction, etc.).*

*With the Energy Balance task you can plot graphs that show the amount of each type of energy. This can be useful for checking the analysis. You can also determine which members in the structure contribute the most to the inelastic dissipated energy. This can be useful for assessing the performance of the structure. This chapter explains the procedure.*

### 23.1 Types of Energy

---

#### 23.1.1 Dynamic Analysis

For a dynamic analysis there can be six different types of energy, as follows.

- (1) Kinetic energy in the masses.
- (2) Recoverable strain energy in the elements.
- (3) Non-recoverable inelastic energy dissipated in the elements.
- (4) Viscous energy dissipated by  $\alpha M$  dampers.
- (5) Viscous energy dissipated by  $\beta K$  damping in the elements.
- (6) Viscous energy dissipated by modal damping.
- (7) Viscous energy dissipated in fluid damper components.

PERFORM-3D calculates each of these energies at each step of the analysis (with certain approximations as noted below). PERFORM-3D also calculates the external work done on the structure. For earthquake analysis this is the work done by the base shear, although it is actually calculated as the work done by the equivalent inertia forces.

### 23.1.2 Static Analysis

For a static analysis there are only two types of energy, as follows.

- (1) Strain energy in the elements.
- (2) Inelastic energy dissipated in the elements.

PERFORM-3D calculates of these energies at each step of the analysis (again with certain approximations). PERFORM-3D also calculates the external work done on the structure (the work done by the external static loads).

If the structure has fluid damper components, and if "push-over" properties have been specified for these components, PERFORM-3D also estimates the amount of viscous energy dissipated by the dampers. This value is approximate, but it can provide useful information. For an explanation of the method see *Chapter 28, General Push-Over Plots*.

### 23.1.3 Calculation of Inelastic and Strain Energies

For any structural component, the sum of the strain energy and the inelastic dissipated energy is equal to the static work done on the element. Figure 23.1 shows a force-deformation relationship for a simple component (e.g. an inelastic bar).

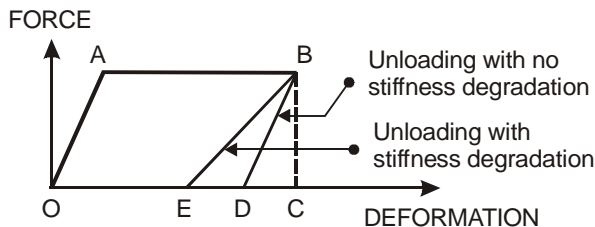


Figure 23.1 Loading and Unloading of an Inelastic Component

If the component is loaded monotonically (i.e., with progressively increasing deformation and no unloading) from its unloaded point O through yield at A to State B, the work done on the component is equal to the area OABC under the force-deformation curve. At this state we do not generally know how much of this work is stored as recoverable

strain energy and how much is dissipated as inelastic energy. The only way we can find out how much energy can be recovered is to unload the component.

Figure 23.1 shows two of many possible unloading paths, as follows.

- (1) Inelastic unloading along BD, with no stiffness degradation. In this case the unloading stiffness is equal to the original elastic stiffness. The strain energy is the recovered work, area BCD. The rest is inelastic dissipated work.
- (2) Inelastic unloading with stiffness degradation. In this case the unloading stiffness is smaller than the original elastic stiffness. The strain energy is again the recovered work, area BCE, which is larger than in Case (1). The rest is again the inelastic dissipated work, which is smaller than in Case (1).

At each step of an analysis, PERFORM-3D calculates the total static work on each component, equivalent to the area OABC for State B. For an inelastic component, PERFORM-3D generally does not know how the total work is divided between strain energy and dissipated energy, since in general this can be determined only by unloading the component. PERFORM-3D thus estimates the amount of strain energy (and hence also estimates the amount of dissipated energy). This estimate assumes that the component unloads with no stiffness degradation.

This means that for a component with stiffness degradation, PERFORM-3D can underestimate the strain energy, and hence overestimate the dissipated energy, at least temporarily. For example, if the component in Figure 23.1 is loaded along path OAB then unloaded to either Point D or Point E, PERFORM-3D calculates the strain energy and the dissipated inelastic energy as shown in Figure 23.2.

Up to Point A the behavior is elastic, and the strain energy increases quadratically with deformation. Between Point A and Point B the total energy increases linearly. If there is no stiffness degradation the strain energy stays constant and the inelastic energy increase linearly. PERFORM-3D calculates the energies using this assumption. If there is actually no stiffness degradation, with unloading along BD, the strain energy decreases quadratically and the inelastic energy stays constant. This is correct. However, if there is stiffness degradation, with

unloading along BE, the change in strain energy is larger and the inelastic energy decreases.

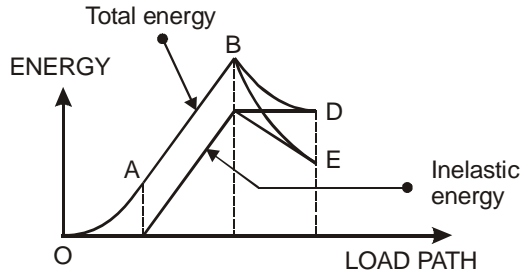


Figure 23.2 Energy Variation for Paths in Figure 23.1

Since inelastic dissipated energy can not decrease, the energies calculated by PERFORM-3D for the degrading case are not correct. However, note the following points.

- (1) The dissipated energy is correct at all points if there is no stiffness degradation.
- (2) For the case with degradation the dissipated energy is correct at Point E, after unloading.
- (3) As the deformation increases, the error in the dissipated energy gets smaller.
- (4) If the component is loaded cyclically, as in a dynamic earthquake analysis, the error in the dissipated energy gets smaller.
- (5) Usually we will be concerned not with the absolute amount of dissipated energy but with the relative amounts contributed by different components (e.g., for a frame structure, the amount of energy dissipated in the beams compared with the amount dissipated in the columns). The relative energies will tend to have smaller errors than the absolute energies.

For a component with stiffness degradation it is possible to get better (but not necessarily exact) energies by calculating the degraded stiffness at each step of the analysis and using this stiffness to calculate the strain energy. However, we have chosen not to do this. For computational reasons PERFORM-3D calculates the degraded stiffness for a component only when it unloads. It is possible to calculate the



degraded stiffness at every step, but this requires substantial additional computation, and is not justified simply to obtain a more accurate energy estimate.

In summary, the inelastic "dissipated" energy calculated by PERFORM-3D is not exact, and does not necessarily increase monotonically. However, any errors are usually small.

## 23.2 Energy Plots

---

### 23.2.1 Complete Structure

To draw an energy plot for the complete structure, choose the **Energy Balance** task and the **Structure** tab. Then choose the load case and press **Plot**. You can draw energy balance plots only for dynamic analyses.

### 23.2.2 Inelastic Energies in Element Groups

It can be useful to determine which element groups contribute most of the inelastic dissipated energy. To show this choose the **Elem Groups** tab, and the **Inelastic** option. Choose an **Element group** from the list, and press **Plot**. The graph shows the total amount of dissipated energy and the proportion of this energy that is contributed by the chosen group. Cycle through the **Element group** list to show the proportion for each element group.

If a structure has viscous dampers, the dissipated viscous energy in these dampers is included in the total dissipated energy. For a static push-over analysis this viscous energy is an approximate estimate.

### 23.2.3 $\beta$ -K Energies in Element Groups

If you specify only Rayleigh damping, it can be also useful to determine which element groups contribute most of the  $\beta$ K dissipated viscous energy. To show this choose the **Elem Groups** tab, and the **Beta-K viscous damping** option. Choose an **Element group** from the list, and press **Plot**. The graph shows the total amount of  $\beta$ K dissipated energy and the proportion of this energy that is contributed by the

chosen group. Cycle through the **Element group** list to show the proportion for each element group.

### 23.2.4 Energy Error

The energies that are plotted are the internal energies from the elements, masses,  $\beta K$  dampers,  $\alpha M$  dampers and modal dampers. PERFORM-3D also calculates the external energy, which is the work done by the external forces. Because the nonlinear analysis does not exactly follow the nonlinear behavior of the structure there will be differences between the internal and external forces (equilibrium errors) and hence between the internal and external energies. This is the energy error for the analysis.

The internal and external energies for any dynamic analysis are shown in the energy plots. The internal and external energies are also shown in the ECHO file. The energy error should be small (no larger than a few percent). A larger error (larger than about 5%) indicates that the analysis may be inaccurate.

The error will tend to be larger for structures with brittle strength loss. The error may also be larger for structures with friction-pendulum dampers. These components can be sensitive numerically, and to get accurate results you may have to use a time step that is shorter than you would need for other structures.

## 23.3 *Approximate Damping Ratio*

---

### 23.3.1 Overview

The concept of a (modal) damping ratio, as a percentage of critical damping, is strictly applicable only for linear structural behavior. However, it is possible to calculate an approximate equivalent damping ratio for nonlinear behavior. This section describes the procedure.

### 23.3.2 Theory

If a linear single DOF structure with viscous damping is subjected to forced sinusoidal vibration at a constant amplitude, energy is dissipated in each cycle. The relationship between the dissipated energy in a complete cycle and the maximum strain energy for the cycle is :

$$\frac{\text{Dissipated Energy}}{\text{Max. Strain Energy}} = 4\pi\zeta \quad (22.1)$$

where  $\zeta$  = damping ratio.

If a linear single DOF structure is subjected to earthquake loading, the strain energy will vary with time as indicated in Figure 23.3(a). Each strain energy peak corresponds to a half cycle. If the dissipated energy is known for a whole number of half cycles, the damping ratio can be calculated as

$$\zeta = \frac{1}{2\pi N} \left( \frac{\text{Dissipated Energy}}{\text{Mean of Strain Energy Peaks}} \right) \quad (22.2)$$

where N is the number of half cycles (strain energy peaks).

For a multi-DOF nonlinear structure the strain energy variation is likely to be more complex, as indicated in Figure 23.3(b). However, a rough estimate of the effective damping ratio can be calculated as

$$\zeta = \frac{1}{2\pi N} \left( \frac{\text{Dissipated Energy}}{2(\text{Mean Strain Energy})} \right) \quad (22.3)$$

which assumes that the mean of the strain energy peaks is 2 times the mean strain energy.

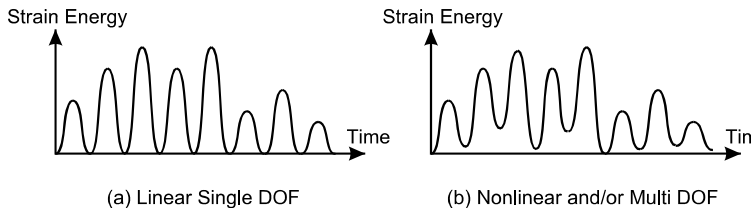


Figure 23.3 Strain Energy Variation for Earthquake Loading

### 23.3.3 Procedure

The above theory is implemented in PERFORM-3D.

Use the **Energy Balance** task. Choose the load case and press **Plot** to plot the energy balance for the structure. You can then enter data defining a range of times (from T1 to T2 seconds) and the number of strain energy peaks over that range.

In the energy plot, choose a strain energy trough and note the corresponding time (move the cursor to the trough to show the time). Then count forward a number of strain energy peaks, and note the time at another trough. Enter the times and the number of peaks, and press **Calculate** to show approximate damping ratios over the range. Ratios are calculated for inelastic energy dissipation and for the four possible types of viscous damping, namely modal,  $\alpha M$ ,  $\beta K$  and fluid dampers.

## 24 Deflected Shape Plots

*Deflected shape plots can be extremely useful for studying the behavior of a structure. A deflected shape plot is even more useful when the elements are colored to show usage ratios. This chapter explains the procedure.*

### 24.1 Deflected Shape

Figure 24.1 shows a deflected shape plot at a point in a dynamic earthquake analysis. You can also draw deflected shapes for gravity and push-over analyses. You can draw the deflected shape for any step in an analysis, and you can animate the deflected shape if you wish.

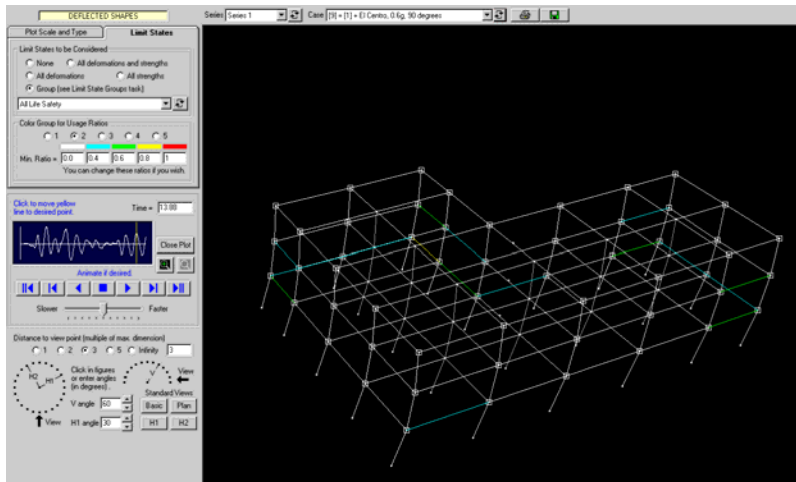


Figure 24.1 Deflected Shape Plot

You have two overall options, as follows.

- (1) Draw the deflected shape only. This can be useful for studying such things as concentration of story drift.
- (2) Draw the deflected shape, calculate usage ratios for chosen limit states, and assign colors to the elements based on their usage ratios. This is valuable for identifying the critical elements in a structure. Be warned that it requires a lot of computation to process the results for this type of plot, and for a large structure there may be a significant delay before plotting begins.

In both cases you can animate the plot if you wish.

## **24.2 Procedure**

---

### 24.2.1 Deflected Shape Only

To start a deflected shape plot press the toolbar buttons for the **Analysis** phase and the **Deflected Shapes** task. In the **Plot Scale and Type** page choose the **Entire elements** option (see later for an explanation of this). Also choose the analysis series from the **Series** list and the load case from the **Case** list.

When you choose the load case a "thumbnail" view of the structure response appears. For a dynamic earthquake analysis this is a plot of reference drift vs. time. For a push-over analysis it is a plot of base shear vs. drift. For a gravity analysis it is a plot of load factor vs. load step number.

To plot deflected shapes only, without usage ratio coloring, the steps are as follows.

- (1) Choose the method for scaling structure displacements. Specify either a displacement scale factor or the ratio of maximum displacement to maximum structure dimension.
- (2) Select the point in the analysis for the deflected shape. The thumbnail plot has a yellow vertical line. To position this line at a point in the plot, click that point. You can do this before or after the deflected shape is drawn. The default is the shape at the beginning of the analysis.

- (3) Press the **Plot** button to draw a static deflected shape at the selected point.
- (4) To animate the plot use the animation buttons. You can move the plot forwards or backwards a single step at a time, jump to the beginning or end of the analysis, and animate forwards or backwards. To control the speed of the animation use the slider. On the fastest setting the animation for dynamic analyses is in real time (but only if your computer is fast enough).

### 24.2.2 Deflected Shape with Usage Ratio Coloring

To add usage ratio coloring you must choose a set of limit states. To do this choose the **Limit States** tab and do the following.

- (1) Choose the limit states that are to be considered. Usually you will choose a limit state group.
- (2) Choose a color group. There are 5 groups with pre-set usage ratio ranges. You can change the ranges for any color group if you wish.
- (3) Press the Plot button and continue as before. As noted above, it may take a little time before the first plot appears.

If you choose the **Entire elements** option in the **Plot Scale and Type** page, the color of each element depends on the usage ratio for that element. In the common case where a beam or column element has plastic hinges at both ends, the element usage ratio is the ratio for the “worst” hinge, and the plot does not show which hinge is critical.

If you choose the **Components** option in the **Plot Scale and Type** page you will get a plot that does show which hinge is critical. With this option all elements are plotted, but only frame-type (beam, column and brace) elements are colored, and the basic components that make up each element are colored rather than the entire element. Moment and PMM hinges are shown as circles. Shear and axial hinges are shown as rectangles. Fiber section segments are shown as thick lines.

### 24.2.3 Some Points on Components vs. Entire Element

- (1) For the *Components* option, only deformation limit states are considered. For the *Entire Elements* option, deformation and/or strength limit states are considered.
- (2) The plots do not use hidden line removal. If you use the *Components* option and many frame-type elements are plotted, the circles for hinge components may overlap in strange ways. It is usually best to set up Frames that are either plane or do not include many elements, and to plot the results for those frames.
- (3) For FEMA Beam and FEMA Column components, each component is divided (by PERFORM) into a hinge component and an elastic segment, in such a way that the FEMA component has the required relationship between end moment and end rotation. In the plot, FEMA components have circles at their ends, as for moment or PMM hinges. The color of each circle is based on the usage ratio for the corresponding FEMA component.
- (4) You can make printer plots for both options, but you can save the results to a text file only for the *Entire Elements* option. Use the buttons on the tool bar.
- (5) You can toggle between the two options without closing the plot.
- (6) To zoom in or out, use the buttons below the Plot / Close Plot button.
- (7) Frame-type elements are drawn using straight lines, not their actual deflected shapes. To see actual deflected shapes, use the Moment and Shear Diagrams task.

### 24.2.4 Other Aspects

If you have defined any frames, there will be a list of frames on the toolbar. If you choose a frame, only the elements in that frame are plotted.

You can zoom in. The zoom-in and zoom-out buttons are in the **Plot Scale and Type** page, not on the toolbar.



In deflected shape plots, the elements are drawn using straight lines only. You can examine the detailed deflected shapes for individual beam, column and brace elements using the Moment and Shear Diagrams task.

### 24.2.5 Save Usage Ratios to a File

If the save to file button on the toolbar is enabled (colored green), you can save the usage ratios to a text file by pressing this button.

You will be asked to input a file name and description. By default the file is saved in the *User* folder (see *Chapter 2, Structures, Folders and Files*). The file contains a description of the data, the element number, the coordinates of node I, and the usage ratio for each element.

The saved usage ratios are for the current deflected shape. For example, if you are considering a dynamic earthquake analysis the usage ratios are for the time that you have currently chosen. To get the final values for the earthquake, be sure to go to the end of the earthquake and draw the color-coded deflected shape at that time. For a push-over analysis you may want to output usage ratios at drifts other than the drift at the end of the analysis (usually you will output them at the estimated drift demand as calculated in the General Push-Over Plot or Target Displacement Push-Over Plot task).



## 25 Time History Plots

---

*Time history plots are useful for checking the behavior of a structure. You can draw time history plots for a variety of node and element results, and also for drifts and structure sections. You will probably use these plots mainly for dynamic earthquake analyses. However, you can also draw plots for gravity analyses (in which case "time" is load factor) and push-over analyses (in which case "time" is reference drift).*

*This chapter explains the procedure.*

### 25.1 Plot History for a Single Node

---

#### 25.1.1 Options

For node results you can plot histories of the following. Velocities and accelerations can be plotted only for dynamic analyses.

- (1) The H1, H2 or V translation or rotation of any node. This is displacement relative to the ground.
- (2) The H1, H2 or V translational or rotational velocity of any node. This is velocity relative to the ground.
- (3) The H1, H2 or V translational or rotational relative acceleration of any node. This is acceleration relative to the ground.
- (4) The H1, H2 or V translational or rotational absolute acceleration of any node. This is the acceleration relative to the ground plus the ground acceleration.

If you choose absolute acceleration and select a supported node you will plot the ground acceleration. This enables you to check that the earthquake load has been specified correctly.

If you plot the absolute acceleration for any node, you can then plot response spectra for that acceleration record. For a node in a floor, this gives the floor response spectrum. For a node at the ground it gives the spectrum for the ground motion.

### 25.1.2 Procedure

For a single node history plot, press the toolbar buttons for the **Analysis** phase and the **Time History** task. In the Time History form choose the **Node** and **Single Node** tabs.

In the toolbar choose the analysis series from the **Series** list and the load case from the **Case** list.

The steps are as follows.

- (1) Choose the result type to be plotted.
- (2) Choose the translation or rotation direction.
- (3) Specify the displacement unit.
- (4) Select the node, by clicking on it in the graphics panel. To make a new selection press the **Clear** button and repeat.
- (5) Press the **Plot** button.

After you press the **Plot** button for absolute accelerations, you can enter damping ratios and other data for a set of corresponding response spectra. Press the **Plot Spectrum** button to plot the spectra.

### 25.1.3 Save Plotted History to a File

If the save to file button on the toolbar is activated (colored green) , you can save the current history to a text file by pressing this button. You can then process this file using a spreadsheet program.

You will be asked to input a file name and description. By default the file is saved in the *User* folder (see *Chapter 2, Structures, Folders and Files*). The file contains a description of the data followed by two columns of X-Y values.

## ***25.2 Save Histories for Multiple Nodes***

---

If you want to save histories for several nodes you can plot and save them individually, but this can be inconvenient. A quicker method is to choose the **Multiple Nodes** tab.

The steps are as follows.

- (1) Choose the result to be plotted and the displacement unit.

- (2) Select the nodes, by clicking on individual nodes or using one or more box selects. To make a new selection press the **Clear** button and repeat.
- (3) Choose whether to save complete histories or maximum and minimum values only.
- (4) Press the **Save** button.

You will be asked to input a file name and description. By default the file is saved in the *User* folder (see *Chapter 2, Structures, Folders and Files*). The file contains a description of the data followed by the results.

## ***25.3 Plot History for a Single Element***

---

### 25.3.1 Procedure

For a single element history plot, press the toolbar buttons for the **Analysis** phase and the **Time History** task. In the Time History form choose the **Element** and **Single Element** tabs.

In the toolbar choose the analysis series from the **Series** list and the load case from the **Case** list.

The steps are as follows.

- (1) Choose the element group from the list. The elements in the chosen group are shown light blue.
- (2) Select the element, by clicking on it in the graphics panel. Its color changes to red. To make a new selection press the **Clear** button and repeat.
- (3) Choose whether you want to plot results for the element as a whole, a structural component or a strength section. Some choices may not be available. For example, you can choose the strength section option only for frame type elements.
- (4) For component or strength section results, choose the component or strength section type. This depends on the element type, as follows.

## Chapter 25. Time History Plots

- (a) If the element consists of a single basic component the component type is shown in the component type list, and you can choose only this component.
  - (b) If the element is a frame type it can have several basic components and several strength sections. These are shown in the list in their order along the element. Choose the component or strength section. If the element has, for example, two plastic hinge components, take care to choose the correct component.
  - (c) If the element is a shear wall element it has a bending component and a shear component. Choose the component.
  - (d) If the element is a general wall element it has two bending components and two shear components. Choose the component.
  - (e) If the element is a viscous bar it has a viscous damper and an elastic bar component. Choose the component.
- (5) Choose the result type for the chosen component or strength section. The choices depend on the component type.
- (6) If desired specify the length and force units.
- (7) Press the **Plot** button to draw the graph. The X axis is time, reference drift or load factor, depending on the load case type. The Y axis is the result quantity.

### 25.3.2 Save Plotted History to a File

If the save to file button on the toolbar is activated (colored green) , you can save the current history to a text file by pressing this button. You can then process this file using a spreadsheet program.

You will be asked to input a file name and description. By default the file is saved in the *User* folder (see *Chapter 2, Structures, Folders and Files*). The file contains a description of the data followed by two columns of X-Y values.

## ***25.4 Save Histories for Multiple Elements***

---

If you want to save histories for several elements you can plot and save them individually, but this can be inconvenient. A quicker method is to choose the **Multiple Elements** tab.

The steps are as follows.

- (1) If you are in a Complete Structure view, the elements in the current Element Group will be shown light blue. If you are in a Frame view, the elements in the current Element Group and current Frame will be shown light blue. The elements for which results will be saved are these light blue elements. That is, you select the elements by choosing the Element Group and Frame, not by clicking on elements. If the current view does not show the elements that you want, you must go to the **Add or Delete Frames** task and set up a Frame with the required elements. There is a reason for this. If you are saving results for multiple elements you will probably do it several times, for different analyses. If you set up a Frame with the required elements, you need to define the elements once only, and you do not have to select the elements by clicking on them each time.
- (2) Choose whether you want to save element level or component level results. Since all elements in an Element Group have the same element level results, you can always choose this option. However, Frame type elements can have different sequences of basic components along their lengths. You can save component level results only if all of the selected elements have similar basic component sequences. If you choose component level results and the elements are not similar, you will get an error message. You must go to the **Add or Delete Frames** task and set up a Frame with compatible elements.
- (3) Choose the result that you want to save, and the units.
- (4) Press the **Save** button and follow the instructions for saving the results.

## ***25.5 Drift or Deflection Histories***

---

For drift or deflection histories, press the toolbar button for the **Analysis** phase and the **Time History** task. In the time history form choose the **Drift/Deflection** tab, and either the **Single** or **Multiple** tab. In the toolbar choose the analysis series from the **Series** list and the load case from the **Case** list.

For a single Drift or Deflection, choose the drift or deflection from the list, and press **Plot** to draw the graph. If the save to file button on the toolbar is activated (colored green), save the current history by pressing this button.

For multiple Drifts or Deflections, highlight the required drifts or deflections in the table, and press **Save**.

Drifts are dimensionless. For Deflections you can choose the displacement unit.

## ***25.6 Structure Section Force Histories***

---

To start a structure section plot, press the toolbar button for the **Analysis** phase and the **Time History** task. In the time history form choose the **Structure Section** tab, and either the **Single Sect** or **Multiple Sects** tab. In the toolbar choose the analysis series from the **Series** list and the load case from the **Case** list.

For a single Structure Section, choose the result type and units, choose the section from the list, and press **Plot** to draw the graph. If the save to file button on the toolbar is activated (colored green), save the current history by pressing this button.

For multiple Structure Sections, choose the result type and units, highlight the required sections in the table, and press **Save**.



## ***25.7 Plot Section Results for Multiple Load Cases***

---

### 25.7.1 Purpose

You can also plot Structure Section results (e.g. base shears) for a single section and multiple load cases. This applies only for static push-over load cases.

You can use this to do the following.

- (1) Compare push-over curves for different lateral load cases. One restriction is that the load cases must be in the same Analysis Series. This means, for example, that you can not directly compare push-over curves with and without P-delta effects. To compare such curves you must save the histories to text files, then combine and plot these files in a spreadsheet program.
- (2) Plot hysteresis loops for story shear. To do this you must have used a General Load Sequence and applied cyclic push-over loads. If you then plot the Structure Section force corresponding to the story shear, you will get a hysteresis loop.

### 25.7.2 Procedure

The steps are as follows.

- (1) In the **Analysis** phase choose the **Time Histories** task.
- (2) Choose the **Structure Section** and **Multiple Loads** tabs.
- (3) Choose the result type and units.
- (4) Choose the Structure Section from the list.
- (5) Highlight the required push-over load cases in the table.
- (6) Press **Plot**.



## 26 Hysteresis Loop Plots

---

*Hysteresis loop plots are useful for checking the behavior of inelastic components. This chapter explains the procedure.*

### 26.1 Purpose

---

Hysteresis loop plots are useful for checking the behavior of inelastic components (and also viscous dampers). Elastic components do not have hysteresis loops.

You will usually plot hysteresis loops for dynamic earthquake load cases, where components are subjected to cyclic loading. You can also plot them for gravity and push-over load cases if you wish. Since there is usually no cyclic loading for these load cases, the hysteresis loops are usually just action-deformation plots with monotonically increasing values.

PERFORM-3D uses a specific model for hysteresis loops. For an explanation of this model see the document *PERFORM Components and Elements*.

### 26.2 Procedure

---

To start a hysteresis loop plot press the toolbar buttons for the **Analysis** phase and the **Hysteresis Loops** task. In the Hysteresis Loops form choose the analysis series from the Series list and the load case from the Case list.

When you choose the load case a "thumbnail" view of the structure response appears. For a dynamic earthquake analysis this is a plot of reference drift vs. time. The steps are then as follows.

- (1) Choose the element group from the list. The elements in the chosen group are shown white.

- (2) Select the element, by clicking on it in the graphics panel. Its color changes to red. To make a new selection click the **Clear** button and repeat.
- (3) Choose the component type. This depends on the element type, as follows.
  - (a) If the element consists of a single basic component the component type is shown in the component type list, and you can choose only this component.
  - (b) If the element is a frame type element it can consist of several basic components. These are shown in the list in their order along the element. Choose the component. If the element has, for example, two plastic hinge components, take care to choose the correct component.
  - (c) If the element is a shear wall element it has a bending component and a shear component. Choose the component.
  - (d) If the element is a general wall element it has two bending components and two shear components. Choose the component.
  - (e) If the element is a viscous bar it has a viscous damper and an elastic bar component. Choose the component.
- (4) Choose the force-deformation type for the hysteresis loop. The choices depend on the component type. There are no choices for elastic components, since they do not have hysteresis loops. For a viscous damper you can plot force vs. deformation, which is the hysteresis loop, or force vs. deformation rate. This is not a hysteresis loop, but it is a useful plot.
- (5) If desired specify the length and force units.
- (6) Select the starting and ending points for the plotted loop. The thumbnail plot has a green vertical line to mark the starting point and a red line to mark the ending point. To position the green line at a point in the plot, left-click that point. To position the red line, right click. You can do this before or after the loop is plotted.

- (7) Press the **Plot** button to draw the loops. The starting point for the selected range is marked by a solid circle, and the ending point by a solid square.
- (8) To expand or contract the loops, left and right click in the thumbnail plot to move the starting and ending points. For fine tuning, or to watch the loop develop, move the starting and ending points up or down one step at a time using the green and red arrow buttons.

### 26.3 Cut-Off Corners

---

A plotted hysteresis loop may have corners as shown in Figure 26.1. This is not an error. PERFORM-3D saves the analysis results only at the end of each time step. If there is an event in a step the true force-deformation path is as shown by the dashed line in Figure 26.1. However, since only the points at the step ends are plotted, the corner is cut off.

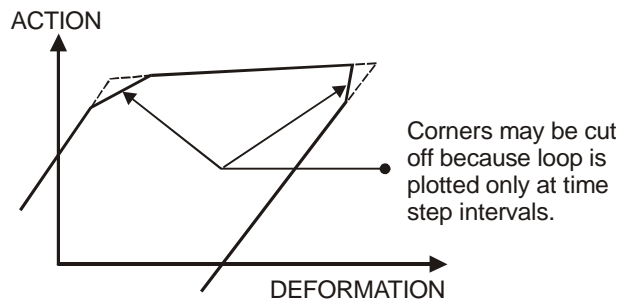


Figure 26.1 Loop With Cut-Off Corners

### 26.4 Save Loops to a File

---

If the save to file button on the toolbar is activated (colored green), you can save the current set of hysteresis loops to a text file, by pressing

## Chapter 26. Hysteresis Loop Plots

this button. You can then process this file using a spreadsheet program. This can be useful if, for example, you want to plot two sets of loops on the same graph to compare them.

You will be asked to input a file name and description. By default the file is saved in the *User* folder (see *Chapter 2, Structures, Folders and Files*). The file contains a description of the data followed by two columns of X-Y values.

## 27 Moment and Shear Diagrams

---

*The Moment and Shear Diagrams task allows you to draw shear force and bending moment diagrams and envelopes for a single element or a line of elements (but not an entire frame), to draw shear force and bending moment diagrams based on cross section forces (for example, plot a shear force diagram for a shear wall), to draw detailed deflected shapes for single frame type elements, and to show the gravity loads on frame type elements. The diagrams can be animated if desired.*

*This chapter explains the procedures.*

### 27.1 Detailed Results for a Single Element

---

#### 27.1.1 Element Selection

Press the toolbar buttons for the **Analysis** phase, the **Moment and Shear Diagrams** task and the **Single Element** page. Choose the analysis series from the Series list in the toolbar, and the load case from the Case list.

Select an element as follows.

- (1) Choose an element group from the list. Only frame type element groups are listed. In the graphics panel the chosen group is shown light blue.
- (2) Select the element, by clicking on it in the graphics panel. Its color changes to red. To make a new selection press the **Clear** button and repeat.
- (3) Choose the units.
- (4) Choose the result type. The options are (a) element loads, (b) moment and shear envelopes, (c) moment and shear histories and (d) deflected shape. See the following sections for the procedure for each option.

### 27.1.2 Element Loads Option

With the element loads option, to show the loads on the selected element press the **Plot** button. For a gravity load case these are the loads at the end of the analysis. You can apply element loads only in gravity load cases. Hence, for push-over and dynamic earthquake load cases the element loads are constant.

In these plots all elements are plotted horizontal. Since only gravity loads are allowed on frame elements, the loads are in the vertical direction, not necessarily along Axis 2 or Axis 3 of the element.

To change the element press **Clear** and select a new element.

### 27.1.3 Moment and Shear Envelopes Option

With the shear and moment envelopes option, to plot envelopes for the selected element press the **Plot** button. For a gravity load case the plot shows the bending moment and shear force diagrams at the end of the analysis. For push-over and dynamic earthquake cases the plot shows the envelopes at the end of the analysis. The diagram in the form shows the sign convention. Use the option buttons at the top of the plot to toggle between the element 1-2 and 1-3 planes.

Use the option buttons at the top of the plot to toggle between the element 1-2 and 1-3 planes.

In these plots all elements are plotted horizontal.

To change the element press **Clear** and select a new element.

### 27.1.4 Moment and Shear Histories Option

For the moment and shear history option a "thumbnail" view of the structure response appears. For a dynamic earthquake analysis this is a plot of reference drift vs. time. For a push-over analysis it is a plot of base shear vs. reference drift. For a gravity analysis it is a plot of load factor vs. load step number.

To draw moment and shear diagrams proceed as follows.

- (1) Choose the force and length units.



- (2) Select a point in the analysis for the diagrams. The thumbnail plot has a yellow vertical line. To position this line at a point in the plot, click that point.
- (3) Press the **Plot** button to draw the diagrams at the selected point.
- (4) To animate the plot use the animation buttons. You can move the plot forwards or backwards a single step at a time, jump to the beginning or end of the analysis, and animate forwards or backwards. To control the speed of the animation use the slider.

Use the option buttons at the top of the plot to toggle between the element 1-2 and 1-3 planes.

In these plots all elements are plotted horizontal.

To change the element press **Clear** and select a new element.

### 27.1.5 Deflected Shapes Option

In the Deflected Shapes task, elements are plotted as straight lines. With the deflected shapes option you can draw detailed deflected shapes, showing element curvatures and hinge rotations.

For the deflected shapes option a "thumbnail" view of the structure response appears. For a dynamic earthquake analysis this is a plot of reference drift vs. time. For a push-over analysis it is a plot of base shear vs. reference drift. For a gravity analysis it is a plot of load factor vs. load step number.

To draw deflected shapes proceed as follows.

- (1) Specify a displacement scale factor.
- (2) Specify whether you want the relative end displacement to be included in the deflected shape. If you are not sure what this means, experiment.
- (3) Select the point in the analysis for the deflected shape. The thumbnail plot has a yellow vertical line. To position this line at a point in the plot, click that point.

- (4) Press the **Plot** button to draw a static deflected shape at the selected point.
- (5) To animate the plot use the animation buttons. You can move the plot forwards or backwards a single step at a time, jump to the beginning or end of the analysis, and animate forwards or backwards. To control the speed of the animation use the slider. On the fastest setting the animation is in real time (if your computer is fast enough).

Use the option buttons at the top of the plot to toggle between the element 1-2 and 1-3 planes.

In the plots, elements that are inclined less than 45 degrees to the horizontal are plotted horizontal, and elements that are more steeply inclined are plotted vertical.

To change the element press **Clear** and select a new element.

## ***27.2 Moment and Shear Diagrams for a Line of Elements***

---

Press the toolbar buttons for the **Analysis** phase, the **Moment and Shear Diagrams** task and the **Line of Elements** page. Choose the analysis series from the Series list in the toolbar, and the load case from the Case list.

The procedure is as follows.

- (1) Choose an element group from the list. Only frame type element groups are listed. In the graphics panel the chosen group is shown light blue.
- (2) In the graphics panel select a line of elements, by clicking the elements individually and/or using box selects. The selected elements must be in a continuous line, usually either a horizontal or vertical line. The selected elements are shown red.
- (3) Choose the units.

- (4) Choose the result type. The options are (a) moment and shear envelopes and (b) moment and shear histories.
- (5) For the histories option, select a point in the analysis for the first plot. The thumbnail plot has a yellow vertical line. To position this line at a point in the plot, click that point.
- (6) Press the **Plot** button.
- (7) To animate a history plot use the animation buttons. You can move the plot forwards or backwards a single step at a time, jump to the beginning or end of the analysis, and animate forwards or backwards. To control the speed of the animation use the slider.

The moment and shear diagrams are plotted on horizontal axes, even for a vertical line of columns.

### ***27.3 Moment and Shear Diagrams Based on Structure Sections***

---

To draw moment and shear diagrams based on structure sections, press the toolbar buttons for the **Analysis** phase, the **Moment and Shear Diagrams** task and the **Section Group** page.

For the remainder of the procedure, see *Chapter 9, Structure Sections, Section 9.5*. You must have specified one or more structure section groups.



## 28 General Push-Over Plots

---

*There are a number of different methods for performance assessment using static push-over results. FEMA 440 has proposed a new Linearization Method, and also a number of changes to the FEMA 356 Coefficient (Displacement Modification) Method. The ATC 40 Capacity Spectrum Method is outdated, but the Capacity Spectrum Method in general is still useful, and has some advantages over other methods.*

*The General Push-Over Plot task that implements all of these methods. This Chapter explains the procedures, illustrates the differences among the methods and covers some essential theory.*

*This Chapter also shows how the Capacity Spectrum Method can be used to assess the effects of fluid dampers.*

### 28.1 Overview

---

#### 28.1.1 Available Methods

You can choose from the following methods.

- (1) FEMA 440 Linearization Method.
- (2) FEMA 440 modification of the Coefficient Method (also known as the Displacement Modification Method).
- (3) FEMA 356 Coefficient Method.
- (4) Capacity Spectrum Method, with options for the ATC 40 procedure or a modified procedure that may be more accurate.

The FEMA 356 Coefficient Method is also implemented, in a somewhat different way, but with the same results, in the **Target Displacement** task (see *Chapter29, Target Displacement Plot*).

#### 28.1.2 Differences Among Methods

All of the push-over methods have the same main steps, as shown in Figure 28.1

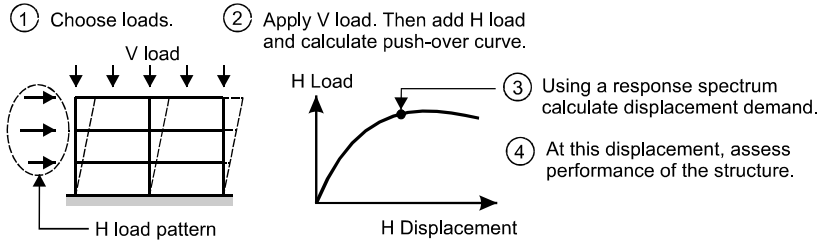


Figure 28.1 Main Steps for Static Push-Over

However, the methods differ substantially in their details. Figures 28.2 through 28.5 illustrate the differences. Each method starts with a push-over curve that relates lateral load,  $H$ , to horizontal displacement,  $\Delta$ . In each figure, follow Steps 1, 2, 3, etc. for the computation sequence.

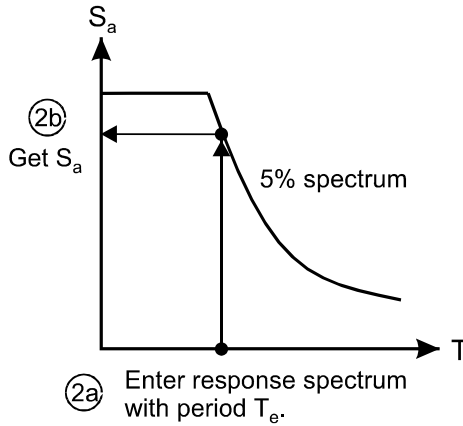
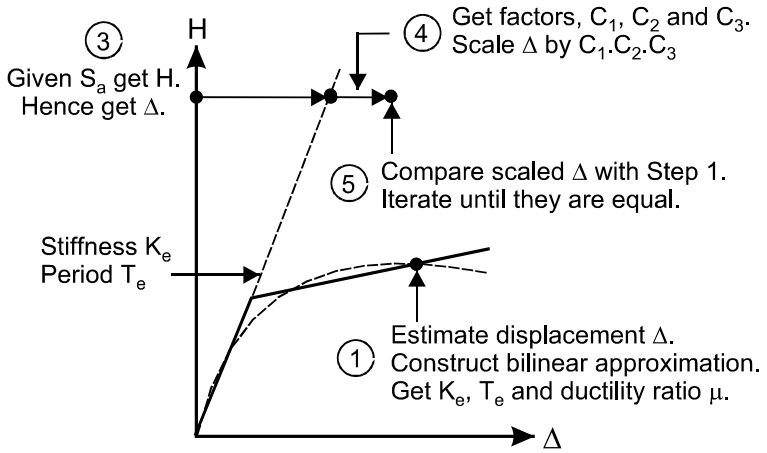


Figure 28.2 Steps in FEMA 356 and FEMA 440 Coefficient Methods

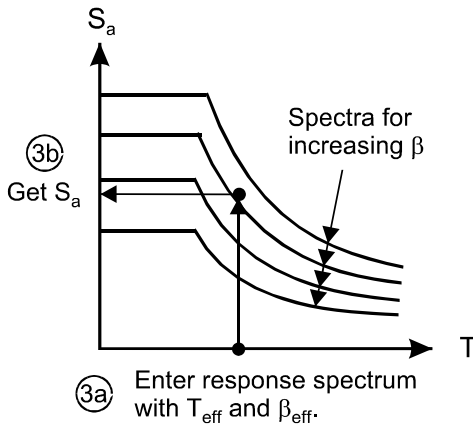
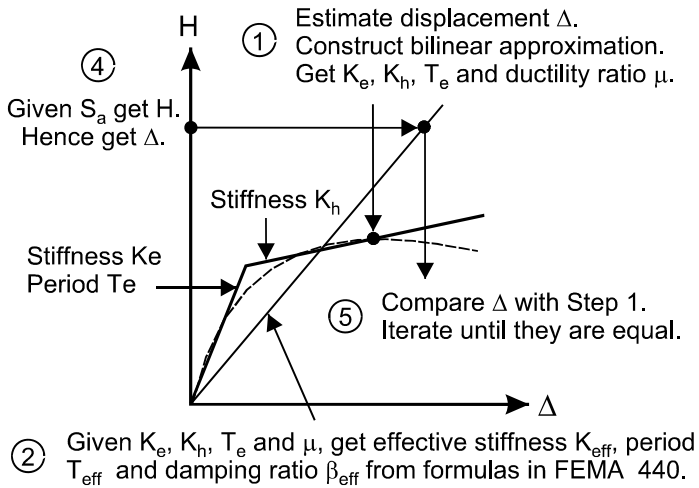


Figure 28.3 Steps in FEMA 440 Linearization Method



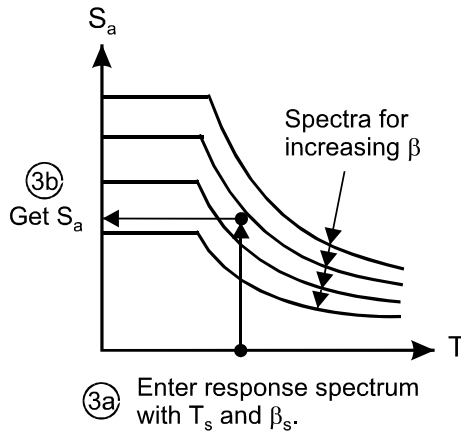
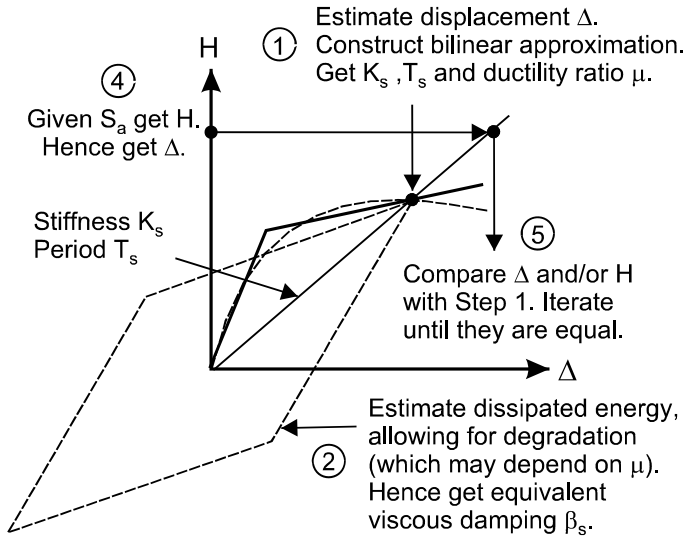


Figure 28.4 Steps in ATC 40 Capacity Spectrum Method

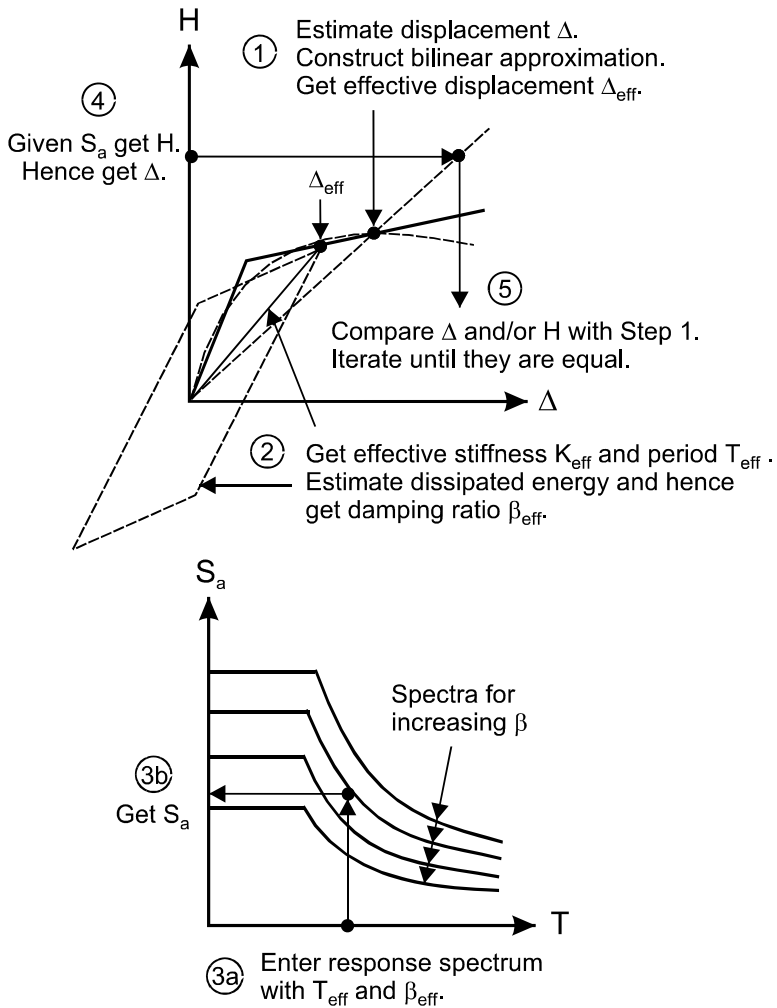


Figure 28.5 Steps in Modified Capacity Spectrum Method

### 28.1.3 Some Key Points

The following are some key points.

- (1) The Capacity Spectrum and Linearization Methods require a response spectrum family, with spectra for a number of damping ratios. The Coefficient Method uses only one spectrum, usually for 5% damping.
- (2) The Coefficient and Linearization Methods use empirical formulas that are obtained by calibration against a large number of dynamic analyses. Empirical formulas are used in the Coefficient Method for the coefficients C1, C2 and C3, and in the Linearization Method for the period  $T_{eff}$  and damping ratio  $\beta_{eff}$ .
- (3) The ATC 40 Capacity Spectrum Method can be interpreted as an extension of response spectrum analysis for a linear structure, but using the secant period and damping ratio rather than the elastic period and 5% damping. With this interpretation, this method is rational rather than empirical (although the implementation has empirical features). The Capacity Spectrum Method is also more flexible than the purely empirical methods. For example, it can be used to assess the effect of additional energy dissipation in fluid damper components (this is explained in a later section).
- (4) It has been shown that the ATC 40 Capacity Spectrum Method is not as accurate as the Coefficient and Linearization methods (see, for example, FEMA 440). It can be argued that this is not a fair comparison, because the Coefficient and Linearization Methods have both been calibrated against dynamic analysis, and the Capacity Spectrum Method has not. The Capacity Spectrum Method has empirical parameters, and if these parameters were calibrated against dynamic analysis, it might be as accurate as the other methods, while still retaining its rational flavor.

One reason why the Capacity Spectrum Method is inaccurate may be that it uses the secant period and damping. This is equivalent to assuming that in an earthquake the structure will experience several cycles with displacements equal to the maximum displacement. It is more likely that there will be only a few cycles with amplitudes close to the maximum, with more cycles at smaller amplitudes. Intuitively, it should be more accurate to use an effective period and damping, based on a displacement that is smaller than the maximum.

This is the main basis of the Modified Capacity Spectrum Method. In this method, the period and damping are calculated at a displacement that is smaller than the maximum. Based on limited experience, a displacement about 0.8 times the maximum appears to be reasonable. The usual effect is to increase the period and decrease the damping, and hence to increase the displacement demand compared with the ATC 40 Capacity Spectrum Method.

Another cause of inaccuracy may be that the Capacity Spectrum Method explicitly relates dissipated energy to damping ratio. The equation for this relationship (see later) applies for linear analysis but may not be accurate for nonlinear analysis. The accuracy of the method might be improved if the relationship included empirically calibrated parameters.

- (5) This is a philosophical point, but worth noting. When dynamic analysis is used, it is common to consider several ground acceleration records, and to use the average result for performance assessment. The goal of push-over analysis is usually to calculate a displacement demand that matches this average result. This is reasonable if push-over analysis is viewed as an analysis method. However, it is not necessarily wise if push-over analysis is viewed as a design tool. Push-over analysis is approximate. Hence, when it is used as a design tool it should tend to err on the conservative side.

#### 28.1.4 Main Steps

In all of the preceding figures, the displacement demand is calculated by successive trials, until the displacement in the last step matches the estimated demand in the first step. The PERFORM-3D implementation is a little different. In this implementation, a Demand curve is plotted, and the displacement demand is the point where the Demand curve crosses the Capacity (push-over) curve. The calculation steps are essentially the same as in the preceding figures. The end result is a plot as shown in Figure 28.6.

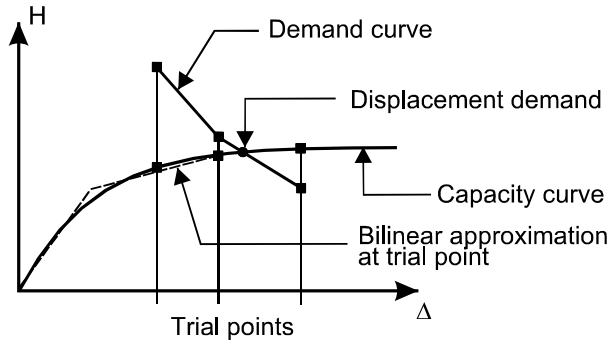


Figure 28.6 Push-Over Plot

In FEMA 356 the displacement demand is the Target Displacement. In ATC 40 the displacement demand locates the Performance Point.

As shown later, this plot can provide useful information on the sensitivity of the displacement demand to changes in the strength and stiffness of the structure.

The main steps are as follows. Detailed steps are given later.

- (1) Choose a push-over load case and plot the Capacity Curve. This is usually a plot of base shear or base shear coefficient vs. reference drift. However, it can be a plot of spectral acceleration vs. spectral displacement.
- (2) Select a number of trial points on the Capacity Curve, and for each point construct a bilinear approximation.
- (3) Choose a push-over method. Depending on the method, choose certain options and specify certain details.
- (4) If you have not done so previously, define one or more new response spectra. If desired, edit existing spectra.
- (5) Choose a response spectrum.
- (6) Plot the Demand Curve. For each trial point a corresponding demand point is calculated. The line passing through the demand points is the demand curve. The point where the demand curve intersects the capacity curve gives the displacement demand (target displacement or performance point).
- (7) Examine the limit state usage ratios at the displacement demand to assess the performance.

## ***28.2 Procedure for Defining Response Spectra***

---

### 28.2.1 Spectrum Types

Each spectrum is actually a spectrum family, with spectra for a number of damping ratios. There are four options for the spectrum shape, as follows.

- (1) *FEMA 356 spectrum.* The shape of this spectrum is as defined in FEMA 356, and the reduction factors specified in FEMA 356 are used to convert the 5% spectrum to spectra for other damping ratios.
- (2) *ATC 40 spectrum.* The shape of this spectrum is as defined in ATC 40, and the reduction factors specified in ATC 40 are used to convert the 5% spectrum to spectra for other damping ratios.
- (3) *Standard shape, user reduction factors.* The shape of the spectrum is essentially the same as that in FEMA 356 (with one extra parameter that can be used to vary the shape), but the user must specify the reduction factors.
- (4) *User shape and reduction factors.* The user must specify the spectrum shape for each of the damping ratios (up to 7 ratios).

The FEMA 440 Linearization Method and the Capacity Spectrum Method require a spectrum family. The Coefficient method uses only the spectrum for 5% damping.

### 28.2.2 Add or Edit a Spectrum

Press the toolbar buttons for the **Analysis** phase and the **General Push-Over Plot** task. Go to the **Spectra** page.

To add a new spectrum, press **New**, choose the spectrum type, and press **OK**. Input the required data, press **Graph** to plot the spectrum, and **Save** to save it.

Alternatively, if there is an existing spectrum that is similar, choose it from the list, press **Save As** to copy it with a new name, edit the data, and press **Save**.

The data for an existing spectrum can be edited at any time. Choose it from the list, edit the data, and press **Save**. To delete an existing spectrum, choose it and press **Delete**.

### 28.2.3 Copy Spectra from a Different Structure

Any spectra that you define for a given structure are stored with that structure, and are not currently available in other structures. If you have defined a number of spectra for one structure and you want to use them for a different structure, you must jump through a hoop, as follows.

The spectra are stored in a file PF3DPOA in the folder for the current structure. This is not a text file, and it can not be edited. To make the spectra available to another structure, copy this file to the folder for the other structure. If there is already a PF3DPOA file for the other structure, it will be replaced. Hence, if you want to keep the existing file, rename it.

Response spectra are also used in Response Spectrum load cases. The response spectra defined for those load cases can not be used for push-over analysis, and vice-versa.

## ***28.3 Procedure for Capacity and Demand Curves***

---

### 28.3.1 Choose Task

Press the toolbar buttons for the **Analysis** phase and the **General Push-Over Plot** task.

### 28.3.2 Plot Capacity Curve

In the **Capacity** page, choose the curve type. The options are as follows.

- (1) Base shear vs. reference drift.
- (2) Base shear coefficient vs. reference drift.
- (3) Spectral acceleration vs. spectral displacement.

If you choose the spectral acceleration vs. spectral displacement option, you must also choose a method for calculating the spectral acceleration values. See the following sections for an explanation.

You usually will also show limit points on the curve. The limit point for a limit state is the point where the limit state usage ratio equals 1.0. There are several options for the limit states that are to be considered, including limit state groups.

Press **Plot** to draw the capacity curve.

If limit points are plotted, they are shown as short red lines. Click on a line to show the corresponding limit state in the box near the bottom of the **Capacity** page.

At the bottom of the **Capacity** page are boxes for optional Scale Factors and Plot Ranges. See later for an explanation.

### 28.3.3 $S_a$ vs. $S_d$ Option

If you choose the spectral acceleration vs. spectral displacement option, the push-over curve is a plot of spectral acceleration capacity vs. spectral displacement. Spectral acceleration demand is obtained from the response spectrum, as considered later.

The spectral acceleration capacity at any point on a push-over curve depends on the deflected shape of the structure. You have three options, as follows.

- (1) Use the “first” elastic mode shape. For a 2D structure this is indeed the first mode. For a 3D structure it is the mode that has the largest base shear factor along the push-over direction. If you choose this option, PERFORM finds this “best” mode. If you are applying FEMA 356 you will usually choose this option. It is shown in later sections that this may not be consistent theoretically.
- (2) For each point on the push-over curve, use the current deflected shape, and otherwise follow FEMA 356. Arguably the current deflected shape is a better choice than the elastic mode shape. With this option, the spectral acceleration capacity is calculated using, in effect, FEMA 356 Equation 3-16. This is explained in more detail in the next section.
- (3) For each point on the push-over curve, use the current deflected shape, but instead of FEMA 356 Equation 3-16, use an equation that is theoretically more consistent. Again, this is explained in more detail in the next section.

### 28.3.4 Spectral Acceleration Capacity

FEMA 356 Equation 3-16 specifies, in effect, that given the base shear capacity at a point on the push-over curve (where the base shear capacity is equal to the applied horizontal load), the corresponding spectral acceleration capacity is given by



$$S_{a, \text{capacity}} = \frac{1}{C_{m\phi}} \frac{H_{\text{capacity}}}{(W/g)} \quad (3.1)$$

where  $H_{\text{capacity}}$  = base shear,  $W$  = structure weight,  $g$  = gravitational acceleration, and  $C_{m\phi}$  = dimensionless effective mass factor for the “first” mode shape. This factor is the effective mass along the push-over direction divided by the total mass, given by

$$C_{m\phi} = \frac{(\{\phi\}^T [M] \{d\})^2}{\{\phi\}^T [M] \{\phi\}} \cdot \frac{1}{(W/g)} \quad (3.2)$$

where  $\{\phi\}$  = mode shape,  $[M]$  = mass matrix, and  $\{d\}$  = matrix of direction cosines for the push-over direction. If the push-over direction is along the H1 axis, matrix  $\{d\}$  has a unit value for each H1 mass in  $[M]$ , and a zero value for all other masses. Similarly, if the push-over direction is along the H2 axis, matrix  $\{d\}$  has a unit value for each H2 mass in  $[M]$ , and a zero value for all other masses.

There are two potential inaccuracies in this equation, as follows.

- (1) It is accurate only if the deflected shape of the structure is proportional to the mode shape,  $\{\phi\}$ .
- (2) It is accurate only if the push-over load distribution is proportional to  $[M]\{\phi\}$ .

As shown in Section 3.7, if the applied push-over load at a point on the push-over curve is  $\{R\}$  and the deflected shape is  $\{r\}$ , a more consistent equation is

$$S_{a, \text{capacity}} = \frac{(\{R\}^T \{r\})(\{r\}^T [M] \{d\})}{(\{R\}^T \{d\})(\{r\}^T [M] \{r\})} \frac{1}{C_{mr}} \frac{H_{\text{capacity}}}{(W/g)} \quad (3.3)$$

where

$$C_{mr} = \frac{(\{r\}^T [M] \{d\})^2}{\{r\}^T [M] \{r\}} \cdot \frac{1}{(W/g)} \quad (3.4)$$

For the special case where the loads,  $\{R\}$  are proportional to  $[M]\{r\}$ , Equation 3.3 reduces to

$$S_{a, capacity} = \frac{1}{C_{mr}} \frac{H_{capacity}}{(W / g)} \quad (3.5)$$

If the current deflected shape is proportional to the mode shape,  $\{\phi\}$ , Equation 3.5 becomes Equation 3.1. Since push-over methods are approximate, Equation 3.1 can be an acceptable approximation.

Equation 3.3 may seem complex. However, since  $\{R\}^T \{d\} = H$ , this equation reduces to

$$S_{a, capacity} = \frac{\{r\}^T \{R\}}{\{r\}^T [M] \{d\}} \quad (3.6)$$

If you choose the first option for the deflected shape ("first" mode shape), PERFORM calculates the spectral acceleration capacity at any point on the push-over curve using Equation 3.1. If you choose the second option (current deflected shape with FEMA 356 Equation 3-16), PERFORM uses Equation 3.5. If you choose the third option (current deflected shape with more consistent equation), PERFORM uses Equation 3.6.

### 28.3.5 Define Trial Points

Go to the **Points** page. You can define up to 6 trial points, and you must define at least 2 points. The properties at the currently defined points are shown in the table.

There are two ways to define a new point, namely "click-and-drag" and "right-click". The click-and-drag method is more general. The right-click method is a lot easier.

To use the click-and-drag method, move the cursor in the graph to the desired drift or displacement value (D), then left click, hold, and drag to set up a bilinear approximation. For the FEMA 356 approximation, the areas under the bilinear and actual curves must be equal, and the initial slope of the bilinear curve must intersect the actual curve at 60% of the distance to the yield point. As you drag the bilinear curve, the area ratio is shown and there is a small circle at the 60% point. Drag until the area

ratio is very close to 1.0 (a value between 0.998 and 1.002 is suggested) and the small circle lies on the actual curve. After you release the mouse button the bilinear curve properties are shown highlighted in the table. If this approximation is OK, press **OK** (actually, this is optional – if you wish you can just start the next point). If the approximation is not OK, press **Clear** and repeat.

To use the right-click method, move the cursor in the graph to the desired drift or displacement value (D), then simply right click. A FEMA 356 bilinear approximation will be drawn automatically (in some cases this may not work – if so there will be a message). As before, the bilinear curve properties are shown highlighted in the table. If the approximation is OK, press **OK**, and if not press **Clear**.

To change a point, choose its number, press **Clear**, and define a new point (or just choose its number and define a new point).

The points do not have to be in order of increasing D. Often you will start with only 2 or 3 points to locate the displacement demand approximately, then add more points close to the displacement demand to get better accuracy.

## 28.3.6 Period Calculation

### 28.3.6.1 Purpose

All push-over methods use a vibration period for each trial point, which depends on the structure stiffness. This period, together with a damping ratio, is used to enter the response spectrum to obtain the spectral acceleration demand. This is illustrated in Figure 28.7.

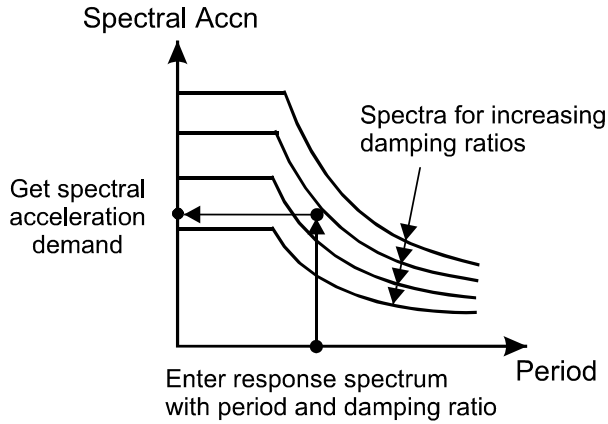


Figure 28.7 Spectral Acceleration Demand

### 28.3.6.2 Stiffnesses

The vibration period depends on the structure stiffness. There are several stiffnesses that may be used. These stiffnesses can be related to slopes in the push-over plot, as shown in Figure 28.8.

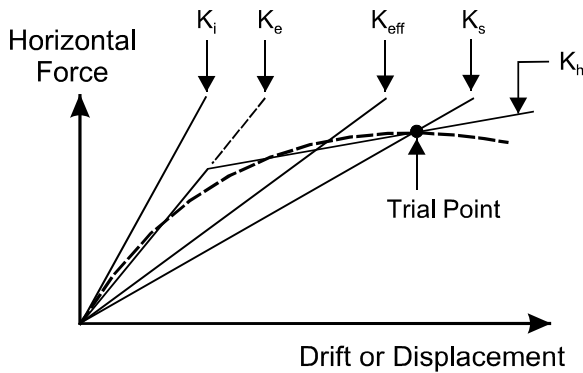


Figure 28.8 Stiffnesses

This figure shows 5 stiffnesses (slopes), as follows.

- (1)  $K_t$  = tangent to the push-over curve at the origin. If the structure remains linear under gravity load, this represents the elastic stiffness of the structure. However, if there is nonlinear behavior such as yield or cracking under gravity load, this may not be an accurate representation of the elastic stiffness.
- (2)  $K_s$  = secant stiffness at the trial point.
- (3)  $K_e$  = elastic stiffness for the bilinear approximation at the trial point. In ATC 40 this stiffness is  $K_0$ .
- (4)  $K_h$  = strain hardening stiffness for the bilinear approximation.
- (5)  $K_{eff}$  = effective stiffness, used by some push-over methods.

### 28.3.6.3 Relationship Between Period and Stiffness

For a single DOF structure, the vibration period is given by the well-known equation

$$T = 2\pi\sqrt{\frac{M}{K}} \quad (3.7)$$

By analogy, if the stiffnesses at two points, A and B, on a push-over curve are  $K_A$  and  $K_B$ , the corresponding periods are related by

$$\frac{T_A}{T_B} = \sqrt{\frac{K_B}{K_A}} \quad (3.8)$$

For example, in Figure 28.8 the period  $T_i$ , corresponding to the initial stiffness  $K_i$ , can be assumed to be the “first” mode elastic period of the structure, which is known. Hence, the period  $T_e$ , corresponding to the stiffness  $K_e$ , can be calculated as

$$T_e = T_i\sqrt{\frac{K_i}{K_e}} \quad (3.9)$$

Alternatively, the period  $T_s$ , corresponding to the secant stiffness  $K_s$ , can be calculated (see below). Hence, the period  $T_e$  can be calculated as

$$T_e = T_s\sqrt{\frac{K_s}{K_e}} \quad (3.10)$$

The Coefficient Method uses the period  $T_e$  to enter the response spectrum, the ATC 40 Capacity Spectrum Method uses the period  $T_s$ , and the FEMA 440 Linearization Method and the Modified Capacity Spectrum Method use an effective period  $T_{eff}$ .

### 28.3.6.4 Options for Period Calculation.

In the **Points** page there is a box with two options for the method of period calculation. The options are as follows.

- (1) Use the elastic mode period. In this case the period  $T_e$  for each trial point is calculated using Equation 3.9. This is the method that is most commonly used. It is specified by FEMA 356, although it is not necessarily the best method.

As noted above, this method can be inaccurate if there is yield or cracking under gravity load. To allow for this case, you have the option of using the initial slope of the push-over curve as the stiffness  $K_i$ , or of specifying  $K_i$  directly. If you choose this second option, you must run a separate push-over analysis for the structure with no gravity load, get  $K_i$  from the initial slope of this curve, and specify this stiffness (slope) directly.

- (2) Use the Rayleigh Quotient. In this case, the secant period  $T_s$  is calculated for each trial point, using the Rayleigh Quotient (see the next section). The period  $T_e$  is then calculated using Equation 3.10. This method has some advantages, and is arguably more consistent theoretically. The Rayleigh Quotient is calculated as follows.

### 28.3.6.5 Rayleigh Quotient

Let the deflected shape at a trial point be  $\{r\}$ . Assume that the structure vibrates with this shape, at a period determined by the secant stiffness at the trial point. The maximum strain and kinetic energies are given by

$$SE = 0.5\{r\}^T [K_s] \{r\} = 0.5\{r\}^T \{R\} \quad (3.11a)$$

and

$$KE = 0.5 \frac{4\pi^2}{T_e^2} \{r\}^T [M] \{r\} \quad (3.11b)$$

where  $[K_s]$  = secant stiffness matrix,  $\{R\}$  = applied push-over loads at the trial point, and  $[M]$  = mass matrix. Equating these energies gives the secant period as

$$T_s = 2\pi \sqrt{\frac{\{r\}^T [M] \{r\}}{\{r\}^T \{R\}}} \quad (3.12)$$

This is the Rayleigh Quotient method. For a given deflected shape it gives the best estimate for the period. It has the advantage that it does not depend on the initial slope,  $K_i$ , of the push-over curve, so it does not matter if there is yield or cracking under gravity load.

If you choose the Rayleigh Quotient method, by default the secant period,  $T_s$ , is calculated, and  $T_e$  is obtained using Equation 3.10. You will usually use this default. However, if you wish you can specify that the Rayleigh Quotient is to be calculated at any point between the bilinear yield point and the trial point. The reasoning is that the deflected shape at a trial point corresponds to the maximum displacement of the structure, and that an effective displacement that is smaller than this maximum may be more representative. A similar argument is used for the Modified Capacity Spectrum Method (see Appendix A). If you specify an effective displacement that is smaller than the displacement at the trial point, Equations 3.12 and 3.10 are modified to use the stiffness and period at the effective displacement. It may be noted that if the displaced shape does not change as the displacements increase, it does not matter which point is chosen.

### 28.3.6.6 Warning on Relationship Between Period and Stiffness

The relationship between period,  $T$ , and stiffness,  $K$ , is accurate only if the structure is well behaved. This is the case if the deflected shape does not change substantially as the deformations increases. It may not be the case if the deflected shape changes substantially. This can be the case if there is brittle strength loss, which can cause local deformations that change the deflected shape. An example is a structure with a brittle bottom story, where the deflected shape can be roughly linear over the building height before the story fails, then can suddenly change to roughly constant over the height.

There can also be inaccuracy if the push-over curve is plotted in terms of reference drift (typically based on roof displacement) and if this drift is not closely related to the masses and the push-over loads. Figure 28.9 shows an example.

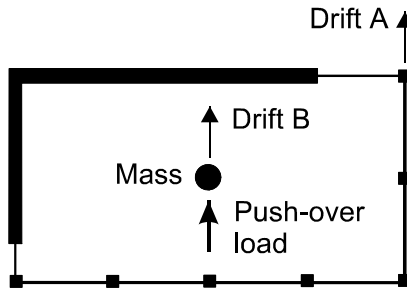


Figure 28.9 Choice of Reference Drift

The structure in this figure is unsymmetrical in plan, and push-over loads applied at the center of mass might cause significant torsional deformation. If the reference drift is Drift A, on one side of the building, this drift may not provide a good representation of the behavior of the building as a whole, and periods based on this drift may not be accurate. A drift such as Drift B is likely to provide a better representation.

As noted later, this effect can be a reason for plotting the push-over curve in terms of spectral acceleration and spectral displacement.

### 28.3.7 Choose Push-Over Method and Response Spectrum

Go to the **Demand** page.

Choose the performance evaluation method from the available options. Then go to the **Details** page and specify the details for the chosen method. These details are essentially as follows.

- (1) For the Coefficient Method, choose the method for calculating the coefficients C1, C2 and C3. There are FEMA 356, FEMA 440 and User-Specified options.



- (2) For the Linearization Method choose the type of hysteresis loop that most closely approximates the H vs.  $\Delta$  loop for structure. FEMA 440 gives equations for effective stiffness and damping ratio for three different hysteresis loops, and a fourth set of equations that are not based on a specific loop. There is also an option for user specified values.
- (3) For the Capacity Spectrum Method, choose an option for energy degradation. To implement the Modified Capacity Spectrum Method specify an Effective Drift Factor. If the structure has fluid dampers, specify an Effectiveness Factor for the energy dissipation in these dampers. These aspects are explained in more detail in later sections.

After specifying the details, return to the **Demand** page. The properties at the trial points are shown in the table in that page. If any user-specified values are to be entered, the corresponding cells in the table are colored beige, and you must enter values for these properties. Otherwise the properties are calculated by PERFORM.

Choose the response spectrum from the list. You must have set up one or more spectra, as described earlier in this Chapter. If you wish you can specify scale factors to be applied to the spectral accelerations and periods.

If the capacity curve is base shear or base shear coefficient vs. reference drift (i.e., if the curve is not spectral acceleration vs. spectral displacement), you must also choose a method for converting spectral acceleration demand (which is obtained from the response spectrum) to base shear demand. The options are as follows.

- (1) Use the “first” elastic mode shape.
- (2) Use the current deflected shape, and otherwise follow FEMA 356.
- (3) Use the current deflected shape, but instead of FEMA 356 Equation 3-16, use a more consistent equation.

These are the same options as for the case where the capacity curve is spectral acceleration vs. spectral displacement, and the theory is exactly the same. The difference is that it is now necessary to convert spectral acceleration demand to base shear demand, rather than base shear capacity to spectral acceleration capacity. For each option the base shear demand at a trial point is calculated as

$$H_{demand} = \frac{S_{a,demand}}{S_{a,capacity}} H_{capacity} \quad (3.13)$$

where  $S_{a,demand}$  is obtained from the response spectrum,  $H_{capacity}$  = base shear capacity at the trial point, and  $S_{a,capacity}$  is calculated as in Section 3.3.4.

Also choose the option for Lines of Constant Period This affects only the sensitivity information contained in the demand curve. It does not affect the displacement demand. The options are (a) lines of constant period are vertical and (b) lines of constant period are radial. The difference is explained in the next section.

### 28.3.8 Plot Demand Curve

To plot the demand curve press **Plot**.

For performance assessment, the key point is where the demand curve crosses the capacity curve. The points on the demand curve are connected by straight lines, and if the trial points are widely spaced the crossing point may be different from the displacement demand. To get the displacement demand more accurately, go to the **Points** page and modify the trial points, then return to the **Demand** page and press **Plot**.

The demand curve also gives useful sensitivity information, as shown in Figure 28.10.

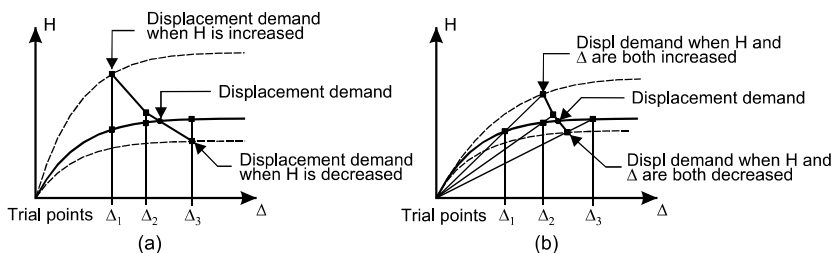


Figure 28.10 Sensitivity Information

In Figure 28.10(a) the capacity curve is changed by changing both the strength and the stiffness, in the same proportion. In this case lines of constant period are vertical. This figure shows how much the strength (and stiffness) of the structure would have to be increased to reduce the displacement demand from its calculated value to  $\Delta_1$ . The figure also shows how much the strength could be decreased if the displacement demand could be increased to  $\Delta_3$ .

In Figure 28.10(b) the capacity curve is changed by changing only the strength, keeping the stiffness constant. In this case lines of constant period are radial.

You can assess the sensitivity to both types of change by selecting the “vertical” or “radial” options for Lines of Constant Period. You can also explore the effects of strength and stiffness changes more directly, as explained in the next section.

### 28.3.9 Capacity Curve Scaling for Sensitivity Studies

The demand curve includes sensitivity information for two special cases, as described in the preceding section. You can do more general sensitivity studies by scaling the capacity curve as follows.

In the **Capacity** page there are two boxes for Scale Factors for Sensitivity Studies, one for a force scale factor and the second for a drift or displacement scale factor. To study the effects of changing strength and/or stiffness, specify values for these factors, press **Plot** to redraw the capacity curve, and proceed as before to draw the demand curve.

Since you have changed the capacity curve, you will have to define a new set of trial points.

## ***28.4 Performance Evaluation***

---

You will usually show limit points on the capacity curve. As a general rule, if all pertinent limit points are to the right of the displacement demand, the performance is satisfactory. In this case the usage ratios for all limit states are smaller than 1.0 at the displacement demand.

To study the limit state usage ratios in more detail, make a note of the load case and the drift demand, then go to the Usage Ratio Graphs task.

For the same load case, enter the drift demand as the maximum reference drift, and draw the usage ratio graph. From this graph you can see how the usage ratios vary for all limit states, up to the drift demand.

## 28.5 Consistency

---

### 28.5.1 General

As noted earlier in this Chapter, FEMA 356 Equation 3-16, which converts spectral acceleration to base shear, is approximate. This can affect the consistency of the results. This section considers this aspect in more detail.

### 28.5.2 Calculation of Spectral Displacement

When the capacity curve is plotted as spectral acceleration vs. spectral displacement, the spectral displacement is needed for each point on the curve.

If the displaced shape at a point is  $q\{\phi\}$ , where  $q$  is the (dimensionless) modal amplitude, the spectral displacement is given by

$$S_d = \frac{q}{\Gamma_\phi} \quad (3.14)$$

where  $\Gamma_\phi$  is the modal mass participation factor along the push-over direction, given by

$$\Gamma_\phi = \frac{\{\phi\}^T [M] \{d\}}{\{\phi\}^T [M] \{\phi\}} \quad (3.15)$$

FEMA 356 does not explicitly specify this equation. However, this equation is specified by ATC 40, and is implied by FEMA 440.

If the displaced shape is  $\{r\}$ , it is arguably more consistent to use

$$S_d = \frac{1}{\Gamma_r} \quad (3.16)$$

where

$$\Gamma_r = \frac{\{r\}^T [M] \{d\}}{\{r\}^T [M] \{r\}} \quad (3.17)$$

It is, however, not necessarily correct to use either of these equations. The reason is that the following relationship should exist between  $S_a$  and  $S_d$  :

$$\frac{S_a}{S_d} = \frac{4\pi^2}{T^2} \quad (3.18)$$

where  $T$  is period. For a point on a push-over curve,  $T$  is the secant period.

For a point on a push-over curve, if  $S_a$  is calculated using Equation 3.1,  $S_d$  is calculated using Equation 3.14, and  $T$  is calculated using Equation 3.9 (based on stiffness ratios), then Equation 3.18 is generally not satisfied. The main reason is that  $S_a$  and  $S_d$  will be consistent only if the push-over load at each point on the push-over curve is proportional to  $[M]\{r\}$ , where  $\{r\}$  is the deflected shape at the point. In general this condition is not satisfied. A secondary reason is that  $T$  may not be given accurately by Equation 3.9 or 3.10.

As a consequence, if you do a performance evaluation using the push-over curve in terms of base shear and reference drift, then repeat the evaluation in terms of  $S_a$  and  $S_d$ , you may not get the same result. For example, if the displacement demand for one form of the capacity curve is exactly at a limit point, the displacement demand for the other form of the curve generally will not be at that limit point. The difference tends to be larger when the push-over load pattern differs substantially from the deflected shape (e.g. for a “uniform” load pattern).

This is the main reason for using “adaptable” push-over analysis, where the push-over load pattern is adjusted at each step to match the deflected shape. Adaptable push-over analysis is currently not available in PERFORM.

### 28.5.3 Suggested Methods

The theoretically consistent approach is to use Equation 3.6 for  $S_a$ , Equation 3.16 for  $S_d$ , and the Rayleigh Quotient for  $T$ . In this case the performance assessment does not depend on the form of the capacity curve (provided that Equation 3.9 or 3.10 is accurate).

However, this is not the approach specified in FEMA 356, FEMA 440 or ATC 40. For practical analysis, the following methods are suggested.

- (1) For FEMA 356 or FEMA 440, for both the Coefficient and Linearization methods, plot the capacity curve in terms of base shear or base shear coefficient. If there is a mode shape that closely matches the push-over direction, convert spectral acceleration to base shear using this mode shape and FEMA 356 Equation 3-16. If there is no such mode shape use the current deflected shape and FEMA 356 Equation 3-16. This is not necessarily a consistent method, but it follows FEMA 356.
- (2) For the ATC 40 Capacity Spectrum Method, plot the capacity curve in terms of  $S_a$  and  $S_d$ . If there is a mode shape that closely matches the push-over direction, calculate  $S_a$  and  $S_d$  using this mode shape. If there is no such mode shape use the current deflected shape. This is not necessarily a consistent method, but it follows ATC 40.
- (3) To be most consistent, for any push-over method, convert spectral acceleration to base shear (or vice versa) using the current deflected shape and the alternative (consistent) method based on Equation 3.6. Also choose the Rayleigh Quotient method for calculating periods.

### 28.5.4 A Reason for Using $S_a$ vs. $S_d$ Option

The spectral acceleration vs. spectral displacement form of the push-over curve is sometimes used because it is consistent with the “Acceleration-Displacement” form of the response spectrum (ADRS). In this form of the response spectrum, both accelerations and displacements are shown, and lines of constant period are radial. In the spectral acceleration vs. period form of the spectrum, displacements are not shown and lines of constant period are vertical.

The spectral displacement is not particularly useful as a displacement measure, and the response spectrum is no simpler if lines of constant period are radial, so there is no inherent advantage in using the ADRS. However, there can be an advantage to using the spectral acceleration vs. spectral displacement form of the push-over curve. The reason is that the spectral displacement is an overall effective displacement of the structure, not just the displacement at one point (e.g., the roof). Hence, push-over plots in terms of spectral acceleration and spectral displacement tend to be better behaved.

As noted in an earlier section, one case where this can be important is where a structure has components that lose strength. In this case, the push-over curve in terms of base shear and reference drift (usually based on the roof displacement) can be badly behaved. A spectral acceleration vs. spectral displacement tends to give a smoother curve.

## ***28.6 Energy Dissipation and Damping***

---

### **28.6.1 General**

When a structure yields, energy is dissipated through inelastic hysteresis. This energy dissipation can affect the results of dynamic analysis, and hence should also be considered in static push-over analysis.

In dynamic analysis the hysteresis loops, and hence the energy dissipation, are considered directly. In static push-over analysis there is no cyclic loading, so energy dissipation must be considered indirectly. This section considers the procedures used by the different push-over methods.

### **28.6.2 Stiffness and Degradation and Energy Ratio**

The dissipated energy for an inelastic deformation cycle is related to the area under the hysteresis loop. A measure of the amount of stiffness degradation is the area of the degraded loop divided by the area of the non-degraded loop. This can be termed the “energy ratio”. The larger the amount of stiffness degradation, the smaller the energy ratio.

Figure 28.11 shows three hysteresis loops for a simple structure. If there is no stiffness degradation, the loop might be as shown in Figure

28.11(a). In most cases, however, there will be stiffness degradation, and the loop might be as shown in Figure 28.11(b) or (c).

For the loop in Figure 28.11(a) the energy ratio is 1.0. For the loop in Figure 28.11(b) the energy ratio is 0.5 for all ductility ratios if  $K_h = 0$ . If  $K_h > 0$  the ratio increases somewhat as the ductility ratio increases. The loop in Figure 28.11(c) also has an energy ratio of about 0.5. There are many possible loops that give the same energy ratio. Other ratios can be obtained by changing the details of the loop.

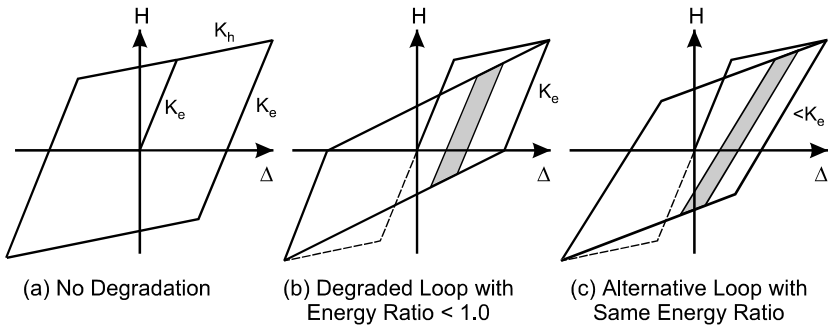


Figure 28.11 Hysteresis Loops

The energy ratio applies for full amplitude deformation cycles. The shaded bands in Figure 28.11 show loops for small amplitude cycles. If the energy ratio for a small amplitude cycle is the ratio of the degraded area to the non-degraded area for the same deformation range, this ratio is not necessarily the same as for a full amplitude cycle. Also, it is not the same in Figures 28.11(b) and (c). This illustrates the inherent difficulty in accounting for stiffness and energy degradation.

### 28.6.3 Equivalent Damping Ratio

In a linear structure with viscous damping, energy is dissipated by the viscous damping. If the structure is vibrating with a constant amplitude and period, it can be shown that the damping ratio, as a proportion of critical damping, is given by

$$\text{Damping Ratio} = \frac{1}{4\pi} \left( \frac{\text{Dissipated Energy per Cycle}}{\text{Maximum Strain Energy}} \right) \quad (27.1)$$



This equation is used in the Capacity Spectrum Method to convert inelastic dissipated energy to an equivalent damping ratio. The equation is not used in the Coefficient Method or the Linearization Method.

For the ATC 40 Capacity Spectrum Method (Figure 28.4), the damping ratio is calculated at the trial displacement. The dissipated energy is the area of the hysteresis loop, and the maximum strain energy is the area under the secant stiffness line, or  $0.5H\Delta$ . For the Modified Capacity Spectrum Method (Figure 28.5), the damping ratio is calculated at the effective displacement. In both cases, the area of the hysteresis loop, and hence the damping ratio, depends on the energy ratio. The procedures for specifying the energy ratio for the Capacity Spectrum Method are described in the next section.

#### 28.6.4 Degradation in Push-Over Methods

Energy degradation is considered in all of the push-over methods, but they use different procedures, as follows.

- (1) In the FEMA 356 Coefficient Method, the coefficient  $C_2$  depends on whether the structure degrades. For a degrading structure,  $C_2$  also depends on the performance level (Immediate Occupancy, Life Safety or Collapse Prevention), increasing from the IO to the CP level. In effect, this means that the energy ratio decreases as the ductility ratio increases, but the relationship is not explicitly specified.
- (2) The FEMA 440 Coefficient Method is calibrated for certain hysteresis loops, two of which are those in Figures 28.11(a) and (b). The equations for  $C_2$  are different in the non-degrading and degrading cases.
- (3) The FEMA 440 Linearization Method is calibrated for certain hysteresis loops, two of which are those in Figures 28.11(a) and (b), with different calibration equations for the non-degrading and degrading cases.
- (4) In the ATC 40 Capacity Spectrum Method the damping ratio is calculated at the trial point, using Equation 3.19. The energy ratio depends on the non-degraded energy and the structure behavior type (Type A, B or C). The energy ratios are specified by ATC 40. For each behavior type the ratio decreases as the non-degraded energy increases. The energy ratios are largest for Type A, and

decrease for Types B and C. PERFORM-3D adds a User type, where the energy ratios are specified by the user.

- (5) In the Modified Capacity Spectrum Method the damping ratio is calculated at the effective displacement, using Equation 3.19. There are three options for the energy ratio. One option is to use the ATC 40 ratios. The second option is to specify the energy ratio as a function of the ductility ratio. This is similar to the ATC 40 procedure, since the non-degraded energy increases as the ductility ratio increases. The third option is to estimate the degraded energy from the properties of the structural components and their deformations, and to calculate the energy ratio directly. This is the “PERFORM method”. It is described in the next section.

### 28.6.5 PERFORM Method for Energy Ratio

The “PERFORM method” for calculating energy ratios is more direct than other methods. The steps in the method are as follows.

- (1) The non-degraded energy is estimated for each inelastic component in the analysis model, based on its current deformation.
- (2) Energy degradation is accounted for using the PERFORM “Energy Factors” (for an inelastic component, see the **Component Properties** task and the **Cyclic Degradation** page). The energy factor is calculated for each inelastic component based on its current deformation. The non-degraded energy is multiplied by this factor.
- (3) The non-degraded and degraded energies are summed for all inelastic components in the structure. The energy ratio is the ratio of these two energies.

### 28.6.6 Extra Damping from Fluid Dampers

If a structure has elements that use fluid damper components, these elements have zero stiffness for push-over analysis. Hence, they develop no resisting forces and have no effect on the capacity curve. However, since fluid dampers increase the amount of damping, they can affect the demand curve, and hence the displacement demand. If you wish, you can account for this additional damping. The method is very approximate, but it may be useful for preliminary design.

To use the method you must do two main things, as follows.

- (1) Specify “static pushover” properties when you define the properties for fluid damper components (in the **Component Properties** task). The default properties are zero.
- (2) Use the Capacity Spectrum Method. This is the only method that explicitly considers the amount of damping. The other methods consider damping only implicitly, and can not account for additional damping due to fluid dampers.

In the **Component Properties** task for Fluid Damper components, choose the **Static Pushover** page. On this page, specify a viscous coefficient,  $C$ , and a damping exponent,  $n$ . For energy dissipation calculations in a static push-over analyses the force,  $F$ , in the component is calculated as

$$F = C\dot{\delta}^n \quad (3.20)$$

where  $\dot{\delta}$  is the deformation rate. The deformation rate is calculated as follows.

During a push-over analysis, PERFORM-3D calculates the axial deformation,  $\delta$ , of each viscous bar element, at each point on the capacity curve. All of this deformation is assumed to be concentrated in the fluid damper component. That is, the elastic bar component that acts in series with the damper is assumed to be very stiff. PERFORM-3D then assumes that the structure is vibrating sinusoidally with a constant amplitude, and that the period of vibration is the current secant period. Hence, the deformation rate also varies sinusoidally, and the force variation in the damper can be calculated (it is sinusoidal only if  $n = 1$ ). Hence the dissipated energy for one cycle of vibration can be found. This energy is summed for all viscous bar elements, and converted to a damping ratio, using Equation 3.19.

When the capacity curve is plotted for a structure that has viscous bar elements, the damping ratio from the fluid dampers is shown on the horizontal axis.

When the demand curve is calculated, the additional damping is considered. The effect of this damping on the displacement demand can be seen by specifying a value for the “effectiveness factor for fluid

dampers” in the **Details** page. To ignore the additional damping specify 0 for this factor. To use the full damping specify 1. Since the method is very approximate, experience may show that a factor between 0 and 1 gives a more accurate estimate.

### 28.6.7 Strength Loss and Degradation

Strength loss can be caused by brittle behavior, by progressive strength degradation under cyclic load, and/or by P- $\Delta$  effects. For push-over analysis it is usual to account for brittle strength loss and progressive strength degradation in the “backbone” curves used for inelastic components. P- $\Delta$  effects are considered rationally, using P- $\Delta$  theory. This theory considers equilibrium of the structure in its deformed shape whereas linear theory considers equilibrium in the undeformed shape. True large displacements theory goes one step further, and considers the nonlinear relationships between node displacements and element deformations. It is rarely, if ever, necessary to consider true large displacement effects for earthquake analysis of building structures.

Strength loss can cause a negative value for the stiffness  $K_h$  in the capacity curve. It can cause unpredictable behavior in dynamic analysis, with some analyses showing satisfactory behavior and others, for similar earthquakes, showing collapse.

The FEMA 356 Coefficient Method considers strength loss in the coefficient C3. The FEMA 440 Coefficient Method uses  $C3 = 1$ , and imposes a limit on the amount of strength loss. The FEMA 440 Linearization Method considers negative values of  $K_h/K_e$  in its calibration against dynamic analysis. The Capacity Spectrum Method does not explicitly consider strength loss.

In general, the effects of strength loss, from whatever cause, are complex, and it is difficult to get reliable results, from static push-over analysis or dynamic analysis.

## ***28.7 Theory for Relationship Between $S_a$ and $H$***

---

### 28.7.1 General

A key step in the computation is as follows: given the spectral acceleration demand,  $S_{a,demand}$ , calculate the corresponding base shear

demand,  $H_{demand}$ . This is Step 3 in Figure 28.2, and Step 4 in Figures 28.3, 28.4 and 28.5.

For a single DOF structure,  $H_{demand}$  is simply

$$H_{demand} = \frac{W}{g} S_{a,demand} \quad (3.21)$$

Where  $W$  is the structure weight. For a multi-DOF structure, FEMA 356 specifies the equation

$$H_{demand} = C_m \frac{W}{g} S_{a,demand} \quad (3.22)$$

where  $C_m$  is the effective mass factor for the “first” elastic mode. For the general 3D case this means the mode that has the largest effective mass factor along the push-over direction. As noted earlier in this Chapter, this equation is accurate only for certain special cases. To show this it is necessary to review some theory.

One of the important aspects of the dynamic response to an earthquake is that there are no actual external forces on the structure. Instead, the earthquake excites a dynamic response, and the forces on the structure are the inertia forces associated with that response.

The same type of behavior is implied when push-over analysis is used. In this case the dynamic response is approximated by applying static external forces. There is implied vibration of the structure, and hence implied inertia forces. These forces are not necessarily the same as the applied external forces. This is an important aspect of the theory.

### 28.7.2 Special Case With Modal Load and Displaced Shape

Consider the following special case.

- (1) The elastic mode shape of the structure is  $\{\phi\}$ .
- (2) The external load  $\{R\}$  is proportional to  $[M]\{\phi\}$ , where  $[M]$  is the mass matrix. In PERFORM this is a diagonal matrix.
- (3) The displaced shape,  $\{r\}$ , is proportional to  $\{\phi\}$ . This will be the case if the structure remains elastic. In general, as the structure

yields the displaced shape will change. For this section assume that the displaced shape does not change.

If the spectral acceleration demand is  $S_{a,demand}$ , its effect is similar to that of a uniform, horizontal gravitational field of this magnitude. In this gravitational field the forces on the structure are

$$\{R_g\} = S_{a,demand} [M] \{d\} \quad (3.23)$$

where  $\{d\}$  = matrix of direction cosines for the push-over direction. If the push-over direction is along the H1 axis, matrix  $\{d\}$  has a unit value for each H1 mass in  $[M]$ , and a zero value for all other masses. Similarly, if the push-over direction is along the H2 axis, matrix  $\{d\}$  has a unit value for each H2 mass in  $[M]$ , and a zero value for all other masses.

The gravity forces are not the dynamic inertia forces on the masses. In effect, the structure is deformed in its mode shape,  $\{\phi\}$ , and is vibrating with some period,  $T$ , with a modal amplitude  $\lambda$  (i.e., the displaced shape at peak amplitude is  $\lambda\{\phi\}$ ). The peak accelerations are

$$\{\ddot{r}\} = \frac{4\pi^2}{T^2} \lambda \{\phi\} \quad (3.23)$$

and hence the peak inertia forces are

$$\{R_I\} = [M] \{\ddot{r}\} = \frac{4\pi^2}{T^2} \lambda [M] \{\phi\} \quad (3.25)$$

To obtain the modal amplitude, equate the work done by the gravity and inertia forces acting through a displacement  $\{\phi\}$ :

$$\{\phi\}^T \{R_g\} = \{\phi\}^T \{R_I\} \quad (3.26)$$

from which the amplitude is given by

$$\lambda = \frac{T^2}{4\pi^2} S_{a,demand} \frac{\{\phi\}^T [M] \{d\}}{\{\phi\}^T [M] \{\phi\}} \quad (3.27)$$

Substitute for  $\lambda$  in Equation (3.25) to get

$$\{R_I\} = S_{a,demand} \frac{\{\phi\}^T [M] \{d\}}{\{\phi\}^T [M] \{\phi\}} [M] \{\phi\} \quad (3.28)$$

Hence the inertia force base shear in the push-over direction is

$$H_{I\phi} = \{R_I\}^T \{d\} = S_{a,demand} \frac{(\{\phi\}^T [M] \{d\})^2}{\{\phi\}^T [M] \{\phi\}} \quad (3.29)$$

or

$$H_I = S_{a,demand} \cdot M^* \quad (3.30)$$

where

$$M^* = \frac{(\{\phi\}^T [M] \{d\})^2}{\{\phi\}^T [M] \{\phi\}} \quad (3.31)$$

is the effective mass. For push-over analysis it is usual to use the dimensionless effective mass factor,  $C_{m\phi}$ , equal to the effective mass divided by the total mass. That is,

$$C_{m\phi} = \frac{M^*}{(W/g)} = \frac{(\{\phi\}^T [M] \{d\})^2}{\{\phi\}^T [M] \{\phi\}} \cdot \frac{1}{(W/g)} \quad (3.32)$$

where  $W$  is the structure weight. Hence the inertia force base shear demand is

$$H_I = S_{a,demand} \cdot C_{m\phi} \frac{W}{g} \quad (3.33)$$

In this special case, the applied push-over loads are proportional to  $[M] \{\phi\}$ , and hence have the same distribution as the inertia forces.

Hence, the inertia force base shear is equal to the actual base shear in the push-over analysis. That is,

$$H_{demand} = H_I = S_{a,demand} \cdot C_{m\phi} \frac{W}{g} \quad (3.34)$$

In this case, FEMA 356 Equation 3-16 gives the correct base shear demand.

### 28.7.3 General Case Where Load Matches Displaced Shape

In the preceding case, the deflected shape is the elastic mode shape, and the load pattern corresponds to this shape. It is also possible for the load pattern to correspond to the deflected shape when this shape is not a mode shape. This is the goal of “adaptive” push-over analysis, where the load pattern at each step of the analysis is adjusted to match the current displaced shape. That is, if the current displaced shape is  $\{r\}$ , the load is proportional to  $[M]\{r\}$ .

In this case, the equations of the preceding section still apply, but with the mode shape,  $\{\phi\}$ , replaced by the deflected shape,  $\{r\}$ . The final result is

$$H_{demand} = S_{a,demand} \cdot C_{mr} \frac{W}{g} \quad (3.35)$$

where

$$C_{mr} = \frac{(\{r\}^T [M] \{d\})^2}{\{r\}^T [M] \{r\}} \cdot \frac{1}{(W/g)} \quad (3.36)$$

Again, FEMA 356 Equation 3-16 gives the correct base shear demand.

### 28.7.4 Case Where Load Does Not Match Displaced Shape

In most push-over analyses, the displaced shape,  $\{r\}$ , is not the elastic mode shape, and the load pattern is not proportional to  $[M]\{r\}$ . Hence, Equation 3.35 does not give the correct base shear demand.



Consider the case with a trial point that has a displaced shape  $\{r\}$  and a lateral load  $\{R\}$ . The equations in this case are as follows.

As before, a spectral acceleration demand of  $S_{a,demand}$  is equivalent to a uniform gravitational field of this magnitude. In this gravitational field the forces on the structure are

$$\{R_g\} = S_{a,demand} [M] \{d\} \quad (3.37)$$

Again, these are not the dynamic inertia forces on the structure. In this case the structure is vibrating with an effective mode shape  $\{r\}$ , a period  $T$  and a modal amplitude  $\lambda$  (i.e., the displaced shape at peak amplitude is  $\lambda\{r\}$ ). Hence the maximum inertia forces are

$$\{R_I\} = \frac{4\pi^2}{T^2} \lambda [M] \{r\} \quad (3.38)$$

To obtain the modal amplitude, equate the work done by these forces acting through the displacement  $\{r\}$ . The work equation is

$$\{r\}^T \{R_g\} = \{r\}^T \{R_I\} \quad (3.39)$$

from which the amplitude,  $\lambda$ , is given by

$$\lambda = \frac{T^2}{4\pi^2} S_{a,demand} \frac{\{r\}^T [M] \{d\}}{\{r\}^T [M] \{r\}} \quad (3.40)$$

Substitute for  $\lambda$  in Equation 3.38 to get

$$\{R_I\} = S_{a,demand} \frac{\{r\}^T [M] \{d\}}{\{r\}^T [M] \{r\}} [M] \{r\} \quad (3.41)$$

Hence the inertia force base shear is

$$H_{I,demand} = \{R_I\}^T \{d\} = S_{a,demand} \frac{(\{r\}^T [M] \{d\})^2}{\{r\}^T [M] \{r\}} \quad (3.42)$$

or

$$H_{I,demand} = S_{a,demand} \cdot C_{mr} \cdot \frac{W}{g} \quad (3.43)$$

where

$$C_{mr} = \frac{(\{r\}^T [M] \{d\})^2}{\{r\}^T [M] \{r\}} \cdot \frac{1}{(W/g)} \quad (3.44)$$

This is the inertia base shear demand, not the required base shear for the analysis, which is the base shear for the push-over loads  $\{R\}$ . To calculate the required base shear demand, consider the relationship between the push-over loads,  $\{R\}$ , and the inertia forces,  $\{R_I\}$ , at a trial point. These are the base shear *capacities* at the trial point.

The base shear capacity from the push-over loads is

$$H_{capacity} = \{R\}^T \{d\} \quad (3.45)$$

The inertia forces are

$$\{R_I\} = \frac{4\pi^2}{T^2} [M] \{r\} \quad (3.46)$$

and the base shear capacity from these inertia forces is

$$H_{I,capacity} = \{R_I\}^T \{d\} \quad (3.47)$$

The ratio between these two base shear capacities applies to force demands as well as capacities. Hence

$$H_{demand} = \frac{H_{capacity}}{H_{I,capacity}} H_{I,demand} \quad (3.48)$$

or

$$H_{demand} = \frac{\{R\}^T \{d\}}{4\pi^2 \{r\}^T [M] \{d\}} S_{a,demand} C_{mr} \frac{W}{g} \quad (3.49)$$

The effective period at the trial point is the secant period. From the Rayleigh quotient, this period is given by

$$T = 2\pi \sqrt{\frac{\{r\}^T [M] \{r\}}{\{r\}^T \{R\}}} \quad (3.50)$$

Substituting this into Equation 3.49 gives

$$H_{demand} = \left( \frac{\{R\}^T \{d\}}{\{R\}^T \{r\}} \right) \left( \frac{\{r\}^T [M] \{r\}}{\{r\}^T [M] \{d\}} \right) S_{a,demand} C_{mr} \frac{W}{g} \quad (3.51)$$

If  $\{R\}$  is proportional to  $[M]\{r\}$ , the terms in parentheses = 1.0, and Equation 3.51 reduces to the equation for the special case.

Equation 3.51 is rather complex. A simpler method is as follows.

From Equation 3.44

$$C_{mr} = \frac{(\{r\}^T [M] \{d\})^2}{\{r\}^T [M] \{r\}} \cdot \frac{1}{(W/g)} \quad (3.52)$$

Substitution in Equation 3.51 gives

$$H_{demand} = S_{a,demand} \frac{\{r\}^T [M] \{d\}}{\{r\}^T \{R\}} \{R\}^T \{d\} \quad (3.53)$$

Since the base shear capacity at the trial point is

$$H_{capacity} = \{R\}^T \{d\} \quad (3.54)$$

Equation 3.53 can be written as

$$H_{demand} = S_{a,demand} \frac{\{r\}^T [M] \{d\}}{\{r\}^T \{R\}} H_{capacity} \quad (3.55)$$

or

$$H_{demand} = \frac{S_{a,demand}}{S_{a,capacity}} H_{capacity} \quad (3.56)$$

where

$$S_{a,capacity} = \frac{\{r\}^T \{R\}}{\{r\}^T [M] \{d\}} \quad (3.57)$$

## 29 Target Displacement Plot

---

*The preceding chapter describes a number of methods for performance assessment using static push-over analysis, including the FEMA 356 Coefficient (Target Displacement) Method. You can use this method in either the General Push-Over plot task, or in this Target Displacement Plot task. The procedures are rather different, but the results should be the same.*

*This chapter summarizes the method and explains the procedure.*

### 29.1 Target Displacement Method

---

#### 29.1.1 Concept

The target displacement (or coefficient) method makes use of the capacity curve from a static push-over analysis, and uses an empirical formula to calculate a displacement demand (the target displacement). You can then use limit states to assess the performance at this displacement.

PERFORM-3D uses drift rather than displacement. The program calculates a target value for the reference drift, which is typically the roof displacement divided by the building height.

Figure 29.1 shows the ingredients of a target displacement plot. These are as follows.

- (1) A capacity curve. This is the relationship between lateral load and lateral drift from a static push-over analysis.
- (2) An estimated (guessed) target drift.
- (3) A bilinear approximation of the capacity curve at the estimated drift.
- (4) A calculated target drift based on this bilinear approximation.

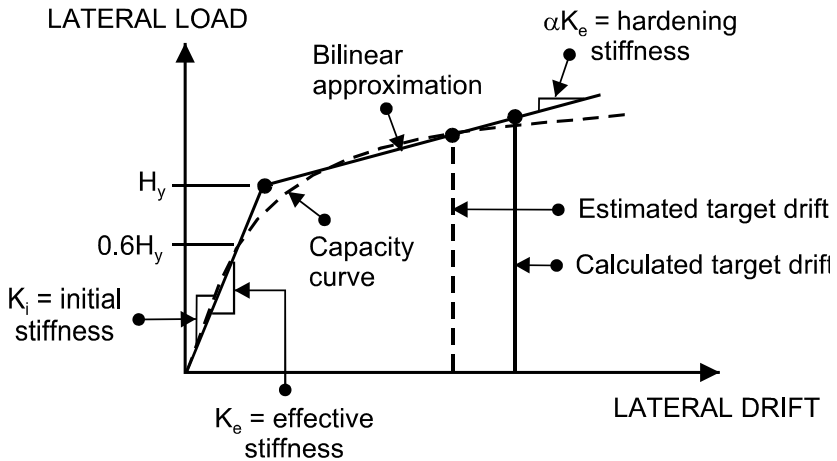


Figure 29.1 Ingredients of a Target Displacement Plot

In Figure 29.1 the calculated target drift is larger than the estimated drift. A new estimate must be made, until the estimated and calculated drifts are close to each other. This defines the target drift.

The key parts of the method are as follows.

- (1) The bilinear approximation. The FEMA 356 approximation is shown in Figure 29.1. The areas under the capacity curve and the bilinear approximation are equal at the estimated target drift.
- (2) The calculated target drift. This is as follows.

### 29.1.2 Target Drift

The formula for the calculated target drift is as follows.

$$\delta = C_0 C_1 C_2 C_3 S_\delta$$

where:

$\delta$  = calculated target drift;

$S_\delta$  = spectral displacement (drift) demand for the elastic structure (see below);

$C_0$  = factor to convert spectral drift to reference drift;

$C_1$  = factor to convert elastic drift to inelastic drift;

$C_2$  = factor to account for framing type and performance level;  
and

$C_3$  = factor to account for P- $\Delta$  effects.

### 29.1.3 Spectral Drift

The spectral drift for the elastic structure is calculated by the procedure specified in FEMA 356. The calculation requires the capacity curve, a response spectrum, a mode shape and a mode period. You must specify the response spectrum. The mode shape is usually the first elastic mode shape (PERFORM-3D gives you other options). The mode period,  $T_e$ , is based on the effective stiffness,  $K_e$ .

See FEMA 356 for the formulas.

### 29.1.4 Factor $C_0$

This factor converts spectral drift to reference (roof) drift. The spectral drift is essentially an average drift value, and the roof drift is numerically larger. For a simple lollipop the values are the same. For a tall building the roof drift is about 1.5 times the spectral drift. The actual value depends on the mode shape.

You have the option of using the mode shape or the deflected shape at the estimated target displacement. You can also specify a user value.

### 29.1.5 Factor $C_1$

This factor converts the drift for an elastic structure with stiffness  $K_e$  to the drift for an inelastic structure with bilinear behavior as shown in Figure 29.1. This factor is largely empirical. For details see FEMA 356.

You have the option of the FEMA 356 formula or a user value.

### 29.1.6 Factor $C_2$

This factor accounts for the quality of the structure and the desired performance level.

You can choose from two structure types, namely Type 1 (worse, with more stiffness degradation) and Type 2 (better, with less stiffness degradation). You can choose from three performance levels, namely Immediate Occupancy, Life Safety and Collapse Prevention.

### 29.1.7 Factor $C_3$

If there are substantial P- $\Delta$  effects the tangent stiffness of the capacity curve can become negative before the target drift is reached. For static load this means that the structure is unstable and would collapse. For dynamic load this is not necessarily the case, but the possibility of serious damage is increased.

This factor accounts for this effect. It depends on the hardening stiffness,  $\alpha K_e$ , in the bilinear approximation. For details see FEMA 356. You can also specify a user value.

There can also be negative stiffness because of strength loss, independent of any P- $\Delta$  effects.

## ***29.2 PERFORM Implementation***

---

To create a target displacement plot you must have results for at least one push-over load case and you must have calculated one or more mode shapes for the structure. To start a plot press the toolbar buttons for the **Analysis** phase and the **Target Displacement** task. In the Target Displacement form choose the analysis series from the Series list and the load case from the Case list.

There are three main steps, each with substeps. First complete Step 1 as follows.

- (1) Define an elastic design spectrum. A diagram in the form shows the required data.



- (2) Choose the framing type.
- (3) Choose the performance level.
- (4) Choose the curve type (base shear vs. reference drift or base shear coefficient vs. reference drift).
- (5) Press the **Check** button to check the data.

If everything is OK the capacity curve will be plotted. Next complete Step 2 as follows.

- (1) Choose the limit points that you want to show on the plot. A plot with limit points is much more useful than one without.
- (2) If you wish, click on any limit point in the plot. Its color changes to blue and the name of the corresponding limit state is shown in the box. You can return to this step to do this at any time.
- (3) Press **OK** to go to Step 3.

Complete Step 3 as follows.

- (1) Estimate a target drift and draw a bilinear approximation. To do this follow the Step 3A instructions in the form. This may take some practice. Press **Clear** to start over.
- (2) Choose options for the C factors. If you choose the User option for any factor, input the value.
- (3) Press **Calculate** to calculate and plot the target drift.
- (4) Repeat from Step (1) until the calculated drift is close to your estimate.

The instructions for Step 3A are for the "click-and-drag" method for constructing a bilinear approximation (see *Chapter 28, General Push-Over Plot, Section 28.3.5*). For a FEMA 356 bilinear approximation you can also use the quicker "right-click" method.

You can judge the performance of the structure by comparing the target drift with the limit point drifts. To get usage ratios in more detail, use the Usage Ratio Plot task, and plot usage ratios at the target displacement.

To change the limit states, or to show the limit state for any limit point, return to Step 2.



## 30 Usage Ratio Graphs

*Usage ratio graphs are valuable tools for assessing whether a structure satisfies the performance criteria. This chapter explains the procedure.*

### 30.1 Usage Ratios

As the load factor increases in a gravity analysis, or as the drift increases in a push-over analysis, or as the time increases in a dynamic analysis, the usage ratios for the limit states will progressively increase. A usage ratio graph shows the variation of usage ratio with load factor, drift or time, depending on the analysis type.

Figure 30.1 shows the ingredients of a usage ratio graph, for a dynamic earthquake analysis.

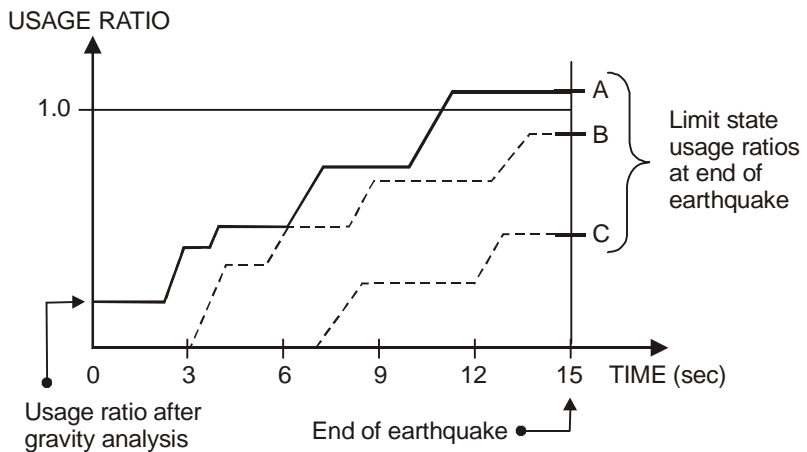


Figure 30.1 Ingredients of a Usage Ratio Graph

In this figure there are three limit states (there will usually be more). The upper line is the maximum usage ratio, considering all three limit

states. The usage ratio lines for the individual limit states are also shown (see below for how to do this). Up to a time of about 6 seconds, Limit State B governs, and after that Limit State A governs. Limit State C is never critical.

The maximum usage ratio at the end of the analysis is less than 1.0 for Limit States B and C, indicating that the structure satisfies the performance criteria for these limit states. However, it is larger than 1.0 for Limit State A, indicating that the structure does not satisfy the performance criteria for the complete limit state group.

### **30.2 Procedure**

---

To start a usage ratio plot press the toolbar buttons for the **Analysis** phase and the **Usage Ratio** task. In the Usage Ratio form choose the analysis series from the Series list and the load case from the Case list.

To draw the graph choose the limit states that are to be considered, then press the **Plot** button. The maximum usage ratio line is drawn, and at the end of the plot the usage ratios for all limit states in the group are shown as horizontal red lines. Click on any of these lines to draw the usage ratio line for the corresponding limit state, and to show the name of the limit state in the box.

You can save the usage ratios at the end of the graph by pressing the save to file button at the right end of the toolbar.

## **31 Load Case Combinations and Envelopes.**

---

*The Usage Ratio Graphs and Deflected Shapes tasks consider usage ratios for only one load case at a time. The Usage Ratio Envelopes task allows load case combinations to be specified, and for usage ratios to be calculated and displayed for these combinations.*

*This Chapter describes the procedure. There are a number of options for setting up the load case combinations and for combining the usage ratios. It is important to understand the combination methods that are used, and to choose the appropriate options for your application.*

### **31.1 Overview**

---

#### **31.1.1 Main Steps**

The main steps in the process are as follows.

- (1) Run a number of dynamic earthquake or static push-over analyses. For dynamic earthquake analyses it will be usual to consider several earthquakes and possibly to apply each earthquake in two horizontal directions. For 7 earthquakes and 2 direction this is 14 analyses. For push-over analysis it will be usual to consider two or more lateral load distributions and to push the structure in two or more horizontal directions. For 2 lateral load distributions and 2 directions this is 4 analyses. For each analysis the usage ratios will be calculated for a number of limit states. For dynamic analyses the usage ratios are at the end of the analysis. For push-over analyses they are at the estimated displacement demand (the target displacement or performance point) for each analysis.
- (2) Specify one or more Load Case Combinations. A combination can consist of just a set of load cases, for example the 7 earthquake cases in one horizontal direction. In this case there would be 2 combinations, each with 7 load cases. Alternatively, a combination can consist of a number of groups of load cases, for example 7 groups of 2 load cases each, with each group containing the

analyses for 2 horizontal directions. In this case there would be a single combinations, with 14 load cases in 7 groups of 2.

- (3) If the combination includes load case groups, choose a combination method to be applied within each group. The options are Maximum, SRSS (square root of sum of squares) and Sum.
- (4) Choose a combination method to be applied across groups (or across load cases if there are no groups). The options are "Max-Max", "Mean-Max", "Max-Mean" and "Max-(Mean+NSigma)", as explained later.

The end result is a combined usage ratio for each limit state. These usage ratios can be displayed in a diagram that is somewhat similar to that used in the Usage Ratio Graphs task, except that final usage ratios are shown, not graphs of usage ratio vs. time or load. Usage ratios for individual elements can also be displayed in a color-coded drawing similar to that in the Deflected Shapes task.

### 31.1.2 Load Case Combinations

A load case combination can consist of up to 50 load cases, divided into groups if desired. If there are groups, they must all have the same number of load cases. A combination can contain both static and dynamic load cases, but this is unlikely.

As one example, consider a structure for which dynamic earthquake analyses have been carried out for 7 earthquakes applied in each of 2 directions. For each analysis, usage ratios are calculated at the end of the analysis. FEMA 356 allows the usage ratios to be averaged over the 7 earthquakes. The load case combination in this case could consist of 7 groups of 2 load cases. Within each group (2 earthquake directions) the usage ratios might be combined using the Maximum method. Across groups (7 earthquakes) the usage ratios might be combined using the Max-Mean method, as explained later.

As a second example, consider a structure for which static push-over analyses have been carried out for lateral load distributions corresponding to 3 mode shapes, in each of 2 directions. For each analysis, usage ratios are calculated at the estimated drift demand (target drift). Usage ratios are to be combined using the Chopra-Goel multi-mode procedure. The load case combination in this case could consist of 2 groups of 3 load cases. Within each group (3 modal load

cases) the usage ratios would be combined using the SRSS method. Across groups (2 directions) the usage ratios would be combined using the Max-Max method.

### 31.1.3 Combination Methods

Within any load case group, the combination options for usage ratios are Maximum, SRSS and Sum, which are self-explanatory. Usage ratios are always positive or zero, so signs are not a concern.

Across load case groups the combination options are Max-Max, Mean-Max, Max-Mean, and Max-(Mean+NSigma). To illustrate the difference, consider a structure with 2 elements (Elements 1 and 2) and 3 load cases (Load Cases A, B and C). Consider one of the limit states, with usage ratios for the elements as shown in the following table.

	Load Case A	Load Case B	Load Case C
Element 1	Usage ratio =0.91	0.77	0.87
Element 2	0.67	0.83	0.75

The limit state usage ratio for the structure is calculated as follows, for each of the four combination methods.

(1) Max-Max.

For each element, usage ratio = Max over all load cases.

For structure, usage ratio = Max over all elements.

Element 1 usage ratio = Max (0.91, 0.77, 0.87) = 0.91

Element 2 usage ratio = Max (0.67, 0.83, 0.75) = 0.83

Structure usage ratio = Max (0.91, 0.83) = 0.91

(2) Mean-Max.

For each element, usage ratio = Max over all load cases.

For structure, usage ratio = Mean over all elements.

Element 1 usage ratio = Max (0.91, 0.77, 0.87) = 0.91

Element 2 usage ratio = Max (0.67, 0.83, 0.75) = 0.83

Structure usage ratio = Mean (0.91, 0.83) = 0.87

(3) Max-Mean

For each element, usage ratio = Mean over all load cases.

For structure, usage ratio = Max over all elements.

Element 1 usage ratio = Mean (0.91, 0.77, 0.87) = 0.85

Element 2 usage ratio = Mean (0.67, 0.83, 0.75) = 0.75

Structure usage ratio = Max (0.85, 0.75) = 0.85

(4) Max-(Mean+NSigma)

For each element, usage ratio = Mean + N times the standard deviation (sigma) over all load cases.

For structure, usage ratio = Max over all elements.

Element 1 usage ratio : Mean = 0.85, Sigma = 0.059

Element 2 usage ratio : Mean = 0.75, Sigma = 0.065

For N = 1, structure usage ratio = Max (0.909, 0.835 ) = 0.909

The Max-Max method is the most conservative. This method is required by FEMA 356 if fewer than 7 earthquakes are considered. The Max-Mean method is the least conservative. This method is allowed by FEMA 356 if 7 or more earthquakes are considered. The Mean-Max method is between the Max-Max and Max-Mean methods. It is not included in FEMA 356. The Max-(Mean+NSigma) method is also not included in FEMA 356, but it may be appropriate for strength-based limit states when capacity design is used. In this case, strengths based on the Max-Mean method may not be sufficiently large to ensure essentially elastic behavior.

The procedure is similar for limit states based on drift, deflection or structure section results, rather than element results. For example, if a drift limit state has several drifts, in the above table replace Element 1 by Drift 1, etc.

## 31.2 Procedure

---

### 31.2.1 Load Combinations

Press the toolbar buttons for the **Analysis** phase and the **Combinations and Envelopes** task. The required analyses must have been run. They do not all have to be in the same Analysis Series.

The steps for setting up or editing a Load Case Combination are as follows.

- (1) To start a new combination, press **New**. Specify the combination name, specify the number of load cases per group (1 if there are no groups) and choose the combination method within groups.
- (2) Set up a list of load cases in the combination, by choosing the Analysis Series and Load Case Name from the lists on the toolbar, and pressing **Add**. Each load case group is assigned a number as



the load cases are added. You can delete load cases, a group at a time, by clicking in the table to highlight the group and pressing **Delete**.

- (3) When the list is complete, press **Save**. You can edit an existing combination at any time, using **Delete** and **Add**.

### 31.2.2 Usage Ratios for Structure

The steps for calculating and displaying usage ratios for the complete structure are as follows.

- (1) Go to the **Structure** page.
- (2) Choose the Combination Method.
- (3) Choose the limit states that are to be considered.
- (4) Press **Plot** to plot the results.

The results are plotted as shown in Figure 31.1.

In this case there are 7 load cases, with only one case per group (i.e., no combination within groups). For each load case there is a blue vertical line, with a short horizontal line for each limit state, showing the limit state usage ratios for the load case. On the left side there is a red vertical line, also with a short horizontal line for each limit state, showing the limit state usage ratios for the load combination. If you click on any of the limit states on the red line, a horizontal red line is drawn showing the usage ratio for the load combination (as shown in the figure), a blue line is drawn connecting the usage ratios for the separate load cases (as shown in the figure), and the name of the limit state is shown in a box at the bottom of the form (not shown in the figure).

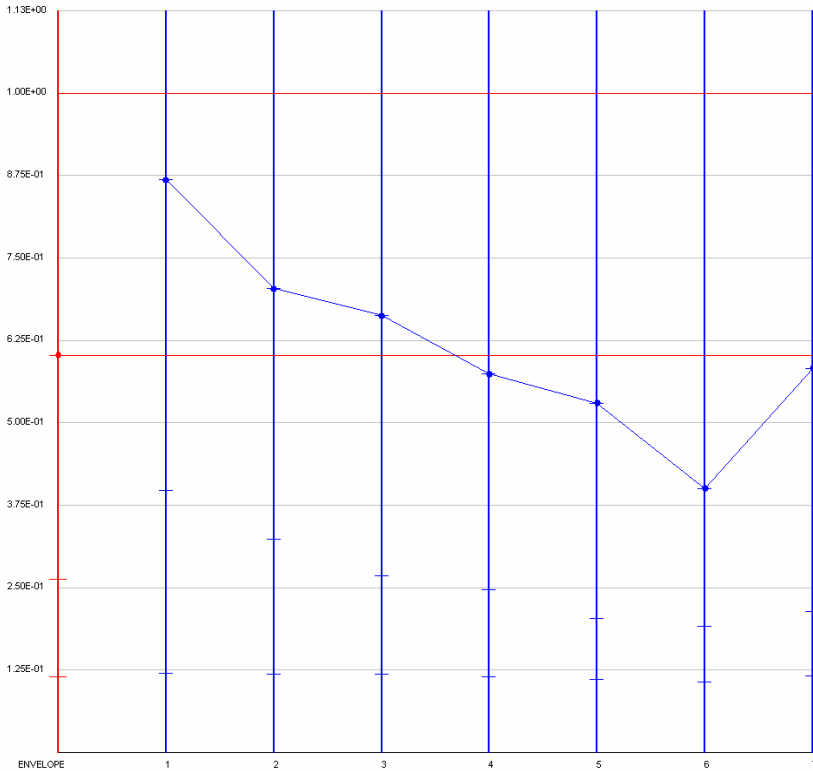


Figure 31.1 Usage Ratios for Structure

### 31.2.3 Usage Ratios for Elements

The steps for showing element usage ratios in a color-coded drawing are as follows.

- (1) Go to the **Element Colors** page.
- (2) Choose the Combination Method. This is the combination type across load cases (or load case groups). The options are Max, Mean and Mean+NSigma. Usage ratios are not combined across elements for this plot.
- (3) Choose the limit states that are to be considered.
- (4) Choose a color group for the element colors.
- (5) Press **Plot** to draw the structure.

## Chapter 31. Load Case Combinations and Envelopes

The drawing is similar to that for the Deflected Shape task, but deflections are not shown. To zoom in, use the two zoom buttons directly below the Plot button.



## 32 Importing/Exporting Structure Data

---

*If you have analyzed a structure using a linear analysis program, you may be able to save time by importing the linear analysis model into PERFORM-3D. You may also wish to export data from a PERFORM-3D model, for import into another PERFORM-3D model or a different computer program.*

*PERFORM-3D can import and export the data for nodes, masses, elements and nodal loads. This chapter describes the procedures.*

*PERFORM-3D can also export and import the data for inelastic, elastic, etc. components. Among other things, you can use this feature to set up libraries of commonly used components, and import them into different PERFORM-3D models. This chapter also describes this procedure.*

### 32.1 Importing Structure Data

---

#### 32.1.1 Text Files

PERFORM-3D can not import structure data by connecting directly to another program. You must export the data from the other program, and organize it into a number of comma-delimited text files. PERFORM can then read those files. You may be able to export the files directly from the other program, or you may need to export the data and modify its format using a spreadsheet program. You can import the following.

- (1) Elements, with or without their nodes.
- (2) Nodes only. It is usually better to import nodes with the elements.
- (3) Nodal masses.
- (4) Nodal loads.

A key point is that nodes must be identified by their H1, H2, V coordinates, not by node numbers. For example, if you are importing element data for 4-node wall elements, the text file must have 12 coordinate values for each element, 3 each for nodes I, J, K and L of the element.

### 32.1.2 File for Elements (with Optional Nodes)

In the PERFORM-3D model, the elements are organized in Element Groups. In a single import operation you can import elements into only one group. This does not mean that you must have a separate text file for each element group. If you wish you can put all of the elements in one file, then read them selectively, as described later. However, you can put the elements for each element group into a separate file if you wish. You can also read elements into a single element group, then divide them into separate groups within PERFORM.

For any element group you have the option of importing only the elements, or elements plus nodes. The difference is as follows.

- (1) If you choose the element-only option, an element is imported only if all of its nodes currently exist in the PERFORM-3D model. The tolerance for whether a node exists is one third of the minimum spacing between nodes for the PERFORM-3D model. This spacing is typically 6 inches or 15 cm, but you can set it to a different value for the import operation if you wish. If all of the element nodes exist with H1, H2 and V coordinates that are within this tolerance (typically  $6/3 = 2$  inches or  $15/3 = 5$  cm), the element is imported. If not, it is ignored. You will be warned if there are some elements that are not imported.
- (2) If you choose the element-plus-nodes option, an element is imported even if all of its nodes do not currently exist in the PERFORM-3D model. If a node does not already exist, a new node is created. The tolerance for whether a node exists is equal to the minimum spacing between nodes for the PERFORM-3D model (typically a tolerance of 6 inches or 15 cm).

The file format is as follows.

- (1) First N lines (as many as you wish) : descriptions of the file. Each line can be of any length, and must end with a line feed (carriage return). These lines are skipped when the data is imported.
- (2) Remaining lines : one line per element, as follows.

- (a) Single node element (e.g., a panel zone element). Each line must contain the H1, H2 and V coordinates of the node, separated by commas.
- (b) Two-node element (e.g., a beam element). Each line must contain the H1, H2 and V coordinates of the node at End I, followed by the H1, H2 and V coordinates of the node at End J.

For a frame type element (beam, column or brace/other) there can be two additional items if desired, describing the orientation of the local 2, 3 axes for the element (the 1 axis is along the element). The first item is the global reference axis for the element. If this item is 1, local Axis 2 of the element is basically parallel to the global H1 axis; if it is 2, local Axis 2 of the element is basically parallel to the global H2 axis; and if it is 3, local Axis 2 of the element is basically parallel to the global V axis. The second item is the twist angle, for cases where local Axis 2 is not along a global axis. This item is the twist angle to the inclined axis, in degrees, measured clockwise about the element Axis 1.

- (c) Four-node element (e.g., a shear wall walls). Each line must contain 12 items, namely the H1, H2 and V coordinates at the element nodes. You have two options for ordering the nodes. The first option is I, J, K, L, where I is usually bottom left, J is bottom right, K is top left, and L is top right. In this case the nodes are ordered zigzag fashion, not around the element. The second option is I, J, L, K, which is counterclockwise around the element.

The length unit for the coordinates must be the same as the global length unit for the PERFORM-3D structure.

You can import only the basic element geometry. You must specify the element properties in PERFORM-3D. You must also specify the local axis orientations of single-node and four-node elements in PERFORM-3D.

### 32.1.3 File for Nodes

Usually you will not need to import nodes directly, since nodes can also be created as you import elements. However, you can import nodes if you wish.

The file format is as follows.

- (1) First N lines (as many as you wish) : descriptions of the file. Each line can be of any length, and must end with a line feed (carriage return). These lines are skipped when the data is imported.
- (2) Remaining lines : one line per node. Each line must contain the H1, H2 and V coordinates of the node, separated by commas. The length unit for the coordinates must be the same as the global length unit for the PERFORM-3D structure.

If a node already exists the line is ignored. The tolerance for whether a node exists is equal to the minimum spacing between nodes for the PERFORM-3D model. This spacing is typically 6 inches or 15 cm, but you can set it to a different value for the import operation if you wish. You will be warned if there are some nodes that do not match.

#### 32.1.4 File for Nodal Masses

In order to import masses you must first create or import the nodes.

The file format is as follows.

- (1) First N lines (as many as you wish) : descriptions of the file. Each line can be of any length, and must end with a line feed (carriage return). These lines are skipped when the data is imported.
- (2) Remaining lines : one line per mass. Each line must contain the H1, H2 and V coordinates of the corresponding node, followed by 6 mass values, namely the H1, H2 and V translational masses and the H1, H2 and V rotational masses at the node, all separated by commas. The length unit for the coordinates must be the same as the global length unit for the PERFORM-3D structure. The masses must be in mass units ( $FT^2/L$ ). The force and length units must be the same as the global force and length units for the PERFORM-3D structure. T is seconds.

If the file contains a node that does not exist (i.e., if the coordinates do not match the coordinates of an existing node), the masses are not imported. The tolerance for whether a node exists is equal to the minimum spacing between nodes for the PERFORM-3D model. This



spacing is typically 6 inches or 15 cm, but you can set it to a different value for the import operation if you wish. You will be warned if there are some nodes that do not match.

If a node already has mass, the imported masses are added to the existing masses. If a node appears two or more times in a file, the masses at that node are added.

You can also use these files for importing nodes only (using the node coordinates but ignoring the mass data). When you use a nodal mass file, this option is available. It can be convenient to use this option to read nodes first, then read masses, using the same file.

### 32.1.5 File for Nodal Loads

You can import nodal loads only. Element loads must be specified in PERFORM-3D.

In the PERFORM-3D model, the loads are organized in Load Patterns. Load Cases are then built up from the load patterns. You can import only load patterns.

In a single import operation you can import loads into only one load pattern. This does not mean that you must have a separate text file for each pattern, although that is probably a good idea.

In order to import loads you must first create or import the nodes.

The file format is as follows.

- (1) First N lines (as many as you wish) : descriptions of the file. Each line can be of any length, and must end with a line feed (carriage return). These lines are skipped when the data is imported.
- (2) Remaining lines : one line per set of loads. Each line must contain the H1, H2 and V coordinates of the corresponding node, followed by 6 load values, namely the H1, H2 and V forces and the H1, H2 and V moments, all separated by commas. The length unit for the coordinates must be the same as the global length unit for the PERFORM-3D structure. The force and length units for the loads must be the same as the global force and length units for the PERFORM-3D structure.

If the file contains a node that does not exist (i.e., if the coordinates do not match the coordinates of an existing node), the loads are not imported. The tolerance for whether a node exists is equal to the minimum spacing between nodes for the PERFORM-3D model. This spacing is typically 6 inches or 15 cm, but you can set it to a different value for the import operation if you wish. You will be warned if there are some nodes that do not match.

If a node already has loads, the imported loads are added to the existing loads. If a node appears two or more times in a file, the loads on that node are added.

You can also use these files for importing nodes only (using the node coordinates but ignoring the load data). When you use a nodal load file, this option is available. It can be convenient to use this option to read nodes first, then read loads, using the same file.

### ***32.2 Note on Element Import***

---

In PERFORM, elements are divided into Element Groups. If you export elements from PERFORM you must do so one element group at a time, and there will be a different file for each group. If you then import the same elements, you must do so one file at a time. You can read the files into different element groups, corresponding to the groups that were exported, or you can import multiple files into one element group. Since the files are text files you can edit them if you wish. You could concatenate a number of exported files into a single file.

If the file containing element data is created by a program other than PERFORM, the elements may not be divided into files that correspond to element groups in the PERFORM model. For example, in PERFORM, beams and columns are both frame type elements, but they must be in separate element groups. If the file is created by another program, beam and column elements may be in the same file. Also, PERFORM has different 4-node element types, namely shear wall, general wall and slab/shell elements. Elements of these types must be in separate groups. Hence, a file may contain elements that you want to assign to different element types. These situations can be handled as follows.

### 32.2.1.1 *Beams and Columns in One File*

If you have Beam and Column elements together in a file, the steps are as follows.

- (1) In the **Elements** task create the element groups that you need, with no elements.
- (2) In the **Import/Export Structure Data** task, import all of the elements into one of these groups.
- (3) In the **Elements** task, choose the **Change Group** tab and move elements into the required groups. PERFORM allows you to move elements from Beam element groups to Column element groups and vice versa.

There is an alternative method, which may be easier if the elements in the text file are well organized. This is to edit the text file, splitting it into two or more files where each file contains only elements that you want to be of Beam or Column type. You can then import each file into a separate element group.

### 32.2.1.2 *Shear Wall and General Wall Elements in One File*

The steps are the same as for beams and columns. PERFORM allows you to move elements from Shear Wall element groups to General Wall element groups and vice versa.

### 32.2.1.3 *Elements of Incompatible Types in One File*

Suppose you have a file that contains a number of diagonal brace elements, and you want to model some of them as Brace/Other Frame elements and the rest as BRB elements. You can not use the preceding method, because PERFORM does not allow you to copy elements from a Brace/Other Frame element group to a BRB element group. In this case you may be able to edit the file to divide the elements. If this is not convenient you can use the following procedure.

This procedure is more complex than that for separating frame or wall type elements. Since you can not use the Change Group operation, you must use the Import/Export task.

First consider the case where you want to move all of the elements from a Brace/Other Frame group to a BRB group. The steps are as follows.

- (1) In the **Elements** task create the BRB element group that you need, with no elements.
- (2) In the **Import/Export Structure Data** task, export the Brace/Other Frame element group to a file. Be sure not to include the element orientation data, otherwise you will get an error message in Step (4).
- (3) Delete the Brace/Other Frame element group.
- (4) In the **Import/Export Structure Data** task, re-import the elements into the BRB element group.

Second, consider the case where you have a file with elements that you want to divide into different types. Suppose that you want to divide the elements into Brace/Other Frame and BRB groups. The steps are as follows.

- (1) In the **Elements** task create two element groups, both of Brace/Other Frame type.
- (2) In the **Import/Export Structure Data** task, import all of the elements into one of these groups.
- (3) In the **Elements** task, choose the **Change Group** tab and move those elements that you want to be BRB elements into the second group.
- (4) Using the above procedure, export the elements in the second group and re-import them as BRB elements.
- (5) If you wish, use the **Elements** task and **Change Group** tab to divide into smaller element groups.

## ***32.3 Import and Export Procedures***

---

### **32.3.1 Import**

You must set up the data in comma-delimited text files, as described in the preceding section. For a structure of modest size the most direct method is to create one file for the elements, a second file for the masses, and a third for the loads. Import the elements first, in blocks that correspond to the element groups. This will also create the nodes. Then import the masses and loads. In between import operations you may wish to do other operations such as adding supports or defining frames.

For a very large structure it may be better to use a larger number of files. For example, you might have a separate file for each element group, and a separate file for each load pattern.

To start an import operation, choose the **Modeling** phase and the **Import/Export Structure Data** task. Choose the **Import** option and the **Elements, Nodes, Masses** or **Loads** page, and follow the instructions.

For each import operation you must specify the name of the text file. If the file is in the standard USER folder you need to give only the name (including any extensions such as ".txt"). If the file is in some other location you must give the complete path. You can browse to do this.

You must then specify the number of lines to be skipped at the start of the file, and the number of lines to be read. You must always skip any description lines at the start of the file, and you may choose to skip other lines.

For example, suppose that you have a file that has 3 description lines, followed by all of the elements in a structure. Suppose that you want to put the first 100 elements into element Group 1, the next 100 into element Group 2, and the remainder into element Group 3, and that you have already created these three groups (with no elements). This requires three import operations. In the first, skip 3 lines and read 100 lines into Group 1. In the second, skip 103 lines and read 100 lines into Group 2. In the third, skip 203 lines and read the rest of the file into Group 3.

For each operation, after you have specified the file name, the numbers of lines, and possibly some import options, use the **Test**, **Cancel**, **OK** and **Undo** buttons in the usual way.

### 32.3.2 Export

You can export data into the same text files as described in the preceding sections.

To start an export operation, choose the **Modeling** phase and the **Import/Export Structure Data** task. Choose the **Export** option and the **Elements, Nodes, Masses** or **Loads** page, and follow the instructions.

## ***32.4 Component Properties Export/Import***

---

### 32.4.1 General

Component properties can be exported from one PERFORM model and imported into another PERFORM model. You can do the following operations.

- (1) Export a complete set of component properties from one structure and import these properties into another structure.
- (2) Export the properties for selected components of a particular type (e.g., selected fiber cross sections, or selected seismic isolator components) and import them into another structure.
- (3) Create libraries of commonly used components, and import components from these libraries into any structure.

The export/import is done by writing and reading files. These are not text files and they can not be edited. PERFORM currently does not allow component properties to be imported from text files.

### 32.4.2 Export/Import Steps

#### **32.4.2.1 1. Create the components.**

First create the components. If you want to set up a library of standard components you can set up a new structure, define components but no nodes or elements, then export those components.

As shown below, it is easier to export and import a complete set of components than selected components, but you can export and import selected components if you wish. This will require careful organization to keep track of the files.

#### **32.4.2.2 2. To export a complete set of components :**

- (1) In the **Modeling** phase choose the **Component Properties** task.
- (2) At the bottom of the form choose the **Export Components** tab and the **Components of all types** option.
- (3) Press the **Export** button and follow the instructions.

You must create a new file for each set of components that you export. The file will be assigned a ".PF3CXX" extension.

**32.4.2.3 3. To import a complete set of components :**

- (1) In the **Modeling** phase choose the **Component Properties** task.
- (2) At the bottom of the form choose the **Import Components** tab and the **Components of all types** option.
- (3) Press the **Import** button and follow the instructions.

You must read a complete file, with a ".PF3CXX" extension. Usually you will use this import option only for a new structure, where you have not yet defined any components. If you have already defined any components for the current structure, these components will be *overwritten* by the imported components. Also, if you have assigned component properties to any elements, the assignments will all be changed to "NONE".

**32.4.2.4 4. To export selected components :**

- (1) In the **Modeling** phase choose the **Component Properties** task.
- (2) Choose the component type that you want to export from the component type list.
- (3) At the bottom of the form choose the **Export Components** tab and the **Components of this type only** option.
- (4) Press the **Export** button and follow the instructions. You can export all components of the current type, or only selected components.

You can either create a new file or add components to an existing file. The file will be assigned a ".PF3CMP" extension. You can export components of several types to a single file if you wish.

**32.4.2.5 5. To import selected components :**

- (1) In the **Modeling** phase choose the **Component Properties** task.
- (2) Choose the component type that you want to import from the component type list. You may or may not already have components of this type.
- (3) At the bottom of the form choose the **Import Components** tab and the **Components of this type only** option.
- (4) Press the **Import** button and follow the instructions.

You can import all of the components of the current type from the file, or only selected components. If you have already defined some components, be sure to specify whether you want

components with duplicate names to be imported (over-write current component) or ignored (keep current component).

The file can contain components of several types (see preceding section).

An important consideration is that some components can depend on others. For example, a plastic hinge component may depend on a cross section component, and a fiber section component will depend on one or more material components. If you want to import a component (say Component A) that depends on another component (Component B), before you can import Component A you must first import Component B. If Component B does not exist, Component A will not be imported (you will get a warning message).



## **33 Procedure For Reporting Errors**

---

*We do our best to keep PERFORM-3D reliable and free of errors, but we are not always successful. If you report an error you may be asked to send us copies of certain structure files. This chapter describes the procedure.*

### **33.1 Reporting Errors**

---

To report an error, please send an email to the following address :

support@csiberkeley.com

Please describe the problem as clearly as you can. If PERFORM-3D has displayed an error message, please include the text of the message. If you get a "we goofed" message, please go to the end of the ECHO file, where there should be an error message. Please zip the ECHO file and attach it to your email. If the zipped file is long (more than about 1MB) please send just the last few lines of the ECHO file, including the error message.

You will be contacted and advised on how to proceed.

We may ask you to send us the data files for your structure. The procedure is described in the next section.

### **33.2 Procedure For Sending Files**

---

#### **33.2.1 Structure Data**

For a description of the way that PERFORM-3D data is stored, see *Chapter 2, Structures, Folders and Files*. Find the folder for your structure.

You can send us the data as an email attachment, or you can upload it to our ftp site.

### 33.2.2 Send As An Email Attachment

Usually you will send only the structure data, with no analysis results. The reason is that the amount of data to be sent in this case is not large, and it can usually be sent as an email attachment. If there are analysis results the amount of data is much larger, and even a zipped file is likely to be too large (the limit for an email attachment is about 5MB)..

The steps are as follows.

- (1) If you do not have any completed analyses for your structure, go to Step 4.
- (2) If you have analyses and do not need to keep them, open the structure, go to the **Run Analyses** task, and delete the analyses in all analysis series (do not delete the analysis series completely). Go to Step 4.
- (3) If you have analyses and want to keep them, press the **Save as a new structure** button on the toolbar. Save the structure without analysis results. Go to Step 4.
- (4) Zip the folder for your structure, including subfolders. There will be one subfolder for each analysis series. If you have deleted all analysis results these subfolders will be empty, but include them anyway.
- (5) Send us an email explaining the problem, with the zipped file as an attachment.

### 33.2.3 Upload to FTP Site

If the amount of data is too large to email, we may ask that you upload it to an ftp site. If this is necessary we will send you instructions.