



**Getting Started**  
**with 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)

# 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.

# 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.



# PERFORM-3D Getting Started

## Contents

---

<b>1</b>	<b>Installation, Manuals, Support</b>	<b>1-1</b>
1.1	<i>Installation</i>	1-1
1.2	<i>Manuals</i>	1-1
1.3	<i>Technical Support</i>	1-1
1.3.1	Availability	1-1
1.3.2	Help Us to Help You	1-2
1.3.3	Phone Support	1-2
1.3.4	Email Support	1-2
<b>2</b>	<b>Overview of PERFORM-3D</b>	<b>2-1</b>
2.1	<i>Screen Layout</i>	2-1
2.2	<i>Structures</i>	2-2
2.3	<i>Phases</i>	2-3
2.4	<i>Modeling Phase</i>	2-3
2.5	<i>Analysis Phase</i>	2-5
<b>3</b>	<b>Quick Tour : Steel Frame Example</b>	<b>3-1</b>
3.1	<i>Overview</i>	3-1
3.2	<i>Preliminaries</i>	3-1
3.3	<i>Open Structure</i>	3-2
3.4	<i>Nodes</i>	3-2
3.5	<i>Component Properties</i>	3-3
3.6	<i>Beam Cross Sections</i>	3-3
3.7	<i>Column Cross Sections</i>	3-4
3.8	<i>Inelastic FEMA Beam Components</i>	3-5
3.9	<i>Inelastic FEMA Column Components</i>	3-7

3.10	<i>Inelastic Connection Panel Zone Components</i>	3-8
3.11	<i>End Zone Component</i>	3-9
3.12	<i>Elastic P/N/M Release Components</i>	3-9
3.13	<i>Strength Section Components</i>	3-10
3.14	<i>Frame Member Compound Components</i>	3-11
3.15	<i>Summary for Component Properties</i>	3-12
3.16	<i>Elements</i>	3-13
3.17	<i>View Direction and Perspective</i>	3-14
3.18	<i>Frames</i>	3-14
3.19	<i>Load Patterns</i>	3-15
	3.19.1 General	3-15
	3.19.2 Nodal Load Patterns	3-15
	3.19.3 Element Load Patterns	3-15
	3.19.4 Self Weight Loads	3-17
3.20	<i>Import/Export Structure Data</i>	3-17
3.21	<i>Drifts</i>	3-17
3.22	<i>Structure Sections</i>	3-18
3.23	<i>Limit States</i>	3-18
3.24	<i>Delete Elements for Gravity Loads</i>	3-19
3.25	<i>Analysis Phase</i>	3-20
3.26	<i>Load Cases</i>	3-20
	3.26.1 General	3-20
	3.26.2 Gravity Load Case	3-20
	3.26.3 Static Push-Over Load Cases	3-20
	3.26.4 Dynamic Earthquake Load Cases	3-22
3.27	<i>Analysis Series</i>	3-23
	3.27.1 General	3-23
	3.27.2 Series 1	3-24
	3.27.3 Series 2	3-24
	3.27.4 Run Analyses	3-24
3.28	<i>Run Analyses</i>	3-25
3.29	<i>Mode Shapes</i>	3-25

<i>3.30 Energy</i>	<i>3-25</i>
<i>3.31 Limit State Groups</i>	<i>3-26</i>
<i>3.32 Deflected Shapes</i>	<i>3-26</i>
<i>3.33 Time Histories</i>	<i>3-27</i>
<i>3.34 Hysteresis Loops</i>	<i>3-28</i>
<i>3.35 Moment and Shear Diagrams, and Detailed Deflections</i>	<i>3-28</i>
<i>3.36 Behavior vs. Performance Assessment</i>	<i>3-29</i>
<i>3.37 Push-Over Plot</i>	<i>3-29</i>
<i>3.38 Usage Ratio Graphs</i>	<i>3-31</i>
<i>3.39 Combinations and Envelopes</i>	<i>3-32</i>
<i>3.40 Conclusion</i>	<i>3-33</i>





# 1 Installation, Manuals, Support

---

## 1.1 Installation

---

Please follow the installation instructions provided in the separate installation document included in your PERFORM-3D package, or ask your system administrator to install the program and give you access to it.

## 1.2 Manuals

---

PERFORM-3D has the following manuals.

- (1) This *Getting Started* manual. If you are a new user, it is essential that you read Chapter 2 (Overview of PERFORM-3D) and that you take the Quick Tour in Chapter 3.
- (2) *PERFORM-3D User Guide*. This gives detailed instructions on the use of PERFORM-3D.
- (3) *PERFORM Components and Elements*. This describes the components and elements that are available in PERFORM-3D and PERFORM-COLLAPSE, and presents other background information.

The manuals are available as PDF files on the PERFORM-3D CD. They currently can not be accessed from within the program using a Help menu.

## 1.3 Technical Support

---

### 1.3.1 Availability

Free technical support is available from Computers and Structures, Inc. (CSI) or your dealer via phone and e-mail for 90 days after the software has been purchased. After 90 days, priority technical support is available only to those with a yearly Support, Upgrade and Maintenance plan (SUM). If you do not have a current SUM

subscription, you can obtain technical support, but via e-mail only and at the non-priority level. Please contact CSI or your dealer to inquire about purchasing a yearly SUM subscription.

If you have questions regarding use of the software, please consult the manuals and any other printed information included with your product. If you can not find a solution, then contact us as described in the following sections.

### 1.3.2 Help Us to Help You

Whenever you contact us with a technical-support question, please provide us with the following information to help us help you:

- (1) A description of your model, including a picture, if possible.
- (2) A description of what happened and what you were doing when the problem occurred.
- (3) The exact wording of any error messages that appeared on your screen.
- (4) A description of how you tried to solve the problem.
- (5) The computer configuration (make and model, processor, operating system, hard disk size, and RAM size).
- (6) Your name, your company's name, and how we may contact you.

### 1.3.3 Phone Support

Priority phone support is available to those with a current SUM subscription. For users in North America, you can contact CSI between 8:30 A.M. and 5:00 P.M., Pacific time, Monday through Friday, excluding holidays, at (510) 649-2200.

When you call, please be at your computer and have the program manuals at hand.

### 1.3.4 Email Support

Email support is available as follows.

- (1) Send an e-mail with a description of the problem to:

*support@csiberkeley.com*

- (2) Visit the CSI web site at [www.csiberkeley.com](http://www.csiberkeley.com), and use the Support link to submit a request for technical support.

If you send us e-mail, be sure to include all of the information requested in the “Help Us to Help You” section.



## 2 Overview of PERFORM-3D

---

*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.*

*This chapter provides an overview of the program.*

### 2.1 Screen Layout

---

Figure 2.1 shows the screen layout for most tasks. The minimum required screen resolution is 1024 x768 pixels. Figure 2.1 is the layout at this resolution. If the resolution is finer the size of the graphics panel is increased.

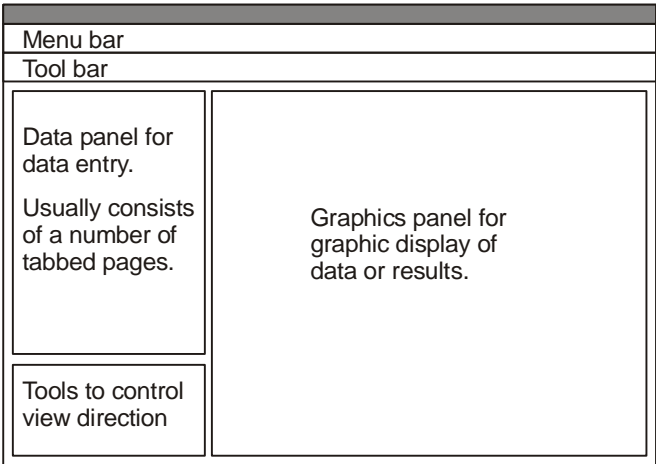


Figure 2.1 Screen Layout For Most Tasks

Figure 2.2 shows the screen layout for specifying the properties of linear and nonlinear components. This screen has diagrams to explain the required properties.

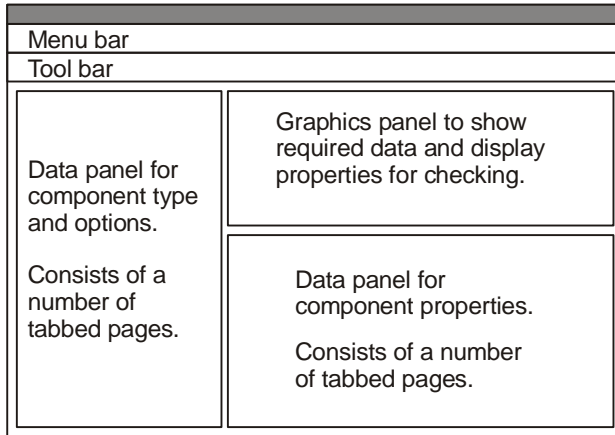


Figure 2.2 Screen Layout For Component Properties

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.

## 2.2 Structures

---

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 (see Chapter 3). 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.

## 2.3 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.

## 2.4 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.



## Overview of PERFORM-3D

- ■ Nodes task. Specify node data, including coordinates, support conditions, slaving constraints (e.g. rigid floor diaphragms) and structure masses.



Component Properties task. Specify properties for nonlinear and linear components. This is the most critical and complex task.



Elements task. Specify element data, including element types, locations and properties.



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.



Load Patterns task. Specify load patterns for node, element and/or self weight loads.



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, you can use this task to import much of the structure into PERFORM-3D.



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.



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.



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.



Inactive Elements task. Specify elements that are inactive for gravity loads.

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.

## 2.5 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.



Run Analyses task. Run static and dynamic 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.



Energy Balance task. Plot graphs that show how much energy of different types is dissipated by the structure.

## Overview of PERFORM-3D



**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.



**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.



**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.



**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).



**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.

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.



**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.



**Usage Ratio task.** For any analysis and limit states, plot graphs of usage ratio versus time, drift or load factor.



Combinations/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.



## 3 Quick Tour : Steel Frame Example

---

*PERFORM-3D includes a simple steel frame example structure. The quickest way to introduce yourself to PERFORM-3D is to take a tour of this example.*

*This chapter leads you through the example. We strongly recommend that you take this tour.*

### 3.1 Overview

---

A nonlinear model of a structure is much more complex than a linear model. The goal of this quick tour is to give you an idea of what can be done using PERFORM-3D. It is not a tutorial or a training. The purpose of the example structure is to introduce you to PERFORM-3D. It is not necessarily a practical example.

For this tour, do not worry about details. You can consider these later, when you develop your own model.

### 3.2 Preliminaries

---

You must have installed PERFORM-3D, and you must have set up the example structures and the earthquake records, as described in the installation instructions.

As you take the tour, do not change any data. You can do this later if you wish (we suggest doing this on a copy of the example structure). If you do change data inadvertently, press the **Cancel** or **UnChange** button to undo the changes. If in doubt, exit PERFORM-3D, and when asked if you want to save the changes to the structure, choose "No".

Start PERFORM-3D. It helps if you have set up a desktop icon.

### 3.3 *Open Structure*

---

In the opening screen, press **Open an Existing Structure**. Then choose the **Default** option.

You should see **Example CSI1** in the table. If you do not, this means that you have not set up the example structure. In this case, press **Cancel**, and then **Exit**.

If you see **Example CSI1**, click on it to highlight it, then press **Open**. Alternatively, double click on it in the table.

You will get an OVERALL STRUCTURE screen. Press **OK**.

Take a couple of minutes to look over the buttons in the toolbar. Hover the cursor over each button to see a brief explanation (tool tip). The current phase is the Modeling Phase, with red task buttons.

### 3.4 *Nodes*

---

Press the toolbar button for the **Nodes** task (or choose **Nodes** from the **Task** menu). When you set up a structure, you will use this task to set up nodes, supports, masses and slaving constraints.

In the data panel you will be in the **Nodes** page. Without worrying about the details, and without entering any data, click the **Grid**, **Move**, **Delete**, **Single**, **Interpolate** and **Duplicate** tabs to get an idea of the tools that are available for setting up nodes.

Click the **Supports** tab. Note that the perimeter column bases are fixed, and the interior column bases are hinged.

Click the **Masses** tab. The masses are lumped at the centroids of the floors. There is only one mass "pattern", based on DL + 0.25LL. There can be up to 6 patterns.

Click the **Slaving** tab. There are 3 rigid floor slaving constraints. Cycle through the list to show each constraint. A convenient way to cycle through a list is to press the "cycle" button adjacent to the list. Left

click the button to cycle forwards, and right click to cycle backwards. Alternatively, choose from the list.

### 3.5 **Component Properties**

---

Press the toolbar button for the **Component Properties** task. This is the most complex and important task. The properties for a nonlinear component can be complex. For this example the properties are kept simple.

There are six main tabs in the data panel, for **Inelastic** components, **Elastic** components, **Cross Sections**, **Materials**, **Strength Sections** and **Compound** components. There are also **Import Component** and **Export Component** tabs. These last two are not used for this example.

This example uses the following main components.

- (1) Three beam cross sections, one each for the perimeter girders, the N-S interior girders and the E-W interior girders.
- (2) Two column cross sections, one for the perimeter columns and one for the interior columns.
- (3) One inelastic FEMA Beam, Steel Type component, for the perimeter girders. These girders can yield.
- (4) One inelastic FEMA Column, Steel Type component, for the perimeter columns. These columns can yield.
- (5) One inelastic Connection Panel Zone component, for the panel zones in the beam-to-column connections in the perimeter frame. These panel zones can yield.
- (6) Two elastic P/V/M Release components. These are explained later.
- (7) A number of Frame Compound Components, for modeling girder and column members. These are explained later.

These are described in the following sections.

### 3.6 **Beam Cross Sections**

---

Choose (click) the **Cross Sects** tab. In the BEAM TYPE SECTIONS part of the **Type** list, choose **Beam, Standard Steel Section**. You will see that there are 3 sections of this type. Also, a form is shown with



three tabs, namely **Dimensions and Stiffness**, **Inelastic Strength** and **Elastic Strength**.

In the **Name** list, choose **Perimeter Girders, W27x94**. The dimensions and section properties are shown in the **Dimensions and Stiffness** page, with green backgrounds. This is a standard steel section, and the properties with green backgrounds are obtained from the standard section properties. The Young's Modulus and Poisson's Ratio values have white backgrounds. These values must be provided directly. The Shear Modulus is calculated from the other two values (the "=" in "Shear modulus =" means that this is a calculated value).

Click the **Inelastic Strength** tab. The Plastic Section Modulus and Shear Area are green, since these are obtained from the section properties. For **Bending at U Point** the **Yes** option has been chosen, and a Yield Stress of 50 ksi has been specified. Hence the plastic bending moment is defined ( $50 \times 278 = 13900$  k.in.). When this cross section is used for an inelastic FEMA Beam, Steel Type component (see later), this plastic moment is used to define the component strength.

A yield stress could also be specified for the Shear at U Point. This strength could then be used for an inelastic Shear Hinge component. This is not needed for this example

There are two other beam sections, namely **E-W interior girders, W18x35** and **N-S interior girders, W24x55**. These girders remain elastic, so inelastic strengths are not specified. Choose **E-W interior girders, W18x35**, and click the **Elastic Strength** tab. The Plastic Section Modulus and Shear Area are green, since these are obtained from the section properties. For **Nominal Bending Strength** the **Yes** option has been chosen, and a yield stress of 50 ksi has been specified. Hence the moment strength of the section is defined ( $50 \times 66.5 = 3325$  k.in.). When this cross section is used for a Moment Strength Section (see later), this is used in a to define its nominal strength

### **3.7 Column Cross Sections**

---

In the **Cross Sects** tab and the COLUMN TYPE SECTIONS part of the **Type** list, choose **Column, Standard Steel Section**. There are 2 sections of this type. As with the beam section, a form is shown with

three tabs, namely **Dimensions and Stiffness**, **Inelastic Strength** and **Elastic Strength**.

In the **Name** list, choose **Perimeter Columns, W14x193**. The dimensions and section properties are shown in the **Dimensions and Stiffness** page. Click the **Inelastic Strength** tab. For **Strength at U Point** the **Yes** option has been chosen, and Yield Stresses of 50 ksi have been specified for tension, compression and bending about both axes. In the **Shape of P-M2-M3 Yield Surface** frame, press **Plot**. This shows the shape of the P-M-M yield surface in the P-M2 and P-M3 planes (M3 is bending about Axis 3 for the column element, which is the section strong axis). The shape of the yield surface can be changed by varying the parameters. When this cross section is used for an inelastic FEMA Column, Steel Type component (see later), this yield surface is used to define the component strength. For an explanation of plastic hinge formation with P-M-M interaction, see the document *PERFORM Components and Elements*.

In the **Name** list, choose **Interior Columns, W14x82**. These are gravity columns that remain elastic. Click the **Elastic Strength** tab. For **Nominal Strengths** the **Yes** option has been chosen, and yield stresses of 50 ksi have been specified. This defines a P-M-M yield surface that could be used for a strength section component to check that the interior columns remain elastic (as the perimeter frame yields under lateral loads there can be significant bending in gravity columns, and it may be a good idea to check that they remain elastic).

### **3.8 Inelastic FEMA Beam Components**

---

The perimeter girders can yield, by forming plastic hinges at the column faces. There are a number of ways to model inelastic behavior in beams. The simplest is to use the "FEMA Beam" model, so called because FEMA 356 explicitly mentions this type of model, and gives deformation capacities. For a discussion of this and other beam models see the document *PERFORM Components and Elements*.

Click the **Inelastic** tab. In the BEAM TYPE COMPONENTS part of the **Type** list, choose **FEMA Beam, Steel Type**. There is one component of this type. A form will be shown with seven tabs, for properties of different types. Only a few of these properties are used for this example.

In the **Name** list, choose **Perimeter girders**. Note that there are a number of options for the properties, namely Shape of Relationship, Use Cross Section, Symmetry, Deformation Capacities, Strength Loss, Cyclic Degradation and Upper/Lower Bounds. The shape of the relationship is E-P-P (elastic perfectly plastic) a cross section is used, and deformation capacities are specified. There is no strength loss, no cyclic degradation, and upper/lower bounds are not used.

The **Section and Dimensions** page shows that the Perimeter Girders, W27x94 cross section is used for this component. Click the **Elastic Stiffness** tab to show the section properties. Since these are all obtained from the cross section they have green backgrounds.

Click the **Basic F-D Relationship** tab to show the action-deformation (F-D) relationship for the component. Note that F is end moment and D is end rotation. Note also that D values are multiples of the yield rotation, DY. For example, when D = 1 a plastic hinge just forms, and when D = 7 the beam end rotation is 7 times the end rotation when the hinge first formed. This is not the same as plastic hinge rotation (for example, D = 1 corresponds to zero plastic hinge rotation, since the hinge has just formed).

Only two properties are required for the F-D relationship, namely the U bending moment (FU) and the X rotation (DX). The figure above the tabbed pages defines FU and DX. FU is obtained from the cross section. DX is input as 20, which corresponds to a large rotation, well beyond the rotation capacity of the component. As shown later, if the rotation in any component exceeds DX, the analysis stops, since there is no point in continuing. Since the component has e-p-p behavior the boxes for trilinear behavior are disabled, and since the properties are symmetrical, the boxes for all negative properties are disabled.

Click the **Strength Loss** tab, to show the properties that would be required if "Yes" were chosen for the Strength Loss option. Since "No" is chosen, the boxes in this page are all disabled. Do not worry about these properties for this example.

Click the **Deformation Capacities** tab, to show the capacity values. Capacities can be input for up to 5 performance levels. For this structure, Level 1 is Immediate Occupancy (IO), Level 2 is Life Safety (LS) and Level 3 is Collapse Prevention (CP). Levels 4 and 5 are not

used. The capacity values are those specified by FEMA 356. Note that these are multiples of the yield rotation.

Click the **Cyclic Degradation** tab, to show the properties that would be required if "YX+3" were chosen for the Cyclic Degradation option. Since "None" is chosen, the boxes in this page are all disabled. Do not worry about these properties for this example.

Click the **Upper/Lower Bounds** tab, to show the properties that would be required if "Yes" were chosen for the Upper/Lower Bounds option. Since "No" is chosen, the boxes in this page are all disabled. Do not worry about these properties for this example.

An important thing to note is that if you consider symmetric e-p-p behavior with no strength loss or cyclic degradation, you do not have to input many properties, and these properties are quite simple. If you want to consider more complex behavior you can do so, but you must input more complex properties.

You might like to choose different options for the properties, to see how the forms change. If you do, be sure to press **UnChange** to restore the properties to their previous values.

### **3.9 Inelastic FEMA Column Components**

---

The perimeter columns can yield, by forming plastic hinges at the top and/or bottom of each story. There are a number of ways to model inelastic behavior in columns. The simplest is to use the "FEMA Column" model, which is similar to the FEMA Beam model except that the plastic hinges have P-M-M interaction.

In the **Inelastic** tab and the COLUMN TYPE COMPONENTS part of the **Type** list, choose **FEMA Column, Steel Type**. There is one component of this type. In the **Name** list, choose **Perimeter columns**. A form will be shown with eight tabs, for properties of different types. Click the various tabs to see the properties that have been, or might need to be, specified. As with the FEMA Beam component, the deformation D is column end rotation, as a multiple of the yield rotation.

### **3.10 Inelastic Connection Panel Zone Components**

In a model for linear analysis you may not need to have an explicit model for the panel zones in the girder-to-column connections. You may be able to ignore or approximate the effects of deformation in the panel zones, and you can calculate the shear forces in the panel zone from the bending moments and shear forces in the connected girders and columns. However, if the panel zone can yield, in a nonlinear analysis, it should be modeled using an explicit panel zone component and element.

In the **Inelastic** tab and the PANEL TYPE COMPONENTS part of the **Type** list, choose **Connection Panel Zone**. There is one component of this type. In the **Name** list, choose **Panel zones, W14x193 col, W27x94 beam**. This is the panel zone component for all connections in the perimeter frame. A form will be shown with seven tabs for properties of different types. Click on the various tabs to see the properties that have been, or might need to be, specified.

Note that  $F$  is the bending moment transferred through the connection (essentially the sum of the column moments above and below the connection, or the sum of the girder moments to the left and right of the connection). The deformation  $D$  is the shear strain in the panel zone, which is essentially equal to the difference between the rotation of the column section and the rotation of the girder section at the connection (when the panel zone deforms, these sections are not exactly at right angles to each other). For a description of the panel zone element, see the document *PERFORM Components and Elements*.

Click the **Basic Relationship** tab. Given the girder and column sizes, the properties with green backgrounds are calculated by PERFORM-3D. The basic  $F$ - $D$  relationship is trilinear. In addition, a "parallel" strain hardening stiffness has been added, equal to 2% of the basic stiffness.

Click the **Deformation Capacities** tab, to show the capacity values. As before, Level 1 is IO, Level 2 is LS and Level 3 is CP. The capacity values (shear strains) are the FEMA 356 capacities.

Click the other tabs to see properties that could be specified.

### 3.11 *End Zone Component*

---

At the ends of beam elements, it is common practice to assume that the region within the beam-to-column connection is stiffer than the body of the beam. Sometimes this region is assumed to be rigid. Also, plastic hinges in beams are commonly assumed to occur at the column face. Similar assumptions are made for columns.

In PERFORM-3D, stiff end zones are used to model the stiffer connection regions in beam and column elements. These are elastic components. Note that stiff end zones and connection panel zones are different, with very different properties.

PERFORM-3D automatically sets up a Default End Zone component. This component is assumed to be 10 times stiffer than the body of the beam or column, and its length is calculated by PERFORM-3D based on the dimensions of the adjacent column (for an end zone in a beam) or beam (for an end zone in a column).

Click the **Elastic** tab. In the BEAM & COLUMN COMPONENTS part of the **Type** list, choose **End Zone for a Beam or Column**. The properties for the Default End Zone will be shown. The form shows the properties, and how the end zone lengths are calculated.

There are no other end zone components for this example.

### 3.12 *Elastic P/V/M Release Components*

---

In an analysis model it is usual to specify moment releases for connections that are non-moment resistant. In PERFORM-3D, releases are treated as components.

In the **Elastic** tab and the BEAM & COLUMN COMPONENTS part of the **Type** list, choose **Linear P/V/M Hinge or Release**. There are two components of this type.

In the **Name** list, choose **Moment hinge for girders**. This is a simple moment release, about the strong axis (Axis 3) of a beam element. As shown later, it is inserted in Frame Member Compound Components to

account for non-moment-resistant connections in the perimeter and interior girders.

In the **Name** list, choose **M2-M3 hinge for columns**. This has moment releases about both axes. As shown later, it is inserted in a Frame Member Compound Component to account for the fact that the perimeter columns have non-moment-resistant connections at the foundation.

### 3.13 *Strength Section Components*

---

The example structure uses strength sections of two types, namely (1) moment strength sections at midspan of the interior girders, to check that they remain elastic in bending, and (2) shear strength sections at bolted (non-moment-resistant) connections in the perimeter girders, to check that they remain elastic.

Click the **Strength Sects** tab. In the BEAM TYPE STRENGTH SECTIONS part of the **Type** list, choose **Moment Strength Section**. There are two strength sections of this type, one for the E-W interior girders and one for the N-S interior girders.

In the **Name** list, choose **E-W interior girder bending**. The form shows the cross section and the strength. The strength is obtained from the cross section. The capacity factors are 1.0 for the IO, LS and CP performance levels. The properties for the **N-S interior girder bending** strength section are similar.

In the **Strength Sects** tab and the BEAM TYPE STRENGTH SECTIONS part of the **Type** list, choose **Shear Force Strength Section**. There is one strength section of this type. In the **Name** list, choose **Perimeter girder bolted connection**. The strength is for the bolted connections where the perimeter girders frame into the weak axes of the perimeter columns. A cross section is not used. Click the **Strength** tab to show the shear strength. The capacity factors are 1.0 for the IO and LS performance levels, indicating slip critical behavior. The capacity factor for the CP level is larger, indicating bearing behavior. The nominal strength and capacity factors are for demonstration purposes only, and are not necessarily realistic.

### 3.14 **Frame Member Compound Components**

---

Click the **Compound** tab. There are nine Frame Member Compound Components. These are as follows.

- (1) *Perimeter girder, no releases.* This is for perimeter girders that have moment resisting connections at both ends.
- (2) *E-W interior girder.* This is a W18x35 section with non-moment resisting end connections.
- (3) *Perimeter columns, bottom story, pinned.* This is for the bottom story of the perimeter columns, where (a) there is no stiff end zone at the bottom (foundation) end, and (b) there is an M2-M3 moment release at the foundation end. Note that the nodes at the column bases are fixed. The moment release accounts for the non-moment resisting connection at the column base. The same behavior can be obtained by specifying simple supports (rather than fixed) and omitting the release component. However, it is more direct to use a release.
- (4) *Perimeter columns, upper stories.* This is for the upper stories of the perimeter columns, which have end zones and no releases.
- (5) *Interior columns, all stories.* These columns have non-moment resisting connections at the base. This is modeled using pinned support conditions. The floor girders also have non-moment-resisting connections, so stiff end zones are not used.
- (6) *Perimeter girder, release at end I.* This is for perimeter girders that have non-moment-resisting connections to the columns at element end I (they frame into the weak axis of the column, with a bolted connection).
- (7) *Perimeter girder, release at end J.*
- (8) *N-S interior girder.* This is a W24x55 section with non-moment resisting end connections.
- (9) *Perimeter columns, bottom story, no pins.* This is not used. It omits the moment release, and can be used to study the effect of using a moment-resisting connection at the foundation level for the perimeter columns (by assigning this component to the columns in story 1, rather than the component with a moment release).

These components are in the order that they were defined. There is currently no way to re-order them in PERFORM. You can, however,



use the "filter" feature. For example, in the "Text for filter" box enter "Perim" (with an upper case "P" and without the quotes) and press **Filter**. Only the perimeter girder and column components are listed. If you use appropriate names, you can use this feature to simplify long component lists.

In the **Name** list, choose **Perimeter girder, no releases**. The **Basic Components** page shows the basic components that make up the compound component. There are two end zones, and two FEMA Beam components. The FEMA Beam components are of equal length, indicating that the inflection point for the girder is essentially at midspan. The **Strength Sections** page shows no strength sections.

In the **Name** list, choose **Perimeter girder, release at end I**. The **Basic Components** page shows the basic components. There is a moment release at end I, and no end zone (the connection is assumed to be at the column web). In this case the inflection point is at end I of the component. Hence, the first FEMA Beam component is very short (0.02 times the clear span, which is the shortest allowable length) and the second FEMA Beam component is long (0.98 times the clear span). The **Strength Sections** page shows a shear strength section at end I, to check the connection strength.

Choose other components from the **Name** list to see how they are constructed. You may notice that the interior girders have end zones. This is inconsistent if the girder frames into a column in the weak direction, since in that case the end connection is at the column web. However, it is simpler and makes no significant difference.

### **3.15 Summary for Component Properties**

---

It can be a lot of work, and requires careful planning, to set up the components. However, there is no shortcut for this important task. The behavior of the nonlinear model depends on the properties of the components, and you must be careful when you specify these properties.

### 3.16 Elements

---

Press the toolbar button for the **Elements** task (or choose **Elements** from the **Task** menu). When you set up a structure, you will use this task to add and/or delete elements, specify the orientations of element local axes, and assign components to elements.

The **Current Group** list at the top of the data panel lists five element groups. Choose each group in turn. In the graphics panel the elements in the current group are shown light blue.

Choose the **Perimeter girders** group. Click the **Group Data** tab. This page shows that the elements are of Beam type and that there are no P- $\Delta$  effects. It is usually necessary to consider P- $\Delta$  effects only for elements that have vertical projections, such as columns or sloping beams. The Scale Factor for Beta-K Damping is 1.0. This will usually be the case.

Click the **Add Elements** tab. This page is for adding elements. There are a number of generation options. Do not worry about the details.

Click the **Delete Elements** tab. This page is for deleting elements. To delete elements (in the current group), you would select them in the graphics panel and press **Test**.

Click the **Properties** tab. This page is for assigning component properties to elements. In the graphics panel, click on any of the light blue elements. Its color changes to red, and the component that is assigned to the element is shown in the **Component Currently Assigned** frame. Press **Show Properties**. This shows the properties for the compound component that is assigned to the element. In the COMPONENT LIST table, click on any basic component to highlight it. Then press **Show Properties**. This shows the properties for the basic component. When you show the properties in this way, you can view them only. If you want to change the property values, you must use the **Component Properties** task. Press **Close** to return to the **Elements** task.

Press **Clear Selected Elements**. Then double click on an element in the graphics panel. Several elements will change color to red. These

## Quick Tour : Steel Frame Example

elements all have the same properties as the element that you clicked (i.e., the same Frame Member Compound Component).

Click the **Orientations** tab. This page is for specifying the orientations of the element local axes. The symbols on the elements show the local axes. Double click on any element. All elements with the same orientation change color to red, and the orientation data is shown in the data panel. For the **Perimeter girders** group the element Axis 2 is vertical for all elements in the group. In the **Current Group** list, choose **Perimeter columns**. Double click on an element to show all elements with the same orientation. You will see which elements have Axis 2 oriented in the H1 direction, and which in the H2 direction.

Do not worry about the **Change Group** tab. This can be used to move elements from one group to another.

### **3.17 View Direction and Perspective**

---

If you wish, experiment with the tools for changing the view direction and amount of perspective (at the bottom of the data panel).

Press **Plan** to show a plan view with no perspective. Choose a **Distance** option and press **OK** to add perspective. Press **H1** and **H2** to show elevations. Change the view angles if you wish. Press **Basic** to return to the basic view. This is usually a convenient view with an appropriate amount of perspective, but you can make another view the basic view if you wish.

### **3.18 Frames**

---

At the right end of the tool bar is a list of "frames". The frame that is currently shown is the default **Entire Structure**. Choose a frame from the list (or cycle through the list using the cycle button). The chosen frame is shown bright, and the rest of the structure is dark gray. Press the **S-F** (Structure-Frame) button located beside the frame list. Only the frame is shown. Press it again to cycle between structure and frame-only views.

Press the toolbar button for the **Frames** task. This task is used to add, modify, delete and duplicate frames. For this example, do not worry about the details.

## 3.19 Load Patterns

---

### 3.19.1 General

Load Patterns are not the same as Load Cases. Two typical Load Patterns might be (1) dead load (DL) and (2) design live load (LL). A Load Case might scale and combine these patterns to apply, say,  $1.1DL+0.25LL$ . Load Patterns are defined in the **Load Patterns** task. Load cases are defined in the **Set Up Load Cases** task (in the **Analysis Phase**).

Press the toolbar button for the **Load Patterns** task. There are three tabs in the data panel, for **Nodal Loads**, **Element Loads** and **Self Weight**.

### 3.19.2 Nodal Load Patterns

If you are in a frame view, return to the Entire Structure view. Then click the **Nodal Loads** tab. There are two nodal load patterns, namely (1) H1 load based on masses and (2) H2 load based on masses. These loads can be used for static push-over analyses.

From the list of nodal load patterns choose **H1 load based on masses**. The loads are shown in the graphics panel. The roof, floor 3 and floor 2 masses (from the Nodes task) are in the ratio 4.3 : 4.37 : 4.37. The nodal loads for this pattern have these values, applied in the H1 direction. For the **H2 load based on masses** pattern, the loads are the same but in the H2 direction.

### 3.19.3 Element Load Patterns

Click the **Element Loads** tab. Be warned that this is a little complicated.

There are two element load patterns, namely (1) Gravity DL and (2) Gravity LL. The gravity DL is a simplified loading for this example, consisting of the following.

## Quick Tour : Steel Frame Example

- (1) 1 kip/ft (0.0833 kip/inch) on the E-W perimeter girders.
- (2) 2 kip/ft (0.1667 kip/inch) on the N-S perimeter girders.
- (3) 1 kip/ft (0.0833 kip/inch) on the E-W interior girders.
- (4) 2 kip/ft (0.1667 kip/inch) on the N-S interior girders.

The gravity LL is exactly the same. However, it is set up as a separate load pattern.

In the list of load cases at the top of the data panel. Choose **Gravity DL**. In the list of element groups, choose **Perimeter Girders**. The E-W girders in this group have loads of 1 kip/ft. This is Subgroup 1 of the Perimeter Girders group. The elements in this subgroup are shown green. The N-S girders have loads of 2 kip/ft. This is Subgroup 2 of the Perimeter Girders group. If you choose **Subgroup No. 2**, the elements in that subgroup are shown green. All of the elements in any subgroup have the same loads.

Click the **Add Loads** tab. In the **Subgroup No.** list choose subgroup 1. In the graphics panel you will see that all E-W perimeter are green, and all N-S girders are white. The green elements are the elements in the current subgroup (subgroup 1). A small window in the graphics panel shows the loads on the elements in this subgroup (0.0833 kip/inch, or 1.0 kip/ft). If the small window obscures the structure, click in it and drag it to another location in the graphics panel.

In the **Subgroup No.** list choose subgroup 2. In the graphics panel you will see that all N-S perimeter girders are green, and all E-W girders are white. Again, the green elements are the elements in the current subgroup (subgroup 2), and the small window in the graphics panel shows the loads on the elements in this subgroup (0.1667 kip/inch , or 2.0 kip/ft).

The loads were defined by (a) specifying subgroups by selecting elements, and (b) specifying loads in the **Add Loads** page. The load on an element can be a combination of distributed (linearly varying) and point loads. For this example there is only uniformly distributed load.

Still within the Gravity DL load pattern, in the list of element groups choose **Interior Girders**. The E-W girders in this group have loads of 1 kip/ft. This is Subgroup 1 of the Interior Girders group. The N-S girders have loads of 2 kip/ft. This is Subgroup 2 of the Interior Girders group. Click the **Add Loads** tab, and choose the subgroup number as

before, to show the elements in each subgroup and the corresponding loads.

You can repeat this for the Gravity LL pattern. For this example the loads are the same as for Gravity DL.

### 3.19.4 Self Weight Loads

Self weight loads can be specified for wall and other elements. There are no self weight loads for this example.

## 3.20 *Import/Export Structure Data*

---

Nodes, elements, nodal loads and masses can be imported from other computer programs using text files. This feature is not used for this example. If you wish, you can press the toolbar button for the **Import/export structure data** task, to get an idea of the procedure.

## 3.21 *Drifts*

---

In a static push-over analysis, the push-over (capacity) curve is usually a graph of base shear vs. roof drift. At least two drifts must be defined, in the H1 and H2 directions at the roof. Drifts can also be used for other purposes, including setting up drift limit states. For this example, drifts are set up at the roof and for each of the stories.

Press the toolbar button for the **Drifts and deflections** task. There are two tabs in the data panel, for **Drifts** and **Deflections**.

In the **Drifts** page, choose each of the drifts in the **Name** list. The drifts are shown in the graphics panel. These are all "simple" drifts, where the drift value is the H displacement of the upper node minus the H displacement of the lower node, divided by the V distance between the nodes (i.e., in PERFORM a "drift" is actually a drift ratio).

Drifts can also be "distortion" drifts. These are not used in this example. Deflections are also not used.

## 3.22 *Structure Sections*

---

Press the toolbar button for the **Structure sections** task. There are three tabs, namely **Define Sections**, **Strengths** and **Groups**. Strengths and Groups are not used for this example.

There are three structure sections for this example, namely (1) Complete Structure at Base, (2) Perimeter Frame at Base and (3) Interior Columns at Base.

Choose **Complete Structure at Base** in the section list at the top of the page. Then cycle through the element groups in the element group list. For the **Perimeter columns** group you will see that all columns in the bottom story are red, with green squares at their lower ends. This means that these elements have been cut, at their lower (I) ends. You will see the same thing for the **Interior columns** element group.

Choose **Perimeter Frame at Base** in the section list. When you cycle through the element groups you will see that only the columns in the **Perimeter columns** group have been cut. Similarly for **Interior Columns at Base** section.

## 3.23 *Limit States*

---

A nonlinear analysis can produce a huge volume of analysis results. Limit states distill the results down to a few usage ratios that tell you whether the structure satisfies the performance requirements.

In the **Component properties** task, deformation and strength capacities were specified. At each step in a structural analysis, PERFORM-3D calculates, for every component, the demand that corresponds to each capacity, and hence calculates demand/capacity (D/C) ratios. At the end of the analysis, the D/C ratio for any component is the largest ratio for that component at any step in the analysis. There can be thousands of components, and hence thousands of D/C ratios, and it is impractical to consider them individually. A limit state covers a number of components. The usage ratio for a limit state is the largest D/C ratio for any component covered by the limit state. For a limit state to be satisfied, its usage ratio must not exceed 1.0.

Press the toolbar button for the **Limit states** task. This is the second most important task, after **Component properties**.

The limit states for this example are simple, and there are only a few of them. A large structure can have many limit states, and they can be more complex.

In the **Type** list choose **Deformation**. There are 9 limit states of this type. In the **Name** list, choose **Perimeter column rotation, IO**. The **LIST OF CONDITIONS** table shows that this limit state has only one condition. There could be up to 20 conditions for a more complicated limit state. The condition has the following properties.

- (1) It covers all elements in the **Perimeter columns** element group.
- (2) It covers only **FEMA Column, Steel Type** components in those elements.
- (3) It covers only **Column End Rotation** deformations and D/C ratios.
- (4) It covers D/C ratios for only the Level 1 (IO) deformation capacities.
- (5) The **Limit on D/C Ratio** is 1. This means that a D/C ratio of 1.0 corresponds to a limit state usage ratio of the same value. If the D/C Limit were, for example, 1.25, a D/C ratio of 1.0 would correspond to a limit state usage ratio of 0.8 ( $=1.0/1.25$ ). In effect, the D/C Limit increases or decreases the deformation capacity of the component, in this case by a factor of 1.25.

The **CONDITION TO BE ADDED** section is used to set up conditions for the **LIST OF CONDITIONS** table. Do not worry about this section.

Cycle through the other deformation limit states to see the corresponding conditions. Also look at the **Strength** and **Drift** limit states.

### **3.24 Delete Elements for Gravity Loads**

---

Press the toolbar button for the **Delete elements for gravity load** task. This task is not needed for this example. It allows you to delete certain elements for gravity load (usually diagonal braces) and restore them for push-over and dynamic loads.



### 3.25 Analysis Phase

---

This completes the tour of the Modeling Phase. Press the toolbar button for the **Analysis Phase**. The task buttons change, and are blue.

### 3.26 Load Cases

---

#### 3.26.1 General

Press the toolbar button for the **Set up load cases** task.

There are six load case types in the **Load Case Type** list. The example structure uses the **Gravity, Static Push-Over** and **Dynamic Earthquake** types.

#### 3.26.2 Gravity Load Case

In the **Load Case Type** list, choose **Gravity**. There is one load case of this type, namely **Gravity DL + 0.25LL**. Choose this load case in the **Load Case Name** list.

The **Analysis Method** for this load case is Linear. This is usually the case for gravity loads. However, in some cases there can be nonlinear behavior under gravity load, and in such cases the method must be Nonlinear.

The **LOAD PATTERN LIST** shows the load patterns that are used to set up the load case. These can be any combination of nodal, element and self weight loads. For this example the load case consists of 1.0 times the **Gravity DL** element load pattern plus 0.25 times the **Gravity LL** element load pattern.

#### 3.26.3 Static Push-Over Load Cases

In the **Load Case Type** list, choose **Static Push-Over**. There are six load cases of this type.

In the **Load Case Name** list choose **H1 push, uniform displ**. This is often the easiest way to specify push-over loads. The **Displacement Pattern** page shows how the load is defined. This page specifies a uniform displacement in the H1 direction over the structure height

(from the ground level at  $V = 0$  to the roof at  $V = 432$  inches). Loads are applied at the mass points, and each load is equal to the mass multiplied by the displacement at the mass. Hence, for this load case there is an H1 load at each mass, with a magnitude equal to the mass.

The other information required for the load case is the Control Information, as follows.

- (1) The No. of Load Steps is 50. This is a reasonable number in most cases.
- (2) The Maximum Allowable Drift is 0.08 (8%). This drift is imposed in 50 equal steps.
- (3) The Max. Events in any Step is 200. PERFORM uses an "event-to-event" solution strategy, in which the structure stiffness is modified at each nonlinear "event" (such as formation of a plastic hinge). For this example there should be only a few events in any load step. If something goes wrong in the analysis, causing a large number of events, the analysis will stop after 200 events.
- (4) The Limit State to Stop Analysis is the default. If any inelastic component is deformed past its X point deformation, the analysis will stop. Recall that the X point deformation for a component is usually a large value, beyond which there is no point in continuing the analysis.
- (5) The Reference Drift is the H1 drift at the mass point in the roof. This is used when push-over curves are plotted as base shear vs. drift.
- (6) For Initial Step to First Event, the "No" option has been chosen. This is a minor point.

Choose the **Controlled Drifts** tab. This shows that all eight drifts are used as "controlled" drifts. The note in the Controlled Drifts page explains how controlled drifts are used. One consequence of using all drifts as controlled drifts is that the reference drift at the end of the analysis is generally not equal to the specified maximum drift (8%). If there is no strength loss in the components, and if there are no P- $\Delta$  effects, the reference drift is usually rather smaller than the specified maximum drift. If there is strength loss, or if P- $\Delta$  effects are considered, the final value of the reference drift may exceed the specified maximum. For this reason it is usually a good idea to specify a generous maximum drift (but not an extremely large value such as 20% - that is a waste of computer time).

In the **Load Case Name** list, choose each load case in turn. There are two load cases based on uniform displacement patterns, two based on linear displacement patterns, and two based on load patterns. The load patterns were based on the masses (see the Load Patterns task). Hence, the loads correspond to a uniform displacement pattern, and the results for the last two load cases will be the same as those for the first two load cases.

For the load cases that are based on nodal load patterns, note that the load pattern Scale Factor is 1.0. This number is arbitrary. For a push-over load, only the distribution of the loads matters, not their magnitudes. The required load magnitude is determined by PERFORM-3D during the analysis. If the Scale Factor were, say, 10 rather than 1, it would not change the analysis results.

### 3.26.4 Dynamic Earthquake Load Cases

In the **Load Case Type** list, choose **Dynamic Earthquake**. There are four load cases of this type.

In this section, if you get a message saying that an earthquake record is not available, it means that the example records have not been set up as described in the installation instructions. You must exit PERFORM-3D and set up the example earthquake records.

In the **Load case Name** list, choose **NR Mulholland, 0.6g**. This is based on an earthquake record from the Northridge earthquake, scaled so that the peak ground acceleration in the E-W direction is 0.6g. The **Q1 Earthquake** frame shows that the E-W record is scaled by a 1.16 factor. For this load case the Q1 axis is the same as the structure H1 axis. The **Q2 Earthquake** frame shows that the N-S record is scaled by the same factor. This is along the structure H2 axis. There is no vertical earthquake (V earthquake should be omitted in most cases).

To show the earthquake record, press **Add/Review/Delete Earthquakes**. Use the REVIEW OR DELETE AN EARTHQUAKE RECORD frame. In the **Earthquake Group** list choose **Examples**. In the File Name list choose **NR-14145M-EW**. Then press **Review**. The graph shows the earthquake accelerogram. Note that there is about 10 seconds of strong shaking. Press **Close**, then press **Return to Earthquake Load Case**.

The **Control Information for Dynamic Analysis** frame shows the control parameters for the analysis. These are self-explanatory.

In the **Load case Name** list, choose **NR Mulholland, 0.6g, 90 degrees**. This is based on the same earthquake records as the preceding load case, but the **Angle from H1 axis** is 90 degrees, which means that the E-W record is applied along the structure H2 axis and the N-S record is applied along the negative direction of the H1 axis.

The last two load cases are based on the 1940 El Centro record, scaled by a factor of 1.92 to give a peak acceleration of 0.6g in the N-S direction.

The **Total Time** (earthquake duration) is 15 seconds for all four load cases. For other earthquakes it may be necessary to use a longer duration, or it may be possible to use a shorter one.

## **3.27 Analysis Series**

---

### **3.27.1 General**

Press the toolbar button for the **Run analyses** task. This brings up the **ANALYSIS SERIES** form.

In each Analysis Series, some of the structure properties can be changed, as follows.

- (1) P-  $\Delta$  effects can be included or ignored.
- (2) The mass patterns can be combined in different ways.
- (3) The viscous damping assumptions can be changed.
- (4) Component strengths and stiffness can be changed, using Upper/Lower Bounds.
- (5) Brittle strength loss in inelastic components can be ignored, without changing the component properties.

The example structure has two Analysis Series. However, analyses have been run in only one of these series (you can run analyses in the other if you wish).

### 3.27.2 Series 1

In the **Series Name** list, choose **Series 1**. Then choose the **Basic + Masses** tab. This shows that the type of load sequence is Standard (this is usually the case), P-  $\Delta$  effects are to be included, 6 mode shapes are to be calculated, and the mass consists of 1.0 times the DL mass plus 0.25 times the LL mass. Also, the event overshoot factor is 1. This will usually be the case for smaller structures. Larger factors can be used for larger structures.

Choose the **Modal Damping** tab. This page shows that there is 3% damping in all modes (i.e., in the 6 modes that are calculated).

Choose the **Rayleigh Damping** tab. This page shows that there is 0.01% Beta-K damping (i.e., a very small amount) at a period  $T/T1 = 0.05$ . Press **Draw Graph** to show how the damping ratio varies with period. Most of the damping is provided by the modal damping. The Rayleigh damping provides a small amount of additional damping to ensure that high frequency vibrations have some damping. This is the suggested procedure for most cases. Press **Close Graph**.

Choose the **U/L Bounds** tab. This page shows how upper/lower bounds can be used. This is for advanced users. Since none of the components had upper/lower bound properties, this feature can not be used for the example structure.

Choose the **Quick'n'Dirty** tab. This page allows you to ignore strength loss for preliminary analyses, without changing the component properties. Since none of the components has strength loss, this feature has no effect for the example structure.

### 3.27.3 Series 2

In the **Series Name** list, choose **Series 2**. This series has different properties.

### 3.27.4 Run Analyses

In the **Series Name** list, choose **Series 1** again. The **Number of analyses** for this series is 9. That is, 9 analyses have already been run in this series. Press **OK** to go to the **SET UP AND RUN ANALYSES** form.

### 3.28 Run Analyses

---

The SET UP AND RUN ANALYSES form allows you to set up and run analyses in the current Analysis Series. The **Previous Analyses in This Series** list shows that 9 analyses have been run in Series 1, namely a Gravity analysis, 4 Static Push-Over Analyses, and 4 Dynamic Earthquake analyses. In addition, the mode shapes and periods have been calculated.

To run more analyses, you could choose a **Load Case Type**, a **Load Case Name** and a **Preceding Analysis** for more analyses, and add analyses to the ANALYSIS LIST. You could then press **GO** to run the analyses. If you would like to do this, it is better to press **Don't Go**, to return to the ANALYSIS SERIES form, and choose Series 2.

The following sections consider how to examine the results for the completed analyses in **Series 1**.

### 3.29 Mode Shapes

---

Press the toolbar button for the **Modal Analysis Results** task. For this example use only the **Modes** page. The other pages are for response spectrum analyses.

In the **Mode number** list choose Mode 1 and press **Plot**. This plot the mode shape. Press **Animate** to animate the mode shape. The **Mode properties** frame shows that the mode period is 1.35 seconds, and that this is almost entirely a mode in the H2 direction. Choose Modes 2, 3 etc. to show the mode properties.

### 3.30 Energy

---

Press the toolbar button for the **Energy balance** task. Choose a load case in the Case list above the graphics panel. Only dynamic analyses are listed. In the **Structure** page, press **Plot** to show the dissipated energies.

Choose the **Element Groups** tab. Choose the **Inelastic** option, and the **Perimeter girders** element group, then press **Plot**. The graph shows

the proportion of the total inelastic dissipated energy that is contributed by the chosen element group. Cycle through the other element groups to see their contributions.

If there is significant Beta-K damping, the **Beta-K viscous damping** option can be chosen to show the proportion of the total Beta-K viscous dissipated energy that is contributed by each element group. For the Series 1 analysis series there is only minimal Beta-K damping, so there is no reason to do this.

### 3.31 *Limit State Groups*

---

In the example structure there are 13 limit states (9 deformation, 2 strength and 2 drift). In a large structure there are likely to be many more than this. Limit state groups allow you to organize the limit states into smaller, related groups.

Press the toolbar button for the **Set up limit state groups** task. There are 5 limit state groups. Groups can be added, deleted and modified at any time.

In the **Names** list, choose a limit state group. The table shows the limit states in the group. For example, the All Immediate Occupancy group contains three IO deformation limit states and two strength limit states. Groups can be added, deleted and modified at any time.

### 3.32 *Deflected Shapes*

---

Press the toolbar button for the **Deflected shapes** task.

In the **Case** list above the graphics panel choose **H1 push, uniform displ.** A "thumbnail" plot of the push-over curve is shown in the data panel. In the **Plot Scale and Type** page and the **Plot Type for Limit State Usage Ratios** frame, choose the **Entire elements are colored** option.

Next choose the **Limit States** page the **Group** option for **Limit States to be Considered**, and the **All Life Safety** group. In the **Color Group for Usage** ratios frame, choose the **2** option. Then press **Plot**.

The initial plot is at zero drift, as shown by the yellow line in the thumbnail plot. In the **Animate if desired** section, press the **Animate forwards** button. The deflected shape is animated, and as the usage ratios for the elements increase, the elements change color.

When the yellow line reaches the end of the plot, the animation ends. Choose the **Plot Scale and Type** tab again, and the **Components are colored** option. The plastic hinges at the ends of the beam and column elements are shown, with colors representing the usage ratios. In the frames list at the right end of the toolbar, cycle through the frames that have been set up. For each frame, only the elements in the frame are shown, which simplifies the plot.

You may wish to experiment with this task, changing the type of plot, the limit states, the color scheme, the position of the yellow line in the thumbnail plot (click in the plot), etc.

### **3.33 Time Histories**

---

Press the toolbar button for the **Time histories** task. You can plot time (response) histories for node, element, drift and structure section results.

For example, choose the **Node** page, and choose a dynamic earthquake load case from the **Case** list above the graphics panel. Choose **Absolute acceleration** for the Result type and **H1** translation for the direction. Then select a node in the graphics panel, say a node at the roof, and press **Plot**. The time history of absolute acceleration is plotted. Then press Plot Spectrum, to plot the "floor" response spectrum for the selected node. If you select a supported node, at the base of the structure, the time history of absolute acceleration is the earthquake ground acceleration. This can be a good way to check that the correct ground motion has been applied.

You may wish to experiment with this task, changing the type of result. You can plot time histories for element results of a variety of types. If you plot Structure Section results you can see how the base shear varies with time (for a dynamic analysis) or drift (for a push-over analysis).



### 3.34 *Hysteresis Loops*

---

Press the toolbar button for the **Hysteresis loops** task. In the element group list in the data panel choose **Connection Panel Zones**. In the **Case** list above the graphics panel choose the **NR Mulholland, 0.6g** load case. In the graphics panel select the lowest panel zone element in the second column. Press **Plot**.

This draws the hysteresis loop for the panel zone element. In the thumbnail plot of the response the green line and red lines are at the start and finish of the response. Left click in the plot to move the green (start) line, and right click to move the red (end) line. Press Plot to draw the hysteresis loops for the range between the two lines. Use the green and red back-forward buttons to change the loops one step at a time.

You may wish to experiment with this task, changing the element group and the load case.

### 3.35 *Moment and Shear Diagrams, and Detailed Deflections*

---

Press the toolbar button for the **Moment and shear diagrams** task. In the data panel choose the **Single Element** tab, and in the element group list **Perimeter girders**. In the **Case** list above the graphics panel choose the **NR Mulholland, 0.6g** load case. For the result type choose **Moment and shear diagrams, time histories**. In the graphics panel select the lowest girder in the first bay in the H1 direction. Press **Plot**.

This plots the bending moment and shear force diagrams for the element. In the thumbnail plot the yellow line is at the start of the analysis, so these are the diagrams at the end of the gravity load case. In the **Animate if desired** buttons, move the slider to **Faster**, and press the **Animate forwards** button. This plots time histories of the moment and shear diagrams.

Click in the thumbnail plot to move the yellow line back near the start of the analysis. Choose the **Deflected shape** option, and press **Plot**. This plots the deflected shape for the element. In the **Animate if desired** buttons, move the slider to **Faster**, and press the **Animate forwards** button. This plots the time history of the element deflection.

Note that there is a rotation discontinuity at the right end of the element, indicating that a plastic hinge has formed.

Choose the **Line of elems** tab. In the element group list choose **Perimeter girders**, and choose the **Moment and shear diagrams, time history** option. In the graphics panel select the five girders in the lowest floor of the nearest frame (i.e., select a line of elements). Move the yellow line in the thumbnail plot to near the beginning of the plot. Press **Plot**, then **Animate forwards**. This plots the time history of moment and shear for the line of elements.

You may wish to experiment with this task, changing the element group, the load case and the selected elements. For a line of columns the moment and shear diagrams are plotted on a horizontal axis.

The Section Group page is not used for this example structure.

### ***3.36 Behavior vs. Performance Assessment***

---

The results processing tasks so far have considered mainly behavior rather than performance assessment (the deflected shape plots with colored elements are an exception). These tasks are useful for checking behavior, but generally do not give direct results for performance assessment. The remaining tasks are specifically for performance assessment.

### ***3.37 Push-Over Plot***

---

There are two tasks for push-over plots, namely **General push-over plot** and **Target displacement push-over plot**. Only the target displacement plot is considered in this quick tour. The general plot is similar, but it covers more push-over methods.

Press the toolbar button for the **Target displacement push-over plot** task. In the **Case** list above the graphics panel choose the **H1 push, uniform displ** load case.

In the **Step 1** page, an elastic response spectrum has been specified. For the **Framing Type** choose **Type 2**, for the **Performance Level** choose **LS**, and for the **Curve Type** choose **Base Shear Coefficient vs.**

**Reference Drift.** Then press **Check**. This draws the push-over curve. Press **OK** to go to Step 2.

In the **Step 2** page, for **Limit Points to be Plotted** choose the **Group** option and the **All Life Safety** group. There will be two short red lines on the push-over curve. Click on the first of these lines. It changes to blue, and the corresponding limit state is shown in the data panel in the **Limit State** box (Perimeter girder rotation, LS). The line marks the "limit point" where the usage ratio for this limit state reaches 1.0. This is the most critical limit state, since its limit point is reached first. Click on the second red line to show the corresponding limit state (Perimeter column rotation, LS). The limit points for the other limit states in the group are beyond the limit of the push-over curve (i.e., they are less critical). Press **OK** to continue to Step 3.

In the **Step 3** page, keep the default choices. In the graphics panel, move the cursor to a drift of 0.03 (i.e., 3.00E-02 on the horizontal scale), click and hold the left mouse button, drag the mouse to change the light blue line until (a) the small circle is on the push-over curve and (b) the area  $A_b/A_c$  is close to 1.0 (say between 0.998 and 1.002), then release. This may require some practice. The 0.03 drift is your estimate for the target drift. The blue line is a bilinear approximation for the push-over curve at the drift estimate, using the FEM 356 method. Next press **Calculate**. A vertical blue line will be drawn on the graph, at a drift of 0.032. This is not quite close enough to the estimate, so press **Clear**, and repeat with a drift estimate of 0.032. The calculated value (the target drift) should be very close to the estimate.

The target drift is to the left of the first limit point. This means that at the target drift, the usage ratios are smaller than 1.0 for all limit states in the chosen group. Hence, the performance is satisfactory.

You may wish to experiment with this task, changing the load case, the limit states, etc.

You may also like to try a feature that makes it much easier to construct the bilinear approximation. In **Step 3**, instead of left click and drag, just right click at the estimated drift. This draws the bilinear approximation without dragging. It may not always work, but it usually does. It can be used for both general push-over and target displacement plots.

### 3.38 Usage Ratio Graphs

---

Ultimately, performance assessment depends on limit state usage ratios. Limit states were specified in the **Limit States** task, and limit state groups in the **Set up limit state groups** task.

Press the toolbar button for the **Usage ratio graphs** task. In the **Case** list above the graphics panel choose the **NR Mulholland, 0.6g** load case. In the data panel choose the **Group** option, and in the list of limit state groups choose **All life safety**. Then press **Plot**.

This plots a graph of usage ratio vs. time. The **All life safety** group contains a number of limit states, and the graph shows the largest usage ratio for all limit states in the group. If the maximum usage ratio does not exceed 1.0, the limit states in the group are satisfied. For this example the maximum usage ratio is smaller than 1.0, indicating that the structure satisfies the performance requirements for the life safety level.

In the graph, on the vertical line at the end of the analysis, there are a number of short red lines. These are the usage ratios for each of the limit states in the group. Click on any one of these lines. A graph of usage ratio vs. time will be shown for the corresponding limit state. A box near the bottom of the data panel identifies (in blue) the corresponding limit state.

In the **Case** list above the graphics panel choose the **H1 push, uniform displ** load case. Recall that the target drift for this load case is 0.032. In the **Specify max. reference drift** box in the data panel, enter .032. Then press **Plot**. Usage ratio graphs will be plotted, with the last drift close to 0.032. The short red lines show the usage ratios for the limit states at the target drift.

If you wish you can experiment with different load cases and different limit state groups.

The usage ratio graph for a limit state shows only the maximum D/C ratio for the elements and components covered by the limit state. To see the D/C ratios for individual elements, use the **Deflected shapes** task. A usage ratio graph also shows only the usage ratios for a single load

case. To combine the results over several load cases, see the **Combinations and envelopes** task in the next section.

### **3.39 Combinations and Envelopes**

---

When dynamic earthquake analyses are used to assess performance, it is usual to run analyses for several earthquakes, and to envelope or average the results. If three earthquakes are used, FEMA 356 requires that component D/C ratios be enveloped (i.e., for any component use the largest D/C ratio from the three earthquakes). If seven earthquakes are used, FEMA 356 allows the D/C ratios for each component to be averaged over the seven earthquakes. This can give substantially smaller limit state usage ratios than the enveloping method. The **Combinations and envelopes** task allows you to set up groups of analyses, and to use enveloping or averaging to get limit state usage ratios for performance assessment.

Press the toolbar button for the **Combinations and envelopes** task. In the **Combination name** list in the data panel, choose **NR and EC earthquakes**. This combination consists of four dynamic earthquake analyses, as shown in the ANALYSIS LIST.

In the **Structure** page, choose **Max-Max** for the **Combination Method** option. In the **Limit States to be Considered** frame, choose **Group** and **All life safety**. Then press **Plot**.

This creates a plot with a vertical red line at the left and four vertical blue lines labeled 1 through 4. The four blue lines correspond to the four analyses in the combination. The left scale is usage ratio, there is a horizontal line at usage ratio = 1.0. On the blue lines, each short horizontal line shows the usage ratio for a limit state. On the red line, each short horizontal line shows the usage ratio for the combination.

Click on any of the short red lines. A horizontal red line shows the usage ratio for the combination. The zigzag blue line connects the usage ratios for the separate analyses. Since the Combination Method is Max-Max, the usage ratio for the combination is the largest of the usage ratios for the analyses. Since the usage ratios for the combination are all smaller than 1.0, the performance requirements are satisfied.

In the **Structure** page, choose **Max-Mean** for the **Combination Method** option, then press **Plot**. The usage ratios for the combination are now obtained by averaging the D/C ratios at the component level, then calculating the limit state usage ratios. This is quite a complex calculation, and the result is not the same as averaging the limit state usage ratios (to get this average, use the Mean-Max option). For details of the combination procedures see the User Guide.

Using the Max-Mean or Mean-Max combination option, it is possible for the usage ratios for some analyses to be larger than 1.0, but for the combination usage ratio to be below 1.0.

### **3.40 Conclusion**

---

This chapter gives a quick tour of PERFORM-3D, covering most (but by no means all) of its features. If you have completed the tour, you will have an idea of the capabilities of PERFORM-3D, for modeling nonlinear behavior, running analyses, and assessing performance.

Nonlinear analysis is not easy. However, if you have a sound feeling for nonlinear structural behavior and an understanding of the performance assessment process, it need not be really difficult.

Please remember that PERFORM-3D is merely a tool to help with design. We hope that you find it useful.