Design guide for steel frames using advanced analysis program

Seung-Eock Kim a,*, Wai-Fah Chen b

a Department of Civil Engineering, Sejong University, Seoul 143-747, South Korea
b School of Civil Engineering, Purdue University, West Lafayette, IN 47907, USA

Received 18 August 1997; received in revised form 2 October 1997; accepted 2 October 1997

Abstract

The aim of this paper is to provide a design guide of the LRFD based advanced analysis method for practical frame design. To this end, the step-by-step analysis and design procedures for the practical advanced analysis method are presented. The proposed advanced analysis program and input data format are discussed. Case studies are drawn to show the detailed design procedures of the practical advanced analysis method. The case studies cover a two-bay frame and a four-story, five-bay frame. Member sizes determined by the advanced analysis procedures are compared with those determined by the LRFD procedure, and a good agreement is generally observed. The advanced methods achieve the LRFD design requirements without tedious separate member capacity checks, including \( K \)-factor calculation. The methods are practical and therefore recommended for general use. © 1998 Elsevier Science Ltd. All rights reserved.

Keywords: Advanced analysis; Analysis program; Frame design; Practical method; Steel frame

1. Introduction

In a recent paper [1], a practical advanced analysis method has been proposed for steel frame design. The method incorporates three models including an explicit imperfection modeling, an equivalent notional load modeling, and a further reduced tangent modulus modeling. The advanced analysis method can sufficiently capture the limit state strength and stability of a structural system and its individual members. As a result, the method can be used for practical frame design without tedious separate member capacity checks, including \( K \)-factor calculation.

Since the power of personal computers and engineering workstations is rapidly increasing, practical use of the advanced method is currently becoming feasible. In this paper, the advanced analysis program is briefly introduced, and the design procedures using the program are presented. Case studies are drawn to show the detailed application procedures of the practical advanced analysis method. The case studies include a two-bay frame and a four-story, five-bay frame.

Since the present study is limited to two-dimensional braced and unbraced steel frames, the spatial behavior of frames is not considered and the lateral torsional buckling of members is assumed to be prevented by adequate lateral braces. A compact W-section is assumed so that sections can develop full plastic moment capacity without local buckling. The frames are subjected to static loads, rather than earthquake or cyclic loads.

2. Practical advanced analysis

The important attributes which affect the behavior of steel framed structures may be grouped into two categories: geometric and material nonlinearities. The geometric nonlinearity includes second-order effects associated with \( P-\delta \) and \( P-\Delta \) effects and geometric imperfections. The material nonlinearity includes gradual yielding associated with the influence of residual stresses and flexure. The key considerations of the practical advanced analysis are discussed in what follows, and summarized in Table 1 compared with the conventional AISC-LRFD method. The proposed method has been verified for a wide range of frames [2].
Table 1
Key considerations of LRFD and proposed advanced analysis method

<table>
<thead>
<tr>
<th>Key considerations</th>
<th>LRFD</th>
<th>Proposed method</th>
</tr>
</thead>
<tbody>
<tr>
<td>Second-order effects</td>
<td>Column curve B₁, B₂ factor</td>
<td>Stability function</td>
</tr>
<tr>
<td>Geometric imperfection</td>
<td>Column curve</td>
<td>Explicit imperfection modeling method</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Equivalent notional load method</td>
</tr>
<tr>
<td></td>
<td></td>
<td>α = 0.002 for unbraced frame</td>
</tr>
<tr>
<td></td>
<td></td>
<td>α = 0.004 for braced frame</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Further reduced tangent modulus method</td>
</tr>
<tr>
<td></td>
<td></td>
<td>( E'_t = 0.85E_t )</td>
</tr>
<tr>
<td>Stiffness degradation associated with residual stresses</td>
<td>Column curve</td>
<td>CRC tangent modulus</td>
</tr>
<tr>
<td>Stiffness degradation associated with flexure</td>
<td>Column curve interaction equations</td>
<td>Parabolic degradation function</td>
</tr>
</tbody>
</table>

2.1. Geometric nonlinear effects

Stability functions are used to capture geometric nonlinear effects, since they can account for the effect of the axial force on the bending stiffness reduction of a member. The benefit of using stability functions is that it enables only one element to accurately predict the nonlinear effect of each framed member [2].

2.2. Stiffness degradation associated with residual stresses

The CRC tangent modulus is employed here to account for gradual yielding effects due to residual stresses along the length of members under axial loads between two plastic hinges. When this model incorporates appropriate geometrical imperfections, it may provide a very good comparison with the plastic zone solutions [2]. From Liew [3], the CRC \( E_t \) may be written as:

\[
E_t = \begin{cases} 
1.0E & \text{for } P \leq 0.5P_y \\
4 \frac{P}{P_y} E \left(1 - \frac{P}{P_y}\right) & \text{for } P > 0.5P_y 
\end{cases}
\]

where \( P_y \) = squash load and \( E \) = elastic modulus.

2.3. Stiffness degradation associated with flexure

A gradual stiffness degradation of a plastic hinge is required to represent the distributed plasticity effects associated with bending actions. Herein we shall introduce the hardening plastic hinge model to represent the gradual transition from elastic stiffness to zero stiffness associated with a fully developed plastic hinge. When the hardening plastic hinges are present at both ends of an element, the incremental force–displacement relationship may be expressed as [2,3]:

\[
\begin{bmatrix}
M_A \\
M_B \\
\rho
\end{bmatrix} = \begin{bmatrix}
\frac{E}{I} \\
0 \\
0
\end{bmatrix} 
\cdot \begin{bmatrix}
\eta_A \left[S_1 - \frac{S_2}{S_1}(1 - \eta_B)\right] \\
\eta_A \eta_B S_2 \\
0
\end{bmatrix} 
\cdot \begin{bmatrix}
0 \\
\eta_B \left[S_1 - \frac{S_2}{S_1}(1 - \eta_A)\right] \\
0
\end{bmatrix} 
\cdot \begin{bmatrix}
0 \\
0 \\
A/I
\end{bmatrix}
\]

where

\[
M_A, M_B, \dot{P} = \text{incremental end moments and axial force, respectively}
\]

\[
S_1, S_2 = \text{stability functions}
\]

\[
E_i = \text{tangent modulus}
\]

\[
I = \text{moment of inertia of cross-section}
\]

\[
L = \text{length of element}
\]

\[
A = \text{area of cross-section}
\]

\[
\eta_A, \eta_B = \text{scalar parameters for gradual inelastic stiffness reduction}
\]

\[
\theta_A, \theta_B = \text{incremental rotations at element ends}
\]

\[
\dot{e} = \text{incremental axial deformation}
\]

The parameter η represents a gradual stiffness reduction associated with flexure at sections. The partial plastification at cross-sections at the end of elements is denoted by \( 0 < \eta < 1 \). The η may be assumed to vary according to the simple parabolic expression as:

\[\eta = 4\alpha \left(1 - \alpha\right)\]  

where \( \alpha \) is the force–state parameter obtained from the limit state surface corresponding to the element end as:
Fig. 1. Smooth stiffness degradation for a work-hardening plastic hinge based on LRFD sectional strength curve.

\[ \alpha = \frac{P}{P_y} + \frac{8}{9} \frac{M}{M_p} \quad \text{for} \quad \frac{P}{P_y} \geq 0.2 \tag{4a} \]

\[ \alpha = \frac{P}{2P_y} + \frac{M}{M_p} \quad \text{for} \quad \frac{P}{P_y} < 0.2 \tag{4b} \]

where \( P, M = \) second-order axial force and bending moment at the cross-section \( M_p = \) plastic moment capacity.

In Fig. 1, the term \( \alpha \) of 1.0 represents the plastic strength surface and \( \alpha \) of 0.5 is assumed to be the initial yield surface which is assumed to have the same shape as the LRFD plastic strength surface. As the \( \alpha \) value varies from 0 to 0.5, the element end remains in the elastic state. When the \( \alpha \) moves from 0.5 to 1.0, the element stiffness changes with a parabolic degradation shape shown in Fig. 2.

2.4. Geometric imperfection

Geometric imperfection models, combined with the CRC tangent modulus model, are discussed in what follows. They are an explicit imperfection modeling, a notional load modeling, and a reduced tangent modulus modeling.

2.4.1. Explicit imperfection modeling

The AISC code of standard practice [4] limits an erection out-of-plumbness equal to \( L_c /500 \) in any story. This imperfection value is conservative in taller frames since the maximum permitted erection tolerance of 50 mm (2 in) is much less than the accumulated geometric imperfection calculated by \( 1/500 \) times building height. In this study, however, \( L_c /500 \) is used for geometric imperfection without any modifications. This is because the system strength is often governed by a weak story which has an out-of-plumbness equal to \( L_c /500 \) [5].

The frame out-of-plumbness may be used for geometric imperfections for unbraced frames but not for braced frames. This is because the \( P_\Delta \) effect caused by out-of-plumbness is diminished by braces. As a result, the member out-of-straightness instead of the out-of-plumbness should be used to account for geometric imperfections for braced frames. The AISC code recommends a maximum fabrication tolerance of \( L_c /1000 \) for a member out-of-straightness. In this study, a geometric imperfection of \( L_c /1000 \) is adopted by calibration with the plastic zone solutions. The out-of-straightness may be assumed to vary sinusoidally with a maximum in-plane deflection of \( L_c /1000 \) at the mid-height. Ideally, many elements are necessary in order to model the sinusoidal out-of-straightness of a beam-column member. It has been found that two elements with a maximum deflection at the mid-height of a member are practically adequate for reflecting the imperfection effects [2].

2.4.2. Equivalent notional load modeling

The geometric imperfections of a frame may be replaced by equivalent notional lateral loads that are expressed as a fraction of the gravity loads acting on a story. In this study, the proposed equivalent notional load for practical use is 0.002 \( \Sigma P_u \), where \( \Sigma P_u \) is the total gravity load in a story. The notional load should be applied laterally at the top of each story. For braced frames, the notional loads should be applied at mid-height of a column since the ends of the column are braced. In this study, appropriate notional load factor equal to 0.004 is adopted. It may be observed this value is equivalent to the geometric imperfection of \( L_c /1000 \).

2.4.3. Further reduced tangent modulus modeling

The idea of using the further reduced tangent modulus method is to further reduce the tangent modulus \( E_t \) to
account for the geometric imperfections. The reduced tangent modulus method is capable of eliminating the somewhat tedious work of explicit imperfection modeling or notional load input. Fig. 3 shows the member stiffness reduction curve of the further reduced tangent modulus $E_t'$. The appropriate reduction factor of 0.85 to $E_t$ is determined by the calibration with the plastic zone solutions [2]. The similar reduction factor of 0.85 may be used for braced as well as unbraced frames.

3. Program and input data format

This section discusses the proposed analysis program for a two-dimensional steel frame design. This program is developed for practical design use from the refined plastic hinge program [3]. The program is divided into three FORTRAN programs including DATAGEN, INPUT, and PAAP. The first program, DATAGEN, reads an input file P.DAT and produces the generated data file INFIL. The second program, INPUT, rearranges INFIL into three data files including DATA0, DATA1, and DATA2. The third program, PAAP (Practical Advanced Analysis Program), provides the output files P.OUT1 and P.OUT2. The P.OUT1 file contains an echo of the information from the input data file P.DAT. The P.OUT2 file contains detailed information on load–displacement relationship and member forces of the structures. The schematic diagram in Fig. 4 shows the operation procedures of the program. The programs may be executed by issuing the batch file command ‘RUN’.

The program has been tested in two computer environments. The first is in the IBM 486 or equivalent personal computer system using a Microsoft FORTRAN 77 compiler v. 1.00 and Lahey FORTRAN 77 compiler v. 5.01. The second is in a Sun 5 machine using a Sun FORTRAN 77 compiler. The program sizes of DATAGEN, INPUT, and PAAP are 8 KB, 9 KB, and 84 KB, respectively. The total size of the three programs is as small as 111 KB (= 0.111 MB), and so the 3.5 inch high density diskette (1.44 MB) can accommodate the three programs and several example problems. The program can generate the output file P.OUT in a reasonable time period. The run time in IBM 486 PC with a memory of 640 K to get P.OUT for the four-story, five-bay frame shown in Figs. 10 and 11, is 4 min, 10 sec and 2 min, 30 sec in real time rather than CPU time by using Microsoft FORTRAN and Lahey FORTRAN, respectively. In the Sun 5, the run time varies approximately 2–3 min, depending on the degree of occupancy by users.

The input format in the use of the proposed program is presented in what follows. Except considerations of geometric imperfections and incremental loads, the input data format of the proposed program is basically the same as that of a usual linear elastic analysis program. The input data consist of 13 data sets including five control data, three section property data, three element data, one boundary data, and one load data set. The coordinate systems both in the input data and in the output are based on global coordinates rather than on local coordinates. Since the unit system is not specified by the program, users may choose the unit in consistency at their con-
4. Analysis and design procedures

4.1. Step 1: live load reduction

The live load reduction is based on the ASCE 7-95 [6] as:

\[ L = \left( 0.25 + \frac{15}{\sqrt{A_i}} \right) L_o \geq \alpha L_o \]  \hspace{1cm} (5)

where \( L = \) reduced design live load, \( A_i = \) member influence area in square feet (\( A_i \geq 36 \text{ m}^2 = 400 \text{ ft}^2 \)), \( L_o = \) unreduced design live load, and \( \alpha = 0.5 \) for members supporting one floor, \( \alpha = 0.4 \), otherwise.

It is important to properly carry out the application of the live load reduction in analyzing a structural system. This is because the influence area for each beam and column is generally different, and different influence area results in a different reduction factor. In the present study, the live load reduction procedures follow the work of Ziemian and McGuire [7]. The method is based on the use of ‘compensating forces’ calculated by:

1. applying beam live load reduction factor to the column connected beams;
2. applying column live load reduction factor to columns; and
3. determining compensating forces due to different reduction factors between columns and beams at the beam-to-column intersections.

The compensating forces are generally directed upward since columns typically have a larger influence area and a larger reduction factor than beams.

4.2. Step 2: load combinations

The load combinations in the proposed methods are based on the LRFD load combinations [4]. The member sizes of structures are determined from an appropriate combination of factored loads.

\begin{align*}
1.4D \\
1.2D + 1.6(L_o \text{ or } S \text{ or } R) \\
1.2D + 1.3W + 0.5L_o \text{ or } S \text{ or } R \\
1.2D \pm 1.0E + 0.5L + 0.2S \\
0.9D \pm (1.3W \text{ or } 1.0E)
\end{align*}

where

\begin{align*}
D &= \text{dead load} \\
L &= \text{live load} \\
L_o &= \text{roof live load} \\
W &= \text{wind load} \\
S &= \text{snow load} \\
E &= \text{earthquake load} \\
R &= \text{rain load}.
\end{align*}

4.3. Step 3: preliminary member sizing

The preliminary member sizing is intrinsically dependent on engineers’ experiences, the rule of thumb, or some simplified analysis. For example, beam members are usually selected assuming that beams are simply supported and subjected to gravity loads only. For the preliminary sizing of column members, the overall drift requirements should be a good guideline to determine preliminary member sizes rather than the tedious strength checks of individual columns.

4.4. Step 4: modeling of element

Columns in braced and unbraced frames may be modeled by two and one element, respectively. Nodal points should be provided where concentrated loads are applied. When a beam is subjected to uniform loads, it may be modeled by two elements. The uniform loads should be converted into these concentrated loads. The adequacy of the number of element recommended herein has been verified in Ref. [2].

4.5. Step 5: modeling of geometric imperfection

Geometric imperfections can be accounted for by one of the three modelings including the explicit imperfection modeling, the equivalent notional load modeling, and the further reduced tangent modulus modeling. The explicit imperfection modeling and the equivalent notional load modeling require the user to input either imperfections or notional loads, respectively. On the contrary, the further reduced tangent modulus modeling automatically reflects the imperfection effects implicitly in the proposed computer program (PAAP) [2].

4.6. Step 6: determination of incremental loads

The input of incremental loads rather than total loads is required so that advanced analysis can trace nonlinear
Table 2
Input data format for the program 'PAAP'

<table>
<thead>
<tr>
<th>Data set</th>
<th>Column</th>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Title</td>
<td>A70</td>
<td></td>
<td>Job title and general comments</td>
</tr>
<tr>
<td>2. Design control</td>
<td>1–5</td>
<td>IGEOM</td>
<td>Geometric imperfection method</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0: No geometric imperfection (default)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1: Explicit imperfection modeling</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2: Equivalent notional load</td>
</tr>
<tr>
<td></td>
<td>6–10</td>
<td>ILRFD</td>
<td>Further reduced tangent modulus</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>0: No reduction factors considered (default)</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1: Reduction factors considered</td>
</tr>
<tr>
<td>3. Job control</td>
<td>1–5</td>
<td>NNODE</td>
<td>Total number of nodal points of the structure</td>
</tr>
<tr>
<td></td>
<td>6–10</td>
<td>NBOUND</td>
<td>Total number of supports</td>
</tr>
<tr>
<td></td>
<td>11–15</td>
<td>NINCRE</td>
<td>Allowable number of load increments (default = 100); at least two or three times larger than the scaling number</td>
</tr>
<tr>
<td>4. Total number of element type</td>
<td>1–5</td>
<td>NCTYP</td>
<td>Number of connection types (1–10)</td>
</tr>
<tr>
<td></td>
<td>6–10</td>
<td>NFTYP</td>
<td>Number of frame types (1–20)</td>
</tr>
<tr>
<td></td>
<td>11–15</td>
<td>NTYP</td>
<td>Number of truss types (1–10)</td>
</tr>
<tr>
<td>5. Total number of element</td>
<td>1–5</td>
<td>NUMCNT</td>
<td>Number of connection elements (1–50)</td>
</tr>
<tr>
<td></td>
<td>6–10</td>
<td>NUMFRM</td>
<td>Number of frame elements (1–200)</td>
</tr>
<tr>
<td></td>
<td>11–15</td>
<td>NUMTRS</td>
<td>Number of truss elements (1–50)</td>
</tr>
<tr>
<td>6. Connection property</td>
<td>1–5</td>
<td>ICTYP</td>
<td>Connection type number</td>
</tr>
<tr>
<td></td>
<td>6–15</td>
<td>M</td>
<td>Ultimate moment capacity of connection</td>
</tr>
<tr>
<td></td>
<td>16–25</td>
<td>R</td>
<td>Initial stiffness of connection</td>
</tr>
<tr>
<td></td>
<td>26–35</td>
<td>n</td>
<td>Shape parameter of connection</td>
</tr>
<tr>
<td>7. Frame element property</td>
<td>1–5</td>
<td>IFTYP</td>
<td>Frame type number</td>
</tr>
<tr>
<td></td>
<td>6–15</td>
<td>A</td>
<td>Cross-section area</td>
</tr>
<tr>
<td></td>
<td>15–25</td>
<td>I</td>
<td>Moment of inertia</td>
</tr>
<tr>
<td></td>
<td>25–35</td>
<td>E</td>
<td>Modulus of elasticity</td>
</tr>
<tr>
<td></td>
<td>46–55</td>
<td>FY</td>
<td>Yield stress</td>
</tr>
<tr>
<td></td>
<td>55–60</td>
<td>IFCOL</td>
<td>Identification of column member, IFCOL = 1 for column (default = 0)</td>
</tr>
<tr>
<td>8. Truss element property</td>
<td>1–5</td>
<td>ITYP</td>
<td>Truss type number</td>
</tr>
<tr>
<td></td>
<td>6–15</td>
<td>A</td>
<td>Cross-section area</td>
</tr>
<tr>
<td></td>
<td>15–25</td>
<td>I</td>
<td>Moment of inertia</td>
</tr>
<tr>
<td></td>
<td>25–35</td>
<td>E</td>
<td>Modulus of elasticity</td>
</tr>
<tr>
<td></td>
<td>36–45</td>
<td>FY</td>
<td>Yield stress</td>
</tr>
<tr>
<td></td>
<td>46–50</td>
<td>ITCOL</td>
<td>Identification of column member, ITCOL = 1 for column (default = 0)</td>
</tr>
<tr>
<td>9. Connection element data</td>
<td>1–5</td>
<td>LCNT</td>
<td>Connection element number</td>
</tr>
<tr>
<td></td>
<td>6–10</td>
<td>IFMCN</td>
<td>Frame element number containing the connection</td>
</tr>
<tr>
<td></td>
<td>11–15</td>
<td>IEND</td>
<td>Identification of element ends containing the connection</td>
</tr>
<tr>
<td></td>
<td>16–20</td>
<td>JDCNT</td>
<td>Connection type number</td>
</tr>
<tr>
<td></td>
<td>21–25</td>
<td>NOSMCN</td>
<td>Number of same elements for automatic generation (default = 1)</td>
</tr>
<tr>
<td></td>
<td>26–30</td>
<td>NELINC</td>
<td>Element number (IFMCN) increment of automatically generated elements (default = 1)</td>
</tr>
<tr>
<td>10. Frame element data</td>
<td>1–5</td>
<td>LFRM</td>
<td>Frame element number</td>
</tr>
<tr>
<td></td>
<td>6–15</td>
<td>FXO</td>
<td>Horizontal projected length; positive for i-j direction in global x-direction</td>
</tr>
<tr>
<td></td>
<td>16–25</td>
<td>FYO</td>
<td>Vertical projected length; positive for i-j direction in global y-direction</td>
</tr>
<tr>
<td></td>
<td>26–30</td>
<td>JFRM</td>
<td>Frame type number</td>
</tr>
<tr>
<td></td>
<td>31–35</td>
<td>IFNODE</td>
<td>Number of node i</td>
</tr>
<tr>
<td></td>
<td>36–40</td>
<td>JFNODE</td>
<td>Number of node j</td>
</tr>
<tr>
<td></td>
<td>41–45</td>
<td>NOSMF</td>
<td>Number of same elements for automatic generation (default = 1)</td>
</tr>
<tr>
<td></td>
<td>46–50</td>
<td>NODINC</td>
<td>Node number increment of automatically generated elements (default = 1)</td>
</tr>
<tr>
<td>11. Truss element data</td>
<td>1–5</td>
<td>LTRS</td>
<td>Truss element number</td>
</tr>
<tr>
<td></td>
<td>6–15</td>
<td>TXO</td>
<td>Horizontal projected length; positive for i-j direction in global x-direction</td>
</tr>
<tr>
<td></td>
<td>16–25</td>
<td>TYO</td>
<td>Vertical projected length; positive for i-j direction in global y-direction</td>
</tr>
<tr>
<td></td>
<td>26–30</td>
<td>JTRES</td>
<td>Truss type number</td>
</tr>
<tr>
<td></td>
<td>31–35</td>
<td>ITNODE</td>
<td>Number of node i</td>
</tr>
<tr>
<td></td>
<td>36–40</td>
<td>JTNODE</td>
<td>Number of node j</td>
</tr>
<tr>
<td></td>
<td>51–55</td>
<td>NOSME</td>
<td>Number of same elements for automatic generation (default = 1)</td>
</tr>
<tr>
<td></td>
<td>56–60</td>
<td>NODINC</td>
<td>Node number increment of automatically generated elements (default = 1)</td>
</tr>
</tbody>
</table>

Continued over
Table 2
Continued

<table>
<thead>
<tr>
<th>Data set</th>
<th>Column</th>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>12. Boundary condition</td>
<td>1–5</td>
<td>NODE</td>
<td>Node number of support</td>
</tr>
<tr>
<td></td>
<td>6–10</td>
<td>XFIX</td>
<td>XFIX = 1 for restrained in global x-direction</td>
</tr>
<tr>
<td></td>
<td>11–15</td>
<td>YFIX</td>
<td>YFIX = 1 for restrained in global y-direction</td>
</tr>
<tr>
<td></td>
<td>16–20</td>
<td>RFIX</td>
<td>RFIX = 1 for restrained in rotation</td>
</tr>
<tr>
<td></td>
<td>21–25</td>
<td>NOSMBD</td>
<td>Number of same boundary conditions for automatic generation (default = 1)</td>
</tr>
<tr>
<td></td>
<td>26–30</td>
<td>NODINC</td>
<td>Node number increment of automatically generated supports (default = 1)</td>
</tr>
<tr>
<td>13. Incremental loads</td>
<td>1–5</td>
<td>NODE</td>
<td>Node number where a load applied</td>
</tr>
<tr>
<td></td>
<td>6–15*</td>
<td>XLOAD</td>
<td>Incremental load in global x-direction</td>
</tr>
<tr>
<td></td>
<td>16–25*</td>
<td>YLOAD</td>
<td>Incremental load in global y-direction</td>
</tr>
<tr>
<td></td>
<td>26–35*</td>
<td>RLOAD</td>
<td>Incremental moment in global u-direction</td>
</tr>
<tr>
<td></td>
<td>36–40</td>
<td>NOSMLD</td>
<td>Number of same loads for automatic generation (default = 1)</td>
</tr>
<tr>
<td></td>
<td>41–45</td>
<td>NODINC</td>
<td>Node number increment of automatically generated loads (default = 1)</td>
</tr>
</tbody>
</table>

*Indicates the real value (‘F’ or ‘E’ format) to input, otherwise integer value (‘I’ format).

load–displacement behavior. The incremental loads can be determined by dividing the combined factored loads by the recommended scaling number of 10–50.

The simple incremental solution method is implemented here for practical use. The errors in using the simple incremental solution method can be minimized with an automatic scaling of the incremental loads, when changes in the element stiffness parameter exceed a predefined tolerance in the program.

4.7. Step 7: preparation of input data

Based on the input data format in Table 2, the input data file P.DAT can be made by users. The input data format is basically similar to that of the linear elastic analysis with proper considerations of geometric imperfections and incremental loads.

4.8. Step 8: program execution

The program execution can be done simply by typing in a batch file command ‘RUN’ on the screen. The program continues analysis for increased loads, and stops when it reaches its ultimate state.

4.9. Step 9: ultimate load carrying capacity check

The proposed methods are based on the limit state approach to strength design. The limit state format may be written as:

$$\sum \gamma_i Q_i \leq \phi R_n$$  

(6)

where \(\gamma_i\) = load factors, \(Q_i\) = nominal design loads, \(\phi\) = resistance factors and \(R_n\) = nominal resistances.

From the output file P.OUT, the ultimate load carrying capacity, i.e. the ultimate strength \(\phi R_n\) of a structure, can be obtained. The resistance factors \(\phi\) are built in the analysis program so that they are automatically included in the resulting load carrying capacity. The resistance factors are selected as 0.85 for axial strength and 0.9 for flexural strength corresponding to AISC-LRFD specification [4]. The adequacy of the structural system can be directly evaluated by comparing the resulting load carrying capacity \(\phi R_n\) with the applied factored loads \(\sum \gamma_i Q_i\) in Step 2. Separate member capacity checks are not necessary since the predicted ultimate load of a structural system will not violate individual member capacity encompassed by the AISC-LRFD equations [2].

4.10. Step 10: serviceability check

According to the ASCE Ad Hoc Committee report [8], the normally accepted range of overall drift limits for buildings is 1/750–1/250 times the building height \(H\) with a typical value of \(H/400\). The general limits on the interstory drift are 1/500–1/200 times the story height. Based on the studies by the Ad Hoc Committee [8] and Ellingwood [9], the deflection limits for girder and story are selected as:

1. floor girder deflection for service live load: \(L/360\),
2. roof girder deflection: \(L/240\),
3. lateral drift for service wind load: \(H/400\), and
4. interstory drift for service wind load: \(H/300\).

The serviceability of a structural system should be checked also to ensure the adequacy of the system and its member stiffness at service loads. The first-order displacements at service loads can be obtained by either using the proposed analysis program or a usual commercial program. The proposed analysis program is recommended since the input data prepared for the ultimate strength checks may also be used for the serviceability checks without major change. Using the proposed analysis program, the first-order displacements can be obtained by setting the number of load increments equal to 1 (NINCRE = 1), and then by multiplying the resulting displacements by the scaling number. This pro-
procedure is based on the fact that the structural system behaves linearly and elastically under a small incremental load. The adequacy of the system and its member stiffness can be evaluated by comparing the resulting displacements with the limited displacements.

4.11. Step 11: ductility check

Adequate inelastic rotation capacity is required for members in order to develop their full plastic moment capacity. The required rotation capacity may be achieved when members are adequately braced and their cross-sections are compact.

Compact sections are capable of developing the full plastic moment capacity \( M_p \) and sustaining large hinge rotation before the onset of local buckling. The compact section in the LRFD specification is defined as

1. Flange:

\[
\frac{b_f}{2t_f} \leq \frac{65}{\sqrt{F_y}} \tag{7}
\]

where \( b_f \) = width of flange, \( t_f \) = thickness of flange, and \( F_y \) = yield stress in ksi; and

2. Web:

\[
\frac{h}{t_w} \leq \frac{640}{\sqrt{F_y}} \left( 1 - \frac{2.75P_u}{\phi_y P_y} \right) \text{ for } \frac{P_u}{\phi_y P_y} \leq 0.125
\]

\[
\frac{h}{t_w} \leq \left[ \frac{191}{\sqrt{F_y}} \left( 2.33 - \frac{P_u}{\phi_y P_y} \right) \right] \geq \frac{253}{\sqrt{F_y}} \text{ for } \frac{P_u}{\phi_y P_y} > 0.125
\]

(8b)

where \( h \) = clear distance between flanges and \( t_w \) = thickness of web.

In addition to the compactness of section, the lateral unbraced length of a member is also a limiting factor for the development of the full plastic moment capacity of members. The LRFD specification provides the limit on spacing of braces for beam–columns as:

\[
L_{pd} \leq \frac{3600 + 2200 (M_1/M_2)r_y}{F_y} \tag{9}
\]

where

- \( L_{pd} \) = unbraced length of the compression flange when using plastic analysis
- \( r_y \) = radius of gyration about y-axis
- \( F_y \) = yield strength in ksi
- \( M_1, M_2 \) = smaller and larger end moment
- \( M_1/M_2 = \) positive in double curvature bending

When the yield stress is equal to 250 MPa (36 ksi), \( M_1/M_2 = -0.5 \), and the radius of gyration is assumed to be approximately 50mm (2 inches), the permissible unbraced length \( L \) results in 3.5 m (11.6 ft). Since the unbraced length of 3.5 m (11.6 ft) is within the range of a typical story height of 3.0–3.7 m (10–12 ft), lateral torsional buckling is usually not a governing factor for the ductility of beam–columns in typical building frames.

4.12. Step 12: adjustment of member sizes

If the conditions of Steps 9–11 are not satisfied, appropriate adjustments of member sizes should be made. Adjustment of member sizes can be carried out referring to the sequence of plastic hinge formation shown in the ‘P.OUT’. For illustration, if the resulting load carrying capacity of a structural system is less than the applied factored loads, the member containing the first plastic hinge should be first replaced with a stronger member. On the contrary, if the loading carrying capacity of a structural system exceeds the applied factored load significantly, members without plastic hinges may be replaced with lighter members. If lateral drift of a structural system does not meet the drift requirement, beam and column sizes should be increased for serviceability, or a braced structural system may be considered instead.

5. Design example 1: two-bay frame

Fig. 5 shows a fixed-supported two-bay portal frame subjected to gravity and lateral loads. Detailed procedures for selecting appropriate member sizes are presented in what follows. For comparison purposes, the LRFD member size is also presented. All members are assumed to be laterally braced, and wide flange sections of W16 and W14 of A36 steel are used for beams and columns, respectively.
5.1. Design by the proposed method

5.1.1. Step 1: live load reduction
The live load reduction is assumed to be made in the load combination in Fig. 5.

5.1.2. Step 2: load combinations
Herein, the critical load combination is assumed shown in Fig. 5.

5.1.3. Step 3: preliminary member sizing
Preliminary member sizes are selected as W16 × 77 and W14 × 53 for the beams and the columns, respectively.

5.1.4. Step 4: modeling of element
Each column is modeled as one element, and the beams are modeled as three and two elements for the left bay and right bay, respectively. The element and node numbering are shown in Figs. 6–8.

5.1.5. Step 5: modeling of geometric imperfection
In the explicit imperfection modeling method, the geometric imperfection of 0.012 m (0.04 ft) at the top of each column is calculated by 0.2% times the column height of 6 m (20 ft) as shown in Fig. 6. In the equivalent notional load method, the notional load of 1.36 kN (0.306 kips) results from 0.2% times the total gravity load of 681 kN (153 kips) and is added to the lateral load shown in Fig. 7. In the further reduced tangent modulus method, the program automatically accounts for geometric imperfection effects shown in Fig. 8. Users can choose one of the three modelings for generating their input data.

5.1.6. Step 6: determination of incremental loads
Herein, the incremental loads are determined as 8.896 kN (2 kips) by dividing the factored concentrated load of 227 kN (51 kips) by the scaling number of 25.5 shown in Figs. 6–8.

5.1.7. Step 7: input data
Based on the input data format in Table 2, three kinds of input data file P.DAT in Table 3 corresponding to three different geometric imperfection methods can be made by users, but only one input data among these three is required for analysis and design.

5.1.8. Step 8: program execution
Program execution is simply done by typing in ‘RUN’.

5.1.9. Step 9: ultimate load check
From the output file P.OUT, the ultimate load carrying capacities of the structure are obtained as 229 kN (51.4 kips) by the explicit imperfection modeling, and the equivalent notional load method, and 231 kN (52.1 kips) by the further reduced tangent modulus method. All three load carrying capacities are slightly greater than the applied factored load of 227 kN (51 kips).
Table 3
Input data for the two-bay frame

<table>
<thead>
<tr>
<th>Two-bay frame, explicit imperfection modeling</th>
<th>1</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 3 100</td>
<td>20.00 723.0 115.0 29 000.0 36.0</td>
<td></td>
</tr>
<tr>
<td>0 2 0</td>
<td>26.20 1300.0 175.0 29 000.0 36.0</td>
<td></td>
</tr>
<tr>
<td>0 8 0</td>
<td>0.48 240.0 1 1 4</td>
<td></td>
</tr>
<tr>
<td>1 20.00 723.0 115.0 29 000.0 36.0</td>
<td>26.20 1300.0 175.0 29 000.0 36.0</td>
<td></td>
</tr>
<tr>
<td>2 0.48 240.0 1 2 7</td>
<td>0.48 240.0 1 3 9</td>
<td></td>
</tr>
<tr>
<td>3 0.48 240.0 1 2 7</td>
<td>90.0 0.0 2 4 5</td>
<td></td>
</tr>
<tr>
<td>4 90.0 0.0 2 4 5</td>
<td>180.0 0.0 2 5 6</td>
<td></td>
</tr>
<tr>
<td>5 180.0 0.0 2 5 6</td>
<td>90.0 0.0 2 6 7</td>
<td></td>
</tr>
<tr>
<td>6 90.0 0.0 2 6 7</td>
<td>300.0 0.0 2 7 8 2</td>
<td></td>
</tr>
<tr>
<td>7 300.0 0.0 2 7 8 2</td>
<td>1 1 1 3</td>
<td></td>
</tr>
<tr>
<td>4 2.00</td>
<td>5 -2.00 2</td>
<td></td>
</tr>
<tr>
<td>8 -2.00</td>
<td>2</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Two-bay frame, equivalent notional load modeling</th>
<th>2</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 3 100</td>
<td>20.00 723.0 115.0 29 000.0 36.0</td>
<td></td>
</tr>
<tr>
<td>0 2 0</td>
<td>26.20 1300.0 175.0 29 000.0 36.0</td>
<td></td>
</tr>
<tr>
<td>0 8 0</td>
<td>0.00 240.0 1 3 9</td>
<td></td>
</tr>
<tr>
<td>1 0.00 240.0 1 1 4</td>
<td>0.00 240.0 1 2 7</td>
<td></td>
</tr>
<tr>
<td>2 0.00 240.0 1 2 7</td>
<td>90.0 0.0 2 4 5</td>
<td></td>
</tr>
<tr>
<td>3 0.00 240.0 1 3 9</td>
<td>180.0 0.0 2 5 6</td>
<td></td>
</tr>
<tr>
<td>4 90.0 0.0 2 5 6</td>
<td>90.0 0.0 2 6 7</td>
<td></td>
</tr>
<tr>
<td>5 180.0 0.0 2 6 7</td>
<td>300.0 0.0 2 7 8 2</td>
<td></td>
</tr>
<tr>
<td>6 90.0 0.0 2 7 8 2</td>
<td>1 1 1 3</td>
<td></td>
</tr>
<tr>
<td>7 300.0 0.0 2 7 8 2</td>
<td>2.012</td>
<td></td>
</tr>
<tr>
<td>8 -2.00</td>
<td>5 -2.00 2</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Two-bay frame, further reduced tangent modulus modeling</th>
<th>3</th>
<th>1</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 3 100</td>
<td>20.00 723.0 115.0 29 000.0 36.0</td>
<td></td>
</tr>
<tr>
<td>0 2 0</td>
<td>26.20 1300.0 175.0 29 000.0 36.0</td>
<td></td>
</tr>
<tr>
<td>0 8 0</td>
<td>0.00 240.0 1 1 4</td>
<td></td>
</tr>
<tr>
<td>1 0.00 240.0 1 1 4</td>
<td>0.00 240.0 1 2 7</td>
<td></td>
</tr>
<tr>
<td>2 0.00 240.0 1 2 7</td>
<td>90.0 0.0 2 4 5</td>
<td></td>
</tr>
<tr>
<td>3 0.00 240.0 1 3 9</td>
<td>180.0 0.0 2 5 6</td>
<td></td>
</tr>
<tr>
<td>4 90.0 0.0 2 5 6</td>
<td>90.0 0.0 2 6 7</td>
<td></td>
</tr>
<tr>
<td>5 180.0 0.0 2 6 7</td>
<td>300.0 0.0 2 7 8 2</td>
<td></td>
</tr>
<tr>
<td>6 90.0 0.0 2 7 8 2</td>
<td>1 1 1 3</td>
<td></td>
</tr>
<tr>
<td>7 300.0 0.0 2 7 8 2</td>
<td>2.00</td>
<td></td>
</tr>
<tr>
<td>8 -2.00</td>
<td>5 -2.00 2</td>
<td></td>
</tr>
</tbody>
</table>

5.1.10. Step 10: serviceability check

The lateral displacement of the first-order analysis corresponding to the load 2 kips (NINCRE = 1) is equal to 1.53 mm (0.06062 inches) at Node 9. The resulting displacement is computed as 40 mm (1.55 inches) ($=0.06062$ inches $\times 25.5$). The overall drift of the first-order analysis is calculated as 1/155 which does not meet the drift limit of $H/400$. As a result, the member sizes should be increased.

5.1.11. Step 11: ductility check

The compactness and lateral unbraced length of Beam BC and Column BE are checked in what follows:

1. check for compactness of Beam BC (W16 × 77):

   \[
   \frac{b_t}{2t_t} = 6.8
   \]

   \[
   \frac{65}{\sqrt{F_y}} = \frac{65}{\sqrt{36}} = 10.8 > 6.8
   \]

   therefore, the flange is compact.

2. check for unbraced length of Beam BC (W16 × 77):

   \[
   \frac{P_u}{\phi_y P_y} = \frac{21.8}{0.9 (22.6) (36)} = 0.0298 \leq 0.125
   \]

   \[
   \frac{640 (1 - 2.75 P_u)}{\phi_y P_y D_y} = \frac{640 (1 - (2.75) (0.0298))}{640 (1 - (2.75) (0.0298))} = 98 > 31.2
   \]

   where $P_u = 21.8$ kips is obtained from the output file P.OUT2. Thus, the web is compact;

3. check for compactness of Column EB (W14 × 53):

   \[
   \frac{b_t}{2t_t} = 6.1
   \]

   \[
   \frac{65}{\sqrt{F_y}} = \frac{65}{\sqrt{36}} = 10.8 > 6.8
   \]

   therefore, the flange is compact.
\begin{align*}
\frac{h}{t_w} &= 30.8 \\
\frac{P_u}{\phi_b P_y} &= \frac{96.6}{(0.9)(15.6)(36)} = 0.191 \geq 0.125 \\
\frac{191}{\sqrt{F_y}}(2.33 - \frac{P_u}{\phi_b P_y}) &= \frac{191}{\sqrt{36}}(2.33 - 0.191) = 68 > 30.8\]

where \( P_u = 96.6 \) kips is obtained from the output P.OUT2. Thus, the web is compact; and

4. check for unbraced length of Column EB (W14 \times 53):

\begin{align*}
L_{pd} &= \frac{[3600 + 2200 \left( \frac{M_1}{M_2} \right) r_y]}{F_y} \\
&= \frac{[3600 + 2200 \left( \frac{1111}{2511} \right) (1.92)]}{36} \\
&= 244 \text{ inches} (= 20 \text{ ft})
\end{align*}

where \( M_1 = 1111 \) kip/inch and \( M_2 = 2511 \) kip/inch are obtained from the output P.OUT2. Since the column height is equal to 20 ft, no lateral intermediate brace is required.

5.1.12. Step 12: adjustment of member sizes

Since the lateral drift of Step 10 does not satisfy the limitation, the beam and column sizes should be increased. Herein, the iteration procedures are not presented, since they are the same as those presented above.

5.2. Design by the LRFD method

In the LRFD method, two first-order analyses are performed, one for the sway case and the other for the non-sway case. The design procedure of the LRFD is usual and thus is not presented herein. The resulting member sizes shown in Fig. 9 are obtained from Ref. [2].

The three proposed methods result in identical member sizes, and they are one or two sizes smaller than those determined by the LRFD method as shown in Fig. 9. The proposed procedures possess the benefit of inelastic moment redistribution that leads to a reduction of steel weight.

6. Design example 2: four-story, five-bay frame

6.1. Description of frame

The AISC frame is provided in the 1991 Lecture Series. The building has four stories with a one-story penthouse on the center of the top story. The story height is 3.7 m (12 ft) except for the first story of 4.3 m (14 ft) and the fifth story of 3.0 m (10 ft). There are five bays in the east–west direction with spacing of 9 m (30 ft). The three bays in the north–south direction consist of the 11 m (36 ft) exterior bays and the 8.4 m (28 ft) interior bay. The frame is supported by exterior moment frames in the east–west direction and laterally braced in the north–south direction. All other columns except those moment frames are simply supported. The typical plan and section of the frame are shown in Figs. 10 and 11.
6.2. Load condition

The load condition is determined by BOCA 1990 [10] as follows:

1. roof loads
   dead: 1440 N/m² (30 psf)
   live: 1000 N/m² (21 psf)

2. floor loads
   dead: 3260 N/m² (68 psf)
   live: office: 3590 N/m² (75 psf)
       lobby and penthouse: 4790 N/m² (100 psf)

3. cladding: 720 N/m² (15 psf)

4. wind load: 130 Km/h (80 mph), Exposure C.

The following three load combinations govern the member sizes of the frame:

1. \(1.2D + 1.6L + 0.5S\)
2. \(1.2D + 1.6S + 0.5L\)
3. \(1.2D + 1.3W + 0.5(L + S)\)

where \(D\) = dead load, \(L\) = live load, \(S\) = snow load and \(W\) = wind load.

The live load reduction is considered in the calculation of loads.

6.3. Analyses

The moment frame with leaning columns in the east–west direction was analyzed and shown in Fig. 11. The yield stress was selected as 250 MPa (36 ksi). The column sizes were assumed to change every two stories. The analysis and design processes were carried out by the proposed three advanced methods. The geometric imperfection was assumed to be \(\psi = 1/500\) and the equivalent notional load was equal to \(0.002 \Sigma P_u\). The further reduced tangent modulus was adopted as \(0.85E_t\).

6.4. Member sizes

Most member sizes of the frame are governed by the wind load combination except that the upper girders are governed by the gravity load combinations. All three methods result in identical member sizes. The resulting member sizes of the moment frame are compared with those by the conventional LRFD method shown in Fig. 12 where the LRFD member sizes are taken directly from Ref. [5]. The member sizes by the proposed procedures are generally one or two sizes smaller than those by the LRFD method. This is because the AISC frame is highly redundant and therefore possesses much benefit of inelastic moment redistribution that leads to a reduction of steel weight.

6.5. Lateral drift

The overall drift of the first-order analysis is calculated as \(H/366\) for the wind loads of \(1.0W\). This does not meet the drift limit of \(H/400\). The maximum interstory drift is equal to \(H/357\) which satisfies the drift limit of \(H/300\). The column sizes should be increased to meet the overall drift requirements. If the interior column sizes in the first and second stories are increased from \(W14 \times 68\) to \(W16 \times 67\), the overall drift limit is found to be \(H/404\), which is less than the drift limit.

7. Conclusions

A design guide for steel frames using advanced analysis is provided. The step-by-step analysis and design procedures of the practical advanced analysis methods are presented. The two cases studied are the two-bay frame and the four-story, five-bay frame. The three practical modelings used are: (1) an explicit imperfection modeling; (2) an equivalent notional load modeling; and (3) a further reduced tangent modulus modeling. Member sizes determined by the proposed advanced analysis procedures are compared with those determined by the LRFD procedures, and a good agreement is generally observed. These comparative studies should provide further confirmation of the validity and simplicity of the proposed procedures for a routine LRFD design. Since the proposed procedures strike a balance between the requirement for realistic representation of actual behavior and failure mode of a structural system and the requirement for simplicity in use without the calculation of \(K\)-factor, it is considered that in both these respects, all three modelings are satisfactory, but the further reduced tangent modulus modeling appears to be the simplest and is therefore recommended for general use.
The proposed program may be used rigorously for practical design by taking advantage of the small program size, short run time, and simple input format.

References