INFORMATION TO USERS

This manuscript has been reproduced from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps. Each original is also photographed in one exposure and is included in reduced form at the back of the book.

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6" x 9" black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.

UMI

A Bell & Howell Information Company
300 North Zeeb Road, Ann Arbor MI 48106-1346 USA
313/761-4700 800/521-0600

Development of an Integrated System for the Geometry Definition, Load Determination and Analysis of Steel Structures

by

Aldo Martino

A thesis submitted to
the Faculty of Graduate Studies and Research
in partial fulfillment of
the requirements for the degree of
Master of Engineering*

Department of Civil and Environmental Engineering
Carleton University
Ottawa, Ontario, Canada
July 1997

***The Master of Engineering in Civil and Environmental Engineering is a joint program with the University of Ottawa, administered by the Ottawa-Carleton Institute for Civil Engineering.**

© 1997 by Aldo Martino

National Library
of Canada

Acquisitions and
Bibliographic Services

395 Wellington Street
Ottawa ON K1A 0N4
Canada

Bibliothèque nationale
du Canada

Acquisitions et
services bibliographiques

395, rue Wellington
Ottawa ON K1A 0N4
Canada

Your file Votre référence

Our file Notre référence

The author has granted a non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of this thesis in microform, paper or electronic formats.

The author retains ownership of the copyright in this thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without the author's permission.

L'auteur a accordé une licence non exclusive permettant à la Bibliothèque nationale du Canada de reproduire, prêter, distribuer ou vendre des copies de cette thèse sous la forme de microfiche/film, de reproduction sur papier ou sur format électronique.

L'auteur conserve la propriété du droit d'auteur qui protège cette thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.

0-612-27003-3

Canada

Department of Civil and Environmental Engineering

**The Undersigned recommend to the Faculty of Graduate Studies and Research acceptance
of the thesis**

**Development of an Integrated System for the Geometry Definition, Load
Determination and Analysis of Steel Structures**

submitted by

Aldo Martino

**in partial fulfillment of the requirements for the degree of
Master of Engineering**

Chair, Department of Civil and Environmental Engineering

Thesis Supervisor

**Carleton University
Ottawa, Ontario, Canada
July 1997**

Abstract

Currently, a second order elastic analysis for factored loads is required by North American standards and specifications for steel structures to determine the internal forces for member design. Conceptual inconsistencies between the analysis required and design using interaction equations for the corresponding limit states exist. These can be eliminated by conducting an advanced analysis which predicts the true behaviour of the structure. Application of commercially available finite element programs which discretize each member into shell elements are computationally inefficient. As a result researchers have developed beam-column elements which capture the second order geometric effects, material nonlinearity and lateral stability effects. Existing computing technology indicates that it is feasible to develop tools which can create and manipulate the large amount of data required for advanced analysis (LOADS 95), and to execute these computationally intensive analysis algorithms (CU-STAND 4.0). These are separate program modules and have not been integrated. The objective of this thesis was to integrate these programs, to develop a common database structure, to develop database translators and to demonstrate the versatility and advantages of an integrated system through examples. As part of this work, the data required for known and future developments in advanced analysis has been identified and the appropriate program structure specified.

Acknowledgments

I would like to extend my thanks and appreciation to my supervisor, Professor Stephen Kennedy, for his guidance, support and financial assistance. The provision of resources required to complete the work was also greatly appreciated.

Special thanks to Jan Chmelicek for his invaluable assistance with the program LOADS 95.

I wish to extend my thanks to Professor Greg Deierlein, Mourad Attalla and Chuin-shan Chen for their assistance with the program CU-STAND 4.0 during my visit at Cornell University.

I would like to thank Dr. Ron Ziemian for providing assistance with the program CU-STAND 4.0.

I would like to thank Dr. Murray Clarke for providing his finite element program, NIFA.

I wish to extend my thanks to my friends for their support and assistance during my studies.

I would like to sincerely thank my parents for their support and financial assistance throughout the course of my studies.

Table of Contents

Abstract.....	iii
Acknowledgments	iv
Table of Contents	v
List of Tables.....	viii
List of Figures	x
List of Symbols.....	xii

Chapter 1 Introduction

1.1 Integrated Analysis, Design, Fabrication and Erection System for Steel Buildings	1
1.2 Objectives and Scope.....	4
1.3 Outline	5

Chapter 2 Structural Analysis: A State of the Art Review

2.1 Introduction.....	7
2.2 First Order Elastic Analysis.....	14
2.3 Second Order Elastic Analysis	17
2.3.1 P- Δ and P- δ Effects.....	17
2.3.2 Curvature Shortening.....	19
2.3.3 Sway Shortening	20
2.3.4 Geometric Stiffness Matrix Formulation - CU-STAND 4.0 ..	20
2.4 Second Order Elastic-Plastic Analysis	25
2.5 Advanced Analysis.....	33

Chapter 3 Interrelationship and Description of LOADS 95 and CU-STAND 4.0

3.1 Introduction.....	37
---------------------------	----

3.2	Database Structure for LOADS 95	44
3.3	Database Structure for Members.....	44
3.3.1	Geometric Data Structure for a Member	46
3.3.2	Load Data Structure for a Member.....	52
3.3.2.1	Gravity Loads.....	57
3.3.2.2	Lateral Loads	57
3.4	Database Structure for CU-STAND 4.0.....	58
3.4.1	Geometric Data.....	59
3.4.2	Load Data.....	65
Chapter 4	Integration of LOADS 95 and CU-STAND 4.0	
4.1	Introduction.....	67
4.2	General Description of Files for the Integration of LOADS 95 and CU-STAND 4.0.....	69
4.3	Description of Load Combination Table, Statically Equivalent Uniform Load Distributions and Nodal Renumbering Scheme	76
4.3.1	load_comb_table.c	78
4.3.2	convert_to_uniform.c.....	78
4.3.3	re_number.c.....	83
4.4	Description of Data Files	85
4.5	Suggested Improvements to LOADS 95 and CU-STAND 4.0	86
Chapter 5	Examples	
5.1	Introduction.....	89
5.2	Data Input for Program NIFA.....	90
5.3	Example 1 - Two Dimensional Rigid Frame	93
5.4	Example 2 - Lowrise Three Dimensional Rigid Frame.....	98
5.5	Example 3 - Multi-storey Steel Building.....	104

Chapter 6	Conclusions	
6.1	Summary	110
6.2	Conclusions	112
6.3	Recommendations and Future Work	113
References		116

List of Tables

Table	Page
1.1 Description of the design process	2
2.1 Description of load intensities for a rigid plane frame.....	10
2.2 Physical attributes and behavioural effects of steel framed structures	35
3.1 General database structure for LOADS 95	45
3.2 Member types with corresponding code numbers and letters in LOADS 95.....	50
3.3 Section code numbers in LOADS 95	51
3.4 Default sections in LOADS 95	51
3.5 Source of load in LOADS 95	52
3.6 Live load reduction factors in LOADS 95.....	53
3.7 Wind load cases in LOADS 95	55
3.8 Wind load directions in LOADS 95	56
3.9 Applied loads in LOADS 95.....	56
3.10 Description of geometric data generated by CU-PREPF	60
3.11 General analysis parameters in CU-STAND 4.0.....	61
3.12 Element properties in CU-STAND 4.0	63
3.13 Section properties in CU-STAND 4.0	64
3.14 Source of loads in CU-STAND 4.0	66
3.15 Load types in CU-STAND 4.0	66
3.16 Description of load data for each load type.....	66
4.1 Description of buttons within main menu page of the program.....	68
4.2 Miscellaneous files to control the program flow.....	72
4.3 Geometric data related files	74
4.4 Load data related files	76

4.5	Ratio of approximate to actual value of maximum shear forces, bending moments and deflections for various end conditions.....	80
4.6	Data files generated for the integration process.....	85
5.1	Nonlinear control data in NIFA	91
5.2	Load definitions for lowrise three dimensional rigid framed structure.....	98
5.3	Dead and live loads (use and occupancy).....	106
5.4	Time required to define the geometric and load characteristics and to perform analyses	109

List of Figures

Fig.		Page
2.1	Load-displacement behaviour of rigid frames.....	8
2.2a	Degrees of freedom for beam-column element.....	13
2.2b	Nodal forces for beam-column element.....	13
2.3	Three dimensional elastic stiffness matrix of a beam-column element	16
2.4	P-delta stability effects	18
2.5	Three dimensional geometric stiffness matrix of a beam column element (including contribution of finite rotations).....	26
2.6	Three dimensional elastic stiffness matrix of a beam column element (including nonuniform torsion effects)	27
2.7	Three dimensional geometric stiffness matrix of a beam-column element (including nonuniform torsion and contribution of finite rotations).....	28
3.1	Model of the design process	38
3.2	Main menu page for the Integrated System for the Analysis, Design, Fabrication and Erection of Steel Buildings	39
3.3	LOADS 95 title screen	40
3.4	CU-STAND 4.0 title screen	42
3.5	Geometric data for a typical beam member in LOADS 95.....	47
3.6	Table of member types to be extracted from LOADS 95.....	48
3.7	Plan view of roof surface accompanied by member data in LOADS 95	49
3.8	Load data for a typical member in LOADS 95.....	54
3.9	Load data definition in CU-STAND 4.0	65
4.1a	Flowchart of files created for the integration of LOADS 95 and CU-STAND 4.0.....	70
4.1b	Files created for the generation of geometric and load data.....	71
4.2	Definition of a web unit vector	77
4.3	Local and global coordinate system in CU-STAND 4.0	77

4.4	Load combination table in modified version of CU-STAND 4.0.....	79
4.5	Statically equivalent uniform load distributions.....	81
4.6	Shear force and bending moment diagrams for approximate and actual load distributions.....	82
4.7	Drifting snow load case example	84
4.8	Interface display for nonproportional loading.....	88
5.1	Definition of two dimensional rigid frame for NIFA.....	94
5.2	Rigid frame definition for ISYSS.....	95
5.3	Applied load ratio vs. deflection curve for node b.....	97
5.4	Applied load ratio vs. deflection curve for node t.....	97
5.5	Lowrise three dimensional frame	99
5.6	Load vs. Displacement curves, load combination 1	101
5.7	Load vs. Displacement curves, load combination 2	102
5.8	Plastic hinges for load combination 1.....	103
5.9	Multi-storey steel building generated in LOADS 95.....	105
5.10	Multi-storey steel frame displayed in modified version of CU-STAND 4.0...	107
5.11	Load vs. Displacement curves for node 5.....	108

List of Symbols

A	=	area of cross-section
B_A	=	warping torque at node A
B_B	=	warping torque at node B
C_{ijkl}	=	elastic constant tensor
C_w	=	warping torsional constant
E	=	modulus of elasticity
E_t	=	tangent modulus
e_k^s	=	linear incremental strain
$\{ F \}$	=	nodal force vector
F_i	=	element nodal force at node i
F_x	=	axial load
F_{xA}	=	axial nodal load at node A
F_{xB}	=	axial nodal load at node B
F_y	=	axial yield load
F_{yA}	=	transverse nodal load in the y direction at node A
F_{yB}	=	transverse nodal load in the y direction at node B
F_{zA}	=	transverse nodal load in the z direction at node A
F_{zB}	=	transverse nodal load in the z direction at node B
G	=	shear modulus

$[G]$	=	gradient vector of yield surface
$\{G_i\}$	=	gradient vector of yield surface at node i
$\{G_j\}$	=	gradient vector of yield surface at node j
H	=	lateral load
I_y	=	moment of inertia about y-axis
I_z	=	moment of inertia about z-axis
J	=	St. Venant torsional constant
K	=	Wagner coefficient
$[K_e]$	=	elastic stiffness matrix
$[K_g]$	=	geometric stiffness matrix
$[K_{ep}]$	=	elastic-plastic stiffness matrix
$[K_p]$	=	plastic reduction matrix
$[K_t]$	=	tangent stiffness matrix
L	=	length of an element
M	=	bending moment
M_{approx}	=	approximate bending moment value
M_{actual}	=	actual bending moment value
M_{xA}	=	St. Venant torque at node A
M_{xB}	=	St. Venant torque at node B
M_y	=	bending moment about the y-axis

M_{yA}	=	bending moment about the y-axis at node A
M_{yB}	=	bending moment about the y-axis at node B
M_{yp}	=	plastic bending moment about y-axis
M_z	=	bending moment about the z-axis
M_{zA}	=	bending moment about the z-axis at node A
M_{zB}	=	bending moment about the z-axis at node B
M_{zp}	=	plastic bending moment about z-axis
m_y	=	ratio of bending moment about y-axis to plastic bending moment about y-axis
m_z	=	ratio of bending moment about z-axis to plastic bending moment about z-axis
P_E	=	elastic buckling load
P_I	=	load intensity of the last plastic hinge causing the formation of a plastic collapse mechanism
P_{L1}	=	load intensity of the first plastic hinge on a first order elastic-plastic analysis
P_{L2}	=	load intensity of the first plastic hinge on a second order elastic-plastic analysis
P_M	=	actual maximum load
P_{P1}	=	plastic collapse load
P_{P2}	=	maximum load intensity on the second order elastic-plastic analysis curve
P	=	axial load
P_y	=	axial yield load
$\{ q \}$	=	nodal displacement vector

$q_{1/10}, q_{1/30}$	=	wind pressure
q_1, q_2, q_3	=	load distribution values
q_i	=	load intensity at node i
q_j	=	load intensity at node j
S_s	=	ground snow load
S_r	=	rain load
V	=	volume of the element
u_{xA}	=	axial nodal displacement at node A
u_{xB}	=	axial nodal displacement at node B
u_{yA}	=	transverse nodal displacement in the y direction at node A
u_{yB}	=	transverse nodal displacement in the y direction at node B
u_{zA}	=	transverse nodal displacement in the z direction at node A
u_{zB}	=	transverse nodal displacement in the z direction at node B
V_{approx}	=	approximate shear force value
V_{actual}	=	actual shear force value
$\overline{W}$	=	web unit vector
w	=	linearly varying load distribution
w_x	=	x component of web unit vector
w_y	=	y component of web unit vector
w_z	=	z component of web unit vector
w_{D1}, w_{D2}	=	uniform dead load distribution
w_{L1}, w_{L2}	=	uniform live load distribution

w_1, w_2	=	uniform load distribution
y	=	distance along the y-axis from the neutral axis to a point on the cross-section
z	=	distance along the z-axis from the neutral axis to a point on the cross-section
Δ	=	lateral displacement of a member relative to its ends
Δ_{actual}	=	actual deflection value
Δ_{approx}	=	approximate deflection value
Δ_b	=	deflection of node b
Δ_t	=	deflection of node t
$\{ \Delta F \}$	=	incremental nodal force vector
$\{ \Delta q \}$	=	incremental nodal displacement vector
$\{ \Delta q_p \}$	=	incremental plastic displacement vector
$\{ \Delta q_{pi} \}$	=	incremental plastic displacement vector at node i
$\{ \Delta q_{pj} \}$	=	incremental plastic displacement vector at node j
Δu	=	incremental nodal displacement in the x direction
Δv	=	incremental nodal displacement in the y direction
Δw	=	incremental nodal displacement in the z direction
$[\Delta \epsilon]$	=	incremental strain vector
Π	=	total potential energy
δ	=	lateral displacement of a member relative to its chord
δW_{int}	=	virtual work done by internal forces

δW_{ext}	=	virtual work done by external forces
δe_{ij}^s	=	linear incremental strain (resulting from a first order transformation matrix)
δe_{ij}^r	=	incremental strain (nonlinear when expressed in terms of rotation components)
$\delta \eta_{ij}^s$	=	nonlinear incremental strain (resulting from a first order transformation matrix)
ϕ	=	yield surface function
$\{ \lambda \}$	=	plastic deformation vector
λ_i	=	plastic deformation magnitude at node i
λ_j	=	plastic deformation magnitude at node j
θ_{xA}	=	nodal rotation about the x-axis at node A
$\dot{\theta}_{xA}$	=	warping displacement at node A
θ_{xB}	=	nodal rotation about the x-axis at node B
$\dot{\theta}_{xB}$	=	warping displacement at node B
θ_{yA}	=	nodal rotation about the y-axis at node A
θ_{yB}	=	nodal rotation about the y-axis at node B
θ_{zA}	=	nodal rotation about the z-axis at node A
θ_{zB}	=	nodal rotation about the z-axis at node B
$[\sigma]$	=	element stress vector
σ_y	=	element yield stress
τ_{ij}	=	Cauchy stress tensor

Chapter 1

Introduction

1.1 Integrated Analysis, Design, Fabrication and Erection System for Steel Buildings

The analysis-design-fabrication-erection process for building structures has exhibited great variability due to the unique nature of these structures and has not been integrated to any significant extent. The process tends to be sequential and segmented without adequate interaction between the architects, engineers and fabricators who are responsible for different parts of the process. The quality of design also varies greatly.

The development of a knowledge-based system that integrates the process would lead to better and more cost effective structures. This would arise from ease of verification, opportunity for increased conceptual development (alternate framing patterns and construction), improved understanding of the loads acting on the structure and the behaviour under those loads, and the unimpeded flow of information throughout the entire process. An integrated system should give the engineer control in the decision-making

process and qualitative aspects of design. The system should also enable the engineer to intervene and make revisions as the process progresses. Within an integrated system, expertise can be programmed to give detailed design logic that can both inform and educate the practicing structural engineer about the multitude of facets of structural engineering including changes in construction practices, improved fabrication methods and advances in structural engineering understanding.

The essential steps for a complete integrated system for the layout, analysis, design, detailing, fabrication and erection of a steel building are given in Table 1.1.

Step	Designation	Objectives
1	Preprocessing	Definition of structural geometry, determination of loads and preliminary sizing of members
2	Structural analysis	Determination of displacements and internal forces, including geometric and material nonlinear effects
3	Design of members	Selection and verification of suitability of members
4	Reanalysis and redesign	Iteration of Steps 2 and 3 until all design requirements are satisfied in a consistent manner
5	Design of connections	Selection and verification of suitability of connections and joints
6	Erection considerations	Investigation of partial structures under construction loadings and bracing conditions
7	Description of structure	Preparation of working drawings and specifications
8	Fabrication	Manufacturing of all elements of the structure for field erection
9	Construction	Erection and completion of finished building

Table 1.1: Description of the design process.

In North American practice there are generally a number of different organizations involved in this process. In a typical building project, the architect may be responsible for portions of steps 1, 7 and 9; the consulting engineer responsible for a portion of 1, all of 2, 3 and 4 and portions of 9; and the general contractor responsible for portions of 6 and most of 9. Consequently, communication and coordination are crucial links and a high probability of litigation exists if problems arise. A complete integrated system would provide a common database for, and the integration of, all the processes involved in all the above steps. Design data would be shared by all the participants in the process.

Currently, material standards and specifications in North America specify the method of analysis (step 2), second order elastic analysis for factored loads, and requires that the members be designed according to the limit states design equations given for the corresponding internal forces (step 3). Recognized inconsistencies in this design approach have led to research and development of advanced analysis methods which would replace steps 2 and 3 as the analysis would model the true behaviour. The Australian Standard, AS4100-1990, and the Eurocode, EC3 (1990), have adopted provisions that allow the use of advanced analysis. Similar provisions are being considered for North American standards and specifications. Note, step 4 would become a resizing algorithm and the advanced analysis would then simply be repeated.

The existing computing technology and benefits suggested by the trends which include; increased computational efficiency, increased storage capacity, improved graphics and user interface development tools and more advanced programming languages (object oriented, C++, Java), indicate that this technology will make practical the use of advanced

analysis for steel buildings. However, the amount of data required to fully describe the structural geometry, members, imperfections, lateral and torsional restraints, residual stresses, loads and load combinations and the geometry of all the connections is significant and requires specially developed tools to make the use of advanced analysis feasible.

The most comprehensive programs in this area are:

- i) LOADS 95, Chmelicek (1993, 1996), the only program available that has the ability to define the structural geometry and to determine the loads and load combinations acting on the members in accordance with the National Building Code of Canada 1990, NRCC 1990.
- ii) CU-STAND 4.0 (Cornell University - static analysis and design), McGuire et al. (1989), a state of the art advanced analysis program that provides the capability of conducting a second order elastic-plastic analysis for three dimensional steel structures and includes some recognition of the effects of residual stresses and the behaviour of flexible and semi-rigid connections on the overall behaviour of the structure.

1.2 Objectives and Scope

Logically, the first step in developing an integrated system and its database is to integrate LOADS 95 and CU-STAND 4.0 to gain an understanding of the data and program structure required. The work involved includes:

- i) a state of the art review of the attributes and limitations of beam-column elements from the simplest to the most complex element that involve both geometric and material nonlinearities.
- ii) a review of CU-STAND 4.0 program and database structure. All design oriented subroutines based on the American Institute of Steel Construction Load and Resistance Factor Design Specifications (AISC-LRFD) were extracted.
- iii) a review of LOADS 95 program and database structure.
- iv) the creation of files required to integrate LOADS 95 and CU-STAND 4.0.
- v) the identification of the limitations of the capabilities of each program.
- vi) the identification of the required data for comprehensive advanced analysis programs.

1.3 Outline

The following section describes the content of the upcoming chapters providing the reader with an overview of the organization of the thesis.

Chapter 2 presents a state of the art review of the attributes and limitations of beam-column elements for first order elastic, second order elastic and second order elastic-plastic analyses. Stiffness matrices related to these analysis methods are also presented. A description of the physical attributes and the geometric and material effects that need to be considered for an advanced analysis is given.

Chapter 3 gives a complete description of the database structure for both LOADS 95 and CU-STAND 4.0.

A description of the program developed for the integration of LOADS 95 and CU-STAND 4.0 is presented in Chapter 4. The limitations of each with respect to the requirements for an ideal advanced analysis program are listed.

The examples given in Chapter 5 verify the integration program and illustrate practical aspects of applying advanced analysis on large complex irregular structures. Also demonstrated is the versatility of the integrated system in that any realistic steel structure can be modeled and analyzed.

A summary of this work and the conclusions drawn are given in Chapter 6. Also described are a list of suggestions for future work that address the existing limitations of LOADS 95 and CU-STAND 4.0 and that incorporate recent advances in program logic and computing technology.

Chapter 2

Structural Analysis: A State of the Art Review

2.1 Introduction

Presented is a state-of-the-art review of the attributes and limitations of beam-column elements from the simplest to the most complex element that involve both geometric and material nonlinearities. The behaviour from the simplest to the most complex element modeled by different analysis methods is illustrated by the load-characteristic nodal displacement curves for a simple portal frame given in Figure 2.1. These include first order elastic, second order elastic, first order rigid-plastic, first order elastic-plastic, second order rigid-plastic, second order elastic-plastic and a finite element analysis including both geometric and material nonlinearities that represents the true behaviour.

In a first order elastic analysis, equilibrium is formulated on the undeformed geometry of the structure resulting in a linear load-displacement response. For a second order elastic analysis, equilibrium is formulated on the deformed geometry of the

Figure 2.1: Load-displacement behaviour of rigid frames.

structure and includes geometric nonlinearities that result in a nonlinear load-displacement relationship. A first order rigid plastic analysis describes the behaviour of a frame that exhibits no deflections until the plastic collapse load has been reached in sufficient locations such that a mechanism develops. In a first order elastic-plastic analysis, the effects of material nonlinearity only are included through the formation of successive plastic hinges whereby the behaviour between plastic hinges is elastic. A second order rigid-plastic analysis differs from the first order case due to the inclusion of deflections in the formulation of the equilibrium equations. In a second order elastic-plastic analysis, both geometric and material nonlinearities are included and the formation of successive plastic hinges occurs until a collapse mechanism is achieved. A finite element analysis includes both geometric and material nonlinearities whereby plastification is assumed to occur over a plastic zone therefore accurately modeling the true behaviour of the frame.

Illustrated in Figure 2.1 are a number of load intensities that have significance in relation to the actual maximum load intensity P_M . Table 2.1 provides a description of the significance of each load intensity that will emphasize their importance in the understanding of frame behaviour.

Upon reviewing the different methods of analysis, it can be said that an advanced structural analysis model is one that approximates the true behaviour of frame structures and includes overall stability, local and out-of-plane stability of members, plastification of structural members as load is increased and redistributed amongst framing members, and all geometric nonlinear effects of loads acting on the deformed configuration.

Load Intensity	Description
P_E	<ul style="list-style-type: none"> • elastic buckling load • highest theoretical load possible
P_{P1}	<ul style="list-style-type: none"> • plastic collapse load • upper bound of P_M
P_{P2}	<ul style="list-style-type: none"> • maximum load intensity of second order elastic-plastic analysis • upper bound of P_M
P_M	<ul style="list-style-type: none"> • actual maximum load
P_I	<ul style="list-style-type: none"> • load intensity where the last plastic hinge occurs causing the formation of a collapse mechanism
P_{L1}	<ul style="list-style-type: none"> • load intensity where the first plastic hinge occurs • lower bound of P_M for frames with low axial loads
P_{L2}	<ul style="list-style-type: none"> • load intensity where the first plastic hinge occurs but unlike P_{L1} second order effects are included • lower bound of P_M

Table 2.1: Description of load intensities for a rigid plane frame.

Currently, material standards and specifications in North America generally require that a second order elastic analysis for factored loads be conducted on the structure to determine the internal forces for which the members must be designed. For CAN/CSA-S16.1-94, these requirements are outlined in Clause 7, Loads and Safety Criterion, and Clause 8, Analysis of Structure. This load intensity is related to the development of the first plastic hinge given by the point marked P_{L2} on the second order elastic-plastic curve in Figure 2.1. There are some inconsistencies in designing these members for the limit states for the internal forces given by this analysis that lead to a combination of unconservative and conservative uncertainties. On the unconservative side, the plastic hinge occurs over a zero length. The rest of the member is assumed to behave elastically

and as a result underestimates the characteristic nodal displacement of the structure and overestimates the load at which the hinge will form. This load estimate is generally conservative as the real structure will develop more than one plastic hinge before it forms a mechanism and the analysis and resulting internal force distributions are based on centerline dimensions which do not include the actual size of beams and columns that reduce effective lengths. Obviously, advanced analysis methods which accurately describe the behaviour of the structure negate these uncertainties and eliminate the need for approximate interaction equations to describe the behaviour associated with the ultimate strength of each member. Recognizing this, McGuire, Deierlein, White, Ziemian, Clarke and others have been developing advanced inelastic analysis methods to model the true behaviour of the structure. The goal is to provide a realistic model of the strength and stability of the overall structural system and to ensure that the interdependence of member and system strength and stability have been accounted for resulting in economical, efficient and more uniformly safe designs.

Ideally, an advanced analysis should have the ability to predict accurately the behaviour of class 1, 2, 3 or 4 sections that are prismatic or non-prismatic, doubly symmetric, singly symmetric or asymmetric with realistic connections and restraint. The analysis should include the effects of geometric nonlinearities ($P-\Delta$, $P-\delta$, curvature shortening, sway shortening, initial imperfections (sweep, camber, initial twist, out-of-plumbness), material nonlinearities, varying residual stress patterns and have the ability to model local buckling, lateral-torsional buckling and flexural-torsional buckling.

To understand the current state-of-the-art for advanced structural analysis of framed structures, a description of the attributes and limitations for the most advanced model of beam-column elements is given for each analysis in order of complexity from first order elastic, to second order elastic and to second order elastic-plastic that includes both geometric and material nonlinearities. The state-of-the-art is represented by research development programs that advance commercial programs in their relative complexity.

The use of beam-column elements to model behaviour is common as they are easy to understand as related to frame behaviour as illustrated in Figure 2.1. Focus has been placed on these types of elements as they are computationally efficient as compared to discretizing structural members into plates or brick elements. The degrees of freedom and nodal forces of a typical beam-column element used in CU-STAND 4.0 is illustrated in Figures 2.2(a) and 2.2(b). Analysis methods are implemented through the use of stiffness matrices that explicitly model the corresponding behaviour of beam-column elements. Solution strategies are based on iterative schemes that provide an efficient means of accurately solving equilibrium equations. As a result, the implementation of advanced analysis techniques that efficiently model the true behaviour of frames is entirely possible in the near future.

The feasibility of advanced analysis can be attributed to analytical capabilities, available computing power and programming capabilities. In the past decade, computer technology has allowed the development of sophisticated analysis tools that can realistically predict the behaviour of framed structures. A number of researchers have been working on the development of such tools to incorporate them into an advanced

Figure 2.2(a): Degrees of freedom for beam-column element.

Figure 2.2(b): Nodal forces for beam-column element.

analysis. At the present time, the practical consideration of all behavioural and physical attributes is not possible but the ability to model in-plane strength and stability by conducting a second order inelastic analysis is achievable. The Australian Standard, AS4100-1990, and the Eurocode, EC3 (1990), allow the use of an advanced analysis. CAN/CSA-S16.1-94 specifically states in Clause 8.6.1 that a second order elasto-plastic analysis be used if the second order effects exceed 40% of the primary effects. The members still must be designed according to the rules given in the standard.

Program development has progressed rapidly due to advances in computing technology. Constant increases in the computational efficiency and storage capacities of computers have facilitated the development of interactive graphical programs. The development of graphical user interfaces have enabled the visualization of structural geometry and behaviour. Extremely large databases can be created with reasonable effort commensurate with the owner's ability to pay for the analysis required. With the development of object oriented programming languages such as C++ and Java, the modularization of programs allows subsequent changes to be made with ease and such programs are easier to integrate with graphic oriented languages. Advances in analytical procedures can therefore be implemented quickly and with little effort. As a result, computing technology will make practical, the use of advanced analysis for steel buildings.

2.2 First Order Elastic Analysis

A first order elastic analysis represents the simplest and most common method of analysis used in practice today. Equilibrium is formulated on the undeformed geometry of the structure resulting in a linear load-displacement response that can be expressed by:

$$\{ F \} = [K_e] \{ q \} \quad (2.1)$$

where, $\{ F \} =$ nodal force vector

$[K_e] =$ elastic stiffness matrix

$\{ q \} =$ nodal displacement vector

Common assumptions applicable to an elastic analysis are that both strains and deflections are small, plane sections remain plane and shear deformations can be neglected. Member lengths are based on gridline dimensions, ie. the effects of finite size joints is not included. A number of methods used in the past that provide exact analytical solutions include slope deflection, virtual work and moment distribution. Procedures used today, as a result of the available computing power, include the force and displacement methods that are based on matrix algebra.

The three dimensional stiffness matrix for a beam-column element with six degrees of freedom per node (three translations and three rotations) is illustrated in Figure 2.3.

Each column of the stiffness matrix represents the forces generated at the member ends by successively applying a unit displacement at one degree of freedom while maintaining all other degrees of freedom fixed.

2.3 Second Order Elastic Analysis

A second order elastic analysis differs from the elastic analysis in that equilibrium is formulated on the deformed geometry of the structure resulting in a nonlinear load-displacement response that can be expressed by:

$$\{ \Delta F \} = \left[[K_e] + [K_g] \right] \{ \Delta q \} \quad (2.2)$$

where, $\{ \Delta F \}$ = incremental nodal force vector

$[K_g]$ = geometric stiffness matrix

$\{ \Delta q \}$ = incremental nodal displacement vector

There are a number of geometric nonlinear effects that contribute to the nonlinearity of the load-displacement relationship. White et al. (1991) compiled a list of geometric and material nonlinear effects affecting the strength and stability of steel frames and is similar to that given in Table 2.2.

2.3.1 P- Δ and P- δ Effects

One of the best understood effects includes both the P- Δ and P- δ effects which are graphically defined and illustrated for a typical beam-column element in Figure 2.4. These are defined by Chen and Lui (1991) as:

“P- δ moments are moments caused by the axial force acting through the lateral displacements of the member relative to its chord.”

“P- Δ moments are moments caused by the axial force acting through the relative lateral displacement of the two ends of the member.”

a) P - Δ effects
(Frame instability effect)

b) P - δ effects
(Member instability effect)

Figure 2.4 P-delta stability effects.

The $P-\delta$ moments have a destabilizing effect on the member and are often referred to as a member instability effect and are included in overall beam column strength interaction equations for design as an amplification factor. This effect becomes more pronounced for slender members or members subjected to large axial loads. $P-\Delta$ moments have a destabilizing effect on the frame and are often referred to as a frame instability effect and are accounted for in design by amplifying the translational load effects obtained from a first order elastic analysis in accordance with Clause 8.6.1 in CAN/CSA-S16.1-94. Both of these effects have been included in the formulation of the geometric stiffness matrix.

2.3.2 Curvature Shortening

Curvature shortening is defined by White et al. (1991) as:

“Curvature shortening - effect of curvature on longitudinal displacements at member ends.”

Longitudinal (axial) displacements in a member can result from either axial load or bending, where for most practical cases, the former is more influential. For the case where bending moments cause significant bending, the longitudinal displacements can be accounted by using a stiffness modifier, Chen and Lui (1991). For rectangular frames, this effect is generally considered negligible and is not included in the geometric matrix formulation presented. In addition to curvature shortening, sway shortening represents another phenomena affecting the stiffness matrix.

2.3.3 Sway shortening

Sway shortening is defined by White et al. (1991) as:

“Sway shortening - effect of large chord rotation on longitudinal displacements at member ends.”

It has been found that this effect is also negligible and is not included.

2.3.4 Geometric Stiffness Matrix Formulation - CU-STAND 4.0

The updated Lagrangian method and the principle of minimum potential energy were used to develop the geometric stiffness matrix. The matrix was developed from a finite element approach that uses a Hermitian shape function (cubic polynomial) to approximate the transverse displacements along the element length. The beam-column element is assumed to be prismatic, straight at the beginning of each load increment, and has six nodal degrees of freedom at each end. The derivation is based on the potential energy expression given below:

$$\Pi = \frac{1}{2} \int_V [\Delta \epsilon]^T [E] \{\Delta \epsilon\} dV + \int_V [\sigma]^T \{\Delta \epsilon\} dV - [\Delta q]^T \{\Delta F\} \quad (2.3)$$

where, Π = total potential energy

$[\Delta \epsilon]$ = incremental strain vector

$[E]$ = modulus of elasticity

$[\sigma]$ = element stress vector

V = volume of the element

The first two terms represent the strain energy resulting from stress and strain increments while the last term is the potential energy of the applied loads. The incremental strain along the length of the member includes both the centroidal strains due to axial loads and bending strains due to biaxial flexure and is given by:

$$\Delta \epsilon = \frac{\partial \Delta u}{\partial x} - y \frac{\partial^2 \Delta v}{\partial x^2} - z \frac{\partial^2 \Delta w}{\partial x^2} + \frac{1}{2} \left[\left(\frac{\partial \Delta u}{\partial x} \right)^2 + \left(\frac{\partial \Delta v}{\partial x} \right)^2 + \left(\frac{\partial \Delta w}{\partial x} \right)^2 \right] \quad (2.4)$$

where, Δu = incremental nodal displacement in the x direction

Δv = incremental nodal displacement in the y direction

Δw = incremental nodal displacement in the z direction

y = distance along y-axis from the neutral axis to a point on the cross section

z = distance along z-axis from the neutral axis to a point on the cross-section

The expression for the normal stress at any cross section along the length of the member is given by:

$$\sigma = \frac{F_x}{A} - \frac{M_z y}{I_z} - \frac{M_y z}{I_y} \quad (2.5)$$

where, F_x = axial load

A = area of cross-section

M_z = bending moment about the z-axis

I_z = moment of inertia about the z-axis

M_y = bending moment about the y-axis

I_y = moment of inertia about the y-axis

By minimizing the potential energy and solving the corresponding equations, the elastic stiffness matrix, Figure 2.3, and the geometric stiffness matrix can be determined. Using the principle of virtual work, improvements were made to the geometric stiffness matrix to include the effects of finite rotations, Chen (1994). The general expression for virtual work is given as:

$$\delta W_{int} = \delta W_{ext} \quad (2.6)$$

where, δW_{int} = virtual work done by external forces

δW_{ext} = virtual work done by internal forces

Using the second Piola-Kirchhoff stresses and the Green-Lagrange strains, Equation 2.6 can be expanded as shown below:

$$\int_V C_{ijkl} e_{kl}^s \delta e_{ij}^s dV + \int_V \tau_{ij} \delta \eta_{ij}^s dV + \int_V \tau_{ij} \delta e_{ij}^r dV = \delta W_{ext} - \int_V \tau_{ij} \delta e_{ij}^s dV \quad (2.7)$$

where, C_{ijkl} = elastic constant tensor

e_{kl}^s = linear incremental strain

δe_{ij}^s = linear incremental strain (resulting from a first order transformation matrix)

δe_{ij}^r = incremental strain (nonlinear when expressed in terms of rotation components)

$\delta \eta_{ij}^s$ = nonlinear incremental strain (results from a first order transformation matrix)

τ_{ij} = Cauchy stress tensor

Evaluation of the first term on the left hand side gives the 12x12 elastic stiffness matrix, shown in Figure 2.3. As before, cubic shape functions were used to model the transverse displacements of the beam-column. The second term represents the geometric stiffness matrix without the effects of finite rotation and the third term gives that contribution. The resulting matrix from the third term is shown below:

$$\int_V \tau_{ij} \delta e_{ij}^r dV = \begin{bmatrix} 0_{3 \times 3} & 0_{3 \times 3} & 0_{3 \times 3} & 0_{3 \times 3} \\ 0_{3 \times 3} & S_A & 0_{3 \times 3} & 0_{3 \times 3} \\ 0_{3 \times 3} & 0_{3 \times 3} & 0_{3 \times 3} & 0_{3 \times 3} \\ 0_{3 \times 3} & 0_{3 \times 3} & 0_{3 \times 3} & S_B \end{bmatrix} \quad (2.8)$$

$$\text{where, } S_A = \begin{bmatrix} 0 & -\frac{1}{2} M_{zA} & \frac{1}{2} M_{yA} \\ -\frac{1}{2} M_{zA} & 0 & 0 \\ \frac{1}{2} M_{yA} & 0 & 0 \end{bmatrix}$$

$$S_B = \begin{bmatrix} 0 & -\frac{1}{2} M_{zB} & \frac{1}{2} M_{yB} \\ -\frac{1}{2} M_{zB} & 0 & 0 \\ \frac{1}{2} M_{yB} & 0 & 0 \end{bmatrix}$$

M_{yA} = bending moment about the y-axis at node A

M_{zA} = bending moment about the z-axis at node A

M_{yB} = bending moment about the y-axis at node B

M_{zB} = bending moment about the z-axis at node B

The complete 12x12 geometric stiffness matrix including finite rotation effects is shown in Figure 2.5. Chen (1994) further refined the elastic and geometric stiffness matrices by including warping torsion. The beam-column element now includes a seventh degree of freedom at each end for warping displacements as illustrated in Figure 2.2(a). Again based on the principle of virtual work, the matrices (14x14) were rederived and are shown

in Figures 2.6 and 2.7. Note, local buckling, lateral torsional buckling and shear deformations are not included.

2.4 Second Order Elastic-Plastic Analysis

A second order elastic-plastic analysis incorporates both geometric and material nonlinear effects. There are two basic methods to model the latter, plastic zone analysis and plastic hinge analysis. A plastic-zone analysis explicitly captures the spread of yielding through the cross section and along the length of the member. This analysis is computationally intensive and is generally limited to research in structural steel behaviour.

In general, plastic hinge analyses model inelastic material behaviour by concentrating yielding at the ends of the element (plastic hinge) while the rest of the element remains elastic. For an elastic-plastic hinge analysis, these hinges form when the plastic capacity of the section is reached which is defined by a three parameter yield surface given as a function of the axial force and bending moments and is expressed by:

$$1.15p^2 + m_z^2 + m_y^4 + 3.67p^2 m_z^2 + 3.00p^6 m_y^2 + 4.65m_z^2 m_y^2 = 1.0 \quad (2.9)$$

$$\begin{aligned} \text{where, } p &= F_x / F_y \\ m_y &= M_y / M_{yp} \\ m_z &= M_z / M_{zp} \end{aligned}$$

F_{xB}/L	0	0	0	0	$-M_{yA}/L$	$-M_{zA}/L$	$-F_{xB}/L$	0	0	0	0	$-M_{yB}/L$	$-M_{zB}/L$
0	$6F_{xB}/5L + 12F_{xB}I_y/AL^3$	0	M_{yA}/L	M_{zA}/L	M_{xB}/L	$F_{xB}/10 + 6F_{xB}I_y/AL^2$	0	$-6F_{xB}/5L - 12F_{xB}I_y/AL^3$	0	M_{yB}/L	$-M_{zB}/L$	$F_{xB}/10 + 6F_{xB}I_y/AL^2$	
0	0	$6F_{xB}/5L + 12F_{xB}I_y/AL^3$	M_{zA}/L	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	$-M_{zA}/3 + M_{zB}/6$	M_{xB}/L	0	0	$-6F_{xB}/5L - 12F_{xB}I_y/AL^3$	M_{zB}/L	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	$-M_{zB}/L$	
0	M_{yA}/L	M_{zA}/L	K/L	$-M_{zA}/3 + M_{zB}/6$	$M_{yA}/3 - M_{yB}/6$	0	0	$-M_{yA}/L$	$-M_{zA}/L$	K/L	$-M_{zA}/6 - M_{zB}/6$	$M_{yA}/6 + M_{yB}/6$	
$-M_{yA}/L$	M_{xB}/L	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	$-M_{zA}/3 + M_{zB}/6$	$2F_{xB}/15 + 4F_{xB}I_y/AL$	0	M_{yA}/L	$-M_{zA}/L$	$-M_{xB}/L$	$F_{xB}/10 + 6F_{xB}I_y/AL^2$	$-M_{zA}/6 - M_{zB}/6$	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	$M_{zB}/2$	
$-M_{zA}/L$	$F_{xB}/10 + 6F_{xB}I_y/AL^2$	M_{xB}/L	$M_{yA}/3 - M_{yB}/6$	0	$2F_{xB}/15 + 4F_{xB}I_y/AL$	M_{zA}/L	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	$-M_{zA}/L$	$-M_{xB}/L$	$M_{yA}/6 + M_{yB}/6$	$-M_{zA}/2$	$-F_{xB}/10 + 6F_{xB}I_y/AL^2$	
$-F_{xB}/L$	0	0	0	M_{yA}/L	M_{zA}/L	F_{xB}/L	0	0	0	0	M_{yB}/L	M_{zB}/L	
0	$-6F_{xB}/5L - 12F_{xB}I_y/AL^3$	0	$-M_{yA}/L$	$-M_{zA}/L$	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	0	$6F_{xB}/5L + 12F_{xB}I_y/AL^3$	0	0	$-M_{yB}/L$	M_{zB}/L	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	
0	0	$-6F_{xB}/5L - 12F_{xB}I_y/AL^3$	$-M_{zA}/L$	$F_{xB}/10 + 6F_{xB}I_y/AL^2$	$-M_{zA}/6 - M_{zB}/6$	0	0	0	$6F_{xB}/5L + 12F_{xB}I_y/AL^3$	$-M_{zB}/L$	$F_{xB}/10 + 6F_{xB}I_y/AL^2$	M_{zB}/L	
0	M_{yB}/L	M_{zB}/L	K/L	$-M_{zA}/6 - M_{zB}/6$	$M_{yA}/6 + M_{yB}/6$	0	0	$-M_{yB}/L$	$-M_{zB}/L$	K/L	$-M_{zA}/6 - M_{zB}/6$	$-M_{yA}/6 + M_{yB}/6$	
$-M_{yB}/L$	$-M_{zB}/L$	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	$-M_{zA}/6 - M_{zB}/6$	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	$-M_{zA}/2$	M_{yB}/L	M_{zB}/L	M_{xB}/L	$F_{xB}/10 + 6F_{xB}I_y/AL^2$	$M_{zA}/6 - M_{zB}/6$	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	0	
$-M_{zB}/L$	$F_{xB}/10 + 6F_{xB}I_y/AL^2$	$-M_{zB}/L$	$M_{yA}/6 + M_{yB}/6$	$M_{zB}/2$	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	M_{zB}/L	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	$-F_{xB}/10 - 6F_{xB}I_y/AL^2$	M_{zB}/L	$-M_{yA}/6 - M_{yB}/6$	0	$2F_{xB}/15 + 4F_{xB}I_y/AL$	

Figure 2.5: Three dimensional geometric stiffness matrix of a beam-column element (including contribution of finite rotations).

F_y = axial yield load

M_{yp} = plastic bending moment about y-axis

M_{zp} = plastic bending moment about z-axis

The reduction in stiffness of the structure associated with the formation of plastic hinges is made through the plastic reduction matrix, $[K_p]$. The equilibrium equations are based on Drucker's normality rule which states that the plastic deformations are normal to the yield surface and orthogonal to the element force increment. From this, the incremental plastic displacement vector can be defined as:

$$\{ \Delta q_{p_i} \} = \{ G_i \} \lambda_i \quad (2.10)$$

where, $\{ \Delta q_{p_i} \}$ = incremental plastic displacement vector at node i

$\{ G_i \}$ = gradient vector of yield surface at node i

λ_i = plastic deformation magnitude at node i

The gradient vector is given by:

$$\{ G_i \} = \left[\frac{\partial \phi}{\partial F_{i_1}} \quad \frac{\partial \phi}{\partial F_{i_2}} \quad \cdots \quad \frac{\partial \phi}{\partial F_{i_n}} \right]^T \quad (2.11)$$

where, ϕ = yield surface function

F_i = element nodal force at node i

For the case where both ends of the element yield:

$$\{ \Delta q_p \} = \begin{Bmatrix} \Delta q_{p,i} \\ \Delta q_{p,j} \end{Bmatrix} = \begin{bmatrix} \{ G_i \} & \{ 0 \} \\ \{ 0 \} & \{ G_j \} \end{bmatrix} \begin{Bmatrix} \lambda_i \\ \lambda_j \end{Bmatrix} = [G] \{ \lambda \} \quad (2.12)$$

where, $\{ \Delta q_p \}$ = incremental plastic displacement vector

$\{ \Delta q_{p,j} \}$ = incremental plastic displacement vector at node j

$[G]$ = gradient vector of yield surface

$\{ G_j \}$ = gradient vector of yield surface at node j

$\{ \lambda \}$ = plastic deformation vector

λ_j = plastic deformation magnitude at node j

Through matrix manipulation, the plastic deformation vector becomes,

$$\{ \lambda \} = \left[[G]^T [K_e] [G] \right]^{-1} [G]^T [K_e] \{ \Delta q \} \quad (2.13)$$

where, $[K_t] = [K_e] + [K_p]$, tangent stiffness matrix

and the element equilibrium equation becomes,

$$\{\Delta F\} = [K_t]\{\Delta q\} - [K_t][G][G]^T[K_t][G]]^{-1}[G]^T[K_t]\{\Delta q\} \quad (2.14)$$

This can be expressed as:

$$\{\Delta F\} = \left[[K_e] + [K_p] + [K_p] \right] \{\Delta q\} \quad (2.15)$$

where, $[K_p] =$ plastic reduction matrix

This analysis does not account for the degradation in stiffness that is associated with the spread of yielding and leads to an overestimation in the stiffness of the structure. Note; (a) for a first order elastic-plastic analysis, $[K_t] = [K_e]$, and (b) the effect of residual stresses can be approximated through the use of an effective tangent modulus which is incorporated in the elastic stiffness matrix when p is greater than or equal to 0.5 and is defined by:

$$E_t = 4E[p(1-p)] \quad \text{for } p \geq 0.5 \quad (2.16)$$

where, E_t = tangent modulus

p = P / P_y

P = axial load

P_y = axial yield load

The quasi-plastic hinge analysis given in CU-STAND 4.0, Attalla et al. (1994), more closely approximates the actual behaviour as the solution gives comparable results to the plastic zone analysis. The corresponding matrix formulation for a second order elastic-plastic analysis has the following form:

$$\{ \Delta F \} = \left[[K_e] + [K_{ep}] \right] \{ \Delta q \} \quad (2.17)$$

where, $[K_{ep}]$ = elastic-plastic stiffness matrix

The elastic-plastic stiffness matrix was developed from flexibility coefficients that were generated from an inelastic cross section model and a yield surface. The inelastic cross section model contains two independent parameters, for major and minor axis bending, that are a function of the normalized axial load and normalized moments. The parameters are obtained from calibration with force-strain data from a fiber element analysis program. The yield surface is of the same form as in Equation 2.9 with the exception that the

constants are generated by curve fitting to data produced by the fiber element analysis program.

Note, the geometric stiffness matrix, $[K_g]$, is based on an elastic slope that is different from that of the elastic-plastic stiffness matrix $[K_{ep}]$. This results in some approximations that were found to be in good agreement with those of existing benchmark problems, Attalla et al. (1994).

2.5 Advanced Analysis

A review was conducted of the following group of selected commercial programs: DAST, RISA 3D, ROBOT V6, ETABS v6.0 and GT STRUDL. In general, these programs provide a second order elastic analysis which is limited to the formation of the first plastic hinge denoted by the load intensity P_{L2} , in Figure 2.1. ROBOT V6 and GT STRUDL provide the capability of modeling out-of-plane stability if each member was discretized with shell elements. Material nonlinear effects are not included in any of these programs. The research development programs, CU-STAND 4.0, McGuire et al. (1989), and NIFA, Clarke and Zablotskii (1995), provide a second order elastic-plastic analysis where the members are defined by beam-column line elements and brick elements respectively. Analytical algorithms to describe the behaviour of members subject to out-of-plane stability problems are currently being developed.

Table 2.2 presents a list of the physical attributes and behavioural effects that should be included in an advanced analysis program for steel framed structures. The

physical attributes include initial imperfections, member characteristics, shape characteristics, boundary conditions and connection details whereas the behavioural effects include geometric and material effects. Programs that provide a second order elastic analysis generally include the geometric effects listed in Table 2.2 with the exception of stability effects and connection response. The material effects are incorporated with this model to provide the ability to conduct a second order inelastic analysis for CU-STAND 4.0 and NIFA. An advanced analysis capable of modeling the true behaviour should include the geometric effects, material effects, stability effects and connection responses listed in Table 2.2. These analyses should also include the capability of applying nonproportional loading.

The viability of advanced analysis methods depends upon the time required versus the time available to complete the analysis/design process and the accuracy required to predict the behaviour of the structure. The time required includes the time to generate the structure definition, to perform the analytical computations, to interpret the results and the time to design the members. It is implicit that as part of an advanced analysis program are algorithms to input and review the data required and to size and resize members. Hence for the specified loads, the advanced analysis program provides a satisfactory design. For simple structures, advanced analysis programs are not warranted unless they can be completed within the same time frame as the current analysis/design process. For complex structures, which are characterized by irregular geometry, multiple framing systems, elaborate loading schemes and by restrictive requirements on behaviour under service load

Physical Attributes	Behavioural Effects	
	Geometric Effects	Material Effects
<ul style="list-style-type: none"> • initial imperfections <ul style="list-style-type: none"> • sweep • camber • initial twist • out-of-plumbness • member characteristics <ul style="list-style-type: none"> • prismatic, nonprismatic • built-up shapes, rolled, fabricated • finite member size • shape characteristics <ul style="list-style-type: none"> • open or closed cross section • doubly symmetric • singly symmetric • asymmetric • boundary conditions <ul style="list-style-type: none"> • torsional and lateral restraints <ul style="list-style-type: none"> • top flange • shear center • bottom flange • initial strains (eg. environmental thermal loading, lack of fit) • connection details <ul style="list-style-type: none"> • flexible, semi-rigid, rigid • finite joint size 	<ul style="list-style-type: none"> • P-δ (member instability effect) • P-Δ (frame instability effect) • curvature shortening • sway shortening • shearing deformations in members • Stability effects <ul style="list-style-type: none"> • local plate buckling¹ • lateral-torsional buckling¹ <ul style="list-style-type: none"> • web distortions • applicable to doubly symmetric open cross section shapes • flexural-torsional buckling¹ <ul style="list-style-type: none"> • applicable to singly symmetric and asymmetric open cross section shapes • Connection response <ul style="list-style-type: none"> • panel zone deformations • connection deformations 	<ul style="list-style-type: none"> • nonlinear stress-strain relationship including strain hardening and elastic unloading at plastic hinges or in inelastic zones <ul style="list-style-type: none"> • Ramberg-Osgood • spread of inelastic zone in members • residual stresses • inelastic interaction of axial force, biaxial bending, shear and torsion

Note: ¹ It is implicit that all buckling modes may be elastic or inelastic.

Table 2.2: Physical attributes and behavioural effects of steel framed structures.

conditions, advanced analysis programs are necessary. To predict the behaviour of a structure within an acceptable time span (computationally efficient), complex beam-column elements provide an advantage over shell elements as the number of elements required to model the structure is significantly smaller. These elements have limitations which may restrict their use in predicting out-of-plane stability behaviour.

Unless the time to complete the analysis/design process is made more competitive and manageable through advances in computing technology, the common use of advanced analysis will be limited by the ability of the owner to bear the cost of structural engineering services in performing this type of analysis.

Chapter 3

Interrelationship and Description of LOADS 95 and CU-STAND 4.0

3.1 Introduction

A flow chart of the design process, described in Table 1.1, and the corresponding main menu page for the Integrated System for the Analysis, Design, Fabrication and Erection of Steel Buildings are illustrated in Figures 3.1 and 3.2 respectively.

LOADS 95 is an interactive computer graphics program developed by Chmelicek (1993, 1996) that defines the geometry of a structure and the loads acting on a steel structure in accordance with the National Building Code of Canada 1990, NRCC 1990. This program was written using the "C" programming language with the corresponding graphics in Hoops Graphics System. It represents the first step in the design process as illustrated in Figure 3.1. The program is activated upon selecting the "Structural Geometry and Load Determination" button shown in Figure 3.2 which displays the title screen with the main menu as illustrated in Figure 3.3. Currently, algorithms for the preliminary sizing and resizing of members are not available and are

Figure 3.1: Model of the design process.

Figure 3.2: Main menu page for the Integrated System for the Analysis, Design, Fabrication and Erection of Steel Buildings.

CLIMATE DATA	WIND LOAD	DEAD LOAD
DEAD LOAD	SNOW OR RAIN	EARTHQUAKE

May 4, 1997 - 14:15:50

LOADS 95

LOAD DETERMINATION FOR STRUCTURAL STEEL BUILDINGS

CARLETON UNIVERSITY FACULTY OF ENGINEERING

VERSION 1.0, MAY 1993 - APRIL 1998

DEVELOPED BY: JAN CHMELICEK

SUPERVISOR: Dr. STEPHEN J. KENNEDY

☒ National Building Code of Canada 1990
☐ National Building Code of Canada 1995
☐ BOCA - Building Officials and Administrators
☐ SBC - Southern Building Code
☐ UBC - Uniform Building Code
☐

Figure 3.3: LOADS 95 title screen.

defaulted for beams, columns and for bracing members as W530x101, W310x118 and HSS102x102x9.5, respectively.

CU-STAND 4.0 is an interactive computer graphics program, developed at Cornell University, that is capable of conducting second order inelastic analyses that include both geometric and material nonlinear effects. The program is written using the Fortran programming language with the corresponding graphics in Hoops Graphics System. It is activated upon selecting the “Analysis” button shown in Figure 3.2 which displays the title screen as illustrated in Figure 3.4. CU-STAND 4.0 is classified as an advanced analysis program with limited capability. Advanced analysis programs of this type or those that evolve from this will eventually replace the traditional analysis/design approach illustrated in Figure 3.1.

To integrate these programs, the data generated by LOADS 95 must be interpreted, formatted and regenerated as input for CU-STAND 4.0. The data relates to the geometry of the primary members of the structure and the loads that are imposed on them. Secondary members are designed independently and need not be included. Members may need to be redefined into subelements that reflect the limitations of beam-column elements to model the application of complex loads. The more complex the analysis, the more data that must be defined to describe each member, the loads acting on that member and the support or restraint provided to that member. Shell elements fall into this category as the definition of boundary conditions, location of restraints, description of loading, discretization of member, and the inclusion of imperfections and residual stresses, requires the generation of complex database structures. It is therefore apparent that database structures for the structural

Figure 3.4: CU-STAND 4.0 title screen.

geometry/load determination and analysis process have different characteristics and requirements. Database translators are therefore necessary to ensure all details relevant to the analysis are translated to accommodate its database structure.

Through the integration of these programs comes an understanding of the format of the database structure for an ideal advanced analysis program. For this particular case, the definition of members by LOADS 95, defines the member length as that length between centerline dimensions of supporting elements and may have any load pattern acting on it. Sophisticated beam-column elements such as those described in CU-STAND 4.0 can only handle simple load cases but not multiple load vectors. As a result, the integration process requires that the members defined by LOADS 95 be redefined into subelements that reflect the limitation of the beam-column elements in CU-STAND 4.0.

The following sections describe the database structure for each program. It is apparent that LOADS 95 and CU-STAND 4.0 have unique requirements regarding the data that must be presented and its format within the database structure. Specific algorithms have to be developed to translate the data to simplify the communication between both programs. Chapter 4 describes the algorithms required for database translation and the redefinition of the structure and the loads to be compatible with the analysis program.

3.2 Database Structure for LOADS 95

LOADS 95 creates a directory with the project description name within the subdirectory `program_control`. The data generated by LOADS 95 for the specified building is housed in a series of directories and files within the building directory and are described in Table 3.1. Of these files, CU-STAND 4.0, which uses sophisticated beam-column elements, only requires information from the `members` file and `loadcomb` file. If shell elements are used to define the structural members and more sophisticated analytical programs are incorporated which use shell elements then additional information generated by LOADS 95 regarding the geometry of standard connection details and bracing details would be required. Obviously in the redesign and reanalysis phase, information from the analysis program would be returned to the loads program and the program re-executed to generate updated geometry and loads.

3.3 Database Structure for Members

This datafile contains the geometric and load data for every member within the structure. At the beginning of the “members” file are three values which specify the maximum identification number of the created member, the maximum identification number of the connection or support and the default column base plate elevation. Subsequently, the geometric and load data for each member is presented in succession.

Directory	General Description of Files
curvedsf	Data relating to curved surfaces
earthqak	Detailed data for earthquake load calculations
snow	Detailed data for snow load calculations
wind	Detailed data for wind load calculations
File	Description of File
ERRORS_ECHO	List of error messages
bays_def	Definition of bays
bays_lib	Library of existing bay definitions
dl_sandw	Default dead load intensities for structural sandwich systems for roof, floor and wall surfaces
eart_def	Default values for earthquake load calculations
gridgrps	Definition of gridline groups
ll_areas	Live load areas and corresponding load intensities
loadcomb	Load combination table
loadcomb.wdd	Wind directions
location	Geographic location and climatic data
members	Member geometry and load data
meshpac	Mesh spacing
nodes	x, y and z coordinates that define a surface
snow_def	Default values for snow load calculations
surfaces	Surface data
wind_def	Default values for wind load calculations
standard connection details	Yet to be developed
bracing details	Yet to be developed

Table 3.1: General database structure for LOADS 95.

3.3.1 Geometric Data Structure for a Member

The geometric data generated for a typical beam member, which contains general data, member mark, surface data, coordinate data, supported member connection data, support data, cross-section data and tributary area data, is shown in Figure 3.5. The general data consists of: i) a member identification number that increases by one for each member read, ii) a code number defining the type of member given in Table 3.2 (the corresponding code letter are also given), iii) the extract flag that has a value of 1 if the member is not to be considered for the advanced analysis and a value of 0 otherwise. These are generally defined for the member types through a pop-up application window illustrated in Figure 3.6. The member mark given on the structural and erection drawings and used by the fabricator is given next. The letter refers to the member type and the number is equal to the member identification number in the general data. An example is illustrated in Figure 3.7 where the marks for beams and columns for a roof surface are shown. The code letter for each member type is shown in Table 3.2. Currently, LOADS 95 is unable to define hangers, fictitious members, shear walls or virtual members, hence their code letters are not currently available. The surface data identifies the number of surfaces associated with the member and the surface number for each of those. Surfaces are defined as being either roofs, walls or floors. The first line of the coordinate data specifies the location of the ends of the member. The second line specifies the offset of each end from the surface

Figure 3.5: Geometric data for a typical beam member in LOADS 95.

[illegible]

Figure 3.6: Table of member types to be extracted from LOADS 95.

Member Type	Code Number	Code Letter
Footing	1	F
Column	2	C
Primary Beam	3	B
Secondary Member	4	S
Bracing	5	R
Girt	6	G
Hanger	7	-
Sag Rod	8	D
Cladding	9	L
Other Support	10	P
Fictitious	11	-
Shear Wall	12	-
Virtual Member	13	-

Table 3.2: Member types with corresponding code numbers and letters in LOADS 95.

plane in which the members lie. These offsets are not applicable to columns or footings. The next group of data identifies the number of members supported by the beam, the connection number, the corresponding identification number of the supporting member, the location of the connection, local gridline members, offsets of the supported member from that gridline and the fixity (0 for hinge connection and 1 for a rigid connection). Example gridlines are shown on a plan view of a roof surface illustrated in Figure 3.7. The support data describes the number of the supports, the connection identification number and the member identification number of the supporting member. Cross sectional data includes a section code number for the section type specified in Table 3.3 (default value is 1 representing a rolled shape), the orientation of the major principal axis with respect to the plane in which the member lies, and the default sections specified in Table 3.4.

Section type	Section Code Number
Unspecified Shape	0
Rolled Shape	1
Built Up Shape	2
Truss	3
Other	4

Table 3.3: Section code numbers in LOADS 95.

Member Type	Default Sections
Beams	W530x101
Columns	W310x118
Bracing	HSS102x102x9.5
Secondary	W310x79

Table 3.4: Default sections in LOADS 95.

The tributary area data defines the number of tributary areas of surfaces associated with the member, the corresponding surface and bay identification members, a flag indicating the load status for gravity and lateral loads (a value of 1 indicates that the load type can be applied on the area, while a value of 0 indicates that the load type is not applicable), and data representing the definition of the tributary area.

3.3.2 Load Data Structure for a Member

The load data generated for a typical member as illustrated by the blocks of data in Figure 3.8 contains uniform, uniformly varying and concentrated load data for each source of load and all of the corresponding load cases within that load.

The blocks of data labeled “a” and “b” represents code numbers which describe the source of load (dead, live, snow, wind, etc.), the surface upon which the load acts, the load case, the type of load (uniform, concentrated, etc.), and the load intensities and their locations respectively. The first code number in block “a” gives the code number for the source of load for those loads defined in Table 3.5. Currently, LOADS 95 only attributes the dead, live, snow and wind loads calculated to the members. Earthquake loads are calculated for the structure but not attributed to the lateral load resisting system.

Source of Load	Code Number
Dead Load	3
Live Load	4
Snow Load	5
Wind Load	6
Earthquake Load	7
Temperature Load	8
Shrinkage Load	9
Settlement Load	10
Other Load	11

Table 3.5: Source of loads in LOADS 95.

The second code number is the surface identification number of the surface whose loads are attributed to that member.

The third code number specifies the load case number for the given source of load. For dead loads, there exists only one load case, hence the code number is assigned a value of 1. For live loads, the code numbers given in Table 3.6 represent the live load reduction factor type. For snow loads, the code number represents the load case for the snow load distributions described in NBCC 1990 for that structure. There are three load cases, values of 1, 2 or 3 for figure H-1 (gable roofs) and figure H-3 (valley roofs). There are five load cases, values of 1, 2, 3, 4 or 5 for figure H-2 (curved roof surfaces). Figures H-4, H-5 and H-6 have only one snow load case therefore a value of 1 is assigned.

Type of Live Load Reduction Factor (LLRF)	Code Number
LLRF_A (Clause 4.1.6.9(1) in NBCC 1990)	1
LLRF_B (Clause 4.1.6.9(2) in NBCC 1990)	2
LLRF_NoReduction (No live load reduction applicable)	3

Table 3.6: Live load reduction factors in LOADS 95.

Figure 3.8: Load data for a typical member in LOADS 95.

The definition of load cases for wind is more complex than the others as the action of the wind on the structure or a part of the structure and the direction of that wind with respect to any surface must be included. The number appears as a two digit number in which the first number describes the wind load action and the second the direction of the wind with respect to the surface. These are described in Table 3.7 and 3.8. For example, a value of 11 indicates that pressure coefficients are based on primary structural actions arising from wind loads acting on the windward wall. The wind load cases are defined by the figures contained in Commentary B of the NBCC 1990. The pressure coefficients for primary action strength and serviceability are given in Figure B-7 while those for the structural member strength, structural member serviceability, cladding and local suction are found in Figures B8 to B10 and Figure B12.

The fourth code number defines the type of applied loads and is given in Table 3.9.

Wind Load Action	Code Number
Primary action strength	10
Primary action serviceability	20
Secondary member strength	30
Secondary member serviceability	40
Cladding	50
Local suction	60
Net on building	70

Table 3.7: Wind load cases in LOADS 95.

Wind Load Direction	Code Number
Windward	1
Leeward	2
Parallel1	3
Parallel2	4

Table 3.8: Wind load directions in LOADS 95.

Applied Load	Code Number
Linearly Varying Load	1
Point Load	2
Moment Load	3
Support Reaction	4
Support Reaction Moment	5
Compensating Force	6

Table 3.9: Applied loads in LOADS 95.

In the next set of data, block “b”, uniform or uniformly varying loads are described by the starting point of the load relative to the member end, the x, y and z load intensity values, the ending point of the load and its corresponding x, y and z load intensity values in global coordinates. For all forms of concentrated loads (code numbers 2 to 6), the location of the load and its magnitude in global three dimensional space are defined. Within the live load data set, the last value indicated by block “c” in Figure 3.8 gives the tributary area for that member in square meters.

3.3.2.1 Gravity Loads

Gravity loads computed by the program include dead loads, live loads due to use and occupancy, compensating forces and live loads due to snow.

a) Dead Loads

The dead load is determined according to Clause 4.1.5 of the NBCC 1990.

The components of the dead load intensity can be created or edited using the existing libraries within the program and is attributed to each member through its tributary area.

b) Live Loads due to Use and Occupancy, Compensating Forces

The live load due to use and occupancy is determined according to Clause 4.1.6 of the NBCC 1990. Live load reduction factors are computed according to the use and occupancy and the corresponding tributary area. Compensating forces that account for the live load reduction factors are computed for the accumulated column tributary area and applied at the beam-column joints, Ziemian et al. (1992).

c) Live Loads due to Snow

Live loads due to snow, ice and rain are determined according to Clause 4.1.7 of the NBCC 1990 and snow load data values found in the Supplement.

3.3.2.2 Lateral Loads

Lateral loads computed by the program include wind and earthquake loads.

a) Wind Loads

Wind loads are determined according to Clause 4.1.8 of the NBCC 1990. The internal and external pressure coefficients are taken from the appropriate figures given in Commentary B of the Supplement to the NBCC 1990.

b) Earthquake Loads

The lateral force at each horizontal roof and floor level is calculated based on the equivalent seismic shear force in accordance with Clause 4.1.9 of the NBCC 1990. The overturning moments and twisting moment are also calculated but requires the input of the design eccentricity. Distribution of earthquake forces to lateral load resisting members is not done currently as the center of rigidity is not yet computed by the program.

3.4 Database Structure for CU-STAND 4.0

Three programs for structural analysis and design, CU-PREPF, CU-STAND 4.0 and CU-QUAND were developed at Cornell University, McGuire et al. (1989). CU-PREPF is a preprocessor for CU-STAND 4.0 and CU-QUAND that allows the user to generate the geometry and to input the loads acting on the members for multiple load combinations and other parameters for analysis. CU-STAND 4.0 and CU-QUAND are static and earthquake analysis and design programs for steel structures. For each structure definition, the data created by CU-PREPF is stored in a binary data file with the extension “.des”. Only part of the data file “.des” generated

by CU-PREPF is required for CU-STAND 4.0 and is described in the following sections under the categories geometric data and load data.

3.4.1 Geometric Data

The geometric data generated in CU-PREPF can be classified into the sections listed in Table 3.10 and those pertaining to CU-STAND 4.0 will be described in the following paragraphs.

The first section of data refers to general analysis parameters and it consists of an array of 30 elements as shown in Table 3.11. Array elements 4 through 10, 15 through 20, 22 through 26, 29 and 30 are not used by CU-STAND 4.0. The others, lightly shaded, represent geometric data that is read by CU-STAND 4.0.

The third and fourth sections, given in Table 3.10, consist of a one dimensional array of six elements containing the maximum absolute displacements and rotations in the x, y and z direction and maximum absolute displacements and rotations per increment in the x, y and z direction respectively. This data is used to limit the total displacements and displacements per increment for a structure within an analysis or load step respectively.

The fifth section of data consists of a one dimensional array of 30 elements entitled float analysis parameters of which only the eighth element is read by CU-STAND 4.0. It represents the global space size of the window that displays the

Section	Description	Required by CU-STAND 4.0
1	General analysis parameters	X
2	Integer analysis control parameters	
3	Maximum absolute displacements and rotations	X
4	Maximum absolute displacements and rotations per increment	X
5	Float analysis parameters	X
6	Earthquake acceleration records	
7	Earthquake response spectrum curves	
8	Nodal coordinates	X
9	Element direction cosines	X
10	Boundary conditions	X
11	Floor number corresponding to each node	X
12	Master node number of each floor	X
13	Load or mass vectors	
14	Initial displacements	
15	Initial velocities	
16	Material properties	X
17	Element identification array	X
18	Section properties	X
19	Material property number	
20	Section no. in AISC section database	X
21	Yield surface type number	

Table 3.10: Description of geometric data generated by CU-PREPF.

Array Element	General Analysis Parameters
1	Number of nodes
2	Number of supported nodes
3	Number of floors
4,5,6	Number of data points describing the earthquake accelerations in the x, y and z directions
7	Number of modes
8,9,10	Number of data points describing the earthquake response spectrum curves for the x, y and z directions
11	Number of material types: E and Poisson's ratio
12	Number of beam-column elements
13	Number of k-vectors, defines 3-D element orientation
14	Number of member cross-section groups for design
15	Number of element fixed-end forces
16	Number of element yield surfaces
17	Number of element end eccentricities
18,19,20	Number of grids in the x, y and z directions
21	Size of the load description array, LOADDES
22	Reserved for future operations
23	Reserved for future operations
24	Number of items in earthquake description array, EARTHQ_F
25	Number of nodal points in the frame data representation
26	Number of lines in the frame data representation
27	Number of user defined sections + 1
28	Warping boundary conditions flag at member ends
29	Number of time steps for transient dynamic analysis
30	Number of time steps to save from a transient dynamic analysis—for playback purposes

Table 3.11: General analysis parameters in CU-STAND 4.0.

structural image and must be larger than the largest dimension of the structure in millimeters.

The eighth section of data consists of a two dimensional array with dimensions 3 and a number equal to the number of nodes in the structure. For each node, the x, y and z coordinates are defined.

The ninth section of data consists of a two dimensional array with dimensions 3 and a number equal to the number of beam-column elements. For each beam-column element, the x, y and z direction cosines are defined. Note that there may be more than one beam-column element per member as required for the definition of loads. A member is defined as one that spans from column to column or floor level to floor level.

The tenth section of data consists of a two dimensional array with dimensions 6 and a number equal to the number of nodes in the structure. For each node, the fixity conditions for displacements and rotations in the x, y and z direction are defined.

The eleventh section of data consists of a one dimensional array of size equal to the number of nodes in the structure and contains the floor numbers corresponding to each node. If the value is 0, the node is not constrained to any floor, otherwise, the node is constrained such that the floor deflects as a rigid diaphragm in the plane of the floor. The twelfth section of data consists of a one dimensional array of size equal to the number of floors in the structure and contains the master node number for each floor. In both sections, a default value of 0 was used to reflect that no rigid diaphragm action was considered. The definition of a rigid floor is performed in CU-PREPF and since LOADS 95 is not capable of defining rigid floors, default values of 0 were used.

The sixteenth section of data consists of a two dimensional array with dimensions 6 and a number equal to the number of different materials used in the structure. For each material, the modulus of elasticity, 200 000 MPa, Poisson's ratio, 0.3, the mass density, 7850 kg/m³, and three values reserved for future operations and assigned an arbitrary value of 0, are defined.

The seventeenth section of data consists of a two dimensional element identification array with dimensions 8 and a number equal to the number of beam-column elements in the structure. For each beam-column element, the eight element properties described are listed in Table 3.12. The k-vector number is simply the beam-column element number that has associated with it x, y and z direction cosines.

The eighteenth section of data consists of a two dimensional array of section properties for each section type with dimensions 20 and a number equal to the number of section types, ie. W530x101. The section properties that are defined for each section are listed in Table 3.13.

Array Element	Element Properties
1	node i
2	node j
3	k-vector number
4	material property number
5	group number
6,7,8	not used by CU_STAND4.0

Table 3.12: Element properties in CU-STAND 4.0.

Array Element	Section Properties
1	area
2	depth
3	flange width
4	flange thickness
5	web thickness
6	strong-axis moment of inertia
7	strong-axis section modulus
8	strong-axis radius of gyration
9	weak-axis moment of inertia
10	weak-axis section modulus
11	weak-axis radius of gyration
12	strong-axis plastic section modulus
13	weak-axis plastic section modulus
14	St. Venant torsional constant
15	Warping torsion constant
16	Yield stress
17	Null
18	Null
19	radius of gyration of compression flange region
20	ultimate tensile stress

Table 3.13: Section properties in CU-STAND 4.0.

The twentieth section of data is a one dimensional array with a dimension equal to the number of section types and contains the identification number associated with a particular section designation in the AISC section database. For the case of a user defined section, the identification number is assigned a negative value beginning at -1. The number of section types available in the database is limited to 218, a value that is used directly within CU-STAND 4.0. This database is rather primitive compared to modern approaches in database structure development.

3.4.2 Load Data

The load data defined for a structure is contained in a one dimensional array called **LOADDES** with the general structure illustrated in Figure 3.9.

Figure 3.9: Load data definition in CU-STAND 4.0.

The size of the array is not fixed and depends on the number of loads defined in the load data structure of CU-STAND 4.0. The first value is an index number which points to the location within the array for the next defined load. The second value by default is set to 1 for static loads. The third value specifies the source of load using the code numbers given in Table 3.14. The fourth value specifies the load type using

the code numbers given in Table 3.15. CU-STAND 4.0 has only been developed for concentrated loads at the nodes and uniform member loads acting along the entire element length. The subsequent set of values depends upon the load type and are described in Table 3.16.

Source of Load	Code Number
Dead	1
Live	2
Wind	3
Other1	4
Other2	5
Other3	6

Table 3.14: Source of loads in CU-STAND 4.0.

Load Type	Code Number
Self-weight (currently not used)	1
Nodal load (concentrated)	2
Uniform load	3

Table 3.15: Load types in CU-STAND 4.0.

Load Type	Description
Nodal Load	concentrated force in the x, y and z direction
	concentrated moment in the x, y and z direction
	number of nodes with this load
	node numbers of nodes with this load
Uniform Load	local uniformly distributed load in the x, y and z direction
	number of members with this uniformly distributed load
	member numbers of members with this load

Table 3.16: Description of load data for each load type.

Chapter 4

Integration of LOADS 95 and CU-STAND 4.0

4.1 Introduction

The following chapter provides a detailed description of the integration between the two programs LOADS 95 and CU-STAND 4.0 and concludes with a discussion of suggested improvements to the data structures of both.

The integrated system for the analysis, design, fabrication and erection of steel buildings has been conceived to emulate the design process through a series of programs which are activated by main menu buttons that represent the major steps of that process as listed in Table 1.1. The corresponding cover page, illustrated in Figure 3.2, consists of three regions. The top displays the main menu button, the middle the title, and the bottom, a list of developers as related to the design steps. The current status of the main menu buttons is listed in Table 4.1. An inactive status signifies that the existing programs under this category have yet to be integrated. To begin the process logically, the designer would define the structural geometry and then determine the loads acting on that structure. This is achieved by activating the corresponding button in the main menu

Main Menu Button	Current Status
STRUCTURAL GEOMETRY & LOAD DETERMINATION	initiates LOADS 95
MEMBER SIZING	no development
ANALYSIS	initiates CU-STAND 4.0
MEMBER DESIGN	inactive
CONNECTION DESIGN	inactive
DRAWINGS & SPECIFICATIONS	no development
FABRICATION	no development
ERECTION SIMULATOR	no development
QUIT	terminates the program

Table 4.1: Description of buttons within main menu page of the program.

and following the appropriate steps outlined in LOADS 95. Once this is completed, then the user is prompted for the filename for the project which must be re-entered. This process executes files which regenerates the geometric data developed by LOADS 95 into the appropriate format required by the modified version of CU-STAND 4.0. The analysis procedure is continued by activating the corresponding button. Within the modified version of CU-STAND 4.0, the user is guided through the appropriate steps where the geometric data is read, the analysis defined, the load combination table created, the load data is read and the analysis conducted. Note, the original version of CU-STAND 4.0 contained design algorithms developed for AISC-LRFD Specifications (1986). These were deleted and will be replaced by others that have been developed.

Specific files developed to integrate the two programs, shown in Figures 4.1a and 4.1b, and relevant data files generated for use during the integration process are described in detail in the following sections. Figure 4.1a provides the basic flowchart that illustrates

the integration of LOADS 95 and CU-STAND 4.0. The files for the highlighted subsections, generation of geometric data and generation of load data and regeneration of geometric data are given in Figure 4.1b. The files have been categorized as miscellaneous files to control program flow, geometric data related files and load data related files, and are presented in section 4.2.

4.2 General Description of Files for the Integration of LOADS 95 and CU-STAND 4.0

Tables 4.2, 4.3 and 4.4, give a general description of miscellaneous files to control program flow, geometric data related files and load data related files, respectively. Specific details for the load combination table, statically equivalent uniform load distribution and nodal renumbering scheme are given in section 4.3. These files are all written in the “C” programming language with the corresponding graphics in Hoops Graphics System, Wiegand et al. (1992). All programming was developed on a SPARC station ELC running SunOS 4.1.3.

Figure 4.1a: Flowchart of files created for the integration of LOADS 95 and CU-STAND 4.0.

Figure 4.1b: Files created for the generation of geometric and load data.

Miscellaneous File	Function
main.c	<ul style="list-style-type: none"> calls subroutines that initiate the program through the display of the main menu illustrated in Figure 3.2
input_control.c	<ul style="list-style-type: none"> based on the selection of any button within the main menu and initiates the appropriate action (message display or program execution)
cover_page.c	<ul style="list-style-type: none"> subroutines "build_upper_screen" and "create_button" creates the buttons for the main menu subroutine "build_middle_screen" creates the title of the program subroutine "build_lower_screen" creates a list of developers
display_message.c, display_message2.c, display_message3.c, display_message4.c, display_message5.c, display_message6.c	<ul style="list-style-type: none"> display appropriate messages to determine the preferred course of action
title.c	<ul style="list-style-type: none"> subroutine "title_file" returns the directory path of the data file "members" for the specified problem if it exists subroutine "make_directory" creates a directory path for the specified problem if it does not already exist
remove_files.c	<ul style="list-style-type: none"> subroutine "remove_files" removes intermediate data files used to manipulate the data and retains the files that provide relevant geometric and load data of the structure
create_des_file.c	<ul style="list-style-type: none"> calls a number of subroutines that generates the geometric data for the structure (refer to Figure 4.1b)
member_length.c	<ul style="list-style-type: none"> subroutine "member_length" computes the length of a member given the initial and final x, y and z coordinates of the member and returns the result
writefile.c	<ul style="list-style-type: none"> subroutine "writefile" calls a number of subroutines located in the file "des_write.c" to write the geometric and load data for a structure.
des_write.c	<ul style="list-style-type: none"> contains a number of subroutines that reads all relevant data files and extracts information to be written to an input file required by the analysis program. This data is described in Section 3.4.1 of Chapter 3.

Table 4.2: Miscellaneous files to control the program flow.

Miscellaneous File	Function
convert_point.c	<ul style="list-style-type: none"> subroutine "member_coord" converts a point expressed as a distance from the beginning of a member into global x, y and z coordinates and returns the result. Nodal points may need to be generated at these locations.
main_header.h	<ul style="list-style-type: none"> main header file for the subroutines found in directory "STEEL_PROGRAM/program_control/source". It contains a number of header files for standard C functions and definitions common to subroutines.
funct_pro.h	<ul style="list-style-type: none"> this header file contains function prototypes that define the existing functions in the program
master.h	<ul style="list-style-type: none"> master header file for the subroutines found in the directory "STEEL_PROGRAM/ANALYSIS/source". It contains a number of header files for standard C functions and definitions common to many of the subroutines
hp_macro.h (Chmelicek 1993)	<ul style="list-style-type: none"> this header file contains definitions of short aliases for Hoops routines used in the file load_comb_table.c

Table 4.2: Continued....

Geometric File	Function
read_mem_prop.c (Chmelicek 1993)	<ul style="list-style-type: none"> • subroutine "Read_MEMBERS" reads the "members" data file generated by LOADS 95 • subroutines "Free_Allocated_Memory_One_MEMBER" and "LOAD_Delete remove the memory taken up by the geometric and load data generated for a member • minor modifications were made to "Read_MEMBERS" to extract all the required geometric and load data
read_coord.c	<ul style="list-style-type: none"> • subroutine "read_coord" assigns node numbers to all coordinates extracted from the "members" data file • subroutine "insertlist" inserts a data pointer to the proper position within the list of nodal coordinates • subroutine "scan_list" assigns a new node number to the set of coordinates if it has not already been assigned
read_coord2.c	<ul style="list-style-type: none"> • subroutine "read_coord2" ensures that nodes are placed at all intersecting points of the primary members. It then generates a datafile of member numbers with corresponding node values and coordinates • subroutines "create_list_1" up to "create_list_5" arrange various node lists into an ascending format • subroutine "print_list" prints the nodal and coordinate information for each member into a data file
re_num.c	<ul style="list-style-type: none"> • subroutine "re_num" creates a data file of member numbers and the corresponding node numbers that are used as input for a renumbering subroutine
re_number.c (Gattass et al. 1994)	<ul style="list-style-type: none"> • subroutine "re_number" renumbers the nodes in a structure to reduce the bandwidth of sparse matrices. Refer to section 4.3.
write_member_type.c	<ul style="list-style-type: none"> • subroutine "write_member_type" generates a value at the end of each data line in data file "new_member_info" indicating whether the member is a beam, column or bracing member
write_nodes.c	<ul style="list-style-type: none"> • subroutine "write_nodes" generates a data file called "node_info" where the x, y and z coordinates for each node are written
write_fixity.c	<ul style="list-style-type: none"> • subroutine "write_fixity" generates a data file called "fixity_info" where the fixity conditions for each node are written

Table 4.3: Geometric data related files.

Geometric File	Function
write_data_size.c	<ul style="list-style-type: none"> • subroutine "write_data_size" computes a value that represents the global space size of the window that displays the structural image and must be larger than the largest dimension of the structure in millimeters
web_unit_vector.c	<ul style="list-style-type: none"> • subroutine "web_unit_vector" computes the x, y and z components of the web unit vector for each member and stores it in the data file "web_unit_vector". A web unit vector is illustrated in Figure 4.2
loadcomb.c	<ul style="list-style-type: none"> • subroutine "Create_Combination_Table" calls other subroutines to generate all load combinations and to display them in a load combination table • subroutine "ReadLoadCombinationsTable" is used to read the data file "loadcomb", that is generated by the LOADS 95 and contains all information required to develop the load combination table, Chmelicek (1993). Minor modifications were made • subroutine "CreateCombinations" creates the data file "load_combinations" that includes all combinations with both snow and wind loads and displays them in a load combination table within the analysis program. • subroutine "reduce_load_cases" eliminates any duplication of load combinations existing in the data file
update_data_files.c	<ul style="list-style-type: none"> • subroutine "update_data_files" calls a number of subroutines to regenerate the data files based upon the load combination selected by the user • subroutine "create_coord_file" creates a new version of the "coord" data file to incorporate new node data generated upon the selection of a load combination • subroutine "update_fixity_info" creates a new "fixity_info" data file to incorporate fixity conditions for the new nodes that are generated

Table 4.3: Continued....

Load File	Function
load_comb_table.c (Chmelicek 1993)	<ul style="list-style-type: none"> contains subroutines required to construct the load combination table. Refer to section 4.3
convert_to_local.c	<ul style="list-style-type: none"> subroutine "create_local_loads_file" opens data files required to read the load data and calls the subroutine "convert_to_local" to transform the global distributed load values to local distributed load values for the coordinate systems illustrated in Figure 4.3
load_comb_selection.c	<ul style="list-style-type: none"> subroutine "load_comb_selection" writes the factored loads for the load combination selected to a data file subroutines "write_loads_1" and "write_loads_2" write the data in the proper format
convert_to_uniform.c	<ul style="list-style-type: none"> contains subroutines required to convert linearly varying loads to uniformly distributed loads. Refer to section 4.3
store_loads.c	<ul style="list-style-type: none"> subroutines "rewrite_files", "rewrite_files2" and "rewrite_files3" reduce intermediate data files to a more compact form subroutine "uniform_loads" and "rewrite_uniform_loads" combine the uniformly distributed loads for each source of load and eliminates any duplication, respectively subroutine "concentrated_loads" and "rewrite_concentrated_loads" read the data and eliminates any duplication subroutine "create_load_array" creates the final array of load data in a format acceptable for reading by the analysis program

Table 4.4: Load data related files.

4.3 Description of Load Combination Table, Statically Equivalent Uniform Load Distributions and Nodal Renumbering Scheme

The following sections give an expanded description of the files

load_comb_table.c, convert_to_uniform.c and re_number.c.

Figure 4.2: Definition of a web unit vector.

Figure 4.3: Local and global coordinate system in CU-STAND 4.0.

4.3.1 load_comb_table.c

This file contains all subroutines required to construct the load combination table and to translate the data for CU-STAND 4.0. The load combination table, illustrated in Figure 4.4, replaces the existing table based on AISC-LRFD Specifications with the appropriate table that meets the requirements of the NBCC 1990. Load factors, importance factor, and a load combination factor for a given load combination must be defined by the user. If the load combination includes wind, then the user must define whether strength or serviceability is being investigated by selecting the corresponding button. Selection of the appropriate load combination from the table then executes a series of files which regenerates the load data developed by LOADS 95 into the appropriate format required by CU-STAND 4.0.

4.3.2 convert_to_uniform.c

The linear varying loads were converted to statically equivalent uniform loads to accommodate the limitations within CU-STAND 4.0. To be computationally efficient, the number of subelements was limited to two and if the maximum load intensity for a trapezoidal load was less than 1.2 times the lesser load intensity, then an average load intensity was used. Figure 4.5 illustrates the statically equivalent uniform load distribution for triangular shaped load distributions with varying boundary conditions. Note, that the division of the member into two equal lengths with the uniform load intensities prescribed, based on a parametric study, give the best correlation with the actual load distribution.

		Expd-2 CU - STAND 28-May-87	
		STRENGTH LIMIT STATE IS ON	

NBCC 1990 LOAD COMBINATIONS

Load Factor	Load Type	Importance Factor	Load Combination Factor
1.25	DEAD	1.00	0.70
1.50	LIVE		
1.50	SNOW		
1.50	WIND		
1.00	EARTHQUAKE		

LOAD COMBINATIONS

- D + L
- D + L + S1 + W1
- D + L + S1 + W2
- (Empty Row)
- D + L + S1 + W4
- D + L + S2 + W1
- D + L + S2 + W2
- D + L + S2 + W3
- D + L + S2 + W4
- D + L + S3 + W1

SELECT LOAD COMBINATION

Figure 4.4: Load combination table in modified version of CU-STAND 4.0.

The corresponding shear force and bending moment diagrams for the actual and approximate load distribution, are illustrated in Figure 4.6, with solid and dashed lines respectively. As shown by this figure and by the summary of the ratio of maximum values in Table 4.5, the approximations are within three percent and the distributions are essentially the same. The only exception is the ratio of deflections for the fixed-pinned case illustrated in Figure 4.5(c). Within the file “convert_to_uniform.c” the subroutine “subdivide” converts the loads, subdivides the member into two elements and locates a new node at mid-length.

End conditions	V_{approx}/V_{actual}	M_{approx}/M_{actual}	$\Delta_{approx}/\Delta_{actual}$
pinned-pinned	0.975	1.029	1.002
pinned-fixed	0.980	1.008	0.980
fixed-pinned	0.991	1.031	0.765
fixed-fixed	0.982	1.021	0.963

Table 4.5: Ratio of approximate to actual value of maximum shear forces, bending moments and deflections for various end conditions.

There are five other subroutines entitled, “determine_loads”, “add_loads”, “determine_new_nodes”, “determine_new_nodes_2” and “create_new_loads_file” that interpret, store and process data related to this task. The process is best described through a simple example of a typical beam loaded with drifting snow as illustrated in Figure 4.7(a). The load intensity at the vertices of the load intensity diagram are identified. For triangular shaped loads, ie. the load intensity at one end is zero, the load is converted to one of the statically equivalent uniform load distributions illustrated in Figure 4.5. For trapezoidal shaped loads the load intensities at either end are compared. For the

a) $w_1 = 0.20w$ $w_2 = 0.80w$

b) $w_1 = 0.20w$ $w_2 = 0.80w$

c) $w_1 = 0.35w$ $w_2 = 0.65w$

d) $w_1 = 0.20w$ $w_2 = 0.80w$

Figure 4.5: Statically equivalent uniform load distributions

Shear Force Diagrams

Bending Moment Diagrams

a) $w_1 = 0.20w$ $w_2 = 0.80w$ b) $w_1 = 0.20w$ $w_2 = 0.80w$ c) $w_1 = 0.35w$ $w_2 = 0.65w$ d) $w_1 = 0.20w$ $w_2 = 0.80w$

Figure 4.6: Shear force and bending moment diagrams for approximate and actual load distributions.

example in Figure 4.7, if the largest load intensity is less than 1.2 times the other, then the statically equivalent uniform load is the average load intensity over the length in which the trapezoidal load acts. If it is greater, then the triangular shaded portion illustrated in Figure 4.7(b) is converted into one of the statically equivalent uniform load distributions given in Figure 4.5 and added to the uniform load intensity, q_2 . The bending moment diagrams for the original load distribution and the statically equivalent uniform load distribution shown in Figure 4.7(c) are in good agreement. Similar results can be demonstrated for a comparison of deflected shapes and shear force diagrams. For CU-STAND 4.0 this beam is subdivided into three elements with the uniform loads shown.

4.3.3 re_number.c

Renumbers the nodes in the structure such that the bandwidth of a sparse matrix, $[A]$, is reduced enabling the solution of the matrix equation $[A]\{x\} = \{B\}$ to be solved efficiently, Gattass et al. (1994) and Gibbs et al. (1976). The matrix $[A]$ and vectors $\{B\}$ and $\{x\}$ are arbitrary. Graph representation of sparse matrices allows the visualization of the relationship between the permutation of rows and columns of a matrix and the renumbering of vertices of a graph. The algorithm by Gibbs et al. (1976) that permutes $[A]$ into a matrix which has a smaller bandwidth than $[A]$. Tests conducted by Gibbs et al. (1976) indicate that this algorithm yields bandwidths comparable to the commonly used Cuthill-McKee algorithm, Cuthill et al. (1969), but is more computationally efficient.

(a) Drifting snow load on a primary structural member.

(b) Statically equivalent uniform load distribution.

(c) Bending moment diagrams of original load distribution and statically equivalent uniform load distribution.

Figure 4.7: Drifting snow load case example.

4.4 Description of Data Files

A number of data files are generated throughout the integration process and those relevant to the problem definition are summarized in Table 4.6. Other data files were generated temporarily for data processing and subsequently deleted. Nodal data found in the file "node_info" is duplicated in the file "new_member_info" to facilitate data manipulation.

Data File	Function
node_info	<ul style="list-style-type: none"> lists the node number and the x, y and z coordinate for each node in the structure.
new_member_info	<ul style="list-style-type: none"> lists the element number, initial node number of the element, x, y and z coordinates of the initial node, final node number of the element and the x, y and z coordinates of the final node for each element in the structure.
fixity_info	<ul style="list-style-type: none"> lists the node number, x, y and z coordinates of the node and the fixity condition of the node (0 for a hinged connection and 1 for a moment connection)
uniform_loads	<ul style="list-style-type: none"> lists the uniformly distributed load data for each element in the structure.
concentrated_loads	<ul style="list-style-type: none"> lists the concentrated load data for each node in the structure.
web_unit_vectors	<ul style="list-style-type: none"> lists the element number and components of the web unit vector, w_x, w_y, w_z, that indicates the element orientation, for each element in the structure.
load_combinations	<ul style="list-style-type: none"> lists all the load combinations for a structure. Each data line consists of an equation that may contain the following symbols; "D" for dead loads, "L" for live loads, "S" for snow loads and "W" for wind loads. The value beside the "S" and "W" indicates the load case number under consideration. The load combination table displayed in the analysis program is illustrated in Figure 4.4.

Table 4.6: Data files generated for the integration process.

4.5 Suggested Improvements to LOADS 95 and CU-STAND 4.0

Throughout the integration process, shortcomings that exist and improvements that might be made to both LOADS 95 and CU-STAND 4.0 became apparent, and are summarized in the following sections in point form.

LOADS 95

- The connecting joints between members are well defined but nodal points should be assigned at these joint locations for subsequent use in the analysis stage. This may not be feasible during the creation of the structure but may have to be done as an intermediate step as it was done for the integration process.
- Construction joints should be specified where they are naturally expected to occur. The program should have the capacity to add construction joints where they are required.
- Should give the capability to subdivide members into any number of elements, where members and elements span between connections and nodes respectively.
- The database requires expansion to include connection types and any other relevant description.
- A member selection table should be included so that member sections can be assigned.
- A preliminary design algorithm should be included for the initial sizing of members.
- The orientation of the members in the structure should be defined.
- The ability to incorporate other codes like BOCA, SBC, UBC, ASCE7 or other versions of the NBCC for the definition of design loads.

- The ability to optimize the location of lateral load resisting elements.
- The rewriting of the program code into an object oriented programming language such as C++ to facilitate maintenance and further development.
- Earthquake load definition should be completed.
- Pattern load combinations should be considered where applicable.

CU-STAND 4.0

- A distinction between members and elements should be made.
- Linearly varying load distributions should be incorporated.
- CU-STAND 4.0 should be rewritten into an object oriented programming language such as C++. This will modularize the user-interface and application program codes, simplifying maintenance and the incorporation of future developments.
- Incorporate finite element code for discretizing members which are subject to out-of-plane stability problems (lateral torsional buckling).
- Should provide a consistent display for the graphical user interface components of the program.
- Incorporate nonproportional loading in an automated manner. Any source of load could vary from design conditions and may need to be amplified without changing the other loads to determine the load(s) to cause failure, as illustrated by the corresponding mock-up interface shown in Figure 4.8. All loads would be incremented proportionally up to the design load. Any load source with a maximum applied design load value greater than 1 or with the designation "ultimate" will be loaded proportionally up to the limit specified.

LOAD COMBINATION		PROPORTIONAL LOADING	
D + L		NONPROPORTIONAL LOADING	
D + L + S1 + W1		LOAD SOURCE	MAXIMUM APPLIED DESIGN LOAD
D + L + S1 + W2			
D + L + S1 + W3			
D + L + S1 + W4			
D + L + S2 + W1			
D + L + S2 + W2			
D + L + S2 + W3			
D + L + S2 + W4			
D + L + S3 + W1			
D + L + S3 + W2			

Figure 4.8: Interface display for nonproportional loading.

Chapter 5

Examples

5.1 Introduction

The versatility of an integrated program with advanced analysis algorithms will be demonstrated through three example problems. The first example, a two-dimensional rigid frame, demonstrates the use of advanced analytical tools and verifies that the modified integrated version of CU-STAND 4.0 produces results comparable to NIFA (Nonlinear Inelastic Frame Analysis), an existing finite element program, described in Chapter 6, Plastic-Zone Analysis of Frames by Murray J. Clarke, of Chen and Toma (1994). The second example, a lowrise three-dimensional rigid framed structure, demonstrates the advantages of integrating these tools with preprocessors such as LOADS 95 and illustrates that the geometric and load data (multiple load combinations) defined in LOADS 95 can be successfully interpreted by CU-STAND 4.0 and analyzed. The third example, a multi-storey steel structure, illustrates the versatility of the integrated program and its ability to define and analyze complex problems readily. Prior to the description of

the examples, a brief discussion of the program NIFA and the corresponding data input is presented.

Section 5.2 Data Input for Program NIFA

NIFA is a finite element analysis program that performs elastic and inelastic nonlinear analysis of two-dimensional frames. Geometric nonlinearities can be modeled using a total Lagrangian, updated Lagrangian or co-rotational formulation. A total Lagrangian formulation is valid for small or moderate displacements and rotations and the reference configuration corresponds to the original undeformed configuration. An updated Lagrangian formulation is valid for small, moderate or large displacements and rotations and the reference configuration corresponds to the last known equilibrium configuration. Note, the updated Lagrangian formulation is computationally more efficient than the total Lagrangian formulation. A co-rotational formulation is valid for small, moderate or large displacements and rotations and all static and kinematic variables are referred to an element coordinate system which moves continuously with the element throughout the analysis, but does not deform with it. Material nonlinearities are modeled by capturing the spread of yielding through the cross section depth and member length and is termed as a distributed plasticity approach. The stress-strain curve for the material can be defined elastic perfectly plastic, multi-linear or by the Ramberg-Osgood curve.

The nonlinear control data consists of parameters that control the incremental nonlinear solution of problems and are listed in Table 5.1. The last three parameters,

degree of freedom to be controlled, iteration strategy code and initial load increment strategy code have specified lists of options which can be selected from the appropriate pop-up menu. Guidance as to the appropriate selection of values and parameters can be found in Chapter 6, Plastic-Zone Analysis of Frames by Murray J. Clarke, of Chen and Toma (1994).

Nonlinear control data	Default Values	Values provided for Example 1
Starting load factor	0.4	0.2
Maximum load factor	3.0	2.0
Tolerance for convergence	0.0001	0.0001
Maximum number of load steps	20	50
Maximum number of iterations per load step	15	15
Desired number of iterations per load step to achieve convergence	4	4
Terminate the program upon reaching ultimate load (yes/no)	No	No
Joint number of the node used in displacement control	2	2
Degree of freedom to be controlled	x displacement	x displacement
Iteration strategy code	Rik's arc length constraint	Rik's arc length constraint
Initial load increment strategy code	Arc length control and desired no. of iterations	Arc length control and desired no. of iterations

Table 5.1: Nonlinear control data in NIFA.

The structural geometry is defined through a series of menus for joint definitions, members definitions, joint restraints and concentrated elastic restraints. Joint definitions

require joint numbers and the x and y coordinates of the centerline intersection of members. Member definitions require member numbers, connecting joint numbers, a member property number indicating the section shape and the number of subdivisions along the length of the member. Member sections can be defined as standard section shapes from existing libraries or as non-standard sections that are defined by specifying the cross section dimensions. The rotational degree of freedom can be defined for each joint (rigid or pinned). Joint restraints are defined for x and y displacement and in-plane rotation about the z axis are defined as free or fixed and pinned or rigid respectively. Concentrated elastic restraints require the number of the restrained node and the spring stiffness for translation in the x or y direction and in-plane rotation.

Other characteristics that may be defined are residual stresses, initial strains and structure imperfections (sway and initial out of straightness for any member). Residual stresses can be defined for both standard and non-standard cross section shapes. For standard section shapes, I-sections or square hollow sections, the residual stress patterns are predefined. For non-standard shapes or user defined shapes, the residual stress pattern must be determined and input. For initial strains, the user must input the member number and the corresponding initial membrane strain and the initial bending curvature. For sway imperfections, the user must define the base and top node of the member along with the angular tilt (sway) with respect to the y-axis of the member. Member imperfections are represented as a half-sine wave along the member whereby the users input the magnitude of the imperfection at midlength as a fraction of the member's length. Further details about program usage are found in the user's manual, Clarke and Zablotskii (1995).

Loads can be defined as body forces and load vectors. For body forces, the acceleration in the direction of the x and y axis along with the density of the member in the structure must be specified. Load vectors include concentrated loads applied at joints and distributed loads that can be applied normal or tangentially (axially) along the length of the member.

Section 5.3 Example 1 - Two Dimensional Rigid Frame

The two dimensional rigid frame example demonstrates the use of advanced analytical tools and verifies that the modified integrated version of CU-STAND 4.0 produces results comparable to NIFA. The example consists of a two-storey rectangular rigid frame that is used as a benchmark problem for advanced analysis, Chen and Toma (1994), and is illustrated in Figure 5.1. The applied loads, geometry, section shapes and material characteristics used for NIFA are also given on this figure. The total Lagrangian formulation was used to account for geometric nonlinearities and an elastic-perfectly plastic stress-strain curve was used to model the material behaviour. Members B2 and B4 were subdivided into 20 elements each while the remaining members were subdivided into 10 elements. The default cross section subdivision was used. The flange was divided into 4 sections through its thickness and 8 sections along its width while the web was divided into 4 sections through its thickness and 12 sections through its depth. Residual stresses, initial strains and structure imperfections were taken as zero.

To duplicate this problem within the integrated program (Integrated SYstem for Steel Structures, ISYSS), it was necessary to define the three dimensional frame configuration illustrated in Figure 5.2 within LOADS 95. The interior frame is highlighted with bold lines and represents the rectangular rigid frame that will be compared to the frame analyzed using NIFA. The member section definitions for the interior frame are the same as those for NIFA. All other members were specified as W200x19 to limit their contribution to the lateral resistance of the frame.

Columns:

C1 - W200x22
 C2 - W360x196
 C3 - W360x179
 C4 - W200x19
 C5 - W360x179
 C6 - W360x162

Beams:

B1 - W690x125
 B2 - W920x253
 B3 - W530x66
 B4 - W690x152

Figure 5.1: Definition of two dimensional rigid frame for NIFA.

Load Definition:

$$w_{D1} = 2.80 \text{ kN/m}^2 \times 5 \text{ m} = 14 \text{ kN/m}$$

$$w_{L1} = 7.20 \text{ kN/m}^2 \times 5 \text{ m} = 36 \text{ kN/m}$$

$$w_{D2} = 6.50 \text{ kN/m}^2 \times 5 \text{ m} = 32.5 \text{ kN/m}$$

$$w_{L2} = 15 \text{ kN/m}^2 \times 5 \text{ m} = 75 \text{ kN/m}$$

$$w_1 = 1.25w_{D1} + 1.5w_{L1} = 71.5 \text{ kN/m}$$

$$w_2 = 1.25w_{D2} + 1.5w_{L2} = 153 \text{ kN/m}$$

Figure 5.2: Rigid frame definition for ISYSS.

Distributed gravity loads were applied to the tributary area of the interior frame to give the same factored loads as those for NIFA. A plastic hinge model with similar material properties and an updated Lagrangian formulation were used in the analysis. All members were subdivided into 4 elements along their length. This was done manually by editing the appropriate data file.

A second order inelastic analysis was performed using both NIFA and ISYSS (modified CU-STAND 4.0). The results are summarized in Figures 5.3 and 5.4 by the applied load ratio vs. deflection graphs for node b and node t (as indicated in Figure 5.2), respectively. The applied load ratio represents the fraction of the total load applied on the structure where the loads acting on the structure may be defined as service or factored loads. Regardless of the source of the load, the program is limited to increasing all loads acting on the structure in proportion to the applied load ratio. For node b and t, the maximum applied load ratio achieved by NIFA and ISYSS was 0.982 and 1.007, and 0.982 and 1.005, respectively. The shapes of the curves are comparable and as expected are essentially the same as those given in Chen and Toma (1994) for NIFA and CU-STAND 4.0. The modeling of the frame in three dimensions, with the other members specified as W200x19, did not appear to affect the lateral stiffness or load ratio appreciably.

Figure 5.3: Applied load ratio vs. deflection curve for node b.

Figure 5.4: Applied load ratio vs. deflection curve for node t.

Section 5.4 Example 2 - Lowrise Three Dimensional Rigid Frame

The lowrise three dimensional rigid frame, shown in Figure 5.5, demonstrates the advantages of integrating these tools with preprocessors such as LOADS 95 and illustrates that the geometric and load data (multiple load combinations) defined in LOADS 95 can be successfully interpreted by CU-STAND 4.0 and analyzed. Beams and columns were specified as W530x101 and W310x118, respectively. The columns were oriented so that bending about the strong axis occurred about the smaller plan dimension of the building. The yield strength and modulus of elasticity were specified as 300 MPa and 200 000 MPa, respectively. The loads defined on the structure are summarized in Table 5.2.

	Dead Load (kN/m ²)	Live Load (kN/m ²)	Snow Load (kN/m ²)	Wind Load (kN/m ²)
Roof	1.20	1.00	S _s = 2.20 S _r = 0.40	
Floor	4.15	2.40		
Walls				q _{1/10} = 0.30 q _{1/30} = 0.37

Table 5.2: Load definitions for lowrise three dimensional rigid framed structure.

A series of elastic and inelastic analyses were conducted on the structure for two load combinations, one for each wind direction as illustrated in Figure 5.5. For the nonlinear analyses, the predictor corrector method was used in the solution.

(a) Plan view

(b) South elevation

Figure 5.5: Lowrise three dimensional frame.

The behaviour of the structure can be described by any characteristic load vs. displacement curve. These have been plotted for the x and y lateral displacement of node a for a first order elastic analysis (curve 1), second order elastic analysis (curve 2) and a second order inelastic analysis (curve 3) for load combination 1 and 2 in Figures 5.6 and 5.7, respectively. The service load level and drift limitations are illustrated on the figures. Figure 5.8 illustrates the plastic hinges developed for load combination 1 that are required to cause a collapse mechanism. The applied load ratios for the ultimate capacity of the structure were found to be 5.43 for load combination 1 and 2.55 for load combination 2. As indicated, the structure is both stiffer and stronger than required for the size of members selected. Obviously these would be resized to provide an economical structure that would meet the serviceability and ultimate states. Future versions of the program will include the capability of increasing one or more sources of load independently of the others (nonproportional loading). For example, increasing snow and wind while maintaining the dead and live loads due to use and occupancy. The integrated program, ISYSS, can model simple three dimensional structures (36 columns, 54 beams), determine the appropriate applied loads and load combinations, assigns loads to members and can conduct advanced analyses to give the true behaviour of the structure within a relatively short period of time.

Figure 5.7: Load vs. Displacement curves, load combination 2.

Figure 5.8: Plastic hinges for load combination 1.

Section 5.5 Example 3 - Multi-storey Steel Building

A modified version of the multi-storey steel building structure, developed by Chien (1987), was used to demonstrate the versatility of the integrated program, ISYSS, and its ability to define and analyze complex structures.

The structure, shown in Figure 5.9, consists of a lowrise section that is three storeys high upon which another eighteen storey highrise section sits. The plan view of the structure appears distorted in the figure and is the result of scaling of the diagram for printing purposes. The generation of the geometric characteristics of the structure in LOADS 95 began by defining the floors, walls and roof surfaces. Horizontal surfaces with similar geometry and member layout may be duplicated, simplifying the process. Gridlines are then defined for any horizontal surface and the corresponding members are automatically generated. These gridlines can be attached to like surfaces which greatly speeds up the generation of members. Columns are automatically generated at gridline intersections. Members may be added or deleted via the editor as required. The dead and live loads (use and occupancy) specified for this structure are given in Table 5.3. The highrise section of the structure is intended for office use and one level for mechanical equipment. The lowrise section of the structure is intended for retail use. For Ottawa, the reference velocity pressures are $q_{1/10} = 0.30 \text{ kN/m}^2$ for primary members and the overall structure and $q_{1/30} = 0.37 \text{ kN/m}^2$ for secondary members and cladding. These are retrieved automatically from the Climatic Information Database, NBCC Supplement 1990, and used to determine the corresponding wind loads acting on the structure.

Figure 5.9: Multi-storey steel building generated in LOADS 95.

	Dead Loads (kN/m²)	Live Loads (Use and Occupancy) (kN/m²)
Highrise section		
Ground Floor	4.15	4.8
Level 2 - Level 18	4.15	2.4
Level 19 (Mechanical)	4.15	7.2
Roof	1.20	1.0
Lowrise section		
Ground Floor - Level 3	4.15	4.8
Roof	1.20	1.0

Table 5.3: Dead and live loads (use and occupancy).

The time to create the geometry to determine the loads and load combinations and to attribute them to the 1437 members in the structure was approximately 10 hours as listed in Table 5.4. The corresponding primary structural members generated in LOADS 95 were successfully interpreted by the modified version of CU-STAND 4.0 and are shown in Figure 5.10. A first order elastic, second order elastic and second order inelastic analysis for the load combination due to dead load, live load due to use and occupancy and wind load was conducted using the appropriate analysis parameters. The corresponding characteristic applied load ratio vs. displacement curves for node 5, labeled 1, 2 and 3, are illustrated in Figure 5.11. The analysis of this complex structure by the modified version of CU-STAND 4.0 can be performed more rapidly and the results are comparable to other finite element analysis solutions. The approximate time required for each analysis is listed in Table 5.4.

Figure 5.10: Multi-storey steel frame displayed in modified version of CU-STAND 4.0.

Figure 5.11: Load vs. Displacement curves for node 5.

Operation	Time Required
Generating the geometry, determining the loads and load combinations and attributing them to members.	10 hours
First order elastic analysis	0.5 minutes
Second order elastic analysis	15 minutes
Second order inelastic analysis	20 minutes

Table 5.4: Time required to define the geometric and load characteristics and to perform analyses.

Obviously, the size and complexity of the model and the user's experience with the program will have an impact on the time required to define and analyze the structure. The advantages that accrue by integrating LOADS 95 with the modified version of CU-STAND 4.0 are more significant as the complexity and size of the structure increases. The ability to generate models of realistic structures within such a short period of time allows the engineer the time to investigate the behaviour of a number of structural schemes efficiently.

Chapter 6

Summary, Conclusions, Recommendations and Future Work

6.1 Summary

Currently, material standards and specifications in North America require that a second order elastic analysis for factored loads be conducted on the structure to determine the internal forces for which the members must be designed. The recognized inconsistencies in designing members by interaction equations for the limit states corresponding to the internal forces given by the analysis has led to the provision for the use of advanced analysis methods, which accurately describe the behaviour of the structure, in the Australian Standard, AS4100-1990, and the Eurocode, EC3 (1990). Similar provisions are being considered for standards and specifications in North America. Application of commercially available finite element programs for this purpose is not feasible as the cost of providing the professional services for most structures would be exorbitant. As a result, researchers McGuire, Deierlein, White, Ziemian, Clarke and others have been developing computationally efficient beam-column elements for advanced analysis of specifically steel buildings. One of these, CU-STAND 4.0, provides

the possibility of conducting a variety of different analyses up to and including a second order elastic-plastic analysis through the use of plastic hinge or quasi-plastic hinge methods.

Trends in computing technology indicate that this technology will make practical the use of advanced analyses for steel buildings. The magnitude of data required to fully describe the structural geometry, members, imperfections, lateral and torsional restraints, residual stresses, loads and load combinations and the geometry of all connections is significant and requires specially developed tools to make the use of advanced analysis feasible. This thesis focuses the integration of one of these tools, LOADS 95, in which the structural geometry can be defined and the loads determined, with CU-STAND 4.0. By successfully completing this integration process, a description of the database structure can now be defined.

Presented within the document is a description of the analysis-design process required for an integrated system; a state of the art review of the attributes and limitations of beam-column elements and stiffness matrices used for these elements in CU-STAND 4.0; a list of the physical attributes, geometric and material effects that need to be considered in an advanced analysis that will provide an accurate assessment of the structural behaviour; a description of the database structure for both LOADS 95 and CU-STAND 4.0; a description of the integration of LOADS 95 and CU-STAND 4.0; and a presentation of three examples which verify the integration process and demonstrate the advantages from integrating these programs.

6.2 Conclusions

1. LOADS 95 has the ability to define the structural geometry, to determine the loads, to determine the load combinations and to apply the loads to the members. Limitations include the inability to accurately define connection details, location of lateral restraint to members, and member orientation. The earthquake load computations are not fully implemented.
2. CU-STAND 4.0 is a state of the art advanced analysis program which provides the capability of conducting a second order elastic-plastic analysis for three dimensional steel structures and includes some recognition of residual stresses and the behaviour of flexible and semi-rigid connections. Its major limitation is that it cannot describe out-of-plane member instability problems. Others include: inability to allow for nonproportional loading in an automated manner, inability to interpret lateral restraint and boundary condition data required for instability problems, inability to offset member centroids from gridlines to reflect the actual location of the member and to fully model connection behaviour.
3. Investigations of advanced analysis programs, White et al. (1991) indicate that curvature and sway shortening have little or no effect on the true behaviour of the rectangular frame structures investigated by them.
4. Existing finite element programs could be used to model the true behaviour of a steel structure but is found to be impractical due to the following limitations: significant effort is required and the cost of providing professional services for most structures

would be exorbitant; lack of user interface tools for geometry definition, load determination, preliminary member sizing, connection definition, and member discretization; lack of user interface tools to interpret the data and to resize the members appropriately; the lack of verification against known test data of steel connections and steel structures.

5. LOADS 95 and CU-STAND 4.0 were successfully integrated to form the first version of an integrated system for the analysis, design, fabrication and erection of steel buildings. It represents a practical design tool that moves the design process towards the elimination of the inconsistencies that are known to exist. The versatility and advantages of the integrated system are clearly demonstrated by the examples presented in Chapter 5 where the last one was a complex structure with irregular geometry and 1437 members. The structural geometry, loads and load combinations were defined within 10 hours. A second order elastic-plastic analysis for one load combination on the structure was executed in 20 minutes.

6.3 Recommendations and Future Work

Based on this work and Chmelicek (1993, 1996), Clarke (Chapter 6, Chen and Toma (1994)), McGuire et al. (1989), Bento (1992), Lehtola (1996) and others who have written modules for analysis and design of members and connections, it is apparent that:

1. The methodology for generating the geometry of the structure, LOADS 95, is superior to methods presented elsewhere, but requires the development of additional user interface screens to define member orientation, generic connection details and generic lateral restraint details. The latter two would be retrieved from a library of standard construction details and applied to the structure as a function of its construction.
2. The program architecture needs to be redesigned and that LOADS 95 needs to be rewritten to provide a parallel hierarchical object oriented structure with the appropriate links and predefined predicates as the structural geometry is defined. Objects would belong to the classes of surfaces, members (beams, columns), lateral load resisting elements and connections. The definition of the geometry provides the links and hierarchy between the objects. Predicates are rules which describe the interrelationship between two objects and are geometry based. For example, should the location of a column line or gridline be adjusted in plan, then the geometry of all objects attached to that would be adjusted accordingly. Obviously, this would flag the program to recompute the loads.
3. A preliminary member sizing module needs to be developed.
4. A module for determining the type, size and optimum location of lateral load resisting elements within the structure needs to be developed.
5. CU-STAND 4.0 needs to be rewritten in object oriented programming code and incorporated into the integrated system. The benefits provided include portability of program among different computer architectures and the ability to easily maintain the program. Note, all shortcomings listed in section 4.5 should be addressed.

6. A finite element code, specially developed to describe complex buckling modes of steel members, Albert (1997), needs to be incorporated into the program. Members subject to potential instability failures need to be identified and discretized semi-automatically or automatically. Any new code must be verified and calibrated for standard load and boundary conditions.
7. Simple moment-rotation curves or characteristic load-deformation curves for connections may be sufficient in describing the influence of connection behaviour on the structural behaviour. However, connection design modules must be developed and linked to the integrated system to give the connection details which generate these simplified curves.

The completing of these recommendations should provide an integrated system that meets or surpasses the intent of future standards or specifications in North America by providing advanced analytical tools that can be used for design directly.

References

AISC. 1986. Load and Resistance Factor Design Specification for Structural Steel Buildings. American Institute of Steel Construction, Chicago, Illinois.

ALBERT, C., KENNEDY, D. J. L., MACCRIMMON, R. A. 1997. Inelastic Incremental Analysis of an Industrial Pratt Truss. "Innovations in Structural Design: Strength, Stability, Reliability", Symposium Honouring Theodore V. Galambos, University of Minnesota, Minneapolis, June 6,7, p.14.

ATTALLA, M.R., DEIERLEIN, G. G., and MCGUIRE, W. 1994. Spread of Plasticity: Quasi-Plastic-Hinge Approach. ASCE Journal of Structural Engineering, 120(8): 2451-2473.

BANAHAN, M. 1988. The C Book. Addison-Wesley Publishing Company, Wokingham, England.

BENTO, J. 1992. Intelligent CAD in Structural Steel: A Cognitive Approach. Imperial College of Science, Technology and Medicine, Rio de Janeiro, Brazil.

CHIEN, E. 1987. Multi-storey Steel Building Design Aid. Canadian Institute of Steel Construction, Willowdale, Ontario.

CHEN, C. 1994. Geometric Nonlinear Analysis of Three Dimensional Structures. M.S. Thesis, Cornell University, Ithaca, N.Y.

CHEN, W. F., and LUI, E. M. 1991. Stability Design of Steel Frames. CRC Press Inc., Boca Raton, Florida.

CHEN, W. F., and TOMA, S. (Editors). 1994. Advanced Analysis of Steel Frames. CRC Press Inc., Boca Raton, Florida.

CHMELICEK, J. 1993. Determination of Loads on Structural Steel Buildings. M.Eng. Thesis, Carleton University, Ottawa, Ontario.

CHMELICEK, J. 1996. LOADS 95. Carleton University, Ottawa, Ontario.

CLARKE, M. J. 1992. The Behaviour of Stressed Arch Frames. Ph.D. Thesis, University of Sydney, Sydney, Australia.

CLARKE, M. J., and ZABLOTSKII, S. V. 1995. NIFA User's Manual. Centre for Advanced Structural Engineering, School of Civil and Mining Engineering, University of Sydney, Sydney, Australia.

CSA. 1994. CAN/CSA-S16.1-94, Limit States Design of Steel Structures. Canadian Standards Association, Rexdale, Ontario.

CUTHILL, E., and McKEE, J. 1969. Reducing the bandwidth of sparse symmetric matrices. Proc. ACM National Conference, Association for Computing Machinery, New York, p.157-172.

EC3. 1990. Design of Steel Structures: Part I - General Rules and Rules for Buildings, Vol. 1, Eurocode edited draft, Issue 3.

GALAMBOS, T. V. 1968. Structural Members and Frames. Prentice-Hall Inc., Englewood Cliffs, N.J.

GATTAS, M., PAULINO, G. H., MENEZES, I. F. M., and MUKHERJEE, S. 1994. Node and Element Resequencing using the Laplacian of a Finite Element Graph - Part I: General Concepts and Algorithms. International Journal for Numerical Methods in Engineering, 37(9): 1511-1530.

GIBBS, N. E., POOLE, W. G., and STOCKMEYER, P. K. 1976. An Algorithm for Reducing the Bandwidth and Profile of a Sparse Matrix. SIAM Journal of Numerical Analysis, 13(2): 236-250.

HSEH, S., and ZHAO, Y. 1989. Programmer's Manual for CU-PREPF, CU-STAND and CU-QUAND. School of Civil and Environmental Engineering, Cornell University, Ithaca N.Y.

KALAB, P. 1984. Computer use in Structural Engineering with Emphasis on Pre-And PostProcessing. M.Eng. Thesis, Carleton University, Ottawa, Ontario.

KALICHARAN, N. 1994. C by Example. Cambridge University Press, Cambridge, England.

LEHTOLA, N. 1996. Desenvolvimento de Sistema Integrado para Projeto de Estruturas de Aço de Edificações. Tese de Doutorado, Pontifícia Universidade Católica do Rio de Janeiro, Rio de Janeiro, Brasil.

SSRC. 1992. Task Group 29 Workshop and Monograph on Plastic Hinge Based Methods for Advanced Analysis and Design of Steel Frames. Structural Stability Research Council, Pittsburgh, Pennsylvania.

McGUITRE, W., DEIERLEIN, G. G., SOOI, T. K., and ZHAO, Y. 1989. Illustrated Primer for CU-PREPF, CU-STAND and CU-QUAND. School of Civil and Environmental Engineering, Cornell University, Ithaca, N.Y.

MEEKINGS, B. A. E., KUDRYCKI, T. P., and SOREN, M. D. 1993. A Book on C. The MacMillan Press Ltd., Basingstoke, England.

NRCC. 1990a. National Building Code of Canada 1990. National Research Council of Canada, Ottawa, Ontario.

NRCC. 1990b. Supplement to the National Building Code of Canada 1990. National Research Council of Canada, Ottawa, Ontario.

ORAM, A., TALBOTT, S. 1991. Managing Projects with Make. O'Reilly and Associates Inc., Sebastopol, California.

ORBISON, J. 1982. Nonlinear Static Analysis of Three Dimensional Steel Frames. Ph.D. Thesis, School of Civil and Environmental Engineering, Cornell University, Ithaca, N.Y.

STANDARDS AUSTRALIA. 1990. AS4100-1990 Steel Structures. Standards Australia, Sydney, Australia.

WHITE, D. W., LIEW, J. Y. R., and CHEN, W. F. 1991. Second-Order Inelastic Analysis for Frame Design. Structural Engineering Report No. CE-STR-91-12, School of Civil Engineering, Purdue University, West Lafayette, Indiana.

WHITE, D. W., and HAJJAR, J. F. 1990. Application of Second-Order Elastic Analysis in LRFD: Research to Practice. Structural Engineering Report No. CE-STR-90-28, School of Civil Engineering, Purdue University, West Lafayette, Indiana.

WIEGAND, G., COVEY, R., and COUCH, P. 1992. Hoops Graphics System Reference Manual Version 3.2. Ithaca Software, Alameda, California.

VALLEY, J. 1992. C Programming for Unix. SAMS Publishing, Division of Prentice-Hall Computer Publishing, Carmel, Indiana.

ZIEMIAN, R. D., and MCGUIRE, W. 1992. A Method for Incorporating Live Load Reduction Provisions in Frame Analysis. AISC Engineering Journal, 29(1): 1-3.

IMAGE EVALUATION TEST TARGET (QA-3)

APPLIED IMAGE, Inc
1653 East Main Street
Rochester, NY 14609 USA
Phone: 716/482-0300
Fax: 716/288-5989

© 1993, Applied Image, Inc., All Rights Reserved