NUMERICAL MODELING OF THREE-DIMENSIONAL
LIGHT WOOD-FRAMED BUILDINGS

by

MING HE

B.A.Sc., Beijing University of Iron and Steel Technology, 1982
M.A.Sc., The University of Science and Technology Beijing, 1988
M.A.Sc., The University of British Columbia, 1997

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF
THE REQUIREMENTS FOR THE DEGREE OF

DOCTOR OF PHILOSOPHY

in

THE FACULTY OF GRADUATE STUDIES
THE FACULTY OF FORESTRY
Department of Wood Science

We accept this thesis as conforming
to the required standard.

THE UNIVERSITY OF BRITISH COLUMBIA

April 2002

© Ming He, 2002
In presenting this thesis in partial fulfilment of the requirements for an advanced degree at the University of British Columbia, I agree that the Library shall make it freely available for reference and study. I further agree that permission for extensive copying of this thesis for scholarly purposes may be granted by the head of my department or by his or her representatives. It is understood that copying or publication of this thesis for financial gain shall not be allowed without my written permission.

Department of Wood Science

The University of British Columbia
Vancouver, Canada

Date March 27, 2002
This thesis describes the development of numerical models for predicting the performance of three-dimensional light wood-framed buildings under static loading conditions and subjected to dynamic excitations. The models have been implemented into a package of nonlinear finite element programs. They satisfy the general requirements in the study of the structural behaviour of commonly applied light-frame construction. The models also deal with building configurations and loading conditions in a versatile manner. The application of these programs, therefore, can provide solutions to a wide range of investigations into the performance of wood light-frame buildings. These investigations may include the analyses of an entire three-dimensional light-frame building, an individual structural component, and a single connection containing one to several nails with varied material and structural components and combined loading conditions. These buildings and components can have irregular plan layouts, varied framing and sheathing configurations, and different nail spacings with or without openings.

The models were verified and tested on theoretical and experimental grounds. Theories of mechanics were applied to examine the models and related algorithms, while experimental results were used to validate the finite element programs and to calibrate the basic parameters required by the models. Besides the test data from previous shear wall
studies, three-dimensional building tests were conducted to provide the data required in the model verification. In the experimental planning phase, the programs were intensively employed to help select the correct configurations of the test specimens.

The experimental session contained four tests of a three-dimensional wood-framed structure: two static tests and two earthquake tests. These tests provided extensive information on the overall load-deformation characteristics, dynamic behaviour, torsional deformation, influence of dead load, overturning movement, failure modes, natural frequencies, and corresponding mode shapes of the test systems. The predicted behaviour of the test specimens by the programs is in good agreement with test results. This indicates that the programs are well suited for the investigation of the general behaviour of wood light-frame systems and for the study of load sharing and torsional effects on three-dimensional buildings due to structural and material asymmetries.
TABLE OF CONTENTS

ABSTRACT ii
LIST OF TABLES x
LIST OF FIGURES xii
NOTATION xviii
AKNOWLEDGMENT xxiv
DEDICATION xxvi
CHAPTER 1. INTRODUCTION 1
1.1. General problems 1
1.2. Objectives of the current study 3
1.3. Origin and contributions of the current study 4
1.4. Applications and future study 6
1.5. Thesis organization 7
CHAPTER 2. BACKGROUND 8
2.1. Wood light-frame buildings and their response to earthquakes 8
   2.1.1. Structural roles of wood light-frame buildings 8
   2.1.2. Earthquake response of wood light-frame buildings 9
2.2. Literature review

2.2.1. Numerical models
18
2.2.2. Experimental studies
18
2.2.3. Studies on the system effects in building
25
2.2.4. Testing methods
31
2.2.5. Connection models
35

CHAPTER 3. THE STRUCTURAL MODEL

3.1. Panel element

3.1.1. Assumptions
48
3.1.2. Geometric and physical relationships
48
3.1.3. Shape functions
52

3.2. Frame element

3.2.1. Assumptions
59
3.2.2. Geometric and physical relationships
60
3.2.3. Shape functions
61

3.3. Connection element

3.3.1. Assumptions
63
3.3.2. Connection model for lateral deformations
63
3.3.3. Load-displacement relations
66

3.4. The requirements of conforming elements
69

CHAPTER 4. FORMULATION OF STATIC NONLINEAR FINITE ELEMENT EQUATIONS

4.1. Principle of virtual work
70
4.2. Internal virtual work in elements
  4.2.1. Internal virtual work in plate element
  4.2.2. Internal virtual work in beam element
  4.2.3. Internal virtual work in connection element
4.3. Formulation of external force vector
4.4. Formulation and solution of system equations
4.5. Element stress calculations
  4.5.1. Panel stresses and central deflection
  4.5.2. Frame bending stress
4.6. Displacement control method
4.7. Convergence considerations
  4.7.1. Convergence checking and criteria
  4.7.2. Ill-conditioned stiffness matrix
  4.7.3. Self-adaptive procedures
4.8. Further expansion of the program

CHAPTER 5. COORDINATE TRANSFORMATION IN A THREE-DIMENSIONAL SPACE
  5.1. Coordinate transformation
  5.2. Stiffness matrix transformation

CHAPTER 6. FORMULATION OF DYNAMIC NONLINEAR FINITE ELEMENT EQUATIONS
  6.1. Formulation of equations of motion
  6.2. Formulation of mass matrix and vectors
6.3. Formulation of damping matrix 106
6.4. Formulation and solution of system equations 109
6.5. Time-stepping procedures 110
6.6. Convergence considerations 113
6.7. Solution of eigenproblems 113
6.8. Further expansion of the program 116

CHAPTER 7. ANALYTICAL-BASED VERIFICATION OF THE FINITE ELEMENT MODELS 118

7.1. Verification of three-dimensional transformations 119
7.2. Verification of the static finite element model 121
  7.2.1. Linear cantilever beams 121
  7.2.2. Beams loaded at shear center 124
  7.2.3. Superposition of linear deformations 127
  7.2.4. Stresses in cantilever beams 128
7.3. Verification of the dynamic finite element model 130
  7.3.1. Harmonic vibration of undamped single degree of freedom system 131
  7.3.2. Harmonic vibration of damped single degree of freedom system 133
7.4. Capabilities of programs 135

CHAPTER 8. PROCEDURES OF EXPERIMENTAL STUDY ON SIMPLIFIED THREE-DIMENSIONAL BUILDINGS 139

8.1. Test facility 139
  8.1.1. Capacity of shake table 140
  8.1.2. Data acquisition system 140
8.2. Determination of test buildings

8.2.1. Wall opening

8.2.2. Structural details of walls in test buildings

8.2.3. Structural details of roof in test buildings

8.2.4. The gravity load applied to test buildings

8.2.5. Selection of earthquake records

8.3. Experimental procedures

8.4. Brief description of two-dimensional shear wall tests

CHAPTER 9. EXPERIMENTAL-BASED VERIFICATION OF THE STATIC

FINITE ELEMENT MODELS

9.1. Experimental results of three-dimensional building under static loading conditions

9.2. Prediction of behaviour of the three-dimensional buildings under static loading conditions

9.3. System effects

9.4. Failure modes and relative movement between panel and frame

9.5. Prediction of the static behaviour of shear walls

9.6. Further expansion of the program

CHAPTER 10. EXPERIMENTAL-BASED VERIFICATION OF THE

DYNAMIC FINITE ELEMENT MODELS

10.1. Experimental results of three-dimensional building under dynamic excitation

10.1.1. Sine wave frequency sweep test and system fundamental natural frequency
10.1.2. Input acceleration signals 193
10.1.3. Basic parameters measured during the tests 196
10.2. Prediction of the dynamic behaviour of three-dimensional buildings 204
  10.2.1. Natural frequencies and mode shapes 204
  10.2.2. Comparison of predicted building behaviour and results from Test Three 206
10.3. Failure modes 210
10.4. Predictions of the dynamic behaviour of shear walls 215
10.5. Further expansion of the program 219

CHAPTER 11. CONCLUSIONS 221

BIBLIOGRAPHY 226

APPENDIX A. PROGRAM USER'S MANUAL 234
  A.1. Introduction 234
  A.2. Modeling 234
  A.3. Input and output files 236
  A.4. Structures and control parameters of input files 238
    A.4.1. Input file one – Titles 238
    A.4.2. Input file two – Input data 239
    A.4.3. Input file three – Cyclic protocol 257
    A.4.4. Input file four – Time-acceleration history 258
    A.4.5. Examples of input files 259
    A.4.6. Examples of the output file of an eigensystem 266
  A.5. Viewing graphic files generated by the postprocessor 267

Table of Contents ix
LIST OF TABLES

2.1 The response of vertical members in the cases of rigid and flexible diaphragms 17

7.1 Comparisons of beam deflections predicted by the finite element model and calculated by the beam theory – Panel-dominant cases 123

7.2 The influence of changing moment of inertia 124

7.3 Comparisons of beam deflections predicted by the finite element model and calculated by the beam theory – Frame-dominant cases 124

7.4 Displacements of four points at the end of beam when loaded at shear center 125

7.5 Displacements at the end of beams when loaded at shear center 127

7.6 Superposition of displacements 128

9.1 Summary of static test results of three-dimensional buildings 167

9.2 Comparison of static test results and numerical predictions 170

9.3 Improved predictions in roof translational motion in $y$ direction and torsional motion 173

9.4 Comparison of the deflection of shear walls with and without system effect 176

9.5 Comparison of energy dissipation of shear walls under cyclic loading predicted by LightFrame3D program and measured from tests 183
10.1 Natural frequencies obtained from sine wave sweep tests 193
10.2 Summary of dynamic test results of three-dimensional buildings 198
10.3 Comparison of natural frequencies obtained from Test Three and predicted by LightFrame3D program (Hz) 204
10.4 Comparison of dynamic test results and numerical predictions in Test Three 207
10.5 Comparison of natural frequencies of shear walls predicted by LightFrame3D program and measured from experiments 216
10.6 Comparison of maximum drifts and accelerations of shear walls predicted by LightFrame3D program and measured from experiments 216
**LIST OF FIGURES**

2.1 Load distribution in vertical members when diaphragm is rigid 14
2.2 Load distribution in vertical members when diaphragm is flexible 17
2.3 Seismic response of a building modeled by Chehab (1982) 19
2.4 Global degrees of freedom for house model by Gupta and Kuo (1987) 20
2.5 A three-dimensional finite element model built by Kasal et al. (1994) 21
2.6 Two-storey building test in the CUREE-Caltech wood-frame project (Folz et al. 2001) 28
2.7 Shake table tests of Earthquake 99 project 30
2.8 Three-dimensional view of test structure (Phillips 1993) 33
2.9 L-shaped test house (Foliente et al. 2000) 35
2.10 General load-displacement relationship of a wood structure 36
2.11 General hysteresis loops of a wood structure with a monotonic push-over curve 37
2.12 Nail load-displacement model (Foschi 1974, 1977) 41
2.13 Modeled hysteresis loops of nailed wood joints by various researchers 42
3.1 Basic element unit 47
3.2 Nodal and element degrees of freedom 47

*List of Figures* xii
3.3 Middle plane extension of plate element
3.4 Derivation of shear strain components
3.5 Natural coordinate system
3.6 Twist deformation of a horizontal beam element
3.7 Connection finite element model for lateral load
3.8 Nail connector stress-strain relationship
3.9 Connection lateral deformations
3.10 Connection load-displacement relationships
4.1 General three-dimension body
4.2 Uniformly distributed load acting on an element
4.3 Newton-Raphson iteration starting from point $a$ and converging at point $b$
4.4 Cyclical searching in the Newton-Raphson iteration method
5.1 Coordinate transformation
5.2 Mapping of stiffness coefficients in element stiffness matrix (only exterior panel is considered)
6.1 Rayleigh damping
6.2 Mass-proportional damping and stiffness-proportional damping
7.1 Test cases used in verifications
7.2 Test cantilever beams with varied cross-section profiles
7.3 Channel sections were loaded at shear center or at corner 1
7.4 Output graphics from program showing the deformations at the end of beams
7.5 Cantilevered angle sections were loaded at shear center
7.6 Output graphics from program showing the deformations at the end of beams
7.7 Test of superposition of linear deformations 128
7.8 The locations of Gauss Points in one element 129
7.9 Comparisons of responses of an undamped single degree of freedom system between theoretical calculation and numerical prediction 133
7.10 Comparisons of responses of a damped single degree of freedom system between theoretical calculation and numerical prediction 135

8.1 Data acquisition layout (plan view) 142
8.2 Proposed configurations of test buildings 144
8.3 Joshua Tree record, 1992 California Landers earthquake, 0.35g 145
8.4 Locations of displacement measurement for torsion calculation 145
8.5 Comparison of drifts from the test buildings with Openings A and B (x₀) 146
8.6 Comparison of relative drifts in x direction from the test buildings with Openings A and B (x₁ - x₂) 146
8.7 Comparison of relative drifts in y direction from the test buildings with Openings A and B (y₁ - y₂) 146
8.8 Wall frame and panel layouts 150
8.9 Plan view of roof construction 152
8.10 Joint reinforcement 153
8.11 Arrangement of steel plates as gravity load 154
8.12 Attachment of the steel plates to building roof 154
8.13 Canoga Park record, 1994 California Northridge earthquake, 0.35g 155
8.14 Sherman Oaks record, 1994 California Northridge earthquake, 0.44g 156
8.15 Fourier spectra of three earthquake records 157

List of Figures xiv
8.16 The influence of earthquake record on building behaviour 159
8.17 The scheme of 3D building testing assembly 160
8.18 The test building on shake table 161
8.19 Static shear wall test assembly 164
8.20 Dynamic shear wall test assembly 164
9.1 Load-displacement relationship of 3D building in Test One 168
9.2 Load-displacement relationship of 3D building in Test Two 168
9.3 Comparison of load carrying capacities of two buildings 169
9.4 Comparison of load-displacement characteristics in Test One 171
9.5 Comparison of load-displacement characteristics in Test Two 171
9.6 Plan view of the displaced position of the top of the test building 172
9.7 Predicted post-peak cyclic load-displacement behaviour of building in Test One 174
         (with static back bone curves)
9.8 Forces in frame members measured at floor level 175
9.9 Failure modes of nail connections 178
9.10 Panel movements of the north shear wall 179
9.11 Panel movements of the south shear wall 180
9.12 Predicted panel positions on the north shear wall 180
9.13 Predicted panel positions on the south shear wall 181
9.14 Static test of shear wall with 4 by 8 regular panels 182
9.15 Cyclic test of shear wall with 4 by 8 regular panels 182
9.16 Static test of shear wall with one jumbo panel 183
9.17 Cyclic test of shear wall with one jumbo panel 183
<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>9.18</td>
<td>Predicted post-peak cyclic load-displacement relationship of shear wall</td>
<td>184</td>
</tr>
<tr>
<td>10.1</td>
<td>Amplitudes in the primary direction in the first frequency range</td>
<td>190</td>
</tr>
<tr>
<td>10.2</td>
<td>Amplitudes in two directions in the first frequency range</td>
<td>190</td>
</tr>
<tr>
<td>10.3</td>
<td>Comparison of the phases of two signals in the first frequency range</td>
<td>190</td>
</tr>
<tr>
<td>10.4</td>
<td>Amplitudes in the secondary direction in the second frequency range</td>
<td>191</td>
</tr>
<tr>
<td>10.5</td>
<td>Amplitudes in two directions in the second frequency range</td>
<td>191</td>
</tr>
<tr>
<td>10.6</td>
<td>Comparison of the phases of two signals in the second frequency range</td>
<td>191</td>
</tr>
<tr>
<td>10.7</td>
<td>Phases of two signals in the primary direction in the third frequency range</td>
<td>192</td>
</tr>
<tr>
<td>10.8</td>
<td>Phases of two signals in the secondary direction in the third frequency range</td>
<td>192</td>
</tr>
<tr>
<td>10.9</td>
<td>Three natural frequencies of building in Test Three</td>
<td>193</td>
</tr>
<tr>
<td>10.10</td>
<td>Three natural frequencies of building in Test Four</td>
<td>193</td>
</tr>
<tr>
<td>10.11</td>
<td>Comparison of the 5% damped acceleration response spectra</td>
<td>195</td>
</tr>
<tr>
<td>10.12</td>
<td>Input earthquake record (Joshua Tree, east-west, 0.35g) used in Test Three</td>
<td>196</td>
</tr>
<tr>
<td>10.13</td>
<td>Input earthquake record used in Test Four</td>
<td>196</td>
</tr>
<tr>
<td>10.14</td>
<td>Response-time history of Test Three</td>
<td>201</td>
</tr>
<tr>
<td>10.15</td>
<td>Response-time history of Test Four in the primary direction</td>
<td>202</td>
</tr>
<tr>
<td>10.16</td>
<td>Response-time history of Test Four in the secondary direction</td>
<td>203</td>
</tr>
<tr>
<td>10.17</td>
<td>Predicted three mode shapes</td>
<td>205</td>
</tr>
<tr>
<td>10.18</td>
<td>Comparison of drift and acceleration in Test Three</td>
<td>207</td>
</tr>
<tr>
<td>10.19</td>
<td>Comparison of acceleration frequency range</td>
<td>208</td>
</tr>
<tr>
<td>10.20</td>
<td>Comparisons of drifts of four walls in Test Three</td>
<td>210</td>
</tr>
<tr>
<td>10.21</td>
<td>Nail withdrawal and pulling through failure modes</td>
<td>211</td>
</tr>
<tr>
<td>10.22</td>
<td>Insignificant permanent deformation in end wall</td>
<td>212</td>
</tr>
</tbody>
</table>
10.23 Comparison of drifts of four walls in Test Four
10.24 Damages in the east end wall
10.25 Two nails in fracture modes
10.26 Dynamic responses of shear wall with 4 by 8 regular panels
10.27 Dynamic responses of shear wall with one jumbo panel
10.28 Comparison of Fourier spectra of shear walls predicted by LightFrame3D program and from tests
**NOTATION**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>area.</td>
</tr>
<tr>
<td>$\mathbf{a}$</td>
<td>degree of freedom vector, function of time.</td>
</tr>
<tr>
<td>$\mathbf{\ddot{a}}$</td>
<td>relative mass acceleration vector.</td>
</tr>
<tr>
<td>$\mathbf{\ddot{a}}_g$</td>
<td>ground acceleration vector.</td>
</tr>
<tr>
<td>$\mathbf{\ddot{a}}_t$</td>
<td>absolute acceleration vector.</td>
</tr>
<tr>
<td>$C$</td>
<td>viscous damping coefficient.</td>
</tr>
<tr>
<td>$c$</td>
<td>element damping matrix.</td>
</tr>
<tr>
<td>$D$</td>
<td>dead load applied to test building, kN.</td>
</tr>
<tr>
<td>$dA$</td>
<td>differential surface area of an infinitesimal element.</td>
</tr>
<tr>
<td>$dV$</td>
<td>differential volume of an infinitesimal element.</td>
</tr>
<tr>
<td>$E$</td>
<td>total energy dissipation of test building, Nm.</td>
</tr>
<tr>
<td>$E_x, E_y$</td>
<td>Young’s moduli.</td>
</tr>
<tr>
<td>$\mathbf{F}_I$</td>
<td>inertia forces.</td>
</tr>
<tr>
<td>$F$</td>
<td>resultant reaction force.</td>
</tr>
<tr>
<td>$F_{\text{max}}$</td>
<td>maximum resultant reaction force.</td>
</tr>
<tr>
<td>$\mathbf{f}_s$</td>
<td>resistant force vector, depending on deformation histories.</td>
</tr>
</tbody>
</table>
\( \tilde{f} \) = force containing the contributions from the linear and nonlinear behaviour of panel and frame members and from the nonlinear behaviour of nail connections.

\( f \) = natural frequency.

\( G_{xy} \) = shear modulus.

\( H \) = height of frame or beam.

\( I \) = unit matrix.

\( I \) = moment of inertia.

\( i, j, k \) = unit vectors.

\( J \) = Jacobian matrix.

\( K \) = generalized stiffness matrix.

\( K_T \) = global tangent stiffness matrix.

\( (K)_L \) = linear part of global tangent stiffness matrix.

\( (K)_N \) = nonlinear part of global tangent stiffness matrix.

\( K \) = initial stiffness; system stiffness.

\( k_T \) = element tangent stiffness matrix.

\( (k)_L \) = linear part of element tangent stiffness matrix.

\( (k)_N \) = nonlinear part of element tangent stiffness matrix.

\( L \) = length of beam.

\( l_i, m_i, n_i \) = direction cosines, \( i = 1, 2, 3 \).

\( M \) = generalized mass matrix.

\( M \) = system mass.

\( m \) = element mass matrix.

\( m_u, m_w \) = mass vectors related to ground acceleration components.
\( N_{0i}, L_{0i}, M_{0i} = \) shape functions.

\( \mathbf{N}_0, \mathbf{L}_0, \mathbf{M}_0 = \) shape functions, function of space.

\( N = \) number of integration points.

\( O_{c1, c2, c3, c4} = \) uplift displacements measured at four corners of building, mm.

\( P = \) force.

\( P_{\text{max}} = \) maximum load carrying capacity of test building (static) or maximum effective earthquake force applied to test building (dynamic), kN.

\( Q_x, Q_y, Q_z = \) shape function combinations.

\( Q = \) first moment of area.

\( Q_0 = \) intercept of the asymptote.

\( Q_1 = \) asymptotic stiffness.

\( \mathbf{R} = \) external load vector.

\( \mathbf{R}_B = \) body force vector.

\( \mathbf{R}_C = \) external concentrated force vector.

\( \mathbf{R}_S = \) external surface force vector.

\( R_x, R_y, R_z = \) projections of a force in \( x, y, \) and \( z \) directions.

\( \mathbf{r}_B, \mathbf{r}_S = \) body forces and surface forces, respectively.

\( r = \) ratio of moment inertia.

\( \mathbf{T} = \) transformation matrix.

\( t = \) thickness of panel; time.

\( \mathbf{u} = \) arbitrary displacement field.

\( \mathbf{u}, \mathbf{v}, \mathbf{w} = \) displacement fields in \( x, y \) and \( z \) directions, respectively, functions of space and time.
\( \mathbf{\ddot{u}}, \mathbf{\ddot{v}}, \mathbf{\ddot{w}} \) = nodal acceleration vectors in \( x, y \) and \( z \) directions, respectively.

\( \mathbf{\ddot{u}}_g, \mathbf{\ddot{w}}_g \) = ground acceleration components in \( x \) (horizontal) and \( z \) directions, respectively.

\( \mathbf{\ddot{u}}_r, \mathbf{\ddot{w}}_r \) = absolute nodal acceleration vectors in \( x \) and \( z \) directions, respectively.

\( u_s, v_s, w_s \) = the small displacements of a point in the middle plane of the plate element during deformation in the \( x, y \) and \( z \) directions, respectively.

\( u, v \) = the small displacements caused by axial deformation only in the \( x \) and \( y \) directions, respectively.

\( \mathbf{\ddot{u}}_g^0 \) = acceleration amplitude of harmonic excitation.

\( V \) = volume.

\( W_i, W_j \) = weighting factors.

\( X_i, Y_i, Z_i \) = global nodal coordinate system.

\( \mathbf{X}_i, \mathbf{Y}_i, \mathbf{Z}_i \) = local nodal coordinate system.

\( \mathbf{X}^o, \mathbf{Y}^o, \mathbf{Z}^o \) = identity vectors along the edges of the element in \( \mathbf{X}, \mathbf{Y} \), and \( \mathbf{Z} \) directions, respectively.

\( x, y, z \) = Cartesian coordinate system.

\( x_C, y_C \) = central coordinates of plate element.

\( \alpha, \beta \) = damping coefficients.

\( \Delta x, \Delta y \) = side lengths of plate element.

\( \Delta \) = displacement or drift.

\( \Delta_{\text{max}} \) = displacement at \( F_{\text{max}} \).
\( \Delta_x \) = ultimate displacement of test building in \( x \) direction at the maximum load \( P_{\text{max}} \) in static tests; maximum displacement of test building in \( x \) direction in dynamic tests, mm.

\( \Delta_y \) = maximum displacement of test building in \( y \) direction in both static and dynamic tests, mm.

\( \delta a \) = virtual displacement vector.

\( \delta W_i, \delta W_E \) = internal virtual work and external virtual work, respectively.

\( \delta W_x, \delta W_y, \delta W_z \) = virtual work done by a force in \( x, y, \text{and} z \) directions.

\( \delta u, \delta v, \delta w \) = components of a virtual displacement in \( x, y, \text{and} z \) directions.

\( \delta \varepsilon \) = virtual strain vector.

\( \gamma_{xy} \) = shear strain.

\( \xi, \eta \) = variables of natural coordinate system.

\( \xi', \eta' \) = natural coordinates for loading patch.

\( \varepsilon \) = normal strain vector.

\( \varepsilon_d, \varepsilon_F \) = tolerances.

\( \varepsilon_x, \varepsilon_y, \varepsilon_z \) = the components of normal strain in \( x, y, \text{and} z \) directions.

\( \varepsilon'_x, \varepsilon'_y \) = the components of normal strain due to stretch.

\( \varepsilon''_x, \varepsilon''_y \) = the components of normal strain due to bending.

\( \zeta \) = damping ratio.

\( \theta \) = maximum torsional rotation of the roof of test building about the vertical axis, which passes through the center of mass of the roof diaphragm, rad.

\( \Lambda \) = a \( p \times p \) diagonal matrix listing the corresponding eigenvalues.
\( \lambda \) = eigenvalue.

\( \nu_{xy}, \nu_{yx} \) = Poisson's ratios.

\( \sigma \) = normal stress vector.

\( \sigma_x, \sigma_y, \sigma_z \) = the components of normal stress in x, y, and z directions.

\( \sigma_y \) = yield stress of steel connector.

\( \tau \) = time instance.

\( \Phi \) = an \( n \times p \) matrix with its columns equal to the \( p \) eigenvectors and its rows equal to the \( n \) system degrees of freedom.

\( \phi \) = eigenvector.

\( \Psi \) = global out-of-balance load vector.

\( \psi_e \) = element out-of-balance load vector.

\( \omega \) = circular natural frequency of a structural system.

\( \omega_0 \) = circular forcing frequency of harmonic excitation.
ACKNOWLEDGMENT

In every stage of fulfilling the tasks associated with the completion of this thesis, I am grateful to the professors, technicians and colleagues who have helped me in many ways.

First, I sincerely thank Dr. F. Lam for the guidance and the financial support that he has given to me during the whole program. His precious advice and opinion have greatly shaped my view of timber structural engineering and my research on the modeling of wood building systems.

I also express my gratitude to Prof. R. Foschi, with whom I have made many fruitful discussions on the subjects related to the research project. His invaluable theoretical instructions and suggestions were very critical and helpful in the problem solving and in the finite element programming and have influenced the whole structure of the current study.

My heartfelt appreciation also goes to Dr. H. Prion. His knowledgeable and informative ideas about wood structural design led our experiments to success. He has always been very generous to me with time, advice and instruction in both theoretical and experimental work over the years.

Whenever I used the fundamental concepts of the structural dynamics as an important tool in my whole Ph.D. study, I could not forget the expert instruction and assistance given by Dr. C. Ventura. I certainly want to give my sincere thanks to him.
During the entire experiment session, it has been my privilege to work with the laboratory technicians, who provided continuous assistance in every detailed piece of work. Special thanks are given to H. Nichol, the technician in the Earthquake Engineering Laboratory of the Department of Civil Engineering, for his significant technical contributions to the experimental studies. Further thanks are given to D. Hudniuk and H. Schrempp, technicians in the Department of Civil Engineering, for their technical support.

Moreover, I gratefully acknowledge the comments and suggestions given by my colleagues F. Yao and Y. T. Wang. The discussions with them regarding finite element programming were always beneficial. J. Durham is thanked for providing the experimental data from her study.

Finally, the Department of Civil Engineering and the Department of Wood Science are thanked for their support with equipment, materials and finances.
To my wife, Yingmei

and my son, Da
1.1. General problems

Low-rise residential houses and small commercial buildings in North America are conventionally light-frame structures using wood-based materials. Typically, they are composed of two-dimensional diaphragm systems (e.g. shear walls, floors, ceilings, and roofs) and are highly indeterminate. These wooden structural systems are generally believed to perform well under wind and seismic loading when carefully constructed. This performance could be attributed to the high strength-to-weight ratio of timber as a building material, the redundancy of the whole system, and the ductility of the connections.

However, as has been shown in past earthquakes and hurricanes, the structural integrity of wood frame buildings under the action of natural hazards is not necessarily guaranteed, especially in multi-storey buildings with asymmetrical geometry. For many years, a large amount of experimental and analytical work has been done to understand the structural behaviour of wood based light-frame systems. The work, to a great extent, has been limited to the study of two-dimensional structural components, such as shear walls, roof and floor diaphragms, and metal connectors and fasteners, under static monotonic or cyclic loading. Experiments on full-scale light-frame houses have seldom been done due to high test costs and demands. The knowledge obtained so far about the structural behaviour of
complete wood buildings is mainly derived from construction practice and a few experimental studies on major structural components.

Analytical methods were also developed by a few researchers to predict the structural performance of an entire building. The numerical studies on the light-frame structures mentioned above reveal that further studies on some important issues still need to be performed. These issues include (1) at component level, the influence of nail type, spacing and pattern, panel discontinuity and reinforcement, wall aspect ratio, and new construction techniques; and (2) at structural level, the influence of dead load and inertia mass, ground acceleration excitation, load sharing among components, and load path in the structure.

A literature review also shows that in the early stages of analytical studies of wood buildings, the three-dimensional timber structures were modeled either by simplified closed-form equations (such as Yasumura et al. 1988, Schmidt and Moody 1989) or by linear finite elements under mainly static lateral loads. In recent years, progress has been made in the simulation of three-dimensional wood light-frame buildings. The complications of both actual structural behaviour and programming algorithm make it difficult to develop a model for nonlinear dynamic analysis with refined finite elements. With the development of analytical techniques and computational tools, and with increasing knowledge of wood structural behaviour, it is of great interest and need to develop rational models to analytically predict the structural performance of three-dimensional buildings by means of the nonlinear finite element method. In the investigation of the performance of light-frame construction, reliable numerical models possess obvious cost advantages over experimental methods, even though the latter cannot be completely replaced. Analytical approaches can dramatically reduce time and cost in experiments. Also, they are applicable to cases that are not feasible
in an experimental study. For example, in a parametric study or in development of a
response surface, a large number of cases with different parameter combinations should be
considered. While a numerical model can easily fulfill the task by altering relevant
parameters, it is very difficult to do so within a short time by experimental methods.

1.2. Objectives of the current study

Based on the literature survey and the research needs, a study was initiated, the
overall objective being to develop robust and reliable numerical models capable of predicting
the response of three-dimensional wood light-frame buildings subjected to static loading
and/or dynamic earthquake excitation. The models should also be applicable to the structural
components and individual connections. The following procedures are followed to achieve
this objective:

(1) Development of a nonlinear finite element program for panelized structures that
are in three-dimensional space and are under static monotonic loading;

(2) Development of a nonlinear finite element program with mechanics based nail
model, capable of analyzing structures under static cyclic loading;

(3) Development of a finite element program to calculate a number of the lowest
eigenvalues (natural frequencies) and corresponding eigenvectors (mode shapes) of the
structure under investigation;

(4) Development of a nonlinear finite element program for a full analysis of structures
under dynamic earthquake excitation;

(5) Verification of numerical models by means of mechanics theories and
experimental results from previous shear wall tests and newly conducted three-dimensional
shaking table tests on a model building; and
(6) Application of the programs with experimental input to study load path, load sharing, and torsion effect, which are significant in a three-dimensional structural assembly.

1.3. Origin and contributions of the current study

The development of three-dimensional structural analysis models is based on a diaphragm analysis program (PANEL) written by Foschi (1997). This program was selected as a starting point because its structure possesses many features that are deemed necessary in a three-dimensional numerical simulation, even though it is only applicable to two-dimensional light-frame components under static monotonic loading. In the new model development, the following theoretical contributions are believed to be important and critical:

(1) Introduction of coordinate transformations into finite element models so that the behaviour of three-dimensional structures can be described. In the process of transformation of local element stiffness matrices into a global stiffness matrix, the "substructuring" technique is adopted. Under this technique, panel elements and frame elements, which are under different coordinate systems, can be assembled into one global system without increasing the complexity of the analysis.

(2) Introduction of a general mechanics-based nail connection model, HYST (Foschi 2000) into the newly developed finite element models. This makes the models capable of performing the cyclic and dynamic analyses by calculating the hysteresis loops in mechanical connections from basic material properties, thus enabling them to adapt automatically to any input history, with respect to either force or displacement. This model considers a nail connector as elasto-plastic beam acting on wood, a nonlinear medium that only acts in compression. It permits gaps to be formed between the connector and the surrounding medium. The model can then develop pinching as gaps are formed, and the energy
dissipation in a connection's deformation history can therefore be accurately represented. During calculation, only basic material properties of the connector and the embedment characteristics of the wood medium are required. These characteristics make the HYST model a distinct one from other empirical curve-fitting models.

(3) Modeling of deformations and displacements of nail connection elements between sheathing panels and frame members in three directions at each nail location. The lateral deformations of nail connection are modeled by a spring element in each of the $x$ and $y$ direction in the panel plane. The nail withdrawal or the panel-frame contact is modeled in a direction perpendicular to the panel plane ($z$ direction).

(4) Contribution of load control and displacement control as two options in simulations. In static and cyclic analyses, sometimes, the displacement control method is more desirable. It allows the analysis to go beyond the system's maximum capacity to provide a complete load-displacement path. Also, it can easily follow the multiple load-unloading-reloading path to present stiffness and strength degradations and pinching effects. This is difficult in the load control method because of numerical stability problems.

(5) Introduction of the cyclic analysis procedure into the finite element models. Most of previously developed numerical programs could not perform cyclic calculation due to both the connection models and the load control mode that these programs used. The cyclic test is an important step in the study of woodframe structures to understand the energy dissipation mechanism and the hysteresis behaviour in a wood structure. The capability of performing a cyclic analysis allows the models to carry out all three major structural analysis procedures and therefore fills the gap between the static analysis and dynamic analysis.

This study is unique and original because it is the first attempt to
(1) predict the behaviour of an entire building, instead of individual components, under static monotonic loading, cyclic loading, and dynamic earthquake excitation, by building three-dimensional refined finite element models;

(2) implement a mechanics-based connection model that makes the analysis more general;

(3) integrate numerical modeling and experimental study into one project, therefore providing instant opportunities to verify the models and to predict the building behaviour;

(4) verify the models extensively by mechanics theories and by using a database from both past building components and present three-dimensional building tests; and

(5) build a database that contains research results from analytical and experimental studies and that can benefit future studies in wood-based light-frame structures and performance-based design in wood construction practice.

1.4. Applications and future study

The programs (LightFrame3D) are designed to evaluate and predict the structural response of a three-dimensional light-frame building with varied material, structural, and loading combinations. Load control or displacement control can be used as input history in the static analysis. The structures can be loaded either monotonically or cyclically. They can also be excited by dynamic ground acceleration with constant concentrated or distributed loads being applied simultaneously. A post-processor of the model provides information on the deformation, strength, and failure mode of the structural components.

As an analytical tool in future studies, the programs possess the capacity to perform extensive parametric studies and system identification. A well-understood structural behaviour and a carefully-defined load distribution in a building would provide useful
guidelines in light-frame building design. The programs, as a benchmark, can be further improved and expanded. It is generally believed that the numerical evaluation of the dynamic response of a nonlinear wood light-frame system with numerous degrees of freedom is computationally demanding. However, with the rapid development of computer technology, this shortcoming will be alleviated. For the time being, to make these programs run more efficiently, new solution techniques have to be introduced. Interfaces between the current finite element programs and commercial software are also needed to enhance the user-friendliness of the pre-processor and post-processors. Furthermore, new functions can be added to the programs to satisfy broader requirements in the research and study of light wood frame systems.

1.5. Thesis organization

This thesis describes all the procedures in developing nonlinear finite element models for three-dimensional light wood frame building systems.

This thesis first presents a survey of previous research work on light-frame building systems with both analytical and experimental methods. Based on this survey, the most important research needs of the current study are identified. The thesis describes in detail the structures of numerical models and the formulations of finite element equations for both static and dynamic situations, followed by the model verifications by mechanics theories. The capacity of the programs and their potential applications are also addressed. The limitations in the study are discussed at the end of appropriate chapters. The experiments, as an integral component of the study in providing a database in model verification, are also presented in detail. Lastly, the comparison between predictions made by the programs and actual results from tests is presented.
CHAPTER 2

BACKGROUND

2.1. Wood light-frame buildings and their response to earthquakes

2.1.1. Structural roles of wood light-frame buildings

In North America, wood light-frame construction is primarily used for single-family housing, low- to mid-rise multi-family residential buildings, and low-rise commercial structures. These buildings usually consist of a few typical structural components, such as vertical diaphragms (shear walls) and horizontal diaphragms (floors and roofs). These components play critical roles in resisting various loads, including those induced by earthquake ground motion, to retain the structural integrity of the whole system.

Shear walls are the major lateral load-carrying components in a wood frame system. The structural details of shear walls, including framing and sheathing layout and nail spacing, have a significant impact on the overall performance of building system (He 1997). Shear walls need to resist the in-plane lateral loads caused by wind and ground motion during an earthquake. These lateral loads can be either the forces directly added on shear walls or the forces transferred from floor and roof diaphragms. Shear walls are also required to carry out-of-plane loads, vertical dead loads, live loads, and vertical components of wind and seismic loads. Roof and floor diaphragms mainly need to withstand dead loads and live loads from building materials, occupants, or rain and snow, to carry transverse and lateral loads.
generated by wind or during earthquakes, and to transfer those loads through the vertical lateral load resisting systems to the foundation.

A well-constructed moderately sized wood light-frame building is a very efficient and ductile system. It has a history of good performance under seismic loading conditions. This could be attributed to the following characteristics:

(1) a high strength-to-weight ratio of wood material;

(2) redundancy of the system due to a large number of closely spaced members and nail fasteners;

(3) high ductility and energy dissipation capacity of connections;

(4) light mass being supported by the system; and

(5) symmetrical plan layouts with small openings (traditional residential buildings).

Even though wood light-frame buildings have a good reputation for seismic performance, they may not always be safe and their structural integrity is not necessarily guaranteed against earthquakes. There were always wood light-frame buildings showing poor or bad performance in past earthquakes. To improve the performance of a light wood-framed system, the response of the system to an earthquake should first be fully investigated.

2.1.2. Earthquake response of wood light-frame buildings

The 1994 Northridge, California earthquake caused extensive damage to residential houses, commercial buildings, and highway systems. This earthquake is believed to be the most intense ground shaking that had been recorded so far in a populated area in North America (Rainer and Karacabeyli 1998). It was notable that the horizontal ground accelerations, combined with the vertical accelerations of comparable amplitude, exceeded the nominal horizontal design acceleration of 0.4g by factors of 2 and more. Whereas most
wood frame buildings performed exceedingly well, attention was drawn to the collapses that occurred in a number of multi-storey woodframe apartment buildings due to weak first storey. During another devastating earthquake, which occurred in Kobe, Japan in 1995, the peak ground acceleration was recorded as high as 0.8g in densely populated areas. Similar to the Northridge earthquake, the horizontal ground motion in this earthquake was accompanied by severe vertical shocks. The acceleration of the vertical ground motion sometimes even exceeded the horizontal one. Major collapses occurred in wood houses built before and immediately after World War II. These structures typically consisted of post and beam wood framing, with walls formed by horizontal board strapping in-filled with bamboo webbing and covered with clay. Numerous recently constructed North American light-frame 2 and 3 storey residential buildings survived the earthquake without visible damage (Rainer and Karacabeyli 1998). These cases demonstrate that wood buildings with different construction styles can act very differently from each other when subjected to earthquake excitation. It is evident that many factors influence the performance of a wood building during an earthquake, some of which are the earthquake motion, the soil quality of the site, and the structural, architectural, and material characteristics. Based on their functions, these factors can be divided into four categories (Rainer and Karacabeyli 2000):

(1) the characteristics of ground movement at the building site: amplitude, duration, frequency content;

(2) the dynamic characteristics of the building: natural modes, frequencies and damping;

(3) the deformational characteristics of the building: stiffness, strength and ductility; and

(4) the building regulations being followed in design and construction: year and type of code and standard, engineered design, or construction by conventional rules.
The nature of an earthquake is unpredictable and the properties of the building site cannot be modified easily; therefore, improvement of the seismic performance of wood light-frame buildings relies mainly on understanding and control of the building itself. For several decades, researchers have made great efforts to understand the behaviour of woodframe systems and components under earthquake excitation. Many of these efforts were limited to the study of structural components, such as shear walls, under static loading conditions. From those investigations, mainly using two-dimensional components as the study objects, many influencing factors could not be detected due to the inherent physical and theoretical difficulties of the problem. As a result, the three-dimensional behaviour of the entire building could not be predicted correctly. Based on the literature survey, some commonly recognized factors are described as follows.

Natural frequency – It is one of the basic building characteristics to be identified, which is determined by the mass and stiffness of the whole system, as shown in (2.1).

\[
\omega = \sqrt{\frac{K}{M}} \quad \text{or} \quad f = \frac{1}{2\pi} \sqrt{\frac{K}{M}}
\]

where

\(\omega\) = natural circular frequency.

\(f\) = natural frequency, \(f = \frac{\omega}{2\pi}\).

\(K\) = initial stiffness of the system.

\(M\) = mass of the system.

Usually, only a few of the lowest natural frequencies are of interest. These natural frequencies, when combined with the corresponding mode shapes, can be used to interpret the response of a structure to different types of forced vibration. Correctly identifying these
quantities is important for the study of the dynamic response of light-frame buildings to earthquakes. Foliente and Zacher (1994) summarized the previous investigations on natural frequencies of low-rise wood-framed buildings, pointing out that most of these natural frequencies ranged from 3.0 Hz to 9.0 Hz. When compared to the frequency content of past earthquakes obtained from spectral analysis, this natural frequency range is within the high energy frequency range of most earthquakes. Therefore, the building could be severely damaged if its natural frequencies coincide with or are close to the predominant frequency of earthquake ground motion that causes resonance or partial resonance. It is desirable to design a building such that its natural frequencies are far different from the expected earthquake frequency, so that the response of the building to the earthquake will be small and damages will be minor. The practical difficulty in building such a system, however, is that the exact frequency content of a future earthquake is unknown in advance and the ratio between stiffness and mass of the whole structural system cannot be adjusted significantly. In addition, during an earthquake, the natural frequencies of a building will decrease as a result of stiffness degradation when damage occurs in the structural components. Therefore, the architectural and structural nature of a building dictates that resonance is always a potential problem. In general, the destruction of a building in an earthquake is the result of accumulated interaction between earthquake and building vibrations.

Building asymmetry and torsion – The modern wood light-frame houses have shifted very much from the traditional ones in terms of their building plan, arrangement of walls and openings, and architectural design. The house is no longer in a rectangular and symmetric plan. Doors and windows are not evenly distributed within the building, and large openings on one side often create a weak load-carrying member. This type of construction practice
results in a much more complicated system. Its geometric asymmetry (variations in location and geometry of structural members) and material asymmetry (mass and stiffness variations within and/or among structural members) could significantly affect the behaviour of building when subjected to an earthquake. One of the inevitable consequences is the torsional motion of the building, which may occur whenever the center of mass and the center of rigidity of the structure (the resultants of the resisting forces are located here) do not coincide. When subjected to earthquake ground motion, an asymmetric structure will undergo both translational and rotational motion even if the earthquake excitation is unidirectional. The structure may also undergo an "accidental" torsional motion even though the building plan is symmetric. This is because the building is usually not perfectly symmetrically built and the building’s base can rotate due to the spatial variations in ground motion. The torsional motion is harmful because it may significantly magnify the displacements and forces induced in certain structural elements. For a building structure that is expected to be strained into the inelastic range, torsional motions are believed to lead to additional displacement and ductility demands and to a failure of the structure (Humar and Kumar 1998).

*Diaphragm rigidity and its role in load sharing* - The rigidity or flexibility of a diaphragm is a relative measurement. If the in-plane deflection of a roof diaphragm is much smaller than deflections in vertical lateral load resistant members, such as shear walls, then the roof is said to be rigid, and vice versa. This property of a diaphragm will directly determine the load distribution among components. In general, if a diaphragm is believed rigid, it will distribute the horizontal forces to vertical members in direct proportion to their relative stiffness; on the other hand, if the diaphragm is flexible, it will distribute the forces to vertical members based on tributary areas, independent to their stiffness.
Consider the system as shown in Figure 2.1. If a rigid diaphragm is assumed, the vertical lateral load resistant members are represented by springs. With an eccentricity \( e = 0.1L \), the lateral load is applied to the diaphragm in a non-uniform way.

![Figure 2.1 Load distribution in vertical members when diaphragm is rigid](image)

By using the equations of equilibrium, the relationships among force, stiffness, and displacement are defined as

\[
\sum F_x = 0 \quad k_1 \left( x + \frac{L}{2} \theta \right) + k_2 x + k_3 \left( x - \frac{L}{2} \theta \right) = V \quad (2.2)
\]

\[
\sum F_y = 0 \quad k_4 \left( y + \frac{b}{2} \theta \right) + k_5 \left( y - \frac{b}{2} \theta \right) = 0 \quad (2.3)
\]

\[
\sum M = 0
\]

\[
k_1 \left( x + \frac{L}{2} \theta \right) \frac{L}{2} - k_3 \left( x - \frac{L}{2} \theta \right) \frac{L}{2} + k_4 \left( y + \frac{b}{2} \theta \right) b - k_5 \left( y - \frac{b}{2} \theta \right) b = 0.1VL \quad (2.4)
\]

or in matrix form
\[
\begin{bmatrix}
 k_1 + k_2 + k_3 & 0 & (k_1 - k_3)\frac{L}{2} \\
 0 & k_4 + k_5 & (k_4 - k_5)\frac{b}{2} \\
 (k_1 - k_3)\frac{L}{2} & (k_4 - k_5)\frac{b}{2} & (k_1 + k_3)\frac{L^2}{4} + (k_4 + k_5)\frac{b^2}{4}
\end{bmatrix}
\begin{bmatrix}
 x \\
 y \\
 \theta
\end{bmatrix} = 
\begin{bmatrix}
 V \\
 0 \\
 0.1VL
\end{bmatrix}
\] (2.5)

In the equations, \(x\), \(y\) and \(\theta\) represent the translations in the horizontal and vertical directions, and the roof rotation about the center of mass, respectively. The reactions or spring forces are

\[
R_1 = k_1\left(x + \frac{L}{2}\theta\right)
\]

\[
R_2 = k_2 x
\]

\[
R_3 = k_3\left(x - \frac{L}{2}\theta\right)
\] (2.6)

\[
R_4 = k_4\left(y + \frac{b}{2}\theta\right)
\]

\[
R_5 = k_5\left(y - \frac{b}{2}\theta\right)
\]

For a special case, let

\[
k_1 = k_4 = 0.5k; \ k_3 = k_5 = 1.5k; \ k_2 = k
\]

\[
b = \frac{L}{2}
\] (2.7)

The new coordinates are

\[
x = 0.420\frac{V}{k}; \ y = 0.065\frac{V}{k}; \ \theta = 0.522\frac{V}{kL}
\] (2.8)

The spring forces and the corresponding displacements are
\[ R_1 = 0.341V; \Delta_1 = 0.681 \frac{V}{k} \]
\[ R_2 = 0.420V; \Delta_2 = 0.420 \frac{V}{k} \]
\[ R_3 = 0.239V; \Delta_3 = 0.159 \frac{V}{k} \]
\[ R_4 = 0.098V; \Delta_4 = 0.196 \frac{V}{k} \]
\[ R_5 = -0.098V; \Delta_5 = -0.066 \frac{V}{k} \]

The new position of diaphragm under force \( V \) is shown as the dashed lines in Figure 2.1. As assumed, the horizontal forces supported by vertical members are proportional to their relative stiffness. It is also seen that the small eccentricity of the lateral force causes a torsional movement, which adds shear forces to the transverse walls.

If the diaphragm is assumed to be flexible, for the system in Figure 2.1, the vertical members now are represented by fixed supports because of their relative high rigidity (Figure 2.2). Based on their tributary areas, if the stiffness in each vertical component is the same as that in the case presenting rigid diaphragm, the reaction forces combined with the corresponding displacements are listed in (2.10). The deformed diaphragm shape is indicated by the dashed lines in Figure 2.2. For easy comparison, these two groups of results are combined in Table 2.1.

\[ R_1 = 0.3625V; \Delta_1 = 0.725 \frac{V}{k} \]
\[ R_2 = 0.5V; \Delta_2 = 0.5 \frac{V}{k} \]
\[ R_3 = 0.1375V; \Delta_3 = 0.092 \frac{V}{k} \]
\[ R_4 = R_5 = \Delta_4 = \Delta_5 = 0 \]
In these examples, the effects of diaphragm and its rigidity on the load sharing mechanism are demonstrated. The diaphragms in both cases represent two extreme situations of either a completely rigid or completely flexible system. The actual diaphragm may lie between these cases. From these results, it is concluded that the diaphragm rigidity is a significant factor in the load sharing mechanism, which controls the load distribution.
among the vertical members.

2.2. Literature review

2.2.1. Numerical models

In recent decades, analysis and modeling of wood light-frame buildings have been of great interest. This work was initialized after researchers recognized that development and application of two-dimensional models would not provide an explanation to the system behaviour of three-dimensional buildings. In the early stage, models were usually oversimplified to overcome technical difficulties and the analyses were limited in linear and static considerations. In recent years, effort has been made to develop models suitable for nonlinear and dynamic analysis of wood light-frame buildings. Although each model had its own features and limitations, a common limitation found in these models was that they all heavily depended on the connection tests and fitted curves. This implies that most models could only be used in specified cases where the connectors' behaviour was known. This dependency greatly prevented the model being applicable for predicting the general performance of wood light-frame buildings. The models for the study of light-frame buildings usually were implemented into specially developed programs. Commercially available computer software was also used for this purpose, but it lacked necessary elements to efficiently model the materials and joints found in wood buildings (Polensek 1983). In this situation, user-defined elements were necessary.

In the analytical field, one of the earlier attempts of modeling entire buildings was made by Chehab (1982), which involved a linear seismic analysis of a typical wood frame house. Even though the element mesh and input properties were approximate, the results pictured the effects observed in earthquake-damaged houses, including torsional effects
resulting from a nonsymmetrical arrangement of the shear walls, as shown in Figure 2.3.

Figure 2.3 Seismic response of a building modeled by Chehab (1982)

Gupta and Kuo (1987) developed a simple linear elastic building model containing seven “superelements” and nine global degrees of freedom to analyze the building tested by Tuomi and McCutcheon (1974). The “superelement” was based on the shear wall model built in their previous work (Gupta and Kuo 1985) to represent shear walls, flange walls (walls perpendicular to shear walls or loading direction), roof diaphragms and ceiling in this model. Among of the nine global degrees of freedom, five \( (u_1 - u_5) \) were defined as the horizontal displacements of the top of four walls and one roof ridge, two \( (v_1 \text{ and } v_2) \) as vertical displacements, and two \( (\Delta v_1 \text{ and } \Delta v_2) \) as uplift along one wall, as shown in Figure 2.4. The study made an effort to identify the influence of other structural components on the building performance. The study concluded that, contrary to the observation in an individually loaded shear wall where the stud uplift and the separation between studs and sole plates were apparent, those phenomena in a building were negligible. It also concluded that the roof component only contributed to the torsional deformation and its influence on the lateral stiffness is minimal. The model was suitable only for small deformation within a linear elastic range and could not predict the entire load-displacement path.
Schmidt and Moody (1989) developed a static model (RACK3D) by using a simple energy approach to analyze three-dimensional wood light-frame structures. This model extended the previous work on woodframe shear walls by Tuomi and McCutcheon (1974), combined with nonlinear fastener models developed by Foschi (1977) and McCutcheon (1985) to predict the nonlinear response of structures to lateral monotonic loading. In this model, all the walls with the same height were connected to a rigid roof diaphragm at a single point, which was located at the mid-span of a shear wall. Walls carried only in-plane racking forces without considering out-of-plane bending or torsion. The model was verified with the tests conducted by Tuomi and McCutcheon (1974) and Boughton and Reardon (1984), in which all the test specimens were single storey houses. The results indicated reasonable predictions in loads and deformations. The load sharing among walls and torsional motion of the building due to eccentric load condition were also studied. The deformations in the shear walls in this study were limited to 0.1 in (2.54 mm) and 0.2 in (5.08 mm). Within this range, the nonlinearity of the structure was not significant.

Kasal et al. (1994) used ANSYS finite element software to statically analyze a one-storey light-frame house, tested by Phillips (1993). In the model (Figure 2.5), the roof and floor systems were considered to be linear and were represented by superelements, while...
shear wall systems were assumed to be nonlinear in shear and linear in bending and torsion. Quasi-superelements comprising truss and diagonal spring were constructed to energy-equivalently represent the shear walls. The nonlinear characteristics of a shear wall were solely described by the diagonal springs, whose nonlinearities were obtained from experimental shear wall results. Intercomponent connections were modeled as nonlinear one-dimensional elements similar to that of an individual nail connector. When loaded monotonically, the model can provide good agreements to the tests in terms of lateral displacements and reaction forces in shear walls; however, the model could not accommodate the stiffness degradation of the structure, which is necessary in modeling a structure subjected to cyclic and dynamic loading. Due to the use of diagonal spring elements in wall members, a large number of shear wall tests had to be conducted to obtain the representative load-displacement characteristics for diagonal springs, whereas the single nail connector tests seemed not helpful. Finally, the study suggested that load sharing among shear walls was a function of shear wall stiffness, roof diaphragm action, and intercomponent stiffness.

Figure 2.5 A three-dimensional finite element model built by Kasal et al. (1994)
In 1995, Foschi developed a three-dimensional finite element diaphragm analysis program (DAP-3D) for a project carried out at the University of Western Ontario (UWO) to examine the performance of typical wooden houses subjected to wind loads (Case and Isyumov 1995). The test houses were composed of vertical shear walls and pitched roofs with a reduced-scale of 1:100. The wind loads were applied to houses by the turbulence generator at the Boundary Layer Wind Tunnel Laboratory of the UWO. The program, which modeled the wood components linear-elastically, was used to obtain influence coefficients for reaction forces at the foundation, structural displacements, and nail forces. The data was then compared with National Building Code of Canada (NBCC) code values.

In a project funded by the Forest Renewal BC (FRBC), a finite element program, PANEL, was developed by Foschi (1997). This program contained many features that either were required in a three-dimensional structural analysis or could be expanded from a two-dimensional space into a three-dimensional space, although it was designed for analyzing two-dimensional light-frame structures under static monotonic loading. For example, in the program, each node was provided with seven degrees of freedom (three translations and three rotations plus one twist), and by using plate element, the out-of-plane deformation can be calculated. The program allowed multiple and combined loading conditions to be applied to a diaphragm simultaneously. These loads included distributed loads acting normal to the panel plane and concentrated loads acting in the panel plane. Some of these loads can be incremental while the others were constant. In the panel to frame connections, besides the lateral deformations, the withdrawal and rotation of a nail connector were also analyzed. The program also tried to consider as many as possible variables affecting the behaviour of a structural component, which provided generality in the evaluation of the response of different
panels with different constructions. A diaphragm can have sheathing panels on both sides with different material properties. The analysis was also applicable to diaphragms with openings. It was the first light-frame structural analysis to consider the influence of insulation material filled between the two sheathing layers. All these features combined with the structure of the model in the PANEL program laid a foundation on which the models in a three-dimensional space can be developed. The major limitation of the PANEL program was that the behaviour of nail connections was predicted based on the experimental backbone functions. Therefore, it could be used only for monotonic pushover loading conditions.

In 1997, Tarabia and Itani presented a three-dimensional dynamic model for light-frame wood buildings. The diaphragms in the model were divided into a few sections with four master nodes for each section. These master nodes were located along the boundaries of a diaphragm connecting to other diaphragms and internal degrees of freedom were condensed out during the solution. The model did not consider out-of-plane deformations in the diaphragms and assumed that the panels can deform in shear only. Nail connections between the panel and frame members were modeled as two perpendicular decoupled nonlinear spring elements with the hysteresis behaviour proposed by Kivell et al. (1981). This connection model, similar to other connection models discussed in Section 2.2.5, used a few linear functions to represent the connection's loading history. This study covered the effect of aspect ratio on the rigidity of horizontal diaphragms, load distribution mechanisms, asymmetric lateral-load resisting systems, and transverse walls. It was found that the rigidity of the horizontal diaphragm was influenced by the aspect ratio. As the aspect ratio increased, the rigidity of the diaphragm decreased. The amount of seismic forces carried by the partition walls depended on the length and rigidity of roof diaphragm, and the stiffness and
aspect ratio of the partition walls. The study also concluded that using the asymmetric lateral-load resisting system induced torsional moments in the building and the transverse walls can carry a significant amount of shear forces generated from the torsional moment.

More recently, a hybrid approach combining deterministic and stochastic methods to model light-frame buildings was proposed (Kasal et al. 1999). In this approach, a complicated 3D building was first subjected to a very simple loading history that produced a cyclic response. By using system identification techniques, the response was then fitted into a simplified two-dimensional model, for example, a shear-building model with a general differential hysteresis developed by Foliente (1995), with only a few degrees of freedom. This shear-building model was based on the assumptions that (1) the entire floor systems were infinitely rigid as compared to the columns, and (2) the rigid floor systems remained horizontal during the ground motion. Stochastic analysis or Monte Carlo simulations could be applied to the model, where the system properties and loads were treated as random variables, under an ensemble of loads to determine a possible range of loading scenarios. Finally, the resultant response statistics would be used to describe the behaviour of the actual 3D buildings. From the above descriptions it can be seen that the model was only capable of predicting the load-displacement behaviour of a particular three-dimensional building based on the behaviour of each individual component. It did not provide any information on the system effects, such as torsional motion, load sharing, and so on.

A 3D dynamic time-history analysis was carried out in Canada (Ceccotti et al. 2000) to investigate the influence of diaphragm flexibility on ultimate state of a four-storey symmetric wood frame building. A three-dimensional non-linear dynamic analysis program (DRAIN®-3D) with a pinching nail model was used in the study. It was concluded that the
flexible floors could reduce the capacity of the wood building by approximate 15% when compared to the rigid floor case. The study was used to quantify the action reduction factor (ARF) in design codes, whereas the influence of varied relative stiffness ratio of floor diaphragms and shear walls on building performance including load sharing and building deformation was not addressed.

2.2.2. Experimental studies

Experiments of full-scale light-frame houses were seldom performed in a laboratory environment in the early stages of study of wood light-frame building due to technical and physical difficulties. Polensek (1983) summarized a few of such experimental studies in a span of more than twenty years by Dorey and Schriever (1957), Hurst (1965), Yokel, et al. (1972), Tuomi and McCutcheon (1974), and Yancey (1979). These studies usually involved simple and symmetric building specimens, loaded statically. None of these tests related with numerical analysis, and often the test results were incomplete. Entering 1980s, more efforts were made on experimental studies of full scale light-frame buildings by Hirashima, et al. (1981), Boughton (1982-1988), Sugiyama et al. (1988), Steward et al. (1988), Yasumura et al. (1988), Jackson (1989), and Phillips et al. (1993). These studies tried to understand the structural performance and characteristics of a complete wooden building. Some of them served as a verification of the design values. In general, the tests consisted of several stages at which the structural components were added to the building gradually to determine the influence of the components on structural behaviour of the building. Investigations of lateral load carrying capacity and drift, load sharing and interaction among components, and variations in structural materials in a full-scaled house were the main objectives. Then the test buildings were developed in irregular arrangement of interior walls and openings with
floor and roof installed. The level of the houses ranged from one storey to three storeys. In most tests, only static lateral loads were applied to the tested specimens. In some cases, cyclic lateral loads were applied to simulate wind loads.

One of the most prominent and comprehensive ongoing projects investigating light-frame buildings under dynamic earthquake ground motion is the CUREE-Caltech wood-frame project (Hall 2000). This is a $6.9 million FEMA (Federal Emergency Management Agency)-funded effort administered by California Office of Emergency Services. The goal of the project is to improve the seismic performance of wood-frame construction through development of cost-effective retrofit strategies, changes in design and construction procedures, and education. The project started in 1998 and is expected to be finished in three to four years. In the Testing and Analysis, one of five elements of the project, the plan incorporates five main research tasks, centering on the shake table tests of large-scale wood-frame systems (Filiatrault et al. 2000). The test specimens include conventional wood light-frame structures from simplified box-type models (Task 1.1.3), full-scale two-storey single houses (Task 1.1.1) to multi-storey apartment buildings (Task 1.1.2). The simplified box-type models (Task 1.1.3) have been tested at the University of British Columbia as part of the work in this thesis. It is an integral component of a parallel study funded by the Forest Renewal BC (FRBC). The test results will be described in detail in the subsequent chapters.

In Task 1.1.2, a two-storey single-family house of 4.88 m × 6.10 m (16 ft × 20 ft) in plan (Figure 2.6), following current California residential construction practice, was tested under the 1994 Northridge earthquake ground excitations. In 2000, an International Benchmark was being organized in which researchers and design professionals, inside and outside California, as well as the international communities were invited to blind-predict the inelastic

Chapter 2 Background
seismic response of this tested two-storey wood-frame building. This provided a unique opportunity to assess the capacity of available numerical models incorporating widely different levels of sophistication and to foster cooperation between the CUREE-Caltech Woodframe Project and other related research activities. In this blind prediction, five international teams completed the benchmark exercise by adopting different types of numerical models. The results ranged from poor to excellent when compared to the test results (Folz et al. 2001). All teams modeled the test building in multiple phases. In general, the test data of the connections between panel and frame were first fit into a hysteresis model to develop a full shear wall model. This shear wall was subsequently reduced to a single spring element. These nonlinear spring elements were then assembled to represent the entire building. The roof and floor diaphragms were assumed to be rigid, which was validated with the test results. The study concluded that almost all commercially available general purpose structural analysis programs did not contain hysteresis models that are suitable for wood fasteners or wood components. It was also found that the appropriate level of viscous damping to be used in the modeling was difficult to determine. The reason was that at low levels of structural response, the hysteresis elements may not dissipate any significant amount of energy, whereas at high levels of structural response, viscous damping may be entirely overshadowed by the energy dissipation of hysteresis mechanisms that were activated in the structure. Each participating team commented that the nonlinear dynamic analysis was a time-consuming task, even if an oversimplified structural model was used. This benchmark study indicated that a greater research effort is required to advance the knowledge base of numerical models for woodframe structures, before these analytical tools can be used effectively in the seismic design process.
The Earthquake 99 Project was another major large-scale study of wood light-frame buildings under earthquake loading conditions, which was designed and managed by TBG Seismic Consultants and Simpson Strong-Tie in collaboration with the University of British Columbia (Pryor et al. 2000). The project was to investigate the seismic performance of narrow Simpson Strong-Walls and conventional shear walls complying with the 1997 Uniform Building Code of the United States by testing a series of one- and two-storey buildings (Figure 2.7). In these buildings, the primary lateral force resisting system consisted of either four conventional shear walls, 1.22 m (width) by 2.44 m (height), or four Simpson Strong-Walls, 0.61 m (width) by 2.44 m (height), installed at both ends of two exterior walls parallel to the shaking direction. A specially designed unidirectional shake table was constructed to accommodate the test specimens with plan dimensions of 6.10 m by 7.62 m.
(20 ft by 25 ft) and an inertial weight of 200 kN (45,000 lbs). The input earthquake histories were the Canoga Park record and the Sherman Oaks record of the 1994 Northridge earthquake. The peak ground acceleration (PGA) of the Canoga Park record was scaled up 20% from the original one to 0.50g and was then further increased after the first test to reach 0.63g. The PGA of the Sherman Oaks record was 0.45g. Preliminary observations indicated that the permanent drift in the building using Simpson Strong-Walls was substantially smaller than that using the conventional shear walls. The tests also showed that the Simpson Strong-Wall system could sustain multiple large earthquake events better than its counterpart.

(a) One-storey test building
It is worth mentioning that in the past few decades, a large number of wood building tests have been carried out, both statically and dynamically, in Japan. Most of these tests were focused on the traditional Japanese post-beam and bracing systems. Only a small number of tests were carried out to study the North American 2 × 4 construction. After the 1995 Kobe earthquake, a number of Japanese researchers conducted shake table tests to study the dynamic performance of shear walls (Yamaguchi and Minowa 1998), wooden houses (Kohara and Miyazawa 1998, and Ohashi et al. 1998), and wood-framed houses with bracing systems (Tanaka et al. 1998). These tests on the structural components or the traditional Japanese building systems are not discussed in this thesis, which specially focuses on light-frame buildings.

Figure 2.7 Shake table tests of Earthquake 99 project

(b) Two-storey test building
2.2.3. Studies on the system effects in building

A light-frame building is an assembly of two-dimensional components, joined by connectors. It is inevitable that load sharing and interaction among the components exist when the building is loaded, which is the major distinct difference in behaviour of a building from the individual components. Identifying the load sharing as well as the load paths, however, is not an easy task, because it can be influenced by many factors. In early studies, this fundamental phenomenon was largely ignored since the emphasis was placed mainly on isolated building components. This issue has gradually attracted researchers' attention, along with the development of analyses and experiments of building systems, but no systematic study has been conducted to provide a general understanding of the load sharing mechanism. The major factors believed to affect the load sharing among the components include stiffness of wall components, rigidity of floor/roof systems, mass distribution within the building (under dynamic load), boundary conditions (how the walls being fixed to the foundation), and inter-component connections.

In a full-scale house test under cyclonic wind loads, Boughton (1988) concluded that the load did not always follow paths through the major structural components and the supporting boundary conditions of the components affected the stiffness, ultimate load, and failure characteristics of components under test. This testing also suggested that the components behaved differently in a building test from the prediction of component model. The design methods based on the component studies may give unconservative results; however, this could be compensated by redundancy of the whole system.

A comprehensive study investigating the system effects in buildings was conducted by Phillips et al. (1993). With sufficient instrumentation, the tests provided extensive data,
which were often used by other researchers in later studies. The overall dimensions of the test building were 4.88 m wide by 9.75 m long (16 ft by 32 ft) (Figure 2.8). The roof trusses spanned in the long direction of the building. The house had three separate rooms with doors and windows in different locations to create an asymmetric structural layout. Concentrated loads were applied to the top corners of the shear walls. Considerable efforts were made to instrument the building to obtain the internal force distributions. Eight load cells were embedded into shear walls to measure the horizontal and vertical loads in each shear wall. The gross displacement at the top of each shear wall, the slip at the base of each shear wall, the uplift of the loaded side, and the transverse displacement of the building were measured by linear variable differential transformers (LVDTs). The study found that the roof diaphragm had contributed significantly to lateral load sharing among the vertical shear walls of the building. The roof behaviour was much closer to a completely rigid response than a completely flexible response, as was typically assumed for the design of light-frame buildings. The observed load distributions were a function of the shear wall stiffness, wall position within the building, and the size of the building. This finding was supported by the evidence observed in other tests (Stewart et al. 1988, Yasumura et al. 1988). It was concluded that the walls perpendicular to the loading direction carried a percentage of the lateral load applied to the building, which was localized around the connection of the transverse walls with the shear walls. The percentage varied between 8% and 25%, and decreased with increasing applied load. The transverse walls did not contribute to load sharing among the shear walls. Similar results were also obtained by the tests conducted by Sugiyama et al. (1988) and Jackson (1989).
Figure 2.8 Three-dimensional view of test structure (Phillips 1993)

A study was carried out at Lund University (Andreasson 1999) to investigate influences of the diaphragm interaction on the distribution of dead load in a wood frame system. The author questioned the present design approach, in which all structural components were designed separately, with no consideration of load sharing in the three-dimensional structure because the approach assumed that the overall performance of a structural system subjected to a horizontal loading can be simplified to two-dimensional analyses, where only the shear walls were active. In the study, a static non-linear finite element model was developed using ANSYS software, of which the multi-storey timber frame building was idealized as horizontal and vertical diaphragms connected by springs. The investigation was focused on the distribution of dead load along shear walls and the transfer of dead load via transverse walls. The study concluded that the distribution of dead
load and the uplift load at anchorage were affected by sheathing configuration, wall length, panel dimension, anchorage stiffness, dead load magnitude, and number of storeys. For example, the dead load transferred to the anchorage in a two-side sheathed shear wall was larger than that in a single-side sheathed shear wall. It was also true in a longer shear wall vs. a shorter shear wall. These results indicated a smaller uplift in the end studs due to increased share of dead load. It was found that the distribution of dead load among studs was not linearly increased with increasing dead load. In a multi-storey building, it seemed that the top floor obtained the largest potion of dead load while the ground floor obtained the smallest, which implied that the multi-storey system acted as one diaphragm over all storeys, and its total load capacity was not a simple multiplication of the load of one storey by the number of storeys.

Foliente et al. (2000) presented preliminary test results of an L-shaped wood frame house in Australia. The primary objectives of the project were to improve the understanding of load sharing and distribution within a light-frame wood building and to generate data needed to validate building analytical models (Figure 2.9). The results from the first phase tests indicated a significant load sharing within the whole building. For example, Walls 3 and 4 took a significant in-plane load when the load was applied to Wall 2 only. This load transfer was partially due to the end walls (perpendicular to shear walls) and partially due to the roof system. Because Wall 3 was not connected to the roof trusses, the load transferred to Wall 3 was only via the end walls. This observation differed from that of the tests by Phillips et al. (1993), which concluded that the transverse walls in a house did not contribute to load sharing among the shear walls, as discussed previously. At this stage, only static tests within elastic range were performed. Further study on more loading configurations, the
stiffness of roof diaphragm, and the torsional stiffness of the whole house is under way. Dynamic testing is also scheduled. Finite element analysis will be in place to conduct parametric and sensitivity studies.

![Diagram of test model framing and plan view](image)

(a) Test model framing  
(b) Plan view and wall numbering

Figure 2.9 L-shaped test house (Foliente et al. 2000)

2.2.4. Testing methods

The most commonly used testing methods in investigating the structural performance of wood light-frame buildings fall in three categories: static monotonic testing method, quasi-static or cyclic testing method, and dynamic testing method. Briefly, they are described as follows.

*Static monotonic testing method* – In experimental study of a wood system, static loading is usually first applied. The approach is fairly simple to provide basic static strength and displacement characteristics of the structure. It is useful because most seismic design codes require that a building be designed to resist specified equivalent static forces. Under static loading, the behaviour of the system can be described by a simple load-displacement
relationship (Figure 2.10).

![Graph showing load-displacement relationship](image)

Figure 2.10 General load-displacement relationship of a wood structure

Unlike steel materials, wood materials do not have an obvious yield point, and the load-displacement relationship presents nonlinearity. To characterize the structures, important parameters, such as stiffness, ductility, ultimate strength, and displacement, have to be defined. Ductility is thought to be the most important quality because it is the ability to deform beyond the elastic limit without substantial reduction in strength and has been used as a criterion in establishing inelastic design response of wood buildings. Most of design procedures (codes) rely on the results from these simple static push-over tests and models on the structural components. Sometimes, the results can be misleading, because

1. the behaviour of wood structures is heavily loading history dependent, which means that the performance of a structure under static loading may not be the same as the performance under seismic loading; and
2. the good performance (e.g. high ductility) of individually tested components does not imply the same performance of an entire building under the same loading condition.

*Cyclic testing method* – As an important step toward a better understanding of the performance of wood structures under wind and seismic loading and improvement in seismic
design methods, a quasi-static or cyclic test is carried out. Cyclic tests are still within the static region, because the loading speed is so low that no dynamic effect or inertia forces can be produced. It can be seen from the literature review that wind loads applied on wood structures are usually simulated as cyclic loading because of their much lower frequencies and cyclical nature. Under cyclic loading, the load-displacement relationship of a wood based structural system is governed by hysteresis characteristics (Figure 2.11) which have been identified by many researchers (Foliente 1995):

(1) nonlinear, inelastic load-displacement relationship without a distinct yield point;
(2) progressive loss of lateral stiffness in each loading cycle or stiffness degradation;
(3) loss of strength when cyclically loaded to the same displacement level or strength degradation; and
(4) pinched and usually asymmetric hysteresis loops.

Figure 2.11 General hysteresis loops of a wood structure with a monotonic push-over curve

One of the purposes of doing cyclic tests is to understand the energy dissipation mechanism in a wood structure. The area of hysteresis loops is used as a measure of the energy dissipated by the structural components when subject to a cyclic loading. Clearly, wider loops will be beneficial to the structure, because more energy can be dissipated to
postpone the structure failure. Most nailed wood components show, however, pinched or area-reduced hysteresis loops, meaning a reduced energy dissipation capability or reduced hysteresis damping of the structure. This conclusion seems inconsistent with the good performance of the wood structures. Researchers have found that the ability for a wood structure to sustain a larger deformation without significant deterioration of the strength (higher ductility) appears to be much more significant. It is believed that the pinched loop as well as the stiffness and strength degradation results from the crushing of the wood around the nail shank and slipping of the nail in panel and frame members. It is noted that the energy dissipation capacity of a structure computed from the area of hysteresis loops is highly influenced by the type of loading history (He et al. 1998). Therefore, the energy dissipation capacities of wood structures tested under different loading histories may not be comparable.

In cyclic tests, several different loading protocols are available, including ASTM-SPD (1993, 1996), CEN (1995), FCC-Forintek (1994) and others used in Australia and New Zealand. When ASTM-SPD and FCC-Forintek protocols were used, the tests often resulted in low cycle fatigue fracture in the nails connecting panels and frame members, which was not commonly observed in wood buildings subjected to earthquakes or dynamic tests. Under CEN short protocol only a few cycles (usually three) with high amplitude were applied, which caused unrecoverable damage in nail joints at the very beginning of the test; therefore, the test results were difficult to justify. Test results from different protocols were not directly comparable because of the difference in defining parameters. A project was under way to develop a consistently recognized international standard for cyclic load testing of wood-based structures (Karacabeyli 1996).
Dynamic testing method – Usually a dynamic test is performed on a shake table with the input of previous earthquake records as ground motion excitations. The excitations could be applied in one or multiple directions depending on the research needs and the capacity of the test facility. The test specimen is tied to the table, carrying a certain amount of mass on its top to generate the required inertial forces. Dynamic testing is preferred, because it reveals the actual behaviour of a system during an earthquake and provides more information than a static or quasi-static test to help better understand the seismic characteristics of a wood light-frame building. Of course, dynamic testing is more costly and more technically demanding. The scale of a test is also restricted in the limitation of dimensions and payloads of a shake table. In a dynamic test, besides the structural considerations of the test specimen, the earthquake records and inertial mass should be carefully selected. If required, an earthquake record could be modified to obtain the right acceleration amplitude, frequency components, and duration. The mass on the top of structure will decide the load applied to the structure under the chosen earthquake excitation. Using properly installed instrumentation, the entire response history of a test system under specified excitation can be recorded. A forced vibration test is usually applied to the test system before and after the main test to determine the natural frequencies and the corresponding mode shapes.

Another type of test, called pseudo-dynamic test, is often performed. The pseudo-dynamic test is performed quasi-statically and provides a realistic simulation of the response of structures exposed to dynamic loadings. In the pseudo-dynamic test, a record of an actual or artificially generated earthquake ground acceleration history is given as input data to the computer. The horizontal displacements of diaphragms, such as floors or roofs at which the mass of the building can be considered to be concentrated, are calculated for a small time.
step. These displacements are then applied to the structure by servo-controlled hydraulic actuators fixed to the reaction wall. The forces necessary to achieve the required deformation are measured and then sent to the computer, combined with analytically modeled inertial and viscous damping forces, for the next step of the calculation. The pseudo-dynamic test method offers the possibility to test very large models and to monitor closely the progression of damage in the structure. Due to high equipment costs, however, the number of suitable hydraulic actuators is a serious limitation, which limits the number of degrees of freedom that can be tested in the laboratory. The method is especially useful for confirmatory tests in full-scale where exact material and construction details can be reproduced, or when multi-point input loading is required.

2.2.5. Connection models

The behaviour of the nail connections between sheathing panels and frame members is generally accepted to be a governing factor in defining the overall performance of the building component. Under dynamic loading, the hysteresis properties of a connection play a central role in deciding its energy dissipation characteristics during the deformation history. It is essential, therefore, in almost every analytical study, to study and model the load-displacement relationship and the hysteresis loops. Foschi (1974, 1977, Figure 2.12) published a model representing the monotonic load-displacement characteristics of nail connections in wood diaphragms. Many researchers used and modified this model to suit specified testing conditions.
Stewart et al. (1988), Dolan (1989), Kasal and Xu (1997), and Davenne et al. (1998) also modeled the hysteresis loops of nailed wood joints in shear wall or diaphragm systems (Figure 2.13). A particularly notable procedure was developed by Baber, Noori and Wen (1981, 1985) and modified slightly by Foliente (1993), using 13 parameters to account for yielding, pinching, strength, and stiffness degradations of a wide variety of possible combinations of materials and joint configurations. Krawinkler et al. (2001) also developed a connector loading protocol for the CUREE-Caltech Woodframe Project.
Figure 2.13 Modeled hysteresis loops of nailed wood joints by various researchers

(a) Stewart, Dean and Carr’s model, 1988; (b) Dolan’s model, 1989;
(c) Kasai and Xu’s model, 1997; and (d) Davenne et al.’s model, 1998.

A commonly used approach in this type of modeling was to fit experimentally observed characteristics of the specified wood joint, with the model basically being a post-test product. The model can then predict the performance of a joint for which the model was developed under static monotonic and cyclic loading because usually the loading paths have very regular patterns. However, it was noted that these hysteresis models could not provide satisfactory results when the material properties and joint configurations varied from the modeled one. The fitted models may not necessarily predict the hysteresis behaviour and energy dissipation of a structure subjected to dynamic loading under which its hysteresis loops were no longer nicely shaped.

An alternative that can be used in modeling the connection was using a general, mechanics-based approach. This type of model was developed (Foschi 2000) and implemented into a subroutine HYST (as it was usually called) for the calculation of hysteresis loops in mechanical connectors from basic material properties of the connector and the embedment characteristics of the surrounding wood medium. This approach considered a
nail connector as an elastic-plastic beam acting on wood, modeled as a nonlinear medium which only acted in compression, permitting the formation of gaps between the beam and the medium. The model can then develop pinching as gaps were formed, and thus the energy dissipation in a connection’s deformation history can be accurately represented. The HYST subroutine could be inserted into any program requiring calculations of nail forces. The subroutine itself was a finite element program calculating the coordinates of the hysteresis loop at each time step, by solving the nonlinear problem of the structural response to the imposed history. Calculating the true response involved a complicated three-dimensional problem, but it can be estimated from basic stress-strain information on the structural member (nail) and from a simple representation of the nonlinear behaviour of the medium (wood) surrounding that member. A beam finite element formulation was used for the member, using high order interpolating shape functions to reduce the number of required elements. The main advantage of this approach was that, starting from basic material properties, it automatically adapted to any input displacement history. Instead of calibrating parameters, to some of which it may be difficult to assign a physical meaning, the approach used constants with which engineers are more familiar, such as moduli of elasticity, yield stress, etc (Foschi 2000). Because the model can well reflect the actual behaviour of general connections in a more reliable manner, it was implemented into the programs developed in this thesis work to make the predictions under cyclic and dynamic loadings possible. It should be noted that the model is more computational intensive in a dynamic analysis when comparing to other curve-fitting nail models because it needs to be invoked for each nail or key nails within a nail line for each time step. The finite element iterative procedures are called to solve this nonlinear problem. It is estimated that more than 90% of CPU time is
spent on the calculation of nail connection behaviour in the programs developed in this study. This shortcoming should not be considered serious when the methods discussed in Chapter 10 are employed. In addition, the computational effort is amply compensated by increased model reliability for dynamic analysis (Foschi 2000).

In this chapter, the literature relating modeling and testing of three-dimensional wood frame constructions is overviewed and some important issues are discussed. It is evident that most of previous work was focused on two-dimensional structural components or simplified three-dimensional buildings. In recent years, the study has shifted to three-dimensional buildings and some progress has been made. Based on the research needs in three-dimensional light-frame structures, this thesis work extends from previous research efforts to develop numerical models. The objective is to use these models as practical tools in predicting and understanding the behaviour of three-dimensional wood light-frame buildings under seismic loadings.
CHAPTER 3

THE STRUCTURAL MODEL

The first step in finite element structural analysis is the idealization of the actual structure (physical model) into a mathematical model. This idealization must consider the configurations and properties of the structural members, the loading conditions, and the boundary conditions. The structural members need to be represented by certain types of elements and their material properties have to follow some idealized rules. The complex loading conditions of the structure will be simplified, for example, as concentrated and distributed forces. The fixities of the structure to the foundation or the connections to the support are idealized as boundary conditions with or without prescribed displacements. A proper idealization of these components has significant influence on how the solution of a finite element analysis closely converges to the actual result. In this chapter, only the idealizations of structural members and material properties are discussed. The loading conditions and boundary conditions should be recognized in the individual cases.

Three types of elements are used in the model development to represent the members in wood light-frame buildings. These elements are panel elements, frame elements, and connection elements. The panel elements are approximated as thin plate elements in a three-dimensional field. Each frame element consists of four sides with each side modeled as a three-dimensional beam element. In this study, a general mechanics-based connection
model, HYST (Foschi 2000), is introduced into the newly developed finite element programs. Using this model allows the finite element programs to perform cyclic and dynamic analyses by calculating hysteretic loops in mechanical connections from basic material properties. The connection element in HYST models the behaviour of a nail and the areas of the panel and frame members that surround the nail. These areas are represented by non-linear spring elements. The nail itself is assumed to obey an elasto-perfectly plastic constitutive relation. The approach used in this model overcomes the disadvantages of commonly used empirical connection models, in which an algorithm is added to trace hysteresis loops fitted to experimental data of a given connection.

In the finite element approximation, a basic element unit contains the above-mentioned three types of elements in a sandwich pattern as shown in Figure 3.1. Each element unit has four nodes with six nodal degrees of freedom and one element degree of freedom (Figure 3.2). The nodal degrees of freedom are the three translations in \(x, y\) and \(z\) directions \((u, v, w)\), and the three rotations about \(x, y\) and \(z\) axes \((\text{Rot}-x, \text{Rot}-y, \text{Rot}-z)\). The element degree of freedom is the twist in \(x-y\) plane \((w_{xy})\). Connection elements have three degrees of freedom at each node \((u, v, w)\).

All three types of elements used in the model are isoparametric elements. An element is isoparametric if one set of nodes and shape functions, represented by natural coordinates \(\xi\) and \(\eta\), can be used to define both nodal displacements \((u, v, w)\) and nodal coordinates \((x, y, z)\). The isoparametric elements are useful because, by selecting appropriate shape functions, the sides of the elements between two nodes can be curved to closely model the deformation of real structures. Those elements have proved to be more effective and accurate than the traditional triangular and rectangular elements by using fewer elements.

Chapter 3 The Structural Model
Chapter 3 The Structural Model

Figure 3.1 Basic element unit

Figure 3.2 Nodal and element degrees of freedom
3.1. Panel element

The panel elements are represented by thin plate elements, which can be loaded by in-plane and/or out-of-plane forces and can deform in three-dimensional space. Under the action of both types of loads, the deflection of plate can be large in comparison with its thickness, but is still small when compared with the other dimensions. Therefore, the analysis has to be extended to include the strain of the middle plane of the plate (Timoshenko 1959, Ch. 13).

3.1.1. Assumptions

The following assumptions are applied to the plate element:

(1) The normals to the middle plane remain normal to it during deformation (Bernoulli-Euler Assumption),

(2) The change in the thickness of plate element is negligible,

(3) The buckling deformation of plate element is modeled by considering bending deflection amplifications due to the effect of axial compressive loads (large deflection),

(4) The plate element has a linear elastic material property and obeys orthotropic stress-strain relationships, and

(5) The extensional or compressive deformation of plate element is linear and the deflected shape of the element is defined by cubic polynomials.

3.1.2. Geometric and physical relationships

Based on assumptions (1), (2), and (3), when the plate element is subjected to axial (tension or compression) and bending deformations in its x-y plane, the displacement in z direction is only a function of x and y. The displacement fields in a plate element are, therefore, determined as the following,
where

\[ u_s = u - z \frac{\partial w}{\partial x} \]
\[ v_s = v - z \frac{\partial w}{\partial y} \]
\[ w_s = w(x, y) \]  

\[ (3.1) \]

and

\[ u_s, v_s, w_s \quad = \quad \text{the small displacements of a point in the middle plane of the plate element during deformation in the } x, y \text{ and } z \text{ directions, respectively.} \]
\[ u, v \quad = \quad \text{the small displacements caused by axial deformation only in the } x \text{ and } y \text{ directions, respectively.} \]

Considering a small section \( AB \) of the middle plane of the plate element in \( x \) direction, the strain due to displacements \( u \) and \( w \), referring to Figure 3.3, is

\[ \varepsilon'_x = \frac{ds - dx}{dx} = \frac{ds}{dx} - 1 = \left[ 1 + \frac{\partial u}{\partial x} \right]^2 + \frac{\partial w}{\partial x} - 1 \]  

\[ (3.2) \]

Using Taylor series expansion and ignoring third and higher order terms, the strain in Equation (3.2) approximates to

\[ \varepsilon'_x = 1 + \frac{1}{2} \left[ \frac{\partial u}{\partial x} + \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial w}{\partial x} \right)^2 \right] - \frac{1}{8} \left[ \frac{\partial u}{\partial x} + \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial w}{\partial x} \right)^2 \right]^2 + \cdots - 1 \]  

\[ (3.3) \]
The strain of the middle plane due to bending is

\[ \varepsilon_x^* = -z \frac{\partial^2 w}{\partial x^2} \]  

(3.4)

Thus the total normal strain in \( x \) direction is obtained

\[ \varepsilon_x = \varepsilon_x' + \varepsilon_x^* = \frac{\partial u}{\partial x} + \frac{1}{2} \left( \frac{\partial w}{\partial x} \right)^2 - z \frac{\partial^2 w}{\partial x^2} \]  

(3.5)

In the same way, the normal strain of the middle plane in \( y \) direction is

\[ \varepsilon_y = \varepsilon_y' + \varepsilon_y^* = \frac{\partial v}{\partial y} + \frac{1}{2} \left( \frac{\partial w}{\partial y} \right)^2 - z \frac{\partial^2 w}{\partial y^2} \]  

(3.6)

The shear strain in the middle plane of a plate element also results from axial deformation and bending. The component of shear strain due to the displacements \( u \) and \( v \) is (Equation (3.1) and Figure 3.4(a))

\[ \gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} - 2z \frac{\partial^2 w}{\partial x \partial y} \]  

(3.7)
The component \((\gamma_1 + \gamma_2)\) of shear strain due to the deflection in \(w\) direction can be obtained from Figure 3.4(b). Vectors \(a\) and \(b\), representing deformed edges of the element, are

\[
\begin{align*}
\mathbf{a} &= \begin{bmatrix} dx \\ 0 \\ \frac{\partial w}{\partial x} \end{bmatrix} \quad \text{and} \quad \mathbf{b} = \begin{bmatrix} 0 \\ dy \\ \frac{\partial w}{\partial y} \end{bmatrix}
\end{align*}
\]  (3.8)

\[
\mathbf{a} \cdot \mathbf{b} = |a| |b| \cos \alpha \approx dx dy \sin (90 - \alpha) = dx dy \sin (\gamma_1 + \gamma_2)
\]  (3.9)

Because

\[
\mathbf{a} \cdot \mathbf{b} = dx \cdot 0 + 0 \cdot dy + \frac{\partial w}{\partial x} dx \frac{\partial w}{\partial y} dy
\]  (3.10)

then

\[
\gamma_1 + \gamma_2 = \frac{\partial w}{\partial x} \frac{\partial w}{\partial y}
\]  (3.11)

Adding these two components (3.11) to the components in (3.7), the total shear strain is

\[
\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} + \frac{\partial w}{\partial x} \frac{\partial w}{\partial y} - 2z \frac{\partial^3 w}{\partial x \partial y}
\]  (3.12)

![Figure 3.4 Derivations of shear strain components](image)

(a) in \(x-y\) plane  (b) due to deflection in \(w\) direction

The stress-strain relationships of plate element are

*Chapter 3 The Structural Model*
\[ \sigma = D \varepsilon \] 

(3.13)

where

\[ \sigma^T = [\sigma_x \quad \sigma_y \quad \tau_{xy}] \]  

(3.14)

\[ \varepsilon^T = [\varepsilon_x \quad \varepsilon_y \quad \gamma_{xy}] \]  

(3.15)

\[ D = \begin{bmatrix}
\frac{E_x}{1-\nu_{xy}\nu_{yx}} & \frac{\nu_{xy}E_x}{1-\nu_{xy}\nu_{yx}} & 0 \\
\frac{\nu_{yx}E_y}{1-\nu_{xy}\nu_{yx}} & \frac{E_y}{1-\nu_{xy}\nu_{yx}} & 0 \\
0 & 0 & G_{xy}
\end{bmatrix} \]  

(3.16)

\( E_x, E_y \) = Young’s moduli.

\( G_{xy} \) = shear modulus.

\( \nu_{yx}, \nu_{xy} \) = Poisson’s ratios.

Equation (3.13) indicates that the plate element with orthotropic material properties loaded in tension or compression along the principal directions exhibits no shear strain with respect to these principal directions. Also, the equation shows that the deformation is independent of \( G_{xy} \). Similarly, a shear stress produces only shear strain. In other words, there is no coupling between tensile and shear strains.

### 3.1.3. Shape functions

The same shape functions can directly achieve the relationship between the element displacements at any point and the element nodal displacements. This is a main objective in the isoparametric finite element formulation. It is found that the generalized element displacements are linear combinations of the element nodal displacements, in which the coefficients are represented as the shape functions. The shape functions are defined in a natural coordinate system and usually are in the form of polynomials.
The element nodal coordinates in terms of natural coordinates are, referring to Figure 3.5

\[ x = x_c + \frac{\Delta x}{2} \xi \]
\[ y = y_c + \frac{\Delta y}{2} \eta \]

(3.17)

where

\( x_c, y_c \) = coordinates of plate element center.
\( \Delta x, \Delta y \) = side lengths of plate element.
\( \xi, \eta \) = variables of natural coordinate system.

The relation between the two coordinate systems is

\[
\begin{bmatrix}
\frac{\partial}{\partial x} \\
\frac{\partial}{\partial y} \\
\frac{\partial}{\partial \xi} \\
\frac{\partial}{\partial \eta}
\end{bmatrix} = J^{-1}
\begin{bmatrix}
\frac{\partial}{\partial \xi} \\
\frac{\partial}{\partial \eta} \\
\frac{\partial}{\partial \xi} \\
\frac{\partial}{\partial \eta}
\end{bmatrix} = \begin{bmatrix}
\frac{2}{\Delta x} & 0 & 0 \\
0 & \frac{2}{\Delta y} & 0 \\
0 & 0 & \frac{2}{\Delta x \Delta y}
\end{bmatrix}
\]

(3.18)

where \( J \) is Jacobian matrix.

Figure 3.5 Natural coordinate system

In the four-node plate element, let the degrees of freedom vector be
\[ \mathbf{a}^T = \left( u_1, v_1, \frac{\partial u_1}{\partial y}, \frac{\partial v_1}{\partial x}, w_1, \frac{\partial w_1}{\partial x}, \frac{\partial w_1}{\partial y}, \frac{\partial^2 w_1}{\partial x^2}, \ldots \right) \]

(3.19)

And then the displacements represented by shape functions and degrees of freedom are

\[ u = N_{01}u_1 + N_{02} \left( -\frac{\partial u_1}{\partial y} \right) + \cdots + N_{07}u_4 + N_{08} \left( -\frac{\partial u_4}{\partial y} \right) = \mathbf{N}_0^T \mathbf{a} \]

(3.20)

\[ v = L_{01}v_1 + L_{02} \left( \frac{\partial v_1}{\partial x} \right) + \cdots + L_{07}v_4 + L_{08} \left( \frac{\partial v_4}{\partial x} \right) = \mathbf{L}_0^T \mathbf{a} \]

(3.21)

\[ w = M_{01}w_1 + M_{02} \left( \frac{\partial w_1}{\partial x} \right) + M_{03} \left( \frac{\partial w_1}{\partial y} \right) + M_{04} \left( \frac{\partial^2 w_1}{\partial x^2} \right) + \cdots + M_{013}w_4 + M_{014} \left( \frac{\partial w_4}{\partial x} \right) + M_{015} \left( \frac{\partial w_4}{\partial y} \right) + M_{016} \left( \frac{\partial^2 w_4}{\partial x^2} \right) = \mathbf{M}_0^T \mathbf{a} \]

(3.22)

where

\[ N_{0i}, L_{0i}, M_{0i} = \text{shape functions.} \]

\[ \mathbf{N}_0, \mathbf{L}_0, \mathbf{M}_0 = \text{shape function vectors.} \]

From assumption (5), which states that the extensional or compressive deformations along the four sides of plate element are linear and the deflections of the element due to bending and buckling are in the third order (cubic), a set of polynomials can be constructed

\[ u = a_0 + a_1 \xi + a_2 \eta + a_3 \eta^2 + a_4 \eta^3 + a_5 \xi \eta + a_6 \xi^2 \eta + a_7 \xi^3 \]

(3.23)

\[ v = b_0 + b_1 \eta + b_2 \xi + b_3 \xi^2 + b_4 \xi^3 + b_5 \xi^2 \eta + b_6 \xi^3 \eta + b_7 \xi^4 \eta \]

(3.24)

\[ w = c_0 + c_1 \xi + c_2 \xi^2 + c_3 \xi^3 + c_4 \eta + c_5 \xi \eta + c_6 \xi^2 \eta + c_7 \xi^3 \eta + c_8 \xi^2 \eta^2 + \cdots \]

(3.25)

\[ + c_9 \xi \eta^2 + c_{10} \xi^2 \eta + c_{11} \xi^3 \eta + c_{12} \xi^2 \eta^2 + c_{13} \xi^3 \eta^2 + c_{14} \xi^2 \eta^3 + c_{15} \xi^3 \eta^3 \]

where \( u \) and \( v \) are related to the first three degrees of freedom and \( w \) to the remaining four degrees of freedom of one node in Equation (3.19). In the construction of shape functions,
the following derivatives of Equations (3.23), (3.24) and (3.25) are also necessary

\[
\frac{\partial u}{\partial \eta} = a_2 + 2a_3 \eta + 3a_4 \eta^2 + a_5 \xi + 2a_6 \xi \eta + 3a_7 \xi^2 \\
\frac{\partial v}{\partial \xi} = b_2 + 2b_3 \xi + 3b_4 \xi^2 + b_5 \eta + 2b_6 \xi \eta + 3b_7 \xi^2 \eta \\
\frac{\partial w}{\partial \xi} = c_1 + 2c_2 \xi + 3c_3 \xi^2 + c_7 \eta + 2c_8 \xi \eta + c_9 \eta^2 + 3c_{10} \xi^2 \eta + 2c_{11} \xi \eta^2 \\
+ c_{12} \eta^3 + 3c_{13} \xi^2 \eta^2 + 2c_{14} \xi \eta^3 + 3c_{15} \xi^2 \eta^3 \\
\frac{\partial w}{\partial \eta} = c_4 + 2c_5 \xi + 3c_6 \eta^2 + c_7 \xi + c_8 \xi^2 + 2c_9 \xi \eta + c_{10} \xi^2 + 2c_{11} \xi \eta^2 \\
+ 3c_{12} \xi^2 \eta^2 + 2c_{13} \xi^3 \eta + 3c_{14} \xi^2 \eta^2 + 3c_{15} \xi^3 \eta^2 \\
\frac{\partial^2 w}{\partial \xi \partial \eta} = c_7 + 2c_8 \xi + 2c_9 \eta + 3c_{10} \xi^2 + 4c_{11} \xi \eta + 3c_{12} \eta^2 + 6c_{13} \xi^2 \eta \\
+ 6c_{14} \xi \eta^2 + 9c_{15} \xi^2 \eta^2
\]

Substituting the four pairs of natural coordinates \((\xi, \eta)\) at nodes 1, 2, 3, and 4 shown in Figure 3.5 to Equations (3.23) - (3.30), respectively, the relations between deformations and natural coordinates yield

\[
u(\xi, \eta) = A \cdot a^o \quad v(\xi, \eta) = B \cdot b^o \quad w(\xi, \eta) = C \cdot c^o
\]

where

\[
u(\xi, \eta) = \left\{ \begin{array}{c} u_i \\ \frac{\partial u}{\partial \eta} \end{array} \right\} \quad v(\xi, \eta) = \left\{ \begin{array}{c} v_i \\ \frac{\partial v}{\partial \xi} \end{array} \right\} \quad w(\xi, \eta) = \left\{ \begin{array}{c} w_i \\ \frac{\partial w}{\partial \xi} \\ \frac{\partial w}{\partial \eta} \\ \frac{\partial^2 w}{\partial \xi \partial \eta} \end{array} \right\} \quad i = 1,..4
\]
A, B, and C are coefficient matrices and $a^0$, $b^0$ and $c^0$ are vectors for the unknowns $a_i$, $b_i$, and $c_i$, respectively.

The vectors $a^0$, $b^0$ and $c^0$ can be obtained by inverting matrices A, B, and C in Equation (3.31). Substitute the results of $a^0$, $b^0$ and $c^0$ into Equations (3.23), (3.24), and (3.25), respectively. Then, reorder them into the form of (3.20), (3.21), and (3.22). After considering the Jacobian transformations, the shape functions of plate element are

$$
N_{01} = 0.25 - 0.25\xi - 0.375\eta + 0.125\eta^3 + 0.375\xi\eta - 0.125\xi^3
$$

$$
N_{02} = -0.125\left(1 - \xi - \eta^2 + \eta^3 + 2\xi + \xi^2 - 2\eta^3 - \xi\eta^2 + \eta^3\right)\frac{\Delta y}{2}
$$

$$
N_{03} = 0.25 + 0.25\xi - 0.375\eta + 0.125\eta^3 - 0.375\xi\eta + 0.125\xi^3
$$

$$
N_{04} = -0.125\left(1 + \xi - \eta^2 + \eta^3 - \xi\eta - \xi^2 + 2\eta^3 + 2\xi + \xi^2 - 2\eta^3 - \xi\eta^2 + \eta^3\right)\frac{\Delta y}{2}
$$

$$
N_{05} = 0.25 + 0.25\xi + 0.375\eta - 0.125\eta^3 + 0.375\xi\eta - 0.125\xi^3
$$

$$
N_{06} = -0.125\left(-1 - \xi - \eta + \eta^2 + \eta^3 - \xi\eta + \xi^2 + \xi^3\right)\frac{\Delta y}{2}
$$

$$
N_{07} = 0.25 - 0.25\xi + 0.375\eta - 0.125\eta^3 - 0.375\xi\eta + 0.125\xi^3
$$

$$
N_{08} = -0.125\left(-1 + \xi - \eta^2 + \eta^3 + \xi\eta - \xi^2 - \xi^3\right)\frac{\Delta y}{2}
$$

$$
L_{01} = 0.25 - 0.25\eta - 0.375\xi + 0.125\xi^3 + 0.375\xi\eta - 0.125\xi^3\eta
$$

$$
L_{02} = 0.125\left(1 - \xi - \eta^2 + \xi^3 + 2\xi + \xi^2 - 2\eta^3 - \xi\eta^2 + \eta^3\right)\frac{\Delta x}{2}
$$

$$
L_{03} = 0.25 - 0.25\eta + 0.375\xi - 0.125\xi^3 - 0.375\xi\eta + 0.125\xi^3\eta
$$

$$
L_{04} = 0.125\left(-1 - \xi + \eta^2 + \xi^3 + 2\xi + \xi^2 - 2\eta^3 - \xi\eta^2 + \eta^3\right)\frac{\Delta x}{2}
$$

$$
L_{05} = 0.25 + 0.25\eta + 0.375\xi - 0.125\xi^3 + 0.375\xi\eta - 0.125\xi^3\eta
$$

$$
L_{06} = 0.125\left(-1 - \xi + \eta^2 + \xi^3 - \xi\eta + \xi^2 + \xi^3\right)\frac{\Delta x}{2}
$$

$$
L_{07} = 0.25 + 0.25\eta - 0.375\xi + 0.125\xi^3 - 0.375\xi\eta + 0.125\xi^3\eta
$$

$$
L_{08} = 0.125\left(1 - \xi + \eta^2 + \xi^3 - \xi\eta + \xi^2 - \xi^3\right)\frac{\Delta x}{2}
$$
\[ M_{01} = 0.25 - 0.375\xi + 0.125\xi^3 - 0.375\eta + 0.125\eta^3 + 0.5625\xi\eta - 0.1875\xi^2\eta - 0.1875\xi^3\eta + 0.0625\xi^4 - 0.1875\xi^5 + 0.125\xi^6 \]

\[ M_{02} = (0.125 - 0.125\xi - 0.125\xi^2 + 0.125\xi^3 - 0.1875\eta + 0.0625\eta^3 + 0.1875\xi\eta - 0.1875\xi^2\eta - 0.1875\xi^3\eta - 0.0625\xi^4 - 0.1875\xi^5 - 0.125\xi^6) \frac{\Delta x}{2} \]

\[ M_{03} = (0.125 - 0.1875\xi + 0.0625\xi^3 - 0.125\eta - 0.125\eta^2 + 0.125\eta^3 + 0.1875\xi\eta - 0.1875\xi^2\eta - 0.1875\xi^3\eta - 0.0625\xi^4 - 0.1875\xi^5 - 0.125\xi^6) \frac{\Delta y}{2} \]

\[ M_{04} = 0.0625(1 - \xi - \xi^2 + \xi^3 - \eta - \eta^2 + \eta^3 + \xi\eta + \xi^2\eta + \xi^3\eta - \xi^2\eta^2 - \xi^3\eta^2 + \xi^4) \]

\[ M_{05} = 0.25 + 0.375\xi - 0.125\xi^3 - 0.375\eta + 0.125\eta^3 + 0.5625\xi\eta + 0.1875\xi^2\eta + 0.1875\xi^3\eta + 0.0625\xi^4 + 0.1875\xi^5 + 0.125\xi^6 \]

\[ M_{06} = -0.125 - 0.125\xi + 0.125\xi^2 + 0.125\xi^3 + 0.1875\eta + 0.0625\eta^3 + 0.1875\xi\eta - 0.1875\xi^2\eta - 0.1875\xi^3\eta + 0.0625\xi^4 + 0.1875\xi^5 + 0.125\xi^6 \]

\[ M_{07} = (0.125 + 0.1875\xi - 0.0625\xi^3 - 0.125\eta - 0.125\eta^2 + 0.125\eta^3 + 0.1875\xi\eta + 0.1875\xi^2\eta + 0.1875\xi^3\eta + 0.0625\xi^4 + 0.1875\xi^5 + 0.125\xi^6) \frac{\Delta y}{2} \]

\[ M_{08} = 0.0625(-1 - \xi - \xi^2 + \xi^3 + \eta + \eta^2 - \eta^3 + \xi\eta - \xi^2\eta + \xi^3\eta - \xi^2\eta^2 + \xi^3\eta^2 - \xi^4) \]

\[ M_{09} = 0.25 + 0.375\xi - 0.125\xi^3 - 0.375\eta + 0.125\eta^3 - 0.5625\xi\eta - 0.1875\xi^2\eta - 0.1875\xi^3\eta - 0.0625\xi^4 - 0.1875\xi^5 - 0.125\xi^6 \]

\[ M_{10} = (-0.125 - 0.125\xi + 0.125\xi^2 + 0.125\xi^3 - 0.1875\eta + 0.0625\eta^3 - 0.1875\xi\eta + 0.1875\xi^2\eta + 0.1875\xi^3\eta - 0.0625\xi^4 - 0.1875\xi^5 - 0.125\xi^6) \frac{\Delta x}{2} \]

\[ M_{11} = 0.0625(-1 - \xi - \xi^2 + \xi^3 + \eta + \eta^2 - \eta^3 + \xi\eta - \xi^2\eta + \xi^3\eta - \xi^2\eta^2 + \xi^3\eta^2 - \xi^4) \]

\[ M_{12} = 0.25 + 0.375\xi - 0.125\xi^3 - 0.375\eta + 0.125\eta^3 + 0.5625\xi\eta + 0.1875\xi^2\eta + 0.1875\xi^3\eta + 0.0625\xi^4 + 0.1875\xi^5 + 0.125\xi^6 \]

\[ M_{13} = 0.0625(-1 - \xi - \xi^2 + \xi^3 + \eta + \eta^2 - \eta^3 + \xi\eta - \xi^2\eta + \xi^3\eta - \xi^2\eta^2 + \xi^3\eta^2 - \xi^4) \]

\[ M_{14} = 0.25 + 0.375\xi - 0.125\xi^3 - 0.375\eta + 0.125\eta^3 + 0.5625\xi\eta + 0.1875\xi^2\eta + 0.1875\xi^3\eta + 0.0625\xi^4 + 0.1875\xi^5 + 0.125\xi^6 \]

\[ M_{15} = 0.0625(-1 - \xi - \xi^2 + \xi^3 + \eta + \eta^2 - \eta^3 + \xi\eta - \xi^2\eta + \xi^3\eta - \xi^2\eta^2 + \xi^3\eta^2 - \xi^4) \]

\[ M_{16} = 0.25 + 0.375\xi - 0.125\xi^3 - 0.375\eta + 0.125\eta^3 + 0.5625\xi\eta + 0.1875\xi^2\eta + 0.1875\xi^3\eta + 0.0625\xi^4 + 0.1875\xi^5 + 0.125\xi^6 \]

\[ M_{17} = 0.0625(-1 - \xi - \xi^2 + \xi^3 + \eta + \eta^2 - \eta^3 + \xi\eta - \xi^2\eta + \xi^3\eta - \xi^2\eta^2 + \xi^3\eta^2 - \xi^4) \]

\[ M_{18} = 0.25 + 0.375\xi - 0.125\xi^3 - 0.375\eta + 0.125\eta^3 + 0.5625\xi\eta + 0.1875\xi^2\eta + 0.1875\xi^3\eta + 0.0625\xi^4 + 0.1875\xi^5 + 0.125\xi^6 \]

\[ M_{19} = 0.0625(-1 - \xi - \xi^2 + \xi^3 + \eta + \eta^2 - \eta^3 + \xi\eta - \xi^2\eta + \xi^3\eta - \xi^2\eta^2 + \xi^3\eta^2 - \xi^4) \]

\[ M_{20} = 0.25 + 0.375\xi - 0.125\xi^3 - 0.375\eta + 0.125\eta^3 + 0.5625\xi\eta + 0.1875\xi^2\eta + 0.1875\xi^3\eta + 0.0625\xi^4 + 0.1875\xi^5 + 0.125\xi^6 \]

\[ M_{21} = 0.0625(-1 - \xi - \xi^2 + \xi^3 + \eta + \eta^2 - \eta^3 + \xi\eta - \xi^2\eta + \xi^3\eta - \xi^2\eta^2 + \xi^3\eta^2 - \xi^4) \]

\[ M_{22} = 0.25 + 0.375\xi - 0.125\xi^3 - 0.375\eta + 0.125\eta^3 + 0.5625\xi\eta + 0.1875\xi^2\eta + 0.1875\xi^3\eta + 0.0625\xi^4 + 0.1875\xi^5 + 0.125\xi^6 \]

\[ M_{23} = 0.0625(-1 - \xi - \xi^2 + \xi^3 + \eta + \eta^2 - \eta^3 + \xi\eta - \xi^2\eta + \xi^3\eta - \xi^2\eta^2 + \xi^3\eta^2 - \xi^4) \]

\[ M_{24} = 0.25 + 0.375\xi - 0.125\xi^3 - 0.375\eta + 0.125\eta^3 + 0.5625\xi\eta + 0.1875\xi^2\eta + 0.1875\xi^3\eta + 0.0625\xi^4 + 0.1875\xi^5 + 0.125\xi^6 \]
The shape functions in (3.33), (3.34), and (3.35) are used to define the element displacements and coordinates in terms of nodal displacements and coordinates. To present all deformations displayed in (3.5), (3.6), and (3.12), the first and second derivatives from those shape functions are necessary:

\[
\frac{\partial u}{\partial x} = \frac{\partial N_0^T}{\partial x} = \frac{2}{\Delta x} \frac{\partial N_0^T}{\partial \xi} = N_{1x}^T a
\]

(3.36)

\[
\frac{\partial u}{\partial y} = \frac{\partial N_0^T}{\partial y} = \frac{2}{\Delta y} \frac{\partial N_0^T}{\partial \eta} = N_{1y}^T a
\]

(3.37)

\[
\frac{\partial v}{\partial x} = \frac{\partial L_0^T}{\partial x} = \frac{2}{\Delta x} \frac{\partial L_0^T}{\partial \xi} = L_{1x}^T a
\]

(3.38)

\[
\frac{\partial v}{\partial y} = \frac{\partial L_0^T}{\partial y} = \frac{2}{\Delta y} \frac{\partial L_0^T}{\partial \eta} = L_{1y}^T a
\]

(3.39)

\[
\frac{\partial w}{\partial x} = \frac{\partial M_0^T}{\partial x} = \frac{2}{\Delta x} \frac{\partial M_0^T}{\partial \xi} = M_{1x}^T a
\]

(3.40)

\[
\frac{\partial w}{\partial y} = \frac{\partial M_0^T}{\partial y} = \frac{2}{\Delta y} \frac{\partial M_0^T}{\partial \eta} = M_{1y}^T a
\]

(3.41)
\[
\frac{\partial^2 w}{\partial x^2} = \frac{\partial^2 M_0^T}{\partial x^2} a = \left( \frac{2}{\Delta x} \right)^2 \frac{\partial^2 M_0^T}{\partial x^2} a = M_{2x}^T a
\] (3.42)

\[
\frac{\partial^2 w}{\partial y^2} = \frac{\partial^2 M_0^T}{\partial y^2} a = \left( \frac{2}{\Delta y} \right)^2 \frac{\partial^2 M_0^T}{\partial y^2} a = M_{2y}^T a
\] (3.43)

\[
\frac{\partial^2 w}{\partial x\partial y} = \frac{\partial^2 M_0^T}{\partial x\partial y} a = \left( \frac{2}{\Delta x} \right)^2 \frac{2}{\Delta y} \frac{\partial^2 M_0^T}{\partial x\partial y} a = M_{2xy}^T a
\] (3.44)

3.2. Frame element

A frame element consists of four components around the edges of a panel element with “rigid” connections at corners. The rigid connection means that nails are not modelled and relative movements and rotations between nodes from two components are not allowed because the nodes share the same degrees of freedom. Three-dimensional beam elements are adopted to represent these side components. The frame can be loaded in all three directions and has the same deformation pattern as the panel element.

3.2.1. Assumptions

The following assumptions are used in frame beam element:

1. The plane sections remain plane during deformation and the shear influence through the cross-section is neglected,
2. The change in cross-sectional dimensions is negligible,
3. The \( P-\Delta \) deformation is modeled by considering bending deflection amplifications due to the effect of axial compressive loads (large deflection and stiffness updating at each step),
4. The beam elements have a linear elastic material property and obey homogeneous stress-strain relationships, and
5. The axial deformation (tension or compression) of beam element is linear and deflected shapes are defined by cubic polynomials.
3.2.2. Geometric and physical relationships

Beam elements, like the plate element, are also subjected to axial and bending deformations. The displacement in the direction perpendicular to the plane of the frame formed by four beam elements (z direction) is a function of only $x$ and $y$. Therefore, the displacement fields in a three-dimensional beam element are,

$$u_s = u - z \frac{\partial w}{\partial x} - y \frac{\partial v}{\partial x} \quad \text{(horizontal beam elements)}$$

$$v_s = v - z \frac{\partial w}{\partial y} - x \frac{\partial u}{\partial y} \quad \text{(vertical beam elements)}$$

$$w_s = w(x, y)$$

For a horizontal beam element, by using the same method as applied in Section 3.1.2, the strain due to tension or compression along its axis in $x$ direction and large deflections in both $x$-$z$ and $x$-$y$ planes is

$$\varepsilon' = \frac{\partial u}{\partial x} + \frac{1}{2} \left( \frac{\partial w}{\partial x} \right)^2 + \frac{1}{2} \left( \frac{\partial v}{\partial x} \right)^2$$

The strain due to bending in $x$-$z$ and $x$-$y$ planes is

$$\varepsilon'' = -z \frac{\partial^2 w}{\partial x^2} - y \frac{\partial^2 v}{\partial x^2}$$

The total normal strain in a horizontal beam element is

$$\varepsilon = \varepsilon' + \varepsilon'' = \frac{\partial u}{\partial x} + \frac{1}{2} \left( \frac{\partial w}{\partial x} \right)^2 + \frac{1}{2} \left( \frac{\partial v}{\partial x} \right)^2 - z \frac{\partial^2 w}{\partial x^2} - y \frac{\partial^2 v}{\partial x^2}$$

The shear strain can be obtained by referring to Figure 3.6.

$$\gamma_{xy} = \rho \frac{\partial \phi}{\partial x} = \rho \frac{\partial^2 w}{\partial x \partial y}$$
Similarly, the strains in a vertical beam element are

\[ \varepsilon_y = \varepsilon'_y + \varepsilon''_y = \frac{\partial \nu}{\partial y} + \frac{1}{2} \left( \frac{\partial w}{\partial y} \right)^2 + \frac{1}{2} \left( \frac{\partial u}{\partial y} \right)^2 - \frac{z}{2} \frac{\partial^2 w}{\partial y^2} - x \frac{\partial^2 u}{\partial y^2} \]  

(3.50)

\[ \gamma_{yx} = \rho \frac{\partial \phi}{\partial y} = \rho \frac{\partial^2 w}{\partial x \partial y} = \gamma_{xy} \]  

(3.51)

In the shear strain expressions (3.49) and (3.51), \( \rho \) is the radial distance of the cross-section of beam element, and \( \phi \) is rotational angle due to twist along the axis, which equals to \( \frac{dw}{dx} \) in \( x-z \) plane and \( \frac{dw}{dy} \) in \( y-z \) plane.

The stress-strain relationships in the beam elements are

\[ \sigma = E \varepsilon \]  

\[ \tau = G \gamma \]  

(3.52)

3.2.3. Shape functions

The beam element uses the same set of shape functions as does the plate element because they deform in the same pattern under the same degrees of freedom definition. Two more derivatives from those shape functions are required, however, to completely represent the deformations in (3.48) and (3.50) besides those shown in (3.36)-(3.44)
\[
\frac{\partial^2 u}{\partial y^2} = \frac{\partial^2 N^T_0}{\partial y^2} a = \left( \frac{2}{\Delta y} \right)^2 \frac{\partial^2 N^T_0}{\partial \eta^2} a = N^T_{2y} a
\]
(3.53)
\[
\frac{\partial^2 v}{\partial x^2} = \frac{\partial^2 L^T_0}{\partial x^2} a = \left( \frac{2}{\Delta x} \right)^2 \frac{\partial^2 L^T_0}{\partial \zeta^2} a = L^T_{2x} a
\]
(3.54)

3.3. Connection element

Connections between the panel and frame members are important components to be modeled in the numerical analysis of entire wood frame structures. The connections consist of mechanical fasteners (nails in this study) and the surrounding wood medium. It is generally recognized that the nonlinearity of the wood frame building subject to lateral loads is mostly due to the nonlinear connection behaviour. Under applied loads, the connection shows a hysteresis response resulting from the elasto-plastic characteristics of mechanical fastener and the nonlinear behaviour of wood medium interacting with the fastener. The accuracy and effectiveness of modeling the nail connection will have a significant influence on the modeling of the whole wood frame structure. In this study, a mechanics-based connection model HYST (Foschi 2000), representing the lateral deformation of connections in \( x \) and \( y \) directions and a nail withdrawal model representing the displacement of connection in \( z \) direction are implemented in the finite element programs. Given the relative displacements between panel and frame members, the both models output the reaction forces from the wood medium as required in stiffness calculations. It should be noted that, in this study, only the connections between panel and frame members are considered. The frame members (up to four three-dimensional beam elements) are considered being rigidly connected at corners.

In the current model, the lateral nail deformation is calculated in both \( x \) and \( y \) directions. Modeling a nail connection in its actual direction of loading is not preferred,
because its loading direction cannot be easily defined in each step and the force between nail and wood may not be necessarily in the same direction as the load. This randomness increases the level of difficulty in keeping track of nail and wood deformation and force history. In addition, wood is not a homogeneous material. At each angle, the force needs to be calculated by using, for example, Hankinson’s formulas. However, the currently used approach, approximating nail connection in $x$ and $y$ directions separately, provides great convenience in the calculation and storage of nail force and deformation. It also complies with the conventions in the finite element formulations, in which all virtual displacements (work) are calculated in $x$ and $y$ directions.

3.3.1. Assumptions

The following assumptions are used in the elements of the mechanics-based connection model:

(1) For simplicity, the lateral deformation in $x$-$y$ plane and withdrawal in $z$ of nail member are defined by individual models, and the frictional forces between the mechanical fastener and the surrounding wood medium are ignored,

(2) The mechanical fastener is modeled as a number of beam elements, obeying an elastic-perfectly plastic constitutive relation, and the wood medium acts nonlinearly in compression only,

(3) The plane sections of beam elements remain plane after deformation, and

(4) The $P$-$\Delta$ effects, due to axial compression loads in connection element, are considered.

3.3.2. Connection model for lateral deformations

The connection finite element model for lateral deformation is shown in Figure 3.7.

The strain in an element is
The first and second terms are considering axial and bending displacements. The third one is considering the deflection amplifications due to the effect of axial compression loads ($P$-$\Delta$ effects). If the previous state is $\{\sigma_0, \varepsilon_0\}$, the stress-strain relationships are (Foschi 2000)

$$F(\varepsilon) = \sigma_0 + E(\varepsilon - \varepsilon_0)$$

if $|F(\varepsilon)| \leq \sigma_y \rightarrow \sigma(\varepsilon) = F(\varepsilon)$

if $|F(\varepsilon)| > \sigma_y \rightarrow \sigma(\varepsilon) = \sigma_y \frac{F(\varepsilon)}{|F(\varepsilon)|}$

(3.56)

where

$\sigma_y$ = yield stress of steel connector.

Equation (3.56) will generate the hysteresis loop as shown in Figure 3.8.

Figure 3.7 Connection finite element model for lateral load
The connector (nail) can be defined as a group of two-node beam elements acting on a nonlinear foundation with linear axial and cubic bending deformation fields. The degree-of-freedom vector is

\[
a_c^T = (u_1, w_1, \frac{\partial w_1}{\partial x}, u_2, w_2, \frac{\partial w_2}{\partial x})
\]  

(3.57)

The displacements represented by shape functions and degrees of freedom are

\[
u = N_{c01} u_1 + N_{c02} u_2 = N_{oc}^T a_c
\]  

(3.58)

\[
w = M_{c01} w_1 + M_{c02} \left( \frac{\partial w_1}{\partial x} \right) + M_{c03} w_2 + M_{c04} \left( \frac{\partial w_2}{\partial x} \right) = M_{oc}^T a_c
\]  

(3.59)

where

\[N_{C0i}, M_{C0i} = \text{shape functions for connection elements.}\]

\[N_{oc}, M_{oc} = \text{shape function vectors for connection elements.}\]

The shape functions, after considering Jacobian transformations, are

\[N_{c01} = 0.5 - 0.5\xi\]
\[N_{c02} = 0.5 + 0.5\xi\]
\[M_{c01} = 0.5 - 0.75\xi + 0.25\xi^3\]
\[ M_{c02} = 0.25 \left( 1 - \xi - \xi^2 + \xi^3 \right) \Delta x \]
\[ M_{c03} = 0.5 + 0.75 \xi - 0.25 \xi^3 \]
\[ M_{c04} = 0.25 \left( -1 - \xi + \xi^2 + \xi^3 \right) \Delta x \]

3.3.3. Load-displacement relations

The deformations of the nail connector in \( x \) and \( y \) directions are determined from the relative movements between panel and frame members, as defined in Figure 3.9 (a) and (b), respectively, whereas the withdrawal in \( z \) direction is determined by calculating the separation of the panel from the frame member. These relations are

\[
\begin{align*}
\Delta x &= u_F - u_p + \frac{H}{2} \frac{\partial w_F}{\partial x} + \frac{t}{2} \frac{\partial w_p}{\partial x} \\
\Delta y &= v_F - v_p + \frac{H}{2} \frac{\partial w_F}{\partial y} + \frac{t}{2} \frac{\partial w_p}{\partial y} \\
\Delta z &= w_F - w_p
\end{align*}
\]

where \( H \) and \( t \) are the height of frame and the thickness of panel, respectively. Subscripts \( F \) and \( P \) represent frame and panel, respectively.

![Diagram of load-displacement relations](image)
Figure 3.9 Connection lateral deformations

The model requires only the basic material properties of nail connector and the embedding behaviour of surrounding wood medium (Foschi 2000). In analysis, the relative displacements $\Delta x$ and $\Delta y$ are inputted to the model. By using the principle of virtual work and applying nonlinear finite element procedures, the lateral reactions $F(\Delta x)$ and $F(\Delta y)$ from wood medium due to compression are calculated. The wood medium acts in accordance with the following rules:

When $\Delta \leq \Delta_{\text{max}}$:

$$F = (Q_0 + Q_1 \Delta) \left(1 - e^{-\kappa \Delta / Q_0}\right)$$  \hspace{1cm} (3.62)

When $\Delta > \Delta_{\text{max}}$:

$$F = F_{\text{max}} e^{Q_3 (\Delta - \Delta_{\text{max}})^{1/3}}$$  \hspace{1cm} (3.63)

$$Q_3 = \frac{\log Q_2}{(Q_2 - 1) \Delta_{\text{max}}^{1/3}}$$  \hspace{1cm} (3.64)

$$Q_2 = \frac{F}{F_{\text{max}}}$$  \hspace{1cm} (3.65)

$$Q_3 = \frac{\Delta}{\Delta_{\text{max}}}$$  \hspace{1cm} (3.66)

where
\[ F = \text{resultant reaction force.} \]

\[ Q_0 = \text{intercept of the asymptote.} \]

\[ Q_i = \text{asymptotic stiffness.} \]

\[ K = \text{initial stiffness.} \]

\[ \Delta = \text{displacement.} \]

\[ F_{\text{max}} = \text{maximum resultant reaction force.} \]

\[ \Delta_{\text{max}} = \text{displacement at } F_{\text{max}}. \]

Assume that the previous state is given by force \( F \) at point \( b \) associating with displacement \( \Delta \), in a loading or reloading. If the current displacement is greater than \( \Delta \), the force will be decided by the rules shown in (3.62) and (3.63). Otherwise, the force is zero until the displacement reaches the point \( a \), after which the load-displacement relationship follows the path \( a-b \) with a slope of the initial stiffness to meet the curve. In the course of unloading, the wood medium recovers slightly from the nail compression so that the load-displacement relationship follows the path \( b-a \). When the fastener does not touch the wood, the force is zero until it reaches the wood material on the opposite side.

The nail withdrawal force or panel-frame contact force \( F(\Delta z) \) can be calculated following a load-displacement relationship as shown in Figure 3.10(b). The same rules in (3.62) and (3.63) apply. When panel separates from frame (nail withdrawal), if \( \Delta z \) in the current step is greater than that in the previous step, the nail withdrawal load-displacement relationship follows the curve \( o-c-b \). Otherwise, it follows the path \( o-a-b \), in which the stiffness \( K_1 \) is assumed to be a small proportion of the initial stiffness \( K \). When the panel touches the frame, the contact force is approximated as a linear function of the initial stiffness \( K \).
3.4. The requirements of conforming elements

The use of conforming elements in the finite element analysis is necessary to achieve a monotonic convergence, which means that the strain energy calculated by the finite element solution converges to the exact strain energy of the mathematical model as the finite element mesh is refined (Bathe 1996). When an element satisfies both completeness and compatibility conditions, it is called a conforming element. The completeness requires that the shape functions of the element must be able to represent the rigid body displacements and the constant strain states. The compatibility requires that the displacements within the elements and across the element boundaries are continuous, meaning no gap occurs between elements when they are loaded. For elements with displacements, rotations, and twist degrees of freedom, the continuity should be preserved to the second displacement derivatives. By using the shape functions as described in the previous sections, the three types of elements used in the model satisfy the requirements of completeness and compatibility and, therefore, are assured to be conforming elements.
CHAPTER 4

FORMULATION OF STATIC NONLINEAR FINITE ELEMENT EQUATIONS

The finite element method was originally developed to analyze structural mechanics problems and was then broadly applied to continuum problems and many other areas. The method, as a numerical procedure, is very effective for problems unable to be solved satisfactorily by classical theoretical methods. Instead of solving the differential equations of equilibrium, the finite element method provides an approximate solution to variational problems by minimizing a functional. In structural mechanics problems, a functional is the expression for strain energy. In the current study, the principle of virtual work is adopted as the basis of the nonlinear finite element formulation.

4.1. Principle of virtual work

In the case of a particle subjected to a system of forces, the principle of virtual work states that if the particle is in equilibrium, the total work of all forces acting on the particle in any virtual displacement is zero (Timoshenko 1970). If \( \delta u \), \( \delta v \) and \( \delta w \) are components of a virtual displacement in the \( x \), \( y \), and \( z \) directions, and \( \Sigma R_x \), \( \Sigma R_y \) and \( \Sigma R_z \) are the sums of projections of forces acting on the particle in the same directions, the principle of virtual work gives

\[
\delta W_x = \delta u \sum R_x = 0 \quad \delta W_y = \delta v \sum R_y = 0 \quad \delta W_z = \delta w \sum R_z = 0
\]

(4.1)
The equations are satisfied for any virtual displacement if

\[ \sum R_x = 0 \quad \sum R_y = 0 \quad \sum R_z = 0 \]  

(4.2)

Consider a general three-dimensional body as shown in Figure 4.1. The body is subject to concentrated forces \( r_C \), surface tractions \( r_s \), and body forces \( r_B \). These forces represent all externally applied forces and reactions in three-dimensional space. The principle of virtual work, when applied to the body, states that equilibrium of the body requires that for any compatible small virtual displacements imposed onto the body, the total internal work is equal to the total external work.

The principle can be expressed as

\[ \delta W = \delta W_I + \delta W_E = 0 \]  

(4.3)

where the internal work is

\[ \delta W_I = \int \sigma \varepsilon \, dV \]  

(4.4)

and the external work is
\[
\delta W_E = -(\mathbf{R}_B + \mathbf{R}_S + \mathbf{R}_C) \delta \mathbf{a} = - \int_V \mathbf{r}_B \delta u dV - \int_A \mathbf{r}_S \delta u dA - \sum \mathbf{r}_C \delta \mathbf{a}
\] (4.5)

In these equations, \( \delta u \) is an arbitrary virtual displacement field, \( \delta \varepsilon \) is the corresponding virtual strain vector, and \( \delta \mathbf{a} \) is the nodal degree of freedom vector. The vector \( \delta u \) satisfies the boundary conditions of the problem and can be expressed as a linear combination of shape functions and nodal degrees of freedom. \( \mathbf{R}_C \) is a vector of the externally applied concentrated forces \( \mathbf{r}_C \) to the nodes of the element assemblage. The \( i \)th component in \( \mathbf{R}_C \) corresponds to the \( i \)th degree of freedom in \( \mathbf{a} \). \( \mathbf{R}_B \) and \( \mathbf{R}_S \) represent the vectors of body forces \( \mathbf{r}_B \) and externally applied surface forces (usually distributed loads) \( \mathbf{r}_S \).

It is noted that the body forces \( \mathbf{r}_B \) due to gravity in Equation (4.5) may be considered as less important than the surface forces \( \mathbf{r}_S \) in statically loaded structures. Therefore, they are excluded in the static finite element analysis. In dynamically loaded structures, inertia forces are generated from the body forces and need to be integrated into the dynamic finite element analysis (Chapter 6).

4.2. Internal virtual work in elements

4.2.1. Internal virtual work in plate element

The constitutive relationships for a plate element in Equation (3.13) can be expressed as follows

\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix} =
\begin{bmatrix}
\frac{E_s}{1-v_{xy}v_{yx}} & \frac{v_{xy}E_x}{1-v_{xy}v_{yx}} & 0 \\
\frac{v_{xy}E_y}{1-v_{xy}v_{yx}} & \frac{E_y}{1-v_{xy}v_{yx}} & 0 \\
0 & 0 & G_{xy}
\end{bmatrix}
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix}
\] (4.6)
Substituting Equation (4.6) into Equation (4.4) yields the internal virtual work of the plate element

$$
(\delta W_i)_p = \int \left[ \frac{E_x}{1 - \nu \nu} \left( \varepsilon_x \delta \varepsilon_x + \nu \varepsilon_y \delta \varepsilon_y \right) + \varepsilon_y \delta \varepsilon_y \right] dV
$$

Using the strain-displacement relationships developed in Chapter 3, Equation (4.7) can be expressed in terms of shape functions and natural coordinates.

$$
(\delta W_i)_p = \frac{\Delta x \Delta y}{2} \left[ \delta a^T \left[ \int \left[ D_i N_i L_i \epsilon_i + K_i M_i M_i^T \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \int \left[ \int D_j \left( \frac{1}{2} M_{1j} M_{1j}^T a + L_{1j} \right) M_{1j} M_{1j}^T + \frac{1}{2} M_{1j} M_{1j}^T N_{1j} N_{1j} a \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \int \left[ \int D_j \left( \frac{1}{2} M_{1j} M_{1j}^T a + L_{1j} \right) M_{1j} M_{1j}^T + \frac{1}{2} M_{1j} M_{1j}^T N_{1j} N_{1j} a \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \int \left[ \int D_j \left( \frac{1}{2} M_{1j} M_{1j}^T a + L_{1j} \right) M_{1j} M_{1j}^T + \frac{1}{2} M_{1j} M_{1j}^T N_{1j} N_{1j} a \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

$$
+ \delta a^T \left[ \int \left[ D_j \left( L_{1j} N_{1j} + N_{1j} L_{1j} \right) + K_i \left( M_{1j} M_{1j}^T + M_{1j} M_{1j}^T \right) \right] d\xi d\eta \right] a
$$

Chapter 4 Formulation of Static Nonlinear Finite Element Equations
\[
+ \delta a^T \left[ \int \int_{\xi \eta} \left[ a^T \left( N_{1y} + L_{1x} \right) \left( N_{1y} + L_{1x} \right)^T + 4K_c M_{2y} M_{2y}^T \right] d\xi d\eta \right] a
\]

\[+ \delta a^T \left[ \int \int_{\xi \eta} \left[ a^T \left( N_{1y} + L_{1x} + M_{1x} M_{1y}^T a \right) \left( M_{1x} M_{1y}^T + M_{1y} M_{1x}^T \right) \right. \right.
\]

\[
+ \left. M_{1x} M_{1y}^T \left( N_{1y} + L_{1x} \right) \right] d\xi d\eta \right] a \}
\]

where

\[
D_x = \frac{E_x t}{1 - v_{xy} v_{yx}} \quad (4.9)
\]

\[
D_y = \frac{E_y t}{1 - v_{xy} v_{yx}} \quad (4.10)
\]

\[
D_v = \frac{v_{xy} E_x t}{1 - v_{xy} v_{yx}} = \frac{v_{yx} E_y t}{1 - v_{xy} v_{yx}} \quad (4.11)
\]

\[
D_G = G_{xy} t \quad (4.12)
\]

\[
K_x = \frac{E_x t^3}{12(1 - v_{xy} v_{yx})} \quad (4.13)
\]

\[
K_y = \frac{E_y t^3}{12(1 - v_{xy} v_{yx})} \quad (4.14)
\]

\[
K_v = \frac{v_{xy} E_x t^3}{12(1 - v_{xy} v_{yx})} = \frac{v_{yx} E_y t^3}{12(1 - v_{xy} v_{yx})} \quad (4.15)
\]

\[
K_G = G_{xy} \frac{t^3}{12} \quad (4.16)
\]

and

\[
E_x, E_y \quad = \quad \text{Young's modulus in } x \text{ and } y \text{ directions, respectively.}
\]

\[
G_{xy} \quad = \quad \text{shear modulus.}
\]

\[
v_{xy}, v_{yx} \quad = \quad \text{Poisson's ratios.}
\]

\[
t \quad = \quad \text{thickness of panel elements.}
\]

All of the definitions of shape functions can be found in Equations (3.36)-(3.44) in Chapter 3.

The above internal virtual work expression includes the work done by the elongation and bending of the middle plane of the plate and the work done by the large deflection due to...
$P-\Delta$ effects. The integration contains constant linear elastic stiffness terms that need to be calculated only once in the whole process and higher order nonlinear stiffness terms that change with respect to the displacements. The contribution of $P-\Delta$ effects is slight until the deflection of the panel member becomes significant. Usually it is convenient to present these two parts of virtual work separately because they are treated differently, as shown in the following equation

$$
(\delta W)_p = \frac{\Delta x \Delta y}{2} \left\{ \delta a^T \int \left[ (\Theta_{LP} + \Theta_{NP}) d\xi d\eta \right] a \right\}
$$

(4.17)

where $\Theta_{LP}$ represents the linear terms and $\Theta_{NP}$ represents the higher order terms required for nonlinear analysis. They are

$$
\Theta_{LP} = D_x N_{1x} N_{1x}^T + D_y L_{1y} L_{1y}^T + D_v (L_{1y} N_{1x} + N_{1x} L_{1y})^T + D_G (N_{1x} + L_{1y}) (N_{1x} + L_{1y})^T
$$

$$
+ K_x M_{2x} M_{2x}^T + K_y M_{2y} M_{2y}^T + K_v (M_{2y} M_{2x}^T + M_{2x} M_{2y}) + 4K_G M_{2x} M_{2y}^T
$$

(4.18)

and

$$
\Theta_{NP} = D_x \left[ a^T \left( \frac{1}{2} M_{1x} M_{1x}^T a + N_{1x} \right) M_{1x} M_{1x}^T + \frac{1}{2} M_{1x} M_{1x}^T N_{1x} a \right]
$$

$$
+ D_y \left[ a^T \left( \frac{1}{2} M_{1y} M_{1y}^T a + L_{1y} \right) M_{1y} M_{1y}^T + \frac{1}{2} M_{1y} M_{1y}^T L_{1y} a \right]
$$

$$
+ D_v \left[ \frac{1}{2} a^T (M_{1x} M_{1x}^T M_{1x} + M_{1y} M_{1y}^T M_{1y}) a
$$

$$
+ \frac{1}{2} \left( M_{1x} N_{1x} L_{1y} + M_{1y} M_{1y}^T N_{1x} a + a^T (N_{1x} M_{1y} M_{1y}^T + L_{1y} M_{1y} M_{1y}) \right]
$$

$$
+ D_G \left[ a^T (N_{1y} + L_{1y} + M_{1x} M_{1x}^T a)(M_{1y} M_{1y}^T + M_{1y} M_{1y}^T) + M_{1x} M_{1y}^T (N_{1y} + L_{1x}) \right]
$$

(4.19)

The linear terms in (4.18) can also be displayed in a conventional matrix form

$$
\Theta_{LP} = B_p^T D_p B_p
$$

(4.20)

where
4.2.2. Internal virtual work in beam element

In beam elements, based on the constitutive relationships described in Equation (3.52), the internal virtual work is

\[
(\delta W_r)_F = \int (E\varepsilon \delta \varepsilon + G\gamma \delta \gamma) dV
\]

(4.23)

For a horizontal frame member, in terms of shape functions and natural coordinates

\[
(\delta W_r)_FH = \int (E\varepsilon \delta \varepsilon_x + G\gamma \delta \gamma_{xy}) dV
\]

\[
= \frac{A\chi}{2} \left\{ \delta a^T \left[ \begin{array}{c} \left[ EAN_{1x}N_{1x}^T + EI_xL_{1x}L_{1x}^T + EI_yLM_{2x}M_{2x}^T + G/M_{2xy}M_{2xy}^T \right] \xi \end{array} \right] \right\} a
\]

\[+ \delta a^T \int \left[ EJ\left( a^TN_{1x}(M_{1x}M_{1x}^T + L_{1x}L_{1x}^T) + \frac{1}{2}(M_{1x}M_{1x}^T + L_{1x}L_{1x}^T)N_{1x}a \right. \right. \]

\[+ \frac{1}{2} a^T(M_{1x}M_{1x}^T + L_{1x}L_{1x}^T)(M_{1x}M_{1x}^T + L_{1x}L_{1x}^T)a \left. \right] d\xi \left\} a \right\}
\]

(4.24)

and for a vertical frame member, in terms of shape functions and natural coordinates
\[(\delta W_i)_{PV} = \int \left( (E\varepsilon_y \delta \varepsilon_y + G\gamma_{yx} \delta \gamma_{yx}) \right) dV\]

\[= \frac{\Delta y}{2} \left\{ \delta a^T \left[ \int_{\eta} \left[ EA L_{1y} M_{1y}^T + EI_x N_{1x} N_{1x}^T + EI_x M_{2y} M_{1y}^T + GJ/M_{2xy} M_{2xy}^T \right] \delta \eta \right] a \right\}

+ \delta a^T \left[ \int_{\eta} \left[ a^T L_{1y} \left( M_{1y} M_{1y}^T + N_{1y} N_{1y}^T \right) + \frac{1}{2} \left( M_{1y} M_{1y}^T + N_{1y} N_{1y}^T \right) L_{1y} \right] a \right]

\[+ \frac{1}{2} a^T \left( M_{1y} M_{1y}^T + N_{1y} N_{1y}^T \right) \left[ M_{1y} M_{1y}^T + N_{1y} N_{1y}^T \right] \delta \eta \] (4.25)

Similar to the virtual work of plate element, the virtual work of beam element also contains contributions from the linear and nonlinear behaviour of the frame members. Like before, they are separated into two parts

\[(\delta W_i)_{FH} = \frac{\Delta x}{2} \left\{ \delta a^T \left[ \int_{\xi} \left[ \Theta_{LFH} + \Theta_{NFR} \right] \delta \xi \right] a \right\} \quad (4.26)\]

\[(\delta W_i)_{PV} = \frac{\Delta y}{2} \left\{ \delta a^T \left[ \int_{\eta} \left[ \Theta_{LFV} + \Theta_{NFR} \right] \delta \eta \right] a \right\} \quad (4.27)\]

The linear terms are

\[\Theta_{LFH} = EA N_{1x} N_{1x}^T + EI_x L_{1y} L_{1y}^T + EI_x M_{2x} M_{2x}^T + GJM_{2xy} M_{2xy}^T \quad (4.28)\]

\[\Theta_{LFV} = EA L_{1y} L_{1y}^T + EI_x N_{1x} N_{1x}^T + EI_x M_{2y} M_{2y}^T + GJM_{2xy} M_{2xy}^T \quad (4.29)\]

and the higher order terms required in nonlinear analysis are

\[\Theta_{NFR} = EA \left[ a^T N_{1x} \left( M_{1x} M_{1x}^T + L_{1x} L_{1x}^T \right) + \frac{1}{2} \left( M_{1x} M_{1x}^T + L_{1x} L_{1x}^T \right) N_{1x} a \right. \]

\[+ \left. \frac{1}{2} a^T \left( M_{1x} M_{1x}^T + L_{1x} L_{1x}^T \right) \left( M_{1x} M_{1x}^T + L_{1x} L_{1x}^T \right) a \right]\]

\[\Theta_{NFR} = EA \left[ a^T L_{1y} \left( M_{1y} M_{1y}^T + N_{1y} N_{1y}^T \right) + \frac{1}{2} \left( M_{1y} M_{1y}^T + N_{1y} N_{1y}^T \right) L_{1y} a \right. \]

\[+ \left. \frac{1}{2} a^T \left( M_{1y} M_{1y}^T + N_{1y} N_{1y}^T \right) \left( M_{1y} M_{1y}^T + N_{1y} N_{1y}^T \right) a \right] \quad (4.30)\]
where

\[ E, \ G = \text{Young's modulus and shear modulus, respectively.} \]
\[ A = \text{cross-section area.} \]
\[ I = \text{moment of inertia.} \]

\[
I_x = \int_A x^2 \, dA
\]
\[
I_y = \int_A y^2 \, dA
\]
\[
I_z = \int_A z^2 \, dA
\]

\[ J = \int_A \rho^2 \, dA \quad (4.32) \]

\[ J = \text{polar moment of inertia.} \]

\[ \rho = \text{radial distance.} \]

The linear terms in Equations (4.28) and (4.29) can also be displayed in matrix form as

\[ \Theta_{LPH} = \mathbf{B}_{FH}^T \mathbf{D}_{FH} \mathbf{B}_{FH} \quad (4.34) \]

and

\[ \Theta_{LPV} = \mathbf{B}_{PV}^T \mathbf{D}_{PV} \mathbf{B}_{PV} \quad (4.35) \]

where

\[
\mathbf{B}_{FH} = \begin{bmatrix}
N_{1x} & 0 \\
L_{2x} & M_{2x} \\
0 & M_{2xy}
\end{bmatrix}
\]

\[
\mathbf{B}_{PV} = \begin{bmatrix}
L_{1y} & 0 \\
N_{2y} & M_{2y} \\
0 & M_{2xy}
\end{bmatrix}
\]

Chapter 4 Formulation of Static Nonlinear Finite Element Equations
4.2.3. Internal virtual work in connection element

From Equation (3.61), the virtual deformations of a connection element represented by the shape functions are

\[
\delta \Delta x = \left( N_0^T - N_0^T + \frac{H}{2} M_{ix}^T + \frac{f}{2} M_{iz}^T \right) \delta a = Q_x^T \delta a
\]

\[
\delta \Delta y = \left( L_{0r}^T - L_0^T + \frac{H}{2} M_{iy}^T + \frac{f}{2} M_{iz}^T \right) \delta a = Q_y^T \delta a
\]

\[
\delta \Delta z = \left( M_{0r}^T - M_0^T \right) \delta a = Q_z^T \delta a
\]

Where the \( Q \)'s are the shape function combinations.

External virtual work done by a connection element in three directions is

\[
(\delta W_E)_E = \sum \left[ - F(\Delta x) \delta \Delta x - F(\Delta y) \delta \Delta y - F(\Delta z) \delta \Delta z \right]
\]

Since

\[ (\delta W_I)_I + (\delta W_E)_E = 0 \]

Therefore, the equivalent internal virtual work in a connection element is

\[
(\delta W_I)_I = \sum \left[ F(\Delta x) \delta \Delta x + F(\Delta y) \delta \Delta y + F(\Delta z) \delta \Delta z \right] = \sum \left[ F(\Delta x) Q_x^T \delta a + F(\Delta y) Q_y^T \delta a + F(\Delta z) Q_z^T \delta a \right]
\]

\[
\mathbf{D}_{FH} = \begin{bmatrix} EA & 0 \\ EI_y & ET_z \\ 0 & GJ \end{bmatrix} \quad (4.38)
\]

\[
\mathbf{D}_{FY} = \begin{bmatrix} EA & 0 \\ EI_x & ET_z \\ 0 & GJ \end{bmatrix} \quad (4.39)
\]
4.3. Formulation of external force vector

Under static loading conditions, the structure to be analyzed is subject to in-plane concentrated loads, out-of-plane uniformly distributed loads, or a combination of both. The concentrated loads in the model are required to be assigned to the nodes of the frame elements, whereas the uniformly distributed loads need to be assigned on the surface of the exterior panels (normal to the surface). These loads represent the weights of the structural components (vertical constant load), the weights of snow and rain (vertical live loads), and the lateral live loads. The lateral loads are applied to structure for the purpose of simulating the action of wind and earthquake. The loading rate is very low to avoid any possible influence due to severe variation of material strain rate. Although this type of loading is usually referred to “quasi-static” loading and is associated with cyclic tests, the unidirectional monotonic loading is often used in the studies of lateral resistance of structures.

The concentrated load vector can be formed by simply accounting all of the external concentrated loads, which is

\[
\mathbf{R}_{c} = \sum \mathbf{r}_{c} \tag{4.46}
\]

Referring to the illustration in Figure 4.2 a consistent load vector for the uniformly distributed loads can be derived. Assume a uniformly distributed load \(q(x, y)\) to be acting on a patch of a panel element. From the expression of the external virtual work by surface forces in Equation (4.5), the virtual work by distributed loads is

\[
(\delta W_{e})_{\delta} = - \int_{A} \mathbf{r}_{s} dA \delta \mathbf{u} = - \int_{A} q(x, y) dxdy \delta \mathbf{w} = - \int_{\xi}^{\eta} q(x, y) \frac{\Delta x}{2} d\xi' \frac{\Delta y}{2} d\eta' \mathbf{M}_{e}(\xi', \eta') \delta \mathbf{a} \tag{4.47}
\]

Based on the geometry in Figure 4.2, the following relations can be found
\[ x = x_i + l_x + \frac{\Delta x'}{2} (1 + \xi) \] (4.48)
\[ y = y_j + l_y + \frac{\Delta y'}{2} (1 + \eta) \] (4.49)
\[ \xi' = \frac{2}{\Delta x} (x - x_c) = \frac{2}{\Delta x} \left[ x - \left( \frac{\Delta x}{2} + x_i \right) \right] = \frac{2}{\Delta x} \left[ l_x + \frac{\Delta x'}{2} (1 + \xi) \right] - 1 \] (4.50)
\[ \eta' = \frac{2}{\Delta y} (y - y_c) = \frac{2}{\Delta y} \left[ y - \left( \frac{\Delta y}{2} + y_j \right) \right] = \frac{2}{\Delta y} \left[ l_y + \frac{\Delta y'}{2} (1 + \eta) \right] - 1 \] (4.51)
\[ d\xi' = \frac{\Delta x'}{\Delta x} d\xi \] (4.52)
\[ d\eta' = \frac{\Delta y'}{\Delta y} d\eta \] (4.53)
\[ l_x = x'_i - x_i \] (4.54)
\[ l_y = y'_j - y_j \] (4.55)

where

\[ \xi, \eta, \xi', \eta' \quad = \quad \text{natural coordinates for loading patch and element, respectively.} \]
\[ x, y \quad = \quad \text{Cartesian coordinates.} \]
\[ x_c, y_c \quad = \quad \text{central coordinates of plate element.} \]

Figure 4.2 Uniformly distributed load acting on an element
Substituting the relations in (4.52) and (4.53) into (4.47), and applying relations (4.50) and (4.51) to the shape functions $M_0$, result in the following,

$$(\delta W_E)_s = -\int \int q(x, y) \frac{\Delta x'}{2} d\xi \frac{\Delta y'}{2} d\eta M_0(x, y) \delta a$$

(4.56)

The consistent load vector for uniformly distributed surface forces is

$$R_s = \int \int q(x, y) \frac{\Delta x'}{2} d\xi \frac{\Delta y'}{2} d\eta M_0(x, y)$$

(4.57)

4.4. Formulation and solution of system equations

The principle of virtual work can be now applied to the whole system to formulate the equations at element level required for the solution. The element tangent stiffness is calculated first and then assembled into a global stiffness matrix using standard finite element procedures. During the assembling, a three-dimensional transformation is required to ensure that all the elements are in a common global coordinate system (Chapter 5). The global stiffness matrix is then solved by a numerical integration method, Gauss quadrature, which is carried out at each Gauss point for the whole system. The global solution vector $a_{i+1}$ is computed by a selected iteration scheme with appropriate boundary conditions imposed. From the previously introduced expressions for panel element, frame elements, connection elements, and the externally applied forces, the total virtual work in an element under static state is

$$(\delta W_E)_p + (\delta W_E)_{FH} + (\delta W_E)_{PV} + (\delta W_E)_C - (\delta W_E)_S - R_c \delta a = 0$$

(4.58)

Differentiating Equation (4.58) with respect to $\delta a$, an element out-of-balance load vector $\psi_c$ can be obtained.
\[
\psi_e = \frac{\Delta x}{2} \frac{\Delta y}{2} \left[ \int \left( \Theta_{LP} + \Theta_{NP} \right) d\xi d\eta \right] a + \frac{\Delta x}{2} \left[ \int \left( \Theta_{LFH} + \Theta_{NFH} \right) H \xi \right] a + \frac{\Delta y}{2} \left[ \int \left( \Theta_{LHV} + \Theta_{NHF} \right) \eta \right] a + \int \sum \left[ F(\Delta x)Q_x^T + F(\Delta y)Q_y^T + F(\Delta z)Q_z^T \right] - R_s - R_c = 0
\]

or

\[
\psi_e = \tilde{f} - R_s - R_c = 0
\]

where

- \( \tilde{f} \) is the force containing contributions from the linear and nonlinear behaviour of panel and frame members, and from the nonlinear behaviour of nail connections. It is

\[
\tilde{f} = \frac{\Delta x}{2} \frac{\Delta y}{2} \left[ \int \left( \Theta_{LP} + \Theta_{NP} \right) d\xi d\eta \right] a + \frac{\Delta x}{2} \left[ \int \left( \Theta_{LFH} + \Theta_{NFH} \right) H \xi \right] a + \frac{\Delta y}{2} \left[ \int \left( \Theta_{LHV} + \Theta_{NHF} \right) \eta \right] a + \int \sum \left[ F(\Delta x)Q_x^T + F(\Delta y)Q_y^T + F(\Delta z)Q_z^T \right]
\]

The vector \( \psi_e \) is obtained first for each element and then assembled into the corresponding global out-of-balance load vector \( \Psi \). The vector \( \Psi \) can be interpreted as the difference between the resisting forces of the structure and the externally applied loads. When this difference is zero, the whole structure is in equilibrium and the unknown nodal point variables will converge to a set of new values corresponding to the preset displacement or load conditions. Since the resisting forces are not single valued but depend nonlinearly on the nodal point displacements, an iterative procedure is needed to find the solutions of the equations \( \Psi \) to satisfy the finite element equilibrium requirements.
One of most frequently used iterative schemes for the nonlinear finite element equations, the Newton-Raphson method, is implemented in the algorithm. This solution method involves repeat calculations of increments in the nodal point displacement, which defines a new displacement vector, until the out-of-balance load vector $\Psi$ to be zero. The currently used Newton-Raphson method usually leads the solution to a rapid convergence within a few iterations when the absolute values of the tangent stiffness are large. However, the convergence by this method is not guaranteed when the tangent stiffness value is close to zero (the flattening parts of the load-displacement curve). In this situation, the solution may converge very slowly or, sometimes, may divergence. To ensure a convergence, some self-adaptive procedures are implemented into the program, as described in Section 4.7.3. An alternative presentation of Newton-Raphson is the modified Newton-Raphson, which differs from the Newton-Raphson only in that the tangent stiffness matrix is not updated during iterations within one step. The benefit of using the modified Newton-Raphson is that the expensive repetitions of forming and reducing the tangent stiffness matrix can be avoided. However, the study found (Cook 1989, Ch. 17) that the modified Newton-Raphson method did not perform well when the load-displacement curve displays a hardening or pinching behaviour. In this situation, the iterations are more likely to converge slowly or fail to converge.

The general equations used in the Newton-Raphson iteration (illustrated in Figure 4.3), for $i=1, 2, 3, \ldots$, are:

\[
K_i \Delta a_i = -\Psi_i \tag{4.62}
\]

\[
a_{i+1} = a_i + \Delta a_i \tag{4.63}
\]

In these equations, the loads are assumed to be independent of the deformations. The
initial conditions are $a_i$, $K_0$, and $P_a$.

![Diagram](image)

Figure 4.3 Newton-Raphson iteration starting from point $a$ and converging at point $b$

If the displacement vector $a_i$ is evaluated in the iterative solution, a Taylor series expansion of the vector $\Psi$, truncated after the first derivative, is

$$\Psi_{i+1} = \Psi_i + \frac{\partial \Psi}{\partial a} \Delta a_i = \Psi_i + \Delta \Psi (a_{i+1} - a_i) \quad (4.64)$$

where the subscript $i$ represents the $i$th iteration.

The component

$$\frac{\partial \Psi}{\partial a} = \Delta \Psi \quad (4.65)$$

is the global tangent stiffness matrix, which is the assemblage of element tangent stiffness matrices at the $i$th iteration.

Assuming that the out-of-balance load vector $\Psi_{i+1}$ is zero in the $(i+1)$th iteration, an updated displacement vector $a_{i+1}$ can be obtained from Equation (4.64) based on the vectors $\Psi_i$ and $a_i$ from the current iteration

$$a_{i+1} = a_i + (\Delta \Psi)^{-1}(-\Psi_i) \quad (4.66)$$
An element tangent stiffness matrix is obtained by differentiating the equation (4.59)

\[
\frac{\partial \psi}{\partial a} = \frac{\Delta x}{2} \frac{\Delta y}{2} \left[ \int_{\xi}^{\eta} \left( \Theta_{LP} + \Theta_{NP} \right) d\xi d\eta \right]
\]

\[
+ \frac{\Delta x}{2} \left[ \int_{\xi}^{\eta} \left( \Theta_{LFH} + \Theta_{NFH} \right) d\xi \right] + \frac{\Delta y}{2} \left[ \int_{\eta}^{\xi} \left( \Theta_{LFV} + \Theta_{NFV} \right) d\eta \right]
\]

\[
+ \sum \left[ \frac{\partial F(\Delta x)}{\partial \Delta x} Q_x^T + \frac{\partial F(\Delta y)}{\partial \Delta y} Q_y^T + \frac{\partial F(\Delta z)}{\partial \Delta z} Q_z^T \right]
\]

\[
\left[ \Theta_{LP} \right] \left[ d\xi d\eta \right] + \left[ \Theta_{LFH} \right] \left[ d\xi \right] + \left[ \Theta_{LFV} \right] \left[ d\eta \right]
\]

\[
\left[ \Theta_{NP} \right] \left[ d\xi d\eta \right] + \left[ \Theta_{NFH} \right] \left[ d\xi \right] + \left[ \Theta_{NFV} \right] \left[ d\eta \right]
\]

\[
\left[ \Theta_{NP} \right] \left[ d\xi d\eta \right] + \left[ \Theta_{NFH} \right] \left[ d\xi \right] + \left[ \Theta_{NFV} \right] \left[ d\eta \right]
\]

\[
= (k)_L + (k)_N = k_T
\]

where

- \( k_T \) = element tangent stiffness matrix.
- \( (k)_L \) = linear part of element tangent stiffness matrix, which is constant and calculated only once.
- \( (k)_N \) = nonlinear part of element tangent stiffness matrix, which is updated for each step.

Accordingly, the global stiffness matrix can be expressed as

\[
\frac{\partial \psi}{\partial a} = K_T = \sum k_T = \sum (k)_L + \sum (k)_N = (K)_L + (K)_N
\]
The integrals in the tangent stiffness matrix presented in equation (4.67) have to be evaluated numerically and updated at each step. The Gauss quadrature has been proven the most useful quadrature rule in numerical integration and is implemented in the LightFrame3D program.

One- and two-dimensional integrals can be expressed approximately as polynomials, as shown respectively in the following equations.

\[
\int_{-1}^{1} f(\xi)d\xi = \sum_{i=1}^{n} W_i f(\xi_i) \quad (4.69)
\]

\[
\int_{-1}^{1} \int_{-1}^{1} f(\xi, \eta)d\xi d\eta = \sum_{i=1}^{n} \sum_{j=1}^{n} W_i W_j f(\xi_i, \eta_j) \quad (4.70)
\]

where

- **n** = order of numerical integration or number of Gauss points.
- **W_i**, **W_j** = weighting factors.
- **f(\xi_i)** and **f(\xi_i, \eta_j)** are **f(\xi)** and **f(\eta, \eta)** evaluated at the Gauss points, respectively.

The determination of the number of Gauss points is based on the orders of **f(\xi)** and **f(\eta)**. Generally, the **n** Gauss points are needed to evaluate those matrices exactly up to the order of (2n-1). In the nonlinear finite element analysis, however, the determination of the number of Gauss points is not as simple, due to the difficulty in examining the order of the nonlinear portion of the stiffness matrix. Although the number of Gauss points can be decided by changing them in the program and comparing the errors in results, it is believed that a satisfactory accuracy is achieved by selecting the number of Gauss points just according to the order of the linear portion of the stiffness matrix, since the influence from the much higher orders on the results is negligible. The final choice of the number of Gauss
points should also consider the balance between the accuracy and the cost of the analysis. Using more Gauss points to achieve slightly improved accuracy of the results may not be practical in the analysis, especially in three-dimensional analysis, because the cost can increase significantly.

From the equations (4.18), (4.28) or (4.29) by referring to the Pascal triangle, the highest order of the linear portion of the stiffness matrices is determined to be six, as shown in equation (4.71). The required number of Gauss points is four, which can evaluate the stiffness matrix exactly up to the order of seven.

\[
f = f(\xi^6, \xi^6 \eta^2, \xi^4 \eta^4, \xi^2 \eta^6, \eta^8)
\]  

(4.71)

The sampling points and weighting factors for four Gauss point integration are listed in double-precision as follows:

<table>
<thead>
<tr>
<th>(\xi_i) or (\eta_j)</th>
<th>(W_i) or (W_j)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\pm 0.86113) 63115 94053</td>
<td>0.34785 48451 37454</td>
</tr>
<tr>
<td>(\pm 0.33998) 10435 84856</td>
<td>0.65214 51548 62546</td>
</tr>
</tbody>
</table>

4.5. Element stress calculations

After element stiffness matrices are assembled into the global stiffness matrix and the nodal point displacement vector is solved using the Newton-Raphson iterative procedure, the element stresses are calculated in the post-processor, the final phase of the analysis.

4.5.1. Panel stresses and central deflection

Recall the expressions of panel normal and shear strains derived in Chapter 3

\[
\varepsilon_x = \varepsilon'_x + \varepsilon''_x = \frac{\partial u}{\partial x} + \frac{1}{2} \left( \frac{\partial w}{\partial x} \right)^2 - z \frac{\partial^2 w}{\partial x^2}
\]  

(4.72)
\[ \varepsilon_y = \varepsilon'_y + \varepsilon''_y = \frac{\partial y}{\partial y} + \frac{1}{2} \left( \frac{\partial w}{\partial y} \right)^2 - 2z \frac{\partial^2 w}{\partial y^2} \]  
\[ (4.73) \]

\[ \gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} + \frac{\partial w}{\partial y} \frac{\partial w}{\partial y} - 2z \frac{\partial^2 w}{\partial x \partial y} \]  
\[ (4.74) \]

Using the first and second derivatives of shape functions (Equations 3.36 – 3.44 in Chapter 3) to represent these strains gives

\[ \varepsilon_x = N_{1x}^T a + \frac{1}{2} a^T M_{1x} M_{1x}^T a - zM_{2x}^T a \]  
\[ (4.75) \]

\[ \varepsilon_y = L_{1y}^T a + \frac{1}{2} a^T M_{1y} M_{1y}^T a - zM_{2y}^T a \]  
\[ (4.76) \]

\[ \gamma_{xy} = N_{1y}^T a + L_{1y}^T a + a^T M_{1x} M_{1y}^T a - 2zM_{2xy}^T a \]  
\[ (4.77) \]

The normal and shear stresses in panel are obtained as

\[ \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = \mathbf{D} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} \]  
\[ (4.78) \]

where

\[ \mathbf{D} = \begin{bmatrix} E_x & \nu_{xy} E_x & 0 \\ \frac{1 - \nu_{xy}}{E_x} & \frac{1 - \nu_{xy}}{E_y} & 0 \\ \frac{\nu_{xy}}{E_y} & \frac{\nu_{xy}}{E_y} & 0 \\ 0 & 0 & G_{xy} \end{bmatrix} \]  
\[ (4.79) \]

The deflection at the center of a plate element is calculated by using the relationship when the natural coordinates \( \xi \) and \( \eta \) are taken to be zero

\[ w = M_0^T a \]  
\[ (4.80) \]

4.5.2. Frame bending stress

The normal strains in a horizontal and vertical beam element are, respectively
\[ \varepsilon_x = \frac{\partial u}{\partial x} + \frac{1}{2} \left( \frac{\partial w}{\partial x} \right)^2 + \frac{1}{2} \left( \frac{\partial v}{\partial x} \right)^2 - z \frac{\partial^2 w}{\partial x^2} - y \frac{\partial^2 v}{\partial x^2} \] (4.81)

\[ \varepsilon_y = \frac{\partial v}{\partial y} + \frac{1}{2} \left( \frac{\partial w}{\partial y} \right)^2 + \frac{1}{2} \left( \frac{\partial u}{\partial y} \right)^2 - z \frac{\partial^2 w}{\partial y^2} - x \frac{\partial^2 u}{\partial y^2} \] (4.82)

When expressed by the derivatives of shape functions, the strains are

\[ \varepsilon_x = N_{ix}^T a + \frac{1}{2} a^T M_{ix} M_{ix}^T a + \frac{1}{2} a^T L_{ix} L_{ix}^T a - z M_{ix}^T a - y L_{ix}^T a \] (4.83)

\[ \varepsilon_y = L_{iy}^T a + \frac{1}{2} a^T M_{iy} M_{iy}^T a + \frac{1}{2} a^T N_{iy} N_{iy}^T a - z M_{iy}^T a - x N_{iy}^T a \] (4.84)

Then the stresses in a horizontal and vertical beam element are, respectively

\[ \sigma_x = E \varepsilon_x \quad \sigma_y = E \varepsilon_y \] (4.85)

4.6. Displacement control method

Displacement and load controls are two options that can be selected as input history in the static finite element analysis by using the LightFrame3D_Static program. In displacement control, the incremental displacement is used as stepping parameter, and in load control, the incremental load as stepping parameter. In terms of the program coding, both methods do not show much difference; however, in practical applications, the selection of the control method should be based on the problems to be studied and the results sought because the two methods do give different outputs. The displacement control method allows the analysis to go beyond the maximum capacity of the system to provide a complete load-displacement path. Usually it is desirable that the analysis could predict the entire deformation history including the post-peak behaviour of the structure. It is also useful when the analytical results are compared with the testing backbone curves. Load control method can not satisfy this requirement because the stiffness matrix will be ill-conditioned at the
maximum load level. The displacement control method can be easily used to follow multiple loading-unloading-reloading paths as in a cyclic procedure, in which groups of cycles with prescribed displacement amplitudes are applied to the structure to study the stiffness and strength degradations and the pitching effects. The load control method, however, is difficult to be used for this purpose.

The only difference between the displacement control and load control is that an incremental displacement step is assigned to the designated degree of freedom in the displacement vector \( \mathbf{a} \) before solving the system equations, and the convergence is checked by using the criterion (4.88) to evaluate the resultant loads of subsequent iterations at that degree of freedom. In Chapter 9, detailed descriptions are given for the predictions made by the LightFrame3D program using displacement control mode.

4.7. Convergence considerations

4.7.1. Convergence checking and criteria

In the nonlinear finite element analysis, convergence rate is strongly problem-dependent. The convergence could be achieved very quickly, such as within two or three iterations, when the structure is mainly in its linear elastic range, while the problem requires more iteration to achieve convergence when the structure is approaching its maximum capacity, at which the tangent stiffness is close to zero. Sometimes, divergence may occur due to the physical instability of the structure. In such cases, the structure can be typically defined as failed. To ensure convergence at each step, convergence-enhancement features were implemented into LightFrame3D programs to activate the self-adaptive procedures if the solution does not converge after required number of iterations (see Section 4.7.3). In all the iterative processes of the current nonlinear finite element analysis, the solutions are only
approximately achieved and some tolerance limits have to be set to terminate the iteration. The selection of tolerances should be realistic. If the convergence tolerances are too liberal, the solution is inaccurate and error can accumulate to cause difficulties for the later steps to converge. If the tolerances are too tight, either the highly accurate solution cannot be obtained or much computational effort is spent to obtain unnecessary accuracy.

The following convergence criteria are used in the nonlinear finite element programs.

Criterion 1.

\[
\frac{\|\Delta a_i\|}{\|a_i\|} \leq \varepsilon_d
\]  

(4.86)

This criterion requires that the norm of the displacements at the end of iteration be within a certain tolerance \(\varepsilon_d\) of the true displacement solution. This criterion is used when the problem is under the load control mode in a static analysis.

Criterion 2.

\[
\|\Psi_1\| \leq \varepsilon_F \|\Psi_1\|
\]  

(4.87)

This criterion requires that the norm of the out-of-balance load vector at the end of an iteration be within a certain tolerance \(\varepsilon_F\) of the initial load increment. In the application of this criterion, a few problems were noticed. The variation in the magnitude of the initial load increment \(\Psi_1\) may cause the change in convergence rate for different steps. In the case of small initial load increment, the convergence in the out-of-balance load may not imply the convergence in the displacements. In addition, the out-of-balance load vector contains quantities in different units. Due to these problems, this criterion is only used for displacement control mode in a static analysis. Considering that the whole structure is in equilibrium under static loading, Equation (4.87) can be simplified to compare the load value
at the nodal point where the specified displacements are applied. The criterion becomes

\[ |P_{i+1} - P_i| \leq \varepsilon_F |P_i| \tag{4.88} \]

4.7.2. Ill-conditioned stiffness matrix

In nonlinear finite element analysis, the tangent stiffness matrix may become ill-conditioned if there is a significant difference in the values of the stiffness coefficients. Under such circumstances, the solution vector (usually nodal displacement vector) can be sensitive to small changes in the stiffness matrix and thus lead to erroneous results. The diagonal coefficients of the matrix can also be zero- or near zero-valued due to the truncation errors, even if double precision is used. There are several reasons for the ill-conditioned stiffness matrix. First, ill-conditioning may reflect the physical reality of a structure with low tangent stiffness because it is near failure or collapse. For example, under the load control mode, when the structure reaches its maximum capacity, the occurrence of ill-conditioned stiffness matrix implies that the analysis should be terminated. Second, ill-conditioning may arise when the elements in the problem become physically unstable. In the mechanics-based nail model in the finite element program, when the gaps between the nail elements and the wood medium are formed, the nail connector can have virtually zero stiffness because the wood medium only acts in compression. This physical instability in nail connections during deformation is believed to be a reason resulting in the ill-conditioned stiffness matrix. Third, ill-conditioning may occur if the stiffness between the contacted elements is very high but there is no room for them to deform. This problem was detected when the panel-frame contact model in the program was developed. A correctly defined relationship between contact stiffness and deformation can effectively prevent the ill-conditioning from occurring. Finally, the mixing elements in the finite element model may be another reason to induce ill-
conditioning.

The stiffness matrix is checked for ill-conditioning before the solution procedure of the equilibrium equations starts. In the solution procedure, a decomposition method, Cholesky factorization in the current finite element model, requires that a stiffness matrix has to be positive definite. In case the ill-conditioned stiffness matrix occurs, the second self-adaptive procedure (Section 4.7.3) will initiate.

4.7.3. Self-adaptive procedures

The following four self-adaptive procedures are implemented in the program algorithms:

Procedure 1. During the analysis, the norm of solution possibly does not converge to the preset tolerance after a large number of iterations because some of the dominant displacements swing from one point to another alternatively. Figure 4.4 illustrates an example occurring in the iteration procedure in which the searching repeatedly goes between $a_3$ and $a_4$ while the convergence is never achieved.

![Figure 4.4 Cyclical searching in the Newton-Raphson iteration method](image-url)

Chapter 4 Formulation of Static Nonlinear Finite Element Equations
In this situation, the procedure will update the unknown displacement vector by adding one-half of displacement increments from the current iteration to the next, as shown in (4.89)

\[\mathbf{a}_i = \mathbf{a}_{i-1} + 0.5\Delta \mathbf{a}_i\] (4.89)

This procedure is applicable to both load and displacement control modes and can effectively improve the solution mechanism. Usually, convergence can then be achieved rapidly.

Procedure 2. If the solution cannot satisfy the convergence requirements after the preset number of iterations or if the stiffness matrix is ill-conditioned, the simulation of the current step will restart with the step size being halved. Usually, repeating the procedure once or twice will lead to a converged solution.

Procedure 3. Procedure 2 sometimes may not be very effective to solve the convergence problem because the structure could be in a physically unstable state. For example, when a nail is situated in a large hole, further reduction in step size does not solve the convergence problem. In this case, an increase in step size is needed to allow the nail to engage the wood medium again in reaching a new stable state. Therefore, if the solution does not converge after the preset number of reduction in the step dimension, the current step is abandoned and the program proceeds to the next step. In both procedures 2 and 3, the calculation needs to restart from the last converged solution.

Procedure 4. In some cases, the norm of displacement vector or the norm of load vector is found to be very close to the tolerances but not within it after a number of iterations. A slight relaxation in the tolerances is given to let the step finish.

If all attempts from these procedures fail, the execution is terminated. This situation
implies that either the structure reaches failure or the structural model is incorrect. In the second case, the model should be carefully checked and appropriate modifications have to be done to improve the analysis.

4.8. Further expansion of the program

The LightFrame3D program requires the exterior or interior panels in a structural model to have the same thickness and material properties. This limitation will not affect the application of the program to the building components, in which usually only one type of panels is used. In the building study, this limitation may cause some difficulties if more than one type of panels need to be used. Modification in the variables defining the panel properties in the program can remove this limitation to allow the panels to have different thickness, orientation, and material properties.

The displacement control of the program is designed to apply to one node of the structural model. Although it is satisfactory in most light-frame building studies, an expansion can be made to suit the requirement for a multiple-point displacement control.
CHAPTER 5

COORDINATE TRANSFORMATION
IN A THREE-DIMENSIONAL SPACE

This chapter describes coordinate transformation, an essential but important procedure in converting finite element formulation from one space to another. In the first part of this process, three-dimensional coordinate transformation is introduced into finite element formulation to expand the finite element models from two-dimensional space into three-dimensional space, thus making description of the structural behaviour in three-dimensional space possible. Then, the "substructuring" technique is applied to coordinate transformation. This technique considers the panel elements and the frame elements as individual substructures and handles them separately. Therefore, the coordinate transformation becomes more effective after this treatment.

Coordinate transformation is necessary whenever the element coordinates, the construction of the global stiffness matrix, load vectors, boundary conditions, and the calculations of element stresses and strains are required to change from a local coordinate system to a global coordinate system, or vice versa. It is more convenient to input the element nodes by their global coordinates. The inverse transformation is performed when nodal coordinates are transformed to local coordinates during the formulation of element equations. Finally, transformation will bring them back to the global system when the assembly of global system is carried out.
5.1. Coordinate transformation

Figure 5.1 shows an element arbitrarily located in three-dimensional space. Let \((X_i, Y_i, Z_i)\) represent global nodal coordinates, \((\overline{X}_i, \overline{Y}_i, \overline{Z}_i)\) local nodal coordinates, and \(\overline{X}^o, \overline{Y}^o, \overline{Z}^o\) be identity vectors along the edges of the element in \(\overline{X}, \overline{Y}, \) and \(\overline{Z}\) directions, respectively.

![Figure 5.1 Coordinate transformation](image)

The local nodal coordinates \((\overline{X}_i, \overline{Y}_i, \overline{Z}_i)\) can be expressed in terms of global nodal coordinates \((X_i, Y_i, Z_i)\) as

\[
\begin{pmatrix}
\overline{X}_i \\
\overline{Y}_i \\
\overline{Z}_i
\end{pmatrix} = T
\begin{pmatrix}
X_i \\
Y_i \\
Z_i
\end{pmatrix}
\]  

(5.1)

where the transformation matrix, \(T\), can be established as follows

\[
T = \begin{bmatrix}
l_1 & m_1 & n_1 \\
l_2 & m_2 & n_2 \\
l_3 & m_3 & n_3
\end{bmatrix}
\]  

(5.2)

\(l_i, m_i, n_i\) = direction cosines, \(i = 1, 2, 3.\)

Vectors \(\overline{X}^o\) and \(\overline{Y}^o\) can be expressed as
\[ \bar{X}^o = l_1 i + m_1 j + n_1 k = \frac{X_1 - X_0}{L_1} i + \frac{Y_1 - Y_0}{L_1} j + \frac{Z_1 - Z_0}{L_1} k \]  
(5.3)

\[ \bar{Y}^o = l_2 i + m_2 j + n_2 k = \frac{X_2 - X_0}{L_2} i + \frac{Y_2 - Y_0}{L_2} j + \frac{Z_2 - Z_0}{L_2} k \]  
(5.4)

where

\[ L_1 = \sqrt{(X_1 - X_0)^2 + (Y_1 - Y_0)^2 + (Z_1 - Z_0)^2} \]  
(5.5)

\[ L_2 = \sqrt{(X_2 - X_0)^2 + (Y_2 - Y_0)^2 + (Z_2 - Z_0)^2} \]  
(5.6)

\[ i, j, k = \text{unit vectors.} \]

Vector \( \bar{Z}^o \) therefore is

\[ \bar{Z}^o = \bar{X}^o \times \bar{Y}^o = l_3 i + m_3 j + n_3 k \]  
(5.7)

where

\[ l_3 = \frac{(Y_1 - Y_0)(Z_2 - Z_0) - (Y_2 - Y_0)(Z_1 - Z_0)}{L_1 L_2} \]  
(5.8)

\[ m_3 = \frac{(X_2 - X_0)(Z_1 - Z_0) - (X_1 - X_0)(Z_2 - Z_0)}{L_1 L_2} \]  
(5.9)

\[ n_3 = \frac{(X_1 - X_0)(Y_2 - Y_0) - (X_2 - X_0)(Y_1 - Y_0)}{L_1 L_2} \]  
(5.10)

For a general case in which the angle \( \theta \) between \( \bar{X}^o \) and \( \bar{Y}^o \) is not 90° (non-rectangular component), the direction cosines of \( \bar{Y} \) axis needs to be recalculated to ensure that all three axes \( \bar{X}, \bar{Y}, \) and \( \bar{Z} \) are orthogonal

\[ \bar{Y}^o = \bar{Z}^o \times \bar{X}^o \]  
(5.11)

The transformation matrix needs to have the following three properties

\[ \sum_1^3 \cos^2(S, \bar{S}) = 1 \quad \sum_1^3 (lm + mn + nl) = 0 \quad (S = X, Y, Z) \]  
(5.12)

\[ |T| = \pm 1 \]  
(5.13)

\[ T^T = T^{-1} \]  
(5.14)
5.2. Stiffness matrix transformation

In the process of transformation, when local element stiffness matrices are assembled into a global stiffness matrix, "substructuring" is applied to avoid introducing unnecessary higher order degrees of freedom, some of which will be meaningless after transformation. The panel elements and frame elements can be considered as different substructures and can be treated separately. This is possible because panels are connected to frame members by nails only; there is no direct physical connection among panels themselves. By using the substructuring technique, panel elements and frame elements, which are under different coordinate systems, can be assembled into one global system, without increasing the complexity of the analysis. The substructuring procedure used here is not the same as that in a "superelement" procedure, in which a single element can have many boundary and interior degrees of freedom with only boundary degrees of freedom connected to other outside elements. In the substructures of panel elements and frame elements, however, all of the degrees of freedom will be connected to each other via nail connectors. In the process, the transformation of stiffness is only performed in frame elements, during which the stiffness coefficients associating with three translational and three rotational degrees of freedom are transformed while those related to element twisting degree of freedom are left intact, as shown in Equations (5.15)-(5.18). The stiffness coefficients of panel elements are still kept in their local systems. An assemblage of the structural stiffness matrix, therefore, contains transformed global frame stiffness values and non-transformed local panel stiffness values. These values match their own global or local degrees of freedom. An identical transformation matrix $T$ (Equation (5.2)) is used in the transformations for both displacement degrees of freedom and rotational degrees of freedom of frame elements.
For one frame element, stiffness transformation is

\[ K_F = T_k^T k_F T_k \]  \hspace{1cm} (5.15)

where

\[
T_k = \begin{bmatrix} T' & 0 \\ T' & T' \\ 0 & T' \end{bmatrix}
\]  \hspace{1cm} (5.16)

\[
T' = \begin{bmatrix} T & 0 & 0 \\ 0 & T & 0 \\ 0 & 0 & 1 \end{bmatrix}
\]  \hspace{1cm} (5.17)

\[
k_F = \begin{bmatrix} k_{11} & & \\ k_{21} & k_{22} & \\ & \ddots & \ddots \\ k_{28,1} & \cdots & k_{28,28} \end{bmatrix}
\]  \hspace{1cm} (5.18)

In the global stiffness matrix, the stiffness coefficients of the connection elements overlap with those of the panel and frame elements (Figure 5.2). The transformation of the stiffness coefficients should be performed in such a way that only the portion relating to frame elements is transformed from local to global, while the portion relating to panel stiffness is kept in their original local coordinates. A special treatment of the stiffness matrix is applied to satisfy this requirement, as shown in Equations (5.19)-(5.21).

Connection stiffness transformation is

\[ K_C = T_k^T k_c T_k \]  \hspace{1cm} (5.19)

where
\[ T_k = \begin{bmatrix} \mathbf{I} & 0 \\ \mathbf{I} & \mathbf{I} \\ 0 & \mathbf{T}' \end{bmatrix} \begin{bmatrix} \mathbf{I} & \mathbf{T}' \\ 0 & \mathbf{T}' \end{bmatrix} \]  

(5.20)

\( \mathbf{I} \) is an identity matrix and \( \mathbf{T}' \) is the same as that in Equation (5.17).

\[ k_c = \begin{bmatrix} k_{11} \\ k_{21} \\ \vdots \\ k_{28,1} & k_{28,28} \\ k_{29,1} & k_{29,28} & k_{56,29} \\ \vdots & \vdots & \vdots \end{bmatrix} \]  

(5.21)

Figure 5.2 Mapping of stiffness coefficients in element stiffness matrix (only exterior panel is considered)
Accordingly, the displacement transformation from local system to global system is

\[ \mathbf{a}_{Global} = \mathbf{T}_k^T \mathbf{a}_{Local} \]  \hspace{1cm} (5.22)

After the residual force vector \( \psi \) is calculated for each element, but before being assembled into a global vector, a transformation is required. This transformation is expressed as follows.

\[ \Psi_{Global} = \mathbf{T}_k^T \Psi_{Local} \]  \hspace{1cm} (5.23)
6.1. Formulation of the equations of motion

The general equations of motion for a non-linear three-dimensional wood structural system can be shown as

\[ m\ddot{a} + c\dot{a} + f_s(a, \dot{a}) - R = 0 \]  \hspace{1cm} (6.1)

where

\( a \) = degree of freedom vector, function of time.

\( \dot{a} \) = absolute acceleration vector.

\( m \) = element mass matrix.

\( c \) = element damping matrix.

\( f_s \) = resistant force vector, dependent on deformation histories.

\( R \) = external load vector.

In a nonlinear system, the resistant forces are shown as a function of displacement and velocity because the forces are not single valued and depend on the deformation history and on whether the deformation is increasing (positive velocity) or decreasing (negative velocity) (Chopra 1995).

If only one horizontal ground acceleration component (\( \ddot{u}_x \)) and one vertical ground
acceleration component \((\ddot{w}_g)\) of an earthquake are considered, which are assigned to \(x\) direction and \(z\) direction, respectively, then the absolute acceleration vector in Equation (6.1) is represented by

\[
\ddot{\mathbf{a}}_i = \ddot{\mathbf{u}}_i + \ddot{\mathbf{v}} + \ddot{\mathbf{w}}_i \\
= \left(\ddot{\mathbf{u}} + \ddot{\mathbf{u}}_g\right) + \ddot{\mathbf{v}} + \left(\ddot{\mathbf{w}} + \ddot{\mathbf{w}}_g\right) \\
= \ddot{\mathbf{a}} + \ddot{\mathbf{a}}_g
\]  

(6.2)

where

\(\ddot{\mathbf{u}}_i, \ddot{\mathbf{w}}_i\) = absolute nodal acceleration vectors in \(x\) and \(z\) directions, respectively.

\(\ddot{\mathbf{u}}, \ddot{\mathbf{v}}, \ddot{\mathbf{w}}\) = nodal acceleration vectors in \(x, y\) and \(z\) directions, respectively.

\(\ddot{\mathbf{u}}_g, \ddot{\mathbf{w}}_g\) = ground acceleration components in \(x\) (horizontal) and \(z\) (vertical) directions, respectively.

\(\ddot{\mathbf{a}}\) = relative mass acceleration vector.

\(\ddot{\mathbf{a}}_g\) = ground acceleration vector.

Substitute Equation (6.2) in (6.1)

\[
\mathbf{m}\ddot{\mathbf{a}} + \mathbf{c}\ddot{\mathbf{a}} + \mathbf{f}_i(a, \ddot{\mathbf{a}}) = \mathbf{R} - \mathbf{m}\ddot{\mathbf{a}}_g = \mathbf{R} - \left(\mathbf{m}_u \ddot{\mathbf{u}}_g + \mathbf{m}_w \ddot{\mathbf{w}}_g\right)
\]  

(6.3)

where

\(\mathbf{m}_u, \mathbf{m}_w\) = mass vectors related to ground acceleration components.

6.2. Formulation of mass matrix and vectors

The virtual work done by inertia forces is

\[
(\delta W)_M = \int_V \delta \mathbf{u}^T \rho dV \ddot{\mathbf{u}}_i + \int_V \delta \mathbf{v}^T \rho dV \ddot{\mathbf{v}} + \int_V \delta \mathbf{w}^T \rho dV \ddot{\mathbf{w}}_i
\]  

(6.4)

Let

\[
\mathbf{u} = \mathbf{N}_0^T \mathbf{a} \quad \mathbf{v} = \mathbf{L}_0^T \mathbf{a} \quad \mathbf{w} = \mathbf{M}_0^T \mathbf{a}
\]  

(6.5)
then

\[
\delta u^T = \delta a^T N \quad \delta v^T = \delta a^T L \quad \delta w^T = \delta a^T M \quad (6.6)
\]
\[
\ddot{u} = N \ddot{a} \quad \ddot{v} = L \ddot{a} \quad \ddot{w} = M \ddot{a} \quad \ddot{u}_g = N \ddot{a}_g \quad \ddot{w}_g = M \ddot{a}_g \quad (6.7)
\]

where

\[u, v, w = \text{displacement fields in } x, y \text{ and } z \text{ directions, respectively, functions of space and time.}\]
\[N, L, M = \text{shape functions, function of space.}\]

Substituting Equations (6.6) and (6.7) in (6.4) yields

\[
(\delta W)_M = \int \delta u^T \rho dV(\ddot{u} + \ddot{u}_g) + \int \delta v^T \rho dV(\ddot{v} + \ddot{v}_g) + \int \delta w^T \rho dV(\ddot{w} + \ddot{w}_g)
\]
\[
= \delta a^T \int \rho (N_0 N^T + L_0 L^T + M_0 M^T) \rho dV \ddot{a} + \delta a^T \int \rho N_0 dV \ddot{u}_g + \delta a^T \int \rho M_0 dV \ddot{w}_g
\]
\[
= \delta a^T m \ddot{a} + \delta a^T (m_u \ddot{u}_g + m_w \ddot{w}_g)
\]
\[
= (\delta W_f)_M + (\delta W_e)_M
\]

where

\[
m = \int \rho (N_0 N^T + L_0 L^T + M_0 M^T) dV \quad (6.9)
\]
\[
m_u = \int \rho N_0 dV \quad \text{and} \quad m_w = \int \rho M_0 dV \quad (6.10)
\]

The required element mass matrix and element mass vectors related to acceleration components are obtained. In the finite element program, either a consistent mass matrix or a lumped mass matrix can be selected, although only the consistent mass formulation is presented in this section. In the analysis of light woodframe buildings, most of the mass is concentrated at the floor levels, therefore, using a lumped mass matrix is justified.

6.3. Formulation of damping matrix

Damping is a mechanism in dissipating energy and of great importance in controlling the response of a vibrating system. Usually, damping in the analysis of nonlinear systems
cannot be explicitly expressed. Instead, estimated values based on previous experience and/or equivalent viscous damping concepts are used, although the latter was originally developed for the linear elastic systems. The basic purpose of introducing the damping in a structural finite element analysis is to approximate the overall energy dissipation during the system response. For wood structures with nailed joints, the initial damping ratio can be 5-7%. When the structure starts to yield, its stiffness and natural frequencies will decrease whereas the damping ratio can increase significantly. A recent experimental study suggested that the damping ratio in a light-frame building can be 10% or higher when it is undergoing a nonlinear deformation (Filatrault 2000, personal communication).

The traditional damping model mostly used in the time-stepping procedures is the Rayleigh damping. It is in the form

\[ c = \alpha m + \beta k \]  

The damping ratio for the \( n \)th mode of a system is (Figure 6.1)

\[ \zeta_n = \frac{\alpha}{2 \omega_n} + \frac{\beta}{2 \omega_n} \]  

Figure 6.1 Rayleigh damping
The coefficients $\alpha$ and $\beta$ can be determined from specified damping ratios $\zeta_i$ and $\zeta_j$ for the $i$th and $j$th modes, respectively. In the matrix form, Equation (6.12) can be expressed for these two modes as

$$\frac{1}{2} \begin{bmatrix} 1/\omega_i & \omega_i \\ 1/\omega_j & \omega_j \end{bmatrix} \begin{bmatrix} \alpha \\ \beta \end{bmatrix} = \begin{bmatrix} \zeta_i \\ \zeta_j \end{bmatrix}$$

(6.13)

If the damping ratios for both modes are assumed to be the same, then

$$\alpha = \zeta \frac{2\omega_i \omega_j}{\omega_i + \omega_j} \quad \beta = \zeta \frac{2}{\omega_i + \omega_j}$$

(6.14)

The modes $i$ and $j$ should be chosen to ensure reasonable values for the damping ratios in all the modes contributing significantly to the response.

When the Rayleigh damping is only partially used, it is called mass-proportional damping or stiffness-proportional damping (Figure 6.2), which is

$$c = \alpha m \quad \text{or} \quad c = \beta k$$

(6.15)

where

$$\alpha = 2\zeta_i \omega_i \quad \beta = \frac{2\zeta_j}{\omega_j}$$

(6.16)
Because of the complexities of the damping mechanism in wood frame buildings, it is not well understood. In general, viscous damping and hysteresis damping can be used to approximate the secondary elastic damping and primary nonlinear damping in a nonlinear system, respectively. In this study, mass-proportional Rayleigh damping is used to represent the damping in the linear elastic portion of the response as:

\[ c = \alpha \mathbf{m} = (4\pi \zeta f_n) \mathbf{m} \]  

(6.17)

where \( f_n \) is chosen as the structure's natural frequency for the first mode. Mass-proportional damping is significant for the few lowest natural frequencies of a structure, but could under-damp the higher modes of vibration. By using mass-proportional damping, instead of the full Rayleigh damping, it was possible to avoid additional stiffness inverting procedures in the program algorithm. The appropriateness of mass-proportional damping, however, needs to be further studied.

The virtual work done by a damping force can be expressed as

\[ (\delta W_D) = \int \delta \mathbf{a}^T \mathbf{c} d\mathbf{V} \mathbf{a} = \delta \mathbf{a}^T \alpha \mathbf{m} \dot{\mathbf{a}} \]  

(6.18)

In the structural dynamic analysis of nonlinear systems, additional damping resulting from structural nonlinearity is typically accounted for through the hysteresis behaviour of connections. This approach has been a standard practice in many dynamic analysis programs and is, therefore, adopted in this study.

6.4. Formulation and solution of system equations

The principle of virtual work is applied to the whole system to formulate the equations at element level required for the solution. Combining the virtual work terms for panel elements, frame elements, connections, and external load vector derived in Chapter 4, the total virtual work in an element under dynamic excitation is
After differentiating Equation (6.19) with respect to $\delta a$, an element out-of-balance load vector $\psi_e$ under dynamic condition can be obtained

$$\psi_e = m\ddot{a} + \alpha ma + \tilde{f} - R_s - R_c + m_u \dddot{u}_g + m_w \dddot{w}_g = 0$$

(6.20)

where

$$\tilde{f} = \text{the force containing the contributions from the linear and nonlinear behaviour of panel and frame members and from the nonlinear behaviour of nail connections.}$$

It is

$$\tilde{f} = \frac{\Delta x}{2} \left[ \int_{\xi}^{\eta} \left[ (\Theta_{LF} + \Theta_{NP}) \frac{dx}{d\eta} \right] d\eta \right] a$$

$$+ \frac{\Delta x}{2} \left[ \int_{\xi}^{\eta} \left[ (\Theta_{LFH} + \Theta_{NFH}) \frac{dx}{d\eta} \right] a + \frac{\Delta y}{2} \left[ \int_{\eta} \left[ (\Theta_{LFY} + \Theta_{NFY}) \frac{dy}{d\eta} \right] a \right]$$

$$+ \sum [ F(\Delta x)Q_x^T + F(\Delta y)Q_y^T + F(\Delta z)Q_z^T ]$$

(6.21)

The vector $\psi_e$ is obtained for each element and then assembled into the corresponding global out-of-balance load vector $\Psi$.

6.5. Time-stepping procedures

Newmark's average acceleration method is used in the time-stepping procedures. This method produces the smallest numerical errors amongst the direct integration methods used in the dynamic analysis. The method is unconditionally stable for linear systems at any time interval $\Delta t$, although it is more accurate with smaller $\Delta t$. In this study, the step increment was controlled by the acceleration increment $\Delta a$, instead of the time interval $\Delta t$, because, for earthquake excitations, the acceleration can vary sharply within an equal time interval. Usually, an acceleration increment of 0.1-0.5 m/s$^2$ is used in the calculation.
depending on the severity of the ground acceleration. The acceleration \( \ddot{a}(\tau) \) for \( t-\Delta t \leq \tau \leq t \) is given as:

\[
\ddot{a}(\tau) = \frac{\ddot{a}(t-\Delta t) + \ddot{a}(t)}{2} \tag{6.22}
\]

Integrating over \( \Delta t \) gives the velocity and displacement terms as

\[
\begin{align*}
\dot{a}(\tau) &= \dot{a}(t-\Delta t) + \int_{t-\Delta t}^{\tau} \ddot{a}(\tau) \, d\tau \\
a(\tau) &= a(t-\Delta t) + \int_{t-\Delta t}^{\tau} \dot{a}(\tau) \, d\tau 
\end{align*} \tag{6.23, 6.24}
\]

After introducing algorithm parameters \( \gamma = \frac{1}{2} \) and \( \beta = \frac{1}{4} \), corresponding to which the method has the most desirable accuracy characteristics, the acceleration and velocity increments are taken as

\[
\Delta \ddot{a}(t) = \frac{\Delta a(t-\Delta t)}{\Delta t^2 \beta} - \frac{\dot{a}(t-\Delta t)}{\Delta t \beta} - \frac{1}{2 \beta} \ddot{a}(t-\Delta t) \tag{6.25}
\]

and

\[
\Delta \dot{a}(t) = \Delta t \ddot{a}(t-\Delta t) + \frac{\gamma}{\Delta t \beta} \Delta a(t) - \frac{\gamma}{\beta} \dot{a}(t-\Delta t) - \frac{\gamma \Delta t}{2 \beta} \ddot{a}(t-\Delta t) \tag{6.26}
\]

The acceleration and velocity at any time \( t \) can be expressed as

\[
\ddot{a}(t) = \frac{4}{\Delta t^2} [a(t) - a(t-\Delta t)] - \frac{4}{\Delta t} \dot{a}(t-\Delta t) - \ddot{a}(t-\Delta t) \tag{6.27}
\]

and

\[
\dot{a}(t) = \frac{2}{\Delta t} [a(t) - a(t-\Delta t)] - \dot{a}(t-\Delta t) \tag{6.28}
\]

Substituting Equations (6.27) and (6.28) in (6.20) yields
\[ \psi_e = m \left[ \frac{4}{\Delta t} \left( a(t) - a(t - \Delta t) \right) - \frac{4}{\Delta t} \dot{a}(t - \Delta t) - \ddot{a}(t - \Delta t) \right] \\
+ \alpha m \left[ \frac{2}{\Delta t} \left( a(t) - a(t - \Delta t) \right) - \dot{a}(t - \Delta t) \right] + \left( \frac{1}{2} \right)_L + \left( \frac{1}{2} \right)_N \dot{a}(t) \]

\[ - \mathbf{R}_s - \mathbf{R}_c + \mathbf{m}_e \ddot{u}_g + \mathbf{m}_w \ddot{w}_g = 0 \]  

(6.29)

After the element vector \( \psi_e \) is assembled into the global vector \( \Psi \), and the displacement vector \( a \), is evaluated in the iterative solution, a Taylor series expansion of the vector \( \Psi \), truncated after the first derivative, is

\[ \Psi_{i+1} = \Psi_i + \frac{\partial \Psi}{\partial a} \Delta a_i = \Psi_i + \Delta \Psi(a_{i+1} - a_i) \]  

(6.30)

where \( i \) represents the \( i \)th iteration.

The component

\[ \frac{\partial \Psi}{\partial a} = \Delta \Psi \]  

(6.31)

is the global tangent stiffness matrix, which is the assemblage of element tangent stiffness matrices at the \( i \)th iteration.

Assuming that the residual in the \((i+1)\)th iteration \( \Psi_{i+1} \) is zero, an updated displacement vector \( a_{i+1} \) can be obtained from Equation (6.30) based on the vectors \( \Psi_i \) and \( a_i \) from the current iteration, where acceleration and velocity vectors are obtained from last step and are kept constant.

\[ a_{i+1} = a_i + (\Delta \Psi)^{-1} (- \Psi_i) \]  

(6.32)

An element tangent stiffness matrix is obtained by differentiating the equation (6.29) with respect to \( a \)

\[ \frac{\partial \psi_e}{\partial a} = m \frac{4}{\Delta t^2} + \alpha m \frac{2}{\Delta t} + \left( \frac{1}{2} \right)_L + \left( \frac{1}{2} \right)_N = \mathbf{k}_r \]  

(6.33)
where

\[(k)_L, (k)_N =\]

linear part and nonlinear part of element tangent stiffness matrix, as
defined in Equation (4.67) in Chapter 4.

Accordingly, the global stiffness matrix can be expressed as

\[
\frac{\partial \Psi}{\partial \mathbf{a}} = \mathbf{K}_r = \sum \mathbf{k}_r
\]

(6.34)

6.6. Convergence considerations

In the current dynamic nonlinear finite element analysis, the convergence criterion is

\[
\frac{\|\mathbf{a}_i - \mathbf{a}_{i-1}\|}{\|\mathbf{a}_i\|} \leq \varepsilon_d
\]

(6.35)

This criterion is an alternative of Criterion 1 (Equation 4.86) in Chapter 4 to better
suit the requirement of convergence checking in the dynamic analysis. The numerator now
uses the absolute difference of the norms from two subsequent iterations and the denominator
is updated after each iteration. In the meantime, Criterion 2 (Equation 4.87) in Chapter 4 is
used as a secondary criterion in the dynamic analysis.

6.7. Solution of eigenproblems

Before proceeding to the general dynamic analysis of a structure, eigenvalues and
eigenvectors of that structure should be calculated and evaluated by a free vibration without
any dynamic excitation. Solving the eigenproblem is to basically find the roots of the
polynomial. Therefore, in the numerical procedure, the solution methods are iterative in
nature. By studying initially a few of the lowest natural frequencies and corresponding mode
shapes, the relevant modifications in structural configurations can be made and the idealized
structural model is therefore rationalized. The fundamental natural frequency is also needed
in setting up the mass-proportional damping (Section 6.3).

The generalized eigenproblem, if the \( p \) smallest eigenvalues and corresponding eigenvectors are sought, is

\[
K\Phi = M\Phi\Lambda
\]  

(6.36)

where

\( \Phi = \) an \( n \times p \) matrix with its columns equal to the \( p \) eigenvectors and its rows equal to the \( n \) system degrees of freedom.

\( \Lambda = \) a \( p \times p \) diagonal matrix listing the corresponding eigenvalues.

They can be expressed as

\[
\Phi = [\phi_1, \ldots, \phi_p] \quad \Lambda = \text{diag}(\lambda_i), i = 1, \ldots, p
\]  

(6.37)

where

\( \lambda \) = eigenvalues.

The eigenvectors satisfy the orthogonality conditions

\[
\Phi^T K \Phi = \Lambda; \quad \Phi^T M \Phi = I
\]  

(6.38)

where \( I \) is a unit matrix of order \( p \). It is important to note that the relation in (6.36) is a necessary and sufficient condition for the vectors in \( \Phi \) to be eigenvectors, while the conditions in (6.38) are necessary but insufficient. In the above relations, \( K \) has to be positive definite and \( M \) can be either positive definite or positive semidefinite, meaning that \( M \) is either a full (consistent) or diagonal (lumped) mass matrix.

For a massive structural system, \( n \) (the number of degrees of freedom of the system) can be in the order of \( 10^2 \) to \( 10^3 \) or even higher. It is expected that, in order to obtain the \( n \) eigenvalues and eigenvectors, much computational effort is required. Clearly, there is no need to do so for a practical structural problem, in which only a few smallest eigenvalues are
of interest. For this purpose, a widely used effective method, the subspace iteration procedure (Bathe 1996) is included in the program for solving eigenproblems.

The subspace iteration method is particularly suited for the calculation of a few eigenvalues and eigenvectors of large finite element systems. The basic objective of this method is to solve for the smallest $p$ eigenvalues and corresponding eigenvectors satisfying both (6.36) and (6.38). In theory, a subspace of a vector space is defined to be a subset vector space in the original space. The solution procedure was named the subspace iteration method because the iteration is equivalent to iterating with a $p$-dimensional subspace and should not be regarded as a simultaneous iteration with $p$ individual iteration vectors. In practice, to increase convergence rate, the iteration starts with constructing $q$ ($q > p$) linearly independent vectors (starting iteration vectors, usually $q = \min\{2p, p+8\}$), which are iteratively improved as calculation proceeds. The starting iteration vectors, as assumed eigenvectors, have to be selected to excite those degrees of freedom with which large mass and small stiffness are associated. The algorithm involved in the subspace iteration is

1. For the iteration from $i$ to $i+1$:

$$K\bar{X}_{i+1} = M\bar{X}_i$$  \hspace{1cm} (6.39)

When $i = 1$, the $X_1$ contains the starting iteration vectors. The equation (6.39) implies that $\Lambda = 1$. Hence $\bar{X}_2$ can be seen as the displacement vector corresponding to the force vector $MX_1$. The iteration suggests that $\bar{X}_2$ may be a better approximation of eigenvectors than $X_1$.

2. Find the projections of the matrices $K$ and $M$ onto the $p$-dimensional subspace:

$$K_{i+1} = \bar{X}_{i+1}^TK\bar{X}_{i+1} = X_{i+1}^TMX_i$$  \hspace{1cm} (6.40)

$$M_{i+1} = \bar{X}_{i+1}^TM\bar{X}_{i+1}$$  \hspace{1cm} (6.41)

where $K_{i+1}$ and $M_{i+1}$ are the projected matrices in the subspace.
(3) Solve for the eigensystem of the projected matrices:

$$K_{i+1}Q_{i+1} = M_{i+1}Q_{i+1}A_{i+1}$$  \hspace{1cm} (6.42)

(4) Find an improved approximation to the eigenvectors:

$$X_{i+1} = \overline{X}_{i+1}Q_{i+1}$$  \hspace{1cm} (6.43)

The iteration is performed with $q$ vectors, but convergence is measured only on the approximations obtained for the $p$ smallest eigenvalues. If the convergence criterion is not satisfied, a repetition of (1) – (4) is needed. If the solution converges, the eigenvalues $\Lambda$ are replaced with the new eigenvalues $\Lambda_{i+1}$, and the eigenvectors $\Phi$ are replaced with the new eigenvectors $X_{i+1}$.

6.8. Further expansion of the program

The LightFrame3D_Dynamic program described in this thesis is capable of analyzing cases subjected to one horizontal and one vertical component of earthquake ground motion. During the development of the program it was not expected to perform an analysis with bidirectional earthquake excitations. The program can be further expanded in the future to include the second horizontal component of earthquake ground motion, thus maximizing the capacity of the program for any combination of input excitation signals.

In the present version of program, the procedure implemented to solve the equilibrium equations in dynamic analysis is the Newmark method. As mentioned previously, this method provides superior numerical accuracy and unconditional stability. However, this method needs to factorize the stiffness matrix in every time step. When the problem size is large, the time spent on the factorization of the stiffness matrix can be considerably large. To reduce the time spent on this part of dynamic analysis without sacrificing the accuracy, the central difference method may be tested. This method does not
require a factorization of the stiffness matrix and is very effective when a lumped mass
matrix can be assumed (which is the case in most structural analysis), because the system of
equations can be solved without factorizing when the mass matrix is diagonal. Furthermore,
the solution can be carried out on the element level and relatively little memory is required.
The drawback of this method is that it is conditionally stable, which means that the time step
has to be sufficiently smaller than a specified value calculated from the frequency of the
system under consideration.
Before the finite element models can be applied to the practical structural problems with confidence, verification and validation of the model algorithms must be conducted. The programs with correct model and coding should yield accurate results in test cases for which answers are known. At each stage of development of the finite element models, relevant example problems were selected as test cases and the corresponding results were compared to those from mechanics theories or from the commercial finite element software ANSYS. This comparison was limited in the elastic range without triggering nonlinear behaviour from the test case. In the verification procedure, idealized example problems with one or two elements per dimension were extensively employed. The advantages of using this type of problems are obvious. The simplified problems possess most of the characteristics of the large and more complex problems to allow testing of most subroutines and functions in the models. This idealization also provides a manageable environment in the programming and debugging stages and saves a great amount of time in the inevitable repeated computing. In addition, the very limited number of elements and degrees of freedom give a problem a very simple mesh, which is easy to construct and modify. In this chapter, the verification procedures were described starting from the three-dimensional coordinate transformation, which was the first step in the model development, followed by that for the models under
static conditions and dynamic conditions, respectively. The stiffness matrix and mass matrix of basic beam elements were examined and verified manually before they were implemented to the static and dynamic finite element models.

7.1. Verification of three-dimensional transformations

The basic requirement for coordinate transformation is that it is able to transform the quantities from one coordinate system to another without introducing any errors and inconsistencies. The locations and signs of the components of one quantity may be modified through the transformation, but this modification must be correct and exact. The value and accuracy of the components in a quantity should not be altered during the operation. As a result, changes in spatial position of an entire structure have no influence on its nodal point displacements, forces, and boundary conditions.

The verification was performed first to examine the sign, location, and value of each coefficient in the stiffness matrix of the test cases when they were transformed from one coordinate system to another. The transformation of the corresponding displacement vector and load vector and the information for the shared nodes was also studied. As discussed in Chapter 5, the transformation of stiffness is only performed to the frame beam elements, whereas the panel plate elements are always in their local coordinate systems. The results in the transformation were compared to those calculated by hand and by ANSYS. The comparison indicated that the transformations were carried out correctly. The examination of the nodal point displacements and forces of the same problem at different spatial positions was then performed. As expected, the results from those test cases were the same, no matter in which spatial positions they were located. Furthermore, the cases containing two or three symmetrically jointed elements were tested. If the loads were symmetrically applied,
symmetric nodal point displacements and forces were obtained. If the loads were anti-
symmetrically applied, the structure rotated about its center of gravity. A few examples of
the test cases are shown in Figure 7.1.

Figure 7.1 Test cases used in verifications
7.2. Verification of the static finite element model

The static finite element model was verified by running a group of linear cantilever beams with varied cross-sectional shapes in three-dimensional space and then comparing the results with mechanics theory. The beams actually were built-up systems due to the nature of the element unit implemented in the model. The results would reflect the contributions from all three types of elements. Therefore, the results given by the current finite element model and the one from classical beam theory may not be comparable directly if the cases used in the theoretical calculations were only constructed by one type of member with homogeneous material property. To configure an approximate uniform and homogeneous system, only one type of element, for example panel element, was considered at one time by minimizing the contribution of the other elements. This can be easily done by appropriately selecting the parameters of panel, frame, and nail connections. In this way, the panel element and frame element in the model can also be examined separately. To make the comparison of the results from both the numerical and the theoretical procedures available, it is essential for the test case to yield a linear response. This was achieved by limiting the nail deformation within its linear elastic region.

7.2.1. Linear cantilever beams

In constructing linear cantilever beams with dominant panel element properties, low Young’s modulus values and small cross-section dimensions were applied to the frame members so that the stiffness of frame member was negligible and therefore the panel would carry the loads solely. In addition, a high nail initial stiffness $K$ was required to ensure the frame members being tightly fixed on the panel member to eliminate relative movement between them. The Young’s moduli in $x$ and $y$ directions ($E_x$ and $E_y$) were assigned the same
value to create a uniform and homogeneous panel element. On the other hand, when the frame member properties were dominant, the panel member was provided with a small Young's modulus value to ensure the loads being carried solely by the frame. The nail connection with a low initial stiffness $K$ was provided to ensure that the frame members were not restrained by panel member. The configured cantilever beams with four cross-sectional profiles are shown in Figure 7.2. The ratio of beam length to height ($L/H$) was set to eight, which minimized the influence from shear deformation on beam deflection.

Figure 7.2 Test cantilever beams with varied cross-section profiles

The deflections of cantilever beams were calculated by using the program with the three-dimensional finite element model implemented. First, the panel-dominant cases were studied. The beams in all cases were simulated by one, four and eight elements on each side, respectively. When compared to the results from the beam theory, the finite element model with one element gave smaller deflections. When the number of elements increased, the finite element results gradually converged to those of the beam theory, as shown in Table 7.1.
(The unit used in all of the following tables is in meter.) The formula from the beam theory used to calculate the deflection of a cantilever without considering the shear deformation is

\[ \Delta = \frac{PL^3}{3EI} \]  

(7.1)

Beam theory also tells that when a beam is undergoing an elastic deformation, its deflection decreases proportionally when increasing its moment of inertia if its elastic properties and loading conditions are kept constant. This was demonstrated by comparing the moments of inertia and deflections in beams (a*) with doubled thickness, (b), (c) and (d) to those in beam (a), as shown in Table 7.2.

Table 7.1 Comparisons of beam deflections predicted by the finite element model and calculated by the beam theory – Panel-dominant cases

<table>
<thead>
<tr>
<th>Cross-Section</th>
<th>No. of Element</th>
<th>FE Model</th>
<th>Beam Theory</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1</td>
<td>0.2238 x 10^{-2}</td>
<td>0.3199 x 10^{-2}</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0.2929 x 10^{-2}</td>
<td></td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>0.3086 x 10^{-2}</td>
<td></td>
</tr>
<tr>
<td>B</td>
<td>1</td>
<td>0.4962 x 10^{-3}</td>
<td>0.6504 x 10^{-3}</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0.6044 x 10^{-3}</td>
<td></td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>0.6504 x 10^{-3}</td>
<td></td>
</tr>
<tr>
<td>C</td>
<td>1</td>
<td>0.2134 x 10^{-3}</td>
<td>0.2985 x 10^{-3}</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0.2905 x 10^{-3}</td>
<td></td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>0.2975 x 10^{-3}</td>
<td></td>
</tr>
<tr>
<td>D</td>
<td>1</td>
<td>0.2187 x 10^{-2}</td>
<td>0.3078 x 10^{-2}</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>0.2869 x 10^{-2}</td>
<td></td>
</tr>
<tr>
<td></td>
<td>8</td>
<td>0.2912 x 10^{-2}</td>
<td></td>
</tr>
</tbody>
</table>
Table 7.2 The influence of changing moment of inertia

<table>
<thead>
<tr>
<th>Cross-Section</th>
<th>( I )</th>
<th>( r = I/I_a )</th>
<th>Deflection, ( \Delta )</th>
<th>( r_d = \Delta_d/\Delta )</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1.042 x 10^{-3}</td>
<td>-</td>
<td>0.3086 x 10^{-2}</td>
<td>-</td>
</tr>
<tr>
<td>a*</td>
<td>2.084 x 10^{-3}</td>
<td>2.0</td>
<td>0.6172 x 10^{-2}</td>
<td>2.0</td>
</tr>
<tr>
<td>B</td>
<td>5.125 x 10^{-3}</td>
<td>4.9</td>
<td>0.6504 x 10^{-3}</td>
<td>4.7</td>
</tr>
<tr>
<td>C</td>
<td>11.167 x 10^{-3}</td>
<td>10.7</td>
<td>0.2975 x 10^{-3}</td>
<td>10.4</td>
</tr>
<tr>
<td>D</td>
<td>1.083 x 10^{-3}</td>
<td>1.04</td>
<td>0.2912 x 10^{-2}</td>
<td>1.06</td>
</tr>
</tbody>
</table>

\( a^* \): with a doubled beam thickness.

In the frame-dominant situation, only beams (a) and (d) were studied. Table 7.3 lists the deflections from both finite element program and the beam theory.

Table 7.3 Comparisons of beam deflections predicted by the finite element model and calculated by the beam theory – Frame-dominant cases

<table>
<thead>
<tr>
<th>Cross-Section</th>
<th>No. of</th>
<th>Deflection, ( \Delta )</th>
<th>FE Model</th>
<th>Beam Theory</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Element</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>A</td>
<td>8</td>
<td></td>
<td>0.21048 x 10^{-2}</td>
<td>0.21053 x 10^{-2}</td>
</tr>
<tr>
<td>D</td>
<td>8</td>
<td></td>
<td>0.16833 x 10^{-2}</td>
<td>0.16842 x 10^{-2}</td>
</tr>
</tbody>
</table>

The comparisons indicate that the finite element program is able to accurately predict the deflections of fabricated beams used in the verifications.

7.2.2. Beams loaded at shear center

The shear center of a beam is defined as the point upon which when a force is applied, the beam will bend without twisting. The shear center may be located inside or outside of beam cross-section, and is determined solely by the cross-sectional geometry. This concept was introduced as a useful tool in the verification procedures. One cantilevered channel
section and one cantilevered angle section (cross-sections (b) and (d), illustrated in Section 7.2.1, respectively) were adopted in the further testing of the program.

For the cantilevered channel section, a force $P$ was first applied to its shear center, as shown in Figure 7.3(a). The displacements measured at points 1 to 4 are listed in Table 7.4. A graphic display output from the program illustrating the deformation of the end of beam is shown in Figure 7.4(a). The results demonstrated that when the beam was loaded at its shear center, it underwent only bending without twisting. Secondly, when it was only subjected to a force $P$ at point 1 (Figure 7.3(b)), the beam showed a deformation at its end caused by both bending and twisting (Figure 7.4(b)).

![Channel sections](image)

**Figure 7.3** Channel sections were loaded at shear center or at corner 1

<table>
<thead>
<tr>
<th>Point</th>
<th>$\Delta_x$</th>
<th>$\Delta_y$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.443 x 10^{-4}</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>0.443 x 10^{-4}</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>0.436 x 10^{-4}</td>
<td>-0.007 x 10^{-4}</td>
</tr>
<tr>
<td>4</td>
<td>0.436 x 10^{-4}</td>
<td>0.007 x 10^{-4}</td>
</tr>
</tbody>
</table>

**Table 7.4** Displacements of four points at the end of beam when loaded at shear center
Figure 7.4 Output graphics from program showing the deformations at the end of beams

The second test case was a cantilevered angle section. The load was applied at the shear center (corner 1) in two directions, respectively, as shown in Figure 7.5(a) and (b). The displacements are listed in Table 7.5. Graphics outputs from the program displaying the deformations of the end of beams subjected to loads in different directions are shown in Figure 7.6(a) and (b). The results also demonstrated that when the beam was loaded at its shear center, it underwent only bending without twisting.

Figure 7.5 Cantilevered angle sections were loaded at shear center
Table 7.5 Displacements at the end of beams when loaded at shear center

<table>
<thead>
<tr>
<th>Load (a)</th>
<th>Point</th>
<th>$\Delta_x$</th>
<th>$\Delta_y$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td>0.287 x $10^{-3}$</td>
<td>0.287 x $10^{-3}$</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>0.281 x $10^{-3}$</td>
<td>0.286 x $10^{-3}$</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>0.281 x $10^{-3}$</td>
<td>0.286 x $10^{-3}$</td>
</tr>
<tr>
<td>Load (b)</td>
<td>1</td>
<td>0.177 x $10^{-3}$</td>
<td>0.110 x $10^{-3}$</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>0.179 x $10^{-3}$</td>
<td>0.111 x $10^{-3}$</td>
</tr>
</tbody>
</table>

Figure 7.6 Output graphics from program showing the deformations at the end of beams

7.2.3. Superposition of linear deformations

For linear cases, the principle of superposition applies. The cantilevered angle section discussed in Section 7.2.2 was used in the program for testing the superposition of deflections. The load procedure is illustrated in Figure 7.7. The results, listed in Table 7.6, verify that the summation of the deflections caused by separate loading equals that when all loads were applied at the one time.
Table 7.6 Superposition of displacements

<table>
<thead>
<tr>
<th>Point</th>
<th>Direction</th>
<th>Δ of Step 1</th>
<th>Δ of Step 2</th>
<th>1 + 2</th>
<th>Δ of Step 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>X</td>
<td>0.177 x 10⁻³</td>
<td>0.331 x 10⁻³</td>
<td>0.508 x 10⁻³</td>
<td>0.508 x 10⁻³</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.110 x 10⁻³</td>
<td>0.531 x 10⁻³</td>
<td>0.641 x 10⁻³</td>
<td>0.641 x 10⁻³</td>
</tr>
<tr>
<td>2</td>
<td>X</td>
<td>0.180 x 10⁻³</td>
<td>0.304 x 10⁻³</td>
<td>0.484 x 10⁻³</td>
<td>0.484 x 10⁻³</td>
</tr>
<tr>
<td></td>
<td>Y</td>
<td>0.111 x 10⁻³</td>
<td>0.526 x 10⁻³</td>
<td>0.637 x 10⁻³</td>
<td>0.637 x 10⁻³</td>
</tr>
</tbody>
</table>

7.2.4. Stresses in cantilever beams

Stresses in the beam (a) (Figure 7.2, Section 7.2.1) with 8 elements obtained from the finite element program were compared with stresses obtained from the beam theory. The stresses calculated by the beam theory were taken from the middle line along the η axis, whereas the stresses calculated by the finite element program were obtained by averaging the results from the two middle columns of Gauss points (Figure 7.8). The unit of the stresses is kN/m². The beam theory assumes that there is no interaction between two horizontal layers in a cantilever beam and therefore gives zero normal stress in y direction (σy = 0). In finite element analysis, the stresses are calculated from strains at Gauss points in each discrete element based on the relationships from Hooke's law (see Equation (4.6)).
relationships describe that normal stress in y direction ($\sigma_y$) is a function of normal strain in x direction ($\varepsilon_x$). Therefore, in each individual element, $\sigma_y$ is no longer zero.

$$\xi=-0.861..., \eta=-0.339..., \xi=0.339..., \eta=0.861...$$

![Figure 7.8 The locations of Gauss Points in one element](image)

The beam theory uses the following formulas to calculate the normal and shear stresses:

$$\sigma_x = \frac{P L}{I} y \quad \quad (7.2)$$

$$\tau_{xy} = \frac{P Q}{I t} \quad \quad (7.3)$$

The beam theory gives:


$$\sigma_y = 0$$
The finite element program gives:

\[
\sigma_x = \begin{bmatrix}
779.28 & 672.44 & 568.43 & 465.02 & 361.67 & 258.33 & 155.00 & 51.65 \\
294.91 & 264.65 & 224.33 & 183.56 & 142.78 & 101.99 & 61.19 & 20.36 \\
-779.28 & -672.44 & -568.43 & -465.02 & -361.67 & -258.33 & -155.00 & -51.65 \\
\end{bmatrix}
\]

\[
\sigma_y = \begin{bmatrix}
232.68 & 199.97 & 170.16 & 139.44 & 108.49 & 77.50 & 46.50 & 15.49 \\
87.85 & 78.51 & 67.12 & 55.04 & 42.04 & 30.60 & 18.36 & 6.11 \\
-87.85 & -78.51 & -67.12 & -55.04 & -42.04 & -30.60 & -18.36 & -6.11 \\
-232.68 & -199.97 & -170.16 & -139.44 & -108.49 & -77.50 & -46.50 & -15.49 \\
\end{bmatrix}
\]

\[
\tau_{xy} = \begin{bmatrix}
10.42 & 8.30 & 7.86 & 7.86 & 7.86 & 7.86 & 7.86 \\
19.15 & 26.50 & 26.60 & 26.64 & 26.64 & 26.64 & 26.64 \\
19.15 & 26.50 & 26.60 & 26.64 & 26.64 & 26.64 & 26.64 \\
10.42 & 8.30 & 7.86 & 7.86 & 7.86 & 7.86 & 7.86 \\
\end{bmatrix}
\]

The above results indicate that the finite element program can provide accurate and consistent answers compared to the output from the beam theory when the measured points were one element away from the support. Deviations of the finite element results in the region close to the support (the first columns in the three matrices) from the beam theory results were due to stress concentration as explained by the Saint Venant's principle (Popov 1952). It is also seen that the errors in the stress calculations by the finite element program were larger than those in the deflection calculations. The reason is that the stresses are higher order derivatives of the deflections and therefore are prone to be more erroneous.

7.3. Verification of the dynamic finite element model

The dynamic finite element model was verified by using a single degree of freedom
system under harmonic excitations and by comparing the results with dynamic theory. A single degree of freedom system can be constructed by idealizing a simplified 2D or 3D structure. This system consists of a mass concentrated at the top of the structure to provide inertia force, and a massless frame that provides stiffness to the system with or without viscous damping that dissipates energy of the system during vibration. The theory regarding this type of system has been well established based on an understanding of the behaviour and response of the system under harmonic excitation.

7.3.1. Harmonic vibration of undamped single degree of freedom system

In dynamics, a harmonic force could be in the following form:

\[ P(t) = -M\ddot{u}_{g0}\sin(\omega_0 t) \]  

(7.4)

where

- \( P(t) \) = harmonic force.
- \( M \) = system mass.
- \( \ddot{u}_{g0} \) = acceleration amplitude of harmonic excitation.
- \( \omega_0 \) = circular forcing frequency of harmonic excitation.

The differential equation governing the forced harmonic vibration of the system, without considering the damping effect, is

\[ M\ddot{u} + Ku = -M\ddot{u}_{g0}\sin(\omega_0 t) \]  

(7.5)

A cubic box with one meter in each dimension was used as the test case. The parameters have been decided in such a way that the deformation of the cubic box will mostly remain in the elastic region to make the numerical results comparable to the theoretical results. The box carried uniformly distributed mass of 10kN/g/m² on its top and the mass from its panel and frame members was neglected. Thus the case can be simplified.
as a single degree of freedom system. When this system was subjected to the harmonic ground acceleration with the initial displacement and velocity being zero, the relative displacement and mass acceleration, including both transient and steady state, were given by the following two equations, respectively (Chopra 1995, Ch. 3):

\[
\ddot{u}(t) = \frac{M\ddot{u}_g}{K} \frac{\omega_0^2}{1 - (\omega_0 / \omega)^2} \sin(\omega_0 t) - \frac{M\ddot{u}_g}{K} \frac{1}{1 - (\omega_0 / \omega)^2} \sin(\omega_0 t) 
\]  
(7.6)

\[
\dddot{u}(t) = \frac{M\dddot{u}_g}{K} \frac{\omega_0^2}{1 - (\omega_0 / \omega)^2} \sin(\omega_0 t) - \frac{M\dddot{u}_g}{K} \frac{\omega_0 \omega}{1 - (\omega_0 / \omega)^2} \sin(\omega_0 t) 
\]  
(7.7)

By performing the modal analysis, the natural frequency of the system \( \omega \) was 30 sec\(^{-1} \) and the stiffness \( K \) was 917,100 N/m. Furthermore, the forcing frequency \( \omega_0 \) was assigned to be one-half of the natural frequency \( \omega \) of system, which ensured the displacement to be in phase with the applied force. The amplitude of exciting acceleration \( \dddot{u}_g \) equaled 1.

Substituting the above values into Equations (7.6) and (7.7) gives

\[
\dddot{u}(t) = 1.339 \times 10^{-4} \sin(30t) - 2.678 \times 10^{-4} \sin(15t) 
\]  
(7.8)

\[
\dddot{u}(t) = 0.333 \sin(15t) - 0.667 \sin(30t) 
\]  
(7.9)

The responses of the same system were predicted by LightFrame3D Dynamic program. The two sets of results from the dynamics and the finite element method are combined into Figure 7.9. The comparisons indicate that the numerical results correctly predicted the vibration frequency, amplitude, and oscillation path. Minor difference between the two curves existed due to the approximation of the cubical box.
Figure 7.9 Comparisons of responses of an undamped single degree of freedom system
between theoretical calculation and numerical prediction.

7.3.2. Harmonic vibration of damped single degree of freedom system

The differential equation governing the forced harmonic vibration of the system,
considering the damping effect, is

\[ \ddot{u} + \frac{C}{m} \dot{u} + \frac{K}{m} u = -\dot{u}_0 \sin(\omega_0 t) \]  \hspace{1cm} (7.10)

where \( C \) is viscous damping coefficient.

The responses of the system to the harmonic excitation including both transient and
steady state are:
\[ u(t) = e^{-\zeta \omega_0} A \sin(\omega_0 t) - B \sin(\omega_0 t) + D \cos(\omega_0 t) \]  
\[ \ddot{u}(t) = e^{-\zeta \omega_0} A \sin(\omega_0 t) \left[ (\zeta \omega_0)^2 - \omega_0^2 \right] + B \omega_0^2 \sin(\omega_0 t) - D \omega_0^2 \cos(\omega_0 t) \]  

(7.11)  
(7.12)

where

\[
A = \frac{\ddot{u}_g \omega_0}{K} \frac{\omega_0 / \omega}{1 - \left(\frac{\omega_0}{\omega}\right)^2} 
\]

(7.13)

\[
B = \frac{\ddot{u}_g \omega_0}{K} \frac{1 - \left(\frac{\omega_0}{\omega}\right)^2}{\left[1 - \left(\frac{\omega_0}{\omega}\right)^2\right]^2 + \left[2\zeta \frac{\omega_0}{\omega}\right]^2} 
\]

(7.14)

\[
D = \frac{\ddot{u}_g \omega_0}{K} \frac{2\zeta \frac{\omega_0}{\omega}}{\left[1 - \left(\frac{\omega_0}{\omega}\right)^2\right]^2 + \left[2\zeta \frac{\omega_0}{\omega}\right]^2} \]

(7.15)

\[
\omega_\phi = \sqrt{1 - \zeta^2} \]

(7.16)

For the same cubic box used in Section 7.3.1, assuming the damping ratio \( \zeta \) to be 5%, Equations (7.11) and (7.12) become

\[ u(t) = e^{-3.528t} \times 1.339 \times 10^{-4} \sin(70.472t) - 2.666 \times 10^{-4} \sin(35.28t) \]
\[ + 1.777 \times 10^{-5} \cos(35.28t) \]  

(7.17)

\[ \ddot{u}(t) = e^{-3.528t} (-0.663) \sin(70.472t) + 0.332 \sin(35.28t) - 0.022 \cos(35.28t) \]  

(7.18)

Comparison of the results from dynamics and the finite element program is shown in Figure 7.10. Excellent agreements were achieved.
7.4. Capabilities of programs

The finite element program package LightFrame3D is designed to analyze three-dimensional wood light-frame structures and their components or subassemblies under static monotonic and cyclic loading or under earthquake excitation. The static problems are solved by using the LightFrame3D_Static program and the dynamic problems by using the LightFrame3D_Dynamic program. Before the solution of a dynamic problem can be sought, eigenvalues and corresponding eigenvectors have to be calculated by using the LightFrame3D_Eigensolver program, from which the natural frequencies and mode shapes
of the system can be derived. To accurately simulate the actual behaviour of wood structures, both material nonlinearity and geometric nonlinearity are considered in the finite element modeling. The former is associated with changes in material properties, as in the nail connections, and the latter is associated with changes in configuration, as in the large deflections (P-Δ behaviour) of panel and frame members.

LightFrame3D programs try to address the common building configurations and loading conditions in a versatile manner. The programs are capable of analysing structures with the following features:

(1) The components in a system can be either single side sheathed or double side sheathed with or without insulation material applied. When double side sheathed, the material properties and dimensions of panels on one side may be different from the other side. The nail connections on two sides may also have varied configurations.

(2) The material properties and dimensions of frame members along four edges of one element can be different. In some situations when sharing frame members occurs, one element may have only one or two or three frame members if those members are accounted by other elements.

(3) On each frame member, nail connections can be in different type, including material properties, size, and spacing. In general, nail connections behave nonlinearly. Any nail line must have a frame member to match it. In some cases, it is desirable to give linear properties to the nail connections in a number of elements which are not at the critical positions. This may simplify the model, especially of a large problem, to save a significant amount of computational time.

(4) In the static analysis, in-plane concentrated loads and out-of-plane distributed loads can
be applied simultaneously. These loads may be constant or incremented individually in fixed or uneven steps. Load control mode or displacement control mode is used as input history in static monotonic analysis, whereas only displacement control mode is used in static cyclic analysis. In the dynamic analysis, the earthquake excitations can be assigned in horizontal and/or vertical directions and the static constant loads can be applied to the structure at the same time.

(5) Window and door openings can be located on a component in different sizes and locations.

(6) There is no limitation on the problem size and the number of elements from the program code because of the use of dynamically allocated arrays. The problem size as well as the computational time, however, largely depends on the capacity of a computer system to be used and the nature of the problem.

The programs are built with three major modules: preprocessor, solution, and postprocessor. The preprocessor generates appropriate text and graphic files to allow the user to review the input data and mesh information prior to execution of the solution procedure. The computational effectiveness and accuracy are based on a well-constructed model. Material properties, nail connection types, loading schemes and boundary conditions in the model must be correctly defined. Different element sizing, nodal numbering and meshing patterns could be tested to investigate the model sensitivity and stabilization. The model should be checked by closely reviewing all the information in the text files and graphic files provided by the preprocessor. For large problems, the graphic display provided by the programs is preferable because the errors in input data files can be easily detected from the visualized graphs. Such common errors in input data include misplaced nodes or
their coordinates, incorrectly numbered elements, misapplied boundary conditions, and loads.

Before conclusions can be drawn from the solution procedures, the most important step is to review all the results generated by the postprocessor based on engineering judgments. These results are in the form of text and graphic files detailing local and global deformation and force information and displaying visualized step-by-step structural deformation. The text files include nodal displacements and forces in six degrees of freedom, panel normal and shear stresses, maximum nail forces and locations, nail bending and withdrawal deformations, and relative movement between frame and panel members. The tearing failure at panel edges due to nail compression and maximum bending stress in frame members are also given if they need to be checked. The graphic files show the deformed shape of the structure being modeled in each step (in the static analysis) or selected steps (in the dynamic analysis). A set of slides is also produced to display an entire continuous load-deformation course of the structure. Detailed user procedures of the programs are presented in Appendix A.
CHAPTER 8

PROCEDURES OF EXPERIMENTAL STUDY ON SIMPLIFIED THREE-DIMENSIONAL BUILDINGS

As an integral component of the numerical modeling of three-dimensional wood-frame buildings, an experimental study was proposed, organized and conducted. The study contained a group of laboratory tests on full-scale simplified three-dimensional 2 by 4 light-frame structures. They were of realistic size and scale, but not installed to represent a real house. The main objective of the experimental study was to provide comprehensive test results required in the verifications of finite element models. These tests can also provide an opportunity for researchers to physically observe the actual behaviour of simplified wood light-frame buildings under static loading and dynamic excitation. This chapter introduces the test facilities for this study with the description of test setup and input signal generating and collecting system. It presents in detail the procedures in determining the test models and their structural components with the help of programs. The test results will be given in Chapters 9 and 10, integrated into the experimental-based model verification procedures.

8.1. Test facility

All of the static and dynamic tests were conducted on the shake table in the Earthquake Engineering Laboratory in the Department of Civil Engineering at UBC. The test buildings were mounted on a base frame that was attached to the surface of the table. The entire test facility consisted of shake table, test base frame, inertial mass, computer
control system, and data acquisition system. The test base frame, composed of steel I-beams and channels, was designed after the dimensions of test buildings had been determined.

8.1.1. Capacity of shake table

The shake table is a 3 × 3 m aluminum cellular structure, capable of simulating multiple-directional earthquakes. In the tests, one of the configurations of the shake table mechanism, 3H, was used. This mechanism generated only horizontal motions, from which two translational motions and one rotational motion can be simulated. Under this configuration, the vertical supports acted as rigid links. For clarity, the direction running in East and West was defined as the primary direction, represented by $x$, and the direction running in North and South as the secondary direction, represented by $y$, as shown in Figure 8.1. The horizontal motion in the primary direction was produced by a table actuator with a maximum load capacity of 156 kN and a maximum displacement of ±76 mm (A1 in Figure 8.1). In the bi-directional dynamic test, the horizontal motions perpendicular to the primary motion were generated by two additional actuators located in the south side of the table (A2 and A3 in Figure 8.1). The motions of the shake table were controlled by specialized software which performed a closed-loop control to reproduce recorded earthquake motions with high accuracy. A data acquisition system with 32 channels was equipped for recording the responses from test specimen.

8.1.2. Data acquisition system

A total of 28 channels in the data acquisition system were employed to measure loads, displacements, and acceleration signals during tests. Figure 8.1 illustrates their positions.

In the static tests, one load cell (G1) with 45 kN load capacity was fixed in between
the rigid arm from a reaction frame and the steel beam connected to the top of the test building to measure the load that the building was subjected to. One Linear Variable Displacement Transducer (LVDT) (D1), mounted on the actuator A1, was used to measure the displacement of the shake table in the loading direction. Three LVDTs in the x direction (B1, B2 and B3) and two in the y direction (B4 and B5) were installed to measure the displacements on the top of the test building. The data from those LVDTs were also used to calculate the building torsional motion. Two dial gauges were vertically mounted at two eastside base corners to obtain the uplift readings of end frame members, assigned as C1 and C2. In addition, two LVDTs (F1 and F2) were installed diagonally on both the north and south shear walls to measure the diagonal displacements, which may help to determine shear wall deformation.

In the dynamic tests, besides using the above-mentioned transducers for displacement measurements, four accelerometers (E1 to E4) were mounted at the center and corners of the top of the test building to monitor the resultant acceleration signals in the building in the primary (_p) and secondary (_s) directions. Two accelerometers were also used to measure the table accelerations in both directions. The dial gauges (C1 and C2) were replaced by transducers, and two more transducers (C3 and C4) were added to measure dynamic uplifts. In the bi-directional dynamic test, two LVDTs (D2 and D3) mounted on actuators A2 and A3 measured the displacements of the shake table in the secondary direction. MTS Seismic Shock Simulation System and Data Acquisition System were used to regenerate the earthquake signals and record the test signals.

The sampling frequency was 50 readings per second in the static tests and 200 readings per second in the dynamic tests.
8.2. Determination of test buildings

The testing specimens were predetermined to be a few full-scale one-storey 2 by 4 light-frame simplified box-type models with a rectangular plan. They would have four walls and one roof, built with commonly used construction materials. Due to the limitation of shake table space, the dimensions of test models were restricted to 3.048 m (10 ft.) in the $x$ direction $\times$ 2.438 m (8 ft.) in the $y$ direction $\times$ 2.438 m in height. Accordingly, the structural details of the test models need to be scaled so that the data for the model verifications can be obtained, while the maximum loading capacity of the shake table would not be exceeded. The tests need to provide the following information:

(1) Load-displacement characteristics under static loading condition;

(2) Natural frequencies and mode shapes under free vibration condition;

(3) Dynamic response under ground excitation;

(4) Torsional response under both static and dynamic loading; and

Chapter 8 Procedures of Experimental Study on Simplified Three-Dimensional Buildings
(5) Failure modes.

To best configure the test buildings and their parameters, the LightFrame_3D finite element programs were employed.

8.2.1. Wall opening

As discussed in Chapter 2, torsional motion is typically generated when the structural layout of a building is asymmetric, which exerts additional risk to the building. In the past, this phenomenon was not well quantitatively studied by experimental method. The prediction made by theoretical model, if any, was not accurate. In the current study, therefore, it was a basic requirement to examine the torsional motion in a building and to predict it using the developed software. For this purpose, a large opening was created on one wall in the x direction to create a significant asymmetric structure. Two opening configurations were considered, as shown in Figure 8.2. In the first configuration (Figure 8.2 (a)), the dimension of the opening was 3.048 m (10 ft) in width, which was equal to the length of the test building, and 2.032 m (80 in) in height. The area of the opening comprised 83% of the area of the shear wall. A narrow lintel was installed to provide vertical support to the roof and dead load. In the second configuration (Figure 8.2 (b)), a partial shear wall was added beside the lintel. Thus the dimension of the opening was reduced to 2.438 m (8 ft) in width by 2.032 (80 in) in height, representing 67% of the area of the shear wall. The first building can be considered as a U-shaped building where most of the torsion is resisted by the two parallel walls. The second building would represent a more typical building where the shear walls add some torsional stiffness. To decide which opening configuration to use in the tests, the LightFrame_3D programs were applied to examine their influence on the building behaviour. Both structures were excited dynamically by the east-west acceleration component of an earthquake in Landers, California in 1992, recorded at the Joshua Tree...
Station (Durham et al. 1997). The peak acceleration of the record was scaled up to 0.35g (Figure 8.3).

Figure 8.2 Proposed configurations of test buildings

(a) Test building with Opening A

(b) Test building with Opening B

Figure 8.2 Proposed configurations of test buildings
Figure 8.3 Joshua Tree record, 1992 California Landers earthquake, 0.35g

Five groups of displacement data on the roof of each test buildings were output by the program, as shown in Figure 8.4, to fully represent the in-plane and out-of-plane motions of all the walls and the rigid body lateral motion of the roof. The drifts in the \( x \) direction (the primary direction) measured at point \( x_0 \) on the test building for Openings A and B are presented in Figure 8.5. The relative drifts in the \( x \) (the difference between \( x_1 \) and \( x_2 \)) and the \( y \) (the difference between \( y_1 \) and \( y_2 \)) directions for two test buildings are presented in Figure 8.6 and Figure 8.7, respectively. The graphs start from the fifth second, because the vibrations of the systems during the first five seconds were insignificant.

Figure 8.4 Locations of displacement measurement for torsion calculation
Figure 8.5 Comparison of drifts from the test buildings with Openings A and B ($x_0$)

Figure 8.6 Comparison of relative drifts in $x$ direction from the test buildings with Openings A and B ($x_1 - x_2$)

Figure 8.7 Comparison of relative drifts in $y$ direction from the test buildings with Openings A and B ($y_1 - y_2$)
As evident in Figure 8.5, the difference between the two predictions in most of the time period is not significant. The test building with Opening A reached a maximum drift of 81.7 mm and that for the test building with Opening B was 66.3 mm. The relative drifts in both the \(x\) and \(y\) directions, however, showed a substantial difference. For example, in the \(x\) direction, the maximum relative drifts in the buildings with Opening A and B were, respectively, 11.0 mm and 7.2 mm, and in the \(y\) direction, they were 6.8 mm and 5.0 mm. On the other hand, if considering the ratio of the relative drifts to the absolute drift in each case, the influence due to the opening on the building torsional motion was at the same level of significance.

It would not be hard to imagine that the stresses in the top beam and lintel of the test house with Opening A were greater than those in the house with Opening B when the top dead load was applied. It is because the former had a longer opening span, whereas in the latter, the framing studs of the narrow shear wall provided additional support to reduce the stresses in the members. Based on the information obtained from the predictions by the program and considering that Opening B was closer to the actual construction layout, the decision was made to use the Opening B in all of the test buildings.

8.2.2. Structural details of walls in test buildings

When constructed with the standard building configurations and materials, the test models in the determined dimensions could be much stiffer. Their strength might exceed the load capacity of the shake table, and their deformation might not reach the expected level. Thus, the structural details needed to be adjusted to create softened test buildings. The change, which would be easily implemented to affect the building behaviour, was considered to be the enlargement in the distance between the framing studs. This change reduced the
total number of nails in the sheathing to framing connections and, therefore, effectively scaled down the building ultimate capacity. The distance of the framing studs was increased from the standard 406 mm (16 in) or 508 mm (20 in) to 610 mm (24 in). The conventional nail distribution pattern was adopted with 152 mm (6 in) nail spacing along the panel edges and 305 mm (12 in) nail spacing in interior panel areas.

Performance Rated W24 Oriented Strand Boards, 9.5 mm (3/8 in) in thickness, were used as sheathing panels. In the majority sheathing area, the panel dimensions were 1.2 m × 2.4 m (4 ft × 8 ft), while in the wall with an opening, the panels were tailored to fit the narrow wall strips besides and above the opening. All the panels were oriented vertically, except for the panel on the wall strip above the opening, which was horizontally arranged. The material properties of panels are: Young’s modulus in panel longitudinal direction $E_x = 6.4$ GPa, Young’s modulus in panel transverse direction $E_y = 4.7$ GPa, Shear modulus $G_{xy} = 1.5$ GPa, Primary Poisson’s ratio $\nu_{xy} = 0.22$, Secondary Poisson’s ratio $\nu_{yx} = 0.3$, and density $= 0.4$. The sheathing panels were attached to the frame members by 2.66 mm × 50 mm (2 in) (6d) pneumatically driven spiral nails. The Young’s modulus $E$ of nails is 200 GPa and the yield stress $\sigma_y$ is 250 MPa. The frame members used No. 2 and Better Spruce Pine Fir 38 mm × 89 mm lumber. The top plates of all the walls and the end studs in shear walls (North and South walls) consisted of double members, whereas the sill plates and interior studs in all the walls used single members. The material properties of frame members are: Young’s modulus $E = 9.5$ GPa, Shear modulus $G = 0.76$ GPa, and density $= 0.4$. The frame members were connected with 3.11 mm × 76 mm (3 in) (10d) pneumatically driven common nails. Instead of constructing the entire building frame on the shake table before sheathing, the individual walls were completely built and then assembled into a building. In assembling the
building, the single end studs of the end walls (East and West walls) were firmly nailed to the end studs of the shear walls (North and South walls). The panel edges in the end walls (East and West walls) were nailed to the end studs of the shear walls with 152 mm (6 in) nail spacing. Hold-downs were applied at four corners of the building and at the opening end stud. In the finite element model, the hold-downs are treated as "rigid connections". A more sophisticated approach is to consider the hold-downs as non-rigid connections and use nonlinear springs to represent them. Implementation of nonlinear springs as future improvement of the model is discussed in Sections 9.3 and 9.4. The material properties for panels, frame members and nails given in this paragraph were used in the entire numerical predictions and analyses.

Figure 8.8 presents framing and sheathing details of shear walls and end walls.

(a) Front (North) shear wall with opening
Figure 8.8 Wall frame and panel layouts

(b) Back (South) shear wall

(c) East and West end walls
8.2.3. Structural details of roof in test buildings

The roof in a test building plays an important role in transferring the forces resulted from the inertia mass on the top to the foundation and in keeping the integrity of the building by holding the vertical components. In the current study, it was expected that the roof should be constructed with enough rigidity to support excessive dead load applied to the building.

The roof had dimensions of 2.4 m (8 ft) by 3.05 m (10 ft), composed of 38 mm (2 in) × 235 mm (10 in) Douglas Fir joists and 19 mm (3/4 in) Douglas Fir Plywood (Figure 8.9). Six evenly distributed boards running in the north-south direction were connected to two continuous end boards with Ø3.11 mm × 76 mm (3 in) (10d) pneumatically driven common nails. Along the span of the joists, three bracing blocks were nailed to the joists with the same type of nails. Three plywood panels, two pieces of 1.2 m (4 ft) × 2.4 m (8 ft) plus one piece of 0.6 m (2 ft) × 2.4 m (8 ft) in the middle, were attached to the roof frame with glue and floor screws. The panels were oriented in the north-south direction (perpendicular to the loading direction).
Figure 8.9 Plan view of roof construction

The Simpson Strong-Tie® light gauge reinforcing angles (A in Figure 8.10) were nailed to each corner of the joist and the bracing block to provide additional strength. During the tests, the roof was very tightly connected to all of the four walls by means of 18 Simpson Strong-Tie® reinforcing angles along the outskirts of the roof in each bay (B in Figure 8.10). These angles were bolted to the roof joists and the top plates of walls by using two 9.5 mm (3/8 in) bolts on each face. The reason to apply this unusual roof-to-wall connection was because it was crucial to the entire test to prevent any relative movement between the roof and the walls.
8.2.4. The gravity load applied to test buildings

The design of wood frame houses requires that the roof/floor diaphragm should be capable of supporting a uniformly distributed gravity load of 4.8 kN·m$^2$ (100 lb·ft$^2$). The load represents the combination of the weight of the structure and furnishings (dead loads), and snow, rain, and/or occupants (live loads). The current design code requires that when subjected to earthquake ground motion, the building carrying this designated load should not collapse. In the future performance-based design code, the design criterion may require that the building not experience any considerable damage. In the proposed dynamic testing, in order to generate adequate inertia force to cause significant displacement and damage in test buildings, the gravity load needed to be increased. In the early stage of designing the test buildings, besides considering the size of the shake table, the dimensions of the existing steel plates in the laboratory as the gravity load were also taken into account in determining the roof area. There were sixteen pieces of steel plates available. They were 1.5 m (59 in) in
length, 0.61 m (24 in) in width, and 64 mm (2.5 in) in height, as shown in Figure 8.11. The total weight of sixteen pieces was 71.1 kN (16,000 lb), which was equivalent to 9.6 kN·m$^2$ (200 lb·ft$^2$).

Figure 8.11 Arrangement of steel plates as gravity load

The steel plates were staggered on the roof in two layers and were fixed to the roof by twenty-four $\phi$25.4 mm (1 in) anchor bolts. To prevent the steel plates from moving horizontally, wood and steel blocking was attached to the edges of the roof (Figure 8.12).

Figure 8.12 Attachment of the steel plates to building roof
8.2.5. Selection of earthquake records

Ideally, the selected earthquake record as the input acceleration history in experimental study should contain a great amount of energy, high acceleration amplitudes, and long enough duration, and its frequency components should match as much as possible the natural frequencies of the test structure. These conditions promise to generate destructive drive to the structure by either strong inertia forces or near resonance effect. Three available earthquake records, with their scales being modified for in-lab dynamic testing, were analyzed and compared in determining the input signal. One of such records was the Joshua Tree Station record of the 1992 California Landers earthquake as demonstrated in Figure 8.3. The other two were from the 1994 California Northridge Earthquake, recorded at Canoga Park Station (Figure 8.13) and at Sherman Oaks Station (Figure 8.14).

The 1992 California Landers earthquake contained two segments of high amplitude of acceleration with long duration. The maximum amplitudes in the two segments were 0.35g and 0.31g and occurred at 9.78 sec and 26.5 sec, respectively. The 1994 California Northridge earthquake only had one strong acceleration peak, which was 0.35g at 7.22 sec measured at the Canoga Park Station and 0.44g at 8.06 sec at the Sherman Oaks Station.

![Graph of Ground Acceleration](image)

Figure 8.13 Canoga Park record, 1994 California Northridge earthquake, 0.35g
The Fourier spectra of the individual earthquake records are shown in Figure 8.15. The results indicate that in all three records the frequencies corresponding to the maximum amplitude are lower than 2.15 Hz. In the Joshua Tree Station and Canoga Park records, most of the energy is concentrated around 1.85 Hz and 1.65 Hz, respectively. The Sherman Oaks record, on the other hand, has a broad frequency band. The amplitudes of frequencies in the three records differ significantly. The peak amplitude of the maximum frequency in the Canoga Park record is approximately six times higher than that in the Joshua Tree Station record and 17 times higher than that in the Sherman Oaks record.
To be able to further judge the effect of each record on the structural behaviour, a simplified 3D model with the same structural conditions was tested when subjected to the individual excitations. The tests were stopped approximately at the sixteenth second for all three cases, because after this moment, the excitation amplitude tended to be diminutive (in the Joshua Tree record, it means the first strong shake.). Figure 8.16 presents the drift-time histories of the test building when excited by three earthquake records. The maximum absolute drifts are 66.3 mm (Joshua Tree record), 68.0 mm (Canoga Park record) and 60.3 mm (Sherman Oaks record). The difference among them is insignificant. The comparison of the earthquake records may indicate that due to the similar acceleration amplitudes and frequency components, these records may not result in significant discrepancy in structural performance in the studied time interval. The advantage of the Joshua Tree record over the other two records is that it has the second high amplitude segment. The damage resulting from the first segment could soften the building and lower the natural frequencies.
Therefore, it is predicted that the building will have a substantial drift when subjected to this second shaking segment, which may result in a further damage in structure. The previous studies (Latendresse and Ventura 1995, Durham 1998) demonstrated the record to be an ideal testing input due to its significant energy. Based on these considerations, the Joshua Tree record was selected and used in the actual tests.

The determined structures of test building are shown in Figure 8.17 and Figure 8.18.

(a) Response resulted from the Joshua Tree record

(b) Response resulted from the Canoga Park record
(c) Response resulted from the Sherman Oaks record

Figure 8.16 The influence of earthquake record on building behaviour

(a) North elevation
Figure 8.17 The scheme of 3D building testing assembly

(a) North-West view

(b) South elevation
The finite element programs were employed prior to testing to predict static and dynamic response of the model buildings. Comparisons made after the tests were conducted indicated that these predictions were in good agreement with the results obtained from testing. The results will be described in Chapters 9 and 10 in detail.

8.3. Experimental procedures

A total of four three-dimensional building tests were carried out. In Test One, the building was statically loaded, undergoing a monotonic pushover. No dead load was applied to the building in this test. The test building in the second test was retrofitted from the first test building by replacing all the panels on shear walls and re-installing all the hold-downs, whereas the frame members were not changed (inspection showed no damage in frame members). The steel plates as the dead load were attached to the roof of the test building. The building was loaded statically in the same scheme as in the first test. In the static tests,
The shake table was moved to the west end to provide a 152 mm (6 in) lateral clearance. This distance was believed to be enough for the test building to reach and go beyond the ultimate strength level. The loading rate was 0.13 mm/sec in compliance with the tests done previously at the UBC. The roof of test buildings was connected to the reaction column with a rigid arm to remain stationary, whereas the base moved with the shake table. After the second test, this building also served as the model to calibrate the Seismic Shock Simulation System and the Data Acquisition System in regenerating the earthquake signals and recording the test signals.

The third and fourth experiments were dynamic tests, with the Joshua Tree Station records as the input excitation history. The specimen used in the third test was built with four new walls. In the third test, the excitation was applied only in the \( x \) direction, whereas in the fourth test, the building was subjected to a combination of earthquake components in both the \( x \) and \( y \) directions. The specimen in the fourth test was retrofitted from that in the third test, with all panels on four walls replaced. The sine wave frequency sweep tests were conducted on both specimens, prior to the destructive earthquake excitations, to determine the buildings' natural frequencies and corresponding mode shapes.

8.4. Brief description of the two-dimensional shear wall tests

In the last few years, a number of experimental projects have been carried out at the University of British Columbia to investigate the structural performance of timber shear walls. The database generated from the projects was capable of providing the required information in the validation of the current numerical models.

Two types of the previously tested shear walls (Durham et al. 1997) were selected as examples in this numerical study, which were tested under the both static loading and
earthquake excitations. The walls, 2.4 × 2.4 m (8 ft × 8 ft) in dimensions, were built by commonly used construction materials. The wall frames were composed of 38 × 89 mm (2 × 4 in) dimension lumber, spaced at 406 mm (16 in) on center. Oriented Strand Board panels, 9.5 mm (3/8 in) thick, were sheathed on the wall frame by 50 mm (2 in) pneumatically driven spiral nails. The differences in these two types of shear walls were the panel dimension and nail spacing. One wall used conventional sized OSB panels, staggered horizontally. The nail spacing on that wall was 152 mm (6 in) on edges and 305 mm (12 in) on interior studs. The other wall was sheathed with one 2.4 × 2.4 m jumbo panel nailed to the frame in the nail spacing of 76 mm (3 in) on edges and 305 mm (12 in) on interior studs. In this way, the numbers of nails used in the two walls were approximately the same.

In the static tests, lateral in-plane load was applied to the top corner of wall specimens either monotonically or cyclically. The cyclic loading schedule consisted of three groups of cycles, with each cycle group containing three identical cycles. The amplitudes of the cycles were taken as 30%, 50% and 80% of the nominal yield displacement, which was defined as the displacement at the load level equal to 50% of the maximum load obtained in the corresponding monotonic test. Both shear walls were subjected to a vertical constant load of 9.12 kN/m and tested in a test assembly displayed in Figure 8.19.

The two types of shear walls were also tested dynamically by using the same earthquake testing facilities as described in Section 8.1. The typical test setup is shown in Figure 8.20. The inertia mass on the top of support frame was 4500 kg. The control earthquake input was the east-west component, recorded in the Joshua Tree Station, of the Landers earthquake of June 28, 1992 in California (Figure 8.3). This was the same record as used in the third three-dimensional building test.
More detailed information can be found in Durham (1998), Durham et al. (1997, 1998, 1999), or Lam et al. (1998).
In this chapter, the test facilities and procedures are described in detail. The experimental results will be presented in the next two chapters to better reflect the integrity of the experimental based verification of the finite element models.
CHAPTER 9

EXPERIMENTAL-BASED VERIFICATION OF
THE STATIC FINITE ELEMENT MODELS

The second stage of verification of the finite element models by using the experimental data from the tests described in Chapter 8 is presented in this and the next chapters. Although they were constructed as idealized models, the test buildings displayed some of the behaviour found in the actual houses. Among those, the torsional effects, deformations, and failure modes in the structural components were of much interest. The finite element programs were operated mainly in the nonlinear range. This range was far beyond the limitation of the verification presented in Chapter 7, which could only be conducted for small deformations and in the linear elastic range. Therefore, the verification described in these two chapters is much more meaningful and critical. Without such a step, the verification of the finite element models would not be thorough and complete. Because full size simplified test structures were used, this step can also be seen as preliminary application of the programs and study of wood light-frame building systems.

9.1. Experimental results of three-dimensional building under static loading conditions

Two three-dimensional buildings were tested under static unidirectional loading. The following notations are defined to represent the characteristics of the test buildings obtained from the tests, which are summarized in Table 9.1.

\[ P_{\text{max}} = \text{maximum load carrying capacity of test building, kN.} \]
A* = ultimate displacement of test building in x direction at the maximum load $P_{\text{max}}$, mm.

$\Delta_y$ = maximum displacement of test building in y direction, mm.

$\theta$ = maximum torsional rotation of the roof of test building about the vertical axis, which passes through the center of mass of the roof diaphragm, rad.

$D$ = dead load applied to test building, kN.

$O_{C1, C2}$ = uplift displacements measured by dial gauges C1 and C2, respectively, mm. Referring to Figure 8.1, C1 was located at the northeast corner of the building and C2 at the southeast corner of the building.

Table 9.1 Summary of static test results of three-dimensional buildings

<table>
<thead>
<tr>
<th>Test No.</th>
<th>$P_{\text{max}}$ (kN)</th>
<th>$\Delta_x$ (mm)</th>
<th>$\Delta_y$ (mm)</th>
<th>$\theta$ (rad)</th>
<th>$O_{C1}$ (mm)</th>
<th>$O_{C2}$ (mm)</th>
<th>$D$ (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>28.3</td>
<td>96.6</td>
<td>11.3</td>
<td>1.04x10^-2</td>
<td>5.08</td>
<td>10.16</td>
<td>0.0</td>
</tr>
<tr>
<td>2</td>
<td>26.8</td>
<td>74.8</td>
<td>9.5</td>
<td>0.84x10^-2</td>
<td>4.57</td>
<td>9.14</td>
<td>71.1</td>
</tr>
</tbody>
</table>

The data from the two static tests were filtered before they could be implemented into the analysis. Figure 9.1 provides the load-displacement relationship of the building in Test One. The load reading was given by a load cell (G1) installed in the rigid arm connecting to the middle of the short span of the building roof, while the displacement was measured at the shake table (D1). Figure 9.2 shows the load-displacement relationship of the building in Test Two. It is notable that the load-displacement curves show a zigzag pattern, which indicate cracking in the wood medium under the nail compression and failure in panels and frame members.
The load carrying capacity of the building in Test Two showed a steep load decline after its peak value was reached compared to that in Test One, even though both buildings had comparable maximum load levels. This phenomenon can be seen more clearly from Figure 9.2.
Figure 9.3 where two load-displacement curves are presented together. This can be attributed to the $P$-$\Delta$ effect caused by the dead load applied to the second test building. The dead load also resulted in smaller torsional motions and uplifts in the second building, because the joints between frame members were firmly compressed downwards and, therefore, resisted separation and rotation. For both tests, the uplift at the southeast corner (C2) was greater than that at the northeast corner (C1). The reason was that the shear wall without opening had a higher stiffness and therefore carried a higher load. The shear wall with the opening was undergoing a larger overall lateral deformation, however.

![Figure 9.3 Comparison of load carrying capacities of two buildings](image)

9.2. Prediction of behaviour of the three-dimensional buildings under static loading conditions

The numerical predictions made by the LightFrame3D program on the maximum load carrying capacities and the maximum torsional motions of the two test buildings are compared with the static test results. Table 9.2 summarized these results, using the same
definitions of notations as in Table 9.1. The numerical results from five points on the roof of the building were recorded to calculate displacement and torsional motions. Among these points, one was located at the center of the roof to measure the displacement in the $x$ direction, two respectively at the north and south corners in the $x$ direction, and two respectively at the east and west corners in the $y$ direction. The difference in the displacements measured at two opposite points (e.g. the north and south corners) was calculated and then divided by the span of the roof to obtain the angle of roof rotation. The rotational results in both $x$ and $y$ directions were similar, but not exactly the same due to slight change in the roof geometric shape, because the roof was not perfectly rigid.

Table 9.2 Comparison of static test results and numerical predictions

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Data Source</th>
<th>$P_{\text{max}}$ (kN)</th>
<th>$\Delta_x$ (mm)</th>
<th>$\Delta_y$ (mm)</th>
<th>$\theta$ (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Test</td>
<td>28.3</td>
<td>96.6</td>
<td>11.3</td>
<td>1.04×10^{-2}</td>
</tr>
<tr>
<td></td>
<td>LightFrame3D</td>
<td>27.9</td>
<td>90.0</td>
<td>6.3</td>
<td>0.31×10^{-2}</td>
</tr>
<tr>
<td>2</td>
<td>Test</td>
<td>26.8</td>
<td>74.8</td>
<td>9.5</td>
<td>0.84×10^{-2}</td>
</tr>
<tr>
<td></td>
<td>LightFrame3D</td>
<td>26.4</td>
<td>68.0</td>
<td>6.3</td>
<td>0.34×10^{-2}</td>
</tr>
</tbody>
</table>

The comparison between the numerical and experimental results indicates that the finite element program can accurately predict the load-displacement characteristics of the three-dimensional test buildings under static loading condition, as shown in Figure 9.4 for Test One and Figure 9.5 for Test Two. The difference between the predicted and the tested values in both cases was less than 4%. The predicted curves, including the initial stiffness, closely matched the curves from the two tests.
The program predicted the torsional motion of the test buildings in the same pattern as that shown in the tests. Figure 9.6 displays the roof deformation values in both the $x$ and $y$ directions obtained from the prediction to the building in Test One.
Figure 9.6 Plan view of the displaced position of the top of the test building

The resultant torsional motion is shown on the displaced diaphragm, illustrated by the dashed lines. The roof experienced a combination of translational and rotational deformations. The four walls slightly deformed out-of-plane because of the uneven out-of-plane and in-plane displacements as well as different drift values. In the $x$ direction, the north shear wall was subject to more severe deformation than the south shear wall due to the larger opening and accordingly, in the $y$ direction the east end wall moved towards the north while the west end wall moved towards the south. The curved deformations in both the transverse end walls indicate that the roof in the model was not completely rigid.

The comparisons of torsional motions of test buildings presented in Table 9.2 indicate that the predicted torsional motions in both buildings were considerably smaller. This consequence can be partially attributed to the connection model used in the frame element, which required four three-dimensional beam elements in a frame element to be rigidly connected. In an actual wood frame building, however, the frame members would usually be
connected with nails and when loaded, the frame members may have relative movement or even separation from each other. Hence the rigid connections in the model eliminated the relative movement between frame members and therefore reduced the overall deformation of the building. In the current version of the finite element program, this discrepancy was compensated for by adjusting the strength of frame members to achieve results closer to reality. In future improvements of the program, it is suggested that spring connection elements be implemented into the frame element to better reflect the actual frame construction. Another factor that contributed to the inaccurate prediction of torsional motion in the test building was that the roof in the simulation model was not completely rigid, whereas the roof in the actual test building was built with high rigidity. Thus the force distribution to the vertical members was not completely proportional to the wall stiffness.

Based on the above considerations, modifications were made in the simulated models. The stiffness of members representing the roof joists was increased to reflect high rigidity of the roof in test buildings. The stiffness of the vertical frame members at the four corners was reduced to provide a flexible frame structure. The simulations were carried out again. The results, as listed in Table 9.3, show improvements in the prediction.

<table>
<thead>
<tr>
<th>Test No.</th>
<th>Data Source</th>
<th>$\Delta_y$ (mm)</th>
<th>$\theta$ (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Test</td>
<td>11.3</td>
<td>$1.04 \times 10^{-2}$</td>
</tr>
<tr>
<td></td>
<td>LightFrame3D</td>
<td>10.0</td>
<td>$0.76 \times 10^{-2}$</td>
</tr>
<tr>
<td>2</td>
<td>Test</td>
<td>9.5</td>
<td>$0.84 \times 10^{-2}$</td>
</tr>
<tr>
<td></td>
<td>LightFrame3D</td>
<td>9.3</td>
<td>$0.61 \times 10^{-2}$</td>
</tr>
</tbody>
</table>

Table 9.3 Improved predictions in roof translational motion in $y$ direction and torsional motion
During the simulation procedures, it was thought that even if a better prediction to an individual test case could be achieved by selecting more carefully calibrated parameters, it is more important for a researcher to focus on building a robust analytical tool that not only can simulate some specified cases, but also can successfully predict structures with a wide range of combinations of material and loading conditions. In testing the finite element program's capacity, the behaviour of a building subject to a post-peak cyclic loading was predicted, as shown in Figure 9.7. This example demonstrates the advantage of the computer simulation over the testing method in terms of cost and time effectiveness. It also indicates that without performing an actual laboratory test, a structure can be investigated by using different parameters, as in a parameter study.

![Figure 9.7 Predicted post-peak cyclic load-displacement behaviour of building in Test One](with static back bone curves)

9.3. System effects

It is generally agreed that a structural component will behave differently depending upon whether it is used in a building system or loaded on its own. This is because the system
effects, including load sharing among components, deformation reassignment, and system eccentricity, will affect the actual forces and deformations in each individual building component. The degree of system effects depends mainly on the material, geometric, and structural layout of a building. For example, an asymmetric arrangement of openings may generate an uneven distribution of forces in walls, which will cause torsion in the building. The system effects, however, cannot be detected in two-dimensional component studies. To reveal the system effects, a preliminary study was performed. From the numerical results simulating the building in Test One, the forces applied to frame members at floor level are presented in Figure 9.8.

![Diagram of forces applied at floor level](image)

**Figure 9.8** Forces in frame members measured at floor level

Figure 9.8 shows that the south shear wall (no opening) carried most of the lateral load applied in the $x$ direction (loading direction) because of its high stiffness. Sharing the corner frame members resulted in notable lateral forces in the direction perpendicular to
loading direction (the $y$ direction) in the frame members of two end walls, even though no external lateral load was applied on them in that direction. In the finite element models, the two adjacent walls are connected by assuming that the end studs in the frames of two walls are sharing a pair of common nodes.

To study the influence of system effect on the deflections of shear walls, the two shear walls (the south and north shear walls) were loaded individually. The deflection results were then compared to those when the two shear walls were integrated into a three-dimensional building system, as the one in the tests. The results, given in Table 9.4, show that the deflections of a shear wall in a system or loaded individually are substantially different, which indicates that the appearance of transverse end walls and roof played roles in adjusting the deformations in shear walls. This phenomenon reveals an important fact that the system effect or load sharing mechanism can virtually redistribute the stiffness of components to prevent the weakest one from undergoing excessive deformation, which implies that the total deformation and load carrying capacity of a system are not simple addition of those possessed by individual structural components in the system. This may also explain the reason why some buildings with wide openings could survive in a severe earthquake ground motion.

Table 9.4 Comparison of the deflection of shear walls with and without system effect

<table>
<thead>
<tr>
<th>Deflection at $P_{\text{max}}$ (mm)</th>
<th>South shear wall (No opening)</th>
<th>North shear wall (Opening)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear walls in 3D building</td>
<td>80.0</td>
<td>87.6</td>
</tr>
<tr>
<td>Shear walls loaded individually</td>
<td>68.0</td>
<td>122.0</td>
</tr>
</tbody>
</table>

It is worth mentioning that the corner studs picked up higher vertical forces than did
the intermediate studs, as shown in Figure 9.8. A higher tensile force in the southeast corner stud than that in the northeast one corroborates the results given in Table 9.1 that the southeast corner stud had a larger uplift than the northeast one ($O_{C2}$ vs. $O_{C1}$). In the current version of the programs, it is not possible to quantitatively predict the uplift values. In order for the program to do so, spring elements containing the same load-deformation characteristics as the hold-downs have to be implemented in between the frame nodes and supports. This can be done when the further improvement of the programs is performed.

9.4. Failure modes and relative movement between panel and frame

As discussed earlier, the shear walls in a wood light-frame system are major load-carrying components. Under lateral loading, the overall performance of the shear wall mostly depends on qualities of the sheathing panels and the connections attaching the panels to framing members because the framing on its own has little lateral load carrying capacity. The panels typically behave as relatively stiff plates, whereas the sheathing-framing connections typically undergo highly nonlinear inelastic deformations. Surveys of previous experimental studies and damaged wood frame buildings demonstrate the importance of the nail connection between panels and frame members because the predominant failure mode almost always involved the above-mentioned connection.

In the present static tests, failure was also first observed starting from the nail connections holding the panels to the frame members. This nail connection failure was mostly located around the edges of sheathing panels. It can be seen that initially the nails bent and crushed into the frame members when relative movement between panel and frame occurred. Along with the panel rotating with respect to or separating from the frame, the nail gradually withdrew from the frame members (Figure 9.9 (a)). As the deformation
progressed, failure in the form of nail heads embedding into and pulling through the panels was observed (Figure 9.9 (b)). The tests indicated that the latter failure mode was more influential because the panel could no longer play any structural role once its connection to the frame was broken.

(a) Nail withdrawal from frame member  
(b) Nail pulling though the panel

Figure 9.9 Failure modes of nail connections

The finite element program predicted that the maximum nail forces would first occur along the bottom panel edges of the shear wall without opening (the south wall) in the early stage of loading and then progressed along the vertical panel edges of the same wall upwards. In the end of simulation, the maximum nail forces were mainly located in the top panel edges. This shift in the maximum nail forces implied a trend of the failure in nail connections from the bottom to the top of the shear wall, which was the same as observed in the tests.

Along with the increasing load and propagating nail failure, the sheathing panels gradually rotated relative to the frame and, at some locations, separated from the frame. The panels on the north shear wall (with opening) and the south shear wall (without opening) stopped in positions as shown in Figure 9.10 and Figure 9.11, respectively. As a comparison,
the panel positions predicted by LightFrame3D program are presented in Figure 9.12 for the north shear wall and in Figure 9.13 for the south shear wall, respectively. A good agreement was achieved between the tests and the simulations. Contrary to considerable damage observed in the two side shear walls, the panels on both end walls did not display any apparent damage or significant movement. It was because those walls underwent only slight deformation and most of it was recovered after the load removal.

Figure 9.10 Panel movements of the north shear wall

(a) Top (Panel perimeters are highlighted)
Figure 9.11 Panel movements of the south shear wall

Figure 9.12 Predicted panel positions on the north shear wall
9.5. Predictions of the static behaviour of shear walls

The numerical simulations were also applied to two-dimensional shear wall cases (Durham et al. 1997), as discussed in Section 8.4 in Chapter 8. The first test case was a 2.4 m × 2.4 m shear wall sheathed with regular size OSB panels, staggered horizontally. The nail spacing for this wall was 152 mm (6 in) on the edges and 305 mm (12 in) on intermediate studs. Figure 9.14 and Figure 9.15 show the prediction of the shear wall performance by LightFrame3D program under either static monotonic or cyclic loading, respectively. The other test case was a 2.4 m × 2.4 m shear wall sheathed with one jumbo size OSB panel. The nail spacing for this wall was 76 mm (3 in) on edges and 305 mm (12 in) on intermediate studs. Figure 9.16 and Figure 9.17, respectively, show the prediction of the shear wall performance by the LightFrame3D program under either static monotonic or cyclic loading. The predicted static monotonic and cyclic load-displacement characteristics of the two walls were in good agreement with the test results. The initial stiffness and
maximum load carrying capacities of tested shear walls were calculated with errors less than 6%. The hysteresis loops generated by LightFrame3D show an obvious pinching shape. The amount of calculated energy dissipation in the shear wall with regular panels (JTest 8) is close to that from tests, as listed in Table 9.5.

![Figure 9.14 Static test of shear wall with 4 by 8 regular panels](image)

**Figure 9.14** Static test of shear wall with 4 by 8 regular panels

![Figure 9.15 Cyclic test of shear wall with 4 by 8 regular panels](image)

**Figure 9.15** Cyclic test of shear wall with 4 by 8 regular panels
Table 9.5 Comparison of energy dissipation of shear walls under cyclic loading predicted by LightFrame3D program and measured from tests

<table>
<thead>
<tr>
<th>Shear Wall</th>
<th>Energy Dissipation (N-m)</th>
<th>Error (%)&lt;sup&gt;a&lt;/sup&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td>JTest 6 - 2.4x2.4 Jumbo Panel</td>
<td>3378</td>
<td>2630</td>
</tr>
<tr>
<td>JTest 8 - 1.2x2.4 Regular Panels</td>
<td>1081</td>
<td>1038</td>
</tr>
</tbody>
</table>

<sup>a</sup>: Error = \( \frac{\text{Ana. - Test}}{\text{Ana.}} \) x 100
The LightFrame3D was also tested on the case of the shear wall with one jumbo sheathing panel to predict its post-peak behaviour, as shown in Figure 9.18.

![Graph showing load-displacement relationship](image)

**Figure 9.18** Predicted post-peak cyclic load-displacement relationship of shear wall

The comparison of the numerical results with the experimental results described in this chapter demonstrate that the finite element model is capable of predicting the static load-deformation characteristics of two- and three-dimensional structures, the pinching and the energy dissipation behaviour of those structures under cyclic loading conditions, and their deformation patterns. The program is useful in doing a parameter study to investigate the system effects and influential factors on the structural performance of wood light-frame buildings and their components. In the next chapter, the comparison will continue in the dynamic domain.

**9.6. Further expansion of the program**

As discussed in the context, to improve the accuracy of the predictions made by the finite element program, some improvements should be considered in future expansions of the program. They are reiterated as follows:
(1) To better reflect the actual frame construction, the rigid connection of frame elements needs to be changed to a flexible connection, which can be done by implementing spring elements in the connections of frame elements;

(2) Spring elements representing the actual material properties of the hold-downs can be added to the frame nodes connecting the support foundation to predict the uplift in the structure under varied loading schemes.
As the continuation of the experimental-based verification of the finite element models, this chapter focuses on comparing the dynamic response of three-dimensional and two-dimensional structures obtained from experimental procedures and numerical analyses. In this chapter, first the test data are presented, followed by the predictions from the LightFrame3D dynamic finite element program, which are then compared to the test results. The natural frequencies and corresponding mode shapes, nonlinear dynamic characteristics, and the failure modes of test systems from both resources are subsequently discussed. Test Four is also described in this chapter in detail, even though the present program is not capable of providing predictions to it.

The raw data from the dynamic tests contained high frequency noise components generated from the electronic apparatus in the test control and signal generation systems. The noise had to be eliminated before the test data could be implemented into the analysis. This is usually accomplished using digital data processing techniques, in which the central step is the Fast Fourier Transform (FFT). The descriptions of the theory and methods in digital data processing can be found in a number of publications, including the books written by Bendat and Piersol (1986) and Newland (1986). In the present testing, an integral data processing routine in both time domain and frequency domain was applied. In the time
domain, the procedures involved were data standardization, data filtering, and data smoothing. In the frequency domain, in addition to the above procedures, auto- and cross-spectral density functions were used to obtain the natural frequencies of the structures.

10.1. Experimental results of three-dimensional buildings under dynamic excitation

The dynamic tests with the application of earthquake ground motion were performed on two three-dimensional simplified test buildings. The two buildings (Test Three and Test Four) were constructed in the same manner, but loaded differently. Test Three was excited unidirectionally (in the primary direction only), while Test Four was excited bi-directionally (in both primary and secondary directions). The basic system parameters, such as natural frequencies and mode shapes, were evaluated by conducting sine wave frequency sweep tests before the main tests were excited by simulated earthquake input.

10.1.1. Sine wave frequency sweep test and system fundamental natural frequency

The sine wave sweeping is a type of forced harmonic vibrations. The purpose of performing the sine wave frequency sweep test is to identify the system’s natural frequencies, usually a few of the lowest ones, and corresponding mode shapes of a structure. The theory of forced harmonic vibration provides the basis of determining the natural frequencies of a structure from its response to input vibration signals. In general, the signals contain a selected range of forcing frequencies, which may cover the possible target natural frequencies of the test structure and, for that reason, are progressively increasing during the test. When the forcing frequency sweeps from a lower value to a higher value and is equal to the fundamental frequency of the system, the forcing frequency is defined as resonance frequency and the state is defined as the resonance at which the system presents the largest response amplitude. By measuring this amplitude from the recorded signals, the natural
frequency of the system can be determined and the associated damping can be subsequently calculated. Theoretically, the resonance frequency is slightly different for displacement, velocity, and acceleration, which, for a damped system, can be described as:

Displacement resonance frequency = \( \omega \sqrt{1 - 2\zeta^2} \)

Velocity resonance frequency = \( \omega \)

Acceleration resonance frequency = \( \omega / \sqrt{1 - 2\zeta^2} \)

The differences among the three resonance frequencies and the natural frequency are negligible because the damping in a typical wood structure is well below the 20%, at which a maximum 4% error is introduced. In the present tests, all system natural frequencies were determined from the acceleration signals. The obtained results would be slightly larger than the true natural frequency, but the error was insignificant.

The sine wave sweep tests performed prior to the main earthquake tests in Tests Three and Four included three forcing frequency ranges to cover the three possible lowest natural frequencies and mode shapes, representing the two translational motions and the one rotational motion. With the help of the LightFrame_Eigen program, the three selected ranges were 0.5 Hz – 6.0 Hz, 4.0 Hz – 10.0 Hz, and 7.0 Hz – 15.0 Hz. During the tests, the sweeps were at constant displacement with amplitude of 0.5 mm. In determining the natural frequencies, FFT analysis was applied to the acceleration signals taken from four accelerometers positioned at three corners and at the center of the top of the test buildings. In each range, the resultant frequency values from the FFT analysis were averaged to provide the final natural frequency. The differences among all the measurements were minor. In determining the mode shapes, the phases and amplitudes of the acceleration signals obtained from different locations were compared. In the first sine sweep frequency range (0.5 Hz –
6.0 Hz), the translational motions measured at the two corners (E2 and E3) on the roof had the same phase in the primary direction (Figure 10.1). The amplitude in the primary direction was significantly higher than that in the secondary direction (E1) (Figure 10.2). The rotation of the roof was not significant when comparing the phases of signals obtained from the two corners on the roof (E3 and E4) as shown in Figure 10.3. These results demonstrated that the mode shape under the first sine sweep excitation was the first mode shape corresponding to the first natural frequency of the test structure. In the second sine sweep frequency range (4.0 Hz – 10.0 Hz), the acceleration waves measured at E3 and E4 in the secondary direction were in the same phase (Figure 10.4). The amplitude in the secondary direction measured at the center of the roof was much higher than that in the primary direction measured at the same location, as shown in Figure 10.5. It was the same as in the first frequency range that the rotation of the roof was not significant when comparing the signals measured at the two opposite corners (Figure 10.6). From these results, it is concluded that the structure was undergoing the translational motion in the secondary direction with the second mode shape corresponding to the second natural frequency. In the third sine sweep frequency range, the acceleration signals measured at the opposite corners (E2 vs. E3 and E3 vs. E4) in both directions were in the reverse phases (Figure 10.7 and Figure 10.8). This phenomenon indicated that the structure was subject to a rotational motion (The third mode shape) corresponding to the third natural frequency.
Figure 10.1 Amplitudes in the primary direction in the first frequency range

Figure 10.2 Amplitudes in two directions in the first frequency range

Figure 10.3 Comparison of the phases of two signals in the first frequency range
Figure 10.4 Amplitudes in the secondary direction in the second frequency range

Figure 10.5 Amplitudes in two directions in the second frequency range

Figure 10.6 Comparison of the phases of two signals in the second frequency range
The natural frequencies and matching mode shapes for two tests are summarized in Table 10.1. Three natural frequencies of Test Building Three are combined in Figure 10.9, and those for Test Building Four are in Figure 10.10. These results show that the fundamental natural frequencies are not well separated. During the data processing, this incidence was observed. The building in Test Four provided lower natural frequency values than those in Test Three. The reason for this may be due to the use of a retrofitted test building in Test Four, in which the sheathing on all four walls from Test Three was replaced with new panels but the original frame members were reused.
Table 10.1 Natural frequencies obtained from sine wave sweep tests

<table>
<thead>
<tr>
<th>Mode Shape</th>
<th>Test Three (f_3, \text{ Hz})</th>
<th>Test Four (f_4, \text{ Hz})</th>
<th>(\Delta=(f_3-f_4)/f_3) (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>First (Translation in (x))</td>
<td>2.9</td>
<td>2.5</td>
<td>13.8</td>
</tr>
<tr>
<td>Second (Translation in (y))</td>
<td>5.1</td>
<td>3.8</td>
<td>25.5</td>
</tr>
<tr>
<td>Third (Rotation about (z))</td>
<td>8.8</td>
<td>8.2</td>
<td>6.8</td>
</tr>
</tbody>
</table>

![Figure 10.9 Three natural frequencies of building in Test Three](image1)

Figure 10.9 Three natural frequencies of building in Test Three

![Figure 10.10 Three natural frequencies of building in Test Four](image2)

Figure 10.10 Three natural frequencies of building in Test Four

10.1.2. Input acceleration signals

In the two dynamic tests, the earthquake waves in The 1992 Landers, California, recorded at the Joshua Tree Station were used as the input excitations to the structures. In
Test Three, the building was excited unidirectionally in the long span with the east-west acceleration component of the earthquake, and the peak acceleration of the record was scaled up to a target value of 0.35g. In Test Four, the building was excited bi-directionally in both the long and short spans. The east-west acceleration component of the earthquake was applied to the long span, and the north-south component was applied to the short span. The target peak acceleration was 0.29g, using the original records without scaling. The reason for using lowered acceleration amplitude in this test was because in Test Three, the building was subjected to very severe damages in the first pulse of the excitation and lost most of its lateral load carrying capacity before the second pulse. Therefore, the purpose of investigating the influence of two pulses on the structural behaviour was not accomplished.

In the tests, the actual peak accelerations of input signals were slightly different from the target levels, due to accuracy limits in the signal generation apparatus and the electrical noise introduced in the testing system. To examine if the shake table can accurately reproduce the input ground motion without losing acceleration frequency components, a 5% damped acceleration response spectrum analysis was conducted. From the 5% damped pseudo-acceleration response spectra (Figure 10.11), it is found that the frequency components in the input acceleration signal and the shake table feedback in the entire range were basically the same.
Figure 10.11 Comparison of the 5% damped acceleration response spectra

The input signals measured on the shake table for the two tests are presented in Figure 10.12 and Figure 10.13, respectively.
10.1.3. Basic parameters measured during the tests

The dynamic tests basically provide two types of data: displacements and
accelerations. These data are recorded as absolute values. In order to extract the building characteristics of deformations, forces, and failure modes, appropriate processing and analysis of the saved data sets are necessary. The building deformations are described by three degrees of freedom defined at the center of mass of the roof. They are the two lateral displacements in the \( x \) and \( y \) directions and the one torsional rotation about the vertical axis. The actual lateral building deformation (relative displacement or drift) is defined as the difference between the total displacement of the mass of the roof and the displacement of the ground (shake table). The actual (relative) mass acceleration comes from the same concept as that for the relative displacement. To define the forces applied to test buildings, the general equations of motion presented in Chapter 6 are described again:

\[
ma_{r} + ca + f_{s}(a, \dot{a}) - R = 0 \tag{10.1}
\]

Because

\[
\ddot{a}_{r} = \ddot{a} + \ddot{a}_{g} \tag{10.2}
\]

By changing the order, the general equations of motion (10.1) can be presented as

\[
ma + ca + f_{s}(a, \dot{a}) = R - ma_{g} = R - P_{\text{eff}} \tag{10.3}
\]

The last term \( P_{\text{eff}} \) (\( R \) are dead loads) on the right-hand side of Equation (10.3) represent the external lateral forces applied to building induced from the earthquake, which are equal to mass multiplied by the ground acceleration and acts opposite to the acceleration. These forces are called the effective earthquake forces. The usefulness of this definition is that the base shear, an essential parameter required in design procedures, can be calculated from the effective earthquake forces. When only one storey is present, the effective earthquake force equals the base shear. Based on this definition, when the same earthquake input accelerogram and mass are applied to the test building and numerical model, the effective
earthquake forces are identical. Note that the effective earthquake forces are distinct from the inertia forces. The total inertia forces $F_i$ consist of the effective earthquake forces and the forces induced due to the relative mass acceleration, which can be seen from the following equation:

$$ F_i = m \ddot{a}_r = m(\ddot{a} + \ddot{a}_g) = m\ddot{a} + P_{\text{eff}} $$

(10.4)

The basic characteristics of test buildings are given in Table 10.2.

Table 10.2 Summary of dynamic test results of three-dimensional buildings

<table>
<thead>
<tr>
<th></th>
<th>Test Three</th>
<th>Test Four</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_{\text{max}}$ (kN)</td>
<td>31.3/-27.8</td>
<td>22.5/-21.7 (p) 25.5/-21.2 (s)</td>
</tr>
<tr>
<td>$\Delta_x$ (mm)</td>
<td>98.6/-95.1</td>
<td>130.2/-77.4</td>
</tr>
<tr>
<td>$\Delta_y$ (mm)</td>
<td>29.8/-19.0</td>
<td>54.9/-77.2</td>
</tr>
<tr>
<td>$\theta$ (rad)</td>
<td>$1.03 \times 10^{-2}$</td>
<td>$4.39 \times 10^{-2}$</td>
</tr>
<tr>
<td>$O_{C1}$ (mm)</td>
<td>0.7</td>
<td>5.6</td>
</tr>
<tr>
<td>$O_{C2}$ (mm)</td>
<td>0.7</td>
<td>3.0</td>
</tr>
<tr>
<td>$O_{C3}$ (mm)</td>
<td>4.0</td>
<td>2.2</td>
</tr>
<tr>
<td>$O_{C4}$ (mm)</td>
<td>1.6</td>
<td>7.1</td>
</tr>
</tbody>
</table>

where $P_{\text{max}} = \text{maximum effective earthquake force applied to test building, kN}$. In Test Four, this force was evaluated in the primary (p) and secondary (s) directions.

$\Delta_x = \text{maximum displacement of test building in x direction, mm}$.

$\Delta_y = \text{maximum displacement of test building in y direction, mm}$.

$\theta = \text{maximum torsional rotation of the roof of test building about the vertical axis, which passes through the center of mass of the roof diaphragm, rad}$.
$O_{c1, c2, c3, c4} = \text{uplift displacements measured at the corners C1 to C4, respectively, mm (see Figure 8.1). They were located, respectively, at the northeast corner (C1), the southeast corner (C2), the southwest corner (C3), and the northwest corner (C4) of the building.}$

Inertia mass applied to the two tests was 7,250 kg.

The data in Table 10.2 reveal that, as expected in Test Four, adding an excitation in the short span activated strong rotational movement of test building, which was not observed in Test Three. In addition, the drifts in both directions were greater than those in the building of Test Three. This result suggests that the damage could be much more destructive when an eccentric building is struck by the interaction of ground excitations from multiple directions. The discrepancies in the effective earthquake forces between two tests and within Test Four were due to different acceleration amplitudes of the input earthquake signals. Uplift displacements in Test Three were smaller than those in Test Four, and furthermore, uplift displacements in both dynamic tests were smaller than those in the static tests.

With the help of the time series plots of the test data, the behaviour of test buildings can be better understood. Figure 10.14 presents the control input and the responses of building in Test Three, including the shake table acceleration (refer to Figure 10.12), mass acceleration, relative mass acceleration, shake table displacement, roof displacement (absolute), and roof relative displacement (drift). The same contents for Test Four in the primary direction are given in Figure 10.15 and those in the secondary direction are given in Figure 10.16.

By examining the curves in Figure 10.14, it is logical to conclude that the building in Test Three responded to the second high amplitude segment with insignificant mass
acceleration. The roof no longer followed the table displacement after the first pulse; instead, it moved slowly with smaller amplitude. All of these indicate that the building was severely damaged by the first strong shaking. Its structure was very much softened and therefore lost most of its lateral load carrying capacity. This interpolation is validated by the observation of Test Three. As the testing was to provide a database for the purpose of verifying the developed dynamic finite element models, the premature failure of the test building was not expected. By applying smaller excitation magnitude in Test Four, when comparing the mass accelerations from two tests, it can be seen that although the building experienced some damages, it survived the first pulse and continuously supported the force induced by the second strong shaking. It was also observed that the stiffness of the building decreased considerably after the first shaking, as indicated by lower mass acceleration frequencies and larger building drifts shown in Figure 10.15 and Figure 10.16. By comparing the deformation of the test building to the displacements of the shake table, it is found that in some ranges, the building underwent larger movements than the input table movements. The substantially increased displacements may imply that the vibration of the building was, at those moments, very close to the resonant frequencies of the input earthquake excitation and, therefore, the deformation was significantly amplified.
Figure 10.14 Response-time history of Test Three

Chapter 10 Experimental-Based Verification of the Dynamic Finite Element Models
Figure 10.15 Response-time history of Test Four in the primary direction
Figure 10.16 Response-time history of Test Four in the secondary direction
10.2. Prediction of the dynamic behaviour of three-dimensional buildings

Because of the limitation of the LightFrame3D programs that the input excitation history can consider one horizontal and one vertical earthquake components, the prediction was only applicable to the building in Test Three exposed to a unidirectional ground motion. The data from Test Four is reserved for future verifications once the program is further expanded.

10.2.1. Natural frequencies and mode shapes

The predicted three fundamental frequencies using the LightFrame3D_Eigen program are listed in Table 10.3, compared to the fundamental frequencies obtained from the sine wave sweep test. Good agreements were achieved. The corresponding mode shapes drawn from the eigenvectors given by the program are presented in Figure 10.17. These mode shapes can also be compared to the results obtained from the sine wave sweep tests as discussed in Section 10.1.1.

Table 10.3 Comparison of natural frequencies obtained from Test Three and predicted by LightFrame3D program (Hz)

<table>
<thead>
<tr>
<th>Data Source</th>
<th>First</th>
<th>Second</th>
<th>Third</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test Three</td>
<td>2.9</td>
<td>5.1</td>
<td>8.8</td>
</tr>
<tr>
<td>LightFrame3D</td>
<td>2.8</td>
<td>6.1</td>
<td>8.8</td>
</tr>
</tbody>
</table>
MODE 1 ($f_1 = 2.8\text{Hz}$)

MODE 2 ($f_2 = 6.1\text{Hz}$)

MODE 3 ($f_3 = 8.8\text{Hz}$)

Figure 10.17 Predicted three mode shapes
10.2.2. Comparison of predicted building behaviour and results from Test Three

Table 10.4 gives the peak values of drifts and accelerations during the first and second large amplitude events from the dynamic tests and the analytical prediction. The torsional motions and the occurrence times from experimental and analytical data are also compared. Similar to the static predictions (see Section 9.2 in Chapter 9), the numerical results from five points on the roof of building were used to calculate drift, relative mass acceleration, and torsional motions. Among these points, three (one at the center of the roof and two at the north and south corners) provided data of the drift and relative acceleration of the building in the $x$ direction, and two (at the east and west corners) provided data in the $y$ direction. The difference in the displacements measured at two opposite points (e.g. the north and south corners) was calculated and then divided by the span of the roof to obtain the angle of roof rotation. The rotational results in both $x$ and $y$ directions were similar, but not identical due to slight changes in the roof geometric shape, because the roof was not perfectly rigid. Figure 10.18 compares the drifts and relative mass accelerations of the building, measured at the center position from the test and the simulation. It can be seen that the numerically predicted response (drift and mass acceleration) follows the trends of the test curves very well with similar cyclic frequency and vibration amplitude. The relative mass accelerations predicted by the program and measured in the tests were processed by FFT to obtain the acceleration response spectra and to compare the frequency components, as shown in Figure 10.19. The measured and predicted frequencies corresponding to the maximum response were 1.37 Hz and 1.39 Hz, respectively. They were very close to the frequency of the input earthquake excitation (1.35 Hz, see Chapter 8). These results demonstrate that the LightFrame3D program can predict the dynamic response of a building subjected to
earthquake ground motion with reasonable accuracy.

Table 10.4 Comparison of dynamic test results and numerical predictions in Test Three

<table>
<thead>
<tr>
<th>Amplitude event</th>
<th>Test Drift (mm)</th>
<th>LightFrame3D Drift (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1st large</td>
<td>98.6/-95.1</td>
<td>77.6/-70.3</td>
</tr>
<tr>
<td>Amplitude event</td>
<td>0.43/-0.45</td>
<td>0.50/-0.48</td>
</tr>
<tr>
<td>2nd large</td>
<td>90.1/-65.1</td>
<td>95.1/-71.7</td>
</tr>
<tr>
<td>Amplitude event</td>
<td>0.36/-0.41</td>
<td>0.47/-0.47</td>
</tr>
<tr>
<td>θ (rad)</td>
<td>1.03×10⁻²</td>
<td>0.84×10⁻²</td>
</tr>
<tr>
<td>Occurrence of θ (sec)</td>
<td>17.11</td>
<td>16.60</td>
</tr>
</tbody>
</table>

Figure 10.18 Comparison of drift and acceleration in Test Three
Figure 10.19 Comparison of acceleration frequency range

Figure 10.20 compares the top drift of all four walls in Test Three to help understand the deformations in the buildings. Generally speaking, the graphs show that the numerically predicted deformations in four walls match the test curves very well. When closely examining the period between the 15th second and the 21st second of the first pulse (Figure 10.20 (a) and (b)), however, it is found that the predicted drifts in the north and south shear walls were smaller than those obtained from the test. The difference in the north shear wall with the opening was more significant than that in the south shear wall. The reasons are believed to be the same as those discussed in the proceeding chapter. The predicted drifts in the north and south shear walls also led a smaller maximum rotational angle, $\theta$, when compared to the value from the test, because both of them occurred within the above-mentioned period. The reason for this discrepancy is likely that in this time period, the severe strike of the first major component of the earthquake to the building has already caused the failure of the test building, which was clearly observed in the test. This outcome exceeded what had been planned for the model verification purpose. Usually, under such an
extreme condition, finite element programs may be difficult to reach a convergence. The newly developed program overcame this difficulty successfully and ran through all designated time steps, owing to the self-adaptive procedures implemented in the program. However, the program may not accurately predict the actual building performance under this situation. It is expected, by reviewing other time periods in the prediction, that the program could provide better results if the input ground acceleration were not so severe. From the curves presenting the deformations of two transverse end walls, it can be seen that these two walls vibrated with higher frequency than the two shear walls. This was an indication that the movements of the two end walls were excited by the building’s second and third natural frequency and their modal responses therefore contributed mainly to the translation in the short span and the rotational motion of the test building.

(a) North shear wall

(b) South shear wall
10.3. Failure modes

Under the seismic loading, similar to the static loading, the observed predominant failure in test buildings first occurred in the connections between the panels and framing members. The local deformation zone of a nail connection under the significant seismic (cyclic in nature) loading, however, showed a different pattern from that under a much slower static monotonic loading. Under the earthquake excitation, the nail connectors bent repeatedly in reversed directions and crushed the surrounding wood medium in different locations. While the nail hole in the framing members was enlarged, a gap between the nail shank and the wood fiber was formed and propagated, and nail withdrawal from wood members occurred. This nail deformation impaired the energy dissipation ability of the system due to the pinching of the hysteresis loops and caused degradation of the stiffness and
load carrying capacity of the system. At some locations, mostly along the edges of panels, the nails pulled through the panels. A combination of both nail actions eventually led to the systematic failure of the entire structure.

The observations from Test Three, under the unidirectional seismic loading, indicated that the building experienced similar damage to the building in the static Test Two. The failures were mostly presented in the two shear walls (north and south walls) in the forms of nail withdrawal from framing and nail pulling through sheathing (Figure 10.21). Although relative movements between the panels and frame members were observed in the two transverse walls, permanent structural deformations were insignificant and nail failures did not occur (Figure 10.22). Whether the failure first originated in the north shear wall with opening or the south shear wall, however, was inconclusive from the test results owing to the rapid excitation.

Figure 10.21 Nail withdrawal and pulling through failure modes
In Test Four, under the bi-directional seismic loading, the four walls displayed great differences in their deformation. From the testing curves as shown in Figure 10.23, the north shear wall experienced greater drift than the south shear wall because of its opening. It was observed that the north shear wall failed first with the separation of the panel from the framing in the narrow wall strip. In contrast to the previous three tests, the previous transverse end walls (east and west) became shear walls in this test because of the introduction of excitation in the $y$ direction. It was observed that the east wall underwent a dramatic deformation (Figure 10.23), with severely damaged the panel to frame connections, as presented in Figure 10.24. In this wall it was also observed that two nails, at the mid height vertical edge of panel, failed as a result of low cycle fatigue (Figure 10.25). This was not considered significant enough to warrant a general conclusion that nail fracture is a contributing factor to seismic failure of woodframe constructions.
Figure 10.23 Comparison of drifts of four walls in Test Four
Figure 10.24 Damages in the east end wall

Figure 10.25 Two nails in fracture mode
10.4. Predictions of the dynamic behaviour of shear walls

The LightFrame3D dynamic finite element program was also used to predict the responses of full-scale (2.4 m × 2.4 m) wood shear walls under simulated earthquake excitations. The predictions were compared to the test results, obtained in the previous series of experimental studies (Durham et al. 1997) as described in the proceeding Chapters 8 and 9. The major differences in the configurations of two shear walls considered in the present dynamic finite element analysis were panel dimensions and nail spacing. One wall (Jtest1) used conventional sheathing layout with one 1.2 x 2.4 m and two 1.2 x 1.2 m OSB panels staggered horizontally. The nail spacing on this wall was 150 mm on the panel edges and 300 mm on interior studs. The other wall (Jtest10a) was sheathed with one 2.4 x 2.4 m jumbo panel nailed to the frame with nail spacing of 75 mm on panel edges and 300 mm on interior studs. In this way, the numbers of nails used in the two walls were approximately the same. The inertia mass on the top of support frame was 4,500 kg. The east-west component, recorded in the Joshua Tree Station during the Landers earthquake of June 28, 1992 in California (Magnitude=7.3; peak acceleration=0.29g), was used as the control earthquake input, which was the same as that used in the three-dimensional building tests. In the tests, the acceleration was scaled 20% higher, to 0.35g, to achieve enough inertia force to produce a significant response in the test specimens.

The natural frequencies of test shear walls, calculated by the LightFrame3D program, were compared to the results obtained from tests, as shown in Table 10.5. The calculated values were slightly lower than the test values. Besides the possible differences in modeling parameters, errors can be attributed to the supporting inertia frame of shake table setup, which provided extra stiffness to the whole system in the tests.
Table 10.5 Comparison of natural frequencies of shear walls predicted by LightFrame3D program and measured from experiments

<table>
<thead>
<tr>
<th>Shear Wall</th>
<th>Natural frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Test</td>
</tr>
<tr>
<td>Jtest 11 – 1.2 x 2.4 Regular Panels</td>
<td>3.3</td>
</tr>
<tr>
<td>Jtest 10a – 2.4 x 2.4 Jumbo Panel</td>
<td>4.5</td>
</tr>
</tbody>
</table>

Table 10.6 gives the peak values of drifts and accelerations in the first and second large amplitude shakes from the predictions and the tests. Figure 10.26 and Figure 10.27 display the predicted structural responses for both shear walls by the program, compared to the test results. It is observed that the predicted drifts and relative mass accelerations follow the test curves very well, with similar cyclic frequencies and vibration amplitudes.

Table 10.6 Comparison of maximum drifts and accelerations of shear walls predicted by LightFrame3D program and measured from experiments

<table>
<thead>
<tr>
<th></th>
<th>Jtest11</th>
<th>Jtest10a</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Test</td>
<td>FEM</td>
</tr>
<tr>
<td>1st large Amplitude event</td>
<td>37.3/-32.6</td>
<td>26.4/-52.4</td>
</tr>
<tr>
<td>Displ. (mm)</td>
<td>0.42/-0.37</td>
<td>0.43/-0.36</td>
</tr>
<tr>
<td>2nd large Amplitude event</td>
<td>51.7/-60.0</td>
<td>43.4/-61.9</td>
</tr>
<tr>
<td>Displ. (mm)</td>
<td>0.28/-0.35</td>
<td>0.52/-0.37</td>
</tr>
</tbody>
</table>
Figure 10.26 Dynamic responses of shear wall with 4 by 8 regular panels

Figure 10.27 Dynamic responses of shear wall with one jumbo panel

The accelerograms on top of shear walls predicted by the program and measured in the tests were then processed by FFT to obtain the Fourier spectra and to compare the frequency components, as shown in Figure 10.28 (a) and (b). For shear wall Jtest11, the maximum normalized amplitudes for both test and prediction have the same frequency of
1.34 Hz, whereas for shear wall Jtest10a, they are 1.44 Hz and 1.41 Hz, respectively. They are all very close to the frequency of input earthquake excitation (1.35 Hz, see Chapter 8). The FFT analysis indicated that the program could accurately predict the behaviour of test specimens.

Figure 10.28 Comparison of Fourier spectra of shear walls predicted by LightFrame3D program and from tests
10.5. Further expansion of the program

In numerical dynamic analysis, it is generally believed that the simulation of a nonlinear wood light-frame system with a larger number of degrees of freedom is computationally demanding (Folz et al. 2001). The dynamic finite element program developed in this study also faces the same problem. The solution for this problem depends on the developments in two areas. The first area is in computer technology and the second is in numerical solution method. With the rapid development in computer technology, the maximum CPU speed has been increased rapidly in recent years. Memory communication bandwidths are much broader and storage capacities are increasing as well. For example, the time spent on a dynamic analysis by using a P4 1.7 GHz computer could be 4.25 times shorter than the time of running the same problem in a PIII 500 MHz computer. In solution method area, the so-called “parallel processing” technique can be adopted to effectively reduce computational time. The fundamental concept of parallel processing is that a program is decomposed so that different data or instructions can be distributed among multiple processors, allowing simultaneous execution of a program. This technique can be applied at matrix level or within a do-loop, such as an element do-loop or Gauss point do-loop (including nail connections). Several commonly used mechanisms in parallel processing are standardized and are available in computer workstations and clusters. To perform parallel processing, the program has to be modified to comply with parallel processing requirements.

It is evident that in an actual three-dimensional wood structure, only a few major components are subject to significant and permanent deformations, while in others, the deformation is insignificant and recoverable. This provides the possibility to save computational time by approximating non-critical components, such as transverse walls,
floors and roofs, as if they behave linearly. In addition, combining a few smaller elements into a larger element by redistributing the nail connections to form an equivalent system can also reduce the number of nodes and elements and, therefore, the computational time. This method may be more appropriate for non-critical components than for major load carrying components. The disadvantages of using these two methods in the simplified structural model are that the solution may not have the required accuracy level and that local deformation details may be overlooked.

It is believed that some other possible improvements are also important:

1. In practice, an actual building might be subjected to multiple earthquakes and the damage might be accumulative over time. To account for this phenomenon, there are two options. If all the earthquake records are known before the simulation, these records can be formed into a long record. In case the record that needs to be used in the simulation is not handy, the program should be modified to be capable of saving data from one case to another case to satisfy the requirement of multiple earthquake simulations;

2. To improve program prediction accuracy and to better reflect the actual behaviour of a modeled wood-frame structural system, the failure models of frame member, panel, and especially the nail connection should be investigated and implemented in the program gradually; and

3. In preprocessing stage, preparing an input data file for a large problem to be modeled in the finite element analysis can be very tedious and prone to errors. To increase the data input accuracy and efficiency, a sophisticated and user-friendly graphic interface in the preprocessor is essential.
CONCLUSIONS

This thesis describes in detail the development of three-dimensional nonlinear static and dynamic finite element models for the study of wood light-frame structures, and addresses the issues related to the behaviour of wood light-frame buildings subjected to external loading. The experimental investigations of simplified three-dimensional buildings were integrated into the thesis work. The accomplishments are as follows:

1. Two three-dimensional nonlinear finite element models were developed. The programs with the models implemented are capable of predicting the structural behaviour of general wood light-frame buildings and their components under either static monotonic and cyclic loading or seismic ground motion. A three-dimensional coordinate transformation system is introduced into the programs to make the analysis of entire building possible. “Substructuring” technique increases the effectiveness of the coordinate transformation procedures and displays the force and deformation fields of panel elements in a conventional way. The displacement history as one of the input options is developed in the programs. This function allows the analysis to follow entire deformation path to trace post-peak behaviour of a structure in a static monotonic analysis and to study the energy dissipation and stiffness and strength degradation characteristics of a structure in a static cyclic analysis. The programs were also equipped
with several self-adaptive procedures to handle the difficulties in solution and iteration, which ensure the programs to be robust and reliable;

(2) The uniqueness of the programs is the implementation of a mechanics based representation of the load-deformation characteristics of individual panel-to-frame nail connections in the diaphragm system. This approach believes that the hysteresis behaviour is the structural response controlled by nonlinear characteristics of nail connector and the surrounding wood medium. By using finite element approach, the model simulates the nail connector as elasto-plastic beams acting on a nonlinear foundation. Differently from other connection models, this model is not a test-dependent curve-fitting model. It requires only the basic material properties of the connectors and the embedding characteristics of the surrounding wood medium. This model represents more accurately the actual hysteresis behaviour of connections and therefore the programs with this model implemented can provide better predictions on the structural behaviour of woodframe systems;

(3) One three-dimensional finite element program was developed to predict the fundamental frequencies and mode shapes of wood structural system, and also provide the damping coefficient required in the dynamic analysis;

(4) The algorithms of programs were carefully examined and tested step by step to ensure the fundamental correctness. The models were thoroughly verified by means of mechanics theories. This verification was carried out mainly in linear elastic range. All of the results testing the beam deflections and stresses given by the models agreed to the theoretical results with an error margin less than 5%, which tended to decrease with increasing the number of elements;
(5) Four tests on the full-scale simplified three-dimensional light-frame buildings were conducted under both static and dynamic loading conditions. The tests provided an excellent opportunity to observe and investigate the structural performance, system effects, and failure modes in three-dimensional buildings. They also provided much information for performing the experimental based model verification; and

(6) The models, combined with the experimental results, were applied to address the issues in system effects, such as load sharing among the structural components, torsional motion, and failure modes. The results suggest that the models can well display the deformation patterns of the system under investigation, but the accuracy needs to be further improved.

In the study of structural behaviour of three-dimensional wood light-frame buildings using the newly developed finite element programs and experimental procedures, it was observed that

(1) Compared to the static experimental results obtained from the three-dimensional building tests and the previous two-dimensional shear wall tests, the maximum load carrying capacities and the deflections at the ultimate load predicted by the static finite element program showed differences less than 6% and 10%, respectively. The finite element program calculated the test buildings with the same deformation pattern as shown in tests; however, the predicted torsional rotations of test buildings were smaller than those in tests. Under the cyclic loading, the finite element program well predicted the pinch shaped hysteresis loops, energy dissipations, and strength and stiffness degradations in shear wall systems. The program also successfully performed predictions of post-peak cyclic behaviour of both three-dimensional building and two-dimensional shear wall;
(2) Compared to the dynamic experimental results obtained from the three-dimensional building test and the previous two-dimensional shear wall tests, the dynamic finite element program provided satisfactory accuracy in the results of drifts, relative mass accelerations, acceleration response spectra, and torsional motions (in 3D cases) of test systems. The numerically predicted curves followed the test curves very well with similar cyclical frequencies and amplitudes. It was also demonstrated that the finite element program can calculate the eigenvalues for the natural frequencies and the eigenvectors for the mode shapes of the test system, excellently matching the test results;

(3) In the preliminary study of system effects, the results obtained from the finite element programs suggested that the shear wall with higher stiffness carried most of the lateral load applied to the building, and the transverse walls perpendicular to the load direction shared notable lateral forces even though no external load was applied to them directly. It was also found that the load sharing mechanism can redistribute the stiffness of components, and therefore the total deformation and load carrying capacity of a system are not simple addition of those possessed by the individual components in the system; and

(4) The finite element programs effectively predicted the movements of sheathing and framing of a test building. The path of nail failure and the maximum forces in nail connectors were also reasonably calculated. Due to the limitations of the program, however, the detailed nail failure modes in nail withdrawal from the framing and nail pulling through the sheathing panel cannot be displayed.

In addition, the following can be concluded for the experimental study:

(1) Under statically applied monotonic load, the buildings with or without dead load showed
similar load carrying capacity. The ability to sustain load capacity after the ultimate load level in the building with dead load was less than that in a building without dead load, which may be explained by the P-Δ effect caused by the dead load on the top of building; and

(2) In Test Four where the test building was subject to bi-directional seismic excitations, the building experienced substantially greater torsional motion with larger drifts in both directions than the building subjected to only unidirectional seismic loading. This implies that when a building is subjected to bi- or multi-direction ground motions, the effect could be cumulative and the damage could be much more severe.

The thesis also discusses the necessary and possible improvements to be implemented into the newly developed finite element programs. The improvements are mainly focused on how to promote the efficiency of the programs by saving significant computational time and how to enhance the ability of the programs to predict the detailed localized deformations with higher accuracy.

The preliminary application of the static and dynamic nonlinear finite element programs shows that they can predict the behaviour of wood light-frame structures with good accuracy and, more importantly, they can suit a wide range of structural, material and loading variations. The programs are expected to be useful either in analysis of an individual structure or in construction of response surfaces for reliability assessment of general wood structures. Such analyses can be used to develop design rules for wood frame systems. It is also believed that the programs will be useful tools in performing the parametric study of three-dimensional buildings to provide better understanding of system effects which influence the overall structural behaviour of wood light-frame systems.


Bibliography


Bibliography

231


Bibliography
APPENDIX A

PROGRAM USER'S MANUAL

A.1. Introduction

This user's manual provides the relevant background information for the LightFrame3D nonlinear finite element programs. The programs are designed to analyze three-dimensional wood light-frame constructions and their components or subassemblies, under both static monotonic and cyclic loading (LightFrame3D_Static) and earthquake excitation (LightFrame3D_Dynamic). Eigenvalues and corresponding eigenvectors can be calculated by using LightFrame3D_Eigensolver, from which the natural frequencies and mode shapes of the system can be derived. These programs address the common light-frame building configurations and loading conditions in a versatile manner.

A.2. Modeling

A basic structural model in the program contains an exterior panel, frames, an optional interior panel, and nail connections. The panels are connected to four frame members by nails in a sandwich pattern. The case to be analyzed is meshed into a number of four-side rectangular element units, which has four nodes with six nodal degrees of freedom and one element degree of freedom. The nodal degrees of freedom represent three translations in x, y and z directions (u, v, w), and three rotations about x, y and z axes (Rot-x, Rot-y, Rot-z). The element degree of freedom is one twist in x-y plane (w_{xy}). Connection
elements have three degrees of freedom at each node \((u, v, w)\). Four-point Gauss quadrature is used in the programs. The nodes on an element should be numbered in a counterclockwise direction. The pattern of numbering the entire mesh of a structure, however, should be carefully decided, because it decides the bandwidth of the stiffness matrix in the problem to be studied. Bandwidth will tremendously affect the size of the problem and hence the computational time. A smaller bandwidth is desired.

The optional interior panels may have different material properties from those of the exterior panels. When an interior panel is present, the space between the exterior and interior panels may be filled with insulation material (EPS). The insulation is assumed to provide support to the sheathing panels when the buckling of the panels produces compression in the insulation. Within an element, the four frame members may have different material properties and dimensions. It can also be assumed that there is no frame member on certain side(s) of the element. On each side of an element, the nail connection can be in different types, material properties, dimensions and spacings. The analysis also applies to the models with openings (e.g. doors or windows).

The programs can handle different loading combinations. In the static analysis, in-plane concentrated loads and out-of-plane distributed loads can be applied simultaneously. These loads may be constant or incremented individually in fixed or uneven steps. Load control mode or displacement control mode is used as input history in static monotonic analysis, whereas only displacement control mode is used in static cyclic analysis. In dynamic analysis, earthquake excitations can be in horizontal and/or vertical directions. At the same time, static constant loads can be applied to the structure.

The dynamic modal analysis procedure can be performed on the structure to obtain
the natural frequencies and the corresponding mode shapes before the dynamic time-history analysis is laid out. The fundamental frequency is needed to decide the damping coefficient required by the program and is also an indicator in model verification. The inertia mass can be consistent or lumped at nodes. The mass of panel and frame members may or may not be considered. When it is considered, the mass of the panel and frame members can also be consistent or lumped. The mass-proportional Reyleigh damping is used in the program.

The programs are written in Fortran 90. There is no limitation on the problem size from the programs themselves, because of the use of dynamically allocated arrays. The actual problem size that can be calculated and the computational time, however, heavily depend on the computer system used. The minimum system requirement for personal computers solving small to medium size problems is Pentium III 500 MHz and compatible processor, 128 MB RAM, and 10GB hard disk drive. For large size problems, a workstation or computer cluster is recommended.

Any unit system can be used in the programs. The unit should be consistent within the problem. For example, if forces are measured in Newton and coordinates in meter, the stresses must be in N/m², distributed loads in N/m², and acceleration in m/s².

For more detailed theoretical information, please refer to the context of this thesis.

A.3. Input and output files

Four input data files may be needed in an analysis:

(1) Input file one: Titles. This file is a batch file containing the names and locations of input data file, output file, and cyclic protocol file for static cyclic loading or earthquake record for dynamic loading,

(2) Input file two: Input data. This is a text file containing all the information of the studied
model. This file includes the three-dimensional nodal coordinates, material properties, 
loading patterns, and boundary conditions,

(3) Input file three: Cyclic protocol. This file contains the displacement points for scheduled 
loading cycles required in the static cyclic analysis, and

(4) Input file four: Time-acceleration history. This file can be a past earthquake record or a 
simulated time-acceleration series required in the dynamic analysis.

The preprocessor generates the following files:

(1) A text file containing all the information given by the input data file (Input file two) and 
relevant resultant parameters (*.tst),

(2) Files written in Lisp language to graphically present the meshes of panel and frame 
elements (Meshtopcover.lsp, Meshframe.lsp, and Meshbtmcover.lsp),

(3) A file containing a new set of time-acceleration history actually used in the calculation 
after defining the maximum acceleration increment (Accelerogram), which may not be 
the same as the original one, and

(4) A file containing the first 10 lowest eigenvalues and corresponding eigenvectors of a 
structure calculated by LightFrame3D_Eigensolver (Eigenvalues) for dynamic analysis.

The postprocessor generates the desired information in the following files:

(1) A text file containing general information of nodal displacements and forces (*.out),

(2) Files containing load-displacement points at specified locations (nodes) (PL*, up to five),

(3) Files containing panel stresses (Panelstress), frame stresses (Framestress), relative 
displacements of panel and frame elements (Reladisp), and nail forces (Nailforce),

(4) File generated in Lisp language to graphically present the deformed shapes of the 
structure (Defoshape.lsp), and
A.4. Structures and control parameters of input files

A.4.1. Input file one - Titles

Line 1. A title describing the problem, which will appear in the output file (*.out) and is up to 80 characters.

Line 2. Index for executive type.

0 – preprocessing.

1 - solving and post-processing.

Line 3. Input data file name.

For example:

D:\anybody\anysubdirectory\anycase.dat

Line 4. Output file name for 0 and 1 in Line 2.

For example:

D:\anybody\anysubdirectory\preproce.tst or
D:\anybody\anysubdirectory\solution.out

Line 5. Index for loading input.

0 - static monotonic loading and no further line in this file is needed.

1 - static cyclic loading.

2 - dynamic loading.

Line 6. If 1 or 2 is chosen in Line 5, give name of the file containing cyclic protocol or dynamic time-acceleration history.
For example:

D:\ anybody\ anysubdirectory\ Joshua.acc

End of the input file one.

This file should be saved in the same directory where the executable file of the program is located.

A.4.2. Input file two - Input data

Note: The appearance of characters ST (for static) and DY (for dynamic) in a line or following a control parameter indicates that the line or the parameter has to be included in the corresponding data file for static analysis or dynamic analysis. If not specified, a line should be in both static and dynamic data files.

Line 1. Number of plotting files, acceleration of gravity (DY). A maximum of five plotting files can be opened and are automatically named as PL1, PL2, and so on. These files show the calculated displacement and load results at specified locations (nodes).

MAXPL, GA

Line 2. Overall numbers of elements and nodes. Enter five numbers. They are:

1. Total number of exterior panel elements.
2. Total number of nodes in exterior panel elements.
3. Index for interior panel elements. 0 - No interior panel element; 1 - At least one interior panel element.
4. If 1 is chosen in number 3, total number of nodes in interior panel elements; otherwise 0.
5. Total number of frame nodes.

NELEM, NNODET, NELEMB, NNODEB, NNODEF

Line 3. Enter the number of Gaussian integration points along the connector span (x...
direction) and over the connector cross-section (y direction). The maximum numbers of them are 9 and 16, respectively.

\( \text{NGX, NGY} \)

**Line 4.** Global nodal coordinates. For each exterior panel node, one at a time, enter the node number followed by its \( X \), \( Y \) and \( Z \) coordinates. For 2-D case, specify one direction in all coordinates to be zero. This line is repeated until all nodes are taken. The origin of the coordinate system is arbitrary, but should be consistent within the whole structure to be analyzed.

\( \text{NOD, XG(NOD), YG(NOD), ZG(NOD)} \)

**Line 5.** Frame and connector types. Enter the number of different frame members and connectors.

\( \text{NFRAM, NFAST} \)

For example:

\[
2 \quad 4 \quad \text{(two different frame members and four different connectors)}
\]

**Line 6.** Index for input method of frame member properties. Enter 1 for inputting frame properties using stiffness, or enter 0 using MOE and cross-section geometry.

\( \text{NPROP} \)

**Line 7.** Frame member properties. If 1 is chosen in Line 6, then enter the following:

**Line 7-1.** Frame type number, \( EI_{\text{max}} \), \( EI_{\text{min}} \), \( GJ \), \( EA \), \( b \) (width of frame member, the dimension in the plane of panel), \( d \) (depth of frame member, dimension normal to the plane of panel), density (DY).

\( \text{NI, EIMAX(NI), EIMIN(NI), GJ(NI), EA(NI), BF(NI), HF(NI), DEN(NI)} \)

**Line 7-2.** Compression characteristics in the direction parallel to the layer fibers. Enter six numbers (\( Q_3 \neq 1.0 \)):
Q₀(force/length) Q₁(force/area) Q₂ Q₃ K(force/area)

D_{max}(length)

Q₀PA(NI), Q₁PA(NI), Q₂PA(NI), Q₃PA(NI), XKPA(NI), DMAXPA(NI)

Line 7-3. Compression characteristics in the direction perpendicular to the layer fibers. Enter six numbers (Q₃ ≠ 1.0):

Q₀ Q₁ Q₂ Q₃ K D_{max}

Q₀PE(NI), Q₁PE(NI), Q₂PE(NI), Q₃PE(NI), XKPE(NI), DMAXPE(NI)

For example:

1 21.21 3.87 1.24 3.21E+4 0.038 0.089

(type no. EI_{max} EI_{min} GJ EA b d)

500.0 1000.0 0.8 1.5 0.15E+06 0.003

500.0 1000.0 0.8 1.5 0.15E+06 0.003

If 0 is chosen in Line 6, then enter the following:

Line 7-1. Frame type number, E, G, b, d, density (DY).

NI, MOE(NI), GM(NI), BF(NI), HF(NI), DEN(NI)

Line 7-2. Compression characteristics in the direction parallel to the layer fibers. Enter six numbers (Q₃ ≠ 1.0):

Q₀ Q₁ Q₂ Q₃ K D_{max}

Q₀PA(NI), Q₁PA(NI), Q₂PA(NI), Q₃PA(NI), XKPA(NI), DMAXPA(NI)

Line 7-3. Compression characteristics in the direction perpendicular to the layer fibers. Enter six numbers (Q₃ ≠ 1.0):

Q₀ Q₁ Q₂ Q₃ K D_{max}

Q₀PE(NI), Q₁PE(NI), Q₂PE(NI), Q₃PE(NI), XKPE(NI), DMAXPE(NI)

For example:
1 9.5E+6 7.6E+5 0.038 0.089
(type no. E G b d)
500.0 1000.0 0.8 1.5 0.15E+06 0.003
500.0 1000.0 0.8 1.5 0.15E+06 0.003

Line 7 is repeated until all type of frame specified in Line 5 are entered.

Line 8. Connector properties. Enter the following five lines subsequently:

Line 8-1. Connector type number.

NI

Line 8-2. Modulus of elasticity (E), Yield stress (σ_y) and Length (L). Length must be greater than the thickness of the panel layer.

E(NI), σy(NI), L(NI)

Line 8-3. Cross-section type. 1 - circular (common nails); 2 - annular (spiral nails).

NCROSS

Line 8-4. Cross-section diameter. If 1 is chosen in Line 8-3, enter one number which is the outside diameter.

D(NI)

If 2 is chosen in Line 8-3, enter two numbers which are the inside diameter and the outside diameter, respectively.

DIN, D(NI)

Line 8-5. Load-slip characteristics for connector withdrawal loads (Q_3 ≠ 1.0).

Q_0(force) Q_1(force/length) Q_2 Q_3 K(force/length) D_max(length)

Q_0W(NI), Q_1W(NI), Q_2W(NI), Q_3W(NI), K(NI), D_maxW(NI)

Line 8 is repeated until all type of connectors specified in Line 5 are entered.

For example:
Line 9. Nodal numbering. For each element, one at a time, enter the following three or four lines subsequently:

Line 9-1. Element number, and index of exterior panel (1 - element is an exterior panel; 0 - if element is an opening).

\[ \text{NE, \ NECOV(NE)} \]

Line 9-2. Associated node number on exterior panels (counterclockwise about the positive direction of the normal to the plane where the component locates).

\[ \text{IET(NE,I), \ I = 1, 4} \]

Line 9-3. Associated node number on frame members (counterclockwise about the positive direction of the normal to the plane where the component locates).

\[ \text{IEF(NE,I), \ I = 1, 4} \]

Line 9-4. If 1 in Number 3 of Line 2 is chosen, associated node number on interior panels (counterclockwise about the positive direction of the normal to the plane where the component locates).

\[ \text{IEB(NE,I), \ I = 1, 4} \]

Line 9 is repeated until all elements specified in Number 1 of Line 2 are entered.

For example:

\[
\begin{array}{cccc}
1 & 1 & \text{ (element no., there is exterior panel) } \\
1 & 4 & 5 & 2 & \text{ (node number on exterior panel) }
\end{array}
\]
10 13 14 11 (node number on frame member)

(no interior panel here)

**Line 10.** Thickness of exterior panels.

TP

**Line 11.** Exterior panel properties. Enter the following numbers:

Line 11-1. \( E_x \) (Young's modulus in panel longitudinal direction), \( E_y \) (Young's modulus in panel transverse direction), \( G_{xy} \) (Shear modulus), \( \nu_{xy} \) (Primary Poisson's ratio), \( \nu_{yx} \) (Secondary Poisson's ratio, satisfying that \( \nu_{xy} E_x = \nu_{yx} E_y \)), usually \( \nu_{xy} < \nu_{yx} \), density (DY).

\[
\text{EXT, EYT, GXYT, NUYXT, NUXYT, DENTOP}
\]

Line 11-2. Compression characteristics in the direction parallel to the panel fibers. Enter six numbers (\( Q_3 \neq 1.0 \)):

\[
Q_0 \text{(force/length)} \quad Q_1 \text{(force/area)} \quad Q_2 \quad Q_3 \quad K \text{(force/area)} \quad D_{\text{max}} \text{(length)}
\]

\[
Q0PA(NFRAM+1), \ Q1PA(NFRAM+1), \ Q2PA(NFRAM+1), \ Q3PA(NFRAM+1), \ XKP(NFRAM+1), \ DMAXPA(NFRAM+1)
\]

Line 11-3. Compression characteristics in the direction perpendicular to the panel fibers. Enter six numbers (\( Q_3 \neq 1.0 \)):

\[
Q_0 \quad Q_1 \quad Q_2 \quad Q_3 \quad K \quad D_{\text{max}}
\]

\[
Q0PE(NFRAM+1), \ Q1PE(NFRAM+1), \ Q2PE(NFRAM+1), \ Q3PE(NFRAM+1), \ XKPE(NFRAM+1), \ DMAXPE(NFRAM+1)
\]

**Line 12.** Frame type associated with element. For each element, one at a time, enter element number and then the frame type number associated with the element, as defined in Lines 5 and 7, counterclockwise about the positive direction of the normal to the plane where the element locates. If there is no frame member on one side of the element, enter 0 for that side.
This line is repeated until all elements are taken.

\[ \text{NE}, \ \text{JMAT(NE,I), I = 1,4} \]

For example:

1 1 1 0 1 \text{(element 1, sides 1, 2 and 4 /w frame; side 3 no frame)}

2 0 1 1 1

3 1 1 0 1

4 0 1 1 1

\[ \ldots \ldots \]

Line 13. Connections between exterior panels and frame members. For each element, one at a time, enter element number and then counterclockwise connector type number, as defined in Lines 5 and 8, associated with the element. Continuing in the same input line, enter the spacing associated with each connector type, counterclockwise. If there is no connection because no frame member on a side of an element is shown or a connection has already been entered as part of another element, enter 0 for connector type number and 0.0 for connector spacing. This line is repeated until all elements are taken. When there is a nail line, there must be a frame member.

\[ \text{NE, NCT(NE,J), J = 1,4, SPT(NE,J), J = 1,4} \]

For example:

1 1 1 0 1 0.15 0.60 0.00 0.15

2 0 1 1 1 0.00 0.60 0.15 0.15

3 1 1 0 1 0.15 0.15 0.00 0.60

4 0 1 1 1 0.00 0.15 0.15 0.60

(element no. connector type spacings )
Line 14. Numbers of elements in a connection between exterior panel and frame and load modification factor. Enter the following five numbers in one line:

The first element number for frame, the last element number for frame, the first element number for panel, the last element number for panel, and finally the load modification factor.

The load modification factor is a factor multiplying to the nail connection force calculated by the SHYST subroutine. The numbering between the last element number for frame and the first number for panel must be continuous. The total number of elements cannot exceed ten.

IBEGFR, IFINFR, IBEGPA, IFINPA, FLD

For example:

1  6  7  10  1.0

If there is no interior sheathing panel, go to Line 21.

Line 15. The thickness of interior panels.

TB

Line 16. Interior panel properties. Enter the following numbers (referring to Line 11-1 for definitions):

Line 16-1. $E_x$, $E_y$, $G_{xy}$, $v_{xy}$, $v_{yx}$, density (DY)

EXB, EYB, GXYB, NUYXB, NUXYB, DENBOT
Line 16-2. Compression characteristics in the direction parallel to the panel fibers. Enter six numbers:

\[ Q_0 \quad Q_1 \quad Q_2 \quad Q_3 \quad K \quad D_{\text{max}} \]

\( Q_{0\text{PA}}(\text{NFRAM}+2), \quad Q_{1\text{PA}}(\text{NFRAM}+2), \quad Q_{2\text{PA}}(\text{NFRAM}+2), \quad Q_{3\text{PA}}(\text{NFRAM}+2), \)

\( X_{\text{KPA}}(\text{NFRAM}+2), \quad D_{\text{MAXPA}}(\text{NFRAM}+2) \)

Line 16-3. Compression characteristics in the direction perpendicular to the panel fibers. Enter six numbers:

\[ Q_0 \quad Q_1 \quad Q_2 \quad Q_3 \quad K \quad D_{\text{max}} \]

\( Q_{0\text{PE}}(\text{NFRAM}+2), \quad Q_{1\text{PE}}(\text{NFRAM}+2), \quad Q_{2\text{PE}}(\text{NFRAM}+2), \quad Q_{3\text{PE}}(\text{NFRAM}+2), \)

\( X_{\text{KPE}}(\text{NFRAM}+2), \quad D_{\text{MAXPE}}(\text{NFRAM}+2) \)

Line 17. Connections between interior panels and frame members. For each element, one at a time, enter element number and then counterclockwise the connector type number, as defined in Lines 5 and 8, associated with the element. Continuing in the same input line, enter the spacing associated with each connector type, counterclockwise. If there is no connection because no frame member on a side of an element is shown or a connection has already been entered as part of another element, enter 0 for connector type number and 0.0 for connector spacing. This line is repeated until all elements are taken.

\( \text{NE}, \quad \text{NCB}(\text{NE}, J), \quad J = 1, 4, \quad \text{SPB}(\text{NE}, J), \quad J = 1, 4 \)

Line 18. Numbers of elements of connections between interior panel and frame and load modification factor. Enter the following five numbers in one line:

The first element number for frame, the last element number for frame, the first element number for panel, the last element number for panel, and finally the load modification factor. The numbering between the last element number for frame and the first number for panel must be continuous. The total number of elements cannot exceed ten. (See graph in Line
14.)

**IBEGFR, IFINFR, IBEGPA, IFINPA, FLD**

**Line 19.** Index for insulation filler. Enter 1 if there is insulation filler, or 0 if no such filler.

**NFOAM**

If 0 is chosen in Line 19, go to Line 21.

**Line 20.** Properties of insulation filler. Enter the following four lines subsequently:

**Line 20-1.** Properties in compression, similar to that used in load-slip relationship of connectors. Enter five numbers:

\[ Q_0, Q_1, K, D_{\text{max}}, K_e \]

\[ Q_{\text{OC}}, Q_{\text{IC}}, X_{\text{KC}}, D_{\text{MAXC}}, K_{\text{EC}} \]

**Line 20-2.** Properties in tension. Enter two numbers:

\[ K, D_{\text{max}} \]

\[ X_{\text{KT}}, D_{\text{MAXT}} \]

**Line 20-3.** Properties in shear. Enter two numbers:

\[ G, G_{\text{max}} \]

\[ G_{\text{K}}, G_{\text{MAX}} \]

**Line 20-4.** Thickness of the filler. Enter one number:

\[ T_f \]

\[ T_{\text{EF}} \]

**Line 21.** Enter the number of nodes with global support conditions.

**NNBC**

**Line 22.** Global support conditions (boundary conditions). For each node, one at a time in one line, enter the following numbers:

1. Node number.

2. Number of support conditions at that node.
3. A list of codes for those support conditions.

\[
\begin{align*}
U &= 1 \\
V &= 2 \\
W &= 3 \\
Rot-X &= 4 \\
Rot-Y &= 5 \\
Rot-Z &= 6
\end{align*}
\]

4. An index specifying the conditions. 0 - all conditions are zero; 1 - some conditions are not zero. If 1 is chosen, Line 22-1 is needed to specify the value of the support conditions.

\[\text{NBC(I), KBC(I), IIBC(J), J=1,KBC(I), KCAS}\]

Line 22-1. Specify the value of support conditions.

\[\text{BC(I,J), J = 1, KBC(I)}\]

For example:

(1) 20 3 1 2 3 0

(node 20 has 3 support conditions: movements in \(U, V,\) and \(W\) directions all are zero)

(2) 20 3 1 2 3 1 1.0 0.0 0.0

(node 20 has 3 support conditions: displacement in \(U\) direction is 1.0 while the movements in \(V\) and \(W\) directions are zero)

Line 23. Selection of control modes for static monotonic analysis. 1 - load control; 2 - displacement control. In static cyclic analysis and dynamic analysis, use 2 only.

\[\text{NCONTR}\]

Line 24. Steps. Enter the number of load or displacement steps in the run (For simplicity,
this line is kept in the dynamic data file). For cyclic loading condition, this parameter is replaced by the step value given by the file of cyclic protocol automatically.

**NSTEPS**

**Line 25.** Enter the number of nodes with concentrated loads. If there is no concentrated load, enter 0 and go to Line 27.

**NCONC**

**Line 26 (ST).** Concentrated loads. For each node, one at a time in the same line, enter the following five numbers:

1. Loaded node number.
2. Three load components in $X$, $Y$, $Z$ directions under global coordinate system.
3. Load increment code. 0 - if the load is constant; 1 - the load will be increased. If displacement control mode is chosen in Line 23, this code must be set to 0. If 1 is chosen, Line 26-1 is needed to specify the value of the load increments.

**ICON(I), PCX(I), PCY(I), PCZ(I), KCON(I)**

**Line 26-1.** Specify the load increments in three directions.

**STCONX(I), STCONY(I), STCONZ(I)**

For example:

1. 14 -2.0 0.0 0.5 0  
   (Node 14 has loads constantly acting on. The load components are $X = -2.0$, $Y = 0.0$, and $Z = 0.5$.)

2. 12 0.1 2.0 0.0 1  
   1.0 0.0 0.0  
   (Node 12 has loads acting on. The load component in $X$ direction has initial value of 0.1 and will increase 1.0 in each step, while those in $Y$ and $Z$...
directions will remain constant during calculation.)

Line 26 (DY). Concentrated loads and masses. For each node, one at a time in the same line, enter the following six numbers:

1. Node number.

2. Three load components in X, Y, Z directions under global coordinate system (The program may then convert those specified load values into masses).

3. Index for dead load: 0 - the load is converted to mass only and is not considered as a dead load; 1 - the load is converted to mass and is also considered as a dead load.

4. Index for load components: 0 - the load is not converted to mass; 1 - the component in X direction is converted to mass; 2 - the component in Y direction is converted to mass; 3 - the component in Z direction is converted to mass.

ICON(I), PCX(I), PCY(I), PCZ(I), KCON(I), ICONMASS(I)

Line 27. Enter the number of elements with distributed loads. If there is no distributed load, enter 0. If it is under load control, go to Line 30; otherwise go to Line 29.

NDIS

Line 28(ST). Distributed loads. For each element, one at a time, enter the following three lines:

Line 28-1. Enter three numbers:

1. Loaded element number.

2. Applied load. It is assumed that this load is applied perpendicular to the element and its positive direction is in the positive direction of the normal of exterior panel plane where the element locates.

3. Load increment code. 0 - the load is constant; 1 - the load is not constant. If
displacement control mode is chosen in Line 23, this code must be set to 0. If 1 is chosen, Line 28-3 is needed to specify the value of the load increments.

\[ \text{IDIS}(I), \text{PD}(I), \text{KDIS}(I) \]

Line 28-2. Enter the global coordinates of two points \((P_1 \text{ and } P_2)\) on the diagonal corners of the loaded area (See the graph below).

\[ \text{XDGl}(I), \text{YDG1}(I), \text{ZDG1}(I), \text{XDG2}(I), \text{YDG2}(I), \text{ZDG2}(I) \]

Line 28-3. Specify the load increments, if 1 is chosen in Number 3 of Line 28-1.

\[ \text{STDIS}(I) \]

For example:

(1)  1   -0.01   0
     0.0 0.0 0.0 0.4 1.2 0.0

(Element 1 has a constant distributed load, -0.01, acting in the area enclosed by \(P_1 (0.0, 0.0, 0.0)\) and \(P_2 (0.4, 1.2, 0.0)\).)

(2)  1   -0.01   1
     0.0 0.0 0.0 0.4 1.2 0.0
     -1.0

(Element 1 has an initial distributed load, -0.01, acting in the area enclosed by \(P_1 (0.0, 0.0, 0.0)\) and \(P_2 (0.4, 1.2, 0.0)\). The will decrease 1.0 in each step.)

Line 28(DY). Distributed loads and masses. For each element, one at a time, enter the
following two lines:

Line 28-1. Enter four numbers:

1. Loaded element number.

2. Applied load. It is assumed that this load is applied perpendicular to the element and its positive direction is in the positive direction of the normal of exterior panel plane where the element locates. When the load is considered to be converted into mass, its direction must along Z direction in a global coordinate system, i.e. it must be applied to a roof.

3. Index for dead load. 0 - the load is converted to mass only and is not considered as a dead load; 1 - the load is converted to mass and is also considered as a dead load.

4. Index for mass conversion. 0 - the load is not converted to mass; 1 - the load is converted to mass.

IDIS(I), PD(I), KDIS(I), IDISMASS(I)

Line 28-2. Enter the global coordinates of two points (P₁ and P₂) on the diagonal corners of the loaded area (See graph in Line 28(ST)).

XDGl(I), YDGl(I), ZDGl(I), XDG2(I), YDG2(I), ZDG2(I)

If load control mode is chosen in Line 23, go to Line 30.

Line 29. Displacement control. Enter the following three numbers in one line (For simplicity, this line is kept in the dynamic data file):

1. Node number where displacement is specified.

2. Degree of freedom specified, using codes provided in Line 22.

3. Displacement specified.

NDISP, NNDOF, ADISP
For example:

24 1 0.001

(Node 24 in $U$ direction is assigned a displacement of 0.001 for each step.)

**Line 30.** Plot nodal load-displacement results. For each plotting file as specified in Line 1, one at a time in one line, enter the following two numbers:

1. Node number to be plotted.
2. Degree of freedom to be shown, using codes provided in Line 22.

$$\text{NMON}(I), \text{KKMON}(I), I=1, \text{MAXPL}$$

For example:

24 1

(The displacement and load values of Node 24 in $U$ direction will be shown in one plotting file.)

**Line 31.** Plot connection deformation results. Enter the following three numbers to decide the connection location:

1. Element number.
2. Side number.
3. Gauss point number (1-4).

$$\text{IELEMTC, ISIDEC, IGPC}$$

**Line 32.** Enter 1 if monitoring the panel edge tearing is desired; otherwise enter 0. If 0 is chosen, go to Line 39.

$$\text{NTEAC}$$

**Line 33.** Enter the maximum edge tearing force for exterior panel.

$$\text{FMAX}$$

**Line 34.** Enter the number of elements of exterior panel where tearing is monitored.
NETOP

Line 35. Tearing. For each element, one at a time in one line, enter the following numbers:

1. Element number.
2. Number of sides where tearing is monitored.
3. Corresponding side numbers.

This line is repeated until all elements specified in Line 34 are entered.

\[ \text{NE}, \text{ NEM}, \text{ NTEA}(J), J = 1, \text{ NEM} \]

For example:

\[ 2 \quad 3 \quad 1 \quad 2 \quad 4 \]

(Element 2 has 3 sides to be monitored for tearing. They are sides 1, 2 and 4.)

If there is no interior panel, go to Line 39.

Line 36. Enter the maximum edge tearing force for interior panel.

\[ \text{FBMAX} \]

Line 37. Enter the number of elements of interior panel where tearing is monitored.

\[ \text{NEBOT} \]

Line 38. Tearing. For each element, one at a time in one line, enter the following numbers:

1. Element number.
2. Number of sides where tearing is monitored.
3. Corresponding side numbers.

This line is repeated until all elements specified in Line 37 are entered.

\[ \text{NE}, \text{ NEM}, \text{ NTEA}(J), J = 1, \text{ NEM} \]

Line 39. Enter 1 if checking frame bending strength is desired; otherwise enter 0.

\[ \text{NBEND} \]

(ST) If 0 is chosen and only one type of loading, either concentrated loads or distributed loads, in the load control mode is considered, the static data file is completed. If both types
of loading are considered in the load control mode, go to Line 41.

(DY) If 0 is chosen, go to Line 42.

Line 40. Enter the bending strength of frame members, if 1 is chosen in Line 39.

SBEND

Line 41 (ST). Load step in plotting files. Enter 1, if the largest load step in concentrated loads is chosen as the load step in plotting files. Enter 2, if the largest load step in distributed loads is chosen as the load step in plotting files. This line is necessary only when both concentrated loads and distributed loads are considered. The static data file is completed.

NPSTEP

Line 42 (DY). Enter 1 for consistent mass or enter 2 for lumped mass.

LUMPFLAG

Line 43 (DY). Panel and frame masses. Enter the following two parameters:

1. Panel mass. 0 - panel mass is not considered; 1 - panel mass is considered.

2. Frame mass. 0 - frame mass is not considered; 1 - frame mass is considered.

MASS_PANEL, MASS_FRAME

Line 44 (DY). Print files. 0 - no output file is printed; 1 - output files are printed. If 0 is selected, the dynamic data file is completed.

PRT

Line 45 (DY). If 1 is chosen in Line 44, give the step interval between two output prints. The dynamic data file is completed.

PRTSTEP

For example:

100  (Print the results to the output files once in every 100 steps)

End of the input file two.
It is recommended to perform a preprocessing to verify all of the input parameters and to obtain the necessary text and graphic information of the case.

To display the graphic mesh from the files generated by the preprocessor, run AutoCAD™ software (R14 or later version). In the command line, type: appload <Return>. A window will pop-up. Select and load Meshtopcover.lsp, Meshframe.lsp, or Meshbtmcover.lsp. Usually these files are saved in the working directory, but exact location may vary when different Fortran compilers are used.

A.4.3. Input file three - Cyclic protocol

Line 1. Enter the total segments in the cyclic protocol.

NSEG

Line 2. Enter the following three parameters:

1. Starting displacement.
2. Increment.
3. Number of steps.

This line is repeated until all the segments specified in Line 1 are entered.

STPT, CYCINCR, INCR

For example:

0.001 0.002 10  (Displacement starts from 0.001 and reaches to 0.021 in 10 steps with equal increments of 0.002)

0.022 0.0005 20  (Continuing from the last segment, the displacement starts from 0.022 and reaches to 0.032 in 20 steps with equal increments of 0.0005)

End of input file three.

This file is sometimes used in static monotonic analysis when the steps are not evenly incremented. In general, larger steps can be adopted in the beginning of an analysis or in the
case of greater stiffness. When the structure is approaching its maximum capacity or it is subject to an unloading (flattening), smaller steps should be used thus to achieve a faster convergence.

A.4.4. Input file four – Time-acceleration history

Line 1. Name of the time-acceleration history.

NAMEACCEFILE

Line 2. Format of the time-acceleration history. Enter 1 if the time steps in the history are even and equal, otherwise enter 2.

IFORMAT

Line 3. Basic control parameters. Enter the following parameters in one line:

If 1 is chosen in Line 2:

1. Total steps.
2. Time step.
3. Maximum allowable acceleration increment in one step.
4. Damping coefficient.
5. Amplification factor of acceleration:

NSTEP, TSTEP, ASTEP, ALFA, FA

If 2 is chosen in Line 2:

1. Total steps.
2. Maximum allowable acceleration increment in one step.
3. Damping coefficient.
4. Amplification factor of acceleration.

NSTEP, ASTEP, ALFA, FA

Line 4. Index for horizontal or vertical acceleration components. Enter the following two
parameters in one line:

1. Horizontal acceleration component. 1 - use horizontal record; 0 - no horizontal record.

2. Vertical acceleration component. 1 - use vertical record; 0 - no vertical record.

At least one parameter is 1.

\[ \text{NAX, NAY} \]

**Line 5. Acceleration points.**

If Format 1 is chosen in Line 2, enter the horizontal acceleration point, if any, followed by the vertical acceleration point, if any.

\[ \text{AX}(i), \text{AY}(i) \]

If Format 2 is chosen in Line 2, enter the time step, the horizontal acceleration point, if any, followed by the vertical acceleration point, if any.

\[ \text{TPT}(i), \text{AX}(i), \text{AY}(i) \]

Line 5 is repeated until the acceleration points specified by Parameter 1 in Line 3 are entered.

The unit of acceleration should be consistent with the unit system used in the whole problem.

For example, if \( N \) and \( m \) are used, then the acceleration must be in \( m/s^2 \).

End of input file four.

**A.4.5. Examples of input files**

1. Static data file

This file describes the data of a conventional light-frame shear wall. The dimension of the wall is 2.4 m by 2.4 m. The shear wall is sheathed one-side with standard-sized oriented strand board (OSB) panels. The wall is loaded at one of its top corners with incremental lateral load. A total of 21.9 kN constant vertical loads is distributed evenly on its top beam. The mesh is in the X-Y plane with a total of 12 elements.
1
12 30 0 0 21
6 10
1 0.0000 0.0000 0.0000
2 0.0000 1.2192 0.0000
3 0.0000 1.2192 0.0000
4 0.0000 2.4384 0.0000
5 0.4064 0.0000 0.0000
6 0.4064 1.2192 0.0000
7 0.4064 1.2192 0.0000
8 0.4064 2.4384 0.0000
9 0.8128 0.0000 0.0000
10 0.8128 1.2192 0.0000
11 0.8128 1.2192 0.0000
12 0.8128 2.4384 0.0000
13 1.2192 0.0000 0.0000
14 1.2192 1.2192 0.0000
15 1.2192 1.2192 0.0000
16 1.2192 2.4384 0.0000
17 1.2192 1.2192 0.0000
18 1.2192 2.4384 0.0000
19 1.6256 0.0000 0.0000
20 1.6256 1.2192 0.0000
21 1.6256 1.2192 0.0000
22 1.6256 2.4384 0.0000
23 2.0320 0.0000 0.0000
24 2.0320 1.2192 0.0000
25 2.0320 1.2192 0.0000
26 2.0320 2.4384 0.0000
27 2.4384 0.0000 0.0000
28 2.4384 1.2192 0.0000
29 2.4384 1.2192 0.0000
30 2.4384 2.4384 0.0000
2 1
0
1 9500.0E+03 760.0E+03 0.038 0.089
150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025
150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025
2.9500.0E-01 760.0E-01 0.038 0.089
150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025
150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025
1
0.2000E+09 0.2500E+06 0.05
2
0.002 0.0025
1.00 0.0 0.8 2.0 1000.0 0.002
1 1
1 5 6 2
31 34 35 32
2 1
3 7 8 4
32 35 36 33
3 1
5 9 10 6
34 37 38 35
4 1
7 11 12 8

Appendix A Program User’s Manual
<table>
<thead>
<tr>
<th>Row</th>
<th>Data</th>
<th>Row</th>
<th>Data</th>
<th>Row</th>
<th>Data</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>35 38 39 36</td>
<td>2</td>
<td>9 13 14 10</td>
<td>3</td>
<td>37 40 41 38</td>
</tr>
<tr>
<td>4</td>
<td>6 1</td>
<td>5</td>
<td>11 15 16 12</td>
<td>6</td>
<td>38 41 42 39</td>
</tr>
<tr>
<td>7</td>
<td>7 1</td>
<td>8</td>
<td>13 19 20 14</td>
<td>9</td>
<td>40 43 44 41</td>
</tr>
<tr>
<td>10</td>
<td>8 1</td>
<td>11</td>
<td>17 21 22 18</td>
<td>12</td>
<td>41 44 45 42</td>
</tr>
<tr>
<td>13</td>
<td>9 1</td>
<td>14</td>
<td>19 23 24 20</td>
<td>15</td>
<td>43 46 47 44</td>
</tr>
<tr>
<td>16</td>
<td>10 1</td>
<td>17</td>
<td>21 25 26 22</td>
<td>18</td>
<td>44 47 48 45</td>
</tr>
<tr>
<td>19</td>
<td>11 1</td>
<td>20</td>
<td>23 27 28 24</td>
<td>21</td>
<td>46 49 50 47</td>
</tr>
<tr>
<td>22</td>
<td>12 1</td>
<td>23</td>
<td>25 29 30 26</td>
<td>24</td>
<td>47 50 51 48</td>
</tr>
<tr>
<td>25</td>
<td>47 50 51 48</td>
<td>26</td>
<td>0.0095</td>
<td>27</td>
<td>6400E+03 4700E+03 1500E+03 0.22 0.3</td>
</tr>
<tr>
<td>28</td>
<td>150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025</td>
<td>29</td>
<td>150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025</td>
<td></td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>1 1 1 2 1</td>
<td>31</td>
<td>2 2 1 1 1</td>
<td>32</td>
<td>3 1 1 2 1</td>
</tr>
<tr>
<td>33</td>
<td>4 2 1 1 1</td>
<td>34</td>
<td>5 1 1 2 1</td>
<td>35</td>
<td>6 2 1 1 1</td>
</tr>
<tr>
<td>36</td>
<td>7 1 1 2 1</td>
<td>37</td>
<td>8 2 1 1 1</td>
<td>38</td>
<td>9 1 1 2 1</td>
</tr>
<tr>
<td>39</td>
<td>10 2 1 1 1</td>
<td>40</td>
<td>11 1 1 2 1</td>
<td>41</td>
<td>12 2 1 1 1</td>
</tr>
<tr>
<td>42</td>
<td>1 1 1 1 1 0.1524 0.3048 0.1524 0.1524</td>
<td>43</td>
<td>2 1 1 1 1 0.1524 0.3048 0.1524 0.1524</td>
<td></td>
<td></td>
</tr>
<tr>
<td>44</td>
<td>3 1 1 1 0 0.1524 0.3048 0.1524 0.1524 0.0000</td>
<td>45</td>
<td>4 1 1 1 0 0.1524 0.3048 0.1524 0.1524 0.0000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>46</td>
<td>5 1 1 1 0 0.1524 0.3048 0.1524 0.1524 0.0000</td>
<td>47</td>
<td>6 1 1 1 0 0.1524 0.3048 0.1524 0.1524 0.0000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>48</td>
<td>7 1 1 1 0 0.1524 0.3048 0.1524 0.1524 0.0000</td>
<td>49</td>
<td>8 1 1 1 0.1524 0.3048 0.1524 0.1524 0.0000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>50</td>
<td>9 1 1 1 0.1524 0.3048 0.1524 0.1524 0.0000</td>
<td>51</td>
<td>10 1 1 1 0.1524 0.3048 0.1524 0.1524 0.0000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>52</td>
<td>11 1 1 1 0.1524 0.3048 0.1524 0.1524 0.0000</td>
<td>53</td>
<td>12 1 1 1 0.1524 0.1524 0.1524 0.1524 0.0000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>54</td>
<td>1 6 7 10 1.0</td>
<td>55</td>
<td>14</td>
<td>56</td>
<td>31 6 1 2 3 4 5 6 0</td>
</tr>
</tbody>
</table>
2. Dynamic data file

This file describes the data of the same shear wall as in the static data file. The wall is subject to an earthquake ground motion when an inertia mass of 5,400 kg acts on its top. The mesh is in X-Z plane with a total of 12 elements.

<table>
<thead>
<tr>
<th>Element</th>
<th>X-coordinate</th>
<th>Y-coordinate</th>
<th>Z-coordinate</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>9.81</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>2.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>3.0000</td>
</tr>
<tr>
<td>4</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>4.0000</td>
</tr>
<tr>
<td>5</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>5.0000</td>
</tr>
<tr>
<td>6</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>6.0000</td>
</tr>
<tr>
<td>7</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>7.0000</td>
</tr>
<tr>
<td>8</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>8.0000</td>
</tr>
<tr>
<td>9</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>9.0000</td>
</tr>
<tr>
<td>10</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>10.0000</td>
</tr>
<tr>
<td>11</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>11.0000</td>
</tr>
<tr>
<td>12</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>12.0000</td>
</tr>
<tr>
<td>13</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>13.0000</td>
</tr>
<tr>
<td>14</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>14.0000</td>
</tr>
<tr>
<td>15</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>15.0000</td>
</tr>
</tbody>
</table>

Appendix A Program User's Manual
16 1.2192 0.0000 2.4384
17 1.2192 0.0000 1.2192
18 1.2192 0.0000 2.4384
19 1.6256 0.0000 0.0000
20 1.6256 0.0000 1.2192
21 1.6256 0.0000 1.2192
22 1.6256 0.0000 2.4384
23 2.0320 0.0000 0.0000
24 2.0320 0.0000 1.2192
25 2.0320 0.0000 1.2192
26 2.0320 0.0000 2.4384
27 2.4384 0.0000 0.0000
28 2.4384 0.0000 1.2192
29 2.4384 0.0000 1.2192
30 2.4384 0.0000 2.4384
3 1
0
1 9500.0E+03 760.0E+03 0.038 0.089 0.4
150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025
150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025
2 9500.0E+03 760.0E+03 0.038 0.089 0.4
150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025
150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025
3 9500.0E+03 760.0E+03 0.076 0.089 0.4
150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025
150.0 2.8E+04 0.8 1.5 0.40E+06 0.0025
1 0.2000E+09 0.2500E+06 0.05
2
0.002 0.0025
1.00 0.0 0.8 2.0 1000.0 0.002
1 1
1 5 6 2
31 34 35 32
2 1
3 7 8 4
32 35 36 33
3 1
5 9 10 6
34 37 38 35
4 1
7 11 12 8
35 38 39 36
5 1
9 13 14 10
37 40 41 38
6 1
11 15 16 12
38 41 42 39
7 1
13 19 20 14
40 43 44 41
8 1
17 21 22 18
41 44 45 42
9 1
19 23 24 20

Appendix A Program User's Manual
Appendix A Program User's Manual
3. Cyclic protocol file

This file contains three small cycles, three large cycles and one unidirectional push-over. The three small cycles have the same amplitude of 16 mm and the three large cycles have the same amplitude of 32 mm. The final unidirectional push-over reaches 70 mm.

Another format can be used, in which one line represents one step:

```
400
0.001 0.0 0
0.002 0.0 0
0.003 0.0 0
0.004 0.0 0
0.005 0.0 0
0.006 0.0 0
```

... ...
4. Time-acceleration history file

Format 1.

JOSHUA_jtest
1
1836 0.02D0 0.02D0 0.3D0 1.D0
1 0
0.057969329
-0.104540805
-0.161530065
-0.051302433
0.059312387
0.055029120
-0.183430398
-0.270305742

Format 2.

JOSHUA_jtest
1
1836 0.02D0 0.3D0 1.D0
1 0
0.200000E-01 0.579693E-01
0.300000E-01 -0.232857E-01
0.400000E-01 -0.104541E+00
0.600000E-01 -0.161530E+00
0.700000E-01 -0.106416E+00
0.800000E-01 -0.513024E-01
0.900000E-01 0.400498E-02
0.100000E+00 0.593124E-01
0.120000E+00 0.550291E-01
0.126667E+00 -0.244574E-01
0.133333E+00 -0.103944E+00
0.140000E+00 -0.183430E+00
0.160000E+00 -0.270306E+00

A.4.6. Examples of the output file of an eigensystem

This file gives the results of eigenvalues and eigenvectors of a system calculated by LightFrame3D_Eigen. A total of 10 columns of eigenvectors (Only four of them are shown here) are corresponding to the 10 eigenvalues. Each eigenvector contains the same rows as the number of degrees of freedom of the system, from which one of the mode shapes can be drawn. The $\alpha$ value for the mass-proportional damping at 5% damping ratio is also given.

Job title: box_open_24e_dy
## METHOD 1: SUBSPACE

**EIGENVALUES AND NATURAL FREQUENCIES**

<table>
<thead>
<tr>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>58978E+03</td>
</tr>
<tr>
<td>92801E+03</td>
</tr>
<tr>
<td>48046E+04</td>
</tr>
<tr>
<td>77289E+04</td>
</tr>
<tr>
<td>20081E+04</td>
</tr>
<tr>
<td>23376E+05</td>
</tr>
<tr>
<td>68356E+05</td>
</tr>
<tr>
<td>12624E+06</td>
</tr>
<tr>
<td>14865E+06</td>
</tr>
<tr>
<td>14891E+06</td>
</tr>
</tbody>
</table>

**EIGENVECTORS**

<table>
<thead>
<tr>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>-39895E-04</td>
</tr>
<tr>
<td>-74332E-04</td>
</tr>
<tr>
<td>16845E-05</td>
</tr>
<tr>
<td>16845E-05</td>
</tr>
<tr>
<td>16845E-05</td>
</tr>
<tr>
<td>16845E-05</td>
</tr>
<tr>
<td>16845E-05</td>
</tr>
<tr>
<td>16845E-05</td>
</tr>
</tbody>
</table>

### POSSIBLE ALFA AT 5% DAMPING RATIO IS

0.24285E+01

A.5. Viewing graphic files generated by the postprocessor

Two files are generated by the postprocessor for visually displaying the deformed shape of the structural system being analyzed. One is Defoshape.lsp, which presents the structural mesh directly in AutoCAD software, and the other is Slides.scr, which is a script file generated by the LightFrame3D program during its execution to create continuously running slides showing the structural deformation step by step in both AutoCAD™ and MS Powerpoint™ software.
To run Defoshape.lsp, start AutoCAD™ (R14 or later version). In the command line, type: appload <Return>. When a window shows up, select and load Defoshape.lsp from the working directory (The exact location of this file may vary for different Fortran compilers). The deformation of the structural system being studied in all recorded steps will be displayed on the screen.

To save slides generated by the LightFrame3D program, it is necessary to create a directory as C:\slides. The slides are automatically saved in two formats. One format is the AutoCAD™ slide format with suffix .sld, which is used for slide shows on the AutoCAD™ screen; and the other is the Windows™ metafile with suffix .WMF, which can be load by many programs, for example, MS Powerpoint™. To present the slides on the AutoCAD™ screen, start AutoCAD™ (R14 or later version) and in the command line, type: script <Return>. From the window, select and open file Slides.scr, which is in the working directory. To run a slide show in MS Powerpoint™, start MS Powerpoint™, then use command: "insert to load slides with suffix .WMF from directory C:\slides and save them into a new file". Usually, creating a slide show in the Windows metafile format is more convenient than doing that in the AutoCAD™ slide format (.sld), because MS Powerpoint™ is more commonly found on computers.